Huntron Workstation Software Tutorial

Huntron Workstation software

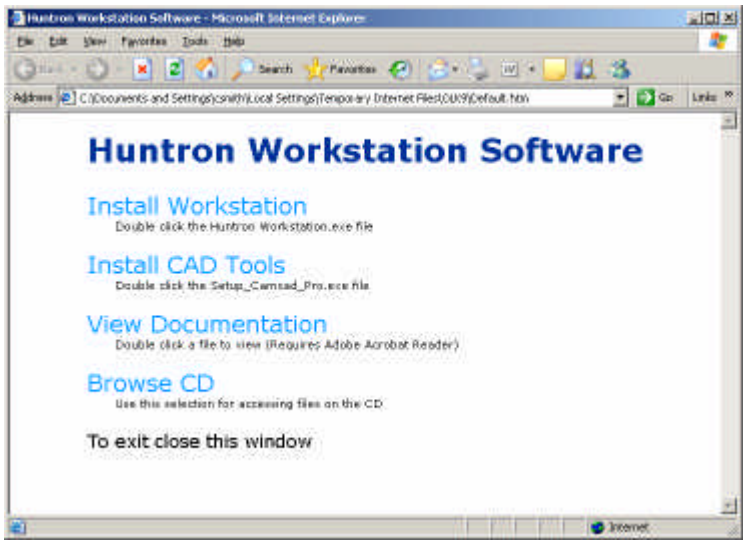
It is very helpful if you have a working knowledge of Microsoft Windows prior to using Huntron Workstation.

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

Installation Instructions

Uninstall any previous versions prior to loading the current version.

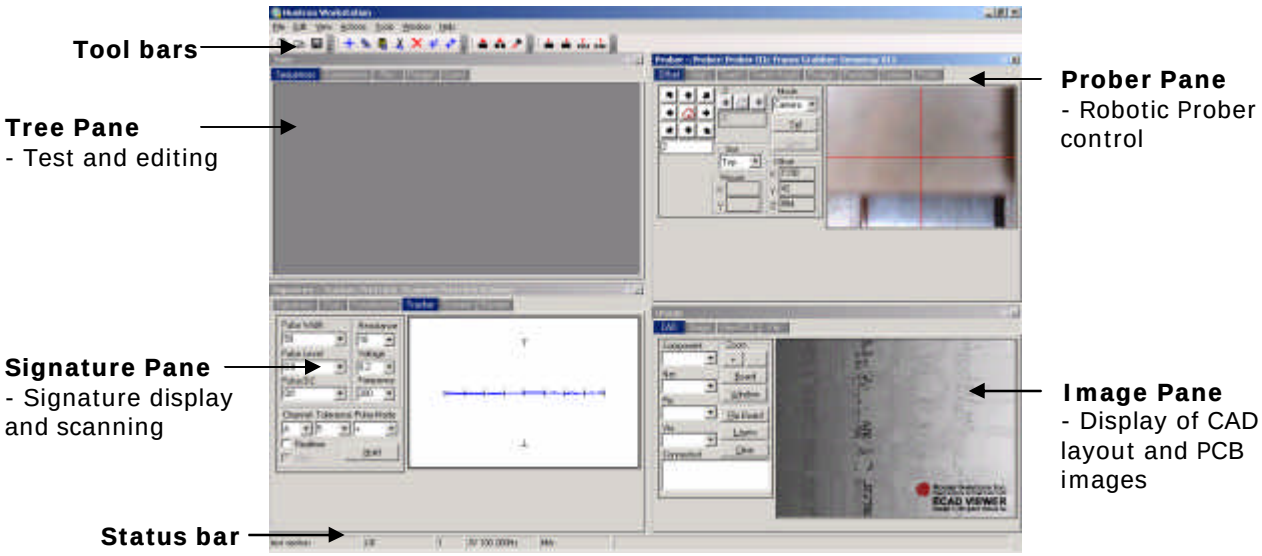
While in Windows (2000 or XP), insert the Huntron Workstation CDROM. The CDROM should Autorun and display the Installation page in a browser window.



Select **Install Workstation** and follow the on-screen prompts for instructions while proceeding. When complete the programs listing will have a new program group called "Huntron" and an icon will be placed on the desktop. Select **Install CAD Tools** if you have purchased addition CAD Tools. You can view support documentation in PDF by clicking **View Documentation**. If you wish to view files on the CDROM, click **Browse CD**.

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

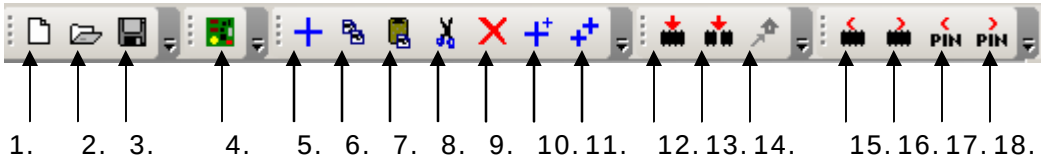
Huntron Workstation 4.0 has built-in conversion utilities that enable conversion of board tests created in 3.X versions of Huntron Workstation.



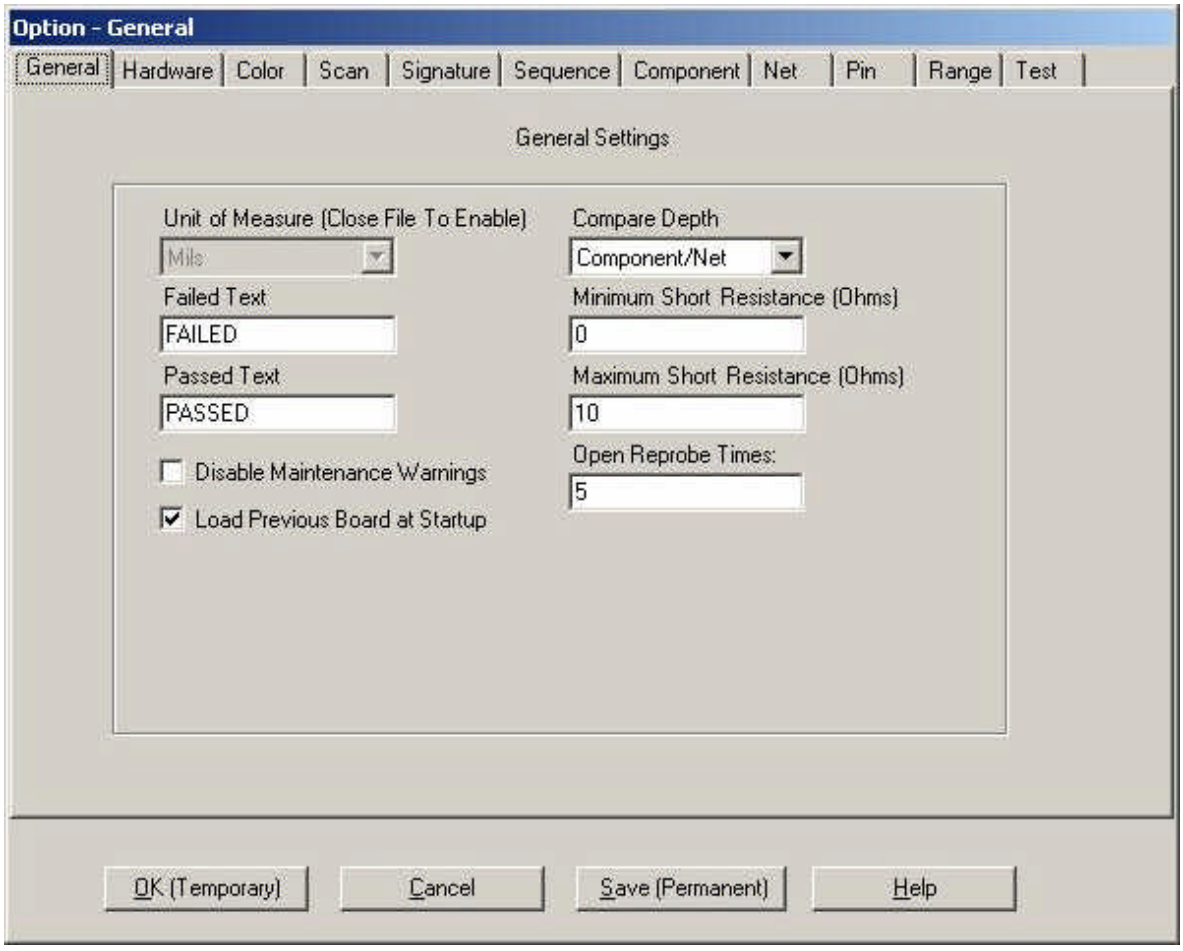
The Main Window for Huntron Workstation 4.0 features a multi-pane layout so many aspects of the test creation and execution can be displayed at the same time. The Panes are interactive to a certain degree in that changes in one Pane will affect other Panes. The Panes can be resized, floated independently 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

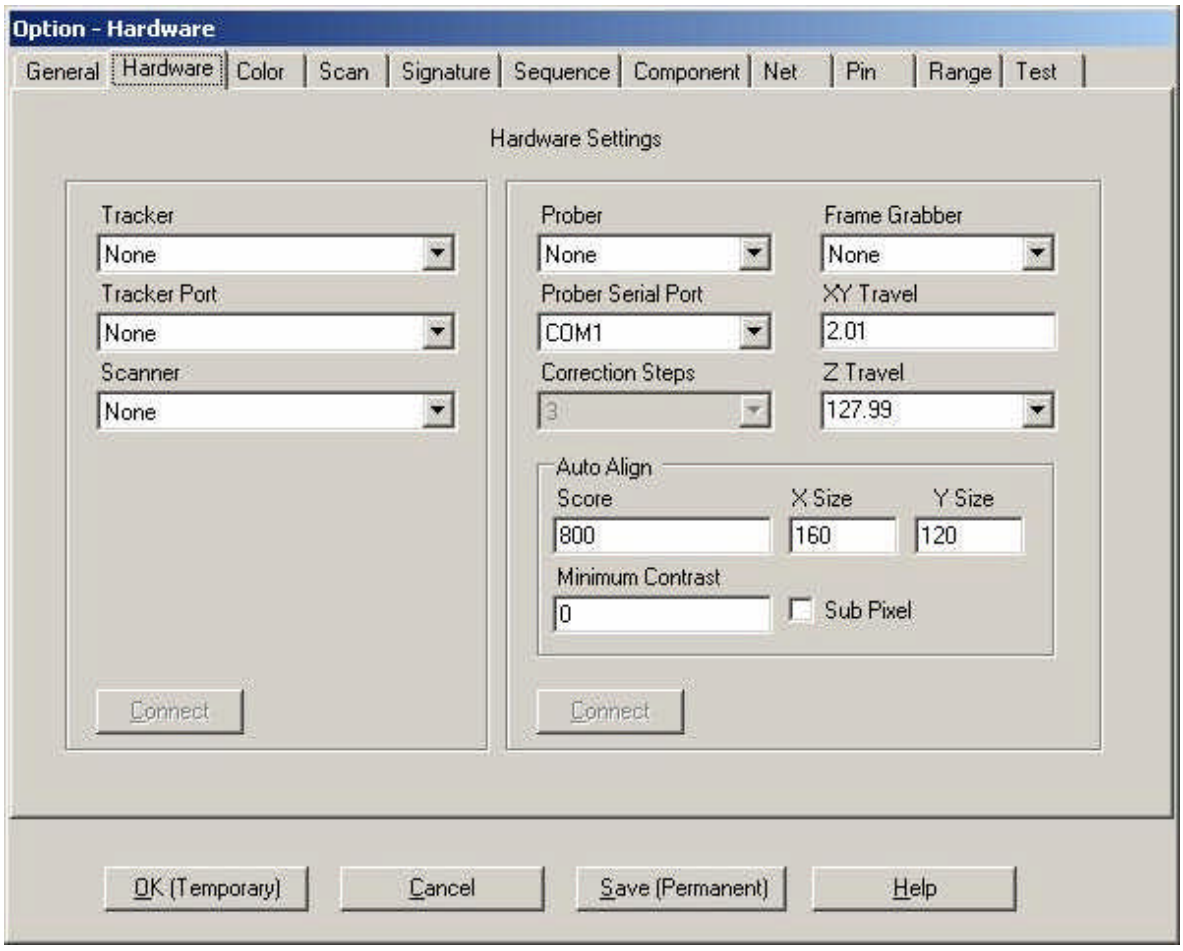
The Toolbar just under the application menus allows for quick access to several program functions. The function of each is broken out in the diagram below.



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



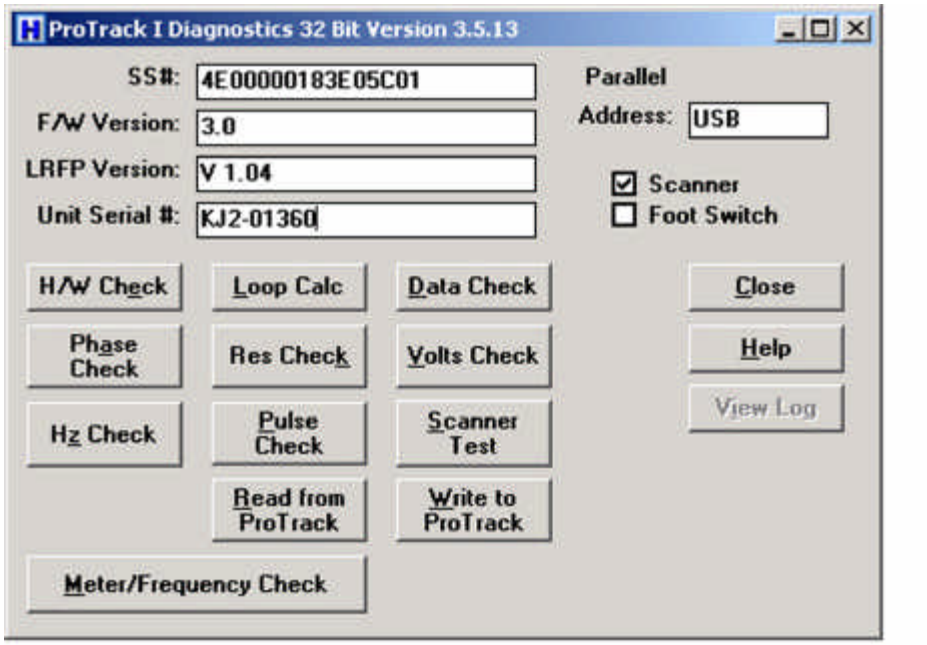
When first starting Huntron Workstation, select **Options** from the Tools menu. The Options window is used to setup default settings used when working in Huntron Workstation. Select the **General** tab and set the desired Unit of Measure (use **Microns** for Huntron Access, **Mils** for any other Prober).



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 (parallel address, USB, etc…) and Scanner (if connected). Prober users will need to select the type of Prober, the serial port for the Prober and the Frame Grabber card installed in the PC. 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.

Using a ProTrack

If a Loop Compensation (Loop Calc) file does not exist for the selected configuration, an error will pop-up asking that a Loop Calc be performed using the ProTrack Diagnostics.



ProTrack Diagnostics

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

- 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 TrackerPXI

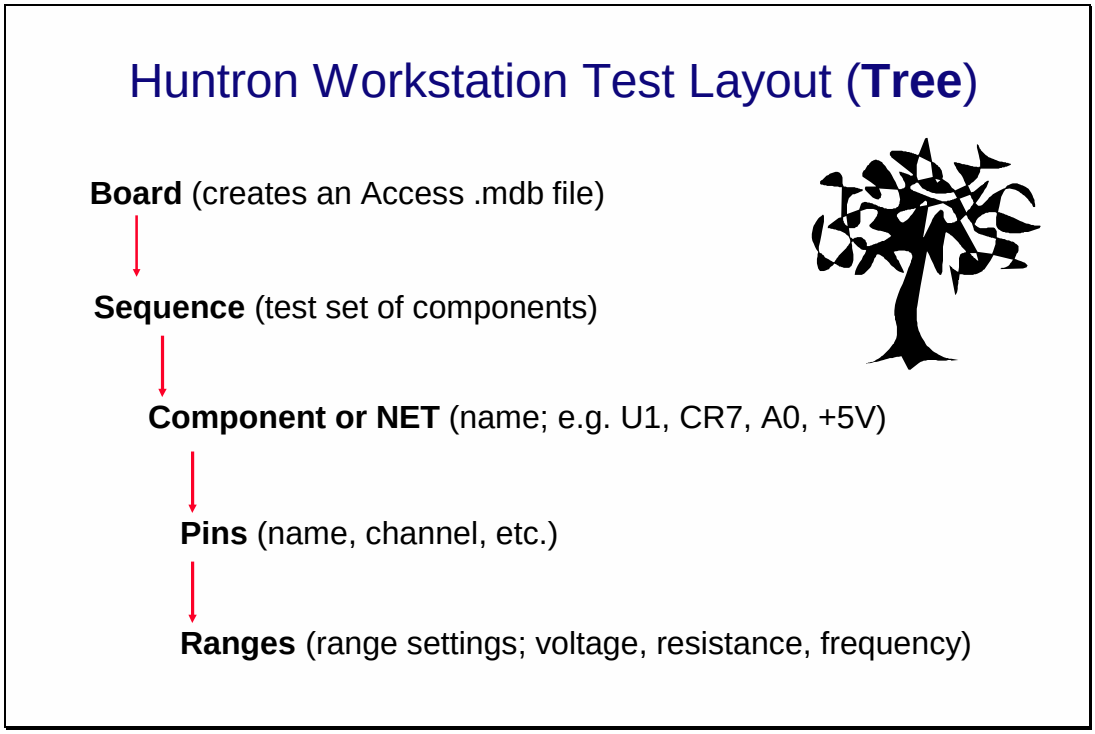
The TrackerPXI comes from the Factory calibrated for standalone use. If the TrackerPXI is connected to a Prober or other cabling, calibration must be performed.

