

**Megger**<sup>®</sup>

**User  
Manual**

**AVTS**

Advanced **V**isual **T**est **S**oftware

Part Number 710023

Rev 2 9/26/2007

## Revision History

<u>Revision</u>	<u>ECN #</u>	<u>Date</u>
1	30524	11/21/2005
2	30960	09/26/2007

# Table Of Contents

Welcome to AVTS Help.....	1
AVTS Quick Links to Topics.....	3
Introduction to AVTS.....	5
Introduction to AVTS.....	5
AVTS Installation Disk.....	6
AVTS Directory Structure.....	7
How to Use AVTS Help.....	9
AVTS General Theory of Operation.....	13
Overview of AVTS.....	15
Menus and Toolbars.....	17
Menus and Toolbars.....	17
Main Menu and Toolbars.....	18
Main Menu and Toolbars.....	18
Main Menu File.....	20
Main Menu File.....	20
File New Relay.....	22
File New Device.....	23
File New Database.....	24
File New Subtest.....	26
File Open Subtest.....	27
File Open Relay via Barcode.....	28
Main Menu Edit.....	29
Main Menu View.....	30
Main Menu View.....	30
View Workbook.....	33
View Workspace.....	35
View Status Bar.....	36
View Toolbars.....	37
View Reports.....	38
View Reports.....	38
View Reports General.....	39
Report Export.....	41
Relay List.....	42
Relay Schedule.....	44
Relay Settings.....	46
Device List.....	48
Device List with Settings.....	50
Test Results.....	51
View Instrument Log.....	53
Main Menu Tools.....	55

Main Menu Tools.....	55
Database Utilities .....	56
Database Utilities .....	56
Create Subset .....	57
Merge Subset.....	59
Results Archive .....	62
Compact Database.....	64
Update.....	65
Initialize .....	67
Calculator .....	68
Fault Calculator.....	69
Fault Calculator.....	69
Fault Calculator .....	70
Equation for Fault Calculator .....	72
SS1 Converter.....	75
SS1 Converter... ..	75
SS1 Converter Dialog .....	76
SS1 File Example .....	80
Tools HyperTerminal .....	85
Tools Battery Simulator .....	91
Tools Import Data.....	92
Tools Import Data.....	92
Import PulseMaster Database .....	93
Import ProTest Data .....	96
Import ProTest Data.....	96
Import ProTesT Data Setup.....	97
ProTest Data Results .....	101
Import ASPEN Relay Database.....	104
AVTS Relay Data File.....	105
AVTS Device Data File.....	107
Tools Export Data.....	108
Tools Export Data.....	108
AVTS Relay Data File export.....	109
AVTS Device Data File export.....	110
Tools Options .....	111
Tools Options .....	111
General Tab .....	112
Default Settings Tab.....	115
Organization Tab.....	116
Channel Labels Tab .....	118
New Password Tab .....	119
Devices Tab.....	124

## Table Of Contents

Devices Tab.....	124
Test Set Inputs and Outputs .....	126
Test Set Ethernet Communication .....	127
Test Set IEEE-488 Communication .....	132
Test Set Serial Communication.....	133
TCM-915S IEEE-488 Communication .....	137
Auxiliary COM Ports.....	139
Devices Tab Lower Area.....	140
Scanner Tab.....	141
Main Menu Window .....	142
Main Menu Help .....	144
Main Menu Toolbar .....	145
Device Menu and Toolbars.....	148
Device Menu and Toolbars.....	148
Device Menu File.....	149
Device Menu Edit .....	150
Device Menu View.....	152
Device Menu Tools.....	153
Device Toolbar .....	154
Relay Menu and Toolbars .....	155
Relay Menu and Toolbars .....	155
Relay Menu File .....	156
Relay Menu Edit.....	158
Relay Menu View.....	161
Relay Menu View .....	161
Current Test Results .....	162
Test Procedures.....	164
Variable Watch.....	165
Add All Watch Variables .....	166
Remove All Watch Variables.....	167
Update Theoretical.....	168
Relay Menu Editor .....	169
Relay Menu Editor .....	169
Editor Align Toolbar .....	171
Editor Nudge Toolbar .....	173
Editor Structure Toolbar .....	174
Editor Zoom / Pan Toolbar .....	176
Relay Menu Tools.....	178
Relay Menu Tools .....	178
Relay Tools Import Data.....	180
Relay Tools Import Data .....	180
Relay Import AVTS Test Data File.....	181

Relay Import MPRT Results.....	182
Relay Import MPRT Settings.....	185
Relay Tools Export Data .....	188
Relay Tools Export Data .....	188
Relay Copy Results to Clipboard .....	189
Relay Export AVTS Test Data File.....	190
Relay Export Results to Word.....	192
Relay Export Current Test Results to Word .....	194
Relay Export MPRT Settings .....	196
Connection Images Toolbar .....	197
Relay Tools Execute Group .....	200
Relay Menu Window .....	201
Relay Toolbar .....	202
Workspace Pane.....	203
Workspace Pane.....	203
Devices Tab Drop Menus.....	204
Relays Tab Drop Menus .....	206
Tests Tab Drop Menus .....	208
MPRT Tab Drop Menus.....	210
MPRT Tab Drop Menus .....	210
MPRT Tab.....	212
Workbook Pane .....	215
Workbook Pane .....	215
Devices Settings Tab .....	216
Devices Settings Tab .....	216
Device Settings Tab Drop Menu .....	218
Devices General Information Tab.....	220
Relays Settings Tab .....	222
Relays Settings Tab.....	222
Settings Tab Drop Menu.....	223
Relay Test Tab .....	224
Relay Test Tab.....	224
Test Tab Drop Menu .....	225
Capture Tab .....	227
Capture Tab.....	227
Capture Tab Drop Menu .....	228
Capture Tab Properties Dialog.....	231
Test Editor Tab.....	234
Test Editor Tab .....	234
Test Editor Tab Drop Menu.....	235
Test Editor Controls Toolbar .....	237
Test Editor Controls Toolbar .....	237

## Table Of Contents

1-Touch Control .....	240
1-Touch Control .....	240
1-Touch Script.....	241
1-Touch Modbus.....	243
1-Touch Modbus .....	243
1-Touch Modbus Properties .....	244
Modbus Relay Settings .....	246
Modbus Device Settings .....	248
1-Touch SS1 .....	251
Analog Input Control .....	252
BASIC Control.....	253
Battery Simulator Control .....	255
Binary Search Control .....	256
Binary Search Control.....	256
Binary Search Control Properties.....	258
Binary Search Control Example .....	262
Calculation Control.....	269
Click On Fault Control .....	271
Click On Fault Control.....	271
Click On Fault Properties .....	273
Click On Fault Graph .....	275
Click On Fault Toolbar .....	277
Click On Fault Shot Tab.....	281
Click On Fault Search Tab.....	284
Click On Fault Settings Tab .....	287
Click On Fault Zones Tab .....	289
Click On Fault Control Examples .....	292
Contact Control .....	296
Contact Control.....	296
Contact Control Properties.....	297
Contact Control Links.....	299
Contact Control Example .....	300
Digital Configuration Control .....	304
Digital Configuration Control .....	304
Digital Configuration Control Outputs.....	306
Digital Configuration Control Inputs .....	307
Digital Graphic Control .....	308
Digital Graphic Control.....	308
Digital Graphic Control Traces .....	310
Digital Graphic Control Overlays.....	311
Digital Graphic Control Properties.....	312
Digital Sequence Control.....	313

Digital Sequence Control .....	313
Digital Sequence Tab.....	315
Digital Default Output States Tab.....	317
Digital Sequence Plot Tab.....	318
Dynamic Control.....	319
Dynamic Control .....	319
Dynamic Control Properties .....	320
Dynamic Control Graph .....	322
Dynamic Control Inputs.....	323
Dynamic Control Outputs.....	325
Dynamic Control Prefault/Fault Info .....	326
Dynamic Control Configuration .....	327
Dynamic Control Sequence .....	330
Dynamic Control Example 1.....	334
Dynamic Control Example 2.....	341
END Control.....	345
GET Control.....	347
Graphic Control .....	348
Graphic Control.....	348
Graphic Control Scale Type .....	350
Graphic Control Traces.....	351
Graphic Control Group Test Overlays .....	352
Graphic Control Graph Properties.....	354
Graphic Control Report Options.....	355
Horn Control.....	356
HW Timer Control.....	357
HW Timer Control .....	357
HW Timer Properties .....	358
HW Timer Links .....	360
Link Control.....	361
List Control.....	363
Log Control .....	365
Loop Control .....	367
Message Box Control.....	369
Node Control.....	375
Output Contact Control .....	376
PC Timer Control .....	377
Ramp Control .....	380
Ramp Control.....	380
Ramp Control Properties.....	381
Ramp Control Links .....	383
Select Control .....	384



Table Of Contents

Start Control.....	385
Status Bar Control.....	387
SubTest Control .....	388
SubTest Control.....	388
SubTest Control Properties.....	389
New SubTest.....	391
New SubTest Example .....	393
Text Control .....	396
TCM-915S Control.....	397
Theoretical Control.....	399
Theoretical Control.....	399
Theoretical - Circle.....	402
Theo Circle - Max Torque .....	404
Theo Line - 2 Point.....	412
Theo Line Point/Slope.....	414
Theo Ellipse .....	417
Theo Lens.....	419
Theo Tomato.....	422
Theo Data - Polar Impedance .....	424
Theo Data - Impedance .....	426
Theo Data - Time-Amplitude .....	428
Theo Data - Generic .....	430
Theo Calculations .....	431
Theo Theoretical Object - Impedance.....	432
Theo Theoretical Object - Impedance.....	432
Theo Impedance Toolbar & Menu.....	433
Theo Quadrilateral Example .....	438
Theo Quadrilateral Impedance Construction.....	439
Theo Quadrilateral Impedance Calculations .....	442
Theo Quadrilateral Impedance Example.....	443
Theo Load Encroachment Example.....	445
SEL Load Encroachment Construction .....	446
SEL Load Encroachment Calculations.....	452
SEL Load Encroachment Example .....	458
Theo Theoretical Object - Time-Amplitude .....	460
Theo Theoretical Object - Time-Amplitude.....	460
Theo Object Time-Amplitude .....	462
Theo Object Time-Amplitude Example .....	464
Theo Object Time-Amplitude Auto Trace.....	468
Theo IEEE/ANSI Overcurrent Curve.....	470
Theo IEC Overcurrent Curve .....	472
Theo RIO .....	474

Theo RIO .....	474
Theo RIO Import .....	475
Theo RIO Example Test .....	480
Vector Control .....	485
Waveform Control .....	487
XMAC Control .....	489
Monitor Editor Tab .....	490
Monitor Editor Tab .....	490
Monitor Editor Tab Drop Menu .....	491
Monitor Editor Toolbar .....	493
Monitor Editor Toolbar .....	493
Select Tool .....	494
Line Tool .....	495
Rectangle Tool .....	496
Text Tool .....	497
PULSAR Tool .....	499
Horizontal PULSAR Tool .....	501
PULSAR Channel Tool .....	503
Timer Module Tool .....	505
Vertical Bar Tool .....	507
Horizontal Bar Tool .....	511
Text Box Tool .....	514
Value Tool .....	515
Connections Editor Tab .....	516
Connections Editor Tab .....	516
Connections Editor Tab Menu .....	517
Connections Editor Toolbar .....	518
Connections Editor Toolbar .....	518
Connections Editor Example .....	519
Select .....	521
Link .....	522
Polyline .....	523
Line .....	524
Rectangle .....	525
Text .....	526
Images .....	527
Node .....	528
Terminal .....	529
MPRT Front .....	530
MPRT Back .....	531
PULSAR Front .....	532
PULSAR Back .....	533

## Table Of Contents

TCM-915S Front .....	534
TCM-915S Back.....	535
GPS-6500 Front.....	536
GPS-6500 Back .....	537
MGTR Front.....	538
MGTR Back .....	539
EPOCH II Front.....	540
EPOCH II Back .....	541
EPOCH 20 Front.....	542
EPOCH 20 Back .....	543
BASIC Tab .....	544
Relay General Information Tab .....	545
Online Controls .....	547
Online Controls .....	547
Online Vector Control.....	548
Online Vector Control.....	548
Vector Polar Diagram.....	550
Vector Voltages and Currents .....	553
Vector List.....	554
Vector VCR Controls.....	555
Vector Input and Output.....	557
Boolean Equation.....	559
Online Ramp Control.....	560
Online Ramp Control... ..	560
Ramp Polar Diagram .....	562
Ramp Voltages and Currents.....	565
Ramp Input and Output.....	566
Ramp VCR Controls .....	568
Online Click On Fault Control .....	569
Online Click On Fault Control.....	569
CoF Menu Toolbar .....	571
CoF Graph View .....	575
CoF Shot Tab.....	577
CoF Search Tab.....	580
CoF Settings Tab .....	583
CoF Zones Tab.....	585
CoF Results Tab .....	588
CoF Example .....	591
Wizards .....	597
Wizards .....	597
New Device Wizard .....	598
New Device Wizard.....	598

New Device.....	599
New Device Settings.....	603
New Device General Information .....	606
New Relay Wizard.....	608
New Relay Wizard .....	608
New Relay .....	609
New Relay Manual Test.....	612
New Test Wizard.....	613
New Test Wizard.....	613
New Test Copy .....	614
New Test Manual .....	615
New Test Generic .....	617
How to Create Test Procedure Example.....	621
How to Create Test Procedure Example.....	621
Pickup Test.....	622
4 Example Tests .....	625
4 Example Tests .....	625
Test Editor OC Time Example 1 .....	626
Test Editor OC Time Example 1 .....	626
Start Control - Example 1.....	628
Calculation Control - Example 1.....	629
Theoretical Control - Example 1.....	630
Graphic Control - Example 1 .....	632
List Control - Example 1.....	633
Calculation Control 2 - Example 1.....	634
Vector Control - Example 1 .....	635
HW Timer Control - Example 1 .....	636
Log Control - Example 1 .....	637
PC Timer Control - Example 1 .....	638
End Control - Example 1 .....	639
Test Results - Example 1 .....	640
Test Editor Charc Z Example 2 .....	641
Test Editor Charc Z Example 2.....	641
Calculation Control - Example 2.....	643
Theoretical Control - Example 2.....	644
1-Touch Control - Example 2 .....	645
Loop Control - Example 2 .....	648
Third Calculation Control - Example 2 .....	649
Ramp Control - Example 2.....	650
Vector Control - Example 2.....	652
Message Box Control - Example 2.....	653
Test Results - Example 2.....	654

## Table Of Contents

Test Editor Microprocessor Time OC Example 3 .....	655
Test Editor Differential Example 4.....	658
Waveform Digitizing Tool.....	663
Waveform Digitizing Tool.....	663
Waveform Digitizing Tool Menu and Toolbar.....	664
Waveform Digitizing Tool.....	667
DFR Waveform Viewer.....	673
DFR Waveform Viewer.....	673
DfrWaveView Menu and Toolbar.....	674
Digital Fault Recorder Testing.....	675
Digital Fault Recorder Testing.....	675
DFRecorder New Convert Test.....	676
COMTRADE To Hexidecimal Conversion.....	680
DfrWaveView.....	682
Appendix A - Basic Language Syntax.....	685
Appendix A - Basic Language Syntax.....	685
Variables.....	686
Expressions.....	691
Appendix B - MPRT Command Set.....	693
Appendix B - MPRT Command Set.....	693
Command Set Constant Definitions.....	694
Command Set Variable Definitions.....	700
Command Set Procedures.....	702
Command Set Procedures.....	702
Generator Select Commands.....	703
Generator Control Commands.....	704
Generator Waveform Commands.....	710
Timer Control Commands.....	717
Binary Input Commands.....	728
Binary Output Commands.....	731
Binary Trigger commands.....	733
Load Error Action Commands.....	735
Synchronization Commands.....	736
Wait Commands.....	738
Restart Commands.....	741
Query Commands.....	742
Query Device Commands.....	747
Query Time Tag Commands.....	750
Query Measurement Value Commands.....	752
Horn Commands.....	757
Directory and File Commands.....	758
Battery Simulator Commands.....	762

Vector Scope Configuration Commands .....	764
Command Set Procedures - New .....	765
Command Set Procedures - New .....	765
Set Wave Index.....	766
Sine Sum .....	767
Point on Wave.....	768
Sweep Frequency .....	769
Set Record ON/OFF .....	770
Set Record Rate .....	771
Record Capture Data .....	772
Record Capture Info.....	773
Record Reset .....	774
Wait for Any Binary Input .....	775
Wait for All Binary Input .....	776
Timer Set Output.....	777
Get Record Data (TIMER).....	778
Wait Phase (TIMER) .....	779
Set Debounce Mode (TIMER).....	780
Start Virtual Timer (TIMER).....	781
Stop Virtual Timer (TIMER).....	782
Query Virtual Timer (TIMER) .....	783
Interface Procedures .....	784
Interface Procedures.....	784
Send Command.....	785
Selecting VI Generators.....	787
Execute Selected Voltage Generator Command .....	790
Execute Selected Current Generator Command.....	791
Execute Timer Command .....	792
Compute and Generate Waveform or DFR Physical Address .....	793
Store DFR Binary Data .....	794
Output Off State Three.....	795
Generator Type Select.....	797
Read Timer Port Command and Data.....	798
Read VI Generator Port Command and Data .....	799
Read Command.....	800
Determine Number VI Generators Present .....	802
Get Timer Query Information .....	804
Get VI Generators Query Information .....	805
MPRT Appendix A - Timer Flash Memory Parameter Sector Physical Layout.....	807
MPRT Appendix B - VI-Generator Flash Memory Parameter Sector Physical Layout.....	808
MPRT Appendix C - VI-Generator Correction factor Flash Sector Physical Layout.....	809
Appendix C - PULSAR General Information .....	811

## Table Of Contents

Appendix C - PULSAR General Information .....	811
PULSAR Error Codes.....	812
PULSAR Error Codes .....	812
PULSAR Error Message Format.....	813
Section 1 - Generator Selection Section .....	814
Section 2 - Generator Controller Section .....	815
Section 3 - Generator Waveform Section.....	816
Section 4 - Timer Section.....	818
Section 5 - General Errors .....	819
Section 6 - Communication Section .....	822
Section 7 - Wait Command Errors .....	824
Section 8 - Macro Errors .....	825
Section 9 - Macro Errors .....	826
PULSAR Command Set.....	827
PULSAR Command Set.....	827
Generator Selection Command.....	828
Generator Control Command.....	829
Generator Waveform Command .....	832
Timer/Monitor Control Command.....	835
Load Error Action Command.....	838
Synchronization Command.....	839
Wait Command .....	841
Unit Reset Command.....	843
Local Mode Command.....	844
Restore Phase Command.....	845
Query Command.....	846
Macro Command .....	851
Xmac Commands .....	852
PULSAR String Commands .....	853
PULSAR String Commands.....	853
Output String Command .....	855
Circle Command .....	856
Clear Command.....	857
ClS - Clear Command .....	858
DFRLoad Command .....	859
Draw Command.....	861
Else Command .....	862
Else If Command .....	863
End Command.....	864
End If Command.....	865
Enter_at Command.....	866
For Command .....	868

Get_Time Command.....	869
GoTo Command .....	870
GoSub Command .....	871
Graph Command .....	872
If Command .....	874
Include Command.....	876
Label Command.....	877
Local Command.....	878
Locate Command.....	879
Monitor Command .....	880
Move Command.....	881
Msg_at - Message At Command.....	882
Next Command .....	883
Off Command.....	884
On Command.....	885
Print Using Command.....	886
PulseMonitor Recorder Macro Command.....	887
PulseMonitor Sequencer Macro Command .....	889
Ramp Command.....	891
ReadUnit Command .....	892
Remote Command.....	893
Reset Command .....	894
Return Command.....	895
Save Command .....	896
Set Command .....	897
SetHdrLn - Set Header Line Command .....	898
SIPULSAR - Select Command.....	899
SITCM915 - Select Command .....	900
Solve Command .....	901
Step Command.....	902
Timer Reset Command.....	903
Trigger Command.....	904
TRGetTag - Time Reference Get Tag.....	905
TRTrig - Time Reference Trigger .....	906
Unit Command .....	908
Wait Command Syntax .....	909
Win_size Command.....	910
WinExec Command .....	911
XmacLoad Command .....	912
XmacUnLd Command.....	913
XMACs .....	914
XMACs.....	914



## Table Of Contents

XMAC Library Functions .....	915
XMAC Library Functions .....	915
XMAC Parser .....	916
Communications Redirection .....	917
Receive .....	918
Output .....	919
Error Message Display.....	920
Gate Status .....	921
Synchronization Reset .....	922
Wait for Synchronization .....	923
Sound PULSAR's Horn .....	924
No System Echo of Messages .....	925
XMAC Read Messages.....	926
Command Syntax Diagrams .....	927
Creating XMACS.....	929
Testing an XMAC Program .....	930
Downloading a PULSAR XMAC.....	931
XMAC Demo Program .....	932
Appendix D - IEEE Codes.....	935
Appendix D - IEEE Codes.....	935
ANSI/IEEE Device Numbers .....	936
IEEE Devices Performing More Than One Function.....	941
IEEE Suffix Numbers .....	942
IEEE Suffix Letters .....	943
IEEE Auxiliary Devices .....	944
IEEE Actuating Quantities.....	945
IEEE Main Device Parts.....	947
IEEE Standard Reference Positions .....	948
IEEE Other Switches .....	950
IEEE Representation of Devices on Diagrams.....	951
Appendix E - AVTS & MPRT Information & Web Links.....	953
Appendix E - AVTS & MPRT Information & Web Links.....	953
AVTS Information.....	954
MPRT Information.....	955
Index .....	957



## Welcome to AVTS Help

This AVTS Help accompanies the latest release of AVTS and MPRT new firmware.

Welcome to the **Megger AVTS** (Advanced Visual Test Software) relay test software. **AVTS** is a friendly user interface for to control the **AVO PULSAR** and the **Megger MPRT** relay test equipment.

AVTS is the industry standard for creating and using modules to test old and modern electromechanical, solid-state and microprocessor relays. In companion to the MPRT, there is virtually no power system protective relay that cannot be tested by the relay technician using these tools. It is ideally suited for those special relays that require test procedures not normal to the older electromechanical or solid-state relay test procedures.

AVTS simplifies the process by which the user organizes the test modules for ease to view and select for any installed relay in the power system.

AVTS contains a variety of Reports that virtually satisfies all the requirements for creating a report of the last and/or previous relay tests. In addition, the user may select from a listing of several categories of reports to generate for record purposes.

The user may select from specific relay test modules from a database provided with the purchased copy of AVTS. In addition to those provided during the initial installation, there are a number of relay test modules available to purchase from Megger.

With only a few moments of exposure to the basics of AVTS, the user is able to create from a Test Wizard a single test or a group of tests that will satisfy the authenticity of the relay settings for any relay in the protective systems.

If a beginning AVTS user, go to the Introduction to AVTS for initial explanation in using AVTS. Examine the four topics in this book for orientation to the Help and AVTS.

If an experienced AVTS user, go to the AVTS Quick Links to Topics to locate normally used work areas.

For more information about additional relay test modules that may be available for new relays, contact the Megger sales representative for your company, or contact the Megger offices directly:

Megger

4271 Bronze Way

Dallas, Texas 75237-1008 USA

Tel: 1.800.723.2861 Ext. 3519

AVTS Help Ver 3.0 -- August 2007



## **AVTS Quick Links to Topics**

**AVTS Help** defines and details Menu Items, Toolbars and Tools for use to create modules for testing relays. The listed topics are typically the most used topics and areas of use in AVTS.

### **Introduction to AVTS**

- AVTS General Theory of Operation
- How to Create Test Procedure
- New Device Wizard
- New Relay Wizard
- New Test Wizard
- Time Overcurrent Test Example 1
- Characteristic Impedance Test Example 2
- Microprocessor Time OC Test Example 3
- Differential Test Example 4

### **Toolbars**

- Main Toolbar
- Relay Toolbar

### **Tools**

- Online Vector Control...
- Online Ramp Control...
- Online Click On Fault Control... - A new feature - Import RIO Characteristics for Tests
- SS1 Converter... - A new feature
- Import Protest Results Database - A new feature

### **Workbook Pane**

- Devices Settings tab
- Relays Settings tab
- Relay Test tab
- Relay Capture tab - A new feature
- Test Editor tab
  - Test Editor Controls Toolbar
    - 1-Touch Modbus - A new feature
    - 1-Touch SS1 - A new feature
    - Binary Search Control - A new feature
    - Click On Fault Control - A new feature - Import RIO Characteristics for Tests
- Monitor Editor tab
  - Monitor Editor Toolbar
- Connections Editor tab
- BASIC tab

AVTS Help 092607

**Relay Menu Tools**

Import Data

Export Data

# Introduction to AVTS

## Introduction to AVTS

### Key Features

AVTS is constructed with three versions of user capability:

- **Basic** is the version supplied with the purchase of a MPRT test set. The user will not be able to access the Test Editor Controls to construct new or revise existing tests. The New Test, New Relay and New Device Wizards are available to construct generic tests. Tests, Relays and Devices may be imported and exported. Database activities are also enabled.
- **Advanced** version will include all the capabilities of the Basic version and enable all the Controls and Toolbars in the Test Editor, Monitor Editor and Connection Editor tabs of an open Relay.
- **Professional** version will include all the capabilities of the Basic and Advanced versions and enables the Waveform Digitizing Tool and the DFR Waveform Viewer for use in performing playback of fault record COMTRADE files.

Two screen tiles to display:

- The Workspace pane for product navigation
- The Workbook pane for operational construction and use

Open a specific relay test procedure by use of Barcode.

Build special tests using Icon objects referred to as Controls.

Use **Online Controls** for real time operation:

- Online Vector Control...
- Online Ramp Control...
- Online Click-on-Fault Control
- Fault Calculator...

Waveform Digitizing Tool

DFR Waveform Viewer

How To ...

AVTS Universal Test Software provides two distinct areas of control upon initial startup of the program. AVTS Workspace pane displays a navigation-friendly split-screen where the table of contents, index, and search tabs remain visible on the left pane while the Workbook pane with tests content displays on the right.

The left pane displays the AVTS Explorer comprised of workbooks for Devices, Relays and Tests. The right pane displays the actual device or relay tests, displays, connections and notes.

Explore the TOC (Table of Contents) to the left of this Help. The organization of the TOC has been created with specific books and sub-books and topics.

**Note:** The AVTS installation disk contains a complete Help folder for Script programming and format. Browse the disk and open the VBScript folder. The folder contains the scrdoc56en.exe file. Double left click on the file to install the file to the default C:\Program Files\Microsoft Windows Script\ScriptDocs folder automatically. Double click on the Help file Script56.CHM to open and use.

## AVTS Installation Disk

The Installation disk for AVTS will open with a dialog to allow the options to Install, Browse or Exit the installation process.

**Install** option will execute the complete Install of the AVTS version resident to the disk.

**Browse** option will open in Windows Explore for view the installation files and information folders, present on the disk.

**Application Notes** folder contains several word documents individually compacted with WinZip. The documents are Application Notes and Guides detailing specific areas of use of AVTS. One very important document is the file labeled "**Application070 - End to End Step by Step**".

**PulseMaster Modules** folder contains files of individual relay test modules. The relay modules provided with PulseMaster Ver. 2.0 have been individually converted to an AVTS database and compacted using WinZip. The extraction of the compacted file will produce the "*relayname*".*mdb* file and all the Macro files for that specific relay.

The individual relay may be placed in an AVTS database using the Tools/Import Data/PulseMaster Database... option. The Macro files from the extracted zip file should be placed in the AVTS/Macros folder.

**VBScript** folder contains the file *scrdoc56en.exe*, a self executable WinZip file. Double click on the file and select a folder for auto extraction to a folder of choice. The extraction will produce the file *Script56.CHM* which is the Help program for using **Visual Basic Script**. The use of the Script will accommodate the need to communicate to a relay to make settings changes if necessary, using the 1-Touch Control and the BASIC Control and BASIC Tab.

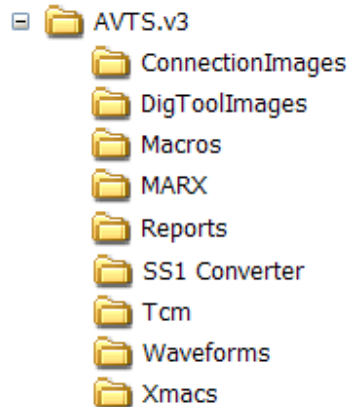
**Exit** option will exit the disk introductory dialog and CD drive.



## AVTS Directory Structure

The installation of AVTS software creates nine (9) sub-directories under the root directory **C:\Program Files\Megger\AVTS**. The path is user determined during AVTS installation.

The *AVTS Directory Structure* installed folders are:



**ConnectionImages** – The directory for images to be used in the Connections Editor. The images may be associated with the 10 controls in the Connections Images Tool Bar located in the upper right Connections Editor Screen. To associate an image with one of the controls, choose Tools/Connections Images... menu item and make the associations desired. The Connections Images dialog may also be opened by clicking on the Connection Images right most button of the Connection Images toolbar. Any image may be used if it is in a bitmap format (<filename>.bmp).

**DigToolImages** – The directory for images to be used as Background Images in the Waveform Digitizing Tool. Typically the images will be a relay's characteristic curve as shown in the manufacturer's instructions for the relay. Any characteristic image may be used if it is in a bitmap format (<filename>.bmp).

**Macros** – The directory in which all basic code accessible to the user is stored. PulseMaster Test Files that are imported into AVTS are located in this directory. Any Basic Control created within AVTS is stored in this directory. Each Basic Control must have a unique name if there are differences in the basic code. The naming structure is <filename>.abm.

**MARX** -- The directory is for the files necessary to enable the MARX parallel or USB dongles to be used with the AVTS Advanced and AVTS Professional versions.

**Reports** – The directory containing all of the reports available in AVTS. Standard reports are located in this directory. If custom reports are developed, they must reside in the Reports Directory. The naming structure is <filename>.rpt.

**SS1 Converter** -- The directory containing all the SS1 (\*.ss1) files by importing State-Sequence Files (SS1) from ASPEN and CAPE power system simulation software programs. The converted files will have the extension \*.m1t.

**TCM** -- The directory containing the sequence (\*.seq) and record (\*.rcd) files used in the TCM Controls. The sequence files are generated using the PulseMonitor software and stored in this directory. The record files contain the data in the AVTS reports. These record files may also be opened using PulseMonitor software.

**Waveforms** – The directory containing the waveforms derived from the COMTRADE Standard File format. Any files converted by the DFR Waveform Viewer must be stored in this directory if they are to be used by a WaveForm Control in an AVTS Test.

**XMACS** -- Xmac files to be downloaded to the Pulsar Universal Test System using the Xmac Control must reside in this directory. All Xmac files are executable files and must be of the file structure <filename>.exe.

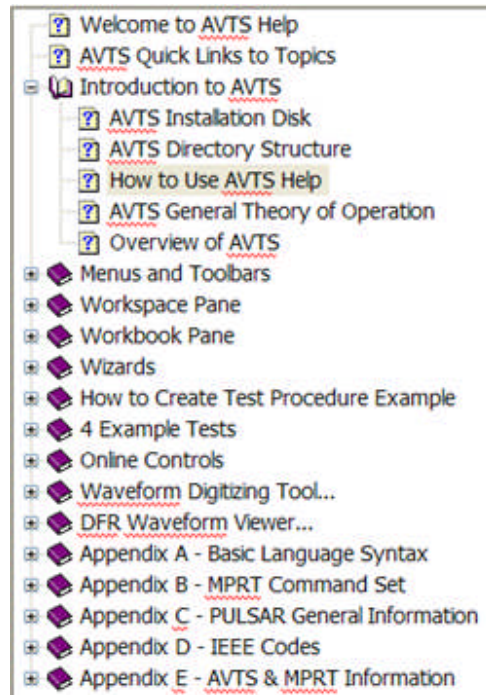
The following folders may be created to segregate other files:

**Click On Fault Tests** - The directory may be created by the user for the particular tests previously created and saved using the Click On Fault online control. The tests are then transferable to other databases and computers by file name.

**html** -- This directory is created when an AVTS test report has been saved using the **Export Format html** in the open test result Workbook pane.

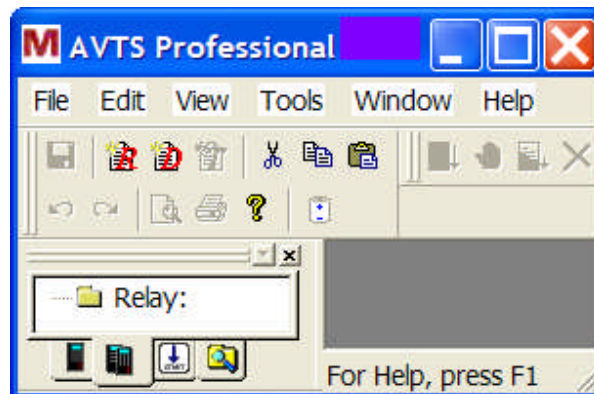
## How to Use AVTS Help

The *Help* topics are arranged in a logical order that will assist the user in locating information to work with AVTS.



The Main parts are the "books" (subjects) with "topics" (book pages) of explanation.

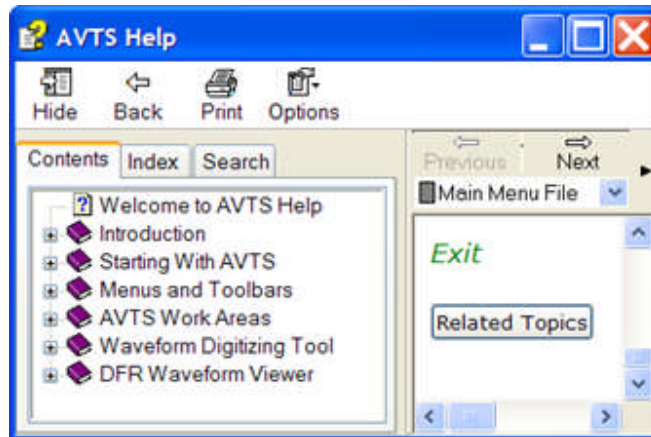
Topics may have hidden detail areas to further explain the given subject. For demonstration, Green italic text will indicate the location of hidden context. For access to the additional detail to the topic, the user should single left mouse **click on the green text to expand a field for additional detail** to open either a **field of teal text** or an *image*.



These expanding text or drop boxes for text and images are designated as **green text not-underlined**.

The expanded area will remain in view until the green italic text first clicked is clicked once more. All expanded fields will remain in view until they are closed one by one or by exiting the topic to another.

Additional information on a topic may be found in another topic set specifically to detail the subject. A left mouse click on blue underlined text, called a *Hyperlink*, will transport the Help screen to the other topic. To return to the originating topic, left mouse click on the Help Menu tool *"Back"*.

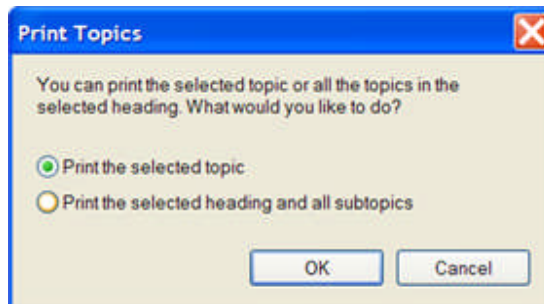


A Hyperlink will be designated by underlined **blue** text.

An example of "other topic" detail. A *Hyperlink* [This is an example of an embedded Hyperlink](#). may also be present in an expanded text box.

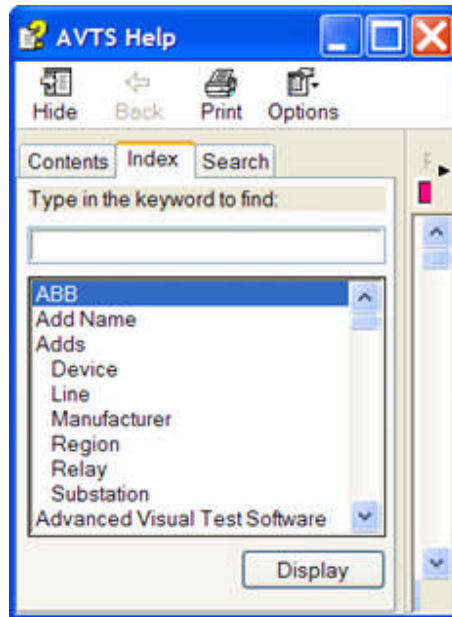
**Note:** Notes will be in this format and highlighted in YELLOW to draw attention to operations of most importance.

To print a topic directly from the Help, select the Print icon on the menu bar. The **Print Topics** dialog will appear for options to "Print the selected topic" or to "Print the selected heading and all subtopics". The second option will print "Book" and all subtopics, sub-books and their subtopics.

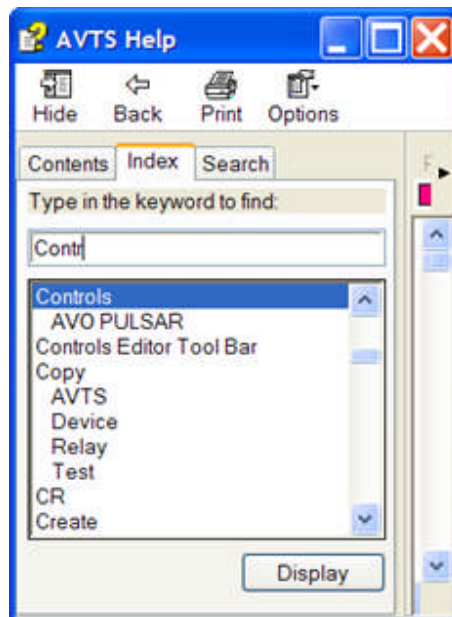


**Note:** The printout of a selected topic or heading with subtopics will NOT print embedded expanded text or drop down images. The user will need to refer to the Word document and print the desired selection to include the embedded functions.

A tab at the top of the Table of Contents is for displaying the Help *Index*.

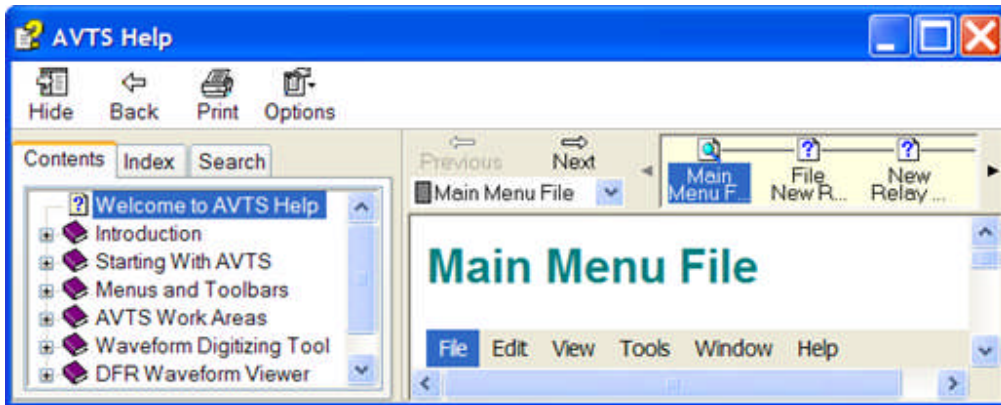


The Index has all the keywords in alphabet order and initially displayed beginning with the first in the list. Type a letter in the insert field to move the Index to the first keyword beginning with that letter. To further refine the search, additional letters may be *typed*.



A function to display an area for locating specific text and subjects, select the tab Help Search. Type a word or phrase in the edit field and to list the related topics, select Display.

The general browse sequence will begin with the Book title and continue through all the topics in that book. There may be a browse sequence set that will link topics within several books in order to explain an AVTS construct or operation procedure in step-by-step format. When available, the top of the topic pane will display a *Browse sequence*.



This is an Example of the jump of a Hyperlink from the previous topic as a result of clicking on the Hyperlink. Go Back.

## AVTS General Theory of Operation

AVTS has been created to give the user a package of tools to utilize specific test procedures or create instruction sets for test equipment. The user is presented with a package utilizing GUI (graphic user's interface) to create tests, active displays during test execution and equipment connection details. This approach allows a user to quickly begin using AVTS and working with the software to perform tests.

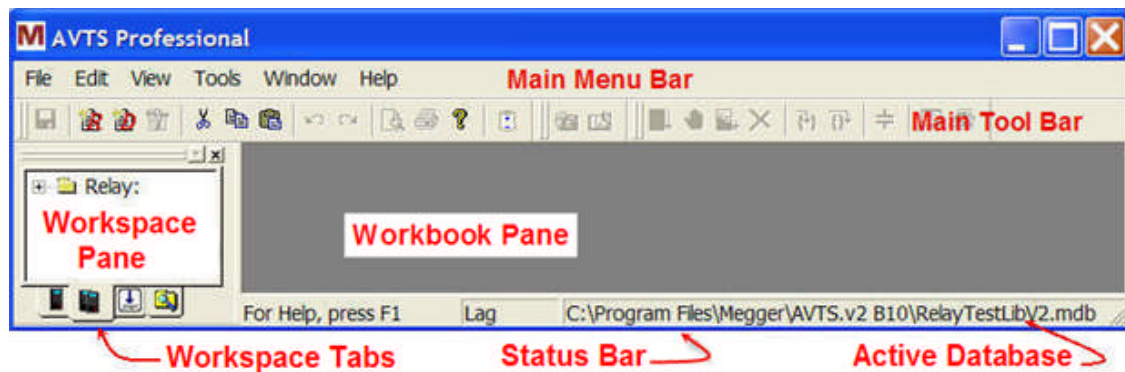
Whereas, with older software, the user was required to have a working knowledge of a programming language in order to create special test procedures, AVTS has been built with tools that replace the need for this special knowledge.

The Main Menu and Toolbars are for selecting the necessary action desired.

The Status Bar is to display the path for the active database and any other items place during a test execution using the Status Bar Control.

The AVTS screen is divided into two panes. A left area called the Workspace Pane and a right area called the Workbook Pane.

The Workspace Pane is to display the Devices, Relays, Tests and MPRT tabs. From these tabs select to open the items from the list trees in each tab. Any item selected and opened will appear in the Workbook Pane.



The Workbook Pane is to display, when selected and opened, Devices, Relays and Tests and MPRT Results and Settings. The Workbook pane is initially blank when AVTS is opened for a session. This pane becomes detailed when the user selects one of the Device or Relay objects in the Workspace. Select a Relay and double-click on it to open and the properties of the test for that relay will display with a number of tabs containing details for that relay.

The Workbook tabs are for:

- Listing the Device Settings for the relay parent device or template,
- Displaying a General Information area for the device,
- Listing the Relay Settings for the relay under test,
- Executing a Test to be viewed in the Test tab.
- Displaying a Test Editor for creating a procedure,
- Displaying a Capture for viewing active tests and exporting completed events to Word.
- Displaying a Monitor Editor for displaying active parameters during a test execution,
- Displaying a Connections Editor for creating graphic displays of equipment connections,
- Displaying the BASIC tab to insert BASIC language, if desired, to provide procedure control, and
- Displaying a General Information area for the relay.

To begin testing relays, expand the list of relays in the Relays tab in the Workspace (left pane). Double click on a specific relay from the list. AVTS will open that relay test module with the Workbook (right pane) Settings tab displayed for accepting the default relay settings or editing them to duplicate the relay settings.

Make setting changes and then click the Execute Test icon on the Relay tool bar. Other methods to execute a test are to right click in the Workspace to drop down a menu containing the Execute command, click on the Tools/Execute menu option, or press F5 on the keyboard. With a test set connected to the computer and the relay, the proper signals are injected to the relay and AVTS will show a mimic of the activity. A list of the relay responses to the test will be listed as test results for documentation.

To build a special test, a list of tools, called Controls in the Test Editor tab, having specific properties that replace program language are displayed on a tool bar. The user simply selects a combination of the controls and links them together to form a procedure from Start to End.



## Overview of AVTS

The first time AVTS is opened, the Main AVTS dialog will be displayed, as shown below. Within this dialog are the working portions of the software.

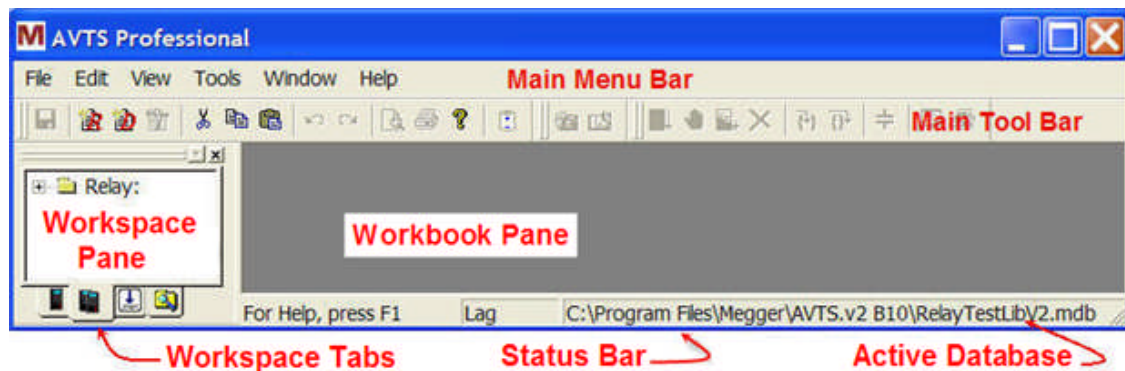
The major portions of use are labeled in red:

Main Menu Bar

Main Menu and Toolbars

Workspace Pane

Workbook Pane



### General

The opening dialog for AVTS displays the two main areas of the program. The pane on the left side is the **Workspace Pane** or the explorer area (similar to Windows Explorer).

The pane on the right hand side is the **Workbook Pane** area (similar to a Microsoft Excel spreadsheet). It is the area that a Device, Relay or Test will be opened to edit or execute a test to a relay. The opened Device, Relay or Test will have several tabs at the bottom of the Workbook to select areas for relay settings, test construction or edit and test operating parameters during execution.

The **Workspace Pane** has three tabs at the bottom of the pane labeled Devices, Relays, and Tests. These same tabs have menu tool icons labeled **R**, **D** and **T**, which call the AVTS New Relay Wizard, the New Device Wizard and the New Test Wizard respectively.

**Note:** The **T** menu tool icon will appear dimmed and not be available to evoke the New Test Wizard until a Relay has been opened.

The **Workbook Pane** has either two or seven tabs at the bottom depending upon whether a Device is open or a Relay is open with Test(s).

When AVTS is initially installed and then opened for use, the Device and Relay tabs are shown as empty folders. The "megger.mdb" database is one of five available databases and is immediately applied when AVTS is opened for the first time. It is suggested that the first time user click on File, Change Database..., Examples.mdb or RelayTestLbV2.mdb. These databases have example Devices, Relays and Tests for the relays..

Care must be taken when changing databases. Five databases are installed with AVTS:

**Megger.mdb** – an empty database that is default upon installation of AVTS. This database is immediately available for creation of Devices, Relays and Tests.

**Blank.mdb** – an unusable database that is the template from which all databases are created, making them all compatible. This database will not be called by AVTS for use.

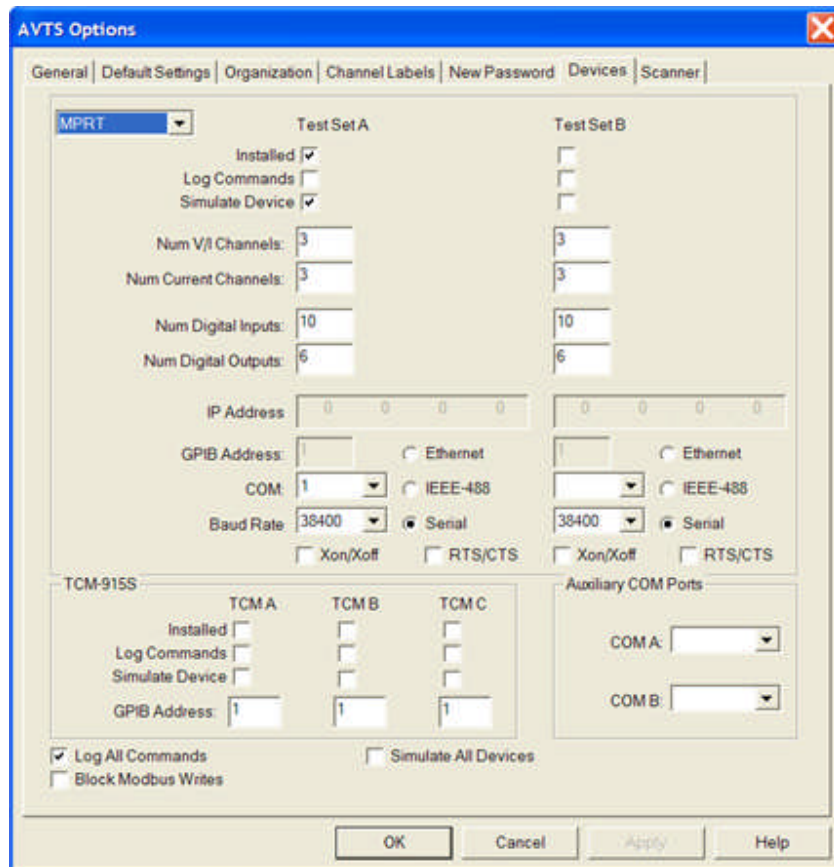
**Default.mdb** – a database that contains all the generic relay tests and copied by the Relay Wizard into the database in use.

*Note: Extreme care must be exercised. This database may be called by AVTS. Should any of the relays and tests be altered in this database, the generic tests copied by the Relay Wizard may not operate correctly.*

**Examples.mdb** – an example database that contains the default Organization structure and a group of relays for which tests have been created. This group of relays is the original group supplied with the first issues of AVTS. The MPRT Results group has been added that reflects several test results reports captured from a database resident to the MPRT test set.

**RelayTestLbv2.mdb** – an example database that contains the default Organization structure and relay modules in several groups listed by relay manufacturer. These relay modules are in addition to those found in the Examples.mdb database.

Prior to exploring the AVTS relay test software, it is suggested that the user also click on the Tools/Options menu item and click on the *Devices tab* to change some of the default installed items.



First uncheck the boxes for Log All Commands and Simulate All Devices.

Then check the boxes for Installed and Simulate Device under MPRT or Pulsar.

Additionally, note that the Pulsar configuration is for three current and three voltage channels. Should the Pulsar for use have a different configuration of current and voltage channels, the correct number of channels should be edited for each. AVTS queries the hardware in the un-simulate mode and if there is a difference in the software configuration compared to the query, then the Pulsar will be unrecognized by AVTS.

## Menus and Toolbars

### Menus and Toolbars

Three major programs for use in AVTS have menus and associated toolbars:

The primary program is AVTS which is used for the usual discrete relay testing with the resident relay test modules. The associated menus and toolbars are described in the links listed below.

The AVTS Main Menu and Toolbars group of topics will have list items that are common to the Relay Menu and Device Menu. Those common items are described in the AVTS Menu and Toolbars group of topics only. The menu items specific to the relay and device menus are described in those respective topics.

Main Menu and Toolbars group of topics. The topic defines the available Standard Menu items. The standard menu items are shared by the other menu items evoked by opening Devices and Relays Workbook panels.

Device Menu and Toolbars group of topics. The topic defines the available Device Menu items that are in addition to the topics in the Standard Menu that are in addition to the topics in the Standard Menu.

Relay Menu and Toolbars group of topics. The topic defines the available Relay Menu items that are in addition to the topics in the Standard Menu.

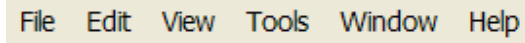
Another AVTS program is the Waveform Digitizing Tool. This program is for the user to import a scanned waveform and digitize it for use as a COMTRADE file to test a relay or a complete system with the DFR Waveform Viewer program. The program tools are described in Waveform Digitizing Tool Menu and Toolbar.

The third AVTS program is the DFR Waveform Viewer. This program is for the user to import COMTRADE files for display, editing and loading into test equipment for playback with the connected MPRT or Pulsar relay test systems. The program tools are described in DFR Waveform Viewer Menu and Toolbar.

## Main Menu and Toolbars

### Main Menu and Toolbars

The *Main Menu* bar is displayed with the opening of AVTS.



Detailed information for each menu item and the drop down list of options for each is available in the following topics: Main Menu File, Main Menu Edit, Main Menu View, Main Menu Tools, Main Menu Window, Main Menu Help.

The *Standard Toolbar* is displayed with the opening of AVTS and is available for use.



Detailed information for each tool is available in Main Menu Toolbar topic.

The **AVTS Device Menu** bar is displayed when a Device has been opened with the Device module properties displayed in the Workbook pane.

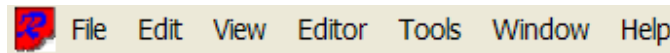
Detailed information for each menu item and the drop down list of options for each is available in the following topics: **Device Menu File**, **Device Menu Edit**, **Device Menu View**, **Device Menu Tools**.

The *Device Toolbar* is displayed with the opening of AVTS and is available for use.



This toolbar is active when a Device is open in the Workspace Devices tab. It is shown in the active state, and detailed information for each tool is available in Device Toolbar topic.

The **AVTS Relay Menu** bar is displayed when a Relay has been opened with the relay module properties displayed in the Workbook pane.



Detailed information for each menu item and the drop down list of options for each is available in the following topics: Relay Menu File, Relay Menu Edit, Relay Menu View, Relay Menu Editor, Relay Menu Tools, Relay Menu Window.

The *Relay Toolbar* is displayed with the opening of AVTS and is available for use.



This toolbar is active when a Relay is open in the Workspace Devices tab. It is shown in an active state, and detailed information for each tool is available in Relay Toolbar topic.

The *Connection Images Toolbar* is displayed when a Relay is open in the Workbook Tests tab and with the Workbook Connections Editor tabs selected. It is shown in the active state, and detailed information for its use is available in Connection Images Toolbar topic.



The *Align Toolbar* is displayed when a Relay is open in the Workbook Tests tab and with the Workbook Test Editor, Monitor Editor or Connections Editor tab selected. It is shown in the active state when one or more objects is selected in the displayed tab, and detailed information for its use is available in Editor Align Toolbar topic.



The *Nudge Toolbar* is displayed when a Relay is open in the Workbook Tests tab and with the Workbook Test Editor, Monitor Editor or Connections Editor tab selected. It is shown in the active

state when one or more objects is selected in the displayed tab, and detailed information for its use is available in Editor Nudge Toolbar topic.



The *Structure Toolbar* is displayed when a Relay is open in the Workbook Tests tab and with the Workbook Test Editor, Monitor Editor or Connections Editor tab selected. It is shown in the active state when one or more objects is selected in the displayed tab, and detailed information for its use is available in Editor Structure Toolbar topic.

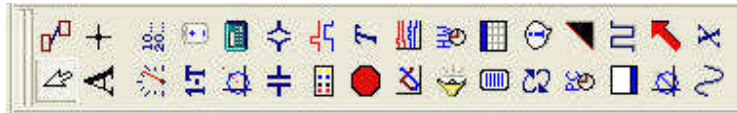


The *Zoom/Pan Toolbar* is displayed when a Relay is open in the Workbook Tests tab and with the Workbook Test Editor, Monitor Editor or Connections Editor tab selected. It is shown in the active state when one or more objects is selected in the displayed tab, and detailed information for its use is available in Editor Zoom / Pan Toolbar topic.



*The following three Toolbars are only visible within an open Relay Workbook pane. They are shown here 90 degrees on side for this topic only.*

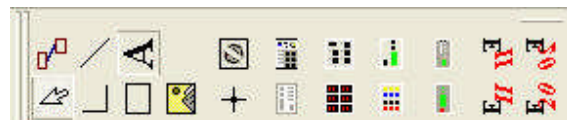
The *Test Editor Controls Toolbar* is located on the right side of the Workbook Test Editor tab. Detailed information for each menu item and the drop down list of options for each is available in Test Editor Toolbar topic.



The *Monitor Editor Controls Toolbar* is located on the right side of the Workbook Monitor Editor tab. Detailed information for each menu item and the drop down list of options for each is available in Monitor Editor Toolbar topic.



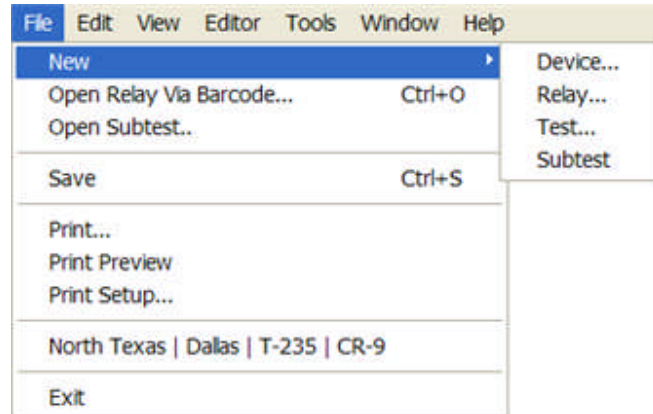
The *Connections Editor Controls Toolbar* is located on the right side of the Workbook Connections Editor tab. Detailed information for each menu item and the drop down list of options for each is available in Connections Editor Toolbar topic.



## Main Menu File

## Main Menu File

The *Main Menu File* drop list has the following items:



### New

**Relay...** item allows selection and creating a new Relay to be displayed in the Workspace Relay Tab. Selection of this item will open the New Relay Wizard. After the necessary steps have been followed to create the new Relay, it will appear in the list of Workspace Relays with the first test displayed in the Workspace Tests Tab and open in the Workbook.

**Device...** item allows the creation of a new Device to be created in the Workspace Device Tab. Selection of this item will open the New Device Wizard. After the necessary steps have been followed to create the new Device, it will appear in the list of Workspace Devices Tab with the Device open and displayed in the Workbook.

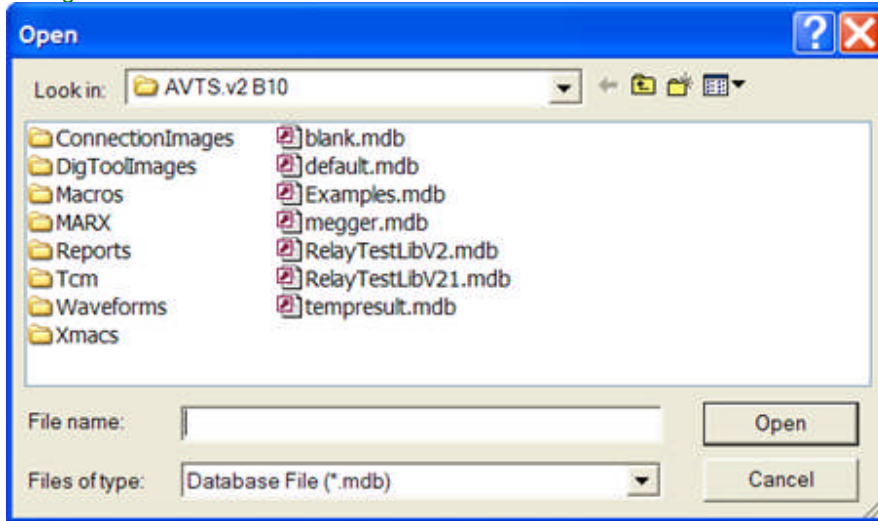
**Database...** Item displays the Windows Open dialog with the default path for the new database. The user will then provide a name of choice for the new database. Once a new file name has been entered for the new database, the presently active database will be closed and a new database void of any Devices or Relays will be opened.

**SubTest...** item allows the creation of a new SubTest to be created in the Workbook Test Editor Tab. Selection of this item will open a Test Editor tab in a new Workbook pane. After the necessary steps have been followed to create the new Test, a dialog will appear to save. Once saved, the test can be recalled and placed into any other Relay Test Editor as part of the total test procedure. See [New SubTest](#).

**Open Subtest...** Item allows the selection of a previously created Subtest. The evoked Subtest dialog displays a list of the Subtest(s) labels. One may be selected to view or edit. File Open Subtest...

**Open Relay via Barcode...** Item allows the opening of a relay by scanning the barcode on the relay or by entering the barcode using the keyboard in the open Relay Barcode dialog.

*Change Database...*



Item displays the Windows Open dialog with the default folder with available databases. The default folder for the database location will typically be the AVTS installed folder. The user will then select a database and Click OK. The Selected database will open in AVTS with the standard Login dialog.

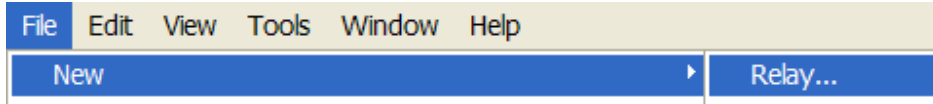
**Print Setup...** Item opens the standard Windows Print Setup dialog for controlling the output print appearance. The user is able to select a printer connected to the computer or network, select paper source and size, and paper orientation.

The **List Area** with the list of "ABB/Westinghouse...." is reserved for a list of files of the most recently opened devices and relays in the current active database. The number of files listed is limited to the last five that were opened.

**Exit** Item if selected will close the AVTS program.

## File New Relay

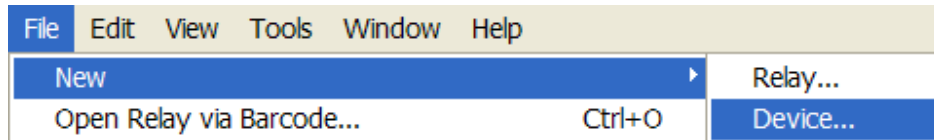
When selected, the File *New Relay...* item allows creating a new Relay to be placed in the Workspace Relay Tab. Selection of this item will open the New Relay Wizard. After the necessary steps have been followed to create the new Relay, it will appear in the list of Workspace Relays and opened with the first test displayed in the Workbook Tests Tab.





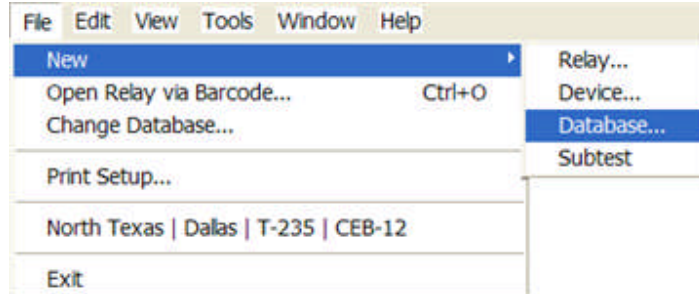
## File New Device

When selected, the File *New Device...* item allows a new Device to be placed in the Workspace Devices Tab. Selection of this item will open the New Device Wizard. After the necessary steps have been followed to create the new Device, it will appear in the list of Workspace Devices Tab and opened in the Workbook with the Settings tab displayed.

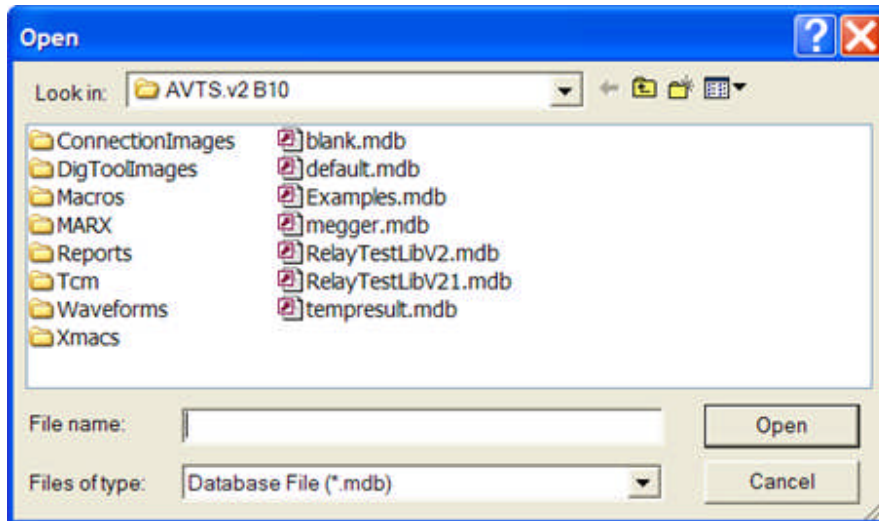


## File New Database

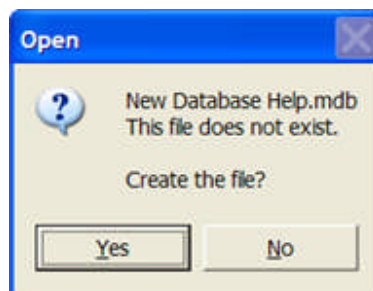
The File *New Database...* is for selecting a database other than the one presently accessed by AVTS.



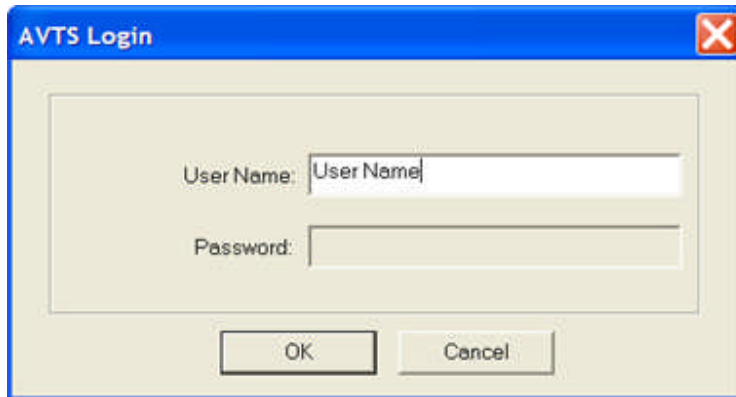
This item displays the *Windows Open* dialog with the default folder for the new database. The default folder for the database location will typically be the AVTS installed folder. The user will then enter into the File name edit field a name for the new database.



After the new database name has been entered, selecting the Open button will evoke the message *Create the file?* query.



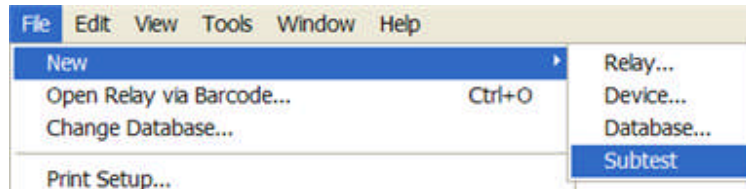
Selection of Yes will open the *AVTS Login* dialog for entry and use of the new database. The previous database will be closed and the new database will be *opened, ready for creating new relays and devices or importing existing subsets of existing databases of relays and devices*, void of any Devices and Relays.



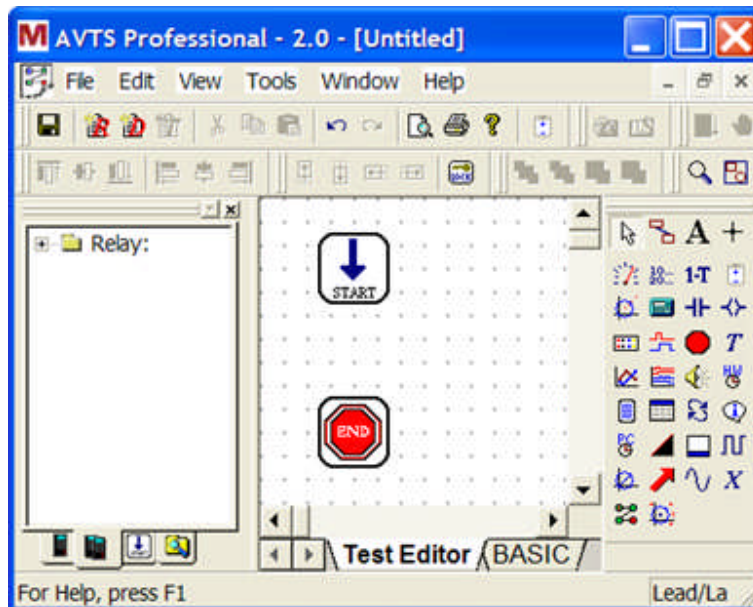
Should an existing database name be entered and the Open button selected, the named database will be opened with the AVTS Login dialog visible. The database will not be altered or different in components from the previous use of that database. There will be no message informing the user that the entered database name is the same as an existing database.

## File New Subtest

The *File New Subtest...* opens a new test in the Test Editor tab of the Relay Workbook. The Subtest is unrelated to any existing or other test for a given relay. The test, once created and saved is then available to be inserted in a test procedure for a relay test. The use is for the creation of a test that can be a standard procedure for any relay; thus, not having to be created from scratch for a new test.



There are only two tabs associated with the open Subtest, the Test Editor and the BASIC tabs. The Start and End Controls are in place in the editor tab with the Test Controls available for creating a special test, the *SubTest*.

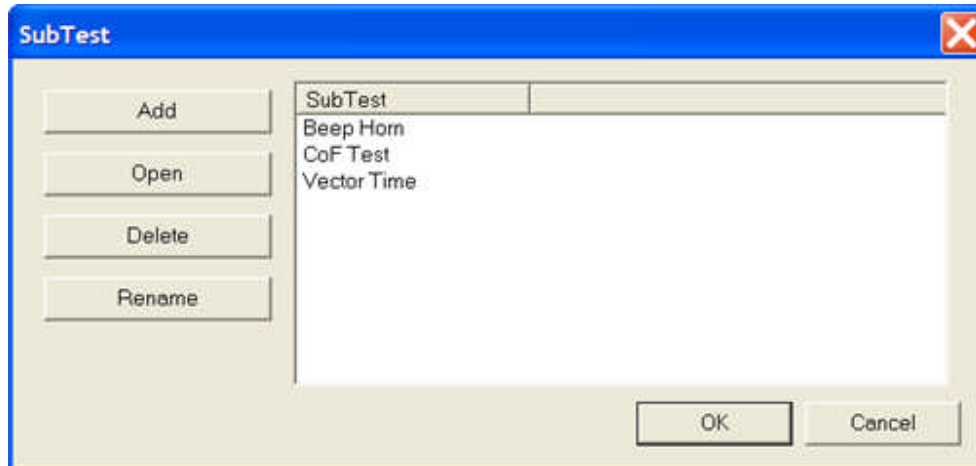


Once the test is created as desired, it may be saved and recalled at any future time for insertion into a test.

The process for using the SubTest in a Relay test procedure is described in the SubTest Control topic.

## File Open Subtest

The **Open Subtest...** item will display the Subtest dialog for selection of a previously labeled and saved subtest routine. The selected subtest will open to display the test procedure for that Subtest.

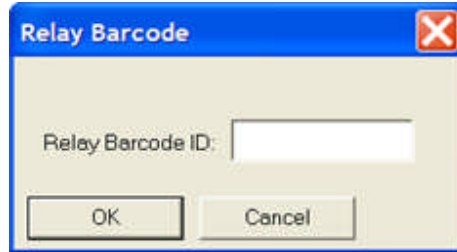


A **Subtest** may be created using the following options:

- Use the File New Subtest option
- Use the Open Subtest... option and select Add
- Use the Subtest Control in the Test Editor tab of an open Relay
- The New Subtest Example topic demonstrates creating a Subtest procedure.

## File Open Relay via Barcode...

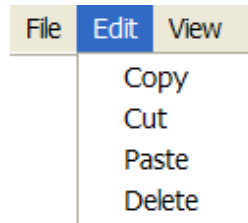
The *Open Relay via Barcode...* item allows the opening of a relay by scanning the barcode on the relay or by entering the barcode using the keyboard in the open Relay Barcode dialog.



In lieu of scanning a barcode to open a relay test module, an alpha-numeric ID for a relay may be substituted with use of the computer keyboard. The entry for a discrete relay is set to the database by opening the Barcode ID entry box in the relay Workbook General Information tab and saving the relay.

## Main Menu Edit

The **Edit Menu** drop down list is displayed below. Additional Edit Menu items will be present in the drop down list when a Workbook Device or Workbook Relay is open in the Workbook pane. Details are in the Relay Menu Edit and Device Menu Edit topics.



The **Copy** option will create a duplicate of a selected object, text or group of objects and hold them in a clipboard for later use.

The **Cut** option will remove a selected object, text or group of objects and hold them in a clipboard for later use.

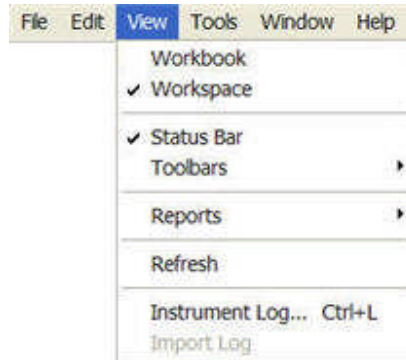
The **Paste** option will place the last object, text or group of objects last placed in a clipboard by a previous Cut or Copy action.

The **Delete** option will remove the object, text or group of objects selected by the cursor.

## Main Menu View

## Main Menu View

The *Main Menu View* menu item drop down list displays the available options as shown below.



The **Workbook** option, if unchecked, will display only the Device or Relay last opened.

If more than one Device and/or Relay is open in the Workbook, each of the open files will be created automatically in the Workbook and the View/Workbook Menu option will be found checked. Having the multiple Devices and Relays open, each with a tab, allows easy selection of the file to be immediately active and displayed. Only one Relay or Device may be active at any time, regardless of the number that have been opened or remain opened in the Workbook.

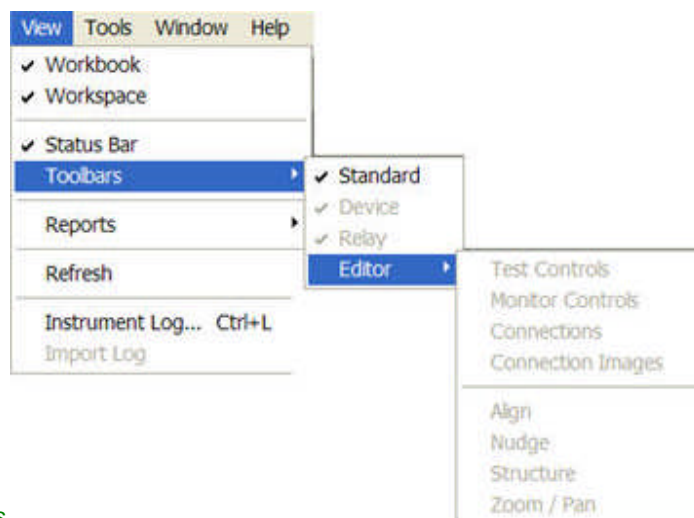
The **Workspace** option, if unchecked, will cause the Workspace pane to hide from view and allow the full AVTS work area to be reserved for the Workbook. If the Workspace pane is hidden, it may be reopened to view by applying a check on the View/Workspace menu item.

The **Status Bar** option, if checked, will display a window at the bottom of an open AVTS. The Status Bar is open by default when AVTS is opened, and displays the set angular displacement mode and the Windows path for the open AVTS database. If a Device is open in the Workbook pane, the Status Bar remains

unchanged. For Help, press F1 Lag C:\Program Files\Megger\AVTS.v2 B10\RelayT

If a Relay is open in the Workbook pane, the Status Bar displays the set angular displacement mode and the 'Device Label:RelayName.TestsTab.TestName'.

For Help, press F1 Lag CO-11\_Ver\_2\_Rev\_1:CO-11\_Ver\_2\_Rev\_1.Tests.Timing Test



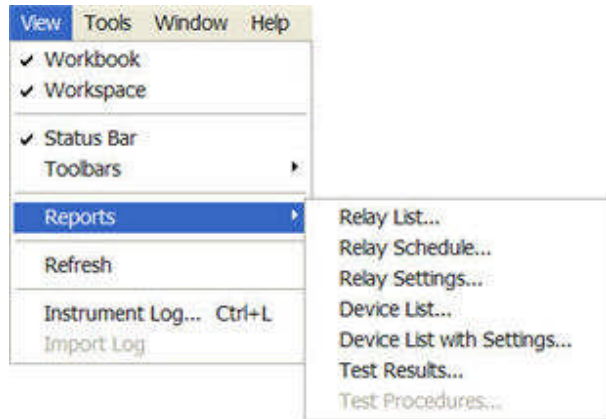
The *Toolbars* option lists several tool bars that when checked will be displayed on the Main Tool Bars under the Main Menu. The Editor Toolbar has several tools that, if checked, may also be displayed on the Main Tool Bars for use in an open



Workbook. If AVTS is just opened and the View/Toolbars item is selected, only the Standard Toolbar is made visible. The Toolbars for Device, Relay and Editor are dimmed and enabled only when a Device or Relay is opened. The Editor Toolbar is enabled only when a Relay is open with the Test Editor, Monitor Editor or Connections Editor tab being selected.

- Editor Test Controls Toolbar
- Editor Monitor Controls Toolbar
- Editor Connections Toolbar
- Editor Connections Images Toolbar

**Note:** Align, Nudge, Structure and Zoom/Pan Toolbars are active in each of the Editor Tabs of an open Relay.



The *Reports* option lists several types of reports that may be created for printing. Each of the listed reports will open a dialog for detailing the selected report type. Relay List..., Relay Schedule..., Relay Settings..., and Test Results... will open the Relay Select Tree dialog for choosing the range of relays from which the report will be created. The Device List... and Device List with Settings... will open a printable report containing the information for all Devices in the Workspace Device tab. The Test Procedures... will open a printable report containing written statements for the Test Editor Controls for particular test.

The **Instrument Log...** option opens the AvtsDevDrv.log in a Notepad file. The Instrument Log is set active in the Menu/Tools/Options... dialog. Open this dialog, select the Devices tab and click the Log Commands check box. When active, the log will capture all communication from and to AVTS and the connected test equipment. The log can be saved by selecting File/Save As... and providing a label for the file. The file is useful for review of the communication and is often used as a trouble shooting tool. Detail information may be found in the View Instrument Log... topic

Detailed information for the drop down list of options for each is available in the following topics:

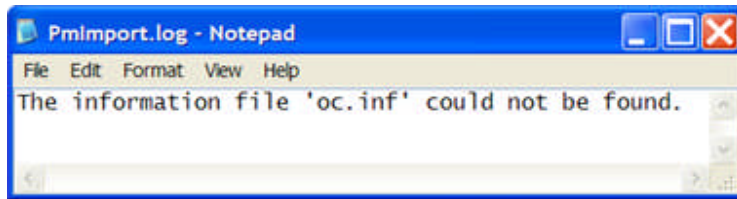
- View Workbook
- View Workspace
- View Status Bar
- View Toolbars
- View Reports

Refresh will update the Workspace Devices or Relays tab. Any tests not visible in the Relay list will be updated. Refresh is not available with a Device or Relay open in the Workbook.

Instrument Log...

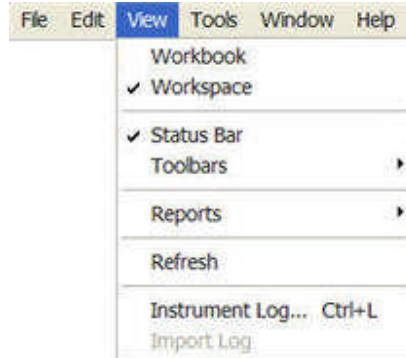
*Import Log* will open a Notepad file of information about the test selected. The "*filename*".inf is of the format produced in information files in AVO PulseMaster Relay Test Software. Information may be added in an open file and saved to the AVTS Macros folder for ready access.

AVTS Help 092607



## View Workbook

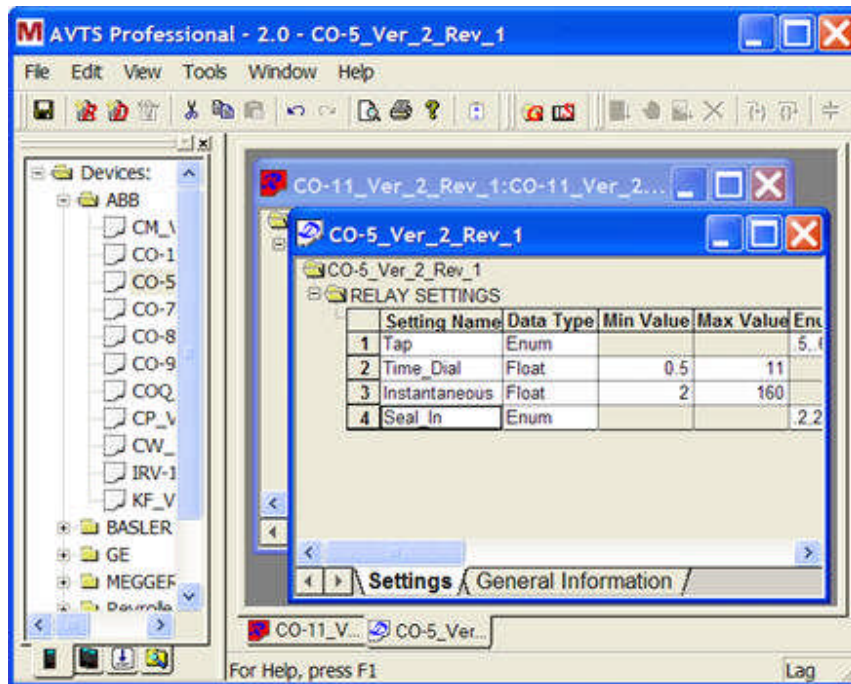
The *Workbook* menu option, if unchecked, will display only the Device or Relay last opened. The AVTS default is to have this item unchecked.



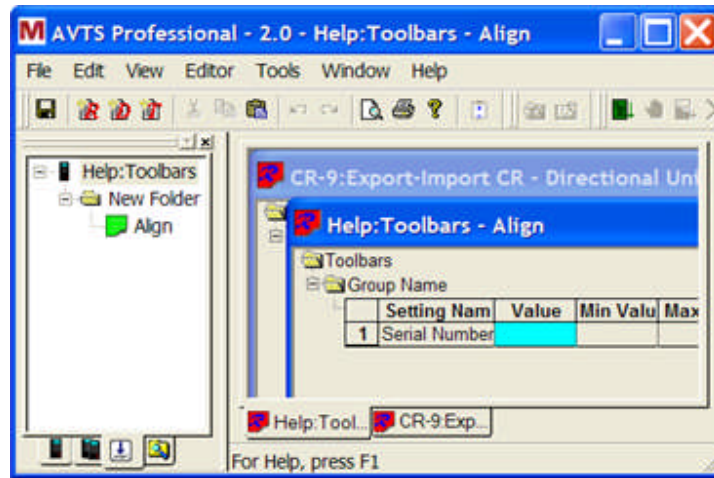
If more than one Device and/or Relay is open in the Workbook, each of the open files will be retained open in the Workbook, but only the last active file will be displayed in the full Workbook pane.

With the View/Workbook option unchecked, there are two methods make active another open Relay or Device:

Reduce the size of the active Relay or Device and all of the open items will be displayed in *Cascade* reduced format. Enlarging one of the items will utilize the full Workbook pane with no identification tabs for the inactive items.



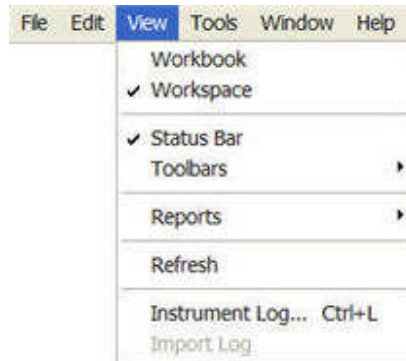
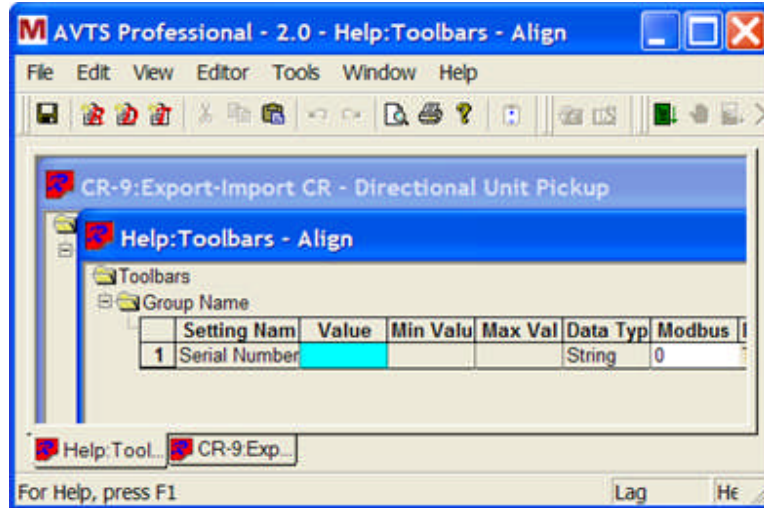
Check the View/Workbook option. Each of the open Relays and Devices will then have an *identification tab* in view across the bottom of the pane. Each open item having a tab, allows ease to select the file to be immediately active and displayed.



Only one Relay or Device may be active at any time, regardless of the number that have been opened or remain opened in the Workbook.

## View Workspace

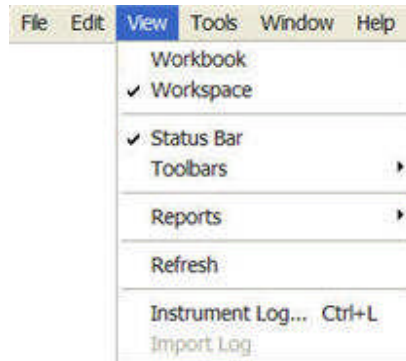
The *Workspace* option, if unchecked, will cause the Workspace pane to hide from view and allow the full AVTS work area to be reserved for the *Workbook*.



If the Workspace pane is hidden, it may be reopened to view by applying a check on the View/Workspace menu item. The user may find best use of AVTS to have the Workspace in view at all times. The AVTS default is for the View Workspace item to be checked.

## View Status Bar

The *Status Bar* option, if checked, will display a horizontal pane at the bottom of an open AVTS. The Status Bar is open by default when AVTS is opened, and displays the set angular displacement mode and the Windows path for the open AVTS database.



If a Device is open in the Workbook pane, the Status Bar remains unchanged.

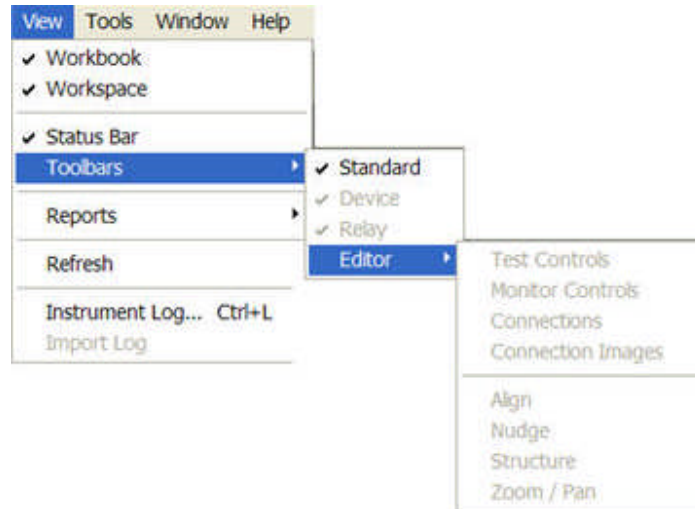
For Help, press F1    Lag    C:\Program Files\Megger\AVTS.v2 B10\RelayT

If a Relay is open in the Workbook pane, the Status Bar displays the set angular displacement mode and the 'Device Label:RelayName.TestsTab.TestName'.

For Help, press F1    Lag    CO-11\_Ver\_2\_Rev\_1:CO-11\_Ver\_2\_Rev\_1.Tests.Timing Test

## View Toolbars

The *Toolbars* option lists several tool bars that when checked will be displayed on the Main Tool Bars under the Main Menu. The Editor Toolbar has several tools that, if checked, may also be displayed on the Main Tool Bars for use in an open Workbook.



If AVTS is just opened and the View/Toolbars item is selected, only the Standard Toolbar is made visible.

The Toolbars for Device, Relay and Editor are dimmed and enabled only when a Device or Relay is opened. The Editor Toolbar is enabled only when a Relay is open with the Test Editor, Monitor Editor or Connections Editor tab being selected.

The Standard Toolbar is defined in the Main Menu Toolbar topic.

The Device Toolbar is defined in that topic.

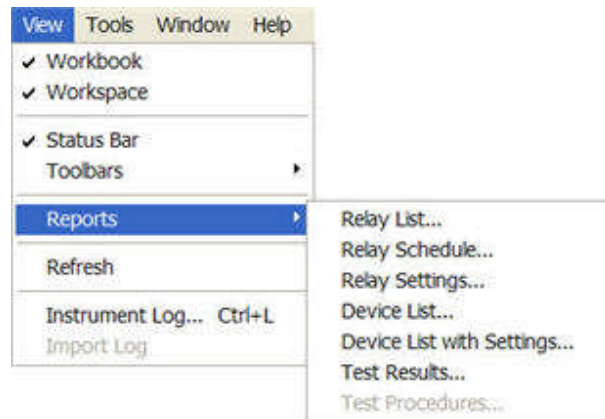
The Relay Toolbar is defined in that topic.

The Editor Toolbars options are defined in the Relay Menu Editor topic.

## View Reports

### View Reports

The *Reports* option lists several types of reports that may be created for printing. For General information about all Reports, refer to the View Reports General topic.



An open Device will have these same Reports available.

An open Relay will have these and additional Reports particular only to an open Relay. The additional options are described in the Relay Menu View topic.

Relay List... is for creating a report listing the Relays present in a selected organization of the database.

Relay Schedule... is for creating a report listing Relays due for test or have been tested in a selected organization of the database.

Relay Settings... is for creating a report listing Relays with all the settings in a selected organization of the database.

Device List... is for creating a report listing Devices in a selected organization of the database.

Device List with Settings... is for creating a report listing Devices with all the settings in a selected organization of the database.

Test Results... is for creating a report listing Relays with current and/or past test results in a selected organization of the database.

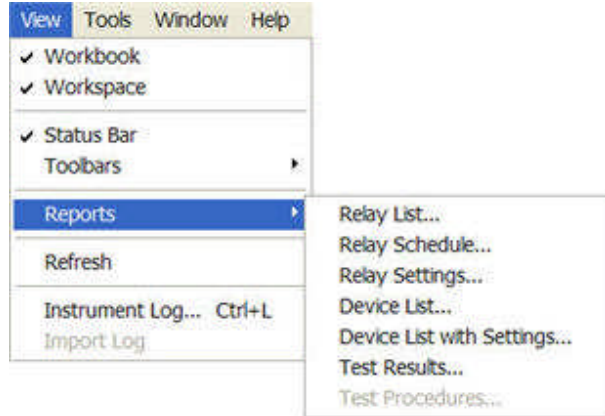
Current Test Results is visible only with a Relay open in the Workbook.

Test Procedures... is visible only with a Relay open in the Workbook, and is for creating a text report of a test procedure open in the Relays Workbook Test Editor tab.



## View Reports General

The *Reports* option lists several types of reports that may be created for printing.



The Report options listed in the Main Menu View are described below. An open Device will have the same Reports available. An open Relay will have these and additional Reports particular only to an open Relay. The additional options are described in the Relay Menu View topic.

When created, all Reports will open in the AVTS Workbook with a menu bar at the top of the Workbook. Reference to the menu bar the left to right items are described below.



The left facing major arrow head is to return to the first page of the report.

The left facing arrow head is to return toward the first page of the report, one page with each click.

The numbers indicate the visible page number of the total pages of the report.

The right facing arrow head is to advance toward the last page of the report, one page with each click.

The right facing major arrow head is to advance to the last page of the report.

The square symbol is to Cancel reading records. If the database has an extremely large results that have been filtered for a report, the report generation may take several minutes to generate. Should it be necessary to stop the report generation, selecting this button will cease the generation and return to the active AVTS.

The standard Microsoft Print icon is shown and transfers the report directly to the Print dialog.

The Report Export icon is to transfer the report using selectable formats and destination types.

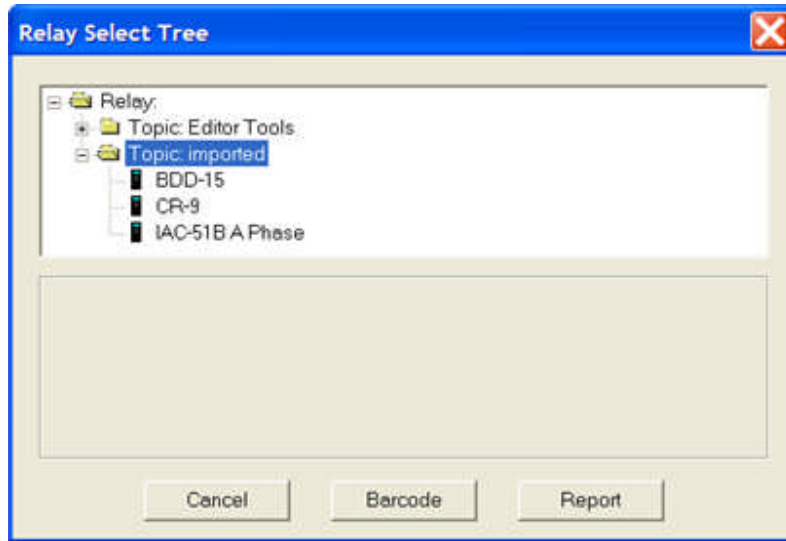
The Zoom window uses a drop down list for sizing the report for ease to view in the Workbook. The list values are 400%, 300%, 200%, 150%, 100%, 75%, 50%, 25%, Page Width and Whole Page. A value may be typed into the size field, but the actual size viewed will be defaulted to a value listed that is closest to the value typed.

The remainder of the menu bar displays a Total:Records, Percent Read, and Records Read. The Total Records are the number of records that have utilized in the database. The Percent Read is the value of the records utilized that are in the report. The Records Read are the number of utilized records of the total number of records in the database. For example, if a Relay has 5 types of test, and only 3 of the tests were executed, had results and then saved, the report for that saved test results would display 3 sets of test results, but would list also the other two test with no results. Hence the Total Records are 3, the Percent Read is 100% and the Records Read would be 3 of 5.

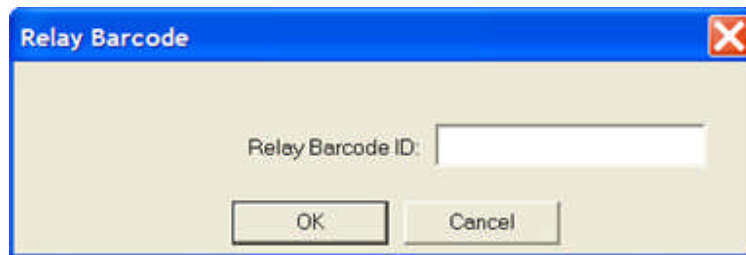
Each of the listed reports will first open the *Relay Select Tree* dialog for detailing the selected report type.

The report will consist of the data in the level of Organization selected. In this manner the report may contain data for a single relay or all the relays contained in any level selected. To report the entire database, select the item Relay, top most level. Each report type requested may have filter options at the

bottom of the Relay Select Tree dialog to further refine the report. The Cancel option will return the AVTS active database.

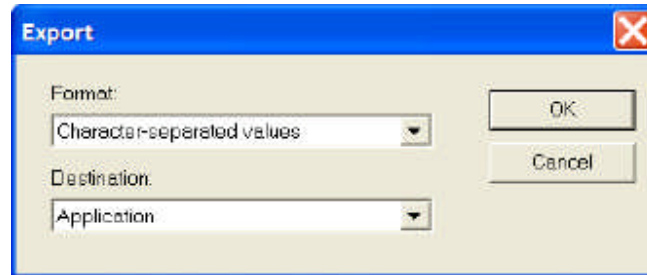


The Barcode option will open the *Barcode* dialog for to scan a relay barcode or to manually enter the barcode of a relay. The report will contain only the filtered information for the barcoded relay. The Report option, regardless of the other options selected or entered, will generate the final report.



## Report Export

The *Export* function allows options for the report to be formatted using several applications.



Use the Disk file destination option allowed for Export, by default, to a Temp folder with the necessary files in \*.html format and the graphics in \*.jpg format. Double clicking on one of the exported files opens to view all or portions of the report including the graphics in HTML format.

**Comma-separated values (CSV)** opens in Excel with only headers and no values.

**Data Interchange Format (DIF)** opens in Excel with only headers and no values.

**Excel 5.0 (XLS), Excel 7.0 (XLS), Excel 8.0 (XLS)** exports to Excel with a box for graphics.

**Excel 5.0 (XLS) (Extended), Excel 7.0 (XLS)** exports to Excel with a box for graphics.

**HTML 3.2, HTML 4.0 (DHTML)** creates new folder (default html label) and places all pages of report into several files, one file for each page of the report with Separate HTML pages checked, for the user to open using Crystal Reports format in Microsoft Internet Explorer.

**Note:** This export function, on the first use, will create a new subfolder under the C:\Program Files\Megger\AVTS folder and place the file in the subfolder.

**Lotus 1-2-3 (WK1), Lotus 1-2-3 (WK3), Lotus 1-2-3 (WKS)** exports to Oracle Lotus with a box for graphics.

**ODBC - dBASE Files, ODBC - Excel Files, ODBC - MS Access** database. Selects known database and creates default **CREXPOR**T file name.

**Paginated Text** creates a text file defaulted into temp folder with \*.rec extension.

**Record Style** (columns no spaces), **Record Style** (columns with spaces)

**Report Definitions**

**Tab-separated text, Tab-separated values**

**Text**

**Word for Windows Document**

The **Destination** drop down options list

**Application**

**Disk file**

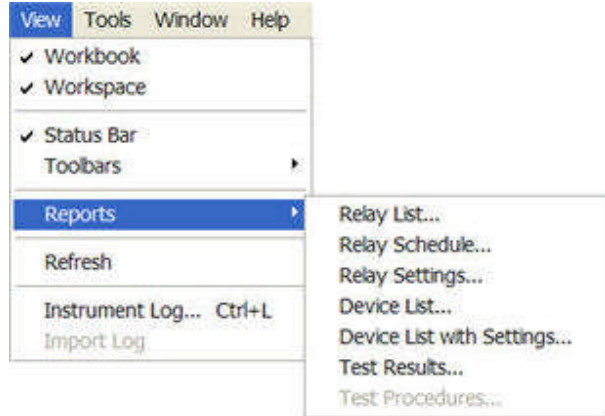
**Exchange Folder**

**Lotus Domino**

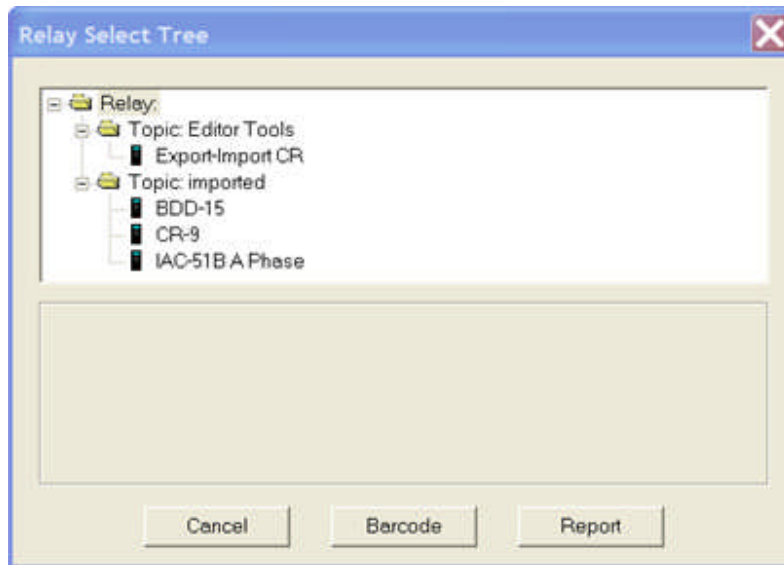
**Microsoft Mail (MAPI)**

## Relay List

The **Reports** option lists several types of reports that may be created for printing.



Relay List... opens the **Relay Select Tree** dialog. Select the area of the Organization level to create a Report with the relay list.



The **Relay List Report** will contain a listing of relays in the selected organization level.

### Relay List Report

Monday, February 27, 2006

Location	Name	Date Created	Test Modified Date	Settings Modified Date	Last Test Date	Maintenance Period (Months)
<b>Topic: imported</b>						
	BDD-15	8/31/1999	1/24/2000	1/24/2000	2/25/2006	0
	CR-9	8/31/1999	1/24/2000	1/25/2006	2/25/2006	0
	IAC-51B A Phase	10/19/1999	2/25/2006	2/25/2006	2/25/2006	0

The report columns will be:

**Location** - The relay location in the Organization level of the database

relay **Name** - The relay label as shown in the Organization level of the database

**Date Created** - The date the relay was placed in the Organization location of the database

**Test Modified Date** - The most recent date the relay test was modified in the Test Editor tab

**Settings Modified Date** - The most recent date the relay settings were modified in the Relay Settings tab

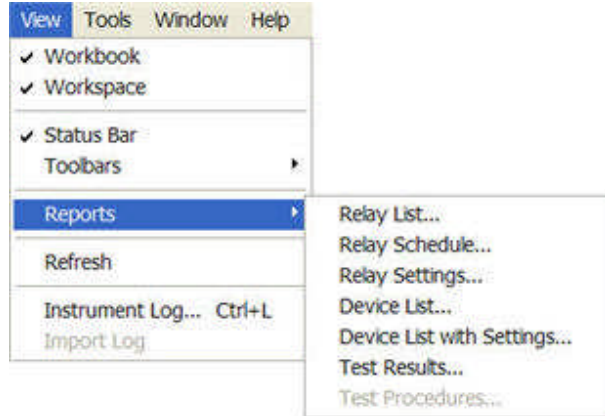
**Last Test Date** - The most recent date of the tests performed and saved to the database

**Maintenance Period (Months)** - The period of time for subsequent periodic testing of the relay

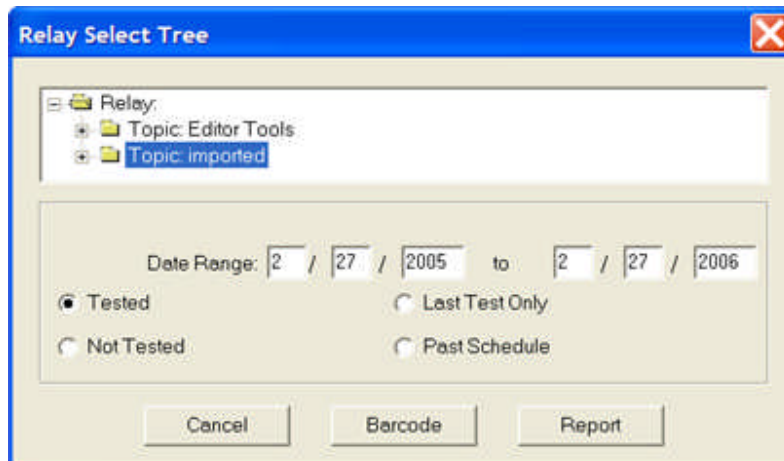
Note the Maintenance Period lists zero as the number of months for each relay. Should the relay have been assigned a maintenance period in the Tools Options Relay General Information tab, that value would appear in this column.

## Relay Schedule

The *Reports* option lists several types of reports that may be created for printing.

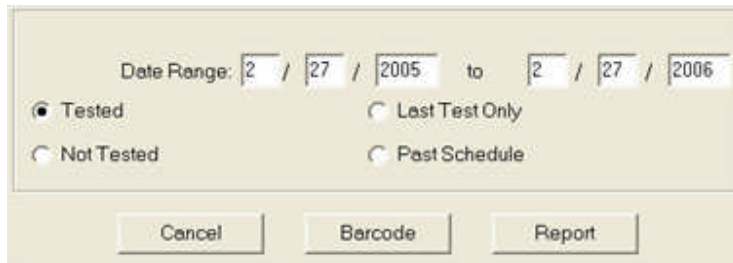


Relay Schedule... opens the *Relay Select Tree* dialog.



Select the area of the Organization level to create a Relay Schedule Report with the schedule list. The date filters are Tested, Last Test Only, Not Tested and Past Schedule.

The *Tested* filter has two date windows for setting the date range in the database from which the report will be generated. The default range will be From a year earlier To the present date.



The *Last Test Only* filter has no filters and generates a report of only the most recent relays tested in the database for the selected relays.

A dialog box with a light beige background. It contains four radio buttons arranged in a 2x2 grid. The top-left radio button is labeled 'Tested' and is unselected. The top-right radio button is labeled 'Last Test Only' and is selected, indicated by a black dot. The bottom-left radio button is labeled 'Not Tested' and is unselected. The bottom-right radio button is labeled 'Past Schedule' and is unselected. Below the radio buttons are three buttons: 'Cancel', 'Barcode', and 'Report', each enclosed in a rectangular frame.

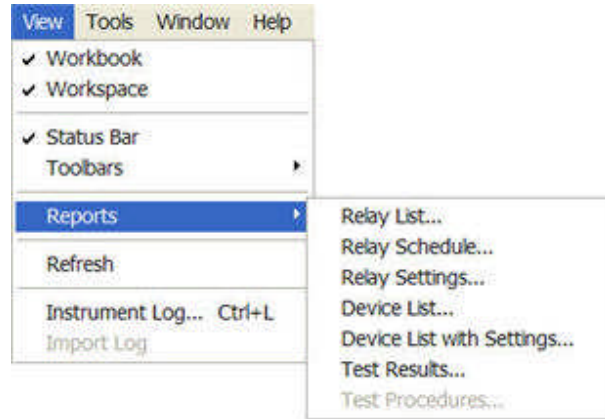
The *Not Tested* filter has one date window for setting the oldest date in the database for the selected relays from which the report will be generated.

A dialog box with a light beige background. At the top, it features a 'Since Date' label followed by three input fields containing the values '2', '27', and '2005', separated by slashes. Below this are four radio buttons in a 2x2 grid. The top-left radio button is labeled 'Tested' and is unselected. The top-right radio button is labeled 'Last Test Only' and is unselected. The bottom-left radio button is labeled 'Not Tested' and is selected, indicated by a black dot. The bottom-right radio button is labeled 'Past Schedule' and is unselected. At the bottom are three buttons: 'Cancel', 'Barcode', and 'Report', each enclosed in a rectangular frame.

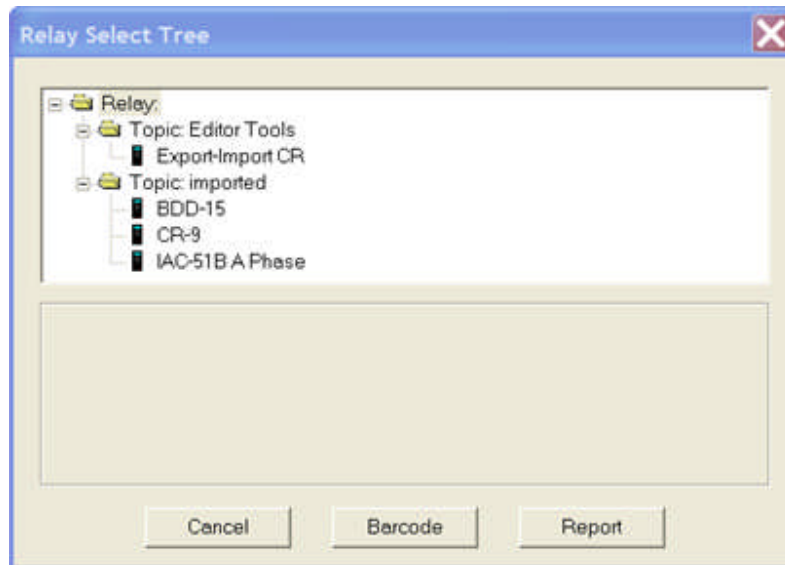
The **Past Schedule** option has no filters and generates a report of only the relays with a maintenance period and which have passed the date for scheduled test.

## Relay Settings

The **Reports** option lists several types of reports that may be created for printing.



**Relay Settings...** opens the **Relay Select Tree** Dialog.



Select the area of the Organization level to create a **Relay Settings Report** with the settings lists. The report will present all the settings data with each relay beginning a new page in the report.



---

## Relay Settings Report

Monday, February 27, 2006

Topic: imported

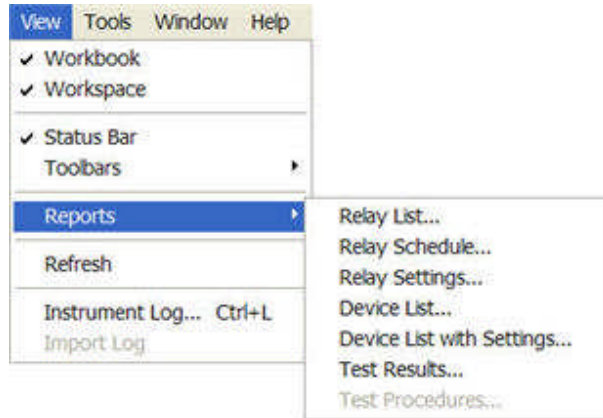
Relay: BDD-15

Barcode:

Setting	Value	Description
<b>BDD-15 Settings</b>		
Winding_#1_Tap	5.0	Winding #1 tap setting
Winding_#2_Tap	5.0	Winding #2 tap setting
%Slope	15	%Slope Tap Setting
Harmonic_Restraint	18.00	% Harmonic Restraint setting
Instantaneous	30.00	Instantaneous Setting in Amps
DC_Volts	48	DC Control Voltage Tap Setting

## Device List

The *Reports* option lists several types of reports that may be created for printing.

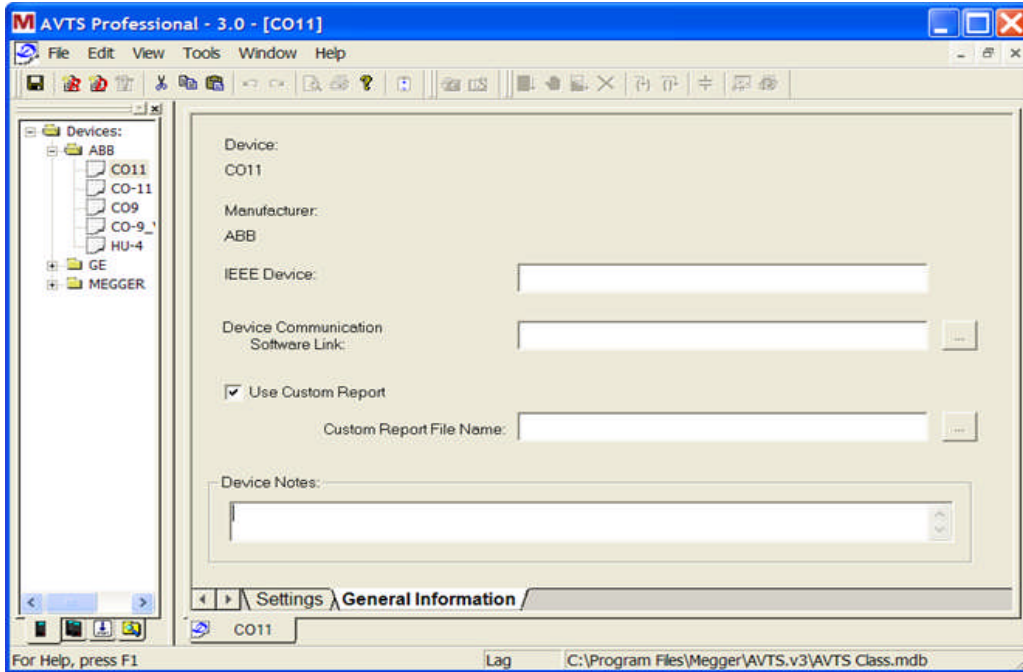


The **Device List...** creates a *Device List Report* containing a list of all Devices in the Workspace Device Tab.

<b>Device List Report</b>			
Monday, February 27, 2006			
<u>Manufacturer</u>	<u>Device Name</u>	<u>IEEE Device</u>	<u>Count</u>
<b>GE</b>	BDD-15	87	1
	IAC-51B	50/51	1
<b>MEGGER</b>	MPRT		0
<b>Westinghouse</b>	CR-9		2

The report columns are **Manufacturer**, **Device**, **IEEE Device** and **Count**. The Manufacturer and Device are the labels viewed in the Workspace Device tab.

The **IEEE Device** label is an entry in the Workspace *Device General Information* tab.

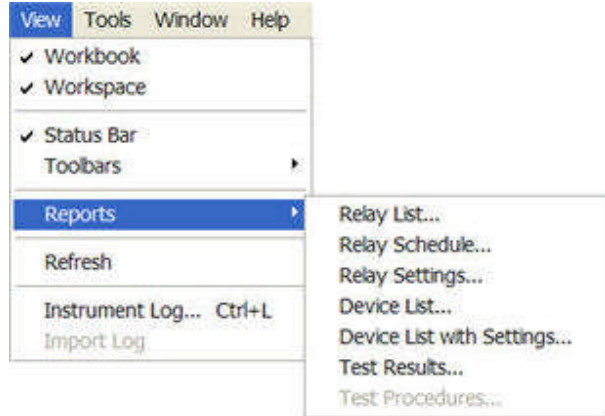


The **Count** is the number of Device related Relays that have been modified one from the other; a relay may be copied from one location to another and one of the Relay settings eliminated or one added. Both relays have a different Settings list or the Settings Name may differ, but both are of the same parent Device. In the *Device List Report*, the CR-9 Count is 2, as one of the CR-9 relays in the Relays tab is different from all the others of the same Device template.

<b>Device List Report</b>			
Monday, February 27, 2006			
<u>Manufacturer</u>	<u>Device Name</u>	<u>IEEE Device</u>	<u>Count</u>
GE	BDD-15	87	1
	IAC-51B	50/51	1
MEGGER	MPRT		0
Westinghouse	CR-9		2

## Device List with Settings

The *Reports* option lists several types of reports that may be created for printing.

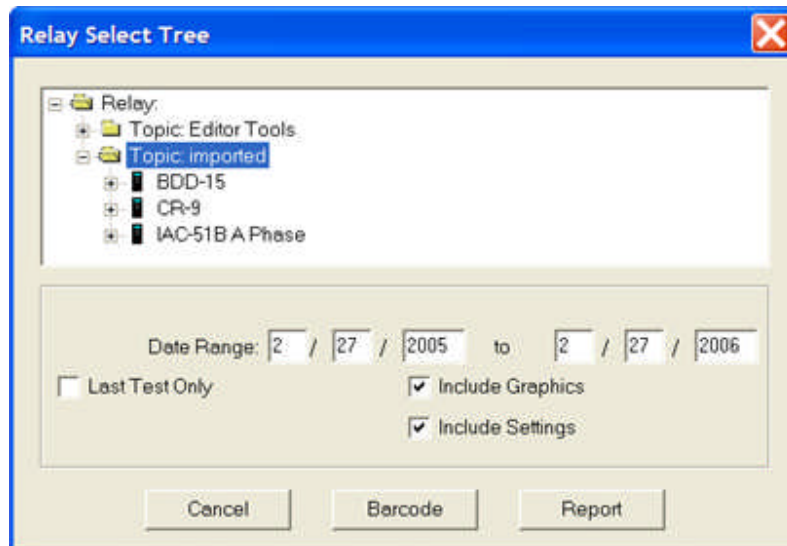


**Device List with Settings...** creates a *Device with Settings Report* containing the setting information for all Devices in the Workspace Device tab.

Device List With Settings Report				
Monday, February 27, 2006				
Device	Setting	Data Type	Range	Description
<b>GE</b>				
<b>BDD-15</b>				
	<b>BDD-15 Settings</b>			
	Winding_#1_Tap	Enum	2,9,3,2,3,5,3,8,4,2,4,6,5,0,8,7	Winding #1 tap setting
	Winding_#2_Tap	Enum	2,9,3,2,3,5,3,8,4,2,4,6,5,0,8,7	Winding #2 tap setting
	%Slope	Enum	15,25,40	%Slope Tap Setting
	Harmonic_Restraint	Float	18.00 to 22.00	% Harmonic Restraint setting
	Instantaneous	Float	0.00 to 150.00	Instantaneous Setting in Amps
	DC_Volts	Enum	48,125,250	DC Control Voltage Tap Setting
<b>IAC-51B</b>				
	<b>IAC 51 Settings</b>			
	Tap	Enum	.5, .6, .8, 1.0, 1.2, 1.5, 2.0, 2.5, 3.0, 4	Available IAC Tap Settings
	Time_Dial	Float	0.50 to 10.00	Relay Time Dial Setting
	Instantaneous	Float	2.00 to 160.00	Relay Instantaneous setting in amps
	T&SI	Enum	.2, 2	Target & Seal In Setting in DC amps

## Test Results

Test Results... will open the *Relay Select Tree* dialog.



Select an organization level or a single relay to generate a *Relay Results Report*. Filters are for results in a range of selectable dates or the Last Test Only. The check boxes Include Graphics and Include Settings allow for the settings and graphics to be included or not.

### Relay Results Report

<b>Topic:</b> imported	<b>Relay:</b> CR-9 <b>Device:</b> Westinghouse: CR-9 <b>Tested By:</b> Bruce Bordelon <b>Date Tested:</b> 2/25/2006
------------------------	------------------------------------------------------------------------------------------------------------------------------

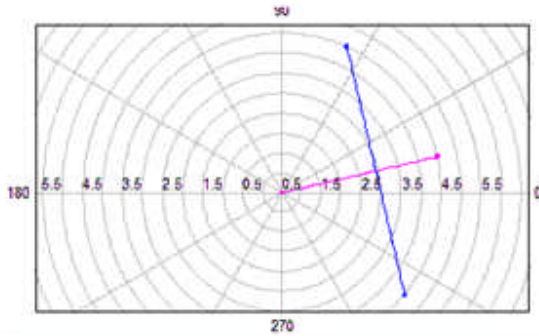
Setting	Value	Description
<b>CO-1 TO CO-11 RELAY SETTINGS</b>		
Tap	5.0	Available CO Tap Settings
Time_Dial	4.000	Relay Time Dial Setting
Instantaneous	20.000	Relay Instantaneous setting in amps
ICS	2	ICS Setting in DC amps
Tap_Range	2	1 = .5-2.5 and 2 = 4-12
MTA	30	MTA set as voltage lagging current

Relay Notes: sdsdf

Test Name: New Folder:Directional Unit Pickup

Pickup	Ia	Va	MTA	Expected_PU	Tolerance	Error	Pa
VA	Amps	Volts		VA	+/- %	%	
3.848	4.00	0.96	30.00	4.000	5.00	-3.800	

Test Name: New Folder:Dir Unit MTA Test



## View Instrument Log...

The **Instrument Log...** option opens the **AvtsDevDrv.log** in a Notepad file. The Instrument Log is set active in the Menu/Tools/Options... dialog. Open Tools/Options/Devices tab and click the Log Commands check box. When active, the log will capture all communication from and to AVTS and the connected test equipment.

The log can be saved by selecting File/Save As... and providing a label for the file. The file is default save location is in the AVTS main folder. The file is useful for review of the communication and is often used as a trouble shooting tool. File may be shared with others during trouble shooting if any problem occurs.

**Note:** The file is very useful when creating a new or editing an existing test procedure to determine if test execution functionality is as expected.

The Instrument Log... displays each command sent to a test instrument and each response received from the test instrument. The send and return codes are those commands listed in the PULSAR Command Set topic and the MPRT Command Set topic.

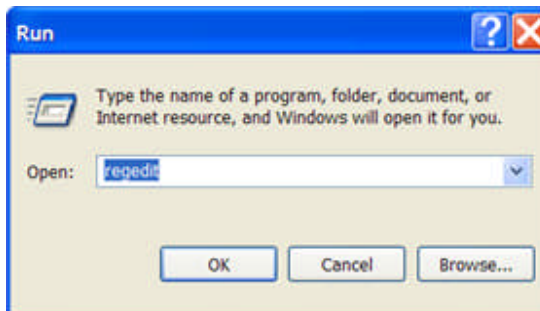
An example of an *Instrument Log...*

```

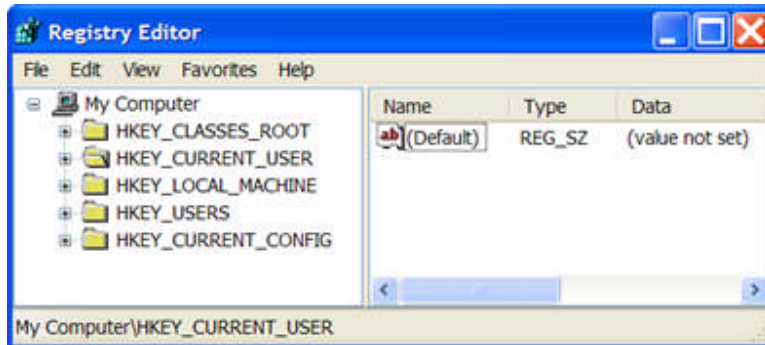
-----
Sat Jan 07 16:12:06 2006
-----
Test Set A is simulated
close serial
Test Set A COM 1 :>Write: su;
Test Set A COM 1 :>Write: qq,
Test Set A COM 1 :>Read: <empty>
Test Set A COM 1 :>Write: t01ad,
Test Set A COM 1 :>Write: t01hd,
Test Set A COM 1 :>Write: t01ca1,
Test Set A COM 1 :>Write: t01sta,
Test Set A COM 1 :>Write: t02ad,
Test Set A COM 1 :>Write: t02hd,
Test Set A COM 1 :>Write: t02ca1,
Test Set A COM 1 :>Write: t02sto,
Test Set A COM 1 :>Write: t03ad,
    
```

The date and time is entered when a test is executed and the Log Commands is enabled.

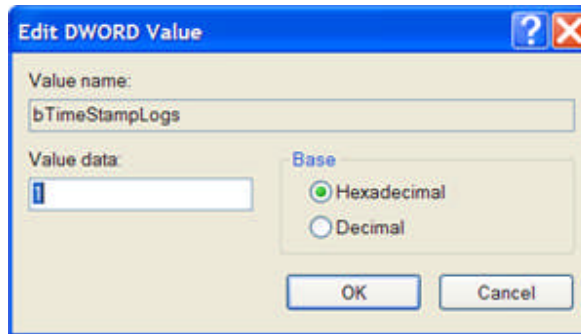
A **time stamp** may be included by entering and making a change in Windows regedit. The location for the necessary change is opened by clicking on Start/Run... The *Run* dialog is displayed and "regedit" must be typed into the edit field.



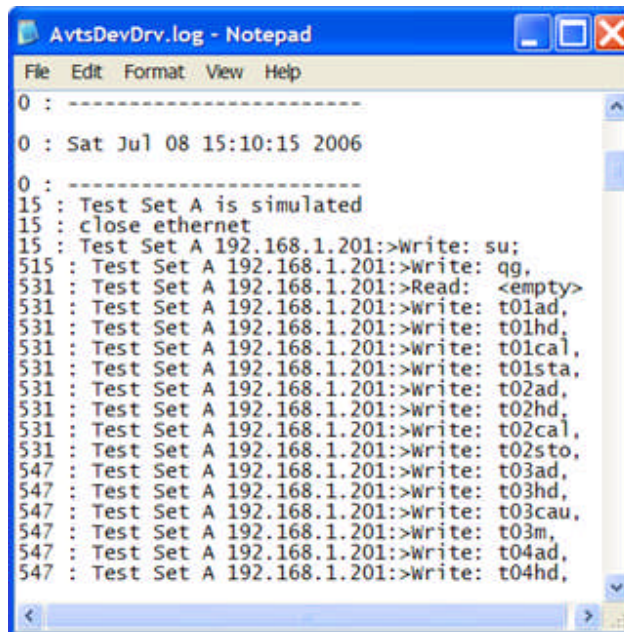
Click on Open to display the *Registry Editor* window. Locate the file "bTimeStampLogs" by clicking on **HKEY\_CURRENT\_USER** folder, **Software** folder, **Megger** folder, **AVTS.vxx** folder, **Settings** folder to display the included files.



Double click on the file **bTimeStampLogs** to display the *Edit DWORD Value* dialog. Enter the number 1 (one) in the Value data: edit field and click OK. Immediately click on the File/Exit menu item.



The time formatted *Instrument Log* will display the time in elapsed milliseconds.



**Note: This is a very dangerous operation. Extreme caution must be exercised. Any other changes made in the Registry Editor could cause software, including Windows, from being available in the future.**

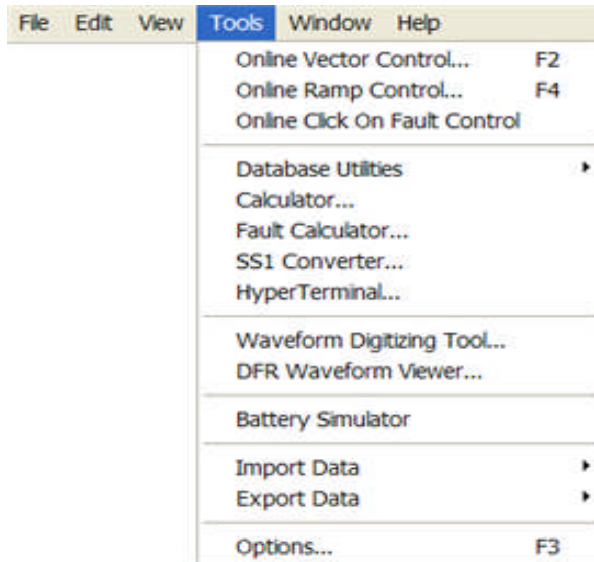
**CAUTION....CAUTION....CAUTION**



## Main Menu Tools

## Main Menu Tools

Details for the *Tools* menu drop down list may be found in the linked topics:

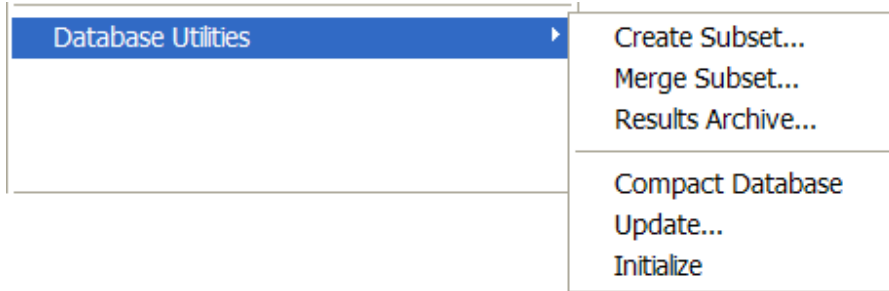


- Online Vector Control...
- Online Ramp Control...
- Online Click On Fault Control
- Database Utilities
- Calculator...
- Fault Calculator...
- SS1 Converter...
- HyperTerminal...
- Waveform Digitizing Tool...
- DFR Waveform Viewer...
- Battery Simulator
- Import Data
- Export Data
- Options...

## Database Utilities

### Database Utilities

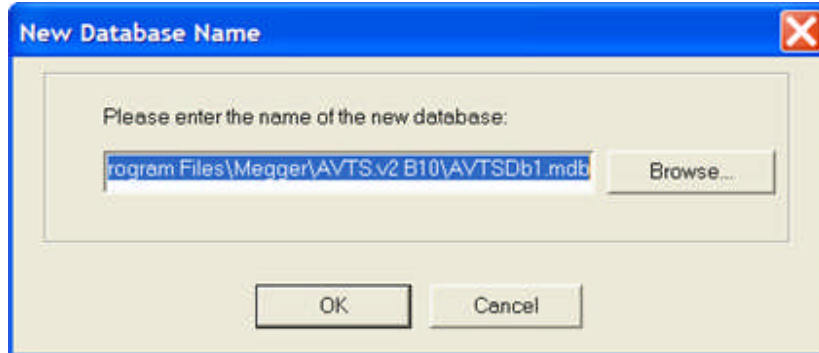
Details to the *Database Utilities* option are found in the following links:



- Create Subset...
- Merge Subset...
- Results Archive...
- Compact Database Update...
- Initialize

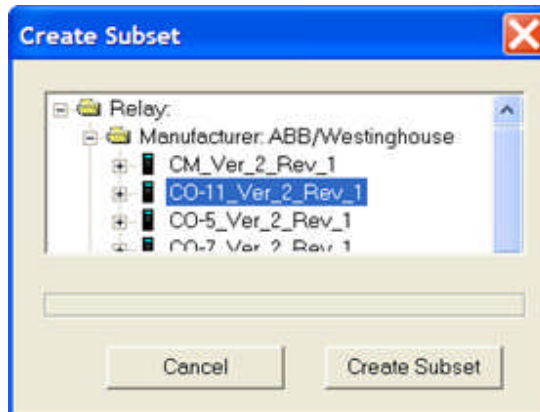
## Create Subset

To create a database of a particular Organization level, the user will select the Menu item **Tools/Database Utilities/Create Subset...** to display the *New Database Name* dialog.

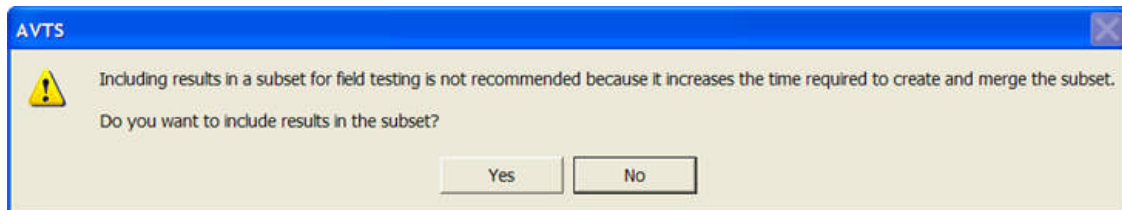


In the example, the new name is defaulted to the path of the presently open database and a default new database name "AVTSDb1.mdb". The Subset to be created will be a complete database; a Microsoft ACCESS database defined by the database name \*.mdb. The database will be of the same construction and properties as the database from which the subset was created. The user may change the default name to one which may be more descriptive than the one initially displayed in this dialog.

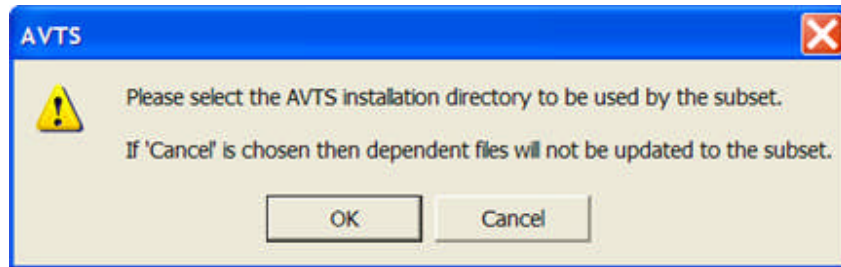
After selecting OK, the *Create Subset* dialog is displayed for selecting the level of the Organization which is to be the data portion of the original database that will comprise the new Subset.



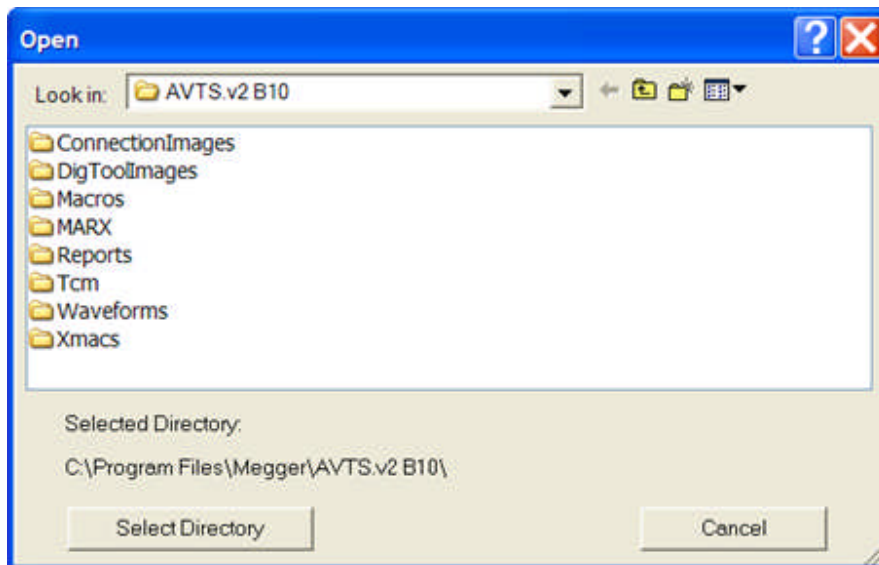
Selection of the Create Subset option, will display a *message dialog* directing the user to select Yes or No for including test results in the creation of the new Subset.



Whether the Yes or No option is selected, a *new message* will be displayed, informing the user to determine if the new Subset will be created in the same AVTS folder as the original database. If Cancel is selected, the Subset is created in the original database folder and all dependent files will be linked to the new Subset.

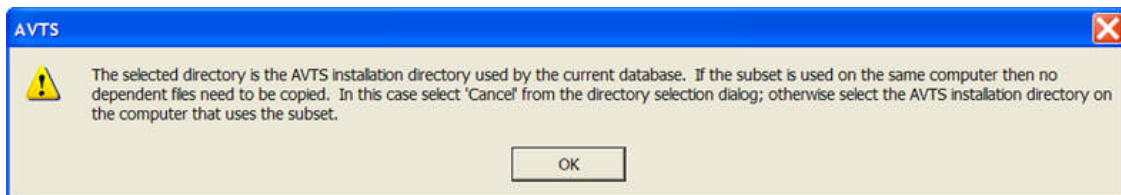


Otherwise, should the user select the OK option, the *Open dialog* is displayed for the user to select the Directory into which the Subset will be saved. The AVTS dependent files will be linked to the new Subset.



Selecting Cancel will open the new subset, replacing the database from which the subset was derived. This new subset (database) will have the name provided earlier by the user and is fully functional. Relays may be added or deleted. Tests may be added or deleted. Devices may be added or deleted. Other databases and subset may be merged using the Database Utilities Merge Subset option. The new subset will also be able to be MERGED back into the original database to update the original with the new information. Note that the Subset can be part of or all of the original database.

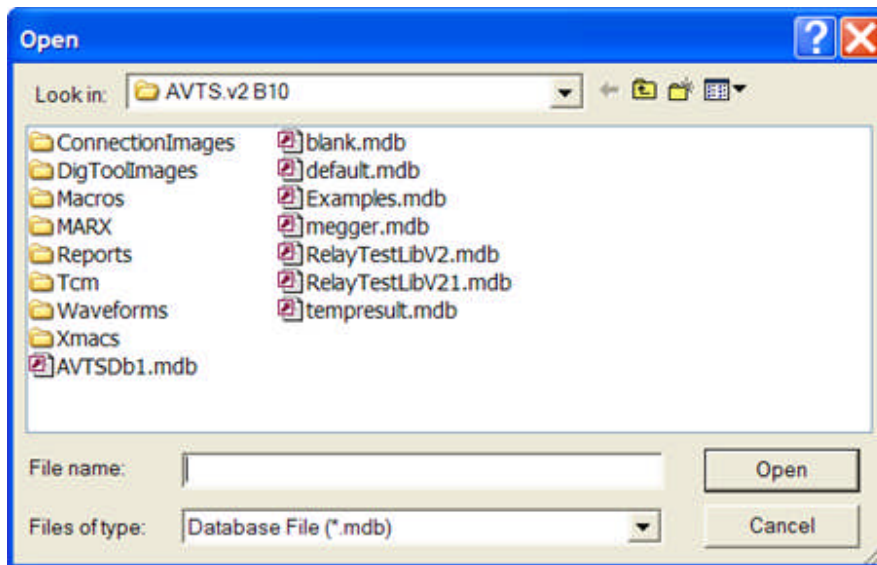
The Select Directory option will create a *message dialog* informing the user that, if the subset is placed in a directory different than the parent AVTS database, dependent files will need to have their links changed manually. Dependent files may include those in the AVTS sub-directories Waveforms, DigToolImages, ConnectionImages, etc. Select OK to return to the previous dialog to select another or the original AVTS directory.



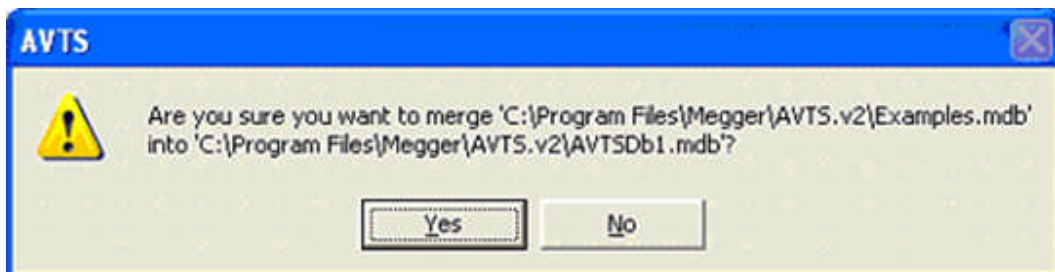
The use of Subsets allows the user to maintain a complete database for a portion or for the entire company on his computer or on a server. For large companies this process allows two or more users of AVTS to be at different locations testing relays and then “Merging” the separate Subsets to the original full database without any one merge over-writing the values of a previous merge by another Subset for a different Organization location.

## Merge Subset

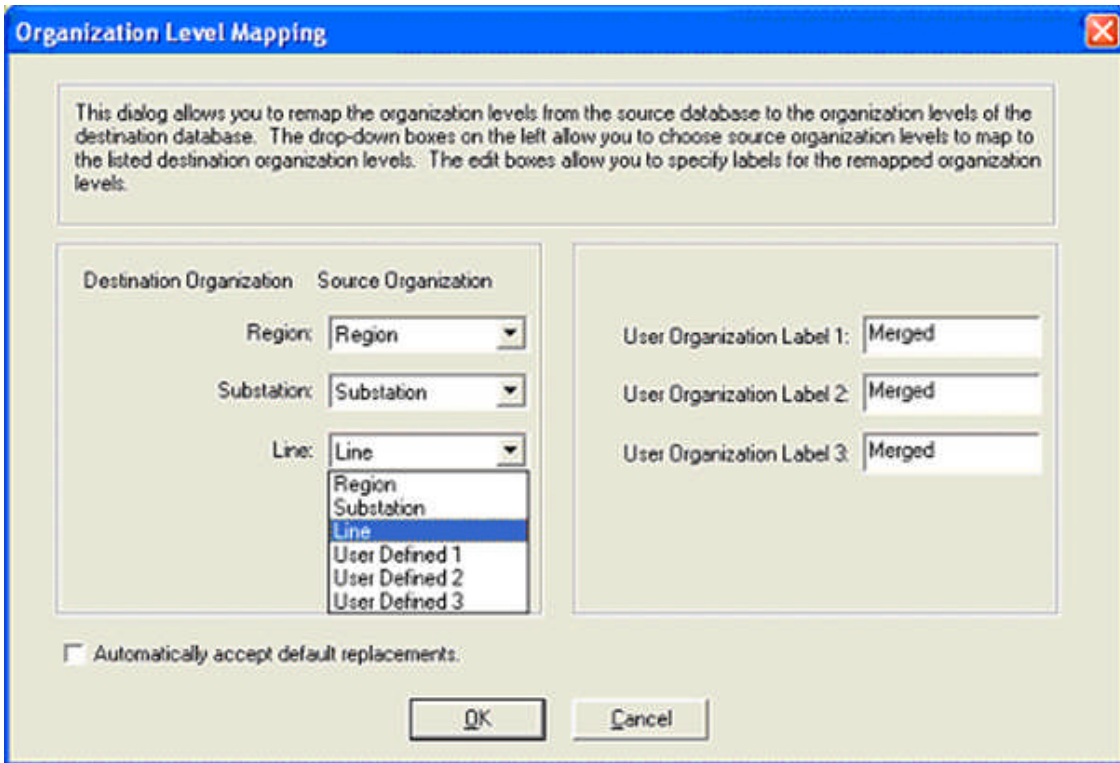
The Database Utilities/*Merge Subset* option menu item is used to combine multiple databases (subsets). This selection will display the Open dialog for the user to locate and select the subset which will be merged into the open database.



In this example, the open database is AVTSDb1.mdb which was previously created and the Examples database was chosen as the Subset to be Merged. Selection of the Open option, displays the *message dialog* directing the user to assure the chosen subset to be merged is the desired subset.

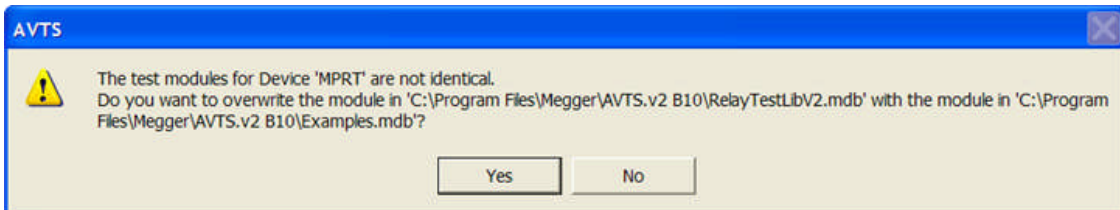


Selection of the No option will return the user to the previous step. Selection Yes will display the *Organization Level Mapping* dialog. The mapping function in the merge process allows the user to merge databases/subsets which have different Organization structures. One database may have more, the same or less levels than the other database.

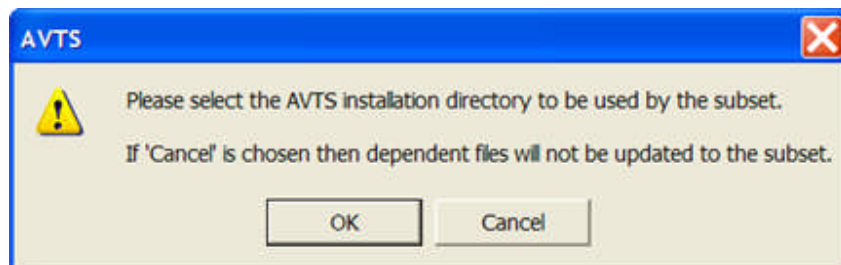


Note: Mapping databases with different levels and/or names of organization is a very important feature. The action will map a database of more or less organization levels into a database of less or more organization levels.

If the database to be merged has an Organization different to the open database, a *message dialog* will be displayed to allow aborting the merge or to allow overwriting the Organization of one to the other.

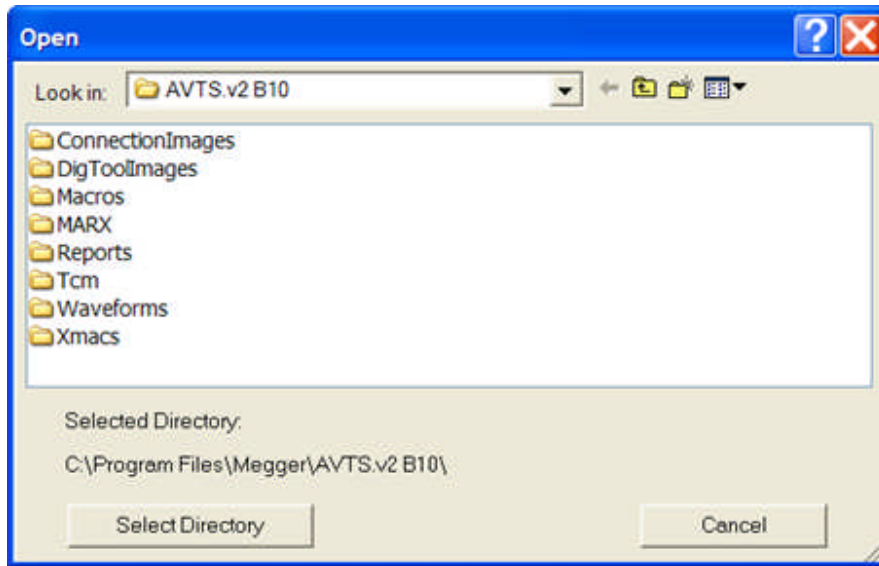


Select Yes to allow the Organization overwrite. Select No and the select AVTS installation directory *message dialog* is displayed.



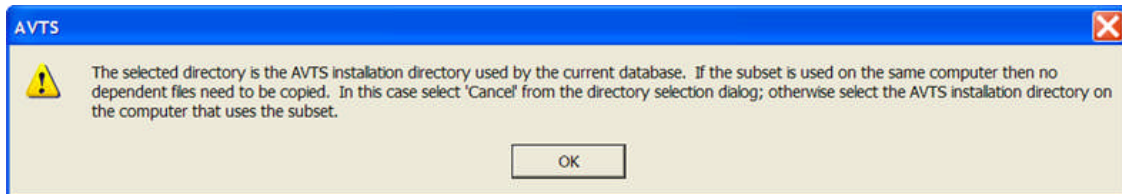
With the mapping selections chosen and the OK option selected, the message will be displayed. The process to complete the merge will follow the same remaining processes as that described in Database Utilities/Create Subset.

Select Cancel to abort the merge or OK to display the *Open* dialog to select the Directory to which the merged database is to be linked for the dependent files.



Selecting Cancel will open the new subset, replacing the database from which the subset was derived. This new subset (database) will have the name provided earlier by the user and is fully functional. Relays may be added or deleted. Tests may be added or deleted. Devices may be added or deleted. Other databases and subset may be merged using the Database Utilities Merge Subset option. The new subset will also be able to be MERGED back into the original database to update the original with the new information. Note that the Subset can be part of or all of the original database.

The Select Directory option will create a *message dialog* informing the user that if the subset is placed in a directory different than the parent AVTS database, dependent files will need be handled. Dependent files may include those in the AVTS sub-directories Waveforms, DigToolImages, ConnectionImages, etc. Select OK to return to the previous dialog to select another or the original AVTS directory.



The use of Subsets allows the user to maintain a complete database for a portion or for the entire company on his computer or on a server. For large companies this process allows two or more users of AVTS to be at different locations testing relays and then “Merging” the separate Subsets to the original full database without any one merge over-writing the values of a previous merge by another Subset for a different Organization location.

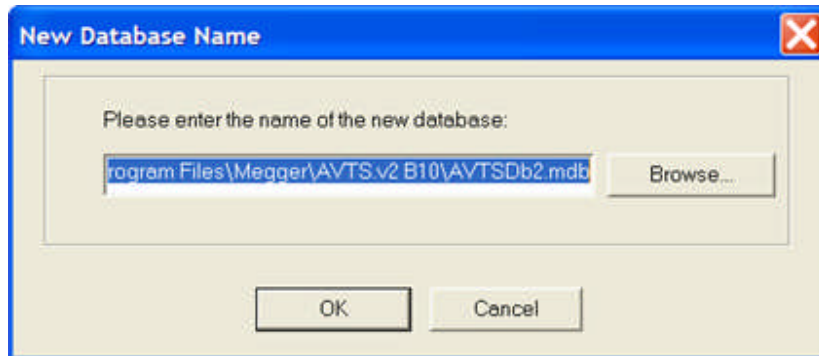
## Results Archive

On occasion, the user may desire to place the test results in an archive file and purge the active database of those historical results. Choose the Results Archive option from the Tools/Database Utilities menu for the archival process.

The original database, once the archive process is completed, will be void of any results that were part of the archive.

An Archive database will contain all the Organization levels and relays selected just as the original database. It may be opened separately in AVTS to view the historic data for study or print.

This selection will display the *New Database Name* dialog and a default database name. A different database name may be applied by typing the new name over the default name.

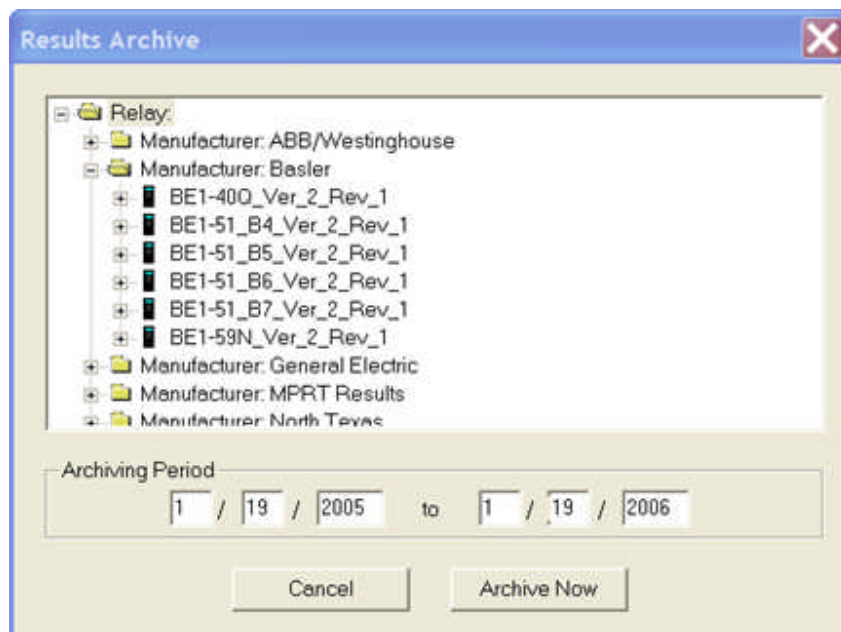


Select Cancel to abort the Results Archive process.

Select OK to display the Results Archive relay tree, which may be expanded by Organization Level.

Highlight with the mouse the "Relay" label, Organization Level or individual Relay that will have the results archived.

Select and enter the *Archive Period* by dates. These will be the results archived from the selected Organization Level. Should the date filter be of a date range not encompassing the entire date range of all the results within the active database, only the test results within the date filter range will reside in the new archive.





The new archived database will be of the given name and will be identical to the active database and contain all the results within the archived date range. The archive database may be opened with AVTS and reviewed at any future time.

The relay modules contained in the archive with the test results will allow the user to relate the results to the associated relay and view the test construction, relay settings and other pertinent data that were used during the test for the results.

**Note:** The archive database will have all relay modules that are associated with the filtered test results. For historical purposes, the archive database has all information about the root of the results and the relay modules may be used like any other database.

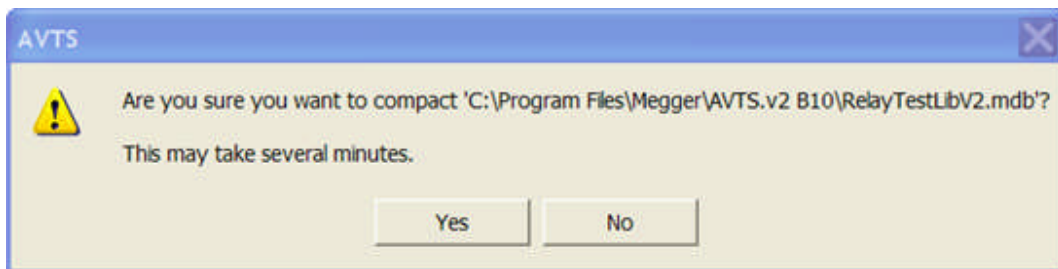
## Compact Database

During the process of creating relay modules, creating tests, modifying tests in the Settings, Test Editor, Monitor Editor, and Connections Editor tabs, and the creating, deleting and modifying of any portions of the database, there will be discontinuities of the database structure. The database may also be of an unnecessary larger size.

The Compact Database function will allow the user to reduce the size of the database. As a result, AVTS may operate more efficiently.

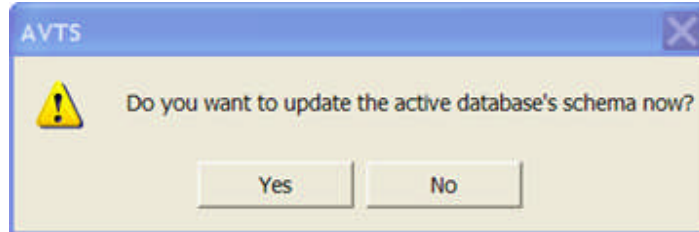
This action is very similar to that of the Windows Disk Defragmenter function used in reorganizing and reducing the active area of a computer hard disk drive.

When selected, the compact will display a *message dialog* for the user to assure whether or not he desires to do so.



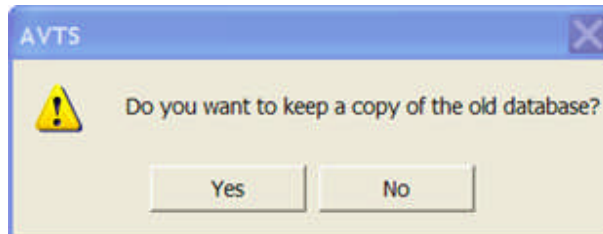
## Update

The Update function will allow the user to update the “schema” of the active database when opened in a later version of **AVTS**. Selection will open the *message dialog* to verify.

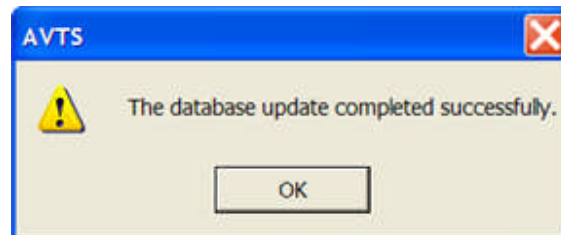


If **No** is selected, the Update process will be aborted and returned to the open **AVTS**.

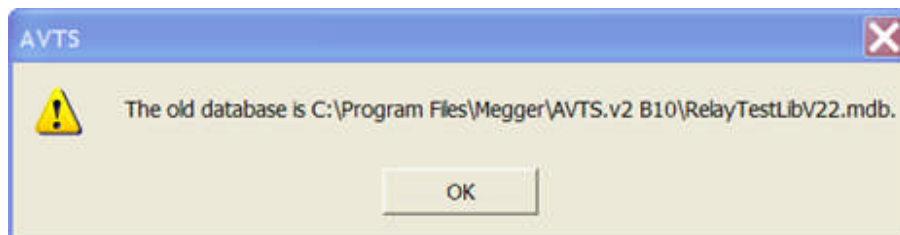
If **Yes** is selected, the database is updated. Then a *message dialog* will be displayed allowing the user to maintain a copy of the original database.



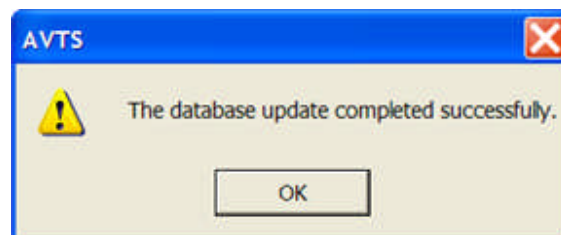
If **No** is selected, the old database will be discarded and no longer available and the successful *update message* is displayed.



If **Yes** is selected, **AVTS** saves the old database with a default path and file name with a *verification message*.



Select **OK**, and the successful *update message* is displayed.



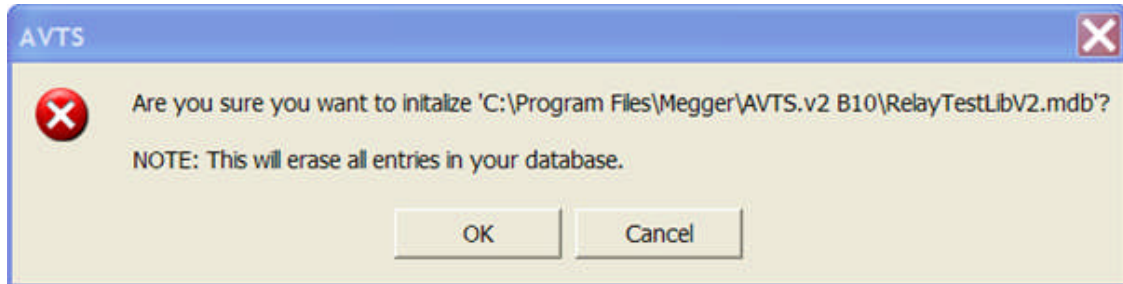
AVTS Help 092607

Select **OK** to complete the Update process to return to **AVTS** and the new active updated database.

## Initialize

The Initialize database function process is one that should be selected only if the database is to be completely purged of all data.

Select the Initialize option and the erase all database entry *WARNING message* is displayed.



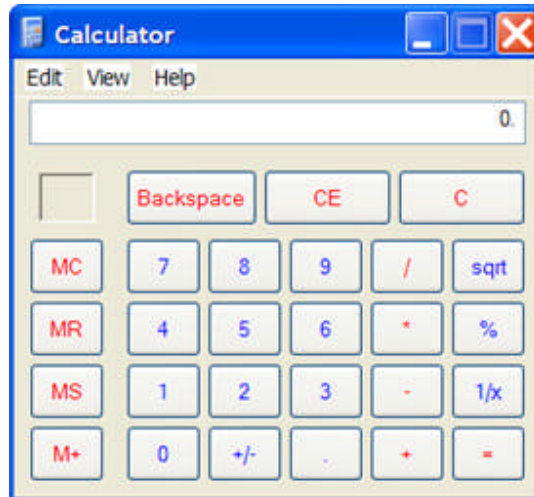
Select Cancel to abort the Initialize process and return to the AVTS active database.

Note: Selection of OK is **IRREVERSIBLE** and the data base is reduced to the size of a newly created database and completely void of any data. The process will end and will return to the AVTS active database with no Devices, Relays, Tests and MPRT tabs entries.

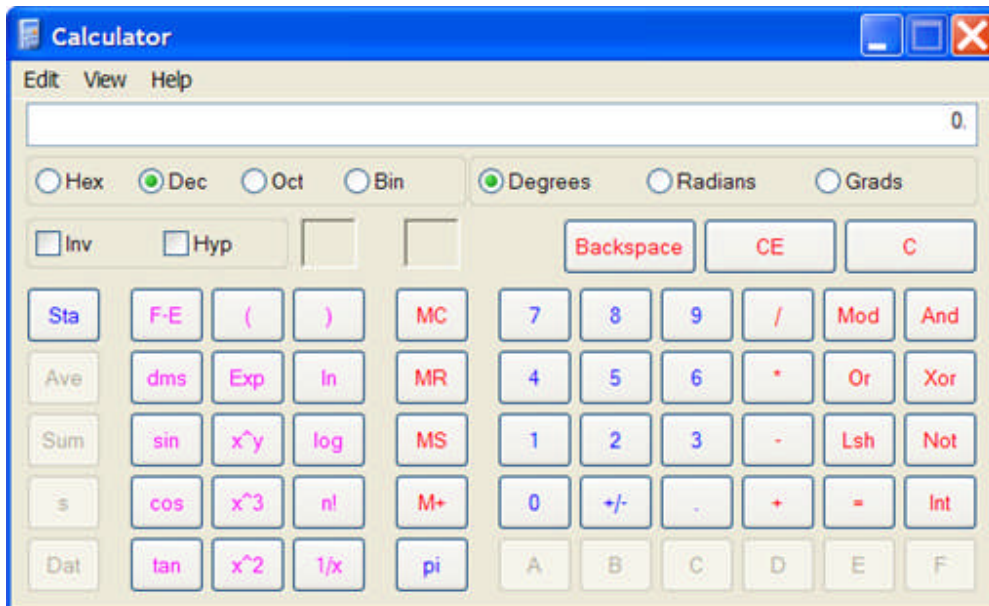
## Calculator

The path for this tool to be called is set in the General tab of the Tools/Options menu item.

The Tools Calculator is opened with the Microsoft Windows *Standard Calculator* displayed.



Select Scientific from the View menu list will change the called display to the Microsoft Windows *Scientific Calculator*.



## **Fault Calculator...**

## **Fault Calculator...**

The **Fault Calculator...** is based on a radial line from a relay location and a source impedance. All fault types are represented. A fault solution will display the line parameters and line-to-line voltages.

Fault Calculator topic defines how to use the Fault Calculator.

Fault Calculation topic defines the symmetrical components solutions for all fault types.

## Fault Calculator

The **Fault Calculator dialog** box allows the user to automatically calculate the fault quantities (phase-to-neutral and phase-to-phase voltages, and phase currents) from a given set of parameters. A fault is calculated based on equations for symmetrical components derived by Dr. Fortescue.

Line Voltage (p-n):		69.28	
Fault Type:	A-G		
Fault Settings			
	Magnitude	Angle	
Line Z:	3	80	
Relay Volts	51.96	0	
Relay Amps	25.98	80	
Source Z:	1	80	
Z0/Z1	0		
Three Phase			
Va	51.96	0	
Vb	65.381	113.41	
Vc	65.381	246.58	
Ia	25.98	80	
Ib	0	0	
Ic	0	0	
Vab	98.358	322.41	
Vbc	119.99	90.000	
Vca	98.358	217.58	

The Fault Calculator allows input variables for:

- Line Voltage (p-n),**
- Line Z** with angle,
- Relay Volts** with angle,
- Relay Amps** with angle,
- Source Z** with angle, and
- Z0 /Z1.**

**Note:** The System Z0 /Z1 Impedance Ratio is applied to both the Source Z and the Line Z for all faults involving ground.

Follow the listed steps to calculate the fault quantities:

1. Enter the normal voltage for the unfaulted Line Voltage (p-n)
2. Choose the desired type of fault from the drop down list box.  
Options are **A-G, B-G, C-G, AB, BC, CA, AB-G, BC-G, CA-G,** and **ABC.**
3. Edit the Line Z for desired magnitude and positive angle leading.
4. Edit the Source Z for the desired magnitude and positive angle leading.
5. Edit the Relay volts for desired magnitude and positive angle lagging.

**Note:** The Relay Amps magnitude and angle are calculated on the fly.

6. If necessary for current module output limitation, edit the Relay Amps magnitude and positive angle lagging.
7. Edit the zero sequence impedance ratio Z0 /Z1. This is linearly equivalent to  $k_0$  and is defined in the Fault Calculation topic.
8. Click Calculate or Tab out of the edit field to complete a calculation based on the inputs. The calculated voltages and currents will be displayed in the corresponding voltage and current fields.

**Note:** Click or select OK to close the Fault Calculator dialog box. The OK selection will not calculate nor apply the fault to the Fault Sequence dialog box.



Given the parameters of the fault condition, the fault calculation is determined by the fault type. The equations employed for fault calculation are provided in the Fault Calculation topic for reference.

## Equation for Fault Calculator

Given the parameters of the fault condition, the fault calculation is determined by the fault type (i.e. A-G, BC-G, ..... ) and the test hook-up. The equations employed for fault calculation are provided reference.

Assumptions:

The fault calculations are developed to recreate system fault conditions utilizing symmetrical components for the derivations.

$Z_1 = Z_2$  and  $\frac{Z_0}{Z_1} = K =$  system impedance ratio. The  $Z_0 = KZ_1$  equality is substituted for  $Z_0$  in the zero sequence network when faults involving ground are calculated.

$$Z_{0S} \angle \Phi_{0S} + Z_{0L} \angle \Phi_{0L} = K(Z_{1S} \angle \Phi_{1S} + Z_{1L} \angle \Phi_{1L}) = K(Z_S \angle \Phi_S + Z_L \angle \Phi_L)$$

All calculations are in engineering angle sense of positive displacement counterclockwise.

The fault calculator requires **positive angles leading** for **Line Z** and **Source Z**. After all calculations for a particular set of parameters with a chosen fault type are completed in engineering sense, the Relay volts and Relay amps are displayed as **positive angles lagging** for application to the PULSAR test equipment. The application to the MPRT depends whether the phase relationships are set for **lead** or **lag**.

Three Phase Fault equation ( A - B - C example)

$$I_F = \frac{V_{LG} \angle \Phi_{LG}}{Z_S \angle \Phi_S + Z_L \angle \Phi_L}$$

$$V_R = \frac{Z_L \angle \Phi_L}{Z_S \angle \Phi_S + Z_L \angle \Phi_L} V_{LG} \angle \Phi_{LG}$$

Line to Ground Fault Equation ( A - G example)

$$I_F = \frac{3V_{LG} \angle \Phi_{LG}}{(2 + K)(Z_S \angle \Phi_S + Z_L \angle \Phi_L)}$$

$$V_R = [V_{LG} \angle \Phi_{LG}] - \left[ \frac{I_F \angle \Phi_F}{3} (2 + K) Z_S \angle \Phi_S \right]$$

Phase to Phase Fault Example ( B - C example)

$$I_B = \frac{V_{LG} \angle \Phi_{LG}}{2(Z_S \angle \Phi_S + Z_L \angle \Phi_L)} (a^2 - a)$$

$$I_C = \frac{V_{LG} \angle \Phi_{LG}}{2(Z_S \angle \Phi_S + Z_L \angle \Phi_L)} (a - a^2)$$

$$V_{RB} \angle \Phi_{RB} = (V_{LG} \angle \Phi_{LG}) a^2 - \left[ \frac{V_{LG} \angle \Phi_{LG} Z_S \angle \Phi_S}{2(Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right] (a^2 - a)$$

$$V_{RC} \angle \Phi_{RC} = (V_{LG} \angle \Phi_{LG}) a - \left[ \frac{V_{LG} \angle \Phi_{LG} Z_S \angle \Phi_S}{2(Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right] (a - a^2)$$

Phase to Phase to Ground Fault Equation ( B - C - G example)

$$I_B = \left\{ \frac{V_{LG} \angle \Phi_{LG}}{\left[ 1 + \frac{K}{(1+K)} \right] (Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right\} \left[ a^2 - \left( \frac{K}{1+K} \right) a - \left( \frac{1}{1+K} \right) \right]$$

$$I_C = \left\{ \frac{V_{LG} \angle \Phi_{LG}}{\left[ 1 + \frac{K}{(1+K)} \right] (Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right\} \left[ a - \left( \frac{K}{1+K} \right) a^2 - \left( \frac{1}{1+K} \right) \right]$$

$$V_{RA} = (V_{LG} \angle \Phi_{LG}) - \left\{ \frac{(V_{LG} \angle \Phi_{LG})(Z_S \angle \Phi_S)}{\left[ 1 + \frac{K}{(1+K)} \right] (Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right\} \left\{ 1 - \left[ \frac{K}{(1+K)} \right] - \left[ \frac{K}{(1+K)} \right] \right\}$$

$$V_{RB} = (V_{LG} \angle \Phi_{LG}) a^2 - \left\{ \frac{(V_{LG} \angle \Phi_{LG})(Z_S \angle \Phi_S)}{\left[ 1 + \frac{K}{(1+K)} \right] (Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right\} \left\{ a^2 - \left[ \frac{K}{(1+K)} \right] a - \left[ \frac{K}{(1+K)} \right] \right\}$$

$$V_{RC} = (V_{LG} \angle \Phi_{LG}) a - \left\{ \frac{(V_{LG} \angle \Phi_{LG})(Z_S \angle \Phi_S)}{\left[ 1 + \frac{K}{(1+K)} \right] (Z_S \angle \Phi_S + Z_L \angle \Phi_L)} \right\} \left\{ a - \left[ \frac{K}{(1+K)} \right] a^2 - \left[ \frac{K}{(1+K)} \right] \right\}$$

**3-Phase Wye:**

Fault will be calculated assuming that the test set is to produce the voltages and currents that a wye connected relay would see. Since non-existent generators are ignored this hookup type can also be used to test single phase relays using the A-G fault type when a three phase test set is not available.

## SS1 Converter...

## SS1 Converter...

In the Tools menu, click on the **SS1 Converter** to open the *SS1 Converter* dialog with default settings.

The screenshot shows the AVTS SS1 Converter dialog box. It features a blue title bar with the text "AVTS SS1 Converter" and a close button. Below the title bar is a "File:" field. The main area is divided into three sections: "Simulations", "States", and "Vectors", each with a table. The "Simulations" table has columns for Test#, Name, Battery Voltage, Debounce Time, Output Freq, Num Channels, and Num States. The "States" table has columns for State#, Name, Time Type, and Duration. The "Vectors" table has columns for Vector#, Name, Freq. Mult., Magnitude, and Phase. At the bottom, there is an "Adjust Time Factor" section with input fields for "MPRT" and "Pulsar" for "Pre-Fault Time Adjustment", "Fault Time Adjustment", and "Post-Fault Time Adjustment". There are also buttons for "Save as Defaults", "Load Defaults", "Create Script File", "View Raw SS1 File", "View Script File", and "Done".

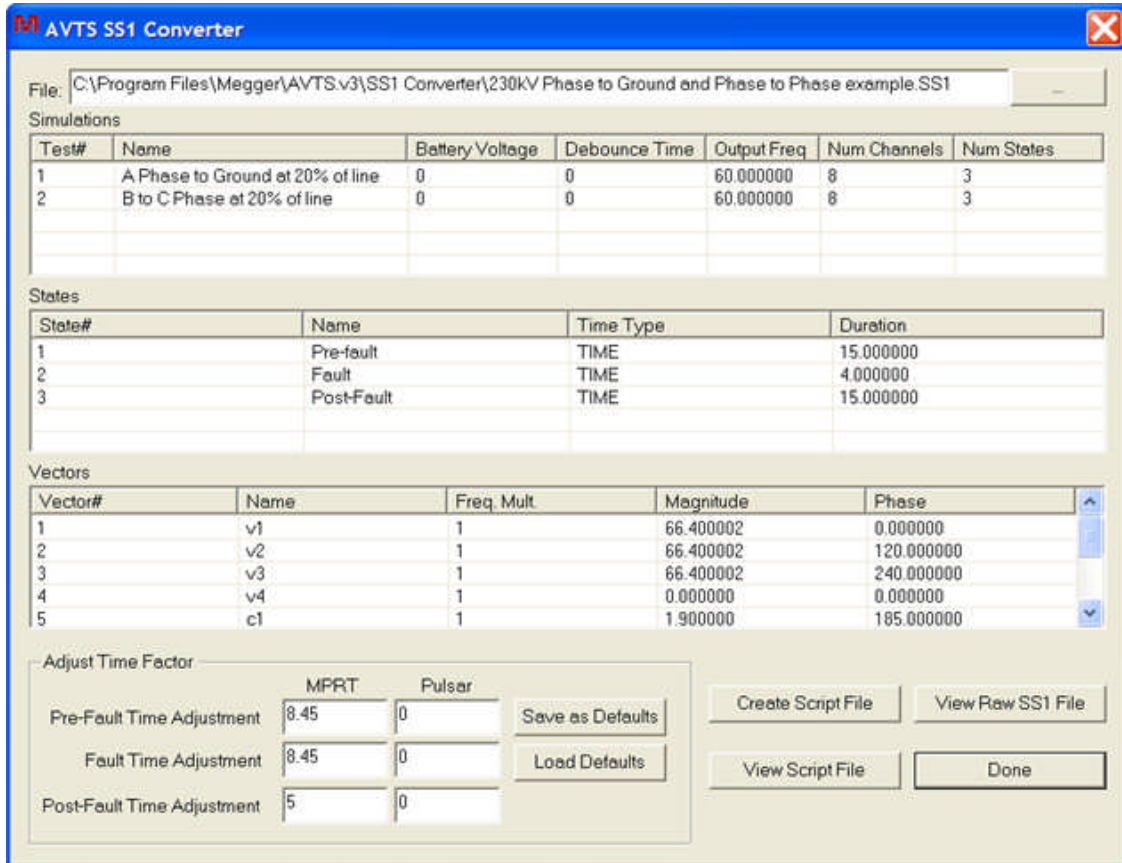
Note: The AVTS SS1 Converter is a self supported executable program resident in the AVTS SS1 subfolder. The program may be used for converting several SS1 files without having to open AVTS.

The SS1 Converter Dialog topic describes the use of this option.

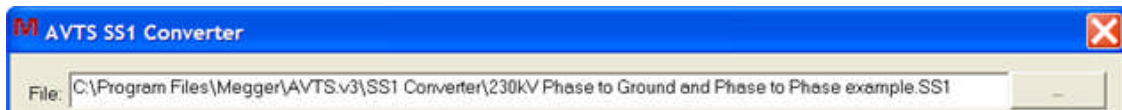
The SS1 File Example topic demonstrates use of a converted SS1 file to a 1-Touch file in the AVTS Test Editor Tab.

## SS1 Converter Dialog

The dialog is divided into five areas and are described with an *imported SS1 file*:

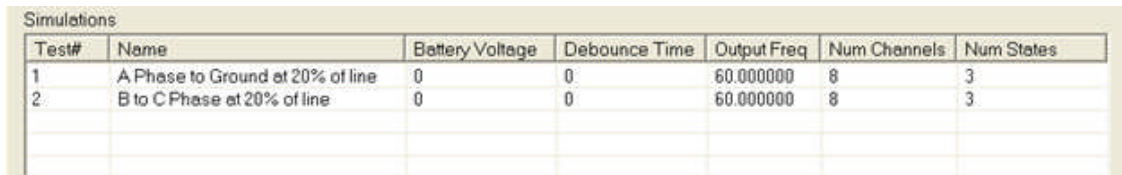


*File: edit field* is to locate, using the browse button, a SS1 file to be converted to an AVTS 1-Touch script file with a \*.m1t extension.



Note: An attempt to change any of the

*Simulations area* is a list of the Test types labeled when initially saved as a SS1 file.



*States area* is a list of the States to be presented for each of the Simulations by Test #.

State#	Name	Time Type	Duration
1	Pre-fault	TIME	15.000000
2	Fault	TIME	4.000000
3	Post-Fault	TIME	15.000000

*Vectors area* is a list of the Voltages and Currents to be exercised in the Simulations.

Vector#	Name	Freq. Mult.	Magnitude	Phase
1	v1	1	66.400002	0.000000
2	v2	1	66.400002	120.000000
3	v3	1	66.400002	240.000000
4	v4	1	0.000000	0.000000
5	c1	1	1.900000	185.000000
6	c2	1	1.900000	305.000000
7	c3	1	1.900000	65.000000
8	c4	1	0.000000	0.000000

Note: Any attempt to change any of the fields in the Simulations, States or Vectors areas will result in an Abort of the SS1 Converter. Any edits to any of the text files must be made in the \*.ss1 or \*.m1t file. A complete new file may be made from a \*.ss1 or \*.m1t file with different values using Notepad and saved as a new file name. If a \*.ss1 file is used for edit of test values, it will need to be converted to a \*.m1t file.

*Adjust Time Factor area* contains three edit fields for time adjustments to the MPRT or Pulsar test sets. The adjustments are to manipulate time in milliseconds for synchronization between test sets involved in playback of the file Tests and States.

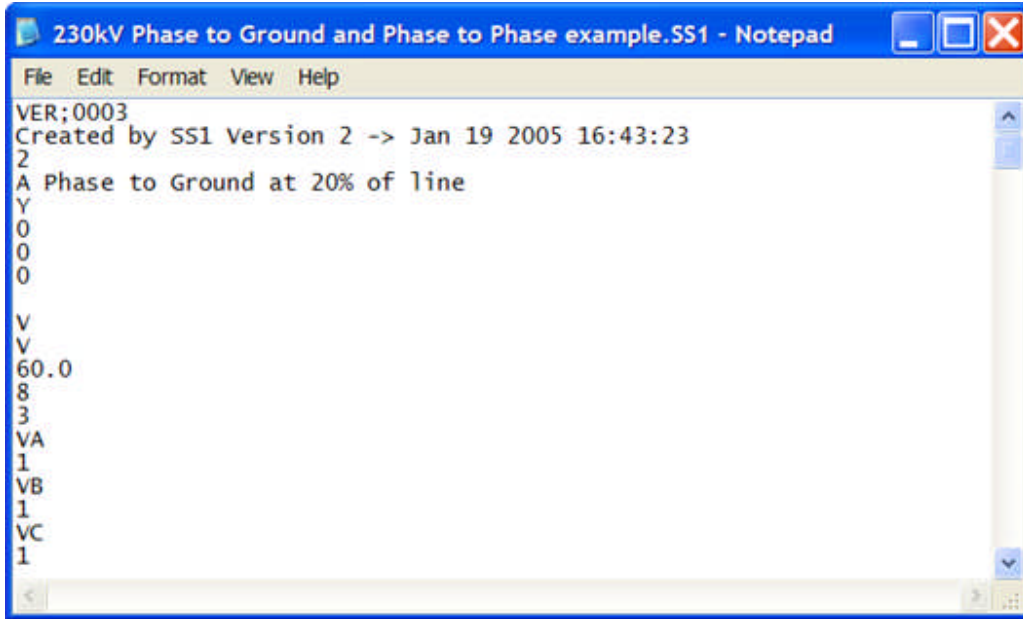
	MPRT	Pulsar	
Pre-Fault Time Adjustment	8.45	0	Save as Defaults
Fault Time Adjustment	8.45	0	
Post-Fault Time Adjustment	5	0	Load Defaults

Time Adjustment edit fields are for necessary skewing of the Prefault, Fault and Post Fault States. These adjustments may be made for the MPRT and Pulsar test sets and saved as Defaults. Default memory is limited to only the values last saved and only that one set is available for repopulating the edit fields with the Load Defaults button.

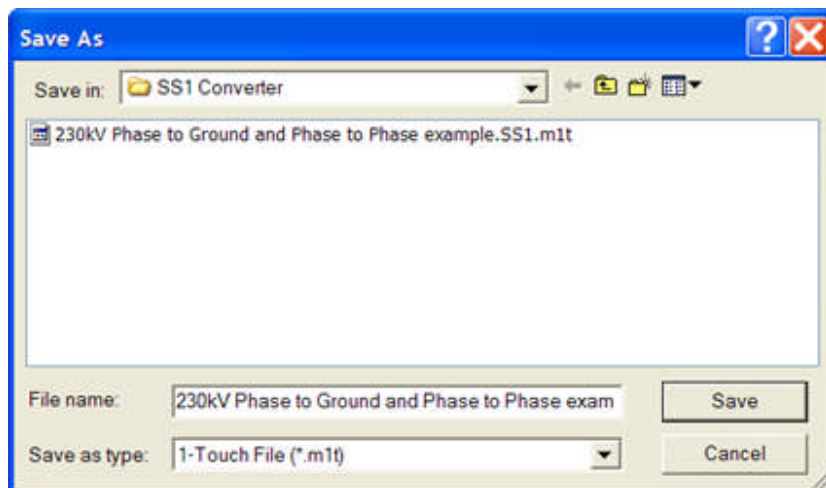
*View Files area* is for viewing each of the original and generated files. All files are displayed as the default text file, typically in Windows Notepad or Wordpad.

Create Script File	View Raw SS1 File
View Script File	Done

*View Raw SS1 File* option will open Notepad and display the data of the SS1 file being viewed in the SS1 Converter dialog.



Create Script File option will open the Windows *Save As dialog*. A File name: must be entered in the edit field to save the converted file. The converted file will be given the 1-Touch File extension \*.m1t and saved in the SS1 Converter AVTS subfolder.



The Save operation in the Save As dialog will open the *Generated AVTS Script File* in Notepad for view. If other files have previously been converted and saved, their labels will appear in the list of files with the \*.m1t extension.



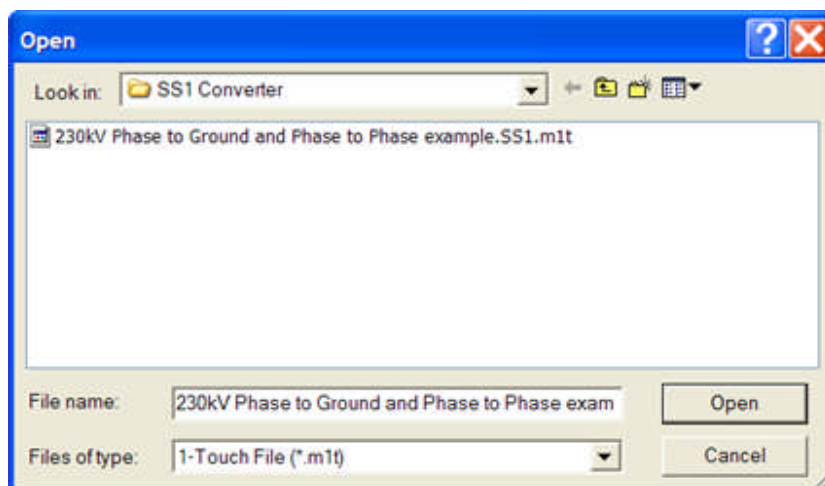
```

230kV Phase to Ground and Phase to Phase example.SS1.m1t - Notepad
File Edit Format View Help
' Generated AVTS Script File
Set AvtsDrv = CreateObject("AvtsDeviceDriverComponent.DeviceDriver")
AvtsDrv.OutputCommand "qc;"
QcStr = AvtsDrv.ReadUnit
Dummy = AvtsDrv.ReadUnit
Dummy = AvtsDrv.ReadUnit
QcStr = left(QcStr, 2)
if QcStr = "<G" then
    TimerStr="t03"
    ReStr = ""
    PreFaultTimeAdjust = 8.45
    FaultTimeAdjust = 8.45
    PostFaultTimeAdjust = 5.00
elseif QcStr = "VG" then
    TimerStr="t3"
    ReStr = "re,"
    PreFaultTimeAdjust = 0.00
    FaultTimeAdjust = 0.00
    PostFaultTimeAdjust = 0.00
else
    TimerStr="t3"
    ReStr = "re,"
    PreFaultTimeAdjust = 0.00
    FaultTimeAdjust = 0.00
    PostFaultTimeAdjust = 0.00
end if
MsgStr1 = "Choose the number of the simulation you want to run."+vbCrLf+_
"1 - A Phase to Ground at 20% of line"+vbCrLf+_
"2 - B to C Phase at 20% of line"
MsgBox "This 1-Touch Script was created from the file '230kV Phase to Ground and Phas
TestNum = InputBox( MsgStr1, "Select a simulation" )

if TestNum = 1 then
    ' Test Name: A Phase to Ground at 20% of line
    if MsgBox( "Running A Phase to Ground at 20% of line") = vbOk then
        MaxCurrents = InputBox("This simulation uses 4 Currents. How many of the Curr

```

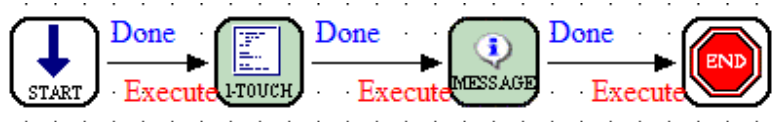
View Script File option will open the Windows *Open dialog*. The dialog opens with default to the SS1 Converter AVTS subfolder and display all the \*.m1t files resident in the folder. Select the file to be viewed so to place the file name in the edit field. Select Open to display the file in Notepad.



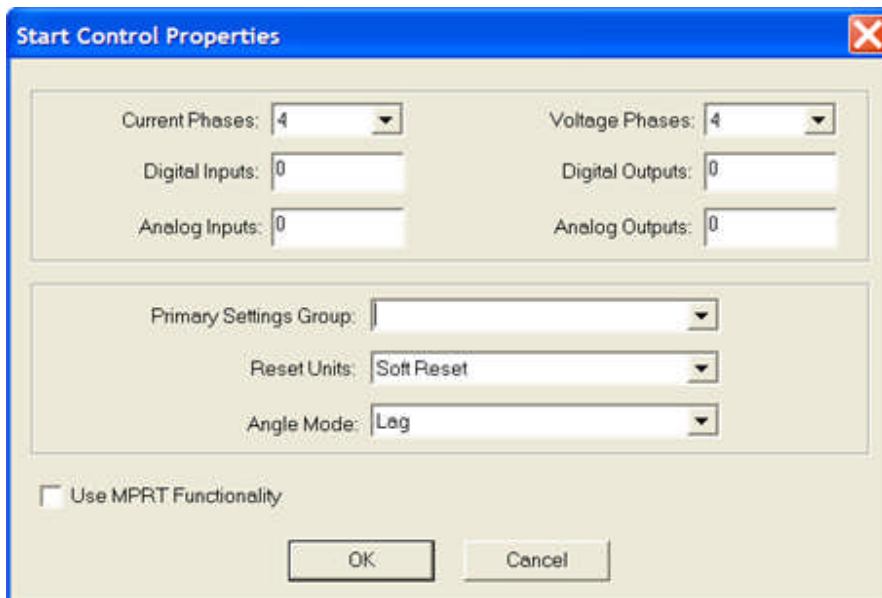
## SS1 File Example

The SS1 file converted to a 1-Touch file may be used with the 1-Touch Control in the Test Editor tab of an open Relay. The use may be a single test from Start Control to 1-Touch Control to another control and then to the End Control. Or the 1-Touch Control may be placed in a given test procedure to incorporate a play of a multi-State fault at some point during the test procedure.

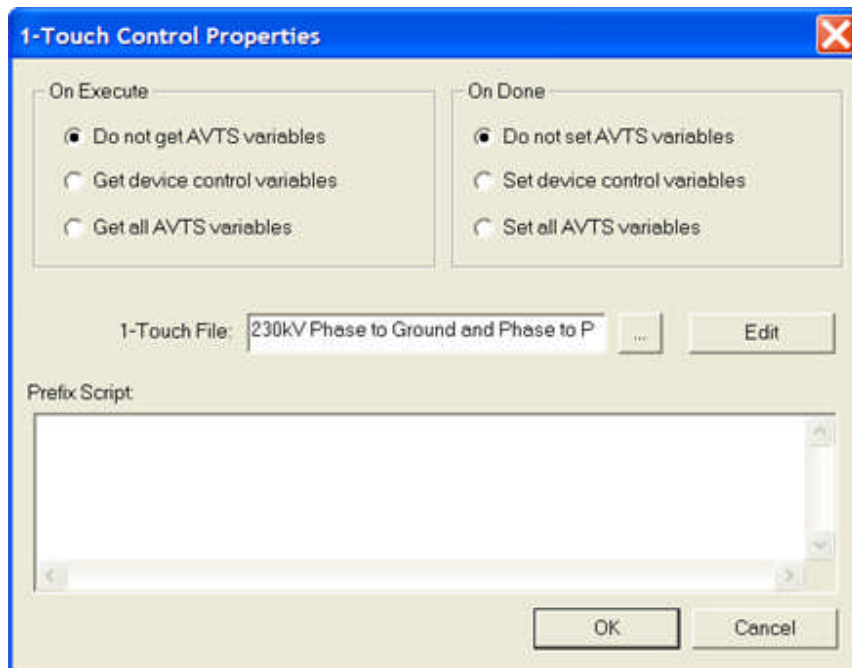
An *Example Test* is to use the converted file in a single step test using the 1-Touch Control. The Message Control is used to indicate the end of the test.



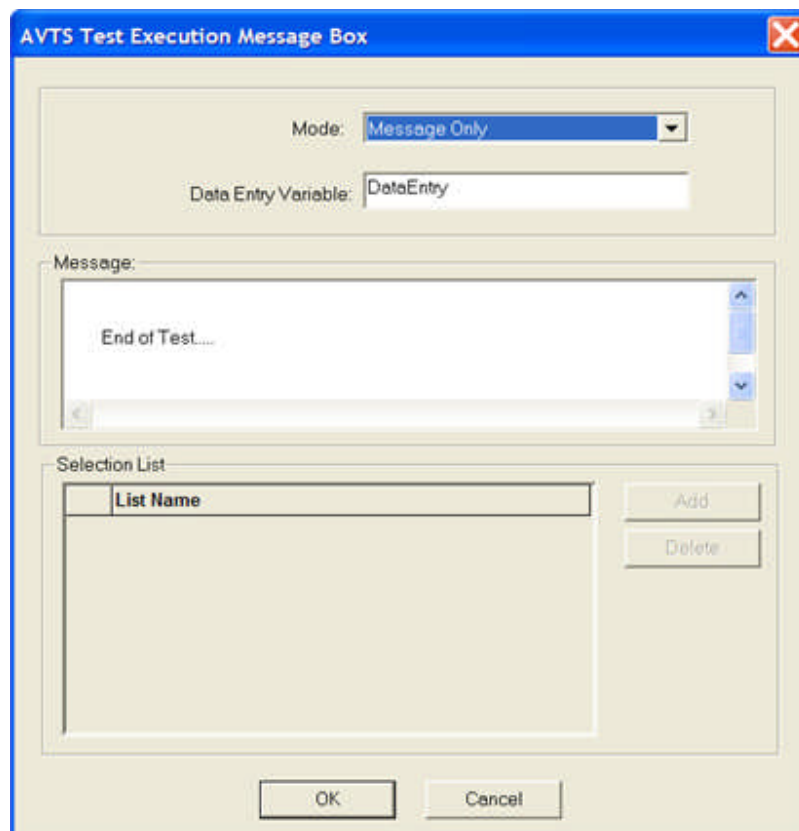
The *Start Control Properties* are shown to be set for use of four Voltage and four Current channels to agree with the number of channels labeled in the converted SS1 text file.



The *1-Touch Control Properties* are shown with the SS1 file converted to the called \*.m1t file.



The *Message Control Properties* are shown with the message to indicate the end of the test.



The *BASIC Tab* is shown with the \*.m1t file called by the 1-Touch Control.

```

TestNum = InputBox( MsgStr1, "Select a Simulation" )

if TestNum = 1 then

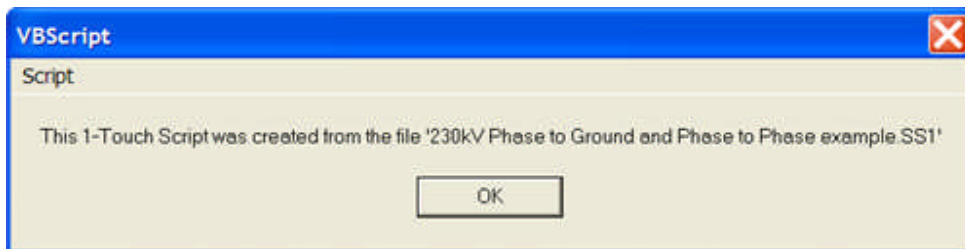
    ' Test Name: A Phase to Ground at 20% of line
    if MsgBox( "Running A Phase to Ground at 20% of line") = vb
MaxCurrents = InputBox("This simulation uses 4 Currents. Hc

MaxVoltages = InputBox("This simulation uses 4 Voltages. Hc

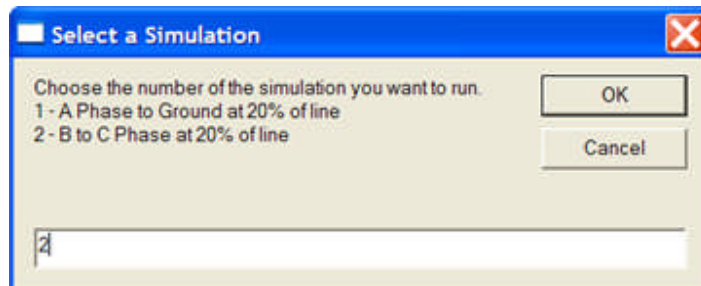
AvtsDrv.OutputCommand TimerStr+"sto,"
AvtsDrv.OutputCommand TimerStr+"cal,"
TimeAdjustment = PreFaultTimeAdjust
' State Name: Pre-fault
if 1 <= MaxVoltages then
    AvtsDrv.OutputCommand "v1,d0,a66.400,p0.000,f60.00000
End If
if 2 <= MaxVoltages then
    AvtsDrv.OutputCommand "v2,d0,a66.400,p120.000,f60.000
End If
if 3 <= MaxVoltages then
    AvtsDrv.OutputCommand "v3,d0,a66.400,p240.000,f60.000
End If
if 4 <= MaxVoltages then
    AvtsDrv.OutputCommand "v4,d0,a0.000,p0.000,f60.00000,
End If
if 1 <= MaxCurrents then
    AvtsDrv.OutputCommand "c1,d0,a1.900,p185.000,f60.0000
End If
    
```

Execution of the example test will produce several queries to the user. These are generated by the script file in the BASIC TAB.

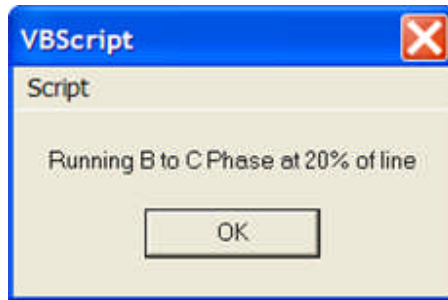
The *first generated dialog* is a 1-Touch message with the name of the file to be used during the test execution.



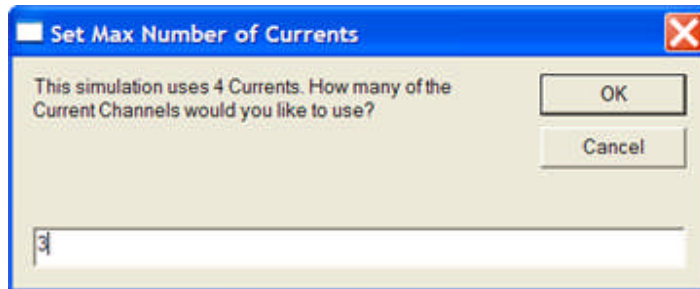
The example SS1 and the converted file to \*.m1t have two tests. Only one test may be run at any given test execution. The *second generated dialog* is for the option to select which test number to execute. Test #1 is for an "A Phase to Ground at 20% of line." Test #2 is for a "B to C Phase at 20% of line.



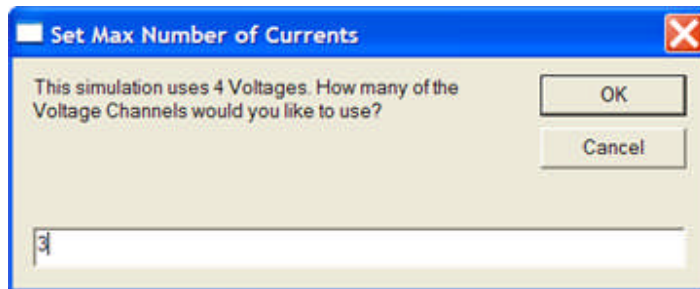
The *third generated dialog* is a message with the name of the selected Test to be executed.



The *fourth generated dialog* is for the option to select the number of Current Channels to be used by the test set during the test execution.



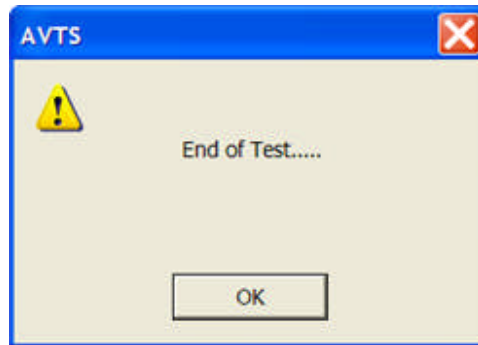
The *fifth generated dialog* is for the option to select the number of Voltage Channels to be used by the test set during the test execution.



The *sixth generated dialog* is an instruction message to hit the ENTER key to end the test.



The *seventh dialog* is generated by the Message Control to indicate the completion of and exit from the 1-Touch Control.



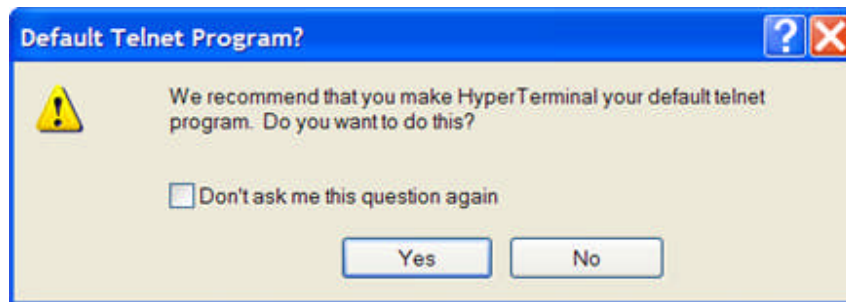
## Tools HyperTerminal

The path for the **Hyperterminal tool** is set in the General tab of the Tools/Options menu item.

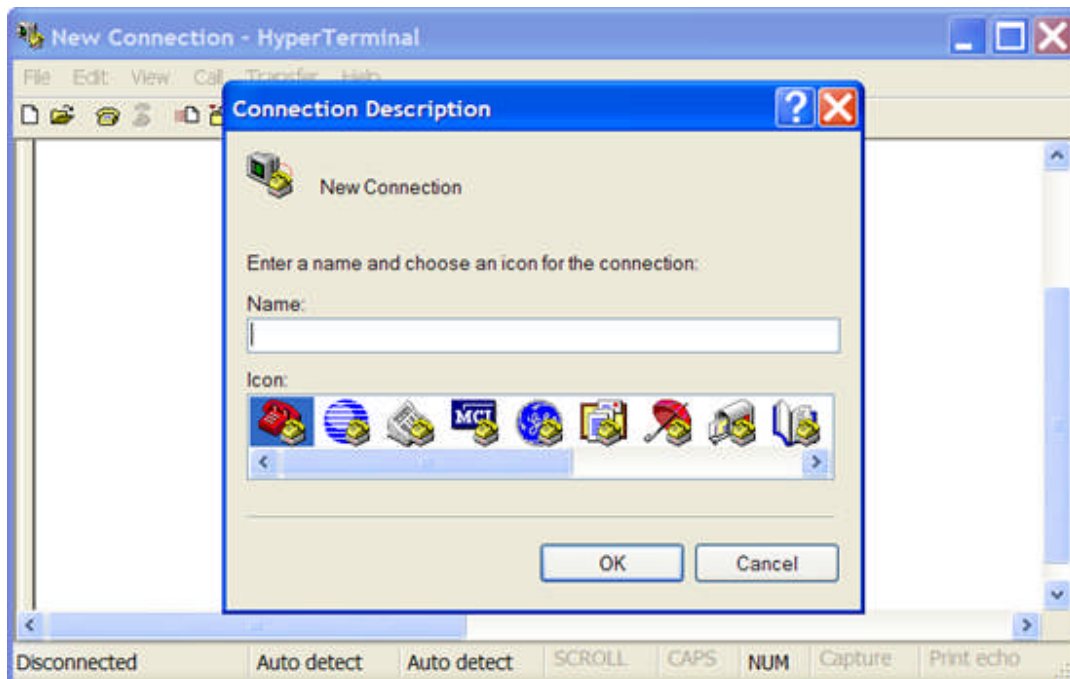
The typical Win 2000/ME/XP location for Hyperterminal is C:\Program files\Windows NT\hypertrm.exe.

Should an older Windows operating system be installed on the computer, the location path can be found by a search for the hypertrm.exe in Windows Explorer.

When first opened, a *Default Telnet Program?* message dialog will offer to make HyperTerminal the default telnet program. The option to do so should be reviewed with the responsible person in the network group of the user's company.

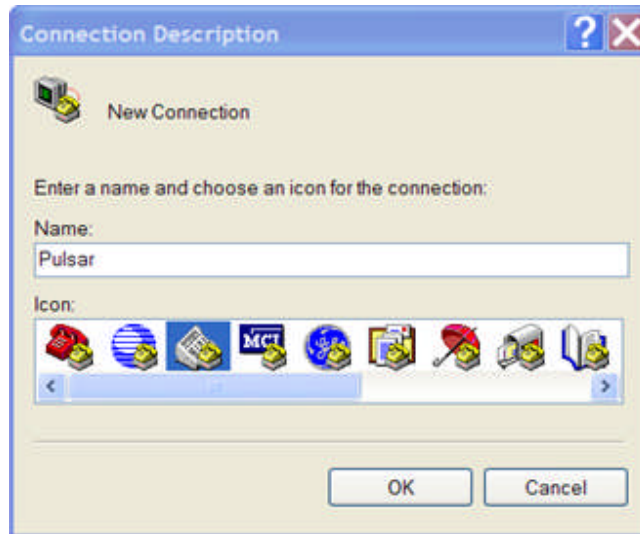


Whether Yes or No is selected, the New Connection - HyperTerminal dialog will open with the *Connection Description* dialog in view.



The Connection Description dialog is for to enter the name and choose on icon for a new connection. It may be useful for the user to create a connection name and icon for each of the communication links required by some relays. If the user selects Cancel without text entry and icon selection, the Connection Description dialog will exit. The user will then have the New Connection - HyperTerminal

An example of a Connection setup begins with an open HyperTerminal. The Connection Description will display. The *Name and Icon* have been typed and selected.



Click on OK and the *Connect To* dialog appears for selection to a telephone line by entering the telephone number to connect to a modem.

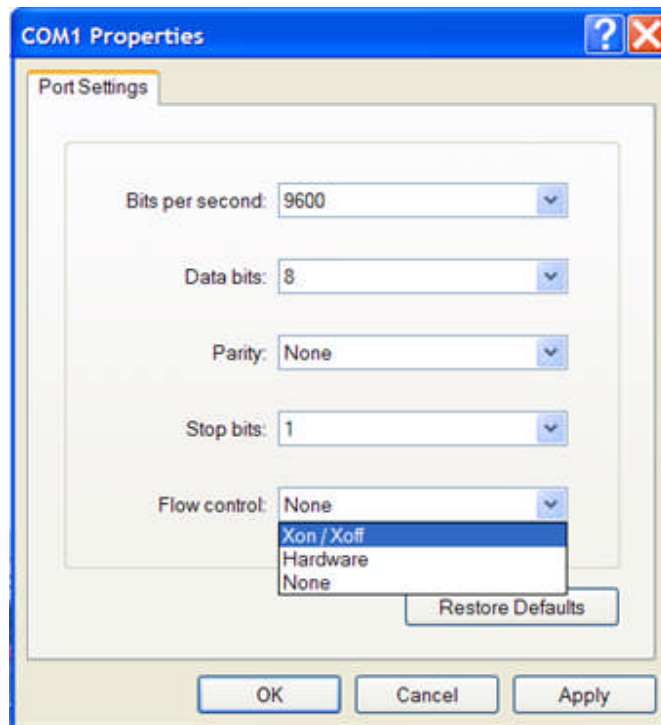


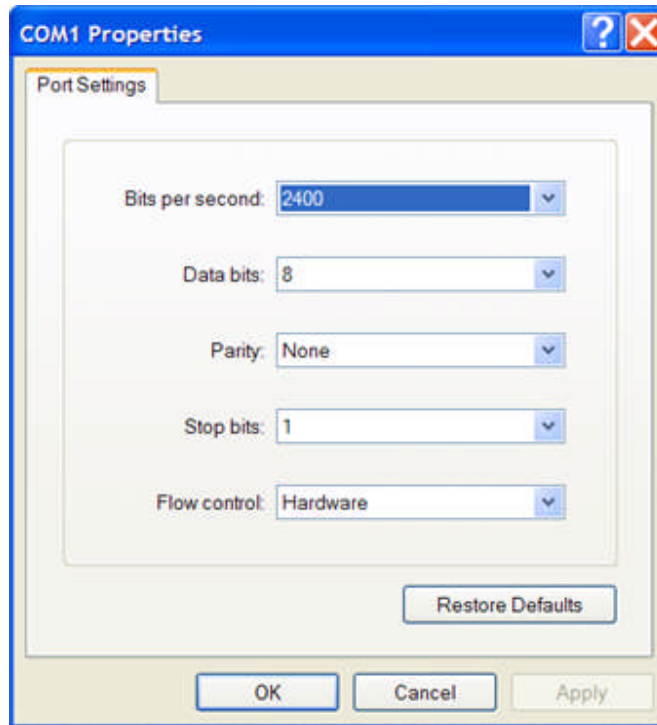
Or, selection of other available computer *communication ports* may be made using the Connect using drop list. In this example the COM1 port was selected as a typical connection to the Pulsar, MPRT or particular solid-state and microprocessor relays.



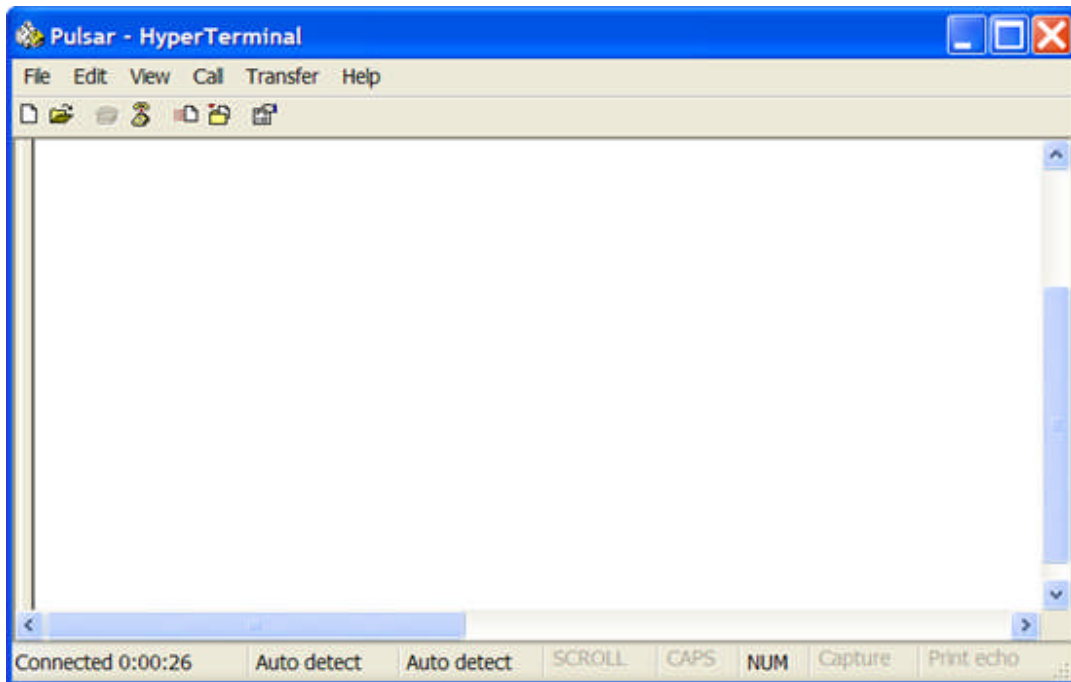


Click OK to view the settings for the selected communication port, exemplified by the *COM1 Properties* dialog. The Windows default settings will be shown for the first use of the new connection. For the equipment to be connected, reference to the equipment vendor's manual should be reviewed for selection of the desired settings for proper choices in the *Port Settings* dialog. The Pulsar has a maximum serial port response bit rate speed of 9600. The remaining settings are suggested as shown, except the Flow Control may be Hardware or Xon/Xoff, which is selected from the drop down list.

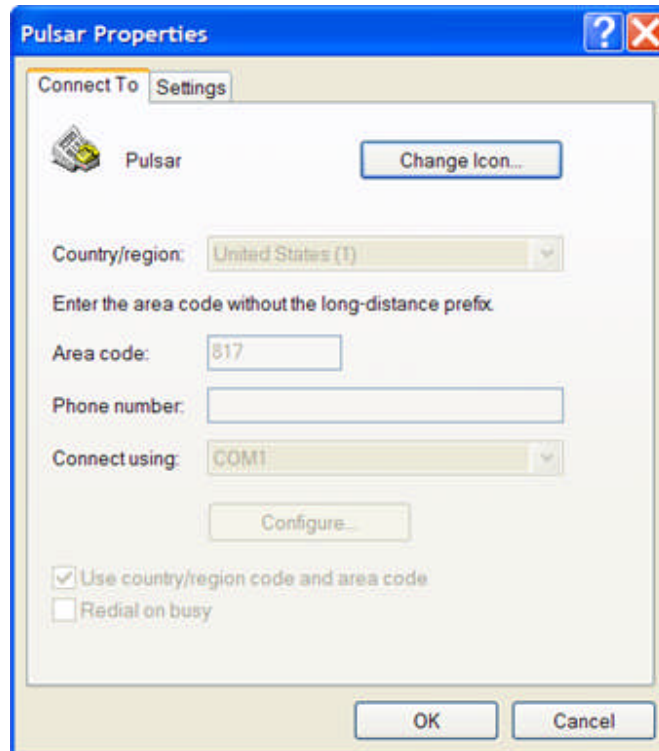




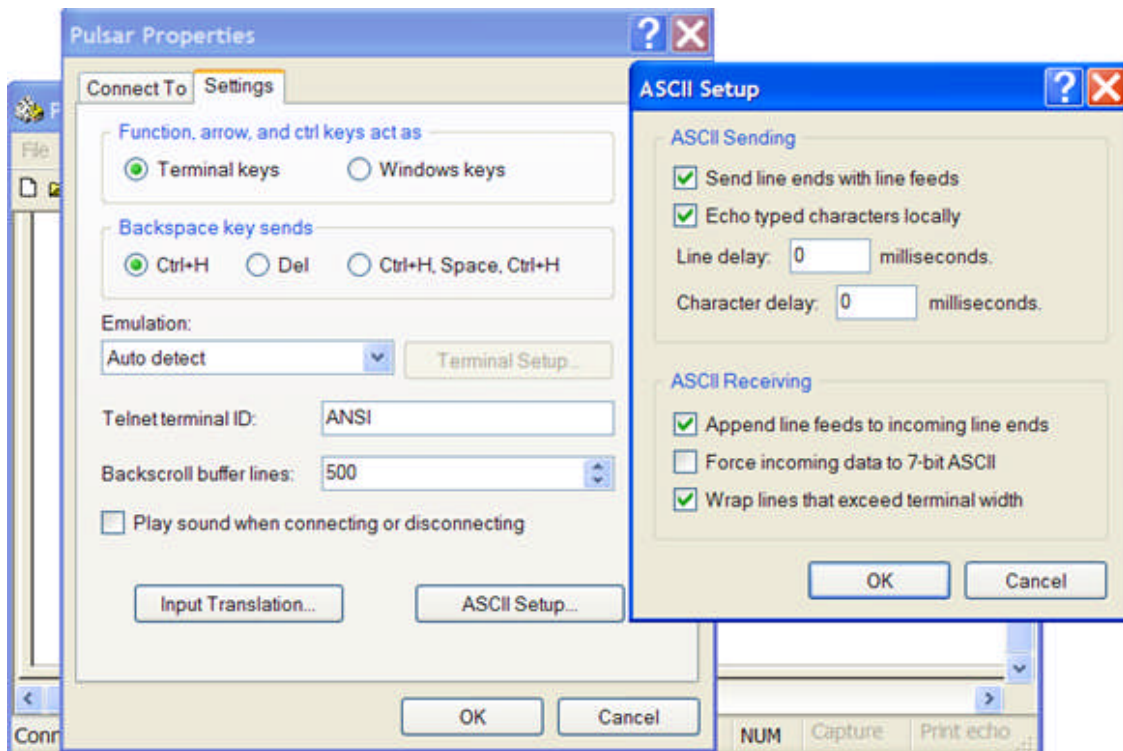
Select Apply to force the settings and then click OK to return the *HyperTerminal* dialog. Note that the dialog is active and communication is immediately available to the connected equipment. Activity is indicated by the scrolling time in seconds at the left end of the Status bar.



Prior to using the connection, select the Call/Disconnect menu item. Then select the File/Properties menu item to view the connection *Properties* dialog. The dimmed settings reflect those chosen in the initial start of the HyperTerminal.



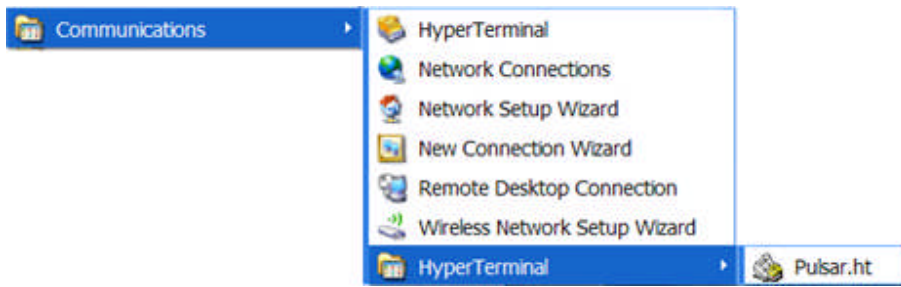
Select the *ASCII Setup* option to display that dialog. Typical settings for the ASCII Setup are shown.



Once the connection and properties have been saved, the connection may be recalled for future use by selecting that connection by name from the All Programs list in the Start selection from the computer desktop. From the All Programs list, select Accessories, then Communications to display another list. The top of the list will display HyperTerminal and at the bottom the new *HyperTerminal connection just created*.

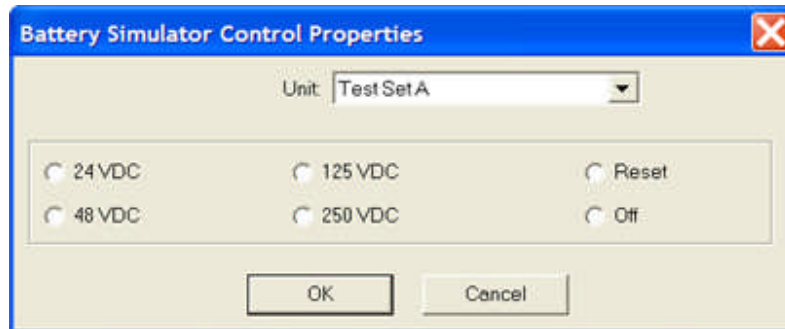
AVTS Help 092607

Expansion of this item will display all the specific connection frames that have been created and saved by the user.



## Tools Battery Simulator

The **Battery Simulator** is to control the Battery output function in the MPRT test set. A MPRT must be the selected Test Set in the Tools/Options/Devices tab to enable the Battery Simulator. The Battery Simulator to be included in a test procedure created in the Test Editor tab is defined in the Battery Simulator Control topic.



When used from the Main Menu Tools option, the Battery Simulator Control Properties dialog becomes visible for selecting the desired output on the MPRT. This control is online active with the MPRT.

The Unit edit field allows selection of either Test Set A or Test Set B. The selected test set must be active in the Tools Options Devices dialog by unchecking any Simulate function in this dialog.

The use is to select the desired DC voltage level of output for the MPRT and then select OK to make the output active.

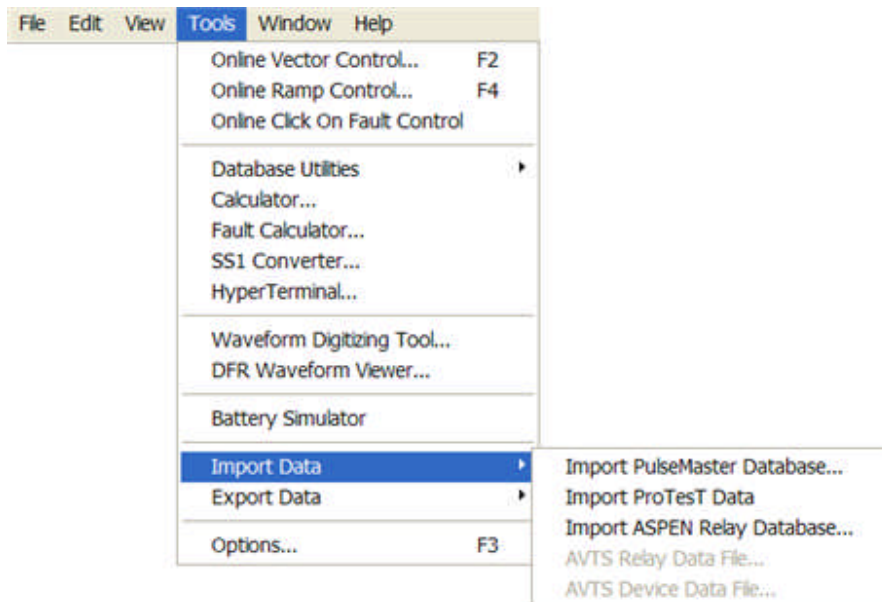
The Off option will turn the MPRT output OFF.

The Reset function will void a voltage level selection, leaving a voltage level options unchecked and send a **batre**; the MPRT (battery reset, execute) command.

## Tools Import Data

### Tools Import Data

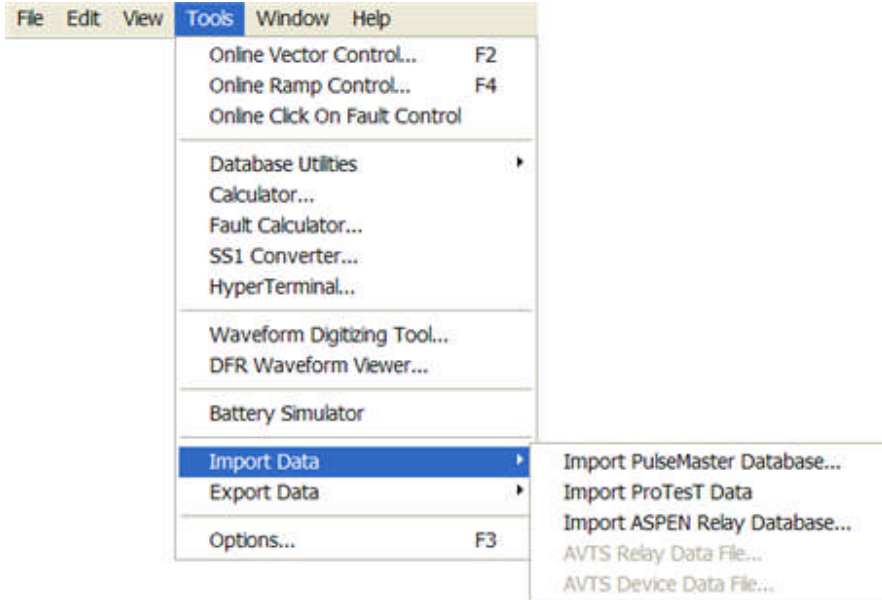
The available options in the Tools *Import Data* menu for importing AVTS data are:



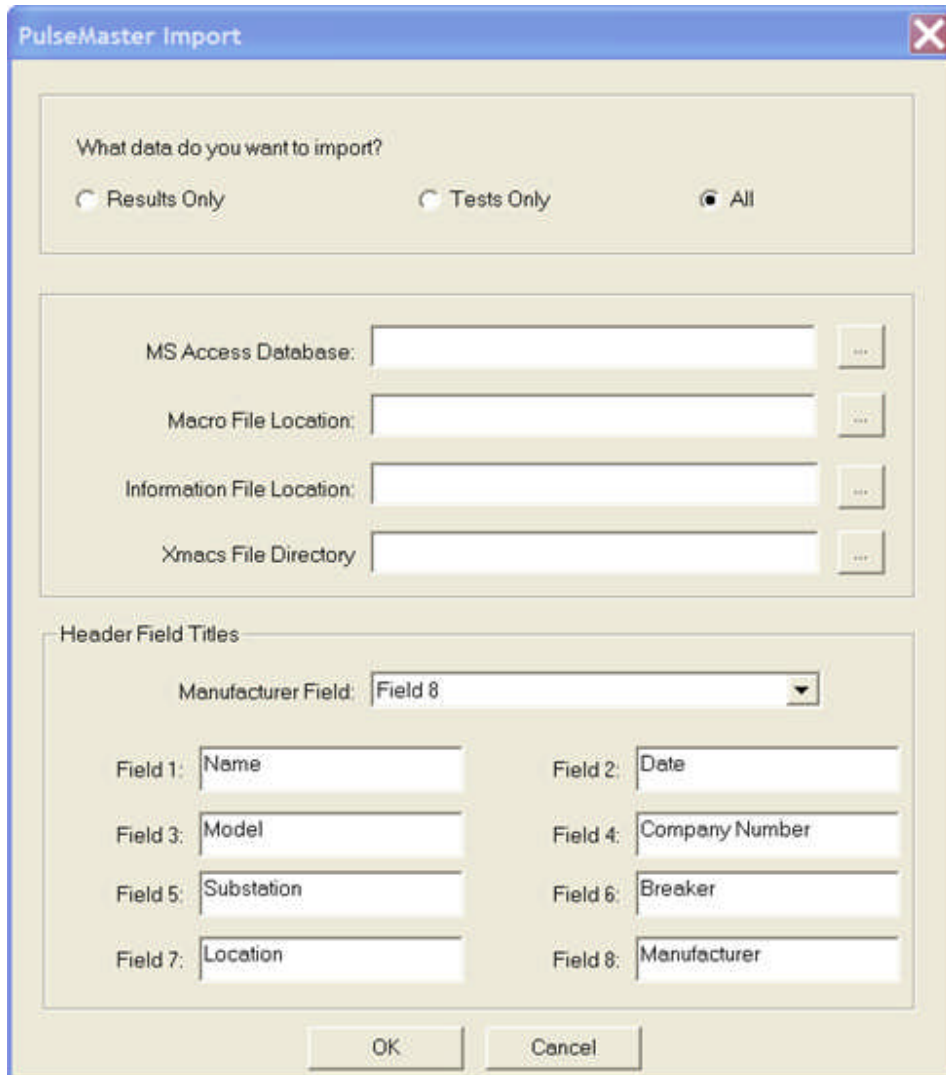
- Import PulseMaster Database...
- Import ProTest Data
- Import ASPEN Relay Database...
- AVTS Relay Data File...
- AVTS Device Data File...

## Import PulseMaster Database

The *Import PulseMaster Database...* option opens the PulseMaster Import dialog. Prior to importing a PulseMaster Database, it must have been converted from the original dBase IV database format to a Microsoft Access 97 database. The database will then meet all Access database requirements, including any version newer than Access 97, for import to AVTS.



Select the Import *PulseMaster Database...* to begin the process.



Prior to the conversion, the user can choose to create a PulseMaster database with a single relay or with many relays or the entire database. The methods for filtering particular relays from the database are provided in the PulseMaster Help files.

A choice of importing the test Results Only, Tests Only or both results and tests with All. If the imported database is to be used for testing during routine maintenance, the Tests Only option may be most satisfactory so as to minimize the importation.

The following four edit fields have a browse button to the right of each field.

The browse allows ready search for:

- the MS Access Database resident in the folder that contains the converted database,
- the Macro File Location folder, usually C:/PM2/AVOMAC having \*.MTM, \*.MTG, \*.MTP, \*.MTT file extensions,
- the Information File Location, usually C:/PM2 having \*.INF file extensions, and
- the Xmacs File Directory, usually C:/PM2/XMACS having \*.EXE, \*.C, \*.H, \*.LIB, \*.MS, \*.BAT file extensions.

Only the files types necessary for successfully executing a test need be imported in the file location/directory fields. The other fields may be left blank.



The Header Field Titles shown are by default and replicate the default fields in the original Headers of the PulseMaster database. If the fields were changed in the original database before conversion, these fields will need to be edited in this dialog for agreement to the database names of the fields in the PulseMaster database. These fields with the associated test information will also appear in the printed/viewed results of a test executed in AVTS.

## Import ProTest Data

### Import ProTest Data

The **Import ProTest Data** menu option is to place in the open AVTS database a copy of a Doble ProTest database. The purpose is to allow the previous users of Doble equipment to import their ProTest database into AVTS. Once the database is imported to AVTS, ProTest test results, filtered by dates on import, are available for view and print.

**Note:** This feature will only Import the test results of the ProTest into the AVTS database. The test procedures or relay settings of the relays present in ProTest will not display in AVTS. However, by purpose, the Import allows the save and print of the Test Results with the settings that may also be present in the results.

The following topics describe the Import Process and an Example of Results Exported to a Word document

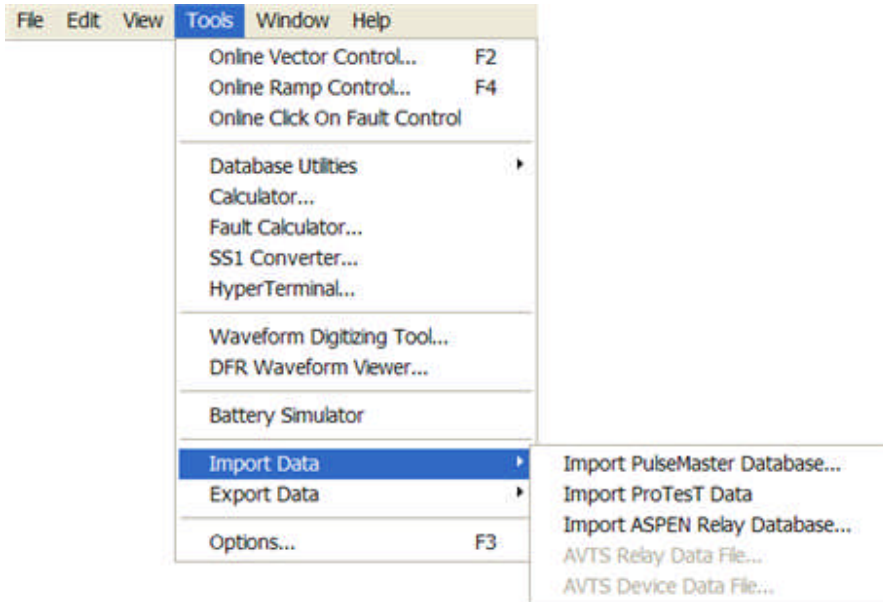
Import ProTest Data

ProTest Data Results

## Import ProTest Data Setup

The Process to Import a ProTest database requires the previous installation of ProTest or a previous installation of SQL Anywhere drivers.

The *Import ProTest Data* option opens the ProTest Import dialog.



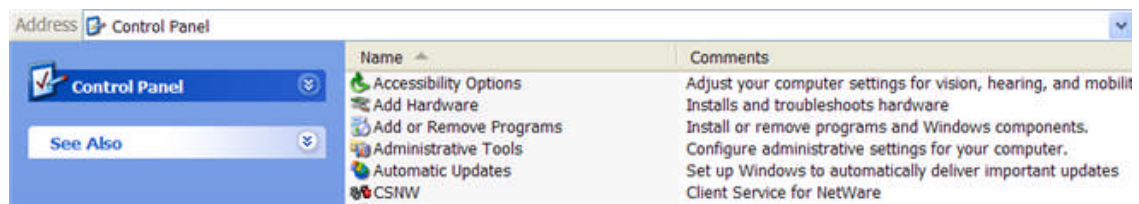
Import of the ProTest database will first require the **"ProTestImport" Device** to be created in the **Workspace Device tab**.

Prior to an attempt to import ProTest data, SQL Anywhere drivers must be installed. Then the following procedure must be followed to setup the computer to allow AVTS to accept the imported data. Typically, the SQLAnywhere drivers are included with the install disk for ProTest.

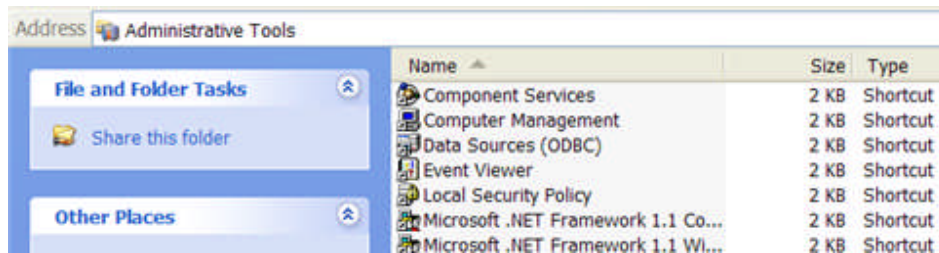
Click on **Start**.

Click on **Control Panel**.

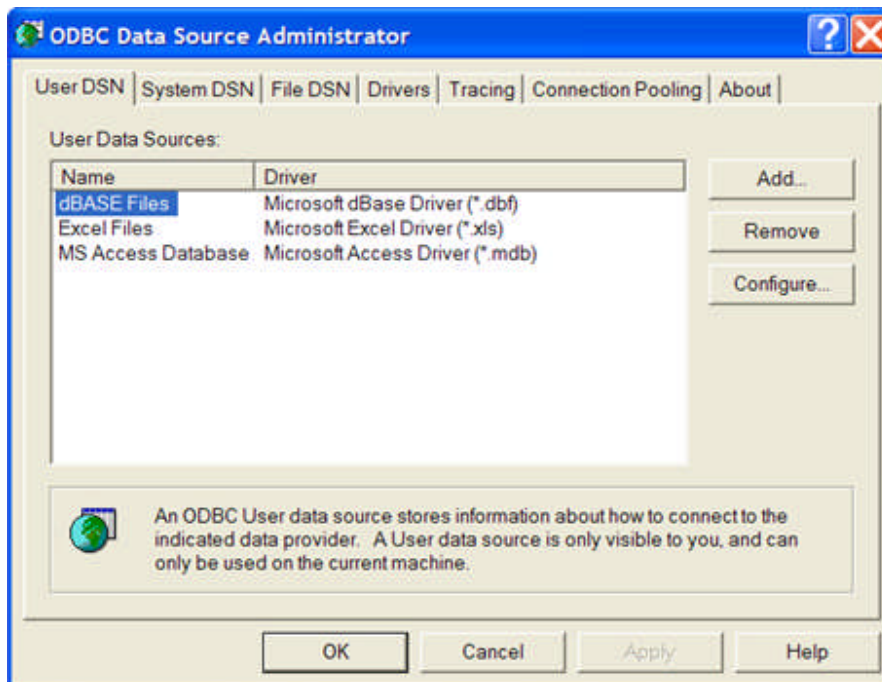
Double Click the *Administrative Tools* option to open the Administrative Tools dialog.



