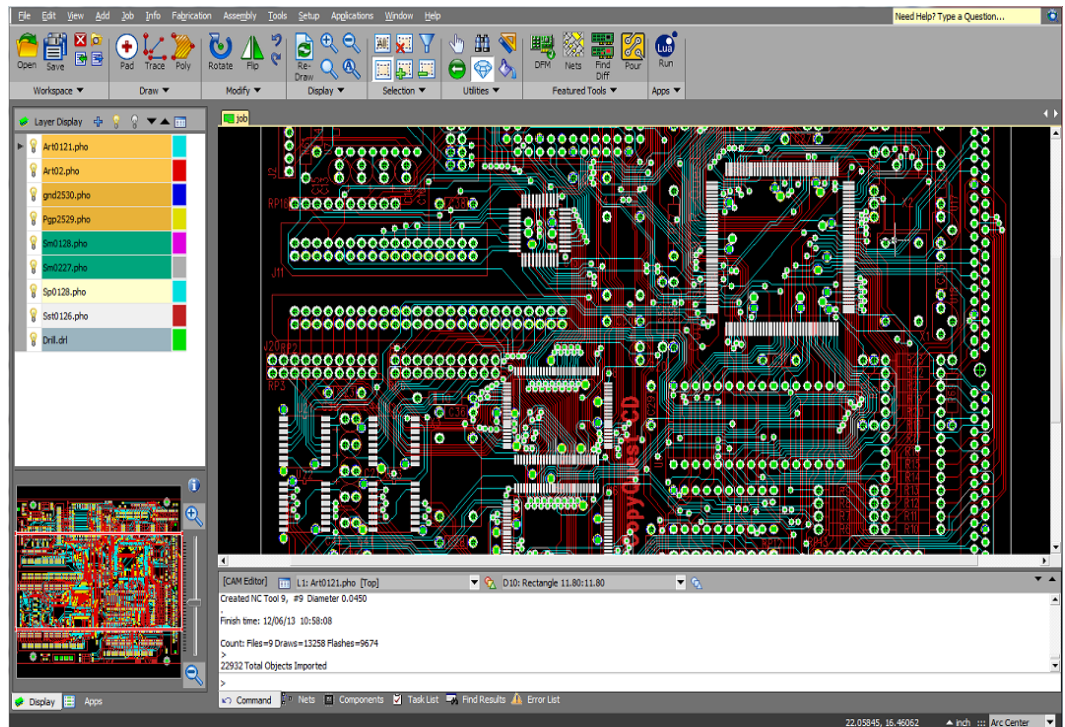


FAB 3000 V7



Version 7

HELP GUIDE

FAB 3000 V7

HELP GUIDE

INTRODUCTION

FAB 3000 serves the needs of a full featured CAM software that assists PCB designers in the analysis and preparation of their designs in Gerber, Drill, & ODB++ formats prior to manufacturing. FAB 3000 goes beyond traditional GUI-based CAM software; providing a set of advanced integrated tools for:

- **Merging PCB Designs to Reduce Manufacturing Costs** by performing all necessary object transformations, Dcode, & drill tool transcoding. Using the ***Merge Wizard***, several PCB designs can be placed on a single panel in order to improve efficiency and reduce manufacturing costs.
- **Intuitively Search for Manufacturing Violations in Gerber Data** prior to sending a PCB design to manufacturing. FAB 3000's custom built DRC/DFM analysis tools (***DFM Checker***) will check your design for all potential manufacturing problems.
- **Generating Netlists from Gerber and Drill Data** by running a comparison against a CAD-generated IPC-D-356 Netlist. By verifying your PCB design electrically, a Netlist comparison will ensure design integrity during each successive stage of PCB manufacturing.
- **Creating X/Y Centroid Files Automatically** for rapid programming of automated surface mount equipment. As this feature is not offered in all CAD packages, FAB 3000 (Professional Version) automatically creates a Part Centroid file from your existing Gerber file.
- **High Resolution Image Exporting with FAB 3000's Film Wizard** allows for the creation of photoplots from Gerber or Postscript files. Output files in either TIFF or Bitmap format and utilize image options such as mirror, invert polarity, trim edges, and data compression. FAB 3000 is the only CAM tool available that directly outputs high-resolution RIPS.
- **Easily Importing DXF Files** using a comprehensive set of control features. Import DXF files by layer, join open/arc boundaries, perform de-embedding, handles solid HATCHes, and more. Additionally, output files to GDS-II format for high-resolution IC artwork, MEMs, and hybrid circuits.

- **Converting Stereolithography (STL) Files** by using FAB 3000 import/export features. Convert your 2D design file into STL format for editing in 3D design tools such as Solidworks, ProEngineer, Rhino3D and more.
- **Creating Custom Scripts for FAB 3000** utilizing the power of an embedded LUA scripting engine. Many CAM tools are limited or outdated in customized application scripting. FAB 3000 supports Open GL (3D-Modeling), IUP (Dialogs), wxWidgets (GUI), SQL, and web server applications.

FAB 3000 V7: WHAT'S NEW

New Features

- ★ Modernized graphical interface with improved usability.
- ★ Embedded LUA scripting engine for custom application design.
- ★ **Generate Error Drawing** - generates marker objects which may be imported into various CAD systems for concurrent verification.
- ★ **Generate CSV Error Report** – Generates a detailed error report in CSV format which may be imported in various spreadsheet applications.
- ★ Auto-Fix invalid polygons during Gerber import.
- ★ Freedom to accept any measurement units while executing commands.

Improvements

- 🔧 Improved and easier to use DFM Check.
- 🔧 Improved rules management for DFM Check.
- 🔧 Improved Gerber file format importation.
- 🔧 Improved ODB++ file format importation.
- 🔧 Improved DXF file importation.
- 🔧 Improved Compare Layers and Compare Jobs functions.
- 🔧 Improved Ariel View Control.
- 🔧 Improved Netlist Comparison.
- 🔧 Over 50 additional improvements in functionality and usability

Bug Fixes

- 🐛 Fixed Alt key commands to operate menu.
- 🐛 Additional 75 minor bugs fixed.

To learn more about FAB 3000's latest features, improvements, and fixes please visit the revision history page at: <http://www.numericalinnovations.com/pages/fab-3000-revision-history>

FAB 3000 HELP GUIDE INTRODUCTION

The FAB 3000 Help menu is divided into 9 sections. Within each section, they are broken down by topic. For example, Section 4 would be the section number of the help document. Section 4.1 would be section 4, topic 1. There are also subtopics; for example, section 4.1.1 (section 4, topic 1, subtopic 1).

FAB 3000 SECTION SUMMARY

Section 1: Getting Started Offers a synopsis on system requirements, installation, and licensing.

Section 2: Working with FAB 3000 Provides an overview of the FAB 3000 GUI, Workspace, & Workbench.

Section 3: FAB 3000 Basics Discusses an overview of control and view manipulation within FAB 3000.

Section 4: FAB 3000 Advanced Features A highlight of FAB 3000's advanced features.

Section 5: FAB3000 Job Editor Menu A complete guide to each drop down menu in the FAB 3000 Job Editor.

Section 6: Import Matrix File Discusses importing Matrix Files with FAB 3000.

Section 7: FAB 3000 Data Format A dictionary of data formats that FAB 3000 can work with.

Section 8: LUA Scripting Engine A guide to FAB 3000's LUA Scripting Engine for custom application development.

Section 9: Purchase & Support Discusses purchasing, licensing, and support options for FAB 3000.

FORUM & VIDEO HELP



Below each topic or subtopic, you'll find forum and video links relevant to the discussion. For additional help on topics beyond this manual, look for this symbol. We'll include links to the FAB 3000 Video library on YouTube as well as direct navigation to relevant FAB 3000 forum topics.

SECTION 1: GETTING STARTED

1.1 Minimum System Requirements

OPERATING SYSTEMS

Windows (32-bit and 64-bit) XP/2003/Vista/7/8

Linux (64-bit) Kernel 2.6

HARDWARE

512+ MB RAM

120+ MB Free Hard Disk Space

Super VGA (1024x768) Display

High Color (16 bit) Graphics Card

4x CD-ROM Drive with 32-bit Drivers

Mouse w/ Center Wheel Button (Intelli-mouse)

Keyboard

Internet Connection Recommended

1.2 FAB 3000 v7.0 Installation

WINDOWS

1. Download FAB 3000 for Windows at <http://www.numericalinnovations.com/pages/download-page>
2. Select FAB 3000 v7 – FREE TRIAL
 - a. For a full featured trial license, fill out the download request form and use a valid email address.
 - b. Upon verification, an email will be sent to your address with download link and trial license key included.
3. Click the download link included in the email address.
4. Unzip **FAB 3000v7.zip** and run the installer **FAB 3000v7.exe**.
5. Follow the install wizard procedures and choose a location for FAB 3000.

