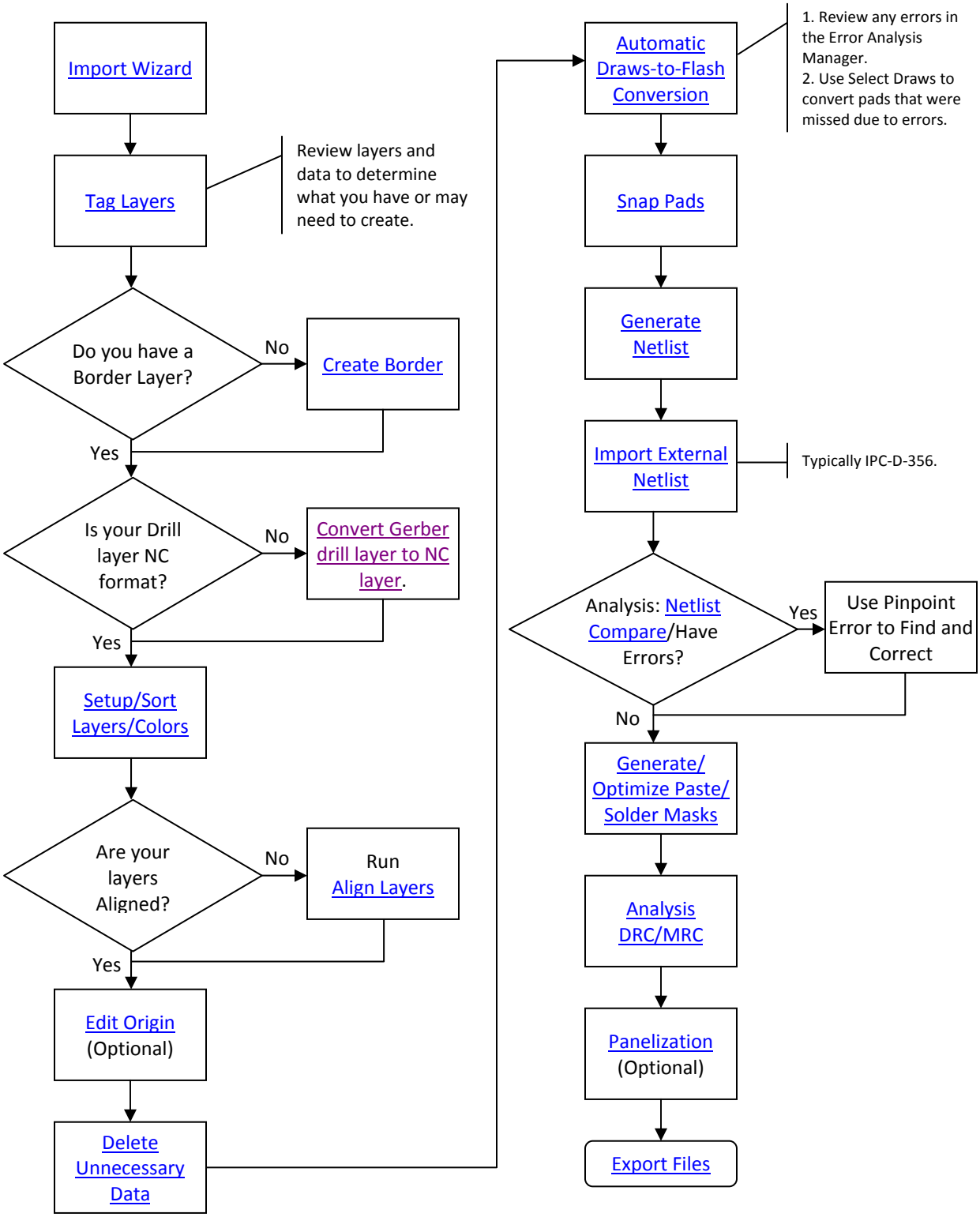
Training Guide

Contents

General Process Flow Chart



VisualCAM Basics

In this lesson we will describe the location and use of the primary toolbars and windows as well as general navigation for the VisualCAM application window.

Purpose

The purpose of this lesson is to familiarize you with the User Interface of VisualCAM, including the location and use of the various windows and toolbars.

Definitions:

- **VC**: VisualCAM
- **UI**: User Interface
- **VCAM**: VisualCAM Database file or job
- **Composite:** A composite is a single layer (and hence, a single Gerber file), which is made up of a set of positive (dark) and negative (clear) layers. They are used to route traces on power planes.

Lesson Objectives

In this lesson you will do the following:

- Start up VisualCAM using 4 methods.
- Learn to navigate the workspace.
- Identify the location and use of toolbars.
- Identify the location and use of the Navigator window.
- Identify the location and use of the Colors window.
- Identify the location and use of the Selection Filter window.
- Identify the location and use of the Properties window.
- Identify the location and use of the online Help.

Estimated Completion Time

Approximately 60 minutes.

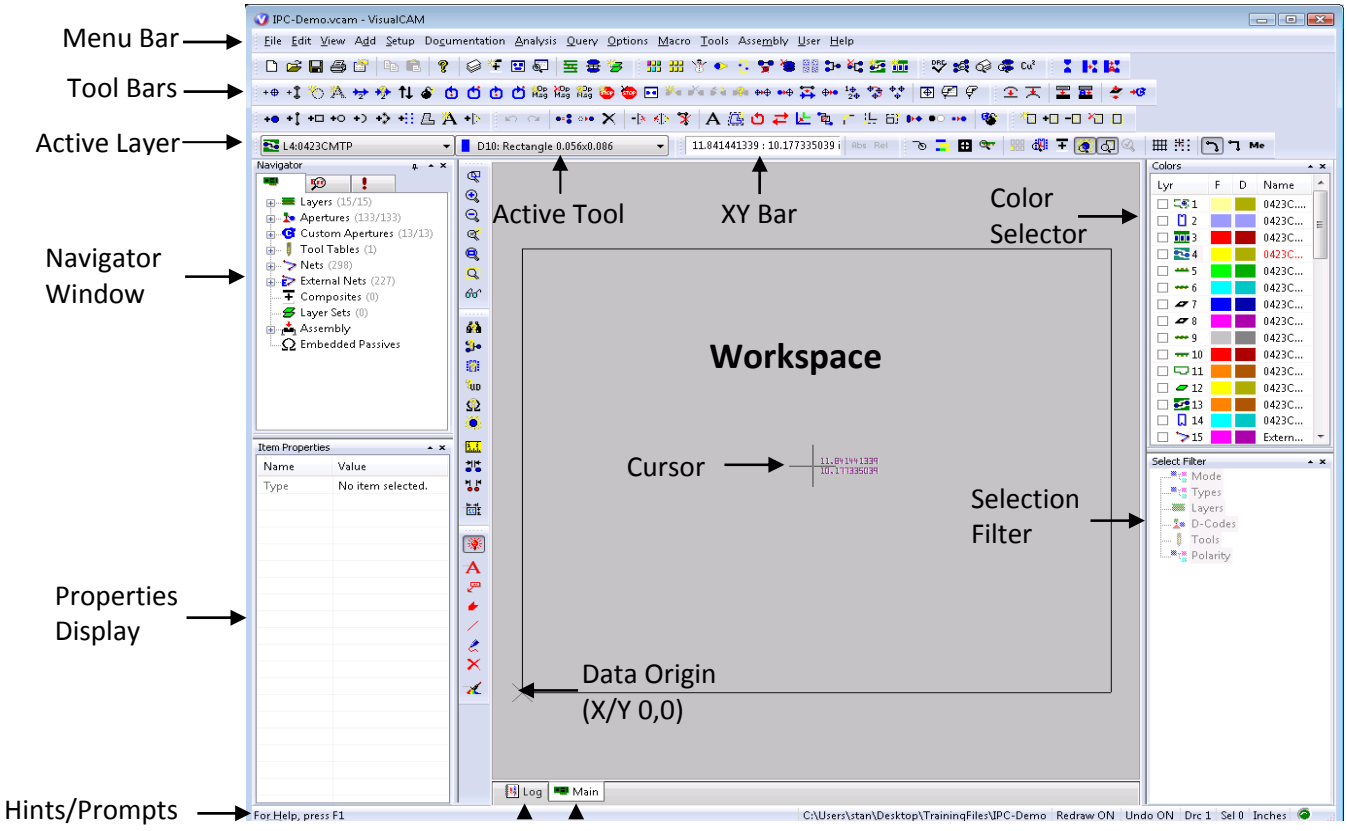
Navigating VisualCAM

The first thing you want to know about any software tool is how to navigate your way around. The graphic below gives you a topographical view of the application window. You will learn how to navigate around this workspace area and within the various windows.

NOTE: To start, copy the "**TrainingFiles**" folder from the course data you were given with this manual and paste it onto your computer desktop. Open the folder and double-click the file "**IPC-Demo.vcam**" to open the training file in the VisualCAM application.

VisualCAM User Interface (UI) Layout

The main VisualCAM workspace, or "desktop," is shown below. All toolbars and control bars may be moved to any location on the desktop by clicking on the bar and dragging it to a new location. They may be docked to an edge of the VisualCAM workspace or floated in a smaller window. Most of the items in the window can also be resized or closed so that you can customize the space to suit your needs.



When started if VisualCAM defaults to Log tab view, click on the Main tab to view the workspace.

Navigating the Workspace

Here we will discuss how to maneuver in the workspace area of the application window and the Left Mouse Button (LMB) and Right Mouse Button (RMB) behaviors.

Left Mouse Button Behavior

The default behavior for the LMB, when no other command is being executed, is to draw a view-window. The first click starts the view-window, and then you drag your mouse to the size of the area you wish to zoom into. The second click executes the zoom command for the area you specified.

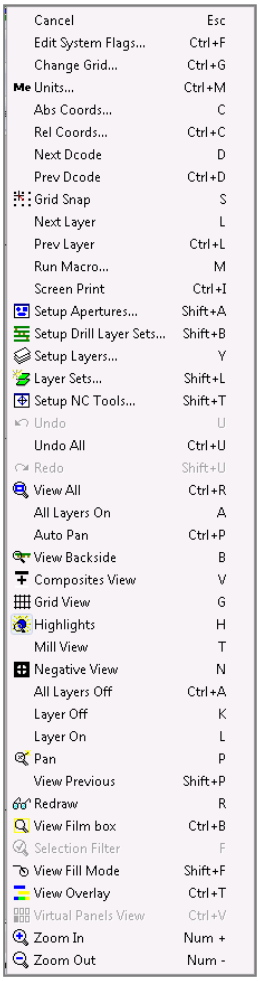
When executing commands from the menus or toolbars your LMB behavior will often be modified for the duration of the command. The exact behavior is determined by the command you selected and will be specified in the Hints/Prompts area in the lower-left corner of the application window. For most commands, your LMB is used to select the object(s) to be affected by the command.

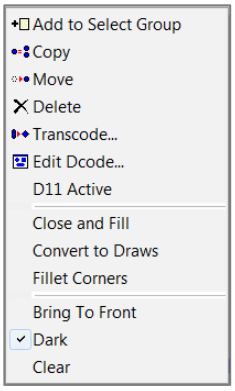
Right Mouse Button Behavior

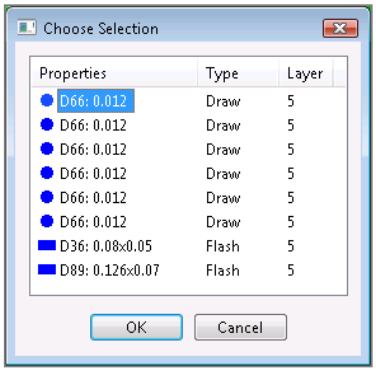
There are two default behaviors for the RMB depending on where your mouse pointer is when you click it.

1. If your mouse pointer is hovering over a blank area in the workspace, when you right-click a pop-up menu (see left image) will appear listing several frequently used commands, such as zooming options and layer surfing.
2. If your mouse pointer is hovering over an object in the workspace, that object is selected and a much smaller pop-up menu (see top-right image) with frequently used edit commands appears.

   If more than one object is located near your mouse pointer when you right-click, such as objects from other layers directly under your mouse pointer, then you will first be presented with a Choose Selection dialog box (see bottom-right image) prompting you to select the specific object to be edited or queried.







Hot Keys & Function Keys

There are many Hot Keys available to VisualCAM users that can make the overall processing of your data that much more efficient. You will find the Hot Keys listed next to each command they execute in the various menus. A complete list of all available Hot Keys is available in the online Help.

VisualCAM comes pre-configured with the following function key assignments:

Key	Assignment	Description
F1	Help	VisualCAM help topics and tutorials.
F2	View > Film Box	Zoom out from object in workspace window.
F3	View > Previous	Return to previous view of object in window.
F4	View > All	Fit object to window and center it.
F5	Setup > Layers	Open Layer Setup dialog box.
F6	Setup > Apertures	Open Aperture Setup dialog box.
F7	Documentation > Reports > Apertures	Open Aperture Report dialog box.
F8	Query > Highlight	Open Select Filter (Highlight) window.
F9	Query > Item	Open Select Filter (Query Item) window.
F10	Menu	Reserved by Windows application.
F11	Edit > Select > Add To	Open Select Filter (Group Add) window.
F12	Edit > Select > Remove From	Open Select Filter (Group Remove) window.

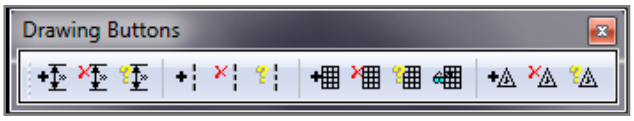
NOTE: The function keys (except for F1 and F10) and the mouse buttons behavior can be modified by going to the **Options > Configure** command and selecting **Function Key/Mouse** tab at the top of the dialog.

Toolbar Button Reference

Each button within the toolbar represents a shortcut to a menu command. When you click on a toolbar button the command associated with that button is invoked. To control which toolbars appear in the window use the **View > Toolbars** command and to change which button appears in each toolbar use the **Options > Customize Toolbar** command.

REMINDER: All toolbars and control bars may be moved to any location on the desktop by clicking on the bar and dragging it to a new location. They may be docked to an edge of the VisualCAM workspace or floated in a smaller window.

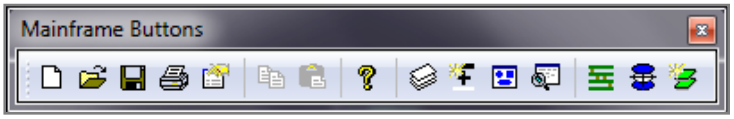
Drawing Buttons are used for working with drawing layers.



Drill Buttons are used for working with NC data.

Edit Buttons are used for working with graphics.

Mainframe Buttons are used for working with files and setting up layer and aperture information.



Parts Buttons execute the utilities used in the reverse engineering process.

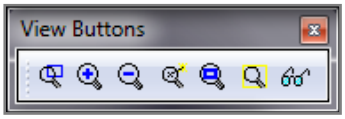
Query Buttons give you quantitative information about specific database items.

Redline Buttons are used for adding comments and other information which are stored separately from the layer design information.

Settings Buttons allow you to control the current state of many display properties and other commonly used program settings.

Tool Buttons execute CAM and Analysis utilities.

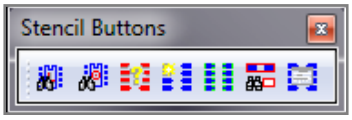
View Buttons manipulate your view of the data in the workspace.



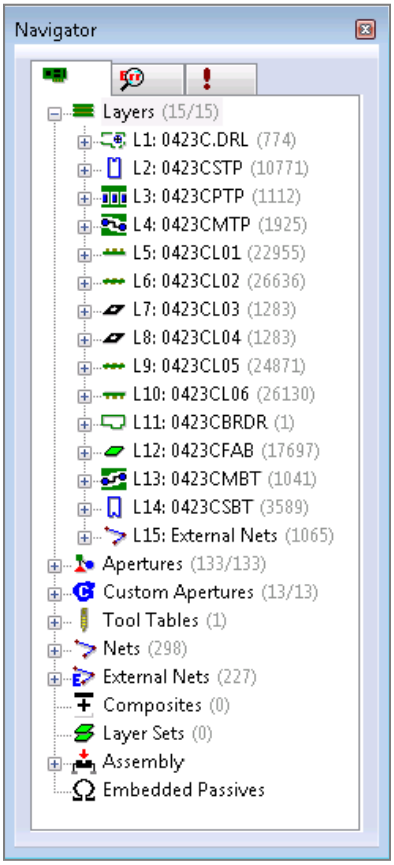
Analysis Buttons are used to setup DRC/MRC, netlist compare, layer compare, find duplicate data, and to calculate the amount of copper used on a layer.

Add Buttons allow you to enter various types of new database items.

Stencil Buttons are used to identify and create special stencil shapes**.**



The Navigator



The Navigator provides a quick-reference list of the database elements, and another way of accessing VisualCAM command functions. Most of the Navigator functions, such as changing layer names, adding apertures, and printing analysis reports are controlled with a right-click shortcut menu.

You can position the Navigator pane anywhere in the main VisualCAM window by clicking on the top of the Navigator bar and dragging it to another location. You can also toggle the view of the Navigator on or off by selecting the View > Navigator command.

The Data Tab

The Data tab provides information about the layers, apertures, NC tools, nets, composites, and layer sets. Functions used for the reverse engineering of parts for assembly and embedded passive data are also available. The information is displayed in a tree format, displaying database elements in an expandable/collapsible hierarchy. To expand an area of the tree, click on the plus box (+ icon) next to the desired database element. The "branches" of the information hierarchy are shown and the plus box becomes a minus box (- icon). To compress or hide the information, click on the minus box. The information hierarchy for that database element is hidden. If there is no Plus symbol next to a topic in the tree, that type of element does not exist in your design, or has been defined but is not used.

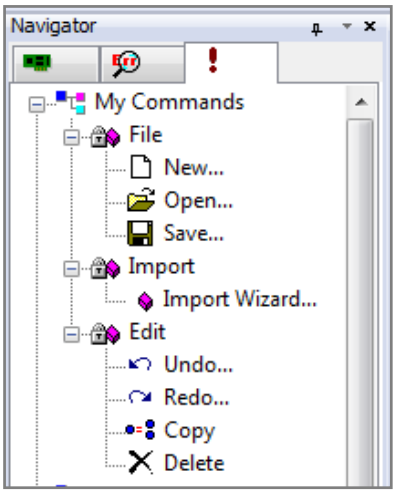
You can edit and delete database elements by right-clicking on the various headings and the branches that are associated with each. This not only provides you with shortcuts to functions available in the main menu, but also some functions unique to the Navigator.

A detailed description for each database element type and how they can be manipulated with the Navigator can be found in the online help. The use of several of these functions will be described in more detail in later chapters.

The Analysis Tab

The Analysis tab provides information about analysis runs and command error lists.

The Commands Tab

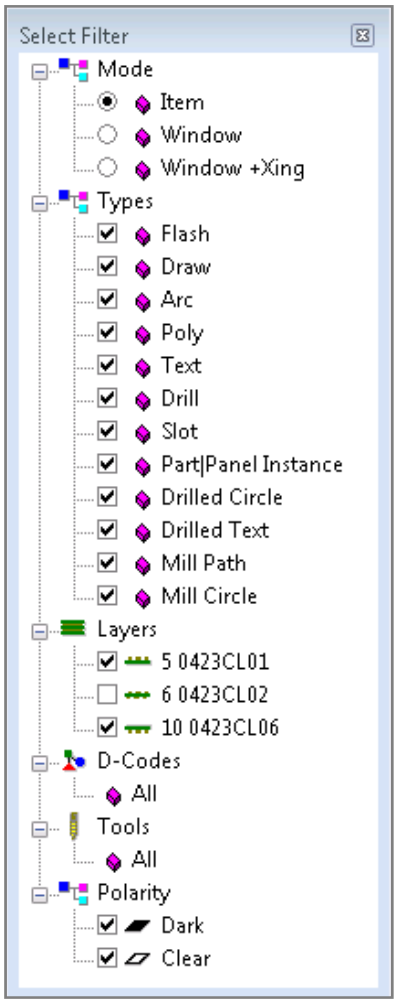


For quick access to commands VisualCAM comes pre-configured with shortcuts to commonly used commands and macros. This can be customized to fit your needs by dragging and dropping the commands from the System commands or Macros section onto the My Commands section.

To use any of these commands just double click on it.

See the Commands Tab section under the Get to Know VisualCAM Navigator activity, page 19 for more information about adding and renaming branches, adding and renaming commands, and reordering commands within the My Commands list.

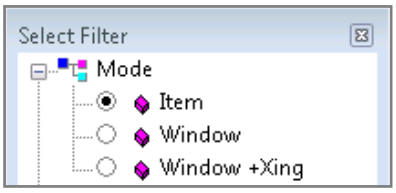
Selection Filter



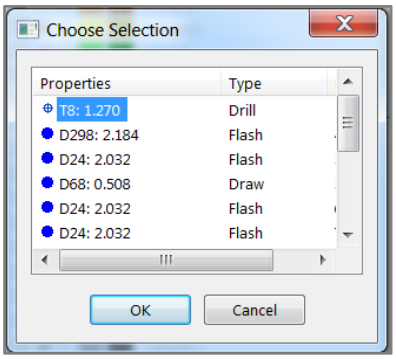
The Selection Filter allows you to define how you want to select items and from what types of items you wish to choose. For example, depending on the command you may choose from single item, window, group, or complete layer selections, as well as restricting your selections to particular layers and/or D-codes, etc.

When you are not in a command that uses the Selection Filter the text in the Selection Filter is gray and you cannot select any options. You also do not have the option of using the **View > Selection Filter** command or the F hotkey when not in a function that uses the Selection Filter.

Mode



These options define how items are selected. The available modes depend upon the function you are in. When you are in **Item**-select mode a bounding box is added to the cursor. Anything that falls within this bounding box is a potential selection. You can increase the size of the box by pressing the **PgUp** hotkey, or decrease the size (for increased accuracy) by pressing the **PgDn** hotkey.

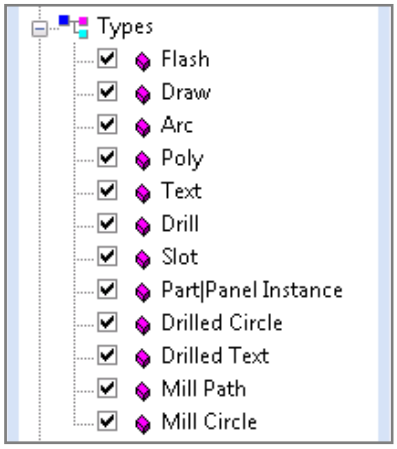


If more than one item exists at that X:Y location the Choose Selection dialog box appears. Select an item and click on the OK button.

In **Window** select mode everything that falls *completely* within your selection window is selected. In **Window +Xing** mode everything inside the selection window, including anything that the window touches, is selected.

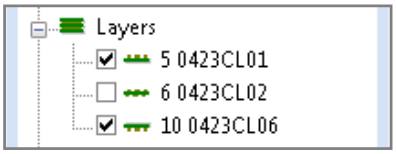
In **Layer** select mode all selected types of data on all selected layers are chosen.

Types



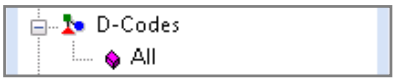
These options allow you to select the kinds of items that will be included. Any type not checked will be excluded from the selection.

Layers



These options allow you to specify items that are created using only a particular D-code.

D-Codes



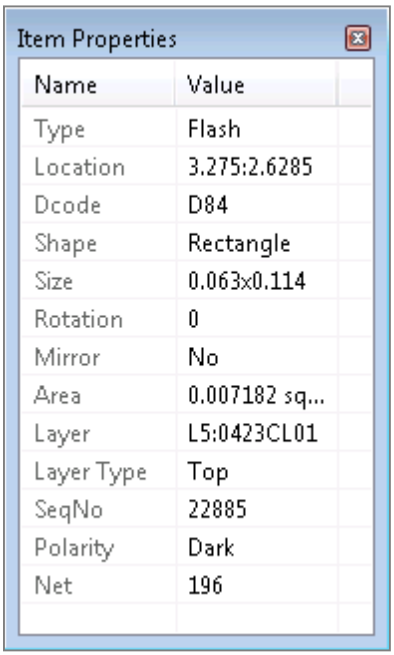
These options allow you to specify drill or mill items that are created using a particular NC tool.

Tools

Polarity

These options allow you to specify only items with a particular polarity.

Item Properties Display



When you query an item in the database its information is shown in the Item Properties display. The type of information displayed depends upon the item being queried.

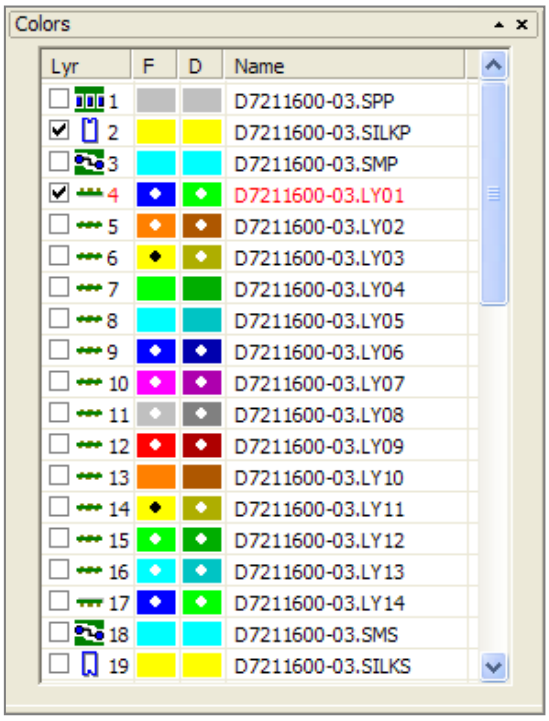
A detailed description for each item type that can be queried can be found in the online Help.

Colors Bar

The Colors Bar is available at all times to change active layer, layer colors, and visibility. Layer names are also presented for your reference, and layer types are represented by the icon next to the layer number.

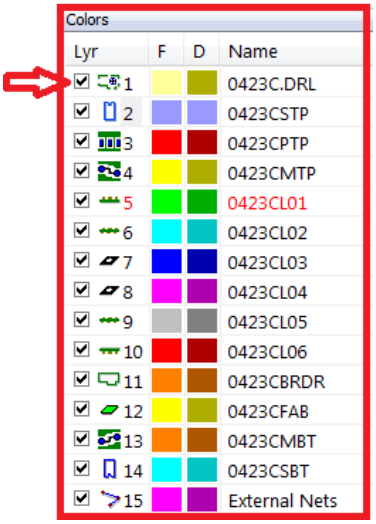
Following is a list of icons and the layer type they represent.

- Top
- Inner
- Bottom
- Plane (Negative)
- Plane (Positive)
- Silk Top
- Silk Bottom
- Mask Top
- Mask Bottom
- Paste Top
- Paste Bottom
- Border
- NC (Drill/Mill)
- Composite
- Drawing
- Insulator
- Etch (Subtractive) Passive
- Screen (Additive) Passive
- External Netlist
- Other
- If a layer contains item-level polarity, the Draw and Flash colors contain a diamond shape, as shown for layer 4 in the above figure.



If you wish to change a layer name or type, use the layer table in **Setup/Layers** and click on the Type icon on the layer you want to change to select it from the menu, or use the **Navigator** and click on the plus box next to the desired layer. In the layer detail list the second item is the layer type. Right-click on the type, and select the new layer type from the menu.

Visibility



Visibility for individual layers, layer sets, and composites is controlled with the Colors Bar by clicking in the check box next to the desired layer number.

A check mark indicates visibility, in which state a layer can be edited; no check mark indicates the layer is not visible and cannot be edited.

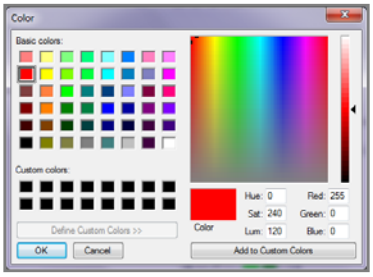
To turn all layers on or off, right-click over the layer in the Colors Bar and select either **All On** or **All Off**. When you select All Off all layers except the active layer are turned off.

Setting the Active Layer

Double-clicking on a layer number makes the layer "active". You can also right-click on the desired layer in the Colors Bar and select Set Active from the shortcut menu. The active layer appears on top of the other layers in the workspace (this does not change the actual layer order). The active layer is also the layer where any newly created objects are placed.

The active layer can also be set using the **Layer Bar** in the Navigator.

Color Selector



You can control the color of Flashes and Draws for each of the layers. To change one of the colors, click on it. The Color Selector appears, and you can select your desired color from the available selection. To change the color choice at a particular location in the Color Selector, right-click on the desired color. This opens the standard Windows color selector that allows you to define a color. To dismiss the Color Selector without choosing a color, press the Esc key.

If you have a specific set of layer colors you would like to use for all your designs, you are provided with a file in the Macros folder called **DEMO.MAC**, which includes a macro called **SetLayerColors**.

1. Select the **Macro > Developer** command from the Menus.
2. Select the **File > Open** command and open the **DEMO.MAC** file located in the WISE Software Solutions folder.
3. **Edit the DEMO.MAC file** to specify your desired colors.
4. **Save** the file and return to the main VisualCAM window.
5. Select the **Macro > Load** command and load the **DEMO.MAC** file.
6. Select the **Macro > Run** command, and run the **SetLayerColors** macro.

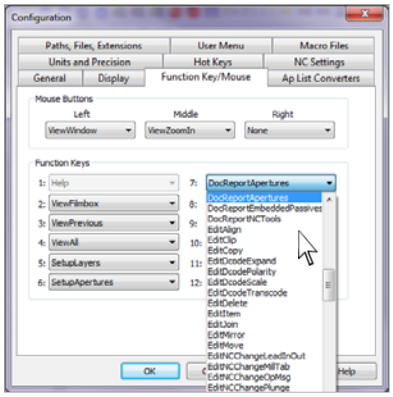
Other Helpful Info

Undoing Edits

The **Undo** command provides a high level of freedom when making database edits. When Undo is enabled you may experiment with edits without fear of data loss. Since undo is available as the **U** hotkey you may undo edits immediately without even having to exit the current command. Undo works for all edits regardless of size, and there is no limit to the number of edits you can undo (*for more information, see Use Common Edit, Query, and View Commands, page 32.*)

NOTE: VisualCAM installs by default with Undo enabled. You can disable Undo on the **Options > Configure** dialog by clicking on the **General** tab and unchecking the box by **Enable Undo/Redo**

Programming the Mouse and Function Keys



VisualCAM's easy-to-use Graphical User Interface (GUI) is further enhanced with the versatility of programmable mouse and function keys. In the **Options > Configure** dialog box click on the **Function Key/Mouse** tab to program both the mouse buttons and function keys F2-F9 and F11-F12 with commands that you frequently use. Click on the key you want to program and you will be presented with a list of commands from which to choose. All macros that you have loaded are available to program a Function Key and they will be presented at the top of the list in the pull down menu (*see Customize the User Menu and Function Keys, page 30 for more information.*)

Interrupting Redraws and Highlights

Any command that redraws the database or highlights a group of items can be sped up by pressing the **Esc** key and canceling the drawing process. This does not affect the operation of the command; only the redraw is affected. Once you are comfortable with the operation of various functions you will find that this ability significantly speeds things up.

Canceling a Command

Any command can be cancelled by pressing the Esc before it is completed. If the command has multiple steps in its process, the Esc key will back up one step at a time until it reaches the beginning of the process and then cancel the command all together when pressed one more time.

Activity: Open the Application File and Setup Toolbars

In this activity you will open VisualCAM using four different methods and modify the toolbar setup.

This activity should take about 20 minutes to complete.

Four Methods to Open the VisualCAM Application

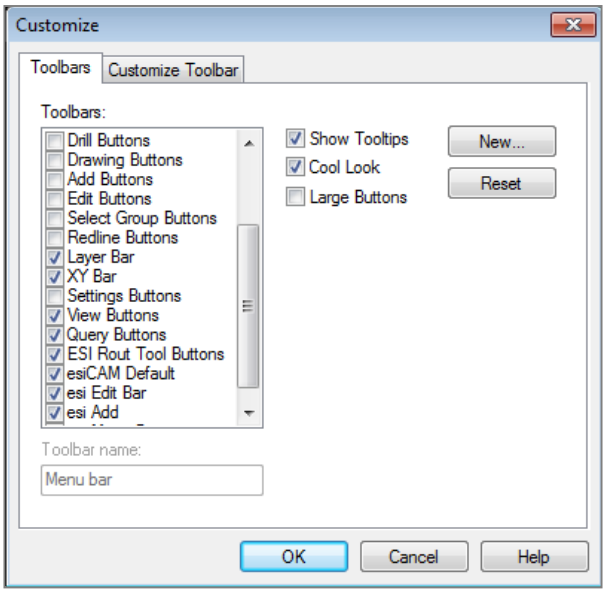
1. On your desktop, **double-click** the VisualCAM icon , OR
2. Click Start, select **ALL Programs** > **WISE Software** > **VisualCAM 16.4** > **VisualCAM,** OR
3. Open by double clicking on your .vcam file and load your design file at the same time. For example, in Windows Explorer browse to the folder, "**C:\Program Data\WISE Software Solutions\VisualCAM 16.4\Samples**". Now double-click on the file "**demo.vcam**", OR
4. Drag and drop the .vcam file onto the VisualCAM icon on the desktop.

Opening a .vcam Design File in VisualCAM

1. On the **File** menu, select **Open** or use the open ( ) Icon on the toolbars, or Press **Ctrl + O**, or drag and drop a .vcam file onto an open VisualCAM application.
2. In the **Open** dialog box browse to the "**TrainingFiles**" folder **C:\Program Data\WISE Software Solutions\VisualCAM 16.4\Samples\Training Files**.
3. Select the file "**IPC-Demo.vcam**" and click **Open**.

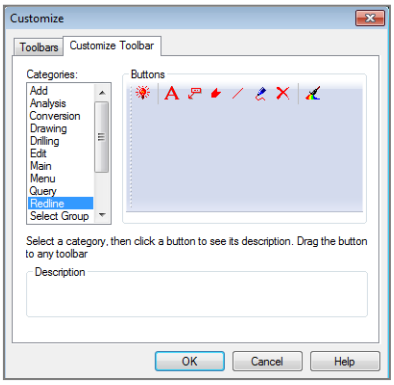
Enable/Disable and Customize Toolbars

1. On the **Options** menu, select **Customize Toolbars…**
2. On the Toolbars tab, the Toolbars with a check next to them are enabled. Uncheck to disable the toolbar. The display immediately updates.

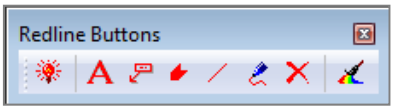


Change the Buttons

Use the **Customize Toolbar tab** to customize the toolbars that are enabled.

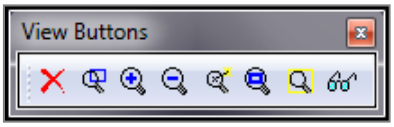


1. On the **Toolbars tab** place a check mark next to Redline Buttons to enable if necessary.
2. On the **Customize Toolbars tab** click on Redline in the Categories list. The available buttons for this toolbar will be shown in the Buttons window.
3. On the Redline toolbar located above or around the workspace window click on the A button and drag it off the tool bar and into the Buttons window of the dialog box. The button is immediately removed from the toolbar.
4. To place the A button back on the toolbar, click on the A button in the Buttons window of the Customize dialog box and drag it back onto the enabled Redline toolbar located above or around the workspace window.



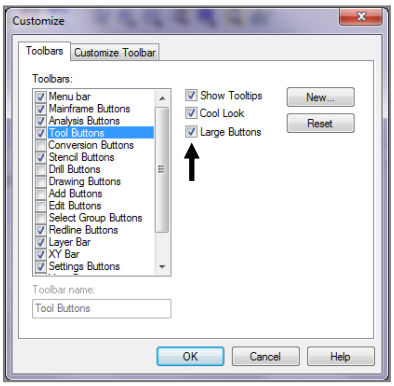
You can also customize enabled toolbars by clicking and dragging buttons within the toolbar itself or from one enabled toolbar to another.

1. With the Customize dialog box open click on the X button on the enabled Redline toolbar and drag it to the right side of the tool bar.
2. On the enabled Redline toolbar click on the X button and drag it to the enabled View Buttons toolbar.



Make Desktop Buttons Appear Larger

Finally, if the icons and buttons on the desktop appear too small, make them appear larger.

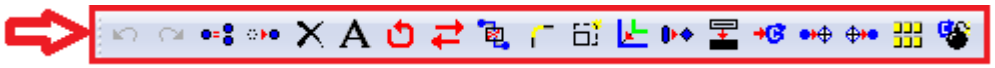


1. With the Customize dialog box open and on the Toolbars tab, place a **check mark** in the **Large Buttons box**.
2. When you are finished, click **OK to exit**, saving the changes; **or Cancel** to exit without changes.

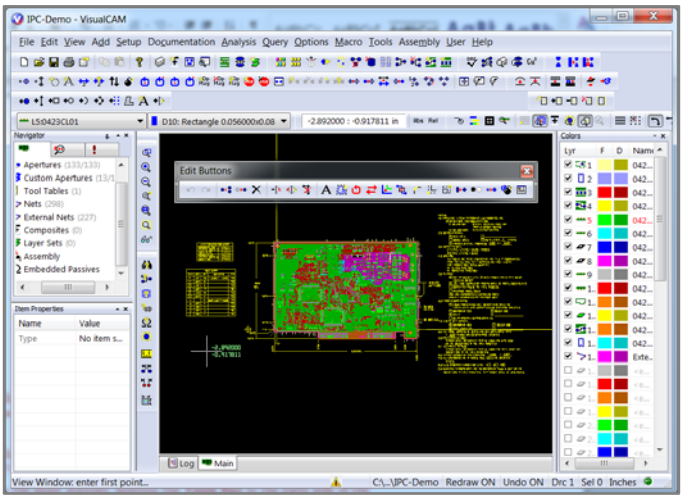
Position and Dock Toolbars

Toolbar positions can be moved to any location within the VisualCAM Main window by clicking on the gripper bar and dragging and dropping to the desired location. If you drag it onto the main window the toolbar will become a floating toolbar. If you drag it to the left or right of the workspace the toolbar will be docked in a vertical position.

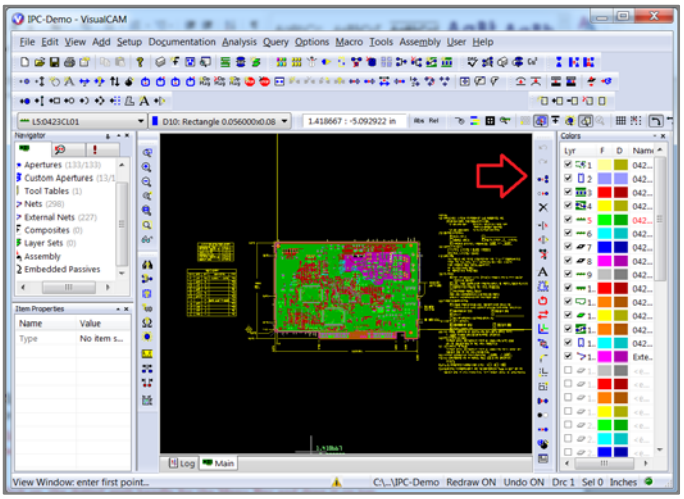
1. For this exercise click on the gripper bar of the Edit Toolbar:



2. Drag it onto the main workspace window. You now have a floating toolbar as shown below.

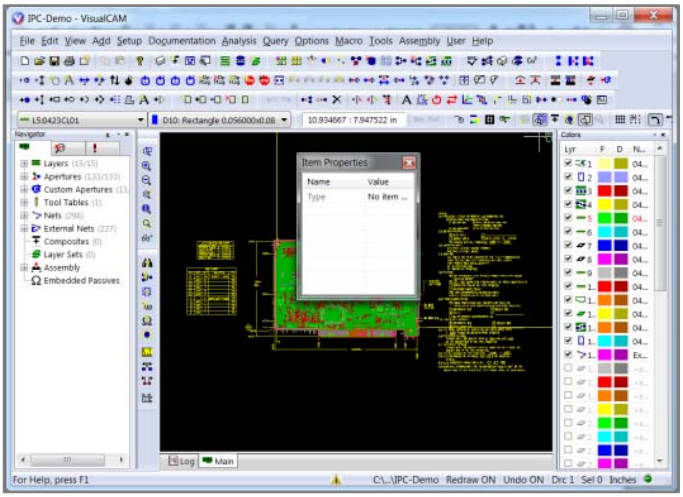


3. Click on the gripper bar of the Edit Toolbar again and drag and drop it on the right side of the screen next to the Colors bar. You now have the toolbar docked on the right side.



You may also resize and position the Navigator, Item Properties, Colors Bar and the Select Filter in the same manner.

4. Click on the Items Properties bar and drag it onto the main workspace to create a floating window.



Once you reposition any of the Toolbars, Navigator, Item Properties, Colors bar or Select Filter, VisualCAM will remember these settings and the next time you open the application these settings will be the same.

Activity: Get to Know VisualCAM Navigator

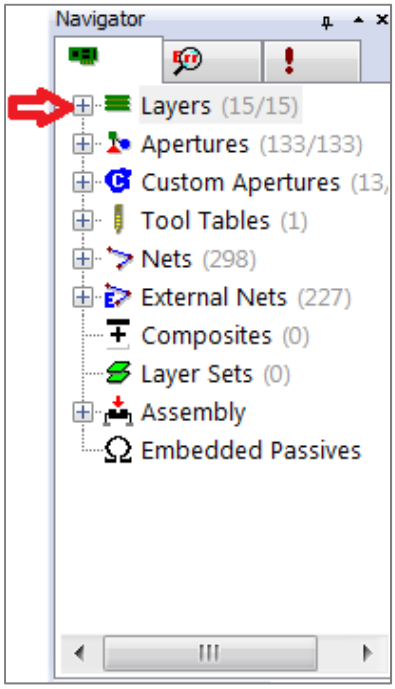
The Navigator provides a tree-type menu for a quick-reference list of the database elements, and another way of accessing VisualCAM command functions.

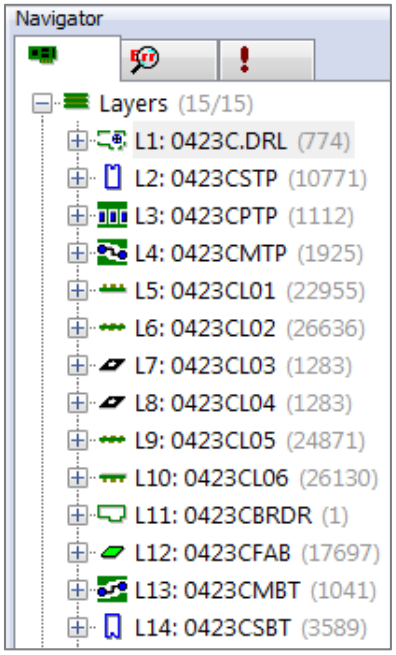
This activity should take about 15 min to complete.

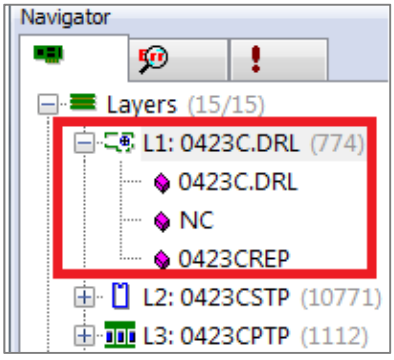
DATA TAB

Layers: displays a list of all layers in the design with the corresponding layer number and name.

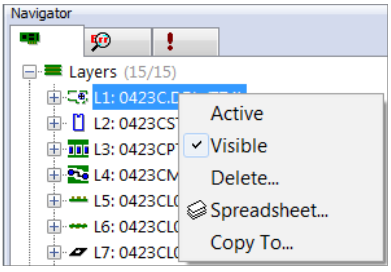
1. In the Navigator click on the plus icon next to Layers to expand the list of layers in the database.
2. To view information about Layer 1, click on the plus symbol next to L1. The layer name and type is shown. The layer type is also indicated by an icon next to the layer number.





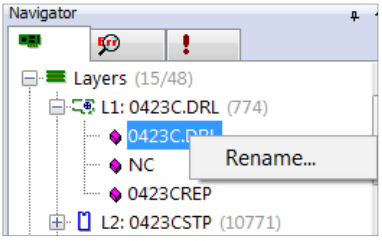


3. All the layer information can be modified using the Navigator. Right click on L1 and you will be presented with a list.

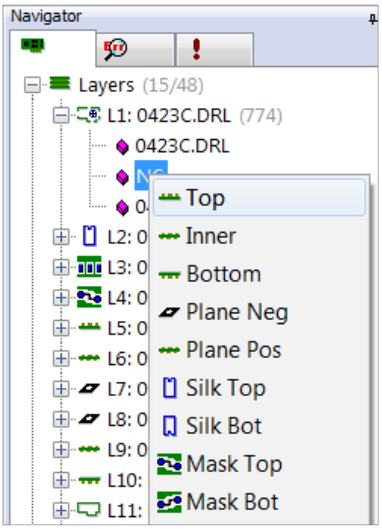


   - **Active** changes the layer to the active layer.
   - **Visible** turns the layer on or off.
   - **Delete** will delete the layer from the database.
   - **Spreadsheet** opens up the Setup Layers spreadsheet so you can modify the Layer Type, Name, and Color.
   - **Copy To…** quickly makes a copy of the layer to one or multiple layers.

