TOTALLY INTEGRATED LAB ENVIRONMENT™

Reference Manual

ETS·LINDGREN™
An **ESCO Technologies** Company

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

Revision	Description	Date
A	Initial Release	August, 2011
B	Updates to selected descriptions and formulas	June, 2012
C	**Updates:** *Working with the Flowchart*-graphic, *Toolbars-Standard Bar*, *Switches*-graphic, *Measurement, Positioner, and Control Actions*-graphic **Additions:** *Windows Menu Bar-Tools*, *EMSwitch*, *Call Action*	July, 2012
D	Updated contact information; miscellaneous edits.	May, 2013
E	Updated *Steps to Create a Profile*	June, 2013
F	Added TILE! 7 information	January, 2014
G	Update	August, 2016
H	Update	February, 2017

USA Headquarters
1301 Arrow Point Drive
Cedar Park, TX 78613
Phone: +1.512.531.6400
http://support.ets-lindgren.com

Please see ets-lindgren.com for a list of all ETS-Lindgren offices.

Table of Contents

About TILE! 7 and the TILE! Reference Manual

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 will be updated as features are added or edits are needed.

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 you, 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. It has a unique, visual interface. The TILE! concept provides a common user interface for all testing, coupled with integration into various standard report writing software (such as Microsoft® Word) as well as 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.

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

TILE! Technical Support

http://support.ets-lindgren.com

1301 Arrow Point Drive

Cedar Park, TX, USA 78613

Phone: +1.512.531.2640

Fax: +1.512.531.6598

TILE! Installation

This section provides a detailed description of installing the TILE!™ software, including steps to check for the correct GPIB driver and a short tutorial on loading the software. In addition to the TILE! Installation CD and TILE! USB Security Key, you will need a computer that meets the following minimum system requirements.

Minimum System Requirements

- Intel Core™ 2 Quad, 3.0 GHz or compatible

- Microsoft® Windows® 7 64 bit (32 will work, but will limit RAM)

- 4 GB RAM or more (8 GB is noticeably better)

- 320 GB free hard drive space or more

- DVD/CD-R/W Drive

- National Instruments™ GPIB card (USB or PCI)

- 20 in monitor

- Speakers

- 2 free Ethernet ports

- 2 free USB ports

Installation Steps

As a general precaution, it is always recommended that you back up the files to an external hard drive or a network folder prior to installing new software.

You will need Administrator access to the computer to perform the software installation.

It is critical that you reboot during the installation process when you are presented with the reboot option. Rebooting will activate the special drivers that are installed.

1. If you are installing this over an existing TILE! installation, uninstall the previous version prior to installation. The existing profiles and data will not be deleted during the uninstall process.

2. Insert TILE! Installation CD into CD drive. If installing from a downloaded file, navigate to the appropriate temporary directory and click Setup.exe.

   Typical users can accept all default choices during the installation process. Contact the TILE! Support Team if you have questions.

   If a different destination location or program folder is desired, make the appropriate selections.

   The first time you install TILE! you will encounter a dialog box prompting you to select the GPIB card you are using. Only NI GPIB adaptors are officially supported. Others may work, but are not warranted, and in some cases have been known to cause sporadic problems.

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.

3. When the setup and installation is complete, click **Finish**.

4. Once TILE! is installed and the computer restarts, insert the TILE! USB Security Key into a USB port. This will install new drivers. The driver installation process could take up to two minutes. Do not launch TILE! until the driver installation process is complete and the red LED on the USB key is illuminated.

Starting TILE!

Upon opening a blank file, the default view has no attributes associated with any window. To activate a new profile select

the New File icon   or ctrl-n or File ->New.

In File ->Options menu allows the user to de-select an instrument interface.

If you are installing on a computer that will not be used for running tests, select the **Disable GPIB** check box to disable the check for GPIB interfaces at startup.
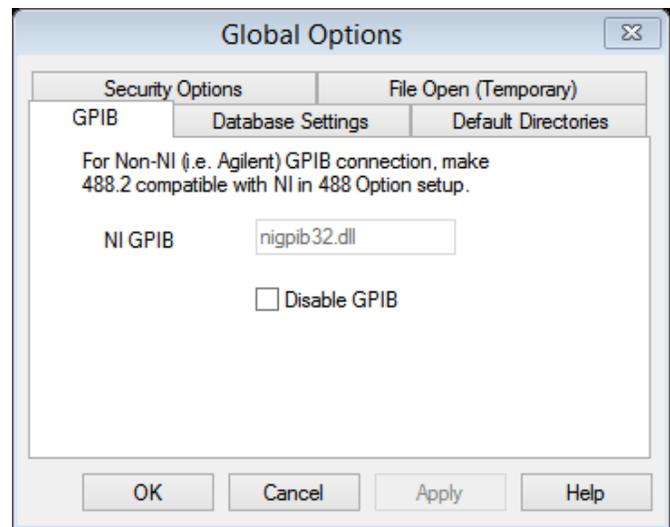
Instrument Control

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).

Most EMC instrumentation is supported by TILE! The emphasis on hardware independence allows for quick introduction of new instrumentation into the laboratory environment. No need to re-create a new profile. Instrument drivers are provided free of charge to registered users except in some situations where equipment has to be leased or timelines require accelerated development.

TILE! Windows

There are seven major windows within a TILE! profile:

- **Flowchart**—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.

- **Data**—Defined data elements are used to store information obtained during the testing, whether for storage of measurement values, specification limits, correction factors, or the results of mathematics upon other data elements.

- **Instrument**—The definition of instruments which will be controlled during this test. This includes the instrument driver and the GPIB or serial port information necessary to address the instrument properly.

- **Log**—Used for debugging purposes, the logging function 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.

- **Audit Trail**—A record, in text format, of the configurations of each action present on the flowchart.

- **Graph**—A visual form of data presentation. The Graph window can be created multiple times within each profile. Complex displays of tests may require many different graphical outputs. TILE! allows each graph to be defined to present different views of the data.

- **Table**—A spreadsheet form of data presentation. A Table window can be created multiple times within each profile.

The Flowchart, Data, Instrument, Log, and Audit Trail windows are considered permanent and cannot be closed by the user (they can only be minimized). The ability to define multiple views of the data, through graphs and tables, is a feature of the program. Multiple graphs and tables can be created with their own unique view of the data elements. Graph and Table windows are considered temporary views of the data since the user can close them, and thus eliminate that view of the data.

Security – Password Protection

Previously the relevant properties regarding security were System Security Active, File Security Active, and Password. The main addition to the security settings in this version is the addition of a second password, File Password. The old password is now called the System Password. Previously, the Password was used to change either the System or File Security Active flags. But the Password really was a "system" password, meaning it was only relevant to a particular 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 type the old password into the new system, but discovered that the password was no longer recognized. This also allowed for a pretty 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 preserved, also.

The Security Options dialog has been updated to handle the new setting. To avoid confusion, the two passwords (System and File) each have their own input box, and there are also separate buttons to change each password. Here is a comparison of the old and new interface:

Old View:
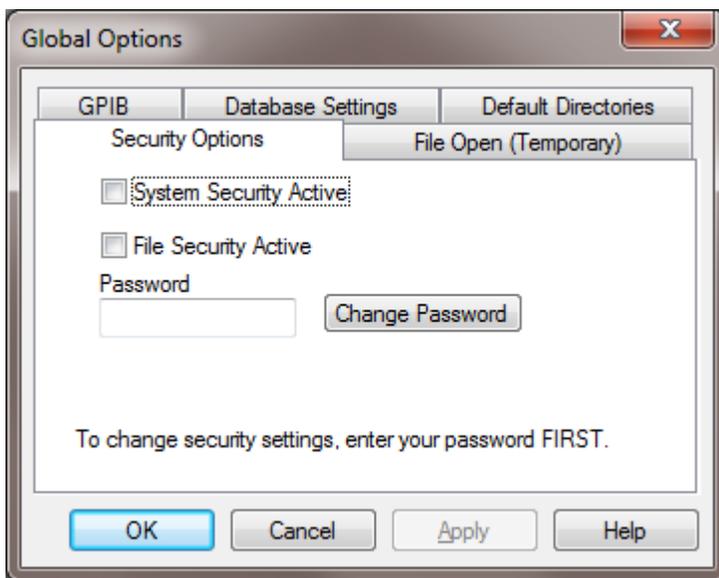
New View:

A common UI is used for changing either password. Whichever Change Password button is pressed will modify that specific password:

There is intelligent enabling of controls based on whether a file is currently open. If the Options Dialog is accessed with no file open, then the File Security settings are hidden:

A valid TILE! key and license must now be present to make any changes to security settings (enabling/disabling System or File Security Active flags, or changing either the System Password or the File Password).

2.0 New Features

New Look

The updated look of TILE! 7 includes new window behaviors, a project view, and the properties pages. 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 is always available either 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 floating.

Views can be saved as user preferences.

Project View

The Project view provides an easily-accessible view that contains all of the components of the profile. Selecting a component from this view will bring it to the forefront for viewing or editing.
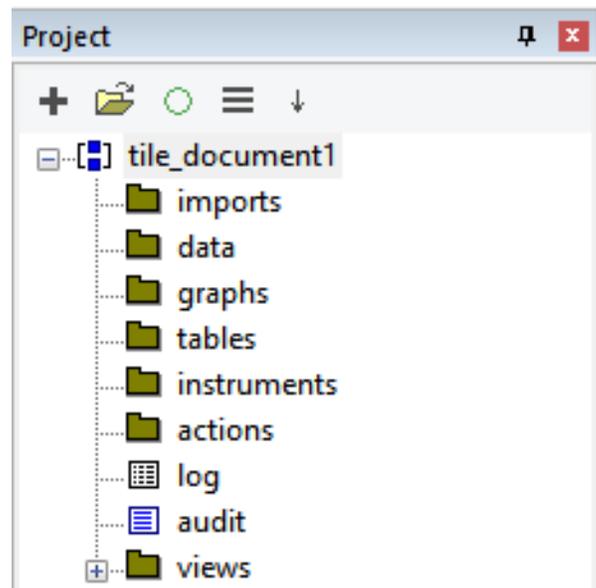
Properties View

The Properties view allows better viewing of each component of the profile. This view is an updated interface of the normal tabbed editing view. It is available for actions, graphs, tables, data elements, and instruments.
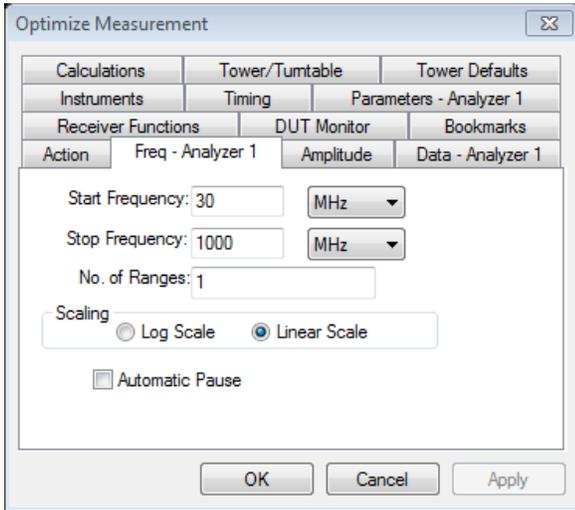
Legacy view of an action
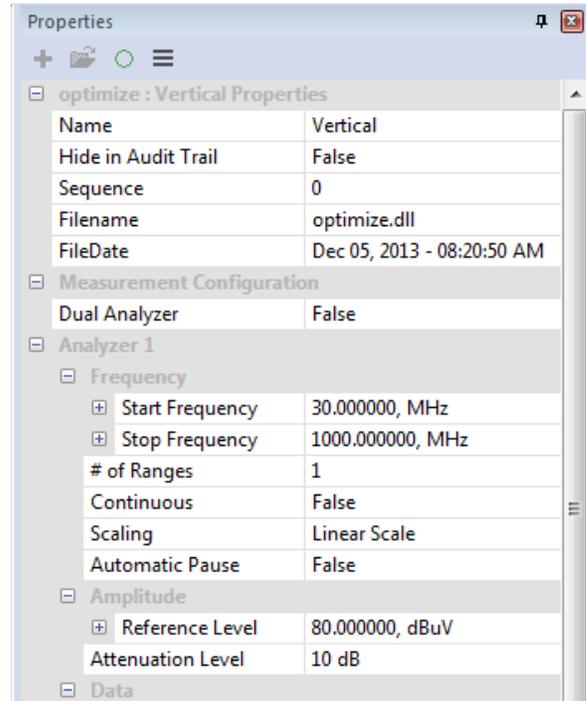
Properties view of an action

Scripting

The TILE! Scripting feature allows users to create scripts to enhance the usability of the program. Scripting does require some knowledge of software 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. 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's really no limit to the features that can be developed with this tool. The Scripting guide as well as the Scripting Linear math guide are found in the TILE7/Help Folder. Scripting and scripting math can be used to enhance the many TILE! math functions already available.

Sequencing

The sequencer 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.

The example below will execute actions in a preferred sequence that are not connected by the flowchart connector 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 1 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.

item \ seq #	seq 1
Clr5.Methods.Action	Execute()
eq_copy_EC_Pos1.Methods.Action	Run()
RVCalc1.Input.Input 1.Fwd Power	EC_P1_IP_Fwd
RVCalc1.Input.Input 1.Max Power Rec	EC_P1_Rx_Max
RVCalc1.Input.Input 1.Avg Power Rec	EC_P1_Rx_Avg
RVCalc1.Input.Input 1.Min Power Rec	EC_P1_Rx_Min
RVCalc1.Input.Input 1.Number of Points	EC_P1_NOP
RVCalc1.Input.Input 1.Probe.Probe RMS Max	EC_P1_Probe_RMS_Max
Eq3.Equation/Script Data.Data	EC_P1_ACF
Eq3.Methods.Action	Execute()
S1.Methods.Action	Execute()
P3.Methods.Action	Go()

item \ seq #	seq 1	seq 2	seq 3	seq 4
Range.Min				
Seq.Continue				
CS1.Frequency.Frequency Items.Start Frequency.Value	100	300	400	600
CS1.Frequency.Frequency Items.Start Frequency.Units	MHz	MHz	MHz	MHz
CS1.Frequency.Frequency Items.Stop Frequency.Value	299.99	399.99	599.99	1
CS1.Frequency.Frequency Items.Stop Frequency.Units	MHz	MHz	MHz	GHz
CS1.Paddles.Positioner 1.Start (deg)	0	360	0	360
CS1.Paddles.Positioner 1.Stop (deg)	360	0	360	0
CS1.Paddles.Positioner 1.Step (deg)	7.2	18	22.5	30
Init1.Methods.Action	Go()			

The sequence shown above illustrates how the sequencer can handle frequency bands in a sequential order.

Instruments Window and Explorer

The instrument window provides a view of the named instruments, their drivers and address information.

IVI	Name	Module	Type	Bd	Addr	Driver	Version	Serial Number	Cal Date	VISA
	Func_Gen	HP 33250 FUNCTION/ARBITRARY WAVEFORM GENERATOR	Signal Generator	Sim		ag33250.ins	7.2.1.5	Unknown	Unknown	TCPIP0::localhost::6350::SOCKET
	RX	Agilent MXE - EMI Receiver Mode	Receiver	Sim		ag_mxe_rx.ins	7.2.1.5	Unknown	Unknown	TCPIP0::localhost::6370::SOCKET
	SA	Agilent MXE Spectrum Analyzer	Spectrum Analyzer	Sim		ag_mxe.ins	7.2.1.5	Unknown	Unknown	TCPIP0::localhost::6371::SOCKET
	Scope	HP EPM 4419 Power Meter	Power Meter	Sim		ag_epm.ins	7.2.1.5	Unknown	Unknown	TCPIP0::localhost::6340::SOCKET

The Instrument Explorer provides a way to find instrument drivers by searching or selecting a vender or instrument type. To enable it, select the + sign in the Instruments window, Select the Driver tab and then the Explore… button

Instrument:\<untitled\>

Instrument | Driver | Driver Info | Address | GPIB

File: None
Path: None

Browse...   Explore...   Load

☐ Do Not Use Driver   ☐ Passive Device
☐ Load User Driver

OK   Cancel   Apply   Help

Explore Drivers (Double Click To Select)

mxg   search   Address (-1 for all) -1   IDN? >>

Amplifier
Burst_Generator
Controller
EFT_Generator
Multimeter
Oscilloscope
Other
Power_Meter
Preselector
Probe
QPAdapter
Receiver
Signal_Generator
Spectrum_Analyzer
Surge_Generator
Switch
Tower
Turntable
unknown

\<unknown\>
Advantest
Aeroflex
Agilent
Amplifier Research
Anritsu
ARINC
Audio Precision
Bonn
Boonton
Bruel & Kjaer
California Instruments
Carnel Labs
Chase
Compumotor
Comtest
CPI
Cytec
DARE
Davis
Device
Dynamic Sciences
Eaton
Electro-Metrics/Tektronix
Elena
EMC Automation
EMCO
Empower
ETS-Lindgren
Fluke
Frankonia
Gauss Instruments
Gigatronics
GraphTec
HD
Hewlett Packard
Holaday
Honeywell
Instruments For Industry
Kalmus
Keytek

name	file	alias	vendor	type
Agilent N518X Series Signal Generators	ag_mxg.ins		Agilent	Signal_Generator

---

Tabs

The standard components of a profile are always available as tabbed windows. They can be moved to any edge, viewed, set to Auto Hide, or windowed to suit the user.

Tabbed View

RE 55022B Demo V9 | **Vertical** ☒ | Turntable | Tower | Table : Top 10 Horizontal | Table : Top 10 Vertical | Graph : Horizontal

Docked Components

Window Behavior (Docked and Tabbed Modes)

From the **View** menu on the toolbar, the user can select a global **Docked** or **Tabbed** mode.

- **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.

Validation

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.

Validation can also run during profile development by selecting it from the **Tools** menu.

Progress Bar and Highlighted Actions

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 in sync with the action progress; however, it does provide a quick look that the profile is running.

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

A blue highlight indicates the current action being processed.
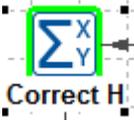
A green highlight indicates that the action has processes successfully.
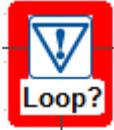
A red highlight indicates that the action was interrupted either by the user or loss of an instrument connection.
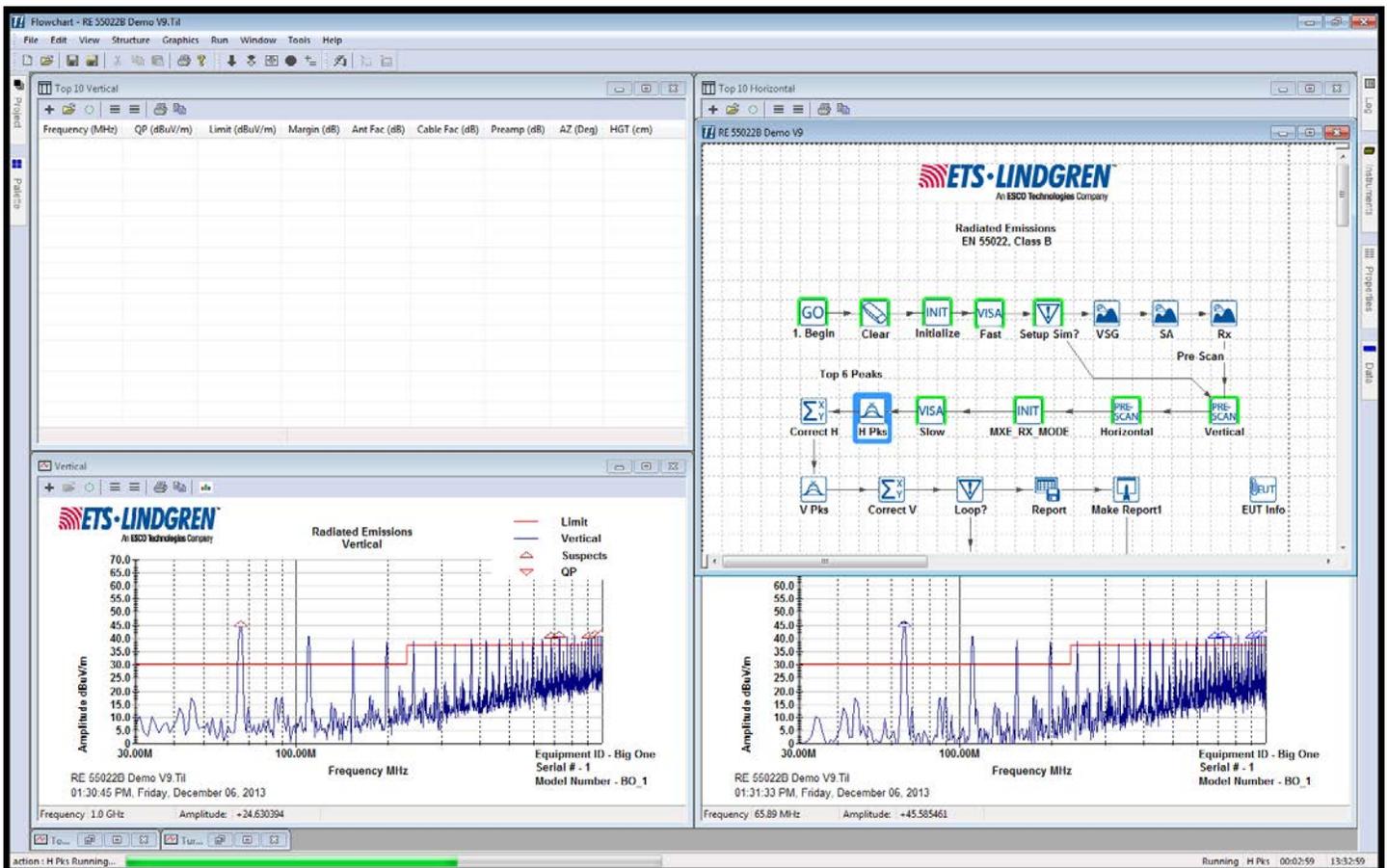
Run Main Action Button (Toolbar)

▶ |

The 'Run Main Action' button (Green 'Play') will allow a single click location to start the profile from a selected action. Typically this would be a Prompt action that would be treated as a Main Menu.

It is enabled by selecting a blank spot in the flowchart and selecting the properties window. The properties of the flowchart will have an input for the 'Main Action'. In the capture below 'Main Menu' is selected.

Properties	📌 ❌
➕ 📂 ○ ☰ │ 🔒	
⊟ Tile Document	
title	RE 55022B V11 with threshold query v4
Main Action	Main Menu
Refresh Rate (ms)	5000

Once this is setup anytime the Green 'Play' button is selected the Main Menu will run. This allows the flowchart to be hidden behind more useful windows such as graphs and tables.

Steps to Create a Profile

This example is used to illustrate how to place a measurement action from the palette onto the flowchart, create data elements and attach an instrument. From there it will show how to run a simulated emissions scan and plot it on a graph. This example is used as a first exercise during hands on training.

There are four steps to create a test profile:

1.   Define the data elements to be used. See *TILE! Data Elements* on page 23.

2.   Identify the instruments used. See *Adding Instruments* on page 39.

3.   Set up the sequence of the test using the flowchart. See *Flowchart Window* on page 45.

4.   Graph and tabulate the data. See *Graphs* on page 48.

For example, follow these steps to create a simple emissions profile using the built-in instrument simulator:

1.   Open a new TILE! workspace.

2.   Select the **Palette** tab.

3.   With a single click, select the **Measure Range Extended** icon and single-click again to place it anywhere in the flowchart.

Measure Range Extended icon

4.   Right-click and select **Edit**.

5.   On the **Action** tab label it **Scan** in the Name: window.

     All tabs will remain in their default states except the **Data1** tab and the **Instruments** tab.

6.   Select the **Data Elements** tab to open the data elements section.

7.   Select the **+** or double-click anywhere in the Data window to open a new data dialogue.
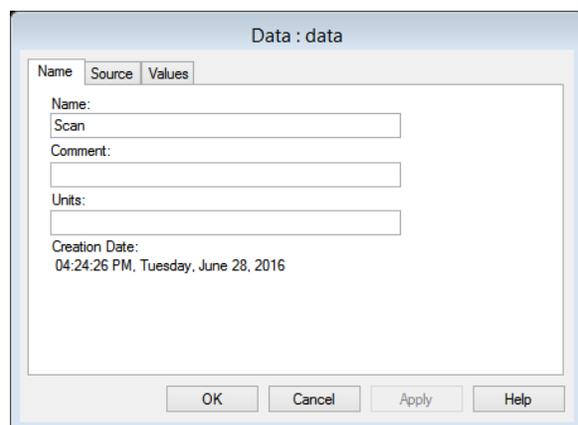
8.   Type **Scan** in the Name: field and then click **OK**.

9. Select the **Instrument** tab to open the instruments section.

10. Select the **+** or double-click anywhere in the Instrument window to open a new instrument dialogue.
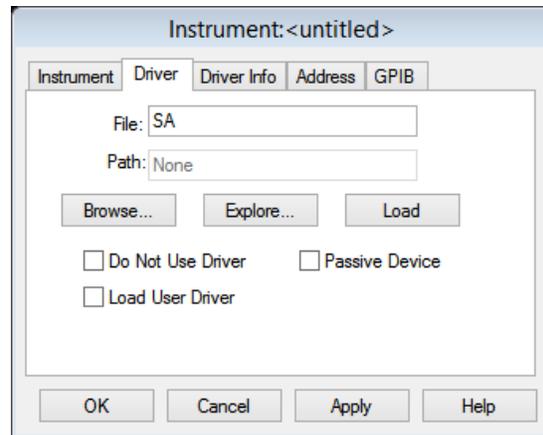
11. Type **SA** in the Name: field.

12. Click the **Driver** tab, and then click **Browse** or the **Explore** button.
(Explore allows selection of instrument drivers via categories or vendors. It also has a feature to find instruments on the bus, however, for the first use or after new installations, an instrument database must be created from the Tools selection on the toolbar.)

Select Build Instrument Database…

13. Type **ag_mxa.ins** in the file name field to find the Agilent MXA driver, and then click **Open**.
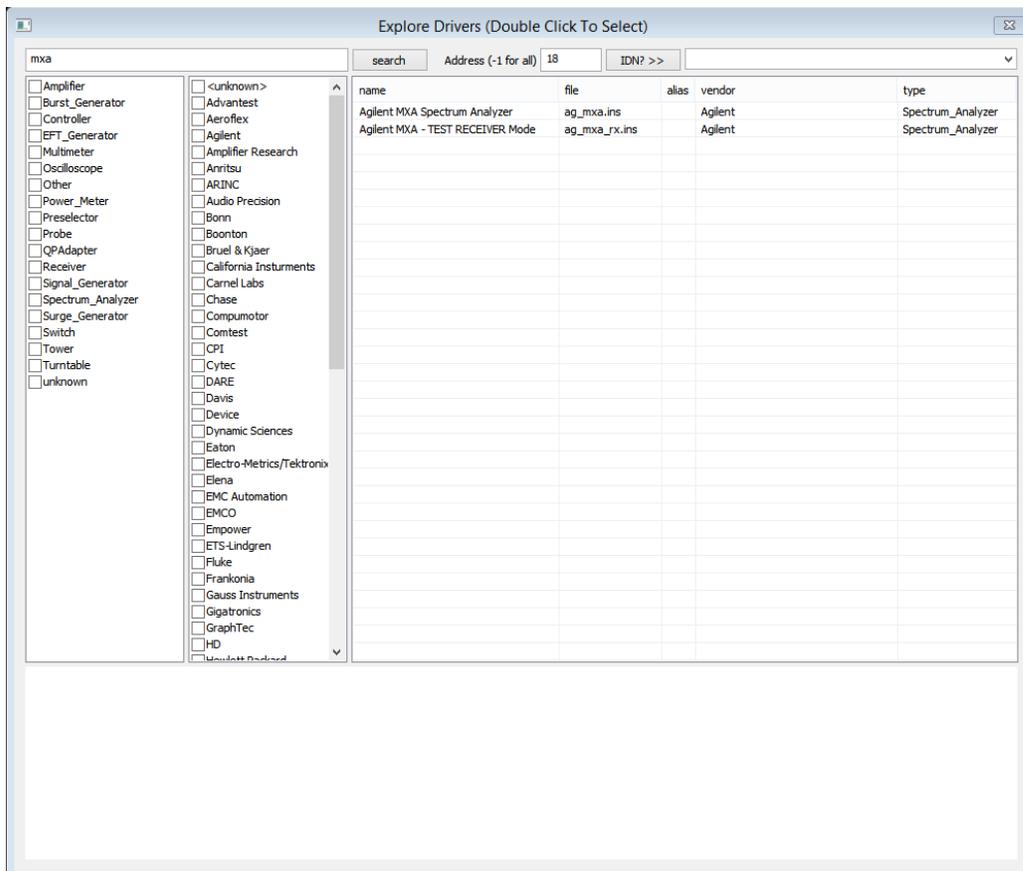
14. On the **Address** tab select **Simulation** from the Communication menu.
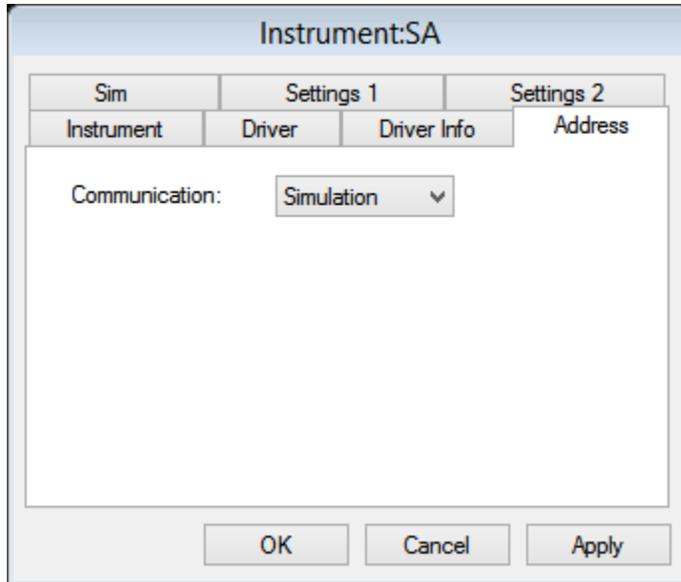
15. On the **Sim** tab select **Analyzer**, and then click **OK** to close the dialogue.

    The simulated analyzer will boot up.

16. On the simulated SA Server 1 panel, select **dBuV** for Data Out Units, and then select **Show VSG**.

    The Virtual Signal Generator will appear.

17. In the Frequency window type in **22e6** (22 MHz) and in the Amplitude window type **0.5**, and then click **OK**.

18. You will again edit the Scan action on the flowchart; on the **Data1** tab select **Scan** from the Measurement Data 1: menu.

19. On the **Instruments** tab select **SA** for the Receiver menu.

20. 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 right-most settings tab, as shown.

- Click the data element **Scan** to move it to the column on the right.

- Look at it tab that appears on the top of the interface and select the data element 'Scan' and ensure that the default color is other than white so it'll show up on the graph.

21. From the top toolbar click the Go arrow and then select **Scan** to execute the emission action you created.

Go arrow

The graph will show a red trace that emulates the signal from the Virtual Signal Generator.

The previous steps illustrate how to create a simple emissions profile from a blank sheet. To change to a real instrument, change the driver (if needed) and then change the communication menu to match the connection type; repeat step 21. If the instrument does not produce a trace, you may have to initialize it by dropping the INIT action onto the flowchart, selecting **SA**, and then executing the INIT action.

Flowchart Window

The Flowchart window is used for placing items in a visual manner and connecting them to create a flowchart.
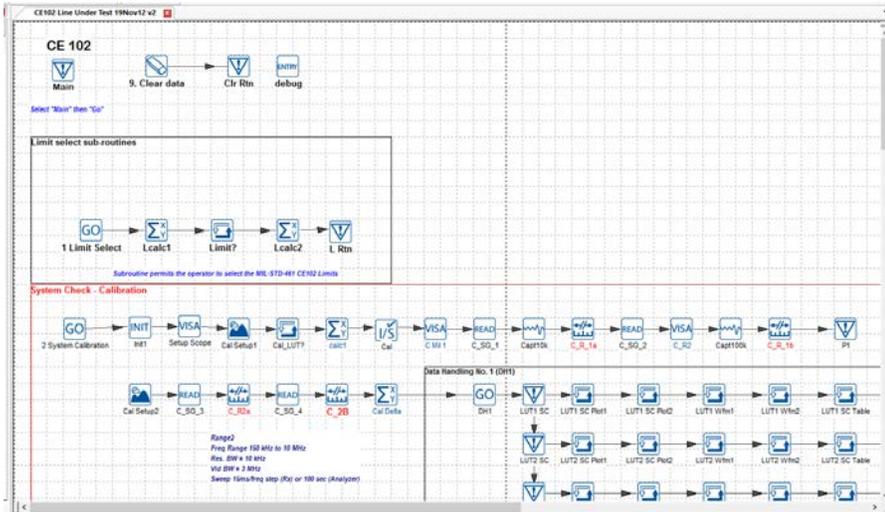
Palette

The palette is natively docked on the left side of the profile along with the project window. The palette can be re-sized by selecting and moving the edges. It can be undocked and moved if desired.

Data Elements

All data such as measurements, limits, correction factors and constants are in the Data Elements section.

Instruments

Instruments are listed with user defined names and attached to drivers that are selected from a list of hundreds of instruments.

Tables

TILE! tables now look more similar to a spreadsheet. The columns can be moved with the cursor instead of editing the order in the Display Data tab. As described in *Window Behavior* on page 15, tables can be moved during operation of test actions as needed.

Tables can be modified using the Edit icons on the toolbar or with the properties editor.

Graphs

As mentioned in *Window Behavior* on page 15, graphs can be moved during operation of test actions as needed.

Graphs can be modified using the Edit icons on the toolbar or with the properties editor.

Gnuplot

Gnuplot has been added as an open source plugin that will allow the user to expand the graphing of data in most any way desired. To enable the Gnuplot application: Select the ▮▮ icon in any graph window and then edit the properties (bottom of the Properties window) to change the type of graph.

New Plot

Like Gnuplot, we're experimenting with different graphical interfaces and New Plot can provide a way to interact with the data on the graph such as the ability to select points to add or delete from a list. Ranges can also be selected from here.

The icon for New Plot. ⊿

Polar

The Polar plot is designed to work with the Polar action **polar** to display results of directional data by interpreting the X axis as degrees instead of Hz. To enable the Polar plot select it from the graph toolbar ⊚. The Polar plot can be used to display other data in degrees and is not limited to the Polar action.

Views

Views can be saved as user favorites in the project view. This allows consistent layouts between uses, or users can save their favorite view.

5.0 Data Elements

Data Elements View

Data Element view is updated to allow moving, re-sizing, and sorting of columns. A **Values** column allows a glimpse into the numerical values of the elements.
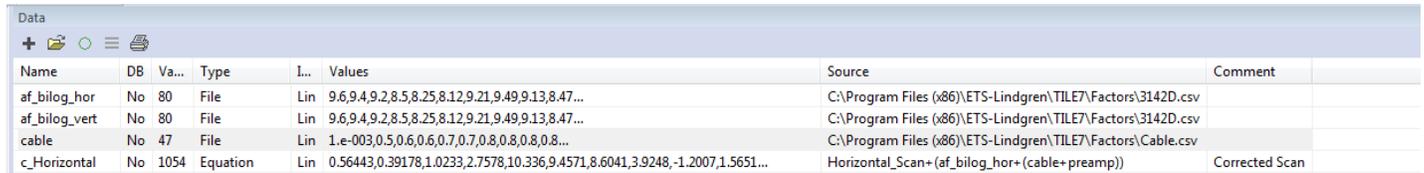
Name	DB	Va...	Type	I...	Values	Source	Comment
af_bilog_hor	No	80	File	Lin	9.6,9.4,9.2,8.5,8.25,8.12,9.21,9.49,9.13,8.47...	C:\Program Files (x86)\ETS-Lindgren\TILE7\Factors\3142D.csv	
af_bilog_vert	No	80	File	Lin	9.6,9.4,9.2,8.5,8.25,8.12,9.21,9.49,9.13,8.47...	C:\Program Files (x86)\ETS-Lindgren\TILE7\Factors\3142D.csv	
cable	No	47	File	Lin	1.e-003,0.5,0.6,0.6,0.7,0.7,0.8,0.8,0.8,0.8...	C:\Program Files (x86)\ETS-Lindgren\TILE7\Factors\Cable.csv	
c_Horizontal	No	1054	Equation	Lin	0.56443,0.39178,1.0233,2.7578,10.336,9.4571,8.6041,3.9248,-1.2007,1.5651...	Horizontal_Scan+(af_bilog_hor+(cable+preamp))	Corrected Scan

TILE! Data Elements

Each data element is an array of amplitude versus frequency values. These data elements may be 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.

See *Creating Data Elements* on page 2 to see how these elements are added to the profile.

Data Types

There are five types of data elements in the TILE! structure: measurements, file, equation, preset, and word. 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, and create arbitrary criteria with a few simples 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 with the test profile and can be retrieved for future tests. The ability to track not only the final number, but each of the intermediate corrections, provides a powerful archiving tool for any test laboratory.

To take advantage of the flexibility of this system, you must understand the different element types and how to intermix them for maximum efficiency.

Measurement Elements

The most common, and the default element type, is the Measurement, and it is defaulted to Continuous interpolation.

Typical examples of this would be a spectrum scan that connects each data point to create a trace. This is also used for any other data that will be displayed in a graph as continuous data.

If Continuous is not checked it is *not* interpolated and is now a discrete data element such as a quasi-peak.

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

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

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 be comma-delimited variables with column one a frequency component and column two a value. All white spaces are ignored. You must use commas (,) or semi-colons (;) as the delimiters. The size of the file is only restricted by the system parameters. You can create these tables quite easily in a text editor or most spreadsheets.

As illustrated, the frequency can either be in raw numerical values or in engineering terminology. The value (the second number) has no specific unit of measure. The unit of measure is a function of 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.

Importing this data into a data element allows us to use this information for graphing purposes, mathematical comparisons, and peak searches. Correction factors take the same form, a table of frequency and value points. Correction tables include antenna correction factor, amplifier gain tables, cable loss tables, and so on. 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.
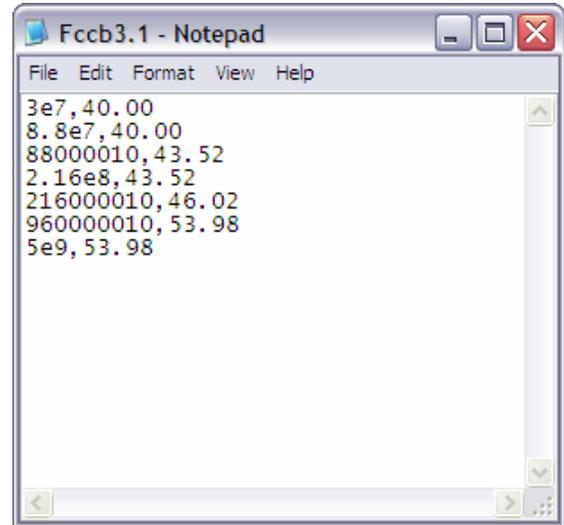
Equation Elements

In 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 some fairly complex mathematics.

The equation writer is the tool for defining an equation element. It is discussed in greater detail in *Equation Tab* on page 2. 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.
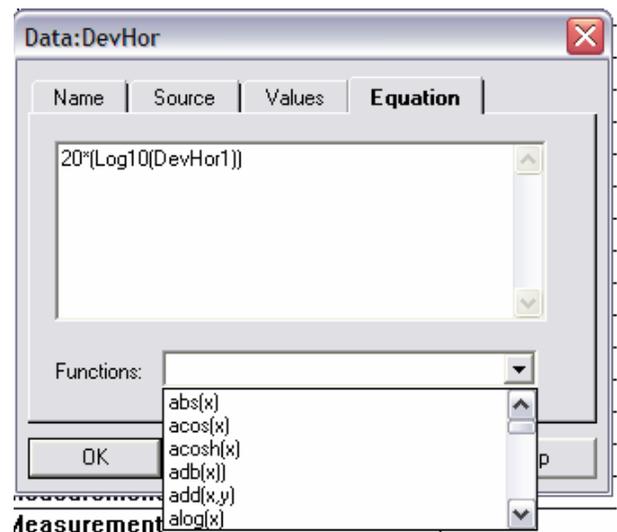
Preset Elements

The Preset data element is a value generator used to create simple data sets. 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, creating a data file for the calibration, and loading it through the file elements, preset allows you to quickly create a set of values across the frequency range. Once created, Preset data elements are treated just like any other data element in the system. For straight lines, you don't need the default 100 steps, so this can be changed to 1 step.

Once the values are entered, select 'generate' and view the results in the 'Values' tab.

This example shows how to create a limit line using bands.

#	start: freq	start: value	stop: freq	stop: value	# steps	step type	termination
1	10.00K	69.00	1.00M	109.00	95	Log	Continue
2	1.00M	109.00	30.00M	109.00	342	Log	Continue
3	30.00M	109.00	200.00M	101.00	381	Log	Continue

Word Elements

There are certain places within the TILE! system where it is necessary or possible 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 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-type 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 less 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 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-type data element from 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.

Creating Data Elements

To create a Data Element, find the Data window and select the '**+**' button.

The Data window displays all the currently-defined data elements. The first column indicates name; the second column identifies whether the database option is selected for this data element; the third column displays the number of valid points in the data element; the fourth column indicates the type of data element; the fifth column indicates whether this is an interpolated data element; the sixth column displays relevant information for the data type; and the seventh column allows room for any necessary comments.
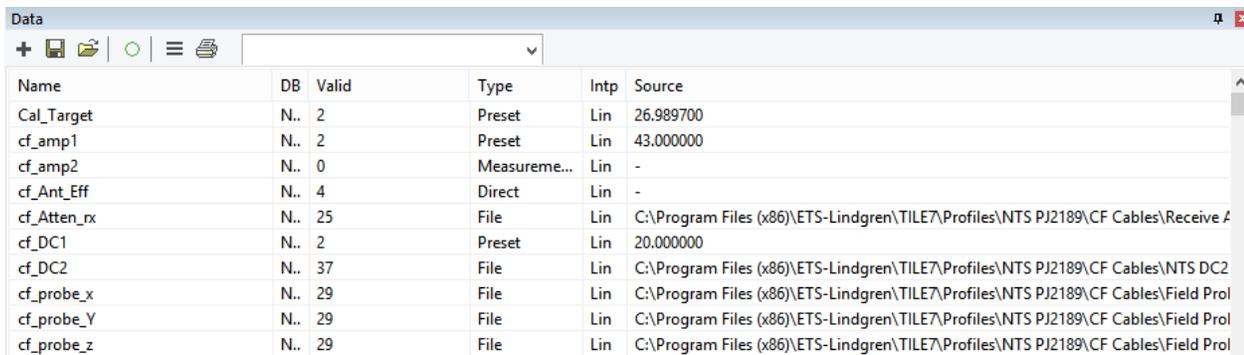
Name	DB	Valid	Type	Intp	Source
Cal_Target	N..	2	Preset	Lin	26.989700
cf_amp1	N..	2	Preset	Lin	43.000000
cf_amp2	N..	0	Measureme...	Lin	-
cf_Ant_Eff	N..	4	Direct	Lin	-
cf_Atten_rx	N..	25	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\Receive A
cf_DC1	N..	2	Preset	Lin	20.000000
cf_DC2	N..	37	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\NTS DC2
cf_probe_x	N..	29	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\Field Prol
cf_probe_Y	N..	29	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\Field Prol
cf_probe_z	N..	29	File	Lin	C:\Program Files (x86)\ETS-Lindgren\TILE7\Profiles\NTS PJ2189\CF Cables\Field Prol

Name Tab

The first tab for a data element has three features. The first is a name. Type a unique data identification name 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.
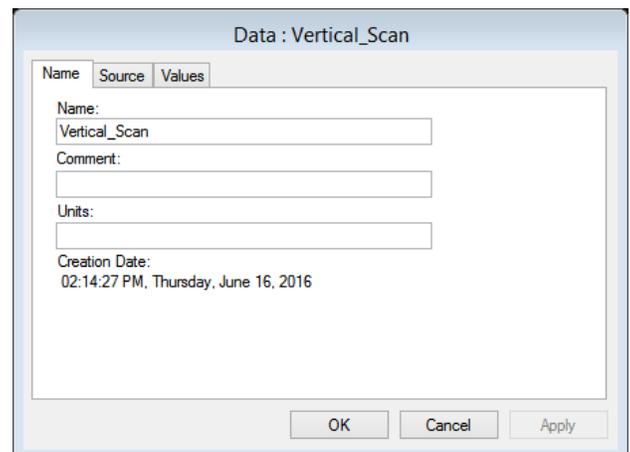
Data : Vertical_Scan

Name | Source | Values

Name:
Vertical_Scan

Comment:

Units:

Creation Date:
02:14:27 PM, Thursday, June 16, 2016

OK | Cancel | Apply

The second feature is the date/time stamp for the data element. This time will reflect the date that the data was last changed or updated. For file-type data elements it will reflect the date/time stamp for the file when this data element was created or last 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**.

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

Units: Will be shown if this 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.

Source Tab

The **Source** tab defines the data elements. There are four choices that need to be made: selecting the data type, determining the interpolation methodology, 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 the **Measurement** option is selected, the only option (tab) available is **Values**; Measurement data elements only show a **Values** tab. When first configured, the values are blank since no measurement has been performed. Select the check box to select a type; for example, **File**. When you select a data type other than measurement, additional tabs are available.

- **Continuous (Log and Linear)**—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. If you are creating data elements that by nature should be discrete (peaks, over limit points, and so on), then deselect these boxes to turn off the Continuous option. In effect there are three choices, either turned on or both turned off.

  NOTE: 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.

- **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.

- **Auto Sort Low/High**—This check box determines whether data is automatically sorted when it is loaded for the first time. This only has meaning for the file-type data elements when they are read from file the first time. You can change the sorting routine from the **Values** tab with a slight difference. With Auto Sort off, data from file will be read in the sequence it is in the file.

- **IO Type**—This an advanced feature used with the Scripting Functions.

See *Data Types* above for more information on the types of data available on this tab.

Values Tab

- **Value Initialization**—The data from the file is read in one of two ways. First, when you accept the settings (click **OK**) the data file is automatically read into the data element. The second method requires moving to the **Values** tab and clicking **Initialize**. On the Data 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.
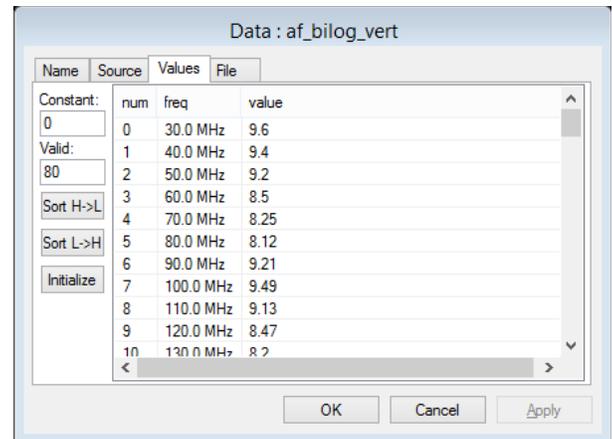
- **Sorting Data Options**—On the **Values** tab you have two choices for sorting the data. Normally, data within TILE! is sorted from low frequency to high frequency. It is important that the data be in one of these two structures. Data that is not sorted will cause strange results during math operations since the interpolation functions looks to the frequency before and after for a valid range.

  Clicking the **Sort H->L** will cause the data list to be sorted starting at the highest frequency and sequencing down to the lowest frequency. The **Sort L->H** tab does the opposite, sorting data starting at the lowest frequency, and sequencing to the highest frequency.

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

File Tab

- **File Name/Directory Path**—Enter the name of the file in the **File** box. To properly identify the file, include the file descriptor, such as .dat or .txt. The same applies to the **Path** box. When you enter the directory path it must be identical to the physical location on disk. If using a network drive it is best to map the network driver to a local drive letter before setting the path.

  You may click the **Browse** button and find the data file. Once selected, the file and path will be filled in automatically.
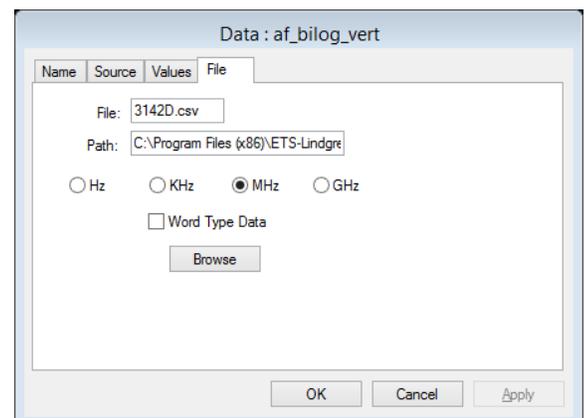
- **File**—To make creating files as easy as possible we use comma-separated variable (.csv) files or text files (.txt) for loading or saving data.

  This format is limited to a two-column format. For our uses the first value is frequency and the second value is amplitude. When looking at the file with a text editor, these two values would be separated by a comma. Within a spreadsheet program make sure that you set the file type to .csv text when doing a save or save as. 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 this function. The first time you are importing data, check this tab to confirm it is imported as text. Be careful 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

The **Equation** tab is an equation writer in which you can enter a formula to define a data element. You can enter a fairly complex equation, but you are limited to nine imbedded parentheses. Each parenthesis must delineate a binary equation. The example here 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.
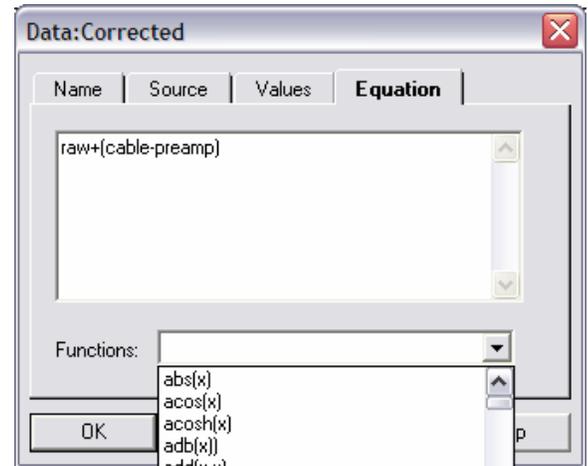
The results of any math operation are stored into the named data element. Until the equation has been executed, no values are present. The **Values** tab will serve no purpose at this point. After execution, you can use the **Values** tab to view the data element values.

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 *Appendix A* on page 278 for a list of math functions.

Preset Data

Ampl. Tab

The **Ampl.** tab appears when the **Preset** option is selected on 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.

