Huntron Workstation 4.3 Oscilloscope Tutorial

HUNTRON WORKSTATION 4.3 OSCILLOSCOPE TUTORIAL 1

HUNTRON WORKSTATION OSCILLOSCOPE SOFTWARE TUTORIAL.....3

Huntron Workstation software 3

Installation Instructions 3

Installation of Additional Support Software 3

Hardware Setup 4

Huntron Workstation Main Interface 4

The Toolbar 5

Hardware Setup 6

Using Workstation with an Oscilloscope 11

Creating a Oscilloscope Board Test 11

Test Building Procedures 12

Creating a Board Database 13

Adding a New Sequence 13

Adding a New Component (Test) 14

Pin Edit 16

Modifying a Range (Oscilloscope Measurement Setup) 16

Mounting a PCB in an Access Prober 19

Prober Setup – Camera Offset part 1 20

Prober Setup – Camera Offset part 2 20

Prober setup – Alignment part 1 21

Prober setup – Alignment part 2 22

Creating a Board Image 23

Prober setup – TP Teach – XY Setup 24

Prober setup – TP Component Teach Height (Z) 25

Manual Modes 26

Scanning a Sequence or Component 27

Viewing Signatures – Troublesheet 28

Viewing Signatures Troublesheet Report 30

Viewing Signatures – Right Click 32

Component Scans Information 33
Component Scans – Right Clicking (Auxiliary menus) 33
Component Scans – Right Clicking (Auxiliary menus) 34
Component Scans – Right Clicking (Auxiliary menus) 34
Range Tab – Right-click Menu 35
Sequence/Component/Pin Editing – Right Click menus 36
Pin Editing – Right Clicking (Auxiliary menus) 37
Huntron Workstation Buttons Feature 38
Technical Support 40

Huntron Workstation Oscilloscope Software Tutorial

Huntron Workstation software

It is very helpful if you have a working knowledge of Microsoft Windows prior to using Huntron Workstation.

You are allowed to create a backup copy of the software disk. *Your purchase agreement allows for copies to be made for backup purposes only- copying for distribution or resale is strictly prohibited.*

Installation Instructions

Install the software **BEFORE** connecting any hardware. Uninstall any previous versions prior to loading the current version. For more installation details follow the "Getting Started" sheet included with your Huntron product.

Uninstall any previous versions of Workstation prior to installing the new version. Insert the Huntron Workstation CDROM. The CDROM should Autorun and begin the installation process. If your PC has Autorun disabled then browse Window and select WorkstationSetup.exe. This will begin the installation process. During the install you will be asked to install additional software for Access Probers. Select the appropriate response based on your Prober hardware.

Installation of Additional Support Software

Installation of software for the support of supported oscilloscopes will be required for the proper operation and control.

Keysight (Agilent) DSO1022A and DSO-X-2022A

Insert the Keysight IO Libraries Suite CD or download the software from keysight.com. Run the IO Libraries Suite installer. The install process should begin. Use default selections as prompted during the installation process as there are no special conditions needed for the oscilloscope hardware.

If the **Agilent/Keysight DSO1022A** is being used copy the "**HuntronExternal3.dll**" file from the path "C:\Program Files\Huntron Workstation\E3DSO1022A" to the path "C:\Program Files\Huntron Workstation". Select "Yes" to overwrite the existing "HuntronExternal3.dll" file in that path. If the **Keysight DS-X-2022A** is being used then use the "**HuntronExternal3.dll**" file from the E3DSOX2022A folder.

Hardware Setup

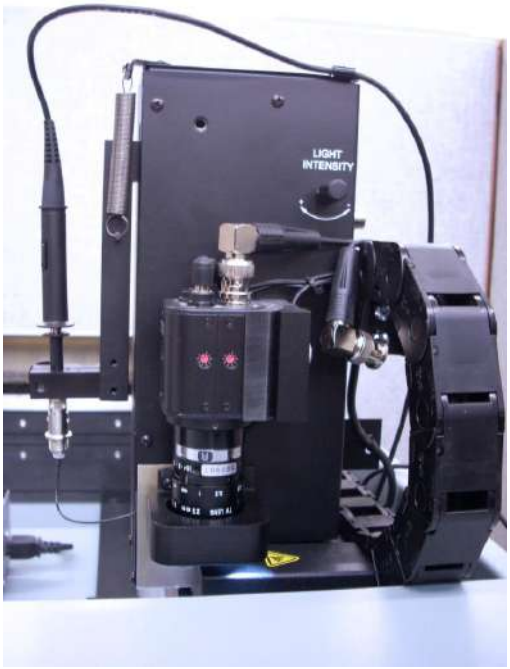
If a Huntron Access Prober is being used, connect it to power.

Connect the Huntron Access to the PC with the USB Cable.

Connect the oscilloscope to Power.

Connect the oscilloscope to the PC using the USB cable.

Using the "Installation of Oscilloscope Probe Mounting Kit" instructions, connect the oscilloscope to the Huntron Access by placing the oscilloscope probe into the probe adapter mounted to the Z axis rail as shown in the image below.

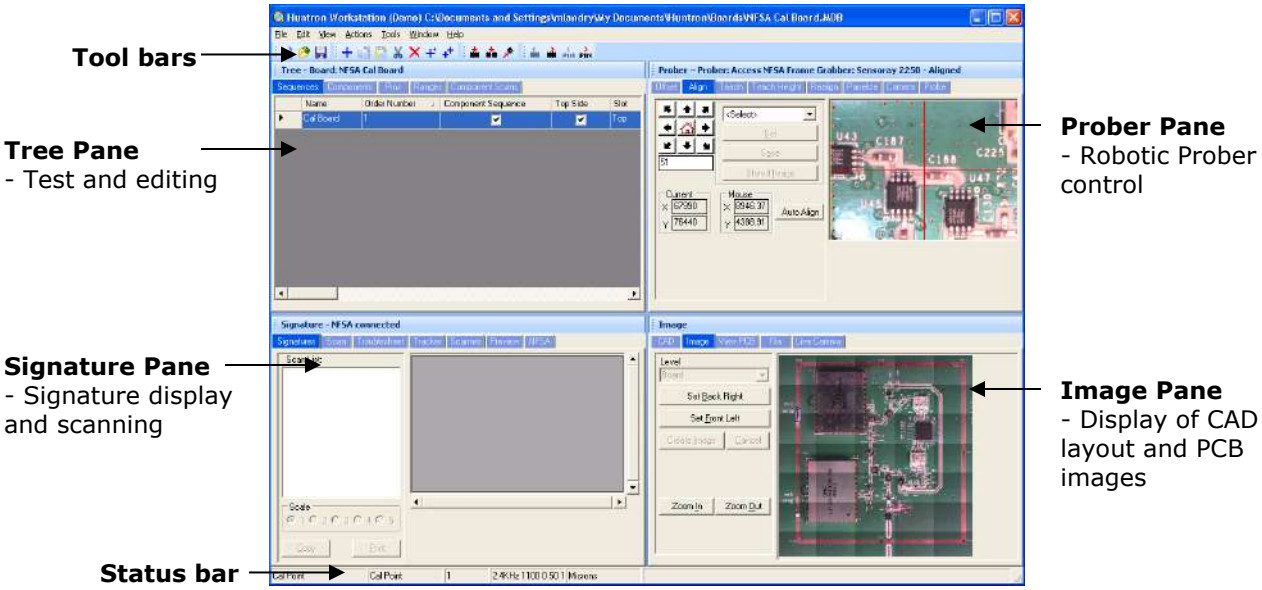


A connection from the oscilloscope ground will need to be made to the UUT.

Huntron Workstation Main Interface

When starting Huntron Workstation for the first time, you will be prompted for an activation code. Type in the code exactly as it is presented on the installation CD.

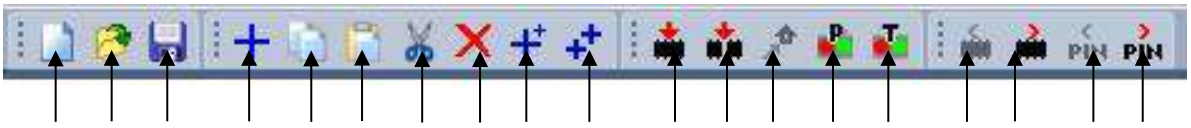
Use upper case letters and note the differences between the letter "O" and the number zero.



The Main Window for Huntron Workstation 4 features a multi-pane layout. The Panes are interactive to a certain degree in that changes in one Pane will affect other Panes. The Panes can be resized, floated independently or "tabbed" to the sides of the window. The Status Bar displays information about items currently selected and status information while processes are running.

The Toolbar

The Toolbar just under the application menus allows for quick access to several program functions. The function of each is broken out in the diagram below.



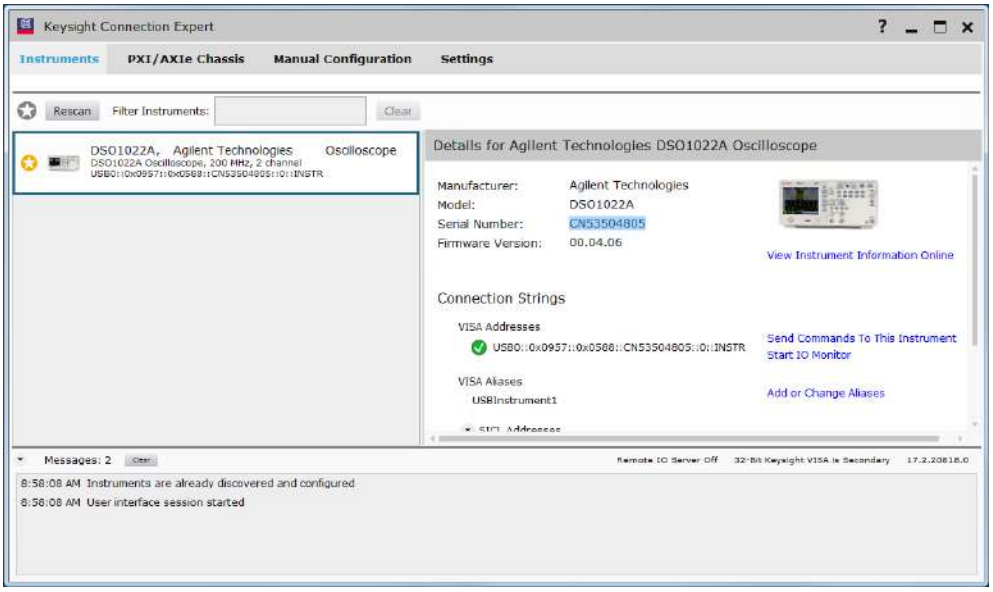
1. Create New Board database
2. Open Board database
3. Save As…
4. Add New (Sequence, Component or Range)
5. Copy (Sequence, Component, Net or Range)
6. Paste (Sequence, Component, Net or Range)
7. Cut (Sequence, Component, Net or Range)
8. Delete (Seq., Comp., Net, Range or Scan)
9. Build New (Sequence, Component)
10. Repeat New (Sequence, Component)
11. Scan Component or Net
12. Scan Sequence
13. Send Prober head to Home position
14. Connect/Disconnect Prober
15. Connect/Disconnect Tracker
16. Previous Component or Net
17. Next Component or Net
18. Previous Pin
19. Next Pin

Hardware Setup

Turn on the Huntron Access and the oscilloscope. Windows will detect New Hardware and install the driver needed to communicate with the oscilloscope.

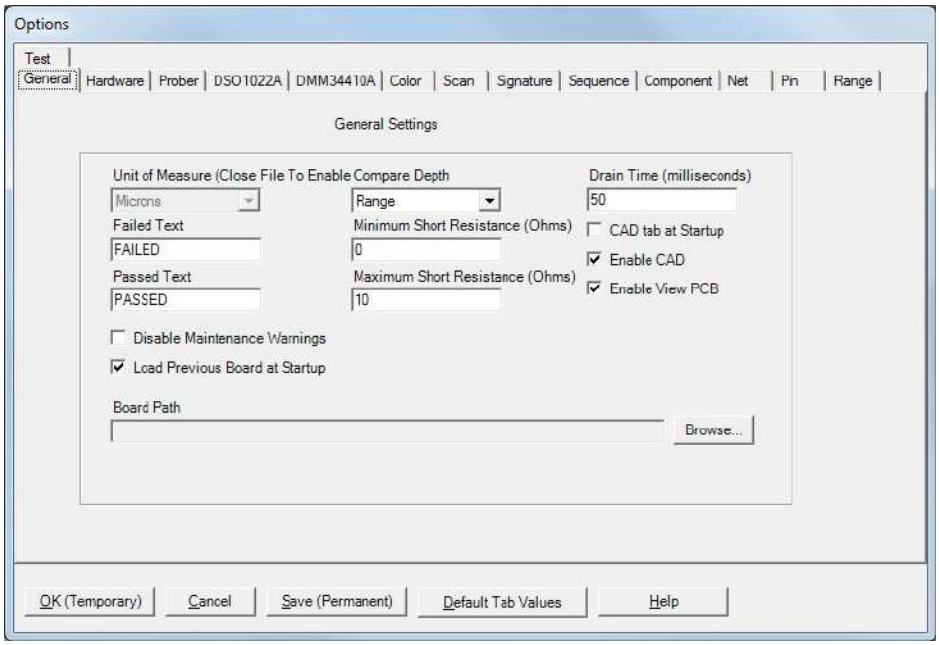
Agilent/Keysight Oscilloscope

Go to the Windows START button, locate and start the **Keysight Connection Expert** application. Locate the connected oscilloscope by selecting your model in the Instruments tab (see image below). Note the **serial number** listed in the Instrument properties.