LINUX

1. Download FAB 3000 for Linux at <http://www.numericalinnovations.com/pages/download-page>
2. Select FAB 3000 v7 – FREE TRIAL

- a. For a full featured trial license, fill out the download request form and use a valid email address.
 - b. Upon verification, an email will be sent to your address with download link and trial license key included.
3. Click the download link included in the email address.
4. Uncompress the file and select a folder where you have read/write permission.
5. We recommend installing to /opt/FAB 3000.

Registered Customers – Windows

1. Start FAB 3000 and navigate to the **Help > Enter License Key > Activate** menu
2. Enter your activation code(s) below and complete the registration process.

Note: Create a unique password that you will remember. To transfer or reinstall your copy of FAB 3000, you will be asked for your password to recover your activation code.

Registered Customers – Linux

1. Start FAB 3000
2. If a license key dialog box does not appear, go to the **Help > Enter License Key** menu.
3. In the dialog box, select **Get Permanent License Key**.
4. Upon filling out the form, a new key will be sent to you



[Video: FAB 3000 in Linux](#)

1.3 Understanding Licensing & Registration

FAB 3000 offers several license types: **Single User**, **Multi User**, **Numerical Cloud**, and **FLEXnet** licensing.

SINGLE USER - ACTIVATION CODE (INTERNET ACTIVATED):

Enter your activation code and it will be automatically verified over the internet. When you are ready to move to another computer, simply de-Activate from your old computer, and Activate on your new computer.

MULTI USER - ACTIVATION CODE (INTERNET ACTIVATED):

Enter your activation code one-time and registration is performed automatically over the internet. This is then repeated for multiple users. You also have access to an online License Control Panel (LCP) allowing you to manage multiple licenses (activate/deactivate).

NUMERICAL CLOUD - ACTIVATION CODE (INTERNET ACTIVATED):

Enter your activation code and it will be automatically verified over the internet. When you are ready to move to another computer, simply de-Activate from your old computer, and Activate on your new computer.

ACTIVATION PROCEDURE FOR SINGLE/MULTI/NUMERICAL CLOUD (INTERNET ACTIVATED)

1. Start FAB 3000.
2. Go to FAB 3000 menu **Help > Enter License Key > Activate**
3. Paste your activation code into the edit box (use key CTRL + V).
4. Create a password that is easy to remember (at least 4 characters).
5. Enter your email address.
6. Select "Next" and follow the activation prompts.

HOW FLEXNET WORKS

FAB 3000 uses FLEXnet Publisher for license management. You can use either a valid license key file or physically install the file (*.lic). If no license is detected, FAB 3000 will start in unregistered mode.

Single User - FLEXnet License

A hardware locked license where the software is locked per machine (no internet activation required). A FLEXnet "Node-Locked" license is also referred to as a single user license.

Copy/Paste a FAB 3000 License Key Via FLEXnet:

1. Start FAB 3000
2. In the Help menu, select **Enter License Key > Enter FLEXnet Key**
3. Your **Product Name** and **Host ID** will already be filled in with relevant registration information.
4. In the window titled **License Key**, paste your license key information.

Your license key information will contain the following format:

```
FEATURE FAB 3000 7.0 numinno 30-dec-2013 uncounted HOSTID=ANY \
ISSUED=30-nov-2013 \
NOTICE=user@demo_license_issue_address.com;FAB 3000v7 TS_OK \
SIGN="1B42 076D 87E3 EAF7 F258 F0DA 3175 654D 4E63 7439 B7C4 \
345A 4348 BD6F 62C6 00BA 17F6 A75E D086 6B1E DE6B 7225 E09C \
86D6 13E2 A7C7 5852 C4AB D8D0 3FBB"
```

FEATURE: This contains information of your product, version number, and licensed duration. For a single user license, HOSTID=ANY.

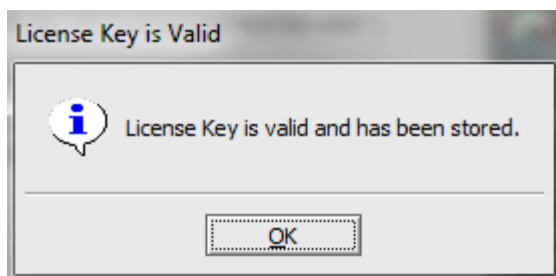
ISSUED: This is the date of license issue.

NOTICE: The address listed is the email address from which the license was sent to you.

SIGN: This includes the string of key characters which make up the license key. Quotation marks should be included. Backslashes are also included.

Notes:

- c. Copying and pasting may produce random characters that replace certain text in the license. Make sure to replace these characters with the original text included in the license.
 - d. Copying and pasting may produce inaccurate spacing of your license characters. Make sure to duplicate the spacing above for proper activation.
 - e. Your operating system's clock must be set to the exact date. If the date is not correct, the license key file will not activate.
5. Once the license information is properly pasted and formatted, select **Activate**.
 6. If your key is valid, you'll receive the following message:



In the case your license key is not valid, you will receive an error message.

Check to make sure your key is properly formatted and there are no phantom characters or spaces. For additional help, contact sales@numericalinnovations.com.

ALTERNATIVE OPTION: INSTALLING A LICENSE KEY FILE

You can also install your FLEXnet license key file (*.lic) by copying the .lic file to the following directory:

Windows

C:\Numerical Innovations\FAB 3000V7

Linux

/opt/Numerical/FAB 3000

FAB 3000 will automatically check this directory for a license key upon startup.

FLEXNET PUBLISHER LICENSING FOR ENTERPRISE ENVIRONMENTS

FLEXnet Publisher offers easier management for concurrent licensing environments. With FLEXnet, multiple licenses can be deployed from a central server (FLEXnet Server) located onsite. The maximum number of seats (concurrent users) and access location of the server computer will be indicated by the invoice issued from Numerical Innovations. Numerical Innovations will provide a Network Bundle Package which contains all necessary files, utilities, and instructions. There are two types of licenses under this program:

Network/Floating - FLEXnet (Concurrent):

This option is ideal for companies that want to manage the number of engineers using FAB 3000 but do not want to manage the individual users of the product.

Enterprise/Site - FLEXnet (Unlimited):

The ultimate freedom in software licensing because it grants your company the right to install UNLIMITED seats of our software onto your workstations, networks, laptops, or home computers.

INSTALLING NETWORK & ENTERPRISE LICENSES VIA FLEXNET

Supported Platforms

Microsoft Windows XP/2003/Vista/7 (32-bit)

Microsoft Windows XP/2003/Vista/7 (64-bit)

Linux 2.4 Kernel or Higher (32-bit)

Linux 2.4 Kernel or Higher (64-bit)

Getting FLEXnet Server

Upon purchase, the Network Bundle Package will be sent to the appropriate administrator. This package contains the following files:

- c. network.txt (document)
- d. Imgrd (Server Daemon)
- e. nummino (Vendor Daemon)
- f. Imtools, Imutils (FLEXnet supplied utilities)
- g. FLEXnet_licensing_end_user_guide.pdf (FLEXnet Publisher instructions)
- h. vcrcdist_x86.exe (for Windows 32-bit only)

1. Extract the files above to a temporary folder.
2. Copy all files to a designated folder of your choice on the FLEXnet server. This folder will also be known as the FLEXnet server.

Note: For Windows platforms, the default copy location is C:/FLEXnet.

For Unix/Linux platforms, you will create your own folder and select a location.

If you have an existing FLEXnet installation, you can choose whether or not to overwrite the old files.

GENERATE YOUR LICENSE FILE (*.LIC)

3. To generate your license file, the following is required: **Platform, Hostid, & Hostname**.
4. Choose your FLEXnet server platform: Windows-32, Windows-64, Linux-32, or Linux-64.
5. Determine your **Hostid** by using the following procedure:
 - a. Open a command line window and navigate to your designated FLEXnet folder.
 - b. Type the following command: **Imutil Imhostid**
 - c. This will return one or more options for hostid. Choose the first one that appears in the list.
6. Enter the hostname. To find the hostname of the FLEXnet server, run **hostname** from a command line on the FLEXnet server. Save this information as it will be required for step 14.
7. Complete the Numerical Innovations "Request License Form" at http://www.numericalinnovations.com/license_request.html
8. After completing the form, the license key will be emailed to you in one business day.

SAVING AND CONFIGURING YOUR LICENSE FILE (*.LIC)

*Upon receiving the license file (**license.lic**), perform the following:*

9. Save license.lic to the FLEXnet folder.

10. In the FLEXnet folder, open **license.lic** in a text editor.
11. In text editor, edit the vendor path to match the path for the FLEXnet folder. For example:

In Windows

VENDOR numinno C:\FLEXnet\numinno.exe

C:\FLEXnet is the path to the FLEXnet folder

numinno.exe is the daemon that authorizes each license installation with Numerical Innovations

In UNIX/Linux

The path should be the exact same path that you chose in **Step 2**.

