Huntron Workstation CAD Import Tutorial using Unisoft

Contents

Installing the Unisoft Software 2
Creating a Unisoft XML file for Import into Huntron Workstation 2
Importing CAD data into the Unisoft software 3
Selecting Reference 1 and Reference 2 (Alignment points) 5
Additional step for Access DH users – Selecting Board Reference 9
Setting a Board Origin 9
Creating a New Point to Use as the Board Origin (Make Point) 10
Export the Unisoft XML File 12
Create a Board database 14
File Import 14
Board Set Up 18
Align and Realign 19
Teach Height 24
Verify Component Position 25
Working with a Net based Test 26
Optimizing Component/Net Probing 27
Scanning 27
Appendix 1: How to Modify the Unisoft XML Output File 29
XML File Basic Layout 29
Editing the XML File 30

Huntron Workstation CAD Import Tutorial

The tutorial will guide the user through the steps require to create a Unisoft XML and FB2 data files. These files contain X-Y coordinates that allow the Huntron Prober to accurately probe a print circuit assembly.

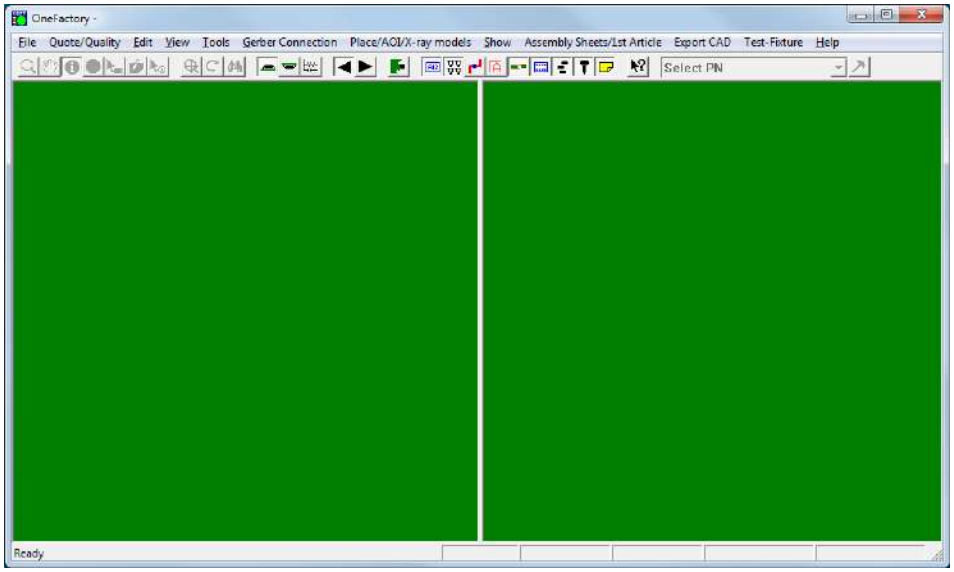
Installing the Unisoft Software

1. Power up your computer. Insert the Unisoft software CD into the CDROM drive.
2. Browse to the CD drive and locate the "**view5.exe**" file and double click to start the installation.
3. Proceed through the installation process click **Next** when necessary, accepting the software license agreement and clicking the **Finish** button when the installation is complete.
4. Browse to the Unisoft CD again and locate the "**update-view.exe**" file and double click to start the software update installation.
5. Proceed through the installation process click **Next** when necessary, accepting the software license agreement and clicking the **Finish** button when the installation is complete.
6. Copy the MARKUP.LIC from the Unisoft CD to the Program Files\Unisoft folder. This is your license file.

Creating a Unisoft XML file for Import into Huntron Workstation

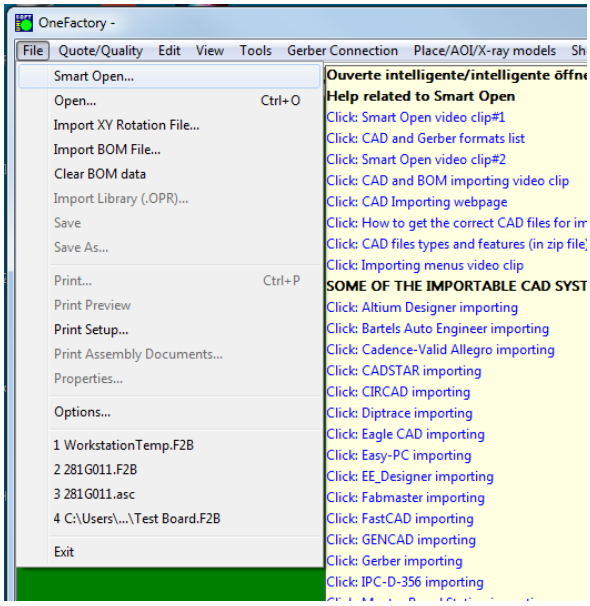
The Unisoft software is used to import ASCII CAD data from various CAD package formats, select Huntron Access Prober alignment points and then export a Unisoft XML that can be read by Huntron Workstation to create a test database. This process describes how to create the Unisoft XML file.

Double click the Unisoft Pronto VIEW-MARKUP icon on your Windows desktop. This will start the program.

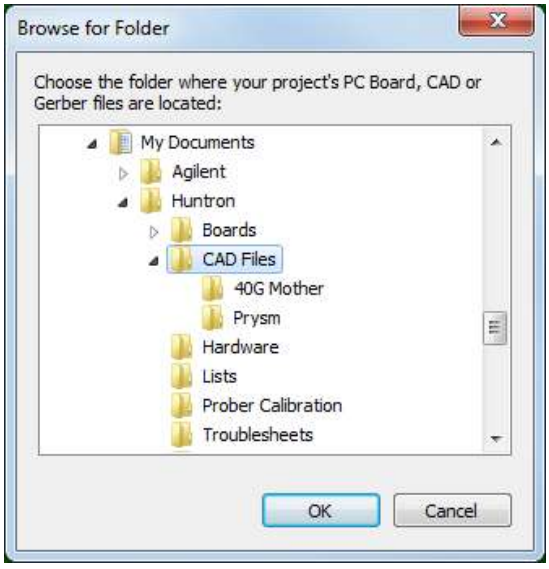


Importing CAD data into the Unisoft software

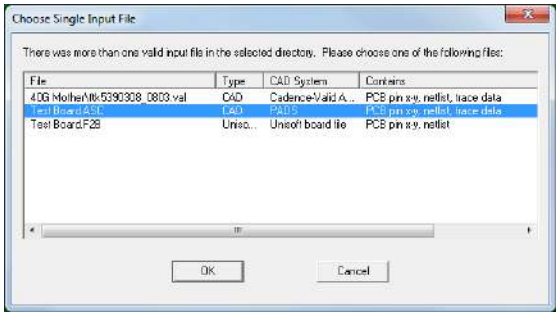
Select **Smart Open** from the **File** menu.



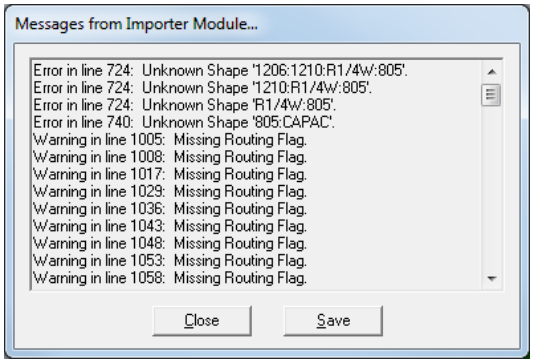
Browse to the location of your CAD source file or files and click **OK**.



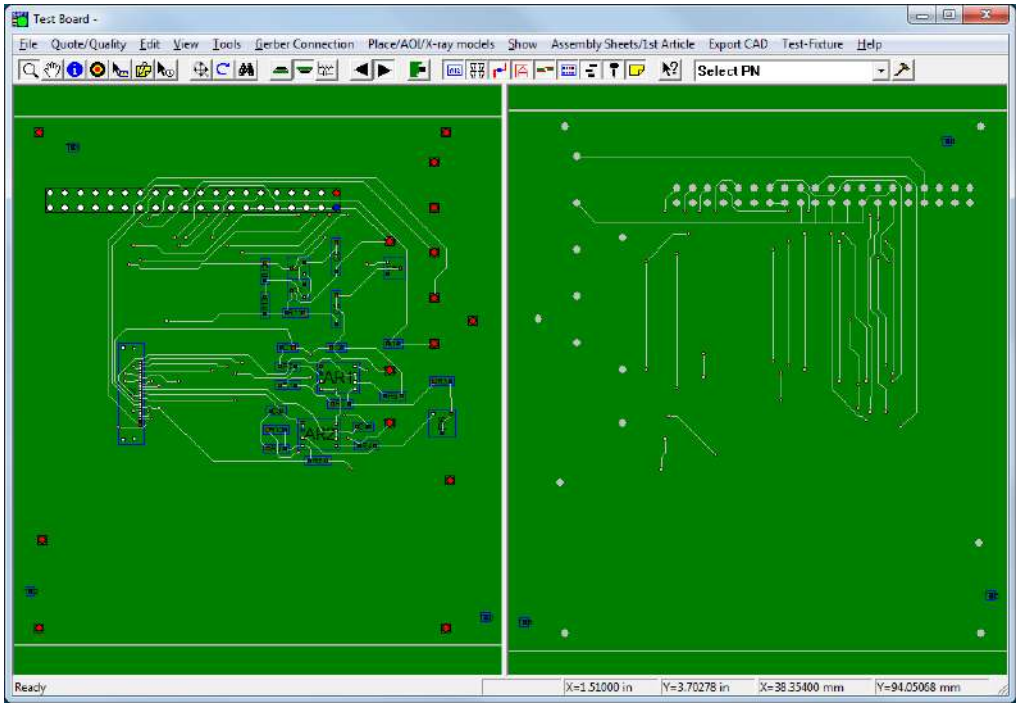
If more than one supported file type is stored at this location, select the correct file and click **OK**.



You will see a Messages textbox displayed during the import process indicating import status and errors. Most errors displayed in this textbox will not affect proper import of the necessary information for creating a test. Click the **Close** button to close the textbox.

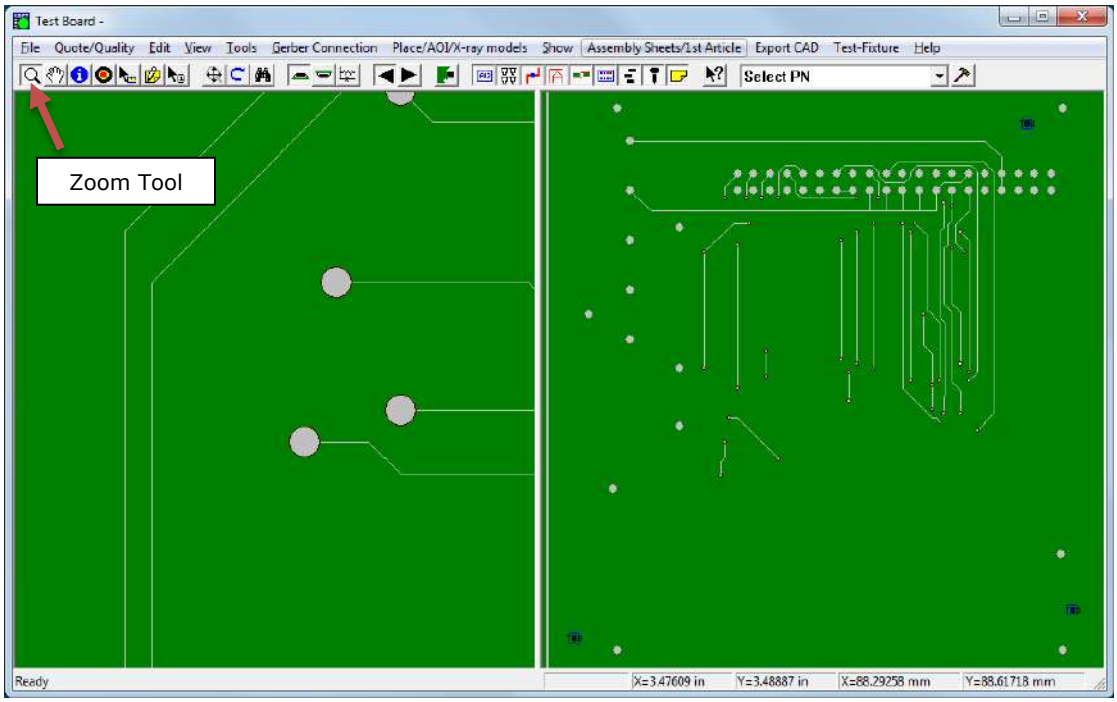


The selected CAD file will be displayed in Unisoft software. By default the Unisoft software will show a split screen with the Top side of the PCB displayed to the left and the Bottom side displayed on the right.

