Reference Manual

ETS-LINDGREN™
An ESCO Technologies Company

Copyright and Update Log

ETS-Lindgren Inc. reserves the right to make changes to any products herein to improve functioning or design. Although the information in this document has been carefully reviewed and is believed to be reliable, ETS-Lindgren does not assume any liability arising out of the application or use of any product or circuit described herein; nor does it convey any license under its patent rights nor the rights of others. All trademarks are the property of their respective owners.

© Copyright 1995-2024 by ETS-Lindgren Inc. All Rights Reserved. No part of this document may be copied by any means without written permission from ETS-Lindgren Inc.

Trademarks used in this document: The *ETS-Lindgren* logo, *TILE!*, *VisionTRX*, *EMCenter, EMSwitch, EMPower, EMControl, EMSense, EMSense 10/40, EMField* are trademarks of ETS-Lindgren Inc.; *National Instruments* and *NI* are trademarks of National Instruments Corporation; *Intel* and *Pentium* are registered trademarks of Intel Corporation; *Microsoft*, *Excel*, *Windows*, and *MS-DOS* are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Gnuplot Plug in: Copyright Statement

Copyright (C) 1986 - 1993, 1998, 2004, 2007 Thomas Williams, Colin Kelley

Permission to use, copy, and distribute this software and its documentation for any purpose with or without fee is hereby granted, provided that the above copyright notice appears in all copies and that both that copyright notice and this permission notice appear in supporting documentation.

Revision Record | MANUAL, TILE, REFERENCE, SOFTWARE | Part #399329, Rev. N

Revision	Description	Date
A	Initial Release.	AUG, 2011
B	Updates to selected descriptions and formulas.	JUN, 2012
C	Updated graphics: Working with the Flowchart, Toolbars Standard Bar, Switches, Measurement, Positioner, and Control Actions. Additions: Windows Menu Bar Tools, EMSwitch, Call Action.	JUL, 2012
D	Updated contact information; miscellaneous edits.	MAY, 2013
E	Updated Steps to Create a Profile.	JUN, 2013
F	Added TILE! 7 information.	JAN, 2014
G	Update.	JUL, 2016
H	Update.	FEB, 2017
J	Updated format, pictures, and definitions. Added NI I/O Trace Section.	JUN, 2018
K	Added data elements, script information, examples and pictures.	FEB, 2019
L	Updated definitions. Added NI MAX connection example.	MAR, 2020
M	Updated: License information, Monitors, and NI I/O Trace. Additions: DO160G Zero Span, Simulation, and IVI.	APR, 2022
N	Updated definitions and graphics.	FEB, 2024

Contact Information and System Requirements

Contact USA Headquarters

1301 Arrow Point Drive

Cedar Park, TX 78613

Telephone: +1.512.531.6400

Fax: +1.512.531.6500

Please see http://www.ets-lindgren.com/ for a list of all ETS-Lindgren offices.

TILE! Support

Support Portal: https://support.ets-lindgren.com/TILE

Support Email: TileSupport@ets-lindgren.com

The more information provided - the better and faster the support team can provide help. To expedite the process, please have the following information available to streamline the support process:

- Full Name
- Email Address
- Phone Number
- Company Name
- Version Number of TILE!
- Relevant Profile[s]
- Screenshots of the error popup[s]
- NI I/O trace of the error[s]

System Requirements

Recommended:

- Intel i7, 3.0 GHz or compatible
- Microsoft Windows 10, 64 bit
- 8 GB RAM
- 320 GB free hard drive space
- DVD/CD-ROM
- 2 free Ethernet ports
- 2 free USB ports
- National Instruments GPIB card
- 20 inch monitor

Minimum:

- Intel® Pentium® 4, 2.4 GHz or compatible
- Microsoft Windows 7, 32 bit
- 4 GB RAM
- 100 GB free hard drive space
- DVD/CD-ROM
- 1 free USB port
- National Instruments GPIB card (USB or PCI card)

License and Maintenance

The TILE! license is on a USB Key. The Key **must** be inserted into a USB slot on the PC for TILE! to **communicate** with an instrument (Spectrum analyzer, signal generator, etc.) The USB Key is also need to **run a test** profile with **simulated** instruments since TILE! does not know the difference between a real and a simulated instrument.

A license maintenance contract is valid for 1 year. When you buy TILE!, the last version that was available before your contract expired will be good forever with a “Permanent” key.

The USB Key never stops working, so TILE! will still run if maintenance expires (TILE! is a permanent license). However, the key will not run versions of TILE! that are released **after** the license expires.

NOTE: A USB Key is **not** needed to **load** a test profile, **edit** the profile, **change** or revise data, rerun calculations, **save** the profile, etc.

There is also a yearly maintenance fee if you want or need access to a version of TILE! that becomes available after the license expires.

The Maintenance covers remote help by email and phone. It also includes access to new software releases and drivers, gives access to our support portal, and being in maintenance enables use of the Simulator for developing profiles.

End User License Agreement

TILE!™ END USER LICENSE AGREEMENT

READ THIS CONTRACT CAREFULLY. BY CLICKING THE "I AGREE" BOX AND/OR INSTALLING OR USING THE SOFTWARE, YOU ARE ACCEPTING ALL TERMS AND CONDITIONS OF THIS AGREEMENT. THIS AGREEMENT GOVERNS THE USE OF TILE!™ SOFTWARE ("SOFTWARE") AND ALL UPGRADES, UPDATES, PATCHES, HOTFIXES, MODULES, ROUTINES, FEATURE ENHANCEMENTS AND ADDITIONAL VERSIONS OF THE SOFTWARE THAT REPLACE OR SUPPLEMENT THE ORIGINAL SOFTWARE (COLLECTIVELY, "UPDATES") ALONG WITH THE ASSOCIATED MEDIA, PRINTED MATERIALS, ONLINE OR ELECTRONIC DOCUMENTATION, DISTRIBUTED BY OR ON BEHALF OF ETS-LINDGREN INC. ("ETSL"), UNLESS A SEPARATE LICENSE AGREEMENT EXISTS BETWEEN YOU AND THE MANUFACTURER OR OWNER OF THE SOFTWARE. SOFTWARE UPDATES, FOR THE PURPOSE OF THIS AGREEMENT, ARE CONSIDERED SOFTWARE AND THIS AGREEMENT, IN AND OF ITSELF, DOES NOT ENTITLE YOU TO ANY FUTURE UPDATE. BY EXPRESSLY ACCEPTING THESE TERMS OR BY DOWNLOADING, INSTALLING, ACTIVATING, AND/OR OTHERWISE USING THE SOFTWARE, YOU AGREE THAT YOU HAVE READ THIS AGREEMENT, AND AGREE TO COMPLY WITH AND BE BOUND BY THE TERMS AND CONDITIONS OF THIS AGREEMENT AND ALL APPLICABLE LAWS AND REGULATIONS. THIS AGREEMENT LIKE ANY WRITTEN AGREEMENT SIGNED BY YOU, IS ENFORCEABLE AND LEGALLY BINDING BETWEEN YOU AND ETSL. IF YOU DO NOT AGREE TO ALL TERMS AND CONDITIONS, DO NOT INSTALL OR USE THE SOFTWARE. IF YOU ARE ACCEPTING THESE TERMS AND CONDITIONS ON BEHALF OF AN ENTITY, YOU ACKNOWLEDGE THAT YOU HAVE THE APPROPRIATE AUTHORITY TO ACCEPT THESE TERMS AND CONDITIONS ON BEHALF OF SUCH ENTITY.

This End User License Agreement ("Agreement") is between ETS-LINDGREN INC. ("ETSL") and the customer that has procured the software license for use as an end user ("Customer" or "You" as used above).

1. TECHNOLOGY LICENSE.

1.1 License Grant. Subject to the terms and conditions of this Agreement, ETSL grants Customer a worldwide, non-exclusive, non-sublicenseable, non-transferable, limited license ("License"), to install or have installed, display, and use one copy of TILE!™ Software and the associated object code therein ("Software"), in a manner consistent with the terms of this Agreement and the User Manual provided.

1.2 Limitations. The License is conditioned upon ETSL receiving Customer's timely payment of all fees applicable to the Software and to any Products in which the Software may be loaded. ETSL reserves all right, title, and interest in and to the Software not expressly granted herein and grants no additional express or implied license, right or interest in any copyright, patent, trade secret, trademark, invention or other intellectual property right of ETSL. The Software is licensed for use as a single product and Customer shall not: (a) disassemble, decompile, or attempt to reverse engineer the Software or derive the underlying source code or algorithms; (b) modify, add features, functionality, or create derivative works of the Software; (c) copy or attempt to copy the Software, or allow others to do so, except that Customer shall be permitted to retain one archival copy for back-up purposes; (d) distribute, sell, sublicense, rent, or lease the Software or Customer's License or attempt to do any of the foregoing; (e) remove any product identification, trademark, copyright, or other proprietary information contained in the Software; (f) use the Software in the development of products or services competitive with those offered by ETSL.

1.3 Deliverables. ETSL will provide one executable copy of the Software and one copy of the User Manual.

1.4 Third-Party Software. The Software may utilize software components from third parties ("Third-Party Software"), including without limitation, source code, object code, and algorithms developed by others and software subject to open source licensing. A list of the Third-Party Software utilized will be provided to Customer upon written request. All Third-Party Software is licensed to Customer pursuant to the relevant third-party license agreement. To the extent the terms or conditions of a third-party license agreement conflict with this Agreement, when required by the conflicting third-party license agreement, the terms and conditions of such agreement will control over this Agreement. ETSL makes no representations or warranties with regard to any Third-Party Software.

2. PAYMENT; TERM AND TERMINATION.

2.1 Payment. Customer agrees to pay all applicable fees associated with the use of the Software License and agrees to comply with the payment terms established by ETSL at the time of sale.

2.2 Term and Termination. This Agreement becomes effective upon the earlier of Customer's installation or use of the Software and shall continue in full force and effect for the duration of the evaluation period, unless terminated in the manner set out below. The License granted under this Agreement is conditioned upon Customer's acceptance of and continued compliance with all terms and conditions stated herein. Customer's noncompliance or breach of any term or condition of this Agreement, including without limitation, Customer's failure to comply with the payment terms established by ETSL at the time of sale, shall result in immediate termination of this Agreement and the accompanying License. Upon notice of termination, Customer shall immediately cease all use of the Software, remove and destroy all copies and components, and certify in writing that such removal and destruction has occurred. ETSL will not have any obligation upon the termination of this Agreement to refund any portion of any License fee. Sections 1.2, 2.1, 5.1, 6, 7.1, 7.3, 7.4, 7.5, 7.9, 7.10, 7.11, and 7.12 shall survive termination of this Agreement.

3. UPDATES; SUPPORT AND MAINTENANCE.

3.1 Updates. ETSL may from time to time make updates to the Software which, in its discretion, may be made available to Customer for download. Installation of such updates will be either optional or mandatory. Upon notice of a mandatory update, Customer agrees to download and install the update within ten (10) days.

3.2 Support. This Agreement does not include software support from ETSL. Maintenance and support services, if any, are provided under a separate agreement.

4. WARRANTIES.

4.1 Title. This Software is licensed and not sold. ETSL warrants that it has clear title in and to the Software and all the necessary rights, power and authority to enter into this Agreement.

4.2 Authority to Bind. Customer warrants that it has the necessary rights, power and authority to enter into this Agreement, assume obligations, and accept a grant of rights on behalf of Customer's organization.

4.3 Viruses, Malware and Malicious Code. Virus, Malware and Malicious Code means any code, program, or sub-program whose knowing or intended purpose is to damage or corrupt data or the operation of the computer system containing such code, program, or sub-program. Prior to the delivery of the ETSL-branded Software, ETSL will use commercially reasonable efforts to prevent the introduction of known computer viruses, malware and other malicious code.

4.4 Limited Warranty. For 30 days from the initial date of installation, ETSL warrants that the Software will perform substantially in accordance with the accompanying documentation, and the

Software media will be free from defects in materials and workmanship. In the event of a defect in performance or materials, ETSL, at its option, shall either (a) refund the price paid for the Software, or (b) repair or replace the Software that does not meet this Limited Warranty. EXCEPT AS OTHERWISE PROVIDED HEREIN, THE SOFTWARE AND THE ACCOMPANYING WRITTEN MATERIALS ARE PROVIDED "AS IS" WITHOUT EXPRESS OR IMPLIED WARRANTY OF ANY KIND. ETSL EXPRESSLY DISCLAIMS ALL IMPLIED WARRANTIES, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. ETSL DOES NOT WARRANT THAT THE SOFTWARE WILL BE ERROR FREE OR THAT IT WILL OPERATE WITHOUT INTERRUPTION.

4.5 High Risk Disclaimer. When monitoring high-risk applications or hazardous environments requiring fail-safe controls including without limitation, operation of nuclear facilities, avionics equipment, safety equipment, communication systems, air traffic control systems, life support systems, and weapons systems, Customer agrees that all results obtained through use of the Software are for reference purposes only. Without limiting the generality of the foregoing, ETSL specifically disclaims any express or implied warranty of fitness for such purposes.

5. INDEMNIFICATION.

5.1 Indemnification by Customer. Customer shall indemnify, defend and hold ETSL harmless from and against any and all claims, damages, losses, and expenses, including without limitation reasonable attorneys' fees and court costs, arising out of, resulting from, or connected with Customer's use of the Software. Customer's obligation under this Section 5.1 applies without limitation to any claim arising from Customer breach of Section 4.5, any infringement or claim for infringement of the intellectual property rights of any third party resulting from or connected with Customer's modification or combination of the Software with any other product not provided or expressly approved by ETSL, and any fines, fees, or penalties imposed as a result of Customer's failure to comply with all applicable laws and regulations.

5.2 Indemnification by ETSL. ETSL shall indemnify and defend Customer from and against all claims of liability for infringement of any United States patent, copyright or trade secret rights of a third party arising from Customer's use of the Software License in accordance with the User Manual, provided that: (i) the infringement is not caused by the combination of the Software with any other item not provided or approved by ETSL, including but not limited to software, data, or hardware, (ii) Customer notifies ETSL in writing within ten (10) days from the date Customer first becomes aware that an infringement claim exists, and (iii) Customer agrees to allow ETSL to fully control any litigation and settlement of such infringement claim.

6. LIMITATION OF LIABILITY. IN NO EVENT SHALL ETSL BE LIABLE FOR ANY CONSEQUENTIAL, EXMEPLARY, INCIDENTAL, INDIRECT, PUNITIVE, OR SPECIAL DAMAGES, INCLUDING WITHOUT LIMITATION ANY CLAIM FOR LOST PROFITS, LOST OR CORRUPTED DATA OR SOFTWARE, OR DOWNTIME, ARISING OUT OF OR RELATING TO THIS AGREEMENT OR CUSTOMER'S ABILITY OR INABILITY TO USE THE SOFTWARE AND LICENSE. THIS LIMITATION ON RECOVERY APPLIES TO ALL CLAIMS OR CAUSES OF ACTION, INCLUDING WITHOUT LIMITATION, ACTIONS ARISING UNDER CONTRACT LAW, WARRANTY LAW, STRICT LIABILITY, NEGLIGENCE, AND EVEN IF NOTIFIED OF THE POSSIBILITY OF SUCH DAMAGES. THE TOTAL LIABILITY OF ETSL FOR ALL CLAIMS MADE BY CUSTOMER RELATING TO THIS AGREEMENT SHALL NOT EXCEED THE TOTAL AMOUNT PAID BY CUSTOMER FOR THE LICENSE PROVIDED HEREUNDER.

7. GENERAL.

7.1 Governing Law and Arbitration. This agreement and performance hereunder shall be governed by the laws of the State of Texas without regard to the conflict of law principles. Customer agrees to resolve any dispute or controversy arising out of this Agreement by arbitration in Travis County,

Texas in accordance with the Commercial Arbitration Rules of the American Arbitration Association. A single arbitrator shall be agreed upon by both parties or, if unable to do so within thirty (30) days, then both parties agree that a single arbitrator shall be appointed by the American Arbitration Association. The award of the arbitrator shall be binding and judgment upon the award may be entered in either the District Courts for Travis County, Texas or the Federal District Court for the Western District Court of Texas. Customer hereby consents to such venue and to the personal jurisdiction of such courts and waive the right to demand a jury trial in any such action.

7.2 Assignment. Customer shall not assign or otherwise transfer the Software, Customer's license, or this Agreement to anyone, including any parent, subsidiaries, affiliated entities or third parties, or as a part of the sale of any portion of its business, or pursuant to any merger, consolidation or reorganization, without obtaining prior written consent from ETSL. Such consent will not be unreasonably withheld. Third parties, contracted by Customer, including without limitation, consultants, subcontractors, or agents, to implement the Software on Customer's behalf at Customer's facility, and who have agreed in writing to use the Software only in accordance with the terms and conditions of this Agreement, do not violate this Section.

7.3 Audit. To confirm compliance with Customer's obligations hereunder, Customer agrees that ETSL and/or its agents, may upon ten (10) days prior written notice and during normal business hours, inspect and audit all of Customer's records relating to the Software and this Agreement. If such inspection or audit reveals that Customer has not complied with Customer's obligations, Customer agrees to bear the full cost of the inspection and audit, and to acquire all additional licenses necessary to maintain compliance.

7.4 Confidentiality. Customer agrees that Customer, its agents and employees shall not in any manner use, disclose or otherwise communicate any information with respect to the Software which might enable the unauthorized use or reproduction of the Software or any portion thereof. Customer shall take all action necessary to protect the confidential and proprietary information included in the Software, including without limitation providing appropriate instruction to Customer's employees and obtaining their consent to be bound by this confidentiality clause.

7.5 Export, Import and Government Restrictions. The Software is subject to U.S. export laws as well as the laws of the country where it is delivered or used. Customer agrees to abide by these laws. Under these laws, the Software may not be sold, leased or transferred to embargoed countries (currently Cuba, Iran, North Korea, Sudan and Syria), other restricted countries, restricted end-users, or for restricted end-uses. Customer specifically agree that the Software will not be used for activities related to weapons of mass destruction, including but not limited to, activities related to the design, development, production or use of nuclear materials, nuclear facilities, or nuclear weapons, missiles or support of missile projects, or chemical or biological weapons. Customer understands that certain functionality of the Software, such as encryption or authentication, may be subject to import or export restrictions in the event that Customer exports the Software from the country of delivery. Customer is responsible for complying with all applicable restrictions.

7.6 Force Majeure. Neither party shall be liable for any noncompliance caused by acts of God, war, terrorism, fire, flood or any other unforeseeable event beyond the reasonable control of both parties. To the extent either party is unable to comply with the Agreement as a result of such an event, that party shall not be in default of this agreement, provided however that such party makes every reasonable effort to become compliant as soon as practicable.

7.7 Entire Agreement. Both parties agree that this Agreement is the complete and exclusive statement of the agreement between Customer and ETSL, which supersedes and merges all prior proposals, understandings and all other agreements, oral or written, between both parties relating to this Agreement.

7.8 Severability. If any provision of this Agreement is invalid, illegal or unenforceable under any applicable statute or rule of law, it is to that extent to be deemed omitted. The remainder of the Agreement shall be valid and enforceable to the maximum extent possible.

7.9 Waiver and Amendment. The failure of either party to exercise any right provided for in this Agreement shall not be deemed a waiver of any further right under this Agreement. Any waiver must be in writing and signed by the party waiving its rights. This Agreement may not be modified, altered, or amended except by written instrument duly executed by Customer and ETSL.

7.10 Conflicting Terms. In the event the terms of this Agreement conflict with the terms or conditions of any other agreement concerning the Software, including without limitation terms or conditions contained in any purchase order, purchase order acknowledgment, invoice, contract, or subcontract, this Agreement shall control.

7.11 Independent Contractors. The parties are independent contractors. Neither party will have any rights, power or authority to act or create an obligation, express or implied, on behalf of the other party except as specified in this Agreement.

Table of Contents

Copyright and Update Log 2
Contact Information and System Requirements 3
Contact USA Headquarters 3
TILE! Support 3
System Requirements 3
License and Maintenance 4
End User License Agreement 5
Table of Contents 10
Introduction 25
Welcome 25
TILE! Datasheet 25
Key Features 26
How TILE! Works 26
Custom Test Profiles 26
TILE! Test Executive 27
Report Generator 27
Simulator 27
TILE! Terminology 27
Security – Password Protection 28
Change Password 28
New Features 29
Frequently Asked Questions 30
Running profiles in different versions 30
Hi-Res monitor scaling adjustment 30
Navigation 31
Toolbars 31
Main Menu Toolbar 31
Expired License 35
Update License 36
Standard Toolbar 38
Command Toolbar 38
View Toolbar 39
TILE! Windows 40
Window Behavior (Docked and Tabbed Modes) 40
Flowchart View 40
Palette View 40
Project View 41

Imports 41
Exports 42
Properties View 42
Drag and Drop properties 43
Data Element View 44
Instrument View 44
Graph View 44
Table View 45
Log View 45
Ignore/Replace Commands 45
Audit Trail View 47
Common Action Features 47
Create a Basic Profile 49
Simulated Emissions Scan 49
Polar Plot with Simulator 52
Data Elements 56
Data Element View Window 56
Element Names 56
Element Database 57
Element Valid Points 57
Element Intp (Interpolated) 57
Element Source 57
Defining a Data Element 58
Editing and Deleting Data Elements 58
Exporting Data 58
Data Types 58
Measurement Elements 58
File Elements 59
Equation Elements 59
Preset Elements 59
Word Elements 60
Word from Equation Elements 60
Word from File Elements 60
Script Elements 60
Standard Elements 61
Data Element Tabs 61
Name Tab 61
Source Tab 62
Values Tab 62

File Tab 63
Equation Tab 63
Addition “+” operator 64
Simple Tab 65
Bands Tab 65
Script Tab 66
Collection Tab 67
Table Tab 67
Standard Tab 67
Database Table Tab 68
Instruments 69
Instrument View Window 69
Defining an Instrument 69
Editing and Deleting Instruments 69
Instrument Tabs 70
Driver Tab 70
Build Instrument Database 71
Address Tab 72
GPIB Tab 72
Serial Tab 73
VISA Tab 74
VISA Resource 74
Using NI Max to configure instruments 75
Sim Tab 76
Example description 76
Quick enable/disable simulation 76
Simulation for Hi-resolution screen 77
Graphs 78
Defining a Graph 78
Copying a graph 78
Graph Toolbar 79
Display Data 80
Controlling Display Conditions 80
Graph Setup Tabs 81
Titles Tab 81
X Labels Tab 82
X-axis Tab 83
Y Labels Tab 84
Y-axis Tab 85

Fonts Tab ... 86
Graph Tab ... 86
Legend Tab ... 88
Additional Information Tab ... 89
EUT Tab ... 89
Comments Tab ... 90
Bitmap Tab ... 91
Page Defaults Tab ... 91
Graph features ... 92
Cursor Position ... 92
Temporary Zoom ... 92
Cursor Interpolation ... 92
Tables ... 93
Defining a Table ... 93
Copying a table ... 93
Table Toolbar ... 93
Display Data ... 94
Controlling Display Conditions ... 95
Table Setup Tabs ... 95
Titles Tab ... 96
Additional Information Tab ... 96
Comment Tab ... 97
EUT Tab ... 97
Page Defaults Tab ... 98
Flowchart and Palette ... 99
Flowchart Window ... 99
Actions ... 100
The TILE! Palette ... 100
Placing Action Icons ... 100
Using Actions ... 101
Highlighted Actions ... 101
Common Tabs for Actions ... 101
Name Tab ... 101
Bookmarks Tab ... 102
Progress Bar ... 102
Manipulate Flowchart ... 103
Structure Actions ... 103
Page Size ... 103
Zoom ... 103

Objects ... 103
Ghost objects ... 104
Ancillary Actions ... 105
Drawing Tool Actions ... 105
Shapes and Colors ... 105
Connector Action ... 106
File Management Actions ... 106
Auto Save ... 106
Save As Tab ... 106
Backup Tab ... 107
Save Windows ... 107
Graph/Table Tab and Data Tab ... 107
Save Options Tab ... 108
HTML Export Tab ... 108
Transfer Data ... 108
Transfer to/from File Tab ... 109
Direct Entry ... 109
Data Element Tab ... 110
Prompt Tab ... 110
Dialog Box ... 110
Clear Data ... 111
Clear Data Tab ... 111
Load Data Tab ... 112
Process Actions ... 112
Start (GO) ... 112
Sequencing ... 112
Prompt ... 114
Message Tab ... 114
Choice Tab ... 115
Sound Tab ... 115
Jump To Tab ... 116
Loop Tab ... 116
Dialog Box ... 117
Math ... 117
Data Tab ... 118
Position Data In/Out Tabs ... 118
Dialog Box ... 119
Print Pages ... 119
Display Tab ... 120

Launch Application 120
Commands Tab 120
Linked Event Tab 121
Pass/Fail Check and Reset Tabs 121
Picture 121
Picture Tab 122
Prompt Tab 122
Dialog Box 122
Timer 123
Sound/Delay Tab 123
Msg Tab 123
Linked Event Tab 123
Dialog Box 124
Information Actions 124
Lab Information 124
Laboratory Information Tab 124
Client Information 125
Client Information/Additional Information Tabs 125
File/Options Tab 125
Operator Information 126
EUT Information 126
Equipment Under Test Tabs — Page 1 and Page 2 126
Equipment Description Tab 127
File/Options Tab 127
Comment 127
Weather 127
Interface Actions 128
Measurement and Control Actions 129
Control Actions 129
Instrument Initialization (INIT) 129
Instruments Tab 129
Instrument Commands (VISA) 129
Instrument Commands Tab 130
Command Format 130
Position Tower 131
Links Tab 131
Position Tab 132
Multiple Positions Tab 133
Dialog Box 133

Position Turntable 133
Position Tab 133
Links Tab 134
Multiple Positions Tab 134
Dialog Box 134
Measure Actions 134
Calibrate Cables/Amplifiers 135
Setup Tab 135
Frequency Steps Tab 135
Data Tab 136
Instruments Tab 136
Dialog Box 137
Amplifier Calibration 137
Frequency Tab 137
Leveling Tab 138
Calibration Tab 138
Instruments Tab 138
Harmonic Test Tab 139
Setup / Monitor Instrument (READ) 139
Monitor Tab 139
Signal Generator Tab 140
Level and Hold Tab 140
Spectrum Analyzer – 1 Tab 141
Spectrum Analyzer – 2 Tab 142
Network Analyzer Tab 142
Read Instrument 142
Defaults Tab 143
Links Tab 143
Trigger Tab 143
Polar / Scalar Measurement 143
Frequency Tab 144
Instruments Tab 144
Input/Output Tab 144
Parameters Tab 145
Call Setup 145
Device Config Tab 145
Setup Calling Test Tab 146
Selecting the Telecom Tester Interface 146
Switch Actions 147

Switch Setup 148
Switches Tab 148
Instrument/Switch Tab 148
EMSwitch 149
Instrument/Switch Tab 149
Switches Tab 149
Switches A, B, C, D, E 149
Emission Actions 150
Range Type Measurement Actions 150
Measure Range 150
Measure Range Extended 150
Frequency Tab 151
Amplitude Tab 152
Data 1 Tab 153
Instruments Tab 153
Parameters Tab 154
Receiver Functions Tab 155
Options Tab 155
Special Tab 156
Monitor Tab 156
Trigger Out Tab 156
Dialog Box 157
Optimize Measurement (PRE-SCAN) 157
Freq – Analyzer 1 Tab 157
Amplitude Tab 158
Data - Analyzer Tab 158
Tower/Turntable Tab 159
Tower Defaults Tab 159
Instruments Tab 160
Timing Tab 160
Parameters - Analyzer 1 Tab 161
Dialog Box 161
Scan Range Measurement 162
Fast Scan Across TT/TWR 162
Peak Type Measurements Actions 162
Scan Peaks and Scan Peaks PSA 162
Frequency Tab 163
Freq Steps Tab 163
Instruments Tab 164

Output Tab 164
Output 2 Tab 165
Search Span Tab 165
Search BW 166
Compare Tab 166
Parameters Tab 167
Recv/Analyzer Tab 167
Optimization Tower / Optimize Turntable Tabs 168
Optimize Parameters Tab 169
Antenna Polarity Tab 170
Timing Tab 170
Power Tab 171
Dialog Box 171
Measure Peaks 172
Frequency Tab 172
Amplitude Tab 173
Search Tab 173
Std Parameters Tab 174
Peak Parameters Tab 174
QP Parameters Tab 175
Avg Parameters Tab 175
Instruments Tab 175
Receiver/Analyzer Tab 176
Power Tab 176
Harmonic Tab 176
GTEM Actions 177
GTEM/OATS 3 Position Correlation 177
GTEM Tab 177
OATS Tab 178
Inputs Tab 179
Results Tab 179
Defaults Tab 179
GTEM/OATS 9-Position Correlation 180
GTEM Tab 180
OATS Tab 180
Alignment Tab 181
Input Tab 181
Results Tab 182
GTEM Optimization 182

Frequency Tab 182
Data Tab 182
Links Tab 183
Parameters Tab 183
Position EUT (GTEM Manipulator) 183
Position Tab 183
Links Tab 184
Dialog Box 184
Other Actions 184
Signal Discrimination Measurement 185
Input Freq Data Tab 185
Input Position Data Tab 185
Output Data - 1 Tab 186
Output Data - 2 Tab 187
Instruments Tab 187
Measurement Parameters Tabs 187
Search Parameters Tabs 188
Amplitude Tab 188
Standards Tab 188
Tower/Turn Position Tab 189
Tower Setup Tab 189
Timing Tab 190
Mode Tab 190
Dialog Box 190
Site Attenuation 193
Frequency Tab 193
Levels Tab 194
Parameters Tab 194
Data Tab 195
Antenna Tab 195
Instruments Tab 196
Dialog Box 196
Occupied BW Mask and FCC Power Spec Limit 197
Immunity/Susceptibility Actions 198
Immunity Actions 198
Immunity / Susceptibility Calibration 198
Frequency Tab 198
Freq Steps Tab 198
Amplitude Tab 199

Leveling Tab 199
TEM Tab 199
P-MID Tab 200
Calibration Tab 200
Results Tab 200
Instruments Tab 201
Samples Tab 201
Pause Tab 201
Dialog Box 202
Immunity / Susceptibility Test 202
Frequency Tab 202
Freq Step 1 Tab 204
Freq Steps 2 Tab 205
Freq Steps 3 Tab 206
Pause Tab 207
Primary Amp Tab 208
Injection Level Tab 209
Sec Amp-Harmonics Tab 210
Leveling Tab 210
TEM Tab 212
Check Tab 213
Reference Tab 213
Threshold Tab 214
Auto Thresholding Tab 215
AM Modulation Tab 215
FM Modulation Tab 216
Pulse Modulation Tab 217
PM-Key Tab 217
Calibration Tab 218
Probe Cal. Tab 218
Results - 1 Tab 219
Results - 2 Tab 220
Calculations Tab 221
Instruments Tab 221
Probes Tab 222
Samples Tab 223
Monitors Tab 224
Position Tab 224
Process Tab 225

Pass/Fail Tab 227
Dialog Box 227
Auto Thresholding 229
Example of Auto-Threshold Settings and Operation 230
Immunity Field Uniformity 231
Input (1/2 - 23/24) Tabs 231
Standard Tab 232
Method Tab 233
Output Tab 233
Output 2 Tab 234
Output 3 Tab 234
CS103/4/5 Measurement 235
Frequency Tab 235
Freq Steps Tab 235
Signal Generators Tab 236
Instruments Tab 236
Switchover Tab 236
Output Tab 237
CS103 Tab 237
Waveform Tab 238
CS104 Tab 238
CS105 Tab 239
Parameters Tab 239
Power Tab 239
Monitor/Monitor 2 Tabs 240
Reverb Actions 240
Reverberation Calibration (CAL) 240
Freq Tab 240
Freq Step Tab 241
Amplitude Tab 241
Leveling Tab 242
Chamber Tab 242
Paddles Tab 243
Power Meter Data Tab 243
Probe RMS Tab 244
Probe X, Y, Z (or 1, 2, 3) Tabs 244
Est. Field Tab 245
Data Tab 246
Instruments 1 Tab 246

Instruments 2 Tab 247
Samples Tab 248
Calibration Tab 248
Timing Tab 249
Dialog Box 249
Reverberation Calculation (EVAL) 249
Chamber Tab 250
Est. Field Tab 250
Antenna Tab 251
Output Tab 251
Output 2 Tab 252
Input 1-9 and Input 1-9 Probe Tabs 252
Reverberation Test (TEST) 253
Freq Tab 253
Freq Step Tab 254
Amplitude Tab 254
Leveling Tab 255
Threshold Tab 255
Auto Thresholding Tab 256
Paddles Tab 256
Power Meter Data Tab 256
Probe Data Tab 257
Data Tab 257
Failed Freq Tab 257
Instruments 1 Tab 258
Instruments 2 Tab 258
Samples Tab 258
Calibration Tab 259
AM Modulation Tab 259
FM Modulation Tab 259
Pulse Modulation Tab 260
PM-Key Tab 260
Monitor Tabs 260
Timing Tab 261
Reverb Standards Tab 262
DO160G Zero Span Tab 262
Dialog Box 263
IEC 61000 Series Test Actions 265
Special Math Functions 266

Available Operators and Conditionals 266
Available Conditionals 266
Special Math Functions 267
Native TILE! NI I/O Trace Support 274
Log Options Dialog 275
Options Tab 275
NI I/O Trace Tab 275
Detailed trace settings 277
NI I/O Trace Scripting 278
Using NI I/O Trace 278
Appendix: Miscellaneous Information 282
How to Configure the Ethernet IP address for an instrument 282
Create new NI MAX VISA resource for TILE! 283
TILE! Simulation Utility 288
Simulation Overview 288
General Operation 288
Methods to connect TILE! instruments to the Simulation Utility 289
Method 1: Open a profile pre-configured with simulated instruments. 289
Method 2: Configure a new instrument with Communication set to Simulation. 289
Method 3: Use the TILE! Tools->Run Simulation menu option. 290
Simulation Server List 290
Instrument Shortcut Buttons 292
Basic and Advanced Interface Modes 292
Connection Active LED 294
Simulation Settings 294
Running Multiple Simulator Instances 294
Simulation Panels description and operation 295
Basic Mode 296
Advanced Mode 297
Virtual Signal Generator 300
Command Config CSV files 300
Supported TILE! Drivers with ver. 2.2.1 CFG files 301
Debugging with Simulation 302
Instrument Interactive Control Utility 302
Response Editor 303
Command Config Editor 304
Command Config Editor Buttons 305
Editing the Command Config Table 306
IVI class drivers 307

Installation 307
Run MAX to configure the IVI device 309
Using the IVI Session in TILE! 310
Using the NI IO Trace Utility with IVI Class Drivers 310
Configuring the NI IO Trace Utility for IVI 311
Selecting APIs to view 312

Introduction

The operational and instructional aspects of TILE! referred to in this manual are applicable to TILE! 7. The features presented are additions to the existing features from previous versions of TILE! as described in the remainder of this manual. This manual is updated as features are added or when edits are needed.

This manual provides the appropriate instructions for building test profiles, debugging the design, and running these on a day-to-day basis. A tutorial helps to provide a short example to getting up and running. The manual chapters describe supporting elements to create a full profile.

Welcome

Welcome to TILE!™ software. TILE! is a dynamic software package that undergoes constant enhancements and improvements. As we strive to streamline the software, inputs from the user, as well as those of our developers, are implemented to support evolving needs. Corrections and additions are welcome to help make TILE! Help as informative as possible.

TILE! is an integrated approach to designing, performing, reporting, and archiving complex Electromagnetic Compatibility (EMC) tests. TILE! provides simple, direct control of EMC measurement instruments with a unique, visual interface.

The TILE! concept provides a common user interface for all testing. This is coupled with integration into various standard report writing software (such as Microsoft® Word) and spreadsheets (such as Microsoft Excel®). It provides the ability to perform EMC tests, as well as manage the data generated during these measurements.

TILE! uses a flowchart to simplify the user interface. Each step in the process is represented on the flowchart with an icon, which are referred to as actions. The actions are each a unique test, information step, or prompt. Once the flowchart is built, is it known as a profile. The profile provides a powerful tool for symbolizing communications with the instruments and data management.

One advantage of the TILE! system is the ability to quickly change test instruments. Most EMC instrumentation is supported by TILE!. The emphasis on hardware independence allows for quick introduction of new instrumentation into the laboratory environment, so there is no need to re-create a new profile.

Instrument drivers are provided free of charge to registered users, except in some situations where equipment must be leased or timelines require accelerated development.

If you design the parameters and frequency ranges to match all the instruments, then you can change instruments during a test by simply changing the linkage in the Instrument window. The names used on the Instrument linkage in each action will automatically adjust to the newly defined instrument.

Instrument control is based on the most popular instrument interfaces that include General Purpose Instrument Bus (GPIB or IEEE-488-2), TCP/IP, USB, and Serial Communications (IEEE-232).

The TILE7 directory on your computer has many sample profiles for your convenience. The shipping profiles are saved under the file path *C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles*. Open a profile and save it under a new name to keep the original template from changing.

TILE! Datasheet

ETS-Lindgren's TILE! Software is an integrated test environment for creating and performing Electromagnetic Compatibility (EMC) Tests. TILE! is available with a library of profiles designed for testing

to international test regulations for radiated emissions and radiated immunity (RE, RI) and conducted emissions and conducted immunity (CE, CI). These profiles produce measurements in compliance with test standards by the IEC, FCC, SAE and others.

TILE! is hardware independent with driver support for most EMC instrumentation available today, with options to add drivers for new instruments in the future. TILE! is designed to perform EMC measurements in any test environment, from an OATS, semi- or fully anechoic chambers, reverberation chambers, GTEMs or any suitable test enclosure.

Key Features

- Named 2017 EMC Product of the Year
- Automates EMC Testing
    - Ready-to-Use Profiles for Testing to RE, CE and RI, CI Test Regulations
    - Design Custom Tests
- Vendor Compatible
    - 1600+ Drivers Available
    - Supports GPIB, Ethernet, USB and Serial Instrument Communications
    - Generates Reports Using Microsoft® Office
- Online User Support and Forum
- Used in Corporate Test Facilities and in Commercial Test Labs

How TILE! Works

An EMC test is performed by a series of steps executed in a specific sequence. TILE! represents each step in the test sequence as an icon on a screen, with the test sequence resembling a flowchart. In TILE!, icons are called “actions” and are grouped in six categories: Emissions, Immunity, Information, Utility, GTEM, and Miscellaneous.

Right-click on an icon to open a dialog box for customizing the settings of that particular action. For example, the icon for Measure Range opens a dialog box with tabs for Frequency, Amplitude, Data, etc. Values related to the test can then be specified for the settings within each tab. Each of these steps have been created in TILE! and a test sequence can be generated by linking necessary steps for testing to a specific test regulation or to a user test specification. These test sequences can be stored as ready-to-use tests or “test profiles”.

Custom Test Profiles

While a library of test profiles are available, the real power and flexibility of TILE! is the ability to create custom test profiles, using the Flowchart window of the application. TILE! Test environment offers four other windows in addition to the Flowchart window:

- The Data window accesses the information being tracked in a profile.
- The Instrument window accesses the instrumentation that will be used in the profile.

- The Log window displays the history of commands sent to the instrumentation.
- The Audit Trail window displays the settings history for each icon on the flowchart.

TILE! offers the ability to create as many graphs and tables as needed. Parameters for the graphs and tables are defined by the user (titles, legends, xy values, ranges, column widths, decimal accuracy, etc.) and are updated and displayed in real-time as the data is collected during the test. Graphs can be exported as Windows metafiles (WMF), and tables can be exported as comma-separated files (CSV) for use in tools like Microsoft Excel® to generate test reports. Both graphs and tables can also be copied to the clipboard for pasting into a Microsoft Word® document.

TILE! Test Executive

This easy-to-use interface enables technicians to run test profiles with the ease of point-and-click menus. This allows engineering resources to focus on higher value tasks.

Report Generator

The new report generator has expanded capabilities, including a fully customizable, MS Word document template with test data, tables, and both graphics and photographs of the test set-up. Reports produced are customer ready within moments.

Simulator

As an exclusive benefit to users in current TILE! Maintenance Support, TILE! features an integrated simulator. This powerful tool enables users to utilize an infinite number of testing environments, without leaving their desk. It also provides a simulated environment, reflecting how each of the selected instruments will interact together. This simulator can also free up chamber time by allowing for activities such as profile development, training, customer demos and sales and validation testing to be done virtually.

TILE! Terminology

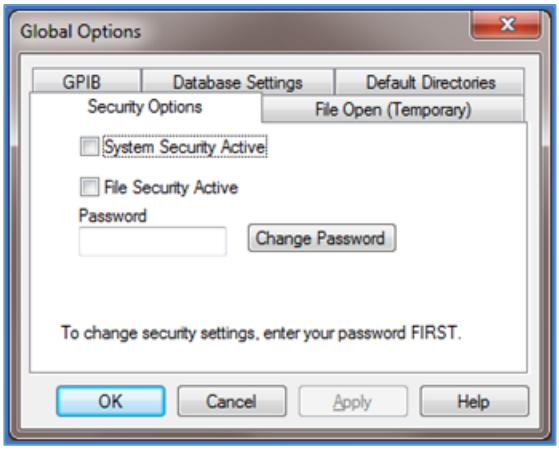
- **Profile** – The test “document” or file that you open that contains all of the actions, settings, and most recently acquired data for a given test.
- **Action** – The items that show up in blocks and are wired on the flowchart. These are the engines of the profile and do the actual communication with instruments as well as calculations and data moving.
- **Data Item** – Raw data, correction factors, presets, and equations for corrected data and parsing.
- **Equation** – A data item that contains math functions to sort or manipulate data.
- **Script** – A section of special code that runs in a Math action to perform a specific process.

Security – Password Protection

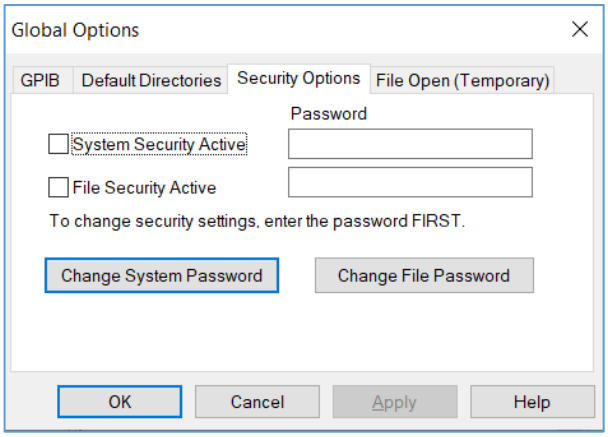
Previously, the relevant properties regarding security were the System Security Active and File Security Active flags, and a Password. The main addition to the security settings in TILE7 is the addition of a second password, “File Password”. The old password is now called the “System Password”.

Below is a comparison of the old and new interface:

Old View:



New View:



In the past, the Password was used to change either the System or File Security Active flags. However, the Password was really a “system” password, meaning it was only relevant to a specific installation of TILE!. This created unexpected behavior when moving a TILE! profile with File Security Active from one TILE! installation to another system.

The File Security flag moved with the profile (enabled or disabled), but the new system would likely have a different password set from the previous one (and a new installation would have a blank password). Users were expecting the password to move with the profile and would type the old password into the new system, but discovered that the password was no longer recognized. This also allowed for an easy bypass of the security feature (turn the flag off with a blank password installation, then move the file back to the original system).

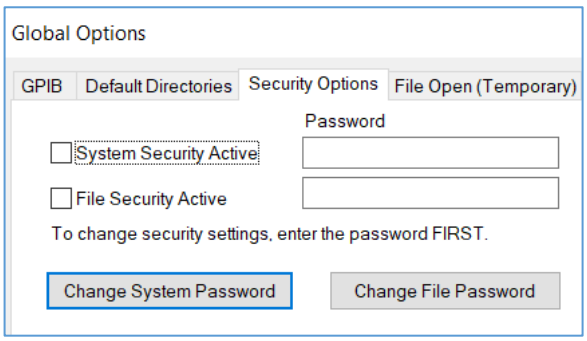
The new File Security Password is now stored in an encrypted format in the profile itself. Thus, when a profile moves from one system to another, not only is the File Security Active flag preserved, but the file’s password is also preserved.

Change Password

To change the security settings, go to the toolbar menu File -> Options, and open the *Security Options Tab*.

There is intelligent enabling of controls based on whether a file is currently open. If the no file is open in TILE!, then the File Security settings are hidden.

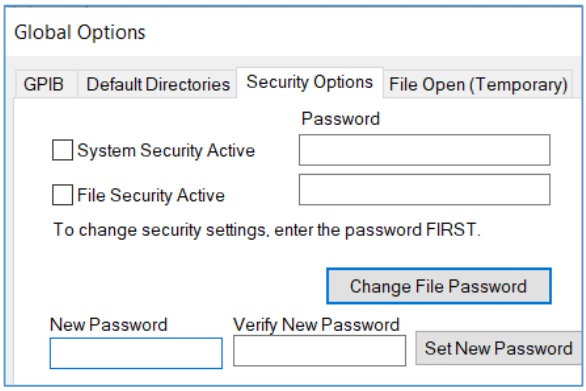
NOTE: A valid unexpired TILE! key and license must be present to make any changes to security settings (enabling/disabling System or File Security



Active flags, or changing either the System or File Password).

To avoid confusion, the two passwords (System and File) each have their own input box, and there are separate buttons to change each password.

Whichever Change Password button is pressed will modify that specific password.



New Features

The updated look of TILE7 includes new window behaviors and while most of the legacy behavior is still available, the updated features are more intuitive for new users.

- One of the more useful features is that each element of the profile either is always available, from the project view or tabs along the margin of the profile. These are available even while an action is in progress.
    - Profile windows can be docked to the edges, tabbed, or float. See *Window Behavior* for more details.
    - The Properties window is a new way to edit actions. See *Properties View* for more details.
- Import features, such as instrument configurations, from other profiles for reusability. See *Project View* for more details.
- Layout views can be saved as user preferences. See *Main Menu Window* for more details.
- The ‘Run Main Action’ button will allow a single click location to start the profile from a selected action. Typically, the Main Run Action would be a Prompt action that would be treated as a Main Menu. See the Command Bar *Run Main Action* for more details.
- Use the Driver Explorer when creating an instrument to search for drivers by vendor or instrument type. See *Instrument Driver Tab* for more details.
- Create scripts to enhance the usability of the program with the new scripting data element. See *Script Elements* for more details.
- The sequencer in the start action can be used to create a table that contains user selectable attributes that can be sent to any action in a flowchart as well as change values of data elements if needed. See *Sequencing* for more details.
- NI I/O Trace has been added for external logging. See *NI I/O Trace* for more details.
- Simulation has been added for simulated instrument testing. See *Simulation Utility* for more details.

Frequently Asked Questions

Running profiles in different versions

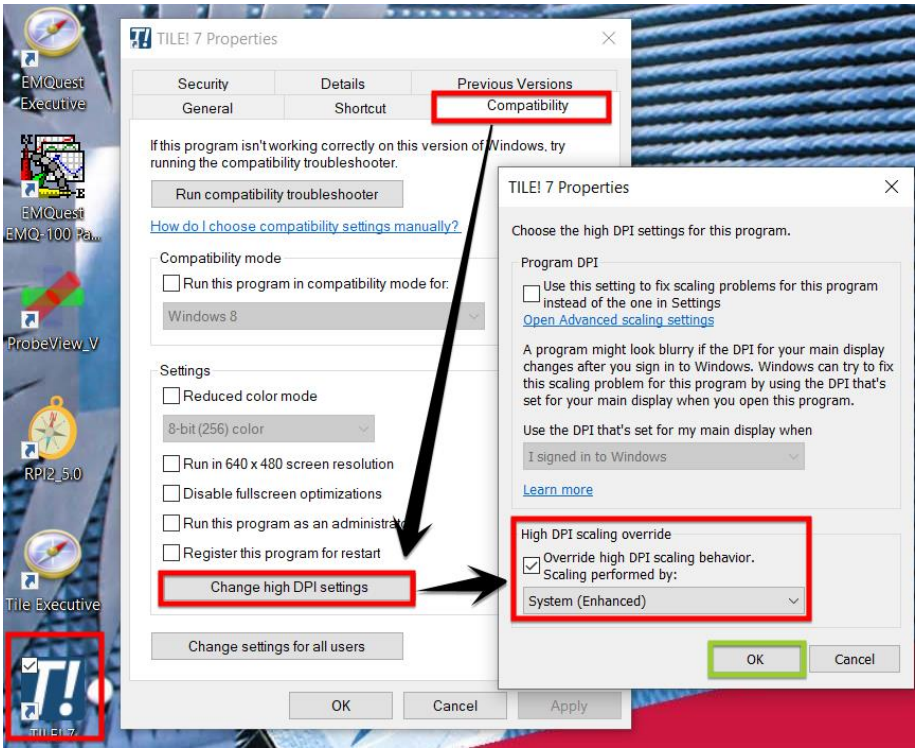
You can open any profiles from TILE! 6 to the current released TILE! Version 7.x.x.x. You can **NOT** open a profile version that is **higher** than **your** licensed system version. For example, you cannot open a TILE! profile created in version 7.9.0.2 using TILE! system version 7.5.3.5.

NOTE: Once you configure a TILE! 6 file forward to TILE! 7, you will be unable to change the profile back to TILE! 6. Therefore always save a backup copy of your profiles before opening an old profile in a newer version of TILE! for the first time. Profiles are unfortunately known to corrupt during this process, and are only sometimes recoverable.

Contact TileSupport@ets-lindgren.com for more help if you have profiles made in TILE! 4 or 5; they are not directly forward compatible with TILE! 7.

Hi-Res monitor scaling adjustment

To help TILE! run on High-Res monitors, go change the High DPI scaling setting in Windows. First right-click the TILE! icon (if you have a shortcut) or tile.exe file, and select Properties. Then on the Compatibility tab press the “Change high DPI settings” button. From there check the “Override high DPI scaling behavior” checkbox and set the dropdown to “System” or “System (Enhanced)”.

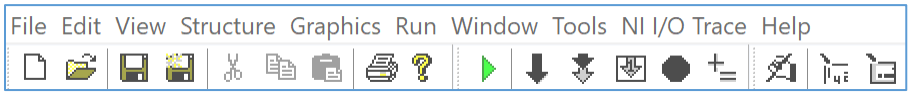


Navigation

This section will cover all the common elements that may be used to develop and execute profiles, such as toolbars, window views, and common features.

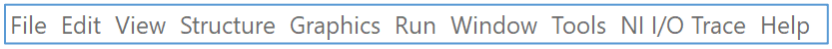
Toolbars

The tool-tip icons on allow you to hover your cursor over the icon to display the name of the tool.

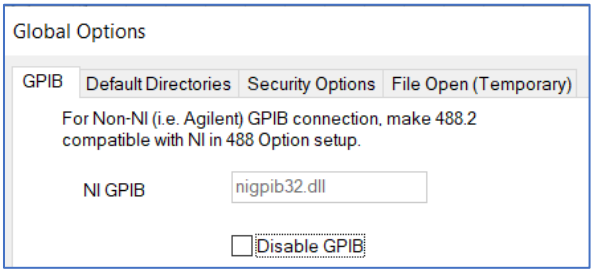


Main Menu Toolbar

The **Main Menu** at the top of the toolbar is used to access features of the window currently being used.



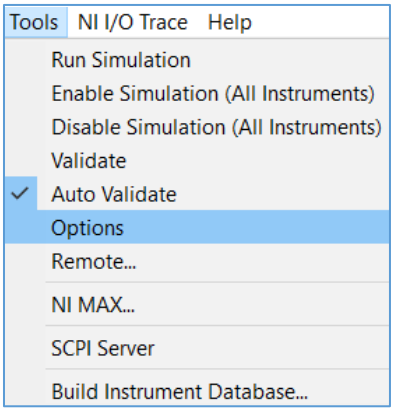
- **File** — This menu shows the typical file functions and their shortcuts such as New, Open, Save and Save As, and Print. There is also a list of recently opened profiles and the system options.
    - **Options** — This opens the **Global Options**, which are used to set up GPIB interface, database settings, preferred directories, and *security options*.
        - **GPIB** — Allows the user to de-select an instrument interface. If you are installing on a computer that will never communicate with instruments or use the simulator, you can select the **Disable GPIB** check box to disable checking for GPIB interfaces at startup.



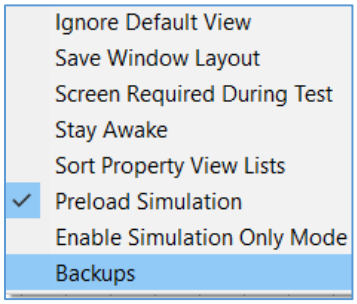
- **Edit** — This menu shows the typical edit functions specific to the selected window. If you accidentally erase an icon, immediately **undo** the mistake. **Redo** repeats a step. Using **Select All** and **Delete** together allow you to clear large areas of the flowchart. For single deletions and selections, it is easier to use the cursor.
- **View** — This menu indicates which view windows are being shown with a check mark.
    - **Snap to grid** — While checked, each action snaps to the grid squares. Uncheck this setting to move actions in smaller increments with the arrows to improve aesthetics.
    - **Zoom** — Zoom in or out on the flowchart to see the actions better.
- **Structure** — This menu manipulates the flowchart visibility settings.
    - **Group/Ungroup** — Select actions to group together for easier maneuverability.

- **Bring to front/ Send to back** — Sometimes actions overlap, so this allows you to choose which action is more visible by bringing it to the front. You can also hide actions by sending them to the back.
- **Align** — Select multiple actions and align them in relation to each other to improve the aesthetics of the flowchart.
- **Page size** — Resize the flowchart in relation to the default 8.5 inch square.

- **Graphics** — This menu will allow customization of the flowchart by adding graphic affects such as color, fill patterns, and fonts. Use this to add a bitmap file to the flowchart.
- **Run** — This menu has many of the same functions as the icons found in the **Command** toolbar. It also has some editing features that can access all the elements of the profile to evaluate or calculate on command.
- **Window** — This menu is used to access any tabbed window or order them in cascade or tile orientations. This menu can also be used to add new graphs, tables, or view layouts.
  - **NOTE**: Add a view to save different layouts of the windows to your profile.
- **Tools** — This menu shows additional tool options like the Simulator, NI MAX, SCPI Server, and Build instrument database.



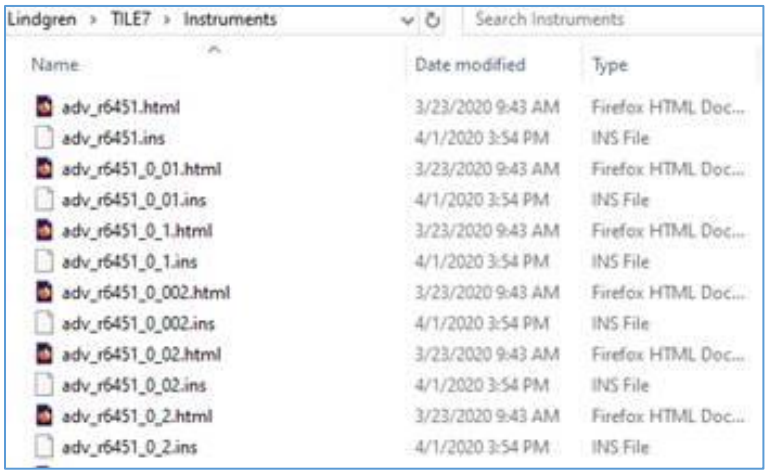
  - **Run Simulation** — Select this to start the instrument simulator.
    - **NOTE:** Not all drivers work with the simulator.
  - **Validate** — Manually validates a profile.
  - **Auto Validate** — Upon opening a profile, TILE! can validate it to find issues that may cause conflicts, such as similarly named actions, data, or instruments. This is useful when updating from prior versions of TILE! that did not check for these issues.
  - **Options** — Has many selections such as **Stay awake** and **Preload Simulation.**



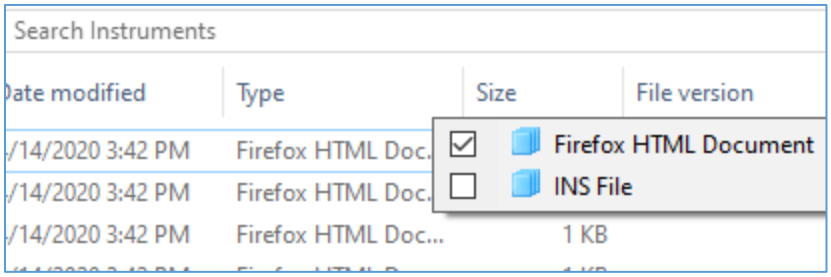
    - **Ignore Default View** — This is a global TILE! setting, not a profile setting. Use this to fix tables and graphs upon opening a profile. This setting does not affect currently opened profiles, only profiles as they are opening.
    - **Sort Property View Lists** — This global TILE! setting enables/disables sorting of these lists. If the Sort option is checked, new items added to the lists are automatically sorted. In addition, renaming an existing item will resort the list (it will be moved to the appropriate location based on its new name).

      NOTE: This sorting is different from using the list control column header sorting. Column header sorting is temporary, which is why closing and reopening a profile does not retain that sort result. The new Sort Data and Sort Instruments settings are permanent, in the sense that saving the profile will save the new order of the list items, and thus will reopen with the list still sorted.
  - **NI MAX** — Launches the NI MAX program. Use this to check instrument connectivity.

- **SCPI Server** — Open and close SCPI ports to control TILE! remotely. See the *TILE! scripting guide* document under the help menu for more instructions on how to use the SCPI server.
- **Build Instrument Database** — Refreshes the instrument database for a profile by removing and replacing deprecated files. TILE! will ask you if you want to update the instruments and overwrite the old ones. Once the update is done, some errors will occur related to the system "looking" for the instrument when the driver loads, but they can be ignored.
    - To clean the instrument database without uninstalling and reinstalling TILE!, do the following steps
        1. Navigate to the **C:\Program Files (x86)\ETS-Lindgren\TILE7\Instruments** folder in Windows File Explorer. You should see a mix of HTML and INS files.

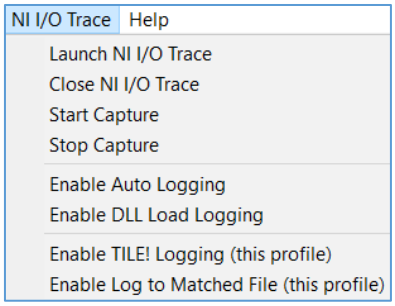


        2. Click the dropdown next to the Type column header and select the HTML Document checkbox.



        3. Click on any file, type Ctrl-A to Select All, then Delete.
        4. Run TILE! and do the Build Instrument Database (or run Explore from and Instrument config and you'll be prompted to do it).
        5. Now you should have a clean instrument DB.

- **NI I/O Trace** — This makes an external log. When asking for assistance from the TILE! support team, they will ask to see this log with the error. See *Native TILE! NI I/O Trace Support* for further details.

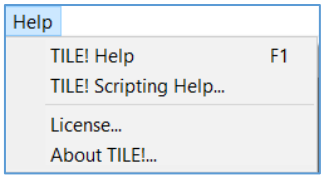


    - **Launch NI I/O Option** — Launches the NI I/O Trace utility if it is not already running.
    - **Close NI I/O Option** — Closes the NI I/O Trace utility if it is running.
    - **Start Capture** — Starts a capture in NI I/O Trace. If the utility is not already running, this option will launch the utility before starting the capture. This is equivalent to pressing the Start button in the NI I/O Trace toolbar, or selecting Tools->Start Capture from the NI I/O Trace menu.
    - **Stop Capture** — Stops a capture in NI I/O Trace. This is equivalent to pressing the Stop button in the NI I/O Trace toolbar, or selecting Tools->Stop Capture from the NI I/O Trace menu.
    - **Enable Auto Logging** — This is an App Level setting that enables Auto Logging mode using the NI I/O Trace utility. This setting completely automates the steps necessary for a user to generate an NI I/O Trace capture file (.nitrace extension) typically requested for debugging customer support issues.

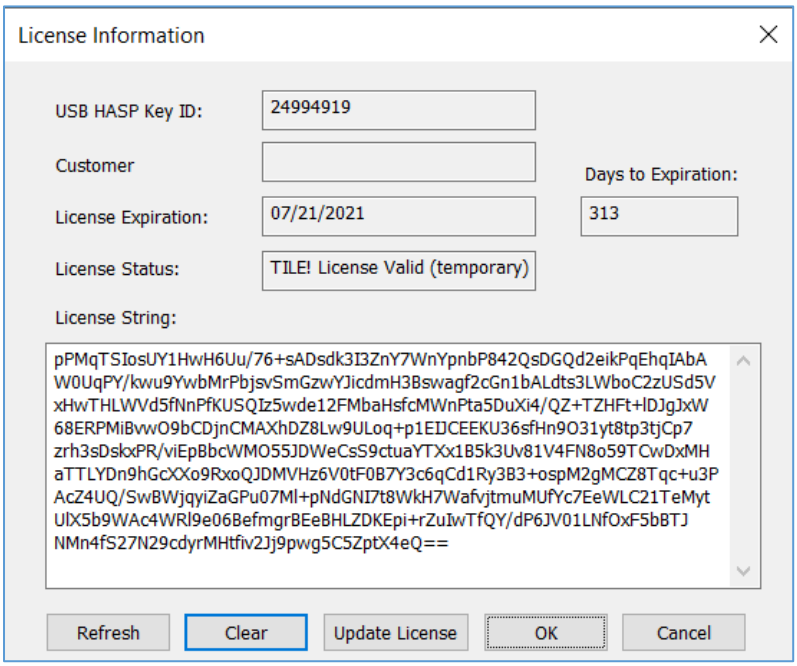
      When the setting is enabled, the following events occur when a profile is opened and closed:

        - When a profile is opened, the NI I/O Trace utility is launched and a capture is started. The File Logging mode is automatically enabled with the name of the log file set to match the name of the profile.
        - User runs the actions that are being investigated using NI I/O Trace debugging.
        - Close the profile. This will automatically close the NI I/O Trace utility, which in turn closes and saves the capture file.
        - Use Ctrl-E or File->Explore to navigate to the current directory and examine the generated .nitrace file. Double-clicking the .nitrace file in Windows Explorer will re-open the trace log in NI I/O Trace utility for viewing.
    - **Enable DLL Load Logging** — This is an App Level setting that enables logging of each action DLL load when a profile is opened. This may be helpful for ETS-Lindgren support engineers to debug a profile that is unable to load.
    - **Enable TILE! Logging (this profile)** — This is a Doc Level setting that enables TILE! log messages to output directly to the NI I/O Trace utility. TILE! log messages are the log entries that would typically be viewable from the TILE! Log View. Enabling this setting is equivalent to checking all five checkboxes under the NI I/O Trace column on the Options tab of the *Log Options* dialog.
    - **Enable Log to Matched File (this profile)** — This is a Doc Level setting that enables file logging in NI I/O Trace and sets the log file name to match the current TILE! profile. Enabling this setting is equivalent to checking the Enable File Logging and Match Filename to Current Profile settings in the *Log Options* dialog.

- **Help** — Access to documentation, and your TILE! license and version number.



    - **TILE! Help Manual** — Opens this manual.
    - **TILE! Scripting Help** — Opens a pdf with an examples of scripting and linear math functions as well as the full command set.
    - **License** — Shows your TILE! Hasp key ID, license expiration date and days to expiration, license status, and license string (used to update license).

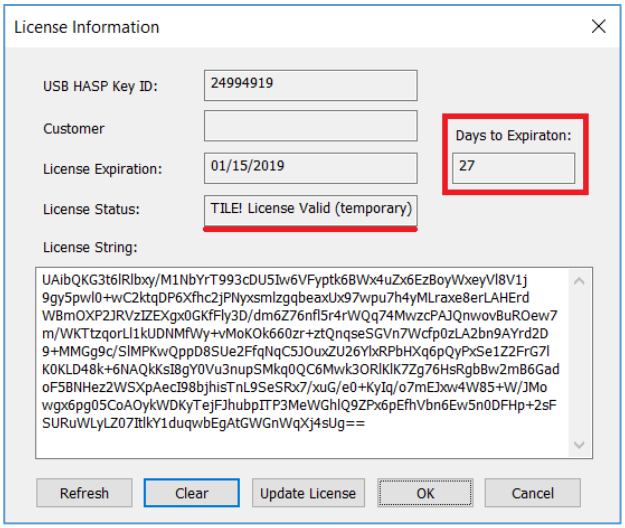


    - **About TILE!** — Displays the copyright, TILE! version number, and ETS-Lindgren contact information.

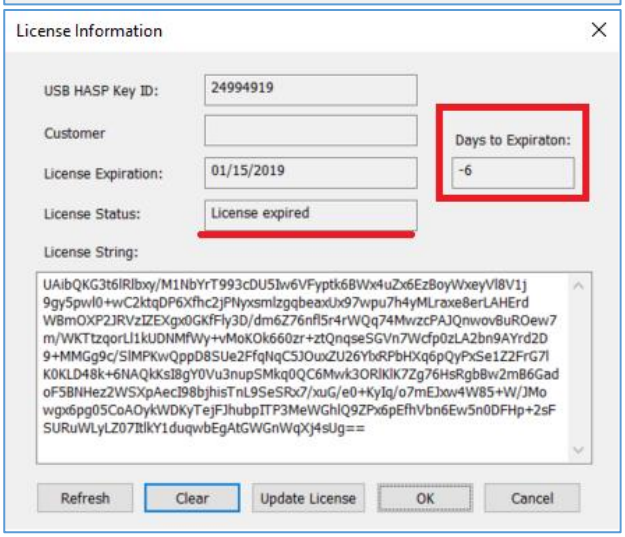
Expired License

To determine whether your TILE! license is expired or not, go to TILE! >> Help >> License.

- **Valid** — If the current date is less than the expiration date, the license status is valid and the Days to Expiration will be a positive number.
    - While valid, the license has access to all functions paid for at start of license term.



- **Expired** — Once the license is no longer valid, the status will change to Expired and the Days to Expiration will be negative.
    - **NOTE**: The license will still work, BUT you will lose access to the portal and maintenance help, as well as some features like Simulation.



Update License

To update your TILE! license, go to TILE! >> Help >> License.

NOTE: If you have any issues or need help, please contact TileSupport@etslindgren.com.

1. Contact the Customer care team to renew your license contract, and they will send you an email with your new license string.
2. Open the email with the TILE! License string:
    a. Click inside the email
    b. Press Ctrl + A
    c. Press Ctrl + C

Here is your TILE! License, which will expire on 12/31/2018.

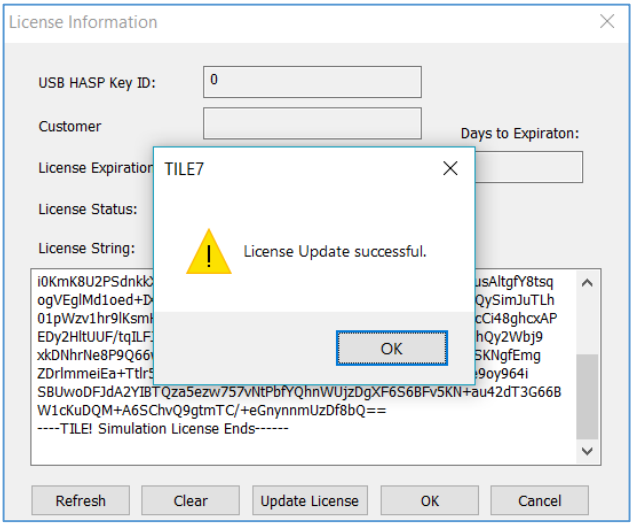
This license is only valid for your HASP USB key with Key ID = 2016260311.

Copy the contents of this e-mail into the TILE! Help->License... menu's License String input and press Update License, while the HASP USB key is plugged into the computer. Make sure all other HASP keys you may have are not plugged in simultaneously while this key is being programmed.

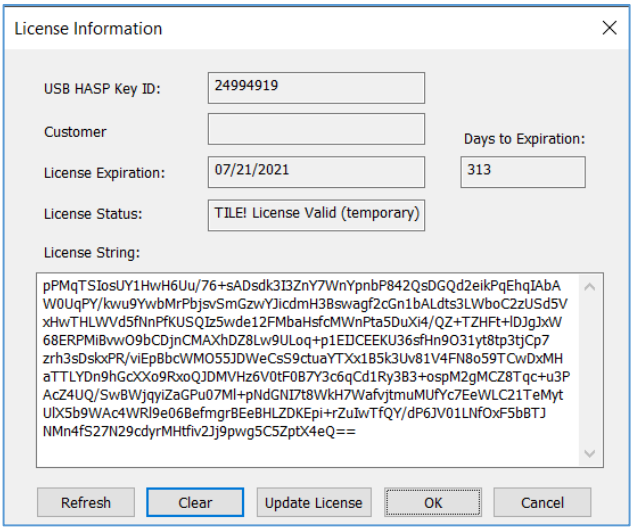
----TILE! Simulation License Begins----
gtvz1+2tU9Z3qI52eFRTTP85aYqhB/S2DdOIoNRqNH4q+Yi8CJMZk1E8NXo5gCW0
i0KmK8U2PSdnkkXU64aMkXumfuS1nHzbSHYNqbpX1p76pH4CLN1musAltgfY8tsq
ogVEglMd1oed+IXao1QIkVMuzuSQPaNRMBk7R6fxMVGN0I3HpPVfFJQySimJuTLh
01pWzv1hr9lKsmHBp3O2QewVGr1w+ml1JzIX+sDF/9oylwEq+5A2WcCi48ghcxAP
EDy2HltUUF/tqILFJzjH7WuoS6+YoylDwMnGweSueum/iZJnPPevLoC1hQy2Wbj9
xkDNhrNe8P9Q66wk9e6vb1bG4hlJ5vuGsv3ythxeK6tIGS7fYScMUR4RSKNgfEmg
ZDrlmmeiEa+Ttlr5kg534BgYh2UfDzYizbLcMpIduIDa4w+QBEgdHonve9oy964i
SBUwoDFJdA2YIBTQza5ezw757vNtPbfYQhnWUjzDgXF6S6BFv5KN+au42dT3G66B
W1cKuDQM+A6SChvQ9gtmTC/+eGnynnmUzDf8bQ==
----TILE! Simulation License Ends------

3. Plug in the TILE! USB key and make sure the red indicator light is on.
    a. NOTE: If you are also installing an updated version when renewing your license, then uninstall the old version, install the new version, and reboot before updating your license.

4. Run TILE! and go to the TILE! menu Help > License
    a. Click the Clear button to remove the current license string
    b. Click inside the license string box
    c. Press Ctrl + V
    d. Click Update License
    e. If successful, you should see the popup to the right.



5. Go to TILE! >> Help >> License and the license status should now say valid.



Standard Toolbar

The **Standard** toolbar has the tooltip versions of the typical file functions and their shortcuts such as New, Open, Save and Save As, and Print.

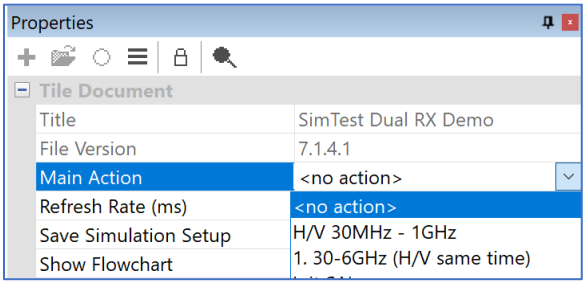
- **Print** — Prints the flowchart pages.
- **Help** — Displays the copyright, TILE! version number, and ETS-Lindgren contact information.

Command Toolbar

The **Command** toolbar is used to execute actions in the flowchart. The arrow icons bring up a window with all the actions in the profile that can be selected as a starting point.

- **Green Triangle** (**Run Main Action**) — This button allows a single click location to start the profile from the selected action. This is normally set to a Prompt action that functions as a Main Menu.

    - To access the Main Action, select a blank spot in the flowchart and go to the properties window. The properties of the flowchart will have an input for the 'Main Action'.
    - Executing from the Run Main Action allows the flowchart to be hidden behind the graphs and tables.



- **Single dark arrow** — Executes the flowchart chain starting with the selected action.
- **Double arrows** — Steps through the actions in order of the flowchart connections.
- **Square single arrow** — Executes a single action on the flowchart.
- **Octagon** — Stops a process.

- **NOTE** that it may not stop a currently running GPIB transaction, but it will stop at the next available step.

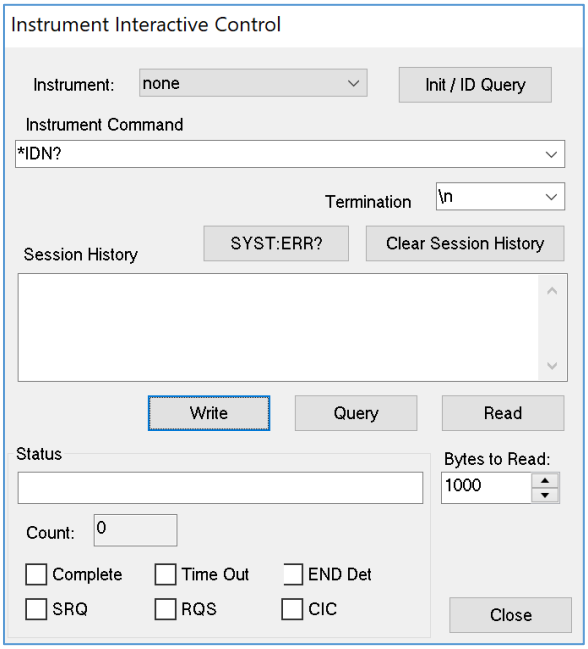
- **+/= button** — Performs all math functions in order from top to bottom as they are listed in the data elements section.

View Toolbar

The **View** toolbar contains TILE! specific icons to add data elements and instruments, and to open the GPIB Interactive Instrument Control.

- **Interactive Instrument Control** — This window interacts with instruments using remote commands. This is useful to validate commands used in the 'VISA' action or to debug driver interaction.

NOTE: You must have at least one instrument configuration to open the Interactive Instrument Control Utility.



  - **Instrument** — Selects an instrument from your list of instruments.
    - **Init / ID Query** — Performs a query to see if the instrument is connected and available to communicate.
  - **Instrument Command** — Allows direct input of commands to send to the instrument. The list will remember recent commands and has a few common commands such as *OPC?.
    - **Termination** — Default is set to line feed '\n'. Carriage return '\r' is also common. Look at the instrument's command set in case of a different termination character.
    - **SYST:ERR?** — Queries the instrument to return the latest error in its queue.
    - **Clear Session History** — Clears previous commands from the Session History.
  - **Session History** — Displays the queries and responses from the instrument.
    - **Write** — Sends the command displayed in **Instrument Command** to the instrument.
    - **Query** — Queries the instrument about the current settings.
    - **Read** — Reads the response from the instrument.
  - **Status** — Shows the status of the communications with the instrument, such as if the operation was successful or an error occurred.

- **NOTE**: The following checkboxes only appear if the selected instrument's communication address is set to GPIB: **Complete**, **TimeOut**, **END Det**, **SRQ**, **RQS**, and **CIC**.

- **Add Data** — One of the ways to add a new data element to the profile. See the section on *Data Elements* for more information.

- **Add Instr** — One of the ways to add a new instrument to the profile. See the section on *Instruments* for more information.

TILE! Windows

Window Behavior (Docked and Tabbed Modes)

From the **View** toolbar menu, the user can select a global **Docked** or **Tabbed** mode. The standard components of a profile are always available as tabbed windows. They can be moved to any edge, pinned, or set to Auto Hide to suit the user.

- **Tabbed mode** — Allows one click tile or cascade orientation from the Window menu (Legacy behavior).

- **Docked mode** — Maximizes the current window and indexes all views as tabs to that view. It will also allow any window to be docked to any edge of the profile view.

Flowchart View

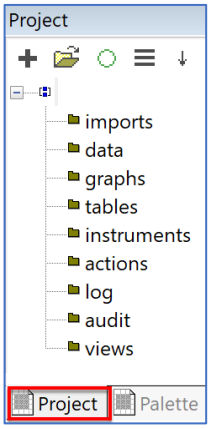
The flowchart is the sequence of actions to perform a specific test. The flowchart is a visual representation of a test profile, where each icon represents an independent step in the test. Each icon represents a miniature program that can be completely configured as a standalone step. The flowchart is natively docked in the center of the profile. See the section *Flowchart Window* for more information.

Palette View

The palette is a collection of all the actions you can use. The palette is natively docked on the left side of the profile along with the project window. It can be re-sized by selecting and moving the edges, as well as be undocked and moved if desired. See the *Palette* section for more information.

Project View

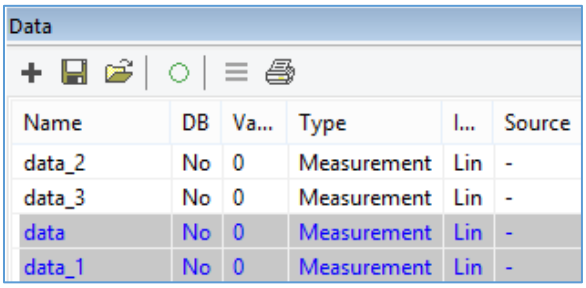
The Project view provides an easily accessible view that contains all the components of the profile. Selecting a component from this view will bring it to the forefront for viewing or editing. The Project view is natively docked on the left side of the profile along with the palette window.



Imports

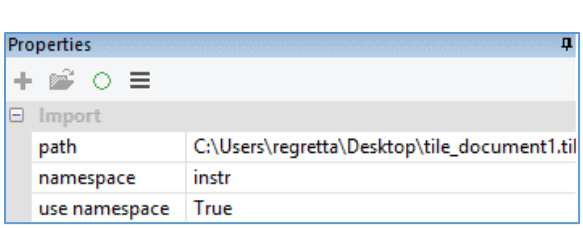
Importing allows the user to include other TILE! profile items into their profile. Importing is like using an include file; anytime the root file is updated, the imported data is also updated.

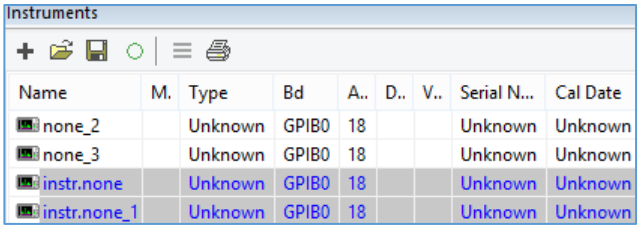
6. Currently we only allow importing of instruments and data items.
7. These imported items can be used in your profile like any other TILE! items.
8. Using this feature, you can have a single profile that describes ALL of your instruments:
    a. The instruments never have to be copied to a new profile.
    b. Changes to the imported profile will trickle through to all profiles that included it.
    c. A change to an instrument or script can be done in a single location.
9. The imported items appear greyed. You can open them, but you cannot save changes to them.



Name	DB	Va...	Type	I...	Source
data_2	No	0	Measurement	Lin	-
data_3	No	0	Measurement	Lin	-
data	No	0	Measurement	Lin	-
data_1	No	0	Measurement	Lin	-

10. If you wish to edit the imported profile, you can double click on the “import profile” in the project tree. This will launch another instance of TILE!.
    d. **NOTE**: These changes will appear in all other TILE! documents that import the profile.
11. An **optional** namespace feature can be turned on via the properties view. This feature adds a prefix name to the imported items. This is valuable to organize instruments and data by a meaningful usage name, for example “ImmunityScripts” and “GtemInstruments”.



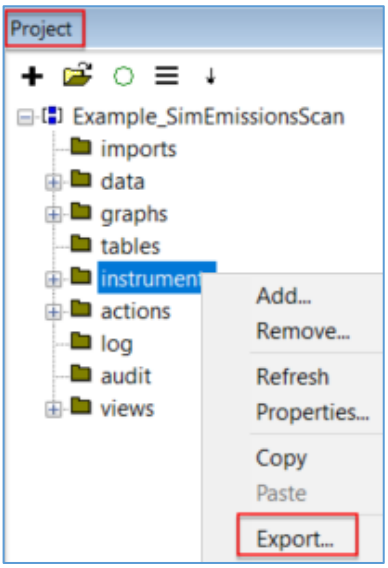


Exports

To export the settings of any object in TILE!, go to the project view and right click on the type of object then select export. This opens the File Explorer to save the object settings as XML or DAT files. You can also do this for anything in the project view.

For example, to save time creating a long list of instruments from scratch, export all the instruments settings from a template profile to use in a new profile that uses the same instruments.

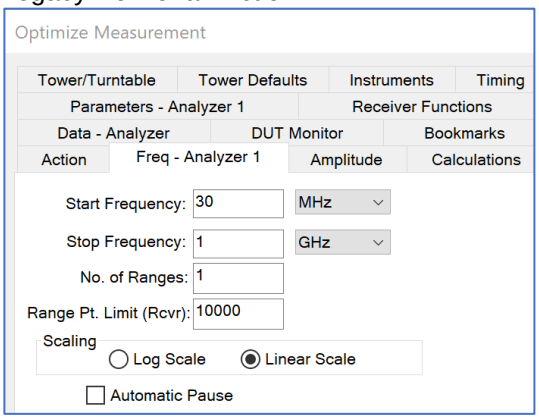
To use the exported xml list in another profile, just drag and drop the file from the File Explorer into the profile and the objects will appear with all the settings from the other profile.



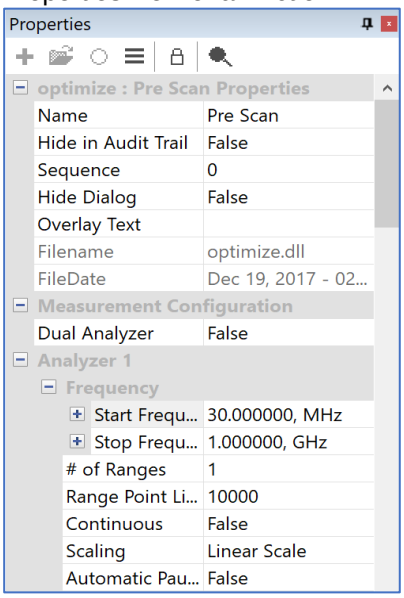
Properties View

The Properties view allows better viewing of each component of the profile. This view is an updated interface of the normal tabbed (legacy) editing view, and both are available to use based on preference. Many of the new or extended features can only be accessed through this view and this feature is available for actions, graphs, tables, data elements, and instruments.

Legacy view of an Action:



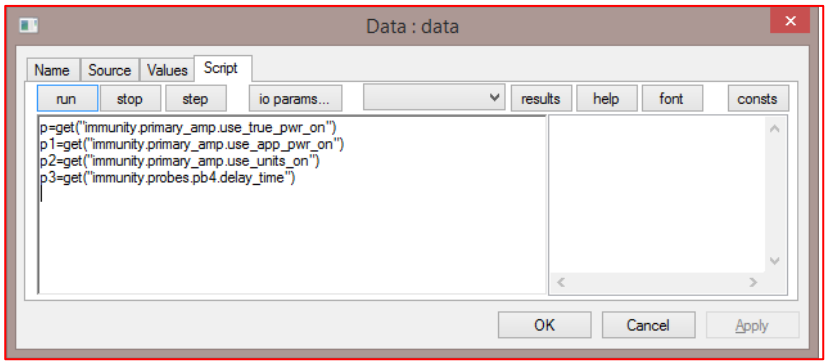
Properties view of an Action:



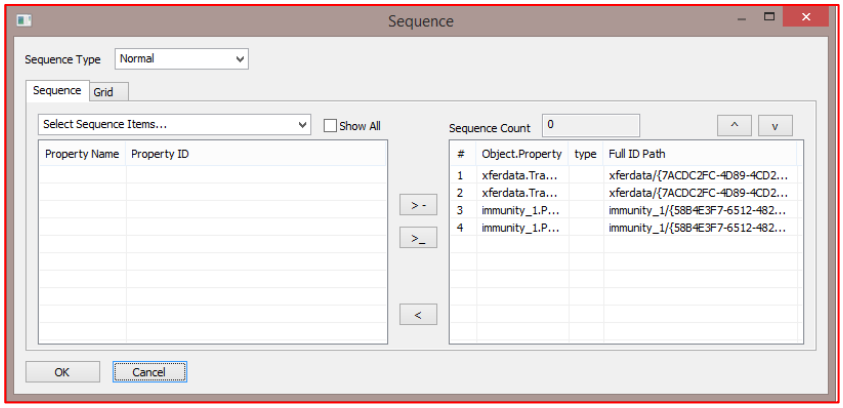
Drag and Drop properties

When a property is drag and dropped, it will be converted into the proper scripting syntax to get the value. You can drop and drop properties onto the following:

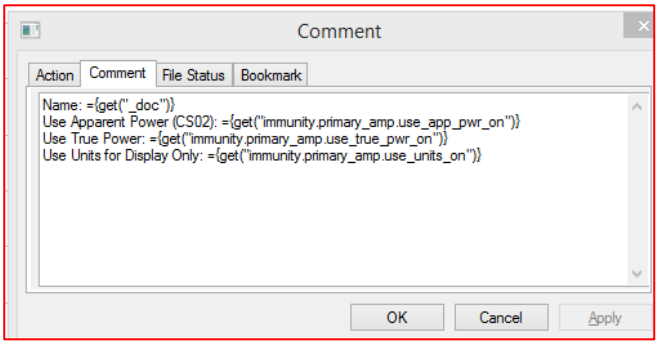
1. Script Editor



2. Sequence Editor



3. Comments Action



Data Element View

Defined data elements are used to store information obtained during the test. Data elements can hold measurement values, specification limits, correction factors, or the results of mathematics upon other data elements.

Data elements can be accessed with the data element window natively docked on the bottom of the profile or by looking under ‘data’ on the project window. See the *Data Elements* section for more information.

Instrument View

The defined list of instruments that will be controlled during the test. Each instrument item includes the instrument driver and the GPIB or serial port information necessary to address the instrument properly. Instruments can be accessed with the instrument window natively docked on the bottom of the profile or by looking under ‘instruments’ on the project window. See the *Instruments* section for more information.

Graph View

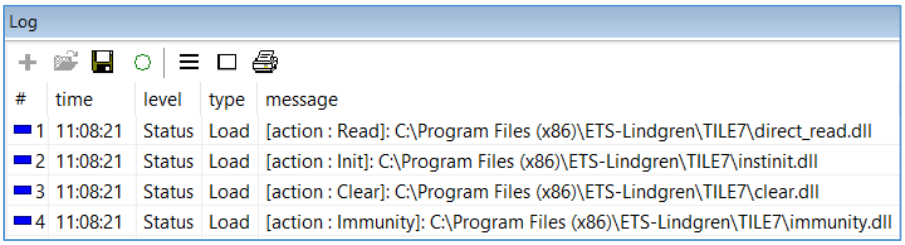
Graphs are a visual form of data presentation. Multiple graphs can be created in each profile on the Graph window. Graphs can be accessed by looking under ‘graphs’ on the project window. See section *Graphs* for more information.

Table View

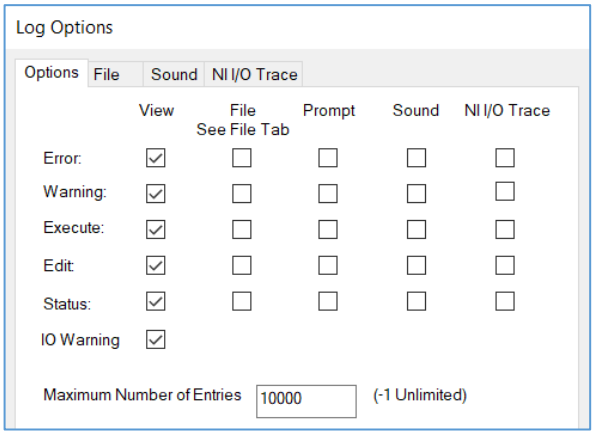
Tables are a spreadsheet form of data presentation. Multiple tables can be created in each profile on the Tale window. Tables can be accessed by looking under ‘tables’ on the project window. See section *Tables* for more information.

Log View

The log view is used for debugging purposes, and allows the user to track the completion/status of each step of the test. The log can show a complete history of the commands sent to the various instruments in the test. The log is natively docked on the bottom of the profile. To change the Log options, click on the 3 horizontal lines icon on the log view window.



It is highly suggested to enable all the View column settings. For the NI I/O Trace suggested settings for trouble shooting, see *Detailed Trace Settings*.



Ignore/Replace Commands

Use the log view right-click feature **Ignore/Replace Commands** to skip, modify, or add specific commands to change the behavior of an instrument object.

1. When the driver is sending out a command that causes the instrument to produce an error.
    a. Use the Ignore Command feature to skip sending out this command.
2. When using a model of the instrument that is slightly different than what the driver expects.
    a. Replace commands to produce an output compatible with the model in use.
3. When the driver is missing an option or command sequence that the instrument expects to be present.

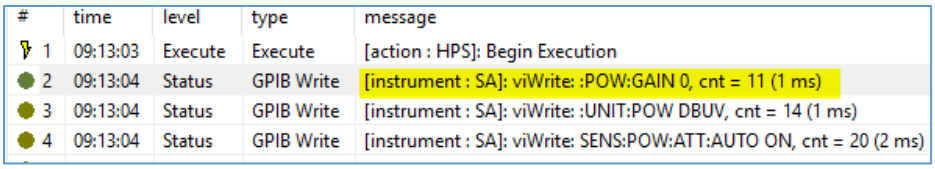
a. You can add entries in the “Pre Command” or “Post Command” slots to add additional commands before or after other commands in the sequence.

NOTE: Using the Ignore/Replace Command feature does NOT permanently change the instrument driver; it only effects the particular instrument object you edited.

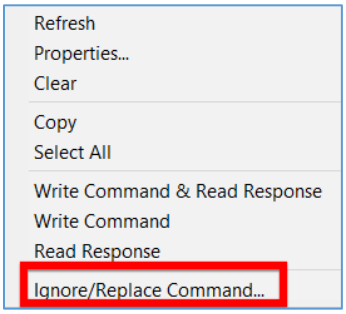
Example of Ignoring a Command:

While running the Measure Range action, there is a settings conflict error for your SA instrument using the “:POW:GAIN 0” command. Rerun the action if needed to generate the log.

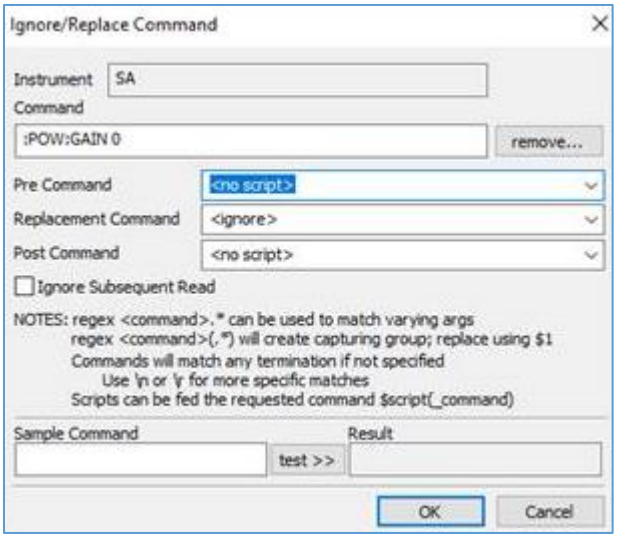
1. In the log, find the “:POW:GAIN 0” command entry.



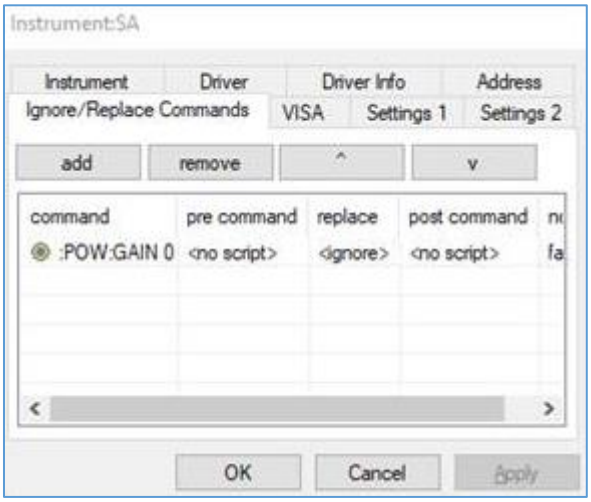
2. Right-click the entry and select Ignore/Replace Command…



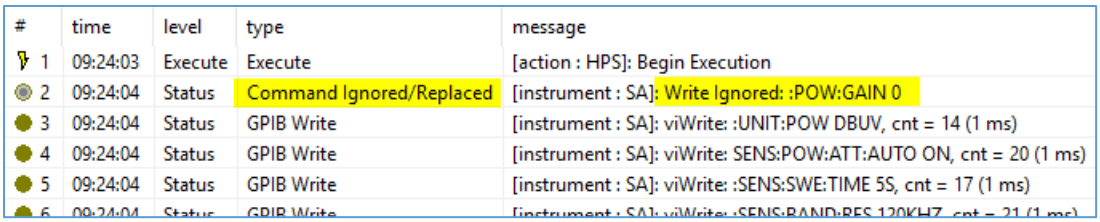
3. To just ignore (not replace or modify) the command, keep the default the dialog entries and click the OK button. This records an Ignore Command setting for “:POW:GAIN 0” to the SA instrument, which means whenever the driver tries to write this particular string it will not output to the bus.



4. You can view the recorded entry by editing the SA Instrument. You should see a new Ignore/Replace Commands tab as part of the instrument configuration. Click that tab to see the ignore entry for ":POW:GAIN 0":

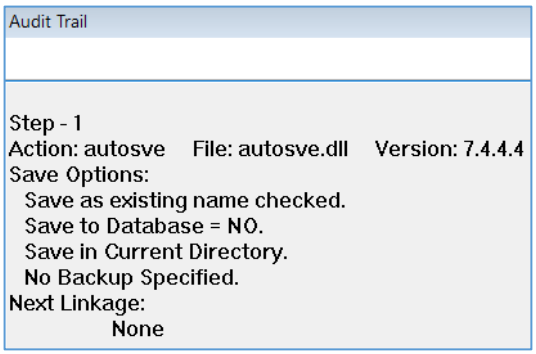


5. Re-run Measure Range and the settings conflict error should not occur. If you view the TILE! log, you will see that the ":POW:GAIN 0" command has been marked as Ignored and therefore was not sent to the instrument:



Audit Trail View

A text format record of the configurations of each action present on the flowchart. The Audit Trail is natively docked in the center of the profile on the tab next to the flowchart when selected under the toolbar Views.



Common Action Features

The Windows operating system environment offers many common tools for dealing with a graphical environment. For ease of use, TILE! uses as many common features as possible. The most common

features are the buttons that appear in most windows. Whenever the following terminology is used, the meaning is the same throughout the program.

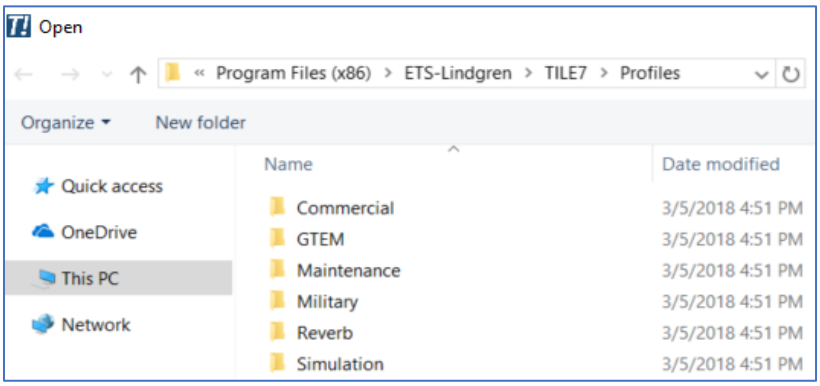
- **Enter Key** — The Enter key accepts the entered information. This may not have the desired effect when you are using a dialog box. The Enter key has the same meaning as the OK button when using a dialog box. When a box has more than one area of the screen requiring information, you must use the Tab key or the cursor to maneuver around the dialog box.
- **Tab Key** — The Tab key is the primary method of maneuvering between different fields in a dialog box when using the keyboard. You can always use the cursor to move between entry points, but when typing use the Tab key.
- **OK** — The OK button accepts the input for the currently open dialog box, and continues with the appropriate modification or operation specified in the flowchart.
- **Cancel** — The Cancel button discards all changes made to the dialog box inputs. The flowchart remains the same as before the dialog box was opened. If you are in the middle of an execution, all execution is halted and the program returns to the flowchart.
- **Apply** — The Apply button is used in certain program applications, and is present in many standard dialog sequences. This button has specific uses in the TILE! program but generally causes the same action as the OK button without closing the dialog box.
- **Help** — This selection will open the reference manual.

Create a Basic Profile

There are four main steps to creating a test profile:

1. Identify the instruments.
2. Define the data elements.
3. Set up the sequence of the test actions using the flowchart.
4. Graph and tabulate the data.

NOTE: ETS-Lindgren has many premade shipping profiles to use in the TILE7 Profiles folder in the Program Files(x86) location of your computer (*C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles*). If you use one of these profile templates, you should save the profile under a new name before making changes, so you do not lose the original profile template.



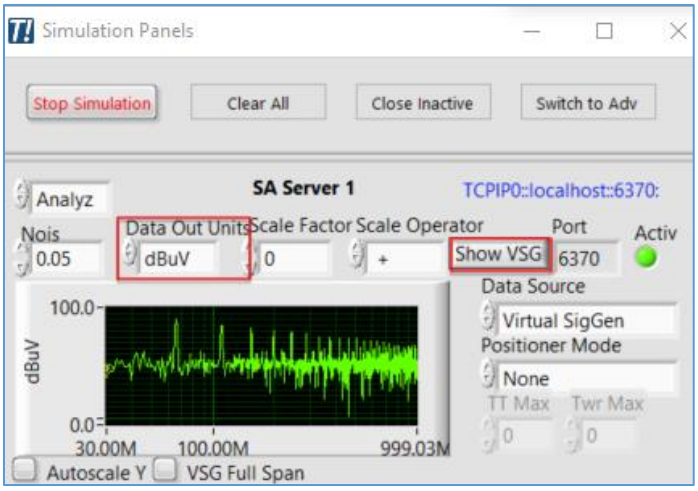
Simulated Emissions Scan

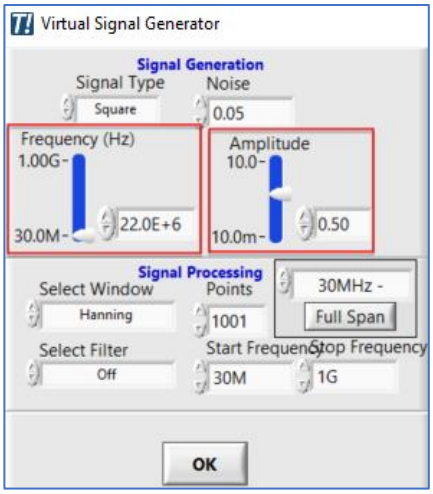
This example shows how to make a simple emissions profile from a blank new file using the built in instrument simulator:

1. Open a new TILE! workspace, File -> New.
2. Create a data element:
   - Go to the Data Elements window and select the **+** to create a new data element.
   - On the Name tab, label it "***Data_Scan***" in the name field.
   - Click OK to save the new data element and close the window.
3. Create an Instrument:
   - Go to the Instrument elements window and select the **+** to create a new instrument.
   - On the Instrument tab, label it "***Instrument_SA***" in the name field.
   - On the Driver tab, and click the Explore button. Type "MXA" in the search field to find the Agilent MXA Spectrum Analyzer driver ***ag_mxa.ins***, and then double click it to select.
   - On the Address tab, set the Communication to ***Simulation***.
   - On the Sim tab, set the Sim Type to ***Analyzer***.

- Click OK to save the new instrument and close the window. The simulated analyzer will now boot up on the simulation panel.

4. Set up the Simulation Panels:
   - On the simulated SA Server 1 panel, set the Data Out Units to ***dBuV***, and then select ***Show VSG***. The Virtual Signal Generator window will appear.
   - On the Virtual Signal Generator window, set the Frequency to ***22e6*** (22 MHz) and set the Amplitude to ***0.5***.
   - Click OK to save the settings and close the window.



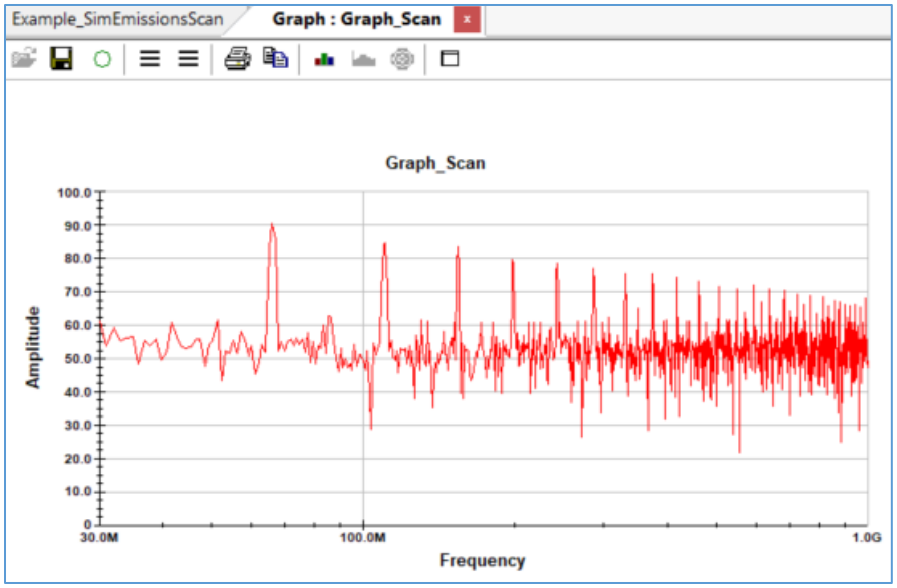


5. Add the Measure Range Extended action to flowchart:

   - Go to the Palette view and with a single click, select the Measure Range Extended icon and single click again to place it anywhere in the flowchart.
   - Right click on the Measure Range Extended icon and select Edit.
   - On the Action tab, label it "***Action_Scan***" in the name field.
   - On the Data1 tab, set the Measurement Data to ***Data_Scan***.
   - On the Instruments tab, set the Receiver to ***Instrument_SA***.
   - Click OK to save the new action and close the window.

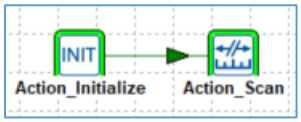
6. Create a graph:
   - On the Project view, select the **Graphs** folder and then click the **+** to add a new graph.
   - On the Graph toolbar, select the **left most 3 horizontal lines** icon to open the display data.
   - On the Titles tab, label it "***Graph_Scan***" in the title field.
   - On the Graph toolbar, select the **right most 3 horizontal lines** icon to open the display data.
   - On the Lines tab, add the data element ***Data_Scan*** to the column on the right.
   - On the Data_Scan tab, set the line color to ***LtRed***.
   - Click OK to save the display data and close the window.