12. Save and close **license.lic** within the text editor.

STARTING THE FLEXNET SERVER AND INSTALLING A CLIENT

13. To start FLEXnet server, open a command line on the FLEXnet server while in the FLEXnet folder. Type the following: **lmgrd -c license.lic**
14. The environment variables must be set as follows:

In Windows

- a. Select **Control Panel > System > Advanced Tab > Environment Variables**. A dialog box will appear.
- b. Under **System Variables**, select **New**. A new dialog box will appear.
- c. In the top entry field, enter **LM_LICENSE_FILE**.
- d. At the bottom field, enter **port@hostname**.
- e. For default port, enter **27000**.
- f. Enter the FLEXnet server hostname (see step 6). This may or may not be necessary.
- g. Click **OK** and close all dialog boxes.

In UNIX/Linux

- a. Type **LM_LICENSE_FILE=port@hostname** in the command line.
- b. The default port is **27000**.
- c. Enter the FLEXnet server hostname (see step 6).

INSTALLING TO A CLIENT

14. On each licensed local client machine, install FAB 3000 by following the instructions included with the CD (or download file).
15. Copy the license file (*.lic) in the FAB 3000 installation folder. Typically this would be labeled as:
C:\Program Files\Numerical Innovations\ ... (whatever directory you installed the software into).

Congratulations! You may now start the FAB 3000 product on each licensed client machine.

Note: For Windows installations, the error message “**Missing MSVCR80.dll**” (or similar) may or may not occur when starting the vendor daemon **numinno.exe**. If it does occur, please install the Microsoft Redistribution Utility known as: **vcredist_x86.exe**. This utility will repair any missing .dll files that you require.



[Forum: How do I move my license to another computer?](#)

[Forum: How do I install my license file?](#)

[Forum: How do I install a Network/Floating License?](#)

1.4 Launching FAB 3000

WINDOWS

1. In the **Start** menu, select **All Programs**.
2. Search for **Numerical Innovations**.
3. Select **FAB 3000 v7**.

The FAB 3000 startup screen will appear.

LINUX

1. Run the shell file: **FAB 3000v7.sh**
2. The script will automatically assign FAB 3000 library paths using LD_LIBRARY_PATH.

The FAB 3000 startup screen will appear.

If FAB 3000 does not startup, it is most likely due to the LD Linker not able to locate the required FAB 3000 shared libraries (*.so). In this case, manually assign the libraries by performing the following:

- a. Execute the Shell/Terminal program.
- b. Type the following commands:

```
LD_LIBRARY_PATH=/opt/FAB 3000/lib
export LD_LIBRARY_PATH
./opt/FAB 3000/FAB 3000
```

The example above assumes FAB 3000 was placed in folder/opt/FAB 3000

If FAB 3000 still does not startup, then your Linux distribution may be missing certain required packages/libraries.

To verify if you have the included packages/libraries installed perform the following:

- a. Execute the Shell/Terminal program.
- b. Type the following commands:

```
ldd /opt/FAB 3000/FAB 3000
```

The example above assumes FAB 3000 was placed in folder/opt/FAB 3000

- c. Review the output of this command. If any library files are missing they will be referenced here.
- d. Install any missing packages/libraries required to install FAB 3000.

For additional installation help, contact support a

<http://www.numericalinnovations.com/pages/customer-support>

In your support inquiry, make sure to include the output from the command ldd.



[Video: FAB 3000 Getting Started Part 1](#)

[Video: FAB 3000 Getting Started Part 2](#)

[Video: FAB 3000 Getting Started Part 3](#)

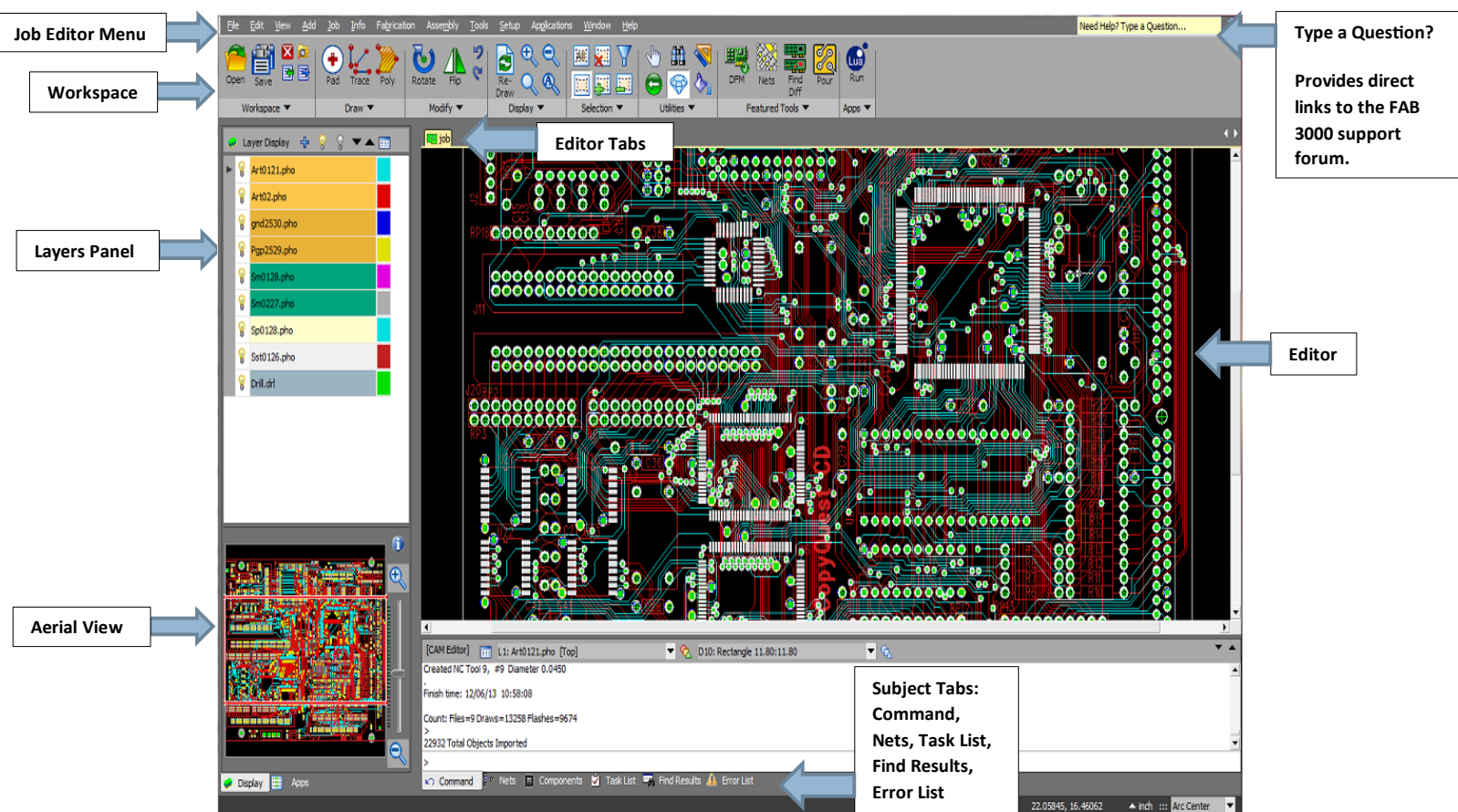
[Video: FAB 3000 Getting Started Part 4](#)

[Video: FAB 3000 Getting Started Part 5](#)

[Video: FAB 3000 Getting Started Part 6](#)