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

Run Calibrate by clicking the appropriate button in the TrackerPXI Diagnostics window. When the Calibrate procedure finishes (about 4 minutes), run Verify Calibration. This also takes about 4 minutes to complete. The Calibration information is stored in the TrackerPXI and will be used by the software. If changes are made to the cabling or attached instruments, simply run Calibrate and Verify Calibration again.

Creating a Board Test

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



Example of Board Test Information:

Board Name: Main Board (creates "main board.mdb" 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.

Test Building Procedure

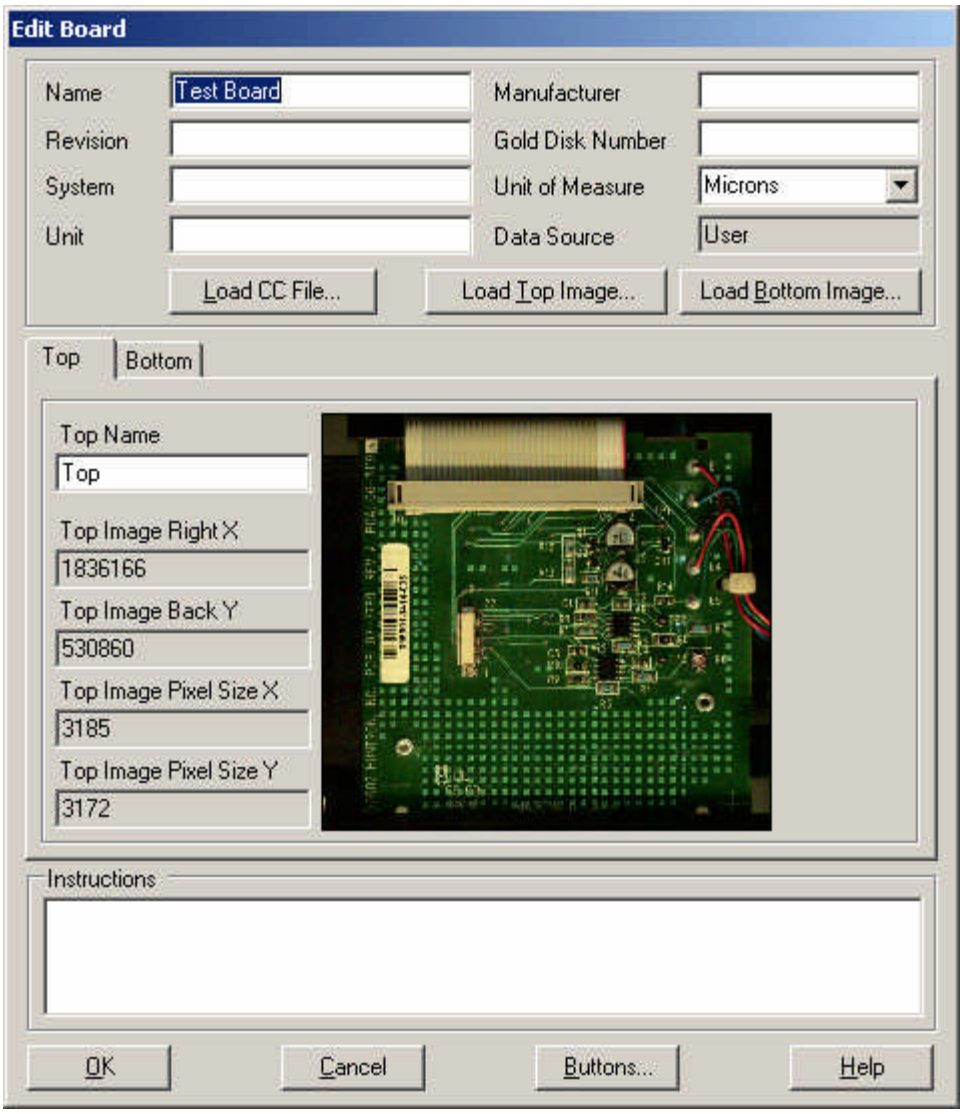
- Create Test Database
  - Add Board, Sequence and Components before teaching or scanning
- Perform Camera Offset
  - For Prober users
- Select Alignment points
  - For Prober users
- Teach component pin locations
  - For Prober users; teach all points before scanning
- Scan, store/merge signatures



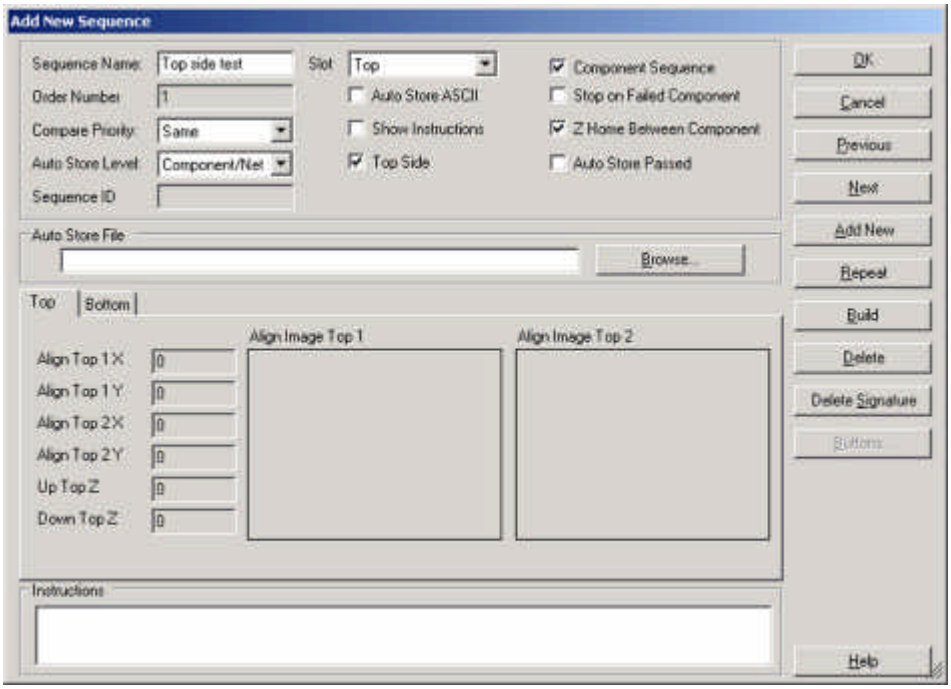
Test Building Procedures

Listed above is a general outline on how you would proceed when preparing 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.

For tests created using version 3.X, Huntron Workstation has a built-in conversion utility. See the section on Covert later in this tutorial for more information.



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.



Add 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 in when scanning.

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

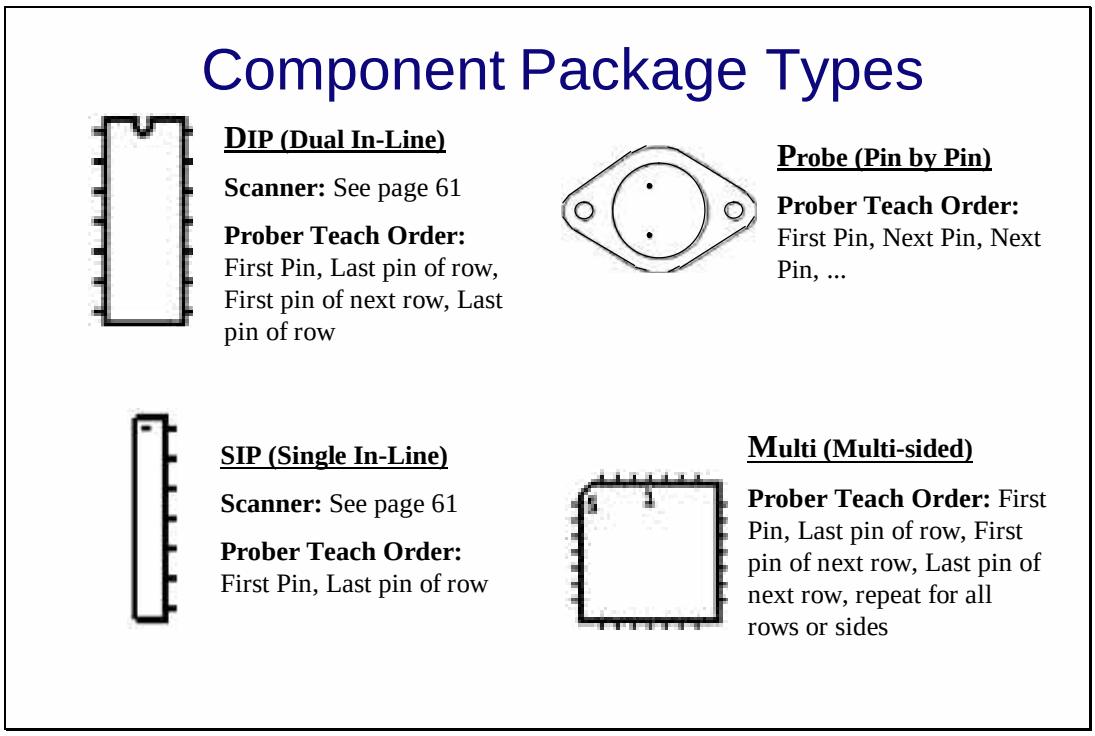
Previous/**Next** buttons: Steps through existing Sequences backward or forward. These buttons are disabled when creating a New Test Sequence.

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

The **Add New** button will allow for a new Sequence to be created.

The **Delete** button will erase the selected Sequence.

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

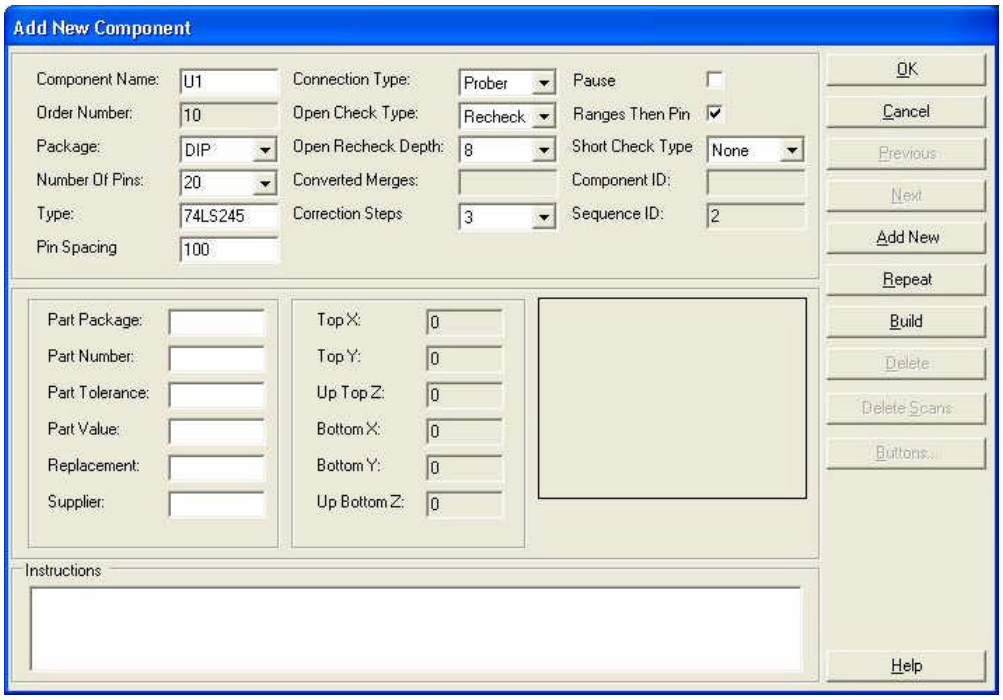


Component Package Types

There are four different package types you will find yourself using when creating components used in the test sequence. They are DIP (dual-in-line), SIP (single-in-line), PROBE (point by point), and MULTI (multiple row or sided).

For ProTrack with Scanner users, the package type determines how the Scanner pins are counted.

For Prober users, the XY locations for each package type are "taught" in slightly different ways. Shown above is a diagram of each package type and a general synopsis on how to proceed with teaching their XY locations.



Add New Component Window

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** field: 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.

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 (in mils)

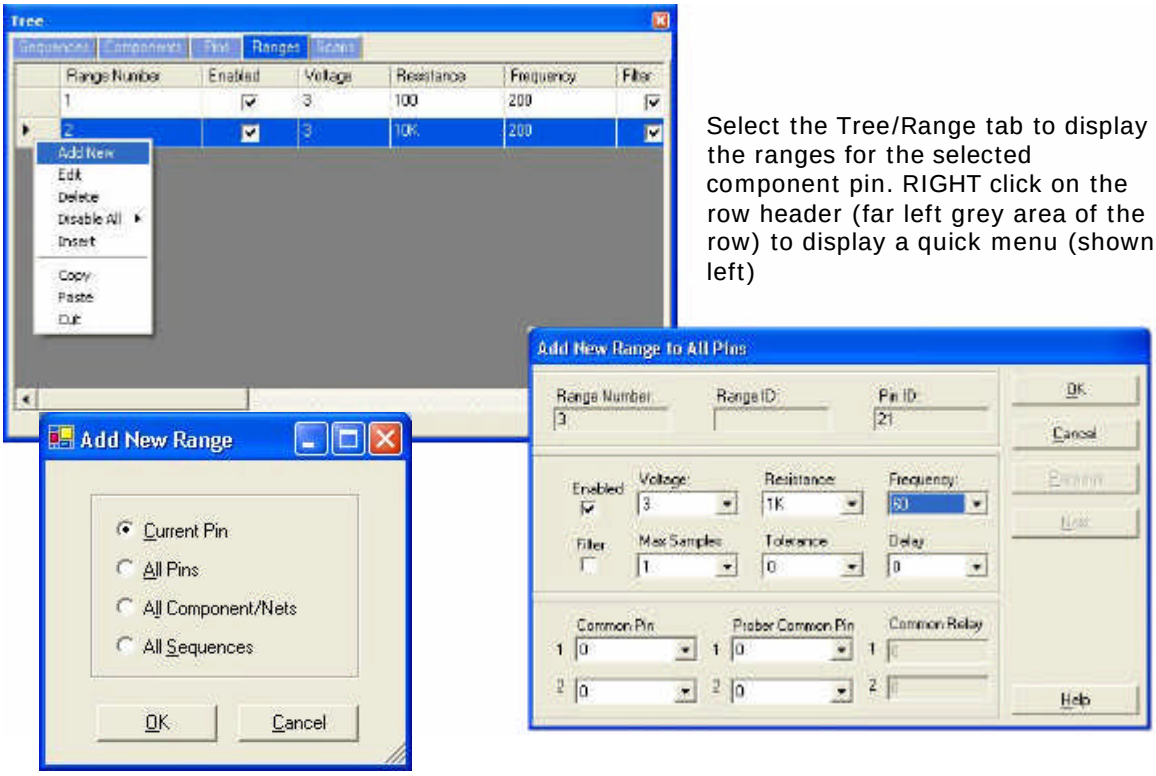
Connection Type: Select the type of hardware used to interface to the component

Open Check Type: Select from None or Recheck 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.

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.

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



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)

Add New Range

To add a new Range, 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 best choice) and click **OK**.

Add New Range Details:

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

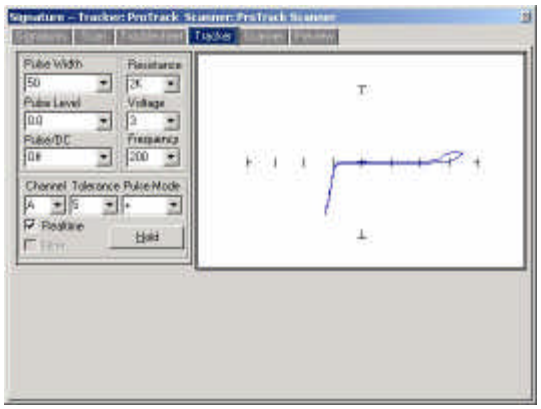
Voltage/Resistance/Frequency fields: Select the desired range settings

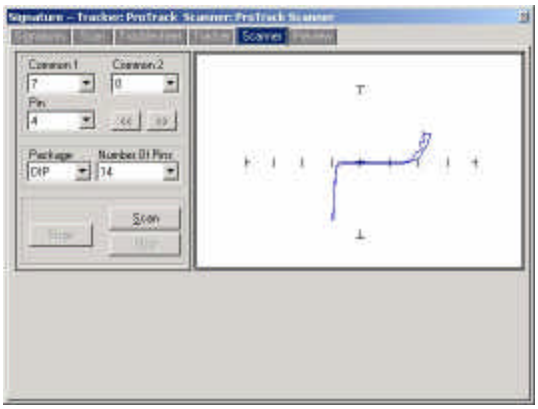
Max Samples/Tolerance/Delay: Select the setting to be used when scanning **this range**

Common Pin/Prober Common Pin: Select common pins to be used when scanning **this range**. Scanner users wishing to use a particular component pin for Common should select the Common pin at this time.

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.

ProTrack with Scanner users can now proceed with actual scanning of a component.