7. Save the program:
    - Save your Simulated Emissions Scan profile, File -> Save As. Save is in your chosen location with the name "***Example_SimEmissionsScan***".
    - NOTE: Always save before testing the profile in case the execution crashes the profile and loses all the data.
8. Test the program:
    - On the Properties window, set the **Main Action** for the flowchart to ***Action_Scan***.
    - Run the profile with **Run Main Action**.
    - As shown below, the graph will show a trace that emulates the signal from the Virtual Signal Generator:



NOTE: This profile can be adapted from simulation to use a real instrument with the following changes.

1. If needed, change the driver for ***Instrument_SA*** on the driver tab.
2. On the Address tab, change the Communication from ***Simulation*** to match your instruments connection type (**GPIB** or **VISA** are the most common). The Sim tab will change to reflect the new type of communication; match the connection settings to the instrument.
3. If the instrument does not produce a trace, you may have to initialize it with the INIT action. Name the INIT action ***Action_Initialize*** and select ***Instrument_SA*** on the instrument tab.



4. Use the connector tool to attach ***Action_Initialize*** to ***Action_Scan***, and change the Main action to ***Action_Initialize***.
5. Save the changes then run the profile with **Run Main Action**. ***Graph_Scan*** now shows the real world instrument trace.

Polar Plot with Simulator

This example shows how to make a polar plot profile from a blank new file using the built in instrument simulator:

1. Open a new TILE! workspace, File -> New.
2. Create a Signal Generator Instrument:
    - Go to the Instrument elements window and select the **+** to create a new instrument.
    - On the Instrument tab, label it "***Instrument_SG***" in the name field.
    - On the Driver tab, and click the Browse button. Search for the driver ***ag_mxg.ins***, and then double click it to select.
    - On the Address tab, set Communication to ***Simulation***.
    - On the Sim tab, set the Sim Type to ***SigGen***.
    - Click OK to save the new instrument and close the window.
3. Create a Spectrum Analyzer Instrument:
    - Go to the Instrument elements window and select the **+** to create a new instrument.
    - On the Instrument tab, label it "***Instrument_SA***" in the name field.
    - Next, go to the Driver tab, and click the Browse button. Search for the driver ***rs_esu_sa.ins***, and then double click it to select.
    - On the Address tab, set Communication to ***Simulation***.
    - On the Sim tab, set the Sim Type to ***Analyzer***.
    - On the Settings tab, set the Points to ***1001***.
    - Click OK to save the new instrument and close the window.
4. Create a Control Instrument:
    - Go to the Instrument elements window and select the **+** to create a new instrument.
    - On the Instrument tab, label it "***Instrument_TT***" in the name field.
    - Next, go to the Driver tab, and click the Browse button. Search for the driver ***EMCO2090.ins***, and then double click it to select.
    - On the Address tab, set Communication to ***Simulation***.
    - On the Sim tab, set the Sim Type to ***Turntable***.
    - Click OK to save the new instrument and close the window.
5. Create a Data element:
    - Go to the Data Elements window and select the **+** to create a new data element.
    - On the Name tab, label it "***Data_TT***" in the name field.
    - Click OK to save the new data element and close the window.
6. Add the INIT action to flowchart:

- Go to the Palette view and with a single click, select the INIT icon and single click again to place it anywhere in the flowchart. Right click on the INIT icon and select Edit.

- On the Action tab, label it "***Action_Initialize***" in the name field.
- On the Instruments tab, add the instrument ***Instrument_SA*** to the selected side.
- On the Instruments tab, add the instrument ***Instrument_SG*** to the selected side.
- On the Instruments tab, add the instrument ***Instrument_TT*** to the selected side.
- Click OK to save the new action and close the window.

7. Add the Polar action to flowchart:

   - Go to the Palette view and with a single click, select the **Polar/Scalar** Measurements icon and single click again to place it anywhere in the flowchart.
   - Right click on the Polar icon and select Edit.
   - On the Action tab, label it "***Action_Polar***" in the name field.
   - On the Frequency tab, set the Frequency to ***675***.
   - On the Instruments tab, set the Receiver to ***Instrument_SA*** and the Turntable to ***Instrument_TT***.
   - On the Input/output tab, set the Turntable to ***Data_TT***.
   - On the Optimization Process tab, select ***Turntable*** and ***Stepped*** and ***Horizontal***. Set the turntable start to ***0***, stop to ***360***, step to ***10***, and margin to ***1***.
   - On the Power tab, set the Signal Generator to ***Instrument_SG*** and set the Signal Level to ***-60***.
   - Click OK to save the new action and close the window.

8. Add the Clear action to flowchart:

   - Go to the Palette view and with a single click, select the **Clear** icon and single click again to place it anywhere in the flowchart.
   - Right click on the Clear icon and select Edit.
   - On the Action tab, label it "***Action_Clear***" in the name field.
   - On the Clear Data tab, add the data ***Data_TT*** to the selected side.
   - Click OK to save the new action and close the window.

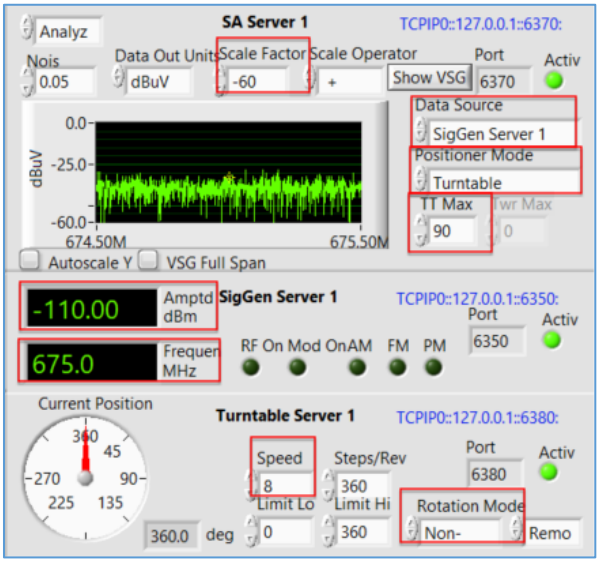
9. Create a graph:
   - Go to the Project view, select the Graphs folder and then click the **+** to add a new graph.
   - On the Graph toolbar, select the **polar** icon to view azimuth centric data.

   - On the Graph toolbar, select the **right-most 3 horizontal** lines to open the display data.
     - ◊ Add the data element ***Data_TT*** to the column on the right.
     - ◊ On the TT tab, set the line color to ***LtRed*** and set the width to ***2***.

- ◊ Click OK to save the display data and close the window.

- On the Graph toolbar, select the left-most 3 horizontal lines to open the graph setup.
  - ◊ On the Titles tab, set the title of the graph to "***Graph_PolarPlot***".
  - ◊ On the X Labels tab, set the min to ***0 Hz*** and the max to ***360 Hz***, unselect the **Log** checkbox, and set the format to ***Decimal***.
  - ◊ On the X-axis tab, set the step to ***36*** and select the **Minor** checkbox.
  - ◊ On the Y Labels tab, set the min to ***-100***, the max to ***100***, and the format to ***Decimal***.
  - ◊ Click OK to save the graph setup and close the window.

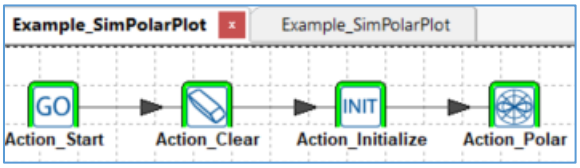
10. On the Simulation Panel:



- To open the Configure Simulation panel, go to Tools -> Run Simulation. On the panel, click the Run Simulator button.
- On the SA Server 1, make sure the scale factor is set to ***-60***, the data source is set to ***SigGen Server 1***, the positioner mode is set to ***turntable***, and the TT Max is set to ***90***.
- On the SigGen Server 1, make sure the amplitude says ***-110 dBm*** and the frequency is ***675*** MHz.
- On the Turntable Server 1, make sure the speed is set to ***8***, and the rotation mode is set to ***non-cont***.

11. Connect the actions on the flowchart and create one start click for the profile:

- Go to the Palette view and with a single click, select the Start (**GO**) icon and single click again to place it anywhere in the flowchart.
- On the Action tab, label it "***Action_Start***" in the name field.
- Use the connector tool from the palette to the actions on the flowchart in the following order: ***Action_Start***, ***Action_Clear***, ***Action_Initialize***, and ***Action_Polar*** action.



- Go to the Properties view and set the **Main Action** to ***Action_Start***. This allows the sequence to run from the toolbar's Run Main Action.

9. Save the program:
   - Save your Polar Plot profile, File -> Save As. Save is in your chosen location with the name "***Example_SimPolarPlot***".
   - NOTE: Always save before testing the profile in case the execution crashes the profile and loses all the data.

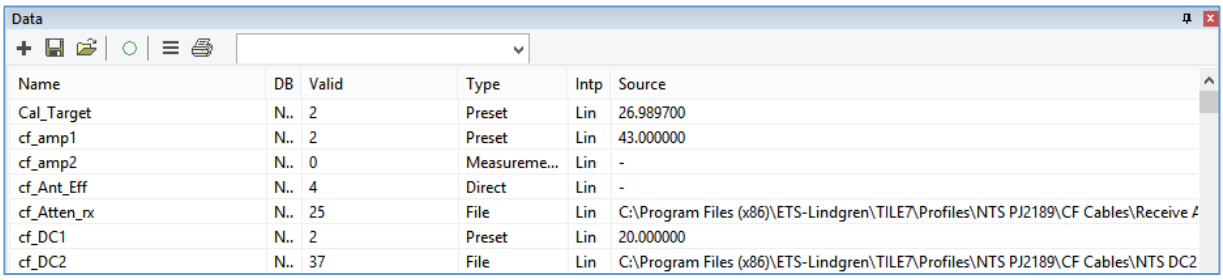
10. Test the program:
    - Run the profile with **Run Main Action**.
    - To alter the results:
        - ◊ If you change the TT Max on the Simulator, then the data will be skewed to make that degree point number the highest as if you are maximizing a signal.
        - ◊ If you increase the Scale Factor on the Simulator, you will see the amplitude increase on the polar plot as if your signal was getting higher.

Data Elements

Each data element is an array of amplitude versus frequency values. These data elements are filled from an instrument measurement, a file, an equation, or a preset value. The only practical limit to the number of data elements you can define is the need and the available memory of the system.

Data Element View Window

The Data window displays all the currently defined data elements. The data element view updates to allow moving and re-sizing of the window, exporting, and sorting of the columns. Each data element displays the **name** of the data element, whether the **database** option is selected, and the number of **valid** points, the data **type**, whether it is **interpolated**, the relevant **source**, the numerical **values**, and any necessary **comments**.



Name	DB	Valid	Type	Intp	Source
Cal_Target	N..	2	Preset	Lin	26.989700
cf_amp1	N..	2	Preset	Lin	43.000000
cf_amp2	N..	0	Measureme...	Lin	-
cf_Ant_Eff	N..	4	Direct	Lin	-
cf_Atten_rx	N..	25	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\Receive A
cf_DC1	N..	2	Preset	Lin	20.000000
cf_DC2	N..	37	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\NTS DC2

Element Names

NOTE that each data element **must** be given a unique name. The name can contain both uppercase and lowercase characters, as well as most other characters. Use under_score or camelCase to differentiate separate words for an element name.

Do not use spaces since a space in the name will make it impossible to use it in an equation.

Do not use periods since it can truncate the rest of the name after it.

You will find when creating test profiles that using a name that gives reference to its source is very convenient. Calling a reading from an analyzer raw will convey the essence of this element as well as its use.

Most lists that utilize data elements will automatically sort alphabetically; keep this in mind when creating data element names. We have started a naming convention for the data elements to group them together to simplify the search of update factors.

For example, E_Pks would be the equation to find peaks in an EMC Scan.

- E = equation
- F = Factor
- L = Limit
- M = Measurement

Element Database

NOTE: This column is only meaningful if the TILE/DB database option is installed.

If this option is installed, use the **Save to Database** check box on the *Source tab* to specify whether to update the database for this data element. If it is marked for saving, the data element is copied to the database table of results.

Element Valid Points

This column displays the current number of valid points in a data element. When first created, measurement elements are empty and display a value of zero. All other data elements are filled with data upon their creation (or file opening), and this number indicates the number of valid points in the table.

You can see the data points when editing the data element on the *Values Tab*. The **Measurement**, **Equation**, and **Word** types will not display any valid points until the data has executed. The **File** and **Preset** types will display valid points as soon as they are defined.

Element Intp (Interpolated)

When EMC measurements are performed, they generally have large data arrays of information. When applying specification limits, correction factors, and other offsets, we need to ensure a valid point exists at each frequency.

Interpolation refers to a switch set during the definition of the data element. This switch tells the system whether to interpolate the data element during math and graphing functions. If the switch is Lin or Log, then interpolation will take place.

There are two interpolation check boxes on the *Source tab*: Continuous (**Linear**) and Continuous (**Log**). In the Data elements view window, the Intp column will indicate '**Lin'** or '**Log'**. If neither Continuous box is checked, then the data type will be discrete and it will show '**No'** in the Intp column.

Linear is the default Intp value for a new data item. Log is typically only used for limit lines with log slopes, so that the math will properly interpolate the value between the start and stop frequency of the slope. A typical discrete data element is QP, or any data that would use a marker on a graph.

If you choose to graph a set of peak signals, the peaks will appear as a line if Interpolate is turned on. With the switch off, the graph would show discrete points, which would require a marker to see on the graph.

Discrete data in a table, along with associated interpolated data, will show artifact type interpolated data. Use the **Show All** option on the table to interpolate these points if selected, else it will only show these points at the correct frequencies if it is un-checked.

Element Source

The source text will vary depending upon the data type. A **File** data element (whether regular or word type) will display the absolute path to the file. An **Equation** data element (whether regular or word type) will display the mathematical equation that has been defined for this element. A **Preset** data element displays the word 'Preset'. A **Measurement** data element displays the word 'Measurement'.

Defining a Data Element

To define a new data element, click the + sign on the Data view or use the **Add Data** icon on the toolbar. Enter a unique name on the *Name tab* and then select the data type on the *Source tab*. Do not press the Enter key without giving the element a name and data type, as this will attempt to save the data element without proper definitions and will generate an error.

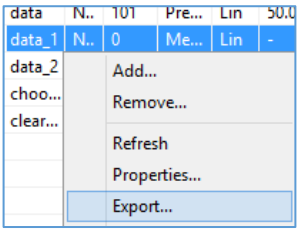
Editing and Deleting Data Elements

To edit an existing data element, bring the Data view to the front focus. Use the cursor and double-click to select the data element.

To delete an existing data element, select the data element and click the Delete key. Deleting permanently removes an existing data element from the profile. When you delete a data element, double check that it does not affect any settings for your actions in the flowchart.

Exporting Data

To export the data from a single data element, go to the data element view and right click on an element then select export. This opens the File Explorer to save the data as a DAT text file. This is useful in tandem with the *Word from File* data type.



Data Types

There are many types of *data elements* in TILE!. Each of these represents a unique capability over competing software. Within the TILE! System, you can take a measurement, add an antenna correction factor, compare it to a specification limit, or create arbitrary criteria all with a few simple clicks of the mouse.

These data elements, when combined in a test profile, can provide a wide range of information on the Equipment Under Test (EUT). This information is saved within the test profile and can be retrieved for future tests. The ability to track each of the intermediate corrections as well as the final number provides a powerful archiving tool for any test laboratory.

Measurement Elements

The Measurement type is the most common and default data element type. Measurement elements are used for any data that will be displayed in a graph as continuous data. If Continuous is not checked then the measurements is not interpolated and is it will be a discrete data element such as a quasi-peak. Typical examples of this type would be a spectrum scan that connects each data point to create a trace.

You can create a correction factor and add them together to form a third data element, but the original information will always be present. This is especially useful in documenting the test process, after the test, or three months later.

You can always look at the raw numbers and trace the corrections and adjustments that have been made to reach the final output numbers. ISO 9000 and ISO 25 both have references to proving the numbers are real. TILE! was designed to solve this problem. The data structure is designed to prove the actual readings are the source of the numbers on the final report.

File Elements

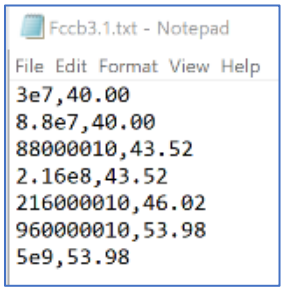
Some types of information used in an EMC test are not a part of the actual measurement, but are used for the standard or the instrumentation. Correction factors, specification limits, and gain tables are all samples of data used in an EMC test, but are normally maintained separately. File elements allow you to import this information into the TILE! system and use them in a test profile.

Importing this data into a data element allows us to use this information for graphing purposes, mathematical comparisons, and peak searches. Correction tables include antenna correction factor, amplifier gain tables, cable loss tables, etc. When used with the other data elements in the TILE! system, these elements add a high degree of flexibility to the ability to configure and control testing.

The default file format is an ASCII text file. The general format for all data elements is a two-column table of information. The table should have delimited variables with column one as the frequency and column two as the value. You must use commas (,) or semi-colons (;) as the delimiters. All white spaces are ignored. The size of the file is only restricted by the system parameters. You can create these tables quite easily in a text editor or in most spreadsheets.

As illustrated to the right, the frequency (the first number) can be either a decimal number or an engineering number. The value (the second number) has no specific unit of measure. The unit of measure is depends on how it is used in the test profile.

In the example, the table represents the FCC Class B specification limit at 3-meter separation. To create a stair step in the levels, there is a transition frequency point at 88 MHz, 216 MHz, and 960 MHz. When creating a stair step effect, there should be at least a 1 Hz difference in the frequencies.



Equation Elements

When performing EMC tests, mathematically manipulating data is an everyday occurrence. The use of an equation as a data element allows you to create a new data element by mathematical reference to two or more other elements. Whether this is simply adding an antenna factor to a raw reading, or performing obtuse mathematics to determine waveform performance, the equation writer has the flexibility to perform complex mathematics.

The equation writer is the tool for defining an equation element; for further details, see the section *Equation Tab*. In the equation writer, you define values using a wide range of mathematical functions. Some are as simple as adding or subtracting elements, but the complexity is only limited by the imagination.

Preset Elements

The Preset data element is a value generator used to create simple data sets. Once created, Preset data elements are treated just like any other data element in the system.

For example, you may have a 26-dB preamplifier that you need for only a simple, non-certified scan. Instead of going through the elaborate calibration routine of creating a data file for the calibration and loading it through the file elements, preset allows you to create a set of values across the frequency range.

Word Elements

There are certain places within the TILE! system where it is necessary to track an alphanumeric comment keyed to a specific frequency. For instance, in Immunity testing if you enter manual mode, you have an option at each frequency to enter a comment about the step in the test.

Whether this comment is a performance characteristic or a comment about the power state of the EUT is strictly up to the user. From the TILE! standpoint, it is an alphanumeric comment.

The word-type data element is a two-column format, like all other TILE! data. The first column is frequency and the second column is the alphanumeric text. This text is limited to 256 characters in length.

Word from Equation Elements

You can perform two math functions using word data elements.

The min and max functions compare the data starting at the first character, and determine the greater or lesser based upon the ASCII text number for the character. As soon as a digit is lesser than or greater than the one being compared, the equation conditions are met and the result is written. These functions are primarily used in automotive testing where you have a single alpha code for performance.

There are also equation functions to produce the words Pass or Fail by comparing a value limit. These functions are *pfless* and *pfgreat*.

Word from File Elements

It is possible to import a word data element from a file (using the same CSV format used in regular data elements). This allows you to move word type data from one profile to another easily.

Script Elements

The TILE! Scripting feature allows users to create scripts to enhance the usability of the program. Scripting does require some knowledge of programing conventions and formatting, but it is very useful in bridging the installed user interfaces/functions to custom features, such as Menus and dynamic data handling.

TILE! scripting is provided through an open source parser called muParser. MuParser already contains many common mathematical functions such as "+-/*, sin, cos, etc.", and most importantly we can extend the parser's functions as needed. For example, this parser has been extended to access and perform array operations on TILE! objects and allow scripts to call other scripts and pass back arguments.

Menus can be created to allow user interaction into settings of actions or numeric values, such as limits/targets/measurement parameters. Loops or dynamic data creation can be used to create new data elements or graphs/tables as data is collected.

There is really no limit to TILE! scripting. Scripting and linear math can be used to enhance the many TILE! math functions already available. See the *script help document* for more information on how to work with scripts.

Standard Elements

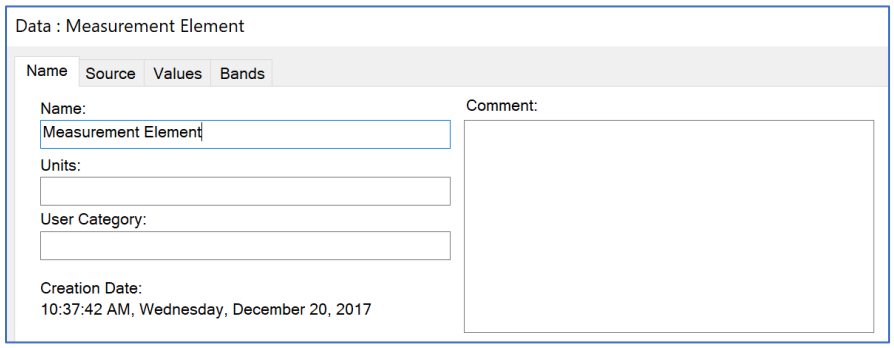
A Standard is a file that contains references to data and multiple other files that are used in a particular profile. The file .stdz extension is a container for the associated files.

The types of files that the standard can contain are varied; they can be data files, sequencer files, tables, scripts, etc. These files can be loaded from the "Standard file" and into a profile using TILE! scripting. How the data files are loaded is left to the developer of the standard's profile.

Data can be extracted from the standard, or the data from the standard file can be inserted into other items. The standards data item is similar to a collection, and therefore collections and standards work interchangeably.

Data Element Tabs

Name Tab



See *Element Names* for details on how to name a data element without later conflicts.

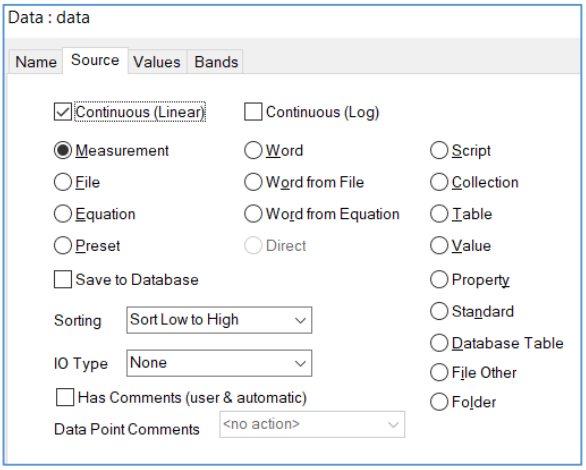
The name is a unique data identification for this data element. There are no specific requirements for data element names except that they cannot contain math symbols; this includes the dash (-), which is seen by the math parser as a minus sign.

The comment line is free text field allows a description of the data element.

Units will be shown if the data is used as a ‘Monitor’ in the Immunity tests. For example, if an O’scope is monitoring Vrms, Vpp or even uS for a rise time, the unit will show next to the value in the runtime user interface. This is a free field and not extracted from the instrument.

The Creation date will reflect the date/time that the data was last changed or updated. If the date/time stamp changes on the disk it will not reflect here unless the data element is specifically updated. This is accomplished by using the Transfer Data icon on the flowchart as part of a test, or by opening the data element on this page and clicking OK.

Source Tab



The **Source** tab defines the data elements. There are four choices that need to be made: determining the interpolation methodology, selecting the data type and the sorting method, and whether to do a database save for this data element if the TILE/DB product is installed.

The default setting for a new data element is a Measurement type. When you select a data type other than measurement, additional tabs are available. See the *Data Types* section above for more information on the types of data available for this tab.

- **Continuous (Linear and Log)** — By default, the Continuous (Linear) check box is selected. This box will direct the system to interpolate data points when drawing graphs or performing mathematics in a linear fashion.

  If the Continuous (Log) check box is selected, then all interpolation will be in log format. Continuous Log is typically only used for limit lines with log slopes so that associated math such as QP margins can be properly interpolated along the line.

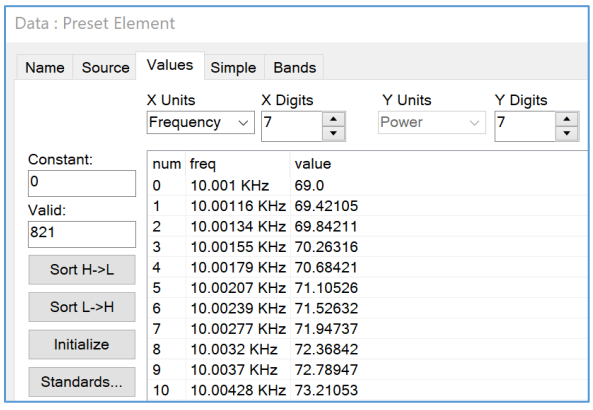
  If you are creating data elements that by nature should be discrete (peaks, over limit points, and so on), then deselect both boxes to turn off the Continuous option.

- **Save to Database** — This check box only has meaning if the TILE/DB product is installed. If this check box is selected and TILE/DB is present, then the software will transfer the contents of this data element to the database whenever a save is performed.

- **Sorting** — This dropdown determines whether data is automatically sorted when it is loaded for the first time. This only has meaning for the file data elements when read from the file for the first time. The default setting is 'Sort Low to High'. You can change the sorting routine from the *Values tab* with a slight difference. With the auto sort set to off, the data from the file will be read in the sequence it is sorted inside the file.

- **I/O Type** — This an advanced feature used with the *Scripting Functions*.

Values Tab



- **Initialize** — The data from the file is read in one of two ways. The first way is when you accept the settings (click OK) and the data file is automatically read into the data element.

  The second method requires using the **Values** tab and clicking **Initialize**. On the *Data elements view window*, this element will now show the number of valid data elements. This gives you the ability to look at the contents of a data file prior to accepting it.

There are two boxes on this tab, Units and Valid. The **Units** field is not active under the current version of TILE!. The **Valid** field displays the total number of points in this data element.

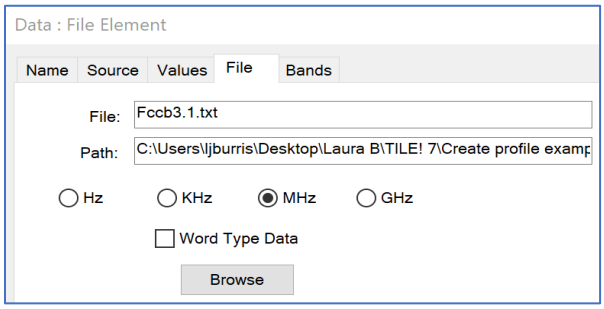
- **Sort H->L/ Sort L->H** — The **Sort L >H** tab sorts the data list starting at the lowest frequency, and sequencing to the highest frequency while **Sort H >L** does the opposite, sorting high to low. Normally, data within TILE! is sorted from Low frequency to High frequency.

  NOTE: Data that is not sorted will cause strange results during math operations, since the interpolation functions look to the frequency before and after for a valid range.

- **Valid** — The valid indicator shows the number of valid frequency points. This is also demonstrated by scrolling down the list with the right scroll bar to look at the list of values.

File Tab

NOTE: This tab only shows up if **File** is selected as the data type under the *Source tab*.



- **File Name/Directory Path** — Use the **Browse** button and find the data file. Once selected, the file and path will be filled in automatically.

  To make creating files as easy as possible, we use comma separated variable (CSV) files or text files (TXT) for loading or saving data. This format is limited to a two-column format. Within a spreadsheet program, make sure that you set the file type to CSV text when doing a save or save as.

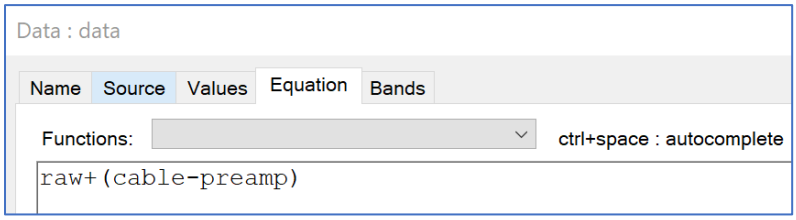
  NOTE: If the file is in the wrong format, the data will not be read.

- **Word Type Data** — There are times when you might need to load word type data into a TILE! profile. Since the CSV file type includes no definitional information, we need to warn the program when we are importing text data, as opposed to numbers. The **Word Type Data check box** serves the function to confirm it is imported as text.

  Do not to use this check box on data that is numerical or you will end up with text that cannot be used in normal mathematics.

Equation Tab

NOTE: This tab only shows up if Equation is selected as the data type under the *Source tab*.



The **Equation** tab is an equation writer that you can enter a formula to define a data element. You can enter a more complex equation, but you are limited to nine imbedded parentheses. Each parenthesis must delineate a binary equation.

The example to the right demonstrates taking a reading, adding the cable losses, and subtracting preamp gain. To keep a binary operand, the cable loss and preamp gain are enclosed in parentheses. The raw data element is then added to this element.

The equation may consist of operations, conditions, and functions. The operators and conditionals allow either matrix or constant operands. All math functions are in lower case and the math parser is **case sensitive**. You must enter functions in **lower case**. Also, avoid naming a data element the same as a math function.

The results of any math operation are stored into the data element. Until the equation has been executed, no values are present. After execution, you can use the *Values tab* to view the data element values.

NOTE: It is important to keep track of the interpolation settings for any data element. If data element (x) is continuous, then all of data element (y) that is bounded by the frequency values of data element (x) will be evaluated. The same is true for data element (y) if it is continuous.

If both data elements are continuous, all values bounded in one data element by the other data elements will be evaluated. If both elements are discrete, only those values with identical frequencies will be evaluated. The above statements are true for conditionals and two parameter functions.

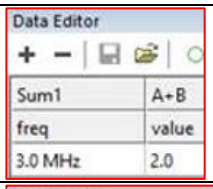
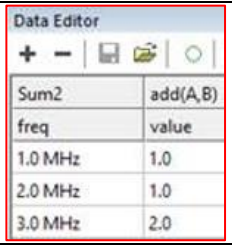
- **Functions** — Displays the available mathematical functions. These are displayed for reference purposes, but when selected the appropriate function will be entered in the **Equation** box for completion. See the *help files* for a complete list of math functions.

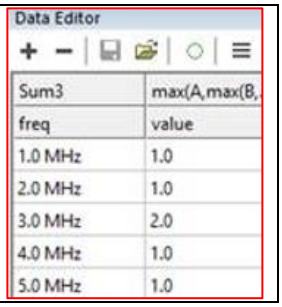
Addition "+" operator

The "+" operator only adds frequencies that appear in both inputs. If either frequency is missing from the opposite input, it will not appear in the output. There is an alternate "add" function which will include frequencies in A if they are not present in B, but it doesn't continue processing B once A is finished (i.e. the number of elements should be the same, but if they aren't then you'll miss frequencies in B).

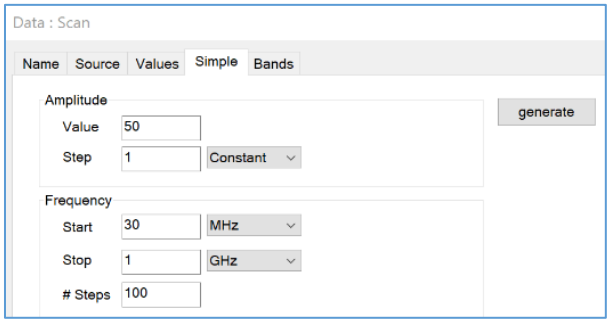
There are interpolation issues for "+" if both inputs have the interpolation (Continuous flag) Off. If either input has interpolation On, it will include an interpolated value as if that point were present (i.e. the "missing" frequency at that point is included in the output, with the interpolated value added to the value of the other input's frequency).

The following example shows how to add frequencies together if they are present in both inputs, else if one or the other is missing then it will copy that frequency to the output as is: max(A, max(B, (A+B))).

Name	DB	Valid	Type	Intp	Source	Values	Output
A	No	3	Preset	Lin	1.000000	1.0, 1.0, 1.0	
B	No	3	Preset	Lin	1.000000	1.0, 1.0, 1.0	
Sum1	No	1	Equation	Lin	A+B	2.0	
Sum2	No	3	Equation	Lin	add(A,B)	1.0, 1.0, 2.0	

Sum3	No	5	Equation	Lin	Max(A, max(B, (A+B)))	1.0, 1.0, 2.0, 1.0, 1.0	

Simple Tab



NOTE: This tab only shows up if **Preset** is selected as the data type under the *Source tab*.

The preset values are defined by two conditions, frequency and amplitude. These are combined in the same fashion as the standard file format.

For straight lines, you do not need the default 100 steps, so this can be changed to 1 step.

- **Amplitude** — Enter the value that is appropriate for the data element. For example, to create a 26-dB preamplifier gain table you would enter 26 (the units are assumed).
    - **Constant** — This the default setting for preset values. When this is selected, the value in the text table is filled into a table across the frequency range defined under **Frequency**.
    - **Add** — Takes the starting value in the **Value** box and sequentially adds the value shown in Step. The number of steps is set under **Frequency**.
    - **Multiply** — Takes the starting value from the **Value** box and multiplies the value by the Step at each frequency.
- **Generate** — Creates the data that in the *Values tab*.

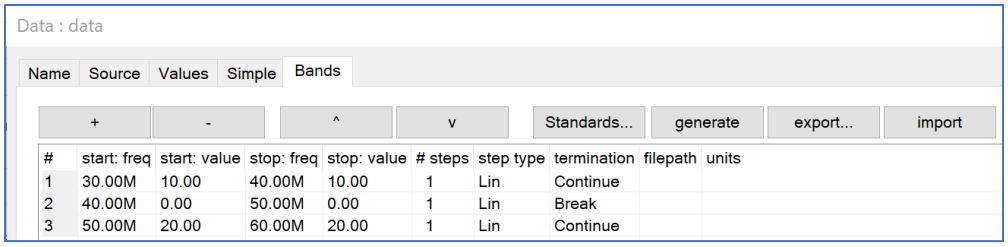
Bands Tab

Bands generate complex data to include in linear and log sections. This is handy to create limit lines. This can also be used to create mask type data used in the graph to hide breaks in data. This tab is generally used in conjunction with *Preset* data.

Click the + button to add a new data band. Each row of the table represents a band, with a Start Freq and Stop Freq value. You can also export/import bands to/from a file with the .bands extension (do an Export and you will see the data format; you can edit the file to add/modify bands, and then Import it back).

Once the band(s) are set as desired, click the Generate button and the bands are copied as frequency pairs into the data values. These values could be generated manually using the Data Editor view, but the Bands UI makes this much easier, and provides a future reference for the band exclusions.

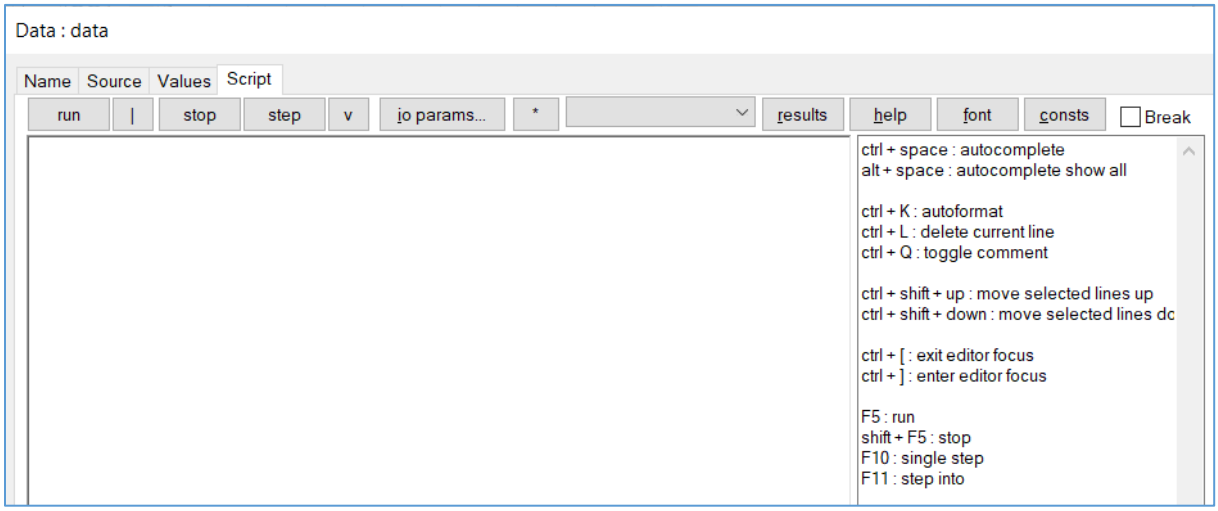
The example below shows how to create a limit line using bands.



Script Tab

NOTE: This tab only shows up if **Script** is selected as the data type under the *Source tab*. See the *help files* for more details about scripting.

NOTE: Use "//" to place comments in your script. The "//" can start a new line or be placed at the end of a line. In either case, any text after "//" to the end of the line will not execute.

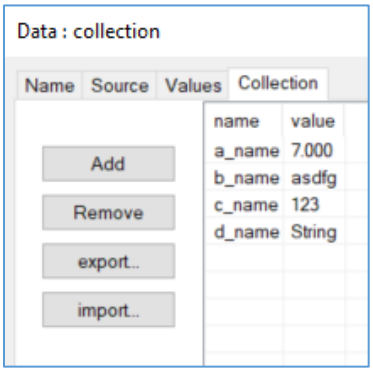


- **Run** — Execute the entire script.
- **Stop** — Force stop the execution of the script.
- **Step** — Execute one line at a time.
    - **NOTE**: Parser variables will always update after every step and in alphabetical order.
- **IO Params** — Generate the IO parameters.
- **'Asterisk'** — Opens an external editor. The first time you use the capability you can set which editor to use, such as Visual Studio or Notepad++.
- **'Blank dropdown'** — Inserts a function's syntax format.
- **Results** — Shows the results of the parser variables when the script executes.
- **Help** — Shows a list of functions in alphabetical order.
- **Font** — Changes the font of the typing window.
- **Consts** — Shows the values for the constant variables Euler's number (*e*) and Pi (*π*).

- _e = 2.71828
- _pi = 3.14159

- **Break** — Sets a break point when the script executes.

Collection Tab

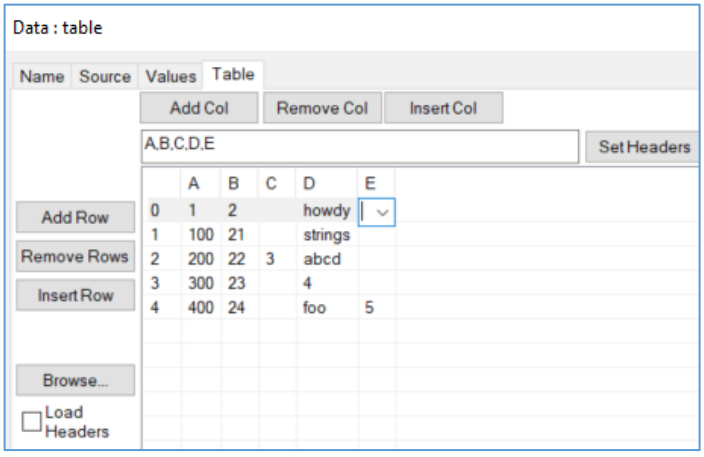


NOTE: This tab only shows up if **Collection** is selected as the data type under the *Source tab*.

Collections are an enhanced data type. A collection is a group of related data. You can give each data item in the collection a name. The value given to a collection item can be a number or a string.

You can access the values from a collection just like any other property from a script. See the *script help document* for more information on how to access collections with scripts.

Table Tab

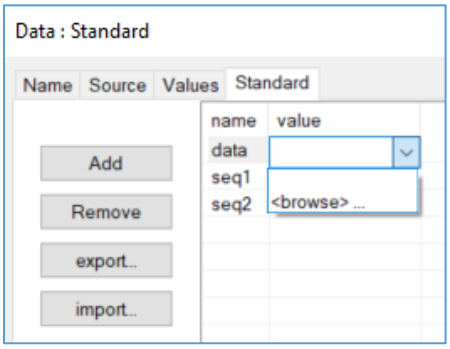


NOTE: This tab only shows up if **Table** is selected as the data type under the *Source tab*.

Tables are a special data type with rows and columns of data. The table can contain numbers or strings.

You can access the values from a table just like any other property from a script. See the *script help document* for more information on how to access tables with scripts.

Standard Tab



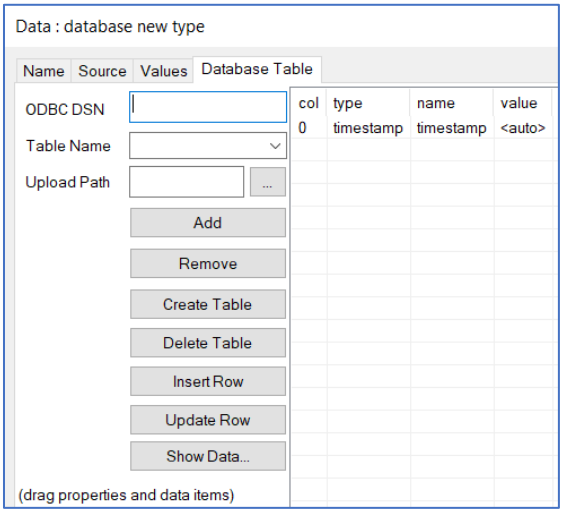
NOTE: This tab only shows up if **Table** is selected as the data type under the *Source tab*.

A standard looks just like a "collection" of "name/value" pairs. The standard developer can add any number of named items to the standard. The named items can be files or data strings. To add a file, select "<browse>" from the value column dropdown editor.

By using the export button on the "Standard" tab, a file with the extension "stdz" is constructed and saved. This a compressed/zipped file containing all of the referenced files. It can be expanded or compressed again using normal ZIP/UNZIP tools.

Database Table Tab

NOTE: This tab only shows up if **Database Table** is selected as the data type under the *Source tab*.



You can hook a Database Table via ODBC to MySQL to a database and use it to store settings, property values and data values. It also would be possible to store whole actions (xml) and reload them. You can then reload the settings and data by double clicking on the desired row like a time machine.

If you look at the example below, the left-hand sided defines your table types/columns and items (data, properties). The right-hand side displays the current rows in the table.

Database Table functions include:

- database_create_table(dataItem) // creates a table based on the data table definition
- database_drop_table(dataItem) // deletes a table
- database_insert_row(dataItem) // inserts a new row sets cursor id to new row
- database_update_row(dataItem, cursor_id) // updates the data at the row/cursor id
- database_load_row(dataItem, cursor_id) // loads the data at the row/cursor id
- cursor_id=database_get_cursor_id(dataItem) // gets the current cursor id

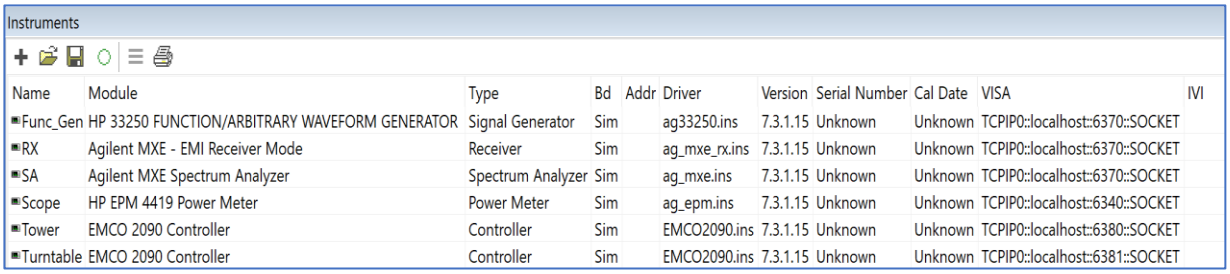
Instruments

One of the biggest advantages with TILE! software is hardware independence. This is achieved by using small, modular programs called instrument drivers. These are recognizable in TILE! by the three-letter extension, .**INS**. The instrument driver contains the unique programming code to allow the software to talk to the instrument. These are dynamically linked libraries (.**DLL**), but we use the .**INS** nomenclature to make it more readable when creating lists of instruments.

Instrument drivers are used in the system to provide communications between the main program, the test profile, and the instrument on the GPIB (or serial) port. An instrument in a test profile needs to be identified by a unique name and then linked to an appropriate instrument driver. The use of a separate name link allows the user to substitute different instruments into a test without having to modify any actions in the test profile.

NOTE: It is recommended that you go to the tools menu and **Build Instrument Database** to refresh the instrument drivers for the profile.

Instrument View Window


Instruments

Name	Module	Type	Bd	Addr	Driver	Version	Serial Number	Cal Date	VISA	IVI
Func_Gen	HP 33250 FUNCTION/ARBITRARY WAVEFORM GENERATOR	Signal Generator	Sim		ag33250.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6370::SOCKET	
RX	Agilent MXE - EMI Receiver Mode	Receiver	Sim		ag_mxe_rx.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6370::SOCKET	
SA	Agilent MXE Spectrum Analyzer	Spectrum Analyzer	Sim		ag_mxe.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6370::SOCKET	
Scope	HP EPM 4419 Power Meter	Power Meter	Sim		ag_epm.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6340::SOCKET	
Tower	EMCO 2090 Controller	Controller	Sim		EMCO2090.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6380::SOCKET	
Turntable	EMCO 2090 Controller	Controller	Sim		EMCO2090.ins	7.3.1.15	Unknown	Unknown	TCPIP0::localhost::6381::SOCKET	

The Instrument view window provides visual access to the currently defined instruments. The displayed columns are the instrument name, module, type, board, address, driver, version number, serial number, and calibration date. The serial number and calibration date default to Unknown unless the user has specifically entered this information.

The name given to an instrument should be a unique reference to a type of instrument. Calling the spectrum analyzer **SA** instead of **HP8568B** gives much greater flexibility in designing test profiles. If you name an instrument with the hardware name, the implication will be that this is the only instrument that is acceptable for this setup. This is not true in the TILE! system. Wherever the name is used in the test profile, you expect to see a receiver or analyzer, not necessarily a specific receiver or analyzer. This is sometimes a subtle difference, but it is very important in properly designing a test profile.

Defining an Instrument

To define a new instrument, click the + sign on the *Instrument view* or use the **Add Instr** icon on the toolbar. Enter a unique name on the Name tab and then select the driver on the *Driver tab*.

Editing and Deleting Instruments

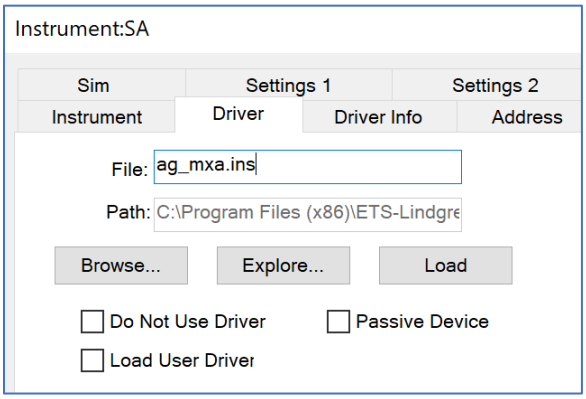
To edit an existing instrument, bring the Instrument view window to the front focus. Use the cursor and double-click to select the instrument.

To delete an existing instrument, select the instrument and click the Delete key. Deleting permanently removes an existing instrument from the profile. Ensure when you delete an instrument that it does not affect any settings for your actions in the flowchart.

Instrument Tabs

Driver Tab

The Driver tab selects the instrument driver used for communicating with the instrument.



Some manufacturers maintain a stable instrument command set cross their product line and in these instances, there may be a generic driver for that manufacturer.

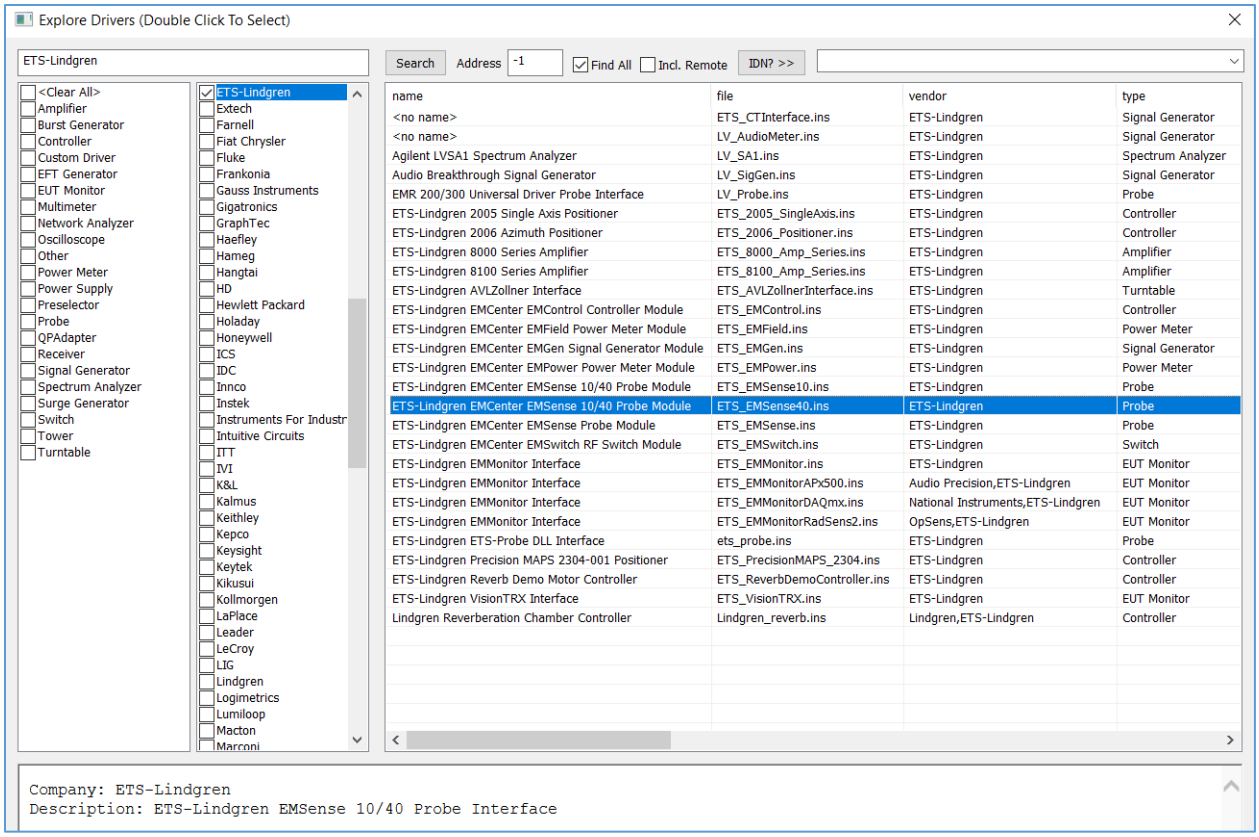
NOTE: Simulation mode does NOT support all drivers. If you just want a general idea of how the instrument will work, then use one of the common default drivers.

NOTE: MXA and MXE drivers are not interchangeable. MXE has its own drivers (and includes the PXE). Everything else (including CXA/EXA/PXA/UXA) uses the MXA drivers.

- **File and Path** — Use the **Explore** or **Browse** to find the instrument driver files. Generally, instrument drivers exist for each manufacturer model number. This method configures the instrument settings automatically based on detected device info (including VISA resource, GPIB ID, GPIB address).
    - **Browse** — This button shows all the instrument files alphabetically. Use this if you already know the name of the driver you wish to select.
    - **Explore** — Find the driver for the model instrument you are defining. Double click to select the driver, and the file path on the driver tab will be filled automatically.

      There is a search box to look up key words, or you can narrow the search results by checking the boxes on the search columns. The first column sorts the driver list by instrument type, and the second column sorts the instrument drivers by vendor.

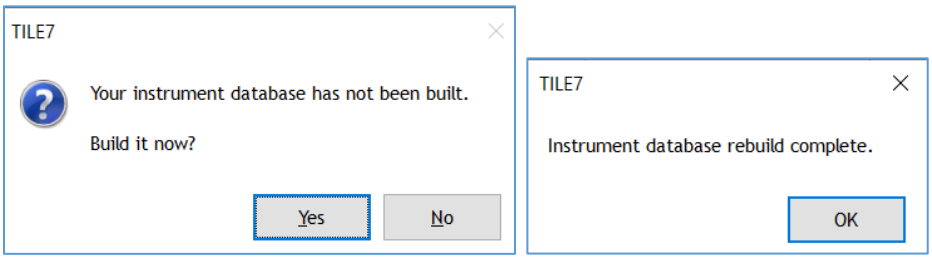
      Use the *IDN? scan (address = -1) to search for GPIB and VISA resources. This means all resources configured in NI-MAX can be discovered (not just GPIB devices).



- **Do Not Use Driver** — Used as a placeholder for an instrument that does not have an instrument driver. These types of instruments can be controlled with the GPIB Control action as described in Instrument Commands (*VISA*) Action. The convention in the GPIB Control requires an instrument name. Using the Do Not Use Driver check box allows you to assign a name and board address to an instrument when a specific driver is not available.
    - **Desc** — When using the **Do Not Use Driver** option, you can enter a description of the item. This will be displayed on the instrument window under **Type**.

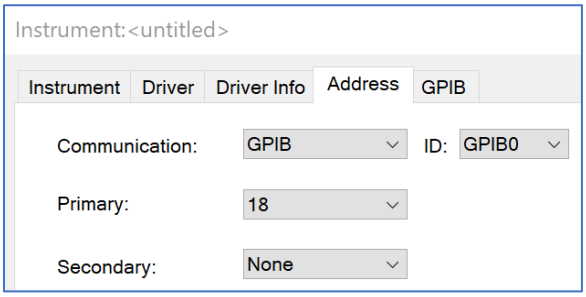
Build Instrument Database

When first using the Explore Button while creating/editing instrument properties, you may this warning; click yes to build the instrument drivers database list:



NOTE: You can also update the instrument drivers by going to the tool menu and selecting *Build Instrument Database*.

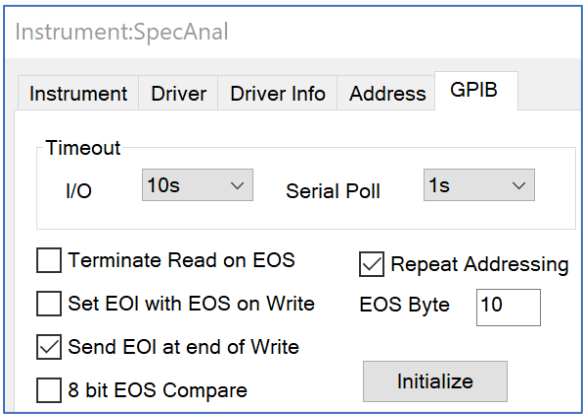
Address Tab



Each instrument must have a unique bus address. If there are two instruments on the same bus, you will get a failure. The **default** settings for the Address tab are **GPIB Board 0** and **Address 18**. You must change these settings to ensure the instrument does not conflict with other instrument settings.

- **Communication** —The TILE! system supports multiple GPIB boards, serial, simulation, and custom PC card interfaces. The GPIB standard supports up to 96 addresses. Typically, a GPIB card can support 32 different devices, so multiple board configurations are unusual.
- **Primary** — The specific hardware address of the instrument relative to the board selected. The primary address is a number in the range of 0 to 30 decimal. Defining addresses above 30 is possible with the secondary address but is recommended only for the most sophisticated users.
- **Secondary** — The secondary address is sometimes used on older test instruments, and is a number in the range of 96 to 126 decimal. The secondary address is a GPIB standard convention, which is rarely used in the TILE! system.

GPIB Tab



NOTE: The GPIB tab only shows up if **GPIB** is the chosen communication on the *Address tab*.

This tabs settings control the communications between the computer and the instrument. If the instrument is an older model, you may need to adjust some of these settings to match the instrument. See the hardware manual for more information on these settings or contact *ETS-Lindgren*.

- **Timeout** — The two timeout settings, **I/O** and **Serial Poll**, refer to how the system responds to timing differences between different instruments and their communications on the bus.
    - **I/O** — The I/O timing determines how long the GPIB bus will wait for a response from an instrument before reporting an error. This default setting is set to 10 seconds, which covers most requirements.

      When you are running long sweep times in emission scans, depending upon the analyzer, the bus may timeout before the instrument is through the measurement if the sweep time is longer than the I/O setting. If you need to run 20 second sweeps, the I/O setting may need to be set at more than 20 seconds.

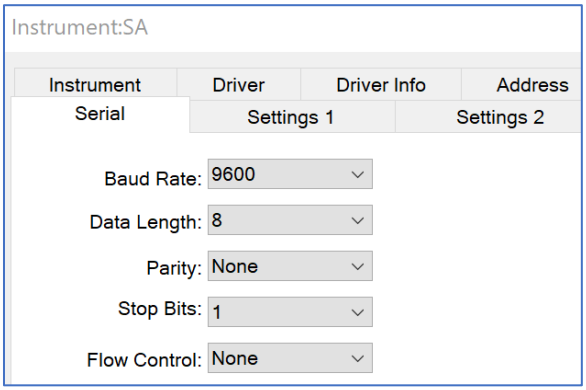
      I/O incompatibility happens when the scan stops in the middle and traces the floor thereafter. In this case, usually the GPIB has aborted and reported an error. Change the I/O time to agree, or exceed, with the instrument sweep time.

- **Serial Poll** — The Serial Poll setting refers to a GPIB standard for SRQ errors. The Serial Poll Timeout list is used to set the wait time for poll responses from the instrument. The default wait time is 1 second. This setting is almost never changed. Please see the GPIB card manual for information on this setting. If you have problems with serial polls, try using a longer timeout value.

- **Terminate Read on EOS (End of String)** — This check box allows the software to terminate a read operation when it receives the EOS byte. The default setting for this option is un-checked.
- **Set EOI (End of Instruction) with EOS on Write** — This check box allows the software to assert the GPIB EOI line whenever it sends the EOS byte. The default setting for this option is un-checked.
- **Send EOI at end of Write** — This check box allows the software to assert the EOI line at the end of each data transfer. The default setting for this option is checked.
- **8-bit EOS Compare** — This check box allows the software to use all eight bits of the EOS byte when checking a match. Only seven bits are used if this option is not selected. The default setting for this option is un-checked.
- **Repeat Addressing** — This check box allows the software to address the instrument before every read or write operation. Some older instruments require this to be set. The default setting for this option is un-checked.
- **EOS Byte** — This box has a range of 0 to 255 and is used by the software for all EOS operations. Make sure this is checked, as there are no known times when this should not be selected. The default is set to 10.

Serial Tab

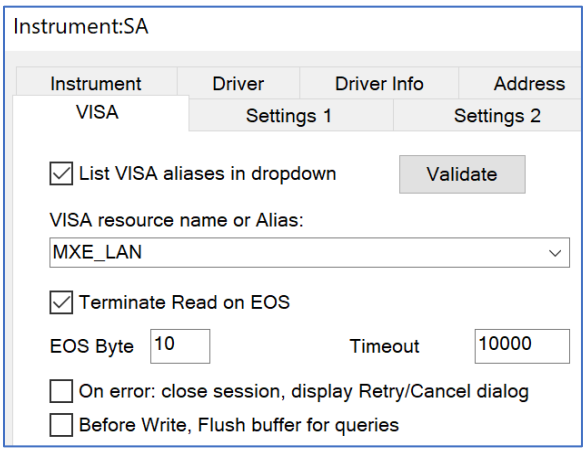
NOTE: The Serial tab only shows up if **Serial** is the chosen communication on the *Address tab*.



These boxes control the five general conditions of the serial bus: baud rate, data length, parity, stop bits, and flow control. See the instrument manual to determine the correct settings.

- **Baud Rate** — The default is 9600. Another common setting is 115,200. There are nine options.
- **Data Length** — The default is 8.The options are 5, 6, 7, and 8.
- **Parity** — The default is None. The options are None, Odd, Even, Space, and Mark.
- **Stop Bits** — The Stop Bits can be set to one or two.
- **Flow Control** — There are three kinds of flow control: default is no flow control (**None**), hardware control (**DTR**), or software flow control (**X On/X Off**). The choice of flow control depends on the equipment supplier.

VISA Tab



NOTE: The VISA tab only shows up if **VISA** is the chosen communication on the *Address tab*.

VISA allows communication to the instruments that are connected to the computer via Ethernet, USB, or TCP/IP.

- **List VISA Aliases in Dropdown** — If this check box is selected, it will show the alias that was assigned to the instrument in the NI MAX interface.

  NOTE: See the section *Create new NI MAX resource* for an example in MAX.

- **VISA Resource Name or Alias** — This specifies the instrument by a VISA resource string or there is a dropdown list of all instruments connected by NI MAX.
- **Validate** — Once the instrument is selected, pushing this button will validate the connection to the device and pop up a Validation Successful window.
- **Terminate Read on EOS (End of String)** — This check box allows the software to terminate a read operation when it receives the EOS byte. The default setting for this option is un-checked.
- **EOS Byte** — This box has a range of 0 to 255 and is used by the software for all EOS operations. The default is set to 10.

VISA Resource

The easy way to set up TILE! is to use NI Max to scan for the instruments and then take the exact VISA resource string from each instrument and paste it into the proper place in TILE!.

1. You should make a flowchart of your instruments and computer setup so there are no conflicting communication addresses. Each IP address must be unique and static.
   a. You should setup all of the Ethernet addresses to be identical for the first three “Octets”, with only a different last one. For example:
      i. 192.168.1.100 = Computer’s USB to Ethernet adaptor
      ii. 192.168.1.200 = EMCenter
   b. You should then set the subnet address to be 255.255.255.0 on all of these devices so they can see each other.
2. NI Max will allow you to scan for the instruments and help you get the VISA resource strings. Unfortunately, some instruments do not respond to NI Max and have to be setup manually based on their Ethernet address.

Type	Format and Example	Description
GPIB	*GPIB[board]::primary address[::secondary address][::INSTR]* *GPIB0::1::0::INSTR*	*EX: GPIB device at primary address 1 and secondary address 0 in GPIB interface 0.*
TCPIP INSTR	*TCPIP[board]::host address[::LAN device name][::INSTR]* *TCPIP0::192.168.9.253::inst0::INSTR*	TCP/IP device using VXI-11 that supports most modern instruments. *EX: TCP/IP device located at IP address 192.168.9.253. This uses the default LAN Device Name of inst0.*
TCPIP SOCKET	*TCPIP[board]::host address::port number::SOCKET* *TCPIP0::1.2.3.4::999::SOCKET*	A raw network socket connection over TCP/IP. Used without TCP/IP Instrument Protocol. *EX: Raw TCP/IP access to port 999 at the specified IP address.*

Using NI Max to configure instruments

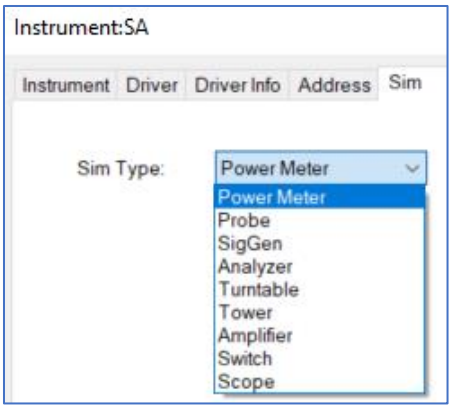
TILE! can access network-enabled VISA devices without first configuring them in NI MAX. It is only necessary to know the correct VISA resource name that will communicate with the device, and enter that into the TILE! VISA resource name input on the instrument's Address tab.

TILE! uses NI Max to simplify the connection of instruments using USB, Ethernet, and GPIB. There are numerous benefits of using a VISA configuration utility such as MAX:

1. Instruments are configured and verified independently of TILE!.
2. Auto-detect feature of MAX may be able to detect and configure available instruments automatically.
    a. Unlike with GPIB/Serial instruments, TILE! will attempt to open VISA sessions and communicate to all VISA-configured instruments in a profile immediately upon loading the profile.
    b. If one of these VISA instruments are not accessible when the profile is opened, then TILE! will generate warning prompts, however this does not prevent the profile from opening.
    c. If the instruments become active after the profile is opened, they will then be accessible by the profile at that time.
3. VISA resource names are produced in MAX using only network-specific information such as IP address and socket number. The specific VISA syntax is generated by the utility.
4. VISA resource names and aliases created in MAX become automatically available in the TILE! resource dropdown list.
    a. Devices that have not been "pre-configured" in MAX will not appear in the dropdown list.
    b. Note that the VISA resource system may take a few seconds to respond before the list is populated.

Sim Tab

NOTE: The Sim tab only shows up if **Simulation** is the chosen communication on the *Address tab*.



The Simulator communicates to simulated instruments to give the tester an idea of how the test would run with real instruments. Simulation in TILE! was never intended to support all possible drivers. That would be nearly impossible to accomplish.

For more information on how Simulation works, see the section Simulation Utility in the appendix.

Example description

For example, the SA Sim is not a spectrum analyzer. The Sim SA would best be describes as a time domain signal generator with a real-time FFT. It does not understand/implement analyzer concepts such as RBW, sweep time, etc. The SA itself is really a “pass-through” of the VSG output (when the SA changes settings, it is telling its connected VSG to generate a different trace, so if you have the VSG open during a test run you can see this effect).

To get a better feel for what type of traces you will get, click Show VSG and play around with the UI a bit (change Signal Type, Signal Processing settings, Frequency/Amplitude). You will see the types of spikes you are observing with multiple ranges when you set the Signal Type to Square and move the Frequency slider. You might have better results using the Triangle signal (it still gets the harmonics, but fewer spikes in between). The Sim runs fine with multiple ranges; it is just like running multiple M_Ranges split up into smaller frequency bins.

Quick enable/disable simulation

There is a feature in recent versions of TILE! that allows you to easily enable/disable simulation for any Instrument, allowing you to toggle back and forth and guarantee that the simulated driver will work. TILE! automatically remembers the original driver and will restore it for you so you do not have to reconfigure it (including GPIB address, etc.). This non-sim information is even saved with the profile, so when you close a profile that has simulation enabled and then reload it, if you disable simulation again the original driver and settings are brought back.

Here is an example that will allow you to simulate ANY instrument without concern whether the native driver is supported by simulation:

1. Start with a Sunol configured Instrument using GPIB address 10.
2. Right-click the instrument in the Instrument View and select Enable Simulation (Default Sim Driver). This will turn on a simulator for the instrument and change the driver to emco2090.ins.
3. Save and close the profile, then reload the profile.
4. Right-click the instrument and select Disable Simulation. The driver is restored to Sunol.ins and the settings are returned to GPIB address 10.

There is also an option by right-clicking on the instrument for Enable Simulation (Retain Driver) that will do the same thing as the Default Sim Driver option, just not changing the driver.

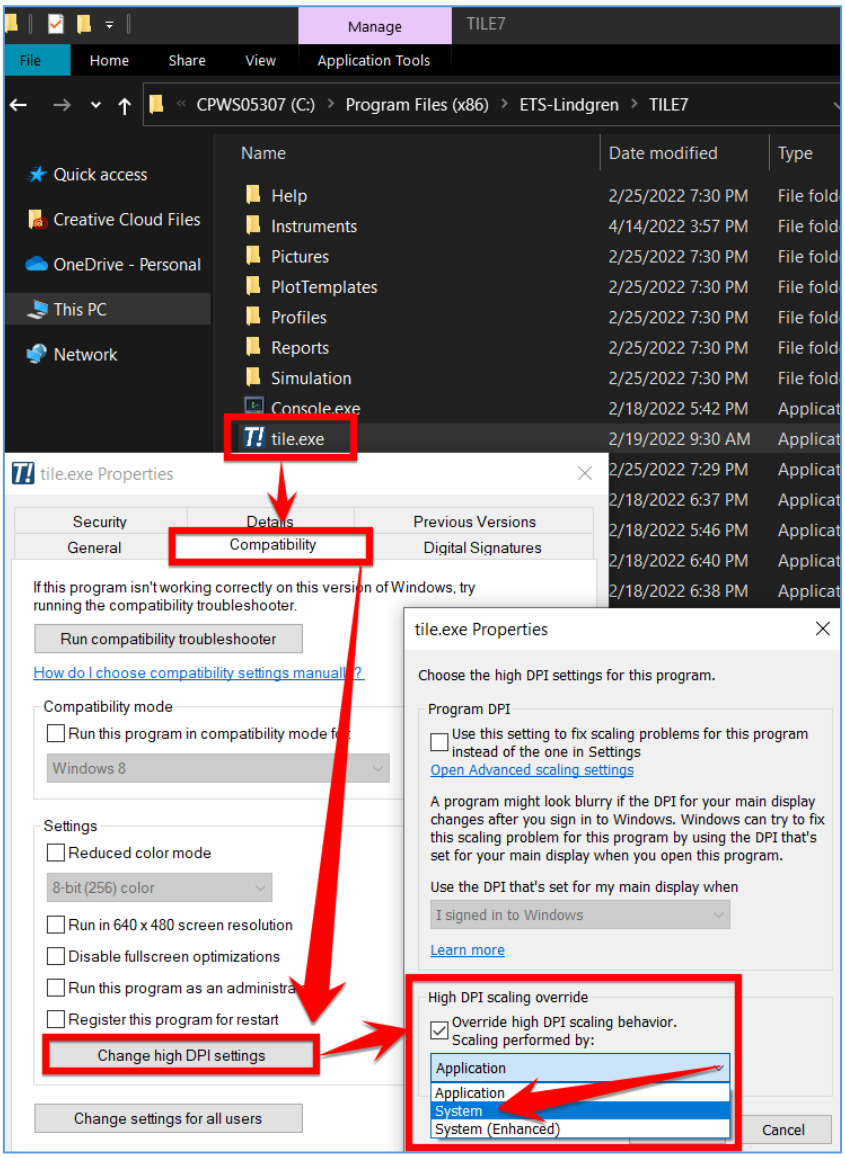
Finally, there are two selections in the TILE! Tools menu, Enable Simulation (All Instruments) and Disable Simulation (All Instruments), which will perform the enable/disable steps on every instrument in the

profile, so you can have a one-click method of enabling/disabling simulation for the entire profile. This way the Default Sim Driver option is automatically selected for every instrument to guarantee that simulation will work correctly.

Simulation for Hi-resolution screen

Here are instructions to fix the garbled fonts on the simulator when using it on hi-res screen.

1. First, exit TILE! if it is currently running.
2. Open a Windows Explorer folder and find the file location for **tile.exe**. The default path is: "*C:\Program Files (x86)\ETS-Lindgren\TILE7*".
3. Right-click **tile.exe** and select **Properties**.
4. Click the **Compatibility** tab.
5. Press the **Change high DPI settings** button.
6. In the **High DPI scaling override** section, check the **Override high DPI scaling behavior** setting.
7. Change the dropdown selection to **System**.
8. Press OK twice to save the settings.



Graphs

Graphs and tables are the primary output of the TILE! program. One of the aims in designing TILE! was to concentrate on instrumentation control. For this reason, the graphs and tables are considered good, solid presentation devices. For very sophisticated reporting, we designed TILE! to interface easily with MS Word and other graphing programs.

Graphs are created to display data in a 2D format, typically showing frequency vs amplitude (X/Y). Either axis can be displayed with linear or logarithmic scales. Lines can be one of 20 colors with one of five format types.

A graph allows you to superimpose test results, specification limits, and other relevant information on one convenient screen. You can use different specification limits on the same graph, as well as putting both raw and corrected data on the same graph. You can have as many graphs as you need, each having a unique set of information. The same data element can appear on each graph. The graphs update dynamically as the graphed data changes.

Defining a Graph

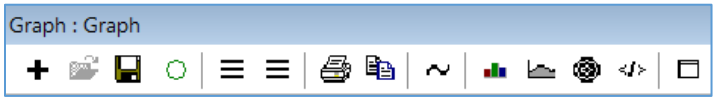
There are two ways to create a new graph. The first way is to go to the **Project** window, right-click **graphs** and select **Add**. The second way is to single-click graphs to highlight it, and then click on the + sign.

Each new graph is started as a blank graph with no selected data elements. The blank graph defaults to a frequency range of 30 MHz to 1 GHz, log frequency scaling, and the amplitude is set to 0 100.

Copying a graph

To duplicate an existing graph, go to the **Project** window, click the + drop down next to graphs, right-click on the graph you want to duplicate, and select **Copy**. Then right-click on **graphs** and select **Paste**. The new graph will be identical except the name will have a numerical appendage (starting at 1) for each time the graph is copied. **Make sure you change the name of the graph after copying**.

Graph Toolbar



The icons on the graph toolbar have the following functions:

- The **plus** icon opens the **Display Data dialog box**. The Display Data dialog box provides a list of available data elements to display on the graph and provides the interface for each element to select the color and format for it. See *Data Display* for further details.

  Due to the evolution of the software, this is the same as the **right-most 3 horizontal** lines. Therefore, data can simply be dragged and dropped onto the graph and a default color will be assigned, so either of these buttons will open the interface to change the data attributes.

- The **folder** icon is currently not functional and will be reserved for future functions.

- The **floppy** icon will open a file explorer to save the graph to a specific location.

- The **green circle** icon is a refresh execution that may be used if a static parameter has been changed and the user wants to force an update to the graph.

- The **left-most 3 horizontal** lines icon opens the Graph Setup that is used to set all the parameters of the graph. See *Graph Setup Tabs* for further details.

- The **printer** icon will open the print interface.

- The **copy** icon will place a copy of the graph onto the clipboard. The new graph will be identical except the name will have a numerical appendage (starting at 1) for each time the graph is copied.

- The **Cursor Interpolate** icon interpolates the current cursor position between points based on whether the plot is linear or log and does not snap to defined data points on a graph. See the section *Cursor Interpolation* for further details.

- The **GNU plot** icon is currently under development to enable expanding of the graphing of data.

- The **new plot** icon allows the user to select points to add or delete from a list or select ranges.

- The **polar** icon will change the graph to a polar plot to display results of directional data by interpreting the X-axis as degrees instead of Hz. This can be used with the Polar action in the flowchart to draw azimuth centric data.

- The **3D graphing** icon is currently under development to enable expanding of the graphing of data.

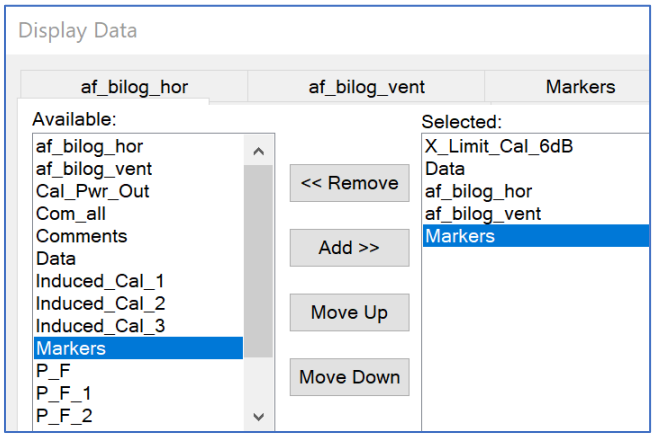
- The **maximize** icon will maximize the graph over the entire screen.

Display Data

With the graph in the front focus, click the **right-most 3 horizontal** lines or the + sign on the graph menu bar to access the Display Data dialog box.

It is divided into two columns, **Available** and **Selected**. They can be selected by double clicking on them or selecting the **Add>>** button. They can be removed by double click or the **<<Remove** button.

On the Selected side, you can rearrange the list by using the **Move Up/Move Down** buttons, which rearranges the sequence of the data element tabs.

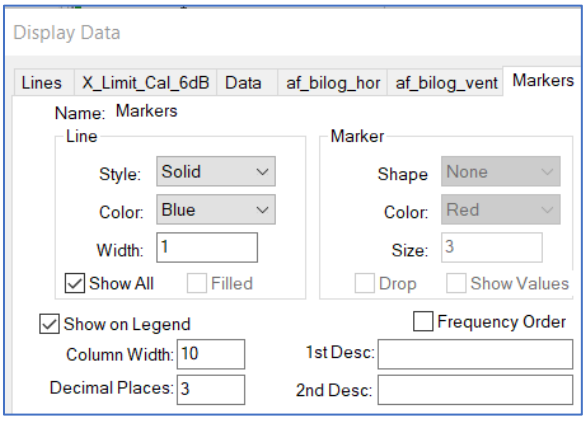


Controlling Display Conditions

Once a data element has been selected, a **Line** tab appears which determines the display parameters for this data element on the graph. Line type, color, and shape are individually defined for each data element.

If a data element was defined as an interpreted element (for example, continuous at every point, which is the default), then the first block labeled **Line** applies. If the data element was defined as discrete, then the box labeled **Marker** applies.

- **Line** — Style, color, and width are the choices for a line. These are only appropriate for data elements that are defined as continuous (either log or linear).



The default settings create a neutral line setting. To highlight the line, choose a different color or shape. This is particularly helpful when displaying the specification limits on the same chart as the readings. By using a dashed line or a different color for the specification limit, you can easily distinguish this line from other readings.

  - **Style** — 5 line styles are available. The default setting is solid.
  - **Color** —20 colors are available. The default setting is blue.
  - **Width** — The default width is 1. Make this number larger to thicken the line. This setting does not work with all line styles.

- **Marker** — Shape, color, and size are the choices for a marker. These are only appropriate for data elements that are defined as non-continuous.

  The default setting will not be visible on a graph. To highlight the marker, choose a different color or shape. This is particularly helpful when displaying the points more than the specification limits on the same chart as the readings. By using a triangular point or a different color for the points over specification limit, you can easily distinguish those points from the readings.

  - **Shape** — 9 shapes (10 if you count None) are available. The default shape is None. It is important to set a shape or nothing will appear on the graph.
  - **Color** — 20 colors are available. The default is red.
  - **Size** — Controls the thickness of the marker. The default value of 3 is useable in most cases.
  - **Drop** — Refers to whether the area under marker is filled in (each element drops to the X-axis). This is very helpful in highlighting the specific points of interest. Drop is only appropriate for graphs.

- **Show on legend** — Shows the line in the legend. The default value is on.

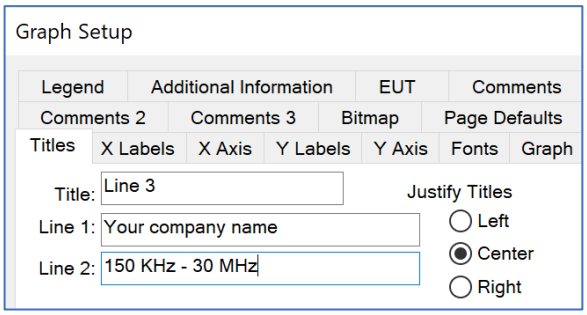
Graph Setup Tabs

The **Graph Setup** has 15 tabs that allow you to edit the title, axis scaling and axial labeling, fonts, and other special features.

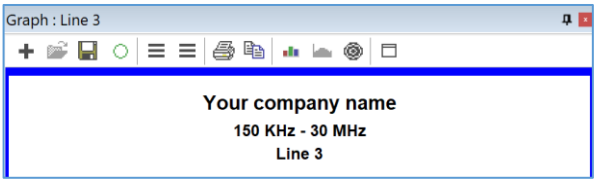
With the graph in the front focus, click the **left-most 3 horizontal** lines on the graph menu bar to access Graph Setup. You can also double click the graph to open the Graph Setup.

Titles Tab

The **Title** box is for the graph name. The next two, **Line 1 and Line 2**, are 128 character boxes that are displayed above the title on the graph. These are used for header type descriptions of the graph, such company name and test descriptions.

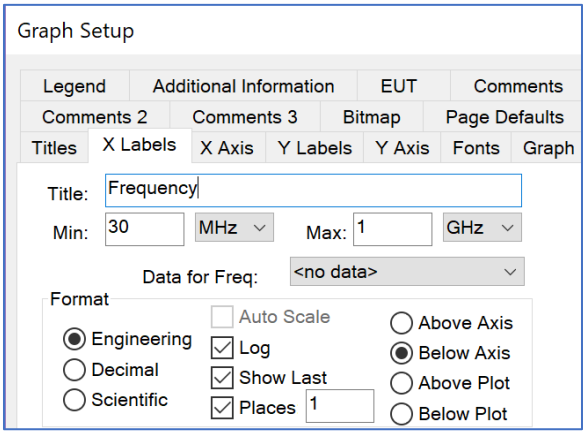


Use **Justify Titles** to specify left, center, or right justification of the titles. All three titles (Line 1, Line 2, and Title) are justified together.



X Labels Tab

This tab includes the title, range (with options), format of the scale, and the position of the label relative to the X-axis.



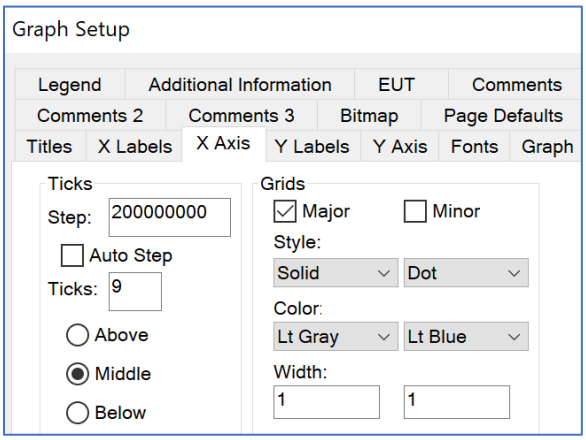
- **Title** — The X-axis label should be limited to 50 characters. In theory, you can use any number of characters, but the scaling of the X-axis makes more than 50 difficult to display. The default title is “Frequency”. For example, if you want to change the title to indicate units, change this field to “Frequency (MHz)”.
- **Frequency Range (Min/Max)** — The frequency range has different effects, depending upon the format setting Log.

  If the **Log** function is **on**, which is the default, the labels will be adjusted to the nearest log scale. In the default setting, the range is 30 MHz to 1 GHz with log scaling, which equates to a graph display of 10 MHz to 1 GHz. If **Log** is **off**, then the actual frequency range shown will be in effect for the graph.
- **Data for Freq** — If a data element is selected, the graph will adjust itself to the starting and ending frequency range of the data element. This is very handy if you are configuring a test where the start and stop frequency will change with each scan.
- **Format** — General settings for the scaling of the graph.
  - **Engineering/Decimal/Scientific** — Engineering, Decimal, and Scientific refer to how the number is displayed. The default is Engineering, which displays in kHz, MHz, and GHz; one megahertz would display as 1 MHz. Decimal is a raw format in which 1 MHz would display as 1000000. Scientific would display 1 MHz as 1e6.
  - **Auto Scale** — This is disabled.
  - **Log** — Determines the scaling factor for the X-axis. When in log mode, the lower range setting will be set to the nearest factor of 10 which encompasses it. For example, 30 MHz would be 10 MHz, and 200 MHz would be 1 GHz. If turned off, then the scaling will be linear starting and stopping at the specified frequencies. The default value is on.
  - **Show Last** — Indicates whether the last data point will be displayed on the chart, regardless of scaling when in a zoom mode. The default value is on.
  - **Places** — Refers to the number of relevant decimal places to be displayed on the X-axis scale: 1st place is 32.1, 2nd place is 32.11, etc. The default value is on at 1st place.

- **Axis/Plot** — Above Axis, Below Axis, Above Plot, and Below Plot refer to the position of the X-axis label and ticks regarding the graph and overall plotting area. The default value is Below Axis.

X-axis Tab

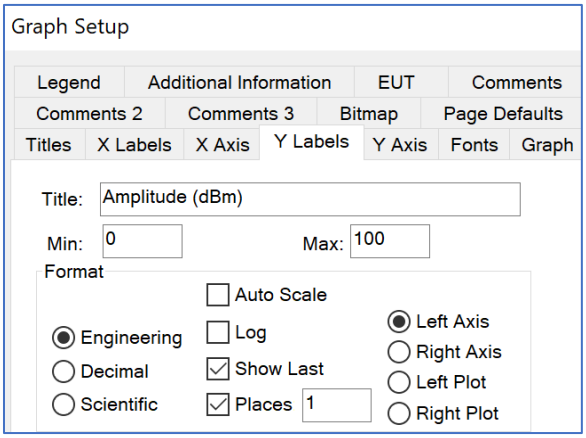
This tab controls the ticks and grid line colors for the X-axis. These can be configured to suit the zoom range of the X-axis.



- **Axis Ticks** — The ticks are the small marks scratched on the axis lines to designate spacing.
    - **Step** — The step scaling determines the position of labels for the X-axis. This is only active when the log mode is turned off. When creating a tick scaling on the X-axis, the range needs to match the desired frequency range. If the log scaling is turned off, the default grid ticking will write a tick mark at every 200 MHz step. If the scaling is zoomed from 20 MHz to 200 MHz, change the tick marking to 2e7 to make them 20 MHz steps. The default value is 200000000.
    - **Auto Step** — When this check box is selected, the vertical scale will be evenly divided into 10 steps. The default is set to off.
    - **Ticks** — Sets the number of tick marks placed across the lower scale between each marker. If you are in the 30 MHz to 1 GHz range with 200 MHz steps, a tick spacing of 9 would cause there to be nine small tick marks on the axis between 30 MHz and 230 MHz, between 230 MHz and 430 MHz, and so on.
    - **Above, Middle, Below** — Specifies whether the tick mark appears above the line, on the line, or below the line.
- **Grids** — Controls the shape, color, and size of the X-axis grid lines. Major grids are those at each label position. Minor grids are at each tick mark location. Major and Minor both have separate settings below, with the left setting as major and right settings as minor.
    - **Major/Minor** — Check each of these boxes to turn on or off the appearance of grid lines. The default is set to Major on.
    - **Style** — The styles available are solid, dot, dashed, dash dot, or dash dot dot. The default style for Major is solid.
    - **Color** —20 colors are available. The default color for Major is Lt Gray.
    - **Width** — Configures the size of the line. It will make the grids thicker or thinner, the minimum (and default) is 1.

Y Labels Tab

This tab includes the title, range, format of the scale, and the position of the label relative to the y-axis.

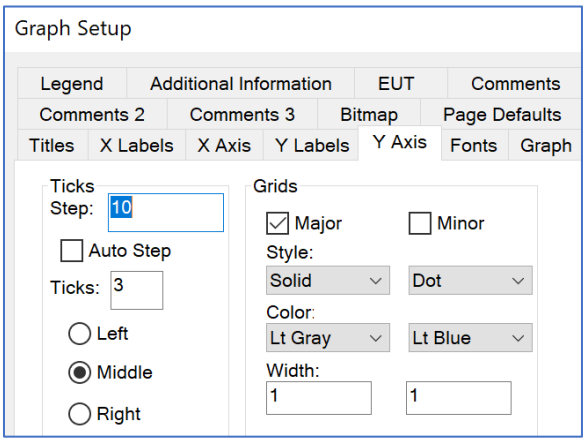


- **Title** — The y-axis label should be limited to 30 characters. In theory, you can use any number of characters, but the scaling of the Y-axis makes more than 30 difficult to display. The default setting is “Amplitude”. For example, if you wanted the title to indicate units, change this field to “Amplitude (dBm)”.
- **Amplitude Range (Min/Max)** — The amplitude range has different effects depending upon the format setting. If the log function is on, the labels will be adjusted to the nearest log scale. In the default setting, the range is 0 to 100. In log scaling, this will be 0.1 to 100.
- **Format** — General settings for the scaling of the graph.
    - **Engineering/Decimal/Scientific** — Engineering, Decimal, or Scientific refer to how the number is displayed. The default is Engineering, which displays in kHz, MHz, and GHz; one megahertz would display as 1 MHz. Decimal is a raw format in which 1 MHz would display as 1000000. Scientific would display 1 MHz as 1e6.
    - **Auto Scale** — The default is set to off.
    - **Log** — Determines the scaling factor for the y-axis. When in log mode, the lower range setting will be set to the nearest factor of 10, which encompasses it.

      For example, 30 MHz would be 10 MHz, and 200 MHz would be 1 GHz. If turned off, then the scaling will be linear starting and stopping at the specified frequencies. The default value is off.
    - **Show Last** — Indicates whether the last data point will be displayed on the chart, regardless of scaling when in a zoom mode. The default value is on.
    - **Places** — Refers to the number of relevant decimal places to be displayed on the y-axis scale: 1st place is 32.1, 2nd place is 32.11, etc. The default value is on at 1st place.
    - **Axis/Plot** — Left Axis, Right Axis, Left Plot, and Right Plot refer to the position of the Y-axis label and ticks regarding the graph and overall plotting area. The default value is Left Axis.

Y-axis Tab

This tab controls the ticks and grid line colors for the Y-axis. These can be configured to suit the zoom range of the Y-axis.

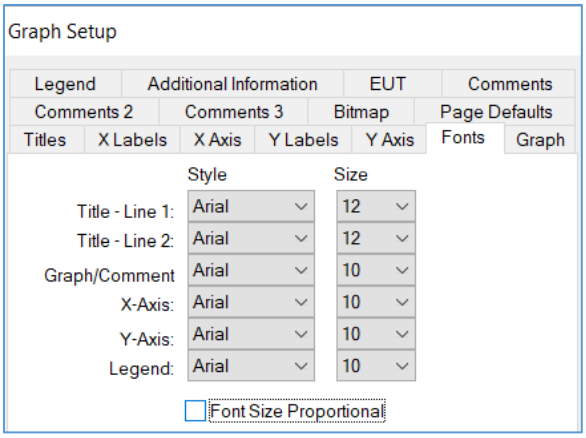


- **Axis Ticks** — The ticks are the small marks scratched on the axis lines to designate spacing.
    - **Step** — The step scaling determines the position of labels for the Y-axis. This is only active when the log mode is turned off. When creating a tick scaling on the Y-axis, the range needs to match the desired amplitude range.

      If the log scaling is disabled, the default grid ticking will write a tick mark at every 10 points. If the scaling is zoomed to one to ten (for example, in looking at probe values in a 3 V/M test), change the tick marking to one to make the vertical scale 1 V/M steps. The default value is 10.
    - **Auto Step** — When this check box is selected, the vertical scale will be evenly divided into 10 steps. The default value is off.
    - **Ticks** — Sets the number of tick marks placed between each marker. If you are in the 0 100 range with 10 steps, a tick spacing of 3 would cause there to be three small tick marks on the axis between 0 and 0, between 10 and 20, and so on. The default value is three.
    - **Left, Middle, Right** — Specifies whether the tick mark appears left of the line, on the line, or to the right of the line.
- **Grids** — Controls the shape, color and size of the Y-axis grid lines. Major grids are those at each label position. Minor grids are at each tick mark location. Major and Minor both have separate settings below, with the left setting as major and right settings as minor.
    - **Major/Minor** — Check each of these boxes to turn on or off the appearance of grid lines. The default is set to Major on.
    - **Style** — The styles available are solid, dot, dashed, dash-dot, or dash-dot-dot. The default style for Major is solid.
    - **Color** —20 colors are available. The default color for Major is Lt Gray.
    - **Width** — Configures the size of the line. This will make the grids thicker or thinner, the minimum (and default) is one.

Fonts Tab

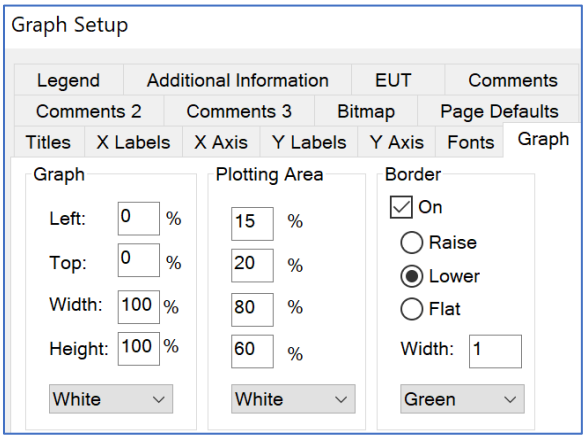
This tab allows the user to customize the fonts used for the major components of the graph. The style and size are available for each choice. The five font choices are Arial, Courier New, Script, Times New Roman, and Decorative. If these fonts are not installed, a close font is the default. The size choices available are 6 – 20.



- **Title-Line 1** — This font is only used on the Line 1. The default is an Arial with 10-pitch font. This makes this the largest font on the screen (to emphasize the line).
- **Title-Line 2** — This font is only used on the Line 2. The default is Arial with an 8-pitch font.
- **Graph/Comment** — This font is used on the graph title as well as the X-axis and Y-axis titles. The default is Arial with an 8-pitch font.
- **X-Axis** — This font is used to set the size of the X-axis labels (and tick indicators). The default is Arial with a 6-pitch font.
- **Y-Axis** — This font is used to set the size of the Y-axis labels (and tick indicators). The default is Arial with a 6-pitch font.
- **Legend** — This font is used to set the size of the legend as well as the font size and style for all additional information (see the *Additional Information Tab* for further details). The default is Arial with a 6-pitch font.

Graph Tab

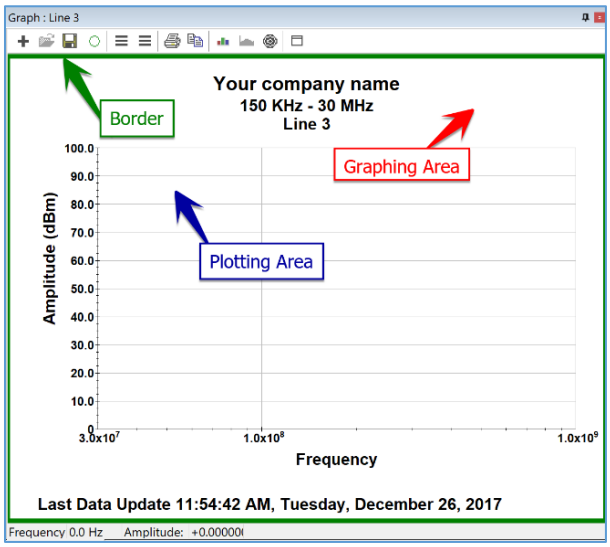
This tab allows you to define the relative size and area of the graph, the plotting area, and the area for text on the graph.



- **Graph** — The overall graphing area is defined by stating how far down from the top of the graph page the graphing area begins. This is stated in percentage of the area. The defaults settings are recommended.
  - **Left** — Represents the starting point for the left side of the graph as a percentage of the page, starting from the upper left-hand corner. The default value is 0%.
  - **Top** — Represents the starting point for the top of the graph as a percentage of the page, starting from the upper left corner. The default value is 0%.
  - **Width** — Represents the width of the graph as a percentage of the page size. The default value is 100%.

- **Height** — Represents the height of the graph as a percentage of the page size. The default value is 100%.
- **Color** — 20 colors are available. The default color is White.

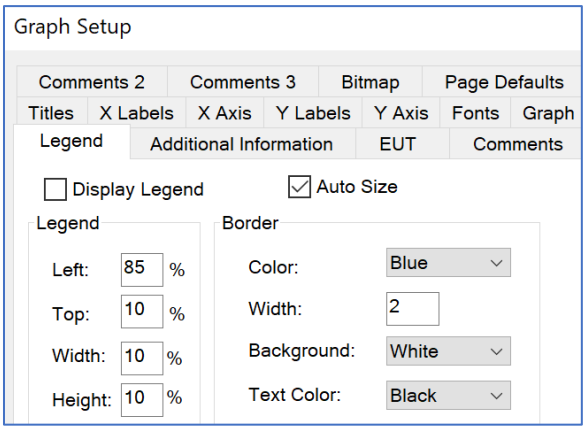
- **Plotting Area** — Uses the same convention as Graph but all percentages are of the graph size, not the page size. It represents the part of the graph area, which includes the X Y presentation of data. All distances are displayed in percentages of graph size and thus automatically scale to whatever graph size is used.



  - **Left** — The default placement of the left side of the plotting is 15% of the graph size. This allows room for the Y-axis labels and titles to be written outside of the Y-axis.
  - **Top** — The default placement of the top of the plotting is 20% of the graph size. This allows room above the plotting area for the Line 1, Line 2, and Title texts.
  - **Width** — The default width of the plotting area is 80% of the graph size. Since there are no default titles or labels on the right side of the graph, the default settings leave 5% of the graph area free along the right side of the graph.
  - **Height** — The default height of the plotting area is 60% of the graph size. This leaves room for the operator, EUT, client, data/time stamp, and file name information that is set on the *Additional Information Tab*. Change this setting if you want a larger graph and no additional information displayed.
  - **Color** — 20 colors are available. The default color is White.
- **Border** — The area of the page outside the graph is defined as the border area. The color of the border can be set as needed. One common use of this feature is to define a top or side region that is all border. This would be where the 3 hole punch would go to place this graph in a bound document. This way none of the graph is lost to the hole placement.
  - **Raise, Lower, and Flat** — These settings are not normally visible unless a high-resolution monitor is used. Generally, assume they are not active.
  - **Width** — The width of the border is stated in percentage terms.
  - **Color** — 20 colors are available. The default color is Green.

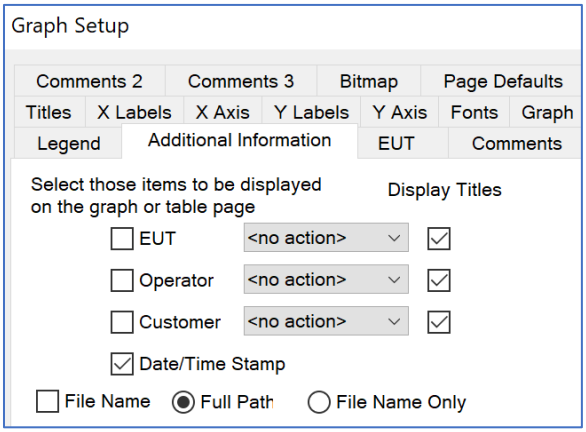
Legend Tab

You can place a legend on the graph to illustrate the data elements displayed. The default names in the legend are the data element names. If you want to control the displayed legend name, go to *Display Data* and type the name you want to appear into 1st Desc.



- **Display Legend** — Select the check box to activate the legend. The legend is un-checked by default.
- **Auto Size** — Determines whether the legend box expands to encompass the defined elements. If this is disabled, the legend box must be manually sized to include the defined elements. The default is checked.
- **Legend** — The Left, Top, Width, and Height of the legend box are placement percentages. They are referenced to the upper left corner of the graphing area. The percentages can be any number between 0 and 100. **These are only used if auto size is un-checked**.
    - **Left** — The default is set to 85% of the graphing area.
    - **Top** — The default is set to 10% of the graphing area.
    - **Width** — The default is set to 10% of the graphing area.
    - **Height** — The default is set to 10% of the graphing area.
- Border — These settings control properties of the legend. The Color, Width, Background, and Text Color boxes allow you to customize the legend.
    - **Color (border)** — 20 colors are available. The default color is Blue.
    - **Width** — Controls the thickness of the border. This can be set to any numerical value between 1 and 10. The default value is two.
    - **Background (Color)** — Refers to the color of the interior area of the legend box. Twenty different colors are available. The default color is White.
    - **Text Color** — 20 colors are available. The default color is Black.

Additional Information Tab



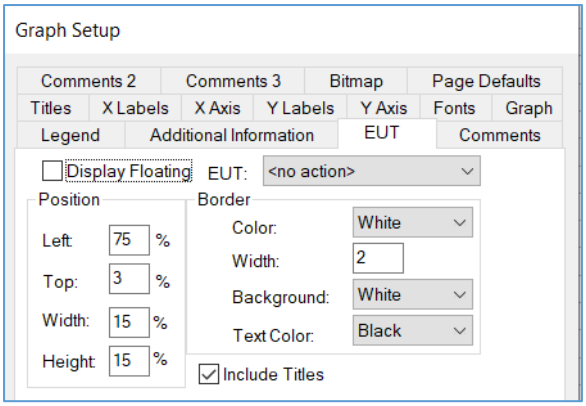
This tab is used to display information about the EUT, operator/customer, date/time stamp, and the file name onto the graph.

This information is shown across the bottom margin of the graph when the checkbox on the left is selected and the positions are fixed. Check the **Display Titles** box to show the titles for the corresponding information.

You can have multiple EUT/Lab/Client actions on the flowchart, but only one on each graph. They can be different for different graphs.

- **EUT** — Select the appropriate *EUT Information* action from the flowchart to display the EUT Equipment ID. Use the **EUT tab** to display more information than just the Equipment ID.
- **Operator** — Select the appropriate *Lab Information* action from the flowchart to display the Operator name.
- **Customer** — Select the appropriate *Client Information* action from the flowchart to display the Customer's company and contact person name.
- **Date/Time Stamp** — Displays the date/time stamp from the newest data element on the graph to reflect the last execution that affected the data shown on this graph.
- **File Name**:
  - **Full Path** — Displays the complete windows path including directories and subdirectories. This can be a long value depending upon the file name and subdirectory situation.
  - **File Name Only** — Displays to the profile name and extension only.

EUT Tab



This tab displays a floating box with all the information from the EUT action from the flowchart. This allows significantly more information to be displayed than is shown when the Additional Information/EUT option is chosen.

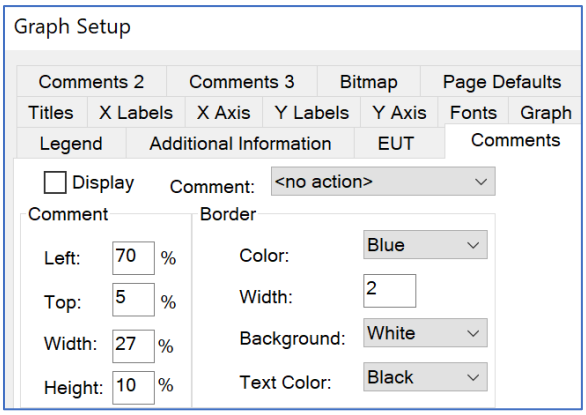
- **Display Floating** — Select this check box to display the EUT information. The default is un-checked.
- **EUT (Action)** — Select an appropriate *EUT action* from the flowchart to display all shown titles from the action.
- **Position** — Allows placement of the EUT information box at any location within the graph. All positions are referenced to the upper left corner of the graphing area and are in percentages of the graph size. The percentages can be any number between 0 and 100.

- **Left** — The default is set to 70%.
- **Top** — The default is set to 5%.
- **Width** — The default is set to 15%.
- **Height** — The default is set to 20%.

- **Border** — You can control the shape, color, and sizes of the floating box using these settings.
  - **Color (Border)** — 20 colors are available. The default color is Blue.
  - **Width** — This setting controls the thickness of the border. This can be set to any numerical value between 1 and 10. The default value is two.
  - **Background (Color)** — 20 colors are available. The default color is White.
  - **Text Color** — 20 colors are available. The default color is Black.
- **Include Titles** — If this check box is selected, the EUT titles will also be displayed. The default is un-checked to save space.