SECTION 2: WORKING WITH FAB 3000

2.1 FAB 3000 GUI Overview

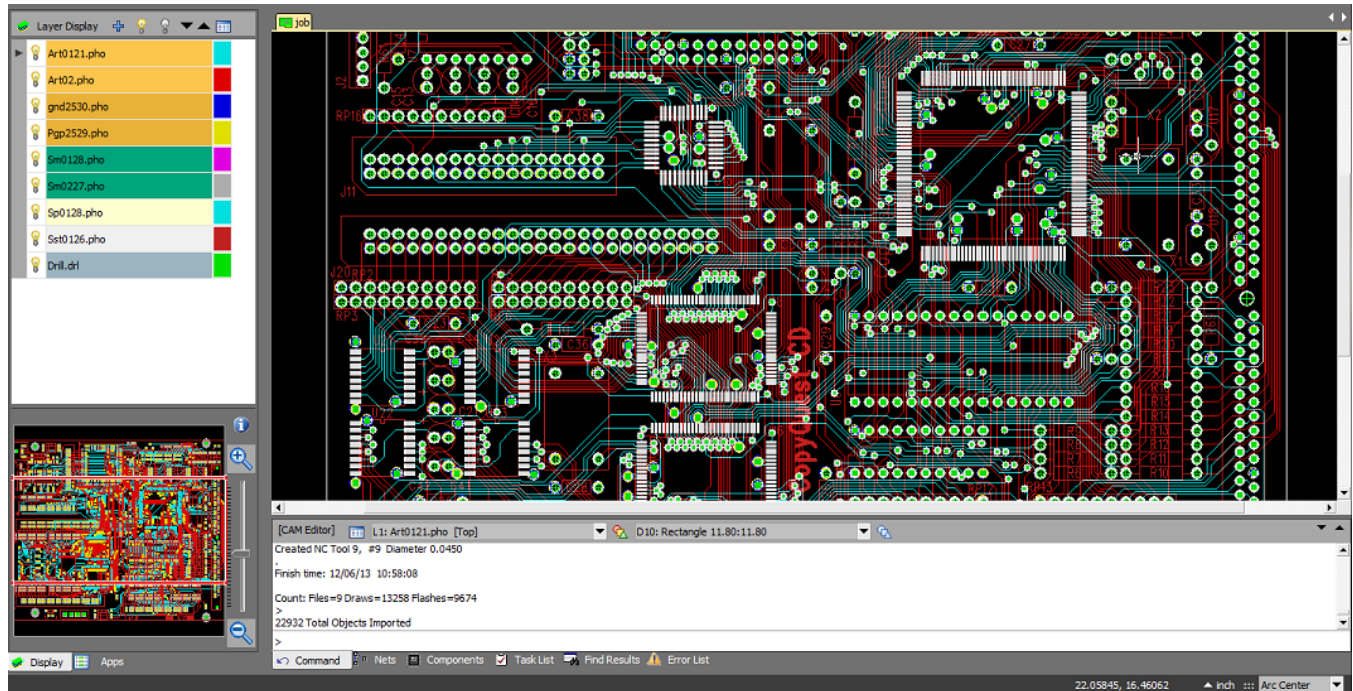


The FAB 3000 GUI provides robust centralized operation for the designer or manufacturer. The graphical user interface aims for maximum flexibility by allowing users to instinctively select, drag, and edit objects. Multiple designs can be separated into tabbed windows while enhanced hierarchy control lets you work with individual layers. It is specifically targeted for the designers of PCB's, MEMs, Microwave and RF circuits, chemically milled parts and optical components; and to photo mask manufacturers.

2.2 Workspace & Workbench

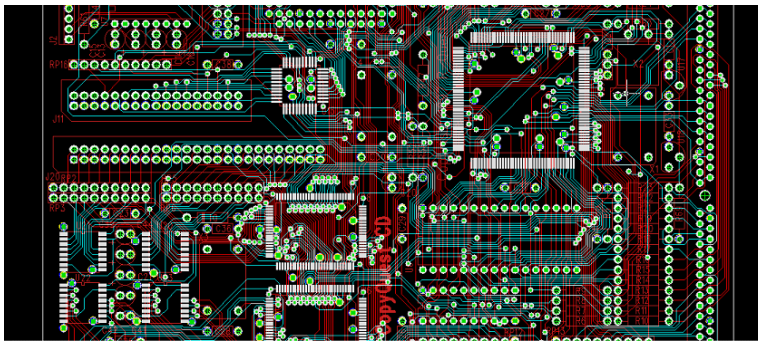
FAB 3000 is divided into two primary components for operation and control: **Workbench** and **Workspace**.

WORKBENCH



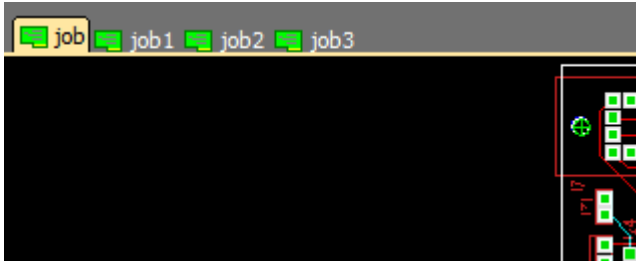
Think of workbench as the central cockpit for controlling FAB 3000 workspaces. With workbench, you can navigate resources and you can view and edit the content and properties of these resources using the workbench menu bars and tool bars. The workbench contains an Editor, Editor Tabs, and 2 Views: Layer View, and Aerial View.

a. Editor

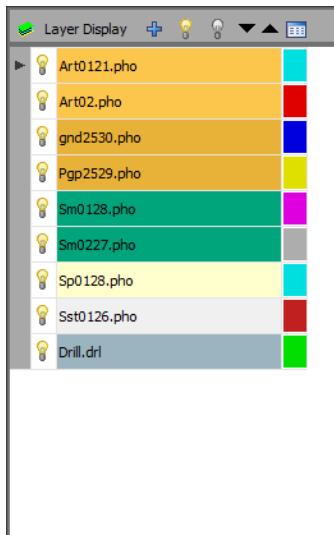


Editor allows the user to open, browse, edit, and save various types of data using the Workbench menu bar. Tools within the editor are first opened, data is modified, saved, and then closed. Multiple instances of editors may exist within a Workbench. They are individually tabbed in the editor area and can be activated by clicking the tab. These are known as **Editor Tabs**. When an editor tab is active, the Workbench menu bar and toolbar will only operate for that particular editor.

b. Editor Tabs



Multiple editor tabs will appear above the editor window. Only one editor can be active at any one time. Clicking an individual editor tab designates that editor as active. Any editing, browsing, saving, etc. using the workspace menu bar or toolbar only affect the active editor tab.



c. Layer View

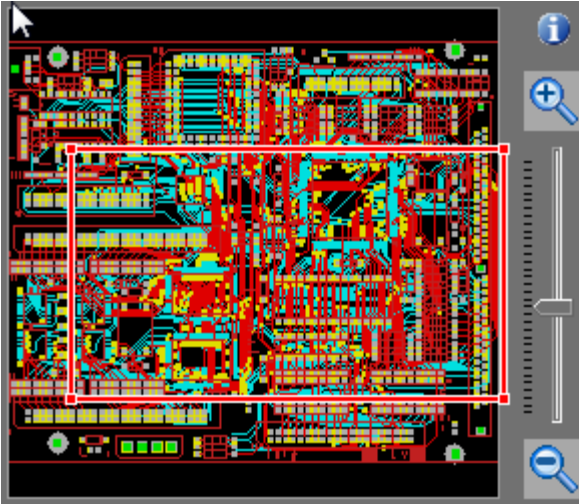
Layer view organizes your design into an interactive table that controls the following attributes: Visibility, Draw Color, Flash Color, and Name. Layer view allows you to organize and control individual objects within your design.

Visibility: Determine if Objects on the layer are visible. If the layer is visible, the icon used is 'bulb on'. If the layer is not visible, the icon used is 'bulb off'.

Name: Name of the layer. (Note: Includes the layer number for reference)

Draw Color: Objects on the associated layer will display that color for all draws (non-flashes)

Flash Color: Objects on the associated layer will display that color for all Flashes.

d. **Aerial View**

The Aerial View is a visual perspective tool providing an overhead view of the active editor window. It provides an excellent reference in determining the general location of the current editor view port within a design.

Zoom Percentage: Ratio of the editor's current view port size compared to the entire design. Use zoom control to increase or decrease your view.

Interactive Panning: Drag the current view port window box with your mouse to orient the view location of the design.

Alternatively, use the aerial widget located inside the aerial view window. Place the cursor just inside the aerial view window, and drag the mouse.

Underneath Layer View, there are two display tabs: **Display & Apps**.

Display: Default Ariel view display of your design.

Apps: Displays a complete list of scripted applications available in FAB 3000.

WORKSPACE

A workspace consists of the Jobs, panels, custom apertures, folders, files that you create within the Workbench and are all stored under a single directory that represents your workspace. A workspace must have at least one or more libraries. Each library may contain one or more resources.

Resources are a collective term for the designs, files, tables, etc. that exist in the Workspace. The most common resources in FAB 3000 are: **Jobs, Inserts, Panels, and Custom Apertures**.

Job: By default, a Job is the base resource of FAB 3000. All data imported and 90% of editing are handled at the Job level. Essentially you can think of a Job as a single design with its own unique layers, Dcodes, and other attributes.