- **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).
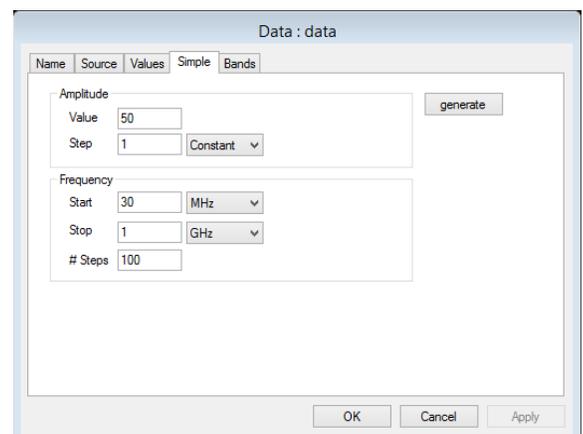
- **Constant**—Constant is the default setting for preset values. When this is selected the value in the text table is filled into a table across the frequency range defined on the **Frequency** tab.

- **Seq Add**—Takes the starting value in the **Value** box and sequentially adds the value shown in **Step**. The number of steps is set on the **Frequency** tab. If the starting value was 20, the step was a value of 2, and there were 10 steps from 100 MHz to 1000 MHz, the example table would be defined. Click the check box to select this option.

- **Seq Mult**—Takes the starting value from the **Value** box and multiplies the value by the **Step** at each frequency. With a starting value of 20, a step value of 2 and 10 steps between 100 MHz and 1000 MHz we would see the values on the example table.

- **Generate**—Will create the data, this can be reviewed in the Values tab.

- **Bands**—Will generate a complex data element to include linear and log sections. This could be handy to create limit lines. This can also be used to create a mask type data that can be used in the graph to hide breaks in data.

Element Names

Each data element must be given a unique name. The name can contain upper and lower case characters as well as most other keys. Although you can enter spaces, such as *Antenna Factor*, **it is highly recommended that you do not use spaces**. A space in the name will make it impossible to use this element in an equation. 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.

As a matter of convenience, all the TILE! samples use the following standard conventions:

	Data : data						
Name	Source	Values	Simple	**Bands**			
+		-		^	v		generate
#	start: freq	start: value	stop: freq	stop: value	# steps	step type	termination
1	30.00M	10.00	40.00M	10.00	1	Lin	Continue
2	40.00M	0.00	50.00M	0.00	1	Lin	Break
3	50.00M	20.00	60.00M	20.00	1	Lin	Continue

OK Cancel Apply

- **cfa_**   :   Precedes all correction factors

- **eq_**   :   Precedes all equations
  For example, **eq_Pks** would be the equation to find peaks in an EMC Scan.

Most lists that utilize data elements will automatically sort alphabetically. Keep this in mind when creating data element names.

Element Database in Use

This column is only meaningful if the TILE/DB database option is installed. If this option is installed you can specify on the source tab whether to update the database for this data element when doing a save. If it is marked for saving, the data element is copied to the database table of results.
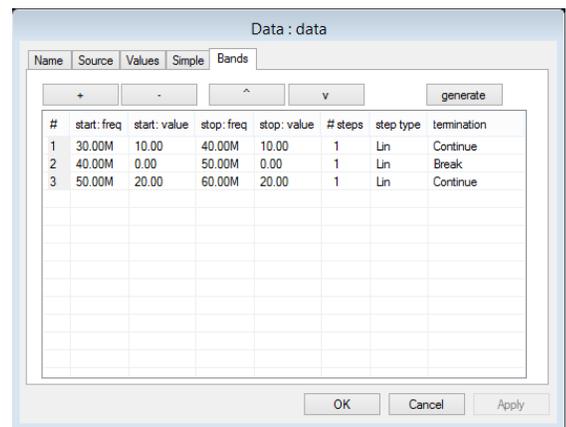
Valid Points

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. This is discussed in *Adding Data Elements* on page 39. The **Measurement**, **Equation**, and **Word** types will not display any valid points until the measurement/equation has executed. The **File** and **Preset** types will display valid points as soon as they are defined.

Intp (Interpolated)

There are two interpolation selections: *Continuous (Linear) and Continuous (Log).* In the Data elements section, the Intp will indicate 'Lin' of 'Log'. Linear is the default and Log is typically only used for limit lines with log slopes so that any math will properly interpolate the value between the start and stop frequency of the slope. The default value is Continuous (Linear).

When EMC measurements are performed we 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. *Interpolated* 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.

For discrete type data this will show '**No**' in the Intp column. A typical discrete data element is the QP or any data that would use a marker on a graph. If you choose to graph a set of peak signals they will appear as a line if **Interpolate** is turned on. With the switch off, they would see discrete points which would require a marker to see on the graph.

Discrete data in a table along with associated interpolated data will may show artifact type interpolated data. There is a selection in the table to 'Show All' that will interpolate these points if selected, or will only show these points at the correct frequencies if it is un-checked.

Source

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

Editing TILE! Data Elements

There are two methods to edit data elements after their creation. Bring the Data window to the front focus. Use the cursor or the arrow keys to select a data element. Clicking the **Edit** command on the Windows menu bar provides the options to add, delete, edit, or initialize a selected data element.

Click the selected data element with the cursor to open the dialog box for editing.

Deleting TILE! Data Elements

There are two methods to delete data elements after their creation. Bring the Data window to the front focus. Use the cursor or the arrow keys to select a data element. Click once on a data element to highlight it. Press the **Delete** key or select **Edit/Delete** from the Windows menu bar to delete the highlighted data element. Deleting permanently removes an existing data element.

Math Functions

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.

Math icon

To do this, drag the **Math** icon to the flowchart where the equation needs to occur. Click and edit the action. You can name it on the **Action** tab.

The second tab is **Data**. Click this tab and a two-column selection window is presented. Select the math data that you want to calculate. This will cause the math equation to be executed at the appropriate time. You can see in the following table that it will calculate and add valid data points to the data elements that were calculated.
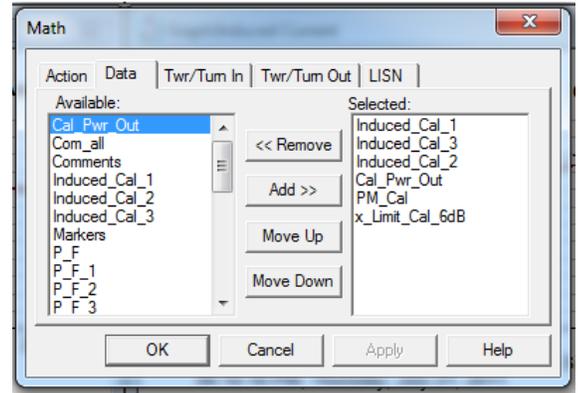
Induced_Cal_1	No	50	Equ	Lin	PM_Cal_FB_1+73	73=107-34 Where 107=0dBm and 34 is 20*log(50 ohms) converting dBm to dBuA
Induced_Cal_2	No	37	Equ	Lin	PM_Cal_FB_2+73	
Induced_Cal_3	No	21	Equ	Lin	PM_Cal_FB_3+73	

Open the data element, select the **Values** tab and look at the information.

See *Appendix A* on page 28181 for a list of math functions.

Adding 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. In reality, 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.

Instrument Window

Name	Module	Type	Bd	Addr	Driver	Version	Serial Number	Cal Date	VISA
SA	Agilent MXA Spectrum Analyzer	Spectrum Analyzer	Sim		ag_mxa.ins	7.1.3.53	3325A00106	05/25/00	TCPIP0::localhost::6370::SOCKET
RX	Agilent MXA - TEST RECEIVER Mode	Spectrum Analyzer	Sim		ag_mxa_rx.ins	7.1.3.53	Unknown	Unknown	TCPIP0::localhost::6371::SOCKET
Tower	EMCO 2090 Controller	Controller	Sim		EMCO2090.ins	7.1.3.53	Unknown	Unknown	TCPIP0::localhost::6385::SOCKET
Turntable	EMCO 2090 Controller	Controller	Sim		EMCO2090.ins	7.1.3.53	Unknown	Unknown	TCPIP0::localhost::6380::SOCKET

The Instrument window provides visual access to the currently defined instruments. The displayed elements are the instrument name, type, board, address, instrument library name (instrument driver link), 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, move the Instrument window to the front. You can then click the **+** on the Instrument window or use the **New Instrument** icon to open the **New Instrument** dialog box.

New Instrument icon

Enter a unique name on the **Instrument** tab and then select the **Driver** tab**. Do not press the Enter key**. This will attempt to save the instrument without proper definitions of the instrument driver and GPIB information, and will generate an error.

Driver Tab

The **Driver** tab has two boxes, **File** and **Path**; three buttons, **Browse, Explore** and **Load**; and three check boxes, **Do Not Use Driver**, **Passive Device and Load User Driver**.

- **File and Path**—Although it is possible to enter **File** and **Path** directly, the best method is to use the **Browse** or **Explore** buttons, which will ensure the correct path for the instrument driver files. Click **Browse** for an **Open File** dialog box. The .ins files will be pre-selected. If the directory is not correct, navigate to the correct directory. Generally, instrument drivers exist for each manufacturer model number. 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. Find the driver for the model instrument you are defining. Click to select the driver. From the keyboard, you may use the arrow keys to highlight the desired file and then press the **Enter** key.

For more information on the Instrument Explorer see the section above 'Steps to create a Profile'.

- **Do Not Use Driver**—Defines an instrument which does not have an instrument driver. These types of instruments can be controlled with the GPIB Control action as described in *Instrument Commands* on page 2. 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**.

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**. Change these settings to ensure the instrument does not conflict with other instrument settings.

- **Board**—The arrow on the **Board** box will display the available settings. The TILE! system supports multiple GPIB boards, serial, VXi, 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. Select the setting appropriate for the instrument by using the arrow key to highlight the selection and the **Tab** key to select an item, or by clicking the selection.

- **Primary**—The specific hardware address of the instrument relative to the board selected needs to be entered in the **Primary** box. Click the arrow and select the appropriate number.

   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.

Setup Tab

Setup is the active tab if a GPIB or VXi board is chosen on the **Address** tab. On this tab are settings which control the communications between the computer and the instrument. The default settings are appropriate for all instruments meeting the IEEE-488-2 standard. 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, and this 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.

   A strong indication of I/O incompatibility is 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 settings refer 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.

Setup Tab–Other GPIB Bus Settings

The following settings are specific to differences between the IEEE-488-1 and IEEE-488-2 standards.

- **Terminate Read on EOS (End of String)**—The **Terminate Read on EOS** check box allows the software to terminate a read operation when it receives the EOS byte. The default setting for this option is disabled.

- **Set EOI (End of Instruction) with EOS (End of String) on Write**—The **Set EOI with EOS on Write** check box allows the software to assert the GPIB EOI line whenever it sends the EOS byte. The default setting for this option is disabled.

- **Send EOI (End of Instruction) at end of Write**—The **Send EOI at end of Write** check box allows the software to assert the EOI line at the end of each data transfer. The default setting for this option is enabled.

- **8 bit EOS (End of String) Compare**—The **8 bit EOS Compare** 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 disabled.

- **Repeat Addressing**—The **EOS Byte** box has a range of 0 to 255 and is used by the software for all EOS operations. The default setting is to turn this on (check box selected). There are no known times when this should not be on.

- **EOS (End of String) Byte (Binary)**—The **Repeat Addressing** 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 disabled.
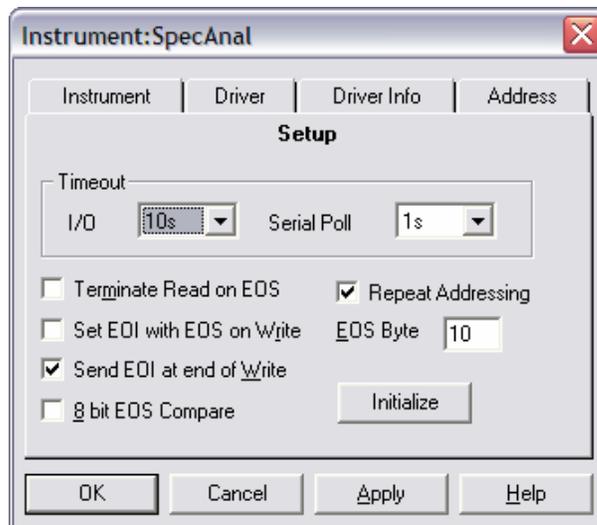
Serial Tab

When the **Serial** option is selected on the **Address** tab, the tab references change to **Serial**. These boxes control the four general conditions of the serial bus: baud rate, data length, parity, and stop bits.

Once you make the appropriate choices on each of the four tabs, click **OK** or press **Enter** to accept. **Cancel** will cancel the definition and exit the dialog box.
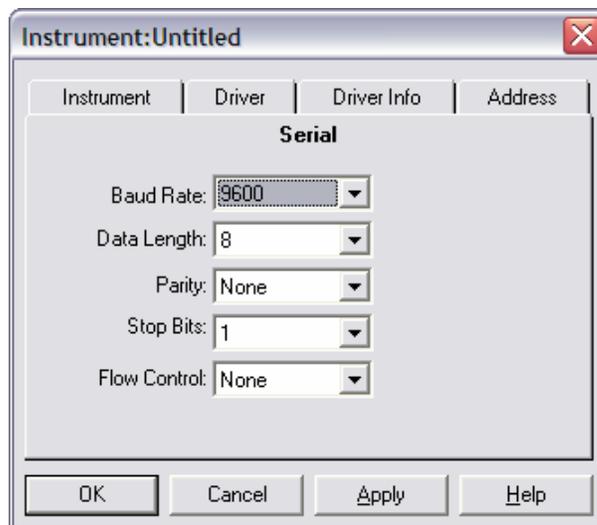
- **Baud Rate**—The **Baud Rate** has nine available settings ranging from 300 to 115,200. See the instrument manual to determine the correct setting. Click the arrow and select from the list.

- **Data Length**—There are four **Data Length** settings: 5, 6, 7, and 8. See the instrument manual to determine the correct settings. Click the arrow and select from the list.

- **Parity**—There are five **Parity** levels: None, Odd, Even, Space, and Mark. See the instrument manual to determine the correct setting. Click the arrow and select from the list.

- **Stop Bits**—The **Stop Bits** can be set to 1 or 2. See the instrument manual to determine the correct setting. Click the arrow and select from the list.

- **Flow Control**—There are three kinds of flow control: None, no flow control; software flow control (**X On-X Off**); and hardware control (**DTR**). The choice of flow control depends on the equipment supplier.

- **Accepting the choices**—Once you have made the appropriate choices on each of the four tabs, click **OK** or press **Enter** to accept. The **Cancel** button will cancel the definition and exit the dialog box.

Visa Tab

VISA will allow communications to instruments connected to the computer via USB or TCP/IP. When the VISA interface is selected, the **VISA** tab will pop up with the following selections.

- **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.

- **VISA Resource Name or Alias**—This is a list of all instruments available in the VISA interface.

- **Validate**—Once the instrument is selected, pushing this button will validate the connection to the device and pop up a Validation Successful window.
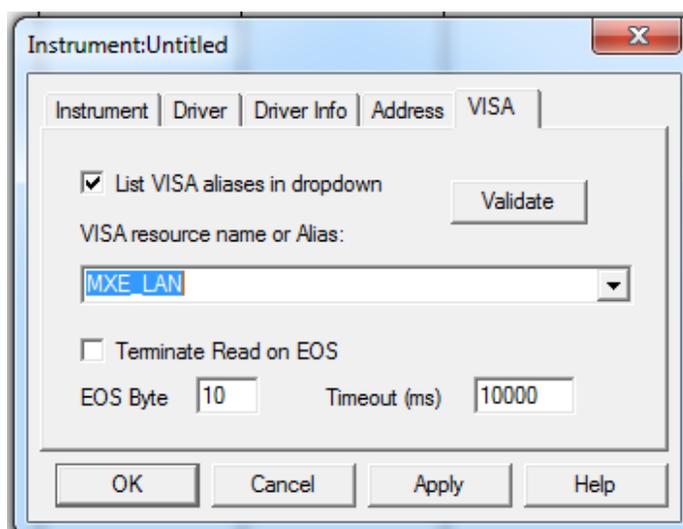
- **Terminate Read on EOS (End of String)**—The **Terminate Read on EOS** check box allows the software to terminate a read operation when it receives the EOS byte. The default setting for this option is disabled.

- **EOS (End of String) Byte (Binary)**—The **Repeat Addressing** 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 disabled.

Editing an Instrument

You can edit an existing instrument by bringing the Instrument window to the current focus. Click to highlight the instrument in question and click to begin editing.

You also can use the arrow keys to highlight the instrument and use the **Edit/Edit** functions from the Windows menu bar.

Using the cursor, click once on an instrument to highlight it. Press the **Delete** key to delete the highlighted instrument.

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.

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 this test. A flowchart defines a complete test program. This can be a simple test, such as one which performs an emission scan for a specific international standard. But 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.

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 chapters for ancillary, emissions, or immunity.

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 Icons

Each icon can be selected and dropped onto the flowchart during the design stage. Click once on the desired icon on the palette. 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.

Editing Icons
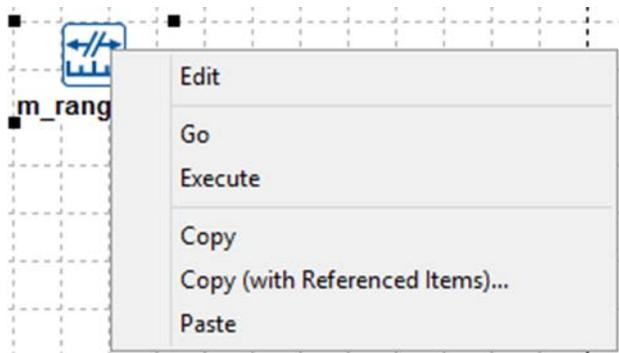
- **Edit**—Reveal the particular icons settings
- **Go**—Run this icon as well as any sequenced after it via connectors
- **Execute**—Run this icon only
- **Copy**—Copies this icon and its settings to be pasted. The pasted copy will be re-named with a sequential number after the original name
- **Copy (With Referenced Items)**—Copies this icon 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)

Linking Actions

Once you have defined a group of actions on the flowchart, you next link them together to form a common chain of test steps.

The **Connect Tool** icon (from the palette) is used to connect one action to the next. Select this tool, position the cursor over the first action, and click once. The single click is defined as an anchor, and is used to attach the first anchor at the starting position. Drag the cursor to the next action to drag a connecting line between the two actions. Double-clicking will attach the endpoint to the second action.

Connect Tool icon

You can only attach two actions together in one operation. To attach multiple actions, you must attach pairs together and then duplicate the steps for the remaining actions. An action can have multiple incoming attachments, but only one outgoing attachment (with the exception of the Prompt action).

Aligning 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.

Two components are available under **Structure**: **Group** and **Align**. **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 it will automatically be scaled to fit on a standard page.

Working with the Flowchart

The flowchart allows you to work with graphical tools to enhance the appearance or efficiency of the test profile. The imbedding of graphics on 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.

Creating a sequence of icons to perform an emissions test is the heart of the TILE! system.

But adding a title to the flowchart makes it obvious to the user what is being accomplished in the test profile. Imbedding a bitmap picture of the test setup could be even more descriptive.

Graphs

Graphs are created to display data in frequency vs amplitude (X/Y) 2D format. Either axis can be displayed with linear or logarithmic scales. Lines can be one of 20 colors with 5 formats such as solid/Dash/Dot.

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.

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.

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.

Creating a Graph

A graph is created from the Project window by selecting Graphs, right click and Add.

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, with log frequency scaling with the amplitude set to 0-100.

The icons on the graph toolbar has the following functions:

1. Plus symbol opens the Display Data window. The Display Data window provides a list of available data elements to display on the graph and provides the interface to select the color and format for it. Due to the evolution of the software this is duplicated with the right-most three horizontal lines. Further, data can simply be dragged and dropped onto the graph though default color will be assigned so either of these buttons will open the interface to change the data attributes.
2. The folder icon is not functional and reserved for future functions
3. The 'Floppy' icon will open a file explorer to save the graph to a specific location

4. The green circle 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
5. The left-most 3 horizontal lines open the graph Setup that is used to set all the parameters of the graph
6. The Printer icon will open the print interface
7. The copy icon will allow a copy of the graph to be placed in the past buffer
8. The bar graph icon will open the GNU plot function that is under development
9. The small graph icon will open the 'New' graph which is currently under development to allow user selectable points or sections
10. The Polar button will change the graph to a polar plot. This can be used with the Polar action that can be used to draw azimuth centric data

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.

88.2 MHz	+50.000000

Data Options

There are two options when selecting data elements: which elements to use, and how to display them. Once you have selected a data element, a new tab appears behind the definition page. For every data element selected a tab with the data element name will be generated. The sequence of the tabs is the same sequence as the list on the selection page. The setup of these tabs is identical for both graphs and tables. Certain features have no effect for graphs.

Selecting Data

When **Display-Data** is selected from the Windows menu bar, a dialog box is opened with an **Available** column and a **Selected** column.

- **Add**—To select a data element, click the available element, automatically adding it to the **Selected** side. You also can use the up/down arrow keys to highlight the data element and use the **Tab** key to move to the **Add** button. Press **Add** to register the selection.
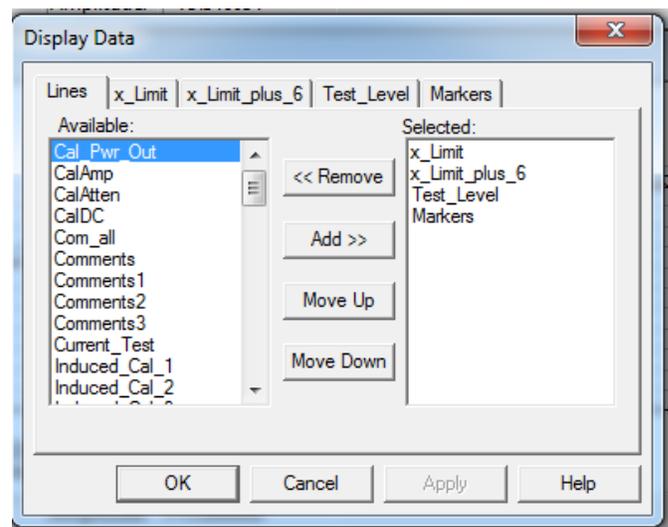
- **Remove**—To remove a selected data element, click the element in the **Selected** column. This will remove it from the list. You also can use the up/down arrow keys to highlight the data element and use the **Tab** key to move to the **Remove** button. Press **Remove** to register the selection.

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). See *Creating Data Elements* on page 33 for more information.

  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 for the specification limit, or a different color, you can easily distinguish this line from the readings.

  Style—There are five line styles that can be chosen. The default setting is solid. Click the arrow to change the selection.

  Color—There are 20 colors that can be selected. The default setting is red. Click the arrow to change the selection.

  Width—The default width is 1. Make this number larger to thicken the line. This setting does not work with all line styles.

- **Show All**—When used on a table, the **Show All** check box controls whether all 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 the normal setting, with 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. See *Creating Data Elements* on page 33 for more information.

  The default setting is none. This 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 in excess of the specification limits on the same chart as the readings. By using a triangular point for the points over specification limit, or a different color, you can easily distinguish those points from the readings.

- **Shape**—There are nine shapes (10 if you count **None**) that can be assigned to a non-continuous data element. The default shape is none. It is important to set a shape or nothing will appear on the graph. Click the arrow to select or change a shape.

- **Color**—There are 20 possible colors. The default is red. Click the arrow to select or change a color choice.

- **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.

- **Column Width**—When defining a Table data element, this field controls the width of the column. The default value is 10.

- **Decimal Places**—When defining a Table data element, this field 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.

- **1st Description**—When defining a table or a graph, this field allows this user to control the visible name for this data element. The default is the data element name.

- **2nd Description**—On a table there are two rows of description for each data element. This field controls the second field. You might use this to display units, dBuV/m, for the reading as a clarifying label.

Display Options

The display option allows you to configure the name, descriptive titling, graph size, axis scaling, and axial labeling. The display option includes the title name for the graph. A unique name is essential when you need to automatically printing or accessing the graph from different applications.

With the graph in the front focus, click **Display/Options** on the Windows menu bar to access **Graph Setup**. You can also click the graph to open **Display Options**.

Graph Setup has thirteen tabs. These control labels, titles, scaling, fonts, legends, default page formats, and special features such as additional information, EUT, comment, and bitmaps.

Graph Setup Tabs

Titles Tab

The **Title** box is for the graph name. As with all TILE! actions and tables, the graphs should have a unique name.

Since you can create multiple graphs, unique names help to keep track of the different graphs.

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.

If you use the **Display/Copy Graph** function the name will be changed to *originalname1* (where the number will be incremented as additional graphs are created). Make sure you change the name of the graph after copying.

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.
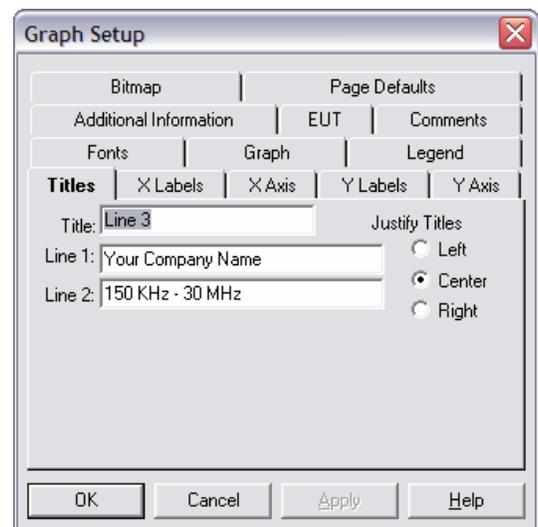
X Labels Tab

The x-axis labels include 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 setting is Frequency. If you want to change the title to indicate units, change this field to **Frequency (MHz)**, for example.

- **Frequency Range**—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. The frequency range can be overridden by using the zoom option. 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**—In this tab there are three numerical formats, as well as log/linear scaling and position information. **Engineering**, **Decimal**, and **Scientific** refer to how the number is displayed. The default is Scientific, which displays in kHz, MHz, and GHz. One megahertz would display as 1 MHz. **Decimal** is raw format in which 1 MHz would display as 1000000. **Scientific** would display 1 MHz as 1e6.

- **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.

- **Show Last**—Indicates whether the last data point will be displayed on the chart, regardless of scaling when in a zoom mode.

- **Places**—Refers to the number of relevant decimal places to be displayed on the x-axis scale: 1 place is 32.1, 2 places is 32.11.

- **Axis/Plot**—**Above Axis**, **Below Axis**, **Above Plot**, and **Below Plot** refer to the position of the x-axis label and ticks with regard to the graph and overall plotting area.

The **X Axis** 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.
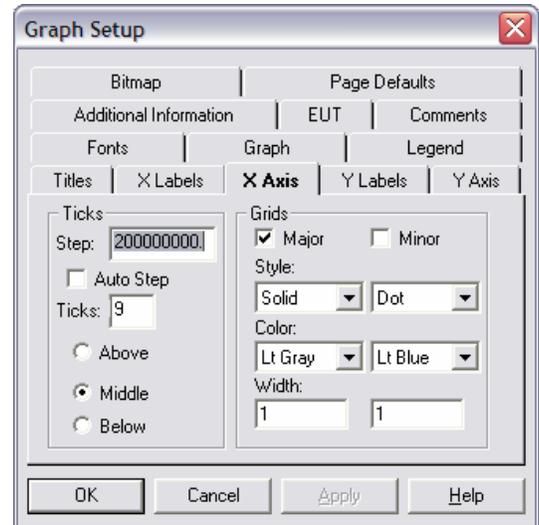
- **Auto Step**—When this check box is selected, the vertical scale will be evenly divided into 10 steps.

- **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/Minor**—Check each of these boxes to turn on or off the appearance of grid lines. If turned off, no grid lines appear for the grid.

- **Style**—You can set the style to solid, dot, dashed, alternating dash-dot, and alternating dash-dot-dot.

- **Color**—The available colors are the standard 20 colors.

- **Width**—Configures the size of the line. It will make the grids thicker or thinner, the minimum (and default) is 1.We suggest the user experiment with these settings to better understand how they work.

Y Labels Tab

The y-axis labels include 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. If you wanted to change the title to indicate units, for example, 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. The amplitude range can be overridden by using the zoom option under **Display** on the Windows menu bar.

- **Format**—General settings for the scaling of the graph.

- **Auto Scale**—This is disabled.

- **Engineering/Decimal/Scientific**—In this tab there are three numerical formats, as well as log/linear scaling and position information. **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** will display the raw format in which 1 MHz would display as 1000000. **Scientific** would display 1 MHz as 1e6.

- **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.

- **Show Last**—Indicates whether the last data point will be displayed on the chart, regardless of scaling when in a zoom mode.

- **Places**—Refers to the number of relevant decimal places to be displayed on the x-axis scale: 1 place is 32.1, 2 places is 32.11.

- **Axis/Plot**—**Left Axis**, **Right Axis**, **Left Plot**, and **Right Plot** refer to the position of the x-axis label and ticks with regard to the graph and overall plotting area.
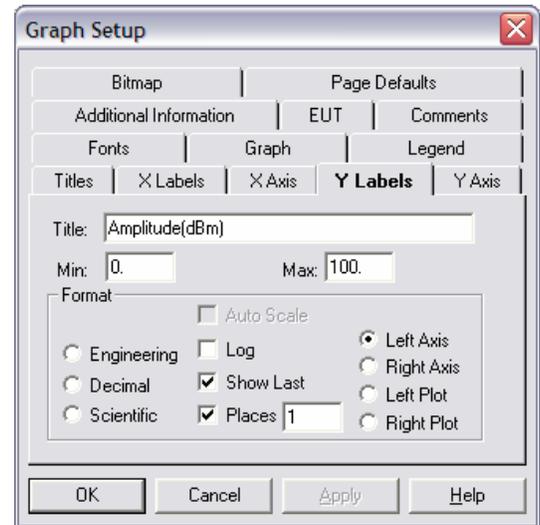
The **Y Axis** tab controls the ticks and grid line colors for the y-axis. These can be configured to suit the zoom range of the y-axis. The tick scaling is only active when the log mode is turned off. In creating a tick scaling on the y-axis, the range needs to match the desired amplitude range.

- **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 frequency range. If the log scaling is turned off, the default grid ticking will write a tick mark at every 10 points. If the scaling is zoomed to 1 to 10 (for example, in looking at probe values in a 3 V/M test), change the tick marking to 1 to make the vertical scale 1 V/M steps.

- **Auto Step**—When this check box is selected, the vertical scale will be evenly divided into 10 steps.

- **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 3 small tick marks on the axis between 0 and 0, between 10 and 20, and so on.

- **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/Minor**—Check each of these boxes to turn on or off the appearance of grid lines. If turned off, no grid lines appear for the grid.

- **Style**—You can set the style to solid, dot, dashed, alternating dash-dot and alternating dash-dot-dot.

- **Color**—The colors available are the standard 20 colors.

- **Width**—Configures the size of the line. This will make the grids thicker or thinner, the minimum (and default) is 1.We suggest the user experiment with these settings to better understand how they work.
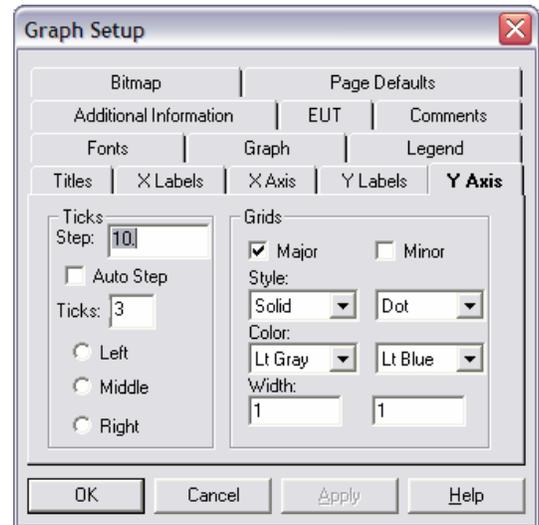
Fonts Tab

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: Arial, Courier New, Script, Times New Roman, and Decorative. If these fonts are not installed, a close font is the default.

- **Title-Line 1**—This font is only used on the Line 1. The default is an Arial font with a 10 pitch font size. 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 Title**—This font is used on the graph title as well as the x- and y-axis titles. The default is Arial with an 8 pitch font.
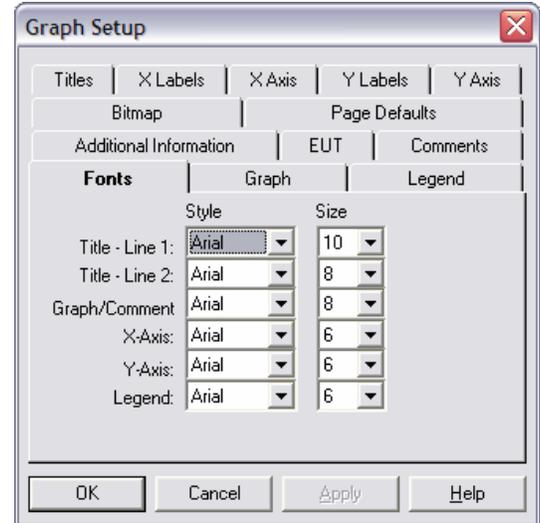
- **X-Axis**—This font is used to set the size of the x-axis labels (and tick indicators). The default is Arial with a 6 pitch font.

- **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 *Additional Information Tab* on page 2). The default is Arial with a 6 pitch font.

Graph Tab

The **Graph** tab allows you define the relative size and area of the graph, the plotting area, and the area for text on the graph.
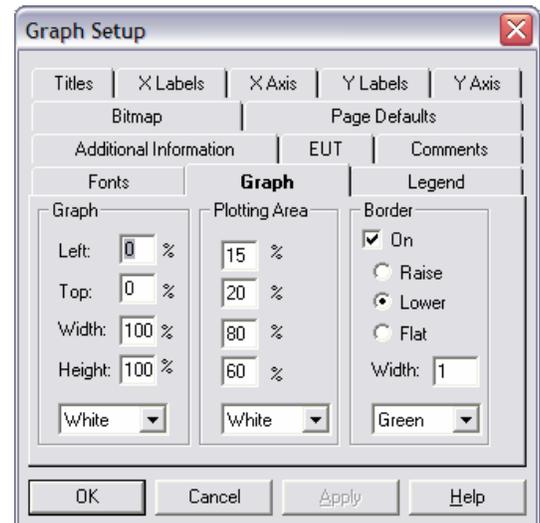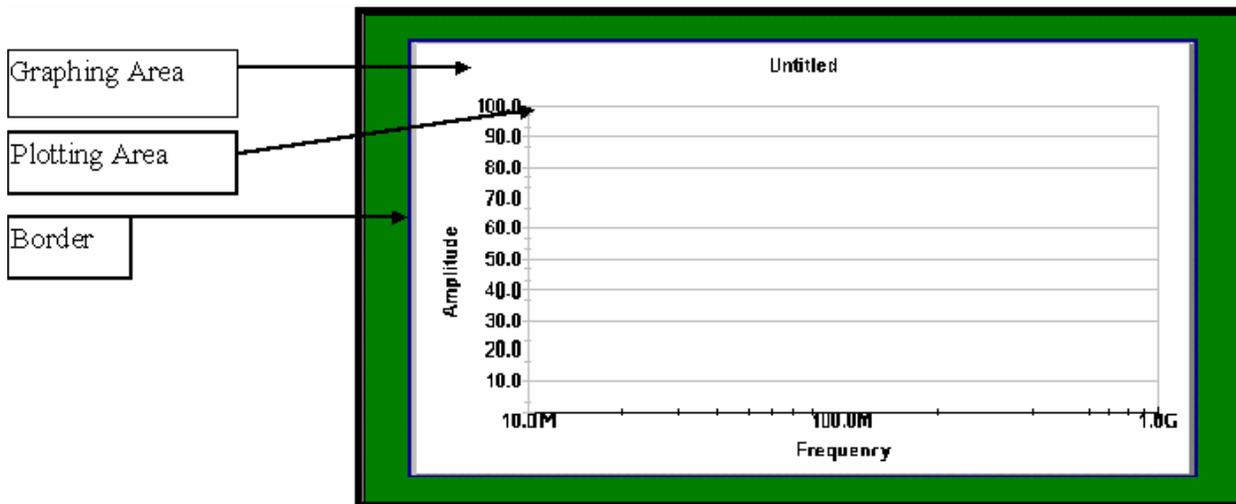
- **Graph**—The overall graphing area is defined by stating how far down from the top of the graph page the graphing area begins. This is stated in percentage of the area. The area of the page outside the graph is defined as the border area. The color of the border can be set as needed. One common use 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.

- **Left**—Represents 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 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 width of the graph as a percentage of the page size. The default value is 100%.

- **Height**—This value represents height of the graph as a percentage of the page size. The default value is 100%.

- **Color**—You can change the graph background color by using the list next to the color setting to view the available colors (the standard 20 colors).

- **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 will include the X-Y presentation of data. All distances are stated 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. Choose a size that fits with the data to be displayed.

- **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. Choose a size that fits with the data to be displayed.

- **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**—Change the color by using the list next to view the available colors (the standard 20 colors).

- **Border**—The **Raise**, **Lower**, and **Flat** settings are not normally visible unless a high resolution monitor is used. Generally assume they are not active. The width of the border is stated in percentage terms. You can change the color of the border by using the list next to the color setting to view the available colors (the standard 20 colors).

Legend Tab

Legend is turned off by default. You can place a legend on the graph to illustrate the data elements that are present. The default names in the legend are the data element names. If you want to control the displayed legend name, change the **Column Description** field in the data definition.

- **Display Legend**—Select the check box to activate the legend.

- **Auto Size**—Determines whether the legend box expands to encompass the defined elements. If this is turned off, the legend box must be manually sized to include the defined elements.
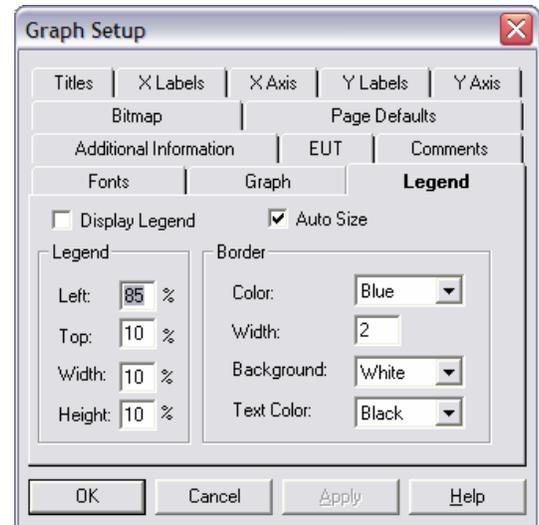
- **Legend**—The **Left**, **Top**, **Width**, and **Heigh**t of the legend box are placement percentages. They are referenced to the upper left corner of the graphing area. These are only used if auto size is turned off.

- **Left**—The default position of the left side of the legend is set to 85% of the graphing area, referenced to the upper left corner of the graph. You can change this to any positive number between 0 and 100.

- **Top**—The default position of the top corner of the legend is set to 10% of the graphing area, referenced to the upper left corner of the graph. You can change this to any positive number between 0 and 100.

- **Width**—The default width of the legend is set to 10% of the graphing area, referenced to the upper left corner of the graph. You can change this to any positive number between 0 and 100.

- **Height**—The default height of the legend is set to 10% of the graphing area, referenced to the upper left corner of the graph. You can change this to any positive number between 0 and 100.

- **Border**—These settings control properties of the legend. The **Color**, **Width**, **Background**, and **Text Color** boxes allow you to customize the colors used in the legend.

  Color is the border color. **Width** is the width of the border. **Background** refers to the color of the interior area of the legend box. **Text Color** determines the color of the text on the legend. All the text is the same color; you cannot individualize the colors.

- **Color (border)**—You can set the color of the border to any of the 20 possible color choices in the list. Click the arrow to select or change the color. The default color is blue.

- **Width**—Controls the thickness of the border. The default value is 1. Change this to any numerical value between 1 and 10.

- **Background (Color)**—The default background color is white. Click the arrow to select or change the color to one of the 20 possible colors.

- **Text Color**—The default text color is light green. Click the arrow to select or change the color to one of the 20 possible colors.

Additional Information Tab

Select the **Additional Information** tab to include information on the graph. This information is shown across the bottom margin of the graph. With the exception of EUT (which has a separate tab) the positions are fixed.

- **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.
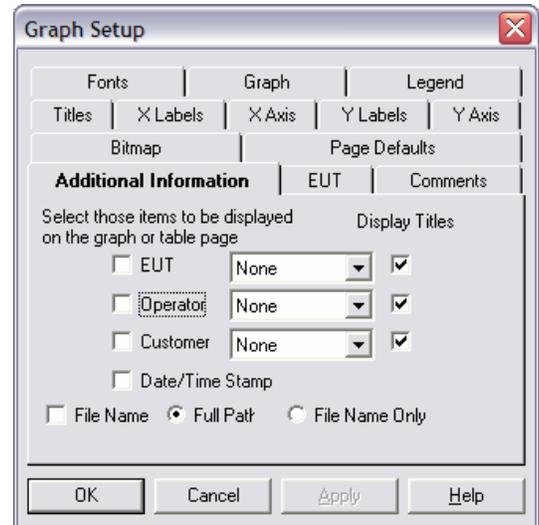
- **Operator**—You can display the operator name across the bottom of the graph. Click the **Operator** check box and select the appropriate action from the flowchart. You can have multiple operator actions on the flowchart, but only one on each graph. They can be different for different graphs. Click the **Display Titles** check box if you want **Operator:** to display.

- **Customer**—You can display the customer contact name and company (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. Under **File Name**, there are two options: **Full Path** will display the complete windows path including directories and subdirectories, and **File Name Only** will limit the display to the file and extension.

EUT Tab

Select the **EUT** tab to display 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. This is turned off by default.

- **Display Floating**—Select this check box to display the EUT information.

- **EUT (Action)**—Click to show a list of all EUT actions from the flowchart. Select an appropriate EUT for inclusion on this graph.

- **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.

- **Left**—The default position of 70 places the upper left corner of the EUT box 70% across the graph (measured from the left side). Change this to any number between 0 and 100.

- **Top**—The default position of 5 places the upper left corner of the EUT box 5% down from the graph top. Change this to any number between 0 and 100.

- **Width**—The default width of 15 sizes the width of the EUT box to 15% of the width of the graph. Change this to any number between 0 and 100.

- **Height**—The default height of 20 sizes the height of the EUT box to 20% of the height of the graph. Change this to any number between 0 and 100.

- **Border**—You can control the shape, color, and sizes of the floating box using these settings.

- **Color (Border)**—You can set the color of the border to any of the 20 possible color choices. Click the arrow to select or change the color. The default color is blue.

- **Width**—This setting controls the thickness of the border. The default value is 1. Change this to any numerical value between 1 and 10.

- **Background (Color)**—The default background color is white. Click the arrow to select or change the color to one of the 20 possible colors.

- **Text Color**—The default text color is light green. Click the arrow to select or change the color to one of the 20 possible colors.

- **Show Titles**—If this check box is selected, the EUT titles will also be displayed. For space considerations, the default is to omit these titles.

Comments Tab

Select the **Comments** tab to allow display of a comment on the graph. Comments are turned off by default. Comments are from the Comment action if used in the flowchart.

- **Display**—Select this check box to display the comments.

- **Comment (Action)**—Click to display a list of all comment actions from the flowchart. Select an appropriate comment for inclusion on this graph.

- **Comment (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.

- **Left**—The default position of 70 places the upper left corner of the EUT box 70% across the graph (measured from the left side). Change this to any number between 0 and 100.

- **Top**—The default position of 5 places the upper left corner of the EUT box 5% down from the graph top. Change this to any number between 0 and 100.

- **Width**—The default width of 15 sizes the width of the EUT box to 15% of the width of the graph. Change this to any number between 0 and 100.

- **Height**—The default height of 20 sizes the height of the EUT box to 20% of the height of the graph. Change this to any number between 0 and 100.

- **Border**—You can control the shape, color, and sizes of the floating box using these settings.

- **Color (Border)**—You can set the color of the border to any of the 20 possible color choices. Click the arrow to select or change the color. The default color is blue.

- **Width**—This setting controls the thickness of the border. The default value is 1. Change this to any numerical value between 1 and 10.

- **Background (Color)**—The default background color is white. Click the arrow to select or change the color to one of the 20 possible colors.

- **Text Color**—The default text color is light green. Click the arrow to select or change the color to one of the 20 possible colors.

Bitmap Tab

Select the **Bitmap** tab to allow display of a bitmap on the graph. The bitmap option is turned off by default. 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.

- **Browse (Bitmap)**—Click the **Browse** button to launch a dialog box. This allows you 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 disabled 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.

- **Left**—The default position of 10 places the upper left corner of the bitmap 10% across the graph (measured from the left side). Change this to any number between 0 and 100.

- **Top**—The default position of 10 places the upper left corner of the bitmap 10% down from the graph top. Change this to any number between 0 and 100.

- **Width**—The default width of 10 sizes the width of the bitmap to 10% of the width of the graph. Change this to any number between 0 and 100.

- **Height**—The default height of 10 sizes the height of the bitmap to 10% of the height of the graph. Change this to any number between 0 and 100.
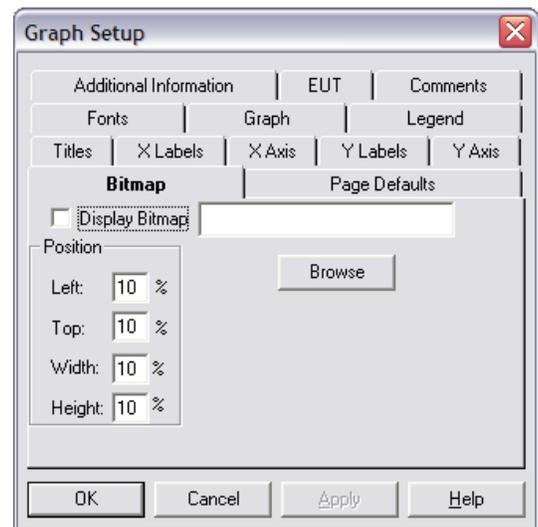
This tab controls the orientation of the graph to the printed page. They are mutually exclusive.

- **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.
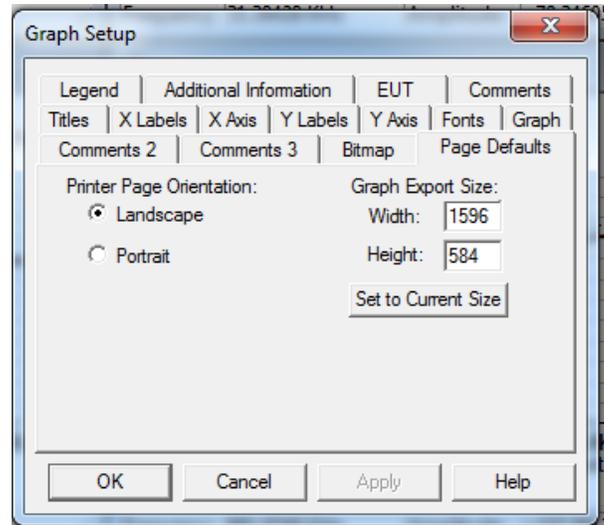
- **Graph Export Size**—This allows the pixilated export file to match what is being viewed on the screen. The EMF and WMF files will inherit the screen resolution as a default. If the exported graph does not look correct, resize this until it is correct.

- **Width/Height**—Sets the number of pixels for each dimension to export.

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

Zoom Graph

The zoom option allows you to change the x-axis or y-axis ranges without having to define or change any other parameters of the graph. With this option off, you can then adjust the scaling of the x-axis and y-axis to exactly fit the desired scale.
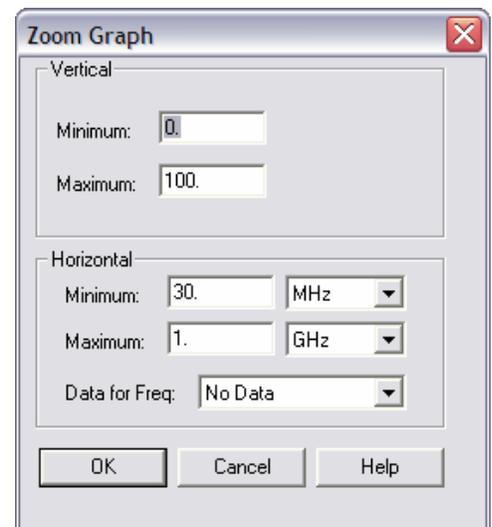
Vertical Specifications

The y-axis range can be changed on this page, but if **Log** on the **Y-Labels** tab is set, the final range may not fit the entered values.

- **Minimum**—The default minimum is 0. Set this to a value consistent with the data or range of interest.

- **Maximum**—The default maximum is 100. Set this value consistent with the data or range of interest.

Horizontal Specifications

The x-axis range can be changed on this page, but if **Log** is set on the **X-Labels** tab, the final range may not fit the entered values.

- **Minimum—**The default minimum is 30 MHz. Set this to a value and unit to be consistent with the data or range of interest.

- **Maximum—**The default minimum is 1 GHz. Set this to a value and unit to be consistent with the data or range of interest.

- **Data for Freq—**You can select a data element and the graph will automatically scale the x-axis to the range of values in this data element. If this data element is empty, the default ranges (or last valid range) will be displayed.

Copy Graph

This copies an exact copy of the current graph to a new graph. The new graph will be identical except the name will have a numerical appendage (starting at 1) for each time the graph is copied.

Editing an Existing Graph

Once a graph is defined, all the tools used to create the graph are used for editing. See *Creating a Graph* on page 2 for more details. You can click anywhere on the graph to bring the **Graph Options** dialog box to the foreground. This allows you to edit an existing graph.

Temporary Zoom

You can zoom in or out of a graph to look at specific details. These are temporary zooms and are never saved. Following are instructions on how to zoom. The originally designed graph is always the saved item.

- **Zoom In—**You can zoom in to look at specific data by holding the right mouse button down and dragging the area that you want to expand. The ranges of the x-axis and y-axis will match the zoomed area unless the Llg option is set on the **X-Labels** or **Y-Labels** tab.

- **Zoom Out—**Double-click the right mouse button to return the graph to the original ranges.

Tables

A Table is created from the Project window by selecting Tables, right click and Add.

Each new graph is started as a blank graph with no selected data elements. The icons on the graph toolbar has the following functions:
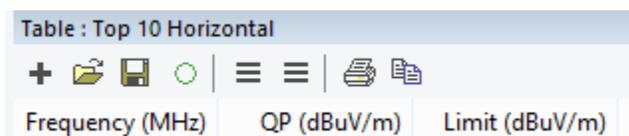
1. Plus symbol opens the Display Data window. The Display Data window provides a list of available data elements to display on the table. Due to the evolution of the software this is duplicated with the right-most three horizontal lines. Further, data can simply be dragged and dropped onto the table.