Double Click the *Data Sources (ODBC)* option to open the ODBC Data Source Administrator.

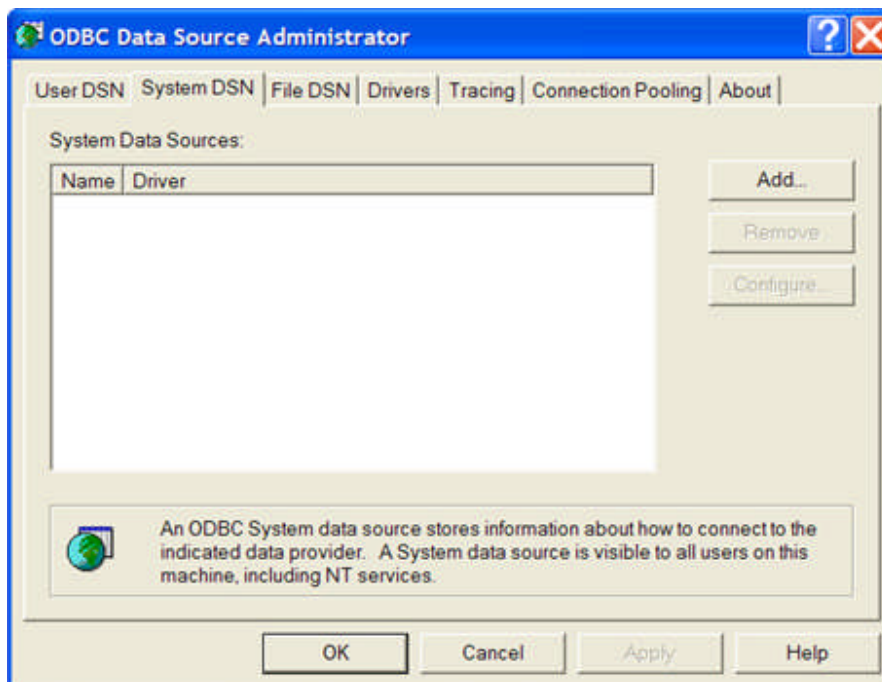


In the *ODBC Data Source Administrator dialog*, select either the User DSN or the System DSN tab.



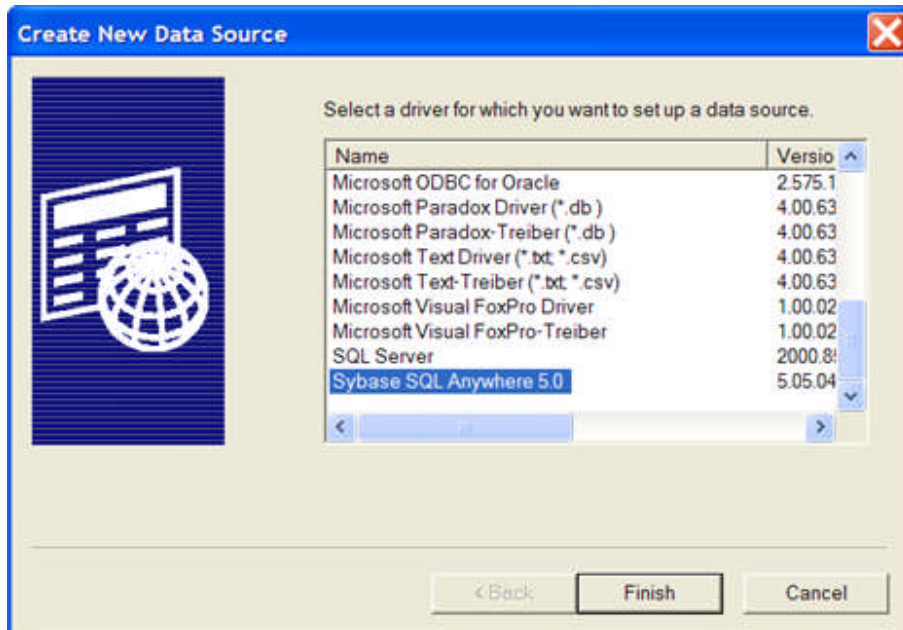
Use of the **User DSN** tab, to create a User Data Source, will limit the source to only the current machine and the current user.

Use of the *System DSN tab*, to create a System Data Source, will allow all users of the current machine to view and select the source.

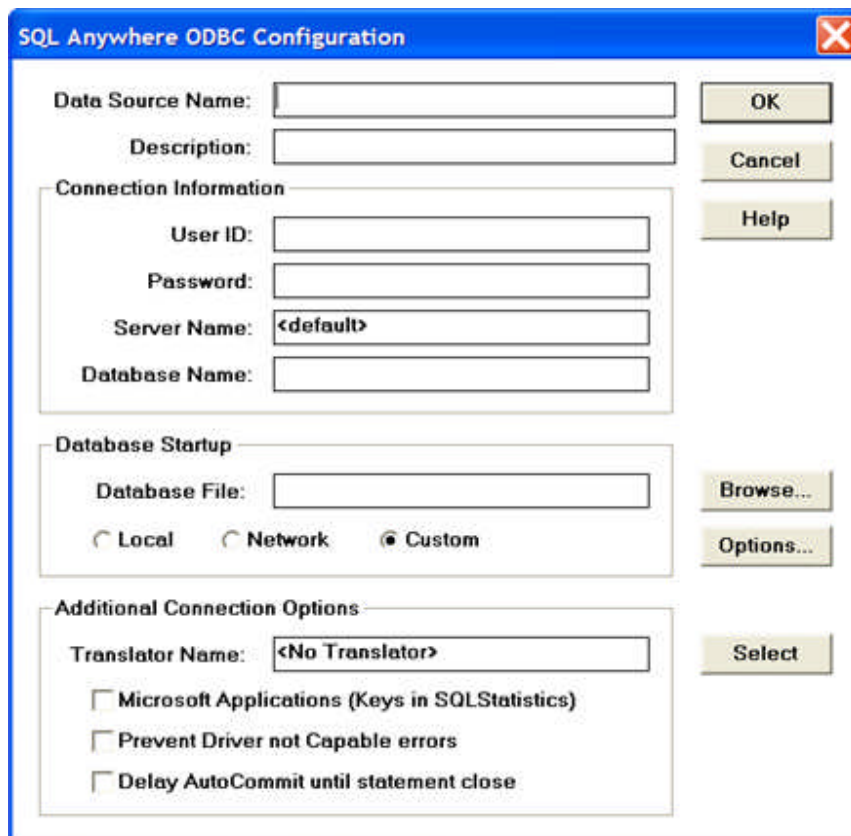


Click the **Add** button to open the Create New Data Source dialog.

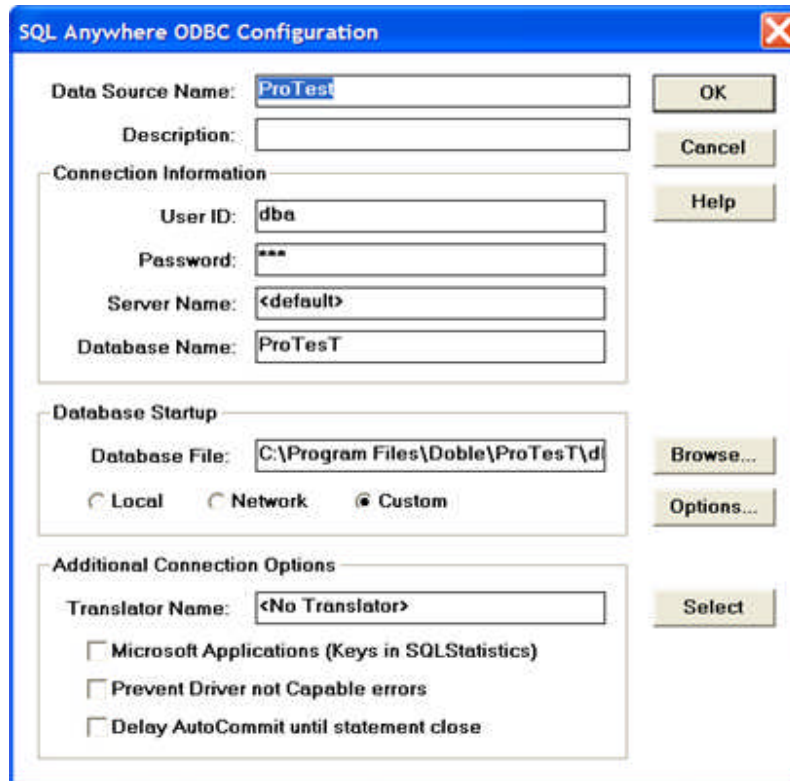
Scroll down the list and select the *Sybase SQL Anywhere* driver Name.



Click on the Finish button to open the **SQL Anywhere ODBC Configuration** dialog.



The *following fields* must be edited as follows:



**Data Source Name:** Enter the name that matches the name entered into AVTS Login dialog. This is usually the name of the user of AVTS. I.e., enter **Protest**.

Enter a **User ID:** for the database. I.e., **dba**

Enter a **User Password:** for the database. I.e., **sql** (entry of the Password will not be displayed as the typed alpha-numeric, but as "stars").

Using the **Browse** button, locate to enter the name of the **Database File:** to be imported. The path to the database will be entered into the edit field after the desired Protest database is located and selected.db.

**Note:** If Protest is installed, the default path for the ProTest database will likely be the **C:\Program File\ProTest\db** folder. The user may create another folder in another location, but the location must reside in the Database File: found using the Browse as above.

**Note:** If more than one database is to be Imported, each must be identified separately with unique names in the procedure above.

## ProTest Data Results

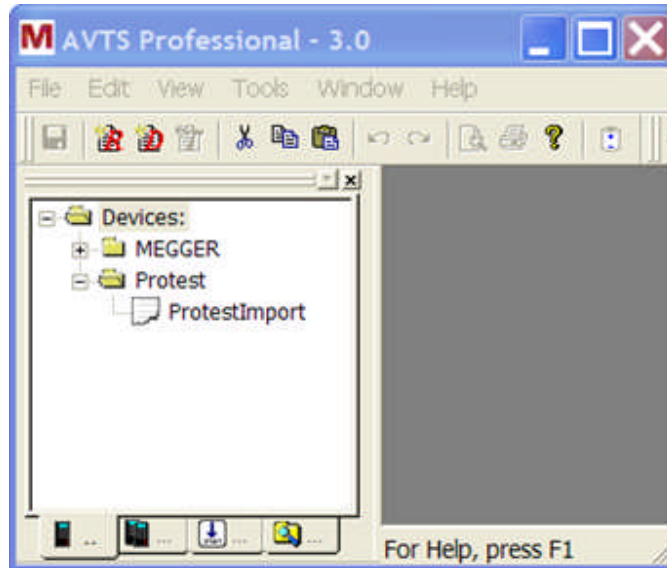
A ProTest Database must be imported to review the saved relay test results.

The Import process will import only test results. Test procedures or test files are not imported.

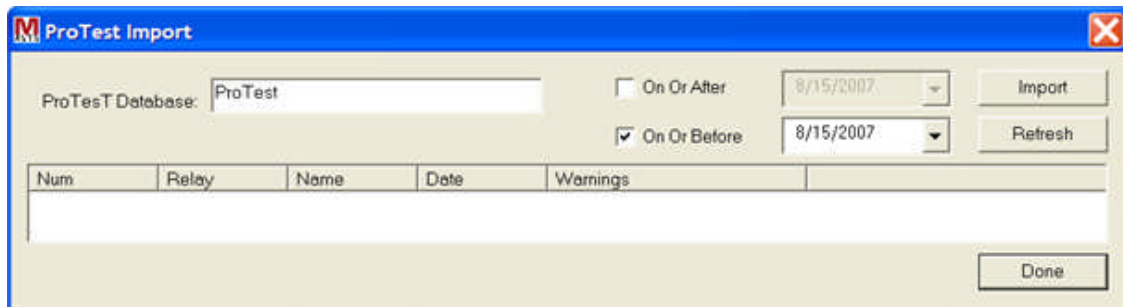
For the first time Import to an AVTS database, create a New Device in the AVTS open Database.

Device **Manufacturer** Name must be **Protest**

**Name** of the *new Device* must be **ProtestImport**, all one word and no spaces



Select the Tools menu item Import Data *ProTest Data* to begin the process.



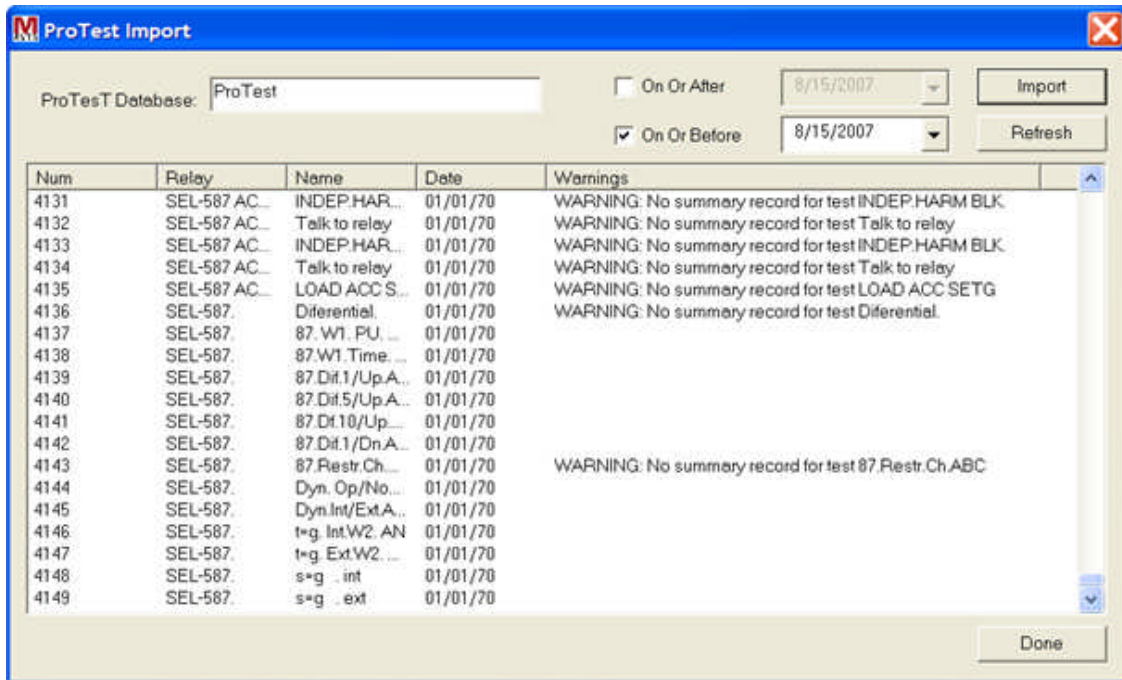
The ProTest Import dialog will have the database name, given in the Setup procedure, defaulted in the ProTest Database: edit field. Should more than one database reside in the default folder, the edit field may be edited to a different name, but the location must be the same folder.

The Check boxes are date filters to segregate those Results files desired.

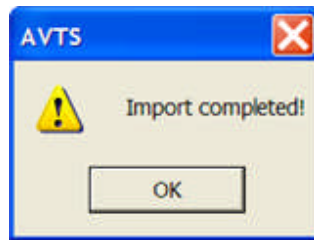
Click on the Import button to begin the Import process.

**Note:** During the Import, the ProTest Import dialog will become blank during the process.

Once complete the *ProTest files will appear* in the listing with the retrieved test number, the **Relay** model, the **Name** of the Relay, the Result **Date** and any **Warnings** associated with any Result.



Upon completion of the Import, an *AVTS message* notifying completion is displayed.



**Note:** The imported database does not immediately appear. Close AVTS and then Open AVTS to complete the Import.

Select from the Organization location in the Workspace the relay for the result wanted. Double click the result date to open the test result for that relay in the AVTS format in the Workbook.

Or, right click the relay test and *Export the results to a Word document.*



### Relay Results Report

Region: <u>Short ProTest Library.</u>	Relay: <u>HU (F2253).</u>
Substation: <u>ABB/VEST.</u>	Device: <u>ProTestImport</u>
Line:	Tested By:
	Date Tested: <u>03/04/1998</u>

**INFORMATION (NOTEBK)**

<u>INFORMATION</u>
--------------------

**Test Connection (NOTEBK)**

<u>Test Connection</u>
------------------------

**Min.PU.Restr.1a (LRAMP1)**

Actual	Percent Error	Min	Max	Result
1.720	14.667	0.000	0.000	Fail

**Min.PU.Restr.1b (PRAMP1)**

Actual	Percent Error	Min	Max	Result
1.630	8.667	0.000	0.000	Pass

**Min.PU.Restr.1c (BSRH01)**

Actual	Percent Error	Min	Max	Result
1.610	7.333	0.000	0.000	Pass

**Min.PU.Restr.2 (BSRH01)**

Actual	Percent Error	Min	Max	Result
1.630	8.667	0.000	0.000	Pass

**Different Up. (LRAMP1)**

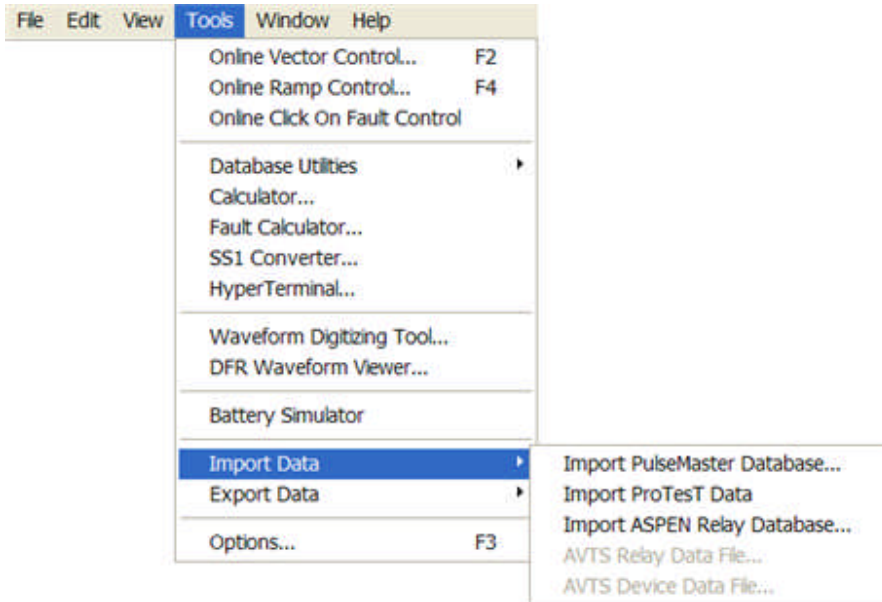
Actual	Percent Error	Min	Max	Result
6.880	-0.290	0.000	0.000	Pass

**Different Down. (LRAMP1)**

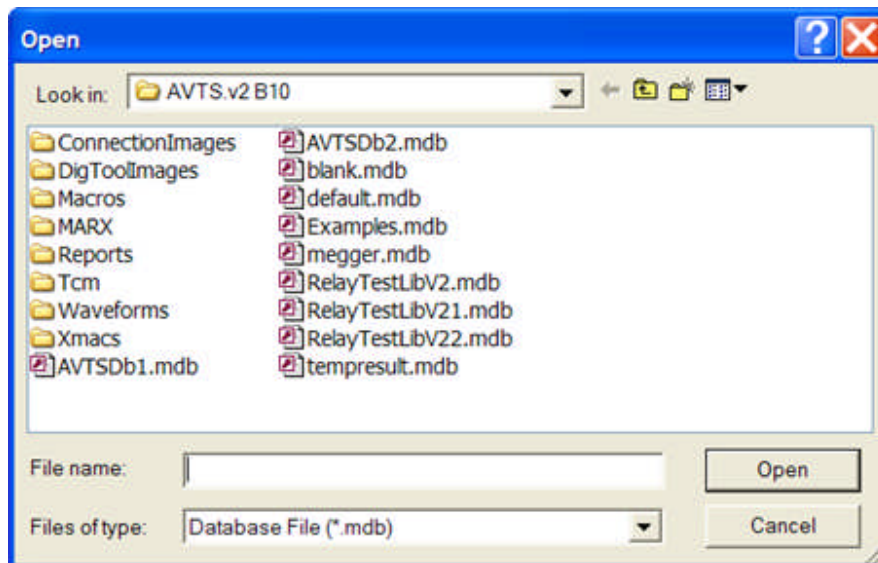
Actual	Percent Error	Min	Max	Result
3.200	-3.030	0.000	0.000	Pass

## Import ASPEN Relay Database

The *Import ASPEN Relay Database...* option opens the Windows Open dialog.



Select the Import *ASPEN Relay Database* to begin the process.

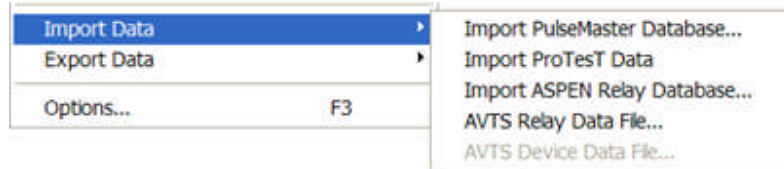


Browse and select the desired ASPEN Relay Database. The selected database name will appear in the File Name edit field.

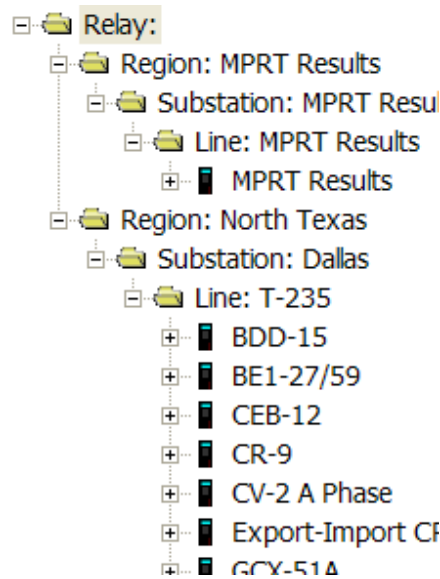
Select the Open button to proceed with the Import ASPEN Relay Database process.

## AVTS Relay Data File

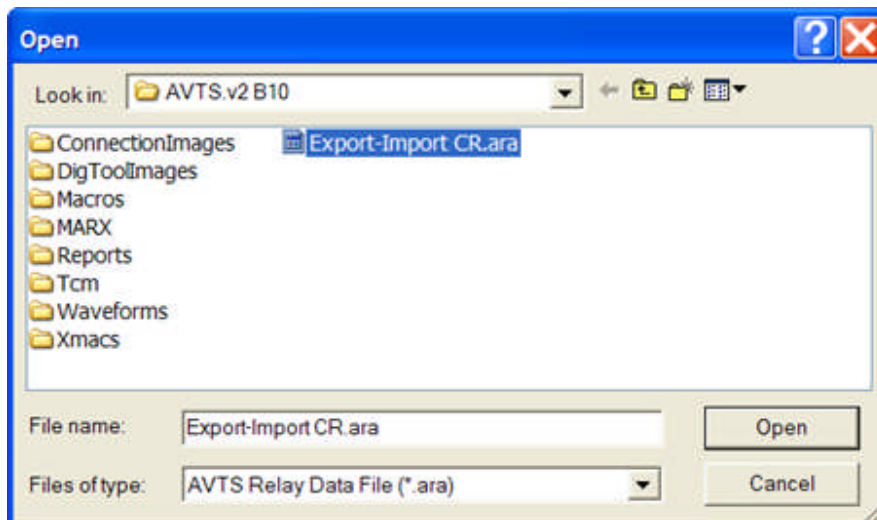
Prior to importing a *Relay Data File...*, the Workspace Relays tab must be selected and the location in the Organization Level must be selected. The purpose is to have the option to import only a relay from one database to another.



Select the relay location in the *Workspace* and then the Import option to begin the process.



The AVTS Relay Data File... option opens the *Windows Open* dialog. The "Files of type:" is defaulted to "AVTS Relay Data File (\*.ara) for segregating the relay data files from all others.



Browse and select the desired AVTS Relay Data File, \*.ara (accessrelayAVTS). The selected database name will appear in the File Name edit field.

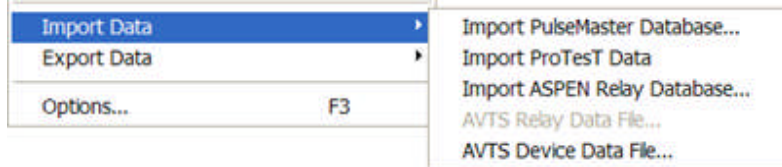
AVTS Help 092607

Select the Open button to proceed with the Import AVTS relay data file process.

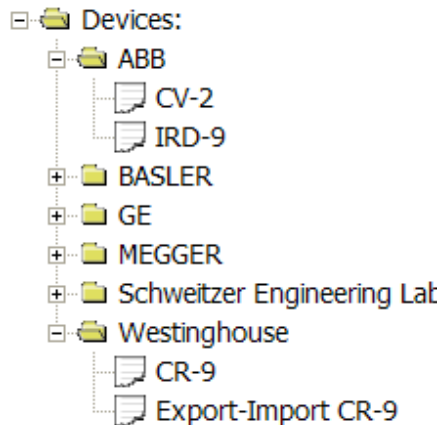
AVTS will place the file, example "Export-Import CR.ara", into the Workspace Relays tab at the selected location.

## AVTS Device Data File

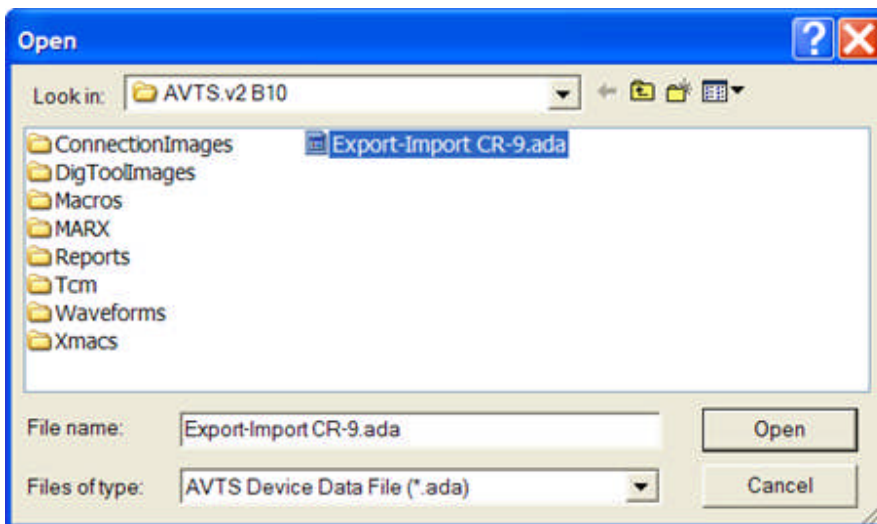
The Import *AVTS Device Data File...* is enable for selection when the Workspace Devices tab has been expanded. A Device does not have to be open for this function to operate.



Open the *Devices* tab and select the Manufacturer and then the Import option to begin the process.



The purpose is to have the option to import only a device from one database to another. Select the Import AVTS Device Data File... option to display the *Windows Open* dialog.



Browse and select the desired AVTS Device Data File, \*.ada (@accessdeviceAVTS). The selected database name will appear in the "File name:" edit field. The "Files of type:" is defaulted to "AVTS Device Data File (\*.ada)" for segregating the relay data files from all others.

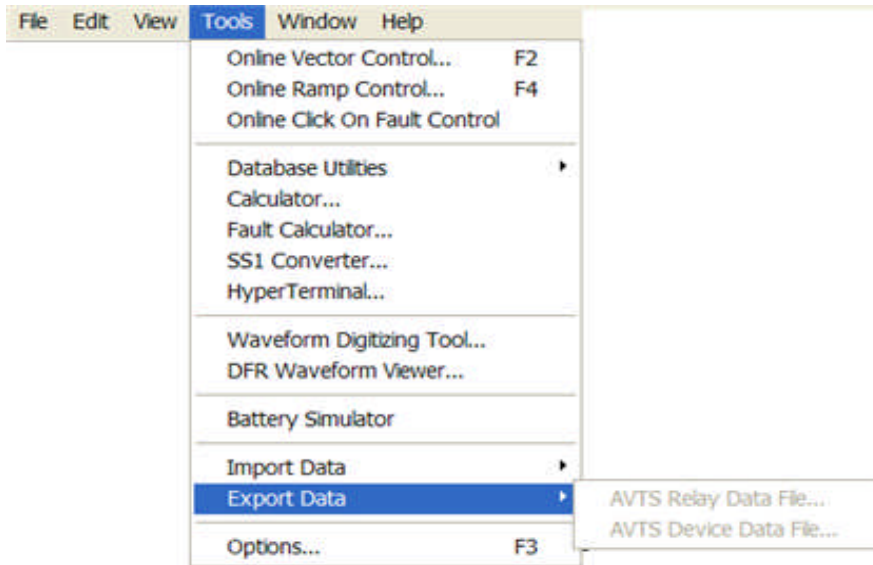
Select the Open button to proceed with the Import AVTS device data file process.

AVTS will place the file, example "Export-Import CR-9.ada", into the Workspace Devices tab.

## Tools Export Data

## Tools Export Data

The available options in the Tools *Export Data* menu for exporting AVTS data are:



AVTS Relay Data File...

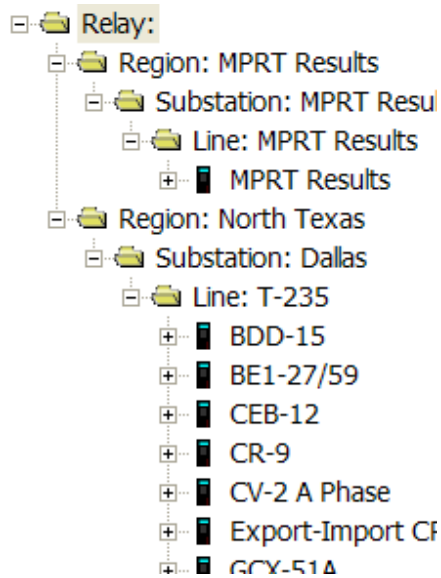
AVTS Device Data File...

## AVTS Relay Data File export

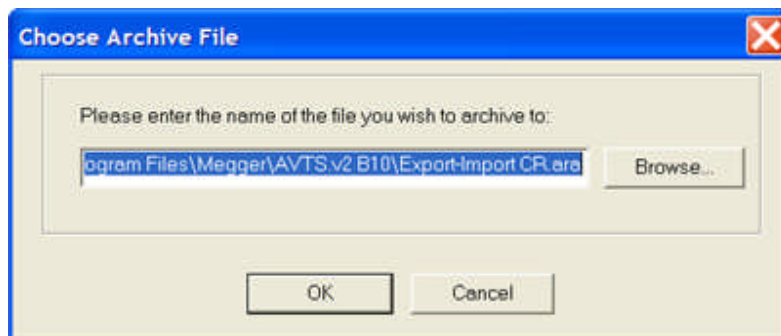
Prior to exporting a *Relay Data File...*, the Workspace Relays tab must be selected and the relay in the Organization Level must be selected.



Select the *Relays* tab then the Export option to begin the process.



The purpose is to have the option to export only a relay from one database to be imported into another. The AVTS Relay Data File... option opens the *Choose Archive File* dialog.



The edit field defaults to the path for the selected relay to be cloned and stored. The name of the relay is also defaulted to the name of the selected relay in the AVTS active database. The database file extension \*.ara is attached to the file name by default.

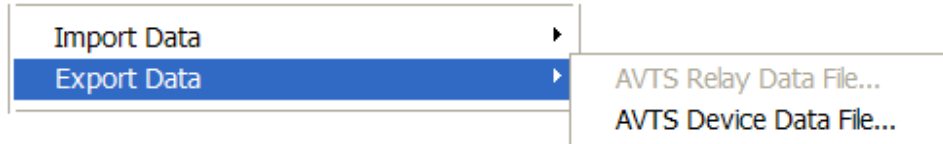
The relay name in the path may be edited separately within the path in the edit field if a name change is desired. It will be necessary to also include the database file extension \*.ara with the file name. The extension will allow an import of the file to be immediately in an import process.

The Browse button may be selected to search for a location other than the default for exportation of the relay data file.

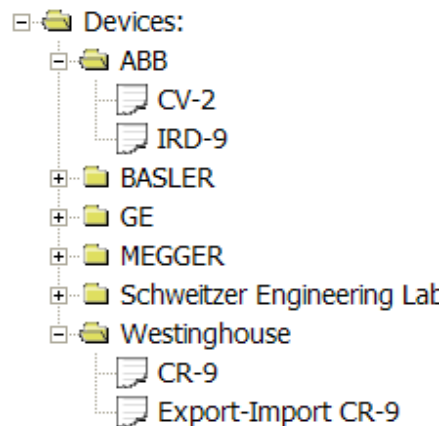
Select OK to complete the exportation process. The Cancel option will abort the exportation process and return to the active AVTS data base.

## AVTS Device Data File export

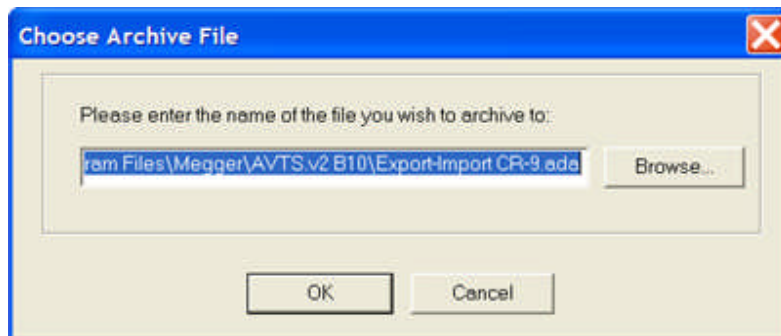
Prior to exporting a *Device Data File...*, the Workspace Devices tab must be selected and the Device must be selected.



Select the *Devices* tab and then the Export option to begin the process.



The purpose is to have the option to export only a Device from one database to be imported into another. The AVTS Device Data File... option opens the *Choose Archive File* dialog.



The edit field defaults to the path for the selected Device to be cloned and stored. The name of the Device is also defaulted to the name of the selected relay in the AVTS active database. The database file extension \*.ada is attached to the file name by default.

The Device name in the path may be edited separately within the path in the edit field if a name change is desired. It will be necessary to also include the database file extension \*.ada with the file name. The extension will allow an import of the file to be immediately in an import process.

The Browse button may be selected to search for a location other than the default for exportation of the Device data file.

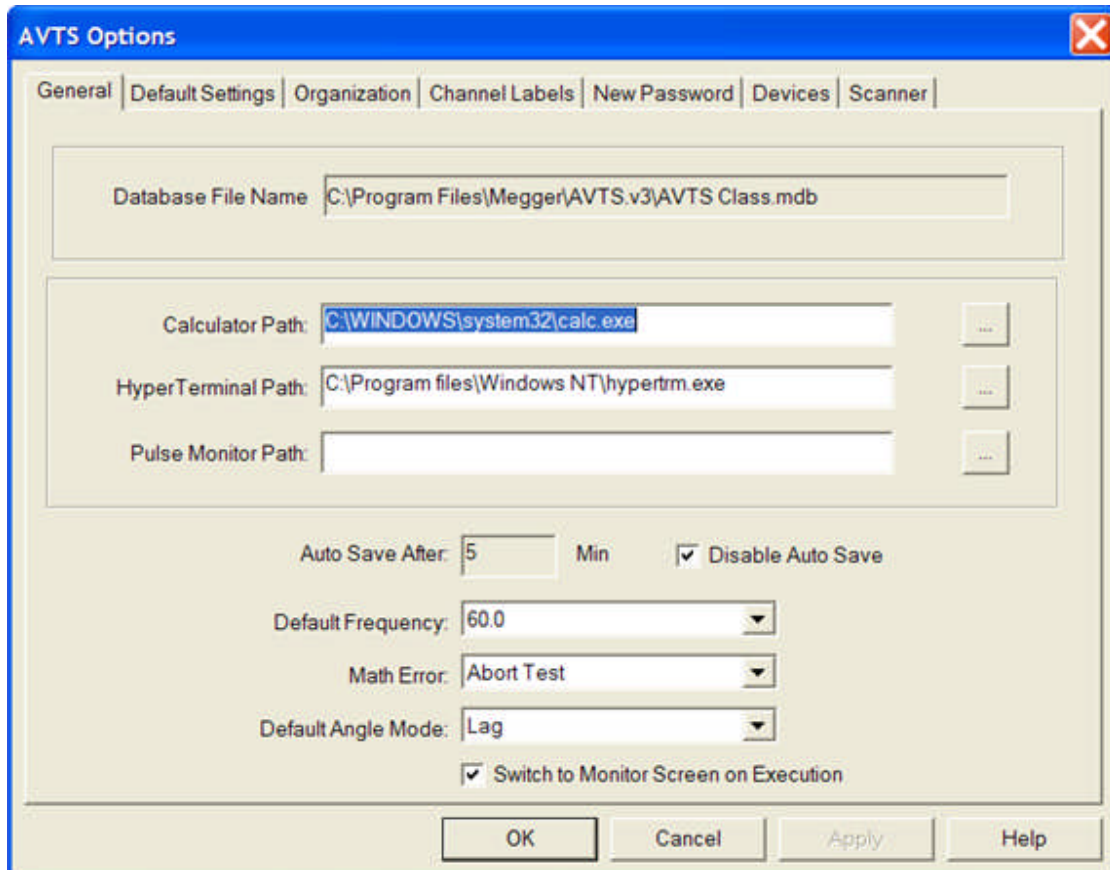
Select OK to complete the exportation process. The Cancel option will abort the exportation process and return to the active AVTS data base.



## Tools Options

## Tools Options

The Tools/Options Menu item opens the *AVTS Options* dialog.

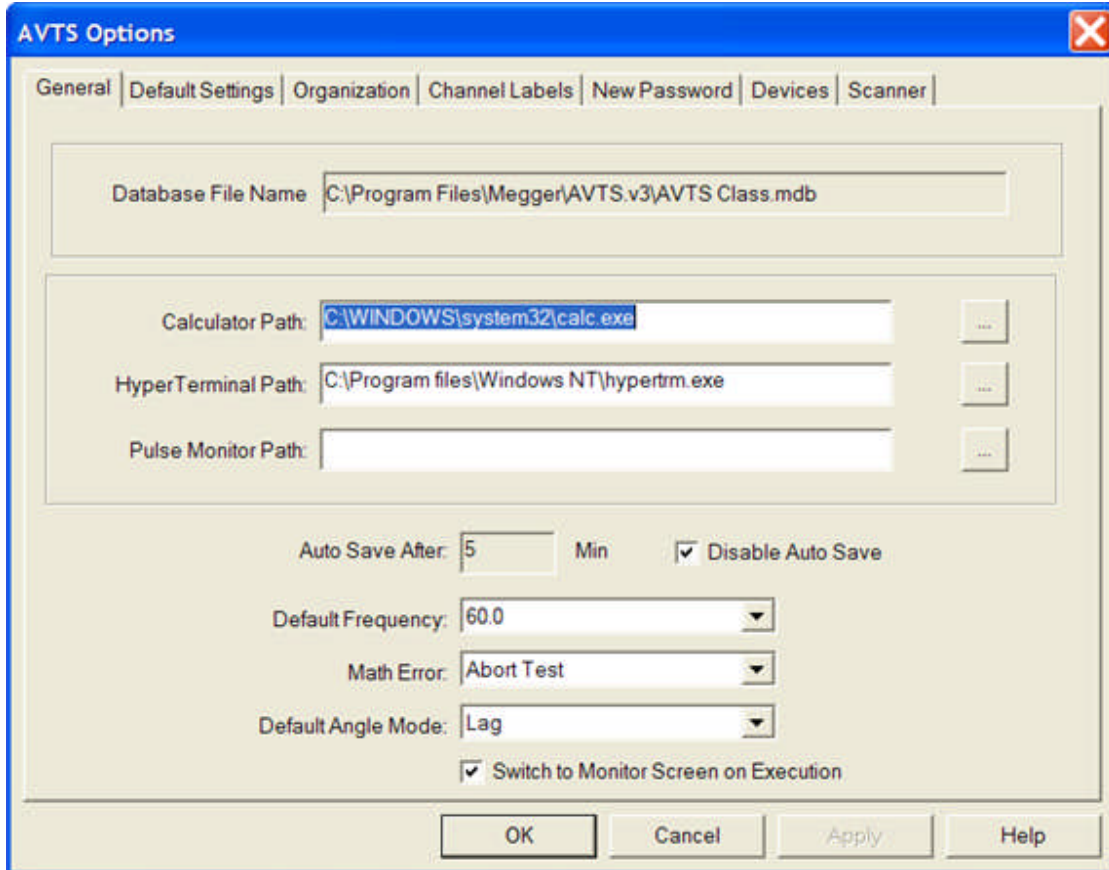


Details to each of the AVTS Options tabs are located in

- General
- Default Settings
- Organization
- Channel Labels
- New Password
- Devices
- Scanner

## General Tab

The AVTS Options *General* tab is the default tab when the dialog is opened from the Tools/Options menu item.



The listed edit fields are therefore immediately available for change while using an active AVTS database.

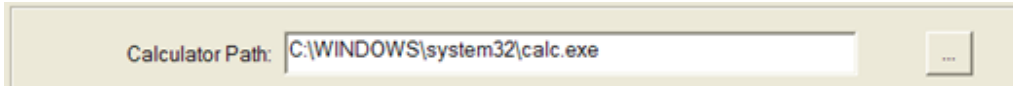
- Database File Name
- Calculator Path
- HyperTerminal Path
- Pulse Monitor Path
- Auto Save After
- Default Frequency
- Math Error
- Default Angle Mode
- Switch to Monitor Screen on Execution

The **Database File Name** edit field displays the default path for the presently open AVTS database. This field is gray and is not available to be edited.



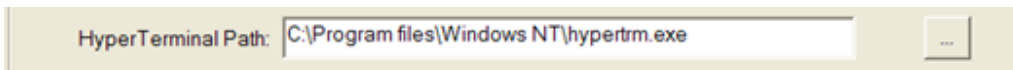
The **Calculator Path** edit field displays the path for the Microsoft Calculator. The path is default to the installation of AVTS. The calculator is made active by selecting the Tools/Calculator... menu item.

Any path for an executable (\*.exe) program may be substituted in this field and made active by selecting the Tools/Calculator... menu item. For example, the path C:\WINDOWS\system32\sol.exe is inserted, the Solitaire card game will become active by selecting the Tools/Calculator... menu item.



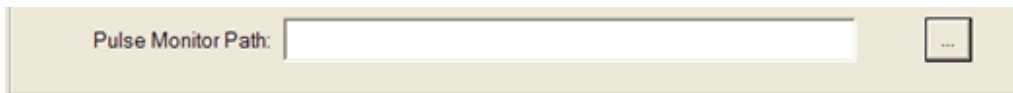
The **HyperTerminal Path** edit field displays the path for the Microsoft HyperTerminal. The path is default to the installation of AVTS. The HyperTerminal telnet program is made active by selecting the Tools/HyperTerminal... menu item.

Any path for an executable (\*.exe) program may be substituted in this field and made active by selecting the Tools/HyperTerminal... menu item. For example, the path C:\WINDOWS\system32\sol.exe is inserted, the Solitaire card game will become active by selecting the Tools/Calculator... menu item.



The **Pulse Monitor Path** edit field displays the path for browsing and selecting the path to the AVO TCM915S Pulse Monitor program executable, switchbox.exe. Sequence Configuration and Record Configuration files must be located in the C:/Program Files/Megger/AVTSv2.x/TCM folder. The path is blank by default on installation of AVTS.

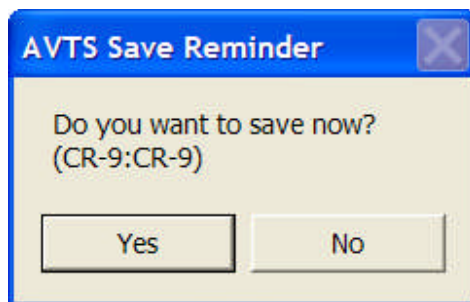
The Pulse Monitor path locates the switchbox.exe to be called by the TCM-915S Control in the Workbook Test Editor tab.



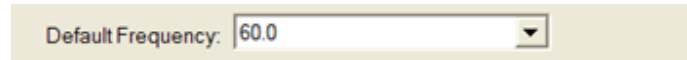
The **Auto Save After** function will automatically, after the time enabled, display the AVTS Save Reminder dialog. Click Yes to save of all work completed in the AVTS Workbook. The field is enabled by unchecking the Disable Auto Save check box. The default is shown with the Auto Save set at 5 minutes and Disabled with the check box. The format for the save time is of integer form and may be set for values of 1-300 minutes.



When enabled and work is being performed in the AVTS Workbook, the AVTS Save Reminder dialog will be displayed after the set elapsed time. Select No to not save the present work since the last save. Select Yes to Save the existing work since the last save.



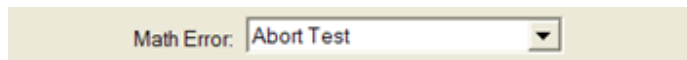
The **Default Frequency** selection is by the drop down arrow. The choice is either 60 or 50 Hertz. The default 60 Hertz is shown and is applied to all timing functions dependent to calculation in cycles and seconds.

A screenshot of a software interface showing a dropdown menu for 'Default Frequency'. The menu is open, and the value '60.0' is selected and displayed in the input field.

The **Math Error** selection is by the drop down arrow. Choice is either *Abort Test* or *Ignore* and is shown in default as *Abort Test*. A Math Error is a calculation performed which yields an undefined result.

During a test execution and there is a necessary calculation during test execution which creates an undefined result, there will be a Math Error and the test will abort with no message. For example, the calculation  $Z = V_a/I_a$  will create a Math Error if it appears early in the test execution and  $I_a$  is initially zero.

Select *Ignore* for Math Error and the same test will begin to execute and continue until the test either completes successfully or terminates early.

A screenshot of a software interface showing a dropdown menu for 'Math Error'. The menu is open, and the value 'Abort Test' is selected and displayed in the input field.

The **Default Angle Mode** selection is by the drop down arrow. Choice is either *Lag* or *Lead/Lag*.

Lag places all vectors in positive degrees displacement clockwise in a polar diagram referenced to zero degrees. Pulsar operates in lagging positive angles only. MPRT allows use of this convention.

Lead/Lag allows for all vectors to be placed in positive degrees displacement counter-clockwise in a polar diagram referenced to 0 degrees. This is considered the textbook engineering convention and may be used with MPRT.

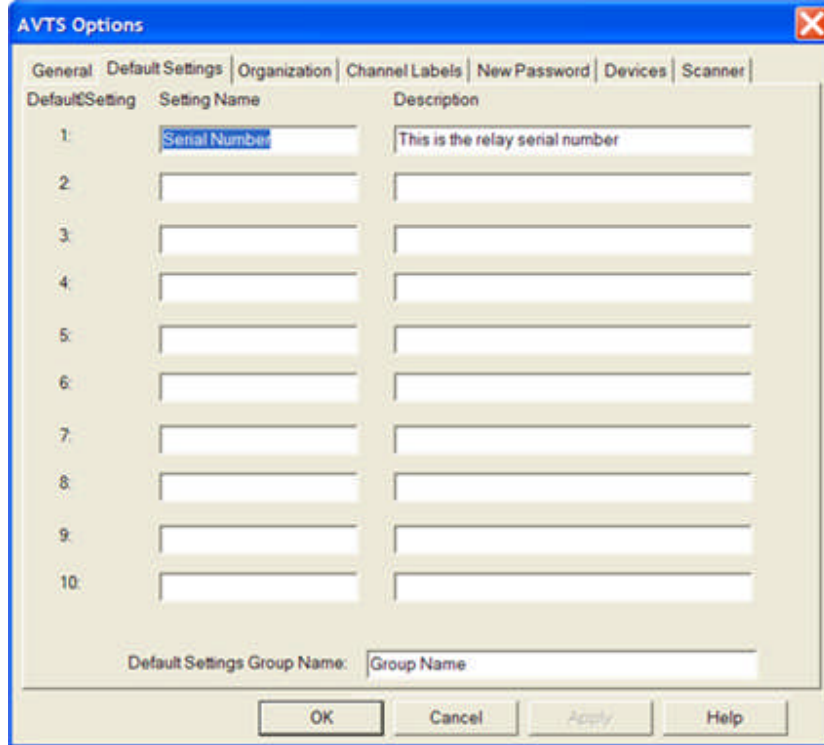
A screenshot of a software interface showing a dropdown menu for 'Default Angle Mode'. The menu is open, and the value 'Lag' is selected and displayed in the input field.

The **Switch to Monitor Screen on Execution** check box is default to Switch to Monitor Screen on Execution. A test may be executed from any selected Workbook tab. Once executed the Test tab becomes automatically selected and displayed for monitoring the test during execution. If unchecked, the Workbook tab displayed when a test is executed will remain in display.

A screenshot of a software interface showing a checkbox labeled 'Switch to Monitor Screen on Execution'. The checkbox is checked, and the text 'Switch to Monitor Screen on Execution' is displayed next to it.

## Default Settings Tab

The AVTS Options *Default Settings* tab is selected from the Tools/Options menu item.



There are ten edit fields for each the Settings Name and the Description columns.

When a New Device is created in the Devices Workspace, it will be created with details in the Workbook Settings tab.

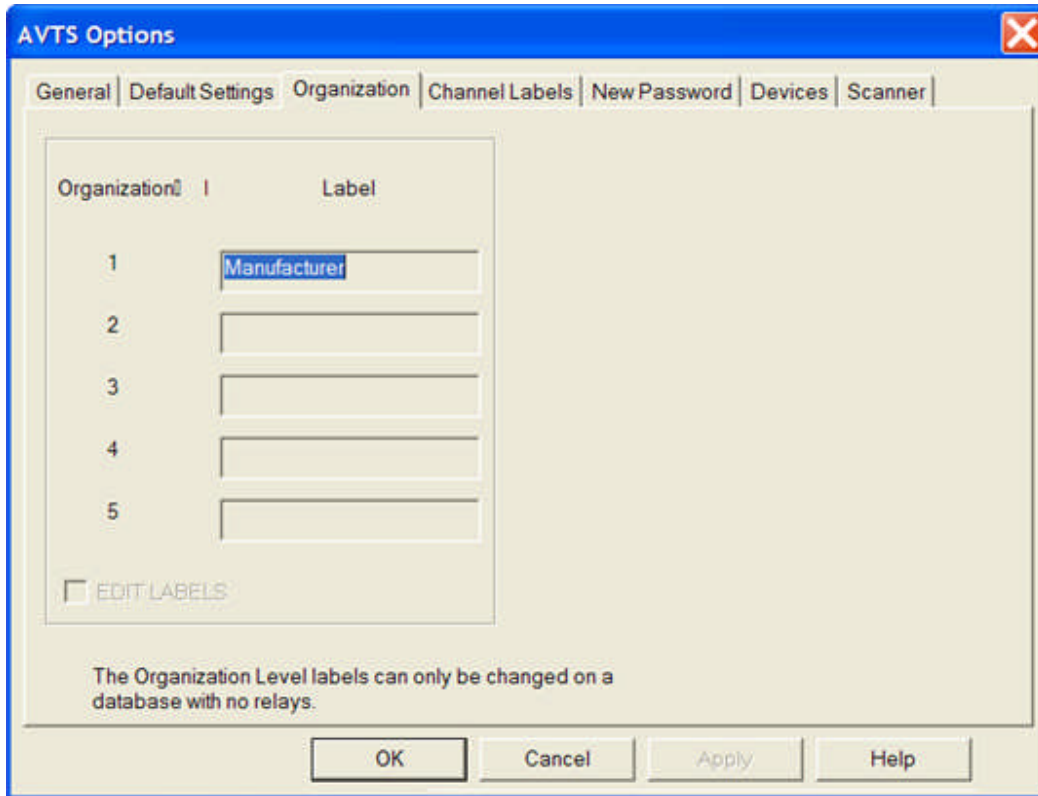
The Settings tab will be populated automatically with a Group of Settings. These will be displayed with the Default Settings shown. The Default Settings may be edited to settings that the user finds most useful. The Default Setting Name and Description columns and the default Settings Group Name shown below are the AVTS installation defaults.

With these existing setting labels, a new created Workspace Device may be opened to the Workbook Settings tab. The new settings will appear as a single group labeled "Group Name" containing a single Setting Name labeled "Serial Number" and a single Description labeled "This is the relay serial number".

Additional Groups of settings and additional Settings may be added while in the Workspace Device Settings tab.

## Organization Tab

The AVTS Options **Organization** tab is selected from the Tools/Options menu item.



Relays may be classified by the Organization level developed in the five edit fields. The AVTS installation default Organization is shown with a single level and labeled Manufacturer. Any number of levels from 1-5 may be defined.

If the given database does not have the desired Organization, it may be merged, as a Subset using Database Utilities, into another database with the desired number of levels. Or it may be merged into a new database with no existing Devices or Relays in the Workspace, but having the desired Organization number of levels and labels. Two databases each with an unlike number of levels may be merged by remapping the levels of the database to be imported to the active database.

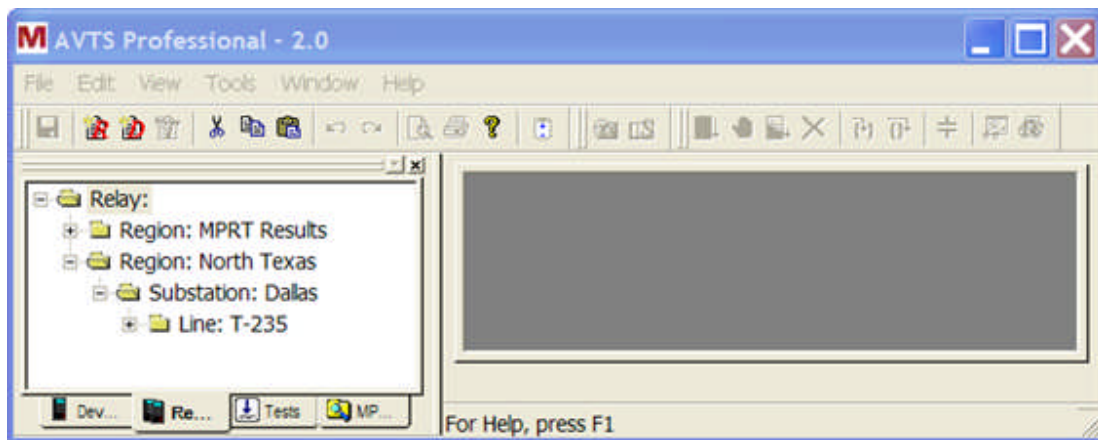
**Note:** EDIT LABELS check box is not able to be checked nor the labels edited unless the database contains no Devices or Relays.

**Note:** Always edit labels starting with level 1 and continuing to level 5. DO NOT leave a label blank between two levels that have labels.

A **three level Organization** may be created in a new Database only.

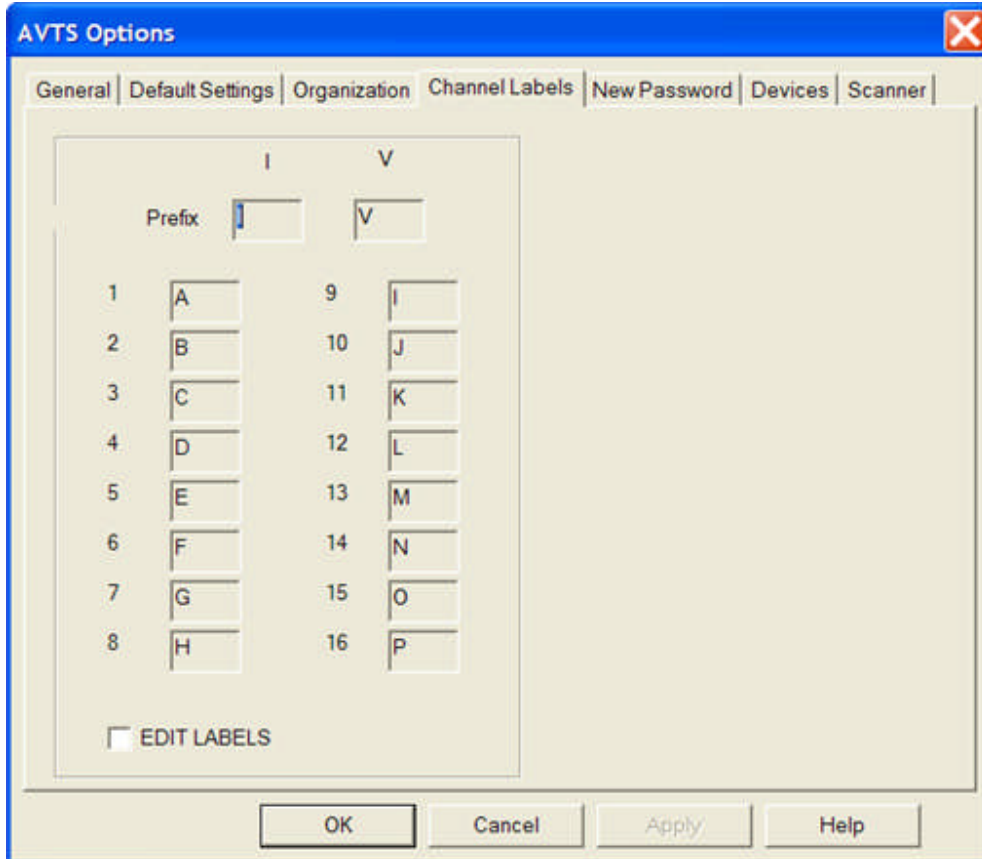


An example of a typical *three level* Organization may have the Relay Workbook appearance shown below. The labels shown after the Organization levels are created with the creation of a new Relay that requires a different location for classification.



## Channel Labels Tab

The AVTS Options *Channel Labels* tab is selected from the Tools/Options menu item.



The Prefix for the I (current) and V (voltage) MPRT output channels may be defined as shown in the AVTS installation default, or edited. Check the EDIT LABELS check box to edit the Prefix and any or all channel labels.

For example Prefix might be A (current) and G (voltage), and the channels 1, 2 and 3 might be edited to R, S and T respectively. These edits will create commands to the MPRT to be Ar, As, At, Gr, Gs, and Gt, assuming only three current and voltage channels exist in the test set.



## New Password Tab

The use of Passwords in AVTS is a user option. They may be exercised by setting appropriate Passwords depending on the type of user to which restrictions may or may not apply.

The AVTS Options *New Password* tab is opened from the Tools/Options menu item.

The screenshot shows the 'AVTS Options' dialog box with the 'New Password' tab selected. The dialog contains the following fields and controls:

- New Master Password:** [Text Field]
- Confirm New Password:** [Text Field]
- Set New Master Password
- New User Password:** [Text Field]
- Confirm New Password:** [Text Field]
- Set New User Password
- New Restricted Password:** [Text Field]
- Confirm New Password:** [Text Field]
- Set New Restricted Password
- New Read-Only Password:** [Text Field]
- Confirm New Password:** [Text Field]
- Set New Read-Only Password

At the bottom of the dialog are four buttons: OK, Cancel, Apply, and Help.

Password Types

Set Passwords

AVTS Login

There are four **Password types**:

- The **New Master** will be able to perform all available functions of and have full control of all users of AVTS. There must be a Master for there to be Passwords set for any or all of the other user types. Once set, the New Password must be re-typed in the **Confirm New Password** field for verification.
- The **New User Password** will be set by the Master while AVTS is open at the Master level. This user has access to all Workbook and Workspace tabs for view. Relay settings may be changed in the Settings tab and a relay may be selected to execute a test. This user will not be able to make changes to any other tabs in an open Relay or open Device. Once set, the New Password must be re-typed in the **Confirm New Password** field for verification.
- The **New Restricted Password** will be set by the Master while AVTS is open at the Master level. This user has access to all Workbook and Workspace tabs for view. Relay settings may be changed in the Settings tab and a relay may be selected to execute a test. This user will not be

able to make changes to any other tabs in an open Relay or open Device. Once set, the New Password must be re-typed in the **Confirm New Password** field for verification.

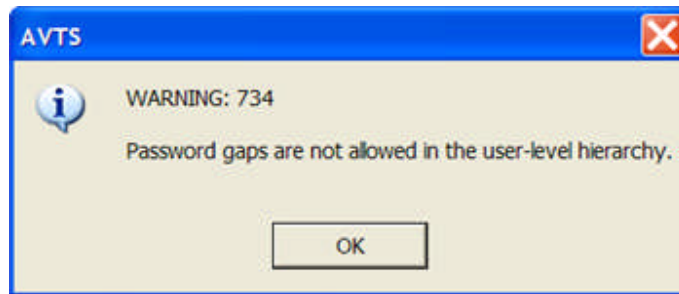
- The **New Read-Only Password** will be set by the Master while AVTS is open at the Master level. This user has access to all Workbook and Workspace tabs for view. A relay may be selected to execute a test. This user will not be able to make changes to any tabs in an open Relay or open Device. Once set, the New Password must be re-typed in the **Confirm New Password** field for verification.

Back to top.

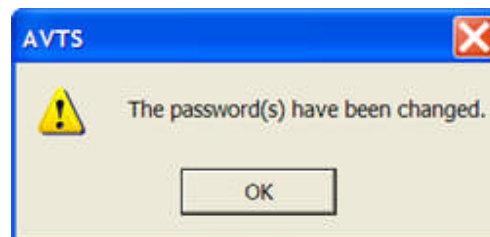
### Set Passwords

For Passwords to be used:

- There must be a valid Master Password to set any other passwords
- There is a hierarchy for the user levels and can make available login to a higher level of use than desired.
- The Set New "user type" Password must be checked to enable the password entries for that user type.
- The Password is case sensitive for confirmation to a new password entry
- The Password is limited to 20 alpha, numeric, or alpha-numeric characters
- If there is no Master password entry, any attempt to enter a password for a lesser user type will create the *Password gaps* message to display. OK will return to the Options New Password dialog



- If the New Master Password entry, or any other user passwords entries there after, are set or altered, the *valid password message* will display. OK will return to the Options New Password dialog



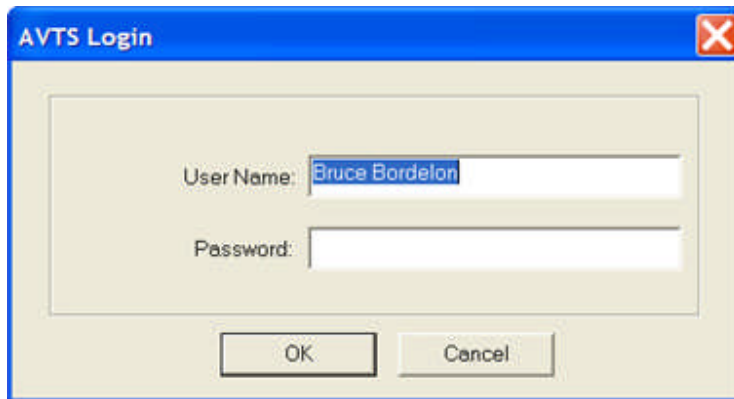
Attempts to enter like passwords for different user types will display the *password error* message



Back to top.

### AVTS Login

Passwords are used to allow user types with or without restrictions to Log into AVTS. If passwords are set and AVTS is opened, the user will be required to enter the assigned password in the *AVTS Login* dialog to be able to exercise his range of capability.



If OK is selected with the proper password, a message will show the restrictions applied to that Password.  
 if OK is selected without a password entry, a message will show the restrictions of the next lesser level user that does not have an assigned password.

**Warning: Should there be a Master user and Restricted or Read-Only user passwords, a login with a blank password field will allow entry at the User level with less restrictions applied.**

Cancel will abort the AVTS application.

An incorrect password entry will cause an *incorrect password* message to be displayed. The message will return for every attempt to enter an incorrect password. If the AVTS Login Password field is void of password attempt, the User level password entry message will be displayed and AVTS opened under those restrictions.



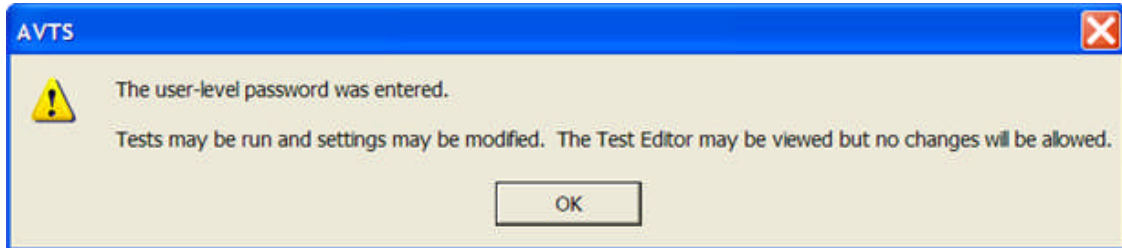
OK will return to the Login dialog.

- Master Login

A valid password entry will open AVTS for full use and no restrictions.

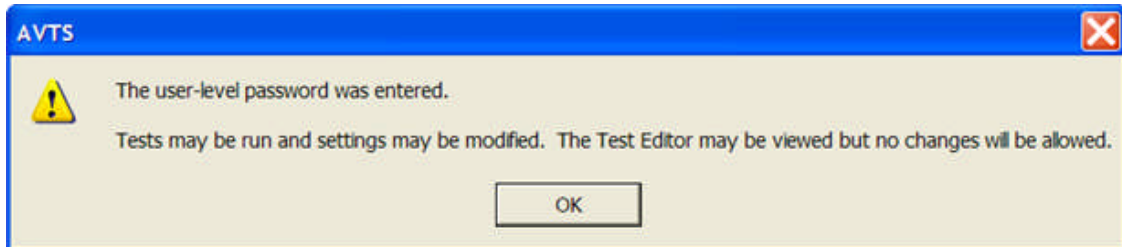
There is no recognition message for a valid Master Login.

Entry of a blank password field will open AVTS to the lowest level of Password types. This example is returned for a blank entry if only a Master password is set. The next lower level of entry is *User-level*.

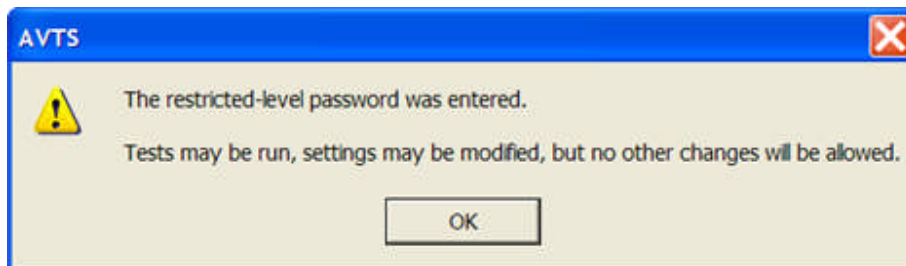


- User Login

A valid password entry will display the *user-level* use restriction message.



Entry of a blank password field will open AVTS to the lowest level of Password types. This example is returned for a blank entry if only a Master password is set. The next lower level of entry is *restricted-level*.

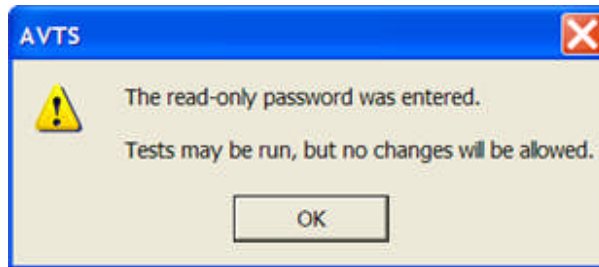


- Restricted Login

A valid password entry will display the user *restricted-level* use message.

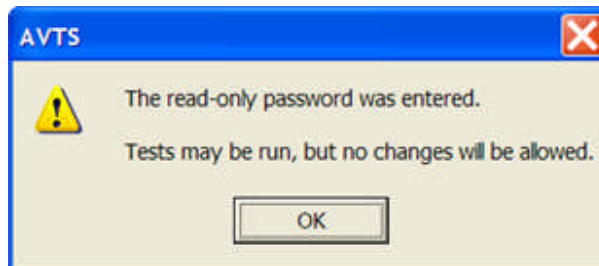


Entry of a blank password field will open AVTS to the lowest level of Password types. This example is returned for a blank entry if only a Master password is set. The next lower level of entry is *read-only*.



- Read-Only Login

A valid password entry will display the user *read-only* level use restriction message.



Entry of a blank password field will not allow opening AVTS. Continued attempts will return the *incorrect password* entry message.

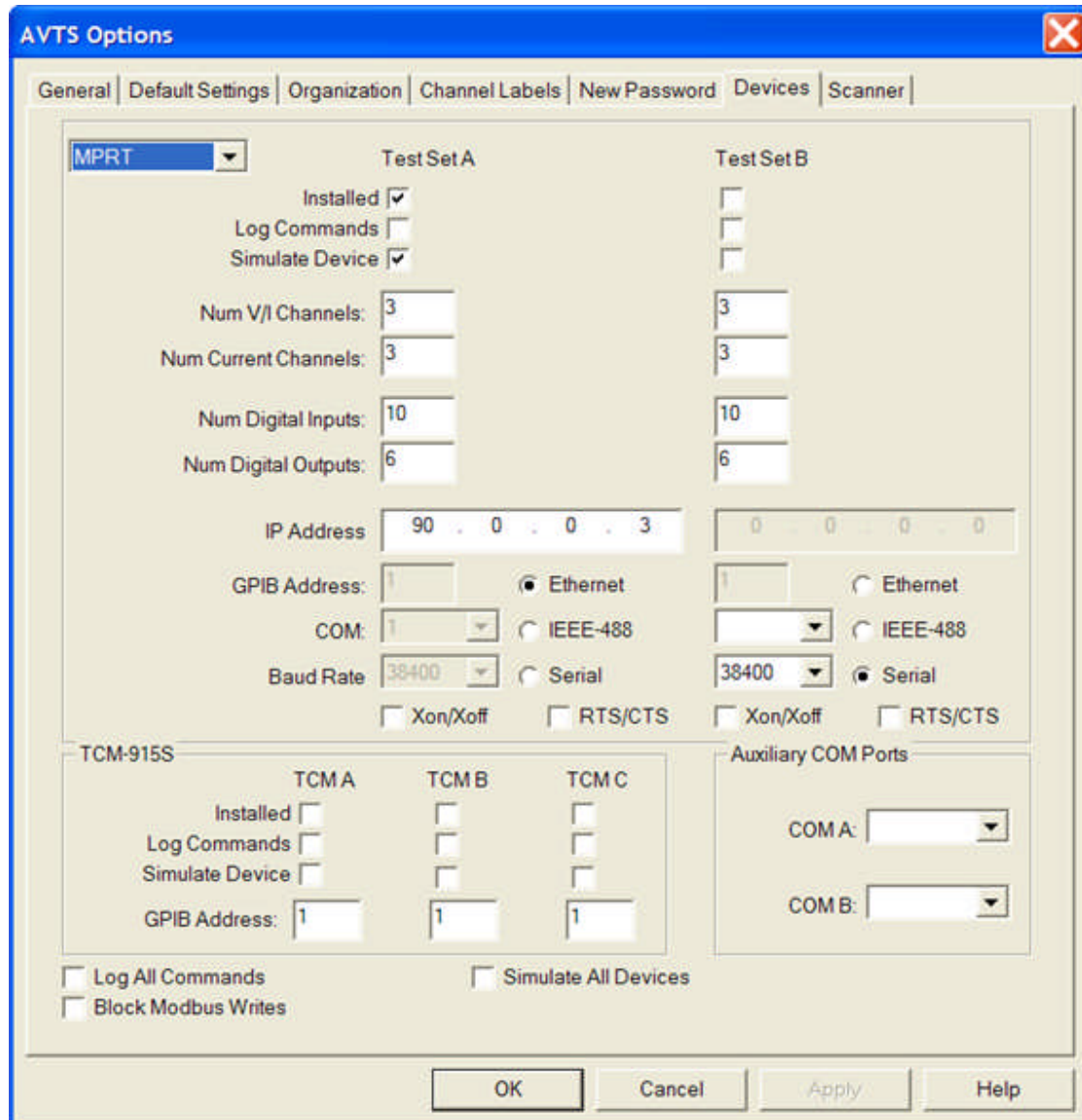


Back to top.

## Devices Tab

### Devices Tab

The *AVTS Options Devices* tab must be set correctly for proper communication to connected test equipment. This is also the location to set AVTS to simulate test equipment when constructing, editing or examining a test module.



The various edit fields are therefore immediately available for change while using an active AVTS database.

The areas of the tab are explained in the following topics:

- Test Set Inputs and Outputs
- Test Set Communication
- Test Set IEEE-488 Communication
- Test Set Serial Communication

TCM-915S Communication  
Auxiliary COM Ports

## Test Set Inputs and Outputs

The *Devices tab upper* area for **MPRT** or **PULSAR** is for selecting the type test set and providing the details. When the **PULSAR** set is selected from the drop down list, the number of available options are reduced by dimming the non-usable fields. The left side is for **Test Set A** settings and the right side is for **Test Set B** settings. Note that this area changes for the set type selected based on the test set capabilities. Both areas are explained together.

Either a **PULSAR** or a **MPRT** may be selected for the Test Set to be used with the **Installed** check box. A mixture of test sets is not allowed; both must be of the same type.

**Log Commands**, when checked, will create a text record of all commands sent to the Test Set Installed and all the responses returned from the Test Set. Should problems occur during the test of edited or newly created test procedures, this record is useful in determining the root of the problem. The problem might also be a result of incorrect relay settings reflected in AVTS compared to the response capability of the relay. The logged commands may be viewed with **Ctrl + L** for to View Instrument Log...

**Simulate Device**, when checked, will allow the execution of a test and allow the computer and user to emulate the relay supposedly under test. Simulate Device is also very useful in checking execution of edited and newly created tests.

**Num V/I Channels**: edit field is for entering the number of V/I Channels present in the connected **MPRT** or the number of Voltage Channels present in **PULSAR**. Note that the **PULSAR** may have as many voltage channels installed as there is space allowed in the chassis.

**Num Current Channels**: edit field is for entering the number of Current Channels present in the connected **PULSAR** or the number of additional auxiliary Current Channels not identified in the Num V/I Channels edit field connected to the **MPRT**.

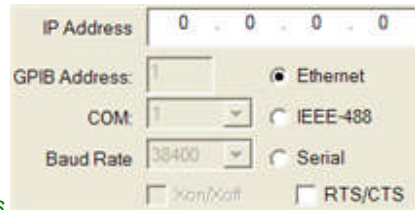
**Num Digital Inputs**: edit field is to enter the number of Num Digital Inputs: to be used by the **MPRT**. The field is dimmed for **PULSAR**.

**Num Digital Outputs**: edit field is to enter the number of Num Digital Outputs: to be used by the **MPRT**. The field is dimmed for **PULSAR**.



## Test Set Ethernet Communication

The software communication to/from the test set has three option buttons for configure option buttons: **Ethernet**, IEEE-488.2 and Serial.



*Ethernet IP Address* is to set the MAC address for the computer and test set.

**Note:** Ethernet communication is applicable to MPRT only.

### Topics

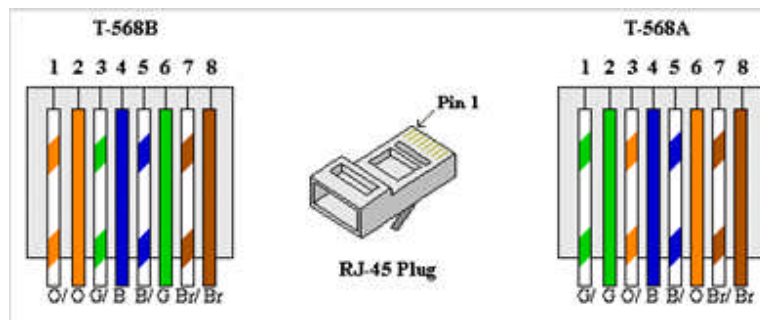
- High Speed Applications for the MPRT Ethernet Port
- Connecting the MPRT to a PC for using AVTS
- Connecting the MPRT to a network for firmware upgrade

## High Speed Applications for the MPRT Ethernet Port

Using ethernet ports and cable provides high speed computer communication to/from the **MPRT**.

Ethernet 10BaseT uses two pairs of wires in a twisted pair configuration, one pair for transmit and the other for receive. Normally a hub is used to connect machines together. When connecting PC's together through a hub or switch, the hub transfers information from all Transmit pairs to all Receive pairs.

Cat 5 (category 5 telecom data transfer speed nomenclature) ethernet 4 pair cable is typically used and readily available. The *wire pair termination standards* are either EIA/TIA T-568A or EIA/TIA T-568B. Most off the shelf termination cable use the T-568B standard.



**Note:** A hub swaps the transmit and receive pairs internally. If a hub is used, standard T-568B cables are used for the interconnections.

The common cable used for an interface through a hub is designated a **Straight-Through** cable. The cable has pin 1 of the plug on one end connected to pin 1 of the plug on the other end, pin 2 to pin 2 and so on for all eight pins. Use a cable with the same standard on both ends.

When interfacing two devices with CPU's directly, each of which is a computer, the **Cross-Over** cable is required. The Cross-over cable has the transmit pair swapped with the receive pair on the same cable end. Use a cable with T-568A standard termination on one end and T-568B standard termination on the other.

**Note:** Be certain to label this cable or purchase one of different color.

## Connecting the **MPRT** to a PC For Using AVTS

The **MPRT** has a single board internal computer (a PC). Both the PC and the **MPRT** have an Ethernet RJ-45 plug and the units may be connected together using a **Cross-Over** cable.

First, obtain the **IP Address and the Default Gateway** from the computer before a connection is made to the **MPRT**.

Click Start

Click Control Panel

Double click network Connections

Select the computer LAN terminal name in the Network Connections dialog

Right click and select Properties to open the LAN\_terminal\_name Properties dialog

In the items list select the Internet Protocol (TCP/IP) item

Click on Properties for the Internet Protocol (TCP/IP) Properties dialog

Make note of the IP Address and the Default Gateway address

Example: IP Address 192.168.1.200

Default Gateway 192.168.1.1

Obtain an IP address automatically

Use the following IP address:

IP address:	192 . 168 . 1 . 200
Subnet mask:	255 . 255 . 255 . 0
Default gateway:	192 . 168 . 1 . 1

**Note:** If the properties are set for "Obtain an IP address automatically", the IP Address and the Default Gateway for the PC may be obtained by pinging in the CMD mode. This will display the IP Address, Subnet Mask and Default Gateway of the computer.

**Note:** When set in the "Obtain an IP address automatically" mode, the PC will always display the first IP Address segments as default. The last two segments will vary on every start of the PC. Thus, only the first two segments must agree between the PC and the **MPRT**.

Click on Start

Click Run, type cmd and click OK to open the DOS window

Type ipconfig

Press the Enter keyboard button

The IP Address will be displayed along with the Subnet Mask.

Check and set the **MPRT Ethernet addresses** using the TVI, Turn on the **MPRT** and select the System Config button

Select the IP Address button to display the IP Address Dialog



The Local Address should be the same IP Address as the computer but removed by one digit and the & Default Gateway should be the same

Example: IP Address 192.168.1.201  
 Default Gateway 192.168.1.1

Click OK

Select File for the System File Manager page

Select File/Save User Sys Def -- Execute -- OK -- Return

The data is now saved to the MPRT and communications is set

The **MPRT** IP Address should be the same as the computer IP Address, except to leave the last two segments as .0.3

For example, If the IP Address is 169.254.162.8, then set the **MPRT** IP Address to 169.254.0.3

Note: From the factory, the **MPRT** default IP Address is 90.0.0.3. When changing the address to agree with the computer address, leaving the last two segments the same as default is less involved.

While still in the CMD DOS Window, Type ping 169.254.0.3

Press Enter keyboard button

An **MPRT response** to the ping will be displayed in the DOS Window and should appear as exhibited here.

```
C:\WINDOWS\system32\cmd.exe
Windows IP Configuration

Ethernet adapter LAN:

    Connection-specific DNS Suffix  . : 
    Autoconfiguration IP Address. . . : 169.254.162.5
    Subnet Mask . . . . . : 255.255.0.0
    Default Gateway . . . . . : 

C:\Documents and Settings\Administrator>ping 169.254.0.3

Pinging 169.254.0.3 with 32 bytes of data:

Reply from 169.254.0.3: bytes=32 time<1ms TTL=64
Reply from 169.254.0.3: bytes=32 time<1ms TTL=64
Reply from 169.254.0.3: bytes=32 time<1ms TTL=64
Reply from 169.254.0.3: bytes=32 time<1ms TTL=64

Ping statistics for 169.254.0.3:
    Packets: Sent = 4, Received = 4, Lost = 0 (0% loss),
    Approximate round trip times in milli-seconds:
        Minimum = 0ms, Maximum = 0ms, Average = 0ms

C:\Documents and Settings\Administrator>_
```

**Next Open AVTS**

Select Tools/Options.../Devices tab

Select Ethernet check button and enter the same IP Address that appears in the System Config dialog on the TVI

Click OK

**Check the connections**

Select Tools/Battery Simulator... and a voltage check button and OK

The Battery simulator voltage indicator should light on **MPRT**

## Connecting the MPRT to a network for firmware upgrade

### Upgrading Firmware in the **MPRT** Using the Ethernet Port and the Internet

In order to download firmware from the internet, the **MPRT** needs access to the internet through a network connection. The **MPRT** integrates into a network just like your Personal Computer. So, what information do you need to connect the **MPRT** to the internet?

After setup of the **MPRT** and the network access port, with assistance from the company resident IT (Information Technologist) this process need not be repeated in the future.

There are several *classes of IP addresses*:

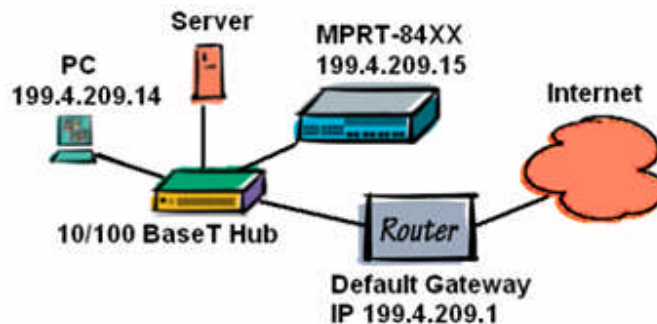
Class A addresses are usually used by backbone providers, those needing many networks (AT&T Cable: 24.X.X.X).

Class B addresses are used by large businesses (Alcatel 143.209.X.X).

Class C addresses are used by small businesses and individuals (The Acme Testing Company 199.4.209.X).

Whichever class of IP address, the MPRT will need a network access port, which will be assigned a unique IP address. Ethernet devices typically have both an IP address and a MAC (Machine Access Control) address. MAC addresses usually can not be changed. It is hard coded into the Ethernet Circuit of the machine and is unique for each PC and the MPRT unit. IP Addresses are eventually mapped to a Mac Address on the destination LAN (Local Area Network). The LAN side of the router has a table of MAC addresses and is able to deliver information from the Internet to that particular machine.

In the network figure, the MPRT has an assigned IP address of 199.4.209.15. The access port will be connected to a hub which is connected to a Router (address of 199.4.209.1), which is the "gateway" to the internet. For MPRT to access the internet, set the IP and Gateway addresses in the MPRT Touch View Interface (TVI). Setting the Local IP Address in the MPRT TVI IP Address Dialog Box will change the Network Mask.



Note: Special cases may exist where sub-net masks are in use. Therefore, IT personnel may need to set the Network Mask address manually.

Example, assume a standard Network Mask has been automatically set.

To complete the address settings, touch the Default Gateway window and set the TVI Gateway IP address to 199.4.209.1.

Press OK, and return to the Main Menu Screen. If the settings are correct and the MPRT is connected to the network, a check should be performed to verify that the addresses are properly connected and set.

To test, "PING" the MPRT unit using your PC (assumes the PC is connected to the network). Use the command "PING" to check the network settings.

Note: "PING" is built into most operating systems. Ping is an easy tool for network troubleshooting.

"PING"ing a system is similar to "Can you hear me now?"

PING is accessed from a DOS prompt.

Go to **Start, Run**, type **Command** (for Win 95/98/ME) or **cmd** (for Win 2000/XP) and click OK to provide a DOS shell.

In the DOS shell, type: **ping X.X.X.X** where each "X" is a network ID, such as "ping 199.4.209.15".

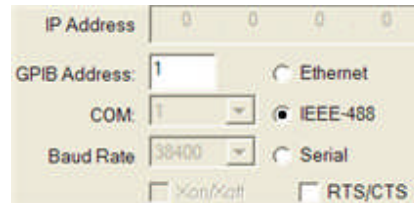
If the ping was successful, upgrade of the MPRT firmware through the internet is ready.

In the MPRT TVI Main Menu, press the Soft/Firm Update button to display the Field Upgrader window. Press the Update Check button for immediate MPRT access to the internet to locate the FTP Server Host for the MPRT firmware upgrade. The process will compare the versions of software and firmware in the MPRT to that which is on the server. If there is a newer version available, a display will ask whether or not to download the new version. To download, press OK to begin. If the process should be interrupted, restart the process. A successful download will be reported in the TVI.

## Test Set IEEE-488 Communication

The software communication to/from the test set has three option buttons: Ethernet, **IEEE-488.2** and Serial. Only one option is available at any one time.

IEEE-488.2 is to set the **GPIB Address**: for the connected instruments. Address settings 1 through 15 are handled by the option.



**Note: IEEE-488.2 communication is applicable to MPRT, PULSAR and TCM-915S.**

### IEEE-488 Overview

Almost any instrument can be used with the IEEE-488 specification, because it says nothing about the function of the instrument itself, or about the form of the instrument's data. Instead the specification defines a separate component, the interface, that can be added to the instrument. The signals passing into the interface from the IEEE-488 bus and from the instrument are defined in the standard.

The instrument does not have complete control over the interface. Often the bus controller tells the interface what to do. The Active Controller performs the bus control functions for all the bus instruments.

### Device Addresses

The IEEE-488 standard allows up to 15 devices to be interconnected on one bus. Each device is assigned a unique primary address, ranging from 0-30, by setting the address switches on the device. A secondary address may also be specified, ranging from 0-30. See the device documentation for more information on how to set the device primary and optional secondary address.

### Physical Characteristics

You can link devices in either a linear, star or combination configuration using a shielded 24-conductor cable. The standard IEEE-488 cable has both a plug and receptacle connector on both ends. This connector is the Amphenol CHAMP or Cinch Series 57 MICRO RIBBON type.

The IEEE-488 bus specifies a maximum total cable length of 20 meters with no more than 20 devices connected to the bus and at least two-thirds of the devices powered on. A maximum separation of 4 meters between devices and an average separation of 2 meters over the full bus should be followed. Bus extenders and expanders are available to overcome these system limits.

Most communication for the TCM-915S is handled by use of a laptop computer PCMCIA card for cable connection. When the card is purchased, operating software install disk accompanies the card.

If a card is not available, it is advised to purchase one from

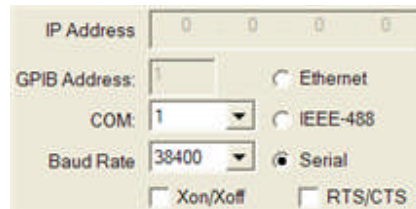
National Instruments  
 Austin, Texas  
 Phone: 1.800.531.5066, or go to <http://www.ni.com/niglobal/> to locate a dealer.

For PCMCIA IEEE-488.2 Card order data, go to  
<http://sine.ni.com/nips/cds/view/p/lang/en/nid/3646> for Windows 2000/XP, or  
<http://sine.ni.com/nips/cds/view/p/lang/en/nid/1234> for Windows ME/98/95

## Test Set Serial Communication

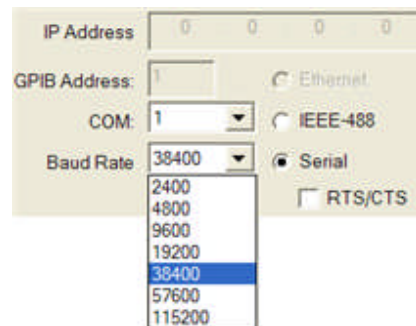
the software communication to/from the test set has three option buttons: Ethernet, IEEE-488.2 and **Serial**. Only one option is available at any one time.

Serial is to set a *COM: port number*, Baud Rate and Flow Control.



**Note:** Serial communication is applicable to MPRT and PULSAR only.

The *communication Baud Rate speed*, using a RS-232 cable and connections, is selected from a drop list of options. The options are 2400, 4800, 9600, 19200, 38400, 57600 and 115200. The MPRT test set has reliable use of all the options. Select to option most comfortable or convenient.

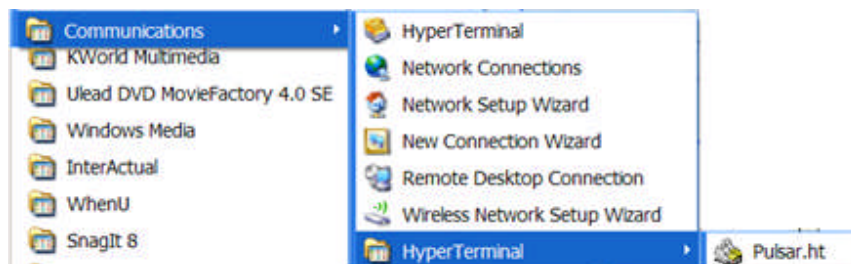


**Note:** The PULSAR test set communication speed via COM serial is device limited to 19200 maximum. However, common usage is 9600 baud for reliability.

Xon/Xoff for Flow control of the data. This may be left unchecked in some cases where the None or Hardware control is set in the COM Properties dialog.

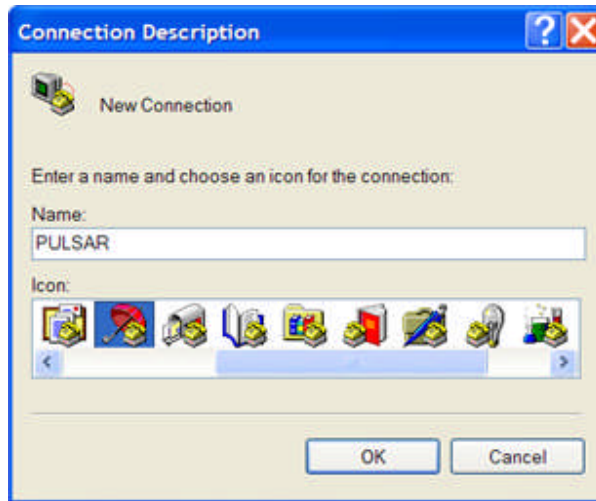
RTS/CTS (Ready To Send/Clear To Send) is to set the option if checked.

Normal *call for serial communication* is usually with the Windows Hyperterminal, found in Windows by clicking Start, Programs, Accessories, Communication and Hyperterminal.

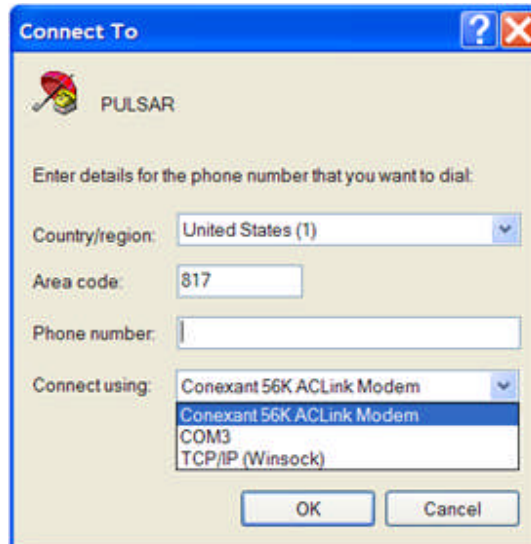


**Note:** A previous setting for the PULSAR has been saved and is ready for use in the example above.

The *New Connection dialog* will appear to enter a name for the connection and an icon for its identity if a shortcut is placed on the computer desktop. Select OK.

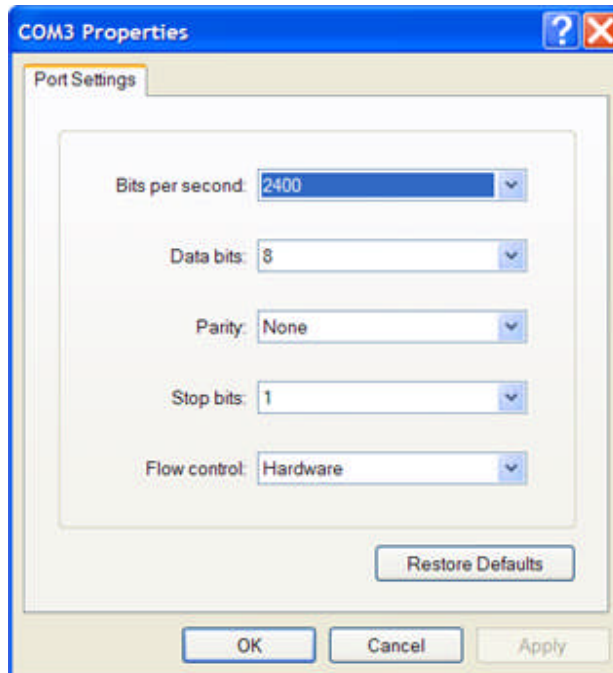


The *Connect To dialog* will appear for selections, which will generally for use with AVTS, the test sets and other devices, will be for an available COM port. Select the COMnum and OK.

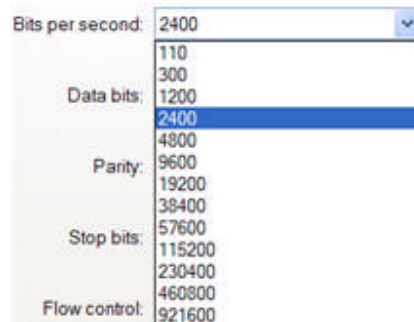


The *COMnum Properties Port Settings* dialog will appear for selecting the

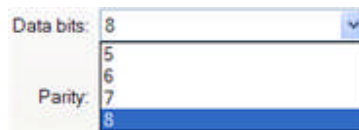




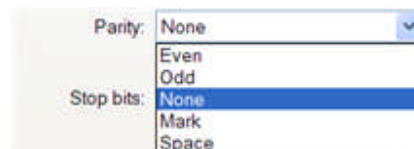
**Bits per second:** (Baud) to select from the drop list the same option selected in AVTS for the equipment.



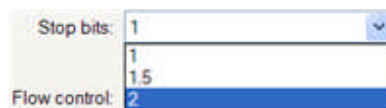
**Data Bits:** to select from the drop list, Select 8 bits.



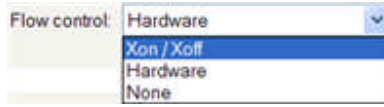
**Parity:** to select from the drop list, Select None.



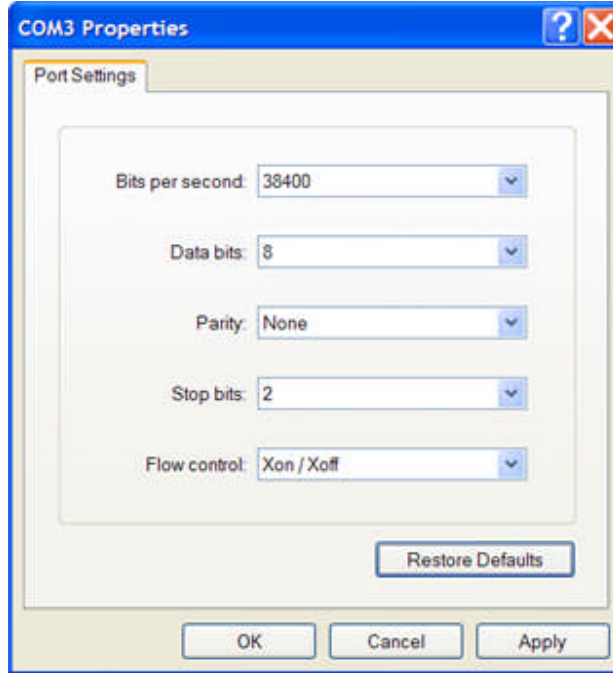
**Stop bits:** to select from the drop list, Select 1.



**Flow control:** to select from the drop list, Xon/Xoff if checked in the Devices tab or None if unchecked in the Devices tab.



The selections in this dialog need be those for proper communication to the test equipment. Typical *settings for MPRT* shown.



Select OK to open the RS-232 COM port and made active for direct communication the test equipment.

## TCM-915S IEEE-488 Communication

The software communication to/from the test set uses the IEEE-488.2 GPIB (General Purpose Interface Bus). This is the only interface option for setting the instrument with a Sequence file and for retrieving recorded and stored event data.

Three TCM-915S instruments may be utilized simultaneously for testing. Each must have a GPIB Address assigned for communication identity. The address is set by dip switches on the back of the instrument and may be set binary 1 through 15. The instrument will need to be turned off and then On for a new address to be recognized. Set the *GPIB Address: edit field* with the numeric binary address for the instrument to be recognized by the TCM A, TCM B or TCM C options.

	TCM A	TCM B	TCM C
Installed	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Log Commands	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Simulate Device	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
GPIB Address:	<input type="text" value="1"/>	<input type="text" value="1"/>	<input type="text" value="1"/>

See the TCM-915S Users Manual for further details. Sequence and Recorder information accompanies the AVO PulseMonitor software Help.

### IEEE-488 Overview

Almost any instrument can be used with the IEEE-488 specification, because it says nothing about the function of the instrument itself, or about the form of the instrument's data. Instead the specification defines a separate component, the interface, which can be added to the instrument. The signals passing into the interface from the IEEE-488 bus and from the instrument are defined in the standard. The instrument does not have complete control over the interface. Often the bus controller tells the interface what to do. The Active Controller performs the bus control functions for all the bus instruments.

### Device Addresses

The IEEE-488 standard allows up to 15 devices to be interconnected on one bus. Each device is assigned a unique primary address, ranging from 0-30, by setting the address switches on the device. A secondary address may also be specified, ranging from 0-30. See the device documentation for more information on how to set the device primary and optional secondary address.

### Physical Characteristics

You can link devices in either a linear, star or combination configuration using a shielded 24-conductor cable. The standard IEEE-488 cable has both a plug and receptacle connector on both ends. This connector is the Amphenol CHAMP or Cinch Series 57 MICRO RIBBON type.

The IEEE-488 bus specifies a maximum total cable length of 20 meters with no more than 20 devices connected to the bus and at least two-thirds of the devices powered ON. A maximum separation of 4 meters between devices and an average separation of 2 meters over the full bus should be followed. Bus extenders and expanders are available to overcome these system limits.

Most communication for the TCM-915S is handled by use of a laptop computer PCMCIA card for cable connection. When the card is purchased, an operating software install disk accompanies the card.

If a card is not available, it is advised to purchase one from

National Instruments  
Austin, Texas

Phone: 1.800.531.5066, or go to <http://www.ni.com/niglobal/> to locate a dealer. Call the dealer to determine the model card to support the GPIB IEEE-488.2 standard.

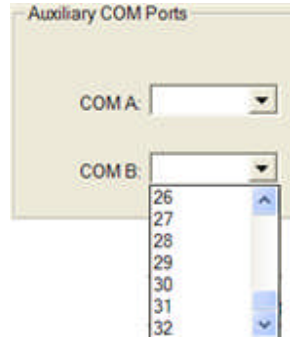
For an existing PCMCIA IEEE-488.2 Card software drivers ready to download, go to <http://www.ni.com/support/gpib/versions.htm#pc> and

Select the appropriate PC operating system column,  
Scroll down to the

PCMCIA-GPIB+ (PC Card) row and select the version software to download.

## Auxiliary COM Ports

*Auxiliary COM Ports* settings are to select an installed COM port to be used to control auxiliary connected equipment.

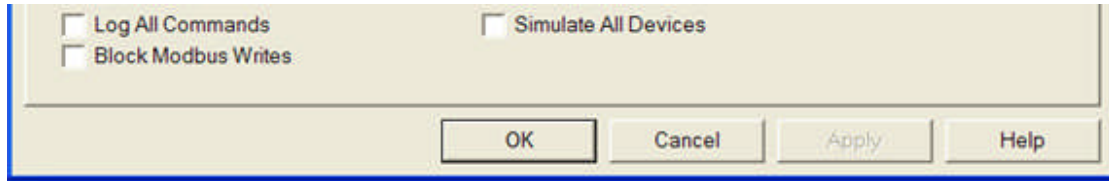


The drop list has a blank and then 1-32 as port number options. A selected port number commits the port to use by AVTS and is under immediate control when a connect device is active.

A typical use is to allow communication to microprocessor relays which may have settings changes occur during test execution. The communication is controlled by the addition of the 1-Touch or BASIC Control in the AVTS Workbook Test Editor tab.

## Devices Tab Lower Area

The *Devices tab lower area* contains three check boxes.



**Log All Commands**, when checked, will create a text record of all commands sent to all test instruments installed and controlled by AVTS. All the responses returned from the installed instruments will also be logged. Should problems occur during the test of edited or newly created test procedures, this record is useful in determining the root of the problem. The problem might also be a result of incorrect relay settings reflected in AVTS compared to the response capability of the relay. The logged commands may be viewed with Ctrl + L for to View Instrument Log...

**Note:** This check box is checked by default with the initial installation of AVTS. It is recommended that this box be unchecked. Left checked, the Instrument Log will ultimately be filled beyond use.

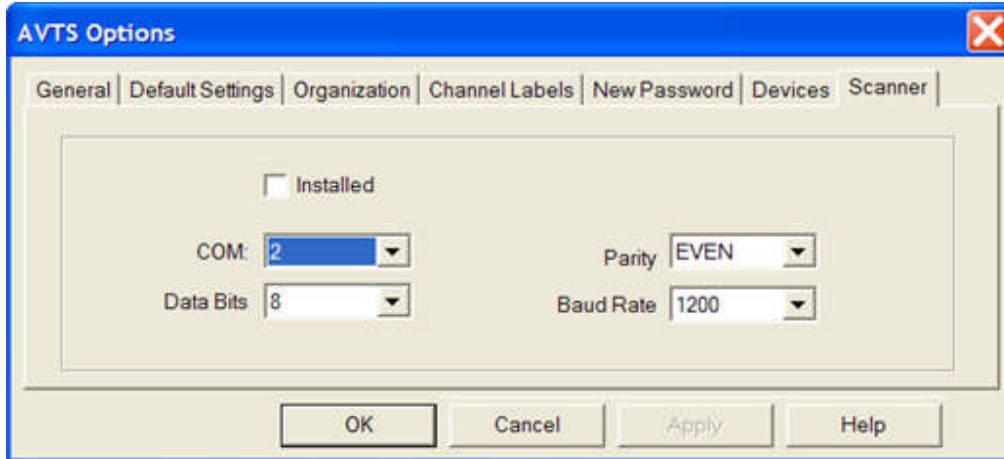
**Simulate All Devices**, when checked, will allow the execution of a test and allow the computer and user to emulate all supposed test instruments and relay under test. Simulate All Devices is also very useful in checking execution of edited and newly created tests.

**Note:** This check box is checked by default with the initial installation of AVTS. It is recommended that this box be unchecked and only be used when the constructed AVTS test utilizes equipment additional to the Test Set A or Test Set B.

**Block Modbus Writes** check box will disable all modbus writes to a relay

## Scanner Tab

The AVTS Options **Scanner** tab is from the Tools/Options menu item.



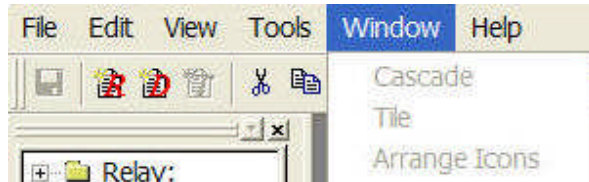
The Scanner tab is for setting the COM port, See Set Serial Communication topic, and settings for a Bar Code Scanner. The user should consult the manufacturer's instruction manual for the desired settings for the scanner in use. These are the settings to be entered into the Scanner tab.

When the proper communication settings are in place, the Installed check box will need to be checked to enable the Scanner. Once enabled the Scanner may be used to scan a bar code on a relay to automatically open the relay test module.

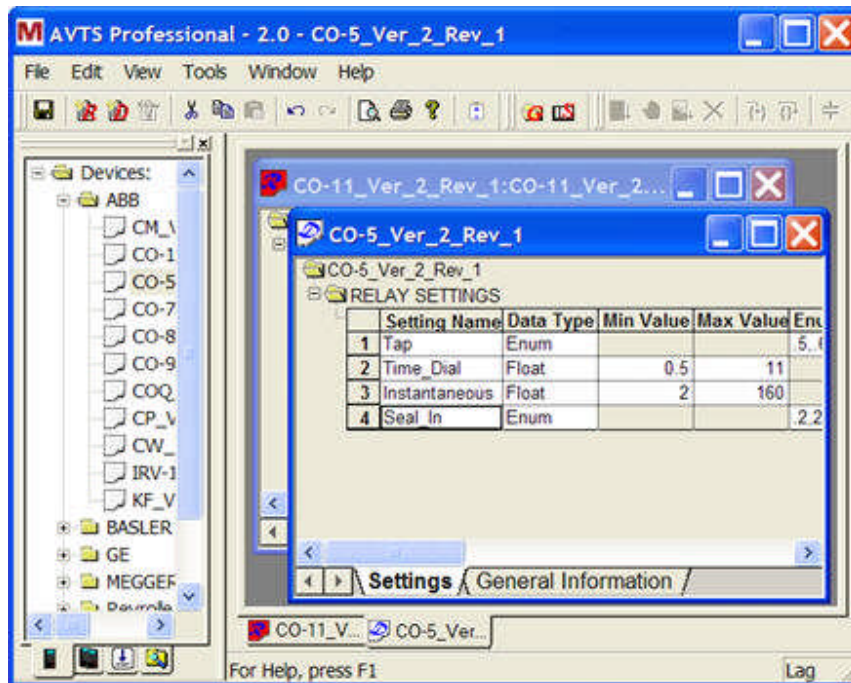
**Note:** When using a "keyboard wedge" type of scanner, do not check the "Installed" check box. Load the scanner software driver from the disk supplied with the scanner. The scanner will be enabled any time AVTS is requesting input from the keyboard. These older scanners were typically connected in the cable between the keyboard and computer. Most new scanners connect directly to the computer through one of the computer USB ports.

## Main Menu Window

The **Window** menu list is used to select display options for the Workbook pane. There must be a Relay or Device open in the Workbook for the Window drop down list to be viewed.

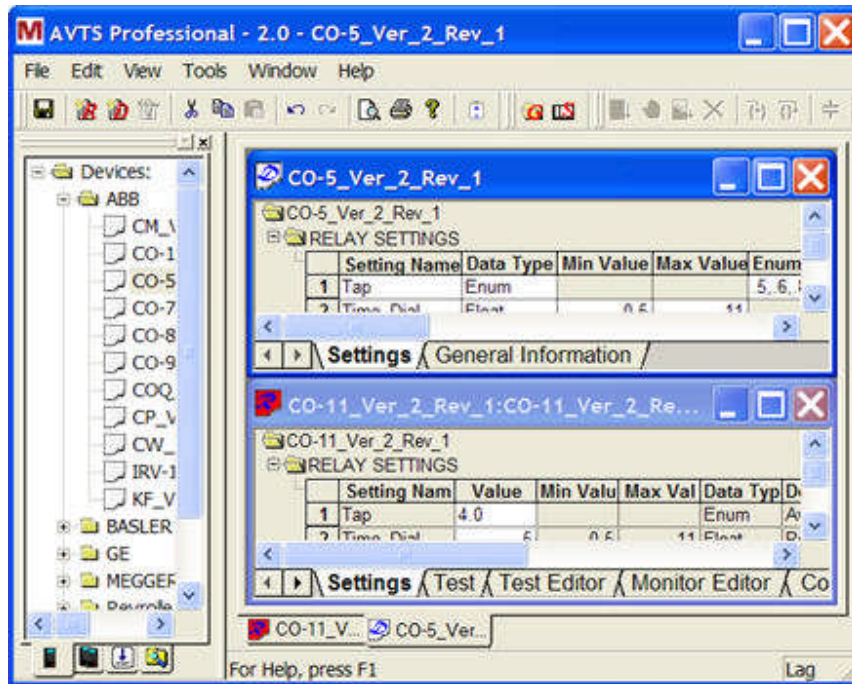


The **Cascade** option is a standard Windows function for the Workbook. With the cursor on the Cascade option, the message "arrange widows so they overlap" will show in the status bar. When two or more Devices and/or Relays open, Cascade will arrange the Workbook to display all the open elements as overlap pages.



The **Tile** option is a standard Windows function for the Workbook. With the cursor on the Tile option, the message "arrange widows as non-overlapping tiles" will show in the status bar. When two or more Devices and/or Relays open, Tile will arrange the Workbook to display all the open elements as shown:

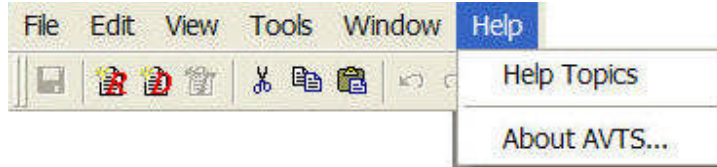




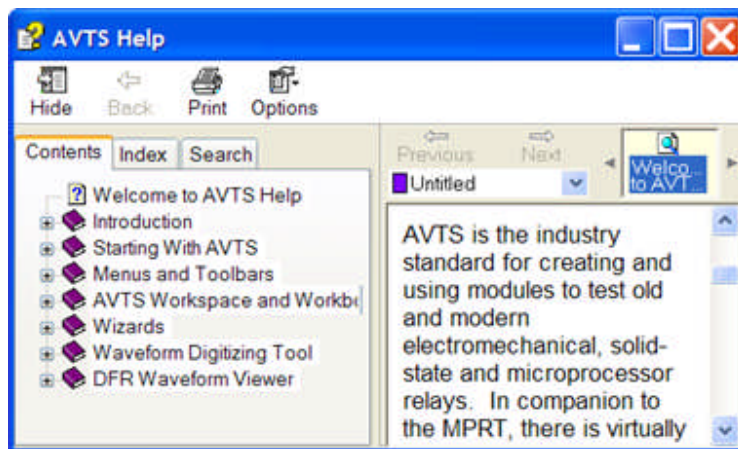
The Arrange Icons option will display the message "arrange icons at the bottom of the window" in the status bar when the cursor is on the Tile option.

## Main Menu Help

The *Help* menu list is a Windows standard drop down.



The *Help Topics* option will open the AVTS Help files in HTML format to display subject topics arranged in a book and topic format for ease in locating information about particular uses of AVTS. The Help also contains Contents, Index and Search tabs for additional assistance in locating particular subjects for help.



The *About AVTS...* option opens the About AVTS dialog to display the present version of AVTS installed.



## Main Menu Toolbar

The Main Toolbar is also a Standard Toolbar when AVTS is first opened.

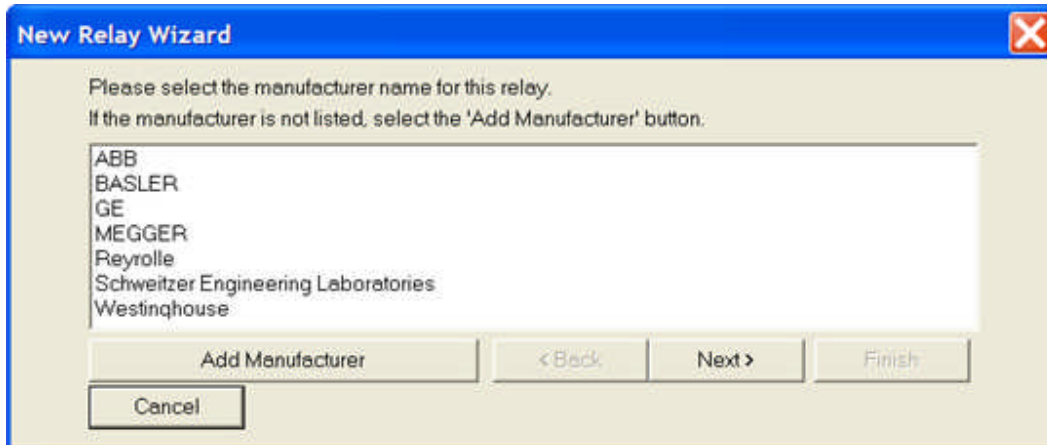
The Standard Toolbar, shown below, displays the tool icons for use in various displays. The dimmed icons are those that are available for use only when specific panes or dialogs are open.

The Standard Tool icons listed from left to right above are:

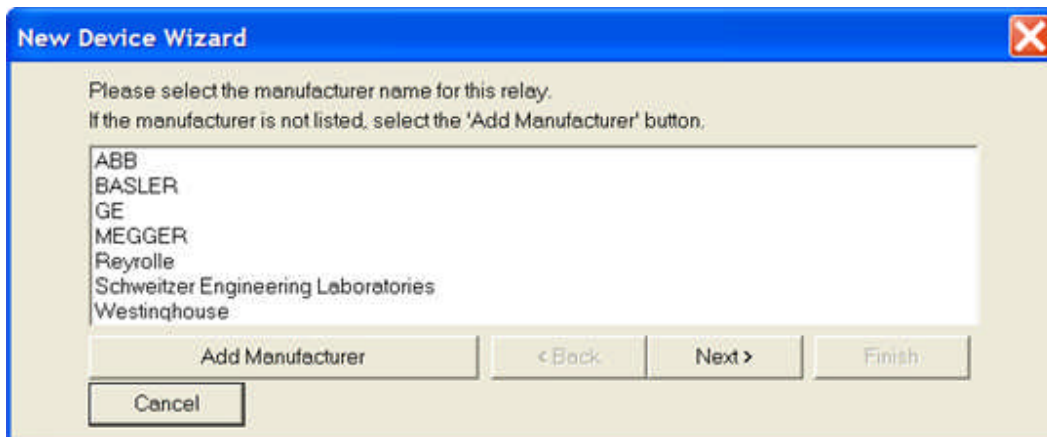


**Save** -- Click on, if not dimmed, will save all work completed in AVTS since the last Save function. This function is a duplicate to that in the File menu item when a Device or Relay in the Workspace has been opened. This tool is part of the Standard Toolbar.

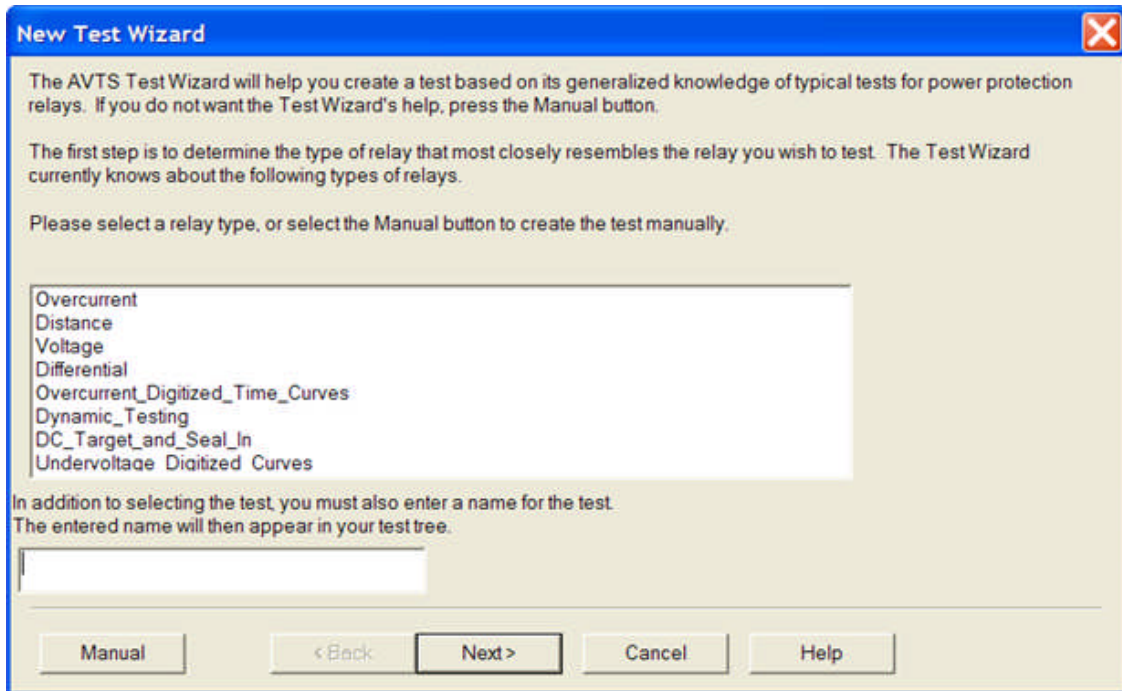
**New Relay** -- Click on will open the *New Relay Wizard* dialog for the user to create a new Relay in the Relay Tab Workspace. This tool is part of the Standard Toolbar. The New Relay Wizard topic has full details on use.



**New Device** -- Click on will open the *New Device Wizard* dialog for the user to create a new Device in the Device Tab Workspace. This tool is part of the Standard Toolbar. The New Device Wizard topic has full details on use.



**New Test** -- Click on, if not dimmed, will open the *New Test Wizard* dialog for the user to create a new Test in the Test Workspace Tab for an open Relay. This tool is part of the Standard Toolbar. The New Test Wizard topic has full details on use.



**Cut** -- Click on will remove a selected object, text or group of objects and hold them in a clipboard for later use. This tool is part of the Standard Toolbar.

**Copy** -- Click on will create a duplicate of a selected object, text or group of objects and hold them in a clipboard for later use. This tool is part of the Standard Toolbar.

**Paste** -- Click on will place the last object, text or group of objects last placed in a clipboard by a previous Cut or Copy action. This tool is part of the Standard Toolbar.

**Undo** -- Click on, if not dimmed, will delete the last action of the user. Multiple click on's will continue deleting the previous actions in historic order. This tool is part of the Standard Toolbar.

**Redo** -- Click on, if not dimmed, will replace a delete action. Multiple click on's will continue the replace action until all the previous delete actions are replaced. This tool is part of the Standard Toolbar.

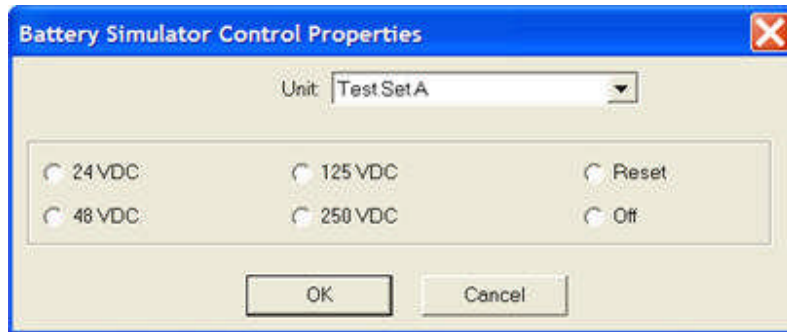
**Print Preview** -- Click on, if not dimmed, will display the existing Workbook display in a view of a printed page, should the user desire to print the selection. This tool is part of the Standard Toolbar.

**Print** -- Click on, if not dimmed, will proceed to invoke action to a connected printer to print the selected display. This tool is part of the Standard Toolbar.

**About** -- Click on will open the *About AVTS* dialog to display the present version of AVTS installed. This tool is part of the Standard Toolbar.



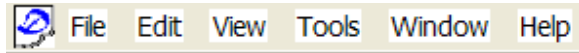
**Battery Simulator** -- Click on, if not dimmed, will open the *Battery Simulator Control Properties* dialog for immediate control of the connected test set. This tool is part of the Standard Toolbar.



## Device Menu and Toolbars

### Device Menu and Toolbars

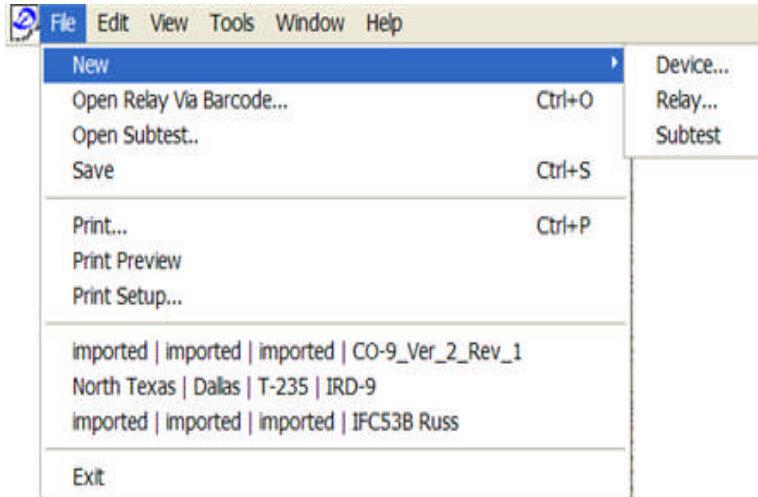
The Device Menu items will have differences when a Device has been opened from the Workbook pane. Any available Tools on the toolbar relating to Devices will become available for use. There are items in the menus that have been defined in the Main Menu and Toolbars topics and will not be redefined in this set of topics.



- Device Menu File
- Device Menu Edit
- Device Menu View
- Device Menu Tools
- Device Toolbar

## Device Menu File

The *Device Menu File* items are defined below.



New Device...

New Relay...

Open Relay Via Barcode...

**Save** operation will place all edited items in the Workspace and all edited items open in the Workbook panes to the active database.

**Print...** is the standard Windows item to evoke the Print dialog for selecting a printer and the print parameters to be sent to the printer along with the material to be printed.

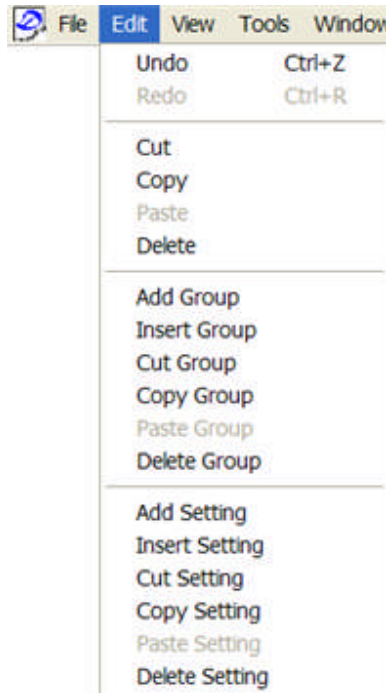
**Print Preview** is the standard Windows preview screen. The screen displays a view of selected subject in a format as it will be printed. The margins and other document settings will be displayed also.

Print Setup...

**Exit** Item if selected will close the AVTS program.

## Device Menu Edit

In addition to the menu items defined in the topic [Main Menu Edit](#), the following items are selectable when a Relay has been opened in the Workbook. Other Edit menu items may be visible when a Relay has been opened in the Workbook. These are defined in the topic, [Relay Menu Edit](#).



The Edit menu items that pertain to an open Device are:

**Undo**, when selected, will return the previous screen prior to an Edit. The operation is not enabled for editing changes in the General Information tab. If in one of the Workbook Editor tabs and an edit has been performed within a Control dialog, the Control will need to be closed to perform the use of the Undo function.

**Redo**, when selected, will return the previous screen prior to an Undo. The operation is not enabled for editing changes in the General Information tab. If in one of the Workbook Editor tabs and an edit has been performed within a Control dialog, the Control will need to be closed to perform the use of the Undo function.

**Cut** option will remove a selected object, text or group of objects and hold them in a clipboard for later use.

**Copy** option will create a duplicate of a selected object, text or group of objects and hold them in a clipboard for later use.

**Paste** option will place the last object, text or group of objects last placed in a clipboard by a previous Cut or Copy action. Text preciously copied to the clipboard may be Paste into the information panes of the General Information tab.

**Delete** Select to remove the object, text or group of objects selected by the cursor. The operation may be annulled by using the Undo option.

**Add Group** Select to add a new Group of settings at the bottom of the list of Groups in the Workbook pane. The new Group will have the default settings added that appear in the Tools/Options/Default Settings tab.

**Insert Group** Select to add a new Group of settings above a selected Group in the list of Groups in the Workbook pane. The new Group will have the default settings added that appear in the Tools/Options/Default Settings tab.



**Cut Group** Select to remove a selected Group. The removed Group is reserved and available to be Insert into a location above a selected Group or Paste at the end of the list of Groups.

**Copy Group** Select to create a clone of a selected Group. The copied Group is reserved and available to be Insert into a location above a selected Group or Paste at the end of the list of Groups.

**Paste Group** Select to place a Cut or Copy Group at the end of the list of Groups.

**Delete Group** Select to permanently remove a selected Group from the list of Groups. The action may be reverted by choosing the Undo menu item.

**Add Setting** Select to add a new setting at the bottom of the list of Settings in the highlighted Group.

**Insert Setting** Select to add a new Setting above a selected Setting in the list of settings of the highlighted Group.

Note: Multiple Settings may be selected for Insert, Cut, Copy, Paste or Delete by holding the Ctrl key and clicking on those to be applied.

**Cut Setting** Select to remove a selected Setting. The removed Setting is reserved and available to be Insert into a location above a selected Setting or Paste at the end of the list of Settings.

**Copy Setting** Select to create a clone of a selected Setting. The copied Setting is reserved and available to be Insert into a location above a selected Setting or Paste at the end of the list of Settings.

**Paste Setting** Select to place a Cut or Copy Setting at the end of the list of Settings.

**Delete Setting** Select to permanently remove a selected Setting from the list of Settings. The action may be reverted by choosing the Undo menu item.

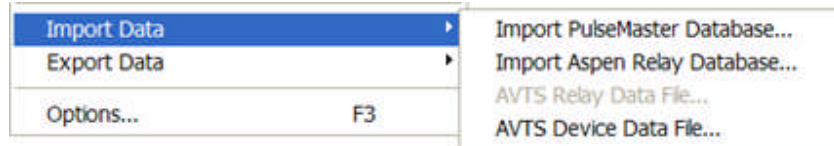
## **Device Menu View**

The Device Menu View items are identical to those defined in the topic Main Menu Edit.

## Device Menu Tools

- Online Vector Control...
- Online Ramp Control...
- Online Click On Fault Control
- Calculator...
- Fault Calculator...
- HyperTerminal...
- SS1 Converter...
- Device Comm. Software...
- Waveform Digitizing Tool...
- DFR Waveform Viewer...
- Battery Simulator

### Import Data -



- Import PulseMaster Database...
- Import ASPEN Relay Database...
- AVTS Device Data File...

### Export Data -



- AVTS Device Data File...
- Options...

## Device Toolbar

The Device Toolbar, shown below, is a portion of the standard toolbar and displays the tool icons for use in various displays. The dimmed icons are those that are available for use only when specific panes or dialogs are open.

The Device Tool icons listed from left to right above are:



**New Group** -- Click on, if not dimmed, will create a new Settings Group in an open Device Workbook pane.

**New Setting** -- Click on, if not dimmed, will create a new Setting in an open Settings Group of a Device Workbook pane.

Details for using these tools are in the topic Device Settings Tab Drop Menu.

## Relay Menu and Toolbars

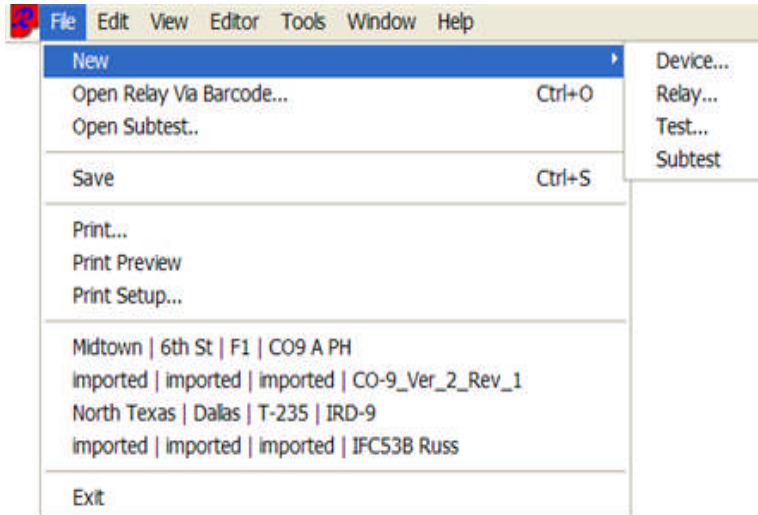
### Relay Menu and Toolbars

The Relay Menu items will have differences when a Relay has been opened into the Workbook pane. Any available Tools on the toolbar relating to Relays will become available for use. There are items in the menus that have been defined in the Main Menu and Toolbars topics and will not be redefined in this set of topics.

- Relay Menu File
- Relay Menu Edit
- Relay Menu View
- Relay Menu Editor
- Relay Menu Tools
- Relay Menu Window
- Relay Menu Help
- New Relay Wizard
- New Device Wizard
- New Test Wizard

## Relay Menu File

The **File** menu, with a relay open in the Workbook pane, has drop down list items additional to the Main Menu File.



### New

**Device...** item allows the creation of a new Device to be created in the Workspace Device Tab. Selection of this item will open the New Device Wizard. After the necessary steps have been followed to create the new Device, it will appear in the list of Workspace Devices Tab with the Device open and displayed in the Workbook.

**Relay...** item allows selection of creating a new Relay to be created in the Workspace Relay Tab. Selection of this item will open the New Relay Wizard. After the necessary steps have been followed to create the new Relay, it will appear in the list of Workspace Relays with the first test displayed in the Workspace Tests Tab and open in the Workbook.

**Test...** item allows the creation of a new Test when a Relay has been opened and the Workspace Tests tab is open. The user will first select the Workspace Test after which the new Test will be placed. Selection of the New Test item will open the New Test Wizard. After the necessary steps have been followed to create the new Test, it will appear in the list of Workspace Tests and placed after a test that was first selected in the Workspace.

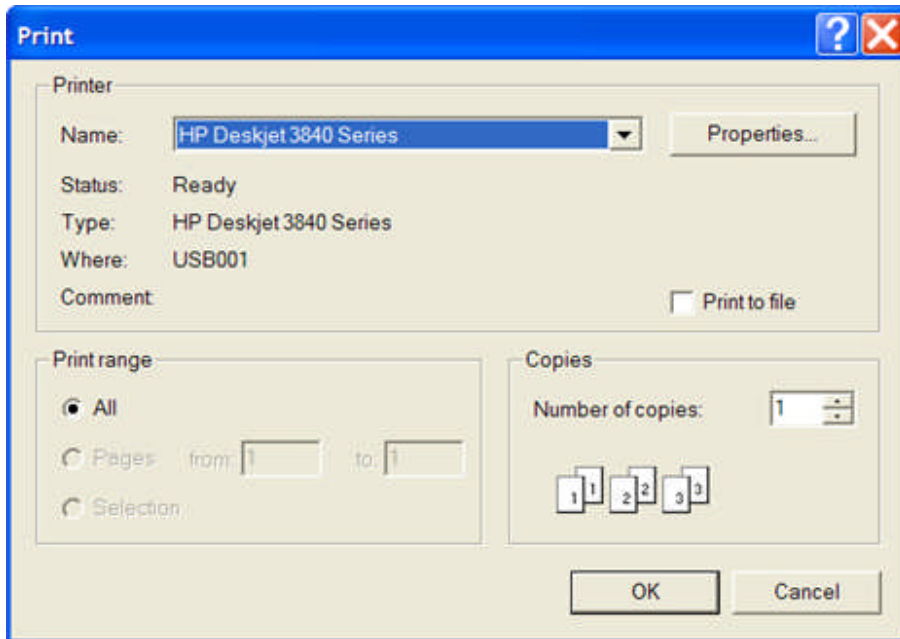
**Subtest...** item is described in the File New Subtest topic.

**Open Relay via Barcode...** item allows the opening of a relay by scanning the barcode on the relay or by entering the barcode using the keyboard in the open Relay Barcode dialog.

**Open Subtest...** item is described in the File Open Subtest topic.

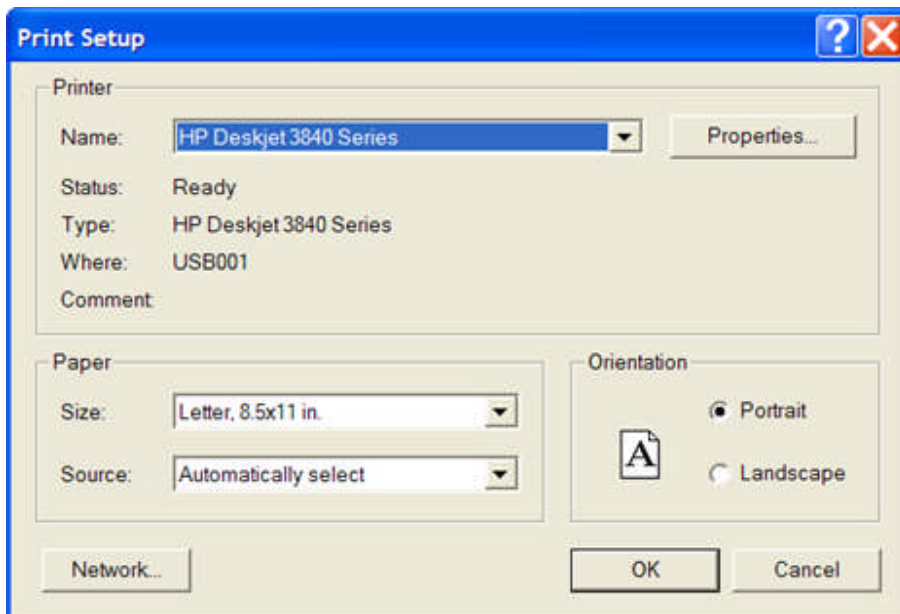
**Save** operation will place all edited items in the Workspace and all edited items open in the Workbook panes to the active database.

**Print...** item evokes the standard Windows **Print** function with the capability to selected paper type, paper arrangement and number of copies to be printed.



**Print Preview...** item will display contents of the Settings, Test Editor, Monitor Editor, Connections Editor and BASIC tabs in the format that the selected tab will be printed if desired. The preview is only available for these tabs in the Workbook. The Test and General Information tabs are not enabled to be printed directly. Therefore, these two tabs are not accessible in Print Preview.

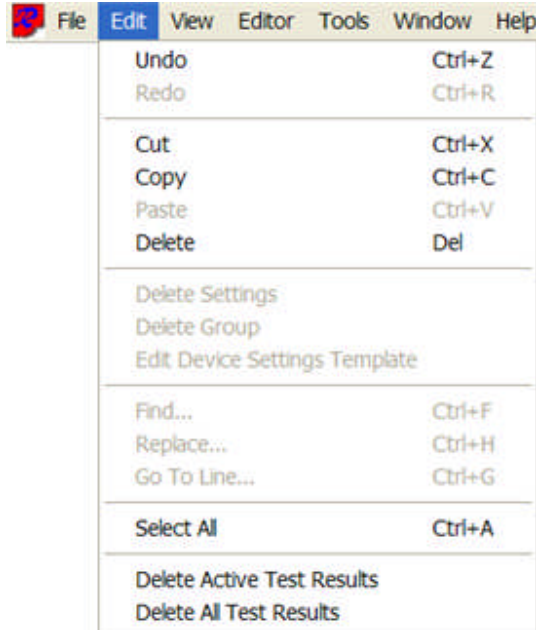
**Print Setup...** item opens the standard Windows *Print Setup* dialog for controlling the output print appearance. The user is able to select a printer connected to the computer or network, select paper source and size, and paper orientation. The next area is reserved for a list of files of the most recently opened devices and relays in the database presently open. The number of files listed is limited to the last six that were opened.



**Exit** item if selected will close the AVTS program.

## Relay Menu Edit

In addition to the menu items defined in the topic Main Menu Edit, the following items in the *Menu Edit* are selectable when a Relay has been opened in the Workbook. The items **Delete Settings** and **Delete Group** are reserved for use with a Device that has been opened in the Workbook and are defined in the topic Device Menu Edit.



The following items are those useful with the open Relay:

**Undo**, when selected, will return the previous screen prior to an Edit. The operation is not enabled for editing changes in the General Information tab. If in one of the Workbook Editor tabs and an edit has been performed within a Control dialog, the Control will need to be closed to perform the use of the Undo function.

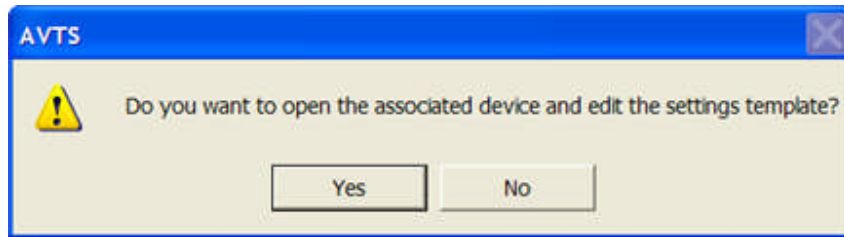
**Redo**, when selected, will return the previous screen prior to an Undo. The operation is not enabled for editing changes in the General Information tab. If in one of the Workbook Editor tabs and an edit has been performed within a Control dialog, the Control will need to be closed to perform the use of the Undo function.

**Edit Device Settings Template** is usable when in the *Settings* tab screen. Should an edit be required in any of the gray fields of a Settings Group, selection of this item will open the Device parent to the open Relay.

	Setting Nam	Value	Min Valu	Max Val	Data Typ	Description
1	Tap	5			Enum	Available CC
2	Time Dial	4	0.5	11	Float	Relay Time
3	Instantaneous	2	2	160	Float	Relay Instan
4	ICS	2			Enum	ICS Setting
5	Tap_Range	2			Enum	1 = 5-2.5 ar
6	MTA	30			Enum	MTA set as
7	Tap	5.0			Enum	Available CC

When selected, an *association message* will appear with the option to open the Device or not.





If Yes, the *Device Settings* tab for the parent Device of the Relay will open for edit.

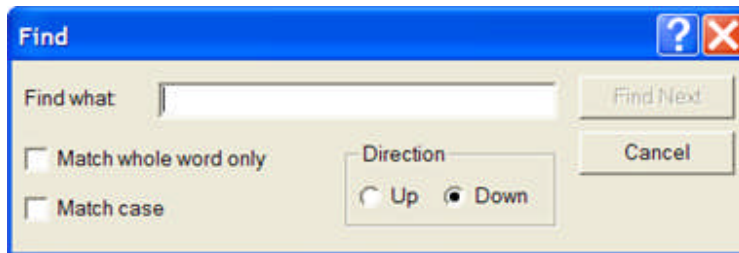
CR-9

CO-2 TO CO-11 RELAY SETTI

	Setting Name	Data Type	Min Value	Max Value	Enum Strings
1	Tap	Enum			5, 6, 8, 10, 15
2	Time Dial	Float	0.5	11	
3	Instantaneous	Float	2	160	
4	ICS	Enum			2, 2
5	Tap Range	Enum			1, 2
6	MTA	Enum			30

If No, a return to the Relay Settings tab will occur.

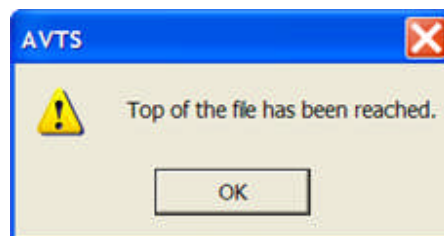
**Find** is usable within the open Relay BASIC tab only. A macro file of any type or any MPRT or Pulsar syntax used in this BASIC tab will usually have several lines of code. When selected, the *Find* dialog is evoked.

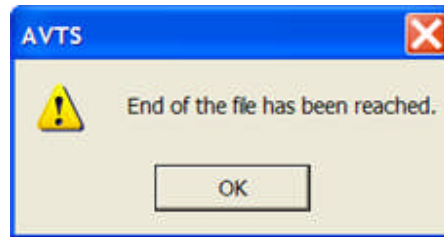


To locate a specific word in the code, type the word into the Find what field, select the Match options desired and the Up/Down direction to search from the present cursor position in code.

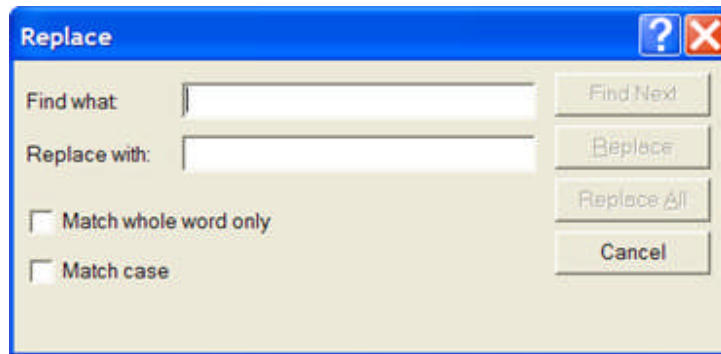
Within the BASIC tab, place the word or phrase desired into the Find What field. Select Find Next and, if the word or phrase exists, it will be brought to view and highlighted. To find other locations for the same Find What entry, select Find Next to locate the next occurrence either Up or Down from the present cursor location.

If no other locations are found, a AVTS message will be generated to advise having reached either the *End of the file* or *Top of the file* has been reached.

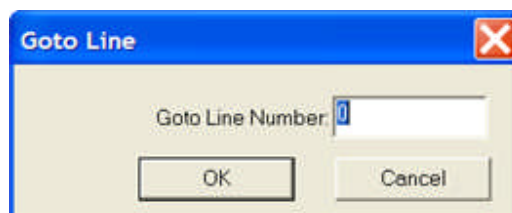




**Replace** is similar to Find. When selected, the *Replace* dialog is evoked. Enter the word or phrase wanted into the Find What field and the word or phrase that will replace the original into the Replace With field. Select the Match option and then Find Next. When located and the proper location highlighted, select Replace to perform the operation. Select Cancel if the found highlighted word or phrase is not the correct location. If no other locations are found, a message will be generated to advise having reached either the end of or top of the file. Select Replace All to replace all locations that might be found with continued searches. Replace All may be selected without first selecting Find Next.



**Go To Line...** when selected, the *Goto Line* dialog is evoked. Place a line number in the edit field and select OK. The cursor will be placed at the beginning of that line and displayed. The line number for the cursor location will also be displayed on the Status Bar below the BASIC tab. If the cursor is on line 50 and **Goto Line Number** 99 is entered, the cursor and line will be at the bottom of the display. If the cursor is on line 100 and **Goto Line Number** 33 is entered, the cursor and line will be at the top of the display.



**Select All** is usable in the Edit and BASIC tabs. In the active Edit tab all the Controls and Links will be selected for editing using the enabled tools on the toolbar.

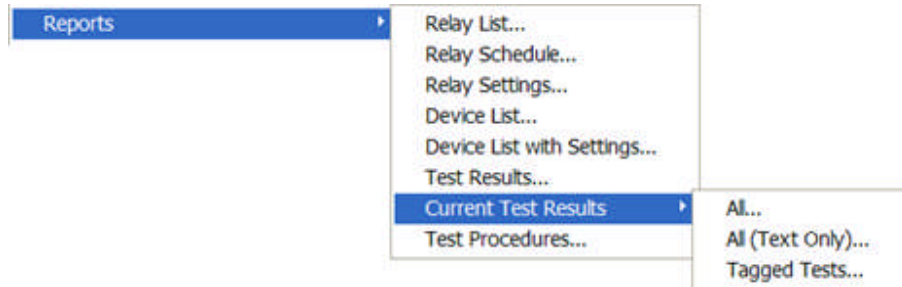
**Delete Active Test Results** will delete the test result, if available, of the active test in the open Relay.

**Delete All Test Results** will delete all the test results for the selected relay.

## Relay Menu View

## Relay Menu View

The *Relay Menu View* displays a menu item drop down list displays the available options as shown below.

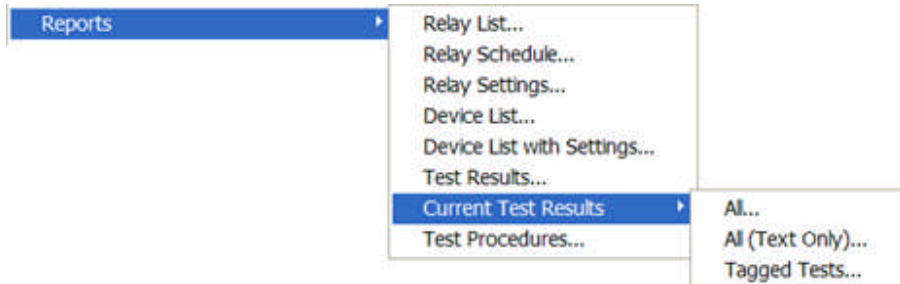


The following items are explained in detail in their linked topics. All other items in the drop down list are detailed in Main Menu View topic.

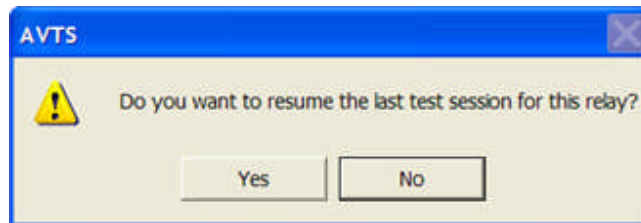
- Current Test Results
- Test Procedures...
- Variable Watch
- Add All Watch Variables
- Remove All Watch Variables
- Update Theoretical

## Current Test Results

To view the Relay View Menu *Current Test Results* a relay must be open in the Relay Workbook.



If a relay has been opened and there have not been any tests executed and test results obtained, then the Current Test Results options are dimmed. If, when the relay was opened, the option to *Resume Last Test* message was allowed, then the Current Test Results options are available to create reports.



The Current Test Results item has three options for report types.

**The All...** selection will generate a report for all the test results of an open relay.

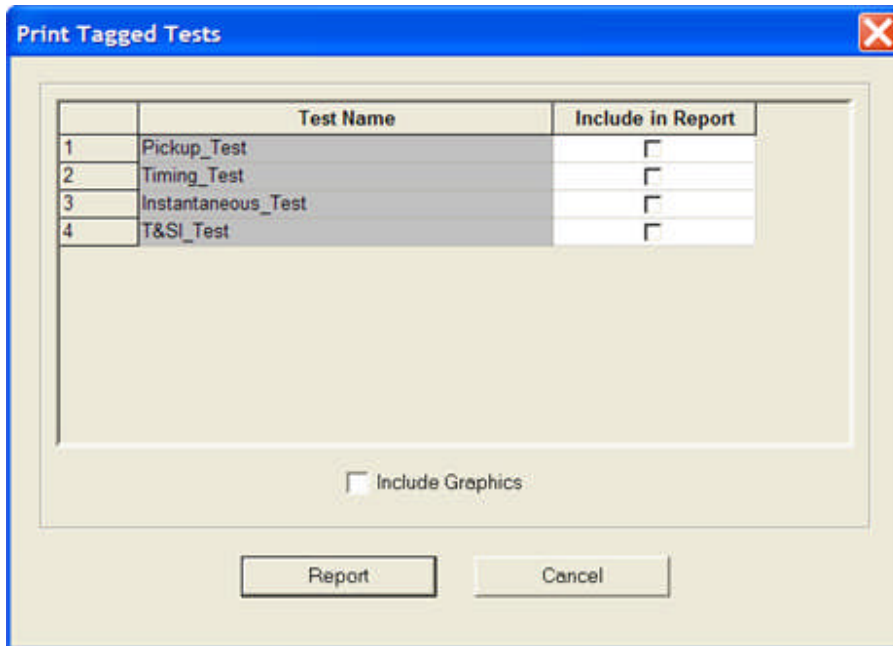
**Note:** If more than one relay is open, each with current test results, the report generated will be for only that relay which is selected to be active.

**All (Text Only)...** selection will generate a report for all the test results of an open relay. The report will be of text form and contain no graphic results.

**Note:** If more than one relay is open, each with current test results, the report generated will be for only that relay which is selected to be active.

**Tagged Tests...** selection will open the *Print Tagged Tests* dialog. The dialog will list all tests by Test Name currently resident in the Workspace Tests tab, whether or not all of the tests have been executed with results obtained. The Include in Report column has check boxes for placing a check to label those tests to be included in the report. An additional check box is available in the dialog to Include Graphics or not.

**Note:** If more than one relay is open, each with current test results, the report generated will be for only that relay which is selected to be active.



## Test Procedures

The Test Procedures... will open the *Test Procedure Report* containing written statements for the Test Editor Controls that are used to create a particular test. A relay must be open in the Workbook and the test for the report must be highlighted in the Workspace.

### Test Procedure Report

For Device IAC-51B  
Manufactured By GE  
Monday, February 27, 2006

Device Notes: IAC-51B Inverse Time Relay  
Test Name: Timing\_Test

Control Name List  
Value N: 5

Direction	Control Name	Source Bar	Target Bar
From	PC Timer	Timeout	Next
To	End	Done	Execute
To	Vector	Continue	Execute
From	Calculation	Done	Execute

Control Name: HW Timer  
Comments:

Unit: Test Set A  
Auxiliary Contact: No Action  
Debounce Time: 1.7  
Time Variable: PulsarTimeA  
Mode: No Action  
Timer Start Latch: Yes  
Timer Start Enable Horn: No  
Timer Stop Latch: Yes  
Timer Stop Enable Horn: No  
Timer Stop Turn Currents Off: Yes  
Start Post: Software  
Stop Software: 3  
Start Mode:  
Stop Mode: Continuity  
Start Active On:  
Stop Active On: Applied  
Stop Max Time to Operate Seconds: 20

Direction	Control Name	Source Bar	Target Bar
From	Vector	Done	Wait for Stop
To	Log	Done	Execute

Note: The Test Editor tab must be selected and the cursor must be clicked once in the Workbook before requesting the report.

## Variable Watch

The Relay View Menu item *Variable Watch* will open a sub-pane in the Workbook Relay Test tab. During the execution of a test, variables not visible may be entered line by line in the watch window. The variables are updated during execution if the variables are changing during the test. This example lists the variable Ia and displays the last know value, 4.9594.

The screenshot shows a software interface for a relay test. On the left, there are several data fields: 'Current A' with a value of 4.959 A, a time field with 0.0000 s, a phase angle field with 0.0°, and a frequency field with 60.000 Hz. Below these are four status buttons: '1 FALSE', '2 FALSE', '3 FALSE', and 'Aux OPEN'. In the center, a cyan box displays 'Pickup Test'. On the right, a table with two columns, 'Variable' and 'Value', shows the variable 'Ia' with a value of 4.9594.

Variable	Value
Ia	4.9594

Additional variables are entered by placing the cursor in a blank Variable column field and typing the variable name. Once entered in the field the Enter key must be pressed. A response to the entry and Enter is immediate.

A list of variables recognized by AVTS may be viewed in the Variables topic. All variables may be viewed by applying the Add All Watch Variables item. Other variables may be created within a test procedure by identifying them by formula using existing known variables and values of floating point and/or integer numbers.

## Add All Watch Variables

The Relay View Menu item *Add All Watch Variables* will open a sub-pane in the Workbook Relay Test tab displaying all know variables presently in the active database. Variables are listed in symbol/alphabetic order. The variables are updated during execution if the variables are changing during the test. This example lists all variables and displays the last know value for each. Notice in the image that the vertical scroll bar is showing only a few variables.

Variable	Value
CofActualTripStateStr	0
CofExpTripState	0
CofExpTripStateStr	0
CofLogPoint	0
CofPassFailFlag	0
CofPassFailStr	0
CofTripTime	0
CofVectorNum	0
CofZMag	0
CofZPhase	0
csrlin	0
DataEntry	1
date\$	
debug	0
debug2	0
debug3	0
debug4	0
delay	0
delay2	0
dfrtoevn	0
dfrtofil	0
dfrtonul	0
dfrwidth	0
distort	0
DownArrow	0
Dropout	0
EnterKey	0
EscapeKey	0

A list of variables recognized by AVTS may be viewed in the Variables topic. Other variables may be created within a test procedure by identifying them by formula using existing known variables and values of floating point and/or integer numbers.



## **Remove All Watch Variables**

The Relay View Menu item Remove All Watch Variables will hide a Variable Watch sub-pane if open in the Workbook Relay Test tab.

## **Update Theoretical**

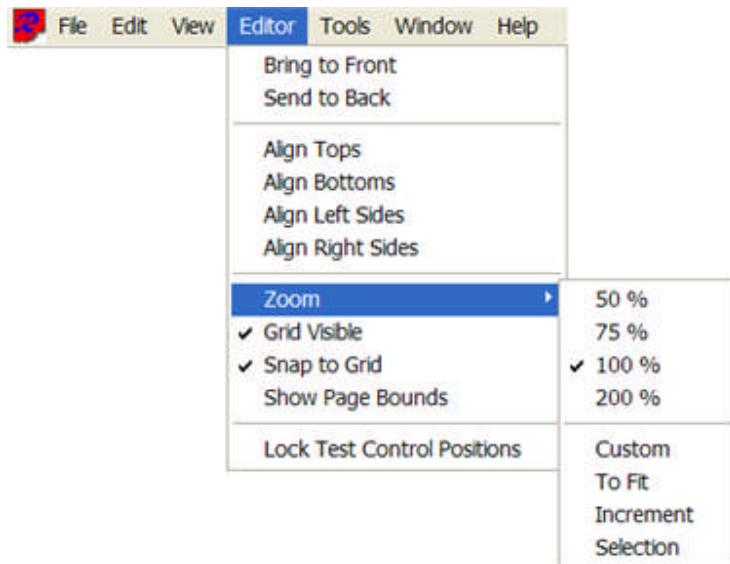
If Workbook Test Editor test procedure has a Graphic Control in the test procedure, there will be a characteristic displayed in the bottom pane of the Workbook Test tab during and after the first test execution.

If edits to the Graphic or Theoretical Control have been made in the Test Editor tab, select the View/Update Theoretical option to have the edits act upon and display a change in the characteristic. The Update Theoretical option alleviates having to execute the test to produce the altered characteristic.

## Relay Menu Editor

### Relay Menu Editor

The *Editor menu list* is associated with and available only with an open relay in the Workbook pane. This menu list allows adjustments and controls while working in the Test Editor, Monitor Editor and Connections Editor tabs of the open relay.

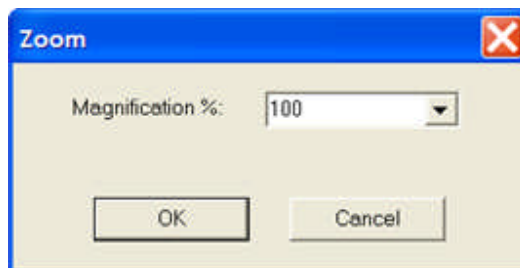


Each section of the drop down list is associated with a toolbar below the Menu bar. In their listed order they are:

The topic Structure Toolbar includes the Bring to Front and Send to Back menu items and are defined in the linked topic.

The topic Align Toolbar includes the Align Tops, Align Bottoms, Align Left Sides and Align Right Sides menu items and are defined in the linked topic.

The **Zoom / Pan** Toolbar drop down menu item Custom invokes the *Zoom dialog* for entry of a specific value of Zoom. A positive integer is required for the Custom zoom. The toolbar includes the Zoom (50%, 75%, 100%, 200%), Zoom to Fit (To Fit), Zoom to Selection (Zoom to Selection) and Pan and are defined in the Zoom / Pan Toolbar topic.



When checked, **Grid Visible** allows a grid of dots to assist in placement and alignment of the Controls in the editor pane.

When checked, **Snap to Grid** forces all controls to be centered to a grid dot. When an object is moved, it will "jump" from dot to dot during the move. When unchecked, an object is not forced to center on a grid position and may be placed between grid dots. The object will be moved in a smooth transposition.

When checked, **Show Page Bounds** will be one dashed horizontal from side to side and two dashed vertical lines from top to bottom dividing the workbook Test Editor pane area into 2 rows of 3 sheets for view and print. For paper conservation, only the sheets containing objects will be printed. The Monitor Editor pane will contain only one page and not display page bounds, even though the menu item is allowed. The Connections Editor pane will contain one vertical page bound separation dashed line to separate the pane into two printable pages.

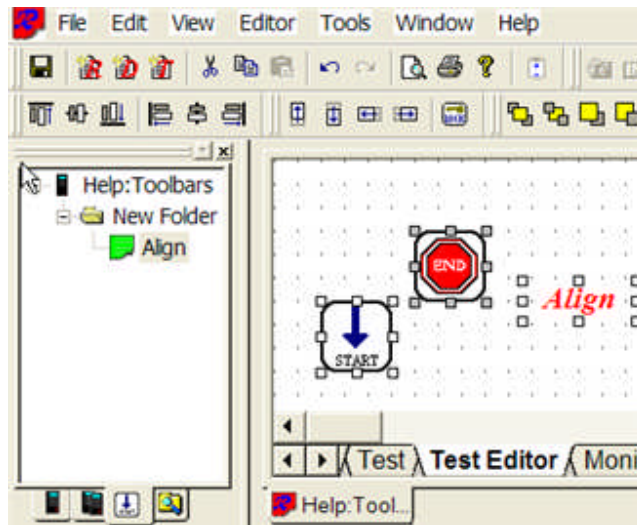
The Lock Test Control Positions Tool drop down menu item is defined in the linked topic.

## Editor Align Toolbar

The **Align** Tools will become active when objects are selected in the Test Editor Tab, Monitor Editor Tab, and Connections Editor Tab topics.



There are six tools that complement the four tools in the Editor menu drop down list. From left to right the Tools are Align Top, Align Middle, Align Bottom, Align Left, Align Center and Align Right. *Relay Align example.*



For the Align tools to be of use, more than one object must be selected. The example above is taken from the Test Editor tab and the Start and End Controls. They have been selected as evidenced by the eight action squares around each Control.


Multiple objects are selected by pressing and holding the keyboard Ctrl key and selecting the objects with a single left mouse click.

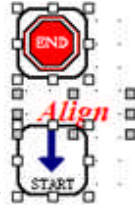
Objects may also be selected by placing the mouse at a location. While holding the left mouse button down, drag the mouse to create a box around the objects to be selected. Once all the desired objects are contained in the box, releasing the mouse button will create the eight squares of selection around each of the objects.

In this example, the darker action squares on the End control indicate it was the last object selected. This will be the anchor Control to which the others will be oriented by the tool used.

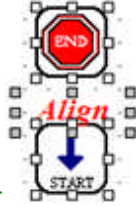

 Select the *Align Top* and the Start Control will move up to the location that places both objects at the same level.


 The *Align Middle* will locate the objects with their horizontal centerlines are at the same level.


 The *Align Bottom* will locate the objects with their bottoms at the same level.



The *Align Left* will locate the objects with their left sides in vertical alignment.



The *Align Center* will locate the objects with their centers in vertical alignment.



The *Align Right* will locate the objects with their right sides in vertical alignment.

## Editor Nudge Toolbar

The **Nudge** Tools will become active when object(s) are selected in the Test Editor Tab, Monitor Editor Tab, and Connections Editor Tab topics. There are items for this toolbar in the Editor menu drop down list.



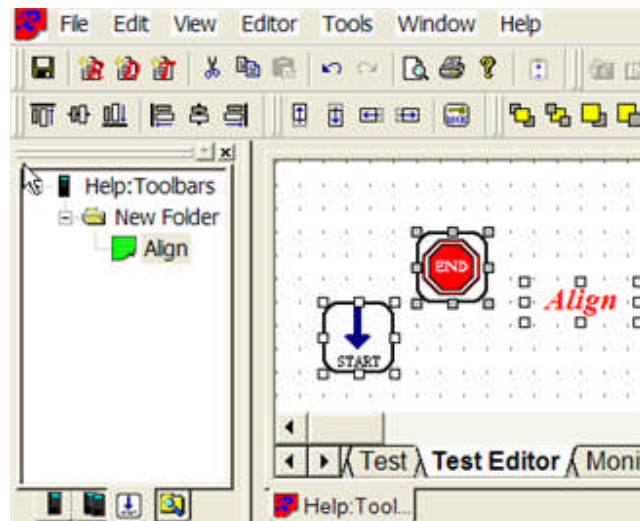
The four tools are Nudge Up, Nudge Down, Nudge Left and Nudge Right. The tool Lock Test Control Positions is included with this toolbar.

For the Nudge tools to be of use, one or more objects must be selected. The example above is taken from the Test Editor tab and the Start, End and Text Controls. They have been selected as evidenced by the eight action squares around each Control.

Multiple objects are selected by pressing and holding the keyboard Ctrl key and selecting the objects with a single left mouse click.

Objects may also be selected by placing the mouse at a location. While holding the left mouse button down, drag the mouse to create a box around the objects to be selected. Once all the desired objects are contained in the box, releasing the mouse button will create the eight squares of selection around each of the objects.

*In this example*, the darker action squares on the End control indicate it was the last object selected.



**Nudge Up** tool will move all objects selected a very small distance in the up direction with each left mouse click.

**Nudge Down** tool will move all objects selected a very small distance in the down direction with each left mouse click.

**Nudge Left** tool will move all objects selected a very small distance to the left direction with each left mouse click.

**Nudge Right** tool will move all objects selected a very small distance to the right direction with each left mouse click.

**Lock Test Control Positions** tool when clicked will make inactive and dim the Test Editor Controls bar on the right side of the pane. The Controls in the pane will also be locked in their current position. When selected and a move of an object is attempted, it will move with the mouse, but will spring back to its locked position when the mouse is released.

## Editor Structure Toolbar

The **Structure** Tools will become active when object(s) are selected in the Test Editor Tab, Monitor Editor Tab, and Connections Editor Tab topics.



The four tools are Front, Back, Forward and Backward.

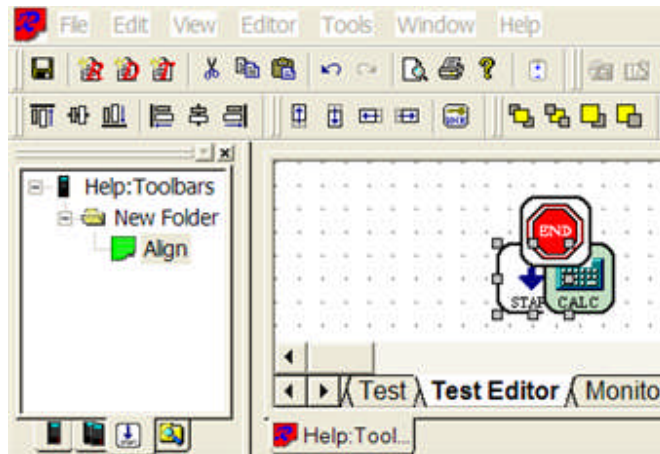
For the Structure tools to be of use, one or more objects may be selected. The example above is taken from the Test Editor tab and the Start, End and Calculate Controls. The Start Control has been selected as evidenced by the eight action squares around the Control.

Multiple objects are selected by pressing and holding the keyboard Ctrl key and selecting the objects with a single left mouse click. The last object selected will be the object that responds to the use of the Structure tools.

Objects may also be selected by placing the mouse at a location. While holding the left mouse button down, drag the mouse to create a box around the objects to be selected. Once all the desired objects are contained in the box, releasing the mouse button will create the eight squares of selection around each of the objects.



*In the example,* selecting the **Front** tool will move the Start Control to the front of the other two Controls.



Selecting the **Back** tool will return the Start Control to the rear of the other two Controls.





Selecting the *Forward* tool will move the Start Control to the middle of the other two



Controls. Selecting the *Forward* tool once again will move the Start Control one more level forward, or, in this example, to the front.



Selecting the *Backward* tool will move the Start Control to the middle of the other two



Controls. Selecting the *Backward* tool once again will move the Start Control one more level backward, or, in this example, to the rear.

## Editor Zoom / Pan Toolbar

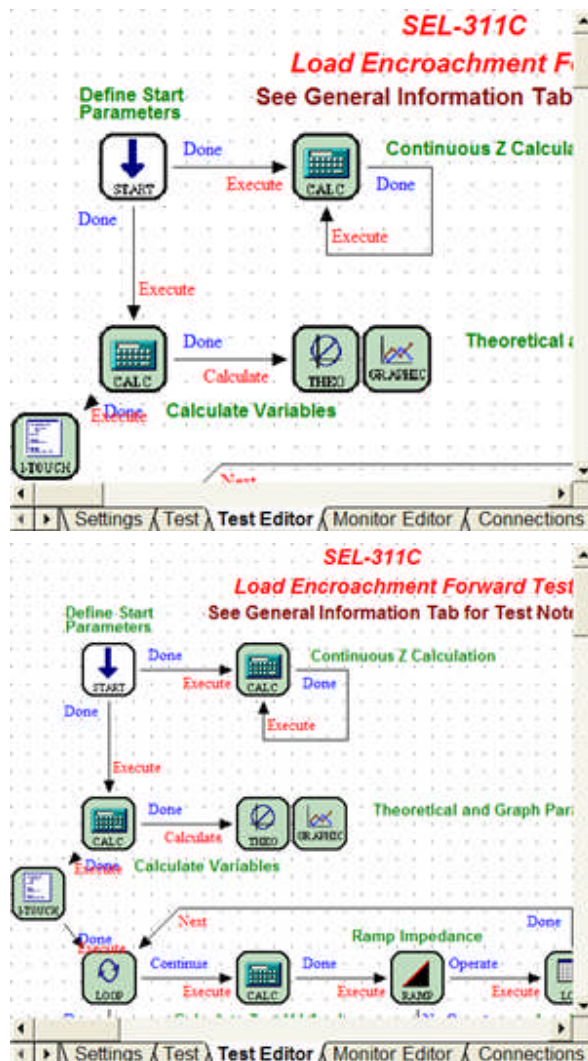
The Zoom/Pan Tools will become active when object(s) are selected in the Test Editor Tab, Monitor Editor Tab, and Connections Editor Tab topics.



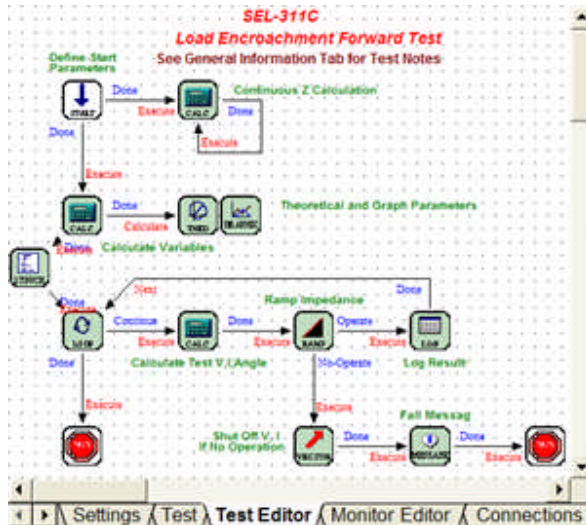
The four tools are Zoom, Zoom to Fit, Zoom to Selection and Pan.

The Zoom / Pan tools will prove very helpful in viewing complex procedures that may not readily be viewed completely without scrolling the pane.

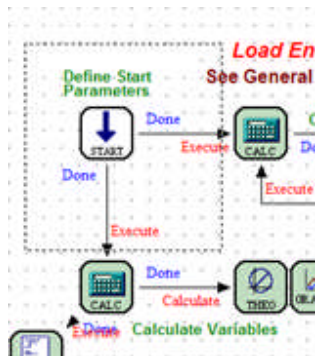
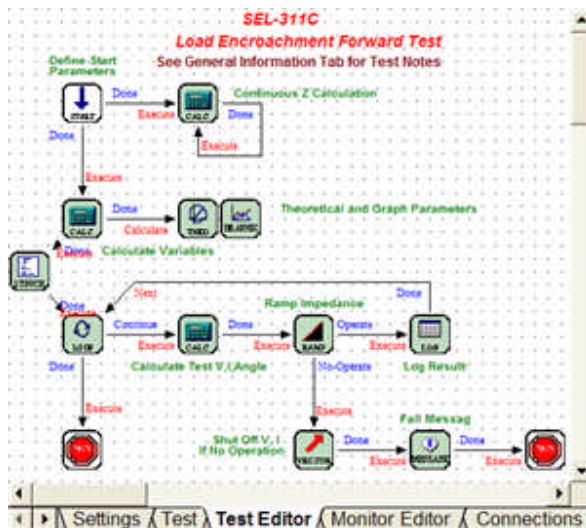
*In the example*, selecting the Zoom tool and executing a single left mouse click in the editor pane will *Zoom-in* the pane in increments. The Undo options are not enabled after a Zoom is executed.



Selecting the Zoom to Fit tool and executing a single left mouse click in the editor pane will *Zoom-out* the pane to make visible all controls in the test procedure within the present Workbook pane size. Should the test procedure occupy a small space in the Test Editor pane, the Zoom to Fit will zoom-in the test procedure to occupy the full pane.



Selecting the *Zoom to Selection* tool is made active only with the selection of one or more objects in the editor pane. The objects are selected by left mouse clicking each while holding the Ctrl key down, or by a left mouse click and drag to box in the wanted objects. The objects selected will then be *zoomed-in* or zoomed-out to occupy the full pane.

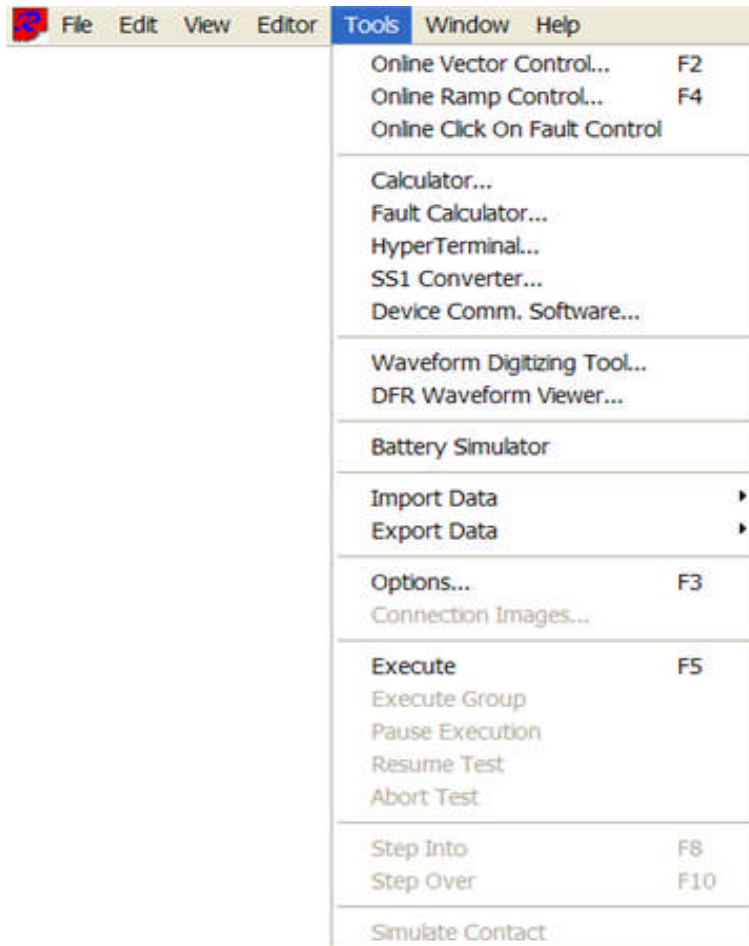


Selecting the Pan tool will alter the normal cursor to a "hand", indicating a Windows Pan tool. Holding the left mouse button down while placing the hand in the editor pane will allow scrolling of the editor pane to view all the available space in the pane. Right clicking in the pane will return the cursor to normal and leave the view just panned.

## Relay Menu Tools

## Relay Menu Tools

In addition to the menu items defined in the topic Main Menu Tools, the following additional *Relay Menu Tools* items are selectable when a Relay has been opened in the Workbook.



Device Comm. Software... is to evoke the communication link to the connected equipment set in the Tools/Options/Devices tab. This item becomes available when a Relay is opened into the Workbook pane and is listed also in the Device Menu Tools.

### Import Data item

- AVTS Test Data File...
- MPRT Results
- MPRT Settings

### Export Data items

- Copy Results to Clipboard
- AVTS Test Data File...
- Results to Word
- Current Test Results to Word
- MPRT Settings

Connection Images...

Execute option is to begin a selected test for a Relay open in the Workbook.

Execute Group option is to begin a group of tests selected from the Tests tab.

Pause Execution option is to stop the test process.

**Note: Caution is necessary as this action does not turn OFF any active test set channel outputs.**

Resume Test is to continue a test which has been stopped at some part of the test process.

Abort Test is to halt an active executed test and turn all test set channels OFF.

Step Into

Step Over

Simulate Contact

## Relay Tools Import Data

### Relay Tools Import Data

In addition to the menu items defined in the topic Main Menu View, the following additional items are selectable when a Relay has been opened in the Workbook.

**Device Comm. Software...** is to evoke the communication link to the connected equipment set in the Tools/Options/Devices tab.

Relay Import AVTS Test Data File... describes the process for the Tools/Import Data.../AVTS Test Data File... menu option.

**Import Data/MPRT Results** is to call from the test files saved from an MPRT and display the Settings and Test Results in a Relay Settings tab.

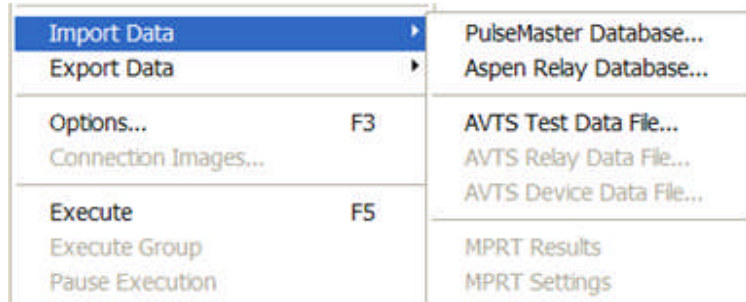
Relay Import MPRT Results describes the process for the Tools/Import Data.../MPRT Results menu option.

**Import Data/MPRT Settings** is to call from the test files saved from an MPRT and display the Settings in a Relay Settings tab.

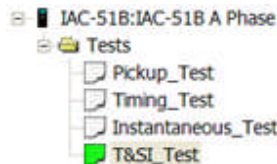
Relay Import MPRT Settings describes the process for the Tools/Import Data.../MPRT Settings menu option.

## Relay Import AVTS Test Data File...

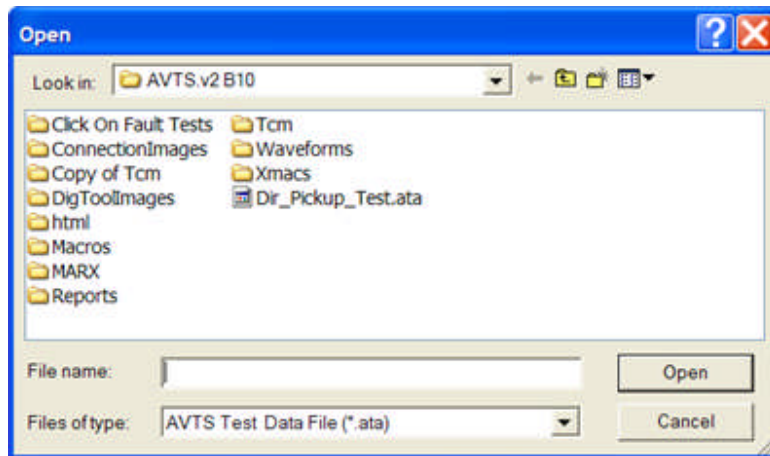
Prior to importing an *AVTS Test Data File...*, the Workspace Tests tab must be selected and the location in the Workspace Tests tab lists must be selected. The purpose is to have the option to import only a relay test from one database to another database.



Select the test location in the *Workspace* and then the Import option to begin the process.



The **AVTS Test Data File...** option opens the *Windows Open* dialog. The "Files of type:" is defaulted to "AVTS Test Data File (\*.ata)" for segregating the test data files from all others.



Browse and select the desired AVTS Test Data File, \*.ata (accessrelayAVTS). The selected database name will appear in the File Name edit field.

Select the Open button to proceed with the Import AVTS test data file process.

AVTS will place the file, example "Export-Import Dir\_Pickup\_Test.ata", into the Workspace Tests tab at the selected location.

## Relay Import MPRT Results

The Tools/Import Data...Import/MPRT Results and MPRT Results Settings menu items allow the population of a particular relay with test results saved from a MPRT.

Use of the Import Settings option is described in the Relay Import MPRT Settings topic.

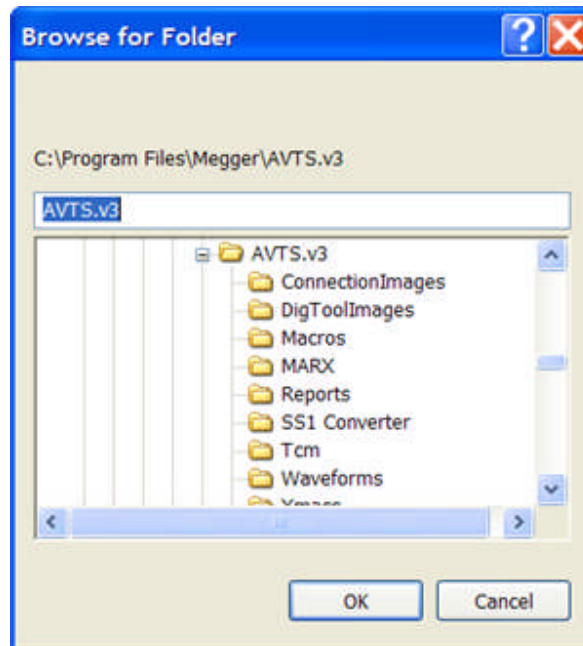
Use of the Export Settings option is described in the Relay Export MPRT Settings topic.

To Import and view the Results in each file and to create a test report for record, the following is the necessary procedure:

Establish communication between the PC and the **MPRT**. If successful, test files previously saved in MPRT using the **MPRT TVI** will display in the AVTS MPRT Workbook tab.

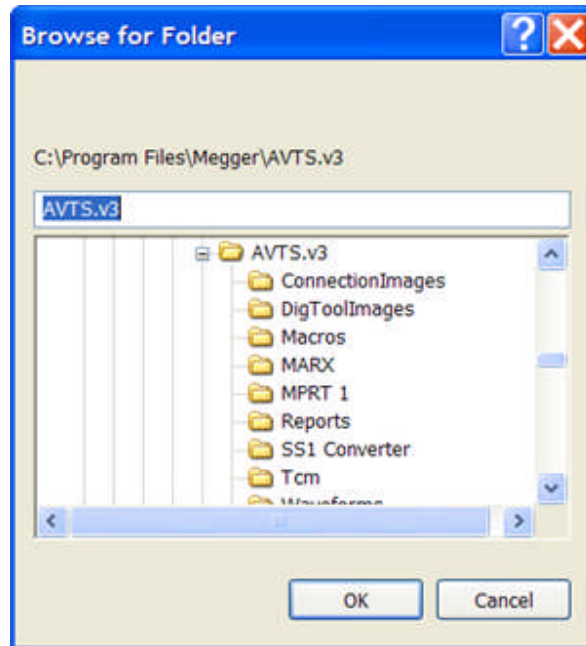
Select from the Workspace dropdown menu the **Save To PC** item.

The first use of **Save to PC** in AVTS will display the *Browse for Folder* dialog to select the folder to store the MPRT \*RTR files without an AVTSv3.0/MPRT 1 folder.



The default **MPRT 1** folder, with the **MPRT TVI** test files in the **MPRT** tab, is auto-created and all the files will be Saved to the PC in that folder. Each successive use will display the new folder in the *Browse for Folder* dialog Explorer list.





Note: The **Save to PC** option operates on the entire **MPRT 1** folder only, not on individual files.

To view and save/print the \*.RTR settings and Test Results, a **New Relay** will need to be created using the New Relay Wizard. The "*newrelayname*" will then be the host for the desired MPRT \*.RTR file import from the Tools/ImportData/ MPRT Results option.

Select File/New/Relay... or the New Relay icon from the menu tool bar to evoke the New Relay Wizard dialog. AVTSv3.0 databases contain an embedded Manufacturer and Device for the **MPRT** import. These must be used for a successful import.

Must select **Manufacturer: Megger**

Must select **Device: MPRT**

Create a location with the next Wizard steps in the **Organization Levels**

Create a name for the **New Relay**

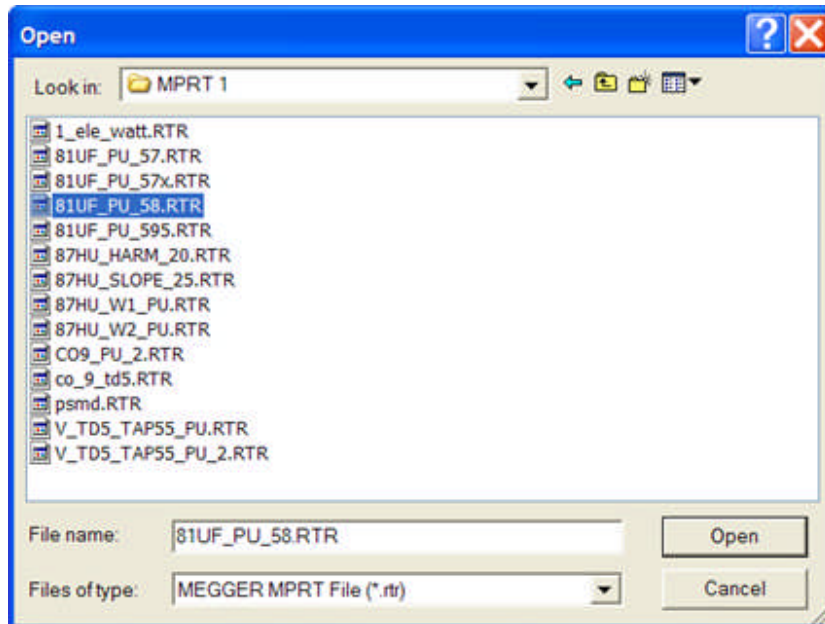
It may be desirable to have Windows Explorer open to the **MPRT 1** folder to label the Relay with the same name as that saved from the **MPRT** database; or, note the name on paper for reference when labeling the New Relay.

Double Click the "*newrelayname*" in the Relay tab so the Workbook Settings tab is displayed.

Note: The Settings tab will remain blank of any Settings.

Select the Tools\Import Data...\MPRT Results menu item

The *Open dialog* will display the MPRT 1 folder contents for selection of the relay to retrieve MPRT Results. The Settings will also be retrieved along with the Results.



To view Settings only, the Tools/Import Data.../MPRT Settings option must be exercised.

The Tools/Import Data.../MPRT Settings menu item will Import only the Settings.

**Note:** To print numerous test results without creating a new relay for each MPRT\*.RTR, Import the Results of several to the same relay. Each set of results will be listed under the relay name.

Test results may be retrieved in the **AVTS format** for archive by double clicking the result to be viewed . The results may also be viewed, saved and printed by right clicking on the result and selecting **Export Results to Word**.

## Relay Import MPRT Settings

The Tools/Import Data...Import/MPRT Settings menu items allow the population of a particular relay with test settings saved from a **MPRT**.

Use of the Import Results option is described in the Relay Import MPRT Results topic.

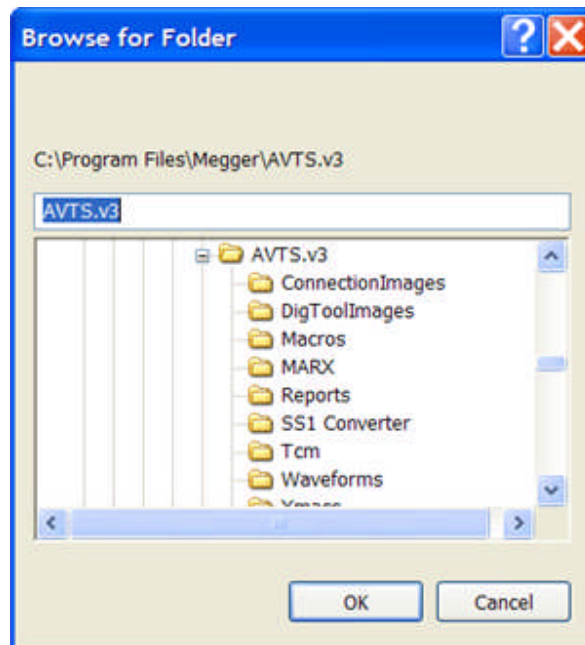
Use of the Export Settings option is described in the Relay Export MPRT Settings topic.

To Import and view the Settings in each file and to create a test report for record, the following is the procedure is necessary:

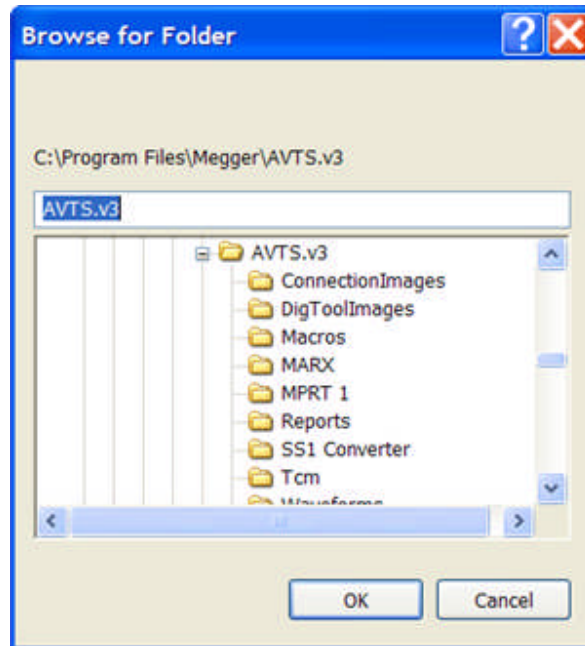
Establish communication between the PC and the **MPRT**. If successful, the saved test files viewable in the **MPRT TVI** will display in the MPRT Workbook tab.

Select from the Workspace dropdown menu the **Save To PC** item.

The first use of **Save to PC** in AVTS will display the *Browse for Folder* dialog to select the folder to store the MPRT \*RTR files without an AVTSv3.0/MPRT 1 folder.



The default **MPRT 1** folder, with the **MPRT TVI** test files in the **MPRT** tab, is auto-created and all the files will be Saved to the PC in that folder. Each successive use will display the new folder the *Browse for Folder* dialog in the Explorer list.



Note: The **Save to PC** option operates on the entire **MPRT 1** folder only, not on individual files.

To view and save/print the \*.RTR settings, a **New Relay** will need to be created using the New Relay Wizard. The "*newrelayname*" will then be the host for the desired MPRT \*.RTR file import from the Tools/ImportData/ MPRT Settings option.

Select File/New/Relay... or the New Relay icon from the menu tool bar to evoke the New Relay Wizard dialog. AVTSv3.0 databases contain an embedded Manufacturer and Device for the MPRT import. These must be used for a successful import.

Must select **Manufacturer: Megger**

Must select **Device: MPRT**

Create a location with the next Wizard steps in the **Organization Levels**

Create a name for the **New Relay**

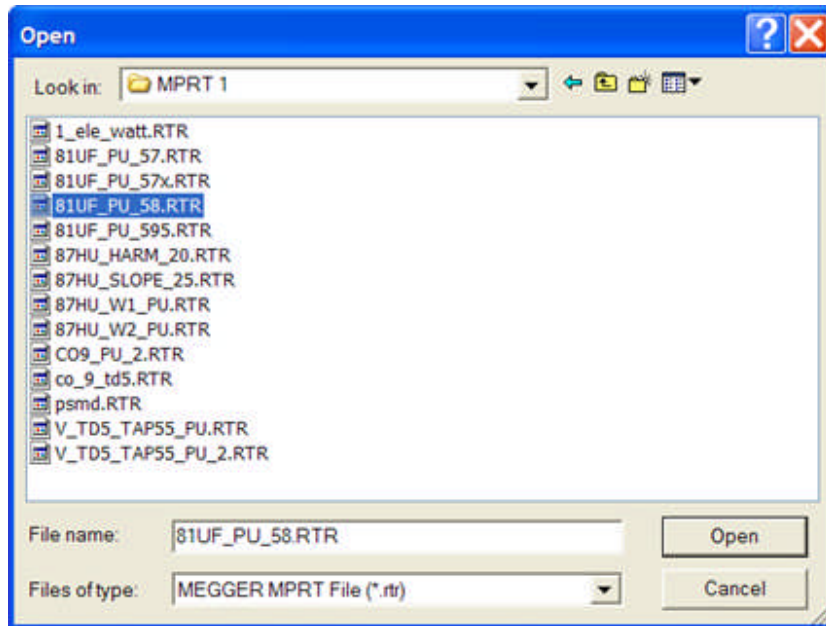
It may be desirable to have Windows Explorer open to the **MPRT 1** folder to label the Relay with the same name as that saved from the **MPRT** database; or, note the name on paper for reference when labeling the New Relay.

Double Click the "*newrelayname*" in the Relay tab so the Workbook Settings tab is displayed.

Note: The Settings tab will remain blank of any Settings.

Select the Tools\Import Data...\MPRT Settings menu item

The *Open dialog* will display the **MPRT 1** folder contents for selection of the relay to retrieve MPRT Settings. The Results will not be retrieved along with the Settings.



## **Relay Tools Export Data**

### **Relay Tools Export Data**

In addition to the menu items defined in the topic Main Menu View, the following additional items are selectable when a Relay has been opened in the Workbook.

- Export Data/Copy Results to Clipboard
- Export Data/AVTS Test Data File...
- Export Data/Export Results to Word
- Export Data/Export Current Test Results to Word
- Export Data/MPRT Settings

## Relay Copy Results to Clipboard

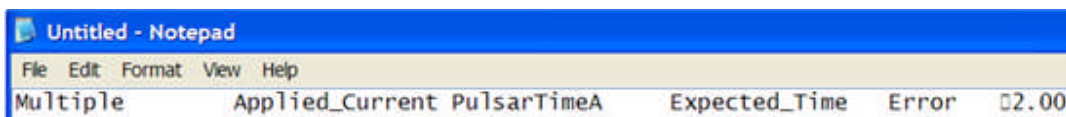
**Copy Results to Clipboard** is to place in the Windows clipboard test results from the Workbook Relay presently open. The test results copied will be only the set of results of the test selected in the Workspace Tests tab.

**Note:** Test results from a previous set of results opened in the Relay Workbook may also be copied.

**Note:** Only data is copied, graphics are not copied.

Once copied to the clipboard, the data in clipboard may be pasted into Notepad, Word or Excel.

A **paste to Notepad** places all data on a continuous line. A notepad indicator will be present to represent a new line for each line of original data. The single pasted line may then be placed into multiple lines by selecting the new line indicator and pressing the Enter key. The data may be aligned to the proper columns using the spacebar for spacing the data.



A **paste to Word** places the data on separate lines but not aligned directly with the column headers. The data is pasted in multiple lines but the columns are not aligned properly. The alignment may be accomplished by using the spacebar. Graphics are not copied to the clipboard and are not available to paste into Word.

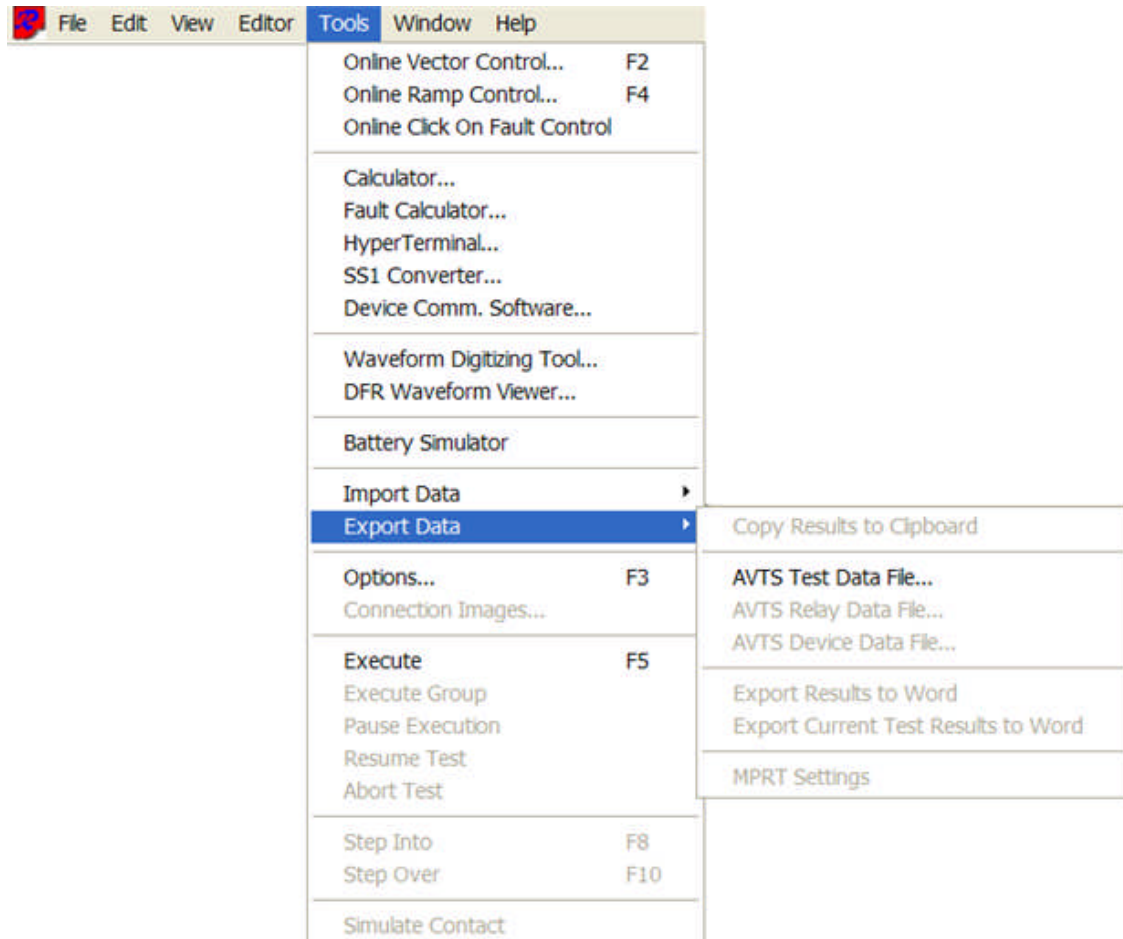
```
Multiple Applied_Current PulsarTimeA Expected_Time Error
2.00 1.00 3.45 3.67 -5.97
3.00 1.50 2.50 2.47 1.02
5.00 2.50 1.48 1.75 -15.09
```

A **paste to Excel** places the data line by line and column by column. All data is placed in individual cells beginning on the cell highlighted. Graphics are not copied to the clipboard and are not available to paste into Excel.

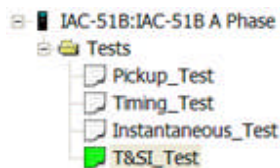
	A	B	C	D	E
1	Multiple	Applied_C	PulsarTime	Expected_	Error
2	2	1	3.45	3.67	-5.97
3	3	1.5	2.5	2.47	1.02
4	5	2.5	1.48	1.75	-15.09

## Relay Export AVTS Test Data File...

Prior to exporting a *Test Data File...*, the Workspace Tests tab must be selected and the test to Export must be selected.



Select the *Tests* tab then the Export option to begin the process.

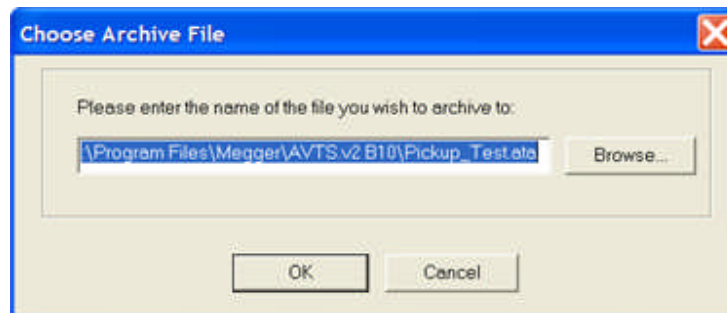


The purpose is to have the option to export only a relay from one database and available to be imported into another relay in the same database or another database.

**Note:** A test may be copied from one relay to the same or another relay in the same database without having to Export as a Test Data File.

The **AVTS Tests Data File...** option opens the *Choose Archive File* dialog.





The edit field defaults to the path for the selected test to be cloned and stored. The name of the test is also defaulted to the name of the selected test in the AVTS active database. The database file extension \*.ata is attached to the file name by default.

The test name in the path may be edited separately within the path in the edit field if a name change is desired. It will be necessary to also include the database file extension \*.ata with the file name. The extension will allow an import of the file to be immediately in an import process.

The Browse button may be selected to search for a location other than the default for exportation of the test data file.

Select OK to complete the exportation process. The Cancel option will abort the exportation process and return to the active AVTS data base.

## Relay Export Results to Word

**Export Results to Word** is available only by selection of a test result archive visible with the Relay in the Workspace Relays tab. Select a relay, expand its folder and select the list of archived dated test results.

The export action will copy all test results from the archived tests and paste them into a new Word document. All data and graphics copied are formatted the same as the normal View/Reports/Test Results... menu option.

When selected the export takes place immediately and the *MS Word Export Options dialog* is displayed.

From the display, the Include Settings, Include Graphics and View While Creating check boxes allow each description if checked. The check boxes are checked by default. Select the options and click OK.



**Note:** During the Export to Word process, the open Word document will be open but underlying the AVTS program. Word document icon will be on the Windows status bar and may be clicked to bring the View While Creating to focus.

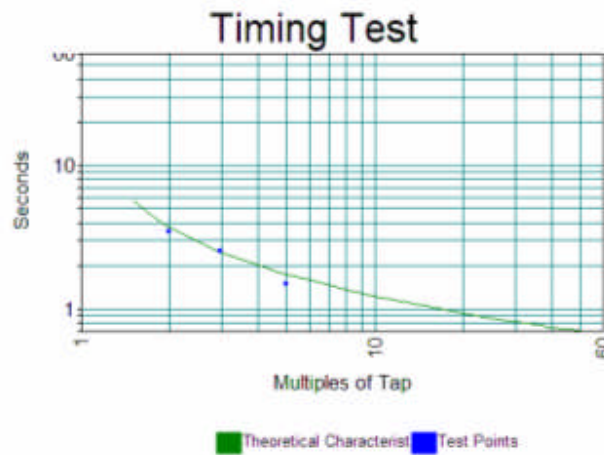
Microsoft Word will open a new document and paste test results of all executed tests with results in the relay Workbook into the *Word document*. The document may be given a name and used in archival files and/or placed in email to another location.

### Relay Results Report

Region: North Texas	Relay: IAC-51B A Phase
Substation: Dallas	Device: IAC-51B
Line: T-235	Tested By: Bruce Bordelon
	Date Tested: 04/08/2006

Setting:	Value:	Description:
<b>IAC 51 Settings</b>		
Tap	.5	Available IAC Tap Settings
Time_Dial	0.500	Relay Time Dial Setting
Instantaneous	2.000	Relay Instantaneous setting in amps
T&SI	.2	Target & Seal In Setting in DC amps

Test: Timing\_Test



Multiple (X Tap)	Applied Current (Amps)	Pulsar time A (Seconds)	Expected Time (Seconds)	Error (%)
1.00	1.00	3.43	3.81	-3.37
1.50	1.50	2.50	2.47	1.02
2.50	2.50	1.48	1.75	-15.09

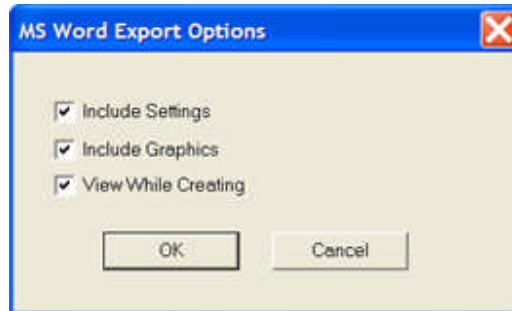
Test: Instantaneous\_Test

Pickup (Amps)	Min_Range (Amps)	Max_Range (Amps)	Pass/Fail
2.40	1.90	2.10	Fail

## Relay Export Current Test Results to Word

**Export Current Test Results to Word** will copy all test results from the active Workbook Relay Tests and paste them into a new Word document. All data and graphics copied are formatted the same as the normal View/Reports/Test Results... menu option.

When selected the export takes place immediately and the **MS Word Export Options dialog** is displayed. From the display, the Include Settings, Include Graphics and View While Creating check boxes allow each description if checked. The check boxes are checked by default. Select the options and click OK.



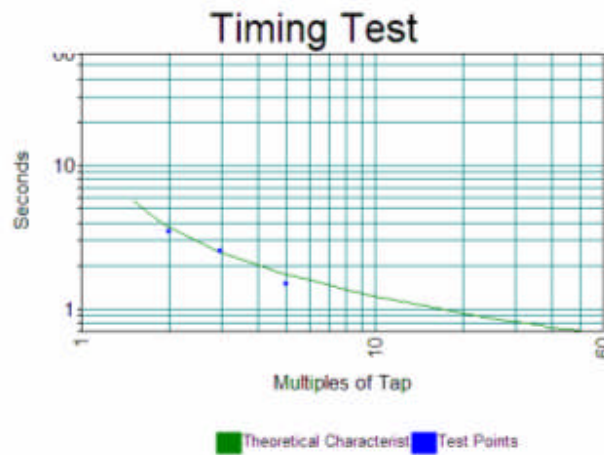
Microsoft Word will open a new document and paste test results of all executed tests with results in the relay Workbook into the **Word document**. The document may be given a name and used in archival files and/or placed in email to another location.

### Relay Results Report

Region: North Texas	Relay: IAC-51B A Phase
Substation: Dallas	Device: IAC-51B
Line: T-235	Tested By: Bruce Bordelon
	Date Tested: 04/08/2006

Setting:	Value:	Description:
<b>IAC 51 Settings</b>		
Tap	.5	Available IAC Tap Settings
Time_Dial	0.500	Relay Time Dial Setting
Instantaneous	2.000	Relay Instantaneous setting in amps
T&SI	.2	Target & Seal In Setting in DC amps

Test: Timing\_Test



Multiple (X Tap)	Applied Current (Amps)	Pulsar time A (Seconds)	Expected Time (Seconds)	Error (%)
1.00	1.00	3.43	3.81	-3.37
1.50	1.50	2.50	2.47	1.02
2.50	2.50	1.48	1.75	-15.09

Test: Instantaneous\_Test

Pickup (Amps)	Min_Range (Amps)	Max_Range (Amps)	Pass/Fail
2.40	1.90	2.10	Fail

## Relay Export MPRT Settings

The Tools/Import Data...Export/MPRT Settings menu items allow saving edited Settings data.

Relay Settings data may be saved to a new file or an existing file name during this process.

Use of the Import Results option is described in the Relay Import MPRT Results topic.

Use of the Import Settings option is described in the Relay Import MPRT Settings topic.

To Export MPRT Settings from the Workbook Relay tab, the following procedure is necessary:

Select the "*relayname*" in the Relay tab to be exported.

Double click the "*relayname*"

Select Tools/Export Data/MPRT Settings

The Save dialog will open for selection of a file resident in the MPRT 1 folder or to create a new "*filename*".

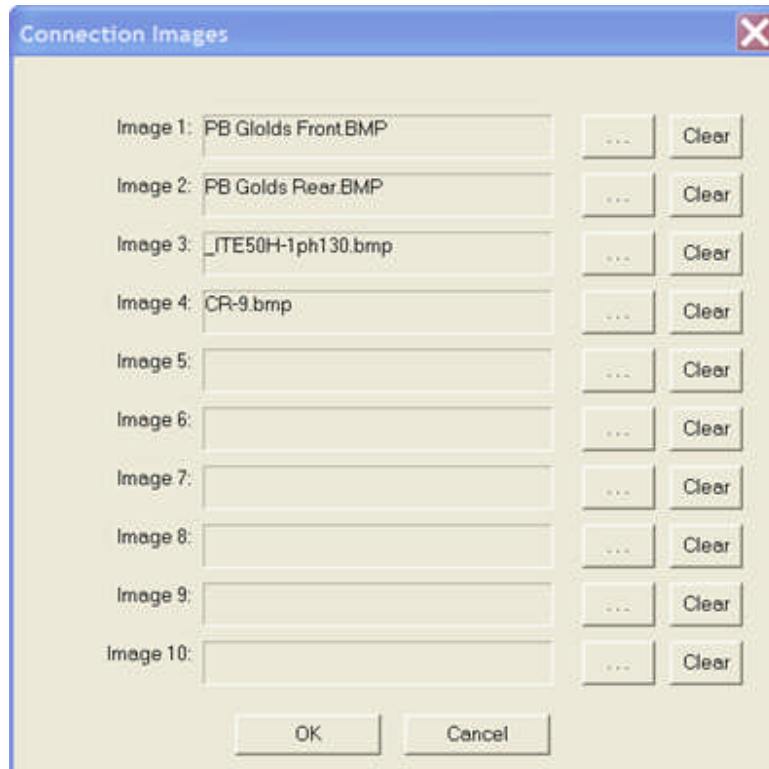
The Settings will be saved to the "*filename*" selected.

## Connection Images Toolbar

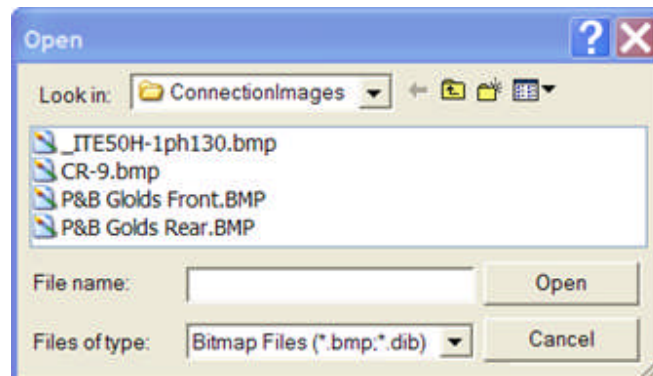
The **Connections Images...** toolbar is usable only with an open relay in the Workbook pane, and with the Connections Editor tab in focus. In this example four of the tools have had images linked to them, evidenced by the tools being highlighted and labeled 1 through 4.



The toolbar has ten tools for linking, in AVTS, bitmap (\*.bmp) images of relay terminals or pictures. When selected, the most right hand tool *Connections Images dialog* for mapping images to each of the ten tools. The dialog has ten label fields, each with a browse and Clear button.

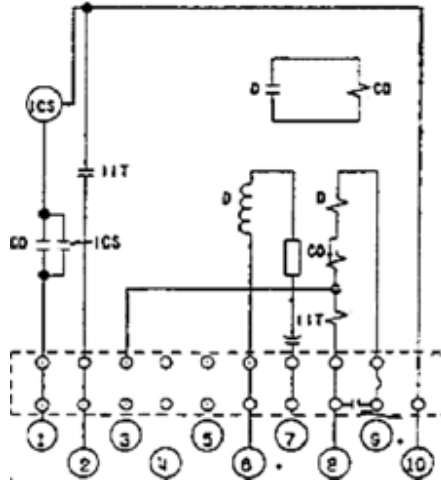


The browse button invokes the *Open images dialog* for selecting the Image to be placed in the field reserved for one of the ten tools. Each image must be selected in turn by using the browse button. The Clear button is reserved for removing the image label from the tool, making that tool available for a different image.

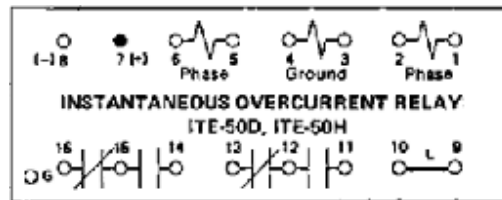


The images may be created by several methods and saved as a bitmap into the ConnectionImages folder of AVTS

- *Scan* an image

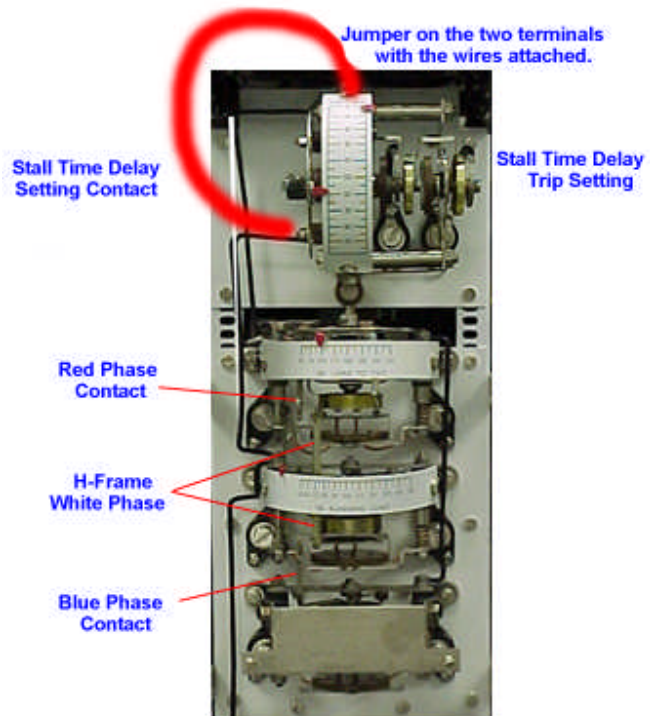


- *Create* an image in Paintbrush or other graphic program



- *Take* a photograph using a digital camera and modify in Paintbrush or other graphic program





Select an image from the Toolbar and click in the Workbook Connections Editor pane to complete the diagram.

## Relay Tools Execute Group

The **Execute Group** Tools menu option is available with a Relay open in the Workbook pane and giving access to the Workspace Tests tab.

When selected, Execute Group executes a group of selected tests. The first test executes and stores the results, then the second test of the group is automatically called and executed. The results of the second execution are stored and the next test is called and executed. The process continues until all tests in the selected group have been executed and results stored.

See Tests Tab Drop Menus

## Relay Menu Window

The **Cascade**, **Tile** and **Arrange Icons** in the *Relay Menu Window* items are defined in the topic Main Menu Window.



The area below the Arrange Icons item is a numeric list of all presently open Devices and Relays in the Workbook. In this example, the Relay "1 IAC-51B:IAC-51B A Phase - Pickup\_Test" is the open relay and is the item presently in view and active, as indicated by the check. The item "2 IRD-9" is an open Device, but is not in view.

The item in view may be changed in three ways:

- Place a check by the item to be displayed and active,
- Select the tab at the bottom of the Workbook pane for the item to be displayed and active, and
- Creating a new Relay or Device.

## Relay Toolbar

The **Relay Toolbar** is a portion of the standard toolbar and displays the tool icons for use in various displays. The dimmed icons are those that are available for use only when specific panes or dialogs are open.

The Relay Toolbar is usable only with a Relay open in the Workbook pane.

Additional details for using these tools are in the topic Relay Menu Tools topic.

The Standard Tool icons listed from left to right above are:



**Execute Test** Click on, if not dimmed, will begin an execution of a live or simulated test when a Relay is open in the Relay Workbook pane.

**Pause Test Execution** Click on, if not dimmed, will cause an active test to cease operation at the present stage of the test. Note that care should be taken to monitor the test set outputs during a live test execution and assure that the outputs are not at a undesirable level.

**Resume Test Execution** Click on, if not dimmed, will cause a active test to continue the operation of the test under Pause.

**Abort Test** Click on, if not dimmed, will cause an active test to cease operation and, if a live test, turn OFF any test set outputs.

**Step Into** Click on, if not dimmed, will allow a test execution, paused due to an inserted Breakpoint, to Step Into the next test control.

**Step Over** Click on, if not dimmed, will allow a test execution, paused due to an inserted Breakpoint, to Step Over the test control and continue to the next test control.

**Simulate Contact** Click on, if not dimmed, will simulate an expected input to an executed test, whether the test is live or in simulated mode.

**Variable Watch** Click on, if not dimmed, open a window in the Workbook Test tab of an open Relay. The user may then option to type into the window known Variable names to monitor the values of the variables before, during and after Test Execution.

**Current Test Results** Click on, if not dimmed, will open a Workbook Report of the test results having occurred during the previous executed test, whether the test was live or in simulated mode.

## Workspace Pane

## Workspace Pane

The **Workspace Pane** is the left hand side of the AVTS Relay Test Software when opened. There are four tabs for use in the Workspace:

**Devices** tab is to locate the Device for a family of installed Relays. The Device is the template to all Relays produced from it and is governed by it. The Devices tab drop down menus are detailed in the Devices Tab Drop Menu topic.

**Relays** tab is to locate a Relay to edit or execute tests. Edits to the Relay will affect all other relays in the database derived from the same Device. The Relays tab drop down menus are detailed in the Relays Tab Drop Menu topic.

**Tests** tab is to locate a Test to edit or execute. Edits to selected Test will affect all other relays in the database derived from the same Device. The Relays tab drop down menus are detailed in the Tests Tab Drop Menu topic.

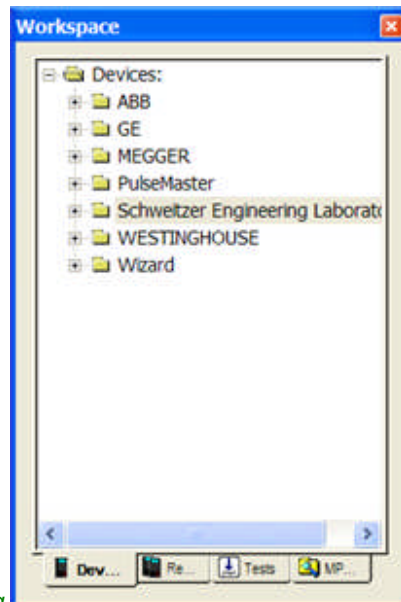
**MPRT** tab is to locate the settings for a connected MPRT. The settings may called from and edited, then returned to the database resident in the connected MPRT. The MPRT Tab Drop Menus are detailed in the MPRT Tab Drop Menu topic.

## Devices Tab Drop Menus

In the Workspace **Devices** tab, a right mouse click will produce a *drop down menu* of options.



The options are:



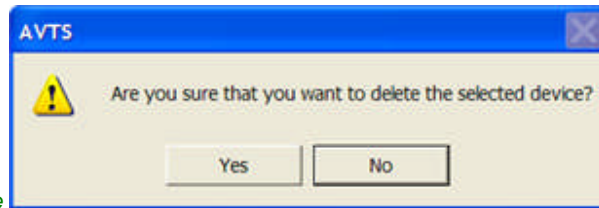
*Allow Docking* is checked and active by default. Left mouse click the option to create a separate Workspace window. The window may be moved and kept separate from the main AVTS and Workbook window. Reproduce the drop down menu and left click the option to re-anchor the Workspace by grabbing the window and dragging back into the AVTS window.

**Hide** is to take the Workspace completely from view. To reproduce the Workspace, select the View/Workspace menu item.

**New Device** is to create a new Device in the Devices tab. Select this option to open the New Device Wizard.

**Copy Device** is not active until a Device has been selected. When selected the device is copied and ready to be pasted to another location in the Devices list.

**Paste Device** is not active until a Device has been copied. When pasted the Device may be renamed to create a new Device (Relay template) completely divorced from the original Device. Any edits made to Relays created with the new Device will not affect the Relays of the original Device. The efficiency is not having to recreate a full Device with all the Tests.



*Delete Device* is not active until a Device has been selected. The selected Device will be removed from the active database. When selected, a delete Device message is evoked to allow Cancel if the incorrect Device was selected.

**Note:** A Device may not be deleted if a Relay derived from the Device is present. The deleted Device can only be recovered with the Edit/Undo or Undo arrow.

**Rename** is to allow edit of the Device name.

**Refresh** will update the Devices list with any recently made edits.

**Float In Main Window** is to place the Workspace in the same AVTS Workbook window as on open Device or open Relay. The open objects will have tabs for identity at the bottom of the Workbook pane. The Workspace may be re-anchored to the left side of the AVTS window by re-selecting the drop down menu option.

See Workbook Pane Device Settings Tab Drop Menu.

## Relays Tab Drop Menus

In the Workspace **Relays** tab, a right mouse click will produce a *drop down menu* of options.



The options are:

Allow Docking

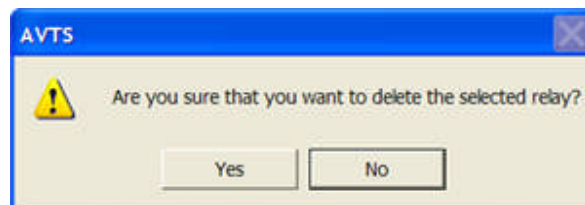
Hide

**New Relay** is to create a new Relay in the Relays tab. Select this option to open the New Relay Wizard.

**Copy Relay** is not active until a Relay has been selected. When selected the Relay is copied and ready to be pasted to another location in the Devices list.

**Paste Relay** is not active until a Relay has been copied. When pasted the Relay may be renamed to create a new Relay cloned from the parent Device template. Any edits made to the relay, other than setting changes in the Workbook Settings tab, will edit all other Relays with the same Device parent. The name may be changed to reflect the substation assigned code without affecting any of the other Relays of the same Device parent. For example, rename a single phase Relay as Phase A. Copy, paste and rename the Relay to Phase B. The efficiency is not having to recreate a full Relay with all the Tests.

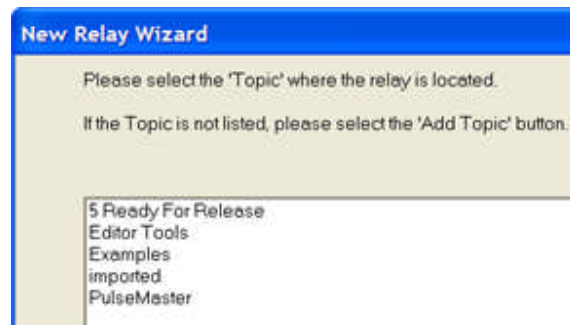
**Delete Relay** is not active until a Relay has been selected. The selected Relay will be removed from the active database. When selected, a *delete Relay message* is evoked to allow Cancel if the incorrect Relay was selected.



**Note:** The deleted relay can only be recovered with the Edit/Undo or Undo arrow. **All results for the Relay will be deleted from the database along with the Relay.**

**Move Relay** is to select a relay and move it from one location to another. To move, the Relay must not be open in the Workbook. The action will open the *New Relay Wizard* for selecting the new location in the active database to which the Relay will be placed.





Note: This action has no Undo. To return the Relay to the original location, repeat the Move Relay process.

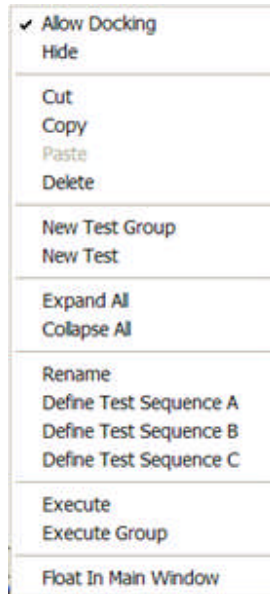
Rename

Refresh

Float In Main Window

## Tests Tab Drop Menu

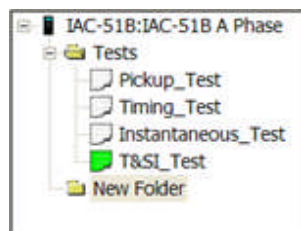
In the Workspace **Tests** tab, a right mouse click will produce a *drop down menu* of options.



The options are:

- Allow Docking
- Hide
- Cut
- Copy
- Paste
- Delete

**New Test Group** is to create tests into groups segregated by function. One group name may be "As Found" and another group may be "As Left". When selected, a *New Group will be placed* in the Test List ready to type a name label. If a test is highlighted, the New Group will be placed as a subgroup to the group in which the selected test resides. To create a separate group, highlight the Relay name at the head of the test list.



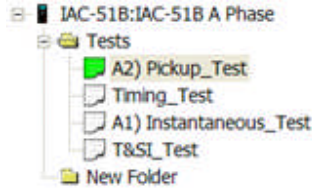
**New Test** is to create a new Test in the Tests tab. Select this option to open the New Test Wizard.

**Expand All** is to open all group folders "expanded" to display all tests within all the groups. If only one group of tests is to be expanded, click on the "plus" sign for the group folder.

**Collapse All** is to close all group folders that have been expanded, whether or not a few or all groups were previously expanded.

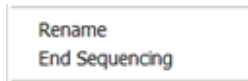
Rename

Define Test sequence A is to *select a group of tests* to execute automatically from the first one selected to the last one selected for the sequence. Select Define Test Sequence A and then begin left clicking on each test in the order desired to be executed.



**Note:** The selected tests will have the sequence alpha and the number of the order of selection appended to the beginning of the test name.

The tests may be selected in any order across several groups. Should a test in the sequence abort, the sequence will halt. To resume the test sequence, select the aborted or any other of the tests in the sequence and Execute Group. The tests will begin with the selected test and continue to the end. After the last test has been selected, open the drop down menu to select *End Sequence* option.



**Note:** End Sequence only appears as a drop down menu option when a Test Sequence has been selected.

**Define Test Sequence B** is to define a separate group of automated tests to Test Sequence A.

**Define Test Sequence C** is to define a separate group of automated tests to Test Sequence A.

**Execute** is to execute a selected test, same as the Relay Toolbar Execute icon.

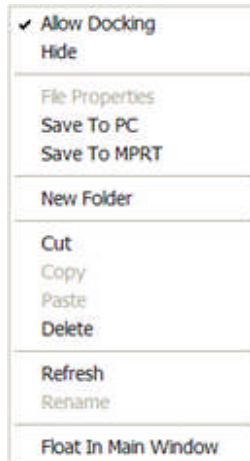
**Execute Group** is to execute the Test Sequence A, B or C by selecting a test in that group prior to performing this option.

Float In Main Window

## MPRT Tab Drop Menus

## MPRT Tab Drop Menu

Use of the *drop down menu* options in the Workspace **MPRT** tab are available with a successful communication link to the **MPRT** using AVTS. A successful link will automatically display all existing test files in the MPRT unit database that were created by the user. These files will have the label created by the user and have a \*.RTR extension.



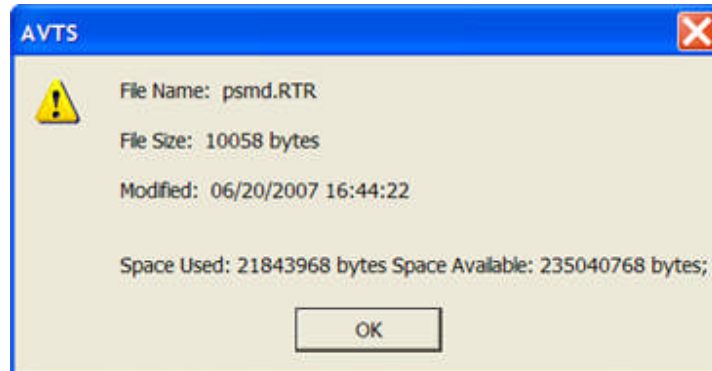
In the Workspace **MPRT** Explorer tab, a right mouse click will produce a drop down menu of options.

The options are:

Allow Docking

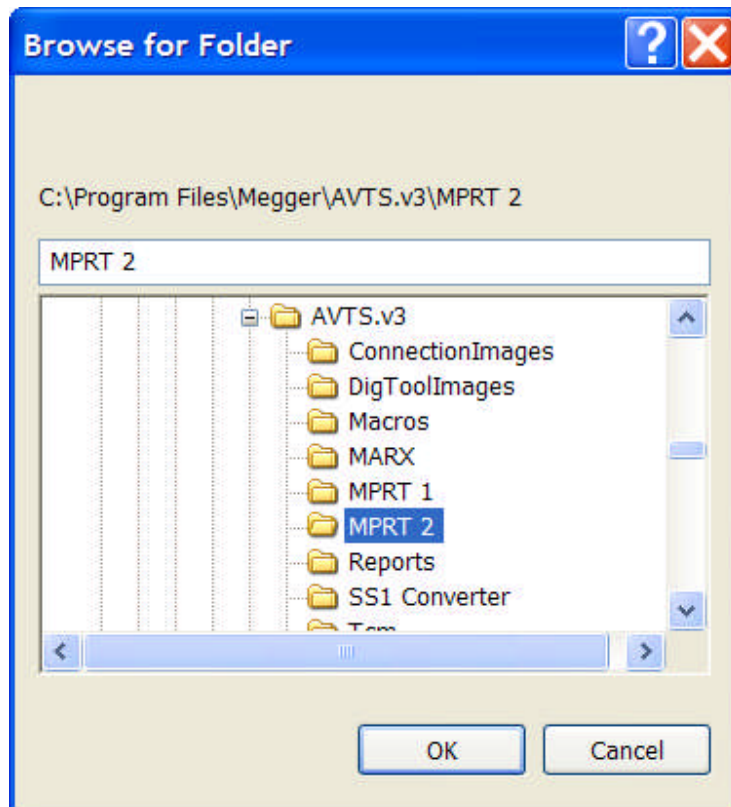
Hide

*File Properties* will display the **MPRT** memory allocation to files and the total memory available



**Save to PC** option will save all the **MPRT** tab files displayed to a folder C:\Program Files\Megger\AVTSv3.x\MPRT 1 folder. The folder will automatically be created the first time the files from any **MPRT** are first saved.

*Save to MPRT* will save all files located in the C:\Program Files\Megger\AVTSv3.x\MPRT 1 folder, or other selected folder, to the connected **MPRT**.



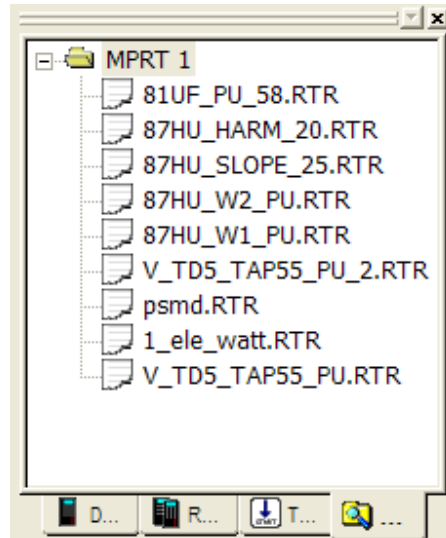
**New Folder** option will create a new folder in the Workspace MPRT tab when communication to the MPRT is established.

**Note:** The folder will not be of use to the **MPRT**. If files are copied to the folder and the **Save to MPRT** option is exercised, the folder name will appear in the **PVI File Manager**, but not usable for testing. **DO NOT USE THIS DROP MENU OPTION.**

- Cut
- Copy
- Paste
- Delete
- Refresh
- Rename
- Float In Main Window

## MPRT Tab

The AVTS Workspace **MPRT** tab will automatically *display all existing test files* in the **MPRT** unit database when a successful communication link is established by AVTS.



Note: First access to the Workbook MPRT Tab will automatically import all test files in the MPRT and display them. There will also be created an AVTS subfolder titled MPRT.

These files will have the "filename" created in the **MPRT TVI** by the user and will be displayed with a \*.RTR extension.

Use of the Import MPRT Test Results from a MPRT \*.RTR file is described in the Relay Import MPRT Results topic.

Use of the Import MPRT Test Settings from a MPRT \*.RTR file in the Relay Import MPRT Settings topic.

Use of the Export MPRT Test Settings to a MPRT \*.RTR file in the Relay Export MPRT Settings topic.

An **Example** of use of this tab is to send new relay settings to the MPRT for residency in the saved or new relay test

Typically, the files saved in the MPRT using the TVI are from the Pre-set Tests options. However, a test may be created using the Manual Tests option and those settings saved to the TVI File Manager.

