Huntron Workstation CAD Import Tutorial

Contents

Huntron CAD Tools Tutorial

The tutorial will guide the user through the steps require to create a Huntron ASCII File (HAF) and a CAMCAD (CC) data files. These files contain X-Y coordinates that allow the Huntron Prober to accurately probe a print circuit assembly.

Installing the CAD Tools

1. Power up your computer. Insert the software CD into the CDROM drive.
2. If your CD was configured for Auto-run, a menu will be displayed. Otherwise run the setup.exe program.
3. Select "Install Products"
4. Select a Source and Target for installation; it is recommended to use the default locations
5. In the Product selection menu, select CAMCAD Professional only, do not install any other application (see image)
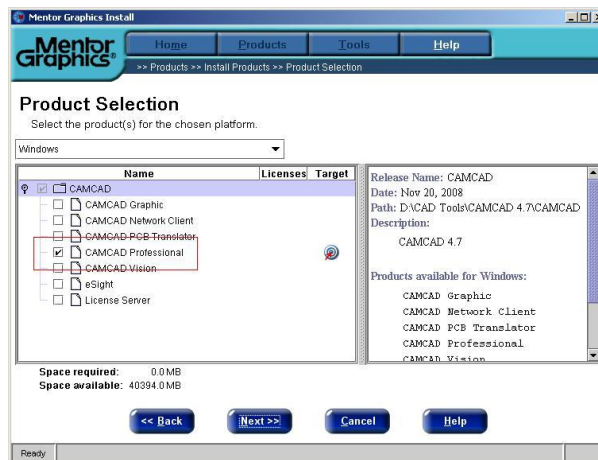
6. After the installation is complete, exit from the CAD Tools CD.

Create a Board database

After starting the Huntron software the main menu is displayed. Select **File/New** from the menu bar. The Add New Board window will be displayed. Type a name into the Name field. Adding information to the additional fields (i.e revision, system, etc…) is not necessary to create a new board. Click **OK** to save the new Board information.

File Import

After created a Board database, select **File/Import** from the menu bar.
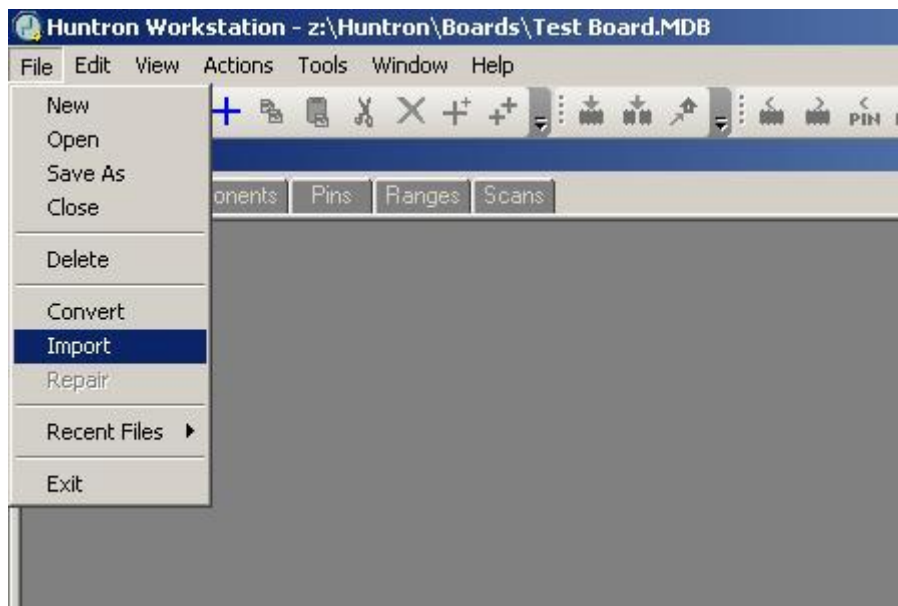
The HAF Import wizard will start and ask you whether a HAF (Huntron ASCII File) has been created. Click **No** to continue.


The HAF Assist dialog box along with the CADCAM software will start. HAF Assist is located on the left side of the screen and CAMCAD is positioned on the right. As you progress through the HAF Assist process, buttons will enable acting as a step-by-step guide. The first is to Import the CAD file.

HAF Assist

HAF Assist helps automate the process of importing a PCB layout and creating the associated files. Each of the processes within the HAF Assist has help information. The five required steps within HAF Assist are Import CAD, Clean Display, Select Align Points, HAF Export and Save CC File.
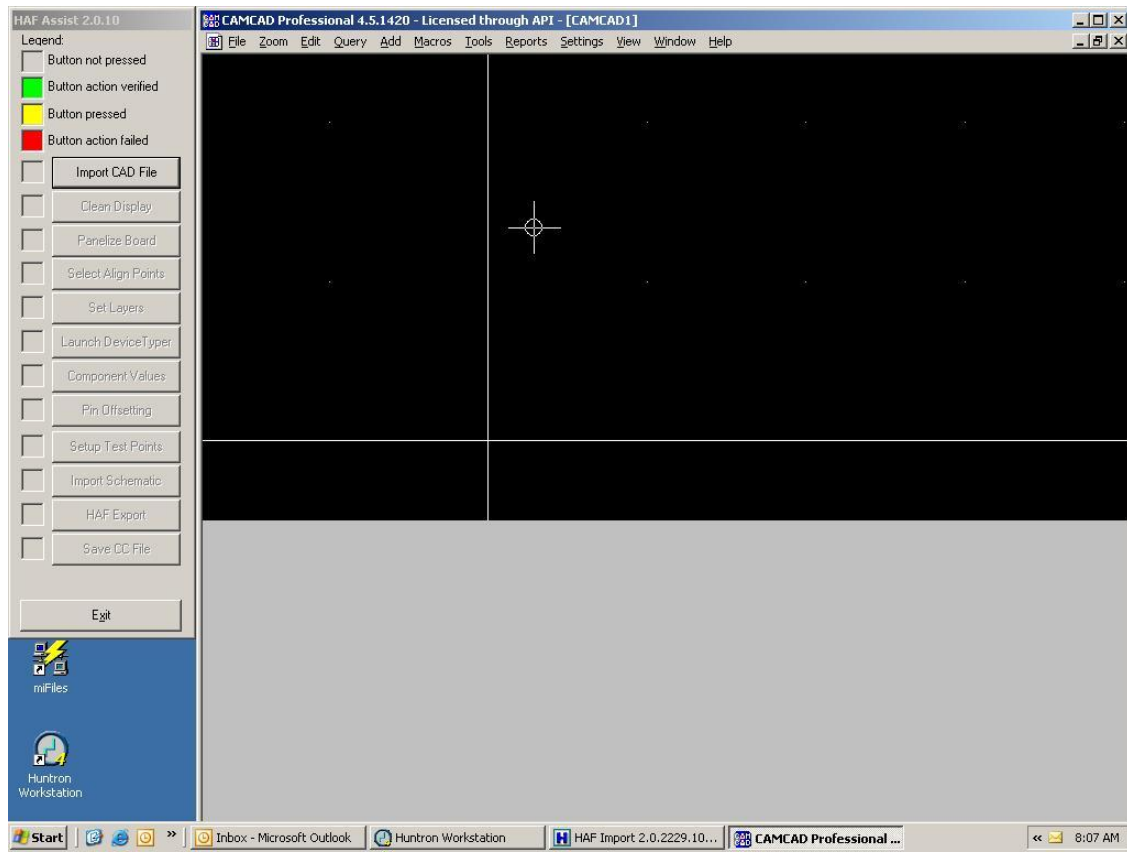
Import CAD

In HAF Assist click on the **Import CAD File** button to start the Import CAD File process. A help file and individual function buttons will be displayed. Click **Import CAD File** and the Import File Format box opens up.
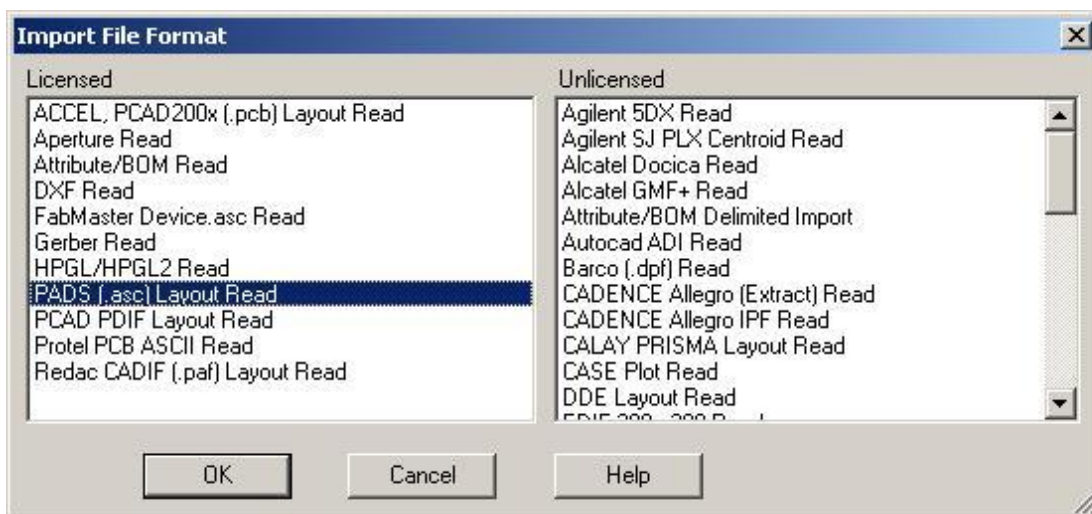
On the left hand side is a list of the licensed formats that are available with the purchased CAD tools. Select the format on the left side that is appropriate for the CAD file that will be imported. In this example the PADS (.asc) file format is chosen. Click **OK** when a format is

selected. Browse to the directory where the PCB CAD data file is located. After finding the file click **Open** and the file will be imported into CAMCAD. A sample PADS file called 2700_LCD_REVA.asc can be found in the My Documents/Huntron/CAD Files directory. The PCB drawing of your board should now be displayed in the CADCAM window.
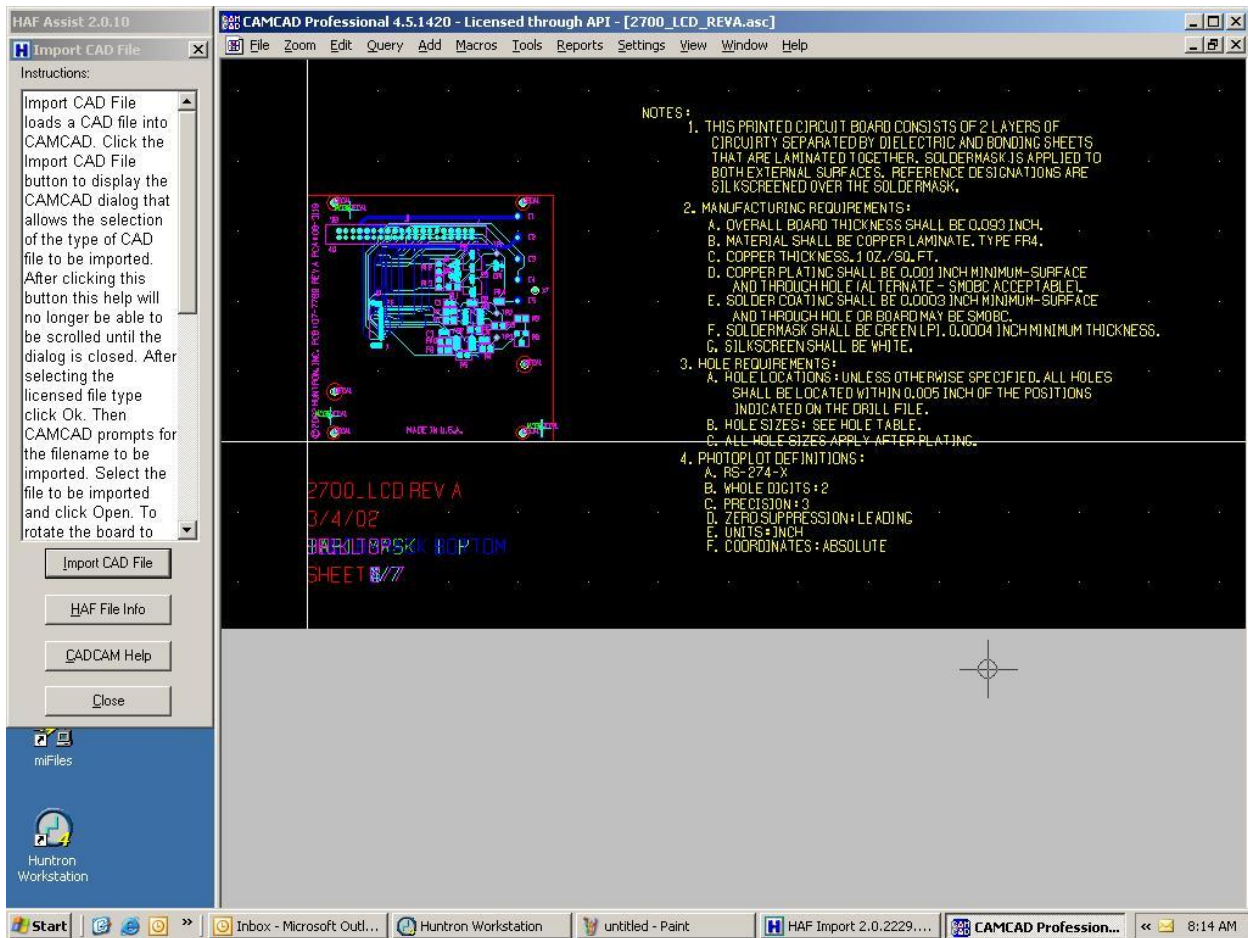
Click the **Close** button at the bottom of the Import CAD File window to return to the HAF Assist window.

Clean Display

Clean Display is used to remove unwanted data such as text that is not needed when the HAF file is created. Click **Clean Display** button and the Clean Display dialog box opens with its help file. Click the **Clean Display** button and the Generic Delete box opens.

Use the default settings and click **OK**. Click **Yes** when asked if you want to proceed with the Clean Display function. The text information will be removed and only the PCB information remains. Click the **Close** button to return to HAF Assist window.

Panelize Board (Optional)

CAMCAD allows the creation of panelized boards. This step is not used for single boards or boards that are already in panels. From HAF assist, click **Panelize Board** button. This displays the Panelize Board dialog. Click **Panelize Board** to display the Panelization window. Click on **Create Single Board Panel** and the Panel Display box appears.
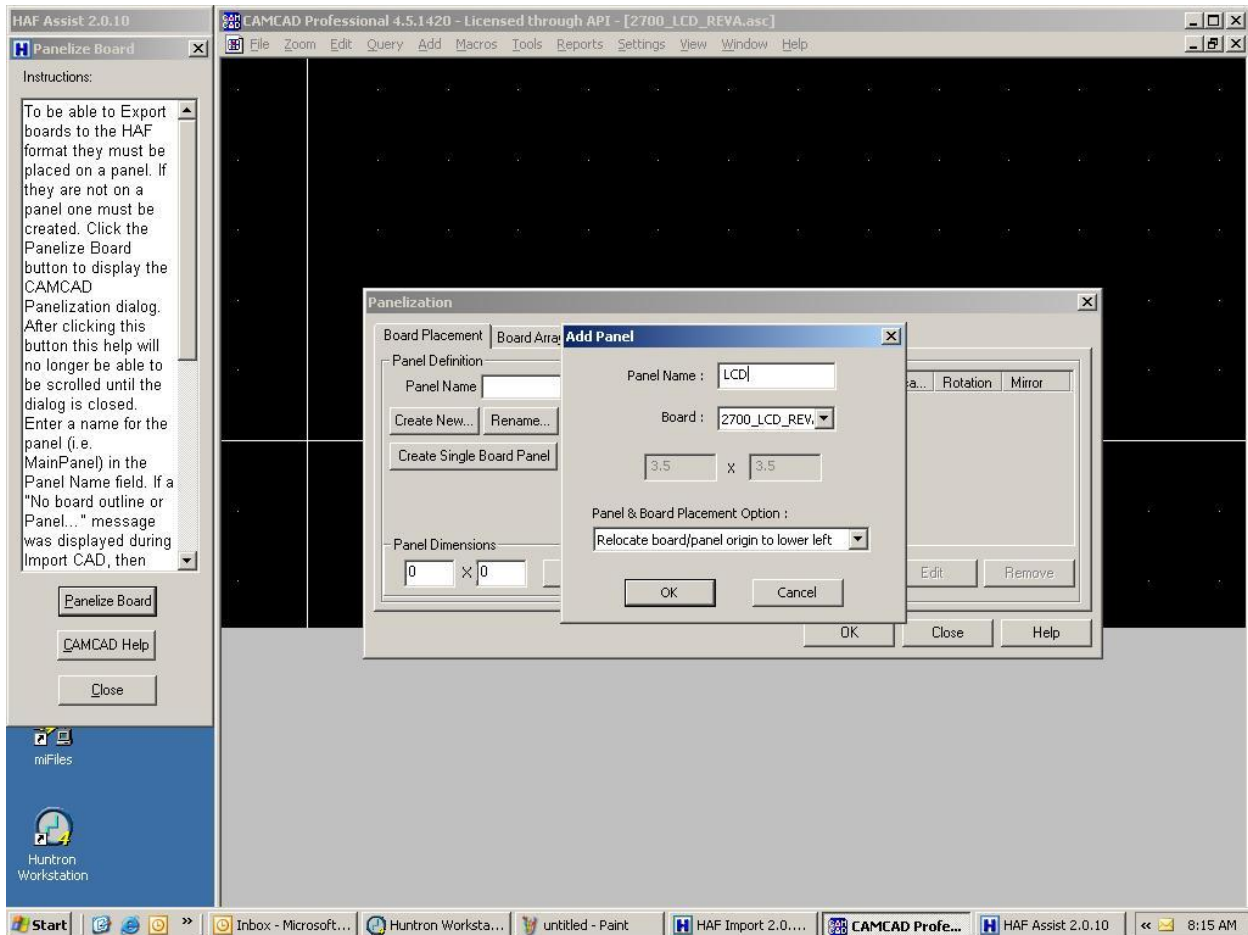

A panel name must be entered and in this example the name "LCD" is used. In the Board placement window, a board file is required. Use the drop-down arrow to select the appropriate file name. The board file is usually the same file name that was used in the import process. In this example the default file 2700_LCD_REVA.ASC is selected. Click **OK** and then click **OK** in the Panelization box. Click the **Close** button to return to the HAF Assist window.