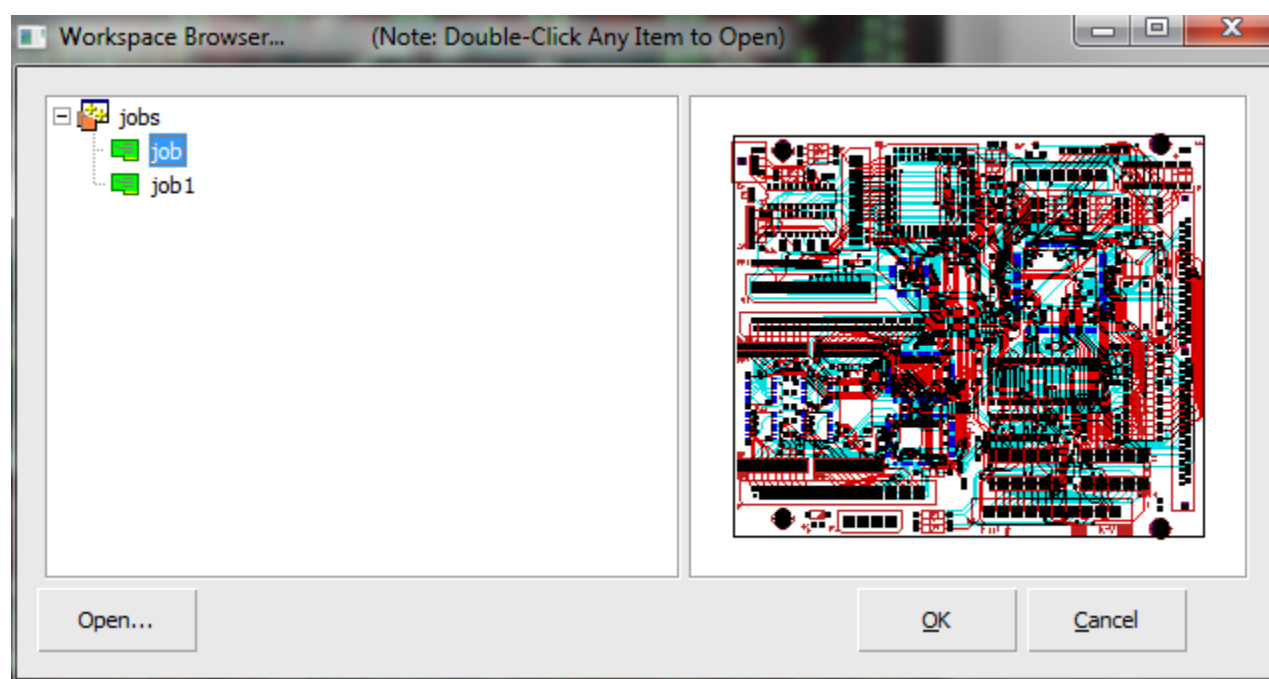
Insert: An Insert represents the inclusion of one Job as a part of the contents of another. The Job containing the Insert is the parent Job, and the Job that is included is the master Job of the Insert. Inserts can create hierarchical designs. The master of an Insert can contain instances of other masters. This hierarchy can continue for as many levels as needed to express a complete design.

Inserts make it easy for the user to combine and visualize the placements of different Jobs resulting in optimum configurations (like pieces of a puzzle) which will save you time & money on manufacturing. Inserts can be transformed (i.e. rotate, mirror, move, etc.) and can be nested into a hierarchy of Jobs/panels.

Panel: The Panel resource is used to easily create and visualize a completed panel used for PCB manufacturing. Panels are essentially templates which generally use one master Insert (which is step & repeated), and can include any number of Test Coupons, Pinning Holes, Fiducial's, etc.

Custom Aperture: The Custom Aperture resource is used to edit and visualize the elements of any custom aperture.

WORKSPACE BROWSER



Workspace Browser is used to navigate resources (from the active workspace) in a hierarchy of libraries, designs, folders and files. From here, you can open resources for editing or select resources for operations such as exporting.

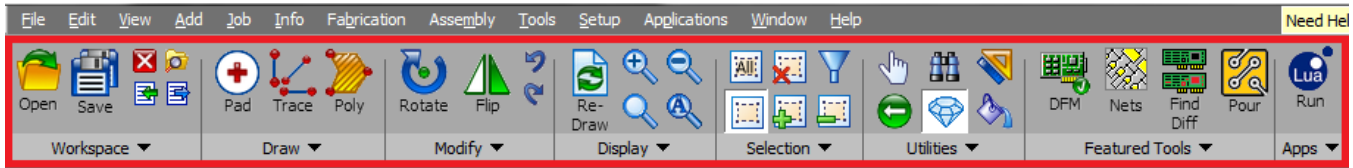
Library

Libraries are the outer containers in the Workspace hierarchy. There are two types of libraries available: **Layout** and **Document**. This has been put here as reference, and rarely will a user need to access or modify libraries directly.

Layout Library: Used for all graphical data. Layout libraries only contain designs as child items.

Document Library: Contains all non-graphical files and folders (i.e. text files, etc.)

2.3 Workspace Menus



Located directly above the editor and layer display, the workspace menus provide a graphical illustration of functions that are also listed throughout the Job Editor menus (File, Edit, View, Add, etc.). Below is a summary of each graphical menu. In Section 4: FAB 3000 Job Editor Menu, we'll discuss every menu function in detail.



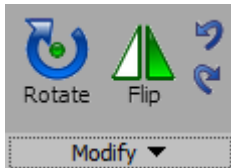
Workspace Menu

The workspace menu provides immediate functions for Opening, Saving, and Closing workspaces. It also includes access to the **Workspace Browser, Import and Export functions.**



Draw Menu

With the draw menu, access important CAD drawing functions such as **Pads, Traces, and Poly**. The draw menu pull down offers access to additional shapes such as **Circle/Arc, Rectangle, Ellipse, Wide Path, Text, Multi-Line Text.**



Modify Menu

The modify menu provides for editing of existing objects. This includes **Rotate, Flip, and Undo/Redo**. Using the modify menu pull down, access additional edits such as **Delete, Move, Copy, Scale, Stretch, and Split**. You can also view an object's properties by selecting the object and choosing **Modify > Properties** from the graphical pull down.



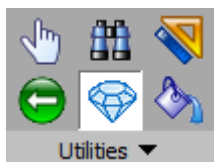
Display Menu

The display menu provides viewing functions within your design. The most prominent are **Re-Draw, Zoom In/Out, Zoom Window, and Zoom All**. Using the display menu pull down, access additional views such as **Pan, Object view filters, and Layer Display**.



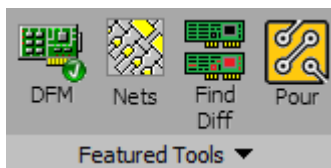
Selection Menu

The selection menu provides selection functions for the objects and shapes within the editor. This includes **Select All, Standard Selection, Add Selection, Subtract Selection, Selection Filter, and Clear Selection**. The selection drop down menu adds additional selection manipulations such as **Select Display, Select Window, Select Crossing Polygon, Select Window Polygon, and Select Fence**.



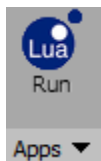
Utilities Menu

The utilities menu provides an array of various features to use within your design. This includes **Query, Find, Measure, Add Rulers, Backside View, Translucent View, and Fill**. Additionally, all reporting can be accessed from the utilities drop down menu (**Aperture, NC Tools, Dcode, Drill, Composite, & Nets Reports**).



Featured Tools Menu

The featured tools menu includes all of FAB 3000's most significant tools for use on your design. This includes **DFM Checker, Netlist Tools, Compare Tools, & Copper Pour Tool**. Using the featured tools dropdown menu reveals additional FAB 3000 special features such as **Convert To, Join Wizard, Film Wizard, Create Fabrication Drawing, Calculate Copper Area, and Build Component**.