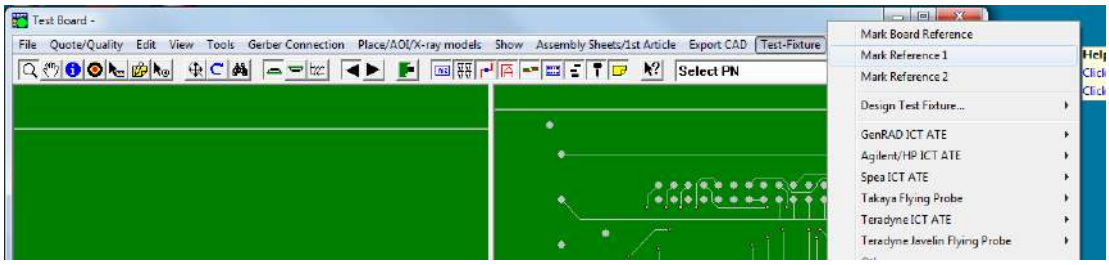


Selecting Reference 1 and Reference 2 (Alignment points)

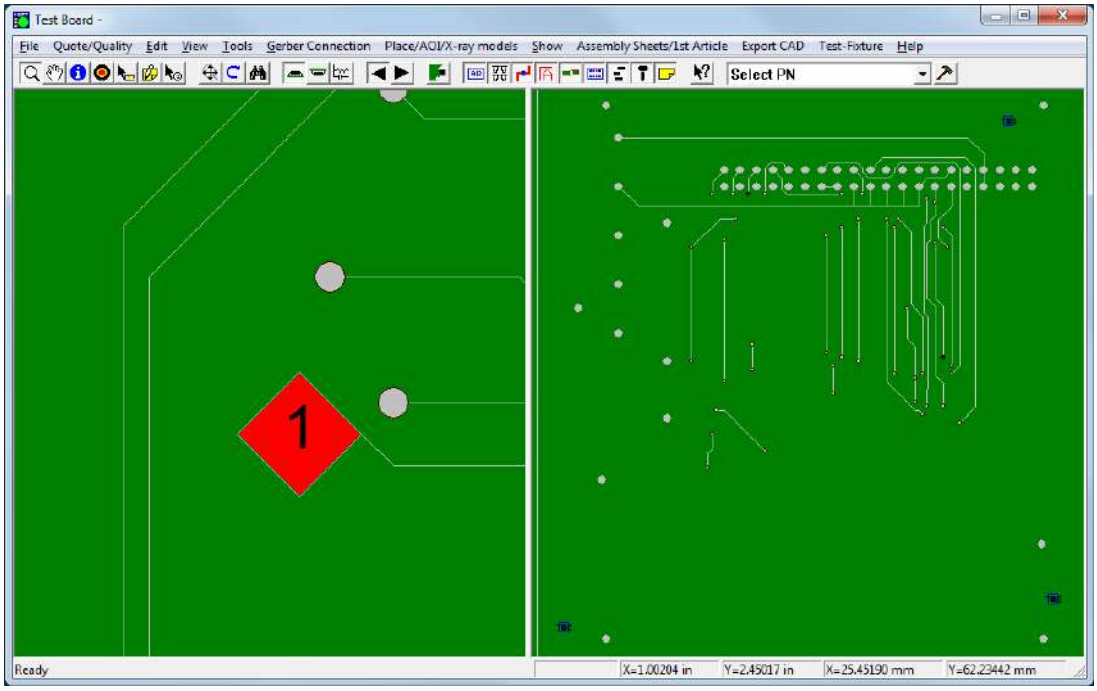
The next step is to select the alignment points to be used by the Huntron Access Prober. Select the **Zoom** tool from the toolbar and click/drag a selection box around the area in the Top side view (left side of window) where your **Top Side** first alignment point is located.



Click the **Test-Fixture menu** and select **Mark Reference 1.**

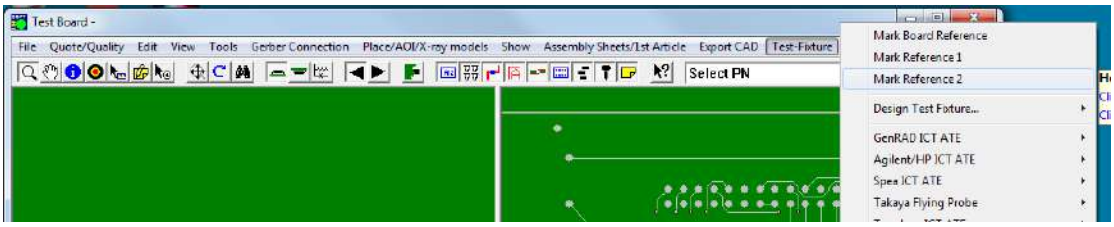


The cursor will change to an arrow with a number 1 label. Click the mark in the CAD view to mark it as the Reference 1. Note that the mark changes to a red color and is labelled as number 1. This will be used as the Alignment 1 for the top side of the PCB.

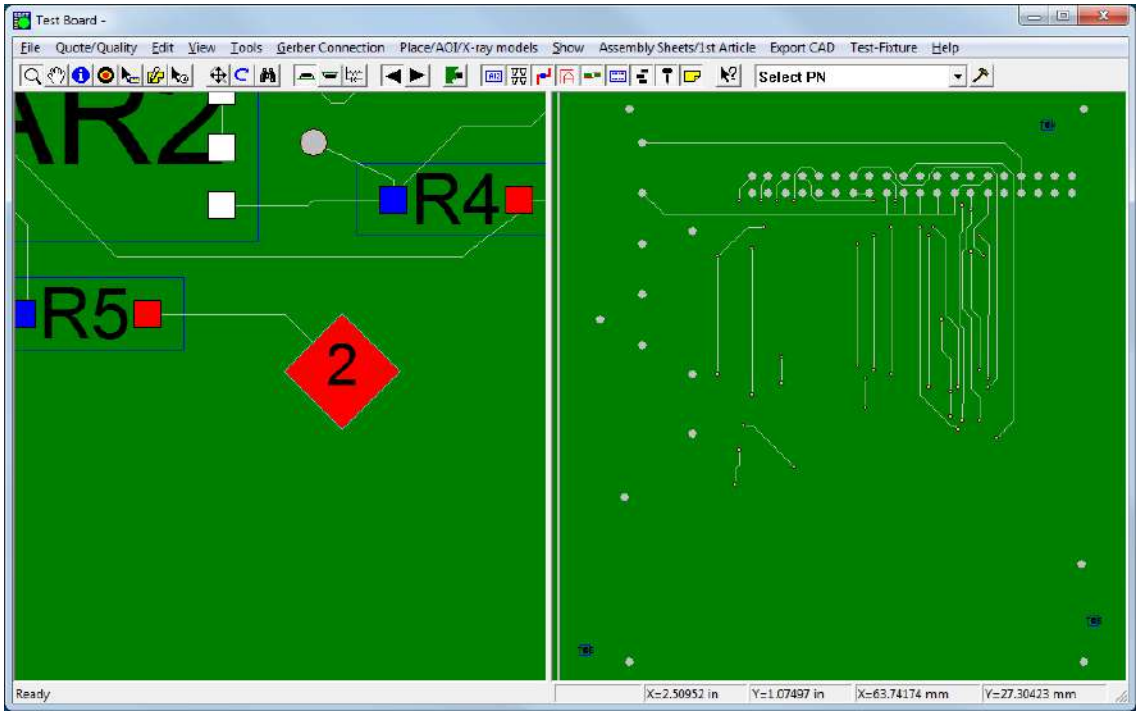


Click the Zoom to Fit button in the tool bar (mouse over the tool bar buttons to see their function) to see the full top side PCB. Select the **Zoom** tool from the toolbar and click/drag a selection box around the area where your **Top Side** second alignment point is located.

Click the **Test-Fixture menu** and select **Mark Reference 2**.

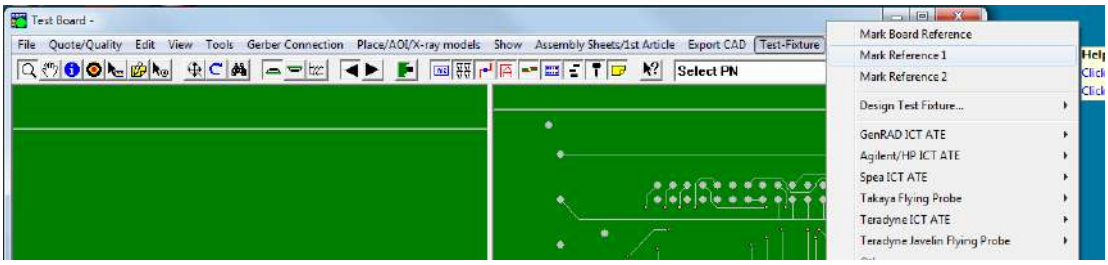


The cursor will change to an arrow with a number 2 label. Click the mark in the CAD view to mark it as the Reference 2. Note that the mark changes to a red color and is labelled as number 2. This will be used as the Alignment 2 for the top side of the PCB.

