GerbTool
Training Guide

Contents

Process Flow Charts

In this section you will find all the flow charts that provide a high level overview of the process for GerbTool.

General Process Flow

Import Wizard

↓

Tag Layers — Review layers and data to determine what you have or may need to create

↓

Do you have a **Border Layer**? —No→ Create Border

↓

Is your Drill layer **NC** format? —No→ Convert Gerber drill layer to **NC** layer.

↓ Yes

Setup/Sort Layers/Colors

↓

Are your layers Aligned? —No→ Run **Align Layers**

↓

Edit Origin (Optional)

↓

Delete Garbage Data

→

Automatic Draw-2-Flash Conversion — 1. Review any errors in the Error Analysis Manager 2. Use Select Draws to convert pads that were missed due to errors.

↑

Snap Pads

↑

Generate **Net List**

↑

Import External Netlist — Typically IPC D356

↑

Analysis: Netlist Compare Have errors? —Yes→ Use Pin-point error to find and correct

↑ No

Generate/Optimize Paste/Solder Masks

↑

Analysis: DRC/MRC

↑

Panelization (Optional)

↑

Export files

GerbTool Basics

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

Purpose

The purpose of this lesson is to familiarize the user with application window (UI) of GerbTool including the location and use of the various windows and toolbars.

- ▸ Definitions
  - o **GT**: GerbTool
  - o **UI**: User Interface
  - o **GTD**: GerbTool Database file or job
  - o **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 GerbTool using 3 methods.
- ▸ Learn to navigate the workspace.
- ▸ Identify the location and use of your 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.
- ▸ Learn to navigate inside the workspace area.

Estimated completion time

This lesson usually takes 30 minutes to complete.

Navigating GerbTool

The first thing you want to know about any software tool is how to navigate your way around. The graphic in Figure 1.1 below gives you a topographical view of the application window. We will also cover the how to navigate your way around the workspace area and the various windows.

Note: Please copy the "**TrainingFiles**" directory from the course data you were given with this manual and paste it onto your computer's desktop. Then open this directory and double-click the file "**IPC-Demo.gtd**"

GerbTool UI Layout

The main GerbTool window, or "desktop", is illustrated below. All toolbars and control bars may be moved to a location you prefer by clicking on the bar and dragging it to a new location. They may be docked to an edge of the GerbTool window, or float in a small 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.

Figure 1.1
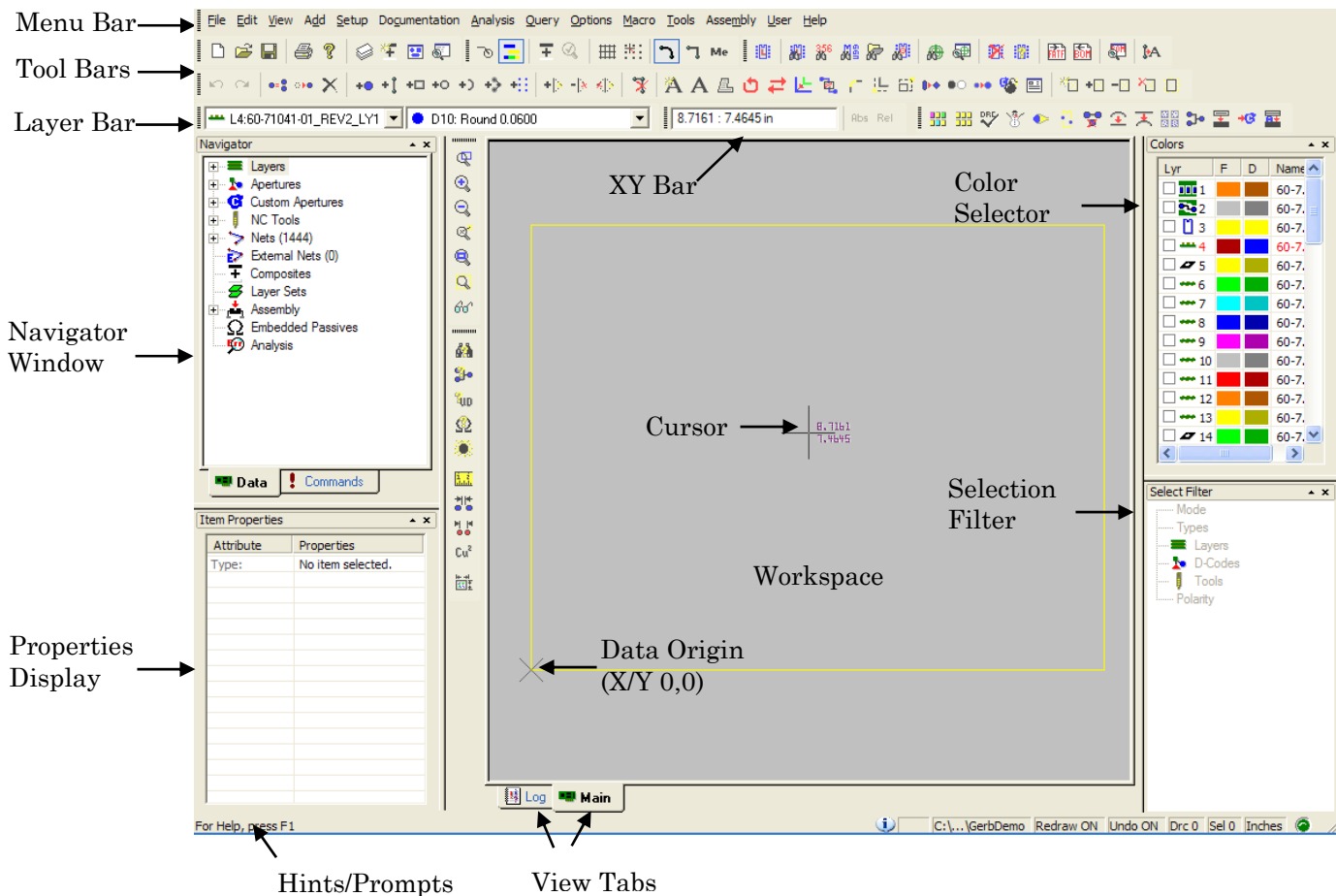
Navigating the Workspace

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

LMB 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. For most commands, your LMB is used to select the object(s) to be effected by the command.

RMB Behavior.

There are two default behaviors for the RMB depending on where your mouse pointer is when you click it.
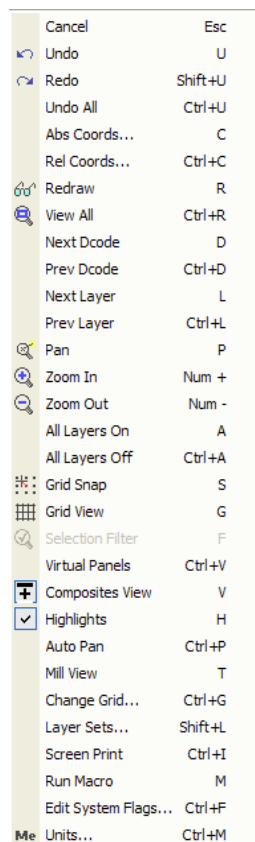
1. If your mouse pointer is hovering over a blank area in the workspace when you right-click, a pop-up menu (see left image) will appear listing several frequently used commands such as zooming options and layer surfing.

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 right image) with frequently used edit commands appears.

   a. 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 selection dialog asking you to select
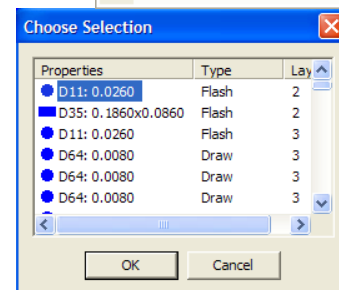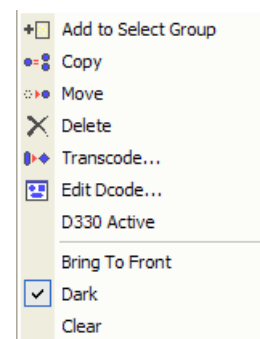
the specific object to be edited or queried.

Hot Keys & Function Keys

There are many Hot Keys available to GerbTool users that can make the overall processing of gerber 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.

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

Key	Assignment
F1	Help
F2	View|Film Box
F3	View|Previous
F4	View|All
F5	Setup|Layers
F6	Setup|Apertures
F7	Documentation|Reports|Apertures
F8	Query|Highlight
F9	Query|Item
F10	Menu
F11	Edit|Select|Add To
F12	Edit|Select|Remove From

Note: Both your function keys and your 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. You can control which toolbars appear in the window with the **View|Toolbars** command. You can change which button appears in each toolbar by using the **Options|Customize Toolbar** command.

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.
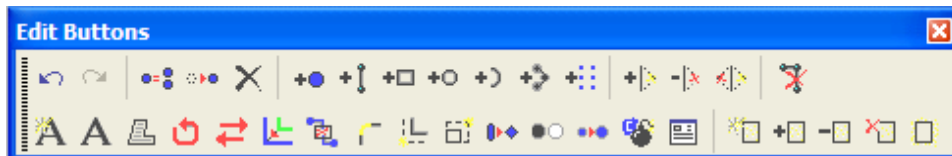
Mainframe Buttons are used for working with files and setting up layer and aperture information.
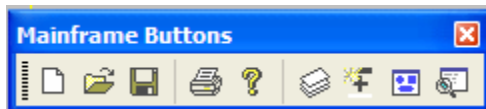
Parts Buttons execute the utilities used in the reverse engineering process.

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.
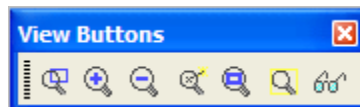
The Navigator

The Navigator provides a quick-reference list of your database elements, and another way of accessing GerbTool 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.

The Data Tab

The Data tab provides information about your layers, apertures, NC tools, nets, composites, layer sets and Analysis runs. 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

Figure 1.2

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. We will be discussing the use of several of these functions in more detail in later chapters.

Selection Filter

Figure 1.3
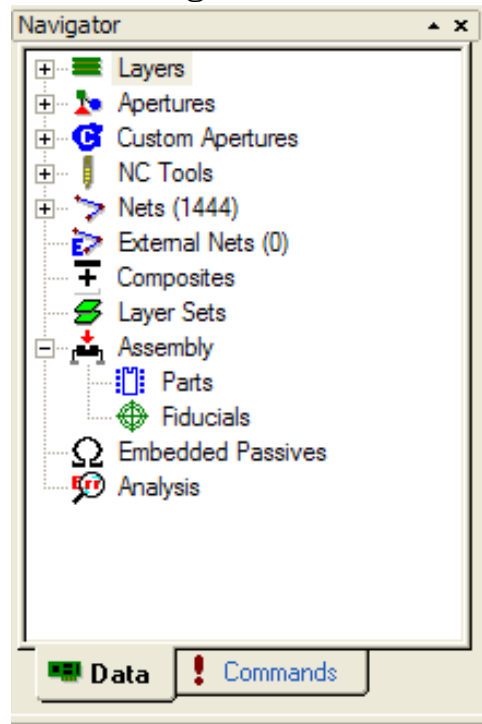
The Selection Filter allows you to define how you want to select items, and what types of items you wish to choose from. For example, depending on the command, you may choose from single item, window, group, or complete layer selections, as well as restricting your selections to particular layers and/or 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 the point (An X:Y location in the workspace.) you select, the Choose Selection dialog box appears.

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 which kinds of items will be included. Any type of item not checked will be excluded from the selection.
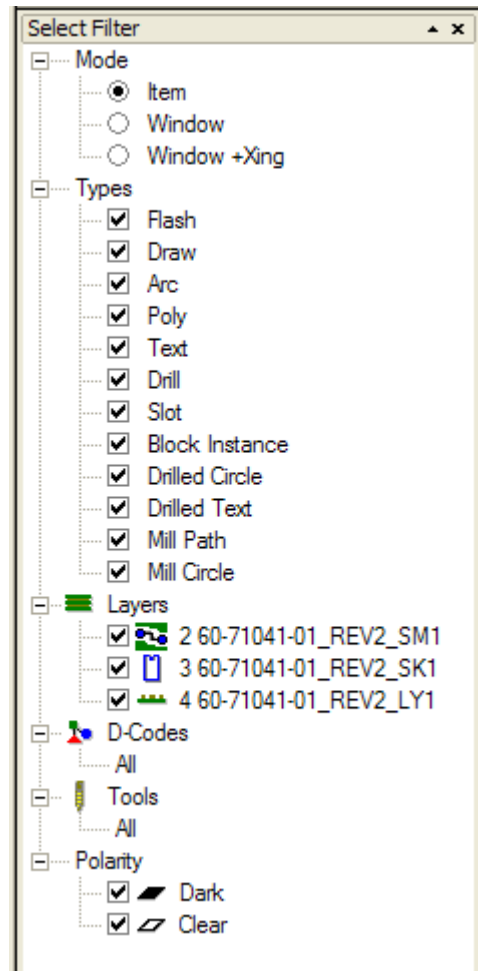
Layers

These options allow you to select which layers will be considered when choosing items.

D-Codes

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

Tools

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

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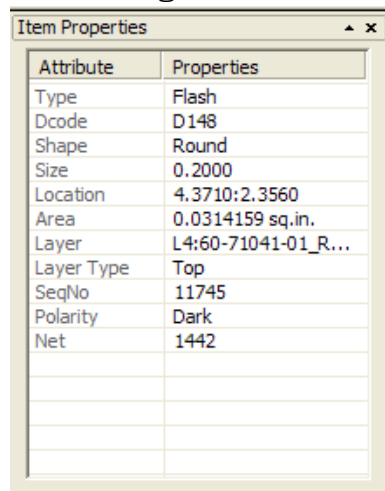
These options allow you to specify only items with a particular polarity.

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.

Figure 1.4
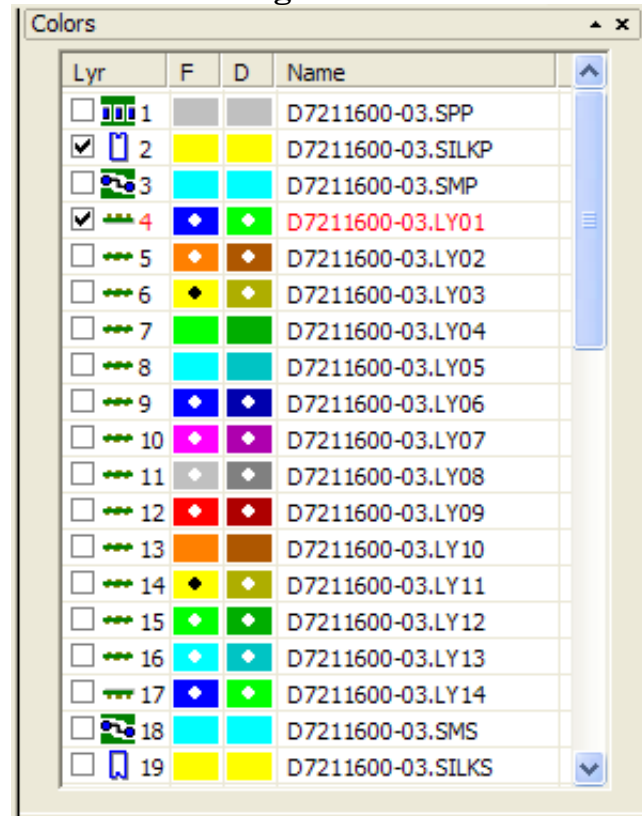
Item Properties	
Attribute	**Properties**
Type	Flash
Dcode	D148
Shape	Round
Size	0.2000
Location	4.3710:2.3560
Area	0.0314159 sq.in.
Layer	L4:60-71041-01_R...
Layer Type	Top
SeqNo	11745
Polarity	Dark
Net	1442

Color Bar

The Color 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.