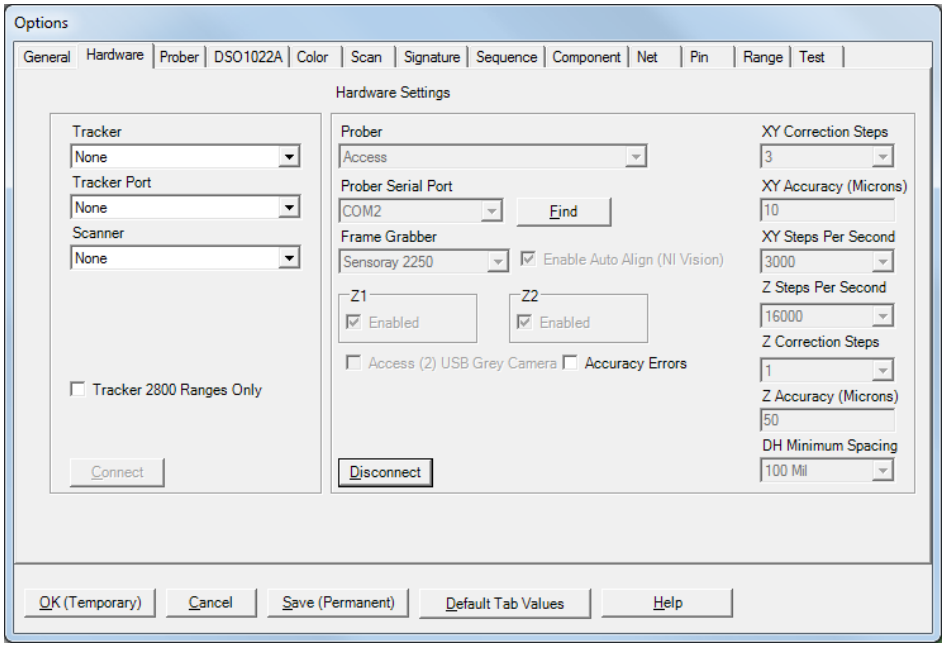
After verifying that the instrument is properly connected, close the Keysight Connection Expert.

When first starting Huntron Workstation, select **Options** from the Tools menu. The Options window is used to setup default settings used when working in Huntron Workstation.

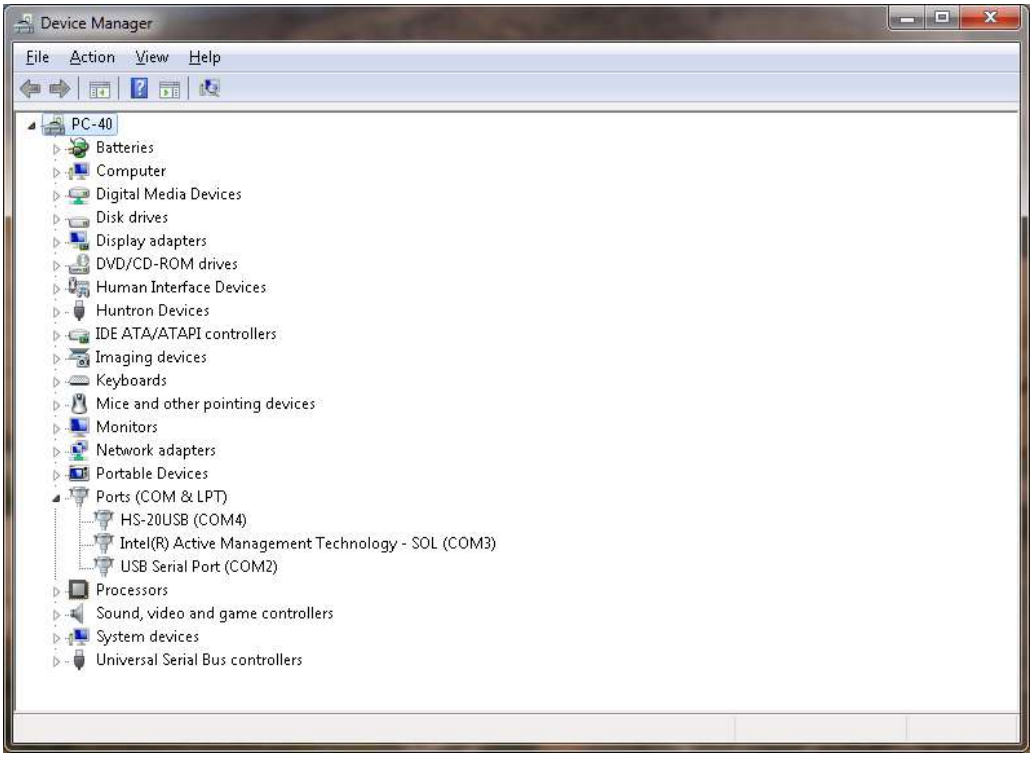


Select the **General** tab and set the desired Unit of Measure (use **Microns** for Huntron Access Probers, **Mils** for any other Prober). Also Select Compare Depth of **Range.**

Select the **Hardware** tab and leave the type of Tracker hardware, the Tracker Port and Scanner options at none.

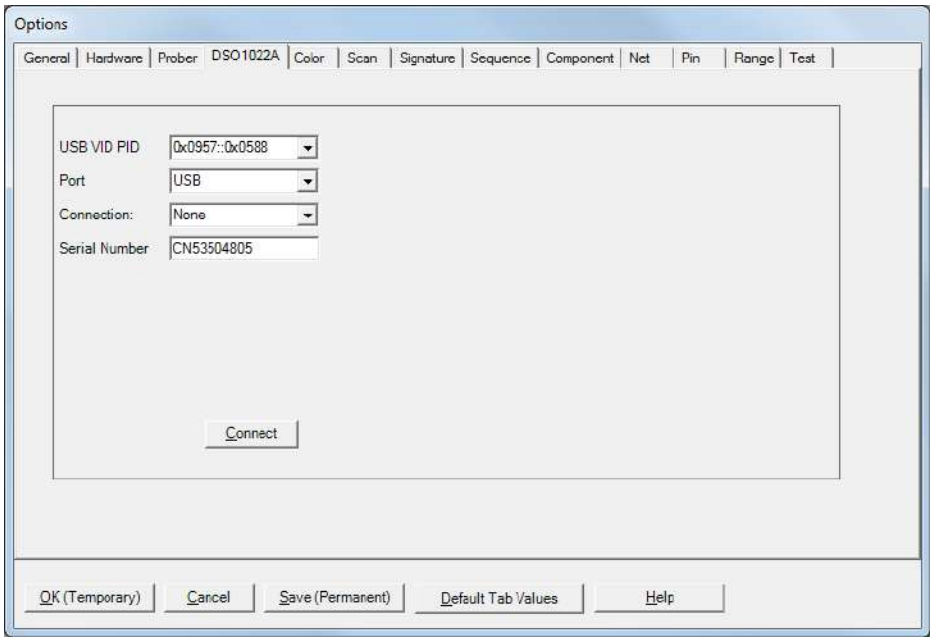


If an Access Prober is being used you will need to select the type of Access Prober, the serial port for the Prober and the Frame Grabber type. Select the COM port the Prober connection is using by clicking the **Find** button which will search available COM ports for the Access USB Prober. Alternatively, go to the Windows **Control Panel**, select **System** and select **Device Manager**. Locate the Ports settings as shown in the image below. Use the COM number shown next to "HS-20USB".

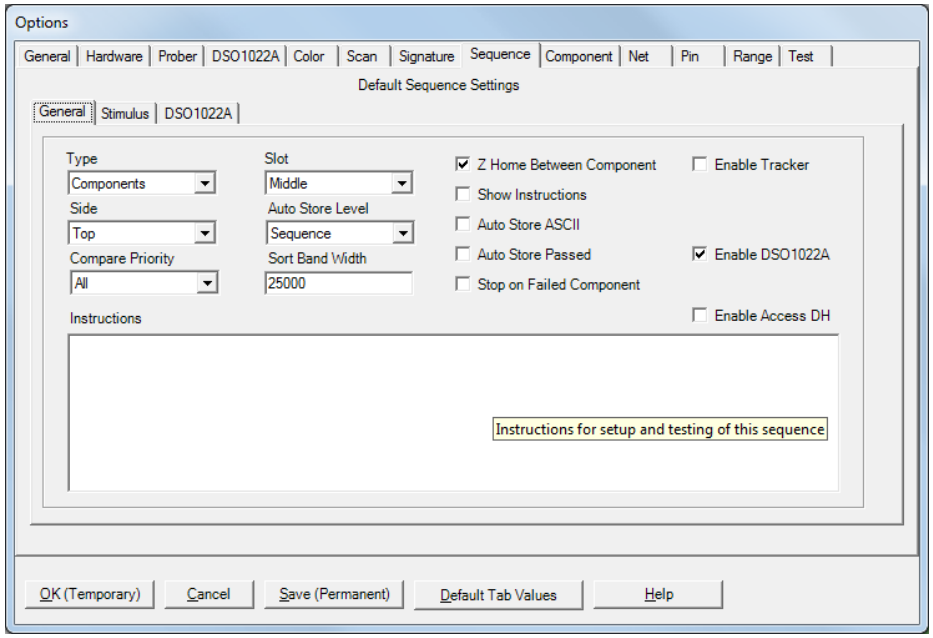


Check the Access (2) USB Grey Camera checkbox if you have an Access 2 Prober where the camera has a grey colored body rather than black.
Click the **Connect** button to initiate communication with the selected hardware.

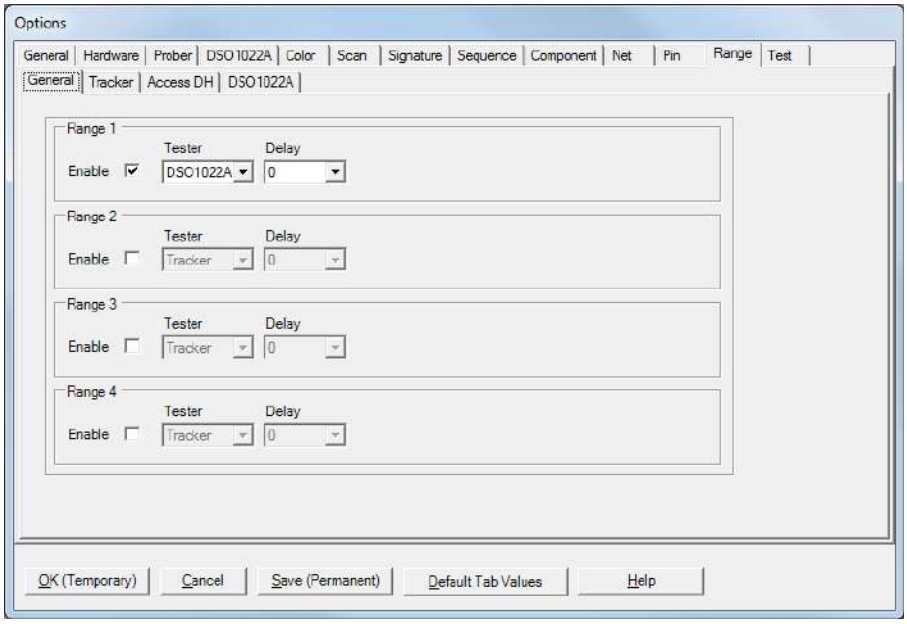
Select the tab that is available for your oscilloscope model (i.e. **DSO1022A**). For most oscilloscopes, set the **USB VID PID** (vendor and product IDs), set the Port for **USB**, Connection to "**None**" and input the serial number of the oscilloscope being connected. Click the **Connect** button to connect with the oscilloscope.



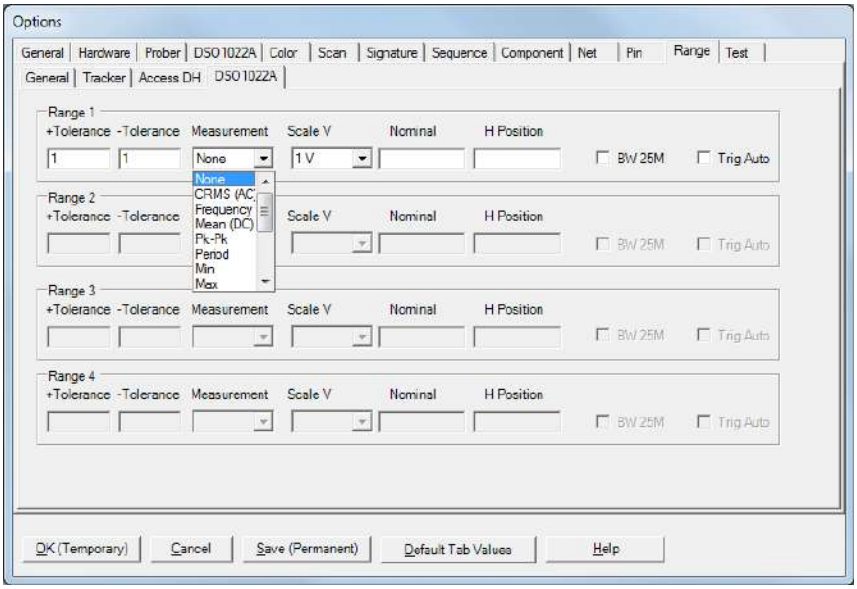
Select the **Sequence tab** and uncheck the Enable Tracker checkbox and check the **Enable (oscilloscope model)** checkbox. Check the **Enable Access DH** checkbox if an Access DH Prober is being used.