Comments Tab

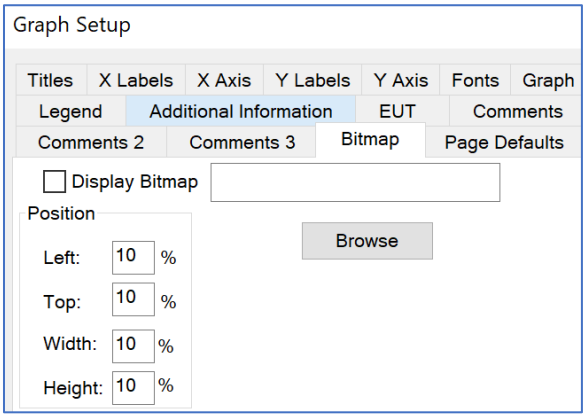
This tab allows a comment to be printed on the graph.



- **Display** — Select this check box to display the comments. The default is un-checked.
- **Comment (Action)** — Select an appropriate *Comment action* from the flowchart for inclusion on this graph.
- **Comment (Position)** — Allows placement of the Comment information box at any location within the graph. All positions are referenced to the upper left corner of the graphing area and are in percentages of the graph size. The percentages can be any number between 0 and 100.
  - **Left** — The default is set to 70%.
  - **Top** — The default is set to 5%.
  - **Width** — The default is set to 15%.
  - **Height** — The default is set to 20%.
- **Border** — You can control the shape, color, and sizes of the floating box using these settings.
  - **Color (Border)** — 20 colors are available. The default color is Blue.
  - **Width** — This setting controls the thickness of the border. This can be set to any numerical value between 1 and 10. The default value is two.
  - **Background (Color)** — 20 colors are available. The default color is White.
  - **Text Color** — 20 colors are available. The default color is Black.

Bitmap Tab

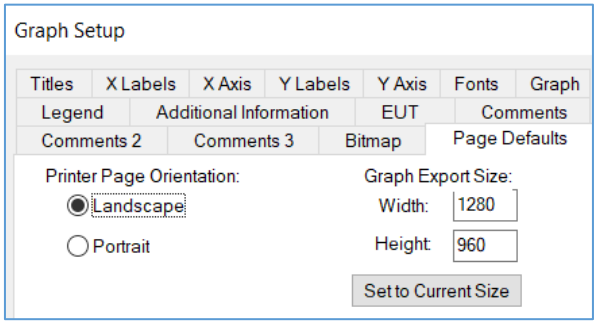
This tab allows the user to place a bitmap, typically a company logo, on the graph.



- **Display Bitmap** — Select this check box to display the specified bitmap. The default is un-checked.
- **Browse (Bitmap)** — Click the **Browse** button to launch a dialog box to specify the directory and name for the bitmap. If you know the name, you can directly type it in this box. If a profile with a defined bitmap is moved to a computer which does not have the bitmap file in its directory, this will be un-checked when loaded.
- **Position** — This allows placement of the bitmap at any location within the graph. All positions are referenced to the upper left corner of the graphing area and are in percentages of the graph size. The percentages can be any number between 0 and 100.
    - **Left** — The default is set to 10%.
    - **Top** — The default is set to 10%.
    - **Width** — The default is set to 10%.
    - **Height** — The default is set to 10%.

Page Defaults Tab

This tab controls the orientation of the graph to the printed page and they are mutually exclusive.

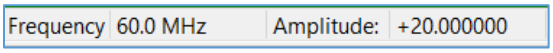


- **Printer Page Orientation**
    - **Landscape** — Prints the graph in landscape orientation regardless of the printer settings and paper size. It will adjust the width and height to the available paper size of the printer. Set the default printer to the desired settings prior to printing.
    - **Portrait** — Prints the graph in portrait orientation regardless of printer settings and paper size. This will leave a large, unprinted area below the graph on most papers since the graph will adjust itself, in height and width, to that of the paper.
- **Graph Export Size** — This allows the pixilated export file to match what is being viewed on the screen. The EMF and WMF files will inherit the screen resolution as a default.
    - **Width/Height** — Sets the dimensions of the graph to a set resolution. If the exported graph does not look correct, resize this until it is correct.

- **Set to current size** — This will leave the export with the dimensions of the graph as seen on the display.

Graph features

Cursor Position



There are two boxes on the lower left corner of the graph. Both boxes display the position of the nearest data point to the cursor. The first box displays the frequency, and the second box displays the value.

Temporary Zoom

You can temporarily zoom in or out of a graph to look at specific details. These temporary zooms are never saved; the originally designed graph is always the saved item.

- **Zoom In** — Hold the right mouse button down and drag the area that you want to expand. The ranges of the X-axis and Y-axis will match the zoomed area unless the Log option is set on the **X Labels** or **Y Labels** tab.
- **Zoom Out** — Double-right click on the graph to return to the original ranges.

Cursor Interpolation

Cursor interpolate does not snap to defined data points on a graph. It interpolates the current cursor position between points based on whether the plot is linear or log.

Tables

Tables are a tabular method of looking at data elements within the TILE! system. They can be created as often as desired and you can have as many different tables as desired. Each table can have different combinations of data elements to fit the requirements. These tables can be printed individually or used to clip data to the spreadsheet. Tables are generally good for looking at specific, limited sets of data, such as peaks or QP values. A table is created with the appropriate number of rows to fit the defined data elements.

Tables are not normally used for listing raw data. If you place raw data onto a table, you might have a thousand or more rows of data. Since each page only holds 66 lines of data, the table must build and hold in memory hundreds, or thousands, of pages of data. This is extremely memory intensive and can account for significant delays in testing. As you add to the data, the complete tabular data must be reformatted in memory. Again, this can cause a significant delay in system performance.

Defining a Table

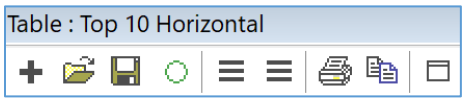
There are two ways to create a new table. The first way is to go to the **Project** window, right-click **tables** and select **Add**. The second way is to single-click tables to highlight it, and then click on the + sign.

Each new table is started as a blank table with no selected data elements.

Copying a table

To duplicate an existing table, go to the **Project** window, click the + drop down next to tables, right-click on the table you want to duplicate, and select **Copy**. Then right-click on tables and select **Paste**. The new table will be identical except the name will have a numerical appendage (starting at 1) for each time the table is copied. **Make sure you change the name of the table after copying**.

Table Toolbar



The icons on the table toolbar have the following functions:

- The **plus** icon opens the **Display Data dialog box**. The *Display Data dialog box* provides a list of available data elements to display on the graph and provides the interface for each element to select the color and format for it.

  Due to the evolution of the software, this is the same as the **right-most 3 horizontal** lines. Therefore, data can simply be dragged and dropped onto the graph and a default color will be assigned, so either of these buttons will open the interface to change the data attributes.

- The **folder** icon is currently not functional and will be reserved for future functions.

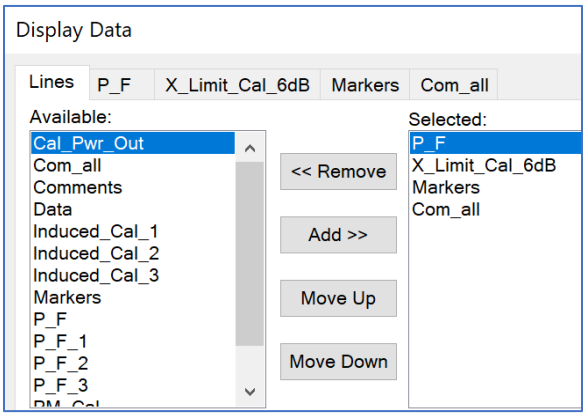
- The **floppy** icon will open a file explorer to save the graph to a specific location.
- The **green circle** icon is a refresh execution that may be used if a static parameter has been changed and the user wants to force an update to the graph.
- The **left-most 3 horizontal** lines icon opens the Table Setup that is used to set all the parameters of the graph. See *Table Setup Tabs* for further details.
- The **printer** icon will open the print interface.
- The **copy** icon will place a copy of the graph onto the clipboard. The new graph will be identical except the name will have a numerical appendage (starting at 1) for each time the graph is copied.
- The **maximize** icon will maximize the table over the entire screen.

Display Data

With the table in the front focus, click the **right-most 3 horizontal** lines or the + sign on the table menu bar to access the Display Data dialog box.

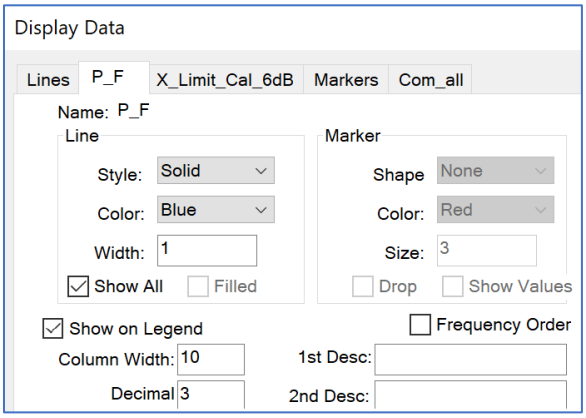
It is divided into two columns, **Available** and **Selected**. They can be selected by double clicking on them or selecting the **Add>>** button. They can be removed by double click or the **<<Remove** button.

On the Selected side, you can rearrange the list by using the **Move Up/Move Down** buttons, which rearranges the sequence of the data element tabs.



Controlling Display Conditions

Once a data element has been selected, a **Line** tab appears which determines the display parameters for this data element on the graph. Line type, color, and shape are individually defined for each data element.



If a data element was defined as an interpreted element (for example, continuous at every point, which is the default), then the first block labeled **Line** applies. If the data element was defined as discrete, then the box labeled **Marker** applies.

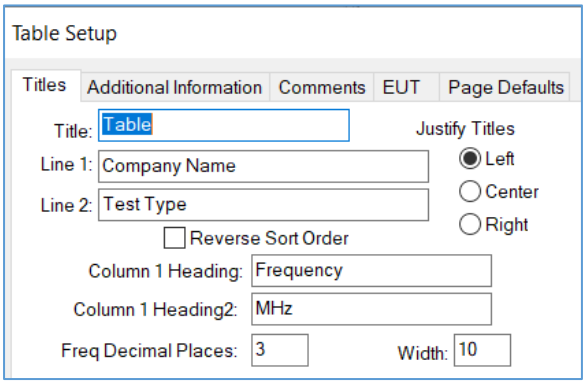
- **Line** — Style, color, and width are the choices for a line. These are only appropriate for data elements that are defined as continuous (either log or linear).
    - **Show All** — Controls whether all the contents of the data element are displayed or only those with matching frequencies to the remaining columns.

      If you have a specification limit on a table with 10 QP readings and have the Show All check box selected, it would also show the frequencies in the specification limit file. This means the number of lines on the table would be greater than 10. If this is off, only the 10 points of interest would be listed but this data column would show the appropriate matching value for this frequency.
- **Marker** — Shape, color, and size are the choices for a marker. These are only appropriate for data elements that are defined as non-continuous.
- **Show on legend** — Shows the line in the legend. The default value is checked.
- **Frequency Order** — Separately sorts all data from low frequency to high frequency. The default is un-checked.
- **Column Width** — Controls the width of the column. The default value is 10.
- **Decimal Places** — Controls the number of decimal places that are visible. This field affects the way tables are copied and pasted. If you export values using the table function, they will be truncated to the value shown in this field. The default value is three.
- **1st Desc** — This field allows the user to control the column header for this data element. The default is the data element name.
- **2nd Desc** — There is an option for a 2nd description for each data element. This field is in parenthesis after the first description, and is typically used for units of measure.

Table Setup Tabs

With the table in the front focus, click the **left-most 3 horizontal** lines on the table menu bar to access Table Setup. The Table Setup dialog box has five tabs that allow the user to control the appearance of the table and some of the characteristics when the table is cut and pasted into other applications. These characteristics include name, descriptive titling, column headers, the inclusion of additional information, comments, page orientation, as well as cut and paste defaults.

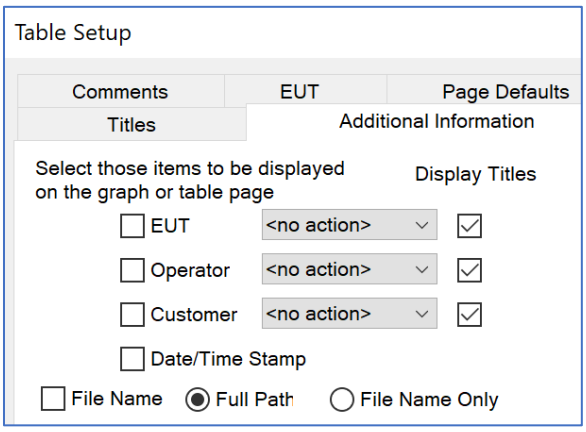
Titles Tab



- **Title** — Enter a unique name to identify this table when automatically printing or accessing the table from different applications.
- **Line 1** — Line 1 is the top line of the table. It is normally used to enter the company name or other appropriate header information. You can enter up to 128 characters, although this might exceed the width of the page.
- **Line 2** — Line 2 is the second line of descriptive headers for the table. It is normally used to display information on the test and its general conditions. You can enter up to 128 characters, although this might exceed the width of the page.
- **Justify Titles** — Specify left, center, or right justification of the titles. All three titles (line 1, line 2, and title) are justified together.
- **Reverse Sort Order** — When this check box is deselected, the frequencies list from low to high frequencies. When this check box is selected, the list will be from high to low frequency.
- **Column 1 Heading** — For a graph, the column 1 heading is the title to be used in the legend. If this field is blank, the data element name will be used. In a table, the default for this is "Frequency"; the user can change this to any appropriate term but the actual data in this column is always the frequency.
- **Column 2 Heading** — There are two headers for each column in the table. This field is in parenthesis after the first description, and is typically used for units of measure. The default value is "MHz".
- **Freq Decimal Places** — The user can control the number of decimal places displayed for the frequency. The default is three (for example, 3.755 MHz). This limitation extends to cut and paste operations.
- **Column Width** — Controls the column width of the first column. If you are specifying more than three decimal places for frequency, you might need to widen the column.

Additional Information Tab

This tab is used to display information on EUT, the operator/customer, date/time stamp, and the file name onto the graph. This information is shown across the top of the table, immediately below line 1, line 2, and the table name.



- **EUT** — You can display the EUT model number (string one from the EUT action) across the bottom of the graph. Click the **EUT** check box and select the appropriate action from the flowchart. You can have multiple EUT actions on the flowchart, but only one on each graph. They can be different for different

graphs. Click the **Display Titles** check box if you want the title from the *EUT action* to display.

- **Operator** — You can display the operator name across the bottom of the graph. Click the **Operator** check box and select the appropriate action from the flowchart. You can have multiple operator actions on the flowchart, but only one on each graph. They can be different for different graphs. Click the **Display Titles** check box if you want the title from the *Operator action* to display.
- **Customer** — You can display the customer contact name and the company name (strings one and two from the Client action) across the bottom of the graph. Click the **Customer** check box and select the appropriate action from the flowchart. You can have multiple client actions on the flowchart, but only one on each graph. They can be different for different graphs. Click the **Display Titles** check box if you want the title from the *Client action* to display.
- **Date/Time Stamp** — If this check box is selected, the date/time stamp from the newest data element on the graph will be displayed. By using the newest date/time stamp, we allow the graph to reflect the date/time of the last execution that affected the data shown on this graph.
- **File Name** — If this check box is selected, the file name will be displayed. This can be a long value depending upon the file name and subdirectory situation.
    - **Full Path** — Displays the complete windows path including directories and subdirectories.
    - **File Name Only** — Displays to the file name and extension only.

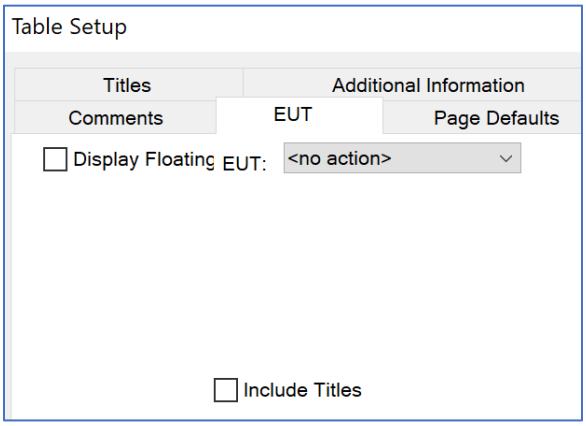
Comment Tab

This tab allows a comment to be printed on the table.

- **Display** — Select this check box to add the comment to the bottom of the table. The comment is the last item printed and will print across the bottom of the page.
- **Comment** — Select the appropriate *Comment action* from the flowchart.

EUT Tab

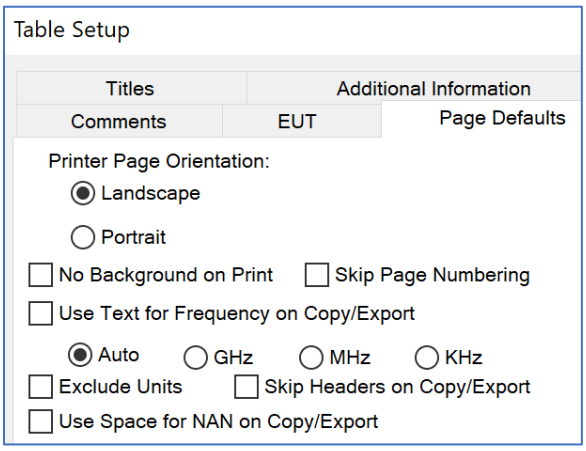
This tab allows selection of an action that contains EUT information. This information is then printed at the bottom of the table.



- **Display** — Select this check box to add the comment to the bottom of the table. The comment is the last item printed and will print across the bottom of the page.
- **EUT** — Select the appropriate *EUT action* from the flowchart.
- **Include Titles** — When selected, the title for each line of the EUT action is included prior to the information.

Page Defaults Tab

This tab allows you to control certain parameters of printing, cut, and paste operations.



- **Printer Page Orientation** — You can select landscape or portrait for printer orientation. This overrides the orientation of the default printer. This is done so that graphs and tables can be printed in different orientations without the user having to change to printer defaults each time.
- **No Background on Print** — When a table is viewed on the screen, there are lines defining each column and row; these print by default. If this check box is selected, the background is not printed.
- **Skip Page Numbering** — When tables are printed, a page number is normally assigned at the bottom of the page; when this check box is selected, no page number will be inserted.
- **Use Text for Frequency on Copy/Export** — When you copy a table, the frequency is normally copied as a value. In this case, 1 MHz would be 1,000,000. When this check box is selected, the frequency is converted to text in engineering format so that 1 MHz would appear as 1 MHz.

  The Auto, GHz, MHz, and kHz selection lets you specify the format. If Auto is chosen, then the label will be varied to fit the size of the data; 1,000 would be 1 kHz, and 1,000,000 would be 1 MHz. If you choose GHz, MHz, or kHz then the data is formatted to that size. For example, if kHz is chosen then 1,000 would be 1 kHz and 1,000,000 would be 1,000 kHz.
- **Exclude Units** — When this option is chosen, the text units are not copied.
- **Skip Headers on Copy/Export** — When copying a table to disk or in a cut and paste operation, it has a WYSIWYG (what you see is what you get) appearance. When this check box is selected, the headers will not be copied. You will only get the columns of data.
- **Use Space for NAN on Copy/Export** — When copying tables of values there is a technical difference between a zero and no value at all. Since these are normally numbers, the absence of a number shows up as a NAN (not a number) value. Choosing this option will substitute a space whenever no value exists so that there is a blank on the copied column, not a NAN.

Flowchart and Palette

The flowchart allows you to work with graphical tools to enhance the appearance or efficiency of the test profile. Imbedding graphics, such as bitmap, into the flowchart is a powerful method of clarifying a test requirement or setup. Labeling can be added to personalize the flowchart, imprint instructions in an obvious manner, or clarify attached graphics. For example, creating a sequence of icons to perform an emissions test is the heart of the TILE! system.

The flowchart has two functions:

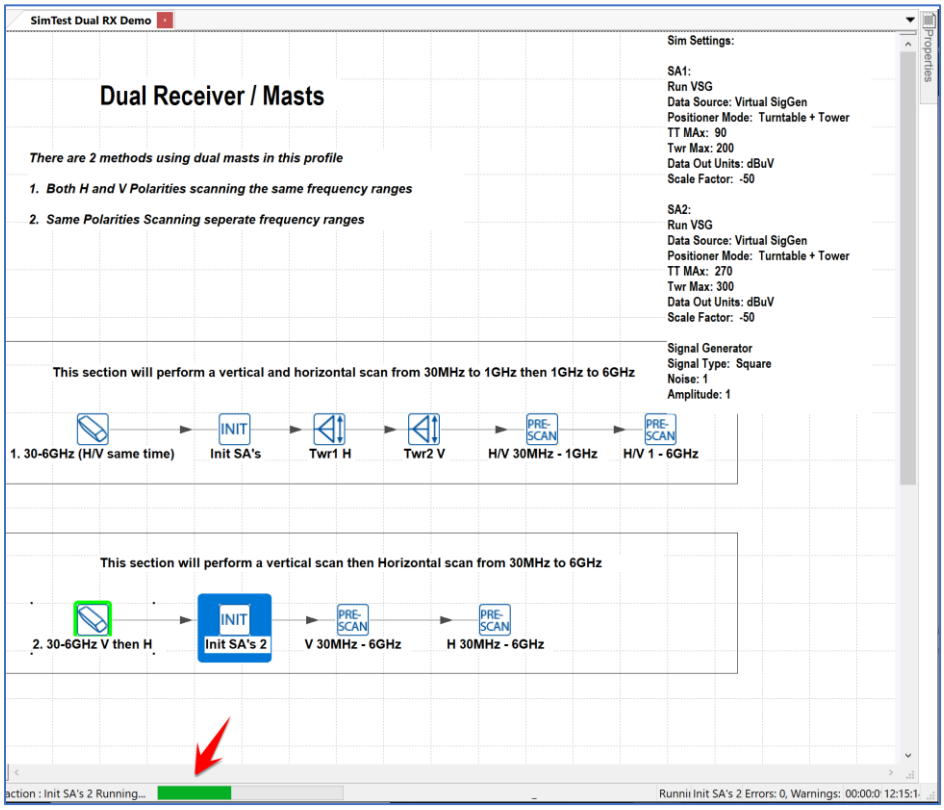
- Each action defines a specific set of information gathering or instrumentation controlling steps.
- It specifies the order of execution for the test.

A flowchart defines a complete test program, such as a simple emission scan test for a specific international standard. However, the real power of the TILE! concept is the ability to define a test profile that is a requirements test. It can include emission as well as immunity and acts as the repository for the final certification test data for a new product.

By consolidating a wide range of requirements into one test profile, the user can automatically store related information in a common file for future reference. This also dramatically simplifies the ability to generate test reports for submissions or internal reporting.

Flowchart Window

The TILE! Flowchart window displays and defines the actions included in the test and depicts the sequence of their execution. The flowchart, with its associated data elements and instruments, is the central component in creating a unique test profile.



Actions

The TILE! Palette

The palette is the gateway to profile development. The icons on the palette represent actions that perform various functions. Each action is a visual representation of a sequence of consolidated code that allows the user to visually create a function or build a sequence. Actions can stand-alone or follow a sequence by connecting them together with the connector tool. All actions will be described in the sections after this.

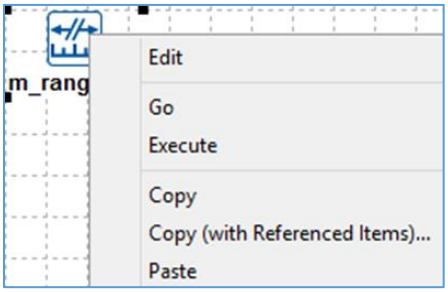
Select	Rectangle	Ellipse	Line
Multi-Line	Polygon	Text	Connect
Start	Auto Save	Save Windows	Print Pages
Lab Information	Client Information	Operator Information	EUT Information
Comments	Prompt	Initialize Instruments	Direct Entry
Math	Clear	Transfer Data	Launch Application
Picture	Timer	Weather	Serial Interface
Measure Range	Measure Range Extended	Scan Range Measurement	Fast Scan Across TT/Turn
Pre-Scan	Scan Peaks Measurement	Scan Peaks Meas (PSA)	Measure Peaks
Signal Disc Meas	Site Attenuation	Instrument Commands	Position Tower
Position Turntable	Read Instrument	Read Instrument (Trace)	GTEM 3 Position Correlation
GTEM 9 Position Correlation	Optimize GTEM Meas	Position EUT	Immunity Calibration
Immunity Test	CS103/4/5 measurement	Field Uniformity	Threshold Data Action
Monitor Interface	Calibrate Cables	Amplifier Cal	Reverb Calibration
Reverb Test	Reverb Eval	Surge	Burst
EFT	Dips	Harmonics	Polar Measurements
FCC Power Spec Limit	Occupied BW Mask	Switch Setup	Pass/Fail Setup Reading (DAQ)
Switch Setup	Pass/Fail Setup Reading	Switch Setup	Switch Setup (DAQ)
EmCenter Switch	Call Setup		

Placing Action Icons

Each icon can be selected and dropped onto the flowchart during the design stage. Click once on the desired icon on the palette to select the action. When you move to the flowchart a small box indicates the icon is ready to be dropped. Click one time at the desired position to drop the icon. It is not important to have absolute position on the drop since the icon can be moved and aligned after placing.

Using Actions

Right-click on the icon to access the following:

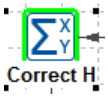


- **Edit** — Reveal the actions' settings.
- **Go** — Run the action and any sequenced after it via connectors.
- **Execute** — Run this action only.
- **Copy** — Copies the action and its settings to be pasted.
- **Copy (With Referenced Items)** — Copies the action and its settings as well as any referenced data items or instruments. This is used for the copying from one profile to another, not to be used in the same profile. If it is copied into the same profile, duplicates of data and instruments will be created.
- **Paste** — Paste the copied icon with an appended name (name_1).

Highlighted Actions

The individual actions in the flowchart will highlight to show the status and progress of the profile while running. This is very handy while developing a new profile that has many actions.

A blue highlight indicates the current action being processed.



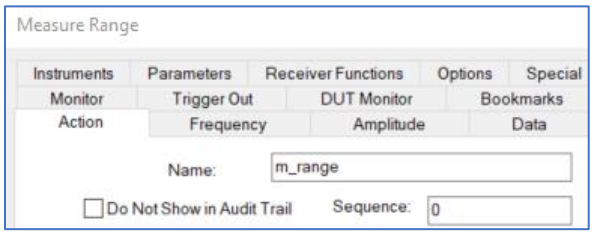
A green highlight indicates that the action has processes successfully.

A red highlight indicates that the action was interrupted either by the user or by loss of an instrument connection.

Common Tabs for Actions

Name Tab

All actions have a common name page. The name is used to add a unique title to the flowchart and refers to this object when using shortcut icons or macro lookups.



- **Name** — Every action is given a name. If not named, they are considered untitled. Although the software does not limit you, we strongly recommend that every action have a name.

  Names are limited to 128 characters, but you should avoid long names since they crowd the flowchart. It is more important to make the titles descriptive of what the step is meant to do. These shortcut names will aid in moving between different sections of a test profile. **Do not use**

spaces, use underscore or camel case instead to differentiate separate words for an element name.

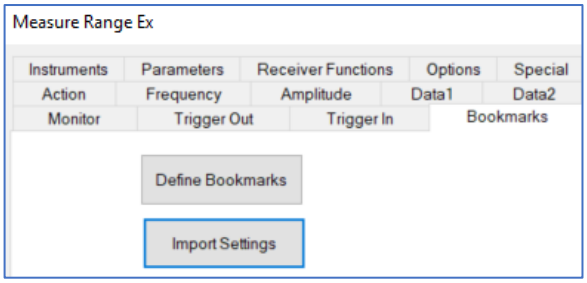
- **Do Not Show in Audit Trail** — If this option is selected, a description of this action will not be included in the audit trail unless the **Show All Actions** option is selected in the audit trail settings.
- **Sequence Option** — If the default sequence number of zero is left in place, the audit trail will sequence in order that the actions were added to the flowchart. The user can override this by entering a sequence number. All numbered items are sequenced in the audit trail first. If the same sequence number is used in different actions, they will be listed on the audit trail at that sequence position but in the order of creation.

Bookmarks Tab

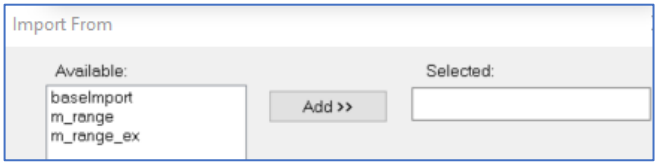
This tab allows the user to specify format for data so it can be seamlessly imported by a macro into a Word document. This tab also allows the user to transfer settings between similar actions, however you will have to reselect the data items since they will not import.

This tab also allows for the import of settings between similar actions like the example below:

- Drop a Measure Range Extended action in the flowchart, and edit it by selecting Import Settings on the bookmark tab.



- You will see a list of the Measure Range and Measure Range Extended actions that you can import.



Progress Bar

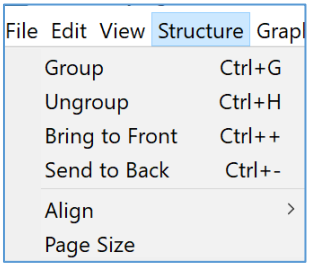
Along with the traditional timer and current action listed in the bottom right status bar, there is also a green progress bar in the status bar and highlighted actions. The status bar is not necessarily synchronized with the action progress; however, it does provide a quick look that the profile is running. The bottom left will show the name of the actions on the palette when the cursor is moved over it.

The only control available displays the line or turns it off. There are no options for configuring this information. This information, except for the current time, only appears when the program is executing.

Manipulate Flowchart

Structure Actions

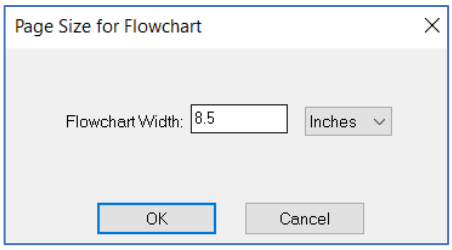
The tools available to align and structure the icons positioned on the flowchart exemplify the graphical nature of the flowchart. When the flowchart is the active window, the Windows menu bar displays the **Structure** option.



- **Group** establishes precedence for bitmaps placed on the flowchart. Align applies to selected items on the flowchart.
- To select multiple actions for alignment purposes, click the first action. Hold down the **Shift** key and click all remaining icons to be aligned. Choose the option desired and the icons will be aligned as selected.

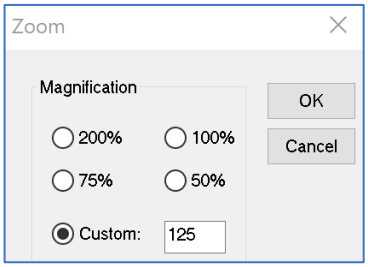
Page Size

The default page size for the flowchart is 8.5 in x 11.0 in. You can make the flowchart large by changing the page size from the Toolbar->Structure->Page Size. The height of the page will automatically scale to the width. When printing the page size will automatically scale to fit on a standard page.



Zoom

You can make the flowchart zoom in and out for easier reading by going to Toolbar->View->Zoom and changing the magnification. The default magnification is 100.



Objects

TILE! is among other things, a collection of objects. Everything you see in the Flowchart and the Profile is what would be called an object; data elements and actions are objects. When you save profiles, the software is storing the entire collection of objects into the profile so that they can be re-loaded and connected later.

Ghost objects

Certain operations related to copy/paste or undo/redo occasionally leave objects in the profile that are no longer needed; these are what we call ghost objects. Ghost objects are not the same as duplicate objects; however, some ghost objects can be created while duplicating an object.

Ghost objects exist in the profile, but since nothing is “referencing” them, they are unnecessary and wasteful in the context of memory use. Another way to describe a ghost object is that it is “invisible” i.e. you cannot see it anywhere on the flowchart. There really is no reason to keep these objects, since they are completely inaccessible.

Ghost objects are more common in older versions of TILE!. Newer versions do a much better job of not creating ghost objects in the first place and they clean up any ghost objects that are loaded into memory during a profile load.

TILE! tries to stay on the conservative side by alerting the user when ghost objects are found and giving the user an option to delete them. However, it is highly recommended that you always delete ghost objects. Once you do that and resave the profile in the new version, the ghost objects should be gone.

It is always good practice to keep a copy of your previous TILE! version profile when loading in a newer version of TILE!. Once you save the profile in the newer version, there is no way to load the updated profile into previous versions of TILE!.

Ancillary Actions

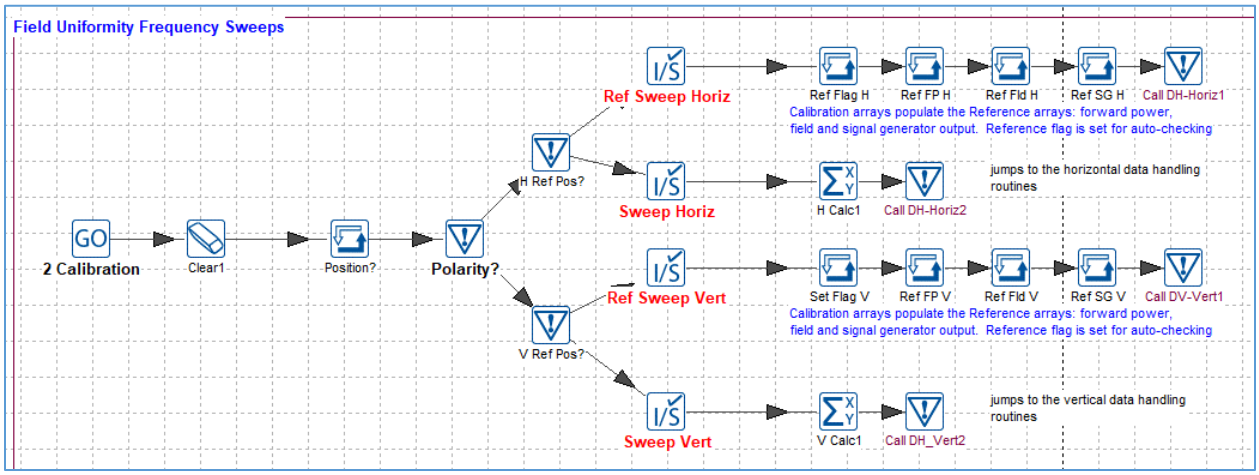
Drawing Tool Actions

The drawing tools are used to draw shapes on the flowchart to add color and improve the appearance of the profile.

Select	Rectangle	Ellipse	Line	Multiline	Polygon	Textbox

Shapes and Colors

- In the palette where the actions are, there are shapes you can place on the flowchart. Once you have your rectangle on the chart, you can select it and use the Graphics menu to change the line color and thickness.
- Select a shape and drop it into the flowchart, the curser is live and you can drag to make the shape you want. If you want to add color or line weight, go to the Graphics area in the main toolbar and select the attributes from there.
- Example: Use color coded text to comment to the user what a framed area and how the connections between actions interact.



IMG example for the shipping profile "IEC 61000-4-3 Field Uniformity Constant Power Method.TIL"

Connector Action

The Connector tool is used to connect one action in the flowchart to the next. Select this tool, then single-click the first action to anchor it as the starting position. Drag the cursor to the next action and single click again to attach the endpoint to the second action. This forms a connecting line between the two actions with an arrow showing the direction of the flow.

You can only attach two actions together in one operation. Therefore, to attach multiple actions, you must attach A to B, then B to C, etc. An action can have multiple incoming attachments, but only one outgoing attachment (except for the Prompt action).

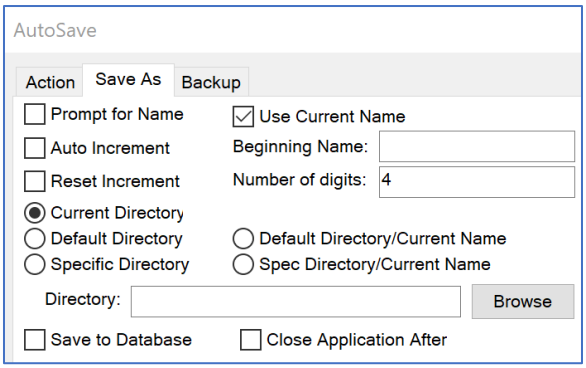
File Management Actions

Auto Save

The Auto Save action automatically saves the current test profile. The options given are extremely useful for historical recording purposes. This option gives the operator the ability to store the entire test profile to be examined later.

Save As Tab

This tab gives the user the options for saving the profile.



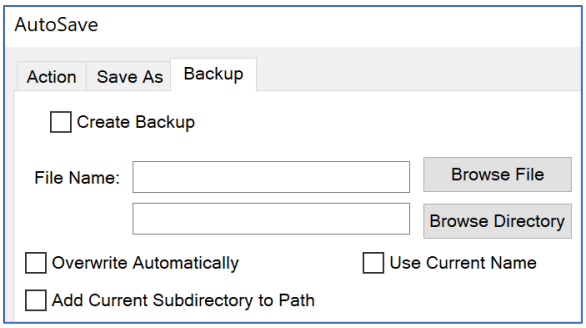
- **Prompt for Name** — When selected, will prompt the operator for the file name. The following Directory Specifications only apply if Prompt for Name is selected:
    - **Current Directory** — The directory in which the TILE! profile was opened. If selected, it will force the user to name the file. You can change the directory during auto save, but the starting directory will be the current directory.
    - **Default Directory** — Uses the TILE! default directory as defined in the File Options settings. The Save As box will show the default directory and it will force the operator to name the file.
    - **Default Directory/Current Name** — If selected, the Save As window will reflect the default directory as defined by the File Options settings, and leave the current name as the default file name in the box.
    - **Specific Directory** — With this option the dialogue for Save As will start in the specified directory as defined by the Directory settings. If this directory is not valid, the operator will receive an error.

- **Specific Directory/Current Name** — If selected, the Save As window will reflect the specified directory as defined by the Directory settings and leave the current name as the default file name in the box. If the specified directory is not valid, the operator will receive an error during execution.
- **Directory** — The user can define a directory for the **Specific Directory** and **Specific Directory/Current Name** options.

- **Use Current Name** — When selected, this setting will automatically save the profile under the current name.
- **Auto Increment /Beginning Name** — select and specify the file naming convention/sequence that will be invoked.
- **Reset Increment** — If this check box is selected, the first save will be reset to zero, and incrementing will begin at zero.
- **Number of digits** — The number of digits for the increment can be set between one and four yielding values of 1 to 9999.

Backup Tab

When activated, this action will create a backup copy of the current profile each time the save is executed.



- **Create Backup** — Check to activate the backup capability.
- **File Name** — Used to specify the location for storing backup files. The box will be filled automatically if the operator browses to find a file. You must add an appropriate file type (DAT, TXT, or CSV for data elements; WMF for bitmaps) if you want this file to be readable by other programs.
- **Browse File/Browse Path** — Displays a dialog box that can be used to search for a desired file. If the OK button is selected the name and path boxes are automatically filled.
- **Overwrite Automatically** — Automatically overwrites a duplicate file if found when saving.

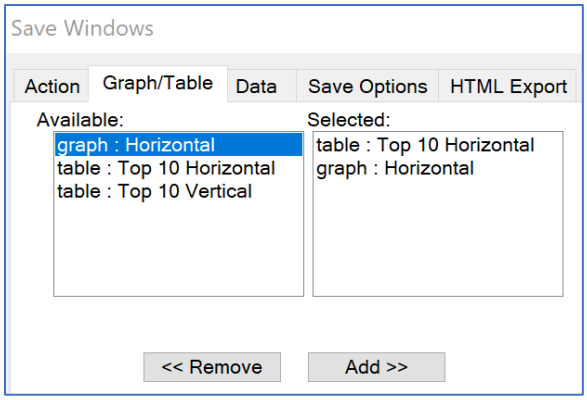
Save Windows

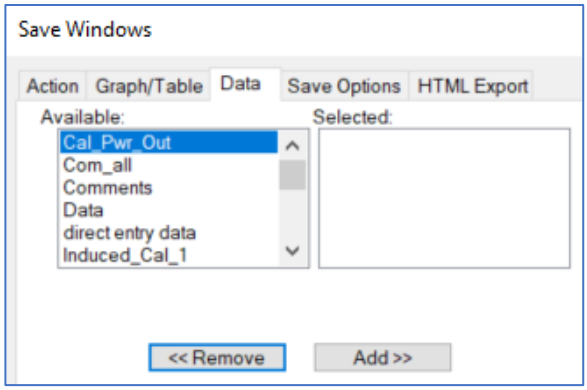
The Save Windows action will save graphs, tables, and individual data elements.

Graph/Table Tab and Data Tab

These tabs each have a column on the right hand side that shows all graphs/tables or data that are available for export.

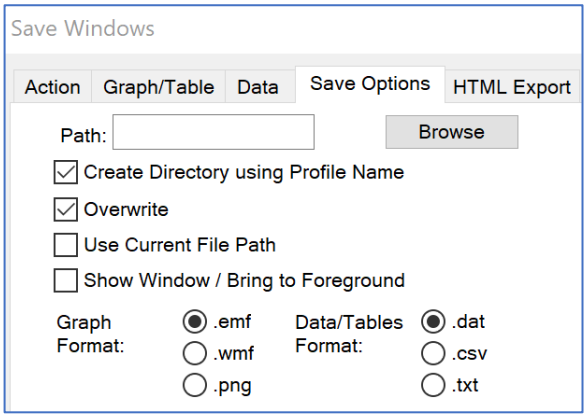
It is divided into two columns, Available and Selected. They can be selected by double-clicking on them or selecting the Add>> button. They can be removed by double-click or the <<Remove button.





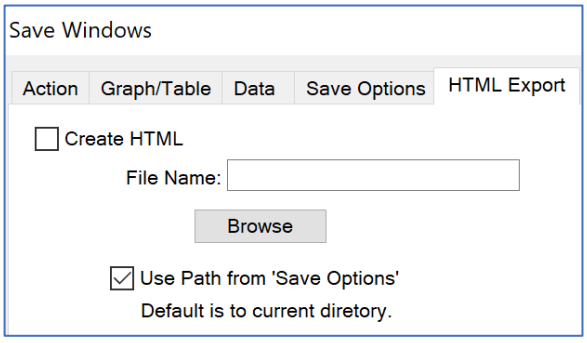
Save Options Tab

This tab allows the selected information to be saved to a selected directory, current directory, or to a new directory created with the name of the current profile name. There is a selection to overwrite any existing data.



HTML Export Tab

This tab will create a file that contains HTML code that can be used to import data into Microsoft Word to generate a report.

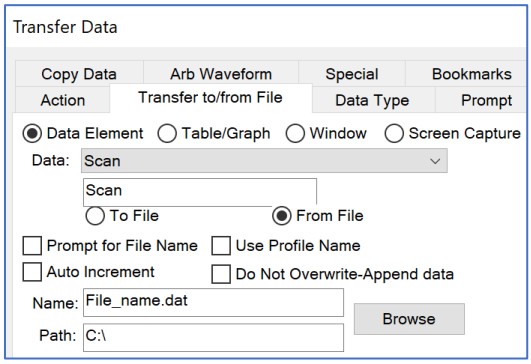


Transfer Data

The Transfer Data action is used to transfer the contents of a data element to or from a disk file.

Transfer to/from File Tab

This tab describes the file to or from which data will be transferred.



NOTE: If location and data type are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **What to transfer**:
    - When Data Element is chosen, the Data list will include the available data elements.
        - **To File/From File** — These option buttons indicate whether the Transfer Data action will save to a file or read from a file. They only appear when Data Element is chosen.

          If From File is selected, the data element defined in the Data window will be filled with the input information. This is the same procedure as defining a data element as a file type and initializing the information, except the data element is already defined (such as an antenna factor file). This procedure is helpful in moving information between different test profiles as well as interacting with Microsoft Excel.

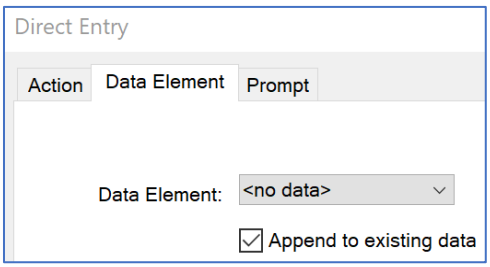
          For example, every morning you run a special cable calibration to ensure that the cables are connected properly and in good condition. The cable calibration information from this test profile is saved to a daily cable file, which is then read into every test profile executed to load the cable correction factors.
    - When Table/Graph is chosen, the Data list will include the available tables or graphs.
    - When Window is chosen, the Data list will include the Data, Instruments, Log, Audit Trail, and flowchart windows.
    - When Screen Capture is chosen, the Data list will include the available instruments.
- **Prompt for File Name** — When selected, this allows the user to browse and select the file of interest during execution. If you enter a file and path on this tab, they become the browse defaults.
- **Use Profile Name** — If this check box is selected, a subdirectory will be created with the name of this profile, and data will be placed into that directory.

Direct Entry

The Direct Entry action gives the operator the ability to declare preset values for a data element.

Data Element Tab

This tab defines which data element will be used to store the results of this action.

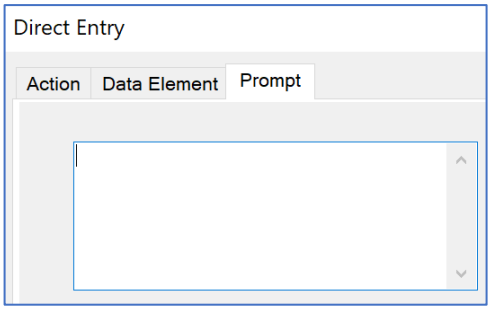


NOTE: If no element is selected and the action is executed, the user can assign a name at that point.

- **Data Element** — Select a data element from the list. Use this to predefine a data element and use it through the flowchart, updating it with this action at the start of a test.
- **Append to Existing Data** — If this check box is selected, the data you enter will be added to the existing data element contents.

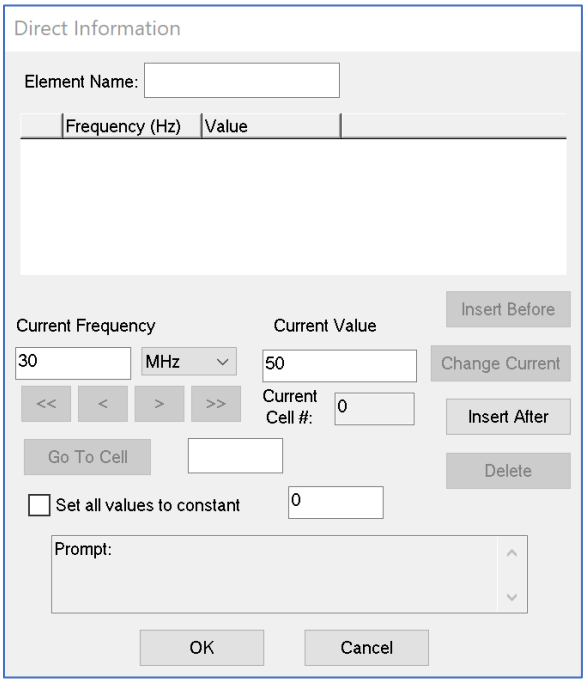
Prompt Tab

A message can be placed here to provide information or instructions to the user.



Dialog Box

This dialog box appears when **Direct Entry** is executed and is the entry point for the frequency and amplitude data.



The frequency and value boxes are the entry points for the respective data. Rows can be selected by clicking on the row number on the left side.

- **Element Name** — The data element that will hold the preset information. If the dialog box is started without an element name chosen, it will be blank. Then the user can assign a name and it will automatically create a preset element with that name.
- Editing the data:
    - **Current Frequency/Value** — Edit the current frequency/value, then select where to insert them into the list:

        - **Insert Before** — Inserts the combination before the current cell.
        - **Change Current** — Inserts the combination at the current cell.
        - **Insert After** — Inserts the combination after the current cell.
    - **Delete** — Deletes the current row from the table.
    - **Set all values to constant** — If selected, this will set all values to the one entered in the box next to the check box.
- Navigating the data:
    - **<< / >> buttons** — Use these buttons to navigate to the very top or bottom the frequency list.
    - **< / > buttons** — Use these buttons to navigate up or down the frequency list.
    - **Go to Cell** — This will navigate to the cell entered here.

Clear Data

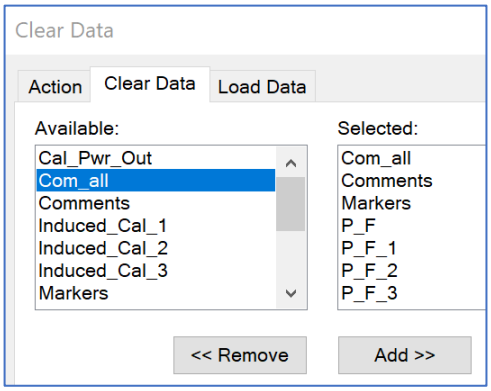
The Clear Data action provides a method of clearing data elements from within the flowchart. This action can be performed between different stages of a test or after a test has been completed. Clear data empties the existing data from a data element and can only clear the data element types measurement, equation, and word.

NOTE: Preset data can never be cleared automatically. From-File data is not refreshed unless the load data information is added. In this case, the data element values are reloaded with the data from disk. If this data is not valid or if the file does not exist, an error is reported and the old data is left in place.

Clear Data Tab

This tab defines the data that will be cleared in this action. Select the data elements that need to be cleared at this point.

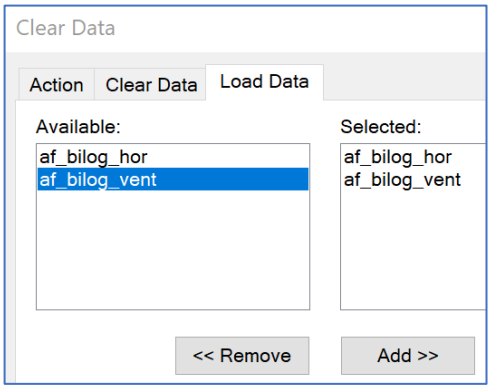
It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.



Load Data Tab

This tab is useful if you always want to load the most current data, such as from an antenna factor file. Select the data elements (only From File types are shown) that need to be reloaded at this point.

It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.



Process Actions

The process actions provide methods to interact with the operator, test equipment, and other programs.

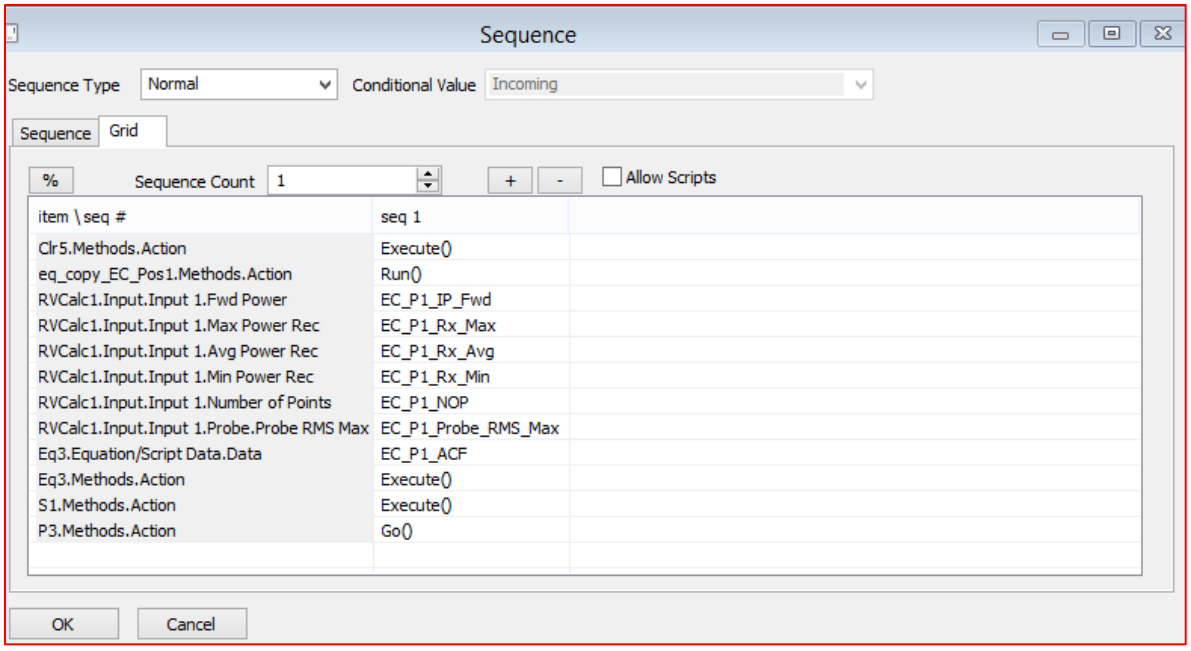
Start (GO)

The Start action provides a logical starting point to a test profile, and therefore is the only action that has no other function than to name it. This action is useful if you want to label a section of the profile that will provide a point to start from when referenced either from the execution arrows, or from an automated prompt action.

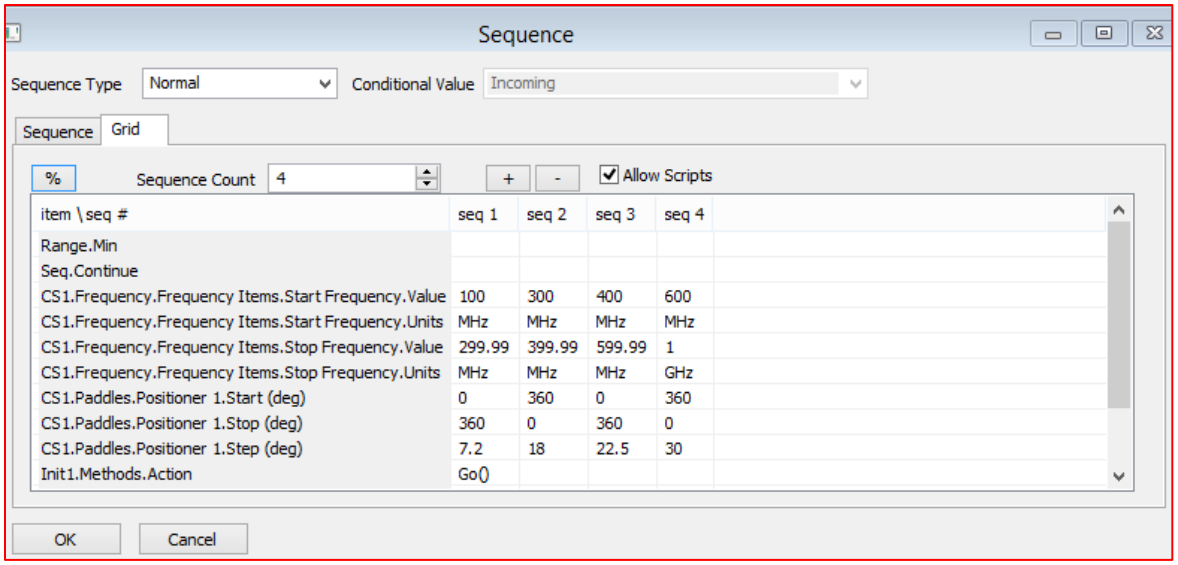
Sequencing

The sequencer can be used to create a table that contains user selectable attributes. These attributes can be sent to any action in a flowchart as well as change values of data elements if needed.

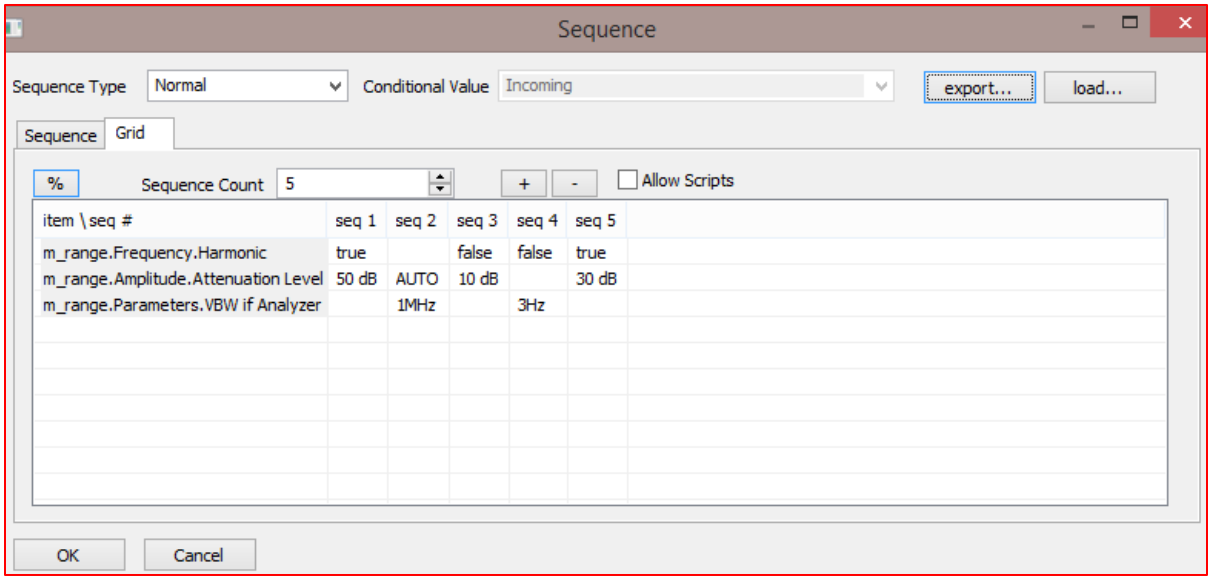
The example below will execute actions in a preferred sequence that are not connected by the flowchart connector, and then it will dynamically insert data elements into an action so that when data is collected it will be recorded in those data elements. When another sequence runs, it will insert the data elements for that measurement. This allows one action to be in a flowchart that will measure multiple facets of the EUT and record the data as needed for each facet or position.



The sequence shown below illustrates how the sequencer can handle frequency bands in a sequential order:



The sequencer displays an "export" button used for saving a sequence to a "seq" file:



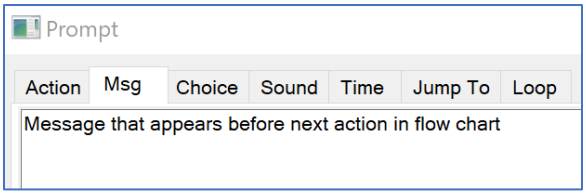
Prompt

The Prompt action displays a message box on the screen and waits for a response from the user before executing the next action in the flowchart. This action is also the only action that will support dual outputs. One option is to specify an accept/reject path for execution.

An optional sound prompt can be played as part of this action. The sound prompt plays a standard WAV file, which requires that the computer be installed with a sound card that is compatible with the Windows operating system.

Message Tab

This tab allows the user to enter a message that will be displayed in the message box during the execution of this action.

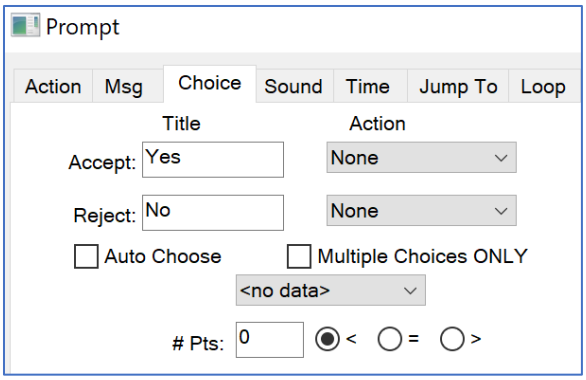


NOTE: The action displays the message without text wrapping. To wrap the text displayed during execution, use CTRL-ENTER where you want the text to break.

When using the Prompt action, we recommend that you put multiple paragraphs of information instead of a single large paragraph. The visual impact of long paragraphs of information can be confusing to the operator. For long messages, please format the message by adding embedded a carriage returns.

Formatting of strings is left to the Profile Writer. When the amount of data exceeds the simple view, you will automatically scroll to add more. This has a very clumsy appearance when it is executing. We recommend that you put long comments into separate Prompt actions so that the amount of information can be easily digested as the operator executes the test.

Choice Tab



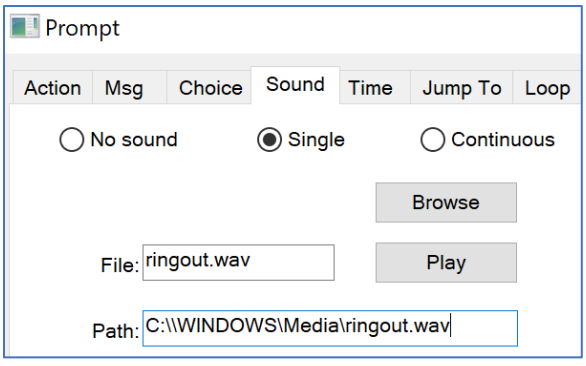
The Prompt action is the only action that allows multiple outward directions to exist; all other actions execute in a serial fashion. This action works like an if/else or switch case function to direct the flow to the next action in the sequence to execute.

- **Accept/Reject** — The first choice on this tab reflects the attachment of actions out of this action. The first arrow is automatically determined to be the accept path. The second arrow is the reject path.
    - **Title** — You can control the title displayed on the dialog box for the Accept and Reject buttons. The default titles are **Yes** and **No**. It is suggested to name each path after the action to be selected to avoid confusion while running.
    - **Action** — The actions displayed here reflect the sequence in which the arrows were attached to the two actions. You can reverse them by dropping down the selection box and choosing the opposite action. You must change both or the change will not be accepted.
- **Auto Choose** — Automatically determines the direction that this action will take based upon a greater than, less than, or equal relationship. Once these are identified, you can display, on the prompt, a data element and the number of data elements that are valid for that data element. If you select Auto Choose, then the action will automatically choose the path to take based upon the appropriate mathematical relationship indicated.

  A common use, for instance, would be to specify that you want to perform a peaks/QP measurement on the specified data element if there are less than 11 data elements. If there are more data elements, the unit has too many failures to justify the additional measurement time.
- **Multiple Choices ONLY** — This check box is to be selected if the **only** options you want available in the prompt are those in the **jump to** list. Be aware that the original arrows, or original options, will not be available to you in the prompt if this check box is selected.

  NOTE: When this checkbox is selected, the message text displayed during execution of the action is not wrapped, so some of the text can be cut off.

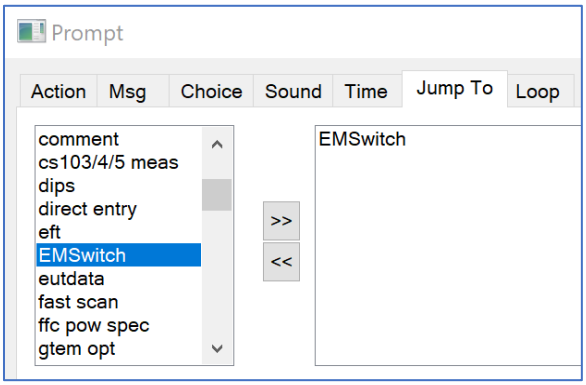
Sound Tab



An optional sound prompt can be included to play a .wav file. This option requires that the computer be installed with a sound card that is compatible with the Windows operating system.

- **No sound** — The default is No sound which will disable this option.
- **Single / Continuous** — The two options for sound are to play a sound file once or play it until interrupted. The sound prompt is played during the execution of the Prompt action in the test profile.

Jump To Tab

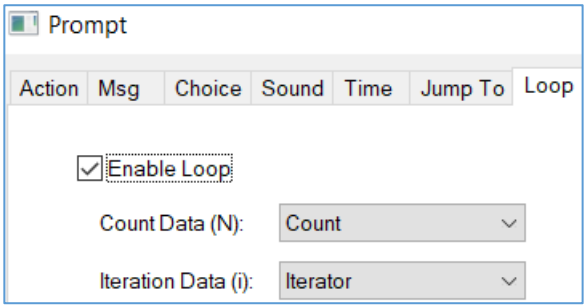


This option allows you to choose multiple continuation points, providing more freedom than the two arrows allotted as connection points from the Prompt icon. You can specify a list of actions to jump to and execute from this prompt.

It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

If a Jump To list is defined, when the prompt executes, an option to jump to an alternate list of actions will be available. However, if the **Multiple Choices ONLY** check box on the **Choices** tab is selected, the prompt will skip the original accept/reject choices and send the user straight to the Jump To options. The original accept/reject choices will not be shown, so only choose this option if you wish to forego the original options.

Loop Tab

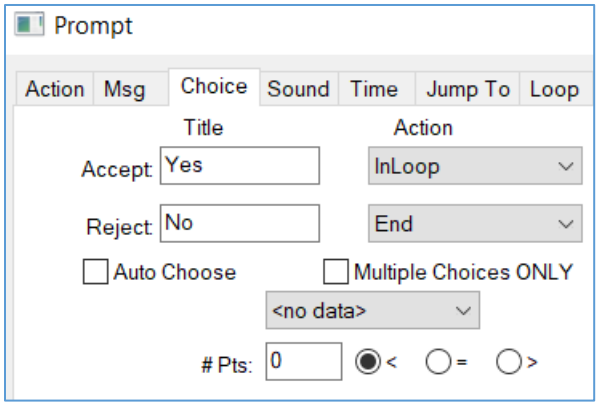


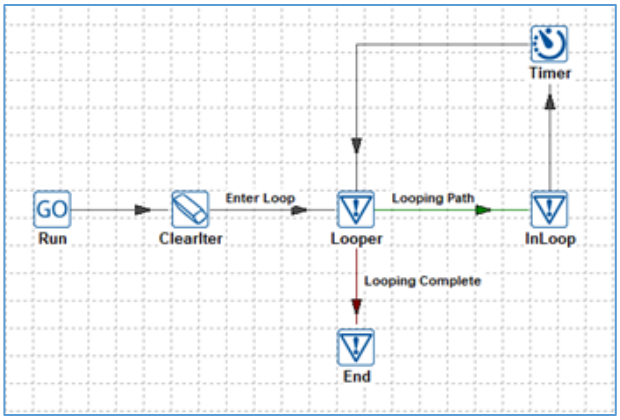
This option allows you to iterate through the 1st path on a set number of loop iterations before continuing to the 2nd path. Use the Connector Tool to link the Prompt action to the first action that will run while the loop is in progress, and then connect it to a second action that will run when the loop iterations have completed**.**

NOTE: The **Enable Loop** check box must be selected to activate this feature.

- **Count Data** — Use this data element to predefine the number of loop iterations that should run. This is an input data element that should be configured with the Preset data type. Set the Amplitude Value setting on the Simple tab to the desired Count value and leave the # of Steps set to 1. The frequency values are ignored.
- **Iteration Data** — Use this data element to keep track of the current loop iteration number. This is an output data element that the Prompt action modifies each time an iteration has completed. Note: This output is optional. The action will still run properly if it is left blank.

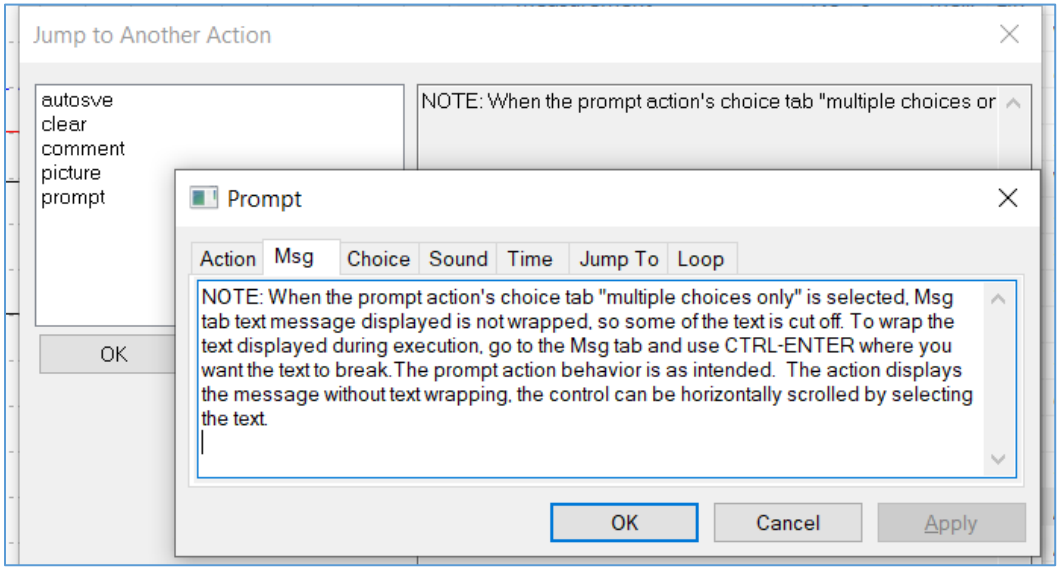
Example: On the **Prompts Choice tab** shown below, the Accept action shows the In Loop path and the Reject action shows the path once the loop iterations have completed. If the Count Data data element has an Amplitude Value of 5, the Looper action will send the program flow to the Looping Path 5 times. After 5 loops have completed, the Looper action will switch the program flow to the Looping Complete path.





Dialog Box

NOTE: The action displays the message without text wrapping. The dialog message can be horizontally scrolled by selecting the text and using the arrow keys.



Math

During the flow of a profile, there are times where equations need to be executed. To execute equations in order, they can be dropped into a Math action and it will run at the appropriate time. Once you select a data element to use the Math action on, you can see the results in the data-element view-window.

For example, if you use the data element equation Induced_Cal_1 in the Math action, it will execute and add the valid data points that were calculated to the data elements view window as shown below:

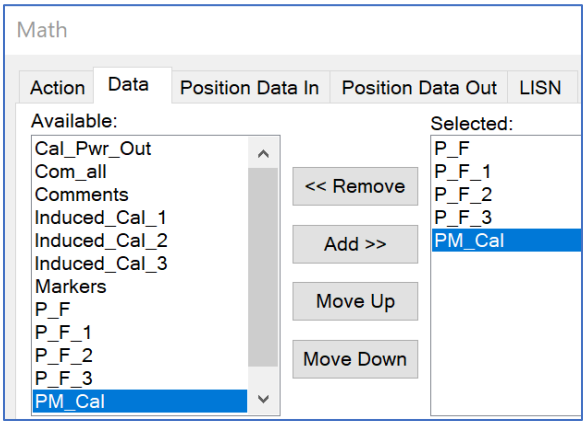
Induced_Cal_1	No	50	Equ	Lin	PM_Cal_FB_1+73	73=107-34 Where 107=0dBm and 34 is 20*log(50 ohms) converting dBm to dBuA
Induced_Cal_2	No	37	Equ	Lin	PM_Cal_FB_2+73	
Induced_Cal_3	No	21	Equ	Lin	PM_Cal_FB_3+73	

Data Tab

The Data tab controls the data elements that will be executed in this step. It is divided into two columns, Available and Selected.

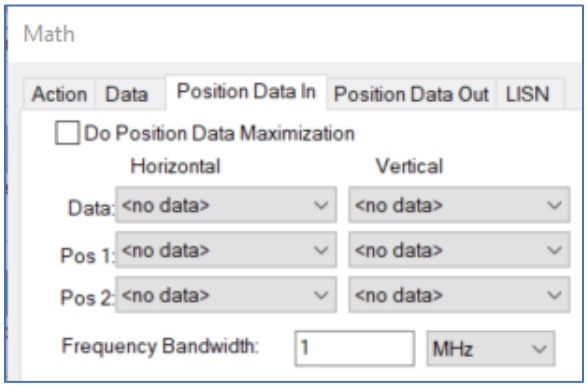
They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

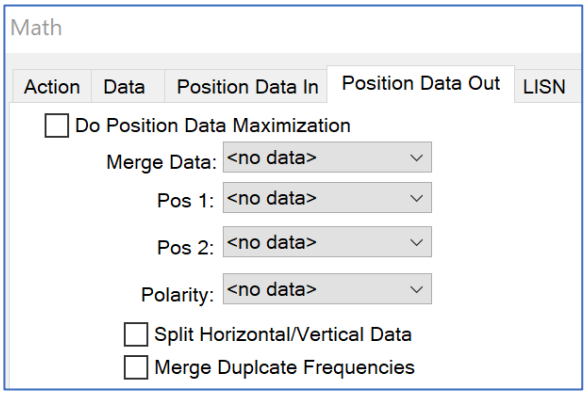
Equations are executed in the sequence they appear in the selected column. If the result of one math equation is part of the equation in a second equation, make sure to execute them in sequence.



Position Data In/Out Tabs

The Position Data In/Out tabs of the Math action take the input tower and turntable positions for both horizontal and vertical, compares the horizontal and vertical amplitudes, determines the higher value, and saves this amplitude plus the related height, angle, and polarity.



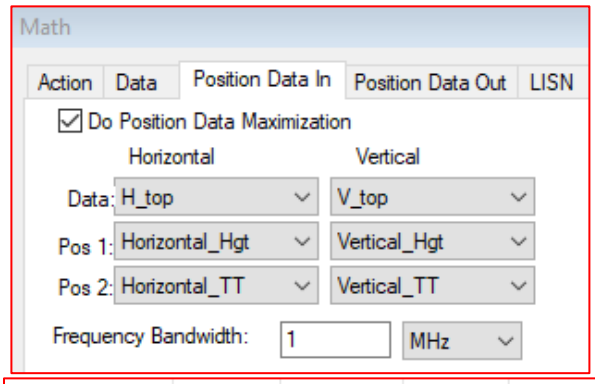


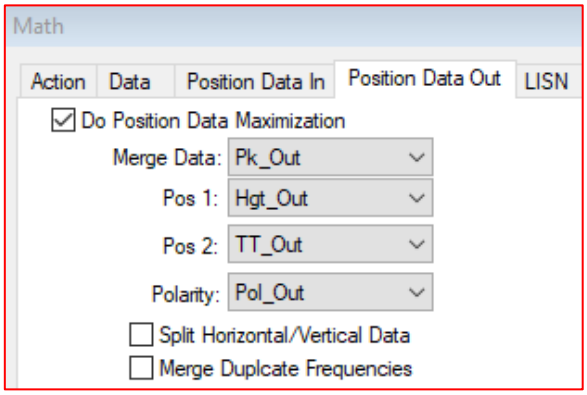
When comparing horizontal and vertical data from an emissions scan, determining the peak signal and orientation becomes complex. Not only do we have to compare the horizontal and vertical for maximum value, it is then necessary to match the polarity, angle of the turntable, and height of the tower. This cannot be done with a simple math function. The tower/turntable in/out tabs allow the user to specify both the horizontal and vertical angle and position information and when selected, the output is the peak signal between the horizontal and vertical, and its related polarity angle, and height.

- **Do Position Data Maximization** — This optimization process is performed when the check box is selected.
- **Split Horizontal/ Vertical Data** — When the check box is selected, the tower/turntable reverses nature and the merged data elements are split based upon their **H** or **V** designation in the Polarity data element.
- **Polarity** — The Polarity data element must be a word type and only this type will be identified in the list.

The data below is a sample of peak type data from each polarity that is compared to find the highest of a similar peak and output the associated polarity (H/V). All of this data should be discreet type (no interpolation).

- **Data** — Peaks that you want to compare.
- **Pos 1** — Either the antenna height or turntable, as long as they match.
- **Pos 2** — Either the antenna height or turntable, as long as they match.

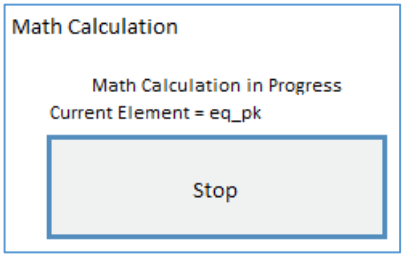




Frequency (MHz)	Pk_Out	H_top	V_top	Hgt_Out	TT_Out	Pol_Out	Hgt_Out	TT_Out
65.987 MHz	51.313	51.313	50.000	100.000	180.000	H	100.000	180.000
109.977 MHz	52.492	52.492	52.492	250.000	180.000	V	250.000	180.000
153.966 MHz	48.264	48.264	48.000	250.000	180.000	H	250.000	180.000
198.004 MHz	45.262	45.262	45.262	250.000	180.000	V	250.000	180.000
418.000 MHz	47.053	47.053	47.000	100.000	270.000	H	100.000	270.000
594.006 MHz	47.135	47.135	47.135	250.000	180.000	V	250.000	180.000

Dialog Box

This dialog box appears when Math is executed, and it provides visual information to the user as well as the ability to halt execution of the test.



Print Pages

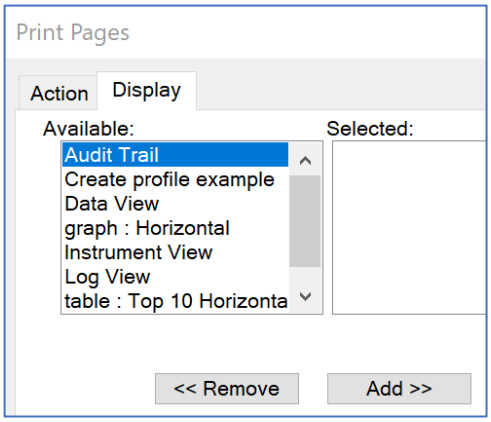
The Print action allows the user to print any window in a profile. This is normally used to print graphs and tables, but you can also print the Instrument, Data, Flowchart, Log, and Audit Trail windows. Placing this icon in a flowchart will result in the printing of the selected items at a specific point in the test flow.

NOTE: You must have a default printer defined or execution of this step will cause a catastrophic failure.

Display Tab

The Display tab defines the graphs, tables, or other windows that will be printed in this action. It is divided into two columns, Available and Selected.

They can be selected by double-clicking on them or selecting the Add>> button. They can be removed by double-click or the <<Remove button.



Launch Application

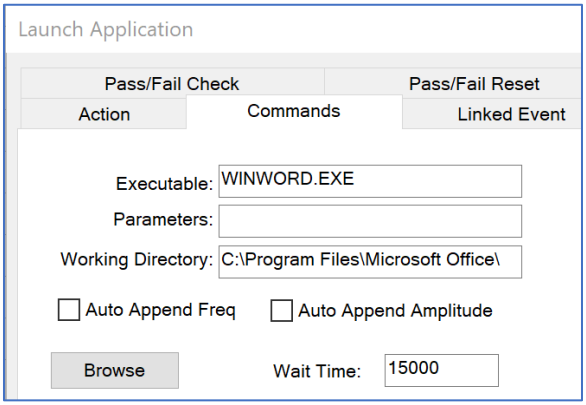
The Launch Application action allows the user to start another application from within the TILE! test profile. This program will run in an independent window. Certain spreadsheet functions or using a word processor for note taking can be accomplished using this action.

You can also launch an application and automatically run a macro. The macro could then transfer information from TILE! into the other application.

Commands Tab

This tab includes Executable, Parameters, and Working Directory; only the first is mandatory. The format for these is the same as the standard OLE launch commands.

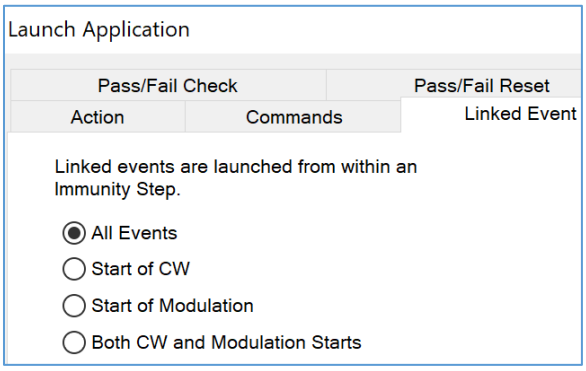
The command line to launch an application is composed of the executable, a space, and the parameter switches (a normal format with a forward slash and switch). The frequency and amplitude selected are appended to the command line as additional switches.



- **Executable** — Type the program executable, including the path name. This needs to include proper disk references, such as C:\msoffice\excel\excel.exe. This indicates the drive location, directory, and executable. Also, you can select the file by using the **Browse** button
- **Parameters** — Certain programs allow starting parameters to be added to the executable line. These parameters can start specific files, set starting switches, or override default switches. The user should read the manual for any program to be launched for the command line command structure.
- **Working Directory** — Type the working directory for the default setting. If this is blank, the TILE! default directory will be used.
- **Auto Append Freq/ Auto Append Amplitude** — If this is selected, the current frequency or amplitude is appended to the MS-DOS® command line that launches with this action.

Linked Event Tab

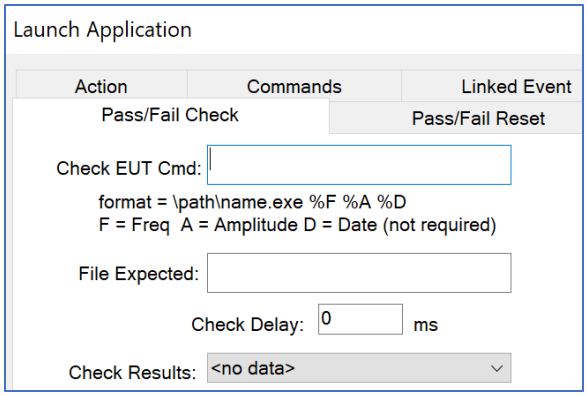
When this action is specified as a pass/fail call in immunity and the immunity achieves a level at each frequency, it will trigger this action. See *Immunity Test* for more information on what events exist. In most cases, you will want to link to either CW or modulation events.

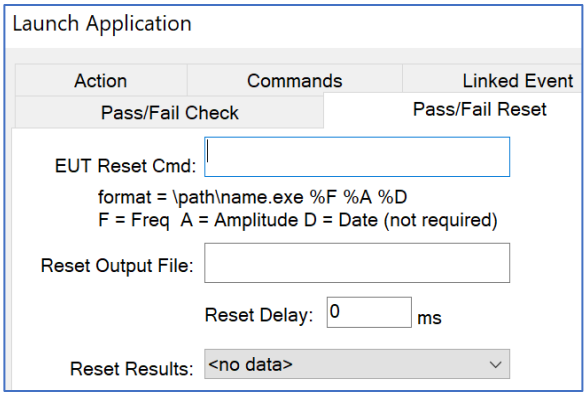


- **All Events** — Click this option for the timer to execute at all events during the Immunity test.
- **Start of CW** — Click this option for the timer to execute only at the start of the CW (after the level has been achieved).
- **Start of Modulation** — Click this option for the timer to execute only at the start of the modulation cycle (after the level has been achieved).
- **Both CW and Modulation Starts** — Click this option for the timer to execute at both CW and modulation events.

Pass/Fail Check and Reset Tabs

These parameters were created to handle the circumstances in which a customer had his own program monitoring EUT failure. It was necessary to create a simple handshaking routine to hand failure statues back and forth. If you wish to explore these possibilities, please contact *ETS-Lindgren*.





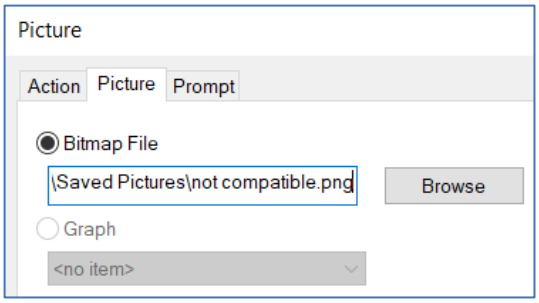
Picture

The Picture action allows the display of a bitmap picture that relates to the test process or setup. Once you have inspected the picture, you can continue or quit.

Picture Tab

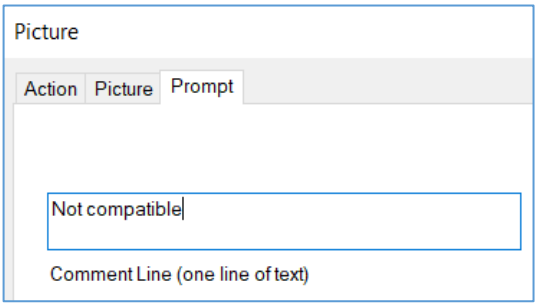
This tab allows selection of the bitmap to be displayed. You can choose PNG, JPG, or BMP picture types.

NOTE: If an image is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

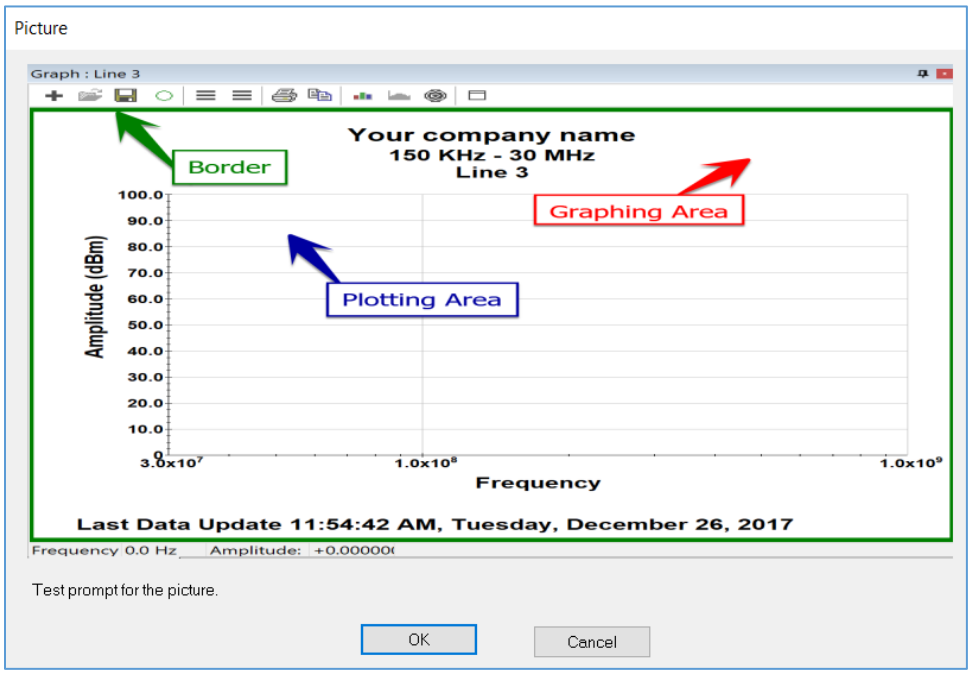


Prompt Tab

The Prompt tab allows for the entry of text to record a comment specific to this step.



Dialog Box

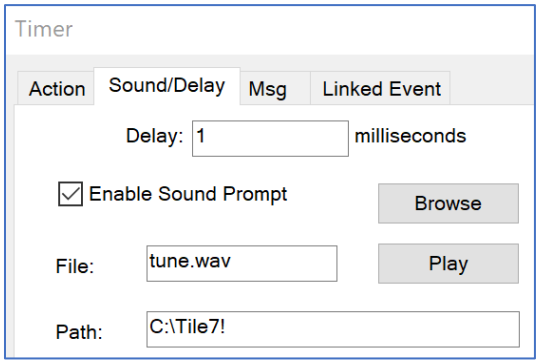


Timer

The Timer action performs a user defined dwell (pause). All action is held for a specified period and then the execution continues.

Sound/Delay Tab

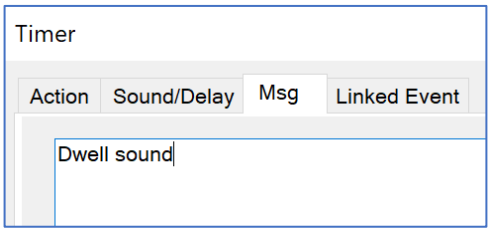
This tab defines a delay and appropriate message to be displayed during the dwell. Sound can be turned on during the dwell.



- **Delay** — Enter the time in milliseconds; for example, 3000 would represent 3 seconds of dwell.
- **Enable Sound Prompt** — Sound will be actuated during the dwell when the check box is selected.
- **File** — The sound waveform file.
- **Path** — Type the path for the file. This will be identified automatically if **Browse** is used.

Msg Tab

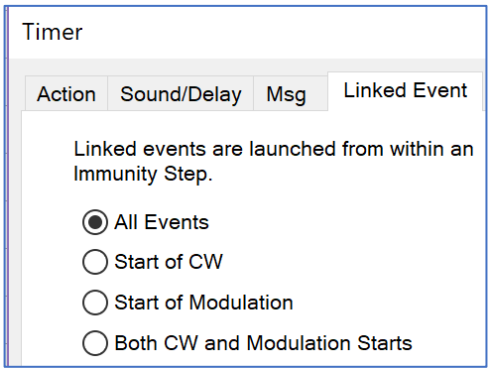
This prompt message will be displayed during the dwell. It is a convenient method of identifying the purpose of the dwell.



Linked Event Tab

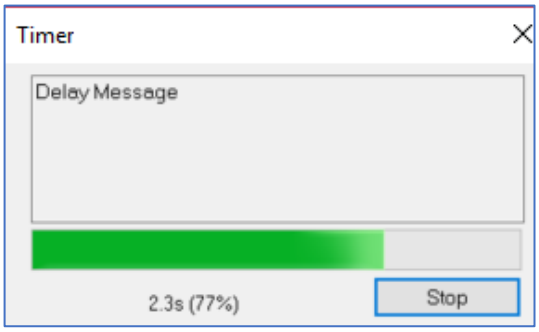
Linked events determine when the timer will be executed after the timer action is called from the *Pass/Fail tab* of the Immunity action.

This tab is similar to other linked event tabs and will not fully be described here. See the *Launch Application Linked Event Tab* for the full description.



Dialog Box

The dialog box displays the message from the **Msg** tab and will delay the length set on the **sound/delay** tab.



Information Actions

The Information actions allow you to attach information to the test file for reference. This information can be transferred to the database when using the TILE! database option. This option is available when the database utility is available.

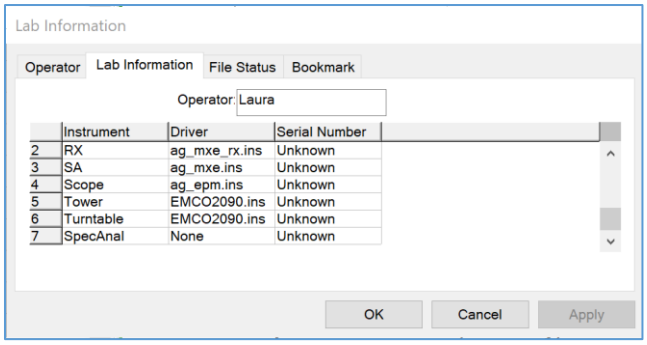
Some of the information will be available to be displayed on the graphs and tables, while some of the information will only be used for documenting the attributes of the profile for reference. This is useful if you want to document the client support engineer attending the testing or other specifics.

Lab Information

The Lab Information action is used to provide two different items to the profile. The first is the name of the engineer or technician performing the test. The other is a matrix of the instruments used during the test.

Laboratory Information Tab

The Operator box permits entry of a single line of text, up to 128 characters. It can be displayed on graphs/tables using the Option/Additional Information features.



The display matrix in the box allows the operator to verify, and change, the serial number and calibration date for the various instruments defined in the profile.

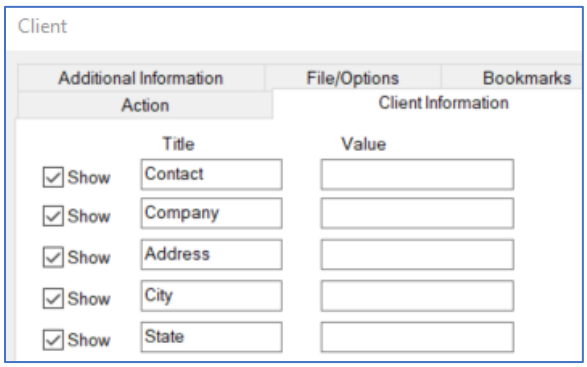
This information is not mandatory and can be skipped by the operator, but it does provide a convenient methodology for verifying that the correct instrument has been used, and to allow storage of the correct calibration date for each instrument in the stack.

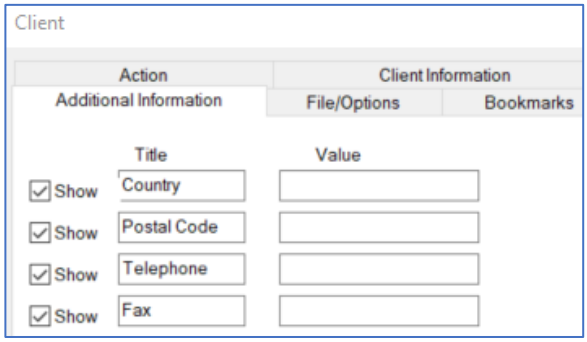
Client Information

The Client Information action is used to record specific information concerning a client. There are nine generic fields with related titles. The user can modify these during editing to match any requirements for labels. When executing, only the value field is changeable.

Client Information/Additional Information Tabs

Contact and **Company** are the only items that appear on tables and graphs even when using the Additional Information options. All other fields are for reference purposes only.



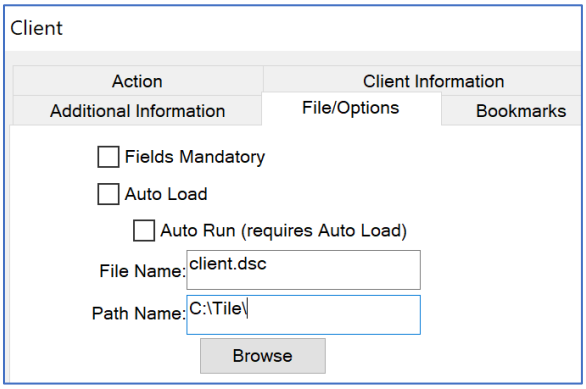


File/Options Tab

When working with different profiles, it is sometimes necessary to re-enter client information into an action. The File option allows you to extract that information from another profile and insert it into this one instead of having to retype it.

To save client information from a profile, you must choose the Save to File option while the Client action is executing. To recall the data in a new profile, use the client/file tab or execute the Client action and choose the Load from File option.

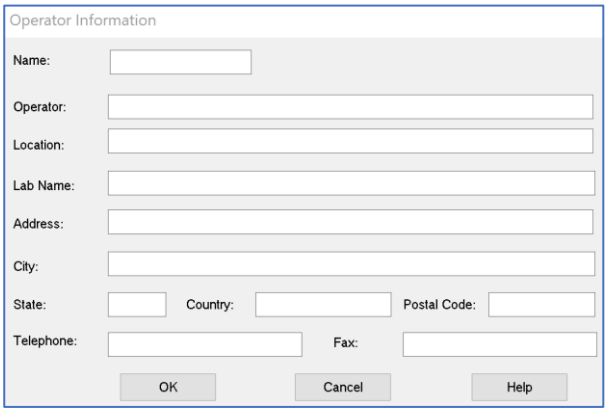
The File tab is used to specify the name of a .wav file.



- **File Name** — The Browse button is available for this step.
- **Path Name** — Contains the path to the .wav file. If the .wav file is in the default directory, no entry is required. This box is filled automatically when the Browse button is used.
- **Browse** — Opens the standard File Open dialog box that is used to select the .wav file that will be used when this action executes. When a file is selected, the file and path boxes are automatically updated.

Operator Information

This is an executable only action that is used to enter information about the laboratory performing the test. This information is not used in the graphs or tables and is only seen when the profile is running.

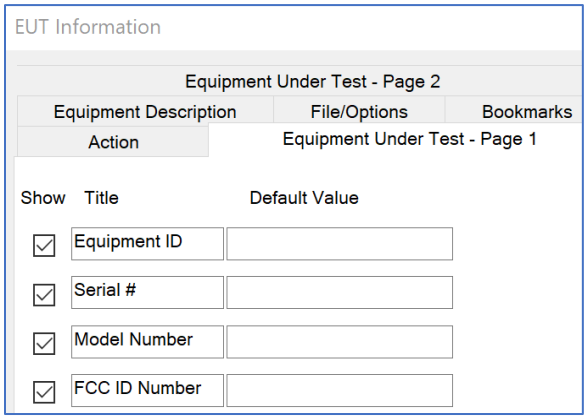


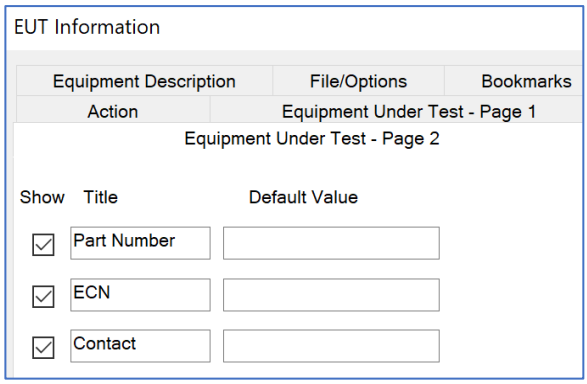
EUT Information

The EUT Information action provides a method to record information unique to the Equipment Under Test (EUT), sometimes referred to as the UUT (Unit Under Test) or DUT (Device Under Test). The EUT action allows all information and titles to be displayed on graphs; see *EUT Tab* for more information about Floating EUT.

Equipment Under Test Tabs — Page 1 and Page 2

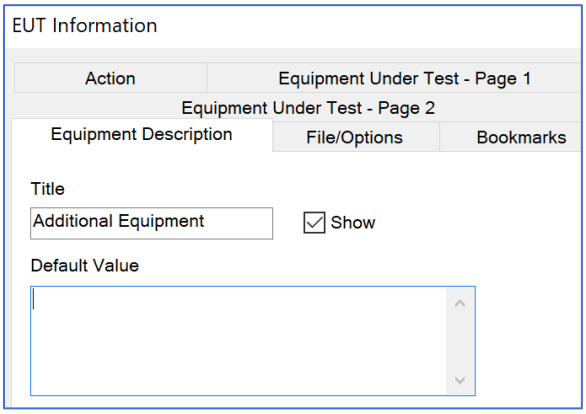
There are seven fields that can be used to describe the EUT. The titles can be modified by the user when editing the action. Only the values can be changed when executing the action. The field **Equipment ID** can be displayed on graphs and tables using the Setup Options/Additional Information features.





Equipment Description Tab

- **Title** — Provides up to 256 characters of descriptive text. This is commonly used to illustrate setups for the EUT or any peripheral equipment that was present during the test. This is helpful when re testing equipment.
- **Show** — Allows the user to turn off display of certain fields during execution. If you decide you only want the first field, deselecting the Show check box for the remaining fields will simplify the entry requirements for the operator during execution.



File/Options Tab

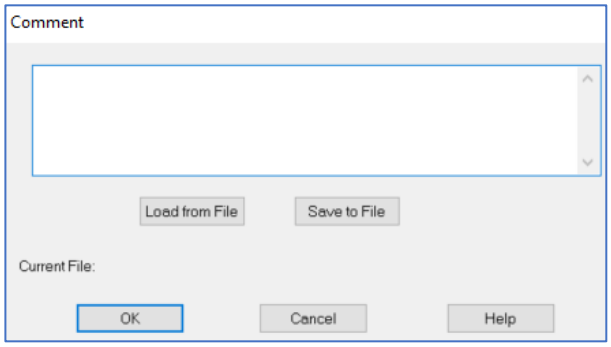
Just as in the Client action, this option allows for importation of information from other profiles.

This tab is similar to other file options tabs and will not fully be described here. See the *Client File/Options Tab* for the full description.

Comment

The Comment action allows the operator to insert comments that relate to the test process. This action can be used liberally where operator input on various stages of the test process is desired. Up to three comment fields can be displayed on the graph.

The Comment box provides space to record comments specific to this profile or step. You can define multiple comments within the same profile; up to three can be displayed on an individual graph or table. Generally, you are limited to 234 characters. We strongly recommend that you enter separate lines of text to ensure a consistent view when placed on a graph. Text will automatically wrap down the screen.



Weather

This action allows the recording of weather conditions when connected to a station that can communicate with TILE!. Contact *ETS-Lindgren* for help with this action.

Interface Actions

The Monitor Interface and Serial Interface actions are not currently documented; ask *ETS-Lindgren* for help with these actions.

Measurement and Control Actions

Control Actions

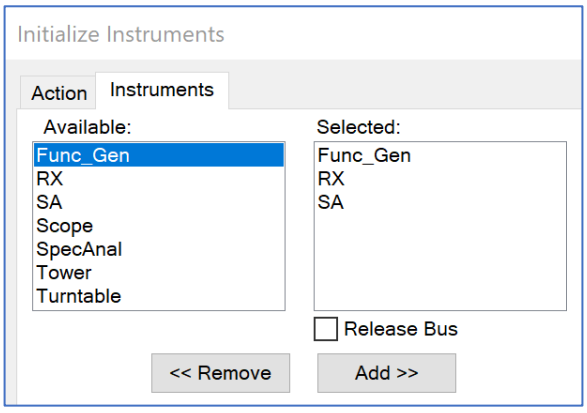
Instrument Initialization (INIT)

Instrument initialization provides a method of sending instrument reset/preset commands to designated instrumentation. This is especially important at the start of long sequences of different types of tests. It is important to ensure the instruments are in a defined starting setup since there is no automatic initialization in any specific instrument commands.

Instruments Tab

It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

The first column shows the instruments defined in this test profile. The second column displays those instruments that need to be initialized at this step.

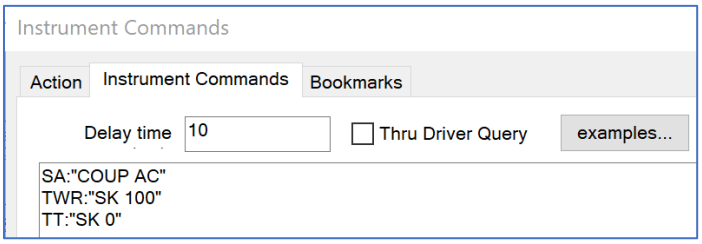


Instrument Commands (VISA)

The Instrument Commands (VISA) action allows the user to send a command, or sequence of commands, to any VISA or GPIB controlled instrument.

Instrument Commands Tab

You need to **define** the **instrument** in the Instrument window **before** you can address it with the Instrument command. If the instrument does not have an instrument driver under the TILE! system, define the instrument with **Do Not Use Driver** set on its **Driver** tab.



- **Delay Time** — You can sequence multiple commands with this action. The delay time is a delay between execution of each command in the sequence.
- **Command box** — The commands in this action are sent directly to the bus address specified in the instrument definition (see *Instrument Control* for more information).

Command Format

The format for a direct command is the instrument name followed by a colon (:), and then the command in a set of quotes (" "). You can embed quotes within the command but must have a set of quotes surrounding the complete string. You can sequence any number of commands to more than one instrument in the same action, but do not send commands that require an answer; this command is not for two way communications.

- **Variable Delay Timing** — You can specify a specific delay time at any point by entering a command using the term delay, Delay, or DELAY as the instrument name and the number of milliseconds as the command.