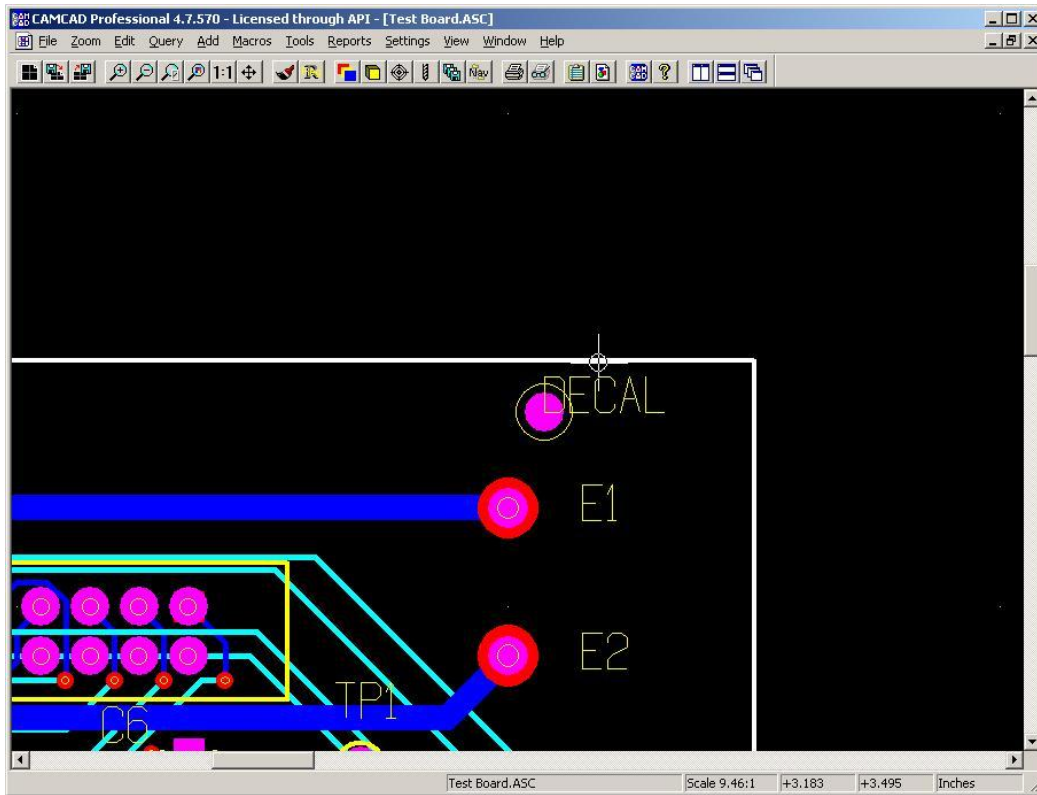## Selecting a Primary Board Outline

On occasion, some imported CAD files will not include a Primary Board Outline for the PCB. This will cause errors when trying to move to the Select Alignment Points step in the HAF Assist process. If you need to select a Primary Board Outline it is necessary to use some of

the functions of CAMCAD. The first step is to find an entity in the CAD display that can be used as the board outline. If one does not exist it can be draw in by hand.

Zoom in on a corner on the displayed board image in CAMCAD. Zoom tools can be selected from toolbar at the top of the display window.
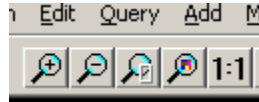

You can place the crosshair cursor on a object in the display and left-click. The selected entity will turn white (see image below).


You may need to search around in the image selecting different objects until you find an entity that will work as the board outline.

To set the Primary Board Outline, place your cursor directly over the selected outline entity and **right-click** to display a sub-menu (see below). Select **Edit Entity** to display the entity attributes.
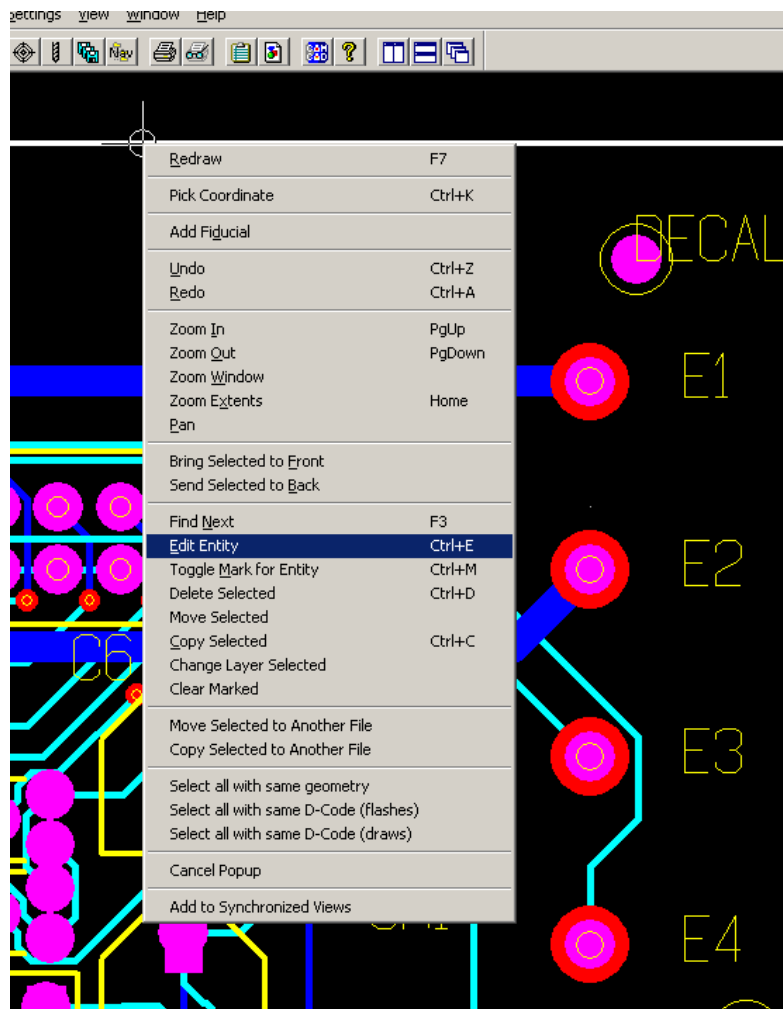
Select the **Class** drop menu and select **Primary Board Outline**. Click the **Apply Changes** button to set the new class and close the attribute window.
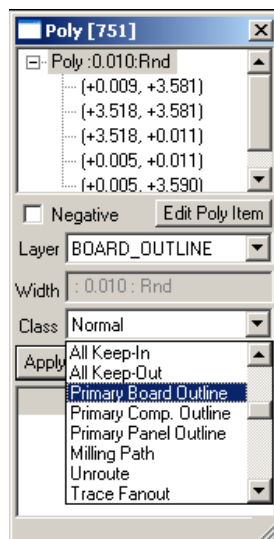
If you cannot find an entity that will work as the board outline or one does not exist, you can create one by drawing one within the CAD image. To manually create an outline entity, select the **Add/Line/Polyline** from the menu (see below).
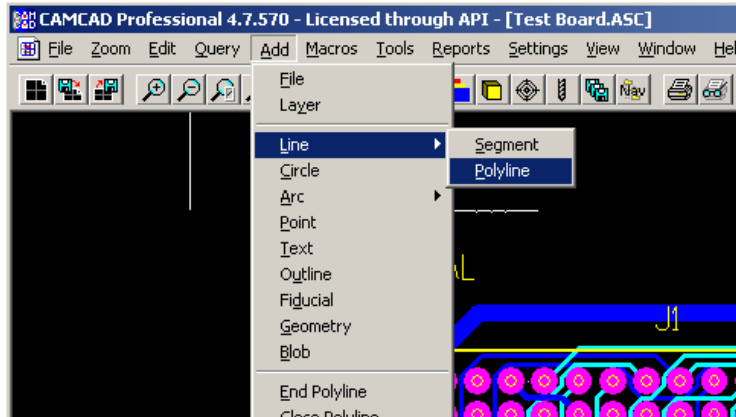
The cursor will change to "X" crosshairs. Click where you want the first corner of your Outline box to be (upper left corner in the example image below) then move the crosshairs to the next corner. A line will automatically follow your movement. Click to set the next corner. Repeat this process until you have drawn a box outline around the board panel. Press the **ESC** key to close the draw mode.
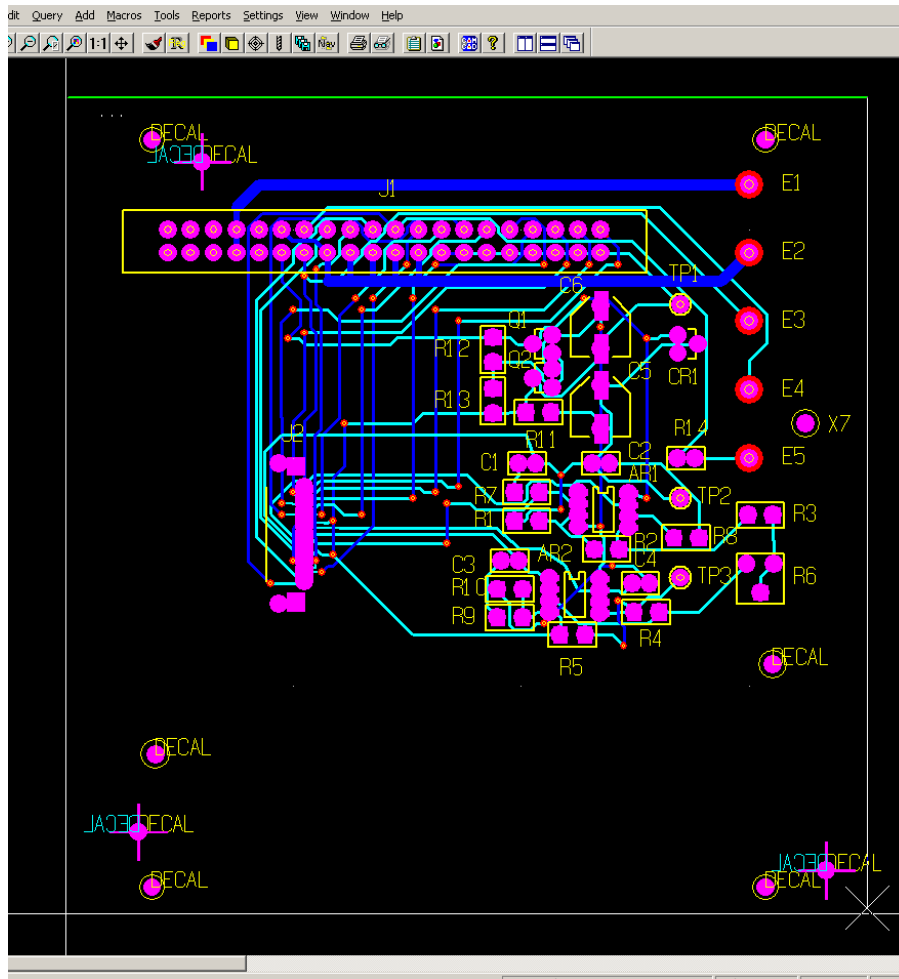
Once the Outline is drawn, select the outline and set it as the **Primary Board Outline** as described earlier.

Set Board Origin

If needed, you may need to also set the board origin. This essentially indicates where the PCB will be placed within a specific space. Generally the board origin is set to the lower left corner of the board for the Top side.
To set the board origin in CAMCAD, select **Tools/PCB/Auto Panel-Board Origin/Lower Left** from the menu (see below).
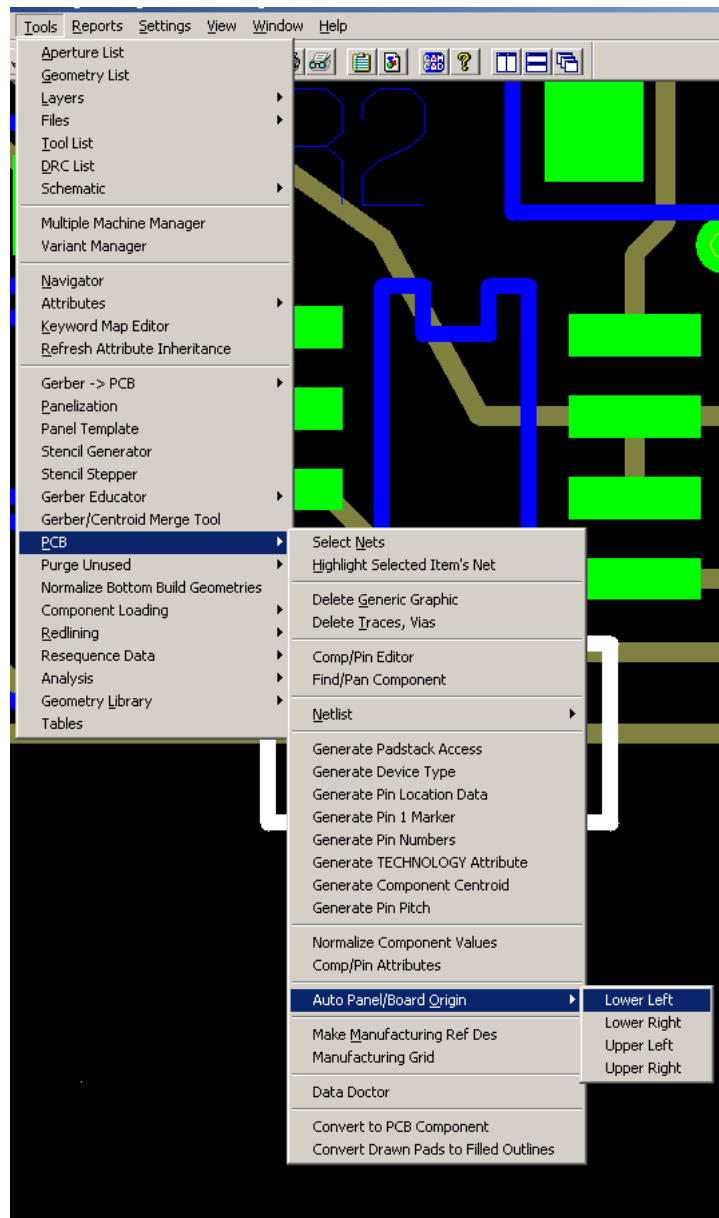

You should see the white board origin lines move to the lower left corner of the board outline in the CAMCAD display.

Select Alignment Points

Alignments points are used as reference points to determine the relationship of the components on the PCA and the Huntron Prober. A set of alignment points are required; two on the top of the board and two on the bottom of the board. Alignment points should be located as far apart as possible for the best overall accuracy. Alignment points are physical points on the PCA that will be accessed by the camera view. The selected points should be small and have their center point easily distinguishable. Fudicials, vias or component pins can be used as alignment points. The software checks to make sure the chosen align point is valid. You may want to physically mark the points selected on an actual PCB for future reference when using the robotic prober. Also, the points selected must be easily visible when using the robotic prober's camera.

Note the rotational board orientation for future reference. In this example, the J1 connector oriented towards the top (back).

In the HAF Assist click the **Select Align Points** button. The Select Alignment Point window appears with the help file and four boxes where the alignment values will be stored. A **Flip Board** button allows views of either the top or bottom of the board and use the **In** or **Out** buttons for zooming. The **Window** button allows you to zoom a specific area of the CAD image by drawing a box around the desired location. The **Board** button will set the Zoom level so the entire PCB is in view.
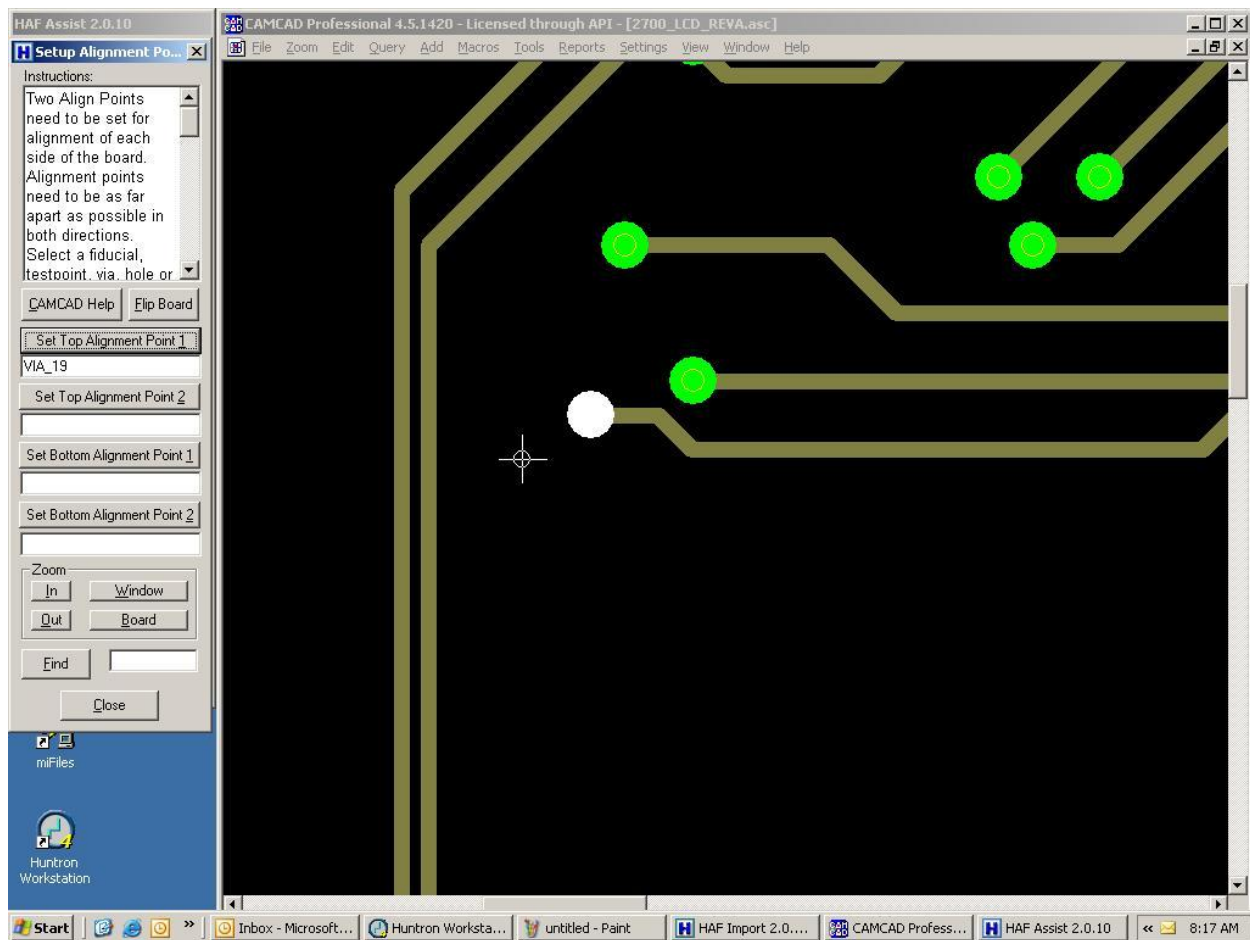