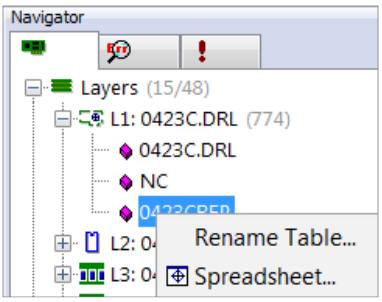
4. To rename the layer, expand L1 and right click on the first item.



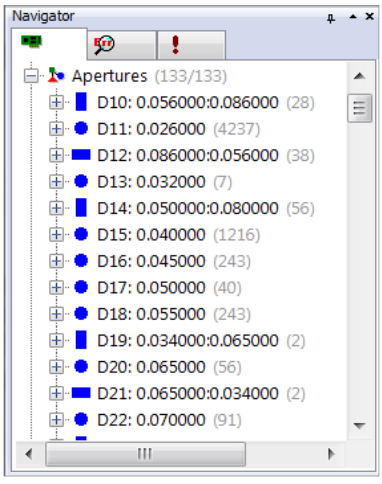
5. To change the Layer Type, right click on the second item and choose the new layer type from the list.



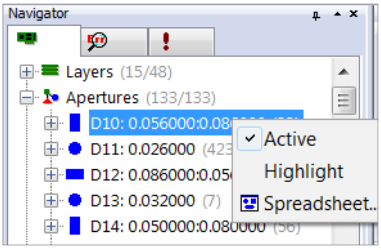
6. If the layer type you have is an NC layer you will have a third item in the list, which allows you to rename the Tool Table or edit the Tool Table by clicking on Spreadsheet.



Apertures area of the Navigator expands to show all the defined apertures in the design with their corresponding D-Code number and size. The number next to the size in parentheses is the quantity used in the design. The blue icon next to each aperture shows the general shape of the aperture.

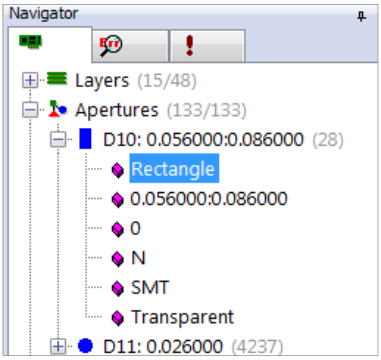


1. To make an Aperture the active D-Code, either double click on the D-Code in the Navigator or right click on the D-Code and choose Active. With the right click the option to Highlight the D-Codes used in the design is also available or choose Spreadsheet to edit the D-Code.

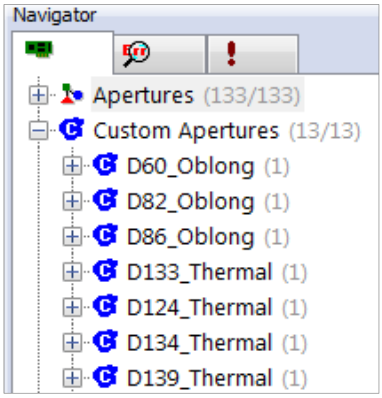


All aperture information that is presented in the spreadsheet may also be modified using the Navigator.

2. Click on the plus icon next to the D-Code to expand it. Change the Shape, Size, Angle, Polarity, Type and Transparency by right clicking on the item to be changed.

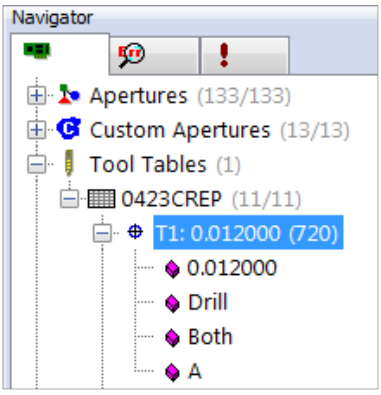


Custom Apertures are listed in the Aperture list as well as a complete list of all customs used in the design under its own heading in the Navigator.



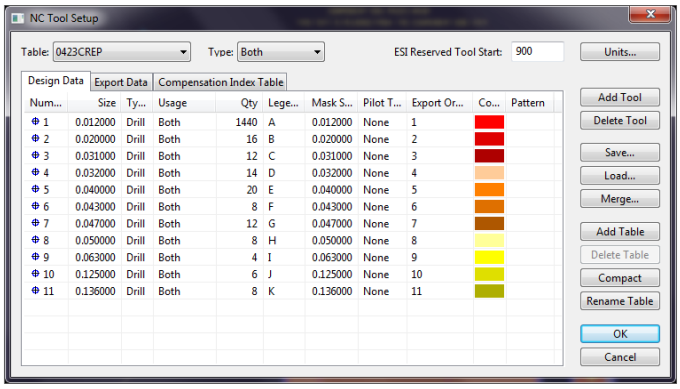
NC Tools: To see a list of defined NC tool tables, click on the plus box next to the NC Tools heading. The NC Tools area expands to show all the defined tool tables. Click on the plus box next to a Tool Table heading to reveal all the tools the table contains.

1. To edit a Tool, click on the plus box next to the Tool to expand it. Most of the basic NC data can be modified from the Navigator by right clicking on the item to be modified.
   - The first item is the tool size.
   - The second item is the tool type.
   - The third item is the plating status.
   - The last item is the legend. (The legend is one or two alphanumeric characters that are used to designate the tool in a drill hole chart).



2. To access the NC Tool Setup dialog box, right click on a tool in the Tool Tables list and choose Spreadsheet from the options, OR click on the Setup NC Tools icon in the toolbar.

   This command opens the **NC Tool Table**, which allows you to edit the drill and mill tools used in your NC layers. You can define tools in multiple tool tables, which can be designated as a "Drill" table (only drill tools are used) "Mill" table (only mill tools are used), or "Both" (drill and mill tools are combined on a single table).



Units

a. To view/change the units of measure click the Units button. The Configuration dialog box appears, with the Units and Precision tab active. Changing the units of measure here affects how measurements are represented to you.

Add Tool

a. Click the Add Tool button. A new tool is created, using the next available tool number. Define your tool, as needed.

Delete Tool (*Warning: You cannot delete a tool that has been used.*)

a. Click on the number of the tool you wish to delete. Click the Delete Tool button. The tool is immediately deleted.

Save

a. To save your tool list as an .nct file, click the Save button. The NC Tool Table Files dialog box appears.

b. Specify a file name for your tool list, and click Save. You can load this file in future databases.

Load

If you have a tool table already loaded, and its tools are used in the design, loading another tool table will make both tables available. If none of the existing tool table's tools are used, loading another tool table will replace the existing table.

a. To load a previously saved .nct file, click the Load button. The NC Tool Table Files dialog box appears.

b. Select the file you wish to load, and click Load. The table appears in the NC Tool Setup dialog box.

TIP: To load a tool list that was created by another program, use the File > Import > NC Tool List command.

Merge

To merge a tool table file with an existing table:

a. Click the Merge button. The NC Tool Table Files dialog box appears.

b. Select the .nct file you wish to merge in, and click the Open button. If you have tools that share the same tool number as the existing table, the NC Tool Mapping dialog box appears.

c. The system attempts to map tools with similar properties to each other (this is also done by clicking the Map button). The existing tool is shown first, then the incoming "mapped" tool is linked to it underneath. To change the assignment manually, click on the mapped tool and drag it to a different existing tool. You can then select the Overwrite Existing Tool Data Of Mapped Tools option, if you wish to overwrite the existing tools with the mapped tools.

   OR-

   You can have the system automatically assign the tools, by using one of the other three Automatic Tool Mapping Algorithms. Click the Overwrite button if you wish to overwrite the existing tools with the mapped tools. Click Ignore to ignore any incoming tools that are mapped to existing tools. Click Add to map the incoming tools to new, unused tool numbers.

d. When you are satisfied with the tool assignments, click OK.

Add Table

a. Click the Add Table button. An empty table appears, with a default name. Define your tool table and tools, as needed.

Compact

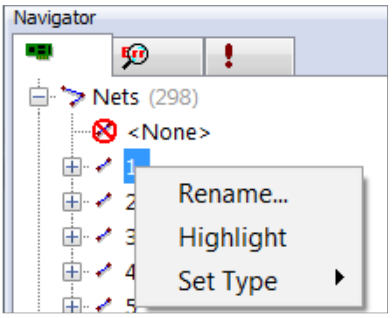
a. To renumber the tools so that they are consecutive, click the Compact button. You are also given the option to delete all unused tools from the tool table, before compacting it.

Rename Table

a. Click the Rename Table button. The Rename NC Tool Table dialog box appears.

b. Change the name as desired, and click OK.

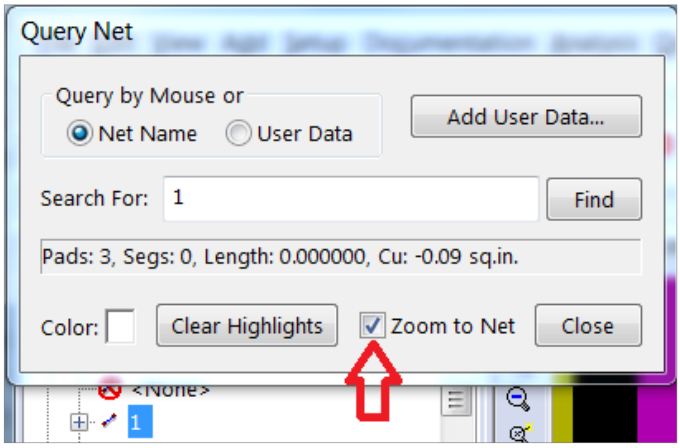
Nets



1. The number of nets in the design is noted next to the "Net" heading, in parentheses. To see a list of all nets in the design, click on the plus box next to the Nets heading. The Nets area expands to show all nets by number.
2. To Rename, Highlight, or Set Type of a Net right click on the Net.

View Nets

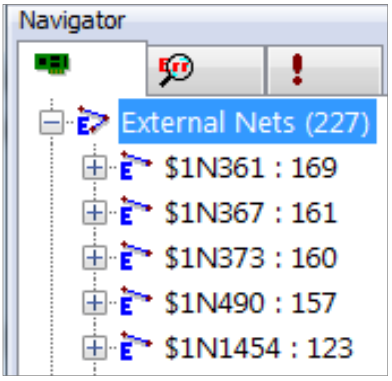
To quickly highlight and zoom to a net, double click on the Net in the Navigator. The net will be highlighted in the Main workspace and the Query Net dialog box opens. Click on the Zoom to Net and click the Find button. Any other nets you double-click on in the Navigator will now automatically snap to the center of the screen and highlight.



NOTE: When working with Gerber files Nets are only available after running the **Generate Netlist** command.

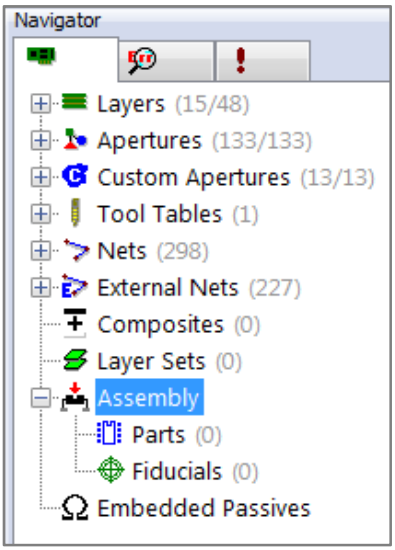
External Nets are imported from IPC-D-356, IPC-2581, ODB++ and PADS ASCII files. Besides doing Netlist Comparisons against the design file you can also assign the External Net Names to the Net instead of using the Net Number assigned by VisualCAM.

After you import the Netlist a new External Net layer is created. Viewed by itself this layer appears to contain no data. However, this layer contains the external Netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points. To view the points, turn on the associated top, bottom, and inner layers, and the external net layer.



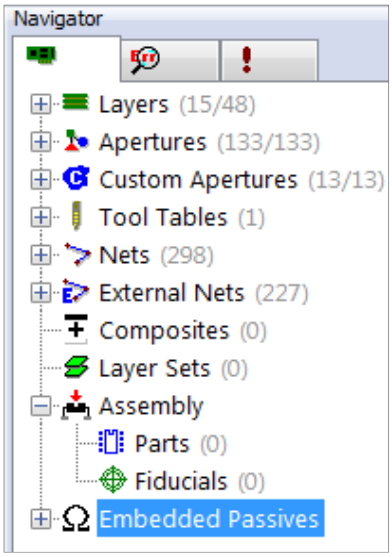
Assembly

If your design has Parts loaded in from IPC-2581, ODB++ or Reverse Engineered to VisualCAM the Assembly area allows you to view information about the parts, as well as perform all functions that are available to you in the Assembly menu of VisualCAM.



Embedded Passives

When you tag a layer type as "Passive Add" or "Passive Sub" it is immediately added to the Embedded Passives area of the Navigator. To view the detail list for a passive layer, click on the plus (+) box next to the desired passive layer.

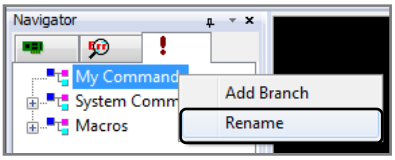


COMMANDS TAB

MY COMMANDS is where the shortcuts to your most commonly-used commands can be listed and organized. To use any of these commands just double click on it.

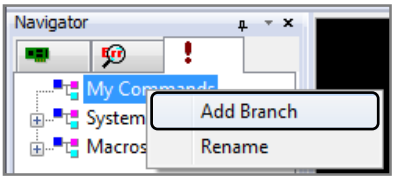
NOTE: Adding, deleting, or renaming a command or macro in the My Commands list has no affect on the command or macro itself. You are only modifying the shortcuts to these functions.

Rename the My Commands List

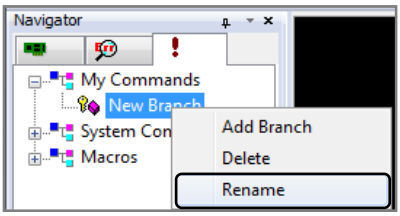


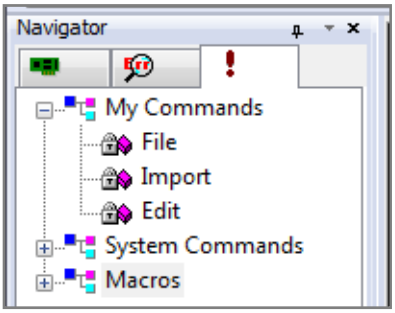
1. In the Navigator, left click on the **Commands tab**.
2. **Right-click on** the **My Commands** heading.
3. In the short-cut menu **select** the **Rename** command.
4. When the edit box appears around the heading, **type in the desired name** (for example, Shortcuts).
5. Press the **Enter** key on your keyboard.

Add and Rename New Branches

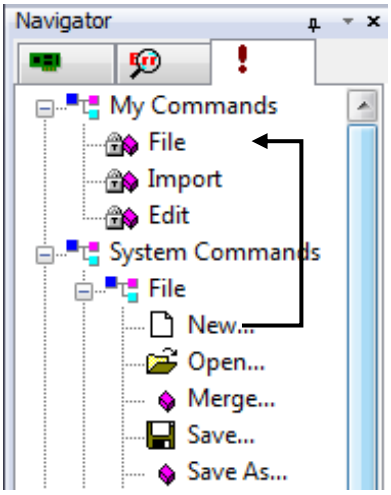


1. On the Commands tab **right-click on** the **My Commands** (or whatever you renamed it) heading.
2. In the short-cut menu **select** the **Add Branch** command.
3. When the new branch appears, **left click** on it to select it.
4. **Right click** on the new branch.
5. In the short-cut menu select the **Rename** command.
6. When the edit box appears around the branch name, type the word **File** in it.
7. Press the **Enter** key on your keyboard.
8. **Left click on** the **My Commands** heading again.
9. **Repeat steps 1 through 8 to add** and rename two more new **branches** named **Import and Edit**.
10. **Repeat the steps if desired** to add and rename other branches**.**





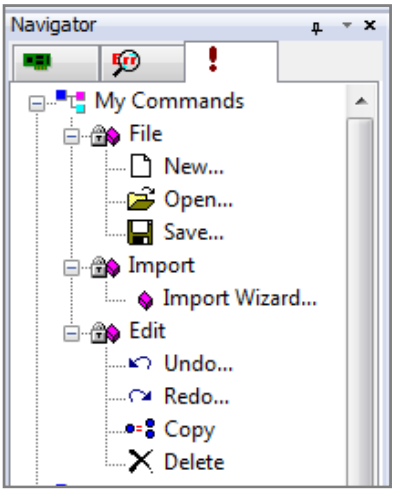
Add System Commands to Your List



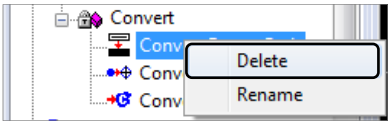
1. **Left click on** the plus sign (+) next to **System Commands** to **expand the tree** and view the available commands.
2. **Left click on** the plus sign (+) next to **File** to **expand the tree** and view the available File commands.
3. Left click on the **New** command, hold down the mouse key and **drag** the command to your My Commands list, **dropping** it on the branch named File.

The command now appears in your list. The command will also still appear in the System Command menu. You have just created a shortcut to it.

4. Left click on the **Open** command, hold down the mouse key and **drag** the command to your My Commands list, **dropping** it on the branch named File.
5. Left click on the **Save** command, hold down the mouse key and **drag** the command to your My Commands list, **dropping** it on the branch named File.
6. **Repeat Steps 2 and 3 to move the Import and Edit commands** you want from the System Commands branch to your My Commands branch.
7. **Rename** the commands if desired.
8. **Reorder** the commands (alphabetically for example) within the branches by dragging and dropping them within each list.



Delete Commands



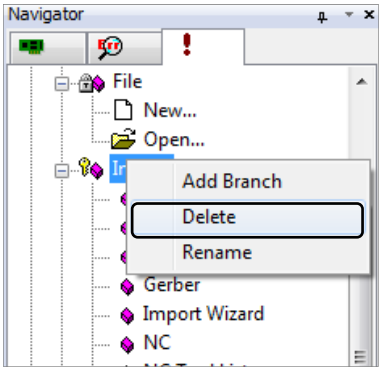
1. **Left click on the desired command** to select it.
2. **Right-click** on the command.
3. In the shortcut menu **select** the **Delete** command.

OR

1. **Left click on the desired branch** to select it.
2. **Press the Delete key** on the keyboard.

To add the command shortcut again to your list, find it in the System Commands menus and drag and drop it back to the list.

Delete Branches



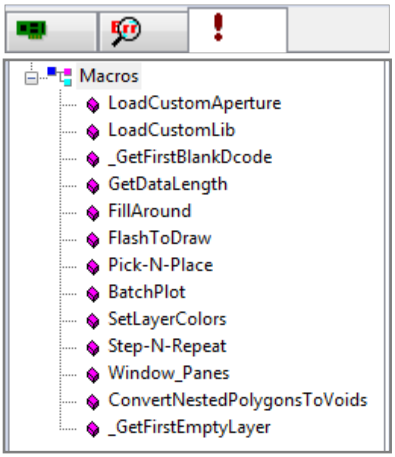
WARNING: Deleting a branch will delete all shortcuts within it as well.

1. **Left click on the desired branch** to select it.
2. **Right-click** on the branch.
3. In the shortcut menu **select** the **Delete** command.

OR

1. **Left click on the desired branch** to select it.
2. **Press the Delete key** on the keyboard.

Macros



The Macros section of the Commands tab contains user-definable commands. The purpose of these is to allow you to make your favorite Macros and favorite commands as accessible and easy to use as any VisualCAM command. These Macros can also be added to the User menu (see Customize the User Menu and Function Keys, page 30 for more information.)

Between the Macros, User Menu, toolbars, and programmable mouse/function keys, the commands you use the most are truly a keystroke or mouse click away.

For information on creating your own Macro commands, see the Macro Developer in the Online Help.

Activity: Use the Colors Bar

In this activity you will use the Colors Bar to turn layers on and off, set the active layer, and modify the layer color scheme to suit your preferences.

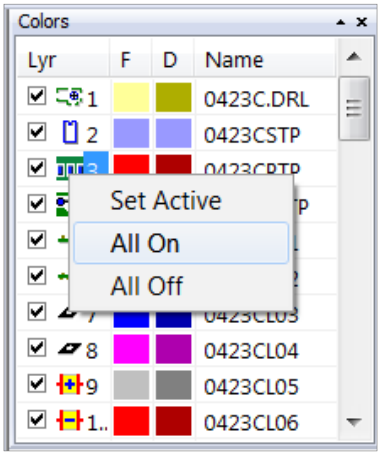
This activity should take about 5 min to complete.

If the Colors Bar does not appear in the workspace:

1. Go to **View > Toolbars…** and place a **check mark next to Colors bar** in the Visible Toolbars dialog box.
2. Click the **OK** button.

All On/All Off

1. In the **Colors Bar** right-click on any of the layers listed.
2. Select **All On.** (All layers are checked and become visible in the workspace.)
3. Now repeat this but select **All Off** this time. (All layers are unchecked and become invisible in the workspace except for the active layer.)



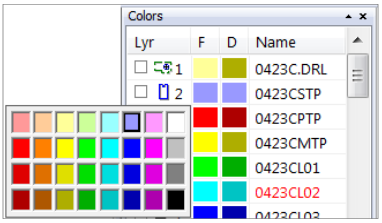
Set the Active Layer

1. In the **Colors** Bar right-click on Layer 3.
2. Select **Set Active.** (The layer name changes to a red font)

You can also set the Active Layer by double-clicking a layer in the Colors bar.

Modify Layer Colors

1. In the **Colors** bar left-click on the color square F column to change the flash color.
2. Select a color of your choice, cyan for example.
3. Now repeat this process for the D column to change the draw colors. All objects are now changed in your workspace to the new colors you selected.

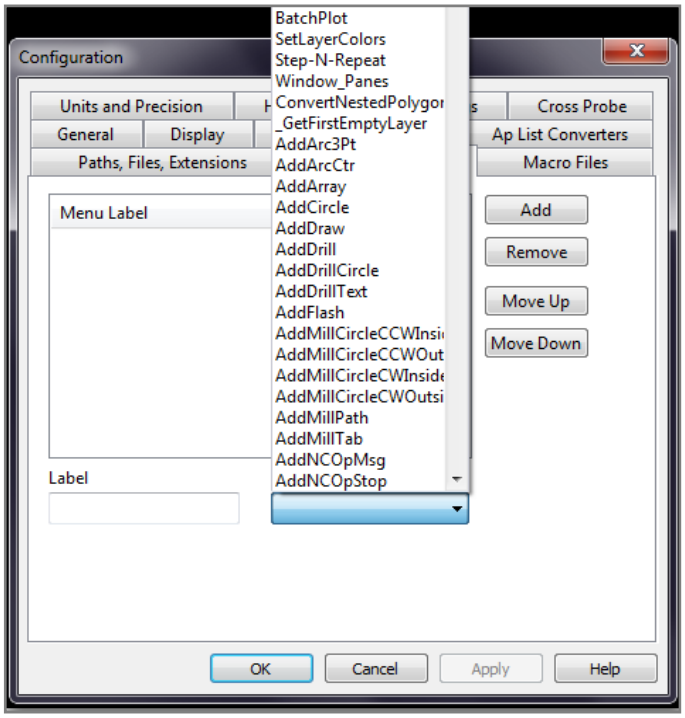


TIP: It is a good practice to use contrasting colors for your Draw and Flash columns. For example on your Top and Bottom copper layers (Green and Blue) This will help you to see graphically if there are drawn pads rather than flashes. Many commands require that the pads be flashes (*see Draws to Flash Conversion, page 77 for more information.*)

Activity: Customize the User Menu and Function Keys

This activity should take about 5 minutes to complete.

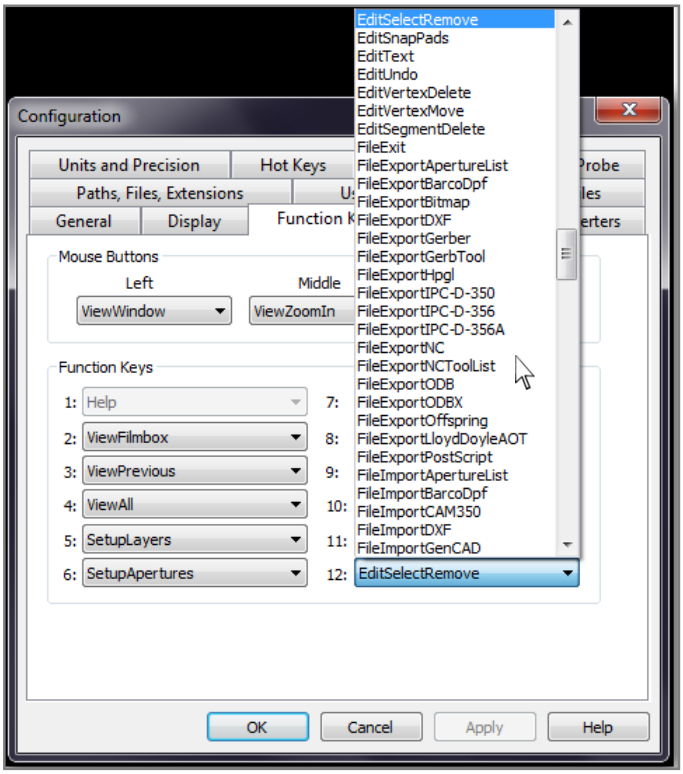
User Menu



1. In the Menu bar, select the **Options > Configure** command.
2. In the Configuration dialog box, click on the **User Menu tab**.
3. To add commands to the menu:
    a. Drop down the **Command list** and **select one**.
    b. In the Label textbox, **change the text** to accommodate the new command that will appear in the menu.
    c. Click on the **Add** button. The new command will be added to the User menu list.
    d. **Repeat** to add more commands.
4. To rearrange commands in the list:
    a. **Select the command** in the list.
    b. Click on the **Move Up or Move Down** button.
5. To remove commands from the menu:
    a. **Select the command** in the list.
    b. Click on the **Remove** button.
6. When you are satisfied with the way you have configured the menu, click on the **OK** button.

Function Keys

Keyboard function keys F2-F9 and F11-F12 can be customized with commands that you frequently use.



1. In the Menu bar, select the **Options > Configure** command.
2. In the Configuration dialog box, click on the **Function Key/Mouse tab**.
3. **Click on the key you want to program** and you will be presented with a list of commands from which to choose.
4. **Choose a macro from the list.** All macros that you have loaded are available to program a Function Key and they will be presented at the top of the list in the pull down menu.
5. **Repeat steps 3 and 4** to change other function keys.
6. When you are satisfied with how you have configured the function keys, click on the **OK** button.

Activity: Use Common Edit, Query, and View Commands

This activity should take about 10 minutes to complete.

Edit > Copy

Toolbar Button: 

You may use this command to copy single items, windowed selections, or a select group of items.

To copy items from one database to another, use the Edit > Copy Selection To Clipboard command. To copy all the data in a layer to another layer, use the Layers list in the Navigator.

If you copy drawn or flashed data from a graphics layer to an NC layer, the system will convert all draws to mill paths, and all flashes to drill hits. If tools the same size as the draws/flashes do not exist in the tool table that is assigned to the NC layer, new tools will be created. You can also use the Tools > Convert > Gerber To NC command to copy and convert an entire graphics layer to an NC layer.

TIP: Hotkeys are available to help you make your selections. For example, you can snap your cursor to the center of an item using the Home hotkey.

To copy a single item:

1. Select the **Edit > Copy** command, or click on the toolbar button.
2. In the Selection Filter, select **Item** mode, as well as the **Type** of item that you wish to select.
3. **Click on the item you wish to copy** in the workspace. If more than one item exists at that location, the Choose Selection dialog box appears, where you may select the desired item.
4. An outline of the item is attached to your cursor. **Move your cursor** to the desired location for the copy, and **click to place it**, **OR you can select exact coordinates** for your placement in the XY Bar.
5. If you wish to place additional copies of the same item in other locations, click on those locations. Otherwise, press the **Esc** key.
6. You can now **select another item to copy**, **or press the Esc key** to end the function.

To copy a windowed selection of items:

1. Select the **Edit > Copy** command, or click on the toolbar button.
2. In the Selection Filter, select **Window mode** if you wish to copy items that fall entirely within your designated selection window. Select **Window +Xing mode** if you want to copy any item that crosses into your designated selection Window. Also choose the type of item that you wish to select.
3. In the workspace, **click on the lower-left corner** of the desired selection area. The selection window is now attached to your cursor.
4. **Move your cursor, and click on the upper-right corner** of the desired selection area. The selected items are highlighted, and you are prompted to select a "copy from point."
5. **Click on an anchor point**, where the items will be attached in relation to your cursor. An outline

of the data being copied is attached to your cursor.

6. **Move your cursor to the desired location** for the copy, and click to place it, **OR you can select exact coordinates** for your placement in the XY Bar.

TIP: If you select data from more than one layer, i.e. more than one layer is visible, then the data will be copied into their respective source layers.

1. If you wish to place additional copies of the same items in other locations, **select those locations**. Otherwise, press the **Esc** key.
2. You can now **select other items to copy**, or press the **Esc** key to end the function.

To copy a Select Group:

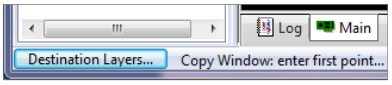
1. Create a Select Group using the **Edit > Select > Add To** command.
2. Select the **Edit > Copy** command, or click on the toolbar button.
3. In the Selection Filter, select **Group mode**. All other types of filters are not applicable.
4. **Click on an anchor point**, where the items will be attached in relation to your cursor. An outline of the data being copied is attached to your cursor.
5. **Move your cursor to the desired location** for the copy, and click to place it, **OR you can select exact coordinates** for your placement in the XY Bar.

TIP: If your Select Group contains items from more than one layer, then the items will be copied into their respective source layers.

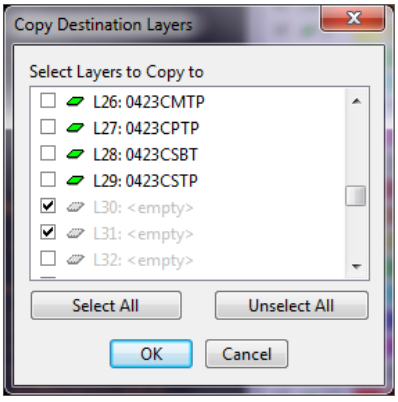
1. If you wish to place additional copies of the same items in other locations, **select those locations**. Otherwise, press the **Esc** key.
2. Press the **Esc** key to end the function.

To copy items to another layer:

1. After having selected your items using one of the methods above, and when they are attached to your cursor, click on the **Destination Layers button** in the Status Bar. The Copy Destination Layers dialog box appears.



2. **Select one or more destination layers** where you want to copy to. Copies of all the selected data (regardless of their layer of origin) will be merged into each selected destination layer.
3. **Specify where to place the copies** by doing one of the following:
   a. Move your cursor to the desired location for the copy, and click to place it.
   b. Enter exact coordinates for your placement by clicking the Abs button in the XY Bar.



c. Press the <End> key to perform an in-place copy.

4. If you wish to place additional copies of the same items in other locations, **click on those locations**. Otherwise, press the **Esc** key.

You can now **select other items to copy**, or press the **Esc** key to end the function.

Edit > Delete

Toolbar Button: 

Select this command when you wish to delete items from one or more visible layers.

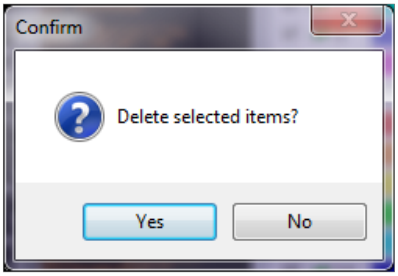
If you wish to delete items from an NC object, such as a mill path segment, use the Edit > NC > Explode command first to explode the object into its individual base elements. In the case of a mill path, it is converted into several mill paths, one for each segment. You can then join the remaining segments back together with the Edit > NC > Combine Paths command.

You can also use the Edit > Clip command to delete windowed sections from a draw or mill path.

TIP: Hotkeys are available to help you make your selections. For example, you can snap your cursor to the center of an item using the Home hotkey.

To delete items:

1. Select the **Edit > Delete** command.
2. **Select an item or items to delete**. Use the Selection Filter to aid your selections, as necessary.
3. Each time you select an item for deletion, **you are prompted for confirmation**. If you do not want to be prompted, click the Disable Prompts button in the Status Bar.



When you are finished, press the **Esc** button.

Edit > Undo

Toolbar Button: 

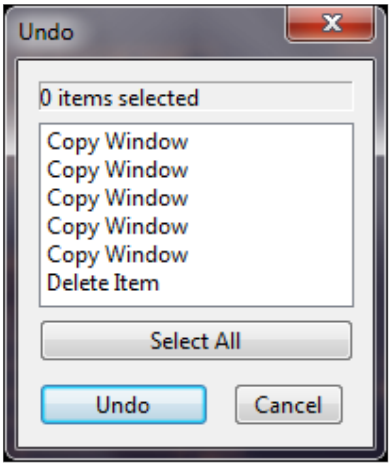
Equivalent Hotkey: U (undo last edit) or Ctrl+U (undo all edits)

The Undo command reverses changes you have made to the currently loaded database. Note that some functions cannot be undone; the Undo command is only available when a function allows it.

If you plan to use the Undo command, it must be enabled with the Options > Configure command prior to making any edits. Undo increases the amount of memory VisualCAM requires. If you do not require the undo capability, you may disable it. Disabling undo releases any memory currently associated with undo information, and prevents further memory use.

To undo a change:

1. Select the **Edit > Undo** command, or use the toolbar button or hotkey.



The Undo dialog box appears, with your last seven edits listed (the most recent edit being listed first). In order to undo one change, you must undo all subsequent changes (e.g. if you add a draw, then a flash, and you wish to undo the draw, you must undo the flash as well).

2. **Select the change(s) you wish to undo** in the dialog box.
3. Click the **Undo** button.

The dialog box closes, and the workspace is immediately updated.

Edit > Redo

Toolbar Button: 

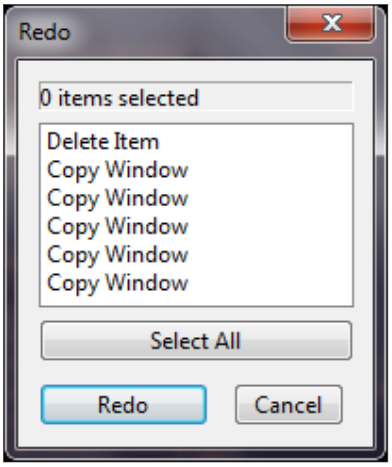
Equivalent Hotkey: Shift+U (reverse last undo)

Where the Undo command reverses changes you have made to your database, the Redo command reverses the Undo command. Your database is returned to the state it was in prior to your selection of the Undo command.

If you plan to use the Undo & Redo commands, the Undo command must be enabled with the Options > Configure command prior to making any edits.

To redo a change:

1. Select the **Edit > Redo** command, or use the toolbar button or hotkey.



The Redo dialog box appears with the last seven "undone" edits listed. You redo changes in the order that they were originally performed, and in order to redo one change, you must redo all changes made prior to it.

2. **Select the change(s) you wish to redo** in the dialog box.
3. Click the **Redo** button.

The dialog box closes, and the workspace is immediately updated.

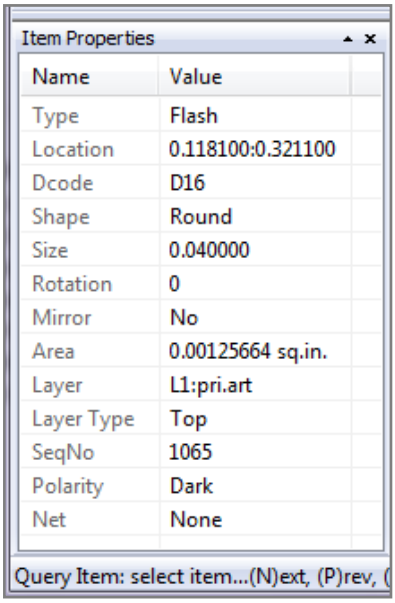
Query > Item

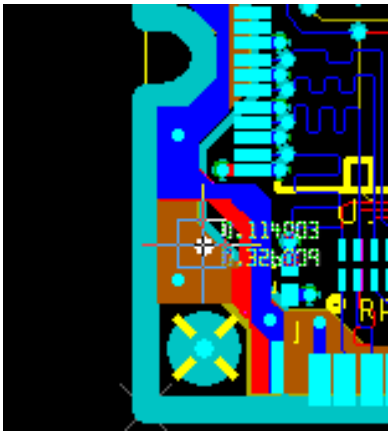
Toolbar Button: 

Equivalent Hotkey: Q

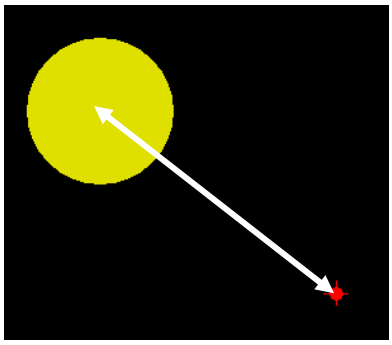
1. Select the **Query > Item** command, or use the toolbar button or hotkey.
2. **Click on the item** in the design.
3. If more than one item exists in the location you select, the Choose Selection dialog box appears. As you **click on items in the Choose Selection dialog box**, each selected item is highlighted, and its attributes are displayed in the Item Properties box.
4. **Repeat** steps 1 through 3 for other items.

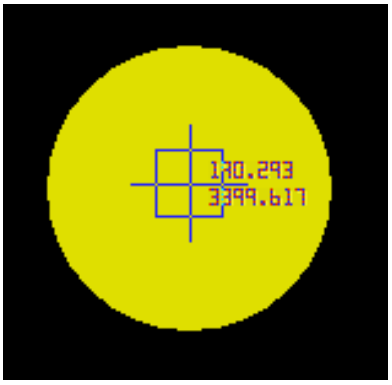
TIPS: When querying an item, you will notice the cursor on the screen is a crosshair with a box in the center. The size of the box dictates how many features will be captured during the query. **Press the "Page Up" and "Page Down" buttons to change the size of the box**. Place the cursor over the item in question, and **press "Home" on the keyboard to snap to the closest feature or intersection**. **Use the "+" key to zoom in, and the "-" key to zoom out**. These functions work for many processes.

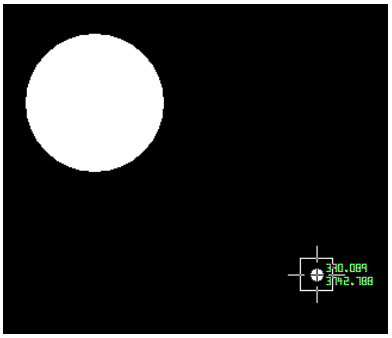




Query > Measure > Center to Center







Toolbar Button: 

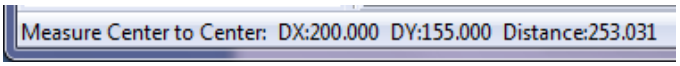
The Center To Center command measures the actual distance between the centers of two items. You can select any item that is on a visible layer.

To measure between the centers of two items:

1. **Zoom or pan to the items you wish to measure** between in the workspace. To aid in your selection, turn off any layers that you do not need to be visible.
2. Select the **Query > Measure > Center To Center** command, or click on the toolbar button.

You are prompted to select the first item.

3. A bounding box is added to the cursor. Anything that falls within this bounding box is a potential selection. You can increase the size of the box by pressing the PgUp hotkey, or decrease the size (for increased accuracy) by pressing the PgDn hotkey.
4. Use the Selection Filter, if necessary, and **click on the item you want to measure from**. If more than one item is at the selected location, a Choose Selection dialog box appears, where you can select your desired item.
5. **Click on the item you want to measure to**. The actual distance between the center of the items in X and Y, as well as true length, is displayed in the Status Bar:



6. To measure between the first selected item and another item, press the **Esc** key, and **select another item**,

   OR to measure between two different items, **press the Esc key twice** (watch the prompt in the Status Bar), and **click on the desired items**.

   OR if you are finished, **press the Esc key three times**.

View > All Fit

Toolbar Button: 

Equivalent Hotkey: Ctrl+R

This command adjusts the size of the viewing window to encompass the extents of the currently displayed layer(s).

If you have added to or deleted data from any displayed layers, you may need to use the Query > Extents command to calculate the current extremes of the database.

View > Redraw

Toolbar Button: 

Equivalent Hotkey: R

The Redraw command refreshes the display in the workspace.

View > Sketch

Toolbar Button: 

The View > Sketch command toggles sketch mode on/off. When sketch mode is enabled, items are shown with an outline only.

An option to this setting is stick mode, which is enabled using the Options > Configure command. (Select the Display Draws As Sticks In Sketch Mode option under the Display tab). When enabled, stick mode displays all draws as a single thin line. This mode can help you spot stacked and buried items.

Review Questions

What is the primary purpose of the Navigator?

_____
_____
_____
_____
_____

What is the primary purpose of the Selection Filter?

_____
_____
_____
_____
_____

What is the primary purpose of the Colors Bar?

_____
_____
_____
_____
_____

What is the primary purpose of the Properties Window?

_____
_____
_____
_____
_____

What are the two functions of the Active Layer?

_____
_____
_____
_____
_____

Importing Gerber Layers

Purpose

In this lesson you will learn the steps and processes used to convert Gerber files to VisualCAM format

Lesson Objectives

In this lesson you will do the following:

- Learn the function and use of the Import Wizard.
- Learn about the different file types that can be imported.

Estimated Completion Time

Approximately 15 minutes.

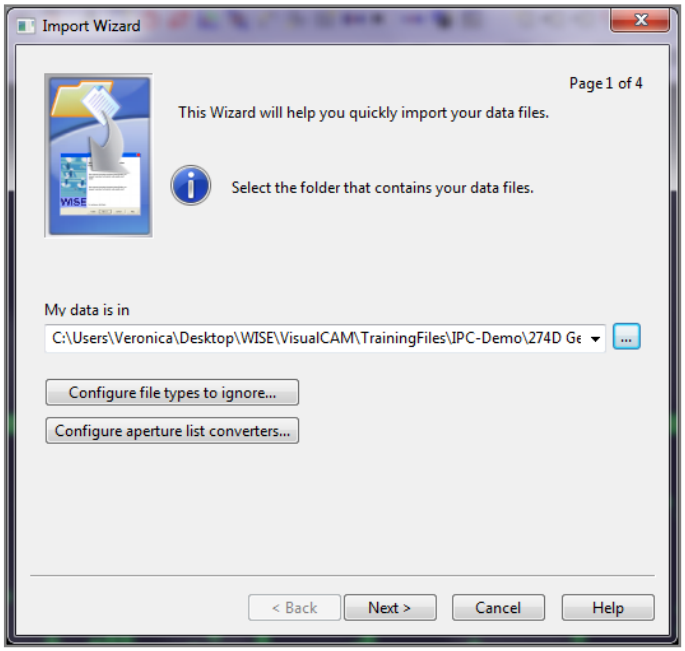
The Import Wizard

The Import Wizard allows you to specify which files (Gerber, aperture lists, VisualCAM databases, Drill files, etc.) you wish to import, assign aperture lists to specific Gerber files, and order the loading sequence. VisualCAM automatically recognizes the file types, and imports the specified files into the first empty block of consecutive layers found. D-Codes with multiple definitions are automatically remapped to new D-Codes as needed.

TIP: To use the Import Wizard, make sure all the files you wish to import are in the same folder.

Wizard: Page 1

Page one is where you select the folder that contains the files you want to import.



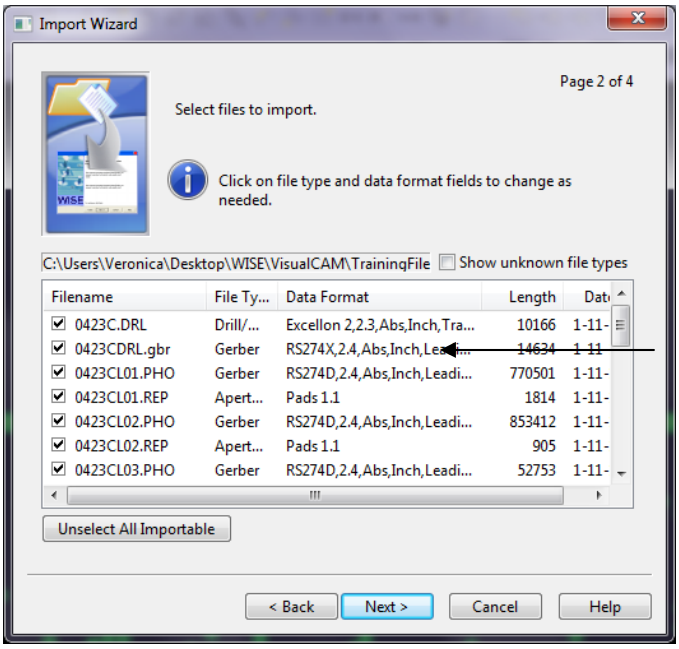
My data is in is used to tell the wizard where the files are located in the computer. Clicking the Browse button ... to the right of the textbox will allow you to search for the folder.