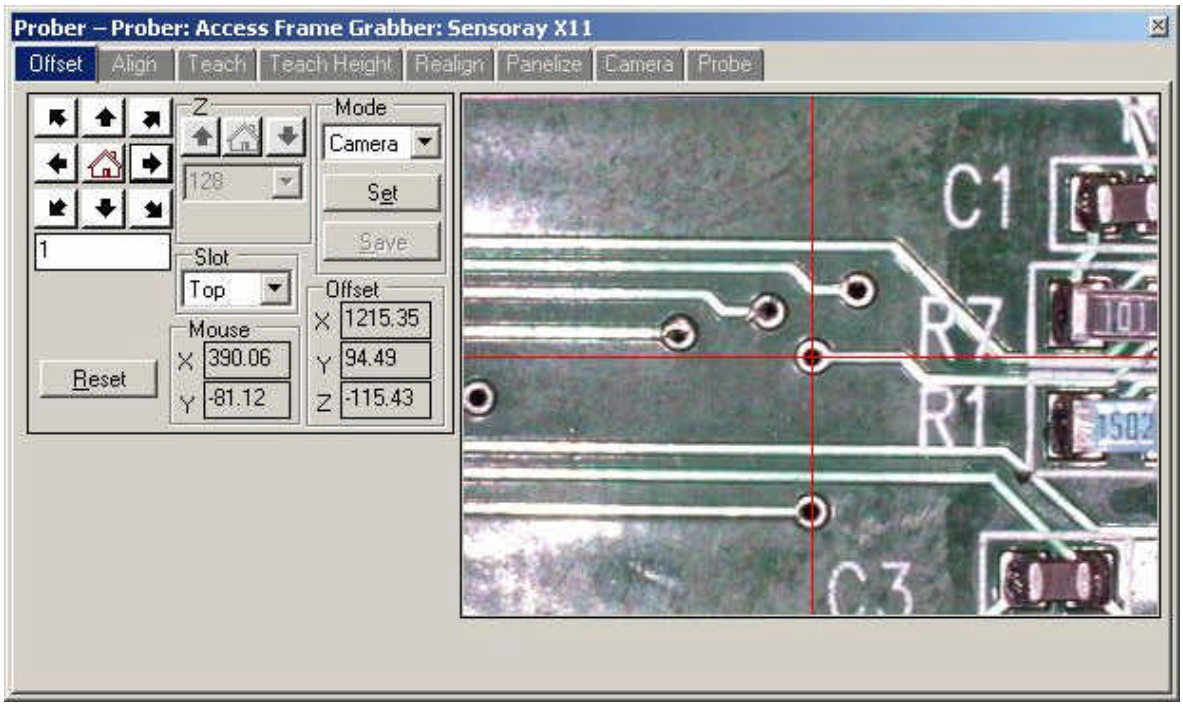


Manual Modes – Tracker and Scanner

When trying to decide on the ranges to use for your test, 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 even set the Pulse Generator settings. 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 ProTrack 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.



Prober Setup – Camera Offset part 1

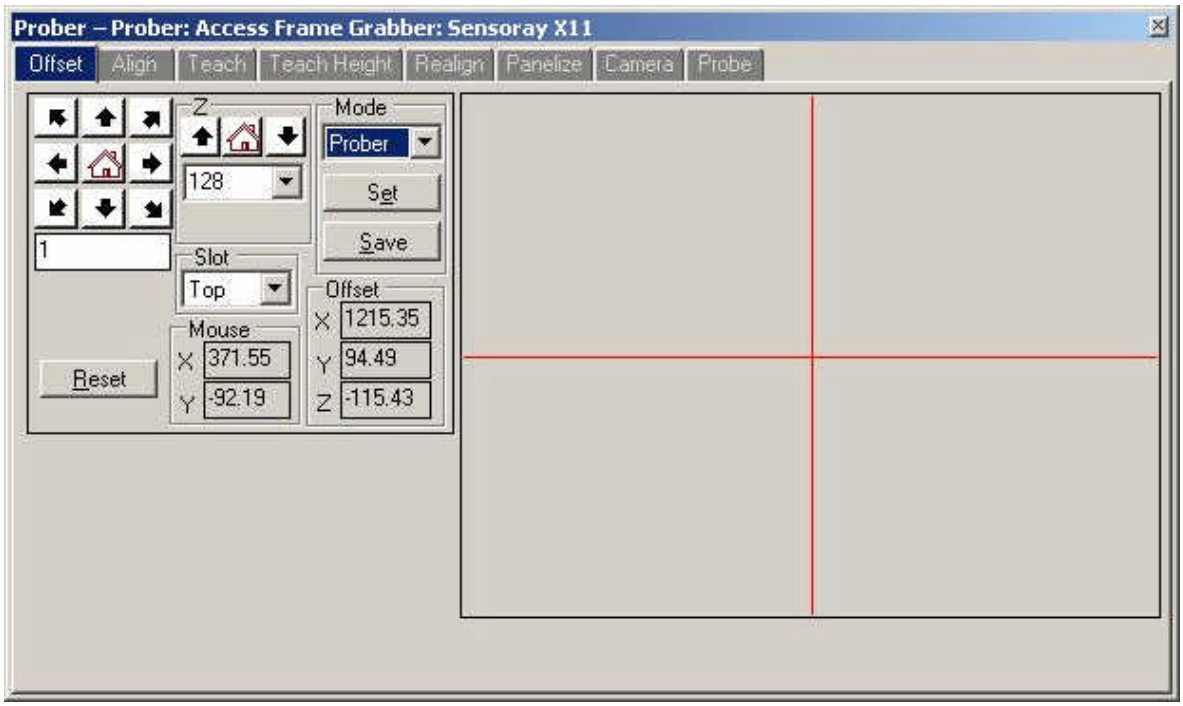
Performing the Camera Offset procedure is essential for the Prober to access test points accurately.

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

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

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

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



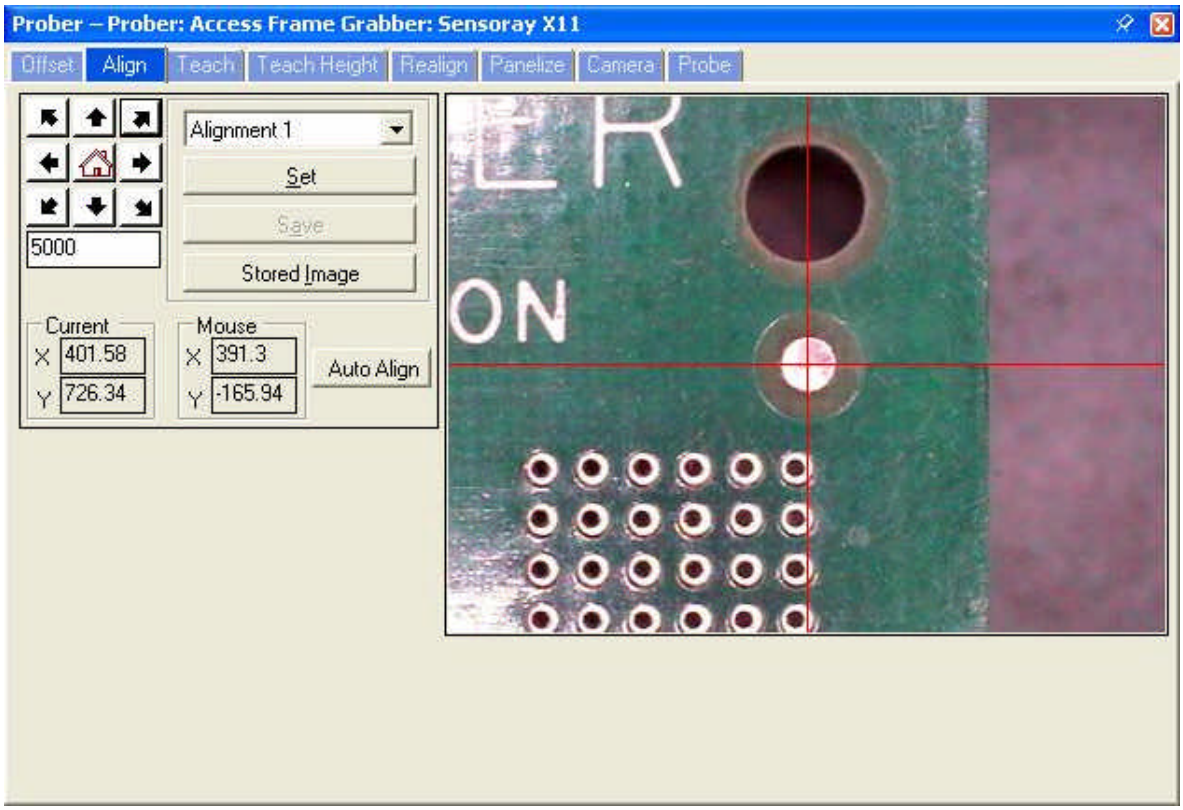
Prober Setup – Camera Offset part 2

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

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

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

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



Prober setup – Alignment part 1

Alignment points that are positioned in opposite corners of the board work best. If the test was created using CAD data, the alignment points are pre-selected and you should proceed to the Teach Height page.

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

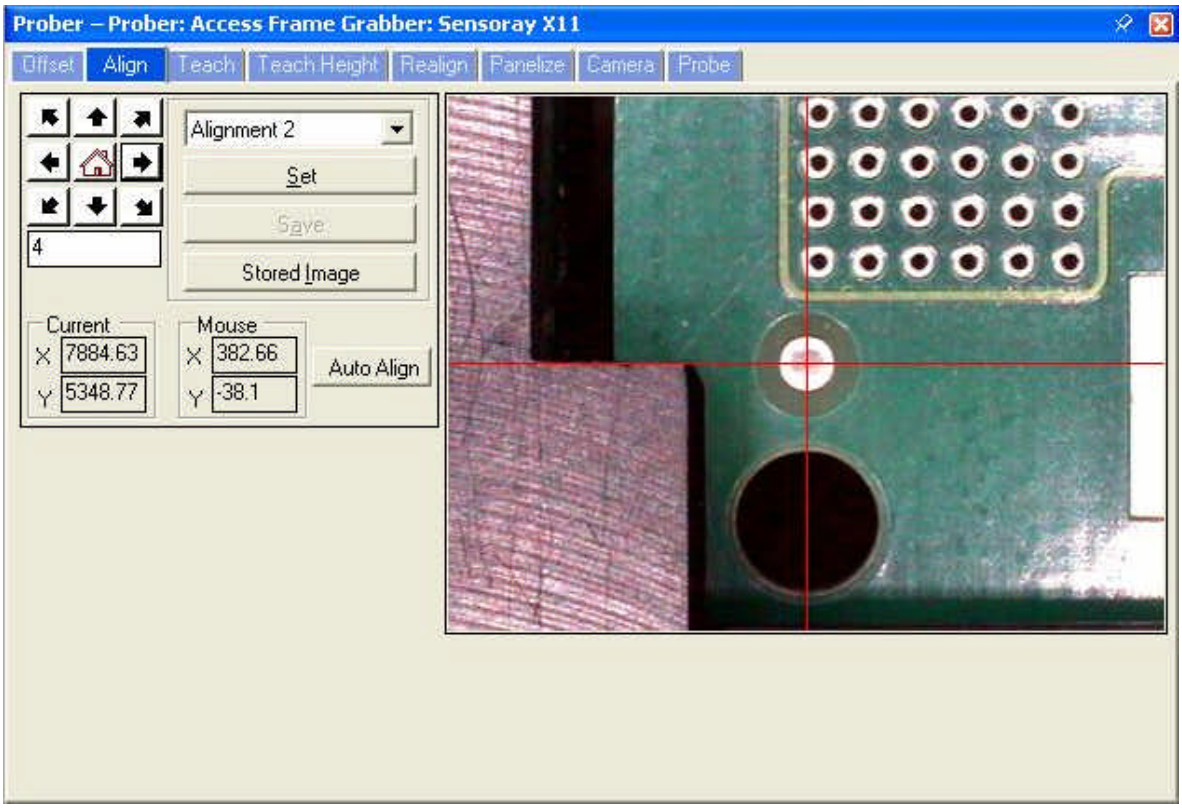
Select **Alignment 1** from the drop menu.

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

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



Prober setup – Alignment part 2

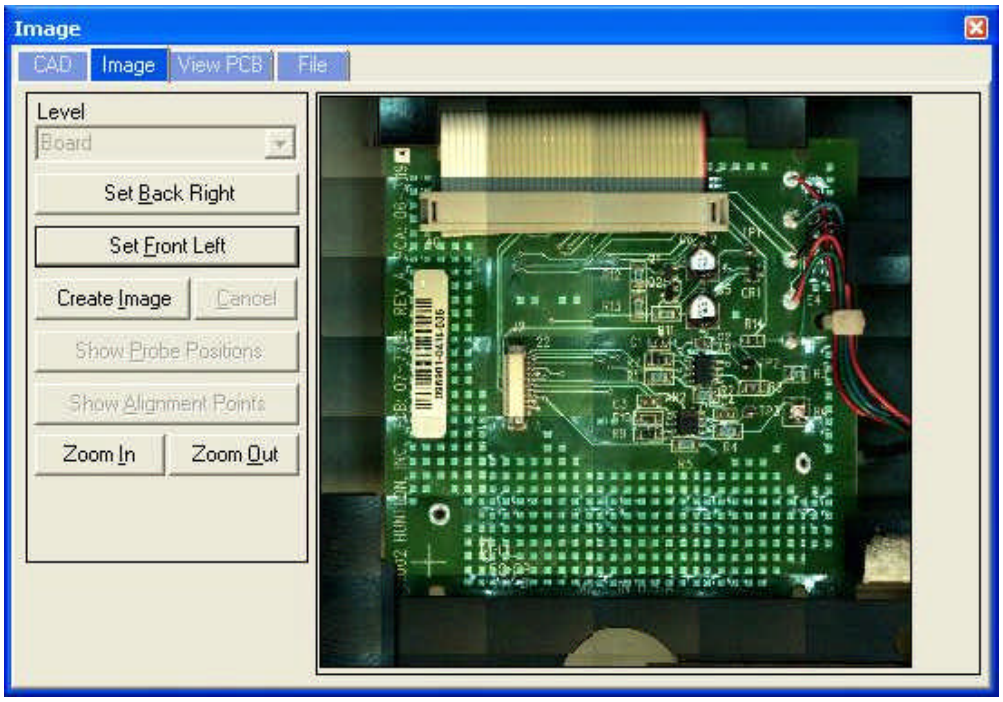
Make sure **Alignment 2** is selected in the Alignment point drop field.

Select and navigate the camera to the second alignment point.

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

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

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



Creating a Board Image

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

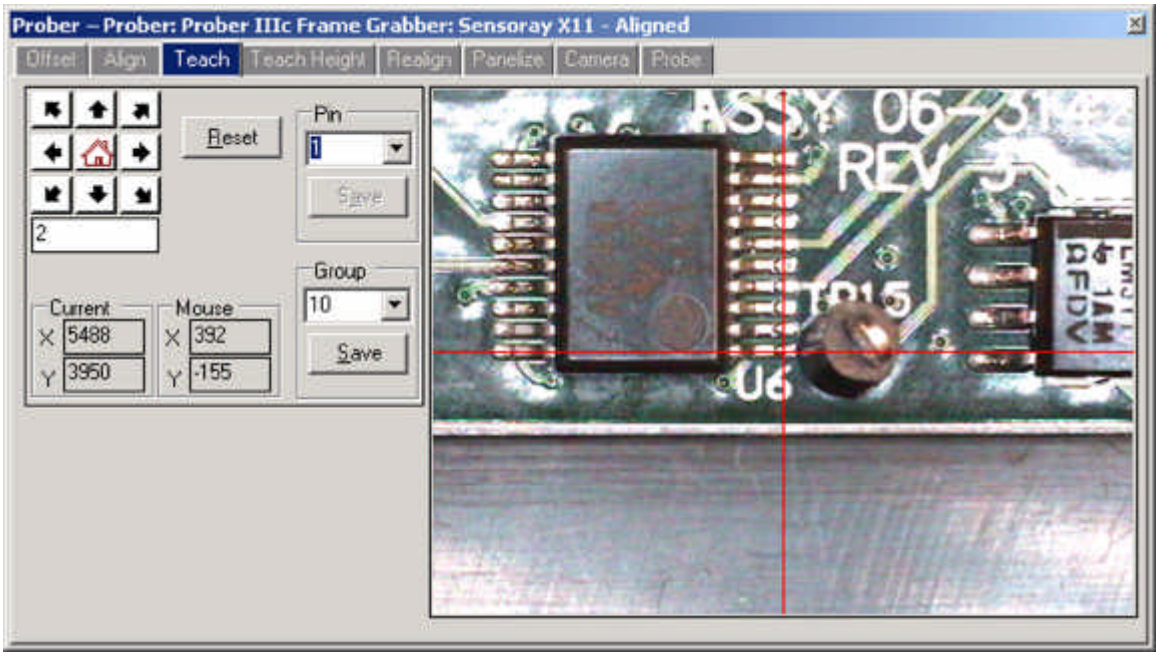
Move the camera to the front left corner of the board.

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.

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



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

Use care when teaching component pin locations. This will ensure accurate probing.

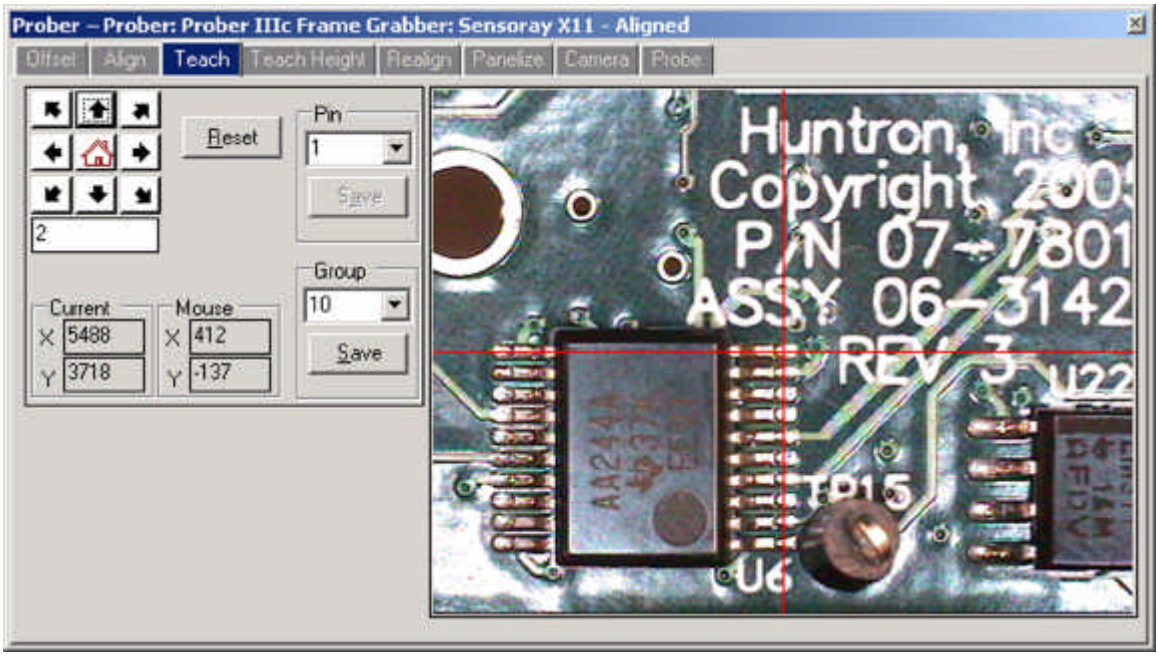
Select the **Teach** tab in the Prober pane.