If you have a preferred alignment point on the board, you can use the **Find** field and associated button to assist in located specific points. In this example, two vias on the board have been chosen for alignment points. The first alignment point is a via located in the

upper left corner. Click the HAF Assist Window button. Drag the cursor to create a box around the via. This will zoom to make selection easier. Click on the via in the CAMCAD image where it will change to a white color. Click **Set Top Alignment Point 1** and the point name will be displayed as Top Alignment Point 1. If you do not see the name displayed then select a different point.
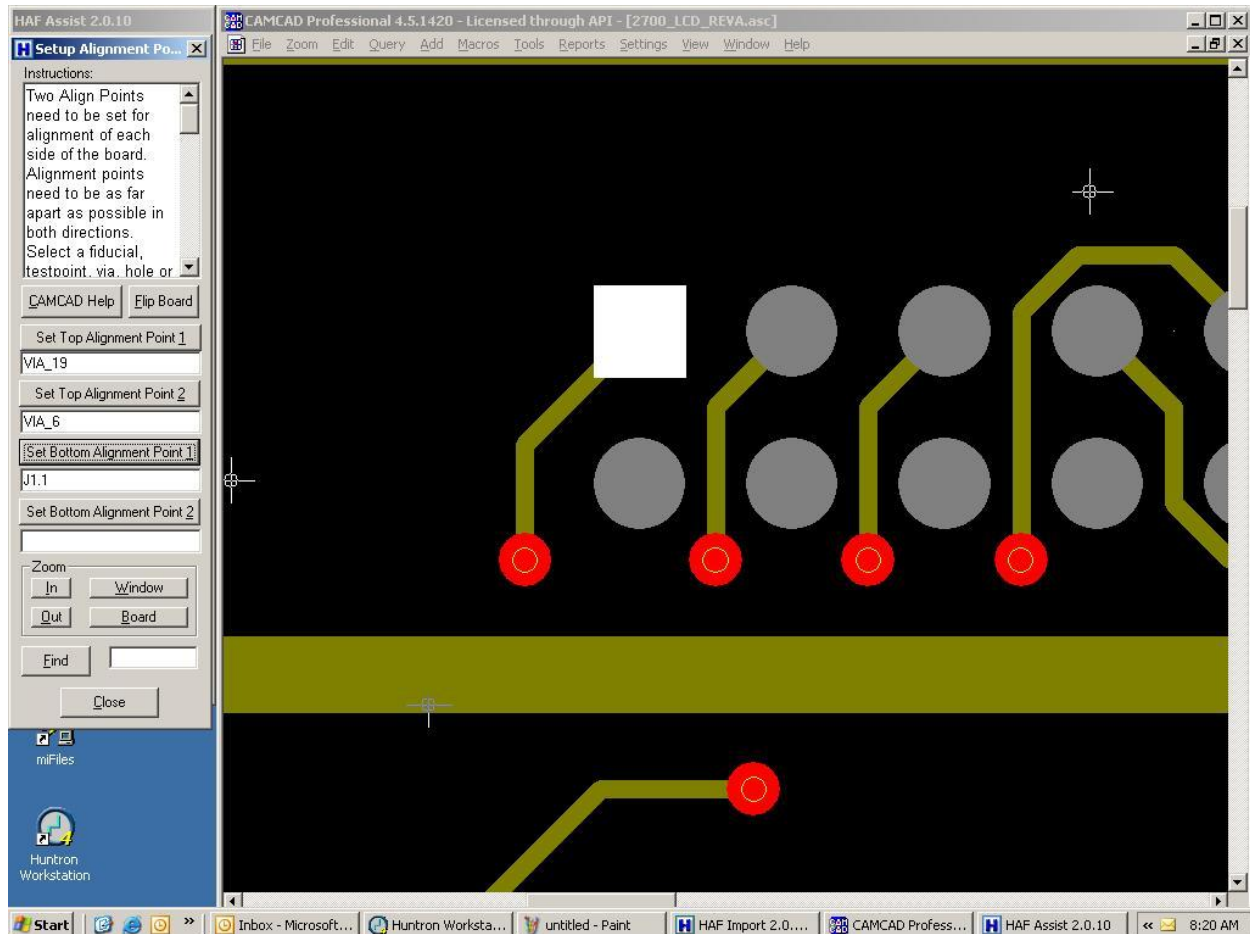
The second alignment point is a via located in lower right corner of the board. Click the HAF Assist Board button to see the whole board again. Click the Window to zoom in on the via. Click on the via so it turns white and click the **Set Top Alignment Point 2** button.
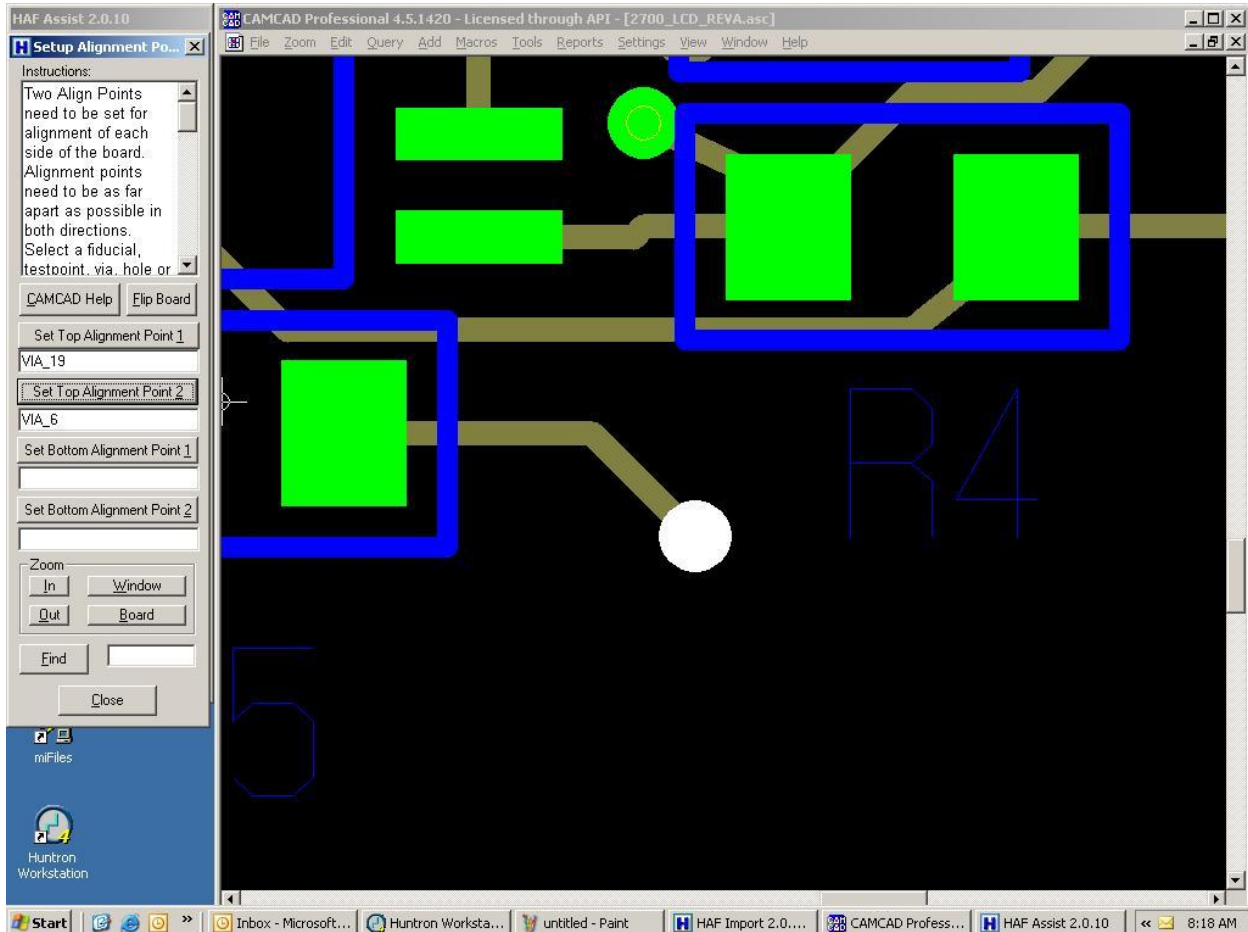
The process is repeated for the bottom side of the board. Click the **Flip Board** button to display the bottom side image. The first bottom side alignment point is pin 1 of a connector located in the upper left corner. Zoom as needed. Clicking on pin 1 in the CAMCAD image will highlight the entire component.
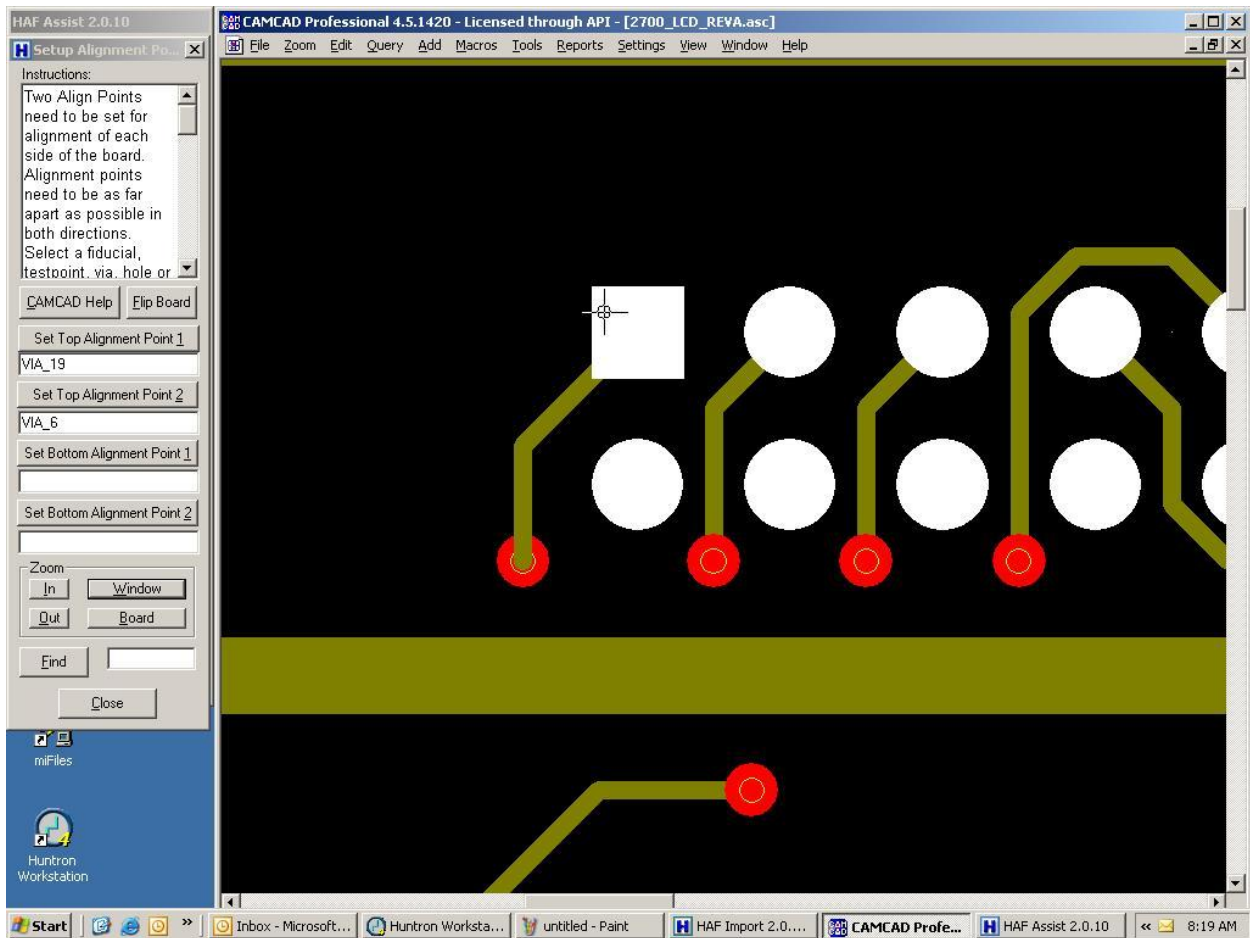
Press **F4** to select or "drill down" to the individual pin where it will change to a white color and the other component pins will change to grey. Click **Set Bottom Alignment Point 1** and the point name will be displayed as Bottom Alignment Point 1.

Select a second bottom side alignment point that is located in a diagonal direction from the first bottom alignment point. In this case, it would be located in lower right corner of the board. Navigate to the lower right corner of the board image and select a suitable feature as the second alignment point. Use zoom as needed. Use an actual board to help select an accessible point. Click the **Set Bottom Alignment Point 2** button once the point the point is highlighted in CAMCAD. Click the **Close** button to return to the HAF Assist window. **Important note:** Do not use a component pad as an Alignment Point if you plan on using the CAMCAD Access Analysis (described later in this tutorial) to modify the component pin centers.
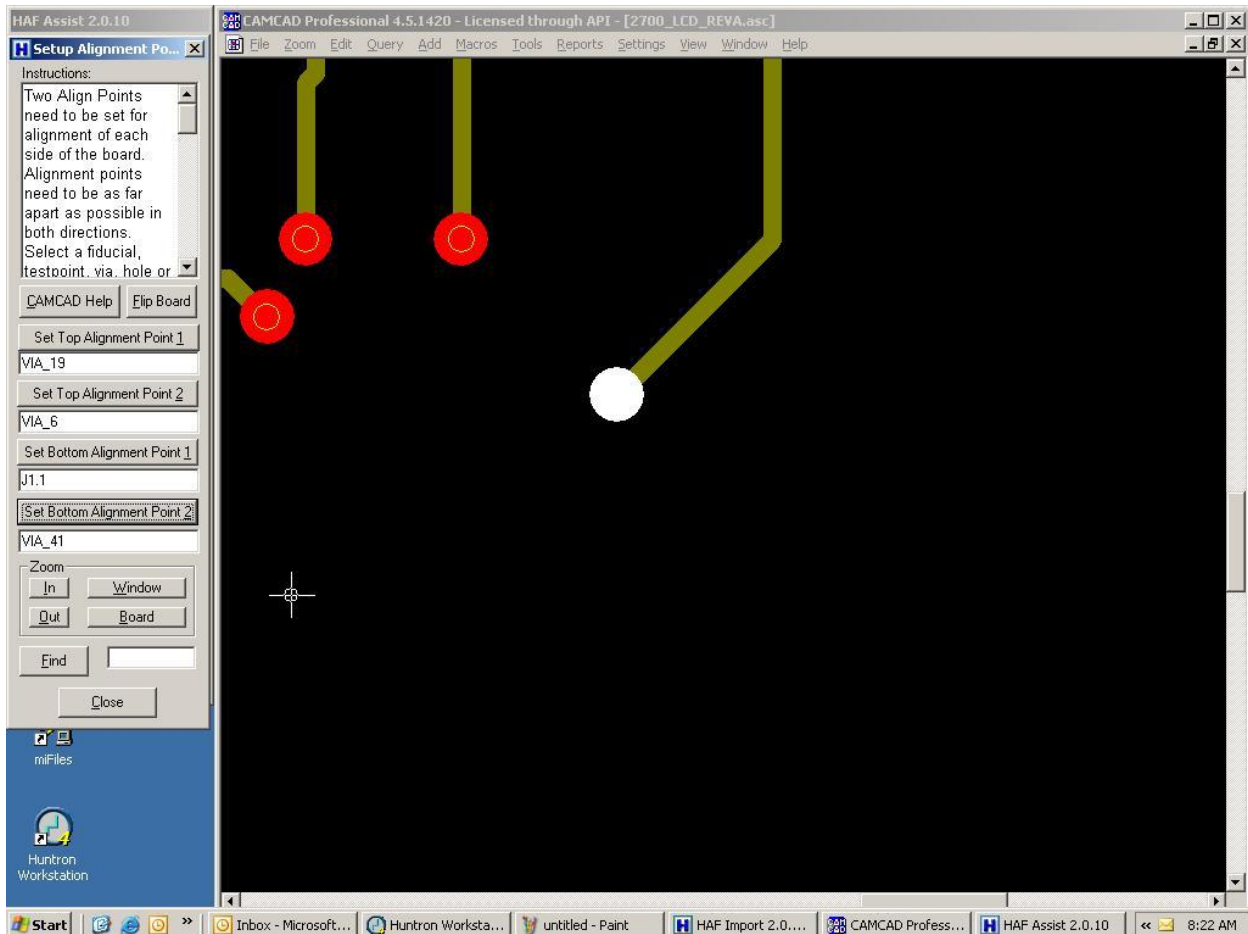
Set Layers (Optional)

CAD files are configured in layers such as top, bottom, silkscreen component, etc. Each layer is represented by a color. To better view of a specific section of the board it may be best to turn off one or more layers.