Configure File Types To Ignore button opens the Configuration dialog box, where you may enter any file name extensions that you want the program to ignore when scanning the folder. The more files that may be safely ignored, the faster the Import Wizard will be able to scan the specified folder.

Configure Aperture List Converters button opens the Configuration dialog box, where you may specify which aperture list converters should be enabled. The more aperture list converters enabled, the longer it will take to scan the specified folder.

Wizard: Page 2

Page two presents the list of files that VisualCAM found in the selected import folder. Clicking once on the list column headers will sort the list based on the data within the respective columns. Clicking once on the Data Format description of the file will open another dialog box based on the corresponding data format type.



Click on the Data Format description to open the Import dialog box for the corresponding data format type.

Filename check marks tell VisualCAM to import the file(s). Clicking the Unselect All Importable button will remove the check marks next to all file names.

File Type lists what types the system can import. If a file cannot be imported, "???" will appear. If VisualCAM detects the file type incorrectly, you may change it by clicking on the file type in question and selecting the correct type from the shortcut menu.

Data Format column indicates the detected data format for each file. Clicking once on the Data Type for a file will present you with a dialog box where you can override the detected format.

- **If the file is Gerber**, the **Gerber Import Data Format** dialog box will appear (a description of the dialog box parameters follows in this section.)
- **If the file is an Aperture list**, the **Select Converter** dialog box will appear. VisualCAM will select an appropriate aperture converter to use for the aperture list if one is available, but you can override the selection by selecting a different converter to use from the Data Format list. You can have the selection applied to all incoming aperture lists by clicking the Apply To All button and you can create a new converter by clicking the New Converter button.
- **If the file is an NC tool list**, the **Select Converter** dialog box will appear. VisualCAM has already selected an appropriate converter to use for the tool list if one is available, but you

can override the selection, by selecting a different converter to use from the Data Format list. You can have the selection applied to all incoming tool lists by clicking the Apply To All button and you can create a new converter by clicking the New Converter button.

- **If the file is NC (drill or mill) data**, the **Import NC Data** dialog box will appear.
- **If the file is DXF**, the **Import DXF** dialog box will appear.
- **If the file is HPGL**, the **HPGL Import** dialog box will appear.
- **If the file is DPF**, there are no format parameters to modify.

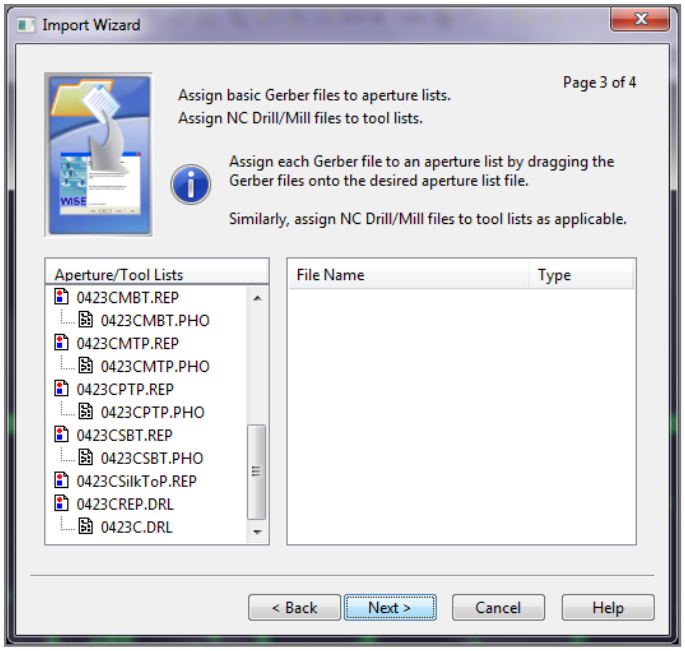
Length is the size of the file in bytes.

Date is when the file was last modified.

Show Unknown File Types. By default the Wizard only displays those files it can identify. Selecting this option will tell the Wizard to display all files contained in the selected import folder.

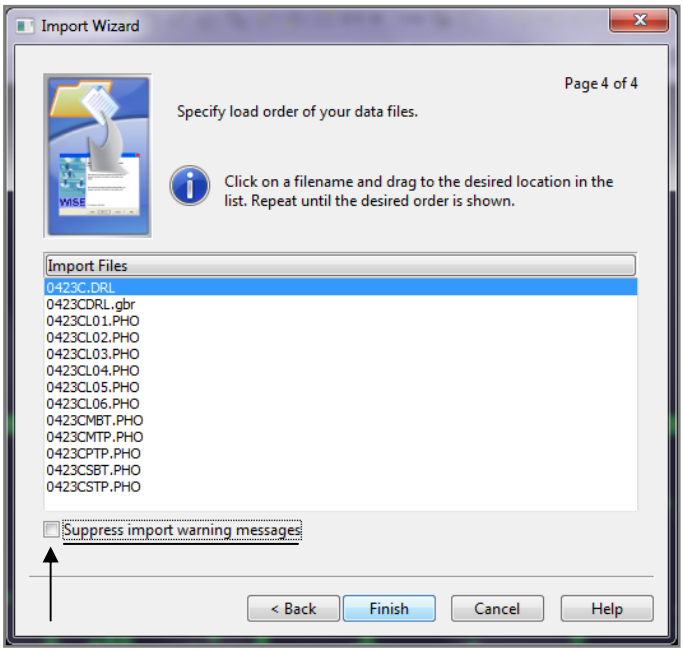
Wizard: Page 3

Page three only appears if you are importing 274-D files and is used to assign 274-D Gerber files to aperture lists. You can drag Gerber files onto any aperture list that appears. If only one aperture list is present, all Gerber files are automatically assigned to it.



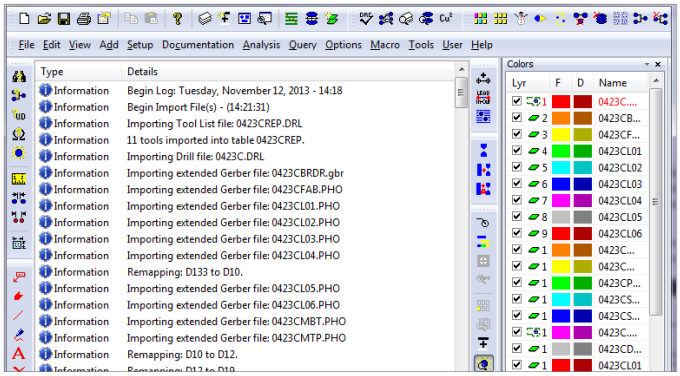
Wizard: Page 4

Page four presents the final list of files that are to be imported. The files are imported to layers in the VisualCAM workspace in the order they are listed in the wizard. Clicking and dragging a file to another position within the list will reorder the files so that your layers will be in the proper order when they appear in the workspace.



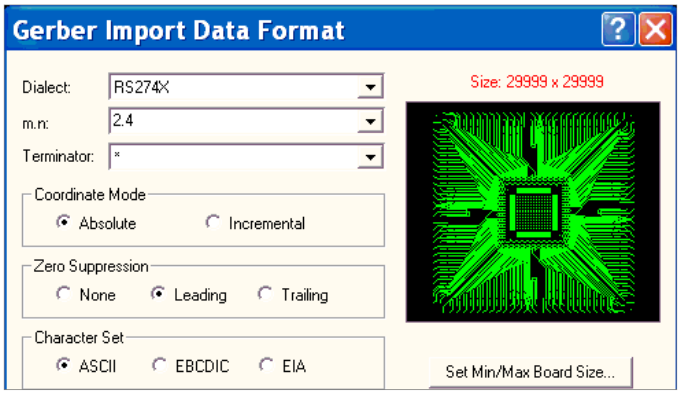
Selecting **Suppress Import Warning Messages** will prevent messages from being displayed during the import process. A log file will still be created, but you will not be prompted to view it.

Once the files are imported any pertinent information or warning messages will be displayed on the VisualCAM Log screen. Save the log by right-clicking on the Log screen and selecting the Save Log command from the shortcut menu.



TIP: Polygon Voids in 274-X files are supported, but are not recognized as standalone entities. Voids are only relative to the closed and filled polygon that contains them. If you query a polygon void (**Query > Item** command), the information returned is for the polygon itself. Polygon voids also cannot be modified, added, or deleted.

Gerber Import Data Format Dialog Box Parameters



Dialect: The format of the imported file. Available options are: RS274D, RS274X, and Fire9000.

m.n.: The coordinate format. Defines the number of digits before and after an implied decimal point.

Coordinate Mode

Absolute: All X:Y coordinates are referenced to a common origin.

Incremental: Each X:Y coordinate is a displacement from the previous coordinate.

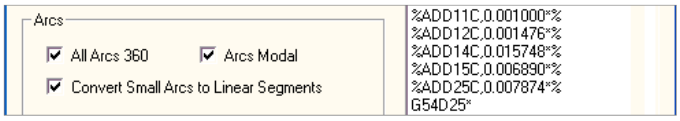
Zero Suppression

None: Zero suppression allows coordinates to have zeroes in front or back of integers.

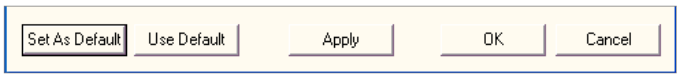
Leading: Shows zeroes only at the end of a number.

Trailing: Shows zeroes only at the beginning of a number.

Character Set: Typically ASCII.



Arcs: On Gerber imports it is recommended that you use “All Arcs 360” as the default parameter.



Set as Default: Use this to save the current settings as the defaults for files whose format cannot be identified by VisualCAM. If an imported file *can* be identified, the format settings will be set accordingly (*not* to the default settings that you save).

Use Default: Use this to apply the saved default settings to imported files. Any detected settings are overridden.

Activity: Import 274D Gerber Files Using the Import Wizard

In this activity you will select all of the necessary files and converters necessary to import the 274D Gerber layers to add to the IPC-Demo job.

This activity should take about 15 minutes to complete.

Create a New Job

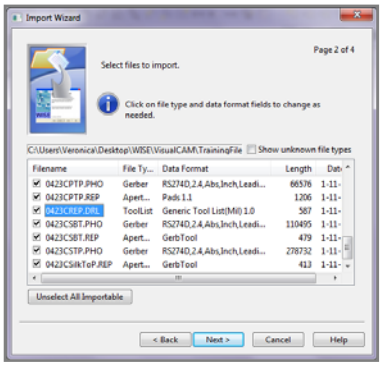
1. Click the **New** icon on the toolbars, or go to **File > New.**

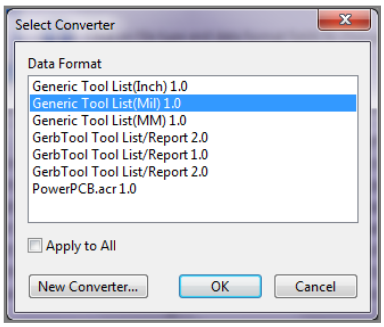
Search for the Files

2. Go to **File > Import > Import Wizard.**
3. On Page 1 of the Wizard, click the ... button to browse to the **…Desktop\Training Files\IPC-Demo\274D Gerber Set** folder saved in your computer.
4. Click the **Next** button.

Change the Detected Data Format

5. On Page 2 **scroll** through the alphabetical Filename list and find the file with the name "**0423CREP.DRL**".
6. Click on the text in the **Data Format** column for file 0423CREP.DRL to open the **Select Converter** dialog box.
7. In the **Select Converter** dialog box, if it's not already selected, click on the **"Generic Tool List (Mil) 1.0"** option.
8. Click on the **OK** button to close the Select Converter dialog box.
9. Back on Page 2 click on the **Next** button.

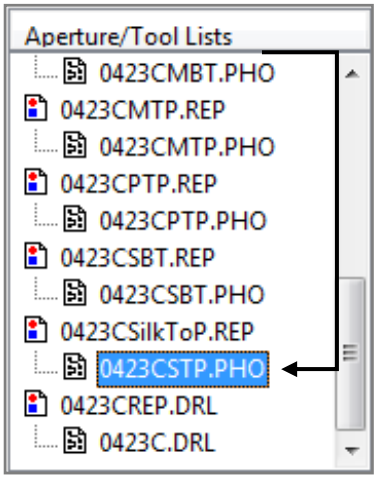




Move a File Within the Aperture/Tool List

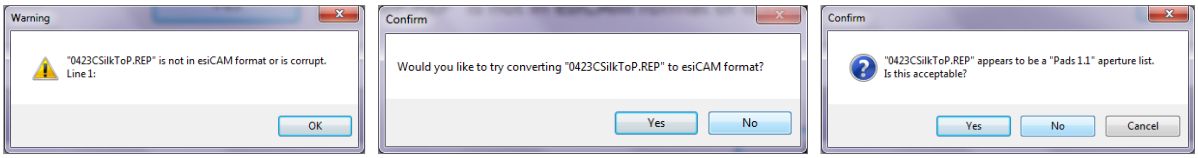
NOTE: In the Aperture/Tool Lists assignment tab, file **0423CSTP.PHO** is assigned to the **0423CL01.REP** aperture list instead of **0423CSilkTop.REP**. This is because VisualCAM automatically assigns Gerber files to the closest reasonable match it can find. If no match is found, then the Gerber file will remain on the right side window. In this case, it considered **0423CL01.REP** to be the first reasonable match.

10. **Click and drag the 0423CSTP.PHO Gerber file** down the list and drop it on 0423CSilkTop.REP.
11. Click on the **Next** button.
12. Click **Finish**.



Respond to Warning Messages

NOTE: If aperture lists do not have the correct converter selected by VisualCAM during the automatic process, the system will display warning messages.



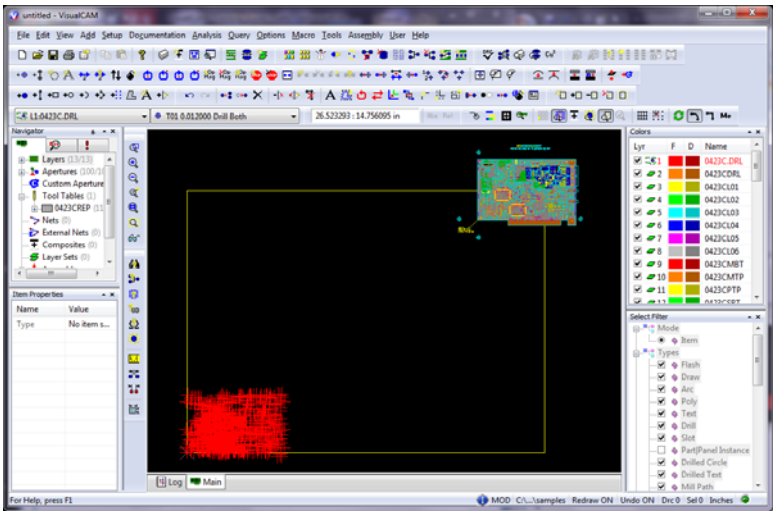
13. Click **OK** to respond to the Warning message.
14. Click **YES** to Confirm the conversion message.
15. Click **YES** to Confirm the newly selected converter.
16. **Repeat** this process for any other warnings during the import process.

Save the Database File

17. Click on the **save** icon , or go to **File > Save**.
18. **Enter a name** for the database such as “IPCDemo_YourName.vcam”.
19. Click on the **Save** button.

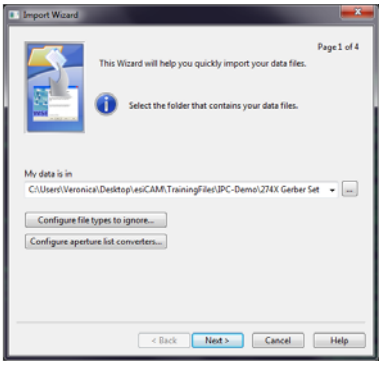
View the Imported File in VisualCAM

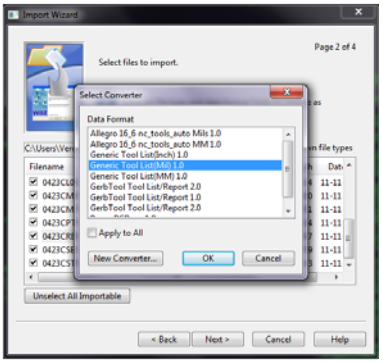
20. Click on the Main tab  at the bottom of your workspace. The workspace should look something like the image below.



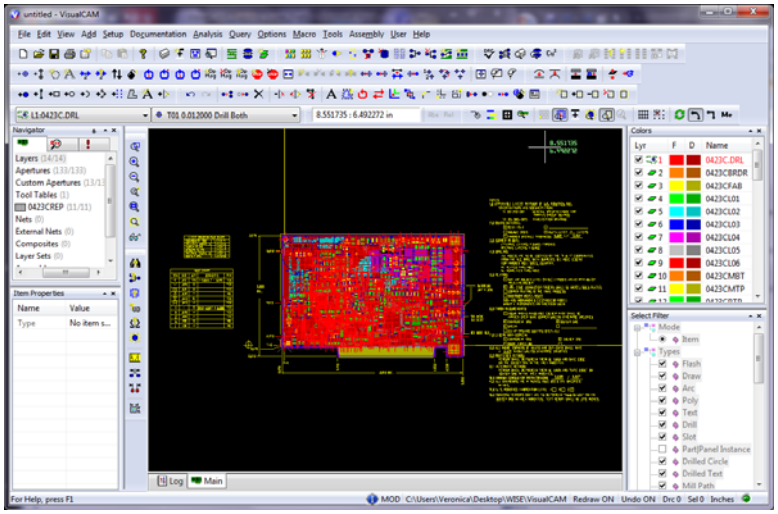
Activity: Import 274X Gerber Files Using the Import Wizard

This activity should take about 5 minutes to complete.





1. Click the **New** icon on the toolbar, or go to **File > New**.
2. Go to **File > Import > Import Wizard**.
3. On Page 1 click the button to browse to the **…Desktop\Training Files\IPC-Demo\274X Gerber Set** folder.
4. Click the **Next** button**.**
5. On Page 2 **scroll** through the alphabetical Filename list and find the file with the name "**0423CREP.DRL**".
6. Click on the text in the Data Format column for file **0423CREP.DRL** to open the Select Converter dialog box.
7. In the Select Converter dialog box, if it's not already selected, click on the "**Generic Tool List (Mil) 1.0**" option.
8. Click on the **OK** button to close the Select Converter dialog box.
9. Back on Page 2 click on the **Next** button.
10. On Page 3 click on the **Next** button
11. On Page 4 click on the **Finish** button.
12. **Respond** to any warning messages.
13. **Save** the database file.
14. Click **on the Main tab** to view the data in the workspace:



Review Questions

True or False; The Import Wizard requires all associated files to be imported to reside in the same windows directory.

_____

True or False; The Import Wizard will automatically identify an aperture list converter to use for importing an aperture table if one is available.

_____

True or False; When using the Import Wizard, it is a good idea to double check the detection of the data format.

_____

Working with Layers

Purpose

In this lesson the functions and processes used to identify the different layer types of the design and how best to sort and color them will be described. You will also learn the method used to create a separate border layer if one was not provided with the original Gerber files.

Definitions:

- **SMD** – Surface Mount Device
- **THD** – Thru Hole Device
- **Tag** – Identify a layer's type
- **PCB** – Printed Circuit Board

Lesson Objectives

In this lesson you will do the following:

- Change the layer type.
- Sort layers.
- Create a border layer.

Estimated Completion Time

Approximately 30 minutes.

ID (Tag) Layers

Prior to any intelligent processing of the design, it is necessary to identify, or Tag, the different layer types that were imported during the Import Wizard steps. By default, all layers are identified with the type "**Other**", except for NC Drill, which is automatically identified due to its unique structure. The minimum layer types required to complete Analysis are:

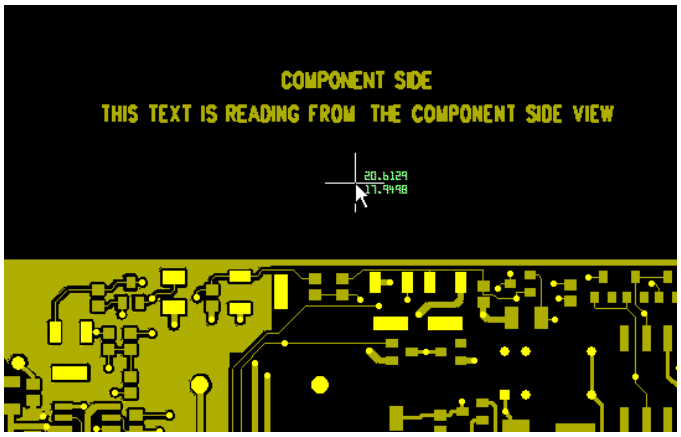
For **Netlist/Netlist Compare:**

- Top and/or Bottom Copper
- All Inner Signal layers
- All Plane layers
- All NC Drill layers

For complete **DRC/MRC**, all the above plus:

- Border
- Top and/or Bottom Mask (Paste and Solder)
- Top and/or Bottom Silk

TIP: In the IPC-Demo job, as will be true for most designs you work with, there is text on each layer identifying what the layer is (see image below). Locating and using these text comments is the easiest method available for identifying layers in VisualCAM. You may also find that the file names for the Gerber layers have a naming convention applied that will help you identify the type. For example, "0423C**SBT**" is a **Silk-Bottom** layer and "0423C**STP**" is a **Silk-Top** layer

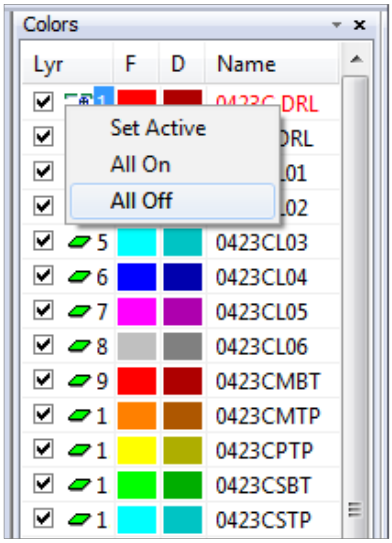


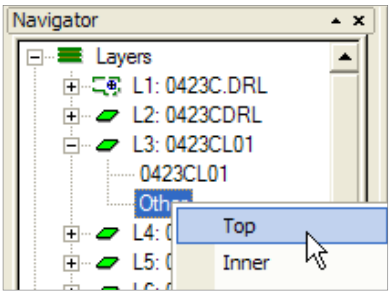
Activity: Change the Layer Type

In this activity you will use the Navigator to tag all the layers for the IPC-Demo job.

This activity should take about 10 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. Right-**click** on any layer in the Color Window and select **All Off**. The active layer remains checked and visible.
3. **Uncheck** the active layer now.
4. Skipping the layer "**0423C.DRL**" for now, click in the check box next to the first undefined layer in your list to make it visible. This should be layer "**0423CL01**".
5. Use the zoom commands to **zoom-in** around the descriptive text label for this layer as shown in the image above.
6. Click the + sign to **expand the Layers list** in the Navigator.
7. Click the + sign next to layer "**0423CL01**".
8. Right-click where it says **Other** and select **Top** for the type.
9. **Repeat this process** for the remaining layers (still ignoring layer "0423CDRL"). Try to use the descriptive text labels on the layers and/or the layer names themselves to determine the type for each. If you have trouble determining the type or to check if your choices are correct you can refer to the following table.
10. **Save** the file**.**





Layer	Type	Layer	Type	Layer	Type
0423CL01	Top	0423CL05	Inner-Signal	0423CPTP	Paste-Top
0423CL02	Inner-Signal	0423CL06	Bottom	0423CSBT	Silk-Bottom
0423CL03	Plane-Neg	0423CMBT	Mask-Bottom	0423CSTP	Silk-Top
0423CL04	Plane-Neg	0423CMTP	Mask-Top	0423C.DRL	Drill

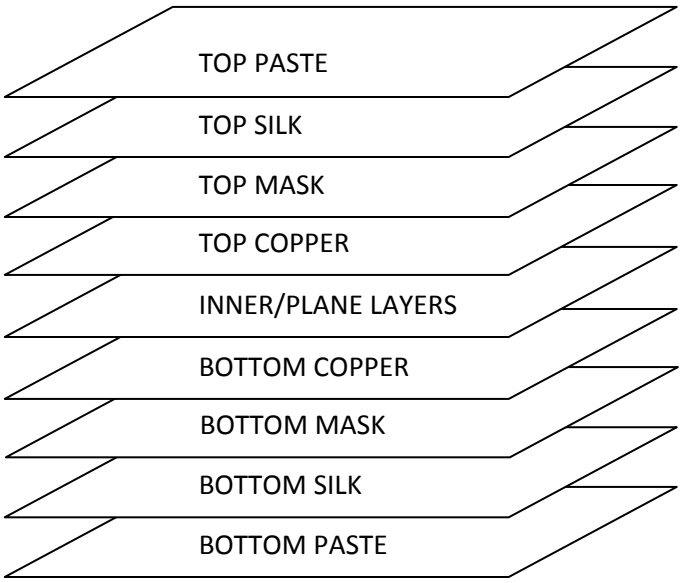
NOTE: If you change the type to an NC layer, a tool table is automatically assigned to it. Its name appears in the layer information in the Navigator.

If you want to change the assignment, right-click on the tool table name, select the Change NC Tool Tables command, and the name of the desired tool table.

Activity: Sort Layers

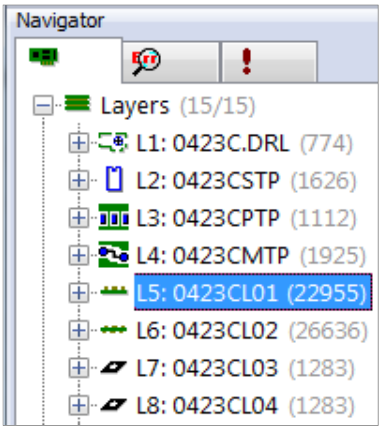
This activity should take about 10 minutes to complete.

TIP: A suggested method for determining how to sort your layers is to order them as if you were looking through the PCB from the top down. So a typical PCB you would use the following sequence to sort your layers:



From there you can sort your extra layers such as Drills, Drawings, and External Nets in any order you wish.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. Under the Layers list in the Navigator, **left click on the layer** you wish to move, and **continue to hold** down the mouse button.
3. **Move (drag)** the mouse cursor until it is over **the layer** (in the Layers list) to which you wish to move the layer.
4. **Release** the mouse button.



All other layers are reordered, as necessary, so that no empty layers are created and no duplicate layer numbers exist.

5. **Repeat steps 2 through 4** to resort the remaining layers.
6. **Save** the file**.**

Activity: Create a Border Layer

When you receive Gerber files it is quite likely that you will not receive a separate layer that only represents the circuit outline or Border. For both DRC/MRC and Advanced Panelization, as well as export to intelligent formats, it is important to have a Border layer with a clean outline.

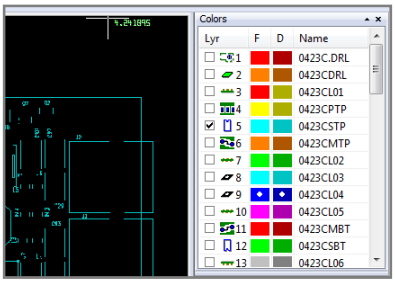
The activity for this section will be to follow the instructions below with the IPC-Demo job using the circuit outline from the Top Silkscreen layer as your master for creating a Border layer.

This activity should take about 10 minutes to complete.

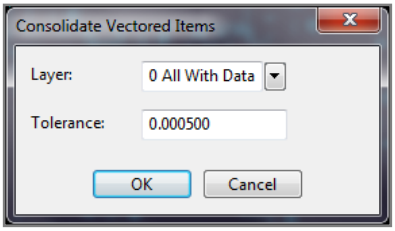
1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.

Consolidate Vectored Items

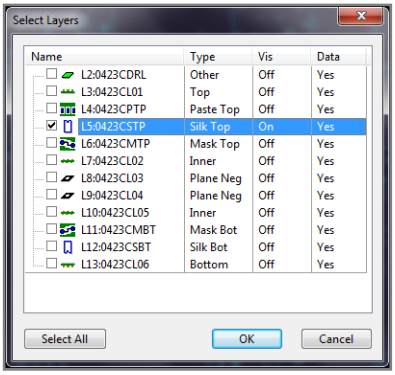
This command converts contiguous lines made of multiple drawn segments into a minimum set of polylines. Items included are draws, arcs, and polylines. Very often your circuit outline is represented by a set of drawn lines on a layer such as the Silkscreen. By converting these drawn lines into a polyline, you can make the selection process for copying this data to a new layer very easy.



2. In the Colors bar **turn on the layers you wish to be processed** (silkscreen in this example), and turn off any layers that you do not want processed.
3. Select the **Tools > Convert > Consolidate Vectored Items** command from the menus.
4. In the Consolidate Vectored Items dialog box, click on the **down arrow** to the right of the Layer textbox.



In the Select Layers dialog box,



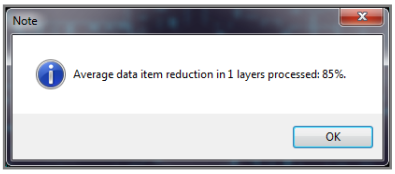
5. **Deselect all layers except for the one** you wish to be processed.
6. Click on the **OK** button.

Back in the Consolidate Vectored Items dialog box:

7. Enter the **Tolerance of 0.0005**.
8. Click on the OK button.

The system will tell you what was processed.

9. In the Note box click on the **OK** button.

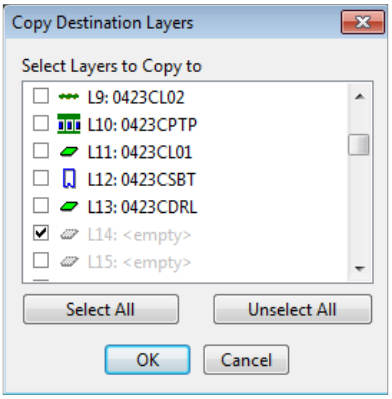


Create Border Layer

10. With the Silkscreen layer still on in the Colors bar, go to the **Edit** menu and select the **Copy** command, **OR** use the toolbar and click on the **Copy icon** .

11. **Click the Destination Layers tab** Destination Layers... at the bottom left corner of your VisualCAM window.

NOTE: If you do not see the Destination Layers tab, go to **View > Toolbars** and make sure the **Status bar** is enabled.

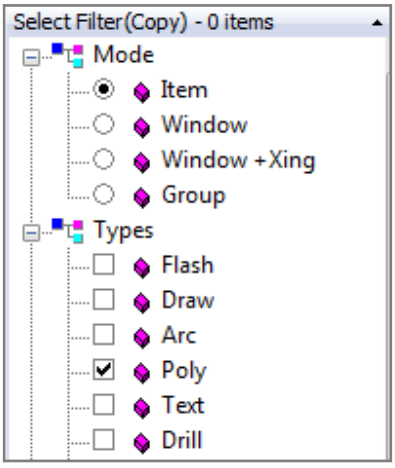


In the Copy Destination Layers dialog box:

12. Select the next available **<empty> layer**.

13. Click on the **OK** button.

In the **Select Filter (Copy)** bar:



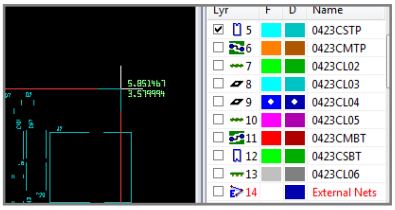
14. Change the Mode to **Item**.

15. Make sure Type **Poly** is checked.

16. In the workspace **left click on the border**.

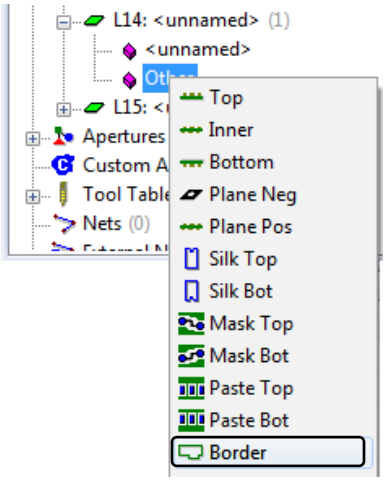
The prompt at the lower-left corner of your workspace should say, "**Copy Window: enter copy from point... (<End> for in-place copy)**."

17. **Press the END key on your keyboard** to copy the data in the same location on the new destination layer you chose.



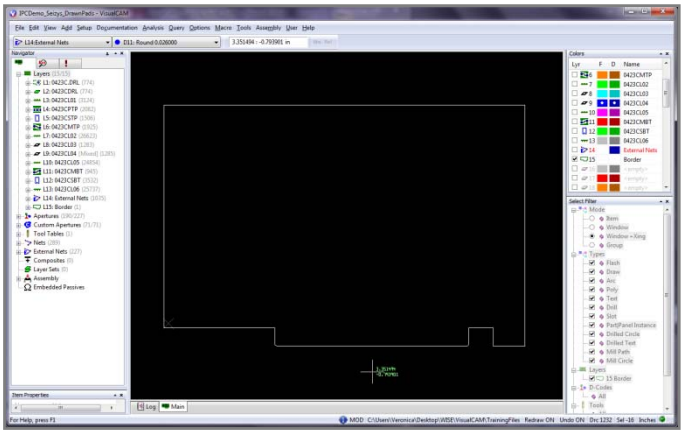
18. In the Navigator window **click on the + by the new layer**.

19. **Right click** on the Layer Type Other and **choose Border**.



20. Right click on <unnamed> and choose **Rename** to name the layer so it is easily identified, for example Border or Outline.
21. In the Colors bar **turn off all layers except the one you just renamed**.
22. **Save** the file.

In the workspace you should now see just the outline on the Border layer.



Review Questions

True or False; By default, all layers are identified with the type “Other”, except for the Border layer.

_____

A tool table is automatically assigned to what layer type?

_____

What is the suggested method for sorting layers?

_____
_____
_____
_____
_____
_____
_____
_____

Creating NC Drill Data from Gerber Layers

Purpose

In this lesson you will learn how to convert a Gerber layer to an NC Drill layer.

Lesson Objectives

In this lesson you will do the following:

- Create an NC Drill layer from a Gerber layer.

Estimated Completion Time

Approximately 10 minutes.

Activity: Create NC Drill Data from Gerber Layers

When you receive Gerber files for a design, sometimes you will receive a Gerber file that represents the Drill layer rather than an actual NC Drill file with a Tool List. It is required to have an NC Drill layer that is recognized as such in order to identify padstacks and make an accurate netlist.

Using the **Tools > Convert > Gerber to NC** command, VisualCAM will automatically convert this layer to an NC Drill layer and create an accompanying Tool list for you.

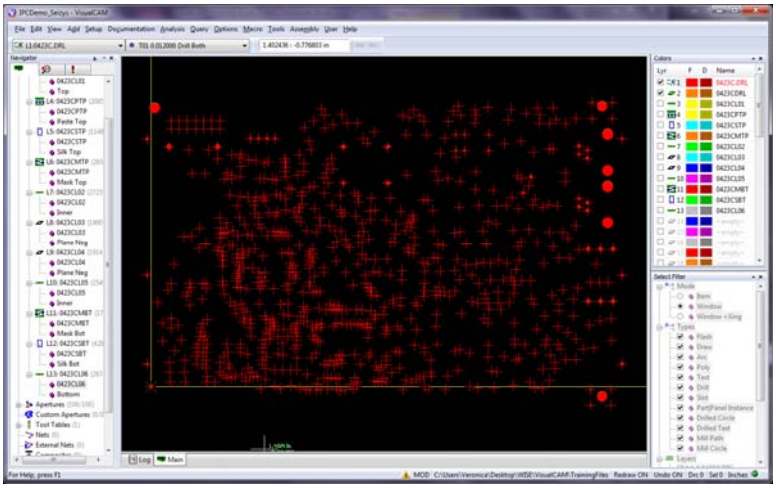
So now it's time to deal with that "0423CDRL" layer we have been ignoring so far. Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

This activity should take about 10 minutes to complete.

NOTE: *Important*! This is the only way to convert Gerber data to Drill data. Simply changing the layer "Type" to Drill does not convert the data.

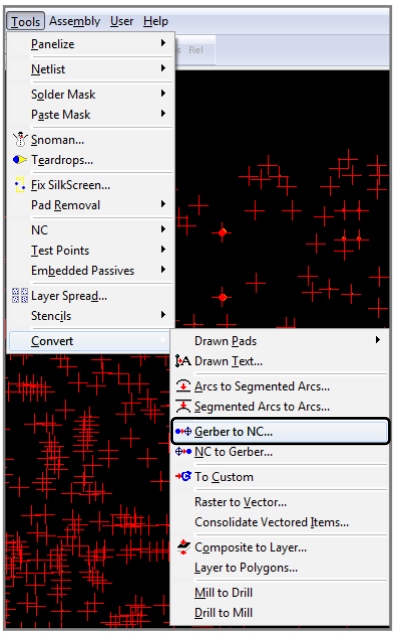
1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. In the Colors Bar turn off all layers except for layer **0423CDRL**.
3. Zoom your workspace out to the **View All** mode.

NOTE: Zooming is not a requirement, but will make it easier for you to see the conversion take place after executing the commands below.



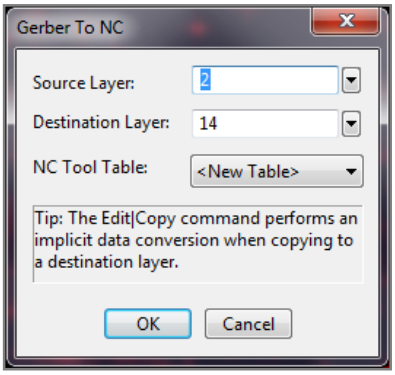
4. On the **Tools** menu select **Convert > Gerber to NC** command,

   OR **click on the icon**  on your toolbar.



In the Gerber To NC dialog box:

5. **Select** the **0423CDRL** layer for the **Source Layer**.
6. **Select** the next available **<Empty>** layer for the **Destination Layer**.
7. **Leave the NC Tool Table** at its default value of **<New Table>**.
8. Click on the **OK** button to execute the command.



A new NC Drill layer has been created on the Destination Layer you selected.

9. **To view the results**, turn this layer on using the Colors Bar.

NOTE: Delete the new NC Drill layer we just created as well as the Gerber 0423CDRL layer. We will be using the 0423C.DRL NC Drill layer we imported for the remainder of this job.

Review Questions

What layers are required to process an accurate netlist?

_____

Once imported, VisualCAM identifies all non-Drill formats as what kind of data?

_____

True or False; Along with using the Convert > Gerber to NC command, you can also convert Gerber data to Drill data by changing the layer type to Drill in the Navigator.

_____

Data Alignment

Purpose

In this lesson you will learn how to use the Align Layers, Edit Origin and Snap Pads commands to align all your data together so that VisualCAM is able to accurately identify parts that are thru-hole technology, as well as finding all the correct nets across the various layers.

Lesson Objectives

In this lesson you will do the following:

- Use the Align Layers command.
- Use the Edit Origin command.
- Use the Snap Pads command.

Estimated Completion Time

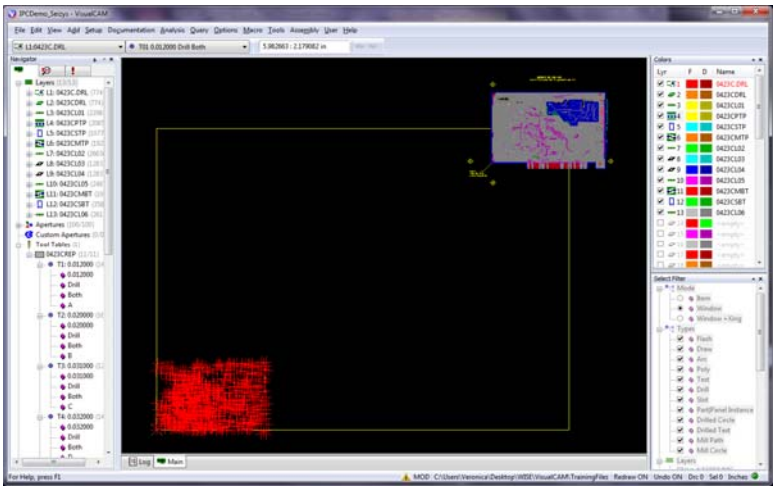
Approximately 15 minutes.

Activity: Align Layers

This command aligns layers for proper layer registration when multiple layers are viewed simultaneously. Proper layer alignment is also crucial to the successful creation of a **netlist** (a file containing groups of X:Y locations for pads that are connected by traces.)

If you wish to verify that your layers are correctly aligned, you can check for layer-to-layer registration using the **Analysis > DRC/MRC** command.

In this activity you will be aligning the layers of the **IPC-Demo** job. With all layers turned on, your job should look like the image below. Note that the drill layer is not aligned with the rest of the Gerber layers. We will correct this problem in this activity.

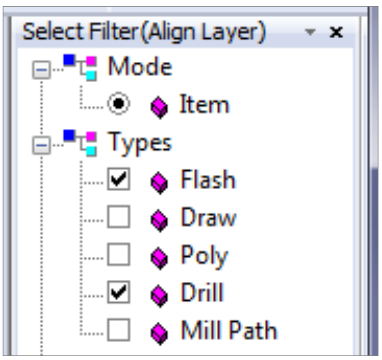


NOTE: When executing commands such as Align Layers, it is often necessary to zoom in and out of certain areas of the design while the command is still active. Use the **"+"** and **"-"** keys on your keyboard to zoom in and out as clicking on any of the zoom icons while executing a command will cancel it.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working ad saving your changes.
2. **Determine** which layer in the design will be the **Master Layer**.

TIP: When selecting a **Master Layer**—the layer all others layers should be aligned with—you should select a layer that is already aligned with the majority of the other layers of your design. Since layer alignment has to be done one layer at time, it is easier to align a few layers with the vast majority of other layers than to align numerous layers to the few that are misaligned.

3. Determine the **Master layer reference point**.



WARNING: Arcs cannot be used as reference points to align layers.

4. Select **Edit > Align Layers** from the menus

   OR click the **Align Layers** icon on the toolbar.

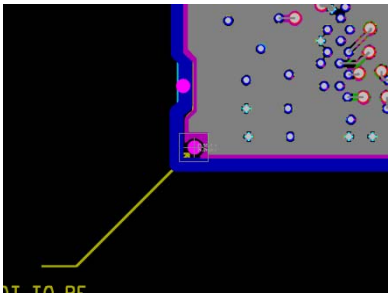
5. In the **Select Filter (Align Layer)** bar, uncheck all the **Types** except for **Flash** and **Drill.**

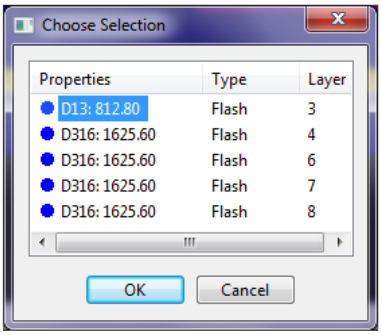
NOTES: Use of the **Selection Filter** may be helpful here. You may also use the zoom in/out and pan hotkeys to make it easier to find the master reference and corresponding items.

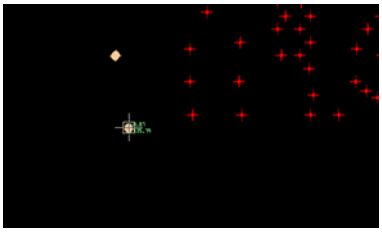
If you wish to select a new base item, press the **Esc** key and choose a new master reference point.

TIP: Choosing objects in one of the far corners of your circuit is the easiest way to ensure that you will be selecting the same reference points for both the master layer and the layer to be aligned.

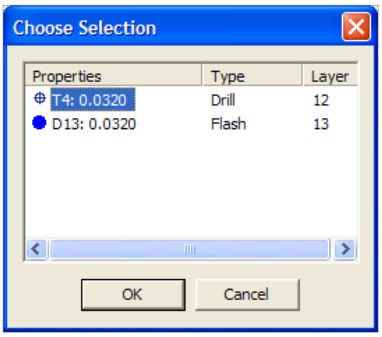
6. Keeping your mouse pointer over the text just below the lower-left corner of circuit, press the "+" key on your Num-Pad several times till you can easily see corner tooling hole as shown here.
7. Click on the tooling hole. This will be the **master reference point**.
8. In the Choose Selection dialog box click on the **OK** button.





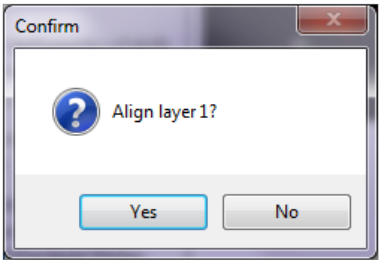


9. Press the "-" key on your Num-Pad several times till you can see the drill layers again.
10. Keeping your mouse pointer over the lower-left corner of drill layer, press the "+" key on your Num-Pad several times till you can easily make out the different drill points.
11. Click on the lower-left most drill point. This will be the reference point on the layer to be aligned.