Figure 1.5

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 (can only be set by GerbTool after importing a 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 illustration.

If you wish to change a layer name or type, use the **Layer table** or **Navigator**.

Visibility

Visibility for individual layers, layer sets and composites is controlled 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 in the Color 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 Color 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**.

Color Selector

You can control the color of flashes and draws for each of your 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.
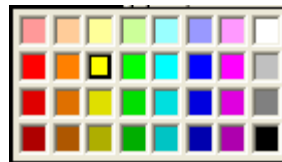
To dismiss the Color Selector without choosing a color, press the Esc key.

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. Use the following steps to utilize this macro.

Open the **demo.mac** file the **Macro Developer**, and edit it to specify your desired colors. Save the file and return to the main GerbTool window.

Select the **Macro|Load** command and load the **demo.mac** file.

Select the **Macro|Run** command, and run the **SetLayerColors** macro.

Tip: If you wish to run the **SetLayerColors** macro automatically when GerbTool starts, rename it to **OnStartup**. To run it whenever you select the **File|Open** command, rename it to **OnOpen**.

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. Just remember to enable Undo with the **Options|Configure/General** command *before* making your edits. Then use the Edit|Undo command, the toolbar button, or the U hotkey to undo changes as desired.

Programming Your Mouse and Function Keys

GerbTool's easy-to-use Graphical User Interface (GUI) is further enhanced with the versatility of programmable mouse and function keys. Using the **Options|Configure** dialog box, you may program both the mouse buttons and function keys F1-F12 with commands that you frequently use and in a layout that you find comfortable.

Interrupting Redraws and Highlights

Any command that redraws the database or highlights a group of items can be sped up by canceling the drawing process. By pressing the **Esc** key, you can halt whatever is redrawing the display. This does not effect the operation of the command; only the redraw is effected. 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.

Review questions

1. What is the primary purpose of the Navigator?

   _____
   _____
   _____

2. What is the primary purpose of the Selection Filter?

   _____
   _____
   _____

3. What is the primary purpose of the Color Window?

   _____
   _____
   _____

4. What is the primary purpose of the Properties Window?

   _____
   _____
   _____

5. What are the two functions of the Active Layer?

   _____
   _____
   _____

Activity: Setup Toolbars

In this activity you will open GerbTool for the first time using three different methods and you will modify your toolbar setup to a more suitable configuration for reverse engineering Gerber to CAD.

This activity should take about 20 min to complete.

1. Open GerbTool.

    ‣ On your desktop, double-click the GerbTool icon.

    ‣ Click Start, select **ALL Programs | WISE Software | GerbTool 15.2 | GerbTool**

    ‣ In Windows Explorer, browse the directory, "**C:\Documents and Settings\All Users\Application Data\WISE Software Solutions\GerbTool 15.2\Samples**". Now double-click on the file "**demo.gtd**".

2. Open the IPC-Demo database.

    ‣ On the **File** menu, select **Open**

    - or- Click the Open (📂) Icon on the toolbars

    - or- Press **Ctrl + O**

    ‣ In the **Open** dialog box browse to the "**TrainingFiles**" directory on your desktop.

    ‣ Select the file "**IPC-Demo.gtd**" and click **Open**.

3. Enable and Disable Toolbars.

    ‣ On the **View** menu, select the **Toolbars...** option**.**

    ‣ Uncheck the **Redline Bar**

    ‣ Check the **Edit Bar** and **Tools Bar**

(The dialog should like the image to the right when you are done.)
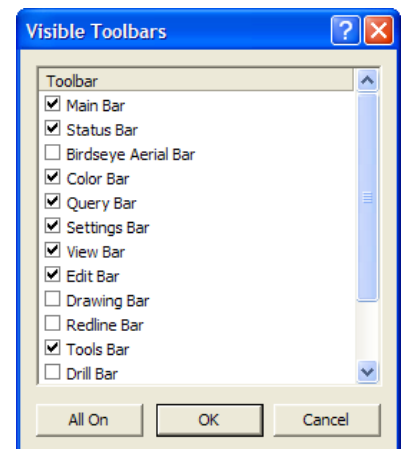
4. Position and dock Toolbars.

‣ Click on the **Edit Bar** and drag it so that shadow outline is on top of the **Menu Bar** then release it.

(The Edit Bar is now docked above the Menu Bar.)
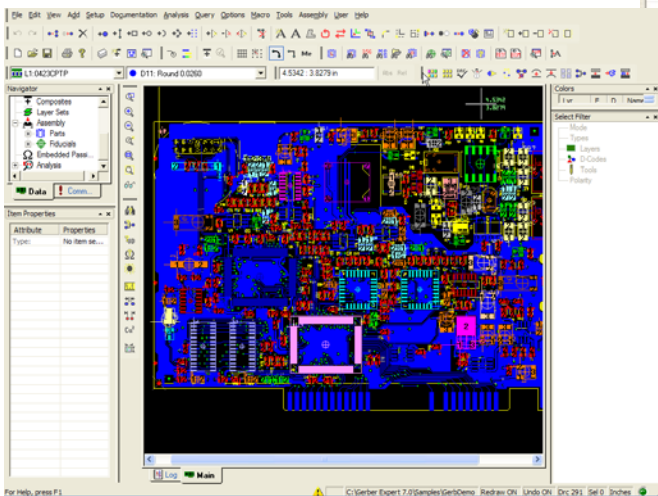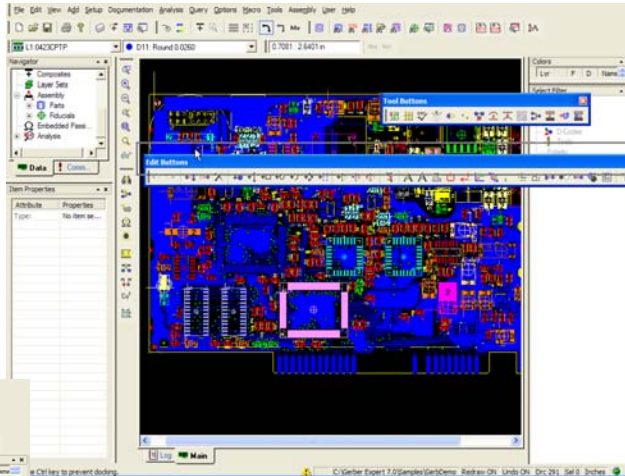
‣ Click on the dotted grab handle for the Menu Bar and drag it on top of the Edit Bar to position it back to the top of the Toolbar stack.

‣ In the same manner, position the **Tools Bar** to the right side of the **XY Bar.** (The application window should like the image to the left when you are done.)

5. Resize the Navigator, Properties, Colors and Selection Filter windows.

‣ Position your mouse pointer in between the **Navigator** and **Properties** windows until it changes to an up/down arrow cursor.

‣ Click and drag down so that the Navigator consumes about 2/3 of the vertical space and the Properties window takes up the remaining 1/3.

‣ Repeat this process for the Colors and Selection Filter
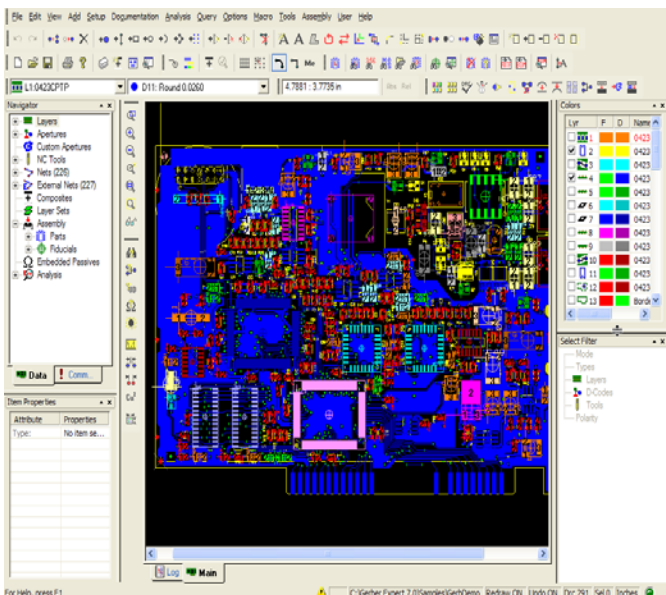
windows, but make the split closer to 1/2 and 1/2. (The application window should like the image to above when you are done.)

Activity: Use the Navigator

In this activity you will search for and modify various database objects in the IPC-Demo job you opened in the previous activity.

This activity should take about 10 min to complete.

1. **View Layers**.

   ▸ Click the plus box (+ icon) next to ▤ Layers item to expand the list of layers in the database.

   ▸ Expand the first layer **0423CPTP**.

   ▸ By right clicking on the first item listed under this layer you see that you have the option to rename the layer. (Do not rename it at this time.)
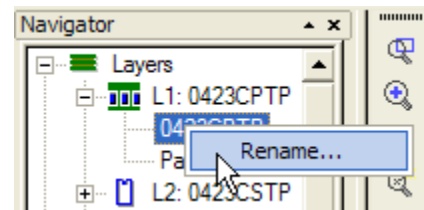
   ▸ By right clicking on the second item listed under this layer you see that you have the option to change the Type for that layer.

   - o Select **Other** and see the icon change for the layer.

   - o Re-expand the layer and change the type back to **Paste Top.**
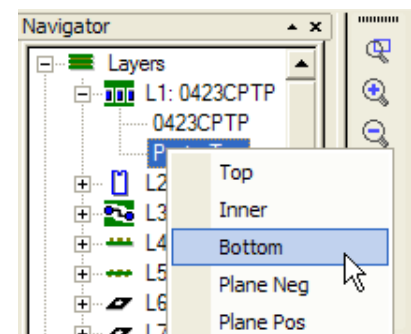
   ▸ Collapse the **Layers** list by clicking the minus box (- icon).

2. **View Nets**.

   ▸ Expand the ⤳ **Nets** list in the Navigator.

Note: Nets are only available after running the **Generate Netlist** command.

   ▸ Zoom in to a small area of the circuit image in the workspace.

   ▸ Double-Click on the first Net in the list, **$1N361**.

   ▸ Check the **Zoom to Net** option in the **Query Net** dialog that just opened.

   ▸ Click the **Find** button.

Net **$1N361** is now centered in your workspace and is highlighted. Any other nets you double-click on will now automatically snap to the center of your screen and highlight.

▸ Collapse the **Nets** list.
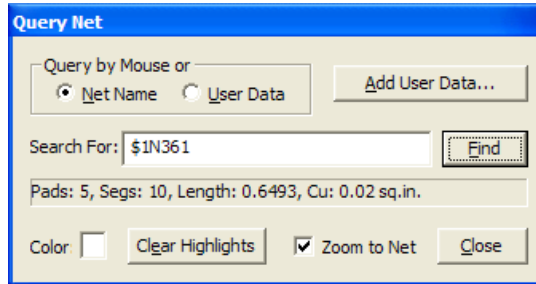
3. **View External Nets**.

▸ Expand the layers list and double click on layer 14 "**L14: External Nets**" to make it visible then collapse the layers list again.

Note: External Nets are only available after importing an external netlist like an IPC-D-356 file.

▸ Expand the **External Nets** list in the Navigator.

▸ Double-Click on the first Net in the list, **$1N361: 123.**
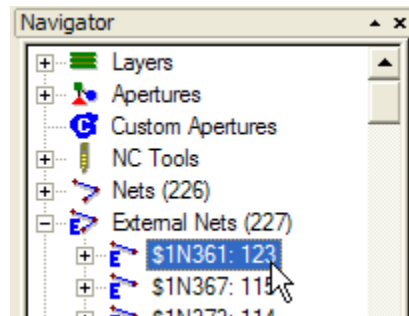
External Net **$1N361: 123** is now centered in your workspace and is highlighted.

▸ Collapse the **External Nets** list.

Activity: Use the Color Window

In this activity you will use the Color Window 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.

1. All On/All Off.

    ‣ In the **Colors Window** right-click on any of the layers listed.

    ‣ Select **All On.** (All layers are checked and become visible in the workspace.)

    ‣ Now repeat this but select **All Off** this time. (All layers are unchecked and become invisible in the workspace except for the active layer.)
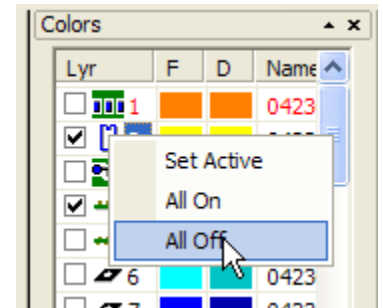
2. Set the Active Layer.

    ‣ In the **Colors Window** right-click on the Top Silk layer.

    ‣ Select **Set Active.** (The layer name and number change to a red font and the Top Paste layer changes to a black font.)

    ‣ You can also set the Active Layer by double-clicking a layer in the Color Window.

3. Modify Layer Colors.

    ‣ In the **Colors Window,** left-click on the color square in the F (Flash) column for the Top Silk layer.

    ‣ Select a color of your choice, cyan for example.
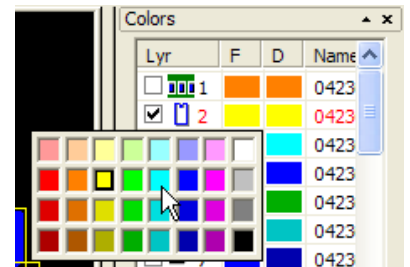
Note: You don't see anything change in your workspace. This is because all objects on the Top Silk layer are drawn rather than flashed.

    ‣ Now repeat this process for the D (Draw) column. Now all the objects on the Top Silk layer are changed in your workspace to the new color you selected.

Tip: It is a good practice to use contrasting colors for your Draw and Flash columns on your Top and Bottom copper layers. (Green ▮ and Blue ▮ for example.) This will help you to see graphically whether pads have been converted from drawn to flash or not before you try to reverse engineer. It is a requirement for all pads to be a flash for the reverse engineering process to work.

Importing Gerber Layers

In this lesson we will describe the steps and processes used to import your data files into GerbTool.

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.

- ▸ Learn to evaluate your data set to determine what reverse-engineering process you will use.

Estimated completion time

This lesson usually takes 15 minutes to complete.

The Import Wizard

The Import Wizard allows you to specify which files (Gerber, aperture lists, GerbTool databases, Drill files, etc.) you wish to import, assign aperture lists to specific Gerber files, and order the loading sequence. GerbTool 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.

To use the Import Wizard

1. Make sure all the files you wish to import are in the same folder.

2. Select the **File|Import|Import Wizard** command. Page 1 of the Import Wizard appears:

   My Data Is In

   Enter the name of the folder containing your files to import. Click the Browse button to search for the folder.

   Configure File Types To Ignore

   This 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.

Figure 2.1
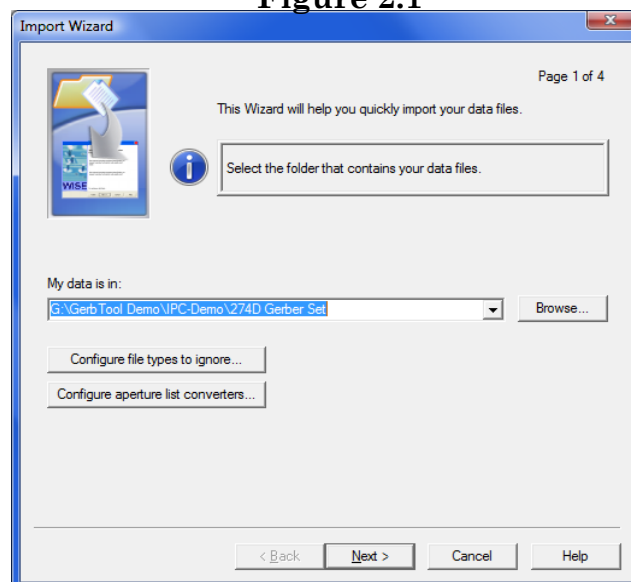
Configure Aperture List Converters

This 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.