From the HAF Assist window click **Set Layers**. This will display the Set Layers dialog box that contains a Help file. Below the HELP is the **Set Layers** button which when clicked on will display the Color Sets Dialog box.
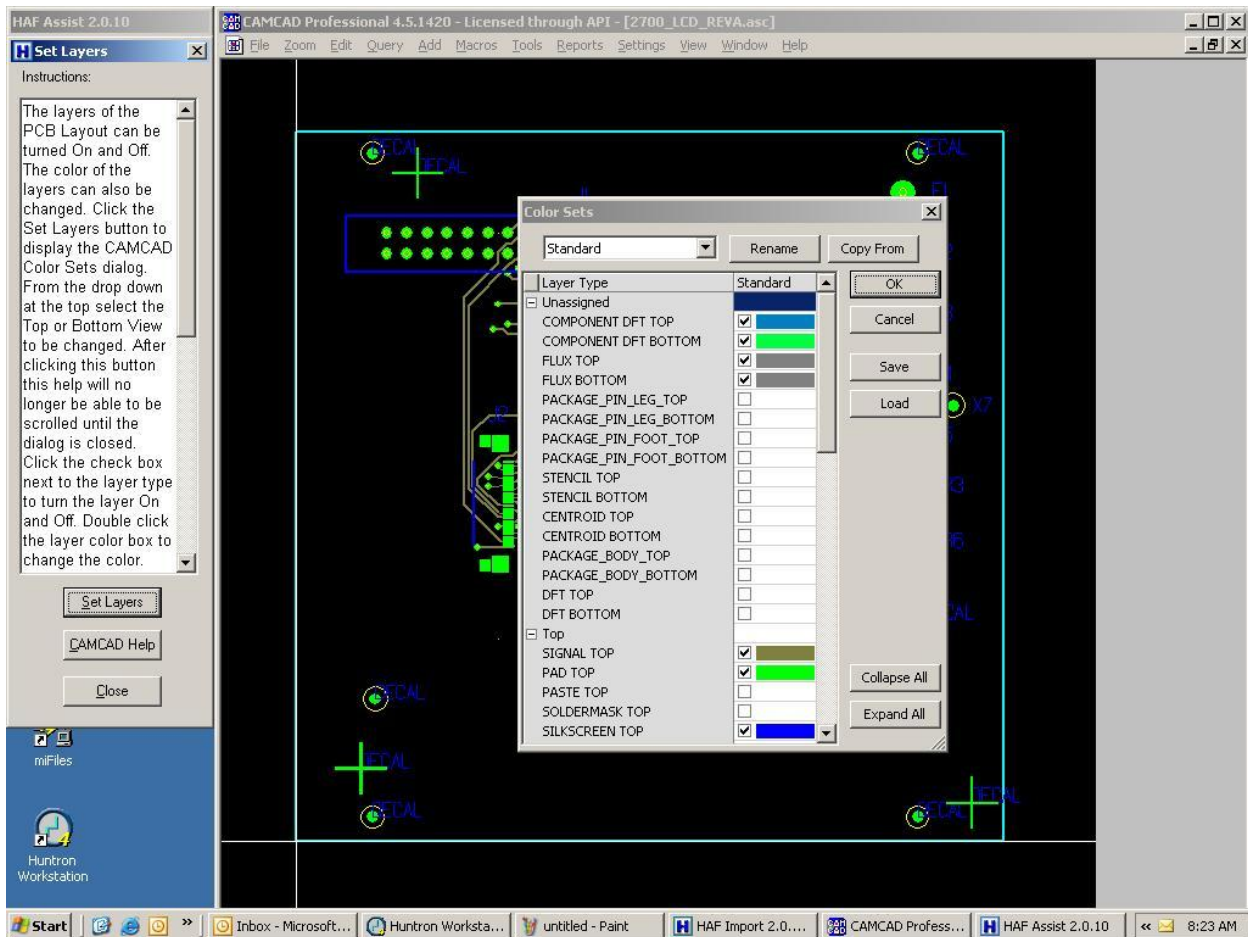
From the Layer Type list you can select which layer(s) that you want to display and assign it a color. Click on the check box which will turn white and click on the **OK** button. The selected layer will be removed from the CAMCAD image. To save a configuration, click the **Save** button. Click the Close button to return to HAF Assist.

Launch Device Typer (Optional)

Device Typer is used to classify the names of components which appear in the Type box that resides in the Component dialog box of the Huntron software. Default names within the PCB file have been predefined. For example a C represents a capacitor, R represents a resistor, Q represents a transistor, etc. The complete definition list can be found in the HAF.OUT file. On the example board is diode CR 1 a 3 pin diode array. Most diodes are 2 pin but CR1 consists of 3 pins and is considered an array. The 'Device Typer' is looking for a 2 pin component. From HAF Assist click **Launch Device Typer** and the Device Typer box with help appears on the left and Generate Device Type dialog box appears on the right.
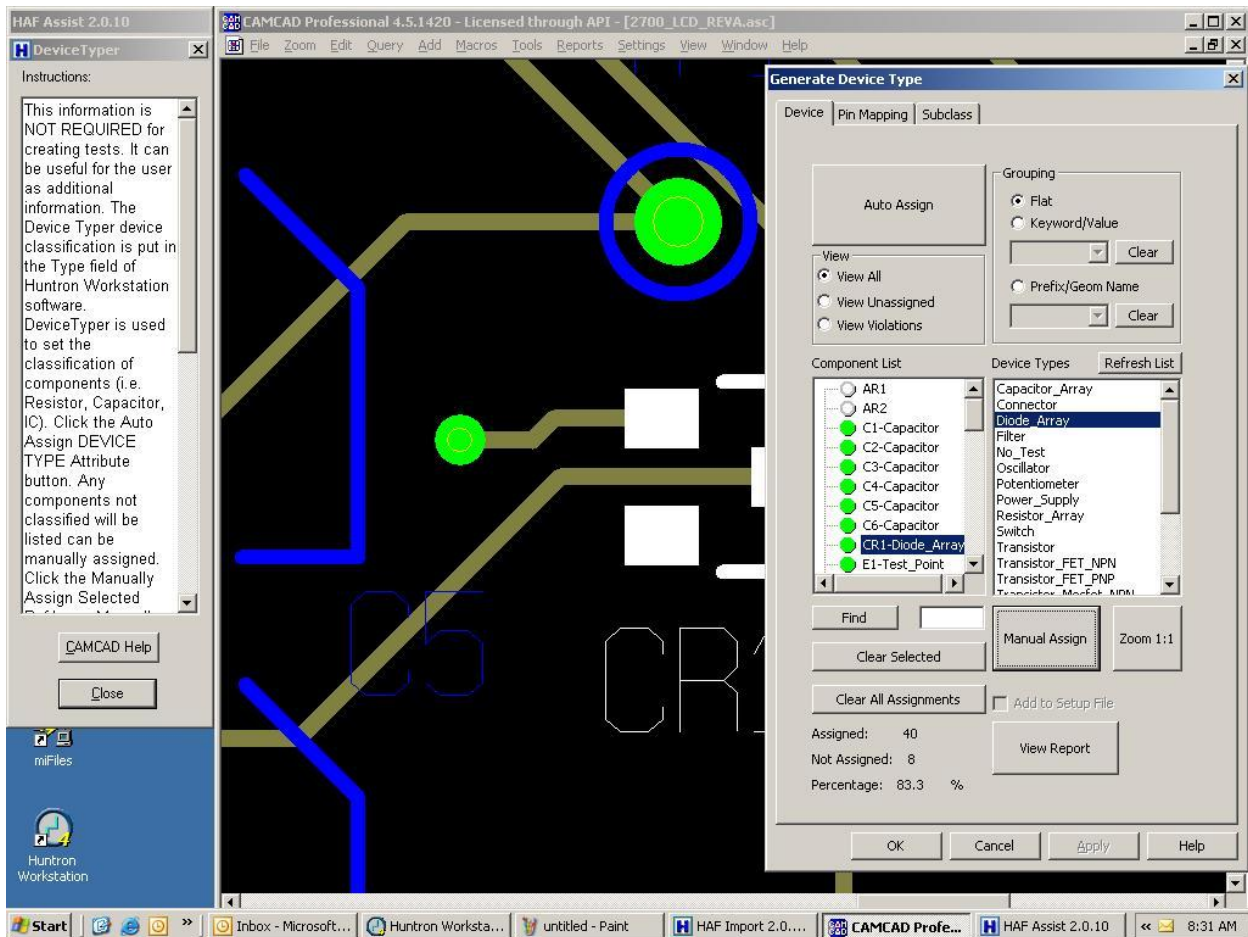
Click **Auto Assign** and the components will turn red, yellow or green showing an attempt to classify them. If a device has a "type" violation the Device turns red. Clicking on CR1 in the component list shows CR1's PCB location. Typically diodes are two pin devices but CR1 is a three pin device, this is why CR1 has a red indicator.

The violation can be removed by selecting 'Diode Array' from the Device Type List, then click on **Manual Assign**. CR1 now has a Green assignment. Click **OK** to close the Generate Device Type window and **Close** to return to HAF Assist.

Component Values (Optional)

The option allows component values (i.e. 100K ohms) to be specified from a attribute in the CAD data. This tutorial does not go into the steps for making these changes.

Pin Offsetting (Optional)

By default the center of each component pad is used as the reference point for the Huntron Prober. There maybe component package types that do not allow the Huntron Prober to connect with the center point of the pad, causing an open condition. The Pin Offsetting feature allows the selected package type to have its Top or Bottom reference point of each pad moved.

To start, the component Outlines must be defined so the pin centers can be repositioned based on the device body. Note that the image below does not indicate the pin center typically shown with a crosshair mark.
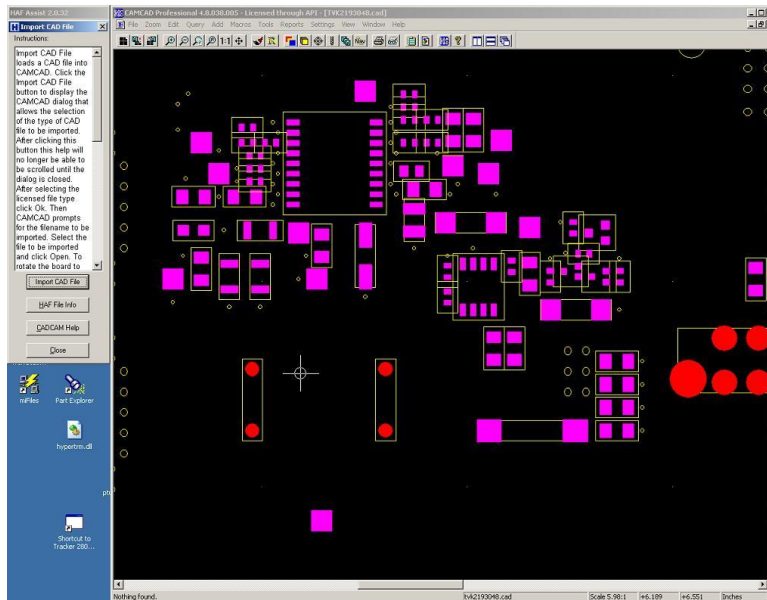

To define the component outline, the CAMCAD Data Doctor must be used. This allows you to set component outlines among many other attributes. The image below shows how to select the Data Doctor by selecting **Tools/PCB/Data Doctor** from the main menu.
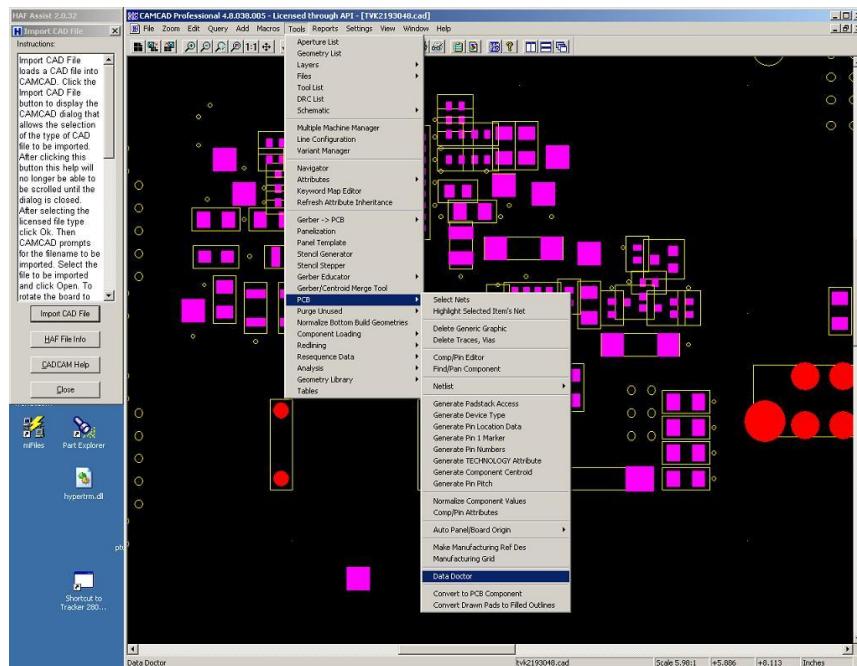

The Data Doctor is shown below. Select the **Packages** tab at the top of the Data Doctor window. Set your **Group Outline** parameters as shown in this image with **Pin Centers** (drop menu) and "**Apply to All Geometries**" selected. Click the **Create Outlines** button to generate (or modify) the component outlines.
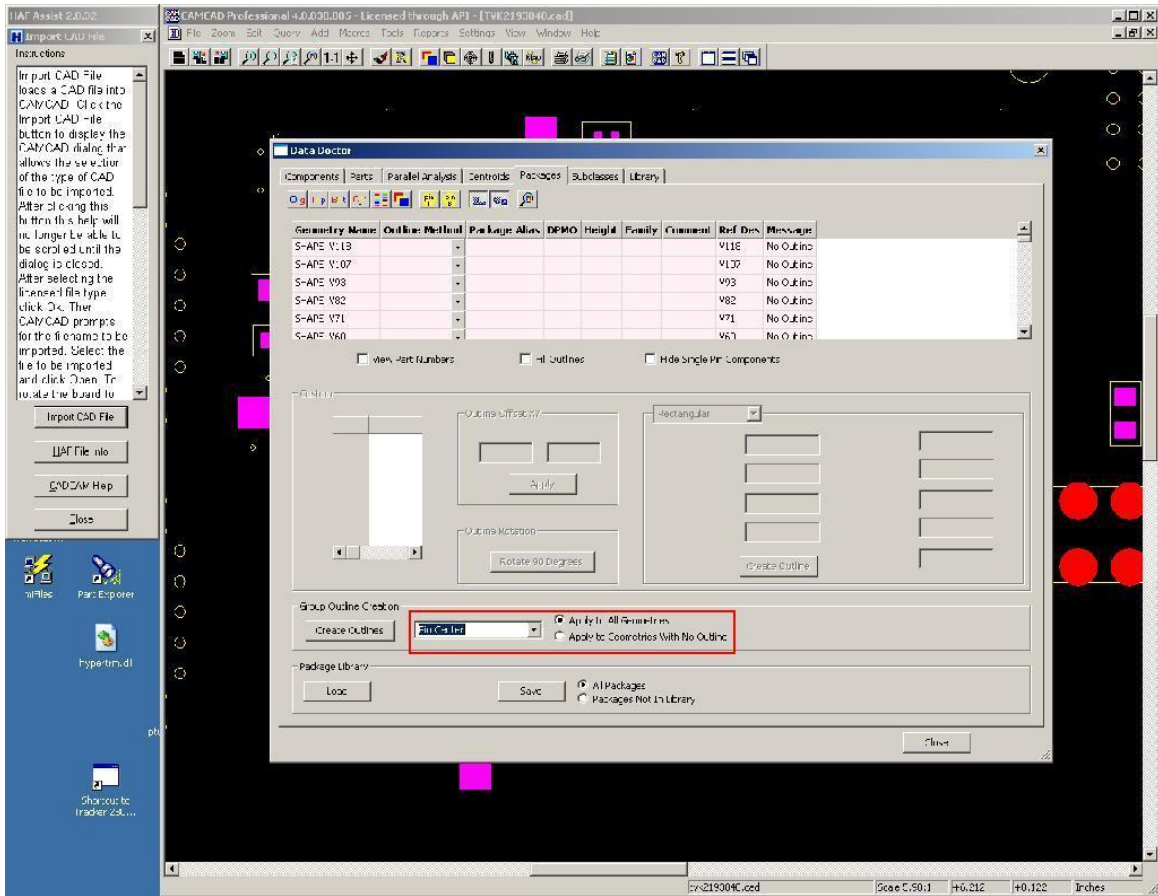
Click the **Close** button to close the Data Doctor. Note how the CAMCAD PCB view now shows component outlines based on the pin centers (blue outline in the image below).
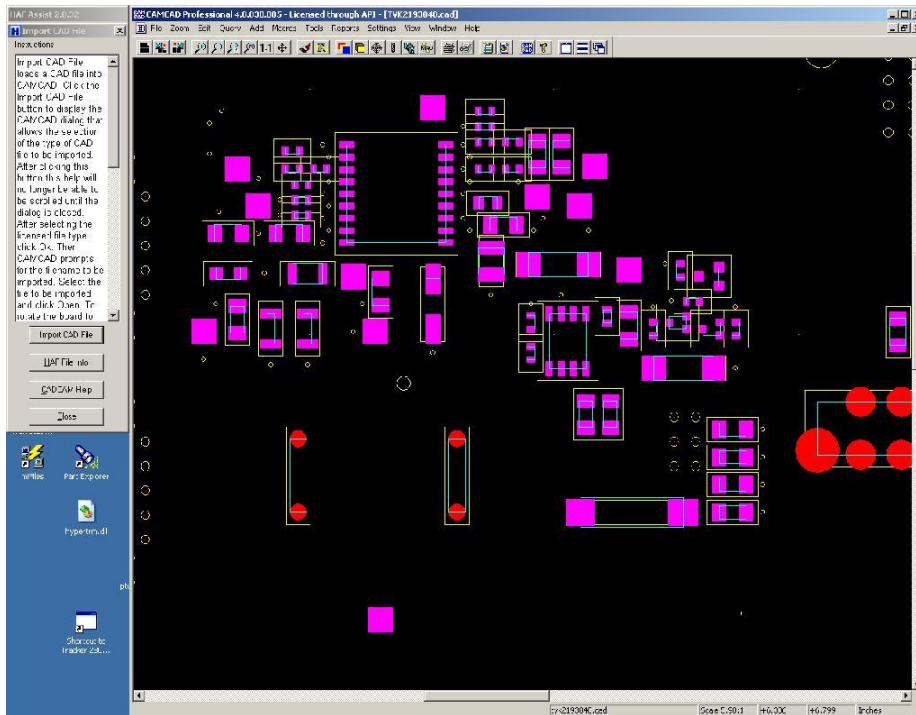
The next step is to use the Access Analysis feature to modify the component pin positions. To open Access Analysis select Tools/Analysis/DFT Analysis/Access Analysis from the main menu.
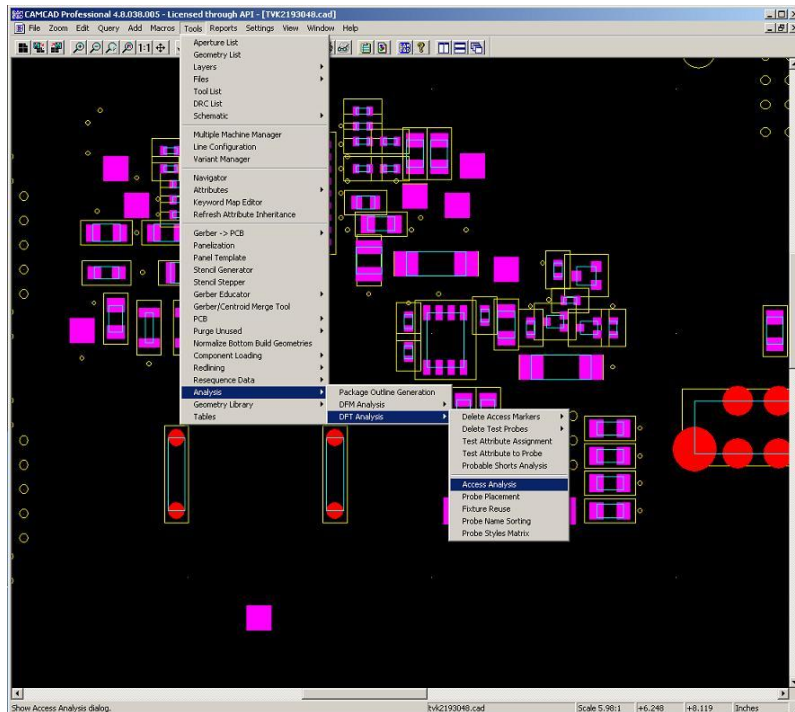