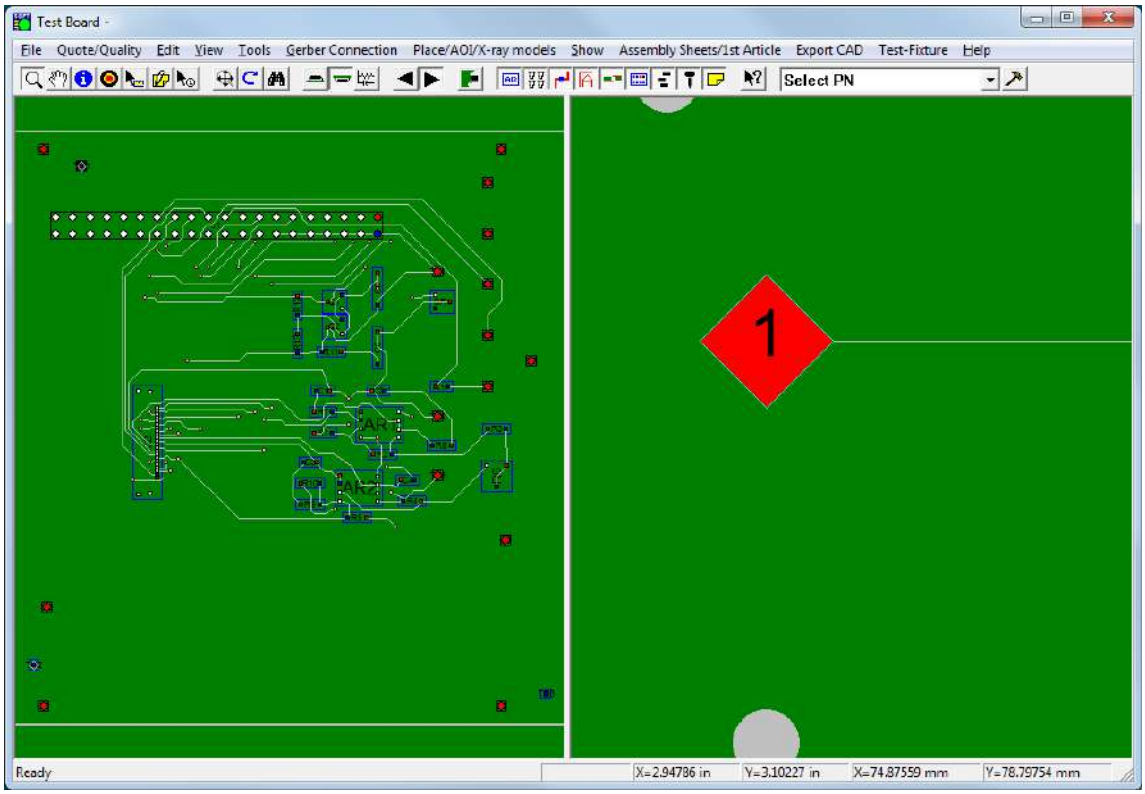


Proceed to select and mark alignment points to the bottom side of the PCB. Select the **Zoom** tool from the toolbar and click/drag a selection box around the area in the Bottom side view (right side of window) where your **Bottom Side** first alignment point is located.

Click the **Test-Fixture menu** and select **Mark Reference 1.**

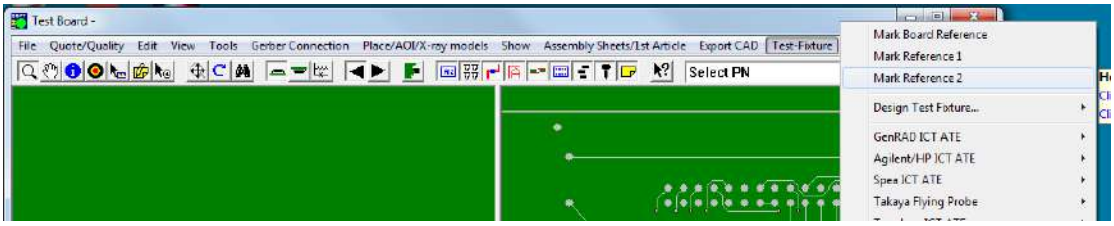


The cursor will change to an arrow with a number 1 label. Click the mark in the CAD view to mark it as the Reference 1. Note that the mark changes to a red color and is labelled as number 1. This will be used as the Alignment 1 for the Bottom side of the PCB.

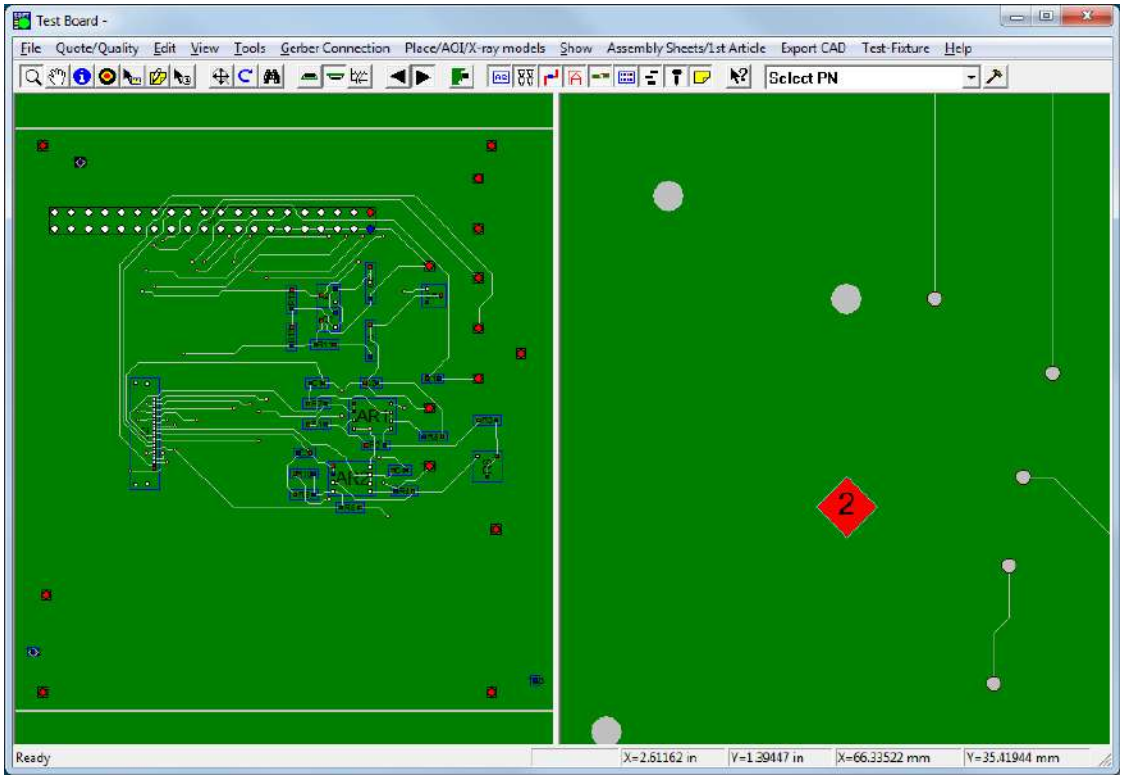


Click the Zoom to Fit button in the tool bar (mouse over the tool bar buttons to see their function) to see the full bottom side PCB. Select the **Zoom** tool from the toolbar and click/drag a selection box around the area where your **Bottom Side** second alignment point is located.

Click the **Test-Fixture menu** and select **Mark Reference 2**.



The cursor will change to an arrow with a number 2 label. Click the mark in the CAD view to mark it as the Reference 2. Note that the mark changes to a red color and is labelled as number 2. This will be used as the Alignment 2 for the Bottom side of the PCB.



Your alignment points are now selected. To change the points simply repeat the process listed above to select different points.

Additional step for Access DH users – Selecting Board Reference

This step is not required but can be very useful if you use the camera alignment function with an Access DH Prober. The Board Reference can be marked in the Unisoft software and then used by Huntron Workstation as the positions for **Z1 Camera** and **Z2 Camera** alignment.
The process to select the Board Reference point is similar to selecting the Reference 1 and 2 points used earlier. Locate the point on the Top side of the PCB view you wish to use for the Board Reference. In the **Test-Fixture menu** select **Mark Board Reference**. The cursor will change to an arrow with a black diamond label. Click the point in the PCB view that you want as the Board Reference and the point will be marked by a red diamond. Complete this same process for the Bottom side of the PCB as well.
End of Additional steps for Access DH users

Setting a Board Origin

It is important to set a board Origin so that the PCB positioning will import properly into Huntron Workstation. **The Origin is set in the TOP side view of the PCB.**

The Board Origin can be set to any selectable object or entity on the PCB (i.e. mounting hole, fiducial mark, etc.) that is in the **extreme bottom right corner** of the board. It is important that the origin point you select encompasses all components above the origin point's X axis and to the left of the Y axis. If do not have a point on the board that will fit these requirements then proceed to the **Make Point** section below then use that New Point as the Origin.

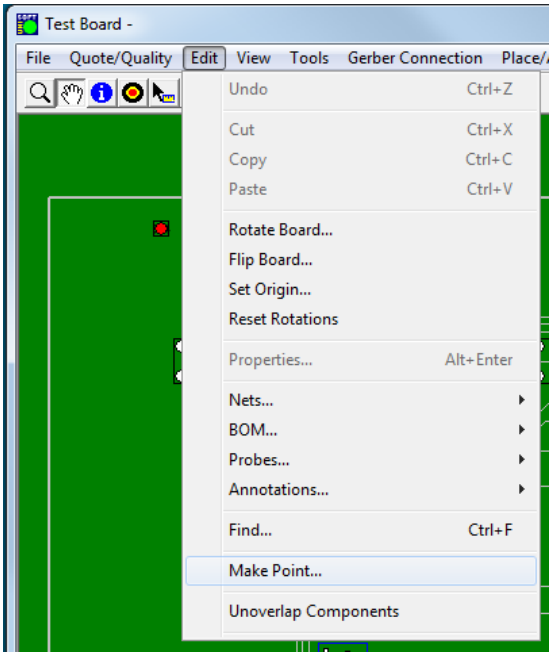
Select **Set Origin** from the Edit menu. The cursor will change to an arrow with a black/white circle.



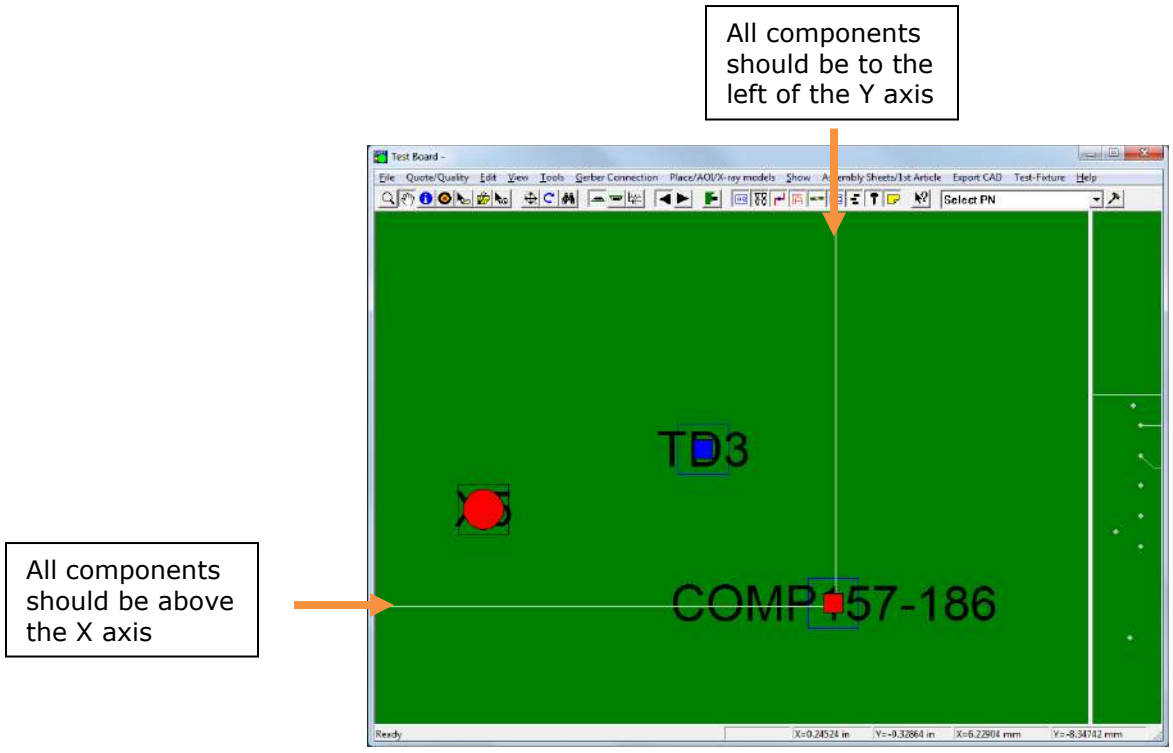
Click the object/entity located in the **bottom right corner** of the PCB that you wish to use as the Origin.