  For example, this would delay for 2000 milliseconds between the two instrument commands:

  analyzer:"command"

  delay:"2000"

  analyzer:"command"

- **Query Commands** — Commands that end with the '?' character will automatically be followed by a Read (a Query is a Write followed by a Read). The returned string from the Read can be viewed in the TILE! Log View. Query commands can also be specified explicitly (even when no '?' is present in the command) using the syntax:

  InstName:query:"command"

- **Read Commands** — A Read function can be explicitly specified using the following format; the quotes are necessary, but the contents inside the quotes are ignored:

  InstName:read:""

- **Hex/Binary Commands** — Commands can be specified using hex or binary format using the syntax; where the command string is interpreted as hex format, with all spaces removed. Two hex characters equals one byte of data:

  InstName:hex:"11 22 33"

  The following example commands will all perform the same function:

InstName:”Hello\n”

InstName:hex:”48 65 6C 6C 6F 0A”

InstName:hex:”48656C6C6F0A”

- **Remarks** — Lines that begin with the characters ‘rem’ (case insensitive) are ignored:

  REM InstName:”command” //this line will be ignored

- **Termination Characters** — The following termination characters can be specified at the end of a standard command:

  \r = carriage return (hex 0D), also called CR

  \n = line feed (hex 0A), also called LF

  \r\n = CR + LF

  Examples:

  InstName:”command\r” //sends ‘command’ + CR

  InstName:”command\n” //sends ‘command’ + LF

  InstName:”command\r\n” //sends ‘command’ + CR + LF

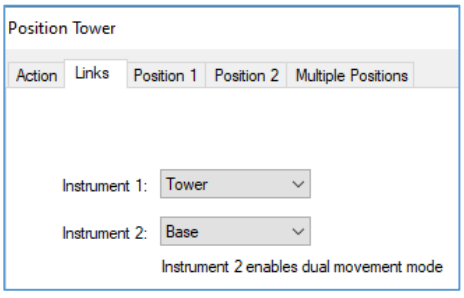
Position Tower

The Position Tower action provides a control interface to move the tower to various positions as well as change the polarization of the antenna. The action can be used for any linear device, such as a two-axis X-Y positioner. If the device supports Seek targeting, the action supports simultaneous movement on two axes when both Instruments are configured.

This is a standalone action that will normally be used for single position movements between different scans or measurements. For tower movement within a Measurement action, see *Scan Range Measurement* or *Scan Peaks*.

Links Tab

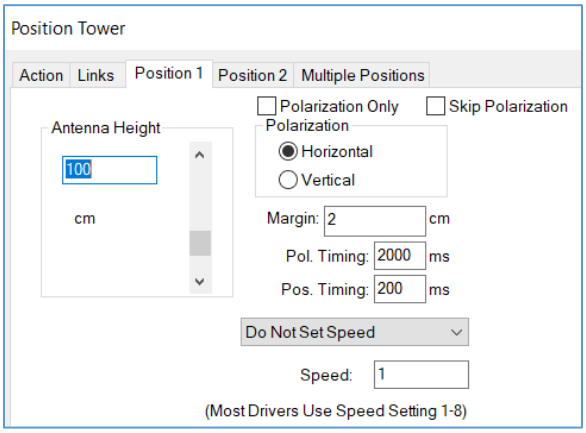
This tab selects the instrument to be used for the action.



NOTE: If the Tower instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Instrument 1** — For single axis movement, click the Instrument 1 dropdown control to display the available instruments, and then select the tower instrument.
- **Instrument 2** — To enable dual movement mode, enable Instrument 1 and Instrument 2. Both device drivers must support “Seek” capability to use this feature. If Instrument 2 is selected, Instrument 1 must also be selected.

Position Tab

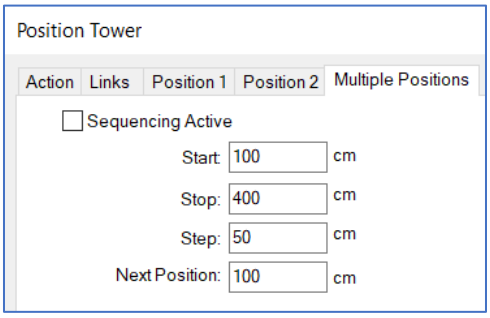


- **Antenna Height** — You can change the antenna height by entering the desired number in the box or by using the slider to move the relative position up or down.
- **Polarization Only** — When the check box is selected, execution of this action will only cause a polarity change on the tower. No movement will be performed.**Polarization** — To move the antenna orientation, select either horizontal or vertical. If the antenna tower is already in this position, no change will take place.
- **Margin** — Depending on the instrument and driver capabilities, the margin setting can be used for two purposes. For newer drivers with "Seek" capability, the margin determines the allowed tolerance between the current position and the target position whether a movement command is sent to the instrument.

  For example, if the current position is 100 cm, the target position is 105 cm, and the margin setting is 5 cm, no movement command will be sent. To ensure that a movement command will always be sent (unless the current and target positions are equal), set the margin to 0.

  For older drivers without "Seek" capability, the margin setting defines when the positioner will be commanded to stop before the actual stop position. When movement reaches the desired position less the margin, the stop command will be sent to the tower. If the margin is set properly, the tower will settle on the desired position. This setting will be site- and test-specific because of the differences between positioners and the equipment placed upon them.
- **Pol. Timing** — Most towers with polarity under remote control use an air control to change the polarity. The amount of time it takes for the antenna to rotate from horizontal to vertical, or vice versa, depends on the weight of the antenna and the air pressure. We want to ensure that the polarity has completed movement prior to any other movement or measurement. Polarization timing in milliseconds reflects how long it takes the positioner to execute a polarity movement. The software will pause this period before continuing its execution.
- **Speed Options**:
  - **Do Not Set Speed** (default) — sets the position, but not the speed
  - **Set Speed and Position** — sets both position and speed
  - **Set Speed Only** — does not set the position, but does set the speed
- **Speed** — Sets the Speed value for the device. Many devices, including ETS_EMGen and EMCO 2090, use a scale of 1-8 to set the speed. Some devices are based on a percentage scale and use a value from 1 to 100.

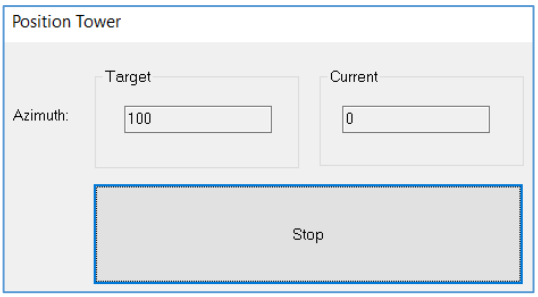
Multiple Positions Tab



- **Sequencing Active** — The Tower and Turntable actions allow the user to place two links out of this action. This is not the normal characteristic of a TILE! action. The purpose of this is to allow the user to define a sequence, with a tower or turntable step between, which will continue to sequence until the positioner has reached one of the limits. Select this check box to make this active and then set the start, stop, and step positions.
- **Start/ Stop/ Step/ Next Position** — At the beginning and end of a sequencing step, the positioner will move to the start position and perform the sequence. It will move one step-position at a time, checking to make sure we are not at or beyond the stop position. It will then perform another sequence, step, and check again. Once the stop position is reached, it will exit to the second arrow out of the action.

Dialog Box

Shows the target position and the current position of the tower. Use the stop button to end the test.

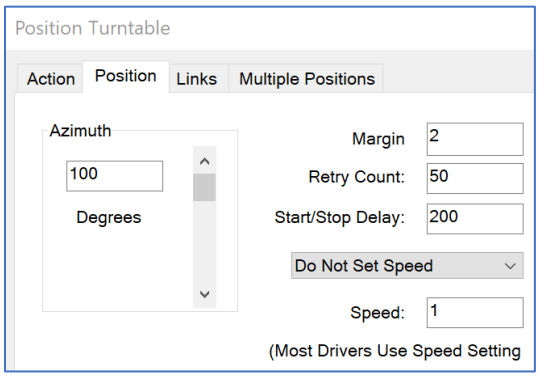


Position Turntable

The Position Turntable action provides a control interface to move the turntable to various positions. This standalone action is normally used for single position movements between different scans or measurements.

Position Tab

The azimuth property is controlled with this tab, and it specifies the turntable position (in degrees). You can move the turntable position by entering the desired number in the box or by using the slider to move the relative position up or down.

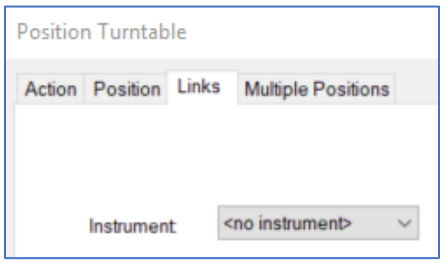


- **Margin** — The margin setting is where the positioner will be commanded to stop before the actual stop position. This setting will be site and test specific due to the differences between positioners and the equipment placed upon them.

- **Retry Count** — When we first send the start movement command, we begin reading the turntable constantly to determine whether it has reached the target. If we read the same position more than the number of times specified in the Retry Count, an error is displayed. Failure to move is an error condition, but large turntables require a large amount of time to start movement. This parameter, along with Start/Stop Delay, works to ensure accurate reading of the tower.
- **Start/Stop Delay** — When the start movement command is sent, the software will delay the time shown here, in milliseconds, before sending other commands to the turntable.

Links Tab

This tab selects the instrument to be used for the action.

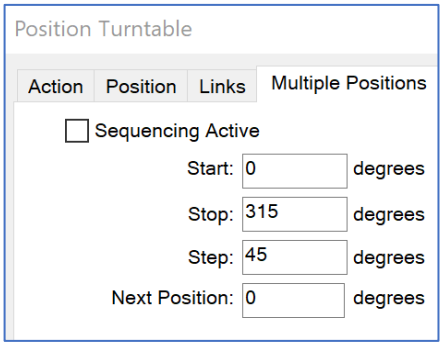


NOTE: If the Turntable instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Instrument** — Select the tower instrument.

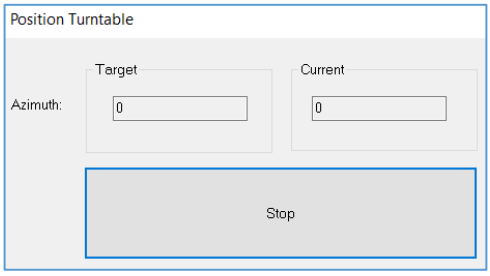
Multiple Positions Tab

This is the same as the Position Tower and will not fully be described here. See the *Position Tower multiple positions tab* for a full description.



Dialog Box

Shows the target position and the current position of the turntable. Use the stop button to end the test.



Measure Actions

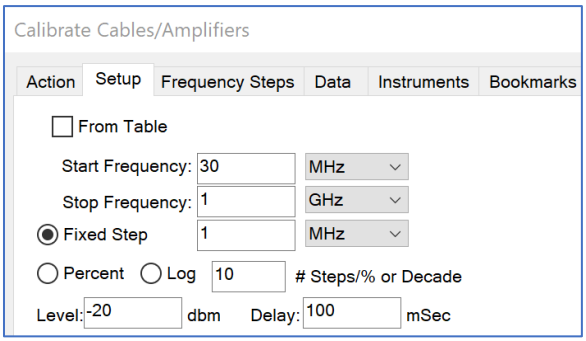
Calibrate Cables/Amplifiers

The Calibrate Cables/Amplifiers action allows the user to perform a calibration and store the results in measurement data elements. These results can then be used to correct to the results of a measurement.

This action is designed as a simple calibration routine. The ability to control step size, signal generator levels, and instrumentation is limited. For more sophisticated control of a calibration use the *Immunity/Susceptibility Calibration* or *Immunity/Susceptibility Test*.

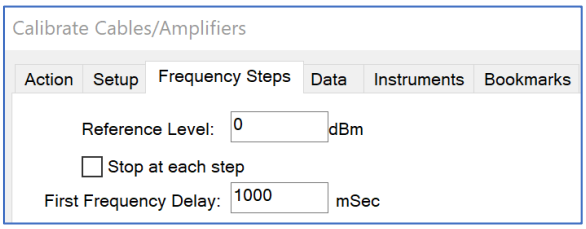
Setup Tab

This tab allows the user to identify the parameters of the calibration.



- **Start Frequency /Stop Frequency** — Enter an appropriate number in the box and select the units.
- **Fixed Step** — Determines the interval between each frequency in a linear fashion.
- **From Table** — When this check box is selected, the frequencies for this action are determined by a data element selected on this tab (the selection box will appear once this check box is selected). The data element can be of any type, except word types. With this choice, you are in complete control of the frequencies and their step sizes.
- **Level** — The signal generator amplitude that will be set at each frequency.
- **Delay** — Sets the desired delay time (in ms) before reading all measurements. Most probes and power meters have a settling time before their readings can be considered accurate. This field allows you to enter a value to create a delay prior to taking these readings. It is typical for power meters to take well over 1 second (1000 milliseconds) to level accurately. Also, older instrumentation is more susceptible to settling time and will require a larger number.

Frequency Steps Tab

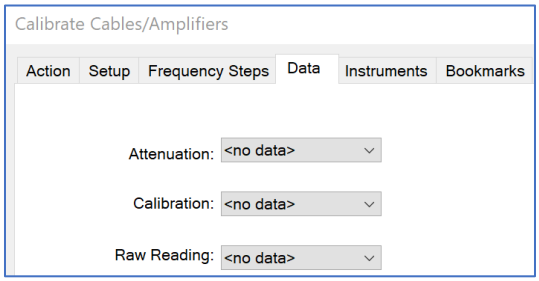


- **Reference Level** — Allows the operator to select the reference level for this measurement. This is a function of both the noise floor of the instrument and the maximum expected signal.

  If you are working in an anechoic chamber, you could set this between 60–80 dB and take valid readings. However, on an Open Area Test Site (OATS) this would put the analyzer/receiver in saturation.
- **Stop at each step** — Causes the action to pause at each frequency step to change a transducer or other components.

- **First Frequency Delay** — Some power meters have a longer settling time when power is first applied. The value entered in the box is a special delay for the first frequency step only.

Data Tab

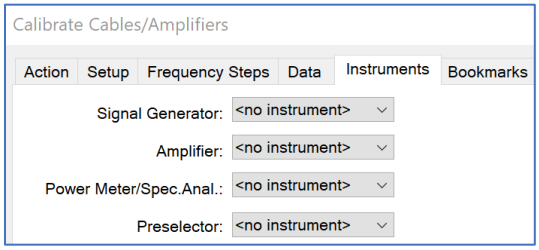


This tab allows the operator to specify data element links. This is the point of coordination between the Data window and the flowchart.

NOTE: If the Attenuation data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Attenuation** — Identifies the data storage element used to record the measured attenuation value. The attenuation value is the measured value less than the value of the Calibration data element. If no Calibration data element is present, then the measured valued is stored.
- **Calibration** — Identifies the calibration values to be used to correct the Attenuation data element during the calibration run.
- **Raw Reading** — The final attenuation value is the direct reading less than the calibration value. If this value is selected, the saved value will be the raw attenuation reading. If No Data is selected, the saved value is the raw plus calibration.

Instruments Tab



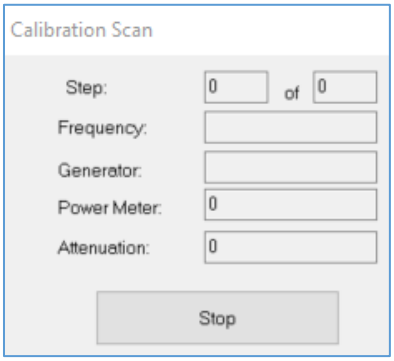
This tab allows the operator to specify the instrument links. This is the point of coordination between the Instrument window and the flowchart.

NOTE: If the Signal Generator and Power Meter/Spec. Analyzer instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Signal Generator** —The instrument selected can be the same as the power meter if you are using a spectrum analyzer with a tracking generator.
- **Amplifier** — Only select an instrument on this tab if the amplifier is GPIB controlled.
- **Power Meter/Spec. Anal.** —The instrument selected can be the same as the spectrum analyzer if this instrument has a tracking generator.
- **Preselector** — Select the appropriate instrument.

Dialog Box

This dialog box appears when Calibrate Cables/Amplifiers is executed.



- **Step** — Displays the current step that is being invoked.
- **Frequency** — Displays the present signal generator frequency.
- **Generator** — Displays the current signal generator level.
- **Power Meter** — Displays the current power level for the designated frequency.
- **Attenuation** — Displays the attenuation level to be recorded to the Attenuation data element.

Amplifier Calibration

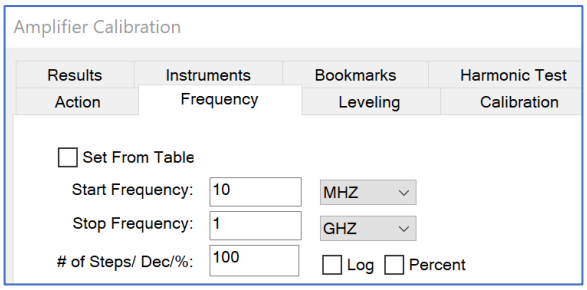
The Amplifier Calibration action is used to find the 1 dB compression point and the harmonics performance of an amplifier. It does this by increasing the input power in steps defined by the user until the measured output step is less than the input step by another user-defined parameter.

Once this value is found, it will output that maximum value in dB of the amplifier and then subtract 1 dB from that value as the 1 dB point. The harmonics portion of this action will set a signal at the 1 dB compression point (as found on the previous step), then measure the value of the desired harmonic and output the difference.

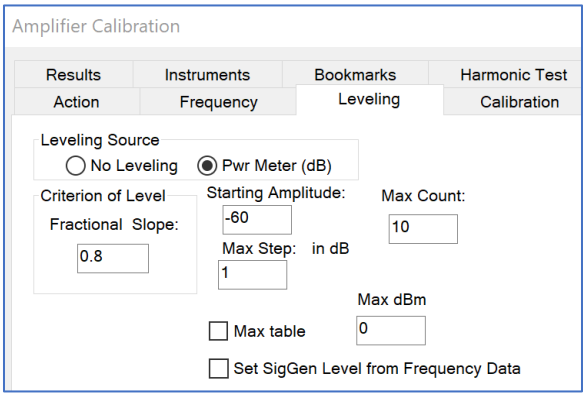
This action can only measure one harmonic number at a time, so it will run through the frequency range but only record the selected harmonic such as the harmonic number 1, 2, or 3 on the *Harmonic Test tab*. This can be set up in the flowchart in a sequence that will allow a single test starting point to do all desired harmonics.

Frequency Tab

The tab is like other frequency tabs and will not fully be described here. See the *Immunity Test Frequency Tab* for a full description.



Leveling Tab



This tab is like the *Immunity Test Leveling Tab* **except** that is uses a Criterion of Level window with a fractional slope user entry.

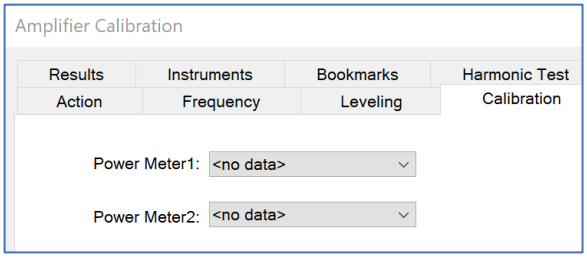
- **Criterion of Level** — Defines the value that the dB will be determined with by comparing the signal generator step with the measured step of the power meter.

  If it is set to 1, the loop will look for a 1 to 1 step before identifying a compression point, so if the power meter steps equal the exact steps of the signal generator, it will not identify the compression point until the power meter registers a delta of less than 1 from step to step.

  If there is noise in the measurement, you may get a false identification of a compression point, so you may want to allow for some tolerance by setting it to 0.8 or slowing down the measurement time on the *Instruments tab*.

- **Max Count** — Sets the number of counts of how many times the leveling will try to meet the criteria for each frequency.
- **Max table** — Used with a cable factor file so that the signal generator will be allowed to go above 0 dBm in proportion to the cable loss at that frequency. This is valuable in the higher frequency ranges where the cable loss is more significant.
- **Max dBm** — The Do Not Exceed Level fed into the amplifier.

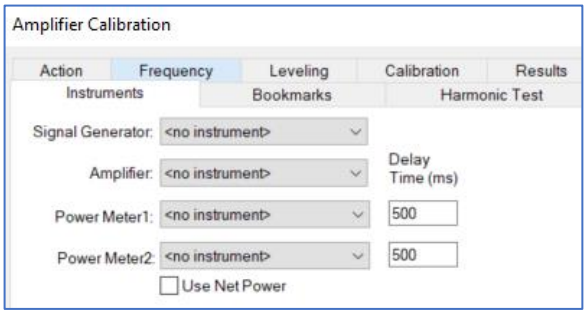
Calibration Tab



This tab will allow the offset of the directional coupler to be accounted for.

NOTE: If the Power Meter 1 data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

Instruments Tab



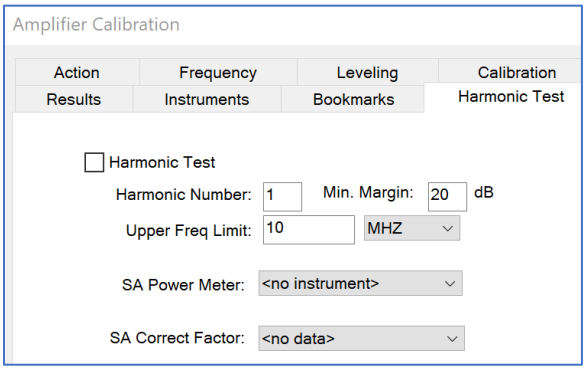
This tab is similar to other instruments tabs and will not fully be described here. See the *Calibrate Cables Instruments Tab* for the full description.

NOTE: If the Signal Generator and Power Meter 1 instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

Harmonic Test Tab

Use this tab to run the harmonics routine.

NOTE: Must select the **Harmonic Test** box to activate this feature.

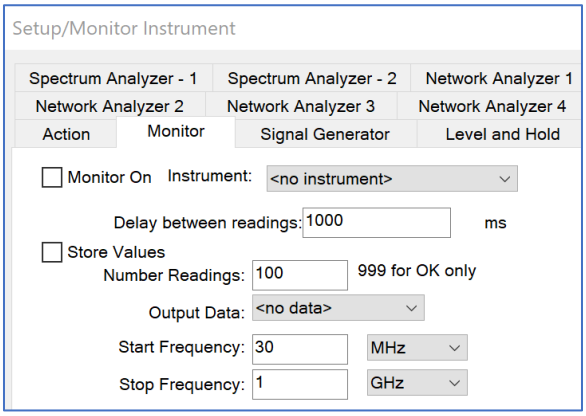


- **Harmonic Number** — Selects which harmonic number will be recorded.
- **Min. Margin** — Sets the minimum margin value that will flag the user if it is exceeded.
- **Upper Freq Limit** — Used to prevent the program from sending a frequency command beyond the range of the instrument.
- **SA Power Meter** — Select the measurement instrument here.
- **SA Correct Factor** — Provides the reference if any correction factor needs to be applied. The directional coupler is already accounted for in the *Calibration tab*.

Setup / Monitor Instrument (READ)

The Setup/Monitor Instrument action is used to monitor an instrument. Specific conditions can be set for the number of readings, and whether to store them or not. In addition, it is used to set up external signal generators, spectrum analyzers, and network analyzers prior to their use in other actions. All setups are performed prior to monitoring.

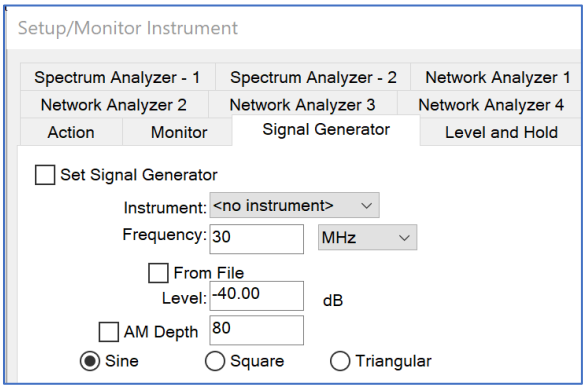
Monitor Tab



- **Monitor On** — When the Monitor On check box is selected, the instrument selected will be read, either continuously or for a defined number of times. If a string of values is desired, the read value will be stored.
- **Instrument** — Select an instrument to be read.
- **Delay between readings** — If multiple readings are specified, there is a dwell time between each reading.
- **Store Values** — Select the check box to save the measured value.
- **Number Readings** — Enter several readings before continuing to the next step of the flowchart.
- **Output Data** — If storing is desired, select a data element. The output will be defined as a Measurement Type data element.

- **Start Frequency /Stop Frequency** — For the created output, enter an appropriate number and select the units.

Signal Generator Tab

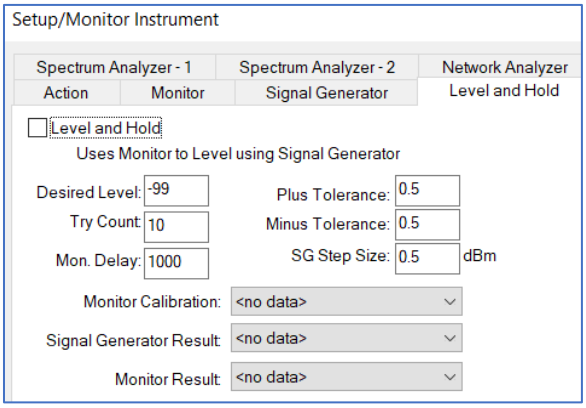


Use this tab to set up a signal generator to be turned on prior to exiting the action.

NOTE: Must select the **Set Signal Generator** box to activate this feature.

- **Instrument** — Select an instrument.
- **Frequency** — Set the frequency of interest. If the From File check box is selected, the value for this frequency (or its interpolated value) will be used for amplitude.
- **From File** — Select the check box if you need the software to find an appropriate amplitude from a table for values.
- **Level** — If the From File check box is not selected, then this is the amplitude that the signal generator will be set to. If it is out of range for the instrument, you will not receive an error.
- **AM Depth** — Select the check box and set the value if you want to turn on AM modulation. The frequency will be the frequency that was specified in the previous field.
- **Sine/Square/Triangular** — If modulation is on, you can select sine, square, or triangular for wave shape.

Level and Hold Tab



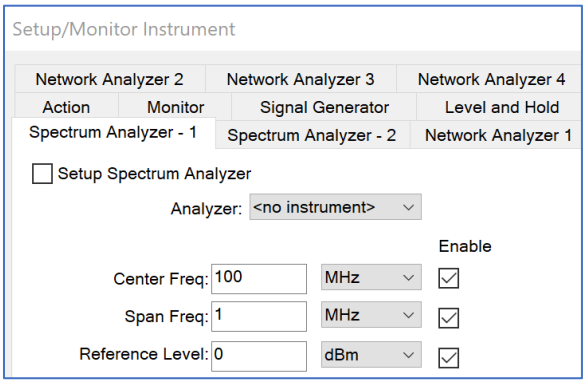
There is a requirement in the MIL STD to establish a known field and use this as a reference to calibrate the emissions test. Turning on the signal generator to an expected value may not be enough; some instrument combinations will be best served if you level to the desired value. In this case, we use this action to read the expected value and adjust the signal generator to achieve the final level within the tolerance.

NOTE: Must select the **Level and Hold** box to activate this feature.

- **Desired Level** — Enter a desired final level. The signal generator will be set to the amplitude on the Signal Generator tab. The monitor will be read and then the signal generator will be adjusted until the desired level is achieved.
- **Plus/Minus Tolerance** — Positive tolerance above the desired level. Any value between the desired level less than the minus tolerance and the desired level, and the plus tolerance will be accepted.
- **Try Count** — The number of steps the signal generator will take to reach the desired level before it will flag a fail to level condition.

- **Mon. Delay** — Delay between each reading of the monitor during the leveling loop.
- **SG Step Size** — Step size for each signal generator step in the leveling loop.
- **Monitor Calibration** — If there are correction factors that need to be added to the monitor reading to ensure an accurate reading, select a data element that contains the correction data. The software will look up the correct value for the frequency this action is setting.

Spectrum Analyzer – 1 Tab



There are certain times when the user needs to set up a spectrum analyzer to a known state prior to other steps in the flowchart executing.

The *Network Analyzer 1* and *Network Analyzer 2* tabs control these settings. The most common time to use this is when an analyzer is being used as a power meter during Immunity testing.

The standard instrument drivers will automatically select a resolution bandwidth (RBW), video bandwidth (VBW), and span and sweep time (ST). There are certain times where the user will want to limit these settings. Using the _**spc** instrument drivers (for example, hp8566b_spc.ins instead of hp8566b.ins) will allow these settings to be preserved.

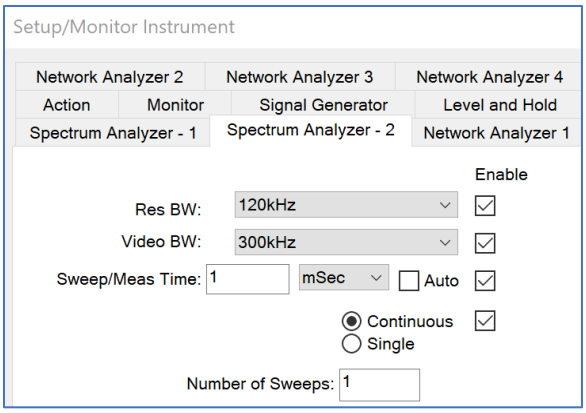
NOTE: Must select the **Setup Spectrum Analyzer** box to activate this feature. If the check box is deselected, all these settings are ignored during execution.

- **Analyzer** — Select an instrument from the list.
- **Center Freq** — The center frequency is entered here for set up purposes. If the instrument is used later during Immunity testing, the frequency will change to match current frequency.
- **Span Freq** — Enter a desired span for the span frequency. This value is preserved during subsequent Immunity testing, regardless of frequency.
- **Reference Level** — Enter a default reference level for the instrument. The instrument will adjust higher if the signal is too strong.

Spectrum Analyzer – 2 Tab

If the **Setup Spectrum Analyzer** box check box on the Spectrum Analyzer - 1 tab is deselected, all these settings are ignored during execution.

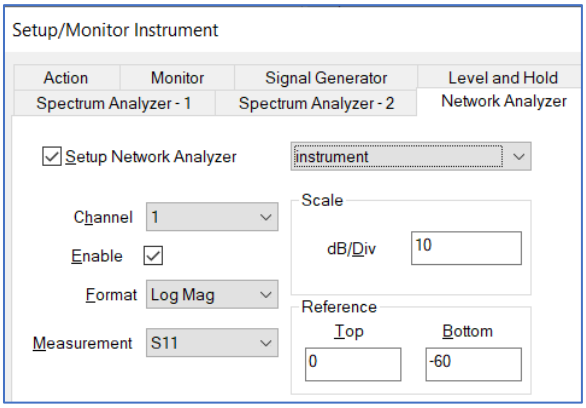
This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



- **Continuous/Single** — Choose whether the instrument will be in single sweep mode or continuous sweep mode. If in single sweep mode, the number of sweeps is read and the number of sweeps is taken in Max Hold for each instance the instrument is read.

Network Analyzer Tab

In certain circumstances, the user will need to set up specific conditions for a network analyzer prior to performing other measurements. Use this tab to set up a Network Analyzer to be turned on prior to exiting the action.



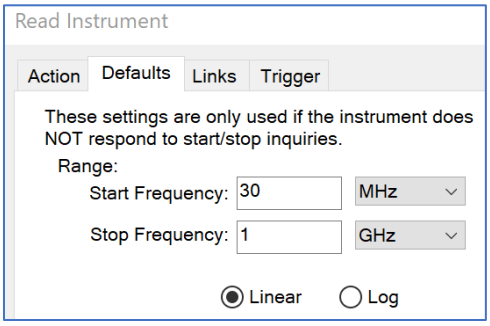
- **Setup Network Analyzer** — Must select this box to activate this feature.
- **Instrument** — Select an instrument.
- **Channel** — Select a channel number 1 - 16.
- **Enable** — Check box enables the rest of the settings.
    - **Format** — Select the mode (S11, S12, S21, S22, or S13) for each channel.
    - **Measurement** — Check the measurement type desired for each channel. Choices are not necessarily available for all instruments. If a type that is not available is selected, the instrument will remain in its default mode.
    - **Scale** — Choose the appropriate dB/Div.
    - **Reference** — Choose the appropriate Top and Bottom for the channel.

Read Instrument

The Read Instrument action is used to retrieve a trace or waveform from an analyzer or oscilloscope. We retrieve the waveform, not a screen dump. The data is parsed into a starting and stopping frequency (time) with the pertinent number of points and amplitudes.

Defaults Tab

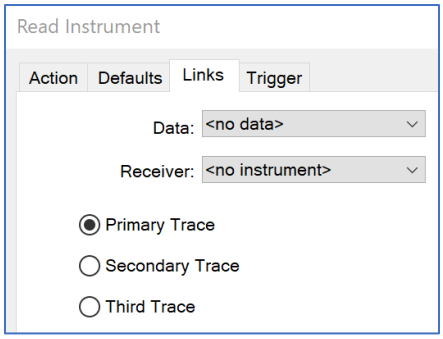
Certain instruments will not return the start and stop values for the trace. The Defaults tab sets a set of default values for this range.



- **Start Frequency /Stop Frequency** — For default readings, enter an appropriate number and select the units.
- **Linear/Log** — Defines how frequencies will be stopped, whether in log or linear interpolation.

Links Tab

This tab controls the instrument to be read and the data element the results will be stored.

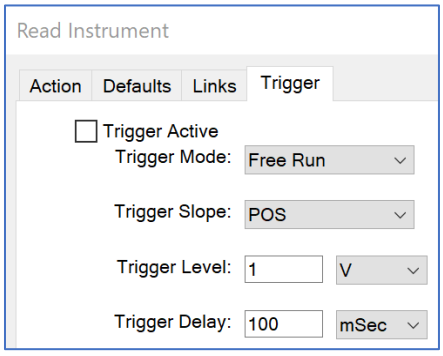


NOTE: If the Receiver instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Data** — Select an appropriate data element to store the trace data.
- **Receiver** — Select an instrument to be read.
- **Trace** — Some instruments have multiple traces. This setting will determine which trace will be retrieved.

Trigger Tab

When reading some instruments, it might be desirable to trigger the sweep prior to reading the trace back. The Trigger tab controls the triggering conditions.



NOTE: Must select the **Trigger Active** box to activate this feature.

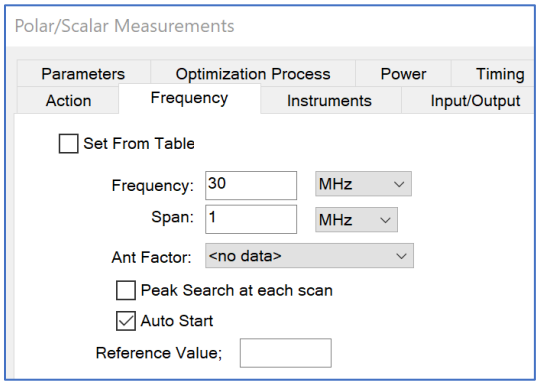
- **Trigger Mode** — Select the desired mode for triggering.
- **Trigger Slope** — Select the slope (POS or NEG).
- **Trigger Level** — Enter a level for the trigger event and select the level type (mV, V, PCT).
- **Trigger Delay** — Delay time prior to reading after the trigger has happened.

Polar / Scalar Measurement

This action is used to perform a polar or scalar measurement across a tower or turntable.

Frequency Tab

There are two ways to specify the frequency in this action. When only one frequency is shown, the polar measurement will perform that frequency only. If the Set From check box is selected, then multiple frequencies will be measured.

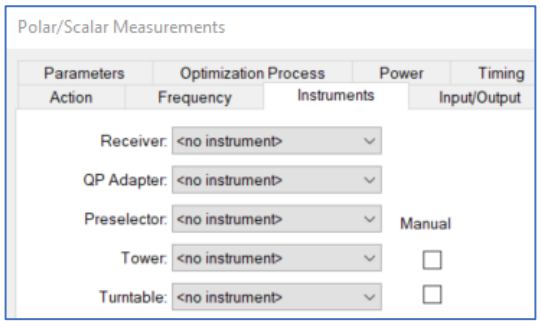


- **Set From** — Select the check box and a data element to perform a measurement using multiple frequencies.
- **Frequency/ Span** — Enter frequency (value and unit) and span at which the polar measurement will be performed.
- **Ant Factor** — Select a data element. The values from this data element will be added to the reading to correct raw data.
- **Peak search at each scan** — When performing a polar measurement, you have two options. The default measurement will tune the analyzer to the desired frequency and perform a polar measurement. If this check box is selected, once the frequency is set up, a marker will move the peak to the center frequency of the strongest point within the span.
- **Auto Start** — If selected, the test will run automatically without any further operator interference. In the not selected, upon execution an intermediate dialogue will prompt the operator for a frequency.
- **Reference Value** — The reference value entered will be set on the spectrum analyzer prior to the start of the measurement.

Instruments Tab

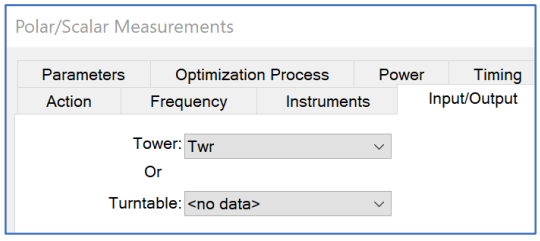
This tab is similar to other instruments tabs and will not fully be described here. See the *Scan Peaks Instruments Tab* for the full description.

NOTE: If the Receiver and Turntable instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



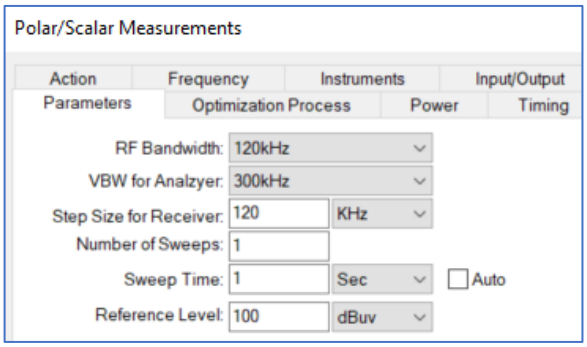
Input/Output Tab

Select a data element to store the results of this measurement.



Parameters Tab

This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.

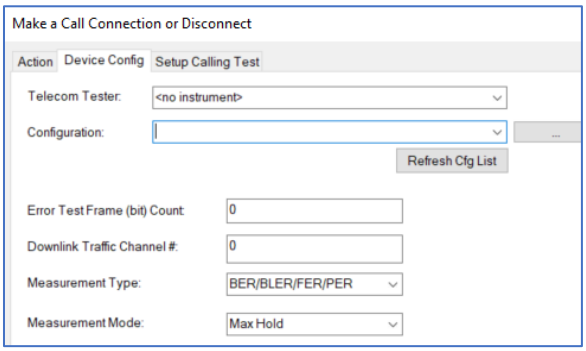


Call Setup

Call Action is used to make a cellular call with a cellular base station emulator or telecom tester. This can be used with the Immunity Test action to perform Audio Breakthrough testing. The telecom tester needs to be installed to complete the setup.

Device Config Tab

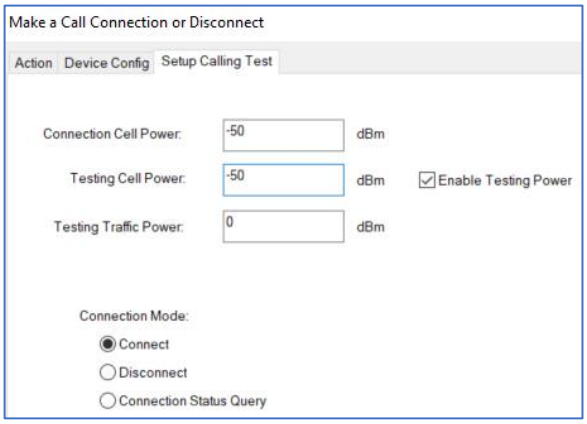
NOTE: If the Telecom Tester instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



- **Telecom Tester** — Select the telecom tester that is listed in the Instruments section.
- **Configuration** — When selected this will provide another interface that will list the available protocol settings (this will be configures in the next section, *Selecting the Telecom Tester Interface*).
- **Error Test Frame (bit) Count** — Select the appropriate call quality parameter and enter the number of units to be used as a reference.
- **Downlink Traffic Channel #** — Enter the channel used for the downlink.

Setup Calling Test Tab

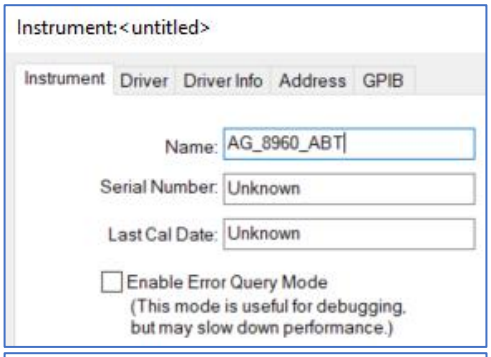
- Connection Call Power — Enter the transmit power level for the telecom tester.
- Testing Traffic Power — Enter the power level for the EUT.
- Connection Mode — These actions can be used to make a call or to disconnect a call.



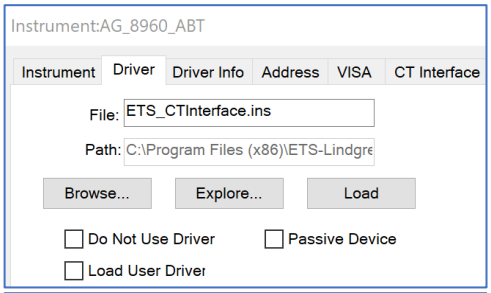
Selecting the Telecom Tester Interface

To install the interface for the telecom testers, the following steps are required for setup in the Instruments section.

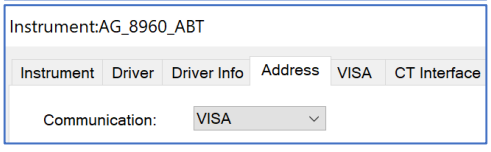
1. Create an instrument to be the Telecom tester and name it AG_8960_ABT.



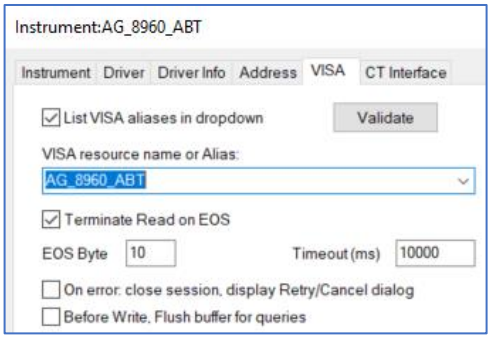
2. Go to the Driver tab and Browse to find the ETS_CTInterface.ins file.



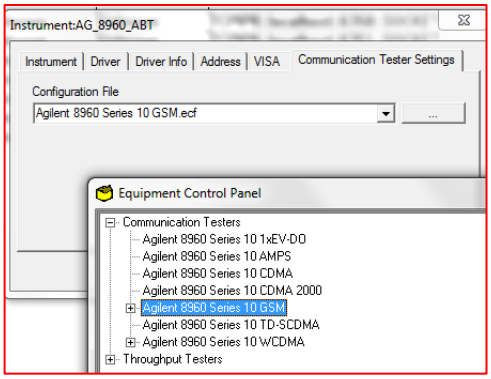
3. Go to the Address tab and change the Communication to VISA.



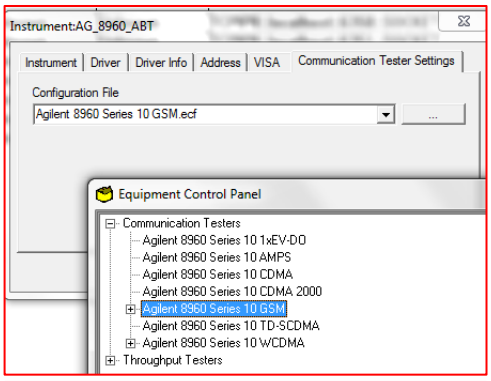
4. Go to the VISA tab and change the VISA resource name or Alias to the telecom tester. Make sure to select Terminate Read on EOS (This is required to work with the ETS_CTInterface as well as any simulated instruments).



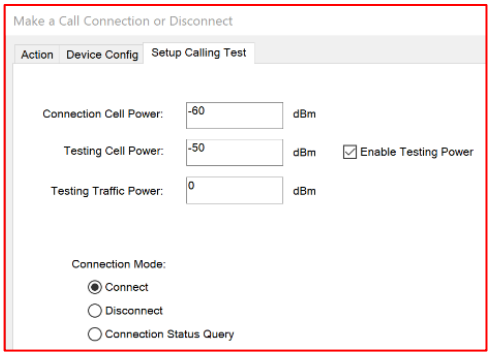
5. Go to the Communication Tester Settings (CT Interface) Tab and select the … button and select the appropriate configuration from the displayed list.



6. Go to the Call action Device Config tab and set the configuration to 8960 GSM PCS, set the error test frame-count to 2000, set the downlink traffic channel # to 512, set the measurement type to RxQ, and set measurement mode to Max Hold.



7. Go to the setup calling test tab and set the connection cell power to -60, set the testing cell power to -50, set the testing traffic power to 0, Enable testing power, and set the connection mode to connect.



Switch Actions

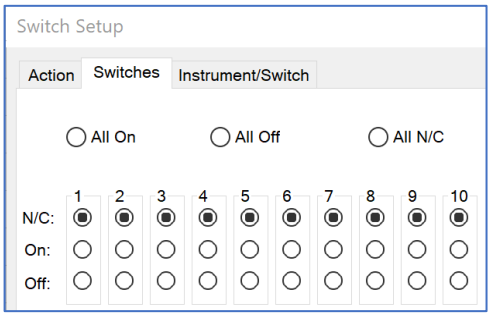
There are seven types of switch interfaces. Some switches allow the user to automate Immunity testing for EUT failures. Most of these are slightly different versions of the Switch Setup action, which are created to allow measurement of voltage rises, voltage drops, open/close relay states, and switch positions.

Switch Setup

This Switch Setup action allows the user to manipulate GPIB controlled switch devices. This action is also used when performing automatic failure analysis. It is designed to use the Keysight 11713 series Attenuator / Switch Driver. It will fully control that device as well others that can emulate it.

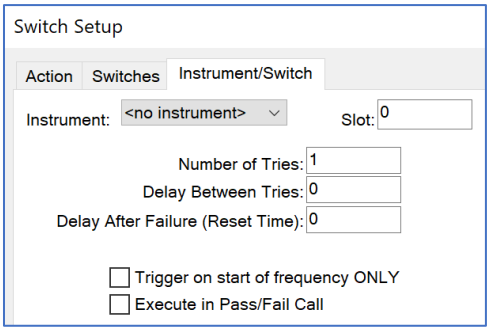
Switches Tab

This tab allows the user to control the state of up to 10 switches. You can selectivity control an individual switch or control the state of all the switches with All On, All Off, or All N/C option buttons.



Instrument/Switch Tab

This tab specifies the instrument to be used during the execution of this action. You must select an instrument from the list.



- **Instrument** —Select the appropriate instrument.
- **Number of Tries** — When used for pass/fail analysis, this setting determines how many times the switch will be read before returning control to the Immunity test. If the EUT has a slow response, you might need to set this value to a larger number to trap failures.
- **Delay Between Tries** — When used for pass/fail analysis, this setting determines the delay time between each try.
- **Delay After Failure (Reset Time)** — Sets the delay time before a Reset command is sent to the switch.
- **Trigger on start of frequency ONLY** — Can be used to trigger an action at the start of testing each frequency, such as the need to reset a switch before testing for a failure. Select this check box if you want to trigger this only on the start of frequency.
- **Execute in Pass/Fail Call** — Will allow the Immunity action to call this action during a Pass/Fail call.

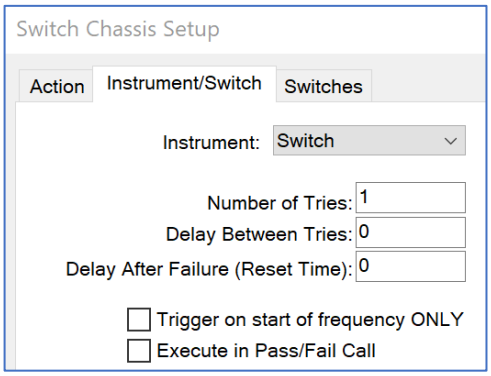
EMSwitch

The switch should be used with the EMSwitch™ RF Switch Plug in Card.

Instrument/Switch Tab

This tab is similar to other switch tabs and will not fully be described here. See the *Switch Setup Instrument/Switch Tab* for the full description.

NOTE: If the Instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

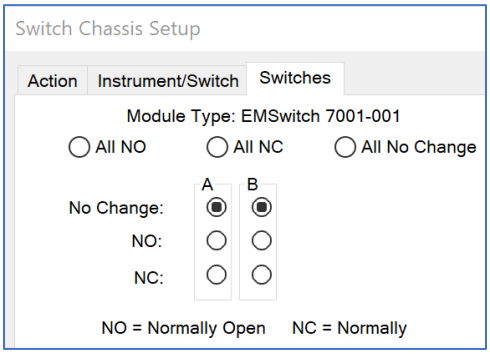


Switches Tab

This Tab configures its settings to the type of switch connected.

NOTE: Must select an instrument using the ETS_EMSwitch.ins on the Instrument/Switch tab.

- **All Positions** — Select All On, All Off, or All N/C option buttons to control the state of all the switches.
- **Switch Positions** — Select switch A or B and the position for each.



Switches A, B, C, D, E

A. Control switches and relays using the HP3497A or other DAQ.

B. Controls switches with multiple digital lines.

C. Controls switches and relays using the HP34970A.

D. Controls remote GPIB switches and reads instrument channels.

E. Controls remote DAQs and reads/stores channels states.

Emission Actions

Range Type Measurement Actions

Range measurements are typical of spectrum analyzers and scanning receivers. They require a start and stop frequency, along with bandwidth and sweep times (SA) and measurement times (Rx). The two main scanning type measurements are *Measure Range* and *Pre-Scan*.

The Measure Range action is the simplest since it does not use positioners. It is a staple of MIL STD tests and conducted emissions tests. The Pre Scan action is used with turntable and tower positioners.

Measure Range

The Measure Range action is used to configure the measurement of a range of frequencies used by a receiver or spectrum analyzer. The measured data is stored in the specified data element. The operating conditions for the receiver or spectrum analyzer are set in the *Parameters tab*.

The Measure Range action requires one data element for storage of the results and at least a receiver or spectrum analyzer. These parameters are set on the *Data tab*. The number of data points is determined by the range times the number of points on the analyzer.

NOTE: All NEW drivers for measurement frequencies are added to *Measure Range **Extended*** as *Measure Range* only supports OLDER instruments.

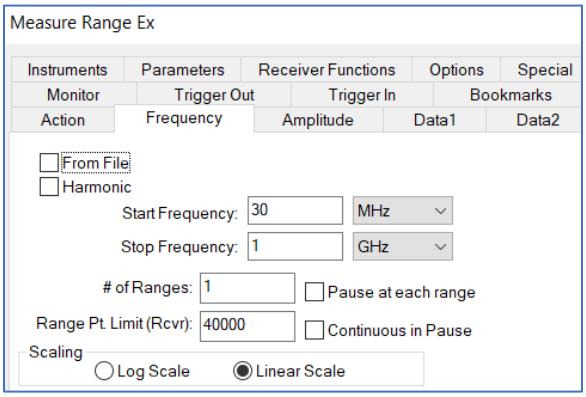
If you already created a *Measure Range* action, you can use the Bookmarks Tab in *Measure Range **Extended*** to import the settings. However, the data items will NOT be set, so you will have to do that manually, as they are the only thing that will not import.

Measure Range Extended

The Measure Range Extended action was developed for the newer, more advanced instruments such as the ESU. It also lets the instrument do the multiple sweeps. It is the **same as** the *Measure Range* action **except** that it adds the ability to measure two ranges (like peak and average) at the same time. It has two data pages, two detectors on the parameters tab, and the Trigger In tab.

Frequency Tab

This tab allows the operator to select the range parameters over which measurements will be taken.



- **Start/Stop Frequency** — Enter an appropriate number in the box and select the units.
- **# of Ranges** — Entering a value sets the number of ranges. When this action is executed, the frequency span between the start and stop frequencies is divided into the number of ranges specified. This is true for spectrum analyzers only. This item is ignored for true receivers.

  Unique scans are made of each frequency range with the results collected into a single data element. The number of points in the data file is the product of the number of ranges and the number of measurement points per scan for the instrument connected to this action.

  In designing the measurement, knowledge of the capabilities of the analyzer/receiver is critical. If the spectrum analyzer has a 401-dot resolution (a common setting), then when you do a scan the instrument will only send 401 points, regardless of the bandwidth you specified in the parameters. If you are making a measurement from 30 MHz to 1000 MHz with a 120-kHz bandwidth, you need 8,083 points ({1,000,000,000-30,000,000}/120,000).

  If the analyzer only reports 401 points, you need to break this frequency range into at least 20 ranges to obtain a reasonable accuracy. For this reason, you might want to design a scan with broad bandwidths, calculate the peaks, and run a measurement of the peaks only with the narrow bandwidth. Experimentation by the user is recommended.

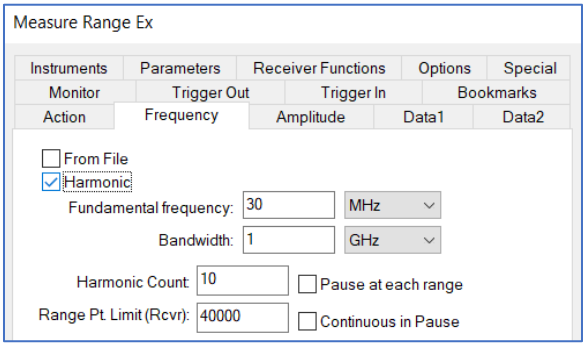
- **Pause at each range** — If there is more than one range, selecting this will cause the software to pause between each range and let the user choose to retry the range, store the value, or cancel the test.
- **Continuous In Pause** — When you pause, the receiver will continue to sweep during the pause.
- **Scaling** — The Log Scale and Linear Scale option buttons are used to select the appropriate frequency range scaling. The default is linear scale. If the range is 30 MHz to 1 GHz with log scaling selected, the frequency range will be adjusted to measure the number of ranges per decade across the stated frequency range.

  For example, if you entered 30 MHz to 1 GHz for the frequency range and set the ranges to two in log scale, you would get a decade from 30 MHz to 300 MHz broken into two ranges and then another sequence from 300 MHz to 3 GHz broken into two ranges. Since our ending frequency is only 1 GHz, actual number of ranges in the last decade would vary depending upon the logarithmic steps.

- **From File** — Select the From File check box if you wish to read in the frequency data from an external file. This data must be imported first into a file type data-element. The data should be comma separated variables with column one identifying the frequency and column two identifying the value.

  For this feature to work, the frequency list (from the data file) should represent pairs of start and stop frequencies. If the number of data elements is not even, this feature will return an error. You can still utilize the Pause at each range and Continuous in Pause options when using From File.

- **Harmonic** — The Harmonic option allows this action to be used to measure a primary frequency (Start Frequency) and then take measurements at a specific number of harmonics. At each frequency, a span for the range measurement will be established using the Bandwidth parameter.

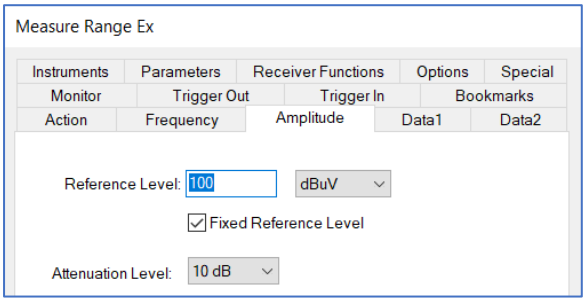


  In the example shown here the software would scan from 29.5 MHz to 30.5 MHz, 59.5 MHz to 60.5 MHz, and 299.5 MHz to 300.05 MHz. These 10 scans (primary plus nine harmonics) are combined into one data array. The **Pause at each range** and **Continuous in Pause** options are available in this mode.

  - **Fundamental Frequency** — Enter the desired primary frequency. This will be the first frequency measured. All harmonics are calculated from this frequency.
  - **Bandwidth** — Choose an appropriate bandwidth for the analyzer to take a sweep. This is the same as a span.
  - **Harmonic Count** — This value determines the number of harmonics measured. The primary frequency is the first harmonic.

Amplitude Tab

This tab allows the operator to select the reference level and attenuation levels to be used with the spectrum analyzer during each scan. These values may be ignored for some EMC receivers, especially the Reference Level specification.



- **Reference Level** — This a function of both the noise floor of the instrument and the maximum expected signal. If you are working in an anechoic chamber, you could set this to 60 80 dB and take valid readings. However, for OATS this would put the analyzer/receiver in saturation.
- **Fixed Reference Level** — During scans the default behavior is to set the reference level specified and is not changed during the various scans. If this field is deselected the behavior changes slightly.

  The specified reference level is set at the start of each scan, but after the information for the analyzer is downloaded, it is checked to determine if there are any values within 10 dB of the reference level. If there are, then the reference level is stepped up 10 dB and rescanned.

  This procedure continues until the maximum signal is within the current range of the analyzer display or we have reached the maximum. This will ensure that it will report accurately the total signal strength. Some analyzers will report a signal that is off the screen, but others report numbers that are not accurate.

  For this reason, this option will allow accurate measurements under these conditions. This procedure does have one side effect that concerns most users: when we combine the multiple scans together, the apparent noise floor will shift depending upon the changes made to the reference level.

- **Attenuation Level** — Use this field to select the attenuation level to be used for the scans. This allows you to adjust the attenuation to match the environment or EUT. You can choose values from 0 60 dB or AUTO.

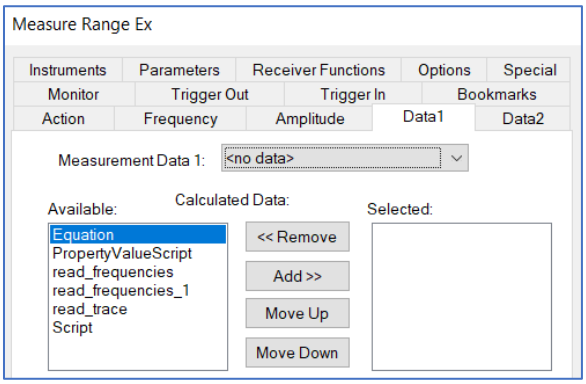
  We do not generally recommend AUTO; some spectrum analyzers change other settings within the analyzer when the attenuation is set to AUTO. If you test this to ensure the performance is what you expect, it can be a very easy method of controlling the analyzer response to high ambient fields.

Data 1 Tab

The Data 1 tab is used to identify the data storage element to be used by this action. When the action is executed, the data element is resized to the product of the number of ranges and the number of measurement points per scan for the instrument connected to this action.

If no errors are detected during the execution, the data is marked as valid and the frequency and amplitude data are set for each point measured.

NOTE: If the Measurement data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



- **Measurement 1 Data** — Select an element to store your data from the list.
- **Calculated Data** — Within this action the normal return value is the raw data from the scan. Often the user will want to view corrected data while a test is still executing. Data is corrected in TILE! through math equations; if an Equation data element is selected at this point, the equation will be executed between each range and thus all graphs and tables that use that data will be updated to reflect corrected values. Select here the Equation type data element that will be executed.

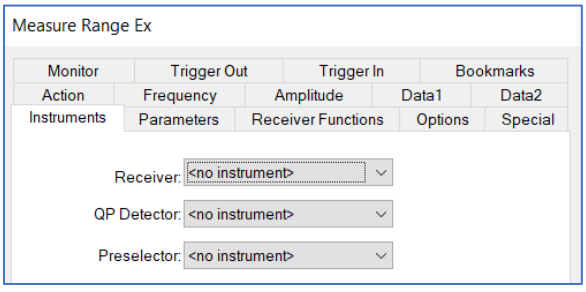
  It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

Instruments Tab

This tab specifies the instruments to be used during the execution of this action.

NOTE: If the Receiver instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Receiver** — Identifies the instrument from which measurements will be taken.



- **QP Detector** — Specifies the QP detector that will be used for the measurement. Only select an instrument if you have a physically separate box for QP measurements. This is true of both the HP85650 QP Adapter and the Anritsu MN1602a.

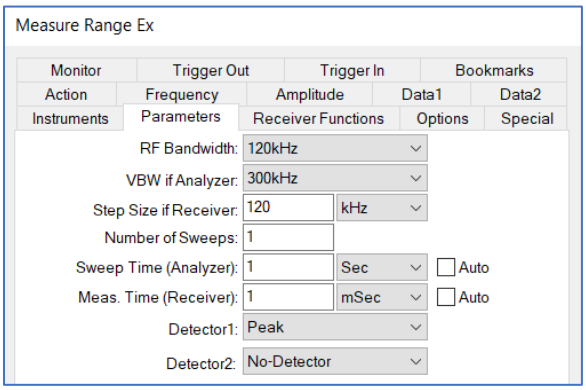
  If the analyzer/receiver has a built in QP function, do not specify an instrument in this box. If a QP adapter is selected, the performance of the test will change slightly. When using an HP8566/8 with the HP85650 QP adapter, the manufacturer specifies a set of matching RF and video bandwidths to match the RF bandwidths of the QP adapter.

  The TILE! software will set the analyzer to these matching bandwidths when using the parameters settings on the QP adapter. If you want a specific bandwidth on the analyzer, do not specify the QP adapter in this box and the software will only set up the analyzer.

- **Preselector** — Specifies the Preselector that will be used for the measurement. You only need to select an instrument if you have a physically separate box acting as a Preselector. If the analyzer has this function built in, do not specify an instrument in this box.

Parameters Tab

This tab is used to specify the instrument setup for this measurement.



- **RF Bandwidth** — The resolution bandwidth (RBW) is the value for the internal filtering used by either the spectrum analyzer or receiver during the measurement. This value is usually mandated by the specification that the user is measuring against.

  If you select an RBW that is not supported by the instrument being used, you will not get a warning message. Instead, the instrument driver will choose the closest bandwidth that is supported. If you specified a QP adapter on the *Data tab*, the RF bandwidth will be set for the QP adapter.

- **VBW if Analyzer** — Display the available video bandwidth (VBW) settings. Not all instruments will be able to use each of these settings, but the instrument driver will pick the closest setting that is valid for the specified analyzer.

  Spectrum analyzers use a video filter as part of the measurement/display cycle. Generally, the video bandwidth (VBW) should be equal to or greater than the RBW. A good rule of thumb is three times the RBW.

  Set this value as needed or required. This value is generally ignored when doing average measurements using spectrum analyzers since the CISPR methodology requires the use of the minimum VBW that is supported by the instrument.

- **Step Size if Receiver** — The step size sets the increment that will be used by the receiver during the measurement scan. Typically, EMI receivers do not actually sweep (as spectrum analyzers do). Instead, they take numerous individual measurements and often display this as a continuous sweep. The step size determines the number of points; it is found by considering the stop frequency minus the start frequency, and then dividing the result by the step size.

- **Number of Sweeps** — A setting greater than one will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the number of specified times and hold the peak value internally.

- **Sweep Time (Analyzer)** — The sweep time (ST) is the amount of time used by the analyzer to perform one continuous sweep (from start to stop). The actual time per measurement step is the ST divided by the number of points per sweep (set by manufacturer).

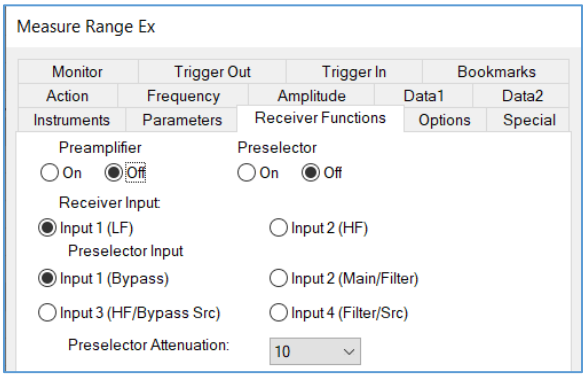
  NOTE: Setting this value to greater than 10 seconds can cause the instrumentation to time out. If this happens, change the GPIB I/O timeout settings in the instrument definition to a value higher than the sweep time. See *GPIB tab* for more detail on sweep times over the GPIB bus.

- **Meas. Time (Receiver)** — True receivers take individual measurements and then group this information into a sweep result. The measure time (MT) is the time increment per frequency step.

- **Detector** — Selects the measurement detector desired for this scan. The available options are Peak, Quasi Peak, Average, and Sample.

Receiver Functions Tab

This tab turns the preamplifier and preselector ON or OFF, as well as sets the receiver input type.

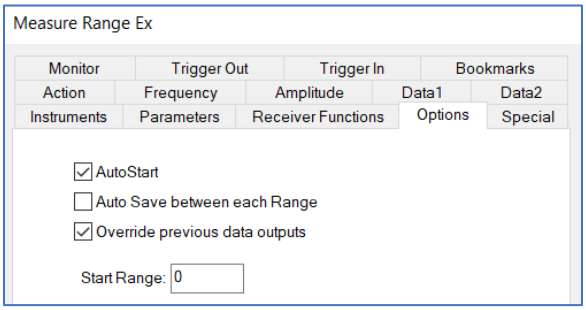


- **Preamplifier** — Enables the preamp.
- **Preselector** — Enables the preselector.
- **Receiver Input** — If you have 2 inputs, this setting chooses which one to use.
- **Preselector Input** — This is used if you have an RF Preselector.

Options Tab

This tab controls the state of the action when it starts execution.

With Auto Start selected, this action will execute a measure range function as defined. The Auto Save between each Range and Start Range settings allow you to restart an action in a place other than the start.



- **Auto Start** — When this check box is selected, this action will execute as a defined executing step such as a compliance test.

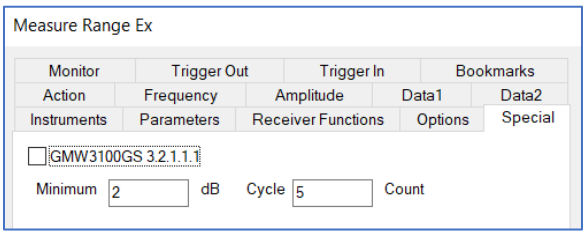
  If this check box is deselected, then an intermediate dialogue will appear, prompting the operator for start frequency/stop frequency. This option is most useful in engineering evaluation where you would be zooming in to take detailed measurements of specific frequency ranges. You will be able to change the harmonic and continuous options each time the action executes.

- **Auto Save between each Range** —Select this option when multiple sub ranges exist, so a complete profile save will execute between each range. In the event of a catastrophic failure you can restart, edit, and change the start range and begin a test at an intermediate sub range.

- **Start Range** — If multiple sub ranges are defined, this value represents the starting sub range for the execution of this action. If the start range is greater than zero and there is no data in the data element, you will not have valid readings in the earlier sub ranges.

Special Tab

NOTE: These settings only apply to General Motors W3100G 3.2.1. In this requirement, GM has defined a very specific methodology for determining the number of sweeps.



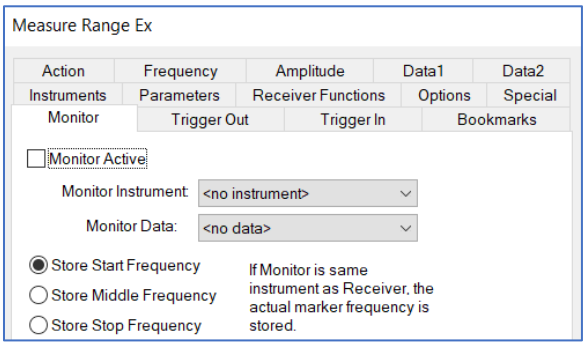
It changes dynamically based upon repetitive max marker readings. Sweeps will continue until the maximum of two repetitive sweeps is less than a defined value in dB.

NOTE: Must select the **GMW3100GS 3.2.1** box to activate this feature.

- **Minimum dB** — Defines the minimum value in dB.
- **Cycle Count** — The maximum total number of sweeps before proceeding to the next sub range. The instrument is in Max Hold during all sweeps.

Monitor Tab

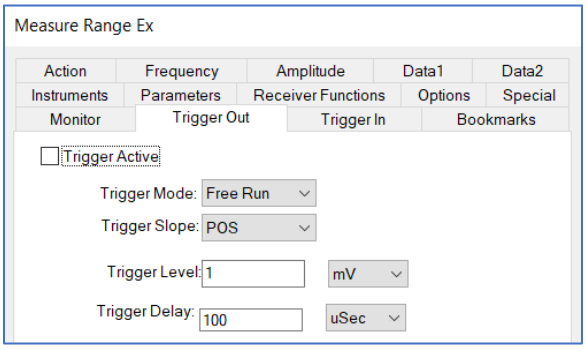
NOTE: Must select the **Monitor Active** box to activate this feature.



- **Monitor** — Defines an instrument to be read during each sub range.
- **Monitor Data** — Defines the data element where the monitor instrument readings are stored.
- **Store Start/Middle/Stop Frequency** — The monitor is read and the amplitude is stored in the monitor data. A frequency will be assigned from these three options.

Trigger Out Tab

This tab is used to trigger an external oscilloscope measurement.



NOTE: Must select the **Trigger Active** box to activate this feature.

- **Trigger Mode** — Specific mode setting for the oscilloscope prior to triggering.
- **Trigger Slope** — Sets the slope of the oscilloscope reading to positive or negative.
- **Trigger Level** — Defines the amplitude of the auto trigger for the oscilloscope.
- **Trigger Delay** — Delay time on the oscilloscope after triggering.

Dialog Box

This dialog box appears when Measure Range is executed and gives the user visual information on the current step, start and stop frequencies, and a Stop button to halt execution of the action.

If the **Stop** button is clicked, the action will terminate at the end of the current GPIB operation (this range scan). The **Pause** button will pause execution after the current analyzer scan, but prior to the next scan. With this option, you can manually check a reading and then retry the scan, capture the current scan, or cancel execution.

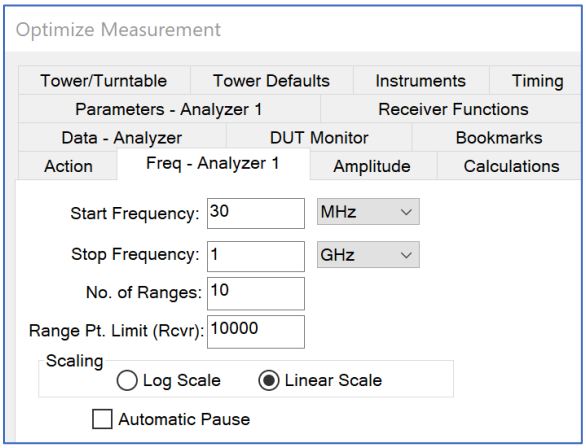
Optimize Measurement (PRE-SCAN)

The Pre Scan action is used to perform a measurement for a range of frequencies, using either a receiver or a spectrum analyzer, optimized for both tower and turntable maximums. This action optimizes two orientations (both tower and turntable), but either orientation can be ignored. The maximum value is stored along with the angle and height that were determined during execution.

NOTE: If you are **only** using **one** positioner, then you can use the actions *Scan Range* or *Fast Scan*.

Freq – Analyzer 1 Tab

This tab allows the operator to select the range of parameters over which measurements will be taken.



- **Start/Stop Frequency** — Enter an appropriate number and select the correct units.
- **No. of Ranges** — When this action is executed, the frequency span between the start and stop frequencies is divided into the number of ranges specified.

  Unique scans are made of each frequency range with the results collected into a single data element. The number of points in the data file is the product of the number of ranges and the number of measurement points per scan for the instrument connected to this action.

  In designing the measurement, knowledge of the capabilities of the analyzer/receiver is critical. If the spectrum analyzer has a 401-dot resolution, then when you do a scan the instrument will only send 401 points, regardless of the bandwidth you specified in the parameters. If you are making a measurement from 30 MHz to 1000 MHz with a 120-kHz bandwidth, you need 8,083 points.

  If the analyzer only reports 401 points, you need to break this measurement into 20 ranges to obtain a reasonable accuracy. For this reason, you might want to design a scan with broad bandwidths, calculate the peaks, and run a measurement of the peaks only with the narrow bandwidth. Experimentation by the user is strongly recommended.

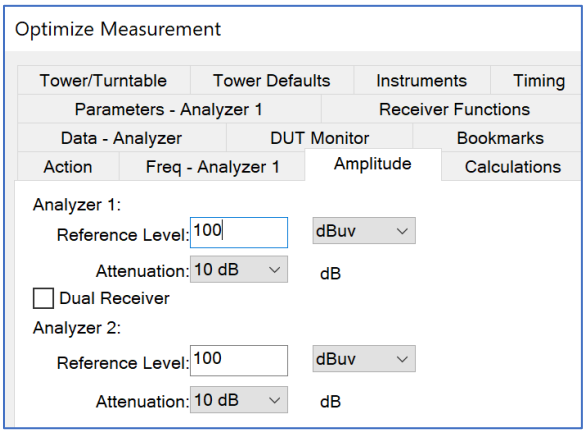
- **Scaling** — The **Log** Scale and **Linear** Scale options are used to select the appropriate frequency range scaling. If the range is 30 MHz to 3 GHz with log scaling selected, the frequency range will be broken into two types of sub ranges. For each decade, you will have the number of ranges specified.

  In this example, we would break the 30 MHz to 300 MHz range into 10 separate scans. The range from 300 MHz to 3 GHz would be split into the same 10 ranges, but we would terminate the measurements on the last scan up to 1 GHz. The default is linear scale.

- **Automatic Pause** — Automatically pauses the test between each sub range. If only one range or sub range is defined, it will not pause.

Amplitude Tab

This tab is similar to other amplitude tabs and will not fully be described here. See the *Measure Range Amplitude Tab* for the full description.

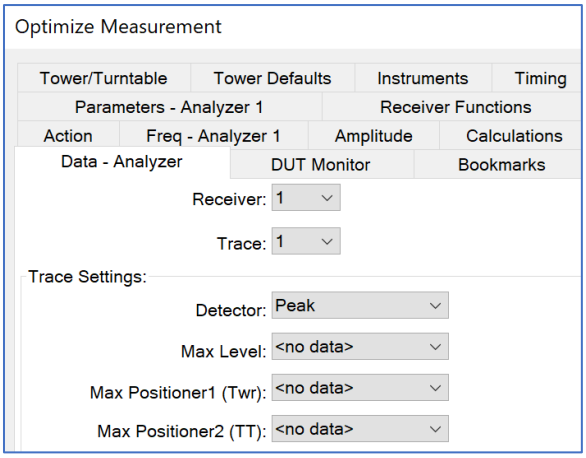


- **Dual Receiver** — This is a special feature that allows for simultaneous measurements of two different analyzers/receivers controlling two different towers during a rotation of the turntable. If selected, additional tabs appear.

  When this option is selected the software will set up both analyzers/receivers at the same time. Move the towers to their start positions and then start the turntable in motion. Both analyzers/receivers will be triggered and read. The towers will then be moved to the next step. The towers are synchronized and have the same number of steps.

Data - Analyzer Tab

This tab allows the user to specify the storage data elements for the maximum value and the related tower and turntable positions for this maximum.



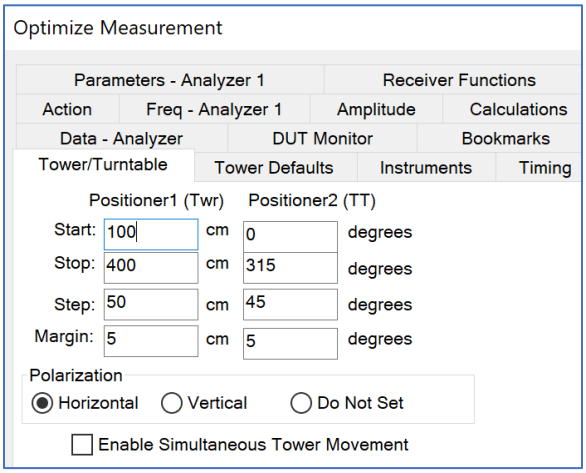
NOTE: If the Max Level data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Max Level** — Identifies the data storage element to be used to record the maximum value. When the action is executed, the data element is resized to the product of the number of ranges and the number of measurement points per scan for the instrument connected to this action. If no errors are detected during the execution, the data is marked as valid and the frequency and amplitude data are set for each point measured.

- **Max Positioner1 (Twr)/ Max Positioner2 (TT)** — Identifies the data storage element to be used to record the height or angle of the maximum value. This is dependent upon whether you are using a tower or turntable controller as Positioner1.

Tower/Turntable Tab

This tab determines the range of motion for the two positioners. If the positioner is a tower is then all references are in centimeters. If the positioner is a turntable is then all references are in degrees.



- **Start** — Enter the appropriate minimum position for each positioner. Typically, towers start at 100 cm and turntable motions are defined to start at 0 degrees.

  TILE! does not support continuous rotation turntable movements. All movements are swung around a 360-degree arc and then reversed.

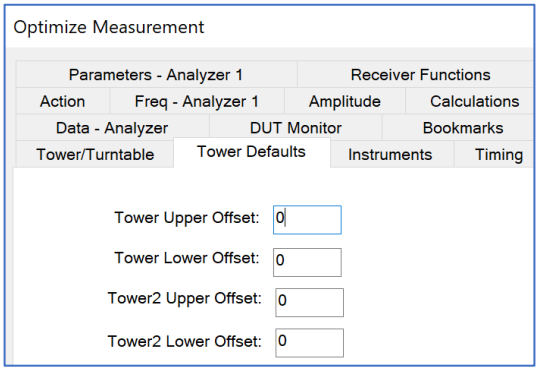
- **Stop** — Enter the appropriate maximum position for each positioner. For three 10-meter emissions measurements, a default height of 400 cm is defined. For 30-meter measurements, this would be set to 600 cm. Turntable maximums are set to 359 or 360 degrees.

- **Margin** — Determines the number of degrees or centimeters prior to the desired position where the positioner will be commanded to stop. This value is the slippage of the positioner. Sending the stop command early will allow the positioner to settle to the desired position. This setting will be site specific and test specific due to the differences between positioners and the equipment placed upon them.

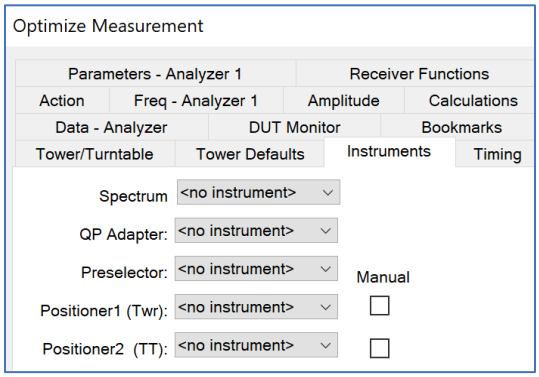
- **Polarization** —If the positioner does not have control of the polarity, no action is performed.

Tower Defaults Tab

- **Tower Upper/Lower Offset** — When rotating an antenna from horizontal to vertical, some antennas will have the potential to touch the floor. The offset value (upper/lower) is the physical amount the tower will move up or down prior to rotating the antenna polarity to avoid this potential problem.



Instruments Tab

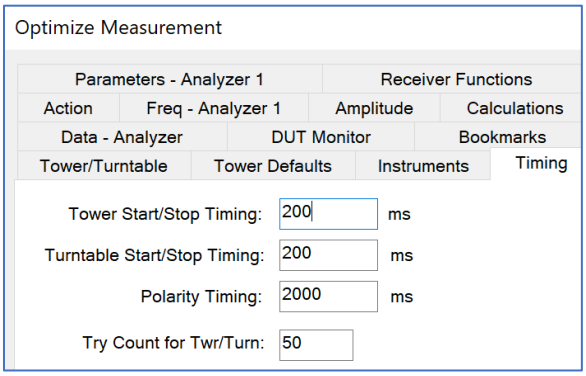


This tab is similar to other instruments tabs and will not fully be described here. See the *Measure Range Instruments Tab* for the full description.

NOTE: If the Spectrum Analyzer, Positioner 1, and Positioner 2 instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Positioner1 (Twr) /Positioner2 (TT)** — The choice of positioner determines whether angular and/or height information is captured during the execution of this action. If Check in Continuous Mode is not set on the *Parameters - Analyzer 1 Tab*, then the logic of this action is to perform the full range of Positioner1 steps prior to stepping to each position of Positioner2. If Check in Continuous Mode is set, then the Positioner2 is rotated completely for each step in Positioner1.

Timing Tab



Use the settings on the Timing tab to improve control of large turntables and older towers.

- **Tower/Turntable Start/Stop Timing** — Between the start command to the tower/turntable and the first read, the system will pause this time to allow the tower/turntable to start its motion. This is a major concern on very large turntable where the start momentum is large.

  Once a stop command is issued, this value is checked to determine a wait time before any other commands are sent to either the tower or turntable. If the braking system responds slowly, or if it has no braking system, this parameter ensures that a command to change direction is not issued before stopping has completed. On some older positioners, if you send a stop command and immediately send a change of direction command, the second instruction will be ignored if the positioner is still in motion.

  When a start command is sent, some towers and turntables exhibit brush noise during the initial start surge. The start/stop timing value is an amount of delay that will be put between the start command and the next instrument read/motion. This will ensure that noise is avoided as much as possible. Some larger turntables take a significant amount of time to start. This delay will ensure that a count error will not take place. A count error happens when the positioner is read more than the amount specified in **Try Count for Twr/Turn** and there has been no motion detected.

- **Polarity Timing** — This setting determines how long the software will wait after issuing the polarization command (either to go to the vertical or horizontal orientation) before any motion is started. Most towers have no feedback to determine whether polarization has executed, or what polarity they are in, so this command allows the user to set a delay time prior to any further motion of the tower.

  Most towers with polarity under remote control use an air control to change polarity. The amount of time it takes for the antenna to rotate from horizontal to vertical, or vice versa, is dependent upon the weight of the antenna and the air pressure. We want to ensure that the polarity has

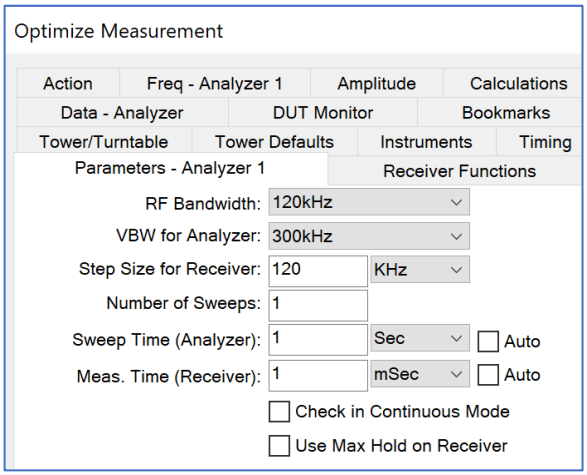
completed its movement prior to any other movement or measurement. Polarity timing is a value in milliseconds that reflects how long it takes the positioner to execute a polarity movement. The software will pause this period before continuing its execution.

- **Try Count for Twr/Turn** — When an up/down or counterclockwise/clockwise motion command is sent, the software reads the controller looking for the current position. If it does not change in a certain number of reads, the tower/turntable is assumed to be non-functioning. This try count value allows the user to control how many times the controller is read before this error is declared.

Parameters - Analyzer 1 Tab

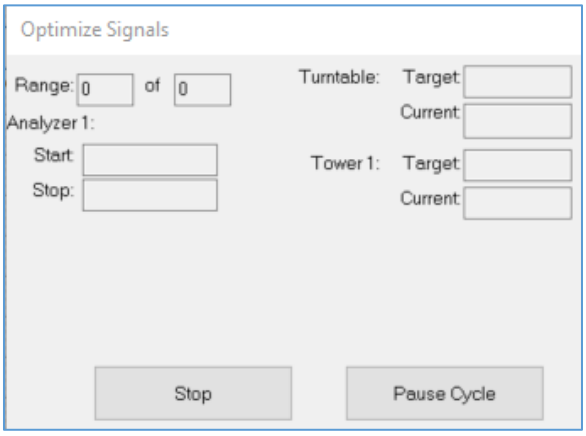
This tab is similar to other parameter tabs and will not be described here. See the *Measure Range Parameters Tab* for the full description.

NOTE: Make sure that the *instrument timeout* is higher than the sweep time.



Dialog Box

This dialog box appears when PRE-SCAN is executed and provides visual information on the current range that is being performed. The start and stop frequencies for this range are displayed. Tower or turntable target and current position information are displayed in the appropriate boxes.



- **Stop** — This button gives the user the ability to halt execution of the test at this point. When pressed, the system will complete the current sequence of GPIB commands and then halt execution. This is to ensure the GPIB is not left in an inoperable condition.
- **Pause Cycle** — This button will pause the test between each movement of the positioner. It will not abort the test, it will only cause a pause until the user continues the test.

Scan Range Measurement

Scan Range is like the *Pre-Scan* action **except** that it will **only** utilize **one** positioner, either a turntable or a tower. It is used to perform a measurement of a range of frequencies, using either a receiver or a spectrum analyzer for the primary instrument with these measurements optimized for either tower or turntable maximums. This action **only** optimizes **one** orientation (either tower or turntable).

NOTE: If the Max level, Spectrum Analyzer, and tower/turntable instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

Fast Scan Across TT/TWR

The Fast Scan Across TT/TWR action is used to measure a range of frequencies using a receiver or spectrum analyzer instrument, optimized for tower and turntable maximums. The analyzer or receiver is constantly in Max Hold for each frequency range while the tower and turntable are rotated through a complete sequence. This action is like the *Pre-Scan* action **except** that it **only** uses **one** positioner.

NOTE: If the Peak data and Receiver instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

Peak Type Measurements Actions

Peak measurements are performed to measure discrete points, such as a quasi-peak, to find a single frequency value. Types of outputs from these are peak, quasi peak, and average.

Measure Peaks is the most basic action and does not use positioners; it is mostly applicable to conducted emissions. The *Scan Peaks* action is used with turntable and tower positioners. The *Scan Peaks PSA* action is specific to the PSA series spectrum analyzer from Agilent.

Scan Peaks and Scan Peaks PSA

These points are usually derived by performing math discrimination on a range of measurement values, but can include fixed frequency points from a table. This is particularly helpful when measuring known harmonics or other fixed-point considerations.

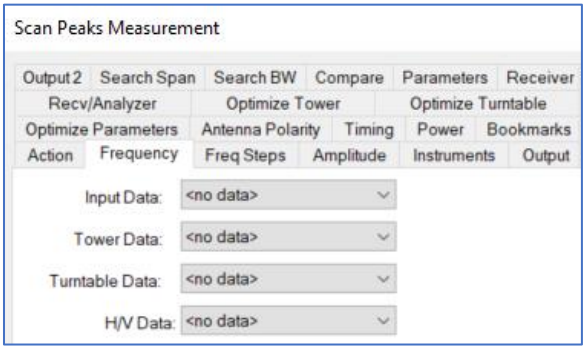
This action has three distinct modes. The first is Search. During this mode, the software sets up the initial frequency settings for the analyzer/receiver. If this choice is also selected in the *Search Span tab*, it will open a search span to find the highest signal in this range, and center the analyzer to this frequency. This is an important step when you have frequency inaccuracies, which needed to be accounted for during the test.

After the Search mode, the software will optimize the tower and turntable to ensure we are locked on the highest signal. It will set the turntable to the default position and then search the turntable for a maximum. After the maximum turntable is found (and the turntable set at this position), the tower is scanned for maximum value.

After searching both tower and turntables, using the selected choices on the *Optimize Parameters tab*, the software will return these positioners to the position consistent with the highest signal. Then the Measurement mode will take final readings of peak/QP/average as specified on the *Output tab*.

The PSA version of Scan Peaks was designed to work with the Agilent PSA Series spectrum analyzer so that it would measure at the marker the peak, QP and average values.

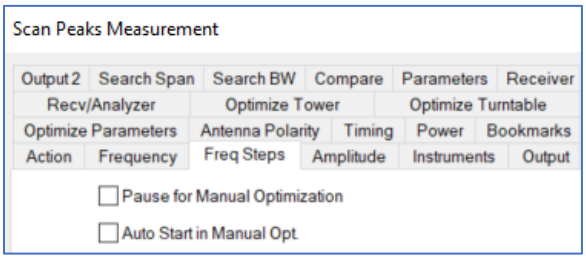
Frequency Tab



This takes a set of peaks from a data element along with the related positioning information and performs designated peak, quasi peak, and average readings on these frequency points.

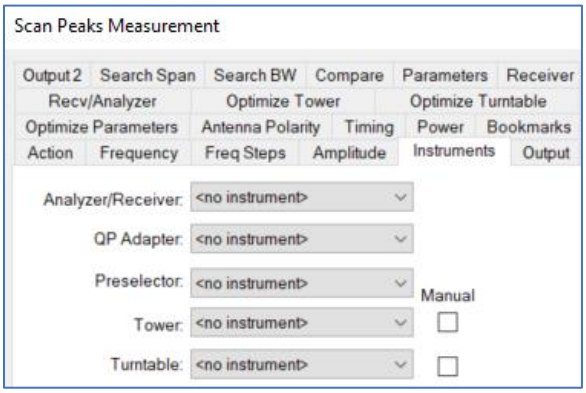
- **Input Data** — Determines the frequency points of interest. This data table can be derived from an existing range measurement through an equation function, by loading a set of data points using a spreadsheet or ASCII text editor or by using the *Direct Entry action*.
- **Tower/Turntable/H/V Data** — The tower, turntable, and H/V data elements contain position information related to the Input Data element. Any of these may be ignored except Input Data. If the position information is available here, then the optimization process can include the Partial Optimize feature.

Freq Steps Tab



- **Pause for Manual Optimization** — When selected, causes the measurement to stop at every frequency and optimize the reading. This is particularly useful when performing measurement to FCC compliance where the operator is required to optimize the signal by moving the cables, and so on.
- **Auto Start in Manual Opt.** — When selected, steps the operator through the measurement process, taking the readings automatically at each frequency point.

Instruments Tab



This tab is similar to other instruments tabs and will not fully be described here. See the *Measure Range Instruments Tab* for the full description.

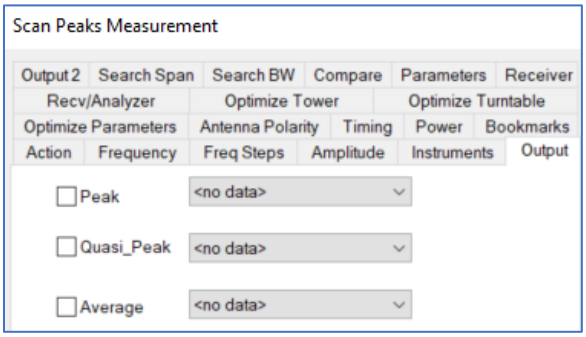
NOTE: If the Analyzer/Receiver instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Tower/Turntable** — Identify tower and turntable controllers as needed or when available. If no tower is available, or it is not GPIB controlled, select the Manual check box to configure an operator interrupt.

  This will allow you to manually set the tower position during the test. We recommend that you create an instrument in the Instrument window even if you are using a manual tower/turntable. This will allow you to upgrade the action in the future with the minimum of changes.

- **Manual** — If the tower or turntable does not support GPIB operation, select the check box to allow operator interruption during the test.

Output Tab



On this tab you choose the type of test and data element to store the results. The selection on this tab determines the combination of detectors for each frequency measurement.

The data element must be defined as a measurement type in the Data window. You can select any or all of these detector types. If selected, a data element must also be selected. If more than one detector is chosen, the tests are always performed in the sequence shown on this tab.

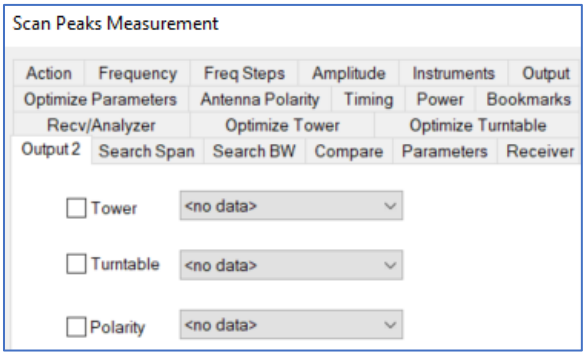
This tab is similar to other output tabs and will not fully be described here. See the *Signal Discrimination Output Data 1 Tab* for the full description.

Output 2 Tab

This tab allows for identification of the tower position, turntable position and a Text data element that will identify the polarity (either H or V) of the final measurement.

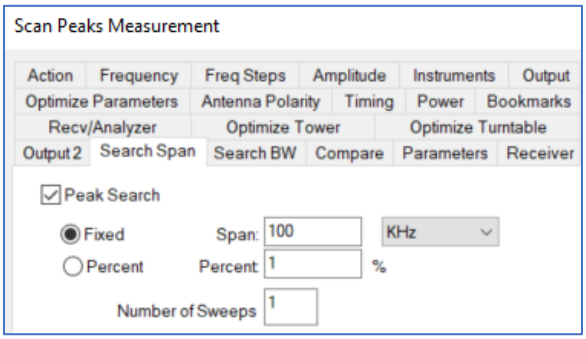
The tower and turntable will identify all Measurement Type data elements. The Polarity data element must be a word type and only this type will be identified in the list.

This tab is similar to other output tabs and will not fully be described here. See the *Signal Discrimination Output Data 2 Tab* for the full description.



Search Span Tab

When performing peak measurements, one problem is the need to ensure the exact peak is found when repeating prior readings. This is especially true if the bandwidth is different than that used with the original measurements.



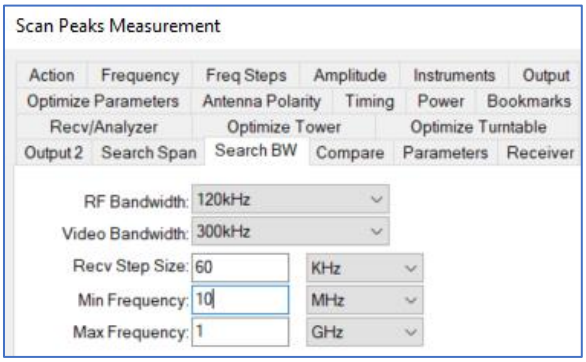
- **Peak Search** — Determines whether the system will attempt to verify the peak by searching around the original point. If you select this check box, you need to specify the band around which you want to search.

  For example, if you have two signals within 2 MHz of each other you might specify a band search of less than 1 MHz. This will ensure that you do not identify the wrong signal when measuring these points. Other options include setting the span to 10 times the bandwidth. Choose a span consistent with the expected signal types and ambients.

- **Span** — Specify a frequency span for the analyzer during peak searching. This fixed span does not vary with frequency. Select the **Percent** option button if you want the span to vary with frequency.

- **Number of Sweeps** — Controls the number of sweeps to be taken while searching for the peak. These settings are only used during the search mode. Choosing a value greater than 1 for the number of sweeps will put the analyzer in Max Hold mode during the measurement sweeps. There are at least two separate sweeps taken to find the maximum signal and move it to the center of the screen.

Search BW



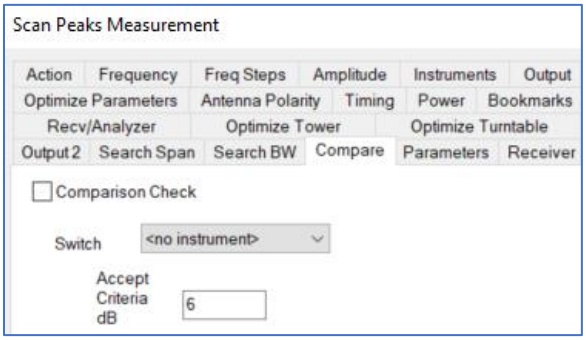
- **RF/Video Bandwidth** — You can select the appropriate RF bandwidth and video bandwidth. These settings are only used during the search mode. You have a completely different set of bandwidths during the optimize mode and the final measurement mode.
- **Recv Step Size** — The step size sets the increment that will be used by the receiver during the measurement scan. Typically, EMI receivers do not actually sweep (as spectrum analyzers do).

  Instead, they take numerous individual measurements and often display this as a continuous sweep. The step size determines the number of points; it is found by considering the stop frequency minus the start frequency then dividing the result by the step size.
- **Min Frequency /Max Frequency** — These values allow you to limit the search span to a defined minimum and maximum frequency. If the requirements do not include any testing above or below a certain frequency, these settings will ensure the software never measures outside the frequency range of interest.

Compare Tab

This is a special feature of the TILE! system that is used on sites with remote, in line reference antennas. When a remote, reference antenna along with a RF switch is available, the software will take measurements from both antennas at each frequency and compare them. If the difference is less than the amount shown in Accept Criteria, the signal will be ignored as an ambient.

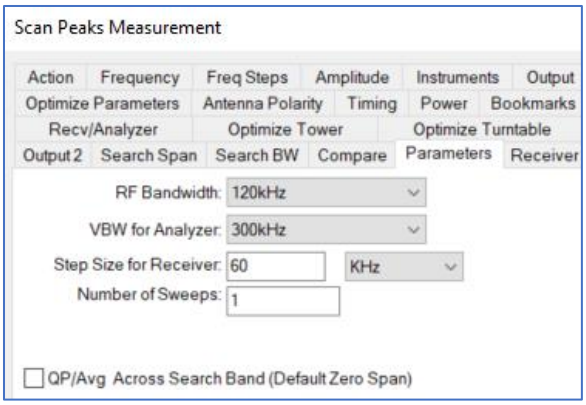
NOTE: Must select the **Comparison Check** box to activate this feature.



- **Switch** — Select an instrument to control the switch.
- **Accept Criteria dB** — This value, in dB, is compared to the difference between the two antennas. If the difference is less than this value, then this frequency is skipped (an ambient signal is assumed). If it is greater than this value, the frequency is measured in full.

Parameters Tab

This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



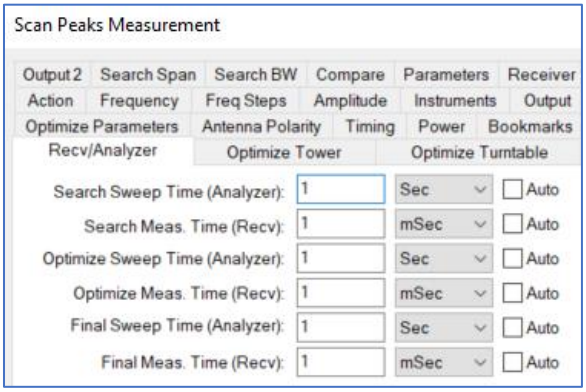
- **QP/Avg Across Search Band** — The default behavior of the QP and average measurements is to go into a zero-span mode and measure the returned value. Sometimes, especially if the signal is frequency varying, you will miss the maximum.

  If you turn on this option, then the QP and average measurements will be done across the defined search span. This takes a little longer, but better captures a moving target.

Recv/Analyzer Tab

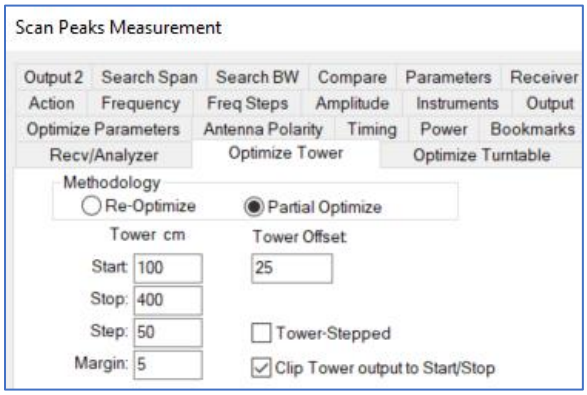
Within this action there are three distinct measurement times used for controlling time for the instrumentation during the various stages of the measurement.

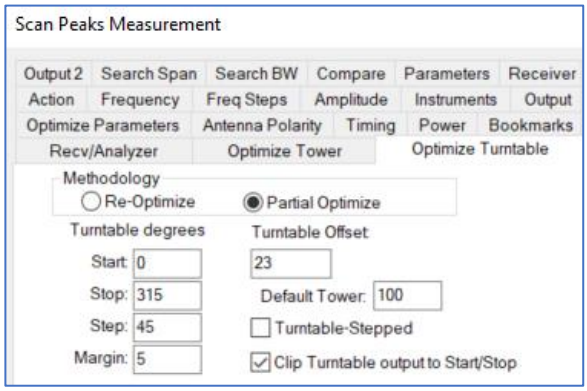
First, a search is performed at fixed tower and turntable height to find the strongest signal. Then, the optimize procedure rotates the turntable and raises the tower to find the angle and height of the maximum emissions. The final measurement is performed at the angle and height where the maximum emission is found.



- **Sweep Time (Analyzer)** — The amount of time used by the analyzer to perform one continuous sweep (from start to stop). The actual time per measurement step is the sweep time divided by the number of points per sweep (set by manufacturer).
- **Measurement Time (Receiver)** — True receivers take individual measurements and then group this information into a sweep result. The measurement time is the time increment per frequency step.

Optimization Tower / Optimize Turntable Tabs





The Optimization process determines how the software will react to height and angle information for each data point. Range measurements are often taken with the tower and turntable rotated through stepped positions. This yields the position at which the maximum was found. Since we do not measure all positions in the range measurements, the Scan Peaks action allows you to scan at a much higher positional accuracy. The Optimization process takes place after the Search mode is performed.

- **Re-Optimize** — Assumes that the earlier readings were not taken with enough tower height, or turntable movement, to adequately predict relative position. In this case, the tower and turntable need to be scanned through their full range of motion to determine the peak signal. This method starts by positioning the tower at the default position. The turntable is then scanned from the start position to the stop position and the peak signal and angle is recorded.

  The turntable is then moved to the position at which the maximum was recorded. The tower is then moved from the start position to the stop position and the peak signal and height are recorded. Then it is moved to this maximum position. The step value is ignored. The margin gives the user control of slippage on the tower and turntable. The stop command is given when motion reaches the limit ± margin.

- **Partial Optimize** — Assumes that the range measurements included the necessary range of motion for the tower and turntable and that these are in the appropriate data elements on the Frequency tab. This may not be the position for the maximum signal due to limited motions, or steps, in the range measurements.