Select the **Range tab** and check the boxes to enable only Range 1. Set the Range 1 Tester to your model of oscilloscope and the Range 1 Delay to 0.



Select the **oscilloscope tab** (for your model) on the **Range Tab.** The settings available in this tab may vary depending on the oscilloscope model used.
You can set the plus and minus Tolerance amount, Measurement type, Scale V, Nominal measurement value and H Position. **Tolerance** is controlled by the Comparison setting at the Range level. It can be in measurement units (i.e. volts, etc.) or by percentage. These steps are simply setting defaults for when you add new components to the test. You can change the values in the Tree pane, Ranges tab at any time.



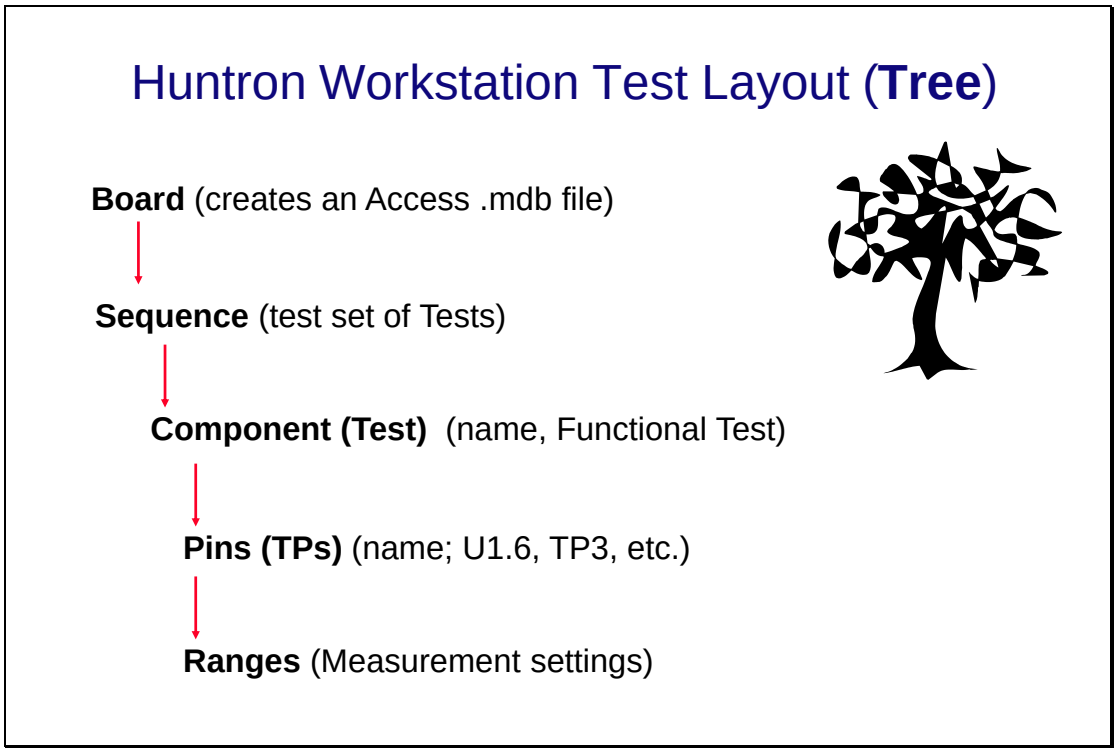
Click the **Save (Permanent)** button to save this configuration as the startup default.

Using Workstation with an Oscilloscope

An oscilloscope based test will typically use the concept of Test Points (TP). The Workstation Software has, for programming purposes, the concept of components and pins. When using an oscilloscope it is best to think of components as tests and pins as Test Points (TP).

Creating a Oscilloscope Board Test

There are a number of steps necessary to creating an oscilloscope board test.



Example of Board Test Information:

Board Name: Sample (creates "Sample.hwdb" file)

Revision: (This field is optional)

Sequence: Voltage Pk-Pk Measurements

Component (Test): Test Points

Pins (TP): Per TP settings for X, Y, Z locations, etc…

Ranges: Per TP measurement settings, tolerance, etc…

The user has total discretion as to the information input into these fields. There is no "right" way as to what information is entered as every user's situation will be different. The above example is just one way the information might be entered.
Entering information into the entry windows is very straight forward. The easiest way to move from field to field is by pressing the **TAB** key. You can also use the mouse to click into a particular field so you can enter information.

Test Building Procedures

- Determine TP Locations on the board and the tests to be performed
- Create Test Database in the Tree Pane
    - Add Board, Sequence, and Components (Tests)

If an Access Prober is being used:

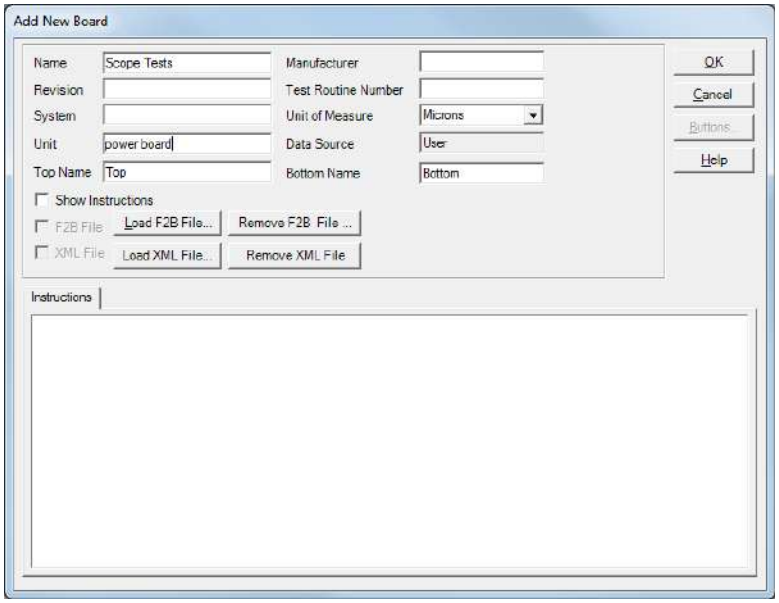
- Perform Camera Offset
- Select Alignment points
- Teach component pin (TP)

Using the test:

- Scan Sequences and Set References

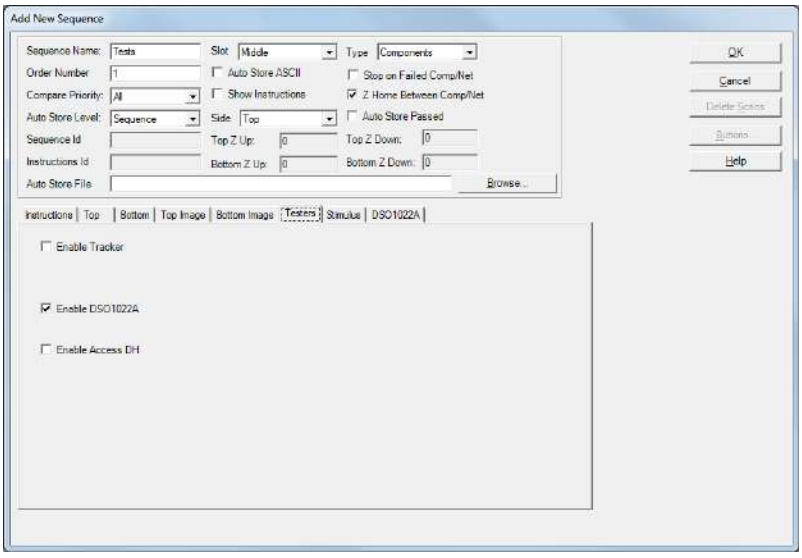
Listed above is a general outline on how you would proceed when preparing to test a board. As with any type of complex test equipment, practice makes perfect so take the time to learn and practice these procedures.

Creating a Board Database



To create a new Board, select **New** from the File menu or use the New button in the toolbar. Input information into the "Add New Board" dialog as needed. The only field that is required is board **Name** *(Example "Sample").* Other fields such as Revision, System, Unit, Manufacturer and Gold Disk Number can be completed with related information if desired.

Adding a New Sequence



To add a new Sequence, select the **Sequence** tab in the Tree pane. Select **Add New Sequence** from the Edit menu or click the Add New button **(+)** in the toolbar.

Details:

Sequence Name Field: Enter Sequence name

Slot field: Select the Prober slot that the board will be in when scanning.

Compare Priority: Selects the signature (or measurement) comparison priority between **Same** (serial number), **All** (serial numbers; this is the recommended setting) or **Min/Max** (Merged min/max signatures)

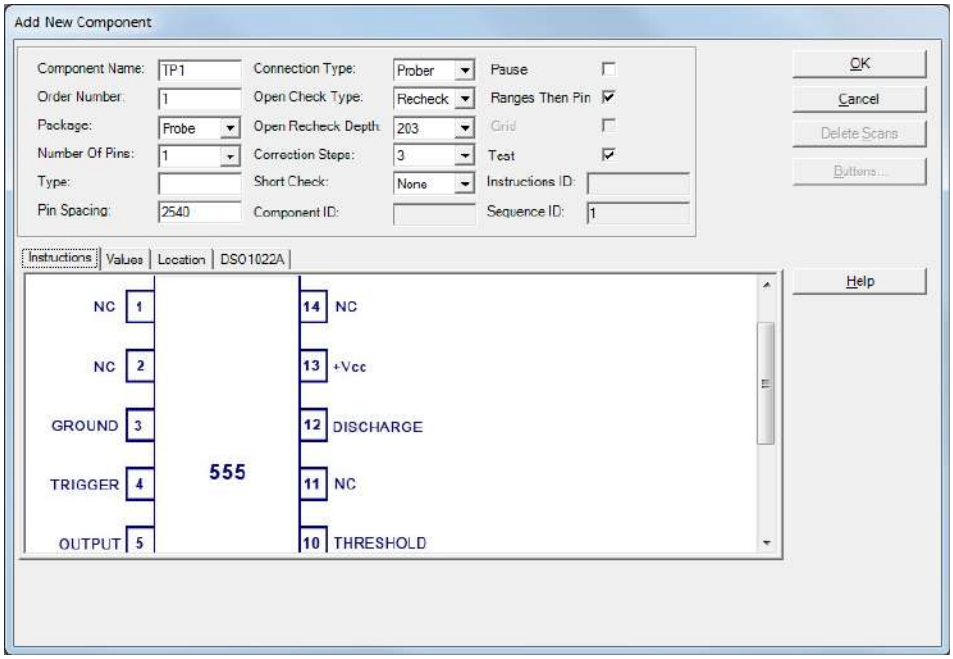
The **Delete Scans** button will erase all scans for the current Sequence.

Instructions field: Select the **Instructions tab** and enter any instructions or system information that you wish to pass on to future users. Note: the **ENTER** key in this field will work as a carriage return.

For oscilloscope tests, select the **Testers tab** and check the **Enable (model number)** checkbox. Uncheck **Enable Tracker** if it is checked.

Other functions within this window can be read by mousing over the item and viewing the Tool Tip or by clicking the **HELP** button.

Adding a New Component (Test)



To add a new Component (Test), select the **Component** tab in the Tree pane. Select **Add New Component** from the Edit menu or click Add New button (**+**) in the toolbar.

Details:

Component Name field: Enter the component name (Test Name) in this field.

Package and **Number of Pins** fields: The Number of Pins should be set to at least the number of TPs in this Test (component). Select more pins than the current number of TPs allows more TPs to be added later. For unused pins (TPs) uncheck the **Scan Pin** check box on the **Pin Tab** after the Component is created.

Type field: This information is optional but you can enter the component type in this field.

Pin Spacing: Not used with oscilloscope tests.

Connection Type: Set to Prober for Access Prober users

Open Check Type: Not used by with oscilloscope tests

Pause checkbox: Sequence tests will pause at this component and display instructions

Test checkbox: Enables or disables testing of the selected component

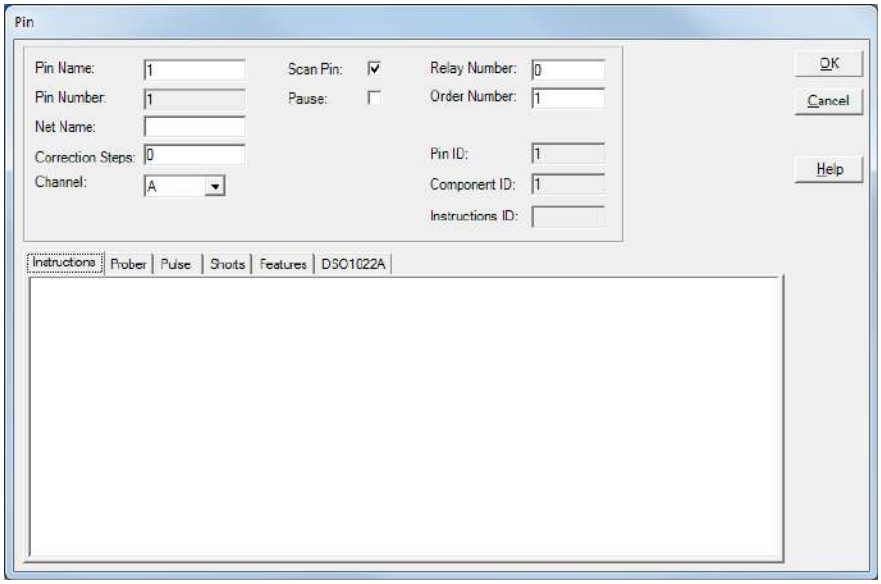
Correction Steps: Sets the maximum amount of correction steps used with the Huntron Access Prober when moving to the first TP of the Test.