3. Click the Next button to continue to Page 2 of the Import Wizard. This page presents the list of files that GerbTool found in the selected import folder. Click on the list column headers to sort the list based on the data within the respective columns.

Filename

Indicate whether GerbTool should import a file by putting a check mark by the filename. Click the Unselect All Importable button if you wish to remove the check marks next to all file names.

File Type

All importable file types are indicated (if a file cannot be imported, "???" appears.) If GerbTool 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

This column indicates the detected data format for each file. To override the detected format, click on the appropriate entry.

If the file is Gerber, the Gerber Import Data Format dialog box appears.

- If the file is an aperture list, the Select Converter dialog box appears. GerbTool has already selected an appropriate aperture converter to use for your aperture list, if one is available. To override the selection, select a different converter to use from the Data Format list. You can have your selection applied to all incoming aperture lists by clicking the Apply To All button. 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 appears. GerbTool has already selected an appropriate converter to use for your tool list, if one is available. To override the selection, select a different converter to use from the Data Format list. You can have your selection applied to all incoming tool lists by clicking the Apply To All button. You can create a new converter by clicking the New Converter button..

- If the file is DXF, the Import DXF dialog box appears.

- If the file is HPGL, the HPGL Import dialog box appears.

- If the file is DPF, there are no format parameters to modify.

- If the file is NC (drill or mill) data, the Import NC Data dialog box appears.

   Length

   The size of the file, in bytes.

   Date

   The date that the file was last modified.

   Show Unknown File Types

   By default, the Wizard only displays those files it can identify. Select this option if you wish to see all files contained in the selected import folder.

4. Click the Next button to continue.

If you are importing any 274-D files, Page 3 appears. This page is used to assign 274-D Gerber files to aperture lists. Drag each Gerber file onto the desired aperture list. (If only one aperture list is present, all Gerber files are automatically assigned to it.) Then click the Next button.

If you are not importing any 274-D files, Page 3 is skipped.

Page 4 presents the final list of files that are to be imported. The files are imported to layers in the order they are listed. If you wish to reorder the files so that your layers are in the proper order, click and drag each file to the desired position.

If you wish to prevent warning messages from being displayed when importing files, select Suppress Import Warning Messages. A log file will still be created but you will not be prompted whether to view it.

When you are satisfied with your selections and ordering, click the Finish button. Your files are imported and any pertinent information or warning messages are displayed in the Log screen. You can 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.

Activity: Import 274D files for IPC-Demo board.

In this activity you will select all of the necessary files and converters necessary to import the gerber layers for the IPC-Demo job.

This activity should take about 15 min to complete.

1. Create a new job.

   ‣ Click the **New** icon ☐ on the toolbars, or

   ‣ On the **File** menu, select **New**

2. Import IPC-Demo data using the Import Wizard.

   ‣ On the **File** menu, select **Import|Import Wizard**

   ‣ Click the **Browse** button

      o Browse to the "…Desktop\TrainingFiles\IPC-Demo\274D Gerber Set" directory.

   ‣ Click **Next**

   ‣ Click on the text in the **Data Format** column for the file **"0423CREP.DRL"**.

   ‣ In the **Select Converter** dialog, click on the **"Generic Tool List (Mil) 1.0"** option.

      o Click **OK**
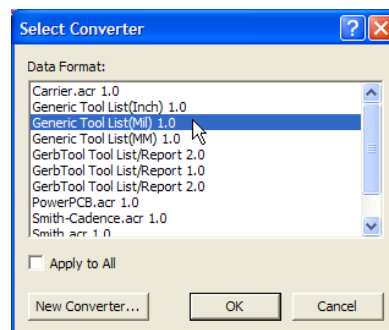
   ‣ Click **Next**
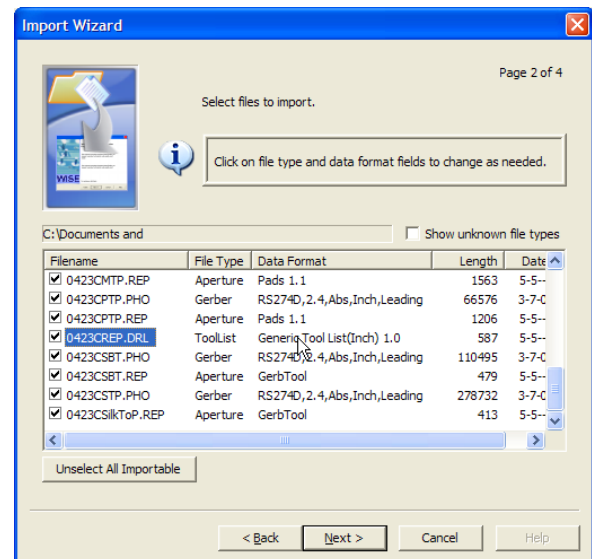
Note: In the Aperture/Tool Lists assignment tab, the gerber file **"0423CSTP.PHO"** is assigned to the **"0423CL01.REP"** aperture list instead of **"0423CSilkTop.REP"**. This is because GerbTool will try to automatically assign gerber files to the closest reasonable match it can find. If no match is 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.

▸ Click and drag the **"0423CSTP.PHO"** gerber file down the list and drop it on the **"0423CSilkTop.REP"** aperture list.

▸ Click **Next**

▸ Click **Finish**
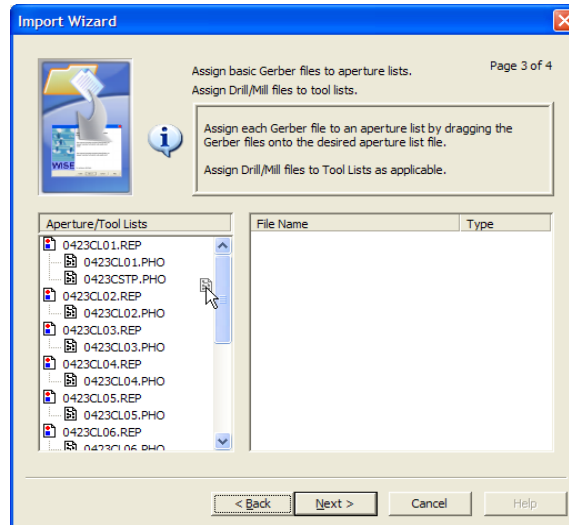
Note: Two of the aperture lists did not have the correct converter selected by GT during the automatic process. When this happens, you will get the Warning message below and the will be given a chance to have GT try to find the right converter.

▸ Click **OK** on Warning message.

▸ Click **YES** to confirm conversion message.

▸ Click **YES** to confirm the newly selected converter.

▸ Repeat this process for the second aperture list converter.
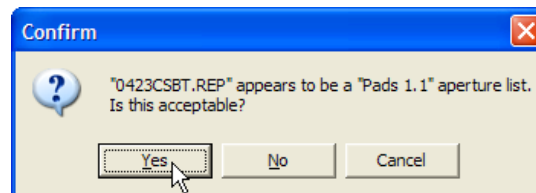
3. Save your GT database.

‣ Click on the save icon 🖵 or

‣ On the **File** menu click **Save.**

‣ Enter a name for your database such as "IPC-Demo_<yournamehere>.gtd".

4. After importing your gerber data in the previous activity you can click on the Main tab **Main** at the bottom of your workspace. Your graphic view should look like the image on the right.
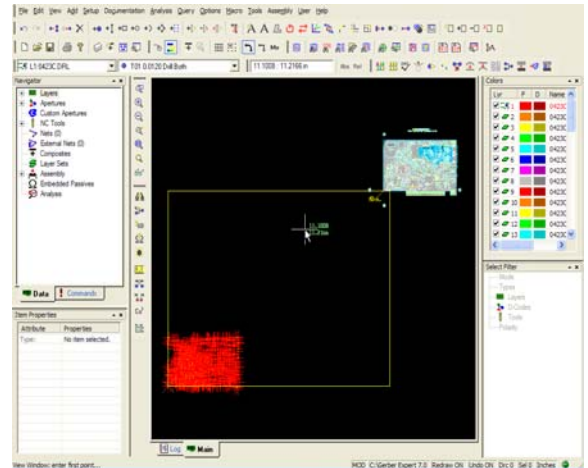
5. Review your data files with your instructor using the decision diagram from Figure 2.2 to decide what your best option(s) is for reverse engineering this job.

Review questions

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

_____

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

_____

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

_____
_____
_____

ID/Sort Layers

In this lesson we will describe the functions and processes used to identify the different layer types of your design and how best to sort and color them. We will also discuss the method used to create a separate border layer if one was not provided with your original gerber files and how to convert a gerber Drill layer to an NC Drill layer. Finally we will remove all unnecessary "garbage" data from the gerber layers of your job.

- ▸ Definitions
    - o SMD – Surface Mount Device
    - o THD – Thru Hole Device
    - o Tag – Identify a layer's type
    - o PCB – Printed Circuit Board
    - o Garbage data – any unnecessary drawn objects on the gerber layers, especially the electrical layers

Lesson objectives

In this lesson you will do the following:

- ▸ Tag layers of your design
- ▸ Create a Border layer
- ▸ Create an NC Drill layer from a gerber drill layer
- ▸ Sort layers into a sensible order
- ▸ Modify layer coloring to provide contrast
- ▸ Delete all "Garbage" data

Estimated completion time

This lesson usually takes 20 minutes to complete.

Identifying Layer Types

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

▸ For **Netlist/Netlist Compare**

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

▸ For complete **DRC/MRC**

  All the above plus:
  o Border
  o Top and/or Bottom Mask (Paste and Solder)
  o Top and/or Bottom Silk

To change a layer's type

1. In the **Navigator,** click on the plus box next to the desired layer.
2. In the layer detail list, the second item is the layer type. Right-click on the type and a shortcut menu appears. The current layer type has a checkmark next to it.
3. Select the new layer type from the menu.
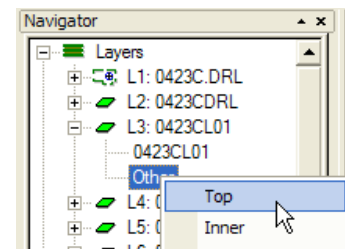
If you change the type to an NC layer, a tool table is automatically assigned to it. Its name appears in the layer information in the Navigator. If you want to change the assignment, right-click on the tool table name, select the Change NC Tool Tables command, and the name of the desired tool table.

Activity: ID (Tag) layers from IPC-Demo job

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

This activity should take about 10 min to complete.

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 GT. 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

1. Right-click on any layer in the Color Window and select **All Off**.

    ‣ Note that the active layer remains checked and visible. Uncheck the active layer now.

    ‣ Skipping the layer "**0423CDRL**" 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**".

2. Use the zoom commands to zoom-in around the descriptive text label for this layer as shown in the image above.

3. Click the + sign to expand the Layers list in the Navigator.

    ‣ Click the + sign next to layer "0423CL01".

    ‣ Right-click where it says **Other** and select **Top** for the type.
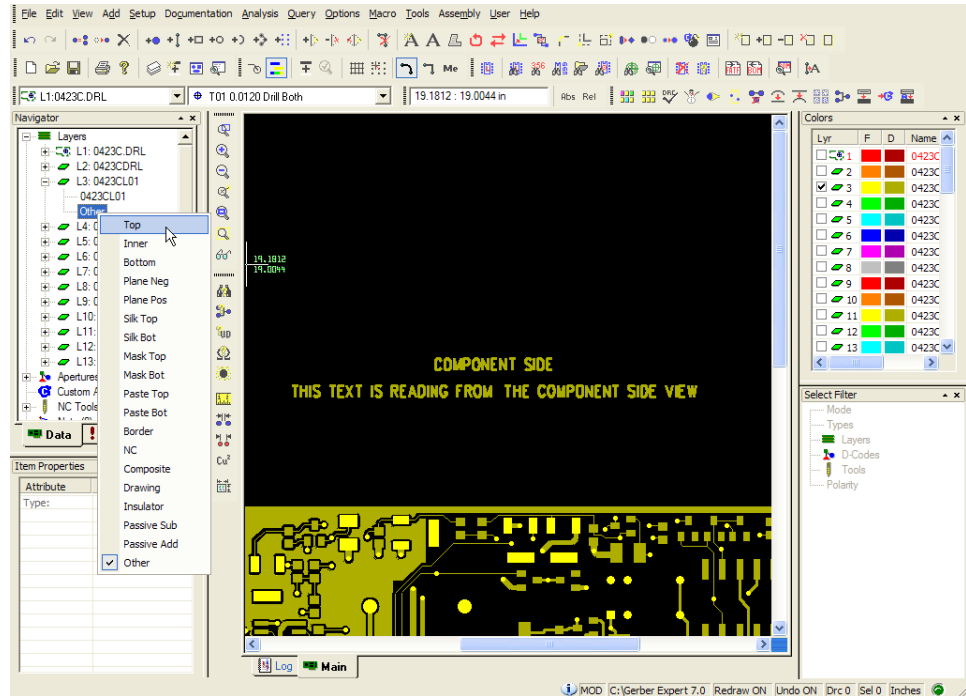
4. Repeat this process for the remaining layers (Still ignoring layer "**0423CDRL**"). Try to use the descriptive text labels on the layers and/or the layer names themselves to determine the type for each. If you have trouble determining the type or to check if your choices are correct you can refer to the table below.

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

Creating a Border Layer

When you receive gerber files for a product 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, it is important to have a Border layer or at least an accurate representation of the extents of your circuit outline for your Border layer. For Advanced Panelization applications you will need this for accurate spacing.

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.

To convert Draws to Polylines

This command converts all contiguous, drawn line segments on visible layers into single polyline (GerbTool's representation of a contiguous line segment) entities. Very often your circuit outline is a 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.

1. Turn on the layers you wish to be processed, and turn off any layers that you do not want processed.
2. Select the **Tools|Convert|Draws To Polylines** command. All visible layers are immediately processed, and you are informed how many draws were converted, and the number of resulting polylines.
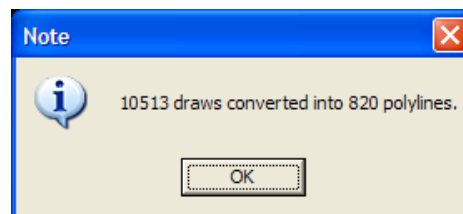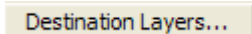
To create a Border Layer

1. Find an existing layer that has the border of your circuit outline defined on it. Silkscreen layers usually have a border.

2. Select the **Edit|Copy** command

   - or -

   From the toolbars click the Copy icon

3. Click the Destination Layers tab at the bottom left corner of your application window.

▸   Select the next available <empty> layer.

4. In the **Selection Filter**

▸ Change the Mode to **Window +Xing**.

▸   Also make sure the Type **Poly** is checked.

5. Click and drag a selection box around any portion of circuit border. (If you ran the Draws to Polylines conversion then this method should allow you to select the entire border.)
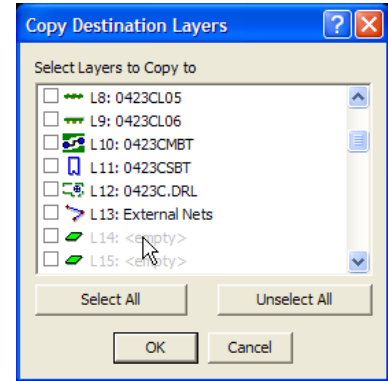
The prompt at the lower-left corner of your workspace says, "**Copy Window: enter copy from point...**" In order to drop it in the correct position on the new layer you will need to use the **Absolute Coordinates** command.

6. Press the **Enter** key and the **Absolute Coordinates** dialog appears.

    ○ Enter a value of **0** for both X and Y fields.