Creating a New Point to Use as the Board Origin (Make Point)

If a suitable object/entity is not available in the bottom right corner of your PCB then you can create a new point to use for this purpose.
Select **Make Point** from the Edit menu.



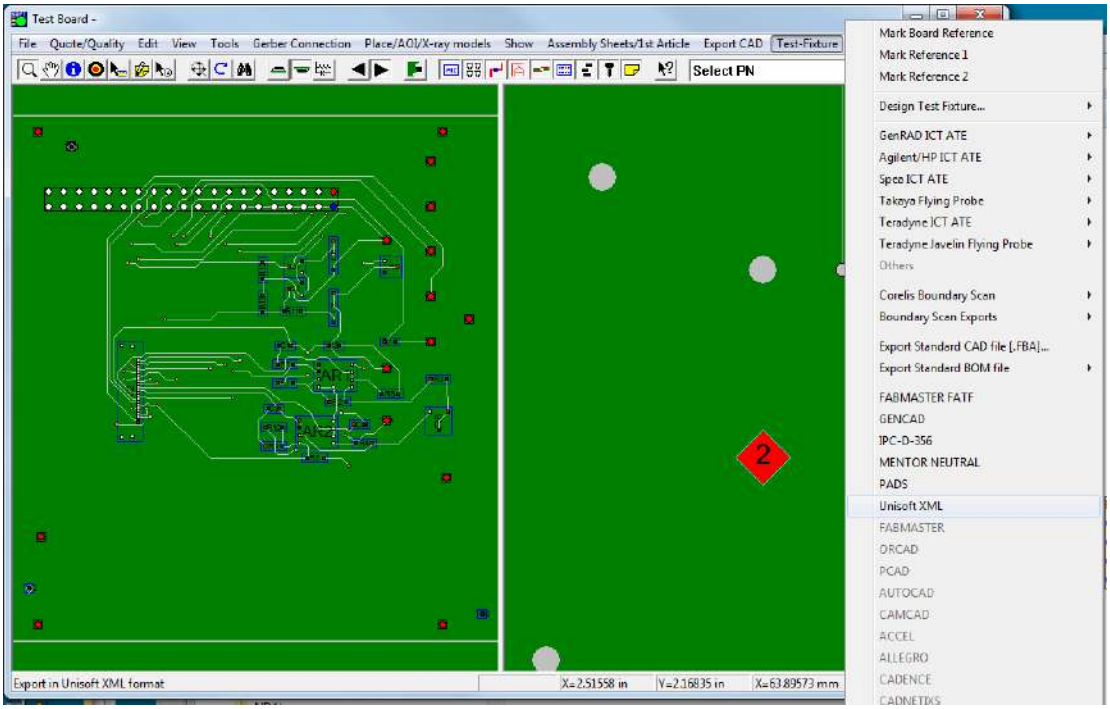
The cursor will change to an arrow with a small pink box. Click at a point in the extreme bottom right corner of the PCB and you will see a new point added. A default name will used to label the new point.



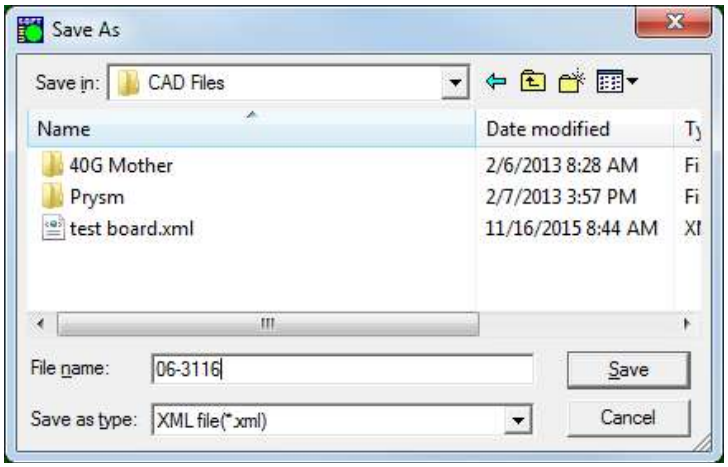
You can know go back to the previous step for selecting the Origin using the new point you created.

Export the Unisoft XML File

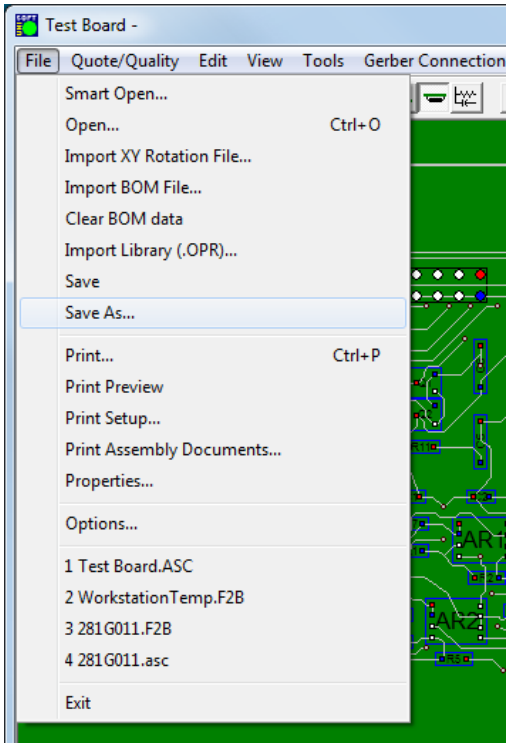
The next step is to export an XML file from the Unisoft software. Click the Test-Fixture menu and select **Unisoft XML**.



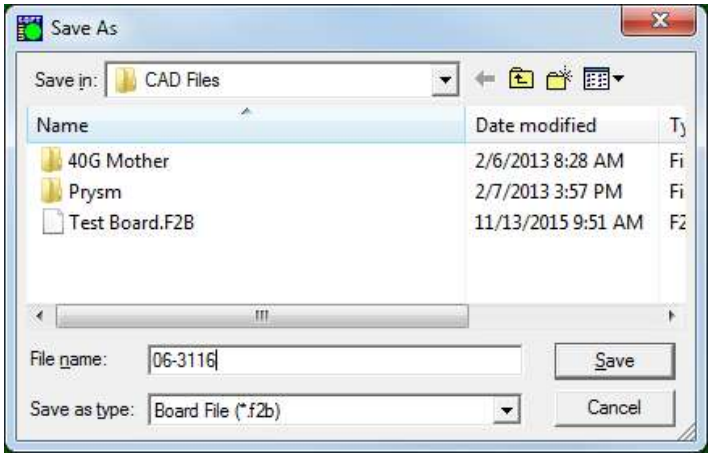
Type in a file name and select the location where you want to save the file. Click **Save** to store the file to the selected location.



Click the **File menu** and select **Save As...** to save a Unisoft F2B file.



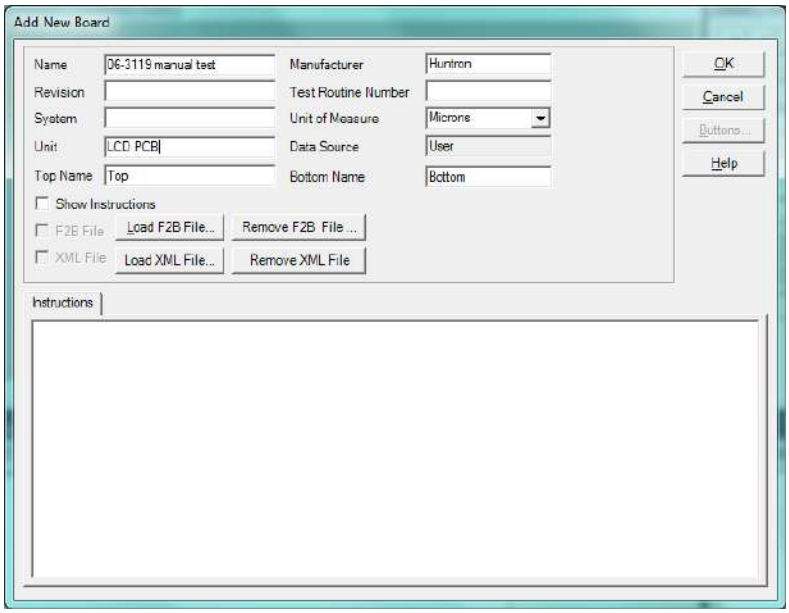
Type in a name and select a location to store the F2B file. It is recommended that you save the F2B file to the same location as the Unisoft XML file saved earlier. Click the **Save** button to store the F2B file.



The creation of the external files needed for building a CAD based Huntron Workstation test is now complete.
The next steps are performed in Huntron Workstation.

Create a Board database

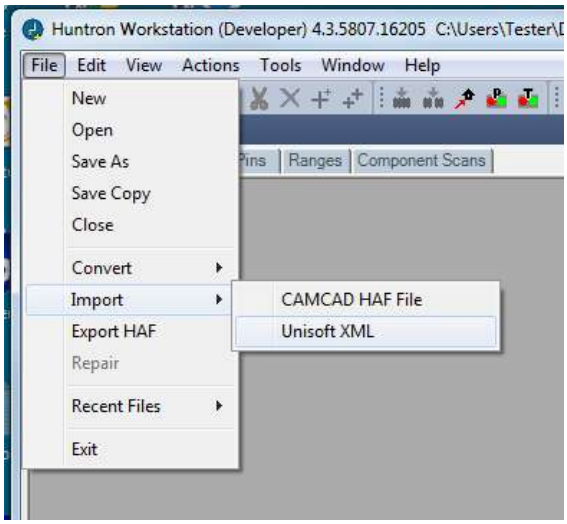
After starting the Huntron software the main menu is displayed. Select **File/New** from the menu bar.