Select the component to teach 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.

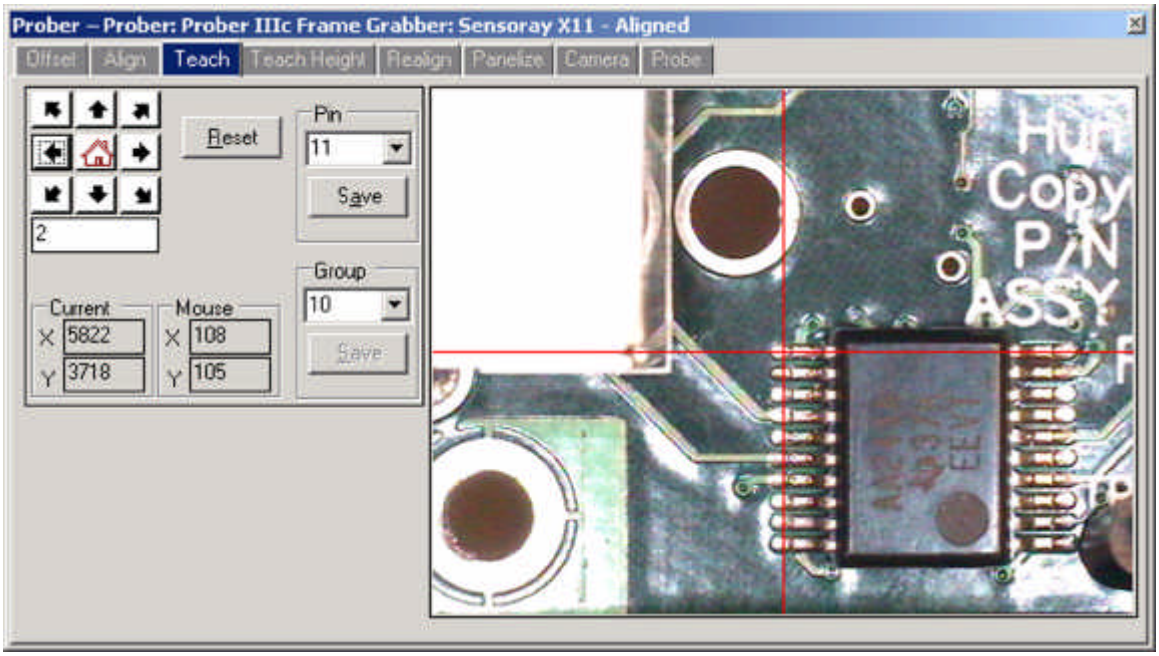


Prober setup – Component Teach; step 2 (using a DIP package example)

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

Set the number of pins in the row in the **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 under the Group drop menu. This sets and calculates the positions of the remaining pins of the first row.

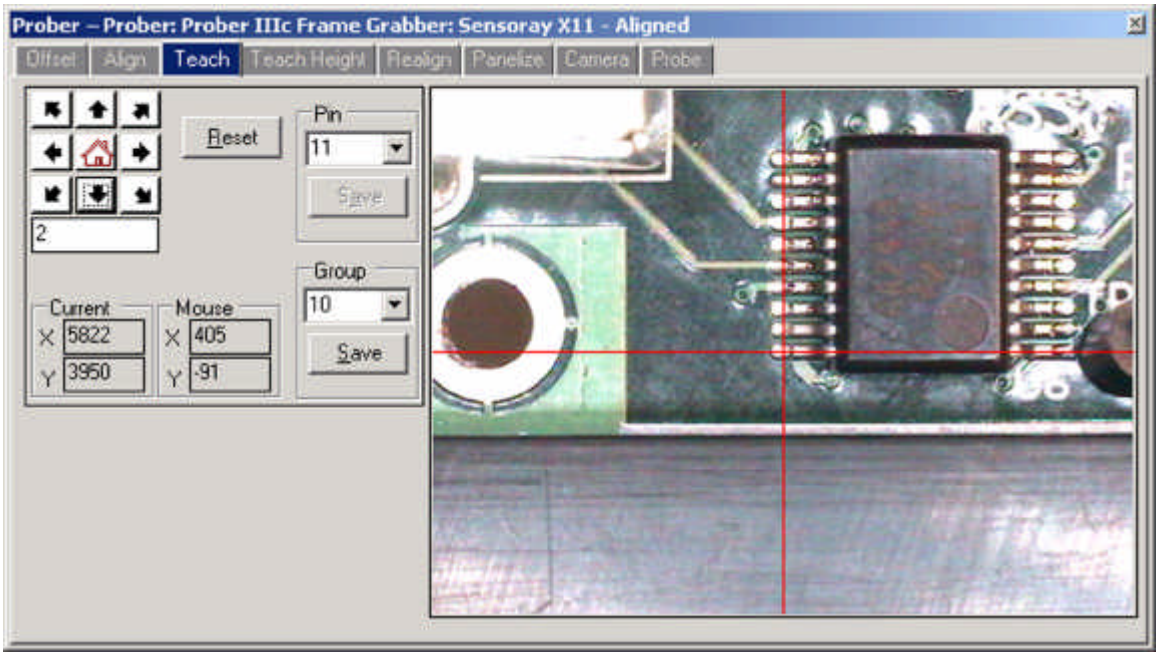


Prober setup – Component Teach; step 3 (using a DIP package example)

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.



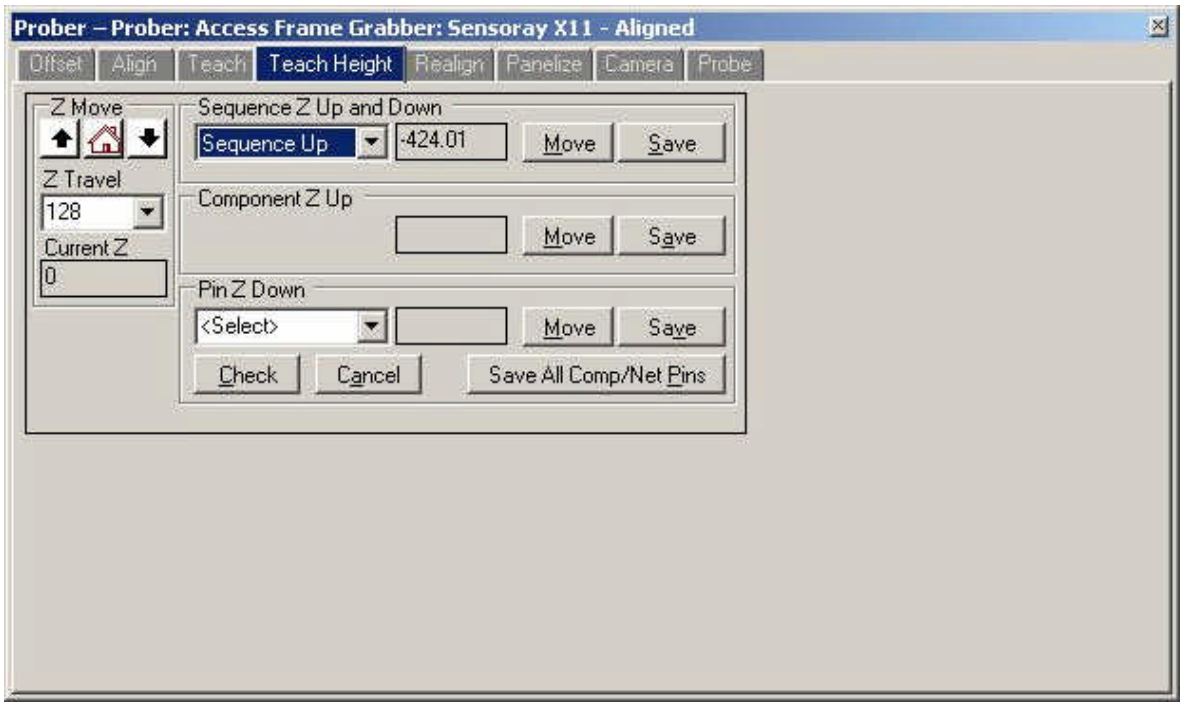
Prober setup – Component Teach; step 4 (using a DIP package example)

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

Set the number of pins in the row in the **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 under 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



Prober setup – Component Teach Height; step 1

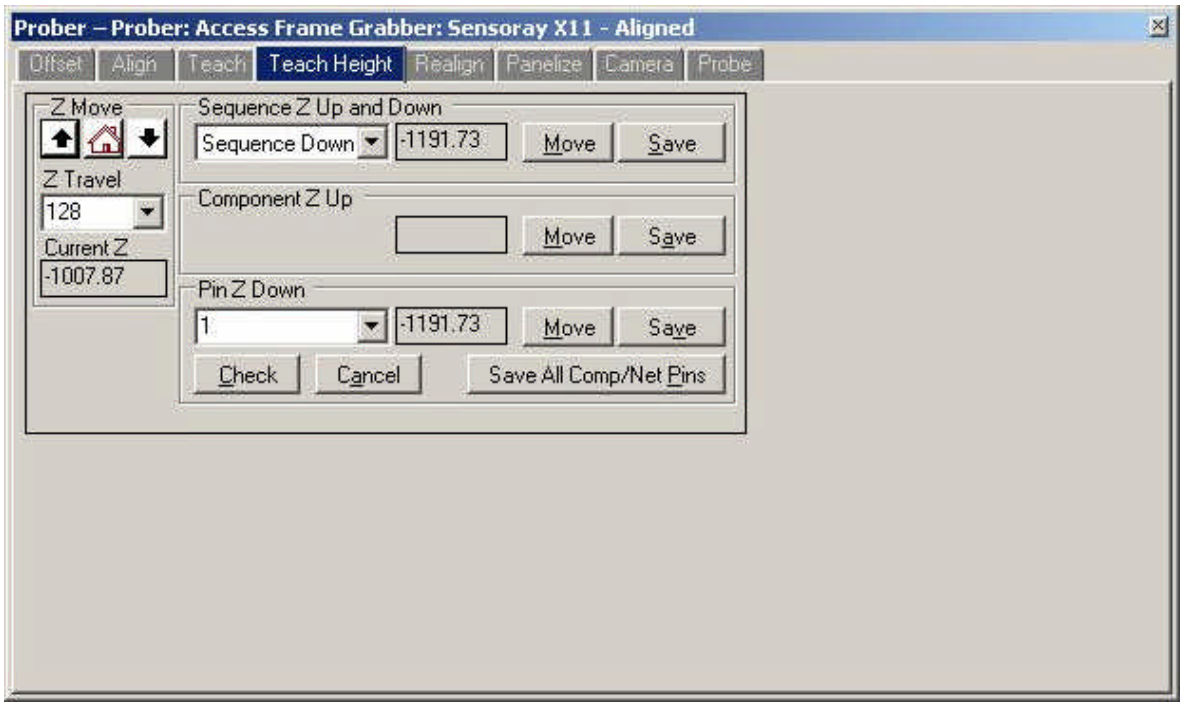
Select the **Teach Height** tab in the Prober Pane.

Select **Sequence Up** from the Sequence Z Up and Down drop menu.

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.

Press the **Save** button in the Sequence Z Up and Down section to set the Sequence Up position.

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



Prober setup – Component Teach Height; step 2

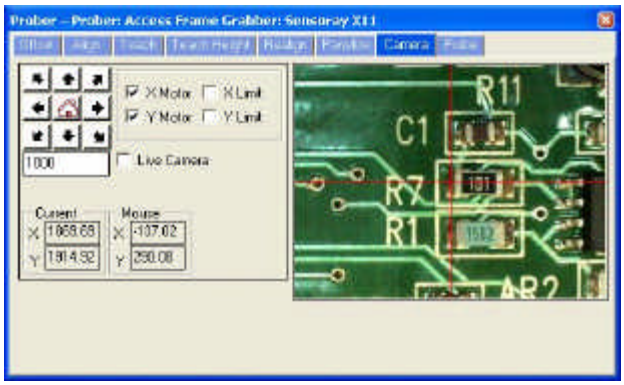
Select a position on the board to use for setting the down position. Select a pin in the Pin Z Down section of the tab.

Select **Sequence Down** from the drop menu in the Sequence Z Up and Down section. 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.

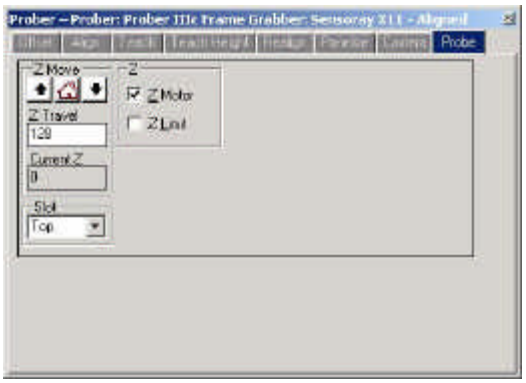
Press the **Save** button in the Sequence Z Up and Down section to set the Sequence Down level. This is the level the Prober will use when probing the component pins.

You can also set a separate height for the components by using **Component Z Up**. Move the probe to the desired Up position for the component and click **Save**.

If needed, you can also set Pin Down levels for individual pins using the Pin Z Down section. In general use, this is not necessary as the Down level is set at the Sequence level. To set an individual pin, select the pin number in the **Pin Z Down** drop menu. Use the **Z Move** buttons to set the desired probe tip level and press the **Save** button to set the level for the selected pin. Press the **Save All Comp/Net Pins** button to set all of the component or net pins to this down level.



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

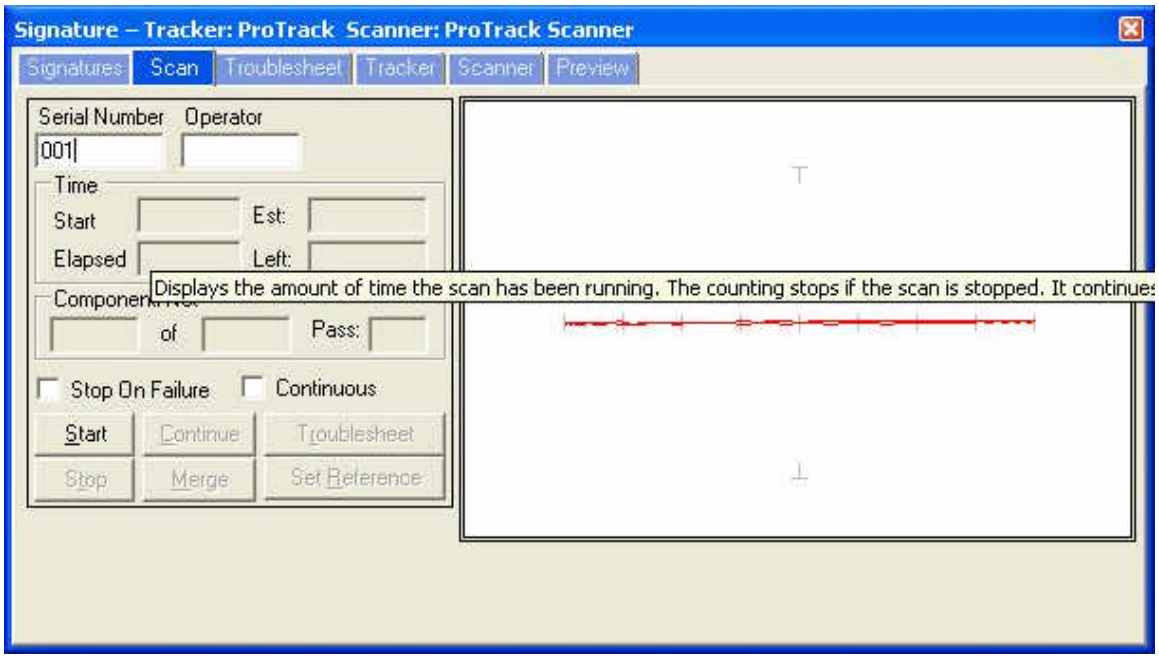


Use the Z Move buttons to lower the probe to the point targeted by the camera.

Manual Modes – Camera and Prober

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.

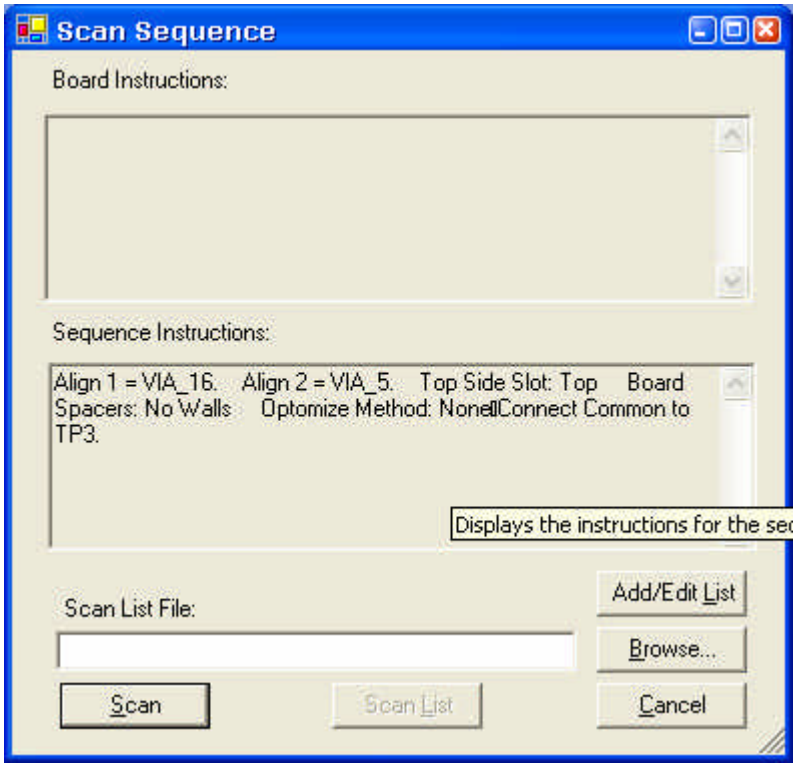
The **Stop On Failure** check box will cause the scanning to stop if a component comparison fails. This is most useful when scanning known good boards to add as references.