    ○ Click **OK**
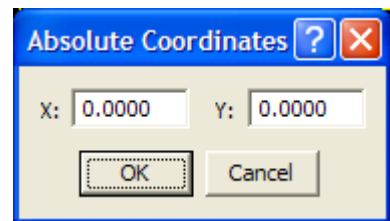
You now have a copy of the border attached to your cursor and the prompt at the lower-left corner of your workspace now says, "**Copy Window: enter insertion point...**"

7. Press the **Enter** key again and the **Absolute Coordinates** dialog appears.

    ○ Enter a value of **0** for both X and Y fields.

    ○ Click **OK**

8. Press the **Esc** key twice to end the function.

9. Your circuit border has now been copied to the destination layer you chose and should be aligned with the layer you copied it from.

▸ Use the **Navigator** to change the layer type to **Border** and to rename it appropriately.
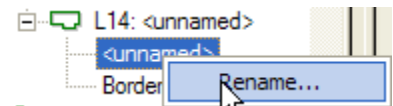
Creating an NC Drill Layer from Gerber

When you receive gerber files for a product, 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 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.

Activity: To convert Gerber to NC Drill

This activity should take about 5 min to complete.

1. Turn off all layer accept for layer **0423CDRL** and zoom your workspace out to the **View All** mode. (This is not a requirement, but will make it easier for you to see the conversion take place after executing the commands below.)
2. Select the **Tools|Convert|Gerber to NC** command from the menu. The **Gerber To NC** dialog.
3. Select the **0423CDRL** layer for the **Source Layer**.
4. Select the next available **<Empty>** layer for the **Destination Layer**.
5. Leave the NC Tool Table at its default value of **<New Table>**.
6. Click **OK** to execute the command.
7. A new NC Drill layer has been created on the Destination Layer you selected. Turn this layer on now to view your results.
8. A new NC Tool List has also been created. Click on the "+" symbol next to the NC Tools item in the Navigator to see it.
9. Delete the new NC Drill layer as well as the original 0423CDRL layer. We will be using the original NC Drill layer we imported for the remainder of this job.
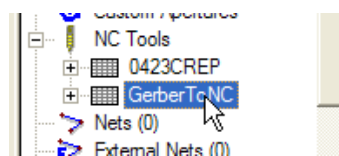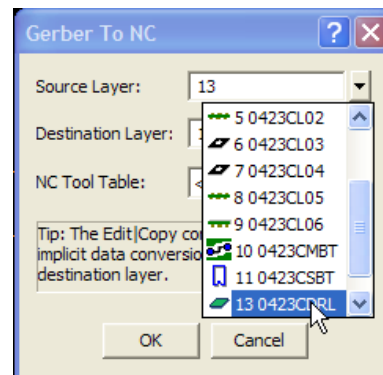
Sorting and Coloring Layers

The **Activity** for this section will be to follow the instructions below with the IPC-Demo job using the method from the **Tip** to sort layers.

Activity: Sort and Color your Layers

This activity should take about 10 min to complete.

1. Under the Layers list in the Navigator, click on the layer you wish to move, and continue to hold down the mouse button.
2. Move (drag) the mouse cursor until it is over the layer (in the Layers list) that you wish to move the layer to.
3. Release the mouse button. All other layers are reordered, as necessary, so that no empty layers are created and no duplicate layer numbers exist.
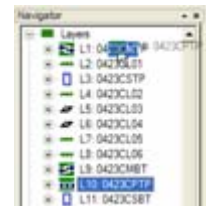
Tip: A suggested method for determining how to sort your layers is to order them as if you were looking thru the PCB from the top down. So a typical PCB you would use the following sequence to sort your layers: Top Paste, Top Silk, Top Mask, Top Copper, Inner/Plane Layers, Bottom Copper, Bottom Mask, Bottom Silk and Bottom Paste. From there you can sort your extra layers such as Drills, Drawings, and External Nets in any order you wish.

4. To color your layers, see the activity "**Use the Color Window**" in Chapter 1 of this manual.

Review questions

1. What layers are required to process an accurate netlist?

    _____
    _____
    _____

2. Why is it important to have a Border layer?

    _____
    _____
    _____

3. What is the suggested method for sorting your layers?

    _____
    _____
    _____

Activity: Delete Garbage Data from all layers

In this activity you will be removing all "garbage" data from the layers of the IPC-Demo job. Garbage data such as targets, layer labels, etc..., especially on electrical layers, will cause invalid nets when you generate a netlist.

This activity should take about 10 min to complete.

1. In the **Colors** window right-click on any layer, select **All On**.

2. 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 garbage data.

3. Select **Edit|Delete** from the menus

- or - Click the **Delete** icon ✕ on the toolbars

4. In the **Selection Filter:**

‣ Set the **Mode** to **Window**

‣ Verify that all **Types** are checked

5. Click and drag a selection window around groups of any obvious garbage data in the workspace. Be careful not to select any of the circuit area.

‣ Click **Yes** to confirm each deletion.

6. Select **Tools | Pad Removal | Stacked**

7. Choose all layers and set the tolerance to .5 mils or .0005" and choose **OK**.

8. Choose **Analysis | Find Duplicates**.

9. Select All layers and choose **OK**.

10. Expand the **Analysis** section of the navigator.

11. Right click the **Duplicate Items** folder and choose fix all.
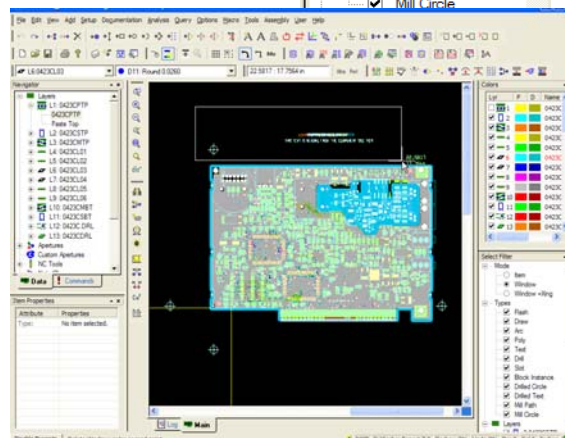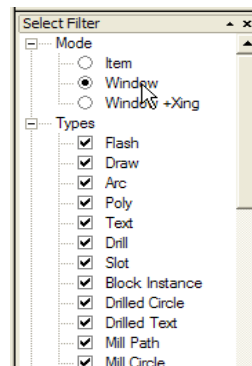
Note: After deleting all obvious garbage data it is a good habit to review each electrical layer individually and look for any text or other drawn objects that are not a part of the circuitry such as a board outline or layer windows and delete these as well.

Data Alignment

In this lesson we will describe how to use the Align Layers, Edit Origin and Snap Pads commands to align all your data together so that GT 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

This lesson usually takes 15 minutes to complete.

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.

To align layers

1. Determine the layer to which all other layers should be aligned with (the master layer).

Tip: When selecting a "Master Layer", 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.

2. Find an item on the master layer to use as a "base item", or reference point.

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

3. Select the **Edit|Align Layers** command.
4. Click on the item you chose as a master reference point. Use the Selection Filter, if necessary.
5. Click on an item on each layer to be aligned that corresponds to the reference point. As you select each item, it is highlighted and you are asked to confirm your selection. The entire layer is then aligned with the master layer.

Again, 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.

6. If you wish to select a new base item, press the **Esc** key and repeat steps 4 and 5.

   -or-

   When you are finished, press the **Esc** key twice to exit the function.

Activity: Align Layers

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.
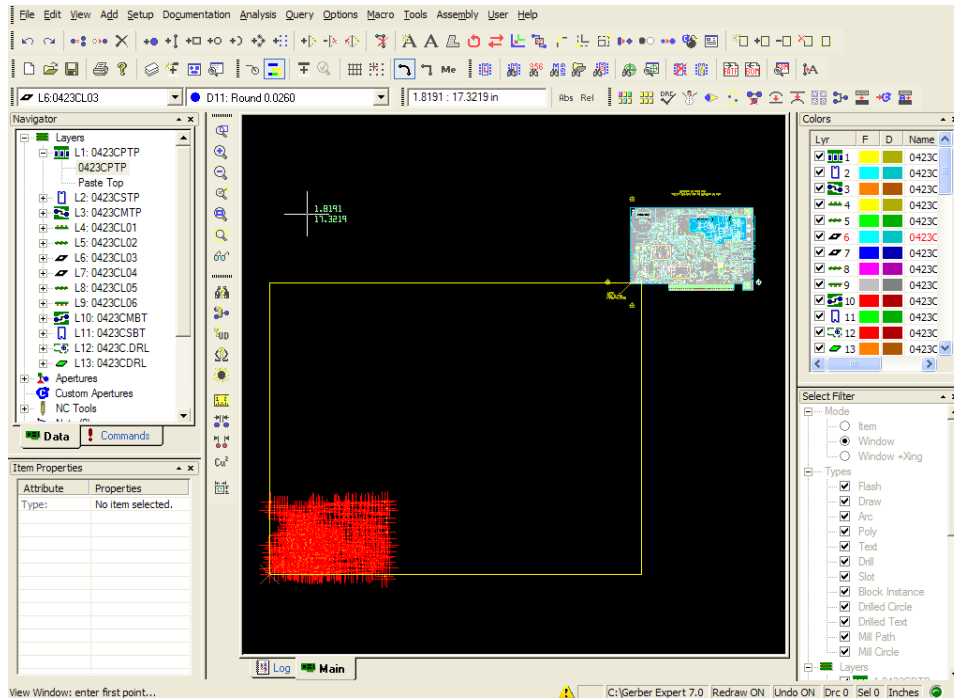
Note: When executing commands such as Align Layers, it is often necessary to zoom in and out of certain areas of the design while the command is still active. **Clicking on any Zoom Icon while executing a command will cancel it!** Some commands provide their own zoom buttons inside their dialog box, but if no dialog is provided the '**+**' and '**-**' keys on your keyboard should be used.

1. Select **Edit|Align Layers** from the menus

   - or -

   Click the **Align Layers** icon on the toolbars

2. In the **Selection Filter**, uncheck all the **Types** except for **Flash** and **Drill.**

3. Select a reference point on the Master Layer.

   ‣ 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 seen here.
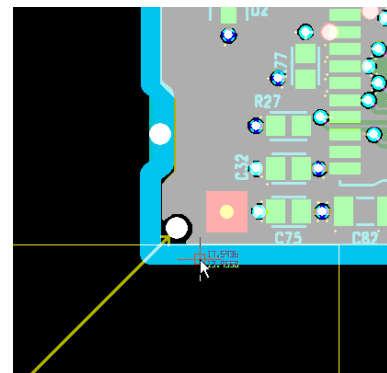
‣ Click on the tooling hole.

‣ Click OK on Choose Selection dialog

4. Select a reference point on the Layer to be aligned.

‣ Press the "-" key on your Num-Pad several times till you can see the drill layers again.

‣ 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.

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

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

‣ The **Choose Selection** dialog appears with two options available. (There are two layers that were misaligned from the rest, so you have an object from each layer listed.) Select the drill object first.

‣ Click **OK** in the **Choose Selection** dialog.
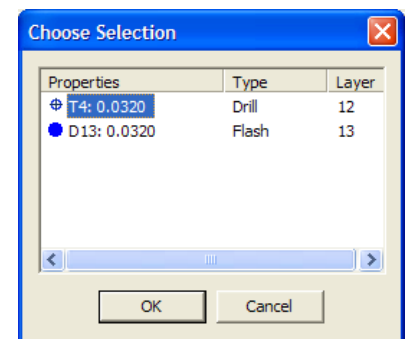
5. The Align Layer function is still running, allowing you to align multiple layers in a consecutive sequence. Repeat step **4** selecting the same reference point on the other misaligned layer.

‣ Press the **Esc** key twice to exit the function.
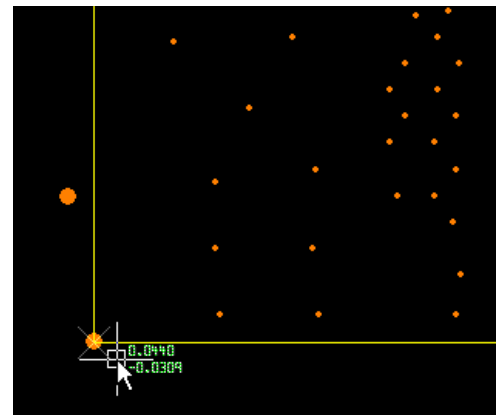
Edit Origin

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.

Activity:To Change the Origin

1. Select the **Edit|Origin** ⊞ command. The Change Database Origin dialog box appears.
2. If you select "**Database Updates Now**", your new origin selection is immediately committed to the database, and the **film box** (A continuous line that represents the film size.) and data in the display are altered to reflect it. If you select "**Database Updates After Next Reload**", the new origin is not committed to the database until after you save the database and reload it. The film box will automatically relocate to the origin you select, but none of the existing data (or imported data) will reflect the new origin until after the reload.

‣ Select **Display This Dialog Only If Shift Key Is Pressed** to cause the dialog box to appear in the future if you press the Shift key at the same time that you select the Edit|Origin command. If you select this option, any setting you select above becomes the default for the Edit|Origin command.

3. Click **OK**, and you are prompted to select a point to define the new origin.
4. Select the new origin point

‣ Hover 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 read the text as seen here.

**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.

‣ Set the **Selection Filter** as necessary

‣ Hover your mouse over the tooling hole and press the **Home** key. (This will snap your cursor to the exact center of the selected object.)

    o Press the **Esc** key to exit the function when finished.
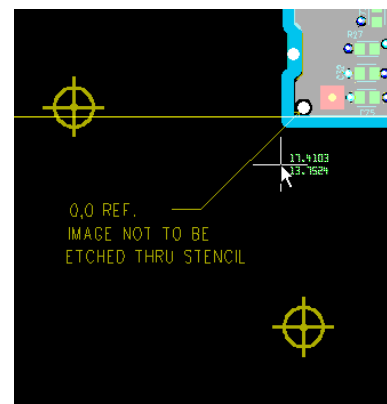
Snap Pads

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.

Activity: To snap pads

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 after converting drawn pads.

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. Select the **Edit|Snap Pads** command. The Snap Pads dialog box appears.
2. Select the **Golden Layer** whose pad or drill locations will be used as a "master" reference to align items on the **Snap Layer(s)**.

   ▶ Select the **Top** layer as your **Golden Layer**.

3. Select the Snap Layers that contain pads or drills to be aligned with those on the Golden Layer.

   ▶ Select All layers.

4. In the **Tolerance** box, enter the distance that coordinates of 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.
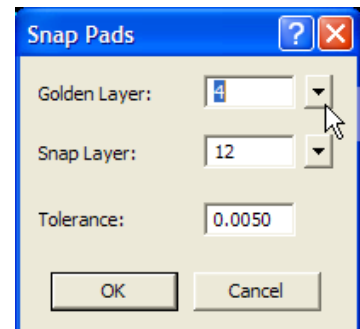
Note: The recommended setting is between 5-10 mils, use 5 mils (0.005 in.) for now. If you use a number larger than 10 mils, you

are risking that the command will snap objects together that are meant to be separate.

5. Click **OK**. Any qualified data are aligned and you are informed when the process is complete.

Review questions

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

_____
_____

2. What option should you select so your new origin selection is immediately committed to the database?

_____
_____
_____

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

_____
_____
_____

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

_____
_____
_____

Draws to Flash Conversion

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

Lesson objectives

In this lesson you will do the following:

‣ Use the Automatic Draw-Flash Converter

‣ Use the Select Draw-Flash Converter to manage pads that were missed by the automated process

Estimated completion time

This lesson usually takes 15 minutes to complete.

Automatic Drawn Pad Conversion

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.

Note: If you do not have any mask layers, use the Select Draws command discussed in the next section to convert your drawn pads to flashes.

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

Activity: To automatically convert draws to flashes

1. Select the **Tools|Convert|Drawn Pads|Automatic** command. The Automatic Drawn Pad Conversion dialog box appears.

    -Or-

    Select the **Automatic Convert Drawn Pads** icon from the toolbar.