Perhaps several tests are desired to test a microprocessor transmission line relay, each test for a particular zone of protection.

First configure the test screen with the appropriate Voltages and Currents and their values. Then save the test file to the File Manager labeling the test with meaningful nomenclature; i.e., **CH\_Line5\_SEL321\_Z1\_PhPh**. This name may imply that the relay is a Schweitzer 321 at CH Station on Line #5 settings are for a phase-to-phase fault within Zone 1.

Establish communication with AVTS and select the MPRT tab.

Save To PC the **CH\_Line5\_SEL321\_Z1\_PhPh** file. Recall that the test will be saved in the MPRT 1 folder along with remainders displayed.

Create a new relay in the relay tab using the Relay Import MPRT Results topic, and import the Settings from the **CH\_Line5\_SEL321\_Z1\_PhPh.RTR** file.

Create another folder in Windows Explorer, perhaps labeled MPRT 2 *Substationname* and save a copy of the original **CH\_Line5\_SEL321\_Z1\_PhPh.RTR** file to that folder.

Edit the file label in the **MPRT 1** folder to reflect the new test settings; i.e., **CH\_Line5\_SEL321\_Z2\_3Ph**

In the open **CH\_Line5\_SEL321\_Z1\_PhPh.RTR** file in the AVTS Relay tab change the Settings Values to reflect the second test to be conducted on the same relay. An example may be for a Z2 3-phase fault.

Export the edited Settings to the new file in the **MPRT 1** folder using the Relay Export MPRT Settings.

Select the **AVTS MPRT** tab.

Select the **MPRT1** folder label in the AVTS **MPRT 1** tab.

Right click in the Workspace pane and select the **Save to MPRT** option.

The MPRT will now have all the original files and the new file in the **TVI** for later use.





## Workbook Pane

## Workbook Pane

The **Workbook Pane** is to view, edit and operate **Devices, Relays** and relay **Tests** when opened.

An open **Device** will have two tabs associated with it:

Devices Settings tab is to create the Device settings list with the complement settings parameters.

Device General Information tab is to provide details concerning the relay family for which the Device is template.

An open **Relay** will have eight tabs associated with it:

Relays Settings tab is to provide the discrete Relay settings values only for the installed application. The Settings names and parameters are edited only in the Devices template.

Relay Test tab is to display the active parameters, test results and graphics during and after the execution of a test.

Relay Capture tab is to display the MPRT test responses during an executed test.

Relay Test Editor tab is to create and/or edit the test procedure using the Controls in the Test Editor Toolbar.

Relay Monitor Editor tab is used to create create displays, of active and inactive parameters emulating attached test equipment, using the Monitor Editor Toolbar.

Relay Connections Editor tab is to create a display of test equipment and relays interconnections using the Connections Editor Toolbar.

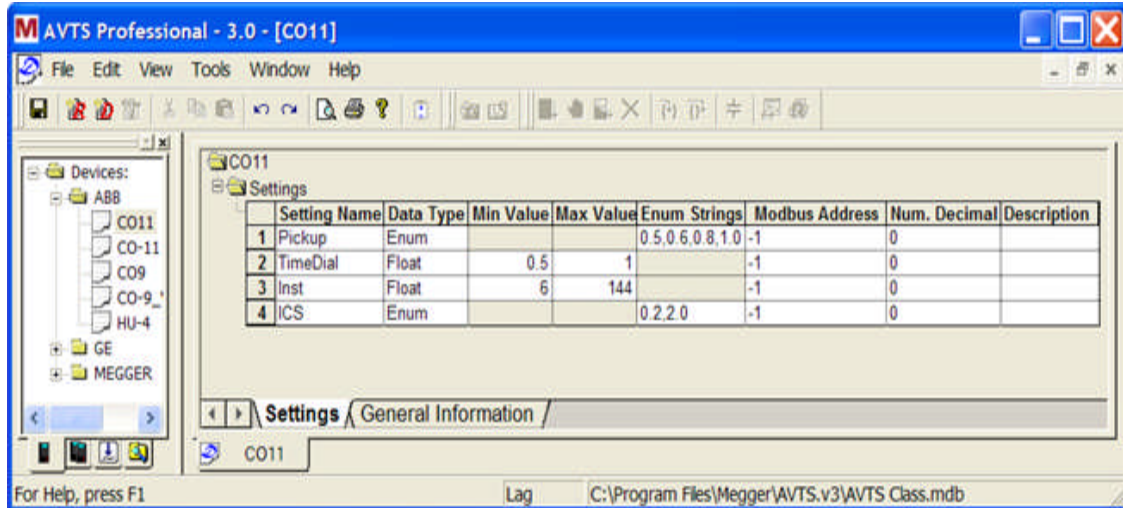
Relay BASIC tab is to create and edit information complementary to the test procedure using Basic Language Syntax and/or script.

Relay General Information tab is to provide details concerning the particular Relay in use.

## Devices Settings Tab

## Devices Settings Tab

The *Settings Tab for Devices* is to define the Names and parameters for each of the settings associated with the relay template or Device. The list may contain many settings all in a single or in multiple groups by Group Name.



The settings are divided into six columns for defining each setting:

**Setting Name** edit field is to enter a name for the setting. The name becomes a variable that may be used as reference to the setting value edited in the Relay Settings Tab. The variable should not have spaces before, interior to or after the name, but may have a dash, underscore or other method to clarify the name. Typically the name is identical to the setting name for the relay provided in the relay instruction manual.

**Data Type** is to identify whether the setting is:

**Integer** is a number of whole value with no decimal. A decimal value placed in an Integer field in the Relay Settings tab will evoke an error message advising that an invalid entry has been attempted.

**Float** is a floating point number indicating that a decimal value may be entered in the Relay Settings tab settings field. The number entered may also be an Integer with no decimal.

**On/Off** is to set the Relay Settings option to either an On or Off string variable. The setting may be used to condition the use or non-use of other settings in the Relay Settings tab or in the Test Editor tab Controls.

**Yes/No** is to set the Relay Settings option to either a Yes or No string variable. The setting may be used to condition the use or non-use of other settings in the Relay Settings tab or in the Test Editor tab Controls.

**Enum** is to create an enumeration of variables and or values for a drop list of valid Values in the Relay Settings tab. The list is entered in the edit field with no spaces with each item separated by a comma.

**String** is to set the variable Name as a non-numeric value. If a String is to have a value, it must be converted from String to Numeric using Expressions.

To convert a string to a number:

`c = val(b$)` results in the string

`b$ = 10$` being set to the number `c = 10`.

The following Data Types to set the how may bytes to read from a Modbus device and how to use the data.

**Long Int** is for a setting with a 32 bit Integer value from -2,147,483,647 to 2,147,483,647

**Unsigned Int** is for a setting with an unsigned 16 bit Integer value from 0 +65,535

**Double** is for a setting for a double float value 64 bit instead of 32 bit

**Note:** Additional information for Modbus is defined in the 1-Touch Modbus topic.

**Max Value** is to set the maximum value allowable for the Setting Name. Default = 99999 for Integer and = 99999.99 for Float. The field is null for the other Data Types.

**Enum Strings** is to enter the comma delimited list of values or string variables to be in the Values drop list in the Relay Settings tab. Default is a blank field and null for all other Data Types.

Entries may be: numeric 0.5,0.6,0.8,1,2,3.0 or string Pass,Fail,Yes,No

**Note:** No spaces may exist in the enumeration of values.

**Modbus Columns displayed** in the Settings tab. Additional information is in the 1-Touch Modbus topic.

The screenshot shows the 'F60 Settings' window with two tables. The first table is under the 'Settings' tab and the second is under the 'Modbus' tab. Both tables have columns for Setting Name, Data Type, Min Value, Max Value, Enum Strings, Modbus Address, Num. Decimals/Size, and Description.

Setting Name	Data Type	Min Value	Max Value	Enum Strings	Modbus Address	Num. Decimals/Size	Description
1 ProductType	Integer	0	100000000		0	0	int
2 ProductVersio	Integer	0	100000000		0	0	int
3 DigitalCounter	Integer	0	100000000		0	0	int
4 IA_RMS	Integer	0	100000000		0	0	int
5 IB_RMS	Integer	0	100000000		0	0	int
6 IC_RMS	Integer	0	100000000		0	0	int
7 OrderCode	String				64	0	string\40

Setting Name	Data Type	Min Value	Max Value	Enum Strings	Modbus Address	Num. Decimals/Size	Description
1 Address	Integer	0	99999		0	0	
2 BaudRate	Integer	0	99999		0	0	
3 Mode	Integer	0	99999		0	0	

**Modbus Address** edit field is to enter the memory map address **Addr** given by the relay manufacturer.

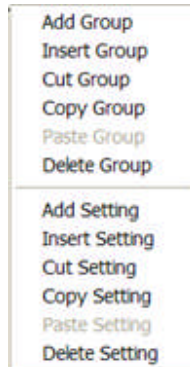
**Num. Decimal/Size** edit field is to enter the memory map address **Step** given by the relay manufacturer for the number of decimal places for the value required for the setting.

**Description** edit field is to enter text with additional information for the given Setting Name.

The drop menu list is described in the Device Setting Tab Drop Menu topic.


## Device Settings Tab Drop Menu

In the Workbook Devices Settings tab, a right mouse click in an edit field will produce a *drop down menu* of options.



The options are:

**Add Group** is to add another **Group** or book to segregate additional settings from the original Group. The New Group name will be *New Group* and highlighted for immediate keyboard edit. The New Group will contain the list of names and parameters as set by the Tools/Options/Default Settings tab menu item and ready for edit. Additional New Groups, if added immediately after each, will have a number suffix appended to differentiate them in the database.

Note: Add Group may be accomplished by selecting the  menu tool. Continued left mouse clicks will continue adding new Groups one after the other with a successive number suffix.

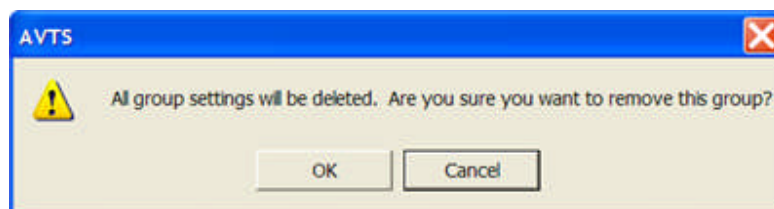
**Insert Group** is to add another Group or book to segregate additional settings from the original Group. The New Group will be inserted above a selected Group. The Add Group attributes apply.

**Cut Group** is to remove an existing Group and place it in a clipboard for paste in another location of this or another open Device.


**Copy Group** is to clone an existing Group and place it in a clipboard for paste in another location of this or another open Device. The New Group will be labeled the same as the one copied and will have a number suffix attached.

**Paste Group** is to place a Group from the clipboard to another location of this or another open Device. The new Group will be appended to the end of the list of the selected Group.

**Delete Group** is to completely remove a Group from the Device. When selected, the *Device delete message* warning will appear. The deleted Group may be recovered by the Edit/Undo menu item, if it is used immediately after the delete.



**Add Setting** is to add another Setting to a selected Group. The new Setting will be labeled *Setting* and highlighted for immediate keyboard edit. The setting will have the default parameters are set by the Tools/Options/Default Settings tab menu item and ready for edit. Additional New Settings, if added immediately after each, will have a number suffix appended to differentiate them in the database.

Note: Add Setting may be accomplished by selecting the  menu tool. Continued left mouse clicks will continue adding new Settings one after the other with a successive number suffix.

**Insert Setting** is to add another setting in the Group selected. The new Setting will be inserted above a selected Setting. The Add Setting attributes apply.

**Cut Setting** is to remove an existing Setting and place it in a clipboard for paste in another location of this or another open Device.

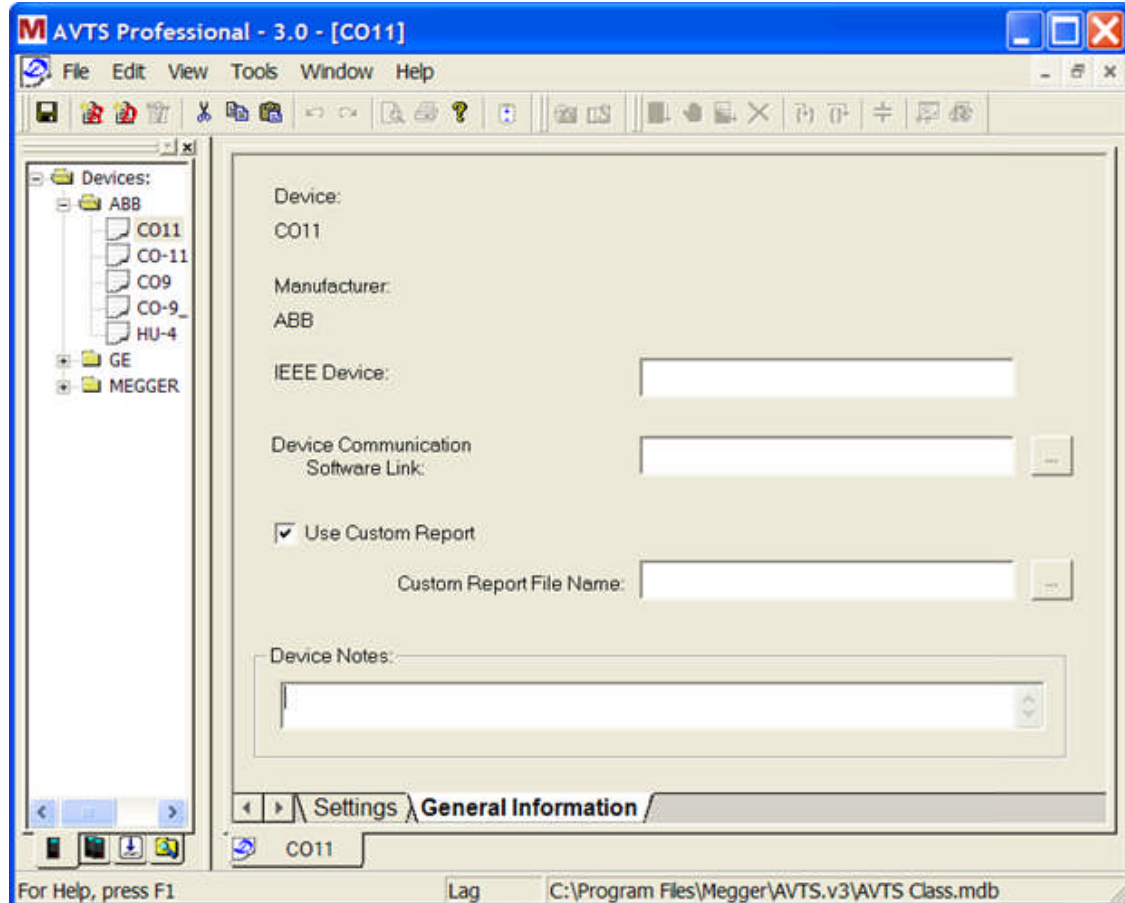
**Copy Setting** is to clone an existing Setting and place it in a clipboard for paste in another location of this or another open Device. The new Setting will be labeled the same as the one copied and will have a number suffix attached.

**Paste Setting** is to place a Setting from the clipboard to another location of this or another open Device. The new Setting will be appended to the end of the list of the selected Setting.

**Delete Setting** is to completely remove a Setting from the Group. When selected, the Setting delete message warning will appear. The deleted Setting may be recovered by the Edit/Undo menu item, if it is used immediately after the delete.

## Devices General Information Tab

The **General Information** tab contains the following fields.



**Device:**

**Manufacturer:**

Edit field for entry of a numeric designation for the device based on IEEE standards described in the IEEE Device topic.

The **Device Communication Software Link:** edit field with browse button. The user may select the appropriate communication software specific to the Device. Once saved in this Device, selection of the **Tools/Device Comm. Software...** item will open the communication path to the Device. If this field is blank or an incorrect software is selected, an **error message** will be evoked.



The **Use Custom Report** check box that, when checked, opens an edit box with a browse button to select the path for the **Custom Report File Name**:

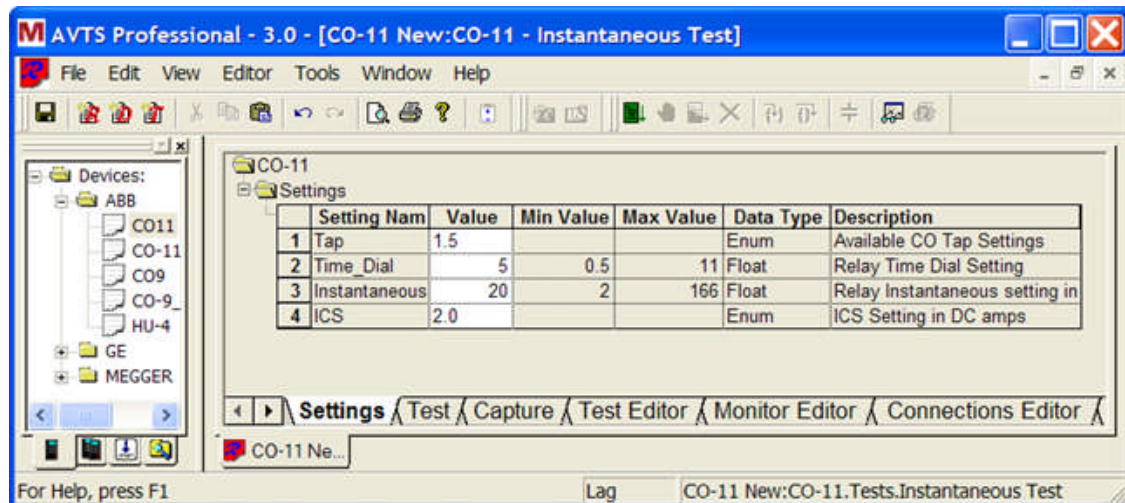
Note that this enables the user, if it is desired, to create a special report for all Relays created with this Device template. This Custom Report must be created with the product Crystal Reports.

The **Device Notes** edit window is specific to this Device and applies to all Relays create from this Device template. Application notes for the Device might be one of the note types placed here. The window is a text file and can be treated as such, and the size can be as large as Windows Notepad. The user can copy and paste information from other sources into this window.

## Relays Settings Tab

## Relays Settings Tab

The *Settings Tab for Relays* is to display the Names and defined parameters for each of the settings associated with the relay template or Device.



The **Settings** tab settings are divided into six columns for defining each setting:

**Setting Name** is to display the variable name defined in the Device as described in Devices Settings Tab.

**Value** is to enter the setting value or string to agree with the relay setting for the application. Devices Settings Tab

**Min Value** is to display the minimum value allowed defined in the Devices Settings Tab.

**Max Value** is to display the maximum value allowed defined in the Devices Settings Tab.

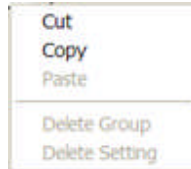
**Data Type** is to display the Setting variable type defined in the Devices Settings Tab.

**Description** edit field is to enter text with additional information for the given Setting Name or value.



## Settings Tab Drop Menu

In the Workbook Relays Settings tab, a right mouse click in an edit field will produce a *drop down menu* of options.



The options are:

**Cut** is to remove an Integer or Float value from the Value cell and place the value in the clipboard for paste to another cell for Integer or Float Values or String.

**Note:** Cut to an Enum field will Copy without removing the Value.

**Copy** is to place a clone of the Value cell in the clipboard for paste to another cell for Integer or Float Values or String.

**Note:** Copy to an Enum field is prohibited.

**Paste** is to place an item from the clipboard to a cell for an Integer or Float Value or String.

**Delete Group** is to remove a Group of settings that do not exist in the Workspace Device. If a Group of Settings has been removed from the Device template and the Device saved to the database, then when a Relay with that Device template is opened, the Delete Group option will be available to remove the Group.

**Note:** An open Relay must first be closed from the Workbook and then reopened to be refreshed from the database. The Delete Group option for a valid Group will be dimmed.

**Delete Setting** is to remove a Setting from a Group that does not exist in the Workspace Device. If one or more settings have been removed from the Device template and the Device saved to the database, then when a Relay with that Device template is opened, the

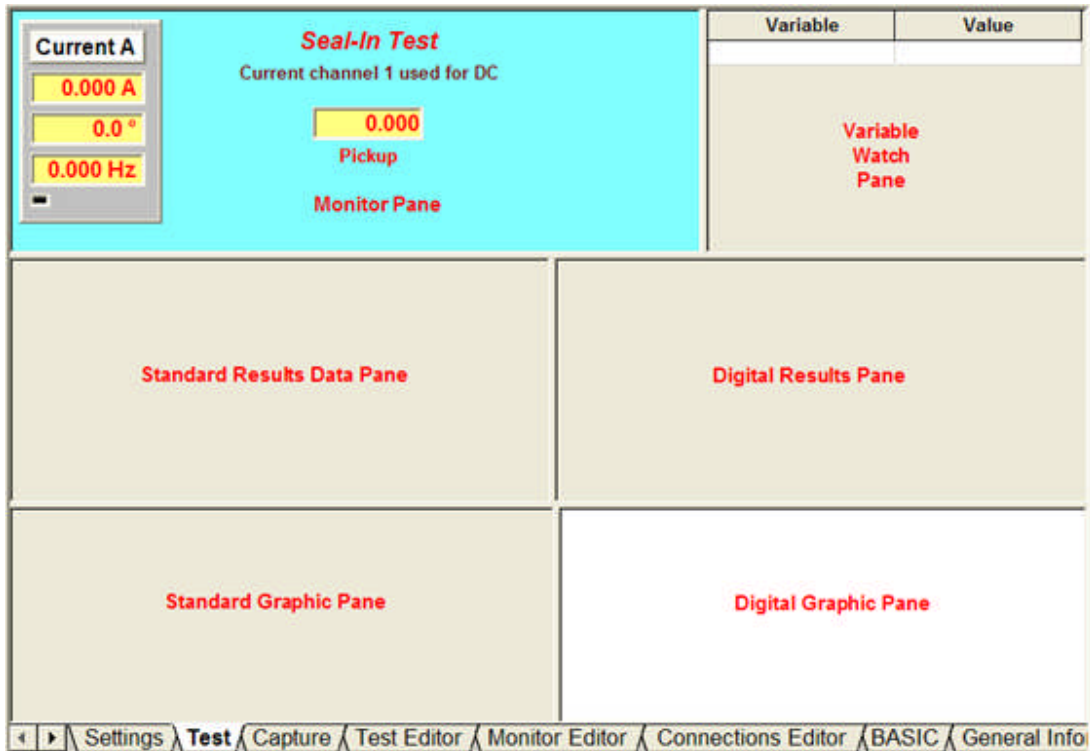
**Note:** An open Relay must first be closed from the Workbook and then reopened to be refreshed from the database. The Delete Setting option for a valid Setting will be dimmed.

**Note:** If the Groups and Settings in both the Device and Relay are identical, the Delete Group and Delete Setting options are dimmed.

## Relay Test Tab

### Relay Test Tab

The Workbook *Test* tab is to view a test execution with the varying parameters that are set in the panes. The three basic panes are the upper Monitor Pane, the middle Results Pane and the bottom Graphics pane. The Monitor and Results panes are always in focus. The Graphics tab is if a Graphics and/or Digital Graphics Control is in place in the Test Editor tab. All available panes are:



The Monitor pane is to display the information set in the Monitor Editor tab.

A sub pane, the Watch Variable pane, is to display variables as desired.

The Standard Results pane is to display the information set in the Log Control of the Test Editor tab.

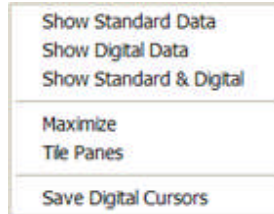
A sub pane, the Digital Results pane, is to view the test results set in the Digital Sequence Control in the Test Editor tab.

The Standard Graphics pane is to display the moving variables set by the Graphics Control in the Test Editor tab to follow the test execution.

A sub pane, the Digital Graphics pane, is to display the active variables set by the Digital Graphic Control in the Test Editor tab to follow the test execution.

## Test Tab Drop Menu

In the Workbook Test tab, a right mouse click in any field will produce a *drop down menu* of options.



The drop down menu options for all panes except the Digital Graphic pane are:

**Show Standard Data** is to display the Monitor, Test Results and Graphics panes.

**Show Digital Data** is to display the Monitor, Digital Results and Digital Graphics panes.

**Show Standard and Digital** is to display the Monitor, Standard and Digital Results, and Standard and Digital Graphics panes.

**Maximize** is to place in full view the selected pane. All other panes will be hidden.

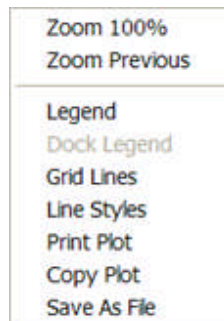
**Tile Panes** is to place in view, equally displaced in the Test tab, the Monitor, Watch Variables, Standard Results, Digital Results, Standard Graphics and Digital Graphics panes. The panes are equally displaced from top to bottom. The Watch Variable pane will occupy about one-fourth of the upper Monitor pane and the Results and Graphics panes equal.

**Save Digital Cursors** is to save in the Results database the cursors set for reference in the Sequence Record of the Digital Sequence Control and displayed in the Digital Results.

The Digital Graphic pane displays the Test Set Digital Inputs and Outputs that have been selected for graphical record by the Digital Graphic Control in the Test Editor tab.

A Zoom-in on a graphic record is accomplished by selecting a location on the trace and performing a left mouse click. Holding the mouse button down, while moving the cursor over an area to expand, and then releasing the mouse button will Zoom in on the trace for more visible detail.

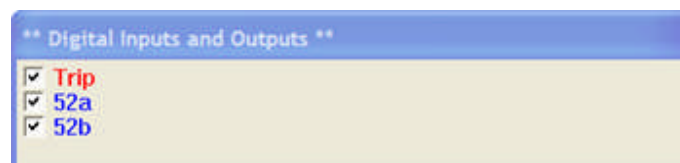
The *drop down menu* options for this Digital Graphic pane are:



**Zoom 100%** places all data collected in this pane to be displayed in one continuous stream per Digital Input and/or Output.

**Zoom Previous** option will return to the previous Zoom In or Zoom Out view.

**Legend** option will display a "floating" Digital Inputs and Outputs dialog. This dialog contains check boxes for the Inputs and Outputs made active in the Test Editor Digital Graphic Control. The check boxes enable or disable the corresponding graph.



**Dock Legend** option will place the Legend "docked" at the bottom of the Digital Graphic pane.

**Grid Line** option will make visible or hide the time grid lines in the Digital Graphic pane.

**Line Styles** option will

**Print Plot** option will display the standard Windows Print dialog to select Printer and Print options to print the Digital Graphic pane data.

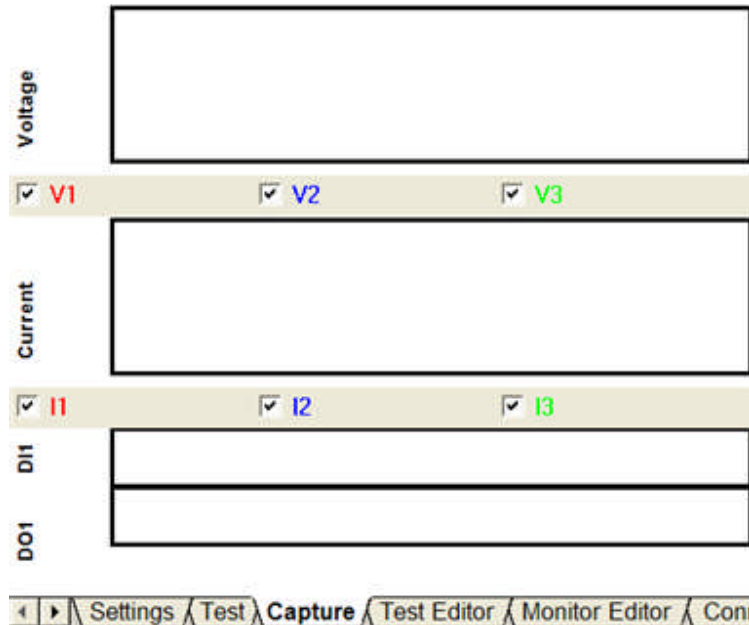
**Copy Plot** option will allow copy of the Digital Graphic pane display for to paste in some other document or program.

**Save As File** option will display the standard Windows Save As dialog to create a file name and location as a bitmap file with the \*.bmp extension.

## Capture Tab

## Capture Tab

The *Capture tab* is to allow recording the analog and digital outputs and the digital inputs to the **MPRT**.



The **MPRT** has metered current and voltage channel outputs. These may be monitored during a test execution for a graphic display to be used as part of the test results. The assignment of the used digital inputs and outputs to the **MPRT** may be displayed to accompany the test results.

For the Capture tab to be populated by test analog and digital data, the Test Editor tab must contain a Dynamic Control in the test procedure.

There are three panes to record the data:

The **upper pane** is for Voltages. The number of Voltage channels displayed is governed by the number of channels selected in the Start Control of the test being executed. If no channels are selected, then there will be no Voltage check boxes for selecting those for record. If four channels are selected, then there will be four Voltage check boxes labeled V1 through V4. The specific channel(s) to be in view is determined by the boxes checked.

The display may be Zoomed to a smaller time range by using the mouse. Place the cursor on an area of the display, hold the left mouse button down and drag a window the size to be Zoomed in. The procedure may be repeated as often as necessary to view the captured data.

The **center pane** is for Currents. The number of Current channels displayed is governed by the number of channels selected in the Start Control of the test being executed. If no channels are selected, then there will be no Current check boxes for selecting those for record. If four channels are selected, then there will be four Current check boxes labeled C1 through C4. The specific channel(s) to be in view is determined by the boxes checked.

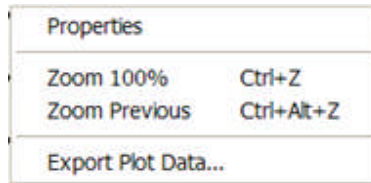
Note: one option will place all analog data in one display.

The **lower pane** is for the digital I/O's used during test execution.

The particular Inputs, Outputs and Analog options for display are selected in the Capture tab drop menu Properties item. The drop menu is described in the Capture Tab Properties Dialog topic.

## Capture Tab Drop Menu

The Capture tab *drop down menu* is for editing the displays and exporting the edited display(s).



The selections are:

**Properties** selection will open the Capture Tap Properties Dialog for display edits.

**Zoom 100%** places all data collected in this pane to be displayed in one continuous stream per Digital Input and/or Output.

**Zoom Previous** option will return to the previous Zoom In or Zoom Out view.

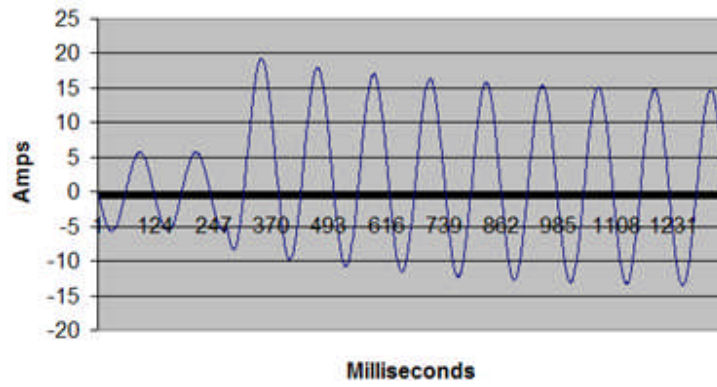
*Export Plot Data...* option opens the Save dialog to create a label and folder to which the Capture tab data will be saved. The user may desire to create an AVTS sub folder labeled Captures for saving these exported data files.

	A	B	C	D	E	F	G
1	Seconds	V1	V2	V3	I1	I2	I3
2	0	0	0	0	0	0	0
3	5.73993	0	0	0	0	0	0
4	5.73993	-0.2719	0.025895	-0.09063	0.001036	-0.00544	-0.00492
5	5.740069	-0.11653	-0.06474	-0.10358	0.010099	-0.00751	-0.0101
6	5.740208	-0.01942	0.045316	-0.18774	0.010617	-0.00829	-0.01372
7	5.740347	-22.7294	-71.5609	89.5061	-0.13543	-0.51169	0.642197
8	5.740486	-20.2046	-82.2879	101.444	-0.10021	-0.56063	0.650483
9	5.740624	-9.45168	-79.2776	91.7072	-0.06551	-0.57772	0.636759
10	5.740764	-5.17253	-81.3557	87.9265	-0.03159	-0.59118	0.620445
11	5.740902	-0.67974	-84.3595	85.3176	0.003107	-0.62019	0.598175
12	5.741041	2.49887	-86.593	82.6634	0.036771	-0.63262	0.583933
13	5.74118	9.97606	-88.9947	79.6142	0.079498	-0.64841	0.558556
14	5.741319	14.8961	-90.7167	76.5845	0.11601	-0.65799	0.541983
15	5.741458	19.5313	-92.3805	73.3412	0.148378	-0.67793	0.517641
16	5.741597	23.9464	-94.0766	69.7547	0.187739	-0.68933	0.4933
17	5.741736	28.7046	-95.3325	66.0517	0.220108	-0.69373	0.466369
18	5.741875	33.8901	-96.2518	62.1351	0.251959	-0.70564	0.434518
19	5.742013	38.7131	-97.1581	58.2055	0.283809	-0.70927	0.407847
20	5.742152	43.3159	-97.4494	53.9911	0.325759	-0.71185	0.381434
65513	14.8384	-83.8157	-1.35949	85.3824	0.012689	-0.00699	-0.01321
65514	14.83854	-81.1226	-7.08876	87.7647	0.012948	-0.00777	-0.00829
65515	14.83868	-78.0994	-11.9829	89.8622	0.013724	-0.01062	-0.01217
65516	14.83882	-75.0244	-16.8771	91.7461	0.006215	-0.00932	-0.00078
65517	14.83896	-71.5609	-21.577	93.3451	0.005438	-0.00725	-0.00414
65518	14.8391	-67.9744	-26.4453	94.8017	0.014501	-0.0145	-0.0088
65519	14.83924	-64.2909	-31.2812	95.9669	0.018903	-0.00544	-0.00777
65520	14.83937	-60.303	-36.1106	96.7762	0.005697	-0.00673	-0.00621
65521	14.83951	-55.752	-40.8364	97.2358	0.019162	-0.00932	-0.00414
65522	14.83965	-52.1979	-45.6788	97.7213	0.010358	-0.0145	-0.00492
65523	14.83979	-47.5692	-50.1716	97.6825	0.012948	-0.00699	-0.01113
65524	14.83993	-42.8239	-54.5025	97.5854	0.014242	-0.0101	-0.00777
65525	14.84007	-38.6418	-58.6328	96.9574	0.003366	-0.00984	-0.00829
65526	14.84021	-33.5405	-62.7048	96.2324	0.016832	-0.00621	-0.00751
65527	14.84035	-28.5298	-66.5049	95.3002	0.010099	-0.00311	-0.00932
65528	14.84049	-23.6098	-70.1367	93.85	0.018903	-0.01139	-0.01062
65529	14.84062	-17.9129	-73.7167	92.4841	0.018903	-0.0101	-0.00233
65530	14.84076	-13.0576	-77.1283	90.7426	0.013983	-0.01424	-0.00337
65531	14.8409	-8.64246	-79.9962	88.5675	0.003625	-0.01269	-0.01139
65532	14.84104	-3.02972	-82.644	86.1786	0.018903	-0.00544	-0.00518
65533	14.84118	1.13938	-85.2658	83.6862	0.011394	-0.0101	-0.00777
65534	14.84132	6.73918	-87.9524	80.9478	0.011135	-0.01683	-0.00829
65535	14.84146	12.3843	-89.8039	77.9634	0.014242	-0.00829	-0.0101
65536	14.8416	17.1425	-91.6295	74.9337	0.012689	-0.00544	-0.0088

Note: Regardless of the number of displays in the tab, the chart or trace where the right mouse click is performed will be the only data exported. The data will be saved to a \*.csv file which may be displayed in an Excel spreadsheet or in Notepad. The data in this file will be two columns, one for the time of each sample and the other for the amplitudes of the chosen trace. ALL RECORDED DATA WILL BE SAVED even if the Capture tab has been Zoomed.

An Excel Graph may be created with the Saved Excel \*.csv file of data can be used in part or all.

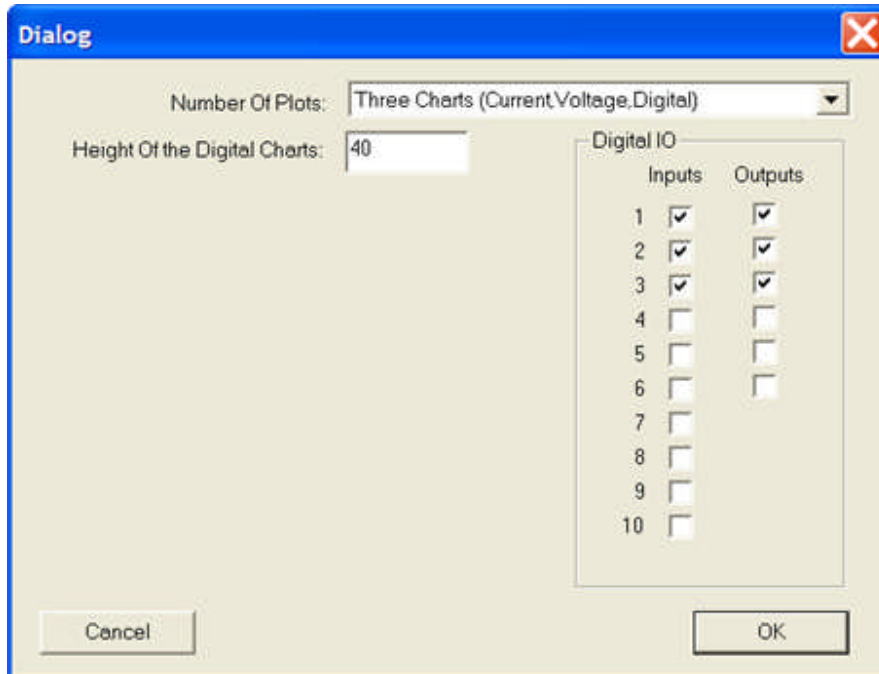
### Exponential





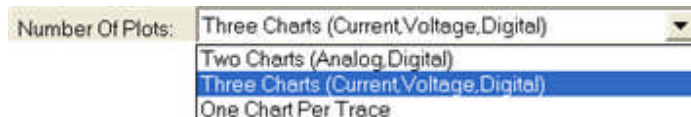
## Capture Tab Properties Dialog

The Capture Tab *Properties dialog* allows editing the displayed channels and I/O's for view and export.

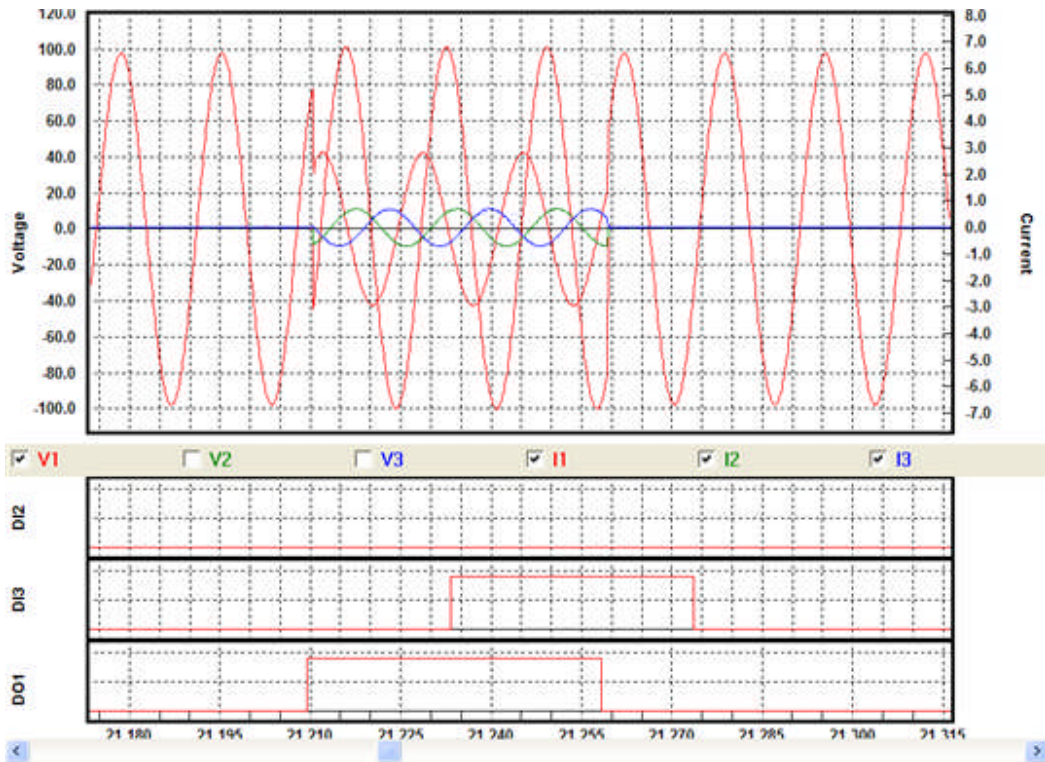


The number of Voltage and Current channels are determined by the number place in the Start Control in the Test Editor. Once the selections are made in the Start Control, they are not visible in the Capture tab until the Relay has been saved and recalled from the database.

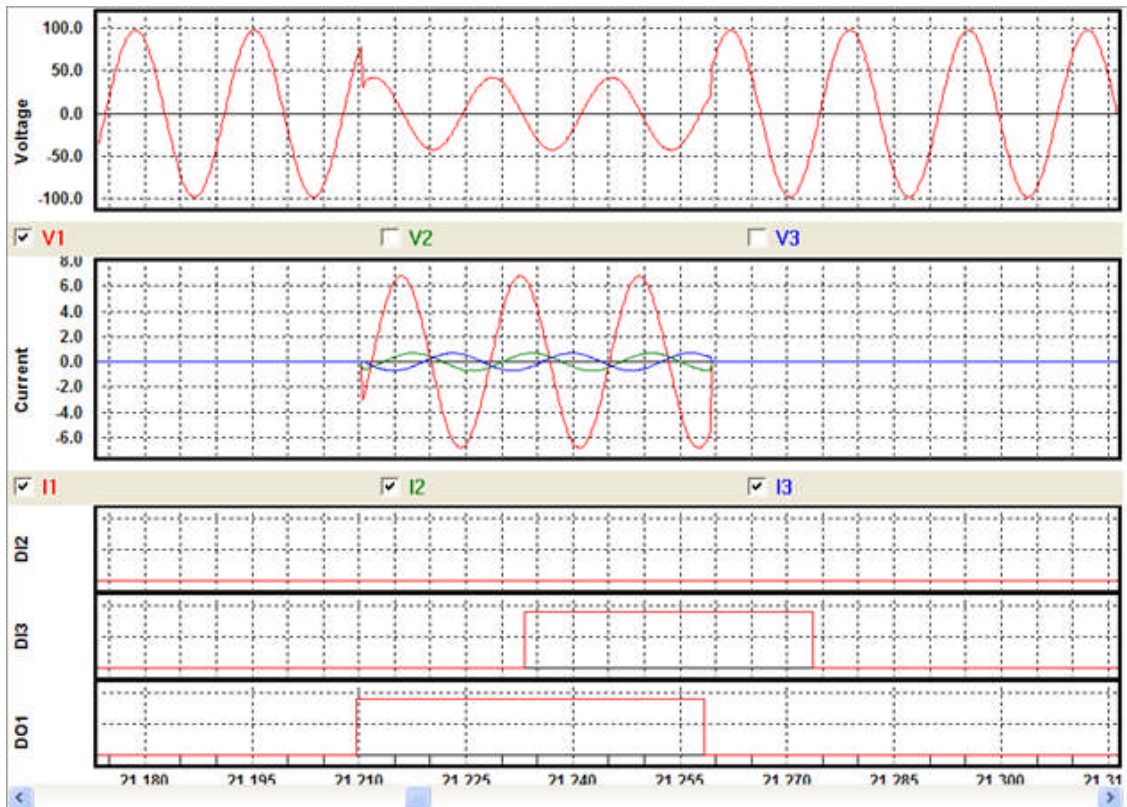
The dialog uses the *Number of Plots*: dialog to select the presentation of the Current and Voltage recordings. The options are:



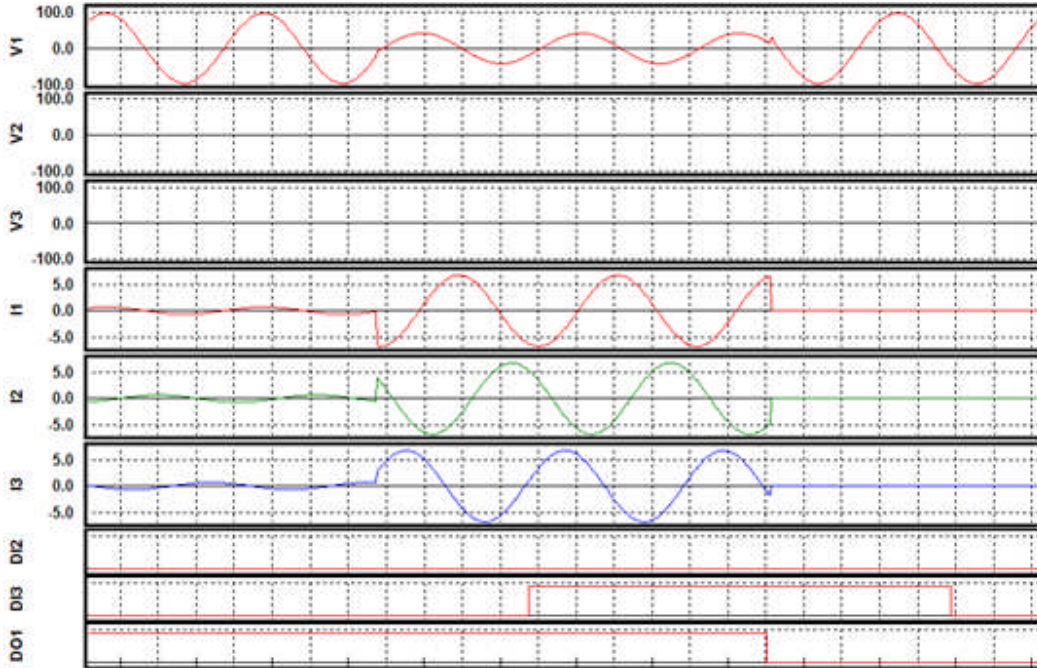
*Two Charts (Analog,Digital)* will set the Capture tab with a chart for the checked combinations of Voltages and Currents.



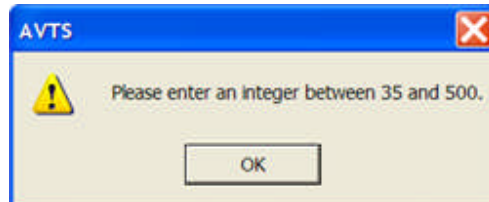
Three Charts (Current, Voltage, Digital) will set the Capture tab with a chart for the checked combinations for the Voltages, a chart for the checked combinations for the Currents and a chart for the Digital I/O's.



**One Chart Per Trace** is determined by the number of Digital IO Inputs and Outputs checked. The display will contain a separate chart for each Voltage and Current Channel set in the Start Control in the Test Editor. There are no check boxes for otherwise selecting the analog channel recordings. The Digital I/O's displayed will be determined by the boxes checked in the Properties dialog.



**Height of the Digital Charts:** edit field allows sizing of the digital charts displayed. The entry range is between 35 and 500 units where the default is 40 units. The message appears when OK is selected if a setting outside the range is made.



## **Test Editor Tab**

### **Test Editor Tab**

The **Test Editor** Tab contains controls linked in a preferential order to conduct a test procedure from Start to End.

The controls for creating a test procedure are displayed in the Test Editor Toolbar at the right hand side of the Workbook pane Test Editor Tab.

Within this tab Test Editor drop menus is available for ready edit of controls and links which are in place in the tab.

Each available control is defined in the Test Editor Toolbar topic.

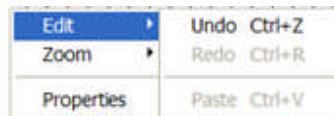
## Test Editor Tab Drop Menu

In the Workbook Test Editor tab, a right mouse click in the Workbook area but not on a Control will produce a *drop down menu* of options.



The options are:

*Edit* is to provide:

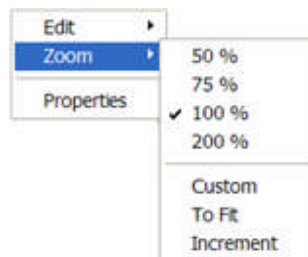


Undo is to return to the preceding edit or delete state.

Redo is to return to the state prior to an Undo.

Paste

*Zoom* is to provide:



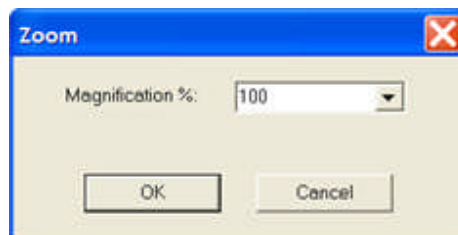
50% is to expand the pane objects to approximately one-quarter size

75% is to expand the pane objects to approximately one-half size

100% is to display the pane objects to the default normal size

200% is to display the pane objects to approximately double size

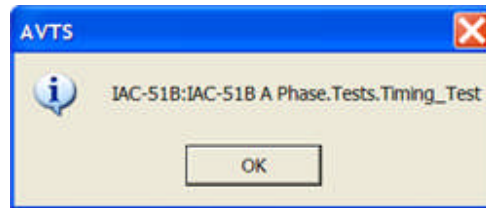
*Custom* is to display the Zoom dialog for edit to make small or enlarge the objects in the pane



To Fit is to display all objects to fit the pane size

Increment to allow the mouse to drag a window about objects and have enclosed objects fill the pane

*Properties* is to display an AVTS message box containing the *DeviceName:RelayName.TestsTab.TestName*

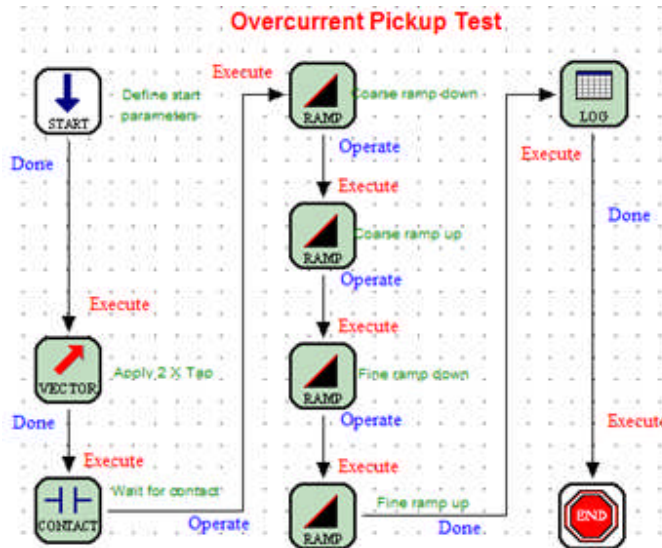


Note: Right mouse click on a Control or a Link between Controls will produce a drop down menu of options to operate on the Control. The Control drop menu options are defined in the topic for each Control in the Test Editor Toolbar group of topics.

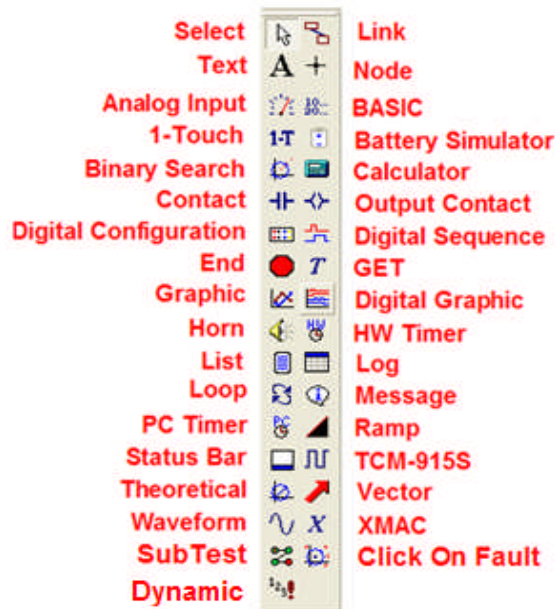
## Test Editor Controls Toolbar

## Test Editor Controls Toolbar

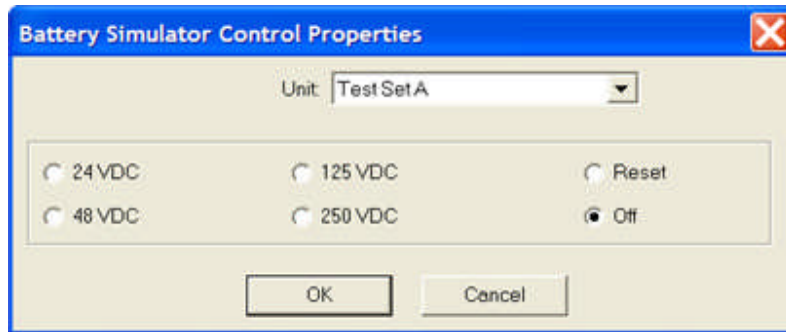
A Test Editor tab test procedure may be created by using Controls connected in a sequence by the Link Control. Each Control topic explains the properties of the Control. The topic will include or link to a demonstration on how to create a test and an *Example Test Procedure* is given.



The *Test Editor tab Controls* tool bar resides to the right hand side of the open Relay Workbook. Each is a program application for placing in the Workbook and linked to create a test sequence.

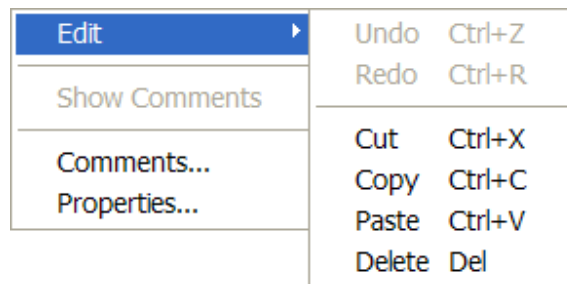


Control Properties are for making activity settings and evoking with a double left mouse click on the Control. An example is viewed by clicking on *Control Properties* menu item. Or, right click the Control to produce a menu of options, one of which is the Control Properties dialog.



Note: Examples provided in the topics for each Control are taken from the RelayTestLibV2.mdb in the C:\Program Files\Megger\AVTS directory.

The *Control Menu Edit* allows Undo, Redo, Cut, Copy, Paste and Delete.



**Undo** negates the previous action taken by the user.

**Redo** replaces an Undo or Delete.

**Cut** removes a highlighted selection and places the detail into the Windows clipboard waiting for further action.

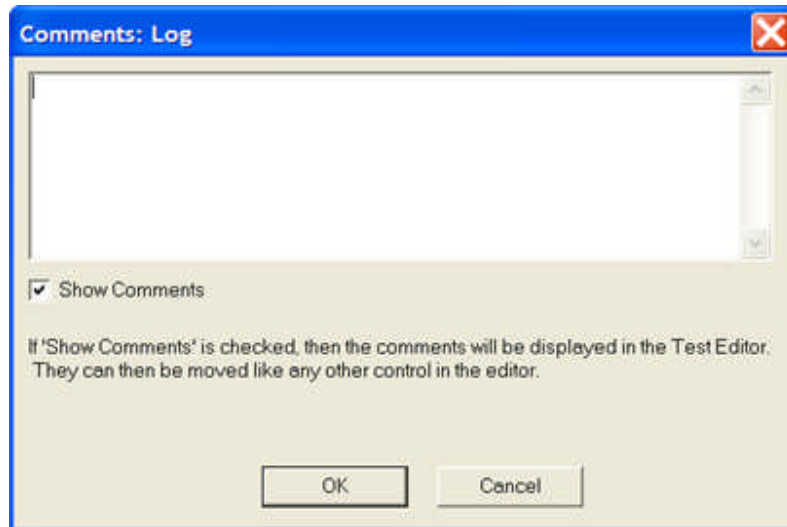
**Copy** places a copy of a highlighted selection and places the detail into the Windows clipboard waiting for further action.

**Paste** places the last selection put into the clipboard.

**Delete** removes a selection from the work area. A deleted selection may be recovered by a Undo action.

The *Control Menu Comments: Log* dialog allows text in paragraph multi-line form. The Comments: Log, once utilized may be Font formatted in the same method as the Text Control. The Show Comments check box must be checked for the comments to be displayed in the Workbook. The comments are able to be positioned in the Workbook separate to the parent Control.





The test procedure will have Controls connected with Links Controls in the sequence of the test procedure from the Start Control to the End Control. The Links have properties and describe the operation when the connected Control is the Target to the Link and when the connected Control is the Source of the Link to another Control.

**Note:** If the Control is deleted, the Comments: Log is deleted also since it is a property of the Control. If the Control is copied or moved, the Comment: Log will accompany the Control.

Each Control and their Links is defined and exemplified in the respective Control topic.

**Note:** The Link Control properties will be explained first Target and then Source. Targets are the link options entering a Control and Sources are the link options leaving the Control.

1-Touch Control Analog Input Control BASIC Control Battery Simulator Control Binary Search Control Calculation Control Click On Fault Control Contact Control Digital Configuration Control Digital Graphic Control Digital Sequence Control Dynamic Control End Control GET Control Graphic Control Horn Control HW Timer Control Link Control List Control Log Control Loop Control Message Box Control Node Control Output Contact Control PC Timer Control Ramp Control Select Control Status Bar Control SubTest Control Text Control TCM-915S Control Theoretical Control Vector Control Waveform Control XMAC Control

## 1-Touch Control

### 1-Touch Control

The **1-Touch Control** is used to enable a script file, embedded in the Workbook BASIC tab, to execute during a test execution. Typical use is to place a macro or script file in the BASIC tab for setting software links to the device under test. The communication to the device usually alters one or more settings in the device for proper response monitoring during test execution.

The 1-Touch Script topic describes the use with Script.

In a very similar method, the 1-Touch Control is used to enable communication to a relay using the Modbus Protocol.

The 1-Touch Modbus topic describes the use with Modbus Protocol.

The 1-Touch SS1 topic describes the use with a CAPE/ProTesT SS1 file converted and used for state record playback.

## 1-Touch Script

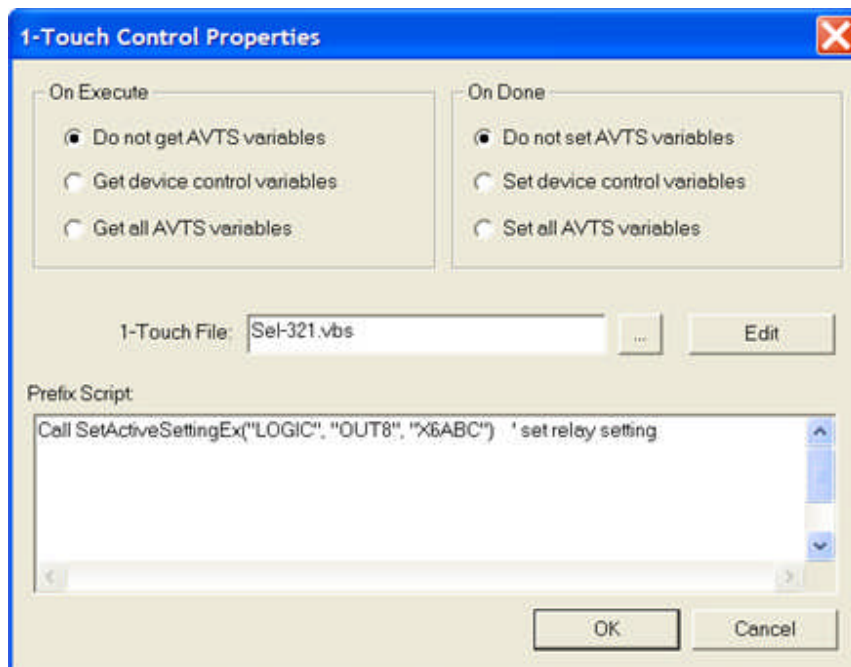
The **1-Touch Control** is used to enable a script file, embedded in the Workbook BASIC tab, to execute during a test execution. Typical use is to place a macro or script file in the BASIC tab for setting software links to the device under test. The communication to the device usually alters one or more settings in the device for proper response monitoring during test execution.

**Note:** The AVTS installation disk contains a complete Help folder for Script programming and format. Browse the disk and open the VBScript folder. The folder contains the scrdoc56en.exe file. Double left click on the file to install the file to the default C:\Program Files\Microsoft Windows Script\ScriptDocs folder automatically. Double click on the Help file Script56.CHM to open and use.



An example of usage of the 1\_Touch Control is explained in Test Editor Charac Z Example.

The *1-Touch Control Properties* has check bullets and entry fields for the Control set up.



The bullet options allow transfer of variables between the AVTS environment and Visual Basic Script.

The AVTS variables and the 1-Touch variables are different. There are functions available in Visual Basic Script with the ability to **get** and **set** variables from AVTS that are used in the visual basic environment (**1-Touch**).

These buttons provide a simple way to move variable values between the two scripting environments without creating script to do so.

**On Execute (On Done)** option "Do not get (Set) AVTS variables" regulates the Control to use variables which are listed in other Controls and the Settings tab.

**On Execute (On Done)** option "Get (Set) device control variables" regulates the Control to call and set the variables that directly set quantities for the test set; i.e., Va, Ib, On.

**On Execute (On Done)** option "Get (Set) all AVTS variables" regulates the Control to call and set the variables available in the active AVTS database.

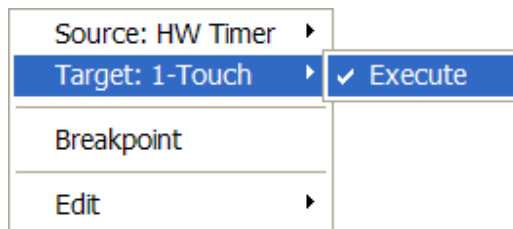
The **1-Touch File** edit field is to enter the name of an existing "script" file to be called and placed in the BASIC tab. The path and file name may be entered via the keyboard or the browse button may be used to locate and select the wanted file.

Typically, a script file will contain the settings to be operated on prior to the execution of the next Control in the test process. For example, the file may contain the interface communications software call to enable a change or disable some relay settings.

The **Edit** button will directly transfer to the Visual Basic Script file in BASIC tab so alterations may be made to the script.

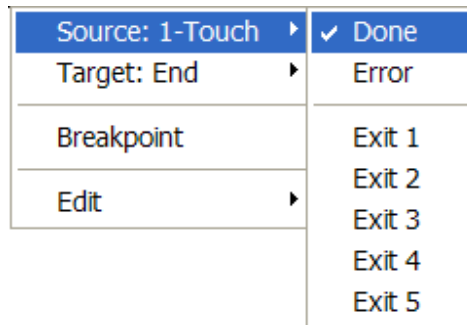
The **Prefix Script** is a keyboard text entry and typically will be the text to directly call a specific routine inside the Visual Basic Script.

The **Target Link** is Execute.



**Execute** command applies immediately after the completion of the properties of the previous Control connected.

The **Source Links** are Done, Error, Exit 1, Exit 2, Exit 3, Exit 4 and Exit 5.



**Done** command concludes the activity of the Control and passes the procedure execution to the next Control.

**Error** command is a branch option from the Control. If an Error occurs within the script, the control can be directed by the script to take this branch option. Usually the Error command will branch to a message dialog and then to an End Control.

**Exit 1** command is a branch option from the Control. This option is executed should a set condition embedded in the properties script or calculation that is met. The Exit 1 is controlled by the properties action.

**Exit 2 through Exit 5** commands are additional branch options governed by the same conditions as the Exit 1 command.

## 1-Touch Modbus

### 1-Touch Modbus

The **1-Touch Control** is used to enable communication to a relay that requires the use of Modbus protocol, such as the GE/Multilin UR series. The created Modbus file to communicate to the relay is named in the Control and the protocol is inserted in the BASIC tab. The use is similar to use of Script files with other relays.

**Note:** The AVTS installation disk contains the Modbus Standard Protocol Specifications document in Adobe pdf format.

The following topics further define the use of Modbus:

- 1-Touch Modbus Properties

- Modbus Device Settings

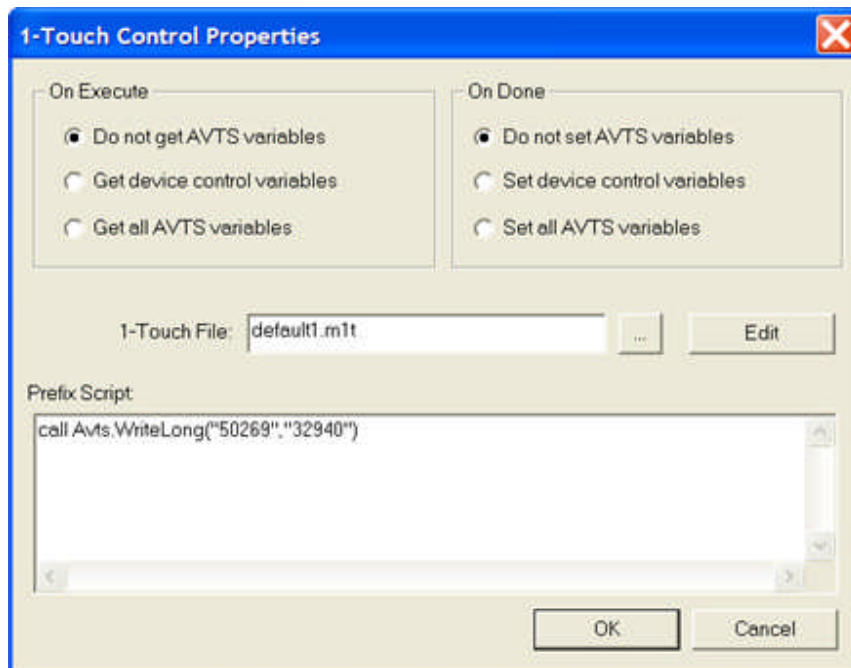
- Modbus Relay Settings

## 1-Touch Modbus Properties

The **1-Touch Properties** for Modbus use require special treatment. Using the 1-Touch Control will enable using Modbus to communicate to particular model relays. Modbus control does not require a separate Script file as the language controls are built within AVTS.



The *1-Touch Control Properties* has check bullets and entry fields for the Control set up defined in the 1-Touch Script topic.



The addition of other settings for the relay in the Settings tab is necessary for the Modbus feature to operate properly.

A new *Group of settings* will need to be added to the Device for the Relay. Modbus will not properly without this Group of settings.

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Address	Integer	0	99999		-1	0	
2 BaudRate	Integer	0	99999		-1	0	
3 Mode	Integer	0	99999		-1	0	

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Contact	Enum			Out01	-1	0	

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Function	Integer	0	1		28976	0	0 = Disabled ; 1 = Enabled
2 Reach	Integer	0	500		28978	2	Reach of the Zone in Ohms
3 Dir	Enum			Forwar	0	0	Direction of the fault
4 VT_Connec	Enum			None	0	0	Xfmr Vol Connection
5 CT_Connec	Enum			None	0	0	Xfmr Curr Connection
6 RCA	Integer	30	90		28986	0	Relay Characteristic Angle, in Deg
7 Comp_Limit	Integer	30	90		28980	0	Comparator limit, in Degree
8 DIR_RCA	Integer	30	90		28987	0	Directional RCA
9 DIR_Comp_Li	Integer	30	90		28988	0	Directional Comparator Limit, in D
10 Quad_Right_Bl	Integer	0	500		0	0	Quad Right Blinder, in Ohms

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 PCTP	Integer	1	65000		17544	0	Phase CT secondary, in amps
2 PCTS	Enum			1,5	0	0	Phase CT primary, in amps
3 GCTP	Integer	1	65000		17546	0	Ground CT Primary, in amps
4 GCTS	Enum			1,5	0	0	Ground CT Secondary, in Amps

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 PVTC	Enum			Wye,D	0	0	Phase VT Connection
2 PVTS	Integer	50	240		17665	1	Phase VT Secondary
3 PVTR	Float	0	24000		17666	0	Phase VT Ratio
4 AVTC	Enum			Vn,Va	0	0	Auxiliary VT Connection
5 AVTS	Integer	50	240		17669	1	Auxiliary VT Ratio
6 AVTR	Float	0	24000		17670	0	Auxiliary VT Secondary

The **Group Name** must be **Modbus** and the Settings must be specific. The values for the Data Types will be entered directly in the Relay Settings tab.

Setting Name 1 = **Address** and Data Type = Integer

Setting Name 2 = **BaudRate** and Data Type = Integer

Setting Name 3 = **Mode** and Data Type = Integer

In a different Device Group, usually the group with the relay settings, add the settings for the relay that need to be addressed in the test procedure.

The settings will be created using the Setting label and made the appropriate Data Type. The Value for the Setting Name will be inserted in the Relay Settings tab and must be in Digital format.

**Note:** The relay instruction manual will have a listing of the recognized labels for all the settings. These names should be used. The manual will also contain the Hex or Digital address for each setting. If given as a **Hex address**, it must be converted to a **Decimal address** to be recognized by Modbus.

## Modbus Relay Settings

The **Relay Settings tab** will display the **Group Modbus** and appropriate edit fields.

The screenshot shows the 'G60 Mod En' interface with several expandable sections. The 'Modbus' section is expanded, showing a table with 3 rows. Below it is the 'Output Contacts' section with 1 row. The 'Phase Distance Z1' section is expanded, showing a table with 10 rows. Below that is the 'CT3 Parameter' section with 4 rows. Finally, the 'VT Parameter' section is expanded, showing a table with 6 rows.

Setting Name	Value	Min Valu	Max Val	Data Type	Description
1 Address	254	0	99999	Integer	
2 BaudRate	19200	0	99999	Integer	
3 Mode	1	0	99999	Integer	

Setting Name	Value	Min Valu	Max Val	Data Type	Description
1 Contact	Out01			Enum	

Setting Name	Value	Min Valu	Max Val	Data Type	Description
1 Function	1	0	1	Integer	0 = Disabled ; 1 = Enabled
2 Reach	5.00	0	500	Integer	Reach of the Zone in Ohms
3 Dir	Forward			Enum	Direction of the fault
4 VT_Connec	None			Enum	Xfmr Vol Connection
5 CT_Connec	None			Enum	Xfmr Curr Connection
6 RCA	75	30	90	Integer	Relay Characteristic Angle, in Degree
7 Comp_Limit	90	30	90	Integer	Comparator limit, in Degree
8 DIR_RCA	75	30	90	Integer	Directional RCA,
9 DiR_Comp_Limit	90	30	90	Integer	Directional Comparator Limit, in Deg.
10 Quad_Right_Blinder	0	0	500	Integer	Quad Right Blinder, in Ohms

Setting Name	Value	Min Valu	Max Val	Data Type	Description
1 PCTP	1000	1	65000	Integer	Phase CT secondary, in amps
2 PCTS	5			Enum	Phase CT primary, in amps
3 GCTP	1	1	65000	Integer	Ground CT Primary, in amps
4 GCTS	5			Enum	Ground CT Secondary, in Amps

Setting Name	Value	Min Valu	Max Val	Data Type	Description
1 PVTC	Wye			Enum	Phase VT Connection
2 PVTS	66.0	50	240	Integer	Phase VT Secondary
3 PVTR	50	0	24000	Float	Phase VT Ratio
4 AVTC	Vag			Enum	Auxiliary VT Connection
5 AVTS	50.0	50	240	Integer	Auxiliary VT Ratio
6 AVTR	10	0	24000	Float	Auxiliary VT Secondary

In the Relay Settings Group the setting **OrderCode** is the GE F60 manufacturer's code for the relay type and used here as an example. The AVTS Settings for Modbus are located in the relay instruction manual memory map of Register Names.

The **Address** may be entered directly for the relay for the value that resides at this Modbus address.

The **BaudRate** may be entered as the rate set to the relay communication setting.

The **Mode** is the Modbus operation mode

Enter **0** = RTU which sends data back and forth between AVTS and the relay

Enter **1** = ASCII which are Modbus commands sent to the relay as ASCII characters

The **1-Touch Control for Modbus** has several script entries for use in communicating with the relay.

The following functions are embedded in AVTS and are called using the proper syntax:

**Avts.ReadSetting(Modbus Address)** placed in the **Prefix Script** field is an AVTS call to read the setting in the relay.

**Avts.ReadSetting "CT2 Parameter", "PCTS"** placed in the **Prefix Script** field is an AVTS call to read the setting in the CT2 Parameter Group labeled PCTS.

The following functions for Modbus control do not require any additional data; there is no requirement for date within the parens other than that shown:

Avts.ReadSettings(): This Function will read and populate all settings into AVTS



Avts.LoadSettings(): This function will read settings and send them to a temp file

Avts.VerifySettings(): alerts the users what settings have changed since the original LoadSettings was completed and asks them if they would like to revert settings to those in the original Load order

Avts.ReadInt( Reg Addr): will read a 16 bit integer from specified address

Avts.WriteLong(Reg Addr,Value):will write a 32 bit number to address specified

(ReadInt)( long RegAddr)

(WriteLong)( long RegAddr, VARIANT Value)

**VARIANT** is a VBScript data type and is handled by the AVTS Modbus features.

(ReadLong)( long RegAddr)

(WriteFloat)( long RegAddr, VARIANT Value)

(ReadFloat)( long RegAddr)

(WriteDouble)( long RegAddr, VARIANT Value)

(ReadDouble)( long RegAddr)

(WriteString)( long RegAddr, long Size, VARIANT Value)

(ReadString)( long RegAddr, long Size)

## Modbus Device Settings

A new *Group of settings* will need to be added to the Device for the Relay. Modbus will not properly without this Group of settings.

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Address	Integer	0	99999		-1	0	
2 BaudRate	Integer	0	99999		-1	0	
3 Mode	Integer	0	99999		-1	0	

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Contact	Enum			Out01	-1	0	

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 Function	Integer	0	1		28976	0	0 = Disabled ; 1 = Enabled
2 Reach	Integer	0	500		28978	2	Reach of the Zone in Ohms
3 Dir	Enum			Forwar	0	0	Direction of the fault
4 VT_Connec	Enum			None	0	0	Xfmr Vol Connection
5 CT_Connec	Enum			None	0	0	Xfmr Curr Connection
6 RCA	Integer	30	90		28986	0	Relay Characteristic Angle, in Deg
7 Comp_Limit	Integer	30	90		28980	0	Comparator limit, in Degree
8 DIR_RCA	Integer	30	90		28987	0	Directional RCA,
9 DIR_Comp_Li	Integer	30	90		28988	0	Directional Comparator Limit, in D
10 Quad_Right_Bl	Integer	0	500		0	0	Quad Right Blinder, in Ohms

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 PCTP	Integer	1	65000		17544	0	Phase CT secondary, in amps
2 PCTS	Enum			1.5	0	0	Phase CT primary, in amps
3 GCTP	Integer	1	65000		17546	0	Ground CT Primary, in amps
4 GCTS	Enum			1.5	0	0	Ground CT Secondary, in Amps

Setting Name	Data Type	Min Value	Max Value	Enum	Modbus Address	Num. Decimals/Size	Description
1 PVTC	Enum			Wye.D	0	0	Phase VT Connection
2 PVTS	Integer	50	240		17665	1	Phase VT Secondary
3 PVTR	Float	0	24000		17666	0	Phase VT Ratio
4 AVTC	Enum			Vn,Va	0	0	Auxiliary VT Connection
5 AVTS	Integer	50	240		17669	1	Auxiliary VT Ratio
6 AVTR	Float	0	24000		17670	0	Auxiliary VT Secondary

The **Group Name** must be **Modbus** and the Settings must be specific. The values for the Data Types will be entered directly in the Relay Settings tab.

Setting Name 1 = **Address** and Data Type = String or Integer

Setting Name 2 = **BaudRate** and Data Type = String or Integer

Setting Name 3 = **Mode** and Data Type = String or Integer

**Note:** The **Modbus Group** settings must have the **Modbus Address** column set to -1 to order AVTS to ignore the setting when communicating with the relay; **Ignore = -1**.

**Note:** Enumerated lists are not recognized by Modbus and must not be used for these type relays.

In a different **Device Group**, usually the group with the relay settings, add the settings for the relay that need to be addressed in the test procedure. The relay settings may be several groups to differentiate the relay functions.

The settings will be created using the Setting label and made the appropriate Data Type.

**Note:** The relay instruction manual will have a listing of the recognized labels for all the settings. These names should be used. The manual will also contain the Hex or Digital address for each setting. If given as a **Hex address**, it must be converted to a **Digital address** to be recognized by Modbus.

The AVTS Data Type drop list has added options to accommodate Modbus and is described in the Devices Settings Tab topic.

The relay *Memory Map* is located in the relay instruction manual and they must be used for Modbus to function properly.

ADDR	REGISTER NAME	RANGE	UNITS	STEP	FORMAT	DEFAULT
7008	Phase Undervoltage 1 Events	0 to 1	---	1	F102	0 (Disabled)
7009	Phase Undervoltage 1 Measurement Mode	0 to 1	---	1	F186	0 (Phase to Ground)
700A	Reserved (6 items)	0 to 1	---	1	F001	0
7013	...Repeated for Phase Undervoltage 2					
<b>Phase Overvoltage (Read/Write Grouped Setting)</b>						
7040	Phase Overvoltage 1 Function	0 to 1	---	1	F102	0 (Disabled)
7041	Phase Overvoltage 1 Source	0 to 5	---	1	F167	0 (SRC 1)
7042	Phase Overvoltage 1 Pickup	0 to 3	pu	0.001	F001	1000
7043	Phase Overvoltage 1 Delay	0 to 600	s	0.01	F001	100
7044	Phase Overvoltage 1 Reset Delay	0 to 600	s	0.01	F001	100
7045	Phase Overvoltage 1 Block	0 to 65535	---	1	F300	0
7046	Phase Overvoltage 1 Target	0 to 2	---	1	F109	0 (Self-reset)
7047	Phase Overvoltage 1 Events	0 to 1	---	1	F102	0 (Disabled)
7048	Reserved (6 items)	0 to 1	---	1	F001	0
<b>Distance (Read/Write Grouped Setting)</b>						
7060	Distance Signal Source	0 to 5	---	1	F167	0 (SRC 1)
7061	Memory Duration	5 to 25	cycles	1	F001	10
7062	Force Self-Polarization	0 to 65535	---	1	F300	0
7062	Force Memory Polarization	0 to 65535	---	1	F300	0
<b>Phase Distance (Read/Write Grouped Setting) (5 modules)</b>						
7070	Phase Distance Zone 1 Function	0 to 1	---	1	F102	0 (Disabled)
7071	Phase Distance Zone 1 Current Supervision	0.05 to 30	pu	0.001	F001	200
7072	Phase Distance Zone 1 Reach	0.02 to 500	ohms	0.01	F001	200
7073	Phase Distance Zone 1 Direction	0 to 2	---	1	F154	0 (Forward)
7074	Phase Distance Zone 1 Comparator Limit	30 to 90	degrees	1	F001	90
7075	Phase Distance Zone 1 Delay	0 to 65,535	s	0.001	F001	0
7076	Phase Distance Zone 1 Block	0 to 65535	---	1	F300	0
7077	Phase Distance Zone 1 Target	0 to 2	---	1	F109	0 (Self-reset)
7078	Phase Distance Zone 1 Events	0 to 1	---	1	F102	0 (Disabled)
707A	Phase Distance Zone 1 RCA	30 to 90	degrees	1	F001	85
707B	Phase Distance Zone 1 DIR RCA	30 to 90	degrees	1	F001	85
707C	Phase Distance Zone 1 DIR Comp Limit	30 to 90	degrees	1	F001	90

Instruction Manual Settings Group label should be used for the AVTS Group label name.

**Phase Distance (Read/Write Grouped Setting)(5 Modules)** may be the AVTS Group label name **Phase Distance Z1**

The **ADDR** column contains the **Modbus Address** for the relay function. The value is a hexadecimal value and must be converted to decimal value. The decimal value is then placed in the **AVTS Device Modbus Address** column for that function.

**ADDR 7070** decimal equivalent is 28784

**ADDR 7072** decimal equivalent is 28786

**Note:** Be certain that the ADDR for the relay to be tested agrees to the Address in AVTS. If there has been a relay firmware change, the change may alter the ADDR of another relay of the same basic model.

The **Register Name** is the label to be used in the AVTS Setting Name column.

**Phase Distance Zone 1 Function** may be the AVTS Setting Name **Function**

**Phase Distance Zone 1 Reach** may be the AVTS Setting Name **Reach**

The **Range** values are the AVTS entries for the Min Value and Max Value columns

The **Step** value is a divisor for the decimal setting decade entry in the AVTS **Num. Decimal/Size** column

A relay Step value of 0.003 has an AVTS setting of 3 or other value provided in the Data Format

A relay Step value = 1 has an AVTS setting of 0 or other value provided in the Data Format

The **Format** column entry is described in the **Date Formats** list after the Memory Map.

Note: Step is regulated by the **Format** column entry. For example:

F001 UR\_UINT16 UNSIGNED 16 BIT INTEGER

F102 ENUMERATION: DISABLED/ENABLED 0 = Disabled; 1 = Enabled

The **Default** column entry is the value of the setting defaulted to the relay memory when the relay is produced.

For a **Data Type String**, the Modbus Address must have a integer = to the relay string character length.

## 1-Touch SS1

The 1-Touch Control is used to enable a script file, embedded in the Workbook BASIC tab, to execute during a test execution.

When used with a SS1 file, the 1-Touch Control references a SS1 \*.m1t file and uses the file directly with the file imported to the BASIC tab.

Use of the SS1 Import and implementation is described in the following topics:

- SS1 Converter... topic describes the conversion of a CAPE/ProTest file to one useful in AVTS.

- SS1 Converter Dialog topic describes the procedure to call a CAPE/ProTest file to convert.

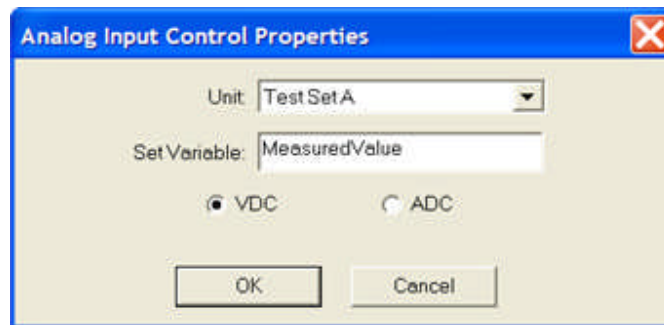
- SS1 Converter Example topic demonstrates the conversion and use of a CAPE/ProTest file converted to AVTS use.

## Analog Input Control

The **Analog Input Control** is used in a test procedure to control and collect the value of inputs to the MPRT test set Analog Input terminals.



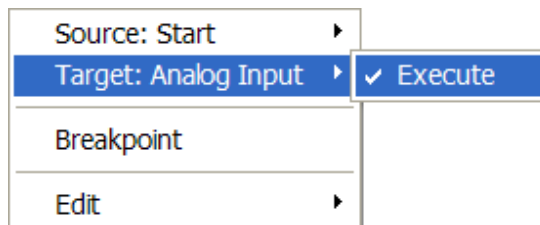
The *Analog Input Control Properties* allows the setting of Test Set A or Test Set B to measure whether DC amperes or DC voltage is to be measured. The value measured may be given a Variable name to Log the test results. The default Set Variable name is Measured Value.



**Note:** The maximum input to the MPRT is either +/- 10 Volts DC or +/- 20 milliamperes DC.

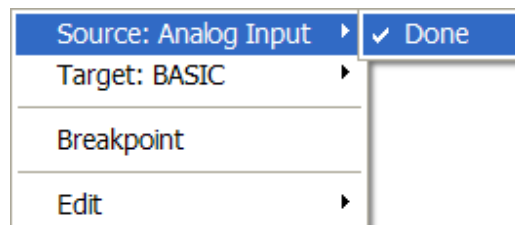
The Control may be placed in a test procedure sequence to set the MPRT outputs as the analog sources to a single or multiphase transducer. The transducer output is connected to the MPRT Transducer DC Input terminals to measure the values and compare to accuracy specifications. Generic test procedures for various transducer types may be created using a Vector and/or Ramp Controls and saved as for use by immediate call from the saved database.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## BASIC Control

The **BASIC Control** is used as access to the Workbook BASIC tab. The BASIC tab is the location for the Macros for Import PulseMaster Database or for script files using the 1-Touch Control. The BASIC Control may be placed within a test procedure at the location where other tables, calculations or specific BASIC programming language is necessary for the successful completion of the test.

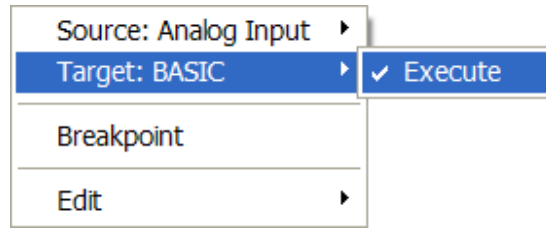


An example of this Control is demonstrated in Test Editor Differential Example.

The *BASIC Control Properties*

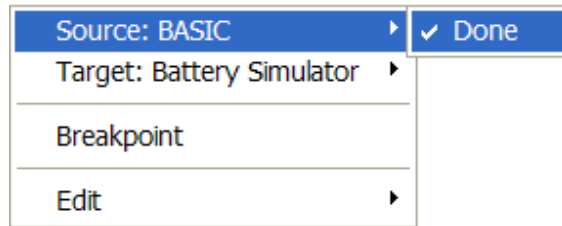
Type	Amplitude	Phase	Frequency	On/Off	Ramp Ampl.	Ramp Phase	Ramp Freq.
VA	0.00	0.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
VB	0.00	120.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
VC	0.00	240.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
IA	0.00	0.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
IB	0.00	120.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
IC	0.00	240.00	60.00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

*Target Link* is Execute.



The Execute command applies immediately after the properties of the previous connected Control are completed.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

If a PulseMaster relay database has been converted for use in AVTS, the Basic Control properties dialog will host the PulseMaster specification screen data that accompanied the relay test selected. These parameters are transmitted to the test set for both initial start settings and the method of Pulsar current, voltage and timer modules operation during the test execution. The entries may be numeric or variables and apply to all the entry fields.

Data entries in the BASIC Control will be passed to the remaining test procedure.

Tests developed for Pulsar and MPRT, using other AVTS Controls, usually do not employ the BASIC Control.

Tests requiring special formatting for communication to other devices will use this Control to allow the use of the Workbook BASIC tab. The BASIC tab can be accessed only with the presence of a BASIC Control or 1-Touch Control resident in the Workbook Test Editor tab.

**Note:** The BASIC Control does not have to be linked to other Controls to have access to the BASIC tab. It will need to be linked for input and output of data during the test execution.

**The File Name:** will be that of the test in focus and each test will have a different file name. The file names are assigned by AVTS automatically from the PulseMaster test module converted to a Microsoft ACCESS \*.abm database. Each file name is the name for the PulseMaster test used.

Each test will have data for the BASIC Control Properties and the Workbook BASIC tab will contain a listing of all the PulseMaster macros used in that test. The macros used may be special instructions to complete the test, or the instructions to collect results, compare to a table of values, print results and print graphics.



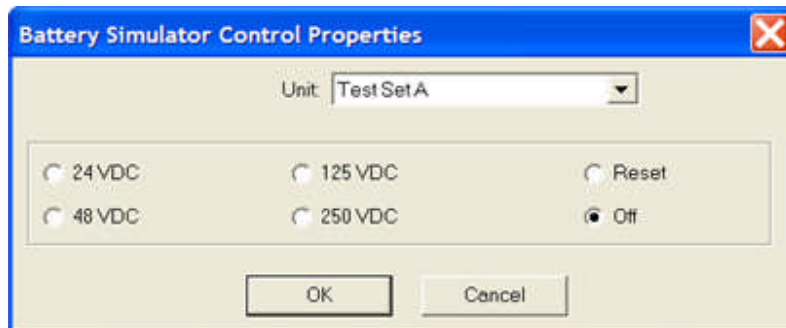
## Battery Simulator Control

The **Battery Simulator Control** is for active control of the MPRT DC voltage output. It may be used without need of a Relay Test by selecting the Tools/Battery Simulator menu item.

**Note:** The Battery Simulator is for use with MPRT only.



*Battery Simulator Control Properties* provides selection of one of the connected MPRT test sets. Once opened, the dialog has no options selected except for the default Unit "TestSetA". Once the test set is selected, the DC voltage discrete output level may be set by placing a check to bullet that level. The selection of a level justifies an ON for the MPRT. The output may be turned OFF by checking the OFF bullet. Several Battery Simulator Controls may be linked throughout a test procedure to turn ON and OFF various levels of output.

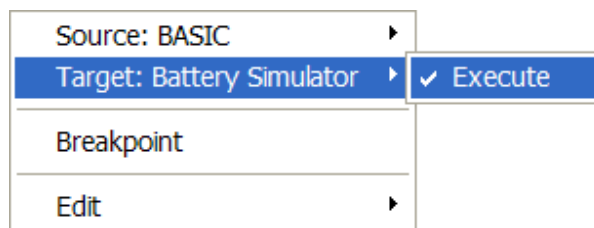


**Ok** acknowledges a selected or de-selected check button.

**Cancel** ignores any selections, closes the dialog and returns to the Test Editor test procedure.

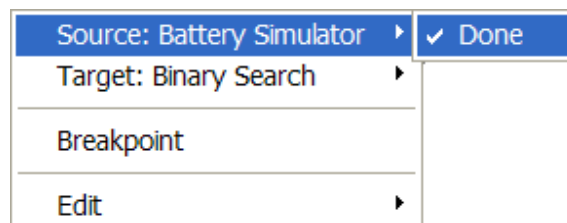
The **Reset** check button is not functional.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Binary Search Control

### Binary Search Control

The *Binary Search Control* is to quickly determine the characteristic of an object with a known or unknown characteristic. Inputting the polar angle to begin the search and setting the operating variables are the only necessary requirements. On execution the Control will incrementally search for the relay pickup, whether known or unknown.

**Binary Search Control Properties**

Voltages

Name	Amplitude	Phase	Frequency	Mode	On
Va	$I_a \cdot Z$	0.0	60.000	AC	<input type="checkbox"/>
Vb	Va	120.0	60.000	AC	<input type="checkbox"/>
Vc	Va	240.0	60.000	AC	<input type="checkbox"/>

Currents

Name	Amplitude	Phase	Frequency	Mode	On
Ia	2.000	$V_a \text{Phase} + Z \text{Phase}$	60.000	AC	<input type="checkbox"/>
Ib	2.000	$I_a \text{Phase} + 120$	60.000	AC	<input type="checkbox"/>
Ic	2.000	$I_a \text{Phase} + 240$	60.000	AC	<input type="checkbox"/>

Show Vectors:  Prefault  Fault

Test Point Definition

MTA:

Test From:  to  in  steps

Operation

Mode:  Max Value:  V

Fault:  Max Value:  Ohms

Min Value:  Ohms

Reset Time:  Seconds

Calculation:

OK Cancel



The Binary Search Control Properties topic describes the properties.

An example for use is described in the Binary Search Control Example.

If a characteristic has been defined in a Theoretical Control object, then operation is checked at the halfway point between the extremes at the max torque angle. If an operation by the relay occurs, then the Center point is defined as the halfway point between the extremes at the max torque angle. If there is no operation, then the theoretical characteristic is ignored.

If there is no theoretical characteristic then operation is checked at the origin. If there is no operation at the origin then a linear search is applied in both directions at the max torque angle until an operation point is found. When an operation is found then a binary search is done in both directions to find the extremes of the characteristic, and the Center point is defined as the halfway point between the extremes.

The test first begins a search at the Test Point Definition MTA to locate the object point farthest from the origin and the point closest to the origin. The tests then establish a Center point for conducting further tests specified by the number of Steps input.

The search starts at the impedance point governed by the entries in the Operation block.

The test then jumps and applies values half the distance between the max impedance and the min impedance. If operation occurs, the next test will be between the operate point and half the distance back to the max impedance; or else, the next test will be between the operate and half the distance to the min impedance point. The process continues up and down the test angle until the characteristic is isolated at that angle.

The resolution, or accuracy, of the two found characteristic locations along the defined MTA is governed by the calculations of Min and Max value entries.

When found, the test calculates a Center Point between the two extreme points to use as the min impedance value for the remain tests in the defined steps.

After the two MTA impedance locations are found, the test then performs similar testing for the characteristic about the MTA. Assume that the MTA is 74 degrees and the Test From = 0, Test to = 180 and steps = 15. The test first determines the Max value and Min value of Z at the MTA.. Then the test establishes a test origin at the half distance between the Max and Min Z's at the MTA. The test continues searching for Z at the various angles from the Test From: through the To; entry in the number of Steps entered. The number of tests will be the number of Steps/2 about the MTA entry.

## Binary Search Control Properties

The Binary Search Control properties are defined below.



*Binary Search Control Properties* include blocks of data fields for Voltages, Currents, Test Point Definition and Operation information.

**Binary Search Control Properties**

Voltages

Name	Amplitude	Phase	Frequency	Mode	On
Va	$I_a \cdot Z$	0.0	60.000	AC	<input type="checkbox"/>
Vb	Va	120.0	60.000	AC	<input type="checkbox"/>
Vc	Va	240.0	60.000	AC	<input type="checkbox"/>

Currents

Name	Amplitude	Phase	Frequency	Mode	On
Ia	2.000	$V_a \text{Phase} + Z \text{Phase}$	60.000	AC	<input type="checkbox"/>
Ib	2.000	$I_a \text{Phase} + 120$	60.000	AC	<input type="checkbox"/>
Ic	2.000	$I_a \text{Phase} + 240$	60.000	AC	<input type="checkbox"/>

Show Vectors:  Prefault  Fault

Test Point Definition

MTA:

Test From:  to  in  steps

Operation

Mode:  Max Value:  V

Fault:  Max Value:  Ohms  
Min Value:  Ohms

Reset Time:  Seconds

Operation Properties

Calculation:

Custom Calculation

OK Cancel

The **Voltages** and **Currents** blocks display the data to be applied for the designated Prefault and Fault conditions selected for the other options in the Binary Search Control Properties. When first opened, the Properties display default values for all edit fields and may be edited for other user inputs.

**Show Vectors** options the check buttons for Prefault and Fault

The **Prefault** option button allows entries in the Voltages and Currents blocks for the data to be applied prior to any applied Fault data. The Prefault apply time is the Reset Time in Seconds. The default *MTA* is 60 degrees

The **Fault** option button allows entries in the Voltages and Currents blocks for the data to be applied after the applied *Prefault* data. These entries may be numeric for a known pickup or may be global by using a *Calculation* drop list selection.

**Test Point Definition** block

**MTA** is to enter the value in degrees for the expected maximum torque angle of the object. The default MTA is 60 degrees.

**Test From:** entry is to enter the phase angle to begin the test. The first test will conduct at the MTA + the Test From entry.

**"to"** entry defines the phase angle value to cease the tests. The test will conduct the test at MTA + degrees "to" the entry.MTA - degrees entry.

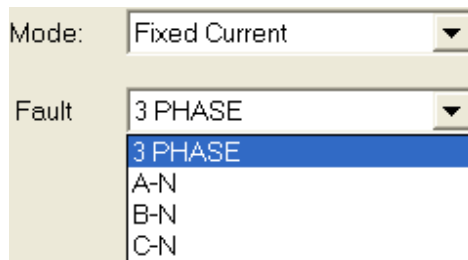
**"in"** entry is to enter the number of **Steps** to perform tests between the From and to entries. The number of steps will include the From and "to" executions. The default is Test From 0 through 360 in 16 steps. The first test is at MTA, second at MTA + 20, third at MTA + 40, ....., to MTA + 360.

**Operation** is to enter data to govern the tests applications

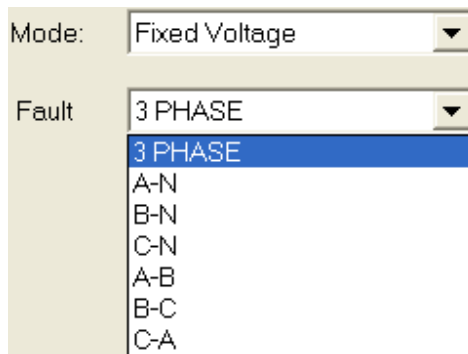
**Mode:** drop list is to select the Voltage or Current set in the Voltage or Current block to be held constant in the calculations of the other parameters for each test application.

**Fault** drop list is to select the fault to be applied for the test process.

*Mode Fixed Current Faults* drop list is to select for Fixed Currents or Fixed Voltages to be held constant while ramping the alternate during test execution. Fixed Current Faults option is default on first use.



*Mode Fixed Voltage Faults* drop list is to select the Fault type to be applied during the test execution. The options are 3-Phase, the three line-to-line and all four of the line-to-ground fault types



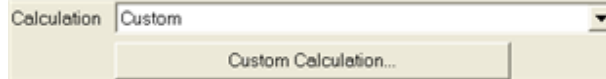
**Max Value V or A** is to enter the maximum quantity allowed for selected Mode. The value will be designated as Volts for Fixed Current Mode and Amps for Fixed Voltage Mode.

**Max Value Ohms** is to set the maximum reach from the origin to begin any test.

**Min Value Ohms** is to set the minimum value of Ohms to test. It is the minimum value of reach between an operate and no-operate point to determine accuracy of the relay characteristic operate point at the test angle.

**Reset Time:** is the time in Seconds that the application between each test and each test jump that the test returns to Prefault values. The setting allows relay logic reset time to assure accuracy in the operate values.

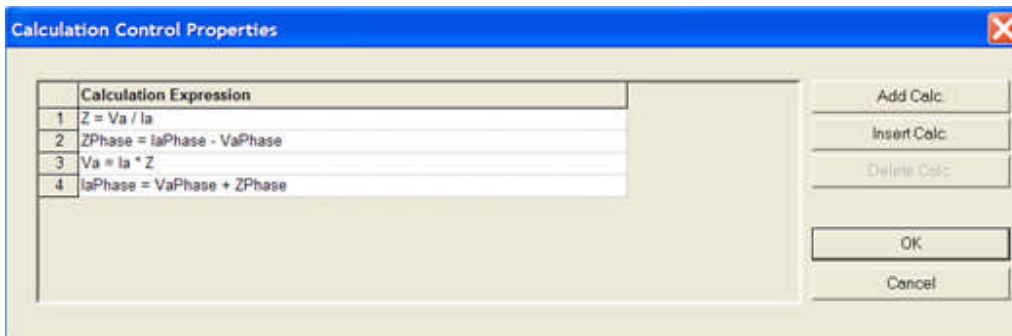
*Calculation* has a drop list and edit field.



The edit field is to enter variables for the calculations necessary for the necessary quantities for the test to operate. The defaults are

$$Z = Va / Ia : ZPhase = IaPhase - VaPhase$$

The *Custom* option makes active the Custom Calculation... button.



The Custom Calculation opens the Calculation Control Properties dialog to enter a user defined listing of BASIC Expressions to use for the test parameters and to pass to a Log Control to store test results. The default entries are:

$$Z = Va / Ia$$

$$ZPhase = IaPhase - VaPhase$$

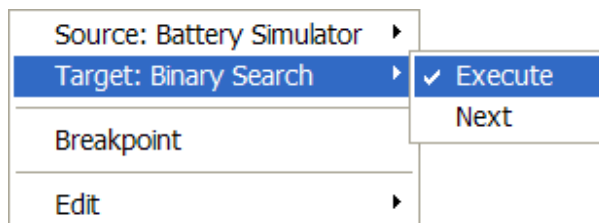
$$Va = Ia * Z$$

$$IaPhase = VaPhase + ZPhase$$

**Note:** These default variable equalities are the default variables in the respective Binary Search Control Properties edit fields.

**Operation Properties** button opens the Contact Control Properties dialog to set the test set mode of contact use for device operate recognition. The Max Time to Operate: may be set to limit the fault apply time in a no-operate region, but should be longer than known operate time of the device.

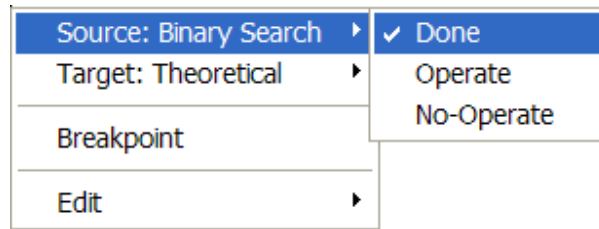
*Target Links* are Execute and Next.



The **Execute** command applies immediately after the completion of the properties of the previous Control connected.

The **Next** command initiates the next item of List or Loop Values in the properties box.

*Source Links* are Done, Operate and No-Operate.



The **Done** command concludes the activity of the Control and passes the procedure execution to the next Control.

The **Operate** command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

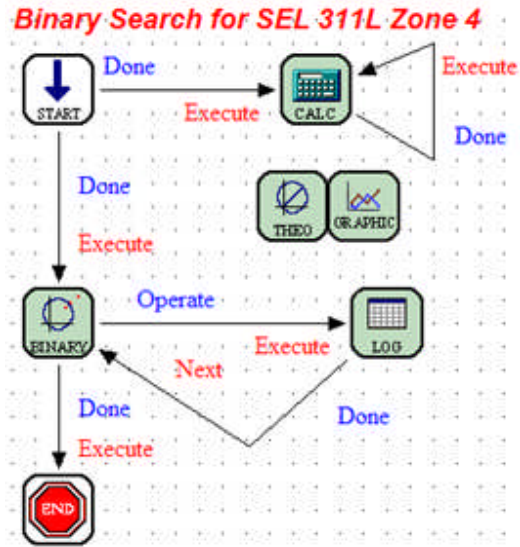
The **No-Operate** command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

**Note:** If there is a non-operation of a required response and the Control properties have expended to the limits, it may constitute a No-Operate and this branch is exercised indicating an improper condition to abort the procedure.

## Binary Search Control Example

A SEL 311L relay was chosen to present an example of use for the Binary Search Control.

The *Test procedure* was established in the Test Editor tab using the following Controls:



*Start Control* is set normally to the number of Voltages and Currents required for the test. The maximum number of each entered must agree to the number of amplifiers entered in the *Tools/Options/Devices tab*.



**AVTS Options**

General | Default Settings | Organization | Channel Labels | New Password | Devices | Scanner

MPRT

	Test Set A	Test Set B
Installed	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Log Commands	<input type="checkbox"/>	<input type="checkbox"/>
Simulate Device	<input checked="" type="checkbox"/>	<input type="checkbox"/>
Num V/I Channels:	3	3
Num Current Channels:	3	3
Num Digital Inputs:	10	10
Num Digital Outputs:	6	6
IP Address	0 0 0 0	0 0 0 0
COM	1	
Baud Rate	38400	38400
Serial	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
Ethernet	<input type="checkbox"/>	<input type="checkbox"/>
IEEE-488	<input type="checkbox"/>	<input type="checkbox"/>
Xon/Xoff	<input type="checkbox"/>	<input type="checkbox"/>
RTS/CTS	<input type="checkbox"/>	<input type="checkbox"/>

TCM-915S

	TCM A	TCM B	TCM C
Installed	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Log Commands	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Simulate Device	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
GPiB Address	1	1	1

Auxiliary COM Ports

COM A:

COM B:

Log All Commands       Simulate All Devices

Block Modbus Writes

OK    Cancel    Apply    Help

**Start Control Properties**

Current Phases: 4      Voltage Phases: 4

Digital Inputs: 0      Digital Outputs: 0

Analog Inputs: 0      Analog Outputs: 0

Primary Settings Group: Group Name

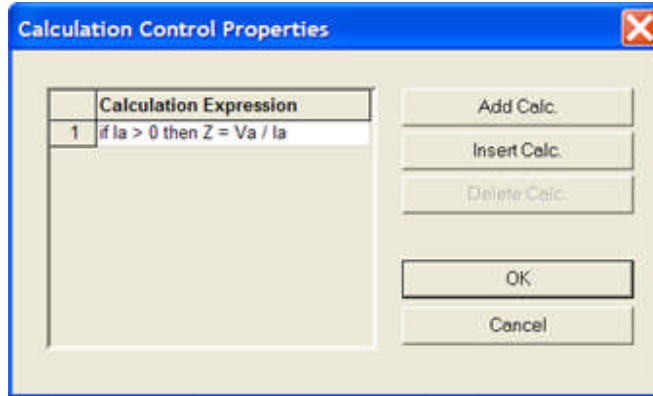
Reset Units: Soft Reset

Angle Mode: Log

Use MPRT Functionality

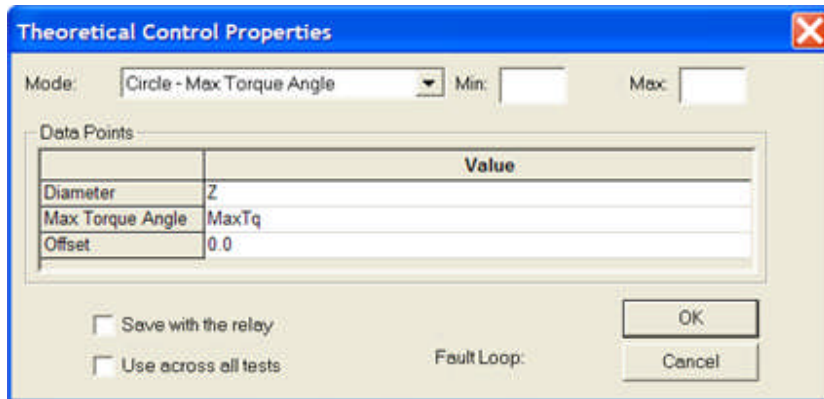
OK    Cancel

*Calculation Control* is to enter a formula to create an active impedance Vector during the executed test.

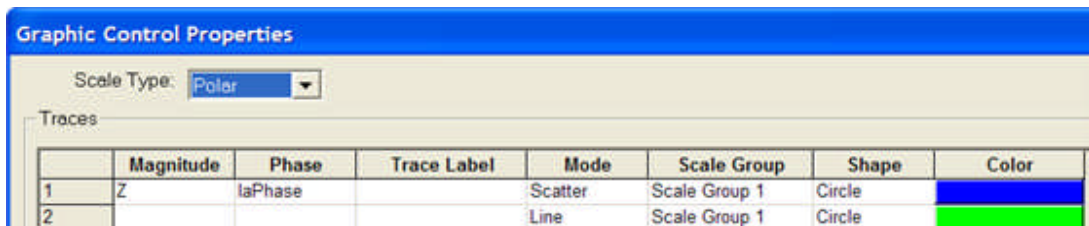


A typical entry is: *if Ia > 0 then Z = Va / Ia* which only allows the calculation when Ia is greater than zero. If the "if" condition is not present, a calculation error when the test is executed.

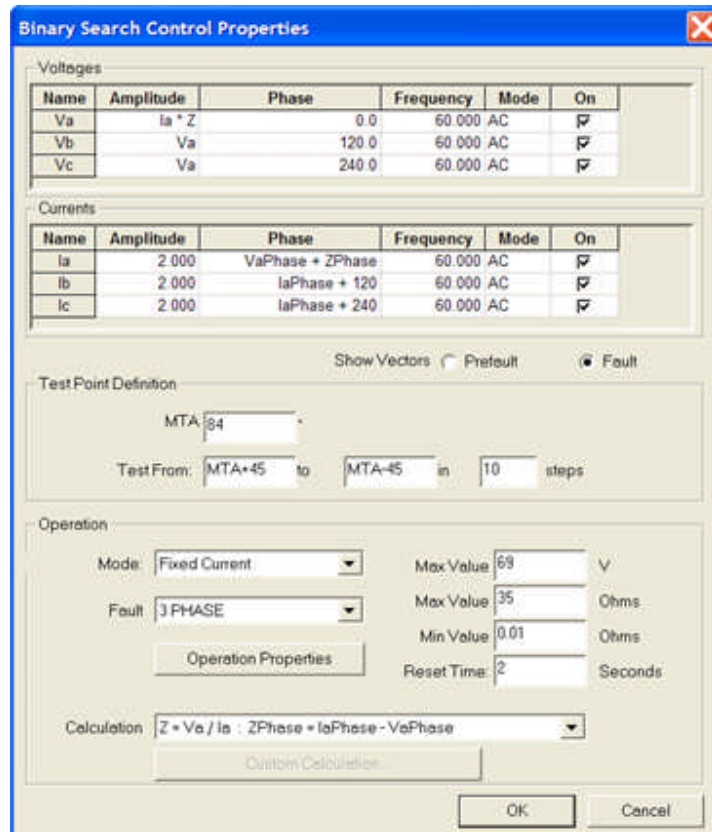
*Theoretical Control* establishes the characteristic of the relay zone under test. The example relay has a Zone 4 mho setting Z = 26 ohms at a MTA = 84 degrees.



*Graphic Control* is set to reflect the conditions to display the mho characteristic, the test points and the active test vector in a Polar display in the Test tab during the test execution. The Magnitude and Phase entries are selected from the list of Variables labeled in the Log Control. The selections are made from a drop down list in the edit field for each.



*Binary Search Control* options remain the Fault option default values when opened. The Test Operation Definitions are edited for the test values.



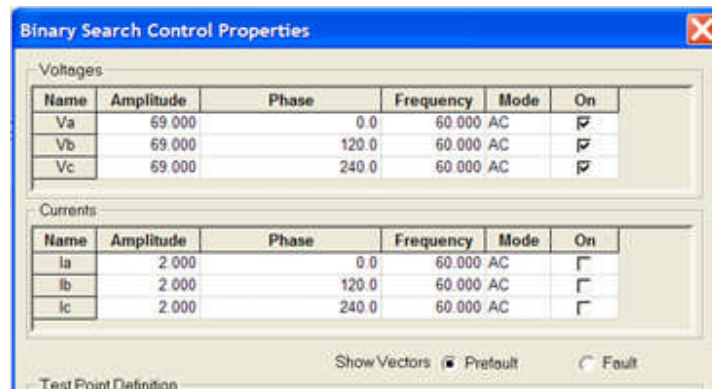
MTA = 84

Step From MTA + 45

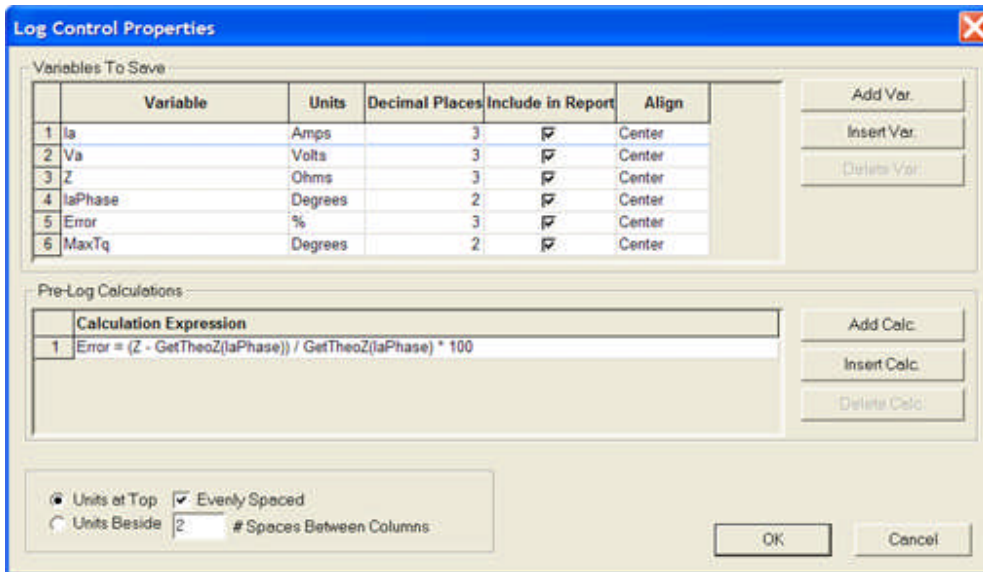
To MTA - 45 in

Steps 10

The *Prefault options* remain the default values.

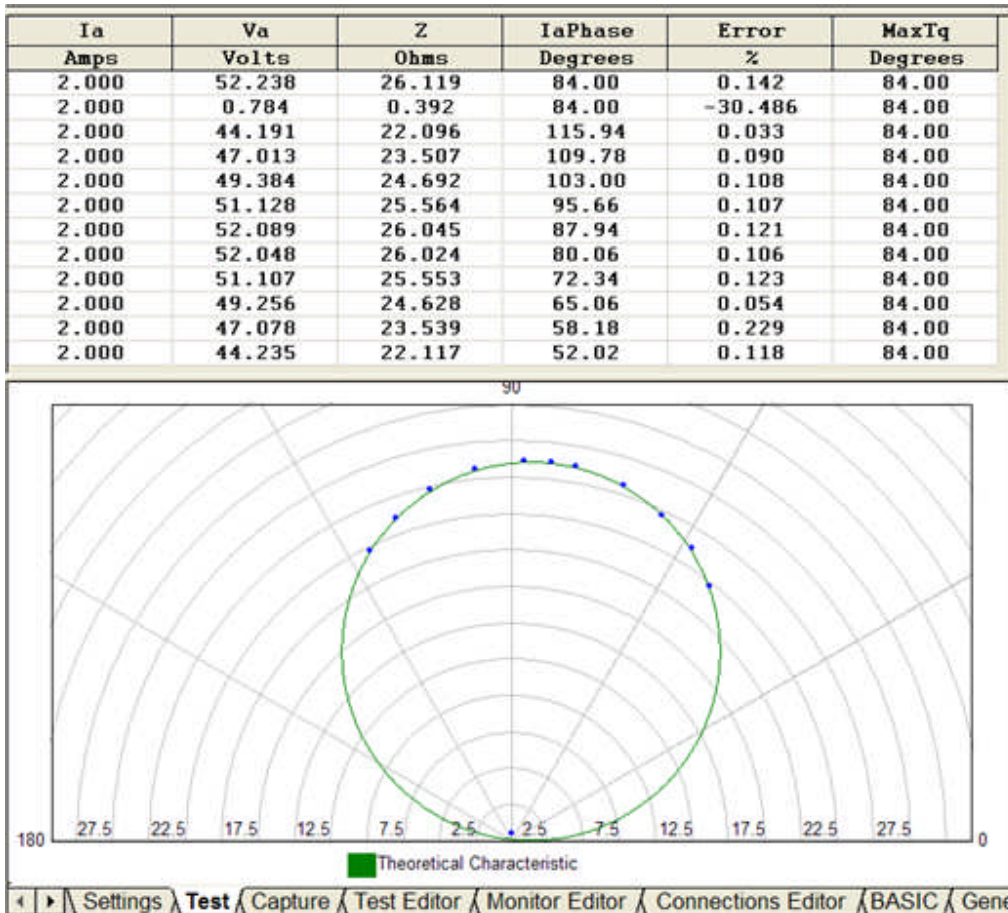


*Log Control* Variable List contains the labels of those recognized by the values of used by the Binary Search Control. The Error and MTA variables and Pre-Log Calculations were added for additional information in the Test Results.

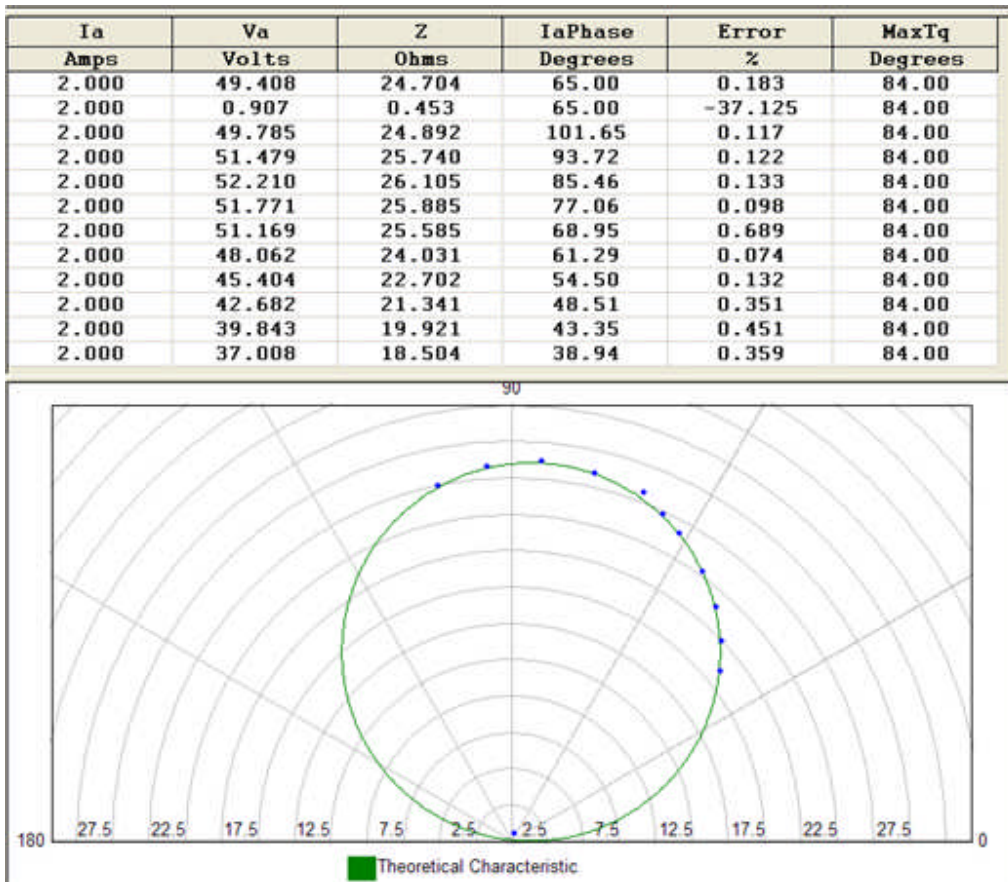


End Control will terminate the test and turn off all active amplifiers.

The *Test Results* for the relay Z4 characteristic using the values above demonstrate the Binary Search Control.

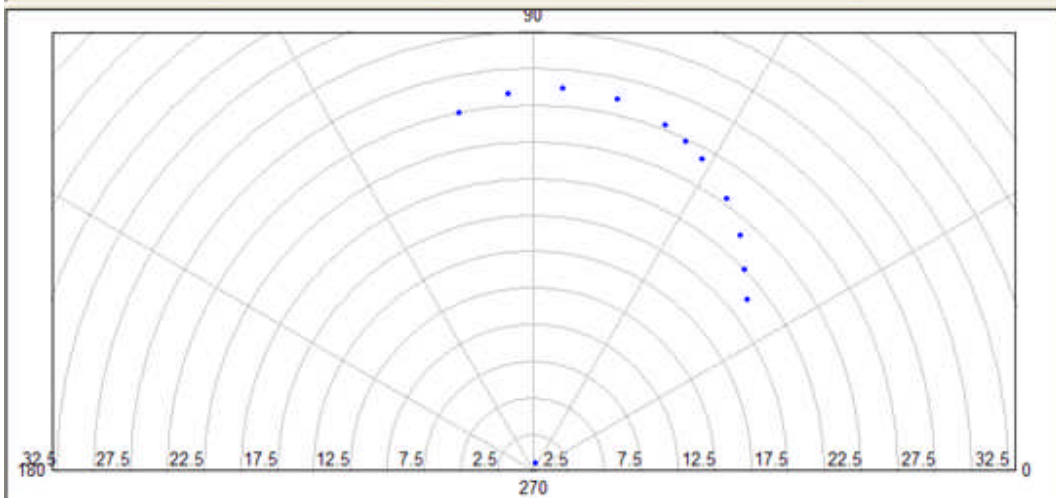


Example 2: The Binary Search Control MTA was changed to 65 Degrees and the *Test Results* demonstrate that the test can run for several relays without knowing the actual characteristic.



Example 3: *Test Results* with relay MTA = 84, Binary Search MTA = 65 and no Theoretical Control (same as Example 2).

Ia	Va	Z	IaPhase	MaxTq
Amps	Volts	Ohms	Degrees	Degrees
2.000	49.344	24.672	65.00	84.00
2.000	0.868	0.434	65.00	84.00
2.000	49.813	24.907	101.69	84.00
2.000	51.505	25.753	93.76	84.00
2.000	52.146	26.073	85.46	84.00
2.000	51.813	25.907	77.08	84.00
2.000	50.351	25.176	68.90	84.00
2.000	48.227	24.113	61.27	84.00
2.000	45.373	22.686	54.49	84.00
2.000	42.648	21.324	48.50	84.00
2.000	39.787	19.893	43.35	84.00
2.000	37.309	18.654	38.52	84.00

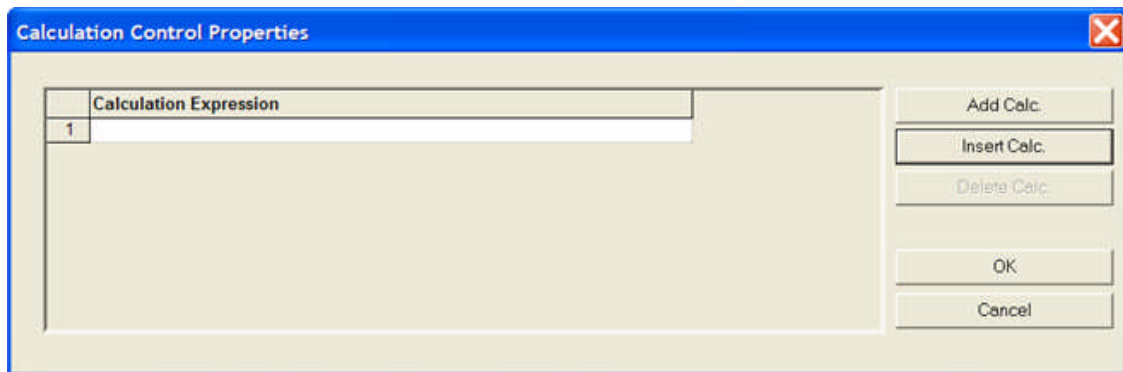


## Calculation Control

The **Calculation Control** may be use to condition variables, create variables, and condition Theoretical objects. An example of this Control is demonstrated in Test Editor OC Time Example and Test Editor Charac Z Example.

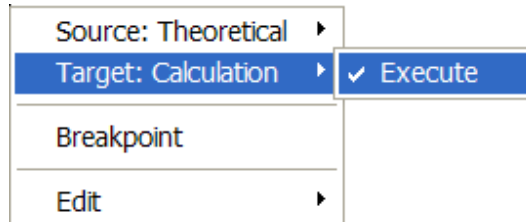


*Calculation Control Properties* allow entry of special set parameters for issue to a test set or for providing calculations using formulas. Entries may be entered for determining graph data, changing variable labels, creating new variables, finding the value of a string, calculating a value from a formula, and making branch decisions during a test execution.



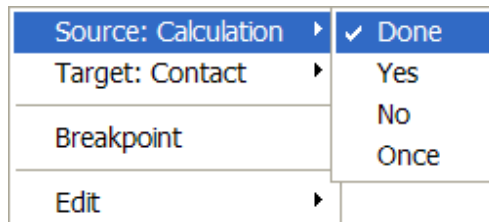
Calculations may also be implemented using most other controls that contain a properties field to do so.

*Target Links* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Links* are Done, Yes, No and Once.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

The Yes command is a branch option from the Control. If the Calculation condition is TRUE, then the branch option is Yes.

The No command is a branch option from the Control. If the Calculation condition is FALSE, then the branch option is No.

The Once command forces all calculations (equations) in the Control to equate one time only when the Control is first entered by a Link Target from another Control or from itself.

**Example:** Using the yes/no decision link. The Calculation Control will have two Links leaving (Source), each Targeting a different Control, one of which may be an End Control.

line 1: if la = 5 then path = 1 else path = 0, or this may have been written if la = 5 then path = yes else path = no.

line 2: path

Set the two output links to yes and no. When la = 5 it will take the yes path.

**Note 1:** The word "path" is required on the last line. The "else" statement is necessary for the If statement and should always be included when using a conditional (yes/no in this example) type statement.

**Note 2:** It is not mandatory to use the word "path"; you may use any variable you like such as "end", "NotFound" or "milkcow", etc. However, the if statement variables must match the variable on the last line. Avoid Key words like "Next" and "Continue" as this may cause undesirable behavior.

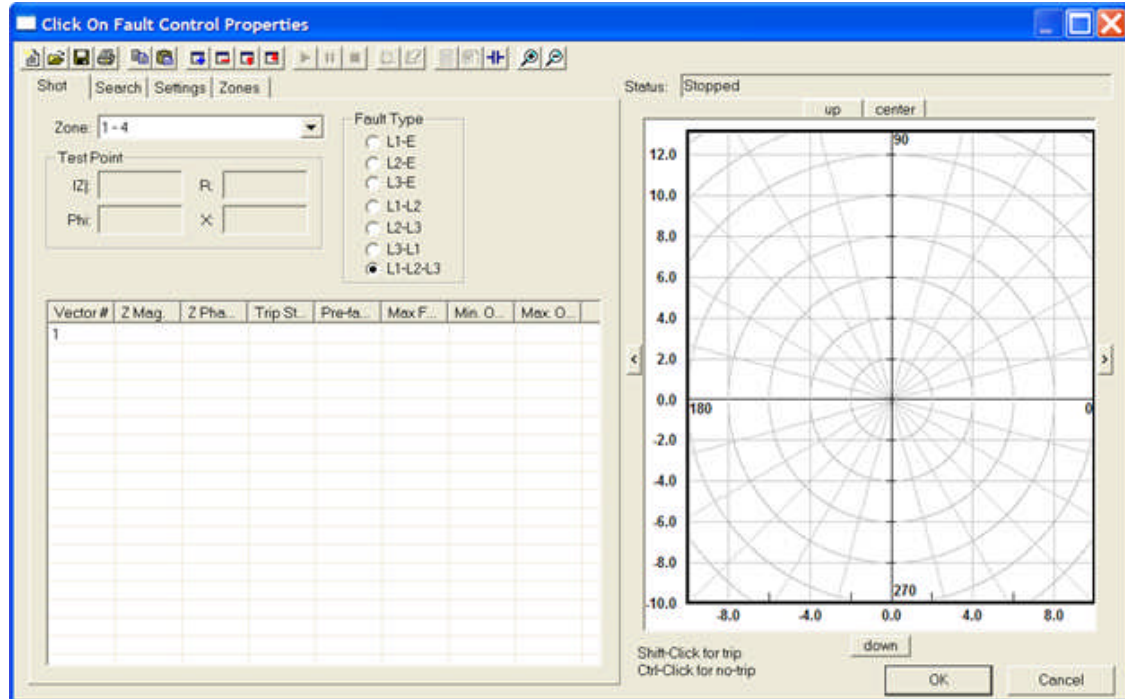
**Note 3:** Do not capitalize the "if" or any other two letter text beginning with "i". In an "if" statement, If (capital I) will be interpreted as CURRENT CHANNEL F.



## Click On Fault Control

## Click On Fault Control

The *Click On Fault Control Properties* is a useful tool and has several methods of use:



Embed into an existing impedance characteristic test procedure.

The Control may be created as a simple stand alone test procedure to be separate from all the other characteristic tests for a particular relay.

The Control may be a separate "relay" created to test various relays. Each file may be called for the relay to test and Results saved.

Create a new Relay with one test.

Create the Shots and/or Search tests for a Relay and save the file.

Create the Shots and/or Search tests for another Relay and save the file.

Create the Click On Fault Tests in a Subtest and save the Subtest for call in the different Relay tests the Subtest Control is placed.

The Control Properties are similar to the Online Click On Fault Control.

When first opened, the Shot Tab is the default tab.

Graphic displays of the Zones of protection including reach, phase angle, offset and other zone parameters are produced by an easy left mouse click and drag. Refinement of each is accomplished in the four tabs at the top of the Control Properties.

The Online Control has a main CoF Menu Toolbar for entries, execution and control of the Online Control.

There are four screens to select for setup and control of the Zones, Tests for each Zone, and Fault Types.

An Impedance characteristic created using the Theoretical Control in a Test Editor procedure will automatically be displayed in the Graph Pane.

Using the same import tool, multiple characteristics that exist in RIO file formats may also be called and used in the Graph Pane for multiple tests.

Test Vectors created in the Graph may be saved with an identifying name to the AVTS default folder Click On Fault Tests.

The file may then be recalled at any time and executed for test results. Therefore, the Test Editor test procedure may be added to any relay for tests specific to that relay.

In addition, the file may be called by placing the Click On Fault Control in a new or existing Relay Test Editor tab and saved with other tests for a particular AVTS Relay Test Module.

**Note:** Data created in the Online Click On Fault Control and saved to the AVTS/Click On Fault Tests folder may be called by the Click On Fault Control in a Relay Test Editor Tab.

Topics defining the Click on Fault Control in the Test Editor tab are:

- Click On Fault Control Properties
- Click On Fault Shot Tab
- Click On Fault Search Tab
- Click On Fault Settings Tab
- Click On Fault Zones Tab
- Click On Fault Control Examples

## Click On Fault Properties

The **Click On Fault Control (CoF)** provides a quick approach to testing an impedance relay. Tabs in the Control Properties dialog allow creation of a set of tests in the selected tabs for call when used in a Test Editor procedure. Tests may be included in the Workspace Test tab for a Relay which may include one or more CoF applications.

A test file may be created using the Online Click On Fault Control and saved to a file. The file can be Opened in the Click On Fault Control in a new or existing Relay Test Editor tab and saved with other tests for a particular AVTS Relay Test Module.

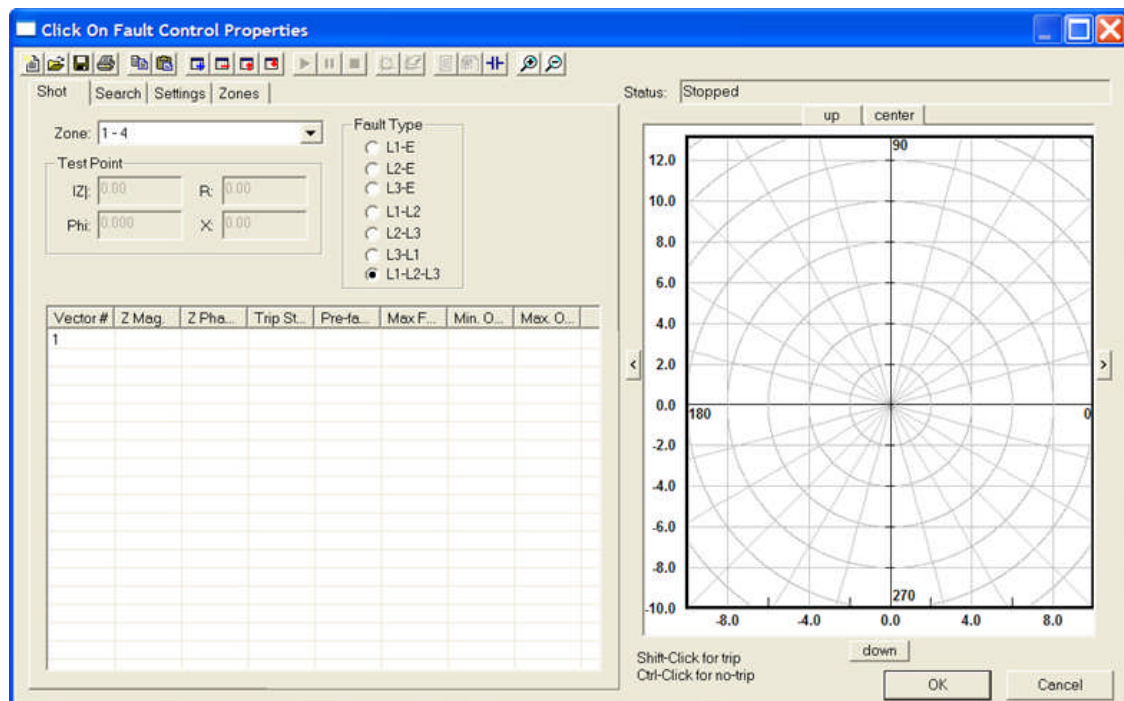
**Note:** Data created in the Online Click On Fault Control and saved to the AVTS/Click On Fault Tests folder may be called by the Click On Fault Control in a Relay Test Editor Tab.

A typical test may include a Start Control, 1-Touch Control to set relay output status, CoF Control, Log Control and End Control. When set to use the Shot or Search Tabs, the series of Vectors or Lines applied may take the place of several other tests that test a discrete relay element.

An example of the use of the Control is in the Click On Fault Control Example topic.



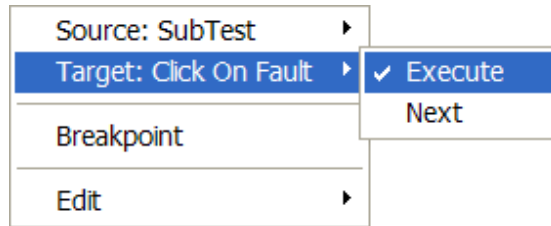
### Click On Fault Control Properties



Definitions for the use of the Control are defined in the Online Click On Fault Control (CoF) topic.

**Note:** A Log Control must be included after the CoF Control to log out the results as detailed by use of the Define Log Control tool in the CoF Control Properties. The Log Control in the test procedure should be void of any Pre-log Calculations and Variables. The results data Variables defined in the CoF Control will create the Variables results in the Test Tab.

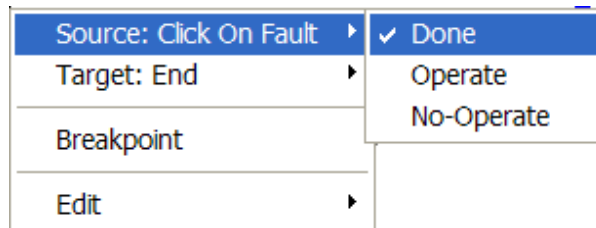
*Target Links* are Execute and Next.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Next command initiates the next item of List or Loop Values in the properties box.

*Source Links* are Done, Operate and No-Operate.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

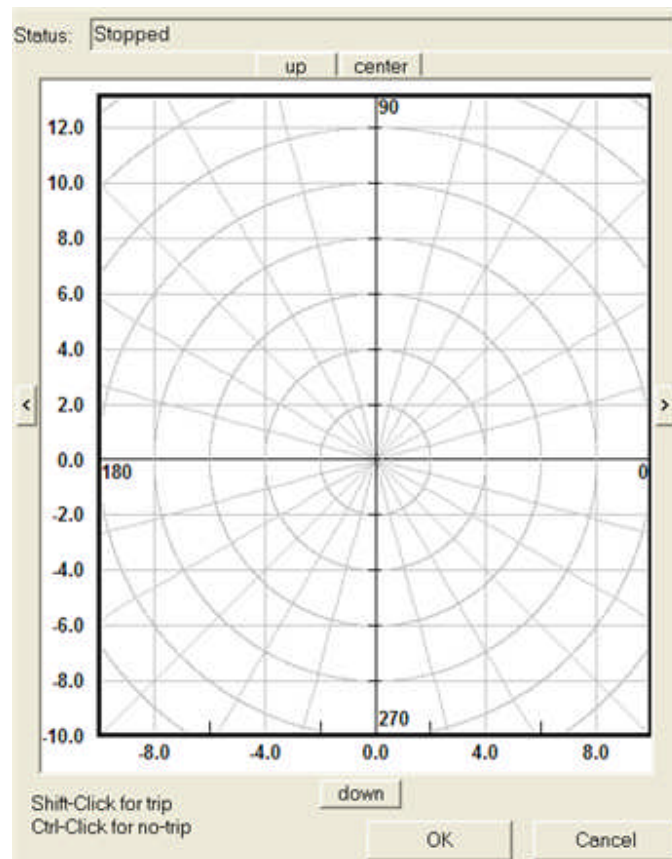
The Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

The No-Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

**Note:** The non operation of a required response and the Control properties having reached the preset limits validates this branch and indicates an improper condition in which to abort the procedure.

## Click On Fault Graph

The **Graph** on the right side of the Click On Fault Control Properties dialog has several elements for edit. The graph is visible for test edits in all tabs. The edits allowed are specific to each tab and dependent on the tab use.



**Test points** in the **Shot Tab** will be shown on the graph referenced by the values in the **Shot Tab**, **Settings Tab** and **Zones Tab**.

**Search Lines** will be shown on the graph referenced by the values in the **Search Tab**, **Settings Tab** and **Zones Tab**.

Note: The Graph will display the Shot Points or Search Lines in the Settings, Zones and Results tabs dependent on the last tab selected, the Shot or Search tab. The Results tab will only display recent test Results of an executed test; else, the Shot Points or Search Lines will show.

The **Status:** field will display the condition of the Run Click On Fault tool, such as

**Stopped** is displayed when the Control is first opened.

**Paused** is displayed when the *Pause Click on Fault Test* tool is selected. Select the *Run Click on Fault Test* tool to continue the tests.

**User Stopped Test** is displayed when the *Stop Click on Fault Test* tool is selected.

**Running** is displayed while tests are executing after the *Run Click on Fault Test* tool was selected.

**Finished Running** is displayed when the executed tests have completed and Results are available to view.

**Communication Error** displayed when the Control has lost connectivity to the MPRT or Pulsar test set.

The graph has left and right arrow buttons to move the center of the graph in the opposite direction.

The **right arrow button** is on the right of the graph and approximately mid vertical distance. Click on this button to move the view to the right, the polar center to the left.

The **left arrow button** is on the left of the graph and approximately mid vertical distance. Click on this button to move the view to the left, the polar center to the right.

The **down button** is on the bottom of the graph and approximately mid horizontal distance. Click on this button to move the view down, the polar center up.

The **up button** is on the top of the graph and approximately mid horizontal distance. Click on this button to move the view up, the polar center down.

The **center button** is on the top of the graph and adjacent to the Up button. Click on this button to move the polar center to the approximate mid vertical, mid horizontal location in the graph..

The **Battery Simulator** button is not displayed in the Click On Fault Control. It utilized by using the Battery Simulator Control in the AVTS Test Editor tab or by the Tools Menu item.

The **OK** button retains any edits made in any of the tabs and exits the Click On Fault Control.

The **Cancel** button exits the Click On Fault Control without retaining any edits made in any of the tabs.

To enter test points in the graph for "Shot" tests use the directives given at the lower left to the graph.

**Shift-Click for center** to enter a test point where, in or out of the characteristic, a trip is expected.

Trip points are entered as Red Triangles and will change to a square when the point is executed. A Green Square after the test indicates a successful test. A Red Square after the test indicates an unsuccessful test.


**Ctrl-Click-Click for search line** to enter a test point where, in or out of the characteristic, a trip is not expected. No-trip points are entered as Blue Circles and will change to a square when the point is executed. A Green Square after the test indicates a successful test. A Red Square after the test indicates an unsuccessful test.

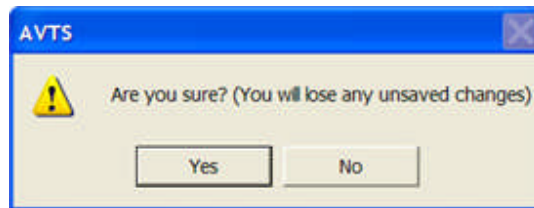
## Click On Fault Toolbar


The Menu Toolbar includes an icon for using and editing the six tabbed screens in the Click On Fault Control Properties.

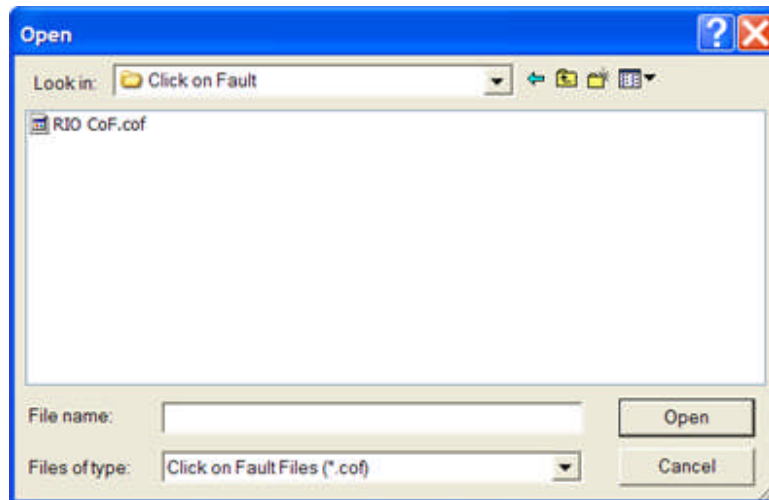


The tool icons are explained from left to right.


**New Click On Fault Test**  icon is for to create a new set of tests. Selection evokes the *Clear Existing Test* message dialog.

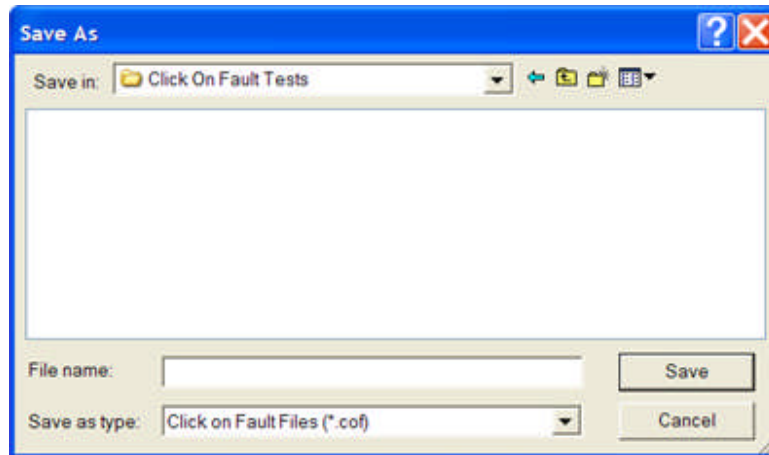


**Open Click On Fault Test**  icon is to open a previously created and saved set of tests. Selection evokes the *Open* dialog. In this example, a subfolder to AVTS has been created and labeled Click On Fault Tests.




Note: The Click On Fault Files will have an extension \*.cof. Only file with this extension may be opened for use in the Online Click On Fault Control.


**Save Click On Fault Test**  icon is to save a created set of tests in the C:\Program Files\Megger\AVTSv2\Click On Fault Tests folder. Selection evokes the *Save As* dialog. By default the dialog opens to the AVTS folder and in this example, the Click On Fault Tests folder has been selected for choosing the desired tests file.





**Note:** The execution results in the Results Tab are not saved with any constructed and executed tests. Results to be kept must be first saved with the **Print Click On Fault Test Results** or the **Export Results to Word** tools.


**Print Click On Fault Test Results**  icon is to directly print the test results in the active file. Selection first opens the computer default word processor software, typically Microsoft Word. Then the default Windows print dialog is evoked for printer settings and command print.


**Note:** If the test results had not been previously saved, they may be saved in the Word document as a \*.doc for later recall to print.


**Copy Test Points**  is to copy a single test point or multiple test points (hold Ctrl key while selecting) and retain for paste to another location within Click On Fault.


**Paste Test Points**  is to place in a new location the test or test points previously retained by use of the Copy control.

**Add New Test Points**  creates a new test number in the Shot tab Vector list, the Search Line list and the edit fields for the Zone Number and Zone Label in the Zone tab. A new test is added in numeric order each time the Tool icon is selected. The Tool is active in Shot, Search and Zones tabs only


**Delete Selected Test Points**  deletes the last test number in the Shot tab Vector list, the Search Line list and the edit fields for the Zone Number and Zone Label in the Zone tab. The next test in reverse numeric order is deleted each time the Tool icon is selected. The Tool is active in Shot, Search and Zones tabs only

**Move Test Point Up In Test**  moves a selected test in the list up one row with each click of the Tool icon.

**Move Test Point Down In Test**  moves a selected test in the list down one row with each click of the Tool icon. The Test Points can not be moved to the last position number.

**Run Click On Fault Test**  is an active test execution tool in the Shot and Search tabs only.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. Test execution is performed with the Test Execute tool in the AVTS Relay test module.

**Pause Click On Fault Test**  is active after a test has been executed by the Click On Fault Test Tool icon. It is active in Shot and Search tabs only.


Selecting this tool prior to test execution will evoke the Contact Control Properties to select the available options for test reaction recognition.

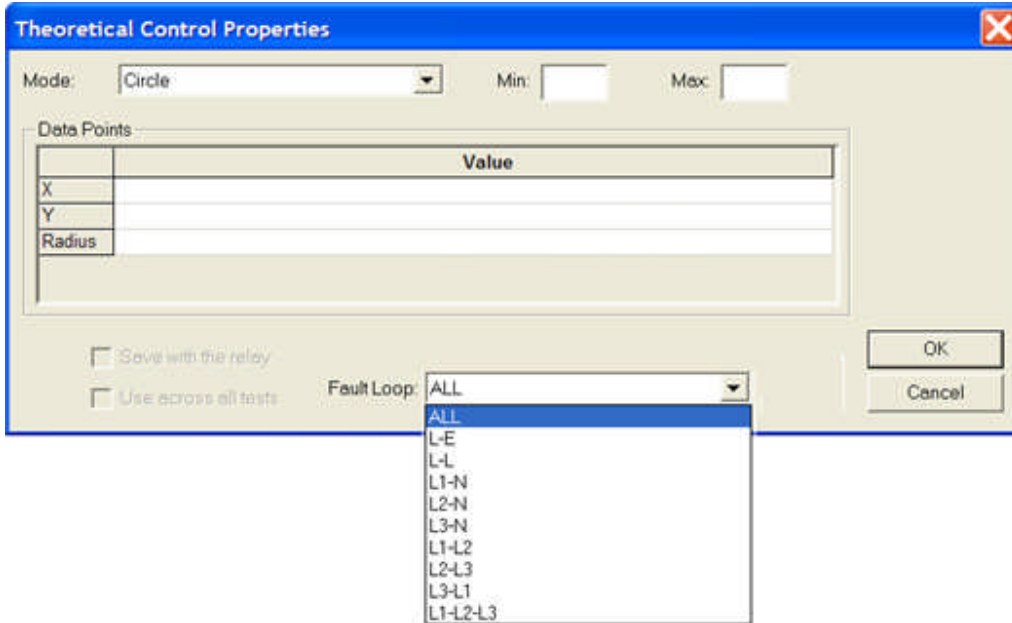
**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. Test Pause is performed with the Test Pause tool in the AVTS Relay test module..



**Stop Click On Fault Test**  is active after a test has been executed by the Click On Fault Test Tool icon. It is active in Shot and Search tabs only.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. Test Stop is performed with the Test Stop tool in the AVTS Relay test module.

**Define Theoretical for Test**  opens the *Theoretical Control Properties* dialog for defining the impedance graph data for plotting on the graph. The data may also be used for accuracy comparison to the recorded results. The comparison is governed by the data provided in the Define Log Control.




**Note:** This option allows import of RIO files directly to the Online Click on Fault Control.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. A Theoretical is created with the Theoretical Control tool in the AVTS Test Editor tab.

**Clear Result Points on Graph**  deletes all previous results from the graph. The action is true for results just accumulated or those from a previously saved and opened test file with results. The test points created remain with their values in the Vector# or Line# tables.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.


**Define Log Control**  opens the Log Control Properties dialog. The details for variables and calculations are placed in this dialog for control of the logged results. The Variables listed in the Log Control will be placed in columns in the CoF Results Screen and the test results listed after tests execution. The appearance is similar then to a test result printed in a usual AVTS test with results governed by the Log control.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. Edits to the Results Log is performed in the Log Control in the AVTS Test Editor tab.

**Export Results to Word**  makes the Results tab active and opens a Word document. The accumulated test results present in the Results tab are placed in the document along with the graph complementary to the results. The Graphic Impedance with test lines, test points and test results is included in the document.

**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab. Export to Word is performed by selecting the Test Result in the Relay results list in the Workspace Relay tab and selected with a right click.

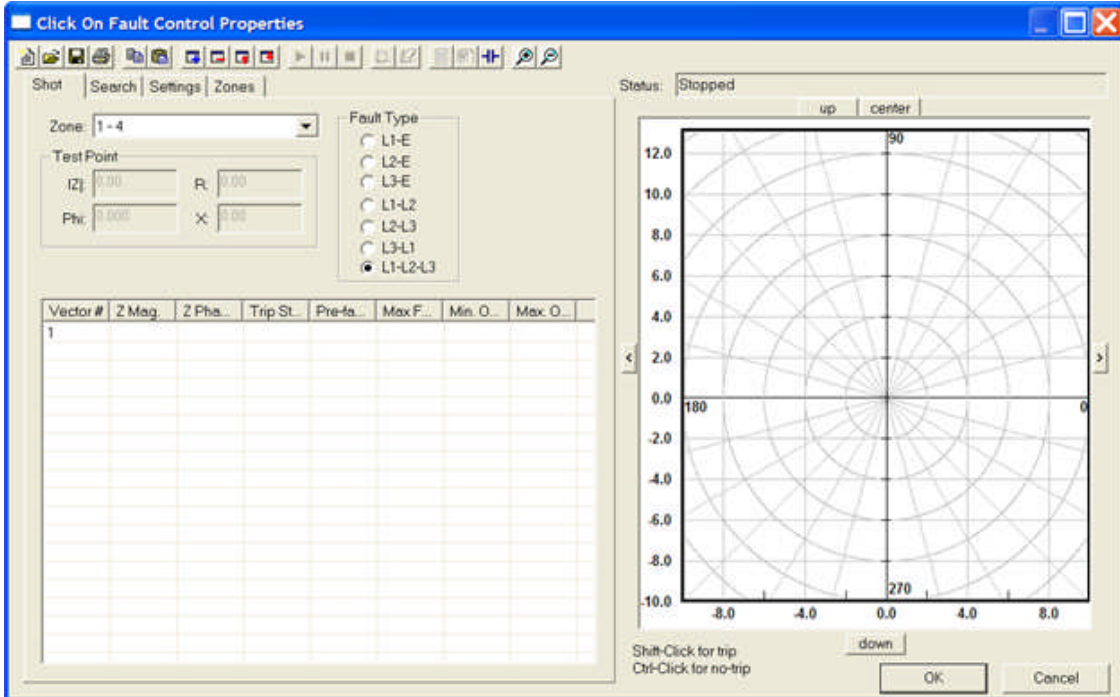
**Define Contact Control**  opens the Contact Control Properties dialog. The options to select test set input contacts for monitoring the active test. The selections allow for biasing a single contact or for setting multiple contacts for a wait any or wait all condition. The options are defined in detail in [Contact Control Properties](#) dialog topic.

**Zoom In**  applies to the Graph display only. The graph zooms in with each mouse click on the icon in 1/2 view increments. The Zoom In feature may be monitored by watching the graph horizontal and vertical scales. The operation is valid in all tabs.

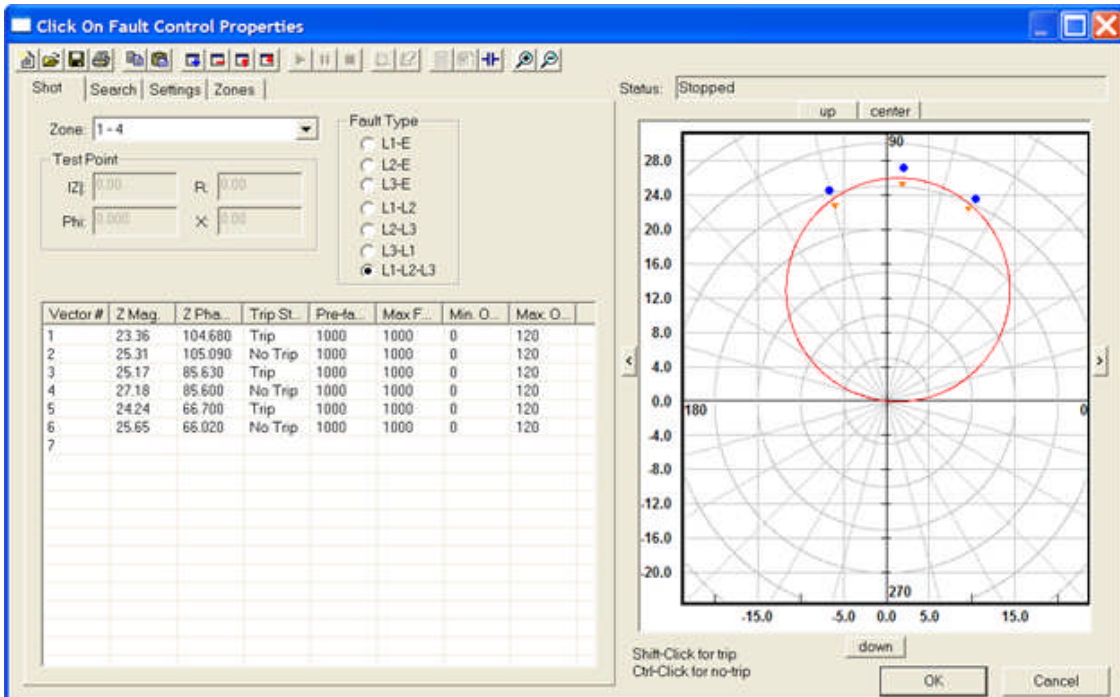
**Zoom Out**  applies to the Graph display only. The graph zooms in with each mouse click on the icon in double view increments. The Zoom Out feature may be monitored by watching the graph horizontal and vertical scales. The operation is valid in all tabs.

## Click On Fault Shot Tab

The *Shot Tab* is to create one or more points of test, each to replicate a fault at a particular magnitude and angle.



Several Trip and/or No-Trip points may be selected for each Fault Type.



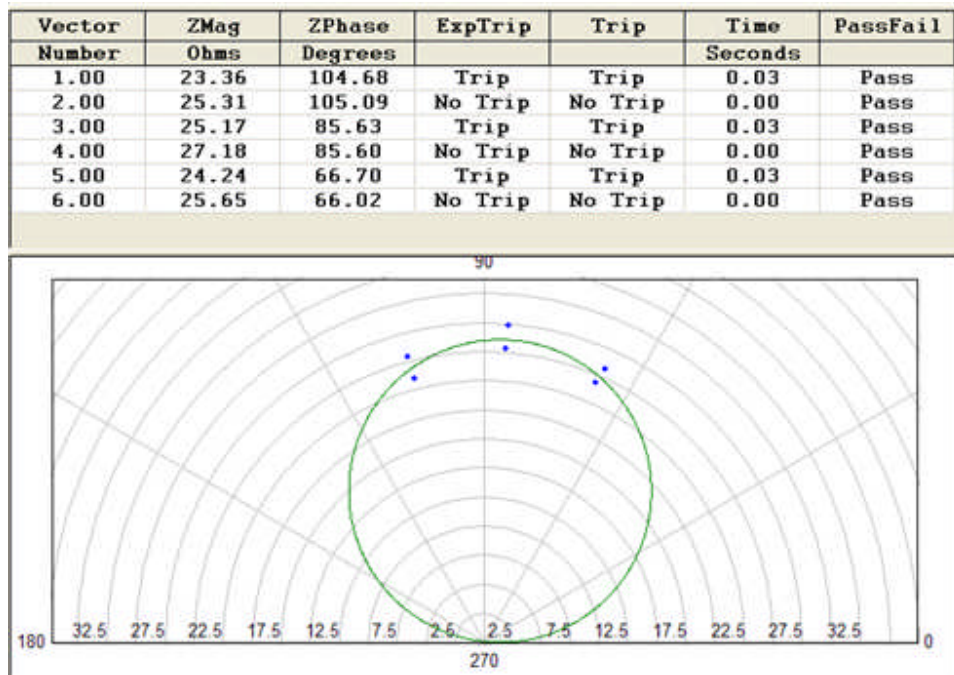
The executed test activity and Results will appear in the Graph pane of the AVTS Test Tab.

When executed, all **Trip** and **No-Trip** points for the selected Fault Type will run in turn beginning with Vector# 1 and continuing to the last Vector# in the list.

Select another **Fault Type** and execute. All created Trip and/or No-Trip points will operate in turn.

Note: An entered Trip point is a **Red triangle**. A valid Trip point will turn to a Green square, an invalid Trip point will turn to a Red Square. An entered No-Trip point is a **Blue circle**. A valid No-Trip point will turn to Green Square; an invalid No-Trip point will turn to a Red square.

*Shot Test Results* are accumulated at each of the created test points and are displayed in the AVTS Test Tab. .



Entries created in other tabs for reference

**Zone** drop list for selection of the characteristic data from the Zones tab

The **Test Point** is the set of values listed both in Mag and Phase Angle, and in Cartesian values that are created in the Graph or used to edit a selected Vector# edit fields

The **Fault Type** List check bullet is to select the fault type to apply to the selected test point.

- L1-E** - applies a single phase fault for the selected test point
- L2-E** - applies a single phase fault for the selected test point
- L3-E** - applies a single phase fault for the selected test point
- L1-L2** - applies a phase-to-phase fault for the selected test point
- L2-L3** - applies a phase-to-phase fault for the selected test point
- L3-L1** - applies a phase-to-phase fault for the selected test point
- L1-L2-L3** - applies a 3 phase fault for the selected test point

The **Graph** pane, when clicked, evokes the *Hold Shift or CTRL keys* to select Trip or No-Trip points message.

A created **Trip** point will show as a **Red triangle** in the graph. When executed, the result of the test on the point(s) created will change to **Green square**. The result will change to a **Red square** if a **No-Trip** occurs for an expected **Trip**.

A created **No-Trip** point will show as a **Blue circle** in the graph. When executed, the result of the test on the point(s) created will change to **Green square**. The result will change to a **Red square** if a **Trip** occurs for an expected **No-Trip**.

Each **Trip** or **No-Trip** point created will be placed in the Vector# list with all the associated parameters of the point.

Continued clicks will produce additional test points at the mouse locations. The graph may contain numerous Trip and No-Trip points for relay response and setting verification.

*Shot Tab Test Point parameters* may be entered by direct clicks on the graph or directly in the edit fields using the keyboard

Vector#	Z Mag	Z Phase	Trip State	Pre-fault ms	Max Fault Time	Min. Oper Time	Max. Oper Time
1	27.25	84.765	No Trip	1100	2000	58	62
2	25.16	85.188	Trip	1100	2000	58	62
3							

**Vector#** is the number of each created Trip or No-Trip test point - This field may NOT be highlighted and edited

**Z Mag** is the ohmic distance of the test point - This field may be highlighted and edited

**Z Phase** is the phase angle of the test point - This field may be highlighted and edited

**Trip State** is a drop list for expected **Trip** or **No -Trip** response to the test point - The Trip State may be changed and the corresponding test point in the Graph will change character also

**Pre-fault ms** is the apply time of normal voltages and currents to the test that have be created in the Settings tab - This field may be highlighted and edited

**Max Fault Time** is the maximum time value in milliseconds that the fault values will be applied before test abort due to a no-operate response that has be created in the Settings tab - This field may be highlighted and edited

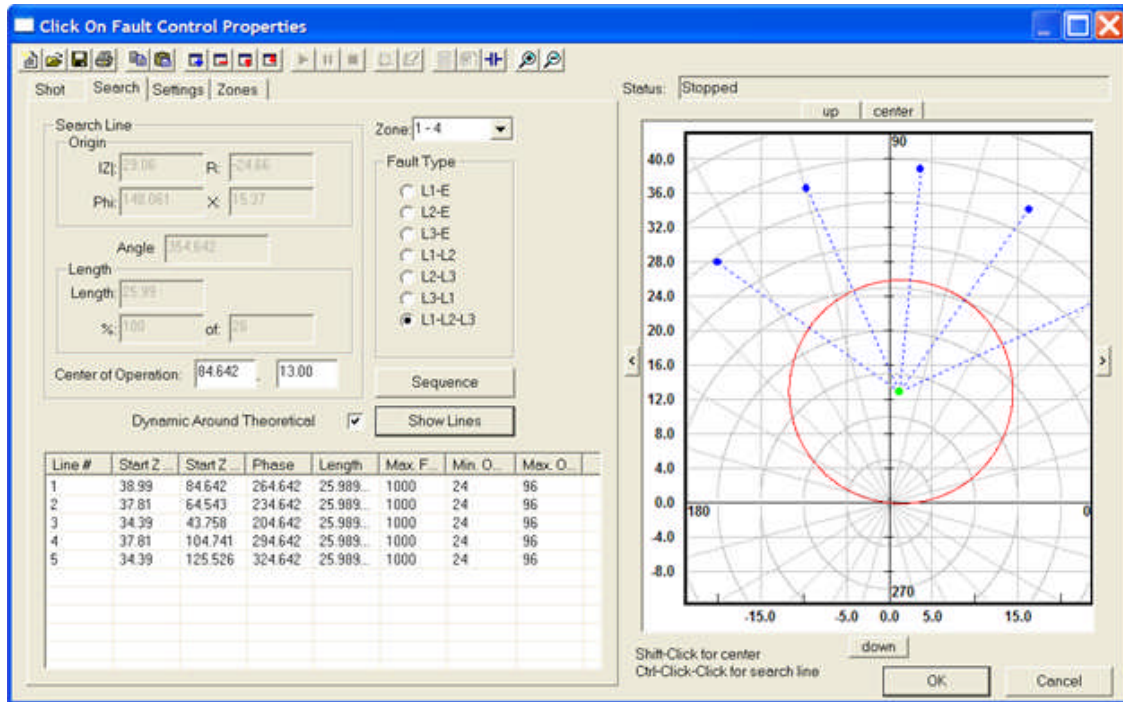
**Min. Operate Time** is the minimum time tolerance allowed for the Zone selected that has be created in the Zones tab - This field may be highlighted and edited

**Max. Operate Time** is the maximum time tolerance allowed for the Zone selected that has be created in the Zones tab - This field may be highlighted and edited

The **Graph** pane, when clicked, evokes the **Hold Shift or CTRL keys** to create a search line message.

## Click On Fault Search Tab

The *Search Screen* is to create a series of tests which ramp from a beginning point along a search line to an end point in that direction.



To create a test line, hold the Ctrl key and left click at a start point in the graph. Continue holding the Ctrl key and drag the mouse to the final test point and left click to anchor the test line. The created start and end points and the test line will be drawn in red, but will turn blue for a saved line with a second mouse click prior to release of the Ctrl key.

A created test will automatically enter the data in the Search Line block of edit fields. Data displayed in these fields are dependent to the Line# (Test line) selected and may be edited to adjust or refine the Line#.

The **Run Click On Fault Test** tool is enabled for this Tab. The executed test will first apply test to the test line point farthest from the graph polar origin for a Trip or No Trip. The next application will be applied to the point closest to the graph polar origin for a Trip or No Trip. Each successive test point will oscillate one-half the distance between a Trip and a No Trip point until a Trip or No Trip occurs within either the Relative (%) or Absolute (Ohms) Impedance Tolerance setting in the Zones tab.

**Note:** The **Run Click On Fault Test** tool is not enabled in the **Click On Fault Control** embedded in a Relay Test Editor tab. The execute for an embedded test is the **Tool bar Execute Test** tool.

The data area contains a Search Line block, Zone: drop list, Fault Type list, Dynamic Around Theoretical check box and the list of created test vectors (Line#s)

### Search Line block

**Origin** block has data fields for

**[Z]:** the magnitude of impedance point first clicked to create a search line

**Phi:** the phase angle associated with the Z magnitude field is the angular displacement of the [Z] selected point (firsts point clicked)

**R:** the Cartesian resistive equivalent of the impedance [Z] and its angle Phi

**X:** the Cartesian reactive equivalent of the impedance [Z] and its angle Phi

**Angle** field displays the angle of the line relative to the phase angle of the search line with the start point is positioned on the graph polar origin

Note: When drawn, the fields display the values for each search line, but gray to indicate a non-active field. The fields are made active for edit when the **Line#** is selected.

**Length** block

**Length:** is the physical ohmic length defined by the two ends of the selected search line.

**%:** entry describes the percent of the system parameter "line length" to be the test limits.

**of:** entry describes the "length" in ohms of the constructed search line.

**Center of Operation** fields

**Center of Operation:** the first field is for the Phase Angle value in degrees

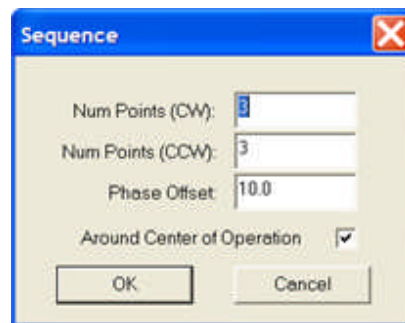
**Center of Operation:** the second field is for the Magnitude of the impedance value autocreated by the Show Lines button

Note: The impedance is automatically created by the Phase Angle entry and the Dynamic Around Theoretical checked box. Entry of a phase angle other than the Zone Max Torque Angle will shift all test entries to the new angle entry when Show Lines button is clicked. The Magnitude will also be changed to 1/2 of the center test Line# intercept to the plotted characteristic.

**Dynamic Around Theoretical** check box

Note: Use of this option will replace any test lines previously constructed and will not be retrievable unless they were saved to a file name before the Show Lines button is selected.

**Sequence** button opens the *Sequence dialog*



**Num Points (CW):** edit field is to enter the number of test points that describe as many test lines clockwise from the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Num Points (CCW):** edit field is to enter the number of test points that describe as many test lines counterclockwise from the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Phase Offset:** edit field defines the angular displacement for each test line from that defined by the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Around Center of Operation** check box will automatically construct the test lines about a constructed Center of Operation either by entering values in the edit fields or by construction directly on the graph.

Note: The middle line of the set will be considered the Maximum Torque Angle and will align with the graph polar origin. The line length will be double the ohmic distance from the Center of Operation point to the impedance characteristic along the line extended from the origin. The maximum torque angle of the impedance characteristic need no be know; the test lines will be determined by the Center of Operation and the details of the Sequence dialog options.

**OK** or **Cancel** buttons to either accept the settings or return to the Search tab without saving the Sequence dialog edits

**Show Lines** button displays all the test Lines created by the checked Dynamic Around Theoretical box and the edits made in the Sequence dialog. Line #1 will be the Center test Line. The succeeding test lines will be in order of the angles selected in the Sequence dialog.

Note: Additional Test Lines may be added to the Dynamic Around Theoretical option by using the Ctrl-Click-Click option in the Graph.

**Zone** drop list is to select the Zone to place the search lines for test

**Fault Type** block lists the check buttons for the types of faults to execute for the selected Zone and Search Line

**Line List Data** columns are defined below:

**Line#** list contains all data associated with a constructed test line. All fields may be edited except for the Line# column. A constructed test line is displayed **Blue** but will be displayed **Green** when the Line# row is selected. The Line# test result will be displayed as a **Red** square on the test line.

**Start Z Mag.** is the magnitude Line test start point displayed in the **[Z]** edit field. Edit in one field will be reflected in the other field.

**Start Z Phase.** is the phase angle of the Line test start point displayed in the **Phi** edit field. Edit in one field will be reflected in the other field.

**Phase** is the phase direction (**Angle**) of the test line from the start point to the end point with the start point referenced to the graph polar origin. Edit in one field will be reflected in the other field.

#### **Length**

**Max. Fault Time** is the Maximum Fault Time in milliseconds to be applied at each test location on the test line. The entry is defaulted by the Max Fault Time entry in the Settings tab.

**Min. Oper Time** field is the Minimum Operate Time in milliseconds of the relay. The entry is defaulted by the Expected Operation Time - the Absolute Operation Time Tolerance entries in the Zones tab.

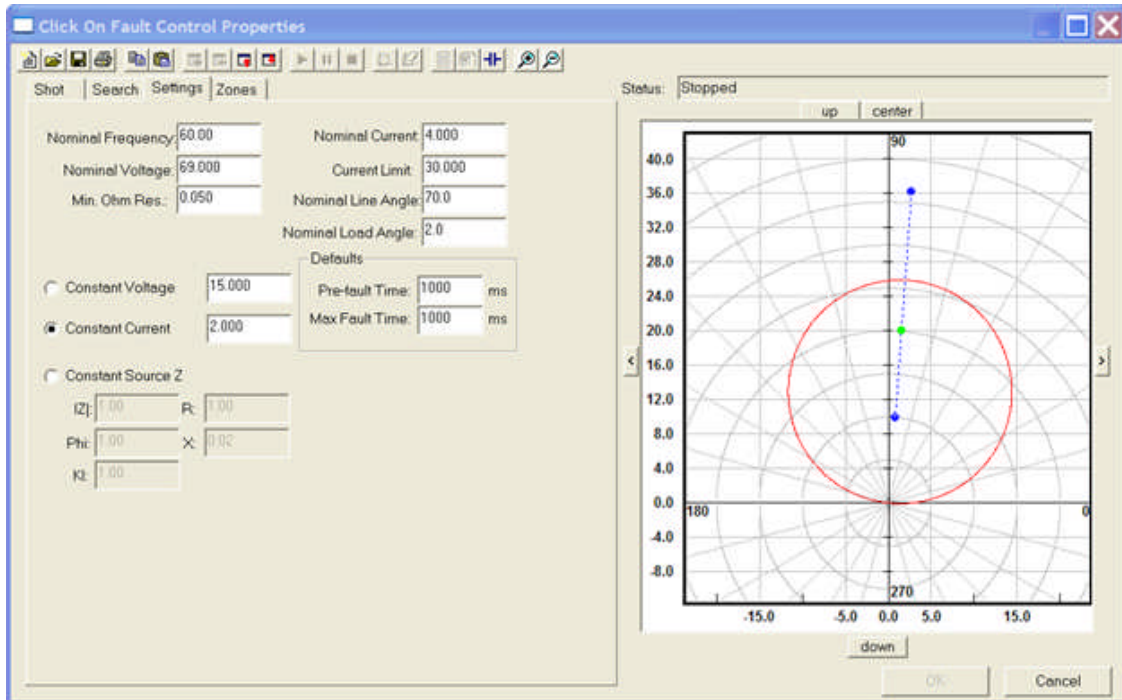
**Max. Oper Time** field is the Maximum Operate Time in milliseconds of the relay. The entry is defaulted by the Expected Operation Time + the Absolute Operation Time Tolerance entries in the Zones tab.

The **Graph** pane, when clicked, evokes the **Hold Shift or CTRL keys** to create a search line message.



## Click On Fault Settings Tab

The *Settings Tab* has edit fields to enter the desired normal system conditions, Default settings and test parameter fault constants.



Neither Shot Points nor Search Lines can be constructed in this tab.

The edit entry fields:

### Nominal Settings

**Nominal Frequency:** is to enter the value of normal system frequency in Hertz. Default value is 60.00.

**Nominal Current:** is to enter the value of normal load current in amperes. This value along with the Nominal Voltage setting to apply to the relay during "Pre-fault" to reset system logic. Default value is 4.000.

**Nominal Voltage:** is to enter the value of normal Line-to-Earth voltage. This value along with the Nominal Current setting to apply to the relay during "Pre-fault" to reset system logic. Default value is 69.000.

**Current Limit:** is to enter the maximum value of current in amperes applied by test set current channels for any test. Default value is 30.000.

**Min. Ohm Res.:** is to enter the value of the minimum resistance allowed in the calculations for all Fault Types and Shot Points or Search Lines. This value is used to determine the first value of test result within the accuracy set by the Impedance Tolerance value in the Zones Tab. Default value is 0.050.

**Nominal Line Angle:** is to enter the value of angle in degrees of the protected transmission line. Default value is 70.0.

**Nominal Load Angle:** is to enter the usual value of angle in degrees of the normal load on the line. Default value is 2.0.

**Constant Test Settings** check buttons allow selection of one only

**Constant Voltage** is to enter the value of volts to be held constant for all Fault Types tests under execution. Default value is 15.000.

**Constant Current** is to enter the value of amperes to be held constant for all Fault Types tests under execution. Default value is 8.000.

**Constant Source Z** is to enter the value of ohms of source to be held constant for all Fault Types tests under execution

**[Z]:** the magnitude of impedance point first clicked to create a search line

**Phi:** the phase angle associated with the Z magnitude field is the angular displacement of the **[Z]** selected point (firsts point clicked)

**kL:** kL is set equal system source grounding compensation factor **kS = 1/3 (ZS0 / ZS - 1)**

**R:** the Cartesian resistive equivalent of the impedance **[Z]** and its angle **Phi**

**X:** the Cartesian reactive equivalent of the impedance **[Z]** and its angle **Phi**

#### Defaults settings

**Pre-fault Time:** is to enter the number of milliseconds that **Nominal** values are applied prior to each Shot test and to each Search Line ramp test. Default value is 1000.

**Max Fault Time:** is to enter the maximum number of milliseconds that an executed value of fault current will be applied at which time the test will abort. Default value is 2000.

**Note:** The **Max Fault Time** is applied to each discrete ramp in a Search Line test

The **Graph** pane displays the test locations that are created in the Shot Tab or Search Tab; the tab last selected.

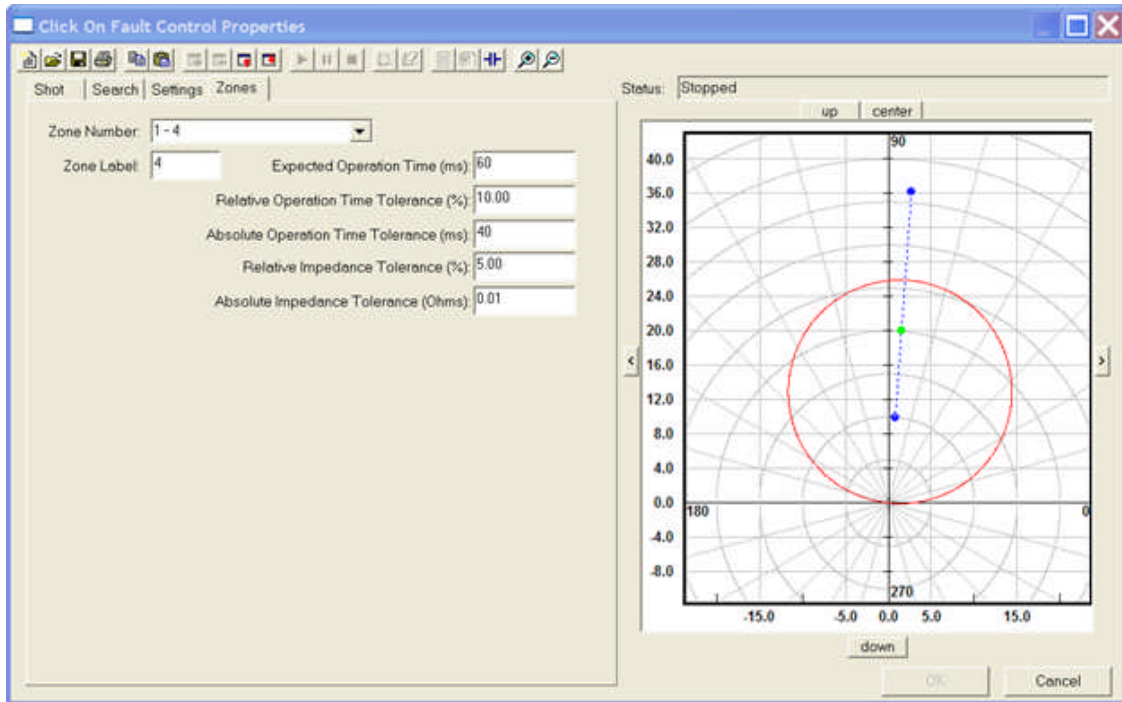
If the Shot Tab was last selected, the graph will display all the Trip and No-Trip Shot points.

If the Search Tab was last selected, the graph will display all the test Search Lines.

The Graph pane, when clicked in this tab, evokes the *Hold Shift or CTRL keys* to create a search line message.

## Click On Fault Zones Tab

The *Zones Screen* has edit fields to enter the desired test accuracy parameters and construct several characteristics for selected fault types.



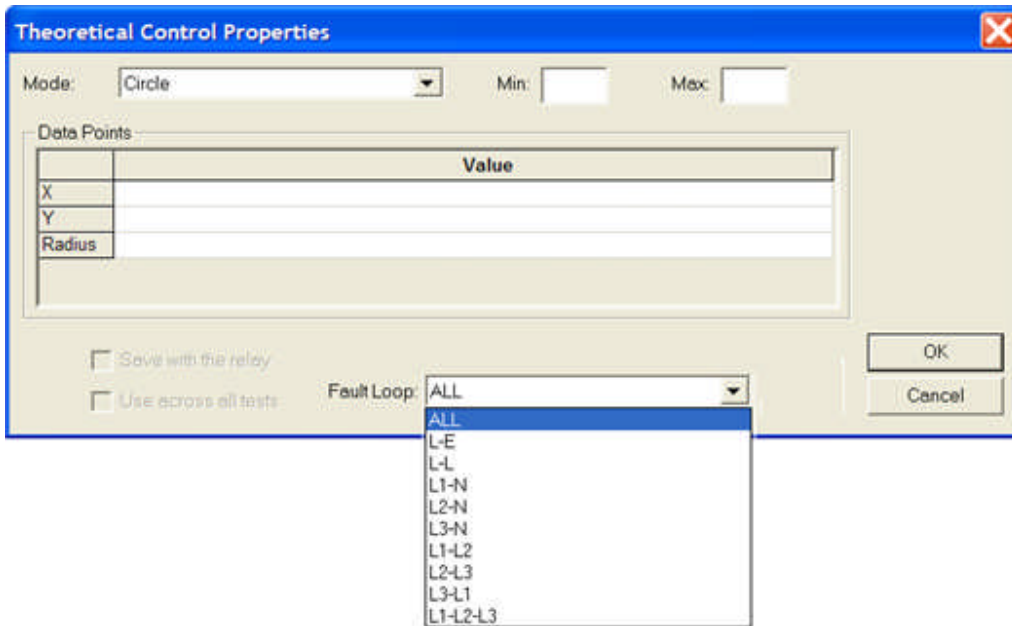
**Run Click On Fault Test** execute tool is not active in this tab.

Neither Shot Points nor Search Lines can be constructed in this tab.

Edit and Selection fields are:

**Zone Number:** drop list displays the Zone Labels created with companion constructed characteristics.

**Zone Label:** edit field allows edit of the Zone Label for the companion *Fault Loop* selected in the Theoretical Control properties dialog.



The Fault Loop selections are:

**ALL** will use the constructed characteristic for all Fault Types selected for test in the Shot and Search tabs.

**L-E** will use the constructed characteristic for all Fault Types involving Earth (ground) selected for test in the Shot and Search tabs.

**L-L** will use the constructed characteristic for all Fault Types involving phase pairs selected for test in the Shot and Search tabs.

**L1-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L2-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L3-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L1-L2** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L2-L3** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L3-L1** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L1-L2-L3** will use the constructed characteristic for the Fault Type involving all phases for test in the Shot and Search tabs.

**Expected Operation Time (ms):** edit field is to enter the expected time delay, if any, set for the selected Zone of the relay.

**Relative Operation Time Tolerance (%):** edit field is to enter the plus/minus percent tolerance acceptable as a result compared to the **Expected Operation Time.(ms):** minus/plus the **Absolute Operation Time Tolerance:**

**Absolute Operation Time Tolerance (ms):** edit field is to enter the plus/minus numeric tolerance acceptable as a result compared to the **Expected Operation Time.(ms):**

Example:

**Zones** tab settings are:

Expected Operation Time (ms): = 60

Relative Operation Time Tolerance (%): = 10  
 Absolute Operation Time Tolerance (ms): = 40

Then **Search** tab values are:

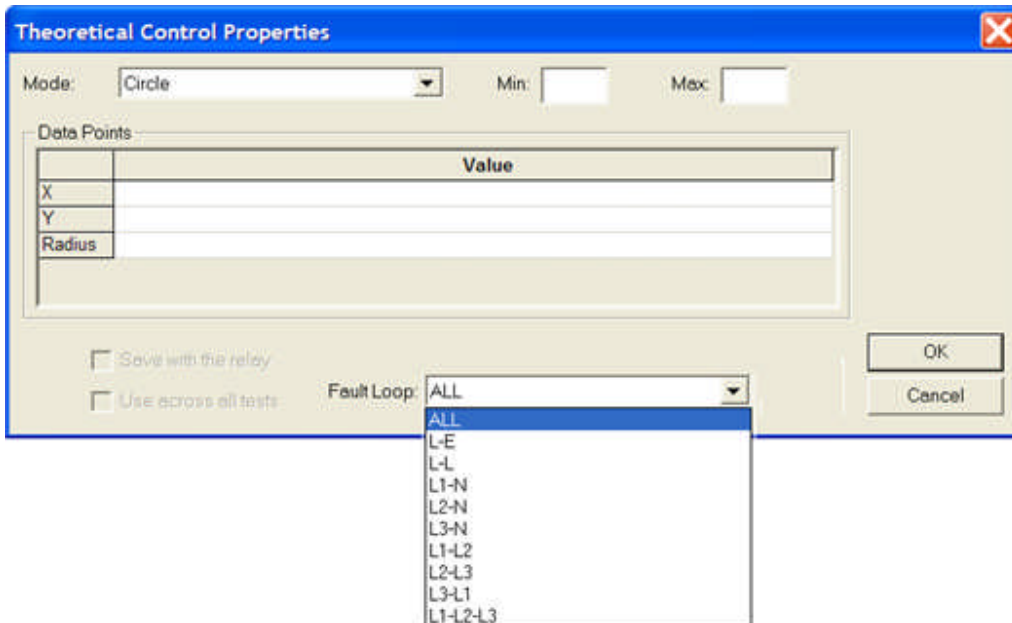
Min. Oper Time =  $60 - 40 - (10\% * 60) = 14$   
 Max. Oper Time =  $60 + 40 + (10\% * 60) = 106$

**Relative Impedance Tolerance (%):** edit field is to enter the plus/minus percent tolerance acceptable as a result compared to the Theoretical Impedance value on the search line. The Theoretical Impedance is determined by the construction of the relay characteristic using the Define Theoretical For Test tool or using the Add and Edit buttons in the Fault Loops block.

**Absolute Impedance Tolerance (Ohms):** edit field is to enter the plus/minus Ohmic tolerance acceptable as a result compared to the Theoretical Impedance value on the search line. The Theoretical Impedance is determined by the construction of the relay characteristic using the Define Theoretical For Test tool or using the Add and Edit buttons in the Fault Loops block.

**Fault Loops** block is to create and/or edit impedance characteristics for companion Zones.

**Add** button evokes the *Theoretical Control Properties* dialog to add a characteristic or to import RIO files. The Fault Loops selections are made in the Fault Loops: drop list in this Control. The selected Fault Loop: option will appear in the Zones tab Fault Loops block.



**Edit** is to evoke the Theoretical Control Properties dialog to edit a previously constructed characteristic. The Fault Loops selections drop list in the Control are dimmed and not available for edit.

**Delete** is to select a Fault Loop and delete the characteristic and its Label from the Fault Loops list.

The Graph pane, when clicked, evokes the *Hold Shift or CTRL keys* to create a search line message.

## Click On Fault Control Examples

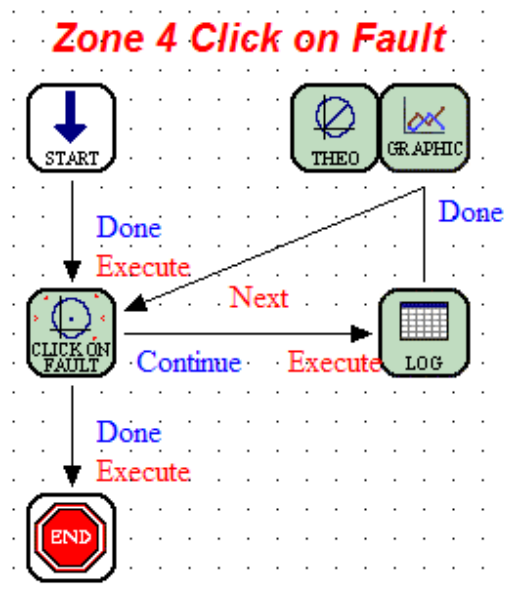
The **Click On Fault Control** may be used as a stand alone test for any relay with an impedance characteristic.

The following example demonstrate the use of the Click On Fault in an impedance test using the Shot and the Search parameters.

The minimum number of Controls required to perform the test and have results logged out are the **Start**, **Click On Fault**, **Log** and **End Controls**. However, selecting Shot test Trip and No-Trip points may produce a number of failed tests since a graphic characteristic is not present in the Click On Fault Shot and Search tabs. Therefore, **Theoretical** and **Graphic Controls** are added to the test procedure.

A Log Control will need to be added with the Click On Fault Control to capture the results generated by the execution of the tests constructed in the CoF Control. The Log Variables set in the CoF Control do not require user input as the necessary variables will automatically pass to the Log Control upon the first execution of the test.

The Example *Test Editor test procedure* Controls are:



**Start Control:** Assure that the number of Current and Voltage Phases agree with no more than in the test set and set in the Tools/Options.../Devices tab.

**Theoretical Control:** Create the impedance characteristic using the Mode drop list. If the characteristic can be described without referencing the Settings tab, then the Start Control Primary Settings Group need not be edited.

**Graphic Control:** Set the Scale Type to Polar and the Mode to Scatter. All Trip and No-Trip points, whether a Shot or Search test is conducted, will be displayed as a Blue Circle by default.

**Click On Fault Control:**

**Shot tab:** Create the Trip and No-Trip points described in the Click On Fault Shot Tab topic. Refined edits may be made in the Vector lists. Constructed Shot tests may coexist with constructed Search tests.

**Search tab:** Create the Search Vectors described in the Click On Fault Search Tab topic. Refined edits may be made in the Vector lists. Constructed Search tests may coexist with constructed Shot tests.

**Settings tab:** Edit the **Min. Ohm Res.**, the **Nominal** entries, the **Defaults** entries and the **Constant** parameter selection as necessary.

**Zones tab:** Edit all the Tolerance and Time fields as necessary to agree with the relay settings.

**OK** button: Ok is not enabled in the Settings and Zones tabs.

**Shot** tests will be conducted if the Shot tab is in view when OK is selected.

**Search** tests will be conducted if the Search tab is in view when OK is selected.

Log Control: Variables supported by the Click On Fault Control will be automatically entered to the *Log Control Variable* list.

Variables To Save					
	Variable	Units	Decimal Places	Include in Report	Align
1	CofVectorNum		2	<input checked="" type="checkbox"/>	Center
2	CofZMag		2	<input checked="" type="checkbox"/>	Center
3	CofZPhase		1	<input checked="" type="checkbox"/>	Center
4	CofExpTripStateStr\$		0	<input type="checkbox"/>	Center
5	CofActualTripStateStr\$		0	<input type="checkbox"/>	Center
6	CofTripTime		3	<input checked="" type="checkbox"/>	Center
7	CofPassFailStr\$		0	<input checked="" type="checkbox"/>	Center

The **Variables** for the Results log out are:

**CofVectorNum**: The Number of either the Shot or Search Vector in the respective tab.

**CofZMag**: The Impedance magnitude of the Shot or Search Vector.

**CofZPhase**: The impedance phase angle of the Shot or the start of the Search Vector.

**CofExpTripStateStr\$**: The desired Trip or No-Trip state for the Shot Vector.

**CofActualTripStateStr\$**: The desired Trip or No-Trip state for the Shot Vector.

**CofTripTime**: The actual trip time for the relay response.

**CofPassFailStr\$**: Pass or Fail for the relay response compared to the values set in the Settings and Zones tabs.

The default Variables are not complete in the available space for print out as the labels are all long.

The Variables may be edited in the *Log Pre-Log Calculations* list to create a more readable print out. Units were also added for clarity in the Results.

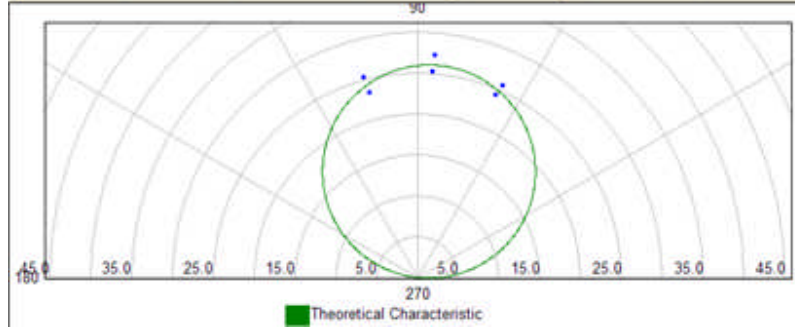
Variables To Save					
	Variable	Units	Decimal Places	Include in Report	Align
1	Vector	Number	2	<input checked="" type="checkbox"/>	Center
2	ZMag	Ohms	2	<input checked="" type="checkbox"/>	Center
3	ZPhase	Degrees	2	<input checked="" type="checkbox"/>	Center
4	ExpTrip\$		0	<input checked="" type="checkbox"/>	Center
5	Trip\$		0	<input checked="" type="checkbox"/>	Center
6	Time	Seconds	2	<input checked="" type="checkbox"/>	Center
7	PassFail\$		0	<input checked="" type="checkbox"/>	Center
8	CofVectorNum		0	<input type="checkbox"/>	Center

Pre-Log Calculations	
	Calculation Expression
1	Vector = CofVectorNum
2	ZMag = CofZMag
3	ZPhase = CofZPhase
4	ExpTrip\$ = CofExpTripStateStr\$
5	Trip\$ = CofActualTripStateStr\$
6	Time = CofTripTime
7	PassFail\$ = CofPassFailStr\$

The *edited Results* are more readable for the print outs or saving to the database for later recall.

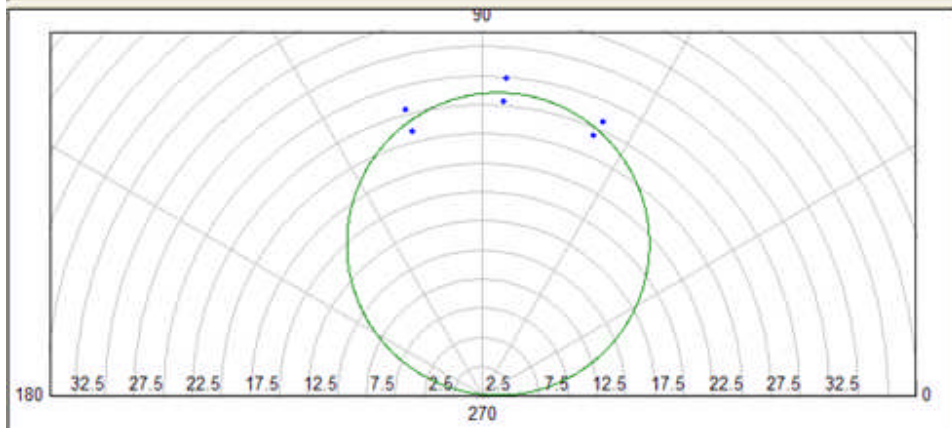
Vector Number	ZMag Ohms	ZPhase Degrees	ExpTrip	Trip	Time Seconds	PassFail
1.00	23.36	104.68	Trip	Trip	0.03	Pass
2.00	25.31	105.09	No Trip	No Trip	0.00	Pass
3.00	25.17	85.63	Trip	Trip	0.03	Pass
4.00	27.18	85.60	No Trip	No Trip	0.00	Pass
5.00	24.24	66.70	Trip	Trip	0.03	Pass
6.00	25.65	66.02	No Trip	No Trip	0.00	Pass



End Control: The End Control retains the Default settings.

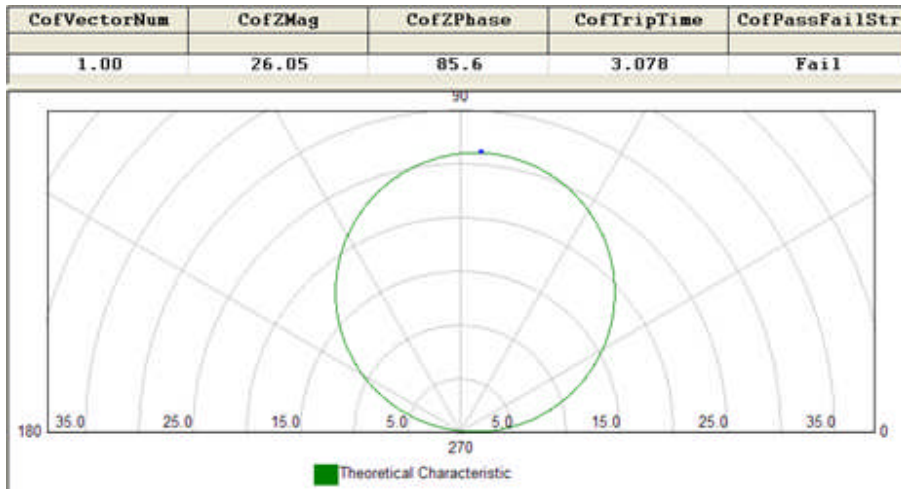
*Shot test results* in the AVTS Test tab contain values for all the Variables listed in the Log Control.

Vector Number	ZMag Ohms	ZPhase Degrees	ExpTrip	Trip	Time Seconds	PassFail
1.00	23.36	104.68	Trip	Trip	0.03	Pass
2.00	25.31	105.09	No Trip	No Trip	0.00	Pass
3.00	25.17	85.63	Trip	Trip	0.03	Pass
4.00	27.18	85.60	No Trip	No Trip	0.00	Pass
5.00	24.24	66.70	Trip	Trip	0.03	Pass
6.00	25.65	66.02	No Trip	No Trip	0.00	Pass



*Search test results* in the AVTS Test tab contain values for all the Variables listed in the Log Control.

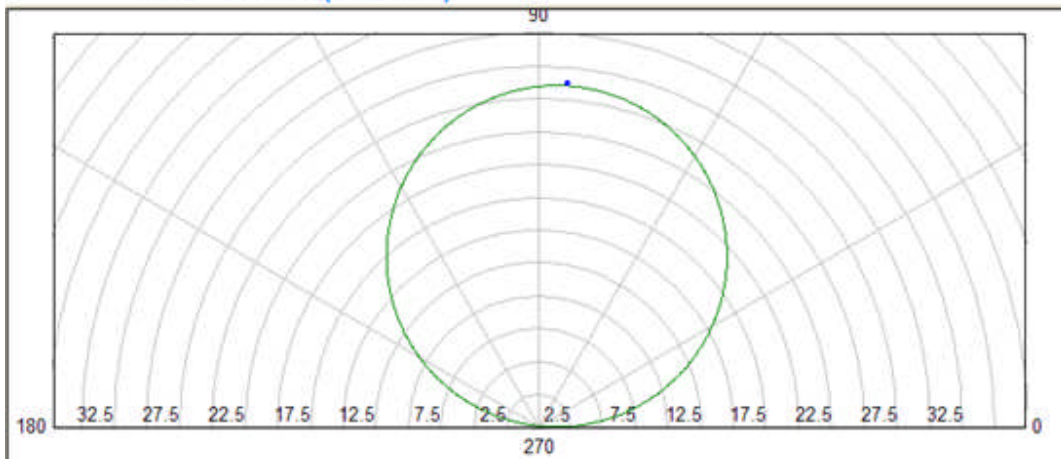




Note: The Log may contain additional Variables to *compare results* with the characteristic. An **Error** Variable might also be added to provide a percent degree of accuracy [  $\text{Error} = (\text{CofZMag} - \text{TheoZ}) / \text{TheoZ} * 100$  ].

CofVectorNu	TheoZ	CofZMag	CofZPhase	CofTripTime	CofPassFail
	Ohms	Ohms	Degrees	Seconds	
1.00	26.00	26.10	85.1	0.082	Pass

TheoZ = GetTheoZ(CofZPhase)



Note: The Log for a Search test does not utilize the Expected and Actual Trip State results and are unselected in the Log Variable list for the Test Results display.

## Contact Control

### Contact Control

The **Contact Control** is used in a process to monitor the status of a device under test. Depending upon the action of the device and the time limit to monitor, the outputs are conditional and constitute a directive decision to continue the test normally or take some other constructed path, such as Abort.



[Contact Control Properties](#)

[Contact Control Links](#)

[Contact Control Example](#)

## Contact Control Properties

The *Contact Control Properties* dialog allows the options for the Unit (Test Set A or B) to monitor and whether or not an Auxiliary Contact is to be used for additional control of the device under test. The Auxiliary Contact is used with PULSAR only.



**Note:** If more than one input is made active, whether in the NO or NC state, the Control adopts a new appearance.



The **Debounce Time** is the amount of time, entered in milliseconds, that the monitored contact must remain in the active state to be recognized as a valid decision. Note: If the Control is used in a timed event, the Debounce Time will be subtracted from the total monitored time so as to register the actual time of the change of state of the monitored contact.

The Post options are displayed as checked, but dimmed. If dimmed, the Post is inactive and ignored by the Control. A single click on a Post will change it from dimmed to active and unchecked.

**Note:** If PULSAR is the active test unit, Posts 1, 2 and 3 apply to the PULSAR Posts 1, 2 and 3 respectively. These are then the only useful posts for monitor.

Checked and dimmed = ignored Post

Unchecked and not dimmed = active Post to monitor a contact to Open or a Voltage to be absent

Checked and not dimmed = active Post to monitor a contact to Close or a Voltage to be present

If more than one contact is made active (undimmed) the "Wait for all inputs" and "Wait for any inputs" options will become active. These options are used to condition the Control presence. Either option will not pass the Control to the next until one is met.

The following options are active for only a single Post to be monitored. If additional Posts are selected to be active, these options will dim for the Post to be ignored.

**Mode:** is to monitor either a Contact or a Voltage input

**Active On:** is to recognize the input Mode

**Applied** is to recognize the presence of Contact continuity or presence of a Voltage

**Changed** is to recognize the change of state of the applied Contact or Voltage

**Removed** is to recognize the opening of a Contact or removal of a Voltage

**Do Action:**

**Ignore** is to disregard any activity on the post and pass the Control if other set conditions are met

**Monitor** is to recognize the state of the post on each event and register the state for later use in the test

**Start** is to initiate the Timer function when the Mode: and Active On: options are met

**Stop** is to cease the Timer function when the Mode: and Active On: options are met

**Max. Time to Operate** edit field allows entry of time monitor in seconds. If monitored activity occurs during the time period, the Control passes to the next control. If monitored activity does not occur within the time period, the test is considered not valid.

**Note:** Max. Time to Operate may be a variable.

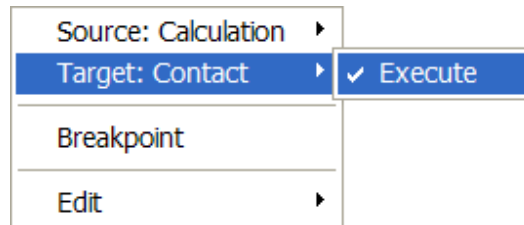
**Latch**, when checked, is to register the first valid condition of a Post after the debounce time setting and to ignore additional changes of state of the monitored Post.

**Enable Horn**, when checked, is to Beep test unit horn on valid activity.

**Turn Currents Off**, when checked, is to Turn all current amplifiers Off upon exit of the Control.

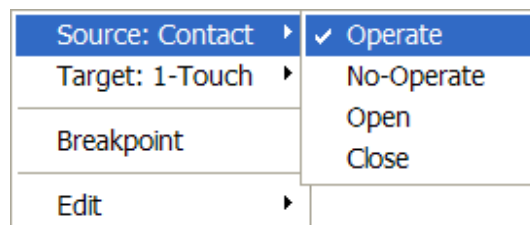
## Contact Control Links

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Links* are Operate, No-Operate, Open and Close.



The Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

The No-Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

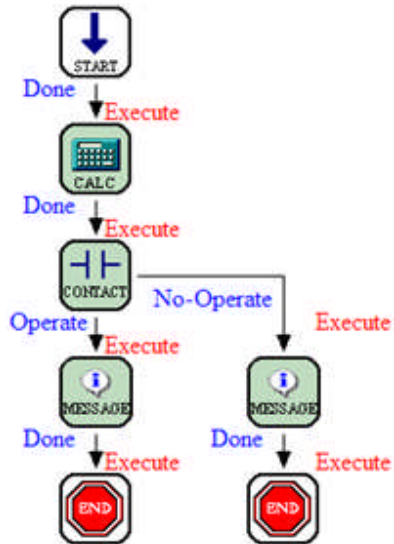
**Note:** The non operation of a required response and Control properties having reached their preset limits may validates this branch and indicates an improper condition in which to abort the procedure.

The Open command is a branch option from the Control. It passes the Control properties either at the completion of an Open contact having closed during the Control execution or completely by-passes the Control if an Open contact is met at the beginning of the Control execution.

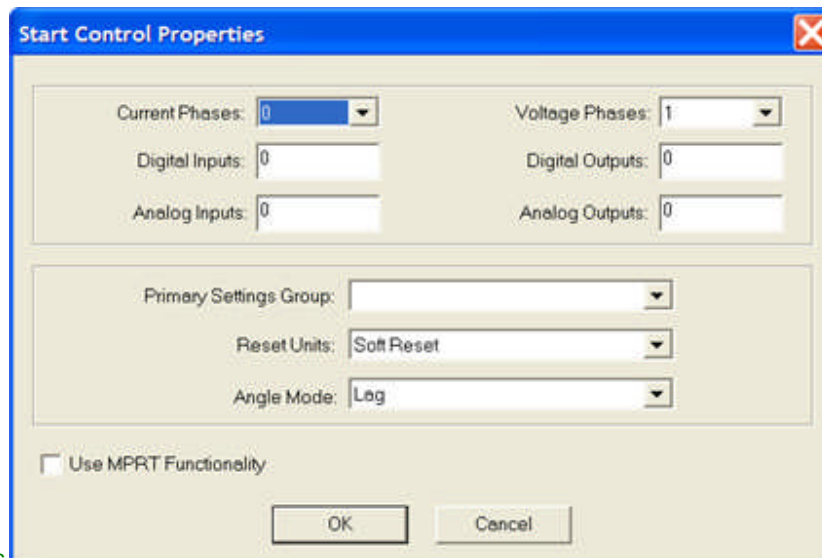
The Close command is a branch option from the Control. It passes the Control properties either at the completion of a Close contact having opened during the Control execution or completely by-passes the Control if a Closed contact is met at the beginning of the Control execution.

## Contact Control Example

*Example conditional Contact Control* with Messages for each condition.

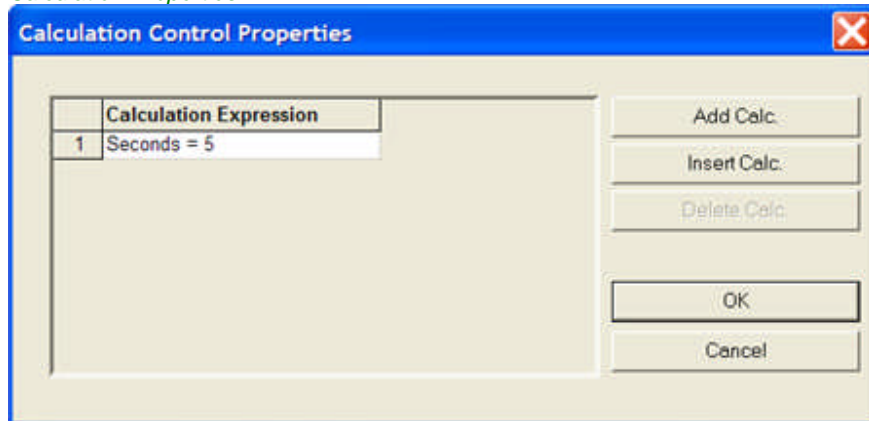


Start to Calculate to Contact. Operate condition Contact to Message to End. No Operate Contact to Message to End.



*Start Properties* uses default options but have set the Current channels = 0 and Voltage Channels = 1

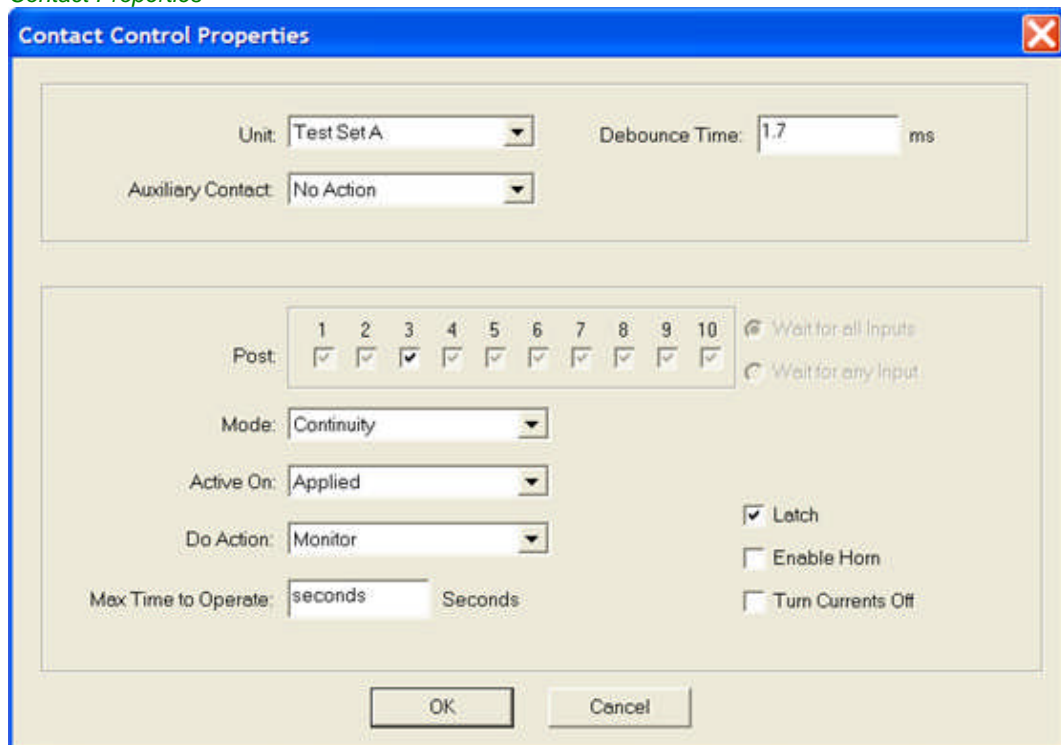
*Calculation Properties*



variable Seconds = 5

places the

*Contact Properties*



has the Max Time to Operate set with the variable = seconds; thus, the Max Time = 5 seconds

Operate Link to Operate *Message Properties*

AVTS Test Execution Message Box

Mode: Message Only

Data Entry Variable: DataEntry

Message:  
Operated

Selection List

List Name
-----------

Add  
Delete

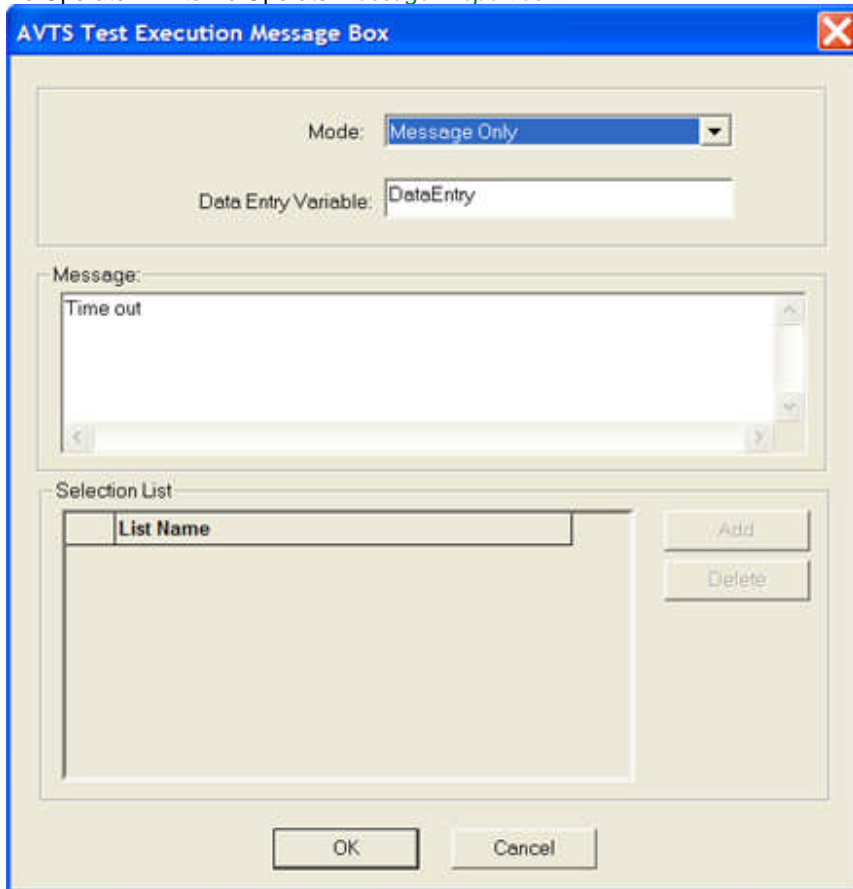
OK Cancel

event occurred

within 5 seconds

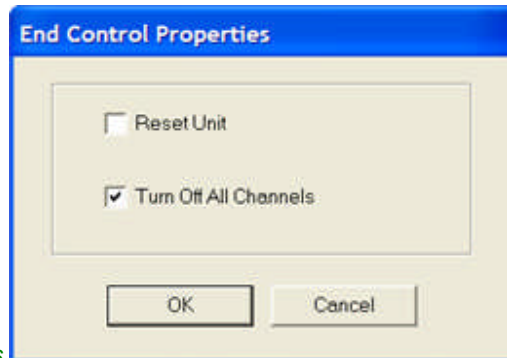


No Operate Link to No Operate *Message Properties*



occur within 5 seconds

event did not



*End Properties*

turn all channels off

## Digital Configuration Control

### Digital Configuration Control

The **Digital Configuration Control** is to develop the conditions for the MPRT Outputs and Inputs and whether or not to record the time stamp of each event.

The Control should be used for the initial Outputs and Inputs configuration on initializing a test; i.e., the Outputs and Inputs conditions at the Start of a test.



**Note:** Should the Control be used during the test procedure or multiple times throughout the procedure, the Execute of the Control will send a reset to the test unit timer. This will cause an interruption to any previously set time function, and may impede any other type test to cause incorrect results.

*Digital Configuration Control Properties* has an Outputs block and an Inputs block for all contact outputs and all input posts.

Outputs		1	2	3	4	5	6	7	8
State:		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
		9	10	11	12	13	14	15	16
State:		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Inputs		1	2	3	4	5	6	7	8
Mode:		Continuity	Continuity	Continuity	Continuity	Continuity	Continuity	Continuity	Continuity
Active On:		Applied	Applied	Applied	Applied	Applied	Applied	Applied	Applied
Do Action:		Start	Stop	Monitor	Monitor	Monitor	Monitor	Monitor	Monitor
Latch:		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Auto Off:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Horn:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Record:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
		9	10	11	12	13	14	15	16
Mode:		Continuity	Continuity	Continuity	Continuity	Continuity	Continuity	Continuity	Continuity
Active On:		Applied	Applied	Applied	Applied	Applied	Applied	Applied	Applied
Do Action:		Monitor	Monitor	Monitor	Monitor	Monitor	Monitor	Monitor	Monitor
Latch:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Auto Off:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Horn:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Record:		<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Unit: Test Set A

OK Cancel

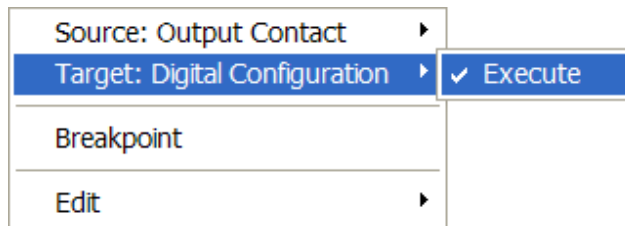
The upper dialog is to set the state of the Digital Configuration Control Outputs contact posts.

The lower block is to set the state of the Digital Configuration Control Inputs monitoring posts.

A drop list is to select either Test Set A or Test Set B, or TCM A, TCM B, TCM C to be influenced by the Control.

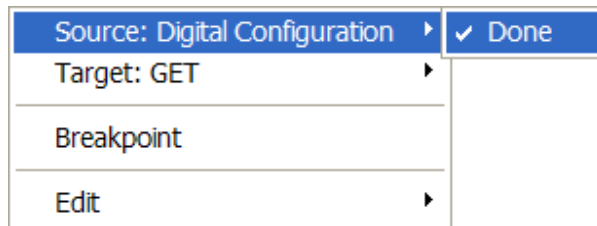
The conditions set in this properties dialog will activate immediately upon entering the Control during a test execution.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

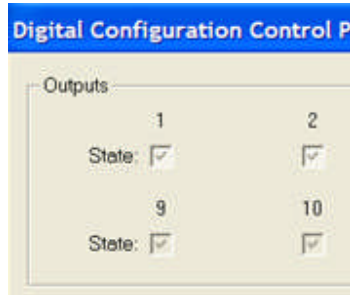
*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Digital Configuration Control Outputs

The *Outputs block* of the Digital configuration Control has 16 check boxes, one for each Output Contact on the MPRT. The MPRT has six outputs on the panel face. Presently, the properties are active for only Contacts 1-6. The remainder are for a future additional accessory to accommodate the Outputs 7-16 and are permanently dimmed.



The Output options are displayed as checked, but dimmed. If Checked and dimmed, the Output is inactive and not in use. A single click on a Output will change if from dimmed to unchecked.

**Note: If PULSAR is the active test unit, Output 1 applies to the PULSAR Aux Contact.**

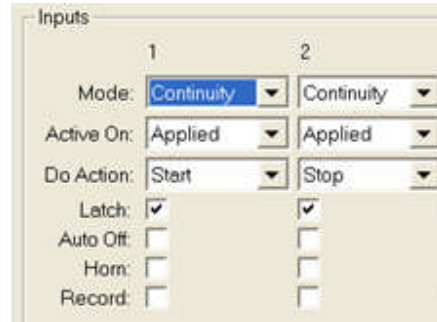
Checked and dimmed = ignored Output

Unchecked and not dimmed = active Output to be recognized in the open (low) state

Checked and not dimmed = active Output to be recognized in the closed (high) state

## Digital Configuration Control Inputs

The *Inputs block* of the Digital Configuration Control has 16 sections, one for each Input on the MPRT. The MPRT has ten inputs on the panel face. Presently, the properties are active for only Inputs 1-10. The remainder are for a future additional accessory to accommodate these inputs 11-16 and are permanently dimmed and unusable.



There are three drop lists and three check boxes for each Input to condition each during the test procedure.

**Mode:** is to monitor either a Continuity or a Voltage input

**Active On:** is to recognize the input Mode that must exist to Start.

Applied is to recognize the presence of Contact continuity or presence of a Voltage and is the default Mode

Changed is to recognize the change of state of the applied Contact or Voltage

Removed is to recognize the opening of a Contact or removal of a Voltage

**Do Action:**

Ignore is to disregard any activity on the post and pass the Control if other set conditions are met

Monitor is to recognize the state of the post on each event and register the state for later use in the test

Start is to initiate the Timer function when the Mode: and Active On: options are met

Stop is to cease the Timer function when the Mode: and Active On: options are met

**Latch:** check box is to recognize the first occurrence of the Mode and Active On options that exceed the debounce time setting and no other subsequence occurrence. The Latch will remain active until reset

**Auto Off:** check box is to turn off all test set channels

**Horn:** check box is to sound the MPRT or PULSAR horn when Mode Active On is TRUE

**Record:** check box is to time tag the selected post occurrences during the test for sequence time plots or data lists

## Digital Graphic Control

### Digital Graphic Control

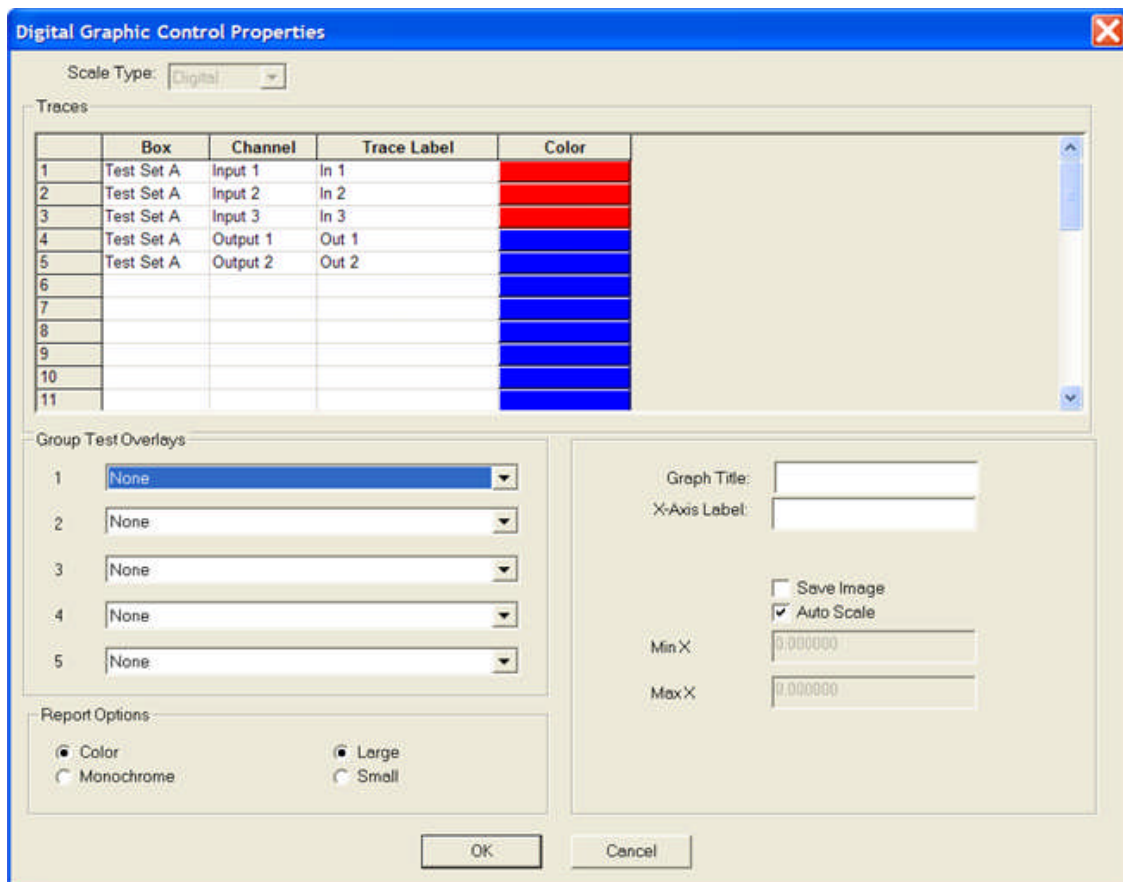
The **Digital Graphic Control** is to set the conditions for display of a graph in the Test tab. The graph will display a time record of the recognized Digital events which occur on the test set Outputs and Inputs. The Digital Graphic Control may be used in tandem with the analog Graphic Control.

The graph will display digital traces of the activities where Contact continuity and Voltage presence will be shown as a high trace, else the trace will be low.

The traces may be reflective of the activity of two test sets and/or three TCM-915S units.

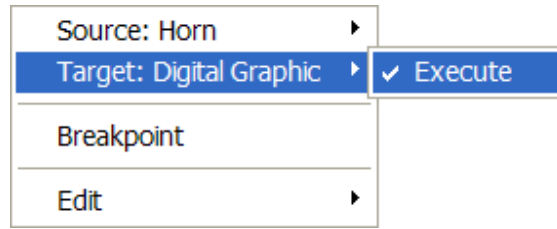


*Digital Graphic Control Properties* are defined in



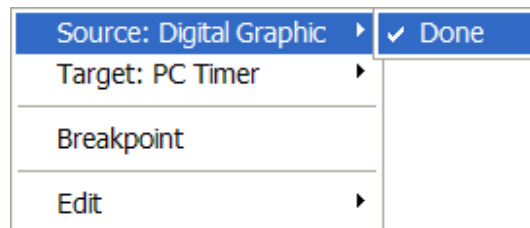
- Digital Graphic Control Traces
- Digital Graphic Control Overlays
- Digital Graphic Control Properties

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

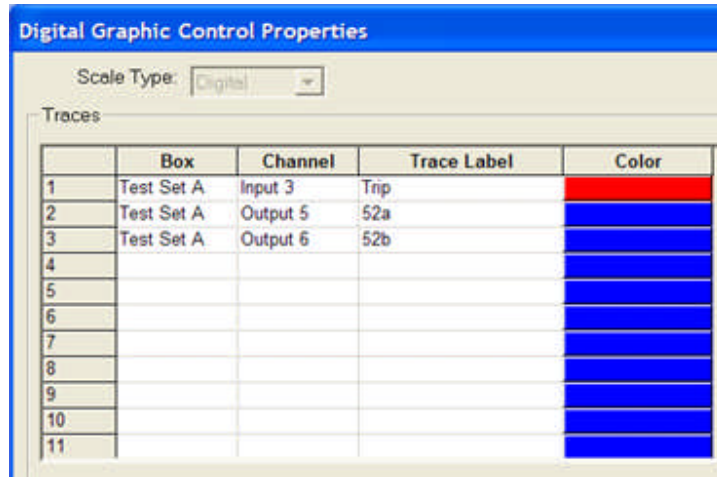
*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Digital Graphic Control Traces

The *Digital Graphic Control Traces* block contains drop lists and edit fields arranged in columns for each of ten rows.



**Scale Type** drops a list of the type graphs to be displayed. The option is Digital only.

Digital graphs are to display, for example, the active and non-active states of contacts and or voltages being monitored.

The **Box** column is to specify from the drop list the Test Set to be used, TCMA, TCMB, TCMC, Test Set A or Test Set B.

The **Channel** column is to specify from the drop list the Output or Input to be displayed for the selected Trace number (row).

**Trace Label** is a text entry for identifying a label for the selected Trace number.

**Scale Group** options in a drop list are Scale Group 1, Scale Group 2 and Scale Group 3.

**Color** will display a palette of 16 colors to select for the color of the graph to be plotted from the test results trace..



## Digital Graphic Control Overlays

The *Group Test Overlays & Reports* boxes allows one or more up to five graphs to be displayed on the graph along with the graph created with this Control and the Report options to be applied.

The image shows a dialog box titled "Group Test Overlays" with a "Report Options" section. The "Group Test Overlays" section contains five numbered rows (1-5), each with a dropdown menu currently set to "None". The "Report Options" section contains two sets of radio buttons: "Color" (selected) and "Monochrome", and "Large" (selected) and "Small".

**Note:** Digital Graphs may also be displayed along side the analog graph, but not overlaid.

The drop list for each of the five overlays contains the test labels of all tests that exist in the Workspace for the active relay. The tests not containing a Graphic Control will not overlay the test but a test execution will conduct the active test with the graphics of the active test. Results from up to five tests may be selected to be overlaid during the test and in the reports generated.

Report **Options** box has two sets of check buttons.

Either Color or Monochrome may be selected for the report.

Either Large or Small graphs may be selected for the report.

## Digital Graphic Control Properties

The *Digital Graph Properties* block contains the edit fields and check blocks to define the test characteristic image.



The screenshot shows a dialog box with a light beige background. At the top, there are two text input fields: 'Graph Title:' and 'X-Axis Label:'. Below these are two checkboxes: 'Save Image' (unchecked) and 'Auto Scale' (checked). At the bottom, there are two more text input fields: 'Min X' and 'Max X', both containing the value '0.000000'.

The **Graph Title:** edit field is to enter a name for the Graph.

The **X-Axis Label:** edit field is to enter a name for the Graph horizontal axis. Option available only in Scale Type = Linear and Digital.

The **Save Image** check box, when checked, will save the image characteristics to the active database for use in other tests.

The **Auto Scale** check box is to allow a change of the Time scale during the test when the plot exceeds the initial boundaries.

The **Min X** edit field is enabled when Auto Scale is unchecked. The minimum time boundary to be displayed is entered in the field in Seconds.

The **Max X** edit field is enabled when Auto Scale is unchecked. The maximum time boundary to be displayed is entered in the field in Seconds.

**Note:** The Auto Scale may be rechecked after the Min and Max time entries have been placed to allow the display to display the time as elapsed. The display will always display the Min time and will not display times greater than Max time.

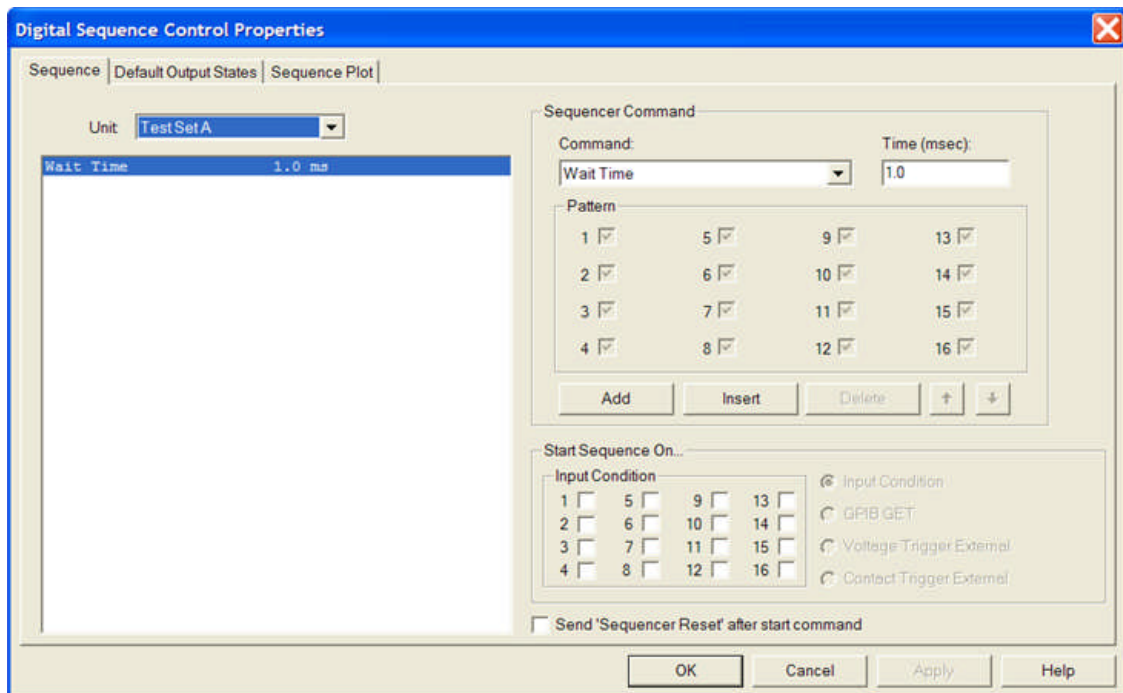
## Digital Sequence Control

## Digital Sequence Control

The **Digital Sequence Control** is to create a step by step sequence to monitor test set Inputs and provide test set Outputs for recording activity of a test under execution.



The *Digital Sequence Control Properties* has three tabs for use in the setup of the Control.

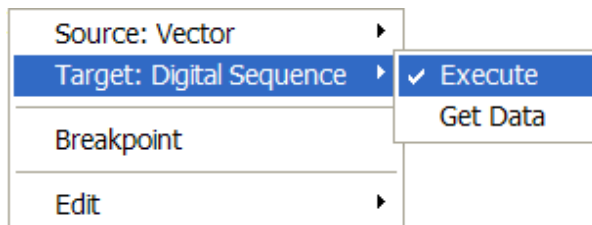


Digital Sequence tab

Digital Default Output States tab

Digital Sequence Plot tab

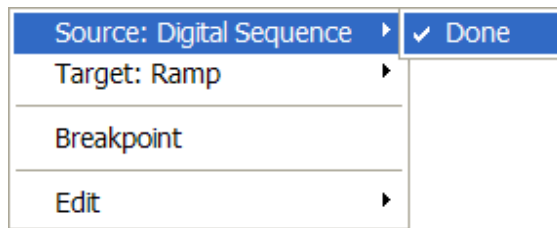
*Target Links* are Execute and Get Data.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Get Data command forces AVTS to read the Record values from the MPRT after or during an execution of a procedure.