2. The folder icon is not functional and reserved for future functions
3. The 'Floppy' icon will open a file explorer to save the table to a specific location
4. The green circle 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
5. The left-most 3 horizontal lines open the graph Setup that is used to set all the parameters of the table
6. The Printer icon will open the print interface so the table can be printed
7. The copy icon will allow a copy of the table to be placed in the past buffer

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 graphs 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. They are not normally used for listing raw data. A table is created with the appropriate number of rows to fit the defined data elements. If you place raw data onto a table you might have 8,000 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 data the complete tabular data must be reformatted in memory. Again, this can cause a significant delay in system performance.

Creating a Table

A table is created from the list on the Windows menu bar. Click or select **Add** to view three choices: **Graphs**, **Tables**, and **Page**. You can create as many different graphs or tables as you want. Each new table is started as a blank table with no selected data elements.

When the new table is formed, the Windows menu bar changes to reveal **Display**. The **Display** selection gives you access to the configurations for the table. There are four choices; **Data**, **Options**, **Set Left Margin**, and C**opy Table**. Each of these controls a slightly different part of the table. You can also click the body of the table to access the options.

Display Data Tabs

Lines Tab

- **Add**—Double-click an element in the **Available** column to automatically add it to the **Selected** column. You can use the **Move Up** and **Move Down** buttons to position the element in the list.

- **Remove**—To remove an element from the list, double-click the element in the **Selected** column. You can also highlight the element and click **Remove**.
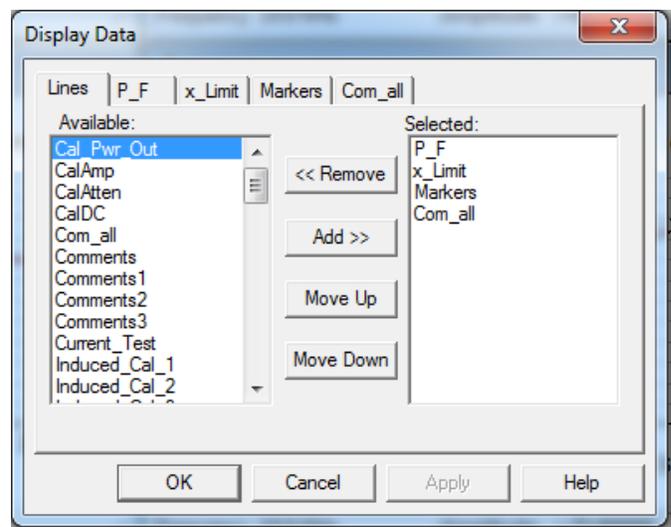
For each data element added to the **Selected** column, a tab will appear with that data element name.

This will allow you to define the line or marker style for that data. 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 (the default), then the first block labeled **Line** applies. If the data element was defined as discrete, then the box labeled **Marker** applies.
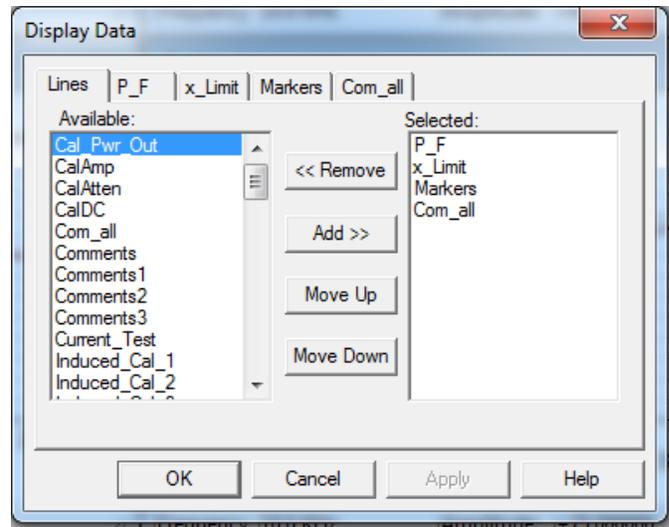
- **Line**—Style, color, and width are only appropriate for data elements that are defined as **Continuous** (either log or linear). The default settings create a neutral line setting. To highlight the line, choose a different color or shape. This is helpful when displaying the specification limits on 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 the readings.

- **Show All**—Controls whether all 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 the normal setting with 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.

- **Filled**—This option is disabled.

- **Marker**—When the data element selected is not interpolated, you can select an appropriate marker shape to represent the data on the graph.

- **Drop**—When this option is selected, a line will be displayed from the marker position extending down and intersecting the zero axis; this is convenient for highlighting the specific location of the data on a graph.

- **Show on Legend**—This feature is disabled.

- **Frequency Order**—When this check box is selected, a separate sort will ensure all data is from low frequency to high frequency.

- **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.

- **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 are two rows of description for each data element. This field controls the second description field. For the reading as a clarifying label, you might use *dBuV/m* to display units.

Options

The **Options** dialog box has five tabs: **Titles**, **Additional Information**, **Comments**, **EUT**, and **Page Defaults**. These 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, and cut-and-paste defaults.

Table Setup Tabs

Titles Tab

- **Title**—Enter the title for the table. A unique name is essential to identify this table when automatically printing or accessing the table from different applications.

- **Line 1**—Line 1 is the top line on 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 value is for the second description of the first column. The default value is MHz. If you want the column header to be blank you must enter a space.

- **Freq Decimal Places**—The user can control the number of decimal places displayed for the frequency. The default is 3 (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 then three decimal places for frequency, you might need to widen the column.
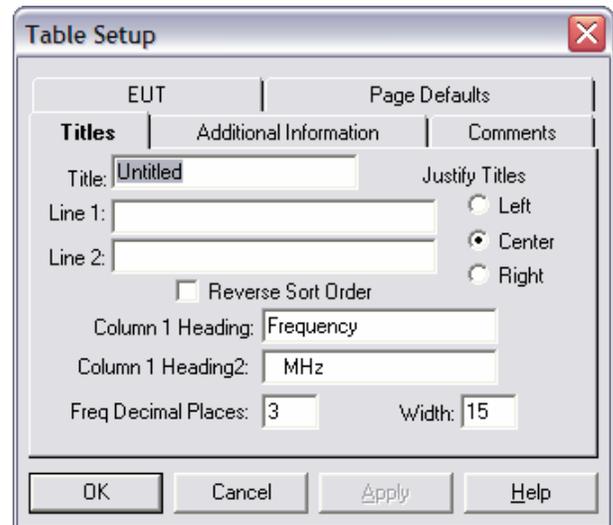
Additional Information Tab

Select the **Additional Information** tab to include information on the table. 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) on the table. 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 table. They can be different for different tables. 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 top of the table. 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 table. They can be different for different tables. Click the **Display Titles** check box if you want the **Operator:** to display.

- **Customer**—You can display the customer contact name and company (strings one and two from the Client action) across the top of the table. Click the **Client** check box and select the appropriate action from the flowchart. You can have multiple client actions on the flowchart, but only one on each table. They can be different for different tables. 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 table will display. By using the newest date/time stamp, we allow the table to reflect the date/time of the last execution that affected the data shown on this table.

- **File Name**—If this check box is selected, the file name will display. This can be a long value depending upon the file name and subdirectory situation. There are two options: **Full Path** will display the complete path including directories and subdirectories, and **File Name Only** will limit the display to the file and extension.
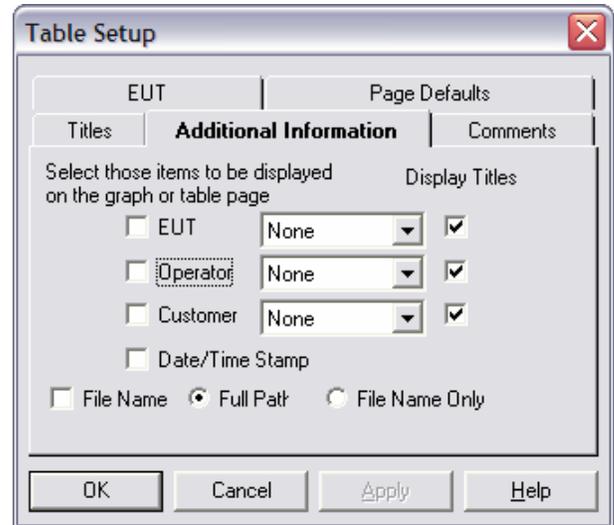
Comment Tab

Select the **Comment** tab to access the option for 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**—Click the arrow and select the appropriate action from the flowchart.
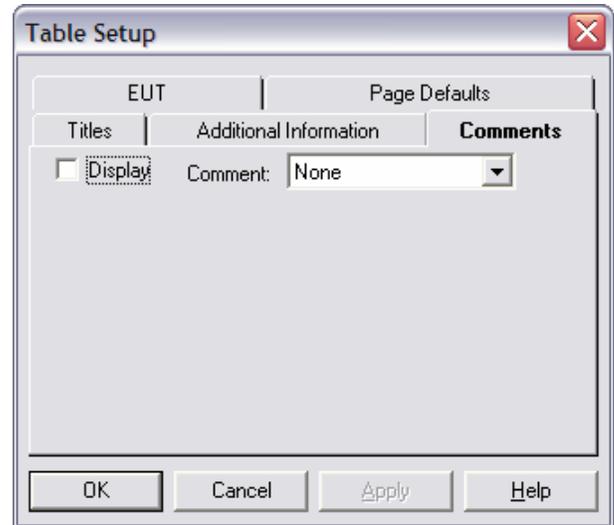
EUT Tab

The **EUT** 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.

- **Include Titles**—When selected, the title for each line of the EUT action is included prior to the information.
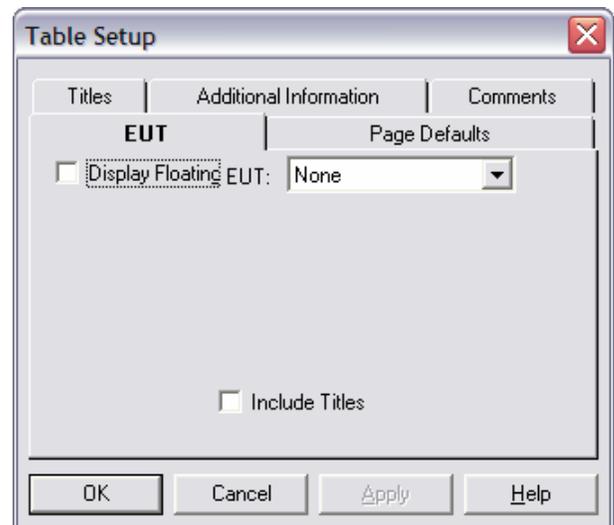
The **Page Defaults** tab allows you to control certain parameters of both printing and cut and paste operations.

- **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.

- **Page Width—**When doing a cut and paste operation, the data is copied to the clipboard in text format. The page width parameter formats the information into an expected page size. Information over this width is separated by a line feed.

- **Tab Spacing—**Columns of information in a cut and paste operation are normally separated by a tab (tab delimited). When writing to disk using copy/export, spaces are substituted for tabs to allow more predictable formatting when opened in word processors with different tab settings. For example, the default in Microsoft Word is no tabs, which causes the information to be pushed together. We have chosen to replace the tab with spaces for consistency across different word processors.

- **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. If **kHz** is chosen 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. This option is only available when **Use Text for Frequency on Copy/Export** is selected.

- **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.
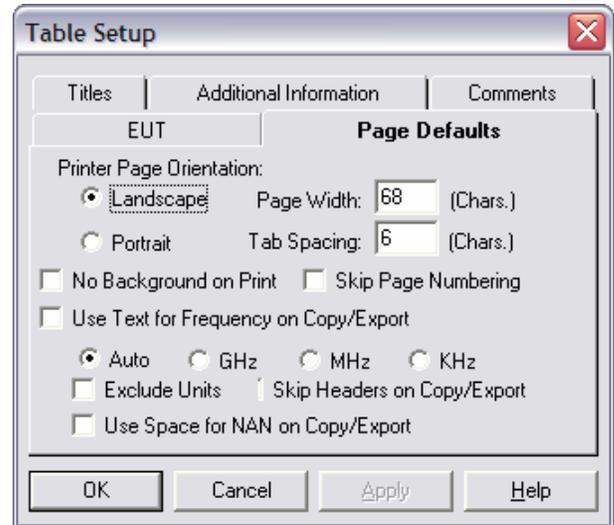
Editing an Existing Table

Once a table is defined, all the tools used to create the table are used for editing. See 'Creating a Table' for more details.

7.0   Common Features for Profile Development

This section will cover all of the common elements that may be used to develop and execute profiles.

Toolbars

Standard Bar

The **Standard** toolbar is common to most Windows operating system applications. It contains **New**, **Open**, **Save**, **Save As**, **Cut**, **Copy**, **Paste**, **Print**, and **Help** icons. Place the cursor over the icon to display the name.

Standard toolbar

View Bar

The **View** toolbar contains TILE!-specific icons that are used to open the flowchart, data elements, instruments, log, audit trail, graphs/tables, Interactive Instrument Control, add instrument and add data functions.

View toolbar

Command Bar

The **Command** toolbar is used to execute actions in the flowchart. Each of the three arrows will bring up a window with all actions in the profile that can be selected as a starting point.

Command toolbar

- **Single dark arrow**—Executes the flowchart starting with the selected action.

- **Double arrows**—Steps through the actions in order of the flowchart connections.

- **Single light arrow**—Executes a single action on the flowchart.

- **Octagon symbol**—Can stop a process. It may not stop a current GPIB transaction but it will stop at the next available step.

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

Windows Menu Bar

The menu at the top of the toolbar is used to access features of the window that is currently selected.
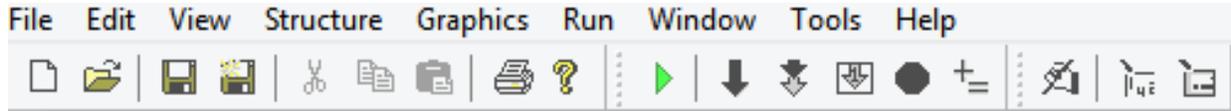
- **File**—Typical file functions with the addition of the **Options** tab. This is used to set up default settings, such as the GPIB interface and preferred directories.

- **Edit**—This will be sensitive to the selected window. The undo/redo options are classic tools. If you accidentally erase an icon, immediately undo the mistake. **Redo** repeats a step. **Delete** and **Select All** allow you to clear large areas of the flowchart. For single deletions and selections, it is easier to use the cursor.

- **View**—This will be sensitive to the selected window but can also be used to zoom in on the flowchart or select other windows. The View command provides a number of tools for controlling the screen of the program. You also have shortcut keys that allow you to move around the program if preferred. The first four choices concern the active areas around the window. The **Toolbar**, **Status Bar**, **Command Bar**, and **Popup Bar** are all shortcut keys or status indicators for the window frame.

- **Structure**—Contains group selections such as group, ungroup, bring to front, send to back, as well as many action alignment functions to improve the aesthetics of the flowchart. There is a page size function that will size 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 has the same functions as the icons found in the **Command** toolbar on page 71. It also has some editing features that can access all of the elements of the profile to delete or calculate on command.

- **Window**—Use to access any window or to order them in cascade or tile orientations. This menu will also be used to add graphs and tables.

- **Tools**—The only option currently available is **Run Simulation**. Select this to start the instrument simulator.

- **Help**—This selection will open this manual.

Status Bar

The status bar at the bottom right of the TILE! window displays information about the status of the window. Items such as current action, time, and elapsed time are displayed. The only control available displays the line or turns it off. There are no options for configuring this information. This information, with the exception of the current time, only appears when the program is executing.

The bottom left will show the tool tips for the actions on the palette when the cursor is moved over it.

Toolbar Icons

Instruments

The **Add Instrument** icon is used to add an instrument to the profile.
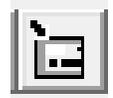
Add Instrument icon

Data Elements

The **Data Elements** icon opens the Data Elements window.

Data Elements icon

Interactive Instrument Control

This function can be used to interact with instruments using remote commands. This is useful to validate commands used in the 'VISA' action or to debug driver interaction.

Interactive Instrument Control icon

- **Instrument**—Selects instrument from list.

- **Init / ID Query**—Performs a *IDN? 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 as well as other common commands.

- **SYST:ERR?**—Queries the instrument to return the latest error in its queue.

- **Clear Session History**—Clears previous commands from **Session History**.

- **Session History**—Displays the queries and responses from the instrument.

- **Status**—Shows the status of the communications with the instrument.

- **Write**—Sends the command displayed in **Instrument Command** to the instrument.

- **Read**—Reads the response from the instrument.

- **Write – Read**—Combines Write and Read as one command.
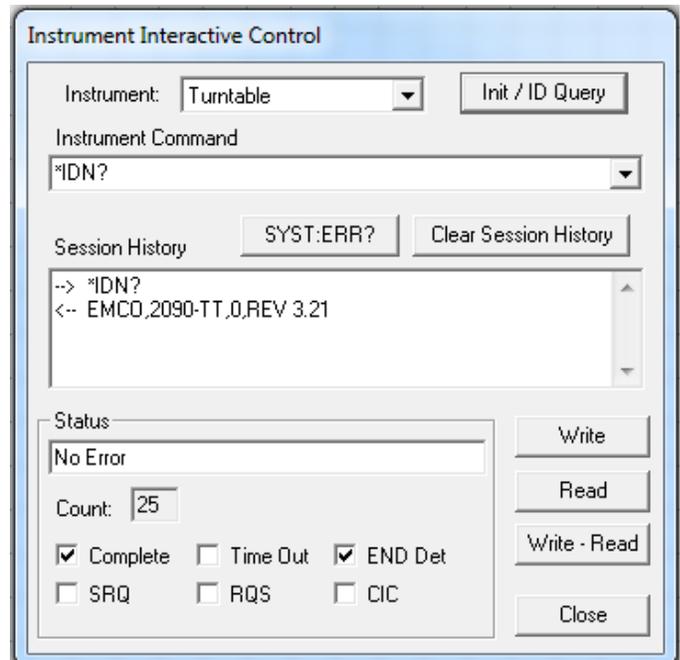
Common Action Commands

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 is left as it was 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.

Help

This selection will open this Reference Manual.

Common Name Page

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 you are not limited by the software, 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 Show in Audit Trail Option

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.

OK, Cancel, Apply, and Help

A description of the **OK**, **Cancel**, **Apply**, and **Help** buttons is found in *Common Action Commands* on page 2.

Import Settings

Along with the cut and paste option when working with actions on the flowchart, the **Import** option lets the user copy the settings from another action on the flowchart. The only items not copied are the output data elements; this is an issue of data integrity that is absolute.

Only the actions of the same type will be displayed as available. Select the action to copy from and click **OK**.

8.0   Ancillary Actions

Ancillary actions are available to add process or information to the profile. Some of the most common actions are Prompt, Math, Auto Save, Transfer Data, and Clear.

- All actions contain the **Action** tab that will allow the action to have a name assigned to it. The **Action** tab will also show the actual name of the .dll file, the version, and the build date.

- Most actions also contain a **Bookmark** tab, which can be used to provide information for an HTML output for report generation.

Drawing Tools

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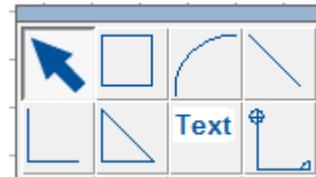
Drawing tools

File Management Actions

There are three File Management actions: Auto Save, Save Windows, and Transfer Data will save the whole profile, the graphs and tables, or transfer data in and out of the profile.

File management actions

Auto Save

The Auto Save action is used to automatically save 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 which can then be examined at a later date.

Auto Save icon

Save As Tab

The **Save As** tab gives the user the options for saving the profile.

- **Prompt for Name**—When selected, will prompt the operator for the file name.

- **Use Current Name**—When selected, will automatically save the profile under the current name.

- **Auto Incr./Beginning Name**—Auto Incr. and **Beginning Name** select and specify the file naming convention/sequence that will be invoked.
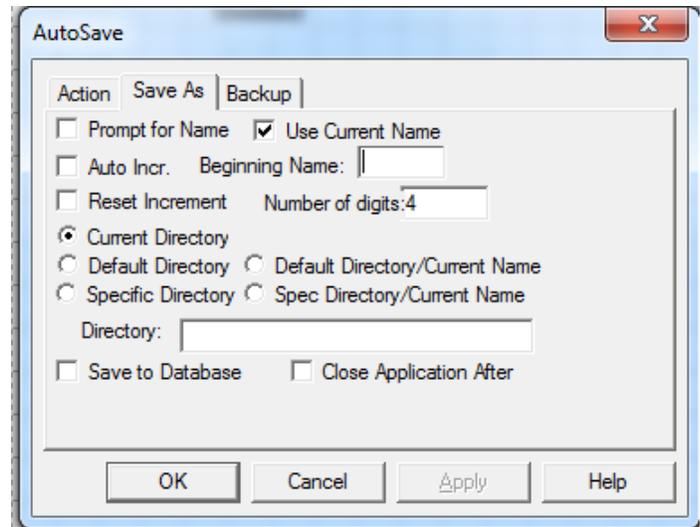
- **Reset Increment**—If this check box is selected, the first save will be reset to zero, and incrementing will begin at zero.

- **Number of digits**—The number of digits for the increment can be set between 1 and 4 yielding values of 1 to 9999.

- **Directory Specification**—These apply if **Prompt for Name** is selected:

- **Current Directory:** The directory in which the TILE! profile was opened. If the **Prompt for Name** check box is selected and **Current Directory** is 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 if **Prompt for Name** is selected it will force the operator to name the file.

- **Default Directory/Current Name:** If **Prompt for Name** is 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 **Prompt for Name** is 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.

Backup Tab

When activated, this 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.
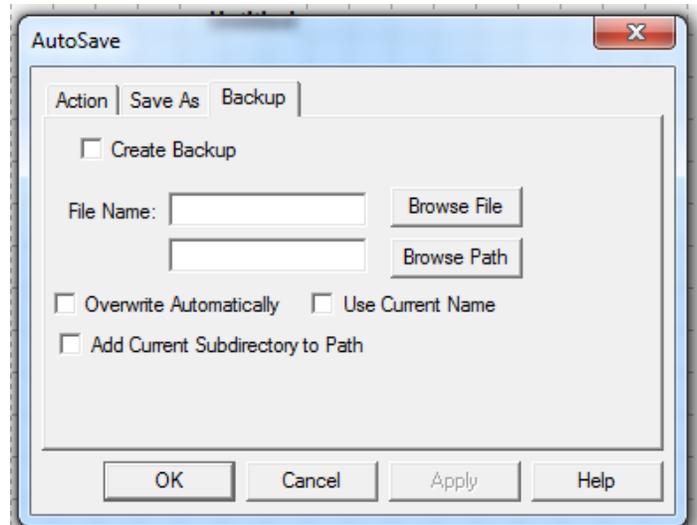
- **Browse File/Browse Path**—Displays a dialog box that can be used to search for a desired file. If the **OK** button is selected the name and path boxes are automatically filled.

- **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.

Save Windows icon

Graph/Table Tab and Data Tab

The **Graph/Table** tab and **Data** tab each have a column on the right that shows all graphs and tables that are available for export. They can be selected by double-clicking on them or selecting the **Add>>** button. They can be removed with a double-click or the **<<Remove** button.
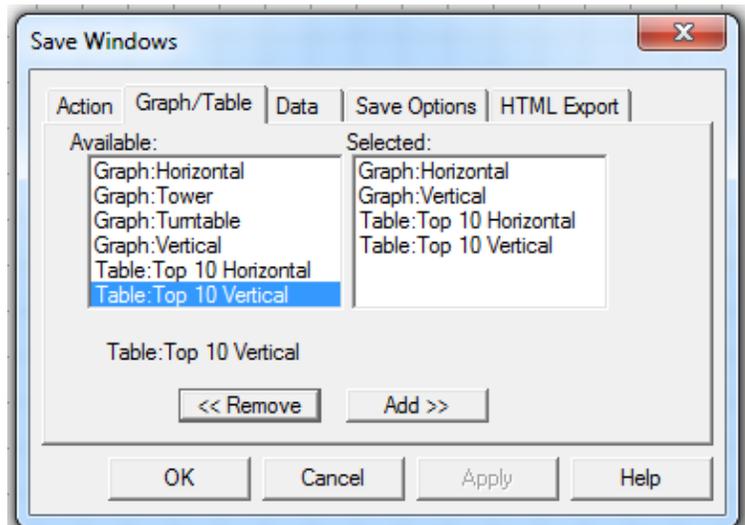
Save Options Tab

The **Save Options** 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.
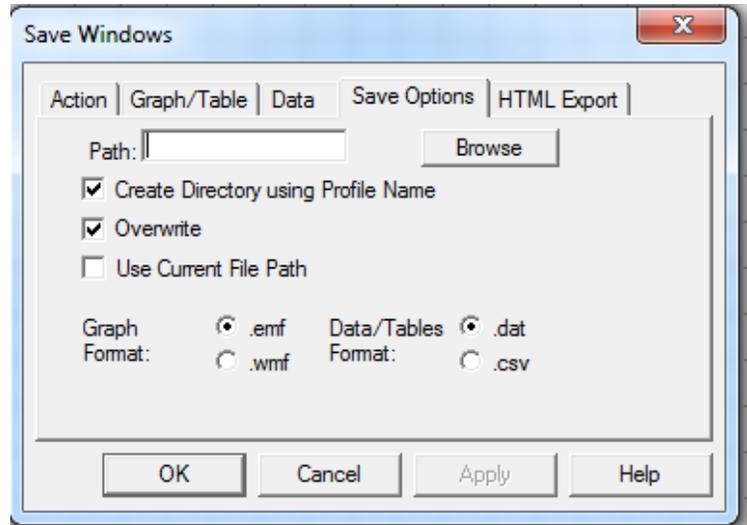
HTML Export Tab

The **HTML Export** tab will create a file that contains HTML code that can be used to import data into Microsoft Word to generate a report.
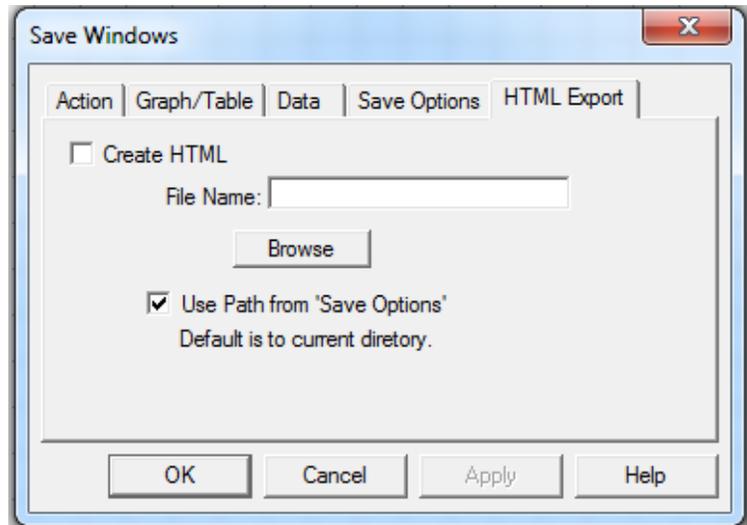
Transfer Data

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

Transfer Data icon

Transfer to/from File Tab

The **Transfer to/from File** tab describes the file from which or to which data will be transferred when **Data Element** is selected. When **Table/Graph** is selected, output to a file is the only option available.

- **Data**

  When **Data Element** is chosen, the **Data** list will include the available data elements. Click and select a defined data element.

  When **Table/Graph** is chosen, the **Data** list will include the available tables or graphs. Click and select a defined data element.

  When **Window** is chosen, the **Data** list will include the Data, Instruments, Log, and Audit Trail windows. Click and select a defined data element.
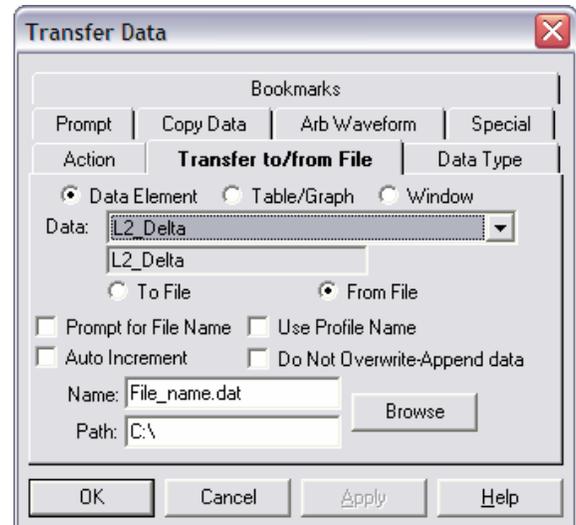
- **To File/From File**—These option buttons indicate whether the Transfer Data action will save to file or read from file. If from file, the data element defined in the Data window will be filled with the input information. This is the same procedure as defining a data element as a file-type and initializing 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 selected, the to file/from file option buttons are not visible.

- **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.

Direct Entry icon

Data Element Tab

The **Data Element** tab defines which data element will be used to store the results of this action. If no element is selected and the action is executed, the user can assign a name at that point.

- **Data Element**—Click the arrow and 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.
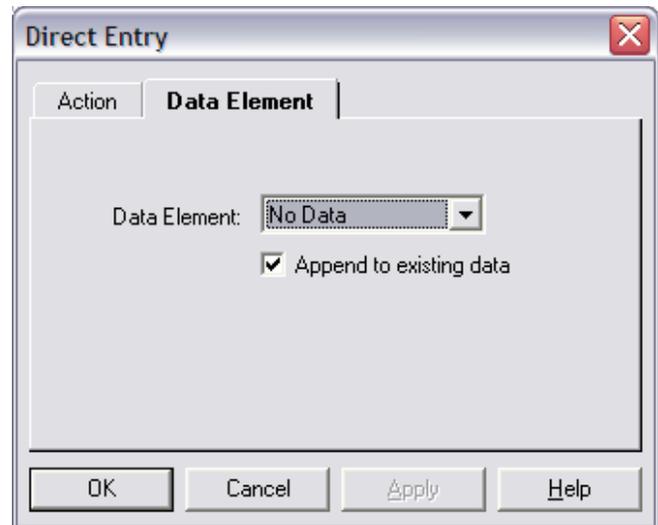
Prompt Tab

A message can be placed here to provide information or instructions to the user.
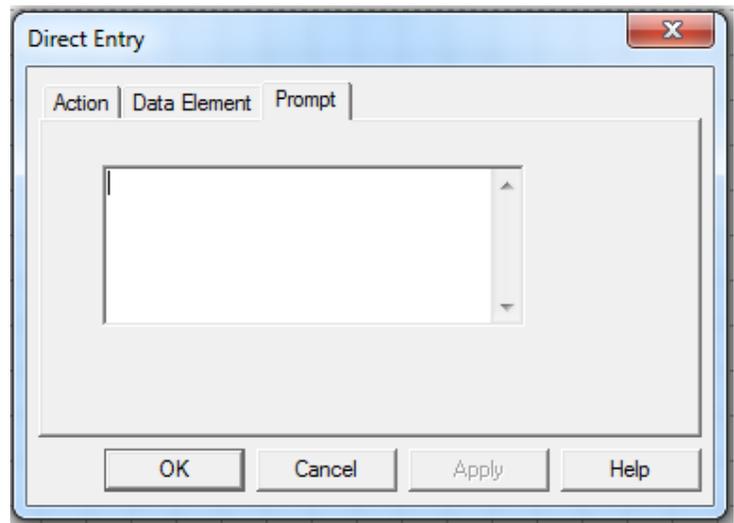
Direct Information Dialog Box

The **Direct Information** dialog box is the entry point for the frequency and amplitude data.

- **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 and the user can then assign a name and it will automatically create a preset element with that name. 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.

- **Insert Before**—Inserts the frequency/value combination in the list before the current cell.

- **Change Current**—Inserts the frequency/value combination in the list at the current cell.

- **Insert After**—Inserts the frequency/value combination in the list after the current cell.

- **Delete**—Deletes the current row from the table.

- **<< / >> buttons**—Use these buttons to navigate up or down the frequency list.

- **Go to Cell**—This will focus the current element on the cell entered here.

- **Set all values to constant**—This will set all values to the one in this window.

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. The Clear Data action can only clear measurement, equation and word-type data elements.

Clear Data icon

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

The **Clear Data** tab defines the data that will be cleared in this action. It is divided into two columns, **Available** and **Selected**.

Available displays the defined data elements. **Selected** displays those data elements that need to be reset or cleared during execution of this action.

- **Add**—The **Add** button will include data elements highlighted from the **Available** column to the **Selected** column. If no data elements are highlighted, this button has no effect. You may also add data elements to the selected column by double-clicking the data element in the **Available** column.

- **Remove**—The **Remove** button will delete highlighted data elements in the **Selected** column. If no data elements in the **Selected** column are highlighted, this button has no effect. You may also delete data elements from the **Selected** column by double-clicking the data element.
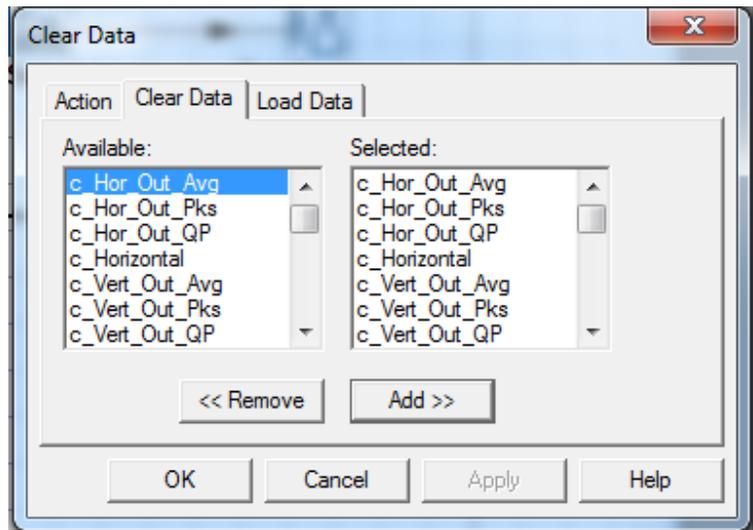
The **Load Data** tab is useful if you want to always load the most current data, such as from an antenna factor file.

Select the data elements (only From File types are shown) that need to be reloaded at this point.
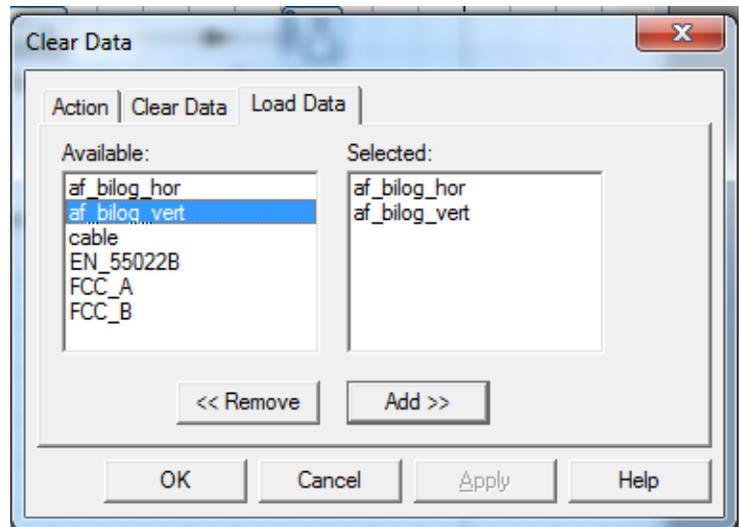
Information Actions

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.

Information actions

Some of the information will be available to be displayed on the graphs and tables where 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 **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.

Lab Information icon

Laboratory Information Tab

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. These can be modified by the user during editing to match any requirements for labels. When executing, only the value field is changeable.

Client Information icon

Client Information Tab

The first two items of the client information, referred to as **Contact** and **Company** in the generic titles, are the only items that appear on tables and graphs even when using the Additional Information options. All other fields, including the four editable fields on the **Additional Information** tab, are for reference purposes only.

File 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 you can either use the client/file tab or you can 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.
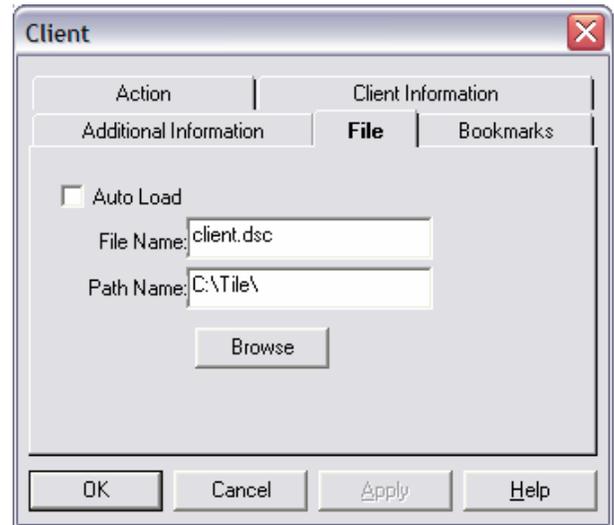
Client Bookmarks Tab

This allows the user to specify format for data so it can be seamlessly imported by a macro into a Word document.

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.

Operator Information icon

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 first field can be displayed on the bottom of graphs and top of Tables when utilizing the Options/Additional Information features of graphs/tables.

EUT Information icon

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 first field, labeled **Equipment ID**, can be displayed on graphs and tables using the 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.

Floating EUT will appear when in edit graph mode; see *Editing an Existing Graph* on page 2. The EUT action allows all information and titles to be displayed on graphs.

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

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.

Comment icon

Comment Tab

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 a total of 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 is not documented but would allow the recording of weather conditions if connected to a station that will communicate with the TILE! workstation.

Go to http://support.ets-lindgren.com for help with this action.

Weather icon

Process Actions

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

Process actions

Start

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

Start icon

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.

Prompt icon

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

The **Message** tab allows the user to enter a message that will be displayed in the message box during the execution of this action. A carriage return can be entered to space text. For this reason, the **Enter** key will not perform as the **OK** button, you must use the cusor or **Tab** key.

The message box includes **OK** and **Cancel** buttons. During operation, if the operator selects **OK**, execution of the next action in the flowchart continues. If the operator selects **Cancel**, execution is halted and the flowchart remains as it was before the dialog box was opened.
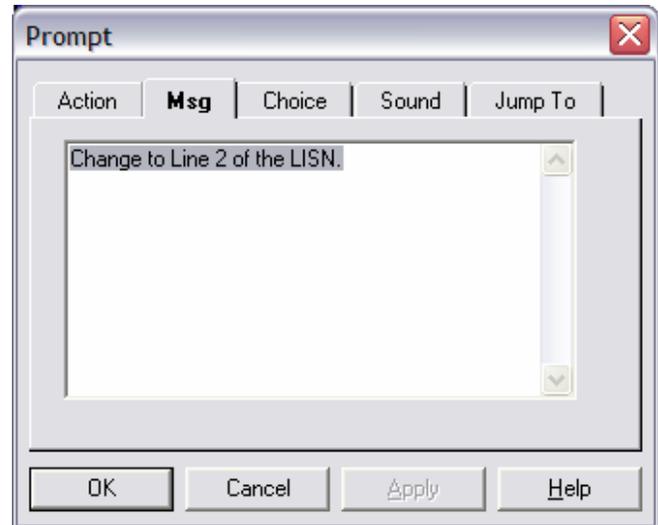
When using the Prompt action we recommend that you put multiple paragraphs of information instead of a single large paragraph. The visual impact of long paragraphs of information can be confusing to the operator. When the amount of data exceeds the simple view, you will automatically scroll to add more. 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 two outward arrows (directions) to exist; all other actions execute in a serial fashion. The Prompt action has an option that allows you to automatically determine the direction that this action will take based upon a greater than, less than, or equal relationship.

- **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 **Accept** and **Reject**.

- **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 of them or the change will not be accepted.

- **Auto Choose**—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 (see *Jump To Tab* on page 2 for more information). Be aware that the original arrows, or original options, will not be available to you in the prompt if this check box is selected.
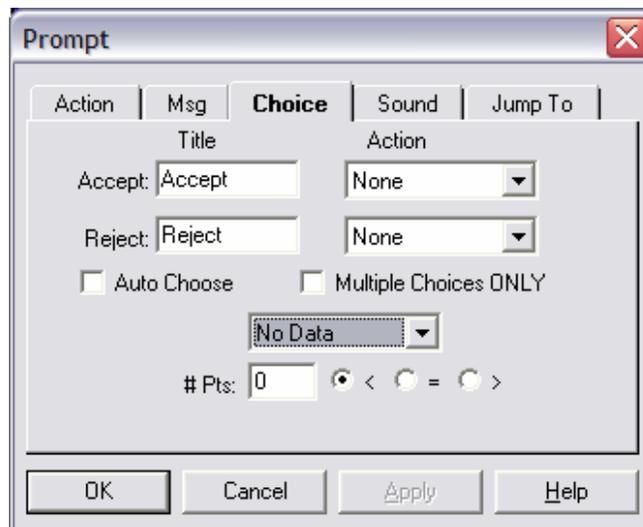
Sound Tab

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. Selecting the **No sound** check box will disable this option. This is also the default.

- **Single / Continuous**—The two options for sound are to play a sound file once or play it until interrupted. Check **Single** or **Continuous**, as desired. The sound prompt is played during the execution of the Prompt action in the test profile.
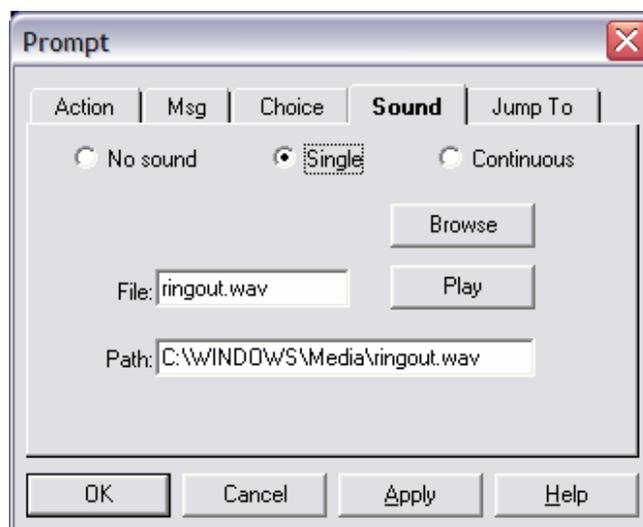
This option allows you to choose multiple continuation points, providing more freedom than the two arrows allotted as connection points from the **Prompt** icon. It allows you to specify a list of any number of actions you could potentially need to execute, allowing you to jump to any action in the flowchart you might need to execute from this prompt.

For example, if under most circumstances you will only need to choose between two options, either action A or action B, but under other circumstances you need to choose action C, D, or even action G.
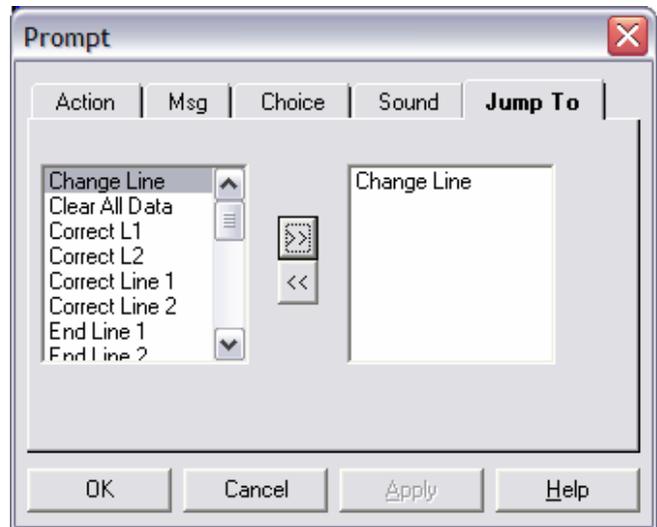
The **Jump To** tab consists of a left column that contains all of the actions defined in the current profile, and a right column (which will be blank in a new prompt) which contains all of the actions available to jump to from this prompt. To specify the list of actions for the prompt, select a predefined action from the left column and press the **>>** key to add it to the right column. Continue doing this for as many actions as are necessary.

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 under Prompt/Choices 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.

Math

The tower/turntable section of the Math action takes 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.

Math icon

Data Tab

The **Data** tab controls the data elements that will be executed in this step. It is divided into two columns: **Available** displays the defined data elements, and **Selected** displays the data elements defined as equations which need to be executed at this step in the profile. See *Equation Elements* on page 2 for more information on equations and creating equation data elements.

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.
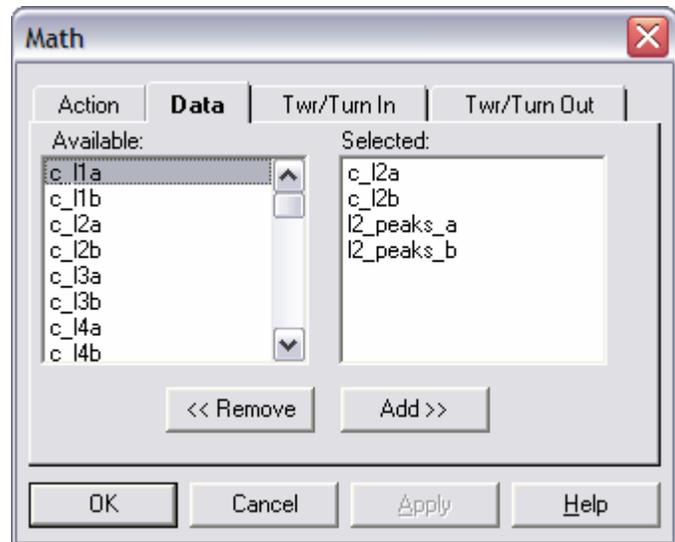
- **Remove**—Deletes highlighted data elements from the **Selected** column. If no data elements are highlighted, this button has no effect. You may also delete data elements by double-clicking the data element.

- **Add**—Includes data elements highlighted in the **Available** column in the **Selected** column. If no data elements are highlighted, this button has no effect. You may also add data elements by double-clicking the data element in the **Available** column.

Twr/Turn In and Twr/Turn Out Tabs

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.
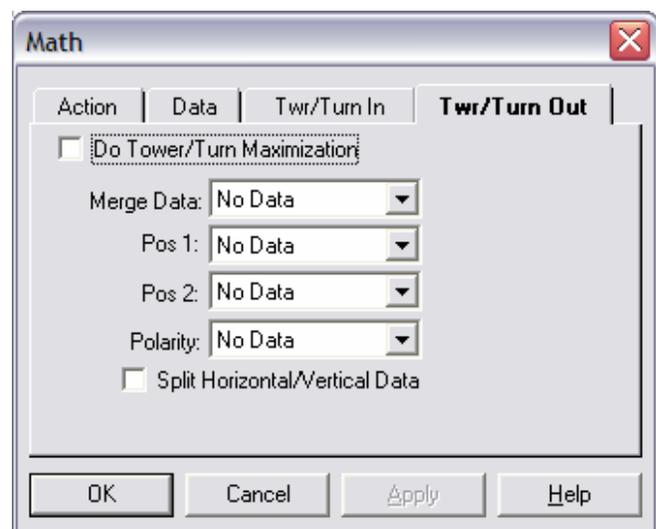
- **Do Tower/Turn Maximization**—This optimization process is performed when the check box is selected.
- **Split Horizontal/ Vertical Data**—When the check box is selected, the tower/turntable reverses nature and the merged data elements are split based upon their **H** or **V** designation in the Polarity data element.

Math Dialog Box

The **Math** dialog box provides visual information to the user as well as the ability to halt execution of the test.

Print

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.

Print icon

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

Print 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** displays the known graphs, tables, and windows, and **Selected** displays those items selected for printing in this action.

- **Remove**—Deletes highlighted items from the **Selected** column. If no items are highlighted, this button has no effect. You may also delete items by double-clicking the data element.

- **Add**—Includes items highlighted in the **Available** column in the **Selected** column. If no items are highlighted, this button has no effect. You may also add items by double-clicking the data element in the **Available** column.
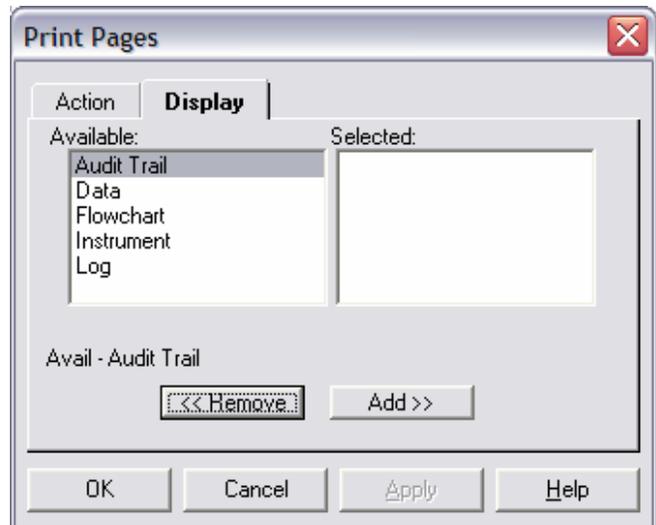
Launch Application

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.