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

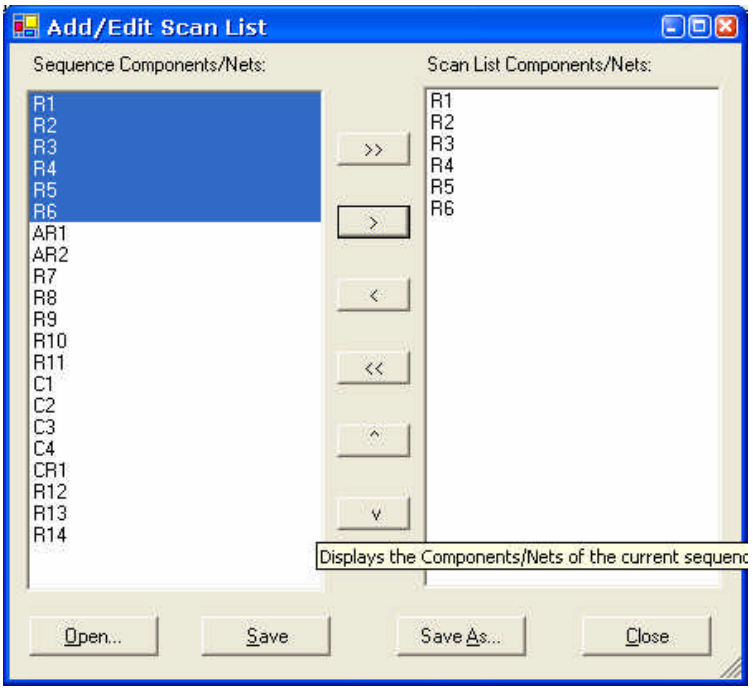
Press the **Start** button to execute the scan.

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

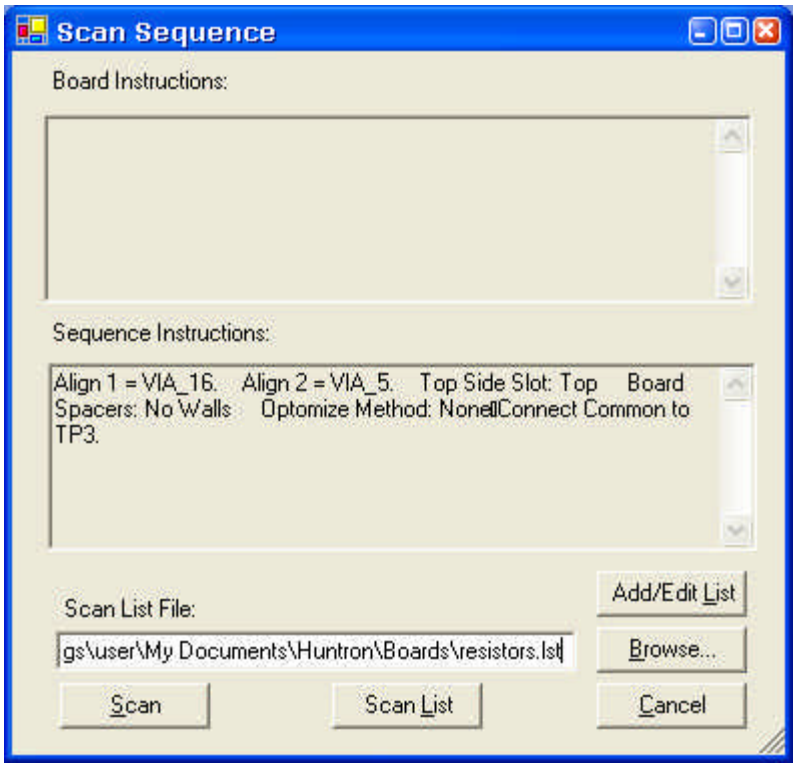
At this point, you have the ability to create a Scan List. A Scan List is simply a list of components to be scanned saved to a text file. To create a Scan List, click the **Add/Edit List** button in the Scan instructions dialog.



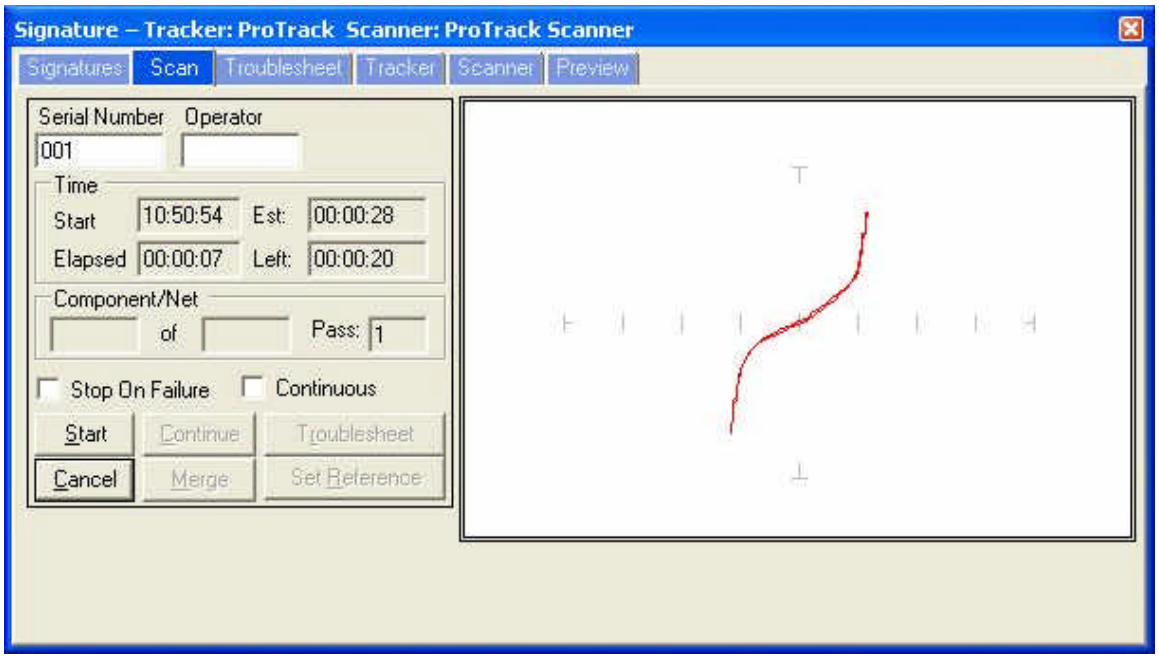
A new dialog will open called Add/Edit Scan list.



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** to save the 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.

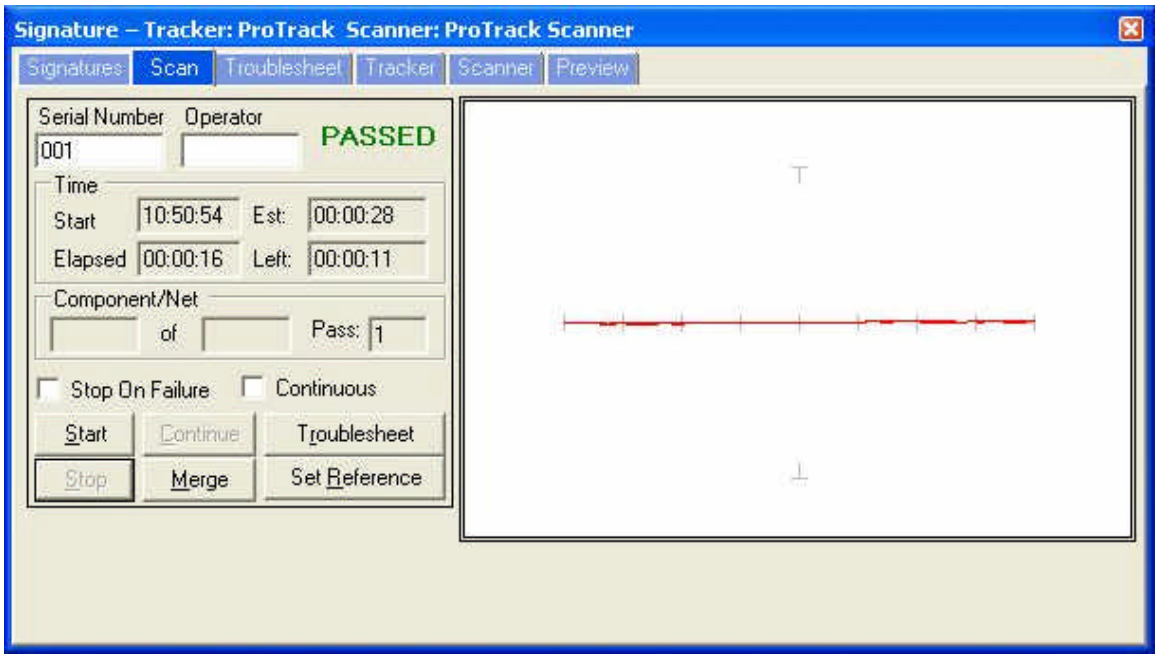


Click **Scan** to start scanning the Sequence or Component selected in the Tree pane or click **Scan List** to scan the selected Scan List File.



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



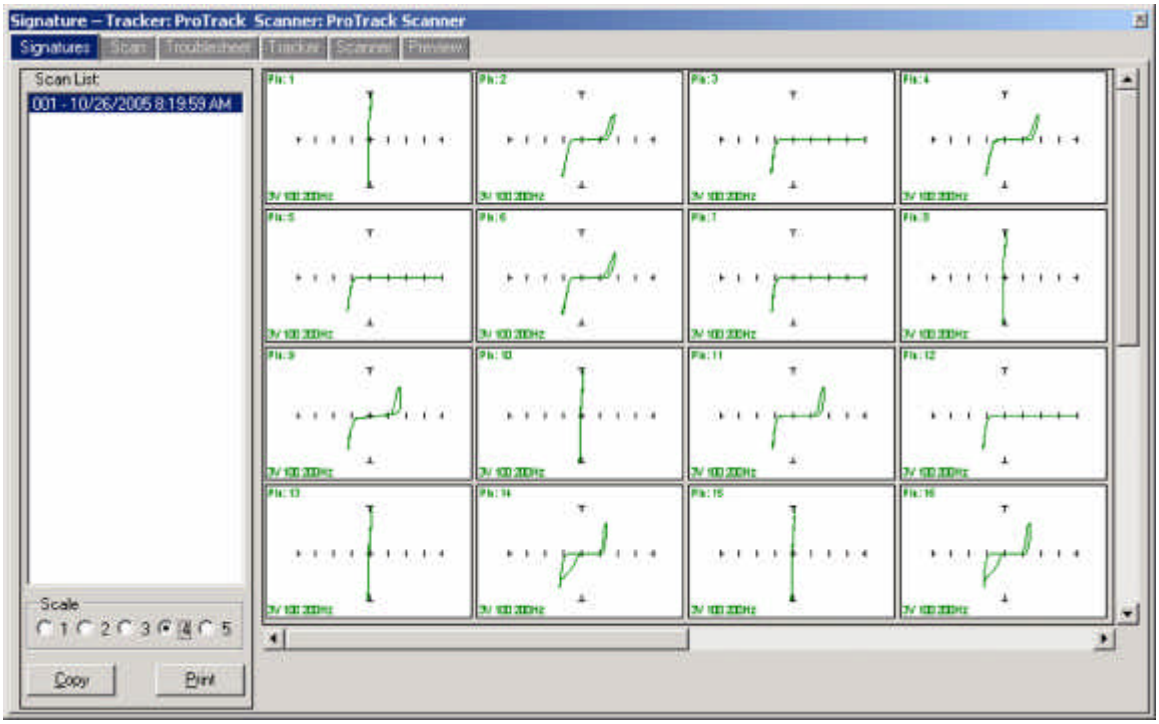
Scan Results

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

To mark a signature set (entire scan) as a Reference, press the **Set Reference** button. This will mark this set of signatures to be used as a known good comparison set when scanning other boards.

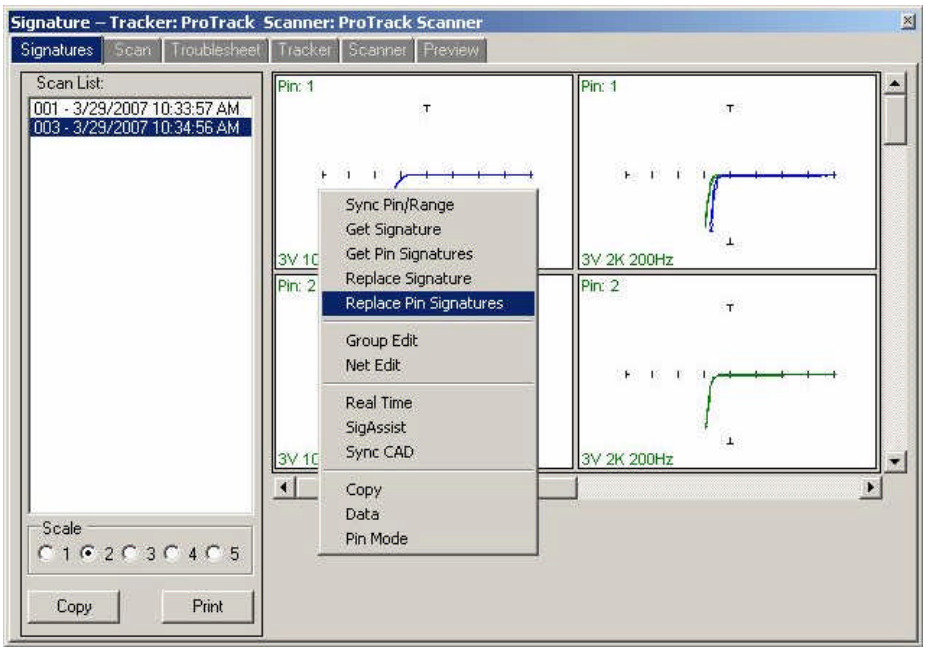
To mark a signature sets a Merged set that can have subsequent signatures added or "merged", press the **Merge** button. Use the Merge function only when potential differences between Merged signature sets are very small.

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



Viewing Signatures

To view scanned signatures, 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.



Viewing Signatures – Right Click

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

Sync Pin / Range: Selects pin and range in the Tree

Get Signature: Capture signature

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

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

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

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

Real Time: Display signature in real time

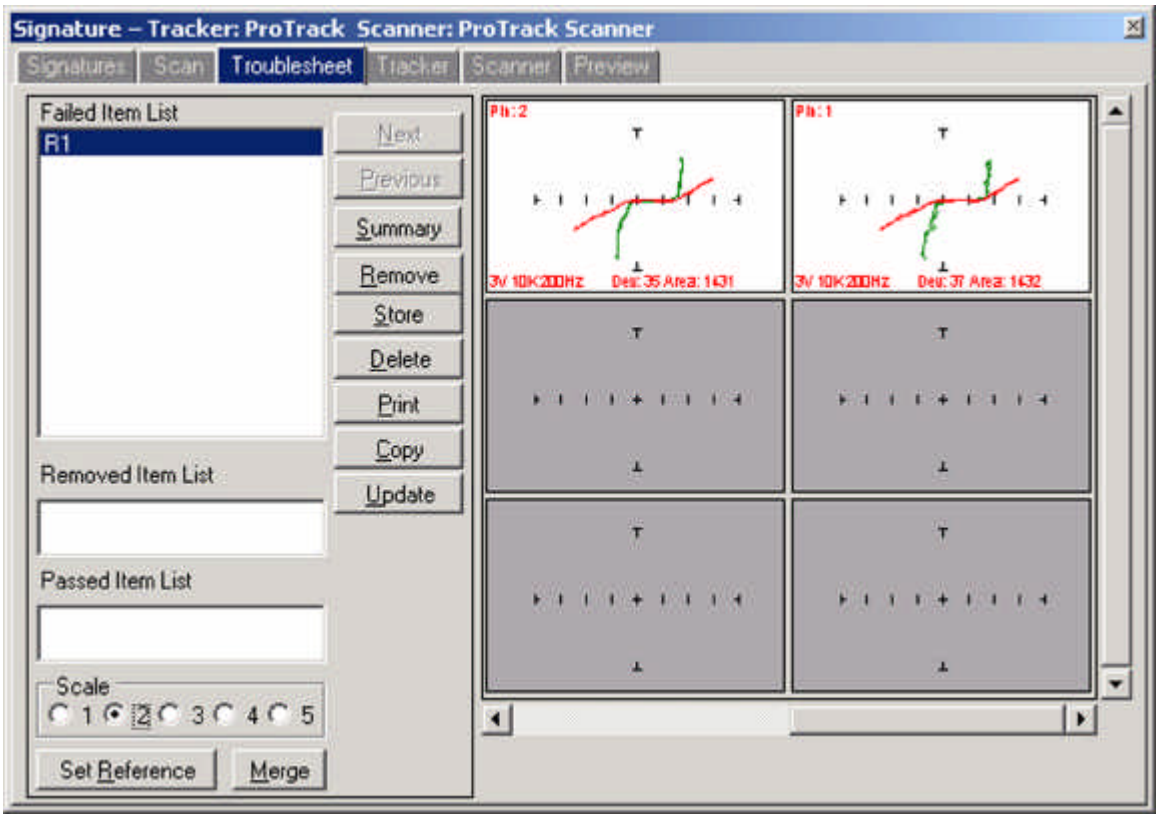
SigAssist: Displays the SigAssist window showing numeric values resistance, capacitance, breakdown voltage, etc. based on signature information

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.