The Access Analysis window allows you to determine the level of test access for the selected PCB. In this case, it will also be used to modify the pin centers. Select the **Options** tab on the top of the Access Analysis window and set the parameters as shown in the image below.
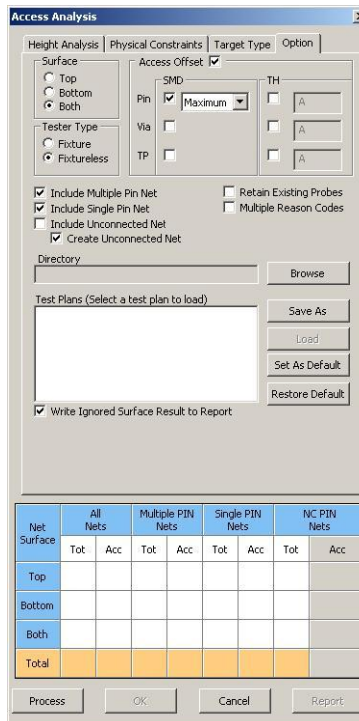
Select the **Target Type** tab and set the parameters as show in the image below.
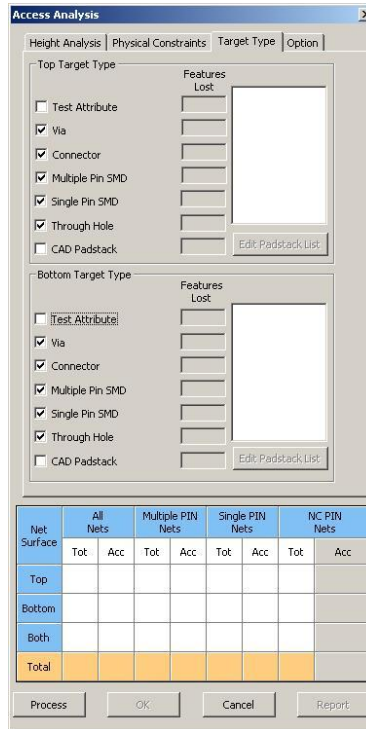

Select the **Physical Constraints** tab and set the parameters as show in the image below. You may need to modify the Outline Distance properties based on the component sizes on your PCB. Click the **Process** button to calculate Access Analysis.
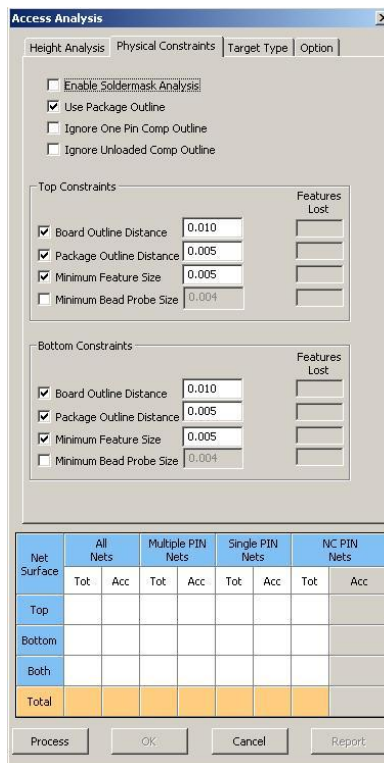
Once the Access Analysis is calculated, you will see the results displayed similar to the image below.
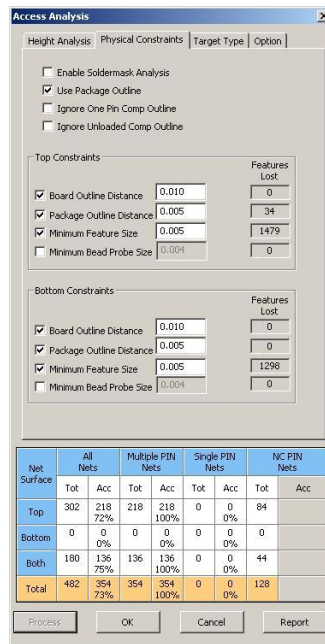
Click **OK** to close the Access Analysis window. Note in the image below that the pin center crosshairs are now displayed in the CAMCAD display and that they are also offset from the pad center.
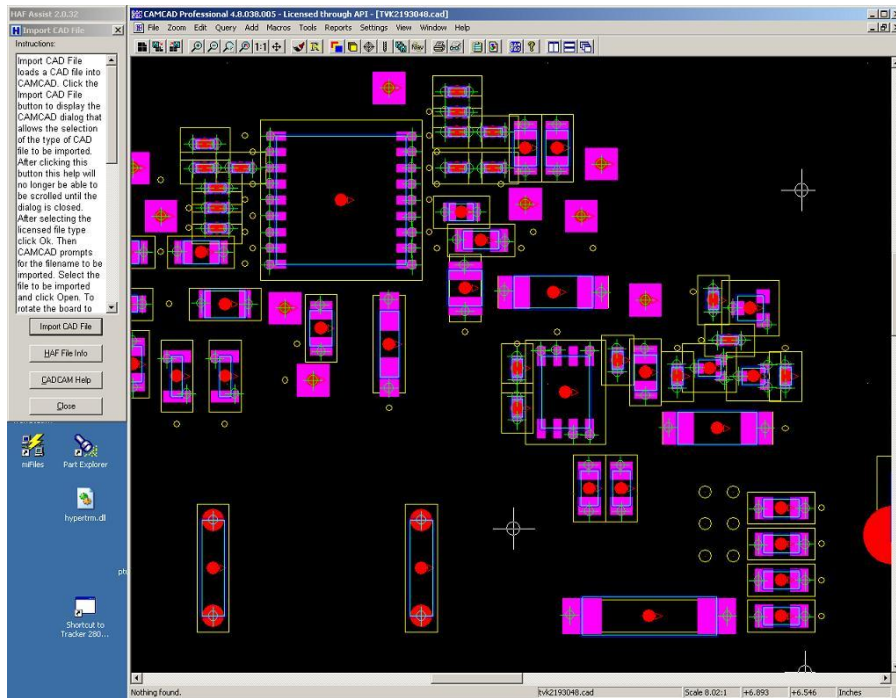

You can now continue with the CAD Import process normally.

Setup Test Points (Optional)

In most cases the Huntron Prober uses physical pins of a component as test points. There may be times when the prober can not make physical access with a pin and a new test point must be created
From the HAF Assist window click **Setup Test Points**. This will display the Setup Test Point window. Below are Help and two buttons, one to set a test attribute or to remove a test attribute.
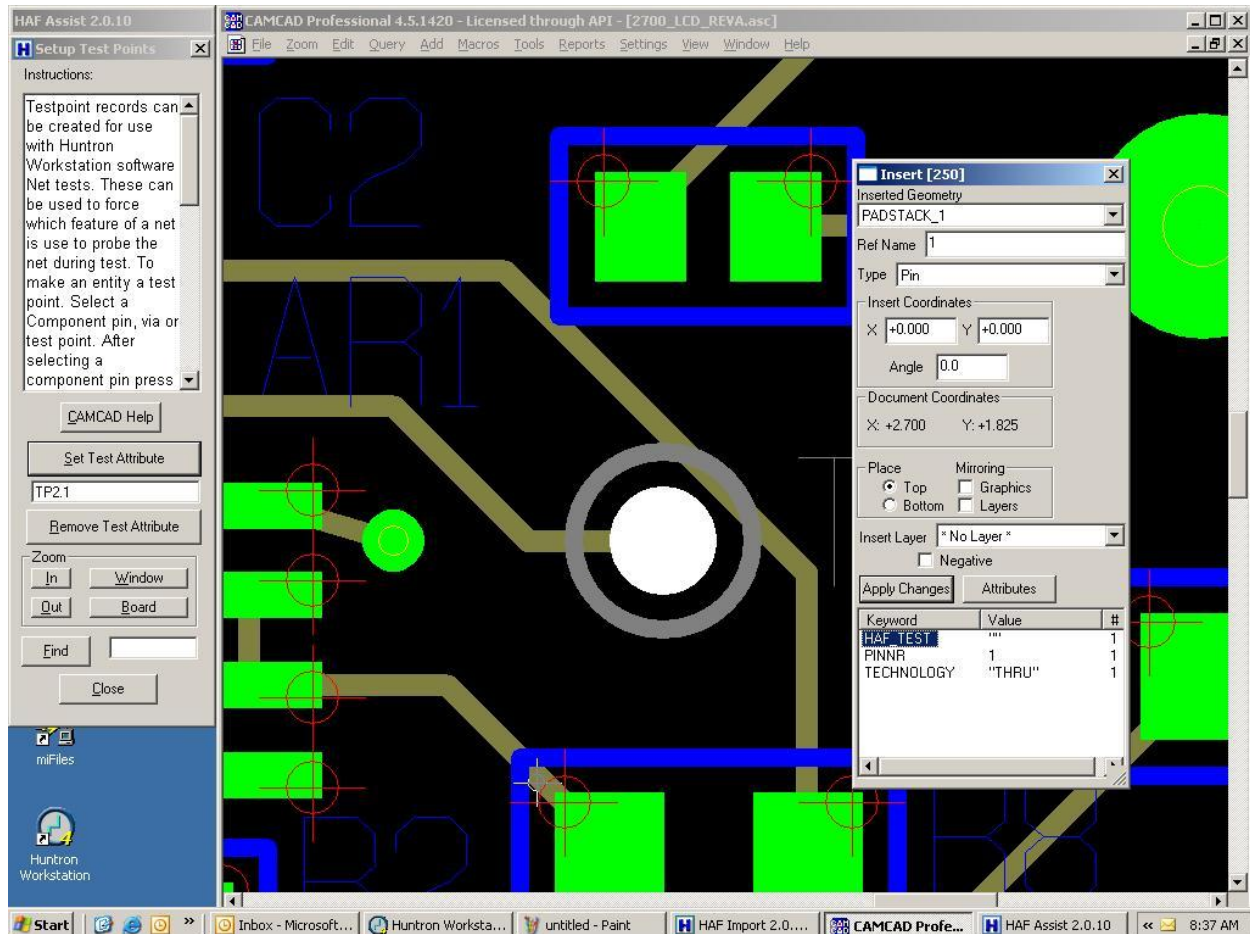
Locate the entity such as a via, component pin that you want to be a test point. The **In** and **Out** buttons will zoom in and out to help in selecting a test point. **Find** can also be used assist in locating a specific point by name. Select a test point and will it highlight. Click the **Set Test Attribute** button and an alphanumeric value will appear (object name). To verify if an entity is a test point, right click the object and select **Edit Entity** from the menu. The Insert box appears – in the Keyword box you should see the word **TEST**. This indicates the entity is a test point.
As many test points can be created as required by repeating this process.
Test points can be removed by highlighting the entity and clicking **Remove Test Attribute**. Return to the HAF Assist window by clicking **Close**.

HAF Export

Creating the Huntron ASCII File (HAF) can be done after the Import CAD, Clean Display, and Select Alignment Points operations are performed. All values from these three mandatory operations plus the values of the optional operations will be exported to the HAF file.
From the HAF Assist window click **HAF Export**. The HAF Export dialog box will open.
Click **Export HAF File** and a standard Windows Save window will open allowing the name of the file to be entered. Browse to the directory where you wish to save the HAF file. Click **Save** to save the file.
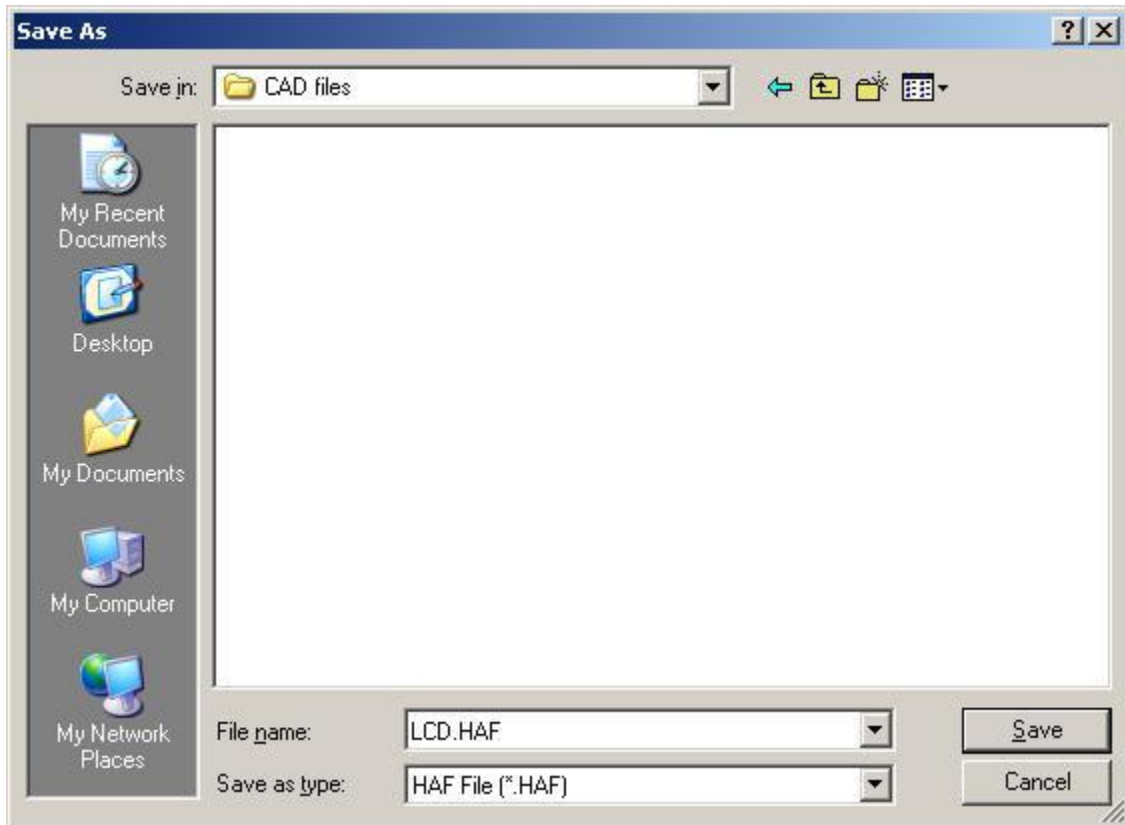
Click the **Close** button to return to HAF Assist.

Save CC File

The CAMCAD (CC) file is created after the creation of the HAF. From the HAF Assist window click **Save CC File**. The Save CC File dialog box will open. Click **Save CC File** and a standard Windows Save window will open allowing the name of the file to be entered. Browse to the directory where you wish to save the CC file (**note:** saving the CC file to the same location as the HAF file is best). Click **Save** to save the file.
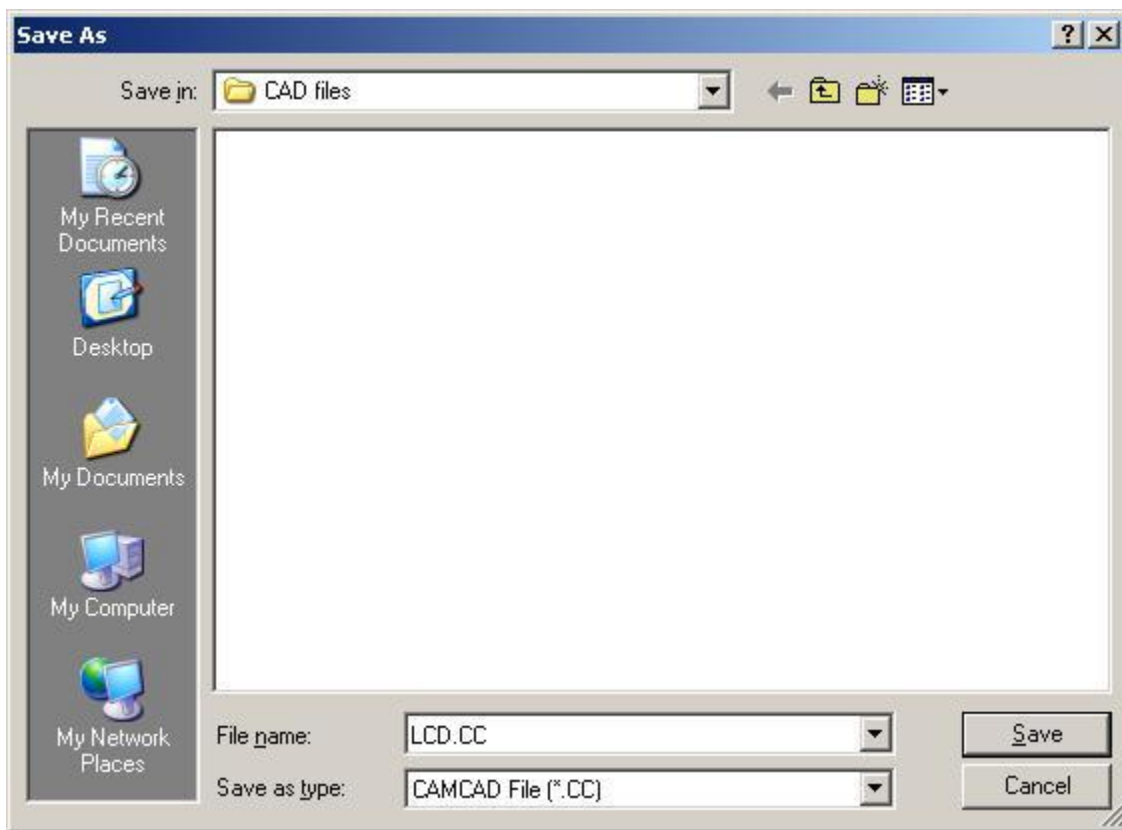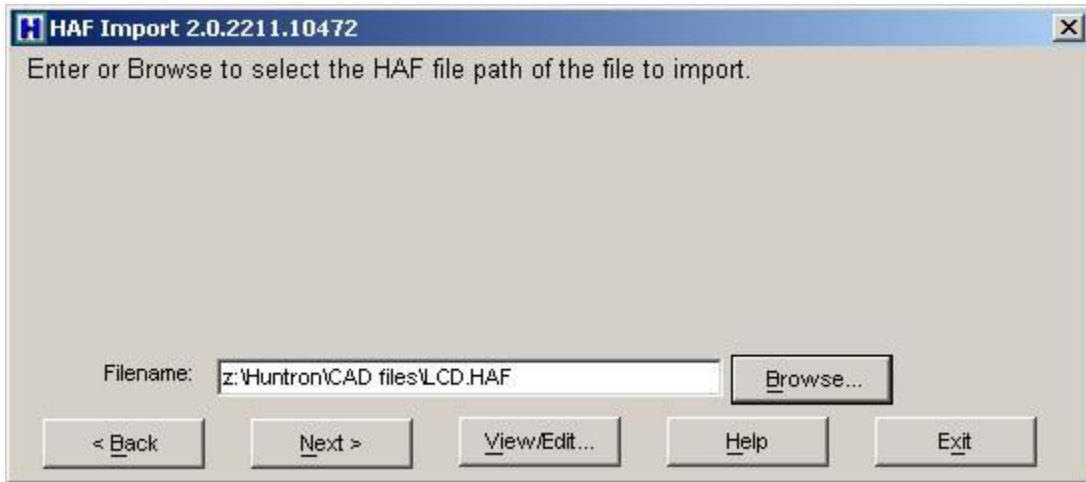
Click **Close** to return to HAF Assist.