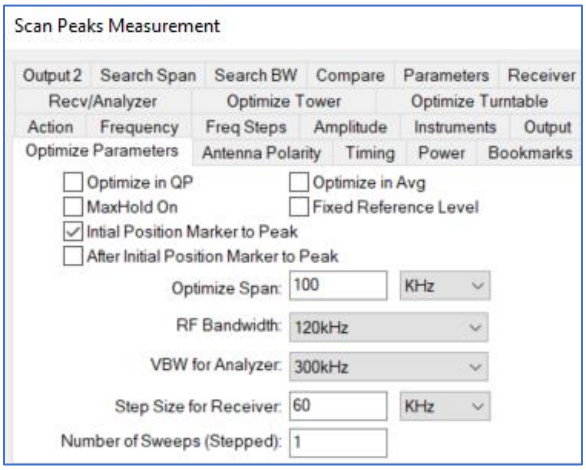
  With this method, the tower and turntable are scanned for the maximum signal over the range of starting positions +/- the offset value. The starting position is read from the input tables for the respective controller.

- **Tower Stepped/Turntable Stepped** — When selected, the tower or turntable will move across the selected range at the defined step interval. In this mode, no continuous readings are taken.

- **Clip output to start/stop** — This feature will clip the output data element to the limits in the Start/Stop fields since some positioners have encoders that will not stop exactly on the number.

Optimize Parameters Tab

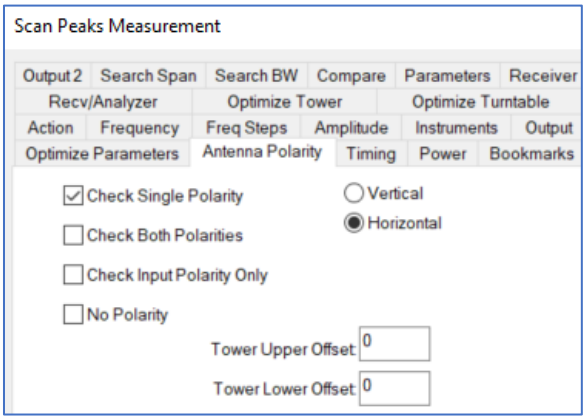
During the Optimization process, the tower is scanned to find the maximum signal. This tab gives you control of certain parameters during this process that will improve the measurement accuracy.



- **Optimize in QP** — Normally the optimization is done in peak detector mode. This choice will turn on the quasi peak detector.
- **Optimize in Avg** — Normally the optimization is done in peak detector mode. This choice will use an average peak method instead.
- **MaxHold On** — In normal mode, the screen is cleared between each sweep and a value is read for comparison purposes. With this check box selected, the analyzer is put in Max Hold and the readings will record the maximum.

  Note that if the EUT is the strongest signal then this setting works well, but if there are strong ambients, there is a possibility of an ambient saturating the Max Hold readings. This will cause an error in the height evaluation.
- **Fixed Reference Level** — When this check box is selected, the reference level will remain fixed. If deselected, which is the default, and then the reference level will be reset whenever a strong signal is within 10 dB of the reference level.
- **Optimize Span** — The span can be set separately from the search and final measurement spans. This allows you to exclude ambients when the tower is in motion.
- **RF Bandwidth / VBW for Analyzer** —Not all instruments will be able to use each of these settings, but the instrument driver will pick the closest setting that is valid for the specified receiver. For example, many older EMC receivers will only do 9 kHz and 120 kHz RF bandwidths. For all bandwidths at or above these settings, the driver will automatically pick the appropriate setting.
- **Step Size for Receiver** — Sets the increment that will be used by the receiver during the measurement scan. Typically, EMI receivers do not actually sweep (as spectrum analyzers do). Instead, they take numerous individual measurements and often display this as a continuous sweep. The step size determines the number of points; it is found by considering the stop frequency minus the start frequency then dividing the result by the step size.

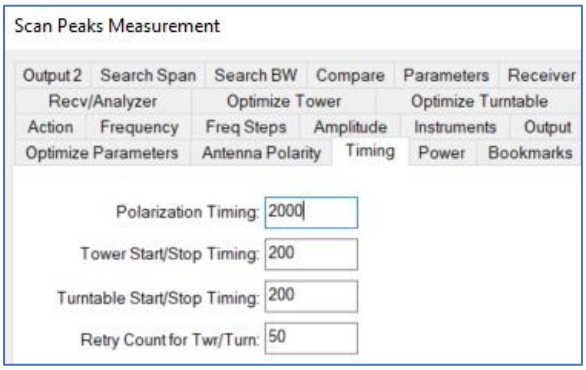
Antenna Polarity Tab



This tab is used to set the respective polarity for the antenna during execution of this action.

- **Check Single Polarity** — With this check box selected, you also specify the vertical or horizontal orientation desired for the action. The antenna will be set to this polarity at the beginning of the execution and left there for the complete sequence.
- **Check Both Polarities** — Performs a complete sequence, including tower and turntable positioning, and stores the maximum value. The polarity will then be rotated and a complete sequence again executed. When this is complete, the maximum values will be compared. The highest reading and its tower position, turntable position, and polarity will be saved.
- **Check Input Polarity Only** — When this check box is selected, the antenna will be set to the polarity specified in the H/V Data list on the *Frequency tab* at the start of each frequency step. The polarity can be different for each frequency.
- **Tower Upper/Lower Offset** — When rotating an antenna from horizontal to vertical, some antennas will have the potential to touch the floor. The offset value (upper/lower) is a physical amount the tower will move up or down prior to rotating the antenna polarity to avoid this potential problem.

Timing Tab



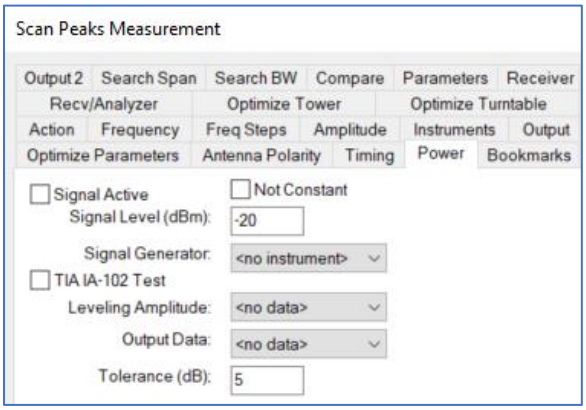
This tab improves control of large turntables and older towers.

- **Polarization Timing** — Different towers execute polarity movements at different speeds. To ensure accurate testing, it is essential that the polarization movement be completed prior to any movement of the tower or turntable. This timing allows the user to match the execution to the characteristics of their tower.
- **Tower/Turntable Start/Stop Timing** — Specifies a minimum timing interval between the stop motion and any other movement command; this will ensure accurate direction and motion.

  On some positioners (both towers and turntables), if they are issued the stop command and then immediately issued a change of direction command, they will ignore the second command until the motion has completed. The software will report an error because the motion commanded did not take place. Setting this value to a higher number will ensure that motion has stopped prior to sending any other command to the tower or turntable.
- **Retry Count for Twr/Turn** — When the start command is issued to a tower or turntable, the software then begins reading the current position to determine when the limit is reached. This

parameter controls how many times the positioner will be read (if the position does not change) before an error message is displayed.

Power Tab

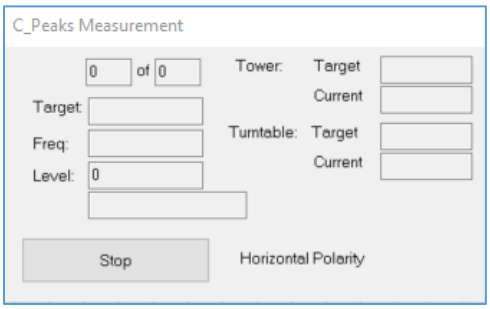


When measuring peak signals, there are times when the user might need to inject a matching frequency signal from a signal generator. There is also a TIA specification, 1A 102, which performs a signal substitution test. The Power tab allows configuration of either of these options.

When power is active, a signal generator is tuned to the current frequency with the specified amplitude. This allows you to perform attenuation measurements, calibrations, and other tests that require an input signal during a measurement.

- **Signal Active** — Selecting this check box will cause the software to address the signal generator specified in the Signal Generator list, setting the amplitude and frequency. The amplitude is usually a constant value like -20 dBm.
    - **Signal Level (dBm)** — The signal level can be set to a fixed value for all frequencies. If the signal generator is active, then the signal generator will be tuned to the current test frequency and the amplitude turned on at the specified value.
- **Not Constant** — If this check box is selected, then you can enter the name of a data element that will contain the desired amplitudes and frequencies.
    - **Signal Generator** — Select the signal generator from the instrument list.
- **TIA IA-102 Test** — When this check box is selected, this feature will compare the reading to the selected amplitude from the Leveling Amplitude list and step the signal generator up or down to achieve a matching level. The signal generator level is stored in the Output Data element.
    - **Leveling Amplitude** — Select a date element that has the desired RF level.
    - **Output Data** — Select a data element to store the signal generator level during this test.
    - **Tolerance** — The margin of error between the actual reading and the desired level (as selected from the **Leveling Amplitude** list).

Dialog Box



This dialog box appears when Scan Peaks is executed and gives the user visual information on the point that is currently being measured. The target and current frequencies are displayed as well as the amplitude at the current frequency. Tower or Turntable target and current position information are displayed in the appropriate boxes.

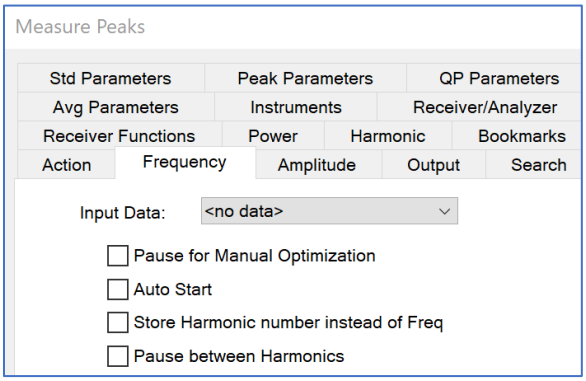
The **Stop** button provides the ability to halt execution of the test at this point. When clicked, the system will complete the current sequence of GPIB commands and then halt execution.

This is to ensure the GPIB is not left in an inoperable condition. The current polarity is noted on the dialog, for reference purposes.

Measure Peaks

Measure Peaks provides the ability to perform measurements on a selected set of frequency points, using a combination of peak, quasi peak (QP), and/or average detectors. These points are usually derived by performing peak selections off a range of measurement values, but can include fixed frequency points from a table. This is particularly helpful when measuring known harmonics or other fixed-point considerations.

Frequency Tab

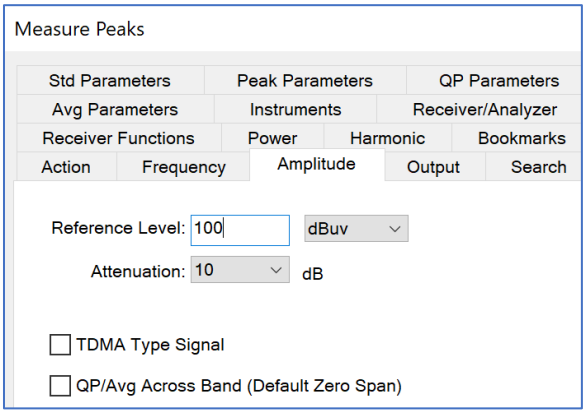


This action takes a set of peaks from a data element and performs a peak, QP, and/or average reading for each data point. The choice of performing peak/QP/average measurements depends upon the settings in the Output Tab. Select the data element that contains the points of interest.

NOTE: If the Input Data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Input Data** — Select a data element.
- **Pause for Manual Optimization** — During the measurement process, it may be required to optimize the signal level at the specified data points. This selection allows the operator to accomplish this. It can be helpful when doing a MIL STD CE 101 type test or when you want to manual search for a signal (for example, when using a small probe on a circuit board). You might call this the appropriate setting if you want to shake the cables.
- **Auto Start** — Automatically steps the measurement through the desired frequency points. Selecting the **Auto Start** check box **overrides** the **Pause for manual optimization** check box, but the Measure Peaks dialog box gives the operator the ability to halt the test and then step through frequency points through manual selections on the dialog box.
- **Store Harmonic number instead of Freq** — When Harmonic is active on the *Harmonic tab*, this determines whether to store the frequency or step number in the frequency column of the data element.
- **Pause between Harmonics** — When Harmonic is active on the *Harmonic tab*, this will pause execution between each frequency step so that the operator can manually adjust the input-signal generator level (or change channels on the TV receiver).

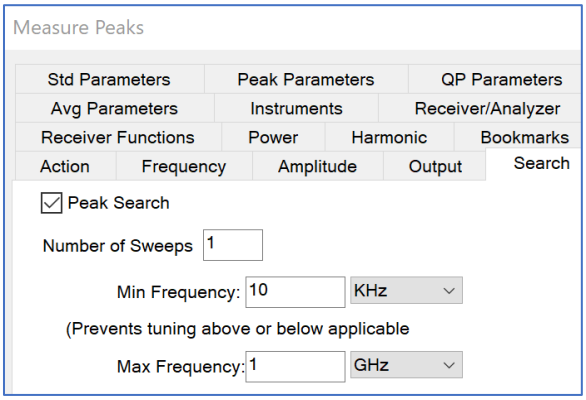
Amplitude Tab



This tab is similar to other amplitude tabs and will not fully be described here. See the *Measure Range Amplitude Tab* for the full description.

- **TDMA Type Signal** — When this check box is selected, the software will set the marker to follow the peak during each sweep. TDMA signals are pulse type, which appear to change frequency each time we take a scan. Another method to handle this would be to set the number of sweeps large enough to ensure that you capture the signal.
- **QP/Avg Across Band** — When performing the QP and average measurements, the default setting is to go to zero Span (for maximum accuracy). When this check box is selected, the software will perform a sweep across the current span with the detector and do a marker to peak, storing the maximum value in the span.

Search Tab



When performing peak measurements, one problem is determining if we are on the signal of interest. Often, you will take a measurement scan across of range of frequencies (using measure range or one of the other emission actions); these readings are wide band scans.

Most spectrum analyzers have significant frequency drift when comparing wide band measurements to vary narrow band measurements (and by this, we mean the bandwidth of the scan).

When you are searching for peak signals, you can determine whether to search for a closer peak. If the Peak Search check box is selected, then the software will perform a peak search routine prior to taking final measurements.

- **Peak Search** — When this box is selected, you then specify the band around which you want to search. For example, if you are taking measurements that will typically see two signals within 2 MHz of each other you might specify a band search of less than 1 MHz. This will ensure that you do not identify the wrong signal when measuring these points. At the same time, this will ensure that the peak signal is properly centered on the screen prior to measurement.
- **Number of Sweeps** — Select an appropriate number of sweeps for the search mode. This setting is different from those used during the final readings, which are set on the *Parameters tab*. If more than one sweep is specified, then the analyzer will be put in Max Hold and the number of sweeps executed.
- **Min Frequency/Max Frequency** — These settings allow you to ensure that the analyzer settings do not exceed a minimum or maximum frequency.

  For example, assume you are measuring conducted emissions from 150 kHz to 30 MHz and have a suspect at 151 kHz. With the search span set to 20 kHz, it would normally open a search span of 141 kHz to 161 kHz. Given the noise floor of some analyzers, it would be possible to walk

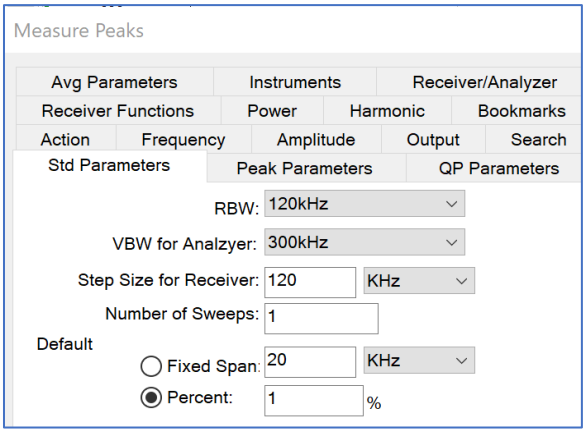
the noise floor down below 150 kHz. Setting the minimum frequency to 150 kHz would limit the search span to 150 kHz to 161 kHz. This would ensure that the peak was found in this span.

A second consideration you are faced with is doing testing outside the range that the standards are written. In this case, setting the minimum and/or maximum frequency will ensure that the searches stay within the frequencies of interest.

Std Parameters Tab

This tab is used to specify the instrument setup for this measurement.

This tab is similar to other parameter tabs and will not be described here. See the *Measure Range Parameters Tab* for the full description.

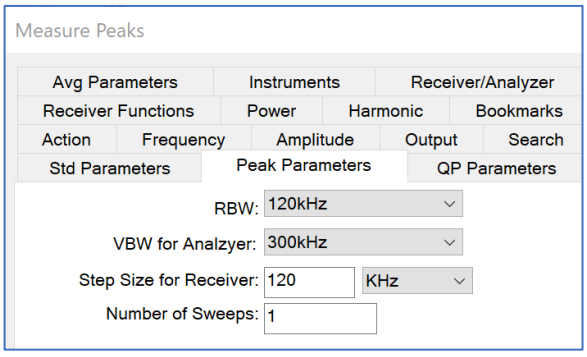


- **Default** — This setting gives the user the choice of using a fixed span or using a span that varies with frequency. The two methods each have their advantages, but the use is usually related to the type of EUT being tested. When testing transmitters, a fixed span is usually appropriate. When testing unknown noise, a percentage will work best.
    - **Fixed Span** — When selected, you can enter the value for the span. Select frequency and unit.
    - **Percent** — When using a floating span, the span is a percentage of the current frequency. Enter a value for this, generally in the range of 1 to 2 percent will work best. Too large a setting can cause the fundamental of a single signal to be measured multiple times with the harmonics lost due to the wide span.

Peak Parameters Tab

The parameters on this tab are used **only** for the final peak measurement after the search has been completed.

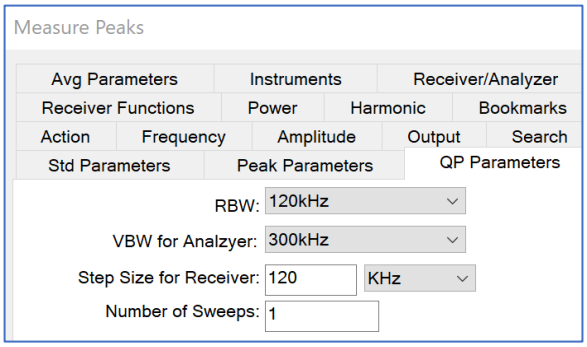
This tab is similar to other parameter tabs and will not be described here. See the *Measure Range Parameters Tab* for the full description.



QP Parameters Tab

The parameters on this tab are **only** used for the final QP measurement after the search has been completed.

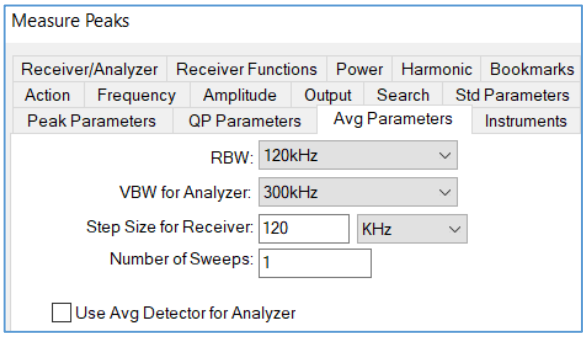
This tab is similar to other parameter tabs and will not be described here. See the *Measure Range Parameters Tab* for the full description.



Avg Parameters Tab

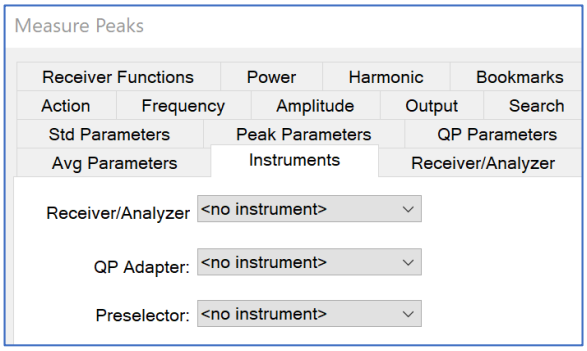
The parameters on this tab are **only** used for the final average measurement after the search has been completed.

This tab is similar to other parameter tabs and will not be described here. See the *Measure Range Parameters Tab* for the full description.



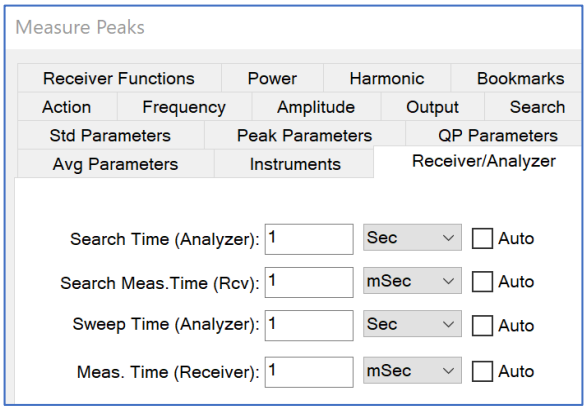
Instruments Tab

This tab is similar to other instruments tabs and will not fully be described here. See the *Measure Range Instruments Tab* for the full description.



Receiver/Analyzer Tab

This tab is similar to other receiver/analyzer tabs and will not fully be described here. See the *Scan Peaks Recv/Analyzer Tab* for the full description.

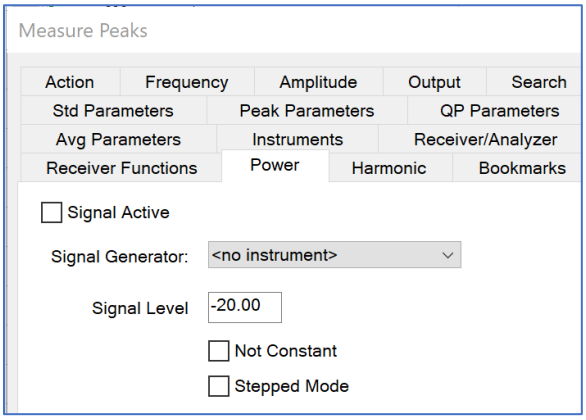


- **Search/Meas. Time (Analyzer/Receiver)** — The time value for the measurements on the analyzer/receiver while we are in a search mode looking for the peak signal.

Power Tab

This tab is similar to other power tabs and will not fully be described here. See the *Scan Peaks Power Tab* for the full description.

NOTE: Must select the **Signal Active** box to activate this feature.



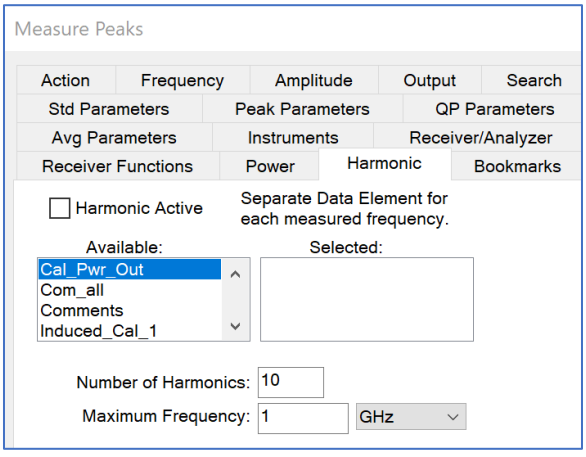
- **Not Constant** — If this check box is selected, a data element can be selected for amplitude. This data element will be read for the current frequency and an appropriate value set for the output.
- **Stepped Mode** — This option is not currently active.

Harmonic Tab

This tab is used when you need to do harmonic measurements on specific frequencies and store their information in separate data elements.

NOTE: Must select the **Harmonic Active** box to activate this feature.

With this option set, the user needs to specify an Output data element for each frequency in the test. The peaks for the input frequency and each of the specified harmonics will be measured and stored in the Sequential data element.



- **Available/Selected** — The Selected column includes the data elements in the order that they will be filled. The first selected data element will be used to store the first frequency in the input data and the harmonics of that

frequency. Then, the second data element will be used to store the second input frequency and its harmonics.

It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

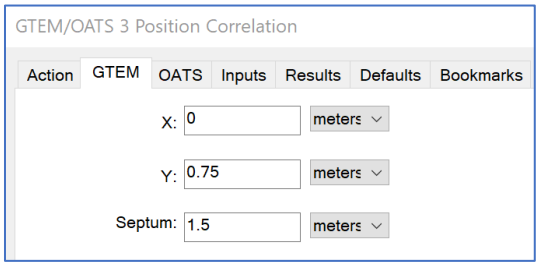
- **Number of Harmonics** — Specifies the number of harmonics to be tested. A value of 1 would indicate a measurement of the fundamental only. A value of 10 would measure the fundamental and the nine harmonics of the fundamental.
- **Maximum Frequency** — Limits the upper frequency for harmonic measurements. For example, if the spectrum analyzer was limited to 1 GHz you could use this value to limit harmonics above 1 GHz. If you had specified 10 harmonics, as the fundamental went above 100 MHz, the software would begin measuring fewer harmonics.

GTEM Actions

These actions will calculate the GTEM correlations and position the EUT.

GTEM/OATS 3 Position Correlation

GTEM Tab

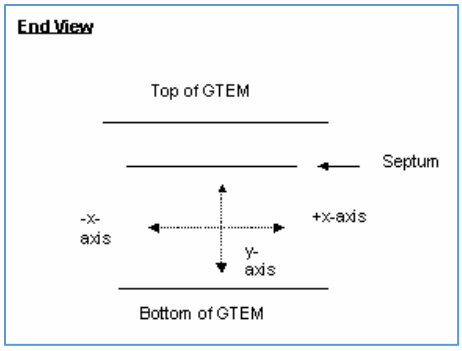


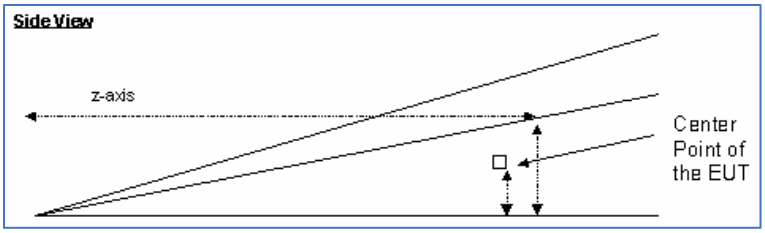
The GTEM tab describes the physical dimensions of the GTEM cell that was used for the input data measurements. As a short explanation of the GTEM physical references, the GTEM is a pyramidal shape in which the x, y, and z-axes describe the center point of a EUT in the GTEM. To illustrate this point, consider the following views of the GTEM:

The **X-axis** is parallel to the width of the septum. Since all measurements are referenced the center septum, consider the x and +x to be the offsets from the centerline of the septum.

The **Y-axis** is the height of the septum from the floor to the septum. This dimension is a positive number.

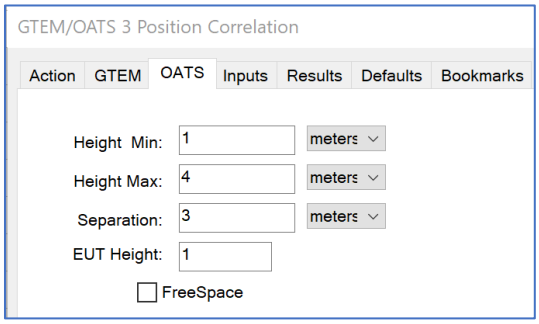
The **Z-axis** describes the length of the GTEM. Because of the 90-degree relationship between the Y-axis and Z-axis and the fixed angle, the length of the Z-axis can always be determined by knowing the height of the Y-axis.





- **X** — Enter the offset from the centerline of the EUT. In an optimum setup the EUT would be positioned in the center of the septum width; this is position 0. If the EUT requires a slight offset to either side, enter the offset; an offset to the left of the centerline is a negative value, and an offset to the right is a positive value.
- **Y** — Describes the physical height from the floor of the GTEM to the center of the EUT. If you are not sure what to enter for the Y value, enter a value that represents half the height of the septum for the position of the EUT.
- **Septum** — Since the geometry of the GTEM is precise, it is possible to define both the septum and Z-axis lengths by mathematical reference to the Y-axis. In this field, enter the height from the floor to the septum at the position where the EUT is to be placed.

OATS Tab



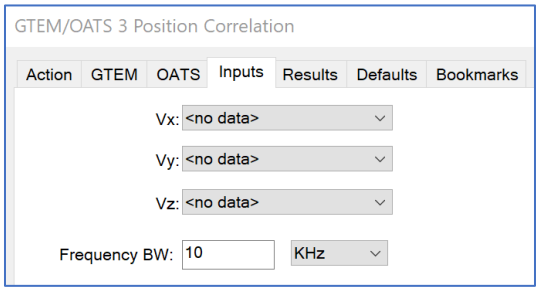
The correlation routine converts GTEM based information to the theoretical field strength that would exist over a ground screen. In calculating this value, the physical dimensions of the OATS are required. The OATS tab allows you to specify which OATS configuration is appropriate for the correlation.

The scan range and separation are the pertinent elements. The standards specify these numbers. Typical pairs are 1–4 meter scan height with a 3-meter separation; 1–4 meter scan height with a 10-meter separation; and 2–6 meter scan height with a 30-meter separation.

- **Height Min/ Height Max** — Enter the minimum/maximum antenna height for the scan range on the OATS. This information is specified by the emissions standards.
- **Separation** — Enter the separation distance for the OATS. This information is specified by the emissions standards.
- **EUT Height** — Enter the height of the EUT. This information is specified by the emissions standards.
- **Free Space** — When this check box is selected, this causes the correlation routine to skip the OATS correction and only combines the three vectors into a free space equivalent value.

Inputs Tab

The input fields identify the three Measurement data elements for the GTEM orthogonal readings. These must be matched; for example, the EUT was in an identical setup with a 120-degree orthogonal rotation.

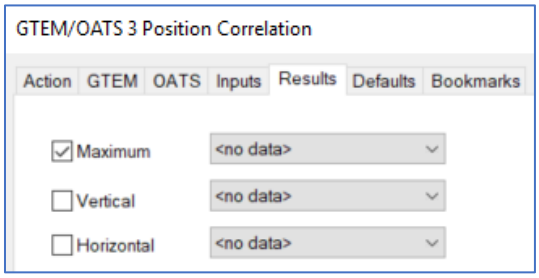


The three readings need to have identical scan ranges, bandwidths, reference levels, and sweep times to ensure accuracy. As with any mathematical calculation, if you put garbage in, you get garbage out. It is essential to ensure the accuracy of the three readings.

- **Vx/Vy/Vz** — This element will be the same as was used during the X/Y/Z orientation measurement on the flowchart.
- **Frequency BW** — There is a slight possibility, especially when comparing peak/QP/avg readings, that the frequencies in the three data elements are not identical. This setting defines how much mismatch there can be before an error will be reported.

Results Tab

The correlation analysis yields three data elements. You have the option of recording each element. The basic correlation yields the maximum. This value is the straight correlated value between the OATS and the GTEM.

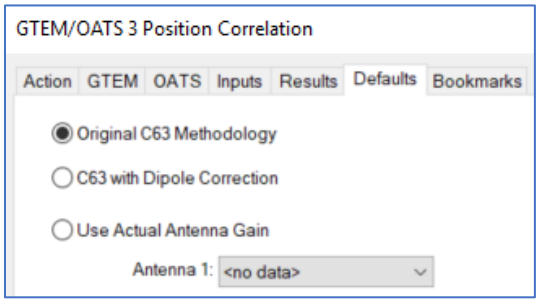


However, certain standards require the maximum signal in the horizontal and in the vertical antenna orientation. These values are derived in the mathematics of the correlation algorithm prior to calculating the maximum.

- **Maximum** — If the maximum value of the correlation results is desired, select an appropriate data element to store these values. This is the default position and there are few reasons not to use this data.
- **Vertical/ Horizontal** — Select an appropriate data element to keep the maximum signal in the vertical/horizontal orientation.

Defaults Tab

There is one industry standard methodology for determining the correlation. There are experimental methods that are shown here.



- **Original C63 Methodology** — This is the original C63 methodology listed in ANSI C63.4.
- **C63 with Dipole Correction** — In the original C63 methodology, there is a small error related to the dipole gain. This method allows the user to correct this error when analyzing differences in correlation methods.

- **Use Actual Antenna Gain** — In the mathematics of the C63 method, the gain of the dipole is inserted. Below 30 MHz, this is a reasonable assumption. As users move above 1 GHz, this method deteriorates in accuracy.
- **Antenna 1** — This data element should have the gain (not antenna factor) of the antenna to be used with the actual antenna gain method.

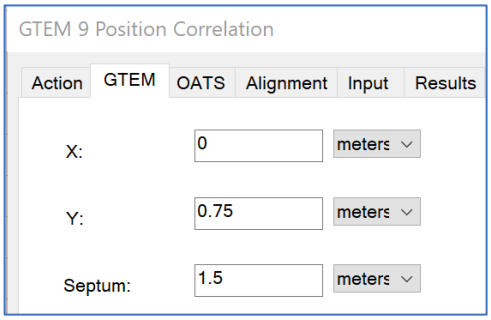
GTEM/OATS 9-Position Correlation

The GTEM/OATS 9-Position Correlation action performs a specialized correlation to determine the magnetic field characteristics for signals measured with a GTEM. This will yield the equivalent field strength as would appear on an Open Area Test Site (OATS).

This action requires nine readings taken with the GTEM. Using these readings in conjunction with the physical dimensions of the GTEM and the OATS, this action performs a mathematical transformation that converts the nine input data readings into the equivalent open field measurement values.

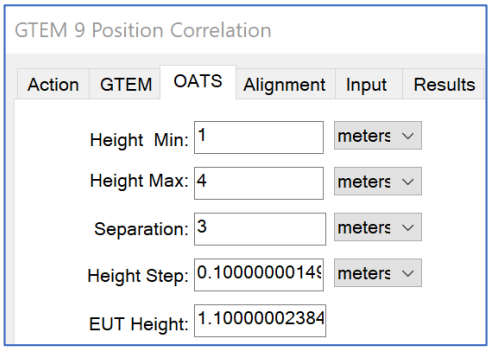
GTEM Tab

This tab is similar to other GTEM tabs and will not fully be described here. See the *GTEM 3 GTEM Tab* for the full description.



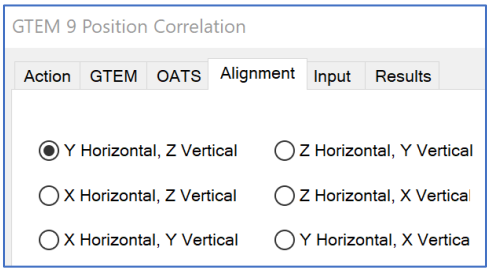
OATS Tab

This tab is similar to other GTEM tabs and will not fully be described here. See the *GTEM 3 OATS Tab* for the full description.



Alignment Tab

When orienting a EUT in the GTEM it is sometimes difficult to use the exact same geometry as was used on the OATS. The Alignment tab allows you to specify which physical orientation was used on the OATS.

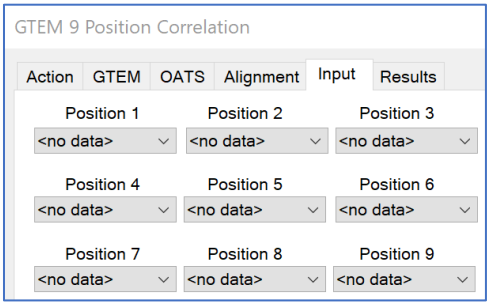


Within the GTEM, when positioning the EUT for proper rotation, it may be necessary to place the EUT in a starting position that is different than the position used when the EUT was tested (or would be tested) on the OATS. This tab allows you to specify the proper home position for the EUT.

The Z-axis is the axis along the length of the septum; the Y-axis is the septum to floor; and the X-axis is the width of the cell. The EUT +Z face is oriented toward the input of the GTEM. Viewed from the input, the +X face is on the left side of the EUT and the +Y face is across the top. The +Y horizontal, +Z vertical (i.e., facing the input of the GTEM) is the most common starting position for a EUT in the GTEM. This would duplicate the physical orientation of the EUT facing the antenna on the OATS.

Input Tab

Performing a nine position correlation requires that nine separate readings be taken of the EUT. You will specify where each of the nine sets of data will be saved.

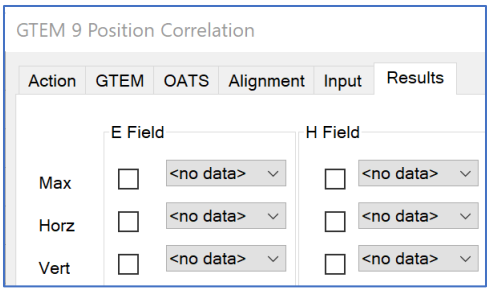


Positions 1, 4, and 7 are the same as those used in the GTEM 3 position correlation analysis.

- **Position 1** — Starting position (SP) referred to as the +X face of the EUT.
- **Position 2** — SP+45 degrees in the X-axis (a counterclockwise movement of the EUT).
- **Position 3** — SP-45 degrees in the X-axis (a clockwise movement of the EUT).
- **Position 4** — First normal rotation. This is referred to as the +Y face of the EUT. From the SP, a 120-degree orthogonal rotation. This is accomplished by rotating the EUT 90 degrees toward the operator and then 90 degrees in a clockwise rotation.
- **Position 5** — +Y+45 degrees in the X-axis (counterclockwise movement of the EUT).
- **Position 6** — +Y-45 degrees in the X-axis (a clockwise movement of the EUT).
- **Position 7** — Second normal rotation. This is referred to as the +Z face of the EUT. From the +Y face position a 120-degree orthogonal rotation. This is accomplished by rotating the EUT 90 degrees toward the operator and then 90 degrees in a clockwise rotation the EUT.
- **Position 9** — +Z–45 degrees in the X-axis (counterclockwise movement of the EUT). Do not confuse the various uses of X, Y, and Z terminology in this section. Using a rotation cube as a reference is very helpful.

  This can be accomplished by taking a cube and placing +X, +Y, +Z, X, Y, and Z labels on the six faces. The + faces should always maintain the same relative plane as the EUT rotates (for example, the + faces always occupy the same three faces of the cube during each rotation).

Results Tab



The correlation analysis yields two sets of three data elements. You have the option of recording all elements or any choice of each element. Use this to save your measurement data.

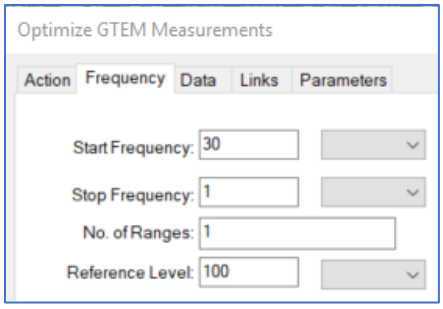
The correlation yields a separate set of three values for both the electric field component and the magnetic field component. The basic correlation for each yields the maximum.

This value is the straight correlated value between the OATS and the GTEM. Certain standards require the maximum signal in the horizontal and in the vertical antenna orientation are separate readings. These values are derived in the mathematics of the correlation algorithm prior to calculating the maximum.

GTEM Optimization

This action is used to perform a Max Hold of a 3-position manipulator in a GTEM using a spectrum analyzer or swept receiver. This action is like other emissions actions except that it adds the positioner instrument and an output for its maximized positions.

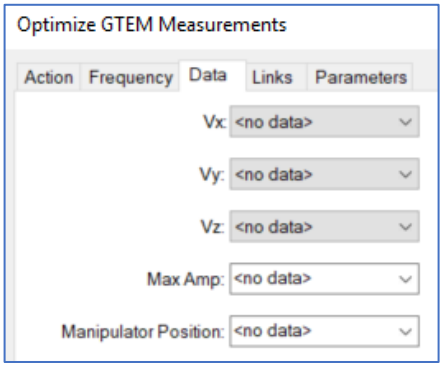
Frequency Tab



This tab is similar to other frequency tabs and will not fully be described here. See the *Measure Range Frequency Tab* for the full description.

- **Reference Level** — This a function of both the noise floor of the instrument and the maximum expected signal. If you are working in an anechoic chamber, you could set this to 60 80 dB and take valid readings. However, on an OATS, this would put the analyzer/receiver in saturation.

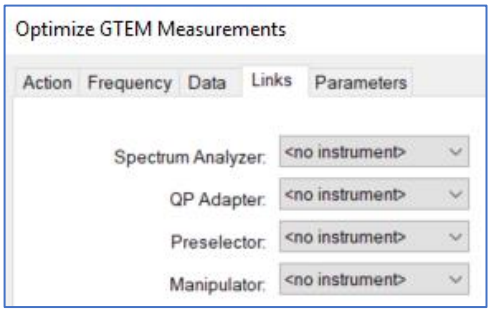
Data Tab



This tab is similar to other data tabs and will not fully be described here. See the *GTEM 3 Inputs Tab* for the full description.

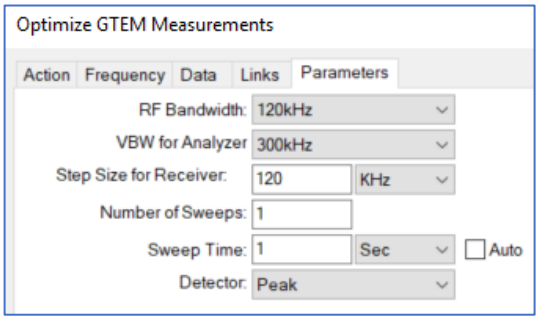
Links Tab

This tab is similar to other links tabs and will not fully be described here. See the *Measure Range Instruments Tab* for the full description.



Parameters Tab

This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.

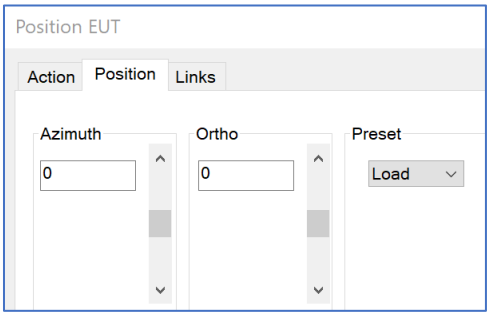


Position EUT (GTEM Manipulator)

The Position EUT action is used to control an automated GTEM manipulator or X–Y positioner. The position can be specified within two degrees of accuracy. The position can be selected from a set of predetermined values or specified in degrees.

Position Tab

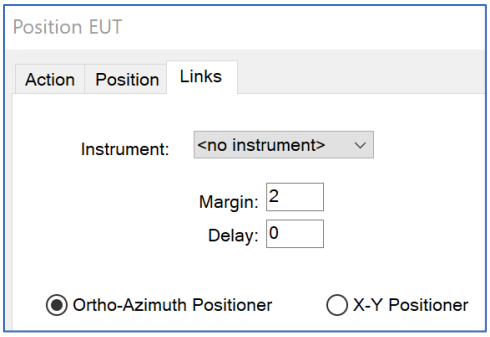
This tab is used to move the current position of the manipulator. The position is defined with the azimuth and orthogonal controls, or chosen from a set of predetermined values.



- **Azimuth** — The azimuth, or X-axis box specifies the angle or position to which the positioner will be moved when the action is executed. The value is **read only** when the value from the **Preset** list is selected, and indicates the value from the selected preset position. When the **custom** option is selected in the **Preset** list, the user can input any value from 0 to 360 degrees/cm.
- **Ortho** — The orthogonal, or Y-axis box specifies the angle or position of the second degree of freedom for the positioner. The value is read only when a value from the Preset list is selected, and indicates the value from the selected preset position. When the custom option is selected in the Preset list, the user can input any value from –180 to +180 degrees or 0 to 400 cm.

- **Preset** — The preset allows the selection of preset position information. The user can select from the list of preset conditions by clicking the arrow. When a different position is selected, the azimuth/X-axis and orthogonal/Y-axis boxes are updated with the preset values from the selected position.

Links Tab



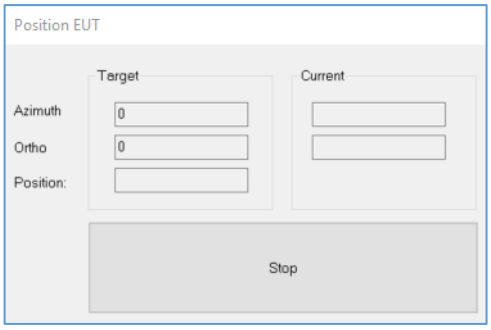
This tab is used to select the instrument that controls the manipulator.

NOTE: If the Instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Instrument** — Select the instrument that controls the manipulator.
- **Margin** — The margin is a defined increment that is the error in executing a stop command. If you are at 120 degrees and need to go to 0 degrees, the action will start motion in the correct direction. As we approach the requested stop position, we are constantly checking current position. As soon as we are within the stop position minus the margin, the stop command is issued. The margin, in effect, is the motion delay required for the positioner to complete a stop command.
- **Delay** — Some positioners will ignore commands if they are still in motion. Sending the stop command causes the positioner to begin stopping, but if we send a command to move in another direction, it will be ignored if the stop has not been completed. This delay is the time between issuing the stop command and sending the next motion command.
- **Ortho Azimuth Positioner/X Y Positioner** — This action will control either an ortho-azimuth positioner or an X-Y positioner. Checking either of these affects the default Presets field on the *Position tab*.

Dialog Box

This dialog box appears when Position EUT is executed and provides visual information to the user as well as the ability to halt execution of the test. The Target field provide information on the desired azimuth and orthogonal angle of the manipulator. The Current fields provide information updates on the manipulator position when it is in motion.



Other Actions

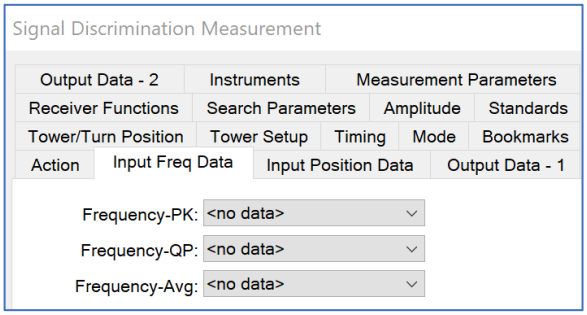
Signal Discrimination Measurement

The Signal Discrimination Measurement action is a manual capture routine. It can take a set of known frequency points as a starting position or operate only on operator input. The action allows the operator to manually tune to a signal, perform height/rotation optimization, and record the information. This action presumes that you have a set of points of interest. These may have been generated through a math step or measured using another emissions action.

For example, if you have taken readings in a chamber and selected the peaks over the specification limit, you will have three data sets: the peak readings, the turntable position of the peak reading, and the tower reading of the peak reading. When started, the action will tune the analyzer/receiver to the first reading, move the tower and turntable to the related position, and wait for the operator input.

Input Freq Data Tab

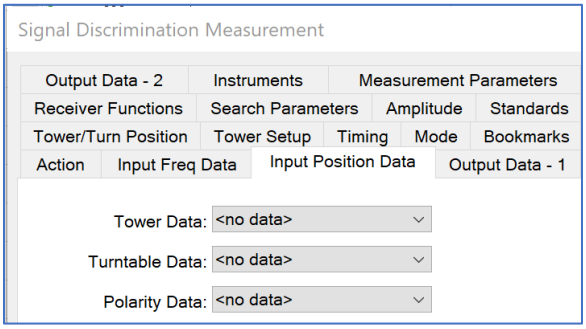
- **Frequency-PK/ Frequency-QP/ Frequency-Avg** — You must have either a peak or QP data element or the action will not start.



Input Position Data Tab

- **Tower/Turntable Data** — This data element will represent the position that the tower was in during the measurement that yielded the peak or quasi-peak.

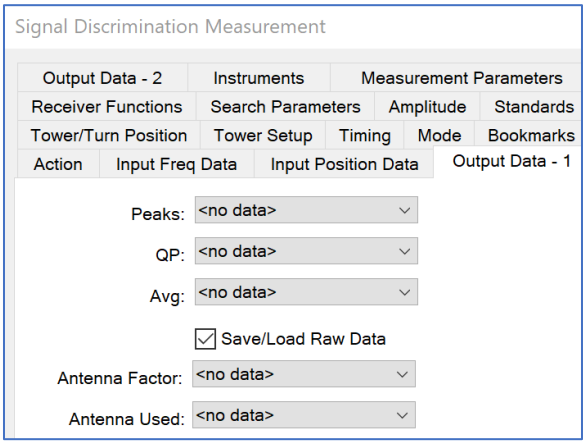
  If Automatically Set Twr/Turn on Frequency Change on the *Tower/Turn Position tab* is selected, the information contained in the specified data element will set the tower/turntable position at the start of each new frequency. If it is not selected, the information in this data set will be displayed for reference purposes only.



- **Polarity Data** — If Automatically Set Twr/Turn on Frequency Change on the *Tower/Turn Position tab* is selected, the information contained in the specified data element will turn the antenna to the correct polarity. If it is not selected, the information in this data set will be displayed for reference purposes only.

Output Data - 1 Tab

When performing the signal discrimination, we can record peak, quasi peak, and average readings. We can also capture information related to these readings, such as antenna used, antenna factor, tower, turntable, polarity, and comments.



- **Peaks/QP/Avg** — Choose an appropriate data element for storing the values measured. If this is the same data element as the input data, this data will overwrite existing data.

  If this is a unique data element, it will store the output values. Generally, it is recommended that you store the output into a unique name. This allows you to compare the input data to the output data for analysis.

- **Save/Load Raw Data** — Generally within TILE!, emissions data is saved in raw format (direct readings from the instrument). If this check box is selected, then we will save a corrected data (inclusive of preamp, cables, and antenna factors).

- **Antenna Factor** — Choose an appropriate data element for storing the actual antenna factor used during each measurement. The Signal Discrimination action allows the user to specify a different antenna for each frequency measurement.

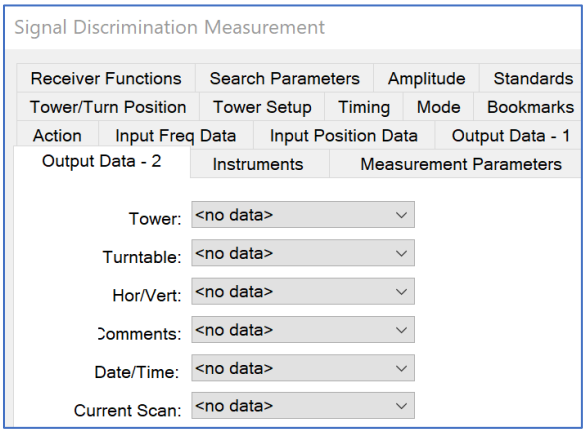
  For example, on an OATS you might take a reading with a bi-conical. If you find yourself close to the limit and substitute a dipole antenna for the final reading, it is important to know which antenna was used at each frequency and to correct the data for the antenna factor.

- **Antenna Used** — Choose an appropriate data element for storing the antenna used during each measurement. This data element must be a word-type data element. It will hold a text message with the name of the Antenna Factor data element used for this measurement. Since these are typically named to identify the antenna, this serves as the name of the antenna used.

  The Signal Discrimination action allows the user to specify a different antenna for each frequency measurement. For example, on an OATS you might take a reading with a bi-conical. If you find yourself close to the limit and substitute a dipole antenna for the final reading, it is important to know which antenna was used at each frequency and to correct the data for the antenna factor.

Output Data - 2 Tab

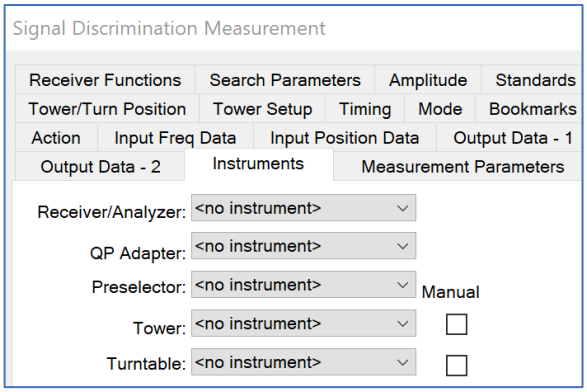
The Output 2 data elements are for Tower, Turntable, Hor/Vert, and Comments. These are continuations of the data available from the Output 1 tab.



- **Tower/Turntable** — Choose an appropriate data element for storing the tower/turntable height/angle for each measurement.
- **Hor/Vert** — Choose an appropriate data element for storing the tower polarity for each measurement. This is a word-type data element that will hold an H or **V**, depending upon the polarity measured during the test.
- **Comments** — Choose an appropriate data element for storing comments entered for each measurement. This is a word-type data element and only this type will be displayed in this list.
- **Date/Time** — Choose an appropriate data element for storing the date and time information. This is a word-type data element and only this type will be displayed in this list.

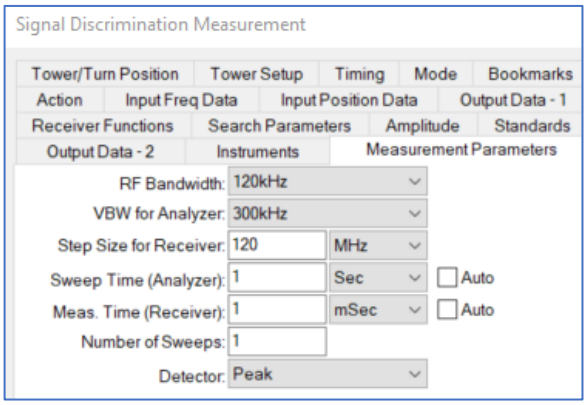
Instruments Tab

This tab is similar to other instruments tabs and will not fully be described here. See the *Scan Peaks Instruments Tab* for the full description.



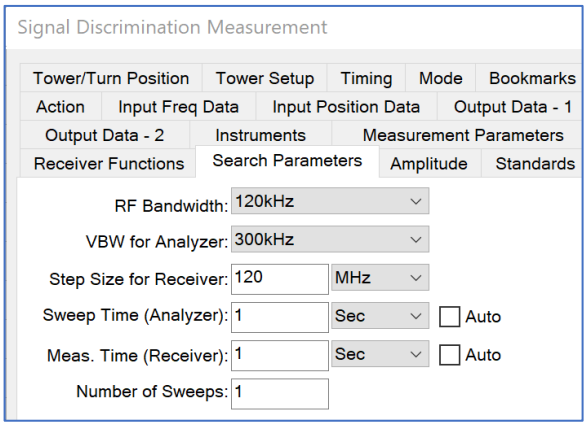
Measurement Parameters Tabs

This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



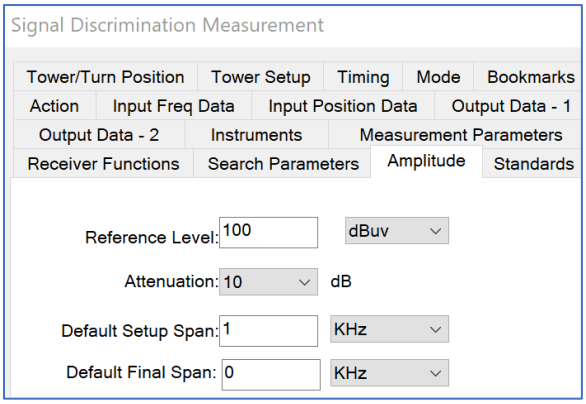
Search Parameters Tabs

This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



Amplitude Tab

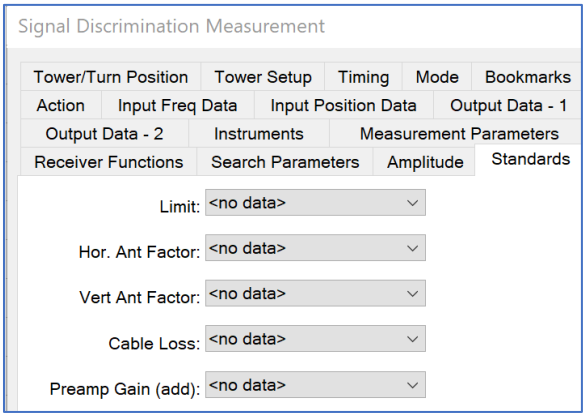
This tab is similar to other amplitude tabs and will not fully be described here. See the *Measure Range Amplitude Tab* for the full description.



- **Default Span** — The default span is the window size for the spectrum analyzer for each frequency. The start frequency for the display will be the current frequency less than one half of the span. The stop frequency will be set to the current frequency plus one half of the span.

Standards Tab

These data elements contain the specification limit information as well as antenna correction information appropriate for each frequency being tested.



- **Limit** — Select the data element with the specification limits. The user has complete control of the definition and use of the specification limit.

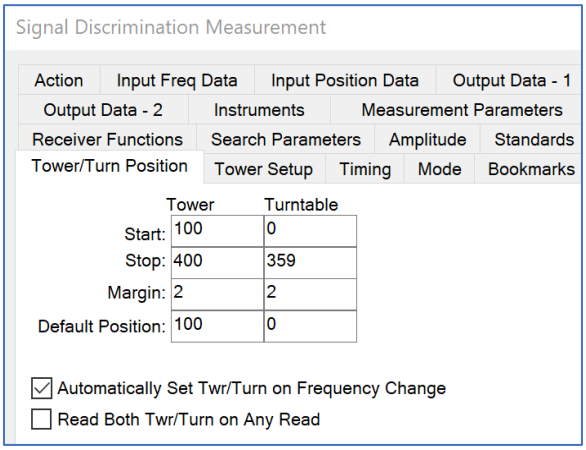
  This value will determine the margin on the display. The limit value will be subtracted from the current reading to determine the margin at the current frequency.

- **Hor/Vert Ant Factor** — In performing signal discrimination the system needs to keep track of the appropriate correction factor for each frequency range in which testing is done. Select the appropriate data element for the default antenna. This is the default setting, but it can be changed interactively any time during the execution of the test.

- **Cable Loss/ Preamplifier Gain (add)** — This value will be added to the actual instrument reading for calculating the margin. If **No Data** is selected, then a default value of 0 dB will be used by the software.

Tower/Turn Position Tab

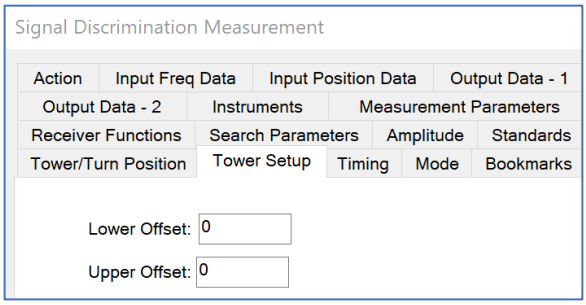
This tab is similar to other tower/turntable tabs and will not fully be described here. See the *Pre-Scan Tower/Turntable Tab* for the full description.



- **Default Position** — Enter a value reflecting the default position for the positioner. When starting the action, this is the position that the tower and turntable will be moved prior to any readings.
- **Automatically Set Twr/Turntable on Frequency Change** — When this check box is selected, the tower and turntable will be set to their position automatically when the user moves to the next frequency. If it is deselected, the tower and turntable will be left in their current positions.
- **Read Both Twr/Turn on Any Read** — Normally, the tower and turntable positions are updated only when the Read button or Optimize button is clicked. With this option, the current position will be updated any time any instrument is read.

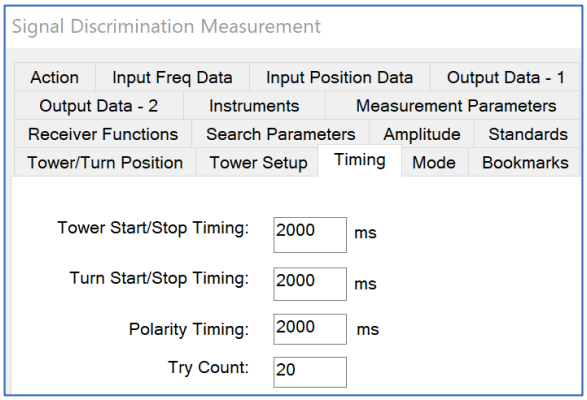
Tower Setup Tab

This tab is similar to other setup tabs and will not fully be described here. See the *Pre-Scan Tower Defaults Tab* for the full description.



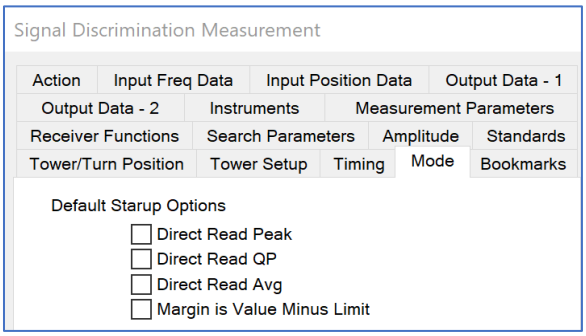
Timing Tab

This tab is similar to other timing tabs and will not fully be described here. See the *Pre-Scan Timing Tab* for the full description.



Mode Tab

The operator can trigger a measurement of the current frequency at any time. In a normal measurement, the software will use the default settings for parameters and bandwidths, set up the instrument to those defaults, and take a reading. If direct read is selected, the software will read the current marker only. This gives the operator finer control over the measurements.



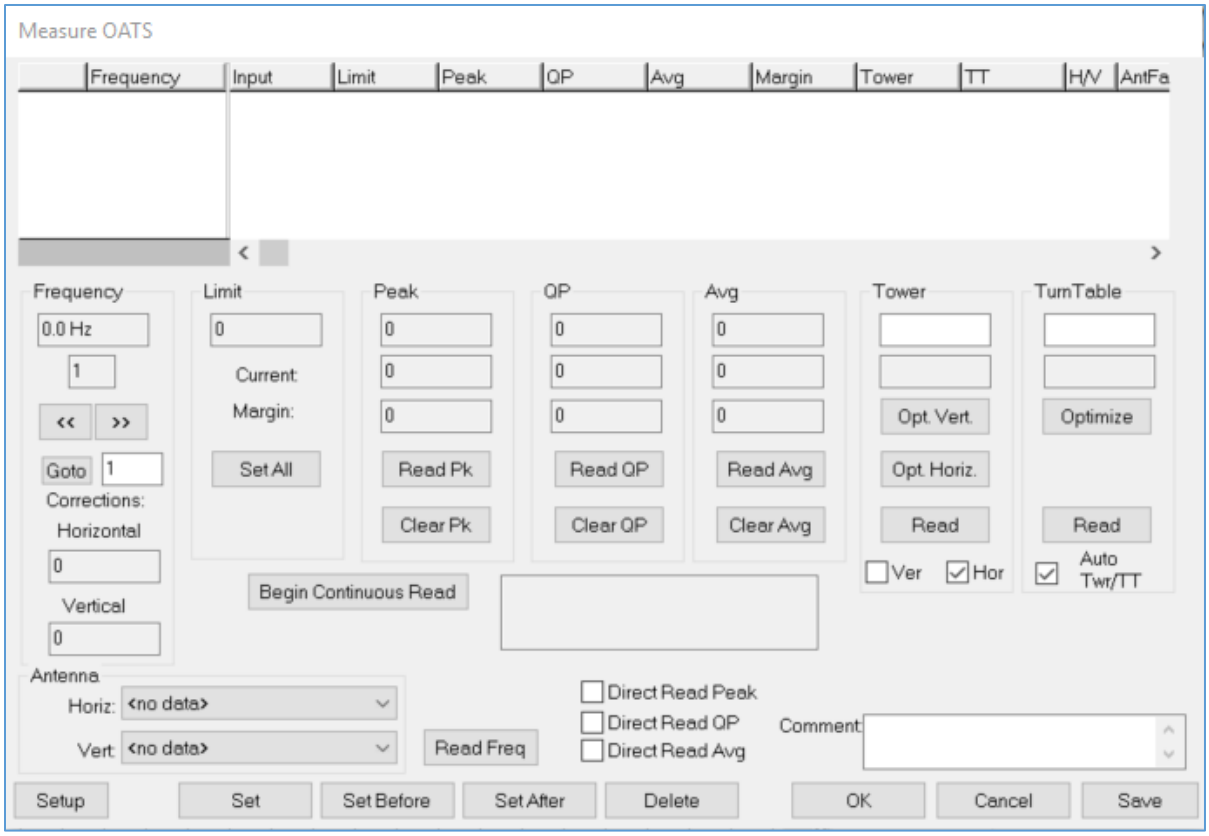
- **Direct Read Peak/ Direct Read QP/ Direct Read Avg** — When taking a reading, this check box determines whether the software will reset the instrument to the default conditions prior to taking a read, or whether it will read the current marker.

Dialog Box

This dialog box appears when Signal Discrimination is executed. The Signal Discrimination Measurement action is a manual action. The input table gives the operator a set of known signals to look at, but the operator can always manually scan on his receiver/analyzer and capture new signals.

The dialog box has a table of known signals, as defined on the *Input Freq Data tab*. The operator then adjusts the receiver/analyzer, searching for new or existing peaks. Once a signal is viewed on the screen, the operator can add to or update the table.

NOTE: The tower and turntable columns will not appear unless an instrument is selected for them on the *Instruments tab*.



- **Frequency** — Displays the current position in the table of frequencies. The second box indicates which step this is in the list (step 1, 2, and so on). The double arrows move up or down the known list of frequencies. These known points are changed by the **Set Before**, **Set After**, and **Delete** buttons.
- **Antenna** — Displays the antenna factor for the current frequency. The readings in any of the remaining boxes will not be corrected if the antenna factor does not exist or is out of range. The Horiz and Vert boxes are the antenna factors for the appropriate antenna if valid data exists at this frequency.
- **Limit** — The Set All button does a peak search within the current window of the instrument. The Limit displayed is from the Standards Limit data element selection.
- **Peak/QP/Avg** — Contains the maximum reading at this frequency, the last reading from the instrument, and the margin between the maximum reading and the specification limit at this frequency. If you read several times, the first/top box will always reflect the highest reading found at any point.

  When you click Set, Set Before, or Set After for this frequency, the maximum number is the saved number. The current number reflects the current (last effective) reading from the instrument. Margin is the difference between current reading and the specification limit. No reading in any box will be valid if the antenna factor is not valid.

  Each have a **Read** and **Clear** button. When the operator has a signal to measure, manually move the marker to the frequency of interest. At this point, you may read the current marker as it sits, or perform a calibrated reading using the default standards. If you click **Read P**k, then a default test will run based on the standard system setups. If you select the Direct Read check boxes, the software will read the current marker with no adjustment.

- **Tower** — The first/top box reflects the position of the tower when the maximum-recorded signal was taken. The second/bottom box reflects the current tower position. You can change the antenna from vertical to horizontal by selecting the **Ver** or **Hor** check box and rereading the tower. If a manual tower is used, the display is changed to reflect the current position and a check box for polarity. Enter the position and check the appropriate polarity.
- **Turntable** — The first/top box reflects the position of the turntable when the maximum-recorded signal was taken. The second/bottom box reflects the current turntable position. If a manual turntable is used, the display is changed to reflect the current. Enter a value for the actual position of the turntable, if desired.
- **Antenna** — Changes the Antenna Reference data element for the current frequency for the horizontal or vertical orientation. Once a data element is selected, the values will be adjusted to reflect the correct antenna factor. If a data element is selected that does not cover the current frequency, a dialog box is displayed. The default Antenna Correction Factor data element is defined on the *Standards tab*.
- **Read Freq** — Click this button to read the current frequency and update that information without changing the values for peak, QP, or avg. This is often used when you want to manual fine tune frequencies of interest prior to executing an automatic measurement using the Scan Peaks action.
- **Direct Read Peak** — Determines whether, when the read peak button is clicked, the software will read the current marker without setup (check condition) or whether it will tune the instrument and take an appropriate measurement using the default settings.
- **Direct Read QP** — Reads the spectrum analyzer after it has been manually configured by the user.
- **Comment** — The operator can enter a comment related to this frequency (assuming a data element has been specified for outputting the comment). These comments can then be placed on graphs or tables as additional information.
- **Setup** — Provides the user some control of the items that display on the table of frequencies. The view of each item can be switched ON or OFF and can be updated at any time by the operator.
- **Set** — Changes the current stored information, replacing it with the maximum-recorded values from the Peak, QP, Avg, Antenna Used, Antenna Factor, Margin, Tower, and Turntable boxes.

  NOTE: If you are physically reading a different frequency than the current point in the table, do not use Set; this will overwrite the frequency point in the table.
- **Set Before, Set After, and Delete buttons** — Control recording of information to file.

  The Set Before and Set After buttons add the current frequency and related information into the frequency table on a new line. Adding will cause two identical points to exist in the table. This will prevent certain math functions from performing correctly. If an additional data point is added by mistake, select this step and then click **Delete**.
- **Delete** — Deletes the currently highlighted frequency point on the table.
- **OK** — Saves the current table of frequencies and exits the action.
- **Cancel** — Aborts the current execution and exits the action without saving any data.

Site Attenuation

The Site Attenuation action is designed to assist in performing an Open Area Test Site (OATS) attenuation measurement. These are generally done per ANSI C63.4, but this action can be used for any attenuation measurement.

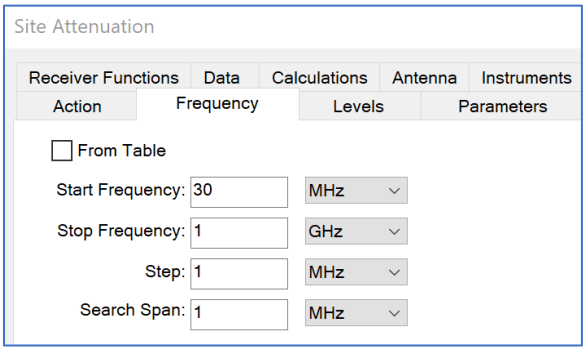
There are two modes to the action, the cable-referencing step and the actual measurement. Generally, when performing site attenuation, you first lay out the test instrumentation and cables with two opposed antennas. Prior to taking readings on the antennas, you will connect the cables end to end, set the generator level at some standard setting, and read the signal path through the cables. This will give you a reference level that includes any losses in the cables and connectors.

The next step is to hook the cables to the antennas and take readings at the same signal generator output-level. The difference between this reading and the cable reference reading is the loss through the antennas and free space. The difference can then be input into the appropriate formula to derive site attenuation. The two different modes are set on the *Levels tab* of this action.

This action can be configured to take a cable reference reading (direct mode) or an OATS measurement. It is not necessary to perform a cable reference reading prior to running this action, but if no cable reading is made, the user must adjust the final readings to include all appropriate correction factors.

Frequency Tab

This tab sets the frequency range, number of steps, and search range of interest for the test. Since we are dealing with radiated signals in free space, the search span allows us to limit the effects of ambient signals on our readings.



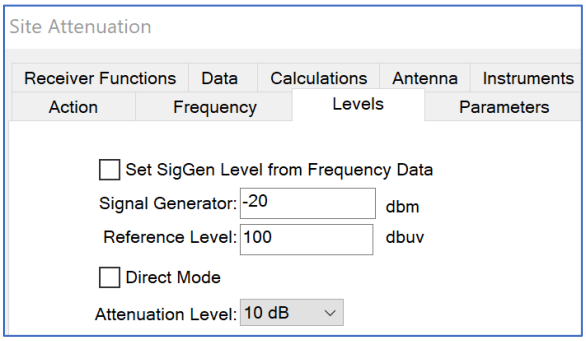
You can enter the start, stop, and step frequencies, or you can enter a data element with a set of specific frequencies of interest.

- **Start Frequency /Stop Frequency** — Enter an appropriate number and select the correct units.
- **Step** — Determines the interval between each reading in the test. When setting this value, each frequency point is scanned for tower height to get the maximum value. The larger this setting is, the longer the test will take to perform.
- **Search Span** — The Site Attenuation action uses a spectrum analyzer/receiver together with a signal generator. Since these two units might have slight differences in absolute frequency accuracy, the search span determines how wide the window will be on the analyzer during scans.

  The system will open this window, search for the peak signal, and record this level. Since most instrumentation used in an EMC laboratory has been calibrated, this might not appear useful. However, the frequency accuracy between analyzers and receivers can affect this test.

Levels Tab

This tab sets the base parameters to be applied to the generator and the receiver/analyzer.

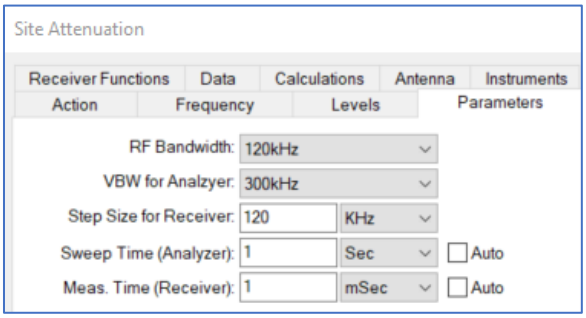


- **Signal Generator** — Sets the signal generator level in dBm. This action uses a fixed signal level in both direct and indirect mode. The frequency is set on the *Frequency tab*. If you are using an amplifier, it will be included in the direct mode readings. Install the amplifier in line with the instrumentation. If you need to use different signal generator levels at each frequency, you should use the *Scan Peaks* action to configure this test.
- **Reference Level** — A reference level for the signal analyzer is set using this value. Using a value too low will cause errors in readings done in an OATS environment because the actual signal will be out of range of the screen parameters. At an OATS, large ambient signals will often saturate the analyzer/receiver. You should set the attenuation for the receiver/spectrum analyzer to prevent this condition
- **Direct Mode** — Measures the cable reference. When this check box is selected, the range of frequencies is scanned with no tower movement. These values are stored in the *Data Tab* Attenuation data element. You then use this data element as the calibration element in a second action configured in a non-direct mode.

  This allows the user to measure only the net attenuation of the transmitted signal during the actual site attenuation readings. When using this mode, all the settings should be identical to those to be used in the final, indirect action.
- **Attenuation Level** — Select an appropriate input attenuation, if required, to protect the front end of the receiver.

Parameters Tab

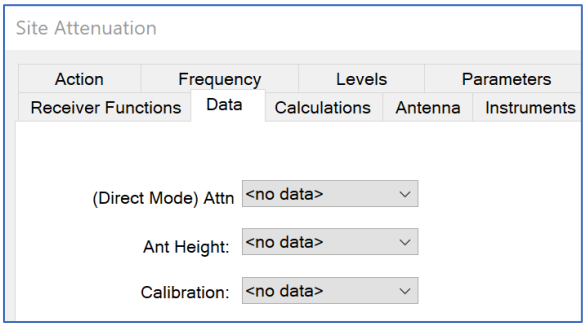
This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



- **RF Bandwidth** — When you have specified a QP adapter on the *Instruments tab*, the RF bandwidth will be set for the QP adapter.

Data Tab

This tab identifies the data elements that will be used to store the information generated by this action.

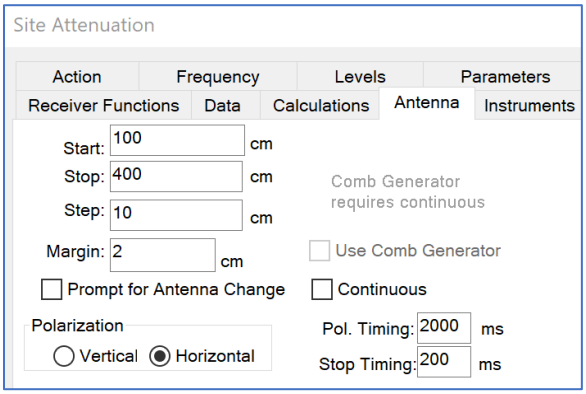


NOTE: If the Attn and Ant Height data are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **(Direct Mode) Attn** — The attenuation level is the net signal level, after correction for the direct mode cable readings. The system will take the analyzer/receiver level and subtract the calibration level prior to recording this number. If there is no calibration value, then the analyzer/receiver reading is stored without correction.
- **Ant Height** — As part of maximizing the signal, the tower is scanned for the peak signal. The peak is stored in **(Direct Mode) Attn**; the actual tower position is stored in **Ant Height**.
- **Calibration** — The Calibration data element is the direct cable-to-cable readings derived when this action was performed in the direct mode. This data is subtracted from the actual signal readings to derive the net signal stored in **(Direct Mode) Attn**.

Antenna Tab

This tab configures the tower scan and step conditions. All tower readings are in centimeters.



- **Start** — The lowest desired tower height is the start position. In normal C63.4 measurements, this will be 1 meter (100 cm).
- **Stop** — The maximum tower height desired during the test is the stop position. In normal C63.4 measurements, this will be 4 meters (400 cm) for 3 and 10 meter OATS and 6 meters for 30 meter OATS.
- **Step** — The step value determines how many centimeters will separate each height reading on the tower.
- **Margin** — The margin setting is a value that allows the user to configure the slippage, or stopping error, of the tower. When the tower is in motion, the software will check for the current position. Whenever we are within the margin distance of the stop position (not the formal stop, but the current move to position) the stop command will be issued.

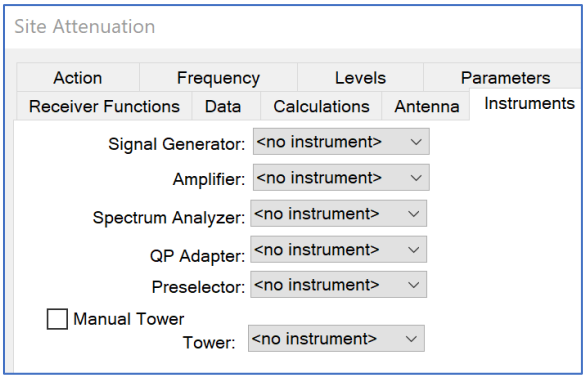
  The presumption is that the tower will then coast into a final position as close to our goal as possible. This setting will be site and test specific because of the differences between towers and the weight of the antennas placed upon them.
- **Prompt for Antenna Change** — This check box is used when a dipole antenna is in place. For each frequency, the dipole is configured, the antenna is stepped across the start/stop range, and then the operator is prompted to configure the dipole antenna for the next frequency.
- **Continuous** — Performs a scan across the designated tower start/stop points recording the maximum value obtained during the scan. The analyzer is put into Max Hold and then the tower is

moved from start to stop. This allows a capture of the highest signal found, but records no height information.

- **Polarization** — This action can be operated with the antenna in either the horizontal or the vertical orientation. If you do not have a tower and are manually adjusting the height or using this action for some other purpose, the polarization element is ignored.
- **Pol. Timing** — Different towers execute polarity movements at different speeds. To ensure accurate testing, it is essential that the polarization movement be completed prior to any movement of the tower or turntable. This timing allows the user to match the execution to the characteristics of their tower.
- **Stop Timing** — Certain towers and turntables require fixed periods to perform a Stop or Start action. If a command is sent prior to completion of the stop, that command is ignored. This setting lets you lengthen or shorten the time interval before execution of the next step.

Instruments Tab

This tab identifies the instruments to be used during the test. See *Measure Range Instruments Tab* for the full description.

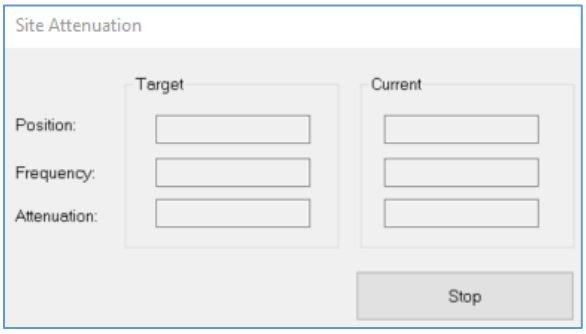


NOTE: If the Signal generator, Spectrum Analyzer, and Tower instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Signal Generator/Spectrum analyzer/Tower** — Select an instrument for signal generator, spectrum analyzer, and tower.
- **QP adapter/Preselector** — If you have a separate QP adapter and Preselector, select these instruments.
- **Manual Tower** — Select the check box if no tower is available, or it is not GPIB controlled, to configure an operator interrupt. This will allow you to manually set the tower position during the test.

Dialog Box

This dialog box appears when Site Attenuation is executed and provides visual information to the user as well as the ability to halt execution of the test.



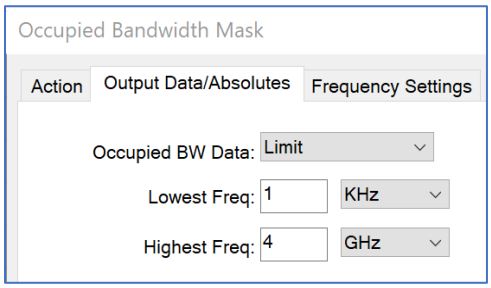
- **Target** — Provides information on the desired position of the tower, the current frequency setting for the receiver/analyzer, and the output level for the signal generator.
- **Current** — Provides information updates on the tower position when it is in motion, on the

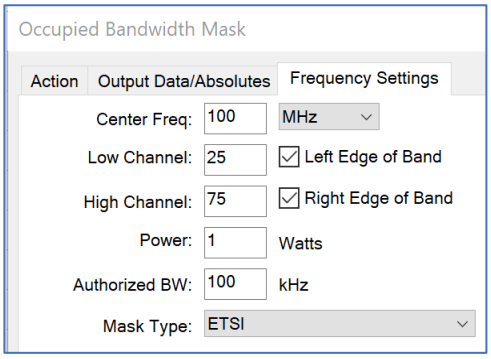
actual frequency setting of the receiver/analyzer, and the corrected reading of the attenuation across the antennas.

Occupied BW Mask and FCC Power Spec Limit

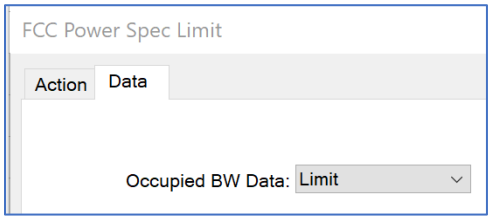
These actions are not documented but the following tabs are the available; ask *ETS-Lindgren* for help with these actions.

The Occupied Bandwidth Mask (BW MASK) will calculate the FCC/ETSI occupied bandwidth mask for transmitters.





The FCC Power Spec Limit action calculates the FCC spec limit for power transmitters by using the calculation (43+10*Log (Power)).



Immunity/Susceptibility Actions

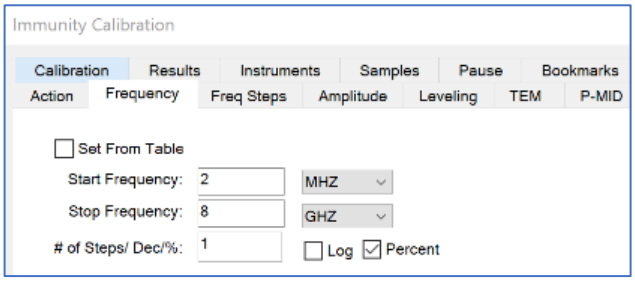
The immunity/susceptibility capabilities in TILE! are adaptable to almost all regulatory, industrial and military applications. A typical application would require a calibration procedure and a EUT test procedure.

Immunity Actions

Immunity / Susceptibility Calibration

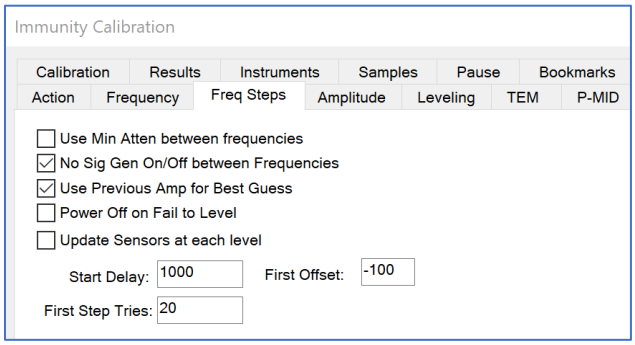
Frequency Tab

This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Frequency Tab* for the full description.



Freq Steps Tab

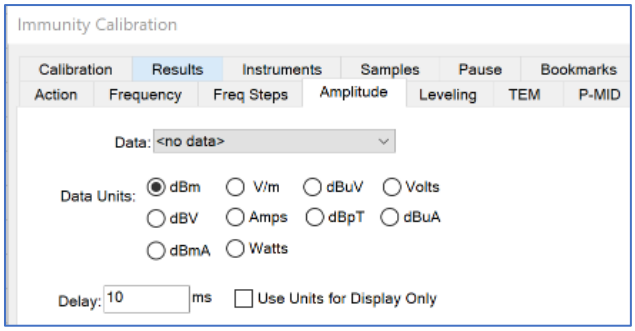
This tab is similar to other freq steps tabs and will not fully be described here. See the *Immunity Test Freq Steps 1-3 Tabs* for the full description.



- **Update Sensors at each level** — When in the leveling loop, the software normally ignores updating the screen to speed up acquisition. If you want the screen values to update at each step, then select this check box.
- **Start Delay** — When certain power meters or probes are first exposed to a field, they require additional time to level. This value is an additional delay time that is substituted at the first frequency only.

Amplitude Tab

This tab is similar to other amplitude tabs and will not fully be described here. See the *Immunity Test Primary Amp Tab* for the full description.

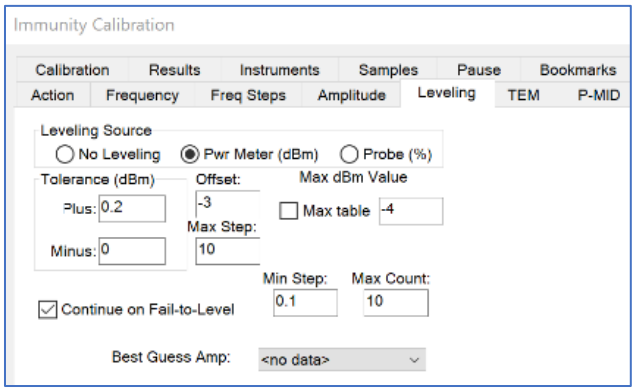


NOTE: If the Data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Delay** — Represents a specified delay between each frequency step. It is primarily of value when testing routines that do no leveling. This will prevent sending frequency/amplitude data to the signal generator faster than it can react.

Leveling Tab

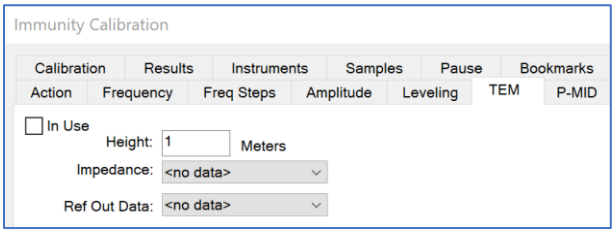
This tab is similar to other leveling tabs and will not fully be described here. See the *Immunity Test Leveling Tab* for the full description.



- **Continue on Fail to Level** — Allows the user to continue with the leveling steps even when a step fails to level.
- **Best Guess Amp** — Sets the signal generator to a predetermined level to speed up the leveling process. This is used when approximate signal generator levels are known. The data set must be a defined CSV data element.

TEM Tab

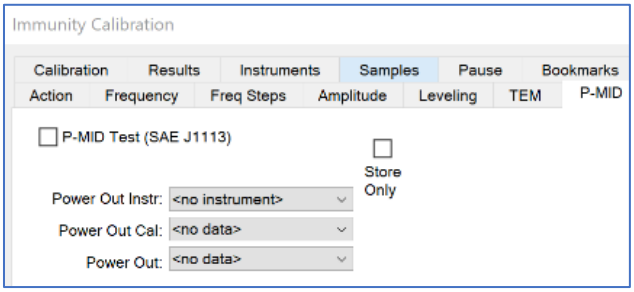
This tab is similar to other TEM tabs and will not fully be described here. See the *Immunity Test TEM Tab* for the full description.



- **Ref Out Data** — Allows the user to store the calculated field strength in a data element.

P-MID Tab

Power at the Mid-Point (P MID) is a specific automotive test first used by Ford Motor Company. The P-MID terminology relates generally to measurement of two different power levels and averaging of power at the midpoint as part of the test requirement.



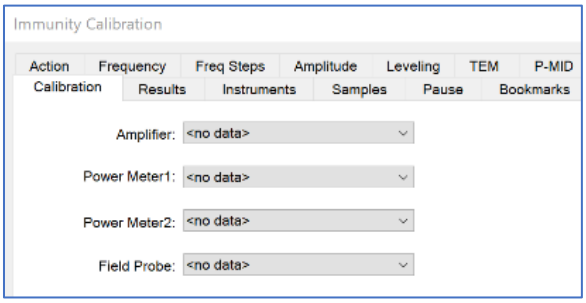
This requirement involves measuring the net power injected into a TEM cell structure and then measuring the output of the TEM cell structure and averaging the two to obtain a final value.

NOTE: Must select the **P-MID Test (SAE J1113)** box to perform this type of test.

- **Power Out Instr** — The power out instrument is used to measure the output power from the TEM cell. Within this specification, the forward power is measured.
- **Power Out Cal** — Select a data element to provide calibration data on the power meter readings. This should be a file-type data element.
- **Power Out** — The data element to store results of this measurement.
- **Store Only** — Allows you to read the TEM cell output power-level but not to use this value as part of the leveling.

Calibration Tab

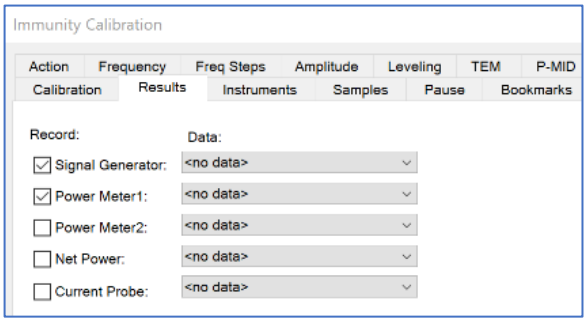
This tab is used to choose the correction factors for the selected instruments. Select the data elements that contain the factors for the losses or gains associated with that instrument. Losses such as directional couplers should have positive numbers.



The field probe calibration should be the correction information from the manufacturer. Probe calibration tables tend to be in the +0.8 to +1.2 range, and are multiplied rather than added.

Results Tab

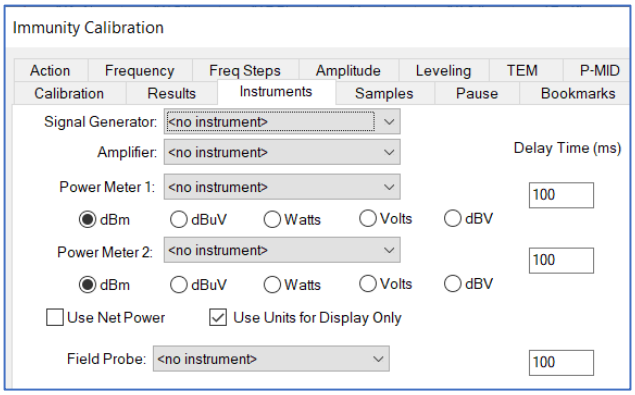
This tab is similar to other results tabs and will not fully be described here. See the *Immunity Test Results 1 Tab* for the full description.



Instruments Tab

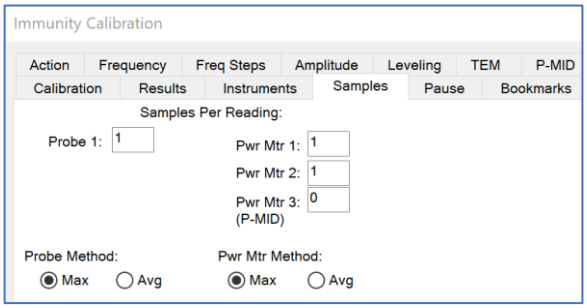
This tab is similar to other instruments tabs and will not fully be described here. See the *Immunity Test Instruments Tab* for the full description.

NOTE: If the Signal Generator instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



Samples Tab

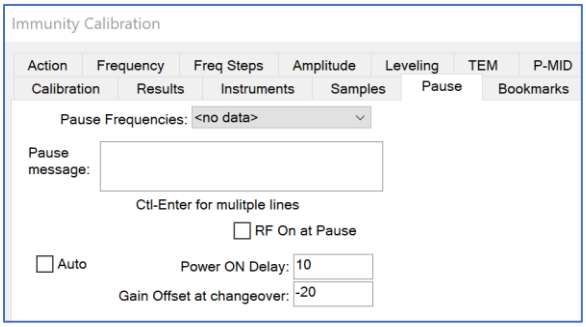
This tab is similar to other samples tabs and will not fully be described here. See the *Immunity Test Sample Tab* for the full description.



Pause Tab

This tab is similar to other pause tabs and will not fully be described here. See the *Immunity Test Pause Tab* for the full description.

- **RF On at Pause** — When pausing the test at a specified frequency, the default behavior is to turn RF off completely. This would be appropriate if you are changing components such as power meters and amplifiers. This option allows the operator to force the RF to remain on at transition points.



Dialog Box

When you are executing the Immunity Calibration, you will have a dialog box displayed showing the current frequency, step number, amplitude target, current amplitude, leveling, try count, and delta.

Delta percentage refers to the amount of change from the previous signal generator level that is being accomplished in this try. The final level is saved when it falls within the tolerances defined.

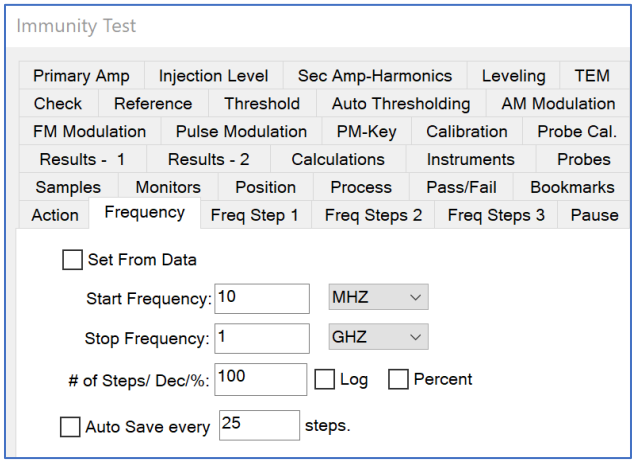
Immunity calibration only saves the information it is configured to save. The **Stop Test RF Off** button will halt execution of the current action; No information is saved.

Immunity / Susceptibility Test

The Immunity Test action performs an Immunity test using either predetermined field data or real time leveling. Many of the features are identical to the Immunity Calibration action in structure, but are used here with slight differences.

Frequency Tab

This tab is used to set the frequency parameters. The Immunity test can be run by selecting the frequency range either by using a start/stop/step mode or by using a data element.



- **Set From Data** — Enables a predetermined set of frequencies to be read in from a data set.

  This is particularly useful when performing IEC testing (or other standards) which require a percent stepped frequency progression but you also have extra frequencies of interest. Create a data element of the IEC values and then merge a data element with the additional frequencies of interest. Then use this option and the software will test all the frequencies of interest.

- **Start Frequency** — Sets the start frequency for this test. The start frequency does not have to be less than the stop frequency. This action will start at the start frequency and then step towards the stop frequency in the manner instructed (it does not matter if this is up or down).

- **Stop Frequency** — Sets the stop frequency for this test. This does not have to be higher than the start frequency.

- **# of Steps/Dec/%** — Defines the number of steps to be used in the range of frequencies set by the start and stop frequencies but does change meaning depending if the log or percent check boxes are selected. If neither are selected then the step size is calculated by dividing the range (stop start) by the number of points.
    - **Log** — Defines how the steps will be taken. If selected, the software will split the frequency range into log decades and then split each decade into the number of ranges specified. For example, if you are testing from 80 MHz to 1 GHz in log steps with the ranges set to 5, the software will split the range 80 MHz–800 MHz into five sub ranges. It will then split the 800 MHz–8 GHz into five ranges, but terminate the test on the last range that covers 1 GHz.
    - **Percent** — This is used when performing IEC testing (or other standards) which require a percent stepped frequency progression. When this check box is selected, you can specify any value for percent, including fractions (entered as decimal places). Some of the MIL STD tests use 0.25% step sizes. The IEC uses 1%. Some calibration schemes use 2% or 5%.

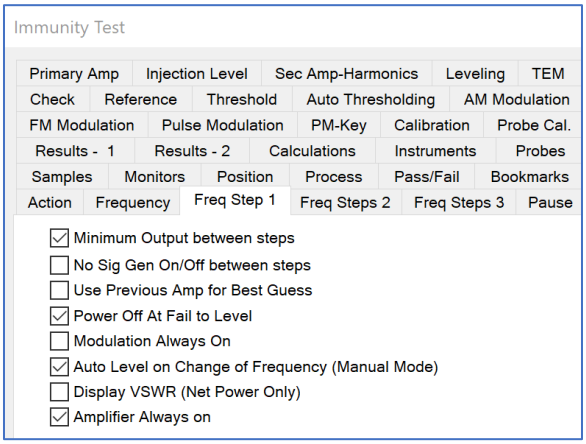
      For example, MIL-STD-461F/G, Table III requires an equivalent step size of 0.25% for test frequencies greater than 1 GHz.

TABLE III. Susceptibility scanning.

Frequency Range	Analog Scans Maximum Scan Rates	Stepped Scans Maximum Step Size
30 Hz - 1 MHz	0.0333 f_o/sec	0.05 f_o
1 MHz – 30 MHz	0.00667 f_o/sec	0.01 f_o
30 MHz - 1 GHz	0.00333 f_o/sec	0.005 f_o
1 GHz - 40 GHz	0.00167 f_o/sec	0.0025 f_o

- **Auto Save every <#> steps** — Normally, saves are only performed as a step on the flowchart, but very complex or long Immunity tests might require intermittent saves. This setting determines how often during the Immunity test to perform a save; this saves the complete profile, not just the frequencies for this Immunity test.

Freq Step 1 Tab



This tab lets the user control several different functions/methods/algorithms during the Immunity test.

- **Minimum Output between steps** — Some signal generators use a relay to set attenuation levels. This relay can wear out over time if used excessively. If you are concerned about this relay, or if the on/off sequence affects the testing, select this box to prevent the command from being sent to the signal generator.
- **No Sig Gen On/Off between steps** — Some signal generators use a relay to turn on and off the signal. This relay can wear out over time if used excessively. If you are concerned about this relay, or if the on/off sequence affects the testing, select this box to prevent the command from being sent to the signal generator.
- **Use Previous Amp for Best Guess** — When leveling, there are three methods of determining the starting amplitude for the leveling loop. If no other choice is made, the software will calculate a beginning amplitude by taking the desired level, the offset and the gain of the amplifier, and guessing at a starting amplitude.

  The second method is found on the Leveling tab. You can specify a fixed set of beginning amplitudes (less than the offset) using the Best Guess data element.

  Generally, the fastest method is to use the previous frequency signal amplitude as a beginning point. If you select this check box, the previous level will be used. In this case, the choices (Best Guess data element and offset) will be ignored, except at the first frequency.
- **Power Off on Fail to Level** — When the leveling loop fails, a dialog box will provide the choice to retry this step, skip this step, or quit. While this dialog box is present, this check box determines whether power is ON or OFF. For safety, RF is turned off when this dialog box is present.

  This prevents potential damage to the EUT. Turning RF OFF does have the disadvantage of making it difficult to examine the state of the equipment to understand the failure. Knowing what the last amplitude was or what the power meter was showing can help in correcting or understanding this failure to level. Selecting this check box will cause the RF to be left ON while the dialog box is present.
- **Modulation Always On** — When this check box is selected, the modulation will be set at the beginning of the test and left on always. All leveling will be performed with modulation on.

  There is a specific set of steps that take place during a test. Normally, when the test starts the signal generator will be in an RF off state. The attenuation will be set to the lowest number supported by the signal generator (normally 123 dBm).

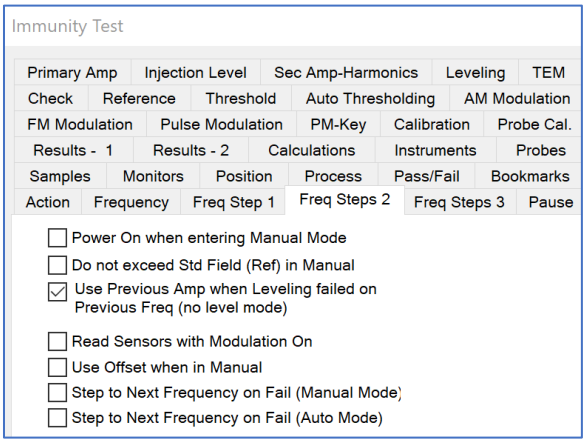
  When the action starts, the generator is set to the first frequency, the signal generator is turned to RF ON. The level is set to the first amplitude. Once the field is established, the modulation is turned ON. After dwelling, we turn off the modulation, set the generator to a minimum attenuation, turn the RF off, and step to the next frequency.
- **Auto Level on Change of Frequency (Manual Mode)** — When in manual mode you have the option to step up or down in frequency. When stepping to another frequency, this check box controls whether the software will perform a level cycle or whether the signal generator will be left at the amplitude level from the previous frequency.

- **NOTE**: In order to prevent extra sensor reads after the level cycle has completed, make sure this setting is enabled.
- This setting translates to the UI dialog's "Auto Lvl On" checkbox at runtime.
  - If you have the Freq Step 1 setting enabled, when you enter Manual Mode during a run, the Auto Lvl On box is initially checked.
  - If you have the Freq Step 1 setting disabled, when you enter Manual Mode during a run, the Auto Lvl On box is initially unchecked.
  - If the box is checked, when you are in Manual Mode and Step forwards or backwards in frequency and the RF is ON, the action will perform the leveling cycle at each freq. change.
  - If the box is cleared, when you change frequency leveling is not performed. Because the setting is "exposed" on the run dialog, you can effectively override the Freq Step 1 setting as the action runs. The action's Edit setting just determines the runtime setting's initial state.

- **Display VSWR (Net Power Only)** — When both forward and reverse power is measured, the software can calculate the VSWR of the transmitted signal. Selecting this check box causes the VSWR to be calculated and displayed as the test is running. This is very convenient for determining the influence that the EUT has upon the test setup.
- **Amplifier Always on** — When the Immunity test starts, if an amplifier is defined, the software will switch it on (in operate mode). If this check box is selected, the amplifier will not be addressed until you exit the immunity step, at which point it will be turned off (put in standby mode). If this is not selected, when the pause button is clicked the amplifier will be put in standby, and when **RF On** or **Start Auto** are clicked, the amplifier will go into operate mode.

Freq Steps 2 Tab

This tab lets the user control several additional functions/methods/algorithms during the Immunity test.

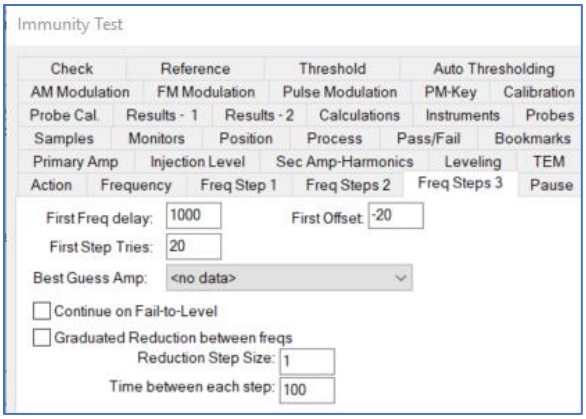


- **Power on when entering Manual Mode** — When the operator clicks the manual button during an automatic Immunity test, the default behavior is to pause the test with RF off. Check this button if you want RF on when entering manual mode.
- **Do Not Exceed Std Field (Ref) in Manual** — When in manual mode, the operator can click the up/down buttons to raise or lower the field level. If this check box is selected the software will limit the up button to the standard field. With this check box selected, you cannot raise the field level above the limit.
- **Use Previous Amp when Leveling failed on Previous Freq (no level mode)** — When not in leveling mode with a secondary leveling loop (*Sec Amp-Harmonics tab*), if you failed to level at a frequency the normal behavior for the next frequency would be to start at the signal generator level for the primary amp field.