Instructions field: Enter any instructions or component information that you wish to pass on to future users.

Other functions within this window can be read about by clicking the **HELP** button.

Pin Edit

Settings for individual pins can be modified by open the Pin Edit window. You can open the Pin Edit window by selecting the Pin tab in the Tree pane, right-click the row header and select Edit Pin.



Name: Type directly into the Name field to change the pin name

Net Name: Type directly into the Name field to change the Net name

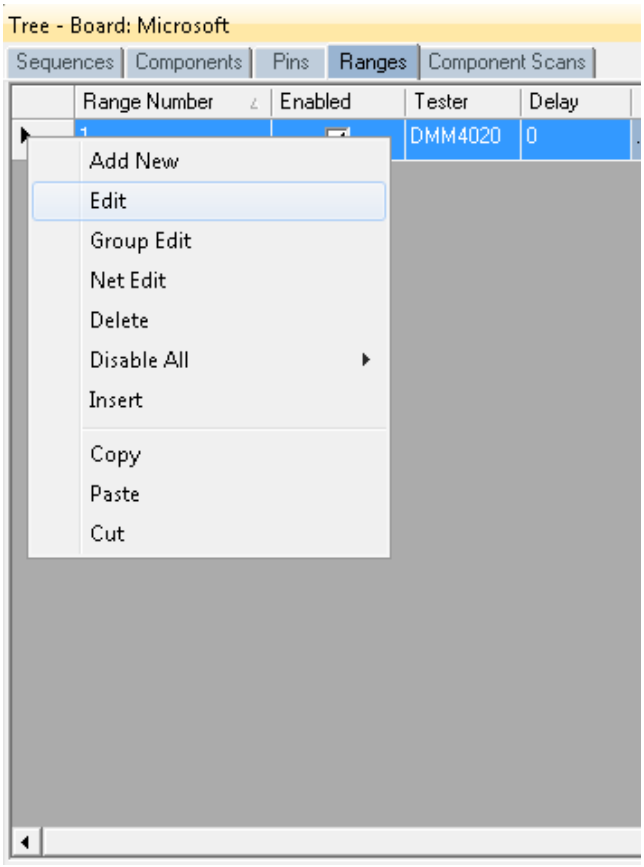
Correction Steps: Modifies the number correction steps used with Access Probers. This may need to be modified for very small pitch devices.

Scan Pin: Enables or disables scanning of the selected pin

Pause: Prober will pause and display Pin Instructions before scanning

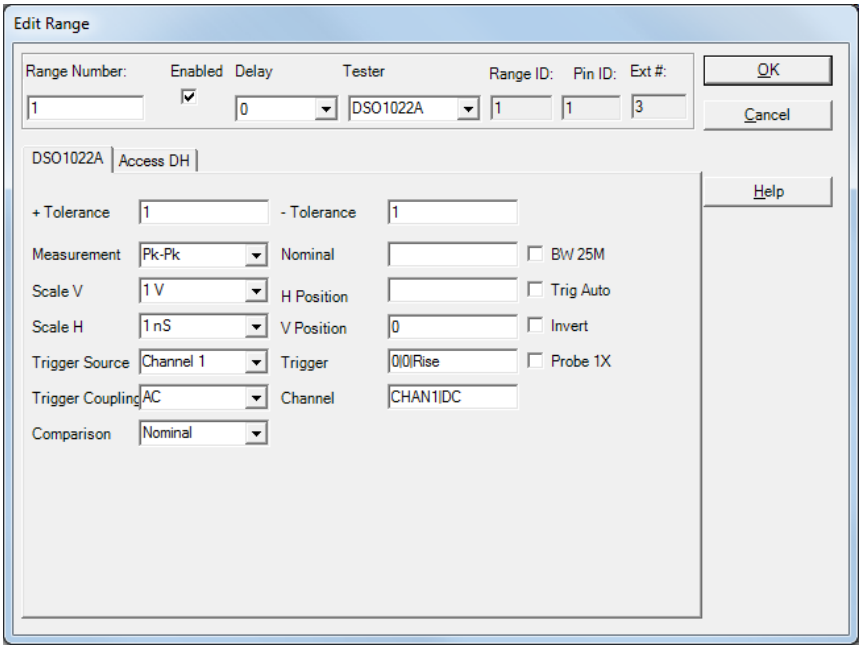
Modifying a Range (Oscilloscope Measurement Setup)

When a new component (Test) is added to a Sequence default range(s) are added to each pin (TP). The defaults are set up in the **Tools/Options/Range/(Oscilloscope model)** tab. These settings need to be modified based on signal being measured at each TP. The goal is to obtain a good measurement at each TP. To stop after a failure of a particular measure it is best to use multiple Components (Tests) at the same XY coordinate each. The test only stops after a component (test) is completed.



Select the Tree/Range tab to display the ranges for the selected component pin. RIGHT click on the row header (far left grey area of the row) to display a quick menu (shown left)

To edit a Range, select the Range tab in the Tree pane. Select **Edit Range** from the **Edit** menu or the quick menu shown above. The Edit Range dialog will be displayed. The **Tester drop menu** shows the instrument used.



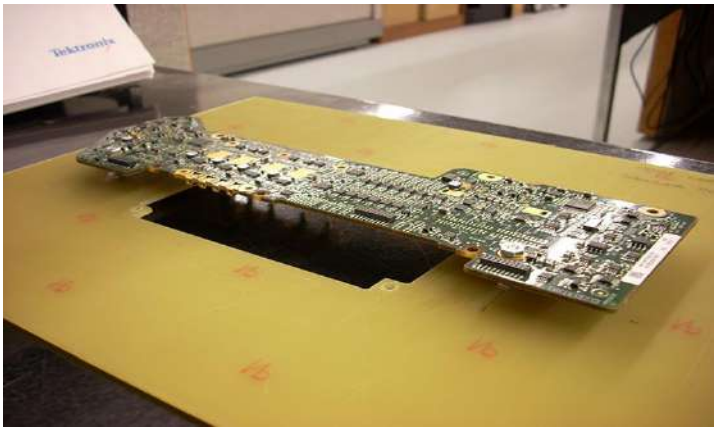
Edit Range Details (settings may be different based on oscilloscope model):
Note: All of the settings can be modified directly in the Range grid.
+Tolerance: The amount above the Reference or Nominal Reading which determines the High Pass Limit. This is used during comparison to the value listed in the Nominal field. This amount can be a value or percentage depending on the Comparison setting.
-Tolerance: The amount below the Reference or Nominal Reading which determines the Low Pass Limit. This is used during comparison to the value listed in the Nominal field. This amount can be a value or percentage depending on the Comparison setting.
Measurement: Sets type of oscilloscope Measurement
Scale V: Sets the scope Volts/Division
Scale H: Sets the scope timebase
Trigger Source: Sets Trigger source i.e. Chan. 1, Chan. 2, External, etc.
Trigger Coupling: Sets Trigger coupling type i.e. AC, DC, etc.
Comparison: Sets the comparison mode. **Nominal** compares the measurement to the entered Nominal value. **Reference** compares the measurement to the stored reference measurements. **Nominal %** uses the Tolerance setting as a percentage of the Nominal value. **Reference %** uses the Tolerance setting as a percentage of the Reference value.
H and V Position: Enter the Vertical Position in divisions | Horizontal Position in seconds; leave blank to defaults
Trigger: Enter the Trigger Level in volts | Trigger Holdoff in seconds | Trigger Slope (Rise, Fall, Both); Example: **1|0|Rise**
Channel: Enter the channel using channel name | coupling (DC, AC or GND). Example: CHAN1|DC
Nominal: Enter the nominal value for the measurement. This is the value used when test **Comparison** is set to **Nominal.**
BW 25M: Check for 25MHz bandwidth, uncheck for maximum bandwidth
Trig Auto: Check to Trigger Auto, uncheck to Trigger Normal
Invert: Check to Invert the signal, uncheck for normal signal
Probe 1X: Check for 1X Probe, uncheck for 10X Probe

Mounting a PCB in an Access Prober

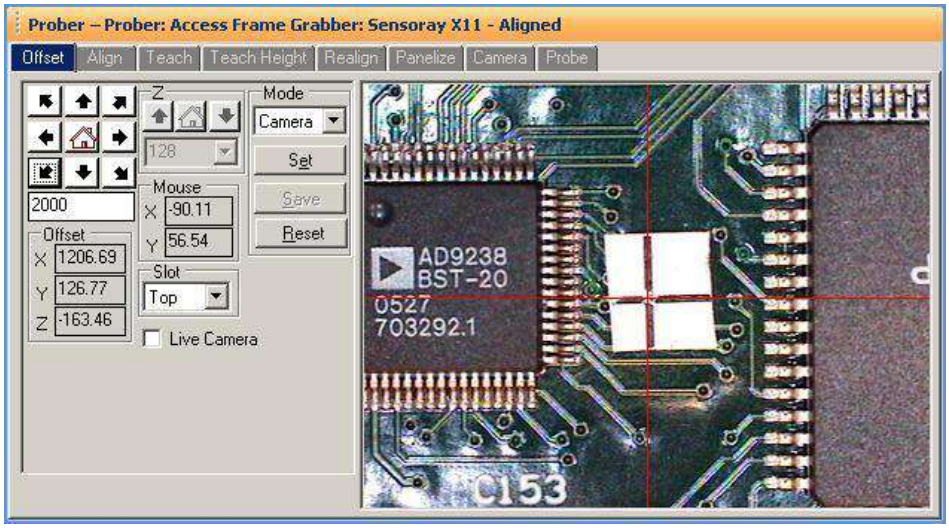
Before software functions relating to an Access Prober can be performed, the PCB under test must be mounted into the Prober test area. Use the mounting accessories supplied with the Prober to securely mount your PCB by its edges.



Refer to the Access Prober Users manual for more information on mounting a PCB. In some cases, a PCB holder is the best solution especially when the PCB cannot be easily mounted using the standard accessories. PCB holders are typically simple to construct and the only prerequisites are that the holder fit into the Prober and that it holds the PCB precisely at one of the designated Prober "Slots". The Slots are referred to by the position in the Prober as TOP, MIDDLE, BOTTOM and BASE (Access 2 and Access DH only) and are set in the Sequence Edit window (see section "**Adding a New Sequence**").



Prober Setup – Camera Offset part 1



Performing the Camera Offset procedure is essential for the Prober to access test points accurately.

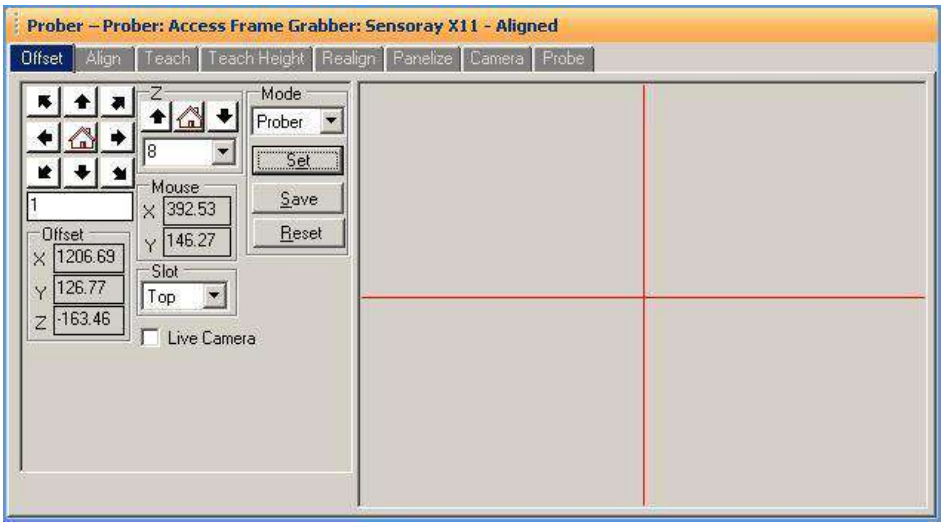
Select the **Offset** tab in the Prober pane.

Navigate the Camera to a target on the PCB. A very small target that is still visible by eye works best. Use the arrow buttons to move the camera the distance set in the travel distance field (directly below the arrow buttons). You can also click directly in the camera image to move the camera crosshairs to the clicked location.

When the target is selected, press **Set**.

Select **Prober** in the Mode drop menu to move to the next step.