2. The Control Layer is the layer number of the solder or paste mask layer that corresponds with the top or bottom layer that you wish to convert pads on. Click the down-arrow button to select the desired layer (to prevent an incorrect layer from being specified, you cannot type a number directly into the box).

‣ Select layer **0423CMTP**
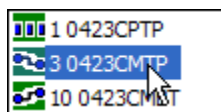
‣ If the selected Control Layer is a paste mask layer, the Paste Mask Expansion option allows you to enter a value by which paste mask openings will be oversized. Over sizing the openings may be required, because often paste mask openings are smaller than the corresponding pads on the top/bottom layer.

3. Click the Select Layers button to specify one or more layers that contain the drawn pads you wish to convert.

> ▸ Click the **Select All** button and it selects all layers and changes to an **Un-Select All** button.

> ▸ Click it again to deselect all layers.

> ▸ Now click to select only the **Top** and **Bottom** layers. Your dialog should look like the image on the right.

> ▸ Click **OK**
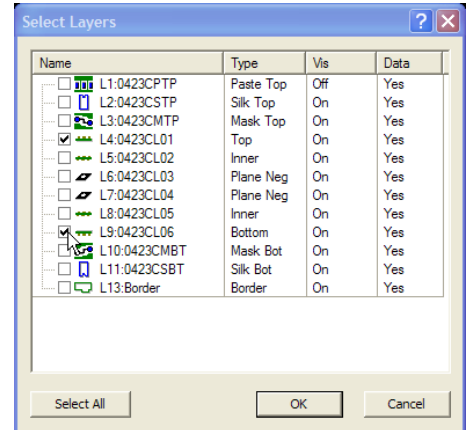
4. Specify the Maximum Size, in the X and Y dimensions that a group of draws may be, and still be considered for conversion to a flash. If a group of draws exceeds the maximum size, they will not be converted.

> ▸ Leave the defaults of 0.25" for this exercise.

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.

5. 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.

> ▸ Leave the default of 0.0005" for this exercise.

6. 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.
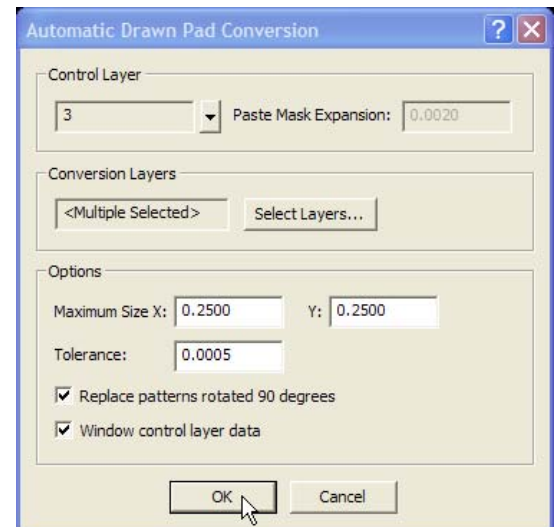
7. Select the Window Control Layer Data option if you want to convert only pads in a specific area. This is useful for designs that have text blocks or other data outside the actual board area, as excluding that extraneous data will speed up the process.

8. Click the **OK** button.

    ▸ If you did not select the Window Control Layer Data option, the
       conversion process begins immediately. If the option was
       selected, you must select two points to define the area to be
       processed.

9. When the process is complete, a message is displayed indicating
    the number of converted pads.
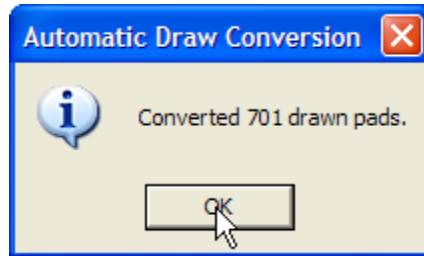
    ▸ If you selected the Window Control Layer Data option, you may
       select another area to process, or press the **Esc** key to end the
       function.

10. 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.
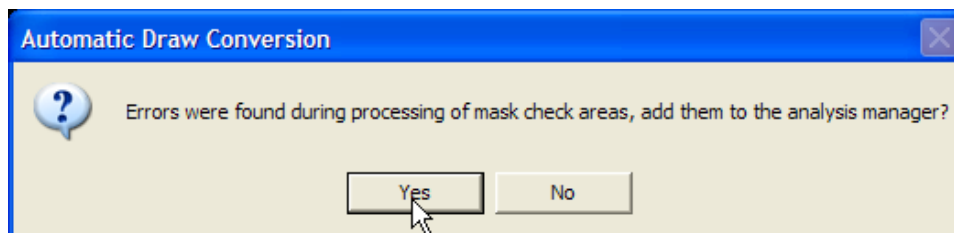
    ▸ Click **Yes** to add these errors to the analysis manager.

11. Repeat this process again for the IPC-Demo job, but select
    layer **0423CMBT** as the control layer this time.

Select Draws

This function converts drawn pads to flashes, based upon data that you manually select for conversion. This function is useful for dealing with any unconverted pads left over as a result of errors from the automated process described in the previous activity. It is also your backup option if you do not have a Solder Mask or Paste Mask layer to use for the automated process.

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

Activity: To convert draws to flashes by manually selecting draws

1. Begin to review the errors from the previous activity as described in step 10 above.

   ‣ You can review these errors using the Error Analysis Manager in the Navigator.

Tip: Prior to reviewing the errors, if you turn all layers **Off**, the analysis manager will automatically turn on the effected layer(s) of an error and zoom your view in when you click on the error in the list. This will give you an uncluttered view of the effected objects.
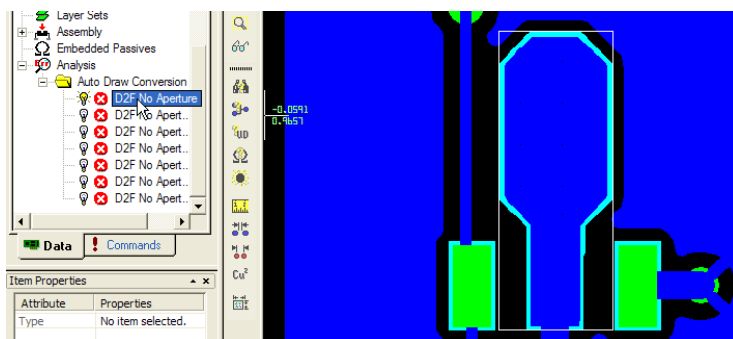
2. After selecting the first error, select the **Tools|Convert|Drawn Pads|Select Draws** command.

-Or-

Select the **Convert Drawn Pads** icon ▾ from the toolbar.

3. In the Selection Filter settings, make sure that the **Type** is set only to **Draw** and the **Mode** is set to **Window**.

4. Under **Layers** in the Selection Filter, uncheck the mask layer so that it will be visible, but not selectable. Otherwise the converter will find both objects on the two layers and it cannot convert more than one unique object simultaneously.

5. Select one occurrence of the drawn pad that you wish to convert. The **Convert Drawn Pads** dialog box appears.

Tip: You may notice that the Error Analysis Manager created a highlight box around the pad that was affected by the error you clicked on. You can use this box as a template for where to click-and-drag your selection window.

‣ 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.

6. Select how you want the draws replaced.

‣ *Create True Size/Shape Custom Apertures*

Select this option if you wish to have selected items converted to a single **custom aperture**, (Special user-created shapes that are normally used as flashes. They can be created or modified in the Custom Aperture Editor.), with the same appearance.

　　o *Use this option for the IPC-Demo job.*

‣ *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 5), then up to 3 apertures may be created.

‣ *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.

7. 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.

   ‣ We will use this option for the IPC-Demo job.

8. 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.

   ‣ We will NOT use this option for the IPC-Demo job.

9. 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.

10. Set the **Tolerance** to increase the system's ability to recognize pads that are the same size, and account for minor variances in drawn data, you can specify a value. 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. Usually a value of 0.0005 inches is sufficient.
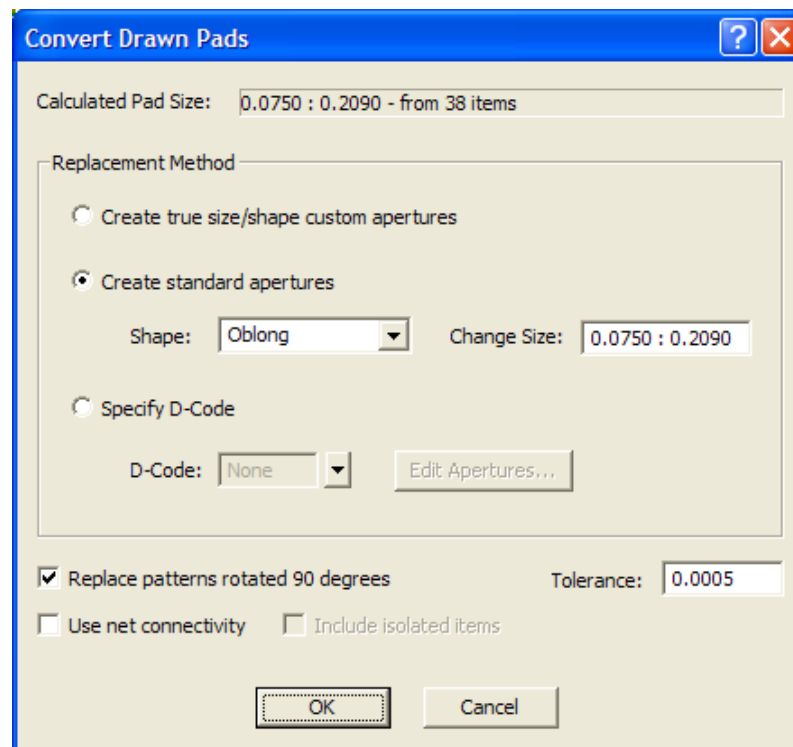
11. After you make your selections and click **OK**, all occurrences of any matching drawn pads are located and you are asked to confirm whether to continue.

    ‣ Click **OK** to confirm the conversion

12. Repeat this process as necessary while reviewing the errors that were found during the automated conversion process or if you are using this process because you have no mask layers available.

▸ There should only be 2 pads that you need to process in total for the IPC-Demo job. The other errors are the result of bits of traces found within the mask areas as seen here.

Tip: The command stays active after each conversion, so you can simply click on the next error in the Analysis Manager and continue selecting the next set of draws.

▸ Press **Esc** key to exit the command.
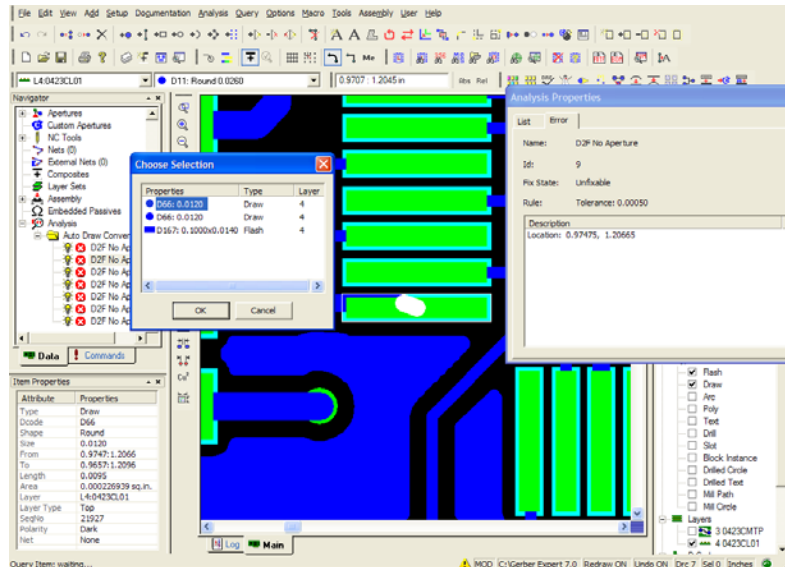
Review questions

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

_____
_____
_____

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

_____
_____
_____

3. In What situation should you use custom apertures when running the Select Draw command?

_____
_____
_____

Netlist Comparison

Graphical netlist comparison allows you to visually compare your design data against the original netlist. GerbTool performs the comparison automatically, and all violations are stored internally like DRC/MRC and DFF violations. 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 ODB++ or PADS ASCII database netlist, the **File|Import|Netlist|ODB++** and **File|Import|Netlist|PADS ASCII** commands import only the netlists from those databases.

Lesson objectives

In this lesson you will do the following:

‣ Generate an internal netlist

‣ Import an IPC D356 netlist

‣ Use netlist compare to find differences between the CAD netlist and the gerber data

Estimated completion time

This lesson usually takes 15 minutes to complete.

Compare Netlists

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

Prior to netlist generation and netlist comparison, the following steps need to have been taken:

- Using the **Setup|Layers** command or the Navigator, make sure that each layer is "tagged" with the correct layer type. This is required so that the system understands what each layer is.
- Make sure that any extraneous data on your electrical layers that are outside the perimeter of the board, such as title blocks, targets, crop marks, etc. is removed.

  Tip: We recommend that any documentation for the board be done on a separate Drawing layer. If you wish to keep the data, placing it on a drawing layer assures it will not interfere with any processing, such as netlist extraction.

- If you have any drawn pads in your database, select one of the Tools|Convert|Drawn Pads commands to convert them to flashes.
- Use the **Analysis|DRC/MRC** command to make sure your layers are properly aligned. If any layers are misaligned, you can use the **Edit|Align Layers** command to align them.
- Use the **Tools|Pad Removal|Stacked** command to remove any redundant pads.
- An NC layer with through-hole drill information is necessary to determine layer to layer connectivity, as well as direct connections to plane layers. The system will automatically display your NC layer in the Thru-hole Drill Layer box. If you want to select a different NC layer, click the 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.

Activity: To compare netlists

1. Select the **Tools|Netlist|Generate** command to extract a netlist from your data.
2. Select the 0423C.IPC (external) netlist using the **File|Import|Netlist|IPC-D-356** command.

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. The net points are displayed in four ways for easy reference. Downward pointing triangles represent test points accessible from the top side of the board. Upward pointing triangles represent test points accessible from the bottom side of the board. X's represent test points which are accessible from both sides. Triangles pointing to the side represent points which are only accessible from the inner layers (for blind and buried technology designs).
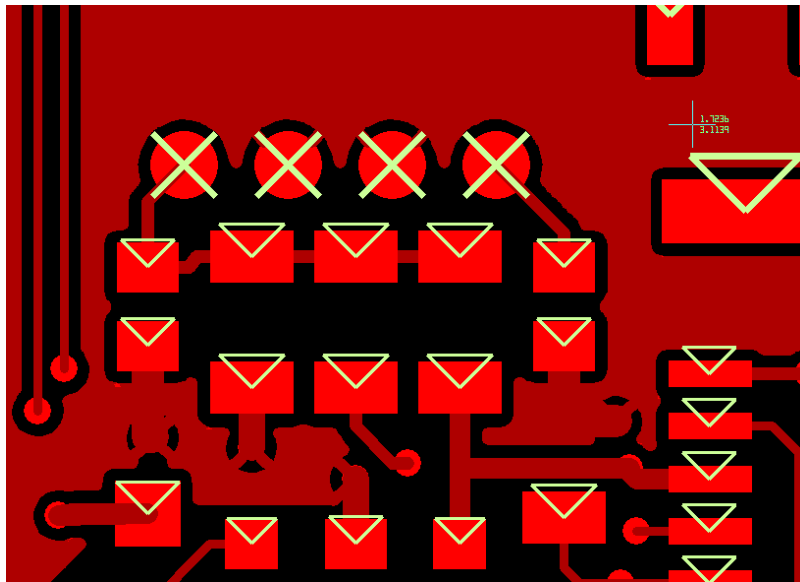
3. Select the **Analysis|Netlist Compare** command.
4. Select the Shorts and Opens checks and click RUN.
5. After performing the analysis, discrepancies are automatically displayed in the **Navigator**, which you may use to view the errors in the workspace and print reports.
6. Double-click on the error in the Navigator to view it in the workspace. The GerbTool internal net in question highlighted, and zoomed to fit the screen. The external net points are shown in a unique color for each net.

