Via Acquanera, 29 22100 COMO
tel. 031.526.566 (r.a.) fax 031.507.984
info@calpower.it www.calpower.it

Huntron Workstation (v 4.3) Tutorial

Installation Instructions 4
Huntron Workstation Main Interface 5
The Toolbar 6
Hardware Setup 6
Using a ProTrack 8
Using a Tracker 3200S 10
Using a Tracker Model 30 10
Manual Modes – Tracker and Scanner 10
Creating a Board Test 11
Converting a Test Created in Workstation 4.2 12
Huntron Convert MDB Application 13
Test Building Procedures 14
Creating a Board Database 15
Adding a New Sequence 16
Component Package Types 17
Adding a New Component 18
Tree/Pin Tab 19
Adding a New Range 20
Mounting a PCB in an Access Prober 25
Prober Setup (Access DH users skip to Camera Offset for 26
Access DH) 26
– Camera Offset part 1 26
Prober Setup – Camera Offset part 2 27
Camera Offset for Access DH 28
Alignment for Access DH 31
Prober setup – Alignment part 1 32
Prober setup – Alignment part 2 33
Creating a Board Image 34
Prober setup – Component Teach; step 1 (DIP package example) 35
(Access DH Users go to "Component Teach with Access DH") 35
Prober setup – Component Teach; step 2 (DIP package example) 36

© Huntron, Inc 2017

Prober setup – Component Teach; step 3 (DIP package example) 37
Prober setup – Component Teach; step 4 (DIP package example) 38
Prober Setup – Component Teach with Access DH 38
Prober Setup – Probe Package Component Teach for Access DH 42
Duplicating XY Teach Position to Multiple Pins and Components 44
Duplicating XY Teach Position for Selected Components 45
Prober setup – Component Teach Height; step 1 (Access DH users go to "Teach Height for Access DH") 47
Prober setup – Component Teach Height; step 2 48
Teach Height for Access DH 49
Manual Modes – Camera and Prober 51
Scanning a Sequence or Component 52
Creating a Scan List 53
Scan Results 56
Viewing Signatures – Troublesheet 57
Viewing Signatures Troublesheet Report 58
Viewing Signatures – Right Click 60
View SigAssist Information 61
Component Scans Information 62
Component Scans – Right Clicking (Auxiliary menus) 63
Component Scans – Right Clicking (Auxiliary menus) 64
Component Scans – Right Clicking (Auxiliary menus) 65
Range Editing – Right Clicking (Auxiliary menus) 65
Sequence/Component/Pin Editing – Right Click menus 66
Component Editing – Component Group Edit 67
Using Component Group Edit to Offset XY Pin Locations 68
Pin Editing – Pin Group Edit and Net Edit 69
Pin Editing 70
ViewPCB Image 71
Test Creation - Panelize 72
Test Creation – Convert step 1 73
Test Creation – Convert step 2 74

Test Creation – Convert step 3 75
Test Creation – Convert Status 76
Huntron Workstation Buttons Feature 77
Technical Support 80
Appendix 1 – Using the Set Size Feature for Access Probers 81
Appendix 2: Huntron Workstation Archive Scans Feature 86

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 4.3 prior to installing the new version. Insert the Huntron Workstation DVD. The DVD should Autorun and begin the installation process. If your PC has Autorun disabled then open a browse window to the DVD 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 (refer to the "Getting Started" documents supplied with the Prober).

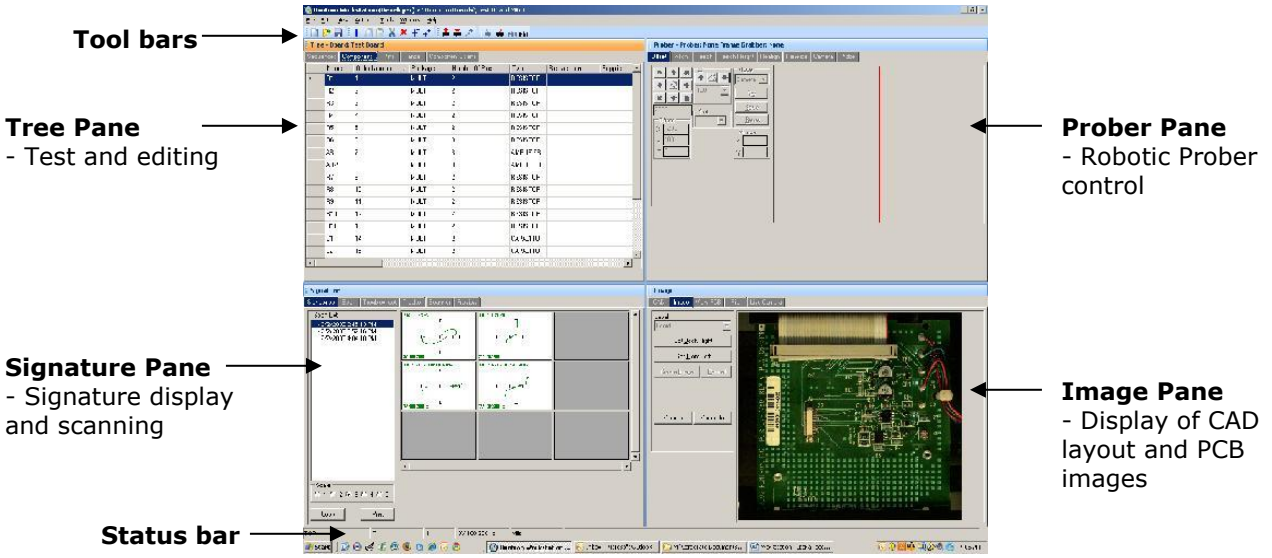
If you purchased CAD Tools then install using the supplied CD. Follow instructions listed in the CAD Tools Tutorial.

If you are using Workstation in conjunction with a digital multimeter, follow the instructions listed in the Workstation DMM Tutorial.

All of the Workstation tutorials are available on the Huntron web site:
www.huntron.com/sales-support/software.htm

Huntron Workstation Main Interface

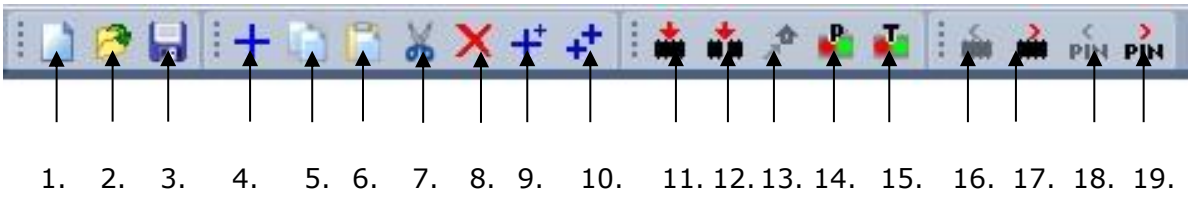
When starting Huntron Workstation for the first time, you will be prompted for an activation code. Type in the code exactly as it is presented on the installation CD. Huntron Workstation 4 has built-in conversion utilities that enable conversion of board tests created in 3.X versions of Huntron Workstation.



The Main Window for Huntron Workstation 4 features a multi-pane layout so many aspects of the test creation and execution can be displayed at the same time. 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.

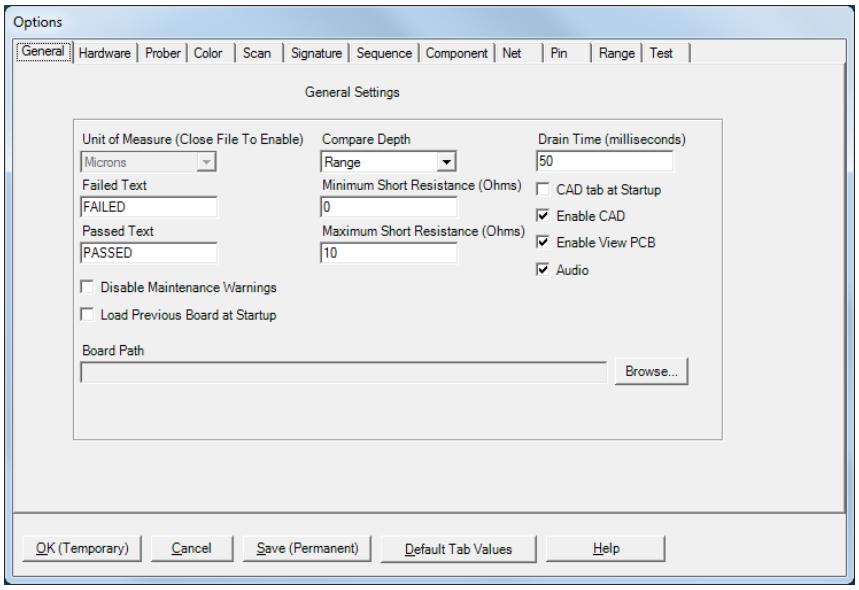
The Toolbar

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.



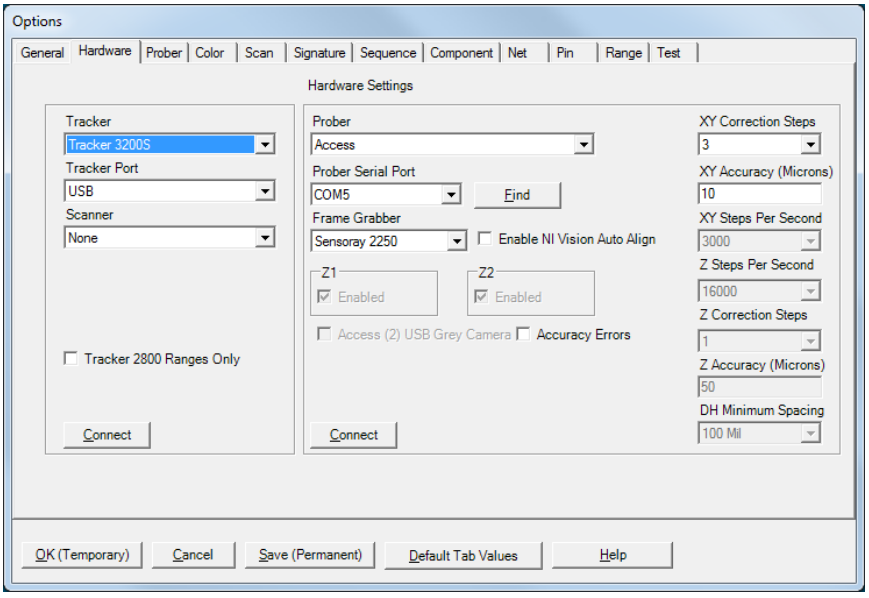
1. Create New Board database
2. Open Board database
3. Save As…
4. Add New (Sequence, Component or Range)
5. Copy (Sequence, Component, Net or Range)
6. Paste (Sequence, Component, Net or Range)
7. Cut (Sequence, Component, Net or Range)
8. Delete (Seq., Comp., Net, Range or Scan)
9. Build New (Sequence, 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



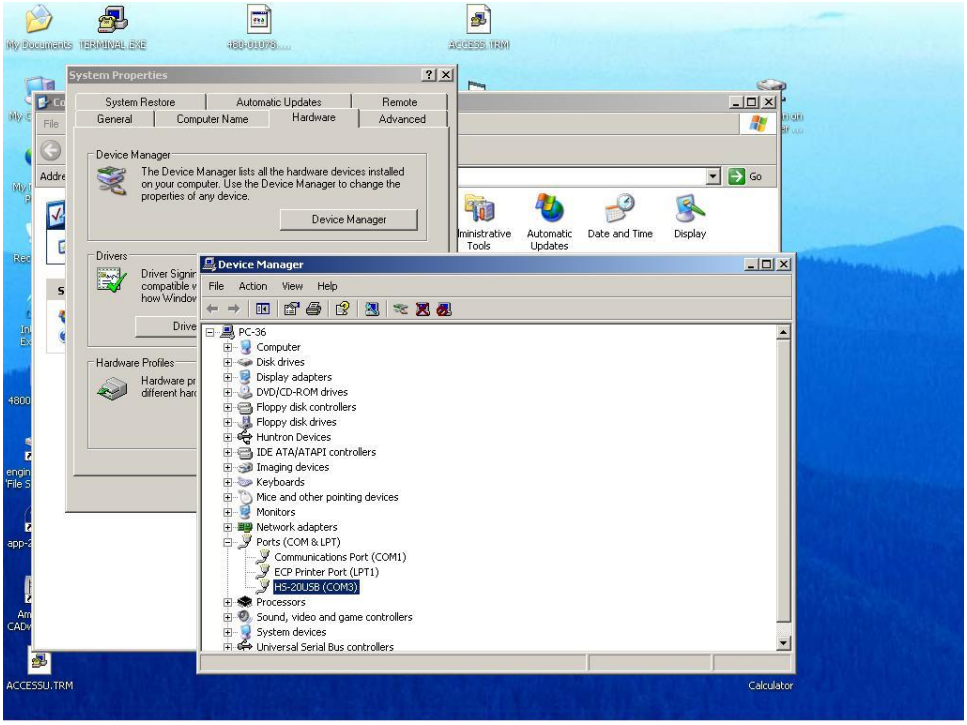
When first starting Huntron Workstation, select **Options** from the Tools menu. The Options window is used to setup default settings used when working in Huntron

Workstation. Select the **General** tab and set the desired Unit of Measure (use **Microns** for Huntron Access Probers, **Mils** for any other Prober).
For this tutorial, the Huntron Tracker 3200S, Tracker Model 30, ProTrack and TrackerPXI will be referred to in general as "Trackers". Specific models will be mentioned by name if certain software features apply a specific Tracker model. The ProTrack Scanner, Scanner II and Scanner 31S will be referred to as "Scanners". Specific models will be mentioned by name if certain software features apply a specific Scanner model.



Select **Options** from the Tools menu. Select the **Hardware** tab and set the type of Tracker hardware that is connected to the PC, the Tracker Port (USB) and Scanner (if connected; Scanner II users should also select the number of Scanners connected in the "Scanners" drop menu**). Access Probers with Tracker** (built-in) should select "**Access Tracker**" in the Tracker menu.

Prober users will need to select the type of Prober, the serial port for the Prober and the Frame Grabber type. To determine which COM port the Access USB (for Access USB Probers only) connection is using, click the Find button for Workstation to detect the Prober port. If this does not work, 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".

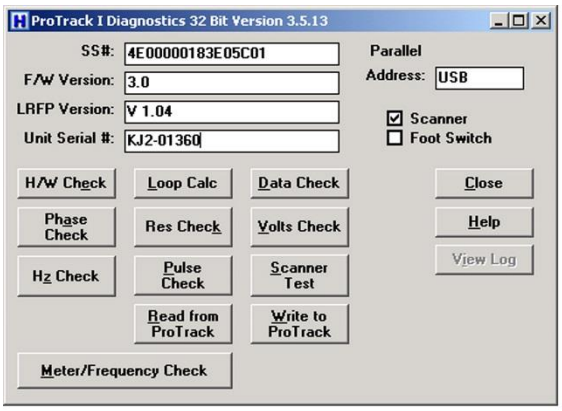


Check the **Enable NI Vision Auto-Align** checkbox if you wish to enable the automatic alignment process that uses the National Instruments Vision Runtime. This option will function only if the NI Vision Runtime has been installed from an earlier Workstation installation (version 4.3.6361 and before). Leave this checkbox **unchecked** to use the new built-in Auto-Align function.

Click the **Connect** buttons to initiate communication with the selected hardware. Once connected, click the **Save (Permanent)** button to save this configuration as the startup default. For initial installation using a ProTrack, you will see a message that Loop Compensation was not found. A Loop Compensation file will need to be created.

Using a ProTrack

If a Loop Compensation (Loop Calc) file does not exist for the selected configuration, a message will appear asking that Loop Calc be performed with ProTrack Diagnostics.



Loop Calc needs to be run before you can use a NEW installation of the Huntron Workstation software with a Huntron ProTrack.

- Select **Tools/Maintenance/ProTrack Diagnostics** from the menu

Run a diagnostic routine by clicking the appropriate button in the ProTrack I Diagnostics window. Clicking **Loop Calc** will execute the Hardware Check (except Scanner Test) before running the Loop Calc routine. The Loop Calc takes approximately 20 minutes to complete. Once complete, a Loop Compensation file is created for the selected hardware configuration. You will also be prompted to save a ProTrack configuration or ".PTC" file (the name will be the serial number you entered in the Diagnostics window). Click **Yes** to save the PTC file to your PC. With newer ProTrack units (serial number prefix KJ3 or KJ4), you will be prompted by another window displaying "Download the LoopCalc settings into the ProTrack for manual control?" Click Yes to download the settings. Exit Diagnostics and select **Tools/Options/Hardware** and reinitialize the hardware by clicking the Connect/Disconnect buttons.

Other functions in the ProTrack Diagnostics allow for reading or writing the ProTrack configuration file. The **Read from ProTrack** button will retrieve calibration and user settings from the ProTrack I unit and store them as a text file on the connected computer in the Program Files/Huntron/Hardware folder. It is recommended this be performed when setting up the system for the first time or after making changes to the ProTrack such as the addition of customs ranges and range groups. The file generated should be stored in a safe location.

The **Write to ProTrack** button will send the calibration and user information to the ProTrack I unit. This can be performed if it suspected that the information stored in the FLASH memory of the ProTrack I has been corrupted.

Using a Tracker 3200S

The Tracker 3200S comes from the Factory calibrated for standalone use. If the Tracker 3200S is connected to a Prober, Scanner or other cabling, calibration must be performed.

- Select **Tools/Maintenance/Tracker 3200S Diagnostics** from the menu.

Click the **Alignment** button the Tracker 3200S Diagnostics window. When the Calibrate procedure finishes (about 10 minutes), run **Verify Alignment**. This also takes about 10 minutes to complete. The Calibration information is stored in the Tracker 3200S and will be used by the software. If changes are made to the cabling or attached instruments, simply run Calibrate and Verify Calibration again.

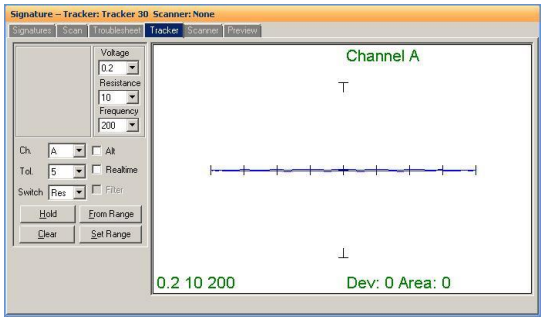
Using a Tracker Model 30

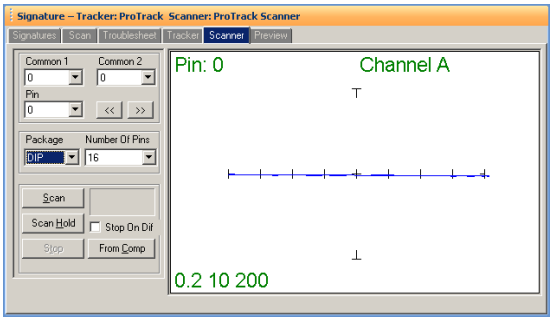
The Tracker Model 30 comes from the Factory calibrated for standalone use. If the Tracker Model 30 is connected to a Prober, Scanner or other cabling, calibration must be performed. **Access Probers with Tracker** inside come from the factory with the calibration already completed.

- Select **Tools/Maintenance/Tracker Model 30 Diagnostics** from the menu.

Click the **Alignment** button the Tracker Model 30 Diagnostics window. When the Calibrate procedure finishes (about 10 minutes), run **Verify Alignment**. This also takes about 10 minutes to complete. The Calibration information is stored in the Tracker Model 30 and will be used by the software. If changes are made to the cabling or attached instruments, simply run Calibrate and Verify Calibration again.

Manual Modes – Tracker and Scanner





You may find it useful to manually test some of your components while adjusting the range parameters. The two manual modes, Tracker and Scanner located in the **Signatures** pane, can work together to display the signature of a component pin.

Tracker mode sets the range parameters, channel and tolerance. You can also set the Pulse Generator settings (Tracker 3200S and ProTrack only) and use **ALT** mode to run channel A versus channel B comparisons. Signatures will be displayed to the right while probing with hand-held probes (select the Real-time check box). The **Hold** button will capture and hold the current signature allowing for quick comparisons. Scanner mode allows you to use a Huntron Scanner to interface to the component pins. You can set the package type, number of pins and common pins. The **Scan** button starts a hands free scan of the component or you manually increment the pins with the **<<** and **>>** buttons. The **Hold** button will capture and hold the current signature allowing for quick comparisons. To store signatures you will need to create a **Board Test**. NOTE: Users with Access Probers with Tracker inside should select **Channel B** when using Manual mode.

Creating a Board Test

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

Workstation Tree Structure

- Board (created Huntron Workstation (.hwdb) database file)
- Sequence (Test set of Components)
- Component or Net (name)
    - Pins (name, channel)
    - Ranges (V, F, R range settings)

Example of Board Test Information:

Board Name: Main Board (creates "main board.hwdb" file)

Revision: Rev. A (This field is optional)

Sequence: IC's

Component: U27 (component name) or A2 (NET name)

Pins: per pin settings for Scan Pin setting, channel, net name, etc…

Ranges: Per pin settings for common reference pin, tolerance, etc…

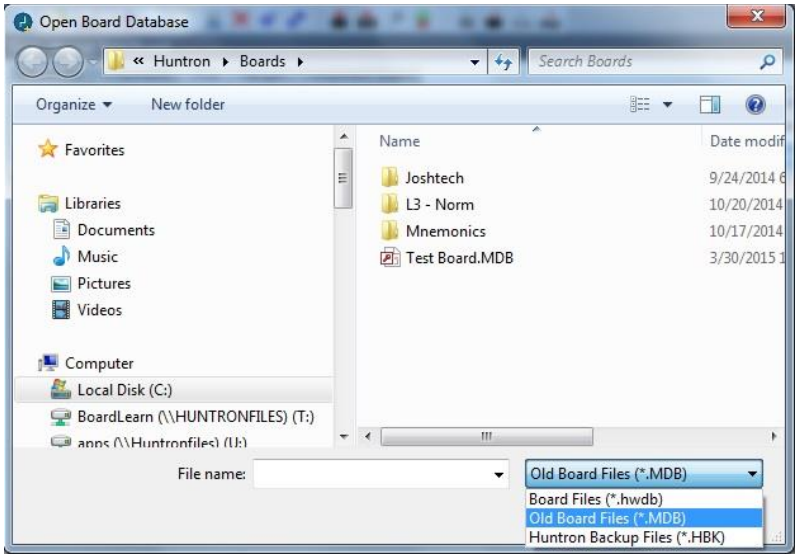
The user has total discretion as to the information input into these fields. There is no "right" way as to what information is entered as every user's situation will be different. The above example is just one way the information might be entered.

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.

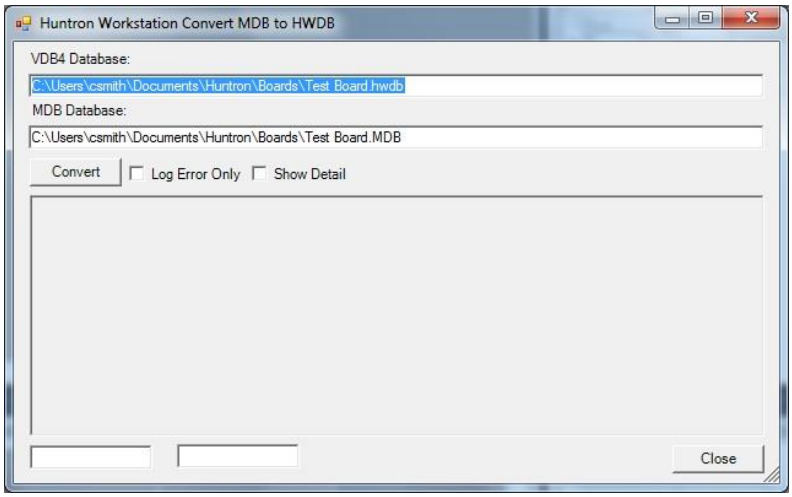
Converting a Test Created in Workstation 4.2

Workstation 4.3 uses a different database type than version 4.2. To update a version 4.2 test to work in version 4.3, select **Open** from the **File** menu. In the Open window, select **.MDB** from the file type drop menu. Select the .MDB test file name and click the **Open** button.

Note: Select **.HBK** if you are converting Workstation version 3.X backup files.



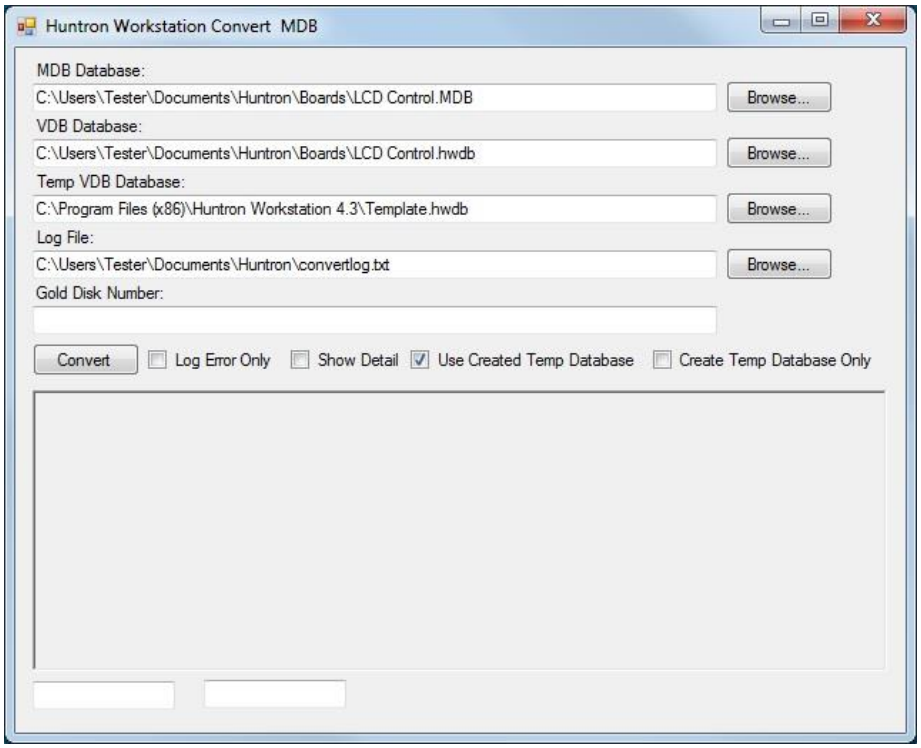
Click the **Convert** button to create the new test for version 4.3. The conversion can take some time based on the original test size.



Click the **Close** button when the conversion is complete. The updated test will be opened in the Workstation interface.

Huntron Convert MDB Application

As an alternative, the conversion process can be run externally by browsing to the Program Files (or Program Files (x86))\Huntron Workstation 4.3 folder and double-clicking the **HuntronConvertMDB.exe** application. This application allows you to select an .MDB file created in version 4.2 and convert it to the 4.3 database format.



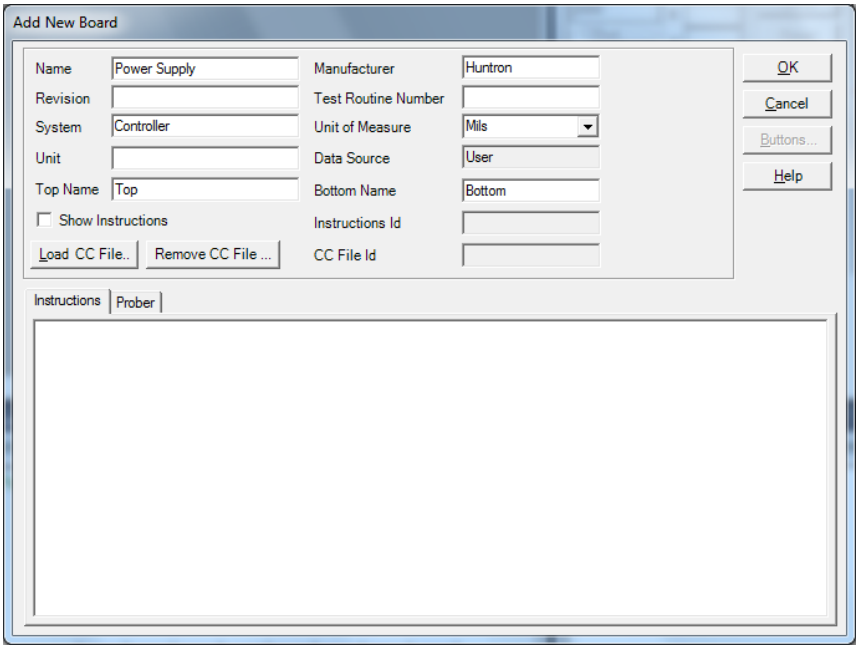
Use the **MDB Database field** to select the version 4.2 test database you want to convert.
Use the **HWDB database field** to select the path and name for the new 4.3 database.
Use the **Template HWDB Data field** to select the preinstalled Template database.
Use the same information as displayed in the image above.
Use the **Log File field** to select the path and name for the application log file.
Leave the **Gold Disk Number field** blank unless you are converting tests used in the US Navy Gold Disk program.
Click the **Convert** button to start the conversion. You will see a pop-up dialog when the conversion is complete.