Prober Setup – Camera Offset part 2



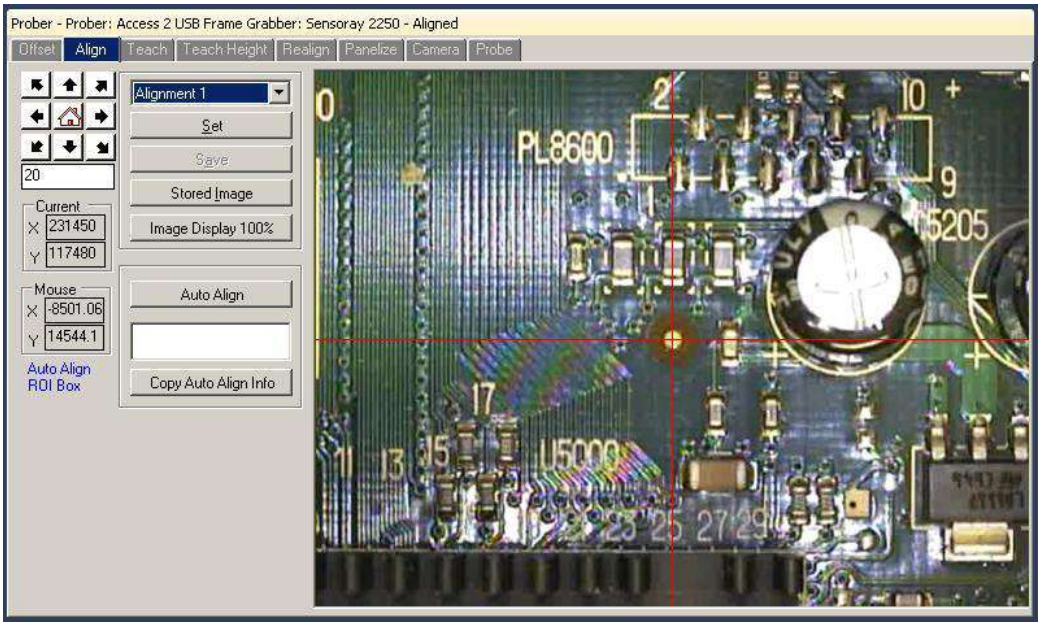
Lower the probe tip down using the **Z** buttons until the probe just makes contact with the PCB.

Visually examine the position of the probe tip; it should be located on the target selected in part 1 of Camera Offset, if not move on to the next step. Use a magnifying glass if necessary.

Use the **XY** buttons to navigate the probe to the target selected in Part 1. The probe will lift when moving in the X or Y direction.

When the probe is placed precisely on the target so it is just touching, press the **Set** and **Save** buttons to calculate the Offset values.

Prober setup – Alignment part 1



Alignment points that are positioned in opposite corners of the board work best. If the test was created using CAD data, the alignment points are pre-selected and you should proceed as described below however be aware that the **Save** button will not become enabled. Refer to the CAD Import tutorial for details on Alignment when creating a test from CAD data.

Select the **Align** tab in the Prober pane.

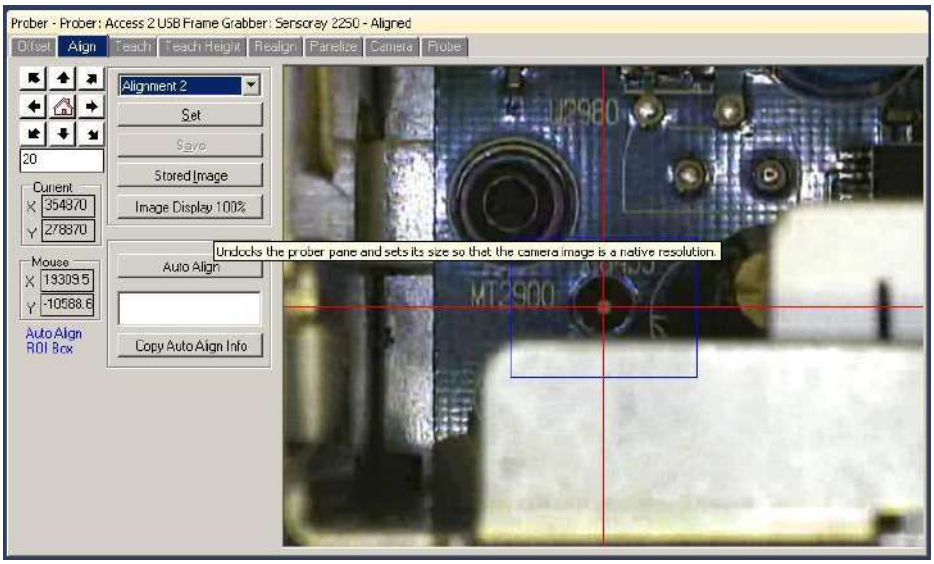
Select **Alignment 1** from the drop menu.

Select and navigate the camera to the first alignment point. Fiducial marks, small vias or traces with 90 degree bends work well for alignment points.

When the camera is set on the first alignment point press **Set.**

The alignment point drop menu will select **Alignment 2** automatically.

Prober setup – Alignment part 2



Make sure **Alignment 2** is selected in the Alignment point drop field.

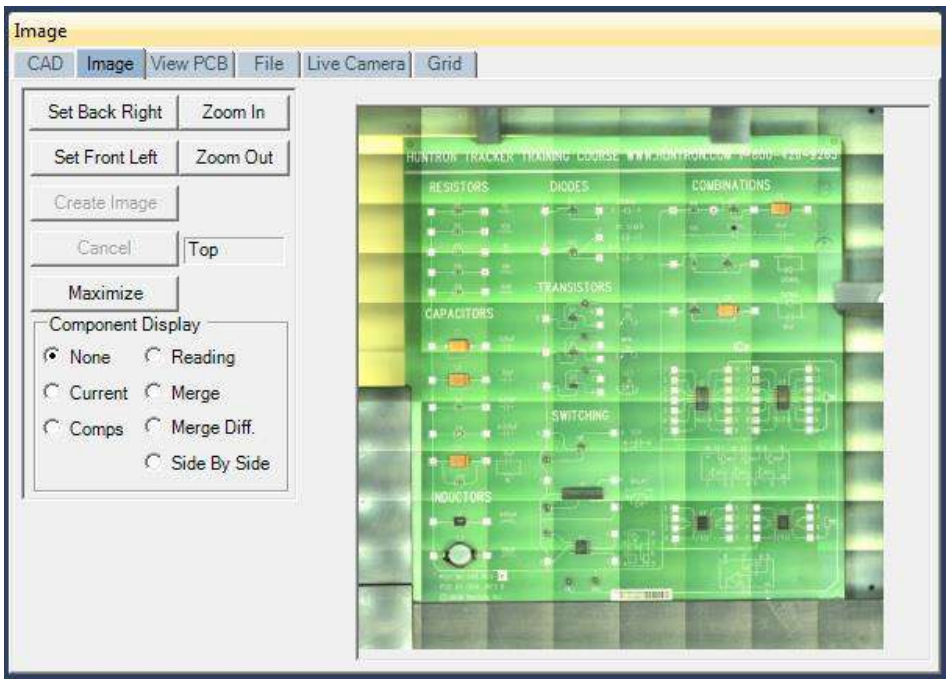
Select and navigate the camera to the second alignment point.

When the camera is set on the second alignment, press the **Set** button.

The **Auto Align** feature will use image recognition to find and automatically center on the alignment point. Auto Align will only work after Align is performed manually the first time.

Press the **Save** button to store the alignment points and the alignment point images.

Creating a Board Image



For Prober users, the built-in color camera can be used to create a "mosaic" of the PCB by capturing images of the board and stitching them together. This is accomplished in the Image/Image Pane. The board must first be aligned before creating the image. Once aligned, you can use any camera view in the Prober pane to set the borders for image capture.

Move the camera to the back right corner of the board.

Press the **Set Back Right** button in the Image/Image Pane.

Move the camera to the front left corner of the board.

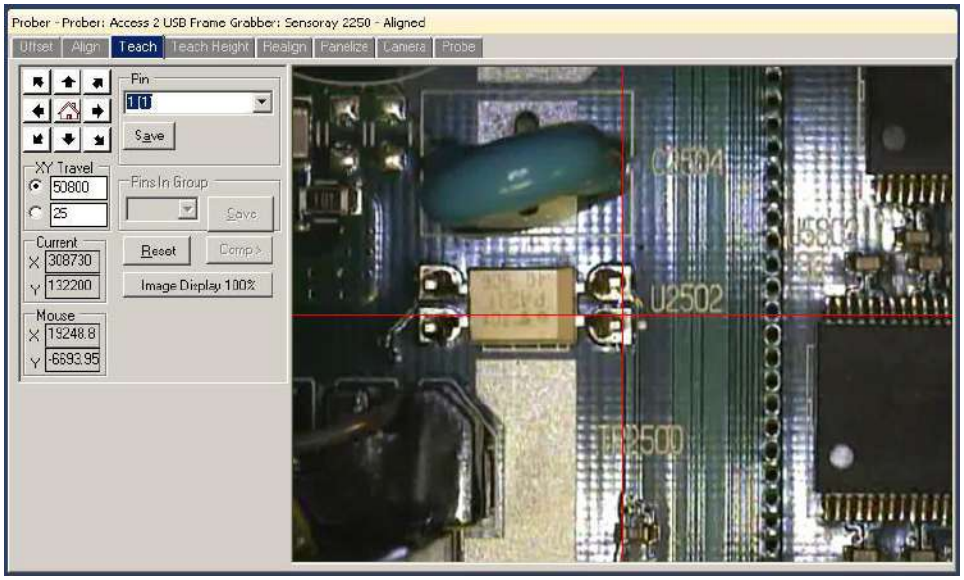
Press the **Set Front Left** button in the Image/Image Pane.

Press the **Create Image** button to create the board image.

Larger boards will require more time to create the board image. Once the board image is created, you can click a point on the board image and the Prober will move the camera to that point.

Right-click on the image to display a menu to **Delete**, **Copy** or **Save As** the board image.

Prober setup – TP Teach – XY Setup



Select the **Teach** tab in the Prober pane.

Select the component to teach the **Tree/Component** pane.

Select **1** in the **Pin** drop menu.

Navigate the camera to the location of the first TP (Pin 1) (see image above).

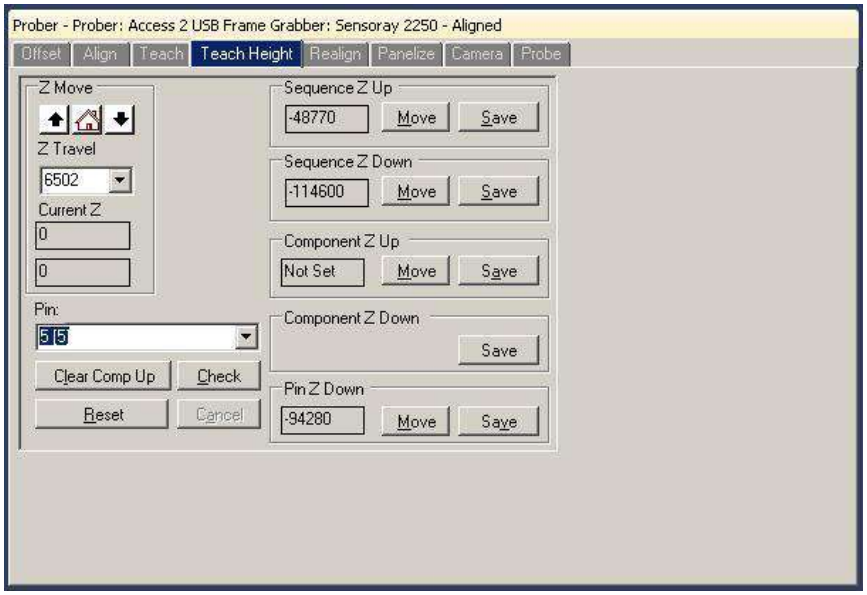
When the camera is set on the first TP, press the **Save** button. The **Pin** drop menu will increment to the next pin.

Navigate the camera to the Location of the 2nd TP.

When the camera is set on the 2nd TP, press the **Save** button.

Continue setting the XY locations of all the TPs.

Prober setup – TP Component Teach Height (Z)



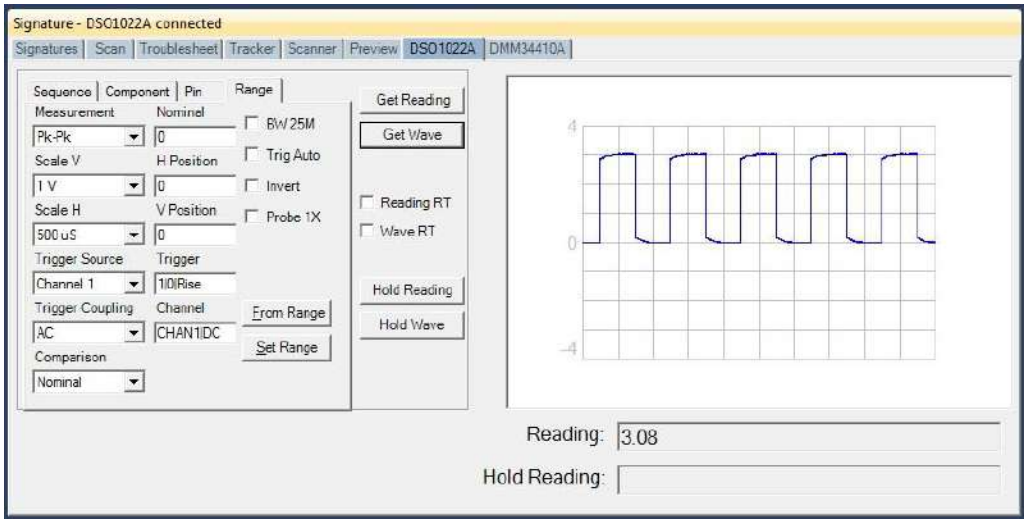
Select the **Teach Height** tab in the Prober Pane.
Select a Pin number in the **Pin Z Down** drop menu. This will position the probe tip at the selected TP.
Use the **Z Move** buttons to lower the Z axis probe to the desired Sequence Up level. Make sure this level is high enough to clear tall components and chassis walls. This is the position the probe will lift to while probing a component.
Press the **Save** button in the **Sequence Z Up** section to set the Sequence Up position.

If desired, you can set an additional Up level for the component. This is the level that the probe will lift to when moving from one pin to the next. Use the **Z Move** buttons to move the probe down to the desired Component Up level. Make sure this level is high enough to clear the component package. Press the **Save** button in the Component Z Up section to set the Component Z Up position.