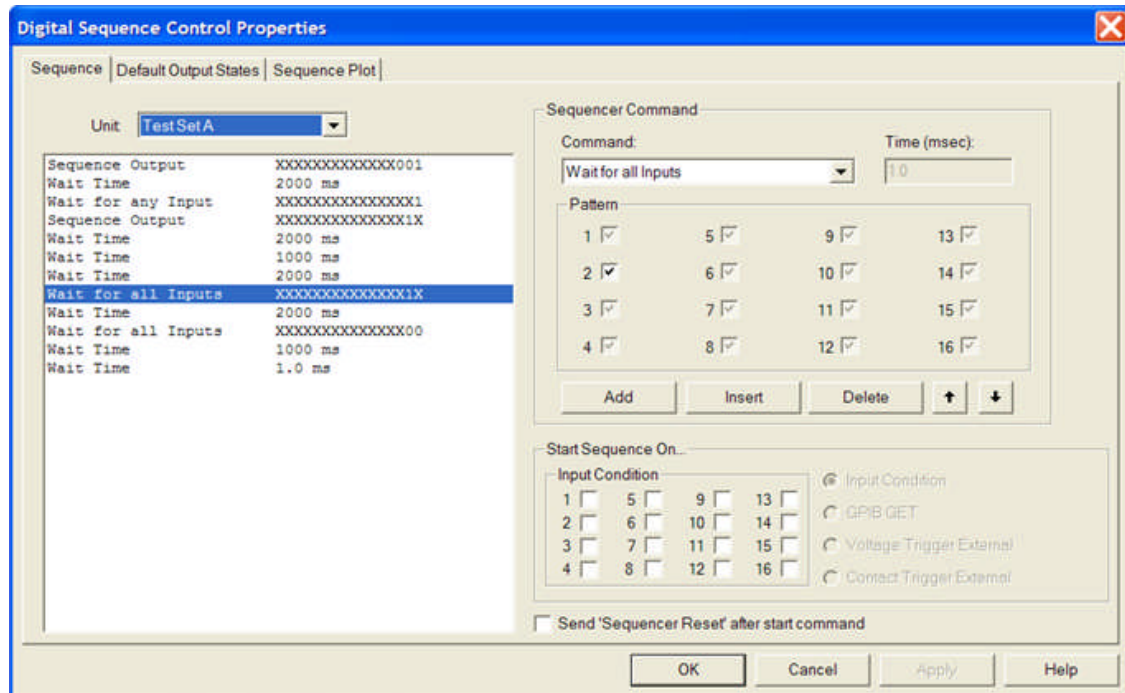
*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Digital Sequence Tab

The *Digital Sequence* tab contains the settings to generate sequenced digital test signals for the Outputs and Inputs for the selected test device. The sequence may have discrete time intervals set and option controls to Start and record the test.



The Unit drop list is to select the test device to generate the developed sequence. The options are Test Set A, Test Set B, TCMA, TCMB or TCMC.

The **Sequence Command** block contains check boxes, drop list, and items to develop the sequence.

**Command** drop list is to select the condition for the sequence entry.

**Wait Time** allows entry of a time value in milliseconds in the Time (msec) edit field. The Wait Time entry and delay Time will be displayed in a row in the Command field. The default Time entry is 1.0 msec.

**Wait for all** inputs allows for selection of the Inputs in the Pattern field to be either Ignored, High or Low. The Wait for all inputs and the created Pattern will be displayed in a row in the Command field. To pass to the next sequence command, the pattern must be TRUE for the entire 16 digit pattern.

**Note:** The Pattern will be displayed in digital format: X = Ignore, High = 1 and Low = 0. The Input 1 pattern will be the right hand entry in the 16 digit field.

**Wait for any** input allows for selection of the Inputs in the Pattern field to be either Ignored, High or Low. The Wait for any inputs and the created Pattern will be displayed in a row in the Command field. To pass to the next sequence command, only one of the High or Low Inputs pattern must be TRUE for the entire 16 digit pattern.

**Note:** The Pattern will be displayed in digital format: X = Ignore, High = 1 and Low = 0. The Input 1 pattern will be the right hand entry in the 16 digit field.

**Sequence Output** allows for selection of the Outputs in the Pattern field to be either Ignored, High or Low. The Sequence Output and the created Pattern will be displayed in a row in the Command field. No time delay is expected for the Test Unit to generate this action and a pass to the next sequence command will ensue.

**Note:** The Pattern will be displayed in digital format: X = Ignore, High = 1 and Low = 0. The Input 1 pattern will be the right hand entry in the 16 digit field.

After the first Command has been created, another sequence Command may be added by selecting the **Add** button. The Add will place a new Command at the bottom of the list of Commands. The new addition will be "Wait Time 1.0 ms". Select the Command just entered with the mouse or use the Up Down arrows to highlight the item for edit with the Sequence Command blocks.

**Insert** will place a new addition, "Wait Time 1.0 ms", in the list just above the highlighted, selected, Command list.

**Delete** will remove the selected item from the Command list.

**Start Sequence On...** settings enable the Sequence based on the specific conditions specified by the four radio button options to the right of the Input Condition block.

**Input Condition** button will Start the Sequence Command list when ALL of the Input Condition check boxes states High are met. Each Input option is either High (unchecked) or Low (unchecked). An Ignore option is not available.

**Note:** This option is the only Start option available for MPRT.

**GPIB GET** button allows a sequence to Start only when a GET (general execute trigger) command is applied on the GPIB interface. This option only applies to the TCM Units.

**Note:** This option is available for TCM-915 Timer Control Monitor sets only.

**Voltage Trigger External** will Start the sequence when an external voltage is applied to a BNC input on the TCM Units. This option only applies to the TCM Units.

**Note:** This option is available for TCM-915 Timer Control Monitor sets only.

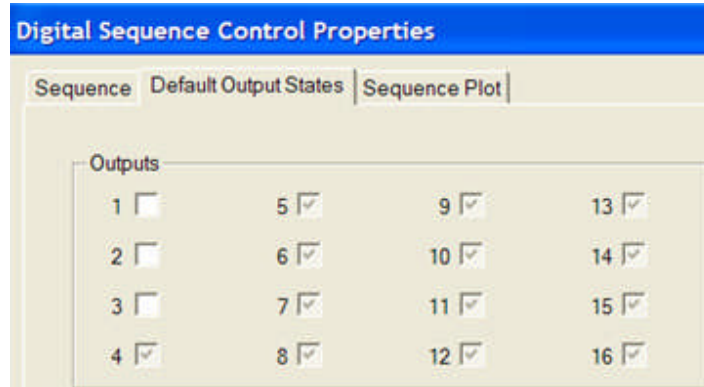
**Contract Trigger External** will Start the sequence when an external contact continuity is applied to a BNC input on the TCM Units. This option only applies to the TCM Units.

**Note:** This option is available for TCM-915 Timer Control Monitor sets only.

Send "**Sequencer Reset**" after start command check box will cause the Sequence to Reset immediately after the test encounters a Start control.

## Digital Default Output States Tab

The *Default Output States tab* of the Digital Sequence Control Window allows setting the default states of each of his 16 digital outputs. The settings in the Outputs check boxes block dictates the state of all outputs of a Sequence command list in the Sequence tab when a Start is incurred.



The Outputs are default set to Xs or Ignore.

**Ignore** is represented by grayed-out and checked check boxes.

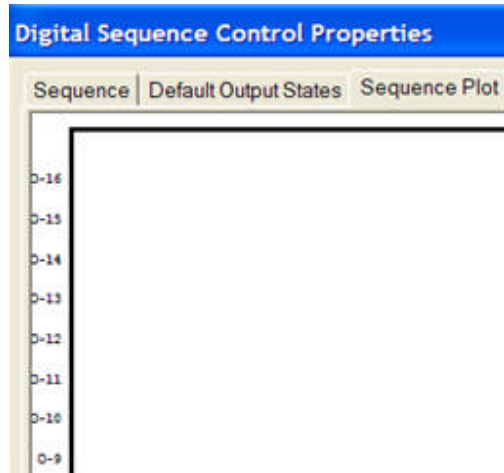
To make an output low/zero, click on it once to clear it.

To make the output high/one, click on it again to make it ungrayed and checked.

To set it back to an Ignore state, click on it again.

## Digital Sequence Plot Tab

The *Sequence Plot tab* enables graphical view of the states of the customized 16 sequenced Digital test signals together in time.



Each trace represents an Output and/or Input and the state of that trace as either high or low at each incident in time. The trace activity will be represented referenced to a time scale at the bottom of the plot.



## Dynamic Control

### Dynamic Control

The **Dynamic Control** allows creation of a step by step sequence to monitor test set Inputs and provide test set Outputs for recording activity of a test under execution.

Use the Wave Index feature to provide State Sequence testing of relays and relay protection systems. The Wave Index feature creates new waveforms, during an applied waveform output, enabling adjustment of the wavefore harmonics real-time.

Use the Dynamic Control to test single or multi-shot reclosing relays.

A Recorder is enabled within the test step sequence to record the Analog and Binary Inputs and Outputs for view and analysis.



The Control allows control of the test set amplifier outputs to the relay under test during the monitor of the test set Inputs and control of the test set Outputs.

Monitoring and recording all the desired test set metered Current and Voltage outputs are available to view and save to file using the Capture Tab.

The following Dynamic Control topics describe the portions of use:

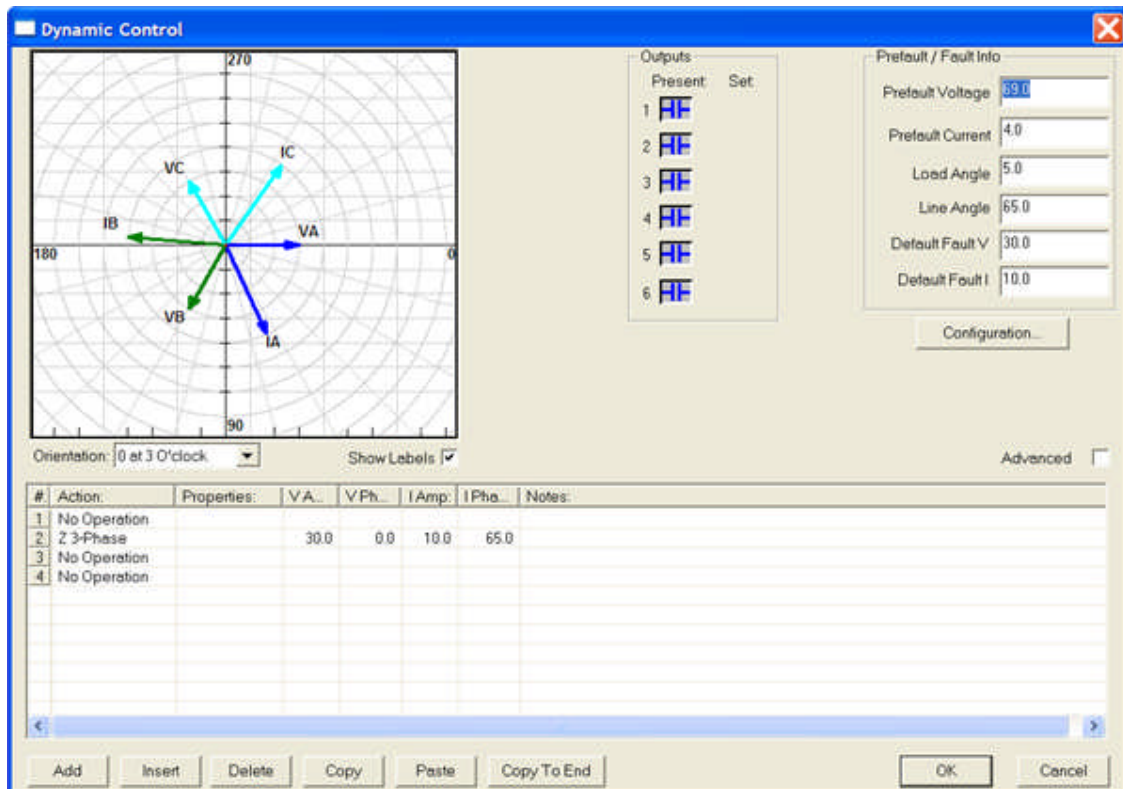
- Dynamic Control Properties
- Dynamic Control Graph
- Dynamic Control Inputs
- Dynamic Control Outputs
- Dynamic Control Prefault/Fault Info
- Dynamic Control Configuration
- Dynamic Control Sequence
- Dynamic Control Example

## Dynamic Control Properties

The **Dynamic Control** Properties allow extended capability of the Control when used in a previously created test or as a stand alone test.



*Dynamic Control Properties* is divided into several areas.



The Graph area is for display of the active Current and Voltage Vectors for the selected portion of the created sequence of activity.

The Inputs block is for adjusting the test set Input state for a given step.

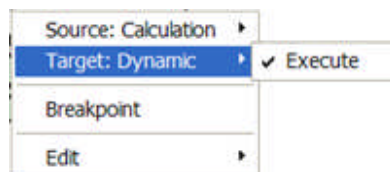
The Outputs block is for adjusting the test set Output state for a given step.

The Prefault / Fault Info block is for edit of the respective variables or values.

The Configuration... button is to adjust Inputs sensing and other data options.

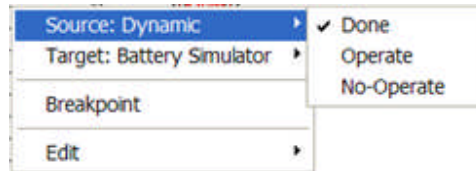
The Sequence block is to create a Step by Step control of the test set Inputs and Outputs.

*Target Dynamic:* Execute



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source:* Done, Operate, No Operate



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

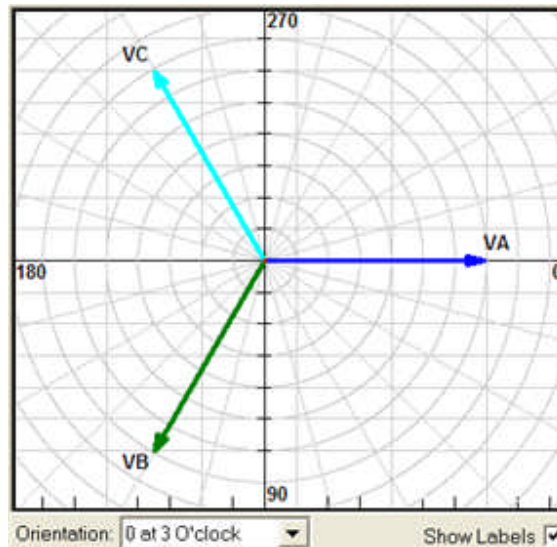
The Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

The No-Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

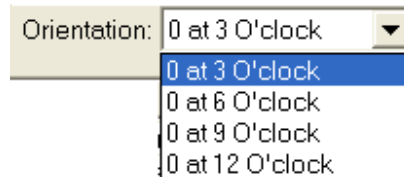
**Note:** The no-operation of a required response and the Control properties having reached their preset limits validates this branch and indicates an improper condition in which to abort the procedure.

## Dynamic Control Graph

The *Dynamic Control Graph* displays the Vectors assigned active in the selected Steps of the Sequence.



The *Orientation* drop down list allows selection of the location of 0 degree reference phase angle. The default is 0 degrees at 3 O'clock.



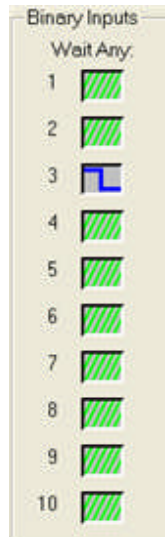
The **Show Labels** check box displays, when checked, each of the Vectors phase label. The default is a checked box. Un-check to remove Vector labels from the Graph.

**Note:** Whether the box is checked or not, the Phase Label, Phase Magnitude and Phase Angle will display in the Graph when the cursor is placed on the Phase arrow head.

## Dynamic Control Inputs

The Binary Inputs block not visible when the Dynamic Control is first opened in the default. All Inputs are set to be ignored by the Sequence until altered to be recognized as a Voltage Absent/Contact Open or Voltage Presence/Contact Closed.

The default *Binary Inputs* block allows selection of any combination of Inputs sense for the selected Sequence Step. The selected Input Sense will remain in the chosen state until altered in another Sequence Step.



The Action options are:

*Wait All* dictates that all ten Inputs set conditions must be met to continue to the next Sequence Step and displays Properties XXXXXXXX10X Input1 = Ignore, Input2 = Open, Input3 = Closed

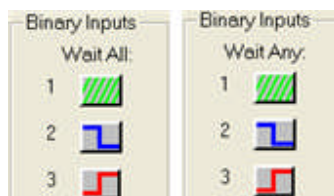



*Wait Any* dictates that only one of the Active set conditions must be met to continue to the next Sequence Step and displays Properties XXXXXXXX10X Input1 = Ignore, Input2 = Open, Input3 = Closed





To change an Input sense, a **Wait All** or **Wait Any** Action must be selected in a sequence step to make visible the Binary Inputs block. The sense of an Input is made by a single left click on the Input in the block. Each successive click alters the available states, Ignore to Open to Closed.

The Inputs block labels the *Wait All or Wait Any* Action



Sense is **Ignore** or InActive 

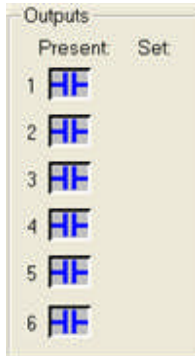
Sense is Active for Input to be recognized as **Open Contact** or **Voltage UnApplied** 

Sense is Active for Input to be recognized as **Closed Contact** or **Voltage Applied** 

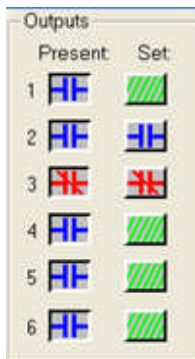
AVTS Help 092607

## Dynamic Control Outputs

The *Binary Outputs* block shows a display of the default Present settings when the Dynamic Control is first opened.. All Outputs are set Normally Open and Ignored by the Sequence until altered to be recognized as either an active Normally Open or Normally Closed Contact.



The default Outputs block will display a parallel column of Output selectors to *Set the Outputs* to the desired Active State.



A Sequence Step Action Set Output option must be selected to the to alter the Output.

The Action options are:


*Set Output* dictates that all six Outputs set conditions must be applied and to continue to the next Sequence Step. and displays Properties XXX10X Input1 = Ignore, Input2 = Open, Input3 = Closed




A single left click on the Set column contact alters the Output state for the selected Sequence Step.

Each successive click alters the available states:

The block labels the Action

**Ignore** sets the Output to an **InActive** state 

**Open** sets the Output to be **Open** and Active applied to the connected apparatus 

**Close** sets the Output to be **Closed** and Active applied to the connected apparatus 

## Dynamic Control Prefault/Fault Info

The *Prefault / Fault Info* block contains six edit fields:

Prefault / Fault Info	
Prefault Voltage	69.0
Prefault Current	4.0
Load Angle	5.0
Line Angle	65.0
Default Fault V	30.0
Default Fault I	10.0

**Prefault Voltage** is to edit the value with a number or a variable for a Voltage to be set

**Prefault Current** is to edit the value with a number or a variable for a Current to be set

**Load Angle** is to edit the value with a number or a variable for a Load Angle to be set

**Line Angle** is to edit the value with a number or a variable for a Line Angle to be set

**Default Fault V** is to edit the value with a number or a variable for a Fault Voltage to be set  
Step Action

**Default Fault I**

**Prefault Values** are automatically set to the **Action** selections

### *Set Phase*

10	SetPhase	A	1	69	0.0	Freq	Sin	4	5	Freq	Sin
----	----------	---	---	----	-----	------	-----	---	---	------	-----

### *Set 3 Phase*

10	SetPhase	A	1	69	0.0	Freq	Sin	4	5	Freq	Sin
11	SetPhase	B	1	69	120.0	Freq	Sin	4	125.0	Freq	Sin
12	SetPhase	C	1	69	240.0	Freq	Sin	4	245.0	Freq	Sin

**Fault Values** are automatically set to the **Action** selections:

### *Z 3-Phase*

6	Z 3-Phase		1	30	0.0	Freq	Sin	10	65	Freq	Sin
---	-----------	--	---	----	-----	------	-----	----	----	------	-----

### *Z Phase-Phase*

7	Z Phase-Phase	A-B	1	30	20	Freq	Sin	10	65	Freq	Sin
---	---------------	-----	---	----	----	------	-----	----	----	------	-----

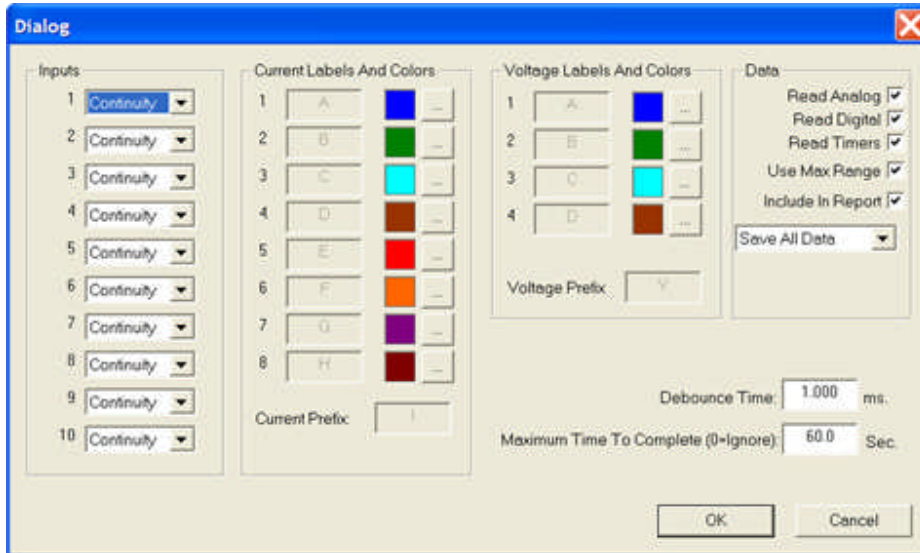
### *Z Single Phase*

8	Z Single Phase	A	1	30	0.0	Freq	Sin	10	65	Freq	Sin
---	----------------	---	---	----	-----	------	-----	----	----	------	-----



## Dynamic Control Configuration

The *Dynamic Configuration dialog* displays several areas for reference and edit.



The dialog is divided into blocks of accessible drop lists, browse buttons, check boxes and edit fields options:

The *Inputs block* contains a drop list, for each of the 10 MPRT Inputs, to select the operation sense:



The Continuity option sets the Input to register activity of a dry contact.

The Voltage option sets the Input to register activity of the presence or absence of a Voltage.

The *Current Labels and Colors block* contains the identity of each of the eight possible current channels:

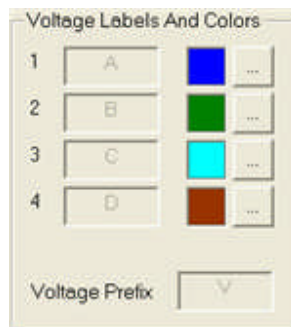


The dimmed field for each channel displays the channel label set in the menu item Tools/Options.../Channel Labels tab.

The colored square displays the assigned color that is selected from the color palate browse button.

The Current Prefix: dimmed field displays the channel variable label set in the menu item Tools/Options.../Channel Labels tab.

The *Voltage Labels and Colors block* contains the identity of each of the four possible current channels:

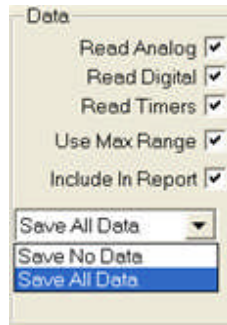


The dimmed field for each channel displays the channel label set in the menu item Tools/Options.../Channel Labels tab.

The colored square displays the assigned color that is selected from the color palate browse button.

The Voltage Prefix: dimmed field displays the channel variable label set in the menu item Tools/Options.../Channel Labels tab.

The *Data block* contains five check boxes and a drop list of options:



**Read Analog** check box

**Read Digital** check box

**Read Timers** check box

**Use Max Range** check box

**Include In Report** check box enables retention of Save All Data option to be included in the test Results.

The Drop List has two options:

**Save All Data**

**Save No Data**

The **Debounce Time** edit field is to enter the milliseconds time to allow recognition of a valid Input operation. The Debounce Time will be subtracted from the recorded time to save the actual response time to the Input.

The **Maximum Time To Complete (0 = Ignore)** edit field is to enter in seconds the time allowed for the complete operation of the constructed Sequence Steps. This will be the maximum time to be displayed in the Capture tab and in a Report. A zero entry ignores a Maximum set time to allow capture of data that may exceed an expected maximum time.

## Dynamic Control Sequence

The Dynamic Control *Steps of Action* contains several columns, default shown, for selection and entry for each step of a sequence.

#	Action:	Properties:	V Amp:	V Phase:	I Amp:	I Phase:	Notes:
1	No Operation						
2	Prefault						
3	Zero Vector						
4	Zero Current						
5	Z 3-Phase		30	0.0	10	65	
6	Z Phase-Phase	A-B	30	20	10	65	
7	Z Single Phase	A	30	0.0	10	65	
8	Set Phase	A	69	0.0	4	5	
9	Set Phase	A	69	0.0	4	5	
10	Set Phase	B	69	120.0	4	125.0	
11	Set Phase	C	69	240.0	4	245.0	
12	Wait Any	XXXXXXXX..					
13	Wait All	XXXXXXXX..					
14	Wait Cycles	0.0					
15	Wait ms	0.0					
16	Wait Phase	0.0					
17	Wait Ext						
18	Set Output	XXXXXX					
19	Start Record	Mode: M..					
20	Stop Record						
21	Start Timer	1					
22	Stop Timer	1					
23	MPRT Cmd						

The default columns are labeled from left to right with the following with the Advance check box unchecked:

# is the step number in the sequence.

**Action:** displays the selected drop list option to be applied for the step. The list options are:

**No Operation** - a space filler with no intended time delay for possible Action option change. This is the default for the Add or Insert step button option. **No entries allowed to any columns.**

**Prefault** - automatically inserts the **Prefault** Voltage and Current magnitudes and angles set in the Prefault/Fault Info block. The values may be edited. **No entries allowed to any columns.**

**Zero Vector** - the option to set all current and voltage amplifiers to zero output without turning the amplifiers OFF. **No entries allowed to any columns.**

**Zero Current** - the option to set all current amplifiers to zero output without turning the amplifiers OFF. **No entries allowed to any columns.**

**Z 3-Phase** - automatically inserts the **Fault** 3-phase values Voltage and Current magnitudes and angles set in the Prefault/Fault Info block. The values may be edited. **No Properties entries allowed. Wave Sum:** drop list options 1 - 4.

**Z Phase-Phase** - automatically inserts the **Fault** phase-phase values for Voltage and Current magnitudes and angles set in the PreFault/Fault Info block. The values may be edited. The **Properties** column allows options of **Variable, A-B, B-C** and **C-A** fault types. The Variable must be defined prior to the Dynamic Control execution. **No Properties entries allowed. Wave Sum:** drop list options 1 - 4.

**Z Single Phase** - automatically inserts the **Fault** single phase values for Voltage and Current magnitudes and angles set in the PreFault/Fault Info block. The values may be edited. The **Properties** column allows options of **Variable, A, B** and **C** fault types. The Variable must be defined prior to the Dynamic Control execution. **No Properties entries allowed. Wave Sum:** drop list options 1 - 4.

**Set Phase** - automatically inserts the **Prefault** single phase values for Voltage and Current magnitudes and angles set in the PreFault/Fault Info block. The values may be edited. The **Properties** column allows options of **A, B, C, D, E, F, G** and **H** Voltage or Current channel. **Wave** drop list options 1 - 4.

**Set 3 Phase** - creates three steps, with default Properties A, B and C respectively, that automatically inserts the **Prefault** single phase values for Voltage and Current magnitudes and angles set in the PreFault / Fault Info block. The values may be edited. The **Properties** column allows options of **A, B, C, D, E, F, G** and **H** Voltage or Current channel. The **Properties** column allows options of **A, B, C, D, E, F, G** and **H** Voltage or Current channel. **Wave** drop list options 1 - 4.

**Wait Any** - automatically displays a series of X's in the **Properties** column. The X's represent the **Inputs** numbered **1 to 10** from **right to left**. **X** = Ignore, **0** = Open or Voltage absent and **1** = Closed or Voltage present. Any Input sensed to the **0** or **1** setting will advance the sequence to the next step. **No entries allowed to all other columns.**

**Wait All** - automatically displays a series of X's in the **Properties** column. The X's represent the **Inputs** numbered **1 to 10** from **right to left**. **X** = Ignore, **0** = Open or Voltage absent and **1** = Closed or Voltage present. All Inputs set to be sensed to **0** or **1** setting must be met to advance the sequence to the next step. **No entries allowed to all other columns.**

**Wait Cycles** - numeric or variable entry to delay by the cycles time advancing to the next step. **Properties** column defaults to 0.0 cycles and may be edited. **No entries allowed to all other columns.**

**Wait ms** - numeric or variable entry to delay by the milliseconds time advancing to the next step. **Properties** column defaults to 0.0 milliseconds and may be edited. **No entries allowed to all other columns.**

**Wait Phase** - numeric or variable entry to delay by the cycles time advancing to the next step. **Properties** column defaults to 0.0 degrees and may be edited. **No entries allowed to all other columns.**

**Wait Ext.** - delay advancing to the next step until an external stimulus is sensed. **No entries allowed to any columns.**

**Set Output** - automatically displays a series of X's in the **Properties** column. The X's represent the **Outputs** numbered **1 to 6** from **right to left**. **X** = Ignore, **0** = Open Output and **1** = Close Contact present. All Outputs set to be set to **0** or **1** setting will be changed immediately and advance to the next step. **No entries allowed to all other columns. No entries allowed to all other columns.**

**Start Record** - begins the Recorder function to store all waveform graphics and digital Input/Output activity that occur from this Step until the Stop Recorder Action occurs or the Maximum Time to Complete ceases the Control. **Properties** drop list for Mode: Low, Mode: Medium or Mode: Low.

**Stop Record** - stops the Recorder function. Usually placed at or near the end of the sequence Steps. **No entries allowed to any columns.**

**Start Timer** - allows selection of the **Main** or any of the **1 - 19** timer functions available. Any or all of the timers may be selected to begin measuring time within the sequence Steps. Time may begin upon the Dynamic Control execution or with a Step set to **Main** or **Start Timer**. **Properties** drop list for Timer selection.

**Stop Timer** - allows selection of the Main or any of the 1 - 19 timer functions available. Any or all of the timers previously set to Start may be selected to cease and store the time register. **Properties** drop list for Timer selection.

**MPRT Cmd** - allows text numeric entry from the MPRT Command Set, where several commands may be set colon delimited. **Properties** allows entries of the desired commands.

**Properties:** provides, depending on the Action option selected for the step, drop lists options, assigned entry options, numeric entries or a null field.

**V Amp:** provides for the Voltage Magnitude, depending on the Action option selected for the step, numeric entry of values, values preset in the Configuration dialog, or a null field.

**V Phase:** provides for the Voltage Phase Angle, depending on the Action option selected for the step, numeric entry of values, values preset in the Configuration dialog, or a null field.

**I Amp:** provides for the Current Magnitude, depending on the Action option selected for the step, numeric entry of values, values preset in the Configuration dialog, or a null field.

**I Phase:** provides for the Current Phase Angle, depending on the Action option selected for the step, numeric entry of values, values preset in the Configuration dialog, or a null field.

**Notes:** allows entry of text to identify the purpose of the created step.

**Advance Check Box** displays *Additional columns*, inserted in the default columns, labeled with the following with the Advance check box checked:

#	Action:	Properties:	Wave Sum	V Amp	V Phase	V Freq	V WF	I Amp	I Phase	I Freq	I WF	IDC AMP	IDC TC	VDC AMP	VDC TC	Note
1	No Operation															
2	Prefault															
3	Zero Vector															
4	Zero Current															
5	Z 3-Phase		1	30	0.0	Freq	Sin	10	65	Freq	Sin					
6	Z Phase-Phase	A-B	1	30	20	Freq	Sin	10	65	Freq	Sin					
7	Z Single Phase	A	1	30	0.0	Freq	Sin	10	65	Freq	Sin					
8	Set Phase	A	1	69	0.0	Freq	Sin	4	5	Freq	Sin					
9	Set Phase	A	1	69	0.0	Freq	Sin	4	5	Freq	Sin					
10	Set Phase	B	1	69	120.0	Freq	Sin	4	125.0	Freq	Sin					
11	Set Phase	C	1	69	240.0	Freq	Sin	4	245.0	Freq	Sin					
12	Wait Any	XXXXXXXXXX														
13	Wait All	XXXXXXXXXX														
14	Wait Cycles	0.0														
15	Wait ms.	0.0														
16	Wait Phase	0.0														
17	Wait Ext.															
18	Set Output	XXXXXX														
19	Start Record	Mode: Medi...														
20	Stop Record															
21	Start Timer	1														
22	Stop Timer	1														
23	MPRT Cmd															

**Wave Sum:** drop list options, numbers 1 through 4, are the number of harmonics that can be added to create a special waveform.

Create a Fault wave with **Wave Sum: 1** and add Exponential for Fault DC offset if desired

Create a second Fault sequence step with **Wave Sum: 2** and create a second harmonic of the first.

**Note:** Wave Sum: 2 - 4 options do not allow adding DC values or Exponential Time Constants.

Create additional harmonics similar to the Second for Wave Sum: 3 and 4 if desired.

**V Freq:** is the **Voltage Frequency** in Hertz edit field for entry of the frequency value to be applied to the **V WF:** option for the Voltage channel assigned for the step #.

**V WF:** is the **Voltage Waveform** type drop list with the options **Sin**, **Square** and **Half** for the Voltage channel to which the **V Freq:** value is applied.

**I Freq:** is the **Current Frequency** in Hertz edit field for entry of the frequency value to be applied to the **I WF:** option for the Current channel assigned for the step #.

**I WF:** is the **Current Waveform** type drop list with the options **Sin**, **Square** and **Half** for the Current channel to which the **I Freq:** value is applied.

**IDC AMP:** entry field is to enter the **Current DC Amplitude** for the Current channel assigned for the step #.

**IDC TC:** entry field is to enter the **Current DC Time Constant** in milliseconds for the **IDC AMP:** entry and Current channel assigned for the step #.

**VDC AMP:** entry field is to enter the **Voltage DC Amplitude** for the Voltage channel assigned for the step #.

**VDC TC:** entry field is to enter the **Voltage DC Time Constant** in milliseconds for the **VDC AMP:** entry and Voltage channel assigned for the step #.

**Option buttons** to edit the Sequence Steps are:

**Add** - option adds a step to the bottom of the existing steps

**Insert** - option adds a step to the existing steps one step above the selected step

**Delete** - option deletes the selected step

**Copy** - option will place the selected step in a clipboard for insertion to another location

**Paste** - option will place a step resident in the clipboard ahead of a selected step

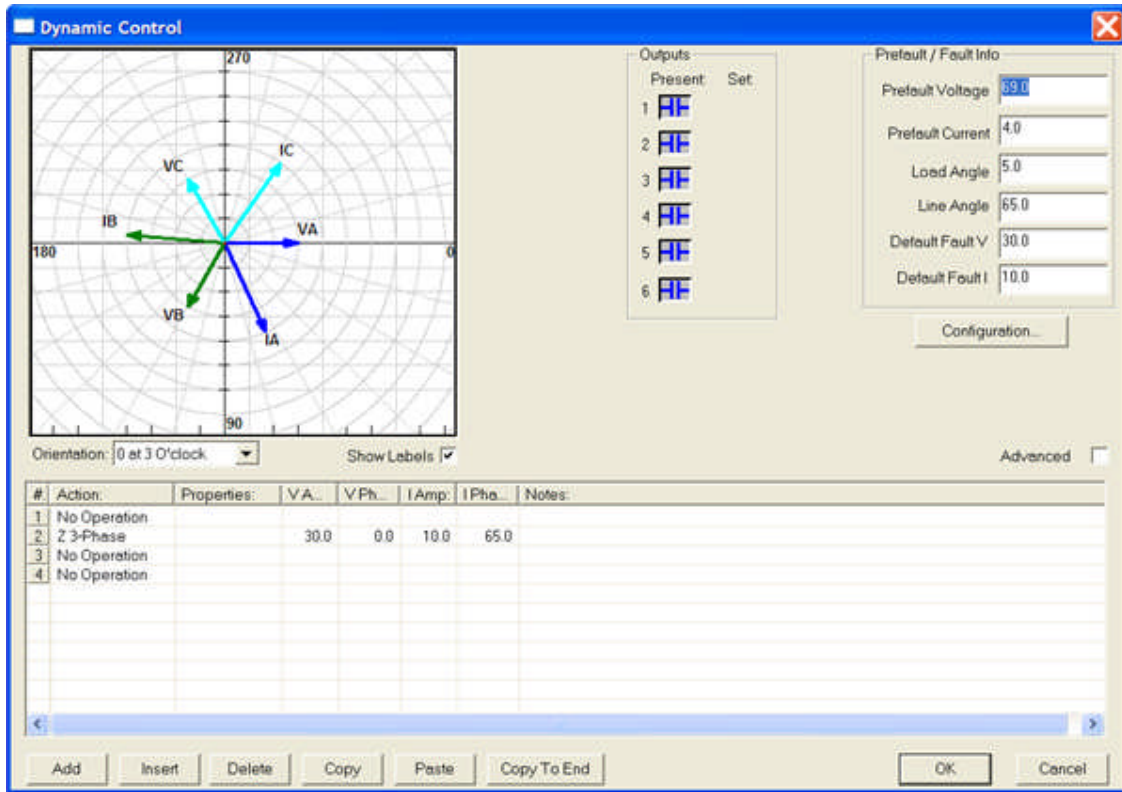
**Copy to End** - option will place a step resident in the clipboard after the last step in the sequence

**OK** - option exits the Dynamic Control dialog and save all edits

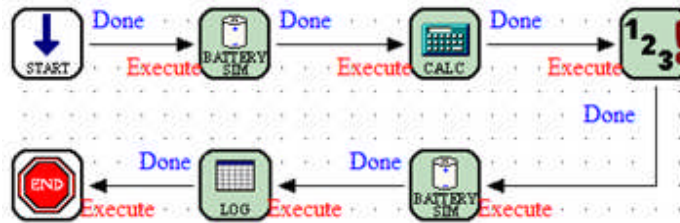
**Cancel** - option produces a warning message that all edits will not be saved upon exit of the Dynamic Control dialog

## Dynamic Control Example 1

The *Dynamic Control* Example 1 is set with steps to test the reclosing sequence of a microprocessor distance or distribution relay with reclose options..

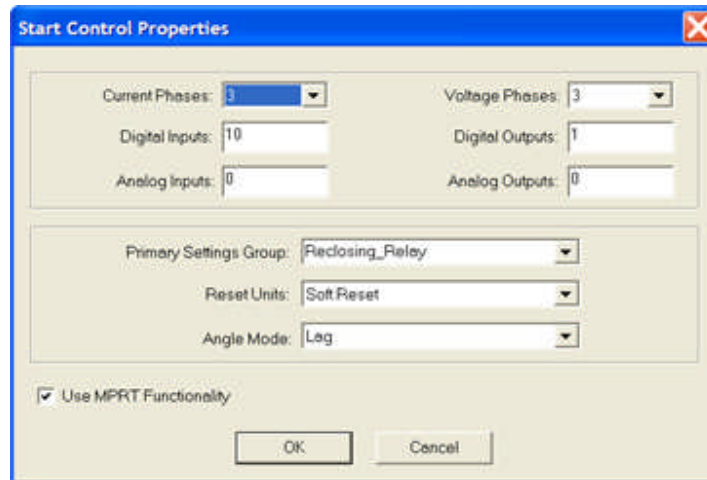


The *test procedure* includes the Battery Simulator to apply DC control voltage to the relay and is maintained until completion of the test.

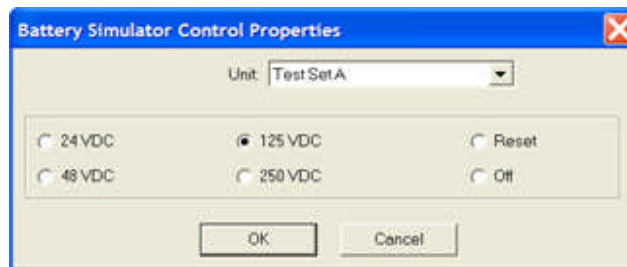


The *Start Control* sets the required amplifiers for the fault applied during the test, the number of Digital Inputs and Outputs that may be required, the Setting Group label, and the use of MPRT Functionality.

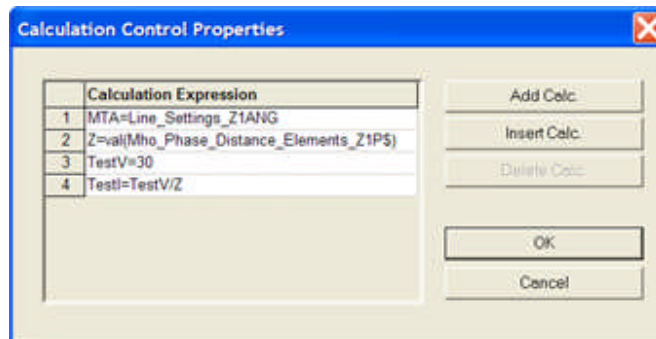




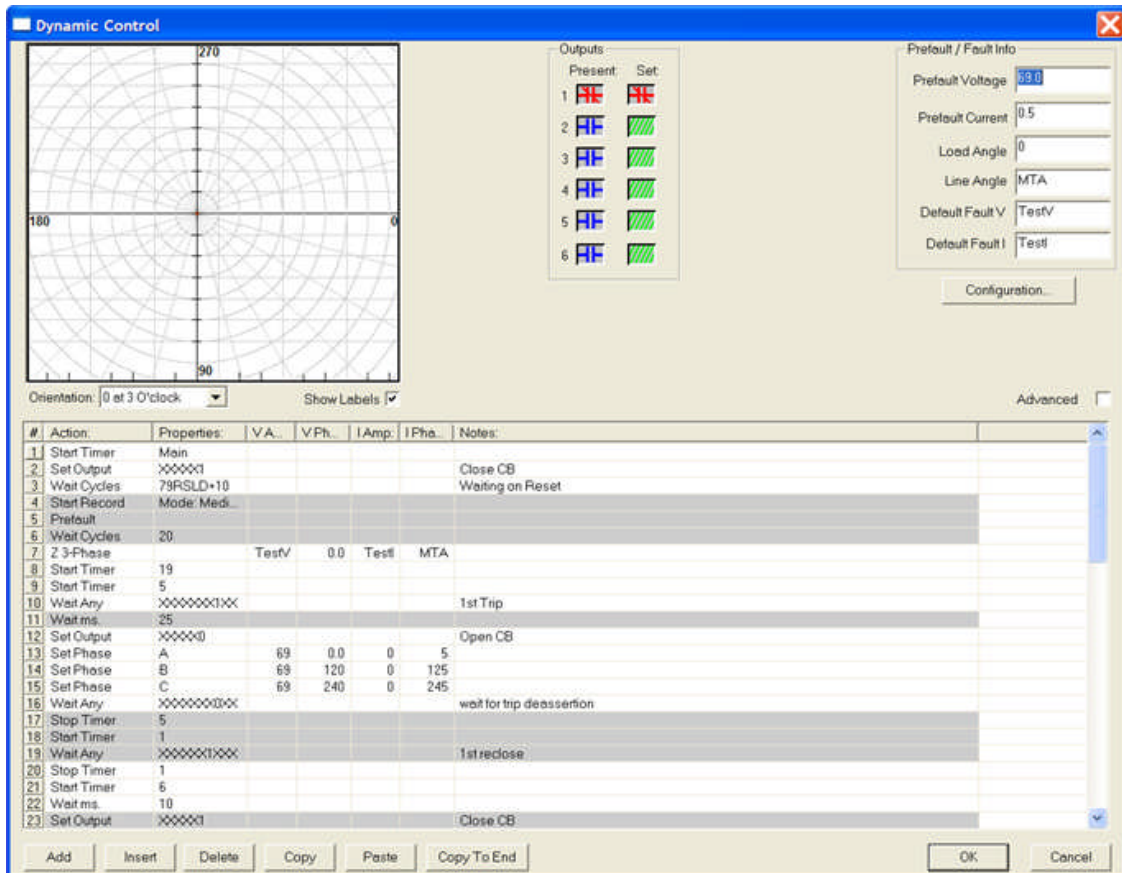
The *Battery Simulator* is set for the relay control Voltage and turned ON.



The *Calculation Control*



The *Dynamic Control* is set to agree with the requirements of the relay test and applies the following steps:



**Setup**

- #1 **Start the Main Timer** to obtain all increments of time activity
- #2 **Set Output** to provide simulated breaker **close** operation to the relay
- #3 **Wait Cycles** to allow the relay to reset internal logic to normal operation
- #4 **Start the Recorder** to record all timed activity and track all test set analog outputs

**Prefault**

- #5 **Apply the Prefault** values defined in the Prefault / Fault Info check blocks
- #6 **Wait Cycles** to record an amount of Prefault

**First Fault and Reclose**

- #7 **Apply Z 3-Phase Fault** defined by the Variables set in the **Calculation Control**
- #8 **Start Virtual Timer 19** to record total time for all Faults and Close operations
- #9 **Start Virtual Timer 5** to record total time for first Fault and Trip
- #10 **Wait Any Input** that indicates first relay trip response to the applied Fault
- #11 **Wait ms** (milliseconds) to simulate breaker response time
- #12 **Set Output** to simulate **breaker trip** operation
- #13 **Set Phase A to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #14 **Set Phase B to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #15 **Set Phase C to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault

- #16 **Wait Any Input** that indicates relay de-asserted trip logic
- #17 **Stop Virtual Timer 5** to record
- #18 **Start Virtual Timer 1** to record total time delay for first Reclose
- #19 **Wait Any Input** that indicates relay has provided for **first reclose**
- #20 **Stop Virtual Timer 1** to record
- #21 **Start Virtual Timer 6** to record total time for second Fault and Trip
- #22 **Wait ms** (milliseconds) to simulate breaker close time
- #23 **Set Output** to simulate **breaker close** operation

#### Second Fault and Reclose

- #24 **Apply Z 3-Phase Fault** defined by the Variables set in the **Calculation Control**
- #25 **Wait Any Input** that indicates second relay trip response to the applied Fault
- #26 **Wait ms** (milliseconds) to simulate breaker response time
- #27 **Set Output** to simulate **breaker trip** operation
- #28 **Set Phase A to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #29 **Set Phase B to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #30 **Set Phase C to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #31 **Wait Any Input** that indicates relay de-asserted trip logic
- #32 **Stop Virtual Timer 6** to record
- #33 **Start Virtual Timer 2** to record total time delay for Second Reclose
- #34 **Wait Any Input** that indicates relay has provided for **second reclose**
- #35 **Stop Virtual Timer 2** to record
- #36 **Start Virtual Timer 7** to record total time for third Fault and Trip
- #37 **Wait ms** (milliseconds) to simulate breaker response time
- #38 **Set Output** to simulate **breaker close** operation

#### Third Fault and Reclose

- #39 **Apply Z Single Phase Fault** defined by the Variables set in the **Calculation Control**
- #40 **Wait Any Input** that indicates third relay trip response to the applied Fault
- #41 **Wait ms** (milliseconds) to simulate breaker response time
- #42 **Set Output** to simulate **breaker trip** operation
- #43 **Set Phase A to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #44 **Set Phase B to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #45 **Set Phase C to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault
- #46 **Wait Any Input** that indicates relay de-asserted trip logic
- #47 **Stop Virtual Timer 7** to record
- #48 **Start Virtual Timer 3** to record total time delay for third Reclose
- #49 **Wait Any Input** that indicates relay has provided for **third reclose**
- #50 **Stop Virtual Timer 3** to record
- #51 **Start Virtual Timer 8** to record total time for fourth Fault and Trip

#52 **Wait ms** (milliseconds) to simulate breaker response time

#53 **Set Output** to simulate **breaker close** operation

**Fourth Fault and Lockout**

#54 **Apply Z 3-Phase Fault** defined by the Variables set in the **Calculation Control**

#55 **Wait All Input** that indicates third relay trip response to the applied Fault

#56 **Wait ms** (milliseconds) to simulate breaker response time

#57 **Set Output** to simulate **breaker trip** operation

#58 **Set Phase A to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault

#59 **Set Phase B to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault

#60 **Set Phase C to Prefault** Voltages and Zero Currents for the relay to recognize clearance of the applied Fault

#61 **Stop Virtual Timer** 19 to record

#62 **Stop Virtual Timer** 8 to record

**Close Recording**

#63 **Wait Cycles**

#64 **Zero Vector**

#65 **Wait Cycles**

#66 **Stop Record**

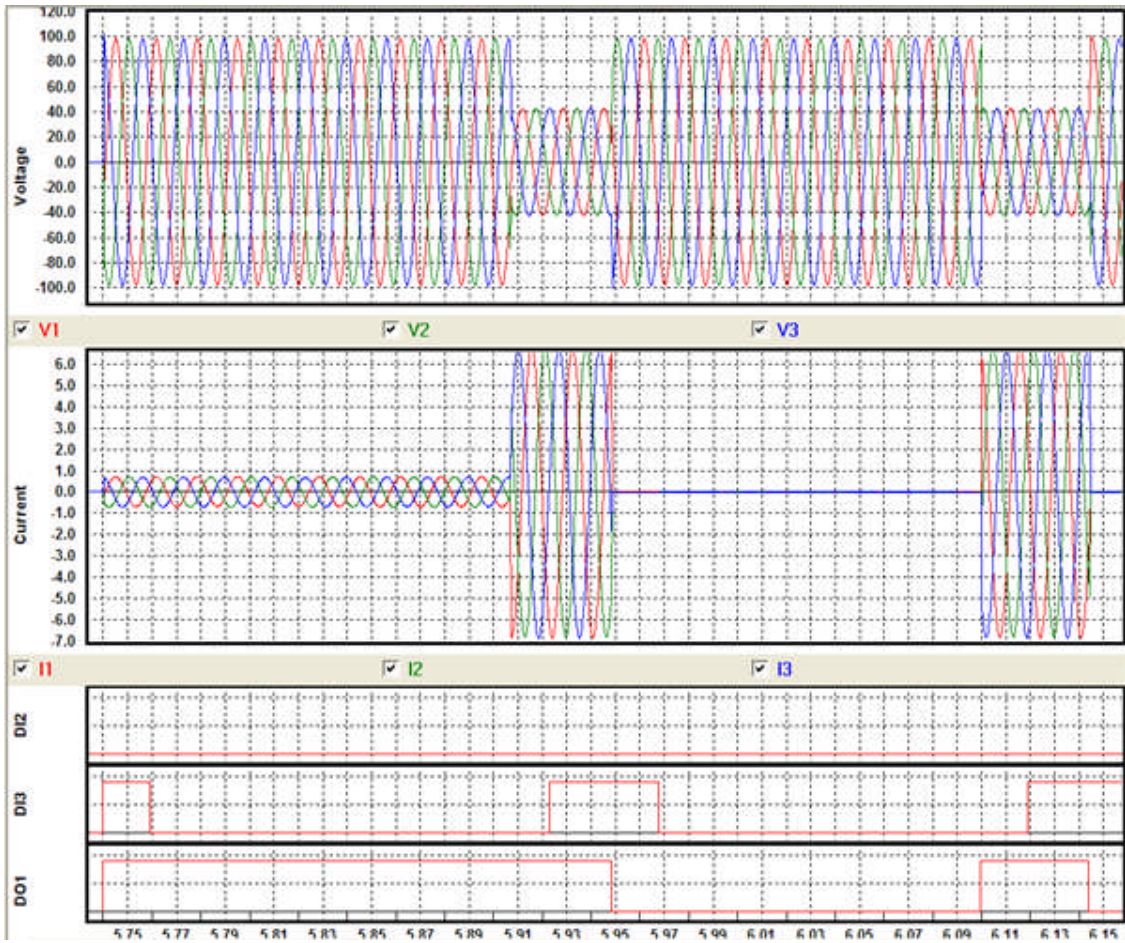
#67 **Stop Main Timer**

Note: The MPRT will remain with all outputs ON while the Dynamic Data is downloaded to the Capture tab. Be patient; this takes a little while. This record may take 15-20 minutes to download via RS-232. Download is much faster using the Ethernet channel.

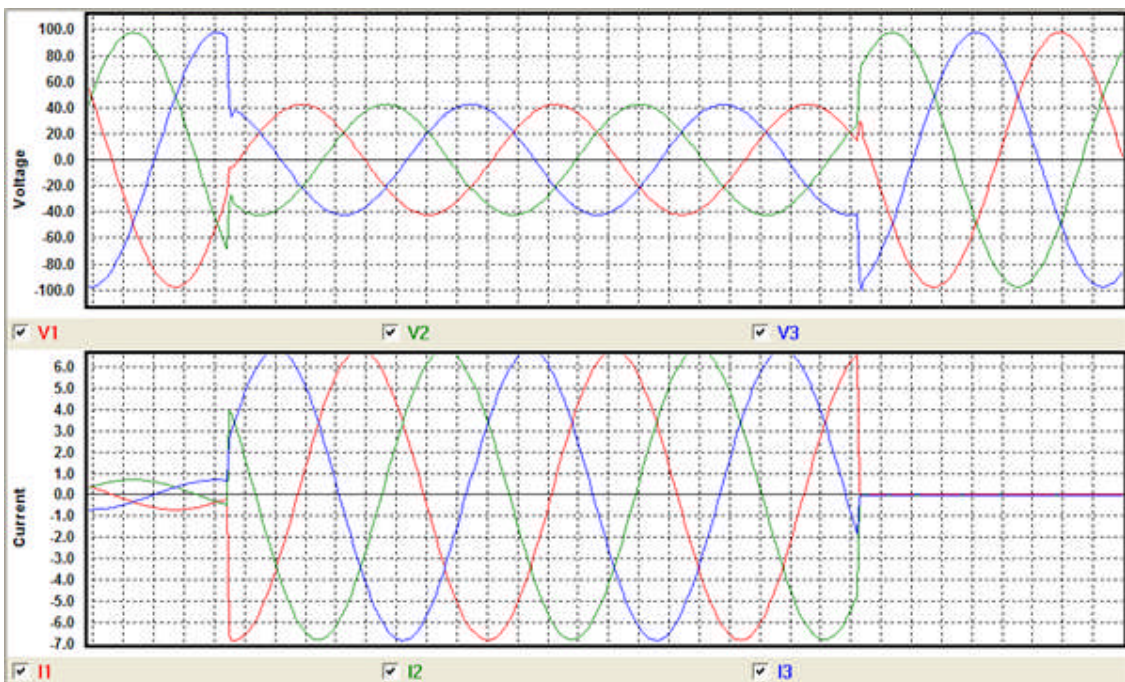
The *Test Result* from the Log Control displays the elapsed time for each of the Trip and Reclose events in cycles and the total sum in seconds.

Tr1	Rec1	Tr2	Rec2	Tr3	Rec3	Tr4	Sum	Reset_L0	Elapsed
VT5	VT1	VT6	VT2	VT7	VT3	VT8	Cycles	VT19	Seconds
3.654	7.320	4.176	902.418	4.446	2102.02	3.024	3027.066	3027.078	55.852

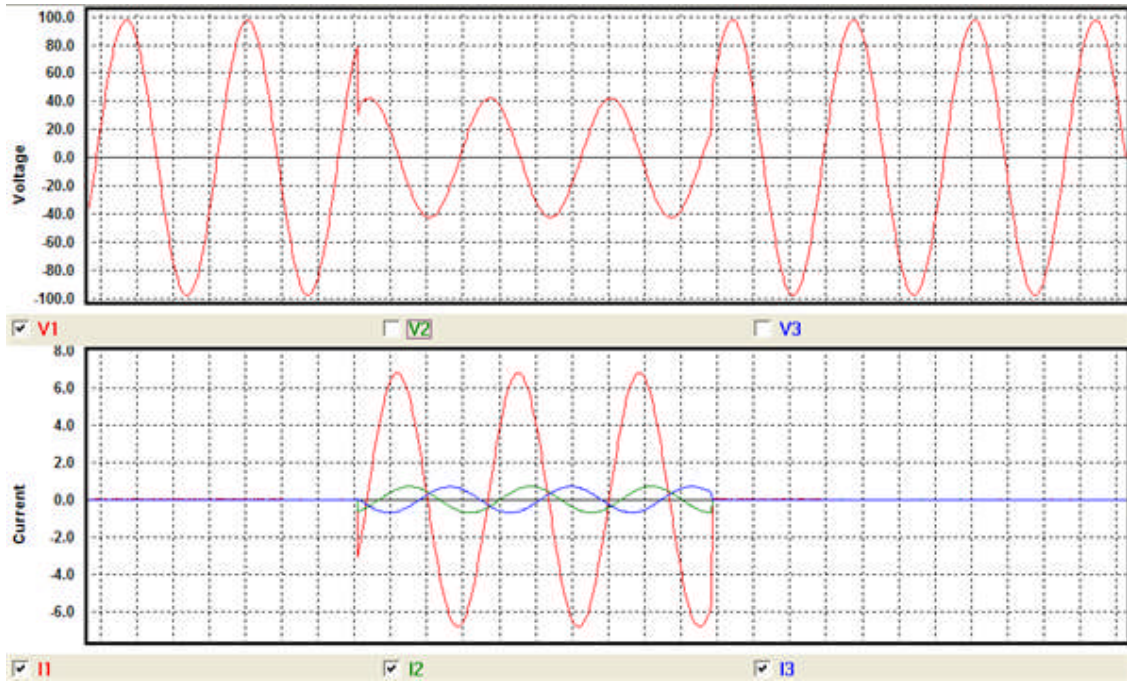
The *Capture Screen* recorded Analog and Digital signals are shown in Zoom for a segment to demonstrate clarity with Time Scale at bottom.



Capture Screen Zoomed with Analog displayed.



*Capture Screen* Zoomed with one Analog displayed.



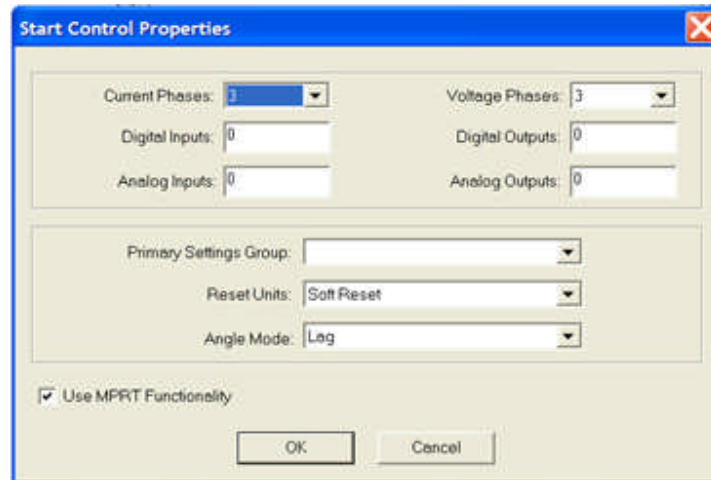
## Dynamic Control Example 2

The Dynamic Control Example 2 has a simple procedure to demonstrate a fault with an exponential component.

The *test procedure* only requires a Start Control, Dynamic Control and the End Control. No logging of data is used.



The *Start Control* only requires the entry of the use of three Current and Three Voltage channels.



The *Dynamic Control* properties have a sequence as follows:

#	Action:	Properties:	Wav...	VA...	VPh...	VFr...	VWF:	I(Amp)	I Pha...	I Freq	I WF:	IDCA...	IDCT...	VDCA...	VDC...
1	No Operation														
2	Start Recorder	Mode: Medi...													
3	Start Timer	Main													
4	Wait Cycles:	5.0													
5	Prefault														
6	Wait Cycles:	5.0													
7	Z Single Phase	A	1	30	0.0	Freq	Sin	10	65	Freq	Sin	6	60	30	60
8	Wait Cycles:	80.0													
9	Zero Vector														
10	Wait Cycles:	5.0													
11	Stop Timer	Main													
12	Stop Recorder														

Start Recorder

Apply Prefault for 5 cycles

Apply a Single Phase Fault

    Include an exponential offset to the Fault

    Exponential Time Constant equal to 60 milliseconds

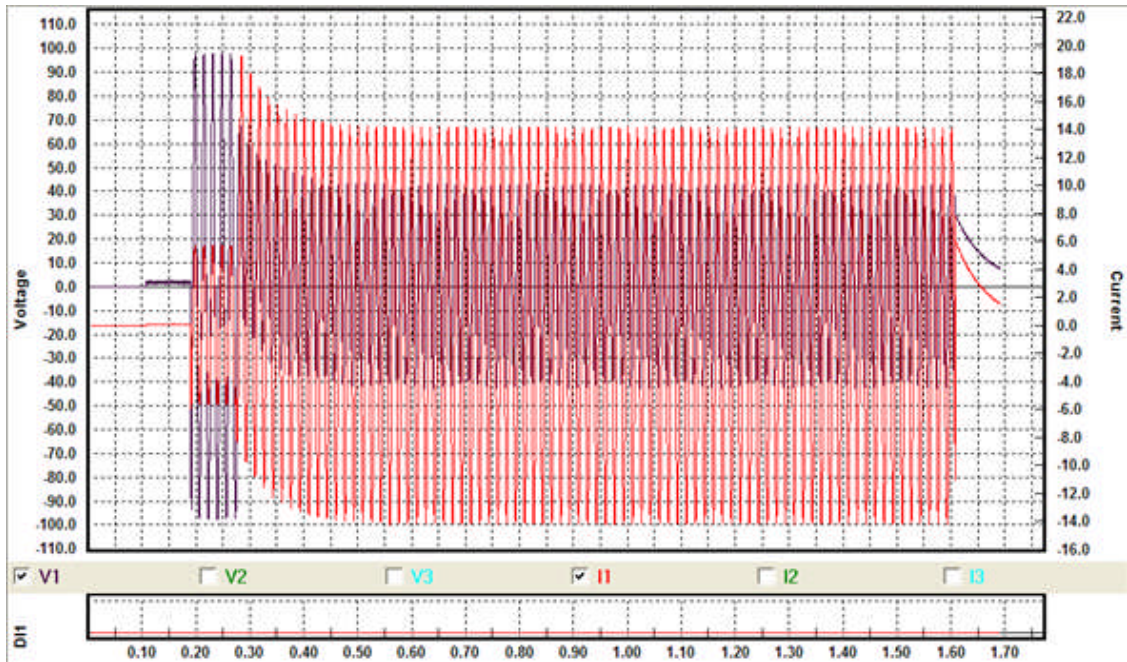
Apply Fault for 80 cycles

Set MPRT outputs to zero amplitude using the Zero Vector

Stop Recorder

**Note:** The MPRT will remain with all outputs ON while the Dynamic Data is downloaded to the Capture tab. Be patient; this takes a little while.

The *Capture tab* displays the resultant fault record

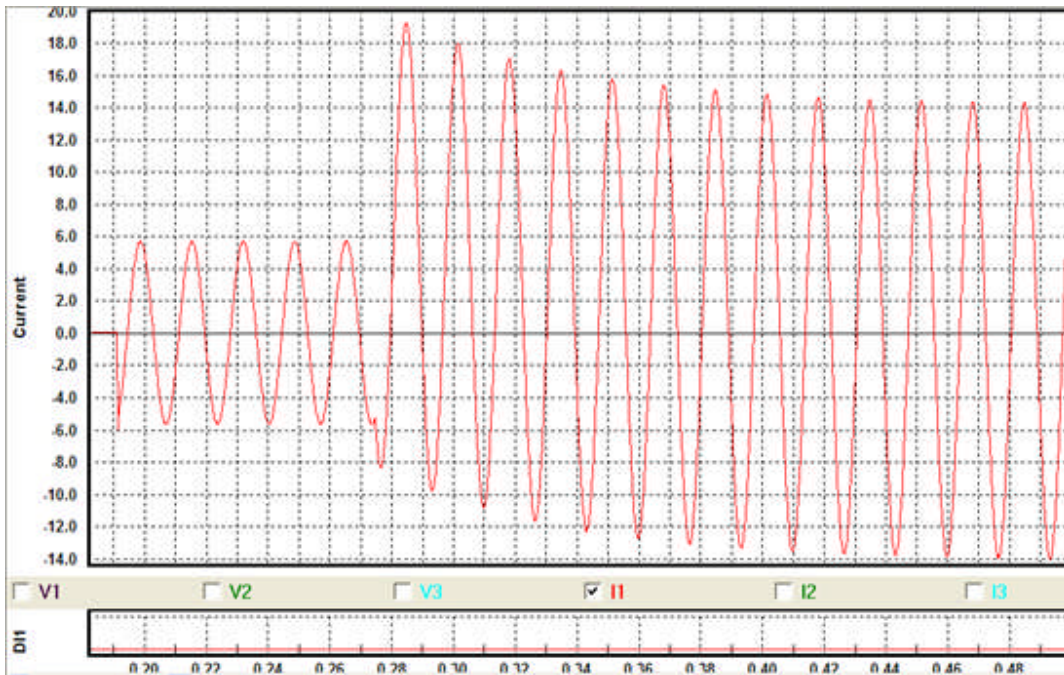


The non-faulted phases have been unchecked for clarity of the fault capture

The A phase current is displayed in **Red**

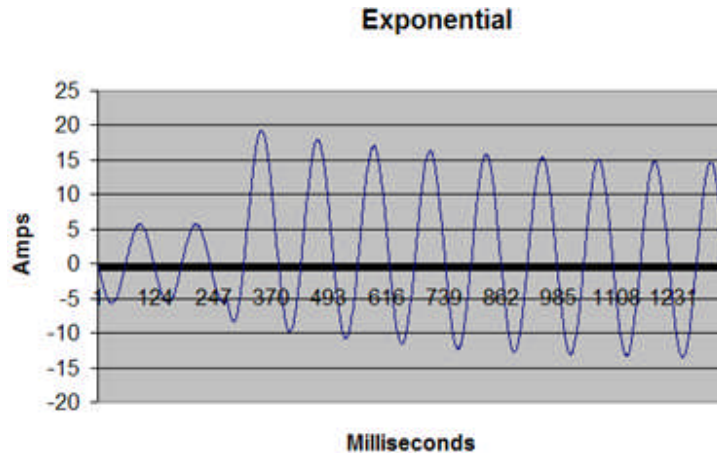
The A phase voltage is displayed in **Black**

A *Capture Zoom* view displays the Prefault and Exponential in clarity



An *Excel Graph* may be created with the Saved Excel \*.csv file of data can be used in part or all.





Add Harmonics to the Faulted Phase by adding *additional Sequence Steps*.

5	Prefault															
6	Wait Cycles:	5.0														
7	Z Single Phase	A	1	30	0.0	Freq	Sin	6	65	Freq	Sin	6	80	30	80	
8	Wait Cycles:	3.0														
9	Z Single Phase	A	2	15	0.0	180	Sin	1	65	180	Sin					
10	Wait Cycles:	6.0														
11	Z Single Phase	A	3	5	0.0	120	Sin	2	65	120	Sin					
12	Wait Cycles:	80.0														
13	Zero Vector															
14	Wait Cycles:	5.0														

The Sequence step includes Wave Sum: 2 to have

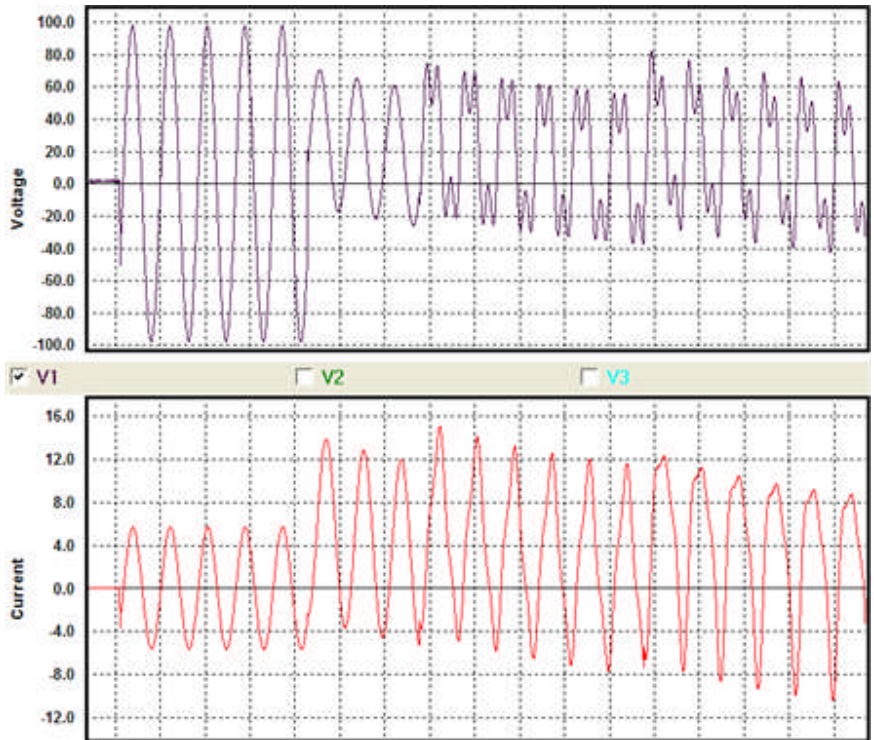
15 Volts of 180 Hz and 1 Ampere of 180 Hz added 3 cycles after the fundamental Fault starts for the remainder of the Fault

The Sequence step includes Wave Sum: 3 to have

5 Volts of 120 Hz and 2 Amperes of 120 Hz added 6 cycles for the remainder of the Fault

**Note:** The example is for all harmonics to apply to the same phase Single Phase Fault. Other phases can be chosen instead in addition to mixing the Fault types.

The *Capture tab* displays the Fault with Exponential and Harmonics added.



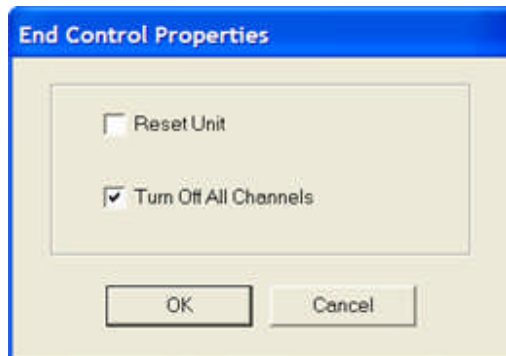
## END Control

The **End Control** is the test termination. Multiple End Controls may exist in a test, any of which first executed will end the test execution.

An example of this Control is demonstrated in Test Editor OC Time Example.



The *END Control Properties* has two available options which produce similar results.

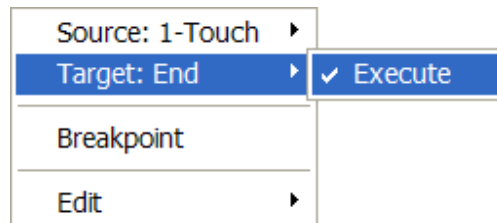


The first is to place the test set into a "soft reset", voiding any stored values and setting the amplifiers to the default OFF setting.

**Note:** A soft reset will not open any binary outputs that were closed during the test execution. To reset binary outputs, set the Start Control to provide a Hard Reset.

The default option is to only Turn Off All Channels, which does not produce a complete reset of the test set.

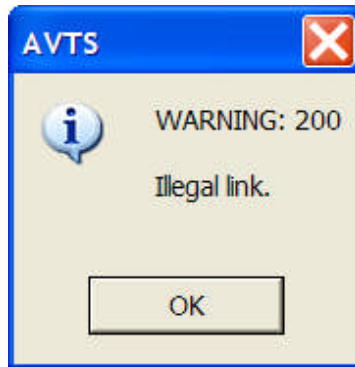
The *Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

The End Control has no **Source Link**. It can only be a **Target Link** for all other Controls.

**Note:** An attempt to link the End Control as a Source to another Control will produce the *Illegal Link* dialog.



## GET Control

The **GET Control**, for General Execute Trigger is the IEEE-488 interface for paralleling multiple devices to receive a trigger simultaneously.

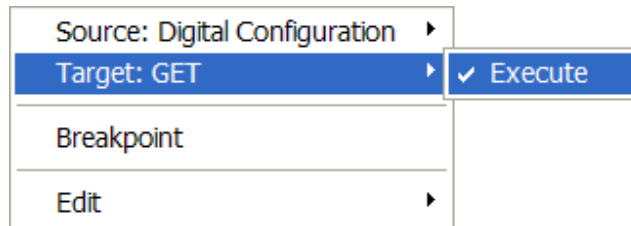
The GET is the only software trigger source for the TCM-915S test unit. The TCM may also be triggered by contact or voltage inputs created with AVTS and MPRT.



*GET Control Properties* displays the dialog that there are no properties for this Control.

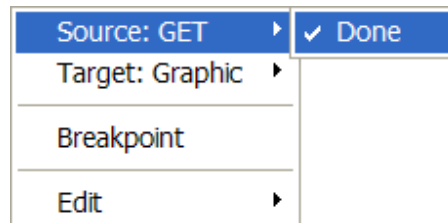


*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected. Trigger signal is immediately and simultaneously applied to all devices connected to the IEEE-488 bus.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Graphic Control

## Graphic Control

The **Graphic Control** is used to define the appearance of and the graph material displayed in the Workbook Test tab graphic pane. Typical graphs are Impedance or Time-Amplitude and are usually a product of the Theoretical Control. The graph characteristics may also be generated, for display by the Graphic Control, by using formulas to reside in the Calculation or the BASIC controls. Only one Graphic Control is allowed in the test, requires no Links and is usually used without links.

The Digital Graphic Control may be used in tandem to display the time sequence operations of the Outputs and Inputs.



An example of this Control is demonstrated in Test Editor OC Time Example and in Test Editor Charac Z Example.

*Graphic Control Properties* are for selecting the variables to be used in the graph. The variables must exist in the Variables list of the Log Control. A Log Control must be present to use the Graphic Control.

**Graphic Control Properties**

Scale Type: **Linear**

Traces

	X	Y	Trace Label	Mode	Scale Group	Shape	Color
1				Line	Scale Group 1	Circle	Blue
2				Line	Scale Group 1	Circle	Green
3				Line	Scale Group 1	Circle	Cyan
4				Line	Scale Group 1	Circle	Red
5				Line	Scale Group 1	Circle	Magenta
6				Line	Scale Group 1	Circle	Yellow
7				Line	Scale Group 1	Circle	Dark Blue
8				Line	Scale Group 1	Circle	Dark Green
9				Line	Scale Group 1	Circle	Teal
10				Line	Scale Group 1	Circle	Dark Red

Group Test Overlays

1: None  
2: None  
3: None  
4: None  
5: None

Report Options

Color  Monochrome  Large  Small

Graph Title:   
X-Axis Label:   
Y-Axis Label:

Save Image  
 Auto Scale  
 Plot Moving Characteristic  
 Plot Theoretical Characteristic  
Theoretical Characteristic:    
 Min/Max Theoretical:  
Min/Max Theoretical Characteristic:

OK Cancel

The Graphic Control Properties dialog is divided into blocks for

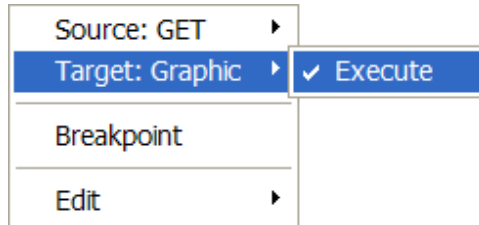
**Scale Type:** drop list described in the Graphic Control Scale Type topic

Traces block described in the Graphic Control Traces topic

**Group Test Overlays** block described in the Graphic Control Group Test Overlays topic

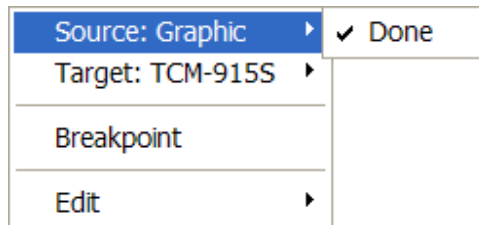
**Graph Properties** block described in the Graphic Control Graph Properties topic  
**Report Options** block described in the Graphic Control Report Options topic

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.

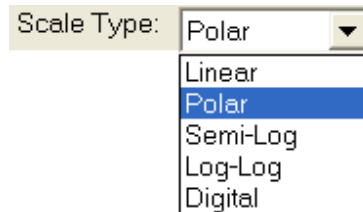


The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Graphic Control Scale Type

The Scale Type options are to accommodate the display of the Theoretical Object constructed in the Relay Test.

*Scale Type* drops a list of the type graphs to be displayed. The options are Linear, Polar, Semi-Log, Log-Log and Digital.



**Linear** will display a graph representative of typical graph paper with a constant size square representing each cell of the graph.

**Polar** is the display normally used for impedance characteristic graphs. The graph has 0 and 90 degree axes with radials about the origin at constant intervals.

**Semi-Log** is typically used for time over/under current/voltage characteristic graphs with either of the X or Y axes represented in linear format and the other in a logarithmic scale.

**Log-Log** typically used for time over/under current/voltage characteristic graphs with both of the X or Y axes represented in logarithmic scales.

**Digital** graphs are to display the active and non-active states of contacts and or voltages being monitored. When selected, the dialog is altered to that of the Digital Graphic Control.



## Graphic Control Traces

The **Graphic Control Traces** block contains drop lists and edit fields arranged in columns for each of ten rows.

Traces							
	Magnitude	Phase	Trace Label	Mode	Scale Group	Shape	Color
1	TestZ	TestAngle	Zone 3 Plot Points	Scatter	Scale Group 1	Diamond	Cyan
2				Line	Scale Group 1	Circle	Green
3				Line	Scale Group 1	Circle	Cyan
4				Line	Scale Group 1	Circle	Red
5				Line	Scale Group 1	Circle	Magenta
6				Line	Scale Group 1	Circle	Yellow
7				Line	Scale Group 1	Circle	Blue
8				Line	Scale Group 1	Circle	Green
9				Line	Scale Group 1	Circle	Teal
10				Line	Scale Group 1	Circle	Dark Red

**X** is a variable for the horizontal variable used in the graph. Left click on the right side of the X edit field to drop down a list of the variables identified in the Log Control.

**Y** is a variable for the vertical variable used in the graph. Left click on the right side of the Y edit field to drop down a list of the variables identified in the Log Control.

**Trace Label** is a text entry for identifying a label for

**Scale Group** options in a drop list are Scale Group 1, Scale Group 2 and Scale Group 3.

**Shape** will display a drop list of options to show the test result points on the graph. The options are Circle, Diamond, Square and None. The selection defines the object.

**Color** will display a palette of 16 colors to select for the color of the graph to be plotted from the test results.

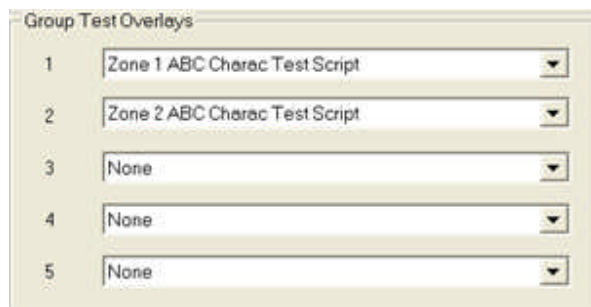
## Graphic Control Group Test Overlays

The **Group Test Overlays** box allows one or more up to five graphs to be displayed on the graph along with the Theoretical Object graph created with this Control.

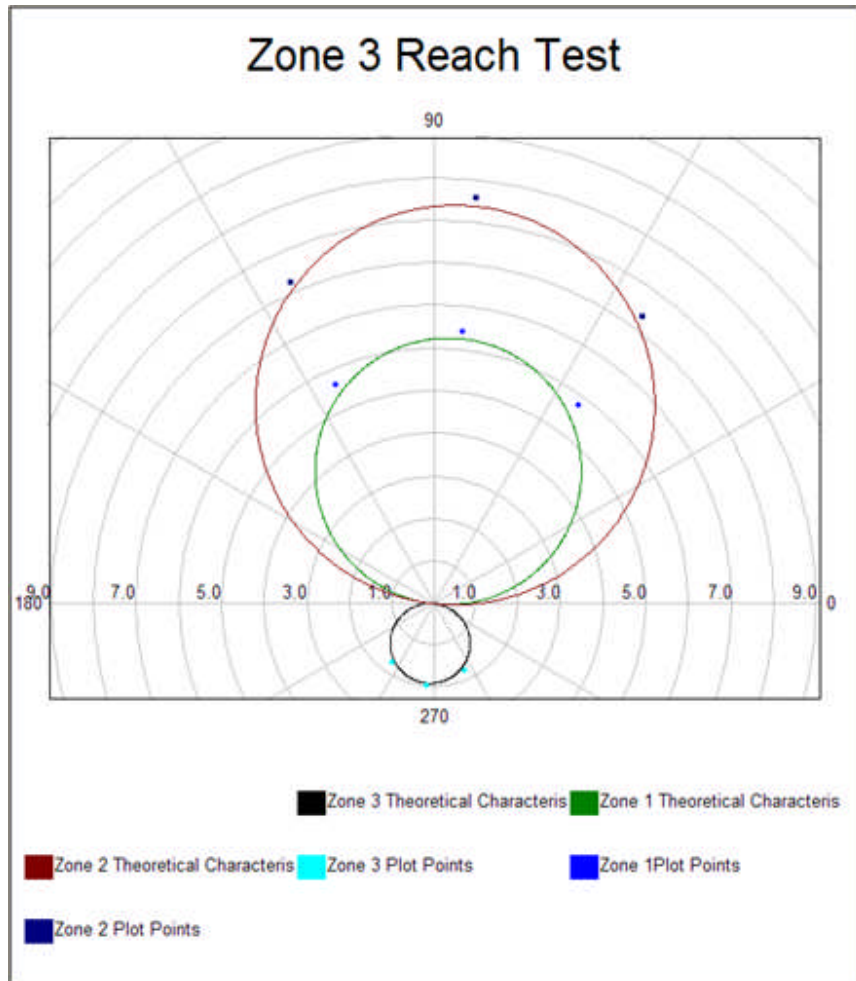
**Note:** Digital Graphs may also be displayed along side the analog graph, but not overlaid.

The drop list for the **Group Test Overlays**, allows for selecting from one to five other test graphics results to be overlaid with the graphic results of the execution of the active test. The drop list for each Group contains the names of all the tests associated with the open relay in the same order as the list of test names in the Workspace.

**Example:** For a multi-zone relay and after having completed the tests for other zones, the other zones may accompany in overlay the zone graph currently being tested. *Zone1 Zone2 Zone3 Group Overlay* settings example for a Zone 3 test.



The "other" zones will be available for selection in the drop down lists for each overlay. *Zone1 Zone2 Z3 Overlay* test result example.



## Graphic Control Graph Properties

The *Graph Properties* block contains the edit fields and check blocks to define the test characteristic image.

The screenshot shows a dialog box titled 'Graph Properties'. At the top, there is a text field labeled 'Graph Title' containing the text 'Zone 3 Reach Test'. Below this, there are three checked checkboxes: 'Save Image', 'Auto Scale', and 'Plot Moving Characteristic'. Further down, there is a checked checkbox labeled 'Plot Theoretical Characteristic' followed by a text field containing 'Zone 3 Theoretical Characteristic' and a small black square color selection block. At the bottom, there is an unchecked checkbox labeled 'Min/Max Theoretical' followed by a text field containing 'Min/Max Theoretical Characteristic' and a small green square color selection block.

The **Graph Title**: edit field is to enter a name for the Graph.

The **X-Axis Label**: edit field is to enter a name for the Graph horizontal axis. Option available only in Scale Type = Linear and Digital.

The **Y-Axis Label**: edit field is to enter a name for the Graph vertical axis. Option available only in Scale Type = Linear.

The **Save Image** check box is to enable the graph image to be placed in the test results for future call.

The **Auto Scale** check box is to allow a change of scale during the test when the plot exceeds the initial boundaries.

If unchecked, the additional columns X Min Value, X Max Value, Y Min Value and Y Max Value are added between the Mode and Scatter columns. These are for entering the desired values for the axes values. The plot will have a constant scale within the X and Y values entered.

**Note:** The Auto Scale must be unchecked, the X and Y boundaries set to a value to display the TheoInstOp in the Test tab Graph Tile. Once set, the Auto Scale may be rechecked and the set boundaries will be retained for the plot.

The **Plot Moving Characteristic** check box is to allow the plot of a varying test parameter.

Example: The time axis for a timing test in an Amplitude Time plot will expand its boundary as the elapsed time continues longer than the initial bounds.

The **Plot Theoretical Characteristic** edit field is to enter the name for the graphic model to which the device under test should accurately respond. A check box is present to select whether or not the Characteristic will be displayed. Click on the color block to evoke the Color selection palate.

The **Min/Max Theoretical** edit field is to enter the name for the minimum and maximum bands of tolerance for the Theoretical Characteristic plotted. A check box is present to select whether or not the Characteristic will be displayed. Click on the color block to evoke the Color selection palate.

## Graphic Control Report Options

The *Report Options* block allows for selection of the graph to be displayed in the test results report to have the colors rendered in the Graphic Control properties and whether or not the graph will be presented in a Large (default) or Small format.



The image shows a dialog box titled "Report Options" with a light beige background. It contains four radio button options arranged in two columns. The left column has "Color" (selected) and "Monochrome". The right column has "Large" (selected) and "Small".

Option	Selected
Color	Yes
Monochrome	No
Large	Yes
Small	No

## Horn Control

The **Horn Control** will operate a MPRT test unit audible horn when placed in a test procedure. The function is bypassed if the selected test unit is PULSAR. A use is to alarm the user should a critical amplifier value be called or time limit reached. This option will not function in Simulate Device mode.

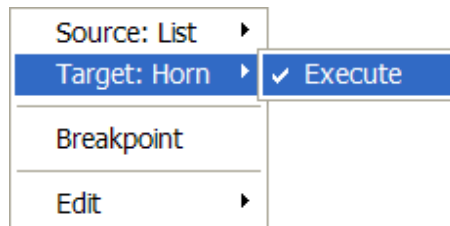
**Note:** The Horn Control is for use with MPRT only.



*Horn Control Properties* include a drop list to select the appropriate connected test unit. A volume control slide button exists to adjust from minimum at left end to maximum at right end. An integer only time value in seconds may be entered in the Seconds edit field.

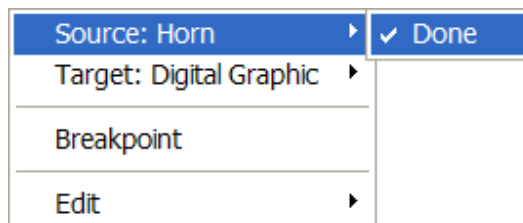


*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## HW Timer Control

### HW Timer Control

The **Hardware Timer Control** is to control the test unit time function. It allows a timer to start and continue until a monitored function occurs. Should a Max Time to Operate limit occur, the process will abort.



An example of this Control is demonstrated in Test Editor OC Time Example.

The hardware timer properties are described in the HW Timer Properties topic.

The hardware timer links properties are described in the HW Timer Links topic.

## HW Timer Properties

*HW Timer Control Properties* dialog is divided into three blocks of settings.

The top block contains:

**Unit:** drop list for selecting the test set to be controlled, the default = TestSetA;

**Mode:** drop list for selecting the time to be Seconds, 50-Hz Cycles or 60-Hz Cycles, default = Seconds

**PULSAR Auxiliary Contact:** drop list for selecting the action for No Action or to Open or to Close, default = No Action

**The Debounce Time:** edit field is to enter the time in milliseconds to delay time record of contact or voltage response for valid recognition. The minimum to maximum recognized times for PULSAR are 0.1 to 1638.3 milliseconds. The default value is 1.7 milliseconds.

**Note:** The debounce time will be subtracted from the total elapsed time of the test.

**The Time Variable:** edit field is dimmed by default and contains the variable name PulsarTimeA. This variable is required for software recognition of the action time and must be placed in the Log Control Variables list if the elapsed time is to be displayed in a moving graphic.

The **Start** block contains:

**Post:** drop list for selecting the time start function to be by PULSAR post 1-3 or MPRT posts 1-15 or by software upon entry to the HW Timer Control. Default is Software.



**Note:** If there has been a Start Timer function set active prior to the execution of the HW Timer, there will be a time inaccuracy. There will be the additional time due to the software transition from the Control that started the timer and the initiation of the HW Timer Control.

**Note:** Software timer start cannot be used for a Sync-Test.

**Mode:** drop list for selecting the time start Post activity for Continuity or Voltage. The Mode: is dimmed for Post: = Software. Default is Continuity.

**Active On:** drop list for selecting the Mode: action for Applied, Changed or Removed of the Continuity or Voltage mode selected. The Active On: is dimmed for Post: = Software. Default is Applied.

The **Stop** block contains:

**Post:** drop list for setting the time stop function to be by PULSAR post 1-3 or MPRT posts 1-15 or to Sync VA and VB upon entry to the HW Timer Control. Default is 3.

**Note:** Software timer start cannot be used for a Sync-Test.

**Mode:** drop list for setting the time stop Post activity for Continuity or Voltage. The Mode: is dimmed for Post: = Software. Default is Continuity.

**Active On:** drop list for setting the Mode: action for Applied, Changed or Removed of the Continuity or Voltage mode selected. The Active On: is dimmed for Post: = Software. Default is Applied.

**Latch** check box. When checked, only the first valid occurrence of action will be recognized. Further post actions will be ignored.

**Enable Horn** check box. When checked, will cause an audible alarm from the test set upon Stop occurrence.

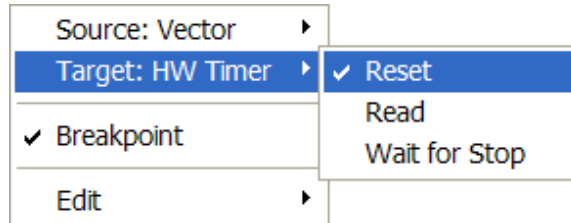
Turn Currents Off check box. When checked, will turn OFF all current channels of the active test set.

**Max Time to Operate:** edit field is to enter the amount of allowable time in Seconds for the device under test to respond to the Stop action.

**Note:** The entry may be a variable identified prior to entry to the HW Timer Control.

## HW Timer Links

*Target Links* are Reset, Read and Wait for Stop.

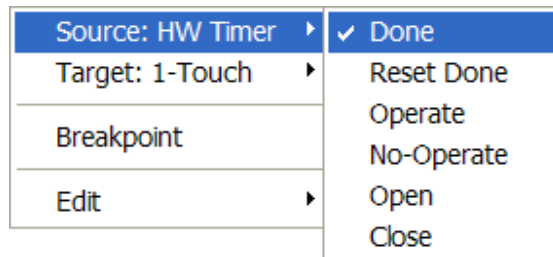


The Reset command sets the Timer function to zero.

The Read command registers the value of time present in the Timer upon entry to the Control.

The Wait for Stop command sets the Timer function to zero, starts the Timer and concludes timing upon a property setting.

*Source Links* are Done, Reset Done, Operate, No-Operate, Open and Close.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control. Done will also start the timer function and immediately pass to the next process Control without regard to any other conditions set in the Control; i.g., wait for a contact or voltage response to stop the timer.

The Reset Done command immediately passes the test procedure to the next Control after a Timer Reset. The elapsed time from a time Start to Stop action will be stored in the Time Variable PulsarTimeA.

The Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

The No-Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

**Note:** The non operation of a required response and the Control properties having reached their preset limits validates this branch and indicates an improper condition in which to abort the procedure.

The Open command is a branch option from the Control. It passes the Control properties either at the completion of an Open contact coming open during the Control execution or completely by-passes the Control if an Open contact is met at the beginning of the Control execution.

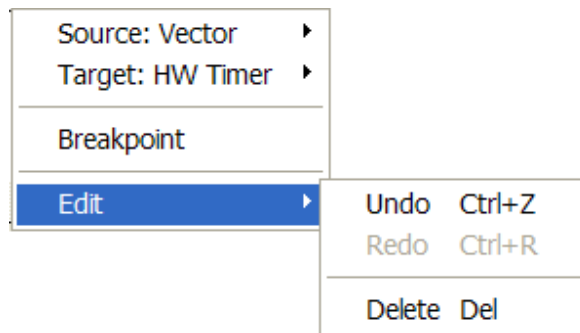
The Close command is a branch option from the Control. It passes the Control properties either at the completion of an Close contact coming open during the Control execution or completely by-passes the Control if an Closed contact is met at the beginning of the Control execution.

## Link Control

The **Link Control** is used to connect one Control to one or more Controls. Selecting the Link Control changes the mouse to a cross-hair. The process of creating a test sequence is to connect the Controls in a from-to sequence of order. When all Links have been placed, the choose the Select Control for additional use of the normal mouse. This linking process determines the procedure that the Controls will be made active for the test execution.

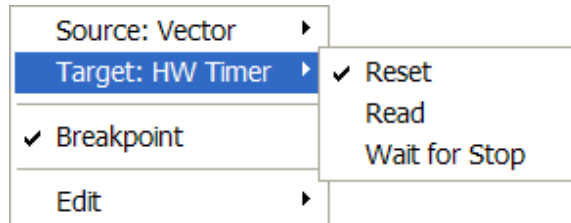


The *Link Control Properties* are made visible with a right mouse click on the Link after it has been placed in the Workbook between two or more Controls. The action produces a menu for the Link in the Workbook. The Edit options Undo, Redo and Delete are the standard Windows actions for the immediately previous choice for the Link.



The Link Control Target and Source properties are different for each Control. This example demonstrates the Link properties from a Vector Control to a HW Timer Control.

The *Target Links* for a linked HW Timer Control are Reset, Read and Wait for Stop.

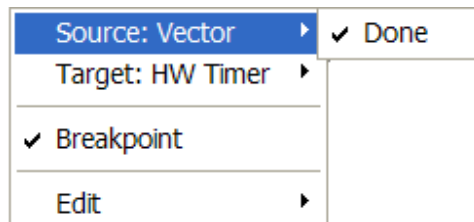


The Reset command sets the Timer function to zero.

The Read command registers the value of time present in the Timer upon entry to the Control.

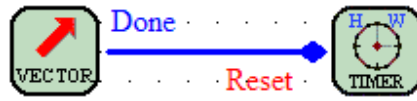
The Wait for Stop command sets the Timer function to zero, starts the Timer and concludes timing upon a property setting.

The *Source Link* for a linked Vector Control is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

The **Breakpoint** item will create a test execution halt at the Link for interaction by the user. Once enabled the Breakpoint menu option will have a check indicating it is active. Remove the Breakpoint from the test procedure by removing the check by Breakpoint in the menu.



Under test execution the test may be followed from Control to Control by placing the Test Editor tab in focus. When the breakpoint is encountered, the Step Over and Step Into main Relay Toolbar items allow to either enter the Control directly or to step over the Control. The Control encountered will determine if the Step Into or Step Over icon is enabled for selection.

See Relay Toolbar topic.

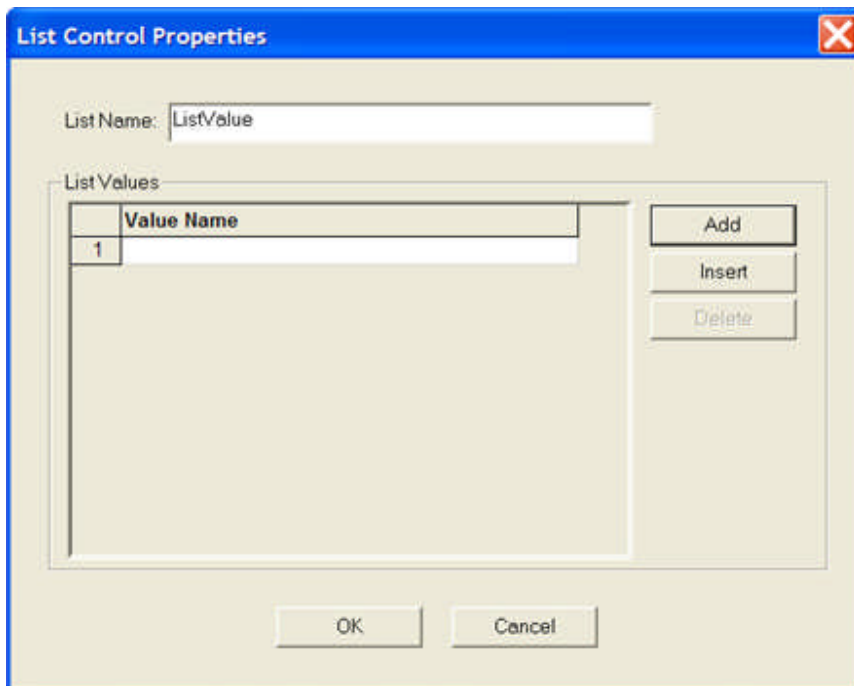
## List Control

The **List Control** is used in a looping function so that a number of passes of testing is performed identically but with different values entered into the Control. Upon completion of the list, the process is passed to the next Control out of the loop. The Control is handled similar to the Loop Control.



An example of this Control is demonstrated in Test Editor OC Time Example.

*List Control Properties* contains an edit field to enter a name for each of the Values to be passed from the List Control to the next Control in a configured loop; default Name: = ListValue.



The Values box may contain as many lines as desired to be passed to the loop action one at a time.

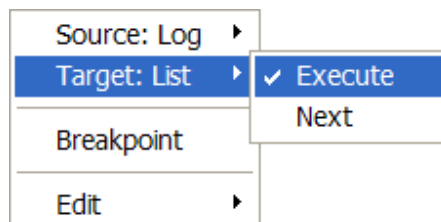
Click the Add button to add lines to the Values block.

The Insert button will place a new Value line above a line selected and highlighted.

the Delete button will remove a selected Value line.

**Note: Value Names may be variables initialized prior to entry to the List Control.**

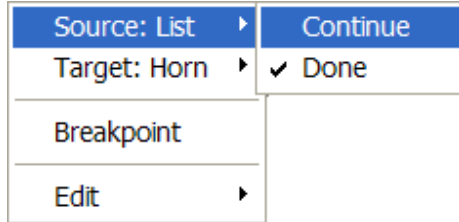
*Target Links* are Next and Execute.



The Next command initiates the next item of List or Loop Values in the properties box.

The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Links* are Continue and Done.



The Continue command registers the next List Value item and forwards the procedure to the linked Control. The completion of the List Values concludes the Loop Control.

The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Log Control

The **Log Control** governs the data stored as a test result for retrieval in a Test Report later. The items to be retained are contained in list of Variables documented in the Control and stored for later use in a test report.



An example of this Control is demonstrated in Test Editor OC Time Example.

*Log Control Properties* contains two blocks for data entry and check/edit boxes for data display control in a test result.

	Variable	Units	Decimal Places	Include in Report	Align
1	Ia		2	<input checked="" type="checkbox"/>	Center

**Variables to Save** block is to enter lines of variables to be saved in the reports of the active database. Each line must have a discrete Variable name in that column with no blank spaces. A blank variable may not be retained.

**Units** column is to enter a description of the Variable type; i.g., Variable = Ia, then Units = Amps. The Unit name may contain only 8 characters and spaces.

**Decimal Places** column is to enter the number of decimals for a floating point Variable.

**Include in Report** column contains a check box for each Variable. A checked box will place the Variable and its result value in a test report when called.

**Note:** Whether checked or not, all Variables results are saved in the active database.

**Align** column drop list, displayed by left click in the column for each Variable, selects Left, Center or Right for placement of the Variable name and data in the report column as selected.

**Width** column is to enter the number of spaces wide for the selected column.

**Note:** The column is visible only when the Evenly Spaced checked box is unchecked.

**Add Var.** button enters a new Variable line. Several new line may be added with successive clicks. Each line must contain a discrete name.

**Insert Var.** button enters a new Variable line above a selected variable.

**Delete Var.** button removes a selected Variable line from the list.

**Pre-Log Calculations** block is to enter Variable names not yet identified and formulas necessary to calculate percent error or other data. The entries will be the last opportunity to identify the Variables to be included in the test results.

**Add Calc.** button enters a new Calculation Expression line. Several new line may be added with successive clicks.

**Insert Calc.** button enters a new Calculation Expression line above a selected variable.

**Delete Calc.** button removes a selected Calculation Expression line from the list.

The bottom block of options are check boxes and one edit field.

The **Units at Top** option will place the Variable Names in a row in a test report. Test results will be placed in a line below the line of Variable Names. Select this option nulls the Units Beside option.

The **Units Beside** option will place the Variable Names in the same row and next to the results in a test report. Select this option nulls the Units at Top option.

**Evenly Spaced** check box spreads all test result columns with equal widths to reside in a full 76 character width. Un-checking the field will produce the Width column explained above in the Variables block. Using this option will allow user column widths entries of varying character widths.

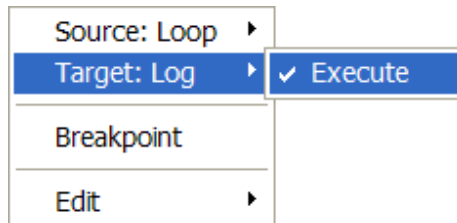
**Note:** More than 76 characters and blank spaces may exist in the total of all the variables listed in the Log Control.

If the number exceeds 76 characters and Evenly Spaced is selected, the Variable labels and the test results will be truncated to fit the 76 character limit in the results printout.

If the number of characters exceeds 76 and Evenly Spaced is un-checked, the Variable labels and the test results will be line wrapped with the results skewed to the columns.

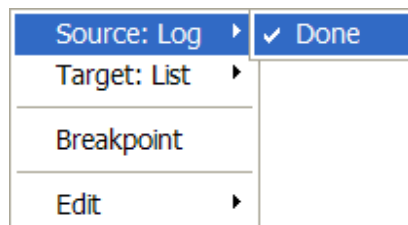
**Note:** Simulate the test and view the simulated test results to check the desired future print out.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.



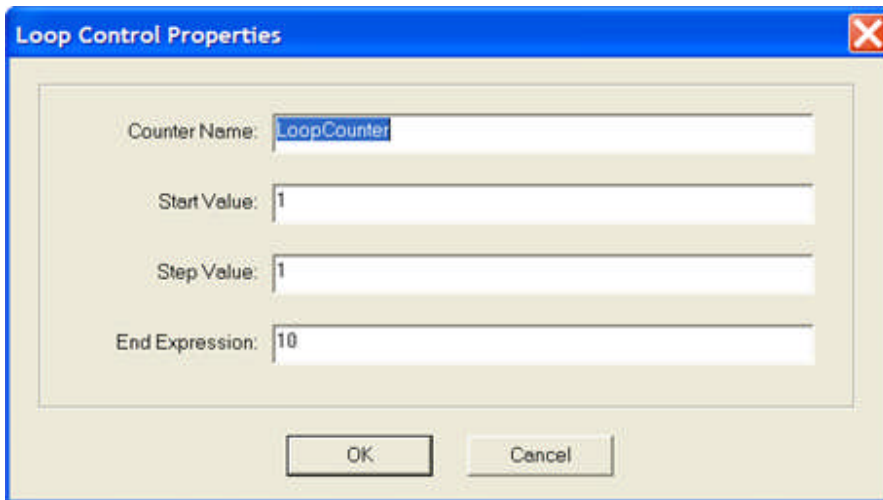
## Loop Control

The **Loop Control** is used in a looping function so that a number of passes of testing is performed identically but with different values entered into the Control. Upon completion of the Control settings, the process is passed to the next Control out of the loop. The Control is handled similar to the List Control.



An example of this Control is demonstrated in Test Editor Charac Z Example.

*Loop Control Properties* contain four edit fields.



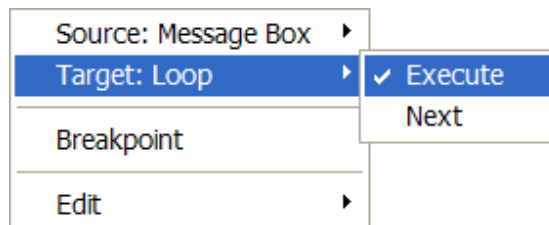
**Counter Name:** is to enter the Variable label that will be used in the following loop for the values determined by the Start Value:, Step Value: and End Expression.

**Start Value:** is to enter a numeric value or an expression to determine a Start Value based on previously identified variables.

**Step Value:** is to enter a numeric value or an expression to determine a Step Value, for each pass of the Loop until the End Expression is recognized, based on previously identified variables.

**End Expression:** is to enter a numeric value or an expression that will be used in the following loop to determine the end of Loop action at the Step Value: that meets the End Expression criterion.

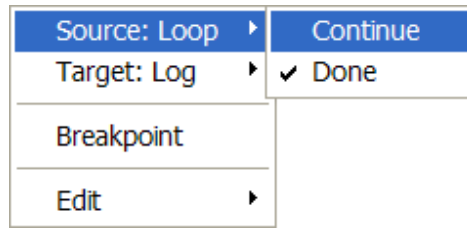
*Target Links* are Execute and Next.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Next command initiates the next item of List or Loop Values in the properties box.

*Source Links* are Continue and Done.



The Continue command registers the next List or Loop Value item and forwards the procedure to the linked Control. The completion of the List Values concludes the Loop Control.

The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Message Box Control

The **Message Box Control** is used to interrupt a process and produce a message, allow data entry, and make a condition decision.



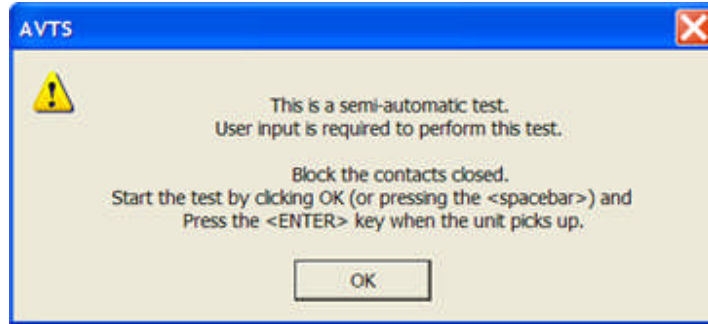
An example of this Control is demonstrated in Test Editor Charac Z Example 2.

*Message Box Control Properties* contains a drop list, an edit field, a Message entry field and a list of condition options available.

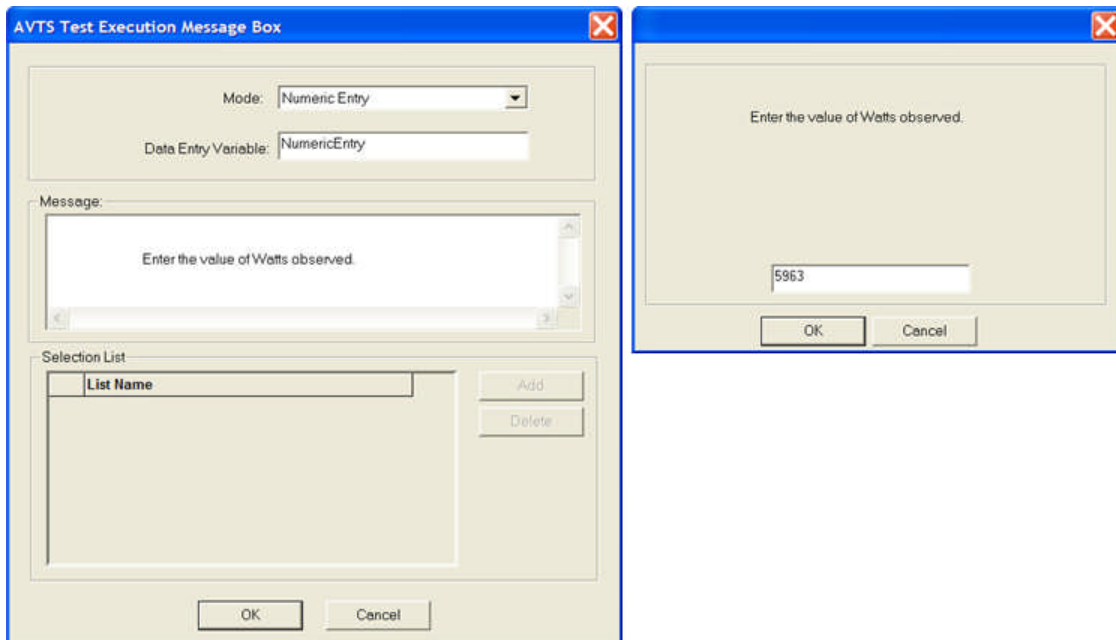
The screenshot shows the 'AVTS Test Execution Message Box' dialog. At the top, the title bar reads 'AVTS Test Execution Message Box' with a red 'X' close button. Below the title bar, there is a 'Mode' dropdown menu currently set to 'Message Only'. Underneath is a 'Data Entry Variable' text field containing the text 'DataEntry'. A large text area labeled 'Message:' is empty. Below that is a 'Selection List' section containing a table with one header row: 'List Name'. To the right of the table are 'Add' and 'Delete' buttons. At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Mode: drop list displays the Message Only, Numeric Entry, Text Entry, Yes/No Entry, Pass/Fail Entry and List Selection.

*Message Only* is to enter text to be displayed when the test process enters the Control. The Message will be displayed until operator action to exit is recognized by the keyboard. The Data Entry Variable: field is default "DataEntry" for the Message Only option.



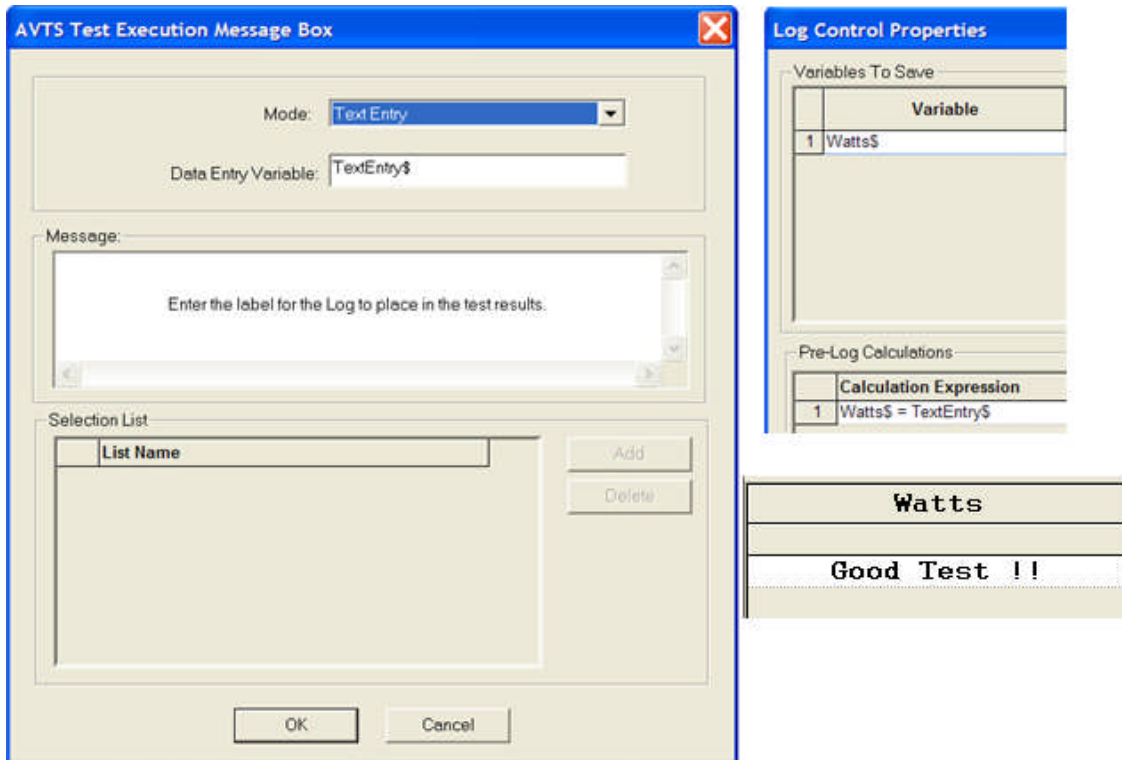
*Numeric Entry* is to input numeric data to the test result. Enter a value and select OK to place the value in memory for use by another Control or to Log into the test results. To place the Numeric Entry in the results, the Log Control must have the Variable Name listed. In this example, the Numeric Entry DataEntryVariable is NumericEntry. In the Log Control, a Pre-Log Calculation may be entered to set Watts = NumericEntry and the entry Watts placed in the Variable Names block. The value will then be logged in the test results; Watts = 5963.00.



**Note:** A short cut would be to place the label Watts in the DataEntryVariable edit field; then a Pre-Log Calculation would not be required.

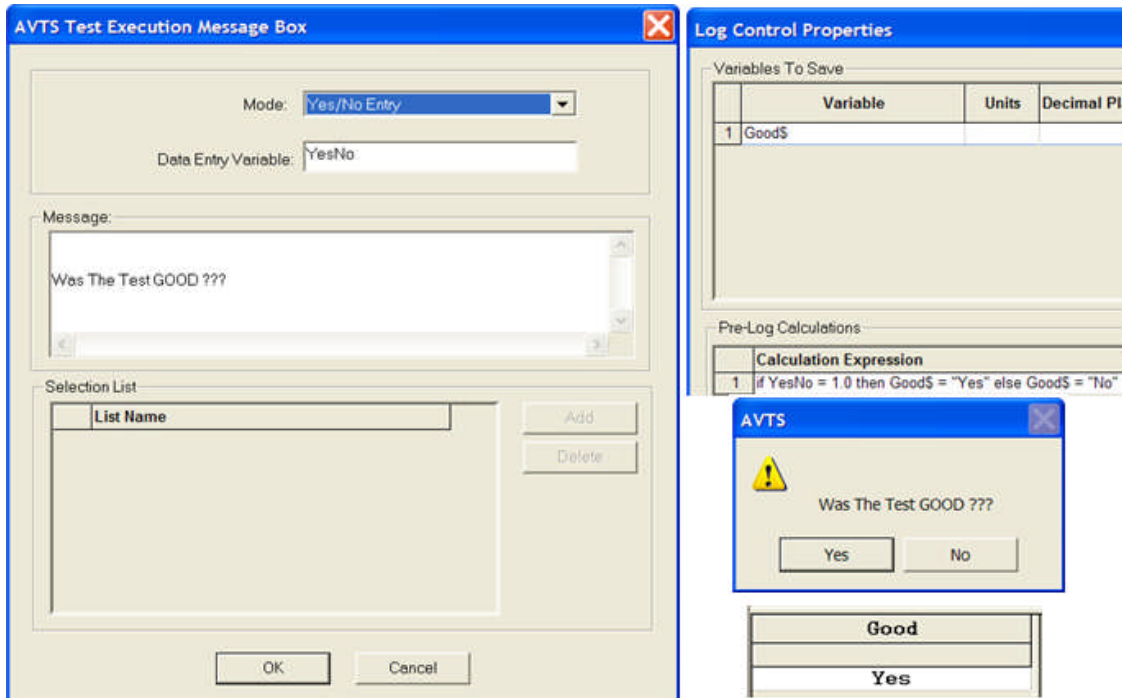
**Note:** Entry of text will produce an output of 0.00.

*Text Entry* is to place non-numeric data into the Log Control for text results. The DataEntryVariable is TextEntry\$ (this is a variable containing string values and must end in '\$'). The Pre-Log Calculation is Watts\$ = TextEntry\$ and the results display Watts with text "Good Test !!".



Note: A short cut would be to place the label Watts\$ in the DataEntryVariable edit field; then a Pre-Log Calculation would not be required.

**Yes/No Entry** is to provide a conditional branch during a test procedure and print to test results. This *Yes/No Entry example* demonstrates for the Yes or No to be printed in a test result. The Message Box YesNo output is either 1.0 or 0.0 and may be displayed as text. (See Expressions).



This *Yes/No Entry example* demonstrates a conditional output of the first Message Box to a second Message Box. Not shown, but a Yes response to the first Message Box provides the same result demonstrated in the first example. Note: The conditional outputs may Target the Ramp, Vector or other Controls.

The screenshot shows the 'AVTS Test Execution Message Box' in 'Text Entry' mode. The 'Data Entry Variable' is 'Goods'. The message text is 'If Bad Test Type "Do again"'. Below the message is a 'Selection List' with an empty table and 'Add' and 'Delete' buttons. To the right is a flowchart showing a 'START' node leading to a 'MESSAGE' node. From the 'MESSAGE' node, a 'No' path leads to another 'MESSAGE' node with the text 'Bad - Do Again..', and a 'Yes' path leads to a 'LOOP' node. Both paths eventually lead to an 'END' node. Below the flowchart is a table representing the output:

Good
Bad - Do Again..

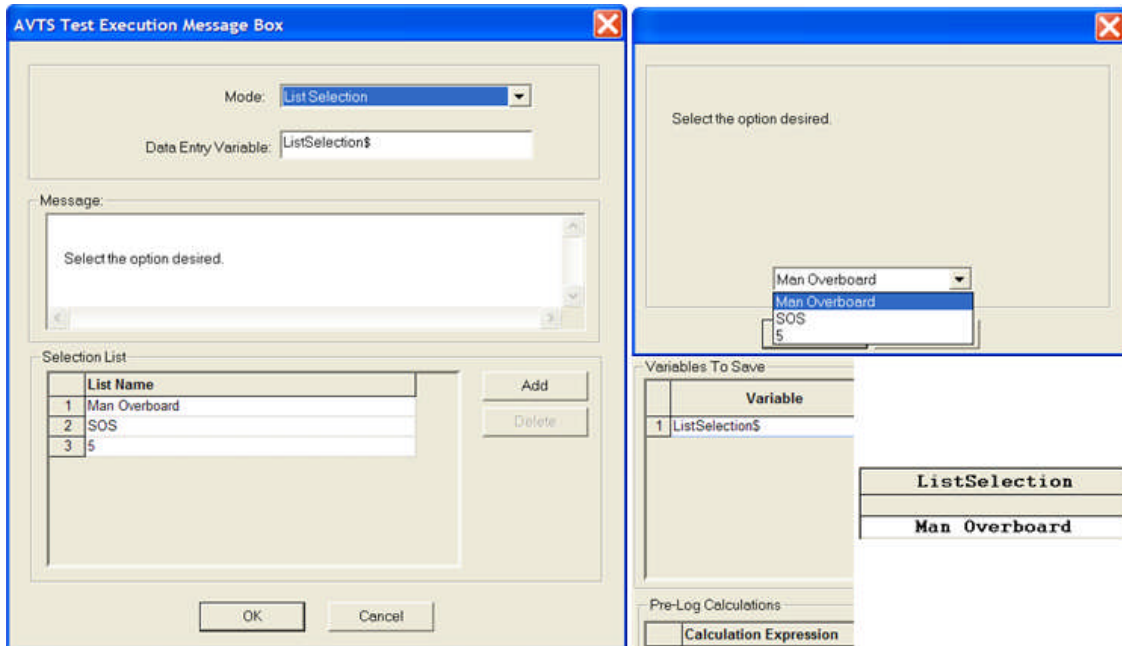
*Pass/Fail Entry* is similar to the Yes/No Entry except to replace the Yes option with Pass and the No option with Fail in the Message Box dialogs and the Links exiting the Message Box.

The screenshot shows the 'AVTS Test Execution Message Box' in 'Pass/Fail Entry' mode. The 'Data Entry Variable' is 'PassFail'. The message text is 'Test Pass? or Fail?'. Below the message is a 'Selection List' with an empty table and 'Add' and 'Delete' buttons. To the right is a flowchart showing a 'START' node leading to a 'MESSAGE' node. From the 'MESSAGE' node, a 'Fail' path leads to another 'MESSAGE' node, and a 'Pass' path leads to a 'LOOP' node. Both paths eventually lead to an 'END' node. Below the flowchart is a table representing the output:

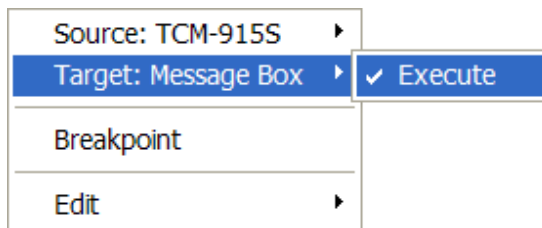
PassFail
1.00

*List Selection Entry* provides ability to create a list of string variables for user choice upon entry to the Message Box. If a number is created as one of the drop list options, it will be treated as a string variable with no value. To create a list of numbers as options to a test, the number is

treated as a string variable and may be converted to a real number in another Control;  
val(number\$) where number\$ is the Data Entry Variable label.

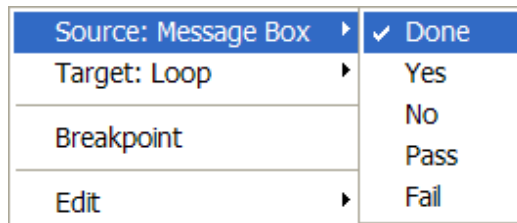


Target Link is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

Source Links are Done, Yes, No, Pass and Fail.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

The Yes command is a branch option from the Control. If the Calculation condition is TRUE, then the branch option is Yes.

The No command is a branch option from the Control. If the Calculation condition is FALSE, then the branch option is No.

The Pass command is a branch option. If conditions present in the properties dialog meet the condition, this branch is active.

The Fail command is a branch option. If the conditions present in the properties dialog meet the condition, this branch is active.



## Node Control

The **Node Control** is to connect more than one Link to a common point. The Control provides the same function as multiple Links to a common Control other than a Node Control.



Note: The Node has this appearance when selected by the cursor and will appear as a dot when not selected by the cursor.

The Node Control has no Properties for setting.

The Node Control has no Target Link.

The Node Control has no Source Link.

Note: The user may consider applying multiple Links to a Control that would be the Target of a Node Control. The purpose is equal but reduces the number of Controls in the procedure.

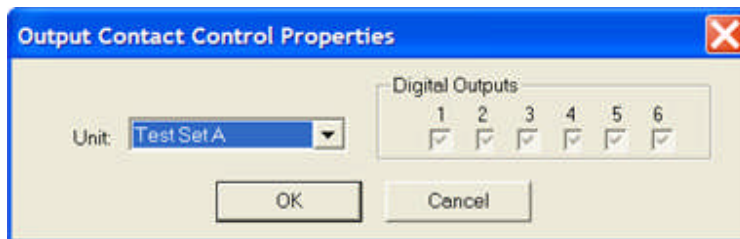
## Output Contact Control

The **Output Contact Control** is for use with the six output contacts on the MPRT test unit. The contact conditions set in the Control occur immediately upon entry to the Control.

**Note:** The Output Contact 1 operates the PULSAR Aux Contact.

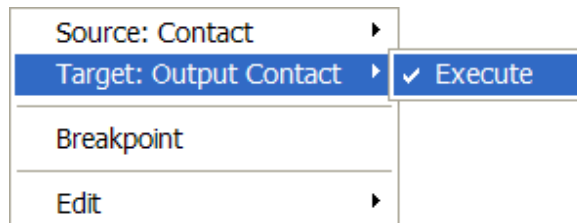


*Output Contract Control Properties* provide a drop list for TestSetA or TestSetB selection and contact conditioning. The default entry dialog displays the contacts as checked but dimmed for "inactive". This is a null state and no contact commands will be sent to the test unit. Single click on a check box will place the selected contact in the open and active state. A second click on the check box will place the selected contact in the closed and active state. A third click returns the selected contact to the default inactive state. Multiple contacts may be made active in the option states where ever the Control is used.



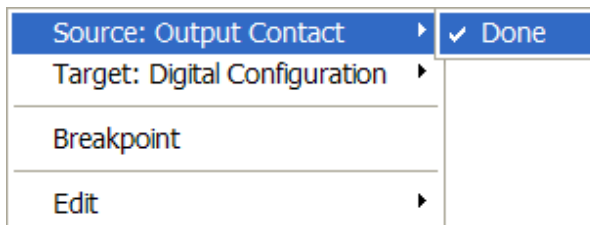
The Control may be used several times through the test procedure. Use in this manner allows varying inputs to provide control logic to a device under test at various times throughout the test process.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## PC Timer Control

The **PC Timer Control** is to provide a wait or idle period of no activity during a test execution. Multiple PC Timer Controls may be used throughout a test procedure.



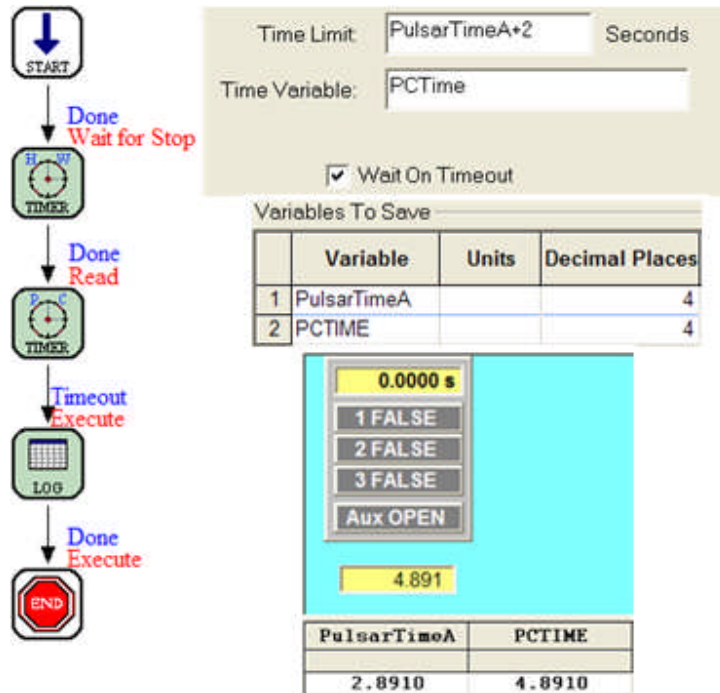
An example of this Control is demonstrated in Test Editor OC Time Example.

*PC Timer Control Properties* set the amount of time to delay within the Control before continuing to the next control.

**Time Variable:** edit field has a default "PcTime" label which may be used to pass the value of time delay in the Control. A variable label different from the default PcTime variable label may be used.

**Time Limit:** edit field enables the Wait On Timeout check box. An entry in this field may be numeric, formula or a variable. Any variables in the Control must be identified prior to Control execution.

*Example:*



Start Control linked to the HW Timer Control.

The HW Timer Properties is started by Software.

The HW Timer Properties is set to be stopped by Simulate Contact.

The HW Timer record time will be saved in the variable "PulsarTimeA".

HW Timer Control linked to the PC Timer Control.

The PCTime variable will save the time expired by the Time Limit value "PulsarTimeA + 2". The saved time will be 2 seconds delay plus the saved HW Timer recorded time.

PC Timer Control to the Log Control.

The Log Control will use the two variable names identified to display as a test result the saved times: PulsarTimeA = 2.8910 and PCTIME = 4.8910.

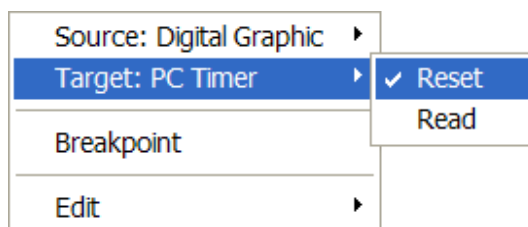
Start to HW Time Target Link = Wait for Stop.

HW Time to PC Time Link Target = Read.

PCTime to Log Link Source = Timeout or Done.

Times displayed in Log Control as results by the variable names.

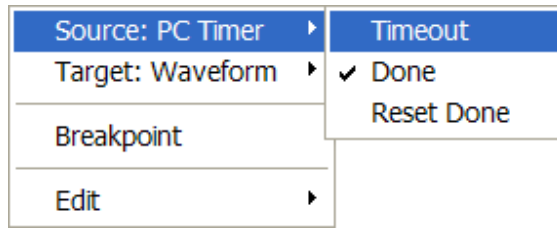
*Target Links* are Reset and Read.



Reset sets the Timer function to zero.

Read registers the value of time present in the Timer upon entry to the Control.

*Source Links* are Timeout, Done and Reset Done.



Timeout executes the Time Limit: set in the Control, providing the definition exists.

Done concludes the activity of the Control and passes the procedure execution to the next Control and will pass the elapsed time to memory. In general, this will allow the same time delay as the Link Source Timeout.

Reset Done will reset the timer and not retain any elapsed timeout.

## Ramp Control

### Ramp Control

The **Ramp Control** is to produce a controlled increase or decrease of current and/or voltage channel outputs of the test unit. It can be set to monitor a device under test to determine its operating characteristic. Typically, the Control is used to determine a Pickup value of a device under test. Additional details for the Ramp Control are detailed in the Online Ramp Control topic.



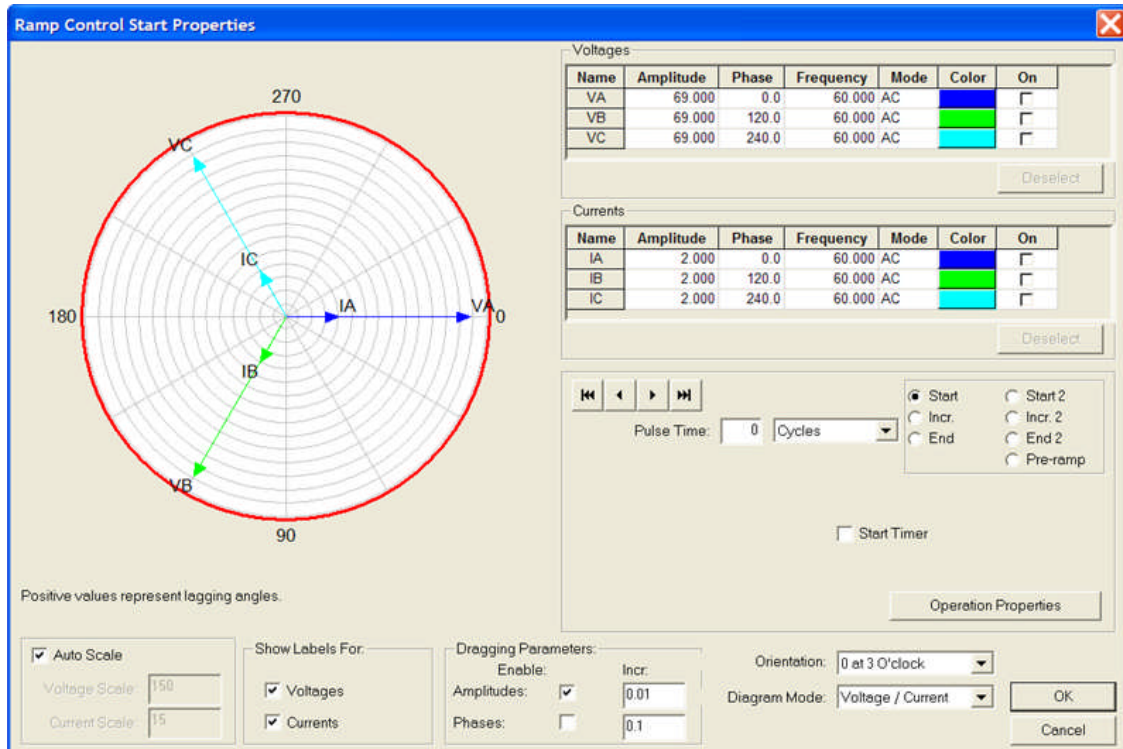
An example of this Control is demonstrated in Test Editor Charc Z Example 2.

The properties dialog is described in the Ramp Control Properties topic.

The link properties are described in the Ramp Control Links topic.

## Ramp Control Properties

*Ramp Control Properties* dialog is to set operating parameters for selected currents and voltages.



The Voltage and Currents edit fields are to set the parameters for each channel

**Name** is default channel label by phases

**Amplitude** edit field is to enter the value of the selected phase. The value may be an integer, floating point or variable.

**Phase** edit field is to enter the value of the phase angle for the selected phase. The value may be an integer, floating point or variable.

Frequency edit field is to enter the value of the frequency in hertz. The value may be an integer, floating point or variable.

**Mode** drop list is to select AC, DC or Half-Wave.

AC is the default for the Frequency edit field.

DC is for entry of a DC amplitude value and sets Frequency edit field to zero.

Half-Wave is for providing the diode test with AC RMS equivalence.

**On** check box will send the ON command to the selected channel when the test is executed.

Initial parameters are set in the Start option dialog.

Incremental ramping up or down parameters are set in the Incr. option dialog.

A second set of parameters are allowed to be set in the Start2, Incr.2 and End2 dialogs.

The stop ramping parameters are set in the End option dialog. The three options are detailed below.

**Start** dialog is to set channel Amplitude, Phase angle, Frequency values, and Mode (AC, DC or Half-wave) and On commands to initialize the test unit for the Control operation. Amplitude, Phase, Frequency and Mode may be variables.

**Pulse Time:** edit field allows a numeric or variable entry to control the pulsing of the channel parameters during the use of the Ramp Control. If set to zero, the ramp will be a continuous non-pulsing operation from Start to monitored operation or End. The Pulse Time drop list dictates the pulse to be in Cycles or milliseconds.

**Start Timer** check box will begin a timer upon initiation of the Start parameters.

**Operation Properties** opens a dialog to set the monitor function to recognize a device under test operation. The monitor function will also Stop the Timer that was set to Start in the Start dialog.

**Incr.** dialog is to set the channel Amplitude, Phase and/or Frequency values to ramp for each time increment of the ramp process. Any parameters not to be ramped should retain a value = 0.

**Increment Time:** edit field is to enter a numeric or variable time to control the value of time that each channel parameter will be applied to the device under test prior to applying the next time value. The Increment Time drop list dictates the pulse to be in Cycles or Seconds.

**Note:** Compare Increment Time: to the Pulse Time: item below.

**End** dialog is to set the final parameter values to be applied should there not be a monitor input from the device under test. On End, the test will abort.

**Note:** The incremental Ramp UP or DOWN is determined automatically by AVTS. If the End value is greater than the Start value, the Ramp will be increasing in value. If the End value is less than the Start value, the Ramp will be decreasing in value.

**Start2** dialog repeats the Start application for the Amplitude, Phase and Frequency only, and will usually have different parameter entries, likely smaller increments for "fine tuning" the monitored response.

**Pulse Time:** edit field allows a numeric or variable entry to control the pulsing of the channel parameters during the use of the Ramp Control. If set to zero, the ramp will be a continuous non-pulsing operation from Start to monitored operation or End. The Pulse Time drop list dictates the pulse to be in Cycles or milliseconds.

**Note:** Compare Pulse Time: to the Increment Time: item above.

**Reset Time:** edit field is to enter a numeric or variable time to delay the Start2 execution, usually to allow a device under test to return to a non operate state.

**Note:** Reset Time is independent to the Pre\_Ramp Delay time.

**Operation Properties** opens a dialog to set the monitor function to recognize a device under test operation. The monitor function will also Stop the Timer that was set to Start in the Start dialog. Settings here are generally the same as those associated with the Start dialog.

**Incr.2** dialog is to set the channel Amplitude, Phase and/or Frequency to values to ramp for each time increment of the ramp process. Any parameters not to be ramped should retain a value = 0. Note: If all values are zero, Start2 will not execute.

**Increment Time:** edit field is to enter a numeric or variable time to control the value of time that each channel parameter will be applied to the device under test prior to applying the next time value. The Increment Time drop list dictates the pulse to be in Cycles or Seconds.

**End** dialog is to set the final parameter values to be applied should there not be a monitor input from the device under test. On End, the test will abort.

**Pre-ramp** dialog is to set the channel parameters to a value to be applied prior to beginning the Start and Start2 execution. The dialog is enabled by setting a non-zero value in the edit field.

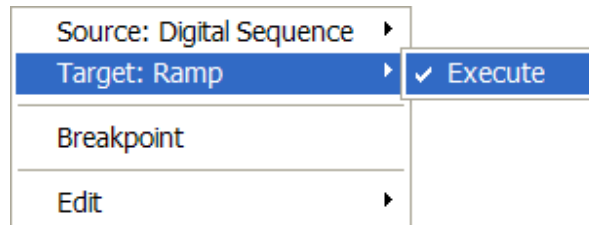
**Pre-Ramp Delay:** edit field is to enter a numeric or time variable to control the time for the Pre-Ramp parameters to be applied prior to passing to the Start or Start2.

**Note:** The Pre-Ramp Delay, if set, will occur prior to the initializing the Start and the Start 2 and apply the values set in the Voltages and Currents edit fields for the value of the Delay. The Delay will occur for each pass to the Ramp Control if placed inside a List or Loop Control.



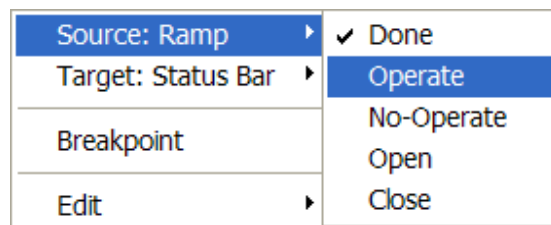
## Ramp Control Links

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Links* are Done, Operate, No-Operate, Open and Close.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

The Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the Operate link upon a successful operation within the set limits.

The No-Operate command is a branch option from the Control. It forces the Control to complete the settings in the properties dialog and to branch to the No-Operate link upon an unsuccessful operation within the set limits.

**Note:** The non-operation of a required response and Control properties have been expended to the limits may constitute this branch and indicate an improper condition to abort the procedure.

The Open command is a branch option from the Control. It passes the Control properties either at the completion of an Open contact coming open during the Control execution or completely by-passes the Control if an Open contact is met at the beginning of the Control execution.

The Close command is a branch option from the Control. It passes the Control properties either at the completion of an Close contact coming open during the Control execution or completely by-passes the Control if an Closed contact is met at the beginning of the Control execution.

## Select Control

The **Select Control** has no properties. It has the actually the mouse cursor and is enabled by "picking" with the mouse the Control from the Control bar.



The Select Control has no properties. It is used to pick controls from the Control bar on the right side of the Workbook pane and place them into the Workbook. It is also used to select one or more controls for moving within the Workbook.

To select several Controls, hold the Ctrl key and click on all desired Controls in the Workbook one after the other to highlight them for further action.

Several Controls may be selected as a group by holding the left mouse button and dragging a box covering all desired Controls in the Workbook to highlight them for further action.

There are no Links associated with the Select Control.

## Start Control

The **Start Control** is placed into a Workbook Test Tab test procedure automatically by AVTS. It is the only Control which cannot be deleted.



An example of this Control is demonstrated in Test Editor OC Time Example.

*Start Control Properties* dialog has several inputs to initialize some settings upon test execution.

**Current Phases:** is to select from the drop list the number of current channels to be active in the test procedure. Only the number selected will be displayed in the Ramp or Vector Controls. A total of 16 channels are allowed to accommodate the use of two MPRT test units.

**Voltage Phases:** is to select from the drop list the number of voltage channels to be active in the test procedure. Only the number selected will be displayed in the Ramp or Vector Controls. A total of 8 channels are allowed to accommodate the use of two MPRT test units.

**Digital Inputs:** is to set the number of MPRT Inputs to be monitored throughout the test procedure. The number used is determined by configuration of the Digital Configuration Control or the Digital Sequence Control. Up to 32 may be used to accommodate two MPRT test units.

**Digital Outputs:** is to set the number of MPRT Output contacts to be utilized throughout the test procedure. The number used is determined by configuration of the Digital Configuration Control or the Digital Sequence Control. Up to 32 may be used to accommodate two MPRT test units.

**Primary Settings Group:** is to select from the drop list the Workbook Settings tab Group name this particular test will collect settings to use throughout the test procedure.

**Reset Units:** is to select from the drop list the Soft Reset, Hard Reset or No Reset application to the test unit upon execution of the test.

**Soft Reset** will replace all parameters to default values and clear all buffers.

**Hard Reset** will reset the test unit in the same fashion as if the test unit is turned on.

**No Reset** will leave all parameters and buffers in their last know values.

**Angle Mode:** is to select from the drop list the angle relationships to be considered by phase angle displacement.

**Lag** places all vectors in positive degrees displacement clockwise in a polar diagram referenced to zero degrees. Pulsar operates in lagging positive angles only. MPRT allows use of this convention.

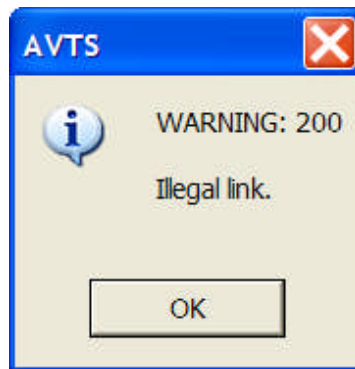
**Lead/Lag** allows for all vectors to be placed in positive degrees displacement counter-clockwise in a polar diagram referenced to 0 degrees. This is considered the textbook engineering convention and may be used with MPRT.

**Note:** PULSAR use allows Lag Angle Mode only.

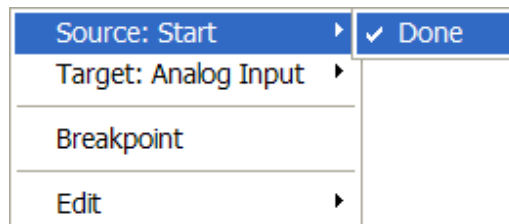
**Use MPRT Functionality** check box **will** enable display of the "Boolean tool" embedded in the Vector Control. The option is to differentiate use between Pulsar and MPRT.

The Start Control has no Target Link.

**Note:** An attempt to link the Start Control as a Target to another Control will produce the *Illegal Link* dialog.



Source Link is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

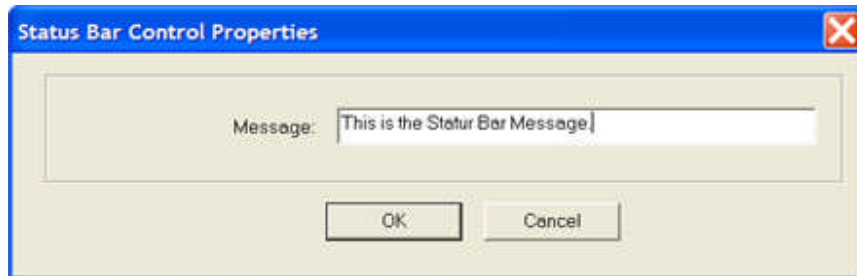
## Status Bar Control

The **Status Bar Control** is for placing a message in the AVTS status bar location at the bottom of the pane.

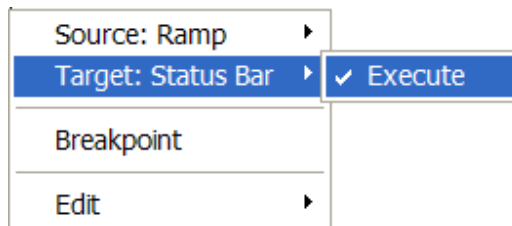


*Status Bar Control Properties* is used to place a *Message in the Status bar*.

For Help, press F1    Lag    This is the Statur Bar Message.

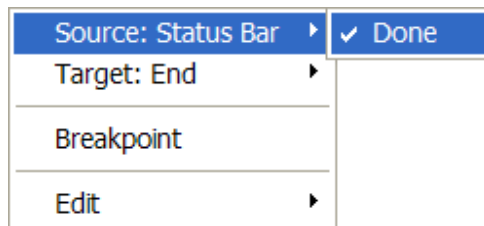


*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## **SubTest Control**

### **SubTest Control**

The purpose of the **SubTest Control** is to create a test(s) that can be added as part of an open relay test. The process allows repetitive procedures to be readily available for insertion at any time to any relay test.

The New SubTest is available from the Main Menu File/New/Subtest option. This option allows creating a SubTest outside an open relay.

Select the SubTest option to create a new SubTest.

- SubTest Control Properties topic describes the Subtest Control properties dialog.

- New SubTest topic details the properties of the SubTest

- New SubTest Example topic for a worked example of use of a SubTest

## SubTest Control Properties

The **SubTest Control** is for recalling sub routines previously created and saved with a test label.

The SubTest to be created is evoked by selecting File/New.../SubTest menu item. The call will open a new test in the Workbook Test Editor tab.

The SubTest may also be created by placing the Control in an existing test procedure. Open the SubTest Control Properties and click on the Add button which will open a new test in the Workbook Test Editor tab for edit.

The Test Editor and BASIC tabs are the only two tabs available to create the test and therefore have no Workspace Device association.

The Test Editor is to create the SubTest using the Controls and Linked to create a test procedure.

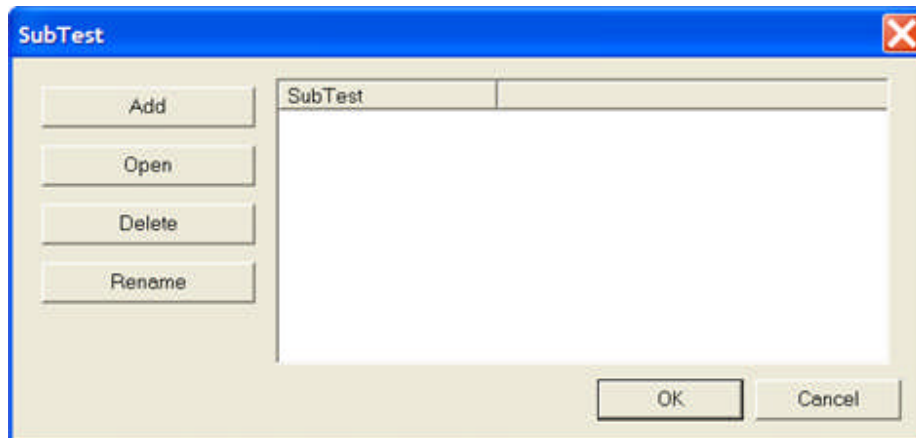
The BASIC tab is available to write a subroutine in BASIC that might not otherwise be able to be created using the available Controls.

A created SubTest using the New SubTest will be saved to the SubTest memory. The SubTest may be recalled from within a list of saved SubTest within the Control for placement in any test procedure, either new or existing.

An example for the construction and use of a SubTest is demonstrated in the New SubTest Example topic.



*SubTest Control Properties* will contain the list of previously saved SubTests, if any, and access buttons.



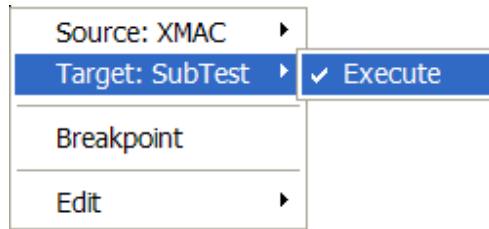
**Add** opens the Test Editor tab with the un-linked Start and END Controls, and a blank BASIC tabs to create a new test.

**Open** will open the selected SubTest highlighted in the list of previously saved SubTests.

**Delete** will remove the selected SubTest from the list.

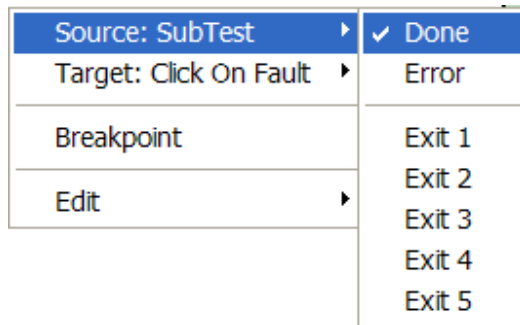
**Rename** will open the SubTest name dialog to edit the label for the selected SubTest.

*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Links* are Done, Error, Exit 1, Exit 2, Exit 3, Exit 4 and Exit 5.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

The Exit 1 command is a branch option from the Control. This option is executed should a set condition embedded in the properties script or calculation that is met. The Exit 1 is controlled by the properties action.

The Exit 2 through Exit 5 commands are additional branch options governed by the same conditions as the Exit 1 command.



## New SubTest

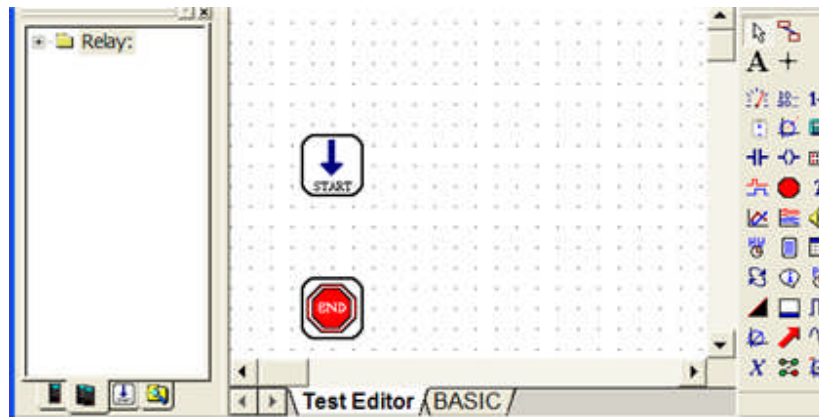
AVTS opens the Workbook pane with the Test Editor and BASIC tabs only.

The Test Editor tab is necessary to construct the test.

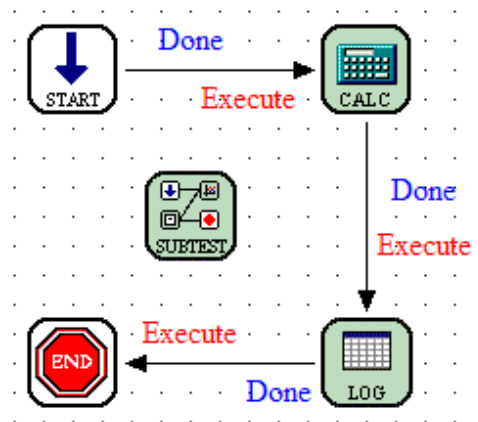
The BASIC tab is necessary to include BASIC macros, script or tables as needed by the BASIC and 1-Touch Controls.

**Note:** The test does not need relationship to a specific relay. No Devices, Relays or pre-constructed Tests are open.

The *Test Editor* tab contains only the Start and END Controls with no Links.



Construct a *test procedure* that may be added to any number of similar relays.

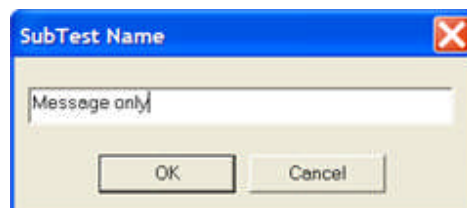


The construction of a test, using the Test Editor Controls, is demonstrated in the How to Create Test Procedure Example topic.

Link all the Controls from Start to End in the Test Editor.

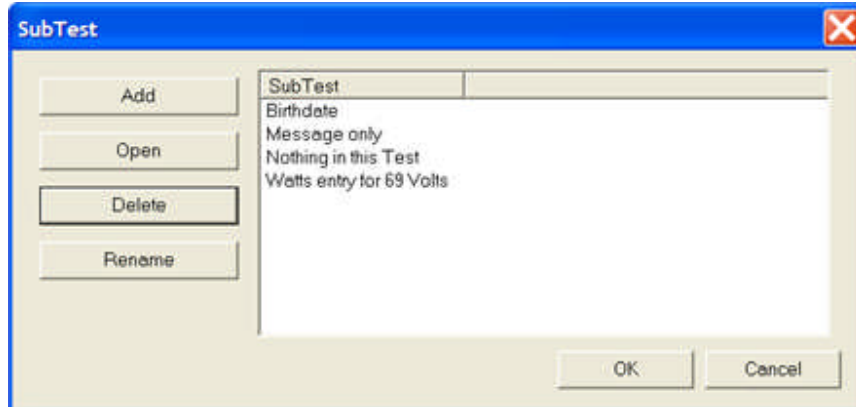
The test must be saved with a name for recall later in another test procedure.

Click on the Save tool on the main toolbar or click on the Close "x" in the upper right corner of the Workbook. The *SubTest Name* dialog will appear for entry of a name of the SubTest just created.



**Note:** Do not click on the major AVTS Close "X".

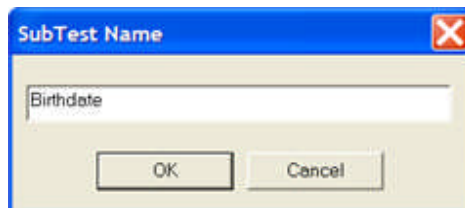
The SubTest has been saved in the database to the SubTest Control and will appear in the *SubTest Control Properties* dialog.



For the Relay SubTest Control insertion to the test procedure:

Insert the SubTest Control.

Double mouse click on the SubTest Control to evoke the Properties dialog. Select the Add button to open the *SubTest Name* dialog. Enter the name and click OK.



**Note:** Accepting the SubTest Name will cause all the test Controls except the Start and END Controls to be absorbed, indicating that those absorbed have been saved, with the SubTest Name, in the database for future insertion to another test.

## New SubTest Example

The **New SubTest** is to create a test procedure that is saved with a label in the active database. The SubTest Control may then be placed in an existing test procedure in the Test Editor tab and become part of the routine or special test for a relay.

This example is to apply a voltage, provide calculations and enter a SubTest to perform a ramping value to determine a pickup value and a calculated value. The test then ends with the results logged and displayed in the Test tab.

The action viewed in the Test tab will be

Voltage = 69 volts and On for 90 cycles and Current = 0 amps and Off

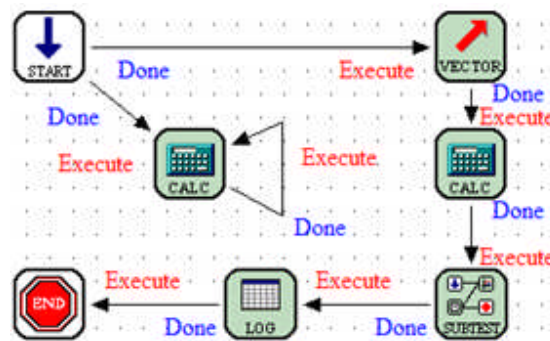
Voltage = 30 volts and On during Ramp to End or to Pickup

Current = ramping from the calculated or 0.5 amperes and Incrementing each 1 Second

Watts box will display values increasing from 0 to the calculated value at the instant of Pickup

The end value of the variables Va, Pickup and Watts are logged as results

The main *Test Editor* tab includes



Calculation Control for continuous update of a ramping parameter

$$\text{Watts} = V_a * I_a$$

Vector Control for applying a momentary stabilizing voltage and may be passed to the SubTest

Va Amplitude = 69 and checked On

Apply Time = 90 cycles

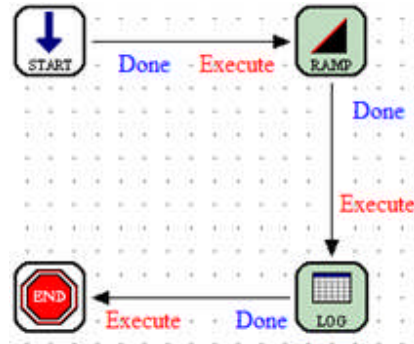
Calculation Control for providing variable data to be passed to the SubTest

$$V_x = 30$$

$$Z_y = 15$$

SubTest Control to include the Controls for the action portion of the test

The *Subtest Test Editor* tab includes



Ramp Control where ramping values are displayed during the test

Start

Va Amplitude = 30 and checked On

Ia Amplitude =  $Z_y / V_x$  and checked On

Incr.

Ia Amplitude =  $Z_y / V_x * 2$

Increment Time = 1 Second

End

Ia Amplitude =  $Z_y / V_x * 30$

Operation Properties left set to default

Post: 3, Mode: Continuity, Active On: Applied, Do Action: Monitor, Latch

Log Control to capture the values to be logged for the final results

Variables are Va and Pickup

Pre-Log Calculations are Pickup = Ia and Watts = Va \* Pickup

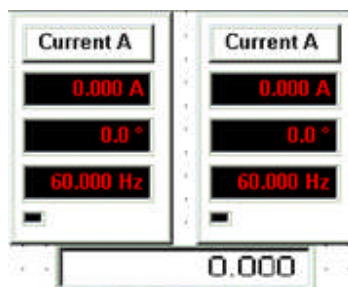
**Note: The Log Control must be present in the Subtest to capture results for the final Log Control.**

Log Control to display the desired results in the Test tab and saved as the relay test results

Variables are Va, Pickup and Watts

Pre-Log Calculations NONE

The *Monitor Editor* tab includes the following



Voltage Channel set for Voltage Channel A

Current Channel set for Current Channel A

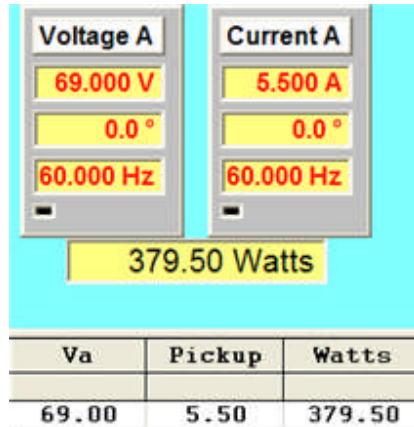
Value box

Variable or Expression:= Watts

Decimal Places = 2

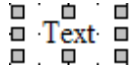
Unit Label = Watts

The **test results** from this example will appear in the Test tab.

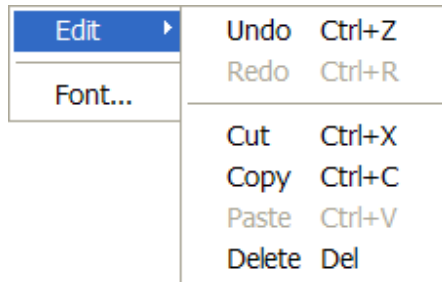


## Text Control

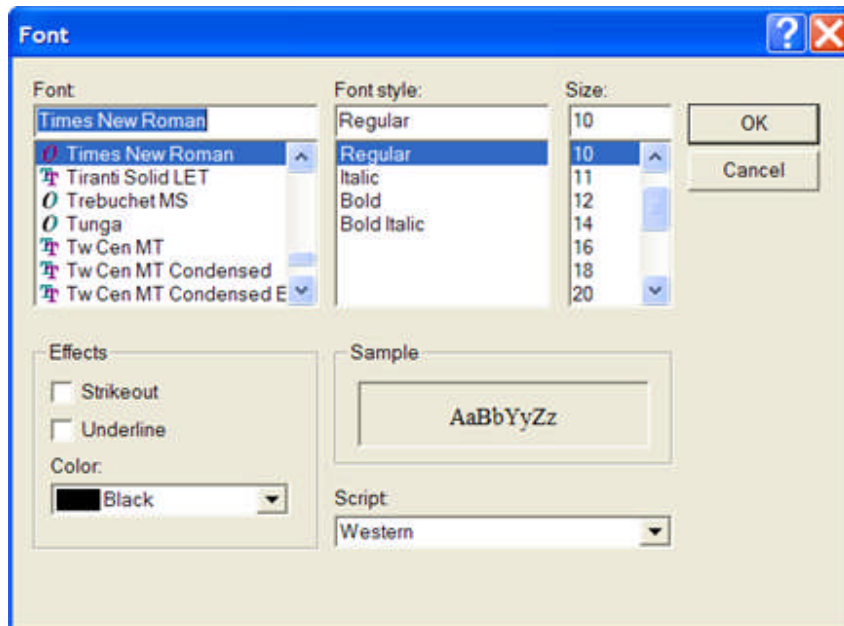
The **Text Control** is used to enter formatted text in desired areas of the Test Editor tab to further detail the test procedure. The Control is shown with the eight squares indicating that it has been placed in the Workbook pane for use.



The desired Text entry is created by double left mouse click, highlighting the text and then entering text with use of the keyboard. Or, right mouse click on the Control to produce the *Text menu* and select the Font option. The Edit options are the standard Windows options. Using the Edit options text may be Cut/Paste elsewhere, Copy/Paste for repetition and Delete.



Font option opens the standard Windows *Font dialog* for selecting the desired font appearance. Note: If "formatted" text is present, it is usually more efficient to copy the existing text, pasting the copied text and then edit the text with a double left mouse click. The action retains the format of the copied text.



An example of formatted text next to a Workbook Control for additional information about the test procedure.



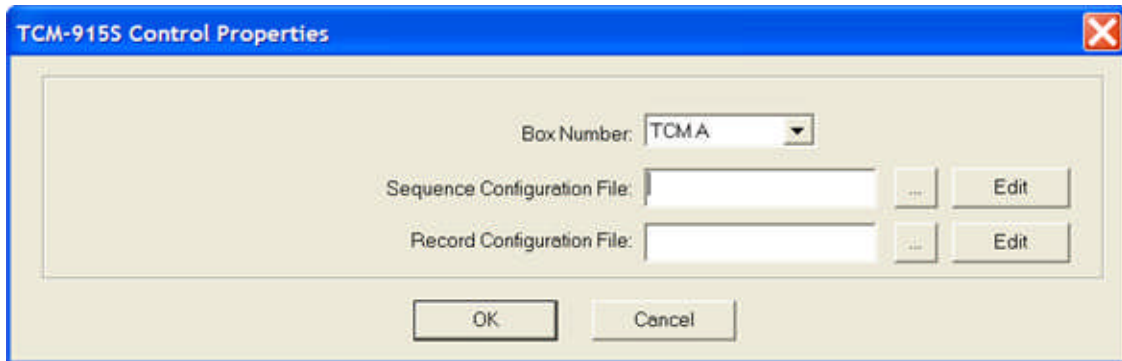
## TCM-915S Control

The **TCM-915S Control** is to enable calls to Sequence and Record files previously set with the PulseMonitor controlling software for TCM-915S. Typical use is to control the 9 output contacts and 15 input monitor posts of the device for logic input and monitored output of the device under test. A time accuracy of +/- 0.1 msec. is maintained throughout the use of the instrument. A Record of the input and output activity can be retrieved from the unit and reproduced in graphic form using the Digital Graphic Control and included in the a test report.

For use, a TCM-915S test unit must be connected to the device under test, and the IEEE 488.2 GPIB interface for communication to and from must be connected between the computer and TCM-915S.



*TCM-915S Control Properties* contains a drop list and fields for initializing two files.



Box Number: drop list is to select TCM A, TCM B or TCM C for use with this Control.

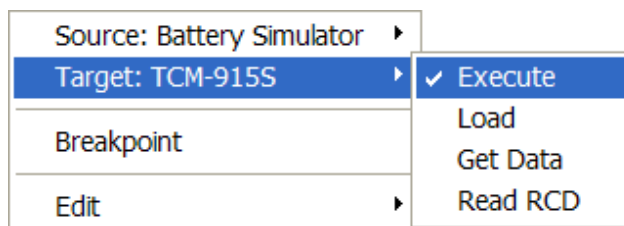
Sequence Configuration File: edit field and browse button allow selection of the path to the \*.seq file to be used for control of the TCM. The file must reside in the C:/Program Files/AVTS/TCM folder.

Edit is to open the Sequence file in switchbox.exe for alteration to the sequence for use during the test.

Record Configuration File: edit field and browse button allow selection of the path to the \*.rcd file to be used. The Record file is to track and display retrieved activity recorded by the TCM during the executing Sequence file. The file must reside in the C:/Program Files/AVTS/TCM folder.

Edit is to open the Record file in switchbox.exe for alteration to the record for use during the test.

*Target Links* are Execute, Load, Get Data and Read RCD.



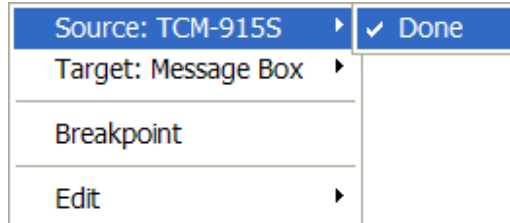
The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Load command forces AVTS to load the Sequence data file \*.seq defined in the Control edit field, or located with the browse button.

The Get Data command forces AVTS to read the Record values from the TCM-915S after or during an execution of a Sequence.

The Read RCD command forces the Control to read the record data file \*.RCD defined in the Control, or located with the browse button.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.



## Theoretical Control

### Theoretical Control

The **Theoretical Control** is to create types of curves and characteristics for devices under test.

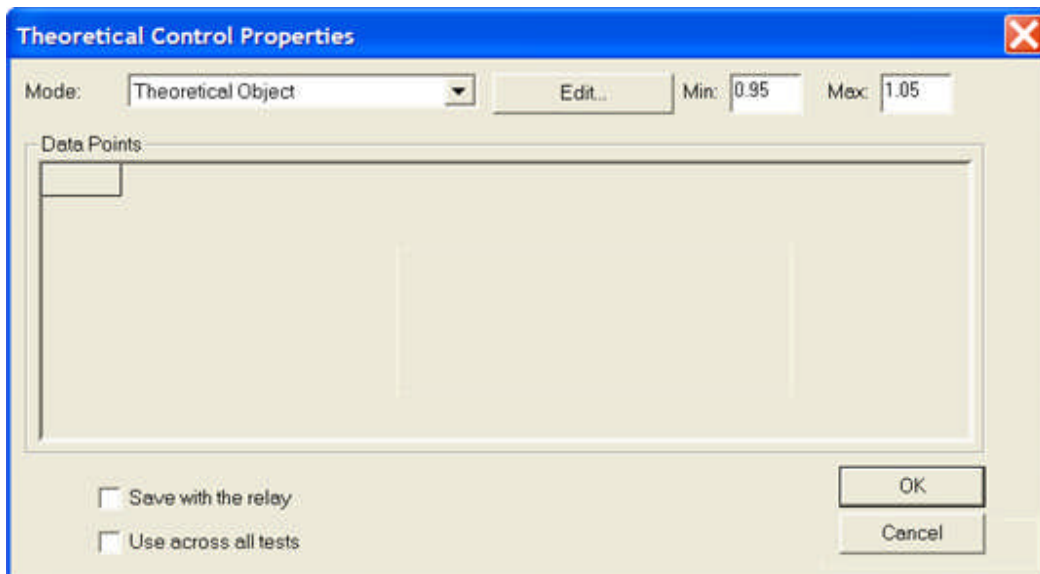
**Note:** The use of the Theoretical Control allows construction of a characteristic by any of the options available. The characteristic will need to be created one time only if all the data to determine the characteristic reside in the Settings tab or in an accompanying Calculation Control.

The theoretical takes information for the characteristics from the device settings, Calculation Control or from within the Theoretical Control itself. During a test execution, the resultant device response is compared to values in the characteristic for accuracy of response. Graphic displays during the test and for the test results are generated by the accompanying Graphic Control.



An example of this Control is demonstrated in Test Editor OC Time Example and Test Editor Charac Z Example.

*Theoretical Control Properties* contains drop list for graph design mode (data type) and an area for Data Point entry. Some mode types enable an Edit button to evoke a graph properties dialog for creating a characteristic with imported bit maps or other graphic data.



**GetTheoZ(Phase)** = get the theoretical impedance from the described data points at the phase angle specified.

**GetTheoZ(index,angle)** = get the theoretical impedance from the described data points (index), Go to Test Editor Charac Z Example for an example of the use of Index.

**Note:** To collect data from the Theoretical Control, AVTS variables must be used. The term (index) is an integer from 1 to n; where index = 1 is default, if not specified, and parentheses may be omitted. The index defines which crossing of the characteristic to identify along the impedance test path at the given torque angle.

**GetTheoPhase(Z)** = get the theoretical phase angle from the described data points for the impedance Z.

**GetTheoTime(Multiple)** = get the theoretical time from the described data points,

**TheoTimeDial** = get the theoretical time from the described data points given for a given time dial from a set of time dial curves; **GetTheoTimeDial** = 3,

**TheoInstOp** = get the theoretical instantaneous operation setting from the described data.

**Mode:** Types are

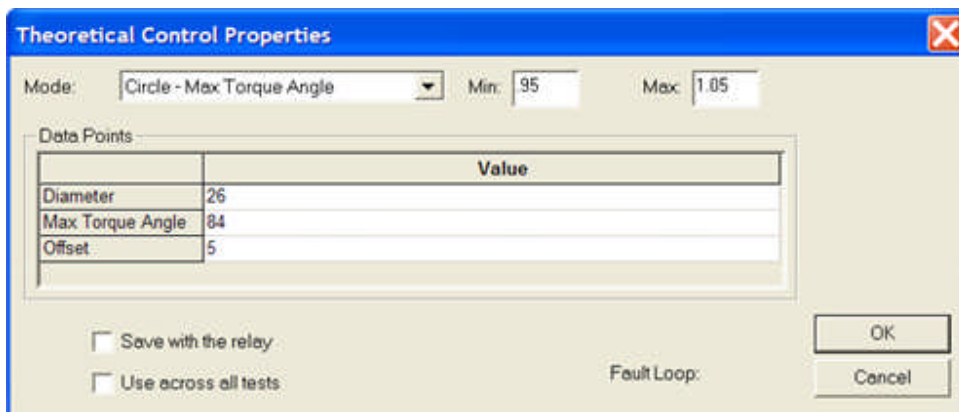
- Circle create impedance characteristic
- Circle - Max Torque create impedance characteristic
- Line - 2 Point create slope characteristic
- Line - Point/Slope create a slope characteristic
- Ellipse create impedance characteristic
- Lens create impedance characteristic
- Tomato create impedance characteristic
- Data - Polar Impedance create impedance characteristic
- Data - Impedance create impedance characteristic
- Data - Time-Amplitude create characteristic with discrete time & amplitude points
- Data - Generic create characteristic with discrete X & Y points
- Calculations create characteristic with formulas
- Theoretical Object for digitizing characteristics - Impedance or Time-Amplitude
- IEEE/ANSI Overcurrent Curve select standards defined characteristic
- IEC Overcurrent Curve select standards defined characteristic
- RIO option is used to import a RIO file data to be used in the Click On Fault Control

**Min:** edit field is to enter a numeric value describing another characteristic parallel at every tangent or with the same center to represent the minimum values of acceptance on the Theoretical Characteristic. A typical value might be 5 to represent the Theoretical Minimum to be described at every point to be 5 units less than the Theoretical.

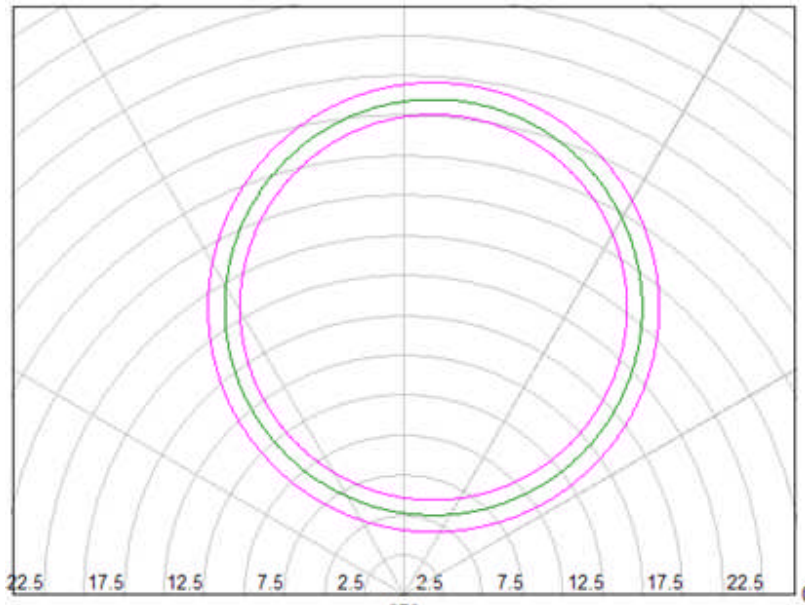
**Max:** edit field is to enter a numeric value describing another characteristic parallel at every tangent or with the same center to represent the maximum values of acceptance on the Theoretical Characteristic. A typical value might be 2 to represent the Theoretical Minimum to be described at every point to be 2 units greater than the Theoretical.

Min Max characteristics with the described Theoretical

**Theoretical Control** with  $Z = 26$ ,  $Z_{ang} = 84$ ,  $Min = 0.95$  and  $Max = 1.05$ .



**Test tab resultant characteristics** show reach of 26 and  $Min = 25.05$  and  $Max = 27.05$ .



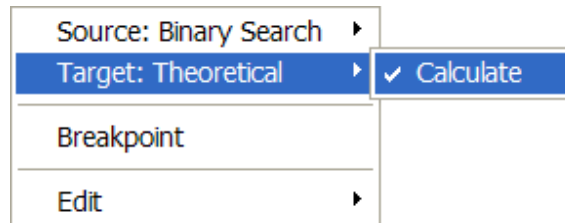
**Save with the relay** check box, when checked, will save the created characteristic to this Relay test only.

**CAUTION:** This option will automatically check and dim the Use across all tests check box.. Other relays of the same Device type will share this characteristic.

**Use across all tests** check box, when checked, will use this characteristic throughout all tests associated with this relay.

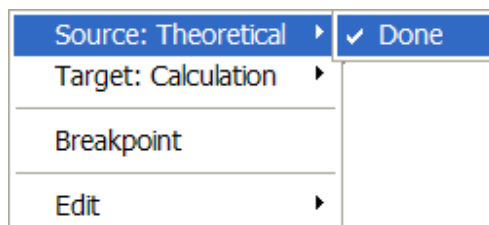
**CAUTION:** This option will make the Theoretical characteristic created with this Relay test associate to all tests in this Relay that have a Theoretical Control in the test procedure. I.e., should a Lens be created in this test, then another test with a quadrilateral or other characteristic type will be changed to a Lens.

*Target Link* is Calculate.



Calculate data points coincident to the active parameters in an executed test.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## Theoretical - Circle

The Theoretical Control properties for the **Mode Circle** are to define the circle diameter and maximum torque angle by using X and Y coordinates and the circle radius.

An example of this Control is to replace the Theo Control in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

Mode selected from drop list: *Circle* Data Points

Theoretical Control Properties

Mode: Circle Min: Max:

Data Points	
	Value
X	0.0
Y	0.0
Radius	0.0

Save with the relay  
 Use across all tests

OK  
Cancel

X is the abscissa for the center of the circle

Y is the ordinate for the center of the circle

Radius is the circle radius

Example:

Z = 4 ohms and MTA = 70 degrees

Calculation for X =  $Z * \cos(\text{MTA}) * 0.5$

Calculation for Y =  $Z * \sin(\text{MTA}) * 0.5$

Calculation for Radius =  $Z * 0.5$

**Note:** Real numbers may be used for a non-varying characteristic.

Place these variables in the formula into the respective edit fields in the *Theo Control Properties*.

**Theoretical Control Properties**

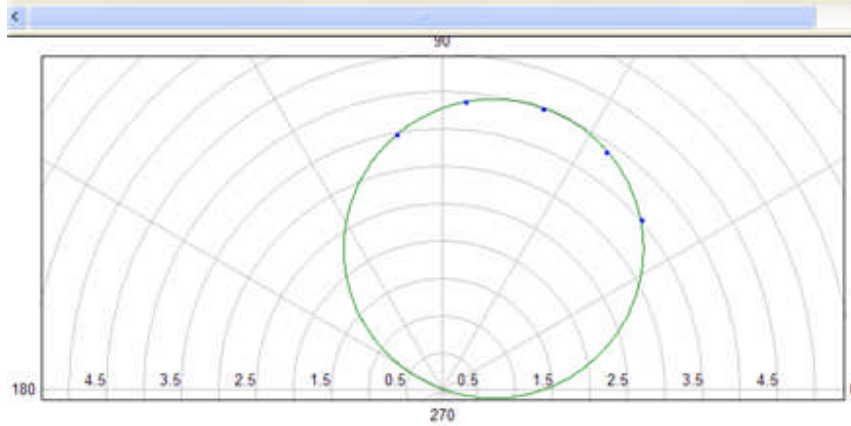
Mode:

Data Points

	Value
X	$z * \cos(\text{MTA}) * 0.5$
Y	$z * \sin(\text{MTA}) * 0.5$
Radius	$z * 0.5$

The *displayed characteristic* will agree with a test with the Settings for the relay are for Z and MTA.

40.00	30.00	8.60	3.49	3.29	3.64	Pass
55.00	30.00	7.79	3.85	3.67	4.06	Pass
70.00	30.00	7.55	3.97	3.80	4.20	Pass
85.00	30.00	7.79	3.85	3.67	4.05	Pass
100.00	30.00	8.71	3.44	3.28	3.63	Pass



Note: An Offset from the origin may be set or varied by increasing or decreasing the Radius while the X and Y values remain constant.

## Theo Circle - Max Torque

The Theoretical Control properties for the **Mode Circle - Max Torque** are to define the circle diameter, maximum torque angle and Offset.

This example is created from the Test Wizard, applying a name to the test, selecting Distance, click Next, select Single Phase Characteristic I, click Next and Finish. The test has been altered to demonstrate a value for Offset.

**Note:** Real numbers may be used for non-varying characteristic.

Mode selected from drop list: *Circle - Max Torque* Data Points

Theoretical Control Properties	
Mode:	Circle - Max Torque Angle
Min:	
Max:	
Data Points	
	Value
Diameter	Ztotal
Max Torque Angle	MTA
Offset	Offset
<input type="checkbox"/> Save with the relay	
<input checked="" type="checkbox"/> Use across all tests	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

**Diameter Value** entries may be a numeric or a variable.

**Max Torque Angle** Value may be a numeric or a variable.

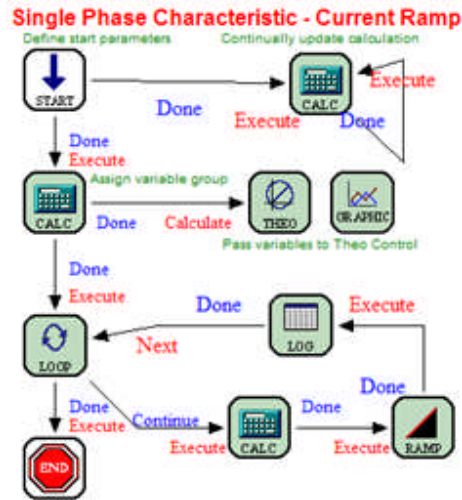
**Offset** Value may be a numeric or a variable.

**Note:** The generated characteristic is dependent upon numeric data values for each of the Data Points variable labels. The values may be given in the Workbook Settings tab or by a calculation or variable equality given in a Control prior to test call to the Theoretical Control.

This Theo Control example may be used in place of the Theo in the Test Editor Charac Z Example topic.

Example: Find three characteristic values for a defined mho characteristic ramping single phase current

Test Editor Controls: *Define test procedure*. Construction of test procedure in Workbook Test Editor tab using Controls with Links for flow control.

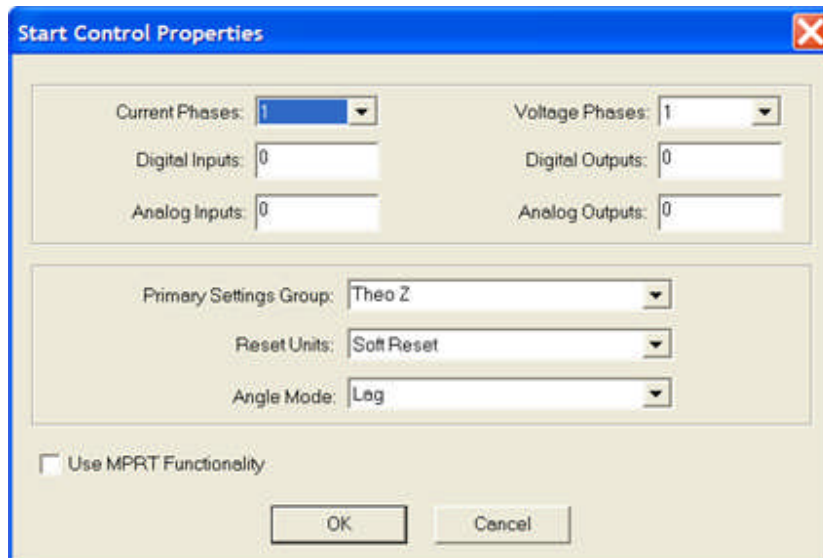


Note: Link Controls Source and Target selections.

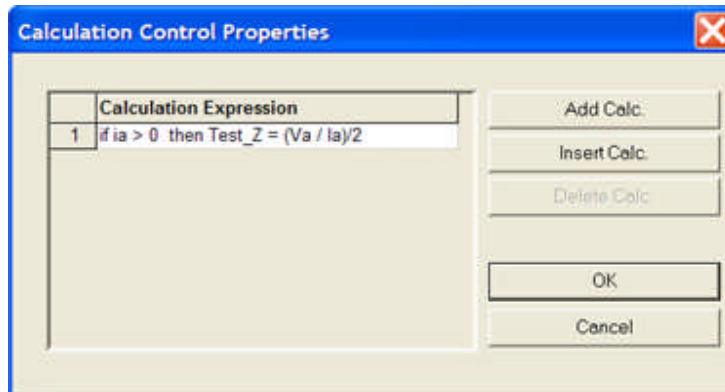
Settings tab entry: *Workbook Settings tab*. Active relay variables list for numeric data entry. Variable Z = Impedance value which is also the diameter of the described mho characteristic.

Setting Nam	Value	Min Valu	Max Val	Data Typ	Modbus	Description
1 Z	4	0	60	Float	0	Impedance setting
2 MTA	70	0	180	Integer	0	Maximum torque setting.

Start Control: *Define required channels and settings*. Using single phase current and voltage and selecting the Group of settings to call from the Workbook Settings tab.

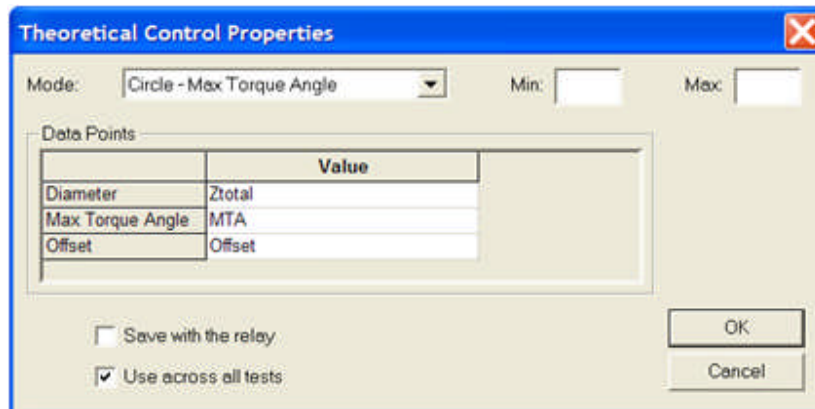


Loop-Back Calculation Control: *Define moving vector*. Used to perform continuous calculation and active display of the impedance vector during test execution. The loop-back Link Control provides the continuous calculation.

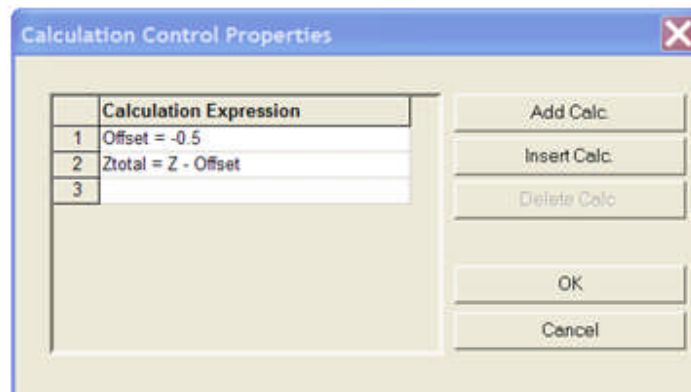


Note: The "if" condition to null the initial Ia = 0 on initiation of test execution; without which, the calculation would yield an undefined value with zero being in the denominator of the calculation of Z.

Theoretical Control: *Define characteristic*. Place variable names for data points entries Diameter, Max Torque Angle and Offset. These variables are defined either in the Workbook Settings tab or in a Calculation Control executed prior to the Theoretical Control.



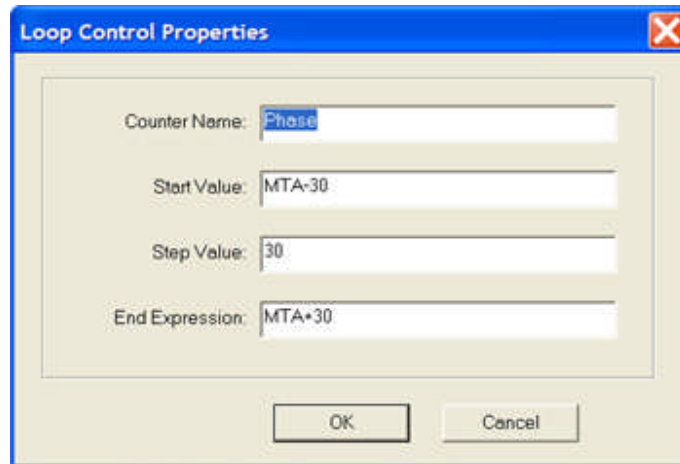
Calculation Control: *Define variable identities*. Defines a new value for Theoretical Control Diameter variable, Ztotal to account for Offset variable. Max Torque Angle variable may remain as MTA variable.



Note: Negative Offset for mho characteristic to not include origin in polar graph; positive Offset for mho characteristic to include origin in polar graph.

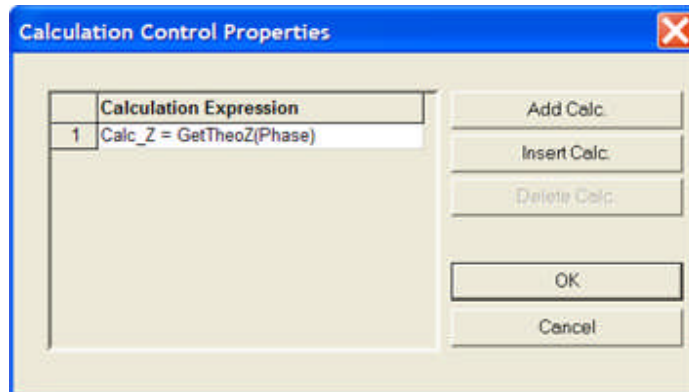
Loop Control: *Define location of test vectors*. Defines the starting test angle to be between the MTA and the zero degree reference; incremental degree test steps of 30 degrees; ending test angle to be the MTA plus the incremental degree setting. The test will begin first at the MTA minus 30 degrees, second point at the MTA and third test point at the MTA plus 30 degrees.



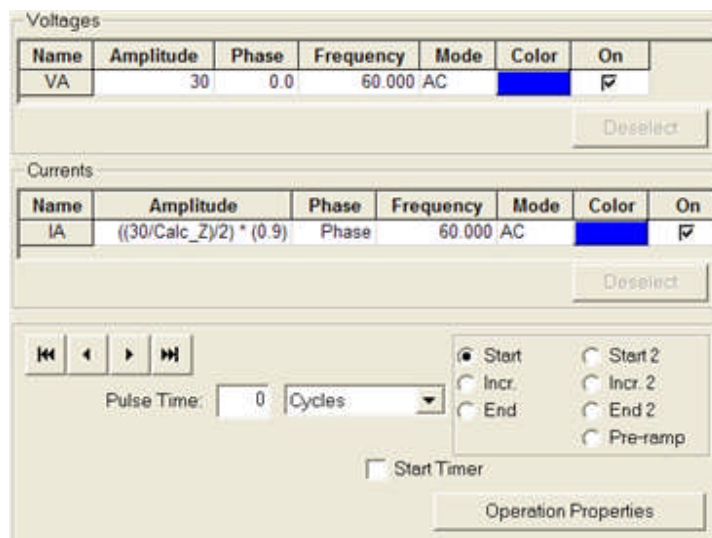


**Note:** The Control Counter Name is Phase which is the angle to be set in the Ramp Control.

Loop Calculation Control: *Define test characteristic test point.* Redefine GetTheoZ(Phase) to Calc\_Z and get theoretical response point for comparison to the device response in the test results.



Loop Ramp Control: *Define ramp Start test parameters.* Set constant voltage amplitude and phase angle output. Set current ramp start amplitude to  $((30/\text{Calc\_Z})/2)$  and Phase to Phase (the Loop Counter Name); or,  $I_a = V_a / \text{Characteristic Z}$  divided by 2 for phase-to-phase impedance test using one current, similar to G.E CEY mho relays.



Operation Properties dialog: *Define monitor contact properties*. Sets a test unit input post to monitor response of device under test.

*Define ramp Incr. test parameters*. Set the increment magnitude of the parameter to be ramped from the Start parameters; Current Amplitude = 0.1.

Hint: Set Incr. ramp rate to a percent of the Start quantity. The ramp values will be directly related to high or low values set in the relay.

Note: Increment Time set to 20 cycles. The Time must be set longer than the relay zone delay or operate time to allow relay response.

*Define ramp End test parameters*. Set end amplitude 150% of Start amplitude; which is much more than the expected accuracy limits of the device in test.

Voltages

Name	Amplitude	Phase	Frequency	Color
VA	0	0.0	60.000	

Currents

Name	Amplitude	Phase	Frequency	Color
IA	$((30/\text{Calc\_Z}/2) * 0.9) * 1.5$	0.0	60.000	

Navigation: Start, Incr., End, Start 2, Incr. 2, End 2, Pre-ramp

Loop Log Control: *Define variables for results.* The test results will display a row of Variables labels and a row of results under each Variable for each set of test parameters to be applied in each loop to Loop Control End Expression.

Log Control Properties

Variables To Save

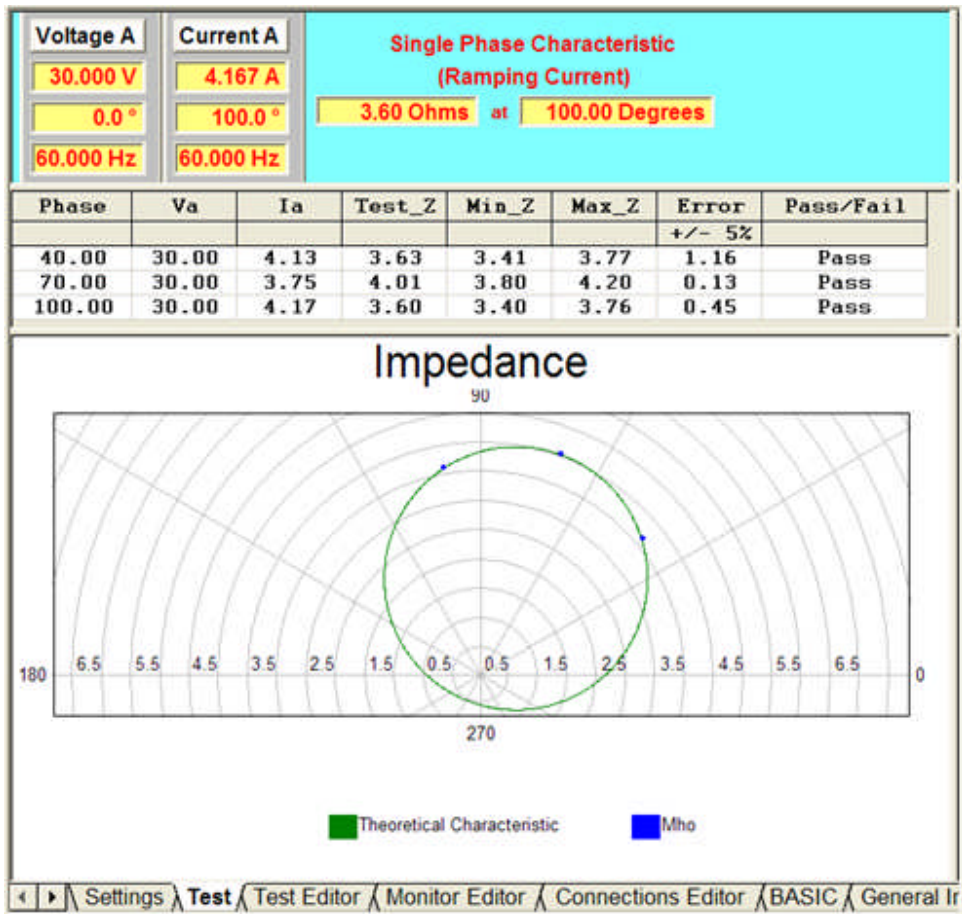
	Variable	Units	Decimal Places	Include in Report	Align	Width
1	Phase		2	<input checked="" type="checkbox"/>	Center	8
2	Va		2	<input checked="" type="checkbox"/>	Center	7
3	Ia		2	<input checked="" type="checkbox"/>	Center	7
4	Test_Z		2	<input checked="" type="checkbox"/>	Center	7
5	Min_Z		2	<input checked="" type="checkbox"/>	Center	7
6	Max_Z		2	<input checked="" type="checkbox"/>	Center	7
7	Error	+/- 5%	2	<input checked="" type="checkbox"/>	Center	7
8	Pass/Fail\$		0	<input checked="" type="checkbox"/>	Center	11

Pre-Log Calculations

	Calculation Expression
1	$\text{Test\_Z} = (V_a) / (I_a) / 2$
2	$\text{Min\_Z} = \text{GetTheoZ}(\text{Phase}) * 0.95$
3	$\text{Max\_Z} = \text{GetTheoZ}(\text{Phase}) * 1.05$
4	$\text{Error} = (\text{Test\_Z} - \text{Calc\_Z}) / \text{Calc\_Z} * 100$
5	$\text{if} (\text{Test\_Z} \geq \text{Min\_Z}) \text{ and } (\text{Test\_Z} \leq \text{Max\_Z}) \text{ then Pass/Fail\$} = \text{"Pass"} \text{ else Pass/Fail\$} = \text{"Fail"}$

Options: Units at Top, Units Beside, Evenly Spaced, # Spaces Between Columns

*Test Result output.*



Graphic Control: *Define the graphic characteristic plot.* Controls the graphic output format. The Magnitude = Test\_Z and Phase = Phase are the coordinate parameters for plotting the Theoretical characteristic. These variable labels **must** be defined and included in the Log Control Variables list, whether or not they are to be printed in the test results.

**Graphic Control Properties** ✖

Scale Type: Polar ▾

Traces

	Magnitude	Phase	Trace Label	Mode	Scale Group	Shape	Color
1	Test_Z	Phase	Mho	Scatter	Scale Group 1	Circle	<span style="background-color: blue; width: 15px; height: 10px; display: inline-block;"></span>
2				Line	Scale Group 1	Circle	<span style="background-color: green; width: 15px; height: 10px; display: inline-block;"></span>
3				Line	Scale Group 1	Circle	<span style="background-color: cyan; width: 15px; height: 10px; display: inline-block;"></span>

Group Test Overlays

1 None ▾

3 None ▾

4 None ▾

5 None ▾

Report Options

Color  Monochrome

Large  Small

Graph Title: Impedance

Save Image

Auto Scale

Plot Moving Characteristic

Plot Theoretical Characteristic

Theoretical Characteristic

Min/Max Theoretical:

Min/Max Theoretical Characteristic

OK Cancel

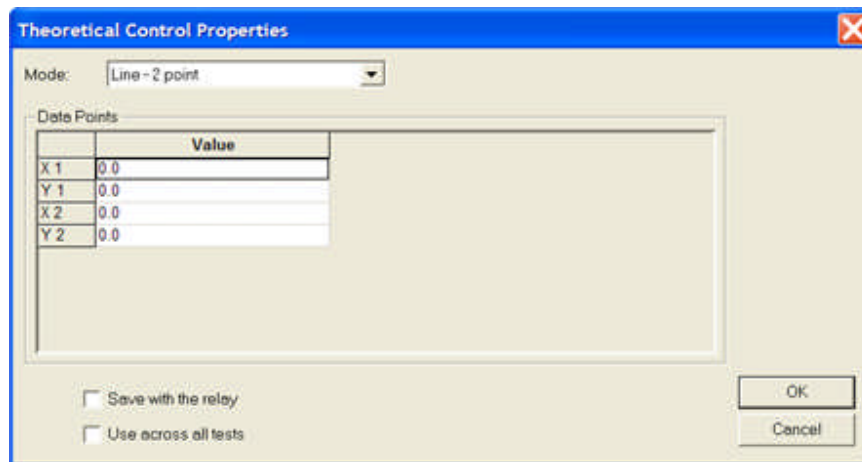
## Theo Line - 2 Point

**Mode Line - 2 Point** characteristic may be used as a valid characteristic. The two points are to describe a line characteristic which may depict an impedance (reactance) or definite time characteristic.

An example of this Control is to replace the Theo Control in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

**Note:** GetTheoTime(Phase) will retrieve proper results for values between the two points used to describe the line. The characteristic does not extend beyond the two points and attempts to perform tests outside will error.

The *Line - 2 Point Properties* dialog has entry fields for the two Cartesian coordinate pairs to construct a line.



Example:

Z = 10 ohms and MTA = 75 degrees

Possible calculations

$$X1 = Z / \tan(\text{MTA} - 35)$$

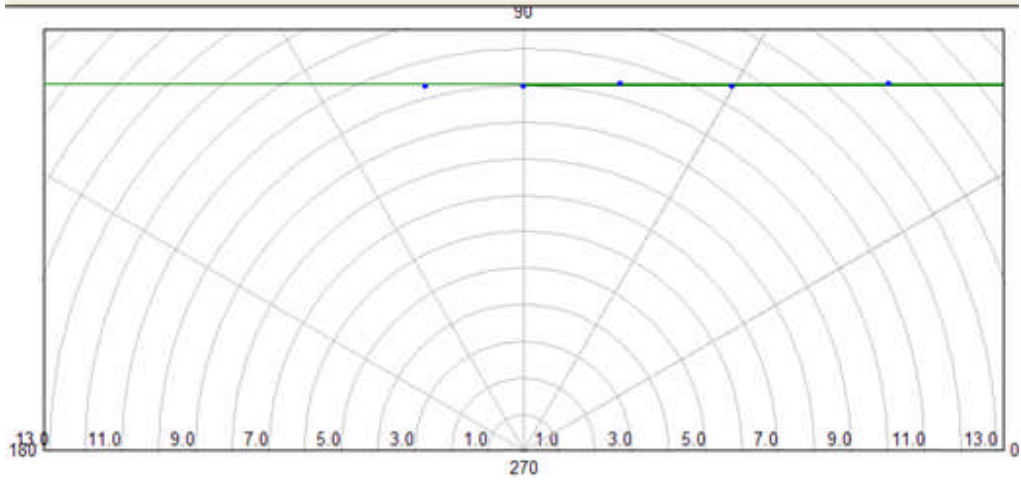
$$Y1 = Z$$

$$X2 = -Z / \tan(180 - \text{MTA} - 35)$$

$$Y2 = -Z$$

The resultant *Impedance characteristic results*.

Phase	Va	Ia	Test_Z	Min_Range	Max_Range	Pass/Fail
45.00	30.00	2.12	14.16	13.44	14.85	Pass
60.00	30.00	2.62	11.45	10.97	12.12	Pass
75.00	30.00	2.90	10.36	9.83	10.87	Pass
90.00	30.00	3.02	9.92	9.50	10.50	Pass
105.00	30.00	2.92	10.27	9.84	10.87	Pass

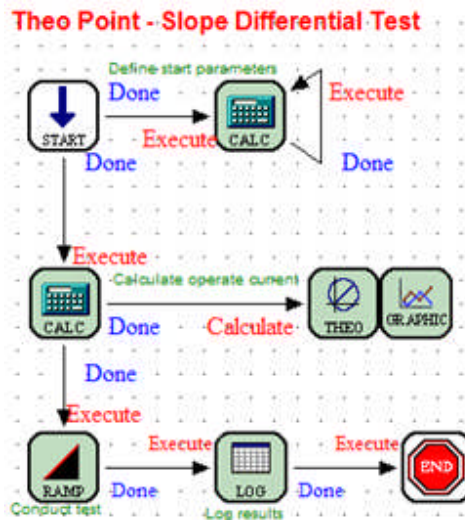


## Theo Line Point/Slope

**Mode Line - Point/Slope** characteristic may be used as a valid characteristic. The point with the slope is used to describe a line at a slope characteristic.

The Line - Point/Slope Properties dialog has entry fields for the two Cartesian coordinate pairs to construct a line. The Graphics Control Properties Scale Type: should also be changed to Linear.

Example *Test Editor* tab Constructed Test with Controls Linked



Settings in the Settings tab

- T1 = value from 2.6 to 8.7
- T2 = value from 2.6 to 8.7
- IRes = Restraint Current = 20
- Slope% = Percent Slope = 35%

*Calculation Control* entries

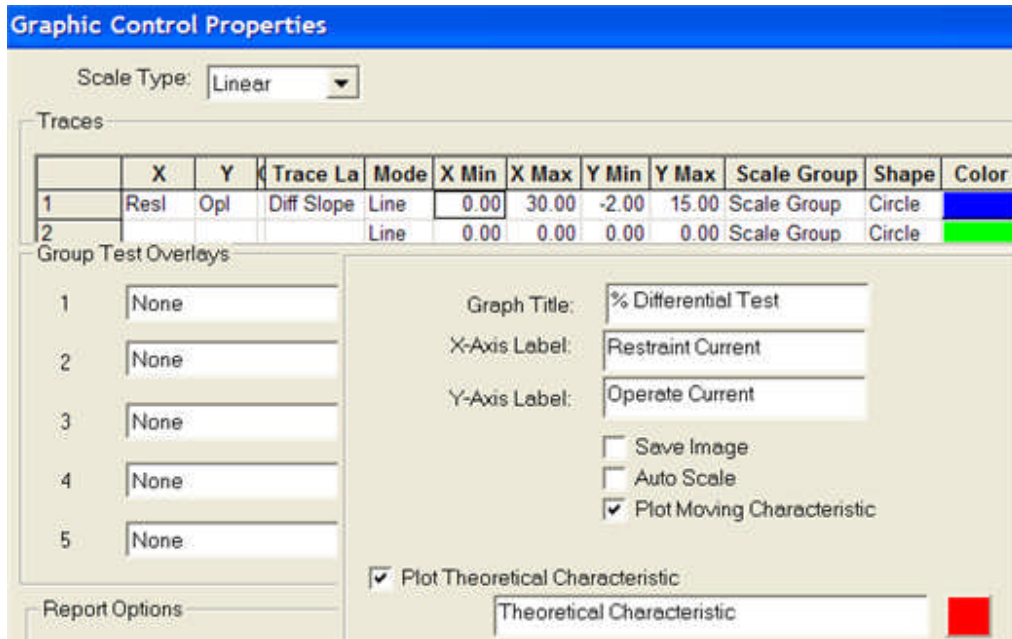
Calculation Control Properties	
	Calculation Expression
1	$Op1 = ((T1 / T2) * IRes * Slope\%) / 100$
2	$Res1 = T1 / T2 * IRes$

*Theo Line - Point/Slope Control* entries

Theoretical Control Properties	
Mode:	Line - Point/Slope
Data Points:	
	Value
X	Ires
Y	Ires * slope% / 100
Slope	slope% / 100

*Graphic Control* entries





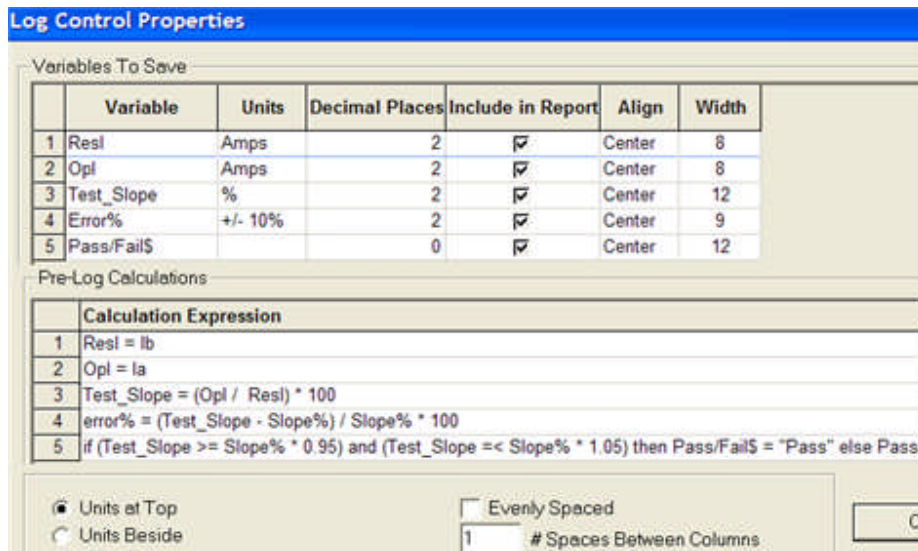
Ramp Control

Start  $I_A = Opl * 0.85$ ;  $I_B = T1 / T2 * IRes$

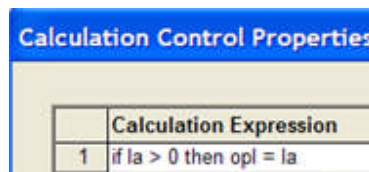
Incr  $I_A = Opl * 0.9 * 0.01$ ;  $I_B = 0$ ; Increment Time = 20 Cycles

Stop  $I_A = Opl * 1.50$ ;  $I_B = 0$

Log Control Entries



Calculation Control for continuous current vector display for executed test



Test tab Executing Vector

ResI	OpI	Test_Slope	Error%	Pass/Fail
Amps	Amps	%	+/- 10%	

### % Differential Test



Test tab *Test Results*

ResI	OpI	Test_Slope	Error%	Pass/Fail
Amps	Amps	%	+/- 10%	
20.00	6.96	34.79	-0.60	Pass

### % Differential Test



## Theo Ellipse

The Theoretical Control properties for the **Mode Ellipse** is very similar to Mode Circle - Max Torque and are to enter the coordinates of the two focus points and diameter necessary to define an ellipse.

**Note:** The Ellipse characteristic may have existed for early model discrete component relays where PC board timers were varied in time to set impedance reach and phase angle. Particular timer settings created this characteristic.

**Note:** Construction of the Ellipse may be more easily accomplished by using the Theoretical Object Mode.

An example of this Control is to replace the Theo Control and add the Calculation Control items in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

An Ellipse is a special case of a circle. Let the circle diameter be on the MTA axis; then a diameter perpendicular to the MTA at the diameter center will be a secondary diameter. For a Circle, the two diameters are equal. The secondary diameter for an ellipse will be less than the main diameter. The ellipse will have 2 focus points and the main diameter. The secondary diameter will be adjusted by the distance between the two focus points.

Mode selected from drop list: Ellipse

Focus 1X is to enter a coordinate value for the horizontal or abscissa axis. The value may be a numeric or a variable.

Focus 1Y is to enter a coordinate value for the vertical or ordinate axis. The value may be a numeric or a variable.

Focus 2X is to enter a coordinate value for the horizontal or abscissa axis. The value may be a numeric or a variable.

Focus 2Y is to enter a coordinate value for the vertical or ordinate axis. The value may be a numeric or a variable.

**Note:** If Focus 1X = Focus 1Y and Focus 2X = Focus 2Y, the two foci become the center of a circle with center at (Focus 1X, Focus 1Y).

Diameter is to enter a value for the distance of the semimajor axis, the two farthest points of the ellipse. The larger the diameter for the same given focus points, the ellipse also becomes wider.

**Note:** The generated characteristic is dependent upon numeric data values for each of the Data Points variable labels. The values may be given in the Workbook Settings tab or by a calculation or variable equality given in a Control prior to test call to the Theoretical Control.

Example: Find characteristic values for a defined Ellipse characteristic, ramping single phase current. Refer to the example in the Theo Circle - Max Torque for the data given to all Controls. The Calculation and Theoretical Controls are the only two alterations to the referenced example.

In the Calculation Control, create definitions for

the number of Parts to divide the Diameter

variable a = the number of parts from the diameter ends to place the two foci

a is a number from 1 to the 1/2 number of parts

when a = parts, characteristic is a circle

Offset = value to add to the Diameter to include the origin and leave the Reach of the Diameter = Z

adding an Offset will increase the secondary diameter, increase the number of parts to reduce secondary diameter

adding a negative Offset will decrease the secondary diameter, decrease the number of parts to reduce secondary diameter

In the Theo Data panel, letting the data reference MTA and Z and

$$X1 = (a * (z-Offset) / Parts ) * \cos(MTA)$$

$$Y1 = (a * (z-Offset) / Parts ) * \sin(MTA)$$

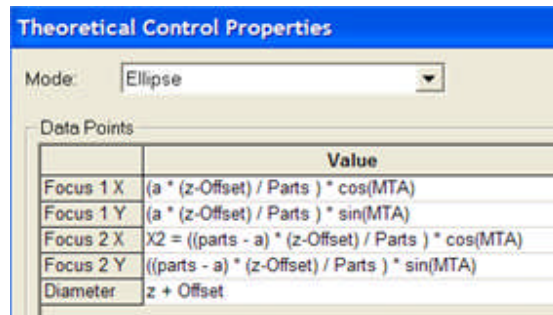
$$X2 = ((parts - a) * (z-Offset) / Parts ) * \cos(MTA)$$

$$Y2 = ((parts - a) * (z-Offset) / Parts ) * \sin(MTA)$$

$$Diameter = z + Offset$$

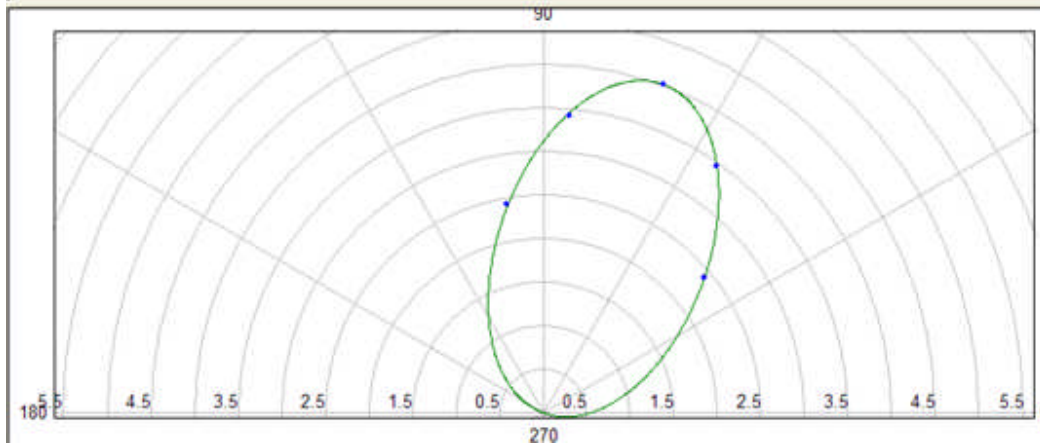
Calculation Control: No data is necessary if the Data points are numeric values.

Theoretical Control: *Define Characteristic*. Place values for data point entries Focus 1X, Focus 1Y, Focus 2X and Focus 2Y. Enter numeric values or define the variables either in the Workbook Settings tab or in a Calculation Control executed prior to the Theoretical Control.



*The resultant impedance characteristic.*

Phase	Va	Ia	Test_Z	Min_Range	Max_Range	Pass/Fail
40.00	30.00	12.52	2.40	2.28	2.52	Pass
55.00	30.00	8.68	3.46	3.28	3.63	Pass
70.00	30.00	7.50	4.00	3.80	4.20	Pass
85.00	30.00	8.80	3.41	3.27	3.62	Pass
100.00	30.00	12.36	2.43	2.26	2.50	Pass



## Theo Lens

The Theoretical Control properties for the **Mode Lens** is very similar to Mode Circle - Max Torque and are to define the Maximum Torque Angle, Offset, Radius and Distance between Centers.

**Note:** The Lens characteristic exists for early model discrete component relays where PC board timers were varied in time to set impedance reach and phase angle. Particular timer settings created this characteristic, particularly for use as a blinder. A relay of this type was the MOD III and possibly others.

**Note:** The GE Multilin D60 modern microprocessor relay may have a characteristic similar to the Lens. The Lens may be easier to create using the **Theoretical Object Mode**.

An example of this Control is to replace the Theo Control and add the Calculation Control items in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

A Lens is a special case of a circle. Let the circle diameter be the MTA axis; then a diameter perpendicular to the MTA at the circle center will be a secondary diameter. For a Circle, the two diameters are equal. For a Lens, the secondary diameter will be less than the main diameter.

Mode selected from drop list: Lens

Max Torque Angle Value may be a numeric or a variable.

Offset Value may be a numeric or a variable.

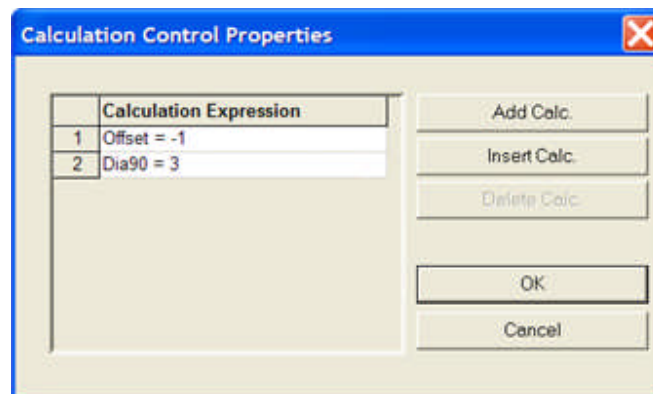
Max Reach may be a numeric or a variable.

Distance between Centers may be a numeric or a variable.

**Note:** The generated characteristic is dependent upon numeric data values for each of the Data Points variable labels. The values may be given in the Workbook Settings tab or by a calculation or variable equality given in a Control prior to test call to the Theoretical Control.

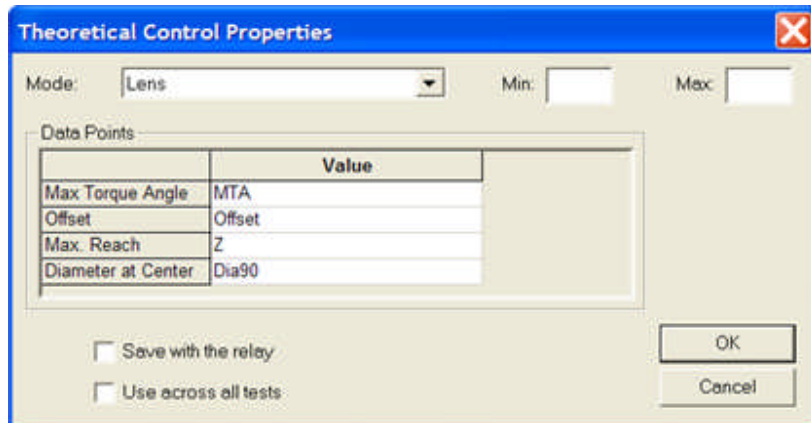
Example: Find three characteristic values for a defined Lens characteristic, ramping single phase current. Refer to the example in the Theo Circle - Max Torque for the data given to all Controls. The Calculation and Theoretical Controls are the only two alterations to the referenced example.

Calculation Control: *Define variable identities*. Defines a new value for Theoretical Control Diameter variable. Max Torque Angle variable may remain as MTA variable.



**Note:** Negative Offset for mho characteristic to not include origin in polar graph; positive Offset for mho characteristic to include origin in polar graph. Zero Offset places Lens at origin

Theoretical Control: *Define characteristic*. Place variable names for data points entries Diameter, Max Torque Angle and Offset. Enter numeric values or define the variables either in the Workbook Settings tab or in a Calculation Control executed prior to the Theoretical Control.



Workbook Settings tab: Z = 4 and MTA = 70.

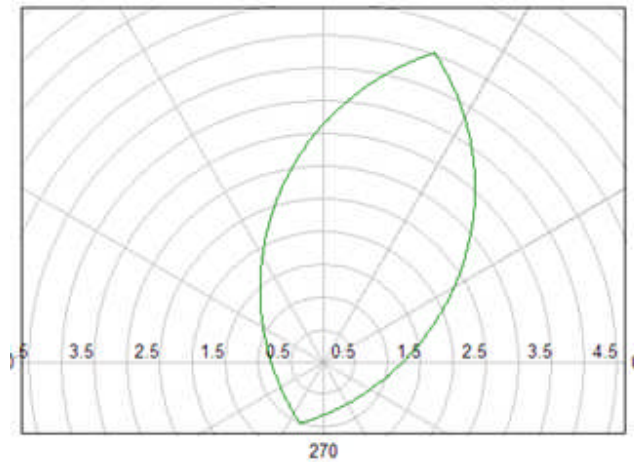
Max Torque Angle = MTA to reflect Workbook Settings tab variable setting. Example MTA = 70 degrees.

Offset = Offset to reflect Calculation Control setting. Example Offset = 0.0. Offset setting may be included in the Workbook Settings tab Group for the relay.

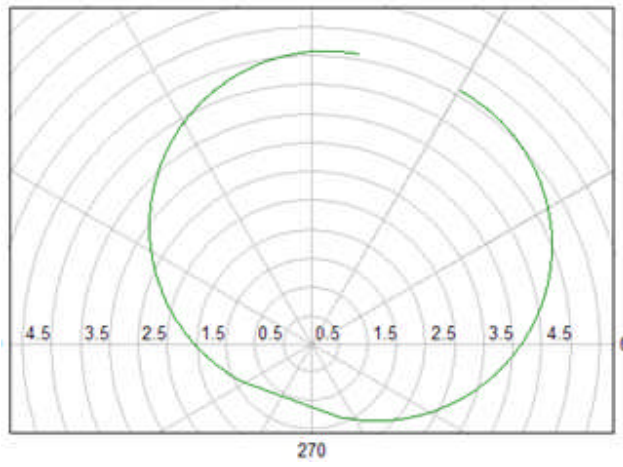
Max Reach = Z to reflect the characteristic. A positive Offset will add to Z and include the origin; a negative Offset will subtract from Z and not include the origin.

Diameter at Center = Dia90 to reflect the ohmic diameter at the characteristic center

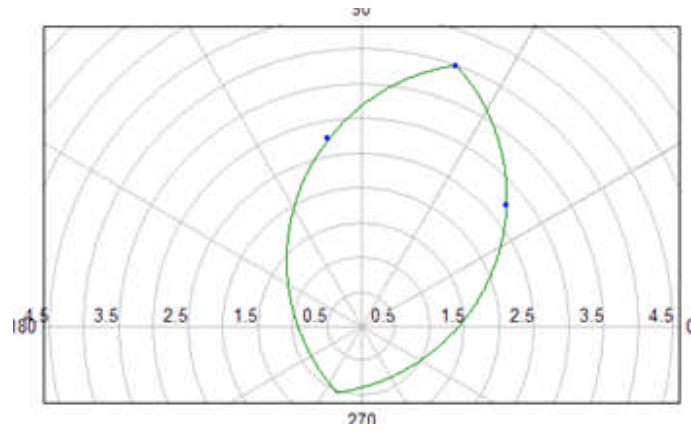
Note: For Lens  $Dia90 < Z_{total}$ , the characteristic is a Lens. If  $Dia90 = Z_{total}$ , the described characteristic is a circle.



Note: For  $Dia90 > Z_{total}$ , then the described characteristic is not valid and may be discontinuous. For this case, use a Theo Control for a Circle or the Theo Tomato.



*The resultant characteristic.*



## Theo Tomato

The Theoretical Control properties for the **Mode Tomato** is very similar to Mode Circle - Max Torque and are to define the Maximum Torque Angle, Offset, Radius and Distance between Centers.

**Note:** The Tomato characteristic exists for early model discrete component relays where PC board timers were varied in time to set impedance reach and phase angle. Particular timer settings created this characteristic. A relay of this type was the MOD III and possibly others.

**Note:** The GE Multilin D60 modern microprocessor relay may have a characteristic similar to the Tomato. The Tomato may be easier to create using the **Theoretical Object Mode**.

An example of this Control is to replace the Theo Control and add the Calculation Control items in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

A Tomato is a special case of a circle. Let the circle diameter be the MTA axis; then a diameter perpendicular to the MTA at the circle center will be a secondary diameter. For a Circle, the two diameters are equal. For a Tomato and holding Z constant, the secondary diameter will be greater than the main diameter, creating two overlapping circles with a common cord.

Mode selected from drop list: Tomato Data Points

Max Torque Angle is the Line angle MTA Value and may be a numeric or a variable.

Offset Value is the distance from the origin for the Line impedance center point and may be a numeric or a variable.

Radius is the size of the circle radii that will determine the common cord of the two overlapping circles and may be a numeric or a variable.

Distance between Centers is determined by the radii of the circles and the length of the common cord and may be a numeric or a variable.

**Note:** The generated characteristic is dependent upon numeric data values for each of the Data Points variable labels. The values may be given in the Settings tab or by a calculation or variable equality given in a Control prior to the test call to the Theoretical Control.

Example: Find characteristic values for a defined tomato characteristic ramping single phase current. Refer to the example in the Theo Circle - Max Torque for the data given to all Controls. The Calculation and Theoretical Controls are the only two alterations to the referenced example.

Let calculations rely on the values for MTA and Z in the Settings tab:

Common Cord of two circles = the relay reach or Z

Max Torque Angle = MTA

Offset =  $1/2 * Z$

Radius taken from the Reach Z and Distance Between Centers of the two circles =  $((Z^2) + (0.5 * Z^2))^{0.5} / 2$

The cord length does not always = Z; the following substitution is made to set the Cord = Z

$(0.85 * Z)^2$  for the first part  $(Z^2)$  of the equation

Distance Between Centers D = use some factor \* Z (user option)

Value should be less than the number 1 to overlap the two circles; if set greater than 1, the two circles separate and are connected by the line between centers.

Calculation Control: *Define variable identities*. Defines a Distance Between Centers D. Max Torque Angle variable may remain as MTA variable.



Calculation Control Properties	
	Calculation Expression
1	Group\$ = "fghgh_"
2	Set\$ = Group\$ + "Z": Z = GetSetting(Set\$)
3	Set\$ = Group\$ + "MTA": MTA = GetSetting(Set\$)
4	D = 0.45 * z

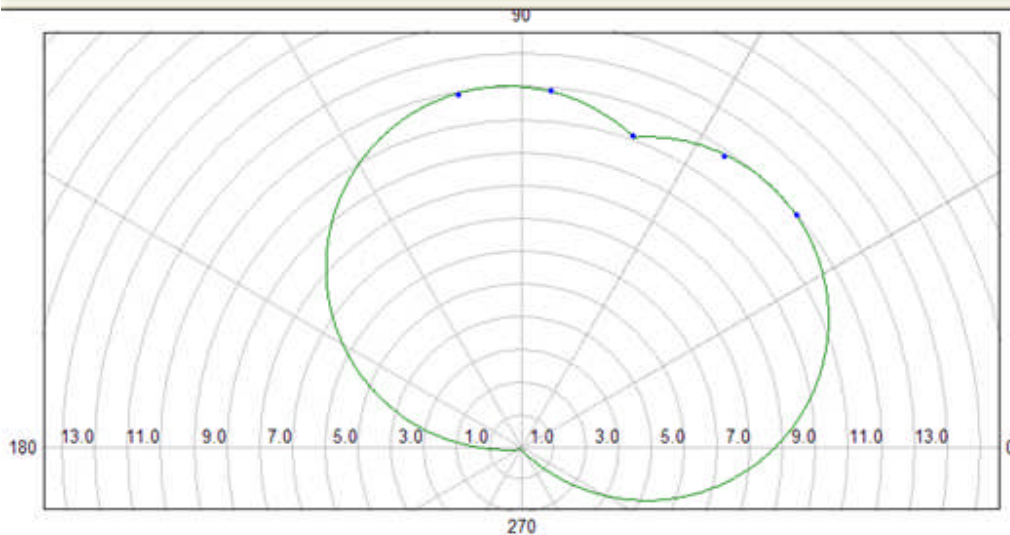
Note: Negative Offset for mho characteristic to not include origin in polar graph; positive Offset for mho characteristic to include origin in polar graph.

Theoretical Control: *Define characteristic*. Place variable names for data points entries Diameter, Max Torque Angle and Offset. Enter numeric values or define the variables either in the Workbook Settings tab or in a Calculation Control executed prior to the Theoretical Control.

Theoretical Control Properties	
Mode:	Tomato
Data Points	
	Value
Max Torque Angle	MTA
Offset	0.5 * Z
Radius	$((Z^2) + (0.5 * Z^2))^{0.5} / 2$
Distance between Centers	D

The example described above creates the *resultant characteristic*.

Phase	Va	Ia	Test_Z	Min_Range	Max_Range	Pass/Fail
40.00	30.00	2.74	10.97	10.41	11.50	Pass
55.00	30.00	2.78	10.78	10.33	11.41	Pass
70.00	30.00	2.98	10.06	9.55	10.55	Pass
85.00	30.00	2.76	10.88	10.33	11.41	Pass
100.00	30.00	2.76	10.87	10.41	11.50	Pass



## Theo Data - Polar Impedance

The Theoretical Control properties for the **Mode Polar - Impedance** are to define a characteristic using polar coordinates by numerous data points of Phase angle and Impedance reach at that angle.

Use this Mode for creating characteristics that may have unusual geometric shapes. In the use of discrete data points, it will be recognized that the characteristic generated will be by a straight line drawn from point to point. Several lines of values will need to be entered to fully describe the characteristic. Data lines are generated using the Add button.

An example of this Control is to replace the Theo Control in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

Mode selected from drop list: Data - Polar Impedance

Phase is to enter a polar phase angle value in degrees to define a value of impedance. The value may be numeric or a variable.

Impedance is to enter a value of impedance in ohms at the entered Phase angle. The value may be a numeric or a variable.

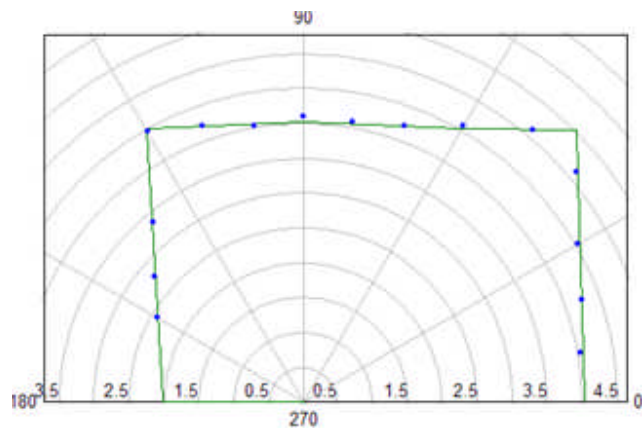
Example: Find three characteristic values for a defined characteristic, ramping single phase current. Refer to the example in the Theo Circle - Max Torque for the data given the to all Controls. The Calculation and Theoretical Controls are the only two alterations to the referenced example.

Calculation Control: No data is necessary if the Theoretical Data Points entries to describe the characteristic are numeric values. If the data points are variables, the values may be taken from another Control for from the Settings tab.

Theoretical Control: *Enter the Data Points*. The data points will need be determined prior to entry in the Phase and Impedance fields.

	Phase	Ohms
1	0	4
2	45	5.5
3	60	4.5
4	90	4
5	120	4.5
6	180	2
7	270	0

The example described above creates the *resultant impedance characteristic*.



## Theo Data - Impedance

The Theoretical Control properties for the **Mode Impedance** are to define a characteristic using numerous data points of X and Y values in rectangular coordinates.

Use this Mode for creating characteristics that may have unusual geometric shapes. In the use of discrete data points, it will be recognized that the characteristic generated will be by a straight line drawn from point to point. Several lines of values will need be entered for fully describe the characteristic. Data lines are generated using the Add button.

An example of this Control is to replace the Theo Control in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

Mode selected from drop list: Real and Imaginary

Real is to enter a coordinate value for the horizontal or abscissa axis. The value may be a numeric or a variable.

Imaginary is to enter a coordinate value for the vertical or ordinate axis. The value may be a numeric or a variable.

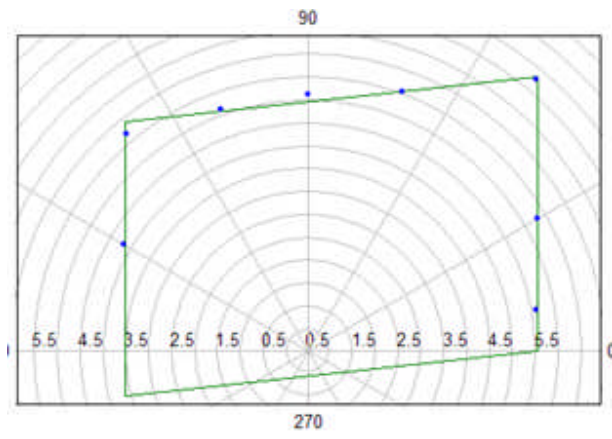
Example: Find three characteristic values for a defined characteristic, ramping single phase current. Refer to the example in the Theo Circle - Max Torque for the data given to all Controls. The Calculation and Theoretical Controls are the only two alterations to the referenced example.