Test Building Procedures

- Create Test Database in the Tree Pane
    - Add Board, Sequence, and Components before teaching or scanning
- Scan Components and Set References (Tracker only or Tracker/Scanner users)
- Perform Camera Offset (Tracker/Probers users)
- Select Alignment points (Tracker/Prober users)
- Teach component pin locations (Tracker/Prober users – teaching all points before scanning will save you time)
- Scan Sequences and Set References

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

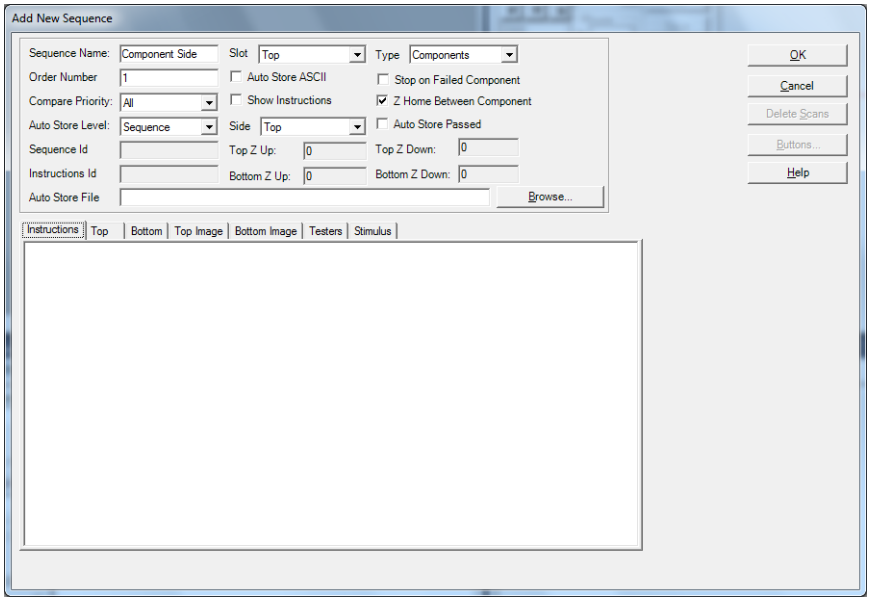
Creating a Board Database



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**. Other fields such as Revision, System, Unit, Manufacturer and Gold Disk Number can be completed with related information if desired.

Access Prober users should select their desired **Units of Measure**. This controls which units (mils, inches, microns, or millimeters) are used in the Prober movement controls.

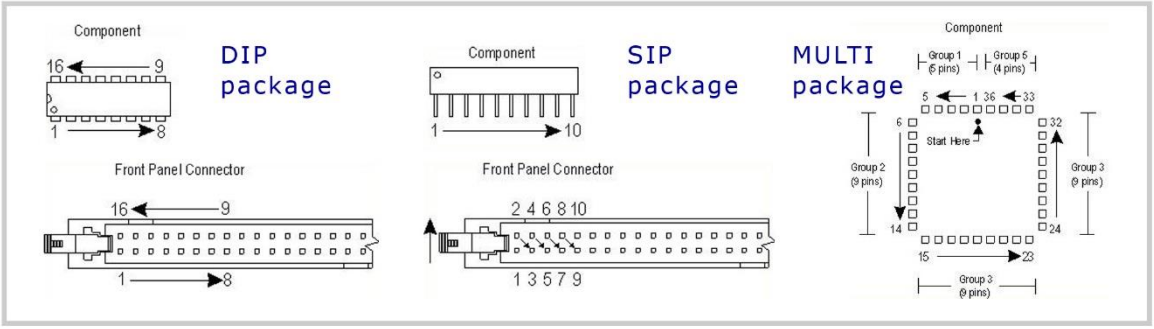
Adding a New Sequence

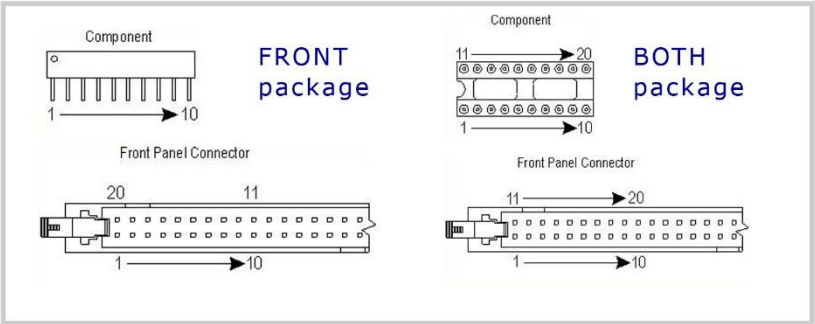


To add a new Sequence, select the **Sequence** tab in the Tree pane. Select **Add New Sequence** from the Edit menu or click the Add New button **(+)** in the toolbar.
Details:
Sequence Name Field: Enter Sequence name
Slot field (Prober users): Select the Prober slot that the board will be mounted into when scanning. Select from TOP, MIDDLE, BOTTOM or BASE (Access 2, Access DH).
Compare Priority: Selects the comparison priority between **Same** (serial number), **All** (serial numbers; this is the recommended setting) or **Merge** (Merged min/max signatures)
Instructions field: Enter any instructions or system information that you wish to pass on to future users.
The **Delete Scans** button will erase all scans stored under the Sequence.
NOTE: Access DH users should select the **Testers** tab and check the **"Enable Access DH**" checkbox.
Buttons... button adds a new user configured Toolbar button. See the Buttons section at the end of this tutorial.
NOTE: You can set your Sequence defaults using the **Tools menu/Options/Sequence tab**. Access DH users should make sure the "**Enable Access DH**" checkbox is checked.

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

Component Package Types



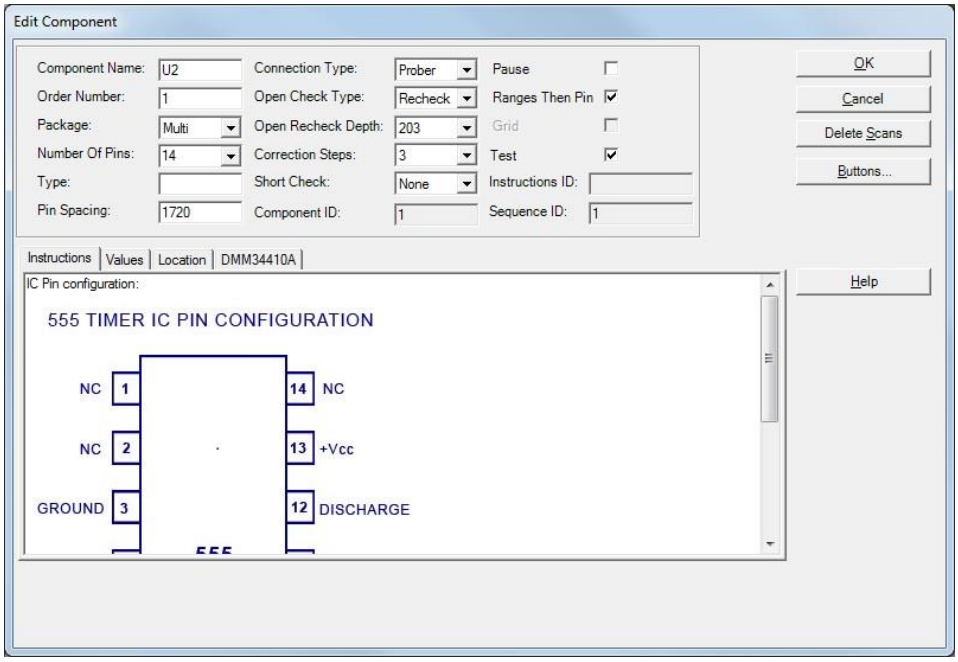


Prober users will use the DIP, SIP, MULTI and PROBE packages.
Tracker with Scanner users will use DIP (including DIP2X), SIP (including SIP2X), FRONT, BOTH, DIPII or SIPII

There are four different package types you will find yourself using when creating components used in a Prober test sequence. The XY locations for each package type are "taught" in slightly different ways. They are **DIP** (dual-in-line), **SIP** (single-in-line), **PROBE** (point by point), and **MULTI** (multiple row or sided).

For Tracker with Scanner users, the package type determines how the Scanner pins are counted. The available Scanner packages are **DIP**, **SIP**, **Both**, **Front**, **SIP2X** (combines the A and B channels for 128 pins) and **DIP2X** (combines the A and B channels for 128 pins). Users of the Scanner II accessory have a **DIPII** and **SIPII** package available allowing them to daisy-chain Scanners together to achieve up to 1024 pins.
Users of Probers ***and*** Scanners have a **MULTI-SIP** package that lets them to probe a component as **MULTI** package while the Scanner counts through the front connector as a **SIP** package. This allows for selection of Common references through the Scanner to the Prober mounted board-under-test using a custom cable.

Adding a New Component



To add a new Component, 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 (reference designator) in this field.

Package and **Number of Pins** fields: Using the mouse, click on the down arrow to the right of the field to bring up the pull-down menu. Select the package type and number of pins from these menus. For **Access DH users** it is recommended that you select **Number Of Pins** as **1** for 2 pin Probe package components such as resistors and capacitors.

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

Pin Spacing: For prober users, you can enter the spacing between component pins in this field (i.e. mils, microns, mm or inches depending on Board level setting). This will be used to calculate distances between key pins when in **Teach** mode.

Connection Type: Select the type of hardware used to interface to the component

Open Check Type: Select from None, Recheck or Reprobe to retest test points that are detected as open. Selecting Recheck will test the point with the probe lowered the distance set by Open Recheck Depth. Reprobe will raise and lower the probe several times then retest. The number of "reprobes" is set in the Tools/Options/Prober tab.

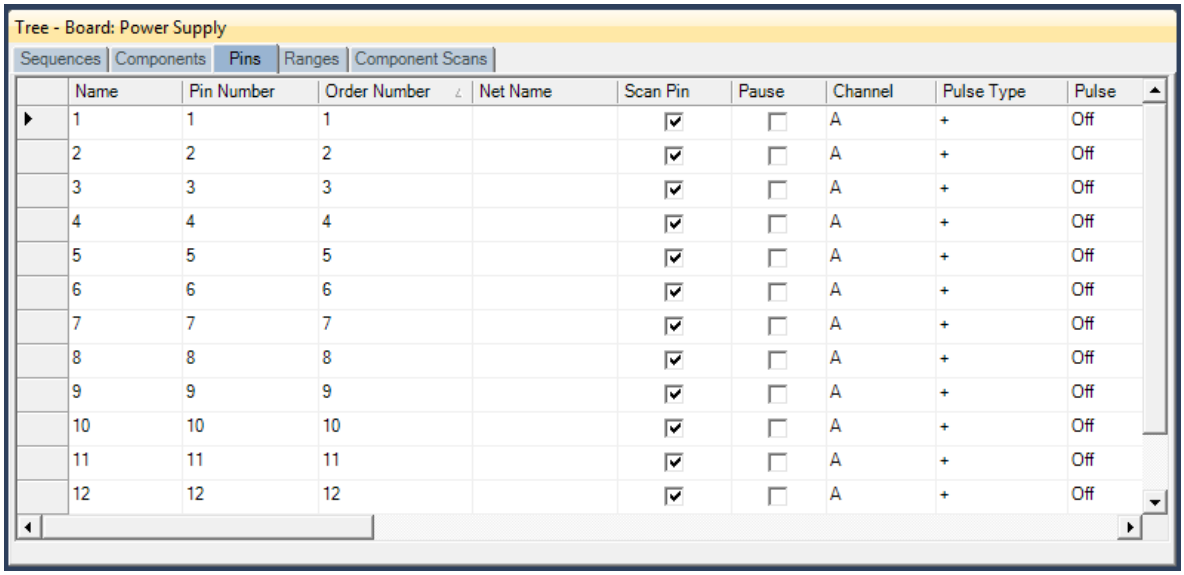
Correction Steps: Sets the maximum amount of correction steps used with the Huntron Access Prober when moving to the first pin of the component.

Instructions field: Enter any instructions or component information that you wish to pass on to future users. Version 4.3 allows you to paste rich text format (RTF) information into the Instructions field. This includes formatted text and images. It is recommended that the JPEG format be used for images to keep the database size under control.

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

Tree/Pin Tab

Once a component is added to the Workstation Tree, switching to the Pin tab will display information for each pin of the selected component.



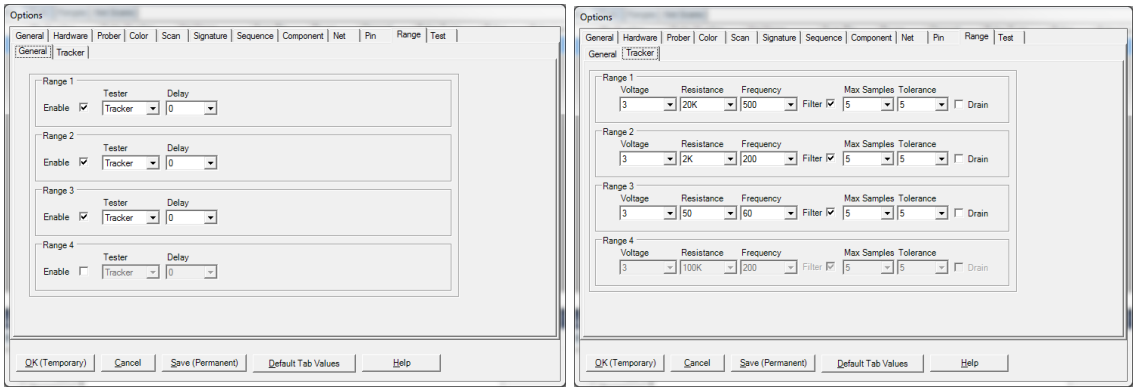
Name	Pin Number	Order Number	Net Name	Scan Pin	Pause	Channel	Pulse Type	Pulse
1	1	1		☑	☐	A	+	Off
2	2	2		☑	☐	A	+	Off
3	3	3		☑	☐	A	+	Off
4	4	4		☑	☐	A	+	Off
5	5	5		☑	☐	A	+	Off
6	6	6		☑	☐	A	+	Off
7	7	7		☑	☐	A	+	Off
8	8	8		☑	☐	A	+	Off
9	9	9		☑	☐	A	+	Off
10	10	10		☑	☐	A	+	Off
11	11	11		☑	☐	A	+	Off
12	12	12		☑	☐	A	+	Off

Text in the **Name** field can be modified directly by clicking in the text field and typing the new name. The **Scan Pin** setting enables or disables scanning of the pin. **Pause** will cause the scan to pause when the selected pin is reached. The **Channel** field controls which channel (**A or B**) is used. Controls for the Pulse Generator (ProTrack and Tracker 3200S) can also be controlled at the Pin level. These control include **Pulse Type** (polarity), **Pulse** (on/off), **Pulse Level** (amplitude) and **Duty Cycle** (up to 50% on pulsed signals).

Information on the other settings in the Pin tab is available in the **Workstation Help**.

Adding a New Range

When a new component is added to a Sequence the ranges used are set by the defaults set up in the **Tools/Options/Ranges** tab. It is best to set up your desired default ranges before adding Components to your Sequence.

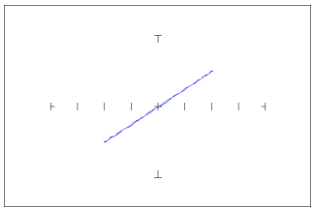


In the Range/General tab, enable the number ranges and set the tester type as Tracker (default). In the Ranges tab, set the Voltage, Resistance and Frequency range parameters. These are the ranges that will be used when adding a new Component. It does not change existing Components. On a new software installation these defaults set three ranges – **3V, 50Kohms at 500Hz**; **3V, 1Kohms at 200Hz** and **3V, 50 ohms at 60Hz**. In general use, these three ranges will work well but you may need to add or modify ranges based on component type and value. As a general reference, the table below can be used to assist you in setting ranges.

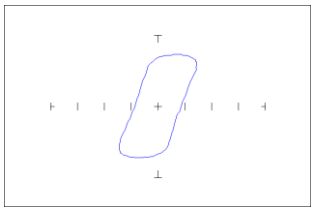
Range Impedance↓	Voltage> 200mV	3V	6V	10V -15V	20V
10Ω - 100Ω	continuity, shorts, caps, resistors & inductors	low impedance components	newer MOS ICs	power supply & regulators	
1KΩ - 10KΩ	resistive faults	latest MOS & surface mount ICs	newer MOS ICs	older MOS ICs	
20KΩ	resistive faults	resistive faults	older MOS ICs	TTL 74XXX & 74LSXXX ICs, older MOS Ics	TTL 74XXX ICs, power transistor
100KΩ	resistive faults	resistive faults	resistive faults	resistive faults	Hi resistance& Hi power

The Tracker signatures are shaped based on the Tracker range settings and the component type/value. The basic goal is to achieve signatures that are halfway

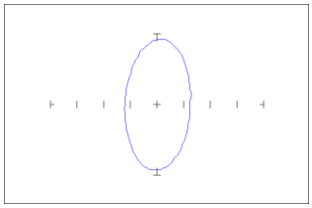
between open and short. The signatures for the four basic types of components shown below illustrate this primary goal.



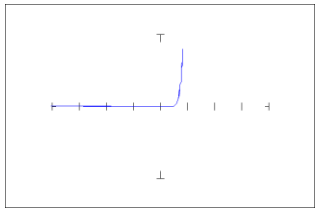
Resistance



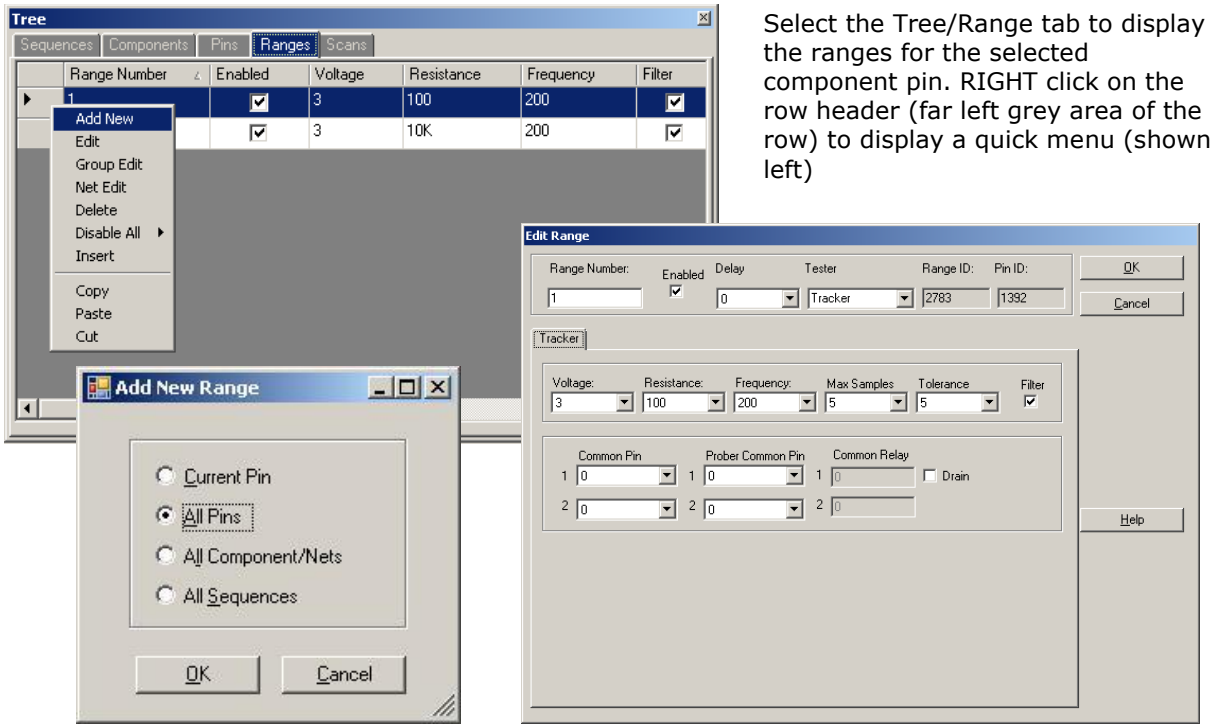
Inductance



Capacitance



Semi-conductance



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 add a new Range to the selected Component, select the Range tab in the Tree pane. Select **Add New Range** from the Edit menu or click the Add New button (**+**) in the toolbar. A pop-up dialog will ask where you want the new range to be added. Select the desired radio button (**All Pins** is generally the desired selection) and click **OK**.

Add New Range Details:

Note: **Voltage/Resistance/Frequency** can be modified directly in the Range grid.

Voltage/Resistance/Frequency fields: Select the desired range settings

Max Samples/Tolerance/Delay: Select the setting to be used when scanning **this range**

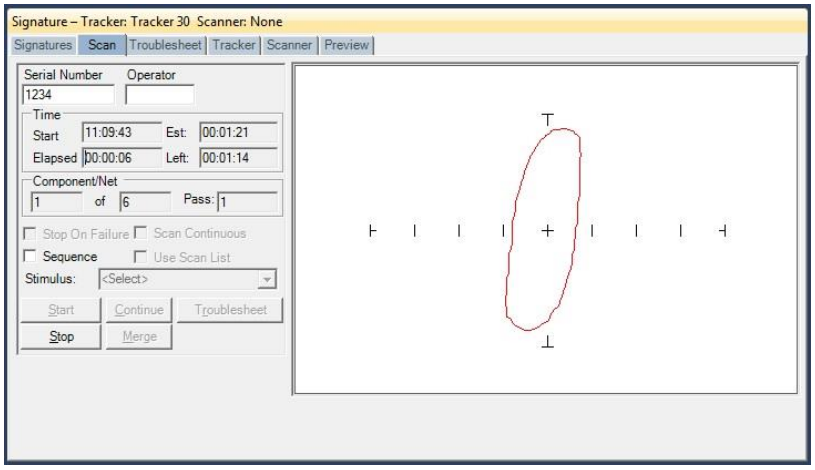
Common Pin: Select Scanner common pins to be used when scanning **this range**. Scanner users wishing to use a particular component pin (i.e. ground pin) for Common should select the Common pin at this time using the **Common Pin 1 field**.

Prober Common Pin: This allow Probers to select a Common connection (1-4) located on the front panel of the Prober.

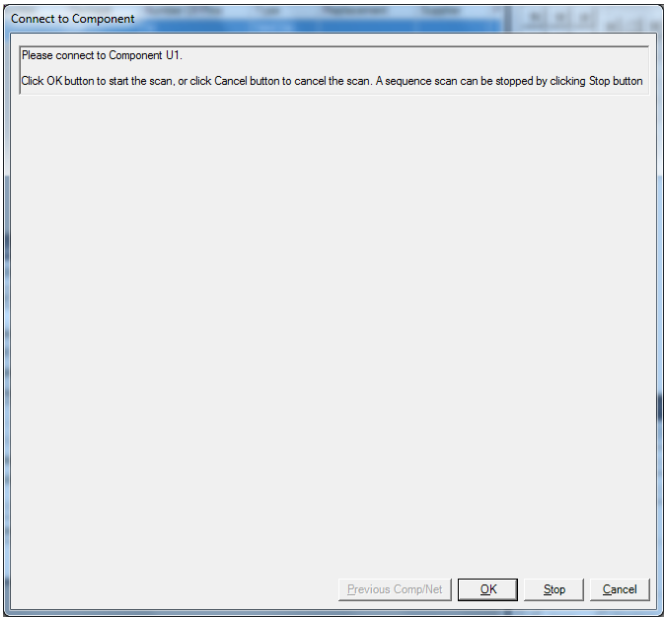
Note: It is easier to set up the ranges used by selecting **Tools/Options/Range** from the Tools menu in the menu bar and pre-configuring the default ranges.

Scanning with a Tracker or Tracker with Scanner

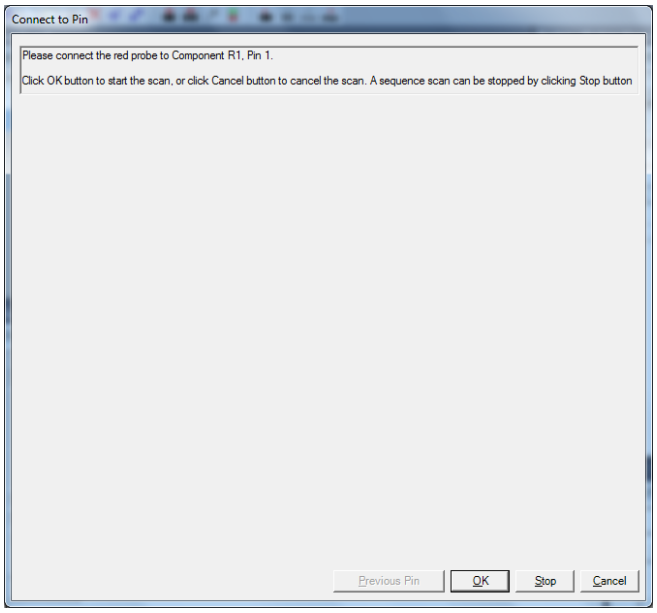
Users of Trackers with hand probes or Scanners can now proceed with actual scanning of a Sequence or Component. To scan an entire Sequence, select the Sequence in the Tree/Sequence tab or enable the **Sequence checkbox** in the Scan tab. To scan an individual component, select the component in the Tree/Component tab. Select the **Scan tab** in the Signatures pane and input a **Serial Number** then click the **Start** button.



You will be prompted to connect to the component. Tracker with Scanner users should connect to the selected component at this time. Click **OK** to initiate the Scan.



If you are performing Sequence scan you will be prompted to continue to the next component until all of the components in the Sequence have been scanned.
If a Tracker with probes is being used your will prompted to connect to pin 1 of the selected component (if performing a single component scan) or of the first component in the Sequence (if performing a Sequence scan). Click the **OK** button to scan the first pin. You will then be prompted to move to the next pin for scanning. Click **OK** to scan and continue in this manner until all of the component pins are scanned. At anytime during scanning you can click the **Previous Pin** button to go back and rescan the previous pin.

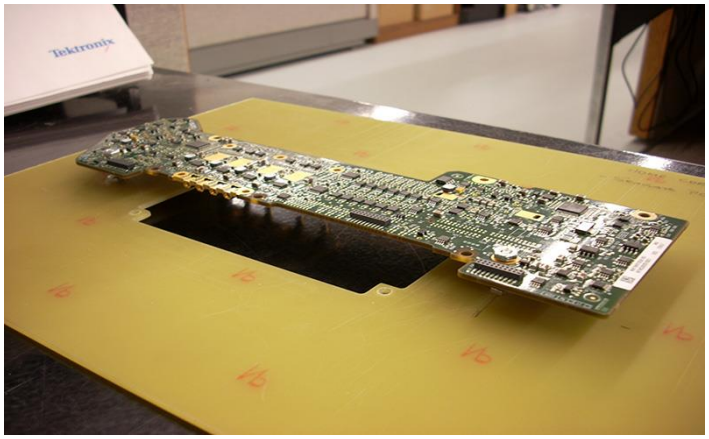


Once all pins are scanned click the **OK** button to continue.
When the Sequence or Component scan is complete the Scan Results window will be displayed. Go to the **Scan Results** section of this tutorial for more information on what to do after scanning.
If the Tracker Scanner connections are being used (i.e. DIP clip and cable) you will be prompted to Connect to the Component before the scan is executed. Connect the Scanner cable to the component to test and click **OK** to start the scan. Once the scan is complete, the Scan results will be displayed.

Mounting a PCB in an Access Prober

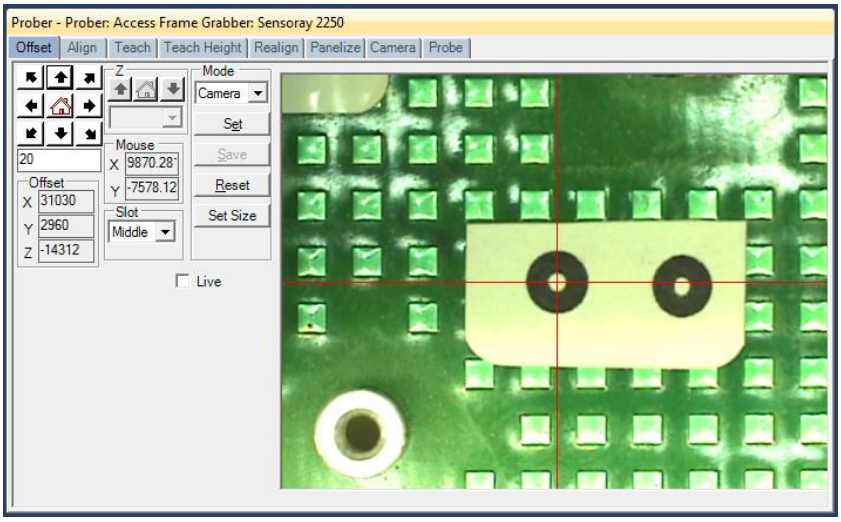
Before software functions relating to an Access Prober can be performed, the PCB under test must be mounted into the Prober test area. Use the mounting accessories supplied with the Prober to securely mount your PCB by its edges.