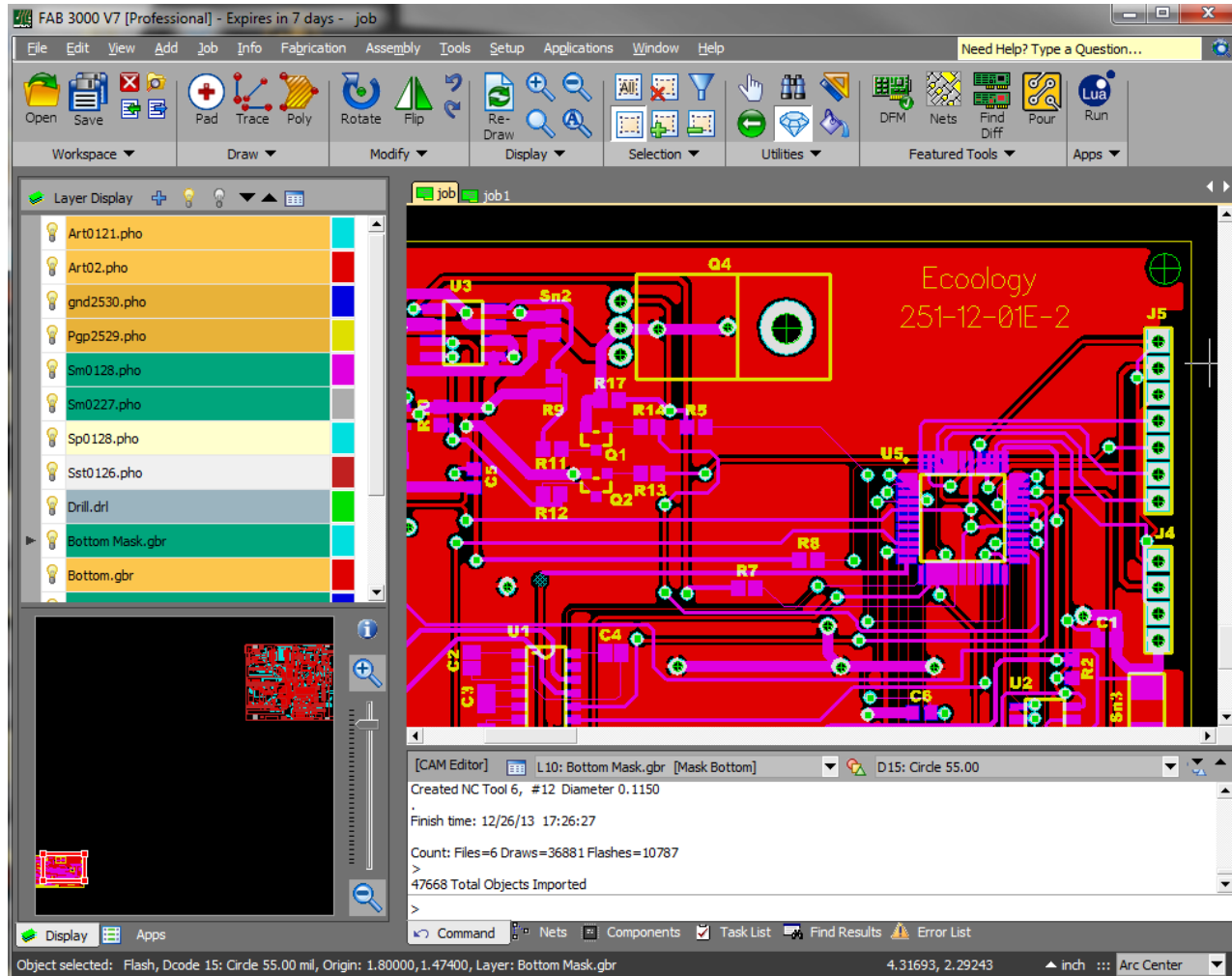
LUA Apps Menu

The LUA apps menu button lets you choose and run custom LUA scripting apps. Using the Apps pull down menu will allow you to create a new LUA app or open an existing LUA app.



[Video: PCB Backside View and Flip using FAB 3000](#)

SECTION 3: FAB 3000 BASICS



3.1 Introduction

FAB 3000 extends beyond typical CAM software for PCB design and manufacturing. It offers a suite of tools to create, analyze, and prepare industry standard files for manufacturing. Integrated with FAB 3000 are a number of features aimed at increasing productivity, enhancing design control, and reducing production costs.

3.2 View & Selection Control

3.2.1 Mouse Control

Right Click: Right click over the editor menu to provide a list of commands for navigating, selecting, and editing your design. For more information on Right Click functions, see **Right Click Menu**.

Left Click: Left click to select a command or left click in editor to select an object. When left double clicking an object inside the editor, the properties of that object will be shown.

Dynamic Zoom: To zoom in and out on a particular region of the editor, use the middle mouse wheel. Scrolling forward will automatically zoom in. Scrolling backward will automatically zoom out.

Dynamic Pan: Pan or move around the editor by clicking and holding the middle mouse wheel down. At the same time, drag across the region of the editor to travel in any direction.

Zoom Window: There are several ways to enable Zoom Window:

1. **Select View > Zoom > Window.**
2. Right click in the editor display menu then select **Zoom > Window.**

By left clicking and dragging anywhere around the region of the editor, a zoom window is enabled over the selected region. Alternatively, by simply left clicking in the editor FAB 3000 will automatically zoom by 2x.

Select an Object: As the mouse hovers over an object in the editor it will add a highlight surrounding the object. Left click to select the object. The object will stay highlighted.

Select Multiple Objects: As the mouse hover over an object in the editor it will add a highlight surround the object. Hold down the **SHIFT** or **CTRL** key and left click to select multiple objects at once.

With your selection(s) active (highlighted), they are now part of the **Active Selection Set**.

Leave or Reset Function or Selection Set: Use the **ESC** key at any time or right click inside the editor and select **Cancel**.

Selection Mode Using the Workspace Menu



Standard Selection Mode: This is the default setting for FAB 3000 upon startup. This selects an individual object. Each time you select an object, all previous objects will be unselected.

Add Selection Mode: This button allows you to add and highlight multiple objects at once; similar to using the SHIFT or CTRL key to add a group of objects to the Active Selection Set.

Sub Selection Mode: This button allows you to subtract highlighted objects from the Active Selection Set.

Clear Selection Mode: This button will subtract all objects from an active selection set. Alternatively, you can use the **ESC** key or **Right Click** on the mouse and select **Cancel**.

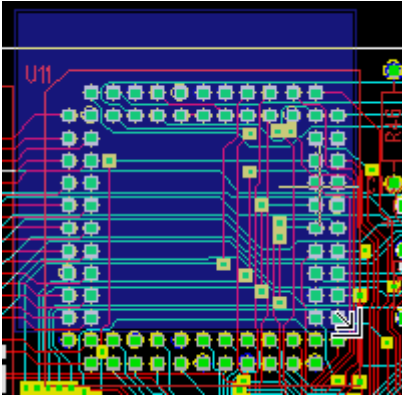
Selection Filter: The selection filter box helps control the content of an active object by allowing you to include or exclude Dcodes, Layers, Composite Levels, NC Tools, Polygons, or Text from being selected. Simply highlight an object then choose the Filter icon. This will open the Selection Filter dialog box for that layer.



[Forum: Crossing Window Selection](#)

USING SELECTION WINDOWS

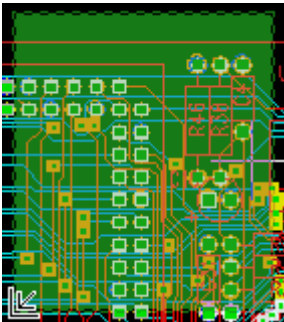
There are several selection window options for selecting multiple objects over a specified area of your design. They include:



Select Window (Blue Box Highlight): The selection window allows you to click and drag over a specific area of a group of objects in order to highlight what is required. Any object contained inside this window will be selected only. **Left Click** anywhere within the editor and then drag across the editor to the **Right**. The selection box is highlighted in blue.

Dynamic Selection Window (Blue Box Highlight): Using dynamic selection, **Left Click** on an empty area in the editor and drag across to the Right to cover any selection area you require. The selection box is highlighted in blue.

Select Crossing Window: Any objects contained inside or intersecting this window will be selected. To activate, go to **View > Selection > Crossing Window**. **Left Click** on an empty area in the editor to define the first point, then **Left Click** again to define the next point.

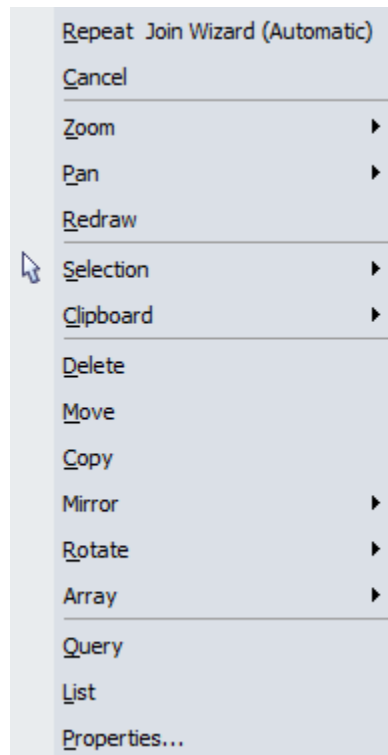


Dynamic Selection Crossing Window (Green Box Highlight): Any object contained inside or intersecting this window will be selected. **Left Click** on any area in the editor and drag over the selection area you require to the **Left**. The crossing window will be highlighted in green.

Quick Query Selection: Move the mouse over any object and **Left Click**. A status window will appear with information about the object. Alternatively, the same information will also appear at the status bar in the bottom of the screen.

Note: When selecting polygons and traces, make sure to select their Edges.