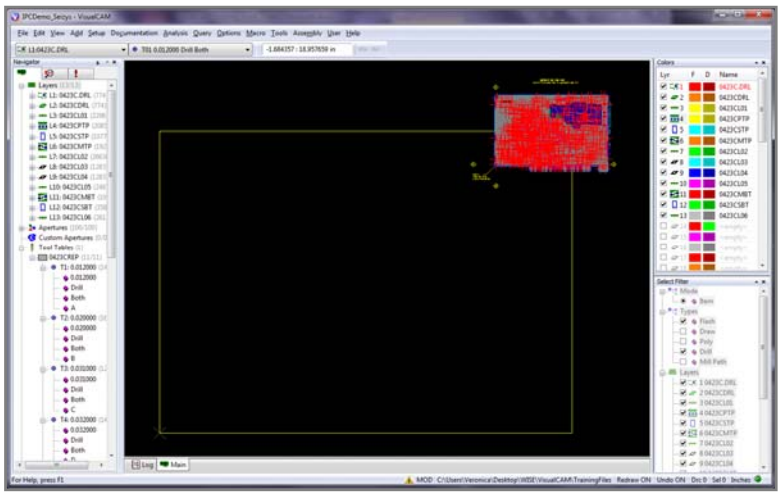


Since the tooling hole on the master layer is also the lower-left most hole in the drill layer, this makes it certain that you are choosing the same reference point on both the master layer and the layer to be aligned.

12. In the Choose Selection dialog box, select the first layer that is misaligned (the Drill object.)
13. Click on the **OK** button.
14. When you are prompted to Confirm the alignment of layer 1, click on the **Yes** button.



15. With the Align Layer function still running, **repeat steps 11 through 14**, selecting the same reference point on the other misaligned layer.
16. Press the **Esc** key twice to exit the function.
17. **Save** the file.
18. **Zoom out** to see the aligned layers.



Activity: Change the Origin

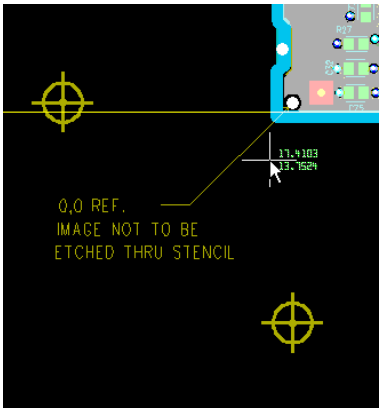
This command allows you to relocate the origin (0:0 point) of the database. Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

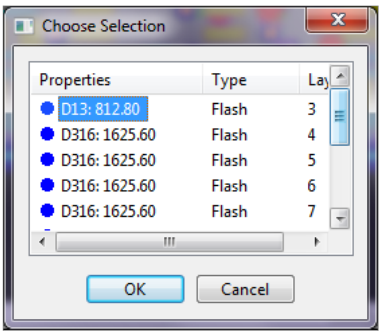
This activity should take approximately 5 minutes to complete.

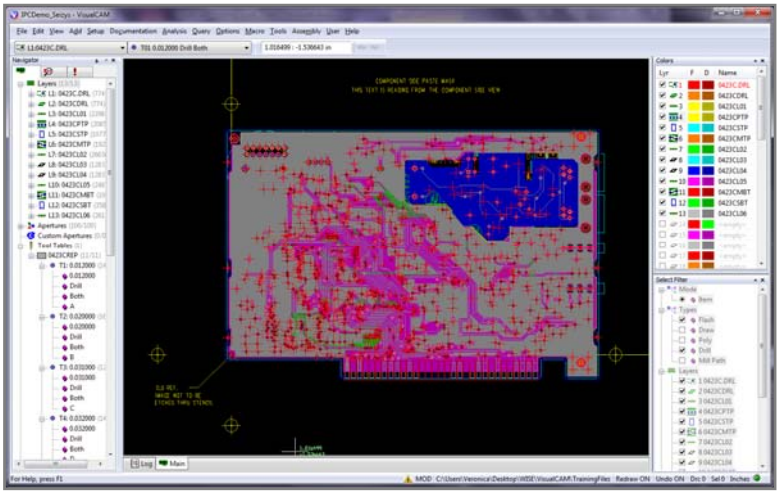
NOTE: The text on the Top Paste layer indicates a tooling hole as the intended origin for the circuit. This is why it can be important not to remove text from non-electrical/physical layers, it can be very helpful.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. In the menus select the **Edit > Origin** command,

   OR click on the **Change Origin icon** .
3. In the design **hover** the mouse pointer **over the new origin**.
4. Press the "+" key on your keyboard Num-Pad or the "I" several times to **zoom in**.
5. **Hover** the mouse pointer **over the tooling hole**.
6. **Press the Home key** on the keyboard to snap to the center of the tooling hole.
7. In the Choose Selection dialog box, **select the object** intended as the new origin.
8. Click on the **OK** button.
9. **Press the Esc key** to exit the function when finished.
10. **Save** the file**.**
11. **Press the F4 key** to zoom out and center the layers in the workspace. It should now look like the image below.







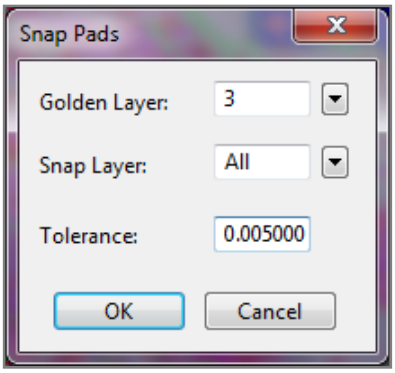
Activity: Snap Pads

Unlike the Align Layers command, which shifts the entire layer based on an individual reference point, the Snap Pads command examines each individual item on a layer and aligns it to the nearest item on the "golden" layer. This is useful if you need to cleanup low resolution artwork that does not line up properly from layer to layer. Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

This activity should take approximately 5 minutes to complete.

TIP: Only visible layers are affected by the Snap Pads command. Only flashed pads are affected by the snap pads command. If your pads are not already flashed, return to this step again after converting drawn pads (*see Draws to Flash Conversion, page 77.*)

The most common problem is that your drill points will be slightly offset, (even after performing the Edit > Align command), from the pads they are paired with on your copper layers. The recommended process therefore is to turn off all non-electrical/physical layers and use the copper layers to snap your drill points into position.



1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Edit > Snap Pads** command. The Snap Pads dialog box appears.
3. Select the **Top** layer as your **Golden Layer**.

   These pad or drill locations will be used as a "master" reference to align items on the **Snap Layer(s)**.
4. Select **All** layers as the **Snap Layer**.

   These layers contain pads or drills to be aligned with those on the Golden Layer.
5. In the **Tolerance** box, **enter 0.005**.

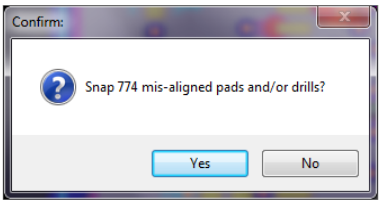
   This is the distance that the items on the Snap Layer can deviate from the items on the Golden Layer to be considered for alignment. Items within this distance are aligned; items outside this distance are ignored.
6. Click on the **OK** button.

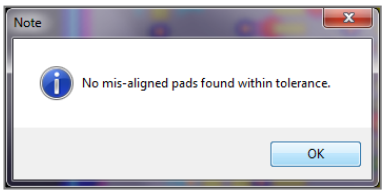
NOTE: The recommended tolerance setting is between 5-10 mils. If you use a number larger than 10 mils, you are risking that the command will snap objects together that are meant to be separate.

7. When prompted **to confirm the command**, click on the **Yes** button.

   If no mis-aligned pads were found within the tolerance, click on the **OK** button.

8. **Repeat the steps if necessary** to align the items within a different tolerance.
9. **Save** the file.





Review Questions

What command will check for layer-to-layer registration?

_____

What two conditions must exist for a layer to be affected by the Snap Pads command?

_____

_____

_____

_____

What is the recommended setting for the Tolerance field in the Snap Pads dialog?

_____

Cleaning Up Your Data

Purpose

In this lesson you will learn how to remove all unnecessary data from the Gerber layers of the job.

Lesson Objectives

In this lesson you will do the following:

- Delete unnecessary data from all layers.
- Remove duplicate pads (stacked pads).
- Remove duplicate data.

Estimated Completion Time

Approximately 20 minutes.

Activity: Delete Unnecessary Data from All Layers

In this activity you will be removing all unnecessary data from the layers of the IPC-Demo job that is outside of the Border, such as layer labels etc. Especially on electrical layers, unnecessary data will cause invalid nets when you generate a netlist.

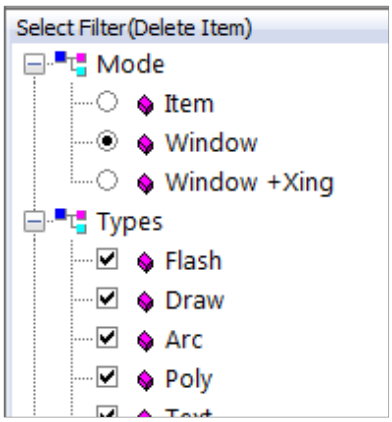
This activity should take about 10 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. In the **Colors** bar right-click on any layer and select **All On**.
3. **Turn off** the Paste Layer **0423CPTP.**

TIP: You will often have additional layers included with your Gerber data, such as paste or drawing layers, which are not a part of the actual electrical or physical layers of the circuit. It is neither necessary nor recommended to remove information from these layers as this data may prove useful later on. You should turn these layers off before deleting unnecessary data.

4. Select **Edit > Delete** from the menus,

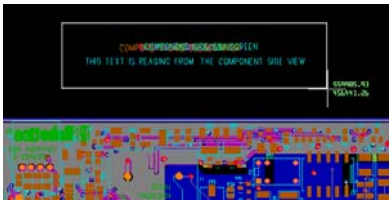
   OR click the **Delete** icon on the toolbar.

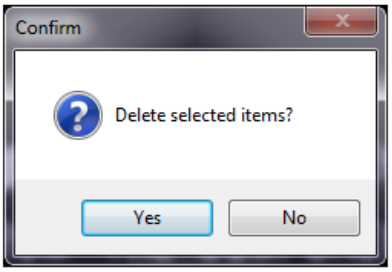


In the Select Filter (Delete Item) window:

5. Set the **Mode** to **Window.**
6. Verify that **all Types are checked.**

7. **Click and drag a selection window** around groups of any obvious data you want to remove in the workspace. Be careful not to select any of the circuit area.
8. Click **Yes** to confirm each deletion.
9. **Save** the file.



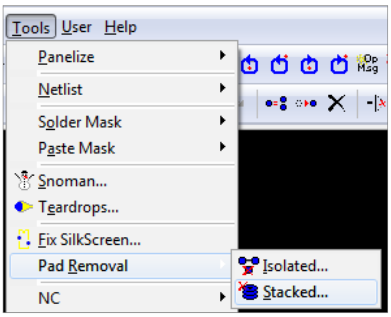


Activity: Remove Duplicate Pads (Stacked Pads)

This command removes pads that are stacked on top of each other. If the data has drawn pads instead of flashes, run the Drawn Pad Conversion first (*see Automatically Convert Drawn Pads to Flashes, page 79.*)

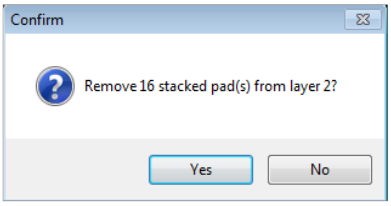
This activity should take about 5 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. In the **Tools** menu select **Pad Removal > Stacked.**
3. Choose **all layers**.
4. **Set the tolerance** to "**0.0005**".
5. Click on the **OK** button.



You will be prompted to confirm the delete for each layer with a message stating that duplicates were found along with the quantity of stacked pads found.

6. Click on the **Yes** button to confirm the delete.
7. Click on the **OK** button to complete command.
8. **Save** the file.



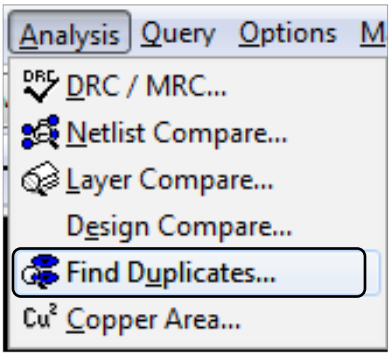
NOTE: After deleting all obvious unnecessary data it is a good habit to review each electrical layer individually and look for any text or other drawn objects that are not a part of the circuitry, such as a board outline or layer windows and delete these as well.

Activity: Remove Duplicate Data

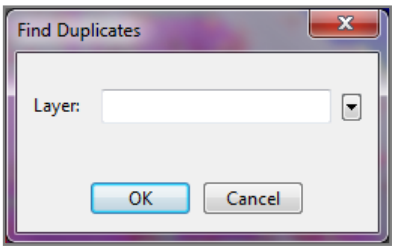
This command finds all occurrences of duplicate data on the same layer and the exact same location and properties. View, highlight and fix errors using the Navigator.

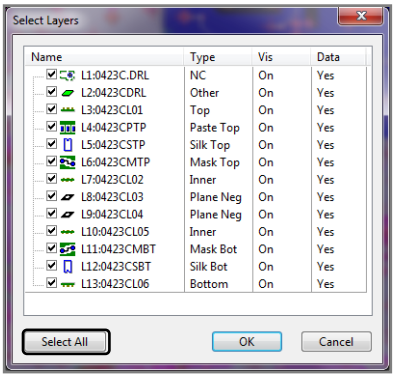
This activity should take about 5 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working and saving your changes.
2. In the **Analysis** menu select **Find Duplicates**.

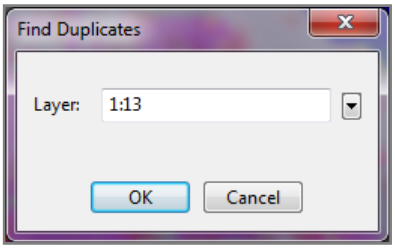


3. In the Find Duplicates dialog box, click on the **down arrow** to the right of the Layer textbox.
4. In the Select Layers dialog box, click on the **Select All** button located at the bottom left corner.
5. Click on the **OK** button.

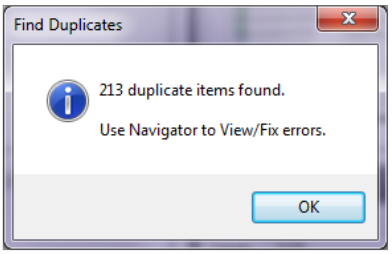




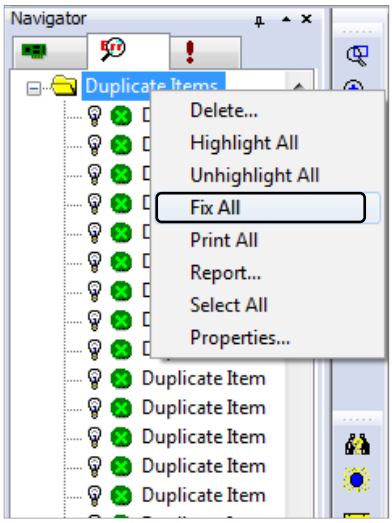
6. Back in the Find Duplicates dialog box click on the **OK** button.



You will be prompted to respond to a message stating how many duplicate items were found.



7. Click on the **OK** button.



8. In the Navigator click on the **Analysis** tab.
9. Right click the **Duplicate Items** folder and choose **Fix All**.
10. **Save** the file.

Review Questions

Why is it necessary to delete unnecessary data from electrical layers?

_____

True or False; It is necessary to convert draws to flashes before removing stacked pads.

_____

True or False; In the Analysis > Find Duplicates command, all occurrences of duplicate data on different layers will be found.

_____

Draws to Flash Conversion

Purpose

In this lesson you will learn the functions and processes used to convert drawn objects in the Gerber data to flashes. The DRC/MRC processes will only recognize flashed apertures as pads so it is required to run this process on the Copper layers as a preliminary step.

Lesson Objectives

In this lesson you will do the following:

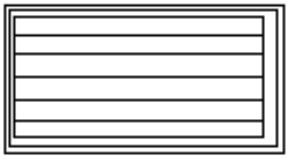
- Automatically convert drawn pads to flashes.
- Convert draws to flashes by manually selecting draws.

Estimated Completion Time

Approximately 20 minutes.

Converting Drawn Pads to Flashes

Many CAD systems still output rectangular pads, such as surface mount pads, using multiple draws to fill in the rectangle rather than a more efficient single flash. This results in bloated Gerber files and increased processing times. Also, it is virtually impossible for high-level CAM functions, such as DRC, to recognize these drawn entities as a pad rather than just a bunch of traces. The example below shows the difference between a typical drawn pad and a comparable flash.



The drawn pad shown on the left requires 27 separate Gerber commands to accomplish what one Gerber flash can accomplish. As you can see, if you have 2000 of these drawn flashes, you will have a Gerber file with at least 54,000 lines when flashes could accomplish the same thing in only 2,000!

It is recommended that the conversion of drawn pads to flashes be the first thing done to your designs. This will usually ensure complete and trouble-free conversion. Also, you must convert all drawn pads to flashes before generating a netlist (a file containing groups of X:Y locations for pads that are connected by traces) or running most other CAM functions.

Activity: Automatically Convert Drawn Pads to Flashes

The automatic drawn pad converter requires a paste or solder mask layer with a corresponding top or bottom layer. For example, a "Mask Top" or "Paste Top" layer requires a "Top" layer to exist. The mask layer must have data on it, but it is not required that the mask data be flashed. The openings on the mask layer are used to look for drawn pads to be converted to flashes.

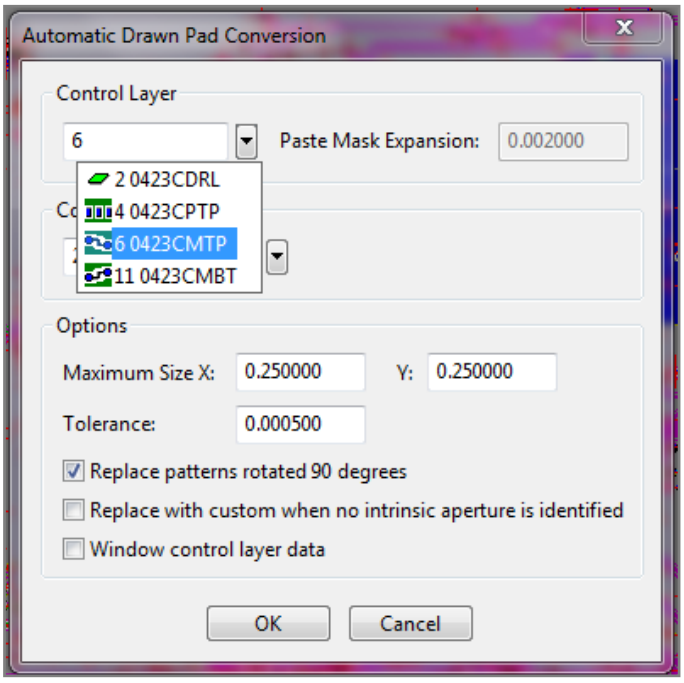
Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

This activity should take approximately 5 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. From the menus select the **Tools > Convert > Drawn Pads > Automatic** command,

   OR click on the **Automatic Convert Drawn Pads icon** on the toolbar.

In the Automatic Drawn Pad Conversion dialog box:



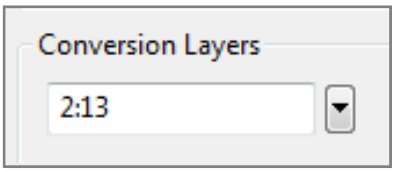
3. From the **Control Layer** drop-down list, select **0423CMTP.**

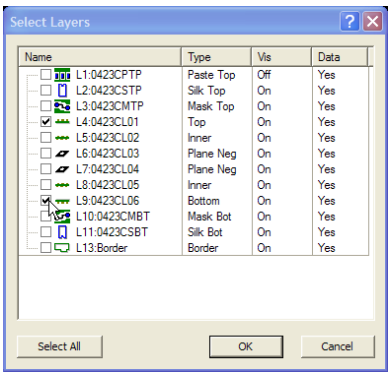
   This is the solder or paste mask layer that corresponds with the top or bottom layer on which you wish to convert pads. To prevent an incorrect layer from being specified, you cannot type a number directly into the box.

4. If the selected Control Layer is a paste mask layer, the Paste Mask Expansion textbox to the right of the Control Layer list allows you to enter a value by which paste mask openings will be oversized.

   Over sizing the openings may be required, because often paste mask openings are smaller than the corresponding pads on the top/bottom layer.

5. Drop down the Conversion Layers list to specify one or more layers that contain the drawn pads you wish to convert.
6. In the Select Layers dialog box click on the **Select All** button to select all layers. When the button changes to **Un-Select All, click it again to deselect all layers.**
7. Now click to select only the **Top** and **Bottom** layers. The dialog box should look like the image to the right.
8. Click on the **OK** button to exit the Select Layers dialog box.





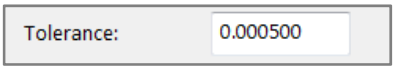
Back in the Automatic Drawn Pad Conversion dialog box:

9. For the Maximum Size, leave the defaults of **0.25**" for this exercise.

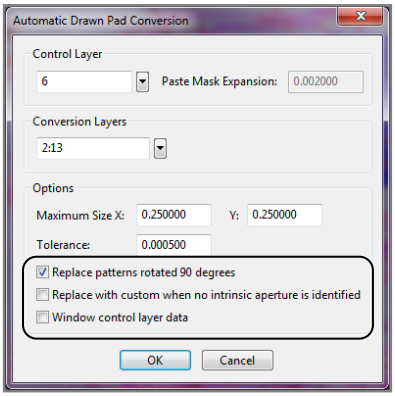
   The Maximum Size in X and Y dimensions is how large a group of draws may be and still be considered for conversion to a flash. If a group of draws exceeds the maximum size, they will not be converted.

10. For the Tolerance, leave the default of **0.0005** for this exercise.

    The Tolerance controls how close the replacement aperture must be in size to the original draws, to be considered a valid replacement. If no replacement is found, then an error will be logged.



11. Select the **Replace patterns rotated 90 degrees** option if you wish to have items of the same shape, which are rotated 90 degrees, converted also.



12. Select the **Replace with custom when no intrinsic aperture is identified** option to allow custom apertures to be created and used as replacements when no matching intrinsic aperture types are identified.

    If this option is not selected and no appropriate intrinsic replacement aperture is found to replace a drawn pad, then an error is logged. Without selecting this option, automatic drawn pad conversion will only find and convert round, oblong, rectangle, and square shapes.

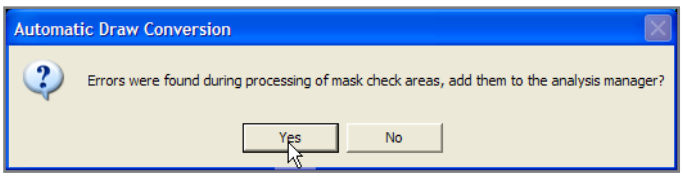
    "No Aperture" errors are generated for shapes that are not recognized; you will need to use **Tools > Convert > Drawn Pads > Select Draws** to convert them.

TIP: Typically you will look for your largest aperture and measure its dimensions to decide these values for each job. However, if it is too large, greater than 1.0” in either dimension, you should not use these values as you may risk converting objects that you don’t want to be flashed.

13. Select the **Window Control Layer Data** option if you want to convert only pads in a specific area. This is useful for designs that have text blocks or other data outside the actual board area, as excluding that extraneous data will speed up the process.
14. Click on the **OK** button to complete the command.

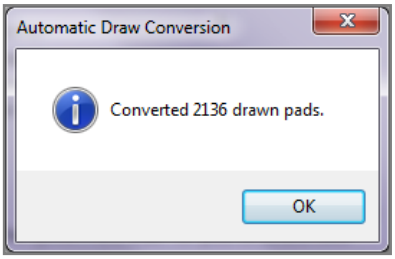
NOTE: **If you selected the Window Control Layer Data option**, the system will wait for you to select two points to define the area of the design to process before continuing.

15. If you receive an Automatic Draw Conversion message stating that errors were found, click the **Yes** button to add these errors to the analysis manager.



If no appropriate replacement aperture is found to replace a drawn pad, then an error is logged. An error is also logged if a mask opening exists, but no drawn data was found in the corresponding location on the conversion layer.

When the process is complete, a message is displayed indicating the number of converted pads.



16. Click on the **OK** button to continue.
17. If you selected the Window Control Layer Data option**, choose another area to process,**

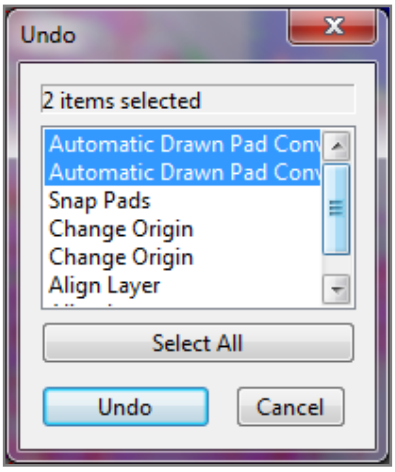
    OR press the **Esc** key on the keyboard to end the conversion process.
18. **Repeat** this process again for the IPC-Demo job, but select layer 0423CMBT as the control layer this time.

Activity: Convert Draws to Flashes by Manually Selecting Draws

This function converts drawn pads to flashes based upon data that you manually select for conversion. Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

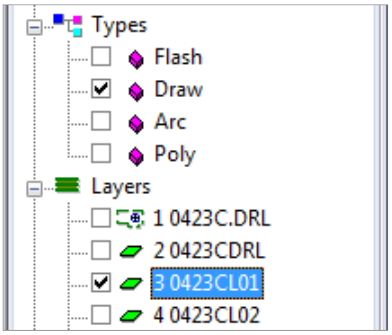
This activity should take approximately 15 minutes to complete.

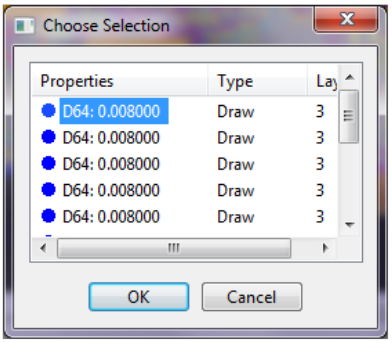
1. Use the **Edit > Undo** command to **undo the last two automatic drawn pad conversions**.
2. From the menus select the **Tools > Convert > Drawn Pads > Select Draws** command, **or** click on the **Convert Drawn Pads button** on the toolbar.



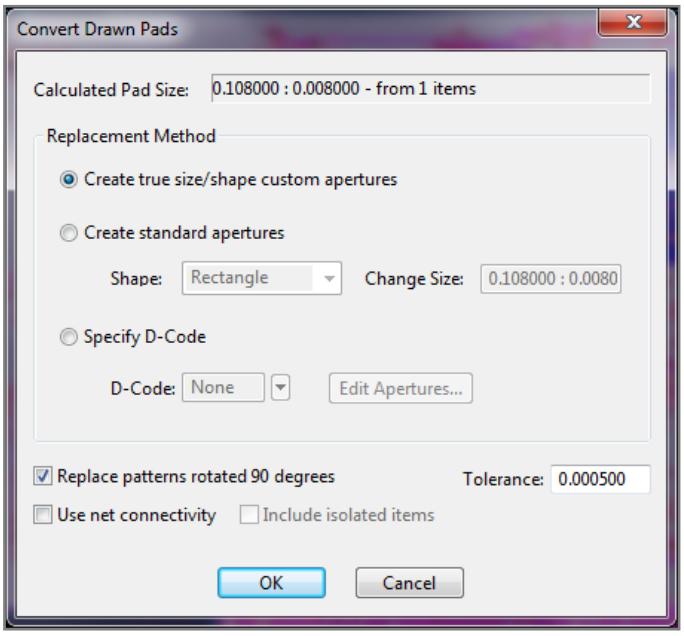
In the Select Filter (Convert Draw Pads) bar:

3. Set the **Type** only to **Draw.**
4. Click on layer **0423CL01.**
5. **Left click anywhere** on the design.
6. In the Choose Selection dialog box, **left click on the first item**.
7. Left click on the **OK** button.





In the **Convert Drawn Pads** dialog box:



Calculated Pad Size: The system locates all drawn pads identical to the one you selected. The Calculate Pad Size area displays information on the selected pads to be converted.

8. **Select the replacement method**:

   a. **Create true size/shape custom apertures**. *This is what you will use for the IPC-Demo job.*

      This option is used to have selected items converted to a single custom aperture—special user-created shapes that are normally used as flashes and can be created or modified in the Custom Aperture Editor—with the same appearance.

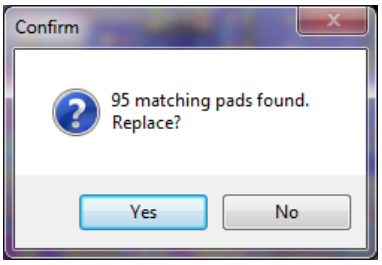
   b. **Create standard apertures**

      Select this option if you wish to have selected items converted to one aperture of the specified shape and size. Select the Shape and Size of the aperture you wish to use. If an aperture of that size and shape already exists, it is used. Otherwise, a new aperture is created. If you also select the “Replace patterns rotated 90 degrees” option (see step 2), then up to 3 apertures may be created.

c. **Specify D-code**

Select this option if you wish to have the selected items converted to an existing aperture. Specify the new D-Code number for the pad created by the conversion. To view the Aperture Table, where you can create or edit apertures as necessary, click the Edit Apertures button.

9. **Select the Replace patterns rotated 90 degrees option** to have items of the same shape, which are rotated 90 degrees, converted also. *This is what you will use for the IPC-Demo job*.
10. Select the Use net connectivity option if you wish to only convert pads that are on the same net as the example pad that you selected. *DO NOT use this option for the IPC-Demo job.*
11. Select the Include isolated items option if you wish to have items that are not part of a net (isolated pads) checked for possible conversion.
12. **Set the Tolerance to 0.0005** inches to increase the system's ability to recognize pads that are the same size, and account for minor variances in drawn data. By doing so, you relax the criteria for determining which pads are the same sizes; pads whose sizes are within the tolerance value are determined to be the same.
13. Click on the **OK** button.
14. When you are prompted to confirm the replacement of matching pads, click on the **Yes** button.



15. With the command still active, **repeat** steps 5 through 7 and 13 on the remaining items.
16. Press the **Esc** key on the keyboard to exit the command function.
17. **Save** the file.

Review Questions

Why is it necessary to convert drawn pads to flashes?

_____
_____
_____
_____
_____
_____

What are the layer requirements for using the Automatic Drawn Pad conversion?

_____
_____
_____
_____
_____
_____

What happens if no appropriate replacement aperture is found to replace a drawn pad?

_____
_____
_____
_____
_____
_____

In the Convert Drawn Pads dialog box, if you choose the “Create standard apertures” and “Replace patterns rotated 90 degrees” options, up to how many apertures will be created?

_____

In the Draws to Flash Conversion, what increases the system's ability to recognize pads that are the same size, and accounts for minor variances in drawn data?

_____
_____
_____
_____

Generating a Netlist

Purpose

In this lesson you will learn the **Tools > Netlist > Generate** command, which processes all conductive layers and creates a netlist (a file containing groups of X:Y locations for pads that are connected by traces) that becomes part of the internal database. The netlist may then be used by other functions that require a netlist, such as Netlist Compare and DRC/DFM Analysis.

Lesson Objectives

In this lesson you will do the following:

- Generate a netlist.

Estimated Completion Time

Approximately 10 minutes.

Activity: Generate a Netlist

This activity should take approximately 10 minutes to complete.

TIP: We recommend that any documentation for the board be done on a separate Drawing layer (a layer used to add information that you do not wish to place on your design layers. This includes redline information, mechanical drawings and diagrams complete with "intelligent" drawing primitives, which automatically update themselves.) If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing.

Prior to Generating a Netlist

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Make sure that **each layer is "tagged" with the correct layer type** (*see Change the Layer Type, page 53*.) This is required so that the system understands what each layer is.
3. Make sure **all layers are properly aligned** (*see Align Layers, page 64*.)
4. Make sure that any **extraneous data** on your electrical layers that is outside the perimeter of the board, such as title blocks, targets, crop marks, etc. is **removed** (*see Delete Unnecessary Data from All Layers, page 72*.)
5. Make sure all **redundant pads** are **removed** (*see Remove Duplicate Pads (Stacked Pads), page73*.)
6. Make sure all **drawn pads** in your database have been **converted to flashes** (*see Draws to Flash Conversion, page 77*.)
7. If you wish to exclude any data from the netlist generation process, **create a Select Group (*optional*.)** Once you have created a select group, the group is added as a Mode in the Selection Filter for use future Edit functions.

   TIP: If you want to add the majority of the items in a database to the Select Group, you can use this function to first create a Select Group of the items you ultimately wish to exclude. Then select the Edit > Select > Invert command to reverse the selection, and create your desired Select Group.

   a. Select the **Edit > Select > New Group** command. If you already had a Select Group defined, you are asked if you wish to clear the current group in favor of a new group. Doing so does not delete or otherwise alter any data.

   b. Using the Selection Filter as necessary, add any desired items to your Select Group.

      TIP: Hotkeys are available to help you make your selections. For example, you can snap your cursor to the center of an item using the Home hotkey.

   c. When you are finished adding items, press the **Esc** key to exit the function. Your selections are automatically saved, and will remain highlighted in the workspace. (To turn off the highlighting, select the View > Highlights command).

   d. **If you later wish to add items to the Select Group**, use the Edit > Select > Add To command. **If you wish to remove items from the group**, use the Edit > Select > Remove From command.

Generate the Netlist

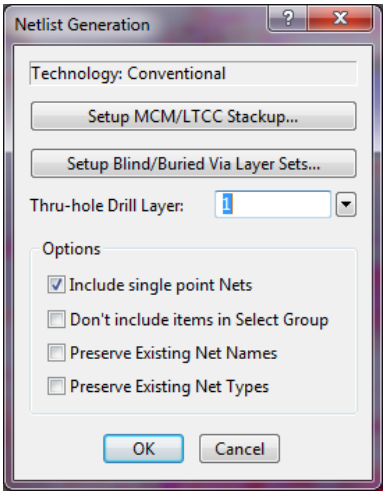
1. Select the **Tools > Netlist > Generate** command.

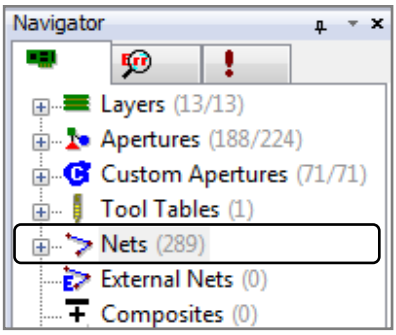
In the Netlist Generation dialog box:

NOTE: By default, the system will generate a "Conventional" netlist, and assume you do not use any MCM/LTCC technology, or Blind/Buried vias. If you are using MCM/LTCC technology, click the Setup MCM/LTCC Stackup button. This opens the MCM/LTCC Setup dialog box, where you define your layer stackup. If you have Blind/Buried vias, click the Setup Blind/Buried Via Layer Sets button. This opens the Layer Set Setup For Blind/Buried Vias dialog box, where you define the pairing of NC layers to circuit layers by creating layer sets.

2. The system will automatically display your NC layer in the **Thru-hole Drill Layer** box. An NC layer with thru-hole drill information is necessary to determine layer to layer connectivity, as well as direct connections to plane layers.

   If you want to select a different NC layer, click the down-arrow button and select the layer. If you do not supply an NC layer (select "None"), the netlist extract function will use the Aperture Table Type field to determine whether each pad is SMT or not.
3. Select the **Include Single Point Nets** option if you wish to designate all isolated pads, pins, or vias as nets.
4. If you created a Select Group of times to ignore during netlist generation, select the **Don't Include Items In Select Group** option.
5. If you previously generated a netlist, and need to recreate it after modifying some data, you can maintain your existing net name assignments by selecting the **Preserve Existing Net Names** option.
6. Click on the **OK** button to generate the netlist.
7. When the system has finished processing, **save** the file.





NOTES: Netlist information is displayed in the Navigator. To save processing time when you generate a netlist, the Nets list in the Navigator is not populated until you first attempt to access it.

When you save your database to a .vcam file, the netlist information is saved with it. You can use the Tools > Netlist > Save command to save the netlist as a simple ASCII file, export the netlist as an IPC-D-356/A file, or export within Gerber files. To export a netlist within Gerber files, select the Netlist option in the Gerber Export Data Format dialog box. VisualCAM uses the G04 command to embed the netlist within the Gerber files. This will cause the Gerber file to increase slightly in size. It is recommended you do not include a netlist when submitting your files to be plotted, due to their increased size and the possibility that the photoplotter may not properly recognize the G04 command

Review Questions

Before generating a netlist, what should be done to the layers?

_____

What should you do to exclude any data from the netlist generation process?

_____

True or False; By default, the system assumes you are using MCM/LTCC technology and Blind/Buried vias. You need to tell it you are not.

_____

True or False; An NC layer with thru-hole drill information is not necessary to determine layer to layer connectivity.

_____

Netlist Comparison

Graphical Netlist Comparison allows you to visually compare your design data against the original netlist. VisualCAM performs the comparison automatically, and all violations, such as DRC/MRC and DFF violations, are stored internally. This means you can quickly scan through and fix connectivity issues.

As an aid to those who wish to compare a netlist created from Gerber data back to an IPC-D-356, IPC-2581, ODB++, or PADS ASCII database netlist, the **File > Import > Netlist** commands import only the "netlists" from these databases.

Lesson Objectives

In this lesson you will do the following:

- Import an IPC-D-356 netlist.
- Use Netlist Compare to find differences between the CAD netlist and the Gerber data.

Estimated Completion Time

Approximately 10 minutes.

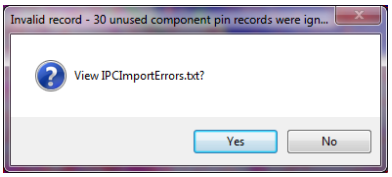
Activity: Compare Netlists

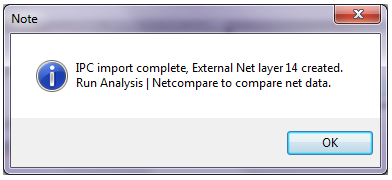
IMPORTANT: Prior to netlist comparison:

- Each layer should be "tagged" with the correct layer type (*see Change the Layer Type, page 53*.)
- All layers should be properly aligned (*see Align Layers, page 64*.)
- Any extraneous data on your electrical layers that is outside the perimeter of the board, such as title blocks, targets, crop marks, etc. should be removed (*see Delete Unnecessary Data from All Layers, page 72*.)
- All redundant pads should be removed (*see Remove Duplicate Pads (Stacked Pads), page73*.)
- All drawn pads in your database should have been converted to flashes (*see Draws to Flash Conversion, page 77*.)
- An internal netlist should have been generated (*see Generate a Netlist, page 88*.)

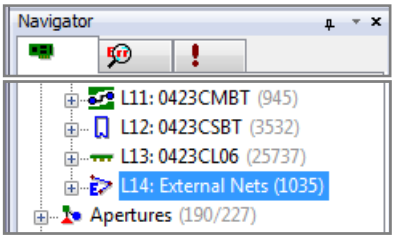
This activity should take approximately 10 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Choose the **File > Import > Netlist > IPC-D-356** command from the menus.
3. In the IPC-D-356 dialog box, select the **…TrainingFiles/IPC-Demo/0423C.IPC** (external) netlist.
4. Click on the **Open** button.
5. If you receive an **Invalid record** message box, click the **Yes** button to view the text file.
6. Click on the **OK** button when prompted to respond to the **IPC import complete** note.

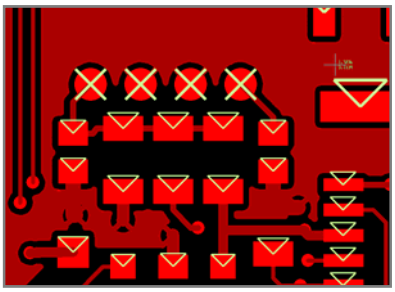




After you import the netlist, a new External Net layer is created. Viewed by itself, this layer appears to contain no data. However, this layer contains the external netlist information, and links the external net points to the appropriate top/inner/bottom layer net test points.



7. To view the points, in the Colors bar turn on the associated top, bottom, and inner layers, and the external net layer. The net points are displayed in four ways for easy reference.
   a. Downward pointing triangles represent test points accessible from the top side of the board.
   b. Upward pointing triangles represent test points accessible from the bottom side of the board.
   c. X's represent test points which are accessible from both sides.
   d. Triangles pointing to the side represent points which are only accessible from the inner layers (for blind and buried technology designs).

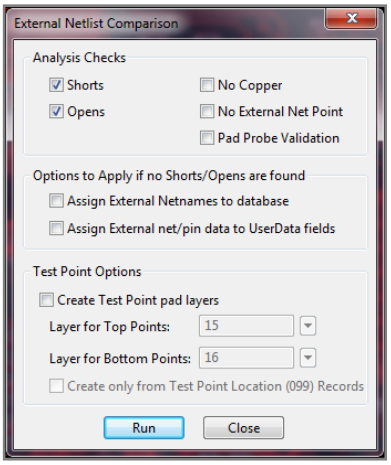


TIP: The external net point graphic colors are arbitrary. If necessary, use the Colors bar to change the highlight, draw, and flash colors for the layers you are looking at for better contrast and easier visibility.

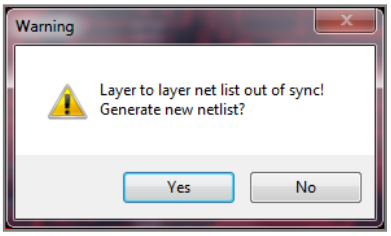
8. Select the **Analysis > Netlist Compare** command from the menus.

In the External Netlist Comparison dialog box:

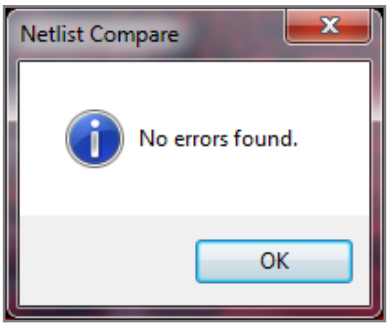
9. Select the **Shorts and Opens** Analysis Checks.
10. Click on the **Run** button.



11. If you receive a Warning that your layer to layer net list is out of sync, click the **Yes** button to generate a new netlist.



12. If no errors are found, click on the **OK** button in the Netlist Compare message box.

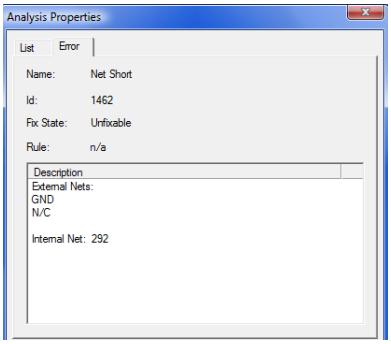


After performing the analysis, discrepancies are automatically displayed in the Navigator, which you may use to view the errors in the workspace and print reports.

13. Double-click on the error in the Navigator to view it in the workspace. The VisualCAM internal net in question is highlighted, and zoomed to fit the screen. The external net points are shown in a unique color for each net.

14. If VisualCAM finds an error, right click on the error and choose properties to view the Analysis Properties box.

15. Find the precise location of the error more quickly by right-clicking on the error in the Navigator, and selecting the Pin-Point Error command.



TIP: If there is a fix to the data needed, you can now change the data as necessary. Once you have done so, you should then re-run the Netlist Compare command to verify that the fix was accurate. Once no more errors exist, you can use the **Tools > Netlist > Apply External Net Names** command to apply the external net names to the database, if you wish.

Review Questions

What are the layer requirements for netlist comparison?

_____
_____
_____
_____

What happens if there is no thru-hole drill layer present?

_____
_____
_____
_____

Where are the netlist compare errors found?

_____
_____
_____
_____

How can you find the actual location of the error?

_____
_____
_____
_____

DRC/DFM Analysis

The VisualCAM Analysis functions are used to find any flaws in your design prior to manufacturing. Using rules that you specify, the system analyzes your design, finds errors, and determines the best fix for those errors. You can then review the errors in the workspace, print error reports, fix the errors yourself, or have the system fix the errors for you.

For maximum efficiency, all your rules can be saved for use in future designs and each check can be run at the same time.

Lesson Objectives

In this lesson you will do the following:

- Perform information checking.
- Perform DRC/DFF Analysis.
- Review and fix problems found during Analysis.

Estimated Completion Time

Approximately 35 minutes.