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 (Access DH users skip to Camera Offset for Access DH)

– 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. Make sure the Mode is set for **Camera**.

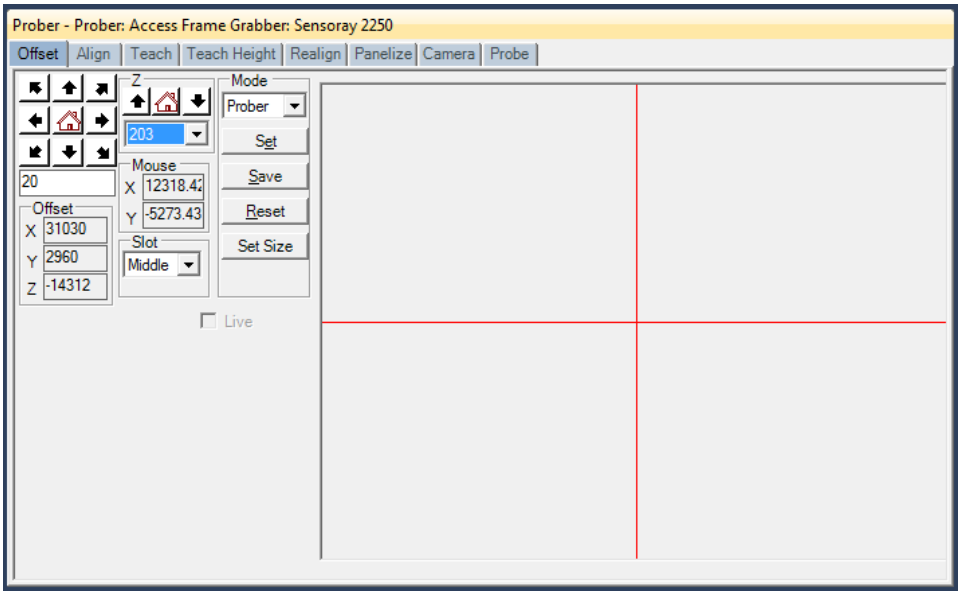
Use the arrow buttons to move the camera the distance set in the travel distance field (directly below the arrow buttons; Note: 1000mils = 1 inch). Navigate the Camera to a target on the PCB that is at least 2 inches from the Prober walls. A very small target that is still visible by eye works best. You can also click directly in the camera image to move the camera crosshairs to the clicked location. **Note:** If you notice that the camera crosshairs do not move to your clicked location, see **Appendix 1** at the end of this tutorial.

This is also the best time to focus your camera. Click the **Live Camera** checkbox to put the camera into Live mode. The aperture ring is the upper ring on the camera lens and the focus uses the lower ring. Set the camera to the lowest f-stop of 1.4 and adjust the focus as precisely as possible. You can change the f-stop to a higher number after focusing to achieve better depth of field. Uncheck the **Live Camera** mode when focusing is complete.

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



Caution: It is possible for the probe to contact tall obstacles on the PCB when being moved. Visually monitor the vertical probe position as it is moved to avoid probe damage.

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.

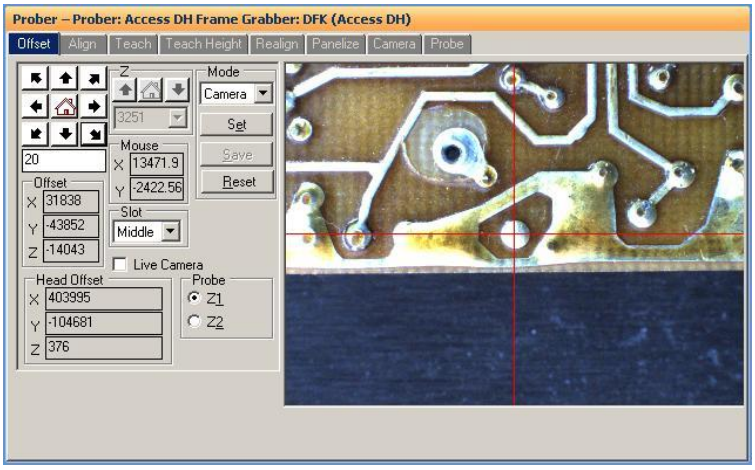
Use the **XY** buttons to navigate the probe to the target selected in Part 1. The probe will lift when moving in the X or Y direction. Access USB Prober users may find the Probe Tip Camera useful in setting the probe tip on the offset target. Select the **Live Camera tab** in the Image pane then click the Select button to display the video source dialog. Select the correct video source and click the **Start** button. A live image will appear in the Live Camera tab. Use the Live Camera only when needed as it can draw on computer resources and slow other software processes. Use the **Stop** button in the Video Source dialog to stop the video feed.

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.

Camera Offset for Access DH

Because the Access DH has two Z heads, setting the Camera Offset for each head is necessary. The procedure is very similar to the single head Prober process.

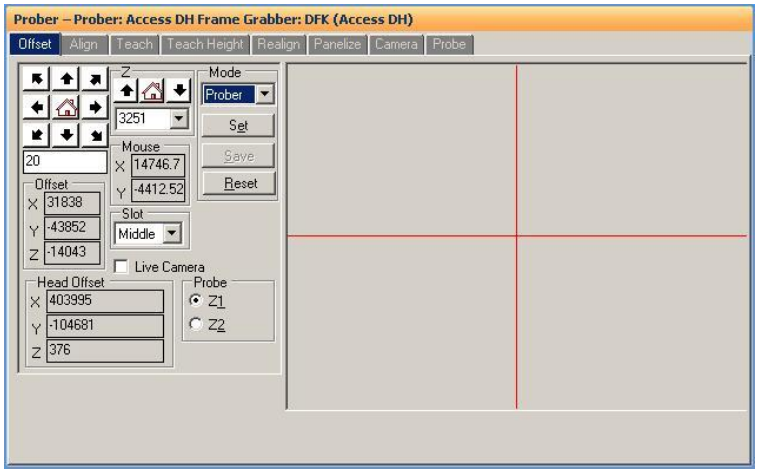
Set the Z1 Probe head first. Select the **Z1** radio button in the Offset tab of the Prober pane. Select **Camera** in the Mode field.



This is a good time to focus the camera if necessary. Enable the **Live Camera** checkbox when focusing. The aperture ring is the upper ring on the camera lens and the focus uses the lower ring. Set the camera to the lowest f-stop of 1.4 and adjust the focus as precisely as possible. You can change the f-stop to a higher number after focusing to achieve better depth of field.

Position the camera crosshairs on the center of your chosen target. **Note:** If you notice that the camera crosshairs do not move to your clicked location, see **Appendix 1** at the end of this tutorial.

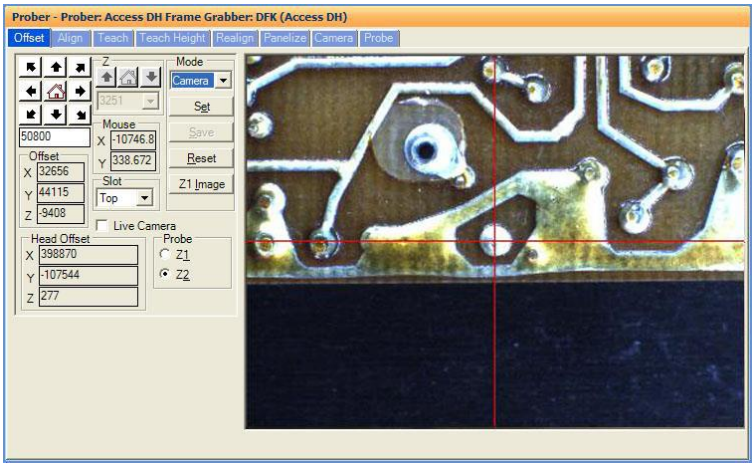
Press **Set**. Select Prober in the **Mode** field. The camera image will be disabled.



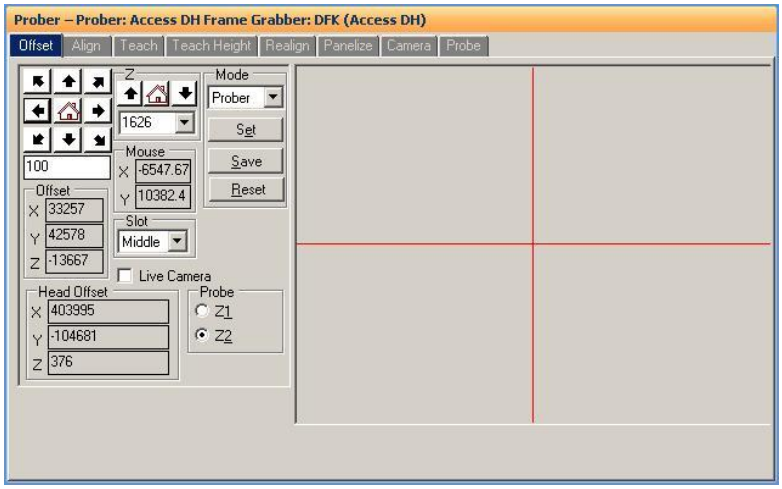
The Probe head will shift to position the probe tip over the target. Use the **Z** travel buttons to lower the probe down to the board surface. Visually examine the position of the probe tip; it should be located on the target selected earlier with the Camera, if not move on to the next step. Use a magnifying glass if necessary.

Use the **XY** buttons to navigate the probe to the target selected earlier. 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**.

Now the Z2 Prober head needs to be set. The procedure is the same as the Z1 head. Select the **Z2** radio button. Move the Z2 camera to the **same point used with the Z1 camera**. Use the **Z1 Image** button to see precisely where the Z2 position should be set.



Line up on your camera target and press **Set**. Select **Prober** from the Mode menu.

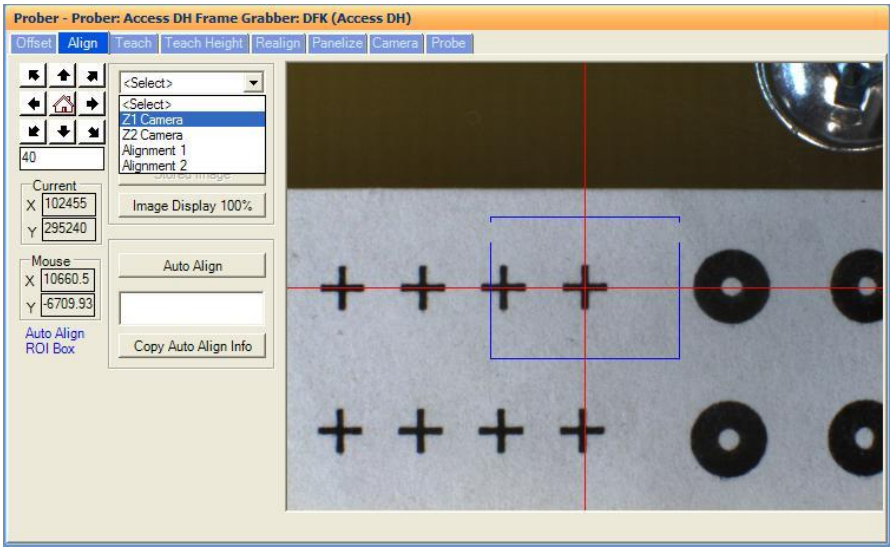


Lower the probe down to the board surface. Adjust the XY position if necessary using the **XY** travel buttons. When the probe is placed properly click **Set** followed by **Save** to store the Offset values.

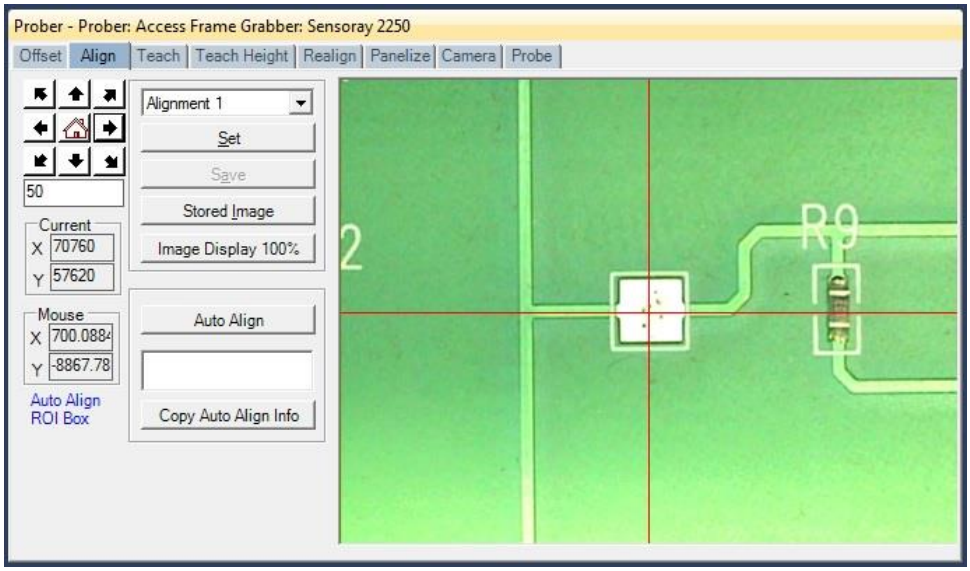
Alignment for Access DH

The Access DH allows for an optional Z Head Align step in the Alignment process to achieve better accuracy. If the **Align Z Heads** checkbox is enabled in the **Tools/Options/Prober tab** you will need to perform this process where each camera is placed over a target on the board under test.

Select the **Align** tab. Select **Z1 Camera** from the alignment drop menu, move the Z1 camera over your selected target and click **Set**. Select **Z2 Camera** and move the Z2 camera to the same target. Match the position as closely as possible (the **Stored Image** button shows the Z1 position) and click **Set**. Once complete, perform the Alignment 1 and Alignment 2 as described below for Access Probers.



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 to the Teach Height page.

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

For best accuracy when positioning the camera on alignment points, click the **Image Display 100%** button. This will display the camera image at full size with the best pixel resolution.

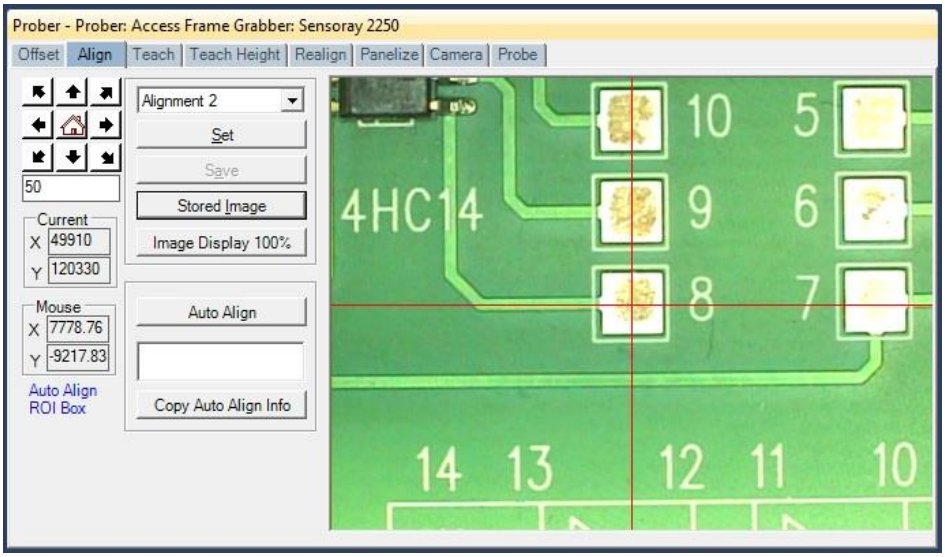
Select **Alignment 1** from the drop menu.

Select and navigate the camera to the first alignment point. PCB 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.

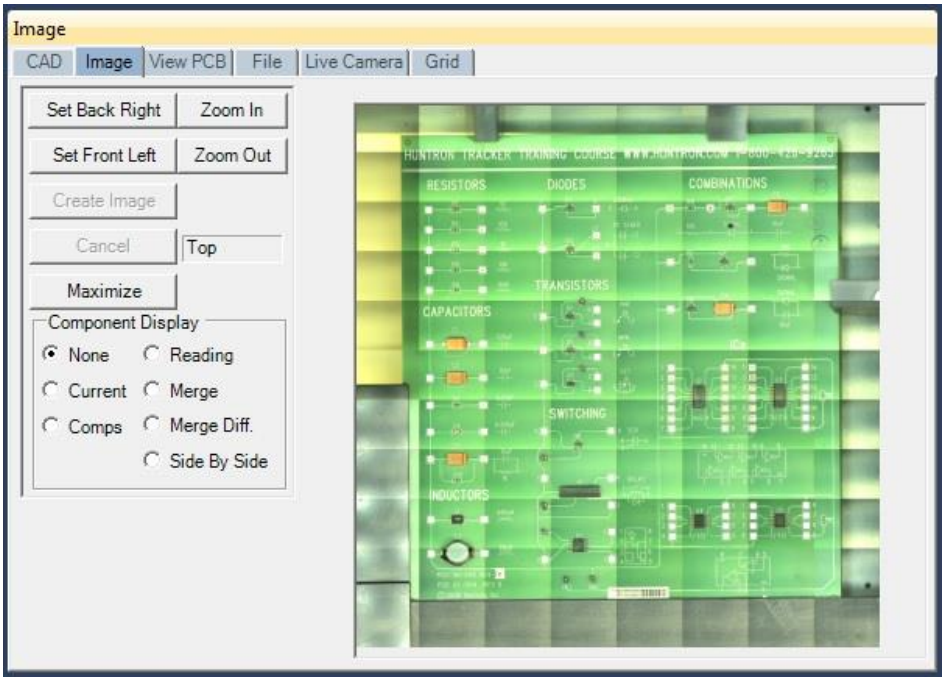
Select and navigate the camera to the second alignment point.

When the camera is set on the second alignment, press the **Set** button. Press the **Save** button to store the alignment points and the alignment point images.

Auto Align Feature

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. Click the **Auto Align button** to start the alignment process. Auto Align may fail to find the alignment points if the lighting, PCB color or camera focus is different than when the original alignment was performed. If this occurs, simply perform the Align manually as described above.

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. We suggest using the Prober/Camera pane rather than the Prober/Align pane or Prober/Teach pane.

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.

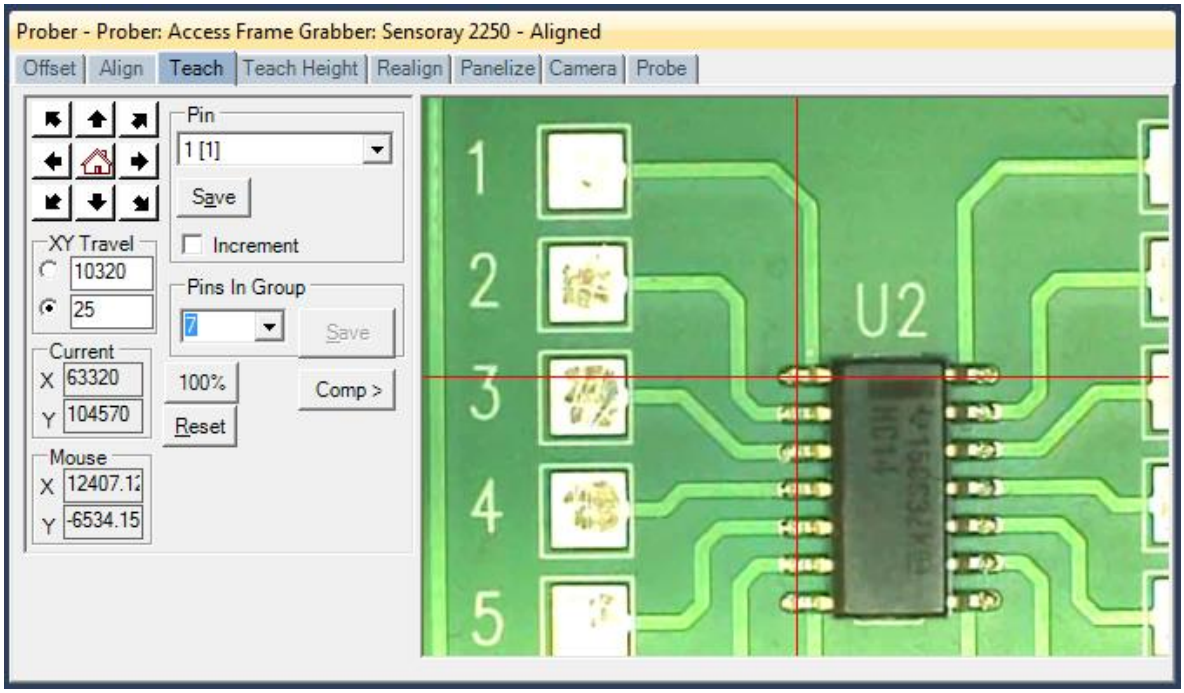
Press the **Set Front Left** button in the **Image/Image Pane**.

The **Create Image** button should now be enabled, press 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. This is especially useful in Teach mode to aide in navigating the camera to component positions.

Right-click on the image to display a menu to **Delete**, **Copy** or **Save As** the board image. The **Component Display buttons** control how component/nets labels overlay on the image. **Note:** If you notice that the image stitching is not precise (overlapped or underlapped images), see **Appendix 1** at the end of this tutorial.

Prober setup – Component Teach; step 1 (DIP package example)

(Access DH Users go to "Component Teach with Access DH")



Use care when teaching component pin locations. This will ensure accurate probing. This example uses a DIP style package to explain the Teach process. Other package types are executed in a very similar fashion.

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

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

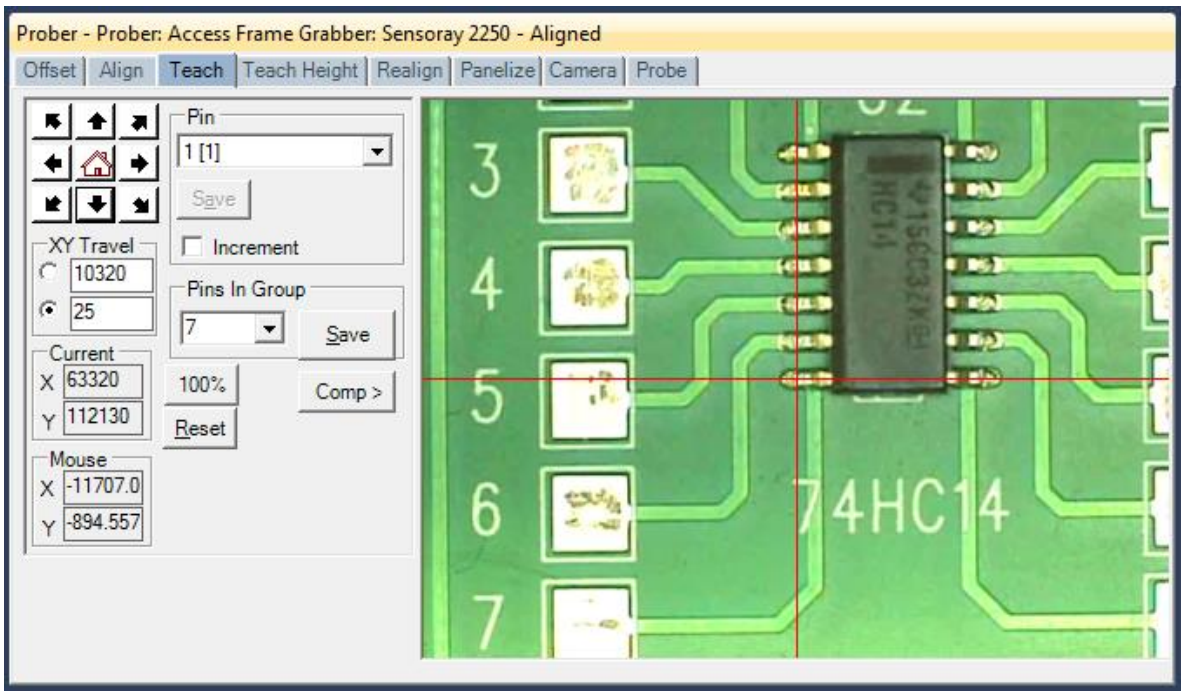
Select **pin 1** in the **Pin** drop menu.

Navigate the camera to the first pin of the selected component (see image above).

When the camera is set on the first pin, press the **Save** button.

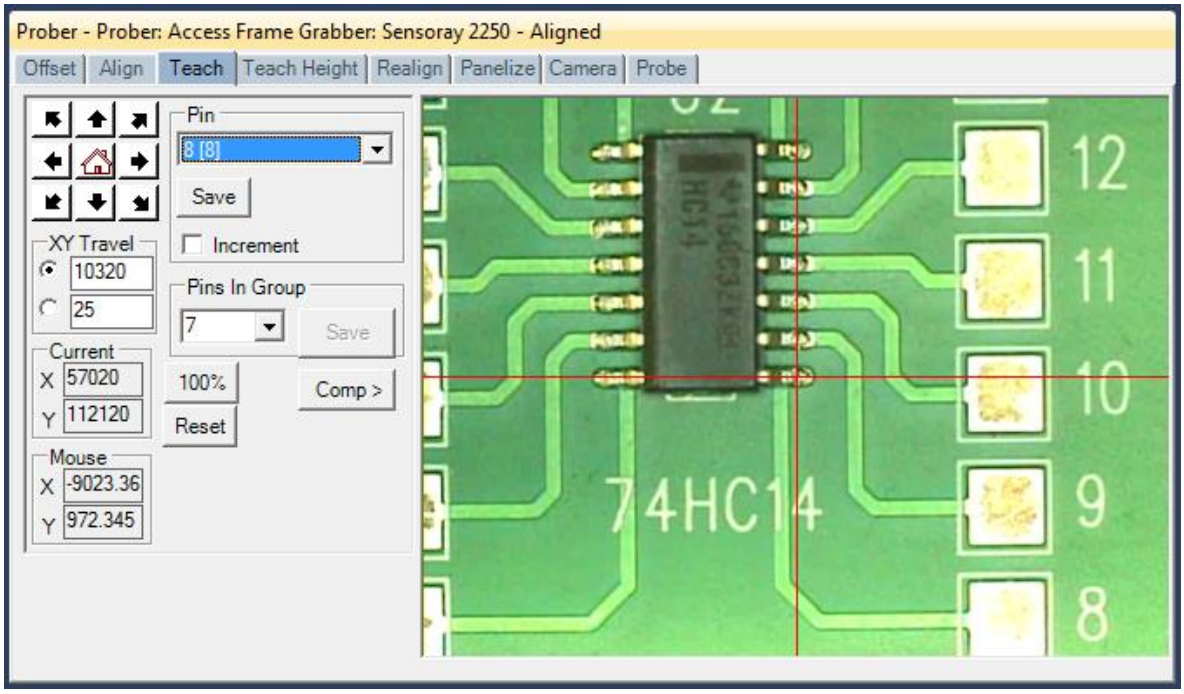
You have two **XY Travel** default fields available underneath the XY travel arrow buttons. It can be useful to set one with a larger travel distance such as 2000 mils and one at a very small distance such as 1 mil.

Prober setup – Component Teach; step 2 (DIP package example)



Navigate the camera to the last pin of the first row (see image above).
Set the number of pins in the row in the **Pins in Group** drop menu (for a DIP package, the number will automatically be set to half the number of pins).
When the camera is set on the last pin, press the **Save** button by the Pins in Group drop menu. This sets and calculates the positions of the remaining pins of the first row.

Prober setup – Component Teach; step 3 (DIP package example)

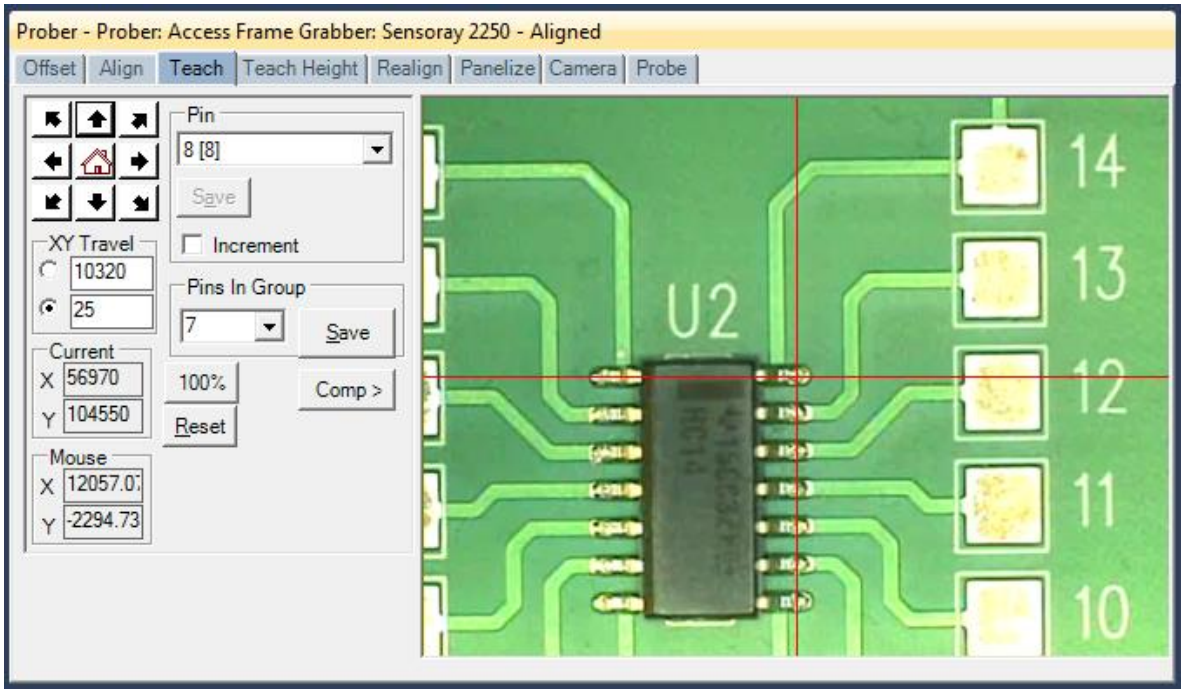


The pin number will automatically increment from the previous step (see above in the Pin drop menu).