Calculation Control: No data is necessary if the Theoretical Data Points entries to describe the characteristic are numeric values. If the data points are variables, the values may be taken from another Control for from the Settings tab.

Theoretical Control: *Enter the Data Points*. The data points will need be determined prior to entry in the Phase and Impedance fields.

	Real	Imag.
1	5	0
2	5	6
3	-4	5
4	-4	-1
5	5	0

*The resultant impedance characteristic* test results graph.



## Theo Data - Time-Amplitude

The Theoretical Control Properties for the **Mode Data - Time-Amplitude** option is to create a characteristic not governed by standard formulae.

An example of this Control is to replace the Theo Control in Test Editor OC Time Example. Scale alterations were also made to the Graphic Control. No other changes are made and the results are shown at the end of this topic.

The abscissa (X axis) is used to enter the Amplitude data.

The Ordinate (Y axis) is used to enter the Time data for each Amplitude entry.

In this example the Amplitudes were selected for the current values for the Multiples of current to be applied. If the Tap = 1, then the Multiple will be equal to the abscissa values.

The *Theo Control* entries.

Theoretical Control Properties

Mode: Data - Time-Amplitude

Data Points		
	Time	Amplitude
1	1	1
2	1	3
3	2	4.5
4	3	6
5	4	7.5
6	5	9
7	6	11.5
8	7	13

The *Graphic Control* entries.

Graphic Control Properties

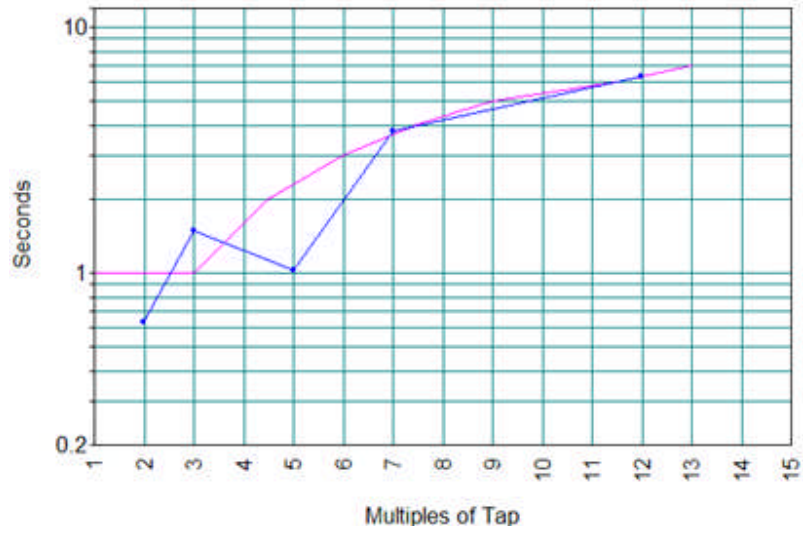
Scale Type: Semi-Log  Logarithmic X-Axis

Traces								
	X	Y	Mode	X Min	X Max	Y Min	Y Max	Scale
1	Multiple	PulsarTime	Scatter	1.00	15.00	0.20	12.00	Scale

The Test tab *Graphic Results* of an executed test. In this example the Graphic Control Mode drop list was changed from Scatter to Line. Line mode connects the results points; whereas, scatter plots only the resultant points.

Multiples	Applied Current	Time	Expected Time	Error
X Tap	Amps	Seconds	Seconds	%
2.00	8.00	0.63	1.00	-37.50
3.00	12.00	1.48	1.00	48.40
5.00	20.00	1.02	2.29	-55.62
7.00	28.00	3.75	3.63	3.19
12.00	48.00	6.33	6.32	0.18

Time Amplitude (Multiple)



## Theo Data - Generic

The Theoretical Control Properties for the **Mode Data - Generic** option is to create a characteristic not governed by standard formulae.

An example of this Control is to replace the Theo Control in Test Editor OC Time Example. Scale alterations were also made to the Graphic Control. No other changes are made and the results are shown at the end of this topic.

The abscissa (X axis) is used to enter the X amplitude data.

The Ordinate (Y axis) is used to enter the Y time data for each Amplitude entry.

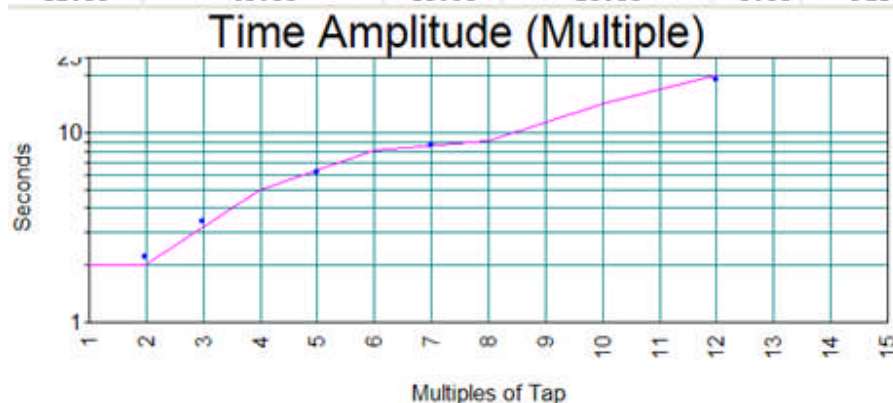
In this example the Amplitudes were selected for the current values for the Multiples of current to be applied. If the Tap = 1, then the Multiple will be equal to the abscissa values.

The *Theo Control* entries.

Theoretical Control Properties		
Mode:	Data - Generic	
Data Points		
	X	Y
1	0	2
2	2	2
3	4	5
4	6	8
5	8	9
6	10	14
7	12	20

The Test tab *Graphic Results* of an executed test. In this example the Graphic Control Mode drop list was changed from Line to Scatter. Line mode connects the results points; whereas, scatter plots only the resultant points.

Multiples	Applied Current	Time	Expected Time	Error	Pass/Fa
X Tap	Amps	Seconds	Seconds	%	
2.00	8.00	2.19	2.00	9.35	Fail
3.00	12.00	3.36	3.16	6.25	Fail
5.00	20.00	6.17	6.32	-2.41	Pass
7.00	28.00	8.52	8.49	0.35	Pass
12.00	48.00	18.98	20.00	-5.08	Fail





## Theo Calculations

The Theoretical Control **Mode Calculations** option is to enter data which may include the formulae for characteristics. The time overcurrent formulae used with microprocessor and solid-state relays typically use the IEEE or IEC standard curves. However, different manufacturers have different formulae for each family of curves. To assure accuracy in performed tests results, using the appropriate formula for a curve for a specific manufacturer is advised. Some manufacturers use more than one formula for a single curve or family of curves. This Control provides the ease to accommodate the required accuracy in testing.

The Phase Time Overcurrent test using the Theoretical Control Calculations option is demonstrated in the Test Editor Microprocessor Time OC Example 3 topic.

**Note:** The Theoretical Calculations has a dialog different to the Calculation Control. Both options can perform the same functions.

The example is a part of the SEL-321 relay included with the installation of AVTSv.2 RelayTestLibV2.mdb.

The *Theoretical Calculations* properties are evoked from the Edit button in the Theoretical Control Properties dialog. The areas of data entries are Type, Variable Name and Calculations.

Type Polar or Linear check buttons to select the appropriate graph display.

Variable Name: area edit fields

X: the variable to be used for the Abscissa label and formulae entered in Calculations or in other Controls.

Y: the variable to be used for the Ordinate label and formulae entered in Calculations or in other Controls.

Iteration Variable: the variable label to use for several test executions in sequence.

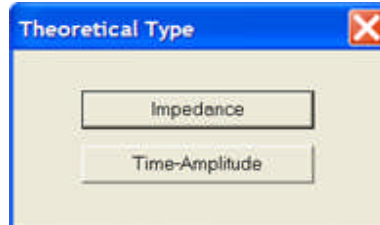
Range: is usually numeric numbers to set the Min and Max values of calculations from the formulae.

Step: is the incremental jump from one calculation to the next.

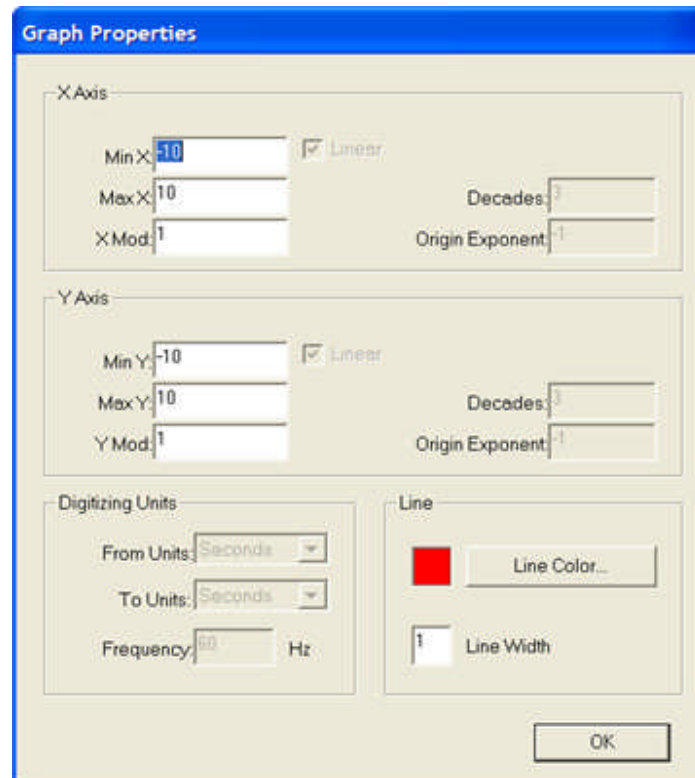
## Theo Theoretical Object - Impedance

# Theo Theoretical Object - Impedance

The Theo Control option Theoretical Object Edit button will display the *Theoretical Type* dialog.



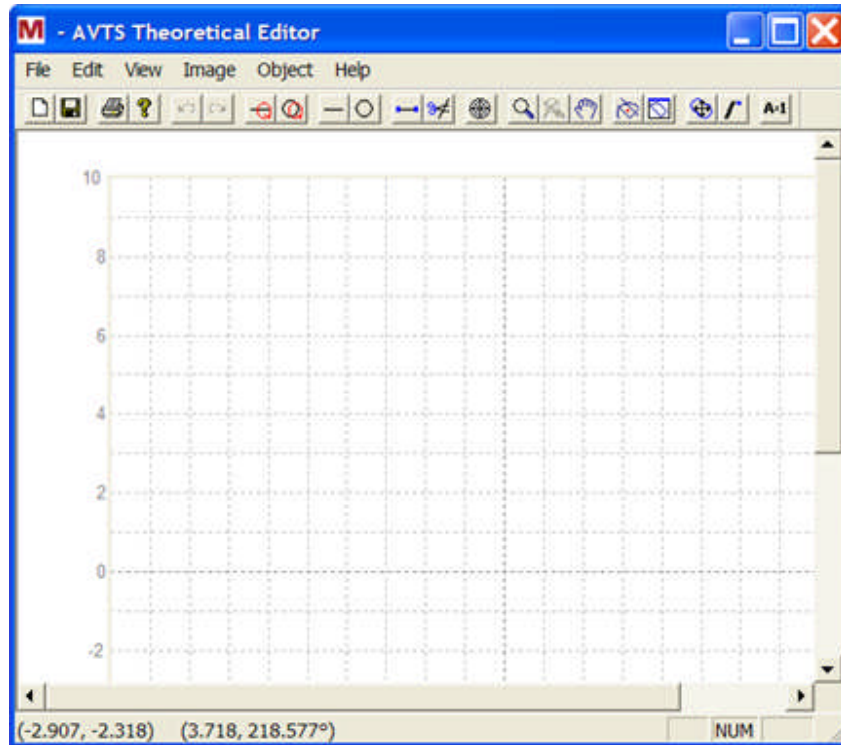
The Impedance and Time\_Amplitude buttons are to select and open the *Graph Properties* dialog, properties for Impedance are demonstrated. The Time-Amplitude option is described in the Theo Theoretical Object - Time-Amplitude group of topics.



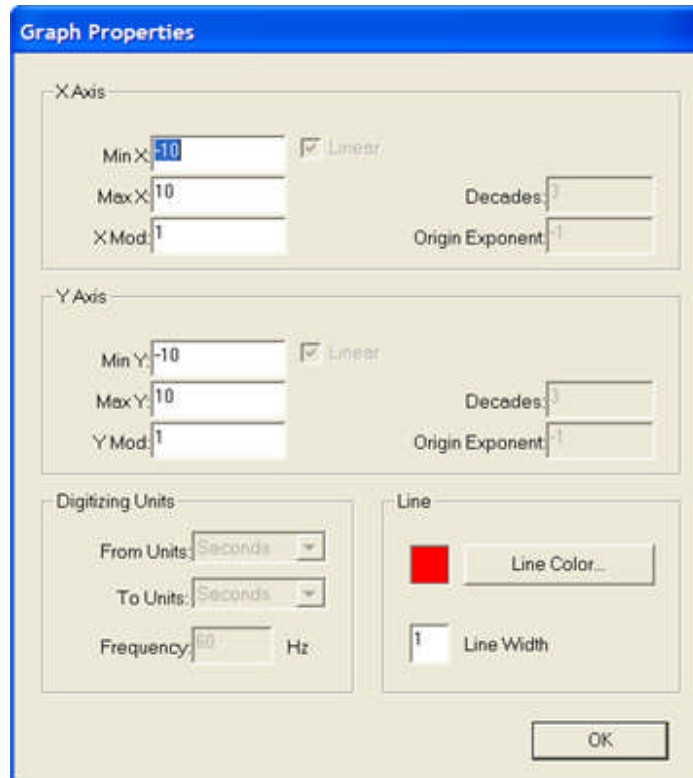
Substitution of this selection in place of the Theoretical Control and editing the associated Calculation Control in the Test Editor Charc Z Example 2 topic demonstrate the use.

## Theo Impedance Toolbar & Menu

When the Impedance option is selected, the *AVTS Theoretical Editor* dialog opens for construction of an Impedance characteristic.



The *Graph Properties* dialog displays four panes for setting the Linear boundaries of the graph.



The characteristic will need to be a closed object using the available tools from the toolbar.

**The toolbar has the following tools defined from left to right:**



**New** (File item) opens the Theoretical Type dialog to select either a new Impedance or Time-Amplitude characteristic.

**Save** (File item) is to place the presently open AVTS Theoretical Editor file in the active database.

**Print** (File item) opens the Windows Printer Selection dialog to print the characteristic.

**About** AVTS Theoretical Editor (Help item) dialog provides the version in the installed AVTS software.

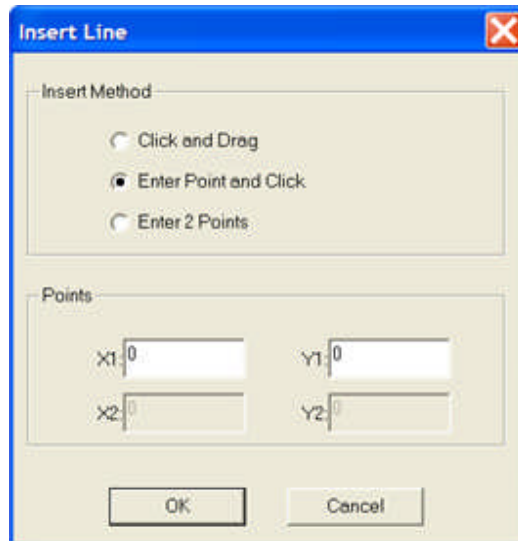
**Undo** (Edit item) is to return to the action previous to the last action performed.

**Redo** (Edit item) is to return to the action previous to the last Undo.

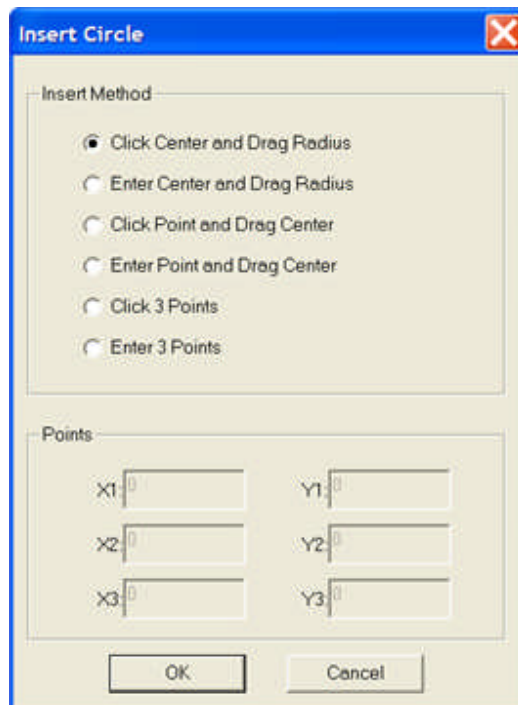
**Quick Line** (Object item), when clicked, highlights the Insert Line tool. The tool allows a single Click and Drag a general line on the graph. Line termination will occur with a second click.

**Quick Circle** (Object item), when clicked, highlights the Insert Circle tool. The tool enables the mouse to create a circle by placing the cursor in the circle center location and single left click. Drag the radius of the circle to the size desired and single left click to anchor the circle.

**Insert Line** (Object item) opens the *Insert Line* dialog to enter the two end points X and Y values or to allow the entry of another line using the same dialog as the Quick Line.



**Insert Circle** (Object item) opens the *Insert Circle* dialog to select the method to construct the circle.



Click Center and Drag Radius enables click a point for the circle center and drag the circle to the radius length.

Enter Center and Drag Radius enables the X1,Y1 point entry fields for the circle center and drag the circle the radius length.

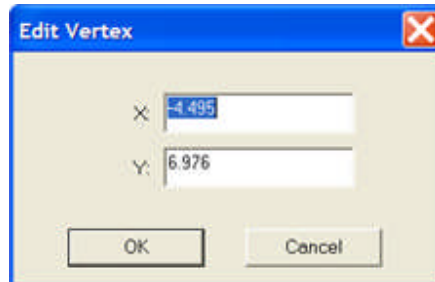
Click Point and Drag Center enables click a point on the circle and drag the circle center the radius length.

Enter Point and Drag Center enables the X1,Y1 point entry fields for a point on the circle and drag the center the radius length.

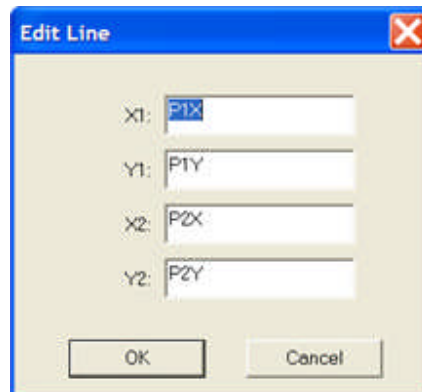
Click Three Points enables three mouse clicks to determine the circle.

Enter Three Points enables Points edit fields for three sets of X and Y entries.

**Edit Object** (Object item) immediately highlights all drawn lines for selection and edit of an end point using the *Edit Vertex* dialog of each as chosen.



Once all points of the Object have been identified, the Object will need to use the Edit Object tool. Select each line to evoke the *Edit Line* dialog and identify in either clock-wise or counter-clock-wise the points variable labels.



**Clip Section** (Object item) allows removing sections of lines and circles (arcs) to create the characteristic.

**View Polar** changes the original AVTS Theoretical Editor from the Cartesian graph view to a polar graph view; select the View Polar again returns the Cartesian graph.

**Zoom** will increase the view size with each mouse click.

**Zoom Normal** returns any Zoom to the original view size.

**Pan** changes the cursor to a hand to grab an area and move the graph view.

**Center of Operation** (Object item) allows placement of an arrow inside the characteristic object, positioned so that there are no hidden edges as seen by the Center of Operation. Failure to position the Center correctly will yield no or inaccurate results when the test is run.

**Note:** Should this occur, return to the Theoretical Editor dialog and edit the Center of Operation.

**View Digitized** displays the characteristic as viewed by the Center of Operation. If the Center of Operation is positioned so that there are hidden edges, the characteristic will change to that where all edges are seen. If the Center is placed so all edges can be seen and if when View Digitized seems to change the shape slightly, it is to move the Center so slightly that all of the edges points can be seen.

**Select and Move** (Object item) enables the cursor to change to crossed-double-headed arrows to select any part of the object to move.

**Snap Points** (Edit item) allows discrete values of line and circle construction and object movement; the values of which appear in the Status bar.

When not selected, the X and Y coordinates of the cursor will be displayed to three decimal places (thousandths). When selected, the X and Y coordinates of the cursor will be displayed to one decimal place (tenths). The polar coordinates are displayed to three decimal places (thousandths) in either case.

**Variables** (Edit item) opens the Variables dialog to display the Variables dialog.

The Name drop list will display the value for each Cartesian point of a created object with the Center of Operation. The dialog may be used to enter new points and values to create an object. Once all points of the Object have been identified and given a Cartesian value, the Object will need to use the Edit Object tool, select each line and identify in either clock-wise or counter-clock-wise the points variable labels.

**The following menu items do not have a toolbar equivalent:**

**File/Print Setup** opens the standard Windows Select Printer dialog.

**File/Exit** opens an AVTS Theoretical Editor message to save the changes to the object, if edited from last **Save**. Yes option saves edits and exits, No exits with no Save; Cancel returns to the Theoretical Editor graph object.

**Edit/Erase All** opens an AVTS Theoretical Editor notice that the Erase All cannot be undone. Continue Yes erases all and Continue No returns to the Theoretical Editor graph object.

**Edit/Select All** (use in Time-Amplitude graph only)

**Edit/Graph Properties** returns to the Graph Properties dialog to edit the graph type and limits previously set.

**Edit/Grid Color**, default gray, opens the Color dialog with the color palate to select the color of the graph grid.

**View/Toolbar**, default checked and visible, hides the Toolbar if unchecked.

**View/Status Bar**, default checked and visible, hides the Status Bar if unchecked. The Status Bar displays Cartesian (X,Y) and Polar (Radius from 0,0,Angle) regardless which graph type is in display.

**View/Background Image** (use in Time-Amplitude graph only)

**View/Theoretical** displays the constructed object.

**View/Max Tolerance** when selected allows the building of a minimum tolerance characteristic. The build is separate from the maximum and the theoretical characteristics first built.

**View/Min Tolerance** when selected allows the building of a maximum tolerance characteristic. The build is separate from the minimum and the theoretical characteristics first built.

**View/Theoretical** with Tolerances displays the object and disables all object construction tools except the Print, Polar, Zoom, Pan and View Digitized tools.

**Image/Insert Background Image** (use in Time-Amplitude graph only)

**Image/Remove Background Image** (use in Time-Amplitude graph only)

**Image/Map Top Left Image Point** (use in Time-Amplitude graph only)

**Image/Map Bottom Right Image Point** (use in Time-Amplitude graph only)

**Image/Cancel Image Mapping** (use in Time-Amplitude graph only)

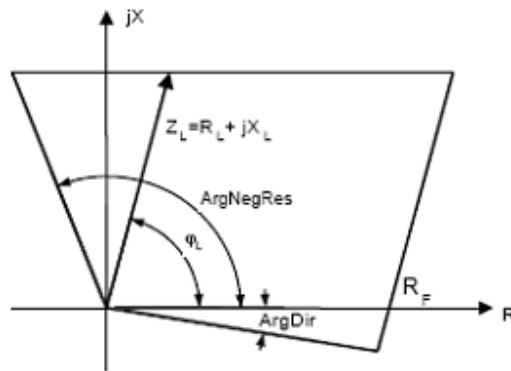
**Object/Edit Center of Operation** opens the Edit Center of Operation dialog to edit the X and Y values for the Center of Operation.

### Theo Quadrilateral Example

## Theo Quadrilateral Example

There exists impedance characteristics which do not fit the options of settings in the Theoretical Control Properties drop list. The Theoretical Object in the list of options allows construction of those characteristics alien to the other listed options.

The *ABB REL 521 impedance quadrilateral characteristic* is selected as an example to construct for this option.



The following topics step through the construction to and include an example test result:

Theo Quadrilateral Impedance Construction

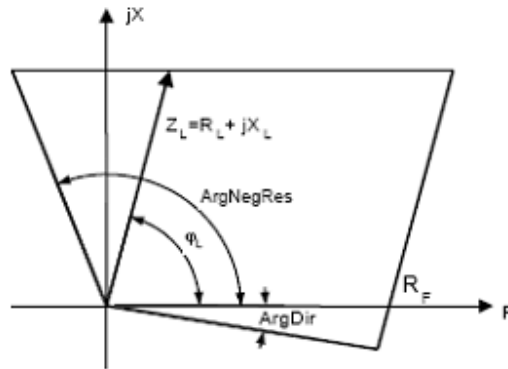
Theo Quadrilateral Impedance Calculations

Theo Quadrilateral Impedance Example



## Theo Quadrilateral Impedance Construction

Example: Construct the **ABB REL 521 impedance characteristic** which consists of the intersections of four straight lines.



Given:  $Z_L$  = the protected line impedance

$Z_{ang}$  = the protected line impedance angle

$R_F$  = the independent resistive reach of a protective zone determined by study

The static characteristic in the reactive direction is a straight line parallel with the R-axis.

A straight line limits the reach of the distance protection zone in the resistive direction. It is generally parallel with  $Z_L$ ; i.e., it forms  $Z_{ang}$  with the R-axis.

With the X-axis, the directional characteristic in the second quadrant forms an  $ArgNegRes$  settable angle (default value  $25^\circ$ ).

**Note: The  $ArgNegRes$  angle is the angle formed in the second quadrant between the line in the fourth quadrant and the ordinate. This is the value used in the REL 521 instruction manual for calculations.**

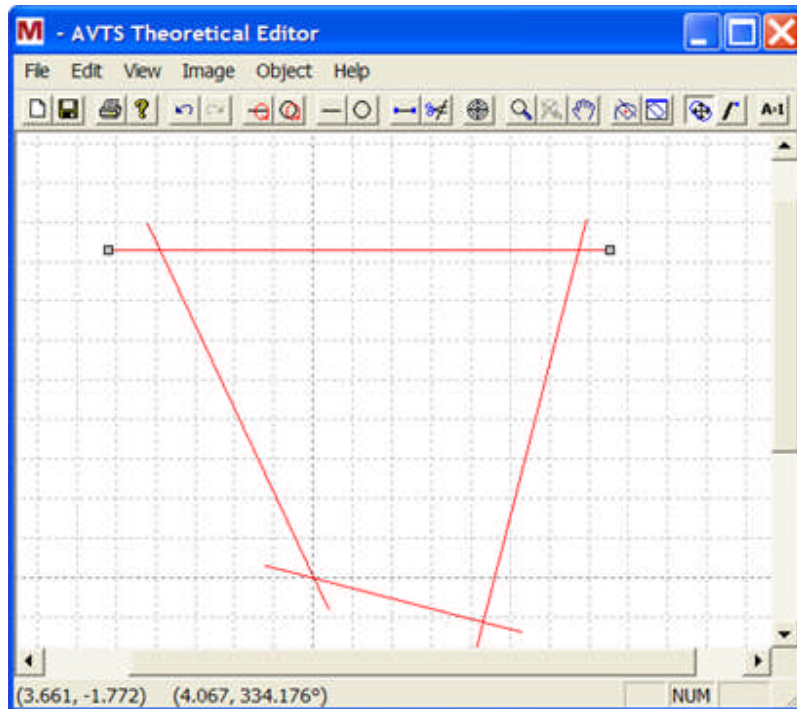
With the R-axis, the corresponding part forms a settable  $ArgDir$  angle in the fourth quadrant (default value  $-15^\circ$ ).


Roughly create the characteristic using the Quick Line tool


Select the Quick Line tool 

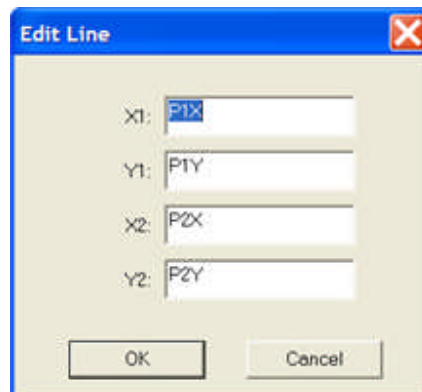
choose a location, click, drag to the second location and second click to anchor line

Continue to construct the remaining three *lines extending past the limits* of each intersecting line




Select the Clip Section tool  to "snip" the ends of the line extensions

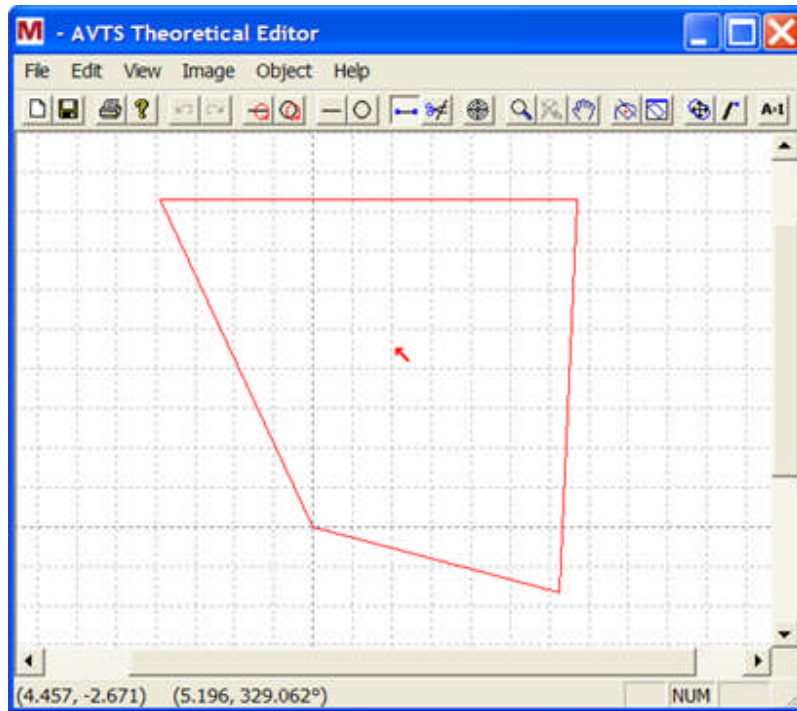
Select the Edit Object tool  to click on a line to label the *Cartesian properties*



The values shown when first opened are the constructed coordinates and the Edit Line properties dialog show the labels of the line ends in the edit fields replacing the values

**Note:** Select a line and enter the coordinates for X1, Y1 and X2, Y2 either CW or CCW. Continue the process placing labels on each line segment in turn labeling in the same direction sense.

Select the Center of Operation tool  and click on the interior of the characteristic to define the *Center of Operation*. The Center of Operation must be in a location that an imaginary line may be drawn to every location on the described characteristic boundaries

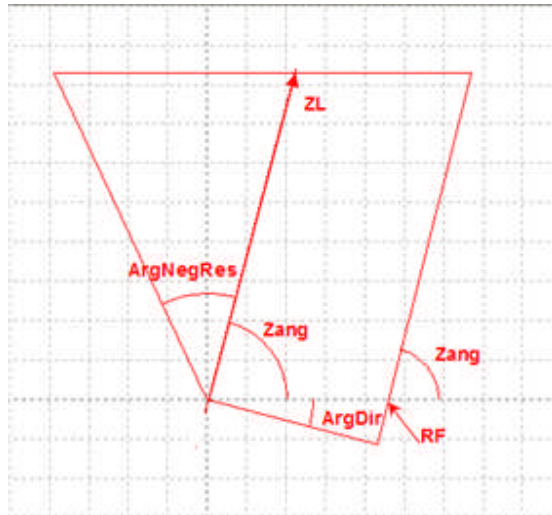


Select File/Save option

Select File/Exit option

## Theo Quadrilateral Impedance Calculations

Example: Determine the formulae for the *ABB REL 521 impedance characteristic* Cartesian coordinates of the characteristic vertexes. Use information in the Theo Object Impedance Construction topic.



For the line in the first and second quadrants consider the vertex in the first quadrant to be P1 and have coordinate labels (P1X,P1Y); this example considers vertex P2 to be in the 2nd quadrant, P3 at the origin and P4 in the 4th quadrant.

Calculate the abscissa and ordinate values ZRL and ZXL of the line impedance

$$ZRL = ZL * \cos(Zang)$$

$$ZXL = ZL * \sin(Zang)$$

**Warning:** Always use lower case letters for the trig functions; the interpreter does not allow upper case for these functions and will yield incorrect calculations. Review the Expressions topic.

Calculate the abscissa value of the negative R value point of the

$$P2X = - ZXL / \cos(\text{ArgNegRes})$$

$$\text{Then } P1Y = ZXL$$

$$\text{Then } P1X = ZRL + RF$$

$$\text{Then } P2Y = ZXL$$

Given that

$$P3X = P3Y = 0$$

Calculate the abscissa for the line P3 to P4; let ArgDir = 360 -15 = 345

$$\text{Length of line P3P4} = (RF * \cos(\text{ArgDir}))$$

$$P4X = (RF * \cos(\text{ArgDir})) * \cos(\text{ArgDir})$$

Calculate the ordinate for the line P3 to P4

$$P4Y = (RF * \cos(\text{ArgDir})) * \sin(\text{ArgDir})$$

## Theo Quadrilateral Impedance Example

In the **Calculation Control** place the variable identities required for the variables labeled for the vertexes in the **Theoretical Control Editor**.

**Note:** The Calculations shown below can be typed into a Notepad file line by line; copied and pasted into the Calculation Control. If an error occurs due to an incorrect formula, the edit may be more easily accomplished in the Notepad rather than the Control.

$$ZL = 8,6 \text{ ohms}$$

$$Zang = 75^\circ$$

$$RF = 4.3 \text{ (this is an arbitrary number for the example)}$$

$$\text{ArgNegRes} = 25^\circ$$

$$\text{ArgDir} = -15^\circ$$

$$P1X = ZRL + RF$$

$$P1Y = ZL * \sin(Zang)$$

$$P2X = -ZXL / \cos(\text{ArgNegRes})$$

$$P2Y = ZL * \sin(Zang)$$

$$P3X = 0$$

$$P3Y = 0$$

$$\tan(\text{AngDir}) = P4Y / P4X \text{ or } P4X * \tan(\text{AngDir}) = P4Y$$

$$\text{and } \tan(90 - Zang) = (RF - P4X) / P4Y$$

$$\text{then } \tan(90 - Zang) = (RF - P4X) / P4X * \tan(\text{AngDir}) \text{ or}$$

$$P4X = RF / [1 + (\tan(90 - Zang) * \tan(\text{AngDir}))]$$

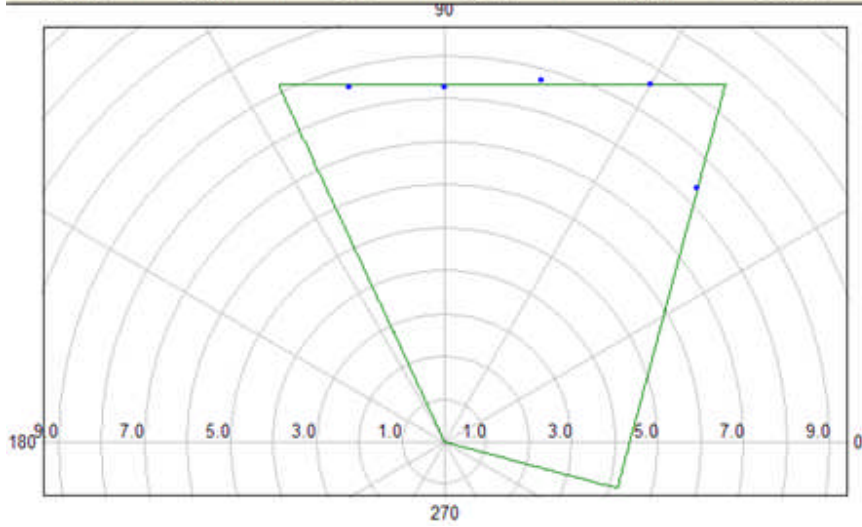
$$P4Y = -P4X * \tan(\text{AngDir})$$

MTA = Zang is a necessary equality to utilize the Distance Characteristic test taken from the New Test Wizard.

**Note:** ZL, Zang, RF, ArgNegRes, ArgDir values may also be placed in the Settings tab instead of the Calculation Control. This example leaves these variable identities in the Calculation Control.

*Execute the Test* and obtain the test results of the relay responses.

Va	Ia	Theo_Z	Test_Z	Error%	Pass/Fail
Volts	Amps	Ohms	Ohms	+/- 5%	
30.00	3.61	8.31	8.32	0.10	Pass
30.00	3.13	9.59	9.60	0.10	Pass
30.00	3.45	8.60	8.69	1.01	Pass
30.00	3.64	8.31	8.24	-0.79	Pass
30.00	3.52	8.60	8.53	-0.79	Pass



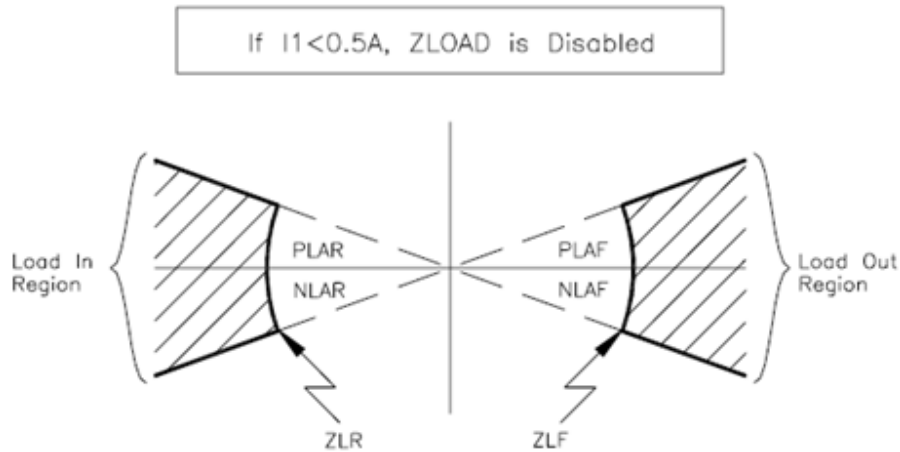
An example of this Quadrilateral is to replace the Theo Control and add the Calculation Control items in Test Editor Charac Z Example. No other changes are made and the results are shown at the end of this topic.

### Theo Load Encroachment Example

## Theo Load Encroachment Example

There exists impedance characteristics which do not fit the options of settings in the Theoretical Control Properties drop list. The Theoretical Object in the list of options allows construction of those characteristics alien to the other listed options.

The SEL 321 impedance mho characteristic with *Load Encroachment* applied is selected as an example to construct for this option.



The following topics step through the construction to and include an example test result:

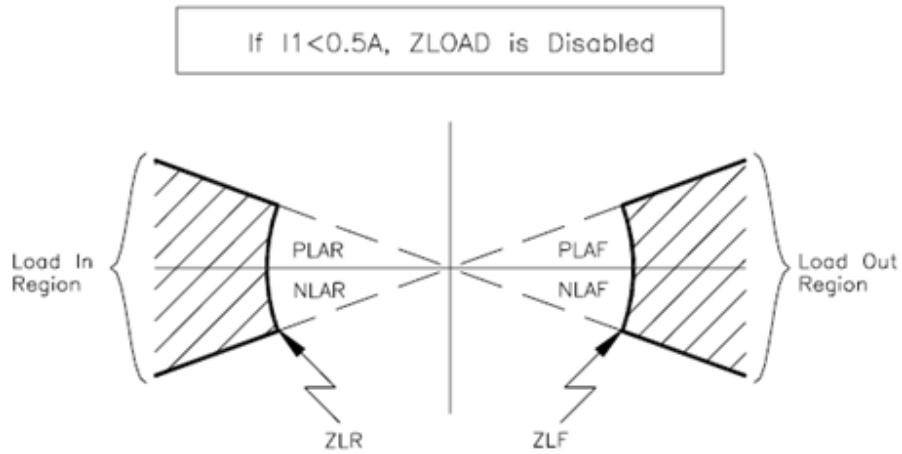
SEL Load Encroachment Construction

SEL Load Encroachment Calculations

SEL Load Encroachment Example

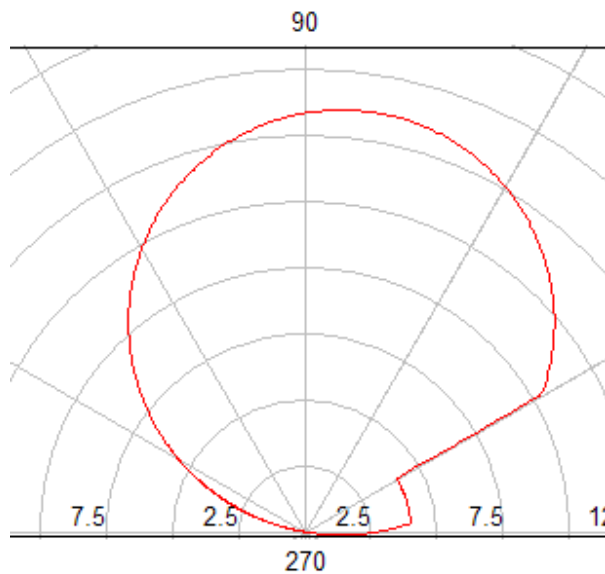
## SEL Load Encroachment Construction

Example: Construct the SEL 321 impedance mho characteristic with *Load Encroachment* applied.



*Zone 4 with Load Encroachment* applied is chosen as the forward zone to apply this discussion.

## Zone 4 Reach Test



Given: Values are used for explanatory purposes and the resultant characteristic derived formulae will apply to any appropriate settings for the relay.

Z4P = 16 Ohms, the protected line impedance setting

Z1ANG = 80 degrees, the protected line impedance angle

ZLF = 3 ohms, the forward load encroachment impedance setting

PLAR = 30 degrees, the positive forward load encroachment angle limit


The SEL Load Encroachment consists of a mho circle with


a circle centered at the origin with a radius = the the ZLF setting and with

a line through the origin in the first quadrant for the positive load angle PLAR



Roughly create the characteristic using the Quick Circle and Quick Line tools

Select the View Polar tool  to change the display from Cartesian to polar coordinates

Select the Quick Circle tool  for the mho circle

place the cursor in the approximate position of the circle center and the max torque angle  
click the cursor in this location and move the mouse to the origin of the plot and click to anchor the mho

Select the Quick Circle tool again to create the Load impedance zone

click the cursor on the origin and move the mouse along the X-axis to the approximate ZLF value

click the cursor in this location to anchor this circle

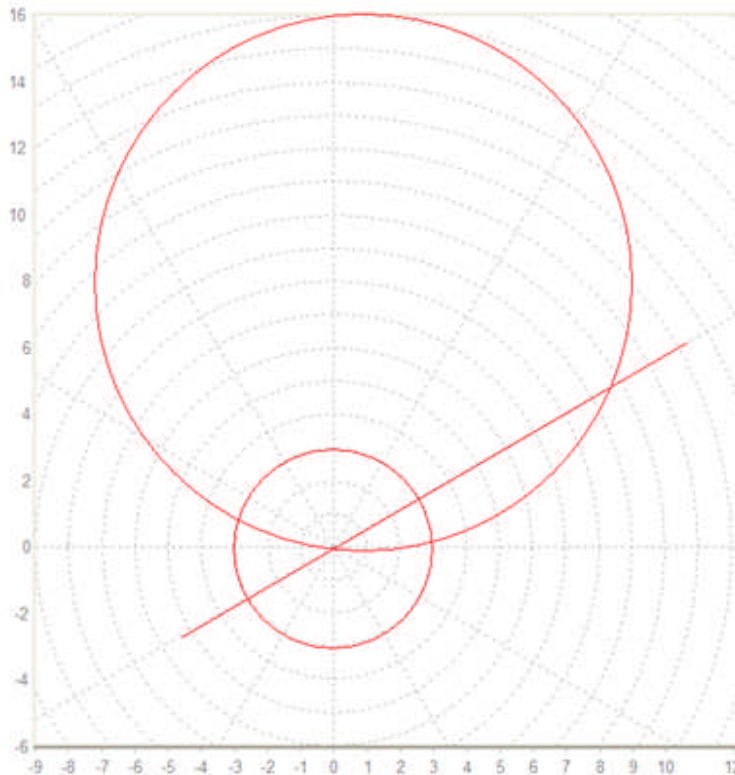
Select the Quick Line tool  for the PLAF angle

click a location in the first quadrant at the approximate positive load angle

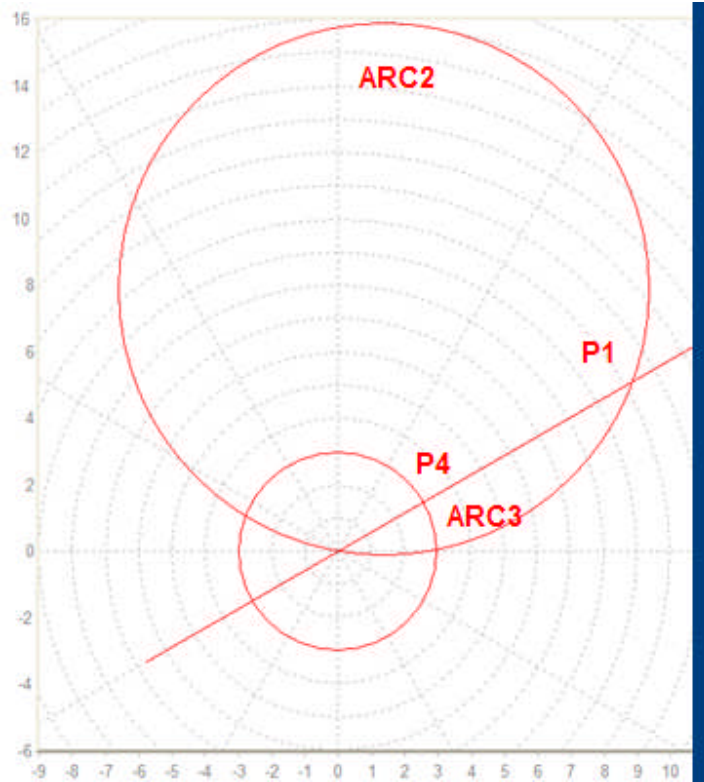
move the cursor through the origin to a location in third quadrant and click to anchor the line for PLAF


Note: Construction from quadrant 1 to quadrant 3 will automatically label the line end points as P1 in quadrant 1 and P2 in quadrant 3. If the line is constructed from quadrant 3 to quadrant 1, the end points will have reverse labels.

The *resultant construction* will then need editing to create the desired characteristic appearance.



Determine a method of labeling the *locations that are changes* in the object circles and line and will be used for reference to the formulae of the complete object. This example identifies the locations as P1, ARC2, ARC3 and P4 labeled in the CCW direction.



Select the Edit Object tool  to click on the line to view the *polar coordinates of the line ends* as the line was created

Property	Value
Magnitude 1	11.927
Angle 1	29.935
Magnitude 2	5.622
Angle 2	210.353

**Note:** Editing the properties after using the Clip Selection tool may create a disjunct object and not able to be saved.

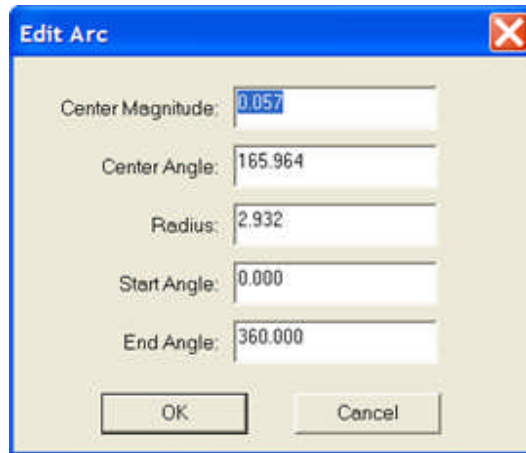
The *values shown when first opened* are the constructed coordinates and the Edit Line properties dialog shows the labels of the line ends in the edit fields

Note: It will be useful to *change the angles* for the line to agree with the 30 and 210 degree locations of the line end points. The magnitudes will reflect values useful for comparison to the calculations of the developed formulae.


The *ARC2 values are displayed* in the Arc Edit dialog when the Edit Object tool is clicked on the zone mho circle

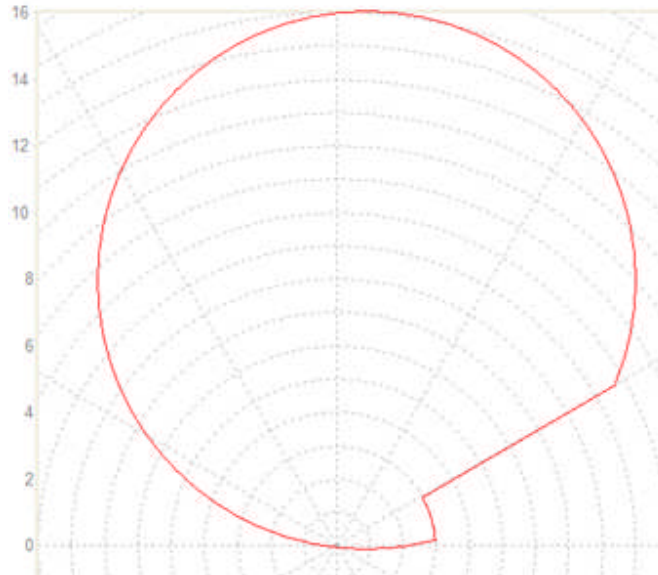
Note: Edit the values for Center Magnitude: 8, Center Angle: = 80, and Radius: = 8

The *ARC3 values are displayed* in the Arc Edit dialog when the Edit Object tool is clicked on the load mho circle



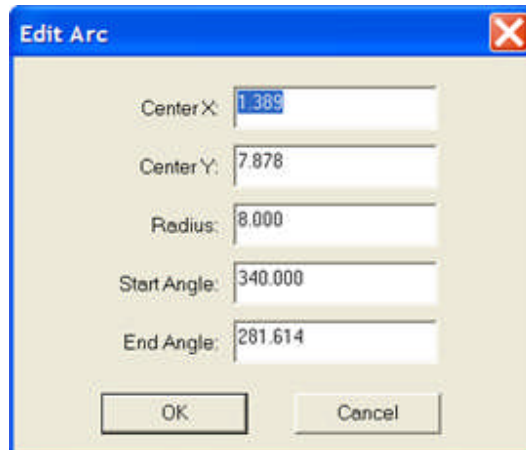
Note: Edit the values for Center Magnitude: 0, Center Angle: = 0, and Radius: = 3

After the properties have been edited, select the Clip Section tool  to "snip" the unwanted *line extensions and arc sections* to obtain the desired view




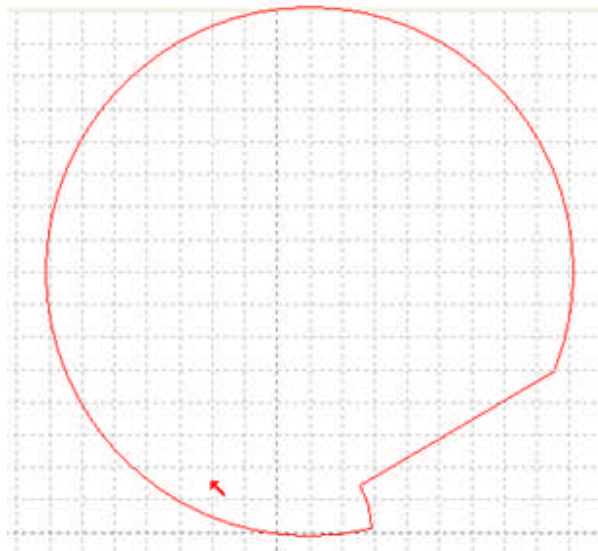
Review the edit properties for the line and two arcs and write the values down for future reference.

Note: The values in the edit dialogs have been viewed and used in the View Polar display. Selection of the View Polar tool will return to the Cartesian display and all arcs and lines will properties will be shown with the Magnitudes and angles converted to *Cartesian X and Y values*.



Select the Save tool or the File/Save menu item.

Select the Center of Operation tool  and click on the interior of the characteristic to define the *Center of Operation*. The Center of Operation must be in a location that an imaginary line may be drawn to every location on the described characteristic boundaries



Select File/Save option

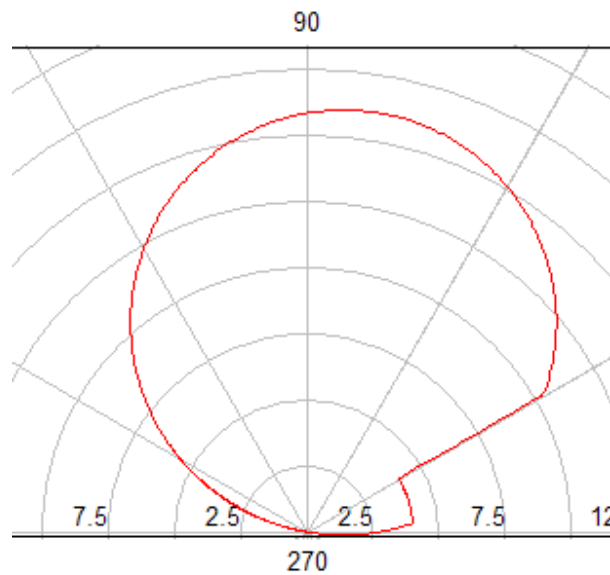
Select File/Exit option

Select the View Polar tool to return the display to Cartesian coordinates.

## SEL Load Encroachment Calculations

Example: Determine the formulae for the *Zone 4 with Load Encroachment* coordinates of the characteristic arcs and line. Use information in the SEL Load Encroachment Construction topic.

### Zone 4 Reach Test



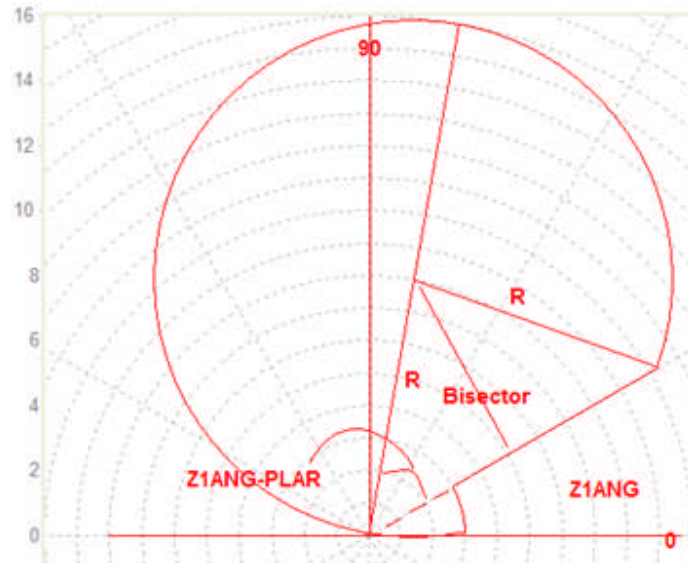
The portions of interest in this characteristic can be viewed as several sets of triangles to determine the formulae to calculate the necessary values for the given Settings.

The following Settings data will be used to verify the developed formulae compared to the construction edited values.

Given:  $Z4P = 8$   $Z1ANG = 80$   $ZLF = 3$   $PLAR = 30$

After the formulae have been determined and checked for accuracy, the Line and ARC2 and ARC3 properties will be given variable labels so that any Settings will in the Settings tab will determine the proper characteristic.

For P1, a *figure is drawn* and information is noted to determine the polar coordinates



A triangle consists of the two radii from the mho center to P1 and to the Origin. A Chord is created from the Origin to P1 at the PLAR angle. This creates a triangle with two know sides, R, and one know (Z1ANG - PLAR) angle.

The angle between the two radii is not known, but bisecting the angle with a line to the center of the chord will create two "right" triangles.

Then:  $\frac{1}{2} * \text{chord} = \frac{1}{2} * Z4P * \cos(Z1ANG - PLAF)$  and the Chord length =  $Z4P * \cos(Z1ANG - PLAF)$

$$P1 = 10.28 \text{ at } 30 \text{ degrees}$$

Additionally, a triangle, formed by the chord and PLAF and the X-axis, is used to determine the Cartesian coordinates

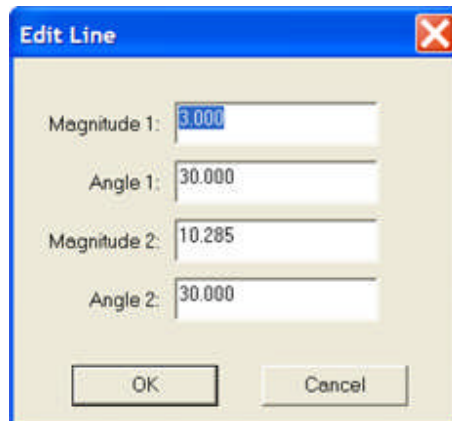
$$P1X = Z4P * \cos(Z1ANG - PLAF) * \cos(PLAF) \\ = 16 * \cos(80 - 30) * \cos(30) = 8.91$$

$$P1Y = Z4P * \cos(Z1ANG - PLAF) * \sin(PLAF) = 5.14$$

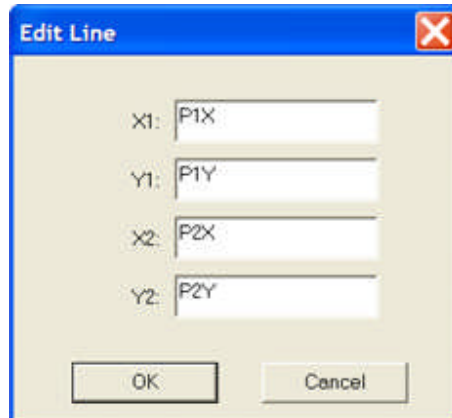
Warning: Always use lower case letters for the trig functions; the interpreter does not allow upper case for these functions and will yield incorrect calculations. Review the Expressions topic.

For P4, the value is simply ZLF at PLAR or  $P4 = 3 \text{ at } 30 \text{ degrees}$ .

The calculated values for P1 and P4 agree with the *constructed properties for the line*

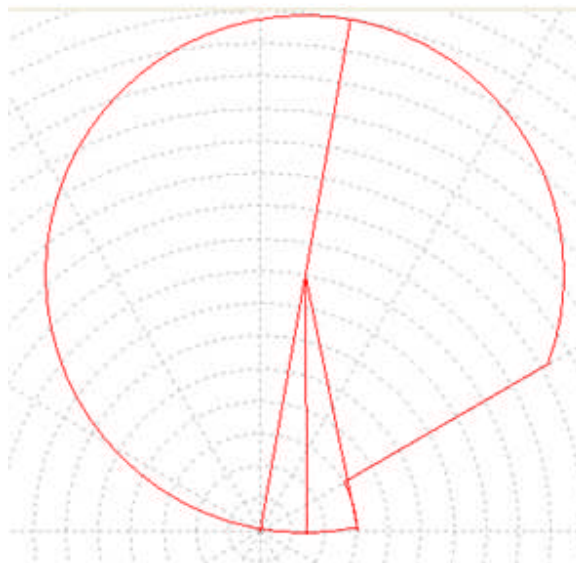


Edit the *Edit Line dialog* with variable names



X1: = P1X   Y1: = P1Y   X2: = P2X   Y2: = P2Y

The ARC3 properties are determined similarly using a *triangle formed by a radius to the origin and a radius to the intersection of the mho and load circles*. The values of the two radii are known, but no angles are readily known.



Determine the angle between the radius to the origin and the Chord to the intersection of the two circles

The bisector of the angle between the two radii forms two right triangles at  $1/2 * ZLF$

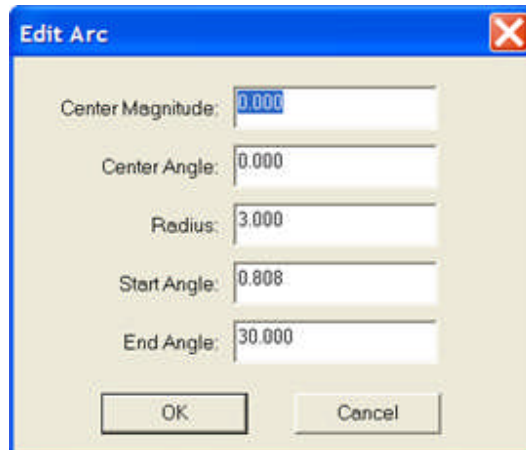
Then: Chord angle =  $\text{acos}(1/2 * ZLF / 1/2 * Z4P)$  and

the angle of the intersection and the X-axis =  $Z1\text{ANG} - \text{acos}(1/2 * ZLF / 1/2 * Z4P)$ . This angle is calculated first as it is necessary to the values of ARC 2.

The angle =  $80 - \text{acos}(3 / 16) = 80 - 79.19 = 0.807$  degrees and the intersection magnitude =  $ZLF = 3$

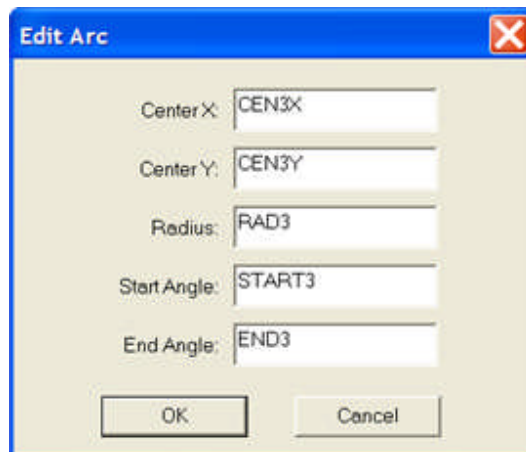
The calculated values for ARC3 agree with the *constructed properties for the ARC3*



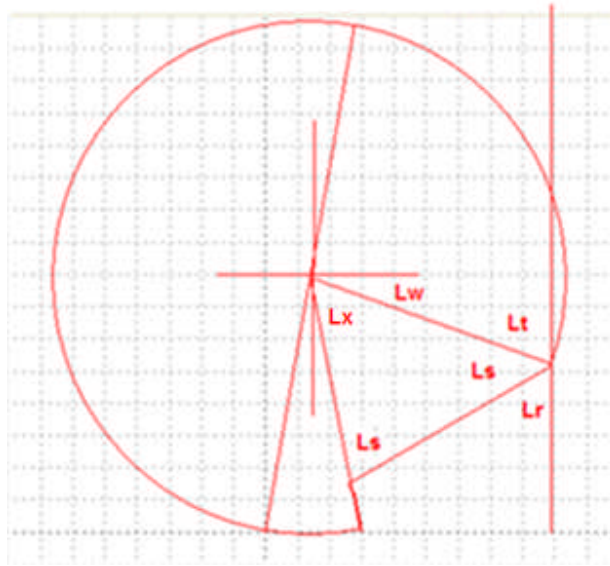


Note: The ARC3 angles begin with the angle that describes the complete arc in a CW direction. Start = 0.807 and End = PLAF.

Edit the *Edit ARC dialog* with variable names ARC3



This ARC2 properties are determined similarly using the triangle formed by the radii to P1 and ARC3 and the included Chord. Magnitudes are determined by the original given information. *Other triangular information is necessary* to determine the angular measurements for the ARC2.



Angle Lr = 90 - PLAF from two angles of a right triangle

Angle Ls = (Z1ANG - PLAF) from given data

Angle Lt = 180 - (90 - PLAF) - (Z1ANG - PLAF) from three angle for 180 degrees

Angle Lw = Angle Lt - 90 = (180 - (90 - PLAF) - (Z1ANG - PLAF)) - 90 from two angles of a right triangle

Start Angle = 90 - (90 - PLAF) - (Z1ANG - PLAF) = -20 degrees or 340 degrees

Angle Lx = 180 - (Z1ANG - PLAF) - (Z1ANG - PLAF)

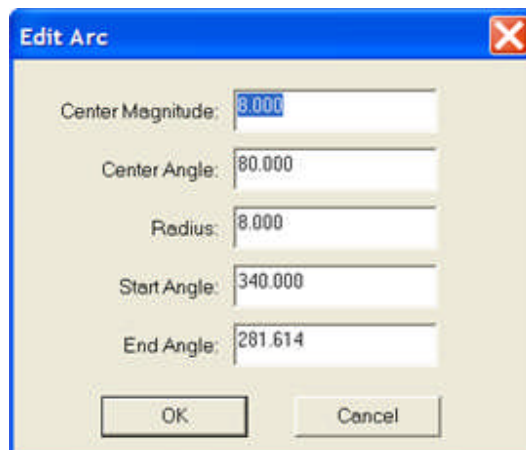
Angle calculated for ARC3 =  $Z1ANG - \arcsin(1/2 * ZLF / 1/2 * Z4P)$  and ARC2 Angle is determined by the angle complement

Angle for ARC =  $Z1ANG - \arcsin(1/2 * ZLF / 1/2 * Z4P)$ , but this is 1/2 the total angle

Angle for ARC3 =  $Z1ANG - 2 * \arcsin(1/2 * ZLF / 1/2 * Z4P)$

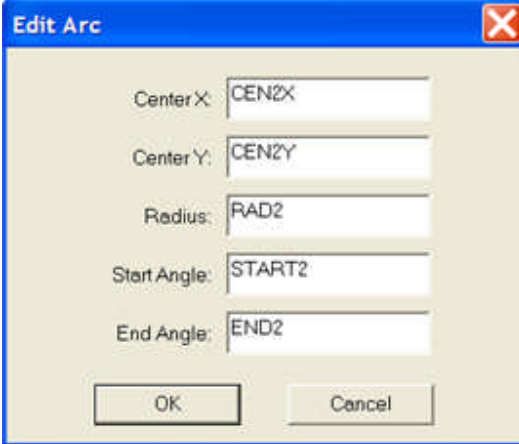
End Angle =  $180 + Z1ANG - 2 * \arcsin(1/2 * ZLF / 1/2 * Z4P) = (180 + Z1ANG) + 2 * \arcsin(ZLF / Z4P) = 281.614$

The calculated values for ARC2 agree with the *constructed properties for ARC2*



Note: The ARC2 angles begin with the angle that describes the complete arc in a CW direction. Start = 340 degrees and End = 281.614 degrees. The angles were calculated in this fashion.

Edit the *Edit ARC dialog* with variable names ARC2



The image shows a dialog box titled "Edit Arc" with a blue header bar and a close button in the top right corner. The dialog box has a light beige background and contains five input fields, each with a label to its left and a text box to its right. The labels and their corresponding values are: "Center X:" with "CEN2X", "Center Y:" with "CEN2Y", "Radius:" with "RAD2", "Start Angle:" with "START2", and "End Angle:" with "END2". At the bottom of the dialog box, there are two buttons: "OK" on the left and "Cancel" on the right.

Center X:	CEN2X
Center Y:	CEN2Y
Radius:	RAD2
Start Angle:	START2
End Angle:	END2

OK Cancel

## SEL Load Encroachment Example

The Variable labels created in the Line and ARC2 and ARC3 edit dialogs will be equated to the developed formulae.

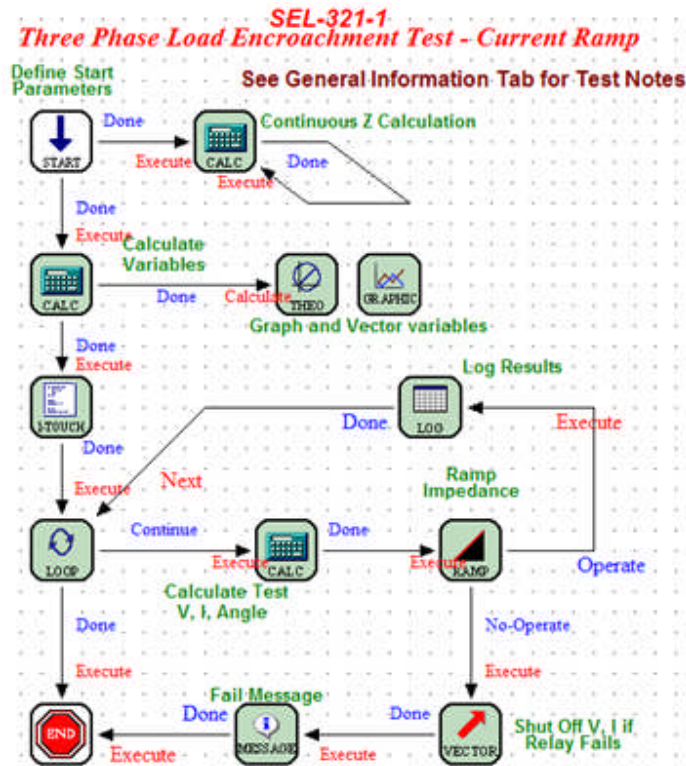
The Variables and formulae may be typed into Windows Notepad and then copied to the Calculate Control that governs the Theoretical Object values.

```
if ZLF = 0.5 then ZLF = 0
P1X = Z4P * cos(Z1ANG - PLAF) * cos(PLAF)
P1Y = Z4P * cos(Z1ANG - PLAF) * sin(PLAF)
CEN2X = (1/2 * Z4P * cos(Z1ANG))
CEN2Y = (1/2 * Z4P * sin(Z1ANG))
RAD2 = 1/2 * Z4P :
START2 = (90 - (Z1ANG - PLAF) - (90 - PLAF))
END2 = ( 180 + Z1ANG) + (2 * asin(ZLF / Z4P))
CEN3X = 0
CEN3Y = 0
RAD3 = ZLF
START3 = (90 - Z1ANG - acos(ZLF / Z4P))
END3 = PLAF
P4X = ZLF * cos(PLAF)
P4Y = ZLF * sin(PLAF)
```

**Note:** All formulae use the same variable label that exists in the Relay Settings tab.

Execute the Test and obtain the test results of the relay responses.

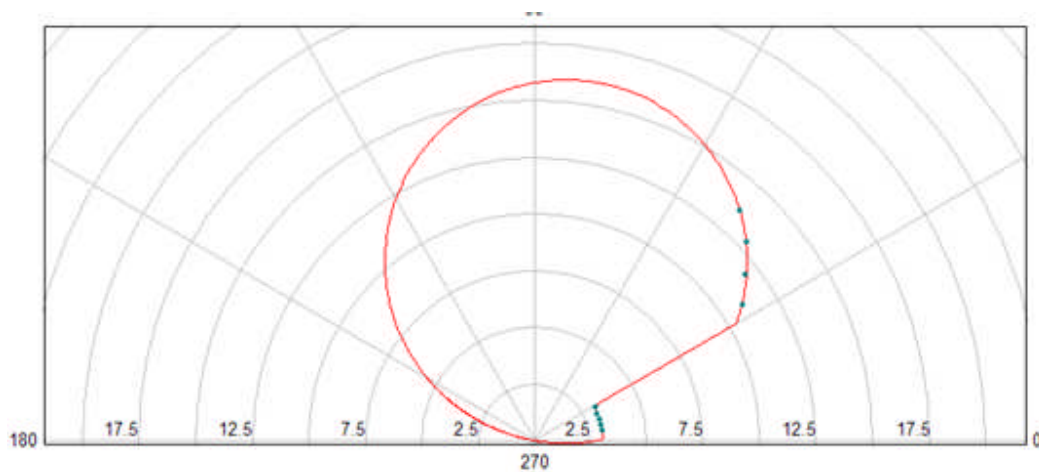
An example of this *SEL Load Encroachment is to replace the Theo Control* and add the Calculation Control items in Test Editor Charac Z Example.



No other changes are made and  
the results table

Impedance Ohms	CalcZ Ohms	Error %	Pass/Fail	TestVolts Volts	TestAmps Amps	TestAngle
13.583	13.57	0.10	Pass	67.00	4.93	48.00
12.790	12.78	0.10	Pass	67.00	5.24	43.00
11.847	11.89	-0.35	Pass	67.00	5.66	38.00
10.920	10.91	0.10	Pass	67.00	6.14	33.00
3.058	3.00	1.94	Pass	67.00	21.91	28.00
3.003	3.00	0.10	Pass	67.00	22.31	23.00
3.044	3.00	1.47	Pass	67.00	22.01	18.00
3.044	3.00	1.47	Pass	67.00	22.01	13.00
3.017	3.00	0.55	Pass	67.00	22.21	8.00

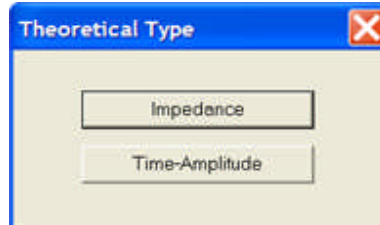
the results graph



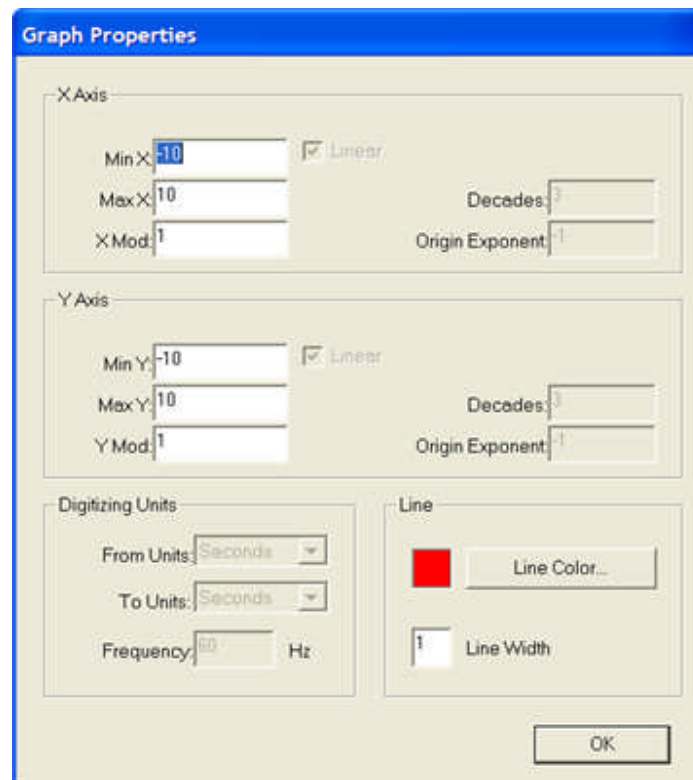
## Theo Theoretical Object - Time-Amplitude

# Theo Theoretical Object - Time-Amplitude

The Theo Control option Theoretical Object Edit button will display the *Theoretical Type* dialog.



The Impedance and Time-Amplitude buttons are to select and open the *Graph Properties* dialog, properties for Impedance are demonstrated. The Impedance option is described in the Theo Theoretical Object - Impedance group of topics.



The Theoretical Object edit option of the Theoretical Control Properties is to digitize a single curve or a family of curves to be used as the expected response of a relay under test at multiple values of current or voltage input. The relays whose characteristic curves are digitized are usually those without a mathematic formula that can accurately predict the relay response. The relays of this type are electromechanical and older solid-state models.

If the relay is newer solid-state or microprocessor models, formulae are provided in the manufacturer relay literature. The use of the Calculation Control and/or the Theoretical Control Theo Calculations edit option is typically used.

The use of this mode is detailed in the following topics

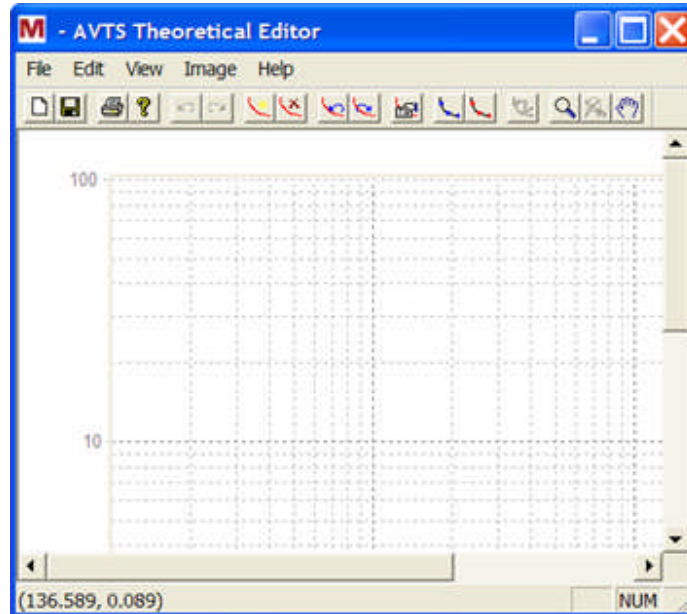
- Theo Object Time-Amplitude details the Toolbar and Menu items.

- Theo Object Time-Amplitude Example details an example to digitize a curve(s).

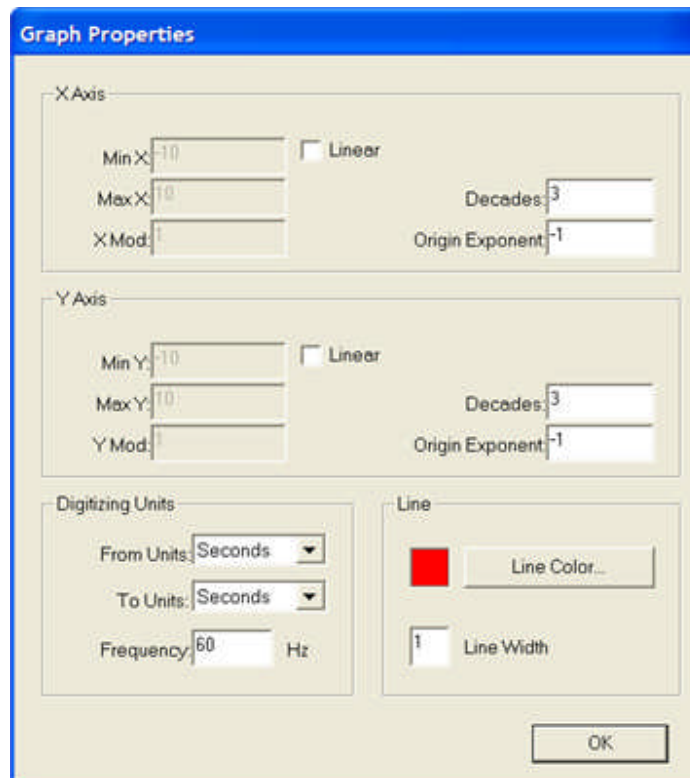
Theo Object Time-Amplitude Auto Trace details the use of the Auto Trace mode to digitize a curve.  
An example use of this Control is demonstrated in Test Editor OC Time Example.

## Theo Object Time-Amplitude

When the Theoretical Object Time-Amplitude option is selected, the *AVTS Theoretical Editor* dialog opens for construction of a Time-Amplitude characteristic. The digitizing of all curves will be accomplished in this dialog.



The *Graph Properties* dialog displays four panes for setting the Linear boundaries of the graph.



The toolbar has the following tools defined from left to right:





**New (File item)** opens the Theoretical Type dialog to select either a new Impedance or Time-Amplitude characteristic. If a previous Background Image had been inserted, the Image will be retained with the mapping removed. Re-mapping the Image will be necessary, or a different Image may be inserted and mapped.

**Save (File item)** is to place the presently open AVTS Theoretical Editor file in the active database.

**Print (File item)** opens the Windows Printer Selection dialog to print the characteristic.

**About AVTS Theoretical Editor (Help item)** dialog provides the version in the installed AVTS software.

**Undo (Edit item)** is to return to the action previous to the last action performed.

**Redo (Edit item)** is to return to the action previous to the last Undo.

**New Curve (Edit item)** evokes the Curve Properties dialog to label the new Time Dial Setting to be digitized. Each new time dial curve to be digitized, will return the dialog with the Description of Curves label entered for the family of curves.

**Delete Curve (Edit item)** will delete all the digitized points of an selected curve of the family of curves.

**Previous Curve (View item)** moves focus to the last digitized curve of the family of curves for view or edit of the points digitized.

**Next Curve (View item)** moves focus to the next digitized curve of the family of curves for view or edit of the points digitized.

**Curve Properties (Edit item)** returns the Curve Properties dialog with the Description of Curves label and Active Curve Time Dial Setting label for the selected curve.

**Insert Curve Points (Edit item Insert Time Amplitude Curve Point)** enables the cursor to pick points along the curve creating locations for the graph to create straight lines from point to point.

**Delete Curve Points (Edit item Insert Time Amplitude Curve Point)** enables the cursor to delete any previously inserted point on a selected curve.

**Auto Trace Mode (Image item)** option changes to the Auto Trace display with a new toolbar. This option is available only with entering a new Time Dial number.

**Zoom (View item)** will increase the view size with each mouse click.

**Zoom Normal (View item)** returns any Zoom to the original view size.

**Pan (View item)** changes the cursor to a hand to grab an area and move the graph view.

The following menu items do not have a toolbar equivalent:

**File/Print Setup** opens the standard Windows Select Printer dialog.

**File/Exit** opens an AVTS Theoretical Editor message to save the changes to the object, if edited from last Save. Yes option saves edits and exits, No exits with no Save; Cancel returns to the Theoretical Editor graph object.

**View/Toolbar**, default checked and visible, hides the Toolbar if unchecked.

**View/Status Bar**, default checked and visible, hides the Status Bar if unchecked. The Status Bar displays Cartesian (X,Y) and Polar (Radius from 0,0,Angle) regardless which graph type is in display.

**View/Background Image** (use in Time-Amplitude graph only)

**Image/Insert Background Image** (use in Time-Amplitude graph only)

**Image/Remove Background Image** (use in Time-Amplitude graph only)

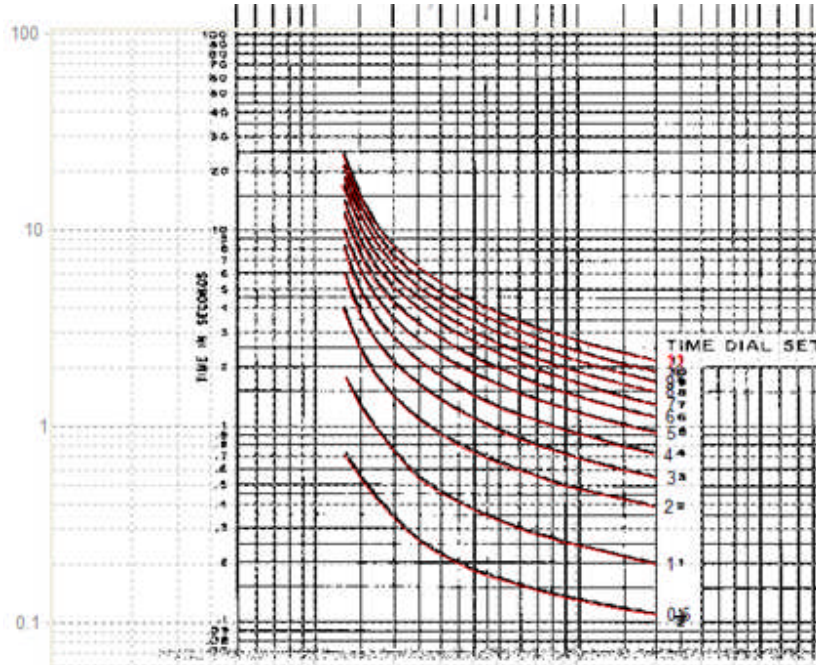
**Image/Map Top Left Image Point** (use in Time-Amplitude graph only)

**Image/Map Bottom Right Image Point** (use in Time-Amplitude graph only)

**Image/Cancel Image Mapping** (use in Time-Amplitude graph only)

## Theo Object Time-Amplitude Example

Example to construct a *family of digitized time overcurrent curves*.



The family of curves to be digitized must first exist as a bitmap (\*.bmp) image and usually saved to the AVTS/DigToolImages folder. The single curve or family of curves may be scanned and saved as a bitmap.

During the scanning process, several scans may be necessary to assure that there is as near as possible perfect vertical and horizontal alignment of the graph axes. AVTS does not possess the tools to make these alterations to a bitmap; and, therefore, must have alignment accomplished when scanning.

The steps to digitize a family are:

Select the Time Amplitude option to open the Graph Properties dialog. Choose whether the graph will be linear X or Y axis or both. If only one is linear, the log setting options for the non-linear axis will be in focus to set the number and range of decades to use.

The AVTS Theoretical Editor will open the graph, in light gray color, with the X and Y axes with the chosen min and max ranges chosen.

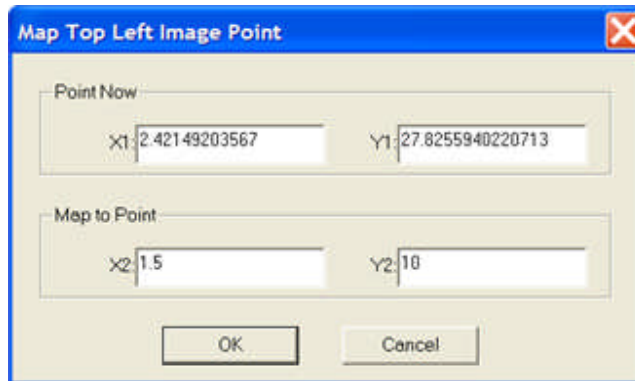
Select Image/Insert Background Image menu item for the Windows Open dialog to select the scanned bitmap. The DigToolImages folder is automatically opened, but if the image had been saved to another folder, the Open dialog browse button is used to locate the image elsewhere. This example uses the co11.bmp file.

The Graph Properties were originally set to log-log scales different from the scanned image. Select Edit/Graph Properties menu item to return to the dialog and edit the scales to fit the Inserted Image.

The co11 X Axis values are Decades: = 2 and Origin Exponent: = 0. The Y Axis values are Decades: = 4 and Origin Exponent: = - 2.

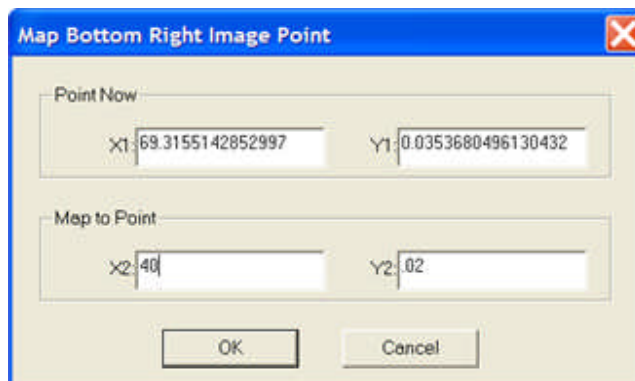
**Note:** A logarithmic scale is in decades of 10 multiples, where 2 decades is 100 when the Origin Exponent is set = 0, which is 1. If the Decades is set to 4 and the Origin Exponent is set = -1, the origin will be = 0.1. Four decades will then yield axis logarithmic values 0.1 - 1, 1 - 10, 10 - 100 and 100 -1000.

Select Image/Map Top Left Image Point to position a Cartesian point on the Image to the same value point on the Graph. The cursor will change to a crosshair to Select the Image point X = 1.5 and Y = 10. The map *Top Left Image Point* dialog will display the Point Now which is the selected Image point referenced to the underlying graph point. Change the Map to Point to X2 = 1.5 and Y2 = 10. Click OK and observe the Image to "map" to the edited graph point.



Note: Use the Zoom tool to enlarge the area to be positioned to more accurately "pick" the Image point. Use the scroll bars if necessary to navigate the zoomed image to the area to map.

Select Image/Map Bottom Right Image Point to position a Cartesian point on the Image to the same value point on the Graph. The cursor will change to a crosshair to Select the Image point X = 40 and Y = 0.02. The map *Bottom Right Image Point* dialog will display the Point Now which is the selected Image point referenced to the underlying graph point. Change the Map to Point to X2 = 40 and Y2 = 0.02. Click OK and observe the Image to "map" to the edited graph point.

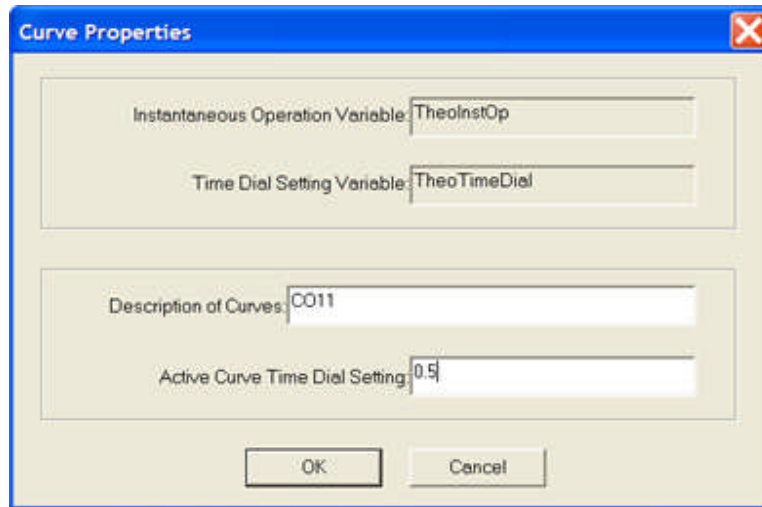


Note: Accuracy in mapping is extremely important. The value of the curve Time vs. the applied Multiple must be as precisely overlaid on the graph to obtain an accurate comparison of the relay time.

Note: It may be necessary to zoom and re-map the top left and bottom right additional times to achieve accuracy of overlay. The Image curves are the only portions that need to be accurately overlaid. The positions of the remainder of the Image are not include in any timing points vs. multiples.

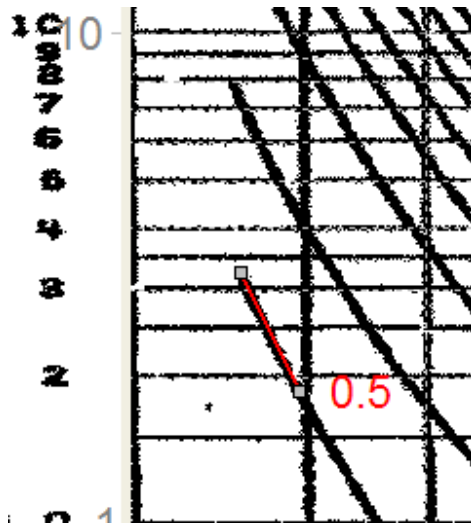
Next to provide labels for the individual curve to be digitized

Select the New Curve tool to display the *Curve Properties* dialog. Insert the curve family name or other label in the Description of Curves: edit field. Insert the curve Time Dial to be digitized in the Active Curve Time Dial Setting: edit field; default = 1. Insert 0.5 and click OK. The cursor is a crosshair and the Insert Curve Points tool is made active.

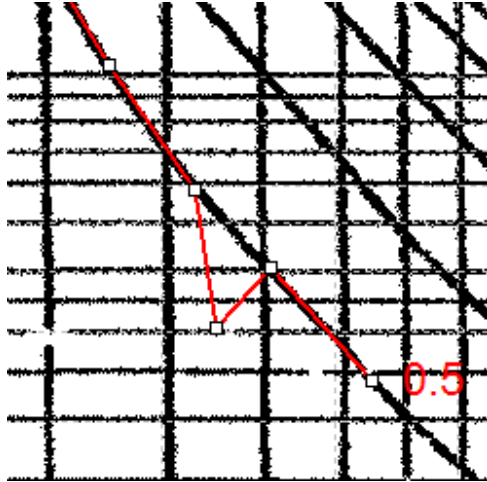


Note: The Instantaneous Operation Variable TheoInstOp is used to format the graph cutoff value on the Time Overcurrent digitized curve and display the vertical Instantaneous characteristic.

Beginning with the top left of the 0.5 Time Dial curve, single click on the end of the curve. Then click on another location on the same curve at a small distance reasonably representing a straight line. The two points will be connected by a *straight line and the Time Dial number* will appear at the position of the last selected point. Continue selecting points along the curve until the end of the curve.



A selected curve point may be corrected by using the Delete Curve Points tool. Place the mouse on the *"bad" point* and click to remove it. The digitized line will re-connect to the two points on either side of the bad point.



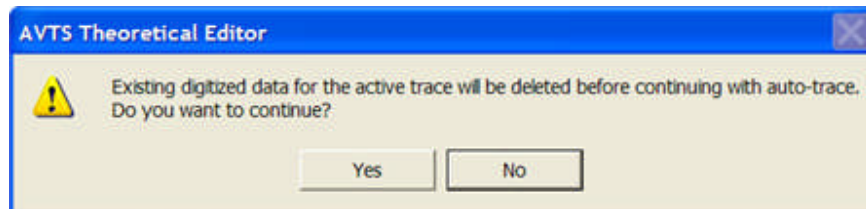
Select the New Curve tool to open the Curve Properties dialog to enter a new Active Curve Time Dial Setting. Enter the number of the next curve to digitize and click OK. Continue digitizing the family of curves for all remaining time dial curves.

**Note:** All time dials must be digitized for accuracy. AVTS will interpolate a setting between any two digitized time dials. The electromechanical relays may not have a precise formula for each time dial and an interpolated value between two extreme time dials, 2 and 7 for example, will likely be inaccurate.

After all curves have been digitized and saved before or during File/Exit, The digitized traces will be saved to the relay Theoretical. The family of curves may be used for other model relays having the same characteristic by copying the Theoretical Control to the other relay.

## Theo Object Time-Amplitude Auto Trace

The Auto Trace selection in the AVTS Theoretical Editor dialog *evokes a message* that entry to the mode will delete all digitized curves in the present view. Only one set of points for a curve may be digitized in this mode. To use the Auto Trace mode for another trace, it must have a new Time Dial number created with the New Trace tool in the original AVTS Theoretical Editor. The Auto Trace mode must be toggled between dialogs for each new curve, or the curves may be digitized entirely in the non-Auto Trace mode.



This mode is a special case for the normal Theo Object Time-Amplitude mode and the use of this mode is similar to the Theo Object Time-Amplitude Example topic.

The Auto Trace mode toolbar tools are defined from left to right:



**Zoom In** (Auto Trace item) is to enlarge the graph in equal increments with each mouse click.

**Zoom Out** (Auto Trace item) is to reduce the graph in equal increments with each mouse click.

**Zoom Normal** (Auto Trace item) returns an enlarged graph to the normal size.

**Auto Trace** (Auto Trace/Run Trace item) enables the tool to trace an characteristic through its curvature. Select the tool and click a start point on the curve. A second click with the cursor on the curve, selects the next point along the curve. A third click enables the Auto Trace to create the digitized trace between the first two clicked points.

**Delete Digitized Range** (Auto Trace item) is to remove all digitized points between any two other points or the end points. The editor message "This action cannot be undone. Are you sure you want to delete all digitized data?" appears for a Yes or NO option.

**Auto Trace Mode** (Auto Trace item) toggles back to the normal AVTS Theoretical Editor graph.

The following menu items do not have a toolbar equivalent:

**File/New** returns to the non-Auto Trace graph. It is to introduce a new Graph and evokes the "Do you want to save your changes?" message dialog. If Yes, the digitized images will be saved to the database. If no, all digitized edits will be cleared and no longer retrievable. The action opens the Theoretical Type dialog to select either a new Impedance or Time-Amplitude characteristic. If a previous Background Image had been inserted, the Image will be retained with the mapping removed. Re-mapping the Image will be necessary, or a different Image may be inserted and mapped.

**File/Exit** opens an AVTS Theoretical Editor message to save the changes to the object, if edited from last Save. Yes option saves edits and exits, No exits with no Save; Cancel returns to the Theoretical Editor graph object.

**View/Toolbar** displays the Toolbar when clicked (default). When clicked a second time, the toolbar is hidden. This is a toggle action.

**View/Status Bar** displays the Status Bar when clicked

**Auto Trace/Delete All Digitized** is to remove all digitized points on the Image.

**Auto Trace/Filter Noise** reduces the weight of image lines and curves. The action is to reduce the weights of the image axes for the Auto Trace tool to more easily follow the curve between selected points.

**Auto Trace/Undo Last Filter** removes the last filter action.

**Auto Trace/Undo All Filters** removes all filter actions to return to the normal inserted image.

**Auto Trace/Zoom** is used to Zoom In of Normal or Zoom Out of a Zoom In in increments of 1X, 2X, 3X, 4X and 5X.

**Auto Trace/Filtered Image** is checked automatically after adding a filter. When unchecked, the image will return to the unfiltered state. The check and un-check will toggle the action.

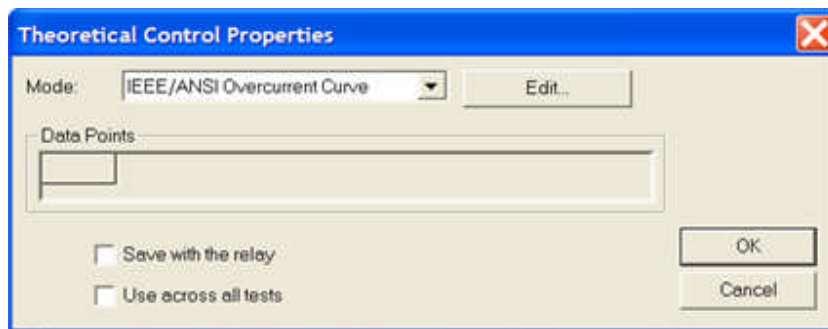
## Theo IEEE/ANSI Overcurrent Curve

The Theoretical Control properties for the **Mode IEEE/ANSI Overcurrent Curve** is to select the IEEE/ANSI Standards Time Overcurrent Curve for use in a relay timing test.

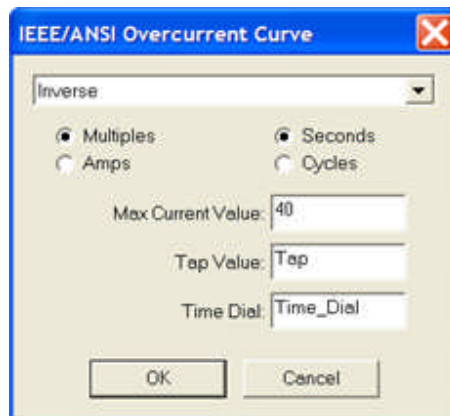
These curves are typically used in electronic and microprocessor relays. Selection of the curve gender along with the tap and time dial settings are immediately made.

Substitution of this characteristic selection in place of the Theoretical Control in Test Editor OC Time Example demonstrates its use. No other edits are necessary in the example.

Mode selected from drop list: *IEEE/ANSI Overcurrent Curve*, then select Edit

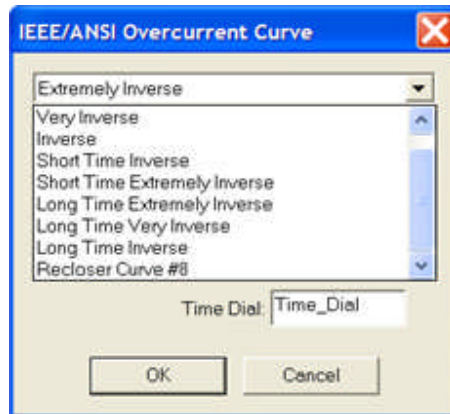


*IEEE/ANSI Overcurrent Curve* dialog is to select the relay settings.



The *Standard Curves drop list*. The curve options are Extremely Inverse, Very Inverse, Inverse, Short Time Inverse, Short Time Extremely Inverse, Long Time Extremely Inverse, Long Time Very Inverse, Long Time Inverse and Recloser Curve #8.





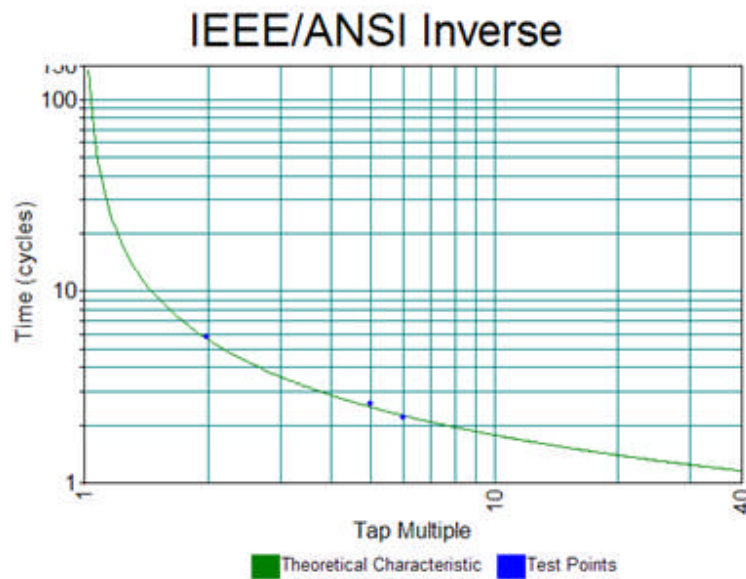
Other selections includes the bullets for either Multiples of Tap Value or Amps and for either Seconds or Cycles for elapsed time to respond.

**Max Current Value:** is to enter a numeric value or a variable name to limit attempts to apply test values of greater value.

**Tap Value:** is to enter a numeric value or a variable name for the tap setting of the relay. Tap has been entered to reflect the value provided in the Settings tab or a previous Calculation.

**Time Dial:** value is to enter a numeric value or a variable name for the time dial setting of the relay. Time\_Dial has been entered to reflect the value provided in the Settings tab or a previous Calculation.

*The resultant time overcurrent characteristic.*



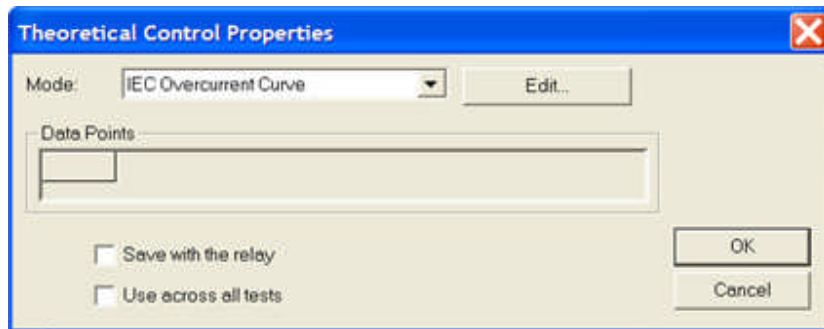
## Theo IEC Overcurrent Curve

The Theoretical Control properties for the **Mode IEC Overcurrent Curve** is to select the IEC Standards Time Overcurrent Curve for use in a relay timing test.

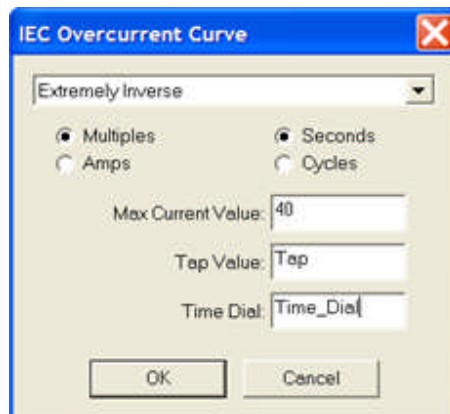
These curves are used in electronic and microprocessor relays. Selection of the curve gender along with the tap and time dial settings are immediately made.

Substitution of this characteristic selection in place of the Theoretical Control in Test Editor OC Time Example demonstrates its use. No other edits are necessary in the example.

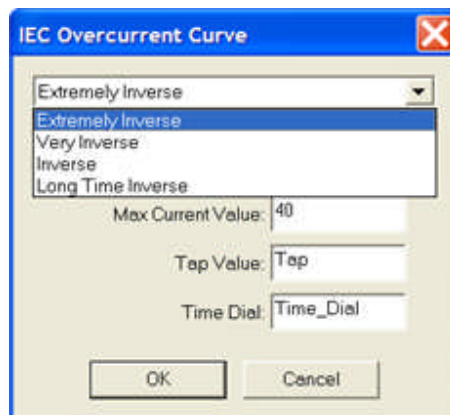
Mode selected from drop list: *IEC Overcurrent Curve*, then select Edit



*IEC Overcurrent Curve* dialog is to select the relay settings.



The *Standard Curves drop list*. The curve options are Extremely Inverse, Very Inverse, Inverse and Long Time Inverse.



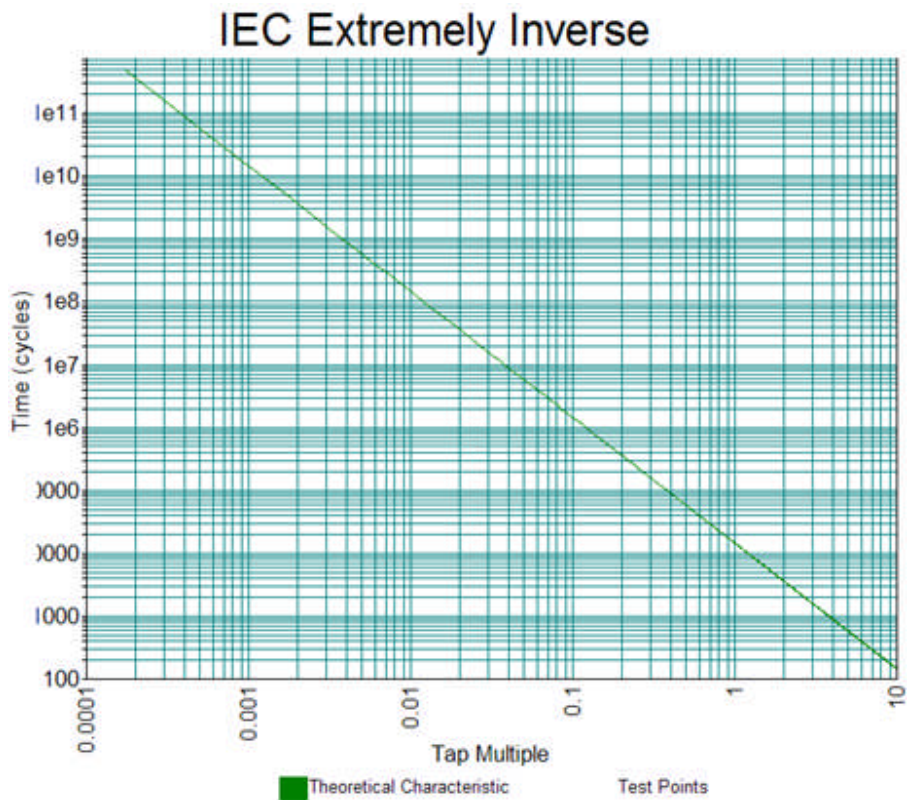
Other selections includes the bullets for either Multiples of Tap Value or Amps and for either Seconds or Cycles for elapsed time to respond.

**Max Current Value:** is to enter a numeric value or a variable name to limit attempts to apply test values of greater value.

**Tap Value:** is to enter a numeric value or a variable name for the tap setting of the relay. Tap has been entered to reflect the value provided in the Settings tab or a previous Calculation.

**Time Dial:** value is to enter a numeric value or a variable name for the time dial setting of the relay. Time\_Dial has been entered to reflect the value provided in the Settings tab or a previous Calculation.

*The resultant time overcurrent characteristic.*



## Theo RIO

# Theo RIO

RIO files are created using relay and relay test software by various manufacturers. They may be considered to be Relay Impedance Objects, but are also allocated to other characteristics such as families of time-amplitude.

**Note:** Only files that contain **RIO impedance characteristics** are usable with the AVTS Import tool. AVTS has the IEEE/ANSI and IEC time-amplitude characteristics available independently in the AVTS Theoretical Object list of options.

RIO files constitute data for the characteristics of a particular relay with specific settings. Some or all the characteristic types may be created in the file and the characteristic relay settings will be included. I.e., the RIO files are relay specific to the settings of the relay when the RIO file is created; the discrete settings are not displayed nor adjustable.

The Theo RIO Import topic describes the file import procedure.

The Theo RIO Example Test topic provides an example of the test of the RIO import and the test results.

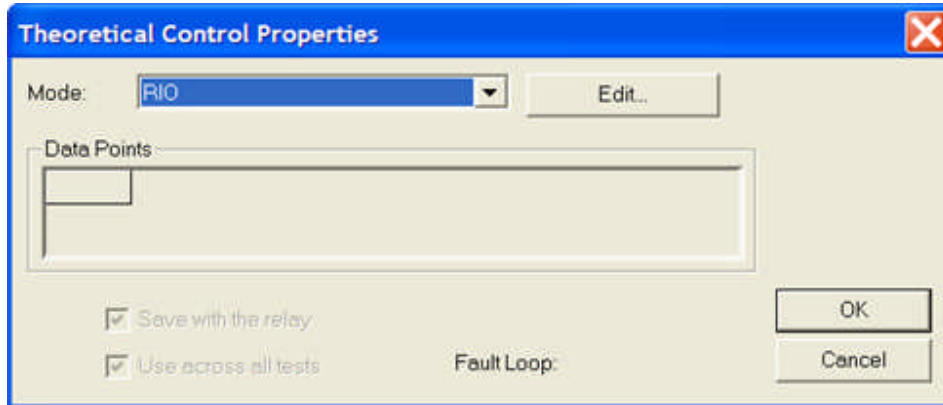
RIO files may also be imported and executed in the Online Click on Fault Control. All Control dialogs, menus and tools are defined in detail in the Online Click on Fault Control.

**Note:** Fault Loops apply to RIO files imported directly using the Online Click On Fault Control and the Click On Fault Control.

## Theo RIO Import

Note: RIO files may also be imported and executed in the Online Click on Fault Control. All Control dialogs, menus and tools are defined in detail in the Online Click on Fault Control.

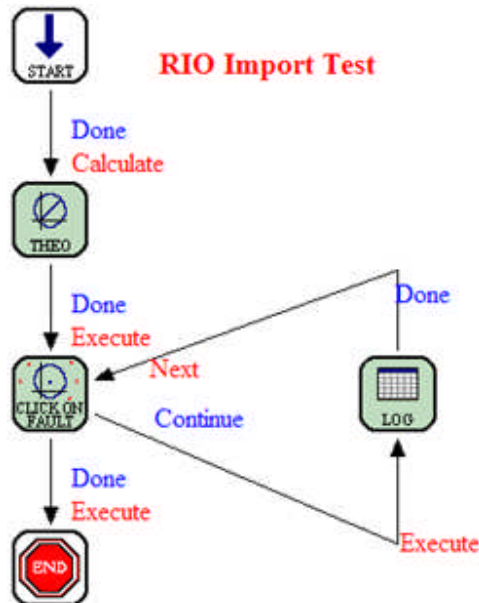
The following procedure describes *Import of a RIO file* and test:



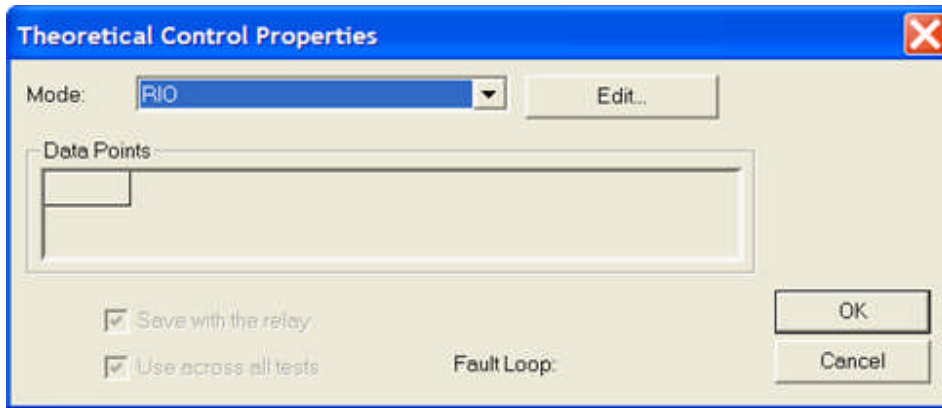
Create a New Device for the RIO file to be created.

Create a New Relay for the RIO file to be imported.

Create a New Test manual, for the Import of the RIO file. *Create a test* that includes a Theoretical Control and a Click on Fault Control.

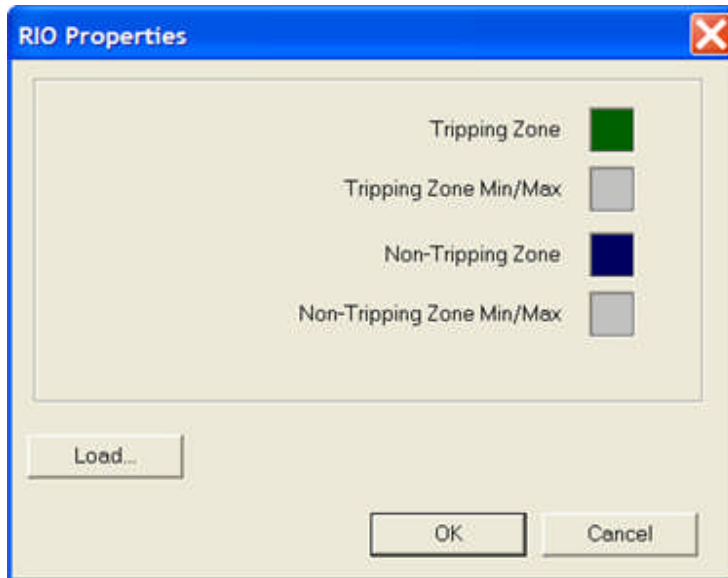


Open the *Theoretical Control Properties* dialog and select RIO in the Mode: drop down list.

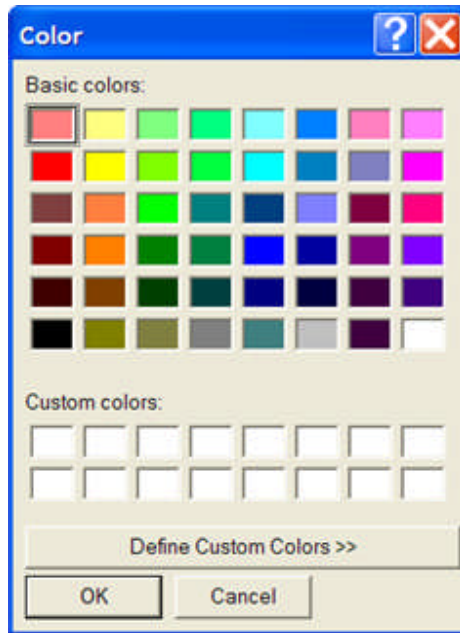


Note: Selecting RIO automatically checks the Save with the relay and the Use across all tests check boxes. Any additional tests for this relay containing a Theoretical Control will have it changed in agreement to this action.

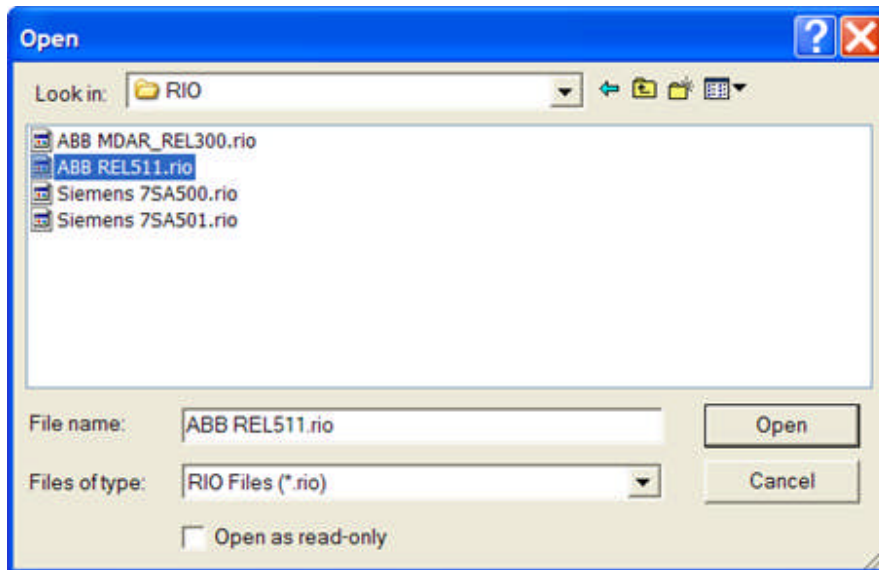
Select the Edit... button to open the *RIO Properties* dialog.



Note: The dialog contains "color" buttons to select the desired colors of the imported characteristics and their parameters. Click on a color button to display the *color pallet*.



Select the Load... button to display the *Open* dialog to locate and select the RIO file to be Imported.

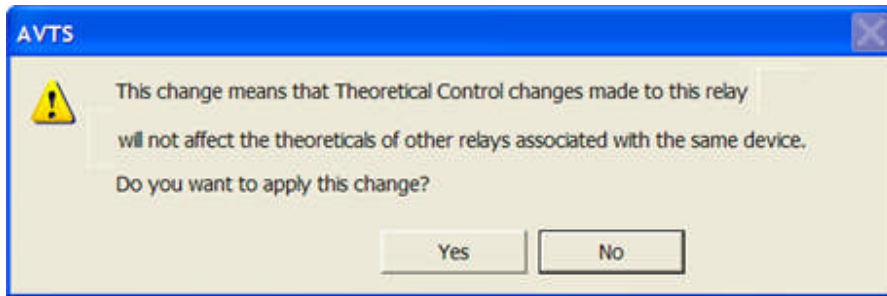


The **ABB REL511.rio** file is selected for the import.

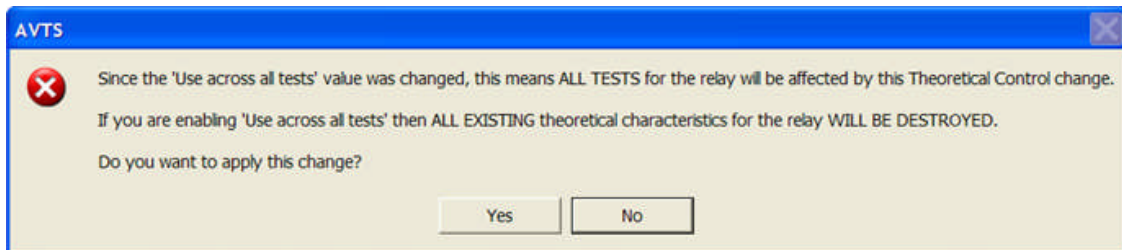
Select the Open button to return to the RIO Properties dialog.

Select the OK button in the RIO Properties dialog to return to the Theoretical Control Properties dialog.

Select OK to display an *AVTS message* with Yes and No options. The message defines the effect this file import change makes to this relay and others of the same Device type.

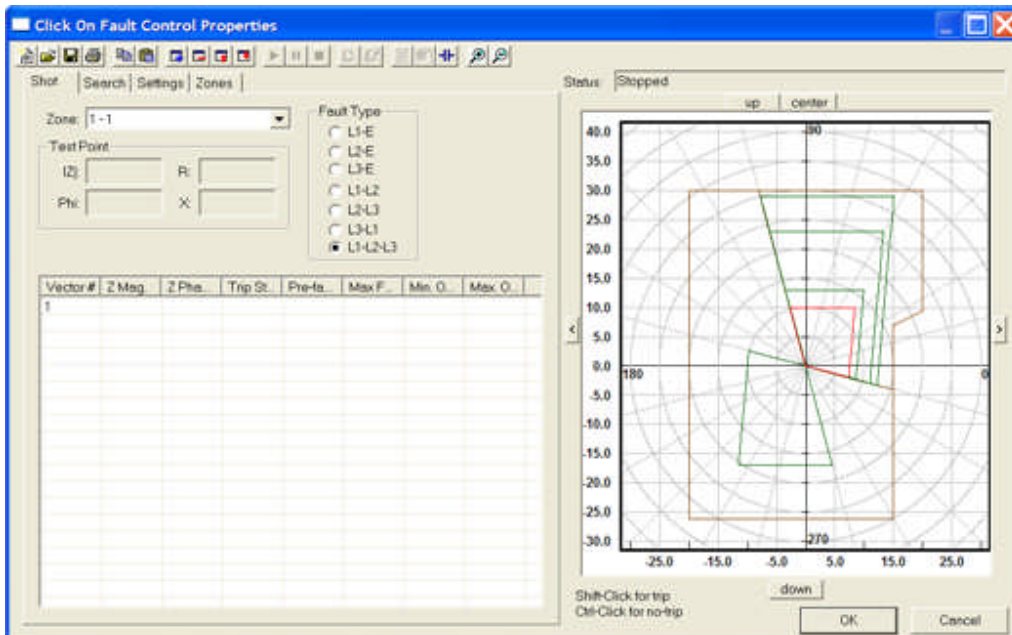


Select Yes to display an *AVTS WARNING message* with Yes or No options. The message warns that accepting this import will change alter all other Theoretical Controls for the open relay in agreement to this import.



Select Yes to return to the Relay Test Editor tab of the open relay with the displayed test.

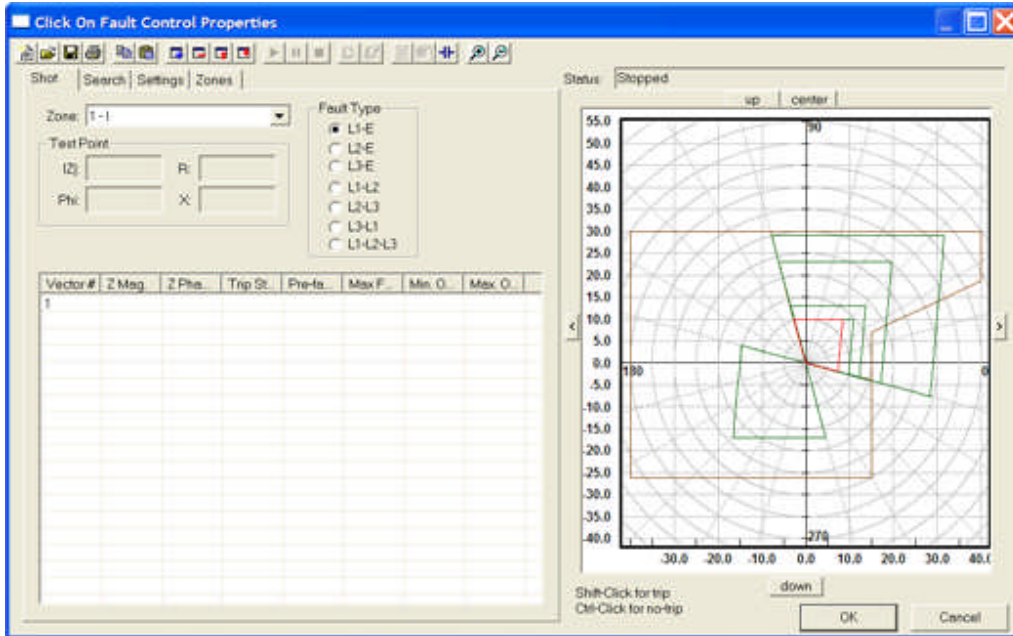
Double click the *Click on Fault Control* to display properties dialog. The dialog contains all the characteristics for the relay contained in the imported RIO file.



**Note:** This display has the L1-L2-L3 Fault Type button selected and the corresponding phase characteristics are displayed graphically.

This example displays all the characteristics set in the relay for Phase faults. If the relay has other characteristics set for faults involving Zero sequence components, *these characteristics* will be displayed, ready for tests, by selecting one of the earth (ground) Fault Type check buttons.





Note: This display has the L1-E Fault Type button selected and the corresponding characteristics are displayed graphically.

## Theo RIO Example Test

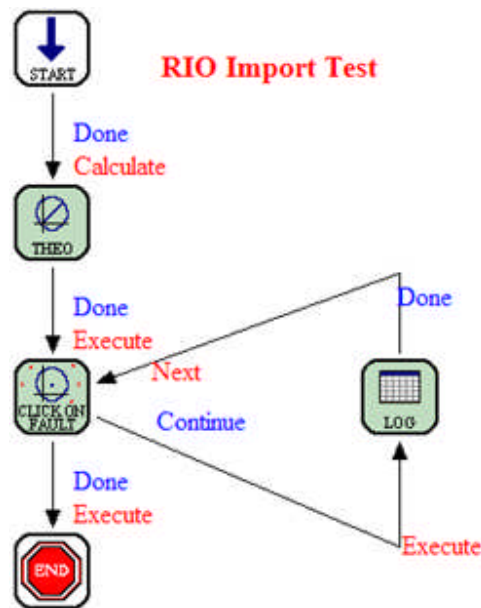
The RIO files available from several relay and test equipment manufacturers are available to be imported directly into AVTS. Various tests may be performed using the files to test relays that have the same developed characteristics and discrete settings.

Note: RIO files may also be imported and executed in the Online Click on Fault Control. All Control dialogs, menus and tools are defined in detail in the Online Click on Fault Control.

The Import process to be exercised in a created and saved Relay Test Editor tab is described in the Theo RIO Import topic.

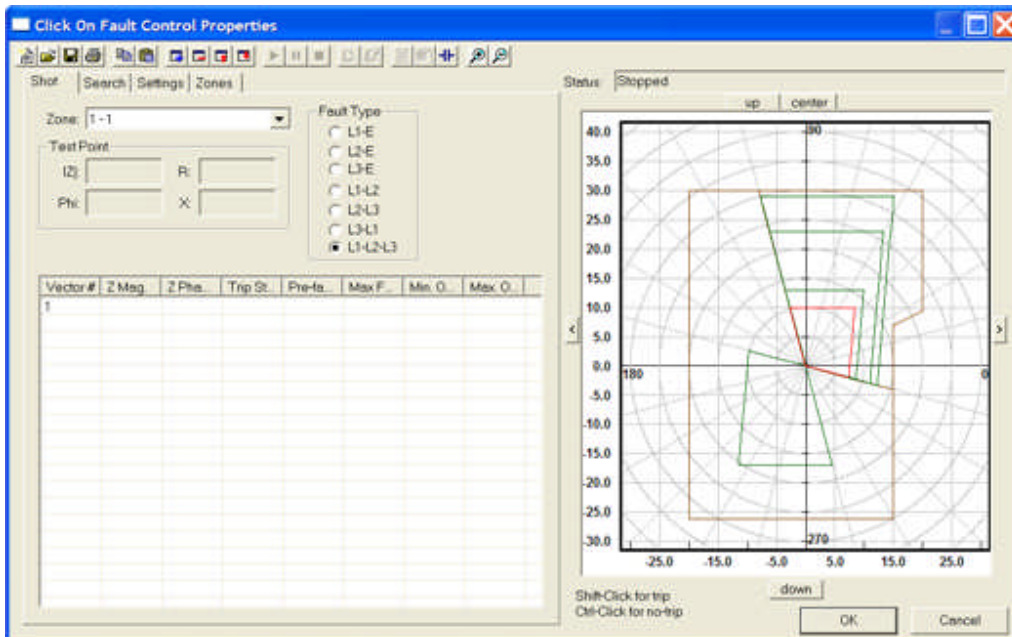
This example, using the **ABB REL511.rio** file, demonstrates the use of the imported RIO file after it has been imported using the Theo RIO option.

The **test procedure** for testing the relay using the imported file includes the Start, Theoretical, Click on Fault, Log and End Controls in the Relay Test Editor tab.

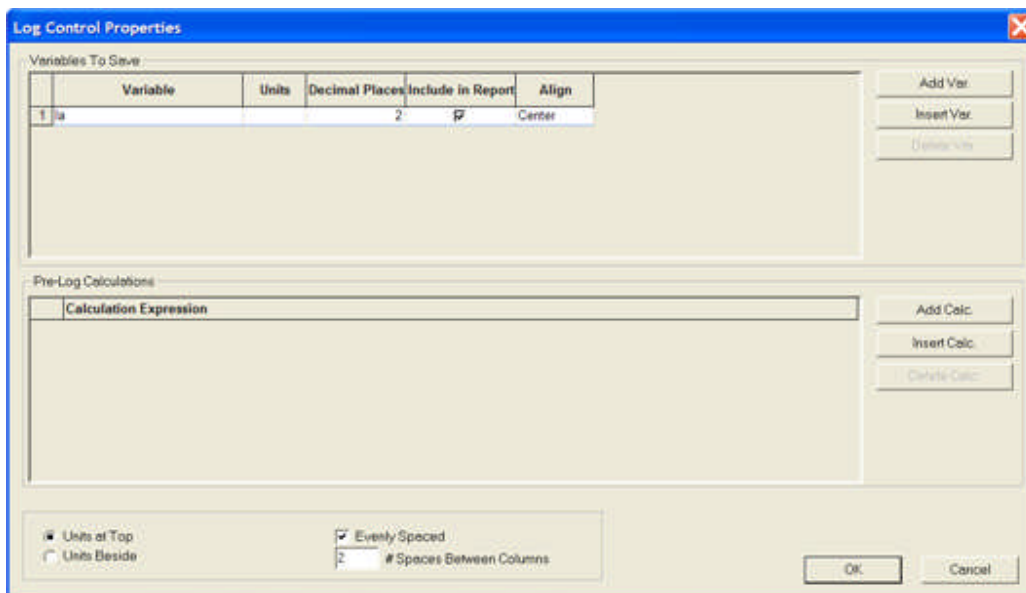


The RIO file is imported using the Theoretical Object option of the Theoretical Control.

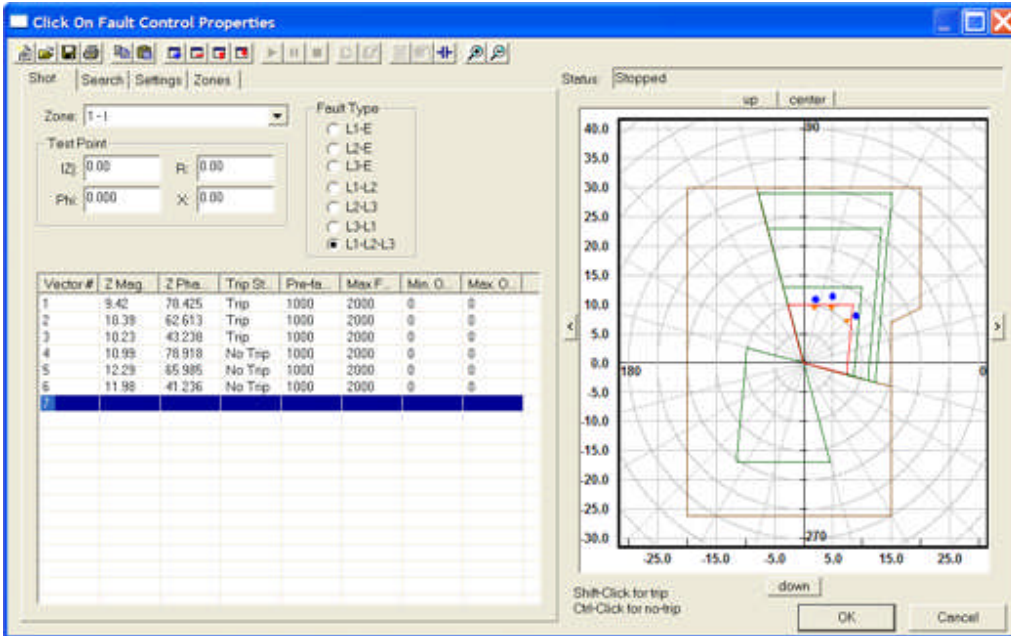
The Click on Fault Control, when opened to its properties, displays the *imported characteristics for a L1-L2-L3 Fault Type*.



The *Log Control Properties* is opened to display the default Variable entry **la**.

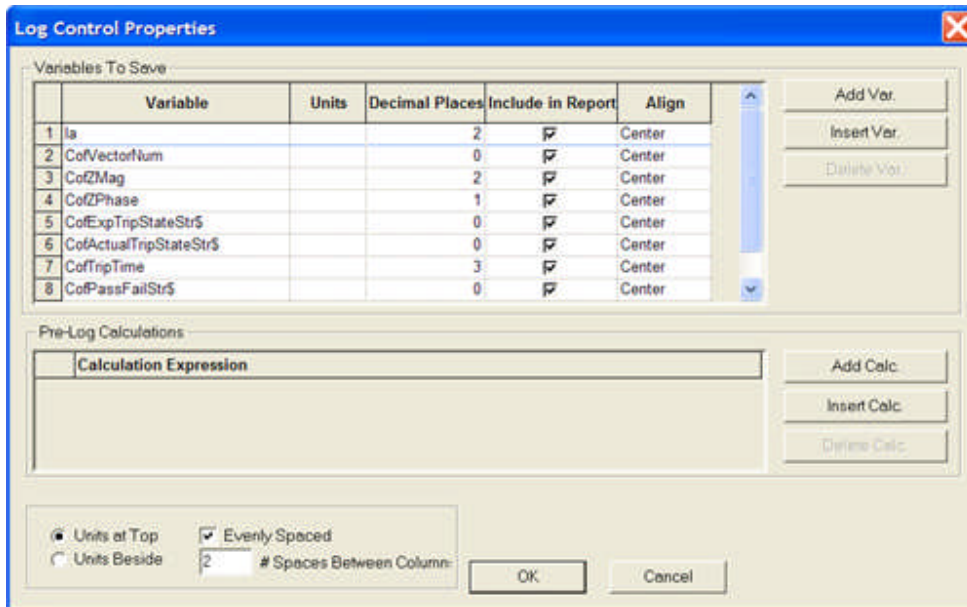


Open the Click on Fault properties dialog and *enter test points in the Graph* area using the Shift-Click for trip and the Ctrl-Click for no-trip test points.

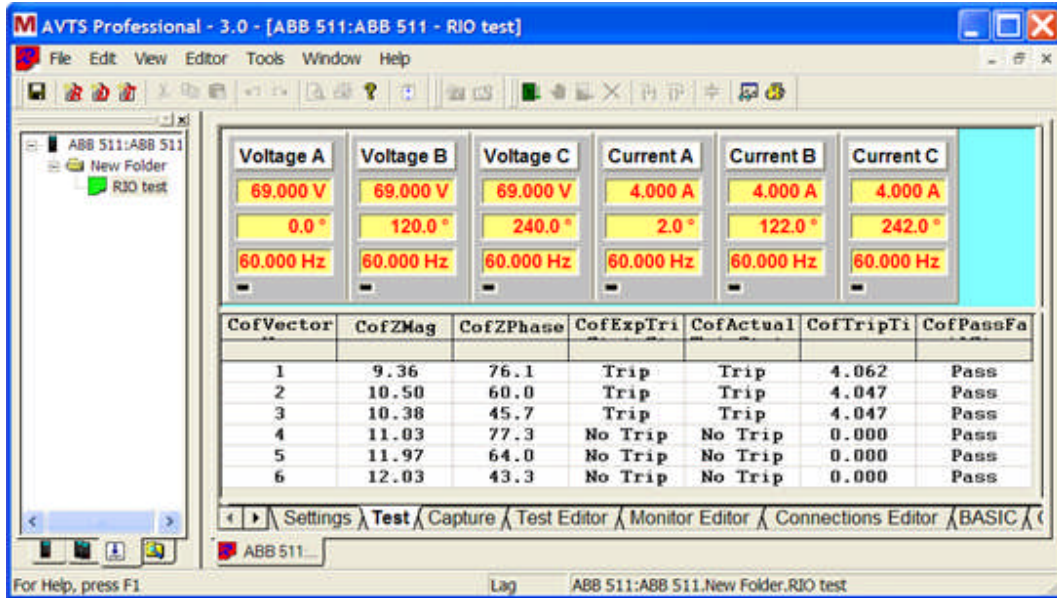


Note: Trip points are displayed as Red Triangles and No-trip points are displayed as Blue Circles. Highlighting one of the Test Vectors will highlight the corresponding graph test point Green.

After selecting test points, the *Log Control Properties* are automatically populated with the Variables that will display the collected results of the executed tests.



Execute the tests that were created and view the *Tests Results of the Relay* responses generated.



Save the test results for future *print using AVTS*. These results are for the L1-L2-L3 executed faults.

### Relay Results Report

Serv Ctr: RIO Sub: RIO Eqp: RIO	Relay: ABB 511 Device: RIO ABB 511: ABB 511 Tested By: Bruce Bordelon Date Tested: 7/3/2007
---------------------------------------	------------------------------------------------------------------------------------------------------

Setting	Value	Description
Group Name		
Catalog Number		This is the relay serial number

Relay Notes:

Test Name: New Folder.RIO test

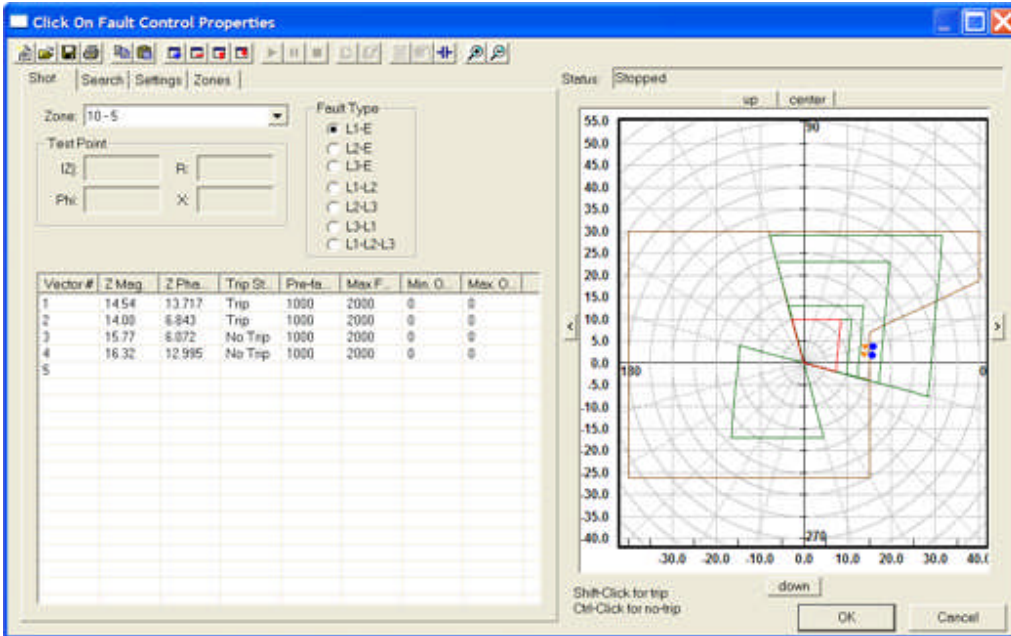
CofVector	CofZMag	CofZPhase	CofExpTri	CofActual	CofTripTi	CofPassFa
1	9.36	76.1	Trip	Trip	4.062	Pass
2	10.50	60.0	Trip	Trip	4.047	Pass
3	10.38	45.7	Trip	Trip	4.047	Pass
4	11.03	77.3	No Trip	No Trip	0.000	Pass
5	11.97	64.0	No Trip	No Trip	0.000	Pass
6	12.03	43.3	No Trip	No Trip	0.000	Pass

Page 1 of 1

Fault tests may be executed for the same test points for any of the Phase to Phase Fault Types by selecting one and executing the tests.

If there are characteristics for other Zones or Fault Types, they may be executed from the Zone drop down list.

Open the Click on Fault properties dialog and *enter test points in the Graph* for the L1-E Fault Type selection.



Execute the tests and save the results.

View the Test Results of the new selections using *Export Results to Word* option.

### Relay Results Report

ServCtr: RIO	Relay: ABB 511
Sub: RIO	Device: ABB 511
Eqp: RIO	Tested By: Bruce Bordelon
	Date Tested: 07/03/2007

Setting:	Value:	Description:
Group Name	Catalog Number	This is the relay serial number

New Folder.RIO test						
CorVectorNu	CorZMag	CorZPhase	CorExpTripSt	CorActualTri	CorTripTime	CorPassFailSt
m			ateStr	pStateStr		r
1	14.54	13.7	Trip	Trip	4.062	Pass
2	14.00	6.8	Trip	Trip	4.063	Pass
3	15.77	6.1	No Trip	No Trip	0.000	Pass
4	16.32	13.0	No Trip	No Trip	0.000	Pass

View and print results may also be accomplished using the AVTS Current Results option on the Toolbar.

## Vector Control

The **Vector Control** is to place quantities of Vector Voltages and Currents and their characteristics into a test procedure. Up to 16 Vector states may be created and saved for applying a series of states, each with its own time period and any other parameters in the Prefix edit field, to a relay.

The Voltage and Currents edit fields are to set the parameters for each channel and defined in the Ramp Control Properties topic.



An example of this Control is demonstrated in Test Editor OC Time Example.

### Vector Control Properties

Positive values represent lagging angles.

Voltages						
Name	Amplitude	Phase	Frequency	Mode	Color	On
VA	69.000	0.0	60.000	AC	Blue	<input type="checkbox"/>
VB	69.000	120.0	60.000	AC	Green	<input type="checkbox"/>
VC	69.000	240.0	60.000	AC	Cyan	<input type="checkbox"/>

Currents						
Name	Amplitude	Phase	Frequency	Mode	Color	On
IA	2.000	0.0	60.000	AC	Blue	<input type="checkbox"/>
IB	2.000	120.0	60.000	AC	Green	<input type="checkbox"/>
IC	2.000	240.0	60.000	AC	Cyan	<input type="checkbox"/>

Apply Time: 0 Cycles

Orientation: 0 at 3 O'clock

Diagram Mode: Voltage / Current

Dragging Parameters: Enable: Amplitudes:  0.01 Phases:  0.1

Auto Scale: Voltage Scale: 150 Current Scale: 15

Show Labels For:  Voltages  Currents

Start Timer:  Stop Timer:  Aux Contact:

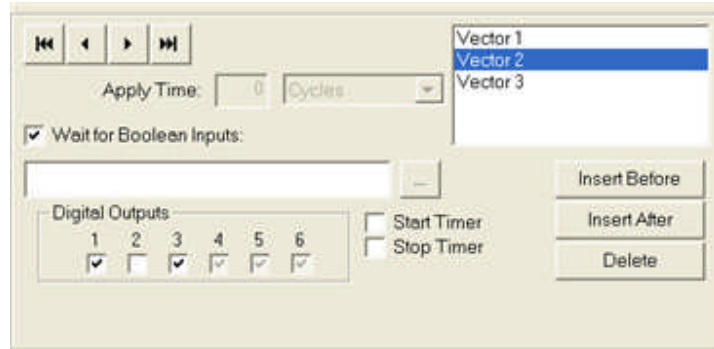
Buttons: Insert Before, Insert After, Delete, OK, Cancel

If the Control is to precede a HW Timer Control, then the Start Timer function is not used. Time will automatically start upon entry to the HW Timer Control.

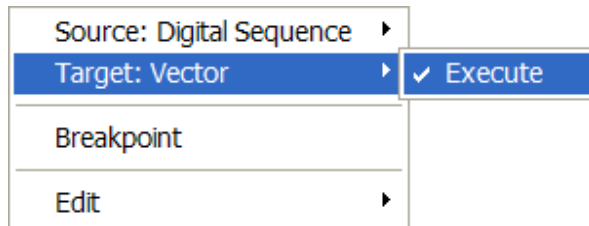
If the Control is to set balanced conditions to a relay for a few moments, then the Start Time may be used to apply the Vector set for a discrete time and then pass to another Control other than a HW Timer Control.

Explanations of Control elements are in the Online Vector Control... topics.

The *Vector Control Properties* has hidden the Boolean Equation logic input capability when used with Pulsar. If Boolean logic for the Inputs of the MPRT are to be used for control of different Vector States, then the MPRT Functionality must be checked in the Start Control Properties dialog.

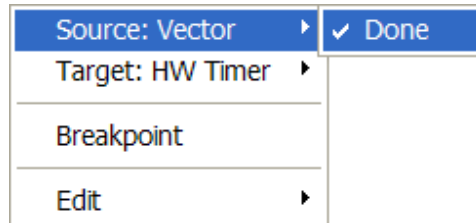


*Target Link* is Execute.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.



## Waveform Control

The **Waveform Control** is to call a file, in COMTRADE converted to HEX format, and have the waveform load and playback using the MPRT or PULSAR test systems.

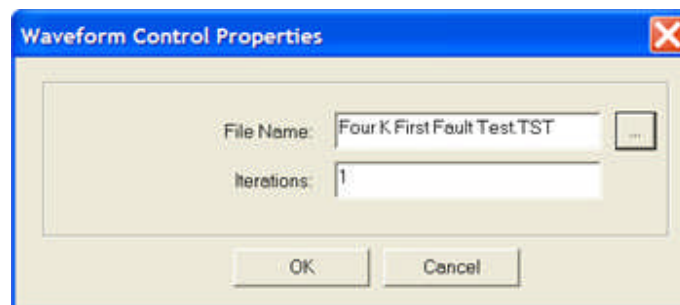
The Waveform Control is used to play a waveform, but there is no capture of a graphic of the waveform. Any resulting operations by the relay under test may be retrieved and logged using the Log Control. The captured data may be saved as test results for review in the future.

An example of the conversion process is described in the COMTRADE to Hexidecimal Conversion topic.



The Waveform Control may also be used to playback waveforms created with the use of the Sin Sum formula, which is useful with the Pulsar and with the latest release of MPRT firmware. Accompanying the Sin Sum command are the new Set Wave Index and the Point on Wave topics. The latest release of MPRT firmware Commands are documented in the Command Set Procedures - New group of topics. These new Commands may be placed in a text file, given a name, saved to the AVTS Waveform subfolder, and called by the Waveform Control to be played back during the test procedure.

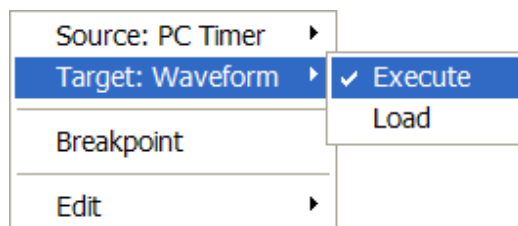
*Waveform Control Properties* contains a file location field and numeric entry field.



The File Name: edit field is to enter the name of the file to be called. Using the browse button defaults to find a waveform in the C:\Program Files\Megger\AVTSv2\Waveforms folder. In order for the file to play it must be a test file, with the \*.tst extension, having been created and saved using the DFR Waveform Viewer.

the Iterations: edit field is to enter an integer for the number of consecutive times the waveform will play without interruption..

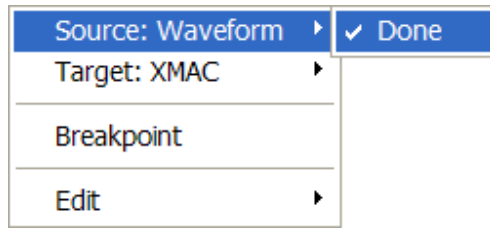
*Target Links* are Execute and Load.



The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Load command registers a Waveform file name from another Control. The properties may then have a variable in place of a file name.

*Source Link* is Done.



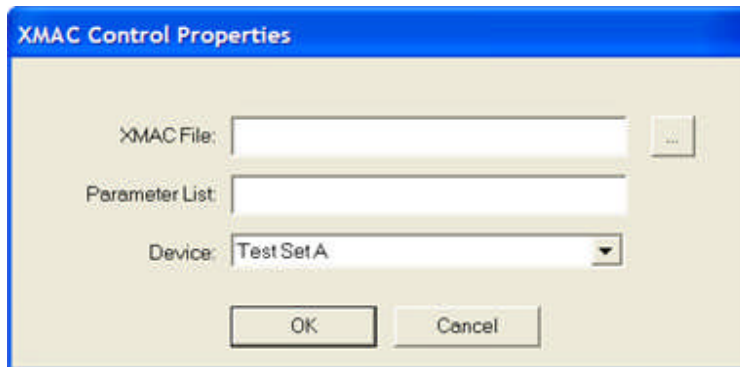
The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## XMAC Control

The **XMAC Control** is to use files for use with PULSAR and may be applied to MPRT, in part. Full explanation for using XMAC is found in the topics under the XMAC Library Functions book. An example of the construction of a XMAC file is in the XMAC Demo Program topic.

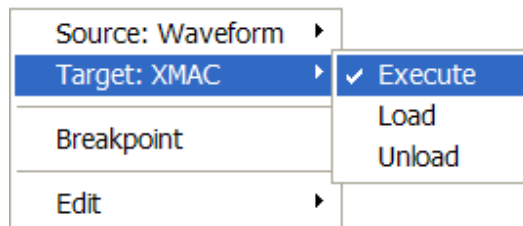


### XMAC Control Properties



A typical use for an XMAC is for use in an end-to-end test to allow synchronization to a GPS receiver for triggering a fault at each end of a transmission line. Test Editor End-To-End Example

*Target Links* are Execute, Load and Unload.

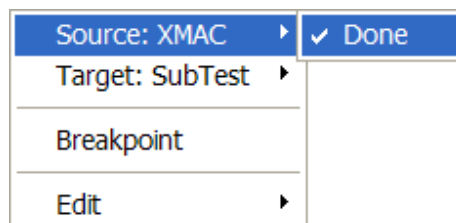


The Execute command applies immediately after the completion of the properties of the previous Control connected.

The Load command loads the XMAC into the PULSAR.

The Unload command removes the previously loaded XMAC from the PULSAR.

*Source Link* is Done.



The Done command concludes the activity of the Control and passes the procedure execution to the next Control.

## **Monitor Editor Tab**

### **Monitor Editor Tab**

The **Monitor Editor Tab** contains a toolbar to place tools into the Workbook pane which will reflect the real time actions of the test set amplifiers, external inputs, external outputs, and any variables properly defined.

The tools for creating the mimics are displayed in the Monitor Editor Toolbar at the right hand side of the Workbook pane Monitor Editor Tab.

Within this tab Monitor Editor Tab drop menu is available for ready edit of the tools which are in place in the tab.

Each available tool is defined in the Monitor Editor Toolbar topic.

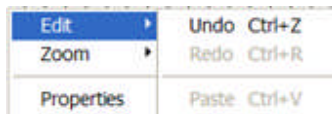
## Monitor Editor Tab Drop Menu

In the Workbook **Monitor Editor** tab, a right mouse click in the Workbook area but not on a Monitor tool will produce a *drop down menu* of options.



The options are:

*Edit* is to provide:

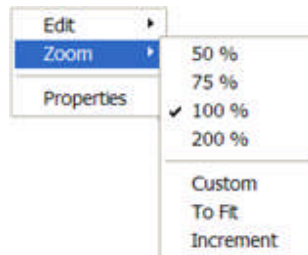


Undo is to return to the preceding edit or delete state.

Redo is to return to the state prior to an Undo.

Paste

*Zoom* is to provide:



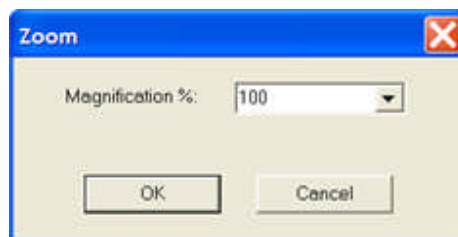
50% is to expand the pane objects to approximately one-quarter size

75% is to expand the pane objects to approximately one-half size

100% is to display the pane objects to the default normal size

200% is to display the pane objects to approximately double size

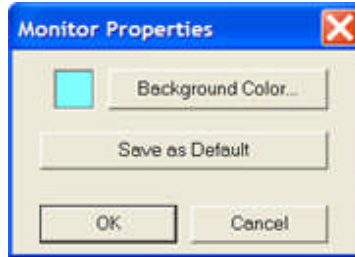
*Custom* is to display the Zoom dialog for edit to make small or enlarge the objects in the pane



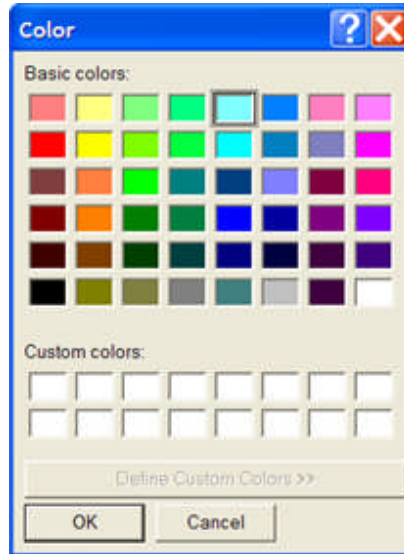
To Fit is to display all objects to fit the pane size

Increment to allow the mouse to drag a window about objects and have enclosed objects fill the pane

*Properties* is to provide:



*Background Color palette* for the Monitor upper display in the Workbook Test tab.



The Save as Default option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

## Monitor Editor Toolbar

## Monitor Editor Toolbar

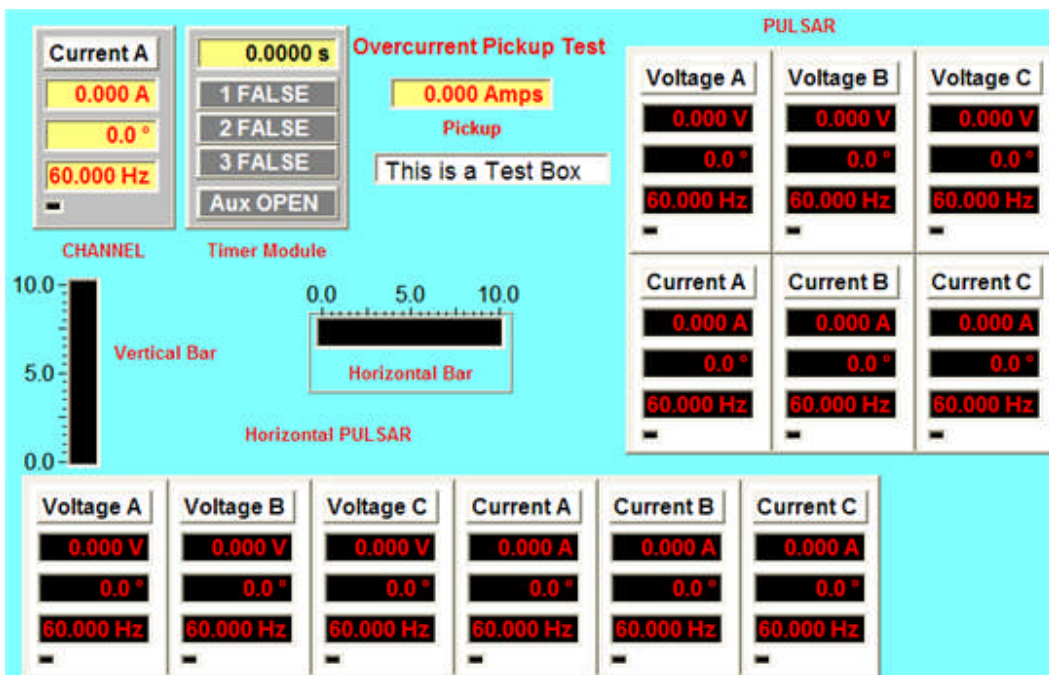
The Monitor Editor tab **Controls** tool bar resides to the right hand side of the open Relay Workbook. Each is a program application for placing in the Workbook and linked to create a test sequence.

Each Control is defined and exemplified in its respective topic.

Select Tool, Line Tool, Rectangle Tool, Text Tool, PULSAR Tool, Horizontal PULSAR Tool, PULSAR Channel Tool, Timer Module Tool, Vertical Bar Tool, Horizontal Bar Tool, Text box Tool, Value Tool



A display with Monitor Tools is in the upper *Monitor pane* in the Test tab.



## Select Tool

The **Select Tool** has no properties. It has the actually the mouse cursor and is enabled by "picking" with the mouse a Tool from the Toolbar. It is also used to select one or more controls for moving within the Workbook.

The Select Tool



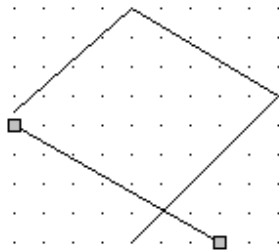


## Line Tool

The **Line tool** is used to "draw" a line dragged with the mouse from one location to another. Select the tool, left click in the Workbook, hold the left mouse key and drag a line. When the second location is reached, release the mouse key to place the desired line.



A line drawn will be active when the second point is produced and have blocks on each end. The blocks are to grab and move the line end to another location. Place the cursor on mid-line to grab and move the entire line without changing the line angle to horizontal.



## Rectangle Tool

The **Rectangle tool** is to draw a rectangle without having to produce one drawing four separate lines. The rectangle may be used to enclose several items in a box.

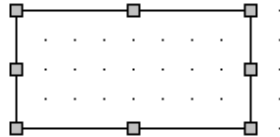
It is used to "draw" a box dragged with the mouse from one location to another. Select the tool, left click in the Workbook, hold the left mouse key and drag a rectangle. When the second location is reached, release the mouse key to place the desired rectangle.

Rectangle Tool



This tool may be resized by grabbing a corner with the mouse and moving in or out.

A rectangle drawn will be active when the second point is produced and have blocks on each corner and mid-sides. The blocks are to grab and change size and location of the rectangle. Place the cursor in the rectangle to grab and move it without changing the line angle to horizontal.



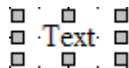
## Text Tool

The **Text tool** is used to enter formatted text in desired areas of the Monitor Editor tab to further detail the test apparatus.

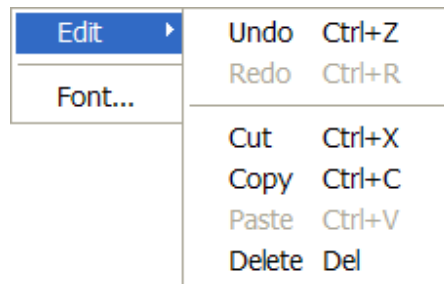
The Text tool



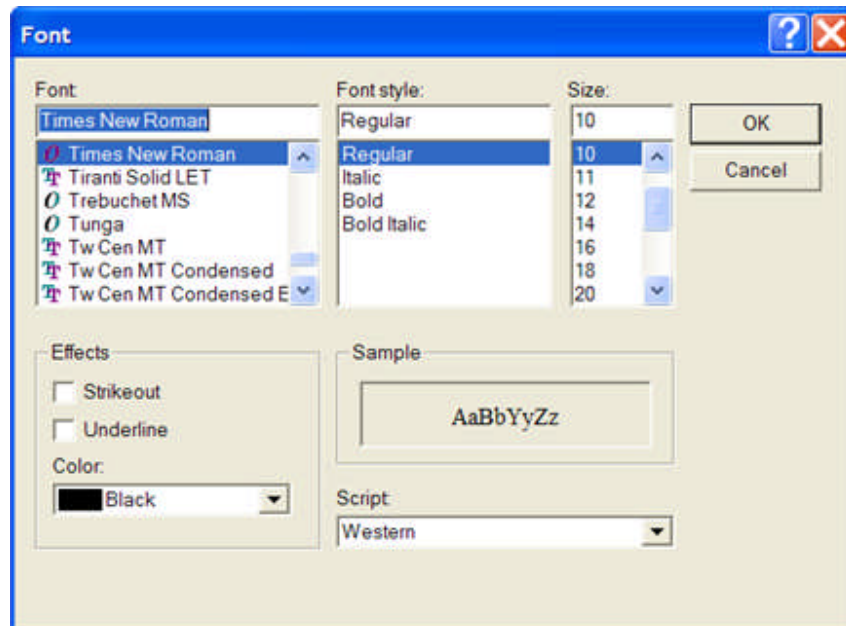
The tool is shown with the eight squares indicating that it has been placed in the Workbook pane for use. This tool may be resized by grabbing a corner with the mouse and moving in or out.



The desired Text entry is created by double left mouse click, highlighting the text and then entering text with use of the keyboard. Or, right mouse click on the Tool to produce the *Text menu* and select the Font option. The Edit options are the standard Windows options. Using the Edit options text may be Cut/Paste elsewhere, Copy/Paste for repetition and Delete.



Font option opens the standard Windows *Font dialog* for selecting the desired font appearance. Note: If "formatted" text is present, it is usually more efficient to copy the existing text, pasting the copied text and then edit the text with a double left mouse click. The action retains the format of the copied text.



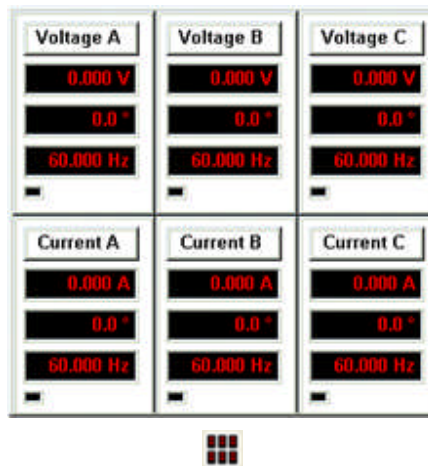
An example of formatted text next to a Workbook Monitor for additional information about the test procedure.

## Overcurrent Pickup Test

## PULSAR Tool

The **PULSAR Tool** is to place a replica in the Monitor tab to display the six channels with amplitude, phase angle and frequency. The three voltage channels are above the three current channels similar to PULSAR. The channels may be used for MPRT.

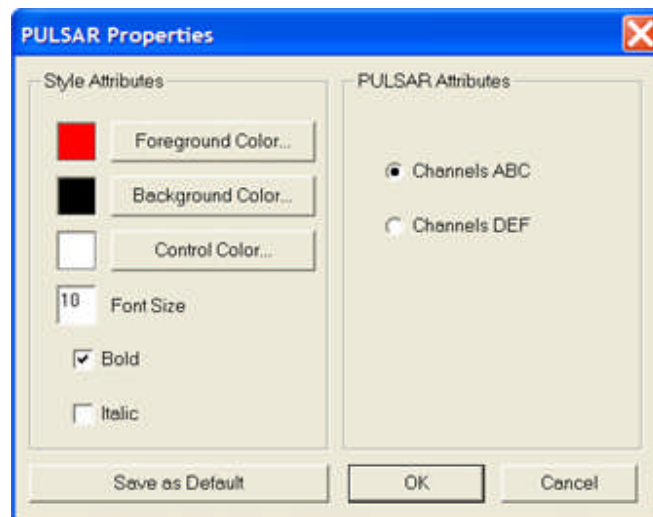
The **PULSAR Tool**



The PULSAR parameters are active during the execution of a test. A small box under the frequency display will indicate in color when the channel has been given the ON command.

This tool can be moved, but cannot be resized.

Right mouse click or double click on the tool to evoke the **PULSAR Properties** dialog. The dialog has Style Attributes block for Tool Color and Font selections and a PULSAR Attributes block to select the active channels designation group.



Style Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the "numbers" for amplitude, phase angle and frequency.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Control Color...** button evokes a color palate for color selection of the Tool facing.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude, phase angle and frequency fields.

**Bold** is whether to place the amplitude, phase angle and frequency numbers bold or not.

**Italic** is whether to place the amplitude, phase angle and frequency numbers in italics or not.

PULSAR Attributes box provides for:

**Channels ABC** check bullet select channel labels ABC only, or

**Channels DEF** check bullet select channel labels DEF only.

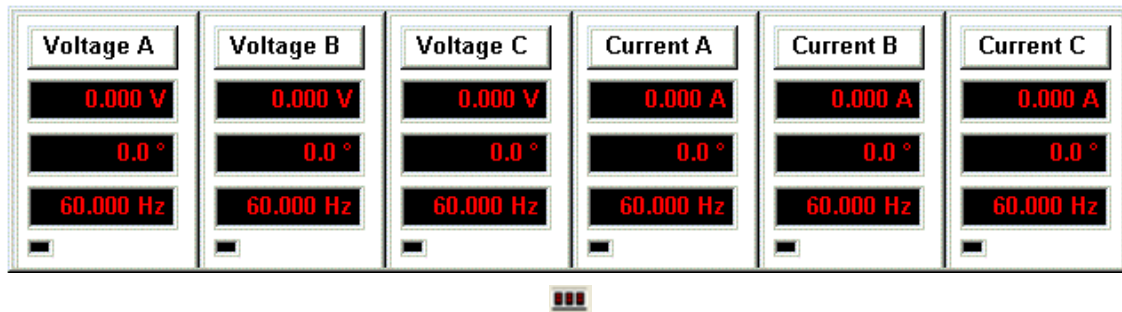
**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

## Horizontal PULSAR Tool

The **Horizontal PULSAR Tool** is to place a replica in the Monitor tab to display the six channels with amplitude, phase angle and frequency. The three voltage channels are to the left of the three current channels similar to PULSAR. The channels may be used for MPRT.

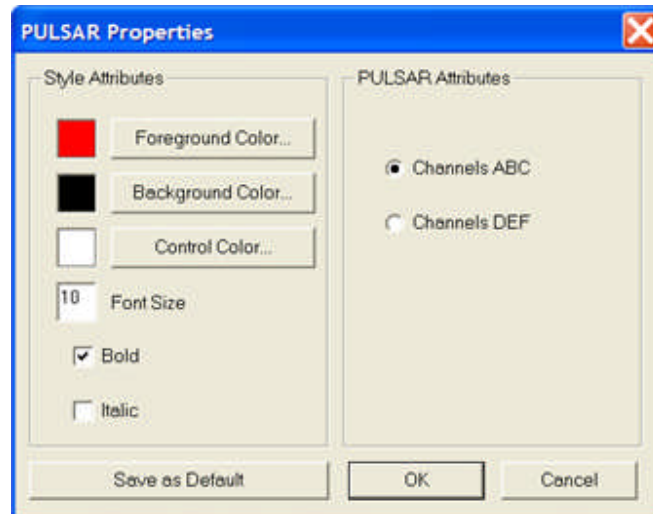
The *Horizontal PULSAR* Tool



The PULSAR parameters are active during the execution of a test. A small box under the frequency display will indicate in color when the channel has been given the ON command.

This tool can be moved, but cannot be resized.

Right mouse click or double click on the tool to evoke the *PULSAR Properties* dialog. The dialog has Style Attributes block for Tool Color and Font selections and a PULSAR Attributes block to select the active channels designation group.



Style Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the "numbers" for amplitude, phase angle and frequency.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Control Color...** button evokes a color palate for color selection of the Tool facing.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude, phase angle and frequency fields.

**Bold** is whether to place the amplitude, phase angle and frequency numbers bold or not.

**Italic** is whether to place the amplitude, phase angle and frequency numbers in italics or not.

PULSAR Attributes box provides for:

**Channels ABC** check bullet select channel labels ABC only, or

**Channels DEF** check bullet select channel labels DEF only.

**Save as Default** option will retain the selection to the database for all new tests created.

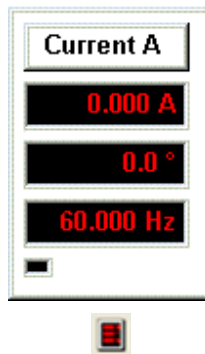
**Note:** The Save as Default will not affect previously created tests.



## PULSAR Channel Tool

The **PULSAR Channel Tool** is to place a replica in the Monitor tab to display a channel with amplitude, phase angle and frequency. Each channel placed in the Workbook may be a voltage or a current channel. The channels may be used for MPRT and as many as needed may be used.

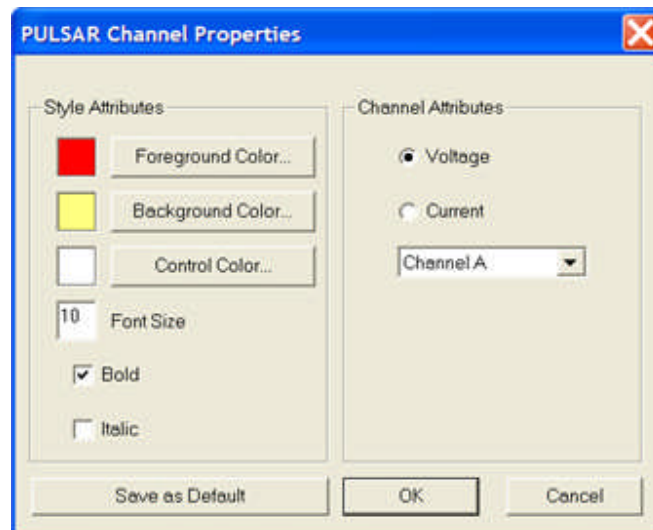
The *PULSAR Channel* Tool



The PULSAR parameters are active during the execution of a test. A small box under the frequency display will indicate in color when the channel has been given the ON command.

This tool can be moved, but cannot be resized.

Right mouse click or double click on the tool to evoke the *PULSAR Properties* dialog. The dialog has Style Attributes block for Tool Color and Font selections and a PULSAR Attributes block to select the active channels designation group.



Style Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the "numbers" for amplitude, phase angle and frequency.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Control Color...** button evokes a color palate for color selection of the Tool facing.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude, phase angle and frequency fields.

**Bold** is whether to place the amplitude, phase angle and frequency numbers bold or not.

**Italic** is whether to place the amplitude, phase angle and frequency numbers in italics or not.

PULSAR Attributes box provides for:

**Channels ABC** check bullet select channel labels ABC only, or

**Channels DEF** check bullet select channel labels DEF only.

**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

## Timer Module Tool

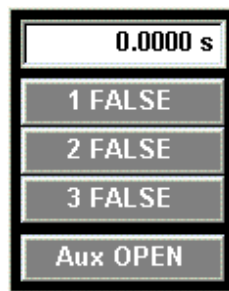
The **Timer Module tool** is to mimic the Timer Module in PULSAR to display active elapsed time.

The upper field is for time amplitude display. The 1 FALSE, 2 FALSE and 3 FALSE are fields to replicate the activity taking place on the Three timer Module monitor inputs. Whether monitor is for presence or voltage or closed contact, the fields will display 1 TRUE, etc. if the module is set for that action to stop timer. FALSE is shown as default in the test tab for a non-executed test.

The first three monitor inputs on MPRT replicate the three monitor inputs on PULSAR; the tool is true for PULSAR or MPRT.

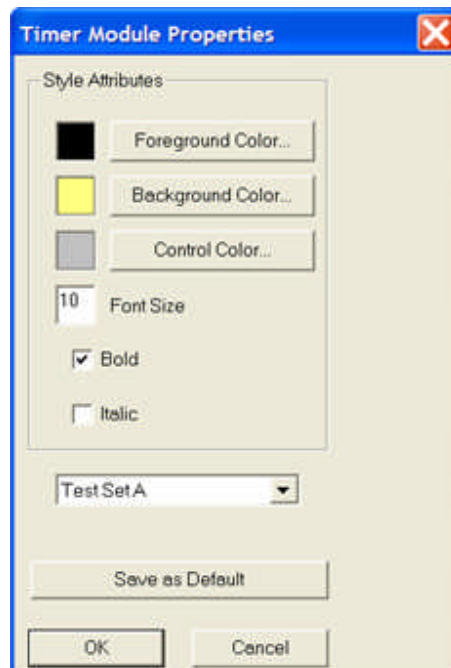
The Aux OPEN field is to replicate the auxiliary contact in the default OPEN mode in the Timer Module. The display is Aux CLOSED for a closed contact.

The *Timer Module* Tool



This tool can be moved, but cannot be resized.

Right mouse click or double click on the tool to evoke the *Timer Module Properties* dialog. The dialog has Style Attributes block for Tool Color and Font selections and a PULSAR Attributes block to select the active channels designation group.



Style Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the "numbers" for amplitude.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Control Color...** button evokes a color palate for color selection of the Tool facing.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude field.

**Bold** is whether to place the amplitude numbers bold or not.

**Italic** is whether to place the amplitude numbers in italics or not.

**Test Set** select drop list to select TestSetA or TestSetB to be in agreement with the Tools/Options/Devices tab.

Save as Default option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

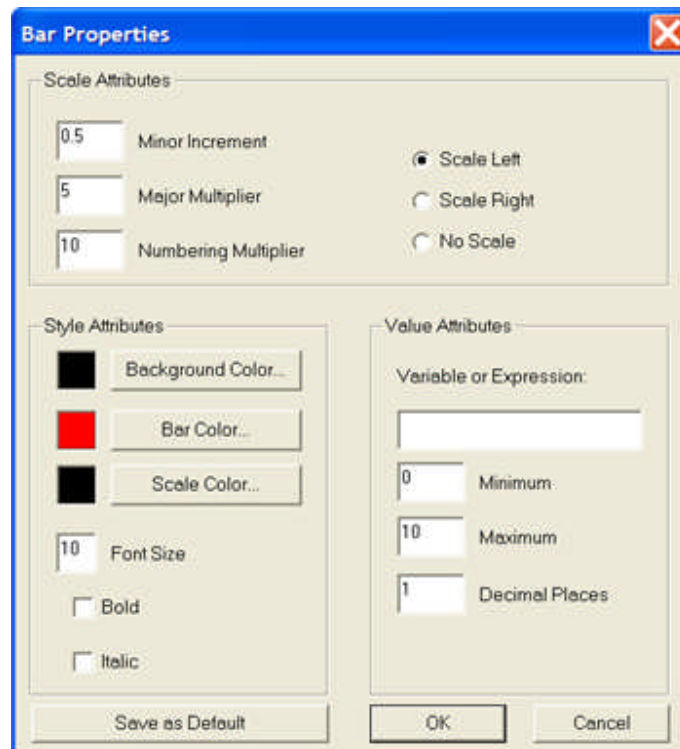
## Vertical Bar Tool

The **Vertical Bar Tool** is to provide an active moving display for a monitored variable.

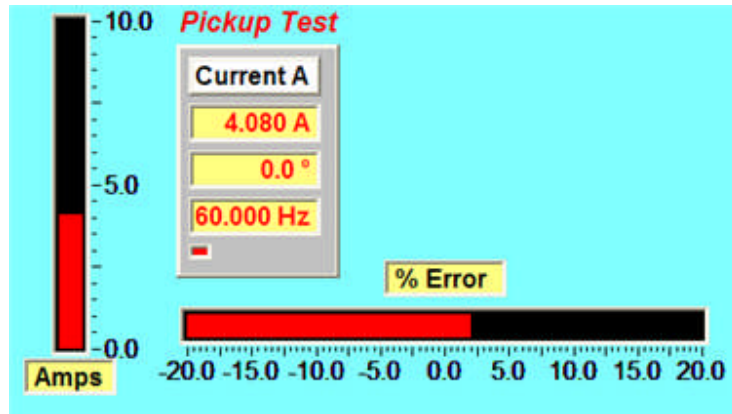
The *Vertical Bar* Tool



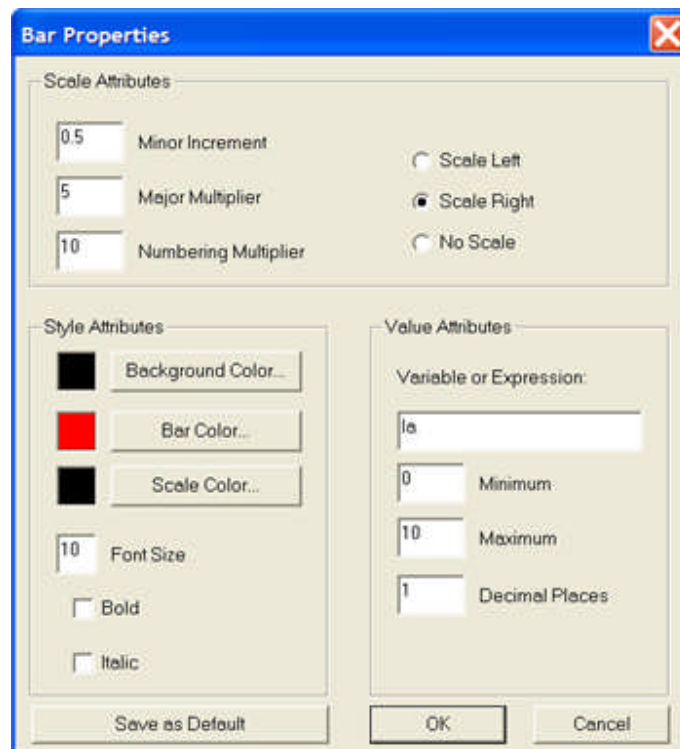
Right mouse click or double click on the tool to evoke the *Bar Properties* dialog.



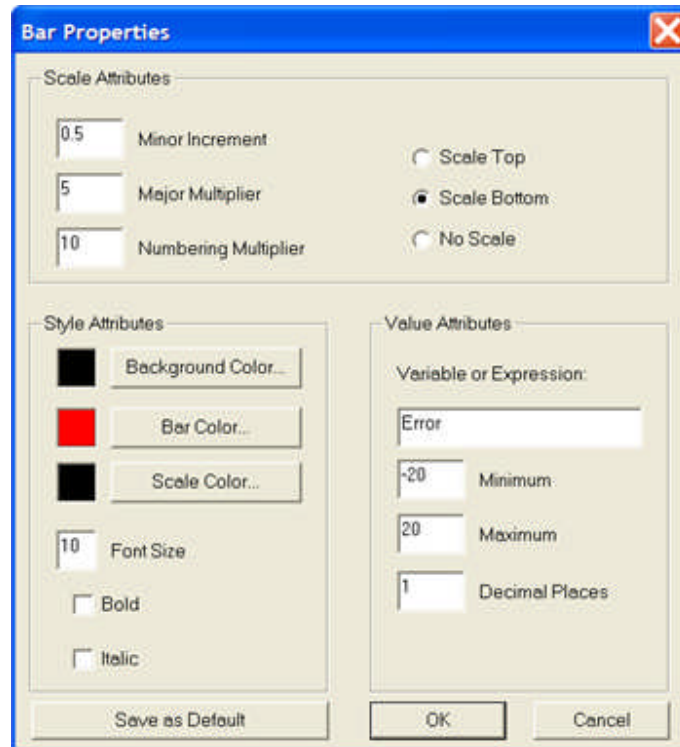
An example for the *Vertical and Horizontal Bar* tools display the value of current and the percent accuracy of an overcurrent ramp to pickup during the ramp process.



The *Vertical bar properties* are for the Amps.



The *Horizontal bar properties* are for the % Error.



This tool may be resized by grabbing a corner with the mouse and moving in or out.

The dialog has Scale Attributes, Style Attributes block for Tool Color and Font selections and Value Attributes block.

Scale Attributes box provides for:

- Minor Increment** is to enter the number for the spacing between marks on the scale.
- Major Increment** is to enter the number for a larger mark at the entered spacing.
- Numbering Multiplier** is to enter the number multiple to be displayed on the scale.
- Scale Left** check button is to select the scale to be displayed only on the left of the bar, or
- Scale Right** check button is to select the scale to be displayed only on the right of the bar, or
- No Scale** check button is to select that no scale be represented by the bar.

Style Attributes box provides for:

- Background Color...** button evokes a color palate for color selection of the inactive or non-encroached area of the bar.
- Bar Color...** button evokes a color palate for color selection of the active moving bar field.
- Scale Color...** button evokes a color palate for color selection scale Font color.
- Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude field.
- Bold** is whether to place the amplitude numbers bold or not.
- Italic** is whether to place the amplitude numbers in italics or not.

Value Attributes box provides for:

- Variable or Expression** edit field for a variable name or an expression such as a formula linked to known test quantities.
- Minimum** number edit field to enter the minimum numeric value on the scale.
- Maximum** number edit field to enter the maximum numeric value on the scale.

**Decimal Places** number edit field to enter the number of decimal places to display on the scale.  
**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.



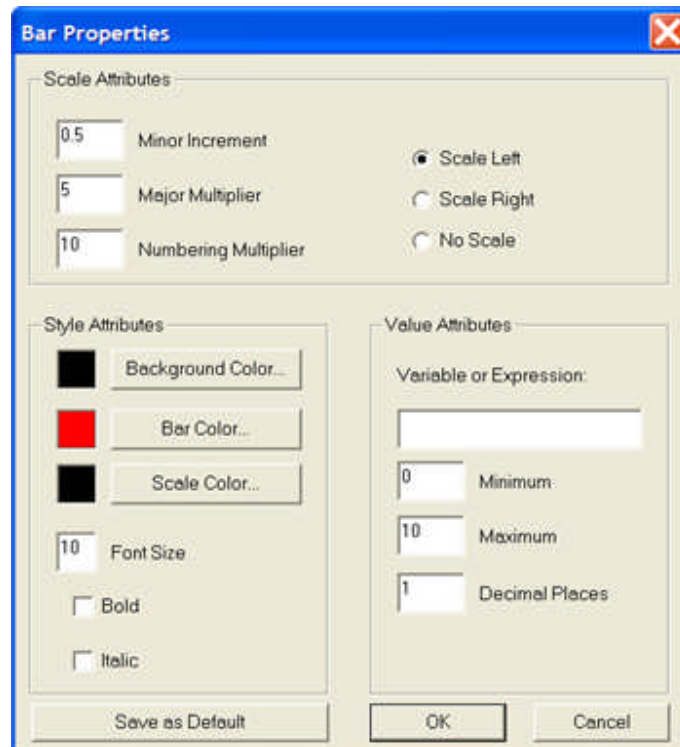
## Horizontal Bar Tool

The **Horizontal Bar Tool** is to provide an active moving display for a monitored variable.

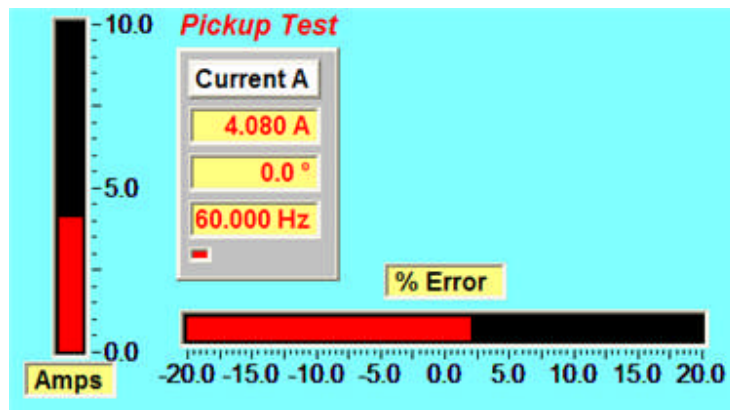
The *Horizontal Bar Tool*



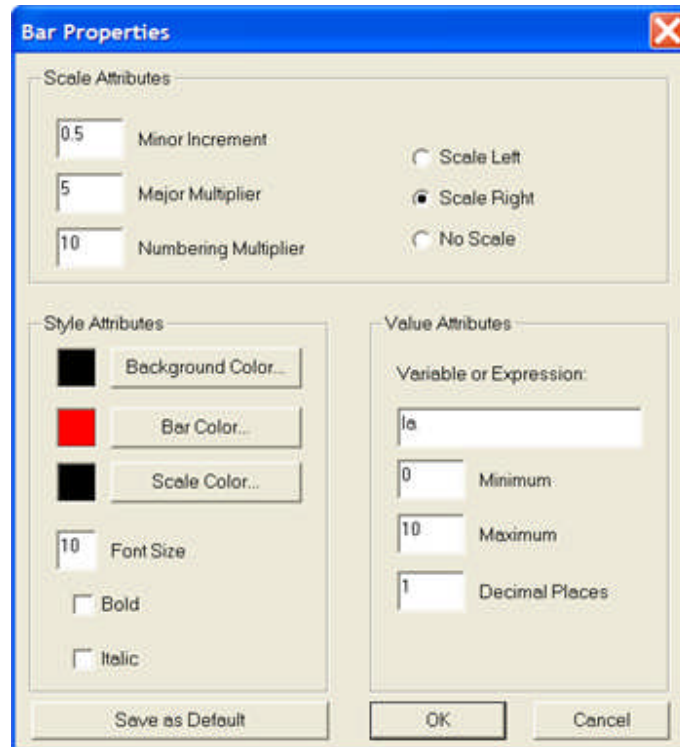
Right mouse click or double click on the tool to evoke the *Bar Properties* dialog.



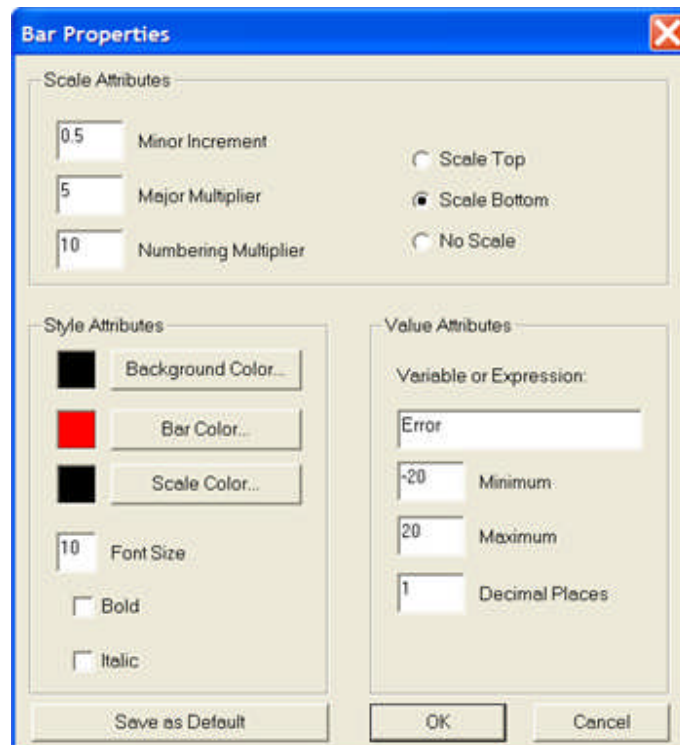
An example for the *Vertical and Horizontal Bar* tools display the value of current and the percent accuracy of an overcurrent ramp to pickup during the ramp process.



The *Vertical bar properties* are for the Amps.



The *Horizontal bar properties* are for the % Error.



This tool may be resized by grabbing a corner with the mouse and moving in or out.

The dialog has Scale Attributes, Style Attributes block for Tool Color and Font selections and Value Attributes block.

Scale Attributes box provides for:

**Minor Increment** is to enter the number for the spacing between marks on the scale.

**Major Increment** is to enter the number for a larger mark at the entered spacing.

**Numbering Multiplier** is to enter the number multiple to be displayed on the scale.

**Scale Left** check button is to select the scale to be displayed only on the left of the bar, or

**Scale Right** check button is to select the scale to be displayed only on the right of the bar, or

**No Scale** check button is to select that no scale be represented by the bar.

Style Attributes box provides for:

**Background Color...** button evokes a color palate for color selection of the inactive or non-encroached area of the bar.

**Bar Color...** button evokes a color palate for color selection of the active moving bar field.

**Scale Color...** button evokes a color palate for color selection scale Font color.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude field.

**Bold** is whether to place the amplitude numbers bold or not.

**Italic** is whether to place the amplitude numbers in italics or not.

Value Attributes box provides for:

**Variable or Expression** edit field for a variable name or an expression such as a formula linked to known test quantities.

**Minimum** number edit field to enter the minimum numeric value on the scale.

**Maximum** number edit field to enter the maximum numeric value on the scale.

**Decimal Places** number edit field to enter the number of decimal places to display on the scale.

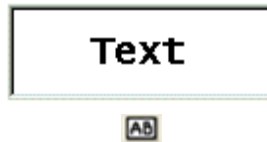
**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

## Text Box Tool

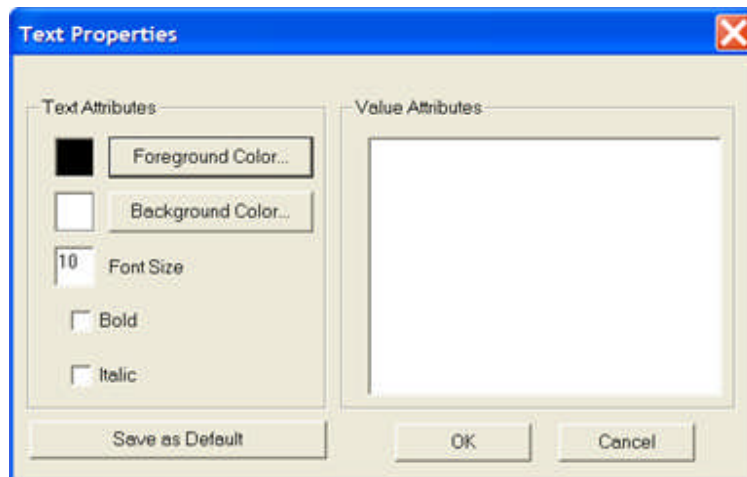
The **Text Box Tool** is to place text in the upper Monitor pane in the Workbook Test tab.

The *Text Box* Tool



This tool may be resized by grabbing a corner with the mouse and moving in or out.

Right mouse click or double click on the tool to evoke the *Text Properties* dialog. The dialog has Style Attributes block for Tool Color and Font selections and a Value Attributes block to enter text.



Text Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the text entered.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude field.

**Bold** is whether to place the amplitude numbers bold or not.

**Italic** is whether to place the amplitude numbers in italics or not.

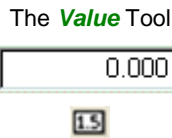
**Value Attributes box** is to enter text to be placed in a box for display. The box may be sized, to accommodate as much text as entered, by dragging one of the corners in the Monitor Editor tab.

**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

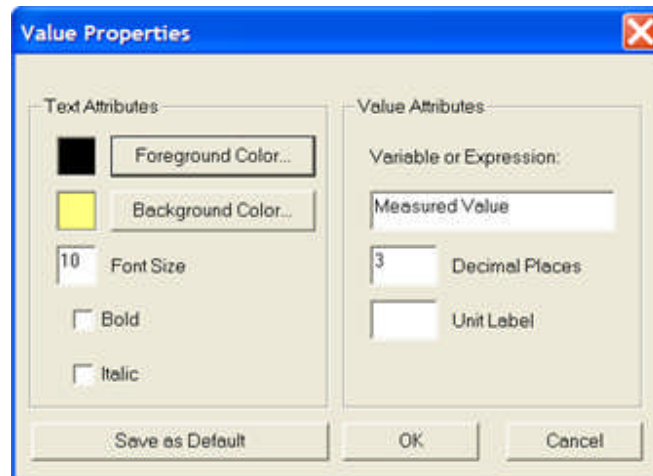
## Value Tool

The **Value Tool** is to provide an active display for a monitored variable.



This tool may be resized by grabbing a corner with the mouse and moving in or out.

Right mouse click or double click on the tool to evoke the **Value Properties** dialog. The dialog has Text Attributes block for Tool Color and Font selections and Value Attributes block.



Text Attributes box provides for:

**Foreground Color...** button evokes a color palate for color selection of the text entered.

**Background Color...** button evokes a color palate for color selection of the field behind the Foreground.

**Font Size** edit field is to place a numeric value for the number sizes to appear in the amplitude field.

**Bold** is whether to place the amplitude numbers bold or not.

**Italic** is whether to place the amplitude numbers in italics or not.

Value Attributes box provides for:

**Variable or Expression** edit field for a variable name or an expression such as a formula linked to known test quantities.

**Decimal Places** number edit field to enter the number of decimal places to display on the scale.

**Unit Label** edit field to enter a label for the Variable or Expression value. If the Expression were  $Z = V_a/I_a$ , then Ohms may be entered. The display will appear in the Test tab monitor pane.

**Save as Default** option will retain the selection to the database for all new tests created.

**Note:** The Save as Default will not affect previously created tests.

## **Connections Editor Tab**

### **Connections Editor Tab**

The **Connections Editor Tab** contains a toolbar to place tools into the Workbook pane which will reflect the connections between the test equipment and relay to be tested.

The tools for creating the mimics are displayed in the Connections Editor Toolbar at the right hand side of the Workbook pane Connections Editor Tab.

Within this tab Connections Editor Tab drop menu is available for ready edit of the tools which are in place in the tab.

Each available tool is defined in the Connections Editor Toolbar topic.

## Connections Editor Tab Menu

The **Monitor Editor Tab** contains a toolbar to place tools into the Workbook pane which will reflect the real time actions of the test set amplifiers, external inputs, external outputs, and any variables properly defined.

The tools for creating the mimics are displayed in the Monitor Editor Toolbar at the right hand side of the Workbook pane Monitor Editor Tab.

Within this tab Monitor Editor Tab drop menu is available for ready edit of controls and links which are in place in the tab.

Each available control is defined in the Connection Editor Toolbar topic.

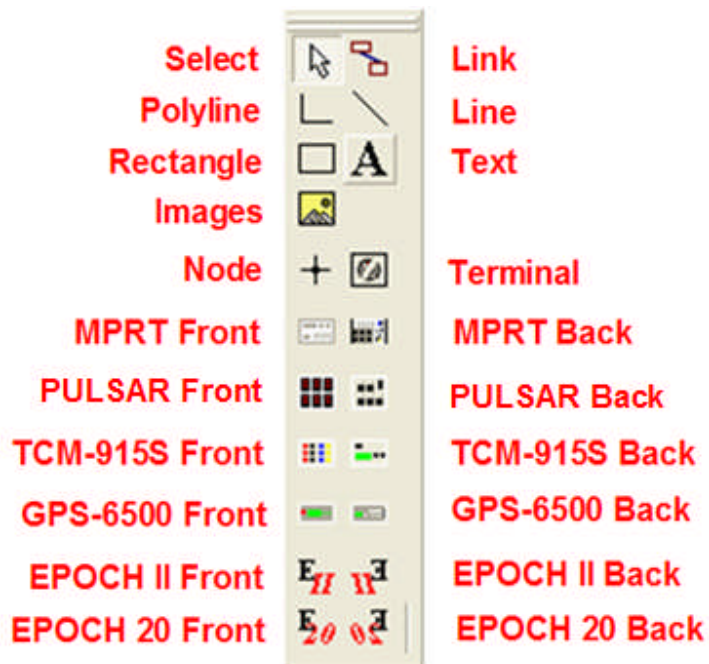
## Connections Editor Toolbar

### Connections Editor Toolbar

The Connections Editor tab **Controls tool bar** resides to the right hand side of the open Relay Workbook. Each is a program application for placing in the Workbook and linked to create a set of test connections.

Each Control is defined and exemplified in its respective topic.

Select, Link, Polyline, Line, Rectangle, Text, Images, Node, Terminal, MPRT Front, MPRT Back, PULSAR Front, PULSAR Back, TCM-915S Front, TCM-915S Back, GPS-6500 Front, GPS-6500 Back, EPOCH II Front, EPOCH II Back, EPOCH 20 Front, EPOCH 20 Back

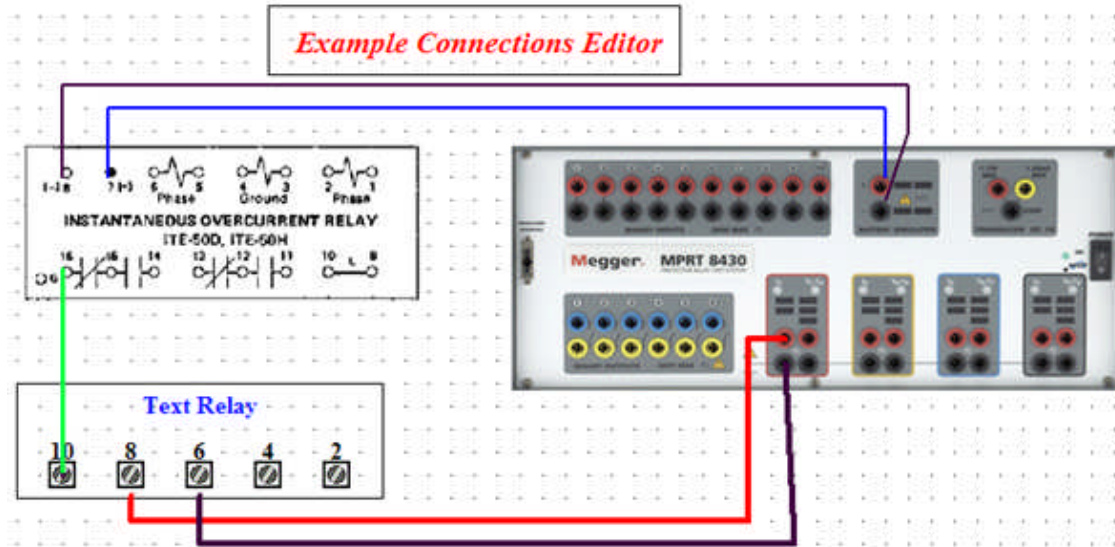




## Connections Editor Example

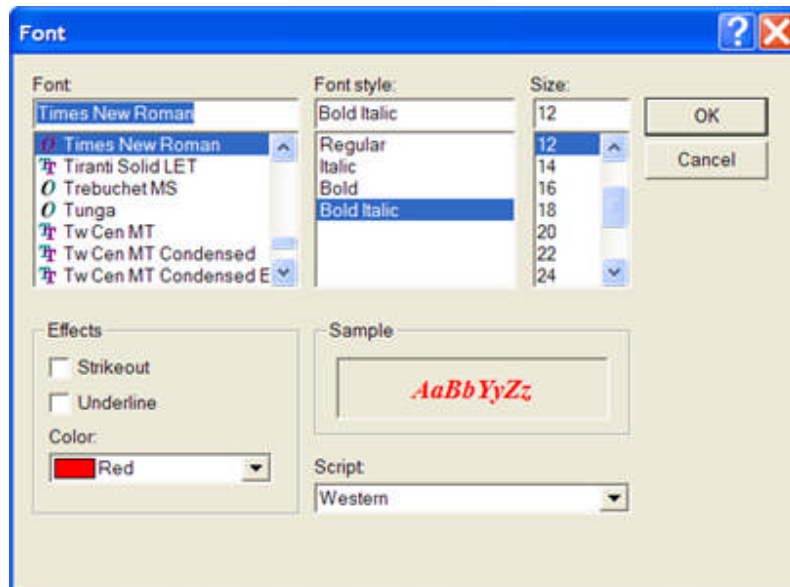
The Connections Editor tab **Controls tool bar** resides to the right hand side of the open Relay Workbook. Each is a program application for placing in the Workbook and linked to create a set of test connections.

The **Connections Editor Example** displays a sampling of several items from the toolbar connected to represent a fictitious set of test connections. The example tools are described below.



The top "Example Connections Editor" is produced using the Text tool.

The Text has been edited using a right click on the text and using the *Font edit dialog*.



The Rectangle surrounding the Text is produced using the Rectangle tool.

The ITE relay Image is a bitmap created in Paintbrush.

The MPRT Front tool created the MPRT "photo" on the right.

**Note:** All tools of the Megger Test Equipment are utilized the same as the MPRT and will not be exemplified.

The Polyline tool created the two thin black and blue lines connecting the terminals of the ITE relay and the MPRT.

The Test Relay text and Rectangle surround the relay terminals created using the Terminal tool. The Terminal Text is derived from the Terminal Tool.

The Line tool created the light green line connecting the ITE relay to the Text Relay.

The Link tool created the Red and Black lines connecting the Text Relay to the MPRT Front.

The Node tool is treated the same as the Terminal tool.

## Select

The **Select Tool** has no properties. It has the actually the mouse cursor and is enabled by "picking" with the mouse a Tool from the Toolbar. It is also used to select one or more controls for moving within the Workbook.

The Select Tool



## Link

The **Link tool** is for connecting terminals emulating wires. The wiring represented is to replicate that which is necessary between test equipment and devices under test.

The Link Tool

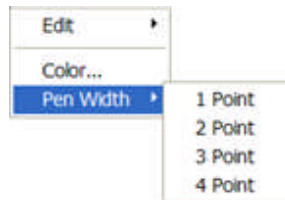


The Link tool can only be used between the Terminal tool and any of the other tools on the tool bar which have terminals.

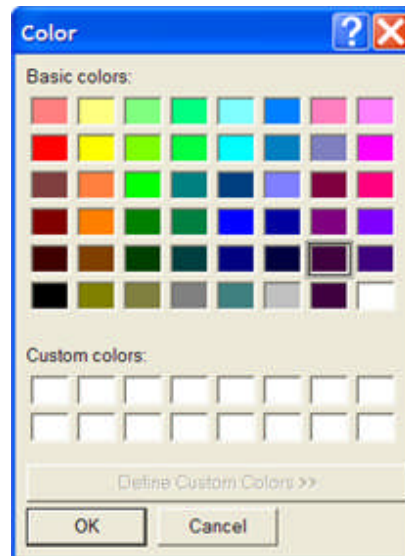
**Note:** The Link tool will anchor to these terminals and move with the object as it is moved.

**Note:** To represent connections between imported Images, Toolbar tools and other Images, it will be necessary to use the **Line** or the **Polyline** tool.

Right mouse click on the Link to display the **drop menu** to change the Link color and line thickness.



Select the Color option to display the **Color** palette for choosing the Link color.



## Polyline

The **Polyline tool** is used to create multiple connected lines.

The Polyline Tool



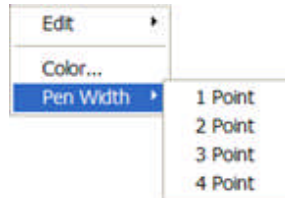
It is created by selecting the tool and using a single left mouse click at the beginning of the line in the Workbook. Move the mouse and provide another left mouse click to end the first straight line. Move the mouse and provide another left mouse click to end the second straight line, which is anchored. Continue this process until the "connected lines" are terminated on two terminals of Images and/of toolbar tools. For the last "leg" of the connection, double left mouse click to end the process.

Each leg is represented with a grab block at the end or the intersection of two lines. Grab the block and that segment of the multiple lines may be moved in the Workbook.

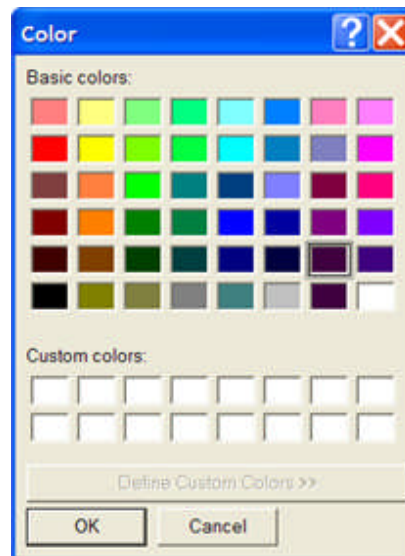
**Note:** The Polyline created between any two Images or devices from the toolbar will not be anchored to the terminals and will not move with the Image or Tool.

**Note:** If connecting two devices from the toolbar, but not an Image, use the Link tool.

Right mouse click on the Polyline to display the *drop menu* to change the Polyline color and line thickness.



Select the Color option to display the *Color* palette for choosing the Polyline color.



## Line

The **Line tool** is used to "draw" a line dragged with the mouse from one location to another. Select the tool, left click in the Workbook, hold the left mouse key and drag a line. When the second location is reached, release the mouse key to place the desired line.

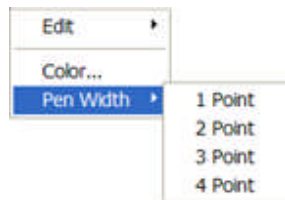


A line drawn will be active when the second point is produced and have blocks on each end. The blocks are to grab and move the line end to another location. Place the cursor on mid-line to grab and move the entire line without changing the line angle to horizontal.

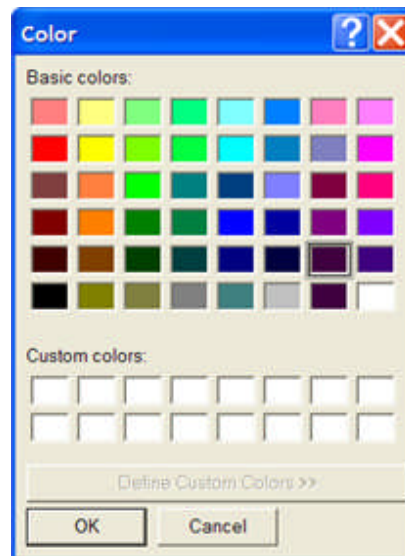
**Note:** The Line created between any two Images or devices from the toolbar will not be anchored to the terminals and will not move with the Image or Tool.

**Note:** If connecting two devices from the toolbar, but not an Image, use the Link tool.

Right mouse click on the Line to display the *drop menu* to change the Line color and line thickness.



Select the Color option to display the *Color* palate for choosing the Line color.



## Rectangle

The **Rectangle tool** is to draw a rectangle without having to draw four separate lines or using the Polyline tool. The rectangle may be used to enclose several items in a box.

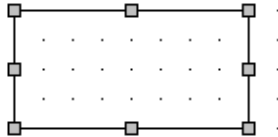
It is used to "draw" a box dragged with the mouse from one location to another. Select the tool, left click in the Workbook, hold the left mouse key and drag a rectangle. When the second location is reached, release the mouse key to place the desired rectangle.

Rectangle Tool



This tool may be resized by grabbing a corner with the mouse and moving in or out.

A rectangle drawn will be active when the second point is produced and have blocks on each corner and mid-sides. The blocks are to grab and change size and location of the rectangle. Place the cursor in the rectangle to grab and move it without changing the line angle to horizontal.



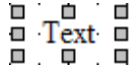
## Text

The **Text tool** is used to enter formatted text in desired areas of the Connections Editor tab to further detail the test apparatus.

The Text tool

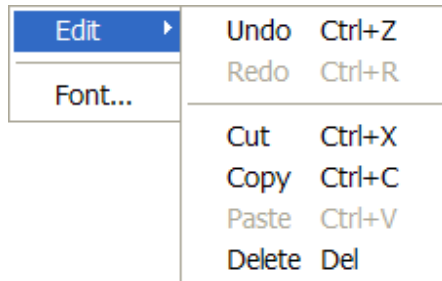


The tool is shown with the eight squares indicating that it has been placed in the Workbook pane for use.

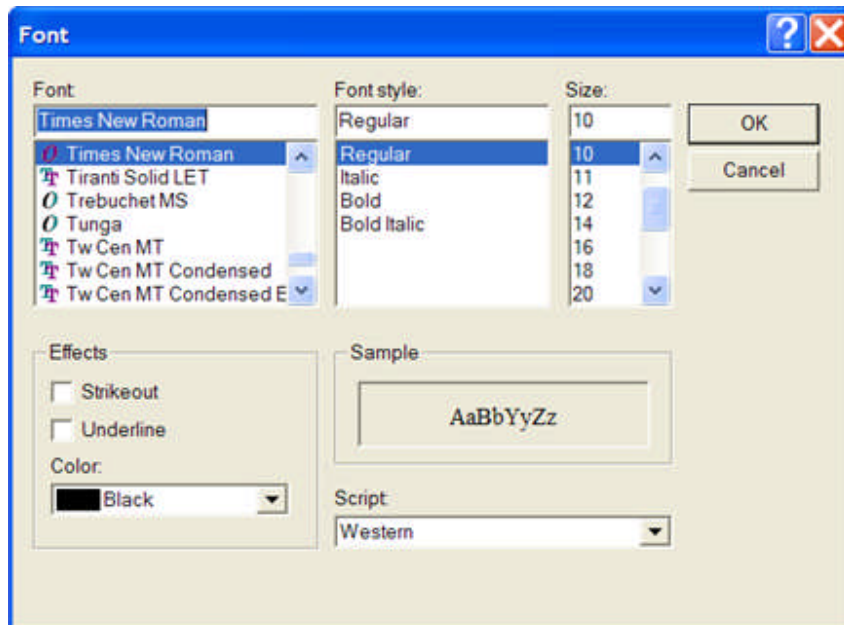


This tool may be resized by grabbing a corner with the mouse and moving in or out.

The desired Text entry is created by double left mouse click, highlighting the text and then entering text with use of the keyboard. Or, right mouse click on the Tool to produce the *Text menu* and select the Font option. The Edit options are the standard Windows options. Using the Edit options text may be Cut/Paste elsewhere, Copy/Paste for repetition and Delete.



Font option opens the standard Windows *Font dialog* for selecting the desired font appearance. Note: If "formatted" text is present, it is usually more efficient to copy the existing text, pasting the copied text and then edit the text with a double left mouse click. The action retains the format of the copied text.



**Overcurrent Pickup Test** is an example of formatted text giving additional information about the test.



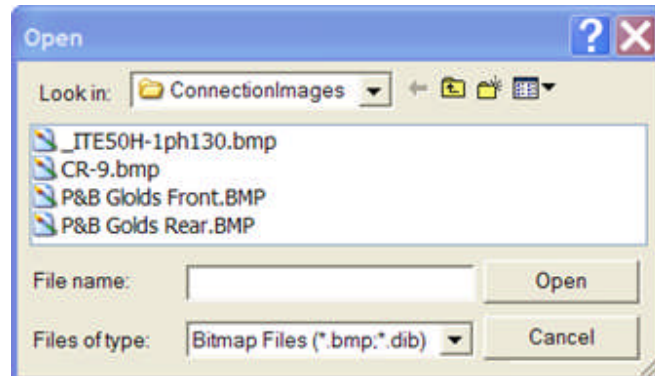
## Images

The **Images tool** is used to place an image or relay with terminals, which does not exist in the Toolbar, in the Connections Editor tab. Associated with the Images tool is the Connection Images Toolbar.

The Images Tool



The Images tool opens the *Open dialog* to the ConnectionImages subfolder of AVTS. Images to be used may be created in Microsoft Paintbrush, scanned or photographed digitally and saved as a bitmap, \*.bmp. They may be called and used as part of the connections to test equipment or other devices representing test interconnections.



**Note:** The **Link** tool will not function as a termination to a bitmap terminal.

## Node

The **Node tool** is to provide a terminal common to several parallel connections.

The Node Tool



The Node tool allows an anchor for the Link tool and may be used with the Line and Polyline tools without anchor. The Node may not exist as a real terminal and may be useful only in the Connections Editor tab.

**Note:** Use one of the device terminals to anchor several parallel connections in lieu of the Node to reduce the number of total terminals used for connections.

## Terminal

The **Terminal tool** may be used in multiple to represent the terminals of a relay or device.

The Terminal Tool



Select the Terminal tool with the mouse. Each left mouse click in the Connections Editor tab will produce a terminal. The terminals may be selected independently or as a group to relocate.

There are two methods to select items as a group:

- Hold the Ctrl key while left mouse clicking on each item to be relocated as a group.

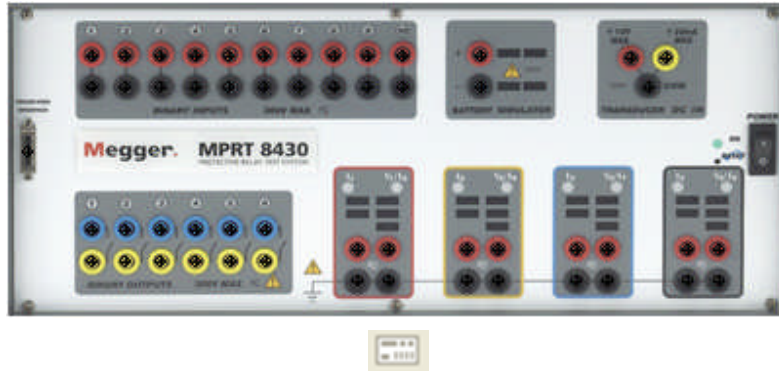
- Hold the left mouse key while dragging a "box" to enclose all the items to be relocated as a group.

Double left click on the Terminal to produce an anchored Text block for the Terminal. Treat the Text the same as the Text Tool.

## MPRT Front

The **MPRT Front tool** is to place in the Connections Editor tab a picture of the front of the MPRT test system. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

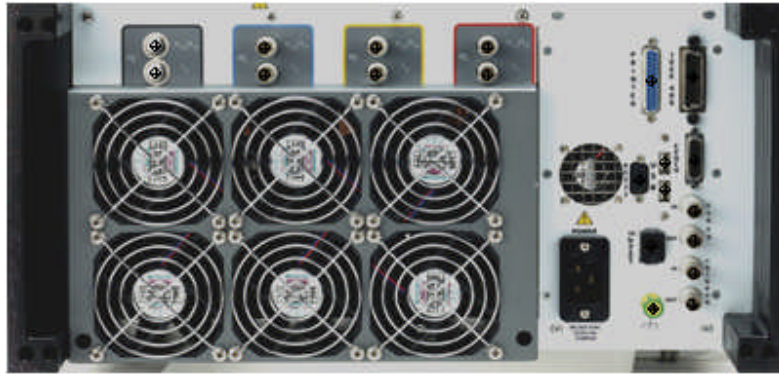
The *MPRT Front* Tool



## MPRT Back

The **MPRT Back tool** is to place in the Connections Editor tab a picture of the back of the MPRT test system. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

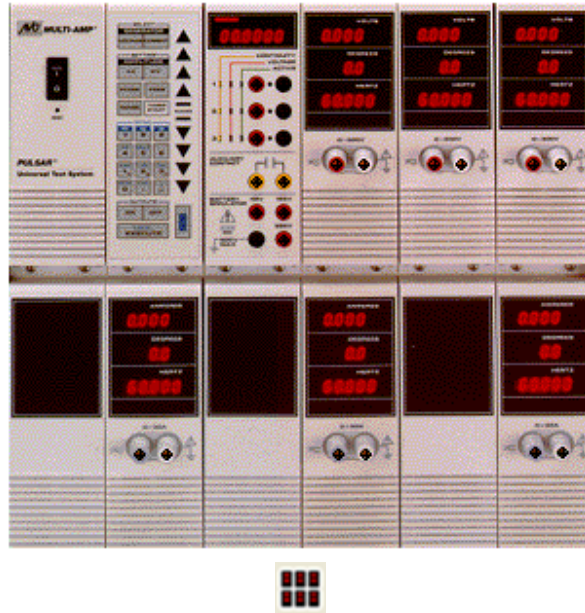
The *MPRT Back* Tool



## PULSAR Front

The **PULSAR Front tool** is to place in the Connections Editor tab a picture of the front of the PULSAR test system. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

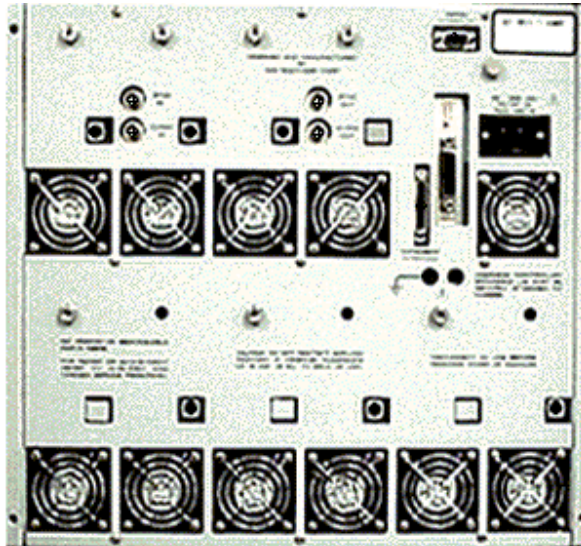
The *PULSAR Front* Tool



## PULSAR Back

The **PULSAR Back tool** is to place in the Connections Editor tab a picture of the back of the PULSAR test system. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

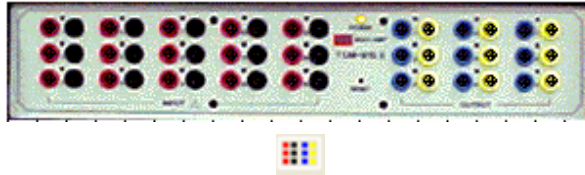
The *PULSAR Back* Tool



## TCM-915S Front

The **TCM-915S Front tool** is to place in the Connections Editor tab a picture of the front of the TCM-915S Timer Control Monitor. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

The *TCM-915S Front* Tool

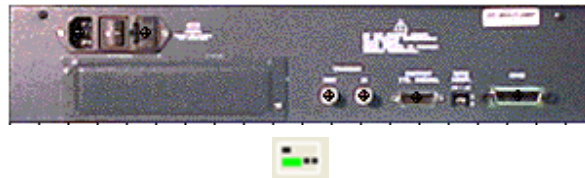




## TCM-915S Back

The **TCM-915S Back tool** is to place in the Connections Editor tab a picture of the back of the TCM-915S Timer Control Monitor. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

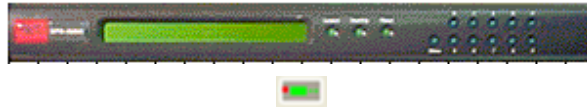
The *TCM-915S Back* Tool



## GPS-6500 Front

The **GPS-6500 Front tool** is to place in the Connections Editor tab a picture of the front of the GPS-6500 Global Position Satellite receiver. There are not termination connections present on this device.

The *GPS-6500 Front* Tool



## GPS-6500 Back

The **GPS-6500 Back tool** is to place in the Connections Editor tab a picture of the back of the GPS-6500 Global Position Satellite receiver. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

The *GPS-6500 Back* Tool



## **MGTR Front**

This Topic is under edit and will be complete in the next release of AVTS Help

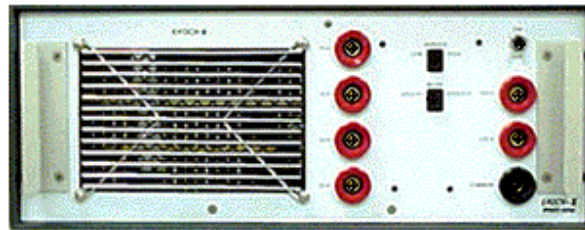
## **MGTR Back**

This Topic is under edit and will be complete in the next release of AVTS Help

## EPOCH II Front

The **EPOCH II Front tool** is to place in the Connections Editor tab a picture of the front of the EPOCH II High Current Interface. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

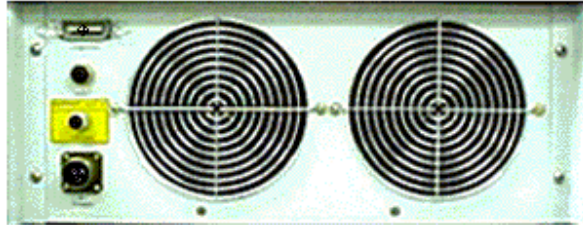
The *EPOCH II Front* Tool



## EPOCH II Back

The **EPOCH II Back tool** is to place in the Connections Editor tab a picture of the back of the EPOCH II High Current Interface. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

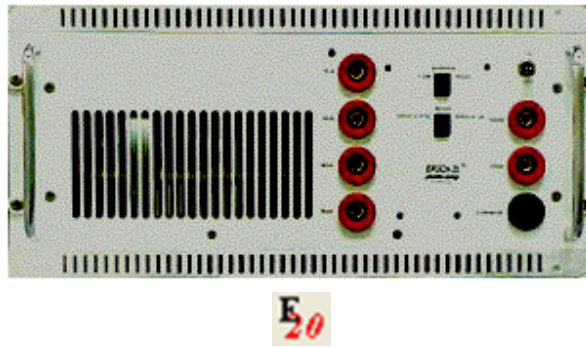
The *EPOCH II Back* Tool



## EPOCH 20 Front

The **EPOCH 20 Front tool** is to place in the Connections Editor tab a picture of the front of the EPOCH 20 High Current Interface. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

The *EPOCH 20 Front* Tool

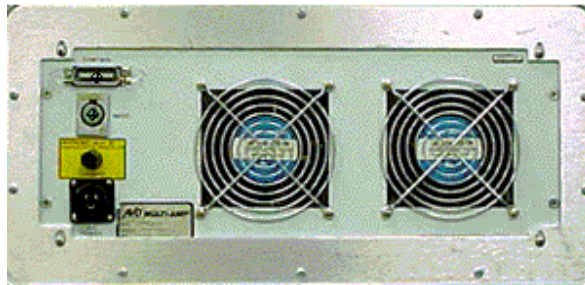




## EPOCH 20 Back

The **EPOCH 20 Back tool** is to place in the Connections Editor tab a picture of the back of the EPOCH 20 High Current Interface. The terminals for interconnect contain hot-spots to provide an anchor for the Link tool.

The *EPOCH 20 Back* Tool



## **BASIC Tab**

The **BASIC tab** is enable by the BASIC Control or the 1-Touch Control when either or both are placed in the Test Editor tab as part of a test procedure.

This tab is the location to place information to control the test procedure whenever an existing Test Editor Control is not able to conduct the desired action.

Perhaps an existing Control or group of Controls are able to provide required action for a test procedure, but are complex in construction. Using BASIC Language Syntax may be simpler and easier to generate to perform the same function

PulseMaster macro files are used when a PulseMaster test module has been converted and Imported into the active AVTS database.

PulseMaster macro files are used as a part of the AVTS constructed test procedure and enabled by the BASIC Control or 1-Touch Control.

Tables constructed using BASIC Language Syntax as demonstrated in the Test Editor Differential Example 4 topic.

Visual BASIC language syntax used to communicate with connected devices under test, as demonstrated in the 1-Touch Control - Example 2 topic.

The PulseMaster relay modules and Visual Basic Script Help are resident in folders on the AVTS installation Disk.

## Relay General Information Tab

The *General Information* tab for an open relay is for entering information.

Relay: **The Relay label in the Workspace Relay tab.**

CR-9

Device: **The Device label in the Workspace Relay tab.**

CR-9 (Westinghouse)

Maintenance Period:  Months

Primary Settings Group:

Barcode ID:

Relay Notes:

Relay Notes may be copied from relay manuals, other sources or typed in via keyboard. The Notes are visible for every Test selected.

Test Notes:

Test Notes are similar to Relay Notes. Each Test selected may have different Test Notes. For Similar Tests, the Notes may be copied from another Test, or typed using the keyboard.

Execution Notes:

Execution Notes are similar to Relay Notes. Each Test selected may have different Execution Notes. For Similar Tests, the Notes may be copied from another Test, or typed using the keyboard.

Navigation:

When open, the Relay name and the parent Device name are displayed for reference. The purpose is to readily identify a selected relay when there are several relays open in the Workbook and the Workspace has been hidden.

**Maintenance Period** edit field allows entry of a period of time of months that determines the schedule for the next test. An entry of months in the field and then saving the relay saves the period in the database. A Report may then be generated to determine the next relays due test at a given time in the future. Another Report may be generated to determine which relays routine maintenance has been missed.

**Primary Settings Group** entry field is governed by the Group Names displayed in the Settings tab, including a group name "None". Using the drop down arrow, the groups for the open relay are displayed and one may be selected for entry.

**Note:** The Primary Settings Group selected in the Start Control in the Test Editor tab takes precedence over the group name selected in the General Information tab.

Note: For a test execution to assume the Primary Settings Group selected in this field, **None** must be selected in the Primary Group Setting in the **Start Control** in the Test Editor tab.

**Barcode ID** edit field is to manually enter a barcode for the open relay and saved to the database. Select the Barcode button to open the Barcode ID dialog to scan a barcode with a barcode scanner for the relay and saving to the database.

**Relay Notes** edit field is to enter any notes the user would like to save with the relay. These notes may have to do with specific uses or settings for the relay. This data is displayed regardless of which relay test is selected in the Workspace. Information which can be copied into Microsoft clipboard, it can be pasted into this field. Relay Notes will be printed with the printed test results data.

**Test Notes** edit field is to enter any notes specific to any test for the relay. Notes entered for one test will not appear for another selected test. Each available test may have separate Test Notes. Information which can be copied into Microsoft clipboard, it can be pasted into this field. Test Notes will not be printed with the printed test results data.

**Execution Notes** edit field is to enter any notes specific to any test execution. Notes entered for one test will not appear for another selected test. Each available test may have separate Execution Notes. Information which can be copied into Microsoft clipboard, it can be pasted into this field. Execution Notes will not be printed with the printed test results data.

## Online Controls

### Online Controls

The **Online Controls**, launched from the AVTS Tools menu item, allow direct real time control of the connected test set. The Controls are enabled by selecting the one desired from the Tools\Online *controlname* Control... list, which evokes the selected Online Control.

The Online Controls are for specific test types to execute without having to create specific tests procedures for relays. The Online Vector, Online Ramp and Online Click On Fault Controls allow any of the settings created for a specific test or sets of test to be saved to a file name .

The Online controls are selected from the menu items Tools/*Online controlname*.

Online Vector Control...

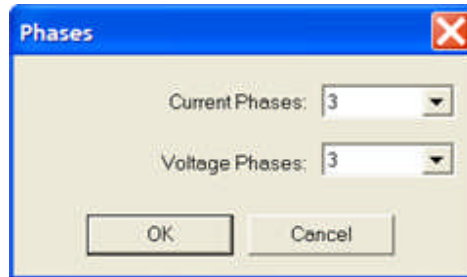
Online Ramp Control...

Online Click On Fault Control

## Online Vector Control...

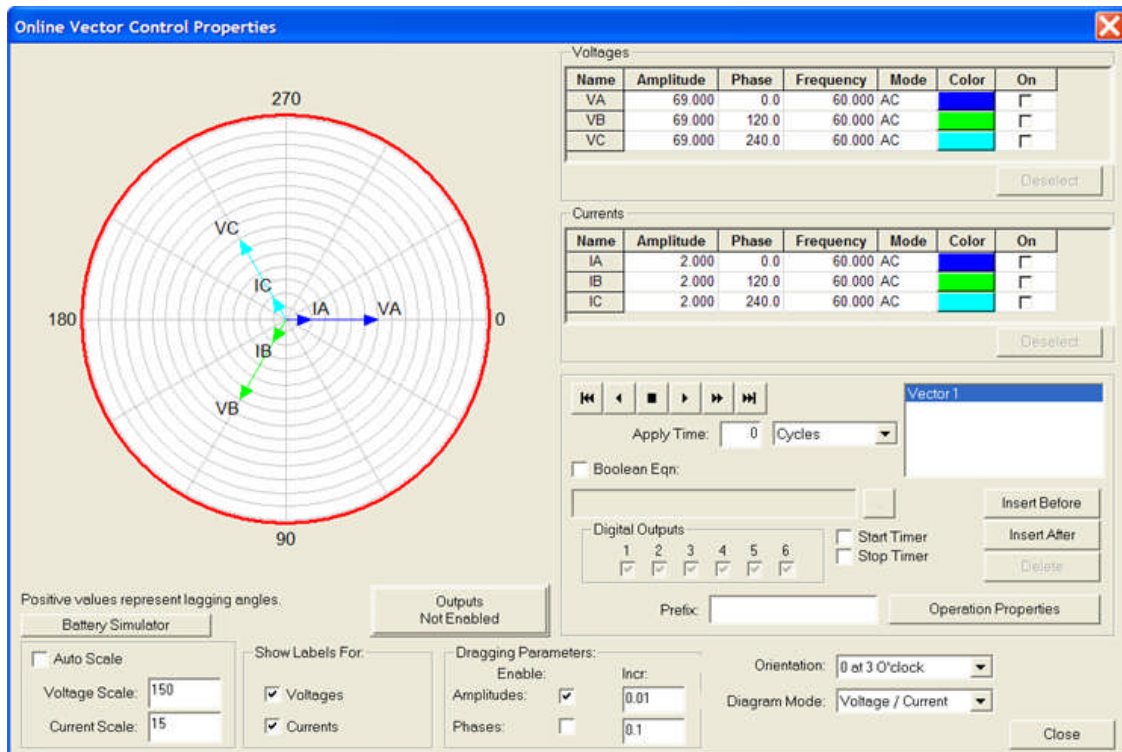
## Online Vector Control...

The **Online Vector Control**, launched from the AVTS Tools menu item, allows direct real time control of the connected test set. The Control is enabled by selecting Tools\Online Vector Control... which evokes the *Phases dialog* for pre-programming the Control for the total number of current and voltage channels to be active.



Up to sixteen vector states may be created and played back through the test set. A timer control is available to enable starting the test set timer at the execution of any one of the vector states. The timer stop is typically controlled by an action, from one of the outputs of a connected device, to an Input used as the timer stop gate. The default view of the Online Vector Control remains visible during all uses of the control.

The default *OneLine Vector Control Properties* Dialog displays a number of areas for parameter entries and control functions.



For manual ramping of amplitudes and phases, a gang control is available through the selection of the vectors to be controlled. The vectors to be controlled in gang are selected with the mouse to grab and alter the vector(s) parameters. Vector selection is made with a left mouse click on a vector channel name to highlight that vector with its parameters. For multiple vectors controlled simultaneously, left mouse click on the wanted vectors while holding the keyboard Ctrl key down to highlight all of the desired vectors. Then

select from the "Dragging Parameters" box whether the amplitudes and/or phases of the selected vectors are to be active. Grab the tip of any of the selected vectors in the polar graph and, while holding the left mouse button down, move the vector(s). The values of the vector(s) will change graphically and numerically, and simultaneously pass the new values directly to the corresponding test set amplifiers.