Information Checking

This section of the Analysis Checks provides information about your design that can be particularly useful in cost estimation and process evaluation. The available checks are:

- **Min Air Gap -** Reports the shortest distance between any two items in the design.
- **Conductive Layer Count** - Reports the total number of conductive layers.
- **Board Size** - Reports the extents of the Border layer, if one exists. Otherwise the extents of the conductive layers is reported (drawing and other graphical layers are ignored).
- **Hole Count** - Reports the tools used on each NC layer, and the number of holes (hits) created by each.
- **Hole Size Count** - Reports the number of tools used on each NC layer (does not report tool numbers or hits).
- **Buried Vias** - Reports the upper- and bottom-most (start and end) layers for buried vias. Blind and Buried layer sets must be defined in order to run this analysis.
- **Blind Vias** - Reports the upper- and bottom-most (start and end) layers for blind vias. Blind and Buried layer sets must be defined in order to run this analysis.
- **Min Pad/Drill** - Reports the minimum annular ring spacing from copper pads to plated thru-hole drill hits.
- **Min Drill/Drill** - Reports the minimum spacing between drill hits.
- **Min NPTH/Copper** - Reports the minimum spacing between non-plated thru-hole drill hits and copper on external and internal layers.
- **Min Pad/Pad -** Reports the minimum spacing from one pad to another for each conductive layer.
- **Min Pad/Trace** - Reports the minimum spacing from a pad to a trace for each conductive layer.
- **Min Trace/Trace -** Reports the minimum spacing from one trace to another for each conductive layer.
- **Min Pad/Mask** - Reports the minimum annular ring spacing from mask openings to their corresponding copper pad, on the top and bottom layers.
- **Min Thermal/Drill** - Reports the minimum annular ring from thermal reliefs to plated thru-hole drill hits.
- **Min PTH Pad Size** - Reports the size of the smallest pad that has been pierced by a plated thru-hole drill hit, for each conductive layer.
- **Min Trace Width** - Reports the smallest trace width in the design.
- **Min SMT Pitch** - Reports the minimum center-to-center spacing of (SMD pads. A pad must be identified as SMT in the Aperture List in order to be checked.
- **Legend on PTH Count & Legend on NPTH Count** - Reports the number of occurrences of the silkscreen legend touching pads on the top and bottom layers.
- **Missing Hole Count** - Reports the number of instances where a thru-hole pad on a conductive

layer appears to be missing a drill hit.

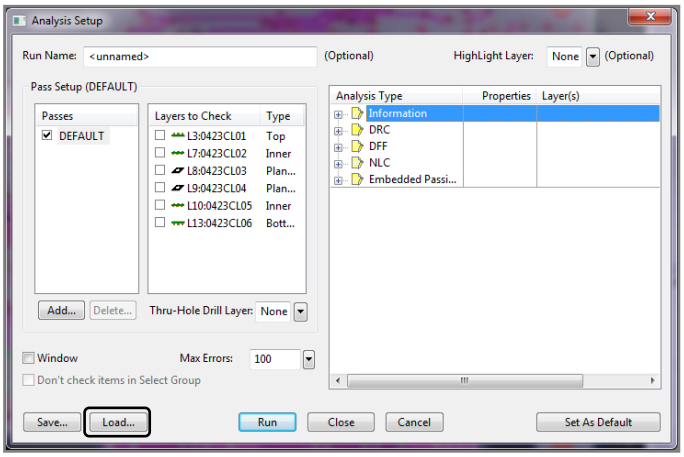
- **Mill Path Length** - Reports the total linear distance each tool travels, per NC layer.
- **Max PTH Registration** - This shows the greatest amount of misalignment of data on drill and conductive layers. It reports the maximum spacing of plated thru-hole drills (on an NC layer) from their corresponding pads (on a conductive layer).
- **Exposed Copper Count** - Reports the number of occurrences of traces exposed by adjacent mask openings.
- **Copper Sliver Count** - Reports the number of occurrences of copper slivers. Any areas of copper that are less than the Size you specify will be considered possible copper slivers.
- **Mask Sliver Count** - Reports the number of occurrences of mask slivers. Any resist areas that are less than the Size you specify will be considered possible mask slivers.

Activity: Perform Information Checking

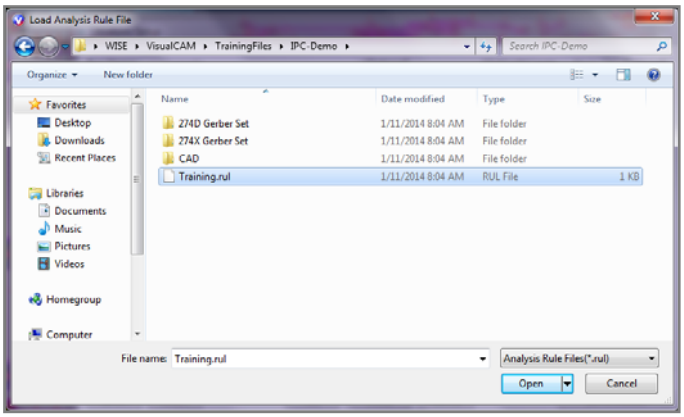
In order to perform analysis functions, each layer type must be set correctly, and the layers that are to be checked must be selected in the “Layers to Check” area of the Analysis Setup Dialog. Only conductive layers and the Thru Hole Drill Layer need to be selected; Mask, Paste, Silk and Blind and Buried drills will be found automatically if they are present in the design with their layer type set.

NOTE: Prior to performing the analysis, make sure that you have generated a netlist for your file (*see* *Generate a Netlist, page* 88.)

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Open the the Analysis Setup by choosing the **Analysis > DRC/MRC** command from the menus.
3. In the Analysis Setup dialog box click on the **Load…** button in the lower left corner.



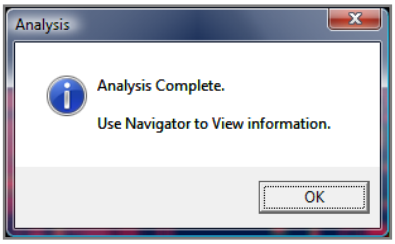
4. In the Load Analysis Rule File dialog box select the **…TrainingFiles/IPC-Demo/Training.rul** file.
5. Click on the **Open** button.



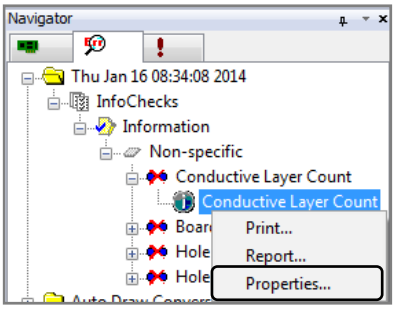
6. Back in the Analysis Setup dialog box click on the **Run** button to run the information checks defined.

After a few moments the Analysis Complete box will pop up.

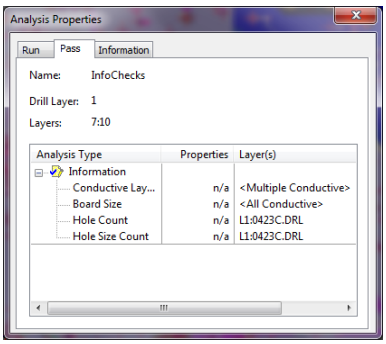
7. Click on the **OK** button.
8. **Save** the file.



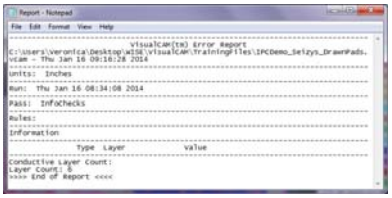
9. **Open the Analysis tree in the Navigator**, and expand the results.
10. **Expand the** Conductive Layer Count **branch**.
11. **Right click on the** Conductive Layer Count **individual check**.
12. Choose **Properties…** to view the results of the individual check.



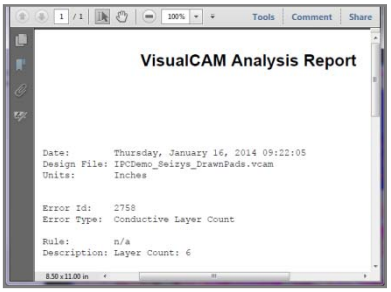
13. **Close** the Analysis Properties box.



14. **Right click on the** Conductive Layer Count **individual check** again.
15. Choose **Report… to save and view a text file** of the information checks performed.



16. **Right click on the** Conductive Layer Count **individual check** again.
17. Choose **Print… to save and view a PDF file** of the information checks performed.



DRC/DFF Checking

In the following activity we will use the Analysis tool, which allows simple rule based checks to determine the manufacturability of your design. This process can help to catch design problems missed by the CAD system (DRC checks) and find manufacturability issues that the CAD system is not capable of checking (DFF Checks). The following is a comprehensive list of the currently available checks:

DRC (Design Rule Check)

- **Pad/Pad** - The minimum spacing allowed between pads.
- **Pad/Trace** - The minimum spacing allowed between pads and traces.
- **Trace/Trace** - The minimum spacing allowed between traces.
- **Border** - The minimum spacing allowed between any item and the border.
- **Min Pad -** The minimum pad size allowed.
- **Min Trace** - The minimum trace size allowed.
- **Stubs** - A trace stub is any trace that touches a pad or trace on one end, but does not on the opposite end.
- **Drill/Drill** - The minimum spacing required between adjacent drill locations.
- **Drill Annulus** - The minimum annular ring required between conductive layers and drill layer.
- **Drill/Copper** - The minimum space required between copper entities and drills. Separate tolerances are available for plated thru-holes (PTH) and non-plated thru holes (NPTH).
- **Pad/Top Mask & Pad/Bot Mask** - The minimum annular ring required between top/bottom side pads and the top/bottom solder mask. (Only checks the size of existing openings, not whether there is an opening missing for a pad. Use the Missing Top Mask and Missing Bot Mask checks to find missing openings, if desired.)
- **Drill/Top Mask & Drill/Bot Mask** - The minimum annular ring required between the drill layer and the top/bottom solder mask.
- **Missing Top Mask & Missing Bot Mask** - Checks for pads on the top/bottom side conductive layer without openings on the top/bottom solder mask.
- **Pad/Top Silk & Pad/Bot Silk -** The minimum clearance required between top/bottom side pads and the top/bottom silkscreen layer. (You can use the Tools > Fix SilkScreen command to remove silkscreen data from pads). If you select a "Mask: Yes" property, then the silkscreen layer is instead compared to the corresponding solder mask layer.
- **Missing Drill** - Checks for pads that do not have a corresponding drill hit.
- **Coincident Drill** - Checks for drill hits that are in the same location but are different sizes.
- **Redundant Drill** - Checks for drill hits of the same size that are in the same location.
- **Mill Path** - Checks compensated mill pats for errors introduced by the compensation: arcs that implode, paths that cross over themselves, and any break tabs that become invalid because the compensated path is now too short for the tab to fit.

DFF (Design For Fabrication)

- **Acid Traps -** An acid trap is an area where etching solution accumulates but does not flow out during manufacturing. This causes over-etching, which hurts yield. This analysis detects potential acid traps in your design. Any areas where acid is likely to accumulate that are smaller than the Size you specify, and any items (traces, pads, etc.) that form an Angle smaller than the one you specify, will be considered possible acid traps.

TIP: Because the acid trap analysis looks for areas that are not fully enclosed, it is recommended that you run the resist sliver analysis in conjunction with this, for a thorough check of your design.

- **Copper Slivers -** Copper Slivers are areas of copper that are so narrow that they will likely flake off. This command detects those potential slivers on the selected conductive layers in your design. Any areas of copper that are less than the Size you specify will be considered possible copper slivers.
- **Resist Slivers -** Similar to acid traps, resist slivers are small areas of resistive material that have a surface area too small to adhere to the board, and can therefore flake or peel off. Where acid traps are open at one end, resist slivers can be fully surrounded by copper. Depending on your process, the resist slivers test will often find items that are also considered acid traps or pin holes. The Resist Slivers analysis may be more applicable to those using a plate-up photo resist process instead of an etch-back process. Resist areas less than the value you specify are considered resist slivers.
- **Top Mask Slivers & Bot Mask Slivers -** Solder mask slivers are areas where the resist is so narrow that they will likely flake off, float, and redeposit themselves in an area that might be soldered later. Any resist areas that are less than the Size you specify will be considered possible mask slivers.
- **Isolated Thermals -** Over-etching of surrounding items can result in a thermal being isolated from the rest of a negative plane. This analysis function allows you to oversize the data on the negative plane layers by a specified amount, and determine its effects on the connection of the thermal to the negative plane. You specify the amount of Over Etch that you wish to apply to the data on the specified plane layer.

Warning: A thermal can be considered isolated by virtue of its design. For example, if a thermal has ties that are 2 mils wide, and the Over Etch is specified as 1 mil or larger, the thermal is isolated.

- **Starved Thermals -** The starved thermal analysis verifies whether each thermal connection to the negative plane is valid, or has been constricted by adjacent data that is too close (or overlapping). You specify the Percentage of the thermal's tie width that can be unblocked by objects. This makes the clearance check relative to each thermal's tie width. The thermal is considered starved if the connection from the interior of the thermal to the exterior, measured across the minimum distance between the cutouts of the thermal and any other nearby data, is less than the specified value. The Clear value is how many thermal ties must be clear in order for a thermal to be considered not starved. If you specify 0, all ties must be clear. If you select 1, one tie must be clear.

Warning: Be aware of the number of ties in all the thermals in your design. If you select 4 as your Clear value, and have thermals with 2 ties, then those thermals would automatically be flagged as starved,

even if both of their ties are clear. If you have several different types of thermals, it is recommended that you leave the Clear value at 0.

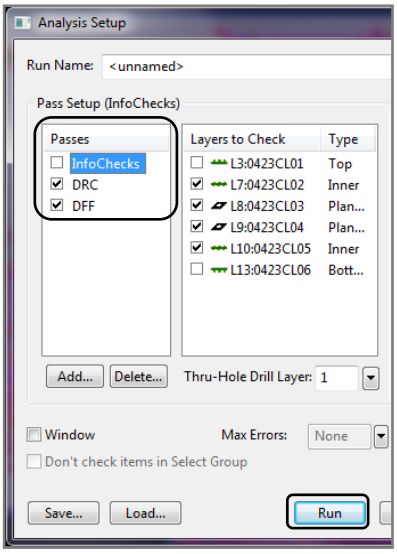
- **Top Solder Bridges & Bot Solder Bridges -** Openings for pads on a mask layer may be oversized too much, and expose an adjacent trace or another conductive object. The copper for that pad may accidentally get too close during fabrication, and create an unwanted connection, or bridge, to the adjacent object. The Bridge Distance is the distance between the pad and objects in the same mask opening, where the solder can possibly create a bridge. If the adjacent object closer to the pad than this distance, it will be identified as a possible bridge. The Position Tolerance tests for shifting of the placement of mask openings. It oversizes mask openings by the amount you specify, to check if other conductive objects are then exposed by the mask opening. If an object is exposed, and is also within the specified Bridge Distance to the pad, then an error is logged.
- **Pin Holes -** A pin hole is a void in an area of solid copper that can cause acid to pool and over-etch the surrounding copper (similar to an acid trap), or can cause resist flakes, potentially causing conductivity problems. Pin holes often result from inaccurate vector-drawn data that leaves tiny voids between the over-lapping draws. You specify the maximum Size of the pin holes to be detected -- areas which are less than this value will be considered pin holes.
- **SMD Pitch -** Checks the center-to-center spacing of adjacent SMD pads, to determine if they are too close. A pad must be identified as SMT in the Aperture List in order to be checked.
- **SMD Spacing -** Checks the edge-to-edge spacing of adjacent SMD pads, to determine if they are too close. A pad must be identified as SMT in the Aperture List in order to be checked.
- **Layer Registration -** Checks your drill layer against the top mask and any other conductive layers you select. If two items are within the tolerance distance you specify, they are considered to be at the same location. The errors reported are those items that, if you use the Edit > Align Layers command or VisualCAM's auto-fix feature to align the layers based on them, will result in 95% or more of the other items on the layer being correctly aligned as well.

Activity: Perform DRC/DFF Analysis

Follow the directions in the activity below using the IPC-Demo job to demonstrate how to use this function.

NOTE: Prior to performing the analysis, make sure that you have generated a netlist for your file (*see Generate a Netlist, page 88.*)

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Choose the **Analysis > DRC/MRC** command from the menus.
3. In the Analysis Setup dialog box, deselect the InfoChecks pass (since you already ran it in the previous activity).
4. Add check marks to the **DRC** and **DFF** passes.
5. Click on the **Run** button to run the checks defined for these passes.



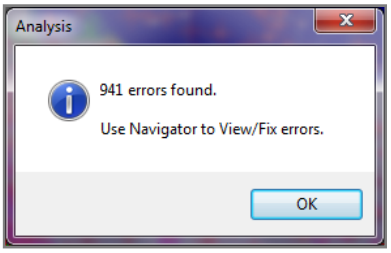
After a few minutes an Analysis box opens reporting that errors were found.

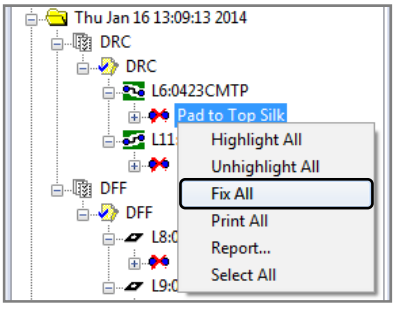
6. Click on the **OK** button.
7. In the Navigator click on the **Analysis tab**.
8. **Expand the** Analysis **branches**.
9. **To fix all DRC** errors at once, right click on the DRC folder and choose **Fix All** from the menu.

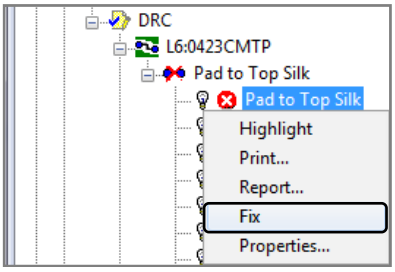
   To fix all errors on individual layers, right click on the layer and choose **Fix All** from the menu.

   To fix all specific errors, for example Pad to Top Silk in the image at right, right click on the layer and choose **Fix All** from the menu.

   To fix individual errors one at a time, expand the error branch, right click on the first error, and choose **Fix**.
10. **Respond to any messages** you receive.
11. **Save** the file.







Review Questions

What are the layer requirements for using the Analysis checks?

What happens if there is nothing selected in “Layers to Check”?

How can you generate a report from the Analysis results?

Why are DRC checks used?

What kind of problems do DFF checks find?

How can I apply all the possible fixes at once rather than clicking each one?

Creating Stencils

In this lesson we will describe the functions and processes used to enhance the Paste Stencil openings in your Gerber data for improved paste application.

Lesson Objectives

In this lesson you will do the following:

- Optimize a paste mask to take care of under-sizing the paste layer.
- Create special stencil paste openings, such as home plates and melfs.
- Create window panes using a macro.

Estimated Completion Time

Approximately 35 minutes.

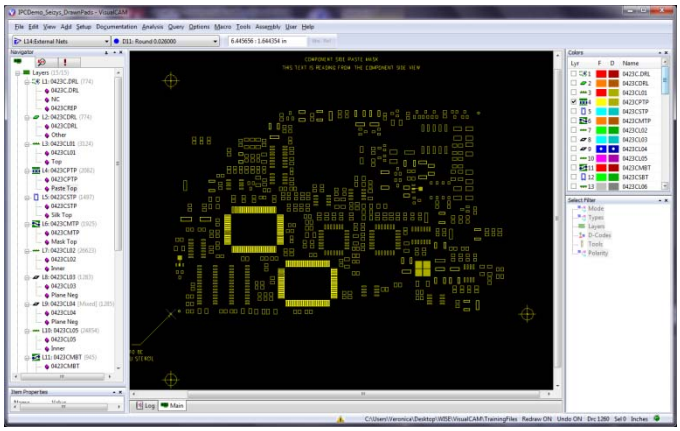
Activity: Optimize or Generate a Paste Mask

This function automatically under-sizes existing paste mask openings on a Paste Mask layer based on the rules given. This function operates on flashed pads only, so in our IPC-Demo design we will need to convert the paste layer to flashed pads first before under-sizing.

NOTE: If no Paste layer exists in your design, use the Generate Paste Mask command instead of Optimize. The Paste Mask Generator produces accurate and usable paste mask layers and uses the size of your conductive pads, and an undersize amount that you define, to create a paste mask layer with openings of an appropriate size. The settings and options for both commands are nearly identical.

This activity should take about 15 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. In the Colors bar, **turn off all layers except for the Paste Mask layer**. If no such layer exists in your design, you will generate one in this activity.

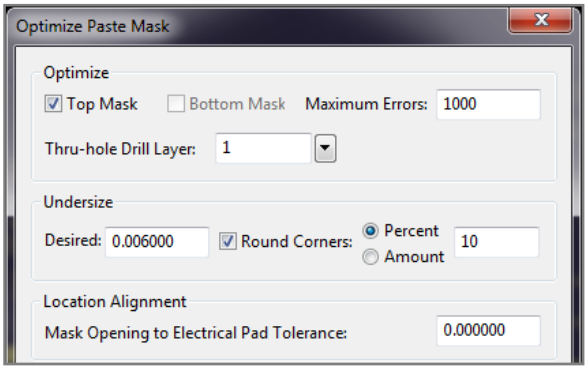


3. Make sure **all drawn pads have been converted to flashes** (*see Draws to Flash Conversion, page 77.*)
4. **If you want any items in your database ignored** during the process, create a Select Group of those items (*see Step #7 in Generate a Netlist, page 88.*)
5. **If a Paste Mask layer exists in your design**, select the **Tools > Paste Mask > Optimize** command from the menus.

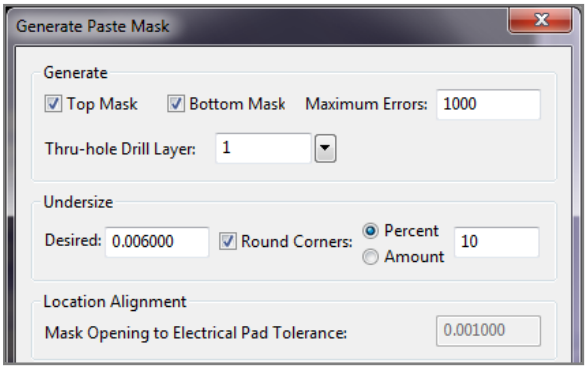
   If no Paste Mask layer exists in your design, select the **Tools > Paste Mask > Generate** command from the menus.

NOTE: The system uses the Thru-hole Drill Layer specified to identify the different types of pads in your design. It will process any pad that is SMT. If you do not have a thru-hole drill layer, then the system will use the Aperture Table Type information.

OPTIMIZE Paste Mask Dialog Box:



GENERATE Paste Mask Dialog Box:



In either the Optimize or Generate dialog box:

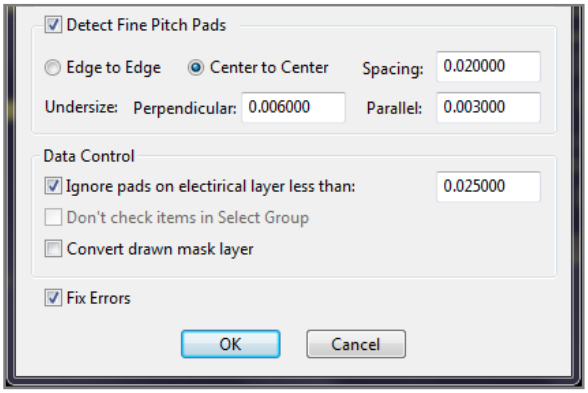
6. **Select whether to optimize or create a Top Mask, Bottom Mask, or both**.
7. In the **Maximum Errors** box, specify the maximum number of allowable errors as **1000**.

   If the system reaches this error count limit, it stops logging errors and warns you that the limit was reached. This limit is provided in case you enter an under-size amount that is not appropriate for your design.
8. Specify your **Desired Undersize** as **6 mils**.

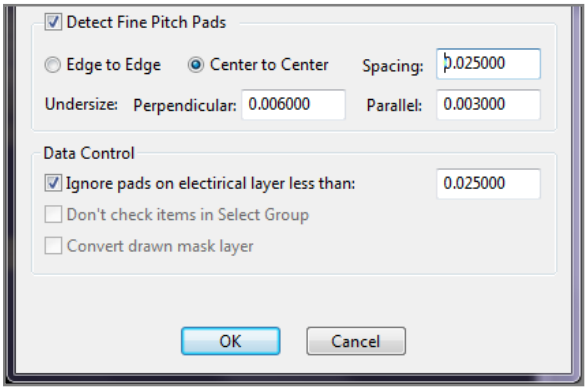
   The **optimization** process looks at all openings on the mask layer at the same location as each pad on the corresponding electrical layer. It determines if the openings are undersized by at least the minimum Desired amount. (It will not report an error if the mask opening is undersized more than the minimum amount.)

   The **generation** process adds an opening of the appropriate shape on the mask layer at the same location as each pad on the corresponding electrical layer. The openings are decreased in size by the specified Desired Undersize amount.
9. Select the **Round Corners** option for the openings. Define the corners as **10 percent** of the shortest side of the aperture.
10. In the **optimization** process the **Location Alignment** specifies the maximum misalignment allowed between the location of the mask opening and the corresponding pad on the electrical layer. **Set this to zero**.

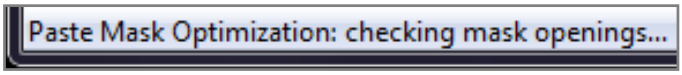
OPTIMIZE Paste Mask Dialog Box:



GENERATE Paste Mask Dialog Box:

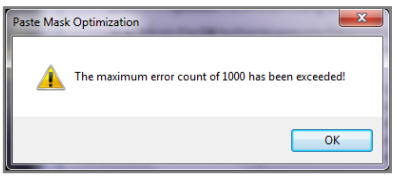


11. Select the **Detect Fine Pitch Pads** option to have the system analyze pads for fine pitch conditions (round pads are ignored). Choose **Center to Center, spacing of 25 mils**. Leave the **default undersize values of 6 mils perpendicular and 3 mils parallel**.
12. Select the **Ignore pads on electrical layers less than:** option and select a **minimum pad size of 0.025**.
13. In the **Optimize** Paste Mask dialog box, **select the Fix Errors option**.
14. Click on the **OK** button to execute the process. It may take a few minutes to finish processing. The status bar in the lower left corner of the workspace will let you know what is happening:

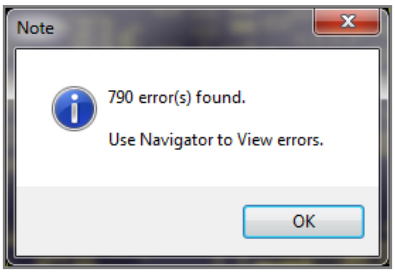


NOTE: The Fix Errors option is only available for the Optimizer. The only error that can occur while generating a paste mask layer is if the undersize amount is too large for a pad, which would mean that the opening size would effectively be less than zero. An opening would not be created, and a Missing Mask Opening error would be logged. To fix this, you must create the paste mask again, decreasing the undersize amount or increasing the size of small pads to exclude (whichever is appropriate in your case).

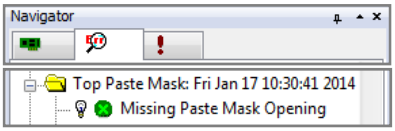
15. If you receive a Paste Mask Optimization message box stating that the maximum error count has been exceeded, click on the **OK** button.



16. The system will let you know how many errors were found. Click on the **OK** button.
17. **Save** the file.



NOTE: Regardless of whether you have the system fix any errors, they are all logged in the Navigator, where you can view them, print reports, and fix/unfix the errors.



The following possible errors are reported:

- **Missing Electrical Pad** There is no pad on the electrical layer corresponding to an opening on the mask layer. To fix this, the mask opening is deleted.
- **Missing Paste Mask Opening** There is no opening on the mask layer corresponding to a pad on the electrical layer. To fix this, a mask opening is added.
- **Misalignment Error** The opening on the mask layer and its corresponding pad are misaligned by a value greater than the Location Alignment value you specified. To fix this, the mask opening is moved to the same location as the pad.
- **Mask Opening Minimum Undersize Error** The mask opening is not undersized by at least the minimum specified undersize. To fix this, the mask opening is undersized by the specified amount.

Activity: Create Special Stencil Shapes

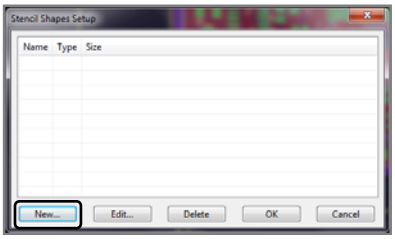
In the following activity we will use the Stencil Shapes tool, which allows simple rule-based conversions of stencil pad pairs to common IPC stencil shapes. Creating special stencil shapes should occur after the general paste reduction has been applied.

Stencil conversion may be done on a single pad or a pair of pads. Therefore, two types of shapes are supported: shapes that are used when a pair of pads are being converted and shapes that are used when a single pad is converted. The paired pad conversion shapes are "Home Plate", "Inverted Home Plate", "MELF", "Oblong", and "Snubnose". The single pad conversion shapes are "Dogbone" and "<Original>".

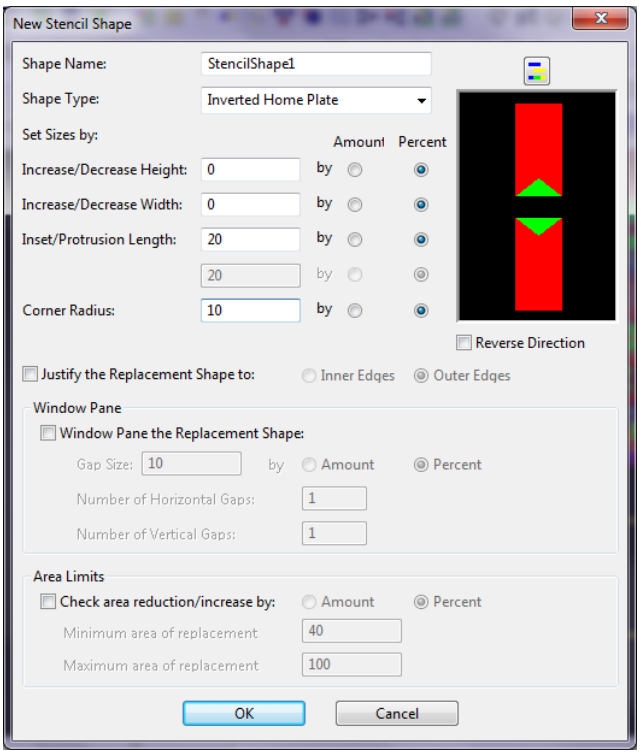
When setting up stencil shapes either type of shape may be created. However, when creating or modifying a shape for specific conversion, such as during Tools > Stencils > Manual Conversion, the type of conversion being done (pair or single) is known and the selectable shapes will be of the appropriate type. In addition, if a new shape is created or an existing shape is to be modified only the correct types for that conversion are available.

This activity should take about 15 minutes to complete.

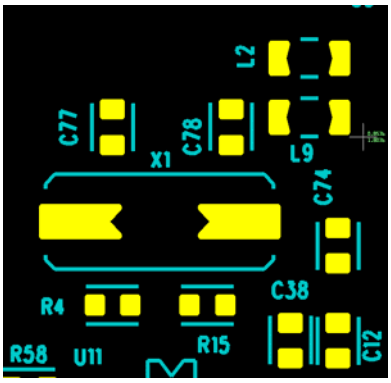
1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Tools > Stencils > Setup Shapes** command.
3. In the Stencil Shapes Setup dialog box, click on the **New** button.



4. In the New Stencil Shape dialog box, enter the values shown in the following screenshot:



5. Click on the **OK** button.
6. Back in the Stencil Shapes Setup dialog box, click on the **OK** button.



7. In the Colors bar turn on off the **visibility** of all layers except for the **Silk Top Layer**.
8. Select the **Tools > Stencils > Manual Conversion** command to teach the system which stencil shapes you want to associate with which original openings over pad pairs.

   Select the pair of pads marked as X1 and L2 on the Silk Top Layer.

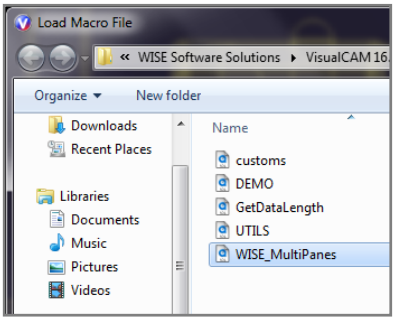
   After selecting each pair of openings, when you are asked whether you want to convert the matching openings, click **No**. It will still save the assignments you make, but the openings will not be converted yet.
9. Select the **Tools > Stencils > Automatic Conversion** command to specify which paste mask layers you wish to convert, what destination layers to place the stencil data on, and convert all the openings.

TIP: Once you have completed the previous steps, you can save the stencil definitions and associations to a file by clicking the Save button in the Automatic Stencil Conversion dialog box. The next time you want to enhance a stencil using the same stencil shape definitions and associations, just select the **Tools > Stencils > Automatic Conversion** command, click the **Load** button to load the file, and run the conversion.

Activity: Create Window Panes

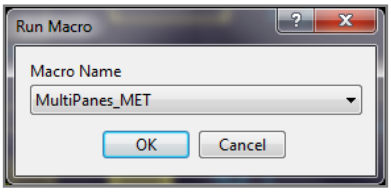
This activity should take about 5 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Macro > Load** command from the menus.



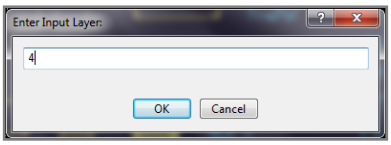
In the Load Macro File dialog box:

3. Choose the **WISE_MultiPanes.mac** file.
4. Click on the **Open** button.
5. Select the **Macro > Run** command from the menus ("M" on your keyboard.)



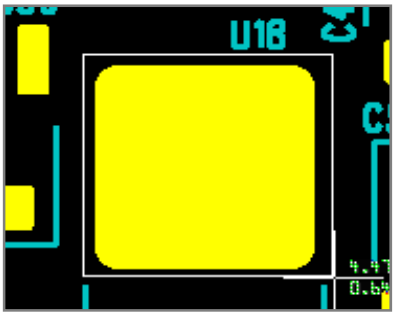
In the Run Macro dialog box:

6. Select **MultiPanes_MET** from the drop-down list.
7. Click on the **OK** button.



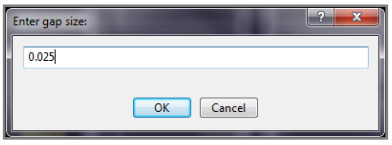
In the Enter Input Layer dialog box:

8. **Type the number of your Paste Layer** in the textbox.
9. Click on the **OK** button.
10. Window around the large ground pad for **U18** as shown in the image to the right.



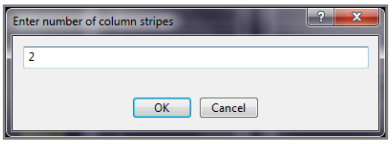
In the Enter gap size dialog box:

11. Enter a **Gap Size** of **0.025**.
12. Click on the **OK** button.



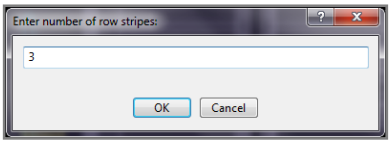
In the Enter number of column stripes dialog box:

13. Enter **2** in the textbox.
14. Click on the **OK** button.

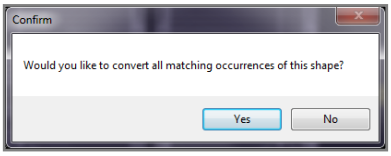


In the Enter number of row stripes dialog box:

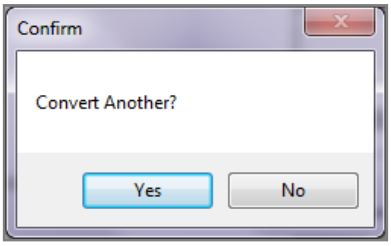
15. Enter **3** in the textbox.
16. Click on the **OK** button.



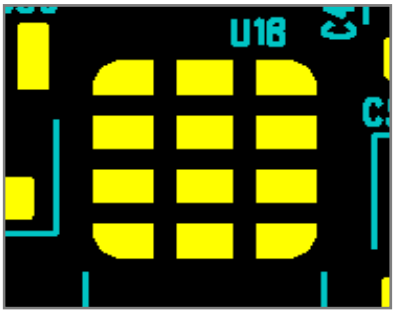
17. When you are prompted to convert all matching shapes, click on the **Yes** button (although in this design there will be only one match.)



18. When you are asked whether or not to Convert Another?, click on the **No** button.



Your pad should now look like the image at the right.



Review Questions

What are the layer requirements for using the Optimize Paste Mask Tool?

_____
_____
_____
_____

What happens if there is no thru-hole drill layer present?

_____
_____
_____
_____

What would you do if there were no Paste Layer to optimize?

_____
_____
_____
_____

Why are home plate and inverted home plate shapes used?

_____
_____
_____
_____

Should you create special stencil shapes before or after general paste reduction has been performed?

_____
_____
_____

V-Score Layer

In this lesson you will learn how to setup a V-Score layer and add V-cuts to this layer to be used later in Advanced Panelization.

Lesson Objectives

In this lesson you will do the following:

- Setup a V-Score Layer and add V-Cuts.

Estimated Completion Time

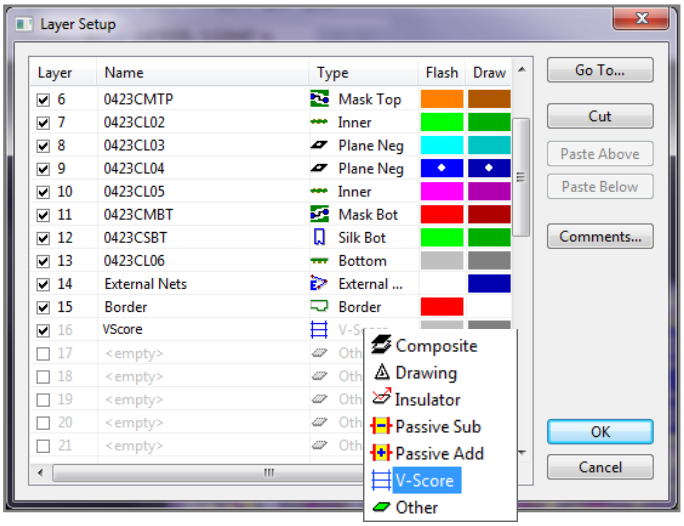
Approximately 10 minutes.

Activity: Setup V-Score Layer and Add V-Cuts

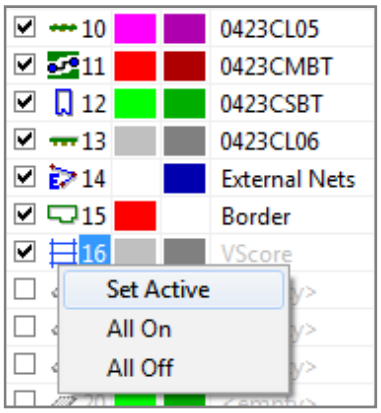
This activity should take about 10 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Setup > Layers** command from the menus.

In the Layer Setup dialog box:

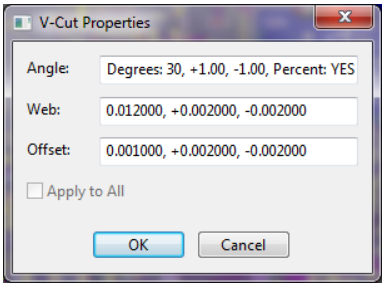


3. Choose an **Empty Layer**.
4. Click in the **Name** box and rename it **VScore**.
5. Click in the **Type** box and choose **V-Score** from the list of options.
6. Click on the **OK** button.
7. In the Colors bar, make the new **V-Score layer the Active Layer**.
8. Select the **Add > V-Cut** command from the menus.

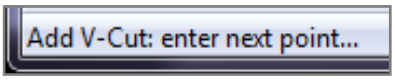


In the V-Cut Properties dialog box:

9. **Set the properties of the V-Cut**. You may adjust the bit angle, web thickness and side-to-side offset properties that will be attached to the V-Cuts you are about to add.
10. Click on the **OK** button.



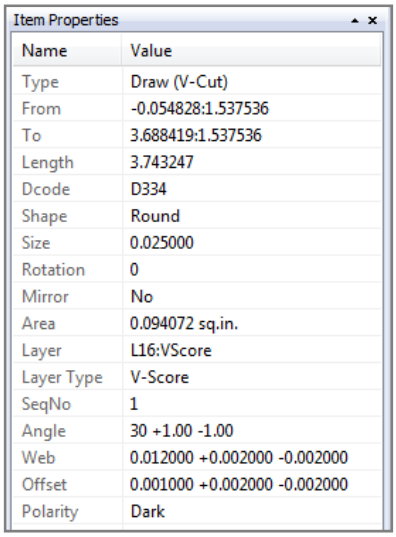
You will see a prompt in the Status Bar for a starting location for the V-Cut.

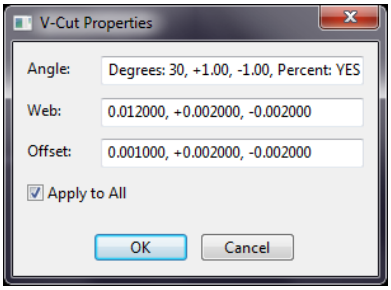


11. Use your mouse to **click on the desired location** in the Workspace.

You will then be prompted to enter the next point.

12. Use your mouse to **click on the desired location**.
13. **After selecting the end point** for the V-Cut, press the **<Esc>** key to end the current V-Cut.
14. You may **add another V-Cut by clicking on a new starting point** (you can change to different V-Score layers at any point), **or press <Esc> again** to exit the function.
15. **To see the properties of a V-Cut in the Item Properties bar:**
    a. **If the Item Properties bar is already active**, right click on the V-Cut item.
    b. **If the Item Properties bar is not active,** right click on the V-Cut item and choose Properties from the list of options.
16. **To change the properties of existing V-Cut items**:
    a. **Right click** directly over a V-Cut item.
    b. Choose **Edit Properties** from the list of options
    c. **Make the changes** in the V-Cut Properties dialog box.
    d. **Choose the Apply to All option** to change the properties of all V-Cut items on a single layer at once.
    e. Click on the **OK** button.
17. **Save** the file.





Review Questions

What properties can be setup or changed when adding V-cuts to a V-Score layer?

_____
_____
_____
_____

Where can you view the properties of existing V-cut items?

_____
_____
_____
_____

When changing the properties of existing V-cut items, what does the Apply to All option in the V-Cut Properties dialog box do?

_____
_____
_____
_____

Advanced Panelization

VisualCAM's Advanced Panelization allows you to panelize single or multiple designs, and lets you store an unlimited number of panel templates so you can use the same panel configuration for multiple jobs.

Lesson Objectives

In this lesson you will:

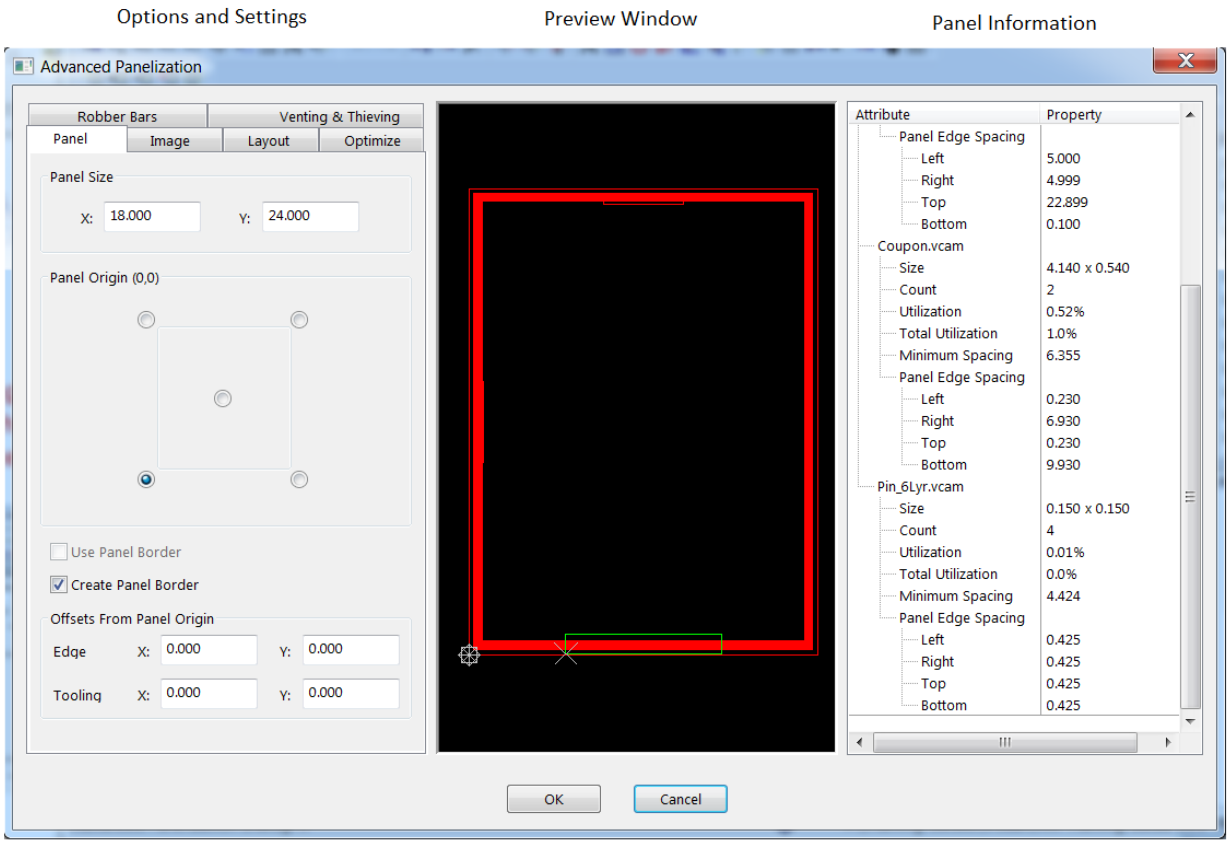
- Create a panel template.
- Panelize using a panel template.