Navigate the camera to the first pin of the next row (see image above).

When the camera is set on the first pin of the next row, press the **Save** button.

Prober setup – Component Teach; step 4 (DIP package example)



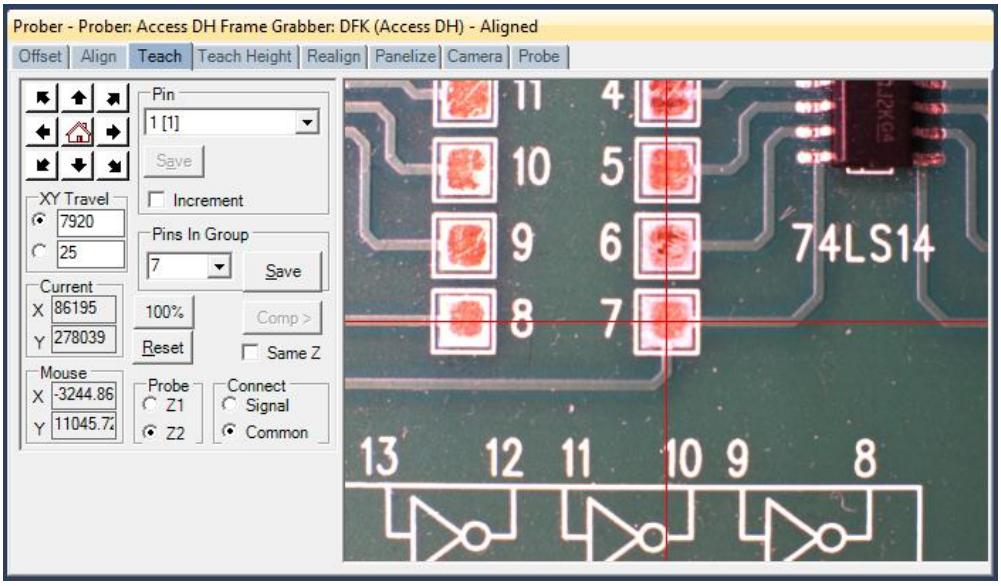
Navigate the camera to the last pin of the row (see image above).
Set the number of pins in the row in the **Pins in Group** drop menu (it will still show the amount selected from step 2).
When the camera is set on the last pin, press the **Save** button by the Group drop menu. This sets and calculates the positions of the remaining pins of this row.

Teach is complete. Make sure to set the Teach Height before scanning the component. Use the **Comp>** button to increment the Tree to the next component. This will allow you to Teach the next component without have to click in the Tree pane.

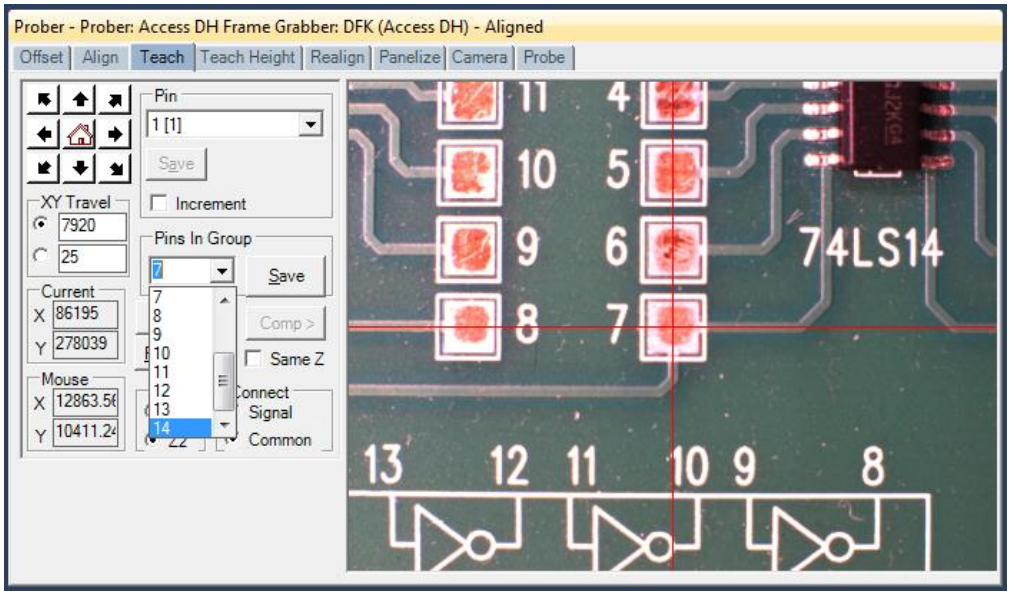
Prober Setup – Component Teach with Access DH

Since the Access DH has two probes, the position where each head will be located during scanning needs to be taught. The process is very similar to a single head Prober. The Z2 probe (typically the Common connected probe) is positioned first followed by the Z1 (Signal connected probe) position. In many cases the Z2 probe is positioned to a single point such as ground and held there while the component pins are scanned by the Z1 probe. **The Z1 camera is used for all component location teaching.**

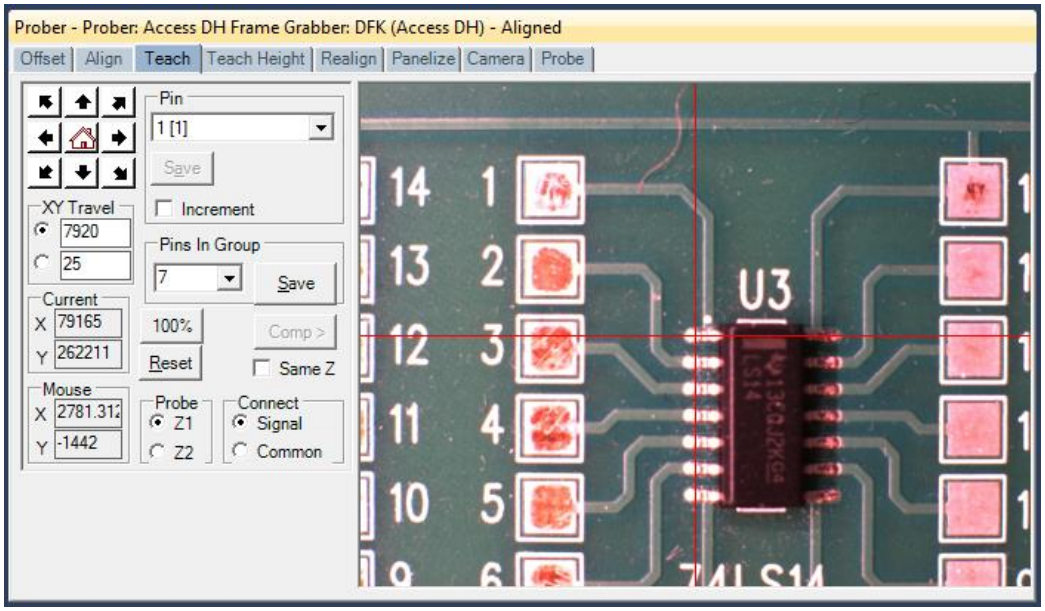
Select the **Teach** tab in the Prober pane. Select pin 1 in the **Pin** dropdown menu.



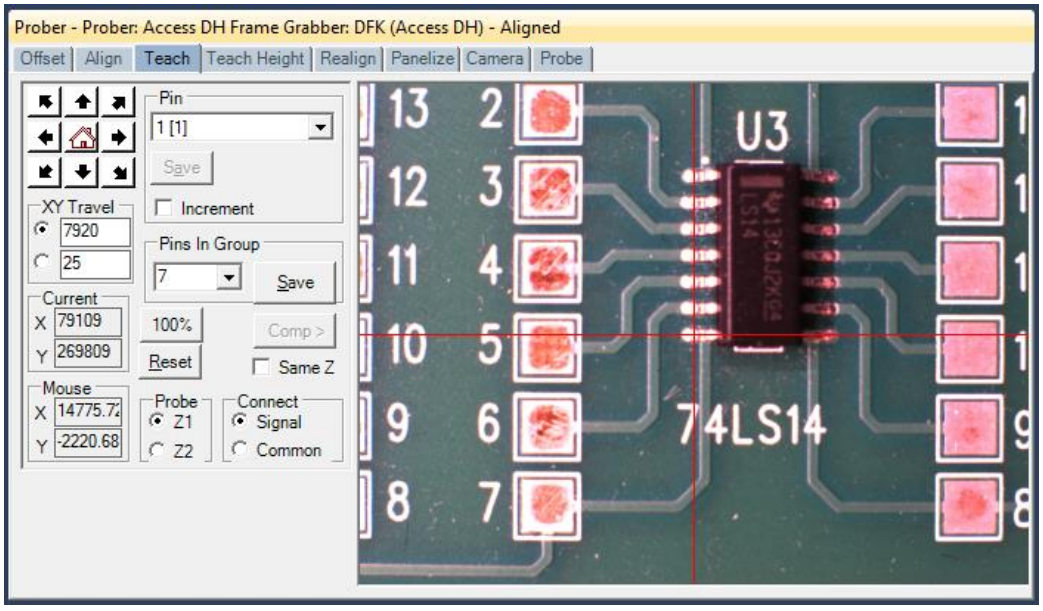
The **Z2** and **Common** buttons should already be selected. Position the camera over the position you want the Z2 probe to probe when pin 1 is probed. It may best to find a point to the **left** of the Z1 head that will not interfere with the position of the Z1 probe. The image above shown a point on the ground trace being used. Click **Save** to set this position. To keep the Z2 probe at this position while the Z1 probe is scanning, select the highest number in the Pins In Group menu (28 for a 28 pin component).



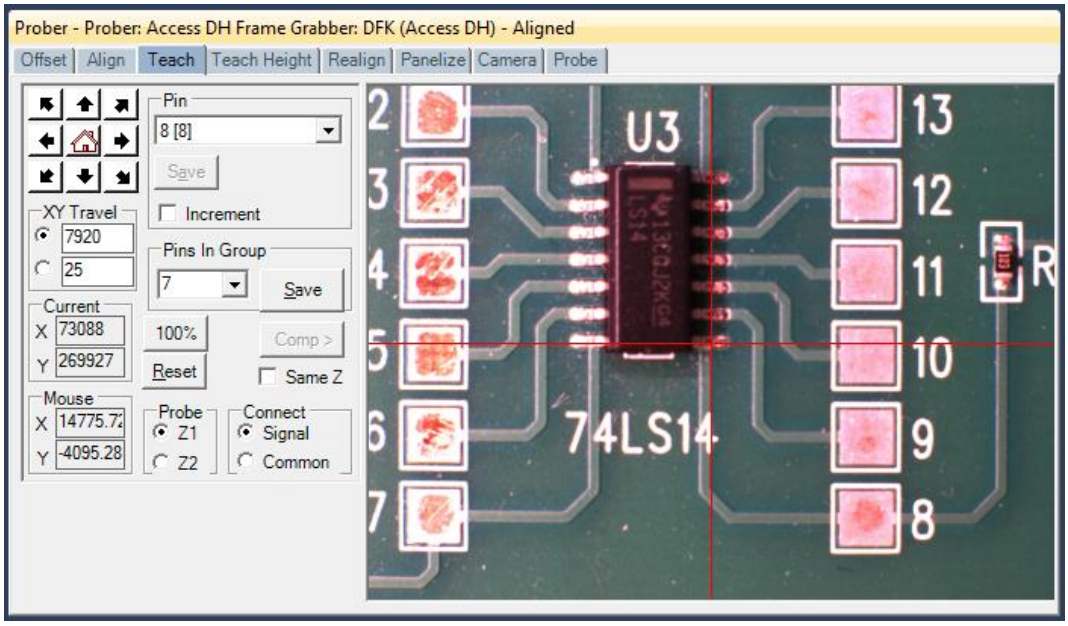
Click the **Save** button for Pins In Group. If you wish to have the Z2 probe change position for each pin scanned by the Z1 head, you will need to teach each pin position individually by selecting the pin number in the Pin menu, moving the camera to the desired position and clicking **Save**. Teaching the position of the Z2 probe can take advantage of the **Pins In Group** function if you wish to speed the process of teaching a row of pins. When the Z2 Teach process is complete, select the **Z1 Button**.



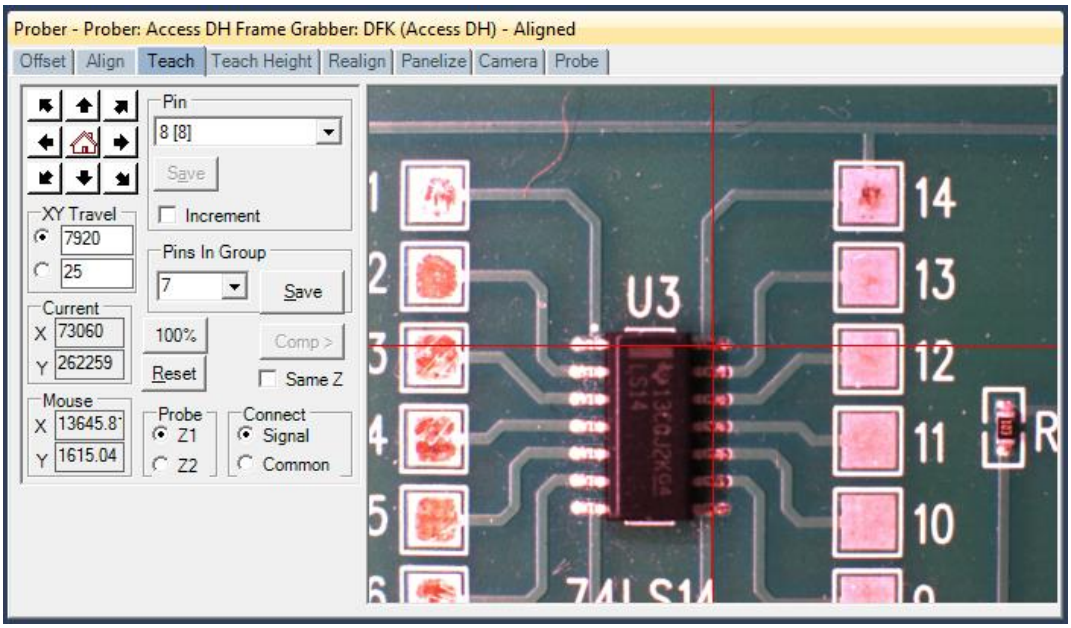
Follow the same procedure used for a single head prober from this point. Select pin 1 from the Pin dropdown menu, position the camera over the pin and click **Save**.
Move to the end of the row of pins, select the number of pins in the row in the **Pins In Group** dropdown menu.



Click **Save** to set the group and move to the beginning of the next row of pins.



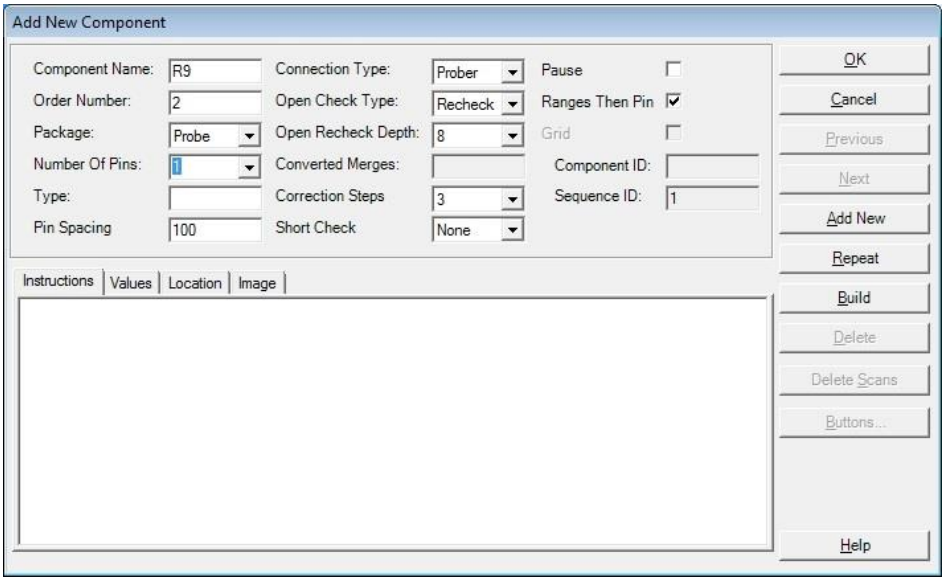
Click the **Save** button to set the beginning of the next row of pins. Move the camera to the last pin of the row.



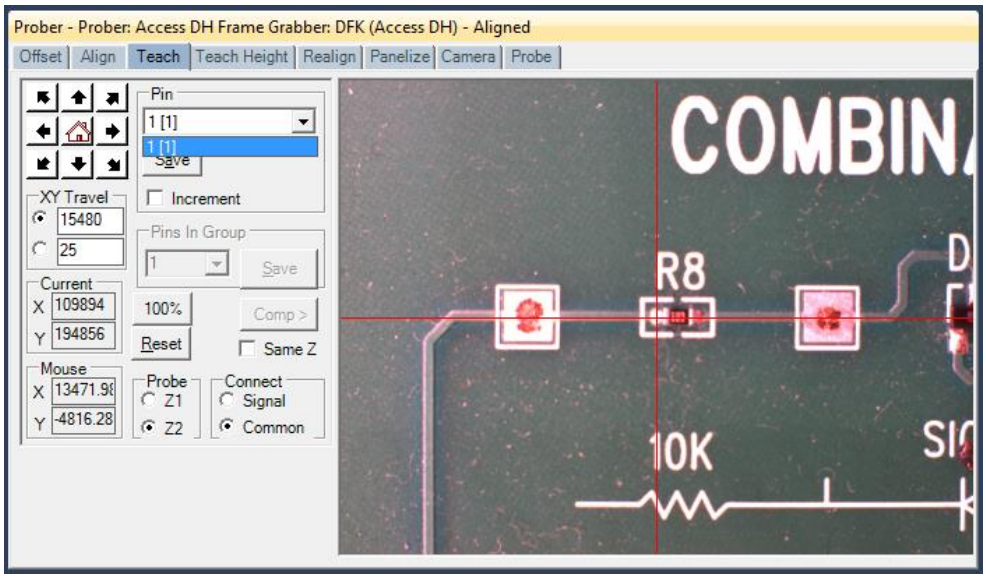
Select the number of pins in the row in the **Pins In Group** menu (it may already be correct) and click **Save** to set the last row of pins. Use the **Comp>** button to select the next component in the Tree.

Prober Setup – Probe Package Component Teach for Access DH

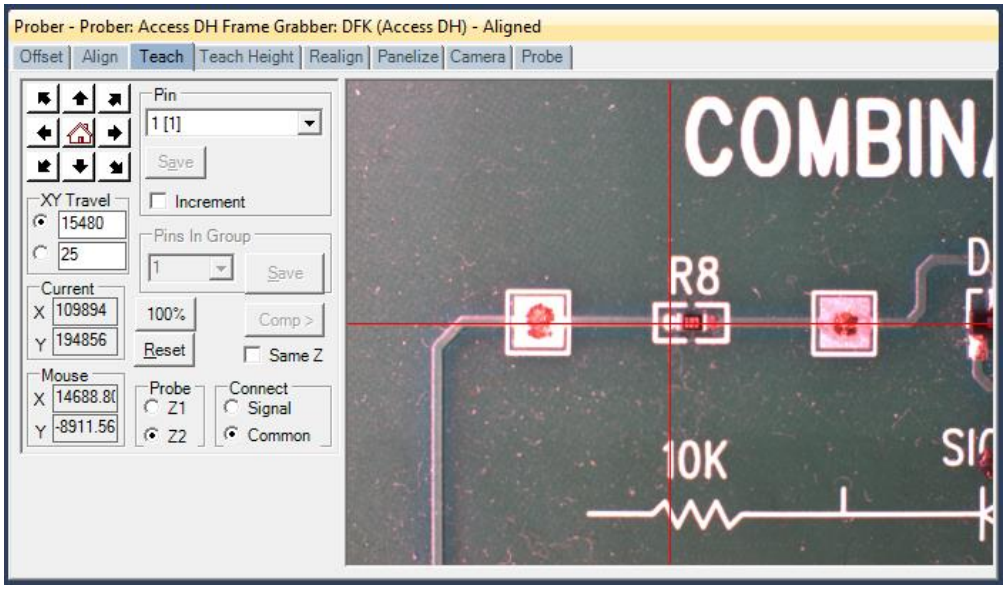
Components that are set to Probe package require that each pin of the component be taught individually. On the Access DH both probes are typically used to probe directly across components such as resistors and capacitors. In these cases the number of pins for the device would be **1** since the Z1 and Z2 probes each have their own pin 1 position. Start by creating a component with **1 pin** and the Package set to **Probe**.



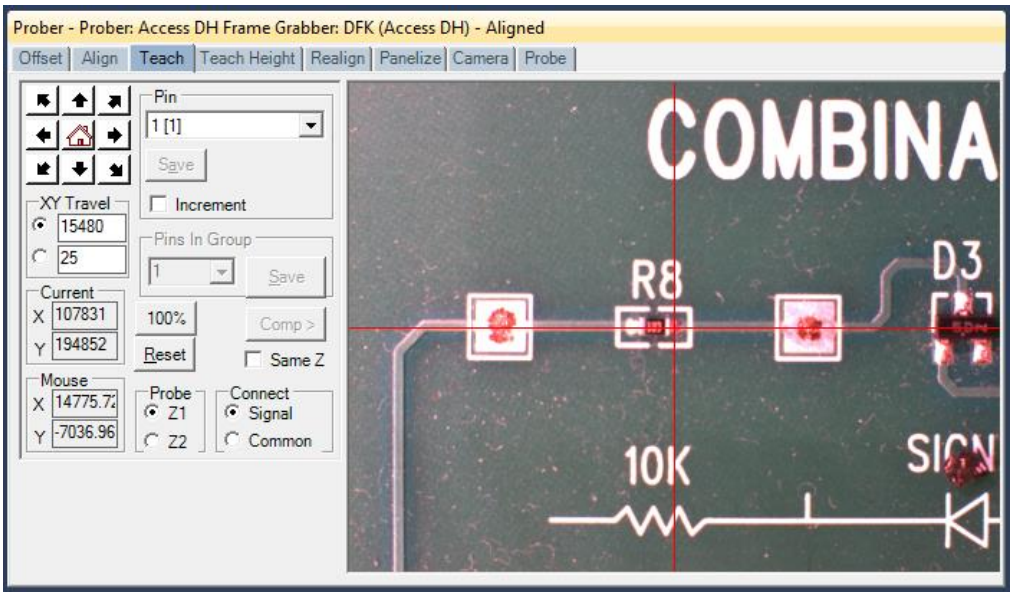
In the Prober pane/Teach tab **select Pin 1** from the drop menu.



Move the Z1 camera over the pin you wish to be used for Z2 (usually Common). Make sure the **Probe** and **Connect** radio buttons are set to **Z2** and **Common** respectively. Click the **Save** button to set the position for Z2.



Note that the **Probe** and **Connect** radio buttons have changed to **Z1** and **Signal**. Move the Z1 camera over the opposite pin and click the **Save** button.



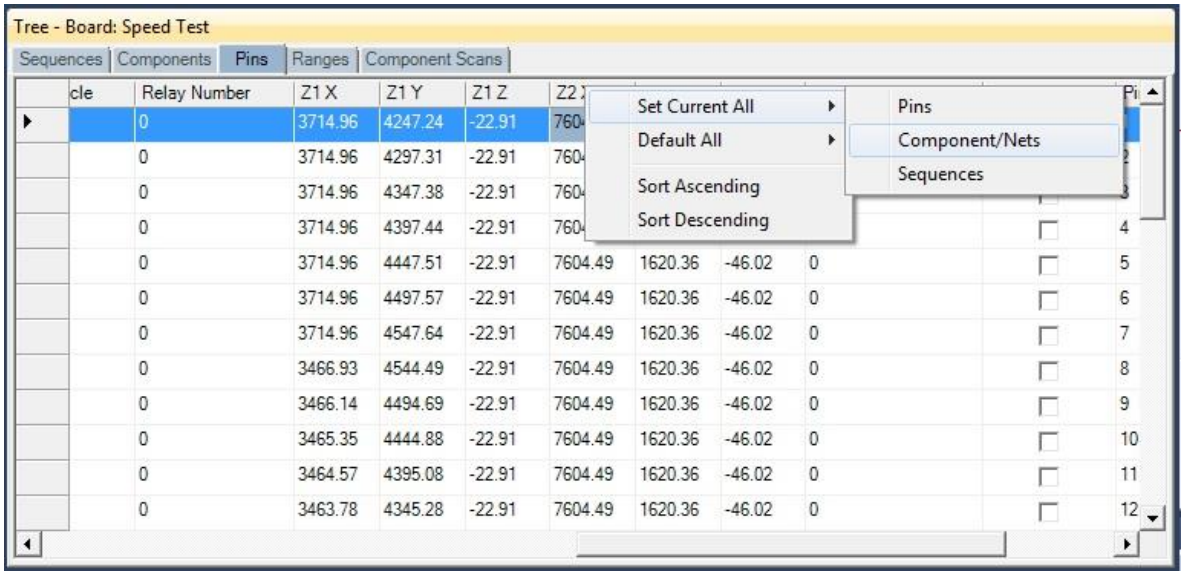
The Teach of this component is now complete.

Duplicating XY Teach Position to Multiple Pins and Components

On occasion it may useful duplicate the XY settings from one pin or component to other pins and components in the Sequence. This particularly useful if you want the Z2 Common position on an Access DH to be the same point for several components (usually ICs where the same Common ground point is used).

Please note that this method is more of a global change to all pins or all components. To make XY Teach Position changes to selected components go to the **Duplicating XY Teach Position for Selected Components** section later in this tutorial.

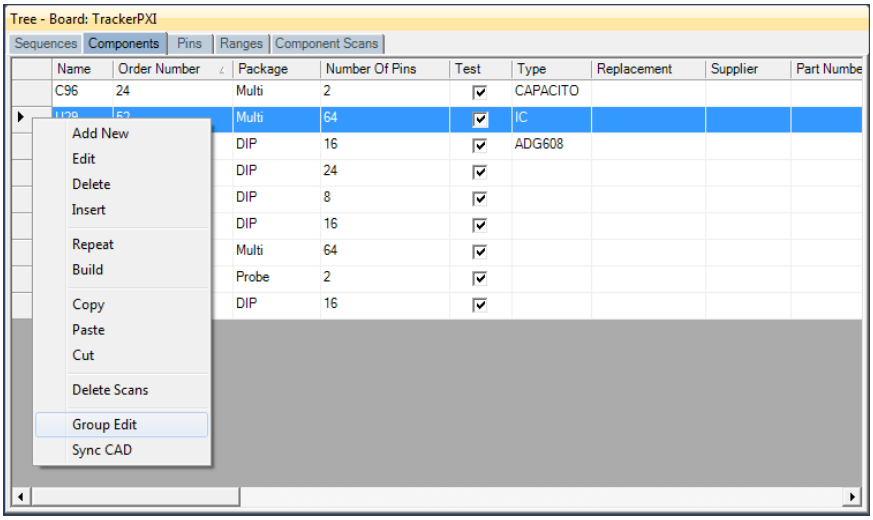
The first step is to teach the Common position to the desired point and set it for all pins of the Component as described earlier. The next step is to select the Pins tab in the Tree pane and scroll to the right until you see the Z2X and Z2Y columns. The XY coordinates in these columns should be the same for all pins. **Right-click** the Z2X column header and select **Set Current All – Component/Nets**. This will set the Z2X coordinate for all components to match the selected component. Do the same for the Z2Y column.



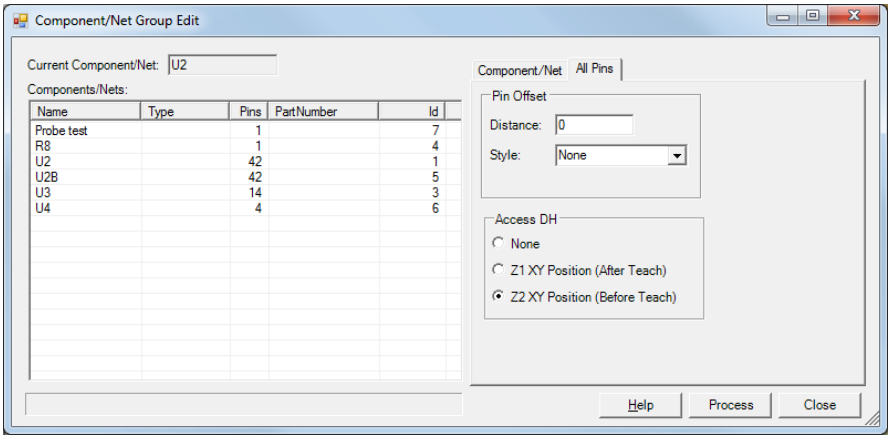
Important Note on Access DH Test Development strategy: Most users will add all of the components that use the same Common first (i.e. ICs), perform the above procedure and then add other components such as transistors and diodes (Probe package components) **AFTER** the Z2X and Z2Y has been modified.