3.2.2 Right Click Menu

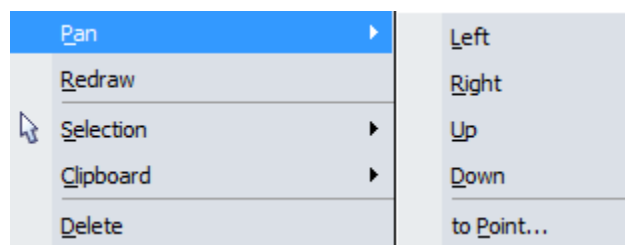
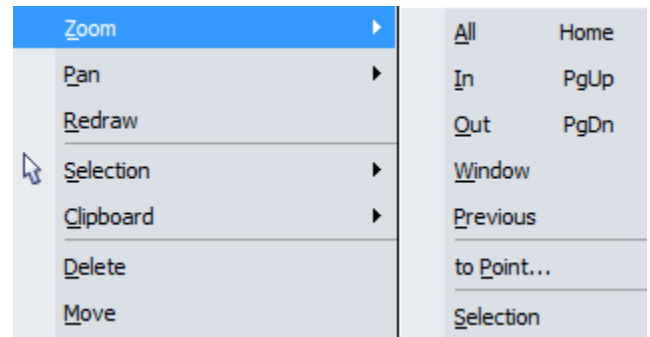


Right clicking the mouse while in editor is an alternate way to access menu options related to editor mode.

Repeat: Repeats the last action taken.

Cancel: Cancels the current action. For example, “canceling an object selection”.

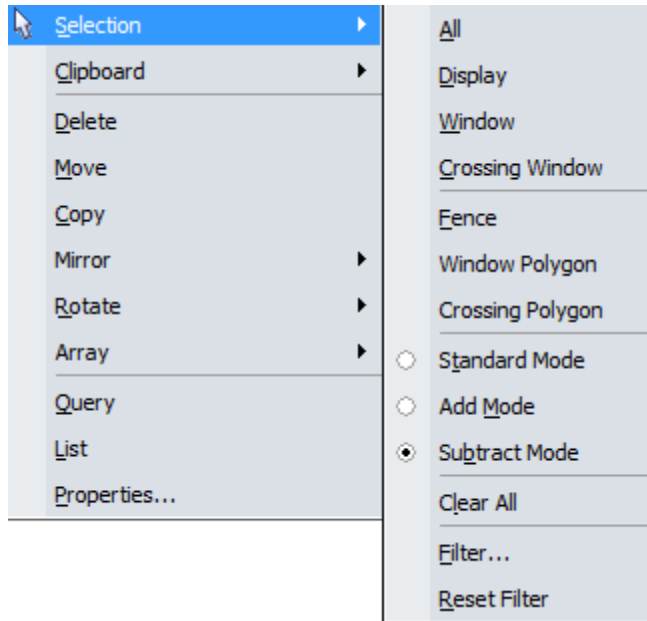
Zoom: Several zoom functions available such as Zoom All (zooms back to entire design), Zoom In/Zoom Out, Zoom Window, Zoom Previous, Zoom to Point, and Zoom to Selection.



Pan: Pan function includes Left, Right, Up, Down, and To Point.

Redraw: Redraws the object.

3.2.3 Right Click - Selection Options



All: Selects and highlights all objects within the editor.

Display: Selects all objects within the Editor.

Window: The selection window allows you to click and drag over a specific area of a group of objects in order to highlight what is required. Objects contained inside this area will be selected only.

Crossing Window: Specify selection by dragging mouse from corner to corner over area of objects. Objects contained inside or intersecting this window will be selected.

Fence: Create a custom selection fence around the area of objects you wish to define as selected.

Window Polygon: Create custom polygon around the area of objects you wish to define as selected.

Crossing Polygon: Specify selection by dragging mouse from corner to corner over area of objects to produce a polygon window. Objects contained inside or intersecting this polygon will be selected.

Standard Mode: This is the default setting for FAB 3000 upon startup. This selects an individual object. Each time you select an object, all previous objects will be unselected.

Add Mode: Allows you to add and highlight multiple objects at once; similar to using the SHIFT or CTRL key to add a group of objects to the Active Selection Set.

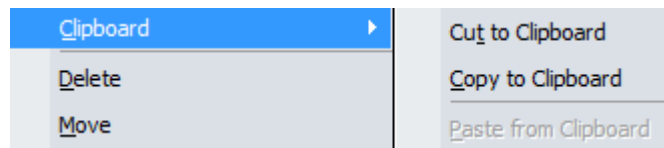
Sub Mode: Allows you to subtract highlighted objects from the Active Selection Set.

Clear All: Will subtract all objects from an active selection set. Alternatively, you can use the **ESC** key or **Right Click** on the mouse and select **Cancel**.

Filter: The selection filter helps control the content of an active object by allowing you to include or exclude Dcodes, Layers, Composite Levels, NC Tools, Polygons, or Text from being selected. Simply highlight an object then use Right Click > Selection > Filter. This will open the Selection Filter dialog box for that layer.

Reset Filter: Resets the selection filter.

3.2.4 Right Click - Clipboard Options



Cut to Clipboard: Cuts selected object(s) to clipboard.

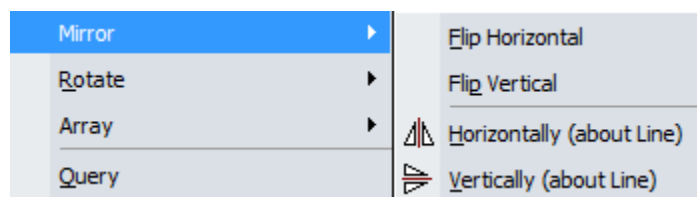
Copy to Clipboard: Copies selected object(s) to clipboard.

Delete: Deletes selected object.

Move: Moves selected object.

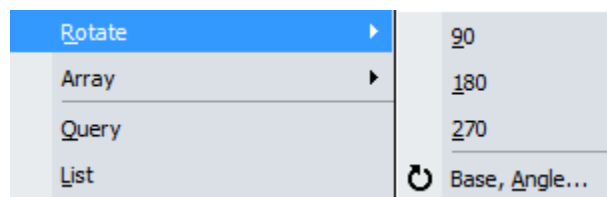
Copy: Copies selected object.

Mirror Options



Mirror: Includes Flip Horizontal, Flip Vertical, Flip Horizontal (Line), Flip Vertical (Line).

Rotate Options



Rotate: Rotates 90°, 180°, 270° or Via a specified base and angle using the command line editor.



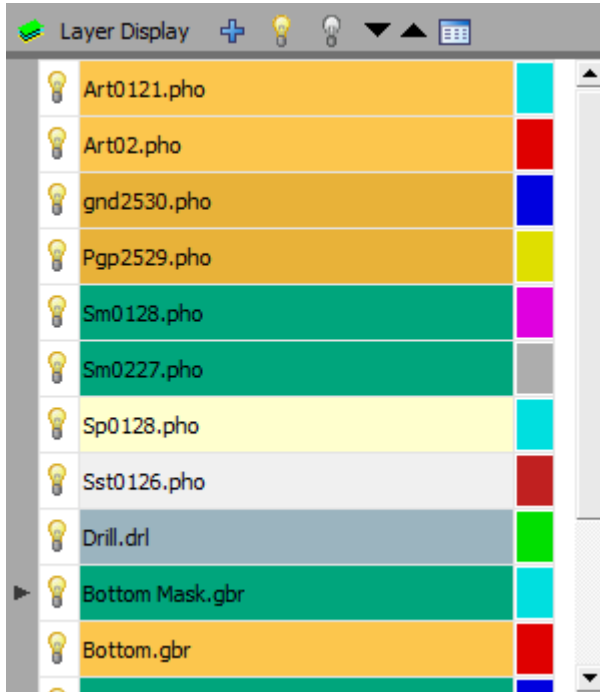
Array: Edit for rectangular or polar array.

Query: After selecting query, select an object. A dialog box will display the object's properties.

List: Opens the command line log.

Properties: Edit the properties of an object. This includes layer attributes and point list.

3.4 Layers Panel



3.4.1 Turn Layers On / Off

Next to each layer name is a light bulb icon that represents the current view state of the layer in a design. The light bulb indicates the status of each layer. Click the light bulb to activate/deactivate each layer.

 If the light bulb is dark, the layer is inactive.

 If the light bulb is yellow, the layer is active.

3.4.2 Adjusting Layer Color

Individual layer color can be adjusted for easier viewing.