Copy: Places the signature image in the clipboard

Data: Displays the signatures numeric data

Range / Pin Mode: Displays signature with Range priority or Pin priority



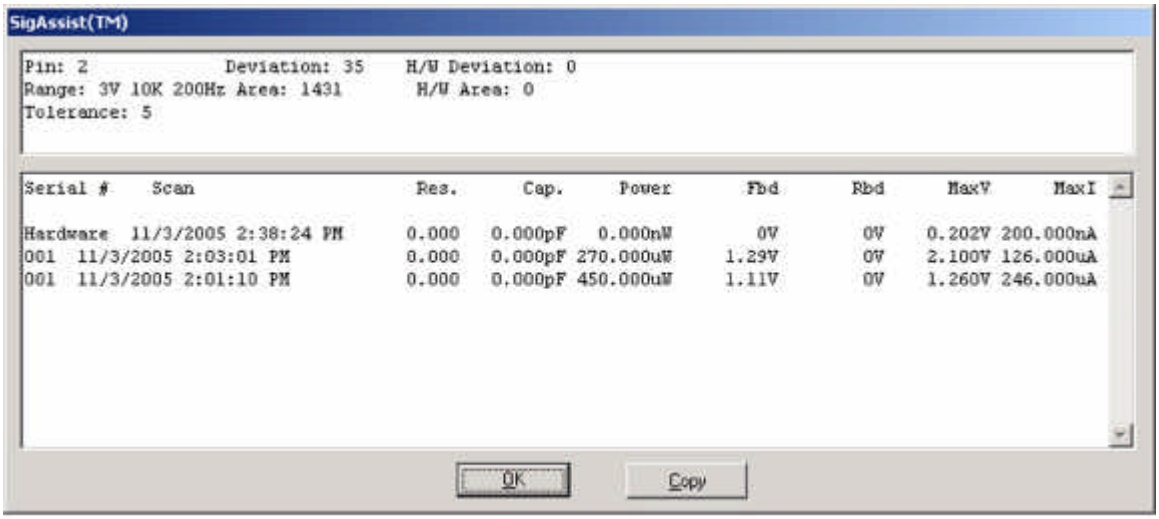
Viewing Signatures – Troublesheet

When a scan is executed signatures are compared against the Reference set (or sets) if it exists. Any signature differences are logged to the Troublesheet. Signatures of the compared signatures are displayed in contrasting colors. By default, the Reference signature is green and the failed signature is red.

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.

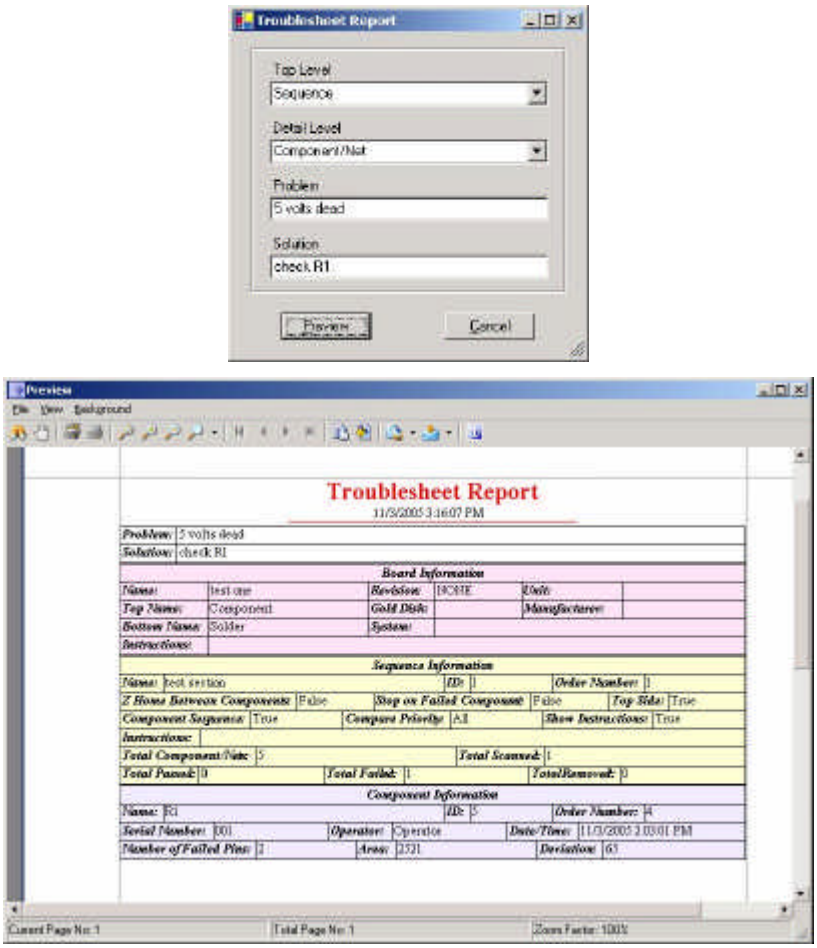
To add the signatures in the Troublesheet as a Reference, press the **Set Reference** button (do this only with known good signatures).

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).



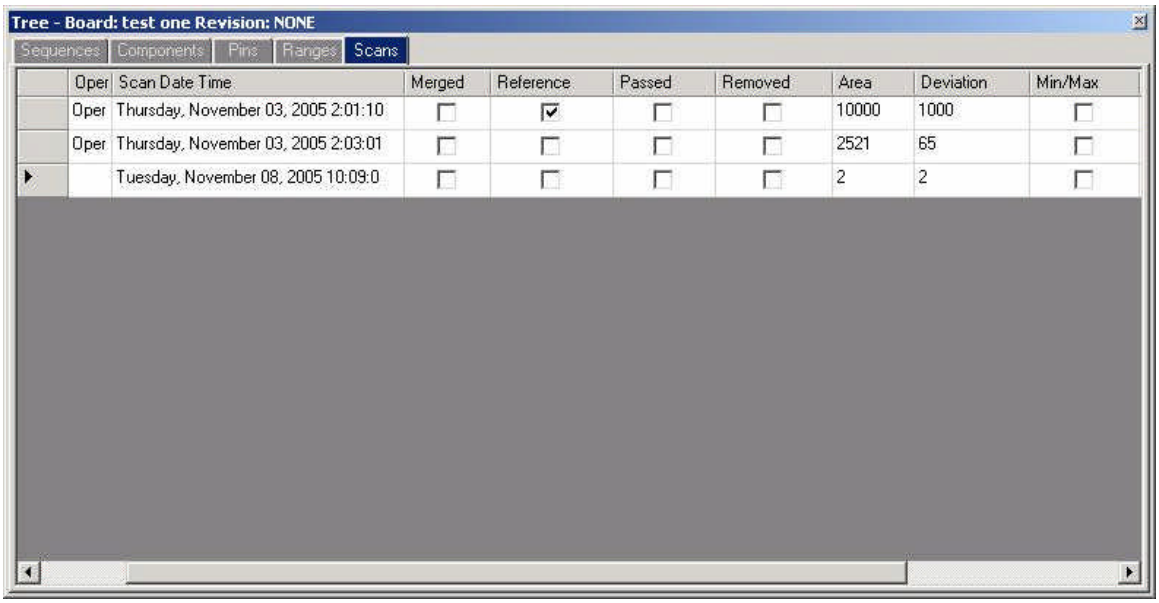
View SigAssist Information

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.



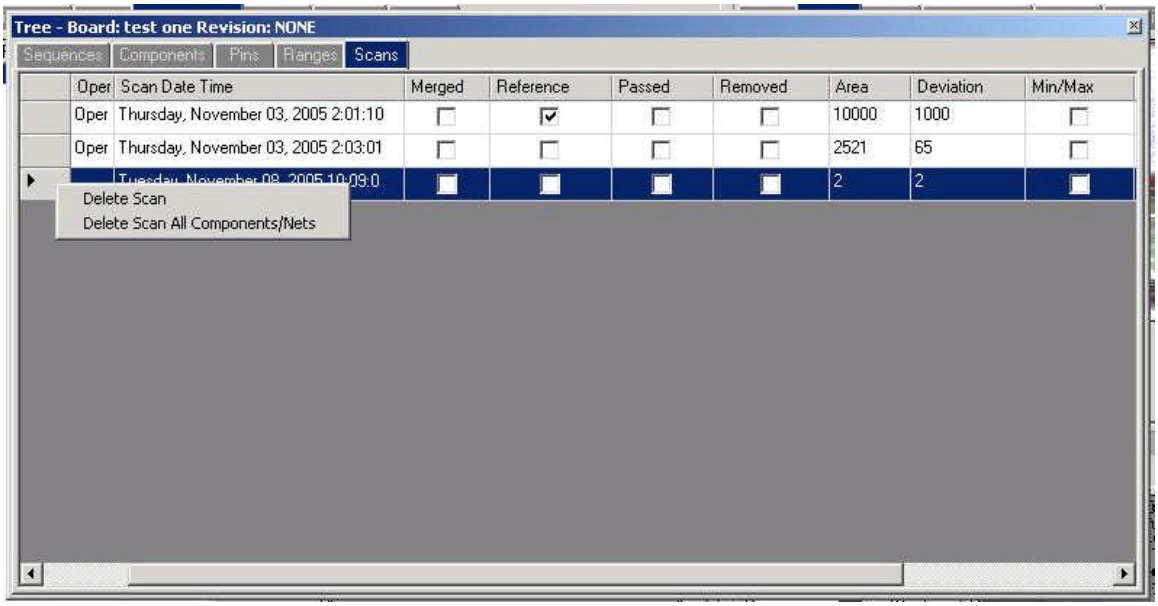
Viewing Signatures Troublesheet Report

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 saved in several different formats including HTML and PDF.



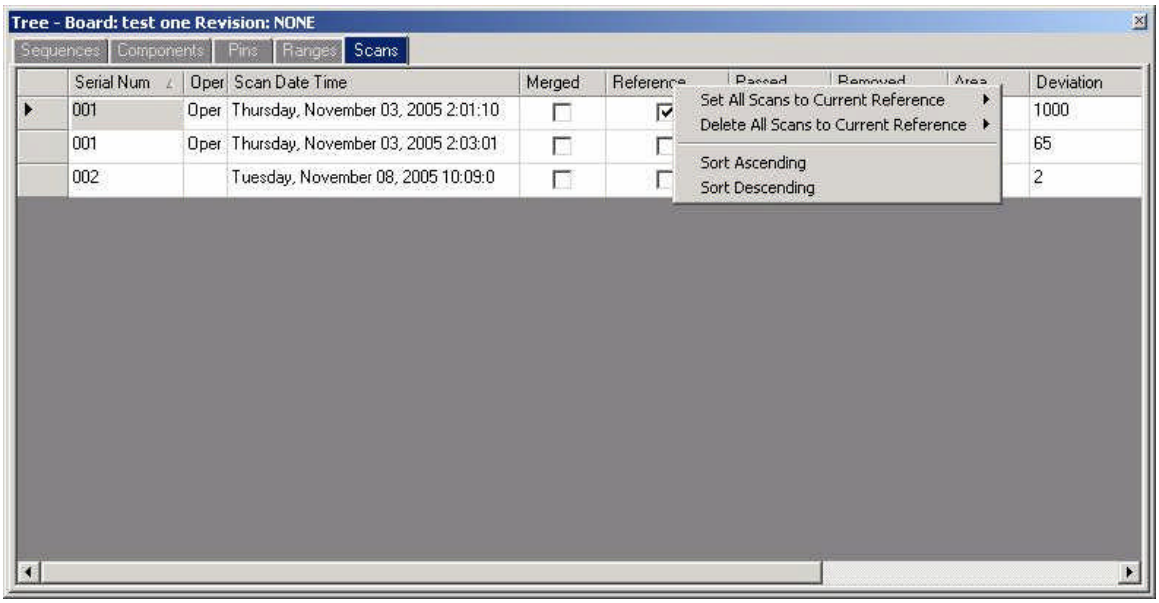
Component Scans Information

Selecting the **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).



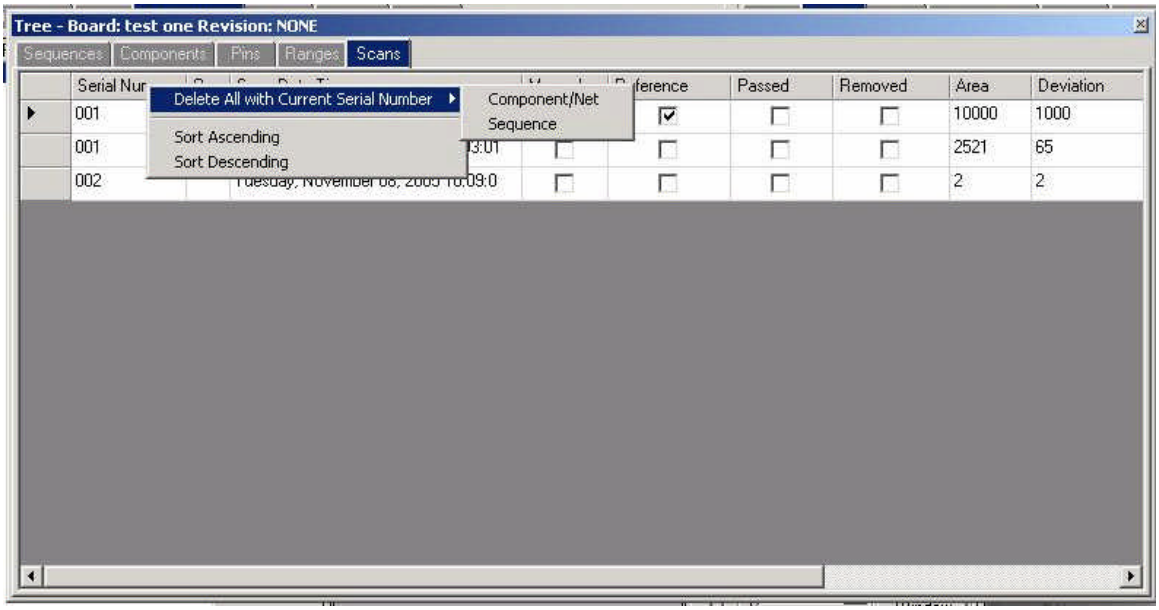
Component Scans – Right Clicking (Auxiliary menus)

Right clicking on the Row bar of the Scan pane displays an additional menu for performing specific tasks such as deleting a scan for the selected components or all components. **Delete Scan** will erase the selected scan. **Delete Scan All Components/Nets** will erase all scans for the selected component or net.



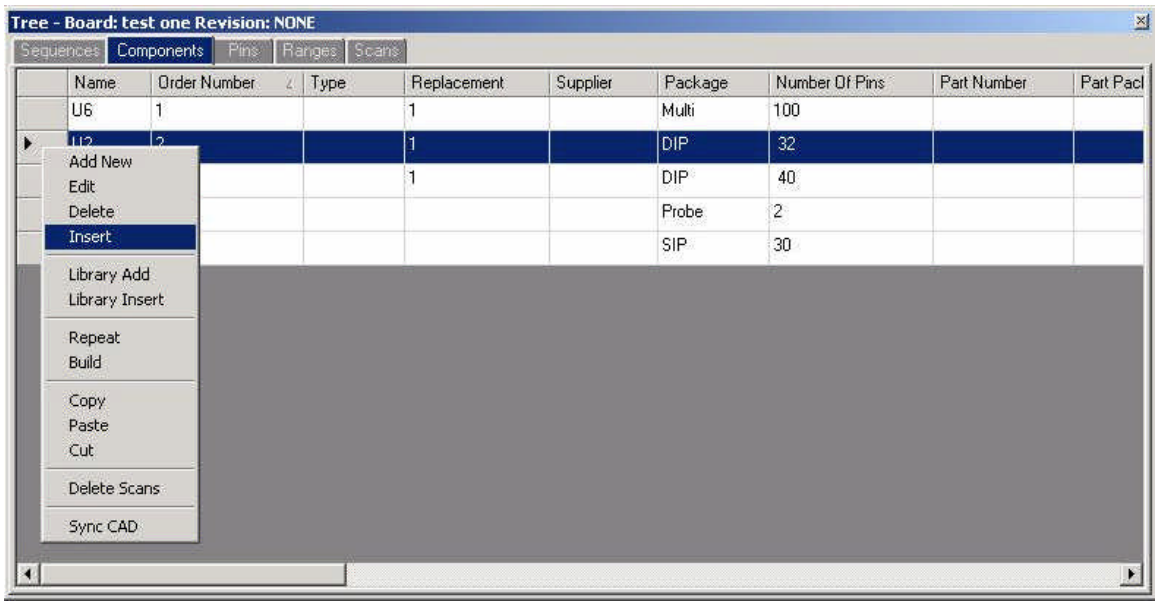
Component Scans – Right Clicking (Auxiliary menus)

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



Component Scans – Right Clicking (Auxiliary menus)