Duplicating XY Teach Position for Selected Components

This feature is for users of the Access DH Prober. It allows you to duplicate the position of the Z1 or Z2 probe from one component to selected components. This is particularly useful for placing a common probe in one location for many components. Setting the Z2 probe position will be used in this example but the same steps apply to using Z1. Keep in mind that teaching the Z1 position takes place **after** teaching Z2. Start by positioning the Z2 probe at your Common position for the component in Teach mode. Save the position for pin 1 then save for all of the pins of the Component using the **Pins in Group Save** button. **Right-click** the row header for the selected Component and select **Group Edit**.

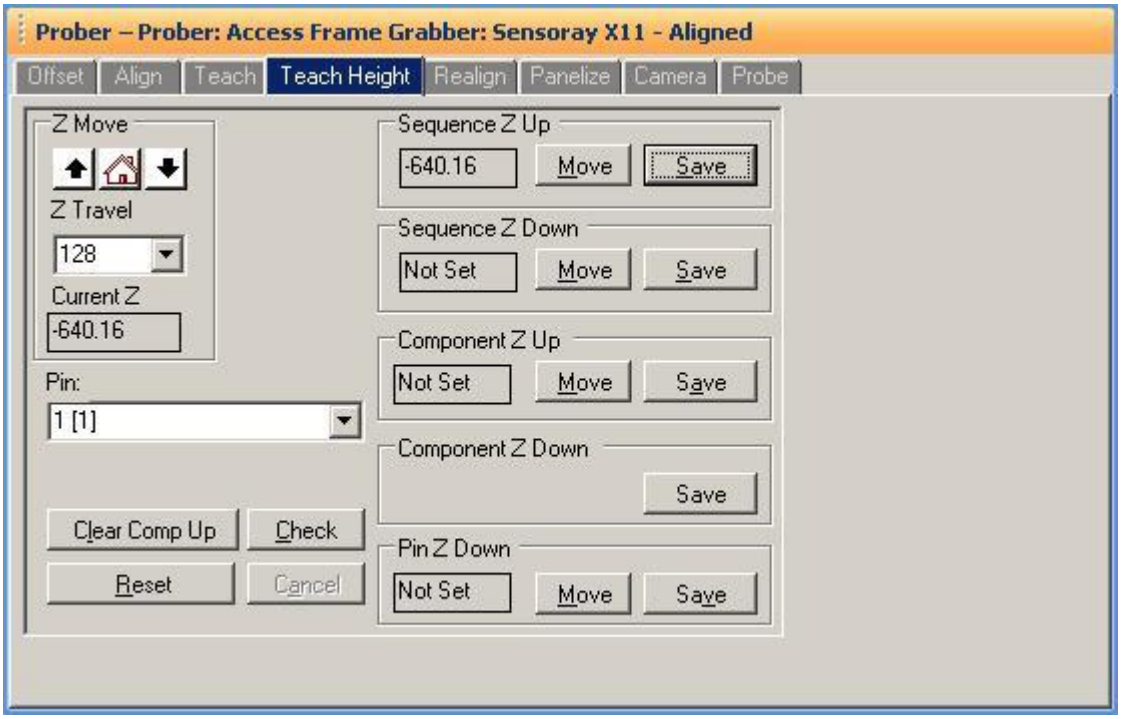


You will now see the Component Group Edit window. Select the All Pins tab and select the **Z2 XY Position** button. Note that it is labeled "**Before Teach**" meaning that this position is taught before Z1 to avoid invalid XY positions.



Select the Components in the list (you can use **Shift** or **CTRL** keys to select groups or individual components) that you want to receive the duplicate Z2 coordinates. When you click **Process**, the selected Components will have their Z2 XY positions set to the same XY coordinates as the selected Component.
The process is the same for duplicating Z1 positions except that you must teach the Z1 positions **AFTER** setting the positions for Z2.

Prober setup – Component Teach Height; step 1 (Access DH users go to "Teach Height for Access DH")



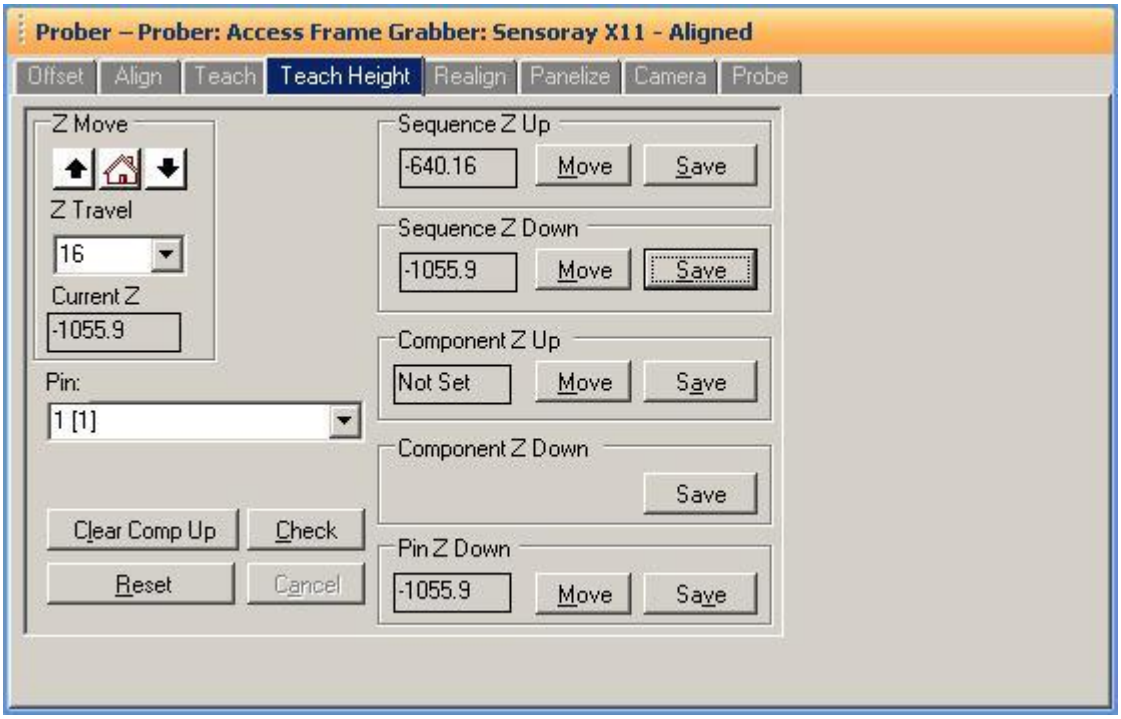
Select the **Teach Height** tab in the Prober Pane.

Select a Pin number in the **Pin** drop menu. This will position the probe tip over the selected component pin.

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. This is the position the probe will lift to while probing a component. Click the **Save** button in the **Sequence Z Up** section of the window.

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.

Prober setup – Component Teach Height; step 2

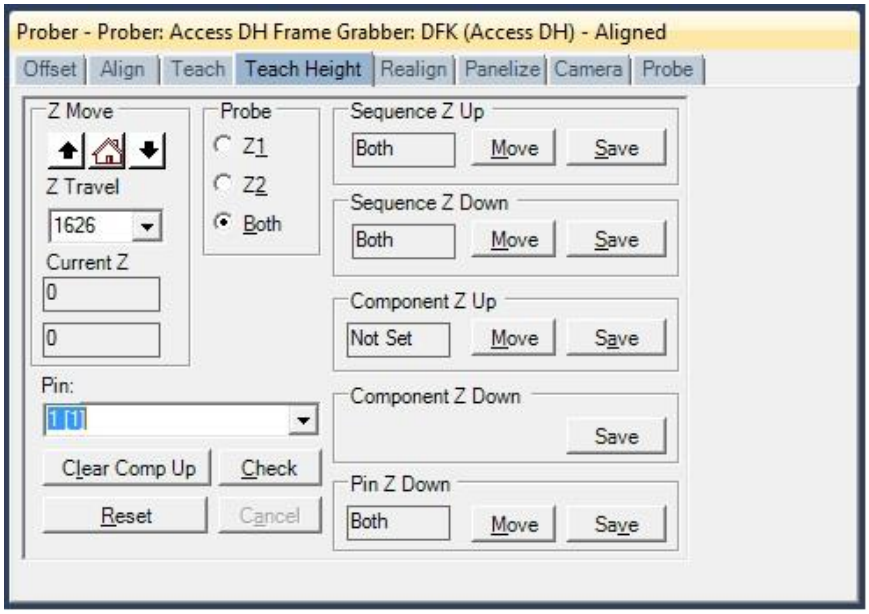


The next step is to set the probe down position. Access USB Prober users may find it useful to use the Probe Tip camera in setting the probe down position. Select the **Live Camera tab** in the Image pane then click the Setup button to select a video source. Select the correct video source and click the **Start** button. A live image will appear in the Live Camera tab. Do not forget to **Stop** the Live Camera when it is not needed.
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 using. In general use, this is not necessary as the Down level is set at the Sequence level.
To save on PC resources, **Stop** the Live Camera when it is not needed.

Teach Height for Access DH

The process of teaching the Z axis position on an Access DH is very similar to that of a single head Access Prober.

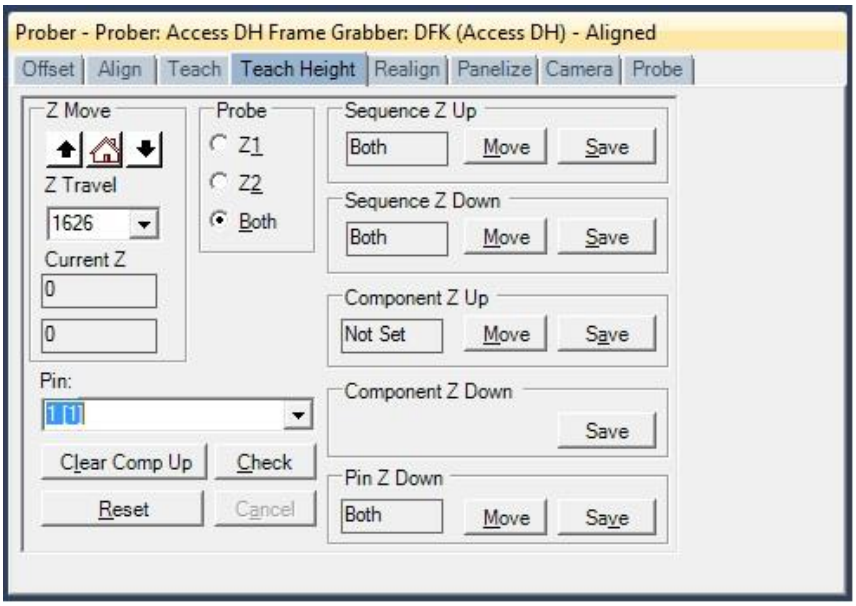
Select the **Teach Height** tab in the Prober pane.



Select the **Z1** radio button. Select a pin in the Pin menu. The Probe head will move to put the probe tip over the selected pin. Use the **Z Move** buttons to lower the Z1 axis probe to the desired Sequence Up level. Make sure this level is high enough to clear tall components. This is the position the probe will lift to while probing a component. Click the **Save** button in the **Sequence Z Up** section of the window. This will set the Up position for **both** Z axis heads.

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. This will set **both** Z axis heads.

The next step is to set the probe down position.

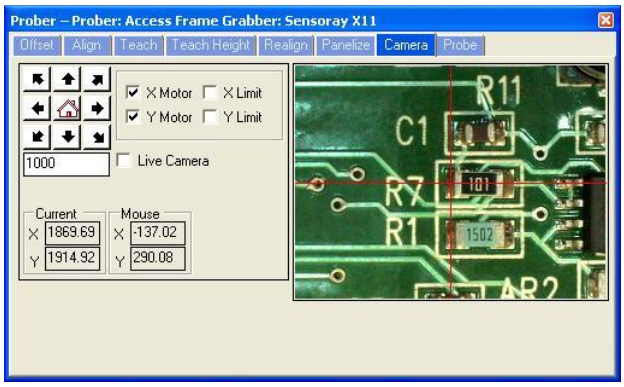


Use the **Z Move** buttons to lower the probe tip to make contact with the component pin or pad. You may want to lower the probe in small increments once contact is made to slightly compress the spring probe and ensure good contact. Do this by changing the Z Travel distance setting. The signature of 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.

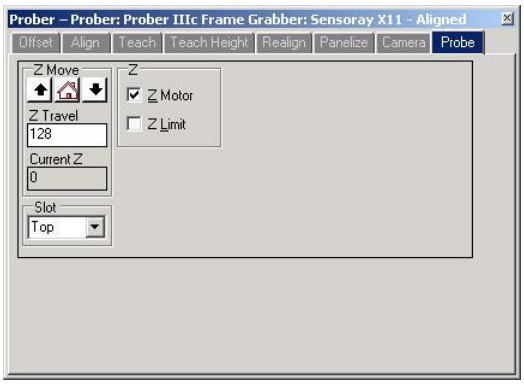
If needed, you can also set **Component Z Down** and **Pin Z Down** levels for individual components and pins using. **Pin Z Down** settings must be set for both heads using the **Z1** and **Z2** radiio buttons. In general use, this is not necessary as the Down level is set at the Sequence level.

Manual Modes – Camera and Prober



Use the XY buttons or click in the camera image to move the prober to a point.

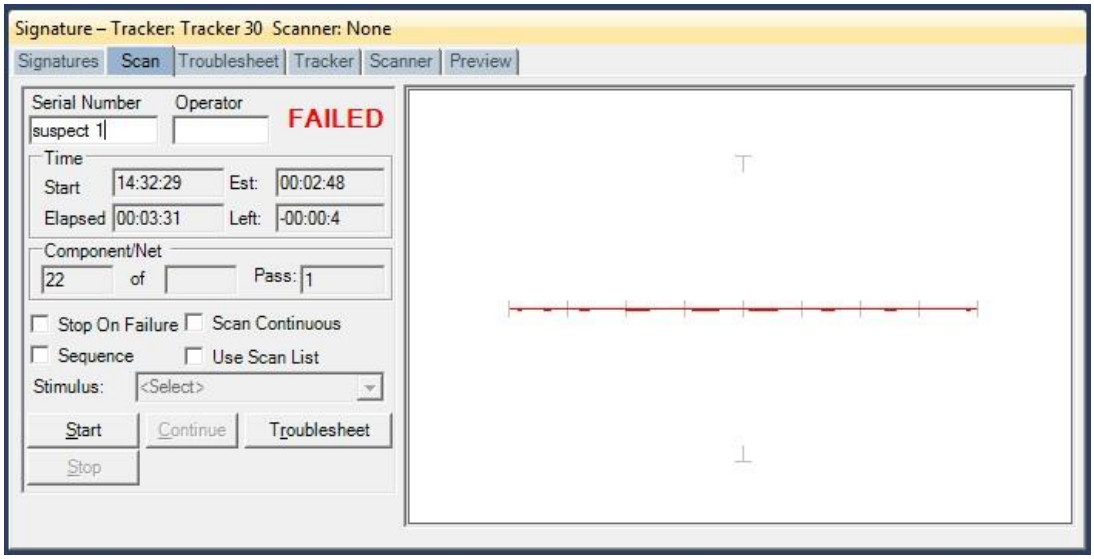
Use the Z Move buttons to lower the probe to the point targeted by the camera.



These panes in the Prober Pane are used in conjunction with the Tracker mode (Signature/Tracker pane) to drive the camera to a point on the board and put the Prober tip on the point. The signature at the point will be displayed and controlled in the Signatures/Tracker pane.

NOTE: Camera Offset and board Alignment must be performed prior to using the Camera and Tracker mode together. This ensures the probe drops to the point indicated in the Camera image.

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** if you wish to scan the selected Sequence.

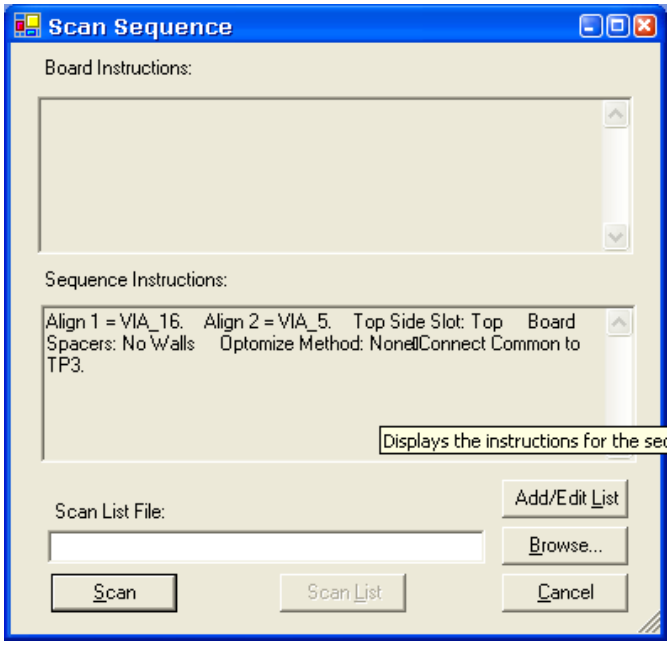
The **Use Scan List** check box will add the ability to use a Scan List when the Scan Sequence dialog appears (see Creating a Scan List section).

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

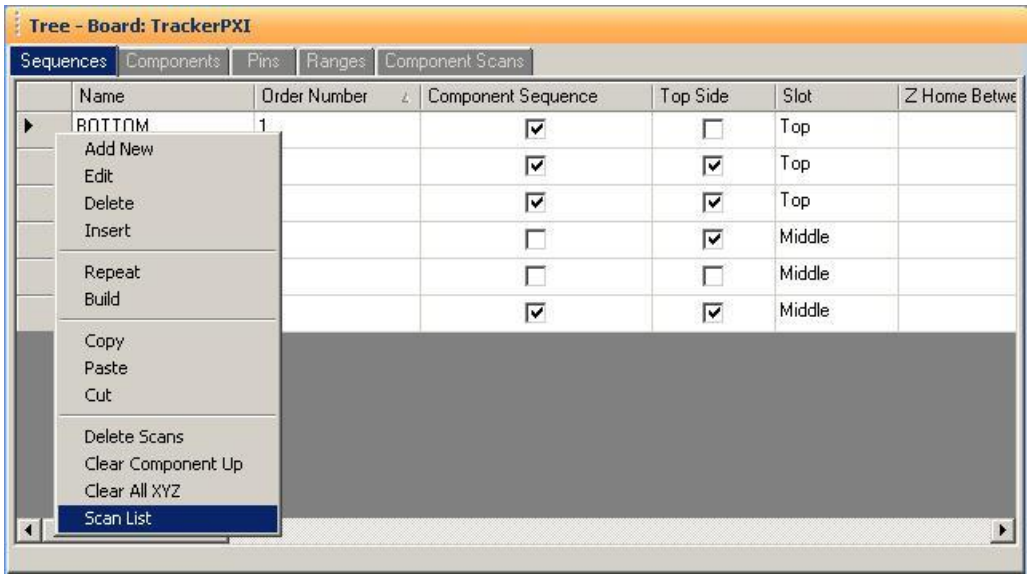
The test instructions prompt will be displayed. Clicking OK will start the scan for probers. Tests using manual probes or a Scanner will prompt the user to connect to the appropriate pin or component.

Creating a Scan List

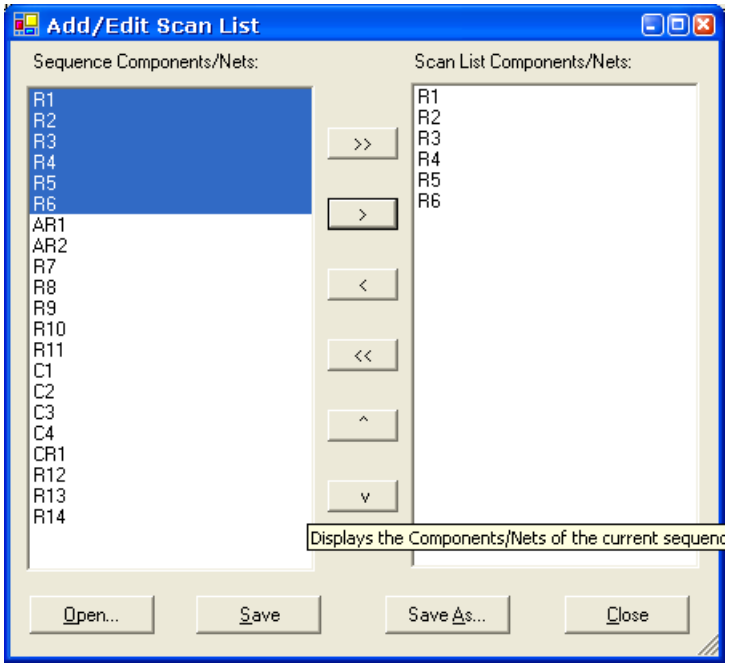
At this point, you have the ability to create a Scan List. A Scan List is simply a list of components to be scanned saved to a text file. To create a Scan List, check the Sequence checkbox and check the **Use Scan List** checkbox in the Signature pane/Scans tab. Click the **Start** button and click the **Add/Edit List** button in the Scan instructions dialog.



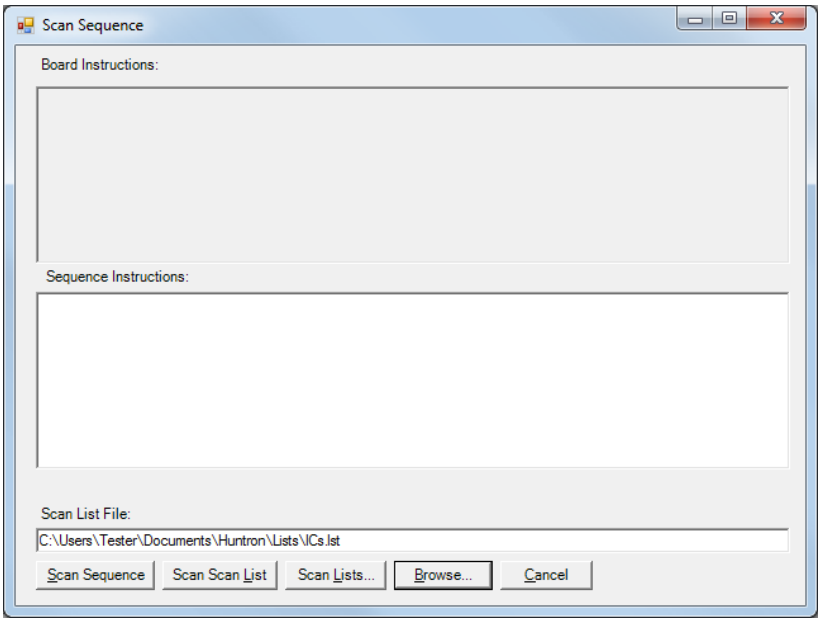
You can also open the Scan List editor by right-click the Sequence row header and selecting **Scan List** (see image).



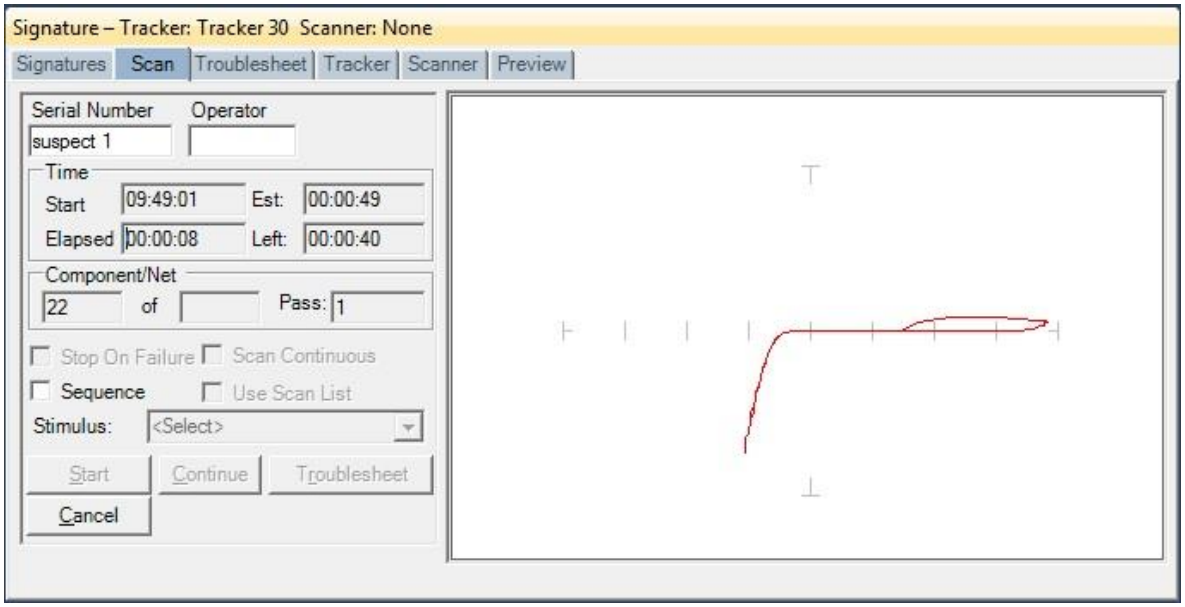
A new dialog will open called Add/Edit Scan list.



To use the Scan List, select components on the left side of the window (you can hold the **Ctrl** or **Shift** keys to assist in selecting groups of components) and click the arrow buttons in the center of the window to add items to the list. Use the UP or DOWN arrow buttons to move a selected component up or down the scan list. When the desired list is assembled, click **Save As...** to save a new Scan List. Once saved, you can select the Scan List in the Scan instructions dialog by selecting the **Browse** button and selecting the desired list.



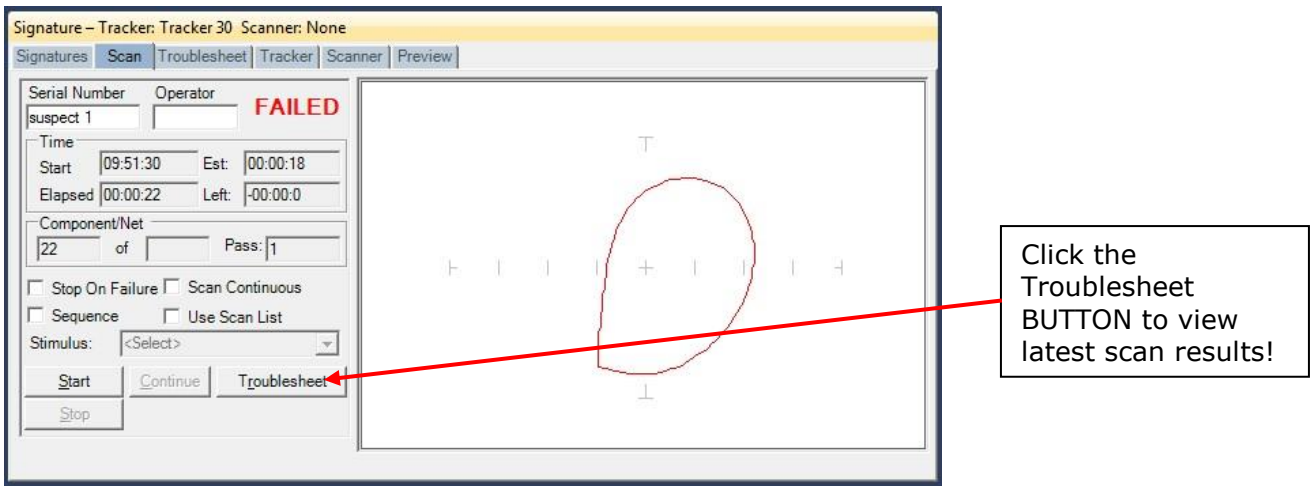
Click **Scan Sequence** to start scanning the Sequence or Component selected in the Tree pane or click **Scan Scan List** to scan the selected Scan List File.



Signatures will be shown in the signature window while scanning. Component name, pin and ranges are displayed in the status bar at the bottom of the main window.

To stop a scan, press the **Cancel** button.
To continue a stopped scan, press the **Continue** button (when enabled).