Exit

In the HAF Assist window click on **Exit** returns the software to the HAF Import wizard.

Find HAF File

Click on the **Next** button; a message appears asking for the path of the HAF file. The path can be entered or located by browsing. Browse to the location where the HAF is stored.

Find CC File

Click on the **Next** button a message appears asking for the location the CAMCAD (CC) file. The path can be entered or located by browsing to the location where the CC is stored. Click **Next** to move to the next step.
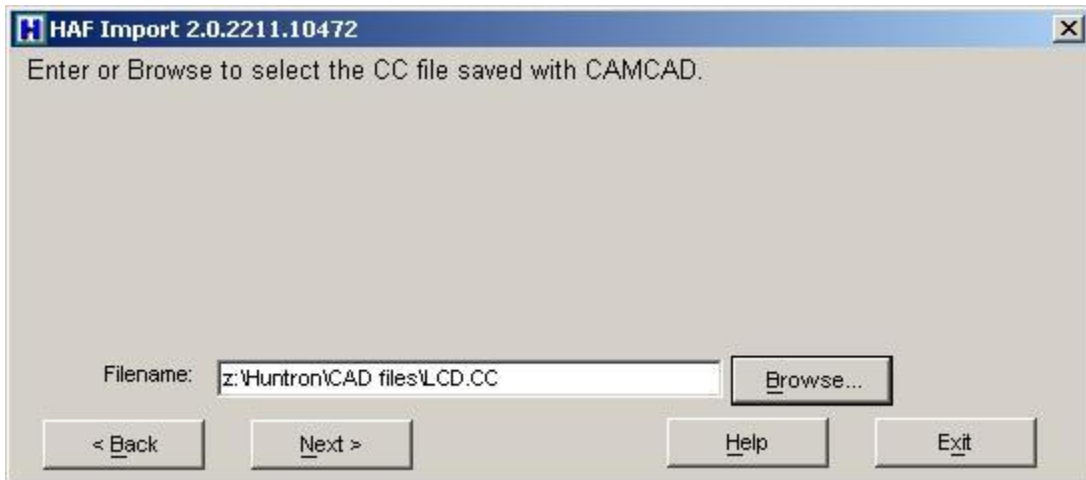
PreScan

A prescan operation will be performed to verify certain features exist in the HAF file. When complete the four alignment points will be displayed along with the message "PreScan Complete".
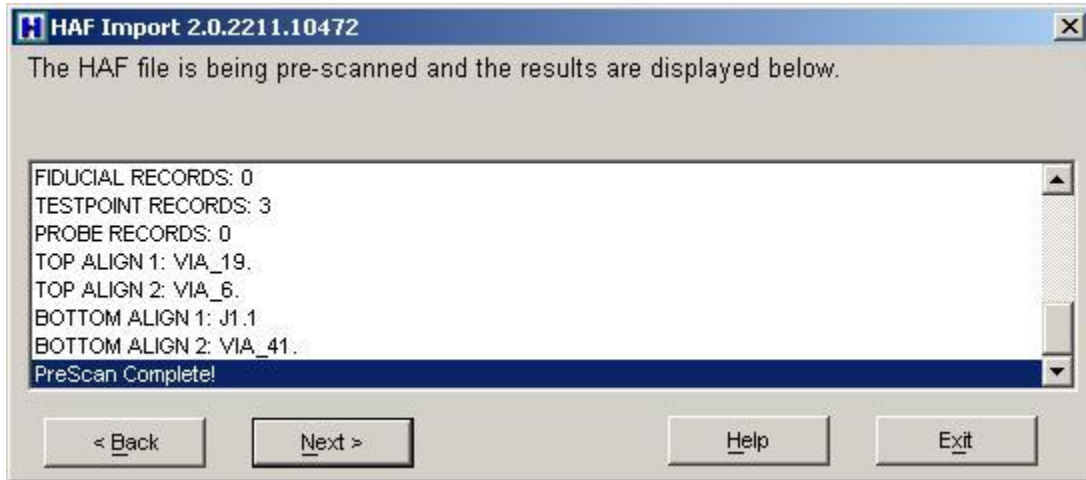
Depending on the file size this operation could take several minutes. Click **Next** when complete.

Test Type

Two types of tests can be created; a Component test or Net test. A component test will access every component pin found in the CAD data. The component test will be slower than a net test but provides better diagnostic information. The net test will be faster because it will test only the unique net points one time. The diagnostic information will isolate problems to the net level. If required both a net test and component test can be created. The net test would be used until a problem is found. The component test could be used to isolate the problem.
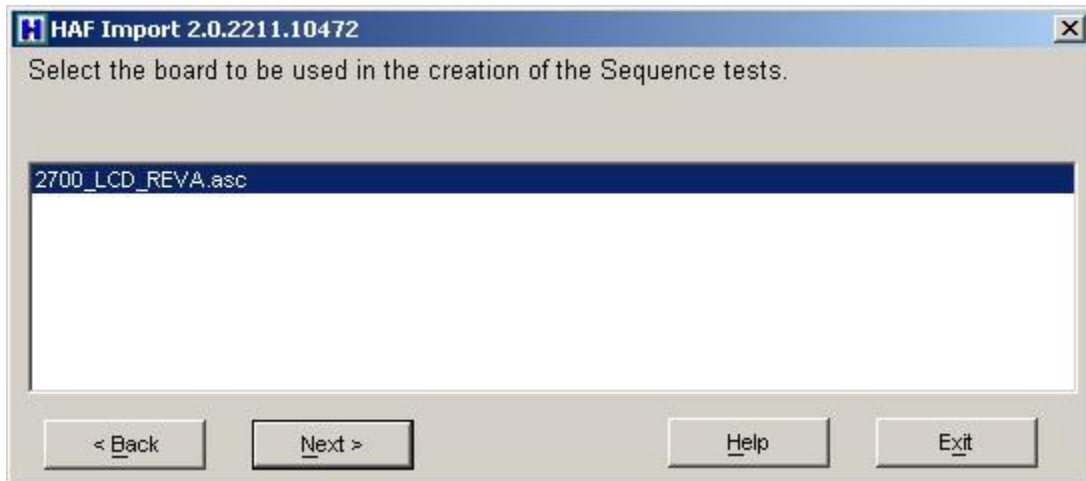
Test Type - Component

Select **Component** and click **Next** to continue. If you want to create a Net based test (Net Test Type) skip ahead to "Test Type – Net".

Select Board

The HAF file can consist of more than one board on a panel. This name is also the name of the board being tested.

This name will appear in the Sequence section in Huntron Workstation after the Process step. If the panel consist of a single board then click **Next**.

Top Bottom Slot Selection

A Huntron Prober has three (or four for Prober III, IIC and Access 2) slot locations where a board under test will reside. Choose a selection for the Top and one for the Bottom.
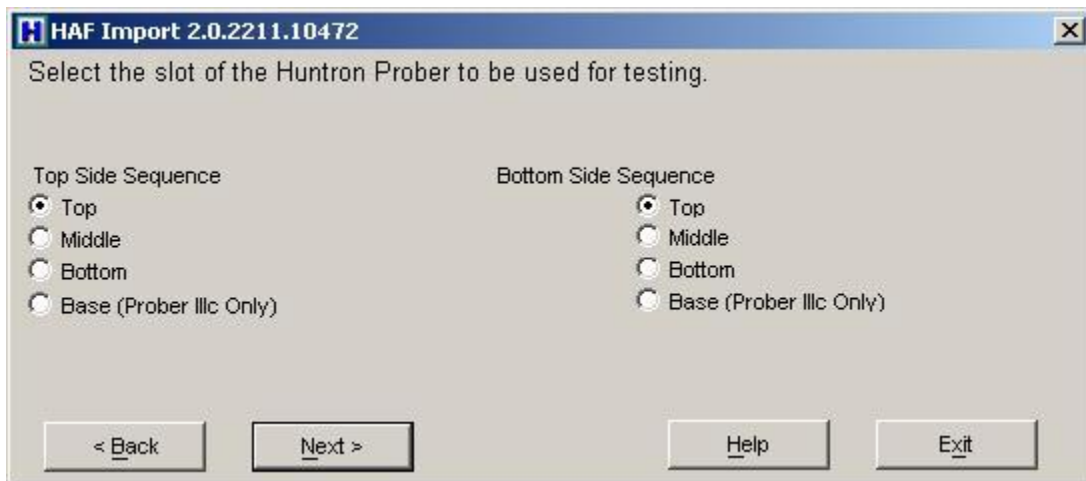
This information is used to set the slot value for the Sequence in the Workstation software. Click **Next** to continue.

Board Spacers

Board spacers are provided with the Huntron Prober. The spacers allow the board under test to be moved away from the walls allowing components near the edge of the board to be accessed.
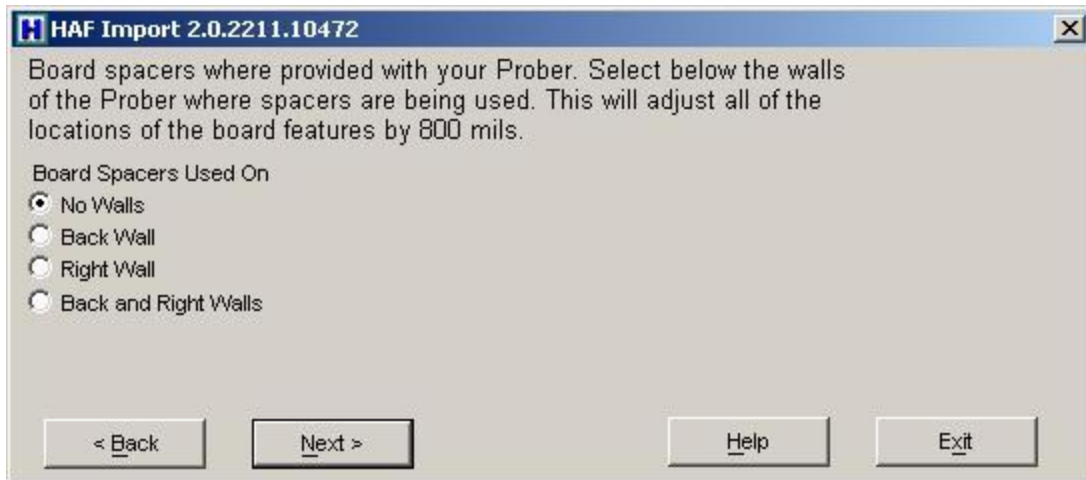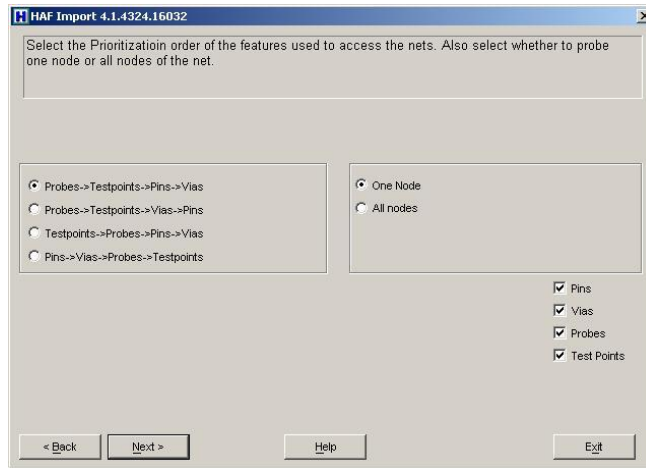
Selecting Board Spacers automatically adjusts the board location by 800 mils. Click **Next** to continue.

Test Type – Net

To create Sequences based on PCB Nets select **Net** as the Test Type. Click **Next** to continue.
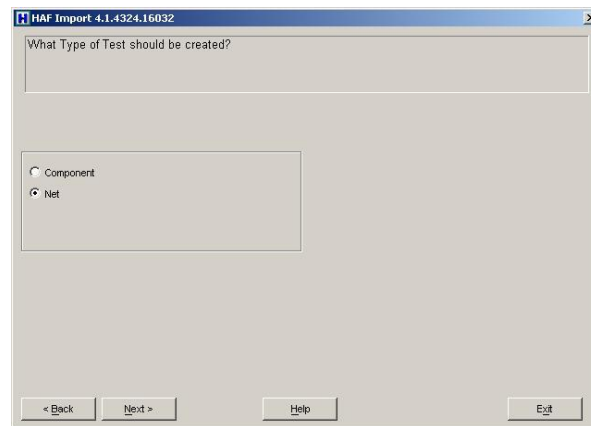
Under Net **Prioritization** there are four priority groups that consist of Probe, Test Points, Pins and Vias.

These groups determine the priority of which point will be used to access a point on the net. The point on the left has highest priority. For most board types select Via or Pins as having the highest priority. Click **Next** to continue.

You will be prompted to select a board side (top or bottom) to be given priority for net test point selection.
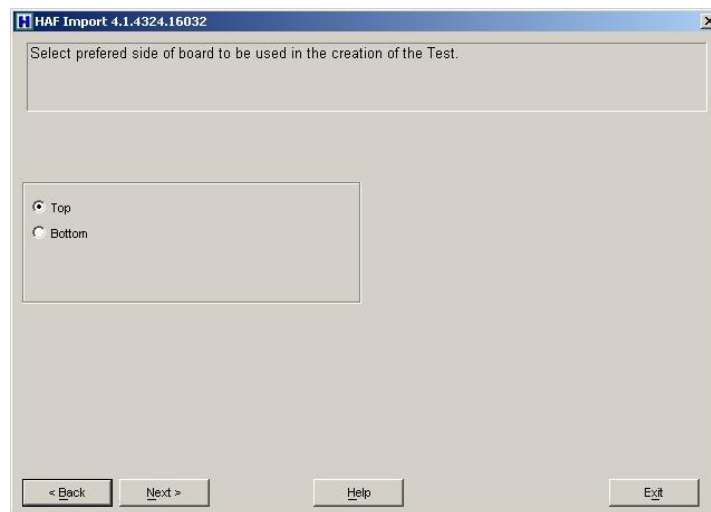

Select the board side you feel offers the best access to test points without having to avoid major obstacles such as connectors, tall components and heat sinks. Click **Next** to continue.

Select a preferred Access Prober slot to be assigned to the imported Sequences. Click **Next** to continue.
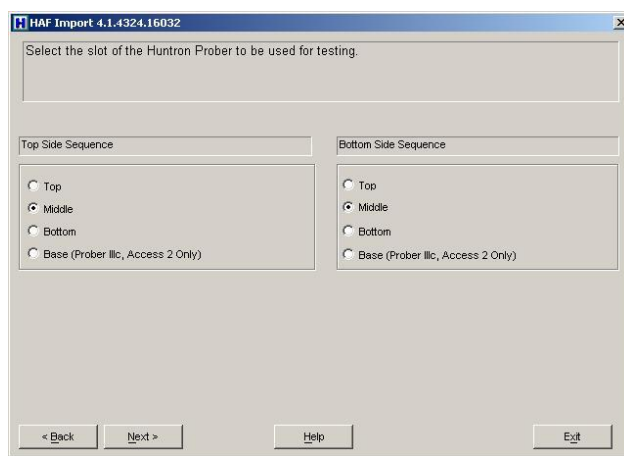
Board Spacers

Board spacers are provided with the Huntron Prober. The spacers allow the board under test to be moved away from the walls allowing components near the edge of the board to be accessed.
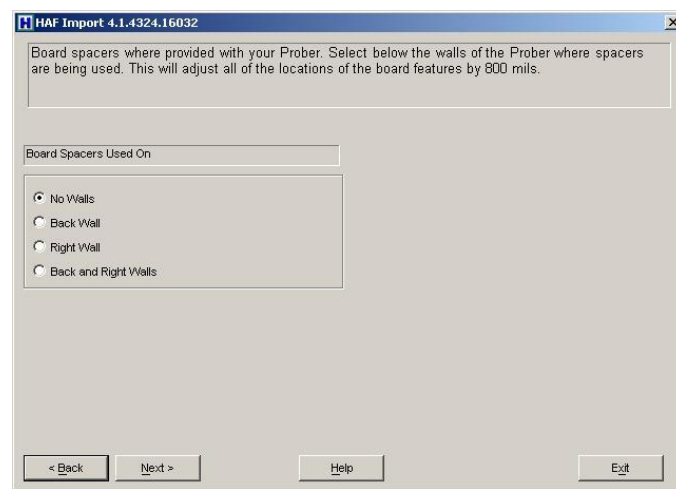

Select the desired setting and click **Next** to continue.