If you failed to level in the secondary loop, this might make you start too high. Select this check box for the secondary loop signal-generator level to be used as the first step. The Max Step size on the Level tab will control the step size for this leveling loop. If you are doing no leveling for the primary amp and you have a secondary amp defined, it is recommended that the Max Step be set to 1 dB.

- **Read Sensors with Modulation On** — When in a leveling loop, sensors are normally read prior to the modulation being turned on. Select this check box if you want to read sensors with the modulation on.
- **Use Offset when in Manual** — On the Leveling tab the offset value is the amount in dB that the signal generator is stepped down at the start of each frequency. When in no leveling mode, this value is ignored unless this check box is selected.
- **Step to Next Frequency on Fail (Manual Mode)** — When the Fail button is clicked, in manual mode the normal behavior is to mark this frequency as a failure, and go to the next frequency. If this check box is selected, this frequency will be marked as a failure, but it will not progress to the next frequency.
- **Step to Next Frequency on Fail (Auto Mode)** — When the Fail button is clicked, in auto mode the normal behavior is to go to manual mode and pause the test. If you click Auto Run at this point, it will retest the current frequency. If this check box is selected, the software will automatically step to the next frequency when Fail is clicked. This allows you to restart the test.

Freq Steps 3 Tab



- **First Freq Delay** — Some power meters and probes have a long leveling time when first subjected to power (due to internal averaging methods).
    - This will cause the first readings to be too low. This value controls a leveling delay for the first frequency step that can be different from the values set on the *Instruments tab*.
- **First Offset** — The first step of a leveling loop sometimes requires different settings than the remaining steps in a test. Because of sensitivity of a test set or unknowns in the leveling loop, the general condition in the first step can be radically different from the conditions in the later steps.
    - The first step offset is a value in dB that the software will automatically step down from the first estimated signal generator drive level.
    - For example, to set the level to half of the target of 100V/m E-field, using the formula below, assuming a 60dB gain amplifier, the 0 offset value out of the sig gen would be -6.99dBm. NOTE: This **must** be accounted for in the 'Calibration tab'.
    - If the desired starting field level is 50V/m, an entry of -6 would set the sig-gen starting value at -12.99.
    - NOTE: In the formula below the height value is for a TEM cell, for all other setups it should be 1m.

$$\text{SigOut} = 10 * \log 10\left(\frac{(\text{Target}*\text{Height})^2}{50*10^{\left(\frac{\text{Gain}-\text{Offset}}{10}\right)}}\right) + 30$$

- **First Step Tries** — The number of tries that will be attempted for the first step before an error is declared. When using Use Previous Amp for Best Guess, the Number of Tries from the *Leveling tab* is used to determine the overall number of tries for a leveling loop.

  On the first step, there is no previous amplitude so the software will start at a much lower level and begin leveling up. This value allows you to let the first step have more tries than the remaining steps. This will allow you to speed the leveling loops of the remaining steps without causing a problem on the first step.

- **Best Guess Amp** — You can use this data element to control the initial signal generator level for each frequency step. It is mutually exclusive with the Use Previous Amp for Best Guess check box on the *Freq Step 1 tab*. If that check box is not selected, then this data element controls the initial signal generator level (less than the offset).

- **Continue on Fail to Level** — Allows the user to continue with the leveling steps even when a step fails to level.

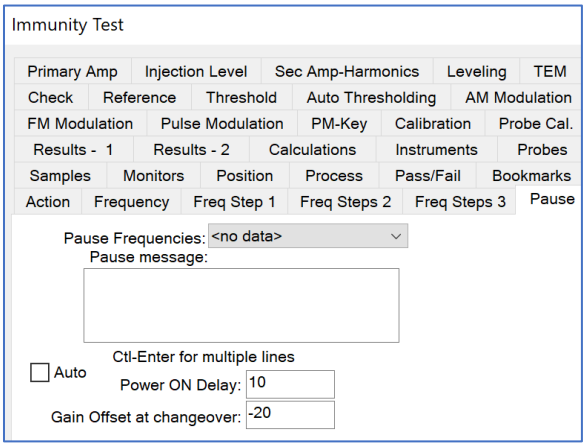
- **Graduated Reduction between freqs** — There are certain specifications that require the amplitude of the signal to be reduced to a fixed level before stepping to the next frequency. This controls the settings for the shape and timing of that step down.

- **Reduction Step Size** — The value in dBm for each step of the reduction. The number of steps is controlled by the offset value on the *Leveling tab*.

- **Time between each step** — The dwell time between each reduction step.

Pause Tab

During some tests, it is useful to prompt the operator to change hardware, throw a switch, or change an antenna. If there are no other substantial differences, you can use this tab to define conditions when a pause will be generated. The pause is defined by two conditions: the frequency at which a pause is requested (there can be multiple frequencies), and the message to display.



- **Pause Frequencies** — You will need to define a data element that has the frequencies, or frequency, at which you want to generate the pause. You can create this data element with the *Direct Entry action*. The software will generate a pause when the next frequency is equal to or greater than the pause frequency.

- **Pause Message** — Enter a text message that will be displayed when a pause is generated.

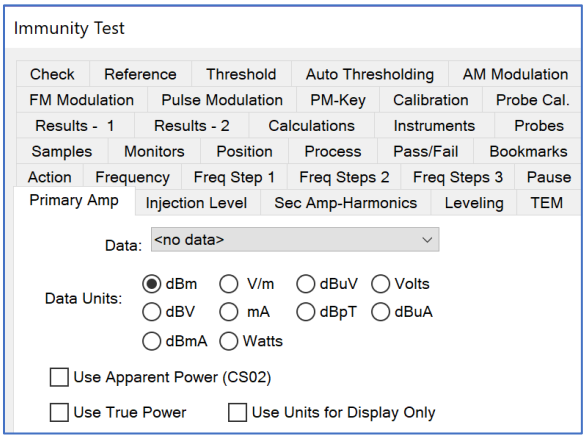
- **Auto** — When this check box is selected and we transition across a paused frequency, the software will perform the gain offset and dwell time and then automatically continue. This setting is primarily designed for dual band amplifiers where the gain of the amplifiers is dramatically different between the two bands.

- **Power ON Delay** — The time between turning the signal generator on and the next point where any valid readings are taken. This is commonly used when you have an amplifier that has a short dwell between the application of RF energy and the output of the full potential of the amplifier. Controls a leveling delay for the pause frequency step that can be different from the values set on the *Instruments tab*.

- **Gain Offset at changeover** — The amount in dB that the current signal generator value will be offset as we apply power after the pause frequency. This value can be positive or negative; positive will reduce the signal generator drive-level, negative will increase the drive level.

  After the dwell time, the software begins the leveling loop again. At this point, you might have made a change in a passive component that affects the efficiency of the current hardware setup. For example, you changed the injection probe, and the new injection probe is substantially more efficient. In this case, if we begin the leveling loop at the previous amplitude level, there is a strong risk of overdriving and potentially damaging the EUT. The gain offset lets you specify the value in dB that the output of the signal generator will be reduced as it begins the leveling loop anew.

Primary Amp Tab



This tab is used to select the amplitude data. Normally, this data element would match the data element used for frequency (if that option was set), but does not have to match.

If you are using No Leveling on the *Leveling tab*, then this data element will indicate signal generator levels. If using Power Meter, this data element will indicate the power level desired (or specification limit converted to dBm).

If using Probe, this will reflect the specification limit in volts/meter. This data element will be interpreted to match the instrument used for leveling. If you use Power Meter for leveling and are using a voltmeter or current meter, then this data element needs to be the value desired on this instrument (for example, 1 V AC or 1 A).

NOTE: If the Data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Data** — Allows the operator to select the desired amplitude data to be used in the test. The frequency component of the amplitude data element must cover the same range as the data element defining the *Frequency tab*, but they do not have to be the same element. In most tests, the same data element will be defined for both frequency and amplitude.

  For example, if the data element has the following points:

2e8	10
2.2e8	10
2.4e8	3
2.6e8	3

  The first frequency, 200 MHz, would have a leveling point of 10 V/M if probe leveling was used. If power meter leveling is specified then the system would level to a power meter reading of 10 dBm (the units used by a power meter). Keep this in mind when defining a test. If you are using a power meter and do not have a probe, you must calculate the expected field strength using the appropriate antenna factors to create this data element.

- **Data Units** —This selection will set the display of the dialog box to reflect the correct units.

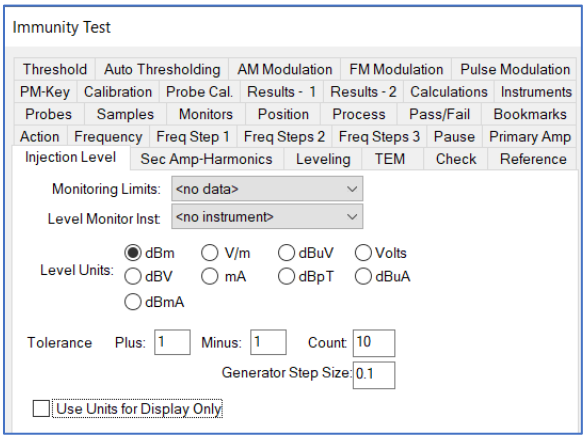
- **Use Apparent Power (CS02)** — When performing a MIL STD 461A Test to Notice 3 you are required to measure apparent power which is defined as:

  Voltage + Current/2 – 90 dB.

- **Use True Power** — When performing certain specialized MIL STD 4611E tests you are required to determine power by summing current and voltage.

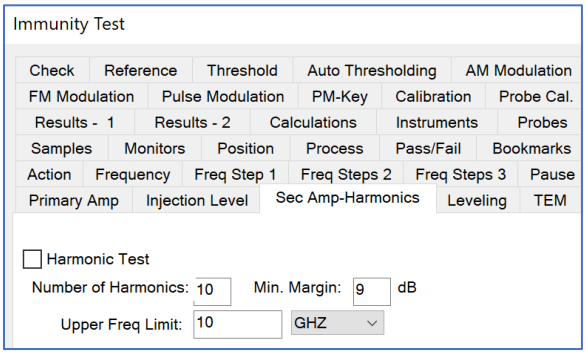
Injection Level Tab

The Injection Level tab is used to define secondary leveling loops, such as Bulk Current Injection (BCI), where there are two standards, the Injected standard and an Induced standard.



- **Monitoring Limits** — The Monitoring Limits data element is used in conjunction with the Level Monitor Inst value as a means of limiting the power/current.
- **Level Monitor Inst** — Defines the instrument that will be used to record the power/current in conjunction with the Monitoring Limits data element.
- **Level Units** — Select this unit to use for display purposes and for instrument control (such as power meters) where units need to be specified.
- **Tolerance Plus/Minus** — You can control the tolerance for the secondary data element. The tolerance is an absolute value.
- **Count** — The number of steps for the secondary loop before an error is declared. The loop will start leveling down by the generator step size. If it cannot achieve the desired pass conditions before the count is reached, then an error message will be displayed for the operator.
- **Generator Step Size** — Controls the step size used when leveling down in the secondary leveling loop. When you exceed the secondary level, the software will enter a loop where it reduces the generator level by the step size shown here. It will then measure the secondary level again. If it is still above the limit, it will reduce the level and repeat these steps until the correct level is achieved or the count is exceeded.

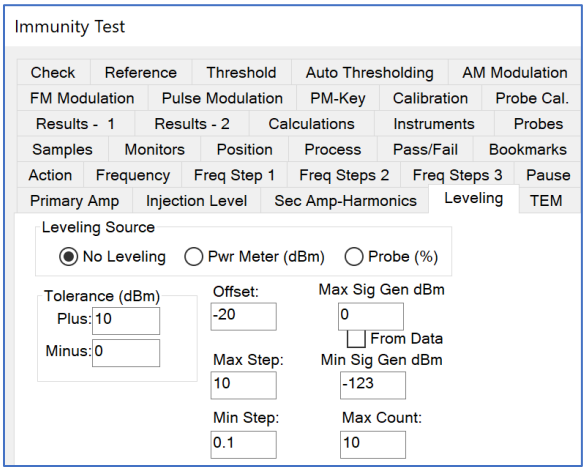
Sec Amp-Harmonics Tab



In automotive testing, the harmonic test selection is used to monitor up to the tenth harmonic of the fundamental frequency. This, along with the minimum margin setting, will limit the field strength accordingly.

- **Harmonic Test** — Select this check box to activate the harmonic test.
- **Number of Harmonics** — Specifies the number of harmonics to be tested at each frequency.
- **Min. Margin** — Sets the tolerance of the field strength for the harmonic test.
- **Upper Freq Limit** — Controls a value above which no harmonics will be performed. For example, if you are doing a GM9112 BCI test up to 400 MHz, but the spectrum analyzer does not go above 1 GHz, then you can limit harmonic test above 100 MHz (assuming 10 harmonics) to not exceed analyzer capability.

Leveling Tab



This tab is used to select the leveling methodology for this run. The TILE! system supports leveling using a probe, a power meter or a pre-calculated field. Pre-calculated fields are only used with control field transducers, such as TEM or GTEM cells.

In these cases, the field can be predicted (at certain frequencies) very accurately. TILE! allows you to create a calculated field in these structures and record the probe level, in effect calibrating the probe.

- **Leveling Source** — Allows the operator to select which instrument will control the leveling cycle. Of the three choices, no leveling causes the test to playback fixed dBm levels from a signal generator with no leveling taking place. Probe and power meter leveling cause specific leveling loops to execute where the software searches for the desired level.
- **Tolerance** — Allows the operator to select the desired tolerances for the leveling process. For example, if you are trying to create a 3 V/M field with a probe then a plus 15, minus 0 would give you a tolerance of 15% starting at 3 V/M. This equates to a minimum acceptable reading of 3 V/M and a maximum of 3.450 V/M.

  This is probably too tight a tolerance for most probes to level to but the user should experiment with these settings to obtain the tightest acceptable margin for the instrumentation. A symptom of too tight a tolerance would be a leveling loop, which jumped below and above these numbers without ever landing within the zone. You will exceed the Max Count specified without leveling. Raise the margin slightly and retry.
- **Offset** — Sets the offset in dBm to the expected signal generator levels for the next frequency. This defines a minimum offset to the calculated field for safety purposes. This value is subtracted

from the expected signal generator level, so a negative value will increase the signal generator level. There are often large swings in field strength given different frequencies in the same setups.

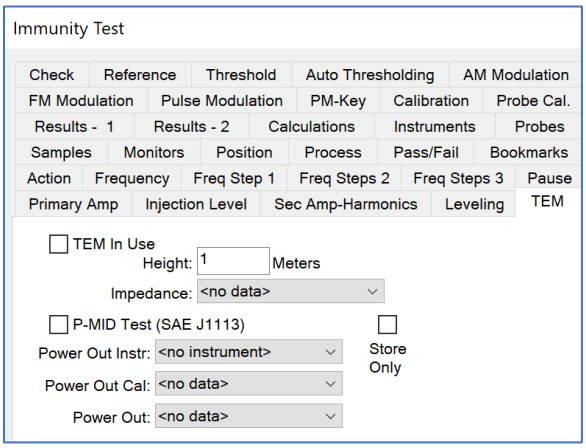
This number determines how far below the calculated value to place the signal generator on the first reading. The system will step up in maximum steps of 10 dB until it begins to receive valid readings from the probe or power meter. Subsequent leveling will follow in internal algorithm. When using the Best Guess Amp data element, the value is reduced by the offset. Typically, a negative number will be used to set a lower level to approach the target.

- **Max Step** — When in a leveling loop, this controls the largest step allowed to the signal generator. This value might be set quite differently for different types of tests. When the software sets the first signal generator level, if the value returned (by either the power meter or probe) is very low, compared to the limit, the software will take a large step. This value limits the size of the step. When performing some secondary leveling loops (either on the *Sec Amp-Harmonics tab* or the *Check tab*), this value also determines the step size for these loops.
- **Min Step** — Sets the minimum amplitude increment that the signal generator will perform.
- **Max Sig Gen dBm** — Controls the maximum RF level that will be output from the signal generator. Use this setting to protect the front end of the amplifier from overload. A typical amplifier cannot handle more than 0 dBm into the input of the amplifier. A value of zero will limit the software by halting the leveling if the anticipated signal generator level will exceed 0 dBm.
- **From Data** — To use a table of sliding values for the maximum dBm value, select this check box. This is most common in low frequency (audio) applications where the maximum capability of the amplifiers is a function of frequency. Selecting this check box has the same effect as a third leveling loop, based upon power inputs.
- **Min Sig Gen dBm** — Some signal generators do not have an RF off mode and the software will set a minimum output; this value defines the minimum output for the signal generator.
- **Max Count** — Sets the maximum number of retries to establish a level measurement reading before the system reports an error. This field needs to be over 10 in situations where you are seeking a large field or in cases in which the margin is extremely small.

  For example, if you are trying to create a 3 V/M field with a 500-Watt amplifier, you will find that small ripples in the amplifier cause relatively large changes in the offsets at this level. You will probably need to give the system more tries to level. An indication of this problem would be a failure to level, but levels quickly if you click **Retry**. This field is also used in the secondary amp and check loops to determine the number of tries.

TEM Tab

TEM or GTEM cells, by design, are calibration devices. The field strength in these cells can be accurately calculated if the height of the septum, impedance, and net power are accurately known.

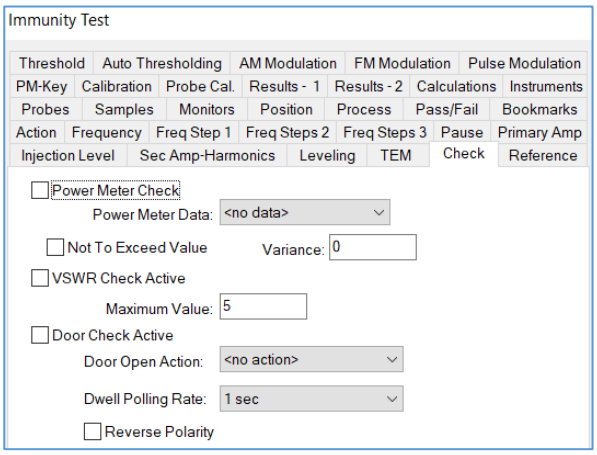


The general formula is **P*I = E^2/H**, where P is the power in Watts, I is the impedance in ohms, E is the field strength in V/M, and H is the height of the septum.

- **TEM In Use** — When using TEM devices such as Striplines, TEM cells, or GTEM cells, you can perform a test in volts/meter using a power meter if you have height and impedance information. Select this check box, enter the appropriate information, and then use the power meter leveling-mode on the *Leveling tab* to level to a value in V/M, not power.
- **Height** — Sets the septum height if you are using a TEM or TEM type device. The internal algorithm for calculating signal levels is adjusted by this reading. If the septum is less than 1 meter (a value of 1), then the calculated signal generator level will be increased by the inverse. If the septum height is larger than 1 meter, the signal generator level will be reduced by the inverse. This value is used in the previous equation to adjust the field strength to an appropriate value.
- **Impedance** — Allows the operator to select a data set as a defined CSV data element. The impedance is the critical in the accuracy of the methodology. You cannot assume a 50-ohm cell. Measured values are necessary. If you do not have these values, it is possible to use TILE! to determine closely approximated values.
- **P MID Test (SAE J1113)** — The power at the midpoint (P MID) test is a Stripline test where the amplitudes are determined by averaging the power in and power out of a Stripline (Tri plate).
- **Power Out Instr** — Selects the instrument to be used to measure the output power. If net power on the output is desired, you will need to use a special driver.
- **Power Out Cal** — Contains the correction factors to be added to the power meter reading to give you a corrected value.
- **Power Out** — Selects the data element used to store the output from the power meter.
- **Store Only** — If this check box is selected, the power out of the Stripline is measured and stored but not used in the leveling loop.

Check Tab

This tab is used for two unrelated purposes: the power meter check is a modified leveling method, and the door check active is used as a safety feature.



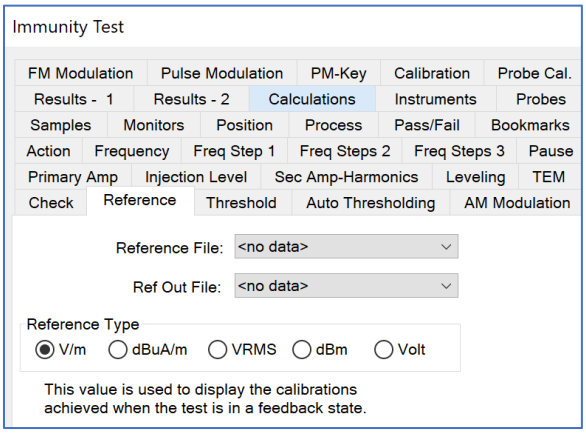
- **Power Meter Check** — A modified form of testing in which you are running no leveling type test, but still want to verify the actual power level present.
- **Power Meter Data** — Data element reflects the expected power meter level. This value will be referenced against the power meter 1 instrument. If you want to perform any other type of checking, either using another instrument or net power, you must configure the *Sec Amp-Harmonics tab* and use that method.
- **Not to Exceed Value** — If this check box is selected and the power level is higher than the desired level, the software will level down to the desired power meter level.
- **Variance** — Reflects the +/- tolerance which will be applied to the power meter reading in determine acceptance.
- **Door Check Active** — When this check box is selected, the action specified as the Door Open Action will be accessed to determine the switch position. If the switch position indicates an open door, the test will be paused (RF turned off). Then an error message will be displayed for the operator.
- **Door Open Action** — This is an icon on the flowchart setup with the appropriate switch settings to measure the open or close position of the door. These are normally accomplished with either Switch actions or serial actions.

Reference Tab

When performing a test, sometimes the unit of the standard is different from the unit currently being used for the test.

For example, if you have calibrated a field using a field probe (to 3 V/M) and saved the signal generator level (or power meter level), you might then perform a test using these values in either the no level or power meter leveling-mode. You will be leveling then in dBm but the standard is still in V/M.

The reference field is related to the current leveling units which reflects the actual standard being measured. When selected, the software changes the display to show the value and units derived from the calibration.



If you are feeding back a set of previously calibrated signal generator levels, the display will show 3 V/M, not –15.8 dBm. This is much more useful. The reference field must be related to the data element used in the *Primary Amp tab*. If there is a discrepancy in this relationship, the displayed values will be incorrect.

- **Reference File** — Displays the field values achieved during the calibration when the test is in a feedback state. The most common use of this field would be to display the field probe level achieved during the calibration step referenced to the current leveling method and its related value from the calibration. Assume that you level to 3.03 V/M with a forward power of 23.0 dBm during Calibration. If you level during the test to 23.2 dBm then the displayed field strength would be:

$$Est\ Fiel = alog10\left(\frac{dB\Delta + 20 * log10(RefVm)}{20}\right)$$

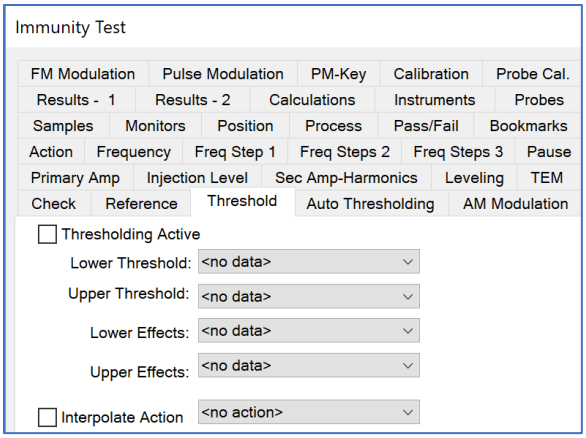
Or

$$3.1 = alog10\left(\frac{0.2 + 20 * log10(3.03)}{20}\right)$$

- **Ref Out File** — Selects the data element to save the reference value. If you go to manual mode and adjust the signal generator to change the field level, the reference is recalculated to display and adjusted value. This value is stored if you pass or fail this step. This allows you to graph the estimated field generated at each step of the test.
- **Reference Type** — Selects the recorded level; these values are for display purposes only. You must have selected appropriate methods of leveling and amplitude/reference data elements to properly level.

Threshold Tab

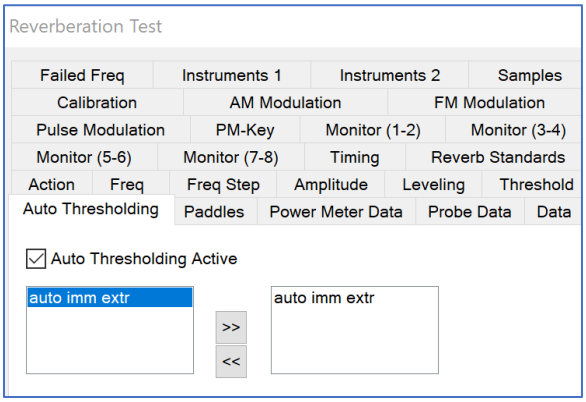
When this function is activated, the software will display the thresholding options during the Immunity test. Thresholding assumes the upper and lower thresholds are identical unless the operator takes manual control and overrides these values. When activated and in manual mode, the operator can identify a level as being the upper or lower threshold and this value will be saved.



- **Thresholding Active** — These settings only apply when the operator is in manual (pause) mode. The operator can then specify levels for upper or lower thresholds. When in automatic mode, the upper and lower thresholds are assumed equal until manually overridden.
- **Lower Threshold/Upper Threshold** — Selects the data element to store the thresholding values. There must be a data element selected for both upper and lower or the software will not allow this option to be activated.
- **Lower Effects/Upper Effects** — Selects the data element to store an alpha code for effects. Some standards allow or require the operator to identify failures by type. Assigning an alphabetic code letter to different failures allows the operator to identify the failure without having to enter the same text repetitively. These data elements must be defined as word type data at this step.
- **Interpolate Action** — Certain standards which level to values that are not measured directly. For example, measuring induced power when no current probe is attached to the line of interest in a

BCI test. When this check box is selected and an appropriate Threshold action is set up, for each injected power value an interpolated induced current will be derived and used as part of the leveling loop.

Auto Thresholding Tab



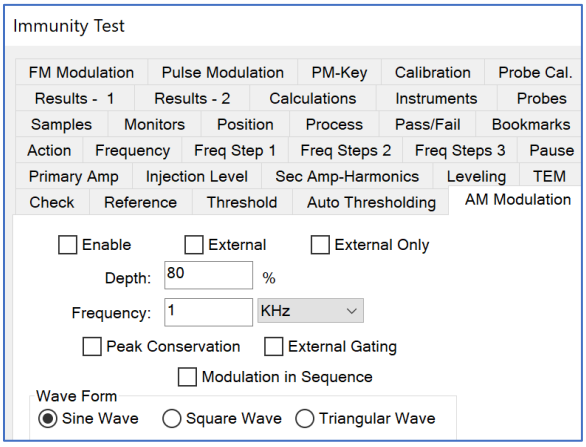
The software allows the user to configure a threshold value for an unlimited number of instruments. As the Immunity test runs, each instrument is read to determine the current value. If the value exceeds the value expected, the software will automatically level down until the EUT passes. If more than one auto threshold is defined, the software will check the thresholds in the order selected until all pass.

See the *Auto Thresholding* action for more details.

NOTE: Must select the **Auto Thresholding Monitor Active** box to activate this feature.

- **Available/Selected** — It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

AM Modulation Tab



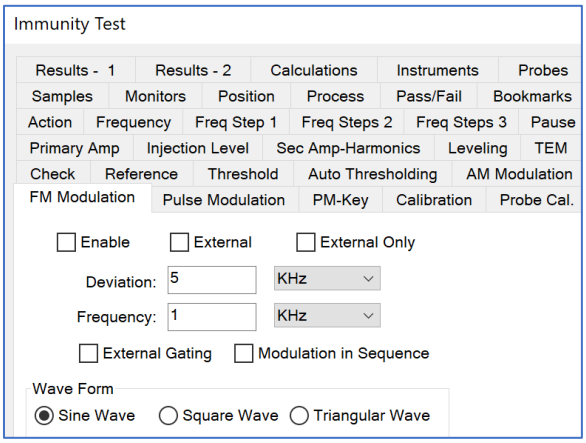
This tab to turns on AM modulation during the Immunity test. The action will turn on AM modulation at the signal generator and turn on the appropriate internal/external switch (if the generator has this feature).

- **Enable** — Select the check box to turn on (enable) AM modulation. Once you select the check box, you must complete the **Depth** and **Frequency** boxes.
- **External** — If this check box is selected and modulation is enabled, the software will access the external input of the signal generator. On some signal generators, this will cause a combination of internal and external modulations.
- **External Only** — If this check box is selected and modulation is enabled, then only the external input modulation will be turned on for all signal generators.
- **Depth** — Determines the relative amplitude of the AM modulation. It is stated in percent, as are most standards. A typical standard is the IEC 1000-4-3, which specifies 80% modulation. The amount of modulation is related to the pitch of the voice being emulated. The standards are trying to duplicate the human voice on a radio transmitter. If the standard is not stated in percent, you must convert the number to a percentage before entering it.
- **Frequency** — In stating a sound (using depth), you must specify how often the sound is manufactured. This is stated in Hz or kHz.

- **Peak Conservation** — There are two methods of handling AM modulation; in the commercial standards, modulations are applied at the CW value causing the peak of the wave to be above or below the peak value, depending upon the modulation period. In other standards, the highest value of the modulated waveform should not exceed the peak value in CW. Peak conservation will reduce the signal generator output level prior to turning on modulation by an amount appropriate for the modulation percentage, so that the peak of the modulated waveform does not exceed the CW amplitude.
- **External Gating** — Select this check box if you need the modulation to be controlled by an external gating signal.
- **Wave Form** — Denotes how the carrier wave will be varied in accordance with the characteristics of the modulation signal.

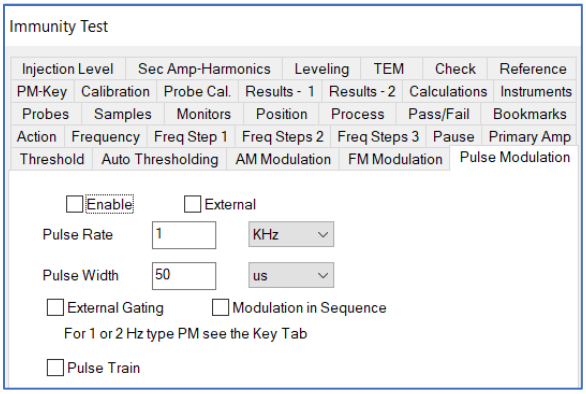
FM Modulation Tab

This tab selects and configures FM modulation. The generator must be capable of this function or these settings will have no effect.



- **Enable** — Select the check box to turn on (enable) FM modulation. Once you select the check box, you must complete the **Deviation** and **Frequency** boxes.
- **External** — If this check box is selected and modulation is enabled, the software will access the external input of the signal generator. On some signal generators, this will cause a combination of internal and external modulations.
- **External Only** — If this check box is selected and modulation is enabled, then only the external input modulation will be turned on for all signal generators.
- **Deviation** — Relates to the relative amplitude of the FM modulation. It is specified in frequency deviation around the primary frequency. If the primary frequency is 100 MHz with a 5-kHz deviation, then the FM modulation will cycle between 99.95 MHz and 10.05 MHz. This simulates the human voice carried on a FM radio transmitter. Enter an appropriate number in the box and select the frequency component from the list.
- **Frequency** — In stating a sound (using depth), you must specify how often the sound is manufactured. This is stated in Hz or kHz.
- **External Gating** — Select this check box if you need the modulation to be controlled by an external gating signal.
- **Modulation in Sequence** — The default behavior in TILE! is for all the desired modulations to be turned on before the dwell cycle starts. If this check box is selected, the software will turn on the first modulation (AM first, FM second, pulse third) and dwell the desired amount of time prior to moving to the next modulation.
- **Wave Form** — Denotes how the carrier wave will be varied in accordance with the characteristics of the modulation signal.

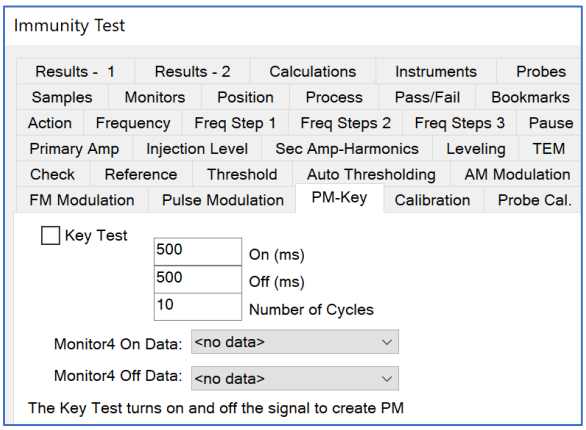
Pulse Modulation Tab



This tab selects and configures pulse modulation that is output by the signal generator on top of the current frequency and amplitude of the CW output. The generator must be capable of this function or these settings will have no effect.

- **Enable** — Select the check box to turn on (enable) pulse modulation. Once you select the check box you must complete the **Pulse Rate** and **Pulse Width** boxes
- **External** — Select this check box if you are using an external modulation generator and need to feed this into the signal generator. Control of the external modulation generator can be configured using the *VISA* action.
- **Pulse Rate** — A pulse modulation is defined as an on/off width of a certain time over a repetition rate. The repetition rate is measured in frequency units.
- **Pulse Width** — The pulse is defined as an on/off width of a certain time over a repetition rate. The parameter is the width of the pulse, and the width is defined in time units.
- **External Gating** — Select this check box if you need the modulation to be controlled by an external gating signal.
- **Modulation in Sequence** — The default behavior in TILE! is for all the desired modulations to be turned on before the dwell cycle starts. If this check box is selected, the software will turn on the first modulation (AM first, FM second, pulse third) and dwell the desired amount of time prior to moving to the next modulation.

PM-Key Tab

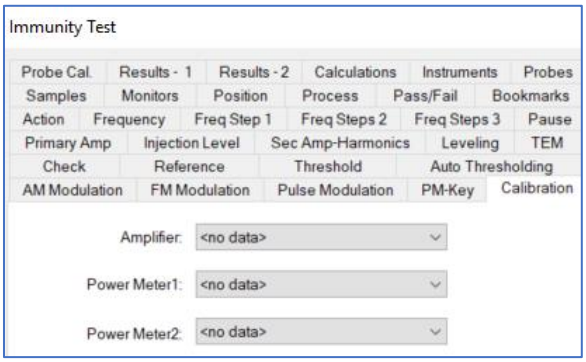


Some standards require 1–3 Hz square wave modulation on top of other forms of modulation. The setting on this tab let you generate a pulse modulation using the output off/on function of the signal generator. It will only work if the signal generator can respond quickly enough to GPIB commands.

- **Key Test** — When this check box is selected, the signal generator will emulate the keying and un-keying of a transmitter at the identified cycle time for the defined number of cycles.
    - **On/Off (ms)** — Defines the number of milliseconds that the RF will be on/off (cycle time).
    - **Number of Cycles** — The pulse shape of on and off signals is repeated the number of cycles defined here.
- **Monitor 4 On/Off Data** — Data elements used to record the results of the key test. If a data element is selected, and an instrument is shown in the Monitor 4 section of the *Monitors tab*, we will read the monitor during the off cycle and save the value.

Calibration Tab

This tab is used to select the calibration data sets for the selected instruments. These data elements should contain the appropriate correction factors for this instrumentation.

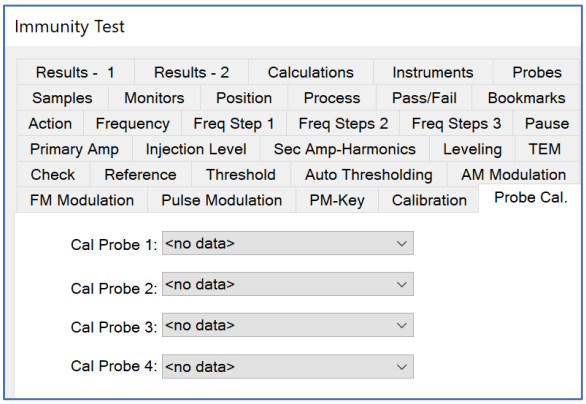


- **Amplifier** — If the amplifier calibration is not entered, the default value is 0 (gain of zero). This will cause serious overshoot if you have an amplifier present in the system. If you are unsure of the amplifier gain, enter a preset of at least 40. The calibration table should be the gain of the amplifier in dB, not watts.
- **Power Meter 1/ Power Meter 2** — Power meter calibration data needs to include cables and couplers. The default value is zero if no data element is selected. You can use TILE! to calculate the calibration data by setting up an Immunity Calibration action with no leveling, establishing a 0-dB signal generator level. Physically hook up the complete system, but install a 50-ohm load in place of the transducer. The power meter readings taken under this circumstance represent the system loss from prior to the power meter.

Probe Cal. Tab

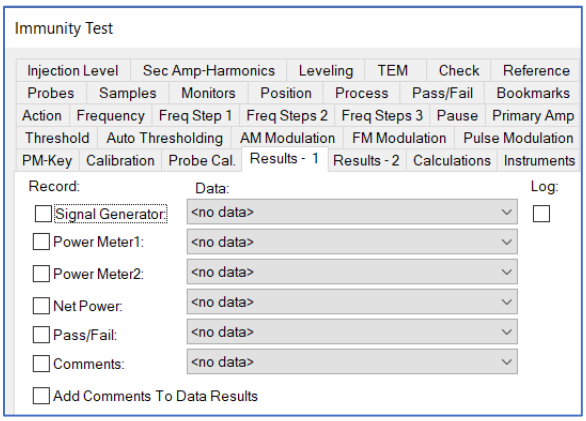
There can be up to four different probes used during an Immunity test. An appropriate Calibration data element must be specified for each probe. The probe calibration table should be the correction information from the manufacturer or a preset value of one.

Never use a value of zero since probe correction factors are multiplicative. We take the reading from the probe and multiply the probe correction factor. A value of zero would cause all results to be read as zero. Probe calibration tables tend to be in the +0.8 to +2 range.



Results - 1 Tab

This tab is used to select which instruments will be recorded and where the recorded results will be stored. The data elements selected must be defined as measurements.



- **Record** — Select the check box for each type of instrument data you want to record. If not selected the results are not saved. This does not mean the instrument is not read, only that the results are not saved. If you choose to level with a probe and do not record the probe levels, then you will have to assume that the desired amplitude was the actual level.

  If you record the probe level you can display this later to show the actual field created compared to the standard. If you are running a no leveling test using calibrated inputs, you might want to record the power meter or probe readings for reference purposes. This will allow you to compare graphically the field established during the calibration run and the field created during the test. If the EUT has an unusually large impact upon the field generation, this will show it.

- **Data** — If a check box is selected to record, a corresponding data element must be selected or the system will record a failure and not run.

- **Signal Generator** — Store the amplitude of the signal generator when the leveling loop has completed. This information is useful for duplicating testing or verifying results.

- **Power Meter1/ Power Meter2** — Store the amplitude of the forward/ reverse power during this test. Power meter1/Power Meter2 always refers to the forward/ reverse power if net power is being measured. The forward/ reverse power value is determined by reading the instrument identified on the *Instruments tab* and adding the calibration value found on the *Calibration tab*. The corrected value is saved.

- **Net Power** — Store the net power measured during this test. You must have the Net Power check box, and both a forward power instrument and a reverse power instrument identified on the *Instruments tab*. The net power value is calculated by the following formula:

  Pf (Watts) = ((Pfm^ (-10))/10)/100000

  Pr (Watts) = ((Prm^ (-10))/10)/100000

  Pnet (dbm) = 10 * Log ((Pf - Pr) *100000)

  Where

  Pf = Forward Power (in Watts)

  Pfm = Forward Power measured in dBm

  Pr = Reverse Power (in Watts)

  Prm = Reverse Power measured in dBm

  Pnet = Net Power (in dBm)

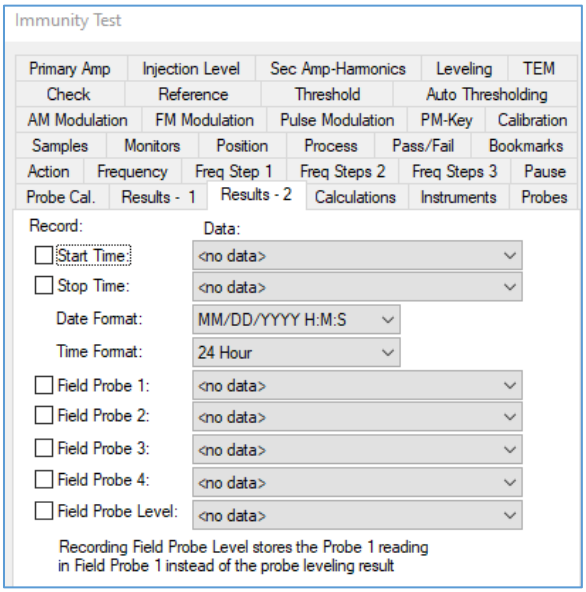
- **Pass Fail** — The Pass/Fail data element stores a value that reflects whether the Pass or Fail buttons were selected during the test. If the step passed, a value of one is entered. If the step failed, a value of zero is entered. This allows the user to perform math that easily distinguished

the failed frequency. A simple equation (data element < 1) will identify those frequencies where a failure occurred.

- **Comments** — When the operator pauses an Immunity test, there must be the option to enter comments. A separate comment can be store for each frequency. To store the values, a word-type data element must be selected here.

Results - 2 Tab

This tab is a continuation of the Results - 1 tab.

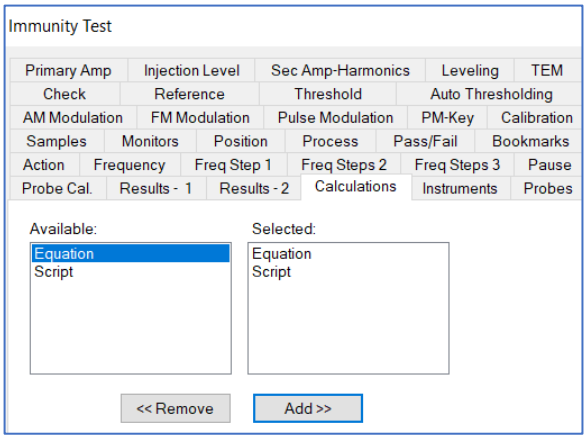


- **Record** — Select the check box for each type of instrument data you want to record.
- **Data** — If a check box is selected to record, a corresponding data element must be selected or the system will record a failure and not run.
- **Start Time/Stop Time** — The start and stop times are time strings requiring a word type data element which can be stored at each frequency. The start time and stop time relate to the modulation envelope.
- **Field Probe/ Field Probe 1/ Current Probe** — This output stores either the probe level result, the Probe 1 result, or the Current Probe result.
    - The data recorded will be the probe level result if Probe Leveling is enabled on the Leveling tab, at least one probe is enabled on the Probes tab, and the Probe Leveling Method on the Probes tab is Min, Avg, or Max.
    - The data recorded will be the Probe 1 data result if the Leveling Method is not Probes, or the Leveling Method is Probes and the Probe Leveling Method is Probe 1.
    - The data recorded will be the Current Probe data result if Probe 1 is not enabled and there is a Leveling Monitor Instrument enabled on the Injection Level tab.
- **Field Probe 2/ Field Probe 3/ Field Probe 4** — These outputs store the result of Field Probes 2, 3 and 4, respectively. Each Field Probe output is determined by reading the corresponding instrument on the Probes tab and multiplying by the calibration value found on the Probe Cal tab. The corrected value is saved.
- **Field Probe Level** — This data element stores the result of the probe leveling when Probe Leveling is enabled on the Leveling tab. Recording this output also causes the Field Probe 1 data element to record the Probe 1 reading instead of the probe leveling result.

Calculations Tab

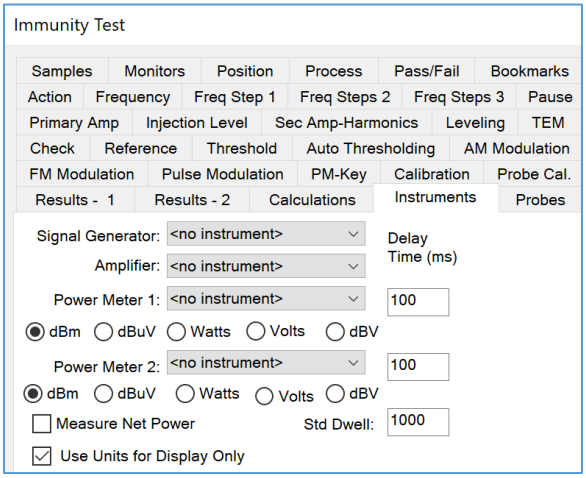
This tab runs calculations on the fly so that they can be shown on the graph in real-time.

This tab is similar to other calculation tabs and will not fully be described here. See the *Math Data Tab* for the full description.



Instruments Tab

This tab is used to select the specific instruments that will be used in the Immunity test. Certain instruments may have functions that imitate others. For example, many spectrum analyzers can be used as power meters.



NOTE: If the Signal Generator is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly. If an instrument other than the signal generator is selected, it must have a calibration entered on the *Calibration tab*. If the instrument is **not in use**, make sure this field is set to **No Instrument**.

- **Signal Generator** — If you select an instrument that does not have the characteristics of a signal generator, the action will not work properly. Some spectrum analyzers have built in tracking generators that can be used for CW type testing using this action. Since a spectrum analyzer can also be a power meter, you can select the same instrument for both signal generator and power meter.
- **Amplifier** — Select an amplifier if the amplifier has a GPIB bus. With this feature, TILE! will put the amplifier into operate mode while running the test, change the bands as needed, and put the amplifier back into standby when the test is complete.
- **Power Meter1/ Power Meter 2** — Within the TILE! system, Power Meter is assumed to be the forward power measurement from a direction coupler. It does not have to be, but be cautious when using a non-standard configuration.

  For the two power meters, you can also specify units for their readings. These units are only valid if the instrument supports them. When using a spectrum analyzer for a power meter, you can read in dBuV (for doing leveling to dBuA). Some power meters will read in Watts or Volts.
- **Delay Time** — Allows the user to enter a delay time (in milliseconds) before a reading takes place. This is particularly useful due to the long settling time inherent with power meters and probes.

It is important to keep in mind that most power meters and probes are designed as constant output devices. When you have an amplitude or frequency change that requires settling these instruments will continue to respond to requests by sending the last valid reading. They do not warn you that they are in the middle of a triggering cycle. For this reason, the results can be inconsistent.

- **Measure Net Power** — This check box requires the use of two instruments. When forward and reverse powers are recorded, the software can calculate the net power reading. The result is recorded in the data element identified on the *Results tab*.
- **Std Dwell** — This setting is the same as the Mod dwell time in the *Process tab*, and is therefore informative, not functional.
- **Use Units for Display Only** — The units in the immunity actions have been designed to convert from dBm (50 ohms) to the selected unit. For example, if the target is 20 Volts ms from an O'scope, TILE! will read the value on the scope and use the dBm to Volts equation to convert the reading 'From' dBm 'To' Volts. This would end up using ~2.3Vrms instead of the 20Vrms level:

  PdBm=10log (V2/ (50*0.001)

  Where 50 is impedance and 0.001 is the relative to 1mW (for dBm).

  Therefore, a level of 2.3 on the scope would be interpreted as:

  20 (dBm (not volts) = 10log (2.32/0.05)

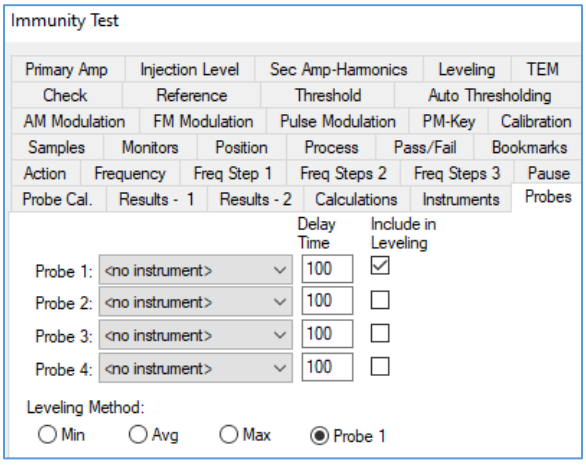
  The reading on the user interface will show 20 'V' though the measured value from the O'scope will only be 2.3Volts

  Selecting the 'Use Units for Display only' will read the actual value from the O'scope without converting to dBm, so the reading on the O'scope will be 20 Vrms and the display and recording of the value will be correct (20).

  NOTE: The *Primary Amp tab* should show the units you want to display (along with Units for Display only) on the user interface when running the test.

Probes Tab

This tab allows you to identify up to four different probes for measurement. During a test, these probes will be read and the values stored depending upon the choices here and on the *Results - 2 tab* and the *Leveling tab*.



- **Probe 1/ Probe 2/ Probe 3/ Probe 4** — You do not have to have all four probes, nor do they need to be identified in order. They are treated independently for reading and saving purposes.
- **Delay Time** — Allows the user to enter a delay time (in milliseconds) before a reading takes place. This is particularly useful due to the long settling time inherent with power meters and probes.

  It is important to keep in mind that most power meters and probes are designed as constant output devices. When you have an amplitude or frequency change that requires settling these

instruments will continue to respond to requests by sending the last valid reading. They do not warn you that they are in the middle of a triggering cycle. For this reason, the results can be inconsistent.

- **Include Leveling** — If *Leveling Method* is set to Min, Max, or Avg, these checkboxes determine which field probe readings are used in the selected leveling algorithm. Probes that are enabled but do not have Include in Leveling set will still be read and recorded. If *Leveling Method* = Probe 1 (or 2,3,4), only one Include In Leveling checkbox can be selected at a time.
- **Leveling Method** — A set of four option buttons that determine how the four probes interact. In the following descriptions, the term "identified probe" refers to a probe that is both selected in its corresponding dropdown, and has its Include in *Leveling checkbox* selected.
    - **Min** — When selected, the software will read all identified probes and level off the lowest value among the probes.
    - **Avg** — When selected, the software will read all identified probes and level using the average of the probe readings. This is defined as the value:

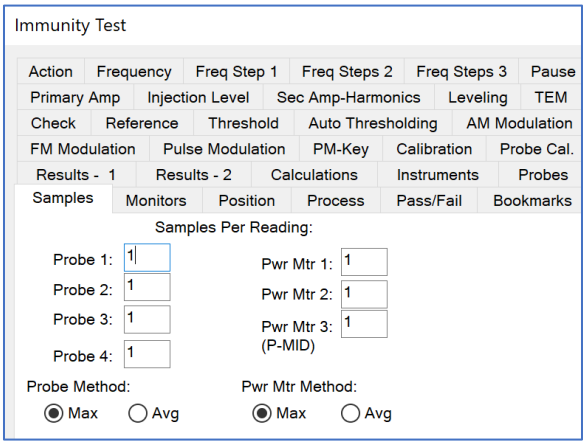
$$1 + \frac{+\text{value4}}{\text{number of probes}}$$

    - **Max** — When selected, the software will read all identified probes and level off the highest value among the probes.
    - **Probe 1/2/3/4** — The default value for probe leveling. With this selection, if probe leveling is being performed, then the system will level on the probe currently selected by the *Include in Leveling* checkboxes. The remaining probes will be read, if selected, and stored, if selected to record their data on the *Results – 2* tab.

Samples Tab

An instrument is read one time to obtain a valid number. There are circumstances where you would want to measure multiple times and take the average or max of these values to determine a final value.

This tab allows you to select the number of samples for each of the different instruments in this step.



- **Samples per Reading** — Defines the number of samples desired per instrument reading.
- **Probe Method /Pwr Mtr Method** — Defines how the value is calculated for readings from each instrument. The user can choose to keep either the maximum or average of multiple samples. If a single sample is taken, that is the only value stored.

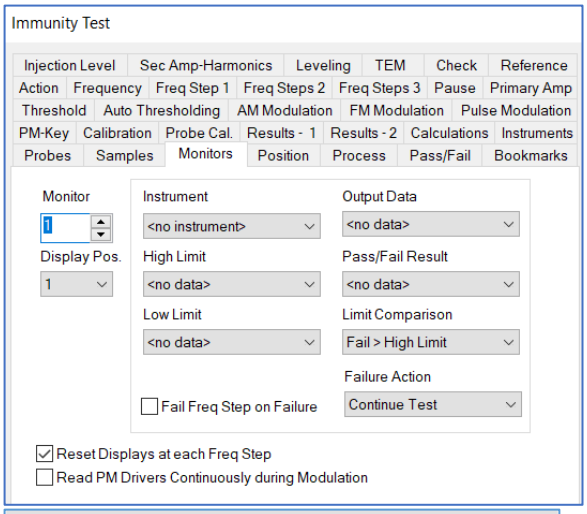
Monitors Tab

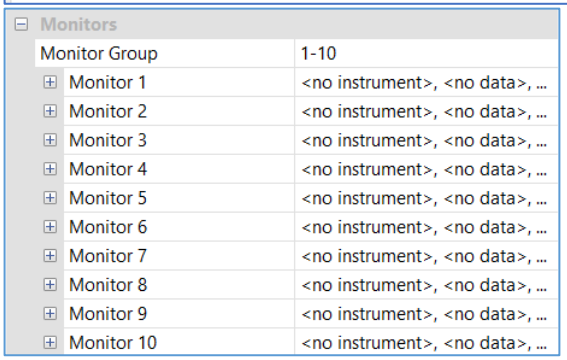
This tab provides the ability to record the results of the test on instruments that are not part of the leveling loop.

Monitoring allows the user to record values from the EUT while the test is running.
Monitor implies that they are looked at, but are not part of the leveling loop.

One possible use of these fields is to record the state of the EUT while doing automatic testing (in a relatively high-speed mode), then using mathematics to determine at what frequencies the EUT was out of band. These frequencies would be the input frequencies for a manual Immunity test with operator intervention.

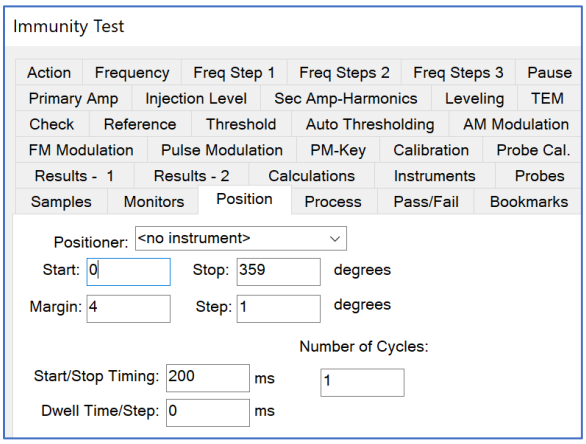
NOTE: The maximum number of monitors that can be configured within a test profile is 100. Use the Immunity actions properties window to manage the monitors in groups of 10 as seen below.





Position Tab

If a positioner is specified on this tab, then after the level is achieved, the positioner will be moved from start to stop position. This will allow all angles of the EUT to be covered in one frequency step.

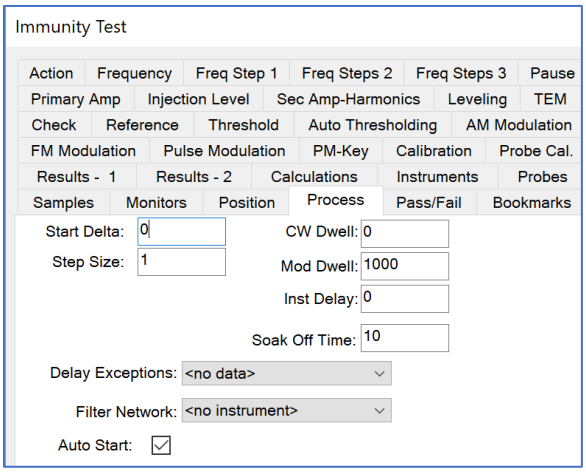


- **Positioner** — Select the instrument that controls position (normally a turntable).
- **Start/Stop** — Enter the start and stop positions desired. For each frequency, the software will look at the nearest position (start/stop) and go to that position, then begin one movement. At frequency 1, the positioner starts at zero and goes to 359 degrees. For frequency 2, since the turntable is already at 359, it will start at 359 and move to zero.
- **Margin** — On certain positioners there is a noticeable delay from when you send a stop command to when motion ceases; this can cause overrun. The margin is the number of degrees prior to the desired position that the program will send the stop command so that it will then stop at the desired location.

- **Step** — If the dwell time per step is greater than zero, then the motion will be non-continuous and the software will step to each position, dwell, step to the next position, and repeat this cycle until the stop position is reached.
- **Start/Stop Timing** — This value is the delay time between sending a start or stop command and any further instructions to the positioner.
- **Number of Cycles** — Determines how many times the software will move from the start to the stop position for each frequency. If the cycle were two, the software would move from the start to the stop position, and then move to the start position again for every frequency.
- **Dwell Time/Step** — If the dwell time per step is greater than zero, the positioning will not be continuous. The software will move using the step size, stop, dwell, move to the next step, and repeat this cycle until the stop position is reached.

Process Tab

This tab allows you to control steps during the Immunity test. The delay feature is particularly interesting. A more complex, but equally powerful tool is the ability to perform step tests. A step test involves stepping up to the final immunity level, pausing at intermediate stages to determine effects. Auto start is only obvious once we have discussed the running of the test.



- **Start Delta** — If you are not performing a stepped test, this box should be set to zero. This will cause an initial offset of zero and the test will begin at the specification limit. If we are performing a stepped test, the value needed here is the offset needed for our first point in specification units.

  If we were performing a 10 V/M test with 1 V/M steps starting at 5 V/M, our start delta would be –5. The system will take the limit and subtract the start delta, yielding the first step level; in this case, 10 V/M – 5 would yield 5 V/M.

  If you are performing an Immunity test from calibrated information, the start delta needs to be stated in the same units as the signal generator since the signal generator units are the amplitude in this defined action. This will probably require some calculations by the user prior to running this test. The units are usually stated in negative numbers and we will want to step up to the specification limit. It is possible to define this action to step down to the specification limit.

  NOTE: If the start delta equals the amplitude level, the system will create a starting point of zero, which is not a valid choice.

- **Step Size** — The step size must be stated in specification limit values. If we are leveling to a power meter, which is stated in dBm, then our specification limit will be in dBm. The step size must match these units.

  Usually when we discuss an Immunity test, we think of a fixed RF level (with or without modulation) at a fixed frequency. However, certain standards call for measuring the effects of RF level changes within a frequency point. The IEC 1000-4-3 sets a pass/fail criterion of 3 V/M at most frequencies and is the only level the standard is concerned with. However, some of the automotive standards discuss plateaus or steps within the frequency point.

If our standard is 10 V/M, the standard might call for starting a 1 V/M, observing the EUT, stepping to 2 V/M, observing the EUT, and continuing in 1 V/M steps until we reach 10 V/M. In defining a stepped test, take the final RF level (the specification limit), and then apply a start delta and step size. These three components will fully define our test requirements.

- **CW Dwell** — After the appropriate level has been achieved, the software will dwell the number of milliseconds defined prior to turning on modulation. If no modulation is defined, and this value is zero, there will be no dwell and the software will automatically go to the next frequency after been achieved.
- **Mod Dwell** — After the appropriate amplitude has been achieved, and after the CW dwell has been completed, the modulation dwell determines how long (in milliseconds) modulation will be left on before returning to the next frequency. If no modulation is enabled, this value is ignored.

  NOTE: This setting is the same as the Std Dwell on the *Instruments tab*, and is therefore informative, not functional.
- **Inst Delay** — This is shown for reference purposes and is the total delays from the different instrument on the *Instruments tab* and *Probe tab*.
- **Soak Off Time** — Used to support automotive testing which requires that the transmitter not be keyed for a certain amount of time (ms) between frequency points.
- **Delay Exceptions** — An exception is a frequency that requires a different dwell time than the standard. This tab includes the specification dwell time, but some standards call for different dwell times at certain frequencies, such as the harmonics of the primary crystal, and so on.

  An exceptions list is a set of frequency points and their required dwell time (in milliseconds). It is critical that the frequency be a point in the Frequency data element, or it will be ignored by this step. One way to ensure that the primary frequencies are covered is to merge the exceptions list and the Frequency data elements together. This will ensure that every point covered by the specification is included.
- **Filter Network** — If a filter network instrument is selected here, at the beginning of each frequency, the filter will be set to the appropriate frequency.
- **Auto Start** — When running an Immunity test with this action you have the option of manually stepping through the frequencies or automatically stepping through them. When first executed, the action displays a dialog box for communication with the technician/engineer. The Auto Start check box determines whether the action will immediately begin automatic execution or whether the user must click the **Start Auto** button in the **dialog box**.

NOTE: How to restart immunity from a point other than the first frequency:

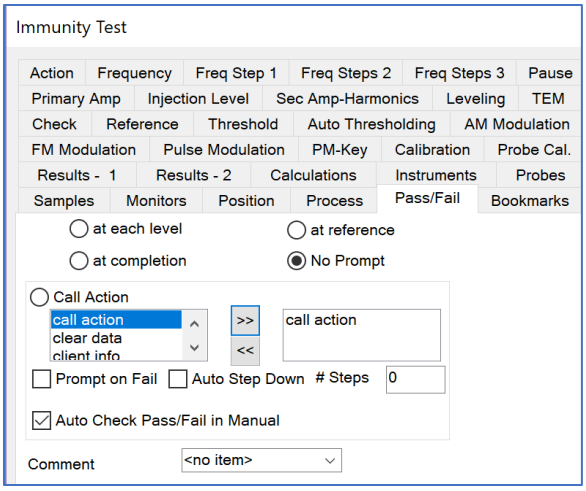
A key point is whether RF is On or Off when you step between frequencies (in Manual mode). There is an option to re-level (or not) at each freq. step when in Manual mode.

To return to previous freq. pt and continue the test:

1. Turn “Auto Start” flag off on Process tab.
2. Run Immunity, RF will start in the Off state.
3. Use the Freq. Next button to step to the desired frequency step.
4. Press Start Auto to restart the test from there.

The flag to re-level at each freq. step is called “Auto Level on Change of Frequency (Manual Mode)”. However, this is only relevant if RF is On while you are stepping between frequencies. If RF is On (whether you re-level or not), the test will re-read the sensors and store the results in the output data elements. If RF is Off, it will not re-read sensors, and thus will not store results.

Pass/Fail Tab



The Immunity action has five settings for pass/fail that are standard in the TILE! system. Four of these presume that the user is manually observing the test. The fifth option allows an interface for automatic failure analysis. Please contact *ETS-Lindgren* for a discussion of these options.

The four option buttons comprise the manual options within the system. When the test is running, you have the option of forcing the user to respond at each step, at the completion of the current frequency point, or at the completion of the complete test. The final (and default) is not to prompt at all.

Since the dialog box present during the test will always allow the user to override the automatic mode, the choice of techniques is left to the user. In some circumstances, especially with complex EUT, it might be necessary to force the user to respond at each step. It is suggested that you try running the test each way to better understand the impact of each option.

- **Call Action** — When selected, this gives the operator the ability to jump to and execute the defined actions. You can select multiple actions to test. Generally, these will be switches, instrument commands, or the Serial Interface action. These icons are set up on the flowchart and configured so that their default behavior is a pass and any deviation is a fail.

  It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

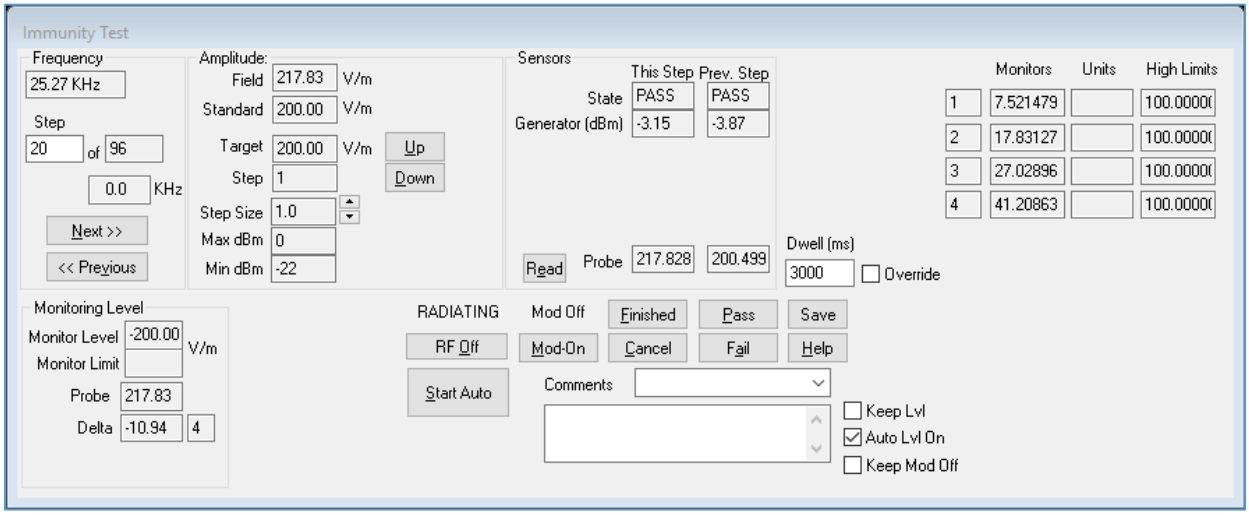
- **Auto Step Down** — If a failure is detected in a Call action, this check box determines whether we will lower the amplitude of the current signal until it turns to a passed condition. This is included here for backwards compatibility; normally, this function should be exercised in auto thresholding.

- **# of Steps** — The number of cycles that the signal generator will be changed and the pass/fail stated reviewed prior to reporting a failure and continuing.

- **Auto Check Pass/Fail in Manual** — This is included here for backwards compatibility; these functions should be used in auto thresholding.

Dialog Box

When you are executing the Immunity test, you will have a dialog box showing a wide variety of information about this test. How this dialog box is used depends on the choice of leveling method, pass/fail, and process settings. The controls for this dialog box are summarized in the control buttons. The remainder of the dialog box is information for the user.



- **Frequency/Step** — Displays current frequency and step number. The step number displayed refers to the total number of frequency steps to be performed. In manual mode, you can use **Next** or **Previous** to move forward or backwards in the frequency table. You cannot enter a new frequency that is not in the current Frequency data element.

  During automatic testing, the user might notice a failure or questionable reading. Stop the test. Move back to the previous frequency. Test this frequency for sensitivity by adjusting the RF level and overwrite the originally pass/fail information with updated information.

- **Monitoring Level** — If using feedback leveling (probe or power meter), then the status of the leveling loop is displayed. The substitution value displays the current RF level for this leveling step. The delta value displays this change in percent. The number in the lower left corner is the count number for the leveling loop. Each time the system changes the RF level in attempting to level, the counter is incremented by one. The value in the **Monitor Limit** box is the maximum dBm depicted on the *Leveling tab*.

- **Amplitude** — Displays the estimated field strength, the current amplitude, the target level, the current step, and the related step size. If you are using steps within the frequency as part of the leveling process (defined on the *Process tab*), the step number displays the current step. Step size, for reference, displays the defined amount by which each step will change. The target level displays the defined specification level. The current level is displayed in the first box and is in the units defined for the Immunity test. If you are using no leveling, then this will be the signal generator level. If you are using probe, then this will be in V/M. If you are using power meter, then this will be in dBm.

- **Sensors** — The sensor section displays the read (or sent) data from the instruments in the test. There are two columns, **This Step** (current) and **Prev. Step** (previous). The current column displays the information from the last read/write to the instruments. The signal generator displays the last level sent to the generator. The **PwrMtr** and **Probe** boxes display the last read levels from these instruments. A value is displayed only if the instrument is identified in the instrument section of the Immunity Test action.

- **Finished** — At any time you can click Finished to save the test to data and exit the action. Finish aborts the operation but saves the information.

- **Cancel** — At any time you can click Cancel to abort the operation and exit the action. No data is saved.

- **Pass/Fail** — When in manual or automatic mode, clicking Pass or Fail will cause the appropriate pass/fail information to be saved and then the program will step to the next frequency point.
- **Help** — This button is not active.
- **Control Buttons** — The control buttons that appear during the test provide direct control of the leveling process and recording capability for pass/fail information. The beginning state for this dialog box depends upon the settings in the process section of the Immunity test. If **Start Auto** was selected the action will start in automatic mode. The action will start with the first frequency point and start stepping through the test. If the Start Auto was not selected the action will start in the manual mode.
    - **RF On (Manual Operation)** — When in manual mode, turning RF on causes the system to level to the first frequency point. Above the RF On button will appear the text RADIATING. When you have the RF on, you can adjust the level using the amplitude section and read the sensors. If you click the **Pass** or **Fail** button the system will record the appropriate value in the data element defined for pass/fail. You can move up or down in frequency across the defined frequency points. You can vary the amplitude and record.
    - **Start Auto/Halt (Automatic Operation)** — When the Start Auto check box is selected the system will begin an automated sequencing of the frequency points. At each frequency, the system will level to the specified RF level. The system will pause if you have selected either Stop at each step or Stop at each frequency and request operator input. The operator will indicate whether the unit has pass or failed and the system will record the appropriate value and continue to the next frequency. If you selected Stop at completion of all frequencies, the system will mark a pass and continue.

      When the **Start Auto** button is clicked, the Halt button replaces it. At any time during the automatic sequencing, you can halt the test, go to manual mode, and make manual adjustments. After completing the manual steps, click Start Auto to resume the automatic sequence from the current frequency point.
    - **Mod-On/Off** — Toggles the AM or FM modulation if they are enabled on their respective pages.
    - **Test** — This button only works if Call actions have been defined on the *Process tab*. If defined, the Test button will force a read of the Call action. The appropriate information will be updated in the status field.

Auto Thresholding

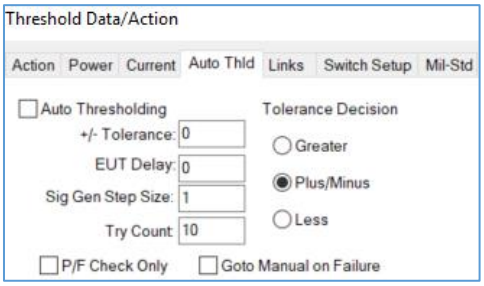
The Auto Thresholding action is used to provide auto-threshold detection and measurement actions during immunity/susceptibility testing for the *Immunity Test* and *Reverb Actions*. It is used to monitor an instrument to evaluate if it exceeds a limit. If it exceeds a limit, it will interact with the action to reduce the signal generator level until the instrument selected indicates a reading that does not exceed the limit.

The Auto-Threshold action monitors the Equipment Under Test (EUT) while the *Immunity Test* action is operating. Should the EUT exhibit undesired performance, the Auto-Threshold action will reduce the interference-condition severity-level until the EUT's operation returns to normal.

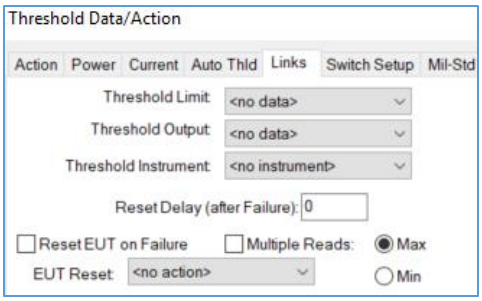
The Auto-Threshold action requires an instrument to monitor the EUT operation condition. The instrument needs to be one of the equipment listed within TILE!'s instrument drivers. The EUT operation should include the allowable tolerances in order to determine if the EUT operation limit meets it susceptibility requirements. The EUT's operational limit and its authorized tolerance are typically defined as the EUT's acceptance criteria, and it is normally found within the EUT's product specifications.

Example of Auto-Threshold Settings and Operation

- Create the data elements EUT_Limit and EUT_Measurement1:
  - Enter the appropriate information within the EUT_Limit as necessary.
  - EUT_Measurement will create any math calculations, if necessary to generate the corrected EUT_level element.
  - Determine the EUT's operation allowable tolerance (acceptance criteria).
- Create the instrument EUT_Monitor:
  - Load the appropriate instrument driver and address.
- Create an Auto Thresholding action:
  - On the action tab, name the action "Auto_T".
  - Go to the Auto-Thld tab, and enter the desired characteristics:
    - Select the Auto Thresholding box
    - Set the +/- Tolerance to 5
    - Set the EUT Delay to 0
    - Set the Sig Gen Step Size to 1
    - Set the Try Count to 50
    - Set the Tolerance Decision to Plus/Minus



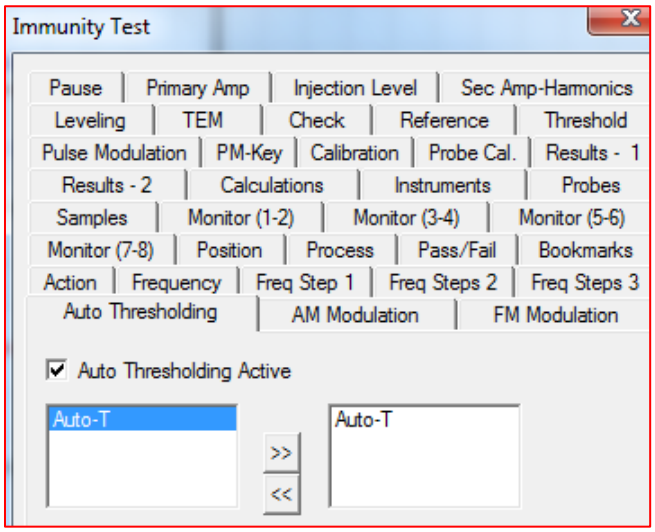
  - Go to the Links tab, and enter the desired characteristics:
    - Set the Threshold-limit to EUT_Limit
    - Set the Threshold output to Auto_T
    - Set the Threshold-Instrument to EUT Monitor



  - If desired, go to the Mil-Std tab to turn on the MIL-STD offset and set the reduction step (default is 6 dB).
    - The Mil-Std tab determines in which data elements the results of the 16-point field calculation will be stored.
    - For example: If selected, the Auto-Threshold action will decrease the interference condition severity level by 6 dB, then slowly increase the interference condition severity level until the EUT operation deviates from acceptable parameters.

The Auto-Threshold action will record this EUT undesired state as a susceptibility response level and increment to the next frequency.

- Create an Immunity/Susceptibility (I/S) action.
    - On the action tab, name the action “TS1a”.
    - On the Auto Thresholding tab, select the Auto-Thresholding box and move Auto-T to the right column to activate the auto thresholding.



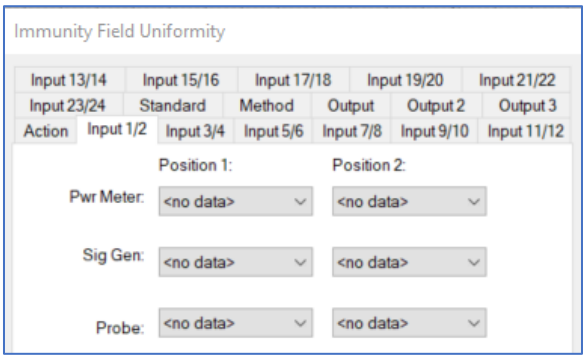
Immunity Field Uniformity

The Immunity Field Uniformity action analyzes from 4 point to 16 point field data to determine if the correct number of points meet the 0 6 dB criteria for acceptance per IEC 61000 4 3 (and other standards with similar requirements). If the criteria are met for field uniformity, the output is the appropriate values to establish a defined field as outlined in the specifications.

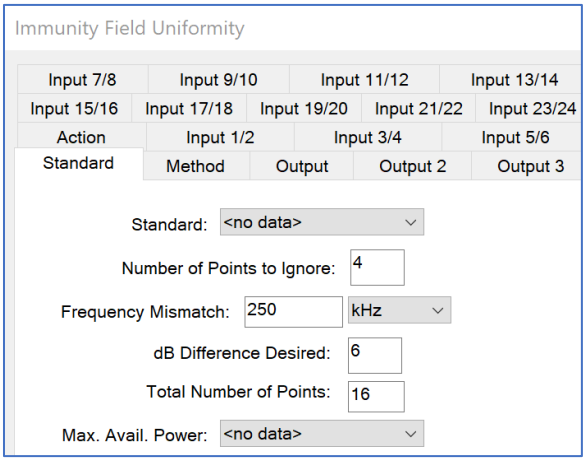
Input (1/2 - 23/24) Tabs

There are twelve input tabs, with information on two different probe positions on each tab. The power meter, signal generator, and probe data can be specified.

At minimum, you must have the signal generator and probe levels for Section 6.2.2 and the power meter and probe levels for Section 6.2.1. If you do not specify the correct input values, the output will be invalid or wrong.



Standard Tab



- **Standard** — If you are performing calculations for Section 6.2.2, it is required as part of the calculation to use the expected field strength. The standard data element should represent that expected value. Section 6.2.2 states that if any of the 12 points (that are in the 0 6 dB band) are less than the desired level, the signal generator level or power level is adjusted by calculation to the level needed to achieve the desired field.

  There is no limitation in the mathematics on adjusting the level to a value higher than the capability of the system to generate efficiently. There is a problem with the modulated signal since the general standard uses an 80% modulation. The actual field envelope into the amplifier can be as much as 6 dB higher than the signal generator drive-level. The user is cautioned to review these calculations closely to ensure the amplitude calculated by the software does not exceed the capabilities of the hardware involved.

  The *Output 3 tab* has a Matching Sig. Gen. -5 dB file output that will compensate for a field level that is calibrated with 80% modulation level to a standard test level. An example would be of a 10 V/m EUT test calibration should be accomplished with a CW level of 18 V/m (10V @80% mod) to show that the amplifier and antenna can meet the level needed for the modulated value. This would output a calculated signal generator level that would be down 5.1 dB so that the 10 V/m EUT test could use this level as a reference.

- **Number of Points to Ignore** — The user can choose how many positions within the 4 16 positions of test data are to be ignored, but if the number chosen exceeds 25%, it will be ignored and 25% will be used. The purpose of this is to allow the user to be more critical in his discrimination, not less.

- **Frequency Mismatch** — Generally speaking, when doing a point-to-point comparison between different positions, the frequency at each point is checked to ensure a matching frequency. Depending upon how the points were generated, you could have fractional errors in the frequency range.

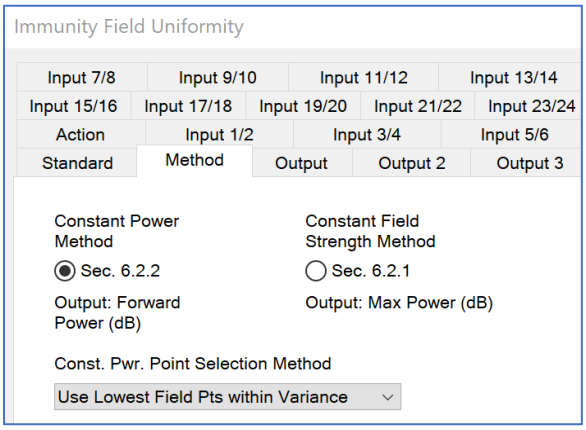
  For example, a 1 2 Hz mismatch in the absolute frequency. The mismatch frequency allows you to specify a bandwidth that all the data needs to fall within to be accepted. If the mismatch was 1 kHz and position 1 had a frequency of 56.123 MHz, and position 2 had 56.1225 MHz, then the software would treat these as the same point.

- **dB Difference Desired** — The 61000-4-3 standard defines a 6-dB tolerance. Within TILE!, we allow you to change this value for non-standard purposes.

- **Total Number of Points** — The total number of probe positions from 4 to 24

- **Max. Avail. Power** — If you have a file with the 1 dB compression point data for the amplifier, this will allow a calculation of how much field strength can be achieved at each frequency. A generic amplifier factor can be used as well.

Method Tab



- **Constant Power Method** — This method involves physically leveling to the desired field at one position and replaying the signal generator drive-level at the remaining positions, measuring the fluctuation in the probe and power meter levels under these circumstances. The choice of which position is the first position does not generally affect the accuracy of the calculation, but it does affect the appearance of the data.

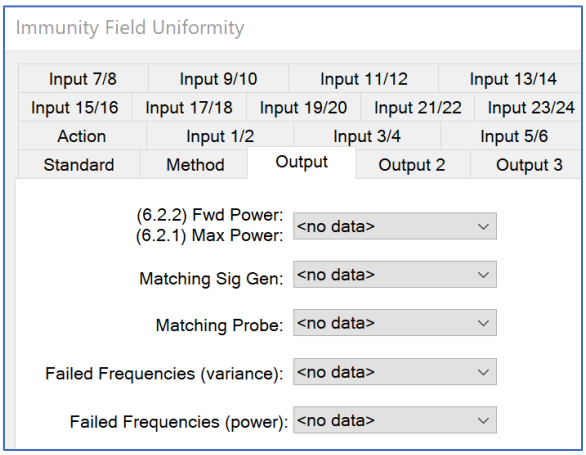
  For instance, in a 16-point field array, if position 1 is an outer corner of the array, you are referencing all the points to the worst position, and not the most-stable position. The field levels and probe levels will show a wide variation from the desired field, but mathematically might pass the 6 dB criteria. If the user were to establish position 1 as one of the center points of the array, then the data will appear to be much more stable. It is an issue of appearance, not of calibration.

- **Constant Field Strength Method** — In this method the field at each position is leveled. In effect, the probe values for all the points should be within the tolerance of the leveling loop, and thus relatively flat. Then, it is a simple matter to calculate the 0 6 dB by direct comparison of the power or signal generator levels.

- **Const. Pwr. Point Selection Method** — These determine how the level will be calculated if more than 75% of points meet the 6 dB criteria. There are three selections: Use Lowest Field Pts within Variance (default), Use Highest Field Pts within Variance, and Use Minimum Variance of Field Pts.

Output Tab

The Output tab determines in which data elements the results of the 16-point field calculation will be stored.



- **(6.2.2) Fwd Power/ (6.2.1) Max Power** — This can be either signal generator level or power meter level, depending upon which data was stored during the 16 probe positions.

- **Matching Sig Gen** — If you measured power level using a power meter and used these values on each of the input tabs, this action will calculate the forward power required to achieve the field strength.

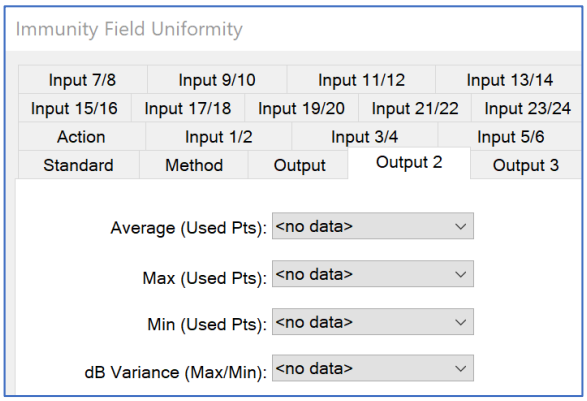
  If the signal generator values were also input, the signal generator level for the lowest position of each frequency will be stored in this data element. This output data can then be used as a best guess starting amplitude. This can substantially shorten the time required to level to the power during the test.

- **Matching Probe** — This output data element reflects the lowest probe reading included in the three 12 points used to calculate the drive level. This data element can then be used for a reference value during the actual test. See *Immunity Test Reference Tab* for further information.
- **Failed Frequencies (variance)** — This data element will contain only those frequencies that failed the field uniformity variance calculation. These frequencies have more than 75% points that are greater than 6 dB.
- **Failed Frequencies (power)** — This data element will contain only those frequencies that, once the uniformity calculation is performed, would exceed the available power from the amplifier. The Max Avail Pwr data element must have the gain of the amp or its 1 dB compression point.

Output 2 Tab

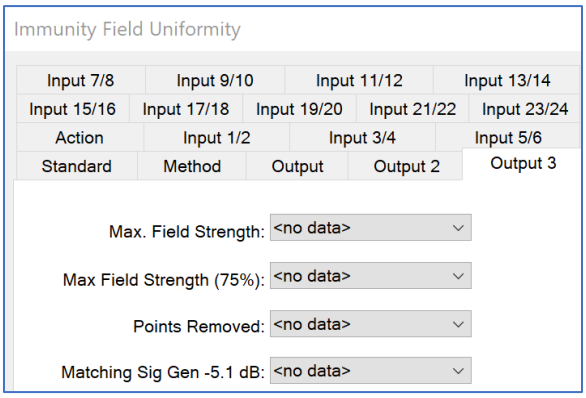
This tabs data elements are for statistical purposes only, and are not part of the required output of the field uniformity measurements.

- **Average/Max/Min (Used Pts)** — This data element will contain the average value of the points that met the 0 6 dB criteria.
- **dB Variance (Max/Min)** — This data element will give you the dB variance from zero to six for each of the positions in the measurement.



Output 3 Tab

This tab assists in determining how much power the system can deliver if there are ≥ 75% points meeting the 6 dB criteria.



- **Max. Field Strength** — Calculates the maximum field strength of all used points.
- **Max. Field Strength (75%)** — Calculates the maximum available power if only 75% of the points were used.

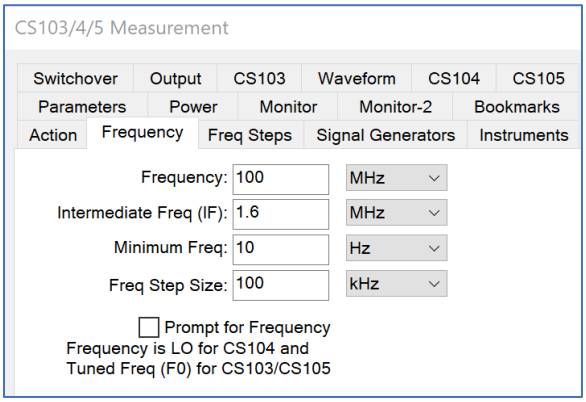
  For instance, if 16 of 16 points meet the 6 dB criteria, the 6.2.2 calculation will use the lowest points as the base point for calculating the driver level needed to meet the field uniformity; but if 25% of the lowest points are ignored, then the overall gain of the amp for the drive level would be reduced. This is valuable in determining if the system can attain higher levels than what you may have calibrated towards.
- **Points Removed** — Creates a data element that shows how many points are ignored after achieving the 6 dB criteria.
- **Matching Sig Gen -5.1 dB** — Creates a data element containing a reference file that is 5.1 dB down from the calculated level required to meet the drive level for the field uniformity.

CS103/4/5 Measurement

The CS103/4/5 Measurement action is used to perform intermodulation and cross modulation testing per MIL STD 461.

CS103 now has the ability to playback previously run calibrations for both SG1 and SG2 (calibrations assumed to have been acquired using Immunity Calibration action). Backwards compatibility of the action is maintained by enabling the playback feature with setting options described below. EUT Passband frequencies can be skipped for both SG1 and SG2. This feature is available whether the playback feature is in use, or not.

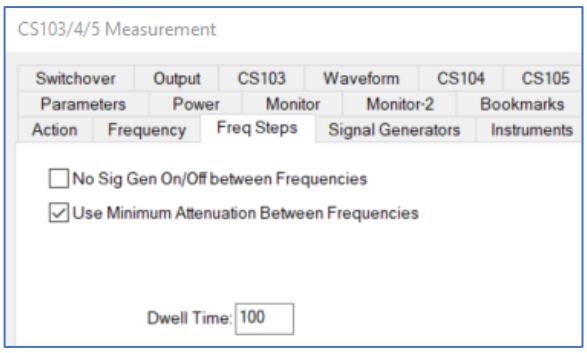
Frequency Tab



- **Frequency** — This is the tuned frequency for CS103/105 and the LO frequency for CS104.
- **Intermediate Freq (IF)** — Specify the IF of the receiver under test.
- **Minimum Freq** — The CS103/104/105 measurements step through a range of frequencies. The minimum frequency is the lowest frequency at which any measurement is done.
- **Freq Step Size** — The step size for frequencies from the lowest to highest frequency of interest.
- **Prompt for Frequency** — Select this check box to prompt the operator at the start of the test to enter the frequency of interest.

Freq Steps Tab

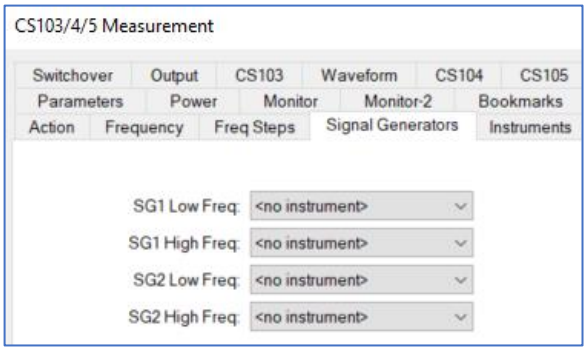
This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Freq Step 1 Tab* for the full description.



- **Dwell Time** — Enter a dwell time. This dwell is used between each frequency.

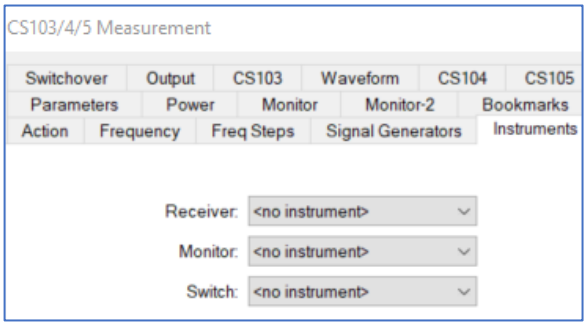
Signal Generators Tab

Select the signal generators to be used for the test. If you are performing a test that crosses over the range of two signal generators, you can enter two signal generators. The frequency transition is defined on the *Switchover tab*.



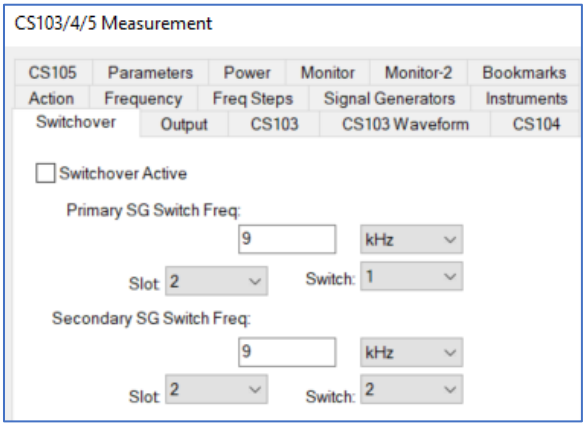
Instruments Tab

Select the appropriate instrument for the measurement receiver (this is mandatory). Select a monitor and/or switch if appropriate for this test.



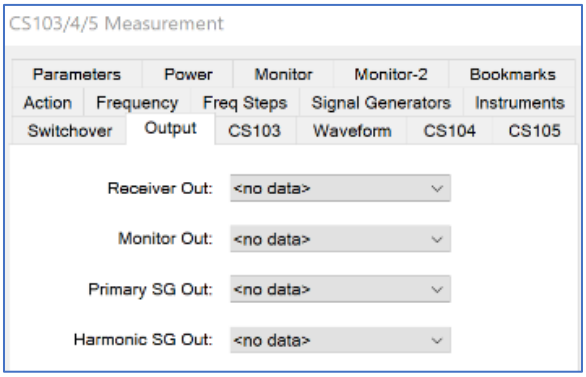
Switchover Tab

These settings control the crossover frequency if an automatic switch is used to control two separate signal generators.



- **Switchover Active** — Allows manual switchover without the switch instrument. If Switchover Active is checked, but no Switch instrument is selected, the action will use Manual Switchover mode. Manual switchover will display dialogs when SG1 and SG2 are crossing over their respective switch frequencies.
- **Primary/Secondary SG Switch Freq** — Enter the frequency where signal generator 1/ signal generator 2 low frequency will cross over to signal generator 1/ signal generator 2 high frequency (see the *Signal Generators tab*).
- **Slot/Switch** — Select an appropriate slot and switch position if a switch instrument is used to change the input outputs to the signal generator.

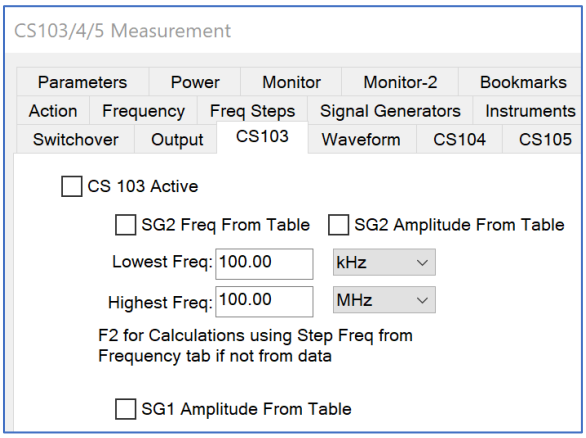
Output Tab



This tab selects the appropriate data element for storing results during this test.

- **Receiver Out** — Select an appropriate data element to store the output level of the receiver at the completion of each frequency.
- **Monitor Out** — Select an appropriate data element to store the output level of the monitor at the completion of each frequency.
- **Primary SG Out** — Select an appropriate data element to store the output level of the primary signal generator at the completion of each frequency.
- **Harmonic SG Out** — Select an appropriate data element to store the output level of the harmonic signal generator at the completion of each frequency.

CS103 Tab



Only one type of CS can be performed at a time. Activation on this tab will disable the **CS104** and **CS105** tabs.

- **CS103 Active** — If selected, CS103 test will be performed using the information from the *Frequency tab* and the general equations as shown on this tab.
- **SG2 Freq From Table** — If this check box is selected, it enables frequency playback on SG2 using the frequency points obtained from a file to be specified by the user.

  NOTE: The test is “driven” off this frequency list only. The F1 frequencies cannot be set independently of the F2 frequencies. CS103 defines the relationship between F1 and F2 based on the intermodulation product equation F0 = m*F1 ± n*F2. This equation is specified (with m, n, and the sign variable as settings) on the Waveform tab. The CS_Peaks action always calculates F1 using this equation, based on the frequency list specified for F2.
- **SG2 Amplitude From Table** — If SG2 Freq From Table is selected, this checkbox is exposed as an additional option. These settings together fully enable the playback feature for SG2, for both frequency and level. If SG2 Freq From Table is enabled, but SG2 Amplitude From Table is not enabled, then the SG2 output is set using the Signal Above Receiver and Receive Sensitivity values on the *Power tab*.
- **Lowest Freq/Highest Freq** — The low/high frequency control the range over which this test is performed using the step frequency from the *Frequency tab*.
- **SG1 Amplitude From Table** — Enables the playback feature for SG1.

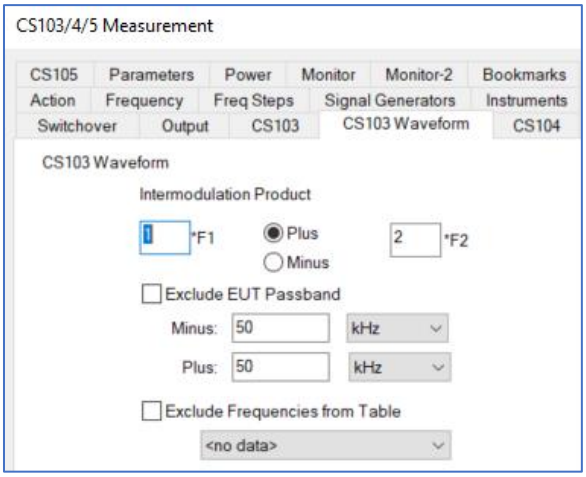
  The F2 frequency is set based on whatever mechanism was used to specify F2 (Lowest/Highest Freq with Step, or From Table). F2 is used to calculate F1 based on the intermodulation product equation settings. The F1 frequency is used to lookup the amplitude from the SG1 Data data-

element. If a data point with an F1 value exists, the amplitude of that data point is used for the SG1 output level.

If a data point with F1 does not exist: If SG1 Data has a Continuous option checked (either Linear or Log), CS_Peaks will use F1 to interpolate the level; If SG1 Data does not have Continuous selected, an Amplitude Data is Missing error will be generated and the action will not run. To ensure that SG1 Data was calibrated using the specific SG1 frequency used in the test, make sure that SG1's Continuous checkboxes are cleared to enable this error checking mechanism.

Waveform Tab

This tab establishes the intermodulation equation to be used in the test.



- **Exclude EUT Passband** — When the Exclude EUT Passband checkbox is enabled, the EUT Passband Bandwidth frequency is used to check all F1 and F2 frequencies in the test, and skip any test points where F1 falls within F1 +- bandwidth/2, or F2 falls within F2 +- bandwidth/2.

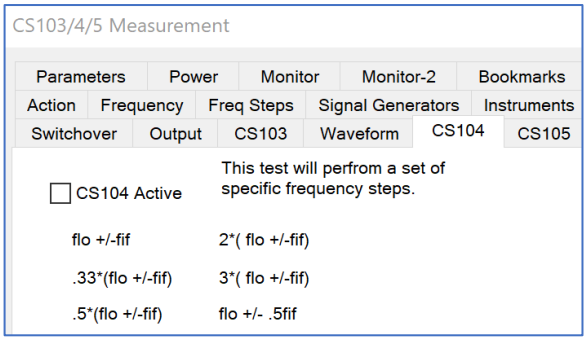
  This feature allows calibrations to be run independently of the corresponding test, and alleviates the need for external calculations to remove these test frequencies beforehand. The actual test frequencies used will be reflected in all the CS_Peaks results data elements (Receiver Out, Primary SG Out, etc.).

- **Exclude Frequencies from Table** — Frequency pairs represent excluded bands that will not be generated by either SG1 or SG2.

  If this data element is the same as the data element in CS_Peaks, they act as band exclusion ranges. If either F1 or F2 would have fallen in any excluded range, the test point is skipped. There is no limit to the number of excluded ranges that can be generated this way (just keep adding more Bands to the data element).

CS104 Tab

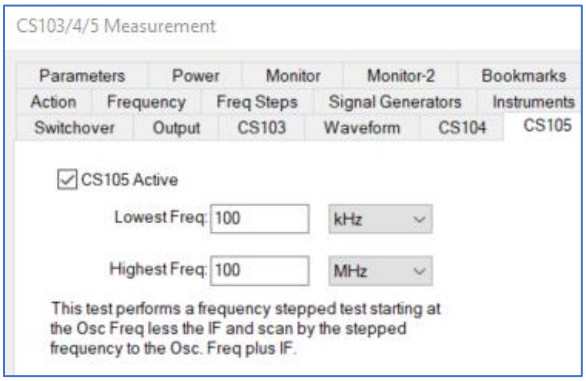
Only one type of CS can be performed at a time. Activation on this tab will disable the **CS103** and **CS105** tabs.



- **CS104 Active** — If selected, the CS104 test will be performed using the information from the *Frequency tab* and the general equations as shown on this tab.

CS105 Tab

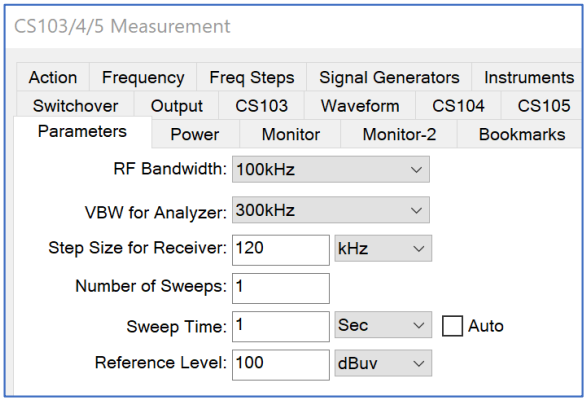
Only one type of CS can be performed at a time. Activation on this tab will disable the **CS103** and **CS104** tabs.



- **CS105 Active** — If selected, the CS105 test will be performed using the information from the *Frequency tab* and the general equations as shown on this page.
- **Lowest Freq /Highest Freq** — This test performs a frequency stepped test starting at the Osc Freq less the IF and scan by the stepped frequency to the Osc Freq plus IF. The start frequency and stop frequency are limited by the low and high frequency shown on this tab.

Parameters Tab

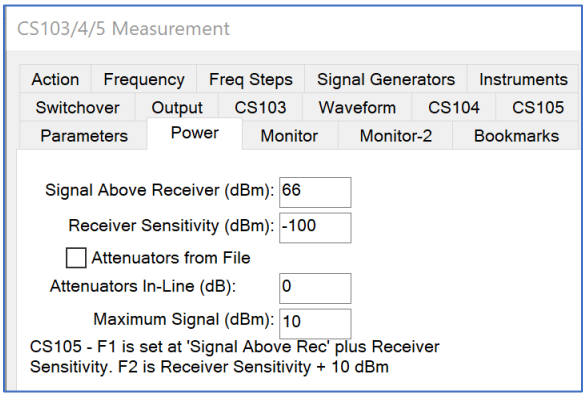
This tab is similar to other parameter tabs and will not fully be described here. See the *Measure Range Parameters Tab* for the full description.



- **Reference Level** — This value will set the reference level of the receiver/analyzer at each frequency step.

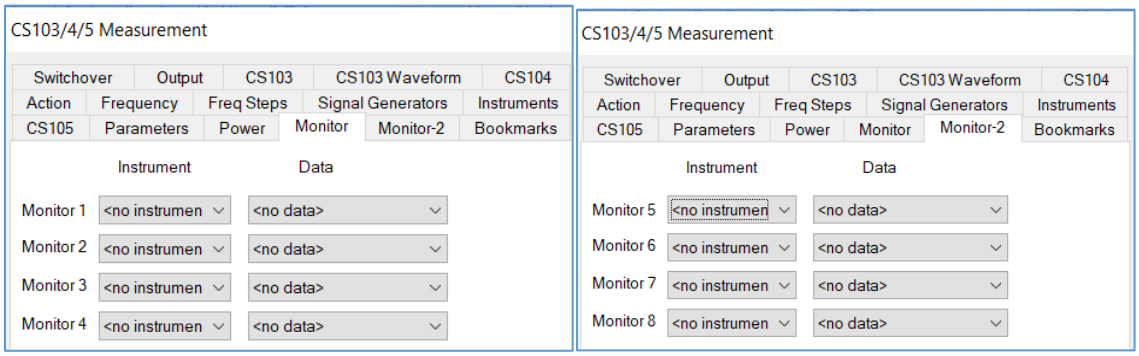
Power Tab

This tab controls the expected values of the EUT.



- **Signal Above Receiver (dBm)** — The margin above the receiver sensitivity for reference level purposes.
- **Receiver Sensitivity (dBm)** — The expected minimum sensitivity of the EUT.
- **Attenuators In-Line (dB)** — Enter the value of any external attenuators inline between the EUT and the measurement receiver.
- **Maximum Signal (dBm)** — The maximum signal to be injected into the EUT.