Launch Application icon

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

The **Commands** 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/ 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/Susceptibility* on page 2 for more information on what events exist. In most cases you will want to link to either CW or modulation events.
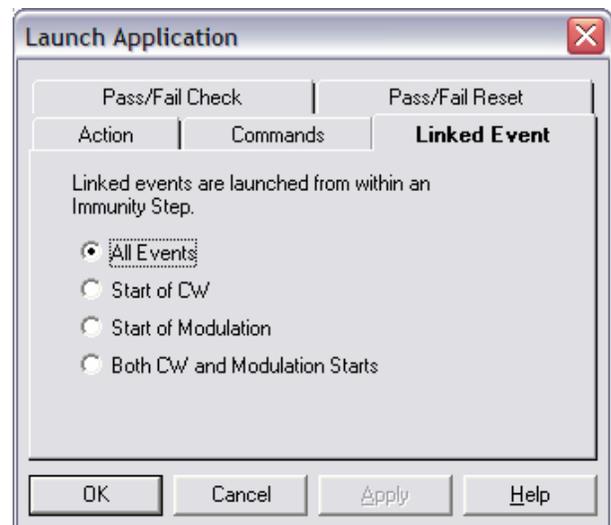
Pass/Fail Check and Pass/Fail Reset Tabs

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. See page 2 for technical support contact information.
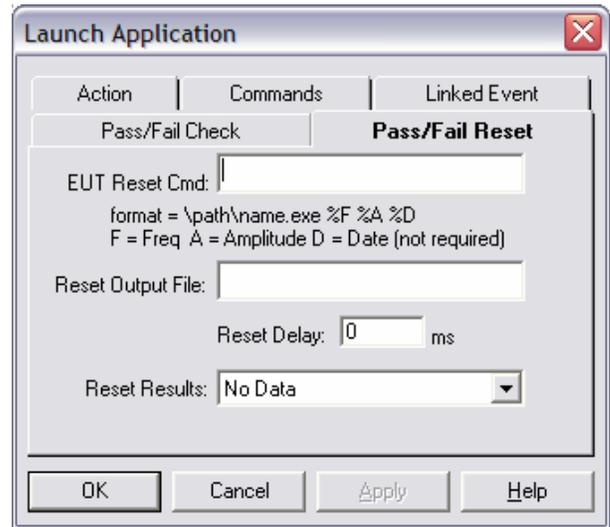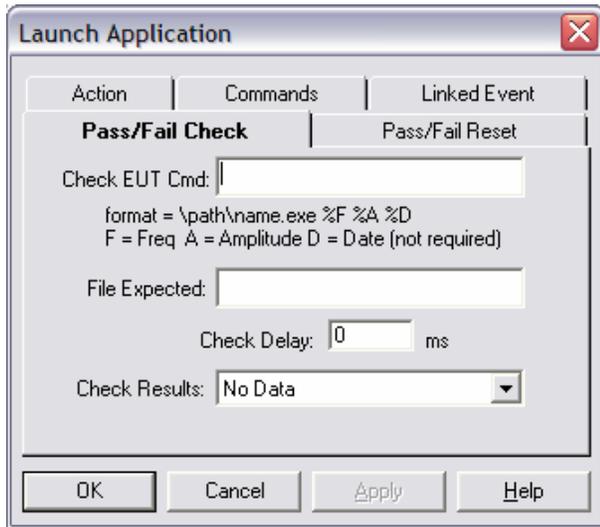
Picture

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 icon

Picture Tab

The **Picture** tab allows selection of the bitmap to be displayed.
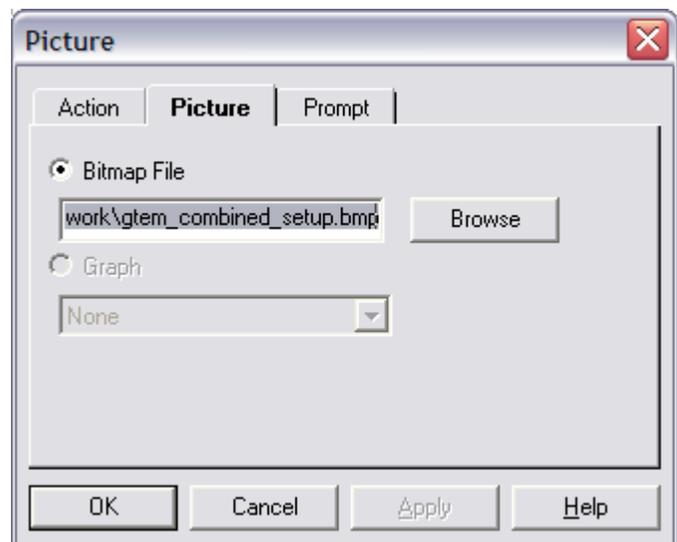
Prompt Tab

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

Timer

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

Timer icon

Sound/Delay Tab

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

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.

Measurement, Positioner, and Control Actions

Measurement, Positioner, and Control actions

Instrument Initialization

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.

Instrument Initialization icon

The **Instruments** tab displays a two-column screen. 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. You can highlight the instrument and click **Add** or **Remove**. Instruments can also be selected and moved by double-clicking the instrument. This will move it to the opposite column (either to add or to remove, as appropriate).
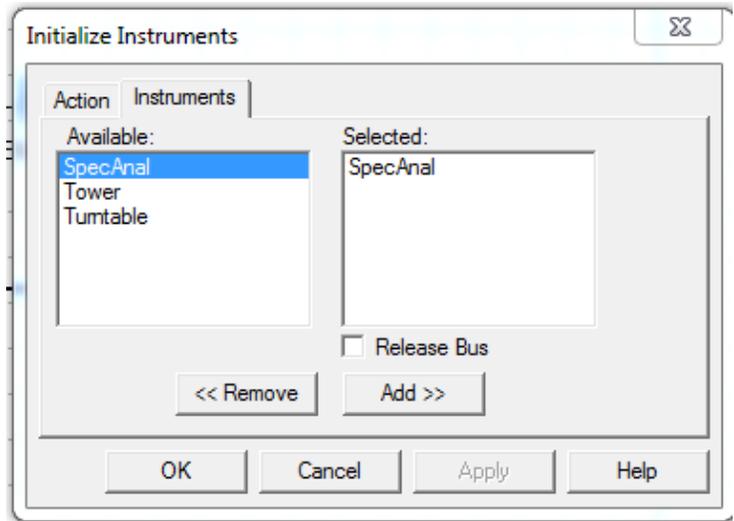
VISA (Instrument Commands) Action

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

VISA (Instrument Commands) icon
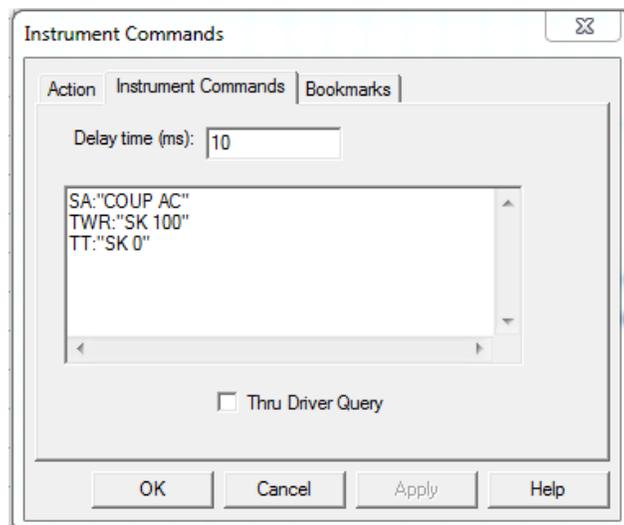
Instrument Commands

You need to define the instrument in the Instrument window. Once the name is defined, 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 the **Driver** tab. These commands in this action are sent directly to the bus address specified in the instrument definition (see *Instrument Control* on page 2 for instrument definition).

- **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 Format**—The format for a direct command is: 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:

      analyzer:"command"

      delay:"2000"

      analyzer:"command"

  This would delay for 2000 milliseconds between the two instrument commands.

- **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 usinf the format

      InstName:read:""

  Note: the quoted are necessary, but the contents inside the quotes are ignored.

- **Hex/Binary Commands**—Commands can be specified using hex or binary format using the syntax:

      InstName:hex:"11 22 33"

  Where the command string is interpreted as hex format, with all spaces removed. Two hex characters equals one byte of data. 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

Calibrate Cables

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

Calibrate Cables icon

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* (see page 2) or *Immunity/Susceptibility Test* (see page 2).

Action Tab

The action tab is a common name page. A complete description is found in *Common Name Page* on page 2.

Setup Tab

The **Setup** tab allows the user to identify the parameters of the calibration.

- **Start Frequency**—Enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—Enter an appropriate number in the box and click the arrow to select the units.

- **Fixed Step**—Determines the interval between each frequency in a linear fashion. Enter an appropriate number in the box and click the arrow to select the units.
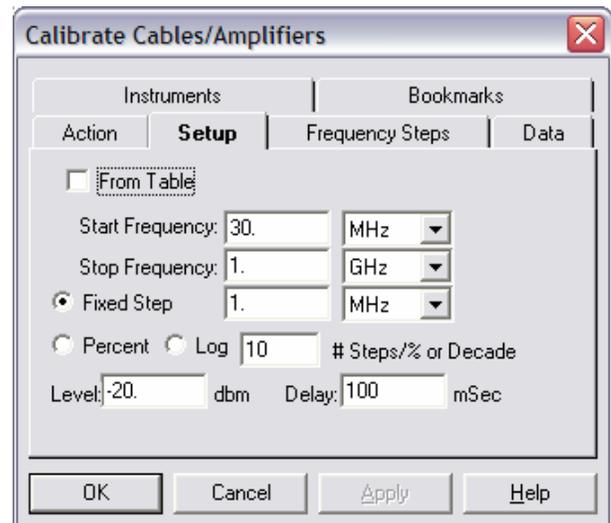
- **From Table**—When this check box is selected, the frequencies for this action are determined by a data element selected on this tab (the selection box will appear once this check box is selected). The data element can be of any type, except word-types. With this choice you are in complete control of the frequencies and their step sizes.

- **Level**—The signal generator amplitude that will be set at each frequency. Enter an appropriate number in the box.
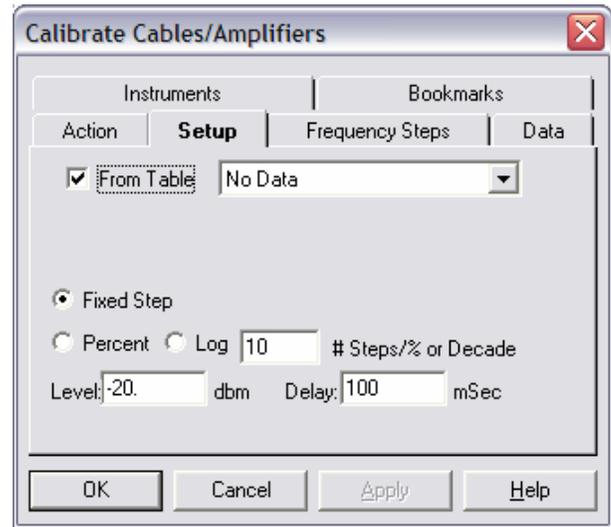
- **Delay**—Sets the desired delay time (in ms) before reading all measurements. Most probes and power meters have a settling time before their readings can be considered accurate. This field allows you to enter a value to create a delay prior to taking these readings. It is typical for power meters 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 to 60 dB–80 dB and take valid readings. But 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.
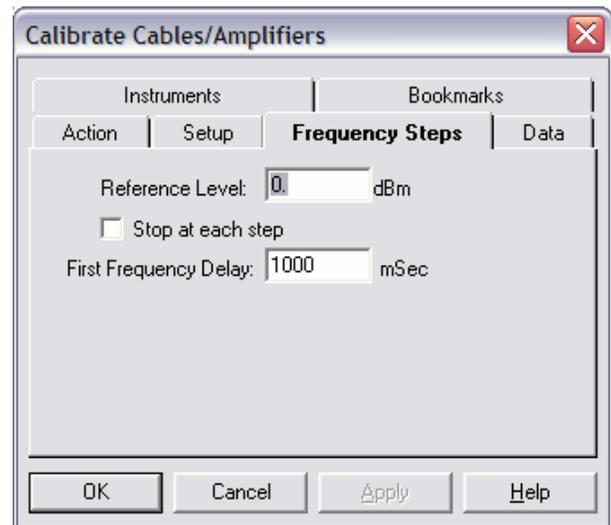
- **First Frequency Delay**—Some power meters have a longer settling time when power is first applied. The value entered in the box is a special delay for the first frequency step only.

Data Tab

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

- **Attenuation**—Identifies the data storage element used to record the measured attenuation value. Click the arrow and select the appropriate data element from the list of defined data elements. The attenuation value is the measured value less 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. Click the arrow and select the appropriate data element from the list of defined data elements.

- **Raw Reading**—The final attenuation value is the direct reading less 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.
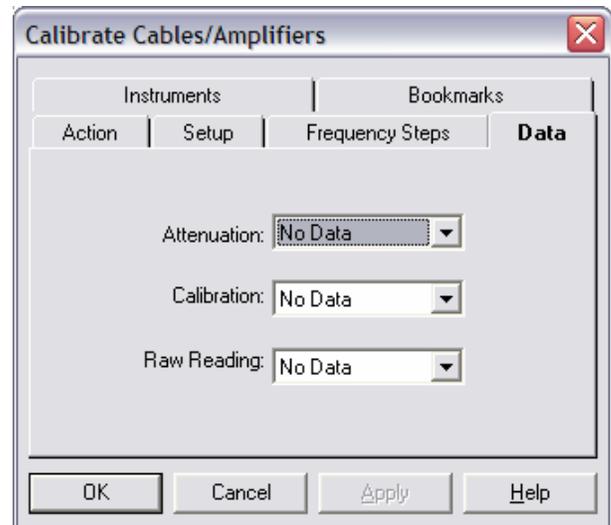
Instruments Tab

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

- **Signal Generator**—Click the arrow and select the appropriate instrument. 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.**—Click the arrow and select the appropriate instrument. The instrument selected can be the same as the spectrum analyzer if this instrument has a tracking generator.

- **Preselector**—Click the arrow and select the appropriate instrument.
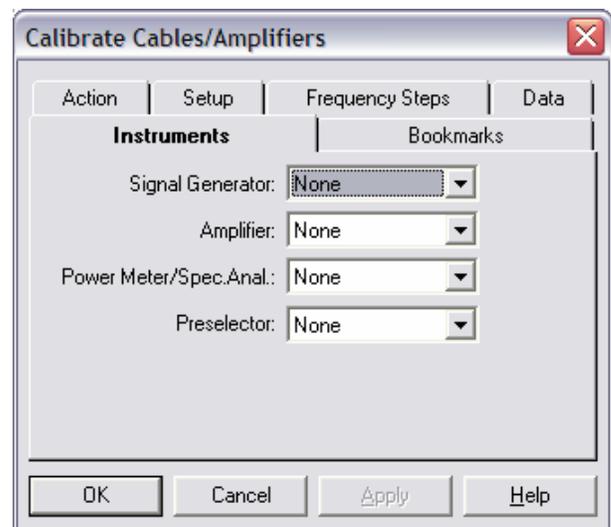
You can execute the Calibrate Cables/Amplifier action by double-clicking the action and selecting **Execute**, by executing it as part of the sequence of tests being executed, or by using the Run/Single options from the Windows menu bar.

- **Step**—Displays the current step that is being invoked.

- **Frequency**—Displays the present signal generator frequency.

- **Generator Level**—Displays the current signal generator level.

- **Power Meter Level**—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 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.

Amplifier Calibration icon

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.
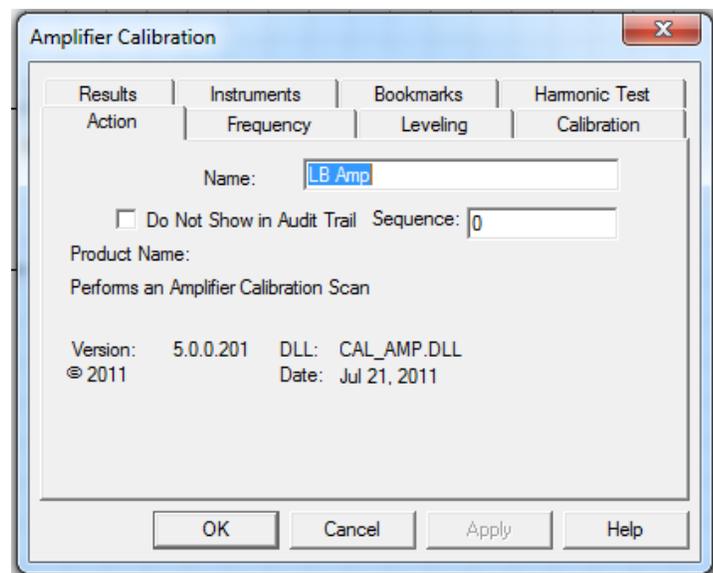
Frequency Tab

The **Frequency** tab is similar to other immunity actions and will not be described here.
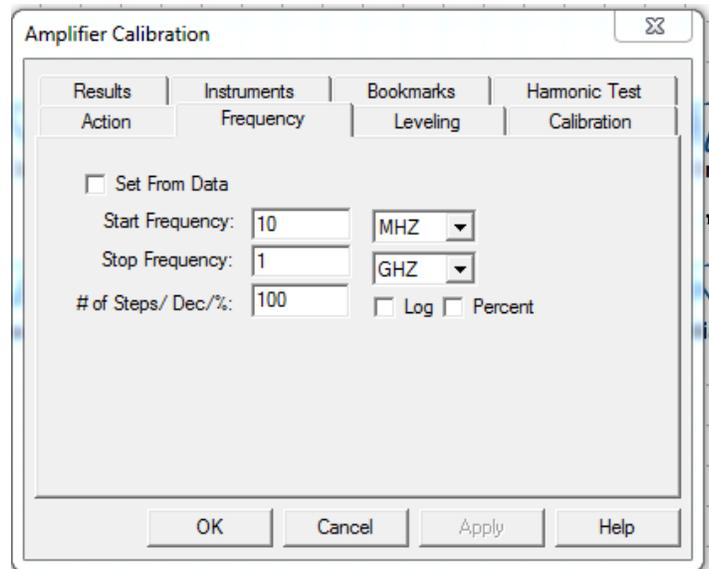
Leveling Tab

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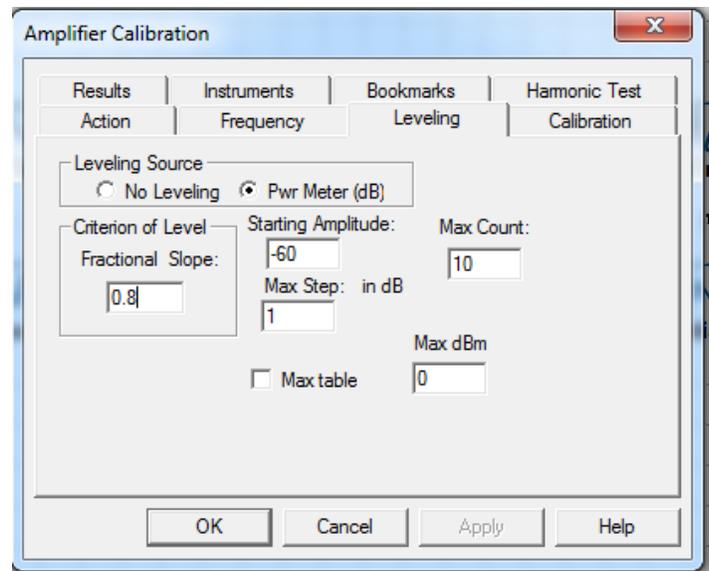
If there is noise in the measurement you may get a false identification of a compression point, so you may want to allow for some tolerance by setting it to 0.8 or slowing down the measurement time on the **Instruments** tab.

- **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

The **Calibration** tab will allow the offset of the directional coupler to be accounted for.
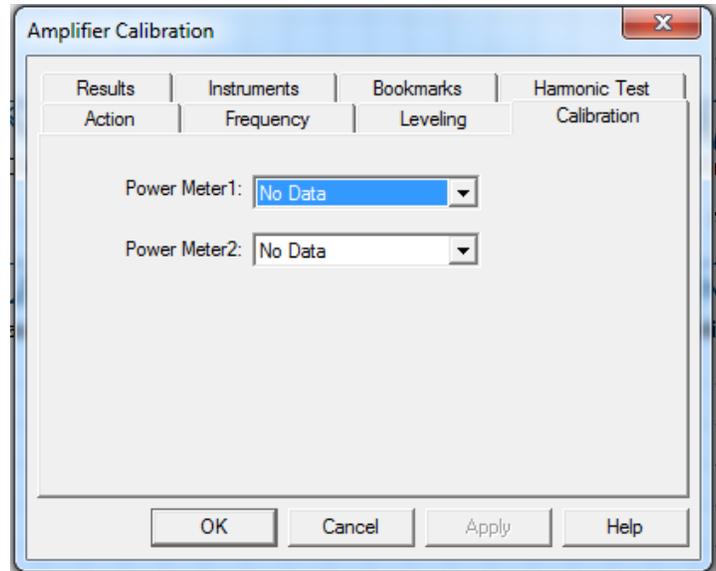
Harmonic Test Tab

The **Harmonic Test** tab selects the harmonics routine.

- **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.
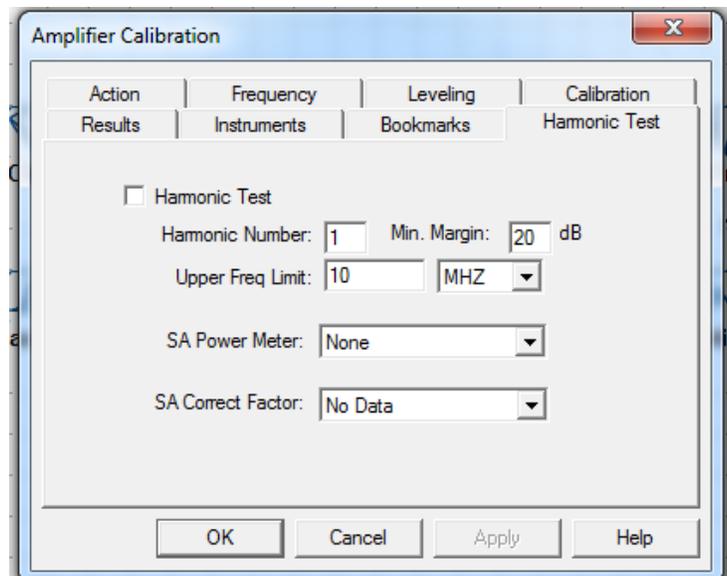
- **SA Correct Factor**—Provides the reference if any correction factor needs to be applied. The directional coupler is already accounted for in the **Calibration** tab.

Setup / Monitor Instrument

The Setup/Monitor Instrument action is used to monitor an instrument. Specific conditions can be set for number of readings and whether to store them or not. Also, 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.

Setup/Monitor Instrument icon

Monitor Tab

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.

- **Monitor On**—Select the check box to activate this feature.

- **Instrument**—Click the arrow and 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 a number of readings before continuing on to the next step of the flowchart.

- **Output Data**—If storing is desired, click the arrow and select a data element. The output will be defined as a Measurement Type data element.

- **Start Frequency**—For the created output, enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—For the created output, enter an appropriate number in the box and click the arrow to select the units.
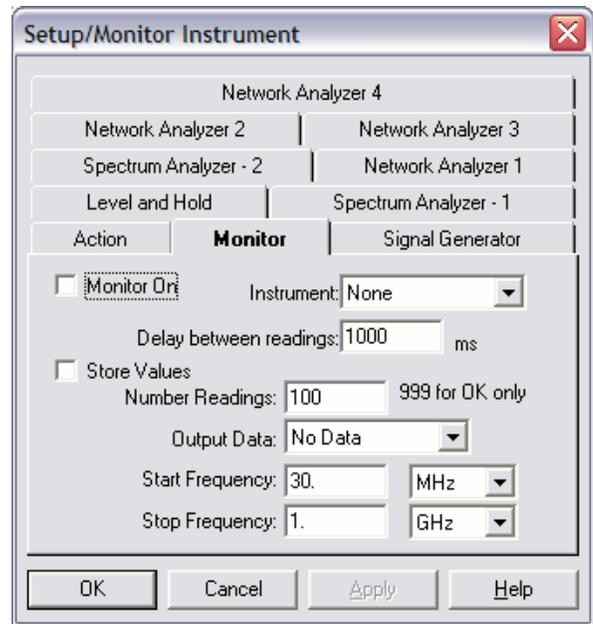
Signal Generator Tab

Use the **Signal Generator** tab to set up a signal generator. The generator will be set up and then turned on prior to exiting the action.

- **Set Signal Generator**— Select the check box to turn on this feature.

- **Instrument**—Click the arrow and 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.
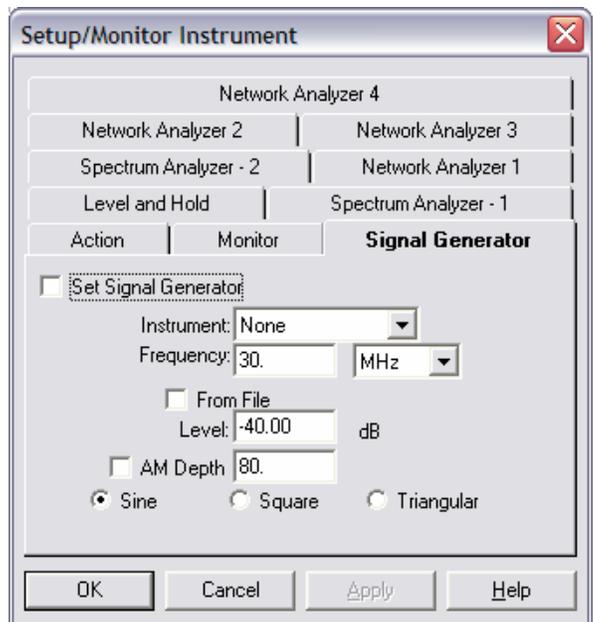
- **AM Depth**— Select the check box and set the value if you want to turn on AM modulation. The frequency will be the frequency that was specified in the previous field.

- **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.

- **Level and Hold**—Select this check 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 Tolerance/Minus Tolerance**—Positive tolerance above the desired level. Any value between the desired level less 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.
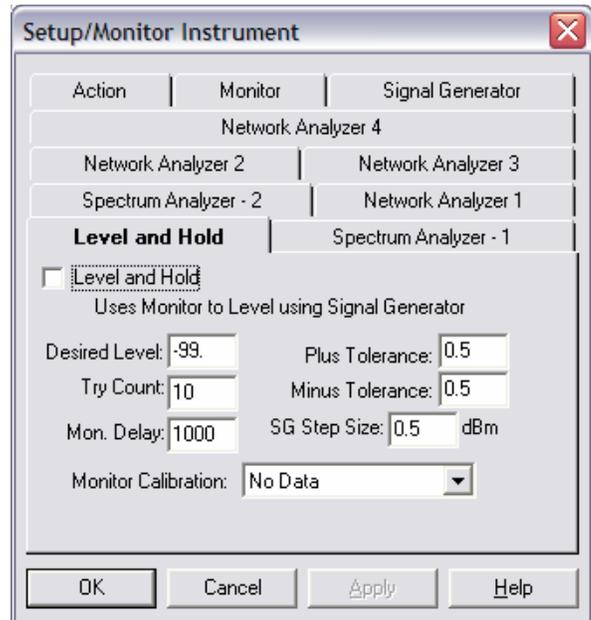
Spectrum Analyzer – 1 Tab

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

These settings are continued on the **Spectrum Analyzer – 2** tab.

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.
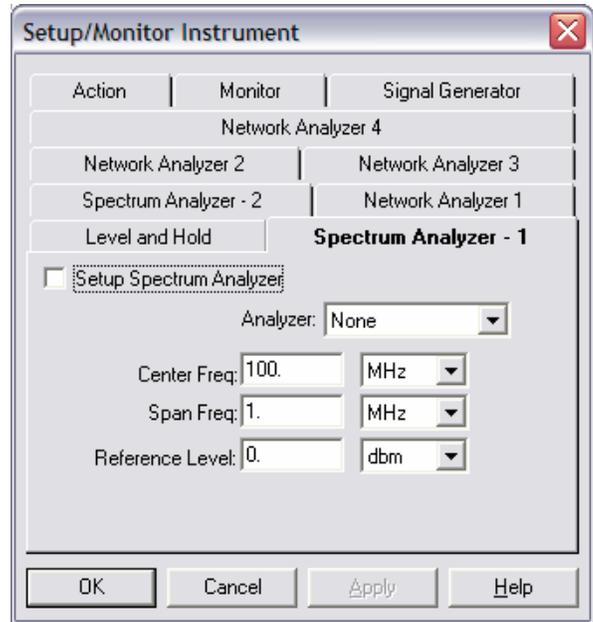
- **Setup Spectrum Analyzer**—Select this check box to activate this feature. If the check box is deselected, all these settings are ignored during execution.

- **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 center frequency. This value is preserved during subsequent Immunity testing, regardless of frequency.

- **Reference Level**—Enter a default reference level for the instrument. The instrument is set to this level, but will adjust higher if the signal is too strong.

Spectrum Analyzer – 2 Tab

This is a continuation of the settings from **Spectrum Analyzer - 1** tab.

- **Res BW**—Set the resolution bandwidth. If you choose a setting not supported by the instrument, it will default to the nearest level supported by the instrument.

- **Video BW**—Set the video bandwidth. If you choose a setting not supported by the instrument, it will default to the nearest level supported by the instrument.

- **Sweep/Meas Time**—Set the sweep time desired. If you choose a setting not supported by the instrument, it will default to the nearest level supported by the instrument.

- **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.

- **Number of Sweeps**—Enter a number of sweeps, if not using continuous mode.
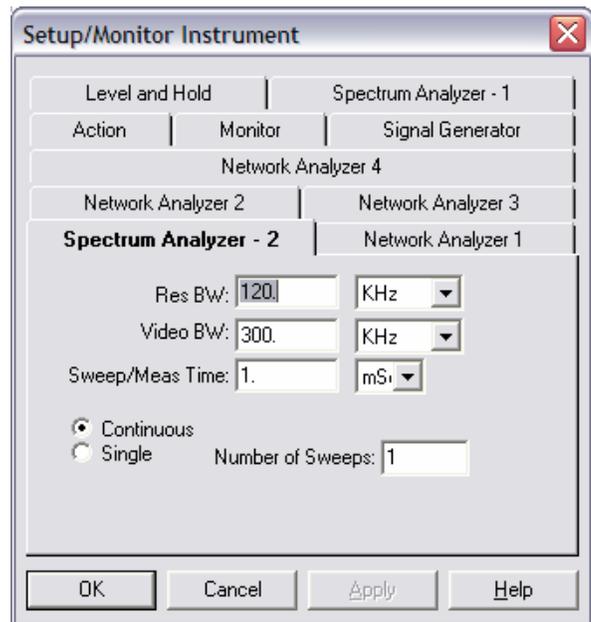
Network Analyzer 1 Tab

In certain circumstances the user will need to set up specific conditions for a network analyzer prior to performing other measurements. The four **Network Analyzer** tabs let you set these conditions before continuing on to other steps in the flowchart.

- **Setup Network Analyzer**—Select this check box to activate the network analyzer settings. This controls all four tabs related to network analyzers.

- **Network Analyzer**—Click the arrow and select the instrument to set up in this step.

- **Channel 1, 2, 3, 4 Active**—Select the check box for each channel that will be active. You can select 1 to 4 channels at the same time.
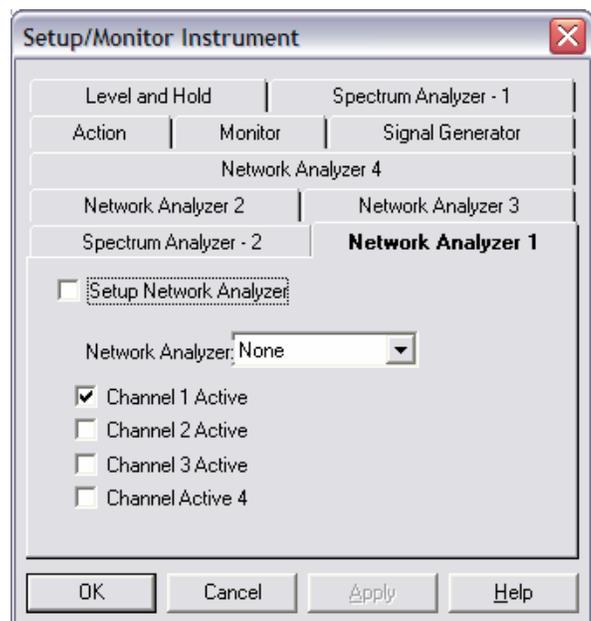
Setup/Monitor Instrument Network Analyzer 2 Tab

This is a continuation of the network analyzer settings. Choose the appropriate dB/Div, Reference Top, and Reference Bottom for each channel.

- **dV/Div**—Enter a dB per division.

- **Reference Top**—Enter the top reference level.

- **Reference Bottom**—Enter the bottom reference level.
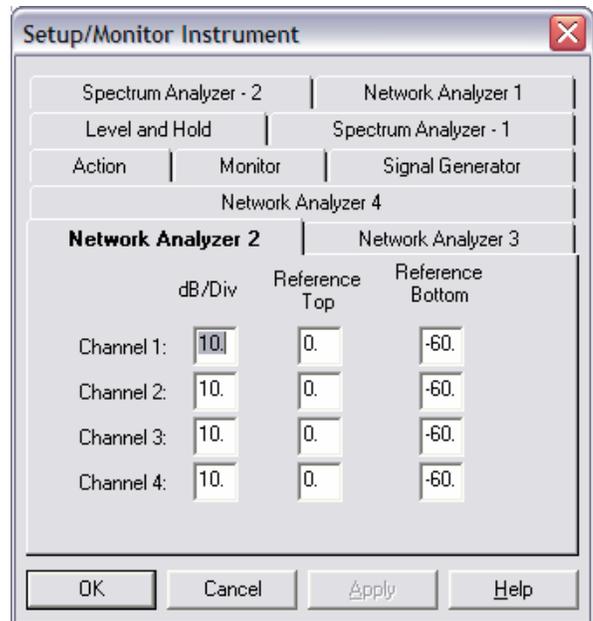
Setup/Monitor Instrument Network Analyzer 3 Tab

This is a continuation of the network analyzer settings. 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.

- **Log Mag**—Turn on log magnitude.

- **Lin Mag**—Turn on linear magnitude.

- **Phase**—Turn on phase mode.

- **SWR**—Turn on SWR mode.

- **Group Delay**—Turn on group delay.

- **Power Out**—Turn on power out.

- **Real**—Turn on real mode.

- **Imaginary**—Turn on imaginary mode.
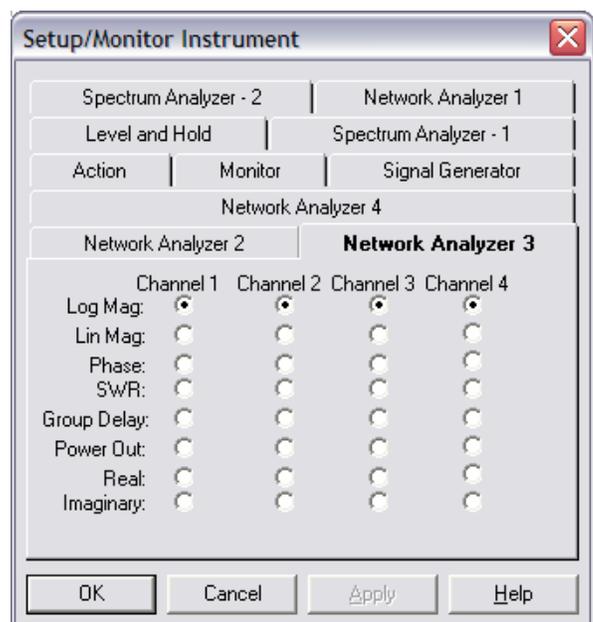
This is a continuation of the network analyzer settings. Select the mode (S11, S12, S21, or S22) for each channel.
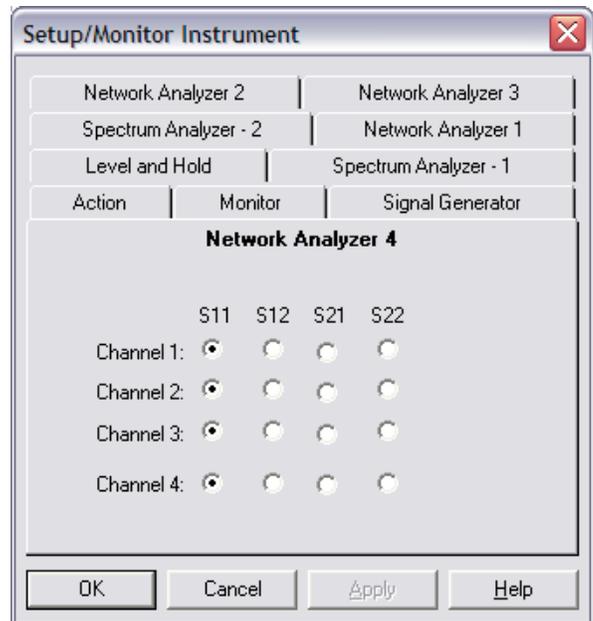
Read Trace

The Read Trace 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.

Read Trace icon

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**—For default readings, enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—For default readings, enter an appropriate number in the box and click the arrow to select the units.

- **Linear/Log**—Defines how frequencies will be stopped, whether in log or linear interpolation.
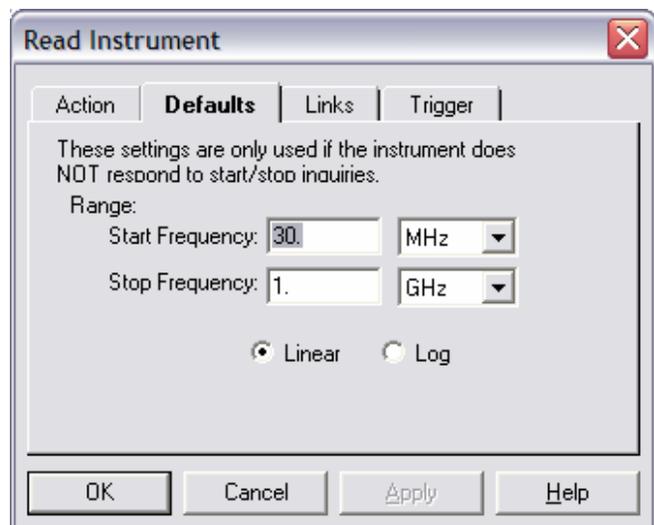
Links Tab

The **Links** tab controls the instrument to be read and the data element to where the results will be stored.

- **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.
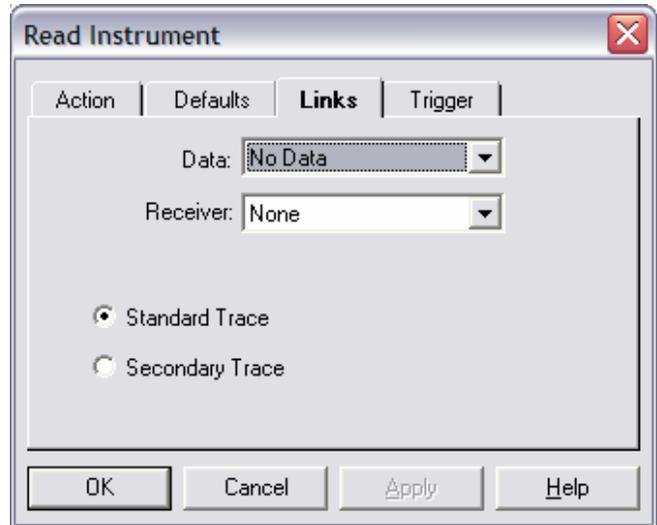
Trigger Tab

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.

- **Trigger Active**—Select this check box for any triggering to be performed.

- **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.

- **Trigger Delay**—A delay time prior to reading after the trigger has happened.
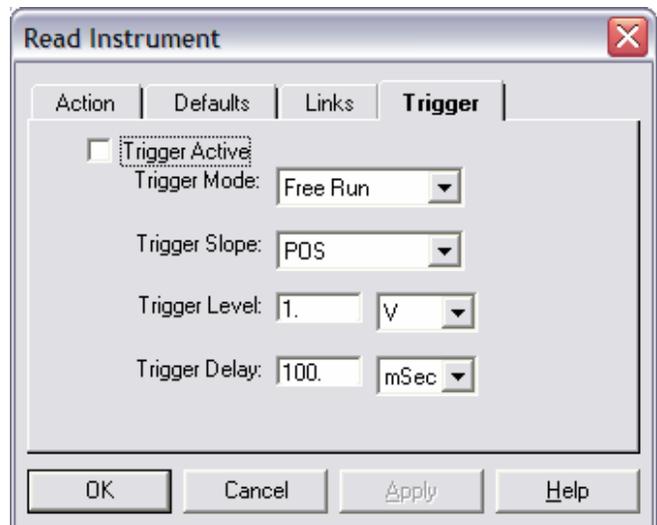
Polar / Scalar Measurement

The Polar action is used to perform a polar (or scalar) measurement across a tower or turntable

Polar / Scalar Measurement icon

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 Data check box is selected, then multiple frequencies will be measured.

- **Set From Data**—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 default/off position, 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.
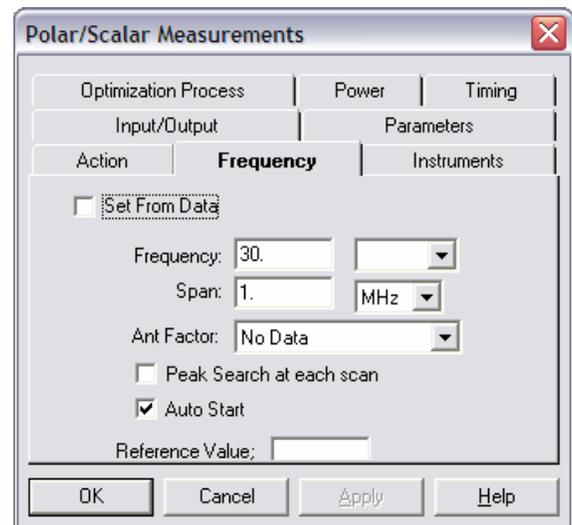
Instruments Tab

The **Instruments** tab allows you to select which instruments are used during this test. A receiver and a tower or turntable is required for this test to operate. If a separate preselector-QP (physically separate bus address) is present it can be specified here. If the quasi-peak and preselector functions are internal to the receiver, they are not specified separately.

If the tower/turntable is not bus-controlled, select the **Manual** check box. During execution this will prompt the operator to move physically to this position.

Select a data element to store the results of this measurement.
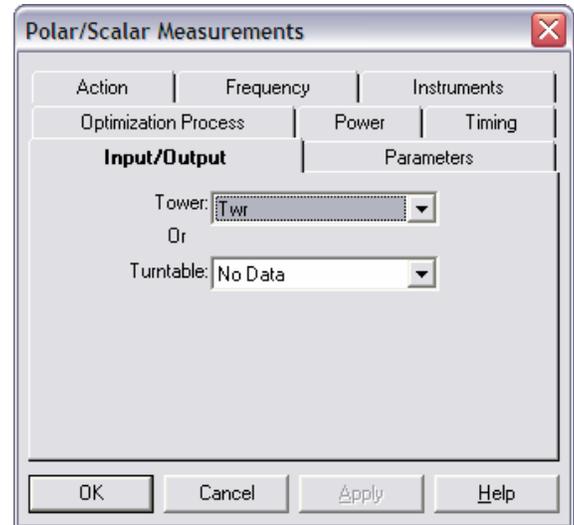
This is the same as other emissions actions and will not be described here.

Position Tower

The Position Tower action provides a control interface to move the tower to various positions as well as change the orientation of the antenna (assuming this function is supported by the tower).

Position Tower icon

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* on page 140, *Scan Peaks* on page 2, and *Common Name Page* on page 2.

Position Tab

Three properties are controlled on the **Position** tab: antenna height (in centimeters), antenna orientation, and margin.

- **Antenna Height**—You can move the antenna height entering the desired number in the box or by using the slider to move the relative position up or down.

- **Polarity 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.
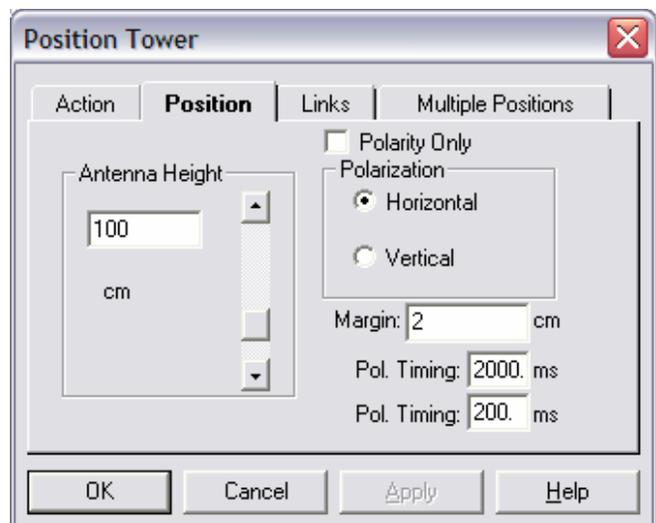
- **Polarization**—To move the antenna orientation, select the desired check box. If the antenna tower is already in this position, no change will take place.

- **Margin**—The margin setting defintes 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 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 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.

Links Tab

The **Links** tab selects the instrument to be used for the action. Click the arrow to display the available instruments, and then select the tower instrument.

- **Clamp Device**—A clamp is a specialized tower placed on the side to facilitate position of a current clamp. This allows the user to move the clamp along the length of a cable to look for maximum emissions. The setup for the clamp is identical to the tower; they both use meters along the length of the positioner. Select this check box if you are using a clamp device.

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.
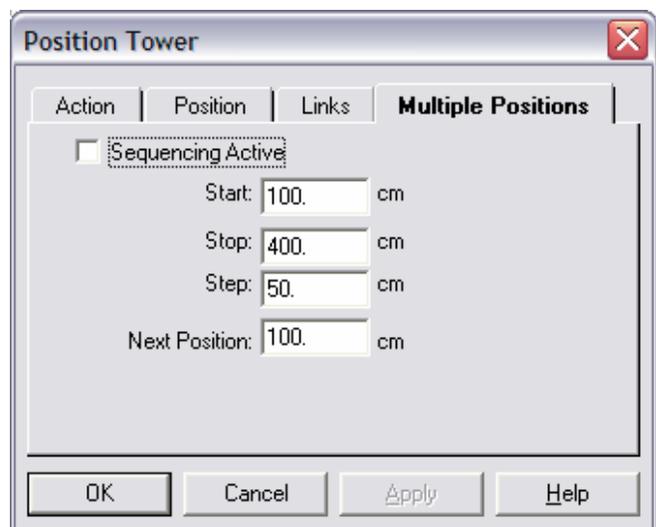
- **Start, Stop, Step, Next Position**—At the beginning and end of a sequencing step, the positioner will move to the start position and perform the sequence. It will move one step position at a time, checking to make sure we are not at or beyond the stop position. It will then perform another sequence, step, and check again. Once the stop position is reached it will exit to the second arrow out of the action. Specify the start, stop, step, and next position values on the **Multiple Positions** tab.

Position Turntable

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

Position Turntable icon

Position Turntable Position Tab

The azimuth property is controlled in the **Position** tab, and 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.
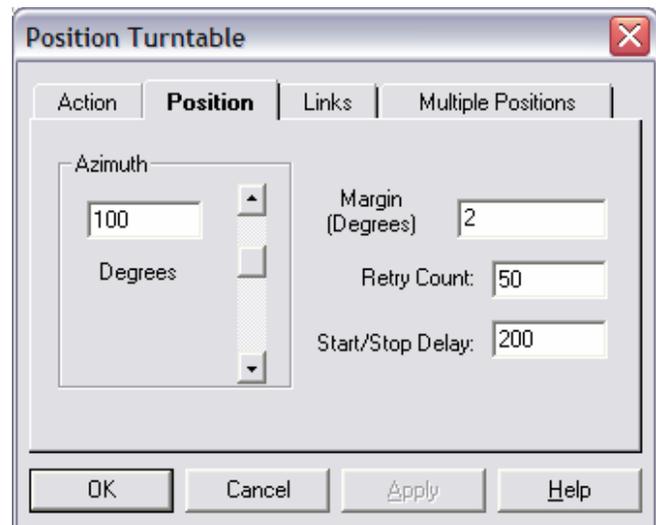
- **Retry Count**—When we first send the start movement command, we begin reading the turntable constantly to determine whether it has reached the target. If we read the same position more than the number of times specified in the Retry Count, an error is displayed. Failure to move is an error condition, but large turntables require a fairly 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

The **Links** tab selects the instrument to be used for this action. Click the arrow to display the available instruments, and then select the turntable instrument.

- **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 its limits.

- **Start, Stop, Step, and 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. Specify the start, stop, step, and next position values on the **Multiple Positions** tab.

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 EUT icon

Position Tab

The **Position** 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, is defined as an X-Y positioner on the **Links** tab; the 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 desired value from 0 to 360 degrees/cm.

- **Ortho**—The orthogonal, or y-axis, is defined as an X-Y positioner on the **Links** tab; the 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 desired value from -180 to +180 degrees or 0 to 400 cm.

- **Preset**—The Preset list allows the selection of preset position information. The user can view a list of preset conditions by clicking the arrow. Any value displayed in the list can be selected by clicking the desired value. 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

The **Links** tab is used to select the instrument that controls the manipulator. The operator can view a list of the instruments by clicking the arrow.

- **Margin**—The margin is a defined increment which 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 period of 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. Selecting either of these affects the Presets default on the **Position** tab.

Switches

There are six types of switch interfaces. At this time documentation is available only for Switch Setup, which is described in the following section.

Switches actions

Some switches allow the user to automate Immunity testing for EUT failures. Most of these are 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.

Switch Setup icon

Switches Tab

The **Switches** tab allows the user to control the state of up to 10 switches, either selectivity controlling an individual switch or by using the All On, All Off, or All N/C option buttons to control the state of all the switches.
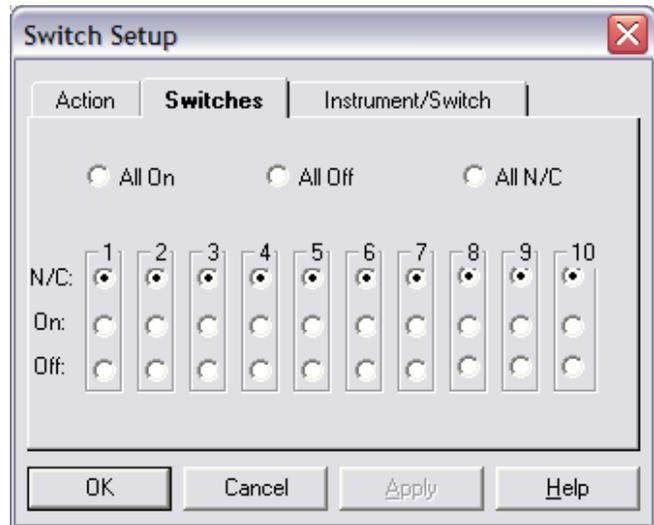
Instrument/Switch Tab

The **Instrument/Switch** tab specifies the instrument to be used during the execution of this action. You must select an instrument from the list.

- **Instrument**—Click the arrow and select the appropriate instrument.

- **Slot/Pin**—When reading switches, it is necessary to specify the slot and pin; enter these here. If you need to monitor multiple switches, configure each separately.