The next step is to set the probe down position.
Use the **Z Move** buttons to lower the probe tip to make contact with the component pin or pad. You may want to lower the probe in small increments once contact is made to slightly compress the spring probe and ensure good contact. Do this by changing the Z Travel distance setting. The signature of the contact point is displayed on the **Preview** tab of the Signature pane. Connect a common lead to the PCB if you see an open signature.

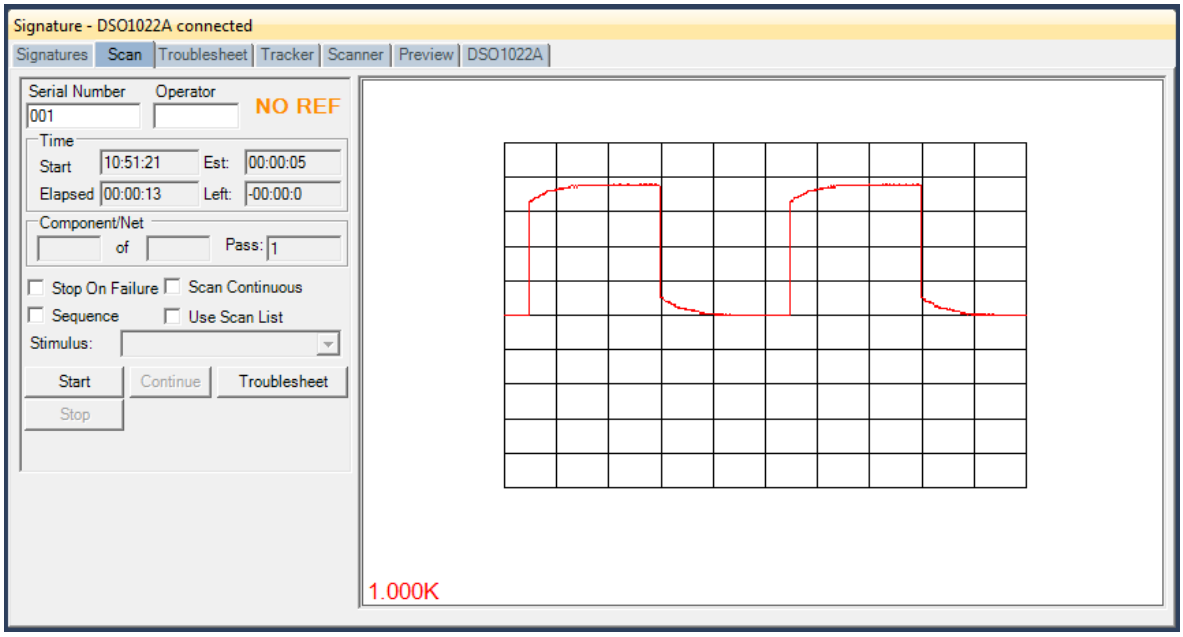
Press the **Save** button in the **Sequence Z Down** section to set the Sequence Down level. This is the level the Prober will use when probing the component pins.
If needed, you can also set **Component Z Down** and **Pin Z Down** levels for individual components and pins. If the TPs of the component have varied heights, then use the **Pin Z Down** settings to modify the height position for each individual pin. Select the pin in the **Pin** drop menu, position the probe to the desired height and click the **Save** button in the **Pin Z Down** section.

Manual Modes



Manual measurements can be obtained using the **(Oscilloscope model)** tab in the **Signature** pane. This pane may be useful when configuring test parameters for a Test Point (TP). Setup the Measurement parameters and click **Get Reading** to capture and display a numeric measurement. Click **Get Wave** to capture and display the oscilloscope waveform. Click **Hold Reading** or **Hold Wave** to keep the reading or waveform displayed while capturing and displaying a second measurement.

Scanning a Sequence or Component



Select the Sequence or Component to be scanned in the Tree Pane.

Select the **Scan** tab in the Signature Pane (shown above).

Input a **Serial Number** into the Serial Number field; this will be the name of the scan.

The **Stop On Failure** check box will cause the scanning to stop if a component comparison fails. This is most useful when scanning known good boards to add as references.

The **Scan Continuous** check box will cause the scan to go into a loop mode where it will scan without stopping until a signature comparison fails.

Enable the **Sequence** checkbox to scan all of the components in the Sequence.

The **Use Scan List** check box will add the ability to use a Scan List when the Scan Sequence dialog appears (see Creating a Scan List section).

Press the **Start** button to execute the scan.

The test instructions prompt will be displayed. Clicking OK will start the scan for probers.

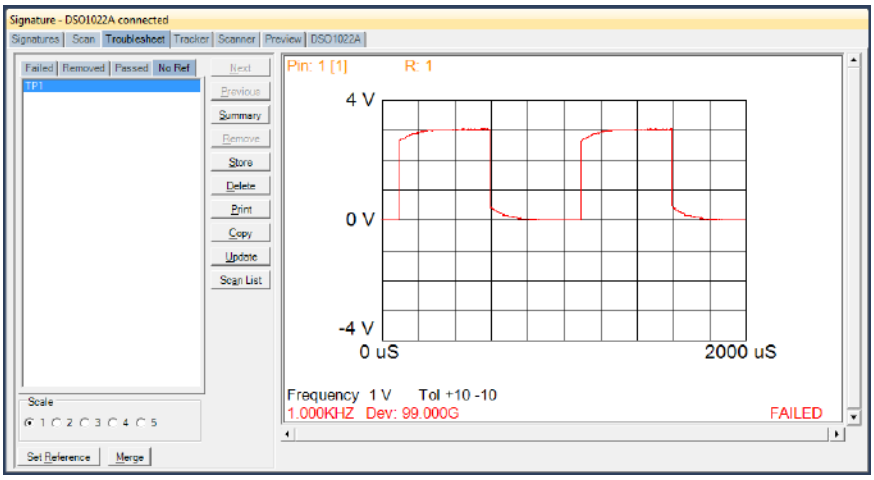
When a scan is complete, the results **NO REF** (NO Reference), **PASSED** or **FAILED** will be displayed (the text of the results message can be changed in the **Tools/Options/General** settings). The **NO REF** message will be displayed on the first scan of any test if the Comparison type is set for Reference or Reference % since there are no scans stored as Reference. If Comparison is set for Nominal or Nominal %

then the result will indicate **PASSED** or **FAILED** depending on how it compares to the Nominal value range setting.

Clicking the **Troublesheet** button will display the signature differences in the Troublesheet tab of the Signature pane.

Viewing Signatures – Troublesheet

To view the Troublesheet, press the **Troublesheet** button in the Scan Results window.



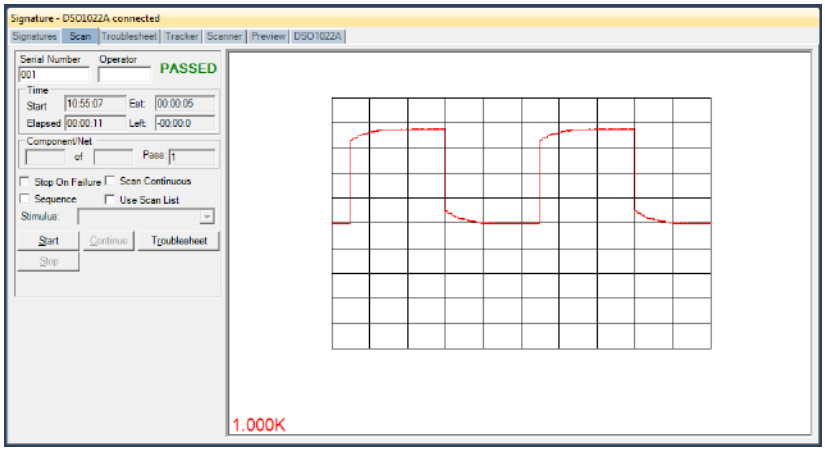
The current measurement, the measurement type, nominal value, tolerances, deviation and test result will be displayed in the signature box.

If this is the first scan and you are using Reference Comparison setting then click the **Set Reference** button if this is a known good scan.

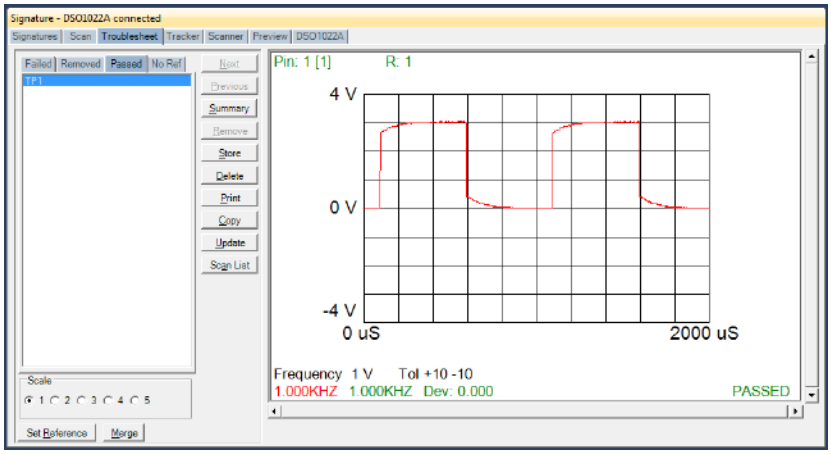
When a scan is executed signatures are compared against the Nominal or Reference set (or sets) if it exists. To add the signatures in the Troublesheet as a Reference, press the **Set Reference** button (do this only with known good signatures).

Signatures of the compared (Reference) signatures are displayed in contrasting colors. By default, the Reference signature is green and the failed signature is red. The **Deviation (Dev)** shown in the signature box is the amount that the measured value deviates from the Nominal or Reference value plus the tolerances set for the range.

If **Set Reference** is used, run the Scan again. A PASSED result should be indicated.

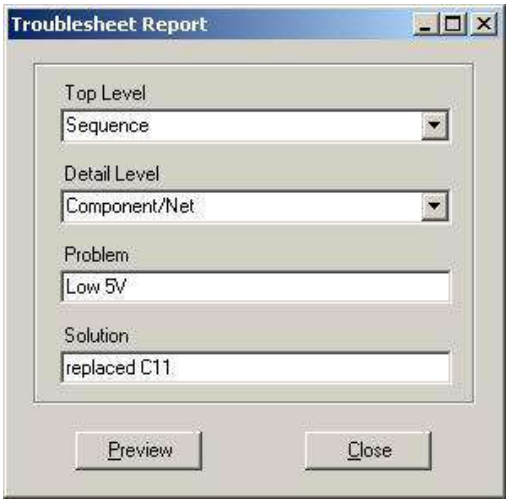


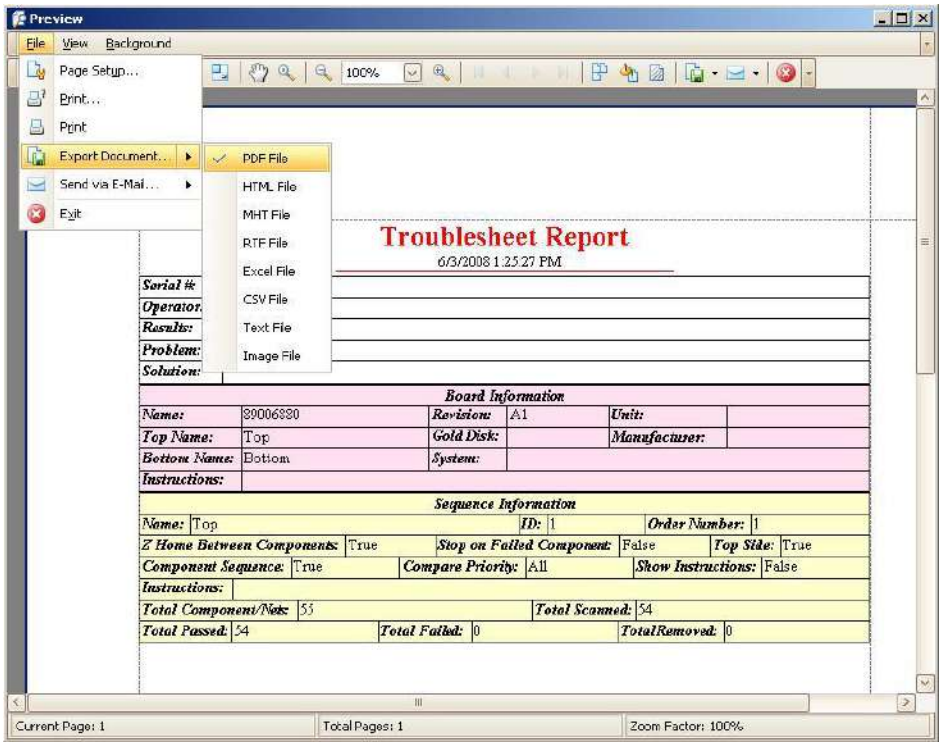
Click the Troubleshoot button to view the results. The Reference and Passed measurements will be displayed as well as the Deviation (**Dev.**) between the two.