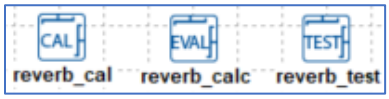
Monitor/Monitor 2 Tabs



These tabs allows you to control instruments that are being monitored during the test.

- **Monitor 1-8** — Enter an appropriate instrument for each monitor.
- **Data** — Enter an appropriate data element to store the output of the instrument at each frequency of the measurement cycle.

Reverb Actions



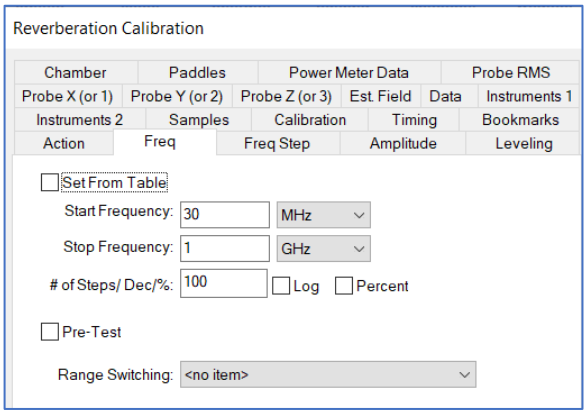
Reverberation Calibration (CAL)

The calibration routine takes the multiple paddle positions and stores the minimum/maximum/average data for the various probes and power meters. This data is then used with data taken from multiple positions in the chamber to generate estimated field levels for given power levels.

Freq Tab

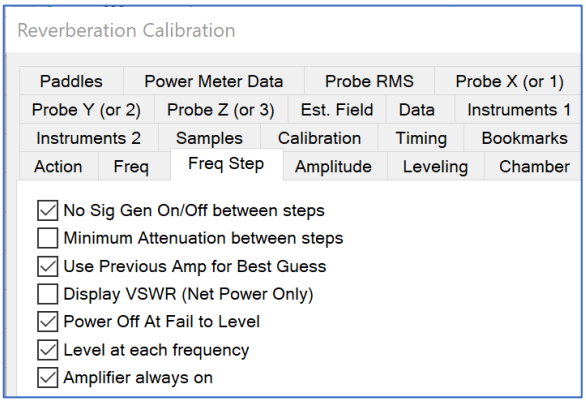
This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Frequency Tab* for the full description.

- **Pre Test** — If this check box is selected, a full calibration is not performed; only the injected signal and receive power are measured. This is normally done as part of a pretest (loaded test) in a reverberation standard.



Freq Step Tab

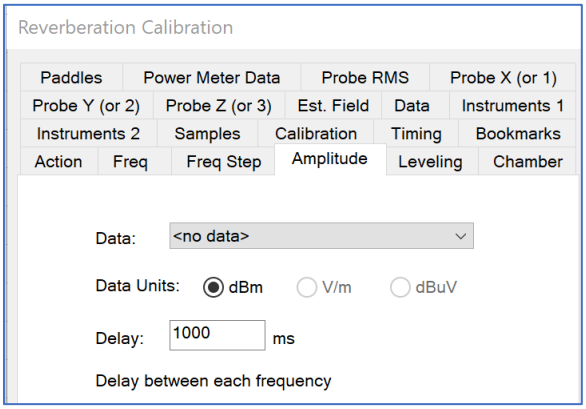
This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Freq Step 1 Tab* for the full description.



- **Level at each frequency** — If this check box is not selected, the software will level to the power at the first paddle position and then step to the same power level at each subsequent paddle position. If this check box is selected, leveling will be performed at every frequency.

Amplitude Tab

This tab is used to select the amplitude data for this test.



NOTE: If the Data is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

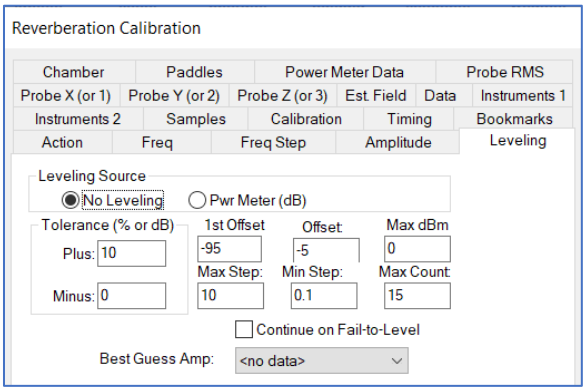
- **Data** — Allows the operator to select the desired amplitude data to be used in the test. Select the appropriate element.

  The frequency component of the Amplitude data element must cover the same range as the data element defined on the *Frequency tab*, but they do not have to be the same element. In most tests, the same data element will be defined for both frequency and amplitude.

- **Data Units** — This selection will set the display of the dialog box to reflect the correct units.
- **Delay** — Represents a specified delay between each frequency step. It is primarily of value when testing routines that do no leveling. This will prevent sending frequency/amplitude data to the signal generator faster than it can react.

Leveling Tab

This tab is similar to other leveling tabs and will not fully be described here. See the *Immunity Test Leveling Tab* for the full description.

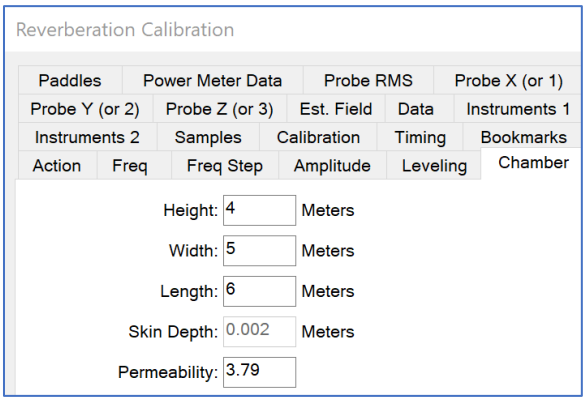


- **First Offset** — This is a safety option. When you first turn on RF power, the signal generator output is set to a reduced level; this avoids potential damage due to overdrive. The initial signal generator output level will start the operator defined x dB value and then step up to read the desired field.
- **Continue on Fail-to-Level** — Allows the user to continue with the leveling steps even when a step fails to level.
- **Best Guess Amp** — Sets the signal generator to a predetermined level to speed up the leveling process. This is used when approximate signal generator levels are known. The data set must be a defined CSV data element.

Chamber Tab

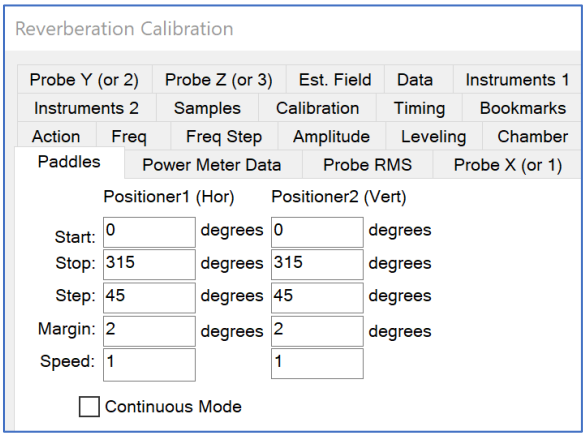
This tab specifies the general dimensions of the chamber and wall materials. This information is used to determine the expected values during the execution.

The default skin depth and permeability are based upon standard shielded room wall materials; see the manufacturer for more specific information. A typical shielded room made of galvanized steel would have a permeability (μ) of 3.79.



- **Height/Width/Length** — Enter the chamber measurements in meters.
- **Skin Depth** — This is derived by equation using the permeability information. This value is disabled.
- **Permeability** — Permeability is the degree of magnetization of a material that responds linearly to an applied magnetic field. Magnetic permeability is represented by the Greek letter μ. In SI units, permeability is measured in Henries per meter or Newtons per Ampere squared. This information can be obtained from the shielded room manufacturer.

Paddles Tab



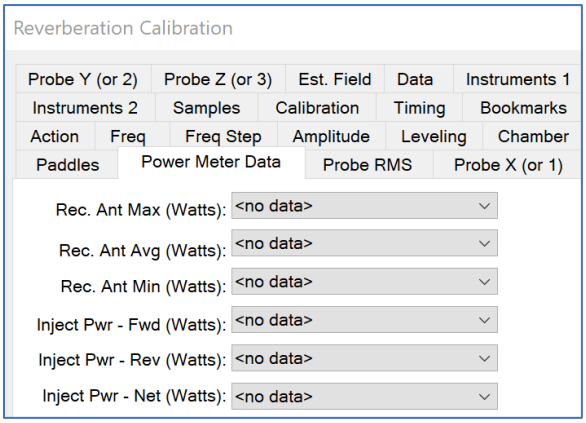
This tab is used to define the number of tuner positions. Although it is setup for two positioners (two tuners), only one is required. If only one tuner is present on the *Data tab*, then set the Positioner 2 values for start/stop to zero.

- **Start/Stop/Step** — Enter an appropriate start and stop positions for the tuners. Typically, the start will be zero and the stop will be a value that equally divides the circumference by the step size.

  Since we are dealing with a zero, it is not necessary to overlap (0 and 360 are the same value). For example, if we want to break the tuner into eight positions, the step size would be 45. The start would be zero and the stop would be 315 (the eighth position starting from zero with a 45-degree step size.

- **Margin** — The stop accuracy of the positioner. Generally, this should be set to zero or one (depending upon the type of positioner). As the positioner is in motion, the software is constantly checking the current position. If it approaches the stop position within the margin the stop command is sent to the positioner.

Power Meter Data Tab



This tab is used to specify the output data elements for the various power meters being read as part of the reverberation calibration.

The injected power value is measured at the input into the chamber (output of the amplifier). The receiving antenna value is the measurement taken inside the chamber from the receiving antenna.

Although these values are taken in dBm (and all correction factors assume a dB relationship), the stored values are converted to watts since all reverberation formulas use linear power.

NOTE: If the Rec. Ant Max, Avg, and Min data are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Rec. Ant Max (Watts)** — Select a data element to store the maximum received antenna value. This is the maximum found for all tuner positions and is used during the evaluation stage.

- **Rec. Ant Avg (Watts)** — Select a data element to store the average received antenna value. This is not used during the evaluation stage, but is provided for statistical purposes.

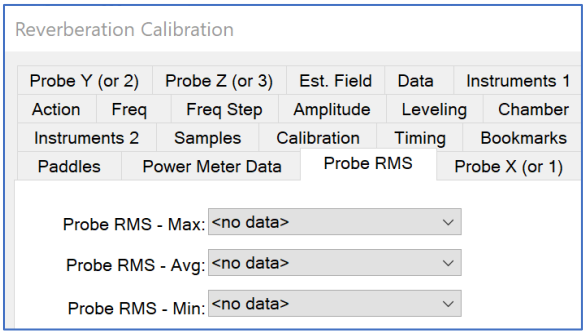
- **Rec. Ant Min (Watts)** — Select a data element to store the minimum received antenna value. This is the maximum found for all tuner positions and is used during the evaluation stage.

- **Inject Pwr – Fwd (Watts)** — Select a data element to store the maximum injected forward power value. This is the maximum found for all tuner positions and is used during the evaluation stage.

- **Inject Pwr – Rev (Watts)** — Select a data element to store the maximum injected reverse power value. This is the maximum found for all tuner positions and is not used during the evaluation stage. The information is provided here for reference purposes.
- **Inject Pwr – Net (Watts)** — Select a data element to store the maximum injected net power value (calculated from the ratio of the forward and reverse powers). This is the maximum found for all tuner positions and is not used during the evaluation stage. The information is provided here for reference purposes.

Probe RMS Tab

The Probe RMS tab is used to select the data elements to store the RMS outputs of the probe during the various positions. The RMS values are used when doing the DO 160 method and are only used for reference in the IEC method.

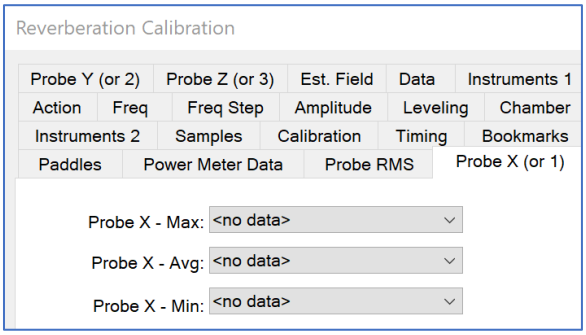


NOTE: The probe instrument drivers setting must be set to Default for proper operation.

- **Probe RMS-Max** — Select a data element to store the maximum probe RMS value. This is the maximum found for all tuner positions and is used during the DO 160 evaluation stage.
- **Probe RMS-Avg/ Probe RMS-Min** — Select a data element to store the average/minimum probe RMS value. This is provided for reference purposes only.

Probe X, Y, Z (or 1, 2, 3) Tabs

The Probe X, Y, Z (or 1, 2, 3) tabs are used to select the data elements to store the axis outputs of the probe during the various positions. The three axis tabs operate identically but the **Probe X (or 1) tab** is described here.

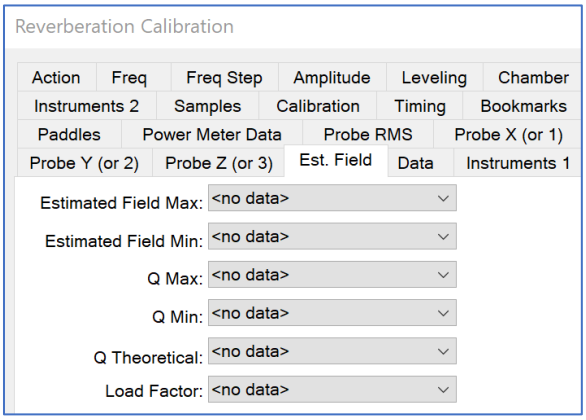


NOTE: The probe instrument drivers setting must be set to Default for proper operation.

- **Probe X-Max** — Select a data element to store the maximum X channel values present during the various paddle positions. These values are used during the evaluation stage.
- **Probe X-Avg** — Select a data element to store the average X channel values present during the various paddle positions. These values are provided for reference purposes.
- **Probe X-Min** — Select a data element to store the minimum X channel values present during the various paddle positions. These values are provided for reference purposes.

Est. Field Tab

The Est. Field tab controls output of some calculated, theoretical or measured values.



- **Estimated Field Max** — Stores the maximum calculated field value found for each of the tuner positions.

  $$E_{max} = \sqrt{\frac{377*8*\pi*(P_{rcvmax})}{\lambda^2}}$$

  This equation uses the receive power to estimate the max field that can be generated at the calibration power level. (Reference IEC 61000 4 21 2011.)

- **Estimated Field Min** — Stores the minimum calculated field value found for each of the tuner positions using same equation as Estimated Field Max.

- **Q Max** — Stores the maximum calculated Q max (or Quality Factor) value found for each of the tuner positions. Q prime was a requirement of the older SAE standard; Q is calculated as follows (Reference IEC 61000 4 21 2011):

  $$Q = \frac{16\pi^2 V}{\eta_{Tx}\,\eta_{Rx}\lambda^3}\left\langle\frac{P_{AveRec}}{P_{input}}\right\rangle_{\eta}$$

  Where:

  V – The chamber volume (m3).

  λ – The wavelength (m).

  $(<P_{AveRec}/P_{input}>)$ – The ratio of the received power to the input power over one complete tuner/stirrer sequence.

  η_{Tx} and η_{Rx} – The antenna efficiency factors for the transmit (Tx) and receive (Rx) antennas, respectively. If a manufacturer's data is not available, then the efficiency can be assumed to be 0.75 for log periodic antennas and 0.9 for horn antennas.

  $\langle\ \rangle_{\eta}$ – Denotes averaging with respect to the number of antenna locations and orientations_ η.

  η – The number of antenna locations and orientations that the Q is evaluated for. Only one location is required as a minimum; however, multiple locations and orientations may be evaluated and the data averaged over them.

- **Q Min** — Stores the minimum calculated Q value found for each of the tuner positions. Q prime was a requirement of the older SAE standard; these values are for reference only.

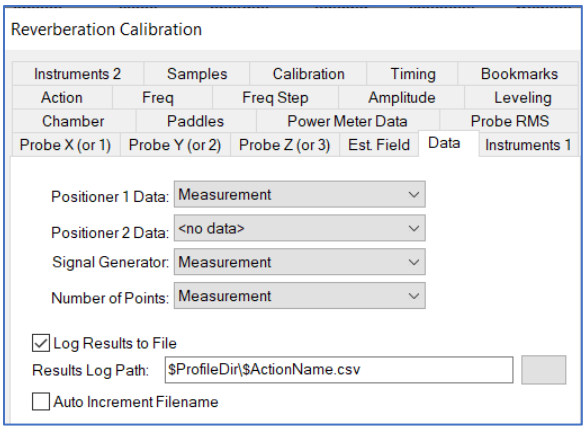
- **Q-Theoretical** — Theoretical Q value can be used as a reference for the chamber. Though this is only useful as a reference, it is calculated each time this action is executed.

- **Load Factor** — Stores the maximum load factor for the multiple paddle positions. Load factor is defined as:

$$10 * \mathrm{Log}_{10}\left(\frac{P_{Fwd}-P_{Rev}}{P_{Rec}}\right)$$

Data Tab

This tab is used to store additional outputs from the calibration. The positioner and signal generator data is used for reference purposes. The number of points is used in the final chamber evaluation.

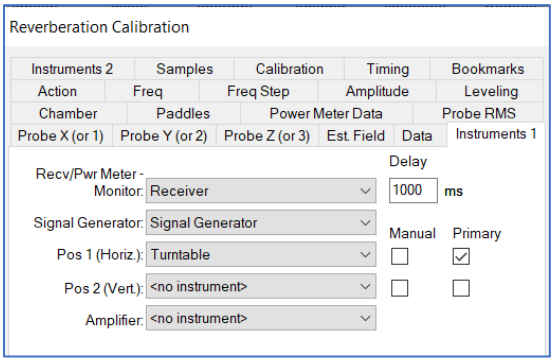


NOTE: If the Positioner 1, Signal Generator, and Number of Points data are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Positioner 1/ Positioner 2 Data** — Stores positioner data to identify the paddle position that had the maximum values. The position at which the highest field was generated is recorded.
- **Signal Generator** — Stores signal generator value for each frequency. Generally, this is a flat value, but if leveling is used, the signal generator level could be different for each paddle position.
- **Number of Points** — Stores the number of paddle positions is used for the evaluation step.

Instruments 1 Tab

This tab is used to specify various pieces of test equipment used during the reverberation calibration. Paddle positioners can be manually controlled and the operator will be prompted for motion if they are so marked.



NOTE: If the Recv/Pwr Meter, Signal generator, and Pos 1 instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

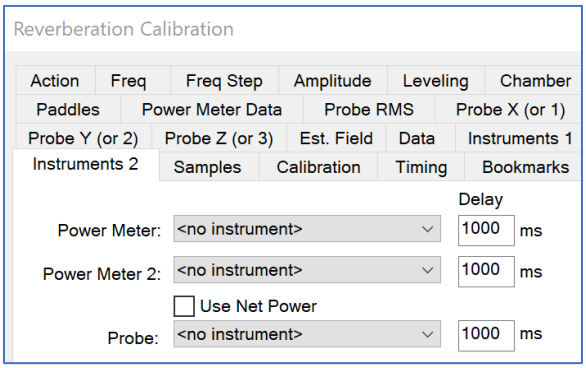
- **Recv/Pwr Meter Monitor** — Select the instrument from the list that will be used to monitor the received power in the chamber on the receive antenna. If you select an instrument that does not have the characteristics of a power meter, the action will not work properly. Since a spectrum analyzer can also be a power meter either type of equipment can be referenced here.
- **Delay** — The time in milliseconds that the instruments needs to level internally prior to reporting values. Basically, this is the time between the changes in signal generator level and the actual reading being taken.

  Typically, spectrum analyzers do not require any delay, since their internal sweep times are adequate delays. Power meters often require some internal leveling time prior to reporting correct values.
- **Signal Generator** — If you select an instrument that does not have the characteristics of a signal generator the action will not work properly.

- **Pos 1 (Horiz.)/ Pos 2 (Vert.)** — If there is only one positioner, it should be identified as positioner 1. Although not required, generally if there are two positioners one is in vertical orientation and one is horizontal. The reference to horizontal and vertical is for identification purposes only, and is not mandated by any standard.
- **Amplifier** — If the amplifier is bus controlled, Select the correct instrument. If the amplifier is not bus controlled, set a Prompt action to instruct the operator to put the amplifier in operate mode prior to executing this step.
- **Manual** — This setting only applies to Positioner 1 and Positioner 2 values if either is manually controlled, select the check box and the software will prompt the operator when it is time to move to each paddle position.

Instruments 2 Tab

Use this tab to set the power meter and probes used during the calibration.



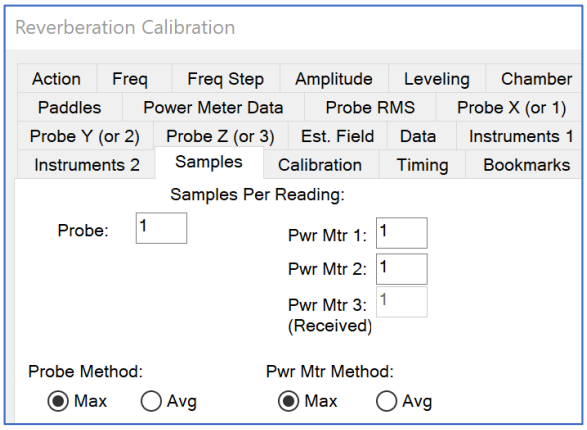
NOTE: If the Power Meter instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Power Meter/ Power Meter 2** — If you select an instrument that does not have the characteristics of a power meter the action will not work properly.

  Since a spectrum analyzer can also be a power meter either type of equipment can be referenced here. If the instrument is a two-channel power meter, the software will address channel 1/channel 2 unless a channel specific instrument driver is used.
- **Use Net Power** — If this check box is selected, the net power will be calculated using power meter 1 as forward and power meter 2 as reverse. Net power is not the norm for reverberation testing and is included here only for R&D purposes.
- **Delay** — Some probes and power meters have internal leveling issues that make it necessary to allow minor delays between changes in the signal generator levels and the reading of each instrument. There is a separate delay time for each instrument.
- **Probe** — If you select an instrument that does not have the characteristics of a probe the action will not work properly.

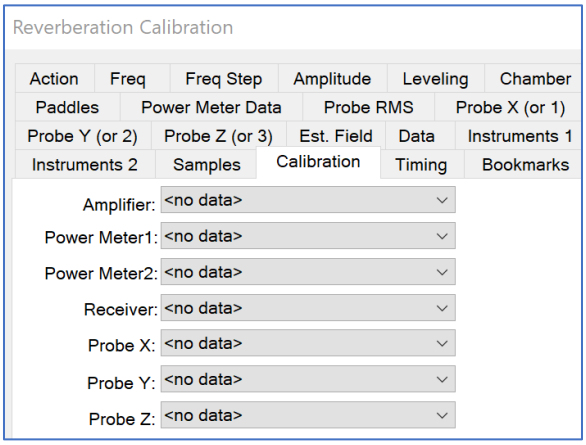
Samples Tab

This tab is similar to other samples tabs and will not fully be described here. See the *Immunity Test Samples Tab* for the full description.



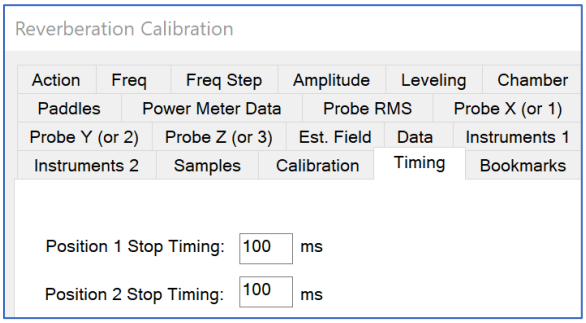
Calibration Tab

This tab controls the various correction factors to be used for each instrument during the test. These factors are added to the instrument value to give a corrected value.



- **Amplifier** — The amplifier calibration is the gain of the amplifier in dB. This is used internally when initial signal generator levels are calculated prior to setting the signal generator on.
- **Power Meter 1/Power Meter 2** — The power meter calibration represents the amplitude in dB of any losses between the sensor and the actual field. For example, if you have a directional coupler, this would be the loss across the monitor port of the coupler. This value is added to the instrument reading to correct it.
- **Receiver** — The receiver calibration represents the amplitude in dB of any losses between the sensor and the actual field. For example, if you have an attenuator on the cable coming from the chamber to the wall to the instrument, this would be the loss of the attenuator. This value is added to the instrument reading to correct it.
- **Probe X/ Probe Y/ Probe Z** — The probe calibration represents the difference between the probe reading and the calibrated value. The probe corrections are typically represented as percentage of full meter deviation when calibrated by the manufacturer. The correction factor for the probes is a value that is multiplied against the actual reading to get a calibrated value. This is a number around 1 (100%). Therefore, a correction factor of 95% would be 0.95; 105% would be 1.05.

Timing Tab



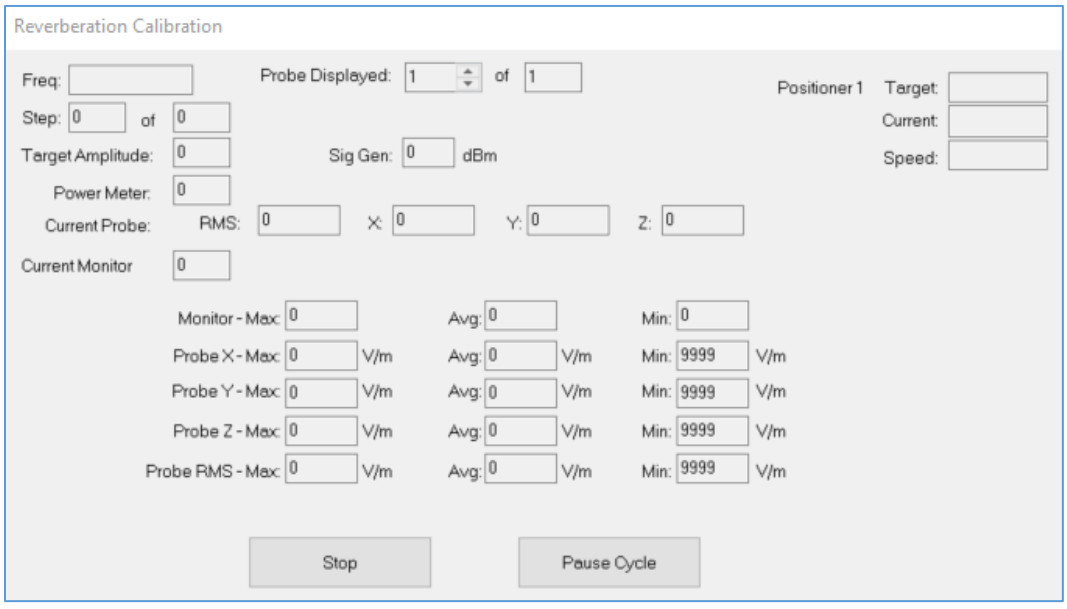
Certain positioners are still vibrating after they have completed motion. It can take a few seconds for the oscillation to fully dampen out. The stop times are delays inserted between the motion commands and any other command that does a measurement.

This ensures that the readings are taken after all motion has fully ceased. In reverberation, the chamber is very sensitive to paddle motion and a paddle that is oscillating will generate bad data.

- **Position 1 Stop Timing / Position 2 Stop Timing** — This is the time after completion of the positioner 1 movement that the software will delay before starting the amplitude measurements.

Dialog Box

When you are executing the Reverberation Calibration, a dialog box will display showing the current frequency, step number, amplitude target, current amplitude (power meter), sig gen level, tuners' positions (targets and current), and probes and monitor data. The final level is saved when it falls within the tolerances defined. Reverberation calibration only saves the information it is configured to save. The Stop button will halt execution of the current action; No information is saved.



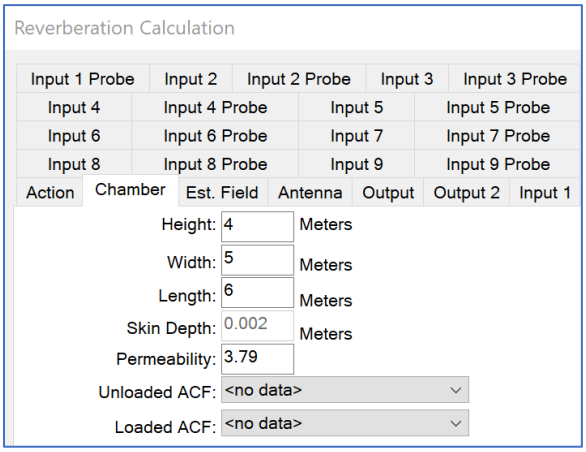
Reverberation Calculation (EVAL)

The various standards in use for reverberation testing all require chamber calibrations. This is the measurement of a full positioner movement(s) with the receiving antenna and probe repositioned multiple times to test the stability of the radiated field.

Using the Reverberation Calibration action, you would run a full sequence, reposition the antenna and probe, run another full sequence, and reposition for the number of positions desired. This data is then combined into a final chamber calibration. The Reverberation Calculation action is designed to perform this merging of data and output the various mathematical results required to use the chamber.

Chamber Tab

This tab is similar to other chamber tabs and will not fully be described here. See the *Reverb CAL Chamber Tab* for the full description.

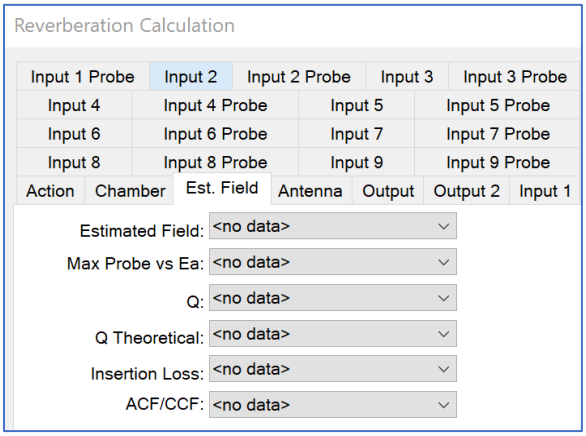


The Antenna Calibration Factor (ACF) is defined as **ACF = (Monitor_Watts/Inject Power Watts)**. It is important not to confuse this term with the commonly accepted EMC usage of this term. In general, the ACF would be the correction factor of the antenna.

In reverberation, ACF is the efficiency of the transmit/receive antenna pair in the geometry of the chamber using multiple paddle positions to find peak amplitudes. The unloaded and loaded ACF are used in the calculation of the CLF and load factors.

- **Unloaded ACF** — This field is only required if the evaluation is being done for EUT loading analysis. This data element is not used when the evaluation is used for chamber acceptance. If this field is blank, then you cannot perform the analysis for the CLF on the *Output 2 tab*.
- **Loaded ACF** — The unloaded and loaded ACF are used in the calculation of the CLF and load factors. Select a data element for this input. This field is only required if the evaluation is being conducted for EUT loading analysis. If this field is blank, then you cannot perform the analysis for the load factor on the *Output 2 tab*.

Est. Field Tab



- **Estimated Field** — Stores the maximum calculated field value found for each of the input positions. Select a data element for this output. The estimated field is calculated as (Reference: IEC 61000 4 21):

$$E_{Est} = \left\langle \frac{8\pi}{\lambda} \sqrt{5 \frac{P_{MaxRec}}{\eta_{rx}}} \right\rangle_{\eta}$$

- **Max Probe vs Ea** — For reference purposes you can store a value that represents maximum field probe vs. the estimated field. Select a data element for this output.
- **Q** — Stores the maximum calculated Q value found for each of the chamber probe positions. Q was a requirement of the older SAE standard; these values are for reference only. (Reference from IEC 61000 4 21. See also in *Reverberation Calibration* for explanation of the symbols.)

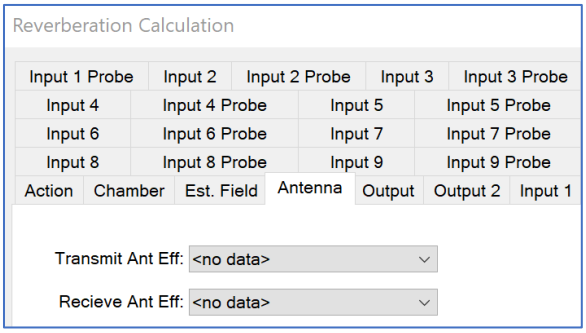
- **Q Theoretical** — Theoretical Q value can be used as a reference for the chamber. Though this is only useful as a reference, it is calculated each time this action is executed.
- **Insertion Loss** — Stores the maximum insertion loss value found for each of the chamber probe positions. These values are used during initial chamber verification as part of the evaluation. Insertion loss is defined as:

  Insertion Loss (L) = Monitor_Max_Watts/Injected Power_Watts
- **ACF/CCF** — Derived from the maximum monitor watts and injected power watts during the multiple probe positions. When multiple probe positions are analyzed, the term changes from ACF to CCF. Antenna Calibration Factor (ACF) is defined as:

  ACF = (Monitor_Watts/Inject Power Watts)

Antenna Tab

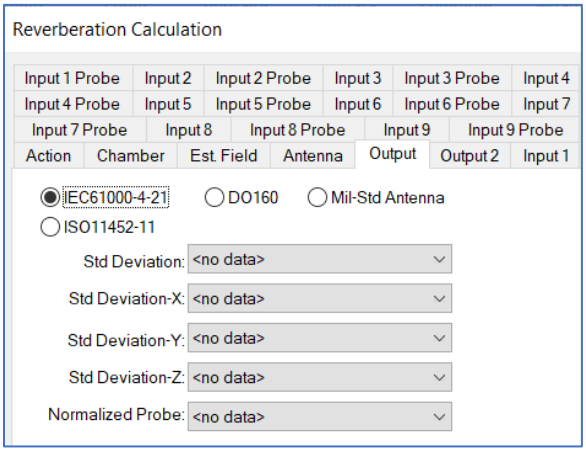
Antenna efficiency is an integral part of the chamber evaluation. For practical purposes, the standards committees have standardized these efficiencies to be 0.75 for a log periodic type antenna and 0.90 for a horn type antenna. Within TILE!, we use a data element so that in the future the option is there to fine tune chamber analysis by more rigorous antenna measurements.



- **Transmit Ant Eff/ Receive Ant Eff** — At a minimum, create a preset type data with a value of either 0.75 (log periodic type antennas) or 0.90 (horn type antennas) and use this data element here.

Output Tab

This tab allows the user to output certain statistical calculations and the normalized probe, which is the most critical component in the evaluation process. Normalized probe is the ratio that is used to calculate the required energy to create a field during the reverberation test.



- **Std Deviation/ Std Deviation-X/ Std Deviation-Y/ Std Deviation Z/Normalized Probe** — Select a data element for the output.
- **Normalized Probe Method** — Select the method for performing the normalized probe calculations. The IEC 61000 4 21 method uses the sum of the individual probe axes. The DO 160 method uses the sum of the RMS of the individual axis. The MIL STD method is illustrated in MIL STD 461E in the section on RS103.

  The standardized deviations are calculated as according to IEC 61000-4-21 2011:

$$\sigma = \sqrt{\frac{\sum(\vec{E}_i - \langle \vec{E} \rangle)^2}{\eta - 1}}$$

These calculations are performed for each axis of the probe and for the sum of the probe.

Where:

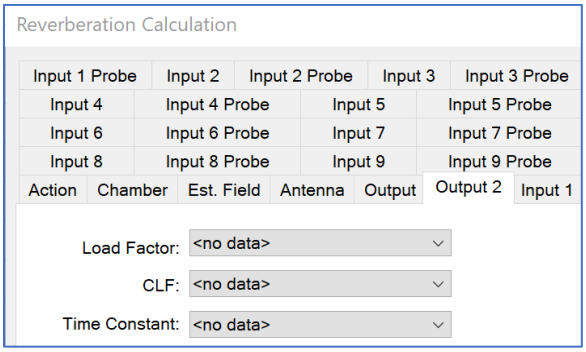
η – The number of measurements *i*.

$\vec{E}_i$ – The individual E-field measurement normalized (to the square root of the input power).

$\langle \vec{E} \rangle$ – The arithmetic mean of the normalized maximum E-field measurements.

Output 2 Tab

For chamber evaluation purposes the load factor, CLF, and time constant are calculated. Use the Output 2 tab to select output data elements for each.



- **Load Factor** — Load factor is the unloaded ACF divided by the loaded ACF.
- **CLF** — The Chamber Load Factor (CLF) is calculated in two steps. First the Chamber Correction factor (CCF) is calculated and then the CLF. Select a data element for this output (Reference: IEC 61000 4 21 2011).

$$CLF = \frac{CVF}{AVF}$$

$$CVF = \left\langle \frac{P_{AveRec}}{P_{input}} \right\rangle_{\eta}$$

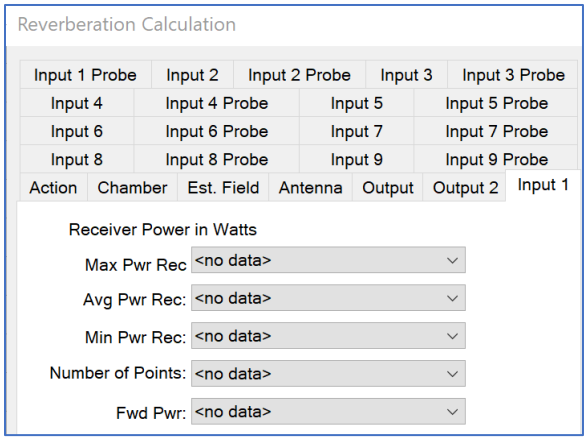
$$AVF = \left\langle \frac{P_{AveRec}}{P_{input}} \right\rangle_{8\ at \leq 10f_0\ or\ 3\ at > 10f}$$

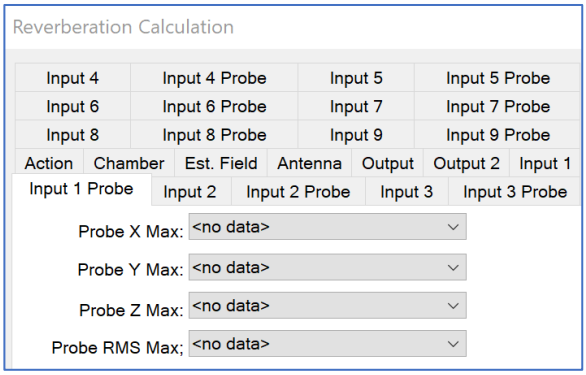
- **Time Constant** — The Time Constant (T) is calculated as:

  T = Q/2πf (where f is frequency in Hz)

Input 1-9 and Input 1-9 Probe Tabs

There are two types of tabs (Input and Input Probe) for each of the nine potential probe positions. Within each probe position, you need specific pieces of data for the evaluation. If any piece is missing, you will be given a warning at the time of execution. Eight positions are required for IEC 61000 4 21. The DO 160 requires nine positions. The information for this tab should come from the calibration done at this probe positions.



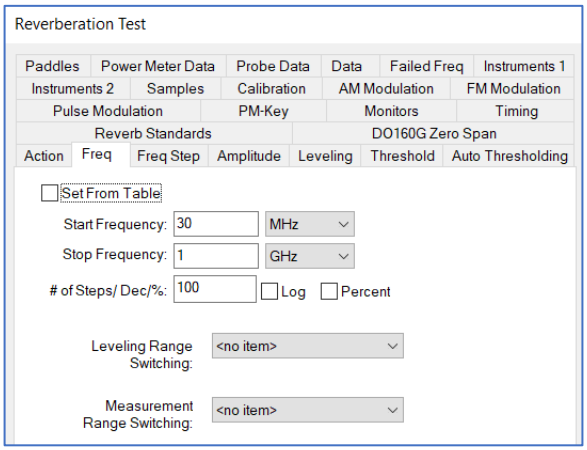


Reverberation Test (TEST)

Reverberation Testing is done using the Normalized Probe information from the Reverberation Evaluation. A desired field is established by multiplying the Normalized Probe by the desired level to get a required input power.

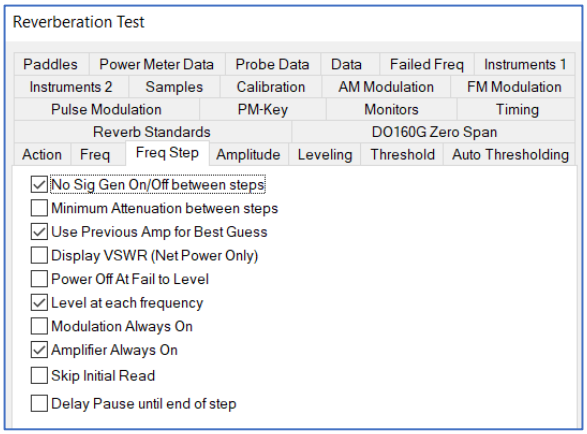
Freq Tab

This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Frequency Tab* for the full description.



Freq Step Tab

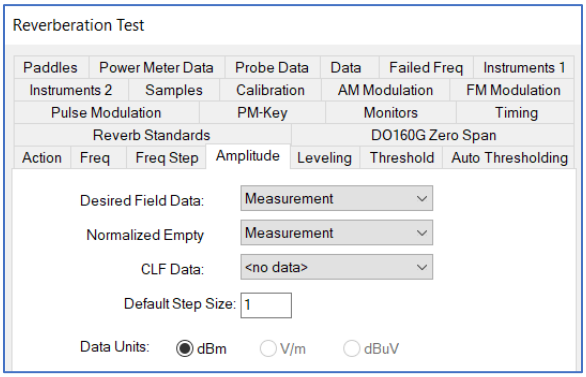
This tab is similar to other frequency tabs and will not fully be described here. See the *Immunity Test Freq Step 1 Tab* for the full description.



- **Level at each frequency** — If this check box is selected, the software will automatically level the field when in manual control as you step to each frequency. When this is off, when changing frequency, the amplitude of the signal generator is left alone.

Amplitude Tab

The field during the EUT test is established using the equations in IEC 61000 4 21 2011. This tab sets the Input data elements that contact the information required to make this calculation.



NOTE: If the Desired Field Data, Normalized Empty, and CLF Data are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Desired Field Data** — This data element is the amplitude of the desired RF field. It can be set from a Preset value or change with frequency using a file-type data element. The field will be determined using the following:

$$P_{Input} = \left[\frac{E_{Test}}{\langle \vec{E} \rangle_{24 \text{ or } 9} \times \sqrt{CLF(f)}} \right]^2$$

Where:

P_{Input} – Forward power injected into the chamber.

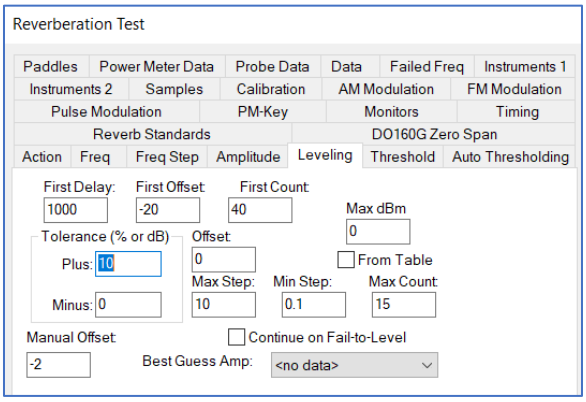
E_{Test} – Required field strength.

$CLF(f)$ – Chamber load factor at frequency.

$\langle \vec{E} \rangle_{24 \text{ or } 9}$ – Average of the normalized E field from empty chamber.

- **Normalized Empty** — This is the normalized field probe from the evaluation step.
- **CLF Data** — This is the Chamber Loading Factor (CLF) determined from the pretest calibration and evaluation.
- **Default Step Size** — This field is disabled.
- **Data Units** — Generally all testing is done in dBm; V/m and dBuV are not active.

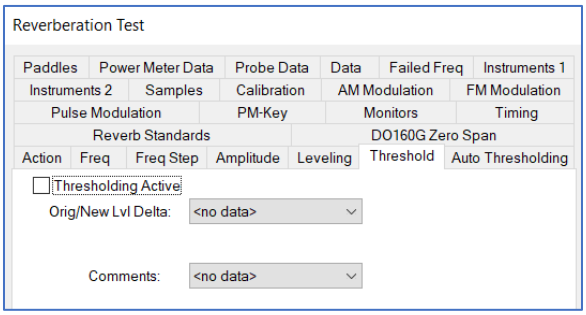
Leveling Tab



This tab is similar to other leveling tabs and will not fully be described here. See the *Reverb CAL Leveling Tab* for the full description.

- **First Delay** — This allows the amp to come up to power.
- **First Offset** — This is the initial signal generator offset for the first frequency of every tuner position. This is useful for amp gain variations requiring a larger offset for each first frequency.
- **First Count** — This is the number of steps to try and level before any failure message is given to the operator. This assumes you are using a large first offset and that you may need more steps for the first one.
- **Tolerance Plus** — For power leveling use dB and for an E-Field probe use percentage. This is a legacy feature.
- **Tolerance Minus** —Set this to zero to prevent leveling below the target. Similar to Plus, this is a legacy feature.
- **Offset** — This is a signal generator offset for all frequencies except for the first frequency of every tuner position. This is useful to create smaller offsets for each subsequent frequency and a larger offset when going back to the start frequency for each tuner position.
- **Max Step** — Max sig gen step size.
- **Min Step** — Min sig gen step size.
- **Max Count** — This is the max number of leveling tries for all steps but the first one of each position.
- **Manual Offset** — When in manual mode (automatic testing paused), this value controls the offset to the signal generator on initial power on. This is the signal generator offset for all releveling attempts due to adjustments made in the pause manual controls screen.

Threshold Tab



- **Thresholding Active** — These settings only apply when the operator is within semi auto mode for thresholding. When in this mode, the operator can pause the test process to launch another dialog when there is EUT failure (required for thresholding).
- **Orig/New Lvl Delta** — Select an appropriate data element to store the thresholding values. These values will be reduced from the paused level when a failure is observed.

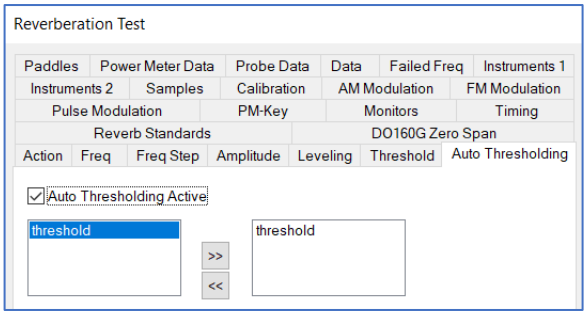
- **Comments** — Comment tables recorded during the semi-automatic thresholding process. A character string and the tested frequency will be stored when the operator chooses to enter comments.

Auto Thresholding Tab

Select the **Auto Thresholding Active** check box to activate thresholding.

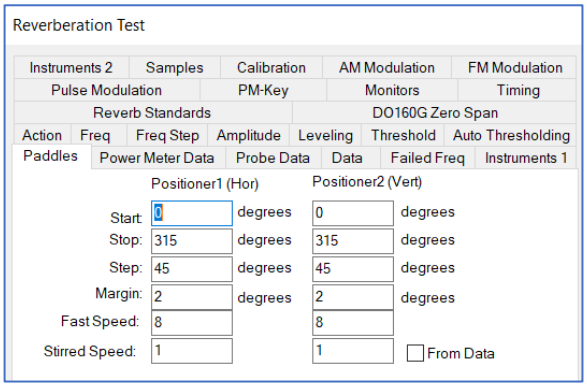
It is divided into two columns, Available and Selected. They can be selected by double clicking on them or selecting the Add>> button. They can be removed by double click or the <<Remove button.

See the *Auto Thresholding* action for more details.



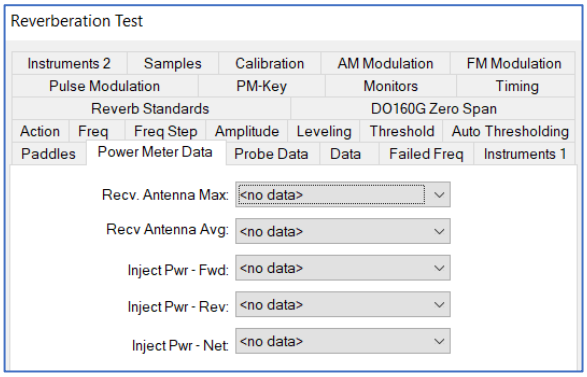
Paddles Tab

This tab is similar to other paddle tabs and will not fully be described here. See the *Reverb CAL Paddles Tab* for the full description.

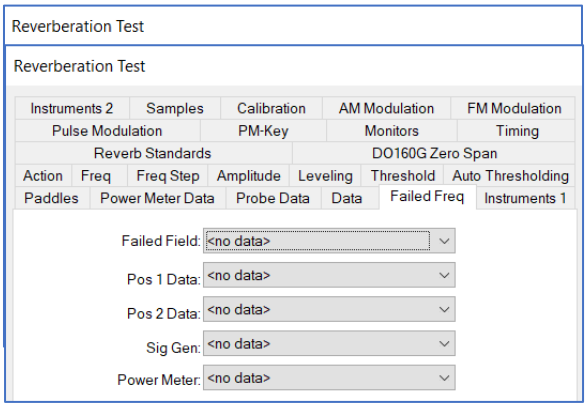


Power Meter Data Tab

This tab is similar to other data tabs and will not fully be described here. See the *Reverb CAL Power Meter Data Tab* for the full description.



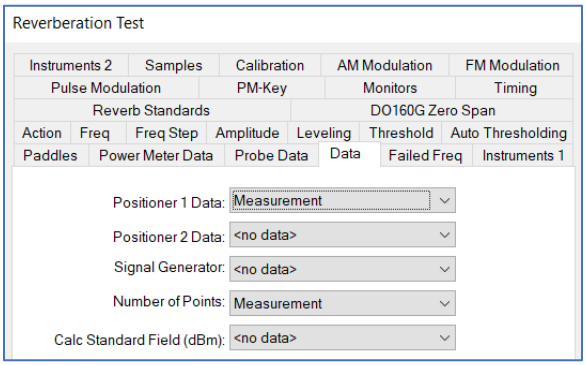
Probe Data Tab



This tab allows you to store various measurements made during the test from actual probes in the field or from the internal calculations done as part of the field generation. During testing, the probe data is for informational use only.

- **Probe X-Max / Probe Y-Max / Probe Z-Max** — The maximum value of the probe for all tuner positions is stored.
- **Probe Max** — The maximum value of the RMS value of the probe for all tuner positions is stored.
- **Calculated Probe** — The calculated probe value is derived from the Expected Field and the ratio of the actual power and the expected power. Given the tolerances of the leveling loop, the actual field might be slightly off from the desired field. Storing this information allows analysis later of the field generated in the chamber.

Data Tab



This tab is similar to other data tabs and will not fully be described here. See the *Reverb CAL Data Tab* for the full description.

NOTE: If the Positioner 1 and Number of Points data are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.

- **Calc Standard Field (dBm)** — In performing a test the desired field and normalize probe are mathematically manipulated to yield the power level required to generate the desired field. The highest level for all paddle positions is recorded.

Failed Freq Tab

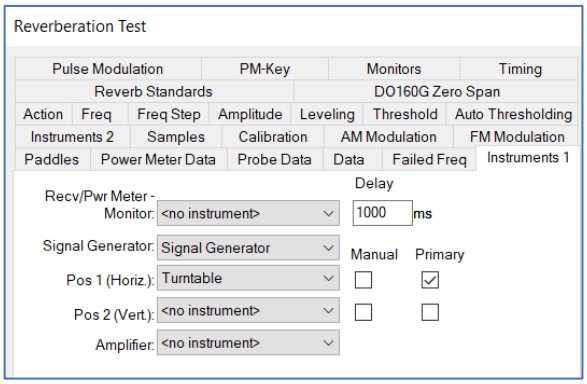
When the operator tags a failure, certain information about the current position is stored for later use, separate from the standard outputs of the program.

- **Failed Field** — Stores the frequencies at which a failure is noted.
- **Pos 1 Data** — The paddle position for each frequency that is marked as a failure is stored. Select a data element for this output (**required**).
- **Pos 2 Data** — If there are two paddle positioners, the position of paddle 2 when a failure is marked is stored. Select a data element for this output (**required only if position 2 is used**).
- **Sig Gen** — The signal generator level at the point when a failure is marked is stored.
- **Power Meter** — The current power level when a failure is marked is stored.

Instruments 1 Tab

This tab is similar to other instruments tabs and will not fully be described here. See the *Reverb CAL Instruments 1 Tab* for the full description.

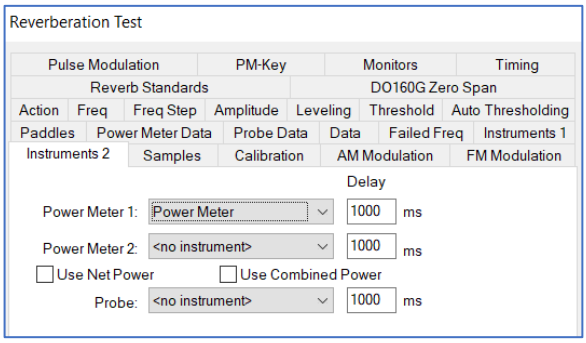
NOTE: If the Signal generator and Pos 1 instruments are not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



Instruments 2 Tab

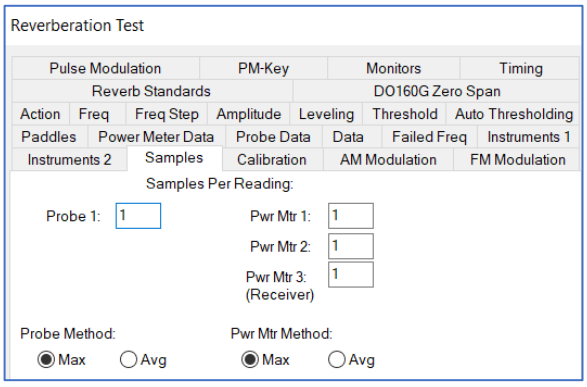
This tab is similar to other instruments tabs and will not fully be described here. See the *Reverb CAL Instruments 2 Tab* for the full description.

NOTE: If the Power Meter 1 instrument is not specified, then an error will appear stating the missing element and the dialog box will not appear and/or not function properly.



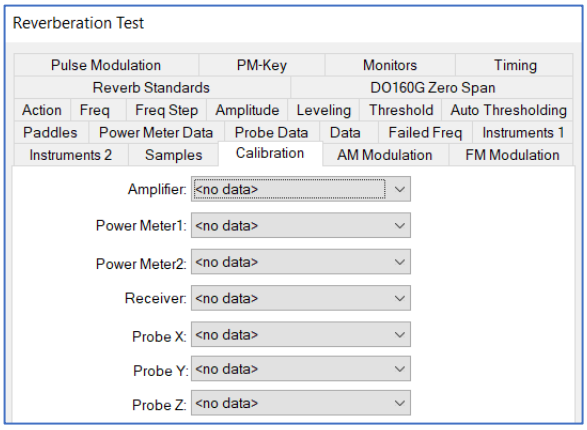
Samples Tab

This tab is similar to other sample tabs and will not fully be described here. See the *Immunity Test Samples Tab* for the full description.



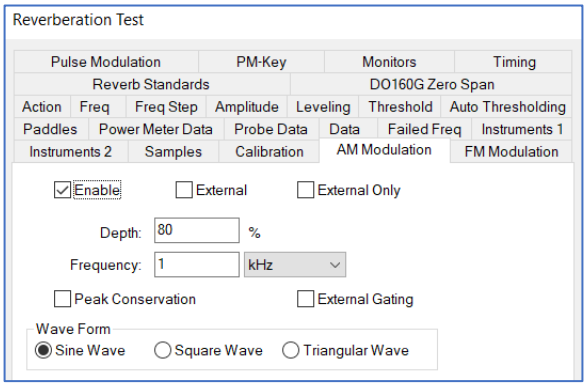
Calibration Tab

This tab is similar to other calibration tabs and will not fully be described here. See the *Reverb CAL Calibration Tab* for the full description.



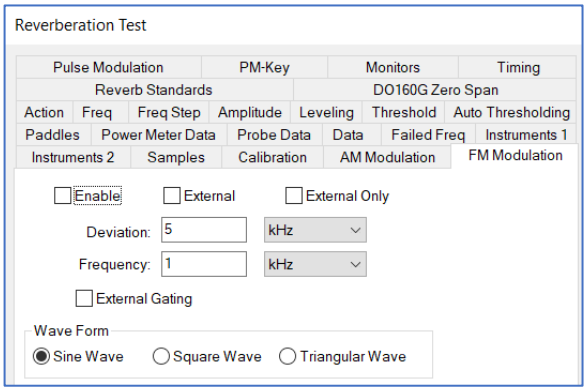
AM Modulation Tab

This tab is similar to other modulation tabs and will not fully be described here. See the *Immunity Test AM Modulation Tab* for the full description.



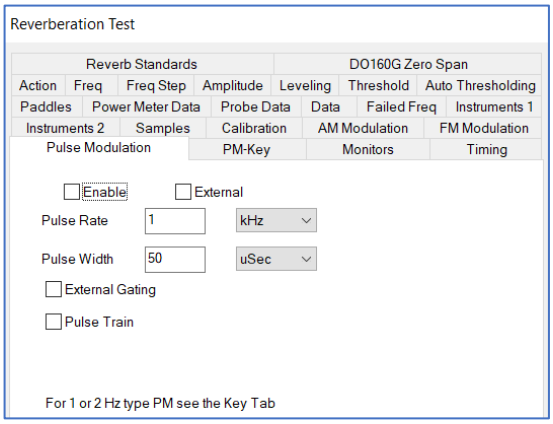
FM Modulation Tab

This tab is similar to other modulation tabs and will not fully be described here. See the *Immunity Test FM Modulation Tab* for the full description.



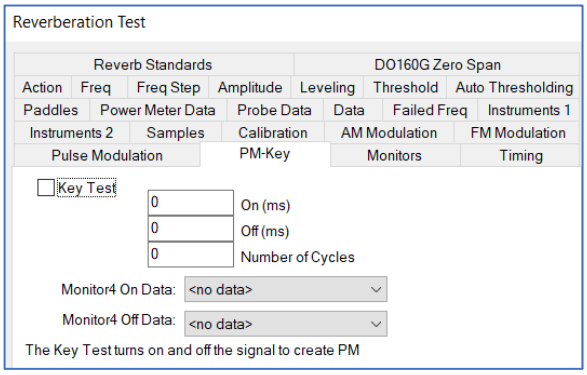
Pulse Modulation Tab

This tab is similar to other modulation tabs and will not fully be described here. See the *Immunity Test Pulse Modulation Tab* for the full description.



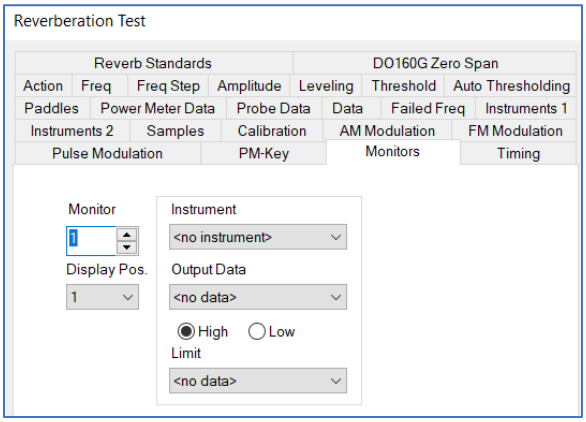
PM-Key Tab

This tab is similar to other PM-key tabs and will not fully be described here. See the *Immunity Test PM-Key Tab* for the full description.



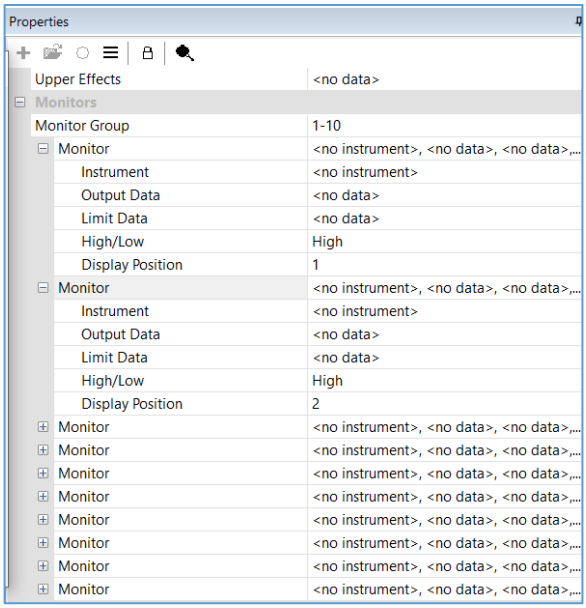
Monitor Tabs

The monitor tab control equipment that will be read and stored during the process time of the action, after all leveling has been completed. This allows you to monitor various conditions of the EUT while the test is running automatically.



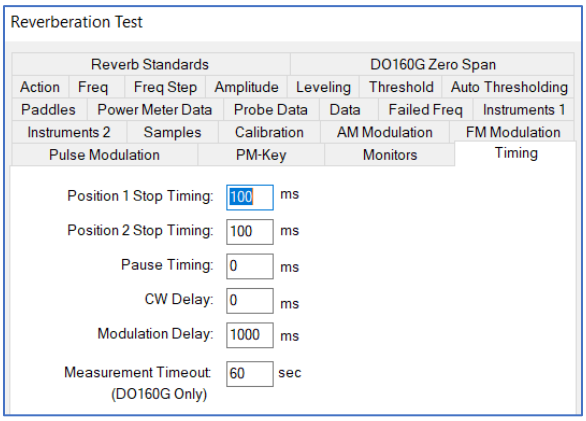
- **Instrument** — Select an instrument to monitor. It must be defined on the *Instruments tab*.
- **Output Data** — Select a data element to store the results of this reading.
- **High/Low** — For multiple reads (CW and modulation stages) specify whether to save the high or low readings.
- **Limit** — If a data element is selected here the values from this data element will be displayed on the screen for the operator to see an expected value next to the actual value.

NOTE: The maximum number of monitors that can be configured within a test profile is 100. Use the properties window to manage the monitors in groups of 10 as seen below.



Timing Tab

The Reverberation action has many timing issues that affect accuracy of the test. Use this tab to set the desired timings.

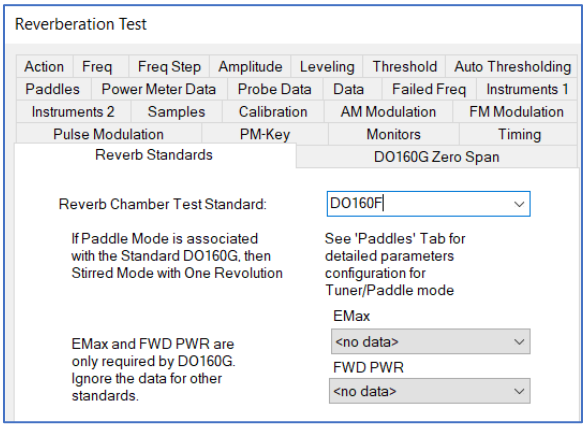


- **Position 1 Stop Timing/Position 2 Stop Timing** — It is imperative that the paddle be completely still before any measurements are taken. The stop timing is the time that the software will dwell between the sending the stop command and any further measurements.

  Paddles driven by stepper motors are less of a concern then those with non-stepped motors. Use the position 1 setting for positioner 1 and the position 2 setting for positioner 2.

- **Pause Timing** — A dwell that is executed between the first RF on command sent and any subsequent measurements.
- **CW Delay** — A dwell time between the completion of the leveling loop (usually done in CW mode) and the start of the modulation cycle.
- **Modulation Delay** — The amount of time that the modulation will be active.

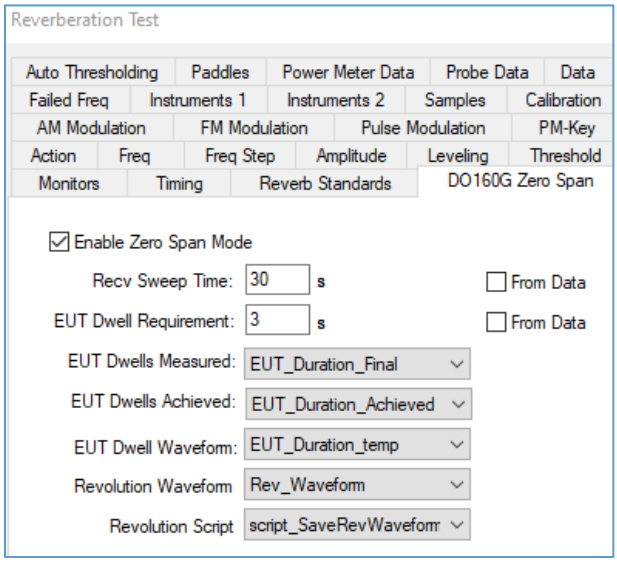
Reverb Standards Tab



The Reverberation action has several reverberation standards supported.

- **Reverb Chamber Test Standard** — Supported standards:
    - **DO160F** — The stepped tuned standard for DO160.
    - **DO160G Load** — The pre-test or EUT Load test for stirred mode standard for DO160G.
    - **DO160G Test** — The stirred mode standard for DO160.
    - **IEC1600-4-21** — The IEEE standard.
    - **MIL-STD461** — The military standards: F or G.
- **EMax** — The max e-field used for DO160G. Produced by the load test or pre-test. Used by DO160G test. The units are in V/m.
- **FWD** — The forward power used for DO160G. Produced by the load test or pre-test. Used by DO160G test. The units are in dBm.

DO160G Zero Span Tab

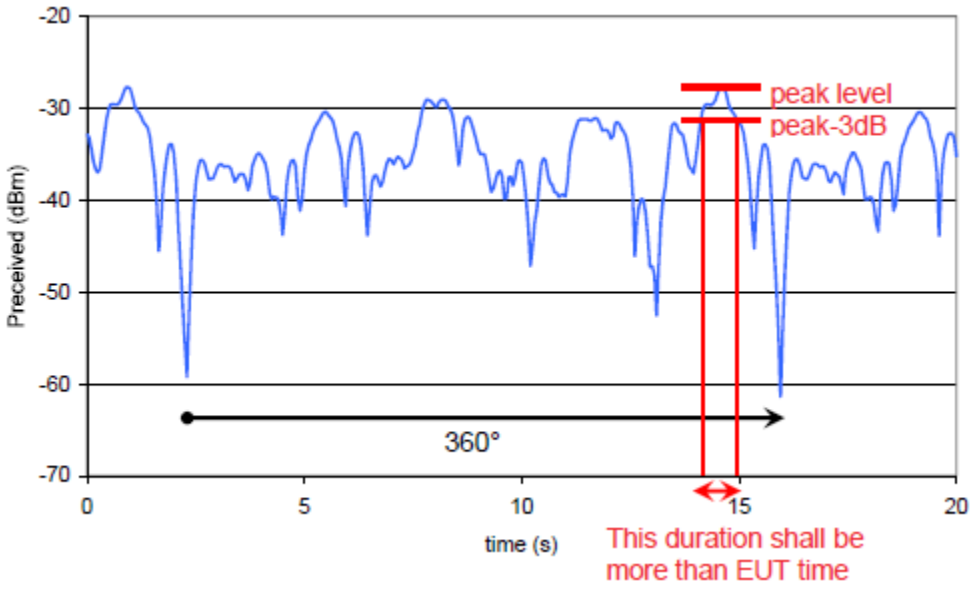


Zero Span Mode implements the EUT Test procedure described in DO160G 20.6.2.c using zero span mode on the signal analyzer. After each zero span trace is acquired from the analyzer, the action then post-processes the trace and determines if the EUT Dwell requirement has been satisfied for that test frequency.

This tab specifies the settings and data elements used for DO160G Zero Span Mode.

- **Enable Zero Span Mode** — Enables Zero Span Mode when the Reverb Chamber Test Standard is either DO160G Load or DO160G Test.
- **Recv Sweep Time** — Sets the sweep time for the receiver/analyzer trace in seconds.
- **Recv Sweep Time From Data** — Enables using a data element to set a frequency based Recv Sweep Time (different sweep time value for each frequency step).
- **EUT Dwell Requirement** — The required EUT Dwell time in seconds. The measured EUT Dwell time will be compared to this value to determine whether the EUT Dwell was achieved at each frequency step.

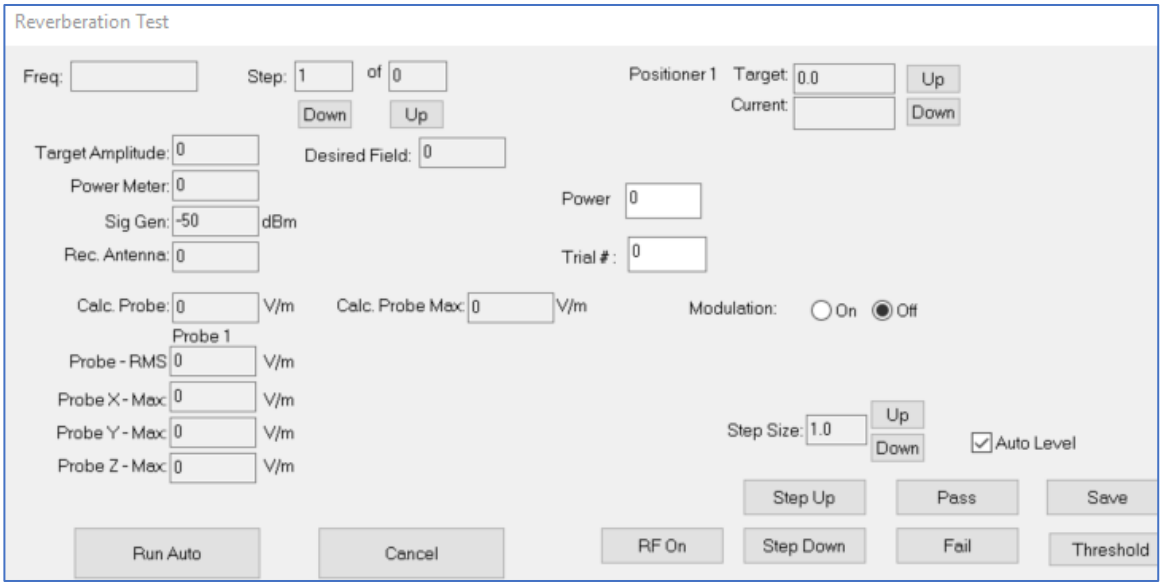
- **EUT Dwell Requirement From Data** — Enables using a data element to set a frequency based EUT Dwell Requirement (different EUT Dwell Requirement for each frequency step).
- **EUT Dwells Measured** — The measured values of the EUT Dwell durations at each frequency step. This calculation is the total consecutive time in seconds that the power of the zero span trace exceeds the peak power minus 3 dB.



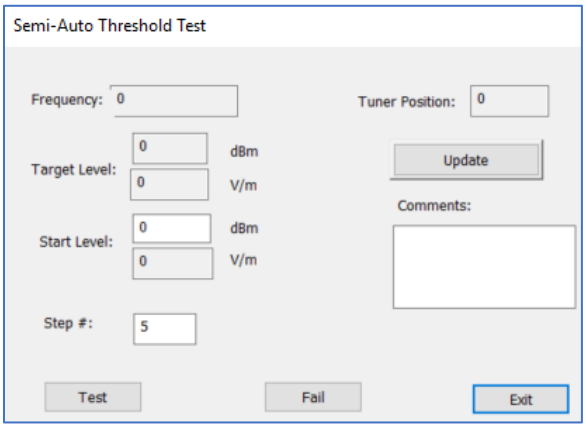
- **EUT Dwells Achieved** — Records a 1 (achieved) or 0 (not achieved) at each frequency step depending on whether the EUT Dwell Measured was greater than or equal to the EUT Dwell Requirement.
- **EUT Dwell Waveform** — The output of the EUT Dwell waveform. This data element is the time domain trace that represents the portion of the zero span trace that exceeds the peak power minus 3dB (see previous diagram). This data waveform is overwritten as each new trace is acquired. It can be accessed at the end of each frequency step using the Revolution Script.
- **Revolution Waveform** — The zero span time domain trace that is retrieved from the receiver/analyzer for each frequency step. This data waveform is overwritten as each new trace is acquired. It can be accessed at the end of each frequency step using the Revolution Script.
- **Revolution Script** — A script that will run (if present) at the end of each frequency step. This script is useful for copying the temporary waveforms EUT Dwell Waveform and Revolution Waveform to permanent data elements that can be accessed after the action completes.

Dialog Box

When you are executing the Reverberation Test, you will have a dialog box showing a wide variety of information about this test. How this dialog box is used depends on the choice of leveling method, pass/fail, and process settings. The controls for this dialog box are summarized in the control buttons. The remainder of the dialog box is information for the user.



To access the Semi-Auto Threshold Test, you must enable thresholding in the settings on the threshold tab. Execute the action and access the manual dialog box as seen above. Select Threshold to see the following window:



- **Frequency/Tuner Position** — These values are determined at the pause mode when the user stops the automatic Process.
- **Target Level** — This is the level when paused by the user. The unit of the level shows in both dBm and V/m.
- **Start Level** — This is the user defined. By default, it is defined in the data element of Threshold tab: Orig/New Lvl delta. User can adjust to lower or higher level based on needs.
- **Step#** — This integer number will be used to define the leveling up step: (Target – Start)/Step#. This can be modified by the user, but cannot be smaller than two.
- **Update/Test On Going** — Toggle when Test button clicked. In the setup stage, user should use Update button to make sure all the inputs are entered. Once in test mode after clicking the Test button, the Test button disappears and Test On Going button will appear, but has no function.
- **Fail** — This button is used to record the failure. The following dialog will pop-up and the mode returns to Setup. You can also pressure the space bar on the key board while the Test On Going to the same effect as click “Fail”. You can make comments in the Comments box. The failure will be record.
- **Exit** — This will finish the Semi-Auto Threshold process without further recording failure.

IEC 61000 Series Test Actions

The IEC 61000 Series Test actions are used to connect and control instruments that are specifically designed for these tests. Since there are no outputs to be captured in the frequency domain, it cannot be used in the graph or tables.

- Harmonics (EN: -2)
- Burst (EN: -4B)
- EFT (EN: -4E)
- Surge (EN: -5)
- Dips (EN: -11)

Special Math Functions

Available Operators and Conditionals

Add {x + y}	Simple addition function.
Subtract {x – y}	Simple subtraction function.
Multiply {x * y}	Simple multiplication function.
Divide {x / y}	Simple division function.
Modulus {x % y}	A function that returns the remainder of a division equation. This is the same result as the function **{x – (x / y) * y}.**

Available Conditionals

The results of the conditionals are dependent upon the interpolation settings for the data element.

Equal {x == y}	Determines those frequencies of the two data elements in which the values are equal (exactly equal). The resultant data element will only have frequencies where this condition is true. If either of the data elements is interpolated, you might see strange results so make sure the definition of each data element is clear.
Not equal {x != y}	Determines those frequencies of the two data elements in which the values are not equal. The resultant data element will only have frequencies where this condition is true. If either of the data elements is interpolated, you might see strange results so make sure the definition of each data element is clear.
Greater than {x > y}	Determines those frequencies where the *x* value is greater than the *y* value. This is the same as the Max function, except the Max function will also work with word-type data elements.
Less than{x < y}	Determines those frequencies where the *x* value is less than the *y* value. This is the same as the Min function, except the Min function will also work with word-type data elements.
Greater than or equal to {x ≥ y}	Determines those frequencies where the value is greater than or equal to the *y* value.
Less than or equal to {x ≤ y}	Determines those frequencies where the *x* value is less than or equal to the *y* value.

Special Math Functions

abs(x)	Absolute value of each value in the data element x.
acos(x)	Determines the Anti-Cosine of each value in the data element x.
acosh(x)	Determines the Hyperbolic Anti-Cosine of each value in the data element x.
adb(x)	Converts the x value in dBA to a value in Amps using the equation **10^(x / 20)**. Same as the function **vdb(x).**
add(x,y)	This function adds values in x by data in y in a sequential step without checking the frequency. The number of elements should be the same for x and y. If there are more elements in x than in y, this function will not execute. If there are more elements in y than in x, it will execute but there will not be any results for the extra elements. For example, if there are five elements in x and four elements in y, there will be only four results. If interpolation is on for the results element, it may populate the results with inaccurate data.
Add1(x,y)	For each frequency in x and y, it will add the two values. If it does not find a matching value, it will linearly interpolate the result. This differs from add(x,y) by not requiring matching number of values and equivalent frequencies.
alog(x)	Anti-log of each value of the data element x. This is the value e raised to the value of x.
alog10(x)	Anti-log of each value of the data element x. This is the value 10 raised to the value of x.
asin(x)	Determines the Anti-Sine of each value in the data element x.
asinh(x)	Determines the Hyperbolic Anti-Sine of each value in the data element x.
atan(x)	Determines the Anti-Tangent of each value in the data element x.
atanh(x)	Determines the Hyperbolic Anti-Tangent of each value in the data element x.
avg(x,c)	Determines the average for **c** number of points in the data array x.
bottom(x,c)	Determines the bottom for **c** number of points in the data array x. The default for **c** is 10 if it is not stated.
bw(x,c)	Finds the highest value in range x and then finds the two frequencies that correspond to the c db down points. It returns the bandwidth (frequency difference) of the c db down point.
bwdiff(x)	Finds the highest value and then calculates the difference between the peak and all other values in the data set of x.
bwpts(x,c)	Finds the highest value in the range x and then finds the two frequencies that correspond to the c db down points. It returns three points with frequency and amplitude for the lower offset, center, and upper.

ceil(x)	Determines the smallest integer that is greater than or equal to x at each frequency.
cos(x)	Determines the Cosine of each value in the data element x.
cosh(x)	Determines the Hyperbolic Cosine of each value in the data element x.
Count(x)	Calculates how many items are in a data element.
dba(x)	Converts the x value in Amps to a value in dBA using the equation **20 * log(x)**.
dbp(x)	Converts the x value in Watts to a value in dBW using the equation **10 * log(x)**. To convert dBm to dB divide by 1 thousand.
dbuvtov(x)	Converts the x value in dBuV to a value in Volts using the equation **10^((x - 120) / 20)**. To convert dBuV to uV divide by 1 million.
dbv(x)	Converts the x value in Volts to a value in dBV using the equation **20 * log(x)**. For dBuV, multiply the voltage by 1 million.
dbw(x)	Converts the x value in Watts to a value in dBm using the equation **10 * log(x * 1000)**.
diff(a,b,c)	Compares the frequencies in data element x to those in data element y. If the difference between the frequencies is greater than the constant C (which you might consider the bandwidth in hertz), the frequency from element x is added to the resultant data element with the value of x.
div(x,y)	This function divides values in x by data in y in a sequential step without checking the frequency. The number of elements should be the same for x and y. If there are more elements in x than in y, this function will not execute. If there are more elements in y than in x, it will execute but there will not be any results for the extra elements. For example, if there are five elements in x and four elements in y, there will be only four results. If interpolation is on for the results element, it may populate the results with inaccurate data.
equal(x,y,c)	Compares the frequencies in data element x to those in data element y. If the difference between the frequencies is less than the constant c (which you might consider the bandwidth in hertz), the frequency from element x is added to the resultant data element with the value of x.
equalrev(x,y,c)	Compares the frequencies in data element x to those in data element y. If the difference between the frequencies is less than the constant c (which you might consider the bandwidth in hertz), the frequency from element y is added to the resultant data element with the value of x.
excl(x,y,c)	Compares the frequencies in data element x to those in data element y. Those frequencies in y that are also in x (within the ± bandwidth c in hertz) are excluded from the resultant data element. This is convenient for gaps in limit lines.
exp(x)	This raises the value of ***e*** to the power of **x**. This is the same result as the equation e^x**.**

Fabs(x)	Returns the absolute value of the floating-point value.
finddupl(x,c)	Scans the input data element x for duplicates and eliminates those frequencies within c range of each other, keeping the higher value. This is convenient for eliminating multiple peaks from modulated or noisy emissions.
firstfreq(x)	This will return the first frequency in x with the frequency and value fields. An example would be that if the first frequency in x is 30 MHz, @ 5 dBuV then the result would be 30e6, 30e6.
floor(x)	Determines the largest integer that is less than or equal to x for each frequency.
freq(x)	Puts the frequency into the value field and allows you to add the frequency to other values.
gmc(x)	Calculates a bin function for emissions data to the General Motors specification. This may not be used in the new world specification.
harmonic(x)	Returns the percentage values of all frequencies in the data element compared to the initial frequency in the array.
harmonicfind (x,c1,c2)	Finds the harmonics of a fundamental frequency of a data element. Evaluates multiples (odd and even) of the fundamental (**c1**) and finds the highest amplitude within the bandwidth (**c2**). **X** is the Data element, **c1** is the fundamental frequency, and **c2** is the Bandwidth window of subsequent harmonics.
iec9554(x)	Finds the maximum value in each bin of the IEC9554 matrix and returns this value for the start/stop frequency of the bin.
iff(x,y,z)	For each frequency in x, it attempts to find the equivalent frequency in y and z. If it does not find a match, it linearly interpolates y, if there is not a match in y, then the value z is returned.
impulse(x,c)	This function strips impulsive signals from a data stream. Impulse is defined as a single signal that exceeds the signal before and after by more than c (in hertz). If the value is defined too broadly or the frequency steps are too large, you may be stripping out real data.
j0(x)	Bessel function of each value in the data element x.
j1(x)	Bessel function of each value in the data element x.
lastfreq(x)	This will return the last frequency in x with the frequency and value fields.
ln(x)	This is known as the natural logarithm and it performs the log function with base **e** on each value in the data element *x*. This is the same result as the equation $\log_e x$.
log(x)	Log of each value in the data element x to the base e (natural base).
log10(x)	Log of each value in the data element x to the base 10.

max(x,y)	Compares the x and y values and returns the maximum of the two. It can be used to find the maximum value from H/V data but they must have the same number of points. This equation can also be used to add multiple trace or limit line data elements to make one element.
maxhv(h,v,x)	This function was created to determine whether the horizontal or vertical data had the maximum value and to create a related word-type data element, in this case x, with the text H or V. The resultant data element has the maximum value of the two (exactly like the Max function). The x data element will have the H/V to designate the polarity.
maxhvtt(h,v,x)	This function compares the polarity (from data element x) and matches the appropriate horizontal and vertical data element.
maxlr(x,y,z)	This function compares data element x to data element y and for each identifies in the word data element z whether the maximum was Line 1 or Line 2.
mergefreq(x,y,z)	Merges three data elements, keeping the maximum values and frequency. The three data elements should be the same.
min(x,y)	Compares the x and y values and returns the minimum of the two. This equation can be used like max(x,y) to link data items together.
mod(x)	Modulus command (same as the symbol %). A function that returns the remainder of a division equation. This is the same result as the function **x – (x / y) * y**.
mult(x,y)	This function multiplies values in x by data in y in a sequential step without checking the frequency. The number of elements should be the same for x and y. If there are more elements in x than in y, this function will not execute. If there are more elements in y than in x, it will execute but there will not be any results for the extra elements. For example, if there are five elements in x and four elements in y, there will be only four results. If interpolation is on for the results element, it may populate the results with inaccurate data.
notempty(x)	This function returns all frequencies in the word-type data element x that are not empty. The return value is the frequency and string. This can be used to find comments in the immunity test output so that only items with comments will be listed in a table.
offsetAdd(x,y)	For each frequency in x, it attempts to find an equivalent frequency in y. If it does not find a frequency, it linearly interpolates, and then adds the two values for that frequency and stores the frequency and new result. This is used to insert only correction factors for measured data so that extra interpolated points are not added to the result. An example would be scan data with 8001 points + factors of 401 points where some frequencies do not line up. Using the Scan + Antenna method may result in something like 8014 points. Using offsetAdd(Scan, Antenna) would result in 8001 points that are forced to align with the frequencies listed in the Scan data.

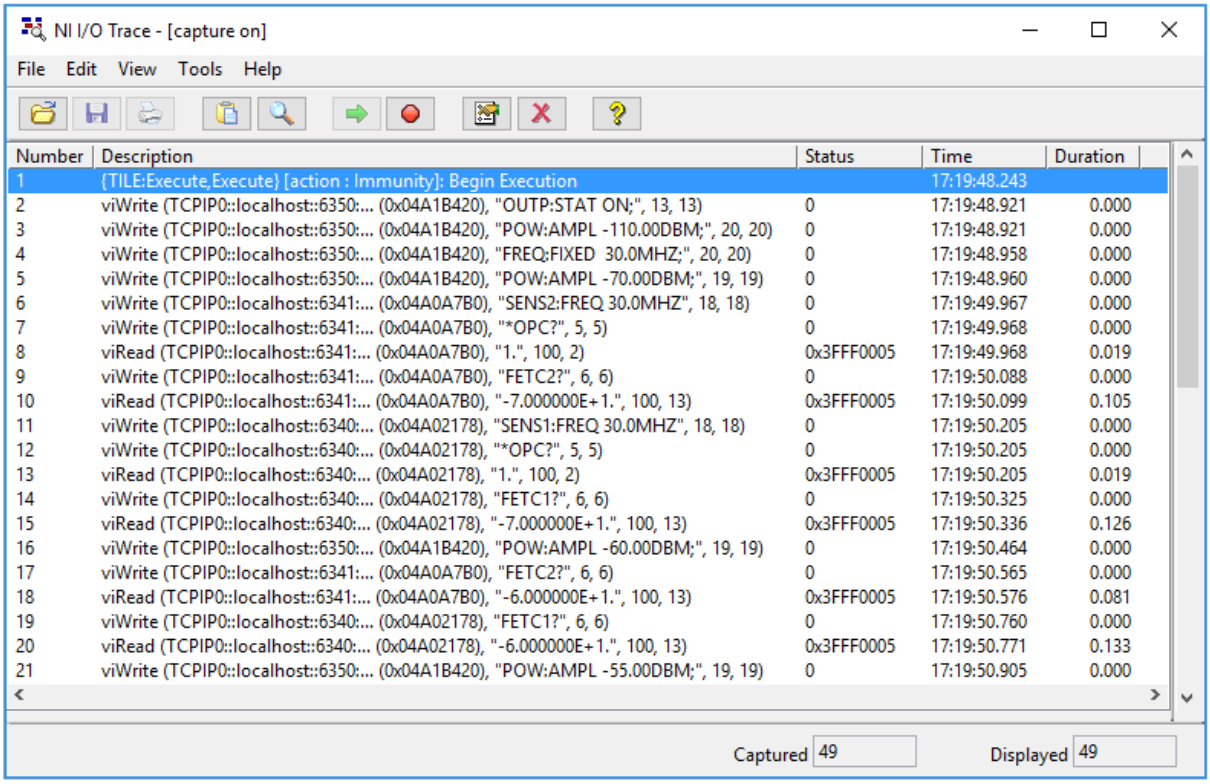
peak(x,c)	Returns all values that are greater than a constant c.
pfgreat(x,c)	Checks if the values in the data element **x** are greater than or equal to **c** and if so it returns the value **Pass**. If the value is less than **c**, then the return value is **Fail**.
pfless(x,c)	Checks if the values in the data element **x** are less than or equal to c and if so it returns the value **Pass**. If the value is greater than **c**, then the return value is **Fail**.
pdb(x)	Converts the *x* value in dBW to a value in Watts using the equation **10^(x / 10)**.
pow(x,c)	This raises the value of **x** to the power of **y**. This is the same result as the equation x^y**.**
reduce(x,c1,c2)	This function compresses the number of points in the data element ignoring any points that are within **c1** dB of the points on either side. To be ignored, the point must be *point before > current point > point after* and the difference between the point before and the current point must be less than c. In addition, the difference between the point after and the current point must be less than **c2**.
rint(x)	Rounds x to the nearest integer.
rms(x)	This function calculates the Riemann sum of the area under the curve represented by the data element *x*. The return value is the final frequency and the value.
scfm(x,y)	A correlation analysis on the data in elements x and y to the Ford specification. Standard Correlation for Ford Motor.
sign(x)	Gets the sign of integer x. Returns -1 if x is less than 0 or 1 if x is greater than 0.
sin(x)	Determines the Sine of each value in the data element x.
sinh(x)	Determines the Hyperbolic Sine of each value in the data element x.
Sorthigh(x)	Sorts data element in descending order.
Sortlow(x)	Sorts data element in ascending order.
span(x)	This will find the span of element x by subtracting the first frequency in x from the last frequency in x. The result will be the frequency of the last frequency with a value of span.
speaks(x,c1,c2)	This function was designed to walk the data and determine when you had strong signals by comparing the previous and next signals, looking for excursions from the lowest to highest signal of *c*. **NOTE: c2** is a bandwidth variable. If you are nesting this in another equation, you will need to populate this variable.
sqrt(x)	Determines the square root of each value in the data element *x*.
sub(x,y)	This function subtracts values in x by data in y in a sequential step without checking the frequency. The number of elements should be the same for x and y. If there are more elements in x than in y, this function will not execute. If there are more elements in y than in x, it will execute but there will not be any results for the extra elements.

<table>
<tr><td></td><td>For example, if there are five elements in x and four elements in y, there will be only four results.
If interpolation is on for the results element, it may populate the results with inaccurate data.</td></tr>
<tr><td>tan(x)</td><td>Determines the Tangent of each value in the data element x.</td></tr>
<tr><td>tanh(x)</td><td>Determines the Hyperbolic Tangent of each value in the data element x.</td></tr>
<tr><td>thd(x,c1,c2)</td><td>This function looks at the entire data element x starting with the frequency c1 with a bandwidth of c2.
Measurements based on amplitudes (e.g. voltage or current) must be converted to powers to make addition of harmonics distortion meaningful. For a voltage signal, the ratio of the squares of the rms voltages is equivalent to the power ratio:
$$THD = \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + \cdots + V_\infty^2}}{V_1}$$
An example would be a data element (discrete points) with harmonics of 1 MHz from 1 MHz to 5 MHz. Each harmonic is half the value the previous point.
THD_1MHz = thd(Data,1e6,100)
Where Data, list of frequencies:
1e6: Fundamental
100: Bandwidth in Hz
<table><tr><th>MHz</th><th>Test_Data</th></tr><tr><td>1.0</td><td>10</td></tr><tr><td>2.0</td><td>5</td></tr><tr><td>3.0</td><td>2.5</td></tr><tr><td>4.0</td><td>1.25</td></tr></table>The equation will return a value of 0.576 at 1 MHz.</td></tr>
<tr><td>top(x,c)</td><td>Determines the top for c number of points in the data array x. The default for c is 10 if it is not stated.</td></tr>
<tr><td>top10(x)</td><td>Determines the top 10 values in the array x.</td></tr>
<tr><td>trim(x)</td><td>There are certain times when a failure during a test will leave an array sized but not filled. The data elements that do not have valid data are empty (freq = 0.0f, amplitude = NAN). This function will remove these invalid data points.</td></tr>
<tr><td>toy(x)</td><td>Calculates a bin function for emissions data to Toyota specification.</td></tr>
<tr><td>Valuetofreq(x)</td><td>Returns an element where the frequency will equal the amplitude value. It can be used to find a certain harmonic of a frequency if used with the freq(x) function. This is the opposite of freq(x).</td></tr>
</table>

	Example: **Harm_2: valuetofreq(2*freq(Tx_Freq))** gets the 2nd harmonic of a frequency
vdb(x)	Converts the x value in dBV to a value in Volts using the equation **10^(x / 20)**. Same as the function **abd(x)**
vswr(f,r)	Calculates the VSWR ratio from a forward and reverse power. The first data element must be the forward power and the values must be higher than the reverse data element.
vtodbuv(x)	Converts the x value in Volts to a value in dBuV using the equation **log(x * 1E6) * 20**.
wdb(x)	Converts the x value in dBm to a value in Watts using the equation **10^((x / 10) / 1E3)**.

Native TILE! NI I/O Trace Support

This section provides detailed information regarding the native NI I/O Trace utility support added in TILE! 7.2.3.1. When asking for assistance from the TILE! support team, they will ask to see this log with the error.



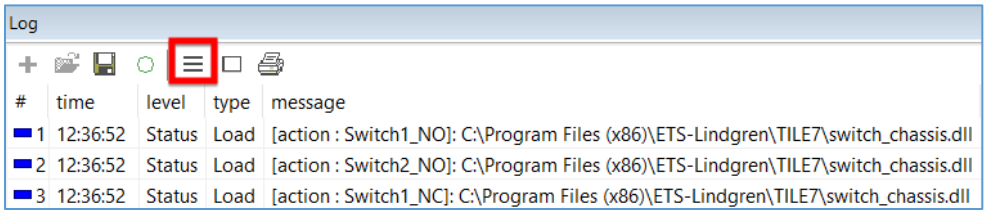
Although standard TILE! logging is sufficient for many applications, using an external logging utility such as NI I/O Trace can provide numerous advantages:

- TILE! menu Integration provides direct control of utility and access to key feature settings.
- TILE! can log messages directly into NI I/O Trace utility.
- It logs all I/O related log events (GPIB, VISA, IVI, and DAQmx) independently of TILE!, including detailed error information and buffer contents.
- NI I/O Trace is a separate application, so logging will continue even if TILE! stops or otherwise becomes locked. It produces a log file independent and separate from the TILE! profile, in various available data formats.
- It provides ms resolution timestamping (vs. sec resolution available in TILE!).
- It supports full search capability in the native environment (without having to export to file first).
- Auto Logging option for 1-step enabling of customer-support log-generation.
- Scripting support.

Log Options Dialog

Use the Log Options Dialog to enable TILE! log messages to the NI I/O Trace utility and to access settings specific to using NI I/O Trace utility with TILE!.

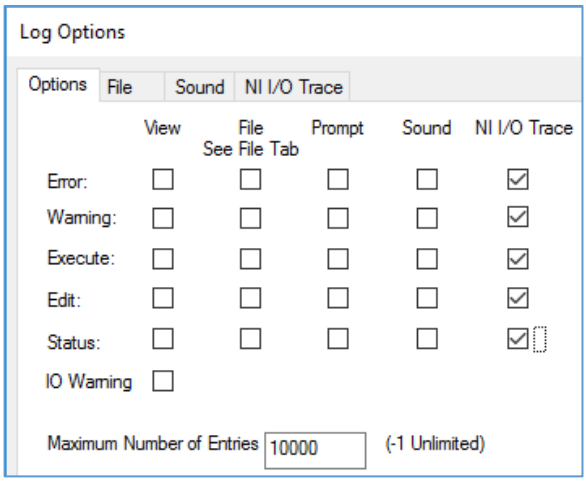
To access the Log Options, open the *Log window* and click the 3 horizontal lines icon.



Options Tab

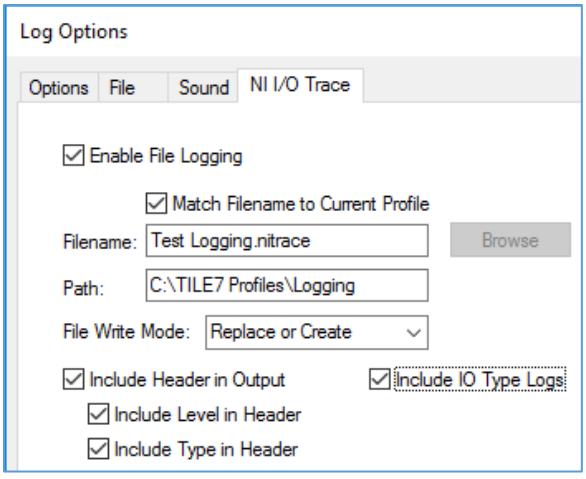
The NI I/O Trace column allows customization of the Log Levels that are sent to the NI I/O Trace utility. Just as each Log Level (Error, Warning, Execute, Edit, and Status) can be configured for the standard TILE! logging options (View, File, Prompt, and Sound), they can also be configured for the NI I/O Trace utility.

NOTE: A shortcut option to enable or clear all five levels at once is available by checking/unchecking the **Enable TILE! Logging** setting in the NI I/O Trace menu.



NI I/O Trace Tab

Settings specific to NI I/O Trace are configured on this tab.



- **Enable File Logging** — Enables the File Logging mode in NI I/O Trace. When a capture is started from TILE! (NI I/O Trace->Start Capture menu), the specified file info is sent to NI I/O Trace to set the log file name.
- **Match Filename to Current Profile** — Checking this box sets the Filename input to the current profile title with a .nitrace extension. The Path input is set to the path of the current profile.
- **Browse** — When Enable File Logging is checked, press Browse to use the File Open dialog to specify the name and path of the log file.

- **Filename** — When Enable File Logging is checked, specifies the filename of the NI I/O Trace log file. The supported file extensions are nitrace, TXT, CSV, and XML.
- **Path** — When Enable File Logging is checked, specifies the path of the NI I/O Trace log file.
- **File Write Mode** — When Enable File Logging is checked, specifies the Write Mode for file logging when a capture is started. The following selections are supported:
    - **Create** — This mode will only create a log file using the specified file name and path if it does not already exist. If the file does exist, the capture will not start.
    - **Append or Create** — This mode will create a new log file if it does not already exist, or append to an existing log file if it does exist.
    - **Replace or Create** — This mode will create a new log file if it does not already exist, or replace an existing log file if it does exist. This mode will overwrite any previous captures that are present in the existing file.
- **Include Header in Output** — Includes a TILE! header as a prefix to TILE! message logging. The full Header has the default format:

  {TILE:Level,Type}

    - **Include Level in Header** — If Include Header in Output is checked, includes the Level designation in the Header string. Unchecking this setting would shorten the Header string to the format:

      {TILE:Type}

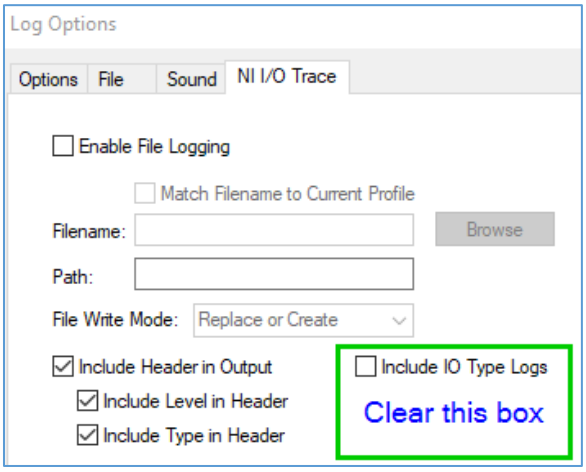
    - **Include Type in Header** — If Include Header in Output is checked, includes the Type designation in the Header string. Unchecking this setting would shorten the Header string to the format:

      {TILE:Level}

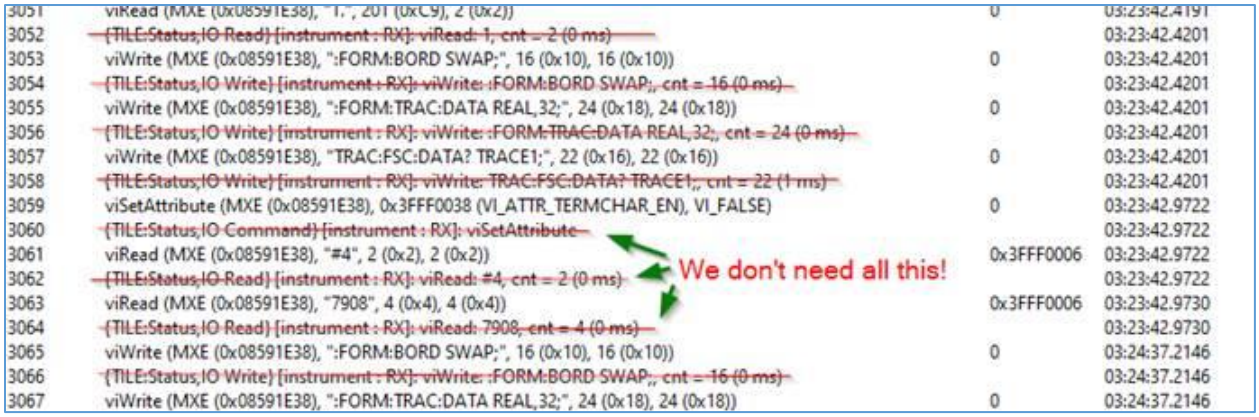
    - **NOTE**: If both **Include Level in Header** and **Include Type in Header** are unchecked, the Header format is:

      {TILE}

- **Include I/O Type Logs** — Checking this sends I/O Type Logs to NI I/O Trace. Uncheck to filter these log messages out. This is a useful feature because the TILE! I/O Type logs (GPIB and VISA writes and reads) are typically redundant to the I/O logging that is natively performed in NI I/O Trace.



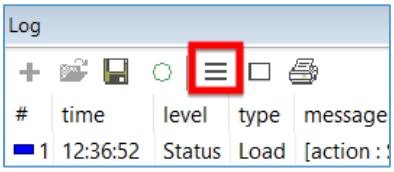
    - **NOTE**: When TILE! sends over its own IO log entries, this is by definition redundant information, makes the logs twice as long (as see in example below), and makes them harder to see what is really going on. The default for new profiles is to have that setting cleared, but if you are using an older profile it may be set and you need to clear it.



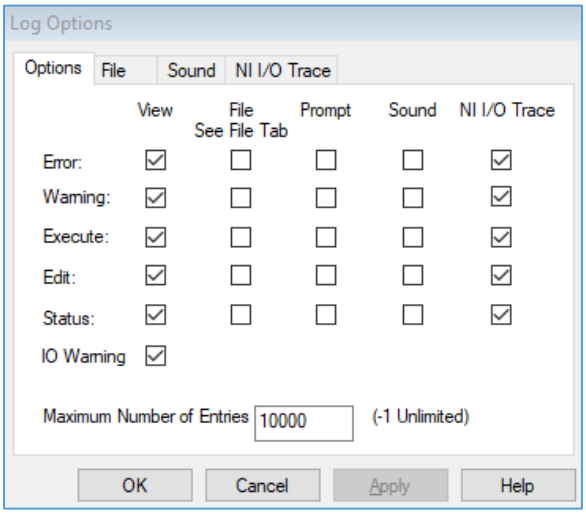
Detailed trace settings

This will create a detailed I/O trace with TILE! commands included so that we can see the data capture related to the SA readings.

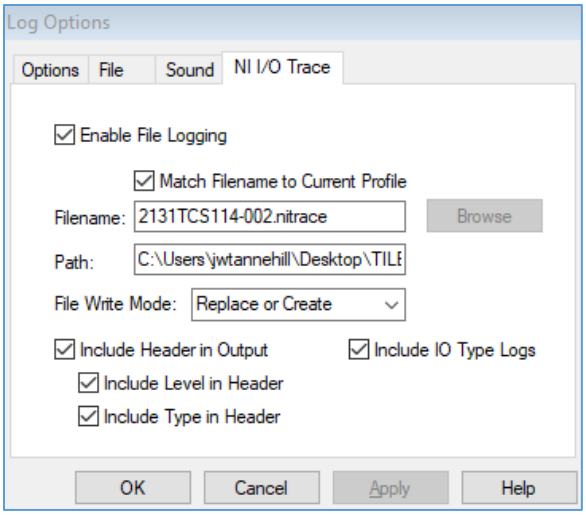
1. In the Log view, there is a button with 3 horizontal lines, select it to open the log options.