Vector Polar Diagram

Vector Voltages and Currents

Vector List

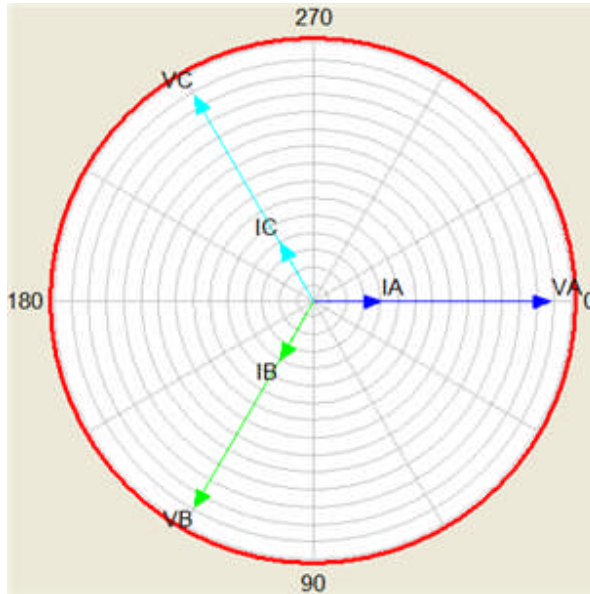
Vector VCR Controls

Vector Input and Output

Boolean Equation

## Vector Polar Diagram

The **Polar Diagram** is for display of the Voltages and Currents vectors whose parameters are initially entered in their respective edit fields. Note: The phase displacement is labeled as "Positive values represent lagging angles." The angles are for 0, 90, 270 and 0 (360) degrees clockwise for lag.



The **Voltage Scale and Current Scale** edit fields become available for numeric entry if the Auto Scale check box is unchecked. The Polar Diagram will zoom, when Auto Scale box is checked, to the maximum Voltages and Currents values as the values increase and decrease.

<input type="checkbox"/> Auto Scale
Voltage Scale: <input type="text" value="150"/>
Current Scale: <input type="text" value="15"/>

The **Show Labels For** area is for enabling display of the Current and/or Voltage phase labels on the polar diagram.

Show Labels For:	
<input checked="" type="checkbox"/> Voltages	
<input checked="" type="checkbox"/> Currents	

The **Dragging Parameters** area is for selecting with the check box whether either Amplitudes or Phases or both are selected and controlled by the dragging of the mouse. The Incr (incremental) value for each of the parameters is determined by entry in the appropriate edit field. The values to enter must be in the range of 0.001 to 100.

Dragging Parameters:	
Enable:	Incr:
Amplitudes: <input checked="" type="checkbox"/>	<input type="text" value="0.01"/>
Phases: <input type="checkbox"/>	<input type="text" value="0.1"/>

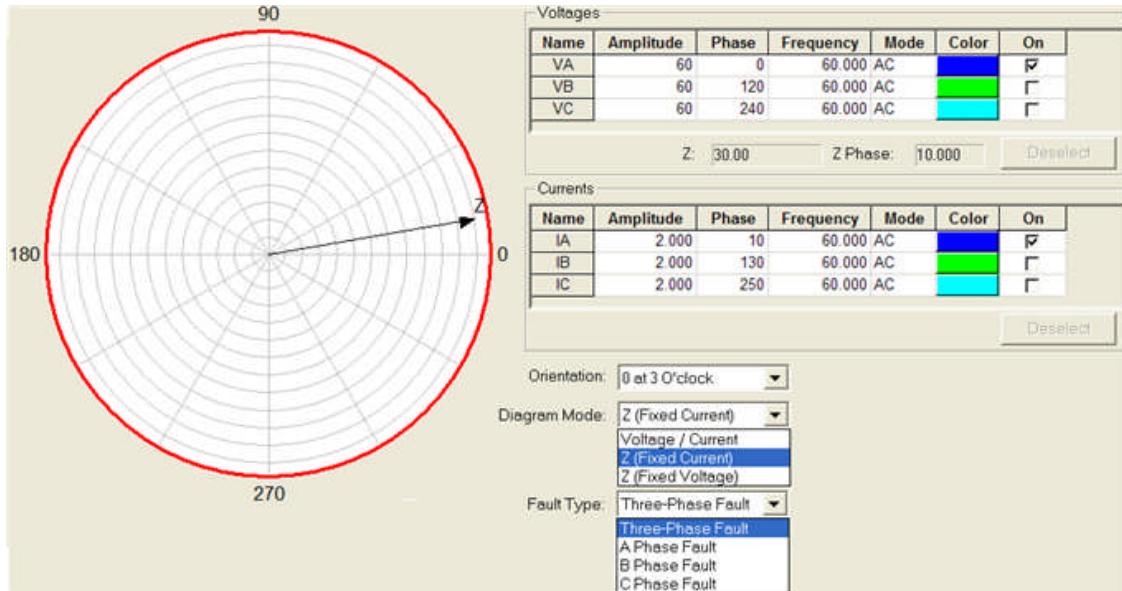
The Polar Diagram Orientation drop down list allows the user to option the location of the 0 degree reference to 3 o'clock (default), 6 o'clock, 9 o'clock, or 12 o'clock.

The Polar Diagram Mode drop down list allows the user to option the phasor modes to Voltage/Current (default), Z (Fixed Current) or Z (Fixed Voltage).

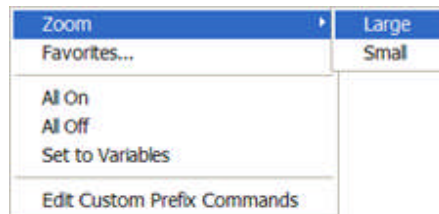


Select Z (Fixed Current) or Z (Fixed Voltage) and a Fault Type drop down list is visible.

The **Polar Diagram now** displays a single vector Z and 0, 90, 270 and 0 (360) degrees counterclockwise. A value (Fixed Current selected) for Z and angle are displayed under the Voltages parameters fields, based on the voltages and currents magnitudes and angles assigned prior.



The **Drop Down Menu** is placed in view with a right mouse click in the area in and around the polar diagram. The options are:

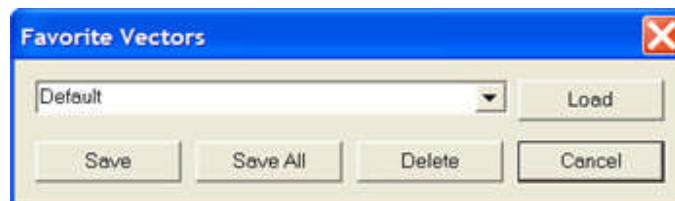


### Zoom

Large option enlarges the Online Vector dialog leaving all sub-boxes normal size.

Small option reduces the Online Vector dialog and all sub-boxes to normal size.

**Favorites...** evokes the **Favorite Vectors** dialog. The Default vector set label in the edit field cannot be deleted. The down arrow displays all other vectors that have been saved in during past use of the Online Vector Control or the Vector Control in the Workbook Editor tab.



The **Load** option places the saved data, in the selected vector set from the drop down list, into the Online Vector Control Properties edit, menu and selections areas of the chosen vector in the Vector list box.

The **Save** option allows the save of a single vector set with all the parameters in the database. The vector set will need a label placed into the edit field for recall at a later use. Save will retain only the selected vector in the vector list. The Edit Customs Prefix Commands are saved along with the generator parameters.

The **Save All** option allows the save of a multiple sets of vectors with all the parameters for each vector set to name entered into the edit field for recall at a later use.

The **Delete** option allows the removal from the Vector Favorites list the vector state selected from the drop list.

The **Cancel** option leaves the Favorite Vectors dialog with a null to any changes during the recent evoke.

**All On** places a check in the On check box for all the voltage and current vectors and turns On all MPRT amplifiers.

**All Off** un-checks all vectors On check box to turn all MPRT amplifiers to off.

**Set to Variables** replaces the numeric values for all the vector parameters to known default variable names. When used in the OnLine Vector Control, the value for the variable will be applied to Pulsar when Outputs are Enabled. The variable values will be those last set equal to a value in the Variable Watch edit field or from a Test Editor tab control of the last executed test. The values will be retained in AVTS until the variable has been set equal to another value. By default, in the Variable Watch window, all variables' Amplitude and Phase are set equal to zero while the variables' Frequency is set equal to 60.

**Note:** This function is commonly used for the Vector control used within a test in the Test Editor tab. The values for the variables are taken from Settings tab, Variable Watch edit field in the Test tab, or in another procedure control prior to the Vector control.

**Edit Custom Prefix Commands** evokes an edit box in the main Online Vector Control dialog for application of additional control to the vector(s) parameters. Various formulas, Pulsar syntax, or other commands may be placed here for application to the Vector prior to the execution of the vector parameters.

## Vector Voltages and Currents

The *Voltages and Currents* fields display the values set for operation of the online control.

Voltages						
Name	Amplitude	Phase	Frequency	Mode	Color	On
VA	69.000	0.0	60.000	AC	<span style="color: blue;">█</span>	<input type="checkbox"/>
VB	69.000	120.0	60.000	AC	<span style="color: green;">█</span>	<input type="checkbox"/>
VC	69.000	240.0	60.000	AC	<span style="color: cyan;">█</span>	<input type="checkbox"/>

Deselect

Currents						
Name	Amplitude	Phase	Frequency	Mode	Color	On
IA	2.000	0.0	60.000	AC	<span style="color: blue;">█</span>	<input type="checkbox"/>
IB	2.000	120.0	60.000	AC	<span style="color: green;">█</span>	<input type="checkbox"/>
IC	2.000	240.0	60.000	AC	<span style="color: cyan;">█</span>	<input type="checkbox"/>

Deselect

The edit fields for the Voltages and Currents are to enter the values of Amplitude, Phase, Frequency, Mode (either AC or DC or Half-Wave), Color (of the vector in the accompanying polar diagram), and On (a check box to turn the Pulsar channel on when Outputs Not Enabled button is clicked to Outputs Enabled).

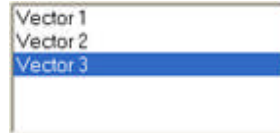
The Voltage and Currents edit fields are to set the parameters for each channel and defined in the Ramp Control Properties topic.

A left mouse click on the Voltage or Current channel in the Name column will select that channel and turn the corresponding vector in the polar diagram Red, indicating that the channel is active. The chosen vector may be dragged with the mouse to change the Amplitude and/or Phase values directly on the polar diagram. Multiple vectors may be selected by holding down the Ctrl keyboard key and clicking on the desired vector names. All of the selected vectors will change to red indicating that a left click and hold of the mouse on any one of the vectors will allow change to the Amplitude and/or Phase values directly on the polar diagram and interactively on Pulsar.

The Deselect button unselects all Voltage or Current edit fields that have been highlighted. When the Deselect button is clicked all vector colors turn from red to the normal color.

## Vector List

The *Vector list* box displays each vector in addition to Vector 1. By default, the list contains one vector, labeled Vector 1, and is also highlighted in BLUE as the selected vector. The label, Vector1, may be renamed by double clicking on the label and typing a new label name. Each state has the full features that are available with the first or default vector.



Insert Before and Insert After enable the addition of up to 16 vector states.

**Insert Before** will enter a new vector state ahead of the vector state highlighted in the list.

**Insert After** will enter a new vector state after the vector state highlighted in the list.

**Delete** button, when clicked, deletes the vector state that is selected.

**Apply Time** edit box allows the numeric entry for the time duration of the selected vector state in either Cycles (default) or Seconds, which ever is selected from the drop down list. The Apply Time is the duration of time that the selected vector state is applied during an Executed Vector Sequence. Should the Timer Start option be selected, the Apply Time will retain the vector parameters for the total Apply Time duration.

**Note:** Apply Time is disabled when the Boolean Eqn: box is checked.

### Example:

Setup for three example vectors as follows:

- Vector 1 contains parameters for load conditions. Set the Operation Properties for Timer Stop on Post 3, Continuity, Applied, Latch, and Turn Currents OFF. Vector 1 has 240 cycles for Apply Time.
- Vector 2 contains parameters perhaps for fault conditions. The Prefix has syntax that changes a parameter or issues some sync syntax, etc. Vector 2 also has the Timer Start box checked and has 180 cycles for Apply Time.
- Vector 3 contains parameters perhaps for Post Fault conditions. Vector 3 has 120 cycles for Apply Time.

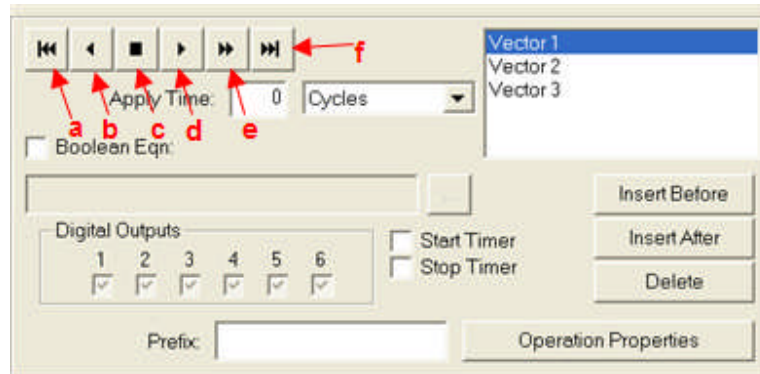
When the Vector 1 is selected and the Execute Vector Sequence is applied, the following will occur:

- Vector 1 will apply all parameters in the generator fields and any syntax placed in the Prefix field. With the lapse of the apply time, the sequence will advance to Vector 2.
- Vector 2 will apply all parameters and syntax present in this vector state. The timer will start and stop time upon received timer input to Post 3. The remainder, if any, of the set Apply Time will lapse and the sequence will advance to Vector 3. Should there be no contact action, the timer will time out with the lapse of the Apply Time (the Pulsar timer will display a time that is longer than the Apply Time due to the communication time for the change to the next vector state).
- Vector 3 will apply all parameters and syntax present in this vector state. With the lapse of the Apply Time, the sequence will stop execution and turn Pulsar generators off.

**Go to End** brings to focus the very last vector state in the existing list. Therefore, the user may single step FORWARD or REVERSE using the buttons “d” (Move to Next Vector) and “b” (Move to Previous Vector) respectively; or the user may execute a complete vector sequence for an automatic run using the button “e” (Execute Vector Sequence). The sequence will begin with the presently selected vector state and (continue automatically to the last vector state or the vector state in action when the Stop Sequence Execution is executed.

## Vector VCR Controls

The “VCR” buttons enable the user to control the application of the listed Vector states, when more than one is in the Vector list.



Each button use is defined from left to right as:

- Go to Start** returns focus to the first vector state from any other vector state.
- Move to Previous Vector** returns focus to the vector state just prior to the displayed vector state. Should the Outputs Enabled button be on and there is direct communication to the Pulsar, this function will maintain that vector state until another vector is selected by the user or the Pulsar outputs be disabled.

**Note:** If checked, the Timer Start and Digital Outputs check boxes will be executed when that vector state is stepped into. These actions will remain active through any succeeding vectors until the Stop Sequence Execution is exercised. Entries in the Prefix edit box are not passed from one to another vector.

- Stop Sequence Execution** halts the Executed Vector Sequence, Disables all the outputs and Resets the test set to the normal manual turn on state.

**Note:** This button also functions as a Reset to the test set at any time, whether the Outputs are Enabled or Not Enabled. Should the outputs be enabled when a Stop is executed, the test set will reset as described and the Outputs Enabled function will remain “active”.

- Move to Next Vector** brings into focus the vector state just after the displayed vector state. Should the Outputs Enabled button be on and there is direct communication to the test set, this function will maintain that vector state until another vector is selected by the user or the outputs be disabled.

**Note:** When checked, the Timer Start and Digital Outputs check boxes will be executed when that vector state is stepped into. These actions will remain active through any remaining stepping until the Stop Sequence Execution is exercised. Entries in the Prefix edit box are not passed with the selection of any vector set.

- Execute Vector Sequence** begins the automatic sequence of the entire list of vector states beginning with the vector state in focus when the sequence is executed.

**Note:** All entries in the Prefix edit box and the Timer Start and Digital Outputs check boxes will all execute for each vector in turn in sequence.

### For PULSAR

Should the Pulsar gate 3 (monitor gate) be assigned as the timer stop gate in the Operation Properties dialog, the assignment will respond only with the first vector set that is assigned a timer start function, whether the assignment is from the Timer Start check box or a Pulsar syntax placed in the Prefix edit field to start the timer.

**Note:** Any MPRT input may be a timer stop gate.

Any succeeding vector sets that start the timer will require stop from the Pulsar gate 2 (the stop gate), unless the use of Pulsar gate 3 is assigned the stop function by syntax entry in the Prefix edit field after the timer start function is issued.

**Note:** Any MPRT input may be a timer stop gate.

Should more than one timer record be desired, the Pulsar timer must be issued Reset syntax prior to the next timer start. This action will allow each timer start to begin from timer zero.

**Note:** The MPRT will continue time for the duration of the sequence and record the elapsed time from timer start for each input and output condition.

Should more than one timer record be desired with time accumulating and added to the last record time, the user must not send Reset syntax to the Pulsar Timer.

Should there be no stop gate input to stop the timer, the Pulsar Timer will have the vector set time out and the timer will continue timing through the remainder of the vector states until a timer stop command, or an input to the normal Pulsar gate 2 (the Stop gate) is received, or the end of the automatic sequence has ended.

## Vector Input and Output

**Operation Properties** button, available only when the first Vector is selected, will make visible the **Contact Control Properties**, Figure 2, dialog when clicked. This dialog enables control of the Pulsar Auxiliary Contact and/or the action of the monitored contact or voltage external to the Pulsar. If the Timer Start check box or the Prefix edit field require action from the Pulsar Timer gates, the user must enter the action desired in this dialog.

The Timer Start check box starts the Pulsar timer when the selected vector state is initiated. Should a contact or voltage be present during the Apply Time, the sequence will continue through the remaining vector states with the Pulsar timer continuing the timing mode until the end of the last vector set. The Pulsar timer can time only one event during a vector sequence execution by use of the Timer Start check box. The selection of this check box on any vector set will uncheck the box in any other vector set. Additional control of the timer may be exercised by the use of Pulsar syntax; i.e., should the syntax "TR;" be placed in the Prefix edit field of any succeeding vector set, the timer will reset without affecting the automated vector sequence.

**Note:** Should more than one vector set have either the Timer Start checked or Pulsar syntax placed in the Prefix edit field to start the timer, multiple times may be captured during the Execute Vector Sequence option.

For Pulsar, Digital Output check box 3 will set the command to close the Auxiliary Contact. The selection of this check box on any vector set will uncheck the box in any other vector set. Additional control of the Contact can be accomplished by placing Pulsar Timer Module syntax in the Prefix edit field; i.e., TAD; will disable or open the Contact should it have been closed by the check box in a previous vector set.

**Note:** The Pulsar Auxiliary Contact will remain in the last state of control set by the vector sequence until it receives another command or the Stop Sequence Execution action is exercised.

The Prefix edit field is made visible by selection of the "Edit Custom Prefix Commands" from a drop down list box. Once selected, it is available for each vector state and allows entry of a formula, Pulsar syntax, or other controlling variable for that selected vector state. Each vector state may have different or no entries in this edit field.

Note: When the Outputs Not Enabled button or Outputs Enabled button is visible and a Move to Previous Vector or Move to Next Vector is selected, the Prefix commands to Pulsar are ignored. The Prefix entries are recognized only with the selection of the Execute Vector Sequence.

Edit Custom Prefix Commands evokes an edit box in the main Online Vector Control dialog for application of additional control to the vector(s) parameters. Various formulas, Pulsar syntax, or other commands may be placed here for application to the Vector prior to the execution of the vector parameters.

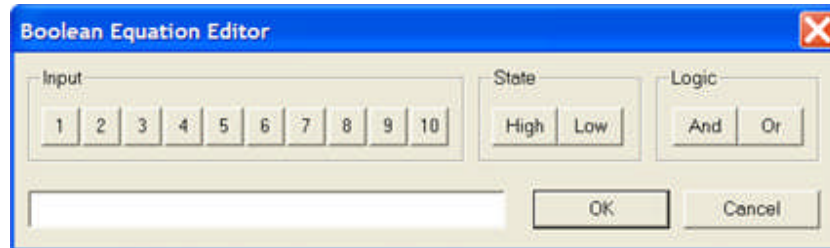


## Boolean Equation

The **Boolean Eqn** check box evokes the edit field and browse button and disables the Apply Time function for the Vector.

Boolean logic may be used to condition the MPRT Timer Start, Timer Stop or advance to the next Vector state. The logic equation created must be met before the advance to the next vector.

The browse button evokes the *Boolean Equation Editor* dialog.



The **Inputs** may be selected in any order. However, evaluation of the equation will be left to right.

The **State** may be either a High (Contact closed or Voltage present) or Low (Contact open or Voltage absent).

The **Logic** may be either AND or OR for any combination of Inputs.

### Example:

i1H\*i3L waits for Input 1 to be High AND Input 3 to be Low

i6H+i7L waits for either Input 6 to be High OR Input 7 to be Low

i5H\*i8H+i3L waits for either (Input 5 High AND Input 8 Low) OR Input 3 Low

The conditions are read left to right and summed in pairs until all Input conditions are read.  
The equation requires 5 and 8 together OR 3.

i1L+i3L\*i4H+i9H waits for ((Input 1 OR Input 3) AND Input 4) OR Input 9

In 1 and In 3 are summed, the result either 1 or 3

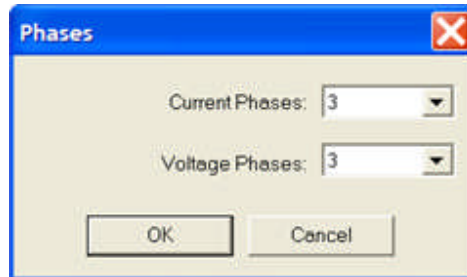
then summed to In 4, the result either 1 or 3 included with 4

then summed to In 9.

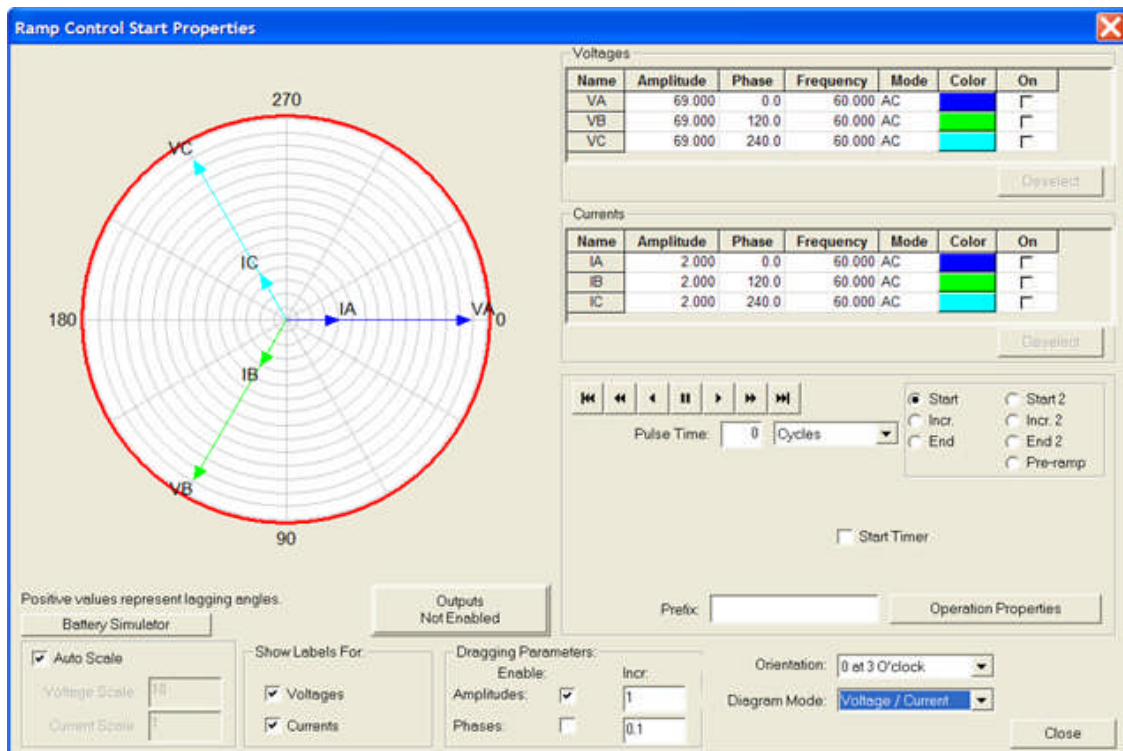
## Online Ramp Control...

## Online Ramp Control...

The **Online Ramp Control**, launched from the AVTS Tools menu item, allows direct real time control of the connected test set. The Control is enabled by selecting Tools\Online Ramp Control... which evokes the *Phases dialog* for pre-programming the Control for the number of current and voltage channels to be active.



After selecting the desired phases to be enabled in the Control, the *Ramp Control Start Properties* dialog is evoked. A Preramp, Ramp 1 and Ramp 2 are available for use to be played back into a protective system. A timer control is available to enable starting the test set timer at the execution of either of the ramp states. The timer stop is typically controlled by an action from one of the outputs of the device connected to the timer stop gate.



For manual ramping of amplitudes phases, a gang control is available through the selection of the vectors (Pulsar amplifiers) to be controlled. The vectors to be controlled in gang are selected by using the mouse to grab and alter the vector(s) parameters. Vector selection is made by clicking on a vector channel name to highlight that vector with its parameters. Should it be desired to simultaneously control more than one vector, click on the wanted vectors while holding down the keyboard Ctrl key. Select from the "Dragging Parameters" box whether the amplitudes and/or phases of the selected vectors are to be active. Once selected, grab the tip of any of the selected vectors in the polar graph and, while holding the left mouse

button down, move the vector(s) on the polar diagram. The values of the vector(s) will change graphically and simultaneously pass the new values directly to the corresponding test set amplifiers.

Ramp Polar Diagram

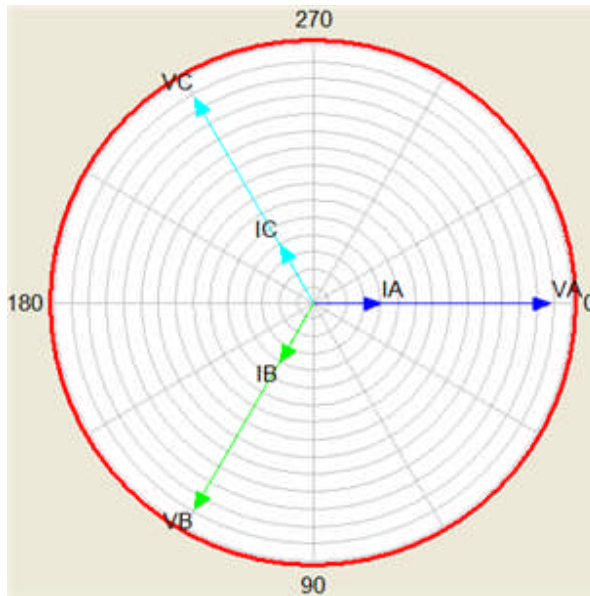
Ramp Voltages and Currents

Ramp Input and Output

Ramp VCR Controls

## Ramp Polar Diagram

The *Polar Diagram* is for display of the Voltages and Currents vectors whose parameters are initially entered in their respective edit fields. Note: The phase displacement is labeled as "Positive values represent lagging angles." The angles are for 0, 90, 270 and 0 (360) degrees clockwise for lag.



The *Voltage Scale and Current Scale* edit fields become available for numeric entry if the Auto Scale check box is unchecked. The Polar Diagram will zoom, when Auto Scale box is checked, to the maximum Voltages and Currents values as the values increase and decrease.

<input type="checkbox"/> Auto Scale
Voltage Scale: <input type="text" value="150"/>
Current Scale: <input type="text" value="15"/>

The *Show Labels For* area is for enabling display of the Current and/or Voltage phase labels on the polar diagram.

Show Labels For:	
<input checked="" type="checkbox"/> Voltages	
<input checked="" type="checkbox"/> Currents	

The *Dragging Parameters* area is for selecting with the check box whether either Amplitudes or Phases or both are selected and controlled by the dragging of the mouse. The Incr (incremental) value for each of the parameters is determined by entry in the appropriate edit field. The values to enter must be in the range of 0.001 to 100.

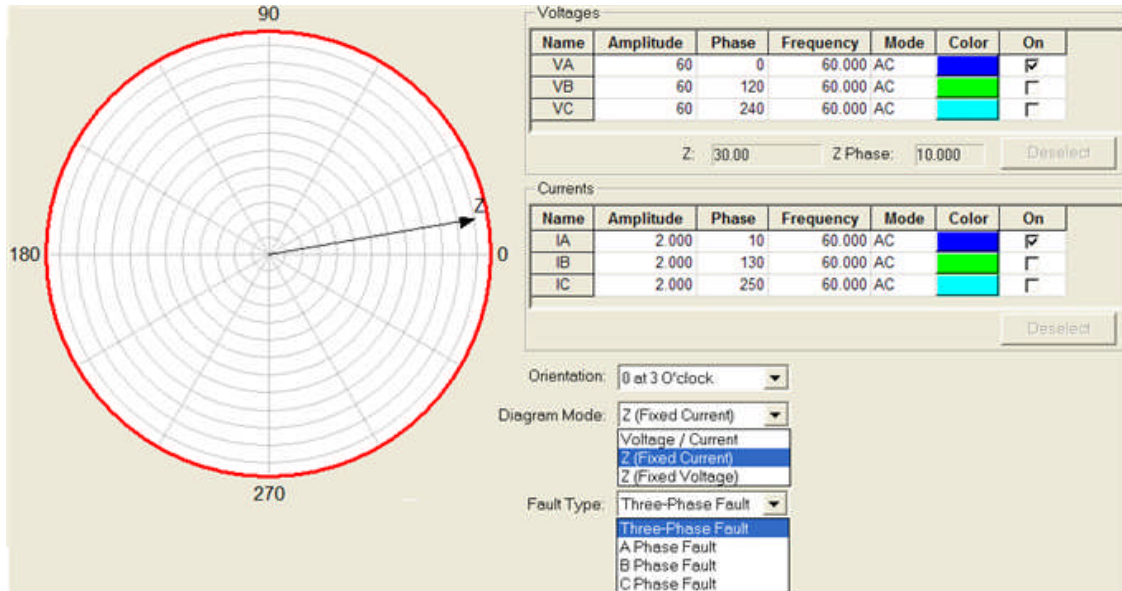
Dragging Parameters:	
Enable:	Incr:
Amplitudes: <input checked="" type="checkbox"/>	<input type="text" value="0.01"/>
Phases: <input type="checkbox"/>	<input type="text" value="0.1"/>

The **Polar Diagram Orientation** drop down list allows the user to option the location of the 0 degree reference to 3 o'clock (default), 6 o'clock, 9 o'clock, or 12 o'clock.

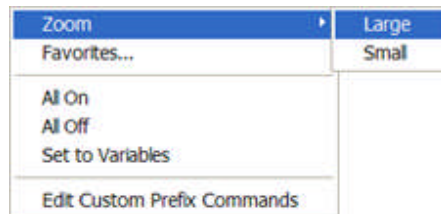
The **Polar Diagram Mode** drop down list allows the user to option the phasor modes to Voltage/Current (default), Z (Fixed Current) or Z (Fixed Voltage).

Select Z (Fixed Current) or Z (Fixed Voltage) and a Fault Type drop down list is visible.

The **Polar Diagram now** displays a single vector Z and 0, 90, 270 and 0 (360) degrees counterclockwise. A value (Fixed Current selected) for Z and angle are displayed under the Voltages parameters fields, based on the voltages and currents magnitudes and angles assigned prior.



The **Drop Down Menu** is placed in view with a right mouse click in the area in and around the polar diagram. The options are:

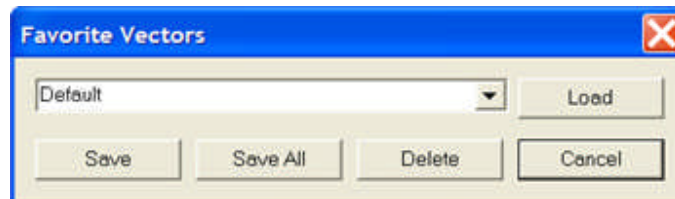


**Zoom**

**Large** option enlarges the Online Vector dialog leaving all sub-boxes normal size.

**Small** option reduces the Online Vector dialog and all sub-boxes to normal size.

**Favorites...** evokes the **Favorite Vectors** dialog. The Default vector set label in the edit field cannot be deleted. The down arrow displays all other vectors that have been saved in during past use of the Online Vector Control or the Vector Control in the Workbook Editor tab.



The **Load** option places the saved data, in the selected vector set from the drop down list, into the Online Vector Control Properties edit, menu and selections areas of the chosen vector in the Vector list box.

The **Save** option allows the save of a single vector set with all the parameters in the database. The vector set will need a label placed into the edit field for recall at a later use. Save will retain only the selected vector in the vector list. The Edit Customs Prefix Commands are saved along with the generator parameters.

The **Save All** option allows the save of a multiple sets of vectors with all the parameters for each vector set to name entered into the edit field for recall at a later use.

The **Delete** option allows the removal from the Vector Favorites list the vector state selected from the drop list.

The **Cancel** option leaves the Favorite Vectors dialog with a null to any changes during the recent evoke.

**All On** places a check in the On check box for all the voltage and current vectors and turns on all Pulsar amplifiers.

**All Off** un-checks all vectors On check box to turn all PULSAR amplifiers to off.

**Set to Variables** replaces the numeric values for all the vector parameters to known default variable names. When used in the OnLine Vector Control, the value for the variable will be applied to Pulsar when Outputs are Enabled. The variable values will be those last set equal to a value in the Variable Watch edit field or from a Test Editor tab control of the last executed test. The values will be retained in AVTS until the variable has been set equal to another value. By default, in the Variable Watch window, all variables' Amplitude and Phase are set equal to zero while the variables' Frequency is set equal to 60.

**Note:** This function is commonly used for the Vector control used within a test in the Test Editor tab. The values for the variables are taken from Settings tab, Variable Watch edit field in the Test tab, or in another procedure control prior to the Vector control.

**Edit Custom Prefix Commands** evokes an edit box in the main Online Vector Control dialog for application of additional control to the vector(s) parameters. Various formulas, Pulsar syntax, or other commands may be placed here for application to the Vector prior to the execution of the vector parameters.

The *Voltage Scale and Current Scale* edit fields become available for numeric entry if the Auto Scale check box is unchecked.

**Outputs Not Enabled** button brightens when clicked and displays Outputs Enabled indicating that the user has full communication with the Pulsar test set and all vector values

## Ramp Voltages and Currents

The *Voltages and Currents* fields display the values set for operation of the online control.

Voltages						
Name	Amplitude	Phase	Frequency	Mode	Color	On
VA	69.000	0.0	60.000	AC	<span style="color: blue;">█</span>	<input type="checkbox"/>
VB	69.000	120.0	60.000	AC	<span style="color: green;">█</span>	<input type="checkbox"/>
VC	69.000	240.0	60.000	AC	<span style="color: cyan;">█</span>	<input type="checkbox"/>

Deselect

Currents						
Name	Amplitude	Phase	Frequency	Mode	Color	On
IA	2.000	0.0	60.000	AC	<span style="color: blue;">█</span>	<input type="checkbox"/>
IB	2.000	120.0	60.000	AC	<span style="color: green;">█</span>	<input type="checkbox"/>
IC	2.000	240.0	60.000	AC	<span style="color: cyan;">█</span>	<input type="checkbox"/>

Deselect

The edit fields for the Voltages and Currents are to enter the values of Amplitude, Phase, Frequency, Mode (either AC or DC or Half-Wave), Color (of the vector in the accompanying polar diagram), and On (a check box to turn the Pulsar channel on when Outputs Not Enabled button is clicked to Outputs Enabled).

The Voltage and Currents edit fields are to set the parameters for each channel and defined in the Ramp Control Properties topic.

A left mouse click on the Voltage or Current channel in the Name column will select that channel and turn the corresponding vector in the polar diagram Red, indicating that the channel is active. The chosen vector may be dragged with the mouse to change the Amplitude and/or Phase values directly on the polar diagram. Multiple vectors may be selected by holding down the Ctrl keyboard key and clicking on the desired vector names. All of the selected vectors will change to red indicating that a left click and hold of the mouse on any one of the vectors will allow change to the Amplitude and/or Phase values directly on the polar diagram and interactively on Pulsar.

The Deselect button unselects all Voltage or Current edit fields that have been highlighted. When the Deselect button is clicked all vector colors turn from red to the normal color.

## Ramp Input and Output

The **Operation Properties** button, available only when either the Start 1 or the Start 2 option is selected, will make visible the *Contact Control Properties* dialog. This dialog enables control of the Pulsar Auxiliary Contact and/or the action of the monitored contact or voltage external to the Pulsar.

**Note:** The Debounce Time for Pulsar may be set for any decimal value, maximum 1638.3 msec., with resolution of 0.1 msec. Debounce time is the time that a change of state for the selected Post activity must remain for the change of state to be recognized as a valid activity. The Debounce Time set is subtracted from the total operate time and will have no effect on the recorded time by test set time out.

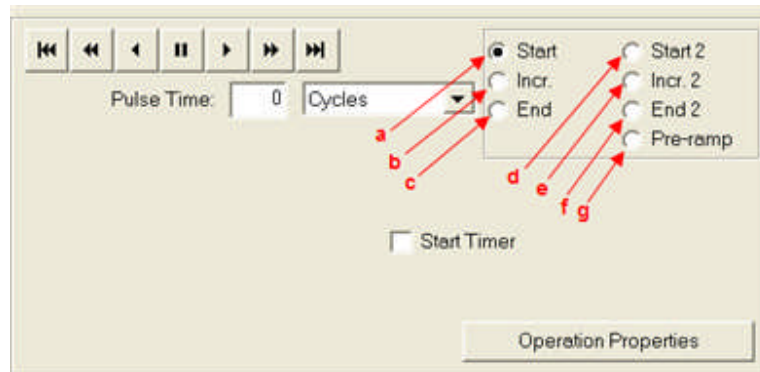
When checked, the Start Timer check box starts the Pulsar timer when any of the options, Automatic Ramp Reverse, Increment Reverse, Increment Forward, or Automatic Ramp Forward, are selected. The timer will stop for the Post conditions met in the Start and Start 2 views selected in the Operation Properties (Contact Control Properties dialog). Post 2 will override any post selected in AVTS and stop the timer and Post 1 will always start the timer after a timer stop.

Should Post 3 be selected for sensing and there is a Ramp 2 with the Post 3 selected for sensing activity, then the timer will stop with activity for Ramp 1 and automatically reset to zero and time again on Ramp 2. However, the timer will only perform the second time stop with activity on Post 2.

The Prefix edit field is made visible and the "Edit Custom Prefix Commands" option is selected from the dropdown menu when the mouse is left clicked in the dialog. The Prefix edit field is visible only in the Start state and allows entry of a formula or other controlling variable using Pulsar syntax. Any control actions from the Prefix edit field will be applied immediately upon execution of one of the Ramp functions. If a Prefix, such as "tae;", is used the auxiliary contact will be enabled (closed) upon execution and will remain closed until the Pulsar is reset or powered down.

The *RAMP Control Properties* contains seven select buttons, each of which will display a separate view of the Voltages and Currents value edit fields.





- a. **Start** – This screen will be the area that the user will enter the vector parameters that will be applied for the start of a RAMP. The Pulse Time edit box allows the numeric entry for the “on” time duration of each applied increment in either Cycles or Seconds, which ever is selected from the drop down list.
- b. **Incr.** – This screen will be the one that the user will enter the value of the RAMP incremental changes to take place during the execution.
- c. **End** – This screen will be the area that the user will enter the vector parameters that will be applied for the last value of execution of the RAMP.
- d. **Start 2** – This screen will be the area that the user will enter the vector parameters that will be applied for the start of a second RAMP.

**Reset Time:** edit field is to enter a numeric or variable time to delay the Start2 execution, usually to allow a device under test to return to a non operate state.

**Note:** Reset Time is independent to the Pre\_Ramp Delay time.

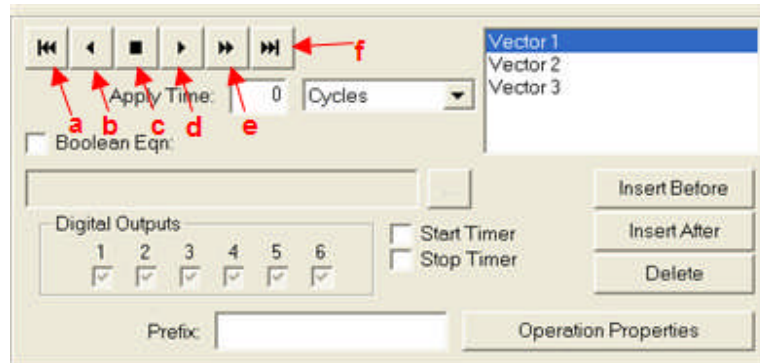
- e. **Incr. 2** – This screen will be the one that the user will enter the value of the second RAMP incremental changes to take place during the execution.
- f. **End 2** – This screen will be the area that the user will enter the vector parameters that will be applied for the last value of execution of the second Ramp.
- g. **Pre-ramp** – This screen will be the area that the user will enter the vector parameters that will be applied for the specified time prior to the Start of a RAMP execution.

The **Time settings** for Pre-ramp Delay, Pulse Times and Increment Times are:

- a. **Pulse Time** - This is a portion of the Increment time that the selected channels to Ramp will be ON. An entry of zero cycles or seconds will cause the ramp to maintain the selected channels to remain ON during the entire change increments sequence from Start to End. When used, a separate Pulse Time for Start 2 is enabled and the value may be different from that of Start 1 Pulse Time.
- b. **Pre-ramp Delay** – This is the time for a set of vector parameters to be applied prior to a Ramp execution and during the Increment time that is not a Pulse Time.
- c. **Increment Time** – This is the total time allocated to each successive increment of change of Ramp. The time may be in Seconds or Cycles by selection from the dropdown menu. Cycle entry will be integer values only. There is a separate Increment Time for Incr. 2 and the value may be different from that of Incr. 1.

## Ramp VCR Controls

The “**VCR**” **buttons** enable the user to control the application of the listed Vector states.



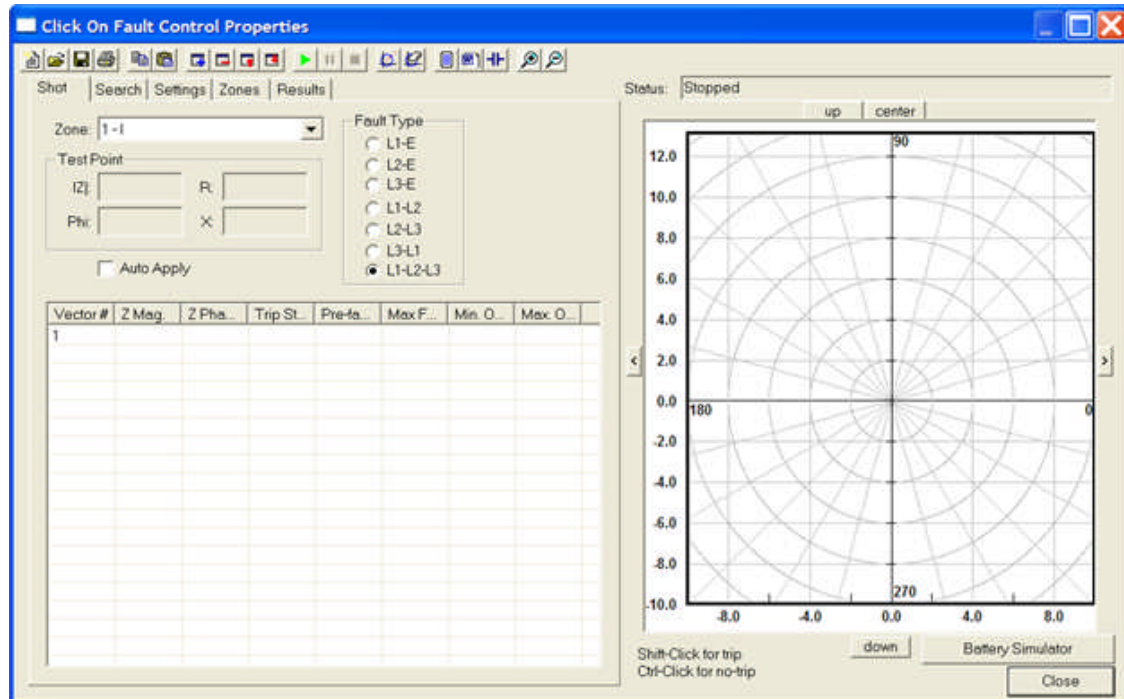
- a. **Go to Start** returns focus alternately to the Start 1 and Start 2 state from any other state. This selection does not enable or disable outputs to Pulsar.
- b. **Automatic Ramp Reverse** enables outputs. This function will begin a reverse ramp from the End state for a single ramp or from the End 2 state for a double ramp on Pulsar.
- c. **Increment Reverse** returns focus to the vector state just prior to the displayed vector after the outputs have been enabled or Pause Ramping has been selected. If the vector state is at the Start or Start 2, the next state selected will be the End or End 2.
- d. **Pause Ramping** halts the Executed Vector Sequence and retains the current parameter values with outputs enabled on Pulsar. The user will be able to select Automatic Ramp Reverse, Increment Reverse, Increment Forward, or Automatic Ramp Forward and have the ramp continue from the current parameter values.
- e. **Increment Forward** brings into focus the vector state just after the displayed vector state after the outputs have been enabled or Pause Ramping has been selected. If the vector state is at the End or End 2, the next state selected will be the Start or Start 2.
- f. **Automatic Ramp Forward** begins the automatic sequence of the entire list of vector states beginning with the Start state for either a single or double ramp on Pulsar.
- g. **Go to End** brings to focus the very last vector state in the existing list.

**Note:** At any time the Outputs have been disabled, the next execution of Automatic Ramp Reverse, Increment Reverse, Increment Forward, or Automatic Ramp Forward will enable outputs and change to the appropriate Start, Start 2, End or End 2 state to continue.

## Online Click On Fault Control

## Online Click On Fault Control

The *Online Click On Fault Control* is to graphically create many tests to a multi-zone relay or a multi-relay system connected together. The Online control is a real time executable tool for quick and easy creation of tests to a given impedance characteristic.



The properties in each of the **Online Click On Fault Control** are available in the Click On Fault Control with minor exceptions.

When first opened, the **Shot Tab** is the default tab.

Graphic displays of the Zones of protection including reach, phase angle, offset and other zone parameters are produced by an easy left mouse click and drag. Refinement of each is accomplished in the five tabs at the top of the Control Properties.

The Online Control has a main CoF Menu Toolbar for entries, execution and control of the Online Control.

There are five screens to select for setup and control of the Zones, Tests for each Zone, Fault Types, and Results.

An Impedance characteristic may be created, by use of the Define Theoretical For Test Tool, in the Graph Pane and saved to a file.

Using the same Import tool, multiple characteristics that exist in RIO file formats may also be called and used in the Graph Pane for multiple tests.

The created tests may be saved with an identifying name to the AVTS default folder Click On Fault Tests.

**Note:** Tests Results are not saved with a file. Results must be either Printed or Exported to a Word document and saved. Both options are controlled by the Print Click On Fault Test Results tool or the Export Results to Word.

In addition, the file may be called by placing the Click On Fault Control in a new or existing Relay Test Editor tab and saved with other tests for a particular AVTS Relay Test Module.

**Note:** Data created in the Online Click On Fault Control and saved to the AVTS/Click On Fault Tests folder may be called by the Click On Fault Control in a Relay Test Editor Tab.

The file may then be recalled at any time and executed for test results complete with an accompanying graph of tests and results.

Topics defining the Click on Fault Control in the Test Editor tab and the Online Click on Fault Control are:


- CoF Menu Toolbar
- CoF Graph View
- CoF Shot Screen
- CoF Search Screen
- CoF Settings Screen
- CoF Zones Screen
- CoF Results Screen
- CoF Example

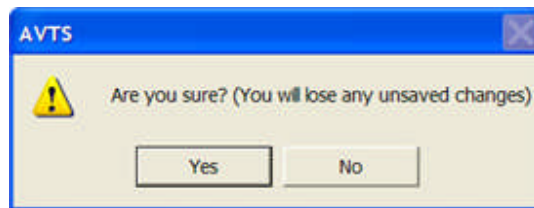
## CoF Menu Toolbar


The **Menu Toolbar** includes an icon for using and editing the six tabbed screens in the Click On Fault Control Properties.

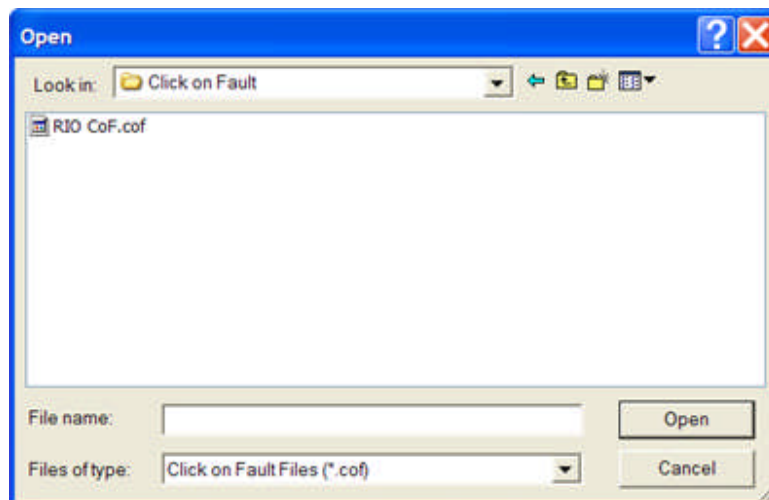


The tool icons are explained from left to right.


**New Click On Fault Test**  icon is for to create a new set of tests. Selection evokes the *Clear Existing Test* message dialog.

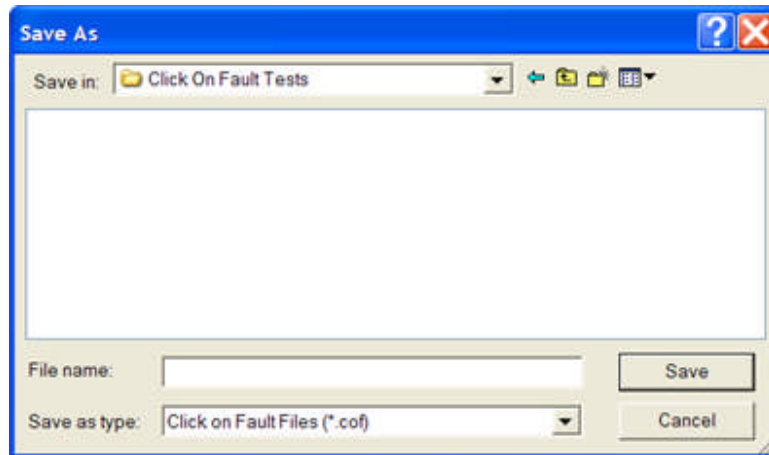


**Open Click On Fault Test**  icon is to open a previously created and saved set of tests. Selection evokes the *Open* dialog. In this example, a subfolder to AVTS has been created and labeled Click On Fault Tests.




Note: The Click On Fault Files will have an extension \*.cof. Only file with this extension may be opened for use in the Online Click On Fault Control.


**Save Click On Fault Test**  icon is to save a created set of tests in the C:\Program Files\Megger\AVTSv2\Click On Fault Tests folder. Selection evokes the *Save As* dialog. By default the dialog opens to the AVTS folder and in this example, the Click On Fault Tests folder has been selected for choosing the desired tests file.





**Note:** The execution results in the Results Tab are not saved with any constructed and executed tests. Results to be kept must be first saved with the **Print Click On Fault Test Results** or the **Export Results to Word** tools.


**Print Click On Fault Test Results**  icon is to directly print the test results in the active file. Selection first opens the computer default word processor software, typically Microsoft Word. Then the default Windows print dialog is evoked for printer settings and command print.


**Note:** If the test results had not been previously saved, they may be saved in the Word document as a \*.doc for later recall to print.


**Copy Test Points**  is to copy a single test point or multiple test points (hold Ctrl key while selecting) and retain for paste to another location within Click On Fault.


**Paste Test Points**  is to place in a new location the test or test points previously retained by use of the Copy control.

**Add New Test Points**  creates a new test number in the Shot tab Vector list, the Search Line list and the edit fields for the Zone Number and Zone Label in the Zone tab. A new test is added in numeric order each time the Tool icon is selected. The Tool is active in Shot, Search and Zones tabs only


**Delete Selected Test Points**  deletes the last test number in the Shot tab Vector list, the Search Line list and the edit fields for the Zone Number and Zone Label in the Zone tab. The next test in reverse numeric order is deleted each time the Tool icon is selected. The Tool is active in Shot, Search and Zones tabs only

**Move Test Point Up In Test**  moves a selected test in the list up one row with each click of the Tool icon.

**Move Test Point Down In Test**  moves a selected test in the list down one row with each click of the Tool icon. The Test Points can not be moved to the last position number.


**Run Click On Fault Test**  is an active test execution tool in the Shot and Search tabs only.

**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.


**Pause Click On Fault Test**  is active after a test has been executed by the Click On Fault Test Tool icon. It is active in Shot and Search tabs only.

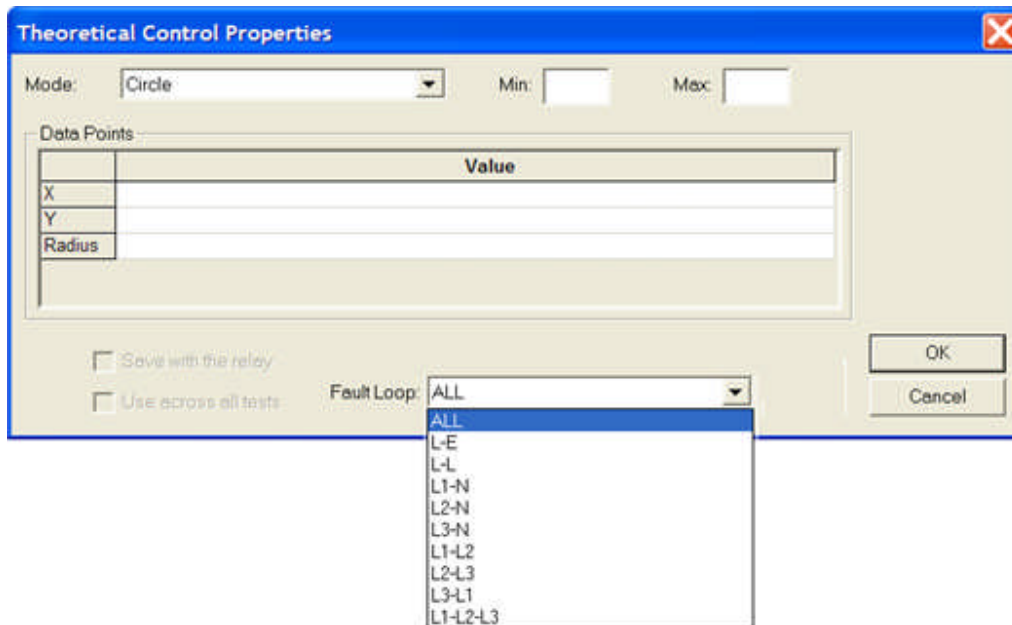
Selecting this tool prior to test execution will evoke the Contact Control Properties to select the available options for test reaction recognition.

**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.

**Stop Click On Fault Test**  is active after a test has been executed by the Click On Fault Test Tool icon. It is active in Shot and Search tabs only.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.

**Define Theoretical for Test**  opens the *Theoretical Control Properties* dialog for defining the impedance graph data for plotting on the graph. The data may also be used for accuracy comparison to the recorded results. The comparison is governed by the data provided in the Define Log Control.




**Note:** This option allows import of RIO files directly to the Online Click on Fault Control.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.

**Clear Result Points on Graph**  deletes all previous results from the graph. The action is true for results just accumulated or those from a previously saved and opened test file with results. The test points created remain with their values in the Vector# or Line# tables.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.


**Define Log Control**  opens the Log Control Properties dialog. The details for variables and calculations are placed in this dialog for control of the logged results. The Variables listed in the Log Control will be placed in columns in the CoF Results Screen and the test results listed after tests execution. The appearance is similar then to a test result printed in a usual AVTS test with results governed by the Log control.


**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.

**Export Results to Word**  makes the Results tab active and opens a Word document. The accumulated test results present in the Results tab are placed in the document along with the graph complementary to the results. The Graphic Impedance with test lines, test points and test results is included in the document.

**Note:** This tool is dimmed in the Click on Fault Control in the Relay Test Editor tab.

**Define Contact Control**  opens the Contact Control Properties dialog. The options to select test set input contacts for monitoring the active test. The selections allow for biasing a single contact or for setting multiple contacts for a wait any or wait all condition. The options are defined in detail in [Contact Control Properties](#) dialog topic.

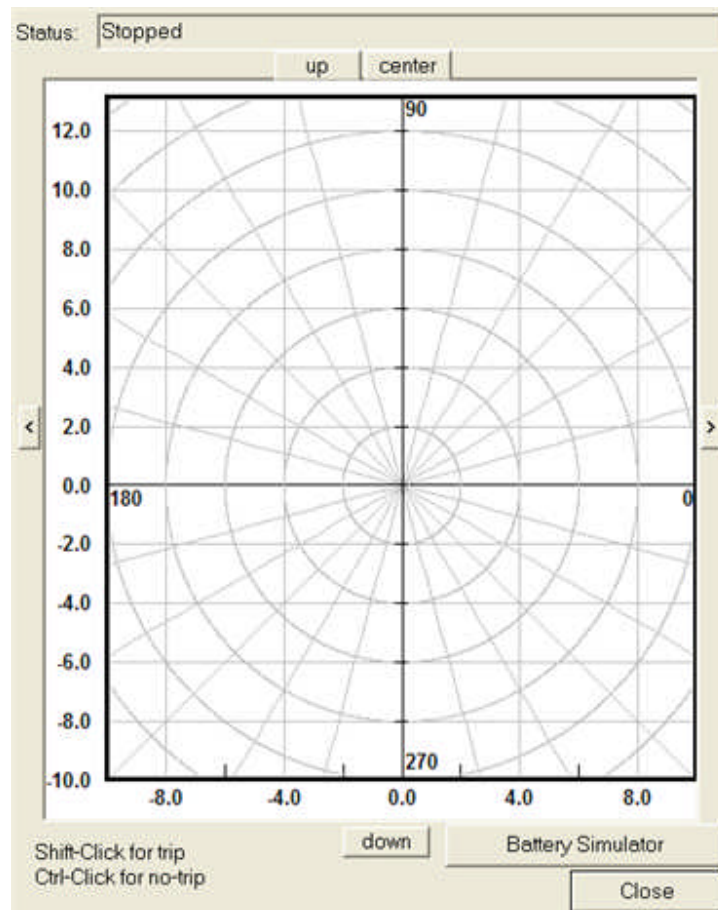
**Zoom In**  applies to the Graph display only. The graph zooms in with each mouse click on the icon in 1/2 view increments. The Zoom In feature may be monitored by watching the graph horizontal and vertical scales. The operation is valid in all tabs.

**Zoom Out**  applies to the Graph display only. The graph zooms in with each mouse click on the icon in double view increments. The Zoom Out feature may be monitored by watching the graph horizontal and vertical scales. The operation is valid in all tabs.



## CoF Graph View

The **Graph** on the right side of the Click On Fault Control Properties dialog has several elements for edit. The graph is visible for test edits in all tabs. The edits allowed are specific to each tab and dependent on the tab use.



**Test points** in the **Shot Tab** will be shown on the graph referenced by the values in the **Shot Tab, Settings Tab** and **Zones Tab**.

**Search Lines** will be shown on the graph referenced by the values in the **Search Tab, Settings Tab** and **Zones Tab**.

Note: The Graph will display the Shot Points or Search Lines in the Settings, Zones and Results tabs dependent on the last tab selected, the Shot or Search tab. The Results tab will only display recent test Results of an executed test; else, the Shot Points or Search Lines will show.

The **Status:** field will display the condition of the Run Click On Fault tool, such as

**Stopped** is displayed when the Control is first opened.

**Paused** is displayed when the *Pause Click on Fault Test* tool is selected. Select the *Run Click on Fault Test* tool to continue the tests.

**User Stopped Test** is displayed when the *Stop Click on Fault Test* tool is selected.

**Running** is displayed while tests are executing after the *Run Click on Fault Test* tool was selected.

**Finished Running** is displayed when the executed tests have completed and Results are available to view.

**Communication Error** displayed when the Control has lost connectivity to the MPRT or Pulsar test set.

The graph has left and right arrow buttons to move the center of the graph in the opposite direction.

The **right arrow button** is on the right of the graph and approximately mid vertical distance. Click on this button to move the view to the right, the polar center to the left.

The **left arrow button** is on the left of the graph and approximately mid vertical distance. Click on this button to move the view to the left, the polar center to the right.

The **down button** is on the bottom of the graph and approximately mid horizontal distance. Click on this button to move the view down, the polar center up.

The **up button** is on the top of the graph and approximately mid horizontal distance. Click on this button to move the view up, the polar center down.

The **center button** is on the top of the graph and adjacent to the Up button. Click on this button to move the polar center to the approximate mid vertical, mid horizontal location in the graph..

The **Battery Simulator** button opens the Battery Simulator Control Properties to apply control voltage to the relay under test.

Note: **Battery Simulator** button is not available **Click On Fault Control** embedded in a Relay Test Editor tab.

To enter test points in the graph for "Shot" tests use the directives given at the lower left to the graph.

**Shift-Click for center** to enter a test point where, in or out of the characteristic, a trip is expected.

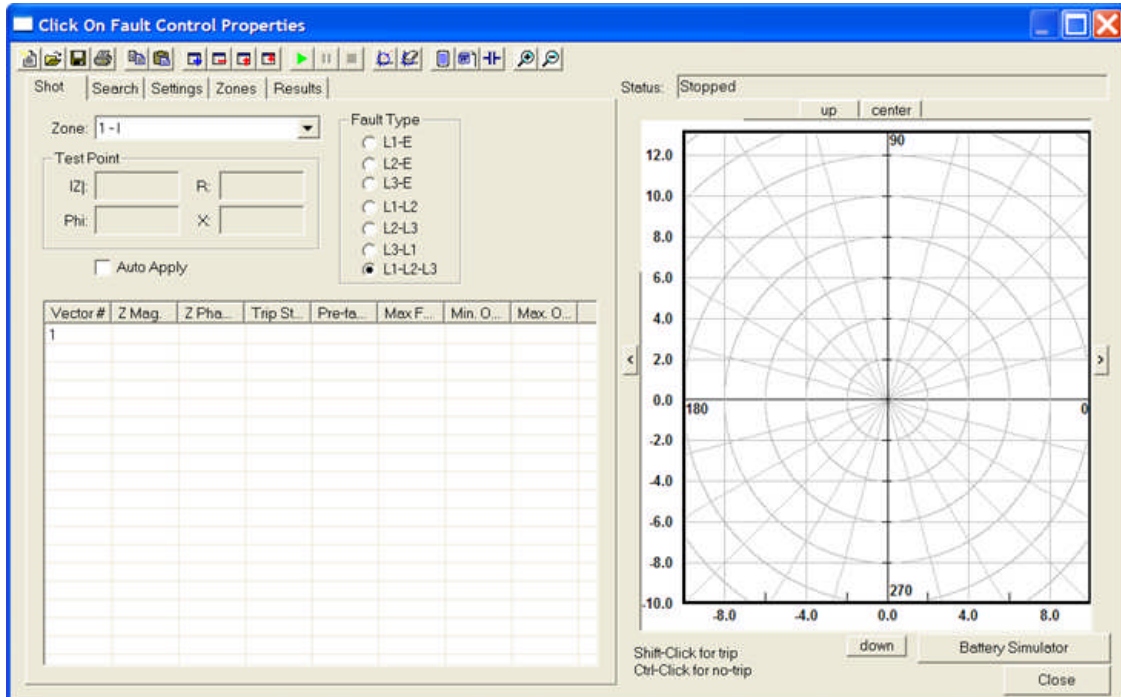
Trip points are entered as Red Triangles and will change to a square when the point is executed. A Green Square after the test indicates a successful test. A Red Square after the test indicates an unsuccessful test.

**Ctrl-Click-Click for search line** to enter a test point where, in or out of the characteristic, a trip is not

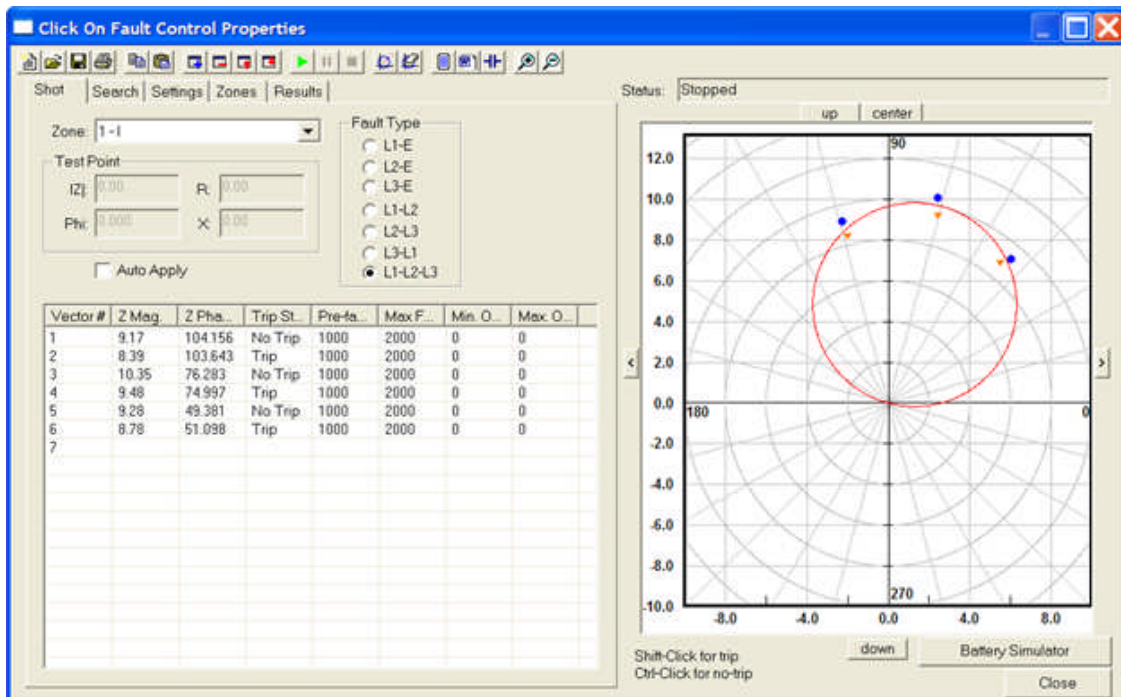
expected. No-trip points are entered as Blue Circles and will change to a square when the point is executed. A Green Square after the test indicates a successful test. A Red Square after the test indicates an unsuccessful test.

## CoF Shot Tab

The **Shot Tab** is to create one or more points of test, each to replicate a fault at a particular magnitude and angle.



Several Trip and/or No-Trip points may be selected for each Fault Type.



The **Run Click On Fault Test** tool is to execute the test for this Tab.

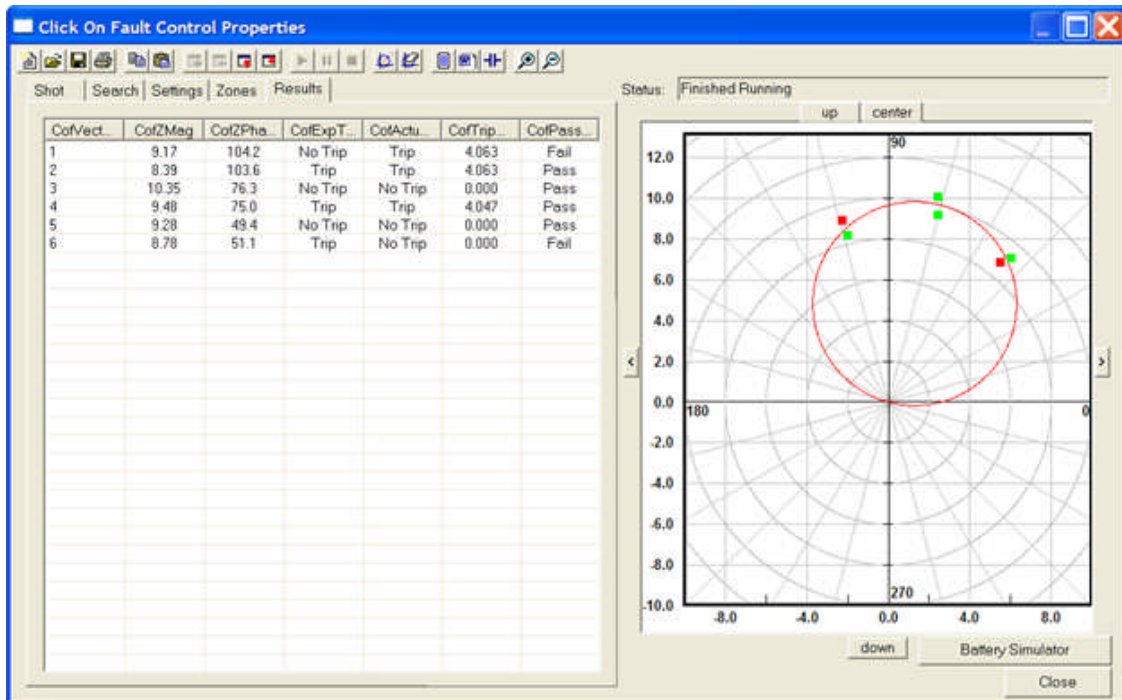
**Note:** The **Run Click On Fault Test** tool is not enabled in the **Click On Fault Control** embedded in a Relay Test Editor tab. The execute for an embedded test is the Tool bar **Execute Test** tool.

When executed, all **Trip** and **No-Trip** points for the selected Fault Type will run in turn beginning with Vector# 1 and continuing to the last Vector# in the list.

Select another **Fault Type** and execute. All created Trip and/or No-Trip points will operate in turn.

**Note:** An entered Trip point is a **Red triangle**. A valid Trip point will turn to a Green square, an invalid Trip point will turn to a Red Square. An entered No-Trip point is a **Blue circle**. A valid No-Trip point will turn to Green Square; an invalid No-Trip point will turn to a Red square.

*Shot Test Results* are accumulated at each of the created test points.



Entries created in other tabs for reference

**Zone** drop list for selection of the characteristic data from the Zones tab

The **Test Point** is the set of values listed both in Mag and Phase Angle, and in Cartesian values that are created in the Graph or used to edit a selected Vector# edit fields

The **Fault Type** List check bullet is to select the fault type to apply to the selected test point.

- L1-E** - applies a single phase fault for the selected test point
- L2-E** - applies a single phase fault for the selected test point
- L3-E** - applies a single phase fault for the selected test point
- L1-L2** - applies a phase-to-phase fault for the selected test point
- L2-L3** - applies a phase-to-phase fault for the selected test point
- L3-L1** - applies a phase-to-phase fault for the selected test point
- L1-L2-L3** - applies a 3 phase fault for the selected test point

The **Auto Apply** check box will apply the selected **Fault Type** to each and every test point in order of the Vector# listing.

Note: The **Auto Apply** tool on the tool bar is not enabled in the **Click On Fault Control** embedded in a Relay Test Editor tab.

The **Graph** pane, when clicked, evokes the **Hold Shift or CTRL keys** to select Trip or No-Trip points message.

A created **Trip** point will show as a **Red triangle** in the graph. When executed, the result of the test on the point(s) created will change to **Green square**. The result will change to a **Red square** if a **No-Trip** occurs for an expected **Trip**.

A created **No-Trip** point will show as a **Blue circle** in the graph. When executed, the result of the test on the point(s) created will change to **Green square**. The result will change to a **Red square** if a **Trip** occurs for an expected **No-Trip**.

Each **Trip** or **No-Trip** point created will be placed in the Vector# list with all the associated parameters of the point.

Continued clicks will produce additional test points at the mouse locations. The graph may contain numerous Trip and No-Trip points for relay response and setting verification.

**Shot Tab Test Point parameters** may be entered by direct clicks on the graph or directly in the edit fields using the keyboard

Vector#	Z Mag.	Z Phase.	Trip State	Pre-fault ms	Max Fault Time	Min. Oper Time	Max. Oper Time
1	27.25	84.765	No Trip	1100	2000	58	62
2	25.16	85.188	Trip	1100	2000	58	62
3							

**Vector#** is the number of each created Trip or No-Trip test point - This field may NOT be highlighted and edited

**Z Mag.** is the ohmic distance of the test point - This field may be highlighted and edited

**Z Phase** is the phase angle of the test point - This field may be highlighted and edited

**Trip State** is a drop list for expected **Trip** or **No -Trip** response to the test point - The Trip State may be changed and the corresponding test point in the Graph will change character also

**Pre-fault ms** is the apply time of normal voltages and currents to the test that have be created in the Settings tab - This field may be highlighted and edited

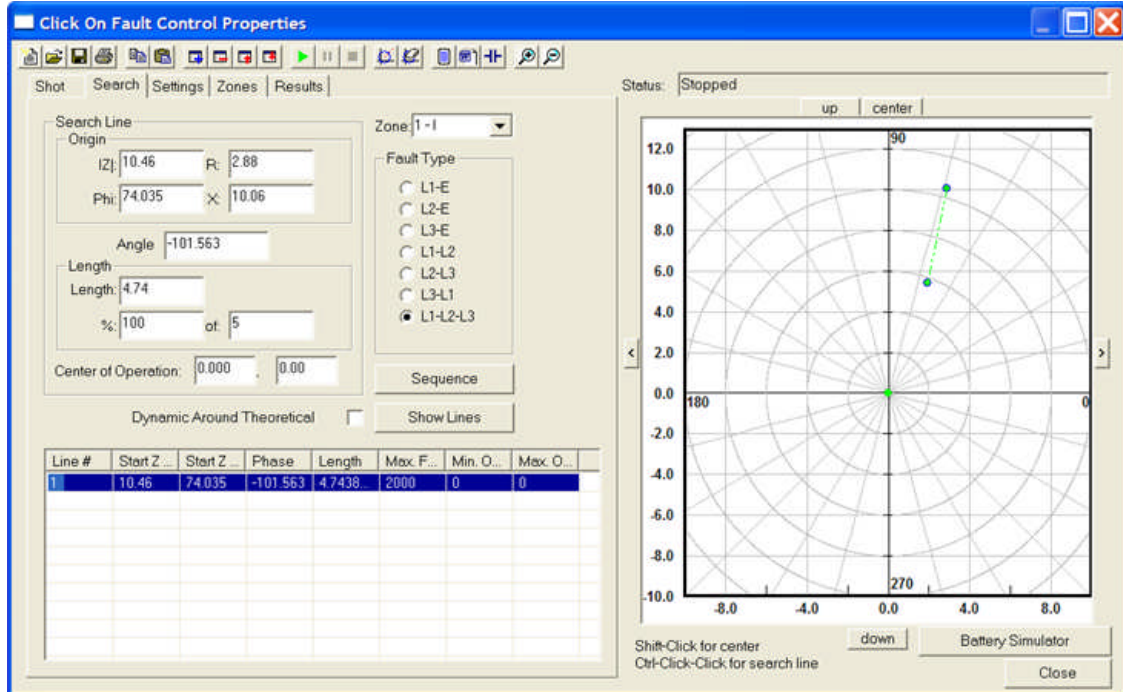
**Max Fault Time** is the maximum time value in milliseconds that the fault values will be applied before test abort due to a no-operate response that has be created in the Settings tab - This field may be highlighted and edited

**Min. Operate Time** is the minimum time tolerance allowed for the Zone selected that has be created in the Zones tab - This field may be highlighted and edited

**Max. Operate Time** is the maximum time tolerance allowed for the Zone selected that has be created in the Zones tab - This field may be highlighted and edited

## CoF Search Tab

The **Search Screen** is to create a series of tests which ramp from a beginning point along a search line to an end point in that direction.



To create a test line, hold the Ctrl key and left click at a start point in the graph. Continue holding the Ctrl key and drag the mouse to the final test point and left click to anchor the test line. The created start and end points and the test line will be drawn in red, but will turn blue for a saved line with a second mouse click prior to release of the Ctrl key.

A created test will automatically enter the data in the Search Line block of edit fields. Data displayed in these fields are dependent to the Line# (Test line) selected and may be edited to adjust or refine the Line#.

The **Run Click On Fault Test** tool is enabled for this Tab. The executed test will first apply test to the test line point farthest from the graph polar origin for a Trip or No Trip. The next application will be applied to the point closest to the graph polar origin for a Trip or No Trip. Each successive test point will oscillate one-half the distance between a Trip and a No Trip point until a Trip or No Trip occurs within either the Relative (%) or Absolute (Ohms) Impedance Tolerance setting in the Zones tab.

**Note:** The **Run Click On Fault Test** tool is not enabled in the **Click On Fault Control** embedded in a Relay Test Editor tab. The execute for an embedded test is the Tool bar **Execute Test** tool.

The data area contains a Search Line block, Zone: drop list, Fault Type list, Dynamic Around Theoretical check box and the list of created test vectors (Line#s)

### Search Line block

**Origin** block has data fields for

**[Z]:** the magnitude of impedance point first clicked to create a search line

**Phi:** the phase angle associated with the Z magnitude field is the angular displacement of the **[Z]** selected point (firsts point clicked)

**R:** the Cartesian resistive equivalent of the impedance **[Z]** and its angle **Phi**

**X:** the Cartesian reactive equivalent of the impedance **[Z]** and its angle **Phi**

**Angle** field displays the angle of the line relative to the phase angle of the search line with the start point is positioned on the graph polar origin

**Note:** When drawn, the fields display the values for each search line, but gray to indicate a non-active field. The fields are made active for edit when the **Line#** is selected.

**Length** block

**Length:** is the physical ohmic length defined by the two ends of the selected search line.

**%:** entry describes the percent of the system parameter "line length" to be the test limits.

**of:** entry describes the "length" in ohms of the constructed search line.

**Center of Operation** fields

**Center of Operation:** the first field is for the Phase Angle value in degrees

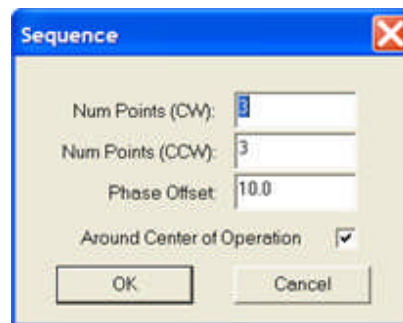
**Center of Operation:** the second field is for the Magnitude of the impedance value autocreated by the Show Lines button

**Note:** The impedance is automatically created by the Phase Angle entry and the Dynamic Around Theoretical checked box. Entry of a phase angle other than the Zone Max Torque Angle will shift all test entries to the new angle entry when Show Lines button is clicked. The Magnitude will also be changed to 1/2 of the center test Line# intercept to the plotted characteristic.

**Dynamic Around Theoretical** check box

**Note:** Use of this option will replace any test lines previously constructed and will not be retrievable unless they were saved to a file name before the Show Lines button is selected.

**Sequence** button opens the **Sequence dialog**



**Num Points (CW):** edit field is to enter the number of test points that describe as many test lines clockwise from the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Num Points (CCW):** edit field is to enter the number of test points that describe as many test lines counterclockwise from the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Phase Offset:** edit field defines the angular displacement for each test line from that defined by the Maximum Torque Angle defined by the Define Theoretical for Test tool.

**Around Center of Operation** check box will automatically construct the test lines about a constructed Center of Operation either by entering values in the edit fields or by construction directly on the graph.

**Note:** The middle line of the set will be considered the Maximum Torque Angle and will align with the graph polar origin. The line length will be double the ohmic distance from the Center of Operation point to the impedance characteristic along the line extended from the origin. The maximum torque angle of the impedance characteristic need no be know; the test lines will be determined by the Center of Operation and the details of the Sequence dialog options.

OK or Cancel buttons to either accept the settings or return to the Search tab without saving the Sequence dialog edits

**Show Lines** button displays all the test Lines created by the checked Dynamic Around Theoretical box and the edits made in the Sequence dialog. Line #1 will be the Center test Line. The succeeding test lines will be in order of the angles selected in the Sequence dialog.

Note: Additional Test Lines may be added to the Dynamic Around Theoretical option by using the Ctrl-Click-Click option in the Graph.

**Zone** drop list is to select the Zone to place the search lines for test

**Fault Type** block lists the check buttons for the types of faults to execute for the selected Zone and Search Line

**Line List Data** columns are defined below:

**Line#** list contains all data associated with a constructed test line. All fields may be edited except for the Line# column. A constructed test line is displayed **Blue** but will be displayed **Green** when the Line# row is selected. The Line# test result will be displayed as a **Red** square on the test line.

**Start Z Mag.** is the magnitude Line test start point displayed in the **[Z]** edit field. Edit in one field will be reflected in the other field.

**Start Z Phase.** is the phase angle of the Line test start point displayed in the **Phi** edit field. Edit in one field will be reflected in the other field.

**Phase** is the phase direction (**Angle**) of the test line from the start point to the end point with the start point referenced to the graph polar origin. Edit in one field will be reflected in the other field.

#### **Length**

**Max. Fault Time** is the Maximum Fault Time in milliseconds to be applied at each test location on the test line. The entry is defaulted by the Max Fault Time entry in the Settings tab.

**Min. Oper Time** field is the Minimum Operate Time in milliseconds of the relay. The entry is defaulted by the Expected Operation Time - the Absolute Operation Time Tolerance entries in the Zones tab.

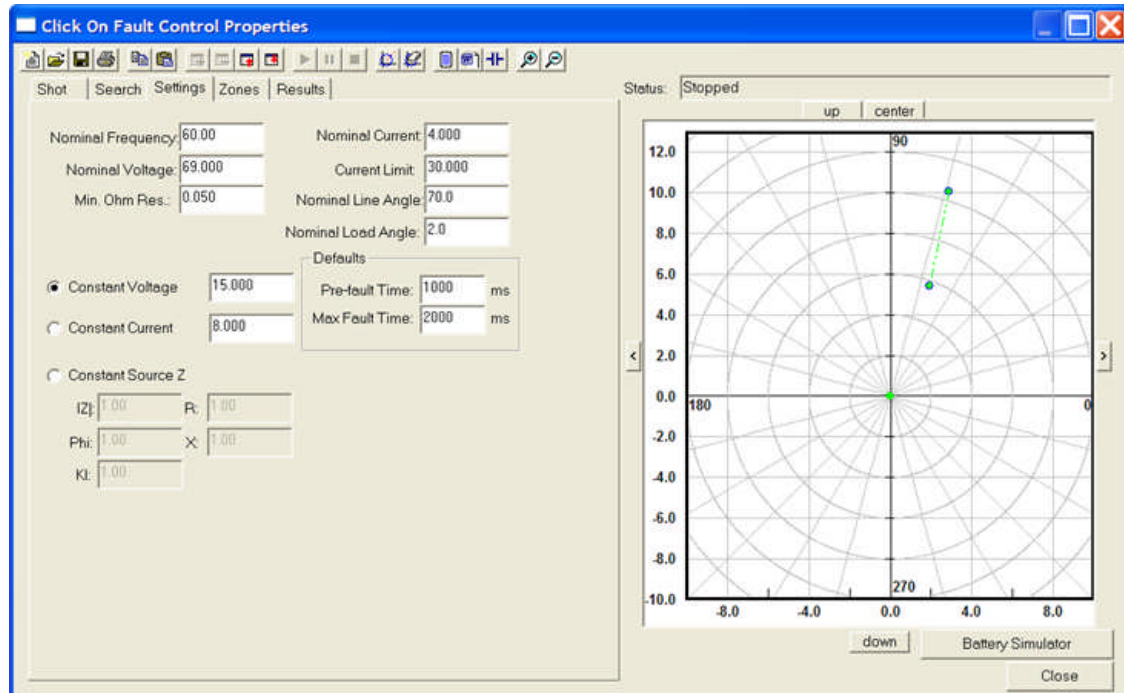
**Max. Oper Time** field is the Maximum Operate Time in milliseconds of the relay. The entry is defaulted by the Expected Operation Time + the Absolute Operation Time Tolerance entries in the Zones tab.

The **Graph** pane, when clicked, evokes the **Hold Shift or CTRL keys** to create a search line message.



## CoF Settings Tab

The *Settings Screen* has edit fields to enter the desired normal system conditions, Default settings and test parameter fault constants.



Run Click On Fault Test execute tool is not active in this tab.

Neither Shot Points nor Search Lines can be constructed in this tab.

The edit entry fields:

### Nominal Settings

**Nominal Frequency:** is to enter the value of normal system frequency in Hertz. Default value is 60.00.

**Nominal Current:** is to enter the value of normal load current in amperes. This value along with the Nominal Voltage setting to apply to the relay during "Pre-fault" to reset system logic. Default value is 4.000.

**Nominal Voltage:** is to enter the value of normal Line-to-Earth voltage. This value along with the Nominal Current setting to apply to the relay during "Pre-fault" to reset system logic. Default value is 69.000.

**Current Limit:** is to enter the maximum value of current in amperes applied by test set current channels for any test. Default value is 30.000.

**Min. Ohm Res.:** is to enter the value of the minimum resistance allowed in the calculations for all Fault Types and Shot Points or Search Lines. This value is used to determine the first value of test result within the accuracy set by the Impedance Tolerance value in the Zones Tab. Default value is 0.050.

**Nominal Line Angle:** is to enter the value of angle in degrees of the protected transmission line. Default value is 70.0.

**Nominal Load Angle:** is to enter the usual value of angle in degrees of the normal load on the line. Default value is 2.0.

**Constant Test Settings** check buttons allow selection of one only

**Constant Voltage** is to enter the value of volts to be held constant for all Fault Types tests under execution. Default value is 15.000.

**Constant Current** is to enter the value of amperes to be held constant for all Fault Types tests under execution. Default value is 8.000.

**Constant Source Z** is to enter the value of ohms of source to be held constant for all Fault Types tests under execution

**[Z]:** the magnitude of impedance point first clicked to create a search line

**Phi:** the phase angle associated with the Z magnitude field is the angular displacement of the **[Z]** selected point (first point clicked)

**R:** the Cartesian resistive equivalent of the impedance **[Z]** and its angle **Phi**

**X:** the Cartesian reactive equivalent of the impedance **[Z]** and its angle **Phi**

**kL:** kL is set equal system source grounding compensation factor **kS = 1/3 (ZS0 / ZS - 1)**

#### Defaults settings

**Pre-fault Time:** is to enter the number of milliseconds that **Nominal** values are applied prior to each Shot test and to each Search Line ramp test. Default value is 1000.

**Max Fault Time:** is to enter the maximum number of milliseconds that an executed value of fault current will be applied at which time the test will abort. Default value is 2000.

**Note:** The **Max Fault Time** is applied to each discrete ramp in a Search Line test

The **Graph** pane displays the test locations that are created in the Shot Tab or Search Tab; the tab last selected.

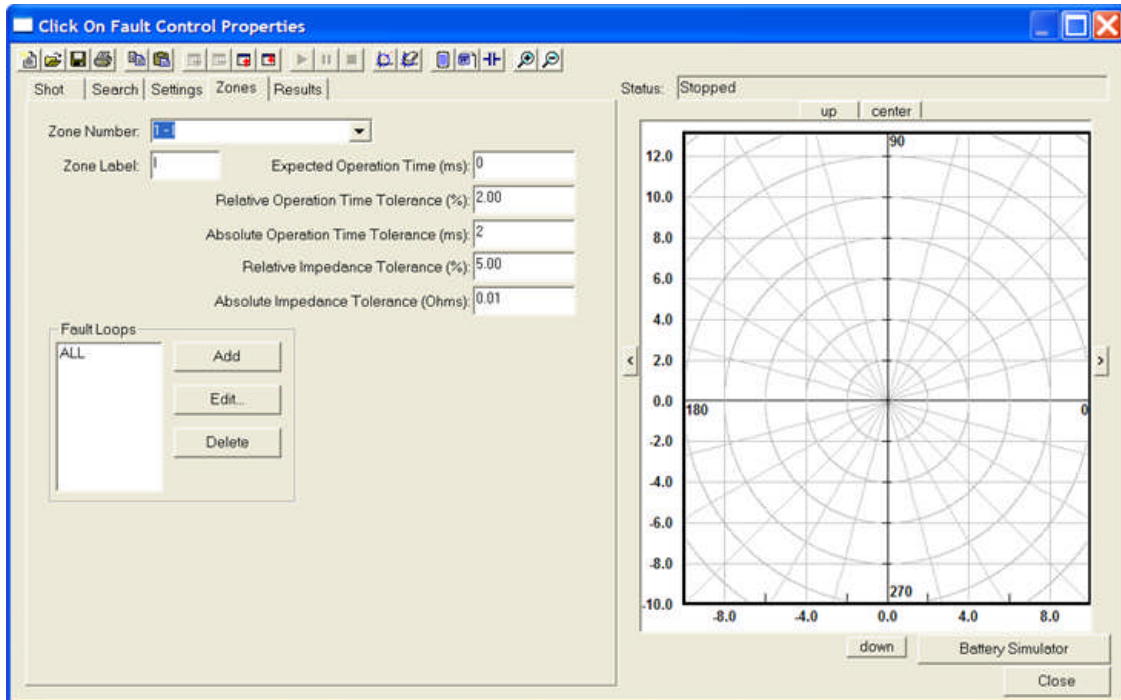
Test are not able to be executed in this tab. If the Shot Tab was last selected, the graph will display all the Trip and No-Trip Shot points. Additional Test Spots may be added using the Shift and Ctrl keys and mouse as defined in the Shot Tab.

If the Search Tab was last selected, the graph will display all the test Search Lines. Additional Search Lines may be added using the Ctrl key and mouse as defined in the Search Tab.

The Graph pane, when clicked in this tab, evokes the *Hold Shift or CTRL keys* to create a search line message.

## CoF Zones Tab

The *Zones Screen* has edit fields to enter the desired test accuracy parameters and construct several characteristics for selected fault types.



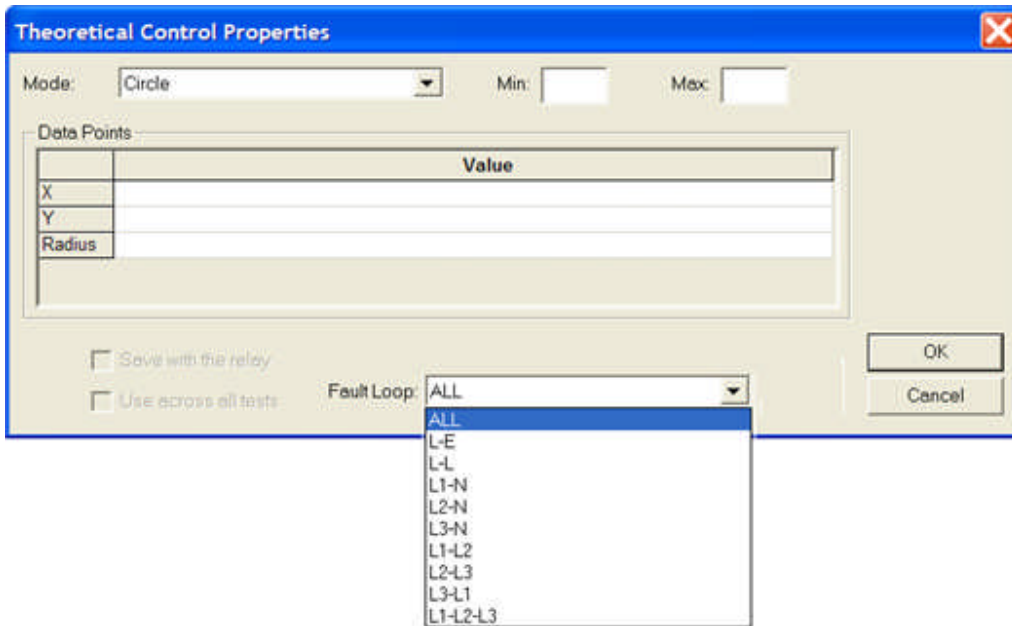
**Run Click On Fault Test** execute tool is not active in this tab.

Neither Shot Points nor Search Lines can be constructed in this tab.

Edit and Selection fields are:

**Zone Number:** drop list displays the Zone Labels created with companion constructed characteristics.

**Zone Label:** edit field allows edit of the Zone Label for the companion *Fault Loop* selected in the Theoretical Control properties dialog.



The Fault Loop selections are:

**ALL** will use the constructed characteristic for all Fault Types selected for test in the Shot and Search tabs.

**L-E** will use the constructed characteristic for all Fault Types involving Earth (ground) selected for test in the Shot and Search tabs.

**L-L** will use the constructed characteristic for all Fault Types involving phase pairs selected for test in the Shot and Search tabs.

**L1-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L2-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L3-N** will use the constructed characteristic for the Fault Type involving the single phase to Earth (ground) selected for test in the Shot and Search tabs.

**L1-L2** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L2-L3** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L3-L1** will use the constructed characteristic for the Fault Type involving the phase pair selected for test in the Shot and Search tabs.

**L1-L2-L3** will use the constructed characteristic for the Fault Type involving all phases for test in the Shot and Search tabs.

**Expected Operation Time (ms):** edit field is to enter the expected time delay, if any, set for the selected Zone of the relay.

**Relative Operation Time Tolerance (%):** edit field is to enter the plus/minus percent tolerance acceptable as a result compared to the **Expected Operation Time.(ms):** minus/plus the **Absolute Operation Time Tolerance:**

**Absolute Operation Time Tolerance (ms):** edit field is to enter the plus/minus numeric tolerance acceptable as a result compared to the **Expected Operation Time.(ms):**

Example:

**Zones** tab settings are:

Expected Operation Time (ms): = 60

Relative Operation Time Tolerance (%): = 10  
 Absolute Operation Time Tolerance (ms): = 40

Then **Search** tab values are:

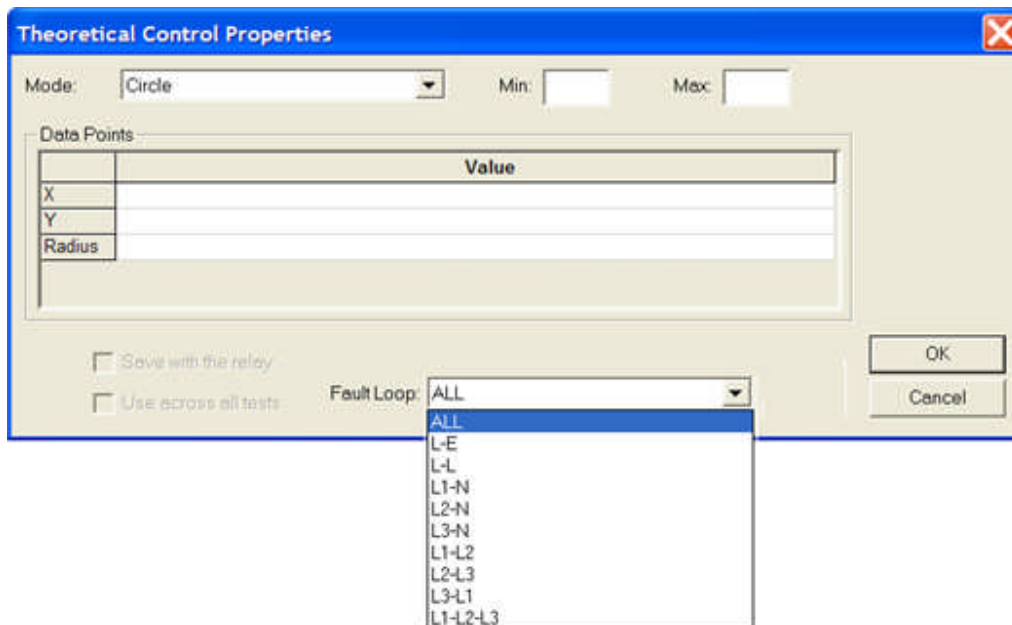
Min. Oper Time =  $60 - 40 - (10\% * 60) = 14$   
 Max. Oper Time =  $60 + 40 + (10\% * 60) = 106$

**Relative Impedance Tolerance (%):** edit field is to enter the plus/minus percent tolerance acceptable as a result compared to the Theoretical Impedance value on the search line. The Theoretical Impedance is determined by the construction of the relay characteristic using the Define Theoretical For Test tool or using the Add and Edit buttons in the Fault Loops block.

**Absolute Impedance Tolerance (Ohms):** edit field is to enter the plus/minus Ohmic tolerance acceptable as a result compared to the Theoretical Impedance value on the search line. The Theoretical Impedance is determined by the construction of the relay characteristic using the Define Theoretical For Test tool or using the Add and Edit buttons in the Fault Loops block.

**Fault Loops** block is to create and/or edit impedance characteristics for companion Zones.

**Add** button evokes the *Theoretical Control Properties* dialog to add a characteristic or to import RIO files. The Fault Loops selections are made in the Fault Loops: drop list in this Control. The selected Fault Loop: option will appear in the Zones tab Fault Loops block.



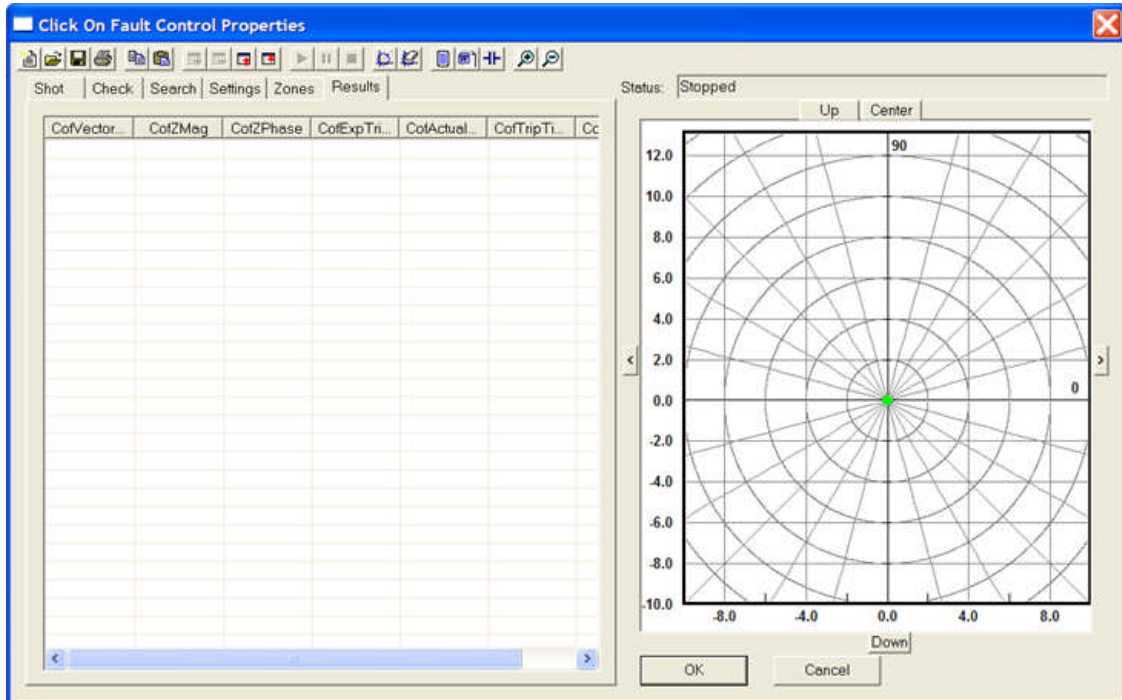
**Edit** is to evoke the Theoretical Control Properties dialog to edit a previously constructed characteristic. The Fault Loops selections drop list in the Control are dimmed and not available for edit.

**Delete** is to select a Fault Loop and delete the characteristic and its Label from the Fault Loops list.

The Graph pane, when clicked, evokes the *Hold Shift or CTRL keys* to create a search line message.

## CoF Results Tab

The *Results Screen* lists the results of either the Shot, Check Tab or the Search Tab execution results. The execution results of one tab will be overwritten by the execution results of another tab and only the results of the last tab will be viewed in the Results Tab.



Run Click On Fault Test execute tool is not active in this tab.

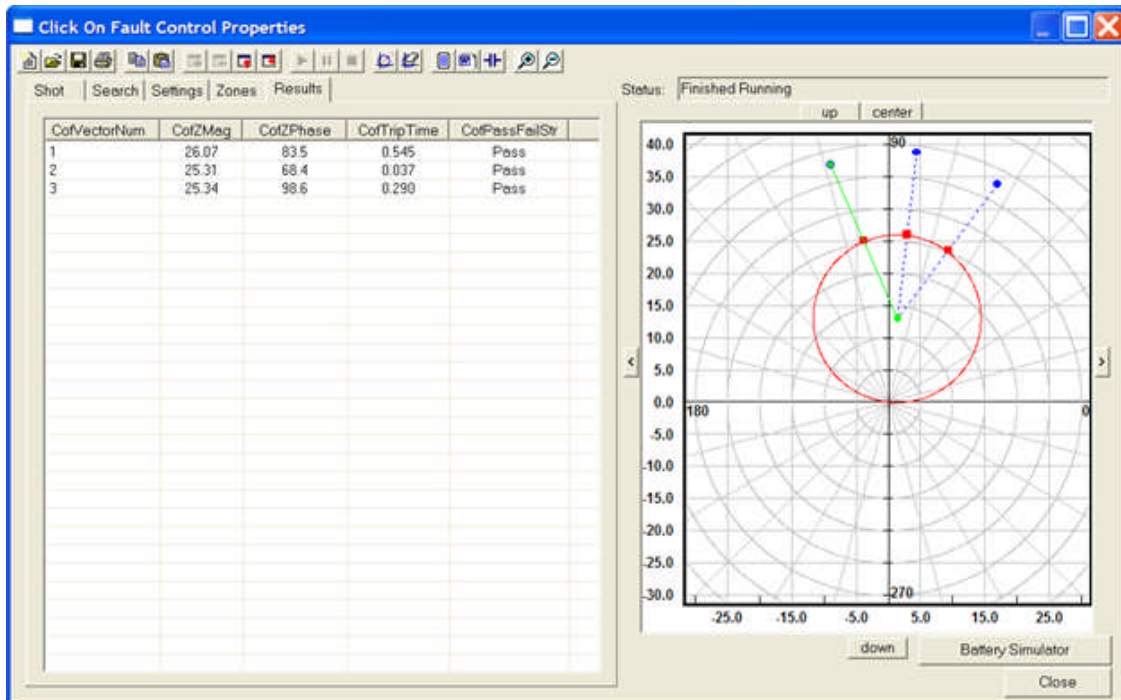
Neither Shot Points nor Search Lines can be constructed in this tab.

The Results are not saved with the save of the constructed Shot, Check and Search Tab test Vectors and Lines.

The Graph pane, when clicked, evokes the *Hold Shift or CTRL keys* to create a search line message. A single Shot Vector for Trip or No-Trip may be created in the Graph, but the Shot Tab must be opened to execute a test. The Shot data will appear as a Vector in the Shot Tab.

The columns in the **Results Tab** data fields are created by using the Define Log Control tool.

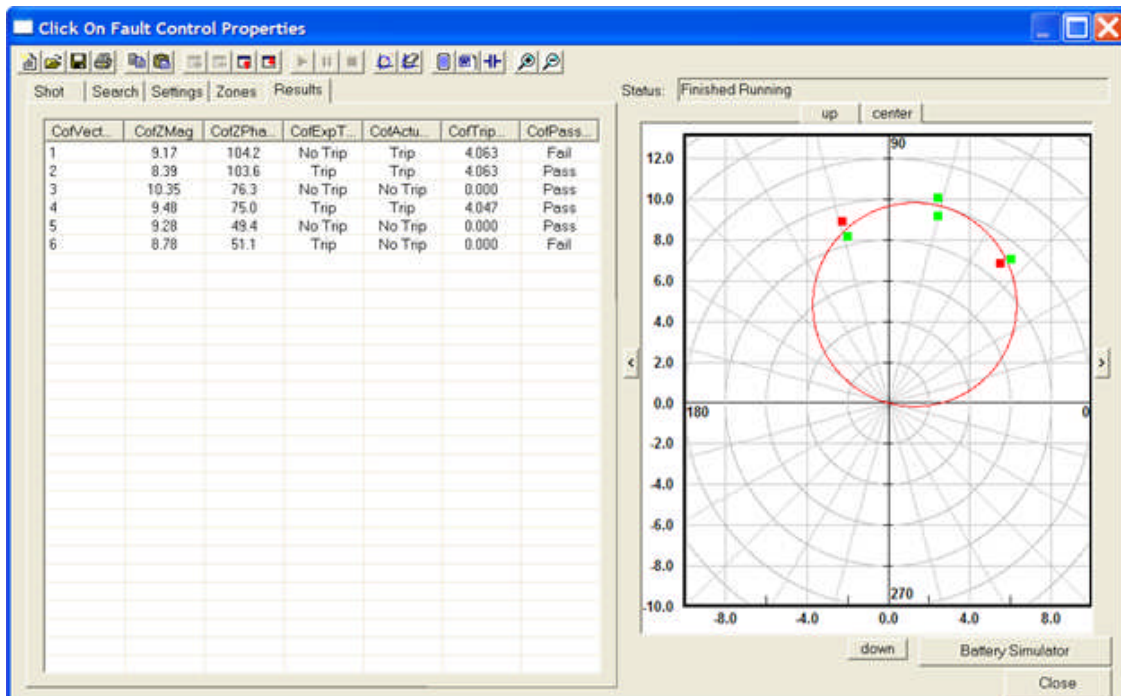
**CofVectorNum** is the row number of data in the *Search Test Result* for the associated search vector Shot or Search test result.



**CofZMag** is the impedance magnitude result for the point or line of Shot or Search test.

**CofZPhase** is the phase angle result for the point or line of Shot or Search test .

**CofExpTripStateStr** is the value of expected *Shot Test Result* of the relay response to a Trip or No-Trip test executed in the Shot tab.



**CofActualTripStateStr** is the resultant relay response for the Shot test point.

**CofTripTime** is the actual time in seconds that the device tripped with the applied fault data in the Shot or Search test.

**CofPassFailStr** will display Pass if the trip was within the tolerance of the protected zone impedance and within the tolerance of expected time delay; else, the display will be Fail in the Shot or Search test.

**Min Pass** = TheoZ – Absol - %Min

**Max Pass** = TheoZ + Absol + %Min



## CoF Example

The object of the **Online Click On Fault example** is to demonstrate a method of constructing an impedance object to test for specific Trip or No-Trip relay responses to selected Fault Types.

Several steps must be taken prior to evoking the Online Click On Fault Control:

**Note:** With Online Click On Fault Control open, access to AVTS Menus and Tools options are not available. AvtsDevDriv.log is not available.

Make all connections to the Test Set and the Relay

Open the Tools/Options.../Devices tab and set enable the communication channels to the Test Set and the Relay under test.

Select Tools/Online Click On Fault Control... to open the dialog.

CoF Example# CoF Shot Test Example

Online Click On Fault (CoF) Shot tests example.

Online Click On Fault (CoF) Search tests example.

Online Click On Fault (CoF) Dynamic Around Theoretical tests example.

CoF opens to the **Shot tab** by default. The procedure to conduct tests in this tab are:

Select the **Fault Type** to conduct

L1-L2 was selected.

Select the **Define Theoretical For Test tool** to open the Theoretical Control Properties dialog.

Create the desired characteristic using the **Mode:** selection

Circle - Max Torque Angle was selected.

Enter the **Characteristic** parameters in the **Data Points** edit fields

Diameter = 26

Max Torque Angle = 84 were entered.

Select the **Fault Loop:** option desired

L-L was chosen for this characteristic to be used with all phase fault pairs which includes a 3-phase fault.

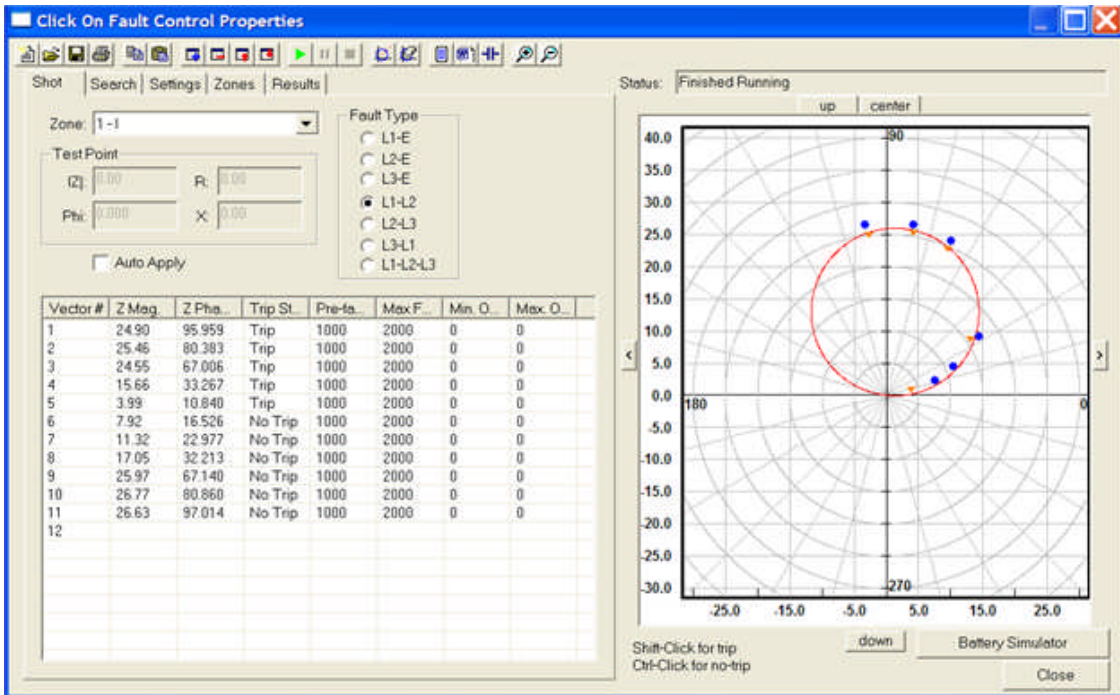
Click **OK** and view the characteristic in the Graph of the Shot tab.

**Note:** None of the phase to ground options display a characteristic.

Use the Shift-Click Graph function to select as many test for **Trip** areas to verify relay Trip inside characteristic executions.

Use the Ctrl-Click Graph function to select as many test for **No-Trip** areas to verify relay No-Trip outside characteristic executions.

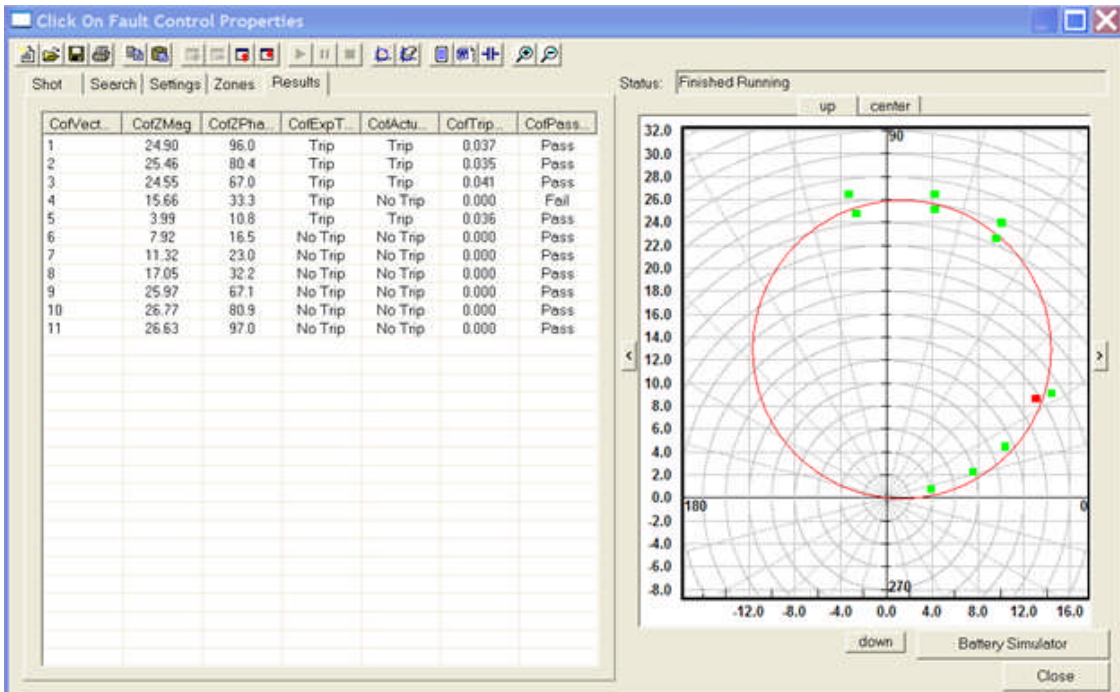
The *Graph display* shows all the constructed tests with **Red triangles** for Trip and **Blue circles** for No-Trip applications.



Note: Selecting for Trip outside and No-Trip inside characteristic executions are valid tests.

Select the **Run Click On Fault Test Tool** to execute a test at each test point created.

The Results tab **Graph display** shows all the test results with **Green squares** for a correct relay response to a Trip or No-Trip and **Red squares** for incorrect responses.



The procedure to conduct binary **Search for characteristic** tests in the Search tab are:

Select the **Search tab**.

Select the **Define Theoretical For Test tool** to open the Theoretical Control Properties dialog.

Create the desired characteristic using the **Mode:** selection

Circle - Max Torque Angle was selected.

Enter the **Characteristic** parameters in the **Data Points** edit fields

Diameter = 26

Max Torque Angle = 84 were entered.

Select the **Fault Loop:** option desired

L-L was chosen for this characteristic to be used with all phase fault pairs which includes a 3-phase fault.

Click **OK** and view the characteristic in the Graph of the Shot tab.

**Note:** None of the phase to ground options display a characteristic.

In the **Zones tab** note that there is a Fault Loop L1-L2-L3 labeled. This is the characteristic just constructed.

Set the following:

**Expected Operation Time (ms):** = 20

**Relative Operation Time Tolerance (%):** = 2.00 ; the default

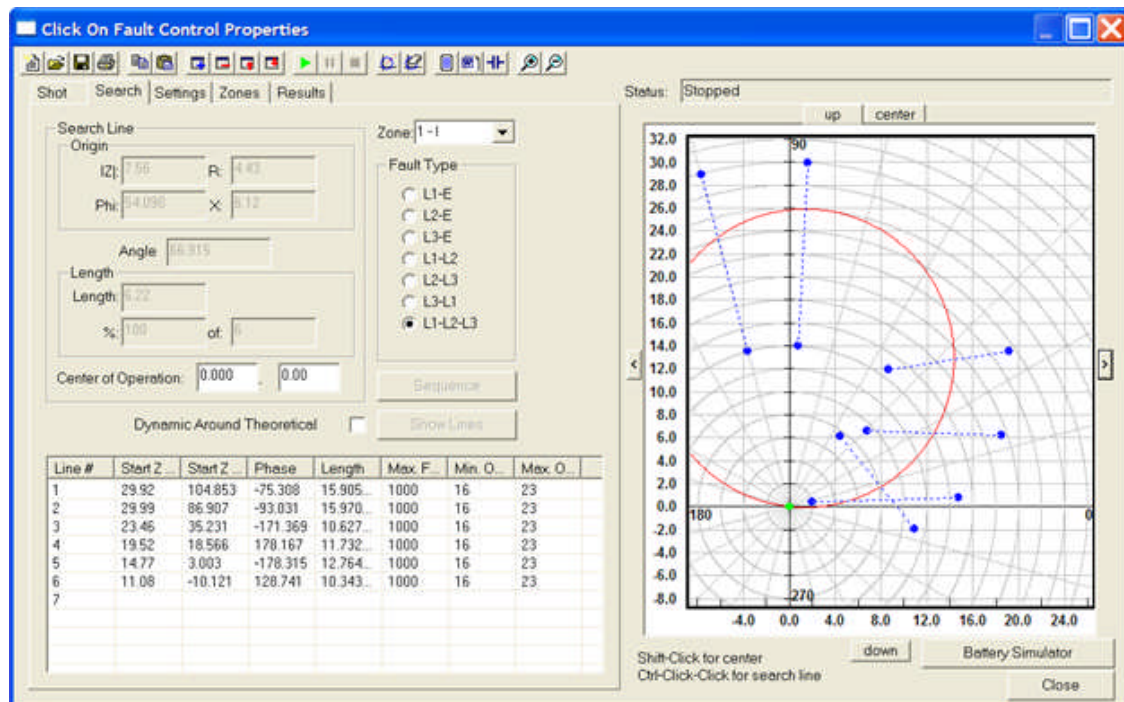
**Absolute Operation Time Tolerance (ms):** = 3

**Relative Impedance Tolerance (%):** =5.0 the default

**Absolute Impedance Tolerance (Ohms):** = 0.5

In the **Settings tab** leave all settings default except edit **Min. Ohm Res.:** = 0.1 and **Max Fault Time:** = 1000 ms

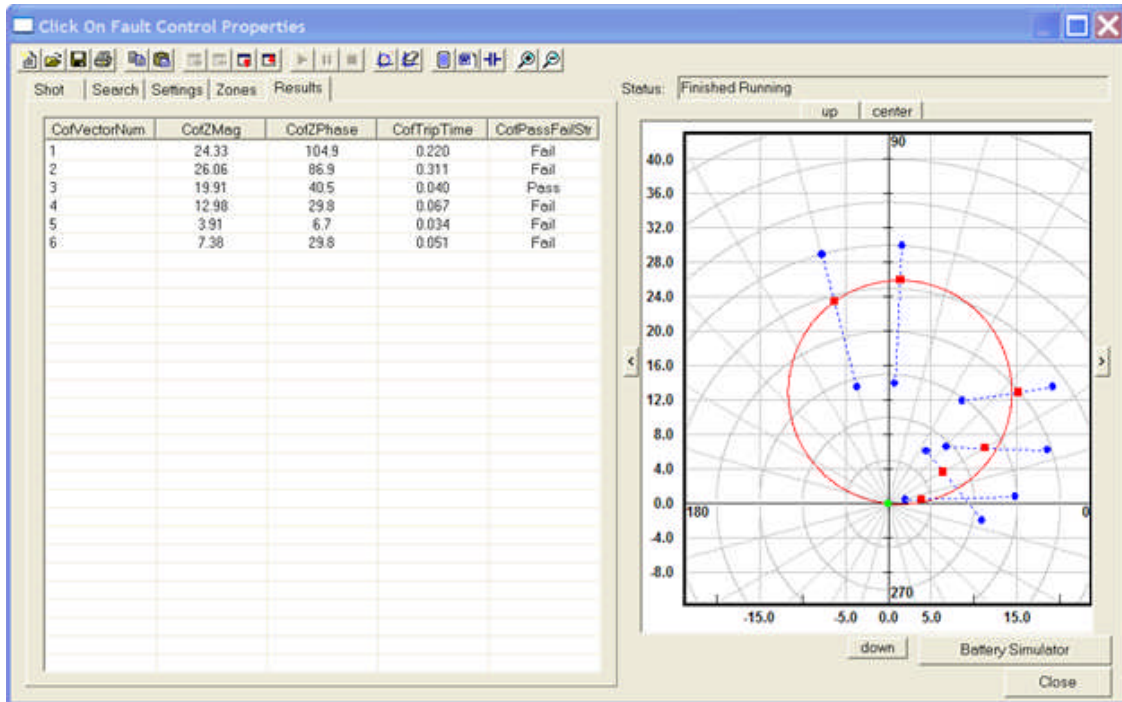
In the **Search tab** Construct the desired tests.



Select the **Run Click On Fault Test Tool** to execute a test at each test point created.

Note: During test execution, a binary search is conducted along the created search vector. The first test is located on the search start point and then the end point is tested. If there is Trip or No-Trip at both ends, the test is aborted. The search must find one end as Trip and the other as No-Trip. Thence, the test is conducted at the half vector, then the next half-of-half between a Trip and a No-Trip, and continuing to narrow in to the characteristic until the Zones and Settings tabs conditions are met.

The Results tab *Graph display* shows all the test results with Red squares for Trip responses. The Pass/Fail is based on the Tolerances set.



Note: The relay was set Diameter = 26 and Max Torque Angle = 84. The test result Vectors 4, 5 and 6 demonstrate that the relay had a Load Encroachment setting of ZLF = and PLAF = 30 and did not meet Ohm tolerance to the mho characteristic constructed.

The procedure to conduct binary Search for characteristic tests **Dynamic Around Theoretical** in the Search tab are:

- Select the **Search** tab.
- Select the **Define Theoretical For Test tool** to open the Theoretical Control Properties dialog.
- Create the desired characteristic using the **Mode:** selection
  - Circle - Max Torque Angle was selected.
- Enter the **Characteristic** parameters in the **Data Points** edit fields
  - Diameter = 26
  - Max Torque Angle = 84 were entered.
- Select the **Fault Loop:** option desired
  - L-L was chosen for this characteristic to be used with all phase fault pairs which includes a 3-phase fault.
- Click **OK** and view the characteristic in the Graph of the Shot tab.

**Note:** None of the phase to ground options display a characteristic.

In the **Zones tab** note that there is a Fault Loop L1-L2-L3 labeled. This is the characteristic just constructed.

Set the following:

- Expected Operation Time (ms):** = 20
- Relative Operation Time Tolerance (%):** = 2.00 ; the default
- Absolute Operation Time Tolerance (ms):** = 3
- Relative Impedance Tolerance (%):** =5.0 the default
- Absolute Impedance Tolerance (Ohms):** = 0.5

In the **Settings tab**, leave all settings default except edit **Min. Ohm Res.:** = 0.1 and **Max Fault Time:** = 1000 ms

In the **Search tab**

Enter a **Center of Operation** = 70 degrees and Center = 20

Check the **Dynamic Around Theoretical** check box

Select the **Sequence** button to open the Sequence Dialog

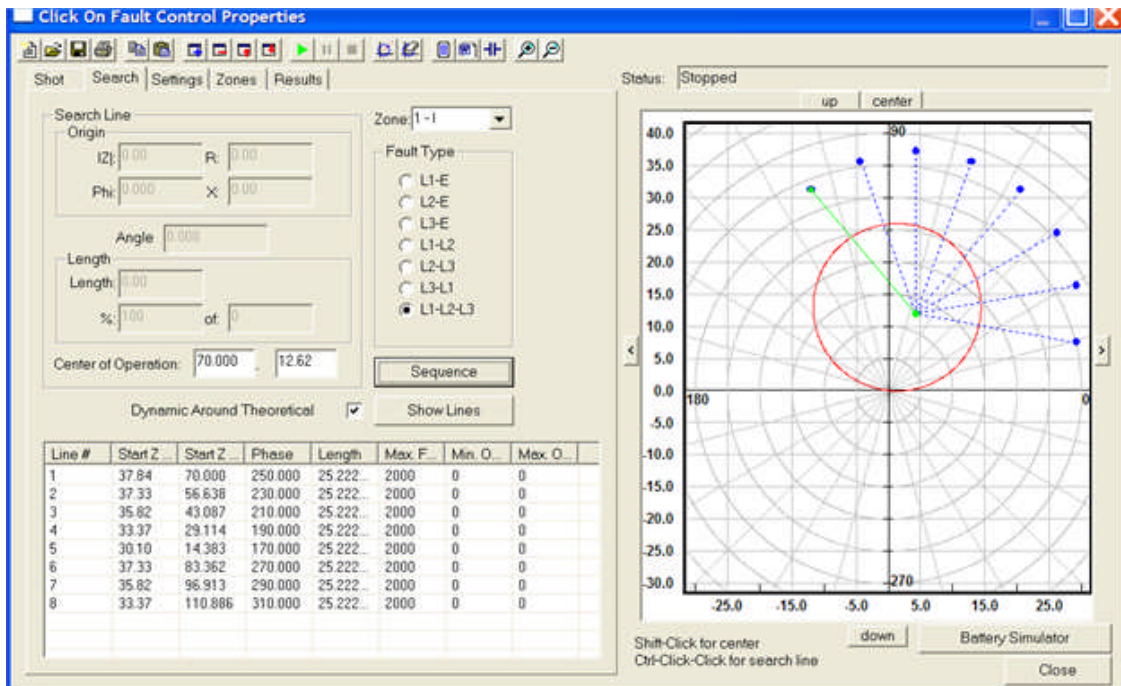
Enter Num Points (CW): = 4

Enter Num Points (CCW): = 3 , the default

Enter Phase Offset:= 20

Select OK

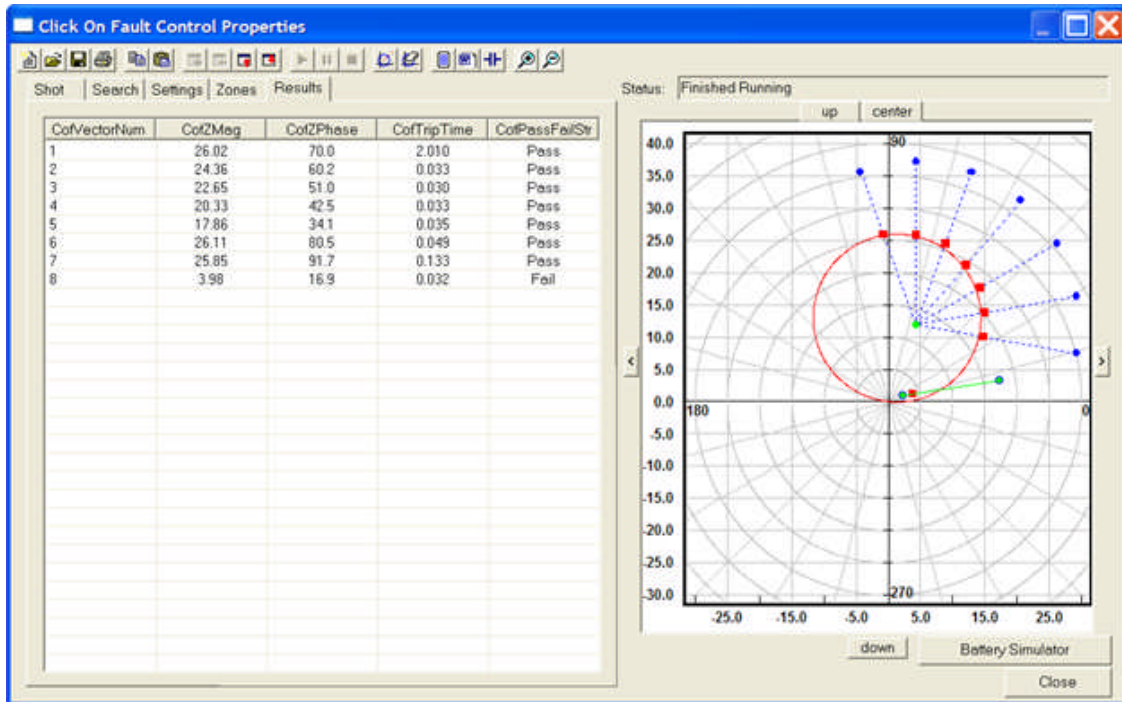
Select the **Show Lines** button to display the automatically constructed *Search Vectors in the Graph*



**Note:** The Center of Operation was automatically edited to 1/2 of the Characteristic chord constructed from the origin to the characteristic maximum Z at the selected Center of Operation Angle. If a Search Vector had previously been constructed, it will no longer be used or displayed.

**Note:** The Dynamic Around Theoretical need not be the Max Torque Angle of the Theoretical Characteristic and that additional Vectors may be constructed separately.

The Results tab *Graph display* shows all the test results with **Red squares** for Trip responses. The Pass/Fail is based on the Tolerances set.



Note: The Vector# 8 is a Fail test. The relay was set Diameter = 26 and Max Torque Angle = 84. The test result Vector demonstrate that the relay had a Load Encroachment setting of ZLF = and PLAF = 30 and did not meet Ohm tolerance to the mho characteristic constructed.

## Wizards

### Wizards

**Wizards** provide a step-by-step method of actions for creating a new **AVTS Relay**, **AVTS Device**, **AVTS Test** or **AVTS SubTest**. The actions are logically oriented using dialogs with informative text to guide in the process.

A **Device** may be created as a sole entity. Future relays may be created using the suitable Device as a parent. A Relay cannot be created unless a parent Device exists.

Each of the products are developed by viewing the topics for each.

New Device Wizard. A Device must exist for a Relay to be created.

New Relay Wizard. A Relay must exist for a Test to be created.

New Test Wizard. A Test may only be created when a Relay has been opened in the Workbook.

New SubTest Wizard.

## New Device Wizard

### New Device Wizard

It is desired that all relays of the same type be consistently tested using the same test procedure. In doing so, accuracy and repetitive results will be achieved.

The **New Device Wizard** will provide a process to create a new Device for new Relays to be added.

**Note:** A Relay cannot exist without first having a Device to define the settings for the Relay.

A Device is a Relay template and becomes the parent for all relays derived from the Device. It is created to define the types of settings that are to be placed in the Device Workbook Settings tab. The Relay is then created from the Device to allow entry of Relay settings in the template made for the Device. The Relay Settings tab is to place the setting values. A Device is created for each relay model and manufacturer. All the Settings for the relay model will be created in the Device that model the relay will be created.

The following topics demonstrate the process:

- New Device


- New Device Settings tab

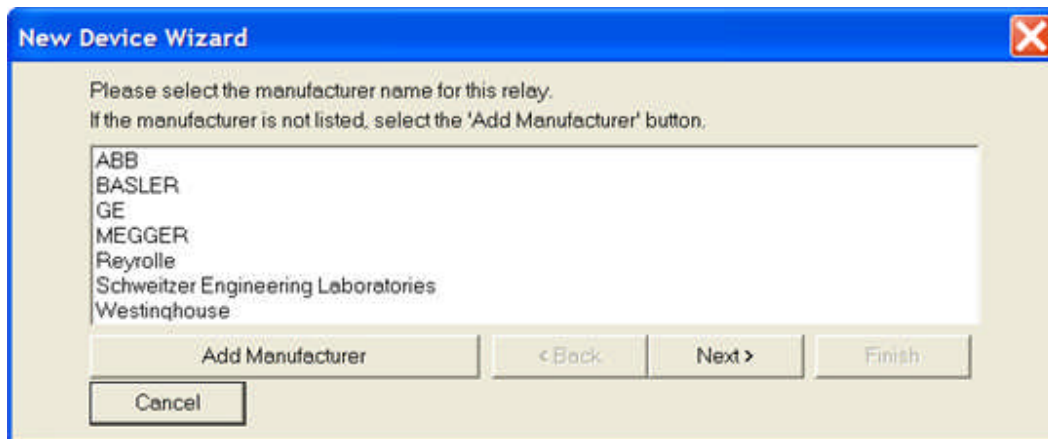
- New Device General Information tab



## New Device

The power of the **AVTS Device** is that all relays in a database will be controlled by the parent Device. If an edit is made to one of several relays having the same parent Device, then all the relays of that Device will receive the same edit in the active database. The exception to edits for a given relay is that the Relay Workbook Settings tab for each relay may have different values for each of the Settings.

The New Device may be created by selecting the File/New.../Device from the main menu. Or the Device may be created by selecting the New Device icon  from the main toolbar. Using either option will display the *New Device Wizard* dialog.

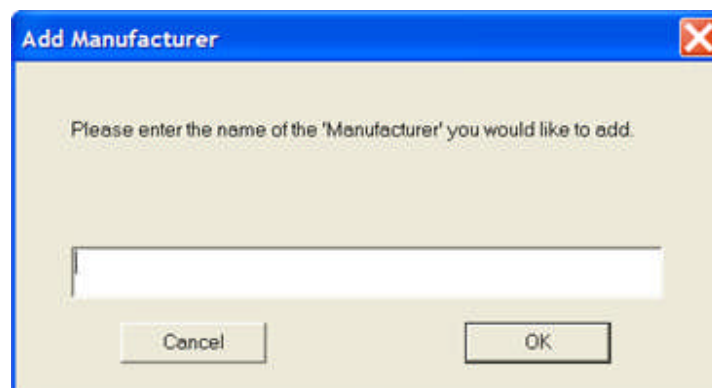


Note: A New Device may be copied in the Worksheet Devices tab, pasted and then renamed in the tab. It will be a new parent to new relays and may have less or additional Settings in the Workbook Settings tab.

The procedure is:

Open the New Device Wizard.

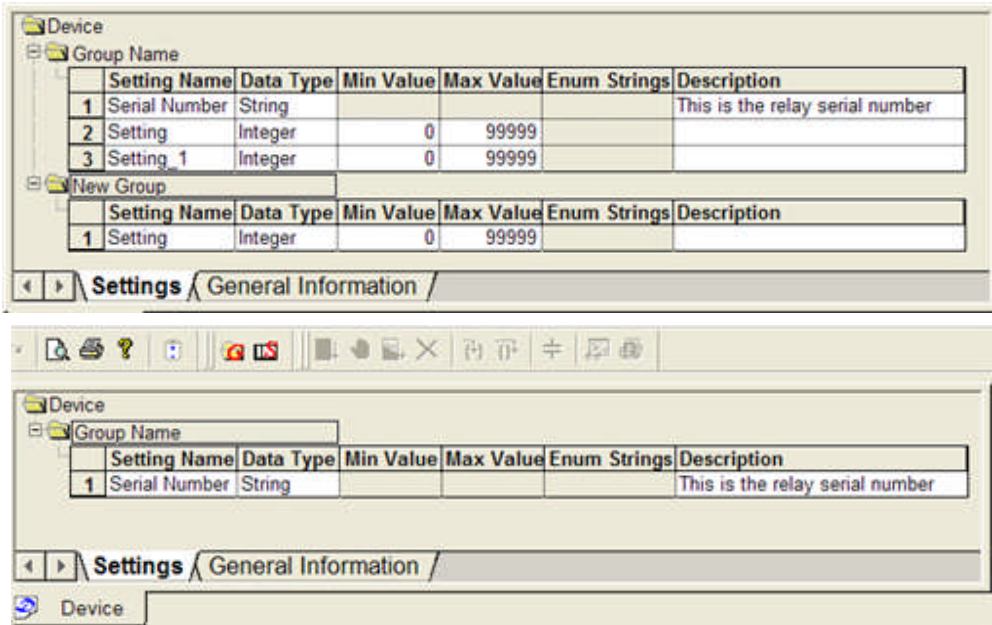
Either select an existing Manufacturer or select the Add Manufacturer button to create a *New Manufacturer* by name. When entered, select the Finish button.



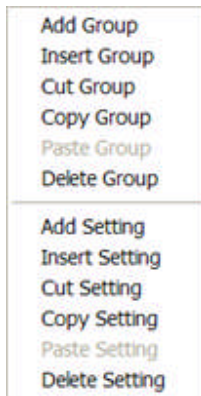
The *New Device created* with the previously assigned name will be displayed in the Workbook. It will be active for adding additional tests and identifying the necessary information for each setting. The



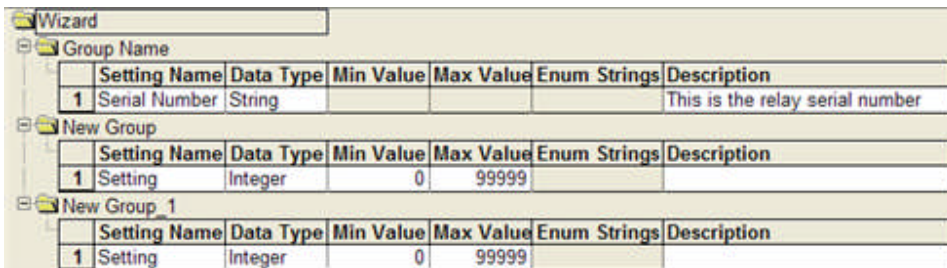
for adding additional "blank" groups and settings will also be made active in on the main toolbar. A *new group or setting* will be added with each click of the respective tool. Add Group and Add Setting options are detailed below.



A *Settings drop down menu* is displayed with a right click on any of the settings columns value fields.



The **Add Group** option will add a *New Group* every time it is selected. The first added group to the default group is labeled *New Group*. Each successive group added is labeled *New Group* with a numeric suffix beginning with the number 1.

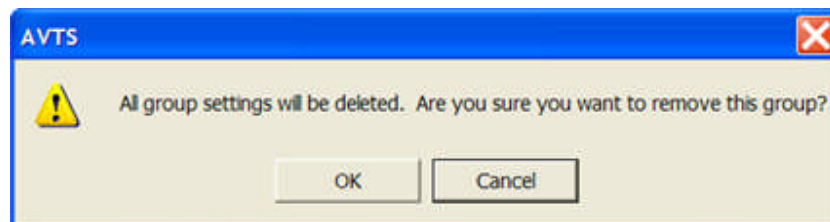


**Note:** Edit Group Name with a left mouse select to position cursor for edit.

The **Insert Group** option will add a *New Group* above the group in which the mouse displayed the drop down menu. The new group will be labeled *New Group* with a numeric suffix beginning with the number 1, each successive group having a label *New Group* numerically increased by 1.

Wizard						
Group Name						
	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Serial Number	String				This is the relay serial number
New Group 2						
	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Setting	Integer	0	99999		
New Group						
	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Setting	Integer	0	99999		

The **Cut Group** option will display the delete *All group settings* message with Ok and Cancel options. Cancel will return to the Workbook with no change to Groups or Settings. OK will remove the Group selected and will retain all settings in a clipboard to Paste Group or to Insert Group in the group order location desired.



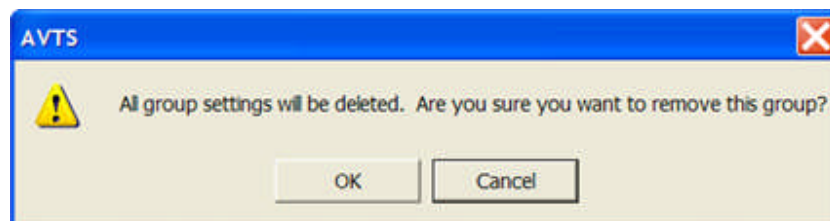
The **Copy Group** option will place a copy of the entire selected group into a clipboard to Paste Group or to Insert Group in the group order location desired. The cloned group will be labeled with the label of the *copied group* with a numeric suffix beginning with the number 1, each successive group having a label New Group numerically increased by 1.

Wizard						
Group Name						
	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Serial Number	String				This is the relay serial number
New Group 1 2						
	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Setting	Integer	0	99999		

Note: if the copied group label had a suffix 3, the cloned group label will have the suffix 3\_1 for the first clone, and will have each successive group having a label New Group numerically increased by 1.

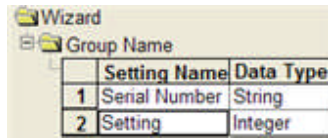
The **Paste Group** option will add a new group below the group in which the mouse displayed the drop down menu. The new group will be labeled New Group with a numeric suffix beginning with the number 1, each successive group having a label New Group numerically increased by 1.

The **Delete Group** option will display the *All group settings* message with Ok and Cancel options. Cancel will return to the Workbook with no change to Groups or Settings. OK will remove the Group selected and will retain all settings in a clipboard.



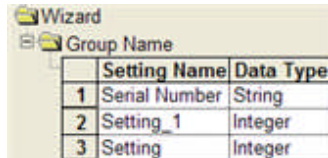
Note: recovery of the deleted group is by selecting the Edit/Undo menu option to restore the group to the original state prior to delete. The Paste Group option is dimmed and the Insert Group will insert a new group only.

The **Add Setting** option will add a *New Setting* every time it is selected. The first added setting to the default setting is labeled Setting\_1. Each successive setting added is labeled "Setting\_x" with a numeric suffix beginning with the number 1.

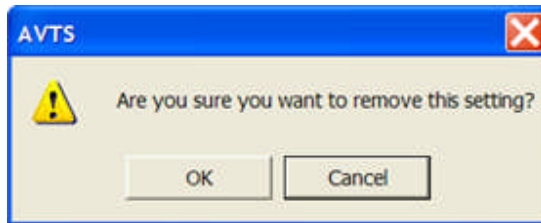


Note: Edit Setting Name with a left mouse select to position cursor for edit.

The **Insert Setting** option will add a **New Setting** above the group in which the mouse displayed the drop down menu. The new group will be labeled New Group with a numeric suffix beginning with the number 1, each successive group having a label New Group numerically increased by 1.



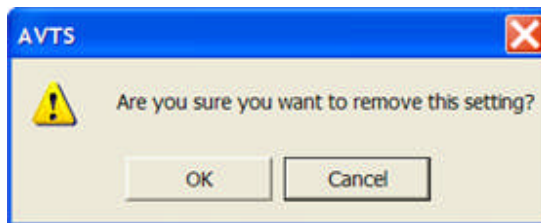
The **Cut Setting** option will display the delete **...remove this setting?** message with Ok and Cancel options. Cancel will return to the Workbook with no change to Groups or Settings. OK will remove the Group selected and will retain all settings in a clipboard to Paste Group or to Insert Group in the group order location desired.



The **Copy Setting** option will place a copy of the selected setting into a clipboard to Paste Setting or to Insert Setting in the setting order location desired. The cloned setting will be labeled with the label of the copied setting with a numeric suffix beginning with the number 1, each successive setting having a label New Setting numerically increased by 1.

The **Paste Setting** option will add a new setting above the setting in which the mouse displayed the drop down menu. The new setting will be labeled Setting\_x\_y with a numeric suffix beginning with the number 1, each successive group having a label Setting\_ numerically increased by 1.


The **Delete Setting** option will display the **...remove this setting?** message with Ok and Cancel options. Cancel will return to the Workbook with no change to Groups or Settings. OK will remove the setting selected and will retain setting in a clipboard.

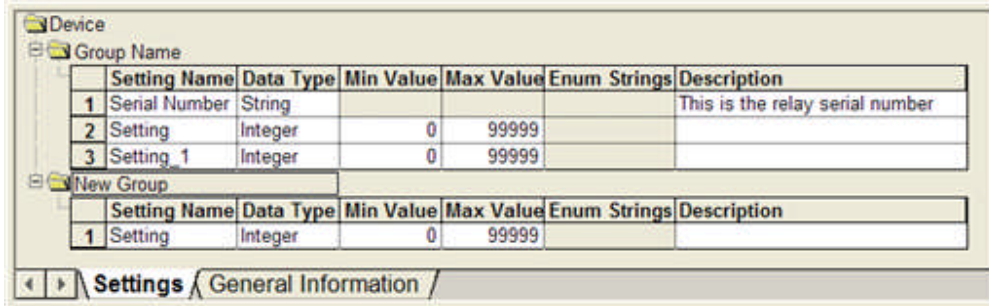


Note: recovery of the deleted setting is by selecting the Edit/Undo menu option to restore the group to the original state prior to delete. The Paste Setting option is dimmed and the Insert Setting will insert a new setting only.

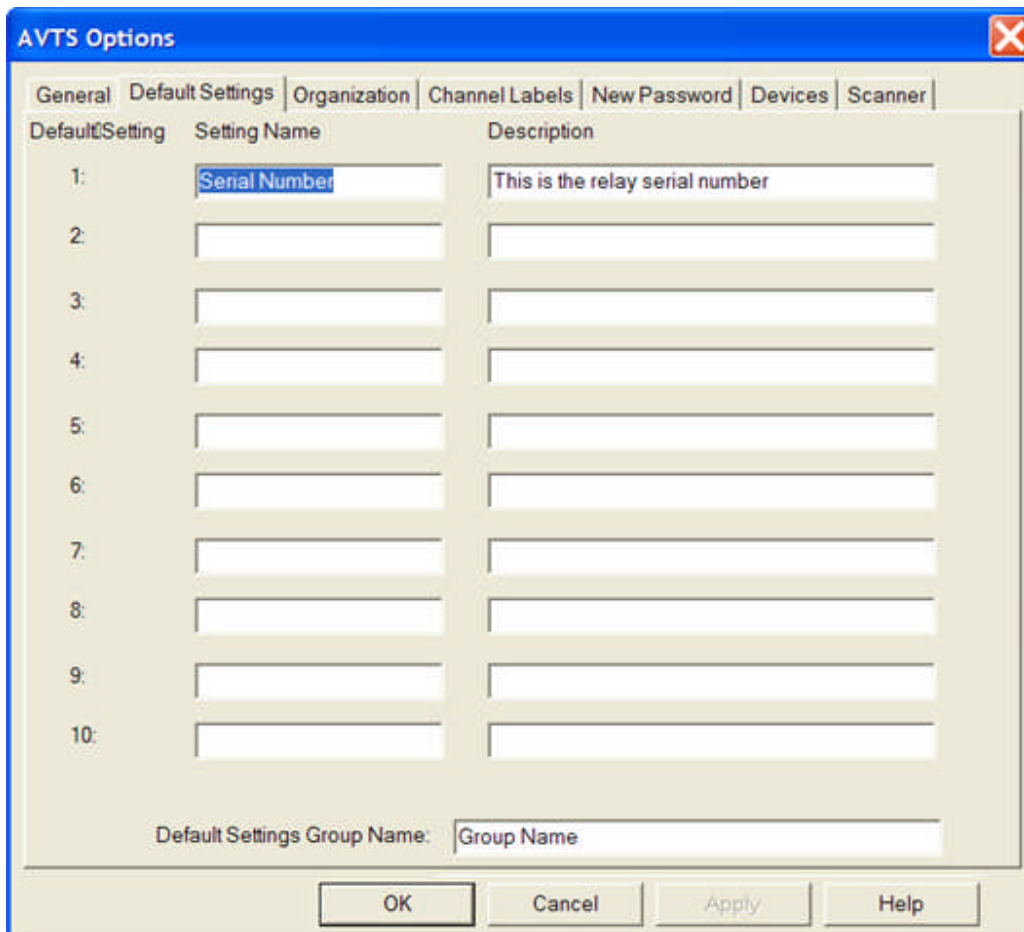
Note: Multiple Settings may be selected those desired while holding the CTRL key.

## New Device Settings

The **New Device Settings** tab displays the number of settings equivalent to the minimum number of settings required to successfully test the relay. Settings are added by clicking the  tool bar icons.



The Group name and single setting displayed in the initial Settings tab are the default labels governed by the **Tools/Options/Default Settings** tab from the main menu. The Group label may be edited by left mouse selection. The Setting label may be edited by left mouse selection.

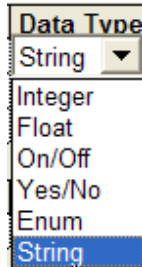


Note: The Setting in the first Group is the only setting name that matches the default label. Each new setting is labeled in numerical sequence as it is created.

Note: The Group name in the first Group is the only group name that matches the default label. Each new group is labeled in numerical sequence as it is created.

The **Settings columns** are labeled to identify the different settings values.

The *Data Type* may be changed dependent upon the type of setting value to be used in testing the relay. A right click in one of the Data Type edit fields produces a drop down menu of the different option types. The selected data type will determine the availability of the other columns for value entry.



Note: The Setting Name, Data Type and Description columns are available for edit for all Data Types. The Description column is a text only entry for all settings created.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Serial Number	String				This is the relay serial number

The **String Data Type** indicates that the Setting Name is a String value and will be treated as text. In some relays, the relay setting may be text or a numeric value. Both the text string and numeric value can be used by conditioning the string in Calculation in the Test Editor tab of an open relay. If the data is a String data type and is text, the text may be used as a condition for branching during a test procedure; such as, data types *Pass Fail* for two different branches. If the data is a String data type but must be used as a numeric value, the Expression `val(Data_Type$)` converts a number string to a number.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Inst	Float	45.0	160.5		This is the relay Instantaneous se

The **Float Data Type** indicates that the Setting Name is a float decimal number and may be used directly in mathematic calculations and relay setting values. The numeric now has the Min Value and Max Value columns for entry of the minimum and maximum numeric values available for setting the relay. The float characteristic is indicated by the decimal placed in the Max Value column.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Plug	Integer	10	100		This is the relay Plug value

The **Integer Data Type** indicates that the Setting Name is a numeric integer with no decimal and may be used directly in mathematic calculations and relay setting values. The numeric now has the Min Value and Max Value columns for entry of the minimum and maximum numeric values available for setting the relay. The integer characteristic is indicated by the whole numeric placed in the Max Value column.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Serial Number	On/Off				This is the relay serial number

The **On/Off Data Type** indicates that the setting is a string value. Only the On or Off option may be used and sets a conditional branch during a test procedure. Typically the branch decision is predicated by a condition met upon entering the Control containing the Setting.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	Serial Number	Yes/No				This is the relay serial number

The **Yes/No Data Type** indicates that the setting is a string value. Only the Yes or No option may be used and sets a conditional branch during a test procedure. Typically the branch decision is predicated by a condition met upon entering the Control containing the Setting.

	Setting Name	Data Type	Min Value	Max Value	Enum Strings	Description
1	ICS	Enum			0.2.2.0	This is the relay ICS value

The **Enum Data Type** indicates that there will be an enumeration of values to select for the setting to be used. The values may be a series of floating, integer or text values. The values may be strings or numeric or a combination, but must be handled the same as the previously described data types. Typically, the values will be entered, comma delimited, with no spaces. The usefulness is to enter discrete values of settings; e.g., tap settings.

The **Description** column allows text entry to describe the intent of the Setting Name, setting limits and perhaps String Data Type information.

## New Device General Information

The *General Information tab* is for the entry of information specific to the specific model of a manufacturer's relay.

The **Device:** entry will be the name of the device taken from name in the Worksheet Devices tab provided when the Device was created.

The **Manufacturer:** entry will be the name of the manufacturer taken from name in the Worksheet Devices tab provided when the Device was created.

The **IEEE Device:** edit field is for the user to enter the IEEE code for the use the relay is to provide. A list of the IEEE Device Numbers are listed in the topic.

The **Device Communication Software Link:** edit field is to locate the specific software program to be used to communicate to the device under test. The software allows commands and setting changes to be transferred directly to the device during a test procedure. This feature may be a direct command from the test procedure or may incorporate a complete script file normally used with a SEL or other microprocessor relay. A button is available to immediately locate the particular file and its location. When selected, the path to the file will display in the edit field.

The **Use Custom Report** check box, when checked, will make visible another edit field labeled "**Custom Report File Name:**" along with a browse button for locating the specific file. Typically, the custom report will be created for use for a specific relay or database. The software required to create the custom report is Crystal Reports.



The **Device Notes**: edit field is for entry of information either copied from another source or typed directly into the field. The field is usually used to provide information about the particular model of relay represented by the Device.

Note: A new or blank line may be created by holding the CTRL key and pressing the ENTER key.

## New Relay Wizard

## New Relay Wizard

The **New Relay Wizard** is to create a new relay which will be of a different model to any other relay in the database. To create a new relay, a New Device is created using the New Device Wizard. The initial steps using the New Relay Wizard automatically begins by selecting an existing Device or to create a New Device.

Note: The copy, paste and renaming an existing Device allows the New Relay Wizard to select the "new device" and create a relay very similar to an existing relay.

The following topics demonstrate the Wizard:

- New Relay

- New Relay Manual Test

## New Relay

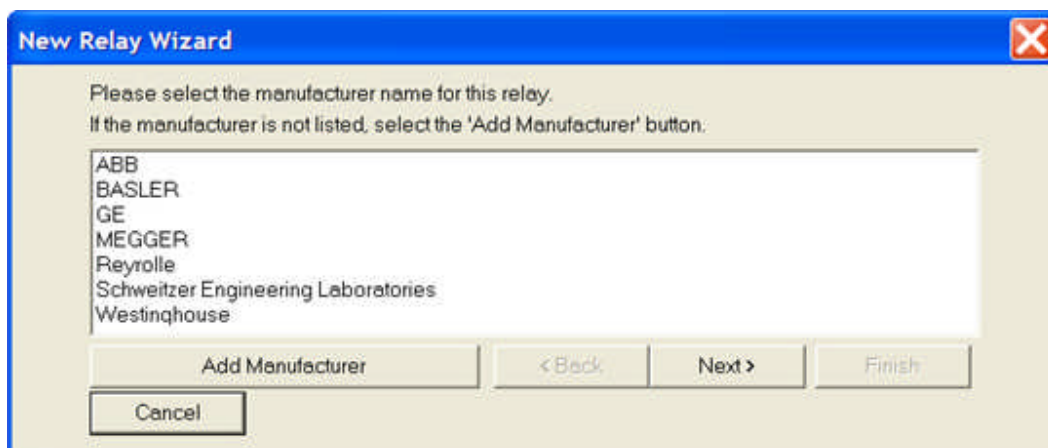
The **New Relay Wizard** is one method of creating a new relay in an active database. The method begins by selection of an existing Device or by creation of a New Device. Another method is by copying an existing relay in the Worksheet Relays tab and pasting that relay in another location, and then editing the relay name is desired.

**Note:** any changes to the Workbook Settings or Editor tabs will be reflected to all other relays in the database that are of the same parent Device.

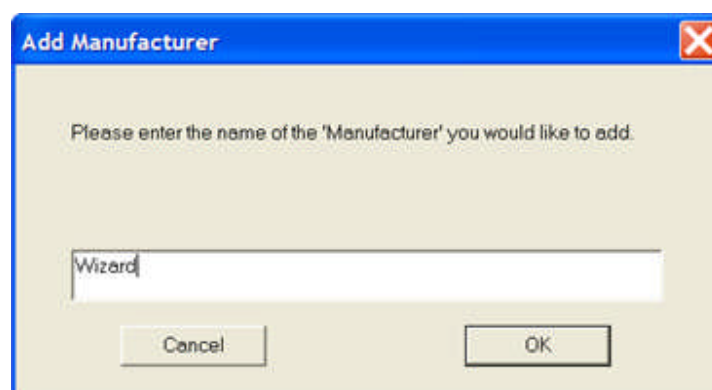
A new AVTS relay may be created in several ways. One method is to copy an existing relay and modifying it. This is a popular method as it uses the existing parent Device and allows renaming the copied relay, modify the settings and immediately conduct tests to a different relay of the same type.

The New Relay Wizard provides a step-by-step process to create a relay and provides the opportunity to create a new Device in the process.

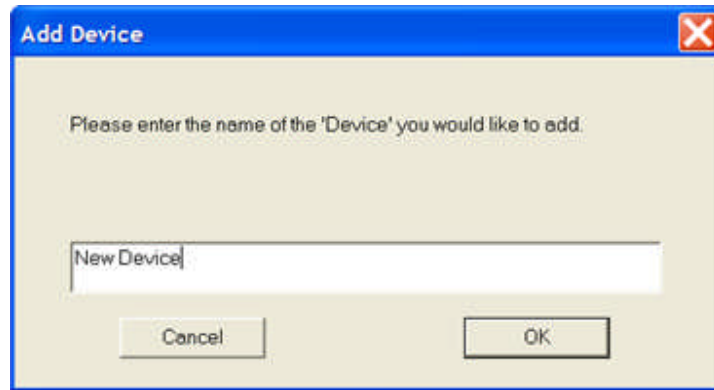
The *New Relay Wizard* proceeds through the dialogs to select an existing manufacturer or Add Manufacturer. Select Finish.



The *Add Manufacturer* opens a dialog to type the name of the manufacturer of the new relay. Select OK.



The **Select Device** dialog is to select an existing Device or add a new Device. Select *Add Device*. Type the name for the Device and select OK.



**Note:** select an existing device will produce a "clone" of any relays present in the database.

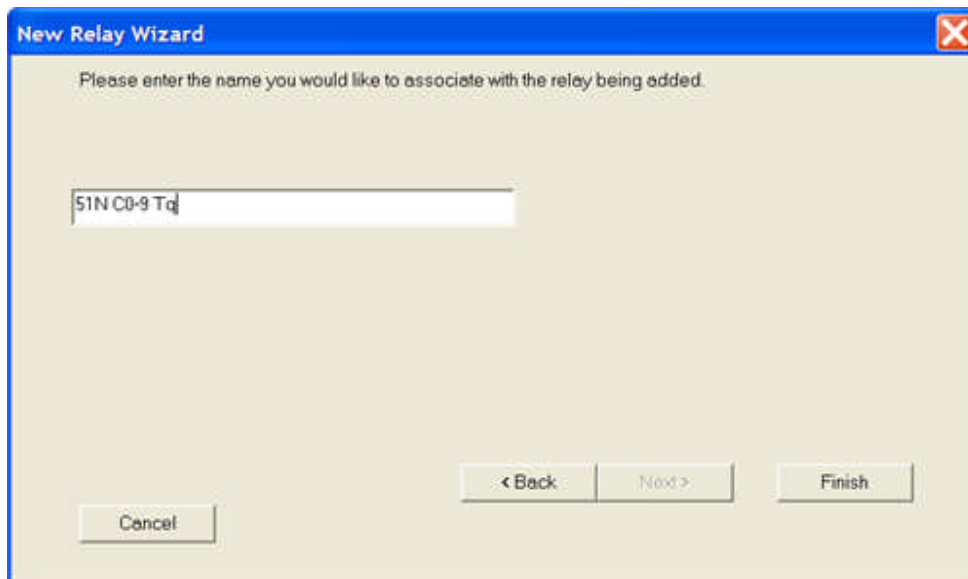
The following dialogs, examples from the default Organization, will select existing or create new location names for the placement of the new relay:

The **Select or Add Region** dialog is to select an existing Region location or Add Region. Select Next, or OK if Add Region.

The **Select or Add Substation** dialog is to select an existing Substation location or Add Substation. Select Next, or OK if Add Substation.

The **Select or Add Line** dialog is to select an existing Line location or Add Line. Select Next, or OK if Add Line.

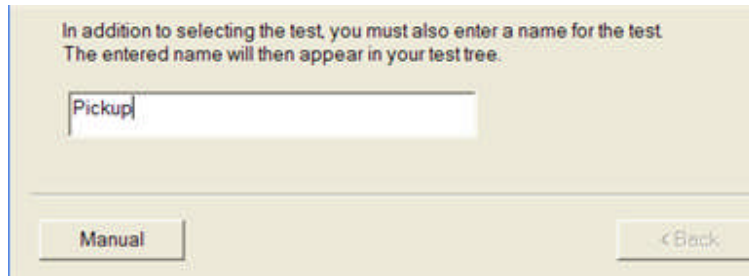
The new *Relay Association* dialog is to name the relay for recognition in the location. If the new relay is placed in an existing location, the name will differentiate it from the other relays both there and perhaps in other locations. Select Finish.



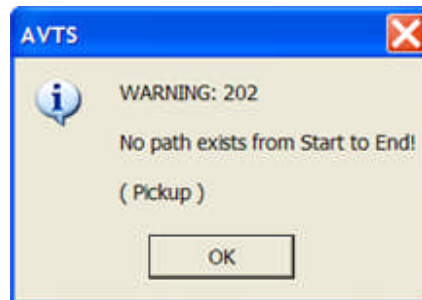
**Note:** if an existing Device was initially selected, Finish will open the new relay in the Workbook with a single test open.

The **New Test Wizard** is opened to place a Test Name for the test tree for the relay. The New Test Wizard is for the selection of existing generic relay test to populate the relay test tree. Type a name to associate the test to the new relay.

Type a *Test Name* and select Manual to place a blank test in the Workbook relay test tree open to the Settings tab.



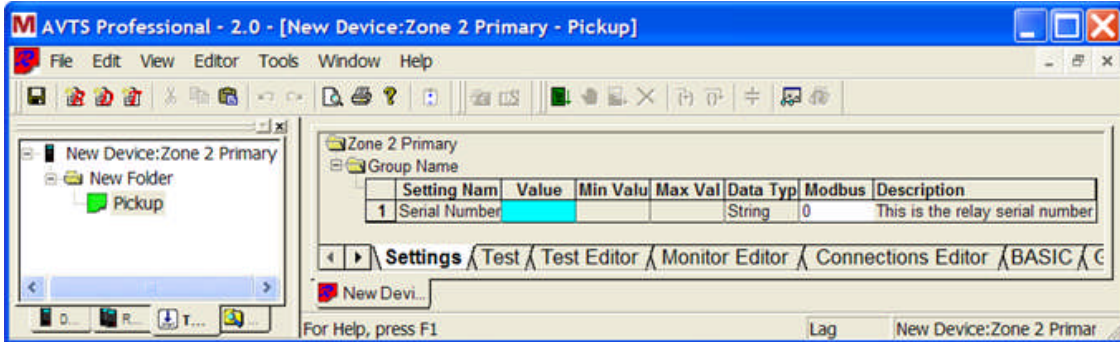
The *No path exists* message appears indicating no test has been developed in the Test Editor tab. The relay test is now ready to be created.



The New Relay Manual Test is NEXT.

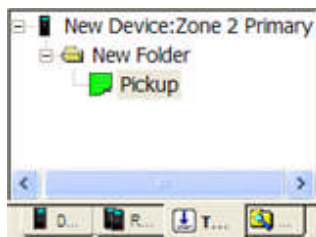
## New Relay Manual Test

The following details the new relay test created by the *New Test Wizard Manual* option.

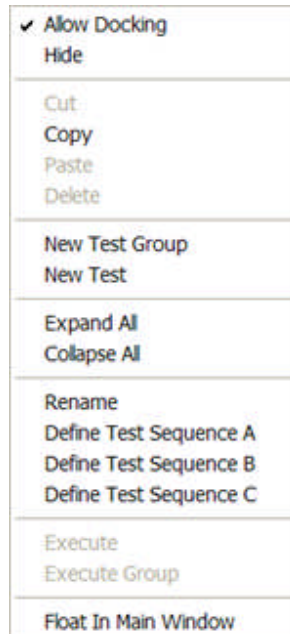


The AVTS title bar displays the name of the New Test and the Workbook is open to the new test. In this example the title DeviceName:RelayName - TestName is [New Device:Zone 2 Primary - Pickup].

The *Worksheet Tests* tab displays the parent Device Name and Relay Name given in the Relay Test Wizard. The DeviceName:RelayName in this example is "New Device:Zone 2 Primary". The test folder name is "New Folder". The test name is "Pickup".



Edits in the Worksheet Tests tab are from a *drop down menu* displayed with a right mouse click in the Worksheet.



## New Test Wizard

### New Test Wizard

It is desired to create a new relay test which will be of a different procedure to any other relay in the database. To create a new relay test, a New Device and a New Relay will be required. The initial steps using the **New Test Wizard** will create the New Test.

The New Test may be created by following the use of the New Test General topic.

Methods to create a New Test are:

New Test Copy an existing Test in the Workspace test tree and paste it where desired. It may then be edited and modified to fit the new test application.

For New Test Manual, evoke the New Test Wizard, enter a test name in the edit field and left mouse click the Manual option button.

For New Test Generic, evoke the New Test Wizard, select a relay type from the list of generic types, enter a test name in the edit field and select the Next option button.

## New Test Copy

A new test for a relay may be very similar to a test that has been constructed in the past and available in the active or other database. A copy of that similar test may easily fit the new relay by simply editing the Workbook Relay Editor tabs.

This method is complementary to the New Test Generic method.

An example of this easy method is to locate the similar test in an open relay.

Select the test from the Workspace Tests tab test tree. If the test is to remain in the same relay or database, right mouse click on the test to place a copy of the test into the clipboard. Locate the new or other relay and highlight the location to place the copied test.

Right mouse click and Paste the item from the clipboard to the relay Tests tab test tree.

Any changes necessary to make the test variables agree to the Workbook Settings tab Setting Name and Value columns may then be made.

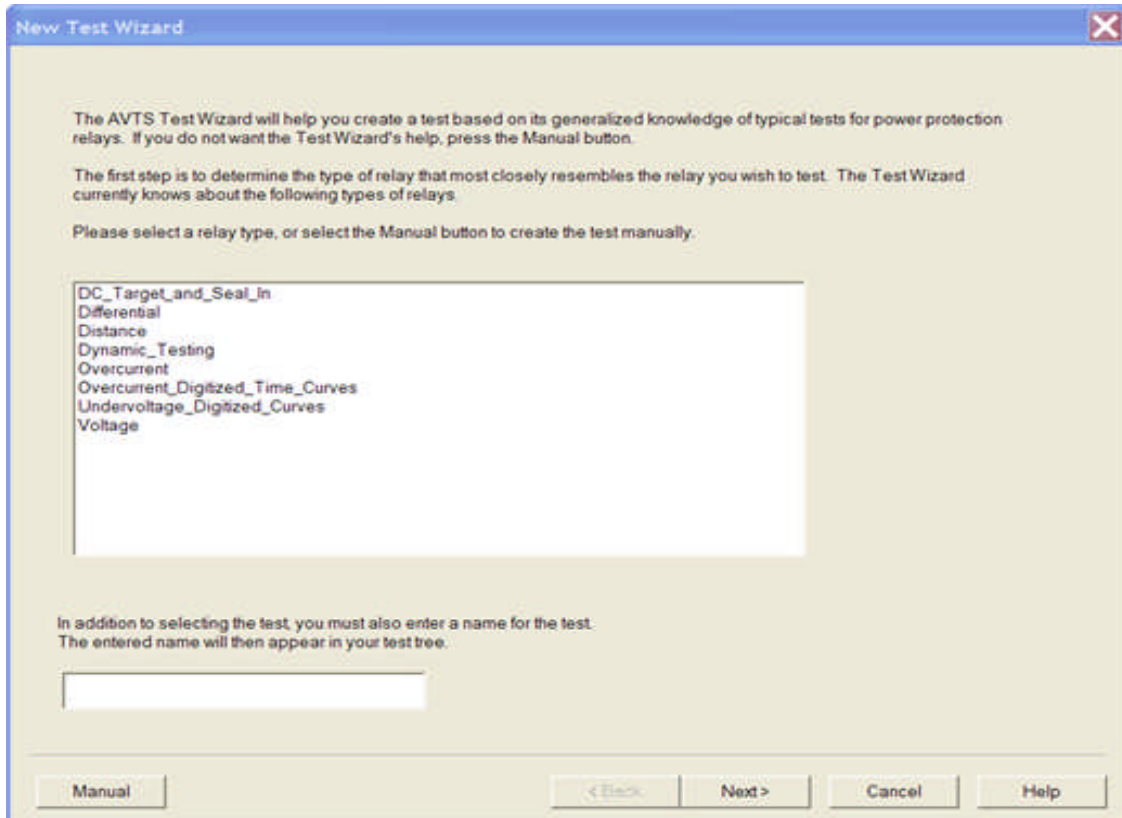
Place AVTS in simulate mode with the Tools/Options/Devices and execute the test to troubleshoot the new test.

Usually this will require less test construction effort than building a new test from the very beginning.



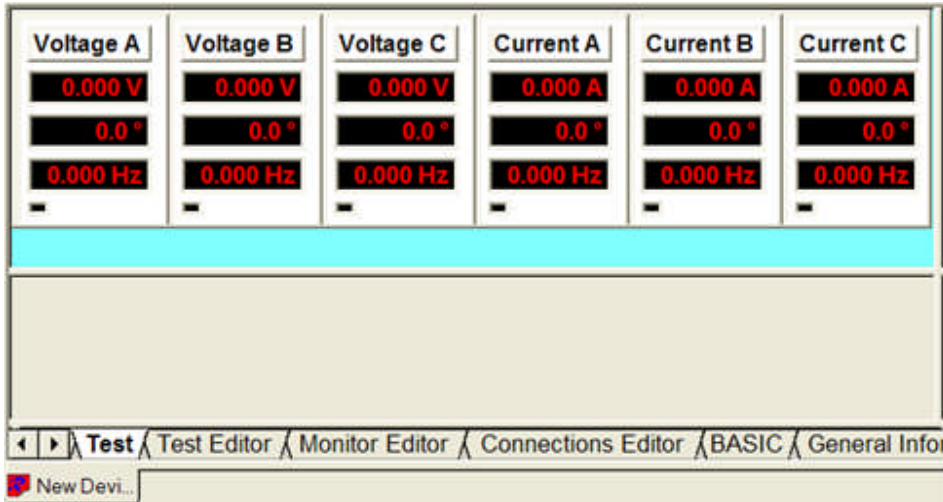
## New Test Manual

Open the *New Test Wizard*, enter a test name in the edit field and left mouse click the Manual option button. The new test is immediately opened in the Workbook with the default settings in the Settings tab and default Monitor and Test tabs. The New Test will be void of a test procedure and the Test Editor tab will contain only Start and End Controls with no Link Control.

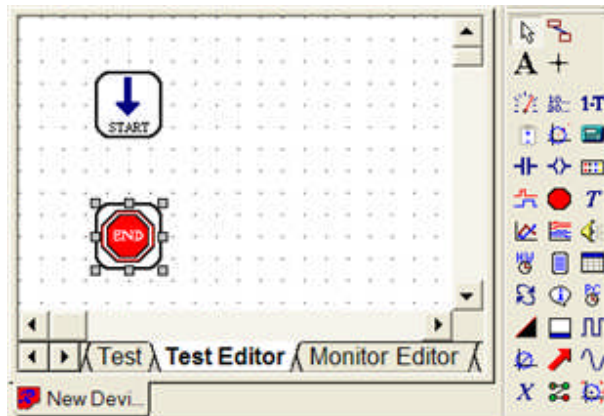


The Workbook Settings tab reflects the number and labels of the defaults resident in the Tools/Options/Default Settings tab. The Settings may be changed or edited within the open parent Device. Definitions and use of the Settings tab are located in the Definition - Settings Tab topic.

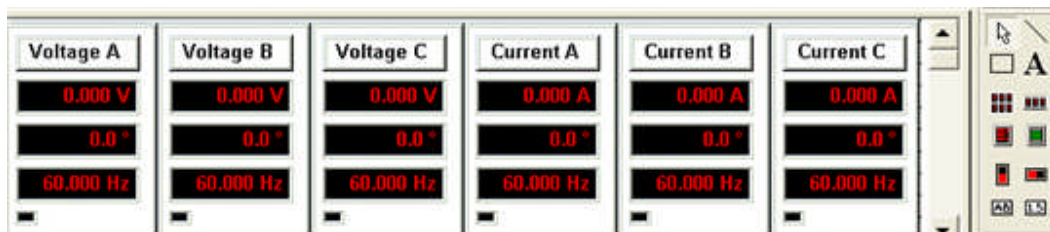
The *Workbook Test tab* upper tile displays the default monitors for test set amplifiers. The lower pane is for display of the test results from an executed test.



The *Workbook Test Editor tab* displays the pane to construct the test procedure. For a new test created from the Manual button of the New Test Wizard, the test contains only the Start and End Controls with no Link Controls. The Controls toolbar is displayed to the right of the edit pane, readily available for selecting the Controls necessary to create a test procedure. The definition and use of each Control is located in the Test Editor Toolbar topic.



The *Workbook Monitor Editor tab* displays the monitor tools used for the Monitor pane in the Workbook Test tab. The tools for use in the Monitor Editor are to the right of the pane. Definitions and use of each tool is located in the Monitor Editor Tab Menu topic.



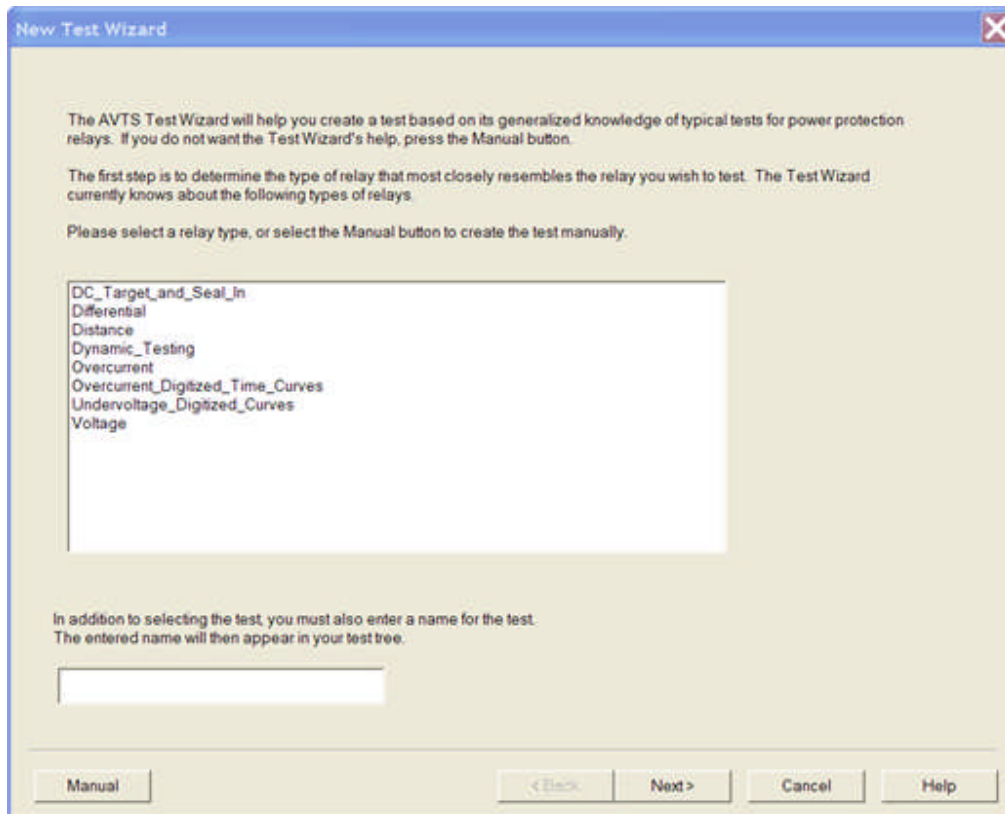
The Workbook Connections Editor tab displays the connections tools to the right of the pane to create test connections from test device to device tested. Definitions and use of each tool is located in the Connections Editor Toolbar topic.

The Workbook Basic tab is defined in the Basic topic.

The Workbook General Information tab is defined in the Relay General Information topic.

## New Test Generic

The *New Test Wizard* provides many tests that are generic in their application and generally applicable to any relay test requirement. Evoking the New Test Wizard displays a selection of generic relay types. The relay types are typical and represent a family of tests specific to the relay type.



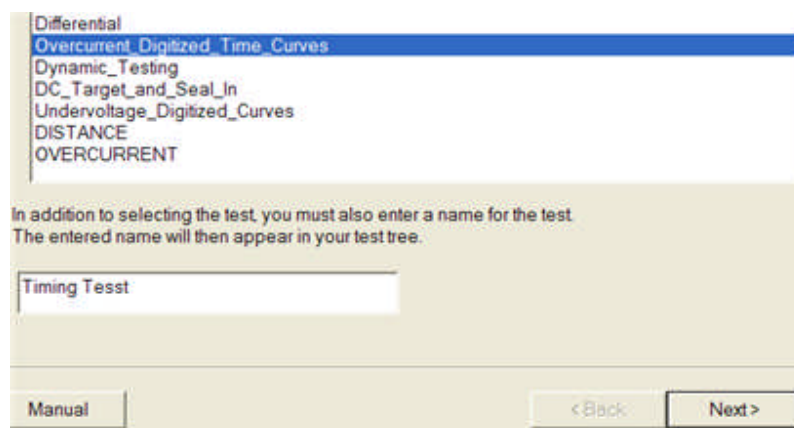
To create a new generic test

Open a Relay or create a new relay using the New Relay Wizard.

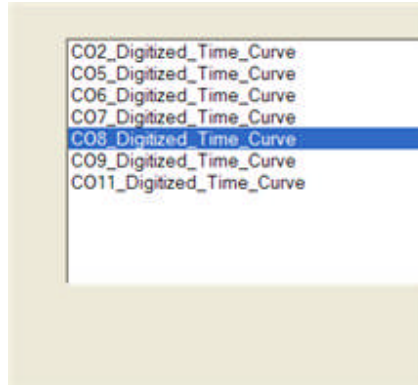
Open the New Test Wizard

Select a relay type similar to the type test to build.

*Enter a name* for the test in the edit field at the bottom of the dialog.



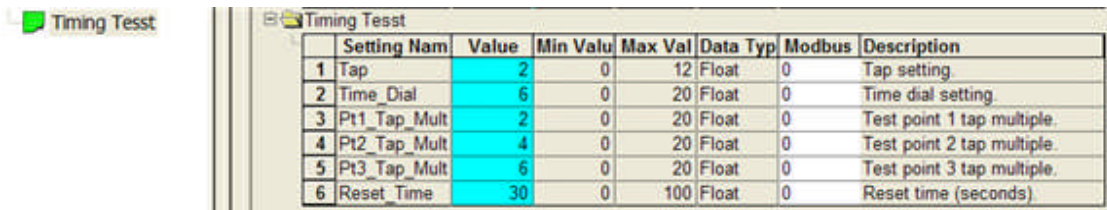
Click Next to select the *Test type* to be used.



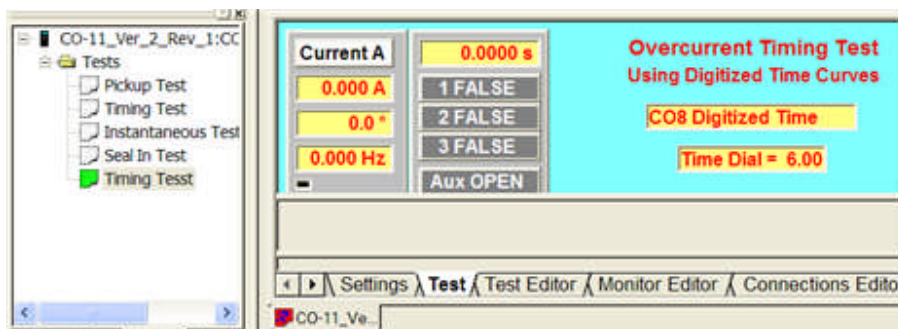
Click Next to view the *new test settings* for edit or import to the relay.

	Setting Name	Value	Min Value	Max Valu	Data Type	F	Description
1	Tap	2	0	12	Float		Tap setting.
2	Time Dial	6	0	20	Float		Time dial setting.
3	Pt1_Tap_Mult	2	0	20	Float		Test point 1 tap multiple.
4	Pt2_Tap_Mult	4	0	20	Float		Test point 2 tap multiple.
5	Pt3_Tap_Mult	6	0	20	Float		Test point 3 tap multiple.
6	Reset_Time	30	0	100	Float		Reset time (seconds).

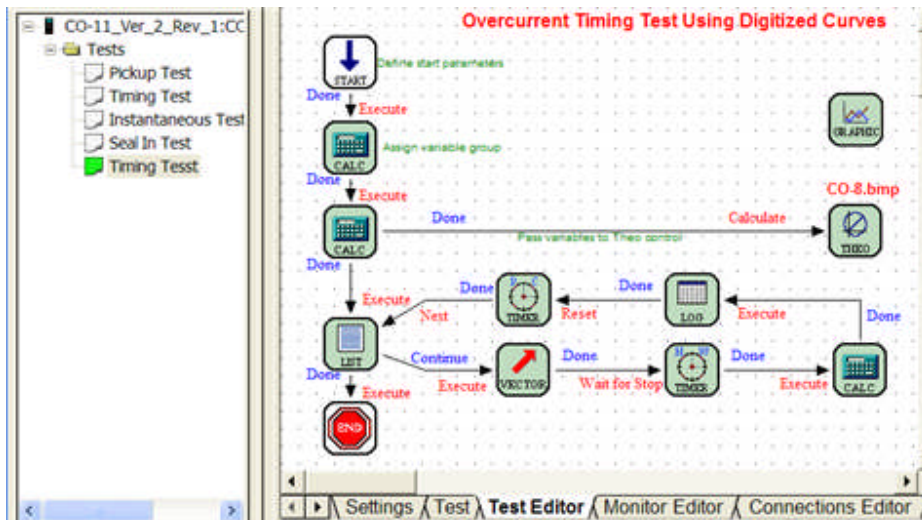
The New Settings Group will be placed into the Workbook Relay *Settings tab*.



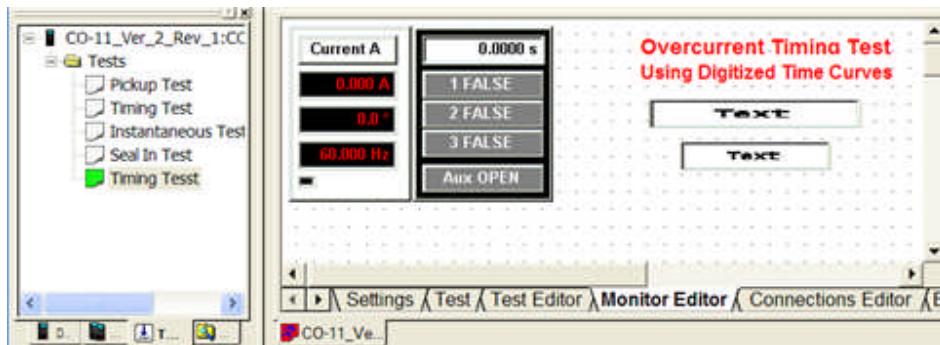
The New Settings Workbook *Test tab*.



The New Settings Workbook *Test Editor tab*.



The New Settings Workbook *Monitor Editor* tab.



The relay types and their respective test types

- Overcurrent*
- Overcurrent\_Pickup*
- Overcurrent\_Timing*
- Instantaneous\_Overcurrent*
- Voltage\_Polarized\_Dir-OC\_Max\_Torque*
- Current\_Polarized\_Dir\_OC\_Max\_Torque*

**Distance**

- Single\_Phase\_Reach\_I*
- Single\_Phase\_Reach\_V*
- Three\_Phase\_Reach\_I*
- Three\_Phase\_Reach\_V*
- Open\_Delta\_Reach\_I*
- Single\_Phase\_Max\_Torque\_I*
- Single\_Phase\_Max\_Torque\_V*
- Three\_Phase\_Max\_Torque\_I*
- Three\_Phase\_Max\_Torque\_V*
- Open\_Delta\_Max\_Torque\_I*
- Single\_Phase\_Characteristic\_I*
- Single\_Phase\_Characteristic\_V*
- Three\_Phase\_Characteristic\_I*
- Three\_Phase\_Characteristic\_V*
- Open\_Delta\_Characteristic\_I*
- Warm\_Up*

**Voltage**

*Over\_Voltage\_Pickup*  
*Under\_Voltage\_Pickup*  
*Over\_Voltage\_Timing*  
*Under\_Voltage\_Timing*

**Differential**

*Harmonic\_Restraint*  
*Slope*

**Overcurrent\_Digitized\_Time\_Curves**

*CO2\_Digitized\_Time\_Curve*  
*CO5\_Digitized\_Time\_Curve*  
*CO6\_Digitized\_Time\_Curve*  
*CO7\_Digitized\_Time\_Curve*  
*CO8\_Digitized\_Time\_Curve*  
*CO9\_Digitized\_Time\_Curve*  
*CO11\_Digitized\_Time\_Curve*

**Dynamic\_Testing**

*Two\_State\_Dynamic\_Test*  
*Three\_State\_Dynamic\_Test*  
*Four\_State\_Dynamic\_Test*

**DC\_Target\_and\_Seal\_In**

*Seal\_In*

**Undervoltage\_Digitized\_Curves**

*CV1\_Digitized\_Curve*

Note: Some tests for one relay may be the generic test for another relay type. The Overcurrent pickup test may be used for a differential relay pickup.

## **How to Create Test Procedure Example**

## **How to Create Test Procedure Example**

This topic will concentrate about the thought and tool processes to create a test. The creation of an AVTS test will be through the use of the Controls in the Workbook Test Editor tab.

Build a typical Pickup Test.

Examples of other test types and use of Controls are in the topics

- Test Editor OC Time Example 1

- Test Editor Charc Z Example 2

- Test Editor Microprocessor Time OC Example 3 using formulae for Theoretical curves.

- Test Editor Differential Example 4

## Pickup Test

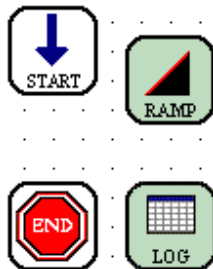
The thought process to create an **Overcurrent Pickup Test** follows the pattern "What comprises the test and what do I use to construct it?" First there must be settings available for the relay in the Settings tab.

Usually "Tap" is used as the variable name for the relay setting. The desire is to determine if the relay reacts properly to the application of test set amplifier outputs connected to the relay.

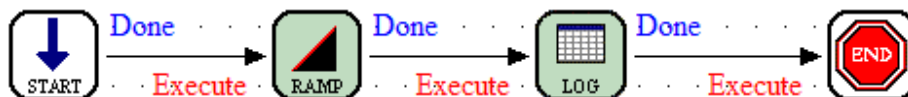
1. The test begins with the *Start and END* Controls resident in the Workbook by default. All that is required is to fill in the Controls in between and connect them in the order of application using the Link Control.



2. The amplifier(s) output must begin at a value below the set value and Ramp up until the relay operates. When the relay operates, the resulting value needs to be captured by the Log Control for saving in the database. Therefore, the *Ramp and Log Controls* are placed in the Workbook.

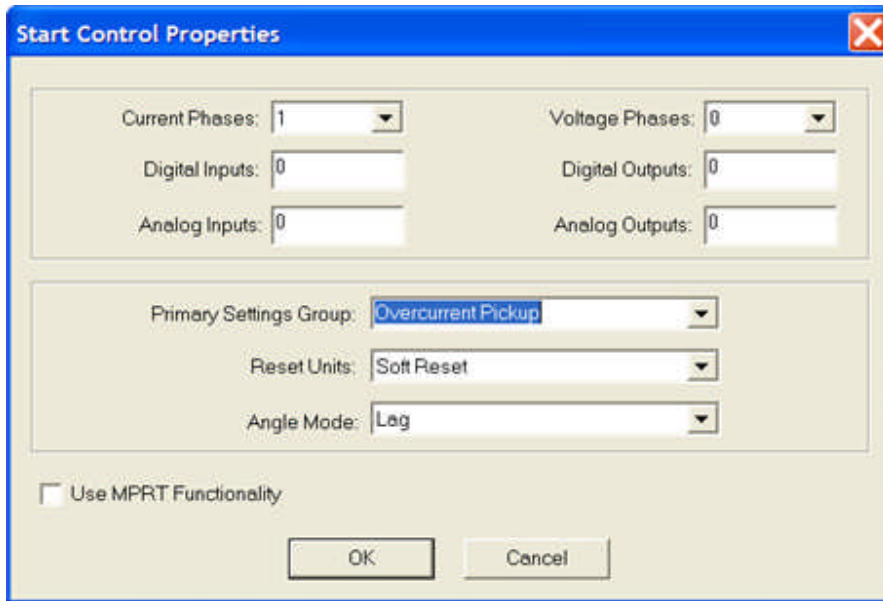


3. *Linking the Controls* for processing will dictate the order of the test procedure. From the Start, a Ramp Control will be used to control the beginning current value, the continuous or pulsed current ramp, the monitoring of relay contact operation and the maximum value to allow ramp for test abort should the maximum be reached. A recognized value of operation will then be passed to the Log Control to save the value for data print. The Log Control may also contain a number of items for relay settings and result comparison to the expected value. The Link default properties are placed with the construction of the Link and are shown as Done for the Source and Execute for the Target.

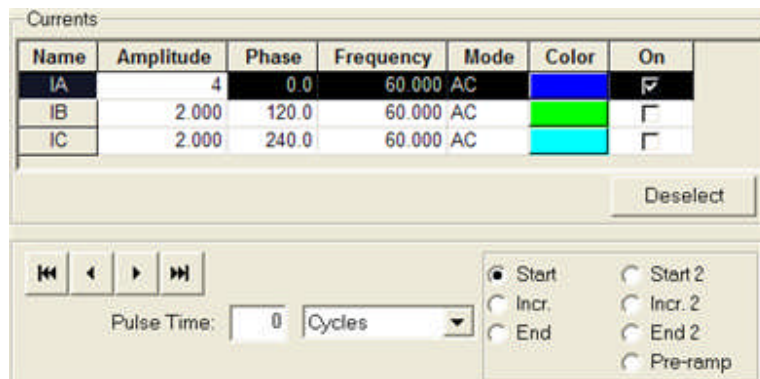


4. The *Start Properties dialog* is opened to set the Current Phases for use to one (1) and the Primary Settings Group to the Overcurrent Pickup group present in the Settings tab.





5. The Ramp Properties dialog is opened to *set the Start value*. The relay is set to pickup at 5 amps, so the Ramp begins at a lesser value; four (4) in this case.



Note that instead of a real value the start value may be a variable; e.g., 90% of tap or "0.9 \* Tap".

The IA amplifier On column is checked so that the amplifier turns on at the beginning of the ramp. The Pulse Time is left at zero (0) Cycles so that the ramp is a continuously increasing value.

The value of each increasing current ramp is determined in the *Incr. dialog*.



Note: The incremental Ramp UP or DOWN is determined automatically by AVTS. If the End value is greater than the Start value, the Ramp will be increasing in value. If the End value is less than the Start value, the Ramp will be decreasing in value.

## 4 Example Tests

## 4 Example Tests

The listed example topics demonstrate the use of the **Test Editor Controls** in test procedures.

Test Editor OC Time Example 1

Test Editor Char Z Example 2

Test Editor Microprocessor Time OC Example 3

Test Editor Differential Example 4

Note: Each example is chosen to demonstrate most of the Controls available. Controls duplicated in a succeeding example are not discussed if they have been discussed in a previous Example. I.e., The Log Control is explained in Example 1 but not in Example 2 or others.

## Test Editor OC Time Example 1

### Test Editor OC Time Example 1

The object of Example 1 is to demonstrate a method of constructing a **Time Overcurrent Timing Test for an electro-mechanical relay**. The example also includes the insertion of the Instantaneous setting and time test for the Instantaneous.

**Note:** The Time OC element and the Instantaneous elements must be connected in series for the test to provide time test results for both elements.

**Note:** All other example overcurrent time tests in the supplied databases do not time test the Instantaneous elements, but perform a separate pickup test.

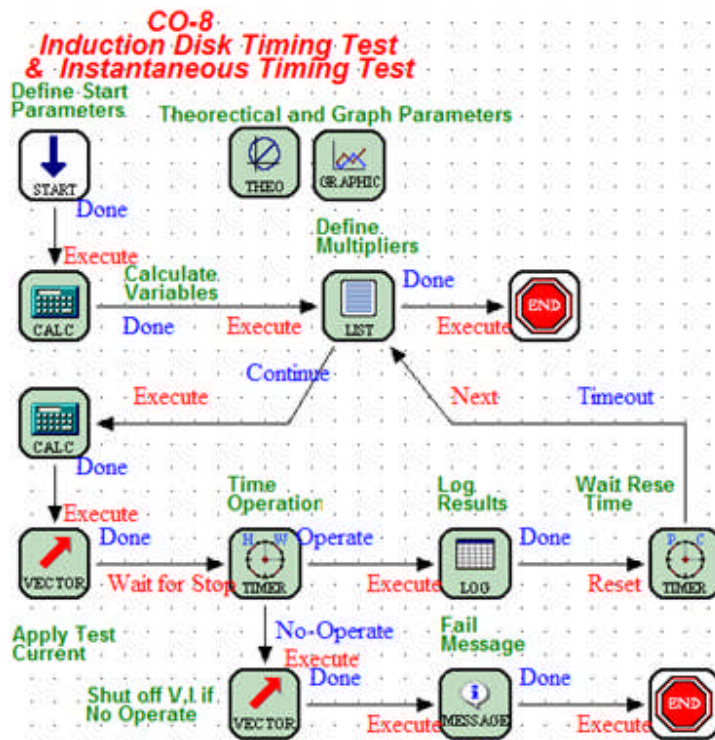
The Controls have been chosen to provide a test with results compared in the Log Control directly to original manufacturer's curves having been digitized in the Theoretical Control.

The List Control is used to "list" the values of tap multiple to be applied for each of the timing tests.

Expanded definitions and details for each of the Controls are given in its own topic in the Test Editor Toolbar book.

Example 1 is taken from the AVTS RelayTestLibV2.mdb CO8 relay. This database is available with the original installation of AVTS Ver. 2.x.

The *example test* is viewable in the Workbook Test Editor tab. The IAC Timing test is chosen for explanation of use of the following:



- Start Control
- Calculation Control
- Theoretical Control
- Graphic Control
- List Control

Vector Control

HW Timer Control

Log Control

PC Timer Control

End Control

Test Results

See How to Create Test Procedure Example

## Start Control - Example 1

The *Start Control Properties* provide the use of Phase A current, Settings Group from the Settings tab from which variables in use will take their values, Provide a soft reset

**Current Phases: 1** 'Only one current channel is to be used and by default will be channel A of the test set.

**Voltage Phases: 0** 'None required.

**Digital Inputs: 0** 'None required.

**Digital Outputs: 0** 'None required.

**Analog Inputs: 0** 'None required.

**Analog Outputs: 0** 'None required.

**Primary Settings Group: IAC 51 Settings** 'The group of settings in the Workbook Settings tab from which values for necessary variables are taken.

**Reset Units: Soft Reset** 'Full test unit reset to default values.

**Angle Mode: Lag** 'The Lag mode for all angles to reference zero degrees in the lagging direction. This is the default mode for PULSAR.

**Use MPRT Functionality Unchecked.**

The **Link** from Start to Calculation Controls displays Source = Done, indicating that the reading of the settings will constitute the only operation for the Start Control. The Target = Execute indicates the action to immediately begin processing settings, variables and equations in the Calculation Control.

**Calculation Control NEXT**

## Calculation Control - Example 1

The *Calculation Control properties* provide the entry of equalities for use by the Theoretical Control.

Calculation Control Properties	
	Calculation Expression
1	TheoTimeDial = Time_Dial
2	TheoInstOp = Instantaneous / Tap

The Settings tab has Setting Name

**Time\_Dial** is not a known AVTS Variable, and must be set equal to the known AVTS variable TheoTimeDial.

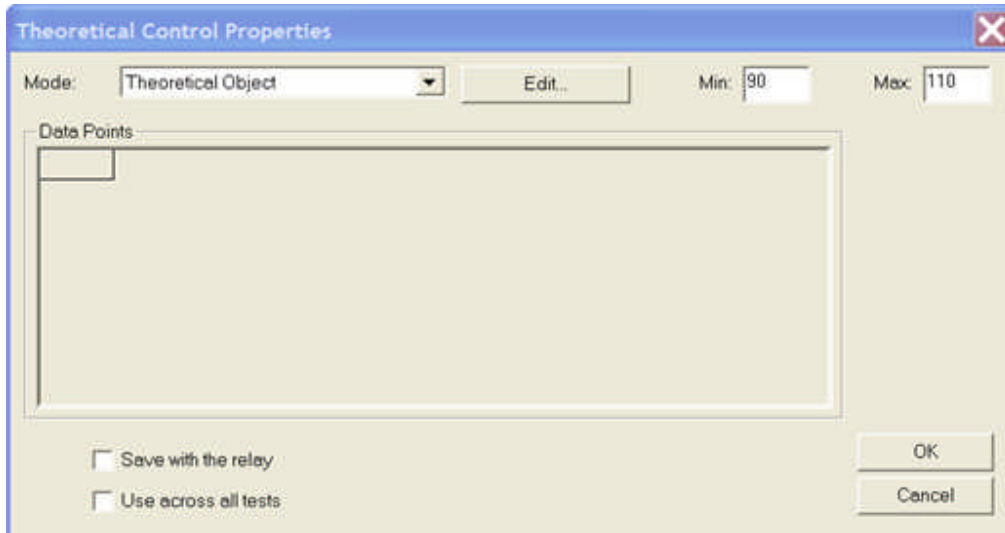
**Instantaneous** is not an AVTS variable and must be set equal to the known AVTS variable TheoInstOp

The **Link** from Calculation to List Controls displays Source = Done, indicating that the reading of settings, variables and equations will constitute the only operations for the Calculation Control as there are no pending actions. The Target = Execute indicates the action to immediately begin processing the entries in the List Control.

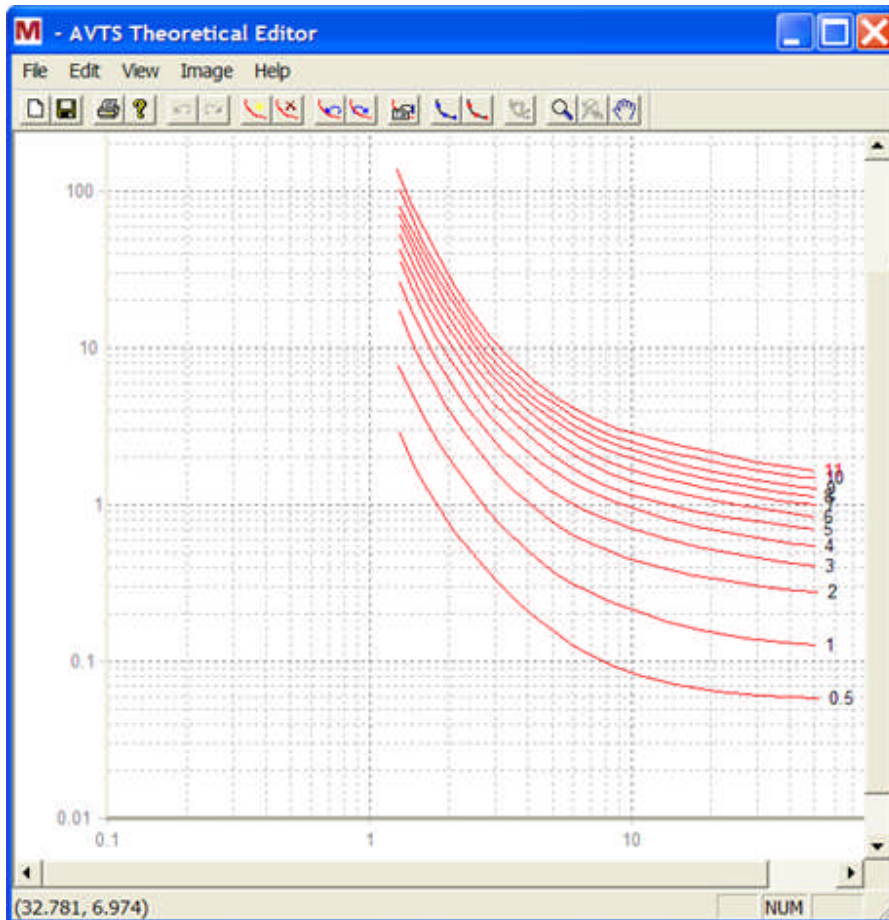
**Theoretical Control NEXT**

## Theoretical Control - Example 1

The *Theo Control properties* provides the selection of the Mode to be used in the test.



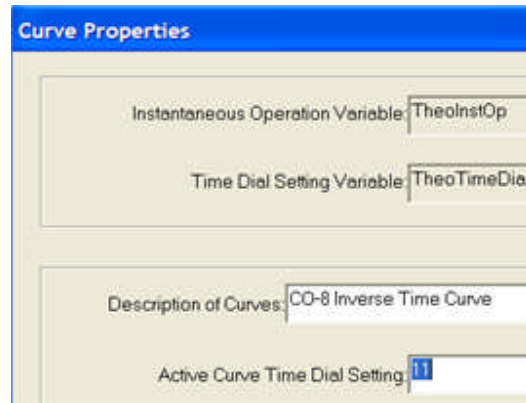
This example uses the Theoretical Object selection to digitize a *family of TOC curves* provided by the manufacturer.





Note: Time dial 11 is the only curve showing the number 11 in RED, as it is the one that is selected. Any other time dial setting value in the Settings tab group, that has not been digitized, will not display a curve during test execution, but will display elapsed time with incorrect error results.

The *Theoretical Control properties* for a Time Amplitude include



The two variable labels that AVTS recognizes for collecting the appropriate curve time for a given multiple of tap test executions.

Instantaneous Operation Variable, "**TheoInstOp**"

Time Dial Setting Variable, "**TheoTimeDial**".

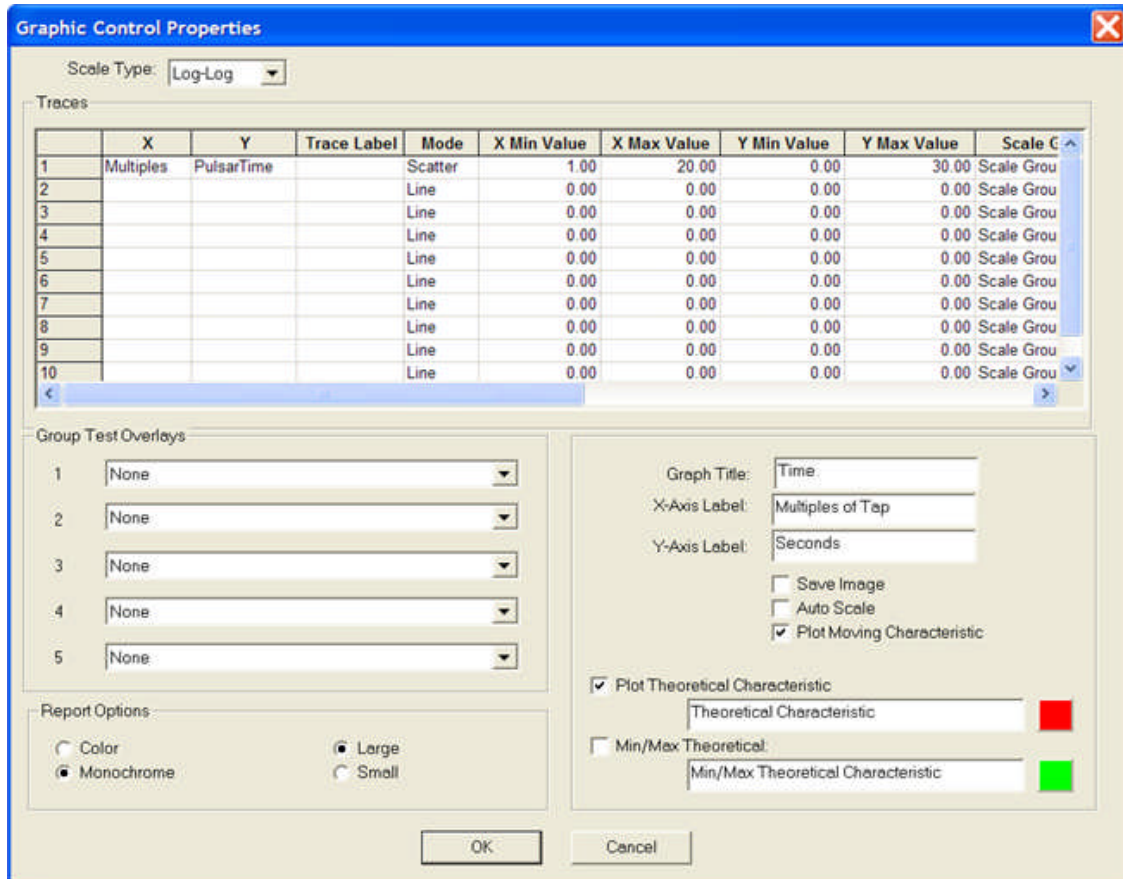
The **Link** from Calculation to Theo Controls displays Source = Done, indicating that the reading of settings, variables and equations will constitute the only operations for the Calculation Control as there are no pending actions. The Target = Calculate indicates that all values and variables be evaluated and used where necessary in the Theo Control.

Note: This link is not necessary and will be found missing in some test modules.

**Graphic Control NEXT**

## Graphic Control - Example 1

The *Graphic Control properties* is for entering the X and Y coordinates values governed by the variable names provided in the Log Control.



Right mouse click on the right side of each of the column edit fields produces a drop down list of the variables which are listed in the Variables block of the Log Control.

This example displays the variables **Multiple**, **Applied\_Current**, **PulsarTimeA**, **Expected\_Time** and **Error**.

The list will be identical for both columns.

The **Trace Label** column entry is Test Points, the Mode column entry is Scatter and the Color column is BLUE. These choices will produce single non-connected test result points on the graph in the color BLUE.

The **Auto Scale** check box has been unchecked to display the columns for the X and Y Min and Max entries for the graph display.

**Note: To display the Instantaneous value, the Auto Scale function must be unchecked and X Max Value must be greater than the TheolnstOp multiple.**

The **Graphic Control** does not require a Link Source or Target and will always be found "at float" in the Test Editor.

**List Control NEXT**

## List Control - Example 1

The *List Control properties* provide the values of Multiple (multiples of Tap setting) that will be applied to the relay in the Vector Control.

	Value Name
1	Instantaneous / Tap + 0.25
2	Instantaneous / Tap - 0.25
3	8
4	5
5	3
6	2 : ResetTime = 1

The **List** has lines with BASIC conditions and values. Each item in the List will be applied to the List Name (Multiple) on each pass of the "loop".

The first pass will apply the variable  $\text{Multiple} = (\text{Instantaneous} / \text{Tap}) + 0.25$ . The entry sets the first test point to the calculated Multiple value based on the Instantaneous and Tap relay Settings plus 0.25 of Tap. The first is to time test the Instantaneous element at 25% of the next highest Multiple.

**Note:** Doing so will always apply the first test point at higher Tap Multiple than the Instantaneous value.

The second pass will apply the next value in the List which has been set equal to a Multiple just less than the Instantaneous setting. This will provide a test of the time overcurrent characteristic at the highest current setting and not pickup the Instantaneous element of the relay.

**Note:** Applying test point for a large value immediate identifies the accuracy of the Time Dial setting and allows immediate adjustment if necessary.

If the value is greater than the first List item, the value will be changed to a value less than the Instantaneous setting.

**Note:** The conditioning test for the List value is accommodated in the second Calculation Control

The remaining List entries will perform timing tests against the time overcurrent characteristic and continue to the last entry.

After the last List entry has been applied, the proceeds to the End Control.

**Note:** Three timing events are shown, but any number may be applied by clicking the Add button and entering a value on the new line of the list.

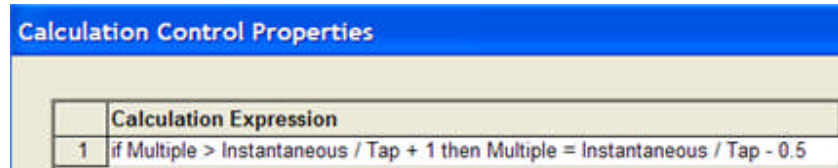
The **Link** from List to Vector Controls displays Source = Continue option in lieu of Done, indicating for the multiple List entries to continue being applied to the appropriate Controls in the "loop" of Controls.

The Target = Execute indicates the action to immediately begin processing the entries in the Vector Control.

**Second Calculation Control NEXT**

## Calculation Control 2 - Example 1

The **Calculation Control** properties provides for the entries of one or more to condition the value of Multiple applied to the Vector Control.



Calculation Control Properties	
	Calculation Expression
1	if Multiple > Instantaneous / Tap + 1 then Multiple = Instantaneous / Tap - 0.5

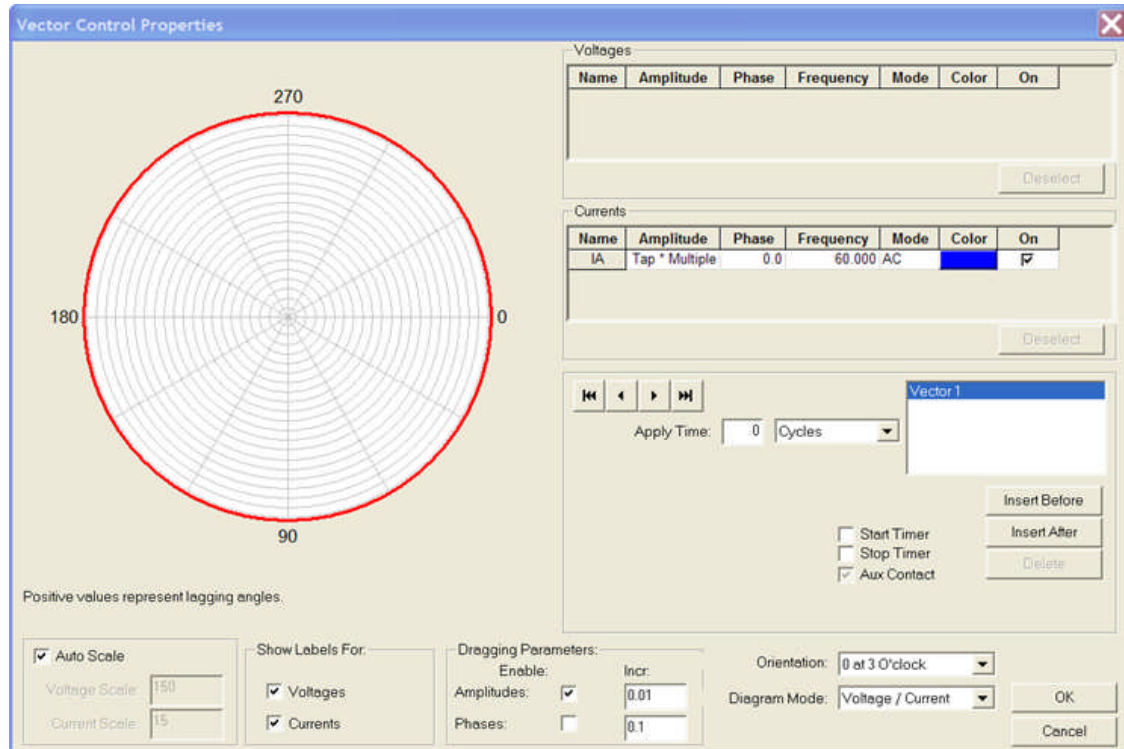
If the Multiple exceeds the Instantaneous setting tested, then the Multiple is set equal to a value less than the Instantaneous setting. The equality produces the test for the high Multiple to be applied to a Multiple that will test the Time Overcurrent curve instead.

Note: The second List Control item was to test the OC characteristic with a value of current 25% Tap less than the Instantaneous setting. This Calculation condition provides a test with a value of current 50% Tap less than the Instantaneous setting.

**Vector Control** NEXT

## Vector Control - Example 1

The *Vector Control properties* show only the current IA which was set in the Start Control with only one current channel being elected.



The Amplitude has the variable name Tap \* Multiple.

Tap is the value taken from the Setting group set in the Start Control and

Multiple is the value taken from the present pass through the List Control.

The **Link** from Vector to HW Timer Controls displays Source = Done, indicating that the settings, variables and equations will constitute the only operations for the Vector Control as there are no pending actions.

The Target = Wait for Stop indicates that the Timer Control will begin timing and continue pending the properties action to wait to stop the timer.

**HW Timer Control NEXT**

## HW Timer Control - Example 1

The **HW Timer Control properties** contains the settings required for the monitoring of the device under test.

Included are the settings for the Start and Stop applications to the Timer.

The Timer Start is via software and at the moment the test execution applies the HW Timer Control.

The timer continues applying the value of current set in the Vector Control until a Stop condition is met.

This example uses Post 3 to monitor a normally open contact making, or providing Continuity to the Timer. The time is collected and passed to the Log Control.

**Note that the Max Time to Operate edit field displays a value of 20 seconds. Should 20 seconds expire with no Stop condition applied, the test will abort.**

The **Link** from HW Timer to Log Controls displays Source = Done, indicating that the required action as stopped the timer and stored the result in temporary memory; unless there was no action within the allotted time, which constitutes a test abort.

The Target = Execute indicates the action to immediately begin processing the entries in the Log Control.

**Log Control NEXT**

## Log Control - Example 1

The **Log Control properties** governs the data stored as a test result for retrieval in a Test Report later. The items to be retained are listed in Variables to Save block and will be "Include in Report" column box is checked.

	Variable	Units	Decimal Places	Include in Report	Align	Width
1	Multiple	X Tap	2	<input checked="" type="checkbox"/>	Center	10
2	Applied	Amps	2	<input type="checkbox"/>	Center	9
3	TheoTime	Seconds	2	<input checked="" type="checkbox"/>	Center	10
4	TestTime	Seconds	2	<input checked="" type="checkbox"/>	Center	10
5	Min	Seconds	2	<input checked="" type="checkbox"/>	Center	8
6	Max	Seconds	2	<input checked="" type="checkbox"/>	Center	8
7	Error	+/- 5%	2	<input checked="" type="checkbox"/>	Center	8
8	Pass/Fail\$		0	<input checked="" type="checkbox"/>	Center	10

	Calculation Expression
1	if Multiple > Instantaneous / Tap then TheoTime = 0.043 else TheoTime = GetTheoTime(Multiple)
2	Applied = Ia
3	TestTime = PulsarTimeA
4	if Multiple < (Instantaneous / Tap) then TheoTime = GetTheoTime(Multiple)
5	Error = (PulsarTimeA - TheoTime) / TheoTime * 100
6	Min = TheoTime * 0.95
7	Max = TheoTime * 1.05
8	if PulsarTimeA => Min and PulsarTimeA <= Max then Pass/Fail\$ = "Pass" else Pass/Fail\$ = "Fail"

**Note:** That the first line of this block is the Variable name "Applied". It is set equal to the known value of ia. "Applied" will be the column header for the Report and the value of ia will be placed in the column as a result for each successive time test.

If a variable is to be calculated for each successive test result, it will be evaluated by the Pre-Log Calculation block.

The first line in Calculation Expression "if Multiple > Instantaneous / Tap then TheoTime = 0.043 else TheoTime = GetTheoTime(Multiple)" is to set the theoretical time for the Instantaneous test equal to about 2-1/2 cycles. Any test after the first Multiple from the List Control will search the time overcurrent characteristic for the value of TheoTime(Multiple).

The fourth line calls for the time overcurrent disk time-delay for the applied Multiple from the List Control to be taken from the digitized curve by use of the variable GetTheoTime(Multiple).

The Min and Max values are calculated from tolerance multiples of the GetTheoTime(Multiple) curve time.

The Pass/Fail\$ if statement compares the Test Seconds value against the Min and Max values. If the Test time is within the boundaries, the test is a Pass; otherwise, it is a Fail.

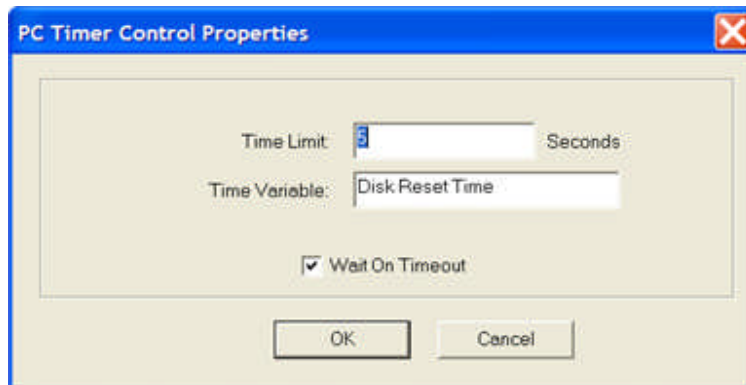
The **Link** from Log to PC Timer Controls displays Source = Done, indicating that any stored results and any calculations to place values for each of the included Log variables has been completed.

The Target = Reset indicates that the stored time value in the PC Timer register be reset to zero and the new time delay begin for the variable time or value in place in the PC Timer Control.

**PC Timer Control NEXT**

## PC Timer Control - Example 1

The *PC Timer Control properties* sets the amount of time to delay within this Control before continuing to the next control. The Time Limit is set to 5 Seconds and the Time Variable is set to DiskResetTime. The Wait On Timeout is checked placing emphasis on the Time Limit edit field for entry of a variable or discrete value. Any value previously set equal to DiskResetTime will be ignored unless the Wait On Timeout is unchecked.



Note: DiskResetTime is not a definite variable. This example might have had a different user created variable; an example may be Reset\_time, Reset, or others. The variable may also be included in the Workbook Settings tab group; thus, allowing time control based on the time dial setting.

The **Link** from PC Timer to List Controls displays Source = Timeout, indicating that the pending process within the PC Timer Control will continue until the completion of the variable or time setting. The Target = Next indicates that the test is to take the next value in the List Control and apply that value to the Vector Control.

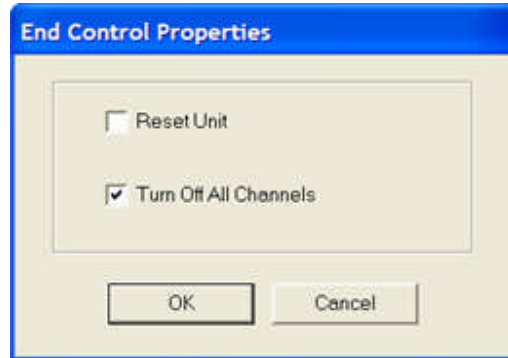
Note: When the last item in the List Control has been applied, there is not another Next value and the test proceeds to the alternate link to the End Control.

**END Control NEXT**



## End Control - Example 1

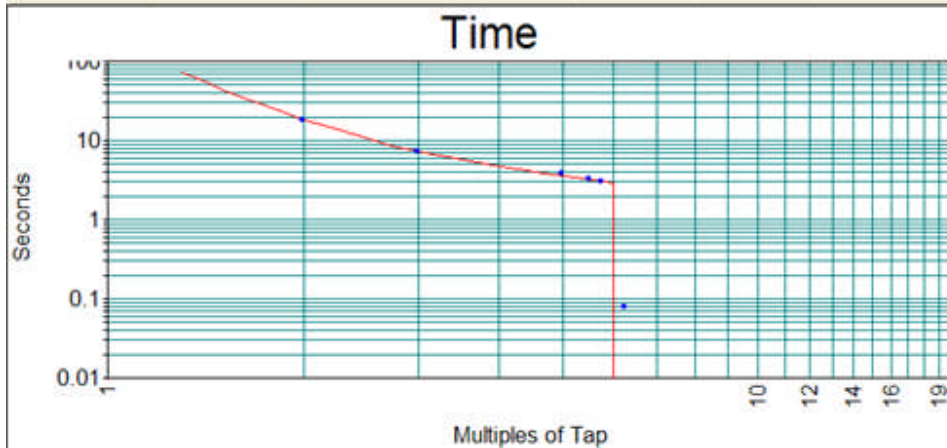
The *End Control properties* provides the necessary action to conclude the test process and execution. The two available options produce similar results. The first is to place the test set into a "soft reset", voiding any stored values and setting the amplifiers to the default OFF setting. The default option is to only Turn Off All Channels, which does not produce a complete reset of the test set.



## Test Results - Example 1

The *Test tab Results* for the Time Overcurrent tests. The results were obtained through simulation of the test.

Multiple	TheoTime	TestTime	Min	Max	Error	Pass/Fail
X Tap	Seconds	Seconds	Seconds	Seconds	+/- 5%	
6.25	0.04	0.03	0.04	0.05	81.40	Fail
5.75	3.00	2.97	2.85	3.15	-1.05	Pass
5.50	3.15	3.28	2.99	3.30	4.25	Pass
5.00	3.50	3.75	3.32	3.67	7.22	Fail
3.00	7.22	7.11	6.86	7.59	-1.59	Pass
2.00	18.19	17.73	17.28	19.10	-2.51	Pass



To also show the Instantaneous setting, the Graph display has the coordinates fixed as explained in the Graphic Control - Example 1 topic.

The test results for the three points along the digitized curve of the selected Time Dial are listed with the labeled columns of variables. Upon exiting the relay, the results may be saved to the active database.

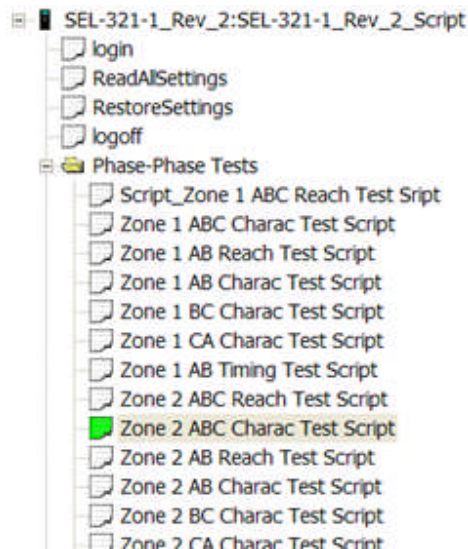
## Test Editor Charc Z Example 2

### Test Editor Charc Z Example 2

The object of Example 2 is to demonstrate a method of constructing an **Impedance Zone Characteristic Test** for a microprocessor relay. The Controls have been chosen to provide a test with results compared in the Log Control directly to original manufacturer's formula and settings given in the Workbook Settings tab.

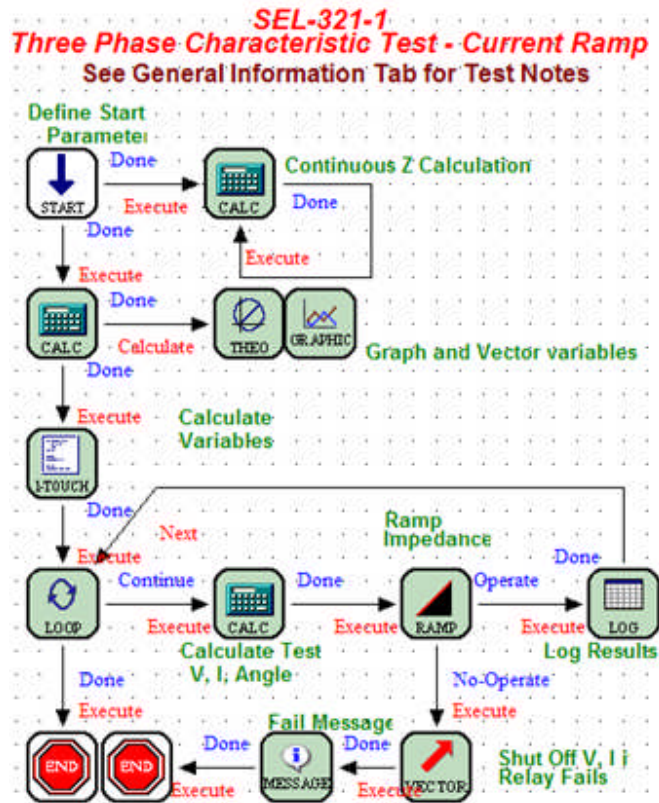
The 1-Touch Control is used to control communications to the relay during the test. Expanded definitions and details for each of the Controls are given in its own topic in the Test Editor Toolbar book.

Example 2 is taken from an AVTS module for a SEL 321 relay available from Megger. For the 1-Touch Control to communicate with the relay, the initial tests in the *Workspace Tests tab* define the tests to log onto the relay (first test) and then to read all the relay settings, and populate the Workbook Settings tab (second test) and place in a Temp file a mirror set of relay settings. At the completion of relay tests, the RestoreSettings will replace all relay settings with the copy of original settings placed in the Temp file. The final test is to logoff the relay.



Note: The login script usually deletes the Temp file used to store the original relay settings. A second login to the relay is performed after a series of performed tests with settings changes being a part of the script, the temp file will be erased and the original settings cannot be retrieved. Check the login script to determine if the script creates a message warning that the temp file contents will be deleted. Some script files contains the message script.

The *example test* is viewable in the Workbook Test Editor tab. The Zone 2 test is chosen for explanation of use of the following Controls:



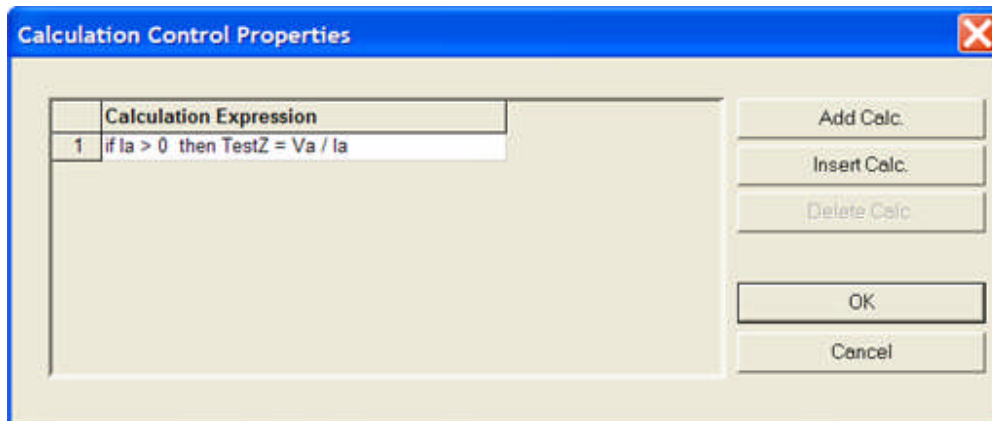
- Calculation Control
- Theoretical Control
- 1-Touch Control
- Loop Control
- Ramp Control
- Message Control
- End Control

Use of the **Start Control**, **Calculation Control**, **Graphic Control**, **List Control**, **HW Timer Control**, **Log Control**, **PC Timer Control** and **End Control** are explained in Test Editor Example 1.

See How to Create Test Procedure Example

## Calculation Control - Example 2

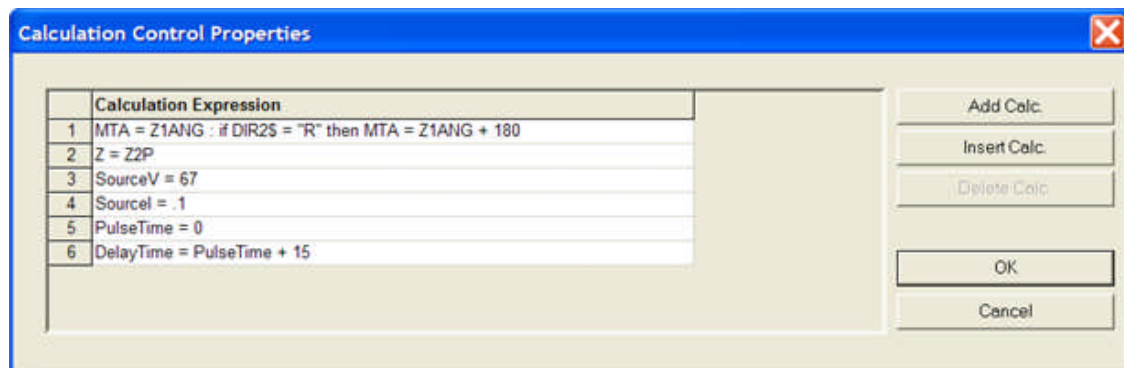
The first *Calculation Control properties* makes available a new variable Z defined by the formulated values of Va and Ia. The Control also conditions the formula so that the test WILL NOT abort immediately upon execution. The formula is  $Z = Va / Ia$ . At test execution all test set channels are set equal zero and the formula yields a calculation which is undefined where the denominator is zero. The conditional statement "if  $Ia > 0$  then  $Z = Va / Ia$ " does not allow the calculation for  $Ia = 0$ ; therefore, the test will not abort.



The **Link** looped on the Calculation Control makes the calculation true for every value of V and I during test Execution. The Target = Execute indicates the action to immediately begin processing the entries in the List Control. The Source = Done indicates that the reading of settings, variables and equations will constitute the only operations for the Calculation Control as there are no pending actions.

**Note:** There may be a test requiring that a calculation be performed to initialize a variable immediate to the Start Control, but to be performed only for initialization. The Source = Once would be the appropriate option for the loop back example.

The second *Calculation Control properties* contains all the formulas necessary to define the Variable names to be used by the Loop, third Calculation, Ramp and Log Controls, where necessary. The first Calculation Expression defines the MTA to be 180 degrees from the original MTA if the Zone is reversed "R" setting in the Settings tab.



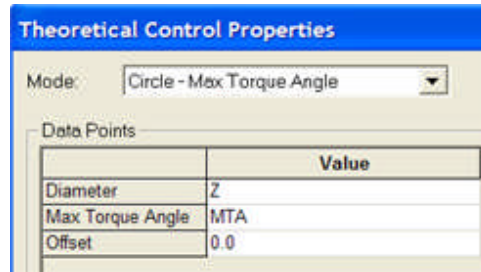
The **Link** from Calculation to Theo Controls displays Source = Done, indicating that the reading of settings, variables and equations will constitute the only operations for the Calculation Control as there are no pending actions. The Target = Calculate indicates that all values and variables be evaluated and used where necessary in the Theo Control.

The Calculation Control topic further defines the control.

**Theoretical Control NEXT**

## Theoretical Control - Example 2

The *Theo Control properties* provides the selection of the Mode to be used in the test.