Viewing Signatures Troublesheet Report

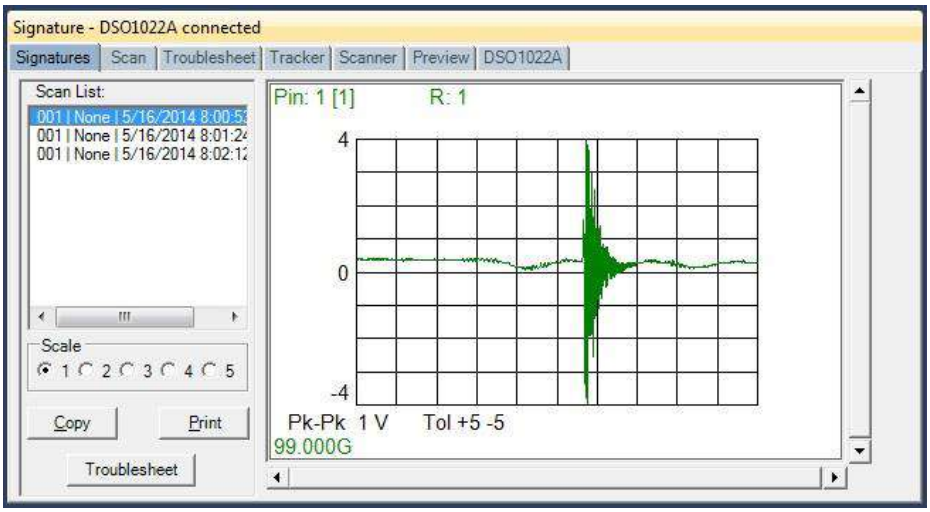
To view and print the Troublesheet report, press the **Print** button in the Troublesheet window. You can select from several options including a Sequence or Component Level report, the level of detail and add problem and solution comments to the report. Click **Preview** to view the report on-screen.





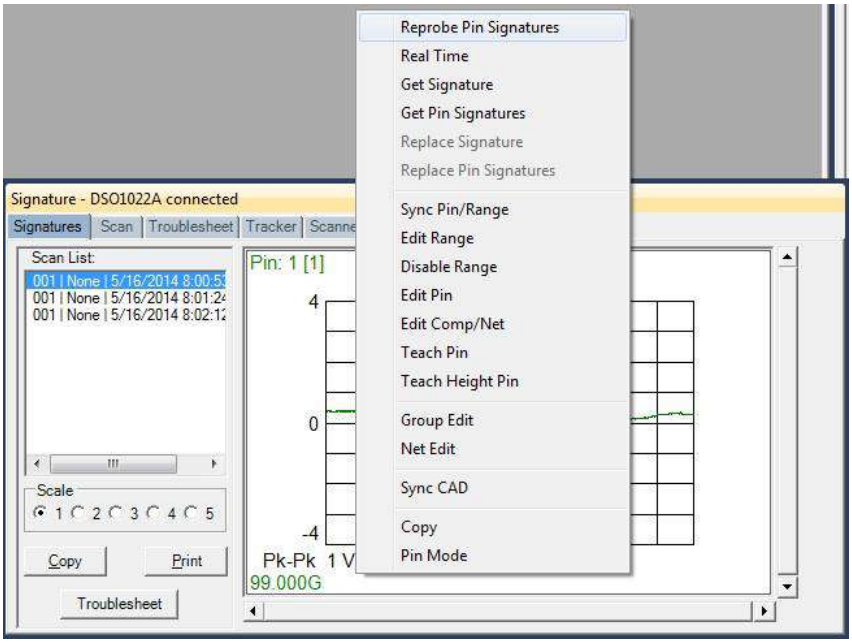
The report can be printed or exported in several different formats including HTML and PDF by selecting **File/Export Document** in the Preview window.

Viewing Signatures in the Signatures Pane



To view scanned signatures, select the **Signatures** tab in the Signature Pane. Signatures marked with an asterisk * indicate that this scan is set as a Reference. The Scan name, date and time will be displayed on the left side of the window and signatures on the right. The **Scale** buttons will change the number of columns displayed. Scaling the window will increase the number of signatures displayed. The **Copy** button will copy the signatures to the Windows clipboard so they can be pasted into another program (in a metafile format). The **Print** button will print the signatures in one of several different user selected formats.

Viewing Signatures – Right Click



Additional options when viewing signatures can be displayed by right clicking in the signature area.

Reprobe Pin Signatures: Gets Signature and prompts to replace the previous one

Real Time: Updates measurement in real time

Get Signature: Capture signature

Replace Signature: Replaces the stored signature with the current signature displayed for the selected pin, range and scan (shown in the Scan List to the left)

Replace Pin Signatures: Replaces the stored signatures with the current signatures displayed for the selected pin, all ranges and scan – works only in Range Mode

Sync Pin/Range: Selects pin and range in the Tree (see default in Tools/Options)

Edit Range: Selects the pin and range in the Tree and opens the Edit Range dialog

Disable Range: Disables the selected range for the selected Pin

Edit Pin: Selects the Pin the in the Tree and opens the Edit Pin dialog

Teach Pin: Selects the pin the Prober Teach pane

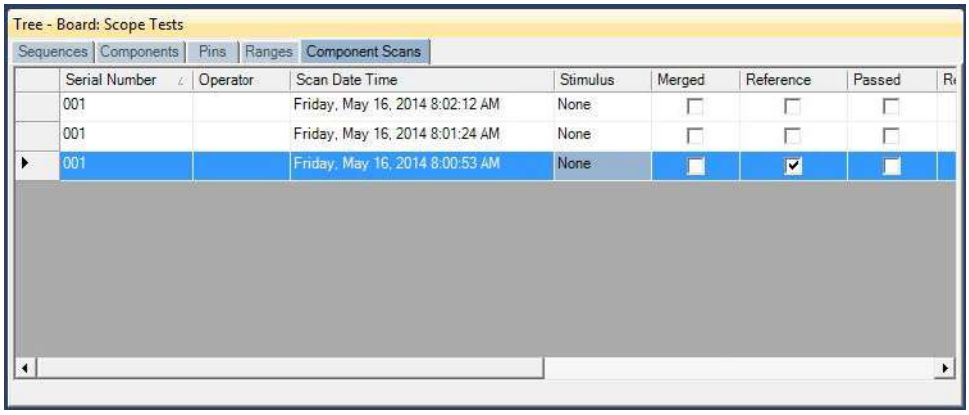
Teach Height Pin: Selects the pin the Prober Teach Height pane

Group Edit: Opens Group Edit dialog for editing of component pin parameters as a group

Net Edit: Opens Net Edit dialog for editing of Net pin parameters as a group

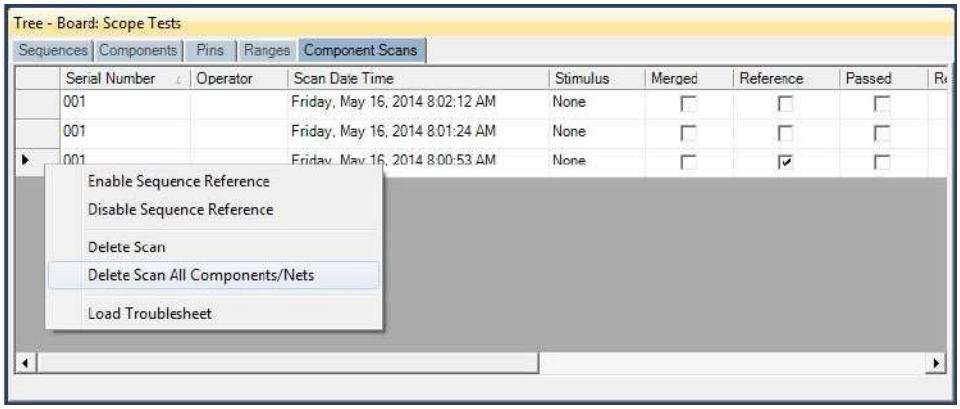
Sync CAD: Will link the selected signature to the CAD image displayed in the Image/CAD pane. **Sync CAD** will be disabled if there is no linked CAD data.
Range/Pin Mode: Displays signature with Range priority or Pin priority; see default in Tools/Options).

Component Scans Information



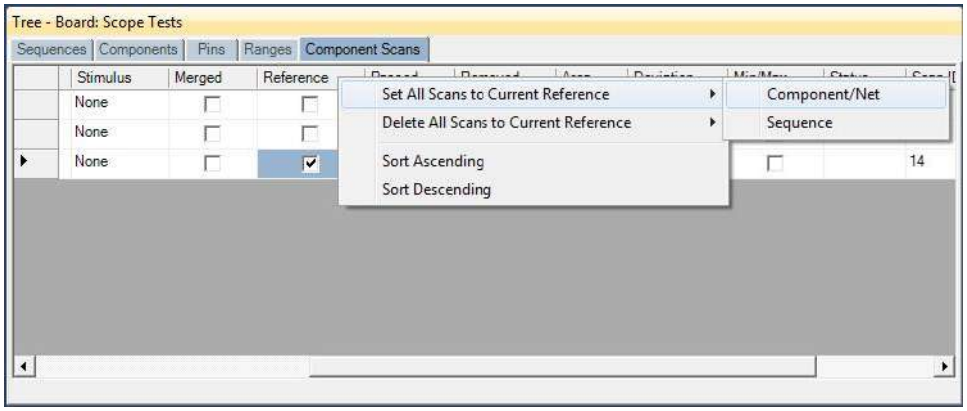
Selecting the **Component Scans** tab in the Tree Pane will display all of the scans performed on the selected component in a table format. The Scan information displayed includes the serial number, operator, date/time, Merge setting (checked if this scan is included in the Merge set), Reference setting (checked indicates it is set as a comparison Reference), Pass status (will be checked if components passed comparison), Removed status (will be checked if component was removed from Troublesheet), Area number (highest of scanned pins), Deviation, Min/Max status and other settings (see Help for more information).

Component Scans – Right Clicking (Auxiliary menus)



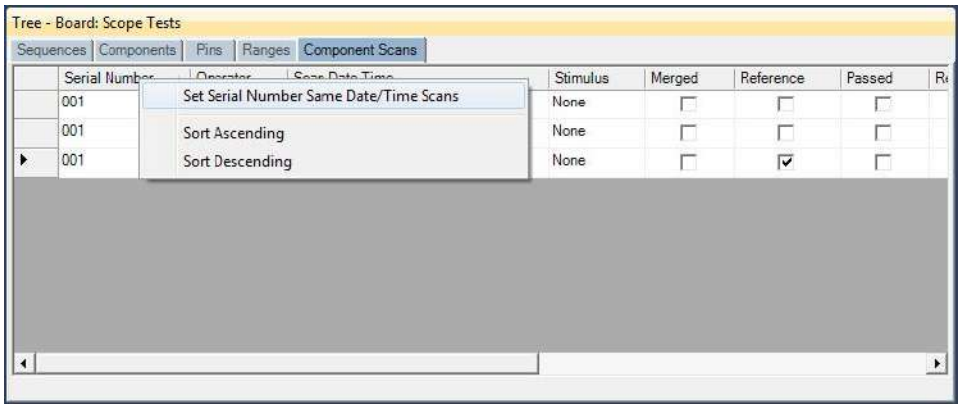
Right clicking on the Row bar of the **Component Scans** pane displays an additional menu for performing specific tasks such as deleting a scan for the selected components or all components. **Enable Sequence Reference/Disable Sequence Reference** will mark or unmark the selected scan as a reference (comparison) scan for all components in the Sequence. **Delete Scan** will erase the selected scan for the current component only. **Delete Scan All Components/Nets** will erase the selected scan for all components or nets in the Sequence. **Load Troublesheet** will display the selected scan in the Signature pane/Troublesheet tab.

Component Scans – Right Clicking (Auxiliary menus)



Right clicking on the Column bar of the Component Scans pane displays an additional menu for performing specific tasks such as sorting or globally setting parameters. The menu will vary depending on the column selected. In the image above, the Reference column header was right-clicked.

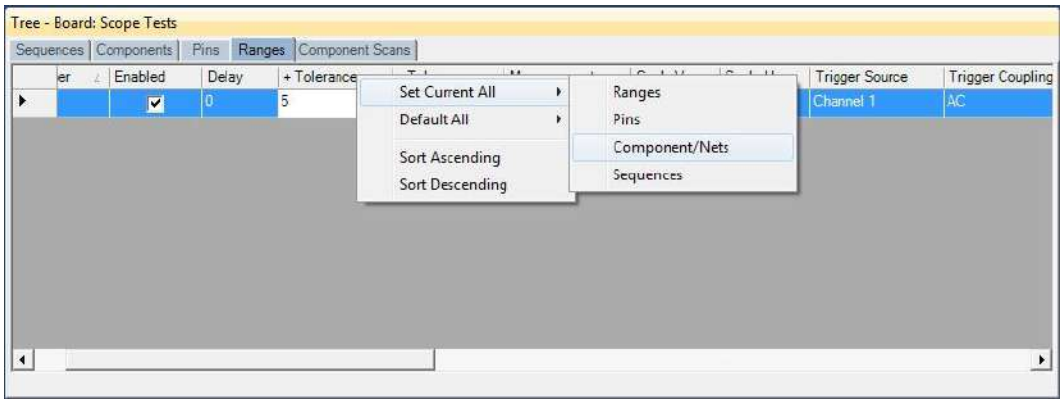
Component Scans – Right Clicking (Auxiliary menus)



Right clicking on the Column bar of the Component Scans pane displays an additional menu for performing specific tasks such as sorting or globally updating items. In the image above, the Serial Number column header was right-clicked.