Estimated Completion Time

Approximately 45 minutes.

Advanced Panelization Dialog Box

There are three main windows within the Advanced Panelization dialog box: on the left is a tabbed window that presents all the options and settings available; in the middle is the viewing window where preview rendering takes place and some user interaction; the right-most window is for information only and contains an Attribute/Property tree that displays all the current settings, both user set and auto calculated.

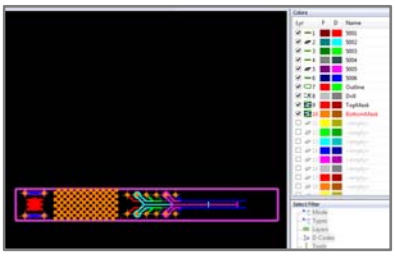


Activity: Create a Panel Template

Before starting the panel template you will want to create your coupons, pinning holes, title blocks and any other images that you want to include in your panel template. Each of these images is created as individual .vcam files. There are several coupons included in your VisualCAM install in the Sample Template folder that you can use or modify to fit your needs.

We will be working with three files from the template folder for this activity. Load the coupon design in and notice it is just like any other VisualCAM design file.

1. Open the file **C:\ProgramData\WISE Software Solutions\VisualCAM\Sample\Templates\Coupon.vcam**.

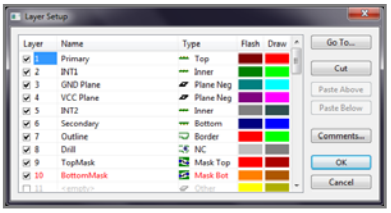


NOTE: When creating a new coupon or image file make sure that each layer is "tagged" with the correct layer type.

Setup Layers and Load Files

2. Use the **Setup > Layers** command or the **Navigator** to **tag your layer types if necessary** (*see Change the Layer Type, page 53*.)

   This coupon is setup for a 6 layer board with all the layer types already set.

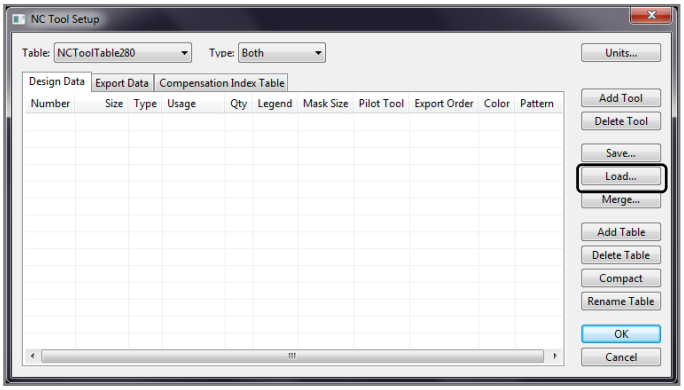


Once you have all your image files created and saved you are ready to define the panel template. We are going to create a 6 layer panel template that may be used to panelize any design that utilizes the same layer structure.

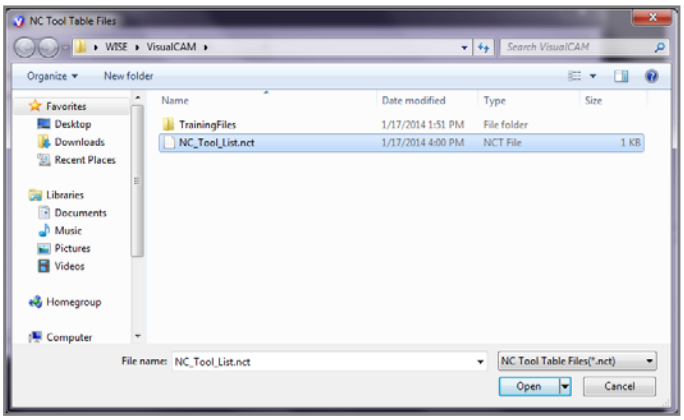
When defining a New Panel Template, Layer Type setup is a very important step to help automate the process of layer mapping. Also, if you use a master NC tool list, import your NC tool list or load in your NC Tool table prior to loading in your image files.

3. Start with an empty design using the **File > New** command.
4. Click on the **Main** viewing tab.
5. Select the **Setup > NC Tools** command from the menus.

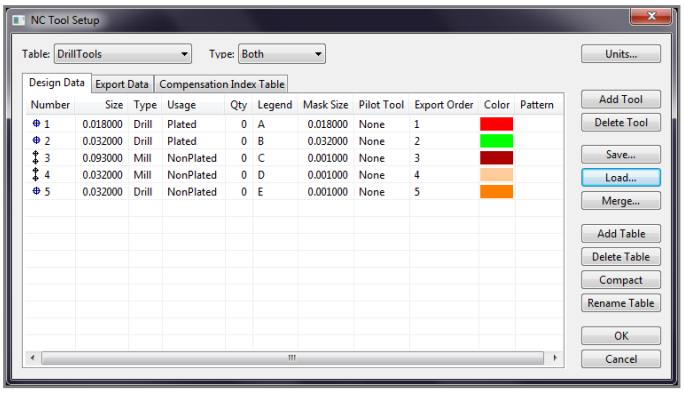
6. In the NC Tool Setup dialog box, click on the **Load** button.



7. In the NC Tool Table Files dialog box, browse for and select the file **NC_Tool_List.nct**.
8. Click on the **Open** button.

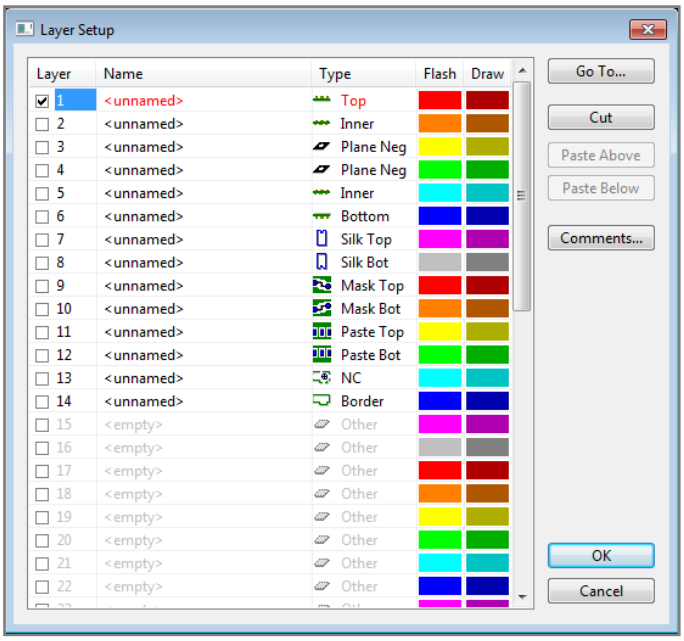


The NC Tool Setup dialog box will now be populated with the **NC_Tool_List.nct** values**.**

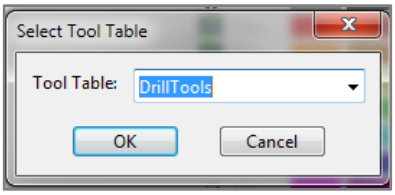


9. Click on the **OK** button.

10. Select the **Setup > Layers command from the menus.**
11. Left click on **Layer 1's Type** field and choose **Top** from the list of available layer types.
12. Left click on **Layer 2's Type** field and choose **Inner**.
13. **Repeat for the remaining layers** using the following image as a guide:



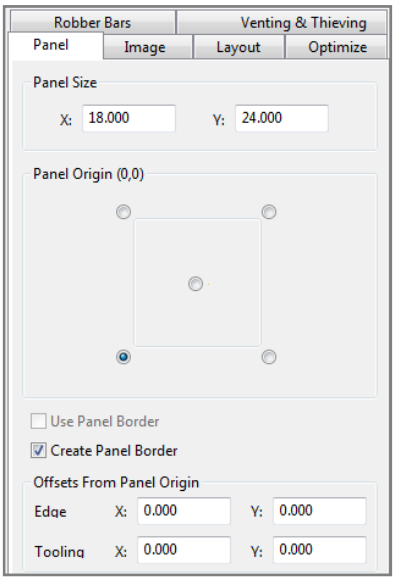
When you choose NC for Layer 13 a Select Tool Table dialog box will open up, prompting you to choose a tool table.



14. Choose **DrillTools** from the available options.
15. Click on the **OK** button.
16. **Finish tagging Layer 14** in the Layer Setup dialog box.
17. Click on the **OK** button.

With all layers types defined and image files created you are now ready to create the panel template.

18. Select the **Tools > Panelize > Advanced** command from the menus.



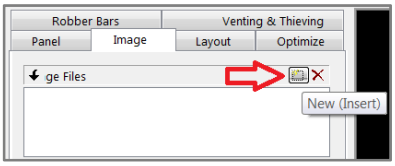
When the Advanced Panelization dialog box comes up you are initially on the **Panel Tab**, this tab lets you define the panel, including the size of the panel, the origin of the panel and a tooling offset.

19. Define the **X** and **Y** dimensions of your panel. We will use the default setting **18 x 24 panel size**.
20. **Select the lower left dot** for the origin of the panel. This establishes the 0:0 point for the panel, and all image data are placed in reference to it.

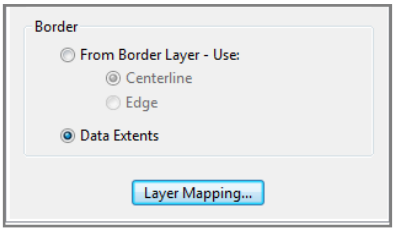
The next step is to load and place the coupons, pinning holes, title blocks and any other images that you wish included in your panel.

Note these files remain separate from the panel design, therefore any updates you make to these .vcam files will automatically be included in the panel template the next time you open it.

21. In the dialog box click on the **Image tab** to load any images that are to be included in the panel template.

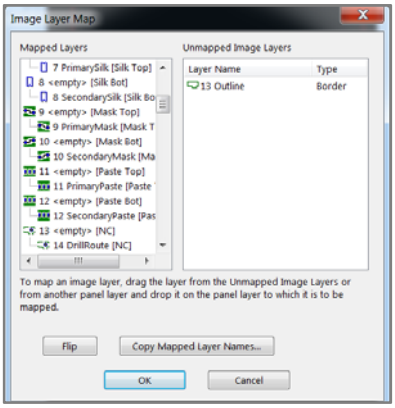


22. Click on the **New** button and then the … to browse for the image file **TitleBlock.vcam**.
23. In the Open dialog box, **select the file**.
24. Click on the **Open** button.



VisualCAM loads the file and activates the Layer Mapping button on the Image tab so you can verify that the image file is defined in the panel correctly.

25. Click on the **Layer Mapping** button.

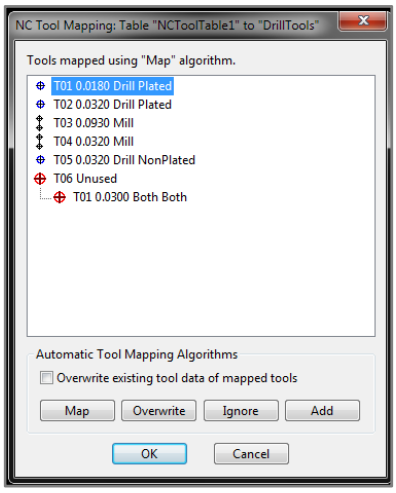


In the Image Layer Map dialog box all layers are automatically mapped appropriately, Top is mapped to Top, Inner mapped to Inner etc.

If any layers are not mapped, for example the Border layer in the image at right, it is because you didn't setup the Border Layer during the layer setup and the Border Layer type is used in the Title Block image.

26. To fix this, **drag and drop any unmapped layers to empty layers** over to an empty mapped layer.
27. Once you have verified all image layers are mapped correctly with the template layers, click the **OK** button and the tool mapping dialog appears.

The NC Tool Mapping dialog displays the tools defined in the panel and the tools from the image file and how they are mapped. There are several automatic mapping algorithms provided:

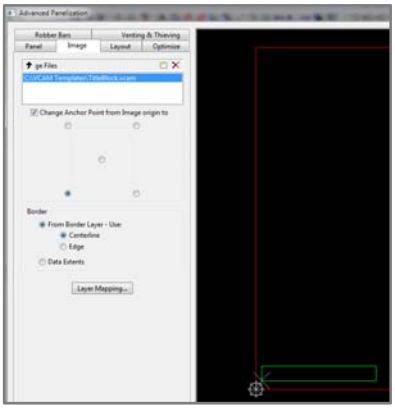


a. **Map:** Tools are mapped by their size to the existing tools

b. **Overwrite:** Tools defined in the image file maintain their tool number and are mapped to the same tool number in the panel tool table

c. **Ignore:** Tools defined in the image file maintain their tool number and are mapped to the same tool number in the panel tool table, the tool definitions from the image file are ignored.

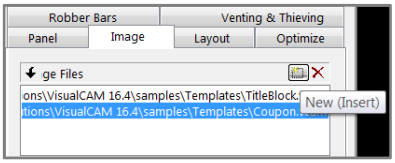
d. **Add**: All tools in the image file are mapped to unused tools in the Panel Tool table.



Using the **Map** method, we can see that Tool 1 in the Title Block image is now mapped to Tool 6 in the Panel Template.

28. Once you have verified that all NC Tools have mapped correctly click on the **OK** button and you will see the green outline of the Title Block Image in the preview display.

29. **Change the Anchor Point from Image origin to lower left**. The anchor point is the origin of the image file, if you need to move or rotate, this is the origin that the image moves from.

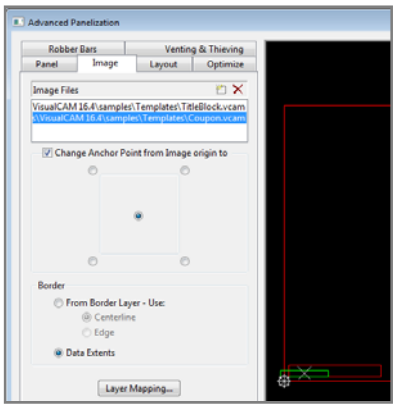


We also have the option of setting how the image is defined within the panel, if the image has a border layer defined use the border; otherwise the extents of the data in the image file are used.

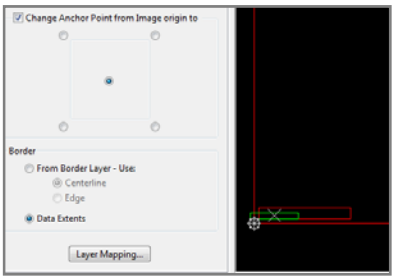
30. For the Title Block image file use **From Border Layer** and **Centerline.**

31. Click on the **New** button again and then the ... to browse for, select, and load the next image file **Coupon.vcam.**

NOTE: When the image is loaded, note in the preview window one image is outlined in green and the other image is outlined in red. Whichever image file is selected in the "Image Files" list is the active image and is indicated by the green outline.



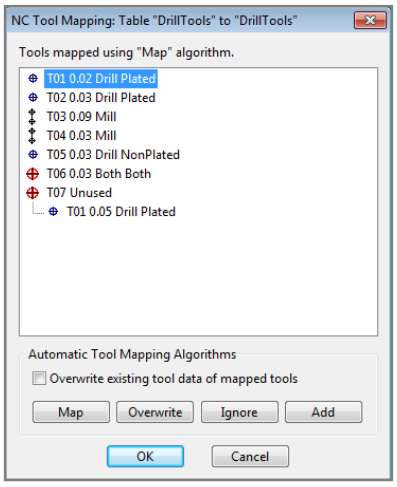
32. With the coupon we will use the center of the data as the anchor so click the "**Change Anchor Point from Image origin to"** to activate the selection controls and click the control in the **center**, note that the image moved slightly in the preview window and the location 'X' went to the middle of the image. We'll use the images extents so leave the "**Data Extents**" control checked.



33. Click on Layer Mapping... to verify that all layers from the coupon are mapped correctly with the panel layer types.

34. Click on the **OK** button.

The NC tool mapping dialog comes up and shows that **Tool 1** from the coupon will be mapped to **Tool 7** in the template.



35. Click on the **OK** button.

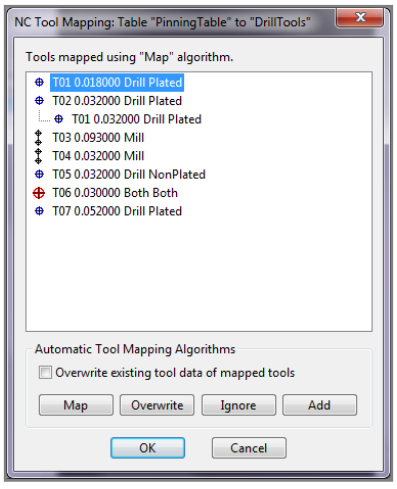
36. Click on the **New** button again and then the ... to browse for, select, and load the final image file **Pin_6Lyr.vcam**.

37. Click the "**Change Anchor Point to"** and select the control in the **center**.

38. Click on Layer Mapping... to verify that all layers from the coupon are mapped correctly with the panel layer types.

39. Click on the **OK** button.

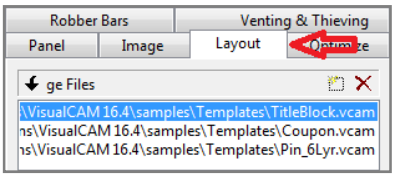
The NC tool mapping dialog shows **Tool 1** from the Pin image will be mapped to **Tool 2** in the panel template. Notice the size and plating in Tool 1 are the same as Tool 2 which has already been defined in the panel template.



40. Click on the **OK** button.

41. **Continue** to the next steps to position the images you have loaded.

To Position Images

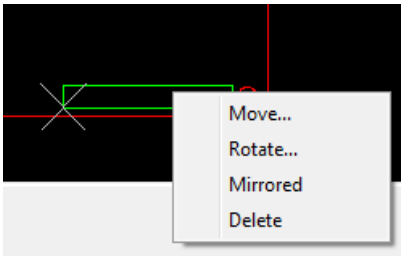


1. In the Advanced Panelization dialog box **click on the Layout tab**.

This page also has an image list like the Image tab, and similarly the highlighted image is the selected image and is the one that is being manipulated.

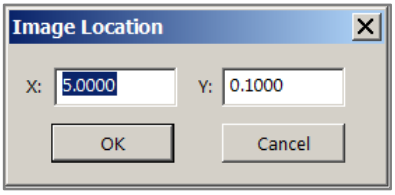
2. In the file list **highlight** the **TitleBlock.vcam** image file.

Since we only want one copy of this image we don't need to modify any of the other settings.

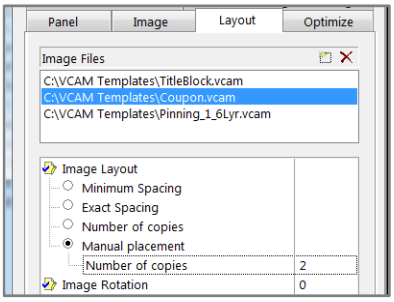


3. **Place your mouse cursor on the location 'X'** in the preview window **and right click**. A pop-up dialog will appear with several options.
4. Select the **Move…** option**.**

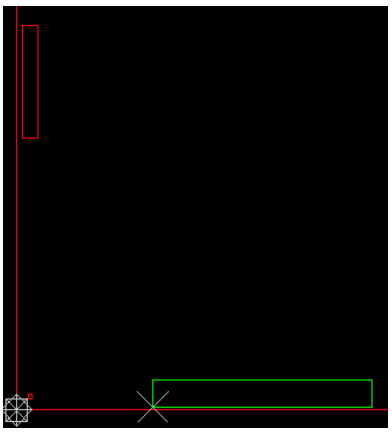
In the Image Location dialog box:



5. **Enter the values** for the location of the image, **X: 5.0, Y: 0.1**, so it is centered at the bottom of the panel.
6. Click on the **OK** button.
7. Back in the file list, **highlight** the **Coupon.vcam** file**.**
8. Under Manual Placement, change the **Number of copies** from 1 to **2**.
9. **Press the Enter key** on your keyboard to make the system accept the change.



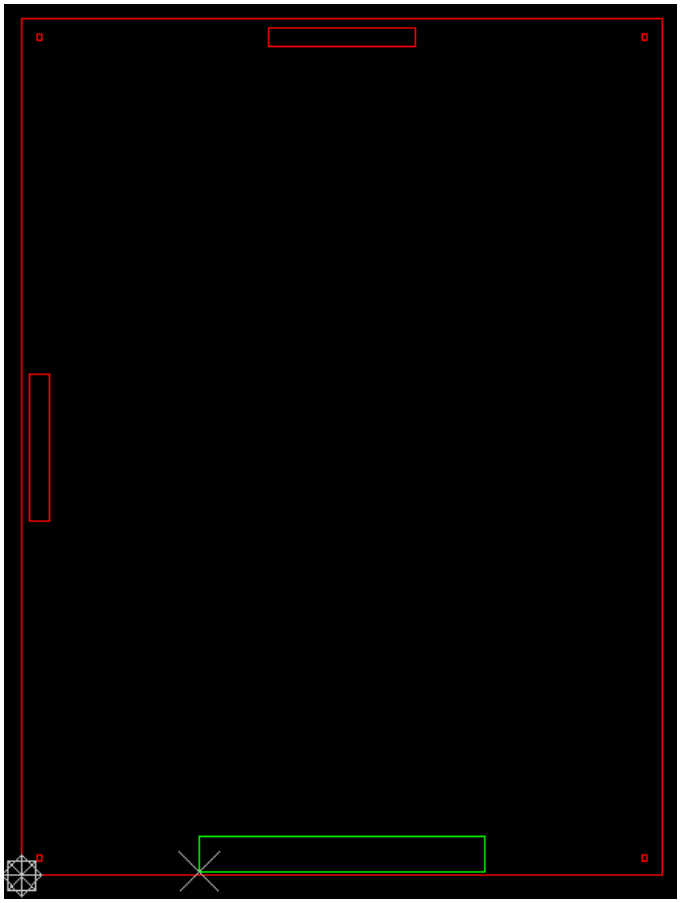
NOTE: In the preview window note that there are now two green outlines. One of the copies we want midway up the left side of the panel, rotated so that lies close to the edge of the panel, and the other copy we want midway across the top of the panel.



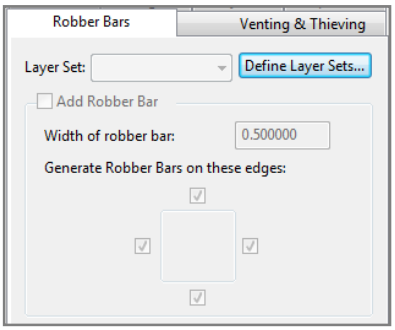
10. **Right click on the location 'X' of one of the images** and select **Move**.
11. Set its position to **X: 0.5, Y: 12.0**.
12. Click on the **OK** button to move the image**.**
13. **On the image you just moved** right click on the 'X' a second time and this time select **Rotate**.
14. In the Rotate Image dialog box enter **90.0.**
15. Click on the **OK** button to position the image correctly.

16. **Right click on** the 'X' of **the second image** and select **Move**.
17. Position it at **X: 9.0, Y: 23.5.**
18. Click on the **OK** button to move it to the correct location.
19. In the file list **highlight** the **Pin_6Lyr.vcam** file**.**
20. Under Manual Placement, change the **Number of copies** from 1 to **4.**
21. Press the **Enter** key.
22. Right click on one of the green images in the preview area and select **Move**.
23. Change its location to **X: 0.5, Y: 0.5**.
24. Click on the **OK** button.
25. Move the second image to location to **X: 0.5, Y: 23.5**.
26. Move the third image to location to **X: 17.5, Y: 23.5**.
27. Move the fourth image to location to **X: 17.5, Y: 0.5**.

When you have finished placing all the panel image files, the Preview screen should look like this:



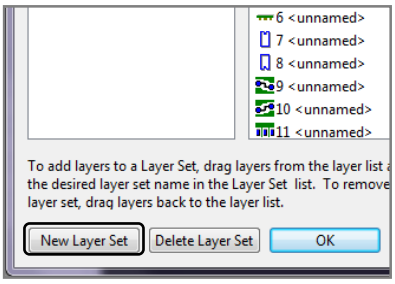
Define Robber Bars



1. In the Advanced Panelization dialog box click on the **Robber Bars** tab.

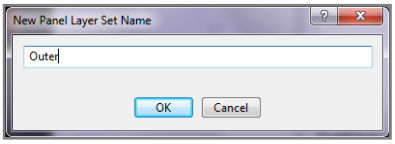
You will notice all the controls are grayed out. This is because Robber Bars are associated with layer sets and we haven't defined any layer sets yet.

2. Click on the Define Layer Sets... control button.



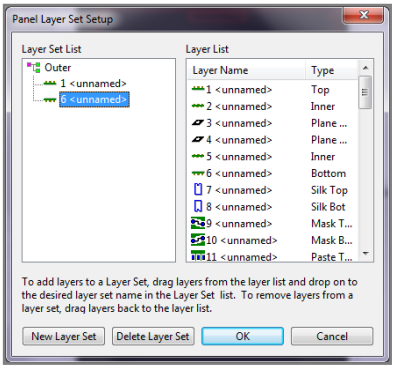
3. In the Panel Layer Set Setup dialog box click on the New Layer Set button to define a layer set.

In the New Panel Layer Set Name dialog box:



4. Enter the name **Outer** for your layer.
5. Click on the **OK** button.

In the left-side pane of the Panel Layer Set Setup dialog box you should now have an entry named **Outer**, to which you're going to assign the two outer layers (Top, Bottom).



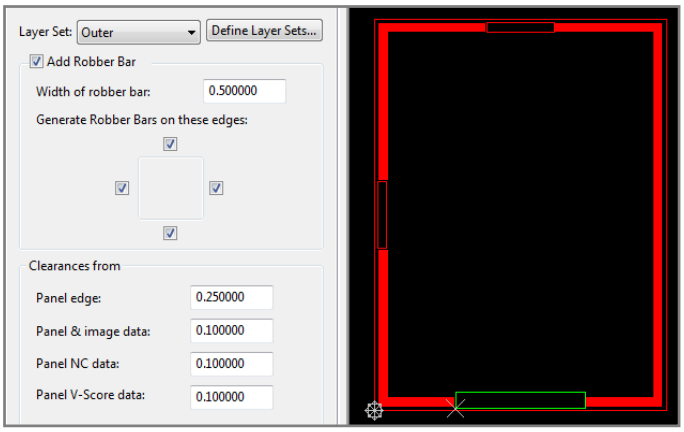
6. **Left click and drag** the **Top** and **Bottom** layers in the right-hand pane to the left pane and **drop them on Outer**.
7. Click on the **OK** button.

Outer is now displayed in Layer Set list control.

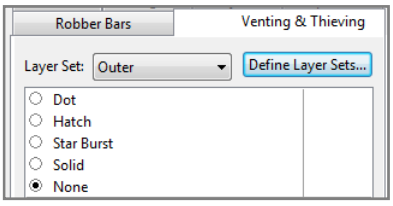
8. Check the **Add Robber Bars** control and you will see the robber bars in the preview window.

The remaining controls are now active and you can define the width of the robber bars and the distance they are placed from the panel edge, and control on which edges the robber bars are defined.

9. For this template we're going to use the default setting ½-inch robber bars located ¼ inch in from the edge of the panel on all four sides, so nothing needs to be changed:

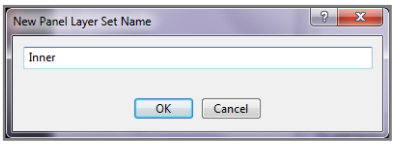


Venting & Thieving



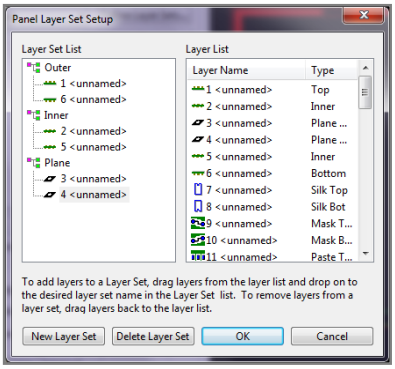
The final step in defining the panel template is to define the venting and thieving. For this we need to create layer sets for the inner and plane layers.

1. Click on the **Venting & Thieving tab**.
2. Click on the Define Layer Sets... control button.
3. In the Panel Layer Set Setup dialog box click on the New Layer Set button.



In the New Panel Layer Set Name dialog box:

4. Enter the name **Inner** for the first layer.
5. Click on the **OK** button.
6. **Repeat to create** another new Layer named **Plane**.



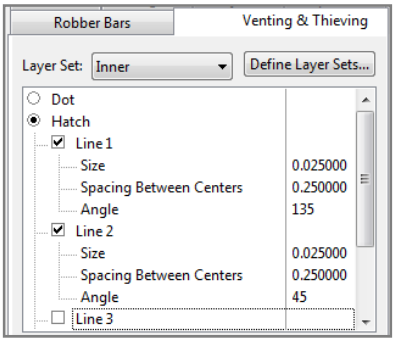
In the Panel Layer Set Setup dialog box:

7. **Drag and drop** the two Inner layers onto the Inner layer set. Drag and drop the two Plane layers onto the Plane Layer set.
8. Click on the **OK** button.

In the Advanced Panelization dialog box:

9. Drop down the Layer Set list and select the **Inner** layer set.

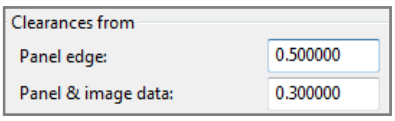
You are going to define a positive hatched pattern with diagonal lines. You can define up to 3 lines. The **Size** is the thickness of the line. The **Spacing Between Centers** is the distance between the lines, measured from their centers. The **Angle** is the angle of each line, in degrees. You can also choose Positive or Negative polarity.



10. Click on the **Hatch** control and you will see the parameter fields necessary to define the hatch pattern.
    a. Change the value for **Line 1** to:
        i. **Size: 0.025**, Press the **Enter** key.
        ii. **Spacing: 0.25**, Press the **Enter** key.
        iii. **Angle: 135**, Press the **Enter** key.
    b. Change the value for **Line 2** to:
        i. **Size: 0.025**, Press the **Enter** key.
        ii. **Spacing: 0.25**, Press the **Enter** key.
        iii. **Angle: 45**, Press the **Enter** key.

You can enter values for Line 3 or unselect the control to leave these out of the hatch pattern.

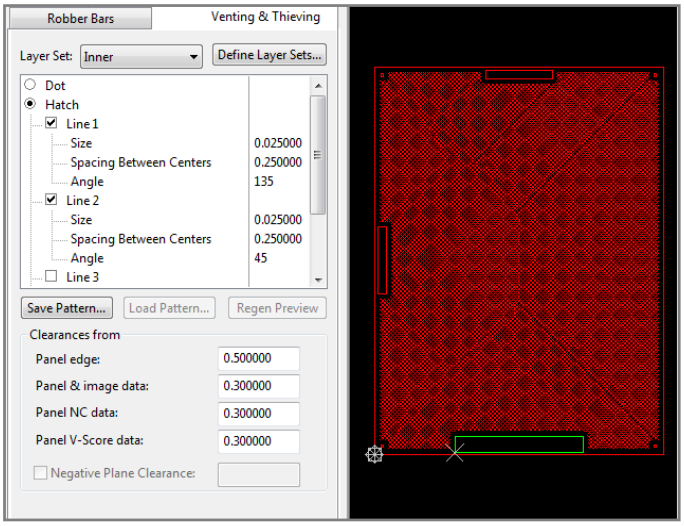
11. In the **Clearances** section, set the values:



    a. From **Panel edge: 0.5.**

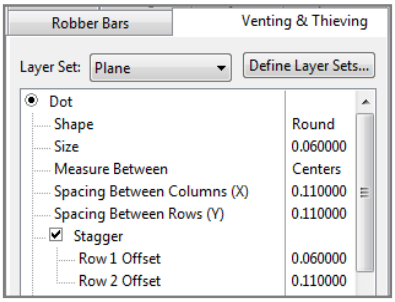
    b. From **Panel & image data**: **0.3**.

12. Click on the Regen Preview button to accept your Hatch pattern changes and update the preview window:



13. Drop down the Layer Set list and select the **Plane** Layer Set.

For these layers you're going to define a **Dot pattern with Stagger offset**.



14. **Change** the Dot controls. Press the **Enter** key after each one:

    a. Shape: **Round**

    b. Size: **0.06**

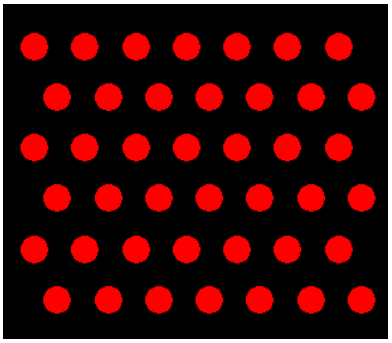
    c. Measure Between: **Centers**

    d. Spacing Between Columns (X): **0.11**

    e. Spacing Between Rows (Y): **0.11**

    f. Row 1 Offset: **0.06**

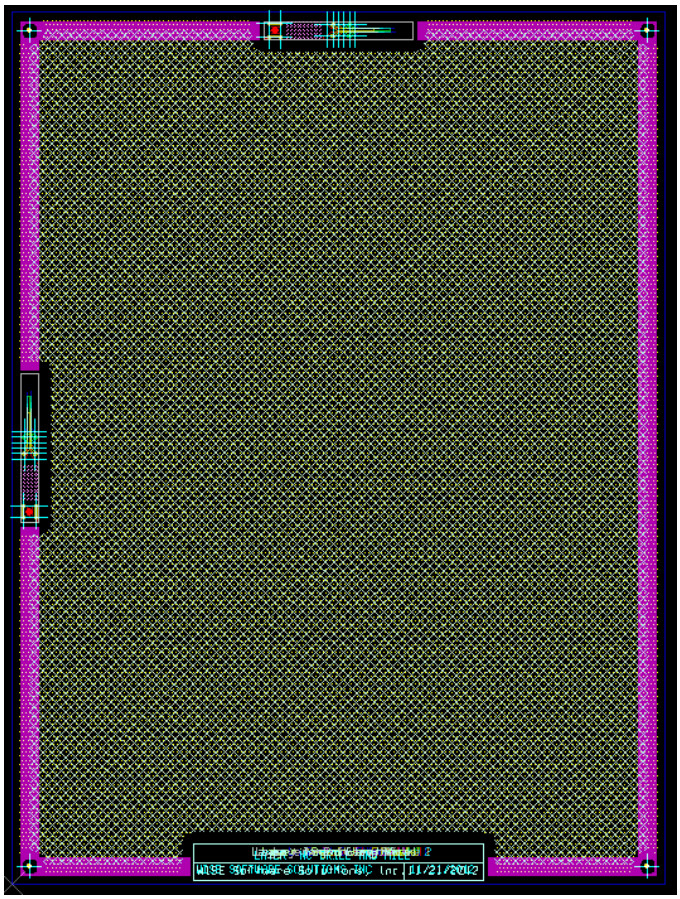
    g. Row 2 Offset: **0.11**



15. Click on the Regen Preview button to make the system accept your Dot pattern changes and to update the preview window. To see a close up view of the pattern as shown in the image at right, check ☑ Pattern Preview below the main viewing window.

16. Click on the **OK** button to create the panel template and your template will be displayed in the main VisualCAM window.

NOTE: If you cannot see the new Venting & Thieving patterns in the VisualCAM window, use the Colors bar to turn on the Inner and Plane layers and to change the colors.



Explore the power of the venting and thieving on your own with the other options available, including Star Burst and Solid patterns and using Negative patterns on complementary layers.

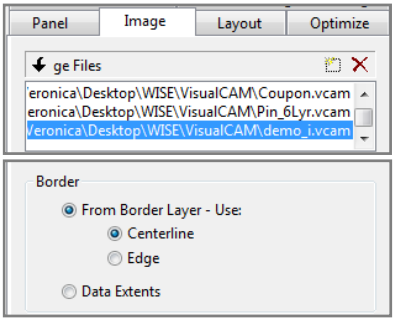
17. Use the **File > Save As** command to save your template as **Panel6lyr18x24.vcam.** Make sure to save it in a folder that you can easily find for later use.

This completes the Creating a Panel Template activity.

Activity: Panelize Using an Advanced Panel Template

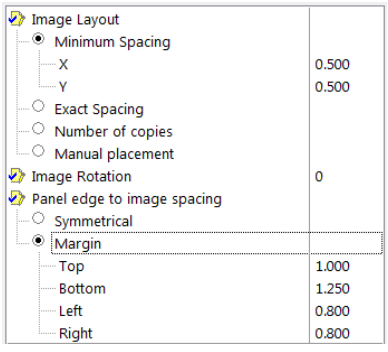
The purpose of this activity is to demonstrate how easy it is to panelize your design using a panel template. You will create a panel using the Template you just created (*see Create a Panel Template, page 122*) and the demo design included in the VisualCAM samples folder.

1. If it is not already, use **File > Open** to open your Panel Template **Panel6lyr18x24.vcam.**
2. Select the **Tools > Panelize > Advanced** command from the menus..
3. In the Advanced Panelization dialog box click on the **Image tab**.
4. Click on the **New** button and then the ... to **browse for, select, and load the file** C:\ProgramData\WISE Software Solutions\VisualCAM 16.4\samples**demo_i.vcam**.
5. Under Border select **From Border Layer** and use the **Centerline**.
6. Click on the **Layer Mapping** button to **verify the demo_i.vcam layers** are mapped to the correct panel template layers.
7. Click on the **OK** button.
8. In the NC Tool Mapping dialog box, **verify** that **the tools** are mapped correctly.
9. Click on the **OK** button**.**
10. Click on the **Layout tab**. All images listed will be included in the panel. The settings below each apply to the image you have highlighted in the list.

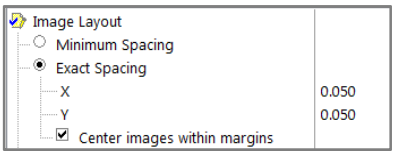


With the **demo_i.vcam** file highlighted you have four options for the **Image Layout placement**. To see how these work, change the settings and watch the Preview Window update:

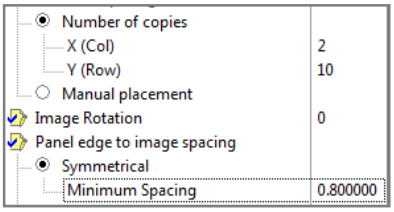
- **Minimum Spacing** defines the minimum amount of X and Y spacing between the images, measured from their borders. Change X: to 0.5 and Y: to 0.5. You will see the images overlay the coupon images that were already in the template. To adjust the images set the “Panel edge to image spacing”. Symmetrical allows you to specify the minimum spacing from the board edge. Margin allows you to define the offset from each individual edge, allowing for better control and placement. Choose Margin and set Top: 1.0 (the image moves away from the top edge), Bottom: 1.25 (the image moves up and closer together, which makes room for the title block on the bottom edge), Left: 0.8, and Right: 0.8.



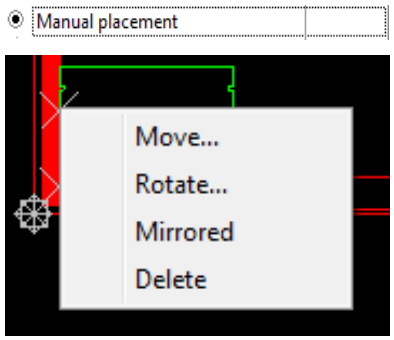
- **Exact Spacing**, where the images are spaced exactly the amount you specify in X and Y. If the resulting layout is not centered on the panel, you can have VisualCAM center the images but maintain the exact border-to-border spacing, by selecting “Center images within margins”.



- **Number of Copies** calculates the image-to-image spacing based upon the number of images you specify in the X and Y directions. For example if you change X: to 2 and Y: to 10 you will get 20 copies. Change the margins below to control the distance from the board edge. Or choose Symmetrical and set it to 0.8 to clear the board edges.



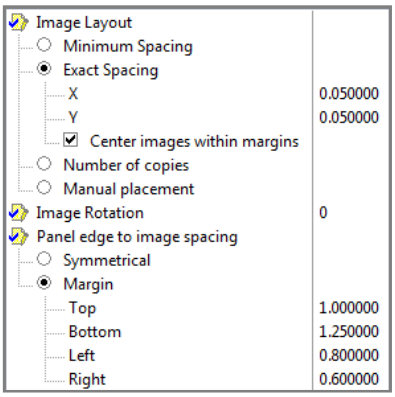
- **Manual Placement** allows you to specify the number of images you want in the panel, and then you can modify each individual image by right clicking within the green outline of the image and choose Move, Rotate, Mirror or Delete.



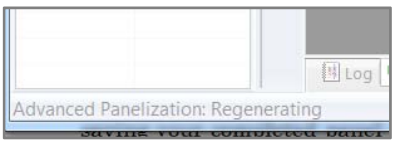
Another option in the Layout menu is the **Image Rotation –** Image Rotation Rotates all the images, around their insertion point, counter-clockwise. Note that this selection will override any manual rotation of individual images that were performed with the right-click shortcut menu. Click on the 0 and change to 90 to see how this works.

That briefly describes the options you have to place your design image in the panel template. The three settings **Image Layout, Image Rotation, and Panel edge to image spacing** all work together to give you better control of the image placement.

11. To complete the panel choose **Exact Spacing:**
    a. X: **0.05,** Y**: 0.05**.
    b. **Center images** within margins.
    c. **Image Rotation** leave at **0**.
    d. Panel edge to image spacing, choose **Margin:**
        i. Top: **1.0**
        ii. Bottom: **1.25**
        iii. Left: **0.8**
        iv. Right **0.6**
12. Click on the **OK** button.



You will notice the Advanced Panel: Regenerating prompt appears in the lower left corner. This may take a minute as VisualCAM is generating the actual data for Venting &Thieving, Robber Bars and placement of all the board images.



Once the panel is done generating you will see your completed panel in the Main window.

Verify that everything looks good, if so you are ready to save and export your files. If you see anything that you want to change, just start up the Advanced Panel command again and make those changes.

We are now finished with the panelization of this design. Save the panel design under a new name so that your template remains untouched.

13. File > Save As **Demo_Panel.vcam.**

You can now:

- Export your finished Panel design to send to manufacturing.
- Export to Gerber, IPC2581 or ODB++.
- Export your NC Drill/Mill files and Tool List if needed.

Review Questions

What types of files are needed in order to perform Advanced Panelization?

_____
_____
_____
_____

What is the best way to include title blocks, pinning holes and coupons in a panel?

_____
_____
_____
_____

What is the first step to begin panelizing with a template?

_____
_____
_____
_____

What are the four options for image layout placement?

_____
_____
_____
_____

Why is Layer Mapping important when placing an image in an advanced panel?

_____
_____
_____
_____

Assembly Reverse Engineering Part Data

Assembly Reverse Engineering (ARE) is an option for VisualCAM and allows for the automated processing of centroid (pick-and-place) extraction from virtually any Gerber data set or other data set where component footprints were not included.

In this lesson we will describe the procedures on how to reverse engineer data, to add part centroid locations, pin locations, part rotation, etc. to a database, for assembly.

Lesson Objectives

In this lesson you will do the following:

- Understand the overall Reverse Engineering process.
- Reverse engineer using IPC-D-356 data.
- Reverse engineer using a Centroid File.
- Reverse engineer using a Silkscreen Layer.
- Reverse engineer using Footprint Library.
- Reverse engineer using Select Pins (Manual Process).

Estimated Completion Time

Approximately 90 minutes.

Evaluating Your Data-Set

Before you can begin Reverse Engineering you need to evaluate the data you have available to you and determine the best method to process the job.