Process

The HAF file will be read to create the Huntron Workstation Sequence test. The file will be displayed in the Board section of the Huntron Workstation window.
Click PROCESS to start. **N**ote: if the file is large this Process can take a long time to complete**.**
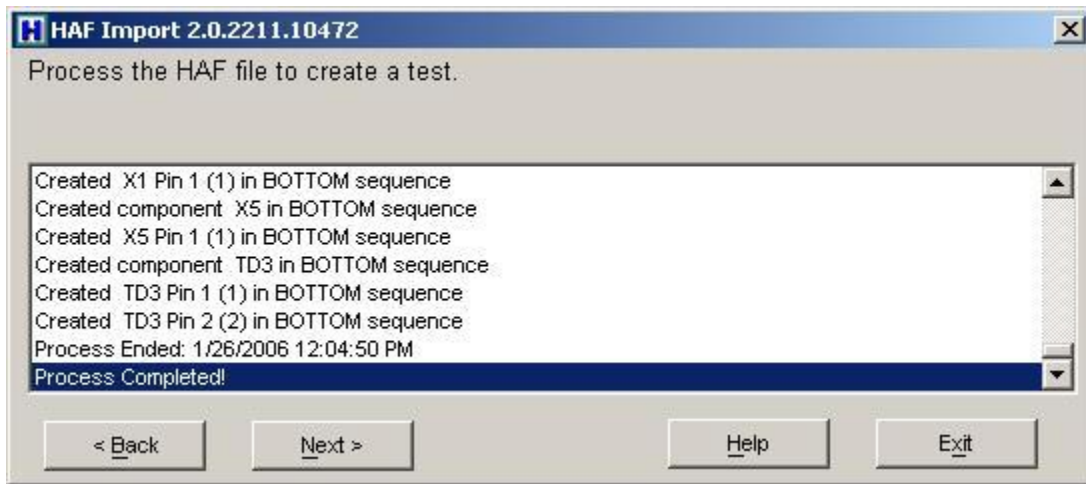
Information will scroll up during the Process. When the Process is complete the words "Process complete" will appear.
Click **Next** to continue.
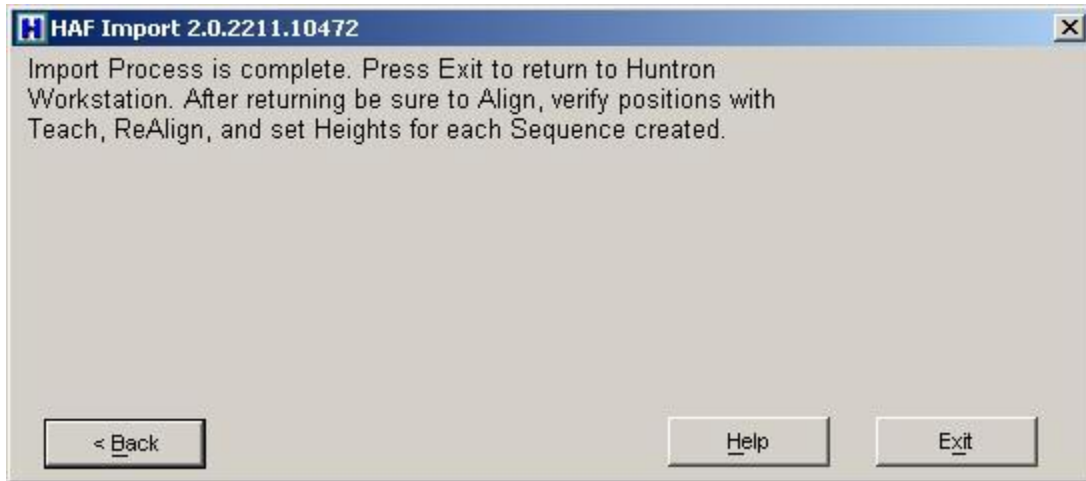
Click **Exit** to return to the Huntron Workstation software.

Board Set Up

Once the HAF Import wizard is closed, you are returned to the Huntron Workstation software. Notice that two Sequences have been created in the Sequences tab of the Tree pane. There should be a sequence for the Bottom side and the Top side of the board. The CAD data obtained from the CC file will be displayed in the CAD tab of the Images pane.
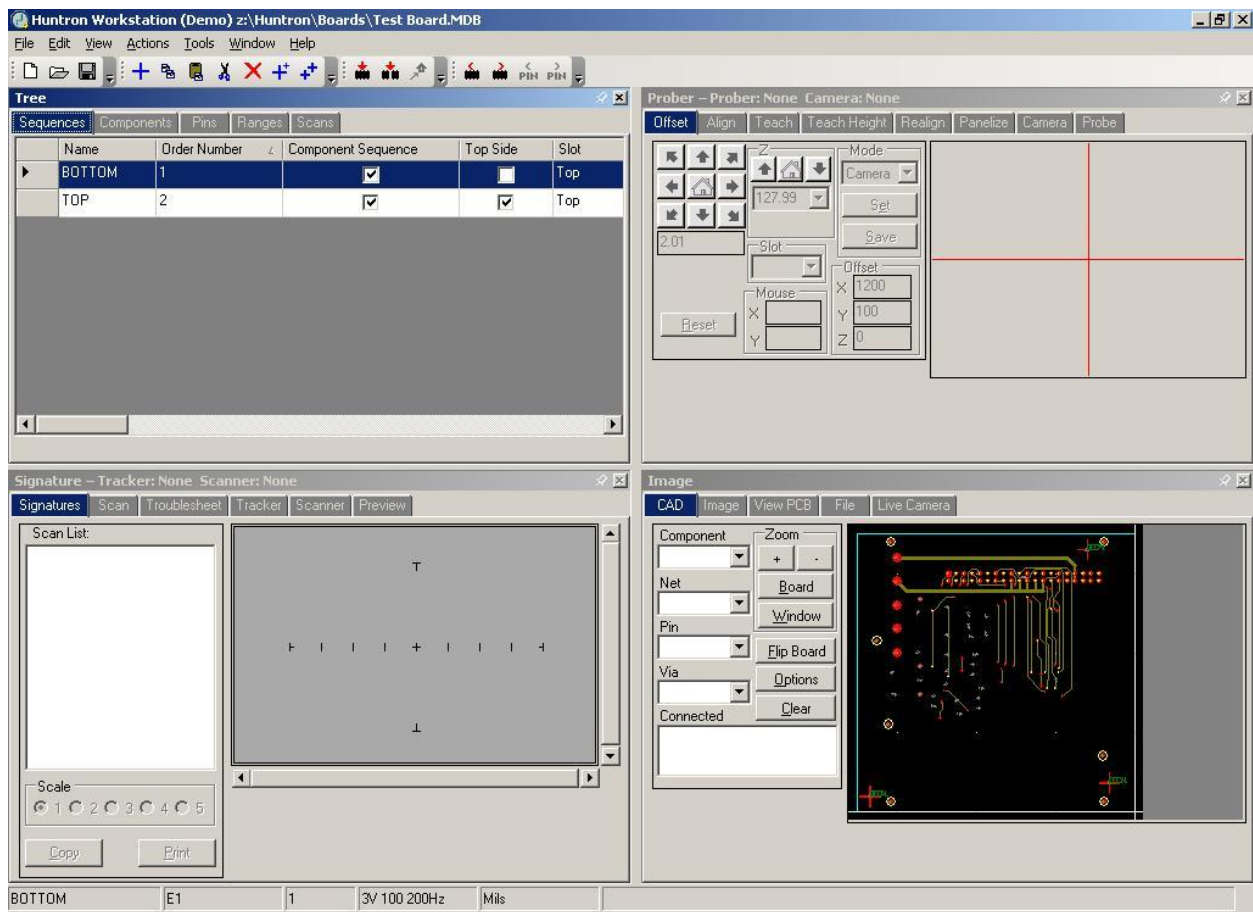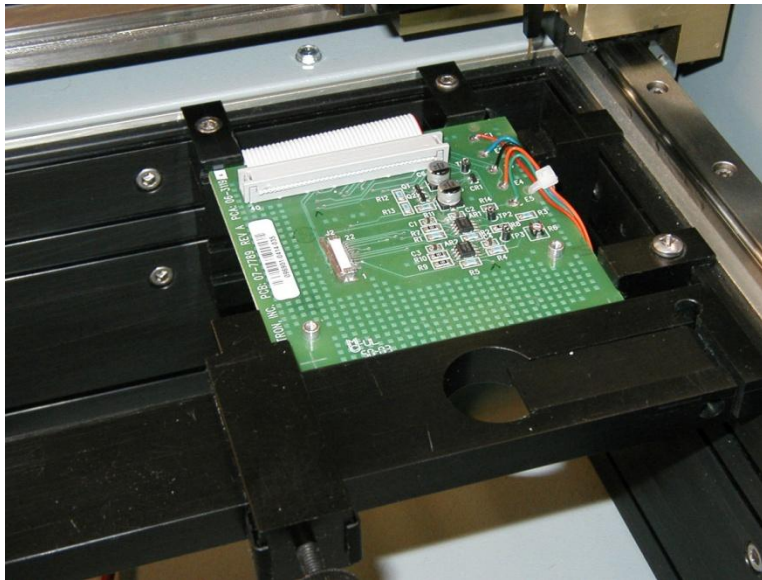
Align and Realign

Select the Sequence you wish to scan in the **Sequence** tab. In the following example, the Top sequence is selected.
Select the **Align** tab in the Prober pane and you will be prompted to mount the board.

Mount the board to test in the Prober making sure the rotational orientation is correct and board spacers (if selected during the Import process) are in place. Click **OK** to continue.

Select **Alignment 1** from the drop menu. The prober will move to the imported XY location for the alignment point. In many cases, you will need to navigate to the point visually using the camera. You can you the **Arrow** buttons to move the camera or you can click directly in the camera image and the camera will move to that point. Place the red crosshairs on the center of alignment point as precisely as possible. Once positioned, click the **Set** button.
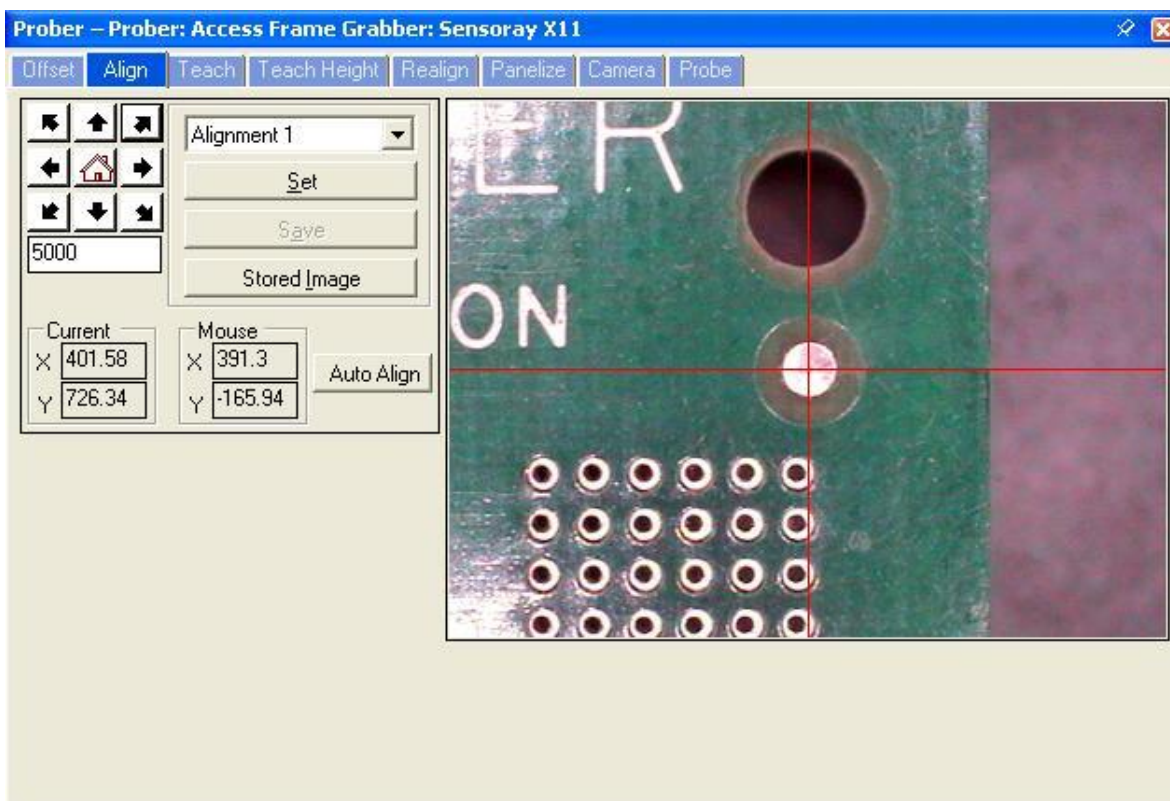
If you are uncertain of where the alignment points are located, click on **View PCB** tab in the Image pane to display the board outline and the two alignment points.
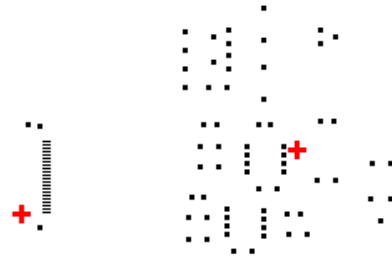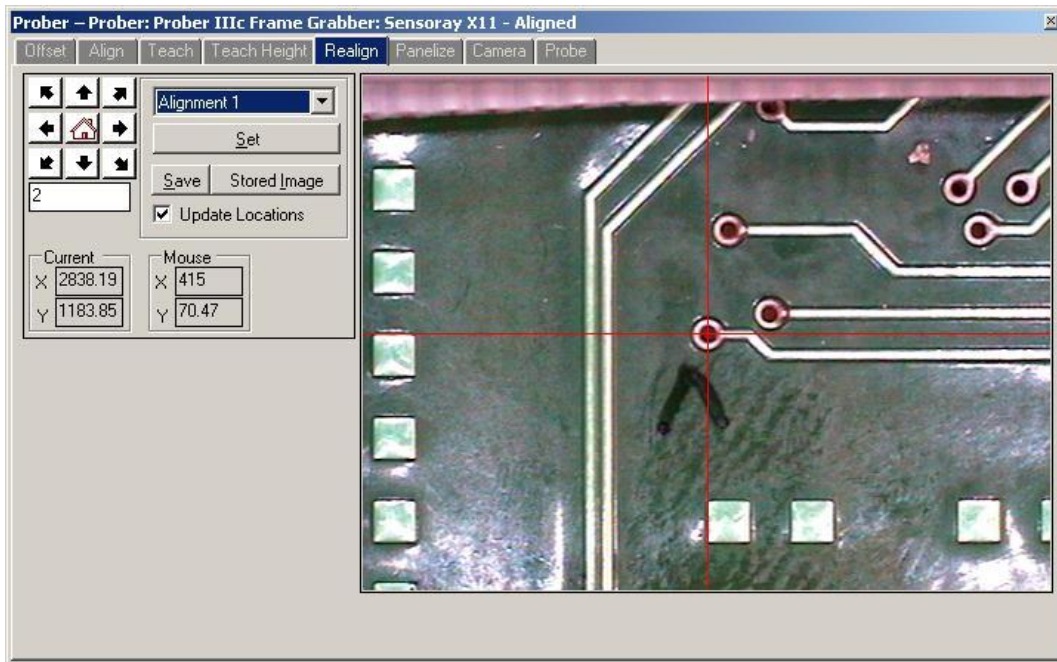

Select **Alignment 2** from the drop field and the prober will move to the imported XY location for the alignment point. In many cases, you will need to navigate to the point visually using the camera. Place the red crosshairs precisely on the center of the second alignment point. Click **Set** to save the point.
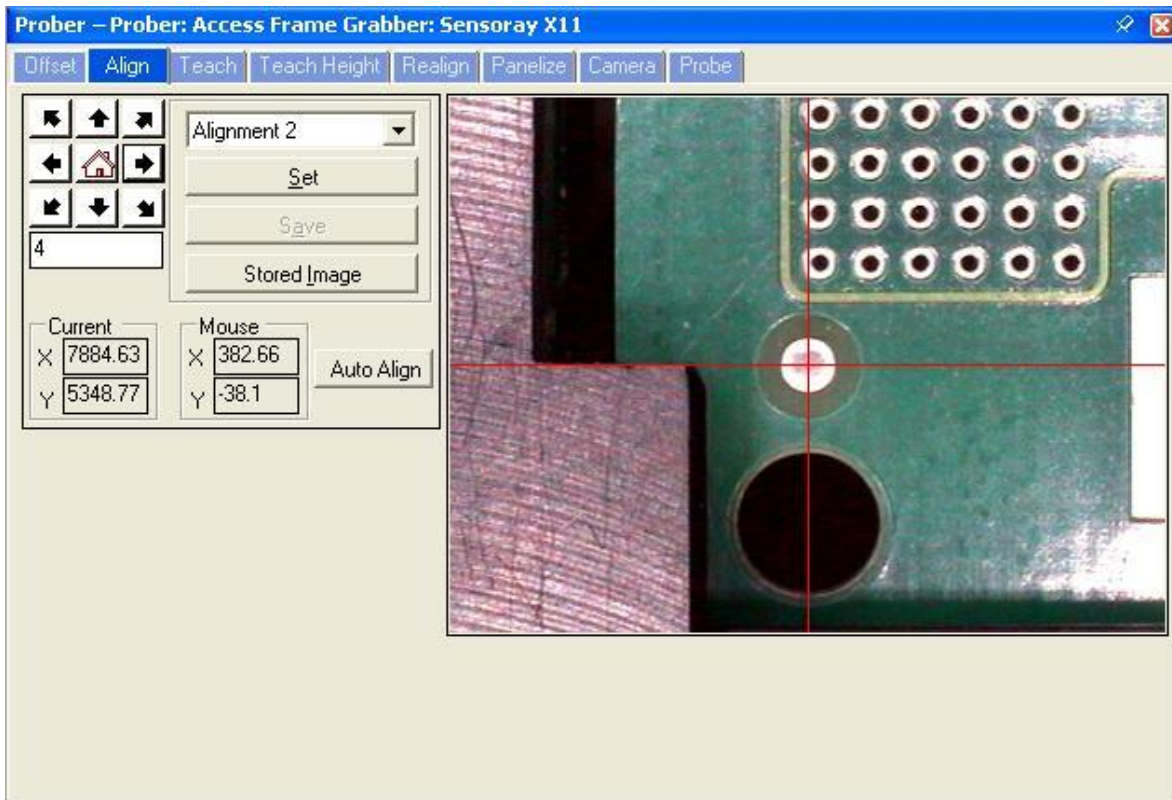

Currently, the alignment point positions are only stored temporarily. At this point, select the **Teach** tab and verify the pin positions of a few random components. This will ensure that the CAD data imported correctly. If the pins positions look good, proceed to **Realign**, if they are off, make sure you have the correct alignment points and recheck the Alignment in the **Align** tab. To make the alignment positions permanent, select the **Realign** tab in the Prober pane. Before proceeding with the Realign, it is **very important** that you align the board precisely in the **Align** tab first. Failure to do so can cause the XY location data to be

modified improperly. Since Realign permanently modifies the XY data, you may also want to select **Save Copy** from the File menu and save a copy of the current database under a new name. This will allow you to go back to the original file if problems during Realign are encountered.