2. Select all the boxes in the view column and all the boxes in the NI I/O Trace column.



3. Go to the NI I/O Trace tab and select all the boxes.



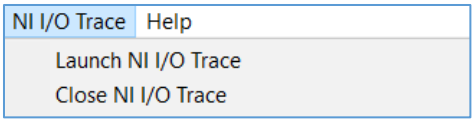
NI I/O Trace Scripting

Many of the NI I/O Trace menu options are accessible via TILE! scripting functions. Refer to the *TILE Scripting and Math Guide* for parameters and syntax.

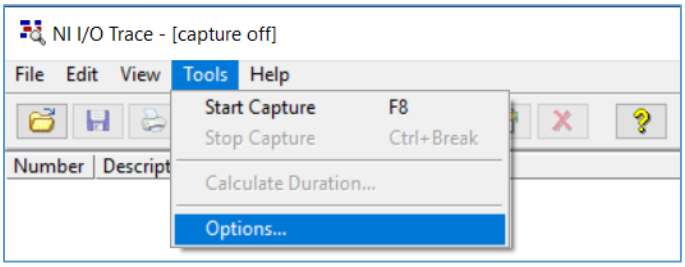
- **niiotrace_launch** — Launches NI I/O Trace utility. This is equivalent to selecting Launch NI I/O Trace from the NI I/O Trace menu.
- **niiotrace_close** — Launches NI I/O Trace utility. This is equivalent to selecting Close NI I/O Trace from the NI I/O Trace menu.
- **niiotrace_start_capture** — Start capturing in NI I/O Trace utility. This is equivalent to selecting Start Capture from the NI I/O Trace menu.
- **niiotrace_stop_capture** — Stop capturing in NI I/O Trace utility. . This is equivalent to selecting Stop Capture from the NI I/O Trace menu.
- **niiotrace_enable_tile_logging** — Enable/disable TILE! logging to NI I/O Trace utility. This is equivalent to checking/unchecking Enable TILE! Logging in the NI I/O Trace menu.
- **niiotrace_enable_match_filename** — Sets or clears the Enable File Logging and Match Filename to Current Profile checkboxes on the NI I/O Trace tab of the Log Options dialog. This is equivalent to checking/unchecking Enable Log to Matched File in the NI I/O Trace menu.
- **log_status** — If TILE! NI I/O Trace Status level logging is enabled, will send a log message string directly to NI I/O Trace utility.
- **log_error** — If TILE! NI I/O Trace Error level logging is enabled, will send a log message string directly to NI I/O Trace utility.

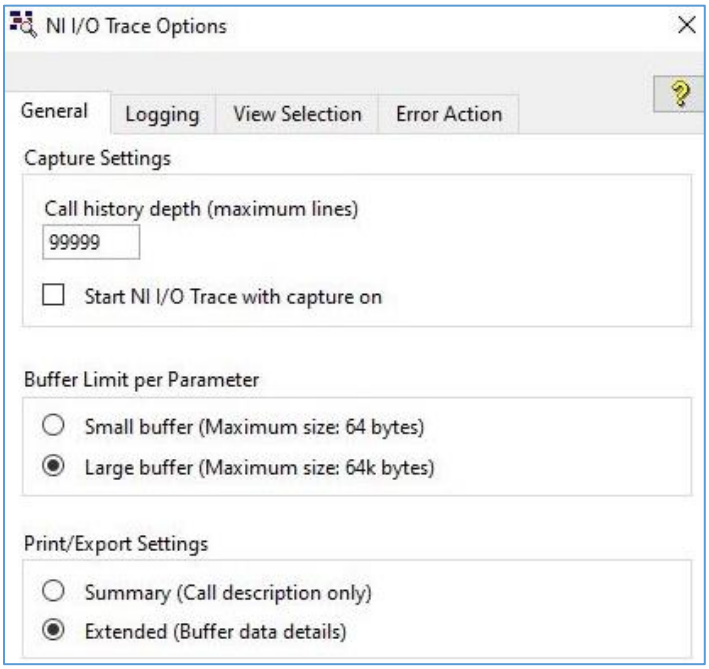
Using NI I/O Trace

1. Go to the main menu toolbar **NI I/O Trace** and select **Launch I/O Trace**.



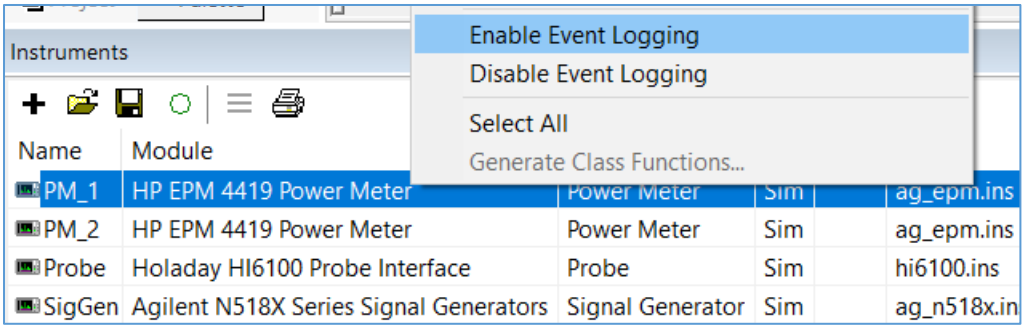
a. This will open the **NI I/O Trace Utility**. Go to the menu **Tools** and the **Options** to see the following window:



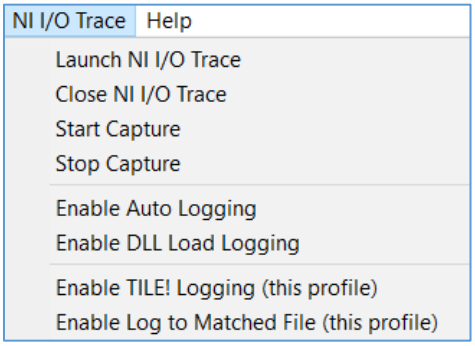


i. Set the **Call History Depth** value to the maximum allowed value of **99999**.
ii. Set the **Buffer Limit** to **Larger buffer**.
iii. Set the **Export Settings** to **Extended**.

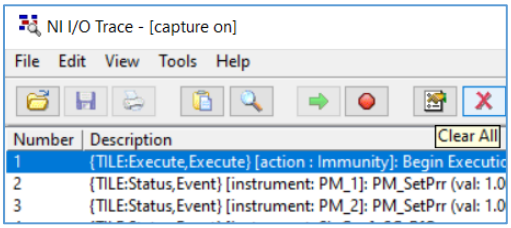
2. Back in the TILE! profile, right-click the instrument you want to see in detail and select **Enable Event Logging**.



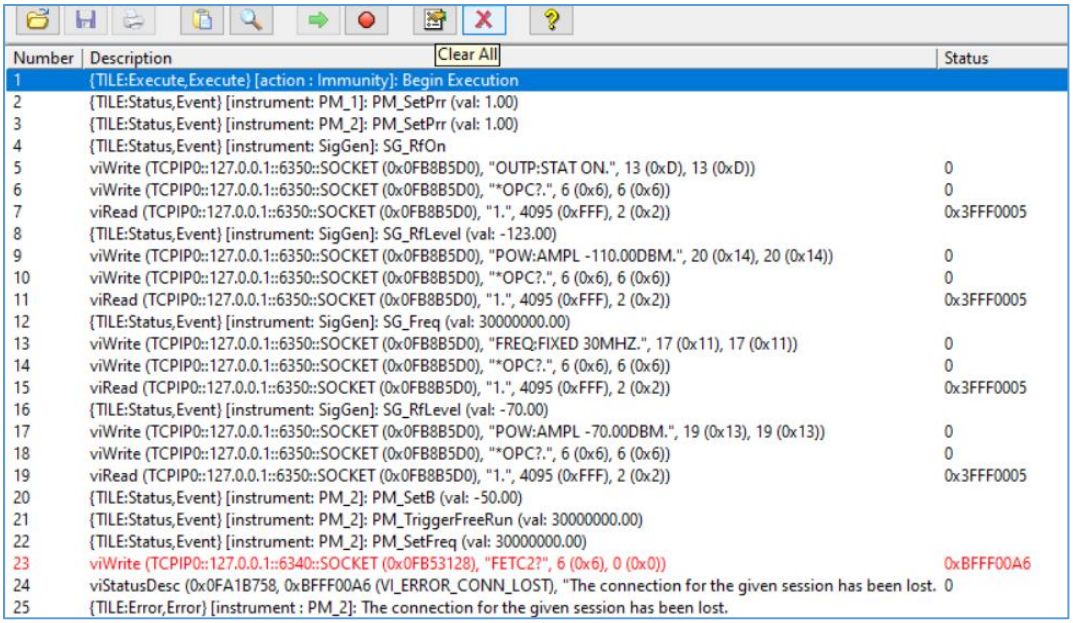
a. Go to the main menu toolbar **NI I/O Trace** and select **Enable TILE! Logging (this profile)**.



3. Back in the **NI IO Trace Utility**, clear the log using the red X button and start a new capture with the green arrow button.



4. Next in the TILE! profile, run the test (including instrument initialization steps) until it's past the issue in question.
   a. Stop the capture once you see the issue (appears in yellow or red) and save both the NI trace and the profile.



Number	Description	Status
1	{TILE:Execute,Execute} [action : Immunity]: Begin Execution	
2	{TILE:Status,Event} [instrument: PM_1]: PM_SetPrr (val: 1.00)	
3	{TILE:Status,Event} [instrument: PM_2]: PM_SetPrr (val: 1.00)	
4	{TILE:Status,Event} [instrument: SigGen]: SG_RfOn	
5	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "OUTP:STAT ON.", 13 (0xD), 13 (0xD))	0
6	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "*OPC?.", 6 (0x6), 6 (0x6))	0
7	viRead (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "1.", 4095 (0xFFF), 2 (0x2))	0x3FFF0005
8	{TILE:Status,Event} [instrument: SigGen]: SG_RfLevel (val: -123.00)	
9	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "POW:AMPL -110.00DBM.", 20 (0x14), 20 (0x14))	0
10	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "*OPC?.", 6 (0x6), 6 (0x6))	0
11	viRead (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "1.", 4095 (0xFFF), 2 (0x2))	0x3FFF0005
12	{TILE:Status,Event} [instrument: SigGen]: SG_Freq (val: 30000000.00)	
13	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "FREQ:FIXED 30MHZ.", 17 (0x11), 17 (0x11))	0
14	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "*OPC?.", 6 (0x6), 6 (0x6))	0
15	viRead (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "1.", 4095 (0xFFF), 2 (0x2))	0x3FFF0005
16	{TILE:Status,Event} [instrument: SigGen]: SG_RfLevel (val: -70.00)	
17	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "POW:AMPL -70.00DBM.", 19 (0x13), 19 (0x13))	0
18	viWrite (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "*OPC?.", 6 (0x6), 6 (0x6))	0
19	viRead (TCPIP0::127.0.0.1::6350::SOCKET (0x0FB8B5D0), "1.", 4095 (0xFFF), 2 (0x2))	0x3FFF0005
20	{TILE:Status,Event} [instrument: PM_2]: PM_SetB (val: -50.00)	
21	{TILE:Status,Event} [instrument: PM_2]: PM_TriggerFreeRun (val: 30000000.00)	
22	{TILE:Status,Event} [instrument: PM_2]: PM_SetFreq (val: 30000000.00)	
23	viWrite (TCPIP0::127.0.0.1::6340::SOCKET (0x0FB53128), "FETC2?", 6 (0x6), 0 (0x0))	0xBFFF00A6
24	viStatusDesc (0x0FA1B758, 0xBFFF00A6 (VI_ERROR_CONN_LOST), "The connection for the given session has been lost.	0
25	{TILE:Error,Error} [instrument : PM_2]: The connection for the given session has been lost.	

b. Go to the toolbar menu **Help** and then to the **About** page to get the version you are running.

   i. Use the links on the About page to contact the TILE! support team through the email TileSupport@ets-lindgren.com or by submitting an SR on the portal at https://support.ets-lindgren.com/Tile

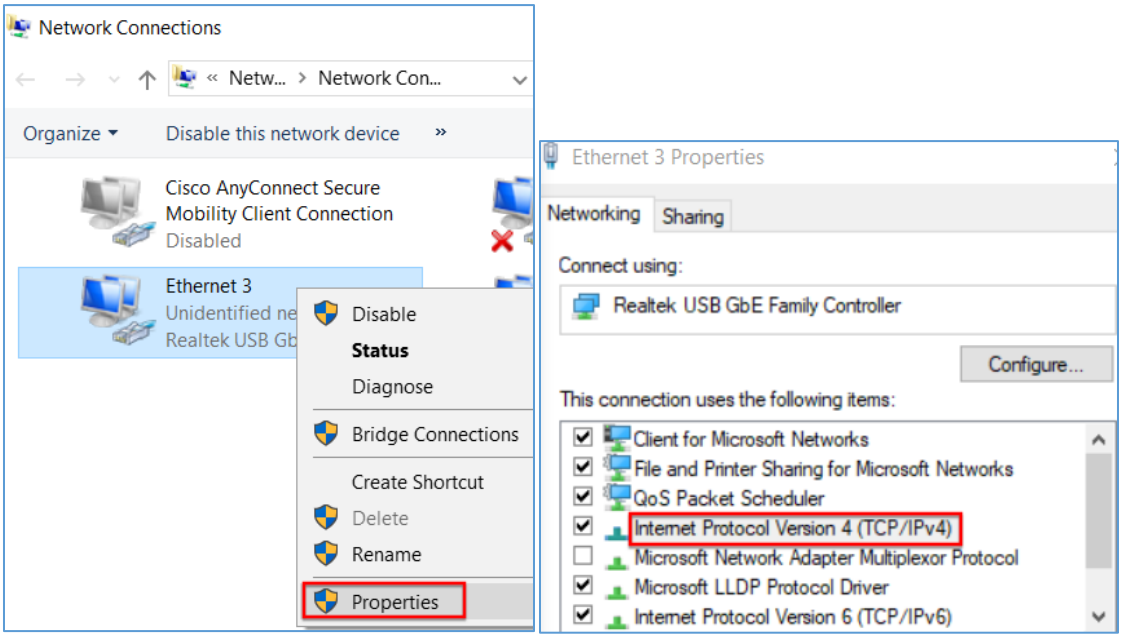
   ii. Send your full name and contact info, version number, the updated saved profile, and the NI I/O Trace log file.

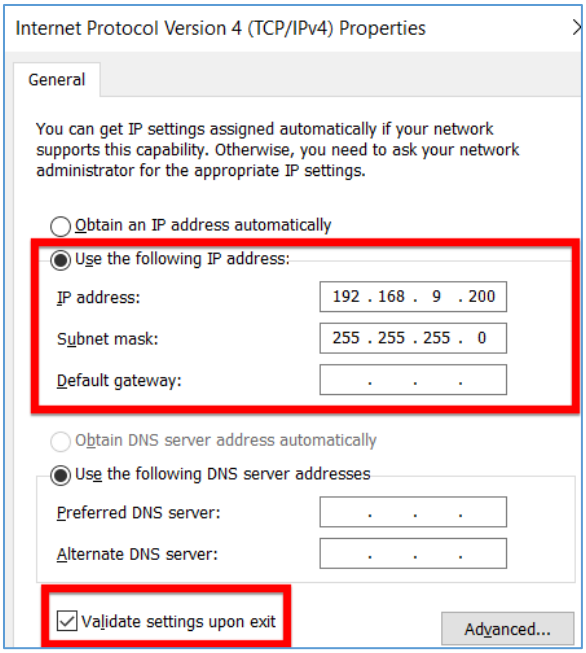
Appendix: Miscellaneous Information

How to Configure the Ethernet IP address for an instrument

The following example shows how to configure the IP address for an EMCenter to communicate with the computer.

1. Go to ***Control Panel*** -> ***Network and Sharing Center*** -> ***Change adapter settings***.
2. Use the Ethernet cord to connect the EMCenter to the computer, and the new Ethernet connection should appear.
3. Right click on the Ethernet connection that appears and select ***Properties***, then Double-click ***Internet Protocol Version 4***.





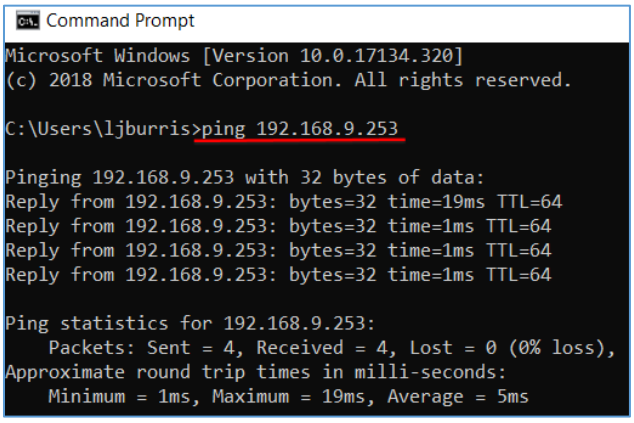
4. Set the ***IP address*** to be similar to the IP address on the EMCenter Config page, but the last numbers must be different or the connection will not work.

   For example if the EMCenter IP address is 192.168.9.**253,** then the computer's Ethernet IP address should reflect that and could be set to 198.168.9.**200**.

5. Set the ***Subnet mask*** to be the same as on the EMCenter (255.255.255.0).

6. Select the ***Validate settings upon exit*** checkbox and click ***OK***.

7. Open the ***Command Prompt*** and ***ping*** the Ethernet IP address to check the connection.



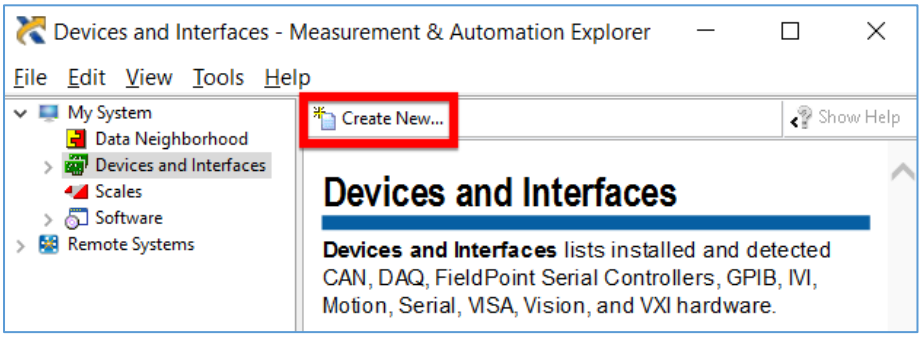
Create new NI MAX VISA resource for TILE!

In order to use a network-enabled (LAN) instrument in TILE!, the easiest way to access it is to first configure it as a Network Device in NI MAX. Follow these basic instructions to configure the device and refer to NI documentation or your instrument's documentation for the specific requirements of the device.

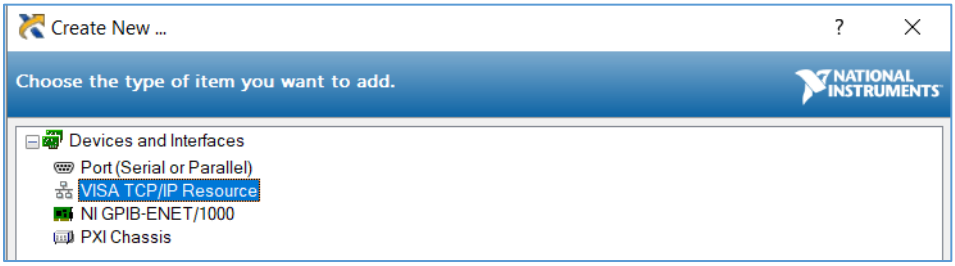
NOTE: Images in this example are taken from NI MAX version 18.5 and TILE! version 7.4.3.2. Other versions may appear slightly different, but the general process should be similar.

1. Run NI MAX (Start Menu->All Programs->National Instruments->NI MAX)

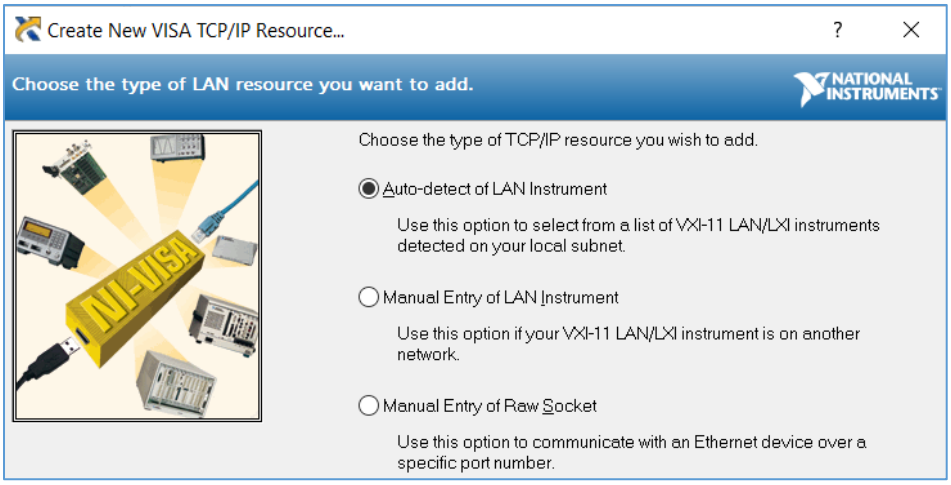
   a. Select **Devices and Interfaces**, then press the **Create New…** button in the right pane (or right-click on Devices and Interfaces and select Create New…):



   b. Select VISA TCP/IP Resource as the type of item to add:



   c. Choose LAN resource:



      i. If the device supports the VXI-11 (LXI) protocol, you can auto-detect the instrument if it is active on the local network.

      ii. If the device is NOT VXI-11 compliant, is not active on the network and/or does not auto-detect on the system, then you will need to use one of the “Manual Entry” options.

NOTE: Refer to your instrument documentation and NI-MAX help for more details on manual configuration of a network device. This may be as simple as knowing the IP address and/or socket of the instrument you are trying to connect to.

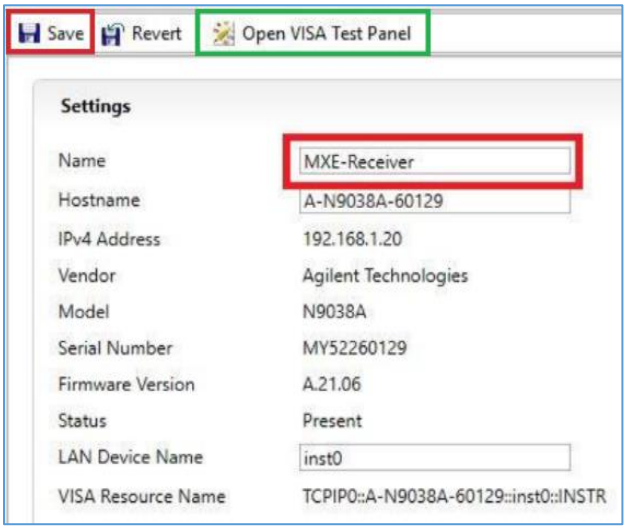
Details of network configuration can be complex and are beyond the scope of this section. Make sure your network-enabled instrument is configured properly and accessible to the computer you are attempting to communicate with via MAX and TILE!.

d. Once your device is properly configured in MAX it will be listed in the Network Devices list, along with the VISA string you will use to communicate with it in TILE!:

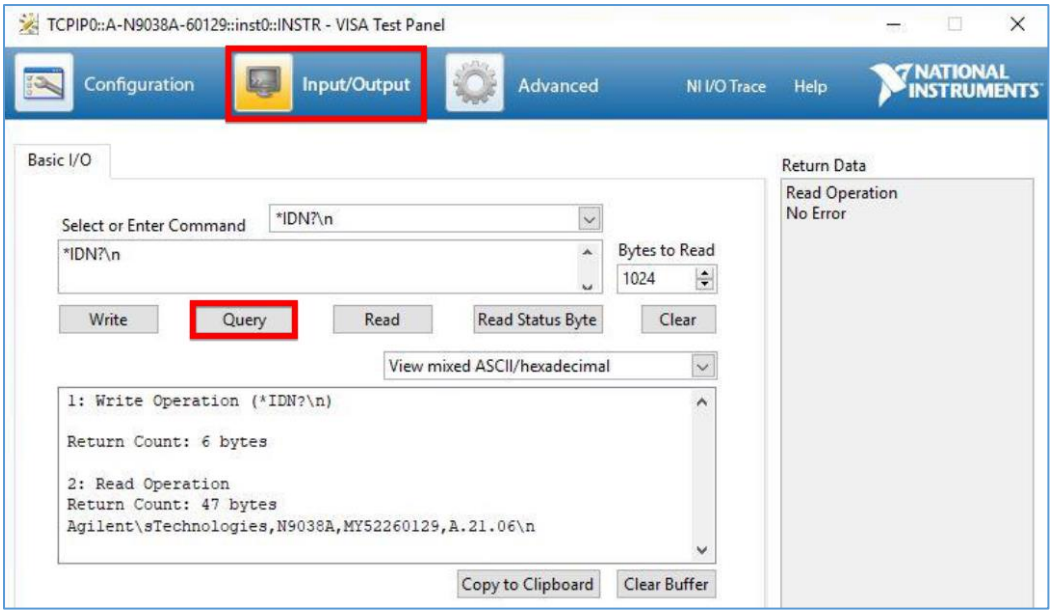
e. OPTIONAL, you can create a VISA Alias name for this device that will make it easier to refer to in TILE!.

   i. Click the General tab on the bottom to enter a VISA Alias name for the device, such as “MXE-Receiver”.

   ii. Make sure to press the Save button after creating a VISA Alias name or making any changes to your device.



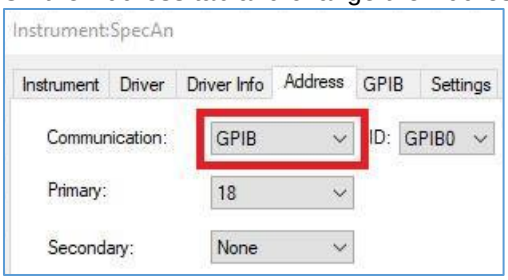
f. To test your connection select “Open Visa Test Panel” above your instrument on the settings page, in the Test Panel, select “Input/Output” and Query the instrument.



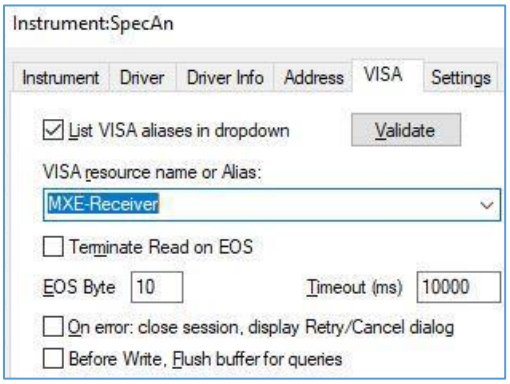
2. Run TILE!:

   a. Add a new instrument in the Instrument View and name it.

   b. On the Address tab and change the Address Communication type to VISA:



   c. On the VISA tab, change the “VISA Resource Name or Alias” to the network device you just configured in NI MAX … “MXE-Receiver”.



      i. Either click the drop-down arrow and view a list of all available VISA instruments on the system.

      ii. Alternatively, copy and paste a VISA resource name from MAX directly into this box. For example, “TCPIP0::A-NP038A-60129::inst0::INSTR”.

   d. Once the VISA resource name has been populated, validate the VISA connection to the instrument by pressing the Validate button. A popup message will appear indicating success or failure.

      NOTE: If an error occurs, it is a good idea to return to MAX to validate proper configuration of the device. If MAX cannot successfully communicate to the device, then TILE! will not be able to either.

e. **NOTE**: Unlike with GPIB/Serial instruments, TILE! will attempt to open VISA sessions (and communicate) to all VISA-configured instruments in a profile immediately upon loading the profile.

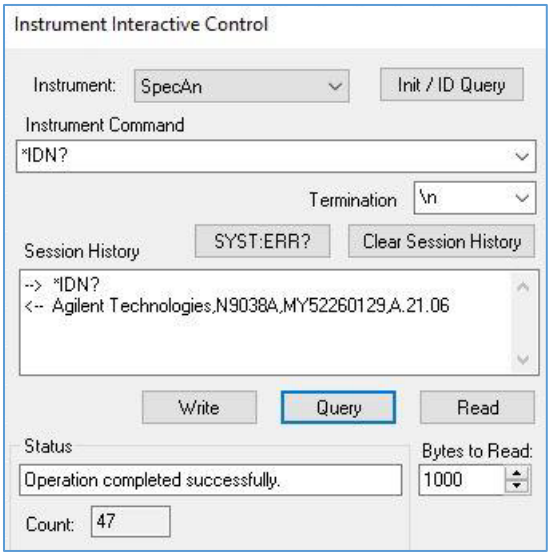
   i. This means that any VISA-configured instruments listed in a profile that are not accessible when the profile is opened will generate warning prompts similar to the one shown above.

   ii. However, this does not prevent the profile from opening. If the instruments become active after the profile is opened, they will then be accessible by the profile at that time.

3. Once the VISA device has been successfully configured, you should be able to communicate to it with any Action or utility in TILE! as if it were a standard GPIB or Serial device.

   a. A useful tool to verify basic device communication is the Instrument Interactive Control utility, accessible via the toolbar button showing the hand and wand icon.

   b. Sending the *IDN? command to most instruments will return an ID string and verify basic device communication is functional:



4. Finally, it should be pointed out that TILE! can access network-enabled VISA devices without first configuring them in NI MAX. It is only necessary to know the correct VISA resource name that will communicate with the device, and entering that resource name in the TILE! VISA resource name input on the instrument's Address tab.

   There are numerous benefits of using a VISA configuration utility such as MAX, however:

   a. Instruments are configured and verified independently of TILE!.

   b. Auto-detect feature of MAX may be able to detect and configure available instruments automatically.

c. VISA resource names are produced in MAX using only network-specific information such as IP address and socket number. The specific VISA syntax is generated by the utility.

d. VISA resource names and aliases created in MAX become automatically available in the TILE! VISA resource name dropdown list. Devices that have not been “pre-configured” in MAX will not appear in the dropdown list.

TILE! Simulation Utility

The TILE! Simulation Utility is installed with TILE! by the TILE! software installer. The Simulation Utility requires a separate license from the TILE! application software and can be obtained (or renewed) by contacting ETS-Lindgren at TileSupport@ets-lindgren.com.

NOTE: In addition to the TILE! software installer, you also need to install the LabVIEW 2018 Runtime Engine installer if it is not already present on the system. This installation can be downloaded from support.ets-lindgren.com.

For more information on how Simulation works, see the Simulation Readme file under *C:\Program Files (x86)\ETS-Lindgren\TILE7\Simulation*.

Simulation Overview

Simulated instruments in TILE! are referred to as Simulation Servers. A Simulation Server instrument panel responds to TILE! IO commands much in the same way that a physical instrument would. A Simulation Server waits to be addressed and communicated with by a TILE! instrument driver, and then if it recognizes the command it takes appropriate action.

Depending on the type of Simulation Server running, it might change an internal state or value, take a simulated measurement, or perform some other action, based on the command received. A SigGen Server, for instance, will store a level value received from TILE! as an internal variable and pass this value to input devices (such as PwrMtr Servers and SA Servers) that are synchronized to that particular SigGen.

A key feature of the Simulation Server instruments is this ability to synchronize a SigGen instrument to a PwrMtr or Analyzer instrument using level and frequency information. This allows multiple Simulation Server instruments running in parallel to model a real-world system.

The SigGen server stores level and frequency data received from TILE! (This simulates outputting a signal into a load). Ensuing read commands sent to PwrMtr servers will retrieve this value, scaled by the simulator, possibly with noise added, and return the calculated value back to TILE!.

Combining simulators in this way makes it possible to test actions and drivers in TILE! that involve leveling feedback loops without having physical instruments present. There is “live” data passing between various simulated instruments such that an actual feedback loop can occur in the TILE! software actions.

General Operation

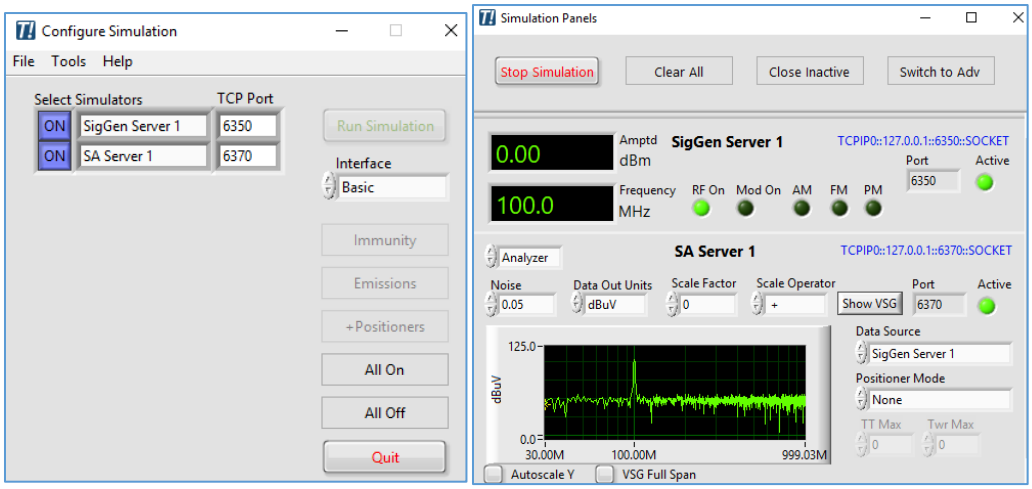
The TILE! Simulation Utility can be started from the TILE! Tools->Run Simulation menu. This will launch the simulation interface panel, Configure Simulation, which will select and run any combination of available Simulation Server panels. Alternatively, TILE! will launch the Simulation Utility automatically when loading a profile with instruments pre-configured for Simulation. There are numerous TILE! example

profiles that ship with instruments pre-configured to use Simulation. These profiles can be found in the [TILE]\Profiles\Simulation directory, where [TILE] is the TILE! installation directory.

Methods to connect TILE! instruments to the Simulation Utility

Method 1: Open a profile pre-configured with simulated instruments.

Several simulation example profiles ship with TILE! in the \Profiles\Simulation directory. TILE! will automatically launch the Simulation Utility when it loads an instrument pre-configured to use simulation (Communication value set to Simulation in any instrument). When run in this manner, the Configure Simulation panel will disable all instruments except for the instruments in the profile that are configured to use simulation.

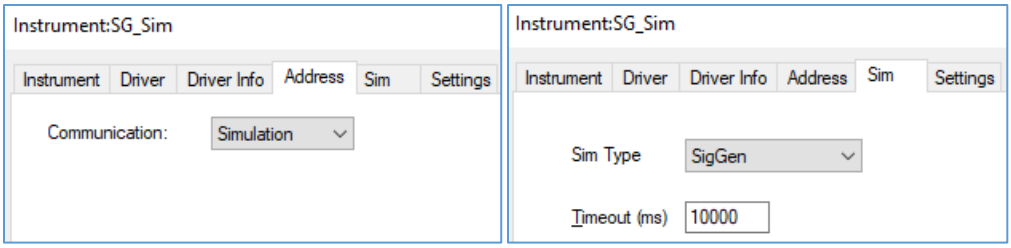


Configure Simulation and Simulation Panels when Synchronized to a TILE! Profile

Method 2: Configure a new instrument with Communication set to Simulation.

To use this method, follow these steps:

1. Add a new instrument to the profile's Instrument list.
2. Switch to the Address tab and select Simulation in the Communication dropdown list.
3. Switch to the Sim tab and select the instrument type (Power Meter, Signal Generator, etc.).
4. Click OK.
5. TILE! will automatically select a supported driver for the instrument class you selected, launch the Simulation Utility, add the appropriate Simulation Server into the Configure Simulation list, and complete the connection between the TILE! instrument and the Simulation Server.



Address and Sim tabs for configuring a Simulated Instrument

Method 3: Use the TILE! Tools->Run Simulation menu option.

Using this option will “disconnect” some of the automation that occurs when using Methods 1 or 2, and thus allow currently unused instruments to remain in the available instrument list even if the profile that used them is closed.

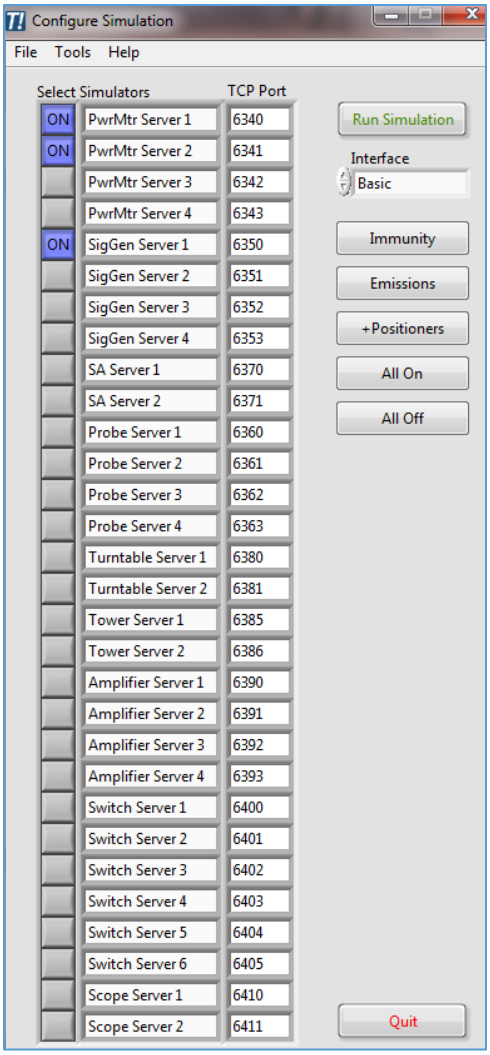
NOTE: If Method 3 is used after the Simulation Utility has already been launched using either Methods 1 or 2, then the Configure Simulation panel may be cleared of all instruments.

To return to the default instrument list, you can select File->Load Settings from the Configure Simulation menu and then select SimulationSettings_default.ini, or you can use the Tools->Options menu and press the Edit Simulator List button, then press Enable All, and OK twice to exit back to the main panel.

You can select Tools->Run Simulation at any time (even while Simulation is currently running) to turn off the clearing of unused instruments feature that occurs when using Methods 1 or 2.

Simulation Server List

The available Simulation Server list indicates the maximum number of each simulation type that can be used. For instance, a TILE! profile can have up to four simulated Power Meters, two simulated Turntables, etc.



Configure Simulation Panel with All Instruments Enabled

The available Simulation Server list is:

1. PwrMtr Server 1
2. PwrMtr Server 2
3. PwrMtr Server 3
4. PwrMtr Server 4
5. SigGen Server 1
6. SigGen Server 2
7. SigGen Server 3
8. SigGen Server 4
9. SA Server 1
10. SA Server 2
11. Probe Server 1
12. Probe Server 2
13. Probe Server 3
14. Probe Server 4
15. Turntable Server 1
16. Turntable Server 2
17. Tower Server 1
18. Tower Server 2
19. Amplifier Server 1
20. Amplifier Server 2
21. Amplifier Server 3
22. Amplifier Server 4
23. Switch Server 1
24. Switch Server 2
25. Switch Server 3
26. Switch Server 4
27. Switch Server 5
28. Switch Server 6
29. Scope Server 1
30. Scope Server 2

Instrument Shortcut Buttons

Specific server combinations can be pre-saved and chosen using the Configure Simulation shortcut buttons:

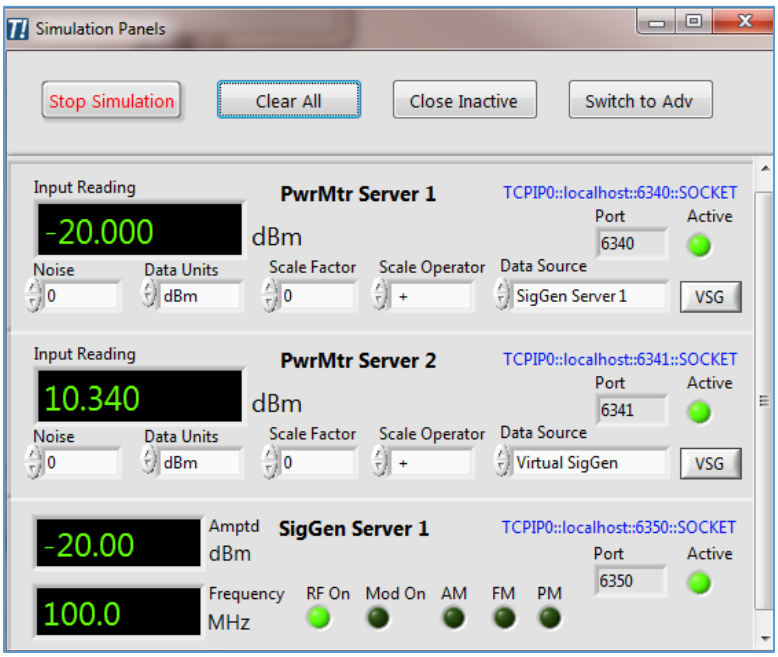
1. Immunity = typically selects PwrMtr Server 1, PwrMtr Server 2, and SigGen Server 1.
2. Emissions = typically selects SigGen Server 1 and SA Server 1.
3. +Positioners = toggles Turntable Server 1 and Tower Server 1 on or off
4. All On = turn on all available servers.
5. All Off = turns off all servers.

NOTE: The Immunity, Emissions, and +Positioners buttons are customizable and may become modified or not as useful when Simulation Utility is synchronized to a TILE! profile. To edit the instruments selected by each button, go to the Configure Simulation Tools->Options menu and press the appropriate Edit Button List button.

Basic and Advanced Interface Modes

There are two interface modes available for each Simulation Server instrument panel: Basic and Advanced. You can press the “Switch to Adv” or “Switch to Basic” button on the Simulation Panels to switch all currently running simulators to the specified mode.

You can also switch an individual panel to a different interface mode by turning it off in the Configure Simulation panel, changing the Interface control, and then re-starting the simulator. Basic interface mode should be used for general use. This interface provides instrument-like front panels for each server, while hiding the low-level details and advanced features.

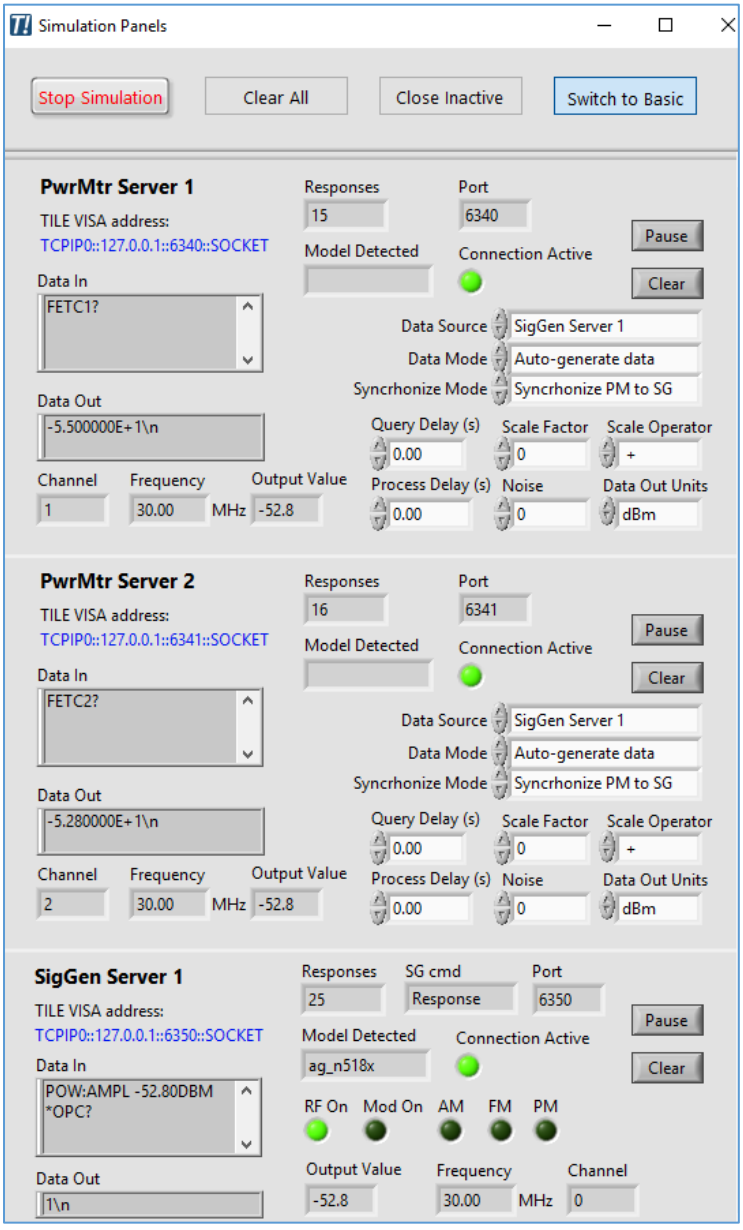


Simulation Servers in Basic Mode

Advanced interface mode provides more details and control of the simulator server's interaction with TILE!, such as a view of the commands received from the TILE! driver and the response strings being sent back to TILE! from the simulator.

Although the simulator server panel keeps no history of past commands, the advanced interface provides a real-time view of TILE!'s command interaction with the simulator, similar to the TILE! IO Log, NI IO Trace Utility, or another IO logging utility.

Advanced mode also provides the ability to pause and step through individual command responses, and in the case of the PwrMtr/Probe Servers, the ability to change Data Modes to "Pause for data prompt" which allows the user to modify the response value before the simulator sends the response back to TILE!.



Simulation Servers in Advanced Mode

Connection Active LED

If a server connection is dropped, the Active light on the server panel will turn off (such as when the profile is closed, or when a server is stopped and restarted). TILE! can usually re-establish a connection to a server when an action initiates communication with the server.

You can also manually establish a connection to a specific Simulation Server by opening the TILE! instrument configuration dialog and then simply click OK to close the dialog. You should see the Connection Active LED re-illuminate, just like when the original connection was being established.

Simulation Settings

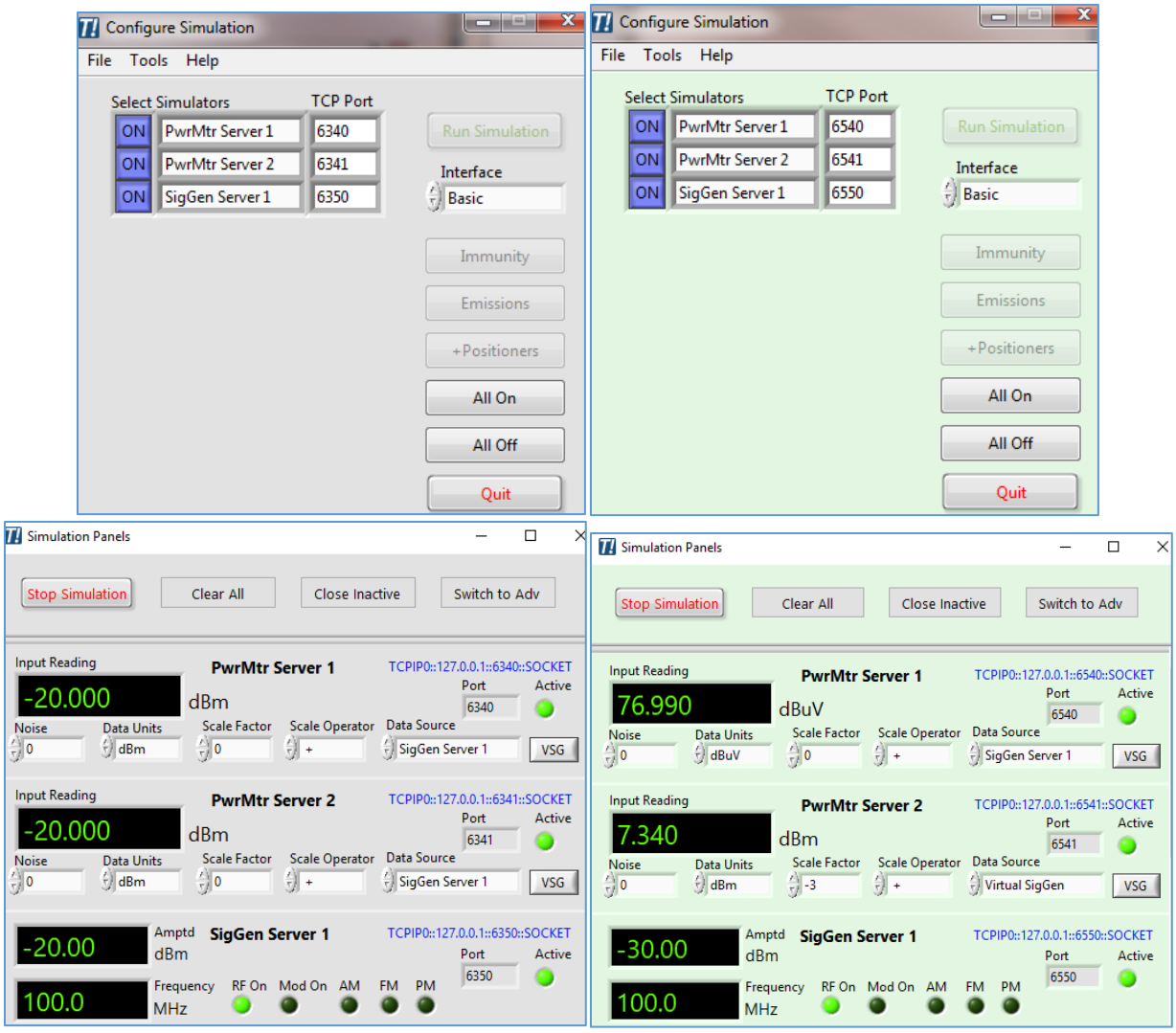
The Configure Simulation panel's settings and the last state of all running Simulation Servers is saved to disk and then reloaded each time the Simulation Utility is run. Thus, each simulator should re-launch in the same state as when it was last running.

Additionally, a snapshot of the current state of the simulator can be saved to disk by selecting Save Settings from the File menu of Configure Simulation. A previously saved settings file can be reloaded by selecting Load Settings from the File menu. This way, simulation settings files can be saved to match their corresponding TILE! profiles.

Running Multiple Simulator Instances

Multiple Configure Simulation windows can control independent simulation systems running at the same time. This allows two or more instances of TILE!, each with a profile of simulated instruments, to run tests concurrently. Duplicate simulation systems will run with different colored panels to identify each simulation system that is running.

In this diagram there are two Configure Simulation panels running simultaneously.



Multiple Simulation Systems

Simulation Panels description and operation

All Simulation Servers are run as “subpanels” in a single window called Simulation Panels. This window provides general control over all running servers:

1. Stop Simulation. Stops all running servers and closes the Simulation Panels window. Returns control to the Configure Simulation panel, where servers can be re-selected and re-started.
2. Clear All Servers. Clears indicators on all running servers. Useful for determining if a specific server is being actively communicated with by TILE!.
3. Close Inactive Servers. Closes all servers that do not currently have their Active light turned on.
4. Switch to Advanced (or Switch to Basic). This button toggles between the Basic and Advanced user interface modes of the server panels as described in a previous section. Most users will use the Basic interface mode. For debugging and low-level information and control of the servers, switch to the Advanced interface mode.

Note that the Configure Simulation window remains active while the simulators are running. Thus, simulators can be added or removed from Simulation Panels in any order, and at any time.

Basic Mode

1. Scale Factor and Scale Operator (PwrMtr/Probe/SA servers only).
    a. The combination of these inputs determines how the PwrMtr Server will scale its data before sending a response to TILE!.
    b. For example, if Scale Factor is 2, and Scale Operator is “+”, then 2 will be added to the measurement value(s).
2. Noise (PwrMtr /Probe/SA servers only).
    a. Specifies the noise level to add to the measurement value(s).
3. Data Out Units (PwrMtr /Probe/SA servers only).
    a. Specifies the units to convert the measurement value(s) into.
4. Data Source (PwrMtr /Probe/SA servers only).
    a. Specifies the data source to read.
    b. Options are:
        i. SigGen Server X.
            1. If SigGen Server 1 and/or SigGen Server 2 are active, these simulators will be listed in the Data Source dropdown of all input servers.
        ii. Virtual SigGen.
            1. Click the Show VSG button to view the Virtual Signal Generator. This runs a separate utility which can generate Sine, Square, or Triangle signals, control Amplitude and Frequency, and manipulate the time domain signal with window and filter settings. Although the VSG is always active, its output will only be synched to an input server when Virtual SigGen is selected as the Data Source.
5. Positioner Mode (SA Server only).
    a. Specifies which positioners, if any, will affect the output of the SA Server. The allowed values are None (default), Turntable, Tower, and Turntable + Tower. For whichever positioners (turntable and/or tower) are enabled, the corresponding Max input (TT Max or Twr Max) specifies the value of that positioner that will produce the maximum SA Server output.
    b. NOTE: SA Server 1 uses the Tower Server 1 position, and SA Server 2 uses the Tower Server 2 position.
    c. For example:
        i. Positioner Mode = Turntable + Tower
        ii. TT Max = 180
        iii. Twr Max = 200

d. This will produce the maximum SA Server output when the Turntable Server position is at 180 degrees, and the Tower Server position is at 200 cm. All other positions for each server will scale the SA Server amplitude spectrum down by a value dependent on the positioner's distance from its maximum value.

e. The scale factors are:

   i. Turntable = 10 dBm / 180 deg.

   ii. Tower = 10 dBm / 200 cm.

f. To continue the previous example, if the Turntable Server position is at 0 degrees and the Tower Server position is at 0 cm, the SA Server amplitude would be scaled down by 10 + 10 = 20 dBm.

g. If the Positioner Mode = Turntable, then only the Turntable Server position would affect the SA Server output, and the amplitude would be scaled down by 10 dBm, instead of 20 dBm.

6. Remote/Local (Positioner servers only).

   a. Local mode allows the Current Position gauge/slider to be controlled interactively from the simulation panel.

      i. Reading the current position in TILE! reflects the value set by the user.

      ii. Also, the SA Servers will respond appropriately to Local mode Positioner changes when they are set to an appropriate Positioner Mode.

   b. Remote mode gives control of the Positioner Server back to TILE!.

Advanced Mode

1. Pause button.

   a. Pauses the server in an active connection.

   b. This allows viewing of specific commands as they are received (Note: TILE! will continue running if a server is paused, and may generate timeout errors if the simulator is paused and not responding).

2. Clear button.

   a. Resets all server panel indicators to default values.

   b. This is useful for determining if the server is currently receiving commands (in particular, Responses will show 0).

3. Data Mode (PwrMtr/Probe servers only).

   a. Select "Auto-generate data" or "Pause for data prompt".

   b. Auto-generate data mode causes the PwrMtr server to generate a value (based on Synchronize Mode selection) and send it immediately back to TILE!.

   c. Pause for data prompt mode will pop-up a dialog box during each ReadPt function and allow the user to modify the simulator's data value before sending it back to TILE!.

   d. Note: when the PwrMtr simulator is paused, the TILE! instrument's VISA timeout setting comes into play. The default VISA timeout is 10000ms (10s).

i. If the user waits longer than 10s before pressing Continue, the VISA Read will timeout and TILE! will generate an error.

ii. If a longer time is required, the VISA timeout setting can be increased on the VISA tab of the instrument configuration.

4. Synchronize Mode (PwrMtr/Probe servers only).

    a. Select “Synchronize PM to SG” to simulate a real-world system, where the last value of the SigGen Server is used by the PwrMtr/Probe Server to generate its next response.

    b. The next response is adjusted by the Scale Factor input, so that the next value = last SigGen output value + Scale Factor.

    c. If Synchronize Mode = Generate random data, the simulator generates a random value between 0 and 1 for the next data value to return to TILE!.

5. Query and Process delay values (PwrMtr /Probe servers only). These are helpful and sometimes necessary delays, depending on the application and instrument being simulated:

    a. Query Delay

        i. a delay specific to functions that return data (such as ReadPt), added after a query command has been received/detected, and before the ensuing TCP Write call is made to return data back to TILE!. This may be helpful to simulate actual instrument delay times.

    b. Process Delay

        i. A delay added after any new characters have been received by TCP Read, but BEFORE the command is processed. This delay was found to be necessary to simulate instruments that have no Query command on ReadPt.

        ii. Instead, the PwrSetA command is used to trigger the simulator’s ReadPt response, and possible race conditions could occur in synchronizing to the SigGen in this scenario.

        iii. The only instrument (so far) tested that needs this delay (50 ms) is the Btn4200.

6. Scale Factor and Scale Operator (PwrMtr/Probe/SA servers only).

    a. The combination of these inputs determines how the server will scale its data before formatting the Data Out string.

    b. For example, if Scale Factor is 2 and Scale Operator is “+”, then 2 will be added to the measurement value(s).

7. Noise (PwrMtr /Probe/SA servers only).

    a. Specifies the noise level to add to the PwrMtr/Probe/SA response before sending the output to TILE!.

8. Data Out Units (PwrMtr /Probe/SA servers only).

    a. Specifies the units to convert the PwrMtr /Probe/SA response into before sending the output to TILE!.

9. Data In indicator.

    a. Displays the most recent IO received from TILE! by the simulation server.

    b. Multiple commands may be included in each received buffer.

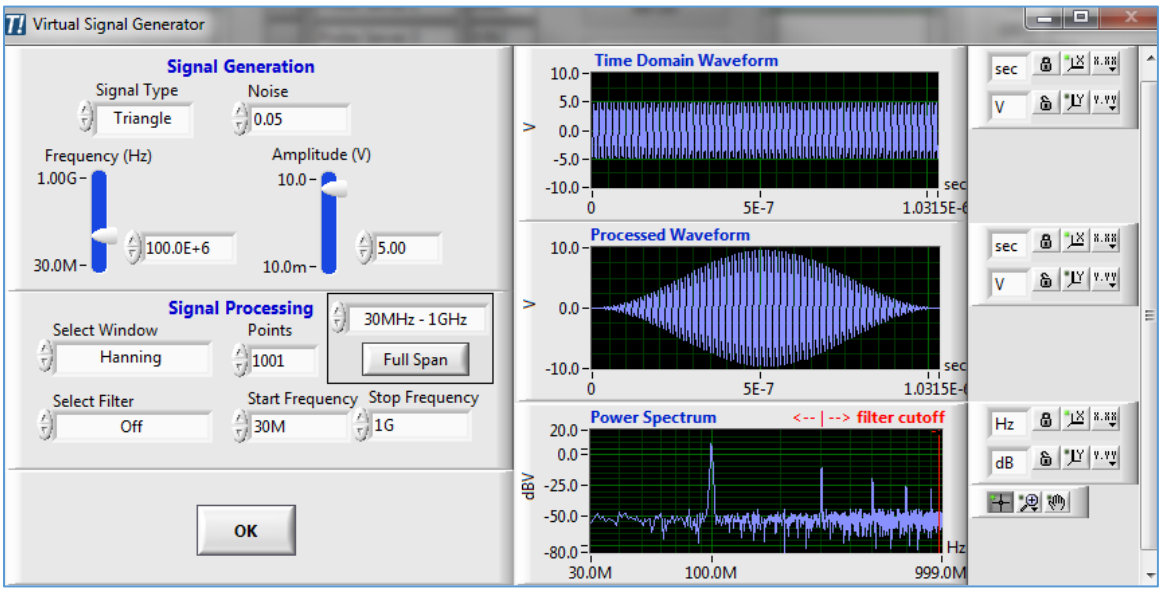
c. Every recognized command will be parsed and processed as a separate response.

10. Data Out indicator.
    a. Displays the most recent IO sent from the simulation server to TILE!.
11. Responses indicator.
    a. Shows the number of responses recorded by the simulation server.
    b. A response corresponds to a single command received by the server.
12. Model Detected indicator.
    a. This string indicates which instrument model response is being used based on the processing of received commands. This indicator can help to identify if the simulator has incorrectly matched a command “belonging” to an instrument other than what is being tested.
    b. For example, if the Agilent EPM driver is being used in TILE!, this indicator should read “ag_epm”. If it says something else, then the simulator may have incorrectly parsed input commands and the config file may need to be adjusted.
    c. Note: some commands such as “*IDN?” will not show a Model Detected value, since this command is shared by many different instrument models.
13. Frequency indicator.
    a. Frequency commands are parsed for both PwrMtr and SigGen servers. If the PwrMtr instrument driver supports Pwr_Set_Frequency, the PM Simulator panel should show the current frequency value (should match the SigGen Simulator frequency indicator, as well as the TILE! action’s frequency indicator). Some PwrMtr drivers do not send frequency information, and the frequency indicator on the PwrMtr Server panel will remain at its default value (-1 Hz).
14. Channel indicator (PwrMtr /Probe servers only).
    a. If a ReadPt or SetFreq command sends channel information, this indicator will parse the channel value and show the last value. If no channel information is sent, the indicator will show 0.
15. Output value indicator (SigGen server only).
    a. This indicator shows the last level successfully parsed and “output” by the SigGen simulation server. A new output value is set when a SetLevel command is detected.
    b. The value is “output” in the sense that this value is now available to an input server when that servers has its Synchronize Mode set to Synchronize PM to SG.
16. Remote/Local (Positioner servers only).
    a. Local mode allows the Current Position gauge/slider to be controlled interactively from the simulation panel. Reading the current position in TILE! reflects the value set by the user.
    b. Also, the SA Servers will respond appropriately to Local mode Positioner changes when they are set to an appropriate Positioner Mode.
    c. Remote mode gives control of the Positioner Server back to TILE!.

Virtual Signal Generator

The Virtual Signal Generator, or VSG, is a utility included with TILE! Simulation that creates a signal source available to any input server (PwrMtr, Probe, SA, or Scope Servers). Regardless of whether an output server (SigGen Server) is currently active, input servers always have the VSG available in their Data Source dropdown.

The VSG panel can be viewed by pressing the “Show VSG” button on an SA Server, or “VSG” button on a PwrMtr/Probe Server. The VSG panel does not have to be showing for its signal output to be active. The VSG is always running in the background when any input simulators are active in the system. You only need to view the VSG panel if you want to change the signal it is producing, after which you can close the panel by pressing the OK button.



Virtual Signal Generator

The Virtual Signal Generator is particularly useful for Emissions applications in order to provide a signal source for the simulated Analyzer or Receiver and emulate the EUT in the system. Using the default input settings of the VSG, the Triangle Signal Type will produce peaks at the primary frequency of 100 MHz, and at each 100 MHz increment for the harmonics. These peaks can then be measured in a TILE! action such as Measure Peaks, Scan Peaks, or Optimize.

Command Config CSV files

Seven Config CSV files must be present for the Simulation Servers to operate. These files are installed in the Simulation\Config Files\ subdirectory. The files are PM Cmd Config.csv, SG Cmd Config.csv, SA Cmd Config.csv, Positioner Cmd Config.csv, Switch Cmd Config.csv, Amplifier Cmd Config.csv, and Scope Cmd Config.csv. These files contain the lists of commands understood by each simulation server type and can be modified to “teach” the simulators commands for new instruments.

An eighth file, Command Configurations.xlw, is an Excel workspace file which loads the other files together for convenience. Generally, the parser syntax follows the PERL compatible regular expression (PCRE) syntax rules (refer to www.pcre.org for more info). This parsing syntax is commonly known as “regex”.

The active Command Config files for Simulation can be viewed by using the Tools->View Config Files option in the Configure Simulation panel. Here are the descriptions of some of the key Command Config file column types.

1. Primary Search field (all files).
    a. This field specifies the regex pattern for each command entry. This column defines the primary parsing step that determines how (or if) each incoming input string maps to a corresponding simulator command.
2. Cmd Type (all files).
    a. This entry specifies the Command Type of the command.
3. Response (all files).
    a. This entry defines the response for Response command types. More than one command type will produce a response, but the Response command type has a fixed response.
    b. A common example of this is the *IDN? query. The simulator will respond the same way whenever this command is received.
4. Frequency Multiplier field (PM Cmd Config.csv).
    a. Some power meters set their frequency in units other than Hz (such as GHz), with no corresponding units characters (i.e. instead of sending “100MHz”, the drivers sends only “0.1”).
    b. The simulator must have specific knowledge of the frequency multiplier to produce the correct value, so this column in the PM Cmd Config file is dedicated to this multiplier.
5. RdPt Format specifier (PM Cmd Config.csv).
    a. This entry tells the simulator how to format its response when returning data. This is useful when extra characters must be appended to the beginning of the response string. To configure a server to send a standard ReadPt response, use the format string: ‘%f’.

For Advanced Debugging techniques and the ability to modify the Command Config tables in real time, refer to the *Command Config Editor* section at the end of this document.

Supported TILE! Drivers with ver. 2.2.1 CFG files

Note: The default TILE! driver for each simulator type is listed in bold as the first entry of the list.

1. Power Meters:
    a. ag_epm, btn4200, btn4232, ETS_EMField, ETS_EMPower, hp4419, rs_nrvd, rs_nrp, rs_ure, t6006_pwm, wavetek_8502, yok_wt1600.
2. Signal Generators:
    a. ag_n518x, ag_n9310a, ag33250, ETS_EMGen, marc_2023, rs_smb100a, t6006_gen
3. Probes:
    a. hi6100, ar_fl7004, ETS_EMSense
4. Amplifier:

a. cpi_j_series_opt_d, ar_amp_modern, ar500a100, milmega_ac001, ETS_EMField, ETS_8100_Series, Bonn, rs_bba100, serial_direct_1

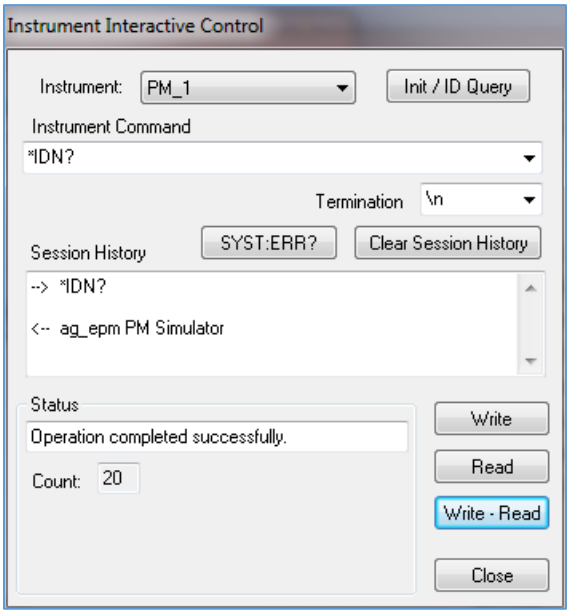
5. Signal Analyzers:
    a. ag_mxa, ag_mxe, rs_esu_sa, ag_e444x, ag_psa
6. Receivers:
    a. ag_mxa_rx, ag_mxe_rx, rs_esu_rx
7. Positioners:
    a. emco_2090, frankonia_fc02, frankonia_fc06, ETS_EMControl, ETS_EMField, anaheim
8. Switches:
    a. ETS_EMSwitch, ETS_EMControl, t6006_swt
9. Scopes:
    a. ag_3104

Debugging with Simulation

Instrument Interactive Control Utility

Just like with a physical instrument, it may be helpful to use an interactive IO utility to communicate to the simulator and test individual commands. The TILE! Instrument Interactive Control utility works well for this purpose.

As an example, send "*IDN?" to PwrMtr Server 1 and query the response. The simulator will send a response indicating its simulator type: "PM Simulator". If the simulator has been linked to a specific TILE! driver it will respond with the name of that driver in its ID string, e.g. "ag_epm PM Simulator".



Using Interactive Control Utility for Debugging

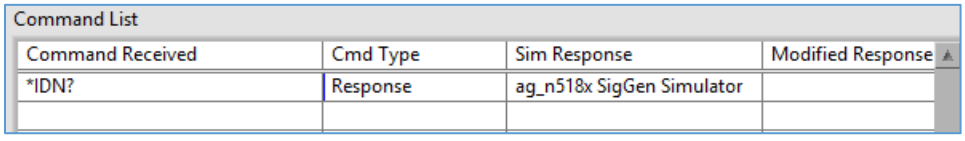
Similarly, you can test other commands recognized by the simulation server, such as “FETC1?”, “SENS1:FREQ 100MHZ”, etc. Refer to the appropriate Cmd Config.csv file (or use Show Command Config Editor) to view specific commands recognized by each simulator type.

Response Editor

The Response Editor provides a real-time log of incoming commands received by the simulation server that the editor was launched from. The interface provides some information about those commands, as well as some limited capabilities to modify the simulator’s behavior when the command is received again.

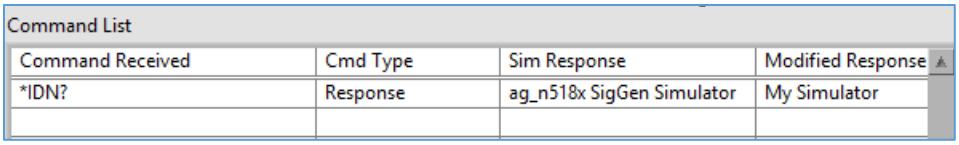
Right-click an empty space on any simulation panel and select Show Response Editor to launch the Response Editor debugging panel.

1. Command Received
    a. This column lists the incoming commands as they are received.
    b. Only unique commands are listed.
    c. The Consolidate Roots checkbox determines whether the uniqueness applies to the command “root” (the part of the command that comes before the first space), or to the entire command. Unchecking Consolidate Roots will produce many more entries in the Command List.
2. Cmd Type
    a. This column lists the simulator’s command type.
    b. Some command types are common to all simulators (e.g. Response), whereas other command types are specific to each simulator type (e.g. SetRFOn).
    c. If Show Unrecognized Commands is unchecked, every Command Received entry should have a corresponding Cmd Type entry.
    d. If Show Unrecognized Commands is checked, unrecognized commands are also listed and the Cmd Type entry will be blank.
3. Sim Response and Modified Response
    a. If the command received produces a corresponding response, the response sent for that command will be listed in the Sim Response column.
    b. The user can cause the simulator to send a different response on subsequent matching commands by entering text in the Modified Response column box.
    c. As an example, follow these steps.
        i. In TILE!, use the Instrument Interactive Control utility to query the *IDN? command. The Response Editor will list the command with the Response Cmd Type.



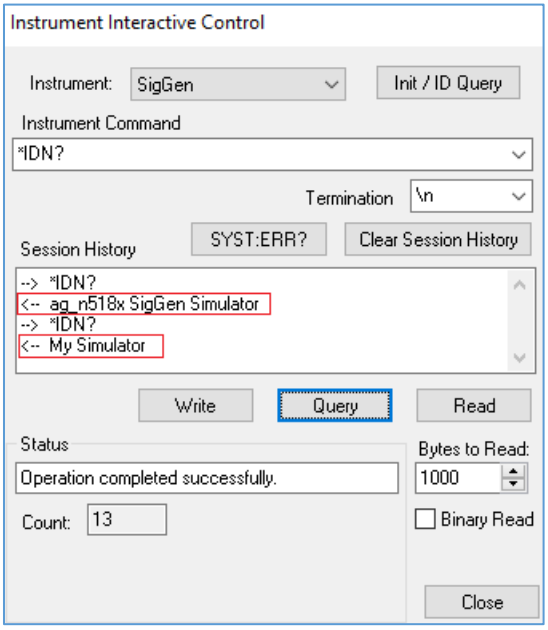
*Response Editor with *IDN?*

        ii. Type “My Simulator” into the Modified Response column box.



Response Editor with Modified Response

iii. Re-query the *IDN? command from TIL! You will see that the simulator's response has been modified from the previous value to the new value.



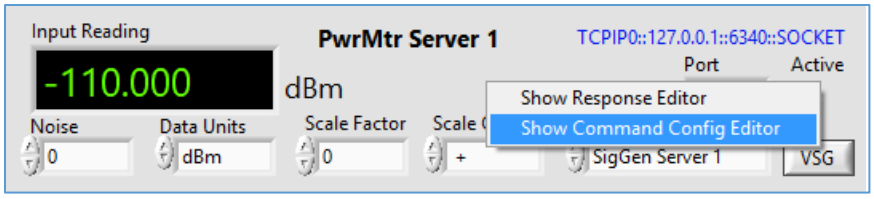
Modified responses in TILE! Instrument Interactive Control Utility

Command Config Editor

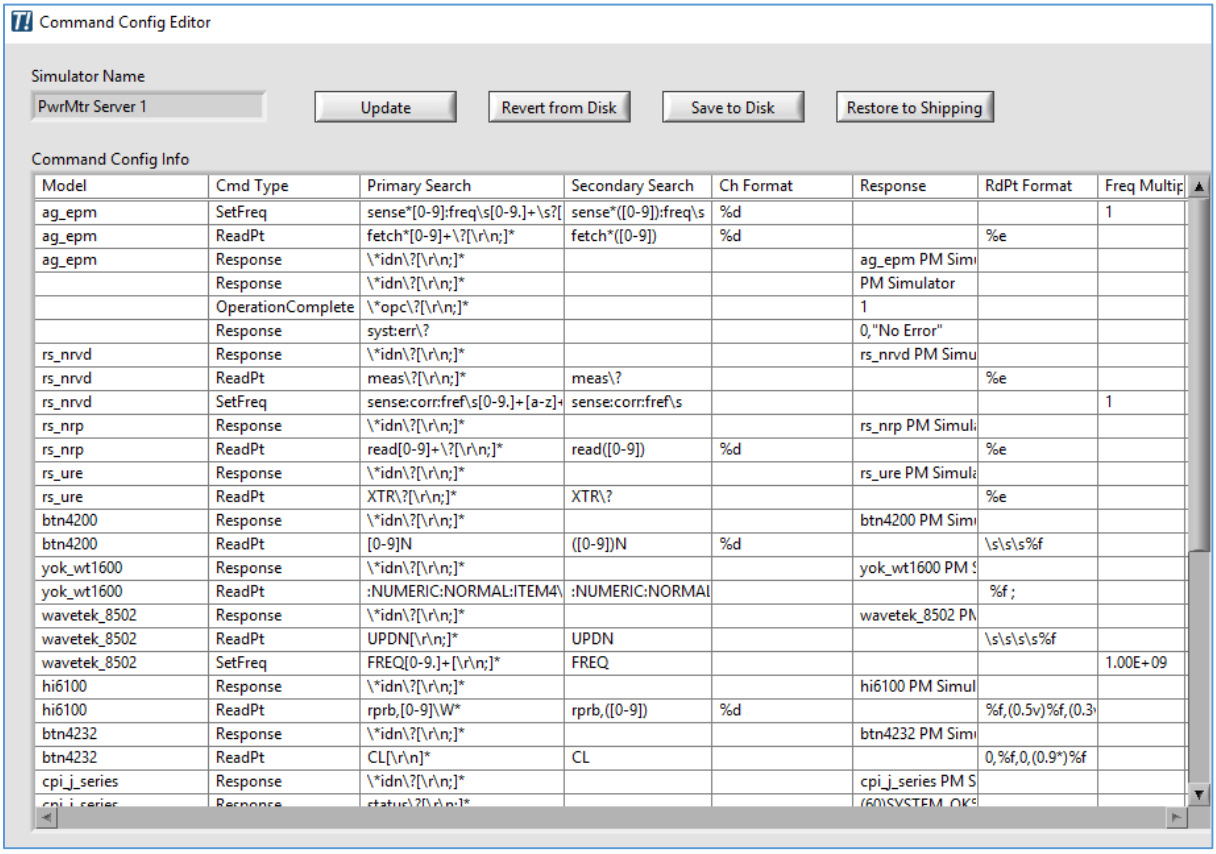
Each simulator type maintains an internal Command Config table of entries that map incoming commands to specific command types and responses. In normal operation, these tables are loaded from CSV files as the simulators launch and then are not modified during the timeframe that the simulator is active.

The Tools->View Config Files menu option in the Configure Simulation panel allows the user to view and edit these files in a spreadsheet or text editor (see the Command Config CSV files section of this document). Modifying the files on disk, however, will not affect the behavior of an already loaded simulator. Using the Command Config Editor, these config tables can be edited while the simulator is active.

Right-click an empty space on any simulation panel and select Show Command Config Editor to launch the Command Config Editor debugging panel.



Show Command Config Editor



Model	Cmd Type	Primary Search	Secondary Search	Ch Format	Response	RdPt Format	Freq Multip
ag_epm	SetFreq	sense*[0-9]:freq\s[0-9.]+\s?[	sense*([0-9]):freq\s	%d			1
ag_epm	ReadPt	fetch*[0-9]+\?[\r\n;]*	fetch*([0-9])	%d		%e	
ag_epm	Response	*idn\?[\r\n;]*			ag_epm PM Simu		
	Response	*idn\?[\r\n;]*			PM Simulator		
	OperationComplete	*opc\?[\r\n;]*			1		
	Response	syst:err\?			0,"No Error"		
rs_nrvd	Response	*idn\?[\r\n;]*			rs_nrvd PM Simu		
rs_nrvd	ReadPt	meas\?[\r\n;]*	meas\?			%e	
rs_nrvd	SetFreq	sense:corr:fref\s[0-9.]+[a-z]+	sense:corr:fref\s				1
rs_nrp	Response	*idn\?[\r\n;]*			rs_nrp PM Simul		
rs_nrp	ReadPt	read[0-9]+\?[\r\n;]*	read([0-9])	%d		%e	
rs_ure	Response	*idn\?[\r\n;]*			rs_ure PM Simul		
rs_ure	ReadPt	XTR\?[\r\n;]*	XTR\?			%e	
btn4200	Response	*idn\?[\r\n;]*			btn4200 PM Simu		
btn4200	ReadPt	[0-9]N	([0-9])N	%d		\s\s\s%f	
yok_wt1600	Response	*idn\?[\r\n;]*			yok_wt1600 PM S		
yok_wt1600	ReadPt	:NUMERIC:NORMAL:ITEM4\	:NUMERIC:NORMAL			%f ;	
wavetek_8502	Response	*idn\?[\r\n;]*			wavetek_8502 PM		
wavetek_8502	ReadPt	UPDN[\r\n;]*	UPDN			\s\s\s\s%f	
wavetek_8502	SetFreq	FREQ[0-9.]+[\r\n;]*	FREQ				1.00E+09
hi6100	Response	*idn\?[\r\n;]*			hi6100 PM Simul		
hi6100	ReadPt	rprb,[0-9]\W*	rprb,([0-9])	%d		%f,(0.5v)%f,(0.3	
btn4232	Response	*idn\?[\r\n;]*			btn4232 PM Simu		
btn4232	ReadPt	CL[\r\n]*	CL			0,%f,0,(0.9*)%f	
cpi_j_series	Response	*idn\?[\r\n;]*			cpi_j_series PM S		
cpi_j_series	Response	status\?[\r\n;]*			(60)SYSTEM_OK		

PwrMeter Server Command Config Editor

Although a complete explanation of the Command Config Table is beyond the scope of this manual, here are some basic explanations that may be helpful. The specific column headers/functions will vary by simulator type, but there are some columns that are common to all simulators.

The most important columns are Primary Search and Cmd Type. Primary Search defines the parsing string (using regex syntax), that is used to map incoming command strings to corresponding simulator command types. Cmd Type defines which simulator function will execute when a matched command is received. Response defines the response string that is sent when an incoming string maps to the Response command type.

Refer to the *Command Config CSV files* section of this document for more information about the Command Config table format.

Command Config Editor Buttons

1. Update

a. Press this button to update the active Command Config table. This allows intermediate entries to be entered in the table without immediately affecting the simulator behavior.

b. Any updates to the Command Config table will only affect this specific simulator and session. It does not affect other simulators, even ones of the same simulation type.

c. Updating the active Command Config table does not affect the CSV files stored on disk. Thus, if the simulator is closed and then later reloads from disk, the behavior will revert to what is represented in the disk copy.

d. Generally, the Ctrl-Z Undo operation allows previous entries to be corrected while modifying the table.

2. Revert from Disk

    a. This button will restore the active Command Config table to the copy that is saved on disk.

3. Save to Disk

    a. This button will transfer the active Command Config table (modified by Update or not) to the copy saved on disk. This makes the changes for this session more permanent, in that they will re-load the next time the simulator is run.

    b. Note that this will affect all simulators of this simulation type (e.g. Power Meters), since a single, common, Command Config CSV file is used for each simulator type.

4. Restore to Shipping

    a. A master copy of each simulation type is maintained separately from the active disk file.

Editing the Command Config Table

Cells can be modified by clicking in the cell and entering new text. Click the Update button to transfer the modified table to the active table.

To delete or create new rows, right-click a row entry and select one of the following options:

1. Delete Row

    a. Deletes the current row.

2. Insert Row Before

    a. Inserts a new row above the current row.

3. Copy Data

    a. Select multiple cells first, then select Copy Data.

    b. This function can be used to copy and paste an existing entry (or entries) into a newly inserted row or rows.

4. Paste Data

    a. Select multiple cells first, then select Paste Data to paste previously copied values into the new location.

    b. NOTE: the shape (rows/columns) of the copied cells must exactly match the shape of the pasted cells.

IVI class drivers

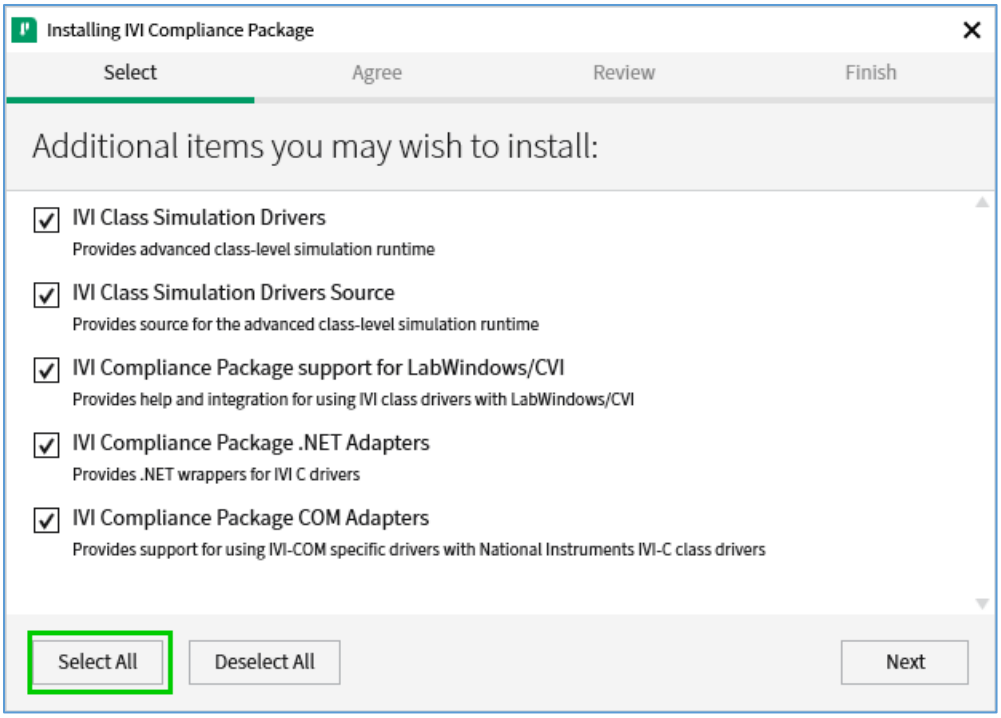
Interchangeable Virtual Instrument (IVI) class drivers provide an interface that allows TILE! to support multiple vendors' instrument models with a single driver for a particular instrument class. TILE! version 7.2.1 and higher has added support for IVI class drivers.

IVI development and run-time tools are maintained by the IVI Foundation, an industry-wide organization with membership that represents over 20 instrument manufacturers. ETS-Lindgren develops IVI class drivers in TILE! using tools provided by IVI. TILE! customers install the necessary IVI run-time utilities, as well as specific drivers provided by the vendors, in order to use the IVI class drivers.

Installation

The following installers are prerequisites to using the IVI class drivers in TILE!. Perform the installations in the specified order.

1. **NI-VISA**
    a. **NOTE**: Most TILE! users will already have NI-VISA installed; If so then you can skip this step.
    b. Download the latest VISA installer from ni.com and run the installer.
        i. https://www.ni.com/en-us/support/downloads/drivers/download.ni-visa.html
    c. Leave the Additional Items selections at their default values. Follow all prompts and restart your computer if directed.
2. **NI IVI Compliance Package**
    a. Download the latest NI IVI Compliance Package installer from ni.com and run the installer.
        i. https://www.ni.com/en-us/support/downloads/drivers/download.ivi-compliance-package.html
    b. Press the Select All button on the Select tab of the IVI Compliance Package installer:



c. Follow all prompts and restart your computer if directed.

3. Vendor's IVI specific drivers for the instrument models to be used

   a. These drivers will usually be available from the instrument vendor's website. If the vendor offers 32-bit and 64-bit options, install the version corresponding to your operating system.

   b. **NOTE**: TILE! supports all three types of IVI specific drivers: IVI-C, IVI-COM, and IVI.NET.

      i. The **IVI Foundation** maintains a registry of IVI drivers:

         1. http://www.ivifoundation.org/registered_drivers/driver_registry.aspx

      ii. **National Instruments** maintains an extensive database of instrument drivers, including IVI vendor drivers; click Technology in the list of filters and select IVI:

         1. http://www.ni.com/idnet

      iii. Example download locations:

         1. **R&S RTE/RTO/RTP (RsScope), RTA/RTB/RTM (RsRtx);** Click your model's link in the results list, then look for IVI.NET x64 driver for the download.

            a. https://www.rohde-schwarz.com/us/search/driver_63451.html?term=drivers+for+oscilloscopes&sort=relevance

         2. **Keysight InfiniiVision**:

            a. http://www.keysight.com/main/software.jspx?ckey=2019021&lc=eng&cc=US&nid=-32540.1150197&id=2019021

         3. **Tektronix DPO/MSO 2000/3000/4000 series:**

a. https://www.tek.com/oscilloscope/dpo2002-software/dpo2000-mso2000-dpo3000-and-dpo4000-mso3000-and-mso4000-mdo3000-and

Run MAX to configure the IVI device

NOTE: MAX can be launched directly from TILE! by selecting NI MAX… in the Tools menu.

- Refer to this NI Tutorial for configuration steps with screen shots:
  - http://www.ni.com/tutorial/3377/en/
- Or, follow these steps:
  1. Under My System->IVI Drivers, right-click on Driver Sessions and select Create New.
  2. Name the driver session (e.g. RsScope).
  3. Select the new session just created.
  4. Click the Software tab on the bottom of the page.
  5. Click the Software Module dropdown and select the specific IVI driver to use.
     a. **NOTE**: Specific IVI drivers will appear in this list automatically after they have been installed. You must install the vendor's specific driver for your instrument before it can be configured in MAX.
  6. Click the Hardware tab.
  7. Click Add and name the Hardware Asset (e.g. RTO GPIB).
  8. Click the Resource Descriptor and select from the entries in the dropdown.
     a. If the instrument is connected to your system, you should see an entry that represents the VISA resource associated with the method of communication for that device.
     b. For instance, a GPIB connection might be listed as "*GPIB::20::INSTR*", or a USB connection as "*USB::……*", etc.
     c. If you do not see a GPIB address corresponding to your connected GPIB device, right-click the GPIB device under Devices and Interfaces and select Scan for Instruments.
  9. Make sure the checkbox to the left of the Hardware Asset just created is checked.
  10. Right-click the Logical Names folder under IVI Drivers in the left pane and select Create New.
  11. Enter a Logical Name for your IVI session. This is the name that will be referred to and appear in TILE!.
  12. Select the Driver Session dropdown and select the Driver Session created earlier.
  13. Click Save IVI Configuration (changes will not appear in TILE! until the MAX IVI configuration is saved).

Using the IVI Session in TILE!

1. Create a new instrument in the profile.
2. On the Driver tab, Browse to select the desired driver.
    a. IVI class drivers will begin with the “ivi_” prefix, e.g. ivi_scope.ins.
    b. **NOTE**: selecting an IVI class driver automatically selects IVI as the Communication type on the Address tab.
3. On the IVI tab, click the IVI Logical Name dropdown to view the list of available Logical Names configured in MAX.
    a. If you do not see the logical name(s) as expected, go back and review the IVI configuration in MAX.
    b. TILE! will not be able to open an IVI session to the device if the MAX configuration is not completed (and the Save IVI Configuration button is pressed after making changes).
    c. **NOTE**: MAX can be launched directly from TILE! by selecting NI MAX… in the Tools menu.
4. Press the Validate button to test the IVI Session, or just click OK on the Instrument dialog.
    a. If the IVI Session initialization generates an error you will see a message dialog.
5. Use the instrument in TILE! actions as desired.
6. **NOTE**: TILE! generates a VISA resource for the IVI instrument that can be used to communicate directly with the instrument. This means that “direct IO” tools such as the Instrument Commands action (VISA icon) and the Instrument Interactive Control utility can still be used.
7. IVI Properties can be viewed in the Property View when selecting the IVI-configured instrument in the Instrument View. Information about the specific driver and class driver in use for this instrument might be helpful for debugging purposes:

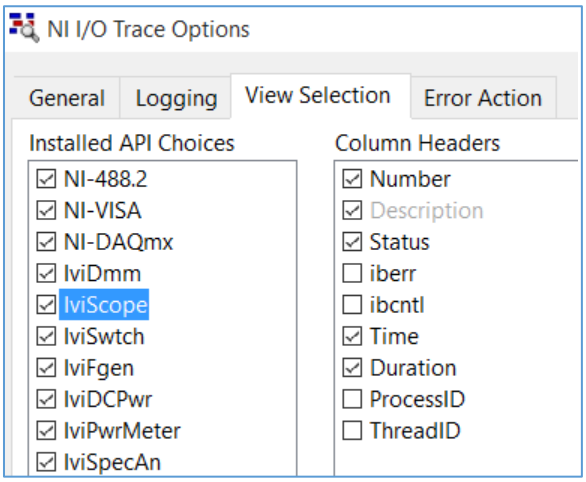
IVI Properties	
Specific Driver Description	IVI driver for the Keysight 2000X, 3000X, 4000X and 6000X family of Oscilloscopes
Specific Driver Vendor	Agilent Technologies
Specific Driver Prefix	AgInfiniiVision
Specific Driver Revision	2.1.9.0
Specific Driver Class Spec	4.1
Class Driver Description	IviScope Class Driver
Class Driver Vendor	National Instruments
Class Driver Revision	Class: IviScope 4.00, Compiler: MSVC 9.00, Components: IVIEngine 16.00, VISA-Spec
Class Driver Class Spec	0.0

Using the NI IO Trace Utility with IVI Class Drivers

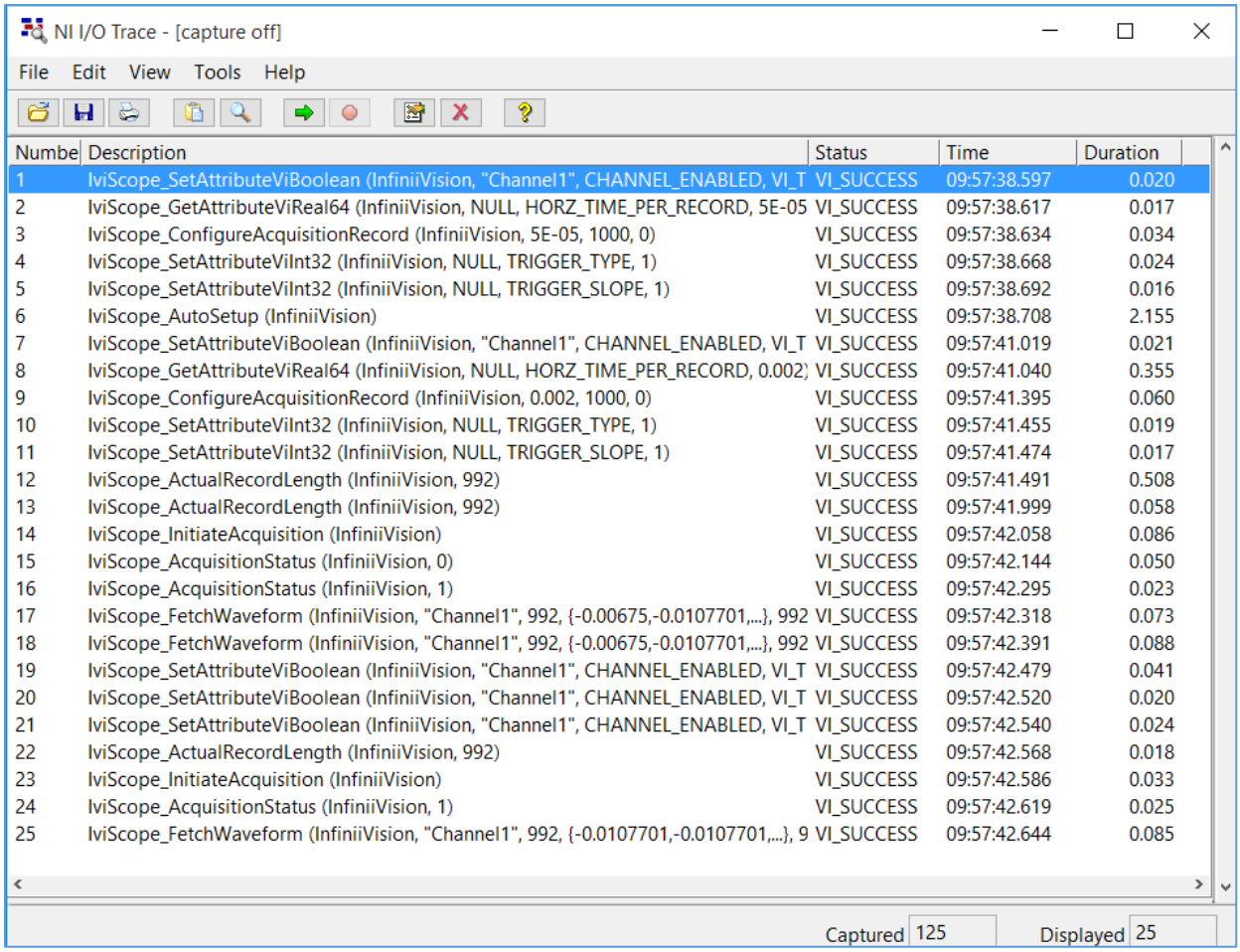
When IVI class drivers are in use, TILE! does not send direct SCPI commands to the device using standard GPIB or VISA API functions. Because of this, the typical IO Log entries will not appear in the TILE! Log View. Instead, you can use the NI IO Trace Utility to show the communication between TILE! and the device.

Configuring the NI IO Trace Utility for IVI

1. Launch NI IO Trace Utility by selecting NI I/O Trace… in the TILE! Tools menu.
2. Select Tools->Options from the menu (stop the capture if this entry is disabled).
3. On the View Selection tab, make sure that the IVI class API in use is selected in the Installed API Choices section:

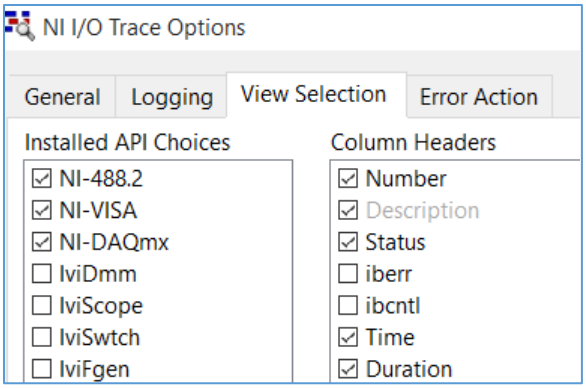


4. Start the Trace Capture and run your TILE! profile actions that use the IVI class driver. You should see the IO captures specific to the IVI class driver API in use. The function calls will begin with a prefix representing the class driver (e.g. IviScope_). Function call parameters and error conditions can be observed in the log.
5. Here is an example IO Capture when running the Initialize Instruments and Read Instrument actions to retrieve a trace from an oscilloscope:

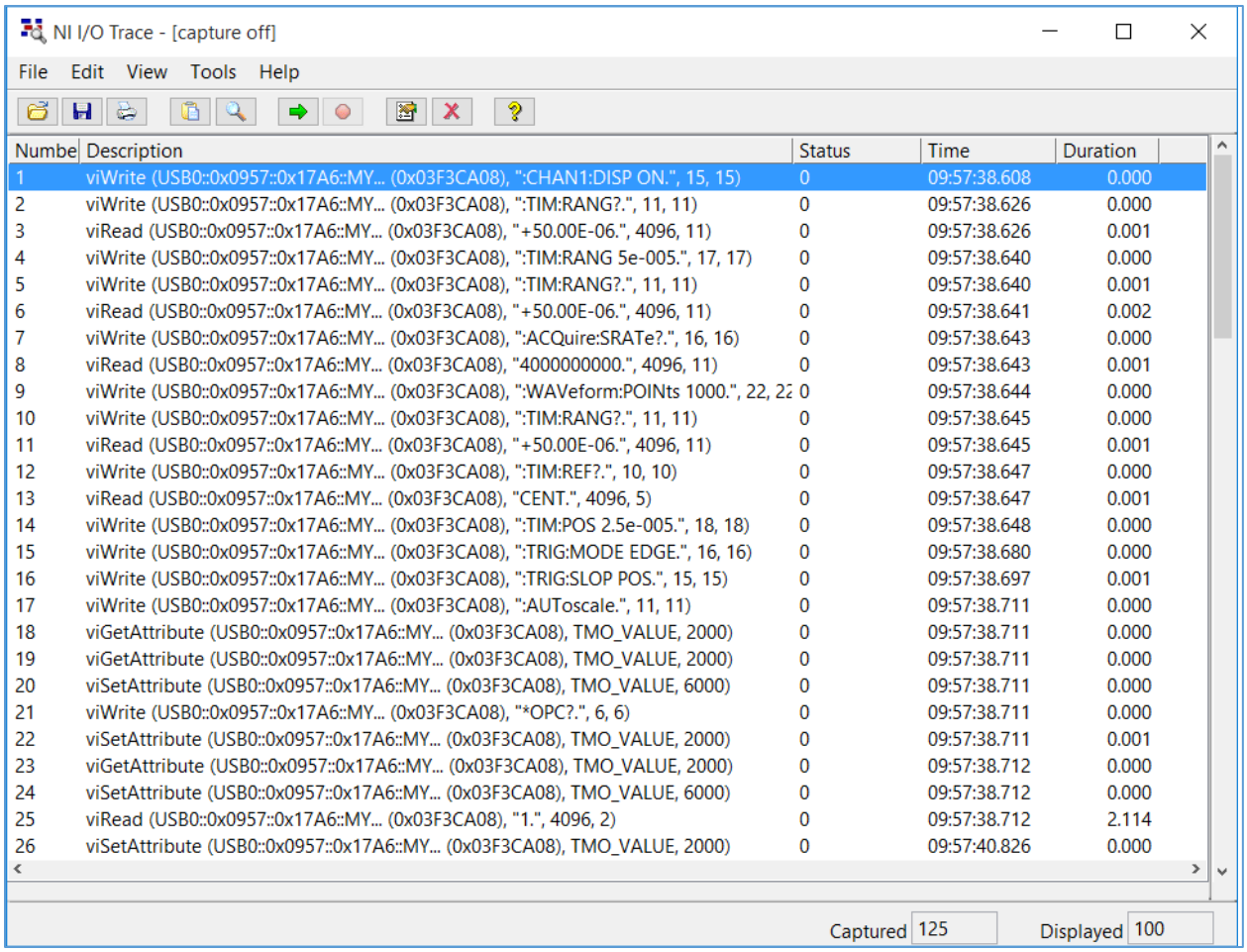


Selecting APIs to view

- IVI class drivers:
    - IVI class drivers create multiple levels of API functions to achieve the desired abstraction (and thus interchangeability) between the application software and the instrument.
    - TILE! makes IVI class API calls IviScope, IviPwrMeter, etc.).
    - The class driver invokes the specific vendor driver, which determines the low level API in use (VISA or native GPIB).
    - Depending on the specific driver in use, these multiple APIs may or may not be able to be viewed simultaneously in NI IO Trace Utility.
- Typically, viewing the IVI class API masks the ability to view the low-level API (even if it is selected on the View Selection tab).
    - If this is the case, you can unselect the IVI class API checkbox in order to allow viewing of the low-level API:



- Now when we re-run the same test as before, we see the low-level API functions appear in the trace capture:



Numbe	Description	Status	Time	Duration
1	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":CHAN1:DISP ON.", 15, 15)	0	09:57:38.608	0.000
2	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:RANG?.", 11, 11)	0	09:57:38.626	0.000
3	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "+50.00E-06.", 4096, 11)	0	09:57:38.626	0.001
4	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:RANG 5e-005.", 17, 17)	0	09:57:38.640	0.000
5	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:RANG?.", 11, 11)	0	09:57:38.640	0.001
6	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "+50.00E-06.", 4096, 11)	0	09:57:38.641	0.002
7	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":ACQuire:SRATe?.", 16, 16)	0	09:57:38.643	0.000
8	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "4000000000.", 4096, 11)	0	09:57:38.643	0.001
9	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":WAVeform:POINts 1000.", 22, 22	0	09:57:38.644	0.000
10	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:RANG?.", 11, 11)	0	09:57:38.645	0.000
11	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "+50.00E-06.", 4096, 11)	0	09:57:38.645	0.001
12	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:REF?.", 10, 10)	0	09:57:38.647	0.000
13	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "CENT.", 4096, 5)	0	09:57:38.647	0.001
14	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TIM:POS 2.5e-005.", 18, 18)	0	09:57:38.648	0.000
15	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TRIG:MODE EDGE.", 16, 16)	0	09:57:38.680	0.000
16	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":TRIG:SLOP POS.", 15, 15)	0	09:57:38.697	0.001
17	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), ":AUToscale.", 11, 11)	0	09:57:38.711	0.000
18	viGetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 2000)	0	09:57:38.711	0.000
19	viGetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 2000)	0	09:57:38.711	0.000
20	viSetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 6000)	0	09:57:38.711	0.000
21	viWrite (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "*OPC?.", 6, 6)	0	09:57:38.711	0.000
22	viSetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 2000)	0	09:57:38.711	0.001
23	viGetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 2000)	0	09:57:38.712	0.000
24	viSetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 6000)	0	09:57:38.712	0.000
25	viRead (USB0::0x0957::0x17A6::MY... (0x03F3CA08), "1.", 4096, 2)	0	09:57:38.712	2.114
26	viSetAttribute (USB0::0x0957::0x17A6::MY... (0x03F3CA08), TMO_VALUE, 2000)	0	09:57:40.826	0.000