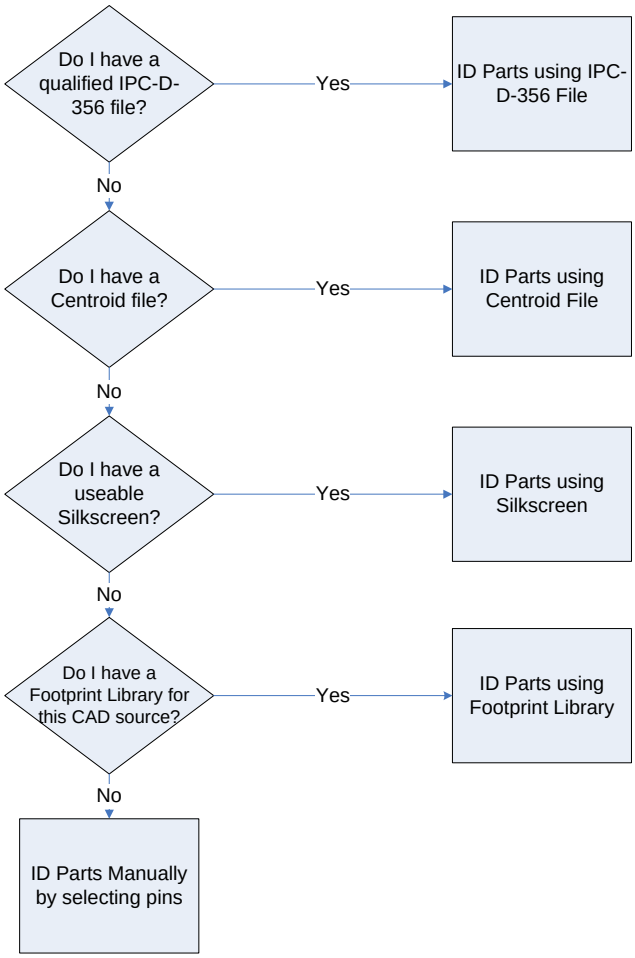
VisualCAM has 5 different automated methods available for reverse engineering component footprints:

1. Using IPC-D-356 data
2. Using a Centroid File
3. Using a Silkscreen Layer
4. Using a Part Footprint Library
5. Manually Defining Parts

These methods are listed in order of the most automated to the most manual process. You may find that you have to use a combination of these methods to complete the reverse engineering process for the entire Printed Circuit Board (PCB).

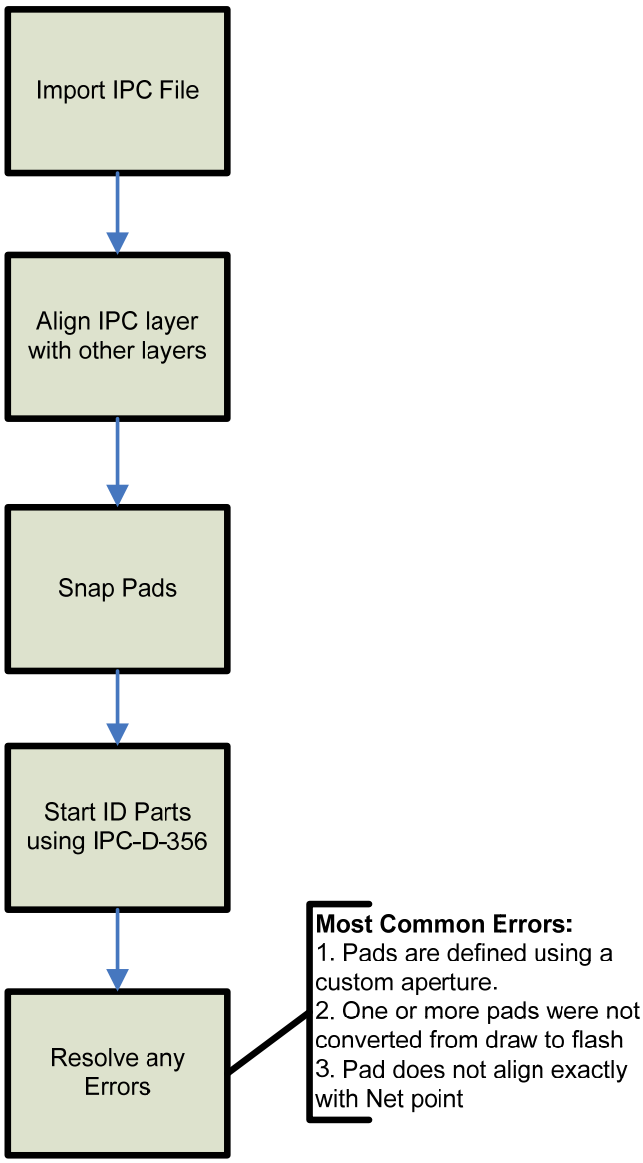
Directly after you have imported your Gerber data into VisualCAM is a good time to evaluate your data.

Decision Time

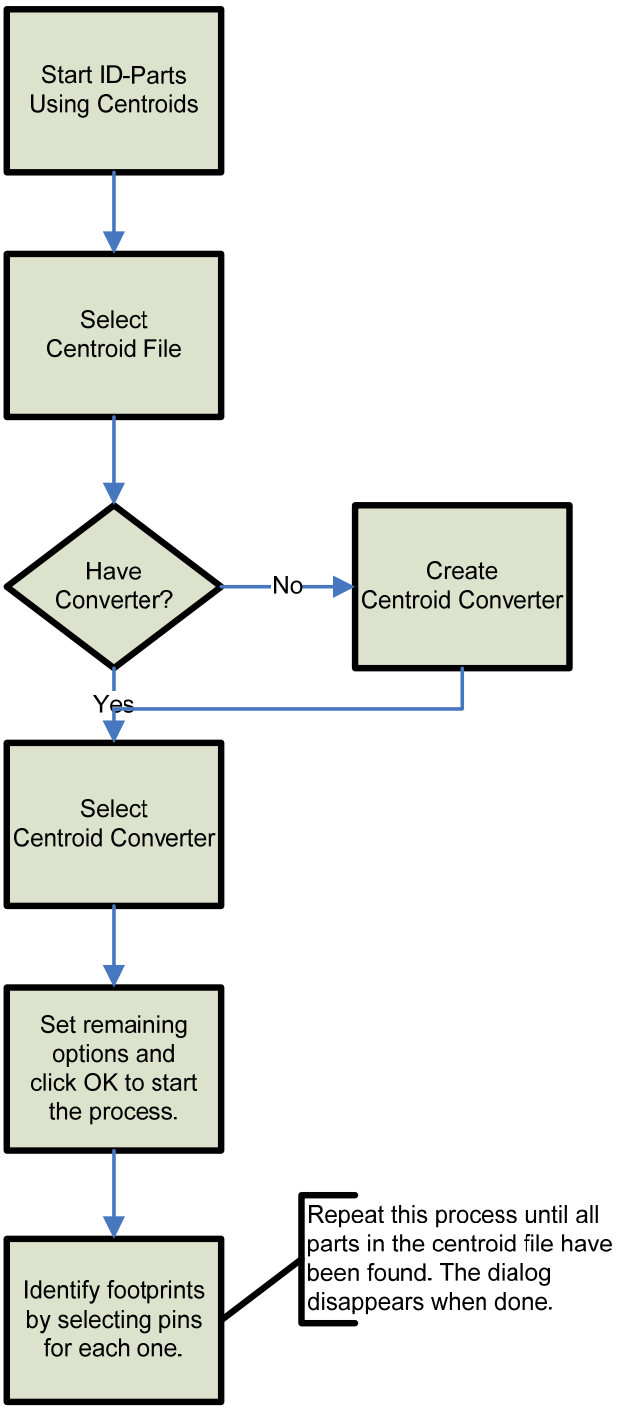


Assembly Reverse Engineering Process Flows

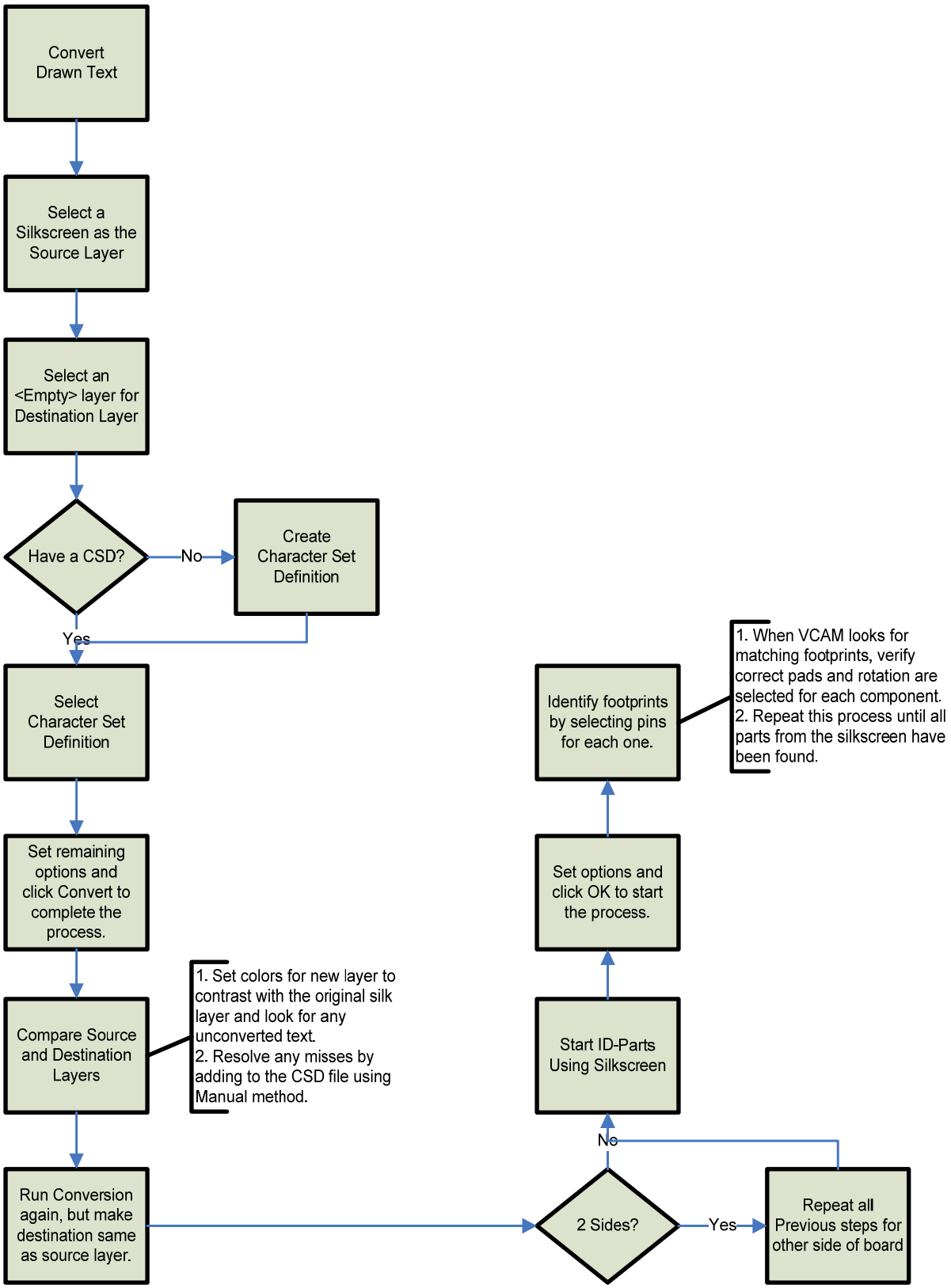
Process Flow Using IPC-D-356



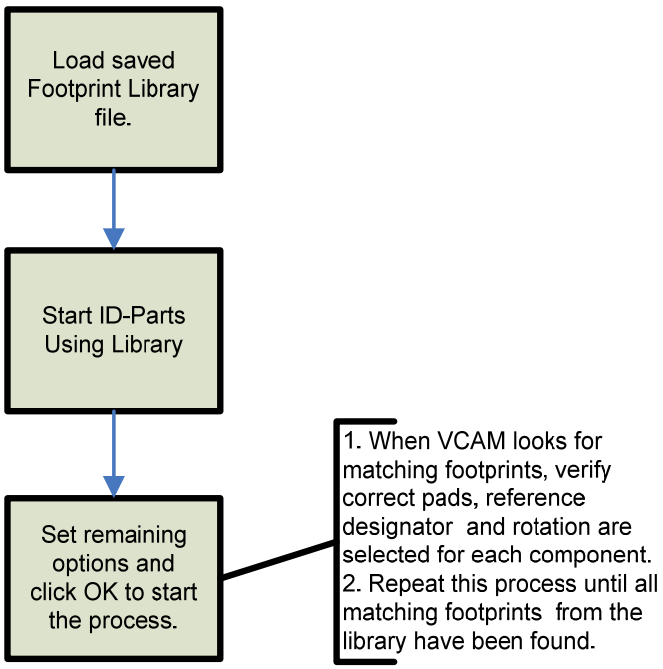
Process Flow Using Centroids



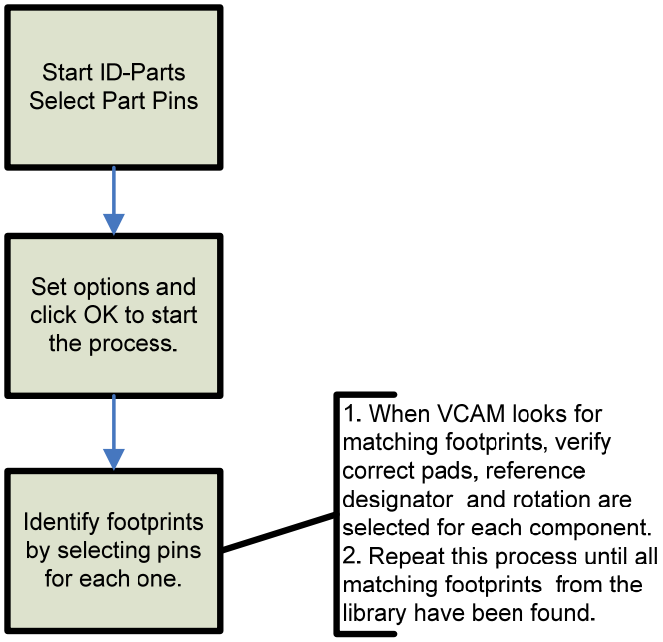
Process Flow Using Silkscreen



Process Flow Using Footprint Library



Manual Process Flow



Reverse Engineering Process Overview

NOTE: All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the Draws to Flash Conversion section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

For the IPC-Demo board you will be using the IPC-D-356 conversion process, page 145. After this board is complete, your instructor will lead you through the remaining activities using other data sets.

Step 1: Based on your findings from Evaluating Your Data-Set, page 139, choose one of the methods described for parts identification and complete the process as explained in the following activities.

Step 2: Part data, such as pin numbers and centroids, are displayed as "virtual entities". Use the **View > Parts** command to display/hide the part data, and the **Assembly > Parts > Query** command to view information on the parts in the **Item Properties Display**.

Step 3: If any of the footprint data, such as pin number assignments, needs to be corrected, use the **Setup > Footprint Library** command to view the footprint information and make the changes. The Footprint Library can be used for any global changes that need to be made to all parts that share a particular footprint. If you have a footprint that is incorrectly assigned to an individual part, use the **Assembly section of the Navigator** to change the assignment. You can also use the Navigator to alter things like rotation, reference designators, or delete a part altogether.

Step 4: If you need to identify any additional parts, see Manual Reverse Engineering, page 165.

Step 5: Identify any fiducials using the **Assembly > Fiducials > Identify** command.

Step 6: If you have a bill of materials (BOM) for your data, select the **Assembly > Import BOM** command to add any additional part or device information to the database prior to exporting. If the BOM contains device names or part numbers that do not match those in the footprint library, they will be added as necessary.

Step 7 When you are finished, use the **Assembly > Export FATF** command to export your data in the FATF format.

Step 8: When you are prompted to **Generate a netlist**, click **Yes**.

Step 9: If you are going to export your data to a FATF file and will use it for a test application, select the **Tools > Netlist > Set Netlist Information** command to specify whether your nets are power or signal, or use the **Assembly > Parts > BOM Report** command to generate an ASCII report that includes BOM and Centroid information.

Activity: Reverse Engineering Using IPC-D-356 Data

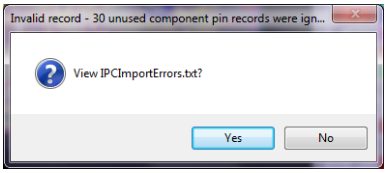
Reverse engineering from IPC-D-356 data is the most automatic and reliable method. However, note that your IPC-D-356 file must contain reference designators and pin numbers.

NOTE: Typically using IPC-D-356 data provides a thorough and complete result. However, it is possible that data inconsistencies may exist between your IPC-D-356 data and your Gerber data. VisualCAM will catch these errors and report them to you in the **Error Analysis Manager** for your review. You may be able to resolve these inconsistencies, but if you cannot then you will need to utilize one or more of the other methods to complete the reverse engineering process for parts that were not successfully identified using the IPC file.

NOTE: All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the **Draws to Flash Conversion** section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

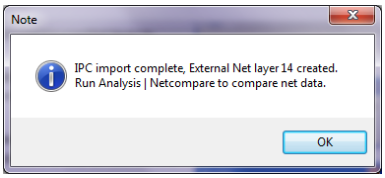
This activity should take approximately 15 minutes to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **File > Import > Netlist > IPC-D-356** command and import the netlist that came with your data set.
3. Select the file **0423C.IPC** found in the …TrainingFiles\IPC-Demo\CAD folder on your desktop.
4. Click on the **OK** button.
5. If you receive an Invalid record message, click **Yes** to view the text file.

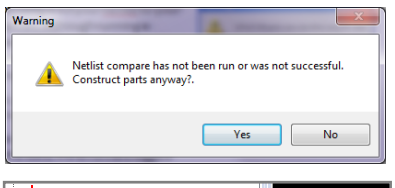


You will receive a message telling you that an IPC layer was created.

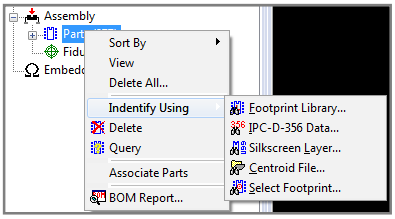
6. Click on the **OK** button.



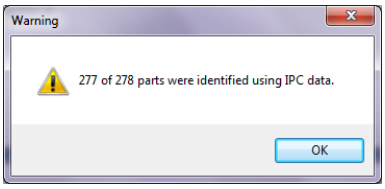
7. Use **Edit > Align Layers** and make sure your netlist layer is aligned to your layer data (*see Align Layers, page 64.*)
8. You may also want to run the **Edit > Snap Pads** command as well (*see Snap Pads, page 68.*)
9. Select the **Assembly > Parts > Identify Using > IPC-D-356 Data** command from the menus, or click the **ID Part – Use IPC-D-356** icon on the toolbar.



TIP: This command is also accessible from the right-click shortcut menu, in the **Assembly** area of the Navigator.



You are warned if you have not compared your netlist to your data, to check for any discrepancies. Although running a netlist comparison is recommended, it is not necessary to complete the reverse engineering process (*see the instructions for **Netlist Compare** at the end of this activity, page 147.*)

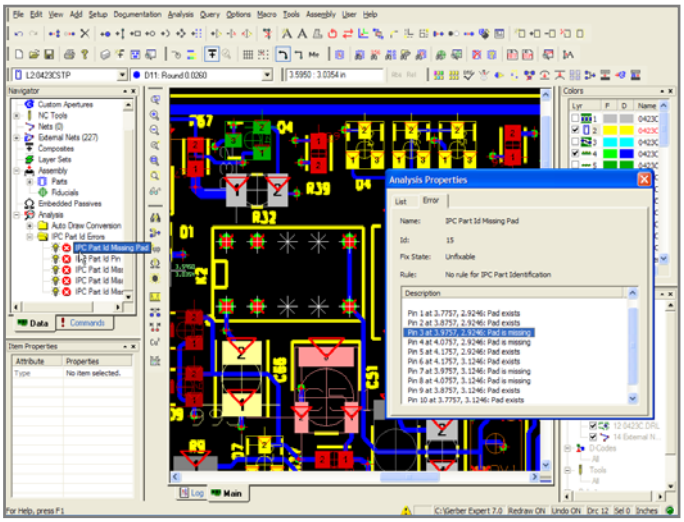


10. Click on the **Yes** button to continue.

VisualCAM scans the database, constructs the **Footprint Library**, and identifies the part instances in the workspace.

11. Click on the **OK** button to continue.

If any parts listed in the IPC file are not found in the database due inconsistencies between the data, the errors are logged in the Analysis area of the Navigator.



12. Review all errors in the Analysis Manager.

    a. The first error is a good example of data inconsistencies. Note in the graphic that the part "**K2**" has net points where there are no pads. The IC has been modified to omit these pins, so the design was modified as well, but the IPC file still has the nets listed. You have two options, delete the nets from the IPC file and run it again, or identify this part using the manual process.

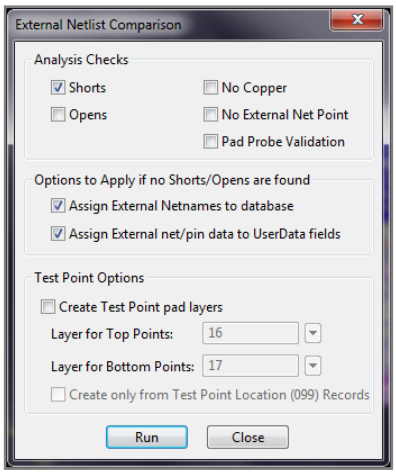
    b. The next two errors are for the edge connector, ignore these.

    c. The last two errors are for the SOT components **U5** and **Q5** where we had to manually convert the draws to flash for the center pads that were odd shaped.

The cause of the errors is that the position of the nets does not align with the center position of the center pads. This is because they are asymmetrical components and the net position was not adjusted for the pad design.

NOTE: Edge connectors (gold fingers) may be identified in the IPC-D-356 file for the purposes of the netlist, but they will show up as errors during reverse engineering because they have pins on both sides of the PCB. They do not actually represent components, so these errors should be ignored.

Netlist Compare

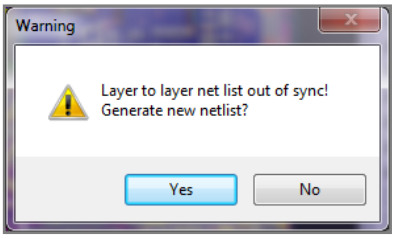


If you are going to export a FATF file for use with your Test Application, it is recommended that you run **Netlist Compare**.

13. Select the **Analysis > Netlist Compare** command from the menus.

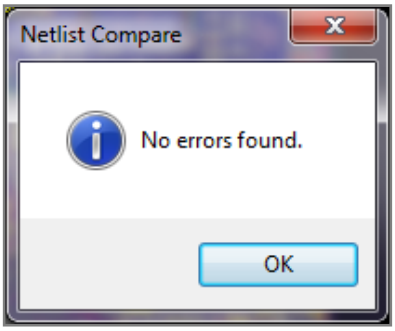
In the External Netlist Comparison dialog box:

14. Choose the Analysis Checks you wish to run. Select **Shorts** check as a minimum.
15. Under the Options, be sure to check:
    a. **Assign External Netnames to database,** and
    b. **Assign External net/pin data to User Data fields.**



This will apply all the netlist info that was imported from your IPC-D-356 file to your internal netlist, which will then be included in your exported FATF file.

16. Click on the **Run** button.
17. If you receive a Warning, click on the **Yes** button to Generate a new netlist (*see Generate a Netlist, page 88 for more information about the dialog box*.)



The system will let you know if any errors were found.

18. Click on the **OK** button to continue.
19. **Respond to any other Warning messages** you may receive.
20. **Save** the file.

Activity: Reverse Engineering Using a Centroid File

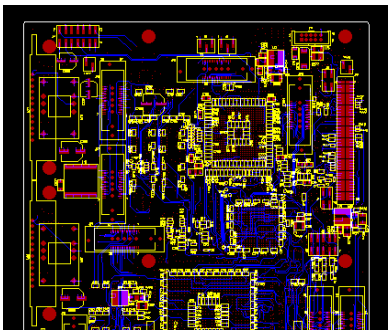
Part Centroid files can be used to reverse engineer part data for assembly. A converter file is also necessary, which is used to instruct the system how to read the data in the various columns of the centroid file. If you do not have a converter file, you can easily create a new one using the interactive Centroid File Converter (*see Create a Centroid File Converter, page 151.*)

NOTE: Typically a centroid file will have a complete listing of all components on the PCB. However, it is possible that you will receive a centroid file that is incomplete. Possibly because it was created with a specific revision of an assembly in mind where some parts are omitted from the BOM but still exist on the PCB. If this occurs, you will need to also utilize one or more of the other methods to complete the reverse engineering process for all parts in your design.

NOTE: All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the **Draws to Flash Conversion** section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

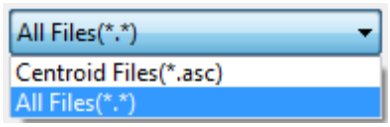
This activity should take approximately 15 minutes to complete.

1. Open the database labeled **cent_silk_demo.vcam** found on your desktop in the "**TrainingFiles\cent_silk_demo**" folder.
2. Select the **Assembly > Parts > Identify Using > Centroid File** command or Click the **ID Parts - Use Centroid File** icon.



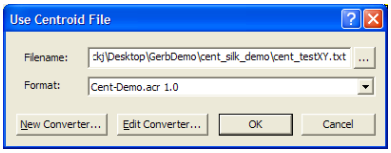
In the **Use Centroid File** dialog box:

3. Click the **Browse** button to search for file **cent_testXY.txt** in the TrainingFiles\cent_silk_demo folder on your desktop.
4. Click on the **Open** button.



NOTE: If you do not see **cent_testXY.txt** in your folder, specify **All Files** in the Use Centroid File dialog box.

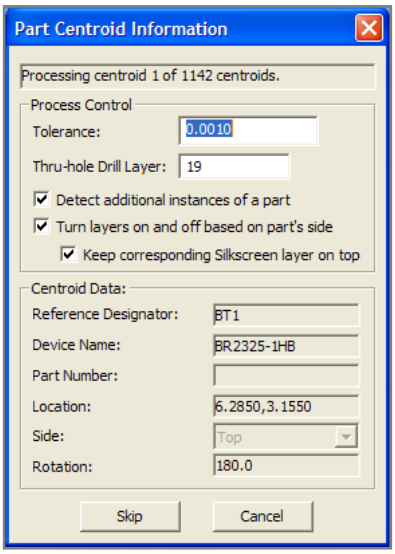
5. Back in the Use Centroid File dialog box, select the correct Centroid File Converter for the centroid file from the Format list. These converter files are located in the **CentroidConv** folder in your main VisualCAM folder



NOTE: If you do not have a Centroid File Converter, go to *Create a Centroid File Converter, page 151* to create one and then return to Step 6.

6. Click on the **OK** button.

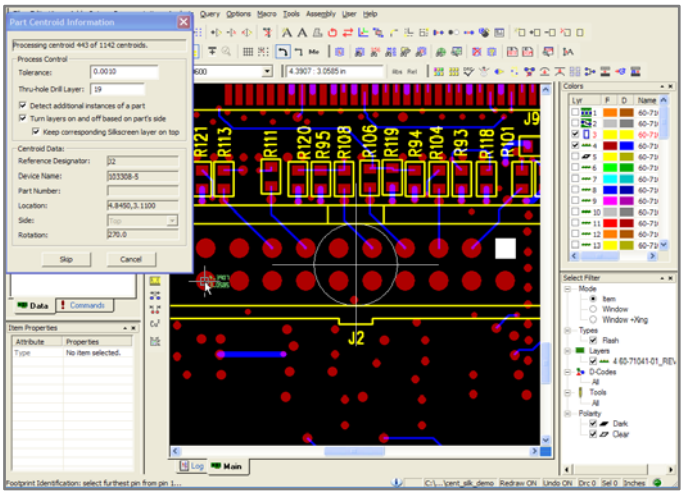
The Part Centroid Information dialog box appears.



7. Specify a **Tolerance** to account for small variances between part instances and for misalignment in a row of pins during the pin identification process.
8. Specify the **Thru-hole Drill Layer**. (This field is defaulted to the first NC-drill layer found in your layer stack.)
9. If you select **Detect additional instances of a part**, once a footprint is identified, all other parts that share the same device name or part number will be identified automatically and their associated reference designators assigned to them.
10. To aid in the visibility of the parts during the footprint identification process, select the **Turn layers on and off based on part's side** option. This will make only the layer that the current part is on visible in the workspace.
11. As an additional aid, you can select the **Keep corresponding Silkscreen layer on top** option to have the silkscreen data appear on top of the other data in the workspace (this does not affect the layer order).

The centroid of the first identified part is displayed as a target in the workspace. All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins.

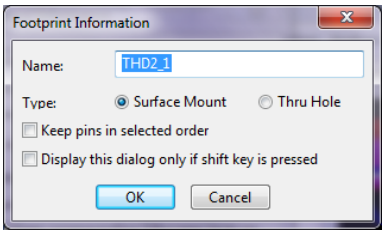
Use the "**+**" or "-" keys to zoom in or out on each part. (The first part will be **BT1** for the IPC-Demo job.)



12. Click on **Pin 1** for that part.
13. Select the pin that is farthest from Pin 1 (typically the opposite diagonal corner of the part).
    a. If the part has more than 2 pins, the system automatically finds other pins in between. If any are missed or selected by mistake, click on the pins to select/deselect them. Use the **Selection Filter**, if necessary.

TIP: When selecting pins for a component, you have the option to select each individual pin one at a time rather than selecting first and last pins. This allows you to force the pin numbering to follow the order you selected the pins. This is useful for parts with small pin counts, but with larger pin counts it is usually easier to allow the default pin numbering and then use the "**Set Pins**" tool in the footprint library to correct it later.

14. Press the **End** key when you are finished.



The **Footprint Information** dialog box appears. The system provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device; THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found.

15. Keep the default footprint name, or assign your own.
16. Click on the **OK** button.

NOTE: You will find that as you add footprints to the library, the Footprint Information dialog box sometimes does not appear after you identify the pins. This is because the system has matched an existing footprint to the part.

If you selected **Detect additional instances of a part** in the Part Centroid Information dialog box, you are told if any other part instances were found.

17. Click **OK**, and the system moves to the next part to identify.
18. You now **repeat the process** to identify the remaining parts. When no more parts remain, the Part Centroid Information dialog box disappears.
19. **You can skip a part at any time** by clicking the Skip button in the Part Centroid Information dialog box.
20. After processing several parts from the centroid file and you feel you are comfortable with the process, go ahead and click **Cancel** to abort the process. We will not process the entire board in this activity.
21. Close the job and **DO NOT SAVE** it! You will need it in the original state for the next activity.

NOTE: If you click **Cancel**, the entire process aborts, but any part information remains. If you select the **Assembly > Parts > Identify Using > Centroid File** command later, you will have the option to Ignore the parts in the centroid file that have already been identified (basically starting where you left off), or to Delete the existing parts and start over. If you delete the existing parts, the footprint information that you created will still remain in the library.

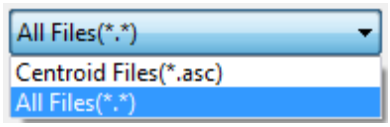
Create a Centroid File Converter

VisualCAM provides an interactive Centroid File Converter Creator that allows you to quickly create a centroid file converter file.

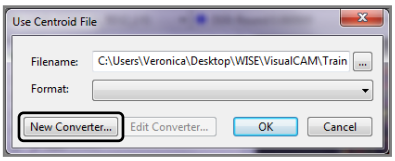
1. If it is not already, **open the cent_silk_demo.vcam** file found on your desktop in the **TrainingFiles\cent_silk_demo** folder.
2. If you haven't done so already, select the **Assembly > Parts > Identify Using > Centroid File** command or click the **ID Parts - Use Centroid File** icon.

If you haven't done so already, in the **Use Centroid File** dialog box:

3. Click the **Browse** button to search for file **cent_testXY.txt** in the TrainingFiles\cent_silk_demo folder on your desktop.



NOTE: If you do not see **cent_testXY.txt** in your folder, specify **All Files** in the Use Centroid File dialog box.

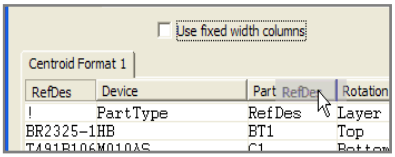


4. Click on the **Open** button.
5. Click on the **New Converter** button.

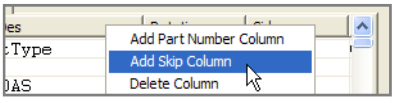
In the **Create Centroid File Converter** dialog box:

- The name of the centroid file being imported appears at the top, and your current centroid file is displayed in the **Centroid Format 1** tab.
- Your centroid file appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the centroid file; **you must reorder and resize the columns to match the centroid file contents**.
- **If your centroid file has columns in a fixed-width**, with no consistent delimiter (instead of tabs or commas, a combination of single, and/or multiple spaces are used--or no spaces at all in some columns--or any number of other text characters are used), select the **Use Fixed Width Columns** option.

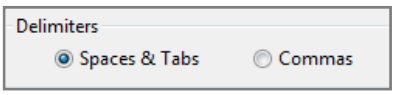
6. **Click on each column header** and drag it to the correct location.



7. **If you wish to delete a column header**, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later (it appears as an Add command in the right-click menu).
8. **If you have a column** of information in your file **that you wish to be ignored**, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.

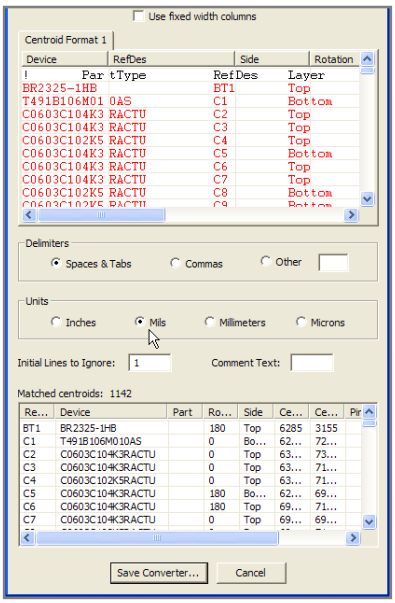


9. If your centroid file uses standard column delimiters (the Fixed Width option does not apply), **specify the appropriate Delimiter.**



NOTE: You do not need to adjust the column widths; you only need to adjust the order of the columns to match your file.

10. Select the Units of measure for your centroid file.
11. If you have any lines in the centroid file, such as header comments, specify in the **Initial Lines To Ignore** box how many lines at the beginning of the file should be ignored by the converter.
12. Also specify, in the **Comment Text** box, the symbol that leads any comment lines that may be intermixed with your centroid information.
13. As you identify the columns in your centroid file, VisualCAM automatically lists recognized centroids in the **Matched Centroids** list. All recognized centroids are also colored red in the Centroid Format view.
14. When you are finished, click the **Save Converter** button.
15. Name your converter and save it in the **CentroidConv** folder



The dialog box closes and your converter appears in the Format list of the Use Centroid File dialog box.

16. Return to Step 6 in the Reverse Engineer Using a Centroid File activity, page 148.

Activity: Reverse Engineering Using a Silkscreen

You can use silkscreen layers to reverse engineer your part data for assembly. This function requires that the reference designators in your silkscreen be "real text", so you will have to do some pre-processing of your silkscreen layer.

NOTE: You should review the silkscreen layer(s) in VisualCAM and get an idea for how complete it is. In other words, does every component on the PCB have their reference designator represented on the silkscreen and is the silkscreen lettering in good condition? (Are the letters and numbers chopped off or broken by other drawn lines or are they complete and contiguous drawn objects?) If the silkscreen is incomplete or choppy, you will need to also utilize one of the other methods to complete the reverse engineering process for all parts in your design.

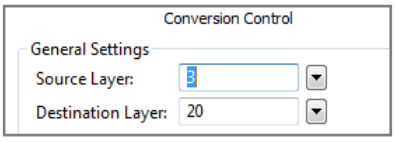
This activity should take approximately 15 minutes to complete.

Convert Drawn Data to Text

1. If it's not already, open the database labeled "**cent_silk_demo.vcam**" found on your desktop in the "**TrainingFiles\cent_silk_demo**" directory.
2. Select the **Tools > Convert > Drawn Text** command.

The **Drawn Text Conversion** dialog box appears. This dialog box contains 3 areas, accessible by tabs: **Conversion Control**, **Alphabetic Characters**, and **Digits and Symbols**.

3. On the **Conversion Control tab**, **select the layer number of your Source Layer**. By default, your silkscreen layer is selected. If you wish to convert the text on a different layer, click the arrow button to select it from a list.
4. The **Destination Layer** can be an empty layer (the default), or it can be the same as your Source Layer. If you make it your Source Layer, the drawn text on your silkscreen will be replaced with the resulting text.

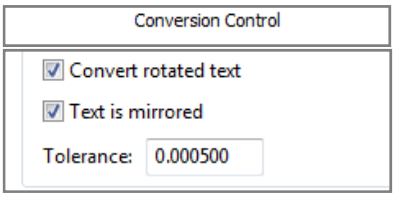


TIP: As a validation step, run the conversion once with an empty layer selected as the destination. After the conversion is run, make sure the colors for your Draws on the original Silk layer and the new layer are contrasted. You can then visually check to see if any needed drawn text from your silk layer was missed during the conversion process.

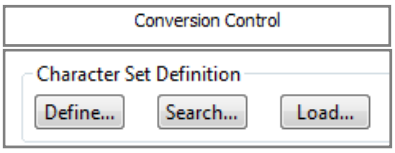
Once the conversion validation checks out, delete the extra layer and re-run the conversion process using a saved CSD file from your first run and select the **Same As Source** option for the destination layer.

NOTE: The converted text strings must exist on the designated Silkscreen layer for each side in order for the ID Parts using Silkscreen to work.

5. Select the **Convert Rotated Text** option if you want the system to look for text that is rotated.



6. Select the **Text Is Mirrored** option if the Source Layer is also a backside layer (the text appears mirrored in the workspace).
7. The **Tolerance** controls how close the replacement text must be in size to the original draws, to be considered a valid replacement. If no replacement is found, the draws are not converted.
8. The **Character Set Definition** features allow you to load, save, and search for files containing definitions of which characters are associated with which set of draws. This is useful if the text you are converting is identical from job-to-job.

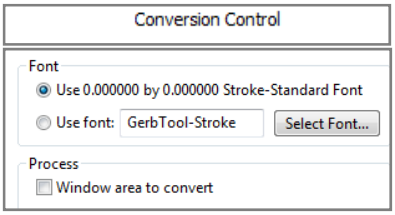


   a. **If you have a Character Set Definition file (see page 156) from a previous job** that you wish to use, click the **Load** button and select the file.

      If you want the system to search a folder containing multiple Character Set Definition files, and determine which one is most appropriate for converting the text on the Source Layer, click the **Search** button and identify the folder that the files are in, **OR**

   b. **To identify each character in your silkscreen and tell the system what it is**, continue with the steps in the section **Define a Character Set, page 156, and then return to Step 9**.

9. **Once you have defined a character set (see page 156)**, you need to tell the system which Font to use for the resulting text. By default, it will use the **GerbTool-Stroke Font** of the same size as the largest character defined in the character set.

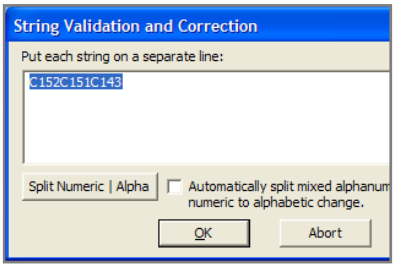


   If you wish to use a different font, or change the size of the GerbTool-Stroke font, click the **Select Font** button and select your desired font and the applicable dimensions of the font.

10. **If you want to convert only draws that are in a specific area** of the Source Layer, click the **Window Area To Convert** option. This is useful if you have different character definition sets that you want to use to convert text on the Source Layer. Otherwise, the entire layer is searched.
11. Click on the **Convert** button.

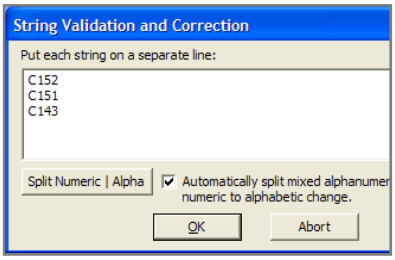
If you selected the **Window Area To Convert** option, you are prompted to define the windowed area. The system searches the Source Layer for text to convert.

Instead of just converting one character at a time, the system groups together your text into strings. Since the primary purpose of converting drawn text is for converting reference designators, this function is designed to insure that reference designators that are placed close together are not accidentally grouped into the same string. Strings consisting of only alphabetic characters (ABC), numeric characters (123), alphabetic characters followed by numeric characters (ABC123) or numeric characters followed by alphabetic characters (123ABC) are automatically converted to text. When strings consisting of multiple groups of alphabetic and numeric characters (AB12CD or 12AB34) are found, the system zooms to it in the workspace, and the String Validation And Correction dialog box appears.



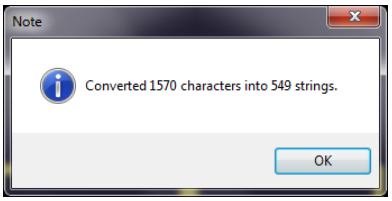
12. You can split the reference designators apart by manually placing them on separate lines in the dialog box, or (since reference designators typically are letters followed by numbers) click the **Split Numeric|Alpha** button to have VisualCAM separate the text between a number followed by a character.

    For example, if "C70R62" is identified, clicking the button will separate the text into "C70" and "R62".



13. You can have VisualCAM separate the text like this automatically by selecting the **Automatically Split Mixed Alphanumeric Strings... option**.

When the conversion process is complete, a confirmation message appears and the new, true text now appears in the workspace.



14. Click on the **OK** button.
15. **If you ran the conversion on an empty layer**, delete the extra layer and re-run the conversion process using the saved CSD file from your first run and select the **Same As Source** option for the destination layer.
16. If you are finished with this process**, close the** Drawn Text Conversion **dialog box and go to** **Reverse Engineer Using a Silkscreen, page 160**.

Define a Character Set

If you select **Automatic Identification** *(the recommended process)*, you specify parameters that the system will use to automatically find characters for you to identify. If you select **Manual Identification**, you are required to create a window around each character, and tell the system what it is.

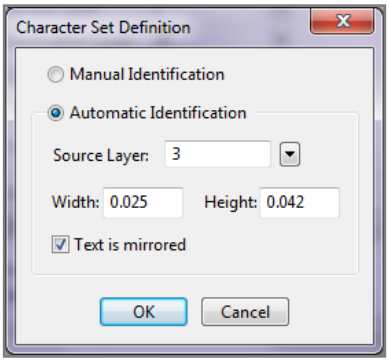
When a character set is used to convert drawn text, you specify a single font and size to use for the true text. Therefore, when defining a character set, you should only include characters of approximately the same size. For example, if you ultimately only want to convert reference designators to text, do not identify characters in title blocks or other non-reference designator text at the same time. Otherwise, the size differences will be lost during the conversion, and you will have some true text characters that are larger than the original drawn text. If you want to convert text of different sizes, and maintain those differences, you must define separate character sets for them.

Similarly, if there are drawn characters using different fonts, and you wish to maintain the different font in the conversion to true text, define separate character sets for the different fonts.

Automatic Identification

1. Click the **Define** button in the **Drawn Text Conversion** dialog box.

The Character Set Definition dialog box appears.



2. Select the **Automatic Identification** option.
3. The **Source Layer** can be the same silkscreen layer that you specified in the main Conversion Control settings, or a different layer that you want to use for creating the character definitions (at this point, you are only creating the character definitions, not actually converting the draws).
4. Specify the Height and Width of the largest character you wish to identify. This helps the system filter out draws that are not characters.

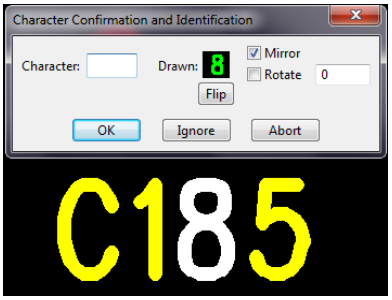
   If you are unsure of the height and width, use the **Query > Measure > Point To Point** command to measure the largest character in the workspace that you wish to identify in this character set. Enter dimensions that are rounded up slightly from the delta X/Y found in lower left corner of your workspace.

   Measure: DX:0.0229 DY:0.0418 Distance:0.0477

5. Select the **Text Is Mirrored** option if the Source Layer is also a backside layer (the text appears mirrored in the workspace).
6. Click on the **OK** button.

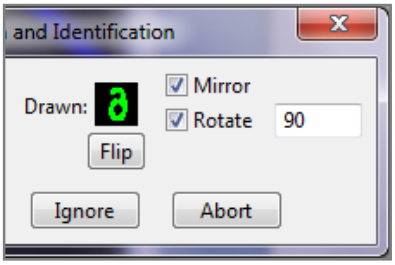
VisualCAM analyzes the Source Layer, and if characters were found, the **Character Confirmation And Identification** dialog box appears.

You now have to teach the system what character each set of draws represents.



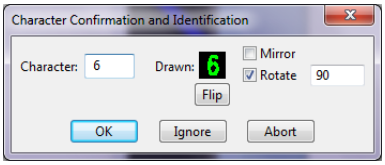
The system highlights an identified character in the workspace, and shows a copy in the **Drawn** box. The copy should appear in the correct (upright, non-mirrored) orientation in the Drawn box.

7. **If the character does not appear correct in the dialog box**, select or deselect the **Mirror** option and/or the **Rotate** option (and specify the angle of rotation) as appropriate.