In the **Realign** tab, select **Alignment 1** from the drop menu. The camera will move to the first alignment point. Click **Set** to save the position.

Select **Alignment 2** from the drop menu. The camera will move to Alignment 2. Click the **Set** button to save the position. Click the Save button to make these alignment points permanent.
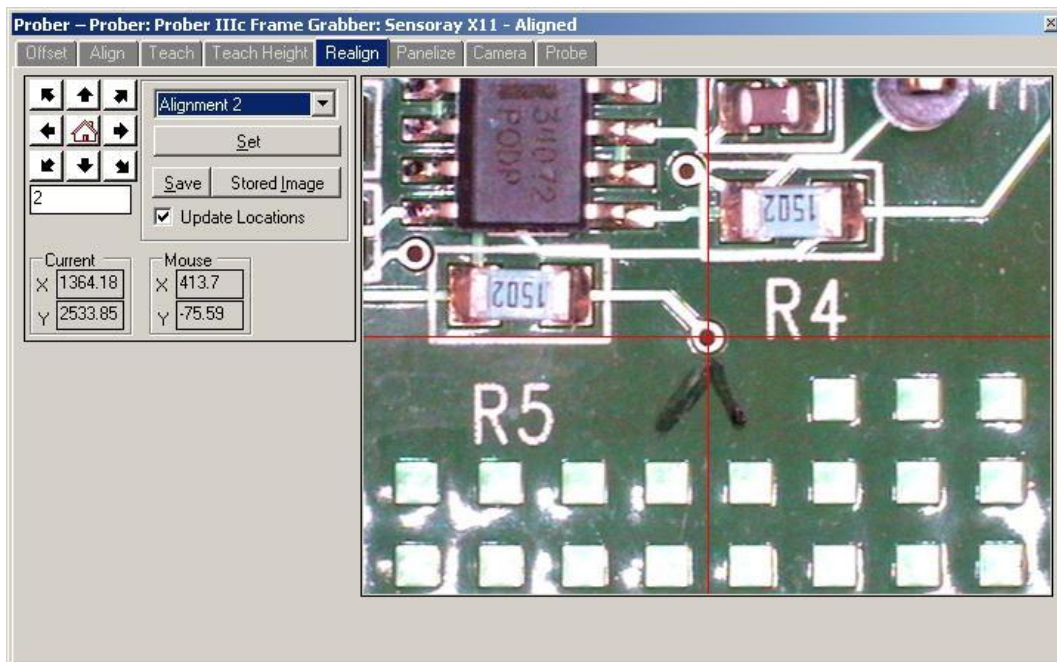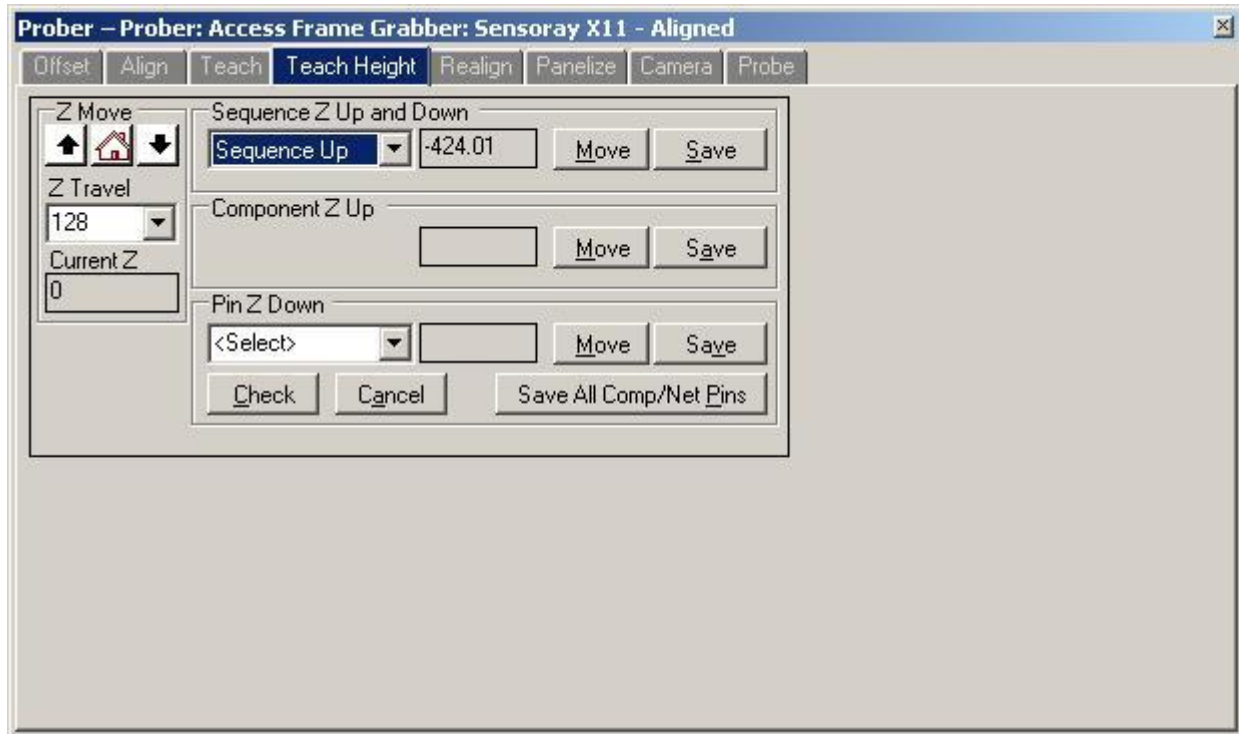
You will be prompted to make these changes. Click **OK** to continue.
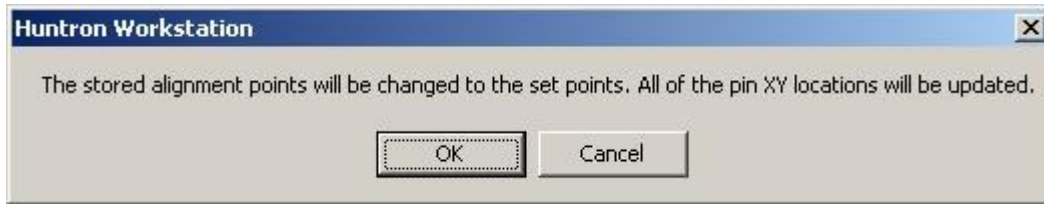
Once the Realign is complete, the Z axis positions need to be set.

Teach Height

To set the up and down probe movement positions for the prober, click the **Teach Height** tab in the Prober pane. Select **Sequence Up** from the drop field in the Sequence Z Up and Down section.


Use the **Z Move** buttons to lower the probe tip to a position above the board where it will clear any tall components or objects on the board. Use the Z Travel field to control the distance the probes with each click. Click the **Save** button to store this Z position. Select a pin from the **Pin Z Down** drop field. Select Sequence Down from the drop field in the Sequence Z Up and Down section.
Use the **Z Move** buttons to lower the probe tip until it makes contact with the board. Make sure the probe tip is down far enough to ensure good contact to the test points. Click **Save** to store the down position. You will be prompted to confirm saving the down position for all components or nets. Click **Yes** to continue.
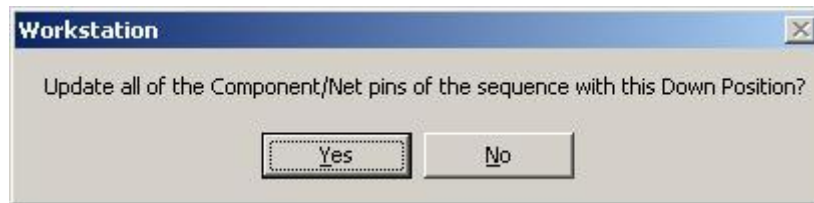
Verify Component Position

To check that all changes have worked properly up to this point, select the Teach tab in the Prober pane.
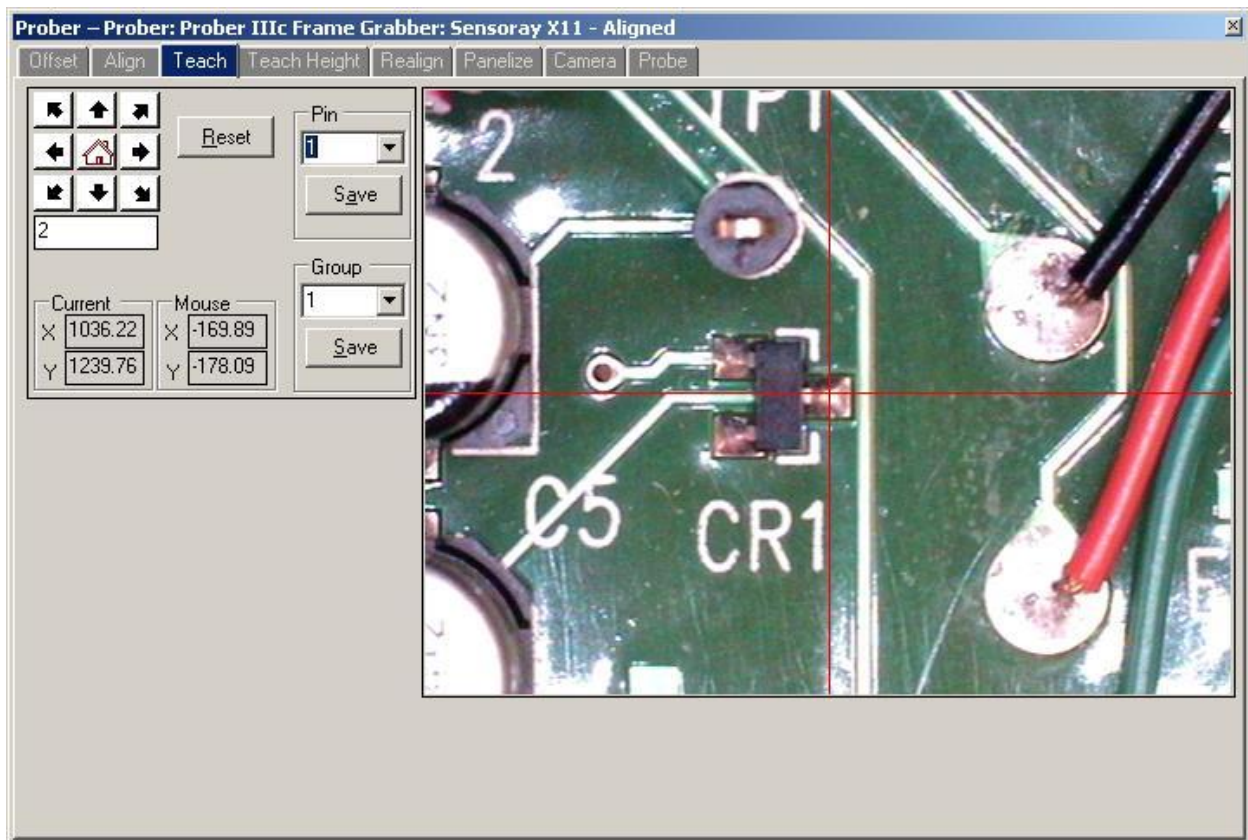
Select **Pin 1** from the Pin drop field and the camera will move to pin 1 of the selected component. Look in the status bar at the bottom of the main window or select the Component tab in the Tree pane to see which component is selected.

Working with a Net based Test

Net based Sequences will be created if **Net** was selected during the Import process. Net based tests differ from Component based tests in that they test each unique electrical point on the PCB only once. This results in a shorter test run than Component based Sequences. Note that the tab to the right if Sequence in the Tree pane is labeled **Net.**

Align the Sequence as described earlier in the Component based test. Similar to Component based tests, correct placement of the Net positions should be verified in the Teach tab of the Prober pane. Once correct placement is verified, perform a **Realign** as described earlier.

The Teach tab in the Prober pane will appear different for Net tests compared to a Component test.
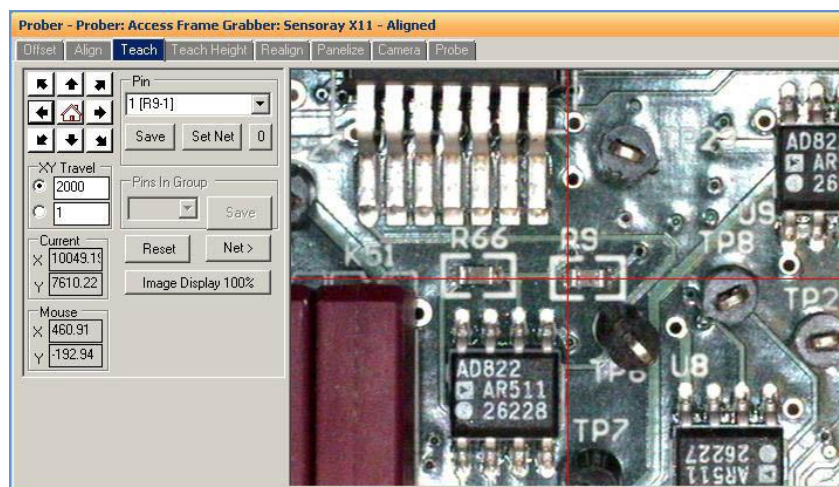

Each pin of the net for the selected board side (Top or Bottom) will be listed in the Pin drop menu. The Prober camera will move to the selected pin. You can modify the position by moving the camera crosshairs to the desired position and clicking the **Save** button. If you wish to set a different pin to be the tested pin, select the desired pin from the **Pin** drop menu and the camera will move to that pin. Click the **Set Net** button to make that pin the tested pin. If you find that none of the pins on the selected Net are accessible with the Prober, clicking the **0** button will disable testing of the Net by setting the test pin to 0 (shown in the **Top Pin** column of the Tree pane/Net tab).

You can use the **Net>** button to step through each of the Nets listed in the Tree pane/Net tab to verify that the net is accessible by the Prober.

Optimizing Component/Net Probing

The best method to optimize the movement of the probe is to use the Set By Location feature. The will reorganize the component order based on their position.

To use Set By Location, right-click the **Order Number** column header in the Tree pane/Component tab and select **Set By Location** as shown below.
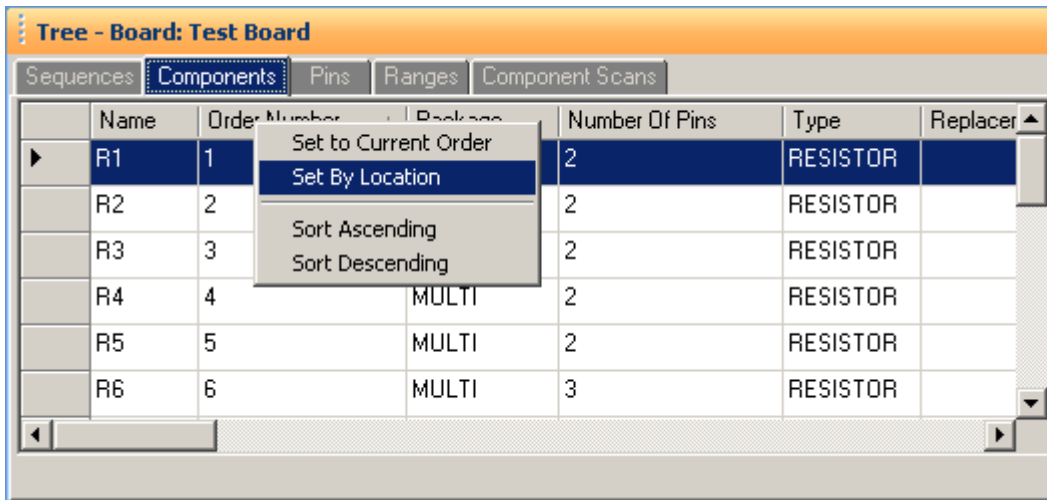
The components will be reordered based on their pin 1 XY coordinate. The pattern will be determined by the longest board length dimension and the **Sort Band Width** setting in the Tools/Options/Sequence tab. The Sort Band Width setting is the width of the path used to sort the components. This will create "snake-like" pattern that the probe head will follow while probing. The smaller the Sort Band Width, the more back-and-forth paths will be used. If the PCB under test is very small, set the Sort Band Width to a small setting such as 500 mils.

Scanning

To execute a scan, select the **Scan** tab in the Signature pane.
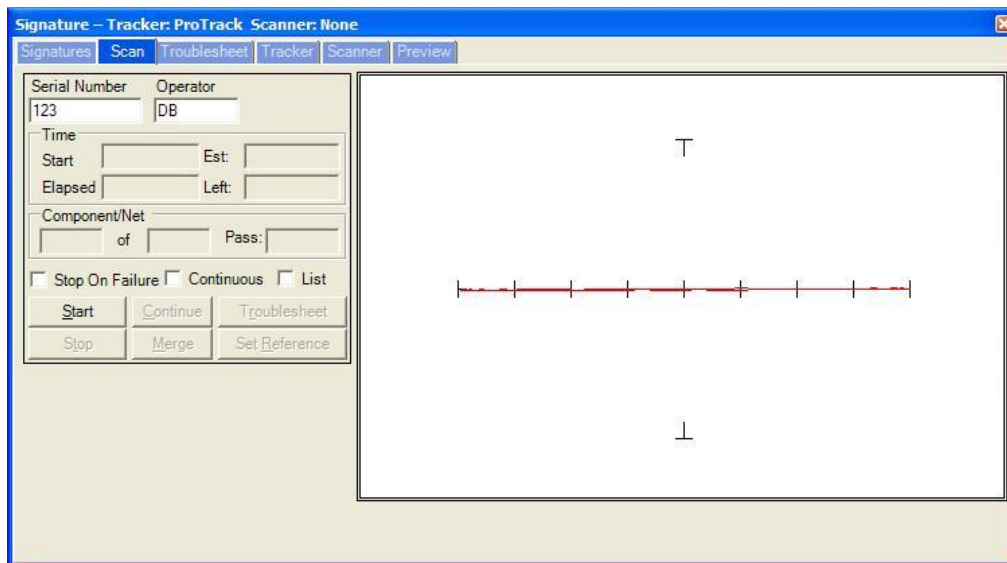
Connect a Common reference lead to the board under test (typically to board ground). Type a serial number into the **Serial Number** field. The serial number is simply a name assigned to the scan and subsequently stored signatures. Click the **Start** button to execute the scanning of the selected component. Signatures will be displayed as the prober scans.

This concludes this tutorial. For more information, see the Help in the Huntron Workstation software or view the Huntron Workstation Tutorial.