- **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.
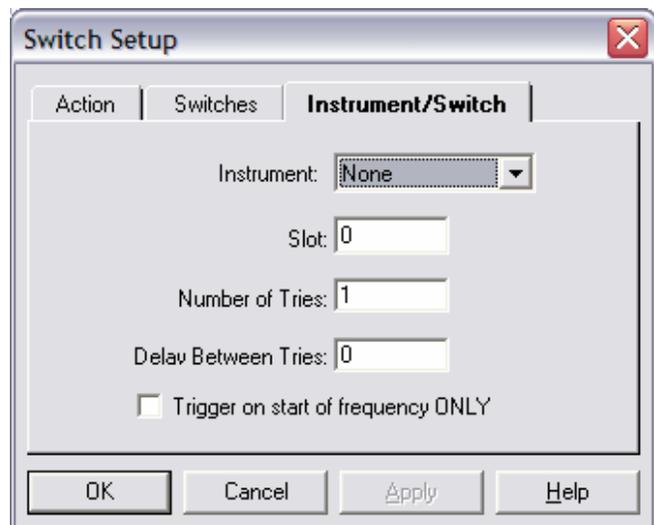
- **Trigger on start of frequency ONLY**—This can be used to trigger an action at the start of testing each frequency, such as the need to reset a switch before testing for a failure. Select this check box if you want to trigger this only on the start of frequency.

About Switches A,B,C,D

Though there is no documentation on these switches, following is a description of the drivers. Go to http://support.ets-lindgren.com for help with these actions.

Icons for switches A, B, C, D, and 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.

EMSwitch

The switch is to be used with the EMSwitch™ RF Switch Plug-in Card.

EMSwitch Icon

- **Instrument**—Click the arrow and 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.
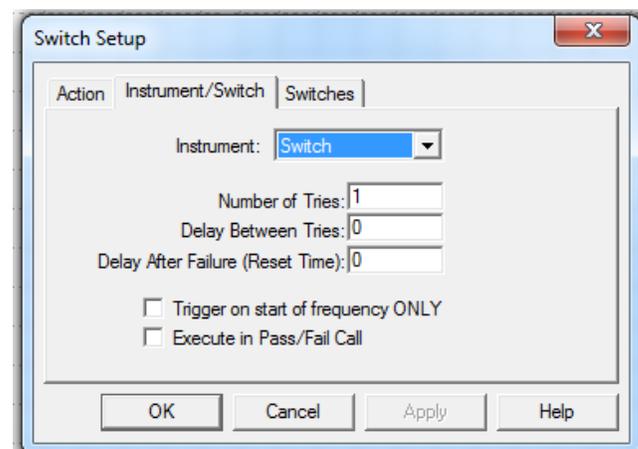
- **Switch Positions**—Select switch A or B and the position for each.
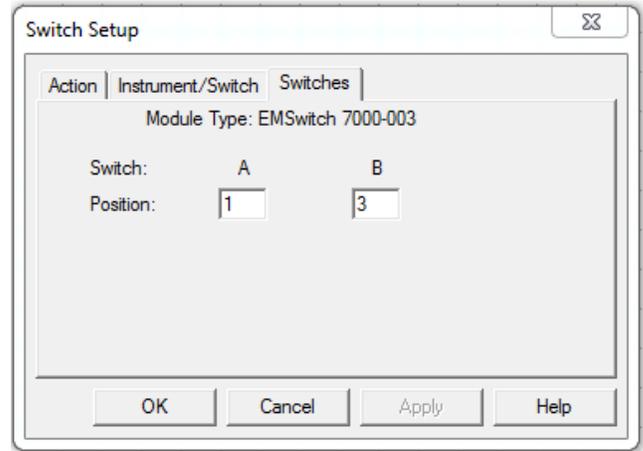
Call Action

Call Action is to be 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. See *Device Config Tab* on page 2 for the steps to setup the telecom tester.

Call Action Icon

Action Tab

- **Name**—Type the name to be displayed in the flowchart.
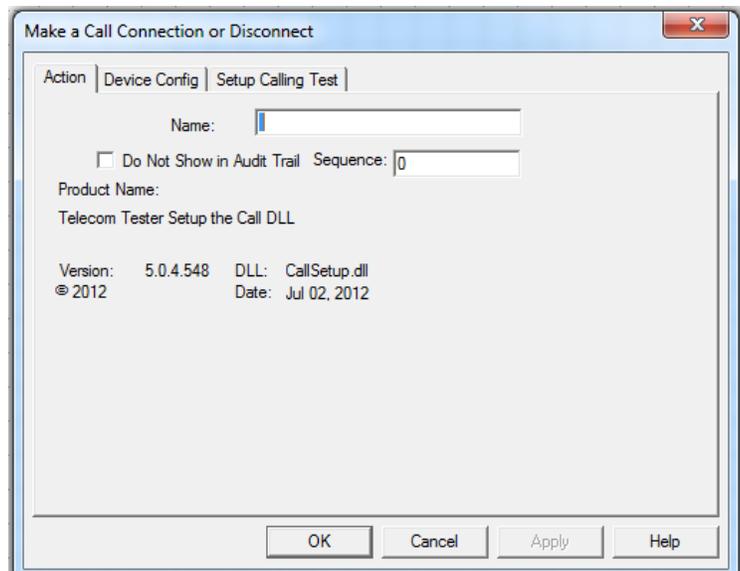
Device Config Tab

- **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 (see 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.
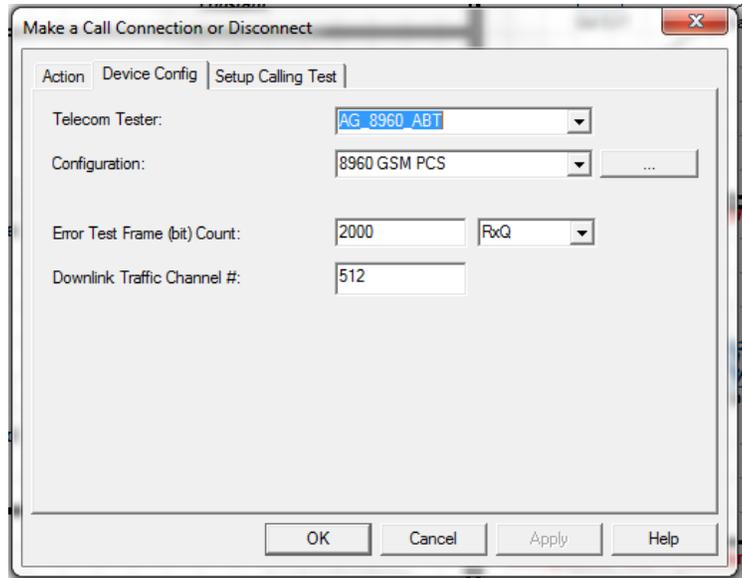
Selecting the Telecom Tester Interface Tab

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

- **File**—Browse to find the ETS_CTInterface.ins file.

A new tab will display, **Communications Tester Settings**.
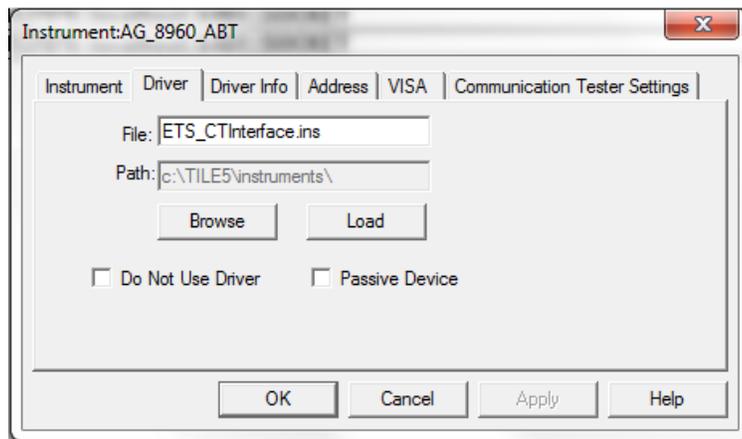
Address Tab

- **Communication**—Select VISA.

- **Primary and Secondary**—These are not used.
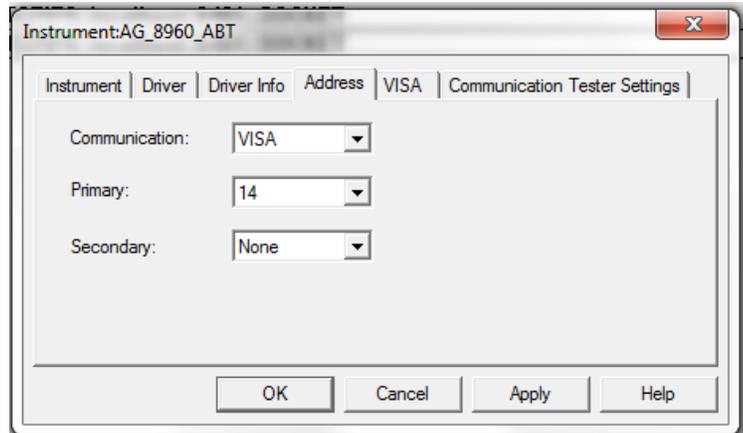
VISA Tab

- **VISA resource name or Alias**—Select the connection to the telecom tester.

- **Terminate Read on EOS**—This is required to work with the ETS_CTInterface as well as any simulated instruments.
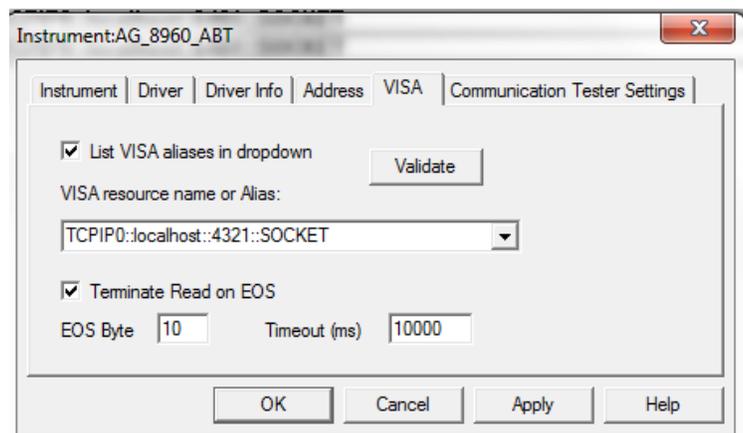
Communication Tester Settings Tab

- **Configuration File**—Select the **…** button and then select the appropriate configuration from the displayed list.
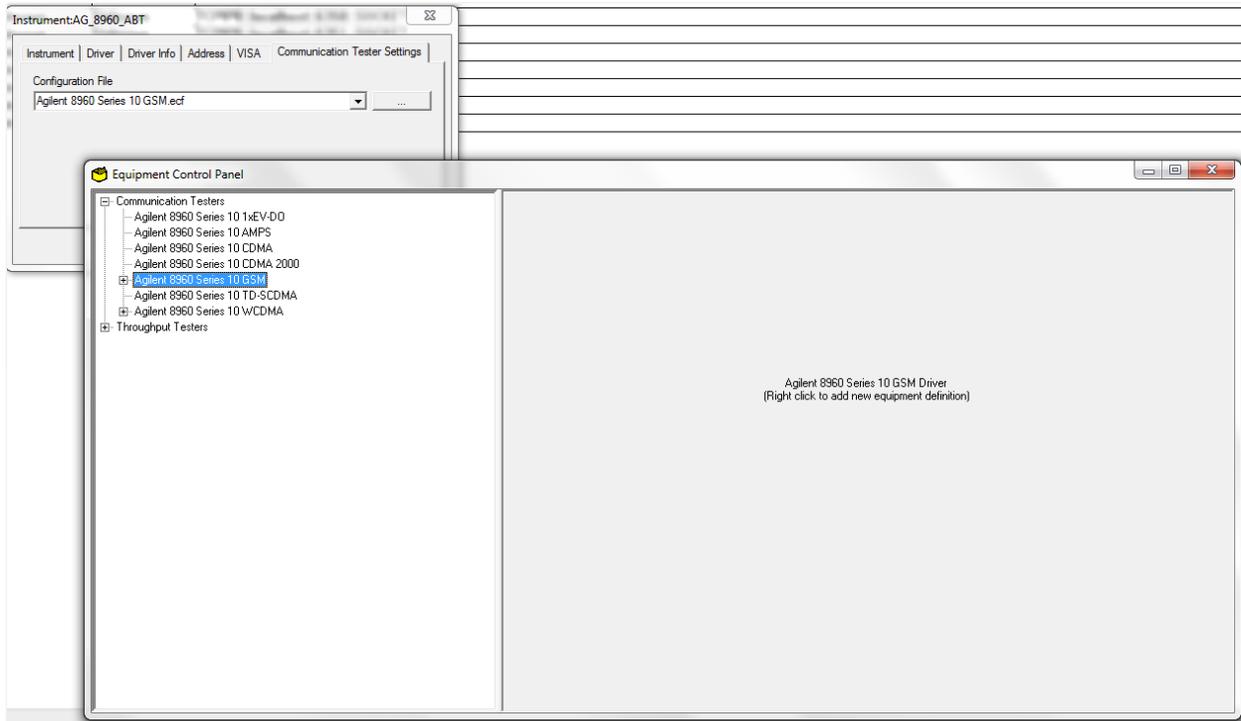
Setup Calling Test

- **Testing Cell Power**—Enter the transmit power level for the telecom tester.

- **Testing Traffic Power**—Enter the power level for the EUT.

- **Time Out**—Set the number in mS for the timeout flag to stop looking for traffic.

- **Connect/Disconnect**—These actions can be used to make a call or to disconnect a call.
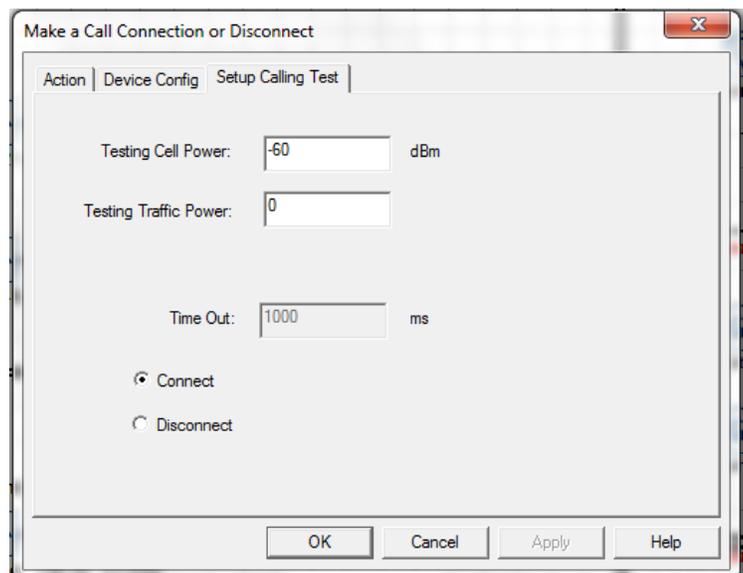
Emissions Actions

Range (Trace) Type Measurements

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 (formally Optimize) is used with turntable and tower positioners.

Range Type Measurement actions

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 **Measure Range 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 **Links** tab.

Measure Range icon

Frequency Tab

The **Frequency** tab allows the operator to select the range parameters over which measurements will be taken.

- **Start Frequency**—Enter an appropriate number in the box and select the units.

- **Stop Frequency**—Enter an appropriate number in the box and click the arrow to 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. 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 2 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. The default is linear scale.

- **From File**—Select the From File check box if you wish to read in the frequency data from an external file. This data must first be imported 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.
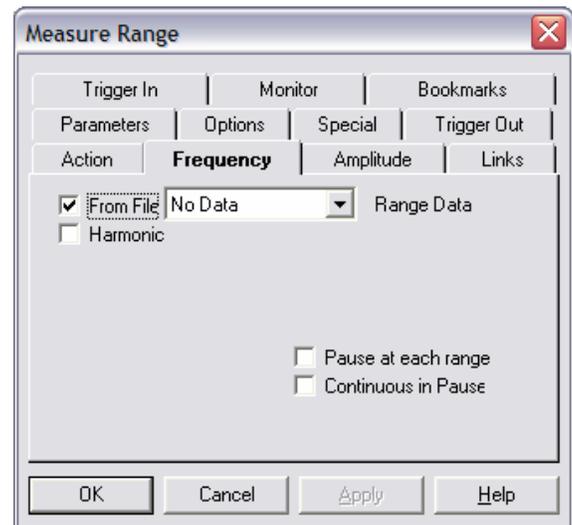
- **Harmonic**—The Harmonic option allows this action to be used to measure a primary frequency (Start Frequency) and then take measurements at a specific number of harmonics. At each frequency, a span for the range measurement will be established using the Bandwidth parameter. In the example shown here 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.
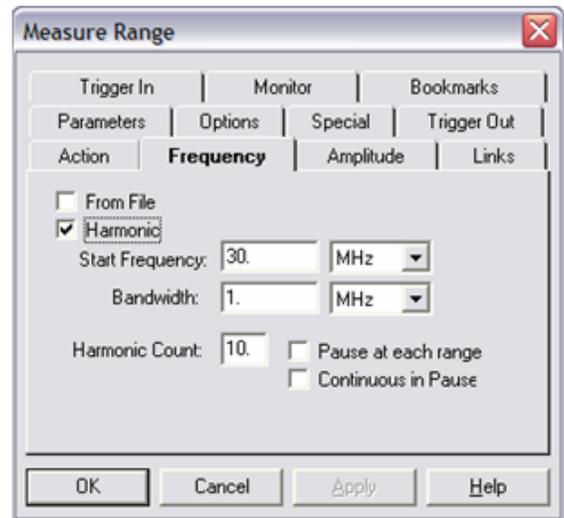
- **Start Frequency**—Enter the desired primary frequency. This will be the first frequency measured. All harmonics are calculated from the frequency.

- **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

The **Amplitude** 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. But on an OATS, this would put the analyzer/receiver in saturation. Enter an appropriate number in the box and click the arrow to select the units.
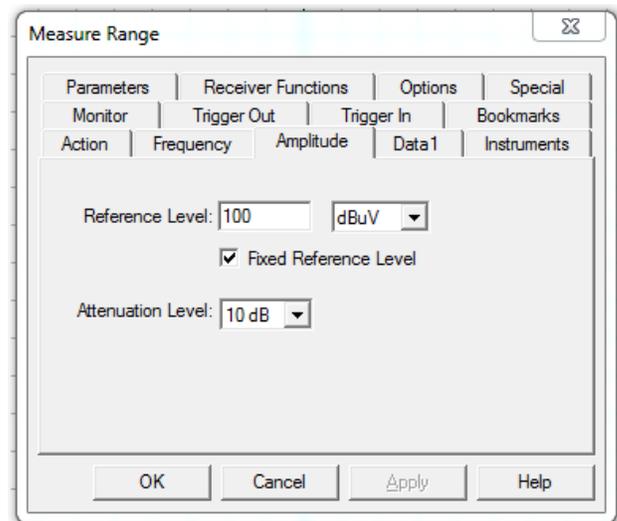
- **Fixed Reference Level**—During scans the default behavior is to set the reference level specified and is not changed during the various scans. If this field is deselected the behavior changes slightly.

  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.

Data1 Tab

The **Data1** 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.

To select the data element click the arrow in the **Measurement Data** list and then choose the appropriate elements from the list.
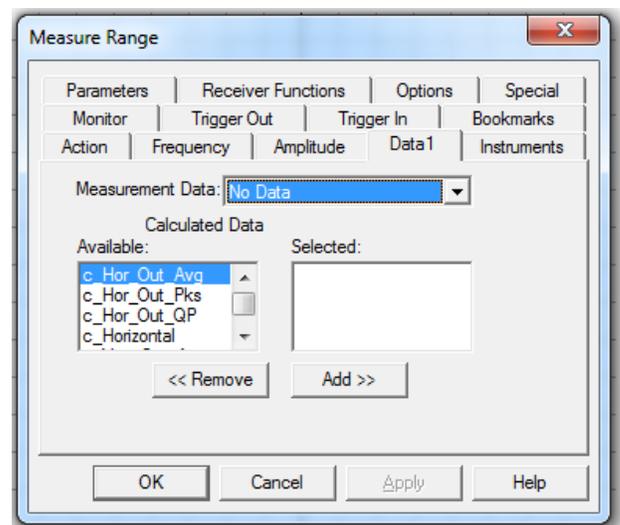
- **Calculated Data**—Within this action the normal return value is the raw data from the scan. Often the user will want to view corrected data while a test is still executing. Data is corrected in TILE! through math equations; if an Equation data element is selected at this point, the equation will 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.

Instruments Tab

The **Instruments** tab is used on almost all measurement actions and is intuitive as to what instrument to install in the appropriate window.

Receiver—Identifies the instrument from which measurements will be taken. Click the arrow to display the defined instruments. One advantage of the TILE! system is the ability to quickly change test instruments. 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.
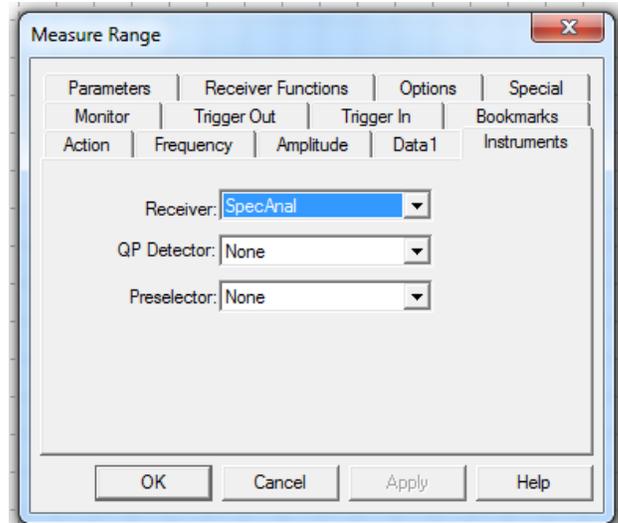
- **QP Detector**—Specifies the QP detector that will be used for the measurement. Only select an instrument if you have a physically separate box for QP measurements. This is true of both the HP85650 QP Adapter and the Anritsu MN1602a. If the analyzer/receiver has a built-in QP function do not specify 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.

Monitor Tab

- **Monitor Active**—Selecting this check box turns on the Monitor action.

- **Monitor Instrument**—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.
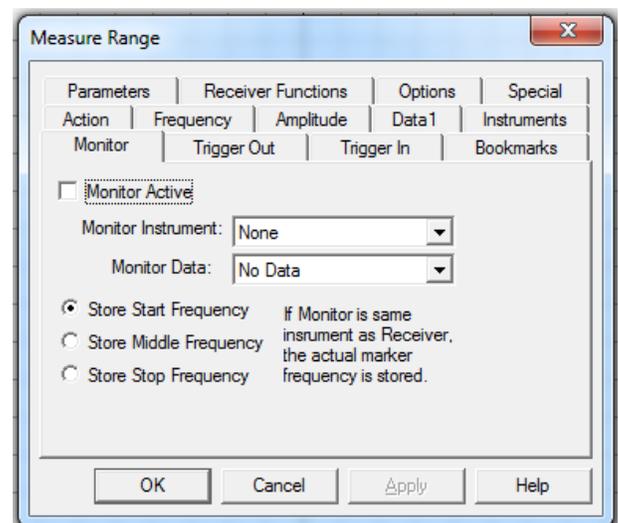
Parameters Tab

The **Parameters** 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 **Links** tab, the RF bandwidth will be set for the QP adapter.
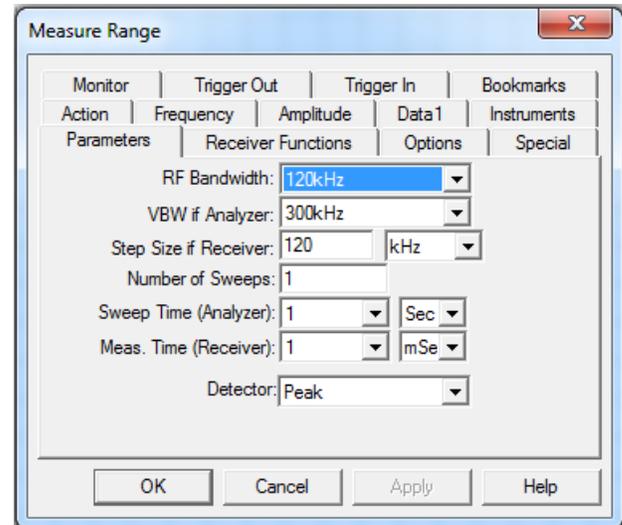
- **VBW if Analyzer—**Spectrum analyzers use a video filter as part of the measurement/display cycle. Generally, the video bandwidth (VBW) should be equal to or greater than the RBW. A good rule of thumb is three times the RBW. Set this value as needed or required. This value is generally ignored when 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 1 will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver 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). Enter an appropriate number in the box and click the arrow to select the units.

Setting this value to greater than 10 seconds can cause the instrumentation to time out. If this happens, change the GPIB/setup/timeout settings in the instrument definition to a value higher than the sweep time. See *Timeout* on page 2 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. Click the arrow to select the appropriate type.

Options Tab

The **Options** tab controls the state of the action when it starts execution.

Generally speaking, 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 someplace other than the start.

- **Auto Start**—The default setting is on; when this check box is selected, this action will execute as defined. 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.

  With Auto Start selected, you are taking a defined executing step such as a compliance test.

  Be aware that if the Auto Start check box is deselected, a different dialogue box will appear when this action is executed. You will be able to change the harmonic and continuous options each time the action executes.

- **Auto Save between each Range**—With this check box is selected, when multiple sub ranges exist, 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 0 and there is no data in the data element, you will not have valid readings in the earlier sub ranges.
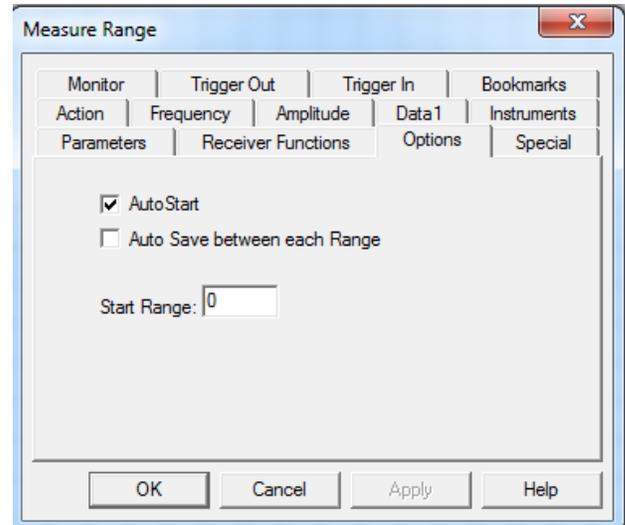
Special Tab

The settings on the **Special** tab 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.

- **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.
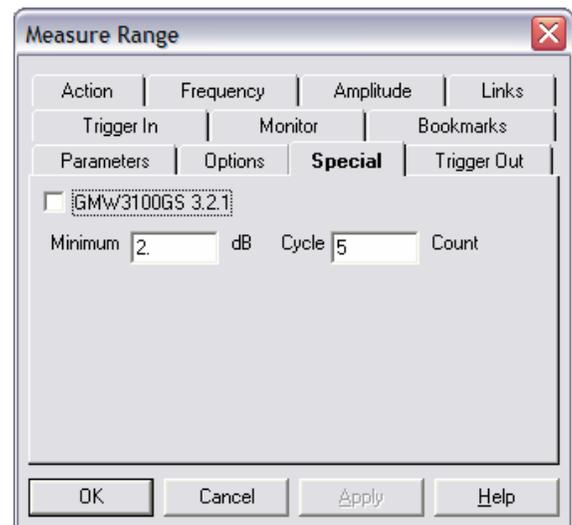
Trigger Out Tab

The **Trigger Out** tab is used to trigger an external oscilloscope measurement.

- **Trigger Active**—Select the check 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.
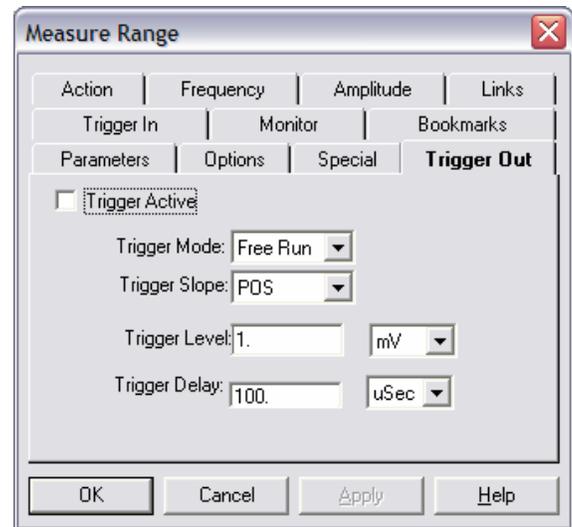
Trigger In Tab

- **External Trigger**—Select the check box for the external event to trigger.

- **Trigger Action**—Defines an action on the flowchart that controls the trigger event.

Measure Range Dialog Box

When executed, the **Measure Range** dialog box will open. This 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.
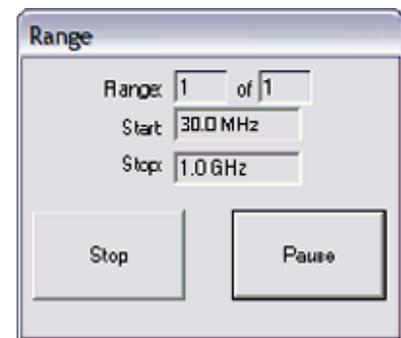
Measure Range Extended

The Measure Range Extended action 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 and two detectors on the parameters tab.

Measure Range Extended icon

Pre-Scan

The Pre-Scan action is used to perform a measurement of 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.

Pre-Scan icon

Freq – Analyzer 1 Tab

The **Freq – Analyzer 1** tab allows the operator to select the range of parameters over which measurements will be taken.

- **Start Frequency**—Enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—Enter an appropriate number in the box and click the arrow to select the units.

- **No. of Ranges**—Enter a value in the box to set 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. 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.
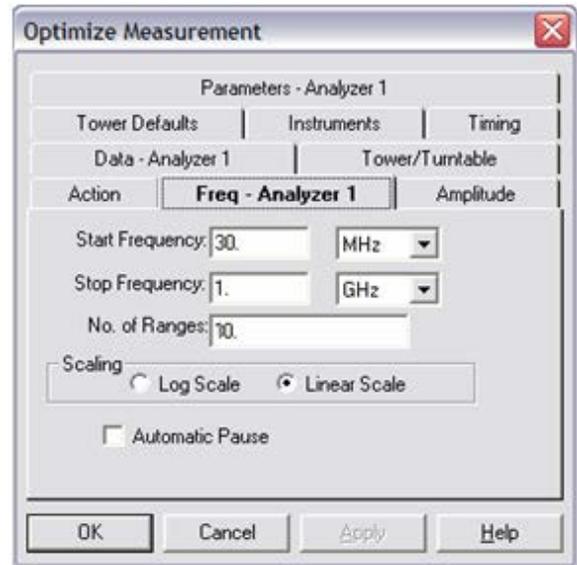
In designing the measurement, knowledge of the capabilities of the analyzer/receiver is critical. If the spectrum analyzer has a 401 dot resolution, then when you do a scan the instrument will only send 401 points, regardless of the bandwidth you specified in the parameters. If you are making a measurement 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

- **Reference Level**—The reference level 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 to 60-80 dB and take valid readings. But on an OATS, this would put the analyzer/receiver in saturation.

    Enter an appropriate number in the box and click the arrow to select the units.
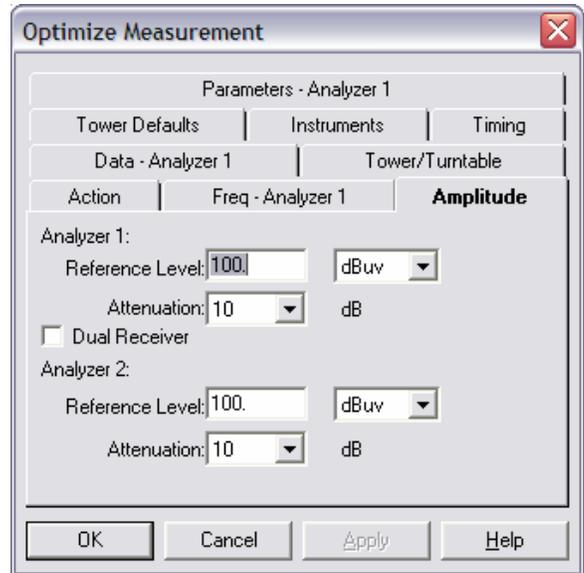
- **Attenuation Level**—Specifies the attenuation level to be used for the scans. This allows you to adjust the attenuation to match the environment or EUT. You can choose values from 0-50 dB or AUTO. We do not generally recommend AUTO; some spectrum analyzers change other settings within the analyzer 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.

- **Dual Receiver**—This action has an option to utilize two different analyzers and towers to speed testing. See *Two Receiver Option* on page 140 for more information.

Data - Analyzer 1 Tab

The **Data – Analyzer 1** tab allows the user to specify the storage data elements for the maximum value and the related tower and turntable positions for this maximum.

- **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.

    To select the data element, click the arrow, and choose the appropriate data elements from the list.
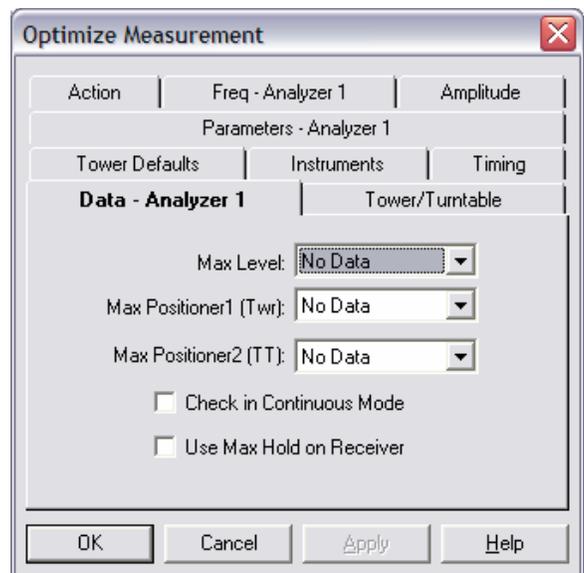
- **Max Positioner1 (Twr)**—Identifies the data storage element to be used to record the height or angle of the maximum value. This is dependent upon whether you are using a tower or turntable controller as Positioner1. If a positioner is identified on the **Instruments** tab, a data element is mandatory.

  To select the data element, click the arrow, and choose the appropriate data elements from the list.

- **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 Positioner2. If a positioner is identified on the **Instruments** tab, a data element is mandatory.

  To select the data element, click the arrow, and choose the appropriate data elements from the list.

- **Check in Continuous Mode**—When this check box is selected the turntable (Positioner2) will be put into continuous motion instead of stepped motion.

- **Use Max Hold on Receiver**—When this check box is selected, the instrument will be put in Max Hold during all sweeps. If this check box is deselected (the default), single sweeps will be taken and the maximum will be kept internally.

Tower/Turntable Tab

The **Tower/Turntable** tab determines the range of motion for the two positioners. If a tower is selected for a positioner (usually Positioner1), then all references are in centimeters. If a turntable is selected, then all references are in degrees.

- **Margin**—The margin setting is used to control the slippage of the positioner. The stop command will be sent to the positioner by an amount equal to the stop position margin. This setting will be site- and test-specific due to the differences between positioners and the equipment placed upon them.

- **Polarization**—Select the appropriate polarization for this measurement test. If the positioner does not have control of the polarity, no action is performed.
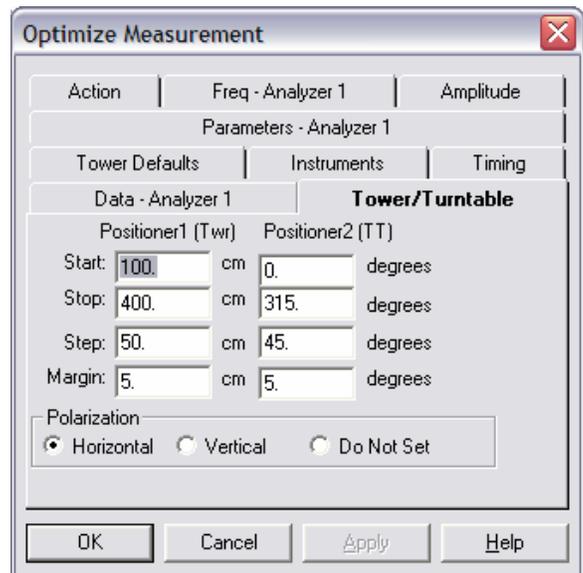
Tower Defaults Tab

- **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 a physical amount the tower will move up or down prior to rotating the antenna polarity in order to avoid this potential problem.
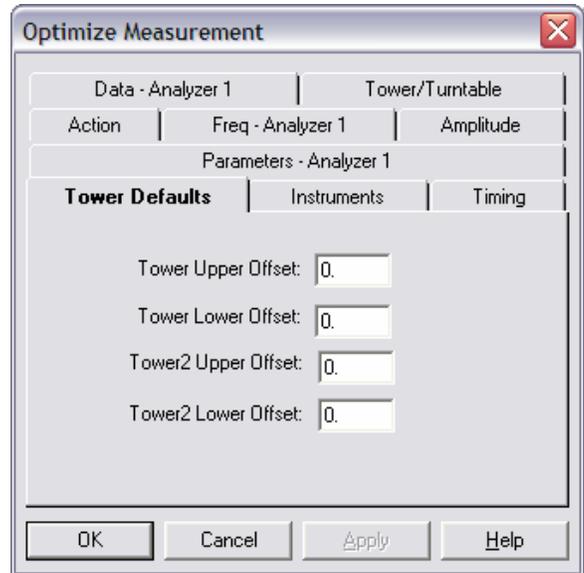
Optimize Measurement Instruments Tab

The **Instruments** tab is used to identify the instruments from which measurements will be taken. One advantage of the TILE! system is the ability to quickly change test instruments. If you design the parameters and frequency ranges properly, 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.

Positioner1 and Positioner2 must be a tower or turntable, but there is no requirement for which instrument occupies which precedence. If Check in Continuous Mode is not set on the **Data-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.
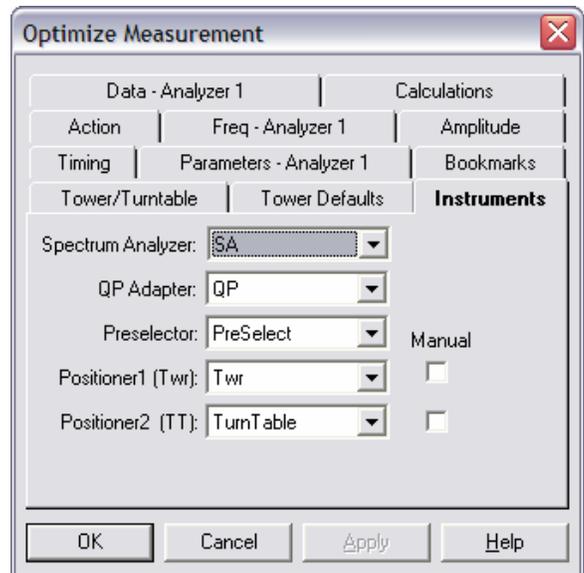
- **QP Adapter**—Specifies a QP detector only if there is a completely separate instrument for this function.

- **Preselector**—Identifies a preselector only if there is a separate instrument. This also needs to be identified. Click the arrow in each instrument list to display the available instruments.

- **Positioner1 (Twr) /Positioner2 (TT)**—The choice of positioner determines whether angular and/or height information is captured during the execution of this action. The instrument needs to be identified on the **Instruments** tab. Click the arrow in each instrument list to display the available instruments.

- **Tower/Turntable Start/Stop Timing—**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.
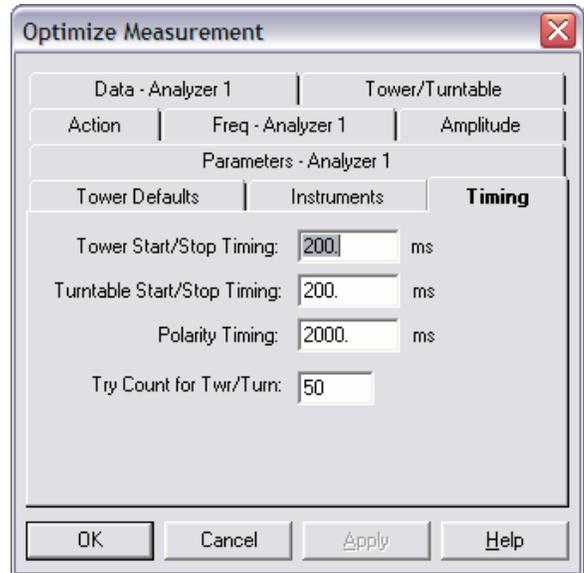
When a start command is sent, some towers and turntables exhibit brush noise during the initial start surge. The start/stop timing value is an amount of delay that will be put between the start command and the next instrument read/motion. This will ensure that noise is avoided as much as possible. Some 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 is 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.

- **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 given 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.

The **Parameters – Analyzer 1** tab is used to specify the instrument setup for this measurement.

- **RF Bandwidth—**Click the arrow to display the available RF bandwidth settings. Not all instruments will be able to use each of these settings, but the instrument driver will select the closest setting that is valid for the specified receiver. For instance, many older receivers only do 9 kHz and 120 kHz RF bandwidths. For all bandwidths at or above these settings, the driver will automatically select the appropriate setting.

- **VBW for Analyzer—**Click the arrow to display the available video bandwidth (VBW) settings. Not all instruments will be able to use each of these settings, but the instrument driver will select the closest setting that is valid for the specified analyzer.

- **Step Size for 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 then dividing the result by the step size.

- **Number of Sweeps—**A setting greater than 1 will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

- **Sweep Time (Analyzer)—**How long the analyzer will take to sweep across the frequency band. Enter an appropriate number in the box and click the arrow to select the units.

- **Meas. Time (Receiver)—**True receivers take individual measurements and then group this information into a sweep result. The measurement time (MT) is the time increment per frequency step.

- **Detector—**Allows selection of the measurement detector desired for this scan. The available options are Peak, Quasi-Peak, Average, and Sample. Click the arrow to select the appropriate type.
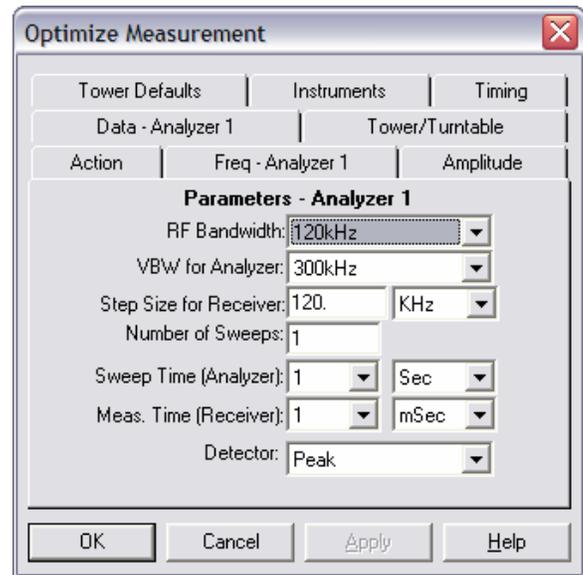
The **Optimize Measurement** dialog box provides visual information on the current range that is being performed. The start and stop frequencies for this range are displayed. Tower or turntable target and current position information are displayed in the appropriate boxes.

- **Stop—**The **Stop** 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—**The **Pause Cycle** 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.
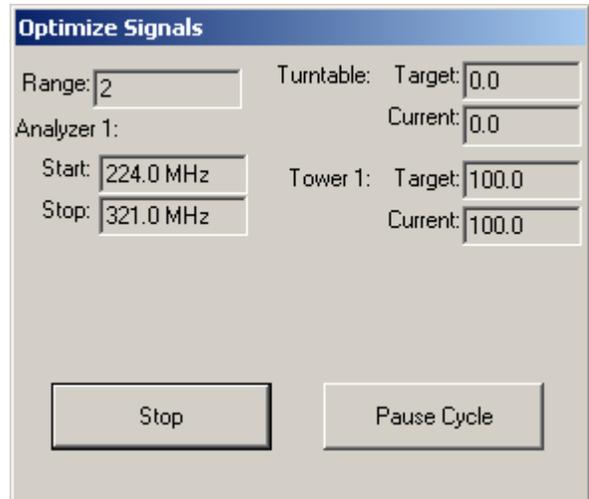
Two Receiver Option

The Optimize Measurement action has a special feature that allows for simultaneous measurements of two different analyzers/receivers controlling two different towers during a rotation of the turntable. On the **Amplitude** tab (see page 2) there is a **Dual Receiver** check box. If selected, four additional tabs appear: **Freq-Analyzer 2**, **Data-Analyzer 2**, **Secondary Instruments**, and **Parameters-Analyzer 2**.

The settings in these four tabs mirror the settings for the corresponding tabs **Freq-Analyzer 1**, **Data-Analyzer 1**, **Instruments**, and **Parameters-Analyzer 1**. There are a few options that are not available, but generally they are the same. 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.

Scan Range

Scan Range is similar to 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).

Scan Range icon

Fast Scan Across TT/TWR

The Fast Scan Across TT/TWR (Quick Scan Measurement) 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 very similar to the Pre-scan action except that it uses only one positioner.

Fast Scan Across TT/TWR icon

Peak Type Measurements

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.

Peak Type Measurement actions

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. There is a Scan Peaks PSA action that 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.

Scan Peaks and Scan Peaks PSA icons

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 selected in the **Search** tab, it will open a search span, 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 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 action; see *Direct Entry* on page 2.

- **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. Click the arrow and select the data element that contains the points of interest.
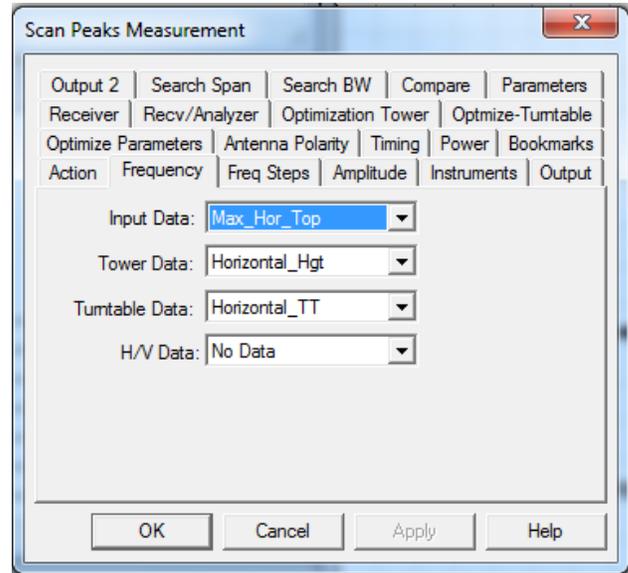
Freq Steps Tab

- **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.
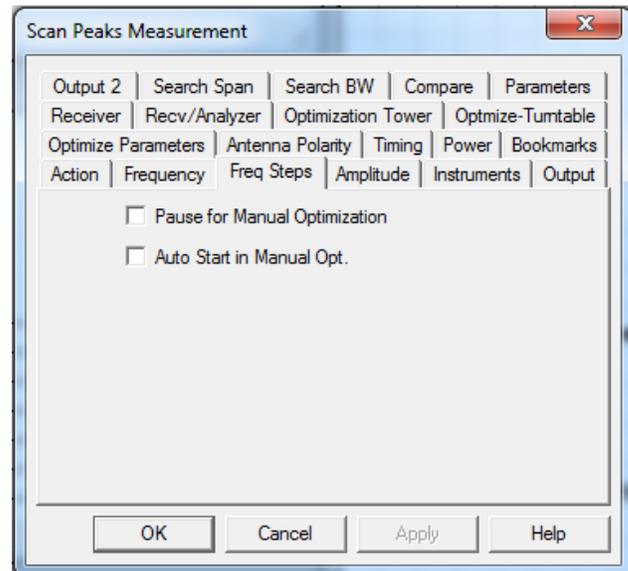
Amplitude Tab

- **Reference Level**—The reference level 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 to 60-80 dB and take valid readings. But on an OATS, this would put the analyzer/receiver in saturation. Enter an appropriate number in the box and click the arrow to select the units.
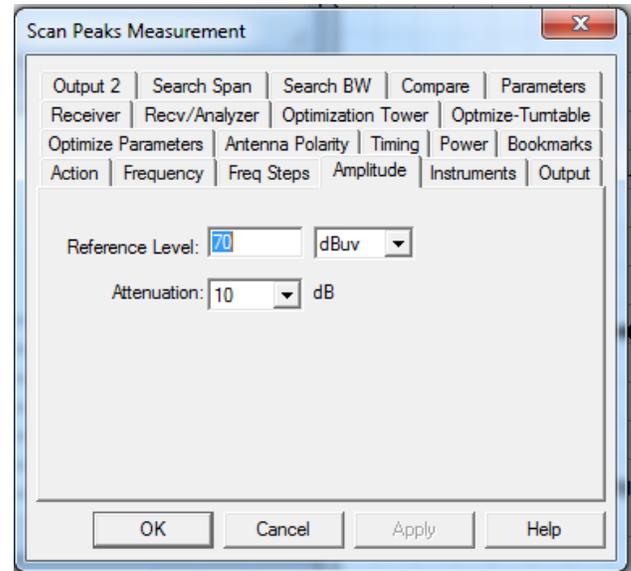
- **Attenuation**—Specifies the attenuation level to be used for the scans. This allows you to adjust the attenuation to match the environment or EUT. You can choose values from 0-60 dB or AUTO. We do not generally recommend AUTO; some spectrum analyzers change other settings within the analyzer when 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.

Instruments Tab

The **Instruments** tab identifies which instruments will be addressed by this action. At a minimum you must specify the analyzer. Click the arrow and select the appropriate instrument from the list.

- **QP Adapter**—If you are making QP measurements and have a completely separate instrument to measure QP, you will need to specify the QP adapter. If the analyzer/receiver has this feature built-in you do not specify it here.
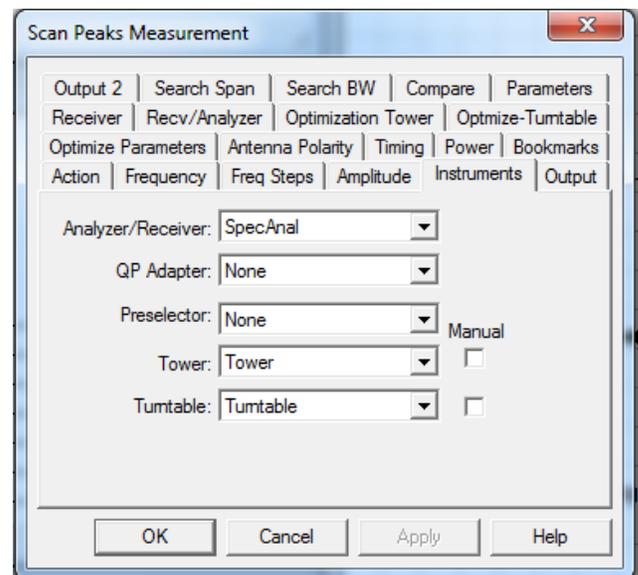
- **Preselector**—The preselector is specified if there is a separate instrument for this function. Identify tower and turntable controllers as needed or when available. If no tower is available, or it is not GPIB-controlled, select the Manual check box to configure an operator interrupt. This will allow 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.