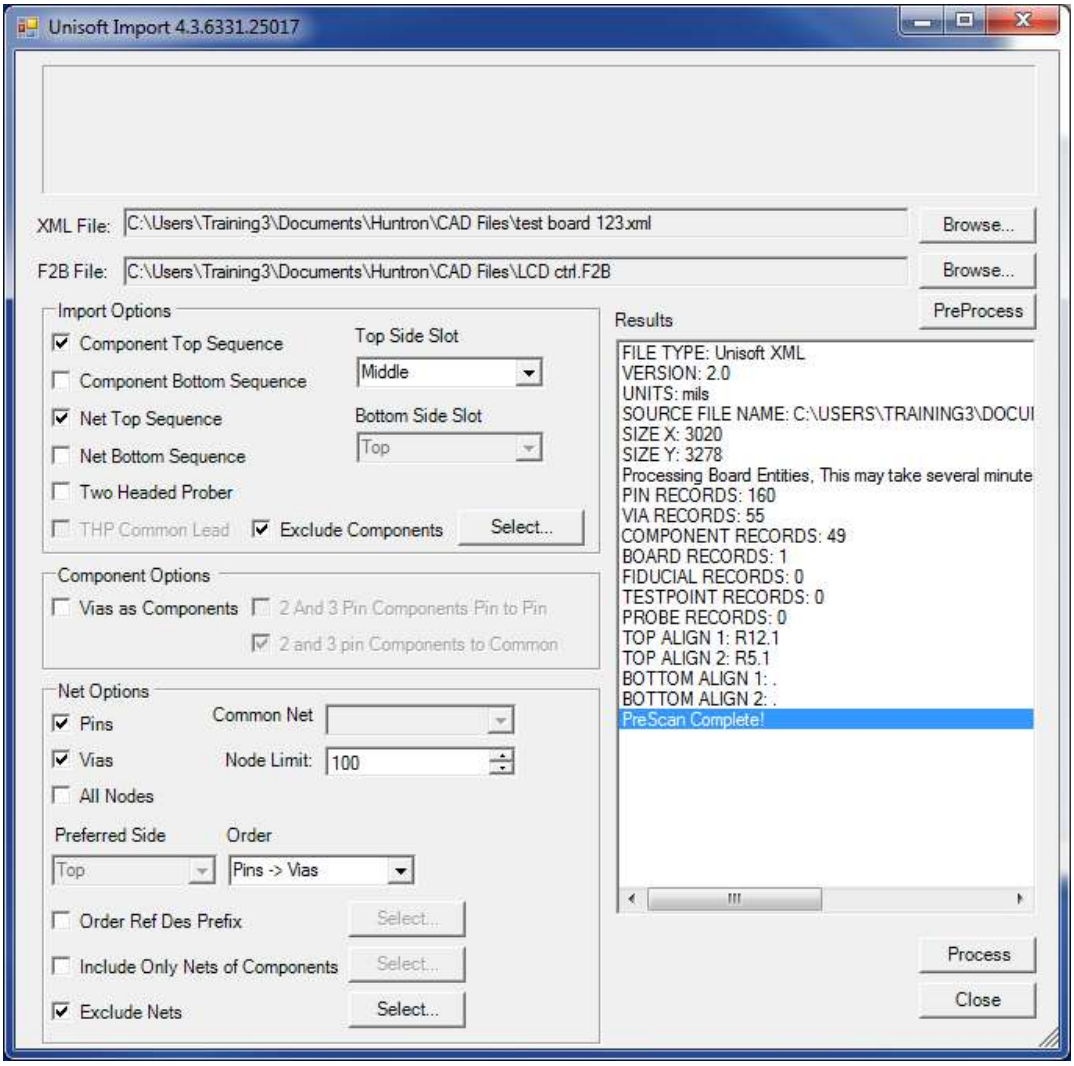
The Add New Board window will be displayed. Type a name into the Name field. Adding information to the additional fields (i.e revision, system, etc…) is not necessary to create a new board. Click **OK** to save the new Board information.

File Import

After created a Board database, select **File/Import/Unisoft XML** from the menu bar.

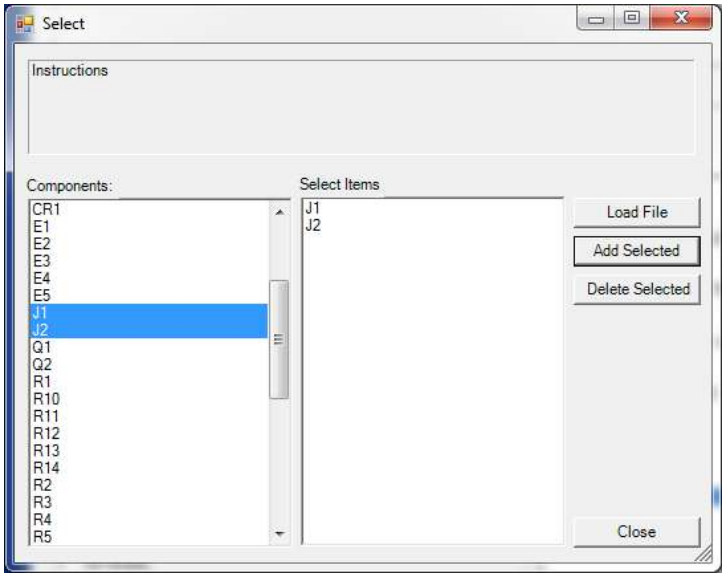


The Unisoft Importer window will open where you will configure your settings for the new Sequence(s).



The process for using the Workstation Importer for Unisoft XML files essentially works from top to bottom. Following is a description of each setting and how it is used.
XML File: Browse to and select the XML file generated from the Unisoft software
F2B File: Browse to select the .F2B file generated from the Unisoft software
PreProcess button: Click this button when the XML and F2B files are selected. Note that all of the fields and checkboxes below are greyed out until this step is completed. Once this step is completed successfully you can move on to the Import Options.
Import Options – Component Top Sequence checkbox: Enable to add a Top side Component based Sequence to the Board
Import Options – Component Bottom Sequence checkbox: Enable to add a Bottom side Component based Sequence to the Board
Import Options – Net Top Sequence checkbox: Enable to add a Top side Net based Sequence to the Board
Import Options – Net Top Sequence checkbox: Enable to add a Bottom side Net based Sequence to the Board
Import Options – Two Headed Prober checkbox: Enable this setting if you are using an Access DH Prober. Note that some additional checkboxes become available (i.e. THP Common Lead).

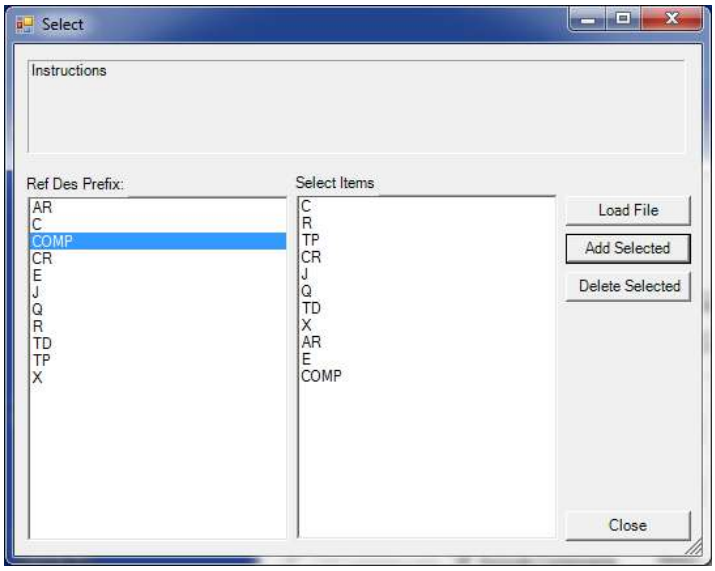
Import Options – THP Common Lead checkbox: For Access DH users, enable this setting if a single Common lead will be used for the test rather than the Z2 head. This setting is rarely used.
Import Options – Top Side Slot drop-down: Select the Prober slot (Top, Middle, Bottom or Base) to be used when the Top side Sequence is created.
Import Options – Bottom Side Slot drop-down: Select the Prober slot (Top, Middle, Bottom or Base) to be used when the Bottom side Sequence is created.
Import Options – Exclude Components checkbox: Enable if you want to exclude Components from being added to the Sequence. This feature can be very useful if you want to exclude components that are not accessible such as BGA devices. Mark the checkbox and click the **Select button** to find and select the components to exclude.



Select the component or components (use the **Shift** or **Ctrl** keys to select multiple components) and click the **Add Selected** button to add the components to the Select Items list. Use the **Delete Selected** button to remove components from the Select Items list .As alternative, you select a text using the **Load File** button. The file must be ASCII text with one reference designator per line. Click **Close** to return to the Importer window.

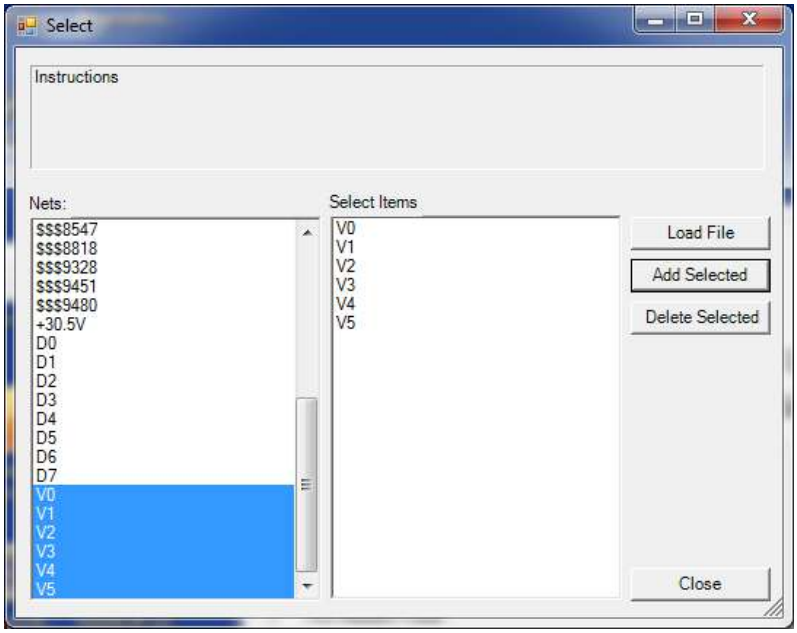
Component Options – Vias as Components checkbox: Enable this setting if you want vias on the board to be added as Components to the Component based Sequences. The vias will be added as "VIA (plus its number)" to the Sequence. Use this setting only if the vias on the board are unmasked and accessible.
Component Options – 2 and 3 Pin Components Pin to Pin checkbox: For Access DH users, enable this setting if you want the Prober to test between each pin combination for 2 and 3 pin devices. Example – the Access DH Prober will use the Z1 and Z2 heads to probe across a resistor.
Component Options – 2 and 3 Pin Components Common checkbox: For Access DH users, enable this setting if you want the Prober to place one head on a board Common point and use the other head to probe each pin of the device. The Common point used is selected in the **Net Options – Common Net drop-down**.
Net Options – Pins checkbox: Enable this setting to import component pins as test points for a net based Sequence.
Net Options – Vias checkbox: Enable this setting to import board vias as test points for a net based Sequence. Do not enable this function if the vias are not accessible or masked.
Net Options – All Nodes checkbox: Enable this setting if you want to test all nodes (pins) of the nets in the Sequence. This setting is useful if you need to test for the same

measurement at different locations on a net. This type of test would essentially be the same as a Component based test in that all pins are tested. The **Exclude Components**, **Node Limit** and **Exclude Nets** settings will affect how many nodes will be probed.
Net Options – Common Net drop-down: For Access DH users, use this menu to select a net to be used as Common when executing the test. When the test is run either Z1 or Z2 will be used depending on where the net is probed. In many cases, a GND net is selected.
Net Options – Node Limit scrollbox: Use this setting to limit the number of nodes (pins) imported for each net. This is very useful when you have common nets such as Vcc or GND that can have a very large number of nodes. Limiting the number of nodes will shorten the import time and also reduce the database size. A typical limit setting is 50.
Net Options – Preferred Side drop-down: Select the board side that you want given priority when a net is tested. In general use, the side with the best visual access to pins and vias is selected. The option will be greyed out if only one side is selected in the **Import Options** above.
Net Options – Order drop-down: Sets the priority used when selecting where a net is probed. Pins=>Vias will give priority to pins over vias. Vias=>Pins will give vias priority over pins. The selection will be controlled by the **Pins checkbox** and the **Vias checkbox** above. Select the priority setting you feel will provide the best access.
Net Options – Order Nodes by Ref. Des. checkbox: Use this setting order the nodes (pins) based on their reference designator (Ref. Des.). Click the **Select** button to open the select window where you can set the order.