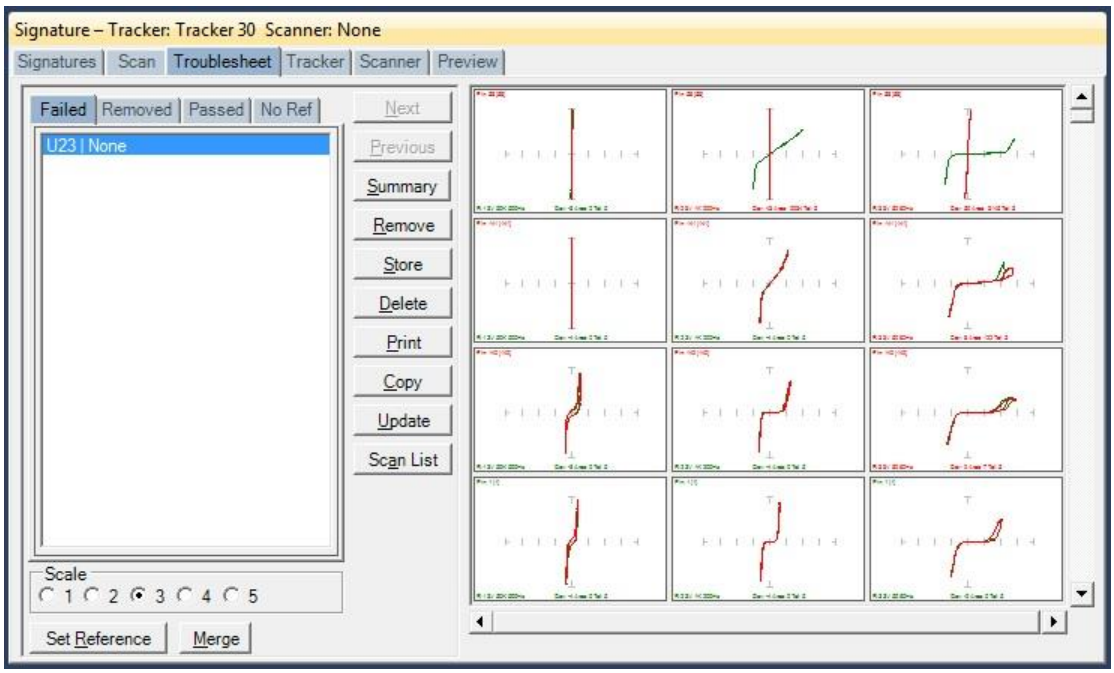
Scan Results



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

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

Viewing Signatures – Troublesheet



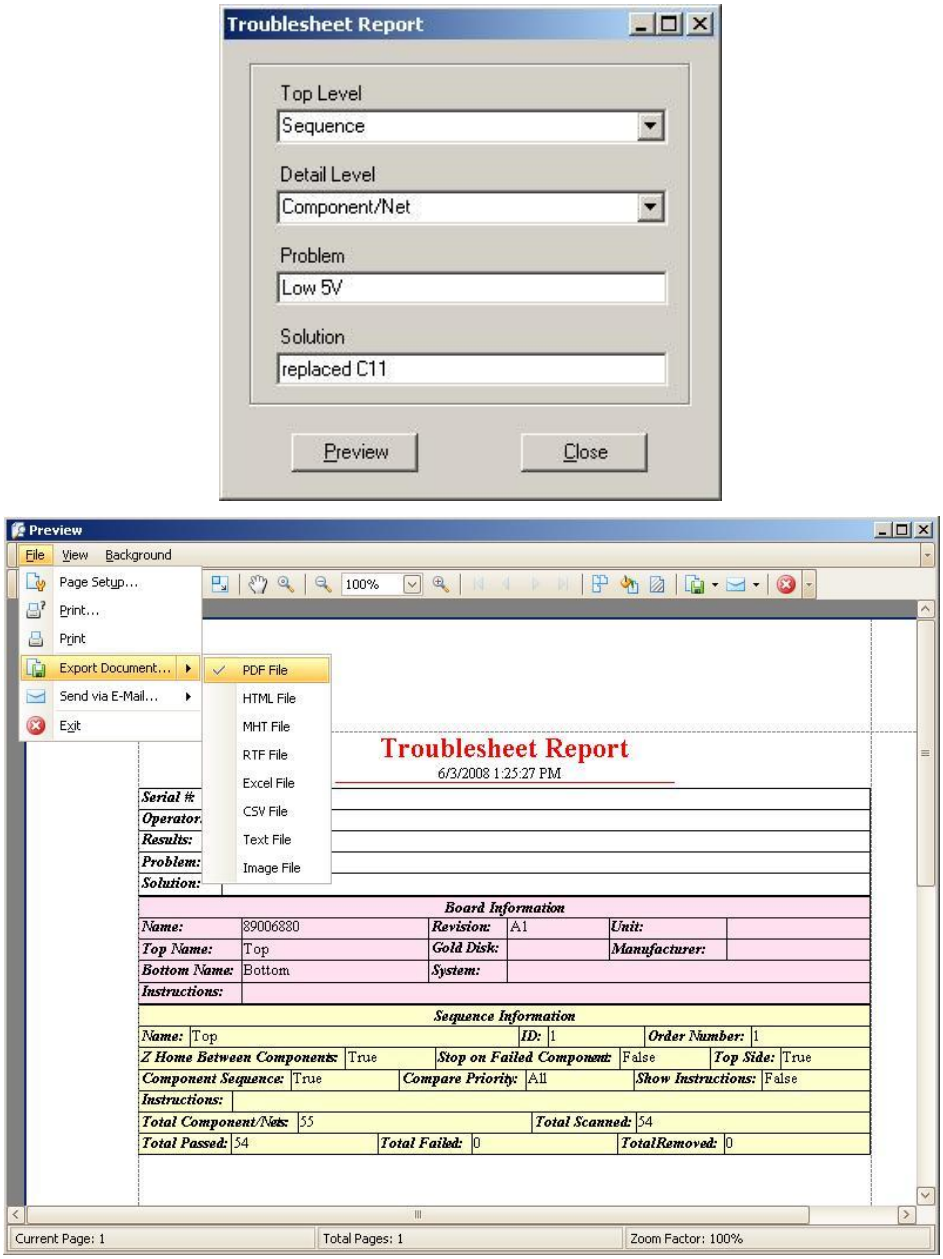
To view the Troublesheet, press the **Troublesheet button** in the Scan Results window. This will display the signatures that compare differently from the Reference signatures.

When a scan is executed signatures are compared against the Reference set (or sets) if it exists. On the first scan, the signatures will be displayed in red in the **No Ref tab**. This occurs because the first scan has no reference to compare against. 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 signatures are displayed in contrasting colors. By default, the Reference signature is green and the failed signature is red.

Use the **vertical scroll bar** to scroll through the scanned pins. Use the **horizontal scroll bar** to scroll the pin ranges especially if the viewing is set to **Pin** mode (see Viewing Signatures – Right Click section).

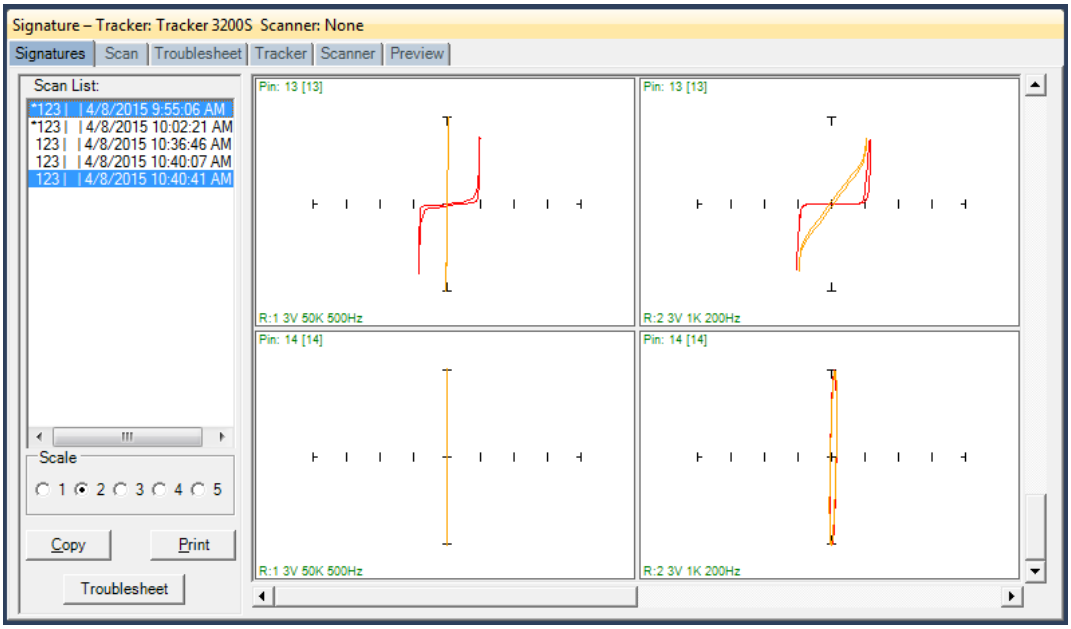
To add the Troublesheet signatures to the Merge signature set, press the **Merge** button (do this only with known good signatures that vary slightly from the original Merge signature set).

Viewing Signatures Troublesheet Report



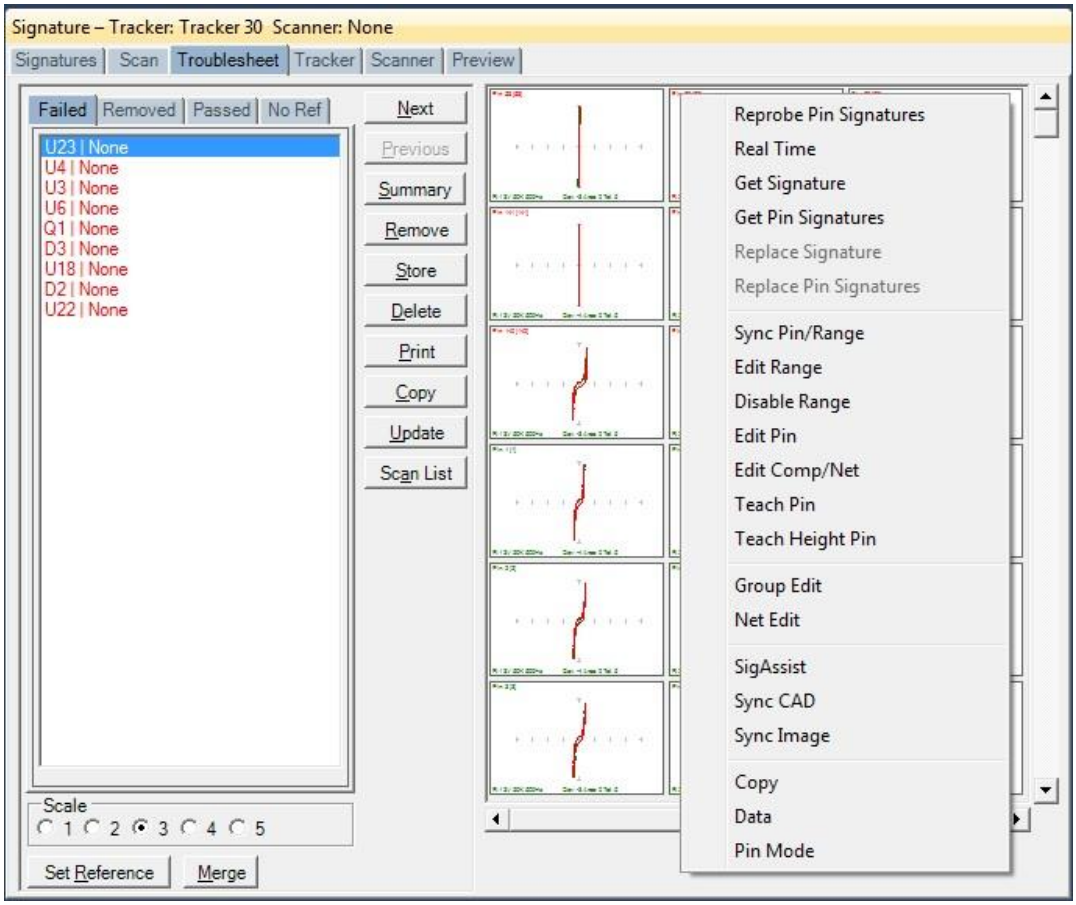
To view and print the Troublesheet report, press the **Print** button in the Troublesheet window. You can select from several options including a Sequence or Component Level report, the level of detail and add problem and solution comments to the report. Click **Preview** to view the report on-screen. 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 of the component selected in the Tree pane/Components tab, select the **Signatures** tab in the Signature Pane. 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. The **Troublesheet** button will switch to the Troublesheet tab and display the Troublesheet of the scan selected in the Scan List. You can use the keyboard **Shift** and **Ctrl** keys to select and stack signatures in the signature boxes (shown above). The signatures will be displayed in contrasting colors and can be very useful if you want to make general comparisons among different scans.

Viewing Signatures – Right Click



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

Reprobe Pin Signatures: Capture signatures for all displayed ranges and prompts to replace signatures immediately – available in Range mode only

Real Time: Display signature in real time

Get Signature: Capture signature of the pin/range that was right-clicked

Get Pin Signatures: Captures signature for all ranges – works only in Range Mode

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

Replace Pin Signatures: Replaces current signatures with the captured 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 (set in the Tree pane, Ranges tab)

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

Edit Comp/Net: Shortcut to the selected Component Edit dialog in the Tree pane

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

SigAssist: Displays the SigAssist window showing calculated approximate values of resistance, capacitance, breakdown voltage, etc. based on signature information

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.

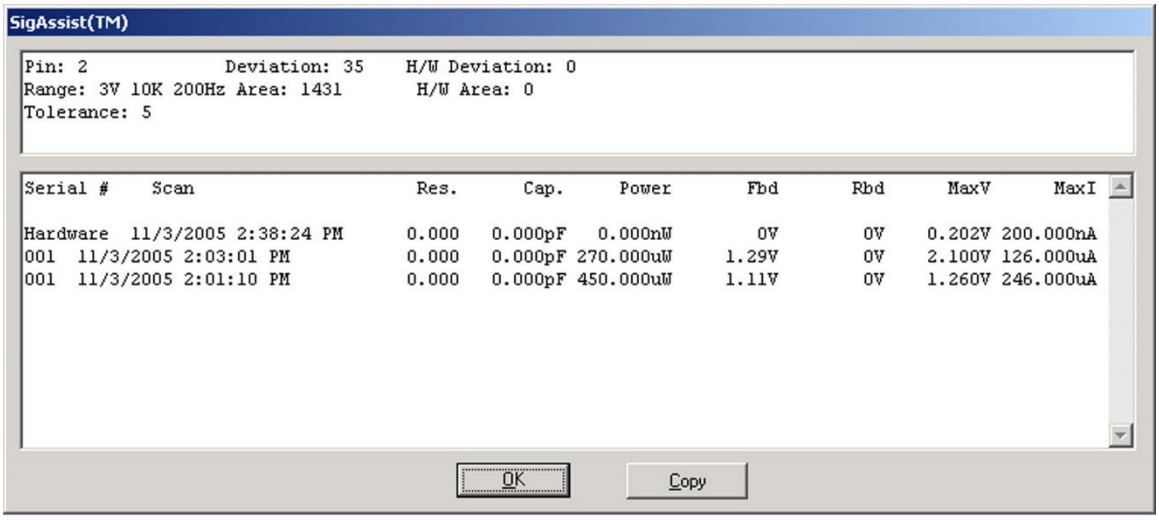
Sync Image: Links current Troublesheet to the image (image created by Access Prober) in the Image pane/Image tab

Copy: Places the signature image in the clipboard

Data: Displays the signatures numeric data

Range/Pin Mode: Displays signatures with Range priority or Pin priority - see default in Tools/Options)

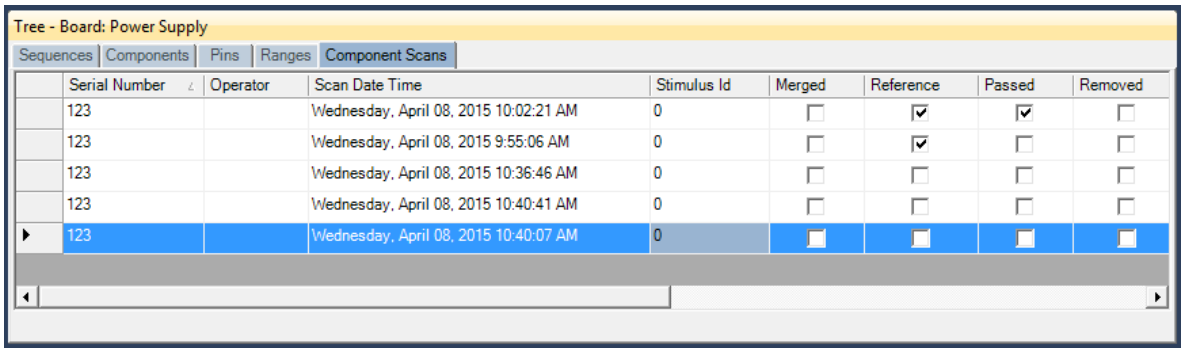
View SigAssist Information



The SigAssist window can be viewed by right-clicking on a displayed signature and selecting SigAssist. It appears as an additional pop-up window. The information presented relates to the individual scans displayed in the signature. The calculated resistance, capacitance, power, forward breakdown voltage (Fbd), reverse breakdown voltage (Rbd), maximum voltage and maximum current are displayed. These values

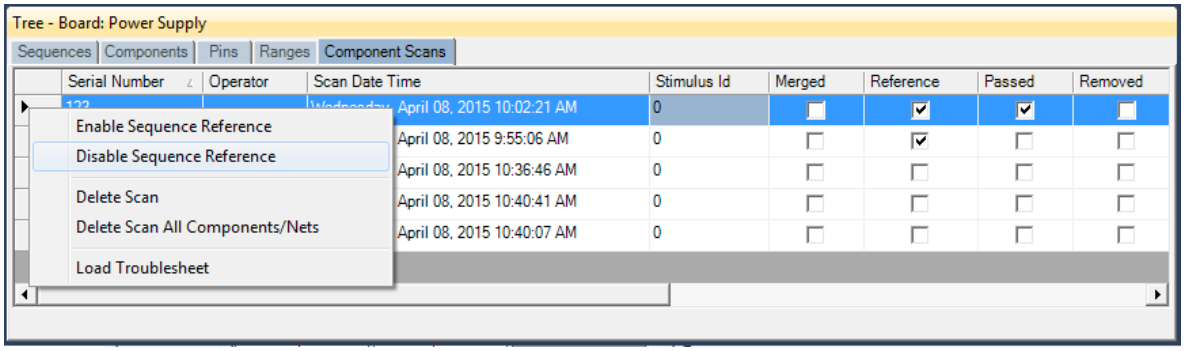
are for troubleshooting purposes only and are not intended to be used as true measurements. The accuracy of the SigAssist values depends greatly on the scanned signature information. SigAssist values may not be calculated for signatures that do not contain enough information for reasonable calculations to take place (i.e. opens, shorts). The Copy button will put a copy of the window into the clipboard for use in other programs.

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)



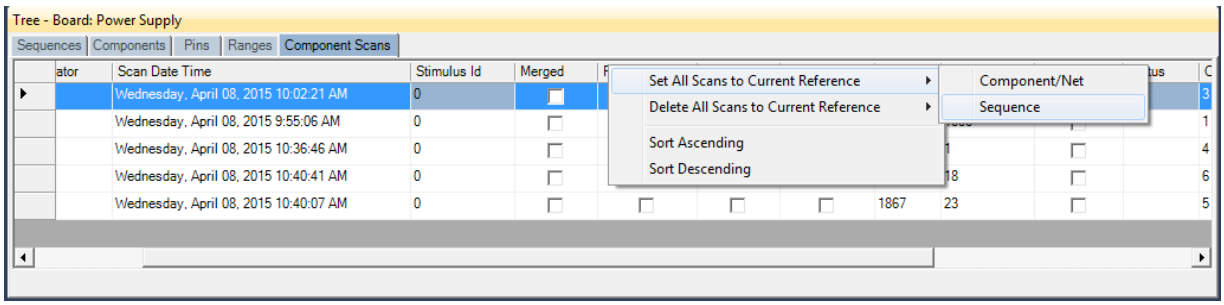
Right clicking on the Row bar of the Component Scans pane displays an additional menu for performing specific tasks such as enabling/disabling References, deleting a scan for the selected components or all components and loading a Troublesheet.

Enable Sequence Reference will mark the scan as a Reference for the entire Sequence. **Disable Sequence Scan** will disable the selected Sequence as a reference.

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 Signatures/Troublesheet pane.

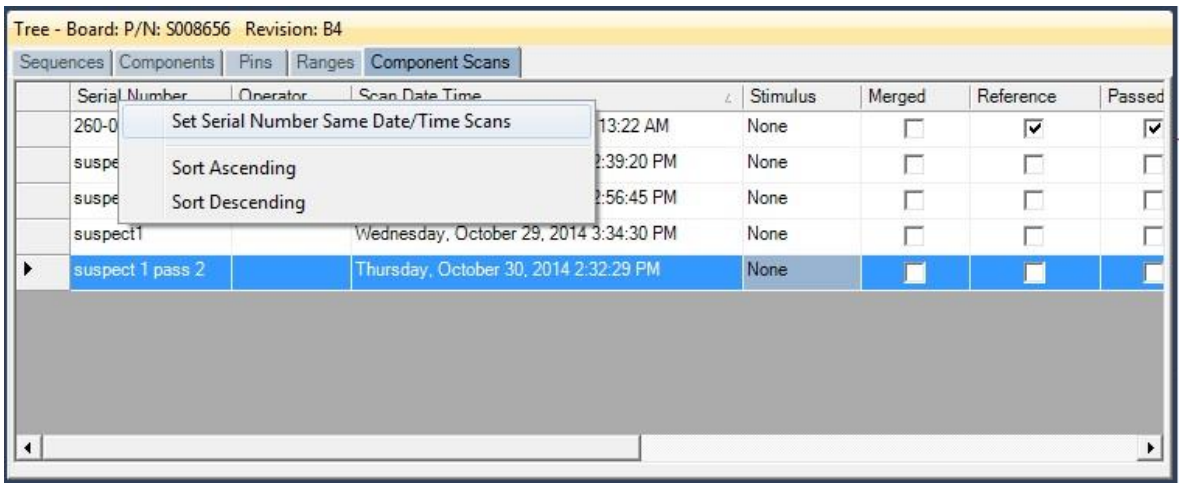
Component Scans – Right Clicking (Auxiliary menus)



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.

This method is the easiest way to delete non-reference scans that are listed in the Tree/Component Scans pane. Select a scan that is a non-reference (the Reference check box is unchecked) the right-click the column as shown above. Select **Delete All Scans to Current Reference/Component/Net** or **Sequence** to delete the non-reference scans for the individual Component/Net or for the entire Sequence.

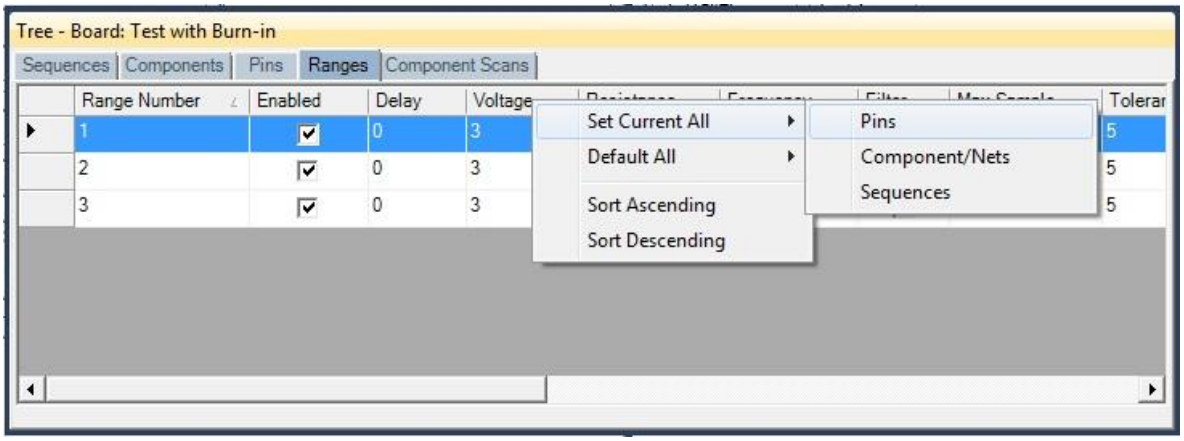
Component Scans – Right Clicking (Auxiliary menus)



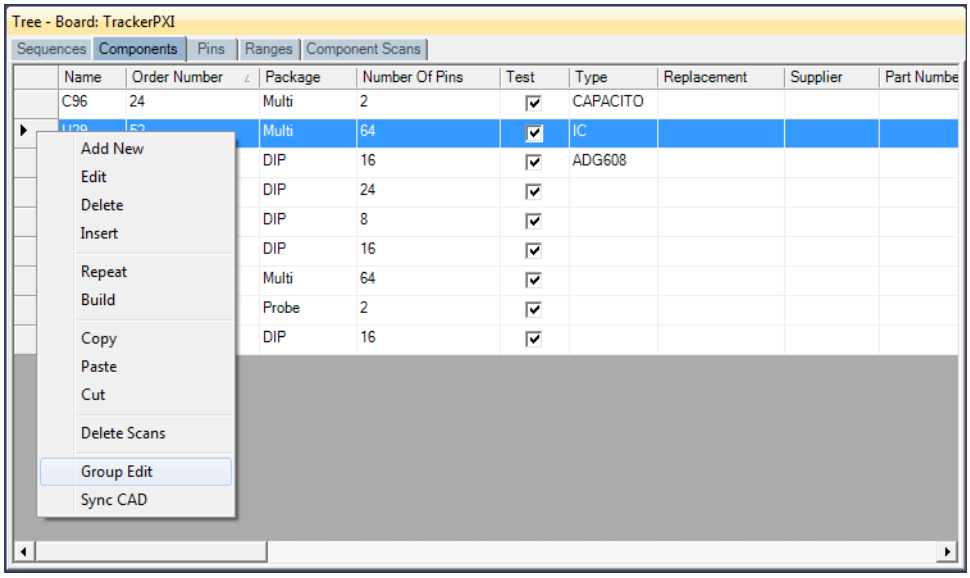
Right clicking on the Column bar of the Component Scans pane displays an additional menu for performing specific tasks such as sorting or globally deleting items. In the image above, the Serial Number column header was right-clicked.

Range Editing – Right Clicking (Auxiliary menus)

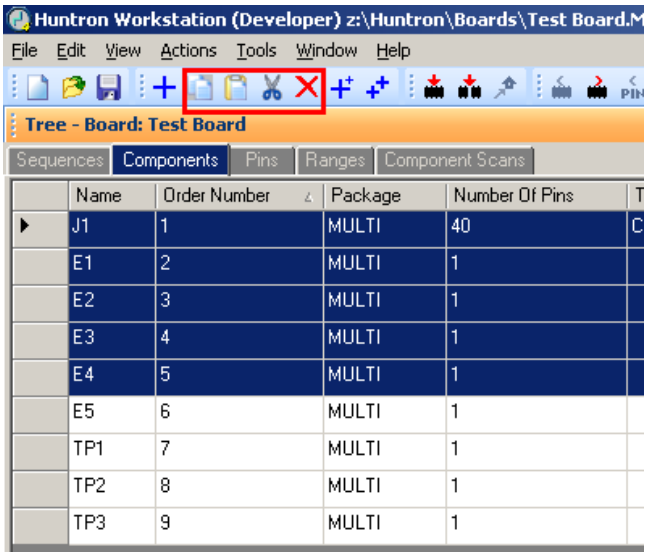
It is common to edit Component Range settings directly in the Tree pane/Range tab. Be aware that making a change in the grid only changes the setting for the selected **Pin** and **Range**. To make your changes global for the Component Pins, all Components or all Sequences, right-click the column header and select **Set Current All – (Pins, Components or Sequences)**.



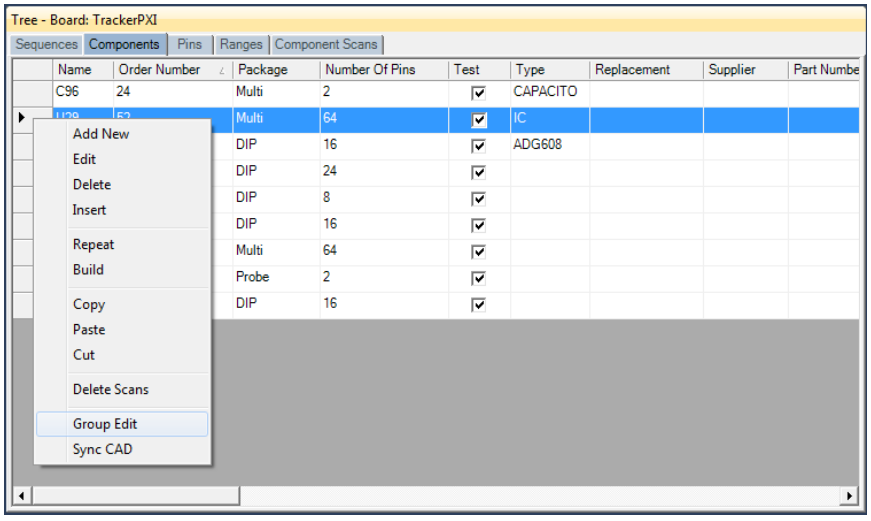
Sequence/Component/Pin Editing – Right Click menus



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.