Output / Output 2 Tabs

On the **Output** 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.

The **Output 2** 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. Select the items wanted and choose an appropriate data element from the list. 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.
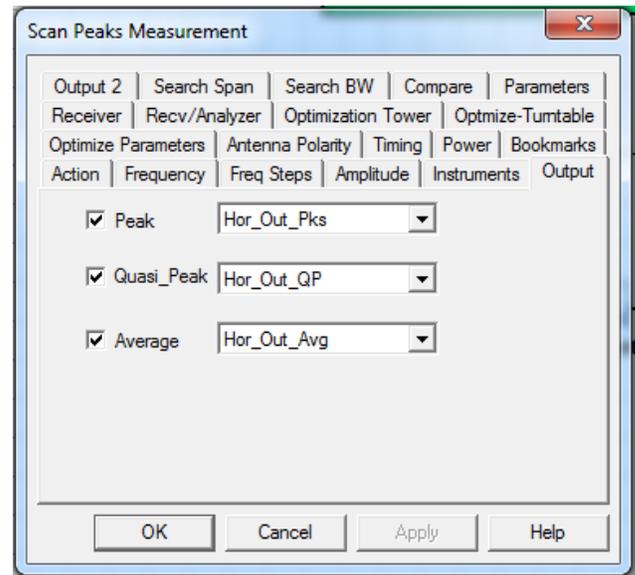
Search Span Tab

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 is a fixed span that 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.
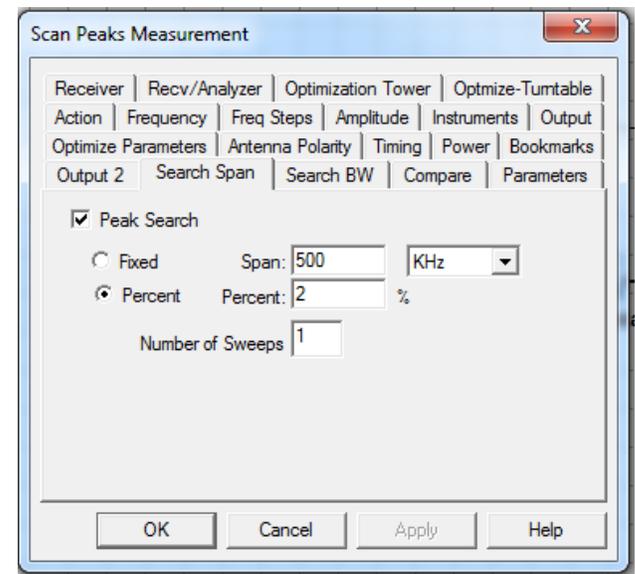
Search BW

- **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/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.
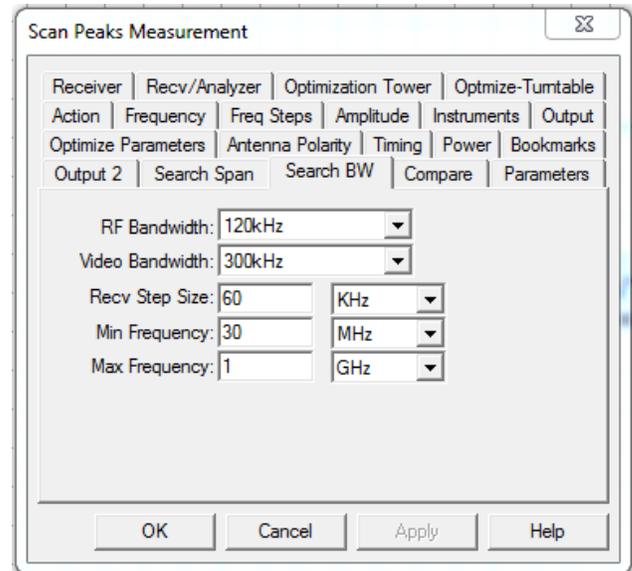
Compare Tab

- **Comparison Check**—This is a special feature of the TILE! system which 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.

- **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.
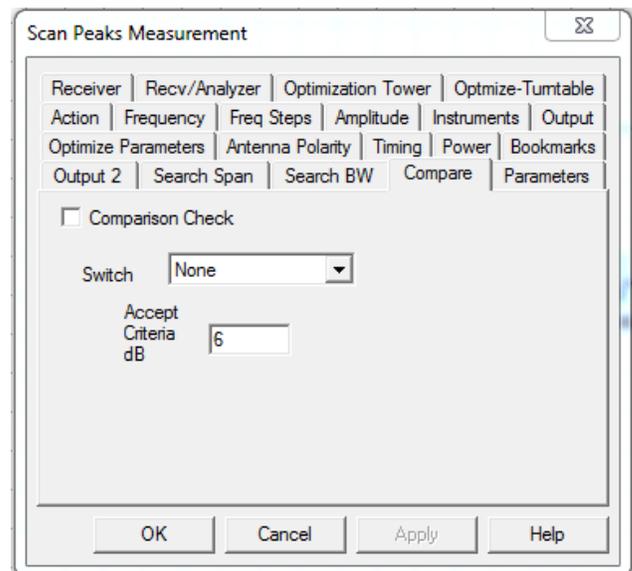
Scan Peaks Parameters Tab

The **Parameters** tab is used to specify the instrument setup for this measurement.

- **RF Bandwidth**—Click the arrow to display the available RF bandwidth 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 receiver. For instance, 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 select the appropriate setting.
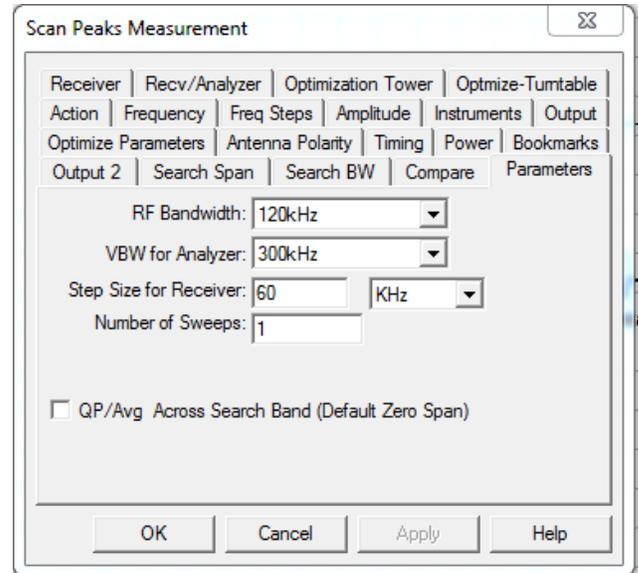
- **VBW for Analyzer**—Click the arrow to display the available video bandwidth (VBW) settings. Not all instruments will be able to use each of these settings, but the instrument driver will select the closest setting that is valid for the specified analyzer. For receivers, this setting determines the step size for readings. Generally speaking, this parameter should be the same or smaller than the RF bandwidth for receivers.

- **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.

- **Number of Sweeps**—A setting greater than 1 will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

- **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. 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.

- **Search Time/ Measurement (Analyzer/Receiver)**—The time value for the measurements on the analyzer/receiver while in a search mode looking for the peak signal.

- **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).

- **Optimize Sweep/Measurement Time**—The time value for the measurements on the analyzer/receiver while in a search mode looking for the peak signal. This is used during the tower and turntable motion.

- **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.
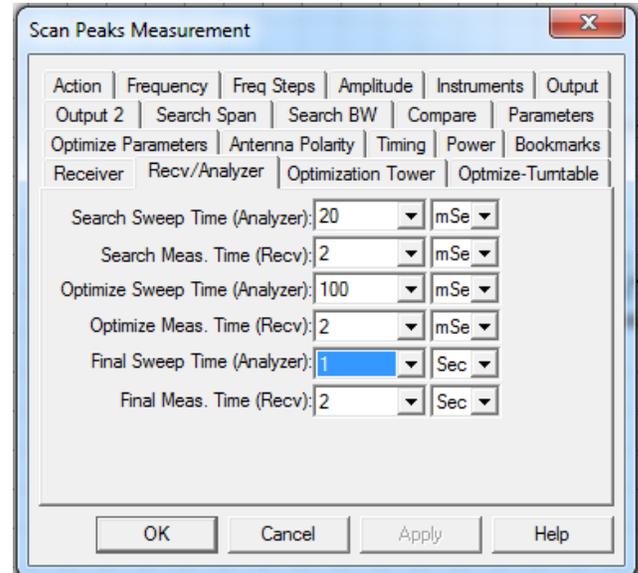
Optimization Tower / Optimize-Turntable Tabs

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.

There are two methodologies available to perform this optimization, re-optimize and partial optimize.
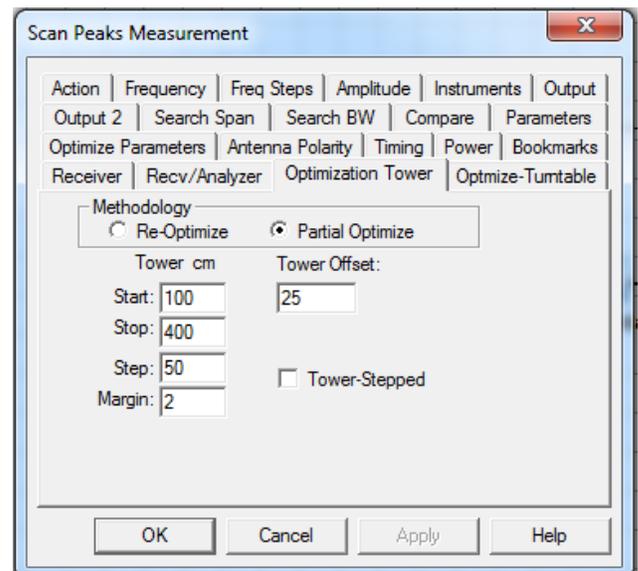
- **Re-Optimize**—Assumes that the earlier readings were not taken with enough tower height, or turntable movement, to adequately predict relative position. In this case, the tower and turntable need to be scanned through their full range of motion to determine the peak signal. This method starts by 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.

Optimize Parameters Tab

During the Optimization process the tower is scanned to find the maximum signal. The **Optimize Parameters** 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. If the EUT is definitely the strongest signal, this will work 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, 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 better exclude ambients when the tower is in motion.
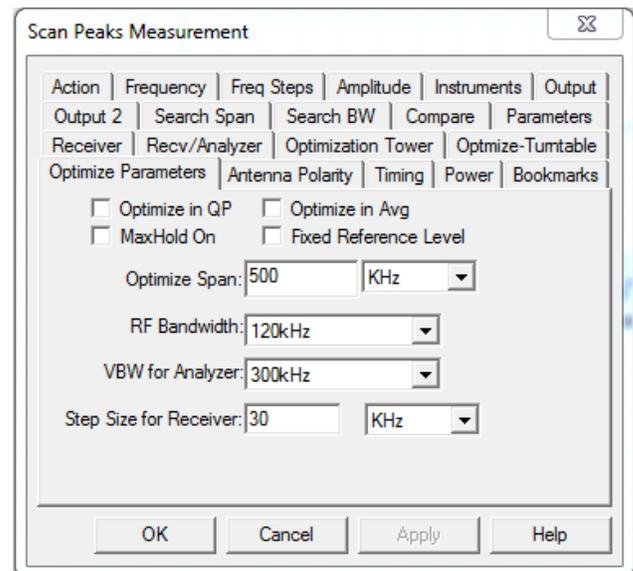
- **RF Bandwidth / VBW for Analyzer**—Click the arrow to display the available RF (RBW) and video bandwidth (VBW) settings. Not all instruments will be able to use each of these settings, but the instrument driver will pick the closest setting that is valid for the specified receiver. For example, many 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

The **Antenna Polarity** tab is used to set the respective polarity for the antenna during execution of this action. Following is a description of the three options.

- **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 in order to avoid this potential problem.
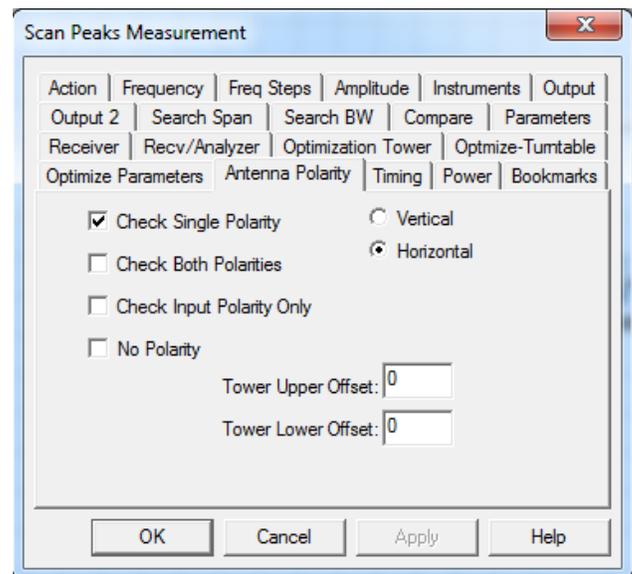
Timing Tab

Use the settings on the **Timing** tab to improve 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.
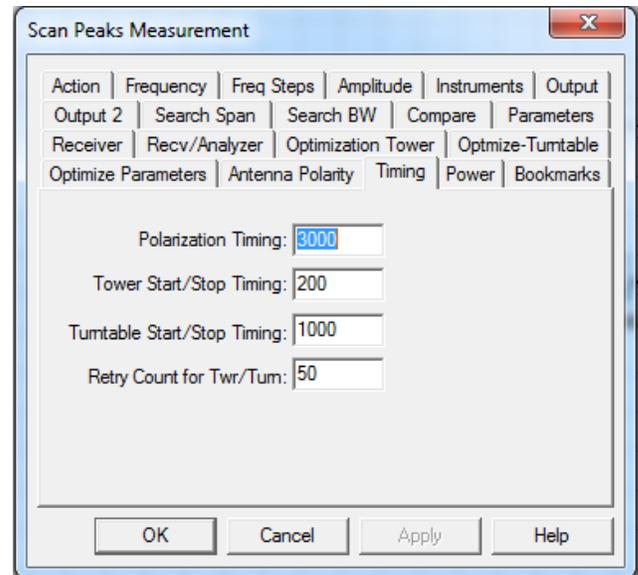
- **Tower/Turntable Start/Stop Timing—**Specifies a minimum timing interval between the stop motion and any other movement command. This will ensure accurate direction and motion. On some positioners (both towers and turntables), if they are issued the stop command and then immediately issued a change 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.

- **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, as shown here, but if the **Not Constant** check box is selected, then you can enter the name of a data element that will contain the desired amplitudes and frequencies.

- **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.
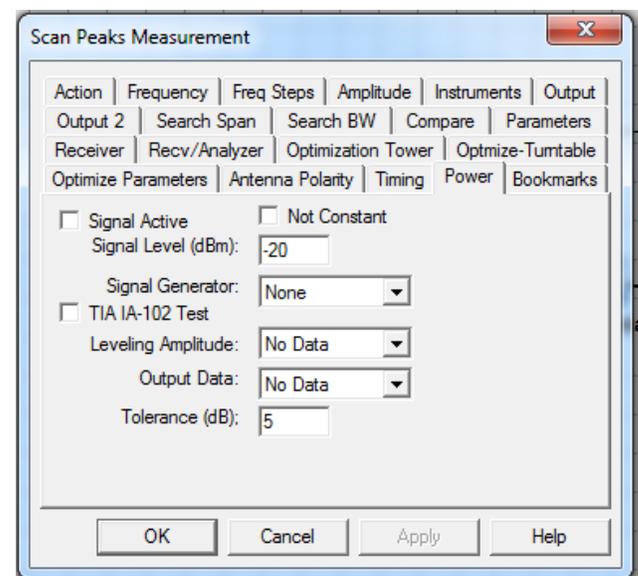
- **Leveling Amplitude**—Select a date element that has the desired RF level.

- **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).

- **Scan Peaks Dialog**—The **Scan Peaks** dialog box 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.
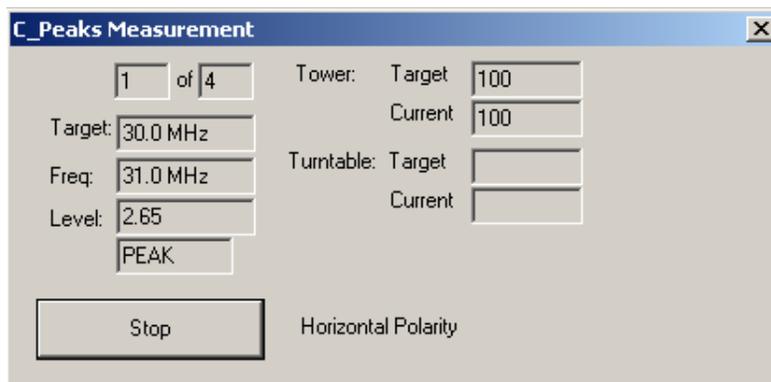
The **Stop** button provides the ability to halt execution of the test at this point. When clicked, the system will complete the current sequence of GPIB commands and then halt execution. This is to ensure the GPIB is not left in an inoperable condition. The current polarity is noted on the dialog, for 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.

Measure Peaks icon

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. Click the arrow and select the data element that contains the points of interest.

- **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.
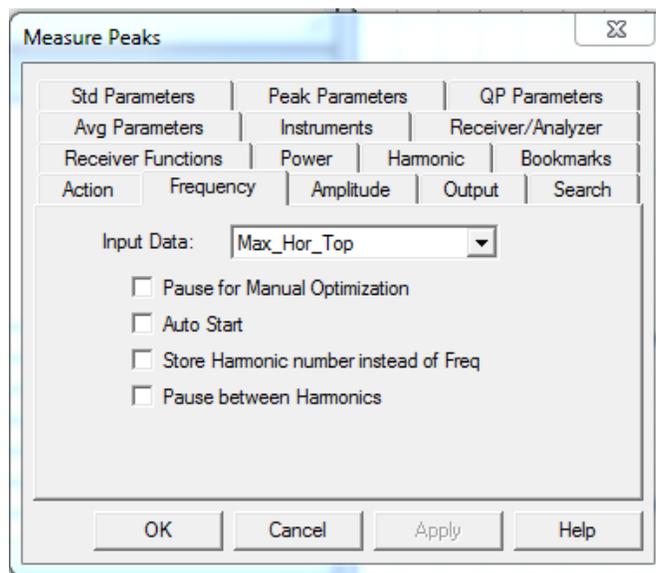
- **Auto Start**—Automatically steps the measurement through the desired frequency points. Selecting the **Auto Start** check box overrides the **Pause for manual optimization** check box, but the **Measure Peaks** dialog box gives the operator the ability to halt the test and then step through frequency points through manual selections on the dialog box.

- **Store Harmonic number instead of Freq**—When Harmonic is active (see **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, 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

The **Amplitude** 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.

- **Reference Level**—The reference level 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 to 60-80 dB and take valid readings. But on an OATS, this would put the analyzer/receiver in saturation. Enter an appropriate number in the box and click the arrow to select the units.
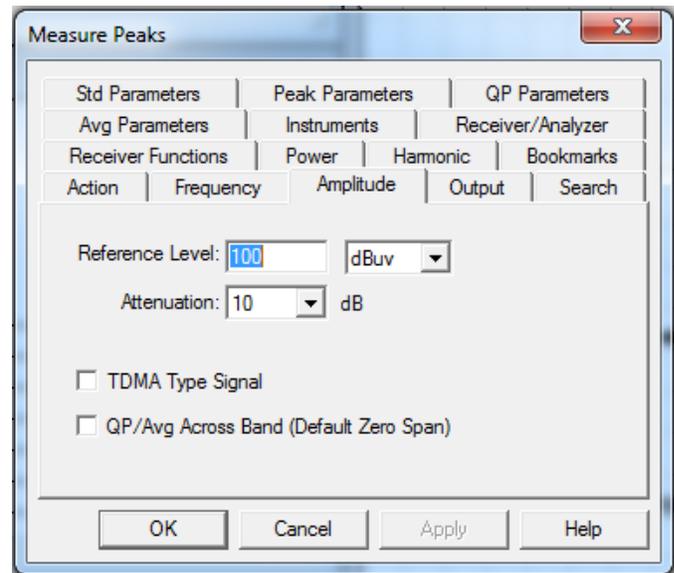
- **Attenuation**—Selects the attenuation level to be used for the scans. This allows you to adjust the attenuation to match the environment or EUT. You can choose values from 0-50 dB or AUTO. We do not generally recommend AUTO; some spectrum analyzers change other settings within the analyzer when the 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.

- **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 signals 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 either detector, and do a marker to peak, storing the maximum value in the span.

Measure Peaks 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 fairly 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.
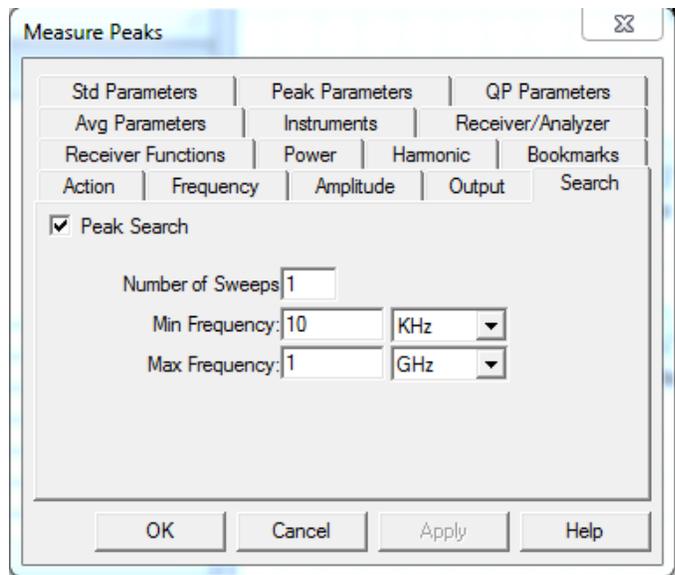
- **Peak Search**—When this box is selected, you then specify the band around which you want to search. For example, if you are taking measurements that will typically see two signals within 2 MHz of each other you might specify a band search of less than 1 MHz. This will ensure that you do not identify 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 than 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

The **Std Parameters** tab is used to specify the instrument setup for this measurement.

- **RBW**—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.

- **VBW for 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 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.

- **Number of Sweeps**—A setting greater than 1 will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

- **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 the **Peak Parameters** tab are used only for the final peak measurement after the search has been completed.

- **RBW**—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 a 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.

- **VBW for 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 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.

- **Number of Sweeps**—A setting greater than 1 will cause the analyzers to be put in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.
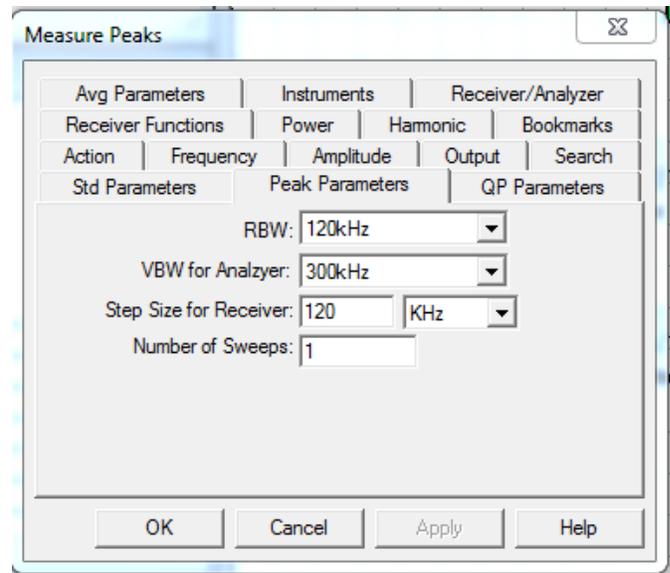
QP Parameters Tab

The parameters on the **QP Parameters** tab are only used for the final QP measurement after the search has been completed.

- **RBW**—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 a 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.

- **VBW for 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 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.

- **Number of Sweeps**—A setting greater than one will cause the analyzers to be put in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

Avg Parameters Tab

The parameters on the **Avg Parameters** tab are only used for the final average measurement after the search has been completed.

- **RBW**—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 a 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.

- **VBW**—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**—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.

- **Number of Sweeps**—A setting greater than one will cause the analyzers to be put in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

Receiver/Analyzer Tab

The settings on the **Receiver/Analyzer** tab are used for controlling time for the instrumentation during the various stages of the measurement.

- **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.

- **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.
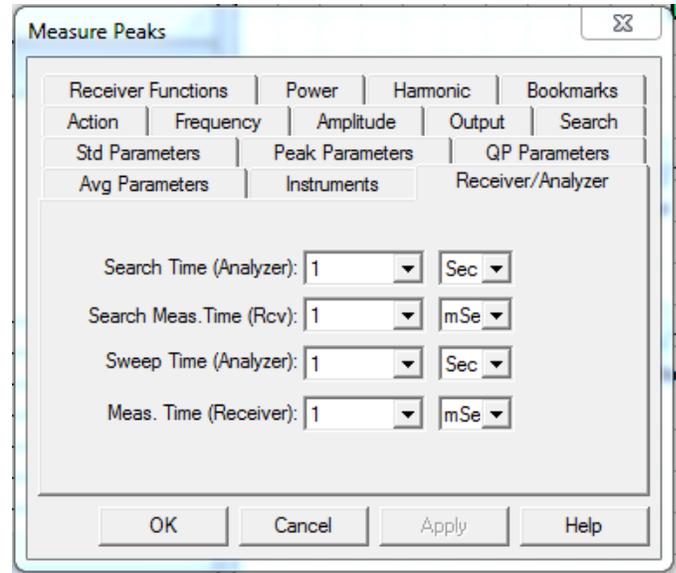
Instruments Tab

The **Instruments** tab identifies which instruments will be addressed by this action. At a minimum you must specify the analyzer. Click the arrow in each list and select the appropriate instrument.

If you are making QP measurements you will need to specify the QP adapter only if a completely separate instrument performs the QP function.

The Preselector needs to be specified if a separate instrument is re-sent.
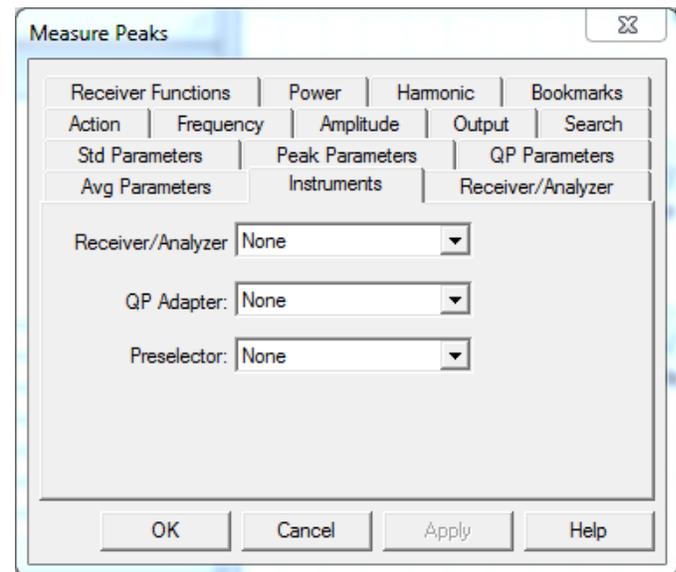
Power Tab

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 which require an input signal during a measurement.

- **Signal Active**—Allows you to track the current frequency with an output from the signal generator. This allows for attenuation measurements, cable losses, shielding effectiveness, and other tests where you need a signal induced into the test setup. Select this check box to make it active.

- **Signal Generator**—Select the signal generator from the instrument list.

- **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, 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.
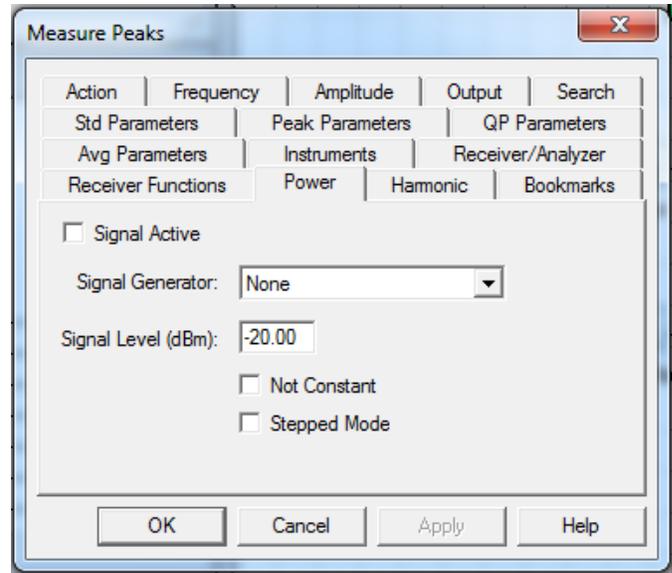
Harmonic Tab

The **Harmonic** tab is used when you need to do harmonic measurements on specific frequencies and store their information in separate data elements.

- **Harmonic Active**—Select on this check box if you want to perform harmonic tests on each frequency in the input data. 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 **Available** column includes all data elements defined on the Data window. 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. To move items back and forth between these two lists, double-click the desired data element.
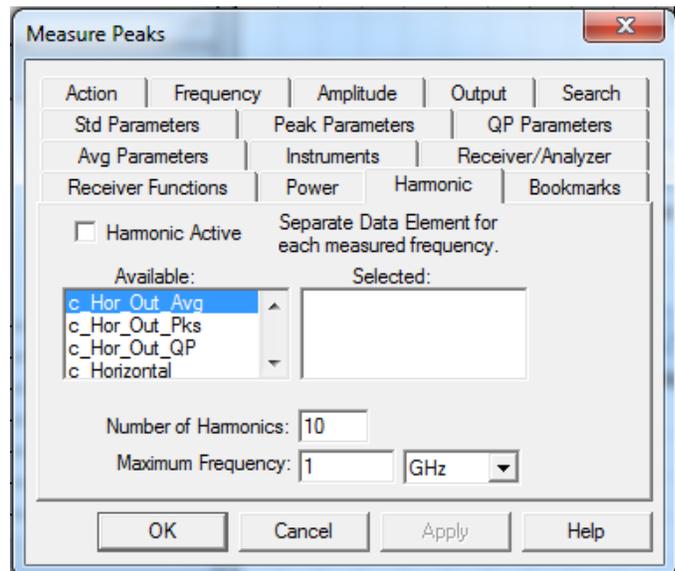
- **Number of Harmonics**—Specifies the number of harmonics to be tested. A value of 1 would indicate a measurement of the fundamental only. A value of 10 would measure the fundamental and the nine harmonics of the fundamental.

- **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.

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.

Signal Discrimination Measurement icon

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, -QP, -Avg**—Click the arrow and select the appropriate input data element. You must have either a peak or QP data element or the action will not start.
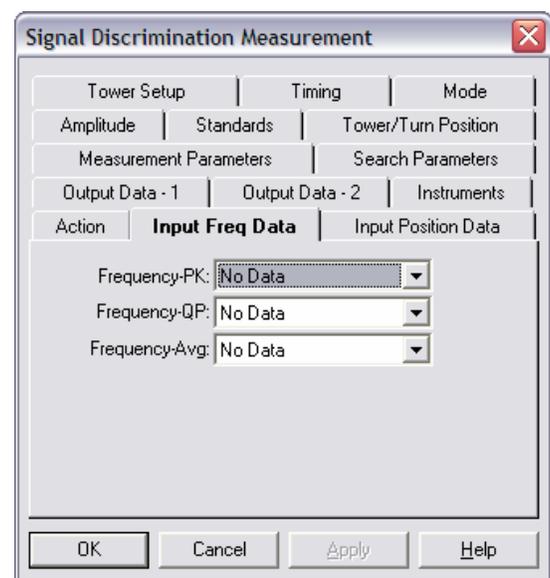
- **Tower Data**—Click the arrow and select the appropriate input data element. This data element will represent the position that the tower was in during the measurement that yielded the peak or quasi-peak. If the **Automatically Set Twr/Turn on Frequency Change** check box on the **Tower/Turn Position** tab is selected, the information contained in the specified data element will set the tower 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.
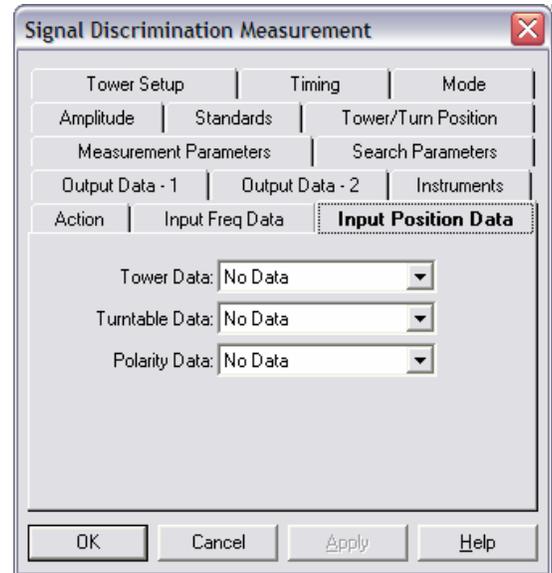
- **Turntable Data**—Click the arrow and select the appropriate Input data element. This data element will represent the position that the turntable was in during the measurement that yielded the peak or quasi-peak. If the **Automatically Set Twr/Turn on Frequency Change** check box on the **Tower/Turn Position** tab is selected, the information contained in the specified data element will set the 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**—Click the arrow and choose an appropriate data element for storing the peak 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.
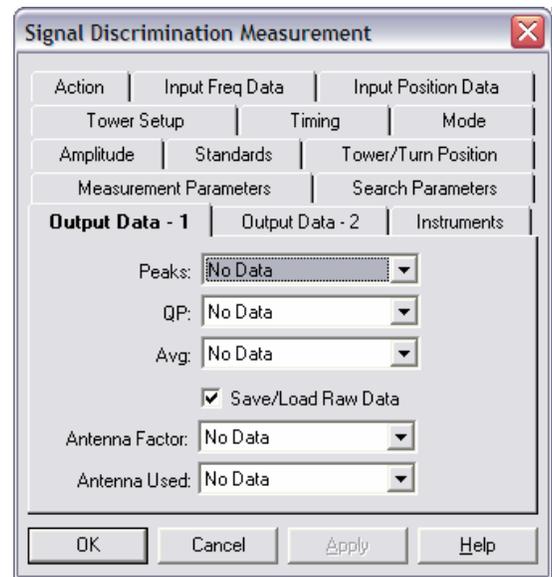
- **QP**—Click the arrow and choose an appropriate data element for storing the quasi-peak values measured during this step. If this is the same data element as the input data, this data will overwrite existing data. If this is a unique data element, it will store the output values. Generally, it is recommended that you store the output into a unique name. This allows you to compare the input data to the output data for analysis.

- **Avg**—Click the arrow and choose an appropriate data element for storing the average 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**—Click the arrow and 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. In particular, on an OATS you might take a reading with a biconical. 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**—Click the arrow and 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. In particular, on an OATS you might take a reading with a biconical. 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**—Click the arrow and choose an appropriate data element for storing the tower height for each measurement.

- **Turntable**—Click the arrow and choose an appropriate data element for storing the turntable angle for each measurement.

- **Hor/Vert**—Click the arrow and choose an appropriate data element for storing the tower polarity for each measurement. This is a word-type data element which will hold an **H** or **V**, depending upon the polarity measured during the test.

- **Comments**—Click the arrow and choose an appropriate data element for storing comments entered for each measurement. This is a word-type data element and only word-type data elements will be displayed in this list.

- **Date/Time**—Click the arrow and choose an appropriate data element for storing the date and time information. This is a word-type data element and only word-type data elements will be displayed in this list.

Instruments Tab

The **Instruments** tab specifies the instruments to be used during the execution of this action. You must have a receiver/analyzer selected. All others are optional.

- **Receiver/Analyzer**—Click the arrow and select the appropriate instrument. You must have an instrument identified at this position for the action to operate.

- **QP Adapter**—Click the arrow and select the appropriate instrument only if you have a separate quasi-peak adapter for the spectrum analyzer. By choosing an instrument here, you change the method of setting RF and video bandwidth. If a QP adapter is present, the RF and video bandwidths chosen on the **Parameters** tab will be set for the QP adapter. A different set of RF and video bandwidths will be set on the spectrum analyzer. The settings for the analyzer are determined by the manufacturer and set by the TILE! program.

- **Preselector**—Click the arrow and select the appropriate instrument only if you have a separate preselector for the spectrum analyzer. By choosing an instrument here, you change the method of setting attenuation. If a preselector is present, you must consider setting its attenuation using a GPIB direct command. It will not be automatically set by TILE!

- **Tower**—Click the arrow and select the instrument. If you are using a manual tower, you must set up an instrument in the Instrument window for the tower. You can either use the Do Not Use Driver feature or the DEMO.INS demonstration instrument driver; see *Defining an Instrument* on page 40.

- **Turntable**—Click the arrow and select the instrument. If you are using a manual turntable, you must set up an instrument in the Instrument window for the tower. You can either use the Do Not Use Driver feature or the DEMO.INS demonstration instrument driver; see *Defining an Instrument* on page 40.

- **Manual**—If the tower or turntable does not support GPIB operation, select the check box to allow operator interruption during the test.
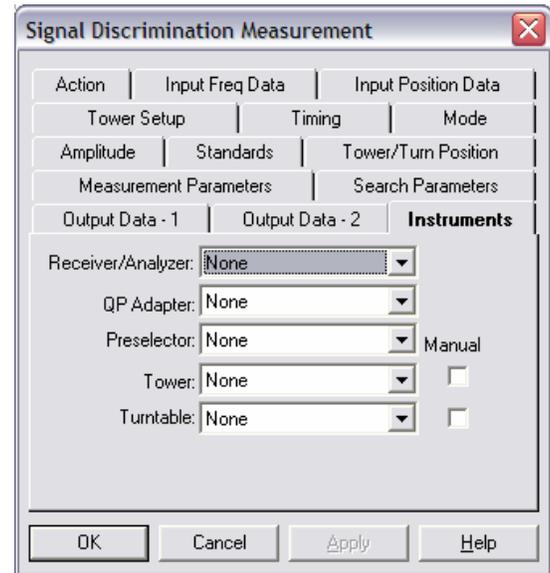
Parameters Tab

This is the same interface found in all other emissions actions so it will not be described here.

The **Search Parameters** tab controls the settings of the spectrum analyzer/receiver during the measurement process.

- **RF Bandwidth**—Click the arrow to display the available RF bandwidth settings. Not all instruments will be able to use each of these settings, but the instrument driver will select the closest setting that is valid for the specified receiver. For instance, some older receivers will only do 9 kHz and 120 kHz RF bandwidths. For all bandwidths at or above these settings, the driver will automatically select the appropriate setting.
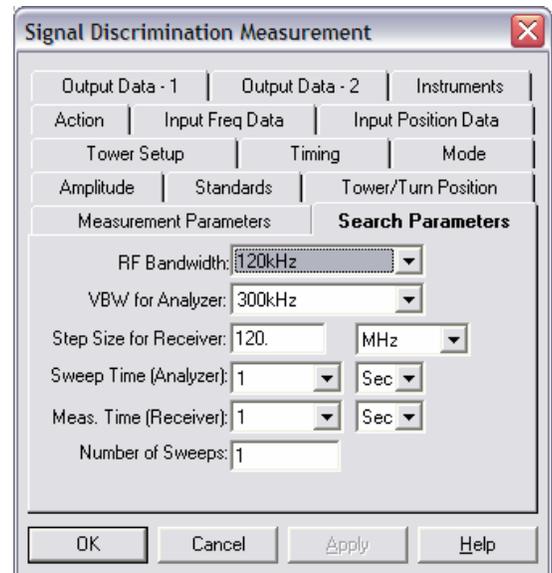
- **VBW for Analyzer**—Click the arrow to display the available video bandwidth (VBW) settings. Not all instruments will be able to use each of these settings, but the instrument driver will pick the closest setting that is valid for the specified receiver.

- **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.

- **Sweep Time (Analyzer)**—Specifies how long the receiver/analyzer will take to sweep across the frequency band. This setting will determine the dwell time at each frequency point. Set these to the required values and click the arrow to select the units.

- **Meas. Time (Receiver)**—True receivers take individual measurements and then group this information into a sweep result. The measurement time (MT) is the time increment per frequency step.

- **Number of Sweeps**—A setting greater than 1 will put the analyzers in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

Amplitude Tab

The **Amplitude** tab is used to set the default reference level, attenuation values, and span value.

- **Reference Level**—Enter a value and click the arrow to select the appropriate units. This value will set the reference level of the receiver/analyzer at each frequency step.

- **Attenuation**—Click the arrow and select an appropriate attenuation value. This will be set on the receiver/analyzer at each frequency step.

- **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 one-half the span. The stop frequency will be set to the current frequency plus one-half the span.
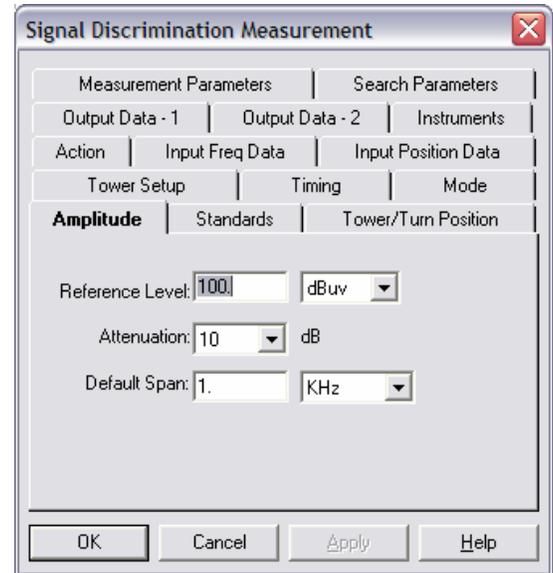
Standards Tab

These data elements contain the specification limit information as well as antenna correction information appropriate for each frequency being tested.

- **Limit**—Click the arrow and 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. Click the arrow and 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**—Select an appropriate data element for the cable loss. This value will be added to the actual instrument reading for calculating the margin. If **No Data** is selected a default value of 0 dB will be used by the software.

- **Preamplifier Gain (add)**—Select an appropriate data element for the preamplifier gain. This value will be added to the actual instrument reading for calculating the margin. If **No Data** is selected a default value of 0 dB will be used by the software.
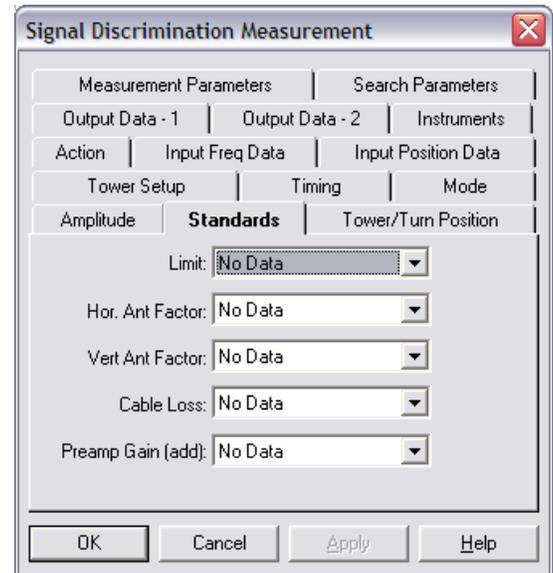
Tower/Turn Position Tab

The **Tower/Turn** tab determines the range of motion for the positioners. For the tower all references are in centimeters. For the turntable 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 3-10 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 a number or 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- and test-specific because of the differences between positioners and the equipment placed upon them.

- **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

- **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.
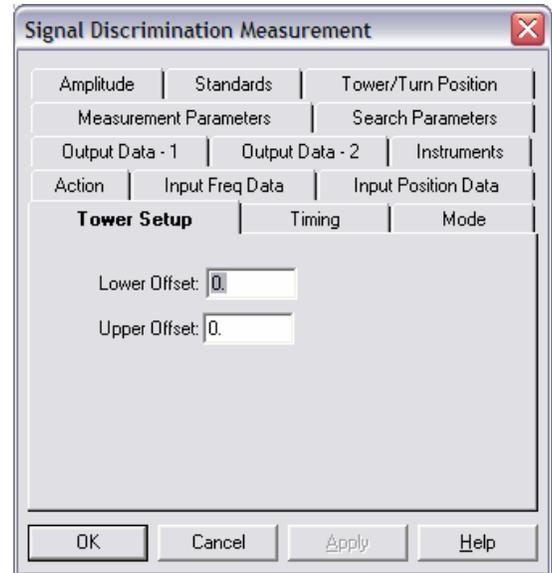
Timing Tab

Certain instruments have timing issues that can be resolved by changing the default values on the **Timing** tab.

- **Tower Start/Stop Timing**—Between the start command to the tower and the first read, the system will pause this time to allow the tower to start its motion.

- **Turn Start/Stop Timing**—Between the start command to the turntable and the first read, the system will pause this time period to allow the turntable to start its motion. This is a major concern on very large turntable where the start momentum is large.
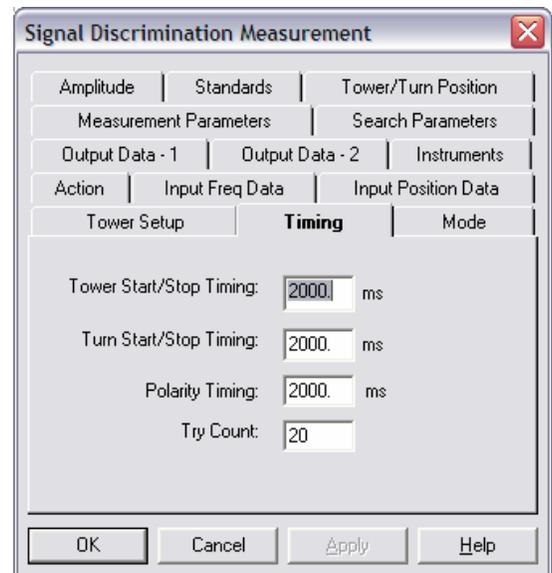
- **Polarity Timing**—Most towers with polarity under remote control use an air control to change polarity. The amount of time it takes for the antenna to rotate from horizontal to vertical, or vice versa, is dependent upon the weight of the antenna and the air pressure. We want to ensure that the polarity 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**—When either the tower or turntable is in motion (after the start command), the software determines when to send a stop command by reading the positioner and comparing the current position to the desired position. In the event that the current reading and the previous reading are the same (but you have not reached the desired position), the software will re-read the instrument the number of times specified in the try count before displaying a dialog box.

Mode Tab

When at a frequency, the operator can at any time trigger a measurement of the current frequency. 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, QP, 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.
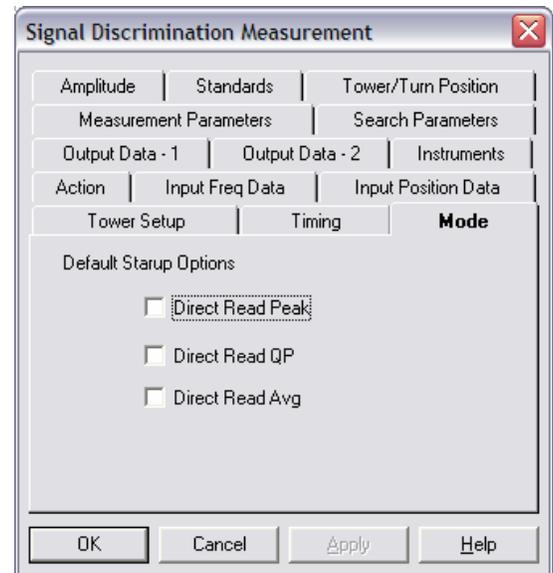
Signal Discrimination Dialog Box

The Signal Discrimination Measurement action is basically 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.

Table of Frequencies Dialog Box

The table of frequencies provides a view of the known signals. This table can be added to or subtracted from by the operator through the use of the **Set Before**, **Set After**, and **Delete** buttons. To control which items are viewed or their sort order, go to the **Set Before**, **Set After**, and **Delete** buttons. The table of frequencies is only updated with the set command buttons.
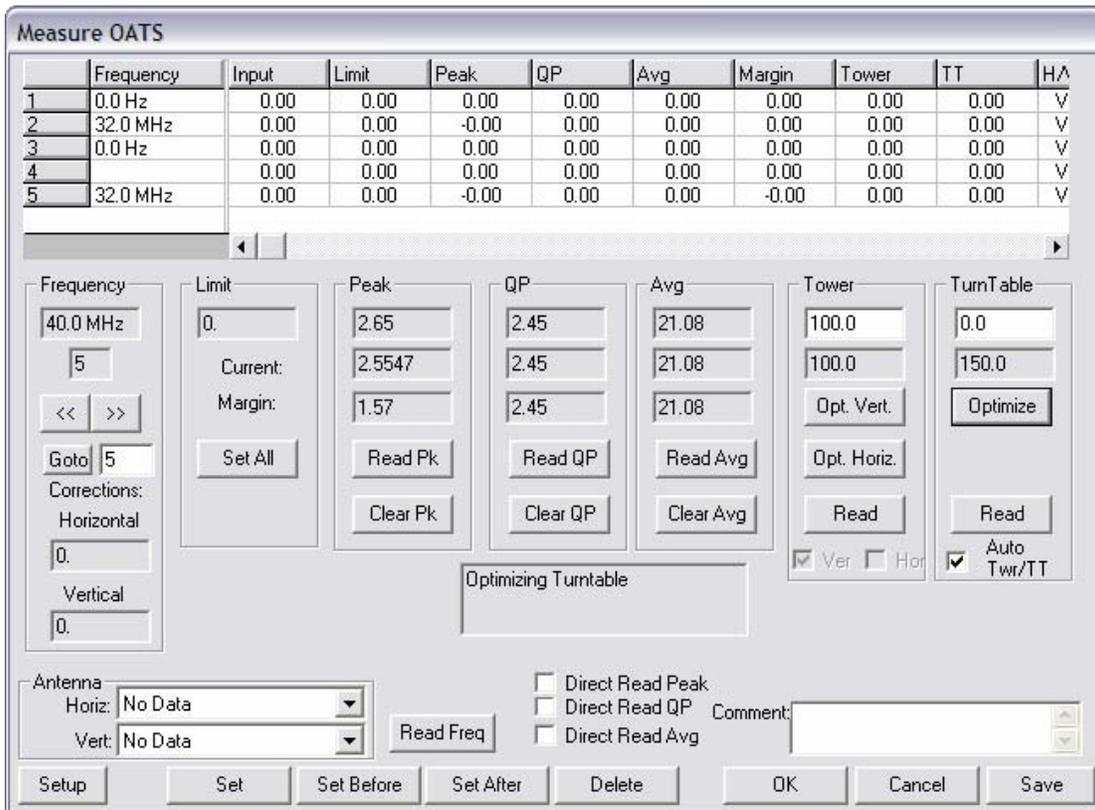
- **Peak, QP, Avg sections**—each have a **Read** and **Clear** button. When the operator has a signal to measure, manually move the marker to the frequency of interest. At this point, you may read the current marker as it sits, or perform a calibrated reading using the default standards. If you click **Read Pk**, 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.