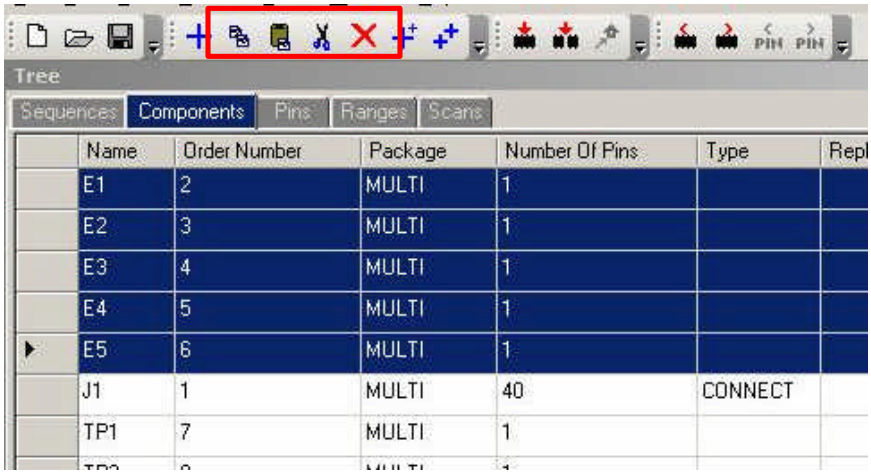
Right clicking on the Column bar of the Scans pane displays an additional menu for performing specific tasks such as sorting or globally deleting items (such as scans based on serial number shown above)

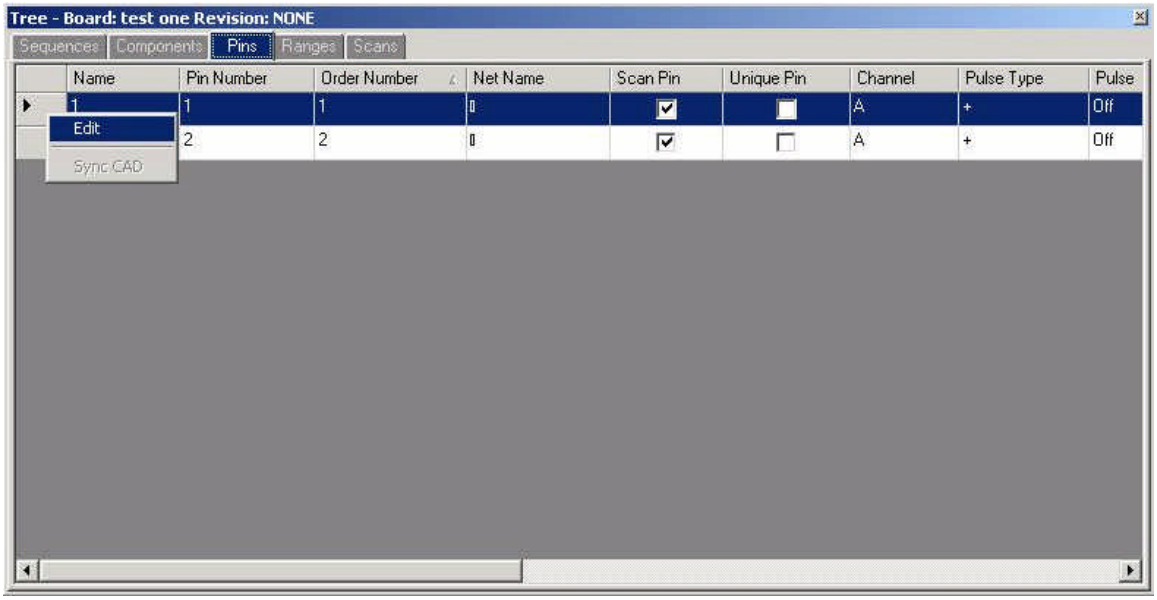


Sequence/Component/Pin Editing – Right Clicking (Auxiliary menus)

Right clicking on the Row header bar of the Tree pane displays an additional menu for performing specific tasks such as **Add New**, **Insert**, **Delete**, **Repeat**, **Build** (repeat but with name incremented by 1), **Copy**, **Paste** and **Cut**. Right clicking works at all levels of the Tree pane including Sequence, Component (shown above), Pins, Ranges and Scans. The **Sync CAD** option available at the Component and Pin levels will synchronize the selected component or pin to the CAD image in the Image pane.

It is also possible to select groups of components by holding the **SHIFT** or **CTRL** keys while selecting components in the Tree. To copy, paste, cut or delete groups of components, use the Copy, Paste, Cut or Delete Toolbar buttons below the main menu.

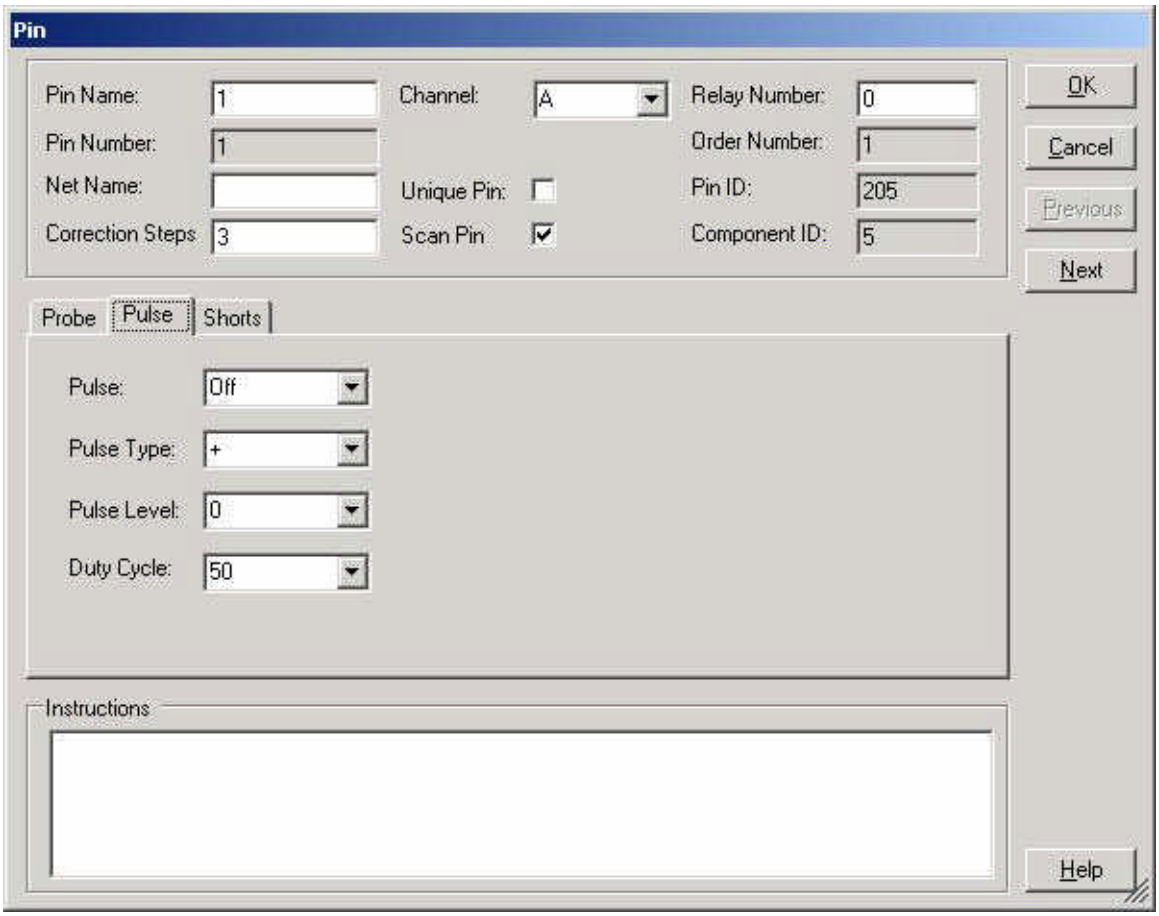




Pin Editing – Right Clicking (Auxiliary menus)

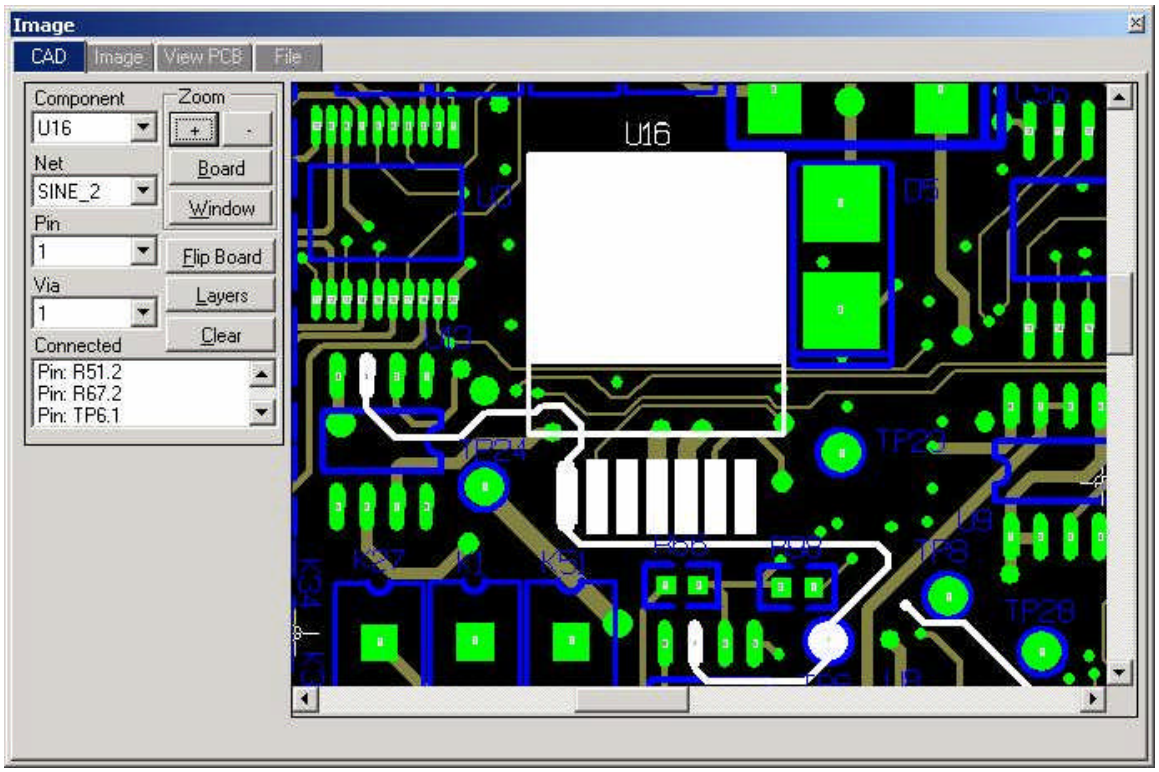
Right clicking on the Row header bar of the Tree/Pins pane allows for editing of the pin information. Select **Edit** to open the Edit window for the selected pin.

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



Pin Editing – Right Clicking (Auxiliary menus)

Individual pin characteristics can be edited in the Pin Edit window. **Net name**, **Channel**, **Correction steps** (for Access Probers), **Scan Pin** (enable/disable scanning of the pin) can be controlled along with controls for the Pulse Generator and Shorts Check. Shorts Check is available for Scanner use only.

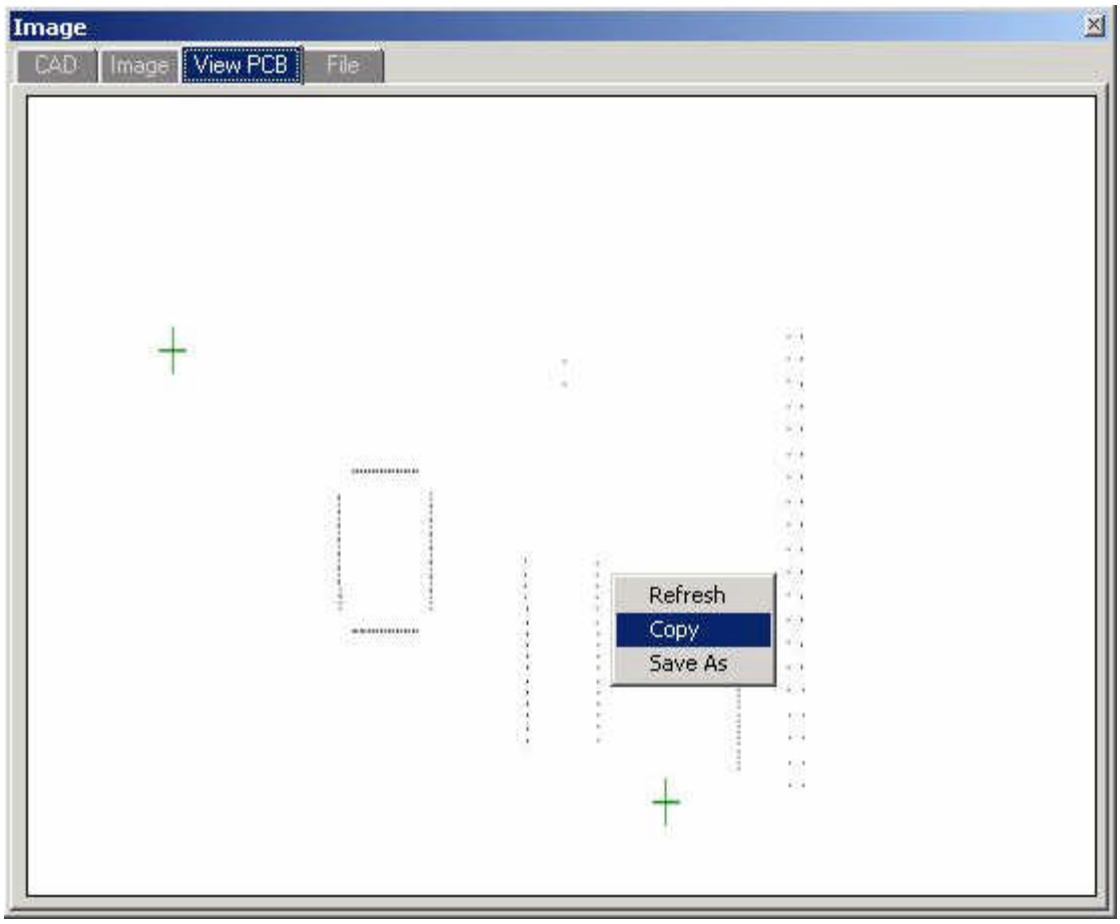


Adding CAD data in the Image Pane

CAD data can be displayed by selecting the **CAD** tab in the Image Pane. Before the board layout can be displayed, a .CC (CAMCAD) file must be loaded. This is accomplished in the Board Edit window by clicking the "**Load CC...** " button and browsing to the associated CADCAM file. Once selected, the board layout will appear in the Image/CAD Pane.

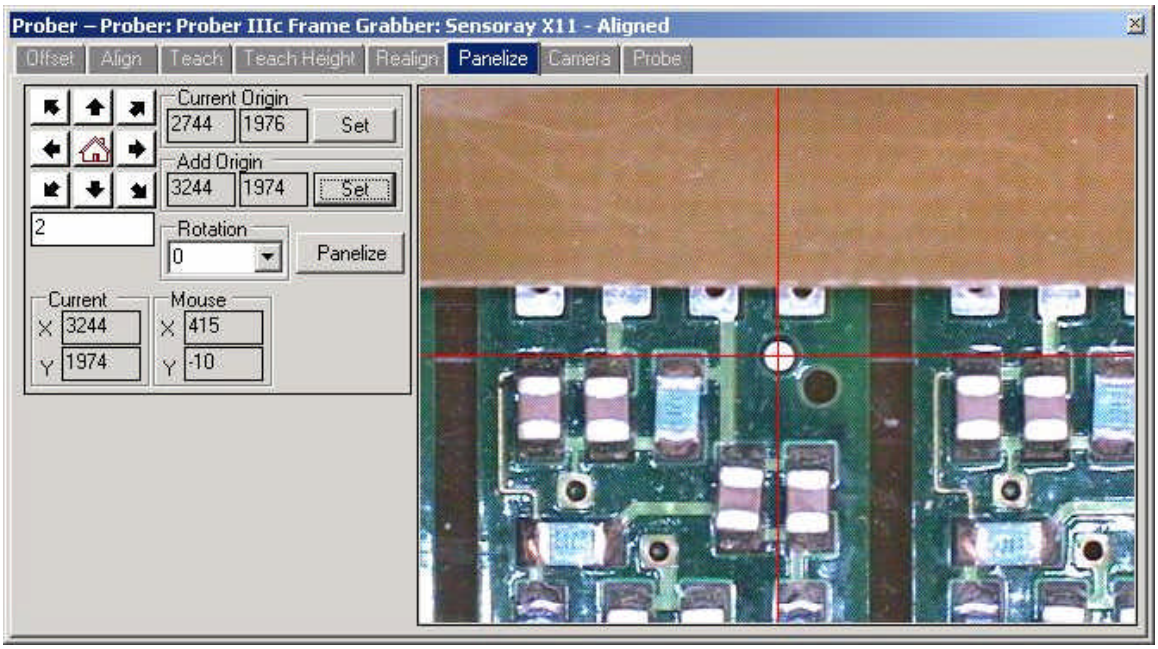
The **Board** button sets the zoom level to view the entire board. **Window** allows you to use a drag box in the CAD image to zoom in on a selected location. **Flip Board** displays the other side of the board. **Options** displays the Layer selection window. **Clear** unselects any highlighted components or nets.

The CAD information is automatically loaded if the test was created using CAD data or if File/Convert was used to convert a test created in version 3.X that had CAD data attached.



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.