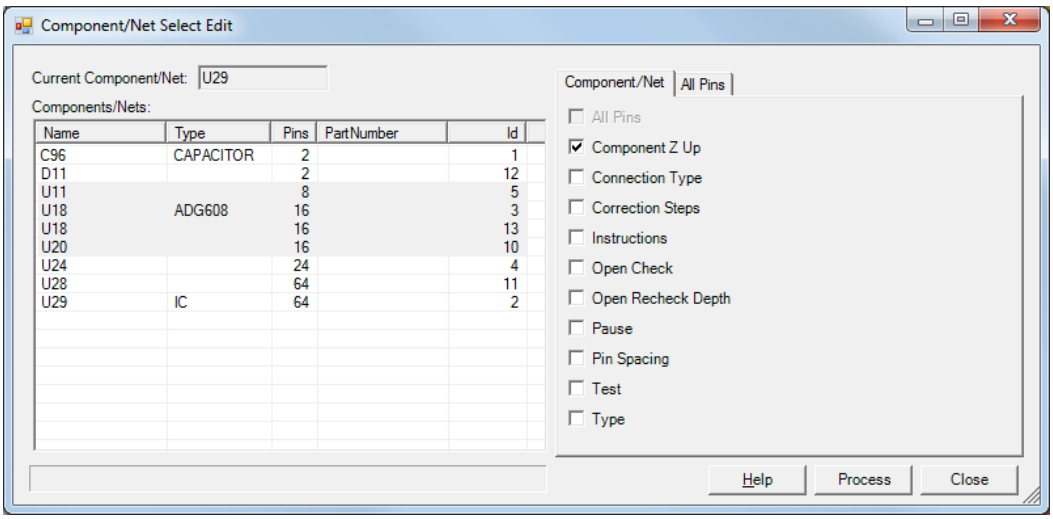


Component Editing – Component Group Edit



Right clicking on the Row header bar of the Tree/Components pane allows for editing of the Component information. Select **Group Edit** to open the Group Edit window for the selected Component. This allows you to duplicate changes made to the selected Component to other Components in the Tree.

Note: Be sure to make your changes to the selected Component **before** using Group Edit to duplicate the settings to other Components.

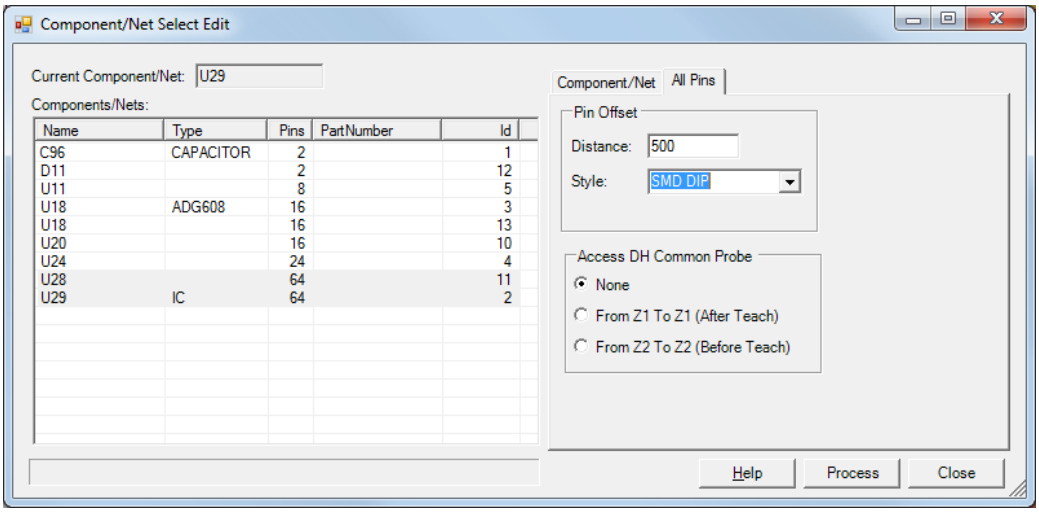


To use the Component Group Edit, select the Components on the left that you wish to change. You can **SHIFT** and **CTRL** keys to select groups or individual Components in the list. Mark the setting you want to change by clicking the checkbox. The example above shows the **Component Z UP** as the setting to change. Click **Process** to make the changes to the selected Components.

Using Component Group Edit to Offset XY Pin Locations

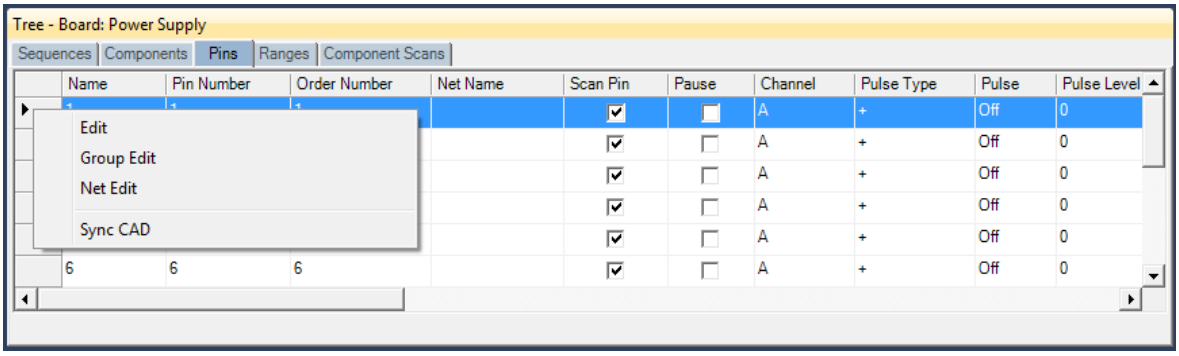
The Component Group Edit can be used to move the XY pin locations for a component a set distance away from the component body. This can be very useful if imported CAD data places the probing location too close to the component body resulting in bad or no probe contact.

Open **Component Group Edit** by right-clicking the Row header for any component in the Tree pane/Component tab. When the Component Group Edit window is displayed select the **All Pins** tab on the right side.



Select the components on the left that you wish to offset pin XY locations making sure that they are the same style. You can **SHIFT** and **CTRL** keys to select groups or individual Components in the list. Input an offset distance in the **Distance** field using the unit set for the Sequence (mils, mm, microns; the above example is using microns). Select the component Style as **2 Pin**, **SMD DIP** (dual inline package) or **Square QFP** (quad flat pack). At this time 3 pin components and rectangular IC packages are not supported. Click Process to make the changes.

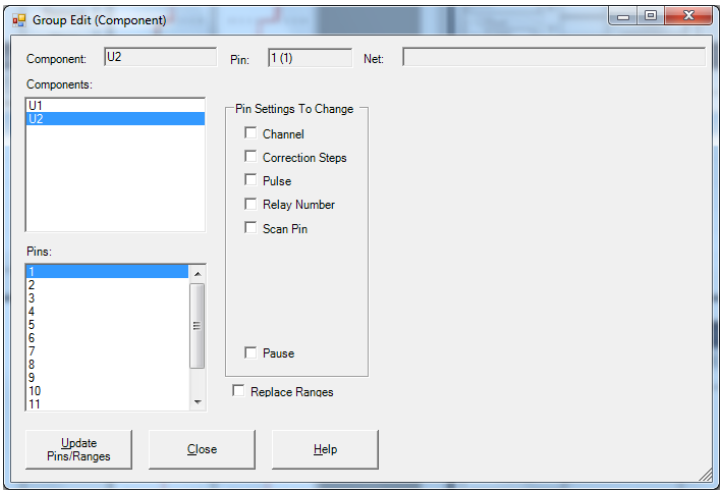
Pin Editing – Pin Group Edit and Net Edit



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.

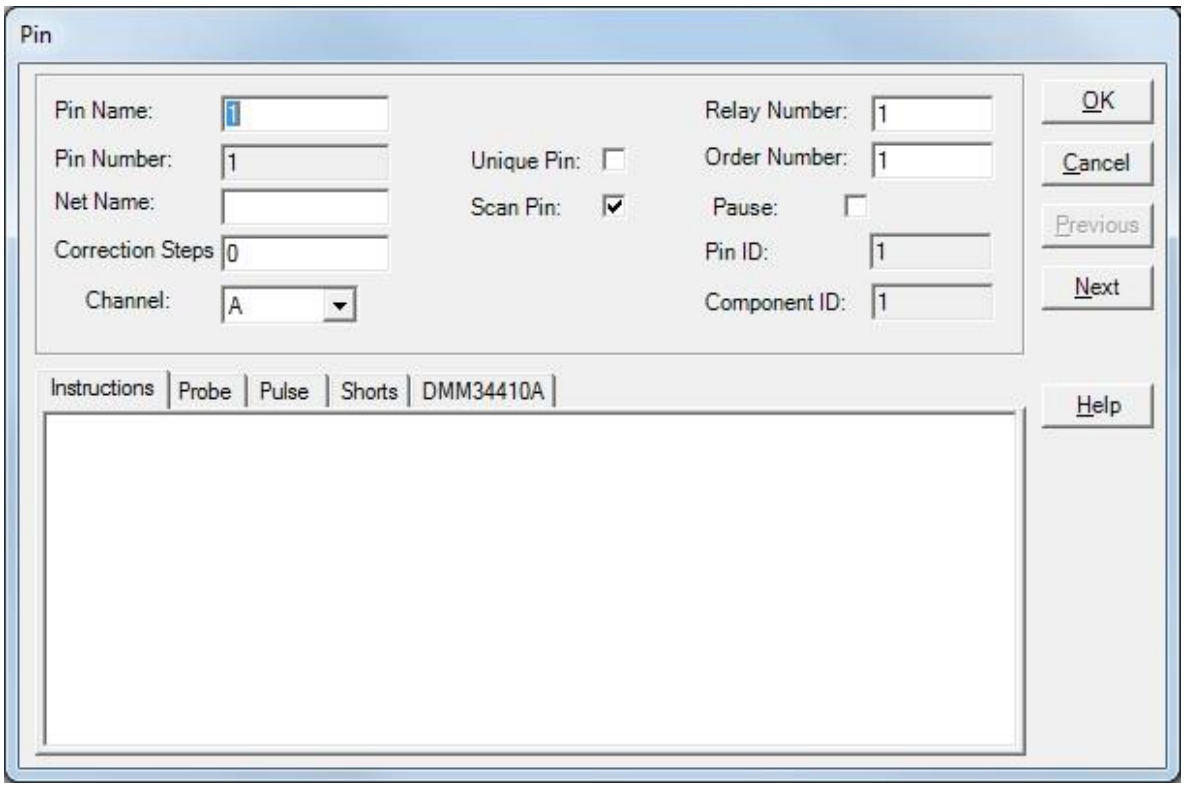
Note: Some Pin settings can also be edited directly in the Pins grid by clicking the appropriate field and modifying the setting.

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.



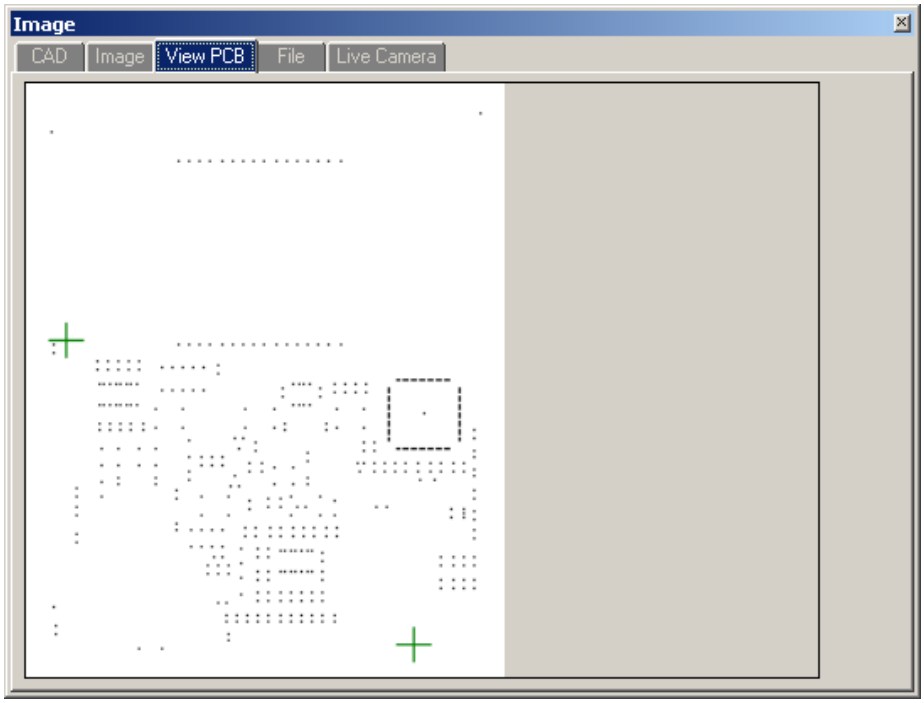
Change the desired settings for the current Pin **first** then select Group Edit from the right-click menu (see above). Components with the same number of pins will be listed. Select the components and pins you want to be changed. Select the Pin Settings check box(s) for the setting(s) you want changed. 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.

Pin Editing



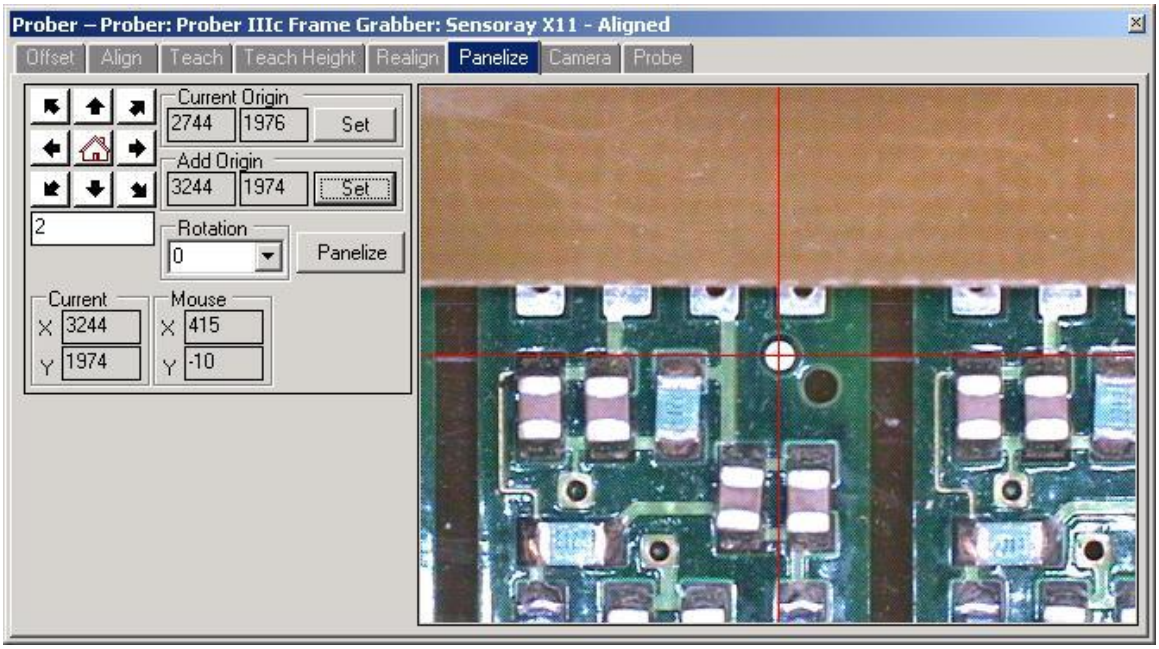
Individual pin characteristics can be edited in the Pin Edit window. **Net name**, **Channel**, **Correction steps** (for Access Probers) and **Scan Pin** (enable/disable scanning of the pin) can be controlled.

ViewPCB Image



For Prober users, when XY locations to test are stored, the Image/ViewPCB pane is automatically updated. To view the Image/ViewPCB image, select the Image/ViewPCB pane. Right-click on the image to display a menu where you can select **Save As** to save the image or use the **Copy** to save it to the clipboard for use in other programs. **Refresh** will update the ViewPCB window.

Test Creation - Panelize



In some cases, a PCB may be part of a panel of similar boards.

Note: Panelize is designed to work with alignments points selected on the actual PCB not a panel frame.

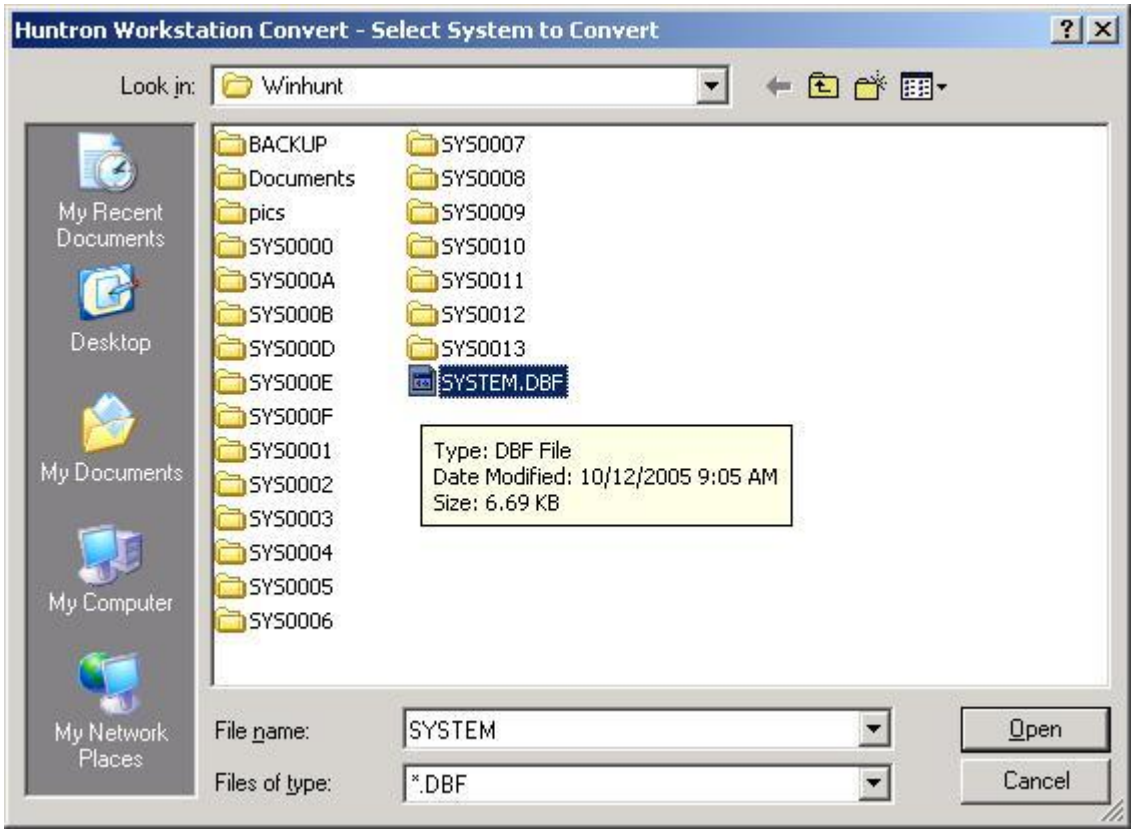
To easily create subsequent panels based off of an original panel, select **Panelize** in the Prober Pane (shown above).

Select a point on the first panelized PCB and press **Set** in the Current Origin section. The XY coordinates of the source origin will be displayed.

Move the camera to the same point on the next panel and press **Set** in the Add Origin area. The XY coordinates of the second origin will be displayed. Set a rotation value in the **Rotation** field.

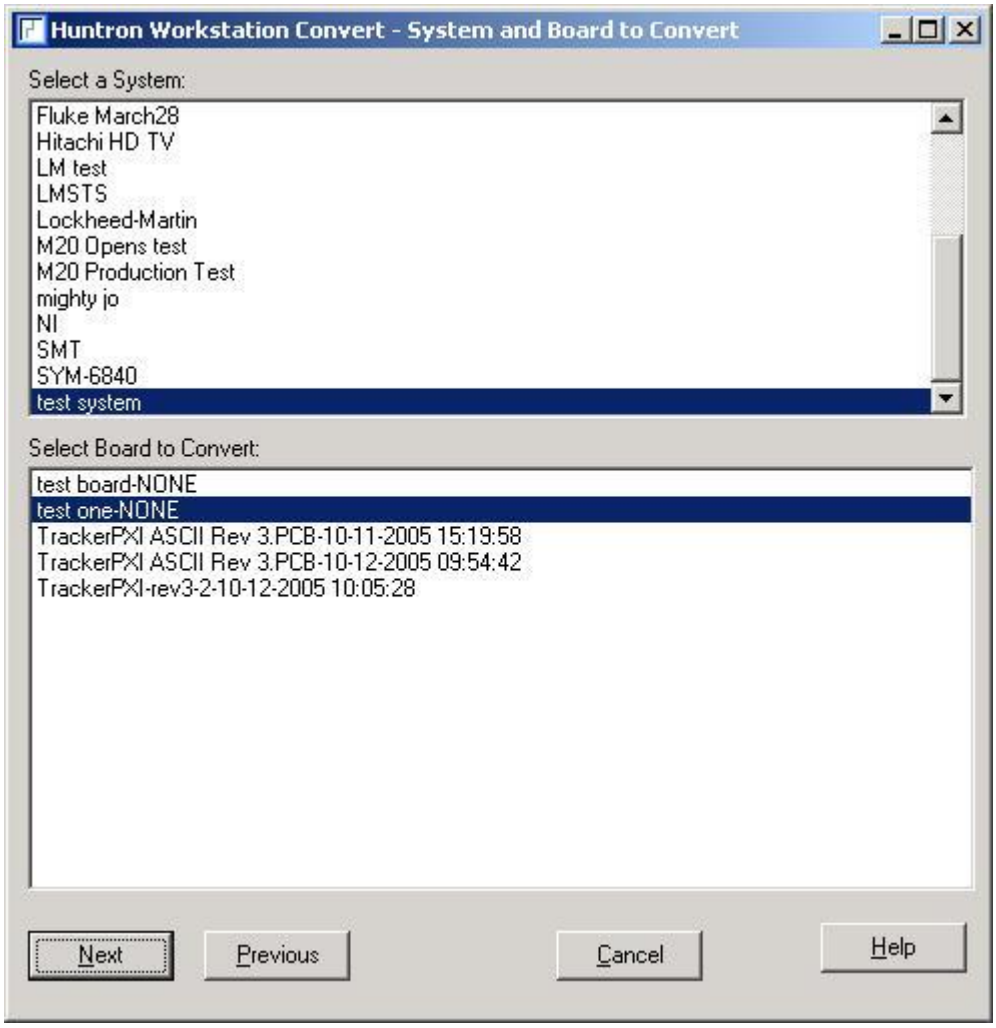
Press the **Panelize** button and a new Sequence will be created in the Tree/Sequence pane. The name will be incremented by 1 or by one letter. The new panel will be an exact duplicate of the original but have its location offset as configured during the Panelize process.

Test Creation – Convert step 1



If you are moving to Huntron Workstation version 4 from Huntron Workstation version 3 then you will want to utilize the tests created in the older version. Version 4 uses a different database structure so the old tests need to be converted for use in version 4. To start, select **Convert** from the File menu. The Select System to Convert window will open. The default directory name for version 3.X is "winhunt". Browse to the winhunt directory. Select the SYSTEM.DBF file and click **Open.**

Test Creation – Convert step 2

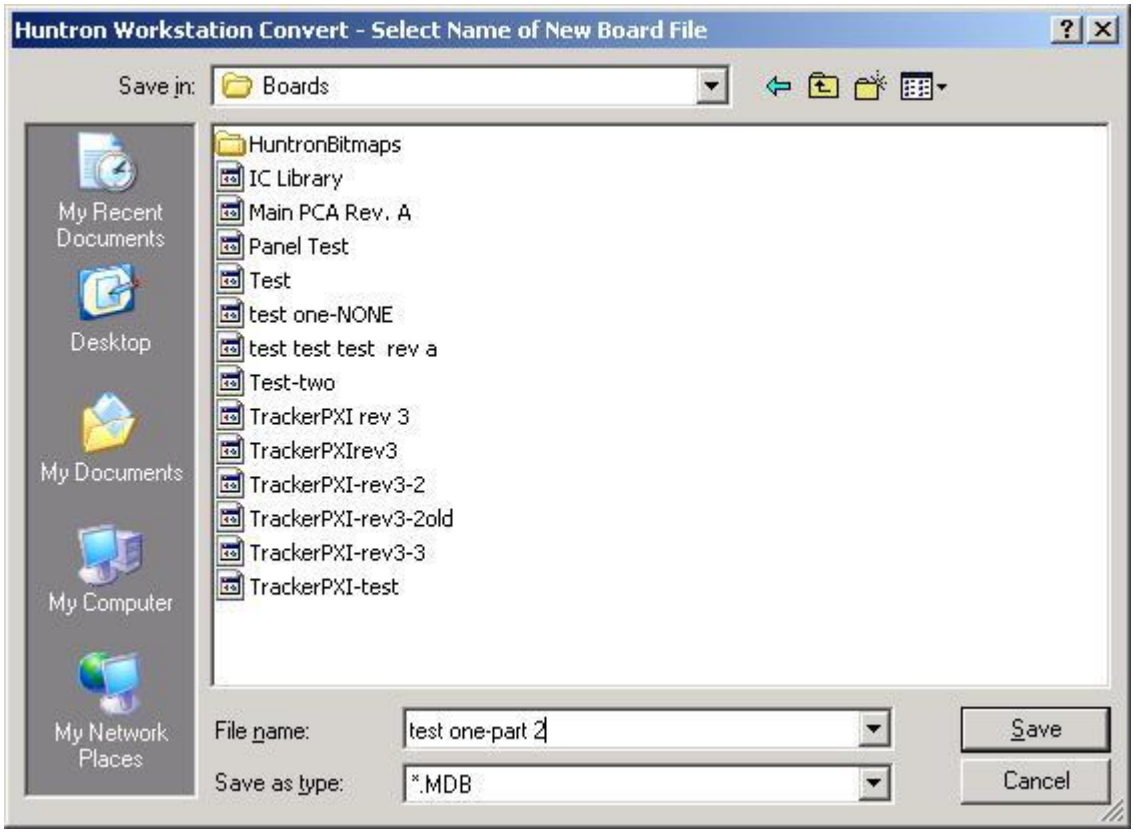


Once SYSTEM.DBF has been opened, a list of the Systems and Boards contained in the DBF will be displayed.

Select the **System** and **Board** to convert as shown above.

Press the **Next** button to continue to the next step.

Test Creation – Convert step 3



Input the **File Name** of the new Board File.

Click **Save** to create and save the new Board that is compatible with Huntron Workstation version 4.

A pop-up window will ask if a Huntron Access prober was used to create the original test. Click **Yes** or **No** accordingly.

Note: It is very important for Prober users to verify the Teach Height settings after Converting a board test.

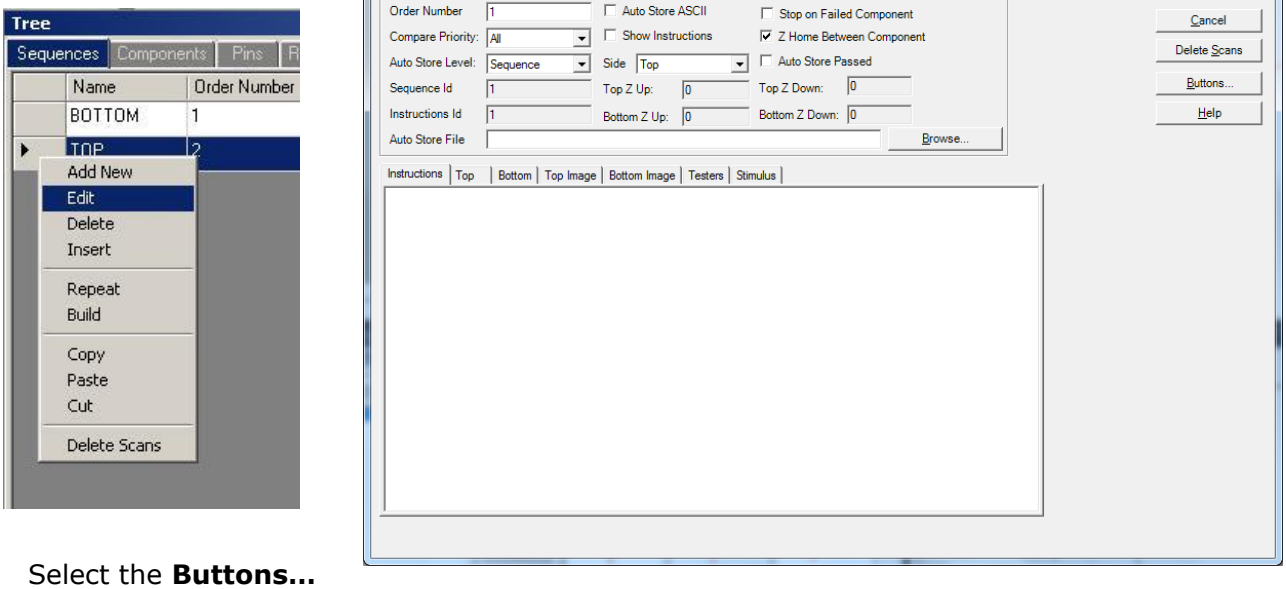
Test Creation – Convert Status

The Convert status (shown above) will be displayed as the Board is converted. You will be prompted when the conversion is finished. The new board will automatically be opened in Huntron Workstation when the conversion is complete.