- **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.
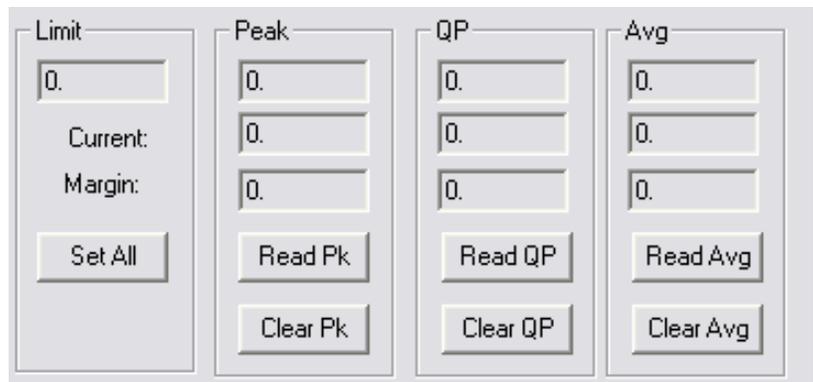
- **Peak/QP/Avg——**Contains the maximum reading at this frequency, the last reading from the instrument, and the margin between the maximum reading and the specification limit at this frequency. If you read a number of times, the first/top box will always reflect the highest reading found at any point. 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.

- **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.
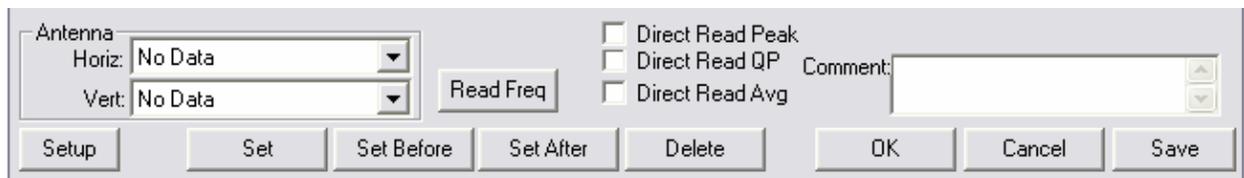
- **Read Freq—**Click this button to read the current frequency and update that information without changing the values for peak, QP, or avg. This is often used when you want to manual fine-tune frequencies of interest prior to executing an automatic measurement using the Scan Peaks action.

- **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 Box—**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. These items can be changed or updated at any time by the operator.

- **Set, 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. If you are physically reading the same frequency as the current point in the table, do not use the **Set** button. 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**.

- **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. 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.

- **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.

Site Attenuation icon

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

The **Frequency** 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 either enter start, stop, and step frequencies, or you can enter a data element with a set of specific frequencies of interest.

- **Start Frequency**—Enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—Enter an appropriate number in the box and click the arrow to select the 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, the longer the test will take to perform. Enter an appropriate number in the box and click the arrow to select the units.

- **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. But the frequency accuracy between analyzers and receivers can impact this test. Enter an appropriate number in the box and click the arrow to select the units.

Levels Tab

The **Levels** 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 (C_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

- **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 receive a warning message; instead, the instrument driver will choose the closest bandwidth that is supported. When you have specified a QP adapter on the **Instruments** tab, the RF bandwidth will be set for the QP adapter.

- **VBW**—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**—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.

- **Number of Sweeps**—A setting greater than   will put the analyzer in Max Hold and the specified number of sweeps will be made. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

- **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 ST divided by the number of points per sweep (set by manufacturer). Enter an appropriate number in the box and click the arrow to select the units.

- **Meas. Time (Receiver)**—True receivers take individual measurements and then group this information into a sweep result. The measurement time (MT) is the time increment per frequency step.

Data Tab

The **Data** tab identifies the data elements that will be used to store the information generated by this action.

- **(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

The **Antenna** 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). Enter an appropriate value in the box.

- **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. Enter an appropriate value in the box.

- **Step**—The step value determines how many centimeters will separate each height reading on the tower. Enter the number of centimeters desired in the box.

- **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.

- **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.

- **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.

- **Polarization**—This action can be operated with the antenna in either the horizontal or vertical orientation. Check the appropriate box. 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

The **Instruments** tab identifies the instruments to be used during this action. Click the arrow and select an instrument for signal generator, spectrum analyzer, and tower. If no tower is available, or it is not GPIB-controlled, select the **Manual Tower** check box to configure an operator interrupt. This will allow you to manually set the tower position during the test. If you have a separate QP adapter and preselector, select these instruments also.
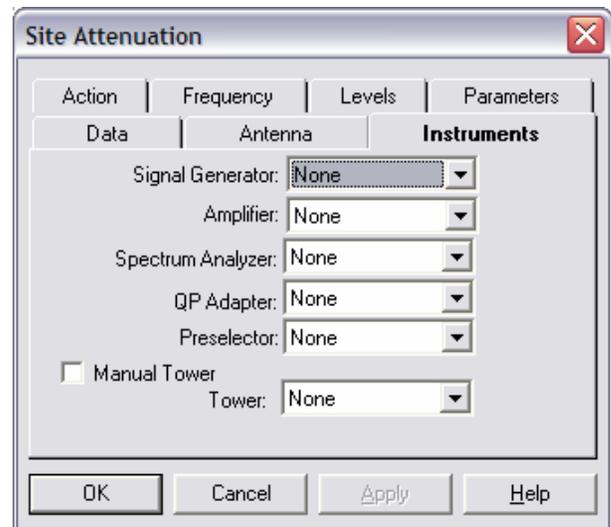
Site Attenuation Dialog Box

The **Site Attenuation** dialog box provides visual information to the user as well as the ability to halt execution of the test.

- **Target**—Provides information on the desired (target) 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.
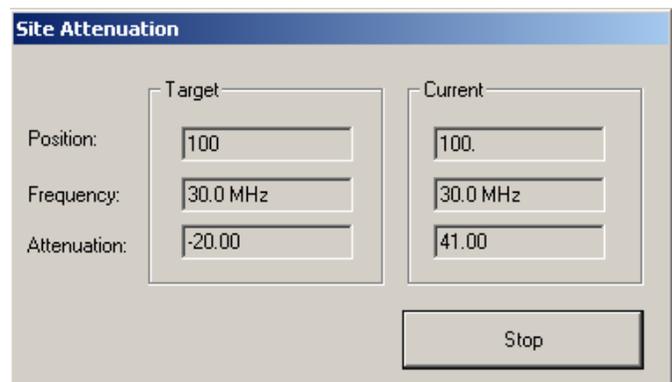
Occupied Bandwidth Mask and FCC Power Spec Limit

The Occupied Bandwidth 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)).

These actions are not documented but following are the available tabs.

Occupied Bandwidth Mask and FCC Power Spec Limit icons

Occupied Bandwidth Mask Tabs

Output Data/Absolutes Tab

Frequency Settings Tab

FCC Power Spec Limit Tabs

Data Tab
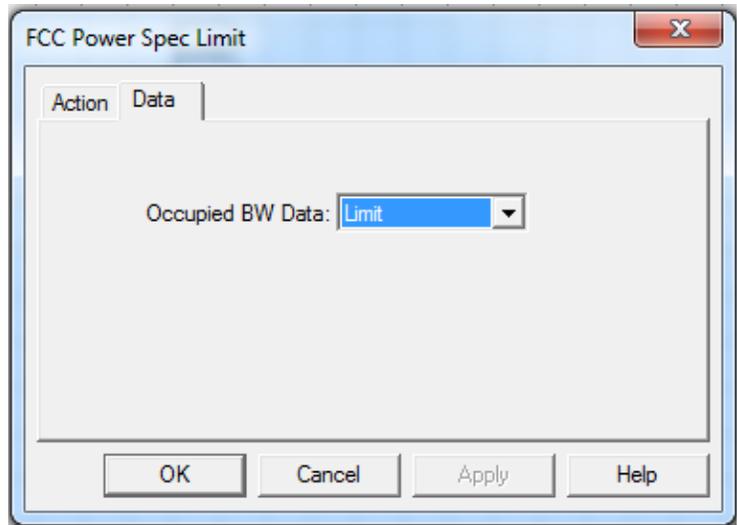
GTEM Actions

These actions will calculate the GTEM correlations and position the EUT.
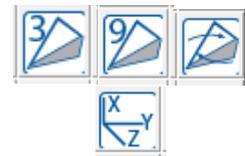
GTEM Actions

GTEM/OATS 3 Position Correlation

GTEM/OATS 3 Position Correlation icon

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 an 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. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Y:—**Describes the physical height from the floor of the GTEM to the center of the EUT. Enter an appropriate number in the box and click the arrow to select the units. If you are not sure, 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. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

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—**Enter the minimum antenna height for the scan range on the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Height Max—**Enter the maximum antenna height for the scan range on the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Separation—**Enter the separation distance for the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **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 to only combine the three vectors into a free space equivalent value.

Inputs Tab

The input fields on the **Inputs** tab 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**—Click the arrow and selected the appropriate data element. This element will be the same as was used during the X-orientation measurement on the flowchart.

- **Vy**—Click the arrow and selected the appropriate data element. This element will be the same as was used during the Y-orientation measurement on the flowchart.

- **Vz**—Click the arrow and selected the appropriate data element. This element will be the same as was used during the 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.
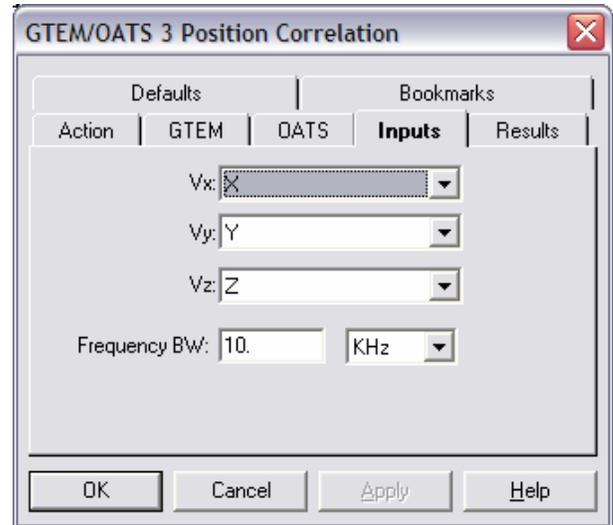
Results Tab

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**—To keep the maximum signal in the vertical orientation, select an appropriate data element.

- **Horizontal**—To keep the maximum signal in the horizontal orientation, select each box for which results are required. Click the arrow for each element and select an appropriate data element.

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/OATS
9-Position Correlation icon

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 an 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. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Y:**—Describes the physical height from the floor of the GTEM to the center of the EUT. Enter an appropriate number in the box and click the arrow to select the units. If you are not sure, 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. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

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**—Enter the minimum antenna height for the scan range on the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Height Max**—Enter the maximum antenna height for the scan range on the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Separation**—Enter the separation distance for the OATS. This information is specified by the emissions standards. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **Height Step**—When performing a height scan on an OATS, the number of steps in height can directly affect the accuracy of the test results. This field is included to allow a closer comparison between the GTEM-based data and the OATS readings. Enter the step size for the height scan and the mathematical model will calculate the expected maximums based upon these stated steps. Click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

- **EUT Height**—The EUT height is a factor in the calculations. Typical settings are between .8 meters and 2 meters depending upon the type of EUT and the actual measurement height for the EUT when placed on an OATS. Enter the appropriate number and click the arrow for the units or use the **Tab** key to move to this field, and then select the units. You can also enter the first character of the units for quick selection.

Alignment Tab

When orienting an 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. The **Alignment** tab allows you to specify the proper home position for the EUT.
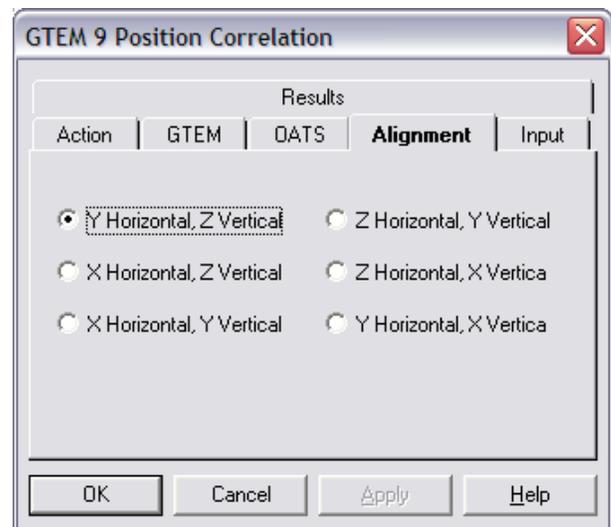
The z-axis is the axis along the length of the septum; the y-axis is the septum to floor; and the x-axis is the width of the cell. The EUT +Z face is oriented toward the input of the GTEM. Viewed from the input, the +X face is on the left side of the EUT and the +Y face is across the top. The +Y horizontal, +Z vertical (i.e., facing the input of the GTEM) is the most common starting position for an EUT in the GTEM. This would duplicate the physical orientation of the EUT facing the antenna on the OATS.

Performing a 9-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.

The separate readings are defined as:

- **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 position +45 degrees in the x-axis (counterclockwise movement of the EUT).

- **Position 6**—+Y position –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 position –45 degrees in the x-axis (counterclockwise movement of the EUT). It is important to 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).

Positions 1, 4, and 7 are the same as those used in the GTEM 3 position correlation analysis.

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. 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.

Click each box for which results are required. Click the arrow
for each element and select an appropriate data element.
These data elements need to be defined as measurement
elements in the data definitions. See *Measurement Elements*
on page 30 for more information.
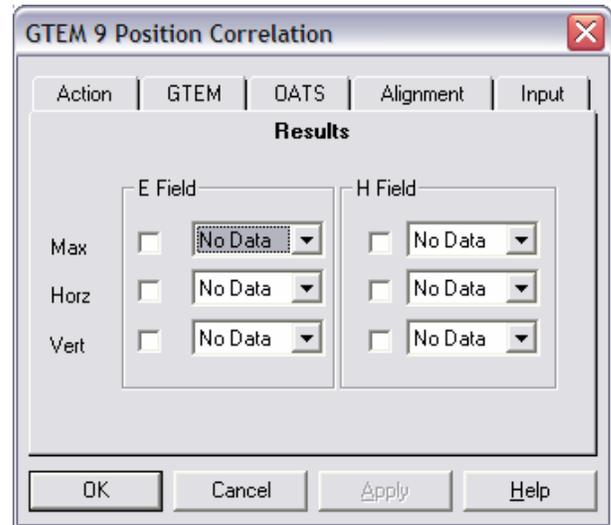
Correlation Dialog Box

During execution of the GTEM correlation, the dialog box provides the user a visual
indication as to the current status of the execution as well as a **Stop** button to
cancel execution of this action. The two display boxes are the current step and total
number of steps. The current step updates as each calculation is performed and will
change during the execution.
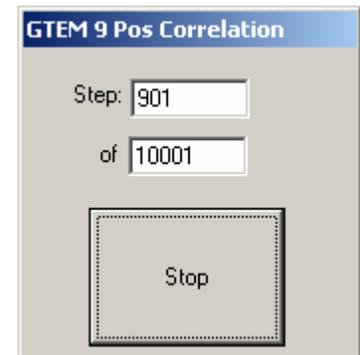
GTEM Optimization

This action is used to perform a Max Hold of a 3 position manipulator in a GTEM
using a spectrum analyzer or swept receiver. The tabs are similar to other emissions
actions except that it adds the positioner instrument and an output for its maximized
positions.

GTEM Optimization icon

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 EUT icon

Position EUT Position Tab

The **Position** 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, is defined as an X–Y positioner on the **Links** tab; the 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, is defined as an X–Y positioner on the **Links** tab; the 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**—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.
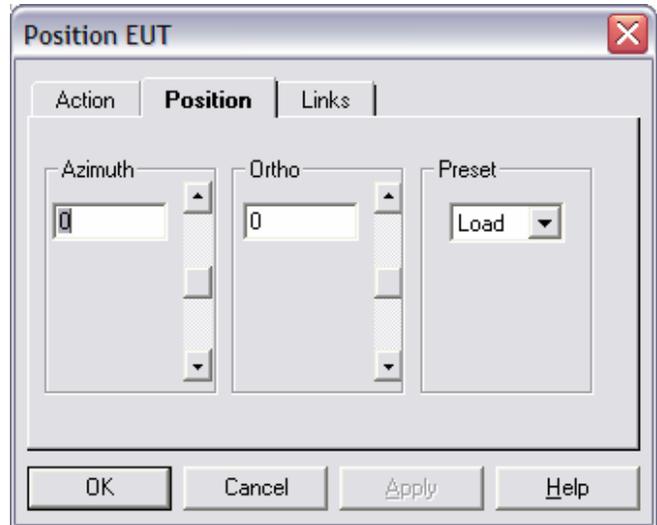
Links Tab

The **Links** tab is used to select the instrument that controls the manipulator.

- **Instrument**—Select from the list of the instruments by clicking the arrow.

- **Margin**—The margin is a defined increment which 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 period of 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.

Position EUT Dialog Box

The **Position EUT** dialog box 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.
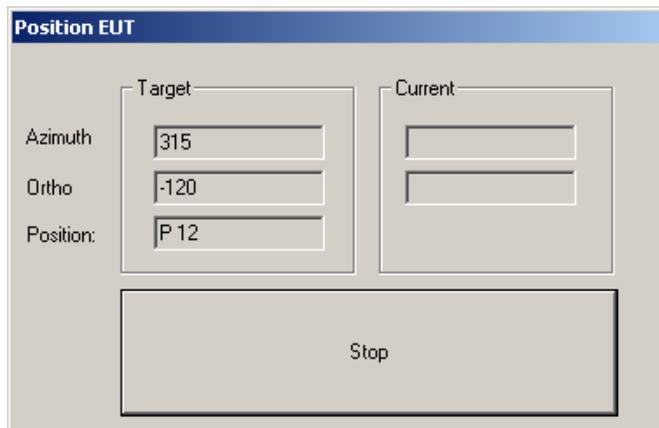
10.0 Immunity / Susceptibility

Immunity / Susceptibility Actions (Radiated / Conducted)

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 an EUT test procedure.

Immunity/Susceptibility actions

Immunity / Susceptibility Calibration

Frequency Tab

The **Frequency** 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 from a data set. When this check box is selected, the data set must be defined using the standard TILE! format of *frequency, value*. This is explained further in *File Elements* on page 30.

  The Set From Data check box is particularly useful when performing IEC testing (or other standards) which require a percent-stepped frequency progression. Creating this progression with standard spreadsheet software is relatively easy. Write this data to file as an ASCII text, comma-separated variable file and define a data element as file-type with this access. This data element will then contain the appropriate information.

- **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). Enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—Sets the stop frequency for this test. This does not have to be higher than the start frequency. Enter an appropriate number in the box and click the arrow to select the units.

- **# 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%. There are some calibration schemes that use 2% or 5%. Enter the value desired and click the check box.

Freq Steps Tab

The **Freq Steps** tab lets the user control a number of different functions/methods/algorithms during the Immunity test.

- **Use Min Atten between Frequencies**—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, deselecting the check box will prevent the minimum attenuation command from being sent to the signal generator.

- **No Sig Gen On/Off between Frequencies**—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, deselecting this check box will prevent the on/off signal 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; see Immunity Test *Leveling Tab* on page 2. You can specify a fixed set of beginning amplitudes (less 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.

- **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.

- **First Step Tries**—The number of tries to check during the leveling for the first step. 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.

- **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, in Bulk Current Injection (BCI): When at the first frequency, the relative efficiency of the injection probe, and therefore the generated field level, are unknown. By using an offset, we can force the first level to start very low ensuring that the EUT is not subjected to an unusually strong field on the first step. The Height portion of the following First Offset equation is from the TEM tab; the default value is 1.

$$SigOut = 10 * log10 \left(\frac{(Target*Height)^2}{50*10^{\left(\frac{Gain-Offset}{10}\right)}} \right) + 30$$

Amplitude Tab

The **Amplitude** tab is used to select the amplitude data.

- **Data**—Allows the operator to select the desired amplitude data to be used in the test. Click the arrow to view the defined data elements and select the appropriate element. 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.

- **Data Units**—Click the appropriate 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

The **Leveling** 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. A choice of no leveling makes the remaining choices inactive. The **Pwr Meter** and **Probe** choices require careful consideration of the tolerance, delay, and maximum count.

- **Tolerance (% or dB)**—The Plus and Minus boxes allow the operator to select the desired tolerances for the leveling process. For instance, 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 that jumps below and above these numbers without ever landing within the zone. You will exceed the maximum 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. 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 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.

- **Max Step**—Sets the maximum amplitude increment that the signal generator will perform.

- **Min Step**—Sets the minimum amplitude increment that the signal generator will perform.

- **Max Count**—Sets the maximum number of tries 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**.

- **Continue on Fail-to-Level**—Allows the user to continue on 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

The **TEM** tab is used to configure parameters when using TEM or GTEM cells.

- **In Use**—Select the check box if you are using TEM or GTEM cells.

- **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.

- **Impedance**—Allows the operator to select a data set as a defined .csv data element.

- **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.

- **P-MID Test (SAE J1113)**—Select the P-MID Test check box to perform this type of test.

- **Power Out Instr**—The power out instrument is the instrument used to measure the output power from the TEM cell. Within this specification, generally speaking, 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

The **Calibration** 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

The **Results** 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 instruments 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 graphically compare 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 the **Record** check box is selected, a data element must be selected. If you fail to select a data element, the system will record a failure and not run. Click the arrow and select from the predefined data elements.

- **Signal Generator**—Choose an appropriate data element if you want to store the amplitude of the signal generator when the leveling loop has completed. This information is useful for duplicating tests or verifying results.

- **Power Meter1**—Choose an appropriate data element if you want to store the amplitude of the forward power during this calibration. Power Meter1 always refers to the forward power, if net power is being measured, or the value read from Power Meter1. The forward 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.

- **Power Meter2**—Choose an appropriate data element if you want to store the amplitude of the reverse power during this calibration. Power Meter2 always refers to the reverse power if net power is being measured. The secondary power level (or reverse power, if in a net power situation) value is determined by reading the identified on the **Instruments** tab and adding the calibration value found on the **Calibration** tab. The corrected value is saved.

- **Net Power**—Choose an appropriate data element if you want to store the net power measured during this calibration. You must have both a forward power instrument and a reverse power instrument identified on the **Instruments** tab and you must have selected the **Net Power** check box on the same 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)

- **Field Probe**—Choose an appropriate data element if you want to store the amplitude of the probe during this calibration.

Instruments Tab

The **Instruments** tab is used to select the specific instruments that will be used in the immunity calibration. Click the arrow for each instrument in use and select the appropriate instrument. If an instrument is selected (other than the signal generator), it must have a calibration entered on the **Calibration** tab. If the instrument is not in use, make sure this field is set to None.

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.
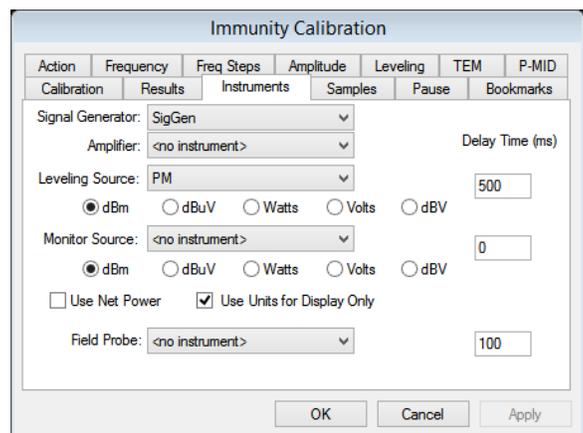
- **Delay Time**—Allows the user to enter a delay time (in milliseconds) before a reading takes place. This is particularly useful due to the long settling time inherent with power meters and probes.

- **Use Net Power**—Requires the use of two instruments to record the net reading between them. The result is recorded in the data element identified on the **Results** tab.

- **Use Units for Display Only Checkbox**—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 Voltsrms 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(V^2/(50*0.001)

Where 50 is impedance and 0.001 is the relative to 1mW (for dBm).

So, 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.

Samples Tab

Generally speaking, 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. The **Samples** tab allows you to select the number of samples for each of the different instruments in this step.

- **Samples Per Reading**—Enter an appropriate value for each of the instruments.

- **Probe Method**—Defines how the value is calculated for readings from each instrument.

- **Pwr Meter Method**—Defines how the value is calculated for readings from each instrument.

Pause Tab

The **Pause** tab allows the user to specify frequencies that will pause the test and prompt the user to take some action.

- **Pause Frequencies**—Select an appropriate data element. The data element should have a list (or single) frequency that is the transition point. When the next calibration frequency is equal to or greater than the frequency in the data element, the program will pause and the pause message displayed until the operator clicks the **OK** button.

- **Pause message**—Enter a message that will be displayed during the pause. When entering multiple lines, use the CTRL+ENTER.

- **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.

- **Power ON Delay**—This is a period of 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.

- **Gain Offset at changeover**—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.

Immunity Calibration Dialog Box

You can execute the Immunity Calibration action either by clicking the action and selecting **Execute**, by executing it as part of the sequence of tests being executed, or by using the run/single options from the Windows menu bar. 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 %** 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.
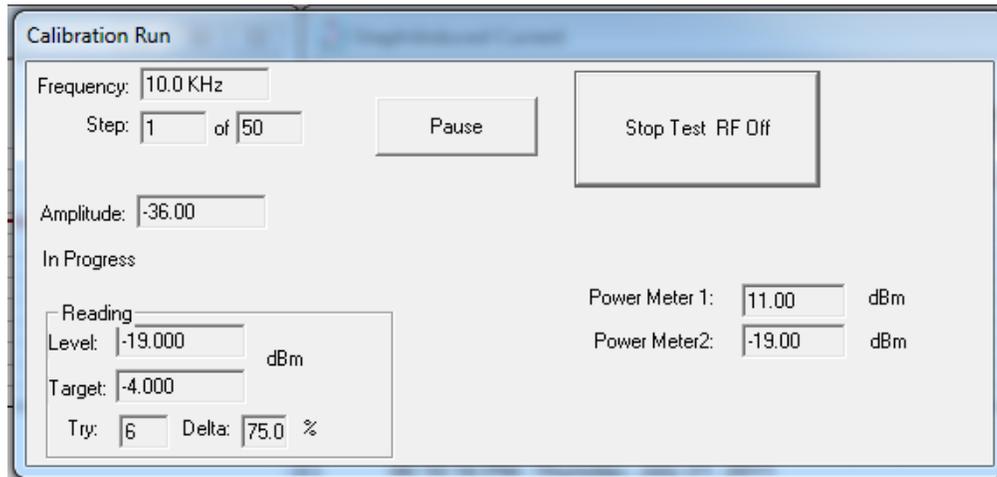
Immunity / Susceptibility Test

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.

Immunity Test icon

Frequency Tab

The **Frequency** 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). Enter an appropriate number in the box and click the arrow to select the units.

- **Stop Frequency**—Sets the stop frequency for this test. This does not have to be higher than the start frequency. Enter an appropriate number in the box and click the arrow to select the units.

- **# 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%. There are some calibration schemes that use 2% or 5%. Enter the value desired and click the check box.

- **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

The **Freq Step 1** tab lets the user control a number of 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; see Immunity Test *Leveling Tab* on page 2. You can specify a fixed set of beginning amplitudes (less 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**—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.

Immunity / Susceptibility | **199**
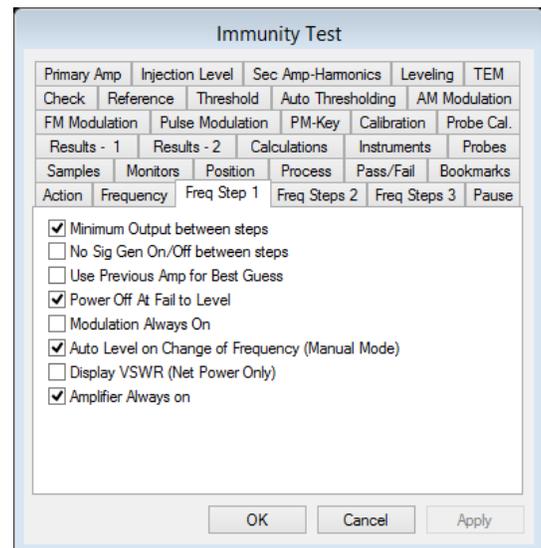
- **Display VSWR (Net Power Only)**—When both forward and reverse power is measured the software can calculate the VSWR of the transmitted signal. Selecting this check box will display this value on the dialog box while it is running. This is very convenient for determining the influence that the EUT 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

The **Freq Steps 2** tab lets the user control a number of 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 in no leveling mode with a secondary leveling loop (**Secondary Amp** 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 will not progress to the next frequency.
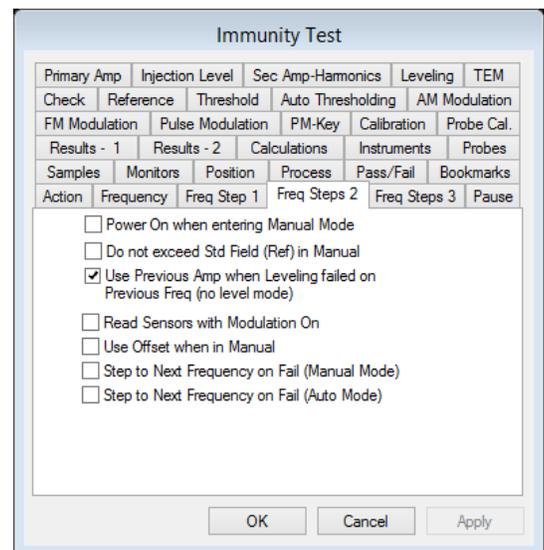
- **Step to Next Frequency on Fail (Auto Mode)**—When the Fail button is clicked, in auto mode the normal behavior is to go to manual mode and pause the test. If you click **Auto Run** at this point, it will retest the current frequency. If this check box is selected, the software will automatically step to the next frequency when **Fail** is clicked. This allows you to easily 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 than the values set on the **Instrument** tab.

- **First Offset**—This value is used as a safety offset prior to turning on the signal generator. Unless a Best Guess data element is used, this value will take the gain of the amplifier and the target value, and then calculate the drive level required. The offset is a value that is added to this drive level prior to turning on the signal generator output. The Height portion of the following First Offset equation is from the TEM tab; the default value is 1.

$$ SigOut = 10 * log10 \left(\frac{(Target*Height)^2}{50*10^{\left(\frac{Gain-Offset}{10} \right)}} \right) + 30 $$

- **First Step Tries**—The number of tries which will be attempted for the first step before an error is declared.

- **Best Guess Amp Data Element**—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 the offset).

- **Continue on Fail-to-Level**—Allows the user to continue on 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 the **Pause** 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. See *Direct Entry* on page 2 for more information. 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 on. 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—**Controls a leveling delay for the pause frequency step that can be different than the values set on the **Instrument** 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 signal generator drive level.

Primary Amp Tab

The **Primary Amp** 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), 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 actually 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).

- **Data**—The **Data** list 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 is 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.

- **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.
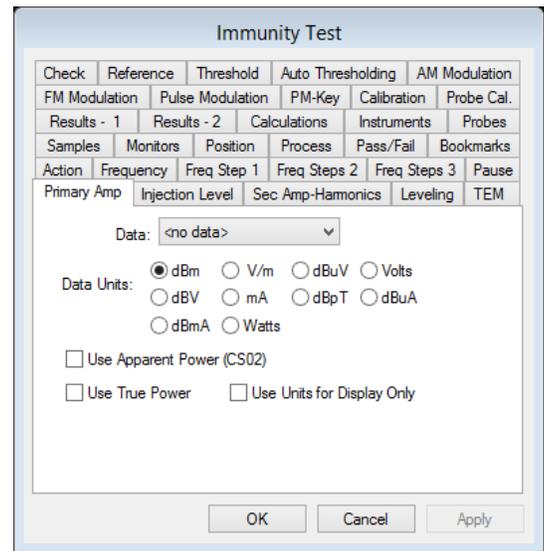
Injection Level Tab

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.

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, you can limit harmonic test above 100 MHz (assuming 10 harmonics) to not exceed analyzer capability.

The **Leveling** 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. 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 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 then 0 dBm into the input of the amplifier. A value of 0 will limit the software by halting the leveling if the anticipated signal generator level will exceed 0 dBm.

- **From Table**—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.

- **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 (either by 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 **Secondary Amp** tab or **Check** tab), this value also determines the step size for these loops.

- **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.

- **Min Step**—Sets the minimum amplitude increment that the signal generator will perform.

- **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. A warning of this problem would be a failure to level which levels quickly if you click **Retry**. This field is also used in the secondary amp and check loops to determine the number of tries.

Immunity Test 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 use power meter leveling mode (on the **Leveling** tab) to level to a value in V/M, not power.

- **Height**—Enter the septum height in meters. This value is used in the previous equation to adjust the field strength to an appropriate value.

- **Impedance**—Choose a data element which contains the impedance of the cell. 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).

- **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.

- **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.

Check Tab

The **Check** 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**—Selects the data element which 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 **Secondary Amplitude** tab and use that method.

- **Allowed Variance**—Reflects the +/- tolerance which will be applied to the power meter reading in determine acceptance.

- **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.

- **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 a field 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 actually 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 to be 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. There are some standards which 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.

- **Auto Thresholding**—Select this check box to activate thresholding.

- **Available/Selected**—Select an action from the **Available** column to move it to the **Selected** column in the order you want the thresholds to be selected.

AM Modulation Tab

Select the **AM Modulatio**n tab to turn 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). A typical AM modulation envelope would appear as follows:

- **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 basically trying to duplicate the human voice on a radio transmitter. Place an appropriate numerical value in the **Depth** box. 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. Enter an appropriate number in the box and select the frequency component from the list.

- **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.

FM Modulation Tab

The **FM Modulation** 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. Enter an appropriate number in the box and select the frequency component from the list.

- **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

The **Pulse Modulation** tab selects and configures pulse 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) 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 is not covered in this action. It can be configured using the GPIB Control action described in GPIB Control *Instrument Commands* on page 2.

- **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. Enter an appropriate number in the box and select the frequency component from the list.

- **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. Enter an appropriate number in the box and select the frequency component from the list.

- **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

- **Key Test**—When this check box is selected, the signal generator will emulate the keying and unkeying of a transmitter at the identified cycle time for the defined number of cycles. The cycle time is defined in the **On (ms)** and **Off (ms)** boxes. The number of cycles is defined in the **Number of Cycles** box.

- **On (ms)**—Defines the number of milliseconds that the RF will be on.

- **Off (ms)**—Defines the number of milliseconds that the RF will be off.

- **Number of Cycles**—The pulse shape of on and off signals is repeated the number of cycles defined here.

- **Monitor 4 On Data/Off Data**—Data elements used to record the results of the key test. They must be defined as measurements on the data page.

Calibration Tab

The **Calibration** 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.

If the amplifier calibration is not entered, the default value is 0 (a gain of zero). This will cause serious overshoot if you actually 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 calibration data needs to include cables and couplers. The default value is 0 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 1. Never use a value of 0 since probe correction factors are multiplicative. We take the reading from the probe and multiply the probe correction factor. A value of 0 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

The **Results - 1** 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 graphically compare 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. Click the arrow and select a data element from the predefined data elements.

- **Signal Generator**—Choose an appropriate data element if you want to 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—**Choose an appropriate data element if you want to store the amplitude of the forward power during this test. Power meter1 always refers to the forward power if net power is being measured. The forward power value is determining by reading the instrument identified on the **Instruments** tab and adding the calibration value found on the **Calibration** tab. The corrected value is saved.

- **Power Meter2—**Choose an appropriate data element if you want to store the amplitude of the reverse power during this test. Power meter2 always refers to the reverse power if net power is being measured. The forward power value is determining 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—**Choose an appropriate data element if you want to store the net power measured during this test. You must have both a forward power instrument and a reverse power instrument identified on the **Instruments** tab and the **Net Power** check box on the same tab must be selected. The net power value is calculated by the following formula:

$$Pf \text{ (Watts)} = ((Pfm^{(-10)})/10)/100000$$
$$Pr \text{ (Watts)} = ((Prm^{(-10)})/10)/100000$$
$$Pnet \text{ (dbm)} = 10 * Log((Pf - Pr)*100000)$$
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 1 is entered. If the step failed a value of 0 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. Select an appropriate data element to store this value.

- **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

- **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, 2, 3, 4**—Choose an appropriate data element if you want to store the amplitude of the probe level during this test. Field probe always refers to the forward power if net power is being measured. The forward power value is determining by reading the instrument identified on the **Probe** tab and adding the calibration value found on the **Probe Calibration** tab. The corrected value is saved.

Instruments Tab

The **Instruments** tab is used to select the specific instruments that will be used in the Immunity test. Click the arrow for each instrument in use and select the appropriate instrument. If an instrument is selected (other than the signal generator), it must have a calibration entered on the **Calibration** tab. If the instrument is not in use, make sure this field is set to **None**.

These four categories of test instruments are generic to an Immunity test. Certain instruments may have functions that imitate others. For example, many spectrum analyzers can be used as power meters.

- **Signal Generator**—Select the correct instrument from the list. 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**—Within the TILE! system, Power Meter1 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. Select the units for this power meter.

- **Power Meter2**—Within the TILE! system Power Meter2 is assumed to be the reverse power measurement from a direction coupler. It does not have to be, but be cautious when using a non-standard configuration. Select the units for this power meter.

- **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 which 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. See the description in **Results - 1 Tab** on page 2. The result is recorded in the data element identified on the **Results** tab.

Probes Tab

The **Probes** 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** tab and the **Leveling** tab.

- **Probe 1, 2, 3, 4**—Click the arrow and select the correct instrument for each probe. 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 which 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.

- **Leveling Method**—A set of four option buttons that determine how the four probes interact. The default selection is **Probe 1**. With this case if leveling is being performed, then the system will level on probe 1. The remaining probes will be read, if selected, and stored, if 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+ +value4/number of probes.

- **Max**—When selected, the software will read all identified probes and level off the highest value among the probes.

- **Probe 1**—The default value in probe leveling.

Samples Tab

Generally speaking, in all leveling loops or for stored values, instruments are read and values stored one time. It might be necessary in certain circumstances to read the instrument multiple times prior to storing. The **Samples** tab allows you to define the number of readings and a method of determining the value.

- **Probe 1, 2, 3, 4/Pwr Mtr 1, 2, 3**—Defines the number of samples desired per instrument reading.

- **Probe Method/Pwr Mtr Method**—If more than one sample is specified, the **Max** or **Avg** selection determines which value of the multiple readings will be used.

Monitor (1-2), (3-4), (5-6), (7-8) Tabs

The monitor tabs provide the ability to record the results of the test on instruments which are not part of the leveling loop. The instruments and data elements are selected by clicking the arrow and choosing the appropriate instruments and data elements.

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.

There are a total of eight monitors that can be configured within a test profile. Choose an instrument and data element for each of the monitors on the monitor tabs. If the user desires to monitor more than eight instruments, use a separate Monitor action for each instrument and specify them on the **Pass/Fail** tab.

Position Tab

If a positioner is specified on the **Position** 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**—Click the arrow and select the instrument which 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 0 and goes to 359 degrees. For frequency 2, since the turntable is already at 359, it will start at 359 and move to 0.

- **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 2, 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

The **Process** tab allows you to control steps during the
Immunity test. Of particular interest is the delay feature. 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 actually running of the test.

Step Test Defined: Usually when we discuss an Immunity test we
think of a fixed RF level (with or without modulation) at a fixed
frequency. But certain standards call for measuring the effects of RF
level changes within a frequency point. The IEC 1000-4-3 sets a
pass/fail criteria of 3 V/M at most frequencies. This is the only level
the standard is concerned with. But 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 on in 1-V/M steps until we reach 10 V/M. In defining a
stepped test, we basically take our final RF level (the specification
limit) and apply a start delta and step size. These three components
will fully define our test requirements.

- **Start Delta—**If you are not performing a stepped test, this box should be set to 0. 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 a 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.

  → If the start delta equals the amplitude level, the system will create a starting point of 0, 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.

- **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 0, 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.

- **Inst Delay**—This is shown for reference purposes and is the total delays from the different instrument on the **Instrument** and **Probe** tabs.

- **Soak Off Time**—Used to support automotive testing which requires that the transmitter not to be keyed for a certain amount of time between frequency points. Enter the appropriate value in milliseconds in the box.

- **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 or not 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)". But 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 which 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; for technical support contact information, see page 2.

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.
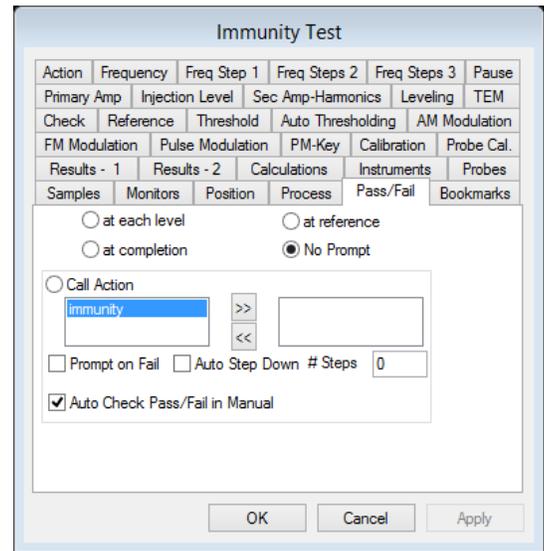
- **Call Action—**When selected, this gives the operator the ability to jump to and execute the defined actions. You can select multiple actions to test. Generally these will be switches, instrument commands, or the Serial Interface action. These icons are set up on the flowchart and configured so that their default behavior is a pass and any deviation is a fail.

- **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.

Immunity Test Dialog Box

You can execute the Immunity Test action either by clicking the action and selecting **Execute**, by executing it as part of the sequence of tests being executed, or by using the run/single options from the Windows menu bar. 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.
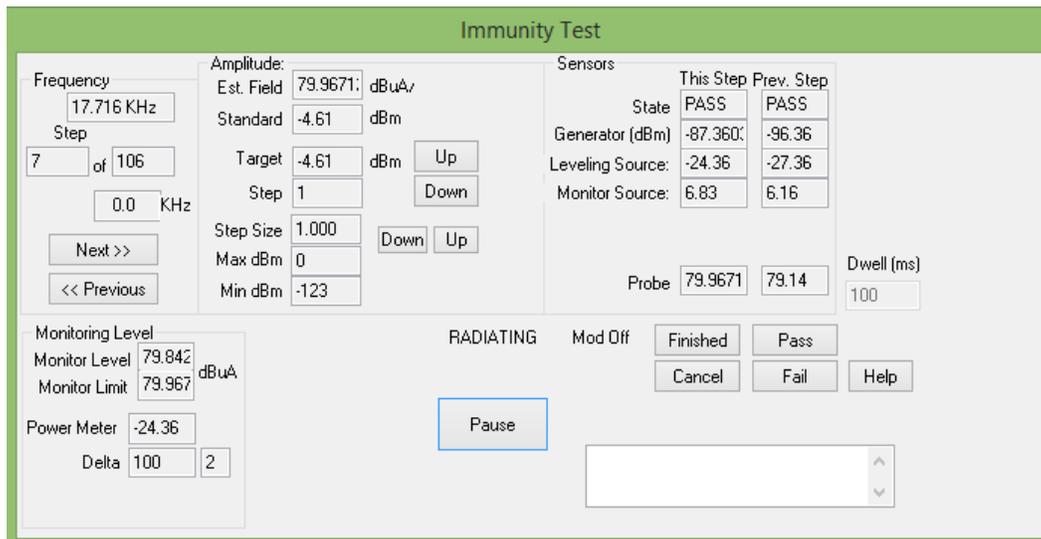
- **Frequency/Step**—Displays current frequency and step number. The step number displayed refers to the total number of frequency steps to be performed. In manual mode you can use **Next** or **Previous** to move forward or backwards in the frequency table. You cannot enter a new frequency that is not 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.

- **Leveling**—If using feedback leveling (probe or power meter), then the current 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 **Max Level** 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 of the Immunity Test action), 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.

- **Help**—This button is not active.

- **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.

- **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 **Halt** button will be present. 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 on.

  When the **Start Auto** button is clicked, it is replaced by the **Halt** button. 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 current status field.

Auto Immunity Extrapolation

The Auto Immunity Extrapolation action is used with the Immunity Test and Reverb Tests 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 Immunity Test action to reduce the signal generator level until the instrument selected indicates a reading that does not exceed the limit.

Auto Immunity Extrapolation icon

There is no documentation at this time to describe the settings, inputs, and outputs of this action.

Immunity Field Uniformity

The Immunity Field Uniformity action analyzes from 4- 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.

Immunity Field Uniformity icon

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.

Click the arrow and select the appropriate data element. 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 efficiently generate. In particular, 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 can 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, in effect, referencing all the points to the worst position, 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.
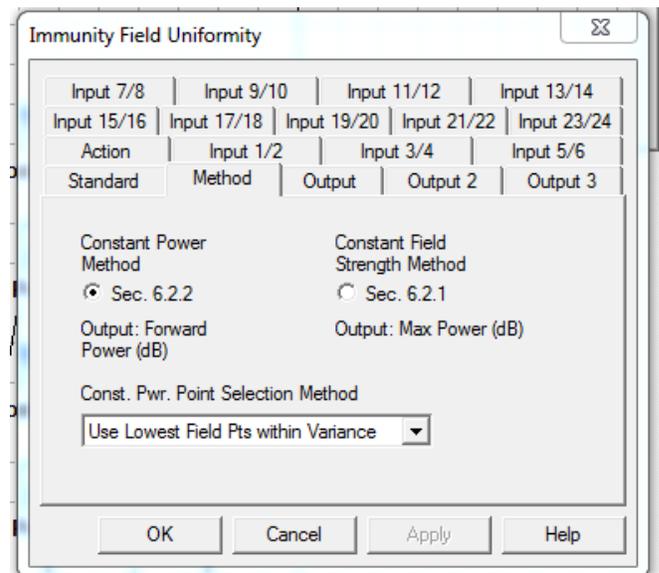
There are three selections for this method in the **Const. Pwr. Point Selection Method** list: **Use Lowest Field Pts within Variance** (default), **Use Highest Field Pts within Variance**, and **Use Minimum Variance of Field Pts**. These determine how the level will be calculated if more than 75% of points meet the 6 dB criteria.

- **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.

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**—Select an appropriate data element to store the required field strength. 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 3-12 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* on page 2 for information.

- **Failed Frequencies (variance)**—This data element will contain only those frequencies that failed the field uniformity variance calculation. These are frequencies that 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.
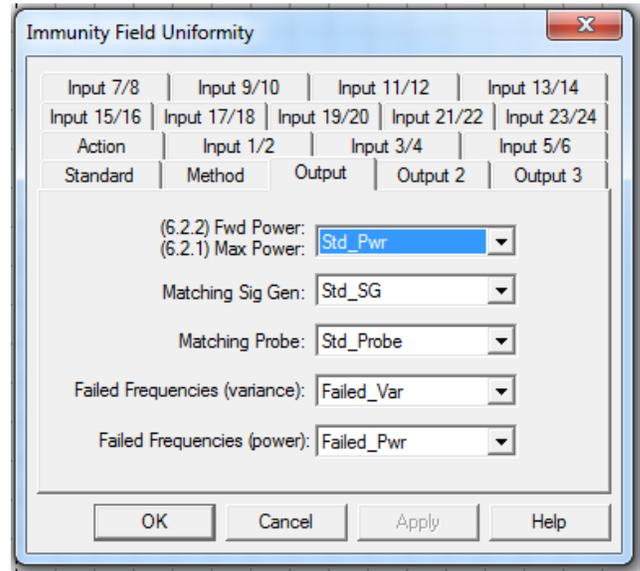
Output 2 Tab

The data elements in the **Output 2** tab are for statistical purposes only, and are not part of the required output of the field uniformity measurements.

- **Average (Used Pts)**—This data element will contain the average value of the points that met the 0-6 dB criteria.

- **Max (Used Pts)**—This data element will contain the maximum value of the points that met the 0-6 dB criteria.

- **Min (Used Pts)**—This data element will contain the minimum 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 0-6 for each of the positions in the measurement.
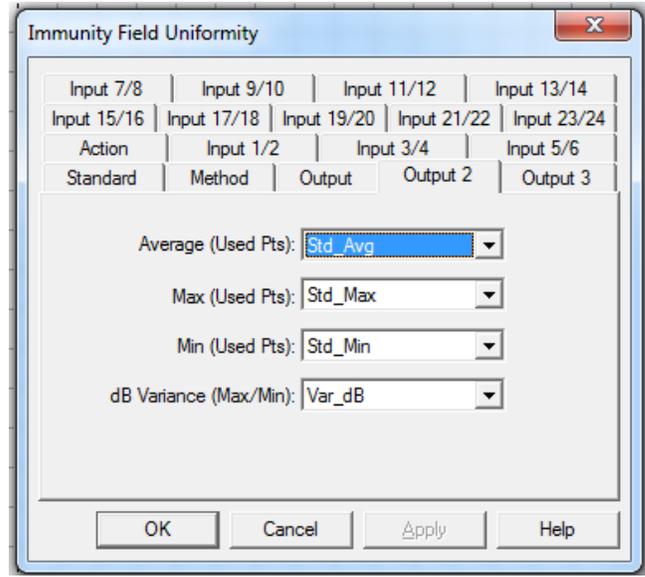
Output 3 Tab

The **Output 3** 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 to.