Select the reference designator prefix on the left side (use the **Shift** or **Ctrl** keys to select multiple prefixes) and click the **Add Selected** button to add to the Select Items list. Use the **Delete Selected** button to remove reference designator prefixes from the Select Items list. As alternative, you select a text file using the **Load File** button. The file must be ASCII text with one reference designator prefix per line. Click **Close** to return to the Importer window.

Net Options – Include only Nets of Components checkbox: Use this setting import only the nets that go to selected components. This feature is useful if you want to probe nets that only connect to a BGA device. Click the **Select** button to open the select window.



Placeholder image

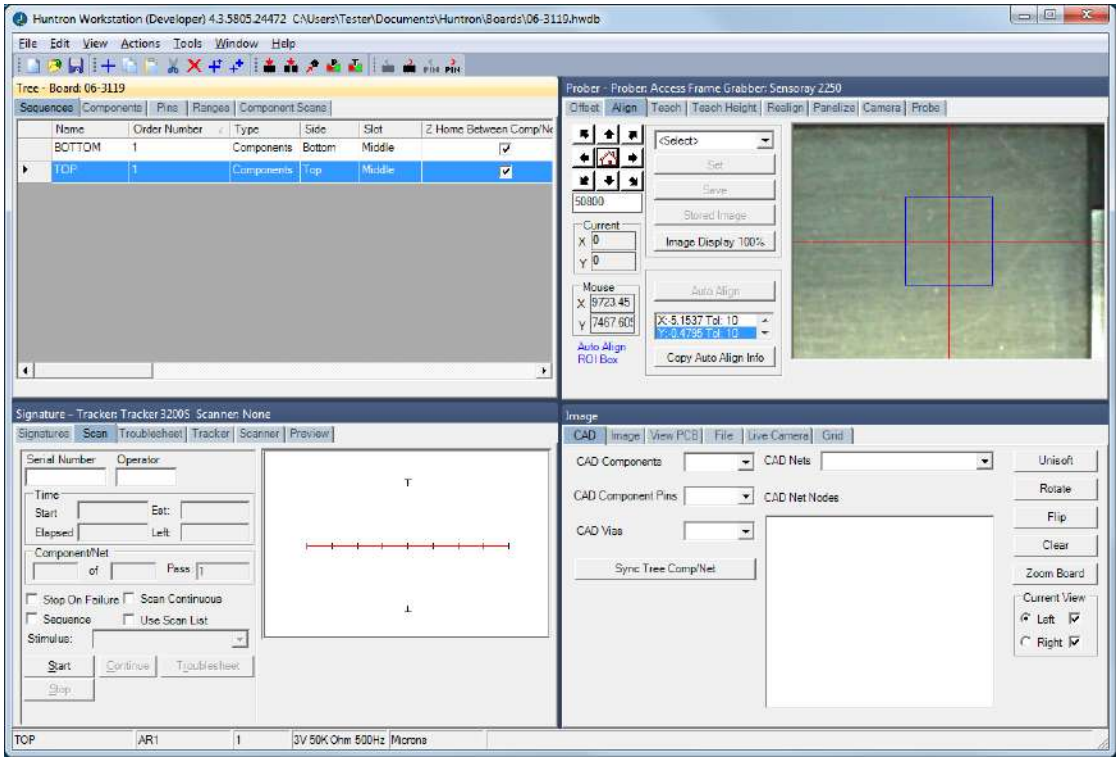
Select the net name on the left side (use the **Shift** or **Ctrl** keys to select multiple prefixes) and click the **Add Selected** button to add to the Select Items list. Use the **Delete Selected** button to remove nets from the Select Items list. As alternative, you load a text using the **Load File** button. The file must be ASCII text with one net name per line. Click **Close** to return to the Importer window.

Click the **Process** button to import the selected CAD file based on the setting you configured in the Import window. The Results area will show the Sequences as they are created into the Workstation database.

When the import is complete, click **Close** to return to the Huntron Workstation software.

Board Set Up

Once the CAD Import 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.



Align and Realign

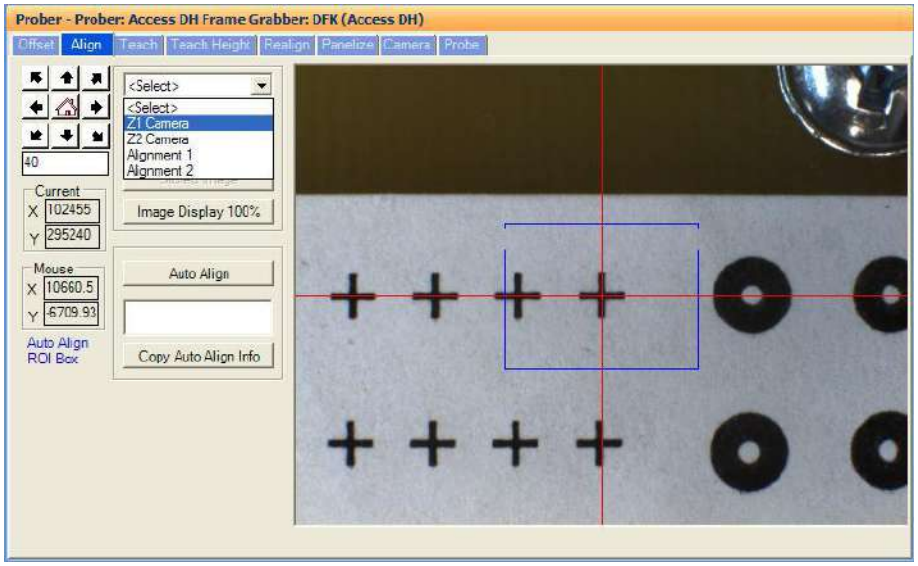
Select the Sequence you wish to scan in the **Sequence** tab. Since Realign permanently modifies the XY data, this is a good time to select **Save Copy** from the File menu and save a copy of the current database under a new name. This will allow you to go back to the original file if problems during Realign are encountered.

Select the **Align** tab in the Prober pane and you will be prompted to mount the board.

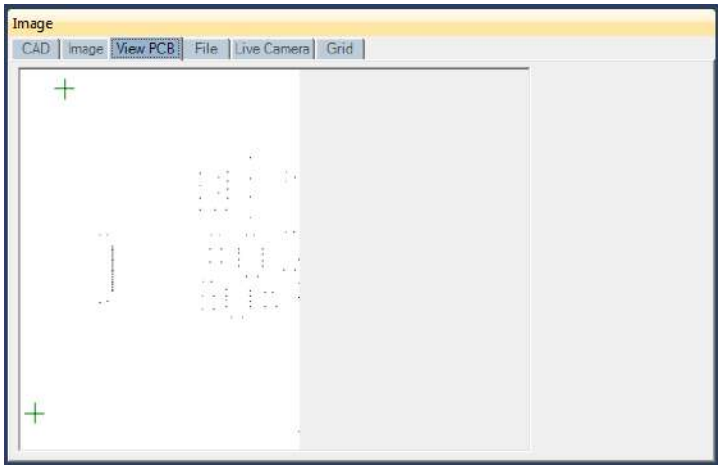


Mount the board to test in the Prober making sure the rotational orientation is correct and the board is positioned at least 0.7 inches (18mm) from the Prober walls. Click **OK** to continue.

Additional steps for Access DH users: Mount the PCB in the center of the probing area so both heads have the best possible access to the PCB.
If the **Align Z Heads** function is enabled (**Tools/Options/Prober tab**) you will need to perform Camera Alignment. This is displayed as **Z1 Camera** and **Z2 Camera** in the Align tab drop menu.



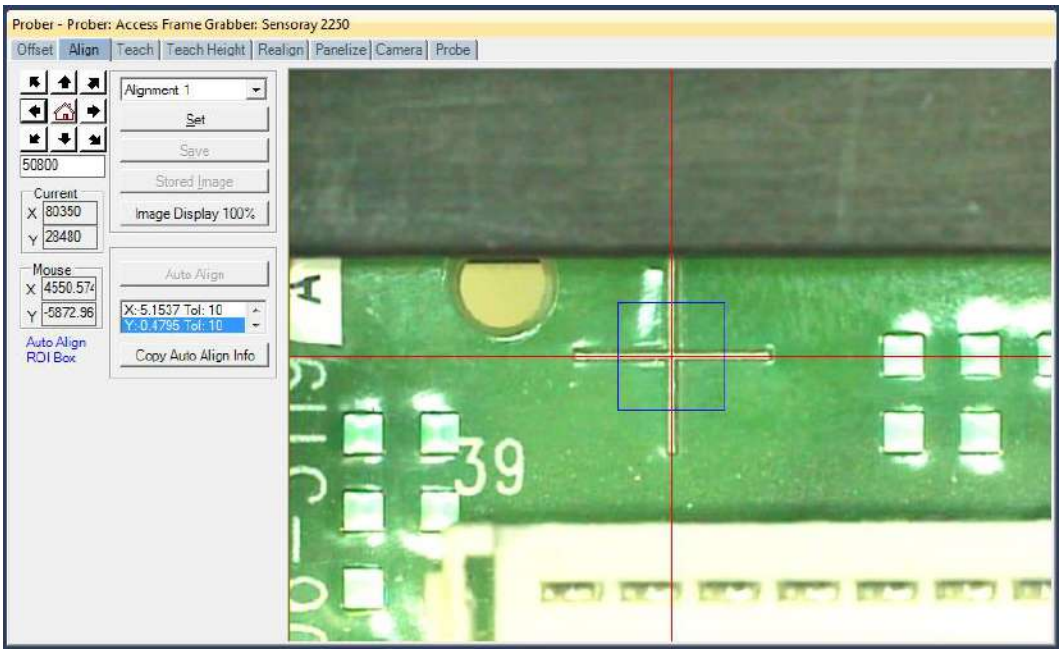
When **Z1 Camera** or **Z2 Camera** is selected, use the Board Reference point you selected earlier in the Unisoft software. Put each camera precisely on this point and click **Set**.
If you are uncertain of where the imported alignment points are located, click on **View PCB** tab in the Image pane to display the board outline and the two alignment points.



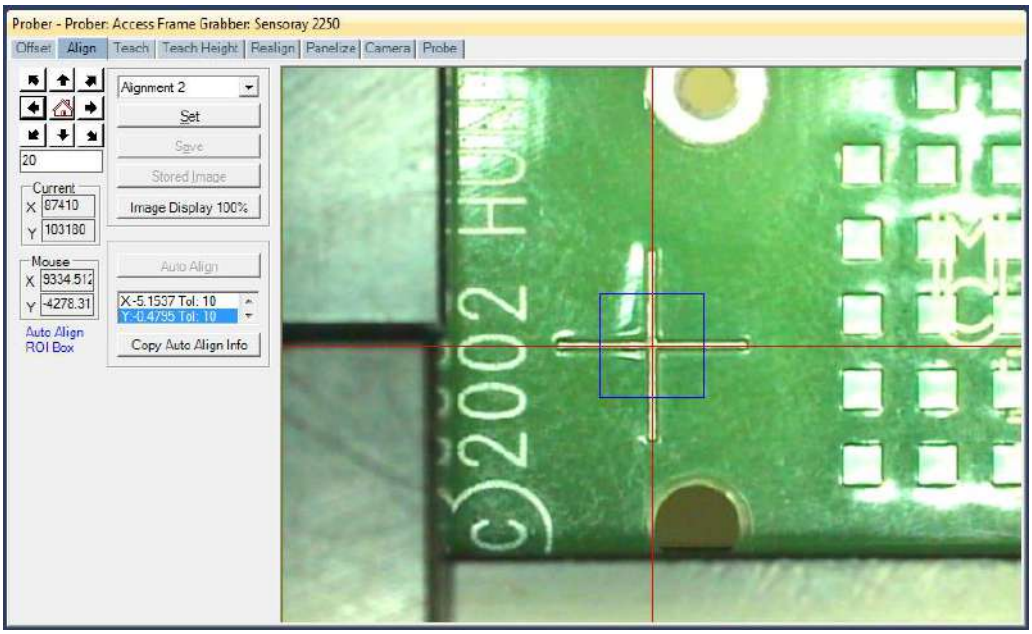
End of Additional steps for Access DH users

Select **Alignment 1** from the drop menu. Since the PCB position within the Prober area is not part of the CAD import, 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.