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

7. No errors are found. If GerbTool finds an error, right click the error and choose properties, an image similar to the following is displayed:
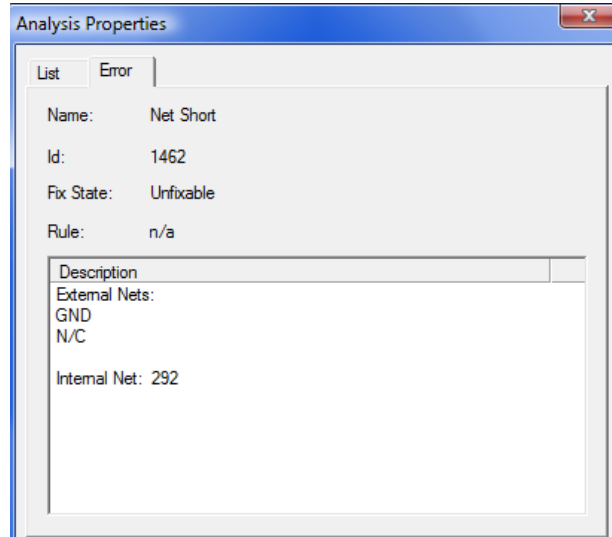
8. Find the precise location of the error more quickly by right-clicking on the error in the Navigator, and selecting the Pin-Point Error command.

    *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

1. What are the layer requirements for netlist comparison?

    _____
    _____
    _____

2. What happens if there is no Thru-hole drill layer present?

    _____
    _____
    _____

3. Where are the netlist compare errors found?

    _____
    _____
    _____

4. How can you find the actual location of the error?

_____
_____
_____

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:

‣ Use the Paste Optimize command to take care of under sizing the paste layer.

‣ Use the Stencil tools to create specially shaped paste openings such as home plates and melfs.

‣ Use the Window Panes macro to create specially shaped paste openings.

Estimated completion time

This lesson usually takes 35 minutes to complete.

Optimize 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. If no Paste layer exists in your design, Generate Paste Mask instead of Optimize. The settings and options are nearly identical for both commands..

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

Activity: To optimize a paste mask

1. Use the **Tools|Convert|Drawn Pads|Automatic** and **Tools|Convert|Drawn Pads|Select Draws** commands to convert your drawn pads to flashes on the paste mask layer as in the previous lessons.
2. If you want any items in your database ignored during the mask optimization process, create a Select Group of those items.
3. Select the Tools|Paste Mask|Optimize command. The Optimize Paste Mask dialog box appears.
4. Select optimize the Top Mask.

   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.

5. In the Maximum Errors box, specify the maximum number of allowable errors as 1000. If the optimizer 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 undersize amount that is not appropriate for your design.
6. Specify your Desired Undersize as 6 mils.
7. Select the Round Corners option for the openings, define the corners as 10 percentage of the shortest side of the aperture.
8. 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.
9. 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.

11. Select the Ignore Pads On Electrical Layers Less Than option and select a minimum pad size of .025.
12. Select the Fix Errors option. 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 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.
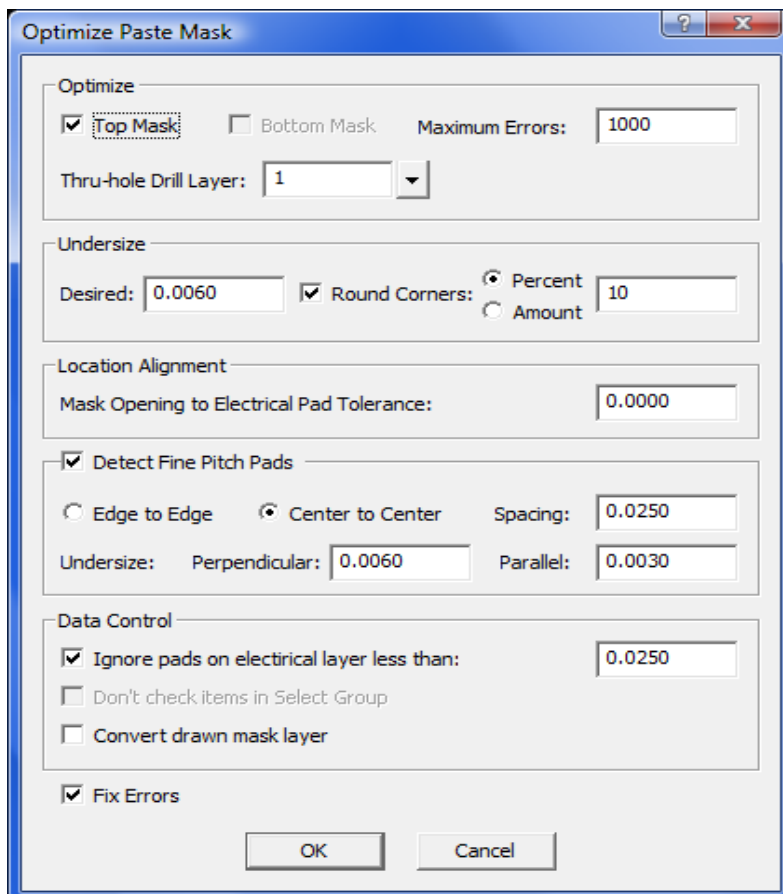
15. After making your selections, click OK to execute the optimization.
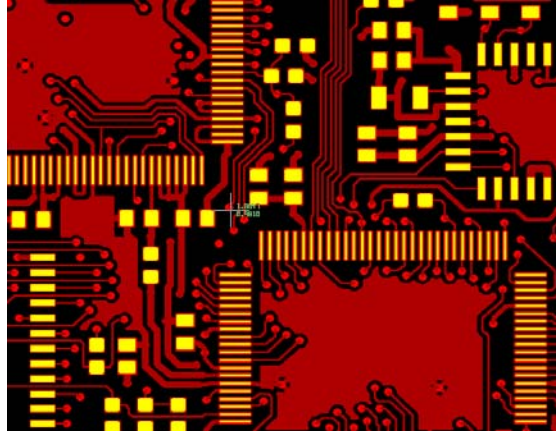
Review questions

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

_____
_____
_____

2. What happens if there is no Thru-hole drill layer present?

_____
_____
_____

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

_____
_____
_____

Create special stencil shapes

In the following activities we will use the Stencil Shapes tool which allows simple rule based conversions of stencil pad pairs to common IPC stencil shapes and a Window Panes macro designed to create ideal solder paste conditions for certain component land to pad situations. Creating special stencil shapes should occur after the general paste reduction has been applied.

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

Activity: Setup stencil shapes

1. Select the Tools|Stencils|Setup Shapes command, and click the new button.
2. Enter the values from the screen shot below and select OK.
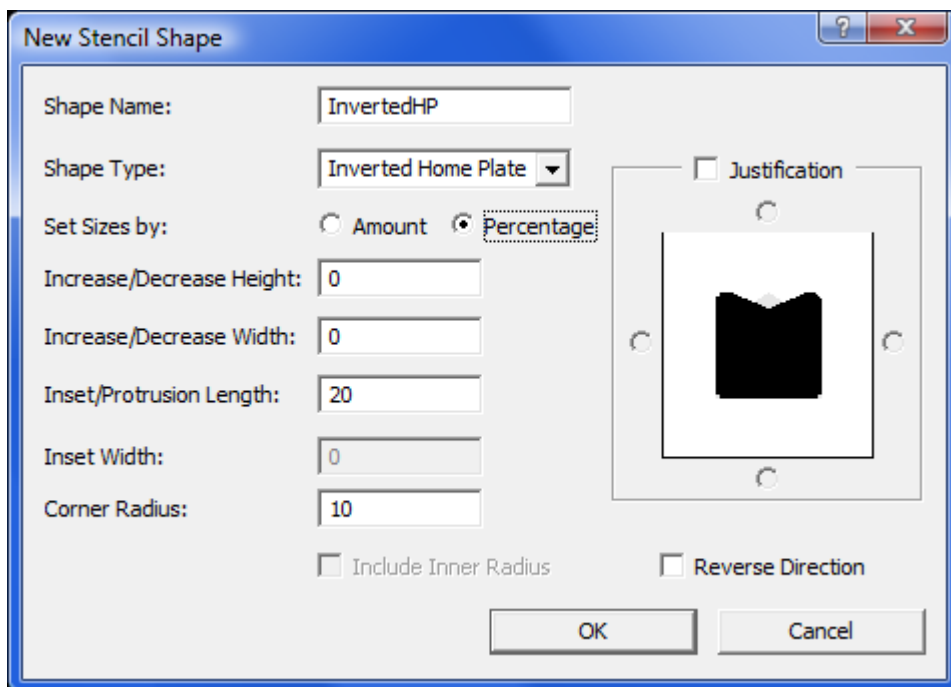
3. Select OK again on the Stencil Shapes Setup Dialog.
4. Turn on the visibility of the Silk Top Layer.
5. Select the **Tools|Stencils|Manual Conversion** command to teach the system which stencil shapes you want to associate with which original openings over pad pairs. Select the pair of pads marked as X1 on the Silk Top Layer. After selecting the pair of stencil shapes, click "No" when you are asked whether you want to convert the matching openings.

6. Next, Select the pair of pads marked as L2 on the Silk Top Layer. After selecting the pair of stencil shapes, click "No" when you are asked whether you want to convert the matching openings.

   The tool will save the assignments you have made, but the openings will not be converted at this time.

7. Select the **Tools|Stencils|Automatic Conversion** command and 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 steps 4 and 5, 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, you can skip steps 3 and 4 above. Just select the **Tools|Stencils|Automatic Conversion** command, click the Load button to load the file, and run the conversion.
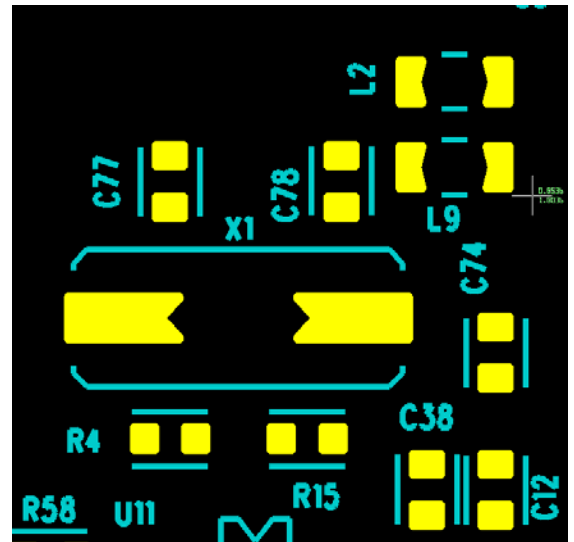
Activity: Window Panes

1. Select the **Macro | Load** command and choose the WISE_MultiPanes.mac and choose open.
2. Select the **Macro | Run** command ("M" on your keyboard) and select "MultiPanes" in the Macro Name listbox.
3. You are prompted to enter an input layer. Type the number of your Paste Layer and select OK.
4. Window around the large ground pad for U18 as shown in the image to the right:
5. You are prompted to enter a gap size. Enter .025.
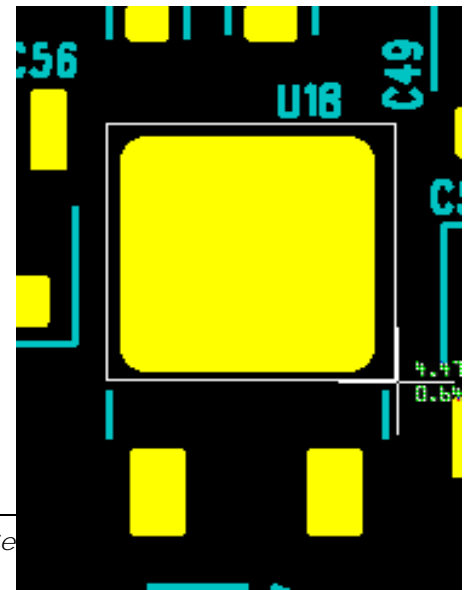
6. Enter a number of column stripes: 2
7. Enter a number of Row stripes: 3
8. Select Yes to convert all matching shapes, although in this design there will be only one match.
9. You are asked whether or not to convert another. In this case we will choose No as there are no other shapes on this design that look as though this technique should be used.
10. Your pad should look like the image below:
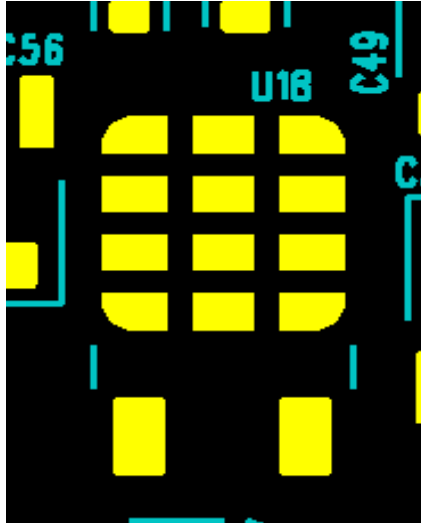
Review questions

1. Why are home plate and inverted home plate shapes used?

    _____
    _____
    _____

2. What is the reason to use the window panes macro?

    _____
    _____
    _____

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

    _____
    _____
    _____

DRC/DFM Analysis

The GerbTool 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:

▶ Use Analysis to collect information about the design.

▶ Setup DRC/DFF checks to find problems in the design.

▶ Review and fix problems found during Analysis.

Estimated completion time

This lesson usually takes 35 minutes to complete.

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 through-hole drill hits.
- **Min Drill/Drill** - Reports the minimum spacing between drill hits.
- **Min NPTH/Copper** - Reports the minimum spacing between non-plated through-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 through-hole drill hits.
- **Min PTH Pad Size** - Reports the size of the smallest pad that has been pierced by a plated through-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 through-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 through-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 Through Hole Drill Layer need to be selected, Mask, Paste, Silk and Blind and Buried drills will be found automatically if they present in the design with their layer type set.

1. Open the the Analysis Setup by clicking Analysis | DRC/MRC.
2. Click the "Load…" button in the lower left corner of the dialog.
3. Select the Training.rul file and click "Open".
4. Click "Run" to run the information checks defined.

5. After a few moments the following dialog will pop up:
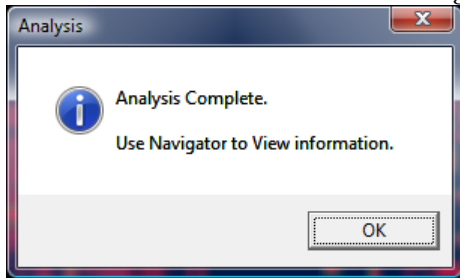
6. Open the Analysis tree in the Navigator, and expand the results.
7. Right click on a particular check such as "Conductive Layer Count" and choose properties to view the results of an individual check:
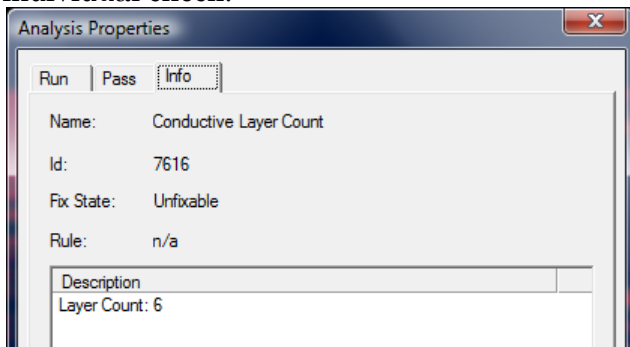
8. Right click on "Information" and choose "Print All" or "Report" to get a single line by line report of all the information checks performed.