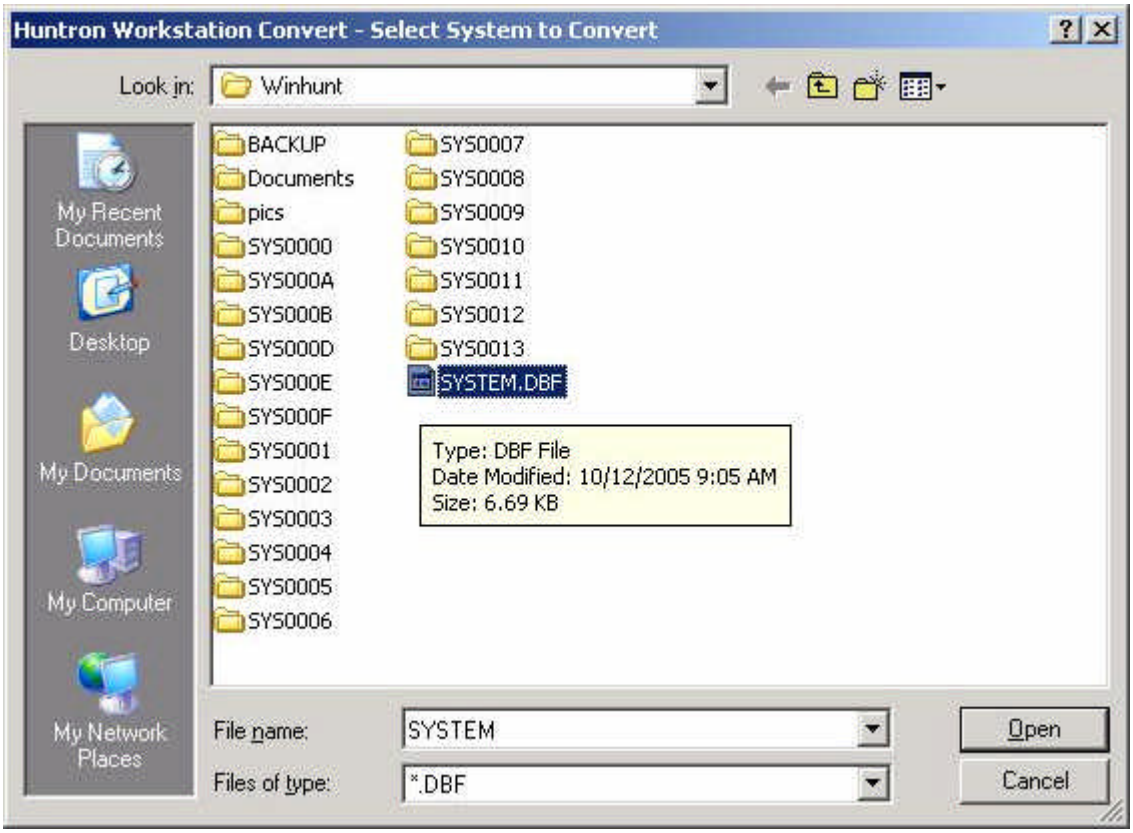
Test Creation - Panelize

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

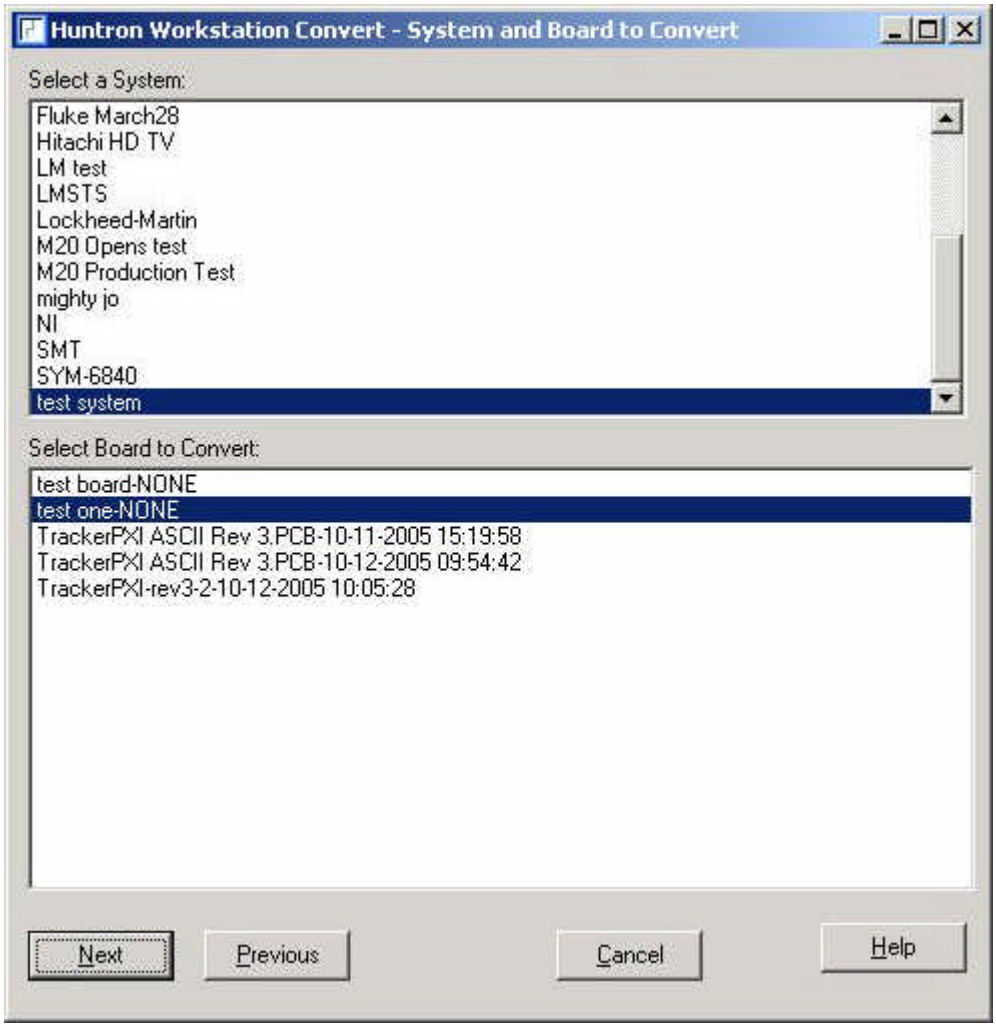
Move the camera to the same point on the next panel and press **Set** in the Add Origin area. The XY coordinates of the second origin will be displayed. Set a rotation value in the **Rotation** field.

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.**

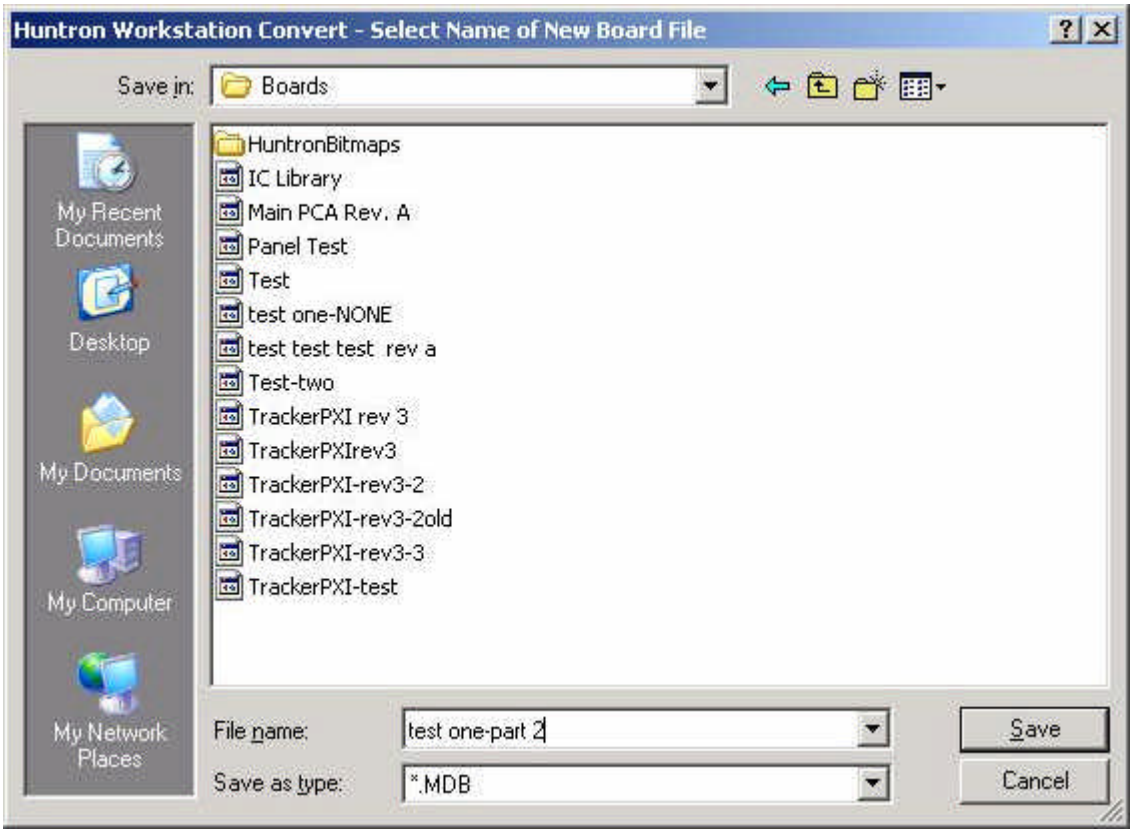


Test Creation – Convert step 2

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.



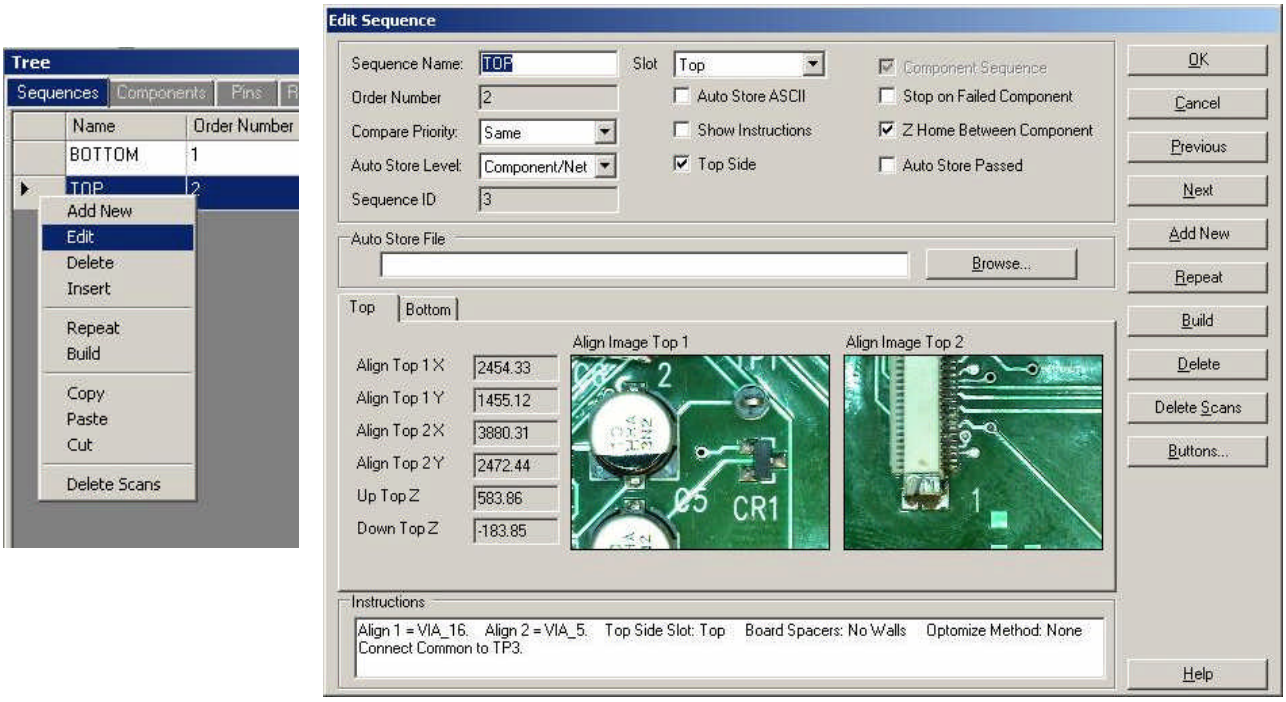
Test Creation – Convert Status

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.

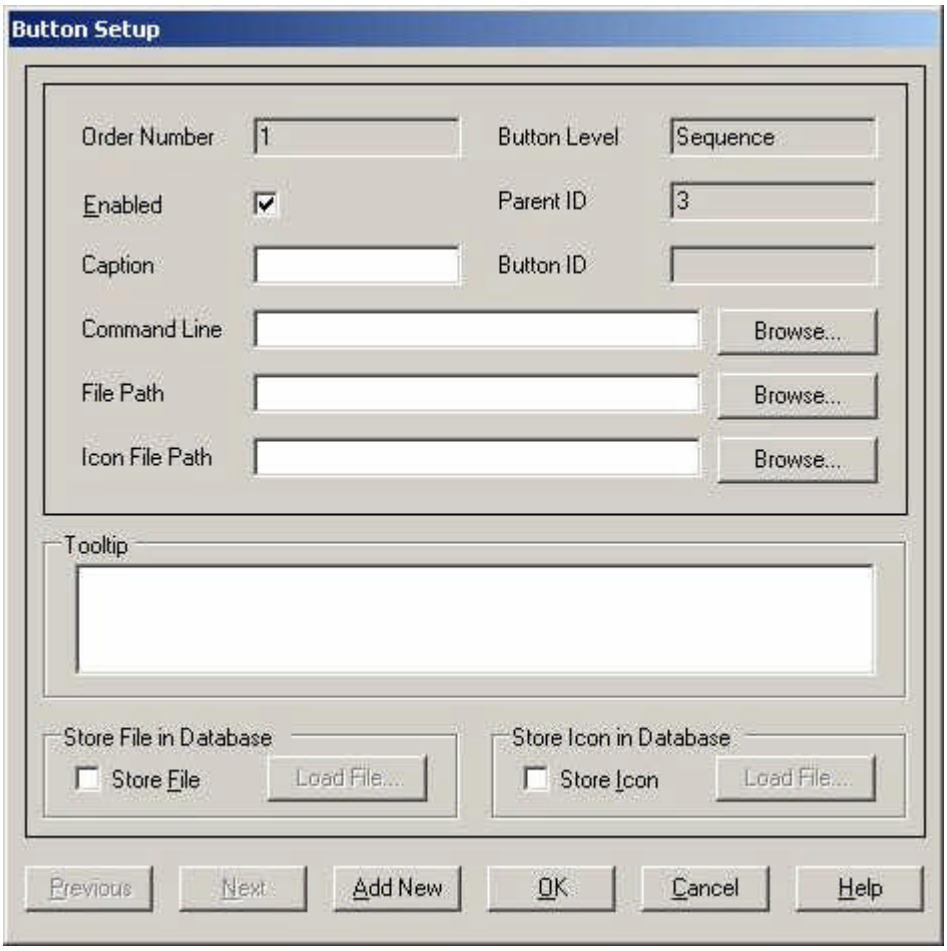
Huntron Workstation Buttons Feature

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.

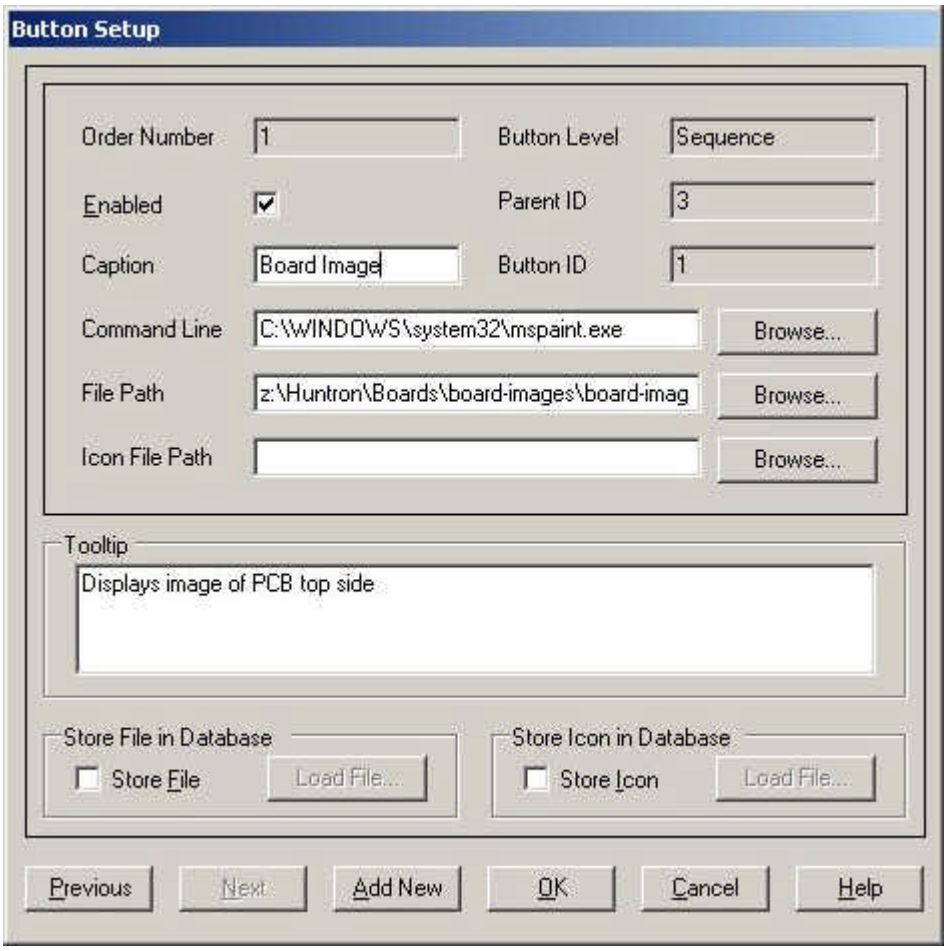
To start, the Sequence edit window is opened by right-clicking the row header for the desired sequence. The Edit window will be displayed (see below).



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

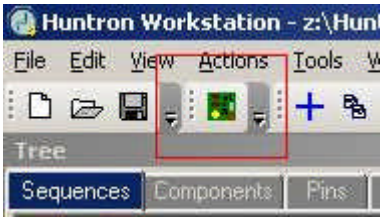


To configure the Button, a Caption (button name), Command Line (the application to be run) and File Path (the file opened by the executed application) need to be input into the corresponding fields. A File path will not always be required such as instances where a browser is started and pointed to a 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. 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.