- **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.
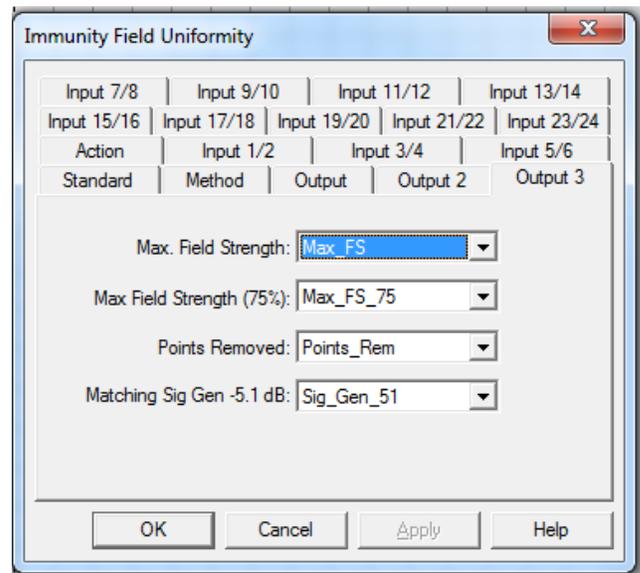
CS103/4/5 Measurement

The CS103/4/5 Measurement action is used to perform intermodulation and cross-modulation testing per MIL-STD 461.

Frequency Tab

- **Frequency**—This is the tuned frequency for CS103/105 and the LO frequency for CS104.

- **Intermediate Frequency (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

- **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 check box to prevent the command from being sent to the signal generator.

- **Use Minimum Output between Frequencies**—When this check box is selected, the signal generator will be set to its lowest output prior to stepping to the next frequency.

- **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 on the **Switchover** tab control the crossover frequency if an automatic switch is used to control two separate signal generators.

- **Primary SG Switch Freq**—Enter the frequency where signal generator 1 low frequency will cross over to signal generator 1 high frequency (see **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.

- **Secondary SG Switch Freq**—Enter the frequency where signal generator 2 low frequency will cross over to signal generator 2 high frequency (see **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.
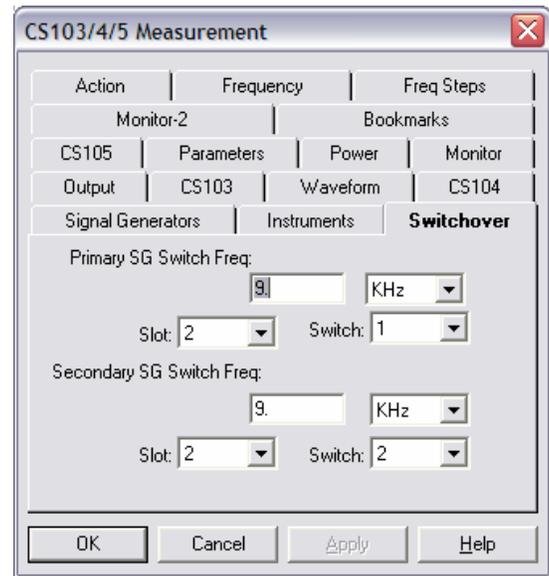
Output Tab

The **Output** 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.
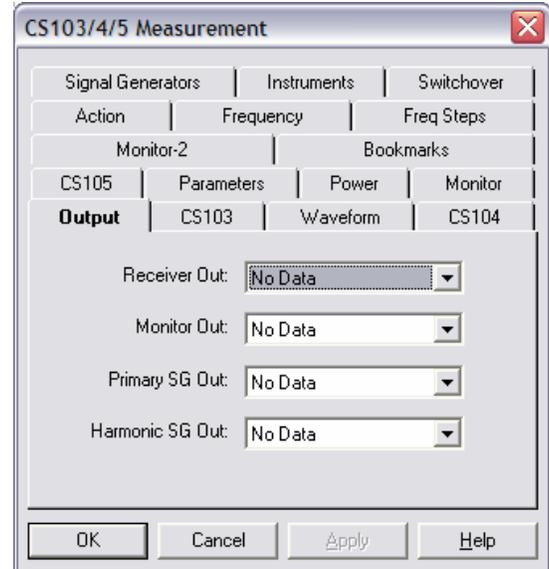
CS103 Tab

The **CS103** tab controls whether a CS103, CS104, or CS105 test is to be performed. Only one type can be performed at a time. Activation on this tab will disable the **CS104** and **CS105** tabs.

If **CS 103 Active** is selected, CS103 test will be performed using the information from the **Frequency** tab and the general equations as shown on this tab.

- **From Data**—If this check box is selected, the low and high frequency is obtained from a file to be specified by the user, and the low/high frequencies on this tab are ignored.

- **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.
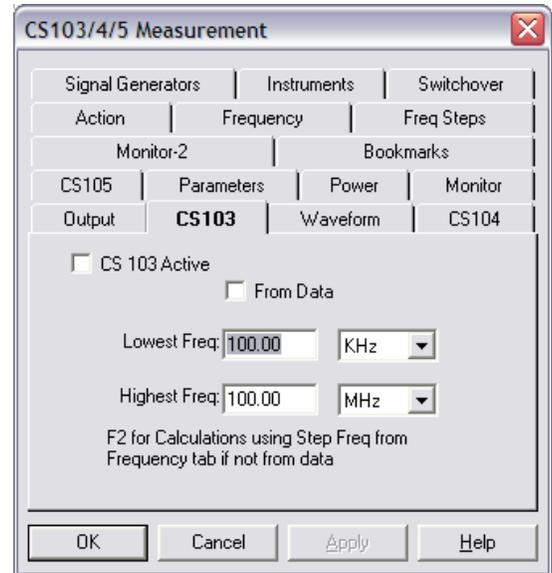
Waveform Tab

The **Waveform** tab establishes the intermodulation equation to be used in the test.

CS104 Tab

The **CS104** tab controls whether a CS103, CS104, or CS105 test is performed. If **CS104 Active** is selected, the CS104 test will be performed using the information from the **Frequency** tab and the general equations as shown on this tab.
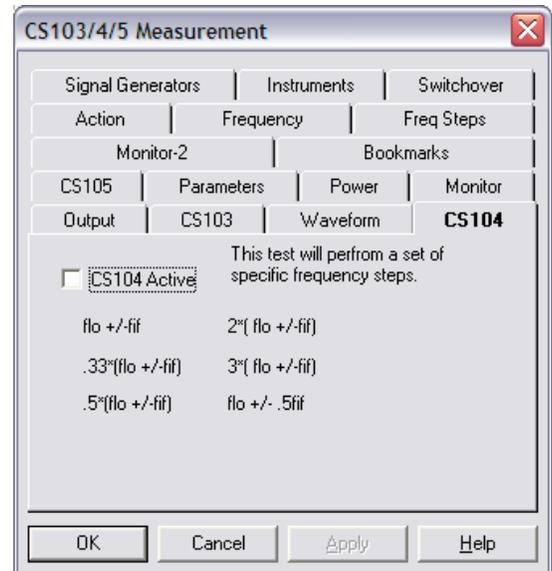
CS105 Tab

If **CS105 Active** is selected, the CS105 test will be performed using the information from the **Frequency** tab and the general equations as shown on this page.

- **Lowest/Highest Frequency**—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

The **Parameters** tab controls the setup of the receiver during this test.

- **RF Bandwidth**—Click the arrow to display the available RF bandwidth settings. Not all instruments will be able to use each of these settings, but the instrument driver will select 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 select the appropriate setting.

- **VBW for Analyzer**—Click the arrow to 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.

- **Step Size for Receiver**—Defines the step size 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.

- **Number of Sweeps**—A setting greater than 1 will put the analyzer in Max Hold and the specified number of sweeps will be performed. For EMC receivers, the software will sweep the receiver the number of specified times and hold the peak value internally.

- **Sweep Time**—This is how long the analyzer will take to sweep across the frequency band. This setting will determine the dwell time at each frequency point when using a receiver. Click the arrow in both boxes to select the settings.

- **Reference Level**—This value will set the reference level of the receiver/analyzer at each frequency step. Enter a value and select the appropriate units.

Power Tab

The settings on the **Power** tab control 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.
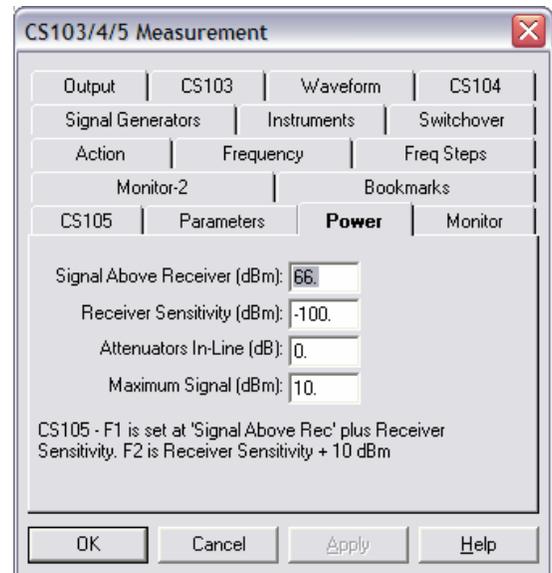
Monitor Tab

The **Monitor** tab allows you to control instruments that are being monitored during the test.

- **Monitor 1, 2, 3, 4**—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.
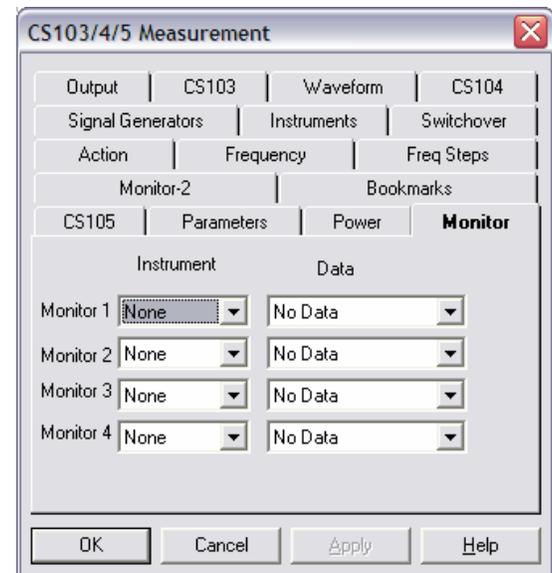
Monitor-2 Tab

The **Monitor-2** tab allows you to control instruments that are being monitored during the test.

- **Monitor 5, 6, 7, 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.

Reverberation Actions

Reverberation actions

Reverberation Calibration

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.

Reverb Calibration icon

Freq Tab

The **Freq** tab is used to set the frequency parameters. The reverberation calibration can be run by selecting the frequency range using a start/stop/step mode or a data element.

- **Set From Data**—Enables a predetermined set of frequencies to be read from a data set. When this check box is selected, the data set must be defined using the standard TILE! format of (*frequency*, *value*). **Set From Data** is particularly useful when performing IEC testing (or other standards) which require a percent-stepped frequency progression. Creating this progression with standard spreadsheet software is relatively easy. Write this data to file as an ASCII text, comma-separated variable file and define a data element as file-type with this access. This data element will then contain the appropriate information.

- **Start Frequency**—Sets the start frequency for this test. Enter data in the box and then click the arrow and select the units. The start frequency does not have to be less than the stop frequency. This action will start at the start frequency and 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, and uses the same conventions as the start frequency. 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 upon the **Log** or **Percent** check boxes. If neither log nor percent 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 this check box is 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 to 800 MHz into 5 sub ranges. It will then split the 800 MHz to 8 GHz into 5 ranges, but terminate the test on the last range that covers 1 GHz.

- **Percent**—This check box 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%. There are some calibration schemes that use 2% or 5%. Enter the value desired and select **Percent**.

- **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

The settings on the **Freq Step** tab control different personalities as we step from frequency to frequency during this test.

- **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 check box to prevent the on/off signal command from being sent to the signal generator.

- **Minimum Attenuation 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 check 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 on page 2. You can specify a fixed set of beginning amplitudes (less 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 Best Guess data element and offset will be ignored, except at the first frequency.

- **Display VSWR (Net Power Only)**—If both forward and reverse power are being measured, then the VSWR can be calculated and displayed on the screen as the test is running.

- **Power Off At Fail to Level**—When the leveling loop fails, a dialog box allows you 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. Generally, 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 box will cause the RF to be left on while the dialog box is present.

- **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.

- **Amplifier always on**—If this check box is selected, the amplifier, if identified as a bus-controlled instrument, will be put into operate mode at the beginning of the test and never put in standby until the test is complete. If this check box is not selected, when the operator clicks the pause button the amplifier will be put in standby.

Amplitude Tab

The **Amplitude** tab is used to select the amplitude data for this test.

- **Data**—Allows the operator to select the desired amplitude data to be used in the test. Click the arrow and 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**—Click the appropriate 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

- **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 into a signal generator with no leveling taking place. The **Probe** and **Pwr Meter** options cause specific leveling loops to execute where the software searches for the desired level; these require careful consideration of the **Tolerance** and **Max Count** fields. Making either of these selections will negate any calibration runs and cause a new leveled test to be performed.

- **Tolerance (% or dB)**—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 first attempt. 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.

- **Max 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 0 here will limit the software by halting the leveling if the anticipated signal generator level will exceed 0 dBm.

- **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 (either by 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 **Secondary Amp** tab or **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 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. A warning of this problem would be a failure to level which levels quickly if you click **Retry**. This field is also used in the secondary amp and check loops to determine the number of tries.

- **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

The **Chamber** 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 permeability (μ) of 3.79.

Enter the appropriate measurements in meters; the value should be as accurate as possible, but more than one decimal place is not necessary.

- **Height**—Enter the chamber height in meters.

- **Width**—Enter the chamber width in meters.

- **Length**—Enter the chamber length 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

The **Paddles** tab is used to define the number of tuner positions. Although setup for two positioners (two tuners), only one is required. If only one tuner is present, set the Positioner 2 values to 0 and 0 for start/stop. Required: Positioner(s) Output data element must be in conjunction with the number of positioners (tuners) used. The Output elements are found in the **Data** tab.

- **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 which 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 0 and the stop would be 315 (the eighth position starting from 0 with a 45 degree step size.

- **Margin**—The stop accuracy of the positioner. Generally this should be set to 0 or 1 (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.

- **SAE Method**—The SAE method of reverberation calibration that was set up for continuous rotation is no longer supported.

- **Tuner Speed Level**—This value will affect both tuners.

Power Meter Data Tab

The **Power Meter Data** 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.

- **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.

Power Cal. Tab

The **Power Cal.** tab is used to select the appropriate correction factors to be used for the injected power meter(s) and the monitor instrument.

- **Forward Max**—Select a data element that has the appropriate correction factor for the forward injected power meter. For example, if the directional coupler had a loss of 40 dB using a Preset data element with an amplitude of 40 or a From File data element representing the actual measured values, this will cause the value read from the instrument to be corrected by this 40 dB (it is an additive correction factor) to yield a corrected value.

- **Reverse Max**—Select a data element that has the appropriate correction factor for the reverse injected power meter. For example, if the directional coupler had a loss of 40 dB using a Preset data element with an amplitude of 40 or a From File data element representing the actual measured values, this will cause the value read from the instrument to be corrected by this 40 dB (it is an additive correction factor) to yield a corrected value.

- **Monitor Max**—Select a data element that has the appropriate correction factor for the monitor power meter (analyzer). For example, you might put a 30 dB attenuator on the cable from the chamber to the receiver used to monitor the antenna in the chamber. This would protect the receiver being used. This correction value is added to the receiver reading to recognize a corrected value.

- **Check Power**—In the older SAE method (continuous rotation) there was a requirement to measure the actual variation in power levels during a single rotation. If the continuous method is selected the user will be prompted for a separate rotational measurement for forward, reverse and receiver power. This setting is ignored if using the newer step tuned method.

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**—Select a data element to store the average probe RMS value. This is provided for reference purposes only.

- **Probe RMS – Min**—Select a data element to store the minimum probe RMS value. This is provided for reference purposes only.
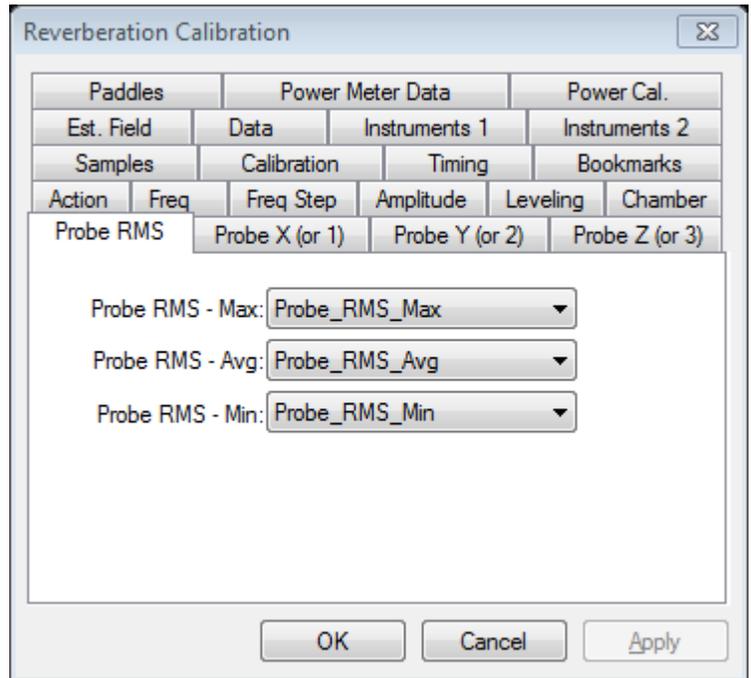
Probe X,Y,Z (or 1,2,3) Tab

The **Probe X,Y,Z (or 1,2,3)** tab is used to select the data elements to store the axis outputs of the probe during the various positions.

Note: The probe instrument drivers setting must be set to **Default** for proper operation.

The three axis tabs operate identically but the X (or 1) tab is described here.

- **Probe X,Y,Z – 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,Y,Z – 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,Y,Z – 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.
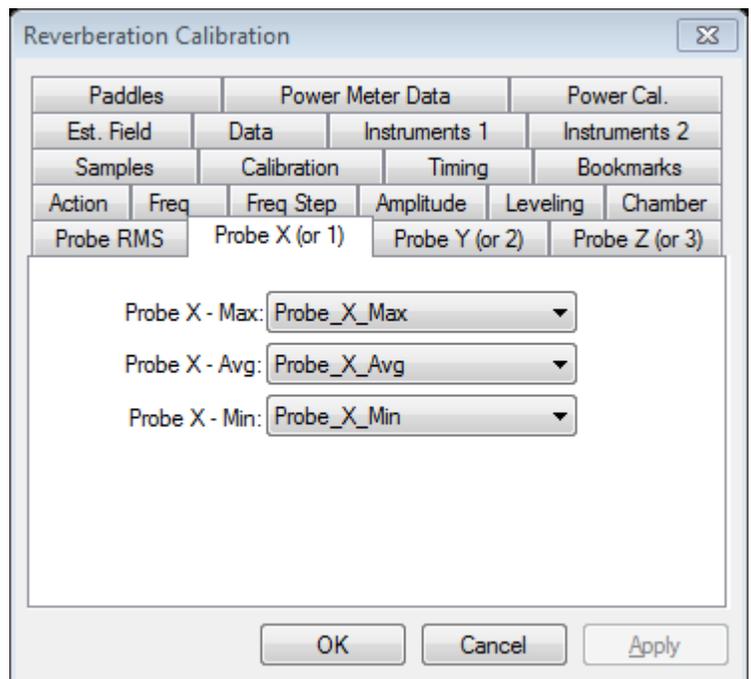
Est. Field Tab

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 Prime Max (Quality Factor)**—Stores the maximum calculated Q prime max (or Q) 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 Prime Min**—Stores the minimum calculated Q prime 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:

$$10 * Log_{10} \left(\frac{P_{Fwd} - P_{Rev}}{P_{Rec}} \right)$$

Data Tab

The **Data** 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.

- **Positioner 1 Data**—Stores positioner 1 data to identify the paddle position that had the maximum values.

- **Positioner 2 Data**—Stores positioner 2 data to identify the paddle position that had the maximum values.

- **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. This information is required for the evaluation step.

Reverberation Calibration

Paddles	Power Meter Data	Power Cal.

Probe RMS	Probe X (or 1)	Probe Y (or 2)	Probe Z (or 3)

Samples	Calibration	Timing	Bookmarks

Action	Freq	Freq Step	Amplitude	Leveling	Chamber

Est. Field	Data	Instruments 1	Instruments 2

Positioner 1 Data: Position_1_Data

Positioner 2 Data: <no data>

Signal Generator: Sig_Gen_Data

Number of Points: Number_Of_Positions

OK   Cancel   Apply
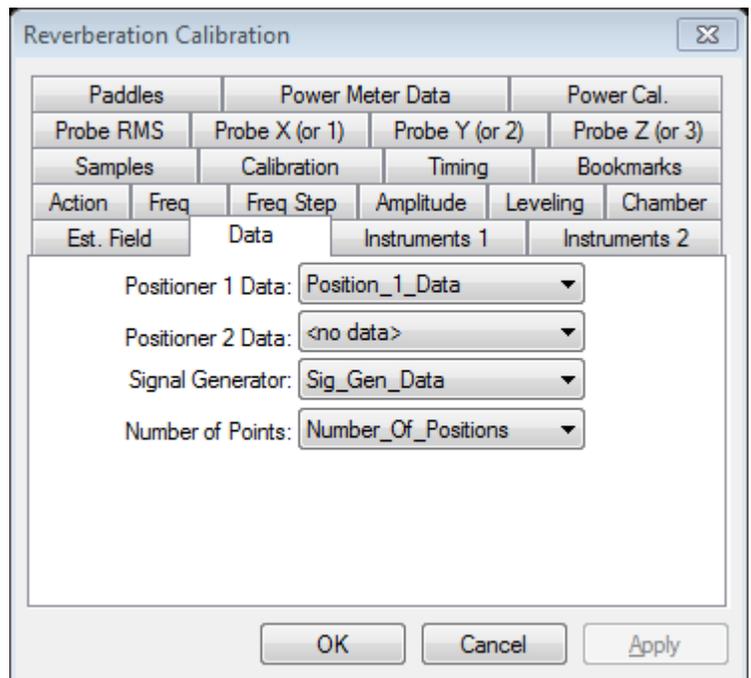
The **Instruments 1** tab is used to specify various pieces of test equipment used during the reverberation calibration. The only mandatory instruments are the signal generator, power meter/monitor, and at least one positioner. Paddle positioners can be manually controlled and the operator will be prompted for motion if they are so marked.

- **Recv/Pwr Meter Monitor**—Click the arrow and select the correct instrument. 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**—Click the arrow and select the correct instrument. If you select an instrument that does not have the characteristics of a signal generator the action will not work properly.

- **Pos 1 (Horiz.)**—Click the arrow and select the correct instrument. If there is only one positioner, it should be identified as positioner 1. The reference to horizontal and vertical is for identification purposes only and is not mandated by any standard.

- **Pos 2 (Vert.)**—Click the arrow and select the correct instrument. If there is only one positioner, it should be identified as positioner 1. 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, click the arrow and 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.

Use the **Instruments 2** tab to set the power meter and probes used during the calibration.

- **Power Meter**—Click the arrow and select the correct instrument. 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 unless a channel-specific instrument driver is used.

- **Power Meter 2**—Click the arrow and select the correct instrument. 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 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**—Both probes and power meters have internal leveling issues that make it necessary to allow small time delays between changes in the signal generator levels and the reading of each instrument. There is a separate delay time for each instrument.

- **Probe**—Click the arrow and select the correct instrument. If you select an instrument that does not have the characteristics of a probe the action will not work properly.

Samples Tab

The **Samples** tab controls the number of readings each instrument will make when triggered. If more than one sample is specified, then the user can choose to use the maximum or the average of the readings.

- **Samples Per Reading**—The number of samples for each instrument. This can be different for each instrument.

- **Probe Method**—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.

- **Pwr Mtr Method**—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.

Calibration Tab

The **Calibration** 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. Click the arrow and select a data element that has the correction factors.

- **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. Click the arrow and select a data element that has the correction factors.

- **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. Click the arrow and select a data element that has the correction factors.

- **Probe/Probe 2**—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%). So a correction factor of 95% would be 0.95; 105% would be 1.05. Click the arrow and select a data element that has the correction factors.

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.
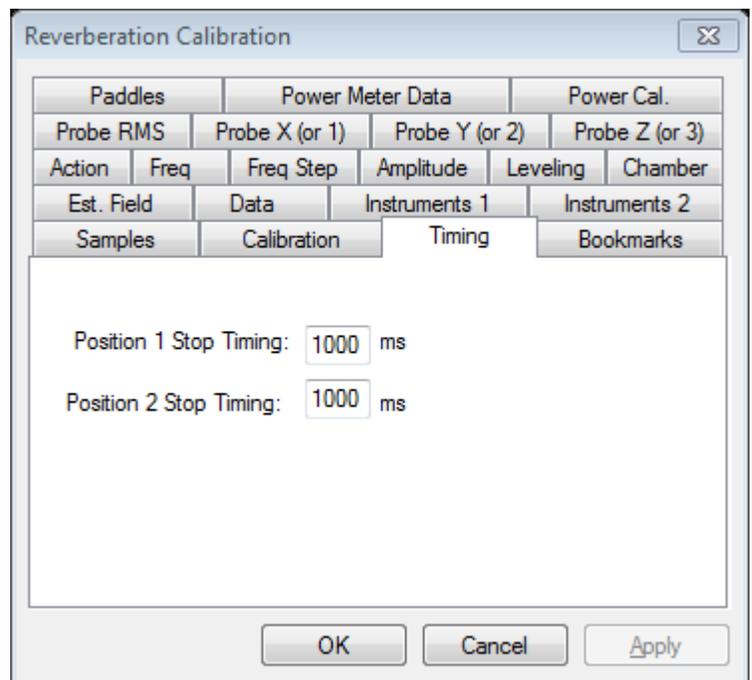
Reverberation Calibration Dialog Box

You can execute the Reverberation Calibration action either by clicking the action and selecting **Execute**, by executing it as part of the sequence of tests being executed, or by using the run/single options from the Windows menu bar. 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

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.

Reverberation Calculation icon

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

The entered values should be as accurate as possible, but more than one decimal place is not necessary.

- **Height**—Enter the chamber height in meters.

- **Width**—Enter the chamber width in meters.

- **Length**—Enter the chamber length in meters.

- **Skin Depth**—The skin depth 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.

- **Unloaded ACF**—The Antenna Calibration Factor (ACF) is defined as: (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, this is the efficiency of the transmit and 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.

    Select a data element for this input. 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

The **Est. Field** tab controls output of some calculated, theoretical, or measured values.

- **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 Prime**—Stores the maximum calculated Q prime value found for each of the chamber probe positions. Select a data element for this output. Q prime 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* on page 2 for explanation of the symbols.)

  See *Q Prime Max (Quality Factor)* on page 2.

- **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. Select a data element for this output. 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. Select a data element to store this output. Antenna Calibration Factor (ACF) is defined as:

    (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—** Select a data element for this output. 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

The **Output** 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, -X, -Y, -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:

$$\sigma = \sqrt{\frac{\sum\left(\overleftrightarrow{E_i} - \langle\overleftrightarrow{E}\rangle\right)^2}{\eta - 1}}$$

These calculations are performed for each axis of the probe and for the sum of the probe.

Where:

η –   The number of measurements *i*.

$\overleftrightarrow{E}_i$ –   The individual E-field measurement normalized (to the square root of the input power).

$\langle\overleftrightarrow{E}\rangle$ –   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. Select a data element for this output.

- **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).

$$CLF = \frac{CVF}{AVF}$$

$$CVF = \left\langle \frac{P_{AveRec}}{Pinput} \right\rangle_{\eta}$$

$$AVF = \left\langle \frac{P_{AveRec}}{Pinput} \right\rangle_{8\ at \le 10f_0\ or\ 3\ at > 10f}$$

- **Time Constant**—Select a data element for this output. The Time Constant (T) is calculated as:

$$T = Q/2\pi f \text{ (where f is frequency in Hz)}$$

Input 1-9 Tabs

There are two 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. Select a data element for the output for each field.

Input 1-9 Probe Tabs

There are two 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. Select a data element for the output for each field.

Reverberation 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.

Reverberation Test icon

Frequency Tab

- **Set From Data**—Enables a predetermined set of frequencies to be read from a data set. When this check box is selected, the data set must be defined using the standard TILE! format of (*frequency*, *value*). This particularly useful when performing IEC testing (or other standards) which require a percent-stepped frequency progression. Creating this progression with standard spreadsheet software is relatively easy. Write this data to file as an ASCII text, comma-separated variable file and define a data element as file-type with this access. This data element will then contain the appropriate information.

- **Start Frequency**—Sets the start frequency for this test. Enter data in the box and select the units from the list. The start frequency does not have to be less than the stop frequency. This action will start at the Start Freq and then step towards the Stop Freq in the manner instructed; it does not matter if this is up or down.

- **Stop Frequency**—Sets the stop frequency for this test. It uses the same conventions as start frequency. Enter data in the box and select the units from the list. 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, and will change meaning depending upon the **Log** or **Percent** check boxes. If neither log nor percent 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 the check box is 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 to 800 MHz into 5 sub ranges. It will then split the 800 MHz to 8 GHz into 5 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%. There are some calibration schemes that use 2% or 5 %. Enter the value desired and select the **Percent** check box.

Freq Step Tab

- **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 check box to prevent the on/off signal command from being sent to the signal generator.

- **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 check 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 on page 2. You can specify a fixed set of beginning amplitudes (less 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.

- **Display VSWR (Net Power Only)**—Selecting this check box causes the VSWR to be calculated and displayed as the test is running.

- **Power Off At Fail to Level**—When the leveling loop fails, a dialog box will allow the user to click **Retry this step, Skip this step, or Quit**. While this dialog box is present, this check box determines whether power is on or off. Generally, 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.

- **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.

- **Modulation Always On**—When this check box is selected, the modulation will be set at the beginning of the test and left on at all times. All leveling will be performed with modulation on.

- **Amplifier Always On**— When this check box is selected, the amplifier will be placed in operate mode at the beginning of the test and left in operate mode until the completion of all steps. It will be returned to standby after the RF is turned off at the last step. If this check box is deselected, when the **Pause** button is clicked and the software halts automatic stepping, the amplifier will be put in standby. When **RF On** is clicked the amplifier will be returned to operate mode.

The field during the EUT test is established using the equations in IEC 61000-4-21 2011.

The **Amplitude** tab sets the Input data elements that contact the information required to make this calculation.

- **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. Select a data element for this output (required). The field will be determined using the following:

$$P_{Input} = \left[\frac{E_{Test}}{\langle \overleftrightarrow{E} \rangle_{24 \ or \ 9} x \sqrt{CLF(f)}} \right]^2$$

Where:

P$_{Inout}$	–	Forward power injected into the chamber.
E$_{Test}$	–	Required field strength.
CLF(*f*)	–	Chamber load factor at frequency.
$\langle \overleftrightarrow{E} \rangle_{24 \ or \ 9}$	–	Average of the normalized E-field from empty chamber.

- **Normalized Empty**—This is the normalized field probe from the evaluation step. Select a data element for this output (required).

- **CLF Data**—This is the Chamber Loading Factor (CLF) determined from the pre-test calibration and evaluation. Select a data element for this output (required).

- **Default Step Size**—This field is disabled.

- **Data Units**—Generally all testing is done in dBm; V/m and dBuV are not active.

During the leveling process, the **Leveling** tab controls parameters such as tolerance, maximum and minimum signal generator step size, and the maximum safety limit for the signal generator to amplifier drive level.

- **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.

- **First Count**—The number of steps to try and level before any failure message is given the operator.

- **Tolerance (% or dB)**—The **Plus** and **Minus** boxes allow 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 that jumps 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. 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 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.

- **Max Step**—Sets the maximum amplitude increment that the signal generator will perform.

- **Min Step**—Sets the minimum amplitude increment that the signal generator will perform.

- **Max Count**—Sets the maximum number of tries to establish a level measurement reading before the system reports an error. This operator-defined field should be over 10 iterations for situations in which you are seeking a large field or in cases where 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**.

- **Manual Offset**—When in manual mode (automatic testing paused), this value controls the offset to the signal generator on initial power on.

- **Continue on Fail-To-Level**—Allows the user to continue on with the leveling steps even when a step fails to level.

- **Best Guess Amp**—This is used to set the signal generator to a predetermined level to speed up the leveling process, when approximate signal generator levels are known. The data set must be a defined .csv data element.

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 Active**—Select this check box to activate thresholding.

- **Available and Selected Columns**—Choose an action from the **Available** column to the **Selected** column in the order you want the thresholds to be selected.
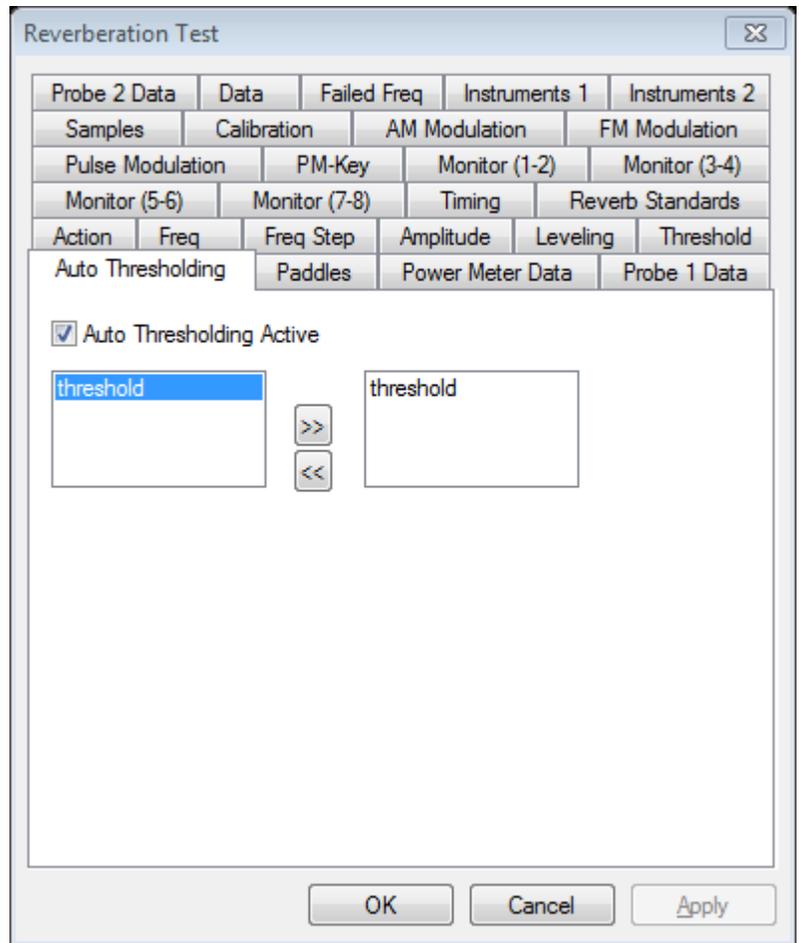
Paddles Tab

The **Paddles** tab is used to define the number of tuner positions. Although setup for two positioners (two tuners), only one is required. If only one tuner is present, set the Positioner 2 values to 0 and 0 for start/stop.

- **Start/Stop/Step**—Enter an appropriate start and stop position for the tuners. Typically the start will be zero and the stop will be a value which 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 0 and the stop would be 315 (the eighth position starting from 0 with a 45 degree step size).

- **Margin**—This is the stop accuracy of the positioner. Generally this should be set to 0 or 1 (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.

- **Speed Level of Faster Tuner**—This value will be sent to the tuner controller to set the speed of the rotation for tuner 1. If two tuners are used, then this will be the faster value, and the second tuner will be a ratio of 2:1 (half-speed) of tuner 1.

Power Meter Data Tab

The **Power Meter Data** tab determines the output data elements for various measured values during the test.

- **Recv. Antenna Max**—The maximum value of the received antenna in the chamber. Select a data element for this output.

- **Recv Antenna Avg**—The average value of the received antenna in the Chamber. Select a data element for this output.

- **Inject Pwr – Fwd**—The measured forward power into the chamber. Select a data element for this output.

- **Inject Pwr – Rev**—The measured reverse power into the chamber. Select a data element for this output.

- **Inject Pwr – Net**—The calculated net power into the chamber. This data is calculated from the forward and reverse power levels. Select a data element for this output.

Probe 1 Data / Probe 2 Data Tabs

These tabs allow 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**—The maximum value of the X channel of the probe for all tuner positions is stored. Select a data element for this output.

- **Probe Y – Max**—The maximum value of the Y channel of the probe for all tuner positions is stored. Select a data element for this output.

- **Probe Z – Max**—The maximum value of the Z channel of the probe for all tuner positions is stored. Select a data element for this output.

- **Probe Max**—The maximum value of the RMS value of the probe for all tuner positions is stored. Select a data element for this output.

- **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. Select a data element for this output.

Data Tab

The **Data** tab controls the output of position, signal generator, and certain calculated fields during the multiple paddle positions defined in the test.

- **Positioner 1 Data—**The position at which the highest field was generated is recorded. Select a data element for this output.

- **Positioner 2 Data—**If there are two positioners, the position at which the highest field was generated is recorded. Select a data element for this output.

- **Signal Generator—**The signal generator level at the position where the highest field was generated is recorded. Select a data element for this output.

- **Number of Points—**Strictly for later use, the number of paddle positions is recorded here. Select a data element for this output (required).

- **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. Select a data element for this output.

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. Use the **Failed Freq** tab to select appropriate data elements.

- **Failed Field**—Stores the frequencies at which a failure is noted. Select a data element for this output.

- **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 if position 2 is used).

- **Sig Gen**—The signal generator level at the point when a failure is marked is stored. Select a data element for this output.

- **Power Meter**—The current power level when a failure is marked is stored. Select a data element for this output.

Use the **Instruments 1** tab to select each
instrument that will be used during the test. A
receiver and signal generator are required.
Positioners and amplifier are optional.

- **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.

- **Delay**—Some power meters require a
  small delay between the time the
  signal level is changed and the actual
  reading of the instrument. This setting
  allows the user to adjust this delay
  time to ensure accurate readings.

- **Signal Generator**—Select the
  instrument from the list that will be
  used as a signal generator.

- **Pos 1 (Horiz.)**—Select the instrument
  from the list that will be used to move
  the first, or only, paddle.

- **Pos 2 (Vert.)**—Select the instrument
  from the list that will be used for
  paddle 2. Positioner 2 is not required.
  Although not required, generally if
  there are two positioners one is in
  vertical orientation and one is
  horizontal.

- **Manual**—If either positioner is not remotely controlled, selecting this check box will cause the software to
  prompt the operator to move to the required position.

- **Amplifier**—If the amplifier is bus-controlled, click the arrow and select the correct instrument.

Instruments 2 Tab

The **Instruments 2** tab is used to select the power meters and probes used during the test. The power meters on this tab are used to measure injected power into the chamber.

- **Power Meter 1**—Click the arrow and select the correct instrument. This will be used to measure forward power. This can be a spectrum analyzer or power meter, but if a two-channel power meter is used it will read channel 1, unless a channel-specific driver is used.

- **Power Meter 2**—Click the arrow and select the correct instrument. This will be used to measure reverse power. This can be a spectrum analyzer or power meter, but if a two-channel power meter is used it will read channel 2, unless a channel-specific driver is used.

- **Delay**—Some power meters and probes require a small delay between the time the signal level is changed and the actual reading of the instrument. This setting allows the user to adjust this delay time to ensure accurate readings.

- **Use Net Power**—If this check box is selected, then all leveling to the power meters (injected power) will be to net power, instead of forward power.

- **Probe/Probe 2**—Click the arrow and select the correct instrument.

Samples Tab

The **Samples** tab controls the number samples taken for each instrument. By default an instrument is read one time, as needed, but when these are changed the instrument will be read multiple times and then the maximum or average will be used for leveling or recording purposes.

- **Samples Per Reading**—The number of samples for each instrument. This can be different for each instrument.

- **Probe Method**—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.

- **Pwr Mtr Method**—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.

The **Calibration** 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. Click the arrow and select and select a data element that has the correction factors.

- **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. Click the arrow and select a data element that has the correction factors.

- **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 from the chamber to the wall to the instrument, this value would be the loss of the attenuator. This value is added to the instrument reading to correct it. Click the arrow and select a data element that has the correction factors.

- **Field Probe**—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%). So a correction factor of 95% would be 0.95; 105% would be 1.05. Click the arrow and select a data element that has the correction factors.

AM Modulation Tab

Select the **AM Modulation** tab if you need AM modulation turned on during the reverberation test. The action will then turn on AM modulation at the signal generator and turn on the appropriate internal/external switch (if the generator has this feature).

- **Enable**—Select this check box to turn on (enable) AM modulation. If this box is selected, 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 basically trying to duplicate the human voice on a radio transmitter. Enter an appropriate numerical value in the **Depth** box. If the standard is not stated in percent, you must convert this number to a percentage before entering it.

- **Frequency**—In stating a sound (using depth), we then specify how often the sound is manufactured. This is stated in Hz (or kHz). Enter an appropriate number in the box and click the arrow to select the units.

- **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.

- **Waveform**—Denotes how the carrier wave will be varied in accordance with the characteristics of the modulation signal.

The **FM Modulation** tab selects and configures FM modulation. The generator must be capable of this function or these settings will have no effect.

- **Enable**—Select this check box if you want to turn on this function. If this check box is selected 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. 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 is not covered in this action. It can be configured using the *GPIB Control Instrument Commands* page 2.

- **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**—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 a value in the box and selecting the appropriate unit from the list.

- **Frequency**—In stating a sound (using depth), we then specify how often the sound is manufactured. This is stated in Hz (or kHz). Enter a value in the box and selecting the appropriate unit from the list.

- **External Gating**—Select this check box if you need the modulation to be controlled by an external gating signal.

- **Waveform**—Denotes how the carrier wave will be varied in accordance with the characteristics of the modulation signal.

Pulse Modulation Tab

This tab controls Pulse Modulation that is output by the signal generator on top of the current frequency and amplitude of the CW output.

- **Enable**—Select this check box to turn on pulse modulation. If this is not selected, pulse modulation (neither internal nor external) will be turned on.

- **External**—If the modulation is being fed into the signal generator on the external input connector, use this setting.

- **Pulse Rate**—Pulse rate in Hz of the signal desired.

- **Pulse Width**—Pulse width in Hz of the signal desired.

- **External Gating**—If the signal is externally gating the signal generator, select this check box to turn on external gating. This is a rare setting, so do not turn it on without a clear understanding of what is expected from the signal generator and external equipment.

PM-Key Tab

There are some standards that require 1–3 Hz square wave modulation on top of other forms of modulation. The setting on the **PM-Key** 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**—Select this check box to turn on this feature.

- **On Time**—The amount of time the RF will be on as part of the pulse waveform.

- **Off Time**—The amount of time the RF will be off as part of the pulse waveform.

- **Number of Cycles**—The number of times that the on/off sequence will be executed.

- **Monitor4 On Data**—If a data element is selected, and an instrument is shown in the Monitor 4 section of the **Monitor (3-4)** tab, we will read the monitor during the on cycle and save the value.

- **Monitor4 Off Data**—If a data element is selected, and an instrument is shown in the Monitor 4 section of the **Monitor (3-4)** tab, we will read the monitor during the off cycle and save the value.

Monitor (1-8) Tabs

The monitor tabs 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 **Instrument** 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.

Timing Tab

The Reverberation action has a number of timing issues that affect accuracy of the test. Use the **Timing** 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 which 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 a number of reverberation standards supported

- **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.

- **FWD**—The forward power used for DO160G. Produced by the load test or pre-test. Used by DO160G test.

Reverberation Test Dialog Box

You can execute the Reverberation Test action either by clicking the action and selecting **Execute**, by executing it as part of the sequence of tests being executed, or by using the run/single options from the Windows menu bar. 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.

- **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)/Setp#. This can modified by the user, but cannot be smaller than 2.

- **Frequency/Tuner Position**—These values are determined at the pause mode when the user stops the automatic Process.

- **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 click 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.
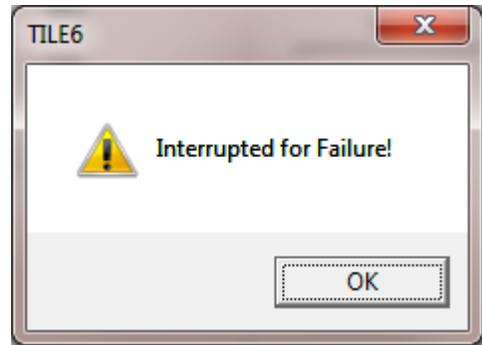
IEC 61000 Series Test Actions

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. At this time there is no documentation for these actions; contact TILE! Support for more information. See page 2 for contact information.

IEC 61000 Series Test actions

11.0   Appendix A: Math Functions

Available Operators

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

There are conditionals available to you, such as the greater than/less than. 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 {**x ≥ y**}	Determines those frequencies where the value is greater than or equal to the *y* value.
Less than or equal {**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**)	Arccosine of each value in the data element *x*.
acosh(**x**)	Arccosine (hyperbolic) of each value of the data element *x*.
adb(**x**)	Converts amps from dBA using **10^(x / 20)**.
add(**x**,**y**)	This function adds values in *x* against data in *y* in a sequential step without checking frequency. The number of elements should be the same. If there are more elements in *x* than *y*, it will not execute. If there are more elements in *y* than *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. The standard addition operator (**+**) should be used for most cases.
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**)	Anti-sine of each value of the data element *x*.
asinh(**x**)	Anti-sine (hyperbolic) of each value of the data element *x*.

atan(**x**)	Anti-tangent of each value of the data element *x*.
atanh(**x**)	Anti-tangent (hyperbolic) of each value of the data element *x*.
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 frequency difference (the bandwidth) 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 frequency points. For the lower offset, center, and upper offset it returns the frequency and amplitude.
ceil(**x**)	The smallest integer, at each frequency, that is greater than or equal to *x*.
cos(**x**)	Cosine of each value in the data element *x*.
cosh(**x**)	Anti-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 dBA using **20 * log(x)**.
dbp(**x**)	Converts the *x* value in watts to dB using **10log(x)**. It is equivalent to dBw(x). To convert from dBm first divide the dBm number by 1,000.
dbuvtov(**x**)	Converts the *x* value in dBuV value to volts using **10^((x-120)/20)**. To convert dBuV to uV divide by 1 million.
dbv(**x**)	Converts the *x* value in volts to dBV. The basic equation is **20 * log(x)**. For dBuV multiply the voltage by 1,000,000.
dbw(**x**)	Converts the *x* value in watts to dBm using **10log(x*1000)**.
diff(**x**,**y**,**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 frequency. The number of elements should be the same. If there are more elements in *x* than *y*, it will not execute. If there are more elements in *y* than *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. The standard addition operator (**/**) should be used for most cases.
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 resultant data element. This is convenient for gaps in limit lines.
exp(**x**)	Returns the exponential value of *x* in log base **e**.
fabs(x)	Returns the absolute value of the floating point value.
finddupl(**x**,**c**)	Scans in the input data element and eliminates those frequencies within *c* of each other, keeping the higher value. This is convenient for eliminating multiple peaks from modulated or noisy emission
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 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. 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=**Data element **c1=**Fundamental frequency **c2=**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 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. An example would be that if the last frequency in *x* is 30 MHz, @ 5 dBuV the result would be 30e6,30e6.
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 writes the maximum of the two to the resultant data element. It can be used to find the maximum value from H/V data but they must have the same number of points. It can also be used to add multiple trace or limit line data elements to make one element. Examples: **limit : max(limit_1,limit_2)** - will add two limit line segments together. **limit : max(max(limit_1,limit_2),limit_3)** - will add three limit segments together.
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 writes the minimum of the two to the resultant data element. This can be used in the same way as the **max(x,y)** equation for linking data items together.
mult(**x**,**y**)	This function multiplies values in *x* by data in *y* in a sequential step without checking frequency. The number of elements should be the same. If there are more elements in *x* than *y*, it will not execute. If there are more elements in *y* than *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. The standard addition operator (*****) should be used for most cases

notempty(**x**)	This function returns all frequencies in the word-type data element *x* which 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.
pdb(**x**)	Converts the *x* value in dBW to watts. The basic equation is **10^(x/10)**. This is identical to the **wdb** function.
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 only insert 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**)	The resultant data element is all values that are greater than a constant *c*.
pfgreat(**x**,**c**)	Checks for values in data element *x* which are greater than or equal to *c* and returns the value **Pass**. If the value is less, the return value is **Fail**.
pfless(**x**,**c**)	Checks for values in data element *x* which are less than or equal to *c* and returns the value **Pass**. If the value is less, the return value is **Fail**.
pow(**x**,**c**)	The value of *x* raised to the *c*. For example, x^2 is **pow(x,2)**.
reduce(**x**,**c1**,**c2**)	This function compresses the number of points in the data element ignoring any points that are within **c1dB** 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*. Also, the difference between the point after and the current point must be less than **c2**.
rms(**x**)	This function calculates the RMS 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**)	Standard Correlation for Ford Motor. Does a correlation analysis on the data in elements *x* and *y* to the Ford specification.
sin(**x**)	Determines sine of each value in the data element *x*.

sinh(**x**)	Determines sine (hyperbolic) 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. Example: **Span_Vert : span(vert) = 1GHz,970E6** Where: **vert** is a span of 30 MHz to 1 GHz
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*. For example, the equation **speaks(x,8)** would look for all signals that were at least 8 dB from low to high amplitude. **Note: c2** is a bandwidth variable. If you are nesting this in another equation you will need to populate this variable. Example: **Suspects : top(speaks(x,8,250E3),6)** - will return the six highest peaks that are separated by a minimum of 250 kHz.
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 frequency. The number of elements should be the same. If there are more elements in than *y*, it will not execute. If there are more elements in *y* than *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. The standard addition operator (−) should be used for most cases.
tan(**x**)	Determines the tangent of each value in the data element *x*.
tanh(**x**)	Determines the tangent (hyperbolic) of each value in the data element *x*.

thd(**x**,**c1**,**c2**)	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: $$\text{THD} = \frac{\sqrt{V_2^2 + V_3^2 + V_4^2 + \cdots + V_\infty^2}}{V_1}$$ 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><td>**MHz**</td><td>**Test_Data**</td></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.
top(**x**,**c**)	Determines the top **c** number of points in the data array. The default is 10 if **c** is not stated.
top10(**x**)	Determines the top 10 values in the array *x*.
toy(**x**)	Calculates a bin function for emissions data to Toyota specification.
trim(**x**)	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.
valuetofreq(**x**)	This is the opposite of **freq(x)**. Will return an element where the frequency will equal the amplitude value. Can be used to find a certain harmonic of a frequency if used with the **freq(x)** function. Example: To get the 2nd harmonic of a frequency use – **Harm_2: valuetofreq(2*freq(Tx_Freq))**

vdb(**x**)	Converts a value in voltage from the dBV value using the equation **10^(x/20)**.
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 a value in voltage to a value in dBuV using the equation **log(x * 1E6) * 20**.
wdb(**x**)	Converts a value in watts from the dBm value using the equation **10^((x/10)/1E3)**.