Select **Alignment 2** from the drop field and 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.

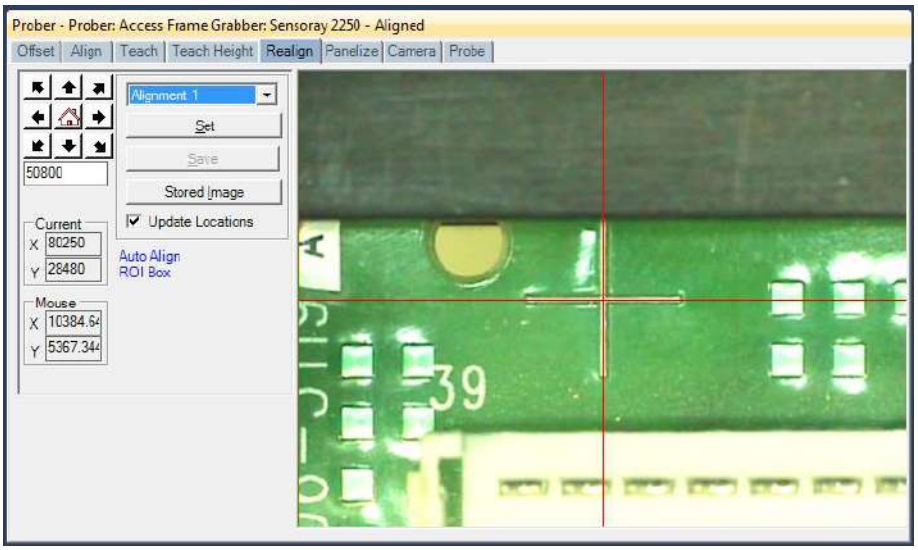


Currently, the alignment point positions are only stored temporarily. At this point, it is **very important** to select the **Teach** tab and verify the pin positions of a few random components. Access DH users should make sure to check component positions for both heads. Checking the component pin positions will ensure that the CAD data imported correctly. If the pin 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. In some cases you will notice that the component positions will be slightly off even after a precise alignment. You can correct the positions by noting the distance and direction the camera needs to be adjusted to be at the pad center. Using this information, go back to the **Align** tab and adjust the alignment points the same distance and direction. Go to **Teach** mode and check the component positions again. Adjust the alignment points again if needed. Once the component positions are correct, proceed to the **Realign** process.

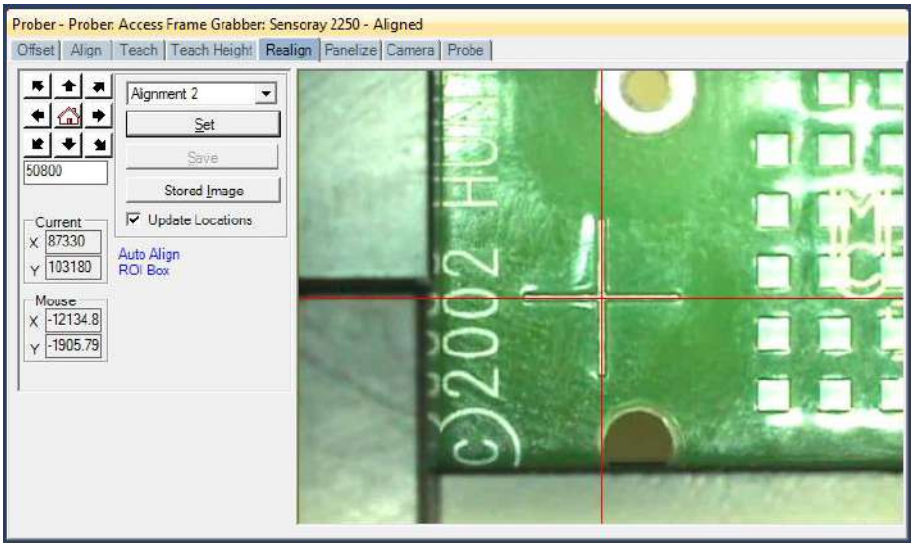
To make the alignment positions permanent, select the **Realign** tab in the Prober pane. Before proceeding with the Realign, it is **very important** that you precisely performed alignment in the **Align** tab first. Failure to do so can cause the XY location data to be modified improperly.

Additional steps for Access DH users: If the **Align Z Heads** function is enabled (Tools/Options/Prober tab) you will need to perform a Camera Re-alignment. This is displayed as **Z1 Camera** and **Z2 Camera** in the **Realign** tab drop menu. When **Z1 Camera** or **Z2 Camera** is selected, use the same point you selected in the **Align** step earlier. Put each camera precisely on this point and click **Set** (do not click Save).
End of Additional steps for Access DH users

In the **Realign** tab, select **Alignment 1** from the drop menu. The camera will move to the first alignment point. You can use this point or select a new point. Click **Set** to save the position selected (do not click Save).



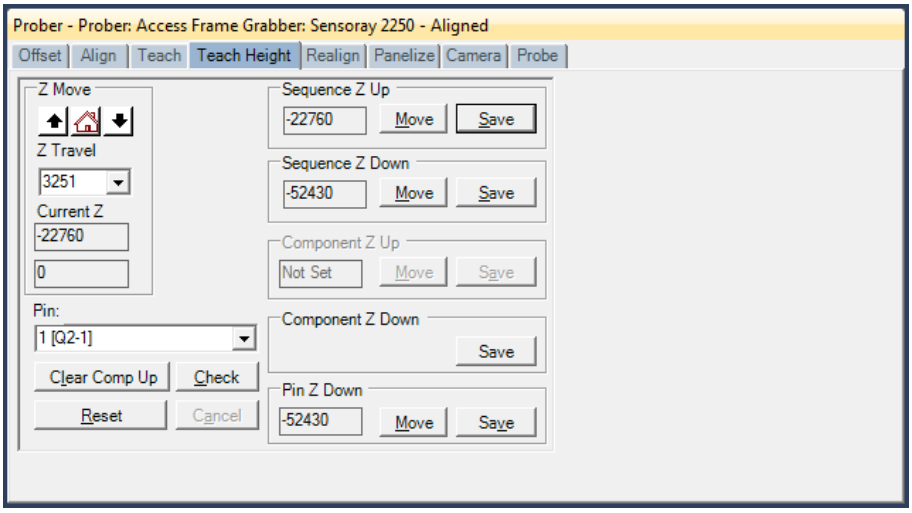
Select **Alignment 2** from the drop menu. The camera will move to Alignment 2. You can use this point or select a new point. Click the **Set** button to save the position. Click the **Save** button to make these alignment points permanent.



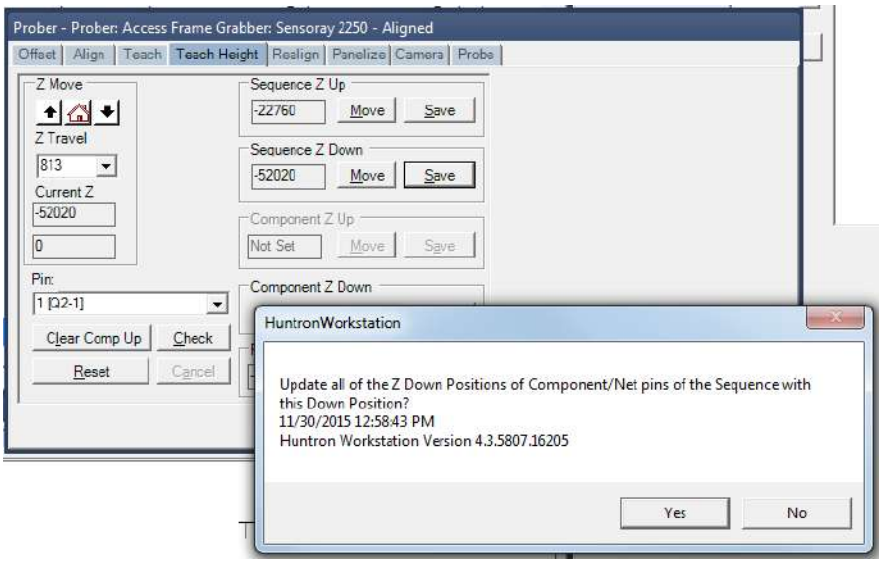
You will be prompted to make these changes. Click **OK** to continue.
Once the Realign is complete, the Z axis positions need to be set.

Teach Height

To set the up and down probe movement positions for the prober, click the **Teach Height** tab in the Prober pane. Select a pin from the **Pin** drop field.

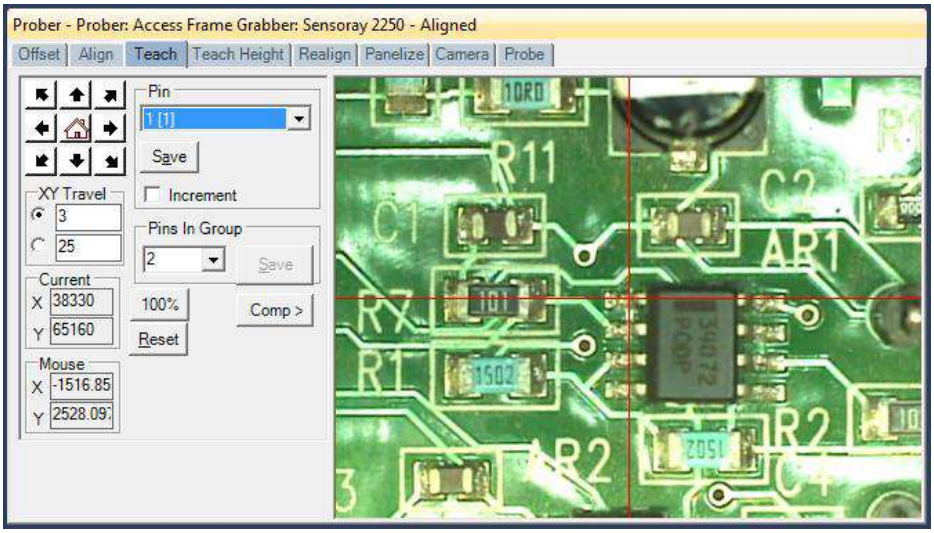


Use the **Z Move** buttons to lower the probe tip to a position above the board where it will clear any tall components or objects on the board. Use the Z Travel field to control the distance the probe moves with each click. Click the **Save** button in the **Sequence Z Up** section to store this Z position.
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 the **Save** button in the **Sequence Z Down** section 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

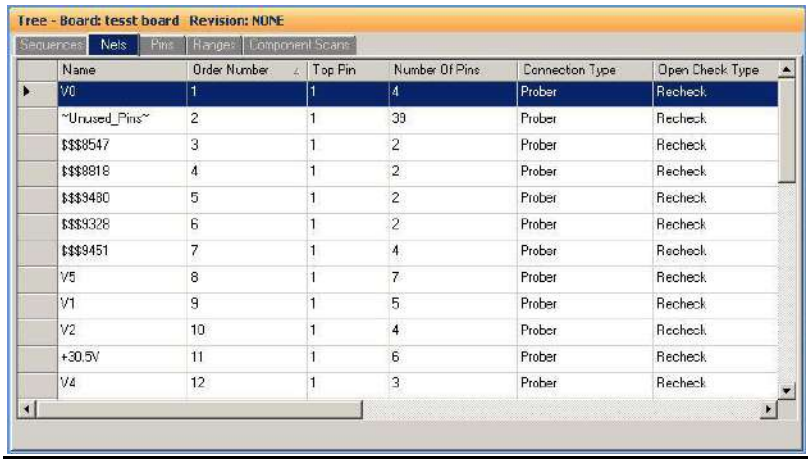
To check that all changes have worked properly up to this point, select the Teach tab in the Prober pane.