This example uses the Theoretical Circle- Max Torque Angle option to describe the characteristic by the Z and MTA settings for the relay in the Settings tab.

The **Link** from Calculation to Theo Controls displays Source = Done, indicating that the reading of settings, variables and equations will constitute the only operations for the Calculation Control as there are no pending actions. The Target = Calculate indicates that all values and variables be evaluated and used where necessary in the Theo Control.

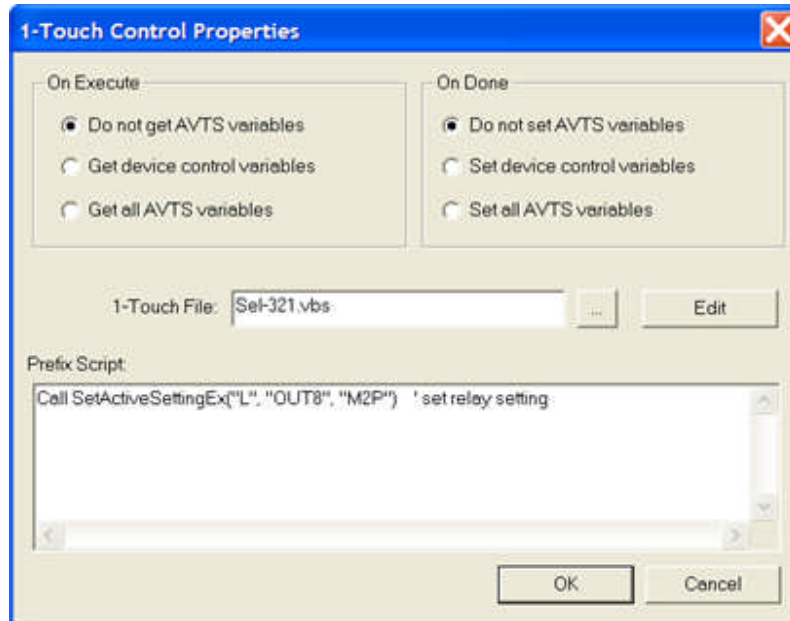
**Note:** This link is not necessary and will be found missing in some test modules.

The Theoretical Control topic defines the options for creating the characteristics.

**1-Touch Control** NEXT

## 1-Touch Control - Example 2

The *1-Touch properties* displays the default option buttons selections and the name of the 1-Touch File. The 1-Touch file will be placed in the BASIC tab for controlling the relay settings for each test.



The **1-Touch File:** entry "**Sel-321.vbs**" is the file in the AVTS/Macros folder that contains the *Visual BASIC Script* file to control the communications between the computer and the relay. The file referenced is included with the SEL-321 Relay in the installed *RelayTestLibV2.mbd* database.

```

dim relay

dim RelayName
dim RelayMapFile
dim RelayTmpFile

dim sleeper

dim fso

dim bVerifyResponse

SUB ShowMessage(ByVal MsgStr)
    call MsgBox(MsgStr, 0, RelayName)
END SUB

FUNCTION CancelOperation()
    CtrlX = CStr(Chr(24))

    CancelOperation = WriteString(CtrlX)
    if CancelOperation = 1 then
        'call sleeper.Sleep(0.5)
        relay.FlushInput()
    else
        ShowMessage "Could not write the cancel operation (C
    end if

END FUNCTION

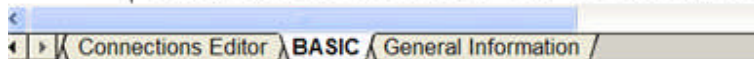
FUNCTION WriteString(ByVal TextToWrite)

    relay.FlushInput()

    WriteString = 1
    StrLen = Len(TextToWrite)
    for i = 1 to StrLen

        CharToWrite = Mid(TextToWrite, i, 1)
        WriteString = relay.WriteString(CharToWrite)
        call sleeper.Sleep(0.1)

        ' Verify that the character we sent was echoed back.
    
```




The **Prefix Script:** entry "**Call SetActiveSettingEx("L", "OUT8", "M2P") ' set relay setting**" uses the Script Language from the called file and resident in the BASIC tab to set the relay Logic Setting Output number 8 to respond to relay action based on the Zone 2 tests.

**Note:** Only the M2P element relay setting is monitored by the Out8 contact.

The **BASIC Tab** contains the complete Script file and may be edited. To edit the Script, use the Script Help program; AVTS Installation Disk. Perhaps the relay baud rate is different and the script requires that to be changed. Open the BASIC Tab and scroll down in *the Script* to locate the command set "**FUNCTION InitRelay()**". The default baud rate set in the Script is 9600, and may be edited directly to a different rate.



```
FUNCTION InitRelay()  
    call InitVars()  
  
    InitRelay = relay.Open(AVTS.AuxComA, 9600, 8, "NONE", "1")  
    if InitRelay = 0 then  
        ShowMessage "Could not communicate with the relay."  
        relay.Close()  
    else  
        relay.SetReceiveTimeout(1000)  
        InitRelay = CancelOperation()  
    end if  
END FUNCTION
```

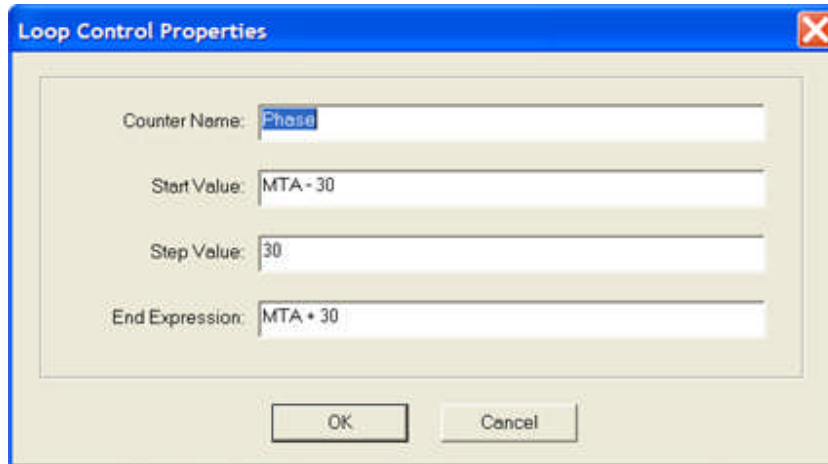
Change baud rate 

The 1-Touch Control topic defines the Control Properties and Prefix Script data.

**Loop Control** NEXT

## Loop Control - Example 2

The *Loop Control properties*



The screenshot shows a dialog box titled "Loop Control Properties". It has a blue title bar with a close button (X) in the top right corner. The dialog contains four input fields with labels to their left: "Counter Name:" with the value "Phase", "Start Value:" with "MTA - 30", "Step Value:" with "30", and "End Expression:" with "MTA + 30". At the bottom of the dialog are two buttons: "OK" and "Cancel".

The Loop Control determines the locations of testing for the characteristic.

The **Counter Name**: is Phase and used as the angle variable in the Ramp Control.

The first test will be the **Start Value**: MTA - 30 degrees.

The test will **Step Value**: 30 degrees and perform the second test at MTA.

The last test will be at the **End Expression**: MTA + 30.

**Note:** The number of tests is determined directly by the number of steps between the Start and End values.

I.e., the **Start Value**: MTA - MTA will produce the first test at 0 degrees and

the **End Expression**: 220 - MTA will locate the last test and

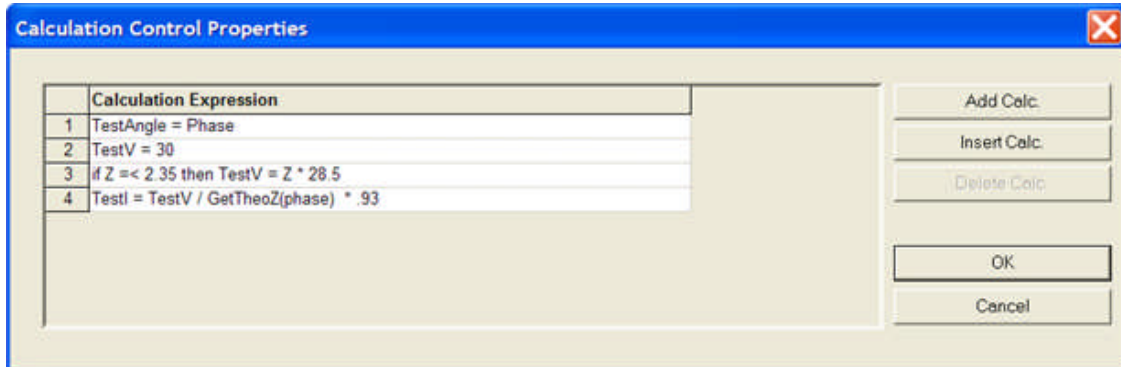
**Step Value**: 10 will create tests at 10 degree intervals from the first test through the last test.

The Loop Control topic further defines the properties.

**Third Calculation Control** Next

## Third Calculation Control - Example 2

The third *Calculation Control properties* contains the formulae necessary to define the Variable names to be used by the Ramp and Log Controls. The first Calculation Expression defines the MTA to be 180 degrees from the original MTA if the Zone is reversed "R" setting in the Settings tab.



The **Log Control** will use the Phase for each test as the test result angle of test.

The test Voltage is set constant at 30 volts.

The **if Expression** is to set the maximum test Current to be no more than 95% of the maximum 30 ampere output of the test set minus the 5% allowance for relay tolerance.

**Note:** Use lower case letters for "if" in an if statement to avoid AVTS interpreting if as Current channel If.

The **Current** calculation TestI is equal to the Test Voltage divided by the characteristic impedance value at the given angle of test. The \*0.93 multiplier is to reduce the ramp start current 7% less than the relay tolerance of 5%.

The Calculation Control topic further defines the control.

**Ramp Control Next**

## Ramp Control - Example 2

The *Ramp Control properties* display the values of the Voltage and Current channels to be executed.

The screenshot shows a software interface for configuring Ramp Control. It is divided into two main sections: 'Voltages' and 'Currents'. Each section contains a table with columns for Name, Amplitude, Phase, Frequency, Mode, Color, and On. Below the tables are control buttons and a 'Deselect' button.

Voltages						
Name	Amplitude	Phase	Frequency	Mode	Color	On
VA	TestV	0.0	60.000	AC	Red	<input checked="" type="checkbox"/>
VB	TestV	120.0	60.000	AC	Green	<input checked="" type="checkbox"/>
VC	TestV	240.0	60.000	AC	Blue	<input checked="" type="checkbox"/>

Currents						
Name	Amplitude	Phase	Frequency	Mode	Color	On
IA	TestI	Phase	60.000	AC	Red	<input checked="" type="checkbox"/>
IB	TestI	Phase + 120	60.000	AC	Green	<input checked="" type="checkbox"/>
IC	TestI	Phase + 240	60.000	AC	Blue	<input checked="" type="checkbox"/>

Control buttons:  Start,  Start 2,  Incr.,  Incr. 2,  End,  End 2,  Pre-ramp. Pulse Time: Pulse Cycles.

The Start values for the Voltages and Currents use values determined by the Variables defined in the Calculate and Loop Controls.

The *Increment of step* for the Ramp is defined as 1% of the Start value of Currents, or  $TestI * 0.01$ .

The time for each step prior to applying the next 1% increment is Delay Cycles defined in the second Calculation Control.

This screenshot shows the same Ramp Control interface as above, but with the 'Increment Time' set to 'Delay Cycles' and the 'Start' radio button selected. The tables show the calculated values for the ramp.

Voltages				
Name	Amplitude	Phase	Frequency	Color
VA	0	0.0	0	Red
VB	0	0.0	0	Green
VC	0	0.0	0	Blue

Currents				
Name	Amplitude	Phase	Frequency	Color
IA	TestI * 0.01	0.0	0	Red
IB	TestI * 0.01	0.0	0	Green
IC	TestI * 0.01	0.0	0	Blue

Control buttons:  Start,  Incr.,  End. Increment Time: Delay Cycles.

**Note:** The incremental Ramp UP or DOWN is determined automatically by AVTS. If the End value is greater than the Start value, the Ramp will be increasing in value. If the End value is less than the Start value, the Ramp will be decreasing in value.

The *End values* for the Voltage and Currents use the values determined by the Variables previously defined where the ending Currents are 150% of the Test Voltage divided by the relay set impedance Z.

Voltages				
Name	Amplitude	Phase	Frequency	Color
VA	TestV	0.0	60.000	Red
VB	TestV	120.0	60.000	Green
VC	TestV	240.0	60.000	Blue

Currents				
Name	Amplitude	Phase	Frequency	Color
IA	(TestV/Z)*1.5	0.0	60.000	Red
IB	(TestV/Z) * 1.5	120	60.000	Green
IC	(TestV/Z) * 1.5	240.0	60.000	Blue

*Ramp Control Operation properties* sets the conditions of the test set monitoring of the test for relay response.

**Contact Control Properties** ✖

---

Unit:       Debounce Time:  ms

Auxiliary Contact:

---

Post:

Mode:

Active On:

Do Action:        Latch

Max Time to Operate:  Seconds       Enable Horn

Turn Currents Off

The third Input post will recognize as relay operation of a closed contact and latched on the first closure of the contact.

Multiple operations for the same test will not be recognized.

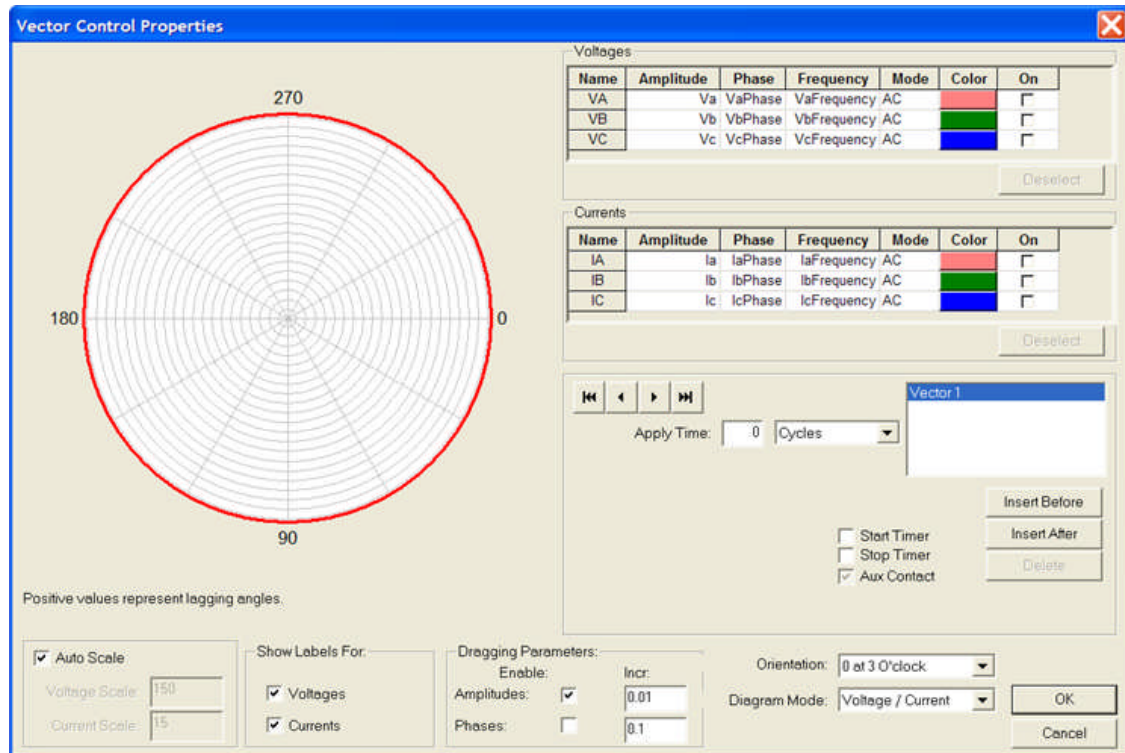
The test will Abort if the Max Timer to Operate exceeds 100 Seconds.

The Ramp Control topic further defines the settings information of the Control.

**Vector Control** NEXT

## Vector Control - Example 2

The *Vector Control properties* display the values of the Voltage and Current channels to be executed. This example demonstrates that the values are "Set to Variables" obtained from the drop down menu from a right mouse click, and turned OFF. The Vector Control has been used to turn OFF all channels due to a device no operate condition.



The Vector Control topic further defines the settings information for the Control.

**Message Control NEXT**

## Message Box Control - Example 2

The *Message Box Control properties* displays the text entered to explain that the test was aborted due to no operation of the device under test.

The screenshot shows a dialog box titled "AVTS Test Execution Message Box". It has a blue title bar with a close button (X) in the top right corner. The dialog is divided into several sections:

- Mode:** A dropdown menu set to "Message Only".
- Data Entry Variable:** A text input field containing "DataEntry".
- Message:** A text area containing the message: "Relay element failed to operate! Check settings and logic and test again."
- Selection List:** A list box with a header "List Name" and one empty row. To the right of the list are "Add" and "Delete" buttons.
- Buttons:** "OK" and "Cancel" buttons are located at the bottom center of the dialog.

The Message Box Control topic further defines all the options for settings of the Control.

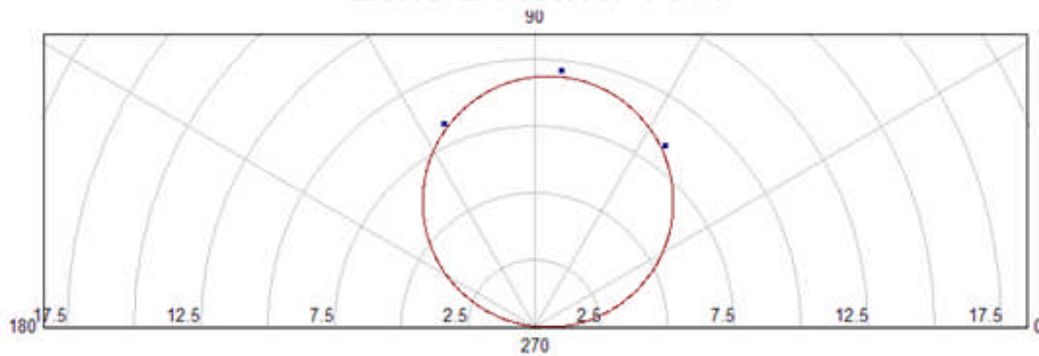
**Test Results** Next

## Test Results - Example 2

The **Test Results** for the Characteristic Z Example.

Impedance	CalcZ	Error	Pass/Fail	TestVolts	TestAmps	TestAngle
Ohms	Ohms	%		Volts	Amps	
8.307	8.11	2.47	Pass	30.00	3.61	54.00
9.552	9.36	2.05	Pass	30.00	3.14	84.00
8.231	8.10	1.63	Pass	30.00	3.64	114.00

### Zone 2 Reach Test



The test results for the three points on the mho reach and maximum torque angle characteristic are listed with the labeled columns of variables. Upon exiting the relay, the results may be saved to the active database.



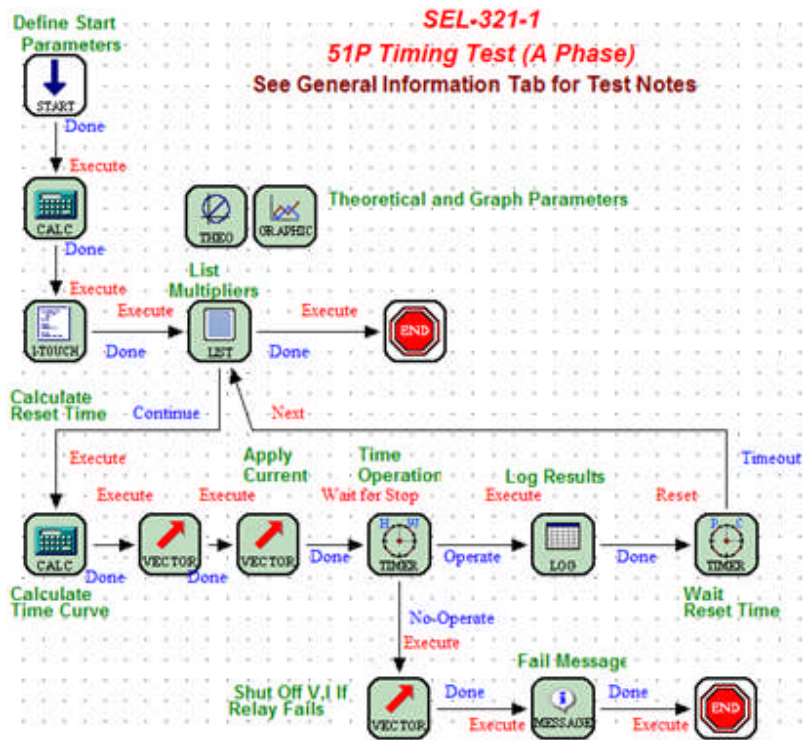
### Test Editor Microprocessor Time OC Example 3

The object of Example 3 is to demonstrate a method of constructing a **Time Overcurrent Timing Test for a microprocessor relay**. The Controls explained in this topic are to detail the use of Calculation and Theoretical Controls to create the time OC characteristic. By using the same formulae as the manufacturer, the most accurate results will be achieved.

Expanded definitions and details for each of the Controls are given in its own topic in the Test Editor Toolbar book.

Example 3 is taken from the **AVTS RelayTestLibV2.mdb SEL-321 relay**. This database is available with the original installation of AVTS Ver. 2.x.

The **example test** is viewable in the Workbook Test Editor tab.



The 51PP Timing test is chosen using the Settings from the *Settings tab* for explanation. Settings used are

80	E51P	Y			Enum	0	Enable Phase TOC Element
81	51PP		4	0.5	Float	0	Phase TOC Pickup (amps sec.)
82	51PC	U4			Enum	0	Phase TOC Curve Family
83	51PTD		5.25	0.05	Float	0	Phase TOC Time Dial US 0.5-15, IEC 0.05
84	51PRS	N			Enum	0	Phase TOC Reset Delay

51PP - Phase TOC pickup setting secondary amps; 4 is chosen

51PC - Phase TOC Curve Family have formulae for each family type; the U4 family is chosen

51PTD - Phase TOC Time Dial Setting; 5.25 TD is chosen

51PRS - Phase TOC Reset Delay in seconds; the setting is either enabled (Y) or disabled (N) - N is chosen

The detail of use of the following:

*First Calculation Control* displays the manufacturer's time delay formulae for the 51PRS setting conditioned by each 51PC setting.

Calculation Control Properties	
	Calculation Expression
1	if 51PCS = "U1" and 51PRSS = "Y" then ResetTime = 51PTD * 1.08
2	if 51PCS = "U2" and 51PRSS = "Y" then ResetTime = 51PTD* 5.95
3	if 51PCS = "U3" and 51PRSS = "Y" then ResetTime = 51PTD* 3.88
4	if 51PCS = "U4" and 51PRSS = "Y" then ResetTime = 51PTD* 5.67
5	if 51PCS = "C1" and 51PRSS = "Y" then ResetTime = 51PTD* 13.5
6	if 51PCS = "C2" and 51PRSS = "Y" then ResetTime = 51PTD * 47.3
7	if 51PCS = "C3" and 51PRSS = "Y" then ResetTime = 51PTD* 80
8	if 51PCS = "C4" and 51PRSS = "Y" then ResetTime = 51PTD*120

51PC\$ is used because the curve type setting in the Setting tab is a string variable

51PRSS\$ is also a string variable set to either Y or N for Yes or No enabled

if 51PC\$ = "U4" and the 51PRSS\$ = "Y" then ResetTime = 51PTD \* 5.67

if 51PRSS\$ = "N" then there will be no emulation of an electro-mechanical disk reset time

Theoretical Control *Theoretical Calculations* sections provide the detail to create the curve characterized by the Settings

Theoretical Calculations

Type:  Polar,  Linear

Variable Name: X: Multiplier, Y: Tm

Iteration Variable: Multiplier

Range: 1.5 to 40

Step: 0.1

	Calculation
1	if 51PCS = "U1" then Tm = 51PTD * (.0226+(.0104/(Multiplier*.02-1)))
2	if 51PCS = "U2" then Tm = 51PTD * (.18+ (5.95/(Multiplier*2-1)))
3	if 51PCS = "U3" then Tm = 51PTD * (.0963+ (3.88/(Multiplier*2-1)))
4	if 51PCS = "U4" then Tm = 51PTD * (.0352+ (5.67/(Multiplier*2-1)))
5	if 51PCS = "C1" then Tm = 51PTD * (.14/(Multiplier*.02-1))
6	if 51PCS = "C2" then Tm = 51PTD * (13.5/(Multiplier-1))
7	if 51PCS = "C3" then Tm = 51PTD * (80/(Multiplier*2-1))
8	if 51PCS = "C4" then Tm = 51PTD * (120/(Multiplier-1))

Buttons: Add, Insert, Delete, OK, Cancel

The displayed data will be **Linear**

The Variable Name for **X**: is Multiplier, which is used as the label for the List Control to perform successive multiples of tap value current to be applied on each test of the curve

The Variable Name for **Y**: is Tm, which is the manufacturer's label for Time

The **Iteration Variable**: is Multiplier, which is in agreement to the List Control label and list of values to apply on each test

The **Range**: is 1.5 to 40, which are the minimum and maximum values for the calculations from the formulae

The **Step**: is 0.1, which is the increment of Range for each calculation of the curve; the Graphic Control will create the curve by "drawing" a straight line from Step calculation to Step calculation; for Step = 0.1, there will be (40 - 1.5) / 0.1 = 385 calculations

Note: The smaller the number for Step, the longer for the initial test execution. Also, the larger the number, the more evident the straight lines "drawn" and perhaps less accuracy in the test result.

Calculations displays the manufacturer's formulae listed for each curve family for each 51PC setting; for 51PC = U4

if 51PC\$ = "U4" then Tm = 51PTD \* (.0352+ (5.67/(Multiplier^2-1)))

The Test Editor Charc Z Example 2 details the 1-Touch Control.

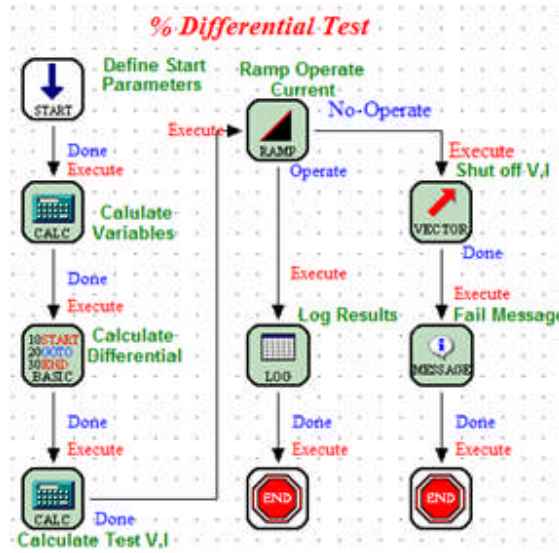
The Test Editor OC Time Example 1 details the remainder of the Controls used in this example of the Test Editor tab.

See How to Create Test Procedure Example

## Test Editor Differential Example 4

The object of Example 4 is to demonstrate the use of the **BASIC Control for the ABB HU differential electro-mechanical relay**.

The *example test* is viewable in the Workbook Test Editor tab.



The *complementary "macro"* is viewable in the Workbook BASIC tab.

```

if t1>t2 then smaller = t2
if t2>t1 then smaller = t1
if t1 = t2 then smaller = t2
i_restraint = smaller * 3

if t1>t2 and t2<8.7 then i_restraint = smaller * 3
if t2>t1 and t1<8.7 then i_restraint = smaller * 3

if t1 = 2.9 and t2 = 2.9 then i_operate = 2.6* 0.9
if t1 = 2.9 and t2 = 3.2 then i_operate = 3.7* 0.9
if t1 = 2.9 and t2 = 3.5 then i_operate = 5.0* 0.9
if t1 = 2.9 and t2 = 3.8 then i_operate = 5.8* 0.9
if t1 = 2.9 and t2 = 4.2 then i_operate = 7.8* 0.9
if t1 = 2.9 and t2 = 4.6 then i_operate = 9.0* 0.9
if t1 = 2.9 and t2 = 5.0 then i_operate = 10.4* 0.9
if t1 = 2.9 and t2 = 8.7 then i_operate = 16.2* 0.9
if t1 = 2.9 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 3.2 and t2 = 3.2 then i_operate = 2.7* 0.9
if t1 = 3.2 and t2 = 3.5 then i_operate = 4.0* 0.9
if t1 = 3.2 and t2 = 3.8 then i_operate = 4.9* 0.9
if t1 = 3.2 and t2 = 4.2 then i_operate = 6.9* 0.9
if t1 = 3.2 and t2 = 4.6 then i_operate = 8.1* 0.9
if t1 = 3.2 and t2 = 5.0 then i_operate = 9.6* 0.9
if t1 = 3.2 and t2 = 8.7 then i_operate = 15.7* 0.9
if t1 = 3.2 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 3.5 and t2 = 3.5 then i_operate = 3.0* 0.9
if t1 = 3.5 and t2 = 3.8 then i_operate = 3.8* 0.9
if t1 = 3.5 and t2 = 4.2 then i_operate = 5.7* 0.9
if t1 = 3.5 and t2 = 4.6 then i_operate = 6.9* 0.9
if t1 = 3.5 and t2 = 5.0 then i_operate = 8.3* 0.9
if t1 = 3.5 and t2 = 8.7 then i_operate = 14.5* 0.9
if t1 = 3.5 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 3.8 and t2 = 3.8 then i_operate = 3.2* 0.9
if t1 = 3.8 and t2 = 4.2 then i_operate = 5.2* 0.9
if t1 = 3.8 and t2 = 4.6 then i_operate = 6.5* 0.9
if t1 = 3.8 and t2 = 5.0 then i_operate = 7.9* 0.9
if t1 = 3.8 and t2 = 8.7 then i_operate = 14.1* 0.9
if t1 = 3.8 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 4.2 and t2 = 4.2 then i_operate = 3.5* 0.9
if t1 = 4.2 and t2 = 4.6 then i_operate = 4.7* 0.9
if t1 = 4.2 and t2 = 5.0 then i_operate = 6.2* 0.9
if t1 = 4.2 and t2 = 8.7 then i_operate = 12.9* 0.9
if t1 = 4.2 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 4.6 and t2 = 4.6 then i_operate = 3.9* 0.9
if t1 = 4.6 and t2 = 5.0 then i_operate = 5.3* 0.9
if t1 = 4.6 and t2 = 8.7 then i_operate = 12.4* 0.9
if t1 = 4.6 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 5.0 and t2 = 5.0 then i_operate = 4.3* 0.9
if t1 = 5.0 and t2 = 8.7 then i_operate = 11.6* 0.9
if t1 = 5.0 and t2 = 8.7 then i_restraint = smaller * 2

if t1 = 8.7 and t2 = 8.7 then i_operate = 5.0* 0.9
if t1 = 8.7 and t2 = 8.7 then i_restraint = smaller * 2
    
```

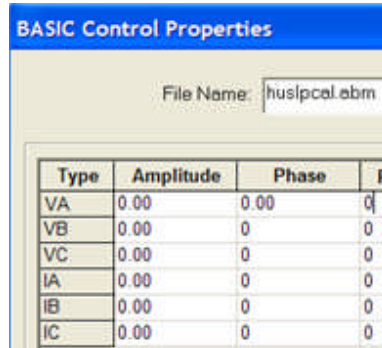
The relay settings are placed in the *Settings tab*.

	Setting Name	Value	Min Val	Max Val	Data Typ	Modbus	Description
1	Winding1Tap	5.0			Enum	0	Winding #1 tap setting
2	Winding2Tap	5.0			Enum	0	Winding #2 tap setting
3	%Sensitivity	35			Enum	0	% Sensivity Tap Setting
4	HarmonicRestraint	14	14	17	Float	0	% Harmonic Restraint setting
5	Instantaneous	0	0	90	Float	0	Instantaneous Setting in Amp
6	Seal_In	0.2			Enum	0	Seal_In tap setting

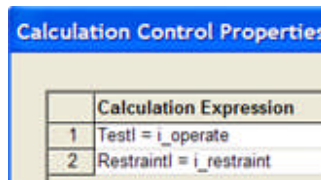
The *first Calculation Control* is used to condition the settings for the relay against the table of operate and restraints used in the test.

Calculation Expression	
1	T1 = Winding1Tap
2	T2 = Winding2Tap
3	T3 = Winding3Tap
4	if Winding1Tap > Winding2Tap then T1 = Winding2Tap
5	if Winding1Tap > Winding2Tap then T2 = Winding1Tap

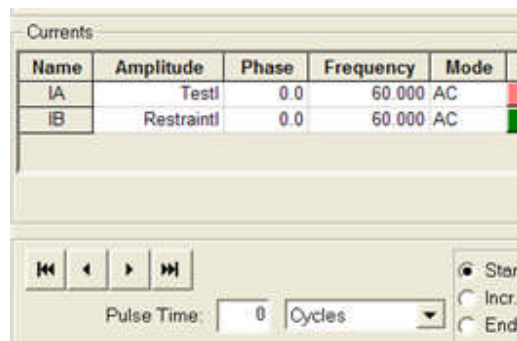
The *BASIC Control* is used to allow access to the BASIC tab. The Control does not contain any settings that may pass to the Voltage and Current channels and create incorrect results. The Control contains the name of the Macro that contains the manufacturer's values for test parameters based on the relay settings. The Macros reside in the AVTS/Macros subfolder.



The *second Calculation Control* is used to set the BASIC tab table values for the operate and restraint currents to use in the test.



The *Ramp Control Start* parameters are shown and the Incr. value is to ramp Start IA current. The ramp rate is 1% of the Start IA current at a rate of change of 10 cycles..



The *Log Control* calculates the values for the maximum and minimum acceptable values for the relay test results.

Log Control Properties						
Variables To Save						
	Variable	Units	Decimal Places	Include in Report	Align	Width
1	Restraint	Amps	2	<input checked="" type="checkbox"/>	Center	10
2	Min	Amps	2	<input checked="" type="checkbox"/>	Center	8
3	Max	Amps	2	<input checked="" type="checkbox"/>	Center	8
4	Operate	Amps	2	<input checked="" type="checkbox"/>	Center	10
5	Pass/Fail\$		0	<input checked="" type="checkbox"/>	Center	12

Pre-Log Calculations	
	Calculation Expression
1	Operate = Ia
2	Min = i_operate * 1.11
3	Max = Min * 1.12
4	Restraint = Ib
5	if Operate >= Min and Operate <= Max then Pass/Fail\$ = "Pass" else Pass/Fail\$ = "Fail"

The **Test tab** displays the final tests results using the applied values from the BASIC tab table and the relay response to the parameters.

% Differential				
Current A		Current B		
4.567 A		15.000 A		
0.0 °		0.0 °		
60.000 Hz		60.000 Hz		
-		-		
Restraint	Min	Max	Operate	Pass/Fail
Amps	Amps	Amps	Amps	
15.00	4.30	4.81	4.57	Pass

See How to Create Test Procedure Example

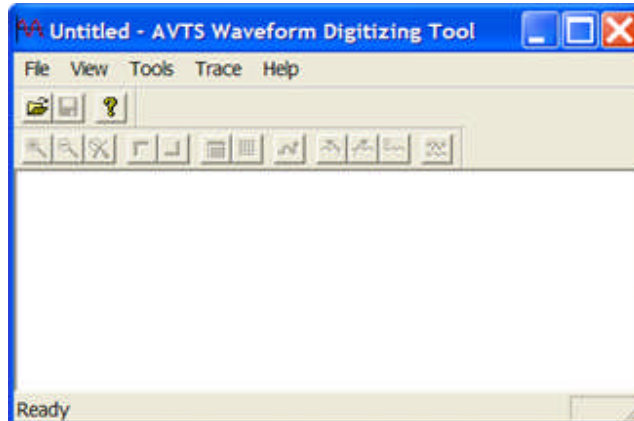




## Waveform Digitizing Tool...

## Waveform Digitizing Tool...

The **AVTS Waveform Digitizing Tool** is a separate software program which may be opened directly from the Windows Start/All Programs Megger list. AVTS does not need to be opened to use this program. However, any waveforms digitized and converted to a COMTRADE \*.cfg file will be placed automatically into the C:\Program Files\Megger\AVTSv2\Waveforms folder.



These files may then be called by the DFR Waveform Viewer, converted to HEX files, loaded into the MPRT or PULSAR test systems and played back into a device under test.

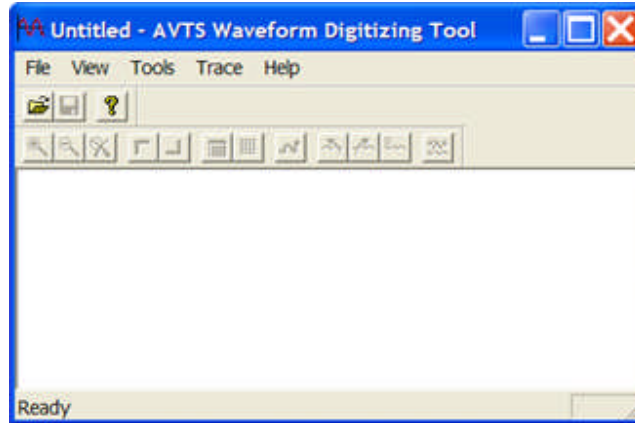
The program begins as a blank dialog ready to Open a bitmap of a waveform previously saved from a scanned document or saved from another software source.

Waveform Digitizing Tool Menu and Toolbar

Waveform Digitizing Tool Example


## Waveform Digitizing Tool Menu and Toolbar

The *Waveform Digitizing Tool* dialog displays a blank dialog when first opened. The File Open and Exit and the Help menu items are available for use when first opened.



Digitizing a set of waveforms is exemplified in the Waveform Digitizing Tool topic.

### File

**Open...**  option opens an existing \*.bmp file and enables the remainder of the menu items and toolbar tools.

**Exit** option will close down the DfrWaveView program.

### View

**Toolbar** option, if checked, displays the Menu Toolbar.

**Status Bar** option, if checked, displays the Status bar at the bottom of the dialog.

### Help

**Help Topics** option opens the AVTS Waveform Digitizing Tool Help files.

**About AVTS Waveform Digitizing Tool...**  option displays program information, version number and copyright data.

With an open bitmap file

### File

**Save**  option replaces the called file with changes made to the active waveforms.

**Note:** Save As is not an available option. To create a new set of waveforms with edited parameters in a new file, first copy the bitmap and rename it in Windows Explorer.

**Export to COMTRADE...** option converts the digitized waveform to COMTRADE \*.cfg and \*.dat files. Each trace saved will be appended to the original Exported trace and available to be played back simultaneously.

**Note:** The two files are automatically placed in the C:\Program Files\Megger\AVTSv2\Waveforms folder.


### View


**Zoom** option displays the choices to zoom in the bitmap by multiples of normal size. The choices are 1x, 2x, 3x, 4x and 5x.

**Zoom In**  option enlarges size the image in multiple steps with each mouse click.


**Zoom Out**  option reduces size the image in multiple steps with each mouse click.

**Note:** The image size cannot be zoomed out past normal size.


**Zoom Normal**  option returns the image to normal size.

**Filtered Image**  option checks the menu item and displays the image with filtering having been previously executed, the tool will be depressed to indicate the filtered state. Either uncheck the menu item or click on the tool to return to the normal state.


## Tools


**Auto Trace**  option changes the cursor to a cross. Place the cursor on a start point of the waveform, left mouse click, place the cursor on a second location of the waveform and left mouse click again. The selected portion of the waveform will then be highlighted in color and the included data points will be stored in a temporary file until the Export to COMTRADE option is selected.


The Auto Trace icon is called a "worm" due to its appearance.

**Delete Digitized Range**  option will delete the digitized section of the waveform between two mouse selected points.

**Delete All Digitized** option will delete all digitized data on the selected waveform. Edit to the waveform parameters will change the data when the waveform is digitized again.

**Define Top Left**  option must be performed prior to digitizing a waveform. Select the tool to change the cursor to a cross and click the top left boundary of the amplitude-time graph of the waveform.

**Define Bottom Right**  option must be performed prior to digitizing a waveform. Select the tool to change the cursor to a cross and click the bottom right boundary of the amplitude-time graph of the waveform.

**Filter Noise**  option opens the *Filter Noise* dialog. The action filters background "noise" from the image, between the top left and bottom right set points, for ease in digitizing. The dialog has a drop list for Radius and Weight.



Radius options are 1-5 (default 2).

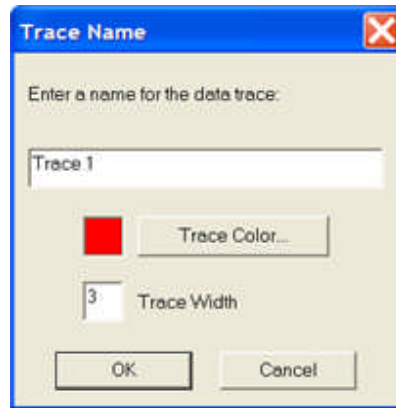
Weight options are 10, 25 and 30 - 100 in 10 multiples (default 75).

**Undo Last Filter** option returns the filtered image to the previous filtered state.

**Undo All Filters** option returns the filtered image to the original unfiltered state.

## Trace

**Properties...** option displays the *Trace Name* dialog.





The text edit field is to enter a name for the trace data which will identify the trace in the COMTRADE file, when exported.


The Trace Color... opens the color palette for color selection.

The Trace Width edit field (default 3) is to enter the physical width of the trace. The larger the number, the wider the trace when digitizing.

**Note:** The trace width has no effect on the size of the \*.data file created.

**Next Trace**  option changes focus to the next trace that has been digitized.

**Previous Trace**  option changes focus to the trace that was previously in focus.

**New**  option enables setting parameters and digitizing a new file trace or an additional trace in the same open file.

**Delete** option removes the digitized trace of the waveform in focus.

## Waveform Digitizing Tool

**Waveforms** from many sources may be digitized and then created as appropriate **COMTRADE** files.

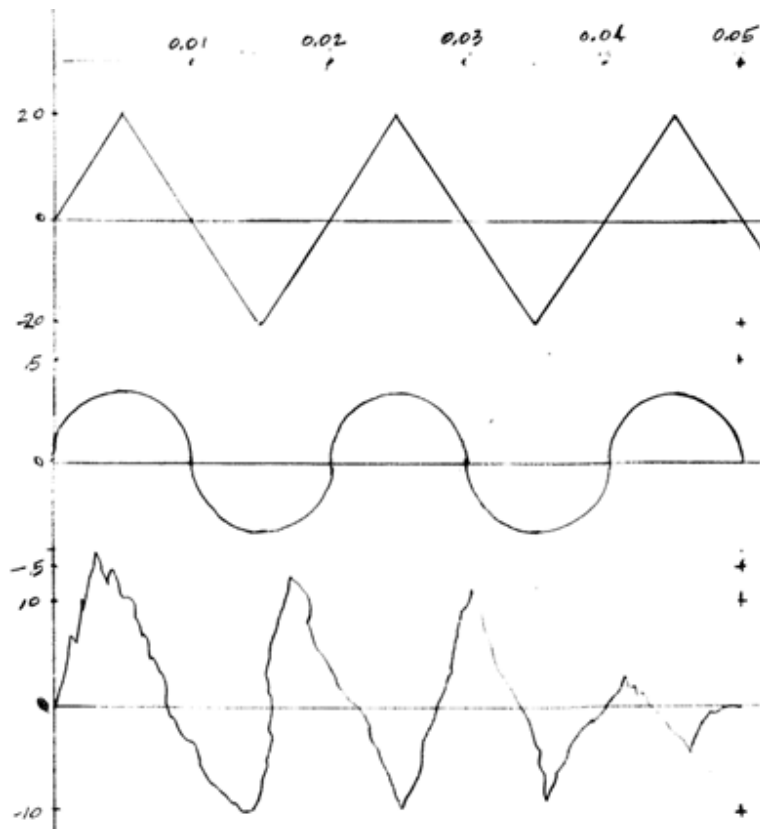
The waveforms may be created by several methods:

- Scanned relay manufacturers software and saved as a \*.bmp file.
- Scanned images from old paper files and saved as a \*.bmp file
- Drawn by the user, scanned and saved as a \*.bmp file
- Other images previously saved as a \*.bmp file.

The created COMTRADE configuration and data files are formatted automatically to conform to the IEEE COMTRADE Standard by the software.


The following example uses a set of hand drawn curves scanned and saved as a bitmap image.

1. Open the **Waveform Digitizing Tool...** to the blank dialog.
2. Import a **bitmap** using **File Open...**




**Note:** The example images were hand drawn on graph paper and points of amplitude and time reference extremes marked and/or labeled.

**Hint:** When creating a hand drawn or manually traced waveform(s), only mark the amplitude and time boundaries. If axes are necessary, draw using ver light marks.

3. Select the **Define Top Left**  tool and click on the zero time maximum amplitude mark. The **Top Left** dialog opens for entry of the Amplitude value. The Time (seconds) is defaulted to zero and constant. The amplitude was defined to be 20.

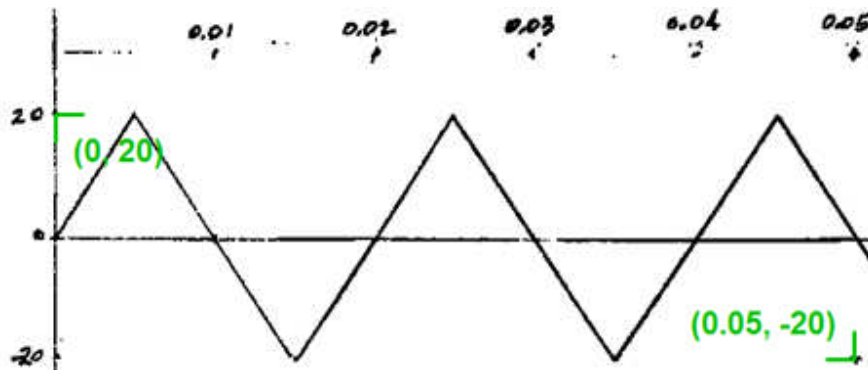



The waveform was then labeled with this corner marker and coordinate values assigned.

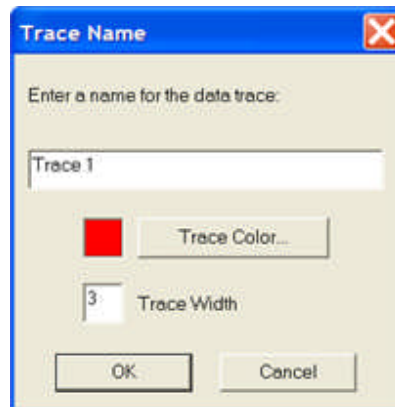
4. Select the **Define Bottom Right**  tool and click on the maximum time axis and minimum amplitude. The *Bottom Right* dialog opens for entry of the Time and Amplitude values.




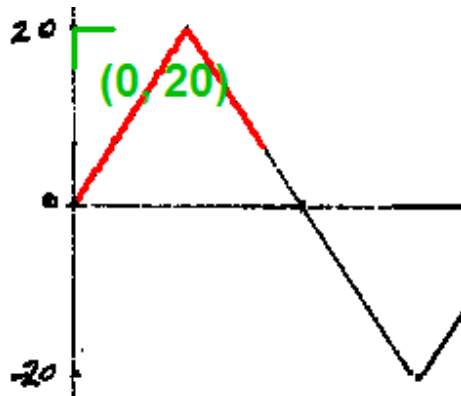
The waveform was then labeled with this corner marker and *coordinate values assigned*.



5. Select the **New Trace**  tool to open the *Trace Name* dialog and assign a name and color to the trace to be created. The Trace Width is default 3 and a good value for use.



6. Select the **Worm**  tool (**Auto Trace**) and click the zero time and first point of the waveform. Move the cursor to a second point on the waveform and click again to "draw" the *first curve trace*.

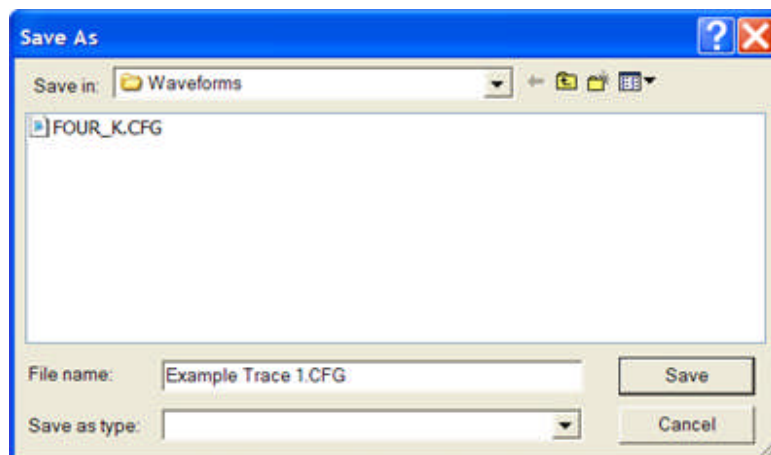


Note: The second click was almost a half cycle from the first click. The drawn curve followed the trace precisely. The worm will generally follow the trace unless another dark line crosses the trace. At that point the worm will usually select the darker line to draw.

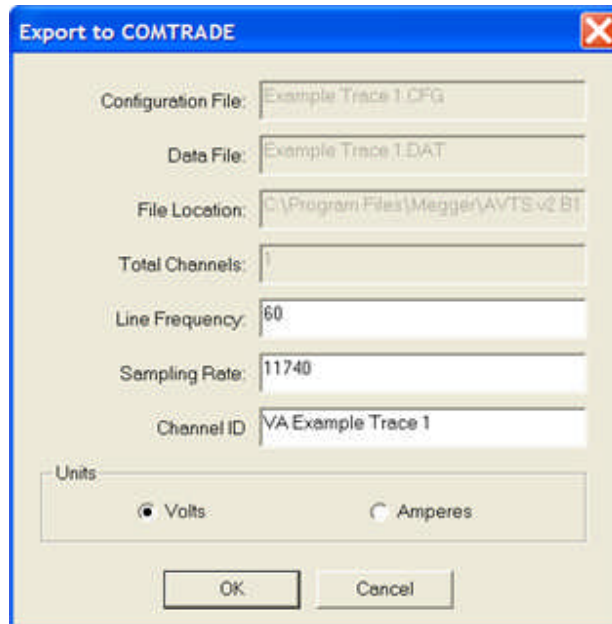
7. Continue using the worm until the trace is drawn to the extremes. Draw the entire curve to extremes and edit incorrect locations afterward using the Delete Digitized Range tool. This tool will usually correct an incorrect worm draw. This example was corrected by a click on the positive side of the axis and a second click on the negative side of the axis.

Note: The worm will not draw a trace outside the defined boundaries.

8. Select **File/Export Trace to COMTRADE...** to open the *Save As* dialog to label the name for the \*.cfg and \*.dat files for the first trace.



9. The Save button will open the *Export to COMTRADE* dialog.



**Configuration File:** default to *cfg\_name.cfg*. The file name may be edited in the DFR Waveform Viewer.

**Data File:** default to *cfg\_name.dat*. The file name may be edited in the DFR Waveform Viewer.

**File Location:** default to C:\Program Files\Megger\AVTSv2\Waveforms folder. Cannot edit.

**Total Channels:** default to the number of channels saved to the \*.cfg and \*.dat files. Cannot edit.

**Line Frequency:** default to 60 and may be edited to 50, 60, 400 or any desired frequency.

**Sampling Rate:** edit field default to a reasonable value and may be edited. The sample rate is per second and was default at 11740 samples per second.

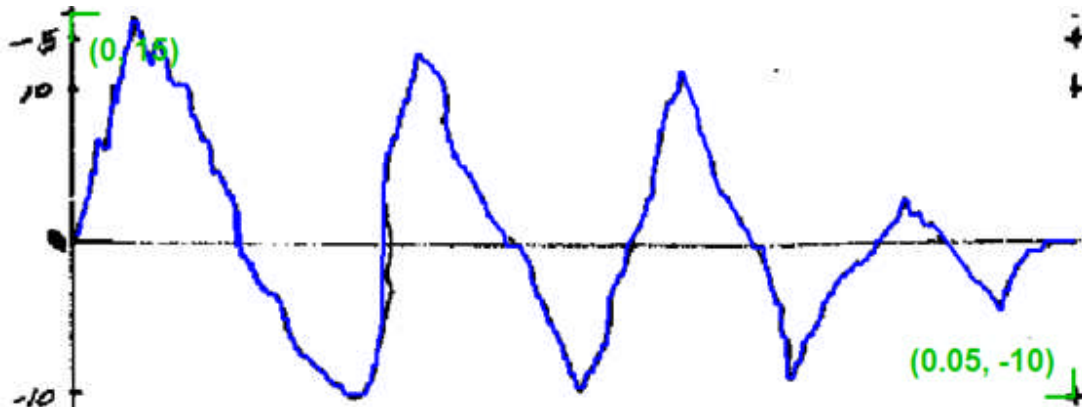
Channel ID edit field defaults to the file name and can be edited.

**Units** check buttons to identify channel trace as a Voltage or Current.

10. Click **OK** to save.

11. Follow the same procedure to add a second trace and **Export** to the same cfg file for all the traces to be saved for multi-amplifier playback.

12. The **third waveform** is crudely drawn. The worm will not backtrack in time to draw a trace, as shown in the transition from the first to second cycle. This is reasonable, because events recorded in time cannot backtrack to history. This trace demonstrates that the worm makes a best estimate between two points to have a time progressive trace.





## 13. The Saved COMTRADE files

The *Example Trace 1.cfg* file.

```

AVTS,Waveform Digitizing Tool,1997
3, 3A, 0D
1,VA Example Trace 1,,,V,6.035504e-004,8.282368e-007,0.000000e+000,-32767,32767,1.000000e+000,1.000000e+000,P
2,Vb Example Trace 2,,,V,1.049074e-004,2.840925e-002,0.000000e+000,-32767,32767,1.000000e+000,1.000000e+000,P
3,Vc Trace 4,,,V,3.712631e-004,2.276786e+000,0.000000e+000,-32767,32767,1.000000e+000,1.000000e+000,P
60.000000
1
11740.000000,588
00/00/0000,00:00:00.000000
00/00/0000,00:00:00.000000
ASCII
1.0

```

Line 1 is the AVTS label for the file

Line 2 is 3 channels, 3A are analog, 0D are digital

Lines 3-5 are the three trace identities

Line 6 is system frequency 60 Hz.

Line 7 is one sampling frequency assigned

Line 8 is 11740.000000 samples per second, 588 samples per 0.05 seconds per trace

Lines 9-10 are date and time stamps for recorded data (none for manual created example)

Line 11 ASCII cfg file type

The *Example Trace 1.dat* file.

```
1,0,0,-270,0
2,0,0,-270,0
3,0,925,-220,-30
4,0,925,14369,-4509
5,0,2776,17602,-4509
6,0,2776,17602,-4447
7,0,2776,17602,-901
8,0,4628,17627,-901
9,0,4628,19768,-828
10,0,4628,19768,1803
11,0,6479,19805,1947
12,0,6479,21934,6012
13,0,6479,21934,6012
14,0,6479,21934,6012
15,0,8330,21986,6299
16,0,8330,24101,12024
17,0,8330,24101,12024
18,0,10181,24101,11937
19,0,10181,24167,10521
20,0,10181,26267,10521
21,0,10181,26267,10521
571,0,10181,21393,-5711
572,0,10181,21393,-5711
573,0,7960,18754,-5711
574,0,7960,18685,-5711
575,0,7960,18685,-5711
576,0,6109,18685,-5711
577,0,6109,15496,-5711
578,0,6109,15435,-5711
579,0,6109,15435,-5711
580,0,3887,15435,-5711
581,0,3887,15435,-5711
582,0,3887,10075,-5711
583,0,3887,10019,-5711
584,0,2036,10019,-5711
585,0,2036,10019,-5711
586,0,2036,3542,-5711
587,0,-925,3520,-5711
588,0,-925,3520,-5711
```

Column 1 is sample number for each trace

Column 3 is data for trace 1 sample by sample

Column 4 is data for trace 2 sample by sample

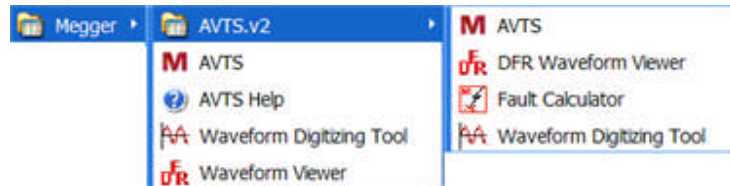
Column 5 is data for trace 3 sample by sample

14. Call \*.cfg file with DFR Waveform Viewer, convert to HEX, load to MPRT and playback.

## DFR Waveform Viewer...

## DFR Waveform Viewer...

The **DFR (Digital Fault Record) Waveform Viewer** is a separate software program which may be started directly from the Windows *Start/All Programs Megger* list. AVTS does not need to be opened to use this program. However, the last used test instrument and available amplifiers will be read from the AVTS database and used by DFR Waveform Viewer



The program allows conversion of **IEEE Standard COMTRADE** files to PULSAR and MPRT Hex files.

EMPT and ATP files may also be converted in like manner. The COMTRADE files are typically generated by a fault recorder vendor's software option. The three files created from the original fault record are the \*.hdr, \*.cfg and \*.dat files. The \*.hdr file is not needed conversion of the \*.dat files needed for conversion to hexadecimal files. The \*.cfg file determines the parameters to be applied to the raw data contained in the \*.dat files.

**Note:** The FOUR\_K.CFG and FOUR\_K.DAT files used in the examples are installed in the C:\Program Files\Megger\AVTSv2\Waveforms folder during the installation of AVTS.

The **COMTRADE Config** file may contain fault records containing the Voltage and Current data for all phases related to several substation lines. The user is able to select a maximum of six phases of Voltages and six phases of Currents to be played back, test set capable dependence.

When the viewer is selected, the dialog opens blank and titled Untitled - DfrWaveView dialog. Use of the DfrWaveView Menu and Toolbar will be necessary to continue use of the program.

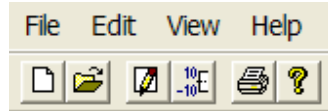
The DFRRecorder New Convert Test or sub-dialog displays the converted files for edit and download to the test equipment.

The COMTRADE To Hexadecimal Conversion dialog, a sub-dialog to the Digital Fault Recorder Testing dialog, is to convert the COMTRADE fault recorder files to Hex format for download and playback.







The DfrWaveView dialog displays, from the converted Hex files, the graphic display of the interconnected samples.

## DfrWaveView Menu and Toolbar

The **Menu and Toolbar** functions are to open existing or new files to view, edit and playback in the MPRT or PULSAR test systems.



The **Toolbar** icons are, left to right,

- New**  Create a new document.
- Open**  Open an existing document.
- Edit Test**  Edit the test data.
- Adjust Scale**  Adjust the plot scale extents.
- Print**  Print the active document.
- About**  Display program information, version number and copyright data.

### File menu items

**New** option opens the Digital Fault Recorder Testing sub-dialog to create a new COMTRADE test file.

**Open** option displays the Windows Open dialog defaulted to the AVTS Waveforms subfolder to select a previously created test file. The file will have the form \*.TST.

Print option displays the Windows Print dialog for selection of the printer name and printer setup options.

**Note:** A \*.TST file must be open for Print to be selected.

**Print Setup** option displays the Windows Print Setup dialog for selection of the printer setup options.

*List of previous opened test files* area will display up to six test files last opened.

**Exit** option will close down the DfrWaveView program.

### Edit menu items

Test option displays the Digital Fault Recorder Testing sub-dialog for edit of a selected \*.TST file and/or download for playback.

**Note:** A \*.TST file must be open for Print to be selected.

### View menu items

**Time** option (default) displays the data samples horizontal axis in milliseconds. The option alternates with the Samples option.

**Samples** option displays the data samples horizontal axis by sample number. The option alternates with the Time option.

**Adjust Scale** option opens the Plot Scale sub-dialog for edit.

**Tool Bar** option, if checked, displays the Menu Toolbar.

**Status Bar** option, if checked, displays the Status bar at the bottom of the dialog.



### Help menu items

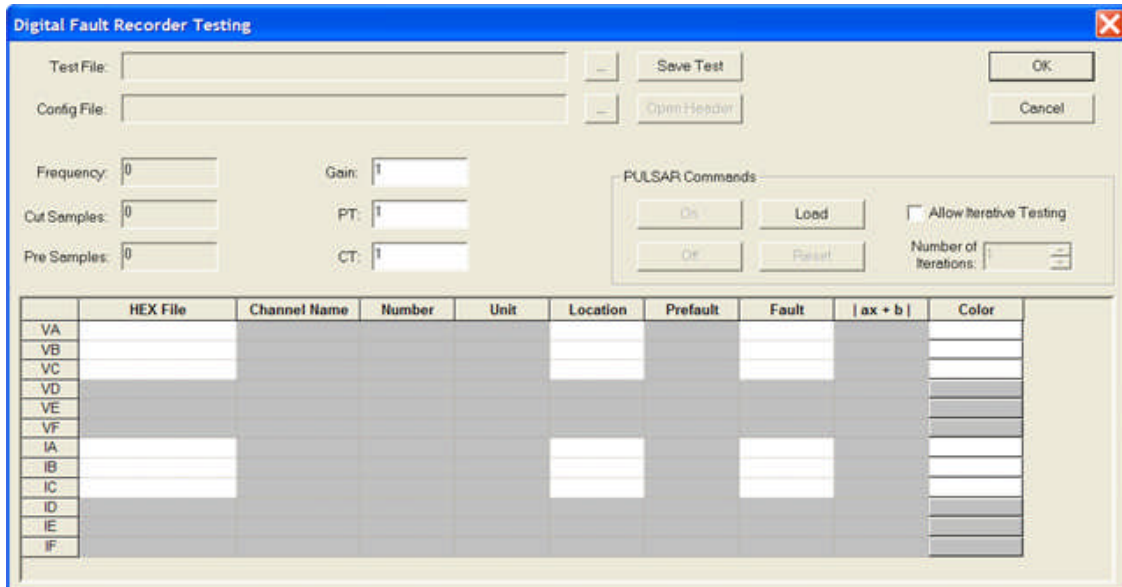
**Help Topics** option opens the DFR Waveform Viewer Help files.

**About DFR Waveform Viewer...** option displays the program version number and date.

## Digital Fault Recorder Testing

### Digital Fault Recorder Testing

The **Digital Fault Recorder Testing** sub-dialog is opened with selection of either the File/New or Edit/Test option. Either the New  or Edit Test  tools will also open the sub-dialog.



	HEX File	Channel Name	Number	Unit	Location	Prefault	Fault	ax + b	Color
VA									
VB									
VC									
VD									
VE									
VF									
IA									
IB									
IC									
ID									
IE									
IF									



Playback Test Files may be created by

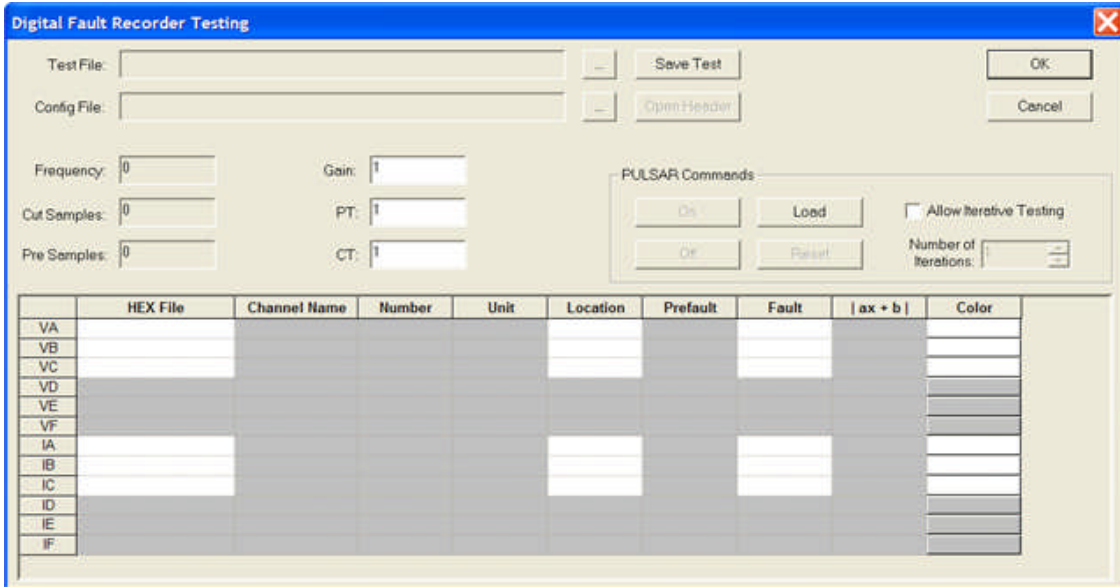
**Convert** an existing COMTRADE \*.cfg to hexadecimal files for view and playback. Save Test the converted file with a unique name. DFRRecorder New Convert Test.

**Open** an existing converted COMTRADE \*.TST file and select other voltages and currents not displayed in the current file. Save the edited \*.TST to a new unique name.

Select the **Config File**: browse button to open the COMTRADE to Hexadecimal Conversion sub-dialog. Select the Edit CFG browse button to open a \*.cfg file in notepad to edit data directly.

## DFRecorder New Convert Test

The **Digital Fault Recorder Testing** sub-dialog is opened with selection of either the File/New or Edit/Test option. Either the New  or Edit Test  tools will also open the sub-dialog.

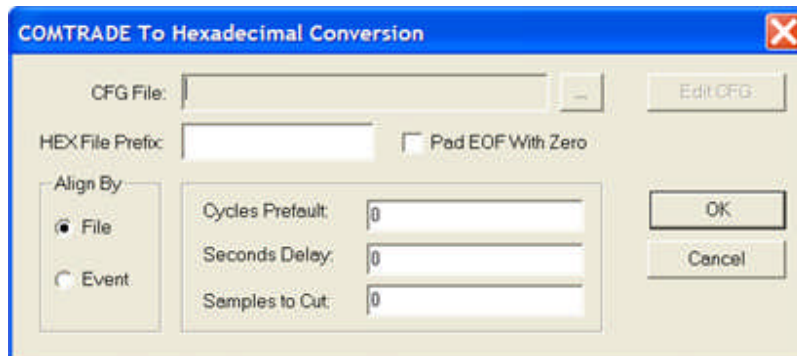


**Playback Test Files** may be created by

**Convert** an existing COMTRADE \*.cfg to hexadecimal files for view and playback. Save Test the converted file with a unique name.

Open an existing converted COMTRADE \*.TST file and select other voltages and currents not displayed in the current file. Save the edited \*.TST to a new unique name.

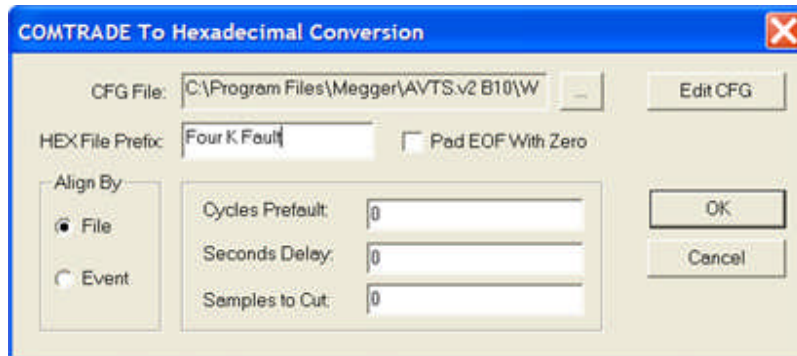
Select the **Config File**: browse button to open the **COMTRADE to Hexadecimal Conversion** sub-dialog. Select the Edit CFG browse button to open a \*.cfg file in notepad to edit data directly.



### Convert COMTRADE

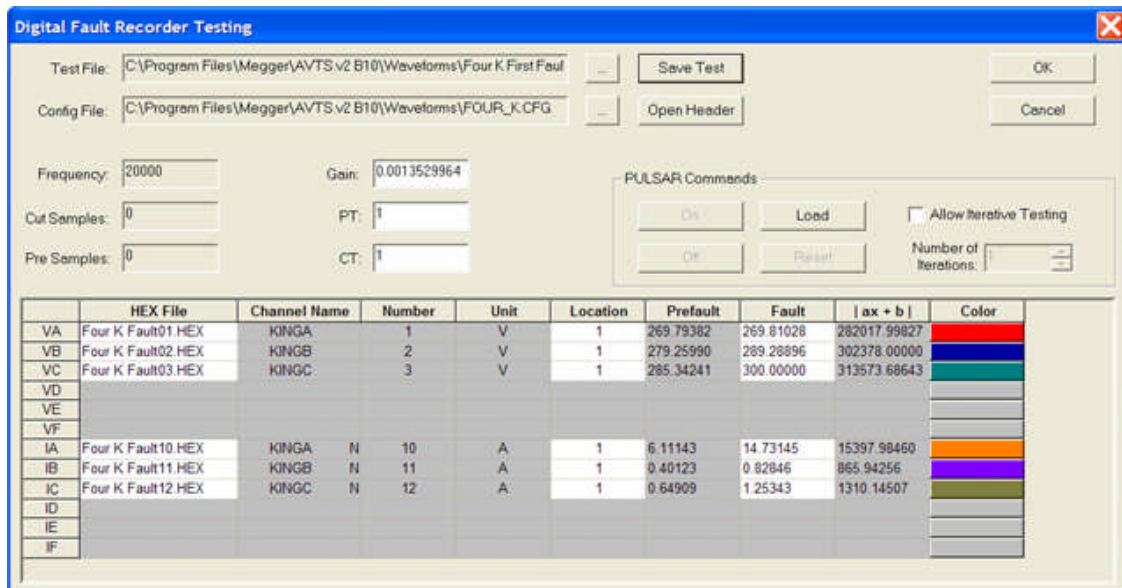
Select **Config File**: browse button to open the COMTRADE to Hexadecimal Conversion sub-dialog.

Select the CFG File: browse button to open the Windows Open dialog to select the \*.cfg file to be converted. The **HEX File Prefix: edit field** must be filled with a label to identify the file for later recall.



Select **OK** to generate the conversion and return to the Digital Fault Recorder Testing dialog.

The *Digital Fault Recorder Testing* dialog is populated with data for selection by the user.



**Note:** There is no initial data visible. The data will fill in as selected.

The **Test File:** field is blank.

The **Config File:** field contains the path and name of the converted \*.cfg file.

**Config file data** from the conversion

**Sample Rate:** data field contains the value of the samples per second of data saved in the \*.dat file. This value is read from the \*.cfg file.

**Cut Samples:** data field contains the number of samples that were cut from the Samples to Cut entry field in the COMTRADE To Hexadecimal Conversion dialog prior to conversion.

**Pre Samples:** edit field contains the number of samples that were added in the Cycles Prefault edit field in the COMTRADE To Hexadecimal Conversion dialog prior to conversion.

**Gain:** edit field contains the numeric value determined by the voltage or current channel with the maximum values in the Fault and [ax + b] columns. The record channel with the maximum data point will produce a value for that channel at the amplifier maximum output. All other channels will be scaled by the same ratio.

In this example, the Gain: has the value = 0.00135299649927776, producing Vc amplifier Fault value = 300.00.

**Note:** The user may choose to reduce this value slightly in order to have the channels not playback an amplifier maximum amplitude. In this example, changing the gain to 0.013 produces a Vc amplifier Fault value = 288.24910.

**PT:** edit field contains the potential transformer ratio which was used to make the recording. If the required channels are identified as secondary values in the \*.cfg file, this number may be set to 1.

**CT:** edit field contains the current transformer ratio which was used to make the recording. If the required channels are identified as secondary values in the \*.cfg file, this number may be set to 1.

**Channels fault data fields**

**Note:** The number of active Voltage and Current channels is determined by the number of phases set in the \*.cfg file.

**HEX File** column has a *drop down list* for each active Voltage and Current phase. The software segregates the Voltage from the Current hex files and displays voltages and currents only in their respective data rows.

	HEX File	Channel Name
VA		
VB		
VC		
VD		
VE		
VF		
IA		
IB		
IC	Four K Fault10.HEX	
ID	Four K Fault11.HEX	
IE	Four K Fault12.HEX	
IF	Four K Fault13.HEX	
	Four K Fault14.HEX	
	Four K Fault15.HEX	

**Channel Name** column fields contain the name for data field derived in the \*.cfg file.

**Number** column fields are the channel number assigned to the Channel name in the \*.cfg file.

**Unit** column fields display the Voltage and Current units as assigned in the \*.cfg file. These may be KV, KA, mv, MV, etc.

**Location** column field is the wave location number in the associated MPRT or PULSAR amplifier. Wave location 1 (usual default for a 4096 sample file) may contain a file up to a 4096 sample data file size. The location number will be 33 for a 64K sample file and 255 for a 128K sample file.

**Note:** Each amplifier location may contain a 4K file stored for use.

**Prefault** column field displays the prefault data magnitudes in the \*.dat file. The amplifier will be that value in agreement to the value [ax + b].

**Fault** column field displays the fault data magnitudes in the \*.dat file. The amplifier will be that value in agreement to the value [ax + b].

**[ax + b]** column field displays the linear relationship between recorded data and stored data as represented by the equation  $y=ax+b$ . The data file is scanned for the maximum amplitude and the absolute value is displayed here. These are the maximum amplitudes recorded for each channel in the displayed units. The maximum value will be aligned in the Fault column as the maximum amplitude available from the corresponding test set amplifier, whether it be a voltage or a current amplifier.

Color column buttons display the color palate for color selection for each waveform when viewed in the DfrWaveView dialog. Selection is made by clicking in the color field and then choosing the desired color from the color display dialog. The color selected will then appear in the Color field for that channel.

**Instrument Commands**

**Load** button sends all the file data to the test instrument and enables the On, Off and Reset buttons.

**On** button turns the test set on.

**Off** button turns the test set off.



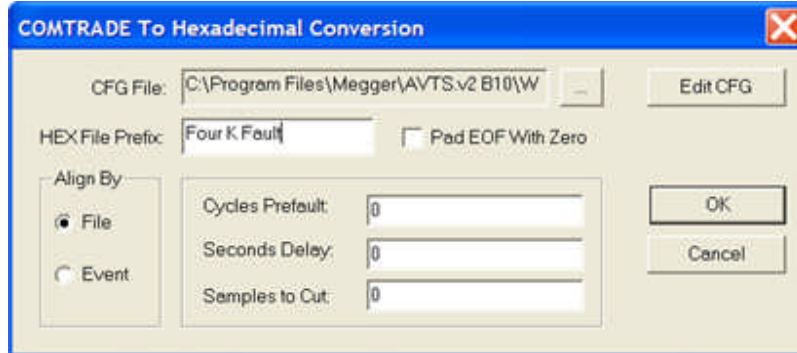
**Reset** button performs a soft reset on the test set to clear all buffers. Load may be required to perform an additional test with the active data.

**Allow Iterative Testing** check box allows for repeated tests of the data files loaded into the test set. There is no delay from one test to the next repeated test.

**Number of Iterations** edit field determines how many times the test is performed by the test set. The scroll buttons allow for numeric entry or the number may be entered with the keyboard.

## COMTRADE To Hexidecimal Conversion

The **COMTRADE To Hexadecimal Conversion** dialog is used to create a new set of hex files or to edit the existing COMTRADE \*.cfg file.

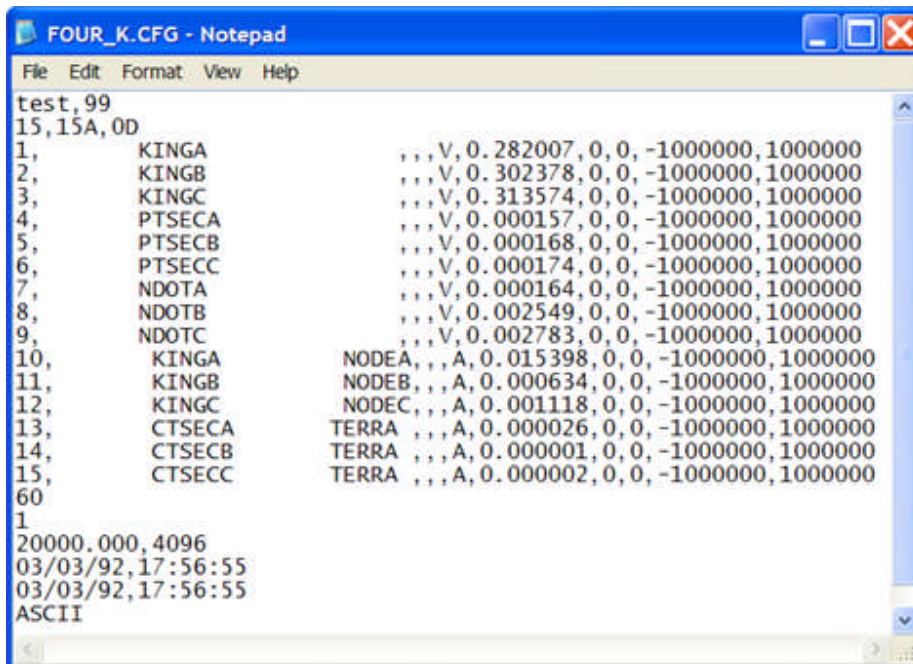


Typically, the user will type the HEX file name and accept the default check boxes and the blank fields as a first trial run of the conversion and playback. Selection of the OK option button returns the Digital Fault Recorder Testing dialog.

### The upper area

The **CFG File:** browse button is to locate the \*.cfg file to be converted or edited. The CFG File: field displays the selected file name and path.

The **Edit CFG** browse button opens \*.cfg file displayed into Windows Notepad for edit purposes.



The **HEX File Prefix:** edit field must have a label inserted by the user to identify in the Waveforms folder the files to be saved from the conversion to hex.

**Pad EOF With Zero** check box is left unchecked for MPRT. The box should be checked for Pulsar firmware version 1.30 or earlier. If checked, the program will convert the COMTRADE data and append null data to create data files the size of a Pulsar wave location.

### Align By

**File** check box selects alignment with the first time tag in the configuration file and is aligned with the first sample in the data file. This is the default option.

**Event** check box selects alignment with the second time tag in the configuration file (the event trigger) and is aligned by the correct number of samples from the front of the data file to the trigger sample.

Data is not in any case thrown away, nor does the prefault sinewave overwrite the data which occurs between the first sample in the file and the time stamp. This check box alters with the File check box.

#### Time block edit fields

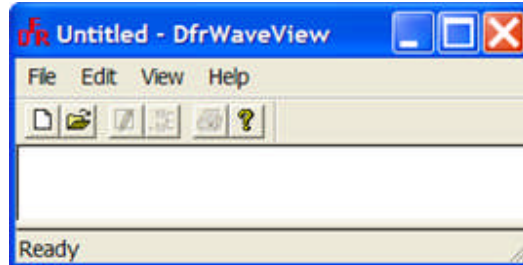
**Cycles Prefault** edit field is to enter the number of cycles desired. This allows the operator to "add" additional time cycles to the original fault record, which might be needed for relay polarization.

**Seconds Delay** edit field is to enter the number of seconds from the first sample to an imaginary trigger. The time will be added to any actual trigger found in the configuration file if in the Align by Event mode. If in the Align by File mode, distance will be measured from the front of the file. The number of seconds from the first sample to an imaginary trigger. Will be added to any actual trigger found in the configuration file if in the Align by Event mode.

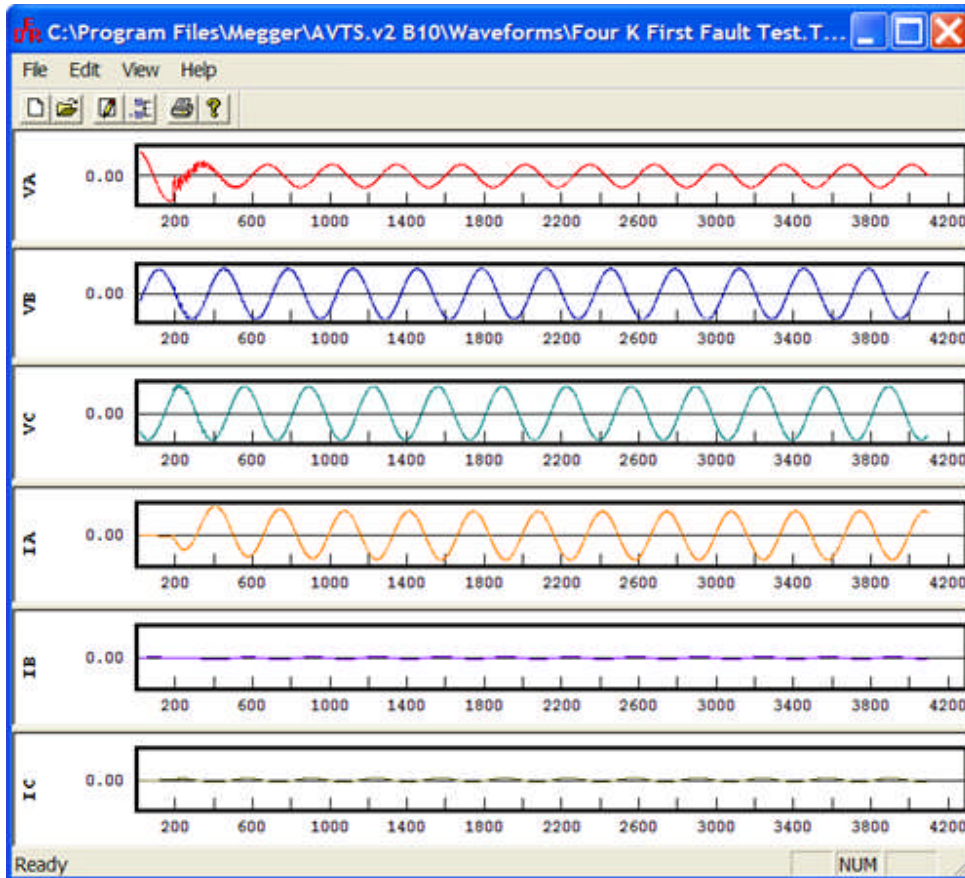
**Samples to Cut** edit field is to enter the number of additional samples subtracted from the prefault to fine tune any alignment. This feature may be used to fine tune any alignment of the recorded fault data to represent the data that might be located at other locations in the power system.

## DfrWaveView

The **Tools/DFR Waveform Viewer...** menu item opens a *DfrWaveView* dialog ready to import a new application or an application previously configured and saved.

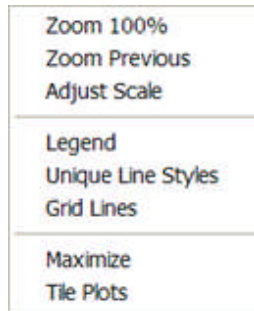


Recall of a previously saved \*.tst will open the test files and display them graphically in the *DfrWaveView* dialog.



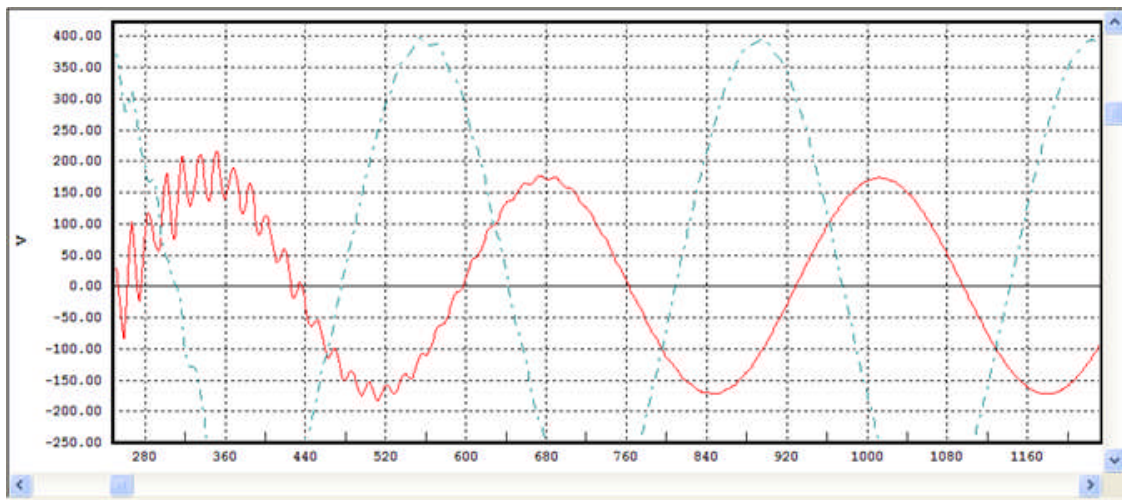
All DfrWaveView Menu and Toolbar items are available for use.

Right mouse click into any graph will produce a *menu dialog* for editing that graph display.



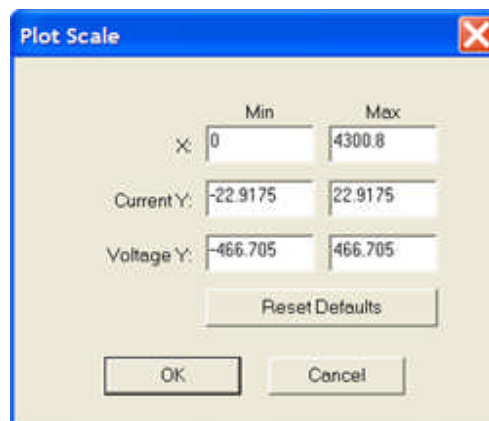
**Zoom 100%** displays the selected graph with all data; the maximum zoom out display.

The user may **Zoom-in** a section of a selected graph with a left mouse click, hold, drag a box to cover the are to be zoomed and release the mouse.

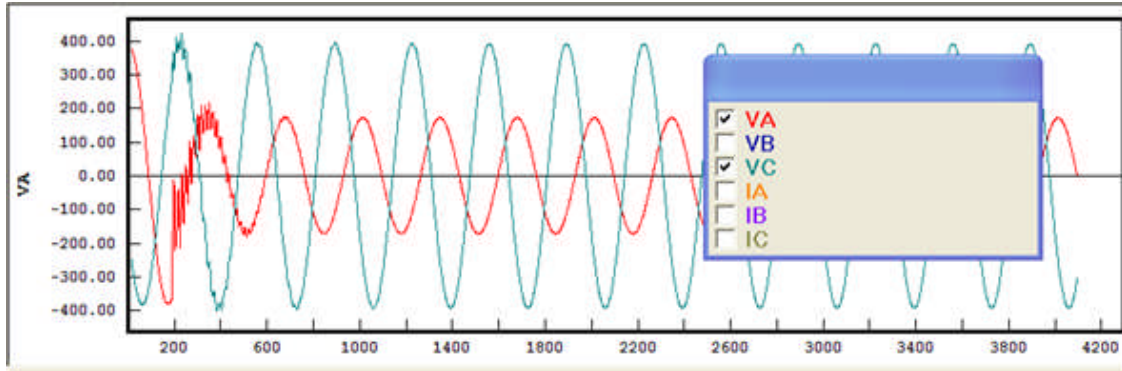


**Zoom Previous** returns to the previous zoomed display.

**Adjust Scale** opens the *Plot Scale* dialog for edit to the graph vertical and horizontal scales. Any adjustment made and OK clicked will affect all graphs.



**Legend** opens the *Legend* dialog in the selected graph with check boxes for all the displayed graphs. Each box checked will overlay those selected in the selected graph.



If Voltage and Current graphs are overlaid, the Voltage amplitude scale will appear on the left and the Current amplitude on the right.

**Unique Line Styles** creates different line for each waveform in the display overlay. This is useful when a monochrome monitor is used and the individual graph colors are not available.

**Grid Lines** produces a grid for the major horizontal and vertical scales values.

**Maximize Plots** zooms the selected graph to full view in the DfrWaveView dialog.

**Tile Plots** produces a single plot of each waveform one below the other in separate tiles. Tiled plots are default when the DfrWaveView dialog is first opened with a \*.tst file.

## Appendix A - Basic Language Syntax

### Appendix A - Basic Language Syntax

**BASIC syntax** has been used for many years and is easily grasped by the user. The introduction of AVTS Relay Test Software and the use of Controls pre formatted with the BASIC syntax functions used by older software "macros" has eliminated the need of creating test procedures with BASIC.

However, there are Controls in which BASIC type syntax may be placed to provide definition to the operation and final result of the test.

The syntax uses Variables and Expressions with some formatting.

The following topics demonstrate the syntax and formatting

- Variables

- Expressions

- PULSAR String Commands

- PULSAR Command Set may be necessary for input to some Controls in a test procedure.

## Variables

**Variable**    Significance

**debug**        When set to 1 single steps through macros.

**debug2**        When set to 1 enables single stepping through macros. Also single steps through successive steps of equation solver.

**debug3**        Steps through each message sent to an instrument.

**debug4**        Performs no function, reserved.

**start**         Contains the value at which all ramps and steps will start. If it is equal to '0' at the time of the ramp or step the variable will be set to the evaluation of the 'start' field of 'Test Specifications'.

**stop**         Contains the value at which all ramps and step will stop. If it is equal to '0' at the time of the ramp or step the variable will be set to the evaluation of the 'stop' field of the 'Test Specifications'.

**inc**           Contains the increment value for all ramps. Prior to the first ramp it is set equal to the evaluation of the 'Increment #1' field of Test Specifications Screen'.

**inc2**         delay2 Some macros support a second ramp of the pickup point. Those macros assign these two variables to 'inc' and 'delay' if 'inc2' is non-zero. 'Inc2' is set to the value of 'Increment #2' on the first ramp.

**pulse**        Contains the time in cycles that the outputs will stay at their fault condition. Prior to the first ramp it is set equal to the evaluation of the 'Pulse Time' field on 'Test Specifications'.

**delay**        Contains the delay time in cycles between each increment of a ramp. Prior to the first ramp it is set equal to the evaluation of the 'delay' field of 'Test Specifications'.

**min\_rang**     Contains the minimum acceptable tolerance. Prior to the first ramp or step it is set to the evaluation of the 'minimum range' field of Test Specifications.

**max\_rang**     Contains the maximum acceptable tolerance. Prior to the first ramp or pulse it is set to the evaluation of the 'maximum range' field of Test Specifications.

**idc**           Contains the value of D.C. current channel.

**idc\_on**        If set to '1' it will turn the 'DC' current channel on. If set to '0' it will turn the 'DC' current channel off.

**idc\_tap**       Sets the 'DC' current channel tap to 'tap 1', 'tap 2' or 'tap 3' if the value of 'idc\_tap' is set to '1', '2' or '3' respectively.

**idc\_ramp**     If set to '1', it will ramp the 'DC' current channel .

**vdc**           Contains the value of the D.C. voltage channel.

**vdc\_on**        If set to '1' it will turn the 'DC' voltage channel on. If set to '0' it will turn the 'DC' voltage channel off.

**vdc\_tap**       Sets the 'DC' voltage channel tap to 'tap 1', 'tap 2' or 'tap 3' if the value of 'vdc\_tap' is set to '1', '2' or '3' respectively.

**vdc\_ramp**     If set to '1', it will ramp the 'DC' voltage channel .

The following variables, up to fa\_ramp, apply to phase 'A' parameters and are repeated for the 'B' and 'C' phases, and 'dc' channels.

**ia**            Contains the value of the 'a' phase current channel.

**iap**           Contains value of the 'a' phase current phase angle.

**ia\_on**         If set to '1' it will turn the 'a' phase current on. If set to '0' it will turn the 'a' phase current off.

**ia\_tap**        Sets the 'a' phase current tap to 'tap 1', 'tap 2' or 'tap 3' if the value of 'ia\_tap' is set to '1', '2' or '3' respectively.

**ia\_ramp**      If set to '1', it will ramp the 'a' phase current.

**iap\_ramp**     If set to '1', it will ramp 'a' current phase angle.

**va**            Contains the value of the 'a' phase voltage channel.



- vap** Contains value of the 'a' phase voltage phase angle.
- va\_on** If set to '1' it will turn the 'a' phase voltage on. If set to '0' it will turn the 'a' phase voltage off.
- va\_tap** Sets the 'a' phase voltage tap to 'tap 1', 'tap 2' or 'tap 3' if the value of 'va\_tap' is set to '1', '2' or '3' respectively.
- va\_ramp** If set to '1', it will ramp the 'a' phase voltage.
- vap\_ramp** If set to '1', 'a' phase voltage phase angle ramps.
- fa** Frequency of channel 'a'.
- fa\_ramp** For future frequency ramps. Set to 1 if ramp checked.
- value** Final reading during a ramp.
- auto\_off** Flag to turn off all current channels after each ramp and step.
- start\_on (start\_on\_step)** Flag which will start an EPOCH timer when the outputs change from their 'start' to 'stop' values during a 'step' function.
- no\_toe** No time out enable. If this flag is set, the test will not be aborted if a ramp reaches stop without detecting a contact closure.
- normal** This variable determines what type of activity will trigger pickup in the ramp and step commands. No commands are sent to reconfigure any binding posts. If set to '1' it indicates a normally closed contact. If set to '0' it indicates a normally open contact.
- Note: The value returned by monitor is not affected.**
- notmrcmd** When this variable is not zero the 'Set' command will not reconfigure PULSAR's timer module or, if the translator is disabled, clear any latches when checking for monitor activity.
- mon\_addr** On program initialization the 'mon\_addr' is set to 1. This tells the program to only look at the first unit on the address screen or logical unit 1. If set to 0 the unit# of first unit to respond to contact activity will be returned in the variable 'add\_ret'. PULSAR: The monitors are bit encoded with the bottom most binding post associated with the lowest bit.
- add\_ret** This variable holds the unit number of the last Epoch unit to detect contact action. The monitors to be checked are defined in the variable 'mon\_addr'. PULSAR uses the bit encoding scheme used by the variable 'mon\_addr'. This variable is updated only if activity on the binding posts selected by 'mon\_addr' is detected.
- latch\_nolatch\_nc** If either is set to 1, then contact monitor will be latched to 1 on the first change of state. The latch is cleared anytime the variable 'monitor' is accessed and at the beginning of a ramp. If both variables are set to zero then a 'ramp' or 'monitor' will return the contact state when sampled. The hardware is configured with the settings when the 'Set' command is issued. PULSAR: If latch\_nc is set the binding post is configured for 'Removed' operation. See also Monitor
- mon\_volt** PULSAR ONLY, TRANSLATOR DISABLED: If not zero then the 'Set' command will configure each PULSAR monitor binding post, as set in 'mon\_addr', to sense voltage instead of continuity. If the variables 'tstart' or 'tstop' are not zero then this command will be ignored by each associated binding post. See also Monitor
- mon1stop** PULSAR ONLY: If not zero then the 'Set' command will configure the PULSAR monitor binding post on the bottom for timer stop instead of monitor. See also Monitor
- pdebtime** PULSAR ONLY, TRANSLATOR DISABLED: If loaded with a legal contact debounce time between .1 and 1638.3 milliseconds, the 'Set' commands will program PULSARs timer module with the specified debounce time.
- tlatch** Sets a timers latch
- 1 = startx
  - 2 = stopx
  - 3 = all
  - 4 = none
- This variable can also be set in the Test Specifications dialog box.
- tstart tstop** Sets a timer start and stop

- 0 = no mode set
- 1 = n.o.modes
- 2 = n.c.
- 3 = voltage applied
- 4 = voltage removed

**mon2conf & mon3conf** PULSAR ONLY, TRANSLATOR DISABLED: If the stop or start binding post is selected by mon\_addr then the 'Set' command will attempt to configure the binding post according to value in one of these variables. If tstart or tstop are set below they will override the settings these variables select.

Sets the start and stop timers to monitor mode.

- 0 = no mode set
- 1 = n.o.                unlatched
- 2 = n.c.                unlatched
- 3 = voltage applied    unlatched
- 4 = voltage removed   unlatched
- 5 = n.o.                latched
- 6 = n.c.                latched
- 7 = voltage applied    latched
- 8 = voltage removed   latched

**time** Returns either the time in an EPOCH timer, the PULSAR timer or the PC.

**epoch\_ti** (epoch\_timer) A flag used in some macros to indicate that an EPOCH or PULSAR timer should be queried for the operate time; it is not a key variable but many macros use it.

**no\_clear** With this variable set to 1, the GPIB bus command 'clear 7' is not issued prior to macro execution. Use the calculator or create a test to set this flag prior to the execution of the test so that the software won't issue a 'clear 7' to the devices that are on the bus. Since 'clear 7' normally initializes a device to its power-on state, this variable was created to allow a test that requires DC control voltage to remain on through out a sequence of tests.

**fix** This variable sets the number of decimal places to be used when numbers are stored in variables. The default number of decimal places is 2. This variable is write only and cannot be read.

**pulsar\_u** This variable can be used to determine the type of the selected Unit.

- 0= Epoch,
- 1-5=PULSAR
  - on 1=COM1, 2=COM2, 3=COM3, 4=COM4 or 5=GPIB.
- 6=TCM915
- 7=General Purpose
- 1=Selected an Empty unit

**pxlatoff** This variable is 0 when PULSAR and the translator are enabled and can be used to disable the translator during a test by making it non zero. At the start of each test this variable is set to reflect the translator settings found in the Addresses configuration dialog. When using this variable the user should first test pulsar\_u to determine if PULSAR is in use. The translator can be re-enabled by clearing pxlatoff.

"EXAMPLE"

```
set      'activate Test Specifications settings
if pulsar_u then  'if PULSAR is on the other end
                svxlat=pxlatoff  'save current setting
                pxlatoff=1      'disable translator
```

```

>"waimi,s!" 'wait, synch immediate
>"v1,a50,waic5,a100,waic5,a50,waic5!" 'pulse
monitor 'check monitor and for errors
pxlatoff=svxlat 'restore previous setting
end if
off :clear :local
end

```

**nodistab** If nodistab is non-zero then a distortion alarm will not automatically cause a test to abort a test. A ramp will automatically stop when the condition is detected. After a ramp or an output command ">" that could cause distortion has executed the users macro should then check the variable "distort" to determine if the ramp ended in distortion, reset the variable "distort" to zero and if in ramp not save any results. If the user macro is for an EPOCH the user should also remedy the cause and issue "CHANGE GPIB 70" to re-enable the run buffer.

**distort** This variable is set to one when PULSAR report a distortion error. The user is responsible for resetting the variable to zero.

**armtcm** This variable determines whether ramp and set also control the first TCM-915 listed in Devices Addressing.

0 = ignore TCM-915.

1 = send Sequencer Reset at start of ramp to first TCM-915.

2 = send Sequencer Reset and GET to first TCM-915 at start of ramp

**dfrtoevn** This is the number of seconds the last waveform downloaded by DFRLoad has to play before the trigger event in the original COMTRADE file is reached.

**dfrtofil** This is the number of seconds the last waveform downloaded by DFRLoad has to play before the data in the original COMTRADE file is reached.

**dfrtonul** This is the number of seconds the last waveform downloaded by DFRLoad has to play before the null data added after the COMTRADE data has been reached.

**dfrwidth** This is the number of seconds the last waveform downloaded by DFRLoad has to play before the end of the data is reached.

**escape up\_arrow down\_arr** These variables are set by the enter\_at function when one of these keys are pressed. These variables should then be reset to zero by the users macro. The users program can use these variables to determine what action the program should take.

**csrlin** The line being processed by Print\_Using. Starts at 0.

**yes, no** Set on exit by "wait key" to 1 if the user hit yes or no. The user should clear these variables prior to calling "wait key" if they are to be used.

**page** During printouts this is the page number currently being printed.

**new\_page** This variable is set to one at the beginning of each new page. The user should clear this variable after the new page has been detected.

**no\_op** This variable is set to one by the ramp command if it did not detect the appropriate monitor before ramping past the value set in stop.

**ph\_ph** A flag used by some macros which states that this is a phase to phase test; it is not a key variable but many macros use it.

**three\_ph** Flag which states that this is a three phase test; it is not a key variable but many macros use it.

### String variables

<b>label\$</b>	The name of the current test.
<b>file\$</b>	Original fields in the Header.
<b>name\$</b>	Original fields in the Header.

<b>date\$</b>	Original fields in the Header.
<b>location\$</b>	Original fields in the Header.
<b>substati\$</b>	Original fields in the Header.
<b>breaker\$</b>	Original fields in the Header.
<b>model\$</b>	Original fields in the Header.
<b>rly_man\$</b>	Original fields in the Header.
<b>co_numbe\$</b>	Original fields in the Header.
<b>hdtname\$</b>	Title of original name field in the Header.
<b>hdtdate\$</b>	Title of original date field in the Header.
<b>hdtloc\$</b>	Title of original location field in the Header.
<b>hdtsub\$</b>	Title of original substation field in the Header.
<b>hdtbrkr\$</b>	Title of original breaker field in the Header.
<b>hdtmodl\$</b>	Title of original model field in the Header.
<b>hdtmanu\$</b>	Title of original manufactures field in the Header.
<b>hdtcono\$</b>	Title of original Co. Number field in the Header.
<b>usr1\$</b>	line 1 of relay settings
<b>usr2\$</b>	line 2 of relay settings
<b>usr3\$</b>	line 3 of relay settings
<b>trigtim\$</b>	Trigger time sent to time reference
<b>smprate\$</b>	Sample rate reported by last Load Transient or DFRLoad
<b>hdtnote\$</b>	note label
<b>noteln1\$</b>	line 1 through 6 of user note. Each line is 76 characters wide with white space add to the end of each line
<b>noteln2\$</b>	ref noteln1\$
<b>noteln3\$</b>	ref noteln1\$
<b>noteln4\$</b>	ref noteln1\$
<b>noteln5\$</b>	ref noteln1\$
<b>noteln6\$</b>	ref noteln1\$

## Expressions

The interpreter supports both string and numeric variables. String variables are distinguished by ending in a '\$'. A variable can be used before a value is assigned to it. An unassigned numeric variable has a value of zero. An unassigned string variable will be an empty string. Variables can also be checked from the Variable Watch window in the AVTS Test tab..

Numeric variables can be manipulated with the following;

<b>operator</b>	description
<b>+</b>	adds two numeric constants/variables
<b>-</b>	subtracts two numeric constants/variables
<b>*</b>	multiplies two numeric constants/variables
<b>/</b>	divides two numeric constants/variables
<b>^</b>	raises left side to the power of the right side.
<b>()</b>	designates order of equation solution.
<b>=</b>	assigns the left side variable to the right side.
<b>:</b>	separates multiple expressions on a single line.

<b>function</b>	descriptions
<b>sqr(x)</b>	returns the square root of 'x'
<b>sin(x)</b>	returns the sine of 'x' degrees
<b>cos(x)</b>	returns the cosine of 'x' degrees
<b>tan(x)</b>	returns the tangent of 'x' degrees
<b>asin(x)</b>	returns the arc-sine of 'x' (in degrees)
<b>acos(x)</b>	returns the arc-cosine of 'x' (in degrees)
<b>atan(x)</b>	returns the arc-tangent of 'x' (in degrees)
<b>log(x)</b>	returns the log base 10 of 'x'.
<b>ln(x)</b>	returns the natural log of 'x'.
<b>val(x\$)</b>	converts a numeric string to a number

**Warning: Always use lower case letters for these functions; the interpreter does not allow upper case for these functions and will yield incorrect calculations.**

String variables can be manipulated by the following:

<b>operator</b>	description
<b>+</b>	concatenates two strings together.
<b>=</b>	assigns the string on the right to the variable on the left.
<b>:</b>	separates multiple expressions on a single line.
<b>" "</b>	double quotes enclose a string of characters.
<b>\"</b>	inside a quoted string signifies that a single character should be entered after the quote mark and the quote mark entered after the single character..

<b>function</b>	description
<b>str\$(x)</b>	converts a number to a string

**Warning: Always use lower case letters for this function; the interpreter does not allow upper case and will yield incorrect calculations.**

Example:

`a = 3 / 2 * 4` results in a equal to 6; e.g.,  $3 \times 4 / 2$

`b = 3 / (2 * 4)` results in b equal to 0.375

`a = 10; b$ = str$(a)` results in 'b\$' being set to the string "10.00"

`c = val(b$)` results in 'c' being set to the number 10

'Logical expression' can consist of constants or variables and may be correlated by the logical operators.

**operator**      function

**=**    true if left operand equals the right operand.

**<**    true if left operand is less than the right operand.

**>**    true if the left operand is greater than the right.

**<>**   true if the left operand does not equal the right.

**and**   used to group two logical expressions together; the overall condition can only be true when both are true (must include a space on either side).

**or**    used to group two logical expressions together; the overall condition is true when either of the two is true (must include a space on either side).

## **Appendix B - MPRT Command Set**

## **Appendix B - MPRT Command Set**

This Topic and all sub books and topics are under edit and will be complete in the next release of AVTS Help

Command Set Variable Definitions

Command Set Constant Definitions

Command Set Procedures

Interface Procedures

Appendix A - Timer Flash Memory Parameter Sector Physical Layout

Appendix B - VI-Generator Flash Memory Parameter Sector Physical Layout

Appendix C - VI-Generator Correction Factor Flash Sector Physical Layout

## Command Set Constant Definitions

### 3.1 Constant Definitions

**Unit reset fpga address** – Hex address 0x25c. Writing any value to this FPGA address will cause a reset to occur on the Timer and all VI generators installed.

**Starting mode Immediate** – Constant value Hex 0x02 define under Parameter Set mode command in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Number of Voltage Generator (NVG)** – Constant value equals 4. The number of Voltage Generators defined for Genesis Test Set.

**Number of Current Generator (NVC)** – Constant value equals 8. The number of Current Generators defined for Genesis Test Set.

**Applied** – Constant value equals 0. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Removed** – Constant value equals 1. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Change** – Constant value equals 2. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Latched** – Constant value equals 0. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Unlatched** – Constant value equals 1. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Enable** – Constant value equals 1. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Disable** – Constant value equals 0. Used as a parameter for the Timer's Voltage Mode and Continuity Mode command codes defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**SPIR** – Address Offset of the Serial port interrupt Status register defined in section 3.7.3 Serial Port Interrupt Status Register (SPIR), of the ISA FPGA Description document.

**SPRR** – Address Offset of the Serial port receiver data register defined in section 3.7.4 Serial Port Transmitter Data Register (SPRR), of the ISA FPGA Description document.

**SPTR** – Address Offset of the Serial port transmitter data register defined in section 3.7.5 Serial Port Transmitter Data Register (SPTR), of the ISA FPGA Description document.

**VI-Monitor-Control-and-Data-Register** – Address Offset of the VI monitor register defined in section, 6.0 VI Monitor, of the ISA FPGA Description document.

**Even-parity-bit** – Even bit defined in section 11. V/I Generator Communication and Commands, of the Genesis Test Set Firmware Specification.

**Odd-parity-bit** – Odd parity bit defined in section 11. V/I Generator Communication and Commands, of the Genesis Test Set Firmware Specification.

**Generator-select-cmd** – Defined in section 11.7 Generator parameter sets, of the Genesis Test Set Firmware Specification.

**Write-index-cmd** – Defined in section 11.7 Generator parameter sets, of the Genesis Test Set Firmware Specification.

**Control-index-cmd** – Defined in section 11.7 Generator parameter sets, of the Genesis Test Set Firmware Specification.



**VI-mode-select-cmd** – Defined in section 11.7 Generator parameter sets, of the Genesis Test Set Firmware Specification

**External Input select-cmd** – Defined in section 11.7 Generator parameter sets, of the Genesis Test Set Firmware Specification.

**Allowable-Distortion-Percentage-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Amplifier-on-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Amplifier-off-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Queued-parameter-Dc-level-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Frequency-of-fundamental-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Filter-time-constant-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification

**Queued-Sine-wave-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Queued-Half-Sine-wave-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Queued-Rectangle-wave-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Queued-Exponential-decay-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Cycle-repetition-count-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Filter-time-constant-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Waveform-playback-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**DFR-playback-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Range-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Parameter-set-mode-cmd** – Defined in section 11.8 Queued Parameter Set Commands, of the Genesis Test Set Firmware Specification.

**Table-write-address-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.

**DFR-sample-rate-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.

**Zero-Waveform-table-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.

**Waveform-Dc-level-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification

**Dc-level-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification

**Sine-wave-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.

**Exponential-decay-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.

- Half-sine-wave-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.
- Rectangle-wave-cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.
- Store-waveform-or-DFR-cmd cmd** – Defined in section 11.9 Waveform and DFR parameters, of the Genesis Test Set Firmware Specification.
- Query-Output-cmd** – Defined in section 11.10 Query and reports to the host, of the Genesis Test Set Firmware Specification.
- Query-parameter-storage-cmd** – Defined in section 11.10 Query and reports to the host, of the Genesis Test Set Firmware Specification.
- Query-DSP-Firmware-Info-cmd** – Defined in section 11.10 Query and reports to the host, of the Genesis Test Set Firmware Specification.
- Link-test-cmd** – Defined in section 11.11 Query and reports to the host, of the Genesis Test Set Firmware Specification.
- Set-debounce-time-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Voltage-Mode-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Start -Timer–Gate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Stop-Timer–Gate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Monitor-Gate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Ignore-Gate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Horn-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Auto-Off-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Software-Start-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Software-Stop-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Cycles-Frequency-Stop-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Display-elapsed-time-in-Seconds-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Aux-Contact-On/Off-cmd- cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Voltage Mode-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Continuity-Mode-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Timer-Reset-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Timer-Sync-Test-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Input-Contact-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

- Binary-Input-Voltage-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Input-don't-care-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Input-Closed-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Input-Open-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Output-Closed-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Output-Open-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Output-don't-care-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Input-Logic-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Input-Wait-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Output-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Output-True-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Output-False-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Output-Wait-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Recorder-Mask-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Recorder-Stop-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Trigger-Software-Start-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Trigger-Voltage-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Trigger-Contact-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Binary-Trigger-Input-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Reset-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Sequencer-Clear-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Load-Horn-Enable-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Load-Horn-Disable-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Load-Shutdown-Enable-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.
- Load-Shutdown-Disable-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Synchronize-Immediate-Mode-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Synchronization-Phase-Reference-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Synchronization-Phase-Reference-Self-Zero-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Synchronization-Self-Zero-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Synchronization-External-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Line-Synchronization-Initiate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Line-Synchronization-Off-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-time-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-post-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Cycles-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Mode-Immediate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Mode-Sync-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Input-Clock-Source-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Input-Clock-Source-External-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Wait-Input-Clock-Source-Internal-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Unit-Reset-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Local-Mode-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Restore-Phase-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Time-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Gate-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Status-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Monitor-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Binary-Input-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Binary-Output-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Event-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Event-Length-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Collected-Record-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Time-Tag-Binary-Input-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Timer-Revision-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Query-Record-Number-cmd** – Defined in section 11.11 Timer Command Set, of the Genesis Test Set Firmware Specification.

**Default-amplitude** – A 4 byte real value. Value equal 1.

**Default-harmonic** – A 1-byte integer value. Value equal 1.

**Default-phase-angle** – A 4 byte real value. Value equal 0.

## Command Set Variable Definitions

### 3.2 Variable Definitions

**Select Voltage Generator (SVG)** – Is an array of four elements index by values 1 through 4. This array will be used to determine the Voltage Generators that will be selected to receive subsequent Generator Control, and Generator Waveform Commands. Setting the contents of this array to 0 indicates that the Current Generator as not been selected. Setting the contents to 1 indicates that the generator has been selected.

**Select Current Generator (SCG)** – Is an array of eight elements index by values 1 through 8. This array will be used to determine the Current Generators that will be selected to receive subsequent Generator Control, and Generator Waveform Commands. Setting the contents of this array to 0 indicates that the Current Generator as not been selected. Setting the contents to 1 indicates that the generator has been selected.

**Cmd-string** – This variable is used to send command information to the V/I generators.

**Command** – This variable is used to send command information to the V/I generators.

**I** - This variable is used as an index value for sequencing through arrays.

**Packet-length** – The length of the packet being sent to the V/I Generator UART hardware interface.

**Port-address** – This variable is set to one of the I/O Memory Map addresses defined in section 2.4 I/O Memory Map of the ISA FPGA Description document.

**Sample-table-index** – This variable is used as an index into the sample table.

**Max-sample-table-index-array** – A 4 byte sample area length returned via a query command.

**Write-index** – This variable is used as an index into the parameter queue table.

**Write-index-array** – Is an array of four elements index by values 1 through 4. Contains indexes into the parameter queue table.

**Max-write-index-array** – Is an array of four elements index by values 1 through 4. Contains 1 byte parameter queue lengths returned via a query command.

**Out-byte** – This variable is used to output a byte of data to a specified Port-address.

**In-byte** – This variable is used to input a byte of data from a specified Port-address.

**Amplitude** – This variable store the AC-amplitude value used in generating waveforms.

**Phase-angle** – This variable store the phase value used in generating waveforms.

**DC-offset** – This variable store the DC offset of the exponent at time zero.

**Generator-number** – This variable store the current selected generator number (0 or 1) and is used in computing the physical address for waveform and DFR.

**Time-constant** – This variable store the time constant of the exponential decay in milliseconds.

**DFR-length** – This variable store the length of the DFR. Used with the DFR playback command.

**Generator-type** – This variable is set to "C" or "V" to determine the type of operation, Current or Voltage Generator, which is currently being executed. Used when a command executes more than one command string.

**Query-results** – this variable stores the results of a Query Command received from the Timer and VI generator interface.

**Number-of-bytes-to-read** – Utilized by the query commands to return variable length query results from the Timer and VI generator interface.

**Query-opcode** – Variable used by the query command to validate the communication between the Host, Timer, and VI Generators interface.

**Configuration-Query-Results** – Variable used to store the results of a Configuration Query command.

**Number-of-VI-Generators-present** – Variable used to count the number of VI generators installed in the Genesis unit.

**Toggle-Battery-Simulator-Reset** – Variable used to reset the Battery Simulator.



## Command Set Procedures

## Command Set Procedures

### 6.0 RTS Command Set Procedures

**RTS** Commands consist of ASCII characters. One or more commands may appear on the same line. The Comma (,), and semicolon (;) are used as terminators for commands.

**Note:** Release of AVTS Ver. 3.0 and the latest MPRT Versio required the additions to this Command Set and are documented in the Command Set Procedures - New group of topics.

The **RTS** command processor will parse the command line calling the procedures outlined below. Each of the procedures below will process only a single command. For instance the command V1C2V3; would call procedure in section 6.1 **Generator Select Commands** three times with the parameters V1, C2, and V3.

- Generator Select Commands
- Generator Control Commands
- Generator Waveform Commands
- Timer Control Commands
- Binary Input Commands
- Binary Output Commands
- Binary Trigger Commands
- Load Error Action Commands
- Synchronization Commands
- Wait Commands
- Restart Commands
- Query Commands
- Query Device Commands
- Query Time Tag Commands
- Query Measurement Value Commands
- Horn Commands
- Directory and File Commands
- Battery Simulator Commands
- Vector Scope Configuration Commands



## Generator Select Commands

### 6.1 Generator Select Commands

Select the current or voltage generators that will be in effect for subsequent Generator Control and Generator Waveform Commands

#### 6.1.1 Select Voltage Generator Command (SVGC) ( V<generator> )

This procedure processes a single Select Voltage Generator Command (V<generator>). This is accomplished by setting and de-selecting locations in the variable arrays SVG and SCG.

If the parameter <generator> equals 0, all four Voltage generators will be selected if the corresponding current generator has not been selected previously.

When <generator> is in the range (1-4) the voltage generator identified by <generator> is selected in the corresponding current generator is de-selected.

This procedure also sets the variable **Generator-number** equal to 0 after a voltage generator has been selected. Table 6.1-1 depicts the valid Select Voltage Generator Command.

Table 6.1-1 Voltage Generator Select Command

COMMAND	Description
V<generator>	Where <generator> is in the range 0-4. When Generator equals 0, All four-voltage generators are selected.

#### 6.1.2 Select Current Generator Command (SCGC) ( C<generator> )

This procedure processes a single Select Current Generator Command (C<generator>). This is accomplished by setting and de-selecting locations in the variable arrays SCG and SVG.

If the parameter <generator> equals 0, all eight Current generators will be selected if the corresponding voltage generator (V1, V2, V3,V4) has not been selected previously.

When <generator> is in the range (0-8) the current generator identified by <generator> is selected in the corresponding voltage generator is de-selected.

This procedure also sets the variable **Generator-number** equal to 0 or 1 depending on the current generator selected.

Table 6.1-2 Current Generator Select Command

COMMAND	Description
C<generator>	Where <generator> is in the range 0-8. When Generator equal 0. All eight current generators are selected.

## Generator Control Commands

### 6.2 Generator Control Commands (GCC)

These commands specify the settings for V/I Generators that have previously been selected. Input to this procedure is **Generator control command**. This procedure executes the command for any of the voltage and current generators that were previously selected by a Generator select command. For any of the commands defined in section 6.2.1 through 6.2.16 the VI-generators will acknowledge receiving the command by returning the commands opcode to the Single board computer.

First, this procedure sets the variable **Generator-type** equal to "V" for voltage, in looks in the array variable **Select Voltage Generator (SVG)** to identify which of the 4 (**Number of Voltage Generator (NVG)**) voltage generators will receive the command. If the location in the array **Select Voltage Generator (SVG)** indexed by the variable (**I**) is set to 1, then, The **Generator-number is set to 0**, and one of the procedure in This section is executed.

Some commands only initialize variable, these commands set the variable **Command** equal to NULL. Procedure 7.3 **Execute Selected Voltage Generator Command (ESVGBC)** is called to perform the command for those commands in section 6.2.1 through 6.2.16 that set the variable **Command** equal to something other than NULL. An Algorithm that depicts this process is demonstrated in Figure 6.3 (a) and Figure 6.3 (c).

Next, this procedure sets the variable **Generator-type** equal to "C" for current, looks in the array variable **Select Current Generator (SCG)** to identify which of the 8 (**Number of Current Generator (NCG)**) current generators will receive the command. If the location in the array **Select Current Generator (SVG)** indexed by the variable (**I**) is set to 1, then, Then **Generator-number** is set to 1, if I equal 1,2, 3 or 4. If I equals 5,6,7 or 8 then **Generator-number** is set to 0, otherwise Generator-number is set to 2. Once the **Generator-number** is determined, one of the procedures in section 6.2.1 through 6.2.11 is executed.

Some commands only initialize variable, these commands set the variable **Command** equal to NULL. Procedure 7.4 **Execute Selected Current Generator Command (ESCGBC)** is called to perform the command for those commands in section 6.2.1 through 6.2.11 that set the variable **Command** equal to something other than NULL. An Algorithm that depicts this process is demonstrated in Figure 6.3 (b) and Figure 6.3 (c).

#### 6.2.1 DC Amplitude Offset Voltage or Current Command (DCOAMP) ( D<real> )

This procedure processes a single DC Amplitude Offset Voltage or Current Command "D<real>". The parameter <real> is expressed in Amps when Current generators are selected, and Volts when Voltage Generators are selected. Table 6.2-1 depicts the DC Amplitude Offset Voltage or Current Command.

Table 6.2-1 - DC Amplitudes Offset Voltage or Current Command

COMMAND	Description
D<real>	Set selected Voltage or Current Generator DC amplitude

Set the variable **Command** equal to the Dc-level-cmd. Convert the parameter <real> into a 4-byte real voltage or current value and append the results to **Command**.

#### 6.2.2 AC Amplitude Offset Voltage or Current Command (ACAOVCC) ( A<real> )

This procedure processes a single AC Amplitude Offset Voltage or Current Command "A<real>". The parameter <real> is expressed in Amps when Current generators are selected, and Volts when Voltage Generators are selected. Table 6.2-2 depicts the AC Amplitude Offset Voltage or Current Command.

Table 6.2-2 - AC Amplitudes Offset Voltage or Current Command

COMMAND	Description
A<real>	Set selected Voltage or Current Generator AC amplitude

Convert the parameter <real> into a 4-byte real amplitude value and store the results and the variable Amplitude. Set the variable Command equal to the NULL.

**6.2.3 Phase Command (PC) ( P<real> )**

This procedure processes a single Phase Command “P<real>”. The parameter <real> is expressed in Degrees. Table 6.2-3 depicts the Phase Command.

Table 6.2-3 - Phase Command

COMMAND	Description
P<real>	Set selected Voltage or Current Generator Phase

Convert the parameter <real> into a 4-byte real phase value and store the results and the variable Phase-angle. Set the variable Command equal to the NULL.

**6.2.4 Frequency Command (FC) ( F<real> )**

This procedure processes a single Frequency Command “F<real>”. The parameter <real> is expressed in Hertz. Table 6.2-4 depicts the Frequency Command.

Table 6.2-4 - Frequency Command

COMMAND	Description
F<real>	Set selected Voltage or Current Generator Frequency

Set the variable Command equal to the Frequency-of-fundamental-cmd. Convert the parameter <real> into a 4-byte real voltage value and append the results to Command.

**6.2.5 Output ON Command (OOC) ( ON )**

This procedure processes a single Output ON Command. Table 6.2-5 depicts the Generator Output ON Command.

Table 6.2-5 - Generator Output ON Command

COMMAND	Description
ON	Set selected Voltage or Current Generator Output ON

Set the variable Command equal to the Amplifier-on-cmd.

**6.2.6 Output OFF Command (OOF) ( OF<state> )**

This procedure processes a single Output OFF Command. Table 6.2-6 depicts the Generator Output OF Command.

Table 6.2-6 - Generator Output OFF Command

COMMAND Description

COMMAND	Description
OF<state>	Set selected Voltage or Current Generator Output OFF Where <state> is optional

If <state> is not specified or is <state> equals 1 or 2, then set the variable Command equal to the Amplifier-off-cmd. If state equals 3, then call routine 7.8 Output Off State three (OOST) in set the variable Command equal to NULL.

**6.2.7 Range Auto-range Command (SGRA) ( RA<range> )**

This procedure processes single Range Auto-range Command “RA<range>”. The parameter <range> is one of the digits 0-3. Table 6.2-7 depicts the Range Auto-range of Command.

Table 6.2-7 - Range Auto-range Command

COMMAND	Description
RA<range>	<p>Set selected Voltage or Current Generator Range to the following:</p> <p>0 = Auto-range will cause both Current and Voltage generators to automatically select a range.</p> <p>1= Low Voltage Range              30V on the 150VA Generator</p> <p>2= Medium Voltage Range              150V on the 150VA Generator</p> <p>3=High Voltage Range              300V on the 150VA generator</p> <p>4 =Low Current Range              6A on the 200VA Generator</p> <p>5 =Medium Current Range              15A on the 200VA Generator</p> <p>6=High Current Range              30A on the 200VA Generator</p>

Set the variable Command equal to the Range-cmd. Append the one byte <range> parameter to the Command.

**6.2.8 Waveform Location Command (WL) ( WAV<location> )**

This procedure processes single Waveform Location Command “WAV<location>”. The parameter <location> is a one byte offset into the waveform table. Table 6.2-8 depicts the Waveform Location Command.

Table 6.2-8 - Waveform Location Command

<b>COMMAND</b>	<b>Description</b>
WAV<location>	Waveform Location  Choose <location> 0 to select a data playback waveform.

Set the variable Cmd-string equal to the Parameter-set-mode-cmd. Append a 1-byte value 8 (waveform mode) to Cmd-string. Call the procedure 7.10 Generator Type Select (GTS) with the Cmd-string.

Call routine 7.6 Compute and Generate Waveform or DFR Physical Address (CAGWPA) with the WAV <location> to select the waveforms physical location. This routine also sets the variable DFR-length depending on the value of the DFR / WAV<location> and returns the physical starting memory address in the variable sample-table-index.

Set the variable Command equal to the Waveform-playback-cmd. Append the 4-byte real Amplitude value to the Command. Append the 4-byte real Phase-angle value to the Command. Append the 4-byte unsigned table start address in the variable sample-table-index to the Command.

**6.2.9 DFR Location Command (DFRL) ( DFR<location> )**

This procedure processes single DFR Location Command “DFR<location>”. The parameter <location> is a one byte offset into the DFR table. Table 6.2-9 depicts the DFR Location Command.

Table 6.2-9 - DFR Location Command

<b>COMMAND</b>	<b>Description</b>
DFR<location>	DFR Location  Choose <location> 0 to select a data playback Waveform.

Set the variable Cmd-string equal to the Parameter-set-mode-cmd. Append a 1-byte value 10 (DFR mode) to Cmd-string. Call the procedure 7.10 Generator Type Select (GTS) with the Cmd-string.

Call routine 7.6 Compute and Generate Waveform or DFR Physical Address (CAGWPA) with the DFR <location> to select the DFRs physical location. This routine also sets the variable DFR-length depending on the value of the DFR / WAV<location> and returns the physical starting memory address in the variable sample-table-index.

Set the variable Command equal to the DFR-playback-cmd. Append the 4-byte real Amplitude value to the Command. Append the 4-byte unsigned table start address in the variable sample-table-index to the Command. Append the 4-byte unsigned integer stored in the variable DFR-length to the Command.

**6.2.10 Exponential restart Command (ERC) ( E )**

This procedure processes single Exponential restart Command “E”. Table 6.2-10 depicts the Exponential restart Command.

Table 6.2-10 - Exponential restart Command

<b>COMMAND</b>	<b>Description</b>
E	Exponential restart

Set the variable Command equal to the Exponential-decay-cmd. Append the 4-byte real DC-offset value to the Command.

Append the 4-byte real Time-constant value to the Command.

**6.2.11 Sample Rate Command (SRC) ( SA<rate> )**

This procedure processes single Sample Rate Command “SA<real rate>”. Table 6.2-11 depicts the Sample Rate Command.

Table 6.2-11 - Sample Rate Command

COMMAND	Description
SA<rate>	Sample Rate  Where <rate> is a value from 0 to 20,000Hz

Set the variable Command equal to the DFR-sample-rate-cmd. Append the 4-byte real <rate> to the Command.

**6.2.12 Filter Time Constant (FTC) ( FT<filter-time-constant> )**

This procedure processes a single Filter Time Constant Command “FT<filter-time-constant> ”. Table 6.2-12 depicts the Sample Rate Command.

Table 6.2-12 - Filter Time Constant Command

COMMAND	Description
FT<filter-time-constant>	Filter Time Constant  Where < filter-time-constant > four byte real value

Set the variable Command equal to the Filter-time-constant-cmd. Append the 4-byte real <filter-time-constant > to the Command.

**6.2.13 Waveform Location On the Fly Command (WLF) ( WAVF<location> )**

This procedure processes single Waveform Location Command “WAVF<location>”. The parameter <location> is a one byte offset into the waveform table. Table 6.2-13 depicts the Waveform Location Command.

Table 6.2-13 – Waveform On the Fly Location Command

COMMAND	Description
WAVF<location>	Waveform Location  Where <location> 0 -127.

Set the variable Cmd-string equal to the Parameter-set-mode-cmd. Append a 1-byte value 4 (on the Fly mode) to Cmd-string. Call the procedure 7.10 Generator Type Select (GTS) with the Cmd-string.

Set the variable Command equal to the Control-index-cmd. Append a 1-byte <location> value to the Command.

**6.2.14 Allowable Distortion Percentage Command (ADP) ( ADP<real> )**

This procedure processes single Allowable Distortion Percentage Command “ADP<real>”. The parameter <real> is a one 4-byte real allowable distortion percentage. Table 6.2-14 depicts the Allowable Distortion Percentage Command.

Table 6.2-14 – Allowable Distortion Percentage Command

COMMAND	Description
ADP<real>	Allowable Distortion Percentage  Where <real> is a 4-byte Allowable Distortion Percentage.

Set the variable Command equal to the Allowable-Distortion-Percentage-cmd. Append the 4-byte <real> Allowable Distortion Percentage to the Command.

**6.2.15 Enable External Input ( ENEXT )**

This procedure processes single Enable External Input Command “ENEXT”. Table 6.2-15 depicts the Allowable Distortion Percentage Command.

Table 6.2-15 – Enable External Input Command

COMMAND	Description
ENEXT	Enable External Input  This command sets the default Range to LOW range.

Set the variable Command equal to the Generator0 or Generator1 External Input select-cmd. Append the 1-byte mode select value of zero to the Command.

**6.2.16 Disable External Input ( DSEXT )**

This procedure processes single Disable External Input Command “DSEXT”. Table 6.2-16 depicts the Allowable Distortion Percentage Command.

Table 6.2-16 – Disable External Input Command

COMMAND	Description
DSEXT	Disable External Input  This command sets the default Range to AUTO range.

Set the variable Command equal to the Generator0 or Generator1 External Input select-cmd. Append the 1-byte mode select value of one to the Command.

## Generator Waveform Commands

### 6.3 Generator Waveform Commands

These commands specify the waveforms for the selected V/I generators. Similar to the Generator control commands. For any of the commands defined in section 6.3.1 through 6.3.10 the VI-generators will acknowledge receiving the command by returning the commands opcode to the Single board computer.

The Generator Waveform commands in sections 6.3.1 through 6.3.10 are processed utilizing the algorithms specified in Figure 6.3 (a) Figure 6.3 (b), and Figure 6.3 (c).

#### 6.3.1 Sinusoidal Analytical Waveform Command (SAW) ( WAV<location>SI1:2:3 )

This procedure processes a single Sinusoidal Analytical Waveform Command "WAV<location>SI#1:#2:#3". Table 6.3-1 depicts the Sinusoidal analytical Waveform Command.

Table 6.3-1 - Sinusoidal Analytical Waveform Command

COMMAND	Description
WAV<location>SI#1:#2:#3	<p>Sinusoidal analytical waveform</p> <p>Where:</p> <p>#1 - amplitude</p> <p>#2 - harmonic</p> <p>#3 - phase-angle</p>

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION.

The parameters #1 (amplitude), #2 (harmonic), and #3 (phase-angle) are option. If any of the parameters are missing, the default values (Default-amplitude, Default-harmonic, Default-phase-angle) for the parameter are used. If only one parameter (#1) is specified that parameter will be mapped to the amplitude parameter. If two of the parameters (#1,#2) are specified, they will be mapped to the amplitude and harmonic parameters respectively. Similar, when all three are specified, they will be mapped to the amplitude, harmonic, and phase-angle parameters respectively.

Set the variable Command equal to the Sine-wave-cmd. Append the 1-byte integer harmonic , 4-byte real amplitude value, and 4-byte real phase-angle value to the Command.

#### 6.3.2 Half-wave Waveform Command (HWC) ( WAV<location>H#1:#2 )

This procedure processes a single Half-wave Waveform Command "WAV<location>H#1:#2". Table 6.3-2 depicts the Half-wave Waveform Command.

Table 6.3-2 - Half-wave Waveform Command



COMMAND	Description
WAV<location>H#1:#2	Half-wave waveform  Where:  #1 - amplitude  #2 – phase-angle

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION.

The parameters #1 (amplitude), and #2 (phase-angle) are option. If any of the parameters are missing, the default values (Default-amplitude, Default-phase-angle) for the parameter are used. If only one parameter (#1) is specified that parameter will be mapped to the amplitude parameter. Similar, when both are specified, they will be mapped to the amplitude, and phase-angle parameters respectively.

Set Command equal to the Half-sine-wave-cmd. Append the 4-byte real amplitude value to the Command. Append the 4-byte real phase-angle value to the Command.

**6.3.3 Square Waveform Command (SWC) ( WAV<location>SQ#1:#2:#3 )**

This procedure processes single Square Waveform command “WAV<location>SQ#1#2#3”. Table 6.3-3 depicts the Square Waveform Command.

Table 6.3-3 - Square waveform Commands

COMMAND	Description
WAV<location>SQ#1:#2:#3	Square waveform  #1 - duty  #2 - amplitude  #3 – phase-angle

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION.

The parameters #2 (amplitude), and #3 (phase-angle) are option. If parameters #2 and #3 are missing, the default values (Default-amplitude, Default-phase-angle) for the parameter are used. If only one parameter (#2) is specified that parameter will be mapped to the amplitude parameter. When both are specified, they will be mapped to the amplitude, and phase-angle parameters respectively.

Set Command equal to the Rectangle-wave-cmd. Append the 4-byte real duty value to the Command. Append the 4-byte real amplitude value to the Command. Append the 4-byte real phase-angle value to the Command.

**6.3.4 Clear waveform Command (CWC) ( WAV<location>C )**

This procedure processes single Clear Waveform command “WAV<location>C”. Table 6.3-4 depicts the Clear Waveform Command.

Table 6.3-4 - Clear waveform Commands

COMMAND	Description
WAV<location>C	Clear selected waveform

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION AND COMPUTE THE DFR-LENGTH.

SET THE VARIABLE COMMAND EQUAL TO THE ZERO-WAVEFORM-TABLE-CMD. APPEND 4 BYTE DFR-LENGTH TO COMMAND.

**6.3.5 Data Playback Command (DPC)** ( WAV<lo>D<terminator><binary data> )

This procedure processes single Data Playback command “WAV<location>D<terminator> <binary data>”. Table 6.3-5 depicts the Data Playback Command.

Table 6.3-5 – Data Playback Command

COMMAND	Description
WAV<location>D<terminator><binary data>	Data Playback Where <terminator> is a comma or semicolon, and <binary data>: indicates binary data in hex format (COMTRADE binary) with the least significant byte of each word sent first. This data is 4096 words (16 bits) in length, 131,072 or 262144 words in length and must immediately follow, non-stop, the next comma or semicolon <terminator>.

Call routine 7.6 **Compute and Generate Waveform or DFR Physical Address (CAGWPA)** with the WAV <location> to select the waveforms physical location. This routine also sets the variable DFR-length depending on the value of the DFR WAV<location> and returns the physical starting memory address in the variable sample-table-index.

Call routine 7.7. Store DFR binary Data to store the <binary data> in the physical memory address.

**6.3.6 Variable Block Data Playback Command (VDPC)** (WAV<loc>V<s><t><data>)

This procedure processes single Data Playback command “WAV<location>V<size><terminator><binary data>”. Table 6.3-6 depicts the Data Playback Command.

Table 6.3-6 – Variable Block Data Playback Command

COMMAND	Description
WAV<location>V<size><terminator><binary data>	Data Playback Where <size> is the number of bytes contained in the <binary data>. <terminator> is a comma or semicolon, and <binary data>: indicates binary data in hex format (COMTRADE binary) with the least significant byte of each word sent first. This data is <size> words (16 bits) in length, and must immediately follow, non-stop, the next comma or semicolon <terminator>.

Call routine 7.6 **Compute and Generate Waveform or DFR Physical Address (CAGWPA)** with the WAV <location> to select the waveforms physical location. This routine also sets the variable DFR-length depending on the value of the DFR WAV<location> and returns the physical starting memory address in the variable sample-table-index.

Set the variable DFR-length equal to <size>, and call routine 7.7. Store DFR binary Data to store the <binary data> in the physical memory address.

**6.3.7 Set Exponential parameters Command (SEPC) ( WAV<location>E<k>:<t> )**

This procedure processes single Set Exponential parameters Command "WAV<location>E<k>:<t>". Table 6.3-7 depicts the Set Exponential parameters Command.

Table 6.3-7- Set Exponential parameters Command

COMMAND	Description
WAV<location>E<k>:<t>	Set Exponential parameters  Where <k> = DC offset as a fraction of the exponent at time zero  <t>= time constant in seconds

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION.

Sets the variable Command equal to the Exponential-decay-cmd. Set the variable DC-offset equal to the parameter <k>. Append the 4-byte real DC-offset value to the Command.

Set the variable Time-constant equal to the parameter <t>. Append the 4-byte real Time-constant value to the Command.

**6.3.8 DC Amplitude Offset Waveform Command (DCAOW) ( WAV<loc>D<real> )**

This procedure processes a single DC Amplitude Offset Waveform Command "WAV<location>D<real>". The parameter <real> is expressed in Amps when Current generators are selected, and Volts when Voltage Generators are selected. Table 6.3-8 depicts the DC Amplitude Offset Voltage or Current Command.

Table 6.3-8 - DC Amplitudes Offset Waveform Command

COMMAND	Description
WAV<location>D<real>	DC Amplitude Offset Waveform

CALL ROUTINE 7.6 **COMPUTE AND GENERATE WAVEFORM OR DFR PHYSICAL ADDRESS (CAGWPA)** WITH THE WAV <LOCATION> TO SELECT THE WAVEFORMS PHYSICAL LOCATION.

Set the variable Command equal to the Waveform-Dc-level-cmd. Convert the parameter <real> into a 4-byte real voltage or current value and append the results to Command.

**6.3.9 Sinusoidal Analytical Waveform On the FLY Command (SAWF) (WAVF<l>SI1:2:#3:#4)**

This procedure processes a single Sinusoidal Analytical Waveform On the FLY Command "WAVF<location>SI#1:#2:#3:#4". Table 6.3-9 depicts the Sinusoidal analytical Waveform On the FLY Command.

Table 6.3-9 - Sinusoidal Analytical Waveform On the FLY Command

COMMAND	Description
WAVF<location>SI#1:#2:#3:#4	<p>Sinusoidal analytical waveform On the FLY</p> <p>Where:</p> <p>#1 - 1 byte component number (0 thru 3)</p> <p>#2 - harmonic (1 thru50)</p> <p>#3 – amplitude</p> <p>#4 – phase-angle</p> <p>(Note: There are 128 physical locations reserved for on the fly waveforms. Each of these reserved locations has default frequency and dc Amplitude offset values that are initialized at power-up. To set a different frequency, and or dc Amplitude offset value for a on the Fly waveform, a frequency command, <b>6.2.4 Frequency Command (FC)</b>, and 6.2.1, and or <b>DC Amplitude Offset Voltage or Current Command (DCOAMP)</b> must be issued after the on the Fly waveform is stored.</p>

Set the variable write-index equal to the value of <location>. Set the variable Command equal to the Queued-Sine-wave-cmd. Append the 1-byte integer component number, 1-byte integer harmonic, 4-byte real amplitude value, and 4-byte real phase-angle value to the Command.

**6.3.10 Half-wave Waveform On the FLY Command (HWCF)**

WAVF<location>H1:#2 )

This procedure processes a single Half-wave Waveform On the FLY Command “WAVF<location>H#1:#2”. Table 6.3-10 depicts the Half-wave Waveform On the FLY Command.

Table 6.3-10 - Half-wave Waveform On the FLY Command

COMMAND	Description
WAVF<location>H#1:#2	<p>Half-wave waveform On the FLY</p> <p>Where:</p> <p>#1 – amplitude</p> <p>#2 – phase-angle</p> <p>(Note: There are 128 physical locations reserved for on the fly waveforms. Each of these reserved locations has default frequency and dc Amplitude offset values that are initialized at power-up. To set a different frequency, and or dc Amplitude offset value for a on the Fly waveform, a frequency command, <b>6.2.4 Frequency Command (FC)</b>, and 6.2.1, and or <b>DC Amplitude Offset Voltage or Current Command (DCOAMP)</b> must be issued after the on the Fly waveform is stored.</p>

Set the variable write-index equal to the value of <location>. Set Command equal to the Queued-Half-Sine-wave-cmd. Append the 4-byte real amplitude value to the Command. Append the 4-byte real phase-angle value to the Command.

**6.3.11 Square Waveform On the FLY Command (SWCF) (WAVF<loc>SQ1:2:3 )**

This procedure processes single Square Waveform On the FLY command "WAVF<location>SQ<amplitude>:<phase>:<duty>". Table 6.3-11 depicts the Square Waveform On the FLY Command.

Table 6.3-11 - Square waveform On the FLY Commands

COMMAND	Description
WAVF<location>SQ#1:#2:#3	<p>Square waveform On the FLY</p> <p>#1 – amplitude</p> <p>#2 – phase-angle</p> <p>#3 – duty cycle</p> <p>(Note: There are 128 physical locations reserved for on the fly waveforms. Each of these reserved locations has default frequency and dc Amplitude offset values that are initialized at power-up. To set a different frequency, and or dc Amplitude offset value for a on the Fly waveform, a frequency command, <b>6.2.4 Frequency Command (FC)</b>, and 6.2.1, and or <b>DC Amplitude Offset Voltage or Current Command (DCOAMP)</b> must be issued after the on the Fly waveform is stored.</p>

Set the variable write-index equal to the value of <location>. Set Command equal to the Queued-Rectangle-wave-cmd. Append the 4-byte real amplitude value to the Command. Append the 4-byte real phase-angle value to the Command. Append the 4-byte real duty cycle to the Command.

**6.3.12 Set Exponential parameters On the FLY Command (SEPCF) (WAVF<loc>E<k>:<t>)**

This procedure processes single Set Exponential parameters On the FLY Command "WAVF <location>E<k>:<t>". Table 6.3-12 depicts the Set Exponential parameters On the FLY Command.

Table 6.3-12- Set Exponential parameters On the FLY Command

COMMAND	Description
WAVF<location>E<k><t>	<p>Set Exponential parameters On the FLY</p> <p>Where &lt;k&gt; = DC offset as a fraction of the exponent at time zero</p> <p>&lt;t&gt;= time constant in seconds</p> <p>(Note: There are 128 physical locations reserved for on the fly waveforms. Each of these reserved locations has default frequency and dc Amplitude offset values that are initialized at power-up. To set a different frequency, and or dc Amplitude offset value for a on the Fly waveform, a frequency command, <b>6.2.4 Frequency Command (FC)</b>, and 6.2.1, and or <b>DC Amplitude Offset Voltage or Current Command (DCOAMP)</b> must be issued after the on the Fly waveform is stored.</p>

Set the variable write-index equal to the value of <location>. Sets the variable Command equal to the Queued-Exponential-decay-cmd. Set the variable DC-offset equal to the parameter <k>. Append the 4-byte real DC-offset value to the Command.

Set the variable Time-constant equal to the parameter <t>. Append the 4-byte real Time-constant value to the Command.

**6.3.13 DC Amplitude Offset Waveform On the FLY Command (DCAOWF) (WAVF<location>D<real>)**

This procedure processes a single DC Amplitude Offset Waveform On the FLY Command "WAVF<location>D<real>". The parameter <real> is expressed in Amps when Current generators are selected, and Volts when Voltage Generators are selected. Table 6.3-13 depicts the DC Amplitude Offset Voltage or Current Command.

Table 6.3-13 - DC Amplitudes Offset Waveform On the FLY Command

COMMAND	Description
WAVF<location>D<real>	DC Amplitude Offset Waveform On the FLY

Set the variable write-index equal to the value of <location>. Set the variable Command equal to the Queued-parameter-Dc-level-cmd. Convert the parameter <real> into a 4-byte real voltage or current value and append the results to Command.

## Timer Control Commands

### 6.4 Timer Control Commands

#### 6.4.1 Timer Volt Applied Latch Command ( T<post>VAL )

This procedure processes single Timer Volt Applied Latch command “T<post>VAL”. Table 6.4-1 depicts the Timer Volt Applied Latch Command.

Table 6.4-1 - Timer Volt Applied Latch Command

COMMAND	Description
T<post>VAL	Timer Volt Applied Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Applied constant, and one byte Latch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

#### 6.4.2 Timer Volt Applied Un-Latch Command ( T<post>VAU )

This procedure processes single Timer Volt Applied Un-Latch command “T<post>VAU”. Table 6.4-2 depicts the Timer Volt Applied Un-Latch Command.

Table 6.4-2 - Timer Volt Applied Un-Latch Command

COMMAND	Description
T<post>VAU	Timer Volt Applied Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Applied constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

#### 6.4.3 Timer Volt Removed Latch Command ( T<post>VRL )

This procedure processes single Timer Volt Removed Latch command “T<post>VRL”. Table 6.4-3 depicts the Timer Volt Removed Latch Command.

Table 6.4-3 - Timer Volt Removed Latch Command

COMMAND	Description
T<post>VRL	Timer Volt Removed Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Removed constant and one byte Latch constant to the Command (See

section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.4 Timer Volt Removed Un-Latch Command ( T<post>VRU )**

This procedure processes single Timer Volt Removed Un-Latch command “T<post>VAL”. Table 6.4-4 depicts the Timer Volt Removed Un-Latch Command.

Table 6.4-4 - Timer Volt Removed Un-Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>VRU	Timer Volt Removed Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Removed constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.5 Timer Volt Change Latch Command ( T<post>VCL )**

This procedure processes single Timer Volt Change Latch command “T<post>VCL”. Table 6.4-5 depicts the Timer Volt Change Latch Command.

Table 6.4-5 - Timer Volt Change Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>VCL	Timer Volt Change Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Change constant, and one byte Latch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.6 Timer Volt Change Un-Latch Command ( T<post>VCU )**

This procedure processes single Timer Volt Change Un-Latch command “T<post>VCU”. Table 6.4-6 depicts the Timer Volt Change Un-Latch Command.

Table 6.4-6 - Timer Volt Change Un-Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>VCU	Timer Volt Change Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Voltage-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Change constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.7 Timer Continuity Applied Latch Command ( T<post>CAL )**



This procedure processes single Timer Continuity Applied Latch command “T<post>CAL”. Table 6.4-7 depicts the Timer Continuity Applied Latch Command.

Table 6.4-7 - Timer Continuity Applied Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CAL	Timer Continuity Applied Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Applied constant, and one byte Latch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.8 Timer Continuity Applied Un-Latch Command ( T<post>CAU )**

This procedure processes single Timer Continuity Applied Un-Latch command “T<post>CAU”. Table 6.4-8 depicts the Timer Continuity Applied Un-Latch Command.

Table 6.4-8 - Timer Continuity Applied Un-Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CAU	Timer Continuity Applied Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Applied constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.9 Timer Continuity Removed Latch Command ( T<post>CRL )**

This procedure processes single Timer Continuity Removed Latch command “T<post>CRL”. Table 6.4-9 depicts the Timer Continuity Removed Latch Command.

Table 6.4-9 - Timer Continuity Removed Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CRL	Timer Continuity Removed Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Removed constant, and one byte Latch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.10 Timer Continuity Removed Un-Latch Command ( T<post>CRU )**

This procedure processes single Timer Continuity Removed Un-Latch command “T<post>CRU”. Table 6.4-10 depicts the Timer Continuity Removed Un-Latch Command.

Table 6.4-10 - Timer Continuity Removed Un-Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CRU	Timer Continuity Removed Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Removed constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.11 Timer Continuity Change Latch Command ( T<post>CCL )**

This procedure processes single Timer Continuity Change Latch command "T<post>CCL". Table 6.4-11 depicts the Timer Continuity Change Latch Command.

Table 6.4-11 - Timer Continuity Change Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CCL	Timer Continuity Change Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Change constant, and one byte Latch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.12 Timer Continuity Change Un-Latch Command ( T<post>CCU )**

This procedure processes single Timer Continuity Change Un-Latch command "T<post>CCU". Table 6.4-12 depicts the Timer Continuity Change Un-Latch Command.

Table 6.4-12 - Timer Continuity Change Un-Latch Command

<b>COMMAND</b>	<b>Description</b>
T<post>CCU	Timer Continuity Change Un-Latch  Where <post> is in the range 0-16.

Set the variable Command equal to the Continuity-Mode-cmd. Append the one byte <post> number to Command. Append the one byte Change constant, and one byte Unlatch constant to the Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.13 Timer Start Command ( T<post>STA )**

This procedure processes single Timer Start command "T<post>STA". Table 6.4-13 depicts the Timer Start Command.

Table 6.4-13 - Timer Start Command

COMMAND	Description
T<post>STA	Timer Start  Where <post> is in the range 0-16.

Set the variable Command equal to the Start-Timer-Gate-cmd. Append the one byte <post> number to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.14 Timer Stop Command ( T<post>STO )**

This procedure processes single Timer Stop command "T<post>STO". Table 6.4-14 depicts the Timer Stop Command.

Table 6.4-14 - Timer Stop Command

COMMAND	Description
T<post>STO	Timer Stop  Where <post> is in the range 0-16.

Set the variable Command equal to the Stop-Timer-Gate-cmd. Append the one byte <post> number to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.15 Timer Monitor Command ( T<post>M )**

This procedure processes single Timer Monitor command "T<post>M". Table 6.4-15 depicts the Timer Monitor Command.

Table 6.4-15 - Timer Monitor Command

COMMAND	Description
T<post>M	Timer Monitor  Where <post> is in the range 0-16.

Set the variable Command equal to the Monitor-Gate-cmd. Append the one byte <post> number to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.16 Timer Ignore Command ( T<post>I )**

This procedure processes single Timer Ignore command "T<post>I". Table 6.4-16 depicts the Timer Ignore Command.

Table 6.4-16 - Timer Ignore Command

COMMAND	Description
T<post>I	Timer Ignore  Where <post> is in the range 0-16.

Set the variable Command equal to the Ignore-Gate-cmd. Append the one byte <post> number to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.17 Timer Horn Enable Command ( T<post>HE )**

This procedure processes single Timer Horn Enable command “T<post>HE”. Table 6.4-17 depicts the Timer Horn Enable Command.

Table 6.4-17 - Timer Horn Enable Command

<b>COMMAND</b>	<b>Description</b>
T<post>HE	Timer Horn Enable  Where <post> is in the range 0-16.

Set the variable Command equal to the Horn-cmd. Append the one byte <post> number to Command. Append the one byte Enable constant to Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.18 Timer Horn Disable Command ( T<post>HD )**

This procedure processes single Timer Horn Disable command “T<post>HD”. Table 6.4-18 depicts the Timer Horn Disable Command.

Table 6.4-18 - Timer Horn Disable Command

<b>COMMAND</b>	<b>Description</b>
T<post>HD	Timer Horn Disable  Where <post> is in the range 0-16.

Set the variable Command equal to the Horn-cmd. Append the one byte <post> number to Command. Append the one byte Disable constant to Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.19 Timer Auto Off Enable Command ( T<post>AE )**

This procedure processes a single Timer Auto Off Enable command “T<post>AE”. Table 6.4-19 depicts the Timer Auto Off Enable Command.

Table 6.4-19 - Timer Auto Off Enable Command

<b>COMMAND</b>	<b>Description</b>
T<post>AE	Timer Auto Off Enable  Where <post> is in the range 0-16.

Set the variable Command equal to the Auto-Off -cmd. Append the one byte <post> number to Command. Append the one byte Enable constant to Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.20 Timer Auto Off Disable Command ( T<post>AD )**

This procedure processes a single Timer Auto Off Disable command “T<post>AD”. Table 6.4-20 depicts the Timer Auto Off Disable Command.

Table 6.4-20 - Timer Auto Off Disable Command

COMMAND	Description
T<post>AD	Timer Auto Off Disable  Where <post> is in the range 0-16.

Set the variable Command equal to the Auto-Off -cmd. Append the one byte <post> number to Command. Append the one byte Disable constant to Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.21 Timer De-bounce Command (TDC) ( TD<milliseconds> )**

This procedure processes single Timer De-bounce command “TD<milliseconds>”. The parameter <milliseconds> is a 4 bytes expressed in milliseconds. Table 6.4-21 depicts the Timer De-bounce offset or current Command.

Table 6.4-21 - Timer De-bounces Command

COMMAND	Description
TD<milliseconds>	Set Timer De-bounce

Set the variable Command equal to the Set-debounce-time-cmd. Convert the parameter <milliseconds> into a 4-byte real and append the results to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.22 Timer Cycles Command ( TC<frequency> )**

This procedure processes single Timer Cycles command “TC< frequency >”. Table 6.4-22 depicts the Timer Cycles Command.

Table 6.4-22 - Timer Cycles Command

COMMAND	Description
TC<frequency>	Timer Cycles

Set the variable Command equal to the Cycles-Frequency-cmd. Convert the parameter <frequency> into a 2-byte integer and append the results to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.23 Timer Seconds Command ( TSE )**

This procedure processes single Timer Seconds command “TSE”. Table 6.4-23 depicts the Timer Seconds Command.

Table 6.4-23 - Timer Seconds Command

COMMAND	Description
TSE	Timer Seconds

Set the variable Command equal to the Display-elapsed-time-in-Seconds-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.24 Timer Reset Command ( TR )**

This procedure processes single Timer Reset command “TR”. Table 6.4-24 depicts the Timer Reset Command.

Table 6.4-24 - Timer Reset Command

<b>COMMAND</b>	<b>Description</b>
TR	Timer Reset

Set the variable Command equal to the Timer-Reset-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.25 Soft Unit Reset Command ( SU )**

The Soft Unit reset command, “SU”, permits a reset of all voltage and current generators with a minimum delay in the automatic testing of relays. Table 6.4-25 depicts the Soft Unit Reset Command.

During automatic testing, output voltage and current values need to be reset without doing a total Unit Reset, as defined in 6.4.23. As a minimum the following items need to be reset with the SU command,

1. Reset Voltage and Current outputs to zero.
2. Reset Phase Angles to the power up default settings.
3. Reset Frequency to the power up default setting.
4. Reset Timer values to zero.
5. Reset the Binary Inputs to the power up default settings.
6. Reset the Binary Outputs to the power up default settings.
7. Reset the Battery Simulator to the power up default setting.
8. DO NOT reset the waveform. Since the user maybe doing a DFR playback, the waveform must not be reset to the power up sine wave.
9. DO NOT reset the main CPU.
10. DO NOT reset any communication channels.

Table 6.4-25 – Soft Unit Reset Command

<b>COMMAND</b>	<b>Description</b>
SU	Soft Unit Reset

**6.4.26 Timer Sync-Test Command ( TSY )**

This procedure processes single Timer Sync-Test command “TSY”. Table 6.4-26 depicts the Timer Sync-Test Command.

Table 6.4-26 - Timer Sync-Test Command

<b>COMMAND</b>	<b>Description</b>
TSY	Timer Sync-Test

The Timer Sync Test is defined in section 13.2.2.5.2.1 Testing Sync-Check, Synchronizing and Auto-Synchronizing Relays, of the Genesis Test Set Firmware Specification. The phase angle referenced in this algorithm below is obtained from a process running on the SBC that periodically constructs a table utilizing

the output messages received from VI generators 1, 2, and 3 **Query-Output-cmd**. An Algorithm that performs a Timer Sync Test is outlined below.

First, Set the Timer to zero. This is equivalent to the RTS command "TR" as described in section 6.4.24 **Timer Reset Command**.

Set Timer Post 1 to Voltage Mode Applied Latch mode. This is equivalent to the RTS command "T1VAL" as described in section 6.4.1 **Timer Volt Applied Latch Command**.

Set Timer Post 1 up as a Timer Start Post. This is equivalent to the RTS command "T1STA" as described in section 6.4.13 **Timer Start Command**.

Select Voltage Generator 1 and 2. This is equivalent to the RTS command "V1V2"; as described in section 6.1.1 **Select Voltage Generator Command (SVGC)**.

Issue a Synchronize Phase Reference command to Selected Voltage Generator 1 and 2. This is equivalent to the RTS command "SP" as described in section 6.14.2 **Synchronization Phase Reference Command**.

Apply 120 volts to both of the selected generators V1 and V2. This is equivalent to the RTS command "A120" as described in section 6.2.2 **AC Amplitude Offset Voltage or Current Command (ACAOVCC)**.

Set the Phase angle of both the selected generators V1 and V2 to zero. This is equivalent to the RTS command "P0" as described in section 6.2.3 **Phase Command (PC)**.

Select Voltage Generator 1. This is equivalent to the RTS command "V1"; as described in section 6.1.1 **Select Voltage Generator Command (SVGC)**.

Set the Frequency of generator V1 to 60. This is equivalent to the RTS command "F60.0" as described in section 6.2.4 **Frequency Command (FC)**.

Select Voltage Generator 2. This is equivalent to the RTS command "V2"; as described in section 6.1.1 **Select Voltage Generator Command (SVGC)**.

Set the Frequency of generator V2 to 60.1. This is equivalent to the RTS command "F60.1" as described in section 6.2.4 **Frequency Command (FC)**.

Select Voltage Generator 1 and 2. This is equivalent to the RTS command "V1V2"; as described in section 6.1.1 **Select Voltage Generator Command (SVGC)**.

First capture and calculate the Pickup-up or closing angle. Obtain the 4-byte real phase angle for voltage generator V2 until the Timer clock starts. The Timer Clock has started when the current time of the clock is not equal to zero. Obtain the current time of the Clock utilizing the RTS command defined in section 6.17.1 **Query Time Command**. If the phase angle is greater than 180 degrees, then set the closing angle equal to 360 minus the phase angle. Otherwise set the closing angle equal to the phase angle.

Next,

Set the variable Command equal to the Timer-Sync-Test -cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.27 Timer Aux-Contact Disable Command ( TAD )**

This procedure processes single Timer Aux-Contact Disable command "TAD". Table 6.4.27 depicts the Timer Aux-Contact Disable Command.

Table 6.4.27 - Timer Aux-Contact Disable Command

<b>COMMAND</b>	<b>Description</b>
TAD	Timer Aux-Contact Disable

Set the variable Command equal to the Aux-Contact-On/Off-cmd. Append the one byte Disable constant to Command (See section 3.1 Constant Definitions). Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.28 Timer Aux-Contact Enable Command ( TAE )**

This procedure processes single Timer Aux-Contact Enable command "TAE". Table 6.4.28 depicts the Timer Aux-Contact Enable Command.

Table 6.4.28 - Timer Aux-Contact Enable Command

COMMAND	Description
TAE	Timer Aux-Contact Enable

Set the variable Command equal to the Aux-Contact-On/Off-cmd. Append the one byte Enable constant to Command (See section 3.1 Constant Definitions). Append Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.4.29 Timer Software Start Command ( TSOSTA )**

This procedure processes single Timer Software Start command "TSOSTA". Table 6.4.29 depicts the Timer Software Start Command.

Table 6.4.29 - Timer Software Start Command

COMMAND	Description
TSOSTA	Timer Software Start

Set the variable Command equal to the Software-Start-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.30 Timer Software Stop Command ( TSOSTO )**

This procedure processes single Timer Software Stop command "TSOSTO". Table 6.4.30 depicts the Timer Software Stop Command.

Table 6.4.30 - Timer Software Stop Command

COMMAND	Description
TSOSTO	Timer Software Stop

Set the variable Command equal to the Software-Stop-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.4.31 Timer Execute Trigger Command ( WAV<loc>; or WAVF<loc>; )**

This procedure processes a single Timer Execute Trigger command. Table 6.4.31 depicts the Timer Execute Trigger Command.

Table 6.4.31 - Timer Execute Trigger Command

COMMAND	Description
WAV<loc>;  or  WAVF<loc>;	Timer Execute Trigger command  When the VI-generator commands defined in section 6.2.8 <b>Waveform Location Command (WL)</b> or 6.2.12 <b>Waveform Location On the Fly Command (WLF)</b> are executed, when the Single Board Computer encounters a semicolon ";" in the RTS command string. The Timer Execute Trigger command is executed to synchronize the VI-generators via the Timer. (Note: The Execute trigger command is only used in conjunction with the VI-generator command.)



When the VI-generator commands defined in section 6.2.8 Waveform Location Command (WL) or 6.2.12 Waveform Location On the Fly Command (WLF) are executed, when the Single Board Computer encounters a semicolon ";" in the RTS command string. The Timer Execute Trigger command is executed to synchronize the VI-generators via the Timer. (Note: The Execute trigger command is only used in conjunction with the VI-generator command.)

Set the variable Command equal to the Execute-Trigger-cmd . Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

## Binary Input Commands

### 6.5 Binary Input Commands

These commands are used to allow the user to set up the Timer Control Module timing.

#### 6.5.27 Binary Input Contact Command ( BI<post>C<logical operation> )

This procedure processes a single Binary Input Contact command "BI<post>C<logical operation>". Table 6.5-1 depicts the Binary Input Contact Command.

Table 6.5-1 - Binary Input Contact Command

COMMAND	Description
BI<post>C<logical operation>	<p>Binary Input Contact</p> <p>&lt;logical operation &gt; is optional.</p> <p>Where &lt;logical operation&gt; is:</p> <p>0 = + = logical OR</p> <p>1 = * = logical AND</p> <p>2 = No Operator</p>

Set the variable Command equal to the Binary-Input-Contact-cmd. Append a one byte <post> value to the Command. If <logical operation> equals "+" then append one byte value "0" to the Command. If <logical operation> equals "\*" then append one byte value "1" to the Command. If no <logical operation> is specified, then append one byte value "2" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Binary-Input-Contact -cmd opcode when the Binary Input Contact Command completes.).

#### 6.5.28 Binary Input Voltage Command ( BI<post>V<logical operation> )

This procedure processes a single Binary Input Voltage command "BI<post>V<logical operation>". Table 6.5-2 depicts the Binary Input Voltage Command.

Table 6.5-2 - Binary Input Voltage Command

COMMAND	Description
BI<post>V<logical operation>	<p>Binary Input Voltage</p> <p>&lt;logical operation &gt; is optional.</p> <p>Where &lt;logical operation&gt; is:</p> <p>0 = + = logical OR</p> <p>1 = * = logical AND</p> <p>2 = No Operator</p>

Set the variable Command equal to the Binary-Input-Voltage-cmd. Append a one byte <post> value to the Command. If <logical operation> equals "+" then append one byte value "0" to the Command. If <logical operation> equals "\*" then append one byte value "1" to the Command. If no <logical operation> is specified, then append one byte value "2" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Binary-Input-Voltage -cmd opcode when the Binary Input Voltage Command completes.).

**6.5.29 Binary Input Don't Care Command ( BI<post>X<logical operation> )**

This procedure processes a single Binary Input Don't Care command "BI<post>X<logical operation>". Table 6.5-3 depicts the Binary Input Don't Care Command.

Table 6.5-3 - Binary Input Don't Care Command

COMMAND	Description
BI<post>X<logical operation>	<p>Binary Input Don't Care</p> <p>&lt;logical operation &gt; is optional.</p> <p>Where &lt;logical operation&gt; is:</p> <p>0 = + = Logical OR</p> <p>1 = * = Logical AND</p> <p>2 = No Operator</p>

Set the variable Command equal to the Binary-Input-don't-care-cmd. Append a one byte <post> value to the Command. If <logical operation> equals "+" then append one byte value "0" to the Command. If <logical operation> equals "\*" then append one byte value "1" to the Command. If no <logical operation> is specified, then append one byte value "2" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Binary-Input-don't-care -cmd opcode when the Binary Input Don't Care Command completes.).

**6.5.30 Binary Input Closed Command ( BI<post>1<logical operation> )**

This procedure processes a single Binary Input Closed command "BI<post>1<logical operation>". Table 6.5-4 depicts the Binary Input Closed Command.

Table 6.5-4 - Binary Input Closed Command

COMMAND	Description
BI<post>1<logical operation>	<p>Binary Input Closed</p> <p>&lt;logical operation &gt; is optional.</p> <p>Where &lt;logical operation&gt; is:</p> <p>0 = + = logical OR</p> <p>1 = * = logical AND</p> <p>2 = No Operator</p>

Set the variable Command equal to the Binary-Input-Closed-cmd. Append a one byte <post> value to the Command. If <logical operation> equals "+" then append one byte value "0" to the Command. If <logical operation> equals "\*" then append one byte value "1" to the Command. If no <logical operation> is specified, then append one byte value "2" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Binary-Input-Closed -cmd opcode when the Binary Input Closed Command completes.).

**6.5.31 Binary Input Open Command ( BI<post>0<logical operation> )**

This procedure processes a single Binary Input Open command "BI<post>0<logical operation>". Table 6.5-5 depicts the Binary Input Open Command.

Table 6.5-5 - Binary Input Open Command

COMMAND	Description
BI<post>0<logical operation>	<p>Binary Input Open</p> <p>&lt;logical operation &gt; is optional.</p> <p>Where &lt;logical operation&gt; is:</p> <p>0 = + = logical OR</p> <p>1 = * = logical AND</p> <p>2 = No Operator</p>

Set the variable Command equal to the Binary-Input-Open-cmd. Append a one byte <post> value to the Command. If <logical operation> equals "+" then append one byte value "0" to the Command. If <logical operation> equals "\*" then append one byte value "1" to the Command. If no <logical operation> is specified, then append one byte value "2" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Binary-Input-Open -cmd opcode when the Binary Input Open Command completes.).

## Binary Output Commands

### 6.6 Binary Output Commands

These commands are used to allow the user to set up the Timer Control Module timing.

#### 6.6.27 Binary Output Closed Command ( BO<post>1<logical operation> )

This procedure processes a single Binary Output closed command “BO<post>1<logical operation>”. Table 6.6-1 depicts the Binary Output closed Command.

Table 6.6-1 - Binary Output closed Command

COMMAND	Description
BO<post>1<logical operation>	Binary Output closed Where <logical operation> is * = Logical AND

Set the variable Command equal to the Binary-Output-Closed-cmd. Append a one byte <post> value to the Command. Append a one byte value “1” to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

#### 6.6.28 Binary Output Open Command ( BO<post>0<logical operation> )

This procedure processes a single Binary Output Open command “BO<post>0<logical operation>”. Table 6.6-2 depicts the Binary Output Open Command.

Table 6.6-2 - Binary Output Open Command

COMMAND	Description
BO<post>0<logical operation>	Binary Output Open Where <logical operation> is: * = Logical AND

Set the variable Command equal to the Binary-Output-Open-cmd. Append a one byte <post> value to the Command. Append a one byte value “1” to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

#### 6.6.29 Binary Output Don't Care Command ( BO<post>X<logical operation> )

This procedure processes a single Binary Output Don't Care command “BO<post>X<logical operation>”. Table 6.6-3 depicts the Binary Output Don't Care Command.

Table 6.6-3 - Binary Output Don't Care Command

<b>COMMAND</b>	<b>Description</b>
BO<post>X<logical operation>	Binary Output Don't Care  Where <logical operation> is:  * = Logical AND

Set the variable Command equal to the Binary-Output-don't-care-cmd. Append a one byte <post> value to the Command. Append a one byte value "1" to the Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

## Binary Trigger commands

### 6.9 Binary Trigger Commands

#### 6.9.1 Binary Trigger Software Start Command ( BTSO )

This procedure processes a single Binary Trigger Software Start command "BTSO". Table 6.11-1 depicts the Binary Trigger Software Start Command.

Table 6.11-1 - Binary Trigger Software Start Command

COMMAND	Description
BTSO	Binary Trigger Software Start

Set the variable Command equal to the Binary-Trigger-Software-Start-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.9.2 Binary Trigger Voltage Command (\*\* NOT IMPLEMENTED\*\*) ( BTV )

This procedure processes a single Binary Trigger Voltage command "BTV". Table 6.11-2 depicts the Binary Trigger Voltage Command.

Table 6.11-2 - Binary Trigger Voltage Command

COMMAND	Description
BTV	Binary Trigger Voltage

Set the variable Command equal to the Binary-Trigger-Voltage-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.9.3 Binary Trigger Contact Command (\*\* NOT IMPLEMENTED\*\*) ( BTC )

This procedure processes a single Binary Trigger Contact command "BTC". Table 6.11-3 depicts the Binary Trigger Contact Command.

Table 6.11-3 - Binary Trigger Contact Command

COMMAND	Description
BTC	Binary Trigger Contact

Set the variable Command equal to the Binary-Trigger-Contact-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.9.4 Binary Trigger Input Command ( BTI<binary input logic> )

This procedure processes a single Binary Trigger Input command "BTI". Table 6.11-4 depicts the Binary Trigger Input Command

Table 6.11-4 - Binary Trigger Input Command

<b>COMMAND</b>	<b>Description</b>
BTI<binary input logic>	Binary Trigger Input  Where <binary input logic> is 0 or 1

Set the variable Command equal to the Binary-Trigger-Input-cmd. Append 1 byte <binary input



## Load Error Action Commands

### 6.11 Load Error Action Commands

#### 6.11.1 Load Horn Enable Command ( LOAHE )

This procedure processes a single Load Horn Enable command "LOAHE". Table 6.13-1 depicts the Load Horn Enable Command.

Table 6.13-1 - Load Horn Enable Command

COMMAND	Description
LOAHE	Load Horn Enable

Set the variable Command equal to the Load-Horn-Enable-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.11.2 Load Horn Disable Command ( LOAHD )

This procedure processes a single Load Horn Disable command "LOAHD". Table 6.13-2 depicts the Load Horn Disable Command.

Table 6.13-2 - Load Horn Disable Command

COMMAND	Description
LOAHD	Load Horn Disable

#### 6.11.3 Load Shutdown Enable Command ( LOASE )

This procedure processes a single Load Shutdown Enable command "LOASE". Table 6.13-3 depicts the Load Shutdown Enable Command.

Table 6.13-3 - Load Shutdown Enable Command

COMMAND	Description
LOASE	Load Shutdown Enable

Set the variable Command equal to the Load-Shutdown-Enable-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.11.4 Load Shutdown Disable Command ( LOASD )

This procedure processes a single Load Shutdown Disable command "LOASD". Table 6.13-4 depicts the Load Shutdown Disable Command.

Table 6.13-4 - Load Shutdown Disable Command

COMMAND	Description
LOASD	Load Shutdown Disable

Set the variable Command equal to the Load-Shutdown-Disable-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

## Synchronization Commands

### 6.12 Synchronization Commands

#### 6.12.1 Synchronize Immediate Mode Command ( S )

This procedure processes a single Synchronization Immediate Mode command "S". Table 6.14-1 depicts the Synchronization Immediate Mode Command.

Table 6.14-1 - Synchronization Immediate Mode Command

COMMAND	Description
S	Synchronization Immediate Mode

Set the variable Command equal to the Synchronize-Immediate-Mode-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.12.2 Synchronization Phase Reference Command ( SP )

This procedure processes a single Synchronization Phase Reference command "SP". Table 6.14-2 depicts the Synchronization Phase Reference Command.

Table 6.14-2 - Synchronization Phase Reference Command

COMMAND	Description
SP	Synchronization Phase Reference

#### 6.12.3 Synchronization Self Zero Command ( SS )

This procedure processes a single Synchronization Self Zero command "SS". Table 6.14-3 depicts the Synchronization Self Zero Command.

Table 6.14-3 - Synchronization Self Zero Command

COMMAND	Description
SS	Synchronization Self Zero

Set the variable Command equal to the Synchronization-Self-Zero-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

#### 6.12.4 Synchronization Phase Reference Self Zero Command ( SPS )

This procedure processes a single Synchronization Phase Reference Self Zero command "SPS". Table 6.14-4 depicts the Synchronization Phase Reference Self Zero Command.

Table 6.14-4 - Synchronization Phase Reference Self Zero Command

COMMAND	Description
SPS	Synchronization Phase Reference Self Zero

Set the variable Command equal to the Synchronization-Phase-Reference-Self-Zero-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command

**6.12.5 Synchronization External Command ( SE )**

This procedure processes a single Synchronization External command "SE". Table 6.14-5 depicts the Synchronization External Command.

Table 6.14-5 - Synchronization External Command

<b>COMMAND</b>	<b>Description</b>
SE	Synchronization External

Set the variable Command equal to the Synchronization-External-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

## Wait Commands

### 6.13 Wait Commands

#### 6.13.1 Wait Time Command ( WAI<time> )

This procedure processes a single Wait Time command "WAI<time>". Table 6.15-1 depicts the Wait Time Command.

Table 6.15-1 - Wait Time Command

COMMAND	Description
WAI<time>	Wait Time  Where <time> is in milliseconds

Set the variable Command equal to the Wait-time-cmd. Convert the parameter <time> into a 4-byte real and append the results to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Wait-time-cmd opcode when the Wait Time Command completes.).

#### 6.13.2 Wait Post Command ( WAIP<post><count> )

This procedure processes a single Wait Post command "WAIP<post><count>". Table 6.15-2 depicts the Wait Post Command.

Table 6.15-2 - Wait Post Command

COMMAND	Description
WAIP<post><count>	Wait Post  Where <count> is optional in defaults to 1

Set the variable Command equal to the Wait-post-cmd. Append the 1-byte post value to Command. If 2 byte <count> is missing append 2 byte value "1" to Command otherwise append 2 byte <count> value to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Wait-post-cmd opcode when the Wait Post Command completes.).

#### 6.13.3 Wait Cycles Command ( WAIC<count> )

This procedure processes a single Wait Cycles command "WAIC<count>". Table 6.15-3 depicts the Wait Cycles Command.

Table 6.15-3 - Wait Cycles Command

COMMAND	Description
WAIC<count>	Wait Cycles  Where <count> is optional in defaults to 1

Set the variable Command equal to the Wait-Cycles-cmd. Append the 1-byte post value to Command. If <count> is missing append 2 byte count value "1" to Command, otherwise append 2 byte <count> to Command. Call the procedure 7.5 Execute Timer Command (ETC) with the Command. (Note: The Single Board Computer will store the command name along with the parameters for this wait command to be used by the Query Status command. status of the current wait command being executed block the execution of all RTS commands except the Query Status command until the wait is complete. The Timer will return the Wait-Cycles-cmd opcode when the Wait Cycles Command completes.).

**6.13.4 Wait Mode Immediate Command ( WAIMI )**

This procedure processes a single Wait Mode Immediate command "WAIMI". Table 6.15-4 depicts the Wait Mode Immediate Command.

Table 6.15-4 - Wait Mode Immediate Command

<b>COMMAND</b>	<b>Description</b>
WAIMI	Wait Mode Immediate

Set the variable Command equal to the Wait-Mode-Immediate-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.13.5 Wait Mode Sync Command ( WAIMS )**

This procedure processes a single Wait Mode Sync command "WAIMS". Table 6.15-5 depicts the Wait Mode Sync Command.

Table 6.15-5 - Wait Mode Sync Command

<b>COMMAND</b>	<b>Description</b>
WAIMS	Wait Mode Sync

Set the variable Command equal to the Wait-Mode-Sync-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.13.6 Wait Input Clock Source Command ( WAI )**

This procedure processes a single Wait Input Clock Source command "WAI". Table 6.15-6 depicts the Wait Input Clock Source Command.

Table 6.15-6 – Wait Input Clock Source Command

<b>COMMAND</b>	<b>Description</b>
WAI	Wait Input Clock Source

Set the variable Command equal to the Wait-Input-Clock-Source –cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

**6.13.7 Wait Input Clock Source External Command ( WAIE )**

This procedure processes a single Wait Input Clock Source External command "WAIE". Table 6.15-7 depicts the Wait Input Clock Source External Command.

Table 6.15-7 - Wait Input Clock Source External Command

<b>COMMAND</b>	<b>Description</b>
WAIE	Wait Input Clock Source External

Set the variable Command equal to the Wait-Input-Clock-Source-External-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

#### **6.13.8 Wait Input Clock Source Internal Command ( WAIII )**

This procedure processes a single Wait Input Clock Source Internal command "WAIII". Table 6.15-8 depicts the Wait Input Clock Source Internal Command.

Table 6.15-8 - Wait Input Clock Source Internal Command

<b>COMMAND</b>	<b>Description</b>
WAIII	Wait Input Clock Source Internal

Set the variable Command equal to the Wait-Input-Clock-Source-Internal-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

## Restart Commands

### 6.14 Restart Commands

Commands that reset or restart the MPRT test system.

#### 6.14.1 Unit Reset Command ( U )

This procedure processes a single Unit Reset command "U". Table 6.16-1 depicts the Unit Reset Command.

Table 6.16-1 - Unit Reset Command

<b>COMMAND</b>	<b>Description</b>
U	Unit Reset

Call the procedure 5.1 Generator Select Commands Initialization (IGSCI) to clear, unselect, all voltage and current generators that were previously selected.

Call the procedure 5.2 Variable Default Initialization (VDI) to set all variables to there default values.

Write the value 0 to fpga address Unit reset fpga address. Delay for 300 milliseconds. Reset the single board computers uarts.

#### 6.14.2 Local Mode Command ( LOC )

This procedure processes a single Local Mode command "LOC". Table 6.16-2 depicts the Local Mode Command.

Table 6.16-2 - Local Mode Command

<b>COMMAND</b>	<b>Description</b>
LOC	Local Mode

The Single board Computer is placed in a mode where it now monitors the Hand Held Controller, RS232, GPIB and Ethernet interface for RTS command input. Set the variable Command equal to the NULL.

#### 6.14.3 Restore Phase Command ( RE )

This procedure processes a single Restore Phase command "RE". Table 6.16-3 depicts the Restore Phase Command.

Table 6.16-3 - Restore Phase Command

<b>COMMAND</b>	<b>Description</b>
RE	Restore Phase

Set the variable Command equal to the Restore-Phase-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

## Query Commands

### 6.15 Query Commands

#### 6.15.1 Query Time Command ( QT )

This procedure processes a single Query Time command "QT<time>". Table 6.17-1 depicts the Query Time Command.

Table 6.17-1 - Query Time Command

COMMAND	Description
QT	<p>Query Time</p> <p>Timer Responds with:</p> <p>4      byte real &lt;time&gt; in seconds</p> <p>The Unit responds with the following :</p> <p>TIME&lt;time in seconds&gt;;</p>

Set the variables Command and Query-opcode equal to the Query-Time-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 4. Call procedure 7.11 Read Timer Port Command and Data (RTPCD), return the results in variable Query-results. Return to the port in control the string

#### 6.15.2 Query Configuration Command ( QC )

This procedure processes a single Query Configuration command "QC<configuration>". Table 6.17-2 depicts the Query Configuration Command.

Table 6.17-2 - Query Configuration Command



COMMAND	Description
QC	<p>Query Configuration The Unit responds with the following :</p> <pre>&lt;G#&gt;&lt;CG&gt;&lt;SNxxxx&gt;&lt;IO&gt;&lt;SNxxxx&gt;,&lt;VER&gt;&lt;MM,DD,YY&gt;&lt;nn.nn&gt;.. &lt;IO&gt;*&lt;VER&gt;&lt;nn.nn&gt;&lt;hh hh&gt;&lt;SNxxxx&gt;,&lt;G&gt;&lt;position&gt;*&lt;VER&gt; &lt;nn.nn&gt;&lt;hh hh&gt;&lt; SNxxxx&gt;,&lt;memory map&gt; ... &lt;VI&gt; /;</pre> <p>Where:</p> <p>&lt;G#&gt; - The number of V/I generators installed</p> <p>&lt;CG&gt;- indicates that the optional high current interface is installed.</p> <hr/> <p>&lt; SNxxxx&gt; - is the serial number associated with the preceding parameter</p> <p>&lt;IO&gt;- indicates that the optional external binary input/output unit is installed.</p> <p>&lt;VER&gt; -Indicates the version number of the SBC Controller firmware in the Genesis unit, TCM firmware, and the V/I generator firmware.</p> <p>&lt;MM,DD,YY&gt; indicates the month, day, and year of the system software.</p> <p>&lt;nn.nn&gt; - Firmware version of the V/I Generators</p> <p>&lt;hh hh&gt; - Hardware version of the V/I Generators</p> <p>&lt;VI&gt; Indicates that the V/I Generator description will follow in the subsequent string for the number of V/I Generator's in the system.</p> <p>&lt;position&gt; - Indicates the location that the V/I Generator resides in the Genesis Unit.</p> <p>&lt;memory map&gt; - Is used to pass the number of memory locations that can be used in conjunction with the waveform location commands.</p>

Call routine 7.14 **Determine the Number of VI Generators present.** This routine will create the <G#> portion of the Configuration-Query-Results.

Note: The optional external high current interface.<CG><SNxxxx> has not been designed. The query configuration specifications for the optional external high current interface will be supplied when a design has been finalized.

Note: The optional external binary input/output unit <IO><SNxxxx> has not been designed. The query configuration specifications for the optional external binary input/output unit will be supplied when a design has been finalized.

Append the string “, VER “ to the Configuration-Query-Results. Append the month, day, and year (<MM,DD,YY>) of the SBC software to the Configuration-Query-Results. Append a comma (,) followed by the version number (<nn.nn>) of the SBC software to the Configuration-Query-Results. Append two commas (,,) to the Configuration-Query-Results.

Call routine 7.15 **Get Timer Query Information (GTQI)**. This routine will create the <IO>\*<VER><nn.nn><hh.hh><SNxxxx> portion of the Configuration-Query-Results.

Call routine 7.16 **Get VI Generators Query Information (GVIGQI)**. This routine will create the ,/<G><position>\*<VER><nn.nn><hh.hh>< SNxxxx>,<memory map> ... <VI> / portion of the Configuration-Query-Results.

**6.15.3 Query Gates Command ( QG )**

This procedure processes a single Query Gate command “QG<gate>”. Table 6.17-3 depicts the Query Gate Command.

Table 6.17-3 - Query Gate Command

COMMAND	Description
QG	<p>Query Gate</p> <p>Timer Responds with:</p> <p>GATE&lt; post active/inactive mask &gt;;</p> <p>Where: &lt;post active/inactive mask &gt; 4 character mask flags indicating which of the input post (16-1)</p> <p>is active or inactive.</p> <p>The Unit responds with the following :</p> <p>GATE0001; indicate that binary input “1” is closed</p> <p>GATE 0021; indicate that binary input “1” and “6” are closed</p>

Set the variables Command and Query-opcode equal to the Query-Gate-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 Read Timer Port Command and Data (RTPCD), return the results in variable Query-results.

**6.15.4 Query Status Command ( QS )**

This procedure processes a single Query Status command “QS”. Table 6.17-4 depicts the Query Status Command.

Table 6.17-4 - Query Status Command

COMMAND	Description
QS	<p>Query Status</p> <p>Timer Responds with:</p> <p>1 byte &lt;status type&gt; + 2 bytes &lt;status data&gt;</p> <p>Where &lt;status type&gt; equals:</p> <p>    Status type 0 - No Wait</p> <p>    Status type 1 - Waiting for Time</p> <p>    Status type 2 - Waiting for Post</p> <p>    Status type 3 - Waiting for cycles</p> <p>Where &lt;status data&gt;</p> <p>    Status Data - 2 byte value 0 - No Wait Status</p> <p>    Status Data - 2 byte time &lt;milliseconds&gt; - Waiting for Time</p> <p>    Status Data - 1 byte "post" + 1 byte "count" - Waiting for Post</p> <p>    Status Data - 2 byte cycle - Waiting for Cycles</p> <p>The Unit responds with one of the following strings depending on the value of the Status type field :</p> <p>    STATUS NO WAIT;</p> <p>    STATUS WAITING FOR TIME &lt;milliseconds&gt;;</p> <p>    STATUS WAITING FOR POST &lt;post&gt; &lt;count&gt;;</p> <p>    STATUS WAITING FOR CYCLES &lt;cycle&gt;;</p>

Set the variables Command and Query-opcode equal to the Query-Status-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 Read Timer Port Command and Data (RTPCD), return the results in variable Query-results.

#### 6.15.5 Query Address Command ( QA )

This procedure processes a single Query Address command "QA". Table 6.17-5 depicts the Query Address Command.

Table 6.17-5 - Query Address Command

COMMAND	Description
QA	<p>Query Address</p> <p>The Unit responds with the following :</p> <p>&lt;GPIB ADDRESS&gt;;</p>

Set the variable Query-results equal to the GPIB Address utilized by the SBC.

#### 6.15.6 Query Monitor Command ( QM )

This procedure processes a single Query Monitor command "QM". Table 6.17-6 depicts the Query Monitor Command.

Table 6.17-6 - Query Monitor Command

COMMAND	Description
QM	<p>Query Monitor</p> <p>Timer Responds with:</p> <p>2 byte &lt;monitor status&gt; flags bit <u>encoding same</u> as that for the Query Gate.</p> <p>The Unit responds with the following :</p> <p>MONITOR&lt; post active/inactive mask &gt;;</p>

Set the variables Command and Query-opcode equal to the Query-Monitor-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 Read Timer Port Command and Data (RTPCD), return the results in variable Query-results. (Note: Query Monitor returns the < post active/inactive mask > as an encoded binary number.

## Query Device Commands

### Device Query Commands

#### 6.18.1 Query Binary Input Command ( QBI )

This procedure processes a single Query Binary Input Current DC Read command "QBI". Table 6.18-1 depicts the Query Binary Input Current DC Read Command.

Table 6.18-1 – Query Binary Input Current DC Read Command

COMMAND	Description
QBI	<p>Query Binary Input</p> <p>Timer Responds with:</p> <p style="padding-left: 40px;">2 byte &lt;active/inactive&gt; flag</p> <p>The Unit responds with the following :</p> <p>I&lt;16 bit active/inactive mask&gt;;</p> <p>Where &lt;active&gt; equals "1" and &lt;inactive&gt; equals "0"</p>

Set the variables Command and Query-opcode equal to the Query-Binary-Input-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. Convert the Query-results to binary, and prefix the Query-results with the letter "I".

#### 6.18.2 Query Binary Output Command ( QBO )

This procedure processes a single Query Binary Output Current DC Read command "QBI". Table 6.18-2 depicts the Query Binary Output Command.

Table 6.18-2 - Query Binary Output Command

COMMAND	Description
QBO	<p>Query Binary Output</p> <p>Timer Responds with:</p> <p style="padding-left: 40px;">2 byte &lt;open/closed&gt; flag</p> <p>The Unit responds with the following :</p> <p>O&lt;16 bit open/closed mask&gt;;</p> <p>Where &lt;open&gt; equals "0" and &lt;closed&gt; equals "1"</p>

Set the variables Command and Query-opcode equal to the Query-Binary-Output-cmd. Call the procedure 7.5 Execute Timer Command (ETC) with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. Convert the Query-results to binary, and prefix the Query-results with the letter "O".

**6.18.3 Query Event Command ( QEV )**

This procedure processes a single Query Event command "QEV". Table 6.18-3 depicts the Query Event Command.

Table 6.18-3 - Query Event Command

<b>COMMAND</b>	<b>Description</b>
QEV	Query Event  Timer Responds with:  2 byte number of sequencer commands executed since trigger  The Unit responds with the following :  EV< number of sequencer commands executed since trigger >;

Set the variables Command and Query-opcode equal to the Query-Event-cmd. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results.

**6.18.4 Query Event Length Command ( QEL )**

This procedure processes a single Query Event Length command "QEL". Table 6.18-4 depicts the Query Event Length Command.

Table 6.18-4 - Query Event Length Command

<b>COMMAND</b>	<b>Description</b>
QEL	Query Event Length  2 byte total command count loaded in sequencer  The Unit responds with the following :  EL< total command count loaded in sequencer >;

Set the variables Command and Query-opcode equal to the Query-Event-Length-cmd. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. Prefix Query-results with the letter "EL".

**6.18.5 Query Collected Record Command ( QCR )**

This procedure processes a single Query Collected Record command "QCR". Table 6.18-5 depicts the Query Collected Record Command.

Table 6.18-5 - Query Collected Record Command

COMMAND	Description
QCR	Query Collected Record  The Unit responds with the following :  CR<number of captured records>;

Set the variables Command and Query-opcode equal to the Query-Collected-Record-cmd. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results

**6.18.6 Query Record Number Command ( QR<num> )**

This procedure processes a single Query Record Number command "QR<num>". Table 6.18-6 depicts the Query Record Number Command.

Table 6.18-6 - Query Record Number Command

COMMAND	Description
QR<num>	Query Record Number  <num> is 2 byte record number  The Unit responds with the following :  R<record-number>; I<binary input mask>;O<binary output mask>; T<time in milliseconds>;

Set the variables Command and Query-opcode equal to the Query-Record-Number-cmd. Append the 2-byte <num> to Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 8. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results.

## Query Time Tag Commands

### 6.19 Query Time Tag Commands

#### 6.19.1 Query Time Tag Binary Input Rising Edge Command ( QTABI<post>R )

This procedure processes a single Query Time Tag Binary Input Rising Edge command “QTABI<post>R”. Table 6.19-1 depicts the Query Time Tag Binary Input Rising Edge Command.

Table 6.19-1 - Query Time Tag Binary Input Rising Edge Command

COMMAND	Description
QTABI<post>R	<p>Query Time Tag Binary Input Rising Edge</p> <p>Where &lt;post&gt; is (01-16)</p> <p>The Unit responds with the following :</p> <p>“BIRE&lt;number of captured-records&gt;, T&lt;time in milliseconds&gt;..... T&lt;time in milliseconds&gt;;”.</p>

Set the variables Command and Query-opcode equal to the Query-Time-Tag-Binary-Input-cmd. Append the 1-byte <post> number to the Command. Append the type value “1” for rising edge to the variable Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. (Note: this will return the number of record return by the command).

From 1 to the number of records return, set the variables Command and Query-opcode equal to the Query-Time-Tag-Record-Number-cmd. Append the 2-byte record number to the Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command. Read the 4-byte time tag records returned by the timer and send the data to the port in control (i.e. RS232 , GPIB, etc).

#### 6.19.2 Query Time Tag Binary Input Falling Edge ( QTABI<post>F )

This procedure processes a single Query Time Tag Binary Input Falling Edge command “QTABI<post>F”. Table 6.19-2 depicts the Query Time Tag Binary Input Falling Edge Command.

Table 6.19-2 - Query Time Tag Binary Input Falling Edge Command

COMMAND	Description
QTABI<post>F	<p>Query Time Tag Binary Input Falling Edge</p> <p>Where &lt;post&gt; is (01-16)</p> <p>The Unit responds with the following :</p> <p>BIFE&lt;number of captured-records&gt;, T&lt;time in milliseconds&gt;,...., T&lt;time in milliseconds&gt;;</p>

Set the variables Command and Query-opcode equal to the Query-Time-Tag-Binary-Input-cmd. Append the 1-byte <post> number to the Command. Append the type value “2” for falling edge to the variable Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. (Note: this will return the number of record return by the command).



From 1 to the number of records return, set the variables Command and Query-opcode equal to the Query-Time-Tag-Record-Number-cmd. Append the 2-byte record number to the Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command. Read the 4 -byte time tag records returned by the timer and send the data to the port in control (i.e. RS232 , GPIB, etc) .

**6.19.3 Query Time Tag Binary Input Both Edges ( QTABI<post>B )**

This procedure processes a single Query Time Tag Binary Input Both Edges command "QTABI<post>B". Table 6.19-3 depicts the Query Time Tag Binary Input Both Edges Command.

Table 6.19-3 - Query Time Tag Binary Input Both Edges Command

COMMAND	Description
QTABI<post>B	Query Time Tag Binary Input Both Edges  Where <post> is (01-16)  The Unit responds with the following :  BIBE<number of captured-records>, T<time in milliseconds>,..., T<time in milliseconds>;

Set the variables Command and Query-opcode equal to the Query-Time-Tag-Binary-Input-cmd. Append the 1-byte <post> number to the Command. Append the type value "0" for both rising and falling edge to the variable Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 2. Call procedure 7.11 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results. (Note: this will return the number of record return by the command) .

From 1 to the number of records return, set the variables Command and Query-opcode equal to the Query-Time-Tag-Record-Number-cmd. Append the 2-byte record number to the Command. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command. Read the 4 -byte time tag records returned by the timer and send the data to the port in control (i.e. RS232 , GPIB, etc).

## Query Measurement Value Commands

### 6.20 Query Measurement Value Commands

#### 6.20.1 Query Measure Input Volts Command ( QMEASIV )

This procedure processes a single Query Measure Input Volts command “QMEASIV”. Table 6.20-1 depicts the Query Measure Input Volts Command.

Table 6.20-1 - Query Measure Input Volts Command

COMMAND	Description
QMEASIV	Query Measure Input Volts  The Unit responds with the following :  MEASIV<volts>;

Write the value zero “0” to the DMA register VI-Monitor-Control-and-Data-Register. Read from DMA register VI-Monitor-Control-and-Data-Register two 16 bit registers. Set the variable Query-results equal to the value of the 16 bit registers.

#### 6.20.2 Query Measure Input Current Command ( QMEASIC )

This procedure processes a single Query Measure Input Current command “QMEASIC”. Table 6.20-2 depicts the Query Measure Input Current Command.

Table 6.20-2 - Query Measure Input Current Command

COMMAND	Description
QMEASIC	Query Measure Input Current  The Unit responds with the following :  MEASIC<current>;

Write the value one “1” to the DMA register VI-Monitor-Control-and-Data-Register. Read from DMA register VI-Monitor-Control-and-Data-Register two 16 bit registers. Set the variable Query-results equal to the value of the 16 bit registers.

#### 6.20.3 Query Measure Generator Volts AC Command ( QMEASG<num>VAC )

This procedure processes a single Query Measure Generator Volts AC command “QMEASG<num>VAC”. Table 6.20-3 depicts the Query Measure Generator Volts AC Command.

Table 6.20-3 - Query Measure Generator Volts AC Command

COMMAND	Description
QMEASG<num>VAC	<p>Query Measure Generator Volts AC</p> <p>&lt;num&gt; will be from one or all the V/I generators:</p> <p>&lt;num&gt; = 0 – All generators</p> <p>&lt;num&gt; = 1 – V/I generator in the Phase ‘A’ slot</p> <p>&lt;num&gt; = 2 – V/I generator in the Phase ‘B’ slot</p> <p>&lt;num&gt; = 3 – V/I generator in the Phase ‘C’ slot</p> <p>&lt;num&gt; = 4 – V/I generator in the Phase ‘D’ slot</p> <p>The Unit responds with the following :</p> <p>MEASIG&lt;num&gt;,VAC,&lt;generator 0&gt;,&lt;volts AC&gt;,&lt;generator 1&gt;,&lt;volts AC&gt;;</p> <p>(Note: If num equals zero, a result is returned for each of the VI generators installed. When more than one V/I generator is installed, the results are separated by a semicolon“;”).</p>

The information for this command is obtained from a process running on the SBC that periodically constructs a table utilizing the output messages received from VI generators 1, 2,3 and 4 Query-Output-cmd.

**6.20.4 Query Measure Generator Volts DC Command ( QMEASG<num>VDC )**

This procedure processes a single Query Measure Generator Volts DC command “QMEASG<num>VDC”. Table 6.20-4 depicts the Query Measure Generator Volts DC Command.

Table 6.20-4 - Query Measure Generator Volts DC Command

COMMAND	Description
QMEASG<num>VDC	<p>Query Measure Generator Volts DC</p> <p>&lt;num&gt; will be from one or all the V/I generators:</p> <p>&lt;num&gt; = 0 – All generators</p> <p>&lt;num&gt; = 1 – V/I generator in the Phase ‘A’ slot</p> <p>&lt;num&gt; = 2 – V/I generator in the Phase ‘B’ slot</p> <p>&lt;num&gt; = 3 – V/I generator in the Phase ‘C’ slot</p> <p>&lt;num&gt; = 4 – V/I generator in the Phase ‘D’ slot</p> <p>The Unit responds with the following :</p> <p>MEASIG&lt;num&gt;,VDC,&lt;generator 0&gt;,&lt;volts DC&gt;,&lt;generator 1&gt;,&lt;volts DC&gt;;</p> <p>(Note: If num equals zero, a result is returned for each of the VI generators installed. When more than one V/I generator is installed, the results are separated by a semicolon“:”.)</p>

The information for this command is obtained from a process running on the SBC that periodically constructs a table utilizing the output messages received from VI generators 1, 2,3 and 4 Query-Output-cmd.

**6.20.5 Query Measure Generator Current AC Command ( QMEASG<num>CAC )**

This procedure processes a single Query Measure Generator Current AC command “QMEASG<num>CAC”. Table 6.20-5 depicts the Query Measure Voltage Generator Input Current AC Command.

Table 6.20-5 - Query Measure Generator Current AC Command

COMMAND	Description
QMEASG<num>CAC	<p>Query Measure Generator Current AC</p> <p>&lt;num&gt; will be from one or all the V/I generators:</p> <p>&lt;num&gt; = 0 – All generators</p> <p>&lt;num&gt; = 1 – V/I generator in the Phase ‘A’ slot</p> <p>&lt;num&gt; = 2 – V/I generator in the Phase ‘B’ slot</p> <p>&lt;num&gt; = 3 – V/I generator in the Phase ‘C’ slot</p> <p>&lt;num&gt; = 4 – V/I generator in the Phase ‘D’ slot</p> <p>The Unit responds with the following :</p> <p>MEASIG&lt;num&gt;,CAC,&lt;generator 0&gt;,&lt;current AC&gt;,&lt;generator 1&gt;,&lt;current AC&gt;;</p> <p>(Note: If num equals zero, a result is returned for each of the VI generators installed. When more than one V/I generator is installed, the results are separated by a semicolon“;”.)</p>

The information for this command is obtained from a process running on the SBC that periodically constructs a table utilizing the output messages received from VI generators 1, 2,3 and 4 Query-Output-cmd.

**6.20.6 Query Measure Generator Current DC Command ( QMEASG<num>CDC )**

This procedure processes a single Query Measure Generator Current DC command “QMEASG<num>CDC”. Table 6.20-6 depicts the Query Measure Generator Current DC Command.

Table 6.20-6 – Query Measure Generator Current DC Command

COMMAND	Description
QMEASG<num>CDC	<p>Query Measure Generator Current DC</p> <p>&lt;num&gt; will be from one or all the V/I generators:</p> <p>&lt;num&gt; = 0 – All generators</p> <p>&lt;num&gt; = 1 – V/I generator in the Phase ‘A’ slot</p> <p>&lt;num&gt; = 2 – V/I generator in the Phase ‘B’ slot</p> <p>&lt;num&gt; = 3 – V/I generator in the Phase ‘C’ slot</p> <p>&lt;num&gt; = 4 – V/I generator in the Phase ‘D’ slot</p> <p>The Unit responds with the following :</p> <pre>MEASIG&lt;num&gt;,CDC,&lt;generator 0&gt;,&lt;current DC&gt;,&lt;generator 1&gt;,&lt;current DC&gt;;</pre> <p>(Note: If num equals zero, a result is returned for each of the VI generators installed. When more than one V/I generator is installed, the results are separated by a semicolon“:”.)</p>

The information for this command is obtained from a process running on the SBC that periodically constructs a table utilizing the output messages received from VI generators 1, 2,3 and 4 Query-Output-cmd.

## Horn Commands

### 6.21 Horn Commands

#### 6.21.1 Horn On Volume Command ( HONV<real > )

This procedure processes a single Horn On Volume command "HONV<real>". Table 6.21-1 depicts the Horn On Volume Command.

Table 6.21-1 - Horn On Volume Command

<b>COMMAND</b>	<b>Description</b>
HONV<real >	Horn On Volume

#### 6.21.2 Horn On Time Command ( HONT<milliseconds> )

This procedure processes a single Horn On Time command "HONT<milliseconds>". Table 6.21-2 depicts the Horn On Time Command.

Table 6.21-2 - Horn On Time Command

<b>COMMAND</b>	<b>Description</b>
HONT<milliseconds>	Horn On Time

## Directory and File Commands

### 6.22 Directory and File Commands

#### 6.22.1 List Directory Command ( DIR<path-name> )

This procedure processes a single List Directory command “DIR<pathname>”. Table 6.22-1 depicts the List Directory Command.

Table 6.22-1 - List Directory Command

COMMAND	Description
DIR<path-name>	List Directory  Note: one or more blanks can appear after “DIR”

The <path-name> used by this command can be a fully qualified pathname, or a partial pathname. Directory listing will include <file type> ('D' for directory and 'F' for file), <file size>, and <modification date/time>, the amount of space used, and the available space left on the solid-state disk. An example of a directory listing is depicted in figure 6.4 below.

Figure 6.4 - Directory Listing format

<file name 1 > <file type> <file size bytes> <modification date/time>

<file name (n)> <file type> <file size bytes > < modification date/time>

Space Used: <xxxxx> byte Space Available: <xxxxx> bytes;

#### 6.22.2 Make Directory Command ( MD<path-name> )

This procedure processes a single Make Directory command “MD<pathname>”. Table 6.22-2 depicts the Make Directory Command.

Table 6.22-2 - Make Directory Command

COMMAND	Description
MD<path-name>	Make Directory  Note: one or more blanks can appear after “MD”

The <path-name> used by this command can be a fully qualified pathname, or a partial pathname. Directory listing will include the amount of space used, and the available space left on the solid-state disk.

#### 6.22.3 Change Directory Command ( CD <path-name> )

This procedure processes a single Change Directory command “CD<pathname>”. Table 6.22-3 depicts the Change Directory Command.

Table 6.22-3 - Change Directory Command



COMMAND	Description
CD <path-name>	Change Directory  Note: one or more blanks can appear after "CD"

The <path-name> used by this command can be a fully qualified pathname.

**6.22.4 Delete File or Directory Command ( DEL<path-name> )**

This procedure processes a single Delete File or Directory command "DEL<pathname>". Table 6.22-4 depicts the Delete File or Directory Command.

Table 6.22-4 - Delete File or Directory Command

COMMAND	Description
DEL<path-name>	Delete File or Directory  Note: one or more blanks can appear after "DEL"

The <path-name> used by this command can be a fully qualified pathname, or a partial pathname. If <path-name> is a directory, the directory cannot be deleted if files exist in the directory.

**6.22.5 Rename File or Directory Command ( REN<opname>\<nname> )**

This procedure processes a single Rename File or Directory command "REN<old-path-name>\<new-path-name>". Table 6.22-5 depicts the Rename File or Directory Command.

Table 6.22-5 - Rename File or Directory Command

COMMAND	Description
REN<old-path-name>\<new-path-name>	Rename File or Directory  Note: one or more blanks can appear after "REN"

The <old-path-name> and <new-path-name> used by this command can be a fully qualified pathname, or a partial pathname.

**6.22.6 Copy File Command ( COPY <old-path-name>\<new-path-name> )**

This procedure processes a single Copy File command "COPY<old-path-name>\<new-path-name>". Table 6.22-6 depicts the Copy File or Directory Command.

Table 6.22-6 - Copy File Command

COMMAND	Description
COPY <old-path-name>\<new-path-name>	Copy File  Note: one or more blanks can appear after "COPY"

The <old-path-name> and <new-path-name> used by this command can be a fully qualified pathname, or a partial pathname.

**6.22.7 Move File Command ( MOVE <old-path-name>\<new-path-name> )**

This procedure processes a single Move File command “MOVE<old-path-name>\<new-path-name>”. Table 6.22-7 depicts the Move File Command.

Table 6.22-7 - Move File Command

<b>COMMAND</b>	<b>Description</b>
MOVE <old-path-name>\<new-path-name>	Move File  Note: one or more blanks can appear after “MOVE”

The <old-path-name> and <new-path-name> used by this command can be a fully qualified pathname, or a partial pathname.

**6.22.8 Read File Command ( FILER<path-name> )**

This procedure processes a single Read File command “FILER<pathname>”. Table 6.22-8 depicts the Read File Command.

Table 6.22-8 - Read File Command

<b>COMMAND</b>	<b>Description</b>
FILER<path-name>	Read File  Note: one or more blanks can appear after “FILER”

The <path-name> used by this command can be a fully qualified pathname, or a partial pathname.

**6.22.9 Write File Command ( FILEW<overwrite>\<pathname>\<size>\<data> )**

This procedure processes a single Write File command “FILEW<over-write>\<pathname>\<size>\<-data>”. Table 6.22-9 depicts the Write File Command.

Table 6.22-9 - Write File Command

<b>COMMAND</b>	<b>Description</b>
FILEW<overwrite>\<pathname>\<size>\<data>	Write File  Where <overwrite> is optional and equals “O”.  Note: one or more blanks can appear after “FILEW”

The <path-name> used by this command can be a fully qualified pathname, or a partial pathname. The variable <overwrite> is optional, and is used to overwrite an existing file. If the write exist and the <overwrite> option is not specified, an error message is generated. The <size> and <data> are the file size and file data associated with the file pathname.

**6.22.10 Get Working Directory Command ( GWD )**

This procedure processes a single Get Working Directory command "GWD". Table 6.22-10 depicts the Get Working Directory Command.

Table 6.22-10 – Get Working Directory Command

<b>COMMAND</b>	<b>Description</b>
GWD	Get Working Directory

This command returns a fully qualified directory <path-name> that is the current work directory.

## Battery Simulator Commands

### 6.23 Battery Simulator Commands

#### 6.23.1 Battery Simulator Off ( BATOF )

This procedure processes a single Battery Simulator Off command “BATOF”. Table 6.23-1 depicts the Battery Simulator Off Command.

Table 6.23-1 – Battery Simulator Off Command

COMMAND	Description
BATOF	Battery Simulator Off  This command turns the Battery Simulator OFF

Set the variable port-address equal to the address of the Battery Simulator defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Logically OR the variable Toggle-Battery-Simulator-Reset with zero (“0”) and set the variable Cmd-string equal to the results. Call the procedure 7.1 Send Command (SC), with the Cmd-string.

#### 6.23.2 Battery Simulator On ( BATONRA<range> )

This procedure processes a single Battery Simulator On command “BATONRA<range>”. Table 6.23-2 depicts the Battery Simulator On Command.

Table 6.23-2 – Battery Simulator On Command

COMMAND	Description
BATONRA<range>	Battery Simulator On  This command turns the Battery Simulator ON.  Where <range> is 24,48,125, or 250

Set the variable port-address equal to the address of the Battery Simulator defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document.

If the parameter <range> equals 24, then logically OR the variable Toggle-Battery-Simulator-Reset with the value “4”, and set the variable Cmd-string equal to the results.

If the parameter <range> equals 48, then logically OR the variable Toggle-Battery-Simulator-Reset with the value “5”, and set the variable Cmd-string equal to the results.

If the parameter <range> equals 125, then logically OR the variable Toggle-Battery-Simulator-Reset with the value “6”, and set the variable Cmd-string equal to the results.

If the parameter <range> equals 250, then logically OR the variable Toggle-Battery-Simulator-Reset with the value “7”, and set the variable Cmd-string equal to the results.

Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string.

#### 6.23.3 Battery Simulator Reset ( BATRE )

This procedure processes a single Battery Simulator Reset command “BATRE”. Table 6.23-3 depicts the Battery Simulator Reset Command.

Table 6.23-3 – Battery Simulator Reset Command

<b>COMMAND</b>	<b>Description</b>
BATRE	Battery Simulator Reset  This command turns the Battery Simulator RESET

Set the variable port-address equal to the address of the Battery Simulator defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document.

If the variable Toggle-Battery-Simulator-Reset equals zero "0" then set the variable Toggle-Battery-Simulator-Reset equal to "1", otherwise set the variable Toggle-Battery-Simulator-Reset equal to zero "0".

Set the variable Cmd-string equal to Toggle-Battery-Simulator-Reset. Call the procedure 7.1 Send Command (SC), with the Cmd-string.

## Vector Scope Configuration Commands

### Vector Scope Configuration Command

#### 6.24.1 Set Vector Scope to 360° Lead Phase Angle Format Command (PLEAD)

The Set Vector Scope to 360° Lead Phase Angle Format Command set the Vector scope phase angle format to 360° Lead. Table 6.24-1 depicts the Set Vector Scope to 360° Lead Phase Angle Format Command.

Table 6.24-1 – Set Vector Scope to 360° Lead Phase Angle Format Command

<b>COMMAND</b>	<b>Description</b>
PLEAD	Set Vector Scope to 360° Lead Phase Angle Format

#### 6.24.2 Set Vector Scope to 360° Lag Phase Angle Format Command (PLAG)

The Set Vector Scope to 360° Lag Phase Angle Format Command set the Vector scope phase angle format to 360° Lag. Table 6.24-2 depicts the Set Vector Scope to 360° Lag Phase Angle Format Command.

Table 6.24-2 – Set Vector Scope to 360° Lag Phase Angle Format Command

<b>COMMAND</b>	<b>Description</b>
PLAG	Set Vector Scope to 360° Lag Phase Angle Format

#### 6.24.3 Set Vector Scope to ±180° Phase Angle Format Command (PAOE)

The Set Vector Scope to ±180° Phase Angle Format Command set the Vector scope phase angle format to ±180°. Table 6.24-3 depicts the Set Vector Scope to ±180° Phase Angle Format Command.

Table 6.24-3 – Set Vector Scope to ±180° Phase Angle Format Command

<b>COMMAND</b>	<b>Description</b>
PAOE	Set Vector Scope to ±180° Phase Angle Format

## Command Set Procedures - New

## Command Set Procedures - New

### 2.0 Command Set Procedures

In addition to the original RTS document, this document contains the latest RTS commands and changes that develop. This document will eventually merge with the original one to make one unified RTS document. Due to the complexity of the original RTS document, this document will, right now, just contain the latest RTS commands.

This document contains the latest release of new RTS commands to accommodate the new DSP (Timer/VIGEN) Software.

These new commands are in addition to those documented in the Command Set Procedures topic.

- Set Wave Index
- Sin Sum
- Point on Wave
- Sweep Frequency
- Set Record ON/OFF
- Set Record Rate
- Record Capture Data
- Record Capture Info
- Record Reset
- Wait for Any Binary Input
- Wait for All Binary Input
- Timer Set Option
- Get Record Data (TIMER)
- Wait Phase (TIMER)
- Set Debounce Mode (TIMER)
- Start Virtual Timer (TIMER)
- Stop Virtual Timer (TIMER)
- Query Virtual Timer (TIMER)

## Set Wave Index

### 2.1 Set Wave Index

There are four wave indices (1 – 4), and the system will automatically select the first wave index if there is no wave index selected. Similar to the V and C commands, this command allows user to select different indices for the settings (amplitude/frequency/phase), and the result is the sum of all the four wave indices. By default, all the wave indices that are not selected will be zero (so the sum of zero won't affect the result).

COMMAND	Description
WAVENUM<index>	Where <index> is 1, 2, 3, 4. Selects the waveform for all subsequent commands.

For example: V123WAVENUM1, A10,f60,WAVENUM2,A2,f120; will set amplitude of 10 at 60 Hz plus 2 amps of 120 Hz to Generators 1, 2, and 3.



## Sine Sum

### 2.2 Sine Sum

COMMAND	Description
WAV<loc>SI<fv>.<freq>.<phase>	Where <loc> is the location (any number is OK for now), <fv> is fractional value of the amplitude, <freq> is the multiplier of the default frequency, and <phase> is the phase value

For example: V1, A10, ON; WAV4SI.5:1:0; WAV4SI.5:2:0; WAV4SI.5:3:0; - you will have the sum of (A5,F60,P0), (A5,F120,P0) and (A5,F180,P0) with assumption that the default frequency is 60.

Note: The amplitude sum may be greater than 1 per unit.

## Point on Wave

### 2.3 Point on Wave

<b>COMMAND</b>	<b>Description</b>
POW<degree>	Where <degree> is a positive floating point value from 0-360

## Sweep Frequency

### 2.4 Sweep Frequency

COMMAND	Description
DF<mode>:<delta>	Where <mode> is an integer and <delta> is a floating point

## Set Record ON/OFF

### 2.5 Set Record ON/OFF

COMMAND	Description
SREC<mode>	Start or stop the recording for both the timer and all channels. <mode> is an integer to turn 1/0 for on/off

This command will set recording on/off to ALL VIGENS and TMR (0xA7=on, 0x9d=off)

## Set Record Rate

### 2.6 Set Record Rate

<b>COMMAND</b>	<b>Description</b>
RECRATE<rate>	Sets the record rate for all channels. Rate is an integer for record rate (0 is high, 1 is medium, and 2 is low)

RECRATE will apply to ALL VIGENS & TMR

## Record Capture Data

### 2.7 Record Capture Data

<b>COMMAND</b>	<b>Description</b>
CAPDATA	Returns the captured data from the selected channel(s).

## Record Capture Info

### 2.8 Record Capture Info

<b>COMMAND</b>	<b>Description</b>
CAPINFO	Returns the capture info from the selected channels

## Record Reset

### 2.9 Record Reset

<b>COMMAND</b>	<b>Description</b>
RECRSET	Resets Timer and ALL VIGEN recording

Reset ALL VIGENS & TMR



## Wait for Any Binary Input

### 2.10 Wait for Any Binary Input

COMMAND	Description
WANY< <u>binarvinput</u> >	Where < <u>binarvinput</u> > is a combination of 0, 1 and X. The system has up to 10 binary inputs, and those 0, 1, and X represent one physical binary input

Note: X = ignore

## Wait for All Binary Input

### 2.11 Wait for All Binary Input

Different from WANY command, WALL command will tell the MPRT will wait for ALL of the following binary input (that user entered) is true for an action to take place.

COMMAND	Description
WALL< <u>binaryinput</u> >	Where < <u>binaryinput</u> > is a combination of 0, 1 and X. The system has up to 10 binary inputs, and those 0, 1, and X represent one physical binary input

Note: X = ignore

## Timer Set Output

### 2.12 Timer Set Output

COMMAND	Description
TSOP< <u>binaryoutput</u> >	Where < <u>binaryoutput</u> > is a combination of 0, and 1. The system has up to 6 binary outputs, and those 0, and 1 represent the physical binary output.

## Get Record Data (TIMER)

### 2.13 Get Record Data (TIMER)

<b>COMMAND</b>	<b>Description</b>
TGD	will return the captured data

## Wait Phase (TIMER)

### 2.14 Wait Phase (TIMER)

COMMAND	Description
TWP<phase>	Will wait for a phase to happen to perform some action.

Previous Frequency setting will be sent down, if no frequency was set, the default frequency will be sent down instead.

## Set Debounce Mode (TIMER)

### 2.15 Set Debounce Mode (TIMER)

COMMAND	Description
TDM<P>.<M>.<T>	Set the <u>debounce</u> mode, where the <P> is binary input post(1-10), and the <M> is the mode (0-leading, 1-trailing, 2-none) and <T> is for time (in millisecond, can be floating point)

## Start Virtual Timer (TIMER)

### 2.16 Start Virtual Timer (TIMER)

COMMAND	Description
STARTVT<T>	Starts a Virtual Timer.  <T> is the Virtual Timer Number ( Integer [1-100] )

## Stop Virtual Timer (TIMER)

### 2.17 Stop Virtual Timer (TIMER)

COMMAND	Description
STOPVT<T>	Stops a Virtual Timer.  <T> is the Virtual Timer Number ( Integer [1-100] )



## Query Virtual Timer (TIMER)

### 2.18 Query Virtual Timer (TIMER)

COMMAND	Description
QVT<T>	Queries a Virtual Timer. Returns the Time in seconds.<T> is the Virtual Timer Number ( Integer [1-100] )

## **Interface Procedures**

## **Interface Procedures**

### **7.0 Interface Procedures**

The procedures in the sections that follow describe the interface require to communicate with the UART controlling the V/I Generators and Timer Unit.

## Send Command

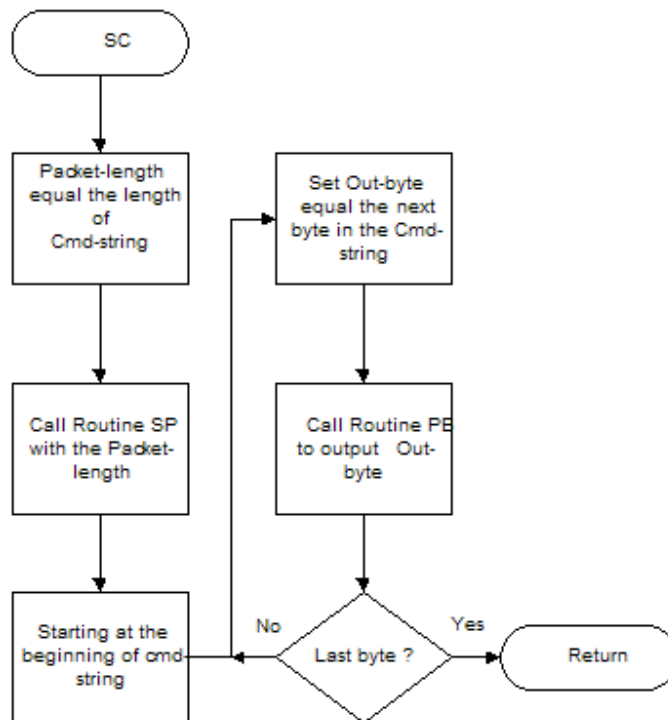
### 7.1 Send Command (SC)

This procedure provides communication between the Host processor, and the UART controlling the V/I Generators and Timer. Input to this routine is Cmd-string with containing a command and optional data bytes.

Set the variable Packet-length equal to the number of bytes in Cmd-string, and call the routine 7.1.1 **Start Packet (SP)** to output the Packet-length to the UART.

Call the routine in section, 7.1.2 Put Byte (PB), to output the command, and any optional data bytes in Cmd-string. An Algorithm that depicts the Send Command procedure is demonstrated in Figure 7.1.

Figure 7.1 - Send Command Flowchart

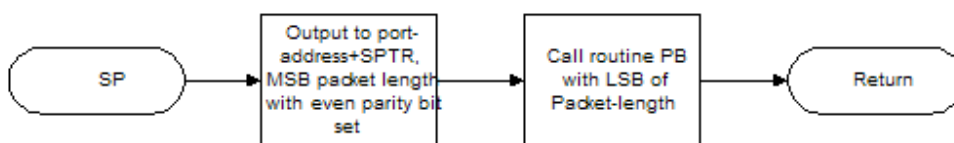


#### 7.1.1 Start Packet (SP)

Input to this procedure is the Packet-length of the current command. This procedure outputs the most significant and least significant bytes of the Packet-length to the UART controlling the V/I generators or Timer. Output the MSB of the packet length to the Serial Port Transmitter Data register (Port-address + SPTR) setting the Even-parity-bit. Call the procedure 7.1.2 Put Byte (PB) with the LSB of the Packet-length setting the Odd-parity-bit.

An Algorithm that depicts the Start Packet procedure is demonstrated in Figure 7.2.

Figure 7.2 - Start Packet Flowchart

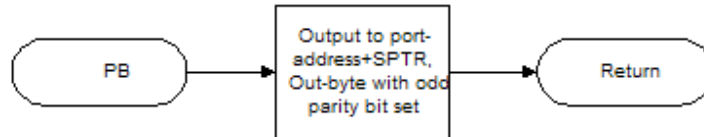


### 7.1.2 Put Byte (PB)

Input to this procedure is Out-byte. This procedure outputs a byte of data, Out-byte, to the UART controlling the V/I generators or Timer. Output the byte to the Serial Port Transmitter Data register (Port-address + SPTR) setting the Odd-parity-bit.

An Algorithm that depicts the Start Packet procedure is demonstrated in Figure 7.3.

Figure 7.3 - Put Byte Flowchart

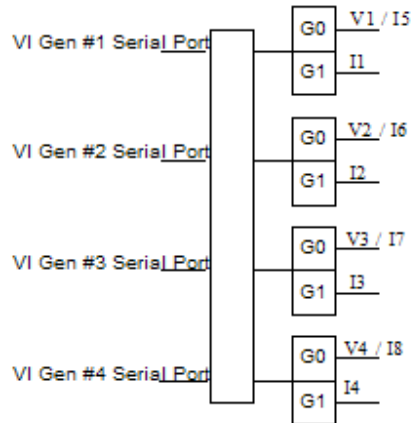


## Selecting VI Generators

### 7.2 Selecting VI Generators

The following sections describe the interface to the V/I Generators and the procedures required to select them. Figure 7.4 demonstrates the hardware interface utilized in the selection of V/I generators. The procedures for selecting one of the four Voltage generators, or eight Current generators are described in sections 7.2.1 and 7.2.2 respectively.

Figure 7.4 -VI Generators Serial Port Hardware Diagram



#### 7.2.1 Set Voltage Generator (SETVG)

This procedure instructs the UART controlling the V/I Generators, to select one of the four voltage generators previously selected via a Generator Select Command. Input to this procedure is I, which identify one of the four voltage generators defined for the Genesis Test Set.

Set the variable port-address equal to the address VI Gen # (I) defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Where I represent one of the Voltage generators (V1,V2,V3,V4). Set the Cmd-string equal to the Generator-select-cmd. Append a one-byte integer value 0 , to the Cmd-string.

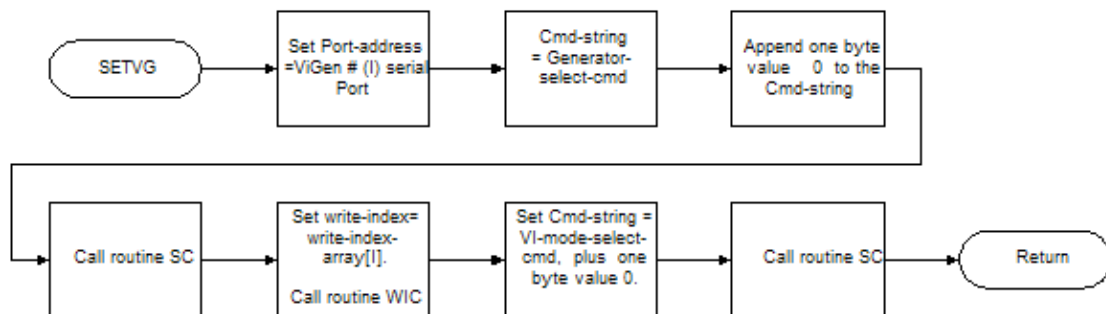
Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string. This selects generator G0 as depicted in Figure 7.4, define in section 7.2 **Selecting VI Generators**.

Set the variable write-index equal to write-index-array[I]. Call the procedure 7.2.3 Write Index Command (WIC), to perform a write-index command to the parameter queue.

Set Cmd-string equal to the VI-mode-select-cmd. Append a one-byte integer value 0 , to the Cmd-string. This will place the selected generator in voltage mode. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string.

An Algorithm that depicts the Set Voltage Generator procedure is demonstrated in Figure 7.5.

Figure 7.5 - Set Voltage Generator Flowchart



### 7.2.2 Set Current Generator (SETCG)

This procedure instructs the UART controlling the V/I Generators, to select one of the eight current generators previously selected via a Generator Select Command. Input to this procedure is I, with identify one of the eight current generators defined for the Genesis Test Set.

First map the current generators to the correct port addresses as illustrated in Figure 7.1, of section 7.2 Selecting VI Generators.

If I equal 1 or 5 then Port-address ViGen #1 serial Port defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable write-index equal to write-index-array[1].

If I equal 2 or 6 then Port-address ViGen #2 serial Port defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable write-index equal to write-index-array[2].

If I equal 3 or 7 then Port-address ViGen #3 serial Port defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable write-index equal to write-index-array[3].

If I equal 4 or 8 then Port-address ViGen #4 serial Port defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable write-index equal to write-index-array[4].

If I is not in the range (1-4 ) then an error has occurred.

Next, set Cmd-string equal to the Generator-select-cmd. If I equal 1,2,3, or 4, then append a one-byte integer value 1, to the Cmd-string. Otherwise, append a one-byte integer value 0, to the Cmd-string.

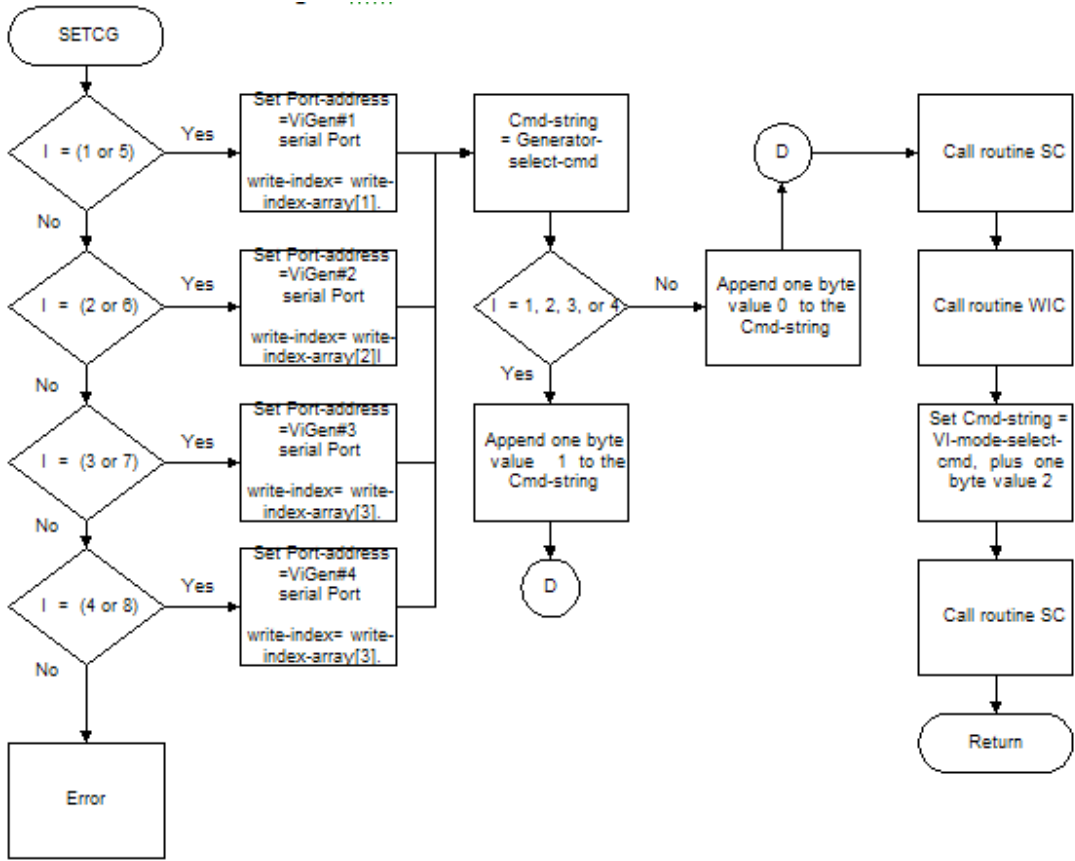
Call the procedure 7.1 Send Command (SC), with the Cmd-string and Port-address. This will selects the correct generator ,G0 or G1, has show in Figure 7.1.

Call the procedure 7.2.3 Write Index Command (WIC), to perform a write-index command to the parameter queue.

Set Cmd-string equal to the VI-mode-select-cmd, and call routine SC. Append a one-byte integer value 2, to the Cmd-string. This will place the selected generator in current mode. Call the procedure 7.1 Send Command (SC), with the Cmd-string.

An Algorithm that depicts the Set Current Generator procedure is demonstrated in Figure 7.6.

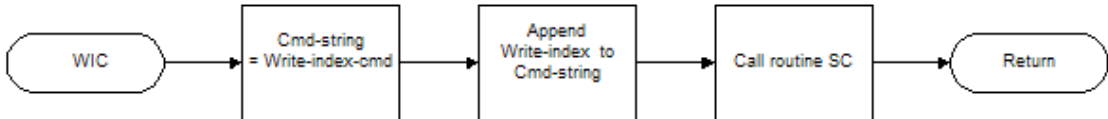
Figure 7.6 - Set Current Generator Flowchart



**7.2.3 Write Index Command (WIC)**

This procedure writes the current one-byte write-index utilizing the write-index-command. An Algorithm that depicts the procedure is demonstrated in Figure 7.7.

Figure 7.7 - Write Index Command Flowchart



## Execute Selected Voltage Generator Command

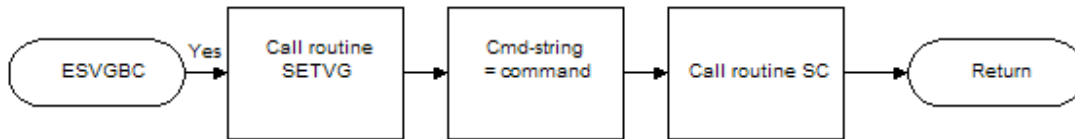
### 7.3 Execute Selected Voltage Generator Command (ESVGBC)

Input to this procedure is Command and I. The variable I identifies one of the selected Voltage generators. The variable I is set by the algorithm in figure 6.3 (a).

This procedure executes the command for any of the voltage generators that were previously selected by a Generator select command

Call procedure 7.2.1 Set Voltage Generator (SETVG), to enable the correct Voltage generator. Set Cmd-string equal to the variable Command, and call procedure 7.1 **Send Command (SC)**, with the Cmd-string. An Algorithm that depicts the procedure is demonstrated in Figure 7.8.

Figure 7.8 - Execute selected generator voltage command Flowchart





## Execute Selected Current Generator Command

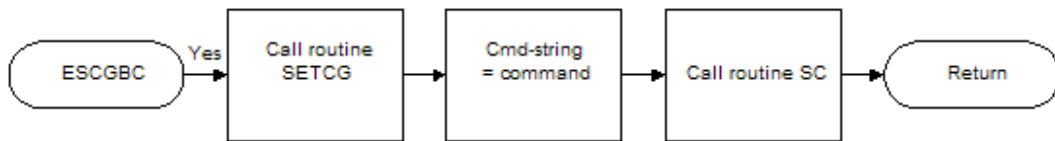
### 7.4 Execute Selected Current Generator Command (ESCGBC)

Input to this procedure is Command and I. The variable I identifies one of the selected Current generators. The variable I is set by the algorithm in figure 6.3 (b).