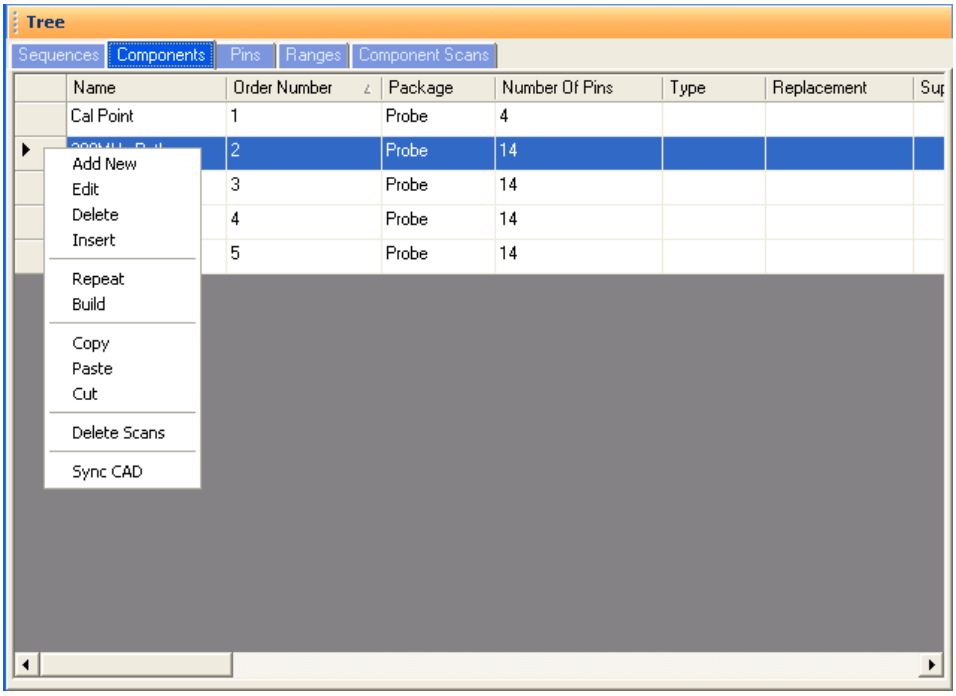
Range Tab – Right-click Menu

One very useful feature making quick global changes to range parameters is the menu available by right-clicking the column header (the title area directly above the column) in the Range tab.

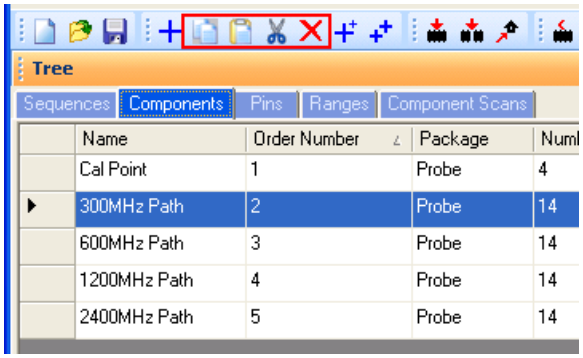


After making a parameter change for the range, **right-click** the column header for that parameter as shown in the image above to display the menu. The most useful selection is **Set Current All/Pins** where all of the pins of the component will be modified to the same setting as the selected Pin. This menu is available for most Range parameters. Remember to make the parameter change first then use **Set Current All/Pins**.

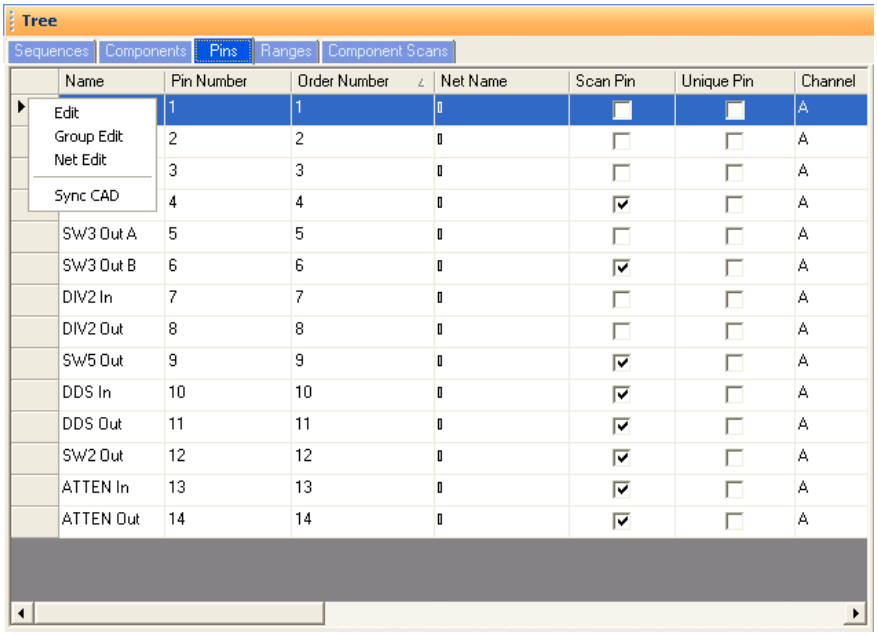
Sequence/Component/Pin Editing – Right Click menus



Right clicking on the Row header bar of the Tree pane displays an additional menu for performing specific tasks such as **Add New**, **Insert**, **Delete**, **Repeat**, **Build** (repeat but with name incremented by 1), **Copy**, **Paste** and **Cut**. Right clicking works at all levels of the Tree pane including Sequence, Component (shown above), Pins, Ranges and Scans. The **Sync CAD** option available at the Component and Pin levels will synchronize the selected component or pin to the CAD image in the Image pane.
It is also possible to select groups of components by holding the **SHIFT** or **CTRL** keys while selecting components in the Tree. To copy, paste, cut or delete groups of components, use the Copy, Paste, Cut or Delete Toolbar buttons below the main menu.



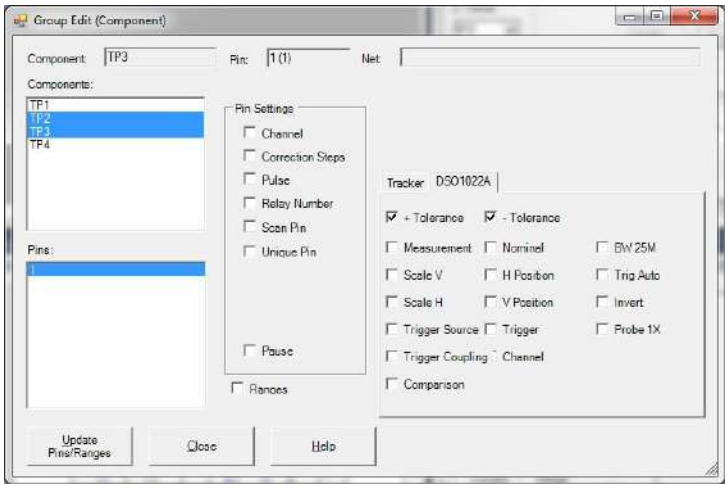
Pin Editing – Right Clicking (Auxiliary menus)



Right clicking on the Row header bar of the Tree/Pins pane allows for editing of the pin information. Select **Edit** to open the Edit window for the selected pin.

Note: Some Pin settings can also be edited directly in the Pins grid by clicking the appropriate field and modifying the setting.

Group Edit and **Net Edit** allow you make changes to the pin of a component and have those modifications also change components that have the same number of pins within the same Sequence.

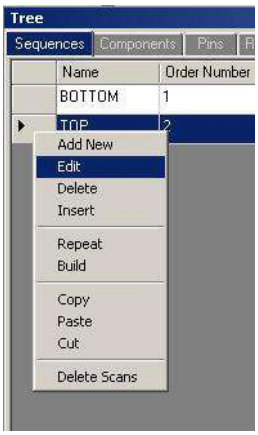


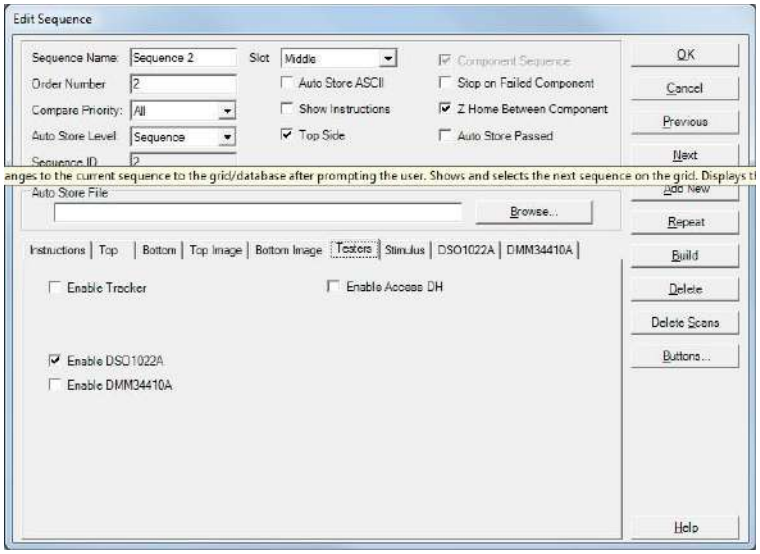
Change the desired settings for the current Pin first then select Group Edit from the right-click menu (see above). Components with the same number of pins will be listed. Select the components and pins you want to be changed. Select the Pin Settings check box(s) for the setting(s) you want changed. Check the Ranges box to match the ranges (this will cause signatures to be deleted). Click the **Update Pins/Ranges** button to make the changes.

Huntron Workstation Buttons Feature

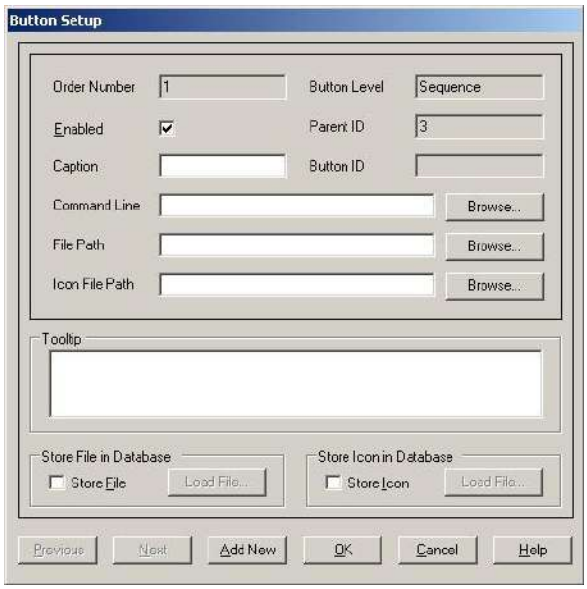
Huntron Workstation has a built-in feature that allows any Windows based program to be started by clicking a button in the Workstation toolbar. It is very easy to use and a "Button" can be attached to any Board, Sequence or Component/Net. In this example, a Button will be created for a Sequence but the process is the same for the Board and Component/Net levels.

To start, the Sequence edit window is opened by right-clicking the row header for the desired sequence. The Edit window will be displayed (see below).

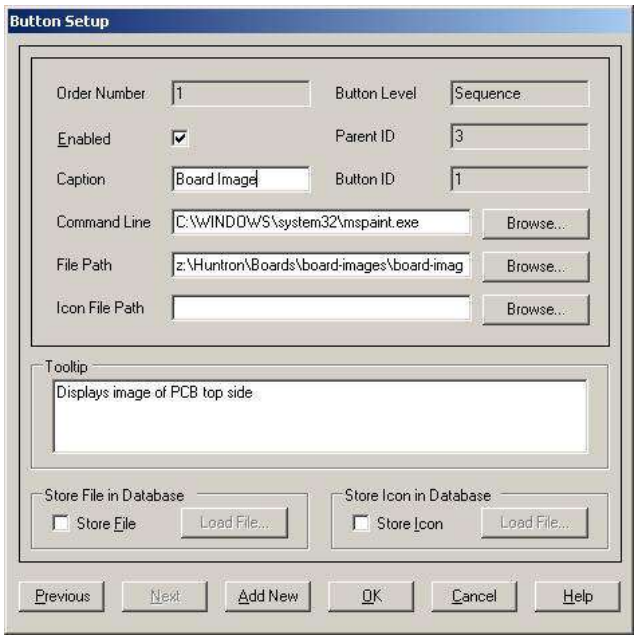




Select the **Buttons...** button located on the right side of the Sequence Edit window to open the Buttons Setup window.

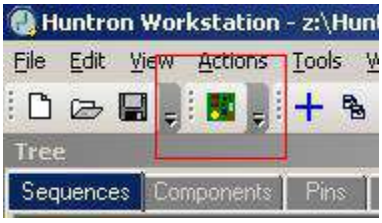


To configure the Button, a Caption (button name displayed in the Tools menu), Command Line (the application to be run) and File Path (the file opened by the executed application) need to be input into the corresponding fields. A File path will not always be required such as instances where a browser is started and pointed to a specific URL. In this example, the button will be configured to start the Microsoft Paint application and open a small PCB image (see image below). If desired, the icon displayed in the main window toolbar can be set by browsing to and setting a path to an .ICO (icon) file in the **Icon File Path** field. Add text that will be displayed when the cursor is placed over the Button in **Tooltip** field. To make the linked file or icon stored as part of the Board database, check the **Store File** and/or **Store Icon** checkboxes.



Click **OK** to accept the Button. To create additional Buttons, repeat this process and click the **Add New** button in the Button Setup window.

When Buttons are created, an additional toolbar will appear below the Huntron Workstation menu. This toolbar will change according to the level (board, sequence or component/net) selected.



The icon shown can be replaced by selecting an appropriate icon in the Button Setup window (the default icon is shown). Click the toolbar button to execute the Button.

Technical Support

For questions or assistance in using Huntron Workstation, contact **VI-Tracer** at **+30-210-59-86-179** or email **technical@vitracer.com**