Select a pin from the **Pin** drop field and the camera will move to that pin of the selected component or net. 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. Access DH users should check pin positions for the **Z1** and **Z2** probes.

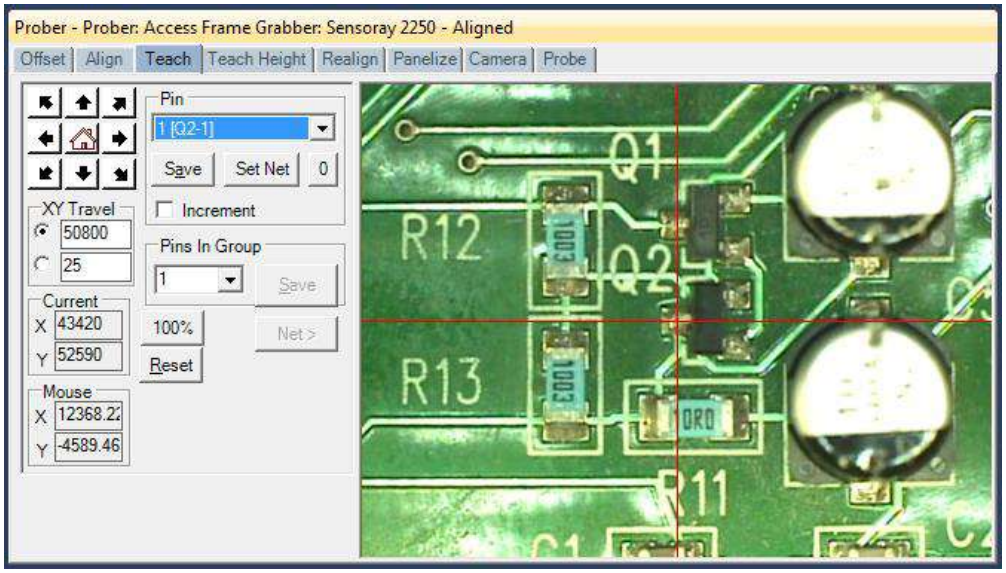
Working with a Net based Test

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.**



Name	Order Number	Top Pin	Number Of Pins	Connection Type	Open Check Type
V0	1	1	4	Prober	Recheck
~Unused_Pins~	2	1	39	Prober	Recheck
$$$8547	3	1	2	Prober	Recheck
$$$8818	4	1	2	Prober	Recheck
$$$9480	5	1	2	Prober	Recheck
$$$9328	6	1	2	Prober	Recheck
$$$9451	7	1	4	Prober	Recheck
V5	8	1	7	Prober	Recheck
V1	9	1	5	Prober	Recheck
V2	10	1	4	Prober	Recheck
+30.5V	11	1	6	Prober	Recheck
V4	12	1	3	Prober	Recheck

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.

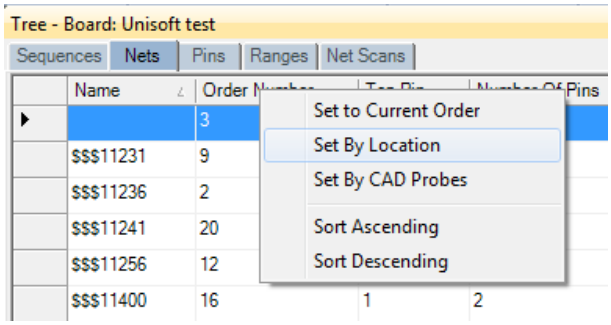


Each pin of the net for the selected board side (Top or Bottom) will be listed in the Pin drop menu. The Prober camera will move to the selected pin. You can modify the position by moving the camera crosshairs to the desired position and clicking the **Save** button. If you wish to set a different pin 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

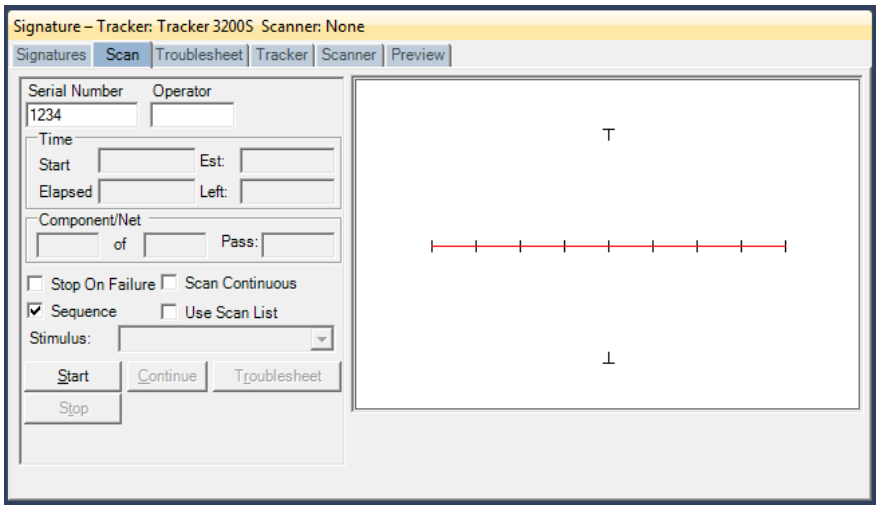
The best method to optimize the movement of the probe is to use the Set By Location feature. The will reorganize the component order based on their position.
To use Set By Location, right-click the **Order Number** column header in the Tree pane/Component tab and select **Set By Location** as shown below.



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. If you wish to scan the whole Sequence click the **Sequence checkbox**. Click the **Start** button to execute the scanning of the selected Component or Sequence. 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.

Appendix 1: How to Modify the Unisoft XML Output File

There may be situations where you may need to modify the XML file output from Unisoft PronoVIEW Markup to provide a more refined list of components or nets that are used when creating a test in Huntron Workstation. This information describes the basic layout of the XML file and how to modify it.

XML File Basic Layout

If you open your XML file in a text editor, web browser or XML editor you will see a fairly simple structure. Each section of information will be enclosed in a "tag" made of a name with brackets < and >. The opening tag will have the structure such as <board> and the closing tag will add a / after the first bracket such as </board>. Each section of information in the XML file will have opening and closing tag according to the XML standard.

The structure of the XML output from Unisoft ProntoVIEW:

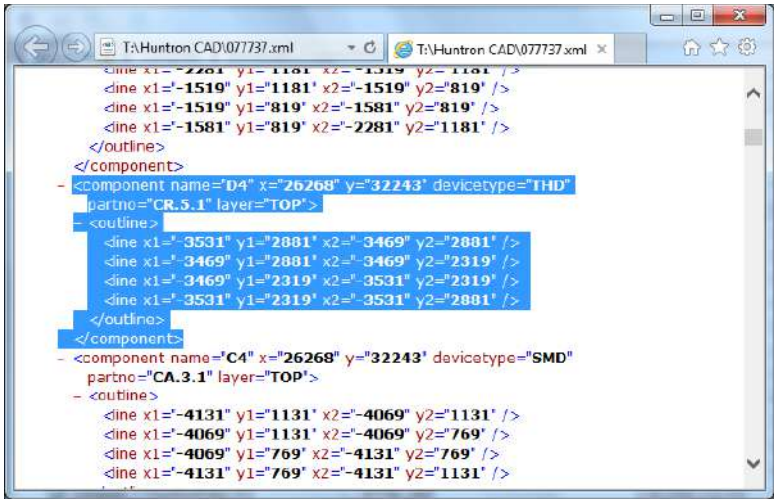
```
<?xml version="1.0"?>
<unisoft version="2.0" units="mils">
   <board name="T:\HUNTRON CAD\077737A.ASC" width="6500" height="4525">
      <marks>
         <mark component="ZP2" pin="1" layer="TOP" type="REF1" x="-6500" y="4000"/>
      </marks>
      <outline>
      <line x1="-6500" y1="4000" x2="-1700" y2="4000"/>
      </outline>
   </board>
<layers>
   <layer name="TOP" top="true" bottom="false" sequence="1"/>
   <layer name="BOTTOM" top="false" bottom="true" sequence="2"/>
  </layers>
<nets>
   <net name="N14439"/>
</nets>
<components>
      <component name="C2" x="262" y="320" devicetype="SMD" partno="C" layer="TOP">
         <outline>
               <line x1="-4131" y1="531" x2="-4069" y2="531"/>
               <line x1="-4069" y1="531" x2="-4069" y2="169"/>
               <line x1="-4069" y1="169" x2="-4131" y2="169"/>
               <line x1="-4131" y1="169" x2="-4131" y2="531"/>
         </outline>
      </component>
</components>
<pins>
   <pin name="1" x="-3600" y="2850" component="D3" net="+VPA" pin1="true"/>
</pins>
   <traces>
      <trace x1="-1700" y1="550" x2="-1900" y2="350" net="N181381" layer="TOP"/>
   </traces>
</unisoft>
```

Editing the XML File

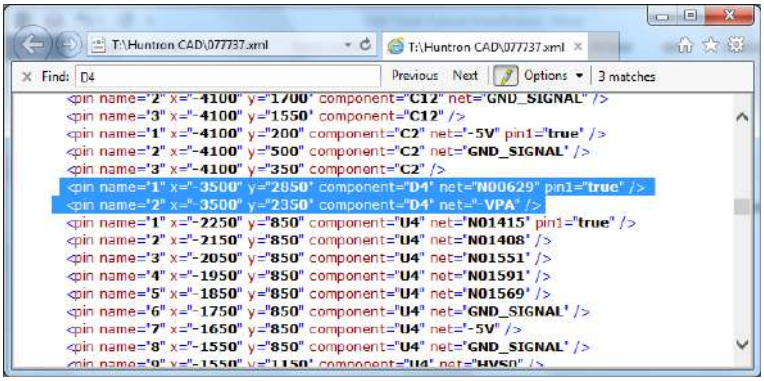
Generally, editing the XML file will involve removing information that you do not want to import into Huntron Workstation. A common situation is to remove BGA devices since they cannot be probed directly. Removing them will help limit Workstation to selecting points that can be probed.

To remove a component:

- Select and locate the component you want delete by reference designator (i.e. U1) in the <Components> section of the XML file.
- Delete all of the information from <component> to </component> for the selected component (see example in the image below deleting D4)



- Locate the selected component pins in the <Pins> section of the XML file
- Delete all of the pins that are part of the selected component; delete all of the information within the <pin (information) /> tag (see example in the image below deleting D4 pins).



It is important that the <Component> and <pin/> information be deleted to avoid error when importing the file. Be sure to keep the tag structure intact by keeping the <, > and /> brackets in their proper place and that each section has an opening and closing tag.