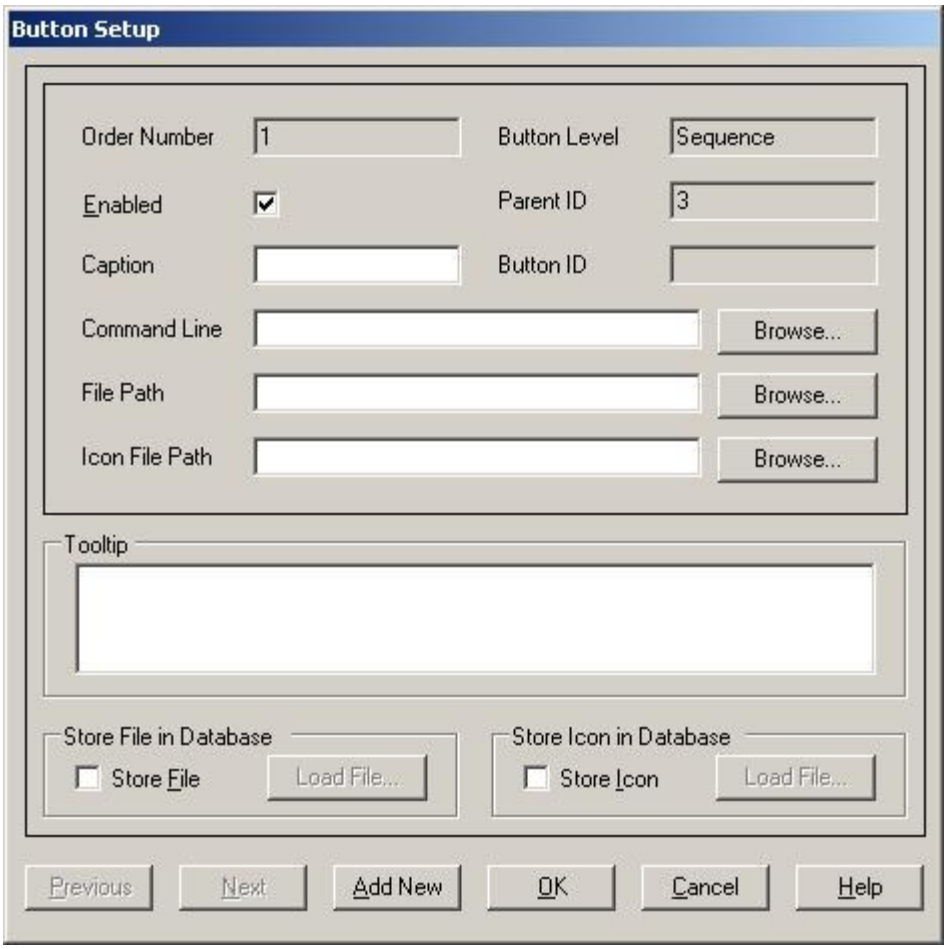
Huntron Workstation Buttons Feature

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.

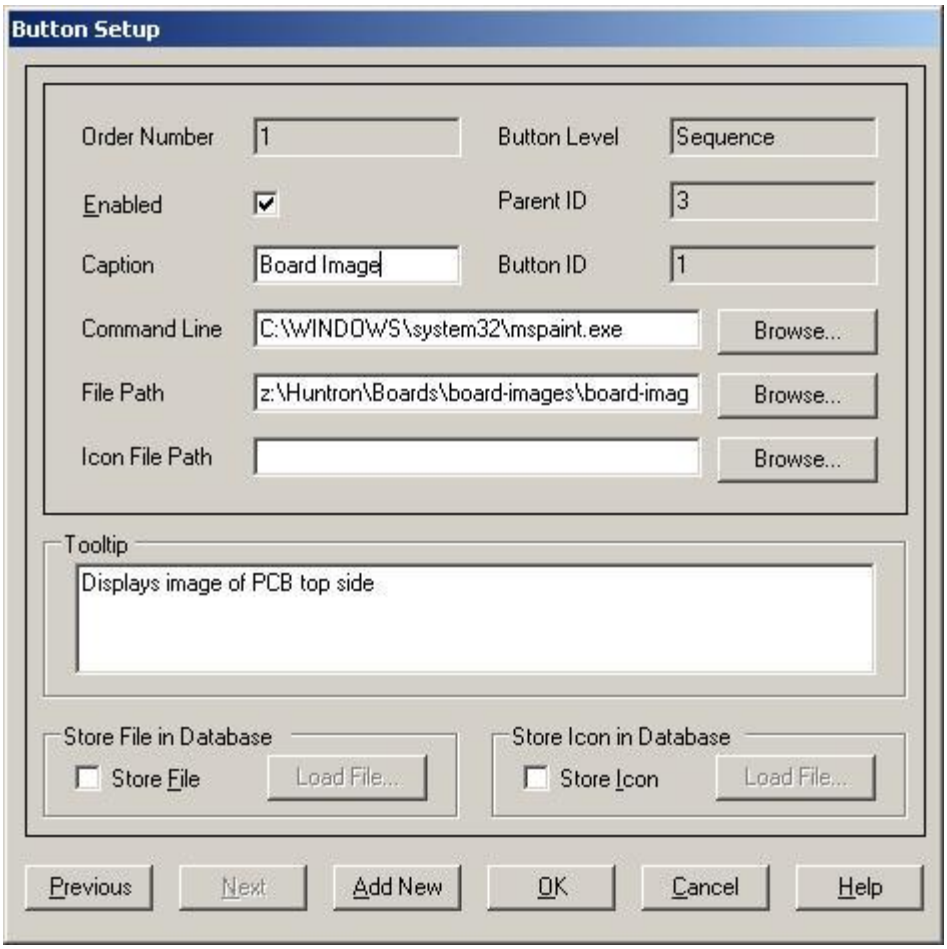
To start, the Sequence edit window is opened by right-clicking the row header for the desired sequence. The Edit window will be displayed (see below).



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 Huntron at 800-426-9265, 425-743-3171 or email info@huntron.com.

There is online assistance for Huntron Workstation at www.huntron.com/support/workstation.htm. This page will contains software updates, a form for reporting software issues and updates to documentation such as this tutorial.

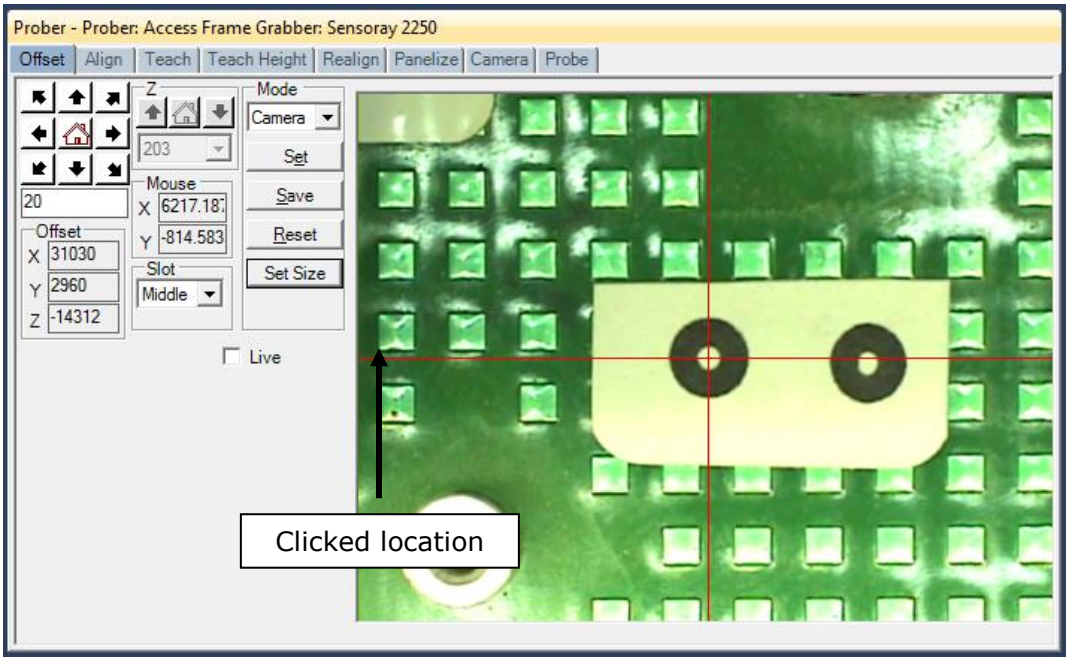
Appendix 1 – Using the Set Size Feature for Access Probers

The Set Size feature allows the user to make adjustments to the dimensions (units of measure per pixel) of the image used in the Offset, Align, Teach and Realign windows. This is useful to adjust accuracy when clicking in the image to move to a point and to create better stitching of the image used in the Image pane/Image tab.
The Set Size feature is available by clicking the **Set Size button** in the Prober pane/Offset tab. The dimensions shown in the Set Size window when opening it for the first time will be factory default values.
The basic rules making adjustments in the Set Size window are:

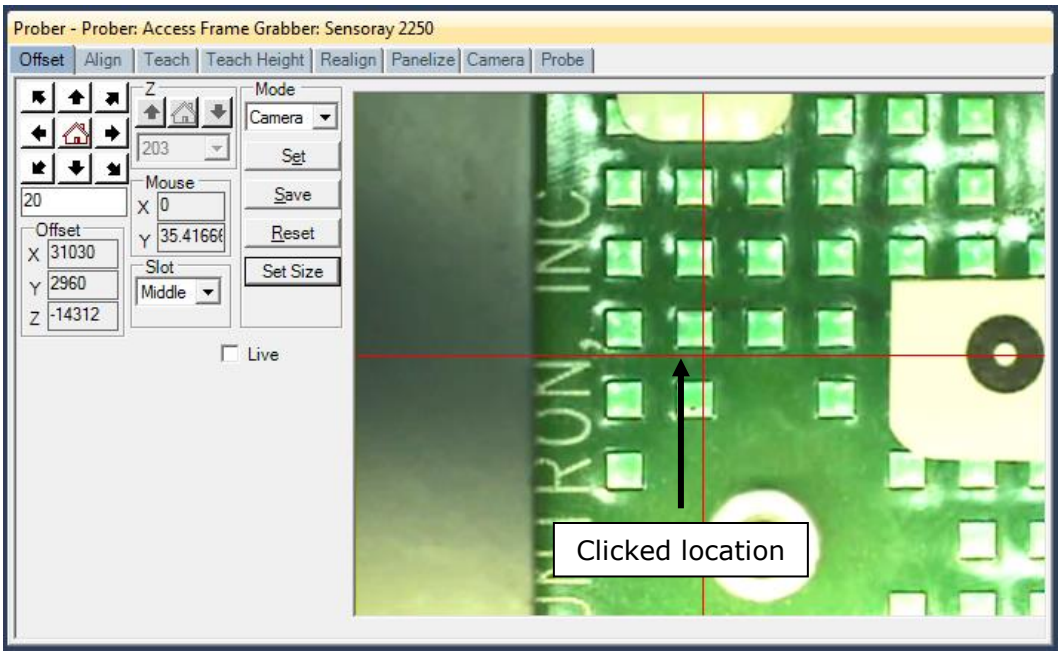
- When clicking on a point in the camera image and the camera moves too far – **decrease** the Horizontal Length and Vertical Height values
- When clicking on a point in the camera image and the camera moves too short – **increase** the Horizontal Length and Vertical Height values

Use the following example to help learn the process of calculating and adjusting the values. In this example we will be adjusting for a camera that moves too short.
Before starting, make sure the circuit board is mounted at the Slot level set for the selected Sequence. The board cannot be above or below the selected Slot level.

Place the camera crosshairs on a point on the circuit board. Next, click on a point at the extreme left or right edge of the image, in this case, the left side was used.

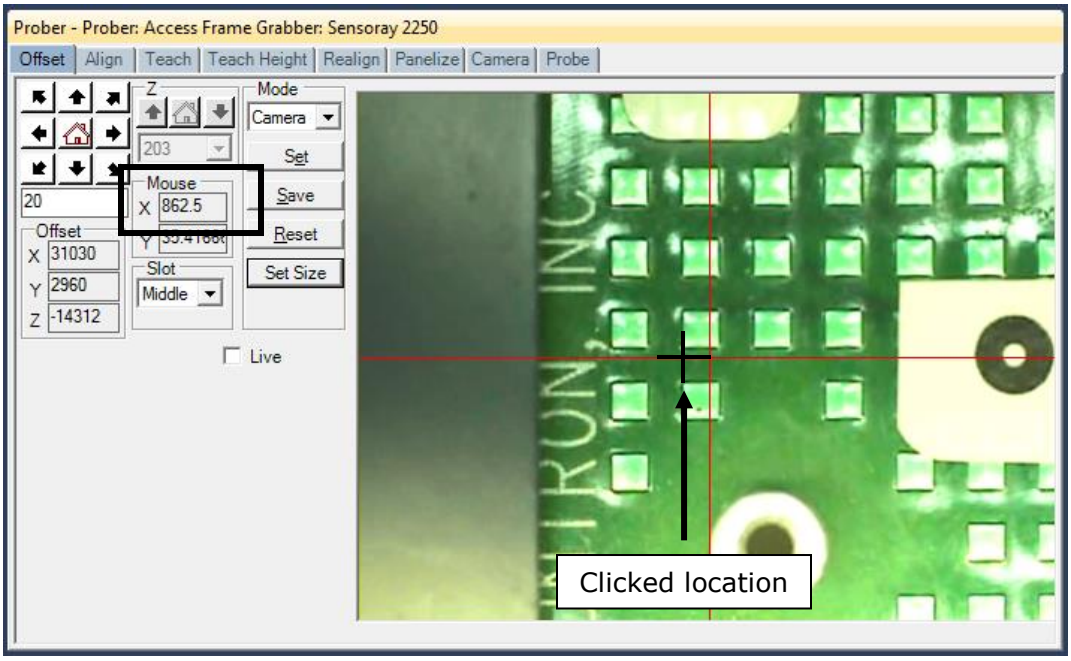


Notice that the camera crosshairs did not go to the clicked location but fell short.

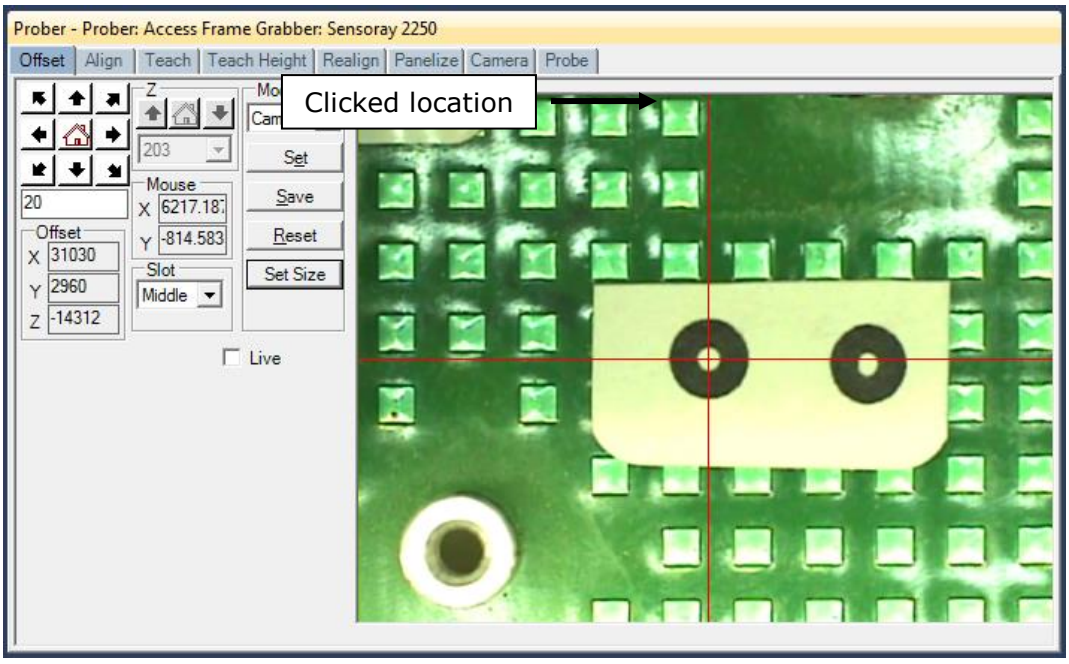


To measure the distance from the crosshairs to the clicked location, move the mouse cursor (crosshairs) to the clicked location but **do not click**. The distance between the

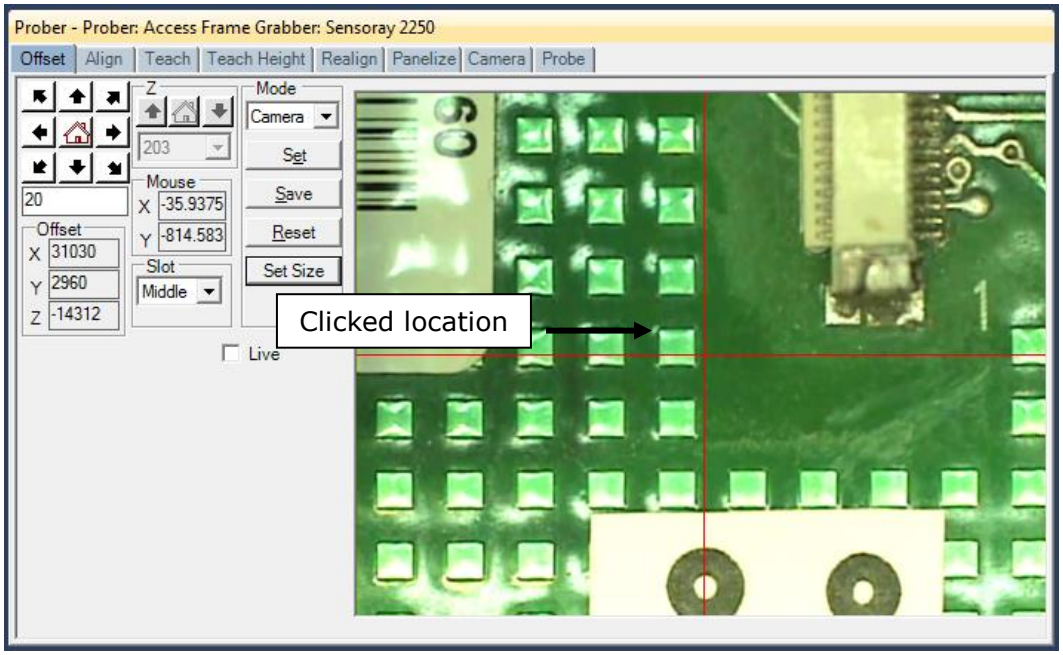
camera crosshairs location and the clicked location will be shown on the **Mouse X field** of the Offset window (red box). Write this value down.



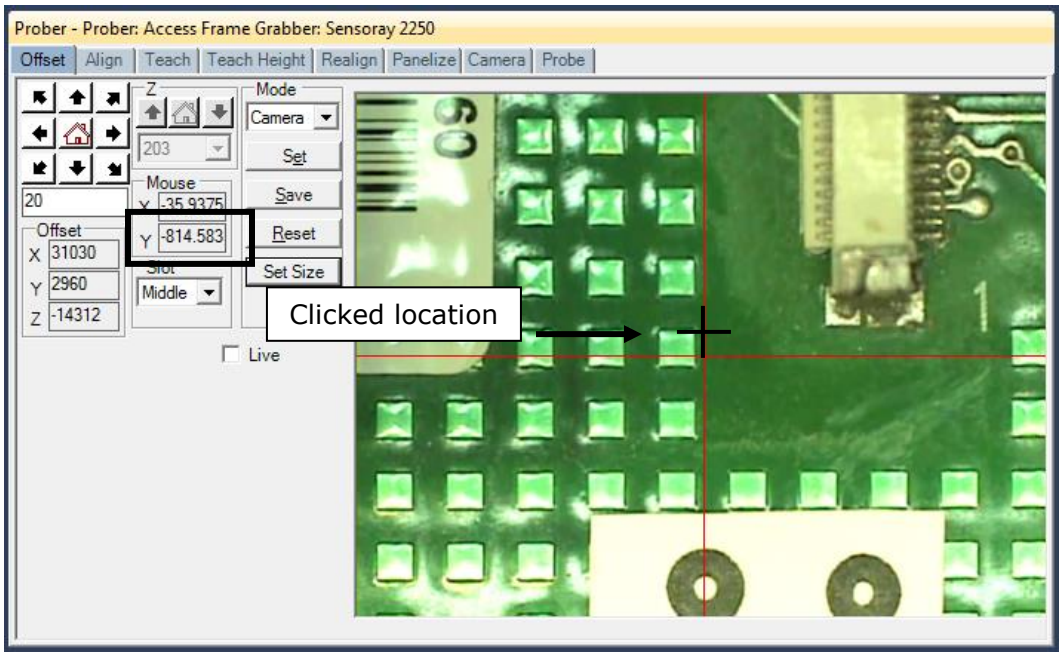
The next step is to calculate the vertical distance. Put the camera crosshairs on a point on the circuit board. Click a location on the extreme top or bottom edge of the camera image. In this example, the top edge was used.



Notice that the camera crosshairs did not go to the clicked location but fell short.

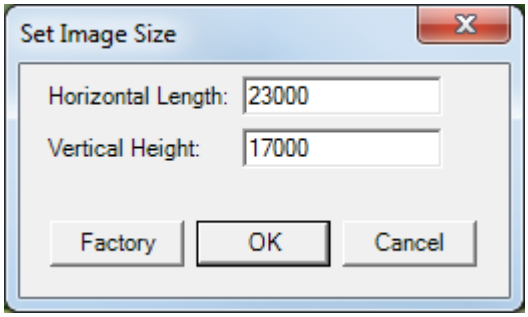


As you did with the horizontal distance, you will need to measure the distance from the camera crosshairs to the clicked location by moving the mouse crosshairs to the clicked location.

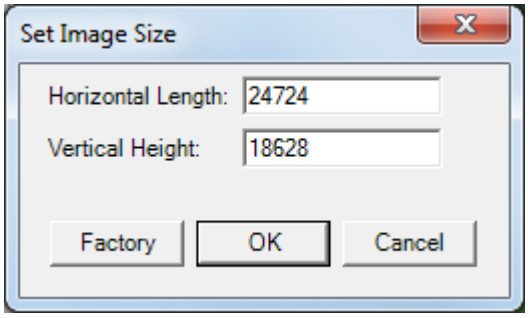


Move the mouse cursor (crosshairs) to the clicked location but **do not click**. The distance between the camera crosshairs location and the clicked location will be shown on the **Mouse Y field** of the Offset window (red box). Write this value down.

Click the **Set Size** button and you will see this window displayed. The values shown will be the factory default unless they were changed previously.



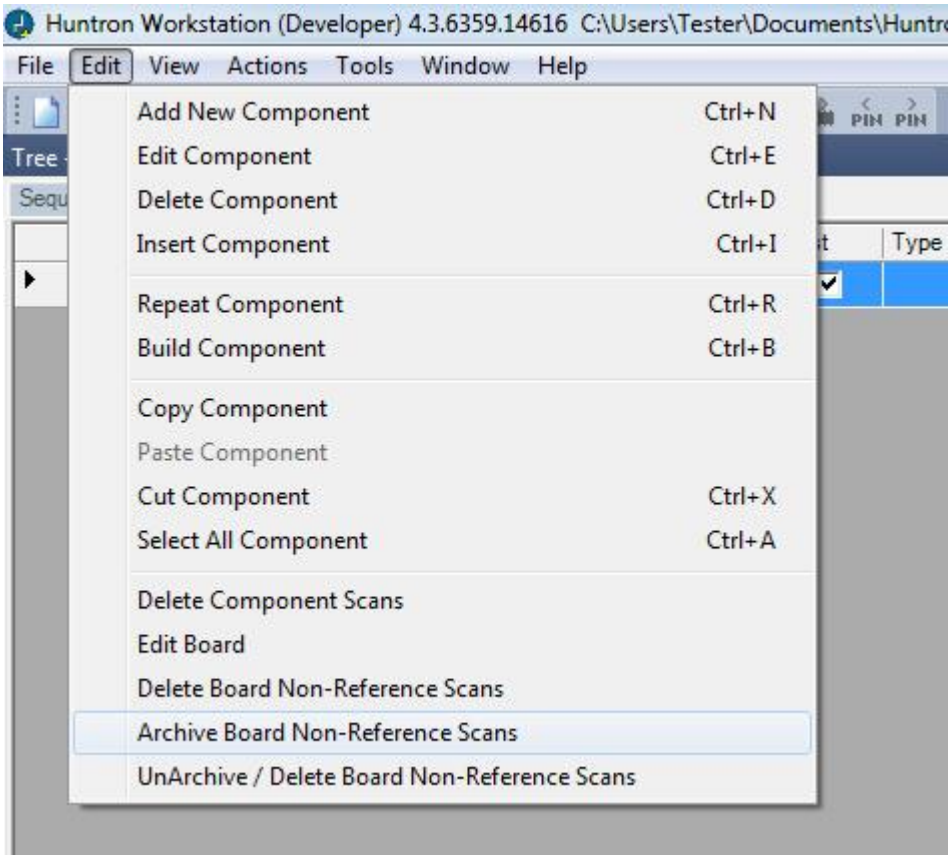
Using the X measurement you wrote down earlier, **multiply this distance X2** and add the result to the **Horizontal Length** value. Next, multiply the Y measurement written down earlier **X2** and add the result to the **Vertical Height** value. It is necessary to multiply by 2 because the dimension needs to be corrected in two directions, top edge to bottom edge and left edge and right edge. The image below shows that values with the measured corrections made.



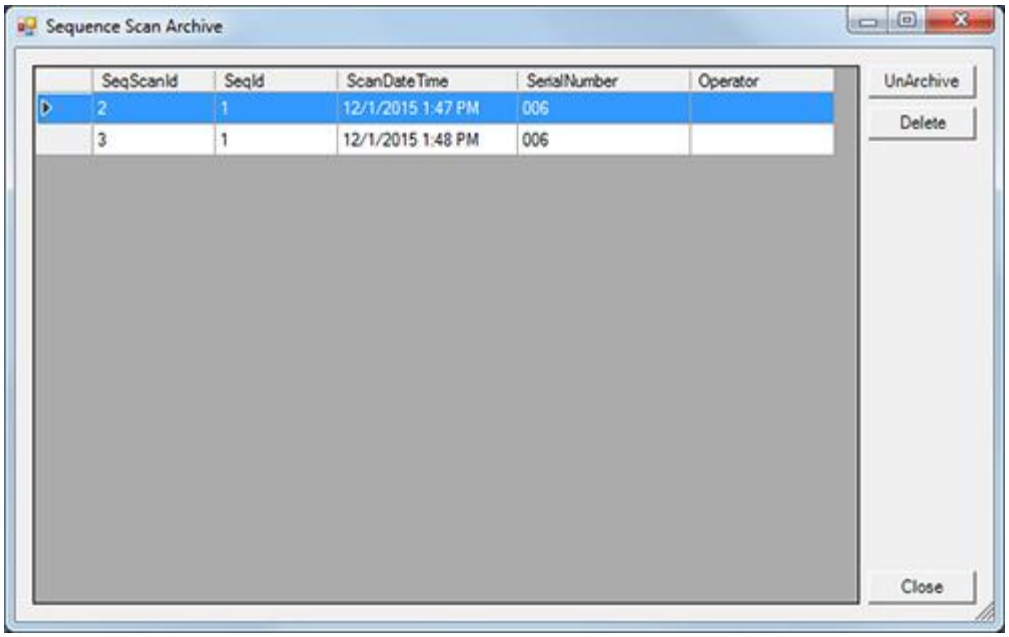
Click **OK** to save the changes and try clicking in the camera image. You will find that the accuracy is much better. You may need to make additional small adjustments to achieve satisfactory accuracy. If you have a board image already created in the **Image pane/Image tab**, you will want to recreate the image again. The result should be better stitching at the image tile seams and an overall better image.

Appendix 2: Huntron Workstation Archive Scans Feature

The new Archive feature built into Huntron Workstation allows you store and remove Sequence scans from the working database to achieve better performance. These archived scans are not deleted but saved away in a different database table and can be restored at any time. It is very simple to use.



- Select "Archive Board Non-Reference Scans" from the Workstation Edit menu
- At this point the software will search the database for non-reference scans that were part of a Sequence scan and remove them from the active database
- If you look at the Components Scans tab in the Tree pane, you will see that the Reference scans remain
- To unarchive a scan, select "UnArchive / Delete Board Non-Reference Scans"
- The Sequence Scan Archive window will be displayed



- You will see the Archive Scans listed where you can select the scan you want to restore to your test Sequence. Use the CRTL or Shift keys to select multiple scans if needed

- Click the UnArchive button and the selected scans will be restored to the Sequence

- Use the Delete button to permanently delete scans that are no longer needed.

- Close the Sequence Scan Archive window and look at your Component Scans tab and you will see that your scans have been restored.

Keeping your test database small is the best way to get the most out Huntron Workstation performance.

CalPower
Via Acquanera, 29 22100 COMO
tel. 031.526.566 (r.a.) fax 031.507.984
info@calpower.it www.calpower.it