Review questions

1. What are the layer requirements for using the Analysis checks?

_____
_____
_____

2. What happens if there is nothing selected in "Layers to Check"?

_____
_____
_____

3. How can you generate a report from the Analysis results?

_____
_____
_____

DRC/DFF Checking

In the following activities 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 through-holes (PTH) and non-plated through 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 GerbTool'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.

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

### Activity: Performing DRC/DFF Analysis

1. Open the the Analysis Setup by clicking Analysis | DRC/MRC.
2. Add check marks to the DRC and DFF passes.
3. Unselect the Information pass (since we've already run it).
4. Click "Run" to run the checks defined for these passes.
5. After a few minutes A dialog opens reporting that errors were found.
6. Open the Analysis tree and expand the DRC checks.
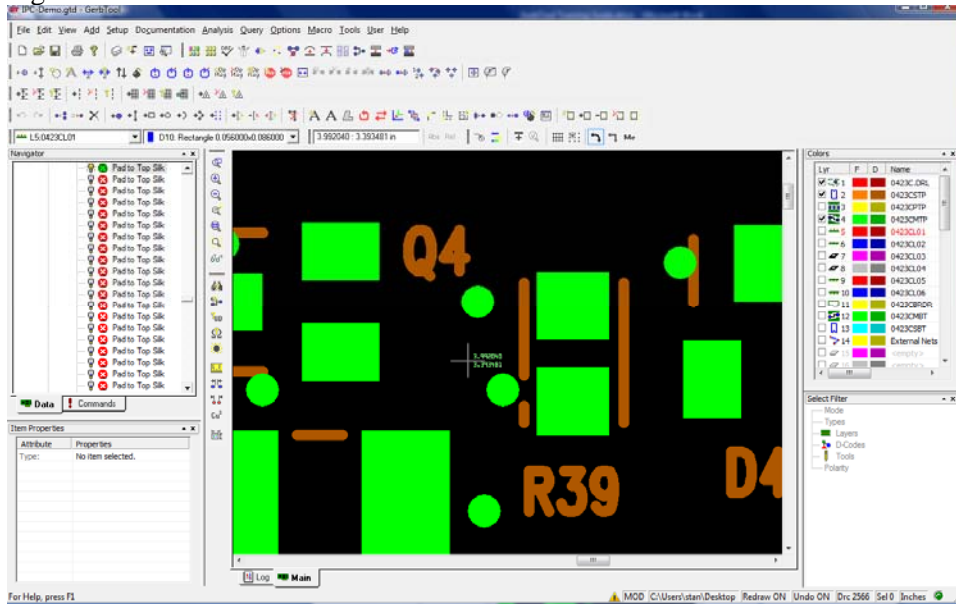
7. Click on a Pad to Border error:
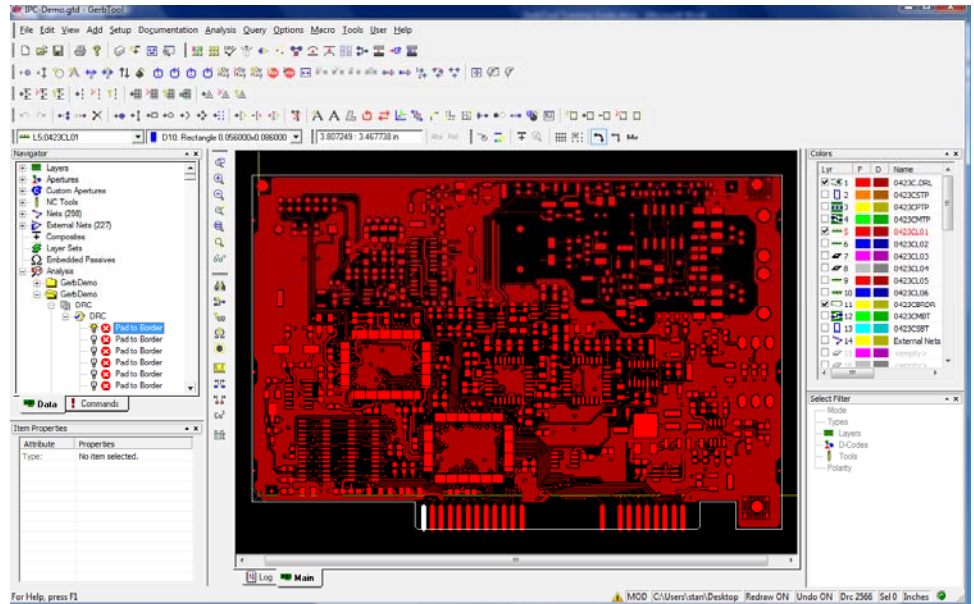
As you can see, there are several pads that break the rule set up in the check.
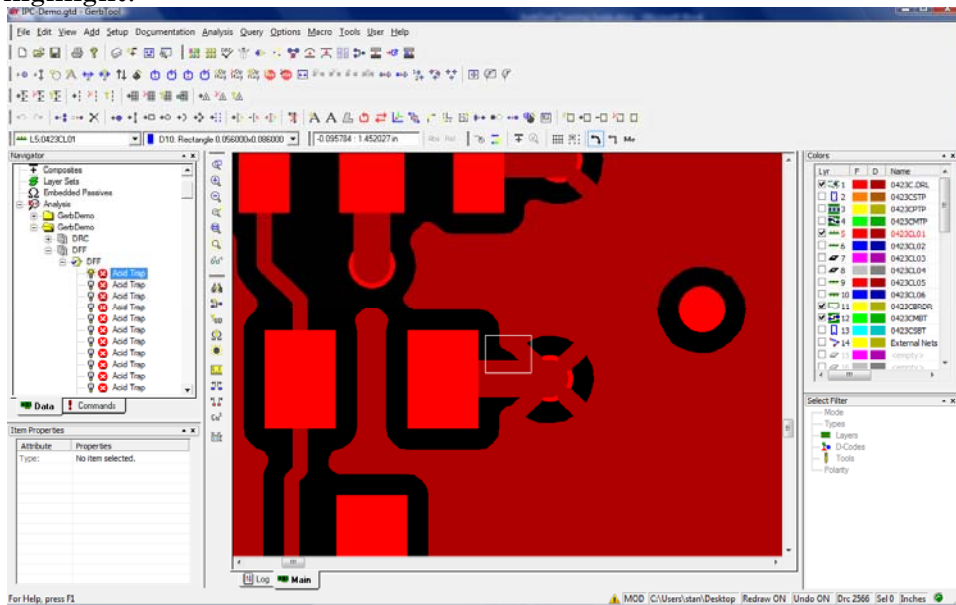
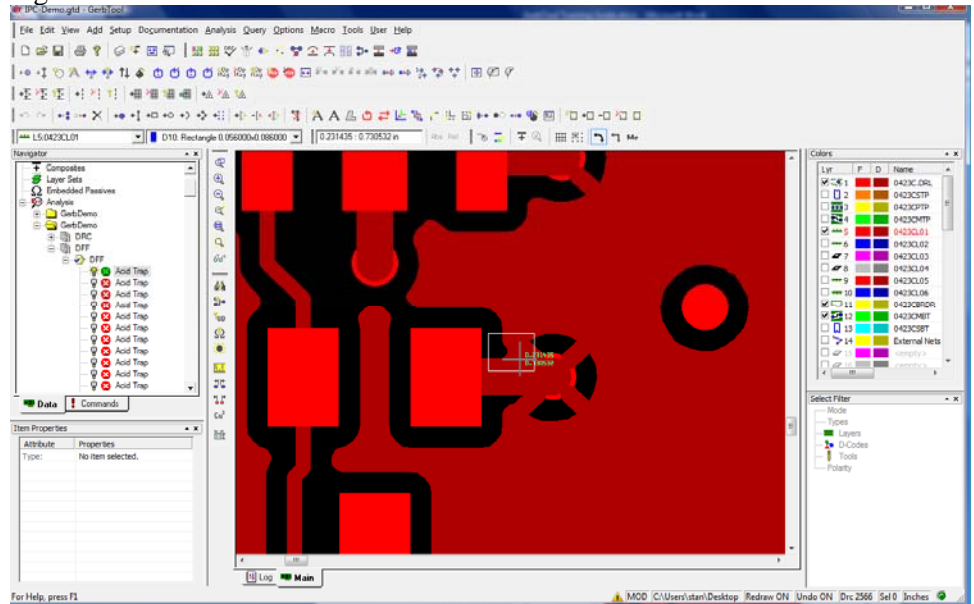8. Click on the first Pad to Top Silk error to highlight:
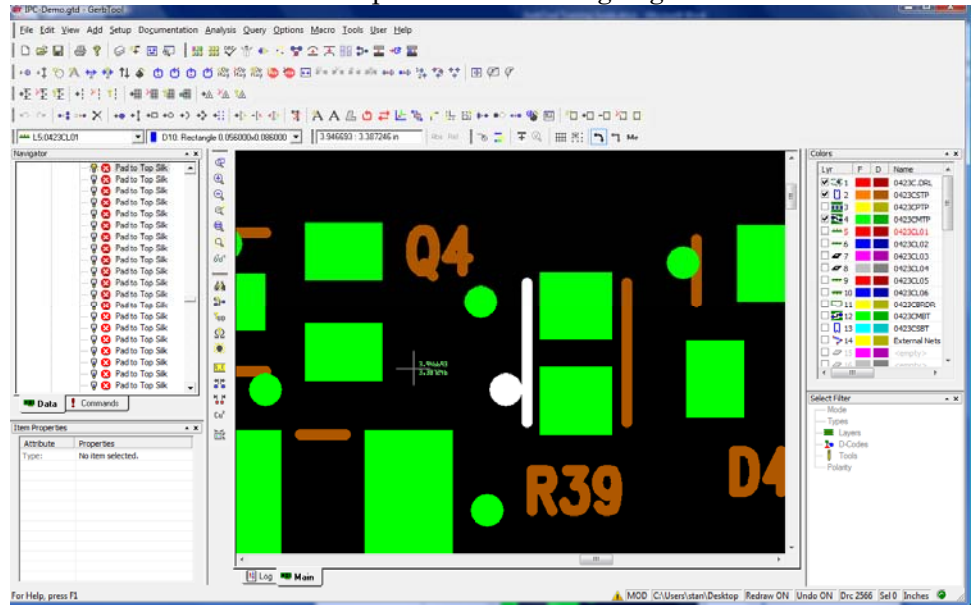
9. Right click the error and choose fix:

10. You may right click on the DRC folder in the tree and highlight all, or fix all to handle these operations en mass.

11. Expand the DFF folder and click the first Acid Trap error to highlight:

12. Right click the error and choose "Fix":

The fix adds copper to smooth the area so that over-etching will not occur due to the acid trap condition.

13. You may right click on the DFF folder in the tree and highlight all, or fix all to handle these operations en mass.

Review questions

1. Why are DRC checks used?

_____
_____
_____

2. What kind of problems do DFF checks find?

_____
_____
_____

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

_____
_____
_____

Panelization

Panelization is the process of creating an array of your PCB for a manufacturing panel. This allows multiple PCBs to be created at one time, optimizing material usage and shortening your manufacturing turnaround time.

Lesson objectives

In this lesson you will do the following:

▸ Use Advanced Panelization to create a finished panel for manufacture.

Estimated completion time

This lesson usually takes 25 minutes to complete.

Advanced Panelization

GerbTool'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.
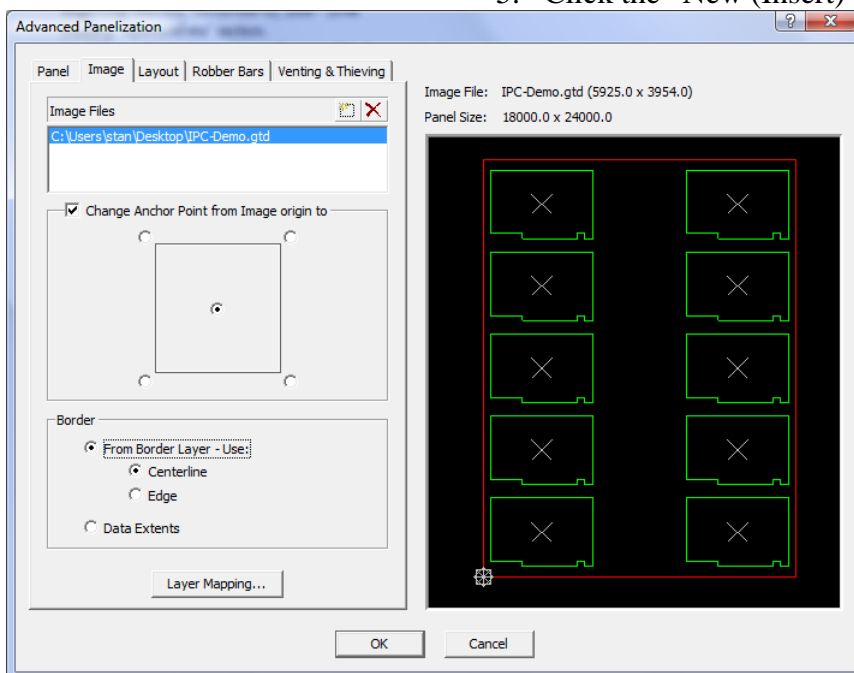
Activity: Perform Advanced Panelization

In order to perform Advanced Panelization, you must have each design already saved as a GerbTool .gtd database (this command does not automatically operate on the data currently loaded in the workspace, unless it is already a panel or panel template). Coupon placement, venting and thieving, pinning holes, and title blocks are supported, as long as those elements have been saved as individual .gtd files.

Essentially, all panel elements are just seen as individual images by GerbTool, that you can add, rotate, and repeat however you wish. As long as you prepare your data properly, panelization is a quick and easy process.

1. Select the File|New command to start with an empty database.
2. Select Tools|Panelize|Advanced.
3. The Panel size should be 18"x24", set it to this value if it is different.
4. Select the "Image" Tab.
5. Click the "New (Insert)" button.
6. Click the "…" button to browse for your "IPC-Demo.gtd" file and select "Open".
7. Place a check mark in the box next to "Change Anchor Point from Image origin to".
8. Select the Center Anchor Point.
9. In the "Border" area, select "From Border Layer - Use" and "Centerline".
10. Click the "New (Insert)" button.
11. Click the "…" button to browse for your "6l2p_250_pin_a.gtd" file and select "Open".