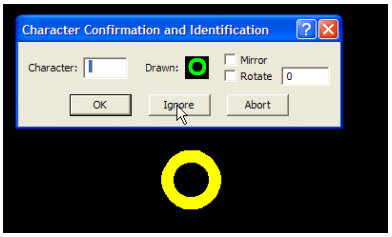
NOTE: If no characters were found, verify that you have specified an appropriate Height and Width for your characters. If the values are too small, nothing will be found.

8. **When the character appears correct**, **type the text character** it corresponds to in the Character box (using the correct capitalization)
9. Click on the **OK** button.



The system asks you to identify another character.

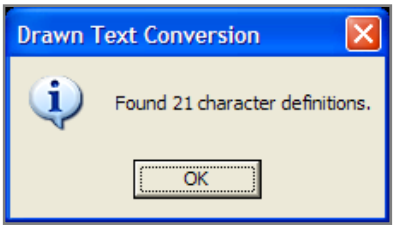
10. **If the system has identified something that is not actually a character**, just click the **Ignore** button to skip it and move on to another character to identify.



NOTE: If you are asked to identify several characters that appear to be the same. It means that each of the characters have slight variations in the way that they were drawn. It is in your best interest to identify each of these for accuracy and ultimate character conversion.

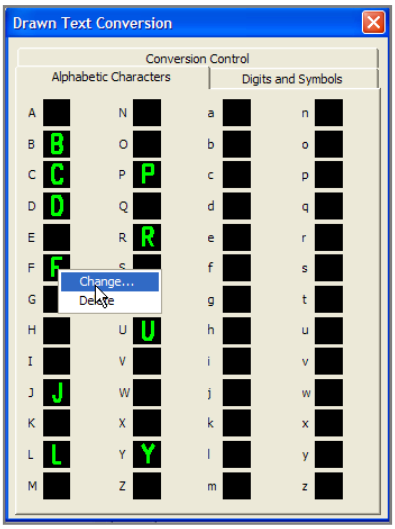
When VisualCAM has gone through all the sets of draws found, a confirmation message appears that tells you how many character variations were identified.

11. **Click** on the **OK** button.



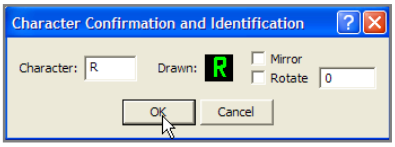
You are returned to the Drawn Text Conversion dialog box.

12. **If you want to see what characters have been defined in the selected character set**, click the **Alphabetic Characters** or **Digits And Symbols** tab at the top of the dialog box. You can see every text character and the corresponding drawn characters that have been associated with them (they are split into two tabs merely because of space limitations in the dialog box). When multiple definitions for the same drawn character exist, then arrow buttons are displayed beside the drawn character so that you can see all of them. The drawn characters are all displayed at the same scale, based on the largest character defined in the character set, so drawn characters will appear to be different sizes in the display.



13. **The characters in the character set may be modified or deleted** by clicking on the drawn character. A shortcut menu appears, with two options.

    If you select **Change**, the **Character Confirmation And Identification** dialog box appears.



    If you select **Delete**, the selected drawn character is deleted from the character definition set. (Use caution, as deleting a character cannot be undone with the **Edit > Undo** command.)

14. **If any characters were missed**, they can be added by using the manual method described below, or rerunning the automatic method with different parameters.

15. **Save the character set definition** for future use in other jobs, or you wish to define multiple character sets for this job.

    a. Click on the **Save** button and save it as a **.csd** file in the VisualCAM **CharDefs** folder.

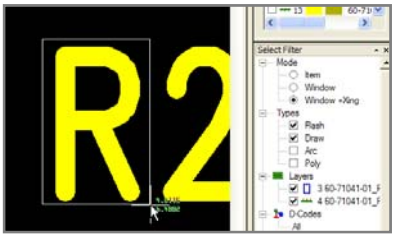
16. **Return to Step 9** in Convert Drawn Data to Text, page 154, **OR continue to the next section, Manual Identification.**

Manual Identification

1. Click the **Define** button in the **Drawn Text Conversion** dialog box.

The **Character Set Definition** dialog box appears.

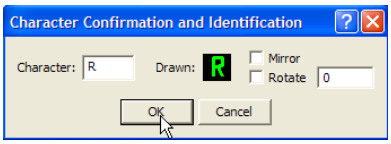
2. Select the **Manual Identification** option.
3. Click on the **OK** button.
4. You are now prompted to create a window around a drawn character in the workspace. Use the **Selection Filter** as an aid, if necessary. By default, you are placed in **Windwo+Xing** mode. Click on a corner of a selection window, and then on another corner to surround a drawn character. The character will be highlighted as soon as it is selected.



TIP: Use the arrows, +, and - hotkeys to pan and zoom to a desired character. You want to make sure that all the draws associated with a character are selected, or you may end up with undesirable results when you perform the conversion to text.

If not all of the desired drawn data is selected, you can continue to select data until everything you want is highlighted. Note that the selecting data works as a toggle when selecting drawn characters. If any data is already highlighted (selected), and you select it again, then highlighting is removed and it is no longer selected. This is useful if more draws are selected than you intended (for example, part of an adjacent character is also highlighted). You can also press the **Esc** key to gradually un-select data.

5. When you have selected the drawn character, click the **End** button.



The **Character Confirmation And Identification** dialog box appears. You now have to teach the system what character that set of draws represents. A copy of the drawn character appears in the **Drawn** box.

6. **Make sure the character appears in an upright, non-mirrored orientation**.
7. **If the character is mirrored or rotated**, or deselect the **Mirror** and/or **Rotate** option (and specify the angle of rotation) as appropriate.
8. **When the character appears correct**, **type the text character** it corresponds to in the Character box (using the correct capitalization)
9. Click on the **OK** button.
10. **Repeat the steps** to identify all the drawn text.
11. When you are done, click the **Esc** key to return to the **Drawn Text Conversion** dialog box.
12. **If any characters were missed**, they can be added by using

the manual method described above, or rerunning the automatic method with different parameters.

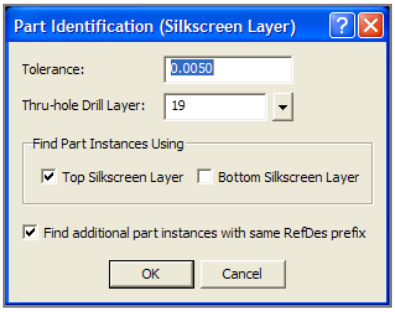
13. **Save the character set definition** for future use in other jobs, or you wish to define multiple character sets for this job.
    a. Click on the **Save** button and save it as a **.csd** file in the VisualCAM **CharDefs** folder.
14. **Return to Step 9** in Convert Drawn Data to Text, page 154.

Reverse Engineer Using a Silkscreen

NOTE: All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the **Draws to Flash Conversion** section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

1. **If you haven't done so already,** convert your reference designators on your silkscreen layers to "real text" (*see **Convert Drawn Data to Text**, page 153.*)
2. Select the **Assembly > Parts > Identify Using > SilkScreen Layer** command from the menus.

The Part Identification dialog box appears.



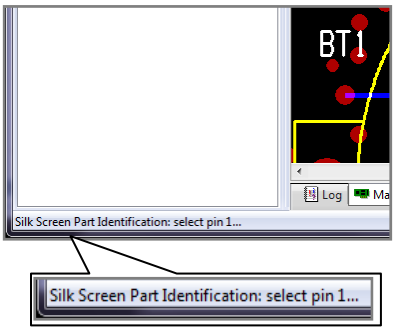
3. Specify a **Tolerance** to account for small variances between part instances and for misalignment in a row of pins during the pin identification process.
4. Specify your **Thru-Hole Drill Layer**.
5. Select whether to find parts using the **Top Silk Screen Layer**, **Bottom Silk Screen Layer**, or **Both**.

During the part identification process, you will be asked to identify the pins that correspond with each reference designator in the silkscreen.

6. Select the **Find additional part instances with same RefDes prefix** option if, after you identify each part footprint, you want the system to automatically identify any parts that the system finds with an identical footprint, that also share the same Reference Designator prefix.
7. Click on the **OK** button.

VisualCAM checks to see if any duplicate reference designators exist on the silkscreen, and if so, you have the option to generate and view a report with the reference designators and their X:Y locations.

The system will highlight the first part and you will be prompted to select the pins for the first reference designator.



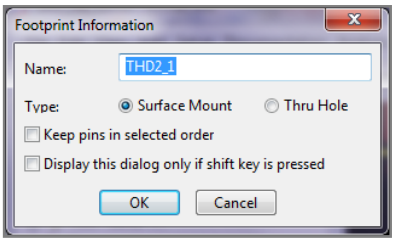
8. Using the Selection Filter and +/- keys to zoom in or out, if necessary**, select the first pin** in the part that corresponds to the highlighted reference designator.
9. **Select the pin that is farthest from Pin 1** (typically the opposite diagonal corner of the part).

   If the part has more than 2 pins, the system automatically finds other pins in between.

   If any are missed or selected by mistake, click on the pins to select/deselect them.

   Use the Selection Filter, if necessary.
10. Press the **End** key when you are finished.

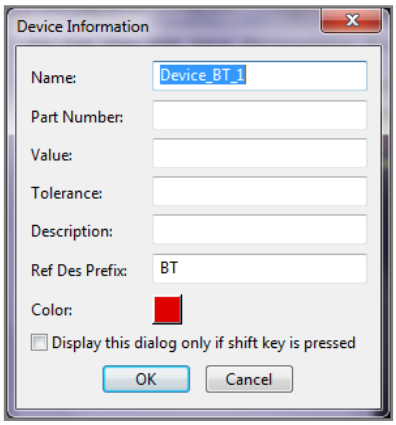
The Footprint Name dialog box appears.



VisualCAM provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device, THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found.

11. **Keep the default footprint name**, or assign your own.
12. Click on the **OK** button.

The Device Information dialog box appears.



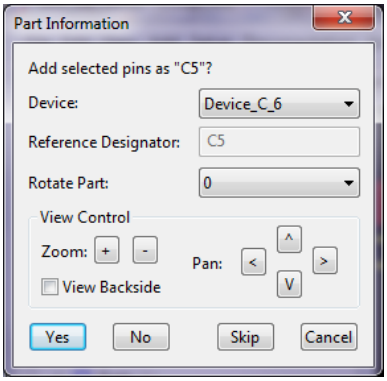
13. **Keep the default information** that the system created for the device, **or specify** the name, number (optional), and the Reference Designator Prefix you want associated with the device that you just identified.

All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins.

14. **Select a color** by clicking on the color in the dialog box, **or accept the system default**.

If you selected **Find additional part instances with same RefDes prefix**, and VisualCAM found other pads that appear to use the same footprint as the part you just created, it highlights the pins and you are asked if they should be identified as another part. The system uses the unused reference designator that is located closest to the

pins.



In the Part Information dialog box:

15. **If the suggested information is correct**, click **Yes** to identify that part.

    If the part information is not correct, you can specify a new device name to associate with it, or rotate the part.

16. **If the identified reference designator is not correct** for the highlighted pins, click **No**.

    The system will move on to any additional footprint matches that it detects, if any.

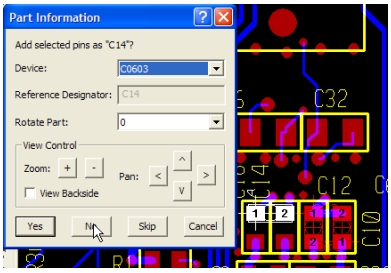
17. **If you find that the footprint type that is being searched on for the reference designator is not correct** type that the reference designator belongs to, click **Skip**.

    The system will move on to any additional reference designators with footprint matches that it detects, if any.

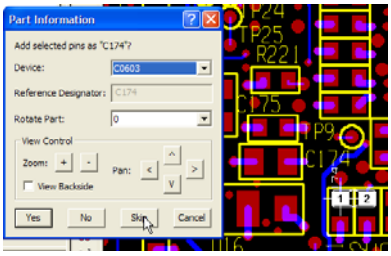
18. **Repeat the process** to identify the remaining parts. When no more parts remain, the process ends.

19. After processing several parts from the silkscreen of the "cent_silk_demo" job and you feel you are comfortable with the process, go ahead and click **Cancel** to abort the process. We will not process the entire board in this activity.

20. Close the job and **DO NOT SAVE** it! You will need it in the original state for the next activity.



Wrong pins for C14. Should be vertical pins next to C12. Click No.



*Wrong footprint for C174, should be larger pads. Click **Skip**.*

NOTE: If you click **Cancel** in the Device Information dialog box, you are prompted to select the pins associated with the highlighted reference designator again. If you want to abort the process entirely, press the **Esc** key. The process aborts, but any part information remains.

If you select the **Assembly > Parts > Identify Using > SilkScreen Layer** command later, you will have the option to **Ignore** the parts that have already been identified (basically starting where you left off), or to **Delete** the existing parts and starting over. (Note that if you delete the existing parts, the footprint information that you created will still remain in the library.)

Activity: Reverse Engineering Using a Part Footprint Library

VisualCAM has the ability to save and load its own footprint library file. If you have a standard footprint library that you use for multiple boards, you can use it in reverse engineering.

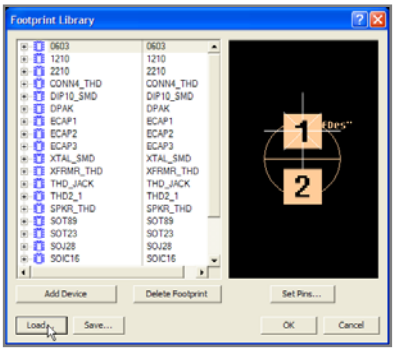
NOTES: This function can also be used after you have used any of the other Part Identification commands, and had to correct any data errors that required deleting a part instance and recreating it or if you had incomplete or inconsistent data sources.

All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the **Draws to Flash Conversion** section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

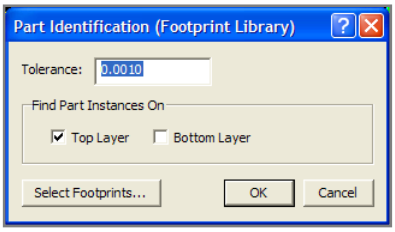
This activity should take approximately 15 minutes to complete.

1. Open the database labeled **FLIB-Demo.vcam** found on your desktop in the **…TrainingFiles\FLIB-Demo** directory.
2. Select the **Assembly > Setup Footprint Library** command, or click the icon in the toolbars.

The Footprint Library dialog box appears.



3. Click on the **Load** button, and select the appropriate **.fpl** file to import.
4. Select the **FLIB-Demo.fpl** file from the**…TrainingFiles\FLIB-Demo** directory.
5. Click on the **OK** button in the **Footprint Library** dialog box.
6. Select the **Assembly > Parts > Identify Using > Footprint Library** command or click the icon on the toolbars to identify the part instances for each footprint.



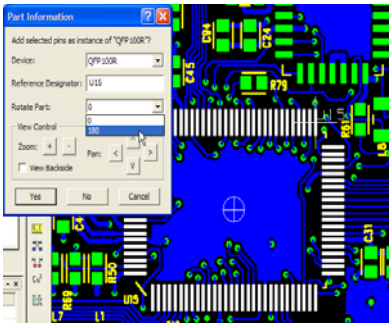
7. **Specify a Tolerance** to account for small variances between part instances and for misalignment in a row of pins during the pin identification process.
8. Select whether to find parts on the **Top Layer**, **Bottom Layer**, or **Both**.
9. Select the **Select Footprints** option if you only want to look for parts with specific footprints in the library.
10. Click on the **OK button**.

The Part Information dialog box appears, and individual part identification process begins. The process starts with the largest footprint in the library.

The pins for a detected part instance are highlighted, with each pin number displayed.

NOTE: View Controls are provided in the **Part Information** dialog box. Since you cannot use the Hotkeys or View commands to zoom and pan around in the workspace use the View Controls to get a better view of your parts.

11. Confirm that the **Device**, **Reference Designator**, and **Rotation** are correct in the **Part Information** dialog box.
    a. If it is, click the **Yes** button. If not, either edit the information in the dialog box, or
    b. Click the **No** button if the pins are not correct and you want the system to look again and pick a different set of pins.
    c. Click the **Skip** button if you simply want to skip the currently selected pin configuration for now (The most common reason being if you didn't run the **Tools > Convert > Drawn Pads** function in the beginning, and you notice that you have some drawn pads.). Unlike pressing the **No** button, when you press **Skip**, VisualCAM will not continue to look for a "correct" pin configuration at that location.
    d. If you click the **Cancel** button, it will skip finding any more instances of the current part footprint on the board, and go to the next footprint in the library.
12. When no **more** part instances are found, a message appears, and you are finished.
13. After processing several parts from the footprint library and you feel you are comfortable with the process, go ahead and click **Cancel** to abort the process. We will not process the entire board in this activity.
14. Close the job and **DO NOT SAVE** it! You will need it in the original state for the next activity.



Pin 1 is on the wrong corner for U15 per the silkscreen. Use Rotate Part to correct it.

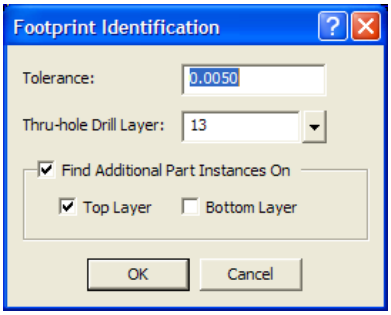
Activity: Manual Reverse Engineering

If you do not have an IPC-D-356 netlist, centroid file, silkscreen layers or footprint library for your database, you can manually identify parts using this function. For the sake of speed and accuracy, if you do have any one of the aforementioned data files, it is recommended that you use it to reverse engineer your part data for assembly.

This function can also be used after you have used any of the other Part Identification commands, and had to correct any data errors that required deleting a part instance and recreating it or if you had incomplete or inconsistent data sources.

NOTE: All pads must be flashed data, and must be made with intrinsic apertures (they cannot be custom apertures). If your pads are created using draws, follow the procedures described in the **Draws to Flash Conversion** section of this manual to correct it (*see Draws to Flash Conversion, page 77.*)

This activity should take approximately 15 minutes to complete.



1. Open the database labeled **FLIB-Demo.vcam** found on your desktop in the …**TrainingFiles\FLIB-Demo** directory.
2. Select the **Assembly > Parts > Identify Using > Select Footprint** command or click the icon in the toolbars.

The **Footprint Identification** dialog box appears.

3. **Specify a Tolerance** to account for small variances between part instances and for misalignment in a row of pins during the pin identification process.
4. Specify the **Thru-hole Drill Layer**.
5. Select **Find Additional Part Instances On** if you want the system to find any other instances of the part footprint you are about to define.
    a. If you do not select this option, the footprint will be added to the library, but you will not be prompted to identify any other part instances that match the footprint.
    b. If you do select this option, select whether you want the system to look for other part instances on the Top Layer or Bottom Layer.
6. Click on the **OK** button.

You are prompted to select the first pin of the part you want to identify.

7. **Click on Pin 1** for that part.
8. **Select the pin that is farthest from Pin 1** (typically the opposite diagonal corner of the part).

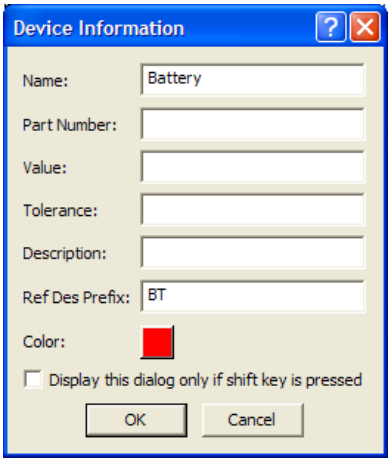
The system finds other pins automatically. If any are missed or selected by mistake, click on the pins to select/deselect them.

Use the **Selection Filter**, if necessary to pinpoint your selection options.

9. Press the **End** key when you are finished.

The **Footprint Name** dialog box appears. The system provides a default name for the footprint you just created, using the type of device, number of pins, and the iteration of the type of footprint that it is. SMD means that it is a Surface Mount Device; THD means it is a Thru-Hole Device. For example, THD6_2 is a Thru-Hole Device with 6 pins, and it is the second footprint of that type found.

10. **Keep the default** footprint name, **or assign your own**.
11. Click on the **OK** button.



The Device Information dialog box appears.

12. **Keep the default information** that the system created for the device, **or specify** the name and the Reference Designator Prefix you want associated with the device that you just identified.

    The other information (Part Number, Value, Tolerance and Description) is optional.

All parts have a color-overlay assigned to them, with the pins appropriately numbered, and the centroids remained marked with a target of the same color. A bounding box is placed through the centerline of the pins.

13. **Select a color** by clicking on the color in the dialog box, **or accept the system default**.

If VisualCAM finds other pads that appear to use the same footprint as the part you just created, it highlights the pins and you are asked if they should be identified as another part.

14. **If the suggested information is correct**, click **Yes** to identify that part.
15. **If the part information is not correct**, you can specify a new device name to associate with it, the correct reference designator, or rotate the part. Then click **Yes**.
16. Otherwise, click **No** to identify another part.

NOTE: If you click **Cancel** in the **Device Information** dialog box, then you are prompted to select the pins for a part again. If you want to abort the process entirely, press the **Esc** key. The process aborts, but any part information remains. You can select the **Assembly > Parts > Identify Using > Select Footprint** command later, without losing any existing parts.

Review Questions

List the five methods for reverse engineering.

_____
_____
_____
_____
_____

Which method is the most automated and reliable?

_____

What is the pre-requisite concerning the pads for all five methods?

_____
_____
_____
_____

What process do you have to run first prior to reverse engineering using silkscreen?

_____
_____
_____
_____

True or False; If you cancel out of any of the reverse engineering processes before you have completed the entire design you will automatically lose all your parts if start the process up again?

_____

True or False; If you are not able to complete the reverse engineering process for the entire design using one method, using a centroid file for example, you can attempt to finish the design using one of the other methods available to you such as manual identification of parts?

_____

Final Steps

In the following activities you will be putting the finishing touches on your job and then exporting the results to the FATF format.

Lesson Objectives

In this lesson you will do the following:

- Modify your results.
- Select Fiducials.
- Import a BOM File.
- Export a FATF File.
- Export an IPC-D356 File
- Export an IPC-2581 File
- Export a GenCAD File
- Export an ODB++ File.

Estimated Completion Time

Approximately 55 minutes.

Activity: Modify Your Results

In this activity we will describe the process and methods used to modify your results if necessary once the reverse engineering process is complete.

The purpose behind this lesson is to instruct the user on how to manipulate various aspects of the component data that was created during the reverse engineering process such as rotation and pin numbering.

This activity should take approximately 10 minutes to complete.

Rotate a Part Instance

NOTE: Rotation is only available if a part can be rotated, and only viable rotations are offered.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. In the Navigator, **expand the Parts list and right-click on the reference designator** of the part instance you wish to rotate. The part highlights in the main window and a shortcut menu appears.
3. Select the **Rotate** command, and the amount of rotation you wish to apply.

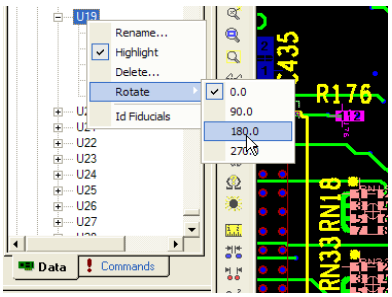
   Rotation is only available if a part can be rotated, and only viable rotations are offered.

   For example:

   a. A square part (having pins on all sides in equal numbers) will offer all 4 angles possible (0, 90, 180, and 270).

   b. A 2 pin device or even a multi-pin device with an equal number of pins on only two sides, such as an SOIC, will only offer 2 values (0 and 180).

   c. An asymmetrical part with an unequal number of pins on any side, such as a SOT23 or a D-Pak, will only offer a value of 0.



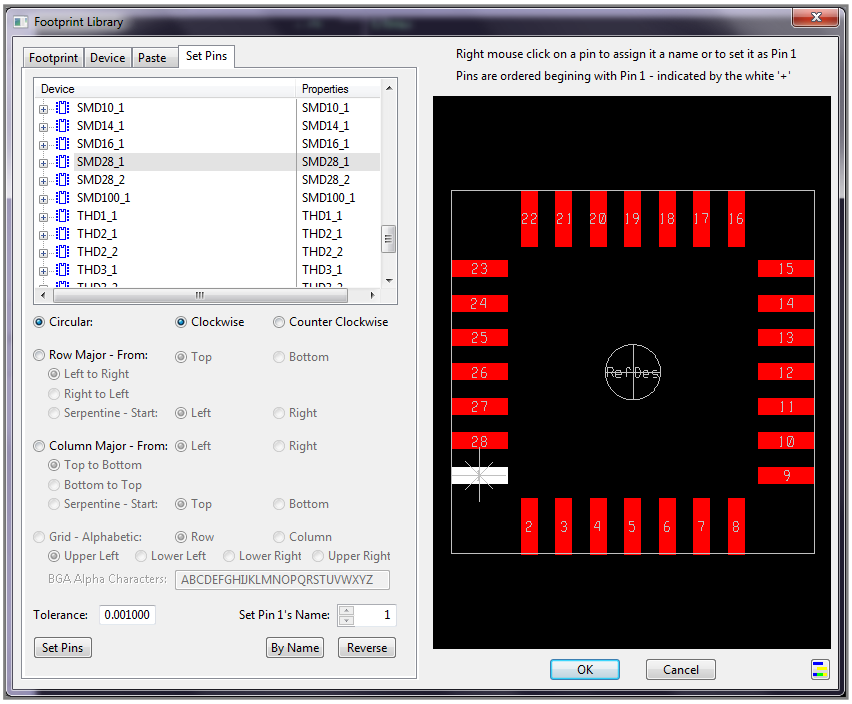
Modify the Pins of a Footprint

The Footprint Library contains information on all part footprints and devices defined for the current database. From the footprint library you can modify these attributes as well as re-arrange the pin numbering if necessary.

TIP: If you want to alter a single part instance (like changing its reference designator, device type, rotation, or to delete it), use the **Navigator**.

1. Select the **Assembly > Setup Footprint Library** command or click the icon in the toolbars.

In the Footprint Library dialog box:

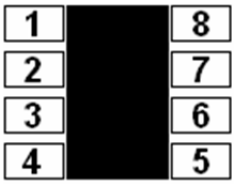


2. Click on the **Set Pins** tab.
3. **Scroll down** the list of Devices and select **SMD28_1**.
4. **Right click on a pin in the diagram.**
5. In the shortcut menu that appears choose **Set As Pin 1**.
6. Under the Device list, **choose the Pin Mapping properties** (*see Pin-Number Mapping Diagrams, page 171 for more information*.)
7. Click the **Set Pins** button.
8. **Try several of the available options** and see how they affect the pin numbering on your selected component.
9. Click on the **OK** button to save the changes and to close the dialog box**.**

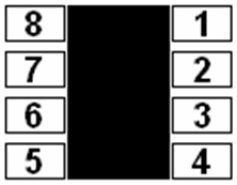
Pin Number Mapping Diagrams

The images below demonstrate how all of the available options will affect the pin numbering on a component.

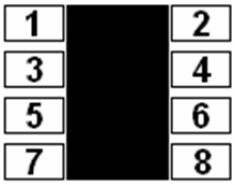
Circular Counter-Clockwise



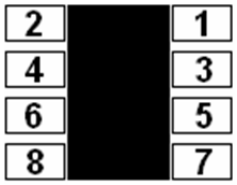
Circular Clockwise



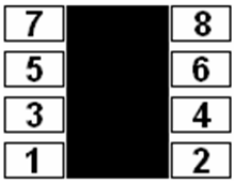
Row Major, Top, Left to Right



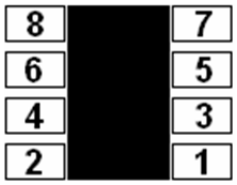
Row Major, Top, Right to Left



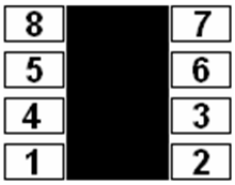
Row Major, Bottom, Left to Right



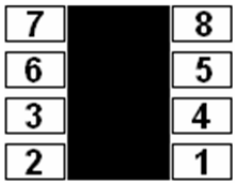
Row Major, Bottom, Right to Left



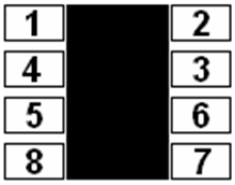
Row Major, Serpentine, Bottom, Left



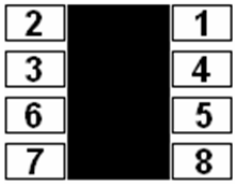
Row Major, Serpentine, Bottom, Right



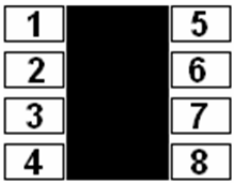
Row Major, Serpentine, Top, Left



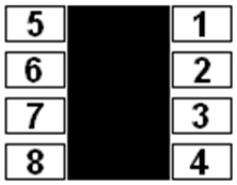
Row Major, Serpentine, Top Right



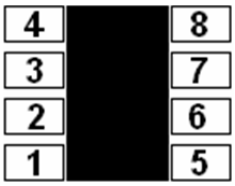
Column Major, Left, Top to Bottom



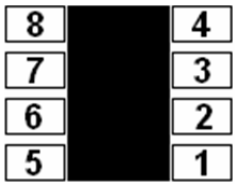
Column Major, Right, Top to Bottom



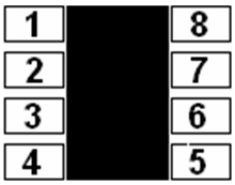
Column Major, Left, Bottom to Top



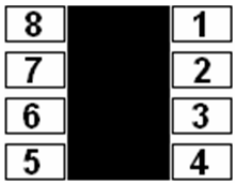
Column Major, Right, Bottom to Top



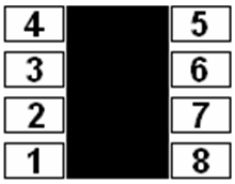
Column Major, Left, Serpentine, Top



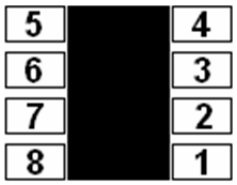
Column Major, Right, Serpentine, Top



Column Major, Left, Serpentine, Bottom



Column Major, Right, Serpentine, Bottom



Activity: Select Fiducials

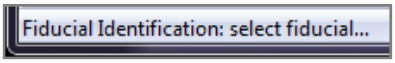
This function can be used to identify either board or part fiducials. If you have not identified any parts in your data using one of the Assembly| > Parts > Identify Using commands, then you can only identify board fiducials.

TIP: This function is also accessible from the right-click shortcut menu, in the **Assembly** area of the Navigator.

This activity should take about 5 min to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Assembly > Fiducials > Identify** command or click the  icon in the toolbars.

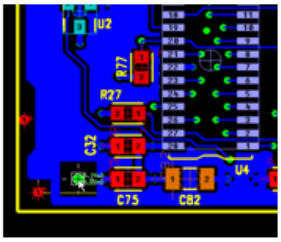
In the Status Bar you are prompted to select a fiducial on your circuit.



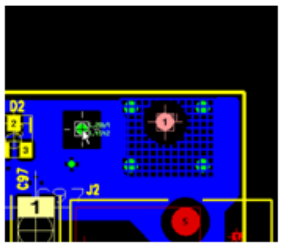
3. **Click on the desired object for the fiducial**.

   In the IPC-Demo job, select the fiducials found in the lower-left and upper right corners of the PCB as pictured in the images below:

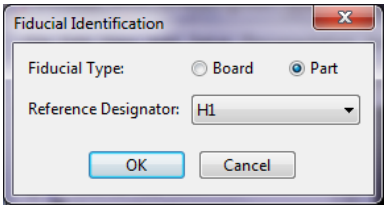
Lower Left Corner



Upper Right Corner

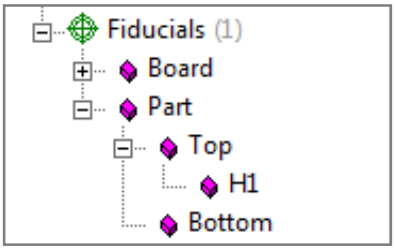


In the Fiducial Identification dialog box:



4. Select whether the fiducial is for the **Board** or a **Part**.
5. **Specify a name** for the fiducial if it is a board fiducial or select the reference designator of the part it belongs to, if it is a part fiducial.
6. Click on the **OK** button.

The fiducial is added to the Assembly section of the Navigator. If you want to view the location of a fiducial, click on its name in the **Navigator** and it is highlighted with a target.

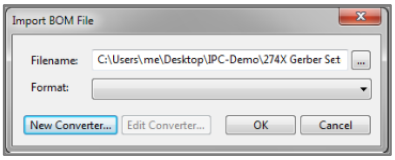


Activity: Import a BOM File

After you have reverse engineered your data for assembly, this function allows you to add additional part information from a bill of materials to the database prior to exporting.

TIP: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator.

This activity should take about 10 min to complete.



1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Assembly > Import BOM** command from the menus.

In the Import BOM File dialog box:

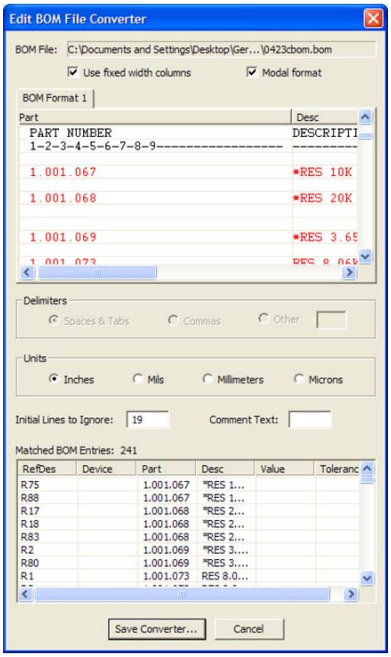
3. In the **Filename** box, specify the bill of materials file to use. Click the **Browse** button if you need to search for the desired .bom file.
4. Select the correct **BOM File Converter** for the BOM file from the **Format** list.

   These converter files are located in the **BOMConv** folder in your main VisualCAM folder C:\ProgramData\WISE Software Solutions\VisualCAM 16.4\bomconv.

   If a converter does not exist, see Create a BOM File Converter activity below, page 175 for instructions.
5. Click on the **OK** button, and the information from the bill of materials is added to the current footprint library.

NOTE: If the bill of materials contains device names or part numbers that do not match those in the footprint library, they will be added as necessary.

Create a BOM File Converter



1. In the **Use BOM File** dialog box, click the **New Converter** button.

The Create BOM File Converter dialog box appears. The name of the BOM file being imported appears at the top, and your current BOM file is displayed in the **Format 1** tab.

Your BOM file appears in a spreadsheet, and default column names are provided at the top. These columns are not automatically matched to the BOM file; you must reorder and resize the columns to match the BOM file contents.

2. **To move a column header**, click it with your mouse and drag it to the correct location.
3. **To resize a column header** to exactly match the width of the BOM file columns, click on the left or right edge of the column header and drag it to the appropriate location.
4. **To delete a column header**, right-click on it and select Delete Column from the shortcut menu. Note that once you delete a column, it can always be added again later (it appears as an Add command in the right-click menu).
5. **To ignore a column of information**, right-click on the column header currently above it, and select Add Skip Column from the shortcut menu. A header titled "Skip" now appears above the contents.

As you identify the columns in your BOM file, VisualCAM automatically lists recognized information in the Matched BOM entries list. All recognized entries are also colored red in the BOM Format view.

6. If you have any lines in the BOM file, such as headers or comments, specify in the **Initial Lines To Ignore** box how many lines at the beginning of the file should be ignored by the converter.
7. Also specify, in the **Comment Text** box, the symbol that leads any comment lines that may be intermixed with your BOM information.
8. **Modal Format** should be selected if your bill of materials uses one line of part information, and then followed by a list of reference designators that are on their own lines (indicating that the same part information applies to each reference designator, without repeating the information on each line).

9. **Fixed Width vs. Delimited**:
    a. If your BOM file uses a single space as the column delimiter, select the **Use Fixed Width Columns** option.
    b. If your BOM file uses column delimiters *other than* single spaces, specify the appropriate Delimiter. Note that the Spaces & Tabs option assumes multiple spaces/tabs are used, as opposed to the Use Fixed Width columns option, that assumes just one space is used.
10. When you are finished, click the **Save Converter** button to save your ACR file in the BOMConv folder.

The dialog box closes and your converter appears in the Format list of the Import BOM File dialog box.

11. Click on the **OK** button in the Import BOM File dialog to complete the process.

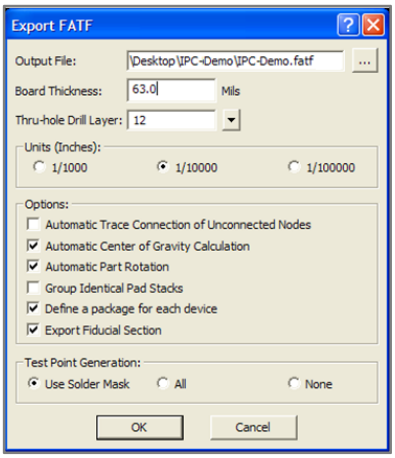
A message box appears indicating how many parts were updated by the BOM import.

Activity: Export a FATF File

After you have finished reverse engineering your data, this function exports a Fabmaster ASCII Transfer Format (**FATF**) file for use by assembly software. For those who are familiar with the FATF format, we have included a description of the FATF option (such as :NOTRACE or :NOAUTOCOG) that corresponds with each dialog box option.

TIP: This function is also accessible from the right-click shortcut menu, in the Assembly area of the Navigator

This activity should take about 5 min to complete.



1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **Assembly > Export FATF** command or click the FATF icon in the toolbars.

In the **Export FATF** dialog box:

3. Specify the **Output File** name and location. Click the **Browse** button, if you want to search for a file/folder.
4. In the **Board Thickness** box, specify the thickness of your board.
5. Specify your **Thru-Hole Drill Layer**.
6. For the **Units (Inches),** select the precision of the output measurements.
7. Select **Automatic Trace Connection Of Unconnected Nodes** to make all traces connect to the exact center of the pads, extending any traces that do not terminate at the center of the pad.

   When this option is not selected, the **:NOTRACE** option is generated.
8. Select **Automatic Center Of Gravity Calculation** to generate the centroids of the silkscreen bodies (openings). This is in addition to the part centroids.

   When this option is not selected, the **:NOAUTOCOG** option is generated.
9. Select **Automatic Part Rotation** to have your assembly software redefine the parts in the design so that the vector components showing the direction of first pin towards the second pin are:
   a. Positive for the X axis (0 < Xpin2 - Xpin1)

b. Positive or zero for the Y axis (0 <= Ypin2 - Ypin1)

When this option is not selected, the **:NOAUTOROTATE** option is generated.

10. Select **Group Identical Pad Stacks** to find padstacks with the same qualities and give them the same definition in the file.

    When this option is not selected, the **:NO_PAD_STACK_GROUP** option is generated.

11. In the FATF database, normally a package is created for each footprint, using the footprint name. The **Define A Package For Each Device** option creates a package for each device, using the device name. This results in a greater number of unique packages.

12. If you are generating the files for **assembly**, then you will want to include the fiducial section, and select the **Export Fiducial Section** option.

13. If you are generating FATF files for **assembly in-circuit test**, then **do not** select the **Export Fiducial Section** option.

14. If you are generating the files for In-Circuit Test, select **All** to output all viable test locations on vias and test point pads. Select **Solder Mask** to only output vias and test point pads that are exposed (are not covered by a mask). Select **None** if you do not want to generate assembly test data.

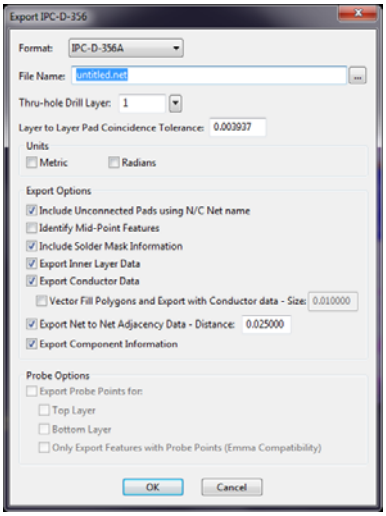
15. Click on the **OK** button.

VisualCAM checks for any data errors, such as whether pads exist on plated thru-hole drills, and informs you of any errors. Your **FATF** file is then created.

Activity: Export an IPC-D-356 File

After you have finished reverse engineering your data, this function exports an IPC-D356 file containing the netlist connectivity as well as reference designator and pin number information contained in your .vcam database. IPC-D356 is a widely accepted format developed for bare board test.

This activity should take about 10 min to complete.



1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **File > Export > IPC-D356** command.

In the **Export IPC-D356** dialog box:

3. In the Format list, **select which IPC-D-356 Format you wish to export**. This automatically enables only the options that are supported by the selected file format.
4. Browse to the destination folder of your choosing.
5. Specify the **Output File** name.
6. **Enter the number of the NC layer** that contains your thru-hole drill information (this is the same NC layer that was used when the netlist was generated).
7. **Layer To Layer Pad Coincidence Tolerance** specifies how far out of alignment pads on different layers can be, and still be considered a padstack in the exported file.
8. **By default, linear dimensions are mils**, and angles are expressed in degrees. If you want linear dimensions to be millimeters, select the Metric option. If you want angles to be expressed in radians, select the Radians option.
9. Select **Include Unconnected Pads Using N/C Net Name** to have all unconnected (isolated) pads output using the IPC-defined net name "N/C". Otherwise, all unconnected pads will be ignored.
10. Select **Identify Mid Point Features** to have all features that are considered mid points (i.e. not end points) exported. Otherwise, no mid-point features will be exported.
11. Select **Include Solder Mask Information** to include data indicating solder mask usage. If no solder mask is present, then this option is not available. (If you have a solder mask but this option is not available, your solder mask layer is not properly tagged in the layer table.)
12. Normally, only top and bottom net locations are exported. Select the **Export Inner Layer Data** option to include inner layer net locations, which is most often used if your design

has blind and buried vias.

13. The **Export Conductor Data** option exports the trace data for the top and bottom layers. If you selected the Export Inner Layer Data option, then the inner layer trace data are also included.
14. **If you are exporting an IPC-D-356A netlist:** IPC-D-356A does not support raster fill polygons. If you want polygon information maintained with your trace data, select the Vector Fill Polygons And Export With Conductor Data option to make the polygons vector fill. Also specify the size of the lines to use for the solid vector fill.

    -or-

    If you are exporting an IPC-D-356B netlist: IPC-D-356B supports raster polygons. If you want polygon information maintained with your trace data, select the Export Raster Polygons option.
15. **Export Net To Net Adjacency Data** to report any nets within your specified distance to each other. (In the netlist, each net is reported along with any other net that is adjacent to it, according to the distance you specify. This option can take considerable processing time and is not recommended unless required.)
16. Select the **Export Component Information** option to include reference designators and pin numbers in your file.
17. Click the **OK** button to export your file.

Activity: Export an IPC-2581 File

After you have finished reverse engineering your data, this function exports an IPC-2581 file containing all the graphical and part information contained in your .vcam database. IPC-2581 is the newest standard, designed for transferring complete board information.

This activity should take about 5 min to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **File > Export > IPC-2581** command.

In the **Export IPC-2581 File** dialog box:

3. Browse to the destination folder of your choosing.
4. Specify the **Output File** name.
5. Click the **Save** button.

Activity: Export a GenCAD File

After you have finished reverse engineering your data, this function exports a GenCAD file which will contain the part and connectivity information contained in your .vcam database. GenCAD is a popular format which is commonly used for assembly.

This activity should take about 5 min to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **File > Export > GenCAD** command from the menus.

In the **Export GenCAD** dialog box:

3. Browse to the destination folder of your choosing.
4. Specify the **Output File** name.
5. Click the **Save** button.

Activity: Export an ODB++ File

After you have finished reverse engineering your data, this function exports an ODB++ format job folder containing all the graphical and part information contained in your .vcam database. ODB++ is a commonly used format for transfer of board design data.

This activity should take about 5 min to complete.

1. If it is not already, **open the IPC-Demo.vcam file** on which you have been working.
2. Select the **File > Export > ODB++** command.

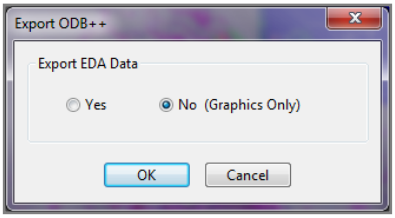
In the **Browse for Files or Folders** dialog box:

3. Browse to the destination folder of your choosing.

The ODB++ job folder will automatically be given the job name of you .vcam file (in this case IPC-Demo).

4. Click on the **OK** button.

In the **Export ODB++** dialog box:



5. Choose **Yes** to Export EDA data (choosing no will exclude part and net information).
6. We suggest that you create a zip archive from the exported ODB++ job folder, for better portability.

Contact Information

WISE Software Solutions, Inc.

2700 E. 9th Street, Suite 100
Newberg, OR 97132
USA
Tel: 1 (503) 554-8855
Fax: 1 (503) 554-1220
Support: 1 (503) 554-8855
E-mail: support@wssi.com
Web: http://www.wssi.com