This procedure executes the command for any of the current generators that were previously selected by a Generator select command.

Call procedure 7.2.2 Set Current Generator (SETCG), to enable the correct Current generator. Then, set Cmd-string equal to the variable Command, and call procedure 7.1 **Send Command (SC)**, with the Cmd-string. An Algorithm that depicts the procedure is demonstrated in Figure 7.9.

Figure 7.9 - Execute selected generator current command Flowchart



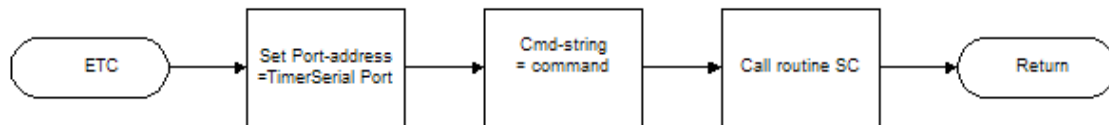
## Execute Timer Command

### 7.5 Execute Timer Command (ETC)

Input to this procedure is Command. This procedure processes timer commands. Set the address variable Port-address to the Timer Serial Port address defined in section 2.4 I/O Memory Map , of the ISA FPGA Description document.

Set Cmd-string equal to the variable Command. Call the procedure in section 7.1 Send Command (SC) with the Cmd-string. An Algorithm that depicts the procedure is demonstrated in Figure 7.10.

Figure 7.10 - Execute Timer Command Flowchart



## Compute and Generate Waveform or DFR Physical Address

### 7.6 Compute and Generate Waveform or DFR Physical Address (CAGWPA)

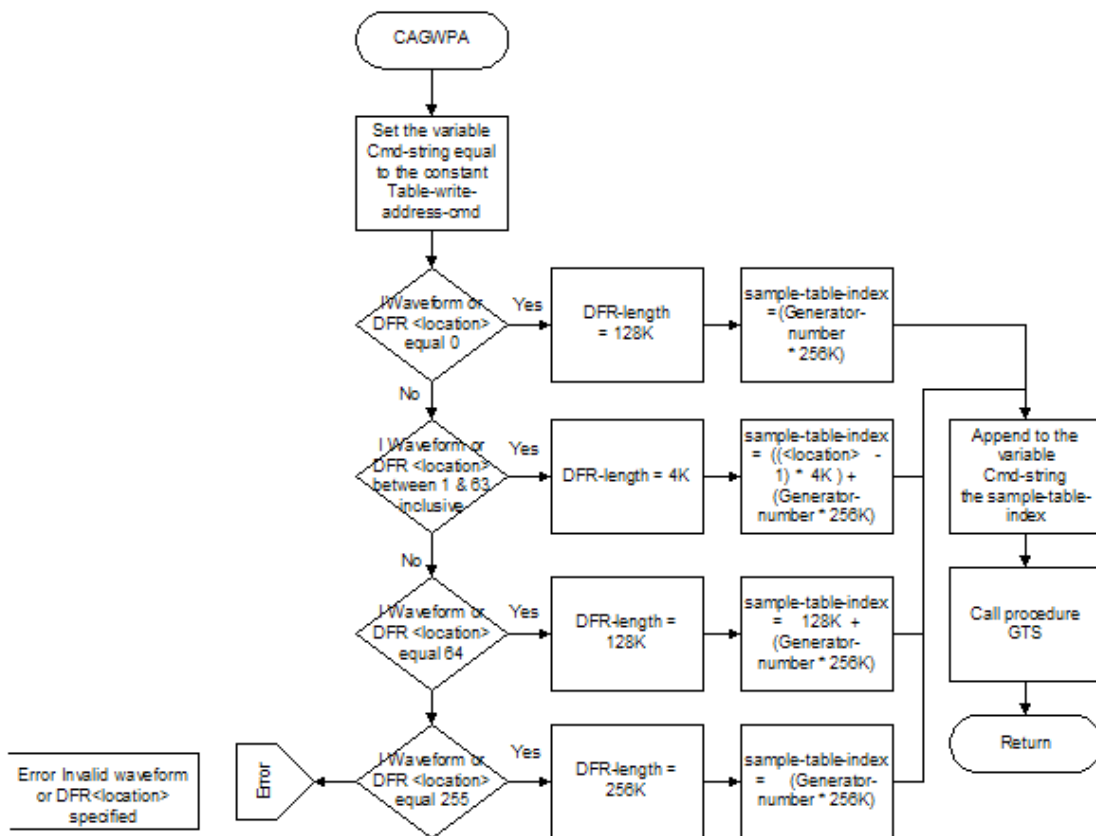
First set the variable Cmd-string equal to the Table-write-address-cmd.

Compute the waveform physical memory address and writes it to the parameter queue. The waveform physical memory location is computed as follow.

If the <location> is zero, then DFR-length equals 128K and the physical memory address is equal to the Generator-number times 256K. If the <location> is between 1 and 63 inclusive, then the DFR-length equals 4K and the physical memory address is equal  $((\text{<location>} - 1) * 4K) + (\text{Generator-number} * 256K)$ . If the <location> equals 64, then DFR-length equals 128K and the physical memory address is equal to the 128K plus (Generator-number times 256K). If the <location> equals 255, then DFR-length equals 256K and the physical memory address is equals (Number-number times 256K). Otherwise an illegal waveform/DFR address has been specified.

Append the 4-byte integer start address (physical memory address) to Cod-string and call procedure 7.10 **Generator Type Select (GTS)** with the String-string. This routine returns the DFR-length, and the physical memory address in the variable sample-table-index. An Algorithm that depicts the procedure is demonstrated in Figure 7.11.

Figure 7.11 – Compute and Generate Waveform Physical Address Flowchart



## Store DFR Binary Data

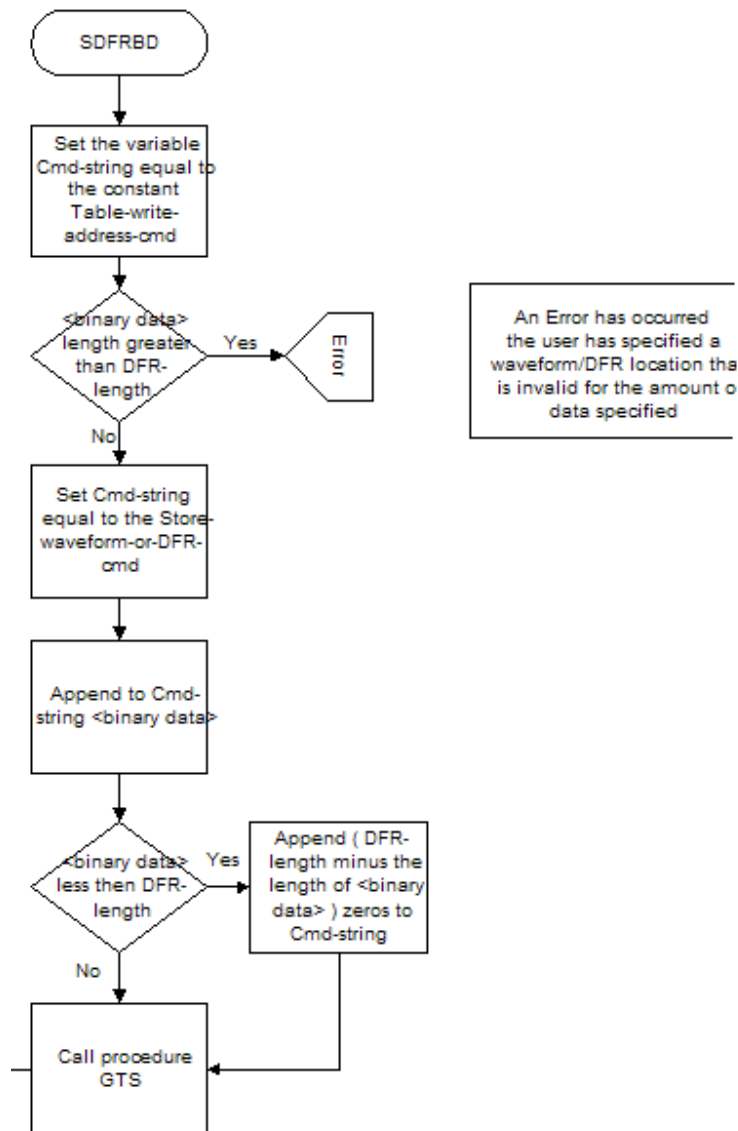
### 7.7 Store DFR binary Data (SDFRBD)

This procedure is used to store a DFR data for the VI Generators. If the length of the <binary data> is greater than the value of the variable DFR-length, then an error has occurred.

Otherwise, set the variable String-string equal to the Store-waveform-or-DFR-cmd. Append to Cmd-string <binary data> (Note: <binary data> can be a maximum length of 256K).

If the length of the variable <binary data> is less then DFR-length, then append (DFR-length minus the length of <binary data>) zeroes to Cmd-string to fill the remaining portion of the buffer. Call procedure, 7.10 Generator Type Select (GTS) with the Cmd-string. An Algorithm that depicts the procedure is demonstrated in Figure 7.12.

Figure 7.12 –Store DFR binary Data Flowchart



## Output Off State Three

### 7.8 Output Off State three (OOST)

Set the variable Cmd-string equal to the Amplifier-on-cmd. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-parameter-Dc-level-cmd. Append a 4-byte real value 0 to Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Parameter-set-mode-cmd. Append a 1-byte range number value Starting mode Immediate to Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Sine-wave-cmd. Append a 1-byte integer component value 0, a 1-byte integer harmonic value 1, 4-byte real amplitude value 0, and 4-byte real phase-angle value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Sine-wave-cmd. Append a 1-byte integer component value 1, a 1-byte integer harmonic value 1, 4-byte real amplitude value 0, and 4-byte real phase-angle value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Sine-wave-cmd. Append a 1-byte integer component value 2, a 1-byte integer harmonic value 1, 4-byte real amplitude value 0, and 4-byte real phase-angle value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Sine-wave-cmd. Append a 1-byte integer component value 3, a 1-byte integer harmonic value 1, 4-byte real amplitude value 0, and 4-byte real phase-angle value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Half-Sine-wave-cmd. Append a 4-byte real amplitude value 0, and 4-byte real phase-angle value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Queued-Rectangle-wave-cmd. Append a 4-byte real amplitude value 0, a 4-byte real phase-angle value 0, and a 4-byte real duty value 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

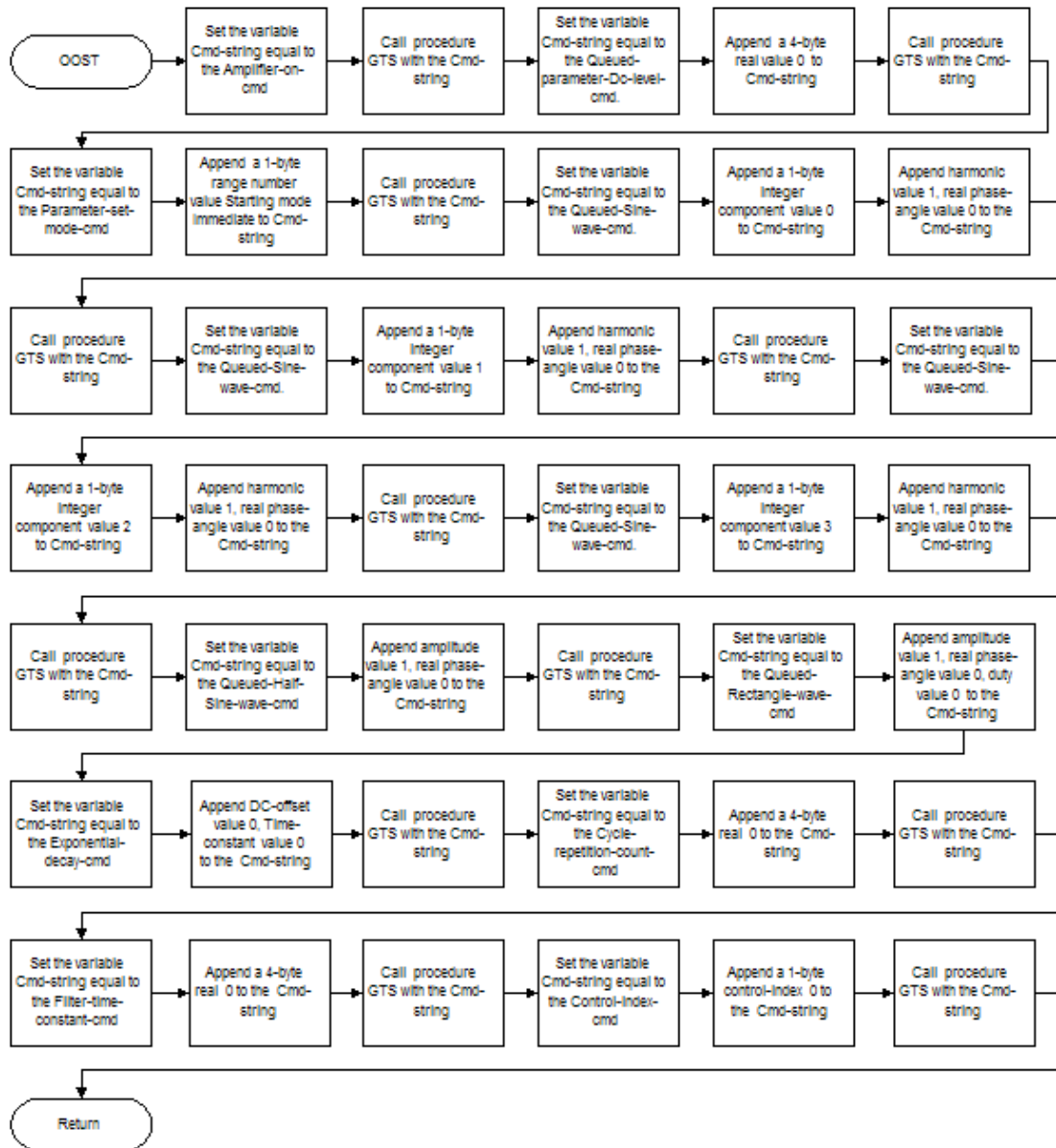
Set the variable Cmd-string equal to the Exponential-decay-cmd. Append a 4-byte real DC-offset 0, and a 4-byte real Time-constant 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Cycle-repetition-count-cmd. Append a 4-byte real 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Filter-time-constant-cmd. Append a 4-byte real 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string.

Set the variable Cmd-string equal to the Control-index-cmd. Append a 1-byte control-index 0 to the Cmd-string. Call the procedure 7.10 **Generator Type Select (GTS)** with the Cmd-string. An Algorithm that depicts the procedure is demonstrated in Figure 7.13.

Figure 7.13 – Output Off State three Flowchart

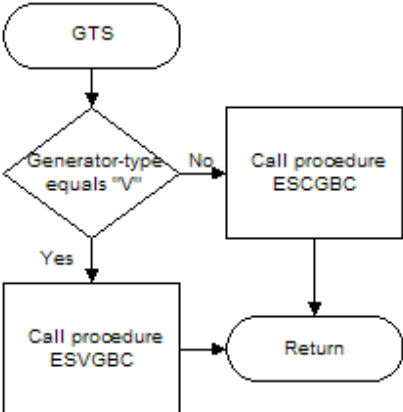


# Generator Type Select

## 7.9 Generator Type Select (GTS)

This procedure executes the command in Cmd-string. If Generator-type equals "V" then call procedure 7.3 **Execute Selected Voltage Generator Command (ESVGBC)** to perform the command, otherwise call procedure 7.4 **Execute Selected Current Generator Command (ESCGBC)**. An Algorithm that depicts the procedure is demonstrated in Figure 7.14.

Figure 7.14 – Generator Type Select Flowchart

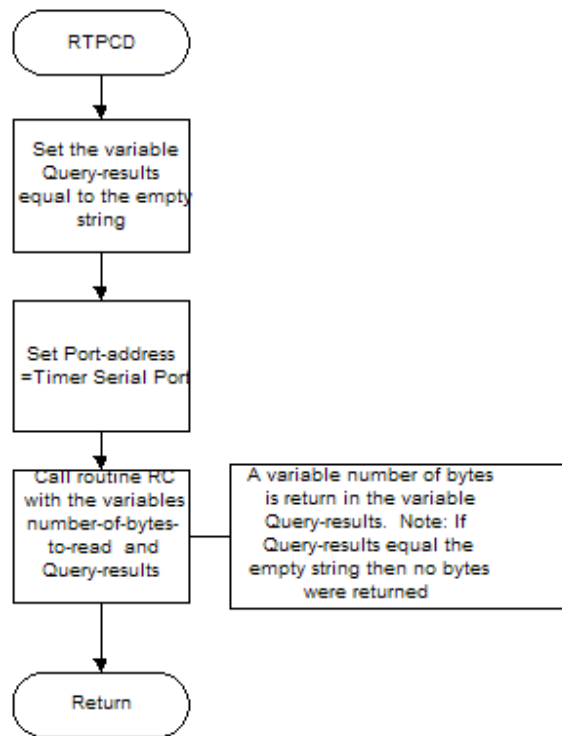


## Read Timer Port Command and Data

### 7.10 Read Timer Port Command and Data (RTPCD)

Input to this procedure is number-of-bytes-to-read. This procedure reads the number of bytes from the Timer port address defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Call the procedure in section 7.12 **Read Command (RC)** with the Query-result and number-of-bytes-to-read variable. The results of the read are returned in the variable Query-results. An Algorithm that depicts the procedure is demonstrated in Figure 7.15.

Figure 7.15 – Read Timer Port Command and Data Flowchart



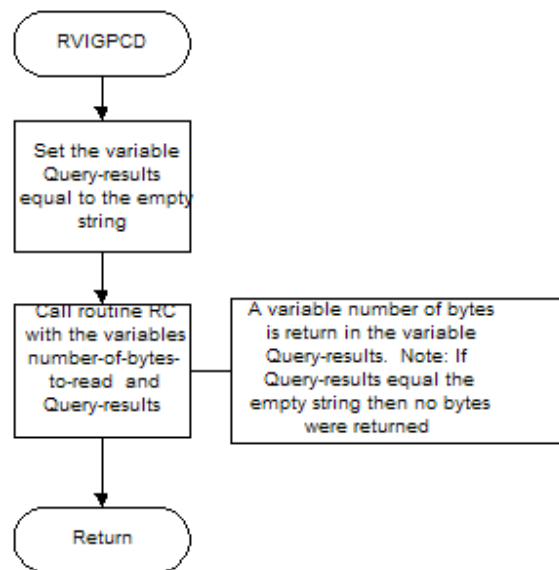


## Read VI Generator Port Command and Data

### 7.11 Read VI Generator Port Command and Data (RVIGPCD)

Input to this procedure is number-of-bytes-to-read, and Port-address. This procedure reads the number of bytes from 1 of the 4 VI Generator port address defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Call the procedure in section 7.12 **Read Command (RC)** with the Query-result and number-of-bytes-to-read variable. The results of the read are returned in the variable Query-results. An Algorithm that depicts the procedure is demonstrated in Figure 7.16.

Figure 7.16 – Read VI Generator Port Command and Data Flowchart



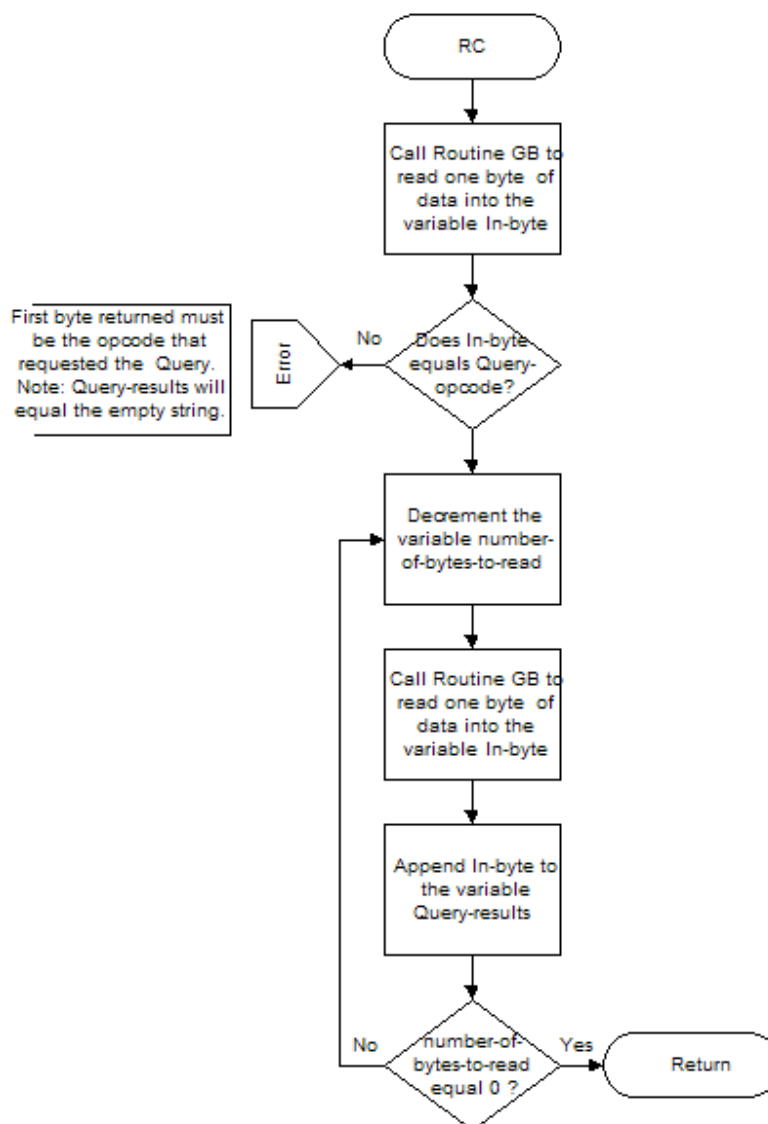
## Read Command

### 7.12 Read Command (RC)

This procedure provides communication between the Host processor, and the UART controlling the V/I Generators and Timer. Input to this routine is the variable Query-opcode, Query-results and number-of-bytes-to-read.

Call procedure, 7.12.1 **Get Byte (GB)**, to read the first byte from the Port-address and compare it to the variable Query-opcode. If the first byte was not equal to the Query-opcode, then an error has occurred. Otherwise call procedure, 7.12.1 **Get Byte (GB)**, to read number-of-bytes-to-read minus 1 bytes from Port-address and store the results in the variable Query-result. An Algorithm that depicts the Read Command procedure is demonstrated in Figure 7.17.

Figure 7.17 – Read Command Flowchart

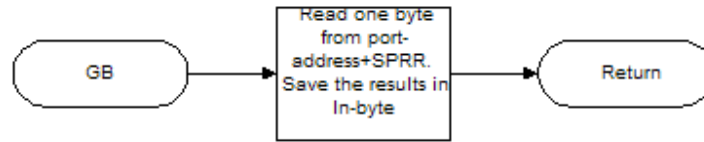


#### 7.12.1 Get Byte (GB)

This procedure returns a byte of data, In-byte, from the UART controlling the V/I generators or Timer. Input a byte from the Serial Port Receiver Data register (Port-address + SPRR).

An Algorithm that depicts the Get Byte procedure is demonstrated in Figure 7.18.

Figure 7.18 - Get Byte Flowchart



## Determine Number VI Generators Present

### 7.14 Determine the Number of VI Generators Present (DNOVIGP)

This routine will create the <G#> portion of the Configuration-Query-Results. Set the variable Configuration-Query-Results equal to the character "G". Set the variable Number-of-VI-Generators-present equal to zero.

Set the variable port-address equal to the address VI Gen # 1 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document.

Set the variables Cmd-string and Query-opcode equal to the Link-test-cmd. Append a one-byte integer value 0, to the Cmd-string. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

Set the variable number-of-bytes-to-read to 1. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

If Query-results are equal to the Cmd-string, then increment by one the variable Number-of-VI-Generators-present.

Set the variable port-address equal to the address VI Gen # 2 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

Set the variable number-of-bytes-to-read to 1. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

If Query-results are equal to the Cmd-string, then increment by one the variable Number-of-VI-Generators-present.

Set the variable port-address equal to the address VI Gen # 3 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

Set the variable number-of-bytes-to-read to 1. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

If Query-results are equal to the Cmd-string, then increment by one the variable Number-of-VI-Generators-present.

Set the variable port-address equal to the address VI Gen # 4 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

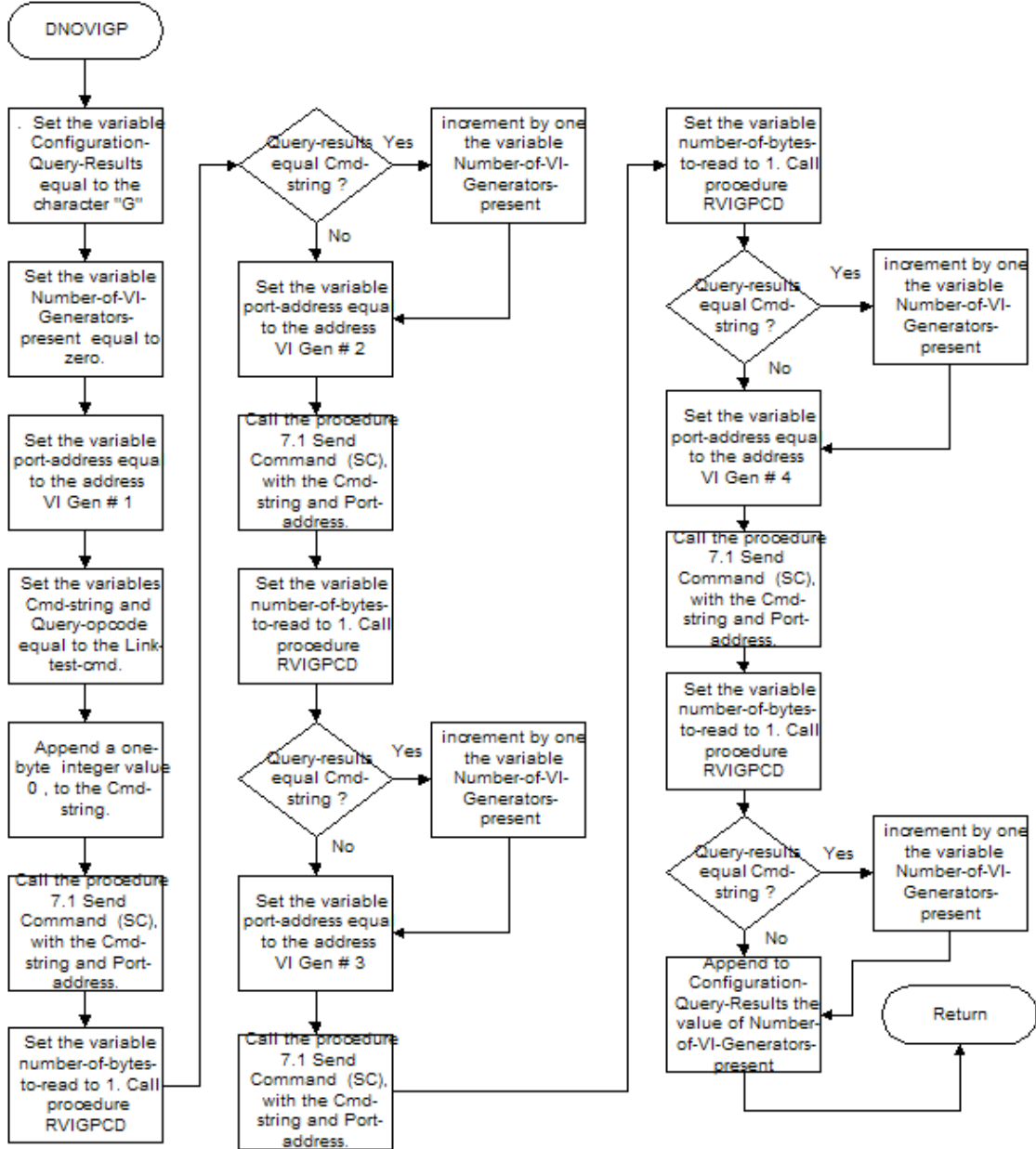
Set the variable number-of-bytes-to-read to 1. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

If Query-results are equal to the Cmd-string, then increment by one the variable Number-of-VI-Generators-present.

Append to the variable Configuration-Query-Results the value of the variable Number-of-VI-Generators-present.

An Algorithm that depicts the procedure is demonstrated in Figure 7.19.

Figure 7.19 - Determine the Number of VI Generators present Flowchart



## Get Timer Query Information

### 7.15 Get Timer Query Information (GTQI)

This routine will create the <IO>\*<VER><nn.nn><hh.hh><SNxxxx> portion of the Configuration-Query-Results.

Set the variables Command and Query-opcode equal to the Query-Timer-Revision-cmd. Call the procedure 7.5 **Execute Timer Command (ETC)** with the Command.

Set the variable number-of-bytes-to-read to 15. Call procedure 7.10 **Read Timer Port Command and Data (RTPCD)**, return the results in variable Query-results.

If the Query-results returned the empty string, then no timer is present, return to the caller.

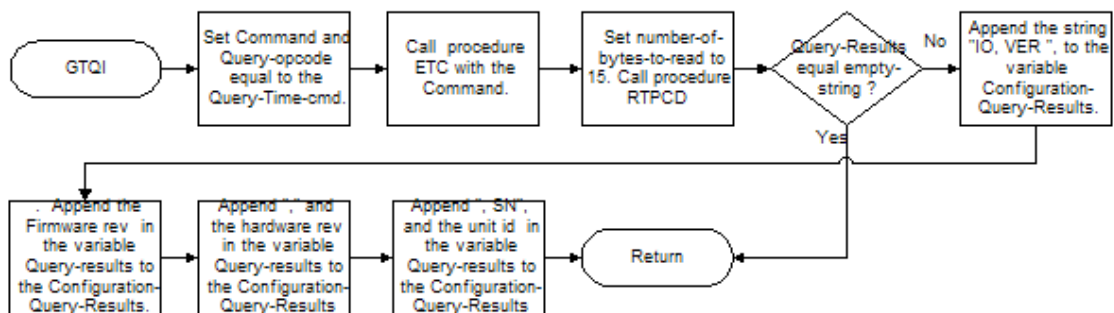
Append the string "IO, VER ", to the variable Configuration-Query-Results. Append the Firmware rev (<nn.nn>) returned in the variable Query-results to the Configuration-Query-Results.

Append a command (,) followed by the hardware rev (<hh.hh>) returned in the variable Query-results to the Configuration-Query-Results.

Append the string ", SN", followed by the unit id (<SNxxxx>) returned in the variable Query-results to the Configuration-Query-Results.

An Algorithm that depicts the procedure is demonstrated in Figure 7.20.

Figure 7.20 - Get Timer Query information Flowchart



## Get VI Generators Query Information

### 7.16 Get VI Generators Query Information (GVIGQI)

This routine will create the, /<G><position>\*<VER><nn.nn><hh.hh>< SNxxxx>, <memory map> ... <VI> / portion of the Configuration-Query-Results.

Set the variable port-address equal to the address VI Gen # 1 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable Number-of-VI-Generators-present equal to 1. Call routine 7.16.1 **Get Selected VI Generator Query Information (GSVIGQI)**.

Set the variable port-address equal to the address VI Gen # 2 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable Number-of-VI-Generators-present equal to 2. Call routine 7.16.1 **Get Selected VI Generator Query Information (GSVIGQI)**.

Set the variable port-address equal to the address VI Gen # 3 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable Number-of-VI-Generators-present equal to 3. Call routine 7.16.1 **Get Selected VI Generator Query Information (GSVIGQI)**.

Set the variable port-address equal to the address VI Gen # 4 defined in section 2.4 I/O Memory Map, of the ISA FPGA Description document. Set the variable Number-of-VI-Generators-present equal to 4. Call routine 7.16.1 **Get Selected VI Generator Query Information (GSVIGQI)**.

Append the semicolon (;) character to the Configuration-Query-Results.

#### 7.16.1 Get Selected VI Generator Query Information (GSVIGQI)

Set the variables Cmd-string and Query-opcode equal to the Link-test-cmd. Append a one-byte integer value 0, to the Cmd-string. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

Set the variable number-of-bytes-to-read to 1. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

If Query-results is not equal to the Cmd-string, then return, the generator is not present.

Otherwise, Append the string “, /G “, to the variable Configuration-Query-Results. Append the <position> number stored in the variable Number-of-VI-Generators-present to the Configuration-Query-Results.

Set the variables Cmd-string and Query-opcode equal to the Query-DSP-Firmware-Info-cmd. Call the procedure 7.1 **Send Command (SC)**, with the Cmd-string and Port-address.

Set the variable number-of-bytes-to-read to 32. Call procedure 7.11 **Read VI Generator Port Command and Data (RVIGPCD)**, return the results in variable Query-results.

Append the string “, VER “, to the variable Configuration-Query-Results. Append the Firmware rev (<nn.nn>) returned in the variable Query-results to the Configuration-Query-Results.

Append a command (,) followed by the hardware rev (<hh.hh>) returned in the variable Query-results to the Configuration-Query-Results.

Append the string “, SN“, followed by the unit id (<SNxxxx>) returned in the variable Query-results to the Configuration-Query-Results.

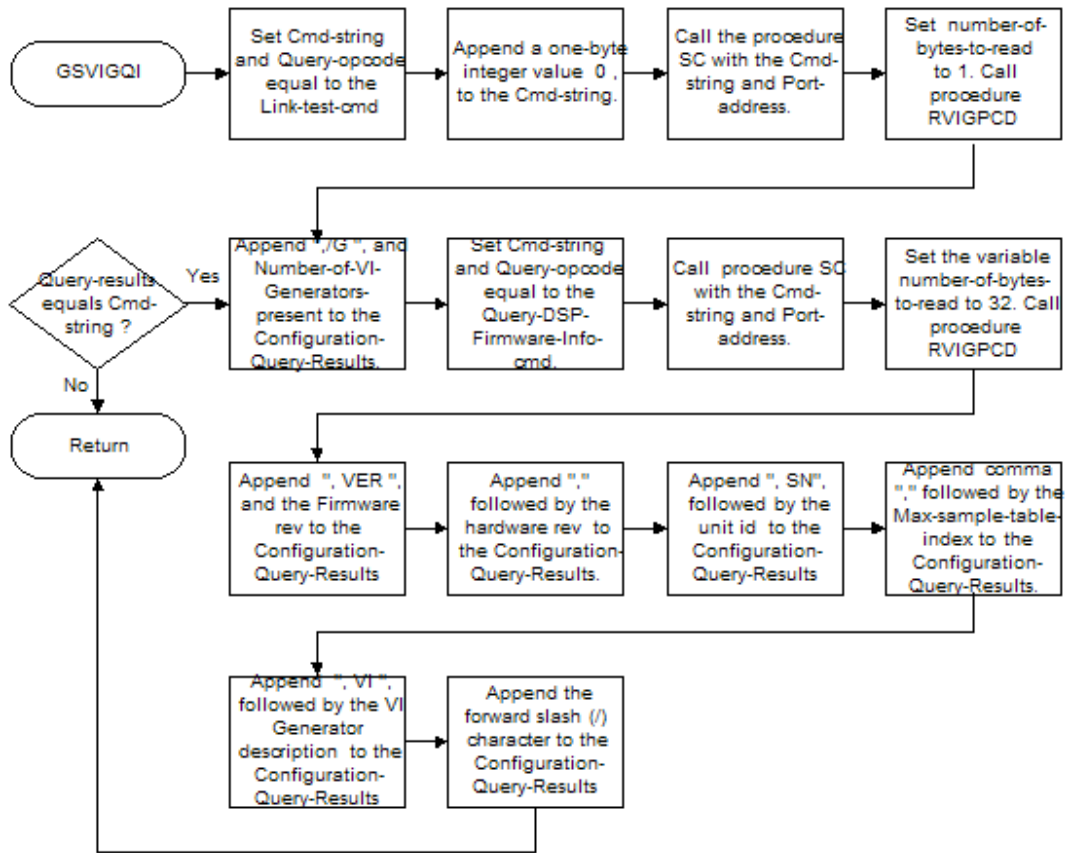
Append the comma (,) followed by the memory map stored in the variable Max-sample-table-index to the Configuration-Query-Results.

Append the string “, VI “, followed by the VI Generator description (<VI>) returned in the variable Query-results to the Configuration-Query-Results.

Append the forward slash (/) character to the Configuration-Query-Results.

An Algorithm that depicts the procedure is demonstrated in Figure 7.21.

Figure 7.21 - Get Selected VI Generator Query Information Flowchart





## MPRT Appendix A - Timer Flash Memory Parameter Sector Physical Layout

Table A-1 – Timer Flash Memory Parameter Sector Physical Layout

<b>MEMORY ADDRESS</b>	<b>FIELD DESCRIPTION</b>
0x07ff00 - 0x07ff03	Firmware Version – 4 byte floating point number
0x07ff04 - 0x07ff07	Hardware Version – 4 byte floating point number
0x07ff08 - 0x07ff1f	Serial Number – 24 Alpha-Numeric Character
0x07ff20 - 0x07ffb6	Reserved – 151 Bytes unused
0x07ffb7 - 0x07ffba	VI-Monitor Zero Current Correction Factor – 4 byte floating point
0x07ffbb - 0x07ffbe	VI-Monitor Zero Voltage Correction Factor – 4 byte floating point
0x07ffbf - 0x07ffc2	VI-Monitor Current Positive Correction Factor – 4 byte floating point number
0x07ffc3 – 0x07ffc6	VI-Monitor Volts Positive Correction Factor – 4 byte floating point number
0x07ffc7 - 0x07ffca	VI-Monitor Current Negative Correction Factor – 4 byte floating point number
0x07ffcb - 0x07ffce	VI-Monitor Volts Negative Correction Factor – 4 byte floating point number
0x07ffcf - 0x07ffe6	Vxworks License Number – 24 Alpha-Numeric Character
0x07ffe7 - 0x07fffe	Unit Serial Number – 24 Alpha-Numeric Character

## MPRT Appendix B - VI-Generator Flash Memory Parameter Sector Physical Layout

Table B-1 – VI-Generator Flash Memory Parameter Sector Physical Layout

<b>MEMORY ADDRESS</b>	<b>FIELD DESCRIPTION</b>
0x07ff00 - 0x07ff03	Firmware Version – 4 byte floating point number
0x07ff04 - 0x07ff07	Hardware Version – 4 byte floating point number
0x07ff08 - 0x07ff1f	Serial Number – 24 Alpha-Numeric Character
0x07ff20 - 0x07fff6	Reserved – 215 Bytes unused
0x07fff7 - 0x07fffa	Max Voltage Output – 4 byte floating point number
0x07fffb - 0x07fffe	Max Current Output – 4 byte floating point number

## **MPRT Appendix C - VI-Generator Correction factor Flash Sector Physical Layout**

Table C-1 – VI-Generator Correction factor Flash Sector Physical Layout



## **Appendix C - PULSAR General Information**

### **Appendix C - PULSAR General Information**

The information contained herein is taken from the PulseMaster Users Manual. Since MPRT and PULSAR can be controlled by AVTS and MPRT must operate with a test executed from an imported PulseMaster database, the information deals with the Control Syntax and the use of XMAC's.

The Syntax, Command Set, XMAC use and Examples are detailed in the following topics:

- PULSAR Error Codes

- PULSAR Command Set

- PULSAR String Commands

- XMACs

  - XMAC Library Functions

## PULSAR Error Codes

### PULSAR Error Codes

Errors are displayed from the unit and are also transmitted over RS232 using a preset Pulsar Error Message Format.

Error codes are organized into sections according to the subsystem involved with the error.

These sections are:

- SECTION 1 - GENERATOR SELECTION SECTION
- SECTION 2 - GENERATOR CONTROLLER SECTION
- SECTION 3 - GENERATOR WAVEFORM SECTION
- SECTION 4 - TIMER SECTION
- SECTION 5 - GENERAL ERRORS
- SECTION 6 - COMMUNICATION SECTION
- SECTION 7 - WAIT COMMAND ERRORS
- SECTION 8 - MACRO ERRORS
- SECTION 9 - MACRO ERRORS

Error codes are given in the form "section-subcode"; i.e., 1-02, 2-02. Generally the meaning of the subcodes changes from section to section, except in the case of syntax errors which have been assigned the following codes and report the same error regardless of section.

#### Syntax

A syntax error code is used for incoming commands that cannot be recognized by the PULSAR system. This error code can occur from the general error classification (5) or from one of the unit type classes (1, 2, 3, 4, or 6). An example of a syntax error is as follows:

The command "SI;" is entered. The error is because this command is invalid and not recognized by the PULSAR system. The command should have been "WAV1SI;"

#### X-01 SYNTAX-GENERAL SYNTAX ERROR

General or non classifiable syntax error. The unit received an unrecognizable series of characters.

#### X-97 SYNTAX-NO GENERATOR SELECTED

A command was received that requires that a current or voltage generator or some other subsystem to have been previously selected. This error can also occur if a timer post configured for auto-off sends the off command.

#### X-98 SYNTAX-BAD COMMAND

The command interpreter did not recognize the command.

#### X-99 SYNTAX-NO NUMBER

A command was entered without a value. This happens when you enter amplitude, dc, phase, or frequency, and leave off the number. "Example: v1,a;" The "a" was entered without a value from 0 to 300 volts. This generates an error



## **Section 1 - Generator Selection Section**

### **1-02 SELECTION ERROR**

A generator selection error is caused when a generator is requested that does not exist. If a system has voltage units 1 and 2 present and voltage unit 3 is requested, the generator selection error occurs. An example of a generator selection error is as follows: The command "V3," is entered. The only valid entries were "V12,", "V1,", "V2;", or "V0;", therefore a generator selection error is created.



## Section 2 - Generator Controller Section

### 2-02 AMPLITUDE VALUE ERROR

An amplitude value error occurs when an amplitude is entered outside the range of the selected generator. Suppose the range of the selected current generator is 3. If a request is made to have the amplitude set to 3.1, which is the next range category, an amplitude error occurs.

### 2-03 DC VAL ERROR

A DC value error occurs when a DC value is entered outside the range of the selected generator. Suppose the range of the selected current generator is 3. If a request is made to have the DC value set to 3.1, which is the next range category, a DC error occurs.

### 2-04 DC AMP SUM ERROR

A DC\_AMP sum error occurs when the absolute value of the DC value plus the AC value caused a range switch when the generator is on, or when the values added together exceed 300, which is the max DC + AC value.

### 2-05 DC AMP EXPONENTIAL SUM ERROR

A DC\_AMP\_EXP sum error occurs when the absolute value of the DC value plus the AC value plus the EXPONENTIAL OFFSET causes a generator to exceed the current range selected by the generator.

### 2-06 PHASE VALUE ERROR

A phase error occurs when the user enters a value greater than 999.9 or less than -999.9. If a phase value of 1000 was requested of PULSAR, then a phase value error would occur.

### 2-07 FREQ VALUE ERROR

A frequency error occurs when the user enters a value greater than 20,000Hz, or when a value of less than 1Hz is entered on the phase reference generator. If a frequency value of 20001 was requested of PULSAR, then a frequency value error would occur.

### 2-08 OFF STATE ERROR

An off state error happens when the user selects an off state greater than 3 and less than 0. The command should be entered as follows: "OFF 1;", "OFF 2;", or "OFF 3;".

### 2-09 WAVE N LOCATION ERROR

The valid wave locations are 1 through 16. If the user enters a waveform location greater than 16 a waveform location error is generated. If the command "WAV17," was requested of PULSAR, then the waveform location error would be generated by PULSAR.

### 2-10 WAVE LOCATION MEMORY ERROR

The waveform location selected has memory errors at that position. Consult the generator configuration response and look into the waveform location - 1 to see the cause of the problem.

## Section 3 - Generator Waveform Section

### 3-02 WAVE LOCATION ERROR

The valid wave locations are 1 through 16 except when specifying Data Playback data. If the user enters a waveform location greater than 16 a waveform location error is generated. This error code is used to represent the selection and execution of an existing waveform location.

The waveform location selected has memory errors at that position. Consult the generator configuration response and look into the waveform location - 1 to see the cause of the problem.

### 3-03 SINE AMP VALUE ERROR

The sine value entered cannot be greater than 1. If the user selects a sine value greater than 1 then a sine value error is generated. The sine amplitude input to PULSAR must be  $\geq 0$  and  $\leq 1$ . A command of "WAV1SI 1.1:10:10;" submitted to PULSAR would generate a sine amplitude value error would be generated, because 1.1 is greater than 1.

### 3-04 SINE AMP OVERFLOW ERROR

The sine value entered cannot be greater than 1. If the user selects a sine value of .3 and .8, when entering multiple sine commands within the same waveform location, or any combination that is greater than 1 a sine amplitude overflow error occurs. Suppose the following command was requested of PULSAR:

```
"WAV1SI.4:20:30 .5:10:10 .2:15:10,"
```

A sine amplitude overflow error would be generated at the ".2" because  $.4 + .5 + .2$  is greater than 1.

### 3-05 SINE HARMONIC VALUE ERROR

A sine harmonic error occurs when a value is enter that is less than 1 or greater than 50. If the command "WAV 5 SIN .7:51:120;" was submitted to PULSAR, a sine harmonic value error would be generated by PULSAR. This occurs because 51 is not in the range of 1 to 50.

### 3-06 SINE PHASE VALUE ERROR

A phase error occurs when the user enters a value greater than 999.9 or less than -999.9. A command of "WAV 7 SIN .2 : 50 : 1000," generates a sine phase value error. The phase value of 1000 is outside the range of -999.9 to 999.9.

### 3-07 TRIANGLE DUTY ERROR

A triangle duty error occurs when the user enters a value less than 0% or greater than 100%. A command of "WAV 1 T 101;," generates a triangle duty error because 101 is not in the range of 0 to 100%.

### 3-08 SQUARE DUTY ERROR

A triangle duty error occurs when the user enters a value less than 1% or greater than 100%. A command of "WAV 1 SQ 101;," generates a square duty error because 101 is not in the range of 0 to 100%.

### 3-09 WAVE LOCATION MEMORY ERROR

The waveform location selected has memory errors at that position. Consult the generator configuration response and look into the waveform location - 1 to see the cause of the problem.

### 3-10 EXPONENTIAL DC VALUE ERROR

An exponential DC value error occurs when a DC value is entered outside the range of the selected generator. Suppose the range of the selected current generator is 3. If a request is made to have the DC value set to 3.1, which is the next range category an exponential DC error occurs. The command string that would cause the error is "WAV 10 E 3.1:100;".

### 3-11 EXPONENTIAL TIME CONSTANT VALUE ERROR

An exponential timer constant error occurs when the user specifies a timer constant decay less than 1 or greater than 10,000 milliseconds. A command of "WAV 12 E 2.5:11000," would generate a exponential time constant value error. A value of 11,000 milliseconds is not supported by PULSAR.

### 3-12 EXPONENTIAL AUTO RANGE ERROR

An exponential auto range error occurs when the user specifies an exponential waveform request and the generator is in auto range mode. The following example would result in an error: First the command "RA 0;" is submitted to PULSAR. Next the command "WAV 1 E 3:1000;" is submitted to PULSAR. The exponential auto range error would be generated by PULSAR. If the first command would have been "RA 30;", then no exponential auto range error is generated by PULSAR.

### 3-13 EXPONENTIAL OVERFLOW ERROR

This message is received from the Sine Gen when the DC offset 'k' of the command "WAVEk:t" causes the exponential decay to go outside the current range set by the RANGE command.

## Section 4 - Timer Section

### 4-02 POST VALUE ERROR

The timer post value error is similar to the generator select error. There are only 3 timer post values (1, 2, and 3). If a selection outside this range is requested a timer post value error is generated. A selection of 0 is allowed, and selects all three post. A selection of "T4;" is invalid and generates a timer post value error.

### 4-03 DEBOUNCE VALUE ERROR

The timer debounce error occurs when the user specifies a number less than 0 or greater than 1638.3 milliseconds. A "TD2000" command generates a timer debounce value error.

### 4-04 CYCLES VALUE ERROR

The timer cycles error occurs when the user specifies a number less than 0 or greater than 20,000Hz. Any command outside the range of 0 to 20,000Hz generates a cycles value error ("TC21000" is and error).

### 4-05 TIMER SYNC TEST ERROR

This error occurs when the user does not have at least 2 generators selected, and an attempt is made to run a synchronization test.

EXAMPLE: V1,TSY;

To prevent an error the user should have entered V12,TSY;

**NOTE: The correct way to setup the test is as follows:**

**T1CAL;T1STA,TSY; This sets up Timer post 1 in the Continuous applied latched mode, configures post 1 to start, and causes the synchronized test to begin.**

### 4-06 THERMAL ERROR TIMER

A thermal overload error condition exists in the timer (battery simulator section).

### 4-07 TIMER POST ERROR

A timer post interrupt was received that was invalid. The only valid are bit 0, bit 1, and bit 3 from the MSR1 (modem status) register.

## Section 5 - General Errors

### 5-02 INPUT BUFFER FULL

The input buffer full error happens when the PULSAR external input buffer is overflowed with incoming commands. The commands requested are flushed. The input buffer could be come full if the incoming commands were being shipped to PULSAR, the generators and timer buffers became filled, and the external device still kept sending command until the main system buffer becomes full. When this occurs there is no more room to store incoming messages and it is determined that the external device has to be shut down until we are able to accept messages again.

### 5-03 INPUT BUFFER EMPTY

The input buffer empty message happens when the PULSAR external input buffer has returned to normal state, and new messages can be sent.

### 5-04 OUTPUT BUFFER FULL

The output buffer full error happens when the PULSAR output buffer overflows. This occurs when the user adds too many loop backs into the commands that can be grouped together without a comma or a semi-colon. An example of this type of command is the sine function, which is located in the generator waveform command section. The generator's input buffer becomes full while other commands are being submitted by the external device and thus this error is generated.

ERRORS 5 THROUGH 8 ARE SIMULATION ERRORS ONLY, AND NOT A PART OF THE TARGET SYSTEM.

### 5-05 FILE OPEN ERROR

The file open error occurs in the PULSAR batch mode. The input file could not be opened because the filename was incorrectly specified.

### 5-06 DEBUG FILE OPEN ERROR

The debug file open error occurs in the PULSAR batch mode. The output file could not be opened because the filename was incorrectly specified, or because there is not enough room on the disk for the file.

### 5-07 WRITE ERROR

The file write error occurs in the PULSAR batch mode and simulation modes. This error occurs because the disk is full or there is problems with the system being used.

### 5-08 READ ERROR

The file read error occurs in the PULSAR batch mode and simulation modes. This error occurs because the disk media has problems or the system being used has problems.

### 5-09 RANGE ERROR

A range error occurs when a request is made to lower the range below the DC value and the amplitude value. If a voltage unit was selected and the amplitude and DC values were 35 and 60 respectively, and a request was made to have the range be 25, a range error occurs. This is an attempt to leave the 300 volt range and move to the 30 volt range with the amplitude and DC values being in the 300 volt range.

### 5-10 DC AMP SUM ERROR

A DC\_AMP sum error occurs when the absolute value of the DC value plus the AC value caused a range switch when the generator is on, or when the values added together exceed 300, which is the max DC + AC value.

### 5-11 DC AMP EXPONENTIAL SUM ERROR

A DC\_AMP\_EXP sum error occurs when the absolute value of the DC value plus the AC value plus the EXPONENTIAL OFFSET causes a generator to exceeded the current range selected by the generator.

### 5-12 THERMAL ERROR GENERATOR AMP

### 5-13 THERMAL ERROR GENERATOR POWER SUPPLY

A thermal overload error condition exists in the generator or power supply.

#### 5-14 EXCESSIVE LOAD ERROR

An excessive load error condition exists in the generator.

#### 5-15 WAIT QUEUE FULL

This error occurs when more than 16 wait commands are presented to PULSAR without a semicolon to cause execution of the wait commands. For example, if the command string

"w1,w2,w3,w4,w5,w6,w7,a3,p3,w1,w2,w3,w4,w5,w6,w7,a20,w2,w3,p30,w4;"

was submitted to PULSAR, a wait queue full error would be generated by PULSAR.

#### 5-16 INVALID WAIT SYNC TIMER 0

#### 5-17 INVALID WAIT SYNC TIMER 1

This error occurs when PULSAR is requested to parse two wait commands of 50 microseconds each. An example of this error message being generated is as follows:

"w1,w1,a30,w2;" causes the two PULSAR timers to time out before the system can get around to processing the "w2" part of the string. Therefore, the invalid wait sync error message is generated by PULSAR.

These errors are used to identify errors to cover cases 010, 101, and other timer errors.. In the non debug system mode, this error is used to identify what state the timer was in when the error occurred.

#### 5-18 WAIT TIMER HARDWARE ERROR

This error occurs when more than four syncs are generated by the system. An example of this error is a wait sync immediate, sync phrase ref, followed by three wait 1's.

"ws,sp,w1,w1,w1;" causes more than four sync's to be generated before one can be processed. Therefore, a timer hardware error is generated.

#### 5-19 WAIT TIMER MISCELLANEOUS ERROR

State 010 or 101 was attempted to be used. These states should be used in test mode only.

#### 5-20 WAIT TIMER STATE 0 TIMEOUT ERROR

A timer timed out at state 0. We expected no timer time out.

#### 5-21 WAIT TIMER STATE 1 TIMEOUT ERROR

Timer 1 timed out in State 1. Timer 0 should have timed out.

#### 5-22 WAIT TIMER STATE 3 RELOAD ERROR

This error occurs in state 3. Timer 0 timed out before we got around to re-loading it. The system has fallen behind and the timing is off.

#### 5-23 WAIT TIMER STATE 3 TIMEOUT ERROR

In state 3 we expect timer 0 to time out, or to receive a packet. We received a timer 1 time out. The system has fallen behind.

#### 5-24 WAIT TIMER STATE 4 ERROR

In state 4 we should receive a new packet. We had a timer time out. Suspected that the system has fallen too far behind.

#### 5-25 WAIT TIMER STATE 6 ERROR

In state 6 we expect to receive a packet or to have timer 1 time out. Timer 0 timed out. The system has fallen behind.

#### 5-26 WAIT TIMER STATE 7 ERROR

In state 7 we expect timer 1 to time out or to receive a packet. We also expect to load timer 1 before it times out. The system has fallen behind and the timing is off.

#### 5-27 WAIT TIMER ERROR

We received a timer 0 time out. We expected to receive a timer 1 time out or to receive a packet.

#### 5-28 NO SYSTEM BUFFERS

This error occurs when no system buffers can be allocated. This is caused when the system is not started properly.

5-29 NO MACRO BUFFERS

This error occurs when no macro slots are available in the MACRO memory table. For example, the user request to load a MACRO program, but there is no slots available for MACRO programs. The suggested action to take is to unload 1 MACRO program to open a slot for the new MACRO program.

5-30 NO MACRO MEMORY

This error occurs when no macro buffers are available to the PULSAR system. For example, the user request to load a MACRO program, but the program will not fit in the Macro memory slot. The suggested action to take is to unload MACRO programs until space is available, or decrease the size of the MACRO program.

5-31 NOT AN EXE

This error occurs when the file requested to be loaded as a macro by PULSAR is not a ".EXE" file. PULSAR checks to see if the file meets ".exe" program specifications. To correct the problem, load a ".exe" file.

5-32 EXE TOO BIG

This error occurs when the file requested is larger than 64k. Currently Macro space is limited to 32k.

5-33 EXE NOT FOUND

This error occurs when the requested MACRO is not found in the Macro table. This error is generated when the EXECUTE, UNLOAD, or CANCEL, macro commands are used and the MACRO does not exist in the Macro table.

5-34 DUPLICATE EXE

This error occurs when the requested MACRO is duplicated in the Macro table. If the MACRO is to replace a MACRO in ROM this error will not be generated. This error is generated when the LOAD macro command is used.

5-35 MACRO SYNTAX

A syntax error was found in part of the macro command. When the Macro commands are parsed, a validity check is made to see that commas, semi-colons and spaces are in the correct positions. If the command cannot be interpreted this error is generated.

5-36 MACRO CHECKSUM ERROR

The MACRO was not loaded properly. A byte or bytes were lost during the transfer of the MACRO.

5-37 MACRO NORMAL COMPLETION

The MACRO has executed normally, and has completed

5-38 MACRO CANNOT BE CANCELED

The MACRO cannot be canceled because it is waiting on system resources. A unit reset can be performed and then a MACRO CANCEL COMMAND can be made to cancel the macro.

5-39 PHASER GENERATOR NOT PRESENT ERROR

If a generator is not in the bottom right slot and a generator is not in the top right slot (satellite box) then a phaser generator error is generated by the system. The "SP" sync phase ref command is not allowed if this error is present.

5-40 INPUT CLOCK SOURCE ERROR

The user specified an incorrect internal clock source.

5-41 TOO MANY MACRO FIXUPS ERROR

This error occurs when the macro relocation table overflows available memory. This problem can be resolved by decreasing the number of references to global data or the number of function calls.

5-80 POWER FACTOR CORRECTOR THERMAL OVERLOAD ERROR

The power factor corrector in the Input Power and Control module has reported thermal overload problems.

## Section 6 - Communication Section

### 6-02 COMMUNICATIONS INIT ERROR

### 6-20 COMMUNICATIONS INIT ERROR A

The external communications port failed during communications. The suspected problem is that communications was lost between the DSP and the System Software. The code used to report the error is dependent on the routine that detected the error.

### 6-3 COMMUNICATIONS OVERFLOW

The external communications port connected to the user has had data overwritten. This occurs when external commands are sent faster than the PULSAR system is capable of accepting and processing the commands.

### 6-04 UNIT UNAVAILABLE

### 6-07 UNIT UNAVAILABLE A

### 6-08 UNIT UNAVAILABLE B

### 6-09 UNIT UNAVAILABLE C

### 6-10 UNIT UNAVAILABLE D

### 6-11 UNIT UNAVAILABLE E

### 6-12 UNIT UNAVAILABLE F

### 6-13 UNIT UNAVAILABLE G

### 6-14 UNIT UNAVAILABLE H

### 6-15 UNIT UNAVAILABLE I

### 6-16 UNIT UNAVAILABLE J

### 6-18 UNIT UNAVAILABLE L

### 6-19 UNIT UNAVAILABLE M

These errors occur when a unit within the PULSAR system has communication problems. This error is a result of a generator or timer module going off line after being detected as present when the PULSAR system was initialized. The particular error codes indicates which section of the system software detected the error.

### 6-06 AUTO SHUT DOWN

This message is used when the timer is sent an auto off message to PULSAR. This is not an error and should not be reported as such.

### 6-22 COMMUNICATION BUSY MAIN ERROR

### 6-23 COMMUNICATION BUSY ERROR

### 6-24 COMMUNICATION BUSY A ERROR

### 6-25 COMMUNICATION BUSY B ERROR

### 6-26 COMMUNICATION BUSY C ERROR

### 6-27 COMMUNICATION BUSY D ERROR

### 6-28 COMMUNICATION BUSY E ERROR

Communications with a DSP has halted. The DSP has locked up. The particular code used depends on the routine that detected the error.

### 6-29 COMMUNICATION BYTE COUNT ERROR

Communications with a DSP experienced a byte count error.

### 6-30 BAD DSP OPCODE ERROR

A DSP received a bad op code during communications with the system board.



**6-31 BAD DSP OPERAND ERROR**

A DSP received a bad operand during communications with the system board.

**6-32 COMMUNICATION SYNC ERROR**

A DSP a sync error during communications with the system board.

**6-33 COMMUNICATION WAIT ERROR**

A DSP detected a wait error during communications with the system board.

**6-34 COMMUNICATION UNDETERMINED ERROR**

A DSP detected an whose source was undetermined during communications with the system board.

**6-35 MACRO LINE SET ERROR**

**6-36 DFR LINE SET ERROR**

**6-37 INPUT CLOCK SOURCE ERROR**

The PULSAR unit asserted SRQ to talk to the controller. The controller did not address PULSAR to talk after approximately 3 seconds. The system sends a message via RS232 and puts the error code 6-37 on the first generator.

The PULSAR unit must be reset.

**6-38 BAD DSP RAM**

An error was found in the RAM for a DSP wave location.

**6-39 EPOCH 20 DISCONNECTED**

The EPOCH 20 has been disconnected, an attempt to send a command to the disconnected unit is not allowed.

**6-83 COMMUNICATIONS RECEIVER ERROR**

The external communications port connected to the user has had data errors. This occurs when data being sent over RS-232 is garbled. Check the PC's baud rate, parity and spacing bits settings as well as the cable.

## Section 7 - Wait Command Errors

**NOTE: "w" is used to stand for "wai" to shorten the length of the demo command strings.**

### CODES

#### 7-02 INVALID WAIT POST

The timer unit within PULSAR has 3 post. If a post value of greater than 3 or less than 1 is requested with the wait command, an invalid wait post error is generated by PULSAR. An example of the invalid wait post error is as follows: "wait post 4".

#### 7-03 INVALID WAIT POST COUNT

The maximum count that can be entered to represent the number of times a post can have the same event occur is 32,768. The minimum number that can be entered is 1, which is the default value. If the post wait count is greater than 32,768 or less than 1, then an invalid wait post count error is generated. An example of this error being generated is as follows:

"wait post 1 0;" or "wait post 2 70000;".

#### 7-04 INVALID WAIT TIME

The invalid wait time error message is generated when the PULSAR unit is requested to process a wait time greater than 32,768.

#### 7-05 INVALID WAIT CYCLES

The invalid wait cycles error message is generated when the PULSAR unit is requested to process a wait time greater than 32,768.

#### 7-06 SYNC ON WAIT QUEUE NOT EMPTY

The sync on wait queue not empty error message is generated by PULSAR when a request is made for PULSAR to switch methods of using the wait timers before all of the wait commands have been emptied from the wait queue. For example, if the string

"wsi,a20,w250,d50,w1000.8,wsp;" was submitted to PULSAR a sync on wait queue not empty would be generated by PULSAR. In this example the requester attempted to change from the sync immediate mode to the sync on phase reference mode while PULSAR timers were still timing.

#### 7-15 WAIT QUEUE FULL

This error occurs when more than 16 wait commands are presented to PULSAR without a semicolon to cause execution of the wait commands. For example, if the command string

"w1,w2,w3,w4,w5,w6,w7,a3,p3,w1,w2,w3,w4,w5,w6,w7,a20,w2,w3,p30,w4;"

was submitted to PULSAR, a wait queue full error would be generated by PULSAR.

#### 7-16 WAIT QUEUE TIMER OVERFLOW ERROR

##### FROM SYSTEM SOFTWARE:

This error occurs when more than 16 wait commands are presented to PULSAR and the system is in the sync phase ref mode, which relies on the frequency and zero crossing to get sync pulses. The command are being sent faster than the system can process sync pulses at the set frequency.

Example:

c3,waimi,sp,f1,re;wai1,wai1,wai1,wai1,wai1,wai1...up to more than 17 wait commands.

## **Section 8 - Macro Errors**

This section is reserved for user defined errors to be generated by Xmacs.

## **Section 9 - Macro Errors**

This section is reserved for user defined errors to be generated by Xmacs.

## **PULSAR Command Set**

## **PULSAR Command Set**

The Command Set for PULSAR is carried through and used in MPRT.

The following topics demonstrate the Command Syntax and full details:

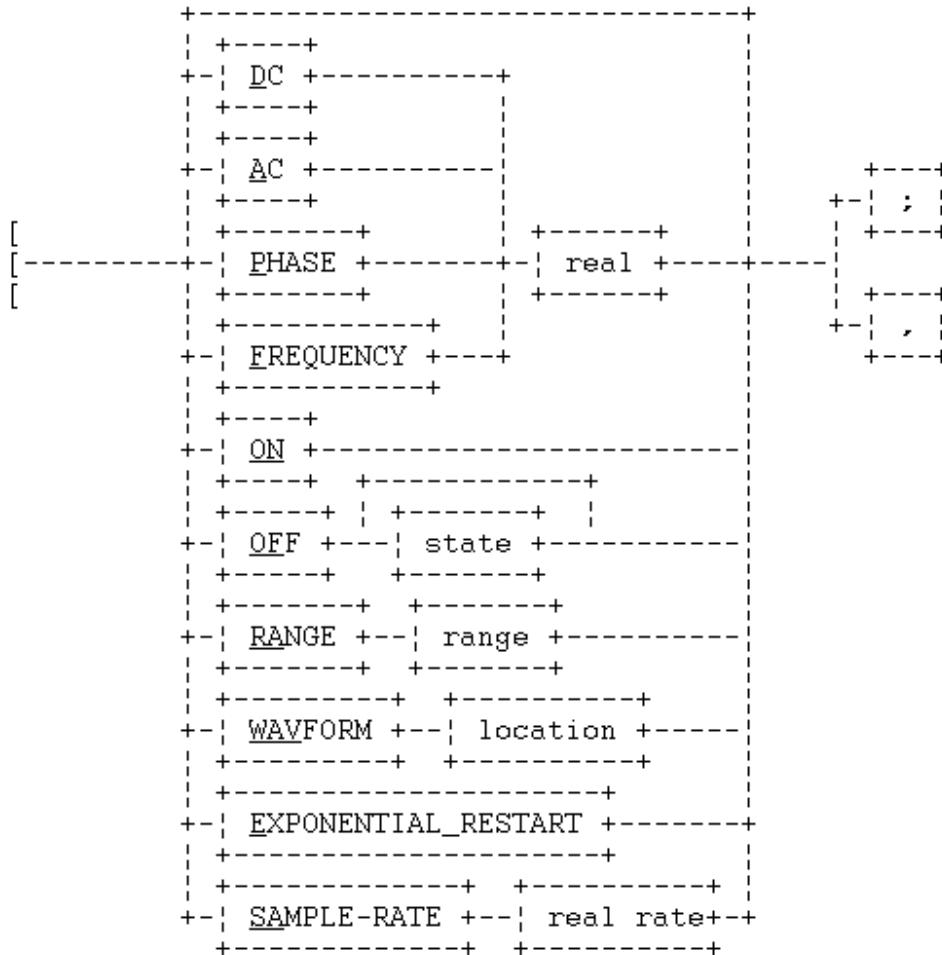
- Generator Control Command
- Generator Waveform Command
- Timer/Monitor Control
- Load Error Action Command
- Synchronization Command
- Wait Command
- Unit Reset Command
- Local Mode Command
- Restore Phase Command
- Query Command
- Macro Command



## Generator Control Command

This command is used to specify the settings of the selected generators (see the Generator Selection Command).

Syntax Diagram



where:

**D** - indicates that the amplitude of the **DC** offset voltage or current for the selected generators is to be set (in Volts or Amps).

**A** - indicates that the **AC** amplitude of the selected generator(s) is to be set (in Volts or Amps RMS).

**P** - indicates that the **phase** of the selected generator(s) is to be set (in **degrees lagging**). The range of the phase angle accepted in the command includes negative phase angles as shown below, however the generator will only display positive numbers in the range 0 to 359.9 degrees.

**F** - indicates that the **frequency** of the selected generator(s) is to be set (in hertz). Whenever the frequency of generator is changed, that generator will be out of phase until the Restore Phase Command. When the frequency is changed, the phase displayed by the altered generators will be replaced with dashes.

When the system is placed in synch. phase ref. mode (Synchronization Command) or self-zero mode the phase reference generator cannot be set below 1 hertz. A lower frequency would prevent zero crossings from occurring, and cause the system to lockup. An AC amplitude of frequency zero (F0;) is not the same

as a DC offset at that amplitude. If a DC value is desired, enter A0; and use the DC command to enter the amplitude (D2;).

**real** - indicates the value of the amplitude, phase, or frequency expressed as a **real** number in decimal form. Allowed values for this variable are:

VARIABLE	RANGE	RESOLUTION
<b>AC Voltage</b>		
range 300	0 to 300 V RMS	.02 V RMS
range 30	0 to 30 V RMS	.002 V RMS
<b>AC Current</b>		
range 30	0 to 30 A RMS	.002 A RMS
range 15	0 to 15.0 A RMS	.002 A RMS
range 3	0 to 3.0 A RMS	.002 A RMS
<b>DC Voltage</b>		
range 300	-300 to +300 V	.04 V
range 30	-30 to +30 V	.004 V
<b>DC Current</b>		
range 30	-30 to +30 A	.004 A (short duration only)
range 15	-15 to +15 A	.004 A
range 3	-3 to +3 A	.0004 A
<b>Phase</b>		
	-999.9 to +999.9	.08382 microDegree
<b>Frequency</b>		
	0 to 20000 Hertz	.00005 Hertz
synchronous phase ref or synchronous self-zero		
	1 to 20000 Hertz	.00005 Hertz

**ON** - indicates that the generator output of the selected generators should be turned **On**.

**OF** - indicates that the generator output of the selected generators should be turned **Off**.

**state** - indicates the off **state** designated by a single digit among the following:

1. generator off, low impedance. (default for voltage generators).
2. generator off, high impedance. (default for current generators).
3. generator on, output zero.

If no state is specified, the generator is set to the default off state.

**RA** - indicates that the generator **Range** is to be set for the selected voltage and/or current generators.

**range** - indicates the generators **range** which is the maximum number desired in volts or amps. The PULSAR unit will choose the lowest range which contains the input value. For example, if an input of 150 is given for a voltage generator range, the 300 volt range will be selected by the PULSAR unit. A range of zero selects AUTO RANGE mode. In this mode, the lowest range capable of handling the current AC or DC amplitude settings is automatically chosen by each generator.



0	Automatic ranging (autoranging).
High range:	
300	Voltage generator: 300 Volt.
30	Current generator: 30 Amp.
Mid range:	
15	Current generator: 15 Amp.
Low range:	
30	Voltage generator: 30 Volt.
3	Current generator: 3 Amp.

Note: If the user desires to ramp a voltage or current generator and the ramp starts in one range and eventually increases to a higher range, the user should first send a Range Command with the highest range sought for the ramp. If this is not done and the range has not been changed, as soon as an Amplitude Command is sent for a higher range, a syntax error will result. If automatic ranging is selected, ramping does not generate a syntax error when the amplitude increases to higher range.

**WAV** - indicates that the previously defined **Waveform** command is to be selected.

**location** - indicates the storage location to use. This must be a number "0" thru "32" or the number "255".

Each generator contains up to 16 locations to use, depending on the available memory. Choose location to indicate which waveform to use (as defined by the Generator Waveform Command). Choose 0 to select the first 64k data playback waveform location (Generator Waveform Command). (for Generators revision ver 2.0 (see the qc, command) use 17-32 to specify the locations added to these generators, 33 to specify the upper 64k of a data playback waveform and 255 to specify the entire 128k block of a data playback waveform.)

**E** - indicates that this is a **Exponential Restart** command. Upon execution of this command the DC offset exponential decay that was defined in the last Generator Waveform Command is executed.

Example:

F15000,P90, = set the previously selected generator(s) to 15kHz and 90 degrees (90 degrees lagging). This will be displayed as 90 degrees.

**SA** - indicates the rate at which the selected waveform is output. When the sample rate is changed, that generator may be out of sync with the output of the other generators. The restore phase command should then be used to synchronously restart the output of the generators. The phase display on the altered generator will be replaced with dashes. Also, the sample command turns the generators zero degree pulse off, which when applied to the master generator, also disables its synch. phase ref. clock (Synchronization Command). If the unit is set for sync phase ref mode then all synchronous command execution will stop until the unit is set to another mode.

**real rate** - indicates a value from 0 to 20,000hz with .00005 resolution. A single period waveform filling a 4096 sample block running under a 20Khz sample clock would create about a 4.88hz waveform.

## Generator Waveform Command

This command defines the type of waveform and the DC offset exponential decay for the selected generator(s) (see the Generator Selection Command). Allowed waveforms are square wave, triangle wave, half-wave sine waves, and analytical sine waves. The default sine wave is a first harmonic sine wave with no phase shift.

Analytical waveforms allow the user to create periodic waveforms with a selectable harmonic content. The following function describes the analytical waveform:

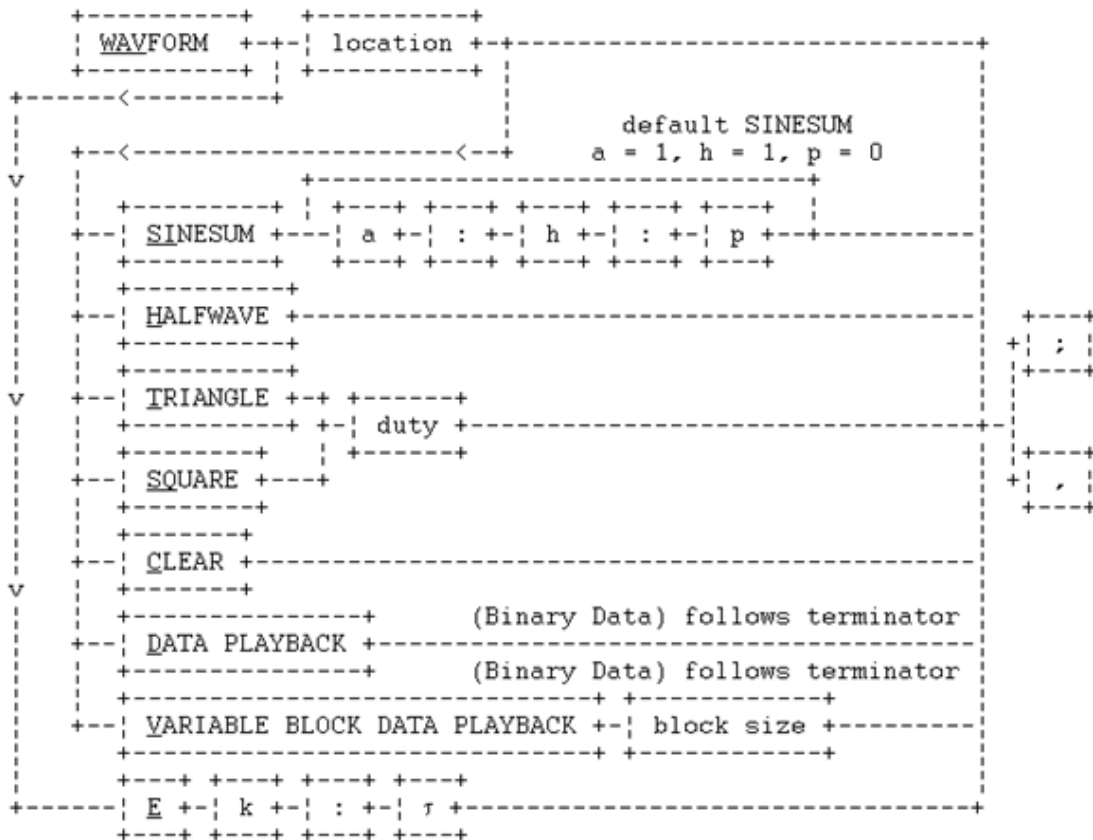
$$a1 \times \sin(h1 \times 360ft + p1) + a2 \times \sin(h2 \times 360ft + p2) + a3 \times \sin(h3 \times 360ft + p3)$$

where f is the frequency as defined in the Generator Control Command.

The DC offset exponential decay modifies the defined waveform by adding an exponentially decaying DC offset in the form of

$$k \exp(-\tau t).$$

Syntax Diagram



where:

**WAV** - indicates that this is a **Waveform** command.

**location** - indicates the storage location to be defined. Except when specifying Data-Playback, this must be a number "1" to "16". Each generator, depending on the memory configuration, contains up to sixteen waveform storage locations which can be independently defined. When specifying Data-Playback, "1" thru "16" indicates a 4096 word waveform. A "0" can only be used with Data-Playback and indicates a 65536 word waveform. (for Generators revision version 2.0 (see the qc, command) use "17" through "32" to specify

the 16 locations added to these generators. And, when specifying Data-Playback, use the location number "33" to specify the new, upper 64k block and "255" to specify the full 128k block.)

**SI** - indicates that a Sinusoidal analytical waveform is to be summed into the existing waveform in the specified location.

Example:

WAV1SI .5:1:0; = specify a location 1 sinusoidal waveform given by the function  $.5 \sin 2 \pi t$ .

**a** - indicates the **amplitude** of the harmonic as a fraction of the amplitude set in the Generator Control Command (a real number in decimal form, in the range of 0 to 1). A non zero positive number not equal to zero.

Note: Each amplitude specification in the analytical specification is defined as fractions (in the range of 0 to 1) of the amplitude set in the Generator Control Command. The vector sum of each component at the waveforms maximum point must not exceed 1.0. (note on a note the same harmonic can be sent more than once, adding new data to the waveform each time while adding to that harmonic as a vector (adding 180 degrees out of phase is subtracting).)

**h** - indicates the **harmonic** of the frequency set in the Generator Control Command (an integer in the range of 1 to 50).

**p** - indicates the **phase** of the harmonic in leading degrees (a real number in the range of -999.9 to +999.9).

**H** - indicates a **half-wave** sine wave. Adds the positive portions of a sine wave to the data in the selected wave location. The wave location clear command is automatically issued first to remove any existing waveforms

**T** - indicates a **triangular** waveform. Adds a triangle wave with the specified duty cycle to the data in the selected wave location. The wave location clear command is automatically issued first to remove any existing waveforms

**SQ** - indicates a **square** waveform. Adds a square wave with the specified duty cycle to the data in the selected wave location. The wave location clear command is automatically issued first to remove any existing waveforms

**duty** - indicates the **duty cycle** of the triangle and square waves (an integer number from 1 to 100 percent ).

**C** - indicates that the selected location should be **cleared**. Each word in the wave location is set to zero. This should be done before defining a new waveform with the **sinesum** command.

**E** - indicates that the **exponential decay** parameters (for the formula  $k \exp(-\pi t)$ ) follow.

Warning: Do not change range after the parameters of the exponential decay have been sent. If the range is to be changed, even if by auto-range, the parameters for the exponential decay will need to be resent.

**k** - indicates the value of the DC offset of the exponential decay at time zero; a real number in the range -FS to +FS of current range. This value is added to the DC and AC values set by the Generator Control command.

Note: Autorange does not check if the resulting waveform will fit with the current range.

Warning: Do not change range after the parameters of the exponential decay have been sent. If the range is to be changed, even if by auto-range, the parameters for the exponential decay will need to be resent.

**t** - indicates the **Time Constant** of the exponential decay in milliseconds. A real number in the range 1 to 10,000.

**D** - indicates Data Playback

(BINARY DATA) indicates binary data in hex format (COMTRADE binary) with the least significant byte of each word sent first. This data is 4096 words (16 bits) in length, 65536 or 131,072 words in length and must immediately follow, non-stop, the next comma or semicolon.

Note: when using GPIB to transmit the binary data it must be transmitted in blocks no larger than 256 bytes.

**V** - indicates Variable Block Data Playback. **For use on Pulsars with system board EPROMs version 1.40, or later, only.** Allows a portion of a wave location to be downloaded. The remainder of the wave location is not changed and may need to be cleared to prevent spurious data from being output.

Example -

"WAV17C,WAV17V2048," followed by 4096 bytes of data.

**block size** - indicates the block size in terms of samples or words. The maximum block size depends on the size of the wave location being loaded. A comma or semicolon must follow the block size and then be followed immediately by N bytes of data where N is twice the block size.

**Note: when using GPIB to transmit the binary data it must be transmitted in blocks no larger than 256 bytes.**



**Note: Post 0 is pre-defined to mean all posts, i.e., posts 1, 2, and 3.**

**V** - indicates that the selected input binding post(s) are placed in the **volt** mode.

**C** - indicates that the selected input binding post(s) are placed in the **continuity** mode. In this mode the post(s) sense continuity only.

**A** - indicates that the selected input binding post(s) are to be active when the input entity is **applied**. Once the timer has started this state cannot be changed until the timer is reset

**R** - indicates that the selected input binding post(s) are to be active when the input entity is **removed**.

**C** - indicates that the selected input binding post(s) are to be active whenever the input entity changes from the **current state**. The **current state** is the state of the input when this command is executed

**L** - indicates that activity on the selected input binding post(s) is to be **latched**, i.e., activity beyond the initial activity will be ignored, unless a command is resubmitted to configure the post(s), i.e. "T2CAL," or the timer reset command is reissued.

**U** - indicates that activity on the selected input binding post(s) is to be **unlatched**, i.e., changes of state are recognized by the post(s).

**Note: If a binding post is configured as timer start then it will not allow the post to be reconfigured while the timer is running.**

**HE** - indicates that the **horn enable** mode is set for the specified binding post. When continuity, or voltage depending on the mode, is applied to the selected post the timer horn will sound until the activity is removed. The horn cannot be configured to be active when the input-entity is in the removed state.

**HD** - indicates that the **horn disable** mode is set for the specified binding post. When valid activity occurs that would cause the timer horn to sound, the timer horn will remain off in this mode.

**AE** - indicates that the **auto-off** mode is set, and that valid activity on the specified binding post will cause the outputs of all generators to be placed in their respective off states. This operation is only performed if the post is configured to be a stop gate.

**AD** - indicates that the **auto-off** mode is reset, and that valid activity on the specified binding post will not cause the outputs of all generators to be placed in their respective off states. The generators must be turned off via the off command.

**STA** - indicates that the **timer will start**, when valid activity on the specified binding post(s) occurs.

**STO** - indicates that the **timer will stop**, when valid activity on the specified binding post(s) occurs while the timer is running.

**M** - indicates the selected post(s) are configured for **monitoring** operation.

**I** - indicates that all input binding post activity for the selected post(s) will be **ignored**.

**D** - indicates the **debounce** time to all input binding posts is to be set. An uninterrupted signal must be present for the specified time duration for the signal to become true.

**NOTE: Valid activity on the timer start and timer stop binding post(s) will cause the timer to compensate for the delay introduced by the debounce logic by adding the debounce time to the currently held time. The timer software start and stop commands are not affected or compensated by the debounce time.**

**milliseconds** - is the debounce time in **milliseconds**. It is a decimal number with a resolution of 0.1 milliseconds. The maximum debounce time is 1.6383 seconds (1638.3 milliseconds).

**CY** - indicates that the output of the timer is in **cycles**, so that the timer's time is measured by the number of elapsed cycles. I.e., a multiplication of time and frequency. The period will be known from the frequency parameter which is required in this command. If no frequency is specified, the previous specified frequency is used. If no frequency is specified and no frequency was previously specified, the jumper selected frequency is used.

**frequency** - is the **frequency** in hertz, a decimal number. See the Generator Control Command for the frequency range and resolution (integer 1-20,000).

**SE** - means that the time that is output by the timer is in **seconds**. The maximum range in seconds is 107374.1824.

**R** - indicates that the timer's time is **reset** to zero and all input binding posts are initialized to their most recent setting and any latches are reset. For post that are configured change of states, the current state becomes the state must change from to go active.

**SY** - indicates that the timer is to start a Sync-Test. This command is executed immediately. The user should first setup the box to operate in Synch Immediate mode (Synchronization Command). The user should then set the frequencies of the two generators to be used in the Sync-Test, set their phase angles to 0, make sure to reselect both generators and then execute this command. This command will then execute restore phase on the selected generators, turn their outputs on and then wait the amount of time necessary for the outputs to **synchronize** (reach the same phase angle). The timer should be started by a contact closure from the device under test, and will be stopped when the two selected generators become **synchronized**. At the end of the test, the generator outputs will be in their off state, and the timer's display will show the sync time with an error of 100 microseconds for every six seconds the test takes to run. The time it takes for a test to run is given by  $1/|f1-f2|$  where f1 and f2 are the frequencies of each generator. Any commands sent while the command is running should not change the generator selection or interfere with a timing event.

**Note:** Since this command executes immediately, all command execution should have completed before the command is sent.

Example:

```
v12,sp,a120,p0,v1,F60,v2,F60.1,v12;
```

pause for at least one cycle to allow commands to execute (PulseMaster might need more time, like 120ms, because RS-232 transmission are done in the background and two output commands sent to close together will be sent as one). TSY; = setup generators and perform a Sync-Test.

**AD - disables** (opens) the auxiliary contact.

**AE - enables** (closes) the auxiliary contact.

**SO** - indicates that the beginning timer software start or stop command.

**STA - start** the timer via software at the next semicolon if preceded by an SO command.

**STO - stop** the timer via software at the next semicolon if preceded by an SO command.

Example (two commands):

```
T1VRU,T1STO; = stop the timer when voltage is removed from the front panel unlatched binding post 1.
```

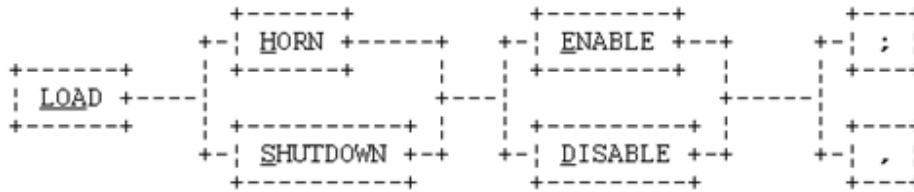
Example (multiple commands):

```
T1VAL,T2VRU;T1STA,T2STO; = The timer will begin timing when an voltage is applied to latched front panel binding post 1. Since the start condition is latched, it will continue timing without regard to the state of post 1. The timer will stop when a voltage is removed from post 2. Since post 2 is unlatched, the timer will resume timing if a voltage is applied to post 2.
```

## Load Error Action Command

This command allows the user to specify the PULSAR's reaction to a generator load error.

Syntax Diagram



where:

**LOA** - indicates the **load** command.

**H** - indicates whether the **horn** will sound when a generator load error occurs.

**S** - indicates whether the generator is to be **shut down** when encountering a load error.

**E** - indicates that the type activity is **enabled**.

**D** - indicates that the type activity is **disabled**.

Example:

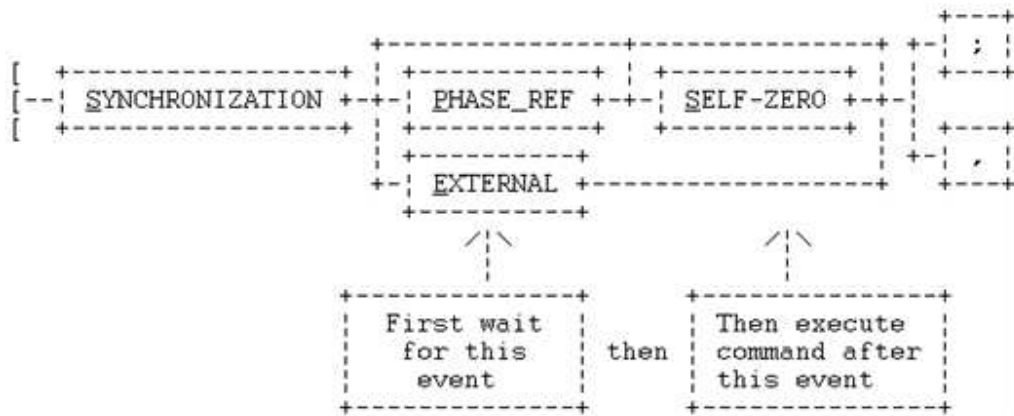
LHE; = enable horn to sound when a generator load error occurs, and the selected generator's display will flash.



## Synchronization Command

This command specifies when commands should be executed. This command should not be sent before a wait command has finished execution.

Syntax Diagram



where:

**S** - indicates that this is the start of a **synchronization** command.

**SP** - indicates that execution of subsequent commands will be synchronized by the **master generator's** phase reference clock. The generator located in the bottom right-most slot is the **master generator**. In this mode, the PULSAR unit's command synchronization is controlled by the synch pulses generated as the master generator's phase angle clock passes through zero degrees. If the master generator does not exist then the unit will switch to **synch immediate** mode. This is the default synch selection after power on and is not affected by the generator's phase angle but is affected by its frequency. If the master generator is running under the Sample Rate command (Generator Control Command) then the phase angle clock will be disabled and SP will not work.

Note: This command goes into affect as soon as the next comma or semicolon is received, it does not wait for a command synchronization signal.

**SS** - indicates that subsequent commands will be executed synchronous with each generator's own positive going zero crossing of the AC signal (**self zero** crossing). See note below.

Note: This command goes into affect as soon as the next comma or semicolon is received, it does not wait for a command synchronization signal.

**S;** - indicates that subsequent commands will be executed immediately in **synch immediate** mode where the synchronization signal is the wait timer's 50us clock.

Note: This command goes into affect as soon as the next comma or semicolon is received, it does not wait for a command synchronization signal.

**SPS** - indicates that subsequent commands will be executed at the first **zero crossing** after the **master generator's** phase reference passes through zero degrees.

Note: This command goes into affect as soon as the next comma or semicolon is received, it does not wait for a command synchronization signal.

**SE** - indicates that the sync pulse will be generated by an **external** source connected to the back of the system. This is provided to allow the adding of generator modules to the system.

Note: This command goes into affect as soon as the next comma or semicolon is received, it does not wait for a command synchronization signal.

Warning: If the user switches from **synch external** to another synch mode before the synch pulses necessary to execute the semicolons and wait cycle commands loaded in the system have been received, then command execution will forever be behind the same number of synchronized events that were outstanding when the external mode was left.

Example: The following commands are sent with nothing connected to the external synch inputs

SE,v0,a1;a2;SI,a3;a4;SP,a5;a6;

The voltage generators will go to 1 volt as soon as the semicolon sent with a3 is received, to 2 volts as soon as the semicolon from a4 is received, to 3 volts upon receiving a synch pulse from the master generator after the semicolon for a5 is received and to 4 volts upon receiving a synch pulse from the master generator after the semicolon for a6 is received. The commands for a5 and a6 will remain loaded until the next synch event is received or the PULSAR unit is reset.

Note: The synchronization source (immediate, phase ref, external) cannot be changed while wait cycle timing events are going on (Ex. waic10;).

Example:

SP; = execute commands at the left-most generator's phase reference.

Note: Sometimes the synchronization mode in use can produce some unexpected results especially when using self zero synchronization. For example the command string "v0c0,s;p0;ss;a1,v1,p0,v2,p120,v3,p240,v0,on;" will cause all generators to come on at the same time at the prescribed phase angles. Whereas, the command string "v0c0,s;p0;ss;v1,p0,v2,p120,v3,p240;v0c0,a1,v0,on;" will cause each voltage generator to come on 120 degrees apart starting with the first generator that generates a positive going zero crossing once the last sequence is received.

In the first command string the on command is executed at the same time because each generator's phase angle is set to zero when the last command sequence is executed. In the second command string the phase angles are set first and then the on command sequence is executed by each voltage generator once the semicolon has been received and that generator produces a positive going zero crossing.

Some other unexpected results are encountered when the restore phase command is used in synch self zero modes. When self zero synchronization is active and a generator is sent the restore phase command, the generator's phase reference is reset on the next positive going zero crossing. If the generator's phase angle is not zero or the current waveform has positive going zero crossings that do not occur at zero degrees then the generator's phase reference will be shifted by the restore phase command.

The restore phase command may also produce unexpected results in synch phase ref. mode and synch immediate when applied to the master generator. In synch phase ref. mode if the command "v0c0,wai2,re;" is issued then each generator's phase reference will be shifted 2 msec and subsequent command synchronization will also be shifted 2msec.

Note: Normally an error is generated when a command to set the synch mode is received while a wait timer is loaded. i.e. when the command string,

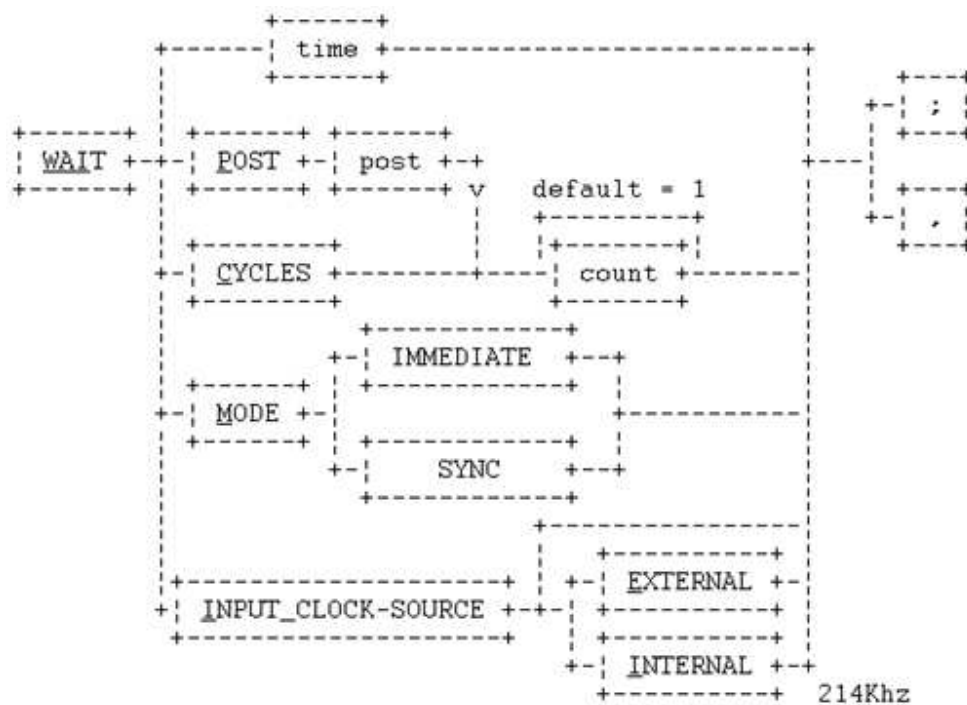
"a1,waic1,a2,waic2,a3,S,wai4;"

is sent. However, it is possible and it is sometimes desirable to change the synch mode while a wait command is in progress. The problem with this, however, is that if the synch mode is being changed as the wait command is expiring it could hang up the wait timer circuitry.

## Wait Command

This command provides a time delay before the next command is executed.

Syntax Diagram



where:

**WAI** - indicates that this is the **wait** command.

**time** - indicates to wait the specified period of **time** in milliseconds (0.3 to 3276.7). This command was designed to work primarily in **Synch Immediate** and **Self Zero** mode. In all other none external synch modes the timer first times out and then must wait for a zero degree pulse from the master generator before it signals all generators to continue command execution. That is, in all other modes the accuracy drops to the frequency of the master generator.

**Pitfall Note:** commands strings like "a0,wai2;a1,wai2;a2,wai2;" must also wait for each semicolon to execute. The command string "v1,waimi,s,waic1,d1,waic1,wai1,d0,waic1,d2;" would wait 0 to 16.666msec waiting for the zero degrees in the first cycle, 17.666msec waiting for the second cycle and the 1msec time and then 15.666msec waiting for the last cycle pulse to be generated. In **wait mode synch** "waic1,wai1"

**P** - indicates timer post events. Wait for the timer post to enter the inactive state.

**post** - indicates the specified timer **post** (1 to 3).

**count**- indicates the number of post events required before generators will resume command execution. If the unit is not in **Synch Immediate** or **Self Zero** mode then the command will also wait for a synch pulse from the Master Generator after the count has expired before allowing the generators to continue.

**Note:** To turn the generators off after a timer post event, enter WAIP1,V0C0,OFF; The first timer post event will start the sequence to turn the generators off.

**C** - indicates wait until a cycle count is complete. This is a count of the master generator's phase ref pulse.

**count**- is the positive whole number with a range of 1 to 65535. The default count is 1.

**Note:** When in **wait mode immediate** and **synch phase ref** or **synch immediate mode**, "waimi,sp;" or "waimi,s," or wait mode synch and synch immediate mode "waims,s," that the minimum allowed count is 2 if the wait starts asynchronously (can be loading while the first count comes in). Counts of 1

are allowed if it must wait for another wait cycle to finish. after other "waic1," is interpreted as a "waic2," command. In **wait mode synch** and **synch phase ref**, "waic1," mode a "waic1," always loads after the first count is received.

Pitfall Note: commands strings like "a0,waic2;a1,waic2;a2,waic2;" must also wait for sync pulses at each semicolon in **synch phase ref**. In **wait mode immediate** the first wait cycle in a series of waits will start asynchronously and produce a timing between (n-1) to n times period of the master generator(Ex. "waic2 ,waic2,;" gives a 16.6msec to 33.3msec wait followed by a 33.3msec wait ).

**M** - indicates the mode of synchronization for the wait command.

**MI** - indicates that timing or counting will begin immediately after the previous timing or counting event has completed. This command takes effect immediately and cannot be sent while a wait command is waiting.

**MS** - indicates that timing will begin on the next master generator's synch signal after the previous timing or counting event has completed. This command takes effect immediately and cannot be sent while a wait command is waiting. This mode is useful in forcing the next wait event to start when the synch pulse is generated and not as soon as the previous wait event timed out.

Note: the synch signal in **synch immediate** or **synch self-zero** is the wait timers 50usec clock. The synch signal in **synch phase ref** and **synch phase ref self zero** is the master generators zero degree pulse.

Pitfall Note: Because the same pulse cannot cause a Wait Cycle time out and synch the next wait events start, Wait Cycle commands will wait an extra cycle when the **synch signal** is the master generators zero degree pulse.

**I** - indicates the input source for the generator's synchronization of wait timer events.

**II** - indicates that the clock source is internal (system generated 214kHz). This is the default entry.

**IE** - indicates that the clock source is external (user defined).

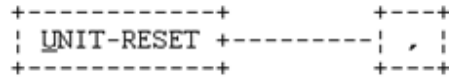
Example:

WAI16.6; = wait 16.6 milliseconds.

## Unit Reset Command

This command resets the unit to its power-up state.

Syntax Diagram



where:

**U** - indicates a **unit reset** command which resets the unit to its power-up state. This command executes immediately upon receipt and does not wait for synchronization command.

Example:

U; = reset unit to its power-up state.

## Local Mode Command

If the PULSAR unit's front panel has been disabled because an attached controller has been communicating with the PULSAR unit via RS-232 or GPIB, the Local Mode Command reenables front panel operation.

Except for enabling or disabling the front panel, the "state" of the PULSAR unit does not change when going from remote mode to local mode or from local mode to remote mode. This command is executed immediately and does not wait for synchronization. If GPIB is in use then this command will be executed upon receiving the GPIB GTL (Go To Local) interface message.

Syntax Diagram

```
+-----+ +-----+
| LOCAL +--| ; |
+-----+ +-----+
```

where:

**LOC** - indicates that this is the Local Mode Command.

**Note:** The change to local mode cannot be made while timing events are going on.

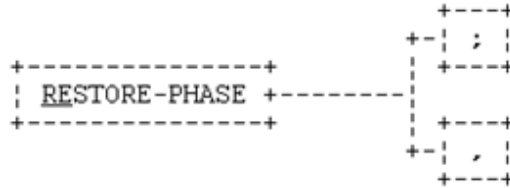
Example:

LOCAL ; = reenables front panel operation.

## Restore Phase Command

This command restores the phase relationship of all selected generators (see the Generator Selection Command). When the frequency of a generator is changed, its phase relationship to the other generators will be unpredictable. To indicate this condition, the generator whose frequency is changed will show dashes at its phase display. The phase display will be restored when a Restore Phase Command is executed.

Syntax Diagram



where:

**RE** - indicates that this is the **restore phase** command which restores the phase relationship of all selected generators.

**Note:** When this command is executed, each of the selected generators immediately jumps to its current phase setting.. If commands are being executed in immediate mode the phase will be restored immediately. For a generator to remain in phase with the master generator in all synchronous modes, except self-zero, the master generator must also be selected to be restored. If the phase of the master generator is not to be changed by the restore phase command then the master generator must be phase restored in synchronous phase reference mode.

Example:

v0c0,sp;RESTORE ; = restore the phase relationship of all generators to the master generator.

v1,sp;re; = restore the phase of v1 with that of the master generator presumed to be c3.

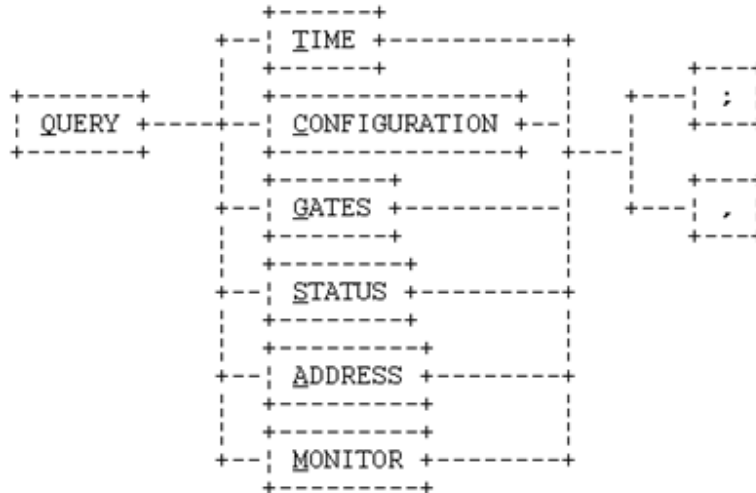
v12,s;v1,re,v2,wai5,re; = phase of v1 and v2 will be 5msec apart;

v1ss;v1,p30,re; = when v1 reaches 0 degrees switch to 30 degrees (execute RE in sync self-zero).

## Query Command

This command asks the PULSAR unit to transmit information back to the controller. This command is executed immediately upon receipt and does not wait for synchronization command.

Syntax Diagram



where:

**Q** - indicates the **query** command.

**T** - requests the timer's **time**.

**C** - requests **configuration**.

**G** - requests the state of the timer **gates**.

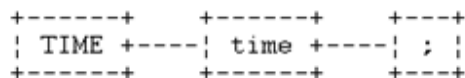
**M** - requests the state of the timer **gates** in binary.

**S** - requests the wait **status** that is currently processing.

**A** - requests the bit encoding of dip switches SW2 on the system board.

### Query Time

For the **time** request, the PULSAR unit responds as follows.



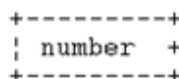
where:

TIME - indicates that this is the **time** response.

time - indicates the **time** in seconds.

### Query address

For responses to **query address** PULSAR will respond with a single number indicating the GPIB address as selected on the system board.



number - indicates the bit encoding of dip switch SW2 on the system board.

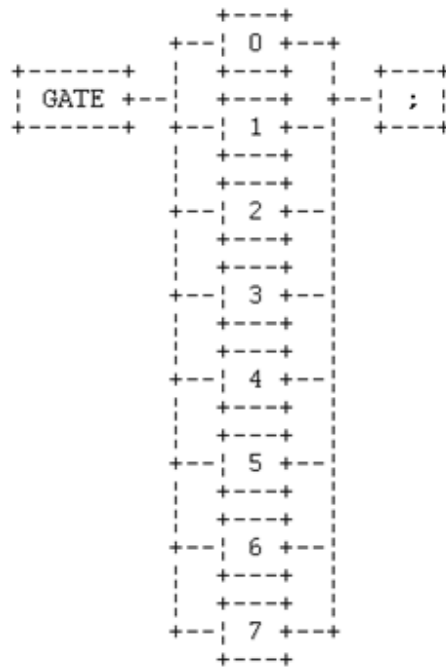
bit 1 bit2 num Baud rate for COM port Switch On = 0, Switch Off=1



0	0	0		19.2 Kbaud	
1	0	1		9600 baud	
0	2	2		4800 baud	
1	2	3		2400 baud	
bit 3 default system frequency					
0	60Hz				
4	50Hz				
bit 4	bit 5	bit 6	bit 7	num	GPIB Address
0	0	0	0	0	Disable GPIB communications
8	0	0	0	8	GPIB address 1
etc...					
8	16	32	64	120	GPIB address 15

**Query Gate**

For the **timer gate** request, the PULSAR unit responds as follows.



where:

GATE - indicates that this is the timer gate response for binding posts 1, 2, and 3. In the following table true indicates that the post is active, and false indicates that the post is not active. Note that the input required to make a binding post active when configured for applied is opposite that of remove.

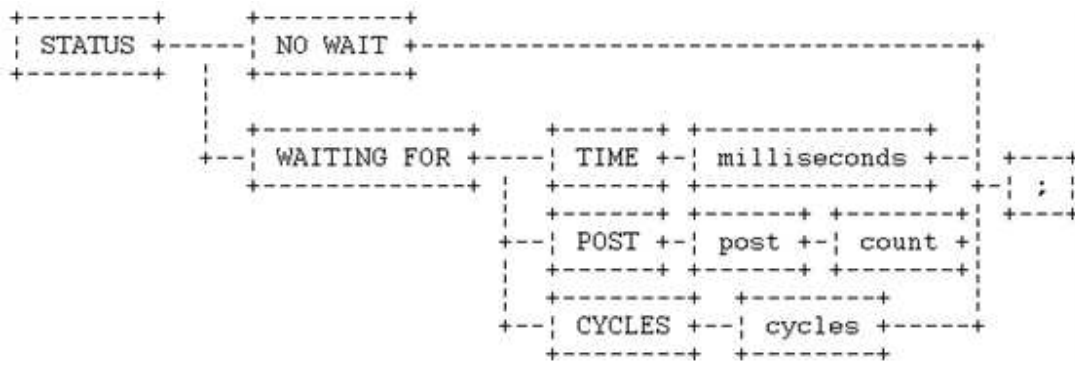
digit sent	binding post 3	binding post 2	binding post 1
0 -	false	false	false
1 -	false	false	true
2 -	false	true	false
3 -	false	true	true
4 -	true	false	false
5 -	true	false	true
6 -	true	true	false
7 -	true	true	true

### Query Monitor

Is similar to **Query Gate** except that it returns one byte that is a bit encode binary number between 0 and 7 set according to Query Gates table.

### Query Status

For the request of the wait status that is currently being processed, the PULSAR unit responds as follows.



where:

**STATUS** - indicates that this is the response to the query for the **wait status** that is currently processing. The status option of the Query Command (for which this is a response) is used for diagnosing user software problems. If system operation appears to be suspended, this may provide a clue to the cause.

**NO WAIT** - indicates that when the status option of the Query Command was processed by the PULSAR unit, there were **no wait** commands currently being processed.

**WAITING FOR**- indicates that when the status option of the Query Command was processed by the PULSAR unit, the PULSAR unit's cpu board was currently **waiting for** the completion of the wait command indicated in the remainder of this command.

**TIME** - indicates that the **time** option of the wait command was in progress.

**milliseconds**- indicates the number of **milliseconds** specified by the user in the time option.

**POST** - indicates that the **post** option of the wait command was in progress.

**post** - indicates that the **post number** option of the wait command was in progress.

**count** - indicates that the **event count** option of the wait command for the selected post was in progress.

**CYCLES** - indicates that the **cycles** option of the wait command was in progress.

**cycles** - indicates the **number of cycles** in the cycles options.

### Query Configuration

For the configuration request, the PULSAR unit responds as follows.

```

+-----+
| VG +| vv +| , CG +| cc +| TT +| tt +| , VER +| MM,DD,YY+| nn.nn +--+
+-----+
|
| +-----+
| +-| , ++ , +| T +| * VER+| nn.nn +| hh,hh +| ,/ ++
| +-----+
+-----+
| +-----+
| +-| C +| Position+| * ver+| nn.nn+| hh.hh+| , +| memory map+| , +-+
| +-----+
+-----+
| +-----+
| +-| V +| Position+| * VER+| nn.nn+| , +| memory map+| , +-----+
| +-----+

```

where:

**VG** - indicates that the number of **voltage generators** installed in the PULSAR unit will be indicated by the subsequent number.

**vv** - is the number of **voltage** generators.

**CG** - indicates that the number of **current generators** installed in the PULSAR unit will be indicated by the subsequent number.

**cc** - is the number of **current** generators.

**TT** - indicates that a **timer** unit installed.

**tt** - indicates the **number** of timer units installed. Currently only one timer unit can be installed.

**MM.DD.YY** - indicates the month, day and year of the system software.

**VER** - indicates the **version number** of the software in the PULSAR unit for the system software, the timer software, and the generator software within the PULSAR system.

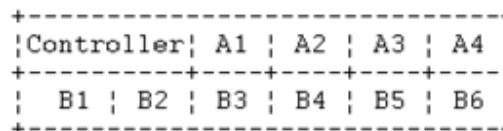
**nn.nn** - is the software version **number** of the generator module.

**hh.hh** - is the hardware version number of the generator module.

**V** - indicates that the **voltage generator** description will follow in the subsequent strings for the number of voltage generators (...vv).

**C** - indicates that the **current generator** description will follow in the subsequent strings for the number of voltage generators (...cc).

**Position** - indicates the location that the generator resides in the PULSAR unit. The positions start at the top with the first slot after the controller being A1. The highest and right most slot is A4. The bottom left most slot is B1 and the right most slot is B6. Current generators always take up the even slots (B2, B4, & B6). The bottom right most slot (B6) hold the systems master generator (Synchronization Command).



**T** - indicates that the **timer unit** description will follow in the subsequent string.

**memory map** - is used to pass the number of **memory locations** that can be used in conjunction with the Generator Waveform Commands location.

, - is a **delimiter** between the parameters that comprise this message ending in semicolon.

Note: The comma in the syntax diagram above has a typical delimiter function rather than the previously defined terminator function. Since this comma is not interpreted within the PULSAR unit's software, its use here is not an inconsistency of the PULSAR unit.

/ - is a **delimiter** between the description of each main module within the PULSAR system (voltage generators, current generators, and the timer unit).

\* - is a **delimiter** between the type of unit within the PULSAR system and the version descriptor.

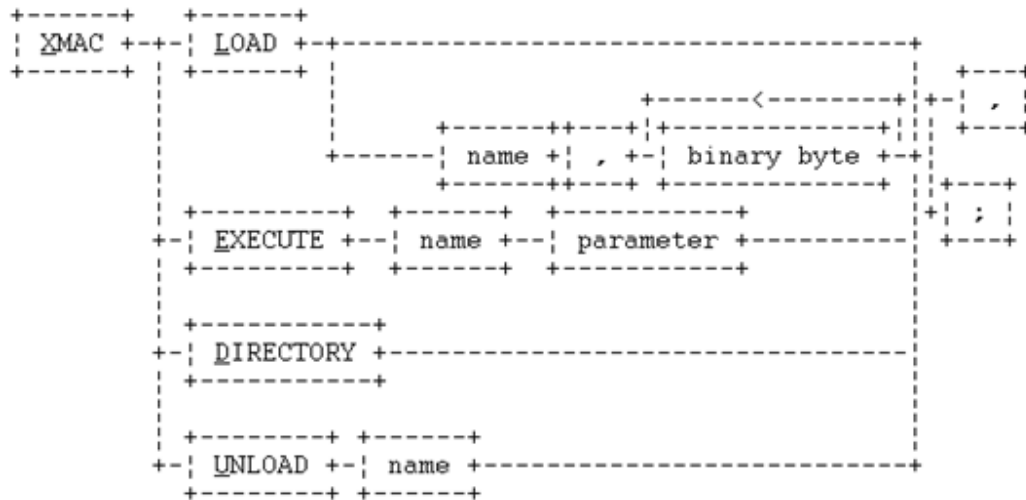
## Macro Command

### XMAC COMMANDS

These command allows programs in ".exe" format that have been linked with the library module, supplied by AVO Multi-Amp, to be managed by PULSAR.

See Also: Guide to XMACS

Syntax Diagram



**X** - Indicates that this is a **Xmac** command.

**L** - Indicates that a **Xmac** program in ".exe" format is to be **loaded** by PULSAR. See also; Downloading a PULSAR Xmac.

**E** - Indicates that a **Xmac** program in ".exe" format, previously loaded by PULSAR, is to be **executed**. The execution may be prematurely terminated if the unit receives the character '%' on RS232 or GPIB.

**D** - Requests that PULSAR output a list of the currently loaded Xmacs. Each item in the list is terminated by a CR LF and the list is terminated with the message "6-00 Transfer Complete".

**U** - Indicates that a previously loaded Xmac program is to be **unloaded** by PULSAR, and that the program space should be freed for reuse. If a macro is executing, it will not be unloaded until execution of the macro has completed. The execution may be prematurely terminated if the unit receives the character '%' on RS232 or GPIB.

**name** - Is the macro **Identification name**. The name can be up to 8 ASCII characters maximum. The name must be followed by a comma. The name in the unload command can be followed by a semicolon.

**binary byte** - Is an eight bit **binary byte** to be loaded by PULSAR.

**parameter** - Is an ASCII string that is passed to the macro program in the form of a **parameter** separated by spaces. The maximum number of parameters that can be passed to a macro is 15 and the combined string of Xmac name and parameters cannot be greater than 128 characters.

**Note:** The number of parameters must match the number expected by the Xmac program.



## **PULSAR String Commands**

### **PULSAR String Commands**

The String Commands are commands that may be used in the AVTS Workbook BASIC tab for special routines not readily available using the AVTS Controls in the Workbook Test Editor tab.

These String Commands are explained in the following topics:

- Output String Command
- Circle Command
- Clear Command
- Clis- Clear Message Command
- DFRLoad Command
- Draw Command
- Else Command
- Else If Command
- End Command
- End If Command
- Enter\_at Command
- For Command
- Get\_Time Command
- GoTo Command
- GoSub Command
- Graph Command
- If Command
- Include Command
- Label Command
- Local Command
- Locate Command
- Monitor Command
- Move Command
- Msg\_at Command
- Next Command
- Off Command
- On Command
- Print Using Command
- PulseMonitor Recorder Macro Command
- PulseMonitor Sequencer Macro Command
- Ramp Command
- ReadUnit Command
- Remote Command
- Reset Command
- Return Command
- Save Command

- Set Command
- SetHdrLn - Set Header Line Command
- SIPulsar - Select Command
- SITCM915 - Select Command
- Solve Command
- Step Command
- Timer Reset Command
- Trigger Command
- TRGetTag - Timer Reference Get Tag
- TRTrig - Time Reference Trigger
- Unit Command
- Wait Command
- Win\_size Command
- WinExec Command
- XmacLoad Command
- XmacUnLd Command



## Output String Command

Syntax:

```
+---+ +-----+  
! > +- ! string expression !  
+---+ +-----+
```

Comments:

This command will send a string expression to all EPOCHS set to listen by the "Unit" command or to the first device set to listen by the last Unit command.

Example:

```
> "V L 67.33;"
```

This example would set the voltage channel in all units addressed to listen to 67.33 volts.

## Circle Command

Syntax:

```
+-----+ ++ +-----+ ++ +-----+ ++ +-----+
| circle +-,+-| xloc +-,+-| yloc +-,+-| Rad. |
+-----+ ++ +-----+ ++ +-----+ ++ +-----+
```

Comments:

This command will draw a circle in graphics mode.

Example:

```
circle,cos(75)*1.5,sin(75)*1.5,1.5
```

This example would plot a three phase characteristic with 1.5 Radius.

## Clear Command

Syntax:

```
+-----+  
| clear |  
+-----+
```

Comments:

This command will reset all devices on the GPIB bus enabled for clear in Device Addressing and clear any Pulsar hooked up to RS-232.

Example:

```
clear
```

This example would reset each device on the bus to its power-up state.

## Cls - Clear Command

Syntax:

```
+-----+  
|  cls  |  
+-----+
```

Comments:

This command will clear a message window of all text and place the cursor back in the upper left corner of the message window.

Example:

```
> cls
```

This example would clear a message window.

## DFRLoad Command

Syntax:

```
+-----+ +-----+
| DFRLoad +--[| string expression |]
+-----+ +-----+
```

Comments:

Attempts to download a DFR test file to the first selected PULSAR. The optional string expression should specify the full or partial pathname of the test file to be used. If a partial pathname is given the function will look for the test file based on the programs current working directory. If the test file cannot be found or wildcards are given, the Get Test file dialog box is opened so that the user may select a test file.

Once loaded the program sets five macro variables with information about the downloaded macro.

DFRTOFIL = time from start of converted data to start of original data. Negative values indicate data has been removed.

DFRTOEVN = time from start of converted data to event time in original data. Negative values indicate the event data was cut off.

DFRTONUL = time from start of converted data to beginning of null data.

DFRWIDTH = How long to play data loaded into the wave location. If equal to DFRTonul then no null data was added.

SmpRate\$ = a string with the sample rate.

```
|<-----DFRTOEVN----->|
+-----+-----+-----+
| added      | -event--->| | null data |
+-----+-----+-----+
|<-DFRTOFIL->| | |
|<-----DFRTONUL----->| |
|<-----DFRWIDTH----->|
```

### WARNING:

DFRLoad forces the extension ".TST" to load the selected test file.

Each target generator should be turned off before the data is downloaded and the PULSAR commands On and Restore Phase should be sent together before the next semicolon is sent so that the generators will turn on playing the first data point. DFRLoad will program PULSAR for Synch Immediate mode to force proper operation in case the master generator is sent the sample rate command.

Note: If the downloaded waveform is to be used in other tests then the macro variable 'no\_clear' should also be set to prevent the clear command from being sent at the beginning of the next test. Clear only clears waveform location 1. If the waveform is not loaded in location 1 then only the sample rate and amplitudes would have to be resent after clear is issued.

Example Macro:

```
*****
* Playback a DFR file
*****

    pxlatoff=1          'make sure translator is turned off
>"v0c0,waimi,s;"      'make sure all outputs off

DFRLoad dfrfile$
DFRLoad "dfr\station1.tst"
'PLAY DOWNLOADED DATA
```

```
>"v0c0,re," 'issue restore phase so that first sample will play first
>"on," 'turn on generator
fix=1 'changes precision for "wai" commands
if DFRTtoEvn<>0 then
  >"wai"+str$(DFRTOEVN*1000)+"," 'add a wait so timer turns on at event
end if
>"tsosta," 'start PULSARs timer
'let fault to play for 1666.6ms or 100 cycles
>"wai"+str$( (1.6666-DFRTOEVN)*1000 ) +","
fix=4
>"off;" 'turn off generators
mon_addr=2 'monitor the default stop timer binding post for activity
for i = 1 to 20
  if monitor then i=100
  wait .1 'wait 100 msec
next i
if i < 100 then
  msg_at 1,1,"No pickup detected."
  wait key
  goto BYE
end if
get_time 'get time recorded by PULSAR timer
msg_at 1,1,"Pickup time "+str$(time)
wait key
:BYE
clear
local
end
```

## Draw Command

Syntax:

```
+-----+ +-----+ +-+ +-----+
| draw +-| x position +-|,+ -| y position |
+-----+ +-----+ +-+ +-----+
```

Comments:

This command will draw a line from the present position on the graph to the position 'x position', 'y position'. The present position can be changed via the 'move' command.

Example:

```
move 0,0
```

```
draw 5.4,3.2
```

A line would be drawn from the position '0,0' to '5.4,3.2'.

## Else Command

Syntax:

```
+-----+  
| else |  
+-----+
```

Comments:

This command is used in conjunction with the 'if' command and indicates that all following commands will be performed if the 'if' condition was evaluated false. If more than one command is grouped to the else condition then an 'end if' termination is required.

Example:

MULTIPLE INSTRUCTION FORMAT:

```
if i < 10 then 'no command(s) follow 'then' for multiple command groups.  
... 'if tested-true, group of commands.  
else 'the 'else' terminates multiple tested-true commands.  
... 'if tested-false, group of commands.  
end if 'the 'end if' terminates multiple tested-false commands.
```

Or,

SINGLE INSTRUCTION FORMAT:

```
if i < 10 then j = 5 else j = 10 'end if' not needed
```

Or,

```
if i < 10 then j=5 'no 'else' statement; 'end if' not needed.
```

Or,

NO ELSE FORMAT:

```
if i<0 then 'no 'else' command format  
... 'group of multiple commands  
end if
```



## Else If Command

Syntax:

```
+-----+ +-----+ +-----+
| else if +-| logical expression +-| then +
+-----+ +-----+ +-----+
      TRY THE else COMMAND FIRST
```

Comments:

This command can be used in situations where the first command after the 'else' is an 'if'.

The command group consists of all following commands and nested if-then blocks up to the next 'end if' not matched to an if-then.

'Logical expression' can consist of constants or variables and may be correlated by the logical operators ...

<b>operator</b>	<b>function</b>
=	true if left operand equals the right operand.
<	true if left operand is less than the right operand.
>	true if the left operand is greater than the right.
<>	true if the left operand does not equal the right.
and	used to group two logical expressions together; the overall condition can only be true when both are true.
or	used to group two logical expressions together; the overall condition is true when either of the two is true.

Example:

MULTIPLE INSTRUCTION FORMAT:

```
if i < 10 then 'no command(s) follow 'then' for multiple command groups.
...          'if tested-true, group of commands.
else if i>10 then 'the 'else' terminates multiple tested-true commands.
...          'if tested-false, group of commands.
end if      'the 'end if' terminates multiple tested-false commands.
```

## End Command

Syntax:

```
+-----+  
| end |   IMPORTANT: This command must be executed  
+-----+           before any results are displayed.
```

Comments:

This command signals the end of macro file execution; once the 'end' command has been executed all normal macro operations cease and the results presentation phase begins. Since print and graph results macros must also run using only the data stored in the results database the end command tries to recreate the environment found when results are presented using data in the database.

After end has executed, all numeric variables are cleared, the expressions in the header are re-evaluated, the last ".MTP" macro found is executed for each results record and the last ".MTG" macro listed is executed once.

Example: (as seen in the Main Program dialog box):

```
include reach.mtm  'file performs the test  
off               'turn all channels off  
local            'turn GPIB off  
end              'prepare form results presentation  
include results.mtp  'this file gives printed results  
include gresults.mtg 'this file gives graphic results
```

## End If Command

Syntax:

```
+-----+  
| end if |  
+-----+
```

Comments:

This command is used in conjunction with the 'if' command and indicates that the command group has ended.

A command group consists of all commands between an 'if', 'else' or 'else if' and the 'end if'.

Example:

```
if i < 10 then  
.....  
.....  
end if
```

Terminates the previous command group.

## Enter\_at Command

Syntax:

```

+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
|enter_at+-|line #+-|,+--|column #+-|,+--|length+-|,+--|string var+
+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+|+-----+
                                                    | +-----+
                                                    +-|numeric var+
                                                    +-----+
    
```

Comments:

This command allows a user to pass a string or numeric variable from the keyboard to the macro. It will place the cursor at 'line #', 'column #' and allow an enter field 'length' characters long. Keyboard entry is then used to enter data into the field. When the user hits the <Enter>, <Esc>, <Cursor Up> or <Cursor Down> keys, the characters entered into the field will be placed in the string 'string var' or converted to a numeric value and placed in the variable 'numeric var'. The program also sets a variable when the <Esc>, <Cursor Up> or <Cursor Down> keys are struck to allow the macro to determine which action to take next. A good guideline for using enter\_at is that, if more than one field is needed, the macro should move between fields until the user hits <Esc>.

**Note: All string variables must end with a '\$'.**

This will occur in the 'message' window. If the window does not exist it will be created although the size will probably be wrong unless win\_size is executed first.

Example:

```

msg_at 3,5,va
msg_at 4,5,vb
msg_at 5,5,vc
:ENTA
  enter_at 3,5,7,va
  if escape <> 0 then goto DATADONE
  if up_arrow <> 0 then : up_arrow=0: goto ENTB: end if
:ENTB
  enter_at 4,5,7,vb
  if escape <> 0 then goto DATADONE
  if up_arrow <> 0 then : up_arrow=0: goto ENTA: end if
:ENTC
  enter_at 5,5,7,vc
  if escape <> 0 then goto DATADONE
  if up_arrow <> 0 then : up_arrow=0: goto ENTB: end if
goto ENTLINE
:DATADONE
set
    
```

This macro will allow the user to enter a value in three fields. Each time the <Enter>, or <Cursor Down> key is hit the cursor will move down one field. The <Cursor Up> key will move the cursor up one field. The <Esc> will exit the data entry loop apply the entered voltages.

**Note: The message window will automatically self-destruct when the first 'non message' window command is executed unless the variable 'no\_purge' has been set to 1.**

Note also: Unusual keys can be tested for using their ASCII code representations in 'if then' commands.

## For Command

Syntax:

```
+----+ +-----+ ++ +-----+ +--+ +-----+ +-----+ +-----+
|for+-|variable+-|=+-|start+-|to+-|stop+-[|step+-|value+|
+----+ +-----+ ++ +-----+ +--+ +-----+ +-----+ +-----+
+-Spaces MUST precede and follow equal sign or the
variable will not be updated and/or the start count
will not be set to the result of the conditional test
```

Comments:

This command is used to execute a group of commands a defined number of times. The command group consists of all commands between the 'for' statement and before the corresponding 'next' statement. The command group will be executed while the variable 'variable' is between the defined 'start' and 'stop' conditions. When the corresponding next command is reached the variable will be incremented by 'value'.

The '[' surrounding 'step value' indicates that this is an optional parameter. If the optional 'value' was omitted then variable will be incremented by '1'. 'Variable' must be a numeric variable. 'Start', 'stop' and 'value' can either be constants, variables or a numeric expression of both.

Example:

```
for i = 1 to 10
  j = i * 2
next i
```

This program segment will cause the command 'j = i \* 2' to be executed 10 times. The value of 'i' will change from '1' to '10' respectively.

**Warning! Goto's that jump outside a 'for' loop are not supported.**

## Get\_Time Command

Syntax:

```
+-----+  
| get_time |  
+-----+
```

Comments:

This command will set the numeric variable 'time' with the test sets timer's reading. If no timer is available the command will hang the GPIB and abort macro execution.

If the Selected Unit is Pulsar, the selected pulsar will be used. If the Selected Unit is EPOCH, the last EPOCH with a timer will be used.

Example:

```
get_time
```

If the EPOCH or PULSAR timer read '2.6453' then this command would set the variable 'time' to '2.6453'.

## GoTo Command

Syntax:

```
+-----+ +-----+
| goto +-; line label |
+-----+ +-----+
```

Comments:

This command will cause program execution to be transferred to the line labeled by 'line label'. A line label is defined by a ':' as the first character followed by an alphanumeric character string containing no spaces. Line labels of up to twenty characters are supported as opposed to the 8 given to numeric variables.

Gotos within "if-then" and "if-then-else" blocks must follow strict rules. You may NOT jump into a lower level "if-then" or "if-then-else" block. You may jump within the same or the next higher level block depending on the placement of the goto within the block. If the goto is placed just before the "else" or the "end if" it must be to a label outside the "if-then" block. A goto placed anywhere else must jump to a label inside the same "if-then" block.

Example

```
:1b1
i=i-1
if i=1 then goto 1b1 else b=2 ' jump out of block
if j=2 then
:1b2
j=j+1
'next is an "if-then" with jump up to next higher block
if j > 4 then goto 1b3 'implied end if follows goto 1b13
goto 1b2 'jumps inside block
:1b3
'if j > 5 then goto 1b1 'illegal jump out of 2 blocks
end if
```

**Warning!** Goto statements inside for-next loops that jump out of the loop are not supported. The program may be OK for one for-next loop, but goto's that jump out of nested loops will definitely cause for-next loops to work improperly. The same is true of jumping out of subroutine called from a subroutine.

Example:

```
goto TEST
...
:TEST j = j + 2
```

Program execution would jump from the 'goto' line to the line

':TEST j = j + 2' when the 'goto' line is executed. Note that it is suggested to capitalize all line labels so that they are not confused with variables.



## GoSub Command

Syntax:

```
+-----+ +-----+
| gosub +-; line label |
+-----+ +-----+
```

Comments:

This command will cause program execution to be transferred to the line labeled by 'line label'. A line label is defined by a ':' as the first character followed by an alphanumeric character string containing no spaces. When the next occurrence of a 'return' command is executed, program execution will return to the line following the 'gosub' command.

Line labels of up to twenty characters long are supported.

Example:

```
gosub TEST
k = k + 1
:TEST j = j + 2
return
```

Program execution would jump from the 'gosub' line to the line ':TEST j = j + 2' when the 'gosub' line is executed. When the 'return' line is executed program execution will then return to the line 'k = k + 1'.

**Note that it is suggested to capitalize all line labels so that they are not confused with variables.**

## Graph Command

Syntax:

```
graph title,x label,y label,l border,r border,t border,b border,
    # increments,polar flag,resolution x,resolution y,
    data field x,data field y,options mask
    [,test #][,test #][,test #][,test #]
```

```
+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+ +-+
|graph+|title+|,+|x label +|,+|y label+|,+|l border+|,+|
+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+ +-+ |
+-----+
| +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
+|r border+|,+|t border+|,+|b border+|,+|# increments+---+
+-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+ |
+-----+
| +-+ +-----+ +-+ +-----+ +-+ +-----+ +-+
+|,+|polar flag+|,+|resolution x+|,+|resolution y+|,+---+
+-+ +-----+ +-+ +-----+ +-+ +-----+ +-+ |
+-----+
| +-----+ +-+ +-----+ +-+ +-----+ +-+
+|data field x+|,+|data field y+|,+|options mask+[|,+---+
+-----+ +-+ +-----+ +-+ +-----+ +-+ |
+-----+
| +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
+|test #+|[|,+|test #+|[|,+|test #+|[|,+|test #+|
+-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
```

### Comments:

This command will cause the data in the fields 'data field x' and 'data field y' to be plotted according to the rest of the command parameters. The optional parameter 'test #' indicates the number of test data to be plotted. This number corresponds to the position of the desired test in the test menu and allows the user to overlay up to four sets of test data. If these values are omitted then the data presently under test is used. The command parameters have the following functions:

- o title        Displays the string 'title' centered, and just above the plotting area.
- o x label     Displays the string 'x label' centered, and just below the 'x' axis.
- o y label     Displays the string 'y label' centered and just to the left of the y axis.
- o l border    Indicates the size of the left border in a percentage of the width of the screen. A left border of '10' would allocate a left border which is 10% of the width of the screen.
- o r border    Indicates the size of the right border in a percentage of the width of the screen.
- o t border    Indicates the size of the top border in a percentage of the height of the screen.
- o b border    Indicates the size of the bottom border in a percentage of the height of the screen.
- o # increments Indicates the number of sub increments (major tic marks) the graph should contain.
- o polar flag   If this value is '1' it indicates that polar data is to be plotted. If its value is '0' it indicates that rectangular data is to be plotted.
- o resolution x   The value 'n.d' would indicate that every 'nth' tic mark should be labeled with 'd' decimal places of resolution ( x axis ).

- o resolution y The value 'n.d' would indicate that every 'n'th tic mark should be labeled with 'd' decimal places of resolution ( y axis ).
- o data field x Indicates that the x axis will represent the data in this field. Valid parameters are 'f1', 'f2' ... 'f9'.
- o data field y Indicates that the y axis will represent the data in this field. Valid parameters are 'f1', 'f2' ... 'f9'.
- o options mask Indicates the type of plot the data will appear on. Each bit in this numeric entry has the following significance:

Dec	bit	function
1	0	Bottom left corner will be origin
2	1	Rectangular grid will be drawn
4	2	Polar grid will be drawn
8	3	Non proportional data (allows x scaling to differ from y scaling)
16	4	Logarithmic scale for the x axis
32	5	Logarithmic scale for the y axis
128	7	Swap x and y axis of polar plot. All graph commands will swap x and y parameters before plotting.

- o test # Indicates the number of test data to be graphed.

Example:

graph Impedance Plot,R,X,35,2,5,10,10,1,2,2,2,2,f4,f1,4

This command would generate a plot labeled 'Impedance Plot'. The x axis would be labeled 'R' and the 'y' axis would be labeled 'X'. The left border would be '35' percent of the width of the screen and the right border would be '2' percent of the width of the screen. The top border would be '5' percent of the height of the screen and the bottom border would be '10' percent of the height of the screen. '10' major increments would be used along the axis.

This value will be slightly adjusted depending on the plot data. All data would be treated as polar data. Every second major x tic marker will be labeled with '2' decimal places of resolution. Every second major y tic marker will be labeled with '2' decimal places of resolution. The field 'f4' will be used for the x axis data and the field 'f1' will be used for the y axis data. A polar grid will also be drawn on the graph.

## If Command

Syntax:

```
+-----+ +-----+ +-----+ +-----+ +-----+
| if +-| (expression)+-| then+ [| (one command)+ |else+|one command;| ]
+-----+ +-----+ +-----+ +-----+ +-----+
```

Comments:

Performs a test and branches on condition. This command is used in conjunction with the 'else' and 'end if' commands and indicates that a command or a group of commands has started.

A command group consists of all commands between a lower case 'if', 'else' or 'else if' and the 'end if'. The results of an expression are true for non-zero results and false for zero results.

```
Example  j=4           'the variable j is set to 4
         if j then goto TEST 'the goto will be executed.
```

The 'expression' may be a logical expression. A logical expression can consist of constants or variables and may be correlated by the logical operators ...

operator	function
=	true if left operand equals the right operand.
<	true if left operand is less than the right operand.
>	true if the left operand is greater than the right.
<>	true if the left operand does not equal the right.
and	used to group two logical expressions together; the overall condition can only be true when both are true.
or	used to group two logical expressions together; the overall condition is true when either of the two is true.

Example:

### MULTIPLE INSTRUCTION FORMAT:

```
if i <=10 then 'no command(s) follow 'then' for multiple command groups.
...           'if tested-true, group of commands
else         'the 'else' terminates multiple tested-true commands.
...           'if tested-false, group of commands
end if      'the 'end if' terminates multiple tested-false commands.
```

Or,

### SINGLE INSTRUCTION FORMAT:

```
if i < 10 and i>3 then j = 5 else j = 10 'end if' not needed
Or,
if i < 10 or i>13 then j=5 'no 'else' statement; 'end if' not needed.
```

Or,

### NO ELSE FORMAT:

```
if i<0 then 'no 'else' command format
...       'group of multiple commands
```

end if

**CHARACTER MATCHING:**

To match single character string acquired through the 'enter\_at' command,

Example: if (a\$="y") or (a\$="Y") then goto OUT 'if user hit y then jump.

## Include Command

Syntax:

```
+-----+ +-----+  
| include +-| file name |  
+-----+ +-----+
```

Comments:

Loads the program 'file name' into the main body of the executing program. This allows the programmer to 'pull in' multiple sub programs and to take a 'modular' approach to his/her programming needs.

If 'file name' has a suffix '.mtp' it will also set the file as the printout file to be used when printing the results.

If 'file name' has a suffix '.mtg' it will also set the file as the graph generation program to be used when printing the results.

Example:

```
include charac.mtm  
include reach.mtp  
include charac.mtg
```

This will load the 'charac.mtm' macro followed by the 'reach.mtp' and the 'charac.mtg' macro files. It will also set 'reach.mtp' and 'charac.mtg' as the printout and graphic macros for future printing.

## Label Command

Syntax:

```
+-----+ +-----+ +-+ +-----+ +-+ +-----+
| label +-| y position +-|,+ -| x position +-|,+ -| string |
+-----+ +-----+ +-+ +-----+ +-+ +-----+
```

Comments:

The string 'string' will be displayed on a graph at the position on line 'y' and column 'x'. If no graph is in progress then the command will be ignored.

Example:

```
label 4,4,"hi mom"
```

This will display 'hi mom' on line '4' and column '4' of the present graph.

## Local Command

Syntax:

```
+-----+  
| local |  
+-----+
```

Comments:

All devices under GPIB control will be returned to their local mode of operation. This will allow front panel control of the devices. This command sends the 'local' command to the GPIB interface board. If PULSAR is being controlled, the string "loc," is sent and if it is controlled from a COM port the port is closed to free the COM port for use by other programs.

Example:

```
local
```

All devices will return to their local mode of operation.



## Locate Command

Syntax:

```
+-----+ +-----+ +-+ +-----+
| locate +-| y position +-|,+ -| x position |
+-----+ +-----+ +-+ +-----+
```

Comments:

Move the present cursor position of the printout window to the line 'y position' and the column 'x position'. If a printout window does not exist then the command will be ignored. All 'print using' commands start at the present cursor position.

Note also that the variable 'cslin' always contains the line number of the present cursor position.

Example:

```
locate cslin-1, 5
```

This will cause the present cursor position to move up one line and go to column '5'.

## Monitor Command

Syntax:

```
+-----+
| monitor |
+-----+
```

Comments:

This function returns the current state of the monitor binding post(s). The binding posts to be checked are defined by the variable 'mon\_addr'. If the function returns '1' then one of the binding post specified by 'mon\_addr' detected activity else it is set to '0'. If the first selected unit is an EPOCH then the unit specified in mon\_addr is an EPOCH.

If the Selected Unit is a Pulsar that unit will be the unit addressed when this command issued (NOTE: Since the Set command only works with Pulsar when it is the first Unit the binding post configuration described will only work for that unit).

PULSAR: Monitor updates 'add\_ret' with the active monitors using the same bit encoding scheme used in 'mon\_addr'. If 'mon\_addr' is zero, then 'add\_ret' is updated with the address of all the active monitors using the bit encoding scheme. If 'mon\_addr' is non zero, then 'add\_ret' is updated with each of the active binding posts specified in 'mon\_addr'. In PULSAR the active state of a binding post depends on whether it is configured for 'Applied' or 'Removed', the variable 'normal' controls whether the 'ramp' and 'step' commands respond to the active or inactive states. Because of this the variable 'normal' only needs to be set when 'ramping' or 'stepping' with normally closed contacts that are not being latched, otherwise the binding post can be programmed to go active in 'Applied' or 'Removed'. Unless disabled by the variable 'NoTmrCmd', the 'Set' command is responsible for configuring the binding posts. The timer variables 'TStart', 'TStop' and 'TLatch' program binding post 1 and 2 for timer start and stop modes respectively and can still be monitored if selected by 'mon\_addr'. Binding post 3, normally used for monitoring, can be programmed for timer stop by setting 'Mon1Stop'.

PULSAR (Translator Disabled):With the translator disabled PULSAR's timer module is more fully supported. If the timer variables 'TStart' or 'TStop' are zero and 'mon\_addr' is set to monitor those posts then the 'Set' command also configures the specified posts to be monitor posts. The 'TLatch' timer variable affects both binding posts in the timer or monitor mode unless it is set to zero. The variables 'latch\_no', 'latch\_nc' and 'mon\_volt' configure each binding post selected by 'mon\_addr' unless overridden by the timer variables or the variables 'mon2conf' and 'mon3conf'. The variables 'mon2conf' and 'mon3conf' can be used to individually configure, respectively, the stop and start binding posts, unless overridden by the timer variables, for modes different from the monitor.

If a binding post has been configured by 'Set' for latch mode, has latched its signal, is selected by 'mon\_addr' and is not selected by TStart or TStop, then the latch will be cleared once it is read. When a binding post is selected by TStart and TStop, latches on those binding posts are not cleared until 'Set' executes again.

If set the variable 'PDebTime' will be used to program the timer module debounce time otherwise a 170msec debounce time is set when no post is configured by 'Set' for latch mode or a 4msec debounce time when a latch mode has been selected.

If the variable 'NoTmrCmd' is set then PulseMaster makes no attempt to configure the timer module or to clear any latches. This allows the user to directly send the commands to configure each binding post but requires the user to also clear each latch.

**Note: The receive buffer will also be checked for errors when this command executes as does the ramp and step commands.**

## Move Command

Syntax:

```
+-----+ +-----+ +-+ +-----+
| move +-| x position +-|,+ -| y position |
+-----+ +-----+ +-+ +-----+
```

Comments:

Move the present graphics position to 'x position', 'y position'.

Example:

```
move 3.4,5.6
```

This will cause the present graphics position to move to x position '3.4' and y position '5.6'

## Msg\_at - Message At Command

Syntax:

```
+-----+ +-----+ ++ +-----+++ +-----+
| msg_at +-| y position +-|,+--| x position +|,+--| string |
+-----+ +-----+ ++ +-----+++ +-----+
```

Comments:

Displays the string 'string' at line 'y' and column 'x' in a message window. If the message window does not exist then it will

be created. The message window will automatically self-destruct when the first 'non message' window command is executed unless the variable 'no\_purge' is set to 1. The message window commands are listed below under 'See also'.

The "Esc" key can be used to abort test execution if the user can't click Abort with a mouse.

Example:

```
msg_at 3,10," Have a nice day ! "
```

The message ' Have a nice day ! ' will be displayed on line '3' and column '10' in the message window.

## Next Command

Syntax:

```
+-----+ +-----+
| next +-| variable |
+-----+ +-----+
```

Comments:

Causes execution to return to the corresponding 'for' loop with the value of 'variable' being incremented. If the 'for' loop has exceeded its boundaries, then program execution will continue with the line immediately following the 'next'.

Example:

```
for i = 1 to 10
  j = 2 * i
next i
i = 0
```

If 'i' is less than '10' then it will be incremented by '1' and program execution will continue with the 'j = 2 \* i' command. If

'i' equals '10' then program execution will continue with the 'i = 0' command.

## Off Command

Syntax:

```
+-----+   +-+
|  off +[--|i|]
+-----+   +-+
          |   +-+
          |   +-+
          +-|v|]
          +-+
```

Comments:

Causes either all voltage and/or current channels to be turned off. If 'i' is specified, all current channels are turned off. Specifying 'v' turns all voltage channels off. Specifying neither turns all channels off. Note that the channel settings are not changed by this command.

**Note: this command only works with the first Pulsar or EPOCHS at the top of the device list.**

Example:

```
off
```

All current and voltage channels will be turned off.

## On Command

Syntax:

```
+-----+  
| on |  
+-----+
```

Comments:

Causes all voltage and current channels to be turned on and the present channel variables to be sent out by issuing the 'Set' command.

**Note:** all channels 'on' variables are set.

**Note:** this command only works with the first Pulsar or EPOCHS at the top of the device list.

Example:

```
on
```

All current and voltage channels will be turned on.

## Print Using Command

Syntax:

```

+-----<-----<-----+
+-----+ +-----+ +-+|+-----+ +-+|
|print using+-|format string+-|;|numeric expression +- [|,+ -]
+-----+ +-----+ +-+|+-----+| +-+
|+-----+ |
+|string expression+--+
+-----+
    
```

Comments:

Displays a line at the present cursor position, in the 'printout' window, according to 'format string' and with the variables 'numeric expression' and 'string expression'. If the 'printout' window does not exist it will create it. For a single test up to 66 lines can be printed. After the 66 line have been printed the print using skip the write portion.

'Format string' can consist of text or variable specifiers. For each variable specified it will display the next variable/expression according to the specifier table below:

specifier	function
\ \	The first 'n' characters of the next string variable will be displayed. 'n' is equal to the number of spaces between the '\ 's plus 2.
\n\	The first 'n' characters of the next string variable will be displayed.
##.###	The next numeric variable/expression will be displayed with two digits to the left of the decimal and three digits to the right of the decimal.
~cn	Will display 'n' of the characters 'c'.
~n	Will display 'n' spaces.
\	States that the command will continue on the next line.

Example:

```

print using "hi!, num=###.####, string = \8\ ~R4 (the end)"; \
i * 2 , label$
    
```

If 'i' were equal to '4.21' and label\$ were equal to 'im a string' then the resulting display would be ...  
 hi!, num= 4.2100, string = im a str RRRR (the end)



## PulseMonitor Recorder Macro Command

**PMonRcd**.*objectid*.*function* <*function parameters*>

**PMonRcd** - the keyword identifying a PulseMonitor Sequencer function call.

*objectid* - variable name, up to 15 characters, that differentiates one PulseMonitor Recorder macro object from another. The object is created the first time a function call is made with reference to that named object and is destroyed when macro execution is finished.

*function* - the PulseMonitor Recorder function to be performed. See below for the list of functions and their associated parameters.

*function parameters* - the parameters that a function may or may not require.

**Example:** PMonRcd.recording1.StartRecorder

### PMonRcd Functions

#### OpenFile *filename*

Reads the PulseMonitor Recorder file specified by *filename*.

*filename* - is a string expression that is the full or partially qualified pathname of the file.

#### SaveFile *filename*

Saves the **PMonRcd** object to the PulseMonitor Recorder file specified by *filename filename* - is a string expression that is the full or partially qualified pathname of the file.

#### StartRecorder

Resets and starts the TCM-915s sequencer which clears and starts the recorder. The command programs the box for operation using the current Recorder Mask, for Group Execute Trigger (GET), waits 50msec and sends the GET.

#### StopRecorder

Stops the recorder by sending the recorder stop command.

#### LoadRecording

Loads the current recording in the TCM-915 to the **PMonRcd** object.

#### GetNoRecords *numeric variable*

Retrieves the number of records stored in the **PMonRcd** object and stores the value in the *numeric variable*.

*numeric variable* - the variable to store the result in.

#### GetRecord *recno numeric expression, input string variable, output string variable, time string variable*

Retrieves the data for record specified by the *recno numeric expression* from the **PMonRcd** object and stores the results in the *string variables* specified.

#### FindEdge *numeric variable, Start Rec, Rising Edge, Edge No., Bit no*

Searches the recording in the **PMonRcd** object for the record where the *nth* rising or falling edge of an input is detected. Edges are assigned to the record that follows the edge.

*numeric variable* - the name of the variable to receive the number of the record found. If set to -1 the record was not found.

*Start Rec* - a numeric expression that identifies the record number to start the search with. When performing the search the record before *Start Rec* is assumed to have the same state as the starting record and cannot count as an edge except when *Start Rec* is 0. When *Start Rec* is 0 then the search starts with the first record and counts it as an edge. *Start Rec* should be a number between 1 and the number of records in the recording.

*Rising Edge* - a numeric expression that selects the rising or falling edge. If *Rising Edge* is 0 then the falling edges are counted otherwise the rising edges are counted.

*Edge no* - a numeric expression that specifies how many edges to count to select the record. *Edge no* should be a number between 1 and half the number of records in the recording.

*Bit no* - a numeric expression that specifies the input bit to be searched. Valid inputs are 1 through 15.

**FindPattern** *numeric variable, Start Rec, input pattern, output pattern*

Searches the recording in the **PMonRcd** object for the record the first record matching *input pattern* and *output pattern*.

*numeric variable* - the name of the variable to receive the number of the record found. If set to -1 the record was not found.

*Start Rec* - a numeric expression that identifies the record number to start the search with. *Start Rec* should be a number between 1 and the number of records in the recording.

*input pattern* - a string expression that specifies the input state to be searched for. The string expression should be a 0 to 15 character string with the characters 0, 1 or X. For strings shorter than 15 characters the undefined characters are assumed to precede the string and default to X (i.e. 1=xxxxxxxxxxxxx1).

*output pattern* - a string expression that specifies the output state to be searched for. The string expression should be a 0 to 9 character string with the characters 0, 1 or X. For strings shorter than 15 characters the undefined characters are assumed to precede the string and default to X (i.e. 1=xxxxxxx1).

**SetBoxNo** *numeric expression*

Sets the TCM-915 box number the **PMonRcd** object will be communicating with. The *numeric expression* is the nth TCM-915 box found in the Set Devices configuration dialog (not the nth Unit).

**GetBoxNo** *numeric variable*

Sets numeric variable to the current TCM-915 box number set in the **PMonRcd** object.

*numeric variable* - the variable to store the result in.

**SetRecorderMask** *string expression*

Sets the recorder mask stored in the **PMonRcd** object. The *string expression* should produce a 15 character string using the characters 1 or 0.

**GetRecorderMask** *string variable*

Gets the current recorder mask setting in the **PMonRcd** object. The result, a 15 character string of the characters 1 or 0, is assigned to *string variable*.

*string variable* - the variable to store the result in.

## PulseMonitor Sequencer Macro Command

**PMonSeq**.*objectid.function* <*function parameters*>

**PMonSeq** - the keyword identifying a PulseMonitor Sequencer function call.

*objectid* - a variable name, up to 15 characters, that differentiates one PulseMonitor Sequencer macro object from another. The object is created the first time a function call is made with reference to that named object and is destroyed when macro execution is finished.

*function* - the PulseMonitor Sequencer function to be performed. See below for the list of functions and their associated parameters.

*function parameters* - the parameters that a function may or may not require.

**Example:** PMonSeq.sq1.ResetSequencer

### PMonSeq Functions

#### LoadSequencer

Loads the TCM-915 with the currently entered sequencer commands.

#### ResetSequencer

Resets the TCM-915 sequencer.

#### ExecuteSequencer

Sends the Outputs and the Trigger mode to the TCM-915 sequencer and arms the TCM-915 to trigger.

#### OpenFile *filename*

Reads the PulseMonitor Sequencer file specified by filename.

*filename* - *filename* is a string expression that is the full or partially qualified pathname of the file.

#### SetBoxNo *boxno*

Sets the TCM-915 box number the PMonSeq object will be communicating with. *boxno* - is a numeric expression specifying the nth TCM-915 box found in the Set Devices configuration dialog (not the nth Unit).

#### GetBoxNo *numeric variable*

Retrieves the TCM-915 box number from in the **PMonSeq** object.

*numeric variable* - the variable to store the result in.

#### SetOutputs *output*

Sets the outputs stored in the **PMonSeq** object.

*output* - a string expression that results in a 9 character string using the characters 0, 1 or X.

#### GetOutputs *string variable*

Gets the current output setting in the **PMonSeq** object. The result, a 9 character string of the characters 0, 1 or X, is assigned to *string variable*.

*string variable* - the variable to store the result in.

#### SetInputSense *input sense*

Sets the input sense stored by the **PMonSeq** object.

*input sense* - a string expression that results in a 15 character *string* with the characters V or C.

**GetInputSense** *string variable*

Gets the current input sense setting in the **PMonSeq** object. The result, a 15 character string of the characters V or C, is assigned to *string* variable.

*string variable* - the variable to store the result in.

**SetRecorderMask** *mask*

Sets the recorder mask stored by the **PMonSeq** object.

*mask* - a string expression that results in a 15 character string with the characters 1 or 0.

**GetRecorderMask** *string variable*

Gets the current recorder mask setting in the **PMonSeq** object. The result, a 15 character string of the characters 1 or 0, is assigned to *string* variable.

*string variable* - the variable to store the result in.

**SendCommand** *command*

Sends *command* to the TCM-915 selected by the box number in the **PMonSeq** object.

*command* - a string expression that results in a legal command string of less than 128 characters.

**PerformQuery** *string variable* , *command*

Sends command to the TCM-915 selected by the box number in the **PMonSeq** object, reads the response to the command and stores it in *string variable*.

*string variable* - the variable to store the result in.

*command* - a string expression that results in a legal command string of less than 128 characters with presumably no more than one query command.

## Ramp Command

Syntax:

```
+-----+
| ramp |
+-----+
```

Comments:

Causes all values to be applied to the EPOCH(s) or PULSAR and all channels that are in the 'ramp' mode to ramp from the defined 'start' value to the defined 'stop' value. Each step in the ramp will occur every 'delay' cycle(s) and will increase/decrease in amplitude by 'inc' value. 'Start', 'stop', 'delay', and 'inc' correspond to interactive variables. If these variables are not set at the time of the 'ramp' then they will be set to the evaluation of their respective fields in Test Specifications. Some macros perform a second ramp using 'inc2', 'delay2' and 'r2backup' (starting at %backup from the first ramp's pickup). The pulse times as short as 1 cycle can be specified however, delay times, are limited by how fast the PC can send commands. For PULSAR without the translator enabled delays of 10 cycles are easily achievable on a 25MHz 386 and 20 cycles using the translator. EPOCHS should also be limited to 20 cycle delay times.

The variable 'normal' specifies the normal state of the monitor. A value of '1' indicates a normally closed contact and a value of '0' indicates a normally open contact. If normal is set then an active monitor will terminate the ramp if normal is '0' then all active monitors must be removed to terminate the ramp.

The variable 'time' will contain the time lapse, in seconds, from the start of the ramp to the completion of the ramp and is used in macros to determine if a ramp started in pickup.

**Note: this command only works with the first Pulsar or EPOCHS at the top of the device list.**

Example:

```
ramp
```

The outputs will change from their 'start' value to their 'stop' by increments of 'inc' occurring every 'delay' cycle(s).

**Note: If the channels are set to ramp and 'pulse' is not zero, then the output will toggle from steady state to the ramp value. This is very useful in dynamic testing. See 'Pulse Time' in 'Test Specifications' for more information.**

## ReadUnit Command

Syntax:

```

+-----+      +-----+
| ReadUnit +--+[-| string variable | ]
+-----+      +-----+
                | +-----+
                | +-----+
                +--+[-| numeric variable | ]
                +-----+

```

Comments:

ReadUnit reads the up to 80 characters from the first selected unit. If a string variable follows the command the string is assigned to the string variable. If a numeric variable follows the command then the first number found is converted to floating point and assigned to the variable. If a variable is not specified the data is discarded and the function returns.

The number of characters actually read depends on whether the Instrument is connected to RS-232 or GPIB. For GPIB the command will read one block of data. The timeout value specified in National Instruments driver configuration file will determine how long the command will wait for the block to be transmitted. If the device is connected to RS-232 the command reads data until the port is quiet for about 10msec or the 80 characters have been read. If the COM port is being used by PULSAR the user may need to wait 50 msec for the reply to appear.

Example:(PULSAR)

```

>"qs," 'query pulsar status
if pulsar_u < 5 then wait .05 'if on com port wait for reply
ReadUnit a$
local
end

```

Example:(EPOCH)

```

>"SET? PH_REF":gosub sb
>"RUN? TIMER":gosub sb
>"CHANGE GPIB 10.15":gosub sb
>"STATUS?":gosub sb
>"ERR?":gosub sb
local
end
:SB: ReadUnit a$ :msg_at 1,1,a$ :wait key:return

```

## Remote Command

Syntax:

```
+-----+  
| remote |  
+-----+
```

Comments:

Causes all devices on the GPIB bus to go under GPIB control. Note that this command must be performed after a 'local' command if the user wishes to send additional data to the devices. This command sends the 'remote' command to the GPIB interface board and renames the Epoch commands Volt, Current, Phase, Level to the single letter commands V, C, P and L.

Example:

```
remote
```

This example would set all devices in the remote mode of operation.

## Reset Command

Syntax:

```
          +---<-----<--+
+-----+|+-----+ |
| reset +-| variable +-+
+-----+ +-----+
```

Comments:

Resets the variables listed to their respective starting values in 'Test Specifications'. If the variable is set to "all" then the amplitude, phase and frequency settings for each channel are reset to the values found in 'Test Specifications'. The individual variables supported by reset are:

va vap vb vbp vc vcp vdc start inc delay min\_range

ia iap ib ibp ic icp idc stop inc2 pulse max\_range

Example:

```
reset start stop
```

If the 'start' and 'stop' fields in Test Specifications contained 'tap \* .8' and 'tap \* 1.5' respectively and 'tap' was equal to '5', then 'start' would be set to '4' and 'stop' would be set to '7.5'.



## Return Command

Syntax:

```
+-----+  
| return |  
+-----+
```

Comments:

Used in conjunction with the 'gosub' command, causes program execution to be returned to the line immediately following the 'gosub' command.

Example:

```
gosub TEST  
k = k + 1  
...  
:TEST j = j + 2  
return
```

Program execution would jump from the 'gosub' line to the line ':TEST j = j + 2' when the 'gosub' line is evaluated. When the 'return' line is executed program execution will then return to the line 'k = k + 1'. Note that it is suggested to capitalize all line labels so that they are not confused with variables.

## Save Command

Syntax:

```
+-----+
| Save |
+-----+
```

Comments:

Causes the values in the variables 'f1' through 'f9' to be added to a results array in memory. Note that this is temporary storage that is not saved to the main database until 'Save Results' is performed.

If the 'Save Results' procedure is not performed, all data will be lost when either the test is run again, when the Test Module is exited, when 'Null Results' is executed or when the 'Load Results Module' is executed.

The array is shared by all tests in the Test Module and can hold a total of 99 elements. The first time a test executes 'Save' two elements of the array are consumed. One element is used to store the names of the test and of the print and graph macros, the other element is used for the data. After that, 'Save' only consumes one element.

The variables/fields 'f1' to 'f9' are typically assigned as follows:

field	description
f1	current phase angle
f2	test voltage
f3	test current
f4	test result #1
f5	test result #2
f6	test result #3
f7	minimum acceptable range
f8	maximum acceptable range
f9	general use/flags

Example:

```
f1 = iap : f2 = va : f3 = ia : f4 = ia
```

```
f7 = min_range : f8 = max_range
```

```
save
```

The first two lines set the temporary variables to the desired values. The 'save' then sends this data to the database.

**NOTE:** The fields in the database for storing these numbers have a fixed size of 8 and the decimal point fixed in the fourth position. Therefore, any number to be 'saved' should fit in the following template: XXXX.XXX .

## Set Command

Syntax:

```
+-----+
|  set  |
+-----+
```

Comments:

Causes all interactive variables which correspond to settings on the EPOCH(s) or PULSAR to be sent to the instruments. If a variable has not changed since the last 'Set' command was executed, the instrument may not be sent the old value. Note that 'ramp', 'step' and 'on' automatically call the 'set' command each time they are used.

Unless 'reset all' was called first, the first time 'set' is called, even if its from ramp, it will solve the expressions in the Test Specification screen associated with variables 'Start', 'Stop', 'Inc', 'Inc2', 'Pulse', 'Delay', 'MinRange', 'MaxRange', and 'Delay'. If 'Start' or 'Stop' are not zero, however, those two expression are not evaluated.

Every call to 'set' then outputs the tap settings, any monitor settings, the settings for each channel, and any external timer settings.

**Note: this command only works with the first Pulsar or EPOCHS at the top of the device list.**

Example:

```
set
ia = 5 : va = 67
set
```

The first line sets the current to five amps and the voltage to sixty seven volts. These values are not applied until the set command in line two is executed .

## SetHdrLn - Set Header Line Command

Syntax:

```
+-----+ +-----+ +---+ +-----+
| SetHdrLn +-! line no. +-! , +-! string +
+-----+ +-----+ +---+ +-----+
```

Comments:

Allows the Relay Settings lines in the Header to be modified by a macro program.

line no.- numeric expression specifying which line is to be modified. Valid numbers are 1 through 3.

string - string expression with the text for the line. Up to 75 characters of the expression are copied.

Example:

```
no_purge=1 'keep message window upr
fix=2     'fix precision to hundredths
```

```
msg_at 1,1,"Reach    = "+str$(Zr)
msg_at 2,1,"Max Torque = "+str$(MxT)
msg_at 3,1,"Report Heading "+rpt$
msg_at 5,1,"Hit Esc when done entering values..."
```

```
:TOP
enter_at 1,14,5,Zr
if escape <> 0 then goto DONE
enter_at 2,14,5,MxT
if escape <> 0 then goto DONE
enter_at 3,16,40,rpt$
if escape <> 0 then goto DONE
goto TOP
:DONE
no_purge=0 'close message window
SetHdrLn 1,"Zr="+str$(Zr)+": Mxt="+str$(MxT) 'Ex; Zr=2.11: Mxt=65
SetHdrLn 3,"hd$=""+rpt$+" Reach "+str$(Zr)+"'" 'Ex; hd$="Relay as
found Reach 2.11"

header 're-evalute header lines
local
end
```

## SIPULSAR - Select Command

Syntax:

```
+-----+ +-----+
| SIPulsar +----| numeric expression |
+-----+ +-----+
```

Comments:

Selects the nth Pulsar to accept input and output from the Pulsar addressable commands.

Pulsar addressable commands are - ">", DFRLoad, GetTime, Monitor, ReadUnit, Timer Reset, XmacLoad and XmacUnload.

Example Macro:

```
testunit=2      'choose second unit
SIPulsar testunit 'select Pulsar
>"TSOSTA;"      'start Pulsar timer
wait .023       'wait 23msec
>"TSOSTO;"      'stop timer
get_time        'use get_time to retrieve the time
>"QT,"         'use Pulsar commands to determine the time
ReadUnit a$     '
'Show both times
msg_at 1,1,str$(time)+" "+a$
wait key
local
end
```

## SITCM915 - Select Command

Syntax:

```
+-----+ +-----+
| SITCM915 +----| numeric expression |
+-----+ +-----+
```

Comments:

Selects the nth TCM-915 to accept input and output from the TCM-915 addressable commands.

The TCM-915 addressable commands are - ">", and ReadUnit.

Macro Example:

```
SITCM915 2
'wait for selected unit to output 000111000
LOOP:
>"qo;"
ReadUnit a$
if a$<>"000111000;" then goto LOOP
local
end
```

## Solve Command

Syntax:

```
+-----+ +-----+
| Solve +--+[-| string expression | ]
+-----+ +-----+
```

Comments:

Tries to open the file specified in the path given by the string expression and evaluates each line of the file as an expression. If no pathname is given or the string expression results in an empty string, the clipboard is checked for text and, if found, each line of text is evaluated as an expression.

The expressions can any numerical or string expression.

## Step Command

Syntax:

```
+-----+  
|  step  |  
+-----+
```

Comments:

Causes all values to be applied to the EPOCH(s) or PULSAR and all channels that are in the 'ramp' mode will step from the defined 'start' value to the defined 'stop' value. They will stay at this value until a key is struck or monitor activity is indicated. Then, the output will return to the 'start' value. If these values are not defined prior to the execution of the 'step', these variables will be set to the evaluation of the 'start' and 'stop' fields of the PulseMaster Test Specifications or BASIC Control screen. With mon\_addr set to 1, both posts 2 and 3 on PULSAR will be monitored.

The variable 'normal' specifies the normal state of the monitor. A value of '1' indicates a normally closed contact and a value of '0' indicates a normally open contact. If normal is set then an active monitor will terminate the step if normal is '0' then all active monitors must be removed to terminate the step.

The variable 'time' will contain the time lapse, in seconds, from the step change of 'start' to 'stop' to the reverse of the contacts.

If the variable 'start\_on\_step' is set to '1' then the PULSAR timer will be started on the step change of 'start' to 'stop'. This time can then be read via the 'get\_time' command.

Note: this command only works with the first Pulsar at the top of the device list.

Example:

```
step
```

The outputs will change from their 'start' value to their 'stop' value.



## Timer Reset Command

Syntax:

```
+-----+  
| timer reset |  
+-----+
```

Comments:

Causes the external timer of the selected EPOCH or Pulsarto be reset to '0'. If no timer is available the command is ignored.

Example:

```
timer reset
```

The timer is reset to zero.

## Trigger Command

Syntax:

```
+-----+  
| trigger |  
+-----+
```

Comments:

EPOCH: Causes all units to trigger any 'GET' that is set in the execution-queue of the EPOCH.

PULSAR: Sends the equivalent of a semicolon.

Note: 'GET' is a MACSART command.

Example:

```
unit 1,2
```

```
trigger
```

Units 1 and 2 would accept and process the trigger command.

## TRGetTag - Time Reference Get Tag

Syntax:

```
+-----+ +-----+ +----+ +-----+
| TRGetTag +-[ -| variable +-[ -| , +---| variable | ]
+-----+ +-----+ +----+ +-----+
```

Comments:

This command waits for the Time Reference to acquire a time tag. Upon detecting a time tag event the function then copies the time tag returned by the device into a string variable or will calculate the difference, in seconds, between the time tag and the last trigger time recorded by the program and copy it into a numeric variable.

The time tag can have .1 usec resolution and includes day month and year information. The full tag time is stored in string variables. The number of seconds since the last trigger is calculated and loaded into the numeric variable instead of the tag time because numeric variables can handle only about 7 digits of significance and would truncate any timings over 10 seconds.

If in simulation mode TRGetTag uses the PC's timer and will immediately return the PC's time.

TRGetTag takes approximately half a second to open the COM port and about 200msec to read and store the time and execute the next command. If TRGetTag has enough time to open the port and wait for the first tag event it should be able read the time in 150msec and allow the time reference to catch the second event. Otherwise, the fastest TRGetTag could operate is every 700msec.

Example:

'the user can hook time tag to event trigger to get a tag event

```
>"v1,a120,of;" 'turn off voltages
```

```
>"v0c0,waims,se," 'switch to synch external
```

```
>"re,v1,a120,waic30," 'load prefault commands
```

```
>"a35,waic10,a120;" 'load fault commands
```

```
TRTrig 'get trigger time from the user and wait for trigger to execute
```

```
>"SP," 'switch to synch phase ref so waic will work (PulseMaster takes up to
```

```
'200msec after trigger event to send this so first wait should be greater
```

```
'than 12 cycles) SEE Note BELOW on changing Synch modes.
```

```
TRGetTag n,n$ 'get next tag
```

```
msg_at 1,1,"Tag Recorded at "+n$
```

```
msg_at 2,1,"Tag Recorded "+str$(n)+"msec after trigger."
```

```
msg_at 3,1,"Trigger at "+trigtim$
```

```
wait key
```

```
clear
```

```
local
```

```
end
```

Note: Normally an error is generated when a command to set the synch mode is received while a wait timer is loaded. i.e. when the command string, "a1,waic1,a2,waic2,a3,S,wai4;" is sent. However, it is possible and it is sometimes desirable to change the synch mode while a wait command is in progress. The problem with this, however, is that if the synch mode is being changed as the wait command is expiring it could hang up the wait timer circuitry.

## TRTrig - Time Reference Trigger

Syntax:

```
+-----+ +-----+
| TRTrig +-[ -| string expression | ]
+-----+ +-----+
```

Comments:

This function programs the Time Reference with a trigger time. The optional string expression can be used to enter a predetermined trigger time and should be of the format:

YYYYDDDDHHMMSSmmmuun

where,

YYYY = the year Ex. 1993  
DDD = is the day of the year 001-366  
HH = hour in 00-23  
MM = minute 00-59  
SS = seconds 00-59  
mmm = milliseconds 000-999  
uuu = microseconds 000-999  
n = is hundred of nanoseconds 0-9

If no string expression is given or it evaluates to empty then the Set Trigger Time dialog box is opened for the user to enter a trigger time.

Set Trigger Time dialog box.

FIELDS

Current Time: Time as reported by Time Reference device.  
Accuracy: Accuracy of time as reported by Time Reference device.  
Trigger Time: Trigger time input field

BUTTONS

Set Mark: rounds the current time up to the next 5 second interval, adds 10 seconds and writes it to the Trigger Time field  
OK: attempt to program the currently entered trigger time.  
Cancel: abort the macro.

Once TRTrig has a trigger time it validates it and checks if the trigger time has already expired. If the trigger time has the wrong format or has already expired, the user is informed and given the option to reenter a trigger time or abort the macro. Once the trigger time is found to be valid, the time reference is programmed with the trigger time and if time permits the Countdown to Event Trigger dialog is popped up to show the seconds remaining until trigger. The trigger function then polls the time reference until the trigger time expires.

Before prompting the user for a trigger time or validating it, the time reference is cleared of any outstanding time tag events. If the time reference is in simulation mode and is talking with PULSAR then the Countdown dialog box will remain open until query status returns "No Wait".

Example Macros:

```
*****
*Playback a DFR file
*****
```

```
DFRLoad "dfr\station1.tst"
xmacunld          'unload all xmacs
xmacload "xmacs\syncwait.exe"
wait 1
>"v0c0,waims," 'make wait timers wait for synch pulses before starting next timer >"SE,"
                'hold off command execution until trigger received'
```

## LOAD COMMANDS TO BE EXECUTED BY TRIGGER

```
>"re," 'issue restore phase so that first sample will play first
>"on," 'then turn on output
fix=1 'changes precision for "wai" commands
>"wai"+str$(DFRTOEVN*1000)+," 'wait so timer turns on at event
>"tsosta," 'start PULSARs timer
'let fault play for 1666.6ms or 100 cycles
>"wai"+str$((1.6666-DFRTOEVN)*1000)+,"fix=2
>"off;" 'turn off generators
>"xesyncwait 1 s@," 'immediately return to synch immediate after trigger 'after 1 synch pulse
issue the command s, 'SEE Note on changing Synch modes.
```

## TRIGGER COMMANDS

```
TRTrig 'get trigger time from user and wait for trigger to execute
TRGetTag a$,n 'wait for and get local pickup time from timer tag
get_time 'get other recorded time from PULSAR timer
msg_at 1,1,"Triger at "+trigtim$
msg_at 2,1,"Pickup Recorded at "+a$
msg_at 3,1,"TRGetTag Recorded "+str$(n)+"msec after trigger."
msg_at 4,1,"Get_Time Recorded "+str$(time*1000)+"msec after trigger."
wait key
clear
local
end
```

## Unit Command

Syntax:

```
      +-----+
+-----+|+-----+  +-+ |
| unit +-| unit number +-[|,+-]
+-----+ +-----+  +-+
```

Comments:

Causes all units in the 'unit number' list to be placed in the listen mode before the commands '>' and 'Trigger' are executed. The 'ReadUnit' command will read from the first unit listed. The DFRLoad, XMacLoad and XMacUnLd commands use the first unit listed by the command.

Each macro begins with the unit 1 selected.

Example:

```
unit 1,2,3
>"VOLT LEVEL 67;"
```

A voltage of '67' volts will be applied on units '1', '2' and '3'.

## Wait Command Syntax

Syntax:

```
+-----+ +-----+
| wait +-| seconds +-+
+-----+|+-----+ |
          |+-----+
          +| key +-----+
          +-----+
```

Comments:

Delays program execution for 'seconds' seconds. If any key is struck during the waiting period the test is aborted.

The 'wait key' command will cause the program to pause until any key on the keyboard is struck.

If the 'Esc' key is struck the macro will abort unless an Enter\_at field is in use.

Example:

```
wait 5.5
```

The program will not execute the next command for '5.5' seconds.

```
wait key
```

The program will wait until a key is struck.

## Win\_size Command

Syntax:

```
+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
| win_size +-|top corner+-,+-|left corner+-,+-|columns+-,+-|rows|
+-----+ +-----+ +-+ +-----+ +-+ +-----+ +-+ +-----+
```

Comments:

Defines the location of the window for displaying messages using the 'msg\_at' command. The top left corner of the window fixes the window's location on the screen. The top left corner of the screen is 0,0. If columns and rows are set to 0,0 only the window borders will exist.

**Note: Message window will automatically self-destruct when the first 'non message' window command is executed unless the variable 'no\_purge' is set to 1.**

Example:

```
win_size 5,4,60,10
```

Causes the subsequent message window to have its upper left corners at row 5 and column 4. The window will be 60 characters wide and 10 rows high.



## WinExec Command

Syntax:

```
+-----+ +-----+
| WinExec +----| string expression |
+-----+ +-----+
```

Comments:

Executes a DOS or Windows program specified in the string expression. The macro will start up the program and resume macro execution when the next Windows message is received. The user should include code to pause macro execution if the remainder of the macro requires that the program finishes execution.

Example 1:

```
'running a DOS program - show directory of c:\
no_purge=1      'dont automatically clear msg_at window
msg_at 1,1,"Hit any key to continue with macro..."
WinExec "command.com /c dir c:\ /p"
wait key
```

Example 2:

```
'start up terminal
WinExec "terminal.exe"
```

## XmacLoad Command

Syntax:

```
+-----+ +-----+  
| XmacLoad +[-| pathname |]  
+-----+ +-----+
```

Comments:

Loads the selected PULSAR with a Xmac .EXE.

pathname - this optional string expression specifies the directory or file to be load. If the file cannot be found or no pathname is given then an dialog box for selecting the Xmac .EXE file is opened to let the user select the file.

Example:

```
XmacLoad "xmacs\demo.exe" 'Load the demo Xmac .EXE  
>"xedemo 1,"           'run demo passing 1 as a parameter  
XmacLoad "xmacs\*.exe"  'let user pick the macro
```

## XmacUnLd Command

Syntax:

```
+-----+ +-----+
| XmacUnLd +[-| name |]
+-----+ +-----+
```

Comments:

This command instructs the selected PULSAR to unload an Xmac program.

name - an optional string expression with the name of the Xmac program to unload. If no name is not given then all Xmacs in PULSAR are unloaded.

Example:

```
XmacUnLd                'UnLoad all Xmacs
XmacLoad "xmacs\demo.exe" 'Load the demo .exe Xmac
>"xedemo 1,"           'run demo passing 1 as a parameter
```

## **XMACs**

## **XMACs**

Xmacs were developed to create programs in .EXE format that are downloaded for execution in PULSAR. Special functions have been added to PULSAR to support communications inside and outside of PULSAR and to allow the synchronization and monitoring of the timing activities in the PULSAR unit.

With this facility, program loops, synchronized to the zero crossings of both the phase reference and the output signal, can be created that allow ramping functions to remain synchronized on a per cycle basis.

Also, the user now has the ability to customize PULSARs interface through the use of input and output redirection.

The use of the Wait for Synchronization XMAC is used with MPRT and PULSAR to synchronize to GPS receivers for trigger and playback of faults on opposite ends of a transmission line.

- XMAC Library Functions

- Command Syntax Diagrams

- Creating XMACs

- Testing an XMAC Program

- Downloading a Pulsar XMAC

- XMAC Demo Program

## XMAC Library Functions

## XMAC Library Functions

Xmacs interfaces with PULSAR are supported by a library file of functions and an include file that contains the declarations of each function in the library.

The Xmac library is named "XMAC.LIB" and was built using the two files; "macro.obj", and "macroc.obj". The files "macroc.obj" and "macro.obj" contain the PULSAR Xmac interface functions. Without linking with this library, Xmacs will not execute properly.

### Function List

- XMAC Parser    Calls to the PULSAR command set parser.
- Communications Redirection    Redirection of communication input to the XMAC routine.
- Receive    Redirection of communication input to the XMAC routine.
- Output    Output directly to RS232 and the IEEE 488 interfaces.
- Error Message Display    Generate an error (On the error message generator's LED).
- Gate Status    Reading of the timer post status.
- Synchronization Reset    Places the PULSAR in a know state to apply a Wait for Synchronization command.
- Wait for Synchronization    Places PULSAR in a wait for synchronization events within the PULSAR.
- Sound PULSAR's Horn    Sounding of the system horn.
- No System Echo of Messages    Turning on or off messages generated by the system during macro execution.
- XMAC Read Messages    Reading of internal messages generated by the system (error messages or query responses).

## XMAC Parser

Pass commands to the PULSAR parser: This function is used to pass PULSAR remote commands (Pulsar Command Set) to the PULSAR parser. This function returns when the command is copied to the parser.

### To Call

```
int macro_parser(command_buffer);  
char *command_buffer; Storage location for command. Pointer to command string.
```

### Returns

Nothing.

### Programmer's Notes

The "macro\_read" function can be used to check for any command errors generated by the parser.

Warning, if a command to the PULSAR parser is greater than 4096 characters, parser error messages will be generated.

### EXAMPLE

```
#include <macroinc.h>  
char command_buffer[] = {"v1c1,a15;"};  
  
main()  
{  
    macro_parser(command_buffer);  
}
```

This program uses macro\_parser to send the amplitude command to voltage generator 1.

## Communications Redirection

Re-direction of the communication data to the Xmac routine: This function is used by the Xmac program to intercept commands sent to PULSAR via RS232 or GPIB before they are sent to the parser. This functionality halts all external command processing, and can be used to insure that only the Xmac program's command strings are processed.

### To Call

```
int redirect_input(redirect_code);
int redirect_code; // 0 - no redirect, 1 - redirect.
```

### Returns

NO\_ERROR (0) or TRUE\_ERROR (1).

### Programmer's Notes

The redirect code is 0 (FALSE) to turn off redirection, and 1 (TRUE) to turn on redirection. The default setting for PULSAR is no redirection.

Caution should be taken when using this function, because it prohibits forcing the cancellation of an Xmac through the '%' character.

Warning: A macro must turn off redirection prior to terminating its execution, otherwise, redirection will be in affect the next time the macro executes. If the PULSAR unit is reset by command, redirection will be turned off.

### EXAMPLE

```
#include <macroinc.h>
char command_buffer[] = {"v1c1,a15;"};
main()
{
  int i, c;
  i = redirect_input(1); // turn on redirection
  if (i == NO_ERROR)
  {
    // normal macro execution
    macro_parser(command_buffer);
    c = get_redirect_char();
    if (c != -1)
    // perform the required operation
    redirect_input(0); //redirect off
  }
}
```

This program uses `redirect_input` to turn off the parsing of external commands. Next the program sends its own command to the parser. The program then retrieves a system message, and, finally, the program turns external parsing back on.

## Receive

Receive a character from PULSAR's input buffer: This function is used by an Xmac program to receive a character RS232 or GPIB once the function `redirect_input` has been called to setup the redirection of external characters.

### To Call

```
int get_redirect_char();
int c; //holds the character received
```

### Returns

The current character from PULSAR's input buffer or -1 if no character is present.

### Programmer's Notes

If more than 4096 bytes are received by PULSAR without an Xmac reading a character using "`get_redirect_char`", the system will generate a buffer overflow error message. This is an indicator that an Xmac left the redirect state on.

### EXAMPLE

```
#include <macroinc.h>
main(){int i, j;
j = 0;
i = redirect_input(1);
if (i == NO_ERROR)
{
while (j < 10)
{
if ((i=get_redirect_char()) > 0)
{
redirect_save_buffer[j++] = i;
}
}
redirect_input(0);
}
}
```

This program uses `get_redirect_char` to receive a character from PULSAR's input buffer. The program first sets character redirection by calling function "`redirect_input`". Next the program loops until 10 characters are received from the input buffer. Finally the program resets character redirection.



## Output

Output directly to RS232 or the to the IEEE488 from a Xmac: This function is used to output a message to the external device connected to PULSAR.

### To Call

```
int macro_output(output_buffer);  
char *output_buffer; //Message buffer for output.
```

### Returns

Nothing.

### Programmer's Notes

The message to be output must be an ASCII string terminated by a NULL (0). Binary messages that contain a 0 cannot be transmitted. If GPIB initiated the execution of the macro, messages will be sent via GPIB. If RS232 initiated the execution of the macro, messages will be sent via RS232.

### EXAMPLE

```
#include <macroinc.h>  
char rs232_msg_buffer[] = {"The MACRO is running"};  
main()  
{  
    macro_output(rs232_msg_buffer);  
}
```

This program uses macro\_output to send a message to the external processor via the communication line.

## Error Message Display

Generate an error message on the display generator's LED: This function is used by an Xmac to display a message on the first generator detected by PULSAR. The displayed message will be formatted as x-yy, where x is an integer 8 or 9 and yy is an integer 00 through 99.

### To Call

```
void error_message(section, code);  
int section;      // 1 digit error group code (8 - 9).  
int code;        // 2 digit error code (00 - 99).
```

### Returns

NO\_ERROR (0) or TRUE\_ERROR (1).

### Programmer's Notes

If the section is not an integer '8' or '9' this function returns TRUE\_ERROR. If the code is not an integer between '00' and '99' this function returns TRUE\_ERROR. This function causes no major damage to the PULSAR system. The next command, terminated by a semicolon, sent by the Xmac will return the display generator's LED to its previous state.

### EXAMPLE

```
#include <macroinc.h>  
main()  
{  
    error_message(9,1);  
}
```

This program uses error\_message to send a message to the display generator's LED.

## Gate Status

Read the timer gate response for binding post 1, 2, and 3. Refer to the Instruction Manual for PULSAR or to PulseMaster help for the status codes.

### To Call

```
int macro_gate_status(void);  
int gate_status;
```

### Returns

The timer post status. This function is equivalent to the "qm," command used by PulseMaster to monitor timer post activity.

### Programmer's Notes

None.

### EXAMPLE

```
#include <macroinc.h>  
int gate_status;main()  
{  
    gate_status = macro_gate_status();  
}
```

This program uses macro\_gate\_status to read the timer gate response.

## Synchronization Reset

This function is used to reset the synchronization counter used by the "macro\_sync\_check" function.

### To Call

```
int macro_sync_reset(void);int previous_sync_count;
```

### Returns

The previous synchronization count.

### Programmer's Notes

This default state after a PULSAR reset is unknown. The macro must reset the synch count to place the system and the Xmac program in a known state. The Xmac program is responsible for keeping in synch with the system.

### EXAMPLE

```
#include <macroinc.h>
int previous_sync_count;main()
{
    macro_sync_reset();
}
```

This program uses macro\_sync\_reset to reset the PULSAR synchronization counter.

## Wait for Synchronization

This function is used to monitor synchronization events within the PULSAR. This function should be preceded by the macro synchronization reset function to place the PULSAR unit and the Xmac program in a known state.

### To Call

```
int macro_sync_check(void);
int current_sync_count;
```

### Returns

The current synchronization count.

### Programmer's Notes

None.

### EXAMPLE

```
#include <macroinc.h>
int current_sync_count;
main()
{
    macro_sync_reset();
    //Send commands and wait for count updates
    do
    {
        //perform other macro functions
    }
    while ((current_sync_count = macro_sync_check() < 3));
}
```

This program uses macro\_sync\_check to monitor PULSAR's synchronization counter.

## Sound PULSAR's Horn

This function is used to sound the horn located inside the PULSAR unit.

### To Call

```
void macro_beep_speaker(int number_of_beeps, int pause_between_beeps, int length_of_beeps);  
int number_of_beeps;  
int pause_between_beeps;  
int length_of_beeps;
```

### Returns

Nothing.

### Programmer's Notes

The function does not return until the sounding of the horn has completed. The beep counts and intervals are not controlled by the PULSAR timer.

### EXAMPLE

```
#include <macroinc.h>  
int number_of_beeps;  
int pause_between_beeps;  
int length_of_beeps;  
main()  
{  
    number_of_beeps = 1;  
    pause_between_beeps = 1;  
    length_of_beeps = 1;  
    // signal the end of macro execution  
    macro_beep_speaker(number_of_beeps,pause_beteen_beeps,  
                        length_of_beeps);  
}
```

This program uses macro\_beep\_speaker to signal the end of Xmac execution.

## No System Echo of Messages

This function is used to halt the sending of messages externally. An Xmac should use this when it wishes to completely control the communication from PULSAR to an externally connected source.

### To Call

```
void no_macro_echo(int echo_code);
```

### Returns

Nothing.

### Programmer's Notes

An `echo_code` of 0 (FALSE) is used to allow system message sending to an external source.

An `echo_code` of 1 (TRUE) is used to halt the sending of messages to an external source (such as PulseMaster). The default is the echoing of system messages. An Xmac can read the messages regardless of whether messages are being echoed or not.

### EXAMPLE

```
#include <macroinc.h>
main()
{
    no_macro_echo(0);
    macro_parser("qt,"); // the Xmac reads the time
    no_macro_echo(1);
    macro_parser("qt,"); // the Xmac and the external
        // connection reads the time
}
```

This program uses `no_macro_echo` to allow only the macro to read system messages.

## XMAC Read Messages

This function is used to allow the macro to read system messages. It can be used to check for errors, to read the time from the timer or any other information that can be retrieved from the PULSAR unit.

### To Call

```
int macro_read(char *read_buffer);  
int count_read;
```

### Returns

The number of bytes read.

### Programmer's Notes

This function will read the last message stored in the PULSAR error and status message buffer.

### EXAMPLE

```
#include <macroinc.h>  
char read_buffer[256];  
int count_read;main()  
{  
    no_macro_echo(1); // optional  
    macro_parser("qc,");  
    count_read = macro_read(read_buffer);  
    no_macro_echo(0); // optional  
}
```

This program uses macro\_read to read in the configuration of the PULSAR unit.



## Command Syntax Diagrams

**SYNTAX DIAGRAM** - A syntax diagram is used to explain the correct sequence and content for sending commands to the PULSAR unit. Such diagrams contain boxes, lines, characters, and sometimes arrows.

By default, horizontal lines between boxes indicate logic flow from left to right. A horizontal line with an arrow pointing left indicates logic flow in the opposite direction.

If a vertical line connects to a line with a left arrow, that vertical line indicates a permissible logic choice only in the direction of that arrow. Other vertical lines that have multiple choices toward the right indicate that each of the choices to the right is logically permissible.

**COMMAND CONSTANTS** - Remotely transmitted command constants consist of the upper case characters that appear in the boxes of syntax diagrams as well as the space delimiter. While the boxes employ lower case letters for variables within boxes, lower case letters are allowed for input. Command constant characters are capital letters, slashes, hyphens, and delimiters (comma, semicolon, space, and colon). The following exceptions limit the general rule that command constants must be transmitted exactly as they appear in the boxes.

The underline symbol is never transmitted.

Non-underlined capital letters, slashes, or hyphens (i. e., all non-underlined characters except delimiters) are optional, but if at least one of these optional characters is transmitted, a single space must be transmitted after all optional characters in a box have been transmitted.

Most transmitted misspellings are accepted for optional text in a box, but if a misspelled option begins with a valid choice of a required (i. e., underlined) starting character in a subsequent box, the logic will usually assume that the subsequent box is desired, so that a syntax error will be assumed if the remainder of the command is not fully valid. In practical terms, this means that in the unlikely case of a syntax error due to a misspelling in optional text, correct the spelling.

If only underlined capital letters, slashes, and hyphens are transmitted for characters in a box, the transmission of a single space after them is optional.

Lower case letters may be used instead of capital letters for input.

Carriage return and line feed input characters are discarded as they are received.

**COMMAND VARIABLES** - A command variable is a set of lower case letters that occurs within a box in a syntax diagram. The form and function of each command variable is defined by each command.

**TERMINATOR** - A terminator is a delimiter that is permitted to be the final character of a PULSAR command. The two valid terminators are semicolon and comma whose use is defined as follows.

```
+----+
|  ,  |
+----+
```

Commands ending with a comma are stored until a command ending with semicolon is received. This eliminates the effect of a communication delay between each pair of commands in a lengthy string of commands, so that all commands between semicolons will be executed nearly simultaneously.

```
+----+
|  ;  |
+----+
```

A command (or string of commands separated by commas) which ends with a semicolon terminator will be executed as soon as it can be processed. Semicolons, that are received by the system before the currently loaded events have finished executing, are loaded as wait time commands and, as such, will be dependent on the current Synchronization and Wait mode. Sequences of events are best implemented by loading the entire sequence using a comma to separate each command and then sending the semicolon to initiate the command sequence.

**MULTIPLE CHOICE BEGINNING SYMBOL** - The vertical symbol below is the multiple choice beginning symbol. This shows the starting point for the diagrammed syntax of a command which has more than one choice for a starting command constant.

AVTS Help 092607

[  
[  
[

## Creating XMACS

An Xmac may be created by anyone who is a skilled C programmer. The user can write, compile, link, and download their programs to PULSAR (with either RS232 or the IEEE488 interface), and execute them. The maximum space available for Xmac '.EXE's is 24k.

### Allowed functions

Standard 'C' features (such as loops, conditionals, data structures, user functions, etc.).

Calls to string and floating point (using the alternate math library) library functions (such as STRCAT, ATOI, etc.) .

Calls to the PULSAR Xmac library functions.

### Disallowed functions

Library functions including FILE I/O, memory allocation that invoke DOS.

Auto initialization of global and static pointers and static variables.

### Compiling the PULSAR Xmac

Write the 'C' program including the header file "XMACINC.H" and observing the allowed and disallowed functions. Compile and link with PULSAR libraries as shown using the Microsoft™ C compiler Version 7.0. The batch program, BLDXMAC.BAT, has also been provided that will build the executable.

```
cl /Alfu /Od /FPa /c /Gs /G2 YOURPROG.C
link /NOI /STACK:64 YOURPROG.OBJ,YOURPROG.EXE,
YOURPROG.MAP/map,XMAC.LIB+YOURLIB.LIB
```

All files created for use with PULSAR Xmacs must be compiled using a custom memory model that is essentially the large memory model with separate stack and data segments.

**NOTE: If the .EXE file is not created using the PULSAR Xmac library, "XMAC.LIB", it will not execute properly inside PULSAR.**

## Testing an XMAC Program

The PulseMaster install disks also contain a library named "XMACDEB.LIB" that can be used to test and debug PULSAR Xmacs on a PC prior to loading them into PULSAR. The library consists of the same functions found in the normal Xmac library except not actions are performed. The source for this library has also been included to allow the user to add simulation code.

To create your test program, compile and link with the PULSAR Xmac test library as shown. The batch program, BLDXMAC.BAT, has also been provided that will build the executable "DEBUG.EXE".

```
cl /Alfu /Od /FPa /c /Gs /G2 /Zi YOURPROG.C
```

```
link /NOI /STACK:2048 /CO YOURPROG.OBJ,YOURPROG.EXE,  
YOURPROG.MAP/map,XMACDEB.LIB+YOURLIB.LIB
```

All files created for use with PULSAR Xmacs must be compiled using a custom memory model that is essentially the large memory model with separate stack and data segments.

Once the program is compiled, CodeView can be used to single step through the execution of the program and check its operation.

## Downloading a PULSAR XMAC

Xmacs are downloaded to PULSAR in binary format.

To transfer an Xmac .EXE to PULSAR use PulseMaster to download the .EXE , or issue the following command to PULSAR:

```
XMAC LOAD MACNAME, (Macro Commands)
```

If PulseMaster is not available, then use DOS COPY or any other method that supports binary transfers, without protocol, to write the file to the serial port:

```
DOS> COPY YOURPROG.EXE COM1 /B
```

Then terminate the transfer by sending a comma.

## **XMAC Demo Program**

This demo program is supplied to illustrate the use of PULSAR Xmac Library functions in a Xmac program. The demo program can be used as a guide for writing Xmacs in C. The Xmac starts at the line "#include <stdio.h>", and ends at the "}" in the function "process\_function".

Following the statement "#include <stdio.h>", are the global data and definitions for the functions used by the Xmac. The Xmac execution begins at the function "void main(int argc, char \*argv[])", and illustrates how an Xmac can be setup to receive and process arguments. The function "main" calls "int process\_function," which performs demonstrations of each Xmac Library function.

The global definitions used in this Xmac are used to support the different execution paths of the Xmac. The parameters passed to the macro are the numbers '1' thru '9' and the letters 'A' and 'B'. Although single character parameter passing is shown in this example, text strings may also be passed to Xmacs. The maximum number of parameters that can be passed to a macro is 15 and the combined string must of Xmac name and parameters cannot be greater than 128 characters.

```

#include "stdio.h"
#include "string.h"
#include "stdlib.h"
#include "macroinc.h"

unsigned char mread_buf[128];
unsigned char redirect_buffer[128];
unsigned char redirect_message_buffer[2];
unsigned char invalid_cmd[] = {"Option not available"};
unsigned char buffer[50];

unsigned char command_buffer[] =
{"v0c0,walmi,sp;v1,a15,tr,tsosta,on,waic2,of,tsosta"};

void main(int argc, char **argv);
int process_function(char *ptr);

void main(int argc, char *argv[])
{
    char *ptr;
    if (--argc) // check how many arguments to process
    {
        do
        {
            if ((ptr = **++argv) == '\0') // retrieve the arguments
                break;
            process_function(ptr++); // process the functional request
        } while (--argc);
    }
}

int process_function(char *ptr)
{
    int char_received, i, receive_cnt;
    switch(*ptr) {
        case '1':
            macro_parser(command_buffer); // send a command to PULSAR
            break;
        case '2':
            redirect_input(1); // turn on input redirection
            for (;;)
                // wait for a character to be received
                if ((char_received = get_redirect_char()) != -1)
                    break;
            redirect_input(0); // turn off redirection
            break;
        case '3':
            receive_cnt = 0;
            redirect_input(1); // turn on redirection
            for (;;)
                // wait for 10 characters to be received
                if ((char_received = get_redirect_char()) != -1)
                    if (receive_cnt == 9)
                        // save the last character received and echo the message
                        // back to the sender
                        buffer[receive_cnt++] = char_received;
                    else
                        // saving first 9 characters received
                        buffer[receive_cnt++] = char_received;
                    else
                        break;
            redirect_input(0); // turn redirection off
            break;
        case '4':
            // output a message externally
            macro_output("The MACRO is running");
            break;
        case '5':
            // inform the external connection of an error condition
            error_message(9,1);
            break;
        case '6':
            // read the timer monitor post activity
            i = get_status();
            redirect_message_buffer[1] = '\0';
            redirect_message_buffer[0] = ((i | 0x30) & 0x7f);
            macro_output(redirect_buffer); // send a message externally.
            break;
        case '7':
            macro_sync_reset(); // reset the PULSAR synchronization counter
            break;
        case '8':
            macro_parser("::"); // generate 2 syncs
            for (;;)
                if ((macro_sync_check()) >= 2) // wait for two syncs to occur
                    break;
            break;
        case '9':
            // tell the operator that the test has completed
            // by sounding the PULSAR horn
            macro_beep_speaker(1,1,1);
            macro_beep_speaker(2,5,2);
            break;
        case 'A':
            no_macro_echo(1); // no external messages
            for (i = 0; i < 128; i++)
                mread_buf[i] = 0;
            macro_parser("qs,"); // check wait timer status
            macro_read(mread_buf,128); // read in the response
            no_macro_echo(0); // allow external messages
            printf(mread_buf, "\nMessage Received");
            macro_output(mread_buf); // send a message externally
            break;
        case 'B':
            // This is a demonstration of generating message
            // from within a macro, reading a message from
            // from PULSAR, and sending what was read externally.
            for (i = 0; i < 128; i++)
                mread_buf[i] = 0;
            macro_parser("qt,");
            macro_read(mread_buf,128);
            macro_output(mread_buf);
            break;
        default:
            // used for an invalid parameter sent to the macro
            error_message(8,2);
            macro_output(invalid_cmd);
            break;
    }
}

```





## **Appendix D - IEEE Codes**

### **Appendix D - IEEE Codes**

The IEEE Codes for devices used and controlled by other devices in control circuits are listed in the following topics

- IEEE Device Numbers
- IEEE Devices Performing More Than One Function
- IEEE Suffix Numbers
- IEEE Suffix Letters
- IEEE Auxiliary Devices
- IEEE Actuating Quantities
- IEEE Main Device Parts
- IEEE Standard Reference Positions
- IEEE Other Switches
- IEEE Representation of Devices on Diagrams

## ANSI/IEEE Device Numbers

**1 MASTER ELEMENT** is the initiating device, such as a control switch, voltage relay, float switch, etc., which serves either directly, or through such permissive devices as protective and time-delay relays to place an equipment in or out of operation.

**2 TIME-DELAY STARTING, OR CLOSING RELAY** is a device which functions to give a desired amount of time delay before or after any point of operation in a switching sequence or protective relay system, except as specifically provided by device functions 48, 62, and 79 described later.

**3 CHECKING OR INTERLOCKING RELAY** is a device which operates in response to the position of a number of other devices, (or to a number of predetermined conditions), in an equipment, to allow an operating sequence to proceed, to stop, or to provide a check of the position of these devices or of these conditions for any purpose.

**4 MASTER CONTACTOR** is a device, generally controlled by device No. 1 or equivalent, and the required permissive and protective devices, that serves to make and break the necessary control circuits to place an equipment into operation under the desired conditions and to take it out of operation under other or abnormal conditions.

**5 STOPPING DEVICE** is a control device used primarily to shut down an equipment and hold it out of operation. [This device may be manually or electrically actuated, but excludes the function of electrical lockout (see device function 86) on abnormal conditions.]

**6 STARTING CIRCUIT BREAKER** is a device whose principal function is to connect a machine to its source of starting voltage.

**7 ANODE CIRCUIT BREAKER** is one used in the anode circuits of a power rectifier for the primary purpose of interrupting the rectifier circuit if an arc back should occur.

**8 CONTROL POWER DISCONNECTING DEVICE** is a disconnective device – such as a knife switch, circuit breaker or pullout fuse block, used for the purpose of connecting and disconnecting the source of control power to and from the control bus or equipment.

**Note:** Control power is considered to include auxiliary power which supplies such apparatus as small motors and heaters.

**9 REVERSING DEVICE** is used for the purpose of reversing a machine field or for performing any other reversing function.

**10 UNIT SEQUENCE SWITCH** is used to change the sequence in which units may be placed in and out of service in multiple-unit equipments.

**11** Reserved for future application.

**12 OVER-SPEED DEVICE** is usually a direct-connected speed switch which functions on machine over-speed.

**13 SYNCHRONOUS-SPEED DEVICE**, such as a centrifugal-speed switch, a slip-frequency relay, a voltage relay, an undercurrent relay or any type of device, operates at approximately synchronous speed of a machine.

**14 UNDER-SPEED DEVICE** functions when the speed of a machine falls below a predetermined value.

**15 SPEED OR FREQUENCY, MATCHING DEVICE** functions to match or hold the speed or the frequency of a machine or of a system equal to, or approximately equal to, that of another machine, source or system.

**16** Reserved for future application.

**17 SHUNTING, OR DISCHARGE, SWITCH** serves to open or to close a shunting circuit around any piece of apparatus (except a resistor), such as a machine field, a machine armature, a capacitor or a reactor.

**Note:** This excludes devices which perform such shunting operations as may be necessary in the process of starting a machine by devices 6 or 42, or their equivalent, and also excludes device 73 function which serves for the switching of resistors.

**18 ACCELERATING OR DECELERATING DEVICE** is used to close or to cause the closing of circuits which are used to increase or to decrease the speed of a machine.

**19 STARTING-TO-RUNNING TRANSITION CONTACTOR** is a device which operates to initiate or cause the automatic transfer of a machine from the starting to the running power connection.

**20 ELECTRICALLY OPERATED VALVE** is an electrically operated, controlled or monitored valve in a fluid line.

**Note:** The function of the valve may be indicated by the use of the suffixes defined at this paper end.

**21 DISTANCE RELAY** is a device which functions when the circuit admittance, impedance or reactance increases or decreases beyond predetermined limits.

**22 EQUILIZER CIRCUIT BREAKER** is a breaker which serves to control or to make and break the equalizer or the current-balancing connections for a machine field, or for regulating equipment, in a multiple-unit installation.

**23 TEMPERATURE CONTROL DEVICE** functions to raise or lower the temperature of a machine or other apparatus, or of any medium, when its temperature falls below, or rises above a predetermined value.

**Note:** An example of a thermostat which switches on a space heater in a switchgear assembly when the temperature falls to a desired value as distinguished from a device which is used to provide automatic temperature regulation between close limits and would be designated as 90T.

**24** Reserved for future application.

**25 SYNCHRONIZING, OR SYNCHRONISM-CHECK DEVICE** operates when two ac circuits are within the desired limits of frequency, phase angle or voltage, to permit or to cause the paralleling of these two circuits.

**26 APPARATUS THERMAL DEVICE** functions when the temperature of the shunt field or the armature winding of a machine, or that of a load limiting or load shifting resistor or of a liquid or other medium exceeds a predetermined value; or if the temperature of the protected apparatus, such as a power rectifier, or of any medium decreases below a predetermined value.

**27 UNDERVOLTAGE RELAY** is a device which functions on a given value of undervoltage.

**28 FLAME DETECTOR** is a device that monitors the presence of the pilot or main flame in such apparatus as a gas turbine or a steam boiler.

**29 ISOLATING CONTACTOR** is used expressly for disconnecting one circuit from another for the purposes of emergency operation, maintenance, or test.

**30 ANNUNCIATOR RELAY** is a non-automatically reset device that gives a number of separate visual indications upon the functioning of protective devices, and which may also be arranged to perform a lockout function.

**31 SEPARATE EXCITATION DEVICE** connects a circuit such as the shunt field of a synchronous converter, to a source of separate excitation during the starting sequence; or one which energizes the excitation and ignition circuits of a power rectifier.

**32 DIRECTIONAL POWER RELAY** is one which functions on a desired value of power flow in a given direction, or upon reverse power resulting from arc back in the anode or cathode circuits of a power rectifier.

**33 POSITION SWITCH** makes or breaks contact when the main device or piece of apparatus, which has no device function number, reaches a given position.

**34 MASTER SEQUENCE DEVICE** is a device such as a motor-operated multi-contact switch, or the equivalent, or a programming device, such as a computer, that establishes or determines the operating sequence of the major devices in an equipment during starting and stopping or during other sequential switching operations.

**35 BRUSH-OPERATING, OR SLIP-RING-SHORT-CIRCUITING, DEVICE** is used for raising, lowering, or shifting the brushes of a machine, or for short-circuiting its slip rings, or for engaging or disengaging the contacts of a mechanical rectifier.

**36 POLARITY OR POLARIZING VOLTAGE DEVICE** operates or permits the operation of another device on a predetermined polarity only or verifies the presence of a polarizing voltage in an equipment.

**37 UNDERCURRENT OR UNDERPOWER RELAY** functions when the current or power flow decreases below a predetermined value.

**38 BEARING PROTECTIVE DEVICE** functions on excessive bearing temperature, or on other abnormal mechanical conditions, such as undue wear, which may eventually result in excessive bearing temperature.

**39 MECHANICAL CONDITION MONITOR** is a device that functions upon the occurrence of an abnormal mechanical condition (except that associated with bearings as covered under device function 38), such as excessive vibration, eccentricity, expansion, shock, tilting, or seal failure..

**40 FIELD RELAY** functions on a given or abnormally low value or failure of machine field current, or on an excessive value of the reactive component of armature current in an ac machine indicating abnormally low field excitation.

**41 FIELD CIRCUIT BREAKER** is a device which functions to apply, or to remove, the field excitation of a machine.

**42 RUNNING CIRCUIT BREAKER** is a device whose principal function is to connect a machine to its source of running or operating voltage. This function may also be used for a device, such as a contactor, that is used in series with a circuit breaker or other fault protecting means, primarily for frequent opening and closing of the circuit.

**43 MANUAL TRANSFER OR SELECTOR DEVICE** transfers the control circuits so as to modify the plan of operation of the switching equipment or of some of the devices.

**44 UNIT SEQUENCE STARTING RELAY** is a device which functions to start the next available unit in a multiple-unit equipment on the failure or on the non-availability of the normally preceding unit.

**45 ATMOSPHERIC CONDITION MONITOR** is a device that functions upon the occurrence of an abnormal atmospheric condition, such as damaging fumes, explosive mixtures, smoke, or fire.

**46 REVERSE-PHASE, OR PHASE-BALANCE, CURRENT RELAY** is a relay which functions when the polyphase currents are of reverse-phase sequence, or when the polyphase currents are unbalanced or contain negative phase-sequence components above a given amount.

**47 PHASE-SEQUENCE VOLTAGE RELAY** functions upon a predetermined value of polyphase voltage in the desired phase sequence.

**48 INCOMPLETE SEQUENCE RELAY** is a relay that generally returns the equipment to normal, or off, position and locks it out if the normal starting, operating or stopping sequence is not properly completed within a predetermined time. If the device is used for alarm purposes only, it should preferably be designated as 48A (alarm).

**49 MACHINE, OR TRANSFORMER, THERMAL RELAY** is a relay that functions when the temperature of a machine armature, or other load carrying winding or element of a machine, or the temperature of a power rectifier or power transformer (including a power rectifier transformer) exceeds a predetermined value.

**50 INSTANTANEOUS OVERCURRENT, OR RATE-OF-RISE RELAY** is a relay that functions instantaneously on an excessive value of current, or on an excessive rate of current rise, thus indicating a fault in the apparatus or circuit being protected.

**51 AC TIME OVERCURRENT RELAY** is a relay with either a definite or inverse time characteristic that functions when the current in an ac circuit exceeds a predetermined value.

**52 AC CIRCUIT BREAKER** is a device that is used to close and interrupt an ac power circuit under normal conditions or to interrupt this circuit under fault or emergency conditions.

**53 EXCITER OR DC GENERATOR RELAY** is a relay that forces the dc machine field excitation to build up during starting or which functions when the machine voltage has built up to a give value.

**54** Reserved for future application.

**55 POWER FACTOR RELAY** is a relay that operates when the power factor in an ac circuit rises above or below a predetermined value.

**56 FIELD APPLICATION RELAY** is a relay that automatically controls the application of the field excitation to an ac motor at some predetermined point in the slop cycle.

**57 SHORT-CIRCUITING OR GROUNDING DEVICE** is a primary circuit switching device that functions to short circuit or to ground a circuit in response to automatic or manual means.

**58 RECTIFICATION FAILURE RELAY** is a device that functions if one or more anodes of a power rectifier fail to fire, or to detect an arc-back or on failure of a diode to conduct or bloc properly.

**59 OVERVOLTAGE RELAY** is a relay that functions on a given value of overvoltage.

**60 VOLTAGE OR CURRENT BALANCE RELAY** is a relay that operates on a given difference in voltage, or current or output of two circuits.

**61** Reserved for future application.

**62 TIME-DELAY STOPPING OR OPENING RELAY** is a time-delay relay that serves in conjunction with the device that initiates the shutdown, stopping, or opening operation in an automatic sequence.

**63 PRESSURE SWITCH** is a switch which operates on given values or on a given rate of change of pressure.

**64 GROUND PROTECTIVE RELAY** is a relay that functions on failure of the insulation of a machine, transformer or of other apparatus to ground, or on flashover of a dc machine to ground.

Note: This function is assigned only to a relay which detects the flow of current from the frame of a machine or enclosing case or structure of a piece of apparatus to ground, or detects a ground on a normally ungrounded winding or circuit. It is not applied to a device connected in the secondary circuit or secondary neutral of a current transformer, or in the secondary neutral of a current transformer, connected in the power circuit of a normally grounded system.

**65 GOVERNOR** is the assembly of fluid, electrical, or mechanical control equipment used for regulating the flow of water, steam, or other medium to the prime mover for such purposes as starting, holding speed or load, or stopping.

**66 NOTCHING OR JOGGING DEVICE** functions to allow only a specified number of operations of a given device, or equipment, or a specified number of successive operations within a given time of each other. It also functions to energize a circuit periodically or for fractions of specified time intervals, or that is used to permit intermittent acceleration or jogging of a machine at low speeds for mechanical positioning.

**67 AC DIRECTIONAL OVERCURRENT RELAY** is a relay that functions on a desired value of ac overcurrent flowing in a predetermined direction.

**68 BLOCKING RELAY** is a relay that initiates a pilot signal for blocking of tripping on external faults in a transmission line or in other apparatus under predetermined conditions, or cooperates with other devices to block tripping or to block reclosing on an out-of-swing condition or on power swings.

**69 PERMISSIVE CONTROL DEVICE** is generally a two-position, manually operated switch that in one position permits the closing of a circuit breaker, or the placing of an equipment into operation, and in the other position prevents the circuit breaker or the equipment from being operated.

**70 RHEOSTAT** is a variable resistance device used in an electric circuit, which is electrically operated or has other electrical accessories, such as auxiliary, position, or limit switches.

**71 LEVEL SWITCH** is a switch which operates on given values, or on a given rate of change, of level.

**72 DC CIRCUIT BREAKER** is used to close and interrupt a dc power circuit under normal conditions or to interrupt this circuit under fault or emergency conditions.

**73 LOAD-RESISTOR CONTACTOR** is used to shunt or insert a step of load limiting, shifting, or indicating resistance in a power circuit, or to switch a space heater in a circuit, or to switch a light, or regenerative load resistor of a power rectifier or other machine in and out of circuit.

**74 ALARM RELAY** is a device other than an annunciator, as covered under device No. 30, which is used to operate, or to operate in connection with, a visual or audible alarm.

**75 POSITION CHANGING MECHANISM** is a mechanism that is used for moving a main device from one position to another in an equipment; as for example, shifting a removable circuit breaker unit to and from the connected, disconnected, and test positions.

**76 DC OVERCURRENT RELAY** is a relay that functions when the current in a dc circuit exceeds a given value.

**77 PULSE TRANSMITTER** is used to generate and transmit pulses over a telemetering or pilot-wire circuit to the remote indicating or receiving device.

**78 PHASE ANGLE MEASURING, OR OUT-OF-STEP PROTECTIVE RELAY** is a relay that functions at a predetermined phase angle between two voltages or between two currents or between voltage and current.

**79 AC RECLOSING RELAY** is a relay that controls the automatic reclosing and locking out of an ac circuit interrupter.

**80 FLOW SWITCH** is a switch which operates on given values, or on a given rate of change, of flow.

**81 FREQUENCY RELAY** is a relay that functions on a predetermined value of frequency – either under or over or on normal system frequency – or rate of change of frequency.

**82 DC RECLOSING RELAY** is a relay that controls the automatic closing and reclosing of a dc circuit interrupter, generally in response to load circuit conditions.

**83 AUTOMATIC SELECTIVE CONTROL OR TRANSFER RELAY** is a relay that operates to select automatically between certain sources or conditions in a equipment, or performs a transfer operation automatically.

**84 OPERATING MECHANISM** is the complete electrical mechanism or servo-mechanism, including the operating motor, solenoids, position switches, etc. for a tap changer, induction regulator or any similar piece of apparatus which has no device function number.

**85 CARRIER OR PILOT-WIRE RECEIVER RELAY** is a relay that is operated or restrained by a signal used in connection with carrier-current or dc pilot-wire fault directional relaying.

**86 LOCKING-OUT RELAY** is an electrically operated, hand or electrically reset, relay that functions to shut down and hold an equipment out of service on the occurrence of abnormal conditions.

**87 DIFFERENTIAL PROTECTIVE RELAY** is a protective relay that functions a percentage or phase angle or other quantitative difference of two currents or of some other electrical quantities.

**88 AUXILIARY MOTOR OR MOTOR GENERATOR** is one used for operating auxiliary equipment such as pumps, blowers, exciters, rotating magnetic amplifiers, etc.

**89 LINE SWITCH** is used as a disconnecting load-interrupter, or isolating switch in an ac or dc power circuit, when this device is electrically operated or has electrical accessories, such as an auxiliary switch, magnetic lock, etc.

**90 REGULATING DEVICE** functions to regulate a quantity, or quantities, such as voltage, current, power, speed, frequency, temperature, and load, at a certain value or between certain (generally close) limits for machines, tie lines or other apparatus.

**91 VOLTAGE DIRECTIONAL RELAY** is a relay that operates when the voltage across an open circuit breaker or contactor exceeds a given value in a given direction.

**92 VOLTAGE AND POWER DIRECTIONAL RELAY** is a relay that permits or causes the connection of two circuits when the voltage difference between them exceeds a given value in a predetermined direction and causes these two circuits to be disconnected from each other when the power flowing between them exceeds a given value in the opposite direction.

**93 FIELD CHANGING CONTACTOR** functions to increase or decrease in one step the value of field excitation on a machine.

**94 TRIPPING OR TRIP-FREE RELAY** functions to trip a circuit breaker, contactor, or equipment, or to permit immediate tripping by other devices; or to prevent immediate reclosure of a circuit interrupter, in case it should open automatically even though its closing circuit is maintained closed.

**95** Used only for specific applications on individual installations where none of the assigned numbered functions from 1 to 94 is suitable.

**96** Used only for specific applications on individual installations where none of the assigned numbered functions from 1 to 94 is suitable.

**97** Used only for specific applications on individual installations where none of the assigned numbered functions from 1 to 94 is suitable.

**Note:** This document was taken from the Westinghouse publication:

Applied Protective Relaying  
A New "Silent Sentinels" Publication  
By Westinghouse Electric Corporation  
Relay-Instrument Division  
Newark, New Jersey  
Circa 1960's

## IEEE Devices Performing More Than One Function

### Devices Performing More Than One Function

If one device performs two relatively important functions in an equipment so that it is desirable to identify both of these functions, this may be done by using a double function number and name such as:

#### 50/51 Instantaneous and Time Overcurrent Relay.

Note: This document was taken from the Westinghouse publication:

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's

## IEEE Suffix Numbers

### Suffix Numbers

If two or more devices with the same function number and suffix letter (if used) are present in the same equipment, they may be distinguished by numbered suffixes as for example, 52X-1 52X-2 and 52X-3, when necessary.

**Note:** This document was taken from the Westinghouse publication:

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's .



## IEEE Suffix Letters

### Suffix Letters

Suffix letters are used with device function numbers for various purposes. In order to prevent possible conflict, each suffix letter should have only one meaning in an individual equipment. All other words should use the abbreviations as contained in ANSI Y1.1 latest revision, or should use some other distinctive abbreviation, or be written out in full each time they are used. The meaning of each single suffix letter, or combination of letters, should be clearly designated in the legend on the drawing or publications applying to the equipment.

Lower case (small) suffix letters are used in practically all instances on electrical diagrams for the auxiliary, position, and limit switches. Capital letters are generally used for all other suffix letters.

The letters should generally form part of the device function designation, are usually written directly after the device function number; as for example, 52CS, 71W, or 49D. When it is necessary to use two types of suffix letters in connection with one function number, it is often desirable for clarity to separate them by a slanted line or dash, as for example, 20D/CS or 20D-CS.

The suffix letters which denote parts of the main device, and those which cannot or need not form part of the device function designation, are generally written directly below the device function number on drawings, as for example.

Note: This document was taken from the Westinghouse publication:

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's

## IEEE Auxiliary Devices

### Auxiliary Devices

#### Separate Auxiliary Devices

X

Y --Auxiliary relay(1)

Z

R --Raising relay

L --Lowering relay

O --Opening relay or contactor

C --Closing relay or contactor

CS --Control switch

CL --Auxiliary Relay, open (energized when main device is in open position)

OP --Auxiliary Relay, Open (energized when main device is in open position)

U --"Up" position-switch relay

D --"Down" position-switch relay

PB --Push button

(1)In the control of a circuit breaker with so-called X-Yrelay control scheme, the X relay is the device whose main contacts are used to energize the closing coil or the device which in some other manner, such as by the release of stored energy, causes the breaker to close. The contacts of the Y relay provide the anti-pump feature for the circuit breaker.

**Note:** This document was taken from the Westinghouse publication:

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's

## IEEE Actuating Quantities

### Actuating Quantities

These letters indicate the condition or electrical quantity to which the device responds, or the medium in which it is located, such as:

- A --Air, or Amperes or Alternating
- C --Current
- D --Direct or Discharge
- E --Electrolyte
- F --Frequency, or Flow or Fault
- H --Explosive
- J --Differential
- L --Level, or Liquid
- P --Power Factor
- PF --Oil
- S --Speed or Suction or Smoke
- T --Temperature
- V --Voltage, Volts, or Vacuum
- VAR --Reactive Power
- VB --Vibration
- W --Water, or Watts
- Main Devices

These letters denote the location of the main device in the circuit, or the type of circuit in which the device is used or the type of circuit or apparatus with which it is associated, when this is necessary, such as:

- A --Alarm or Auxiliary Power
- AN --Anode
- B --Battery, or Blower, or Bus
- BK --Brake
- BL --Block (Valve)
- BP --Bypass
- BT --Bus Tie
- C --Capacitor, or Condenser, Compensator, or Carrier Current or Case or Compressor
- CA --Cathode
- CH --Check (Valve)
- D --Discharge (Valve)
- E --Exciter
- F --Feeder, or Field, or Filament, or Filter, or Fan
- G --Generator, or Ground(2)
- H --Heater, or Housing
- L --Line or Logic
- M --Motor, or Metering
- N --Network, or Neutral(2)

P --Pump or Phase Comparison  
R --Reactor, or Rectifier, or Room  
S --Synchronizing or Secondary or Strainer or Sump or Suction (Valve)  
T --Transformer, or Thyatron  
TH --Transformer (high-voltage side)  
TL --Transformer (low-voltage side)  
TM --Telemeter  
U --Unit

(2)Suffix "N" is generally used in preference to "G" for devices connected in the secondary neutral of current transformers, or in the secondary of a current transformer whose primary winding is located in the neutral of a machine or power transformer, except in the case of transmission line relaying, where the suffix "G" is more commonly used for those relays which operate on ground faults.

**Note:** This document was taken from the Westinghouse publication:

Applied Protective Relaying  
A New "Silent Sentinels" Publication  
By Westinghouse Electric Corporation  
Relay-Instrument Division  
Newark, New Jersey  
Circa 1960's

## IEEE Main Device Parts

### Main Device Parts

These letters denote parts of the main device, divided in the two following categories:

1. All parts, except auxiliary contacts, position switches, limit switches, and torque limit switches.

BK --Brake

C --Coil, or Condenser, or Capacitor

CC --Closing Coil

HC --Holding Coil

M --Operating Motor

MF --Fly-Ball Motor

ML --Load-limit Motor

MS --Speed adjusting, or Synchronizing, Motor

S --Solenoid

SI --Seal-in

TC --Trip Coil

V --Valve

2. All auxiliary contacts and position and limit switches for such devices and equipment as circuit breakers, contactors, valves and rheostats and contacts of relays. These are designated as follows:

a --Contact that is open when the main device is in the standard reference position, commonly referred to as the non-operated or de-energized position, and that closes when the device assumes the opposite position

b --Contact that is closed when the main device is in the standard reference position, commonly referred to as the non-operated or de-energized position, and that opens when the device assumes the opposite position.

aa --Contact that is open when the operating mechanism of the main device is in the non-operated position and that closes when the operating mechanism assumes the opposite position.

Bb --Contact that is closed when the operating mechanism of the main device is in the non-operated position and that opens when the operating mechanism assumes the opposite position.

**Note: This document was taken from the Westinghouse publication:**

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's

## IEEE Standard Reference Positions

**Standard reference positions of some typical devices are as follows:**

Device Standard Reference Position  
Power Circuit Breaker Main Contacts Open  
Disconnecting Switch Main Contacts Open  
Load-break Switch Main Contacts Open  
Valve Closed Position  
Gate Closed Position  
Clutch Disengaged Position  
Turning Gear Disengaged Position  
Power Electrodes Maximum Gap Position  
Rheostat Maximum Resistance Position  
Adjusting Means(1) Low or Down Position  
Relay(2) De-energized Position  
Contactor(2) De-energized Position  
Relay (latched-in type) See 2-9.7.2 (C37.2 – 1970)  
Contactor (latched-in type) Main Contacts Open  
Temperature Relay(3) Lowest Temperature  
Level Detector(3) Lowest Level  
Flow Detector(3) Lowest Flow  
Speed Switch(3) Lowest Speed  
Vibration Detector(3) Minimum Vibration  
Pressure Switch(3) Lowest Pressure  
Vacuum Switch(3) Lowest Pressure; i.e., Highest Vacuum

Note: If several similar auxiliary switches are present on the same device, they should be designated numerically 1, 2, 3, etc. when necessary.

(1)These may be speed, voltage, current, load, or similar adjusting devices comprising rheostats, springs, levers, or other components for the purpose.

(2)These electrically operated devices are of the non-latched-in type, whose contact position is dependent only upon the degree of energization of the operating or restraining or holding coil or coils which may or may not be suitable for continuous energization. The de-energized position of the device is that with all coils de-energized.

(3)The energizing influences for these devices are considered to be, respectively, rising temperature, rising level, increasing flow, rising speed, increasing vibration, and increasing pressure.

The simple designation "a" or "b" is used in all cases where there is no need to adjust the contacts to change position at any particular point in the travel of the main device or where the part of the travel where the contacts change position is of no significance in the control or operating scheme. Hence, the "a" and "b" designations usually are sufficient for circuit breaker auxiliary switches.

**Note: This document was taken from the Westinghouse publication:  
Applied Protective Relaying  
A New "Silent Sentinels" Publication  
By Westinghouse Electric Corporation**

Relay-Instrument Division  
Newark, New Jersey  
Circa 1960's

## IEEE Other Switches

### Other Switches

These letters cover all other distinguishing features or characteristics or conditions, which serve to describe the use of the device or its contacts in the equipment such as:

- A --Accelerating, or Automatic
  - B --Blocking, or Back-up
  - C --Close, or Cold
  - D --Decelerating, Detonate, or Down, or Disengaged
  - E --Emergency or Engaged
  - F --Failure, or Forward
  - H --Hot, or High
  - HR --Hand Reset
  - HS --High Speed
  - L --Left, or Local, or Low, or Lower, or Leading
  - M --Manual
  - OFF --Off
  - ON --On
  - P --Polarizing
  - R --Right, or Raise, or Reclosing, or Receiving, or Remote, or Reverse
  - S --Sending, or Swing
  - T --Test, or Trip, or Trailing
  - TDC --Time-delay Closing
  - TDO --Time-delay Opening
  - U --Up
- Note: This document was taken from the Westinghouse publication:

Applied Protective Relaying  
A New "Silent Sentinels" Publication  
By Westinghouse Electric Corporation  
Relay-Instrument Division  
Newark, New Jersey  
Circa 1960's



## IEEE Representation of Devices on Diagrams

### Representation of Device Contacts on Electrical Diagrams

On electrical diagrams the "b" contacts of all devices, including those of relays and those with suffix letters or percentage figures, should be shown as closed contacts, and all "a" contacts should be shown as open contacts.

For those devices that have no de-energized or non-operated position, such as manually-operated transfer or control switches (including those of the spring-return type) or auxiliary position indicating contacts on the housings or enclosures of a removable circuit breaker unit, the preferred method of representing these contacts is as an "a" switch. Each contact should, however, be identified on the elementary diagram as to when it closes.

In the case of latched-in or hand-reset locking-out relays, which operate from protective devices to perform the shut-down of an equipment and to hold it out of service, the contacts should preferably be shown in the normal non-locking-out position. In general, any devices, such as electrically-operated latched-in relays, which have no de-energized or non-operated position, and have not been specifically covered in the above paragraphs should have their contacts shown in the position most suitable for the ready understanding of the operation of the devices in the equipment, and sufficient description should be present, as necessary, on the elementary diagram to indicate the contact operation.

**Note:** This document was taken from the Westinghouse publication:

Applied Protective Relaying

A New "Silent Sentinels" Publication

By Westinghouse Electric Corporation

Relay-Instrument Division

Newark, New Jersey

Circa 1960's



## **Appendix E - AVTS & MPRT Information & Web Links**

## **Appendix E - AVTS & MPRT Information & Web Links**

Product information for AVTS and MPRT appear in the following topics

AVTS Information topic

MPRT Information topic

The free Adobe Reader 7.0.8, or the latest version available, for all Windows operating systems and in any language is located on the web site [http://www.adobe.com/products/acrobat/readstep2\\_allversions.html](http://www.adobe.com/products/acrobat/readstep2_allversions.html)

Adobe maintains this site and is updated as needed.

## AVTS Information

An AVTS Software Users Forum is available to all users of the AVTS Software.

The **General Information** is found on <http://www.megger.com/us/story/Index.php?ID=174>

The location to join in on **the Forum** is found on

Available **Relay Test Modules**, in addition to those included with the installation of AVTS, are found on <http://www.megger.com/us/login/Index.php?status=1> and then select the first line "Click here for a complete listing of available modules." The option module list page will be displayed.

**Application Guides** for the use of AVTS are in the AVTS Information subfolder.

**Microsoft Visual Basic Help** file for using Script with the 1-Touch Control and in the BASIC tab is in the AVTS Information subfolder.

The **Modbus Application Protocol Standard** document is in the AVTS Information subfolder.

The **Converted PulseMaster Modules** are in the AVTS Information subfolder.

## MPRT Information

The following links are for additional information about the MPRT Relay Test Set.

These files are for the information in a AVTS installed folder.

The **English language MPRT User Manual** is available in Adobe pdf format in the AVTS subfolder MPRT Information.

The **French language MPRT User Manual** is available in Adobe pdf format in the AVTS subfolder MPRT Information.

The **German language MPRT User Manual** is available in Adobe pdf format in the AVTS subfolder MPRT Information.

The **Spanish language MPRT User Manual** is available in Adobe pdf format in the AVTS subfolder MPRT Information.

These links will access the information directly to a Megger web site.

The **MPRT Protective Relay Test Set** general information site is  
<http://www.megger.com/us/story/Index.php?ID=61>

The **MPRT Specifications English Data Sheet** is available in Adobe pdf format and available in the AVTS subfolder MPRT Information.

[http://www.megger.com/us/products/OpenDoc.php?FileName=MPRT\\_DS\\_en\\_V17.pdf&ProductID=567&DocType=DataSheet&PermissionLevel=a&Site=us&ShowDocType=DataSheet&ref=/us/products/ProductDetails.php?ID=567&Description=mprt&ModelFamily=MPRT&BusinessSectorCode=P&Session=1:43:11PM](http://www.megger.com/us/products/OpenDoc.php?FileName=MPRT_DS_en_V17.pdf&ProductID=567&DocType=DataSheet&PermissionLevel=a&Site=us&ShowDocType=DataSheet&ref=/us/products/ProductDetails.php?ID=567&Description=mprt&ModelFamily=MPRT&BusinessSectorCode=P&Session=1:43:11PM)

The **MPRT Specifications Spanish Data Sheet** is available in Adobe pdf format and available in the AVTS subfolder MPRT Information.

[http://www.megger.com/us/products/OpenDoc.php?FileName=MPRT\\_DS\\_sp\\_V01.pdf&ProductID=567&DocType=DataSheet&PermissionLevel=a&Site=us&ShowDocType=DataSheet&ref=/us/products/ProductDetails.php?ID=567&Description=mprt&ModelFamily=MPRT&BusinessSectorCode=P&Session=1:43:11PM](http://www.megger.com/us/products/OpenDoc.php?FileName=MPRT_DS_sp_V01.pdf&ProductID=567&DocType=DataSheet&PermissionLevel=a&Site=us&ShowDocType=DataSheet&ref=/us/products/ProductDetails.php?ID=567&Description=mprt&ModelFamily=MPRT&BusinessSectorCode=P&Session=1:43:11PM)



# Index

## A

Adds.....	15
Advanced Visual Test Software .....	1
Analog Input Control .....	253
And/or .....	1
AVO PULSAR	
control.....	1
AVO PULSAR .....	1
AVTS	
copy.....	1
exploring.....	15
issues .....	15
Overview .....	15
AVTS .....	1
AVTS .....	13
AVTS .....	15
AVTS contains .....	1
AVTS displays.....	15
AVTS General Theory	
Operation.....	13
AVTS General Theory.....	13
AVTS Help	
Welcome .....	1
AVTS Help .....	1
AVTS New Relay Wizard	
call.....	15
AVTS New Relay Wizard .....	15
<b>B</b>	
BASIC .....	13
Blank.mdb.....	15
Bronze Way .....	1
<b>C</b>	
Call	
AVTS New Relay Wizard.....	15
Workbook .....	13
Workspace .....	13
Call.....	13
Call.....	15
Care.....	15

Change Database.....	15
Connections Editor	
Displaying.....	13
Connections Editor.....	13
Contact	
Megger .....	1
Contact.....	1
Controls	
AVO PULSAR.....	1
Controls.....	1
Controls.....	13
Controls.....	253
Copy	
AVTS .....	1
Copy.....	1
Copy.....	15
Create .....	1, 13, 15
<b>D</b>	
Dallas .....	1
Default.....	15
Default.mdb.....	15
Device	
lists .....	13
Device .....	13
Device .....	15
Displaying	
Connections Editor .....	13
General Information.....	13
Monitor Editor .....	13
Displaying .....	13
<b>E</b>	
Ease	
tiew .....	1
Ease.....	1
Electromechanical.....	1
End	
Start.....	13
End.....	13
Examples .....	15

Examples.mdb ..... 15

Exploring

    AVTS ..... 15

Exploring ..... 15

Ext ..... 1

**F**

First uncheck ..... 15

**G**

General Information

    Displaying ..... 13

General Information ..... 13

GUI

    utilizing ..... 13

GUI ..... 13

**I**

Installed ..... 1, 15

Issues

    AVTS ..... 15

Issues ..... 15

**L**

Lists

    Devices ..... 13

Lists ..... 1

Lists ..... 13

Lists ..... 15

Log All Commands ..... 15

**M**

Manufacturer ..... 15

Megger

    contact ..... 1

Megger ..... 1

Megger AVTS ..... 1

Megger MPRT ..... 1

Megger.mdb ..... 15

Menu ..... 15

Microsoft Excel ..... 15

Monitor Editor

    Displaying ..... 13

Monitor Editor ..... 13

MPRT ..... 1, 15

MPRT Results ..... 15

**N**

New ..... 1, 15

New Device Wizard ..... 15

New Group ..... 609

New Test Wizard ..... 15

Notes ..... 15

**O**

Opens ..... 13, 15

Operation

    AVTS General Theory ..... 13

Operation ..... 13

Options ..... 15

Organization ..... 15

Overview

    AVTS ..... 15

Overview ..... 15

**P**

Pulsar ..... 15

Purposes ..... 1

**R**

Relay Wizard ..... 15

Relays

    Selecting ..... 13

Relays ..... 1

Relays ..... 13

Relays ..... 15

RelayTestLbV2.mdb ..... 15

Reports Wizard ..... 1

**S**

Select

    Relay ..... 13

Select ..... 1

Select ..... 13

Settings ..... 1

Simulate All Devices ..... 15

Simulate Device ..... 15

Start

    End ..... 13

Start ..... 13

**T**

Test Editor ..... 13

Test Editor Tab ..... 13



Test Wizard.....	1	AVTS Help.....	1
Tests .....	1, 13, 15	Welcome .....	1
Tools .....	1, 13, 15	Windows Explorer .....	15
Tools menu item .....	15	Workbook	
<b>U</b>		called .....	13
Use .....	15	Workbook.....	13
User .....	1	Workbook.....	15
Utilizing		Workspace	
GUI .....	13	called .....	13
Utilizing .....	13	Workspace .....	13
<b>W</b>		Workspace .....	15
Welcome			