12. Place a check mark in the box next to "Change Anchor Point from Image origin to".
13. Select the Center Anchor Point.
14. In the "Border" area, select "From Border Layer - Use" and "Centerline".
15. Click the "New (Insert)" button.
16. Click the "…" button to browse for your "6l2p_cpn_a.gtd" file and select "Open".
17. Place a check mark in the box next to "Change Anchor Point from Image origin to".
18. Select the Center Anchor Point.
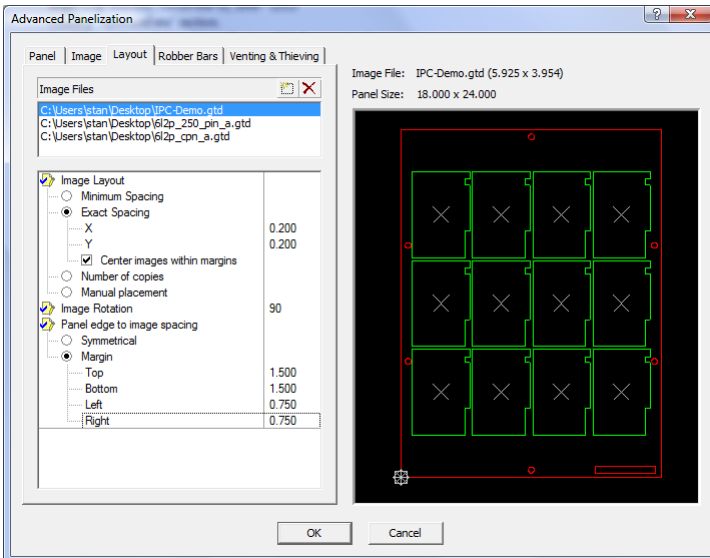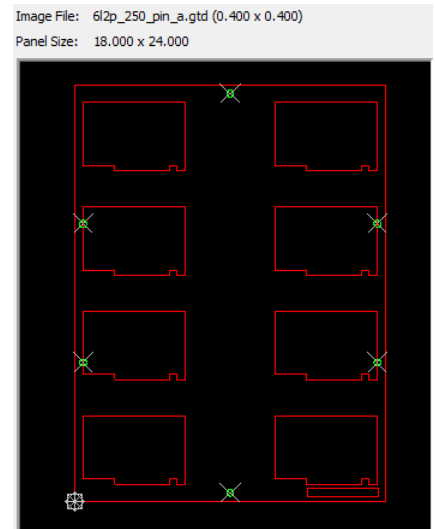19. In the "Border" area, select "From Border Layer - Use" and "Centerline".
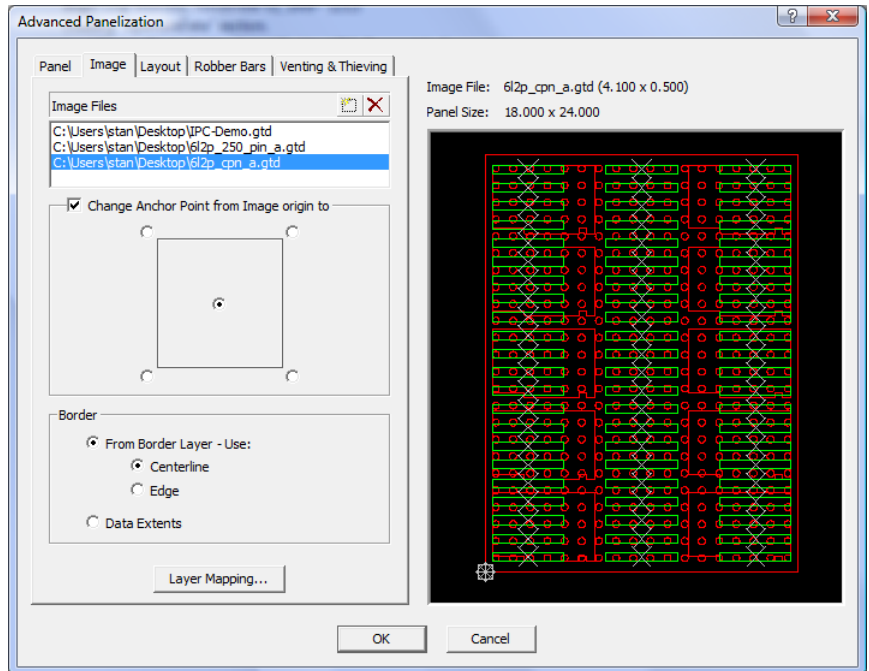
Tip: You do not need to worry about the layer order in the original .gtd files. The first file you bring into the panel establishes an initial panel layer structure. As you add new images to the panel, you can then use the Layer Mapping function to match each image layer to the appropriate panel layer, and create new panel layers as necessary. In the image to the right, all layers are automatically mapped appropriately in this example.
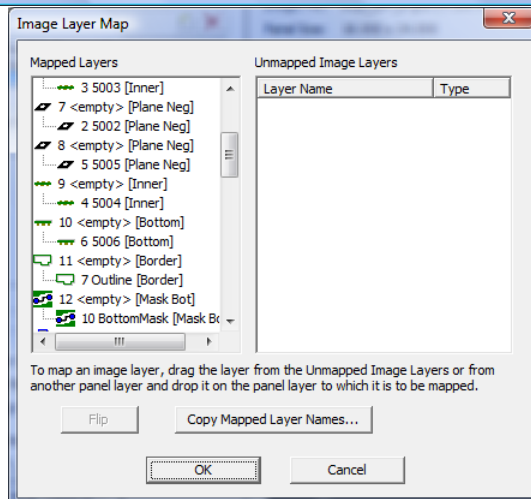
20. Select the "Layout" Tab.
21. With the "6l2p_cpn_a.gtd" file highlighted, select "Manual Placement", and change the "Number of copies" to 1.
22. Use the right mouse button to click on the green image in the preview panel and choose "Move".
23. Enter coordinates of 15.5" for the "X" and .5" for the "Y".
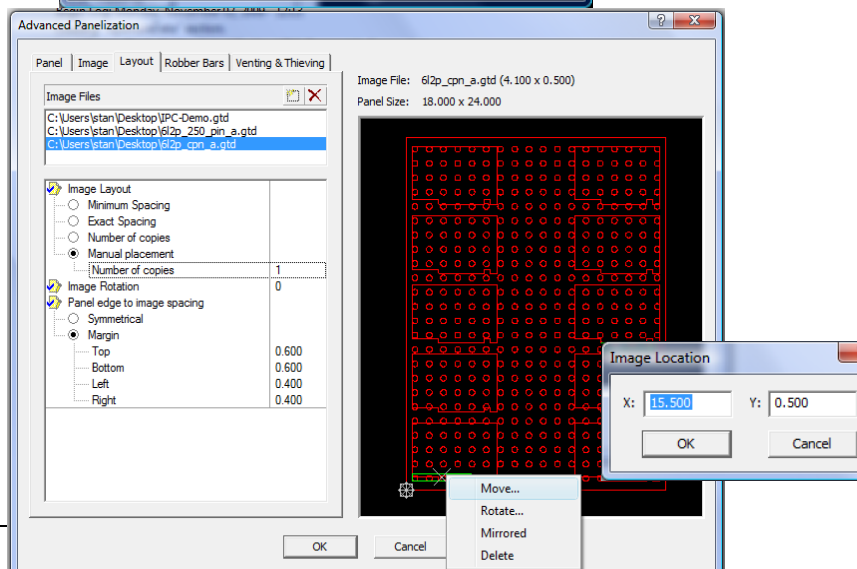
24. Select the "6l2p_250_pin_a.gtd" file in the "Image Files" box.
25. With this file highlighted, select "Manual Placement", and change the "Number of copies" to 6.
26. Use the right mouse button to click on one of the green images in the preview panel and choose "Move".
27. Enter coordinates of 9.0" for the "X" and .5" for the "Y".
28. Use the right mouse button to click on a different one of the green images in the preview panel and choose "Move".
29. Enter coordinates of 9.0" for the "X" and 23.5" for the "Y".
30. Continue selecting and moving each unmoved green image and enter the following coordinates respectively for each move:

    - .5" for the "X" and 8.0" for the "Y".
    - .5" for the "X" and 16.0" for the "Y".
    - 23.5" for the "X" and 8.0" for the "Y".
    - 23.5" for the "X" and 16.0" for the "Y".

31. Select the "IPC-Demo.gtd" file in the "Image Files" box.
32. With this file highlighted, select "Exact Spacing".
33. Change the spacing in the "X" and "Y" to .2".
34. Place a check mark in the box labeled "Center images within margins".
35. Choose "90" for the "Image Rotation".
36. In "Panel edge to image spacing", select margin and enter the following values:

    - Top: 1.5"
    - Bottom: 1.5"
    - Left: .75"
    - Right: .75"

37. Select the "Robber bars" tab.
38. Click the "Define Layer Sets" button.
39. On the "Panel Layer Set Setup" dialog, click "New Layer Set".
40. Name the layer set "Outer" and click "OK".
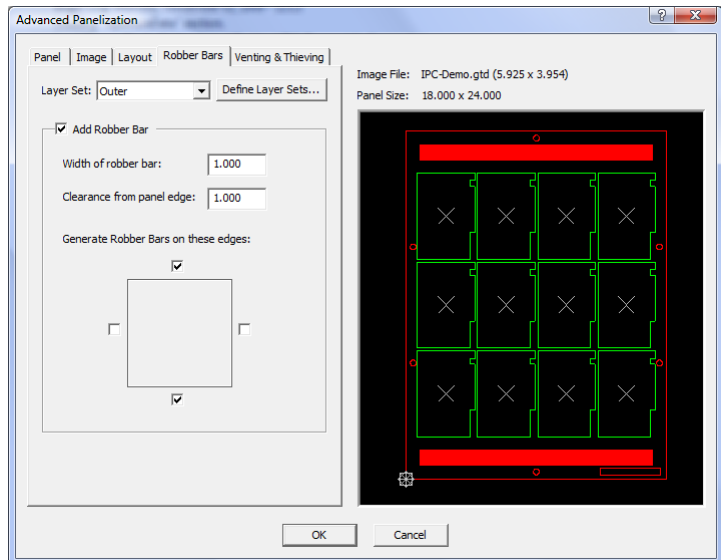41. Drag and drop the Top and Bottom layers onto the "Outer" Layer set.
42. Click "New Layer Set" again.
43. Name the layer set "Inner" and click "OK".

44. Drag and drop the Inner layers onto the "Outer" Layer set.
45. Click "New Layer Set" again.
46. Name the layer set "Plane" and click "OK".
47. Drag and drop the Plane layers onto the "Plane" Layer set.
48. Click "OK" on the "Panel Layer Set Setup" dialog.
49. You should now be on the "Robber Bars" Tab of the "Advanced Panelization" Dialog.
50. Select the "Outer" layer set.
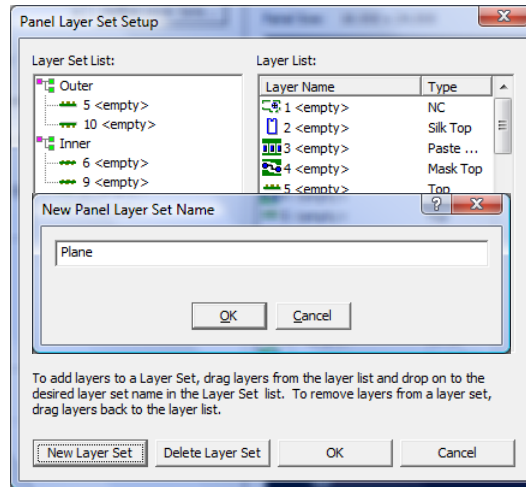
51. Place a check mark in the "Add Robber Bar" box.
52. Enter a value of 1" for "Width of Robber Bar" and "Clearance from Panel Edge".
53. Remove the check marks from the left and right sides in the "Generate Robber Bars on these edges" area.
54. Select the "Venting & Thieving" Tab.
55. Select the "Inner" Layer Set.
56. Choose the "Hatch" radio button.
57. Unselect the box for "Line 3".
58. Change the value for "Line 1" to:
    - Size: .05
    - Spacing: .25
    - Angle: 135
59. Change the value for "Line 2" to:
    - Size: .05
    - Spacing: .25
    - Angle: 45
60. Change "Polarity" to negative.
61. In the "Clearances" section, set the values:
    - From panel edge: 0.0
    - From panel & image data: 0.1
62. Select the "Plane" Layer Set.
63. Choose the "Hatch" radio button.
64. Unselect the box for "Line 3"
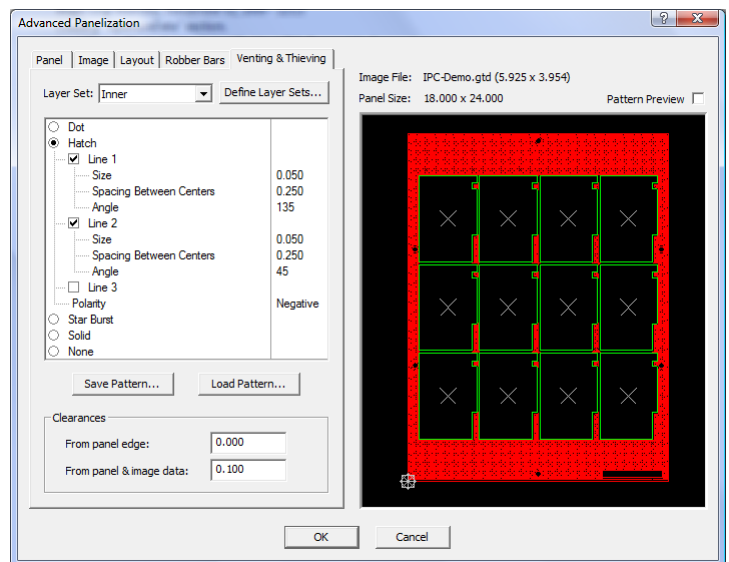
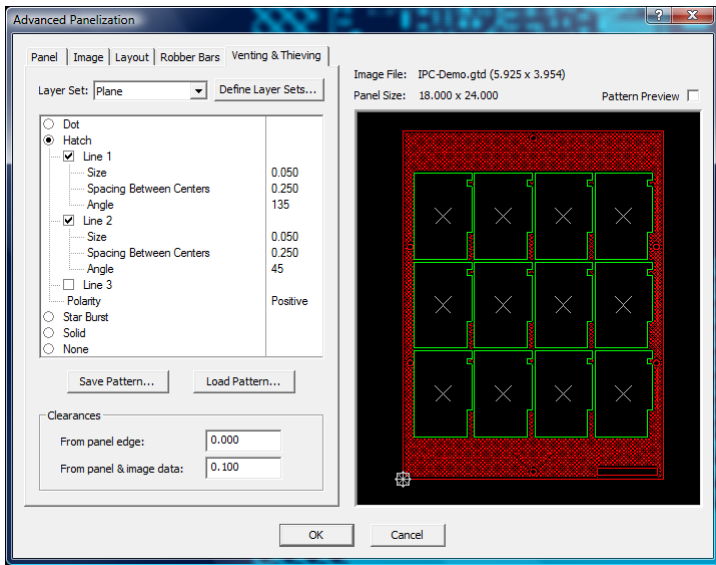65. Change the value for "Line 1" to:
- Size: .05
- Spacing: .25
- Angle: 135

66. Change the value for "Line 2" to:
- Size: .05
- Spacing: .25
- Angle: 45

67. In the "Clearances" section, set the values:
- From panel edge: 0.0
- From panel & image data: 0.1

68. Click "OK" to complete the Advanced Panel setup.

69. Click "OK" when prompted to map unmapped Tools.

70. Your Panel is now complete.

Creating a Panel Template

Many manufacturers have standard panel attributes that are used for several jobs. You can create pre-defined panel "templates" for use on multiple jobs, to streamline the panel creation process. There are several pre-defined panel templates and elements provided in the GerbTool program directory, in the Samples/Template folder.

1. Define a panel as described above, with whatever attributes you wish (size, venting, panel elements, layer setup, etc.).
2. If you have a standard tool table, use the Setup|NC Tools to define your shop's tooling. If you do not define a tool table in your template, the first image that you load into your template upon panelization will determine the tool table definitions.
3. Save the template as a .gtd file.
4. To use the template in the future, use the Advanced Panelization function and load the saved panel template image first.
5. Add whatever images you want to step & repeat, and define their placement, layer mapping, etc. in relationship to your panel template.
6. When you define the layer mapping, if any NC data exists in the images, you are also required to map tool definitions to your template's tool table. Again, if you did not define a tool table in your template, the first image you load into the template will define the tool table. If you load additional images into the table, you must then map their tools based upon the definitions from the first image.
7. When you have completed your panel, save it as a new .gtd file or export it as necessary. By saving your completed panel as a new file, your template file remains untouched, and you have a new panel ready for manufacturing.

Review questions

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

_____
_____
_____

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

_____
_____
_____

3. How can a panel template be created and used?

    _____
    _____
    _____

Contact Information

WISE Software Solutions, Inc.

2700 E. 9th St.
Suite 100
Newberg, OR 97132
USA
Tel: 1 (503) 554-8855
Fax: 1 (503) 554-1220
Support: 1 (503) 554-8855
Web: http://www.wssi.com