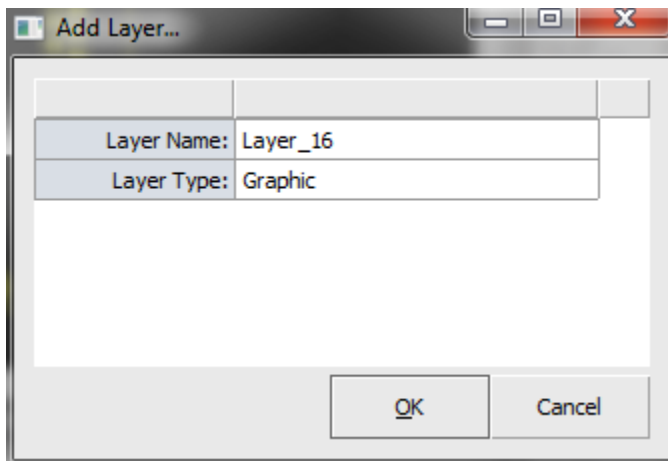
To adjust layer color, left click the adjacent color box. Choose or adjust to a different layer color from the color selection box.


3.4.3 Isolating Layers

You can isolate a particular layer for viewing while hiding the remaining layers.


To isolate a layer, right click within the layers panel and select **Isolate** from the menu.

3.4.4 Add Layer

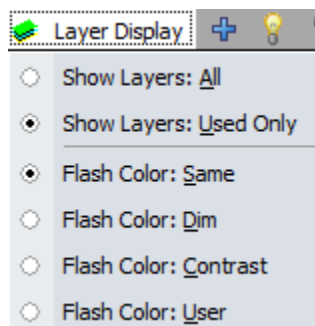


To add a layer, select the  symbol. The Add Layer dialog box will appear. Enter the new layer name and layer type. Select OK.

3.4.5 View Individual Layers


To view an individual layer within the editor, double click the layer within the layers panel. To cycle through the list of individual layers, use the  arrows.

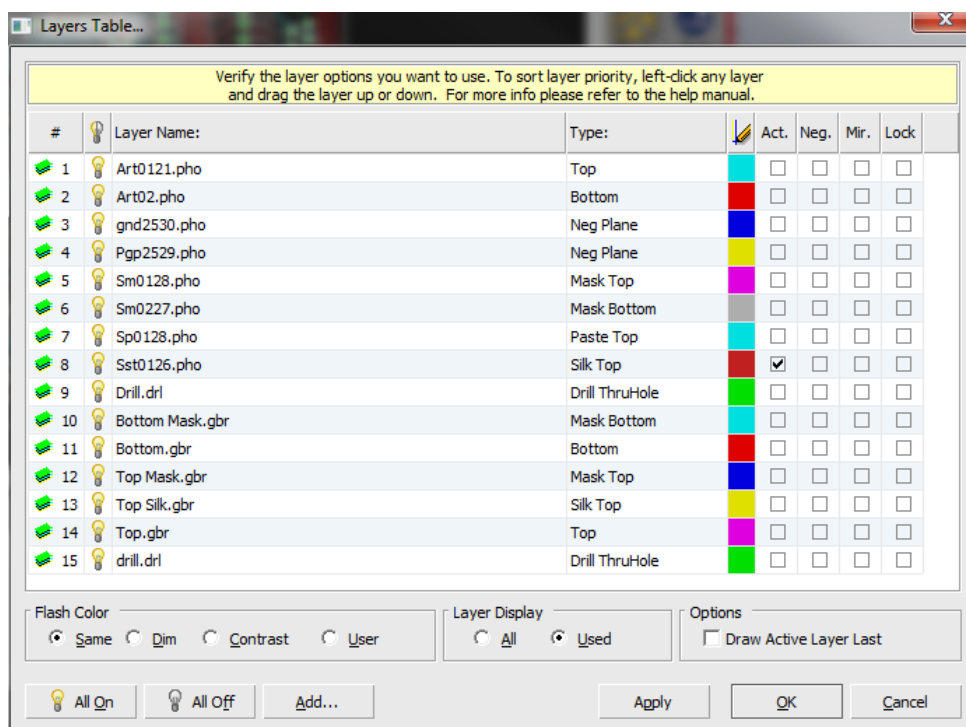
3.4.6 Layer Display



Use the layer display drop down menu to display layers by All, Used Only, or Flash Color.

3.4.7 Layers Table

To add, edit, and verify individual layers, select the Layers Table Icon. 



3.4.8 Right Click Layer Control

Type: Bottom ▶	Right click within the layers panel to activate a number of functions for layer manipulation. The standard options include:
💡 Isolate Layer	Type: Opens parameters based on type of layer selected.
💡 All Layers On	All Layers On: Turns all layers on.
💡 All Layers Off	All Layers Off: Turns all layers off.
<input type="checkbox"/> Negative Layer	Negative Layer: Displays as a negative layer.
<input type="checkbox"/> Lock Layer	Lock Layer: Locks an individual layer.
Add...	Add: Adds a new layer.
Rename...	Rename: Renames a selected layer.
Clear Layer	Clear Layer: Clears a selected layer.
Delete Layer	Delete Layer: Deletes selected layer.
Reload...	Reload: Reloads a selected layer.
Merge...	
Edit...	
Sort Layers	
Select Layer	

Merge: Merges a new layer into the selected layer.

Edit: Edits layer information.

Sort Layers: Sorts layers.

Select Layer: Selects a layer.

Depending on the layer selected, **Type** options will change. For example, if right clicking over an **SMT Stencil Layer**, the menu options will change and accommodate for this type of layer. If right clicking over a **Silkscreen Layer**, the menu options will change as well.



[Video: Manage Layers and Extract a Netlist](#)

3.5 Opening & Saving Workspaces

3.5.1 Opening a Workspace

1. To open a workspace, go to **File > Open Workspace**.
2. Choose the workspace file name you wish to open.
3. Select **Accept**.

3.5.2 Saving a Workspace

1. To save a workspace, to File > Save Workspace.
2. Choose a file name for your workspace. This file is in (*.wrk) format.
3. Select **Accept**.

3.6 Using Snaps

Snaps control the cursor location within the editor. Snaps provide accurate placement of a cursor or can help measure an object accurately. Snaps can be activated by selecting the **View > Snap** menu. Alternatively, pressing **F9** will automatically activate Auto Snap (snaps to center + end of object).

Snap	<input type="radio"/> Snap Off F9	Snap Off – Deactivates snap.
Ortho	<input type="radio"/> to Grid F9	Snap to Grid – Snaps cursor to FAB 3000 background grid.
Object View Filter...	<input checked="" type="radio"/> to Center F9	Snap to Center – Snaps cursor to the center of an object.
Layer Display	<input type="radio"/> to Edge F9	Snap to End – Snaps to end of an object.
Highlight Mode	<input type="radio"/> to End F9	Snap to Edge – Snaps cursor to the edge of an object.
<input checked="" type="checkbox"/> Translucent	<input type="radio"/> to Object Center F9	Snap to Object Center – Snaps to the center of an object.
<input checked="" type="checkbox"/> Origin Icon	<input type="radio"/> to Insertion F9	
<input type="checkbox"/> Film Box	<input type="radio"/> Auto (Center + End) F9	

Snap to Insertion – Snaps to the Insertion point of an object.

Auto – Auto snaps based on the center and end of an object.

3.7 Using Measure

Measure	Measure Point-Point
Rulers	Measure Object-Object
Reports	Measure Net-Net

FAB 3000's measure feature allows you to measure objects from point to point, object to object, and net to net. To activate measure, select **Info > Measure**

or select **Measure** from the toolbar under **Utilities**. Choose whether to measure point to point, object to object, or net to net.

Note: Using the Snap feature (2.6) will help ensure accurate measurement readings.

3.7.1 Measuring Point to Point

1. Select an anchor point within the object and **Left Click**.
2. Move the mouse to the 2nd desired location. As you the mouse travels over the object, a window will display the distance & angle relative to the original anchor point.

Note: Using the Snap feature will ensure accurate measurement reading. Choose a snap variation depending on the type of measurement you're doing. For example, to measure the distance of your anchor point to the center of an object, choose **Snap to Object Center**. To force orthogonal (90°) snapping, select **View > Ortho > 90 Degrees**.

3. When complete, **Left Click** in the editor again. This will reset your measure and allow you to define a new anchor point.

3.7.2 Measuring Object to Object

1. Select the base of an object and **Left Click**. You'll notice a white boundary will outline around the object. Base objects can be nested within an Insert.
2. Move the mouse over to the next object and **Left Click**. A window will appear displaying the distance & angle relative to the original anchor point.
3. The base object you selected will stay active until you reset for a new measurement set. To reset the base object and start again, use the **ESC** key or **Right Click** and select **Cancel**.

3.7.3 Measuring Net to Net

1. Select the base of an object and Left Click. You'll notice a white boundary will outline around the object.
2. In order to successively measure, the base must contain a Net.
3. Move the mouse over the next object and Left Click. A window will appear displaying the distance & angle relative to the original anchor point.
4. The base object you selected will stay active until you reset for a new measurement set. To reset the base object and start again, use the **ESC** key or **Right Click** and select **Cancel**.

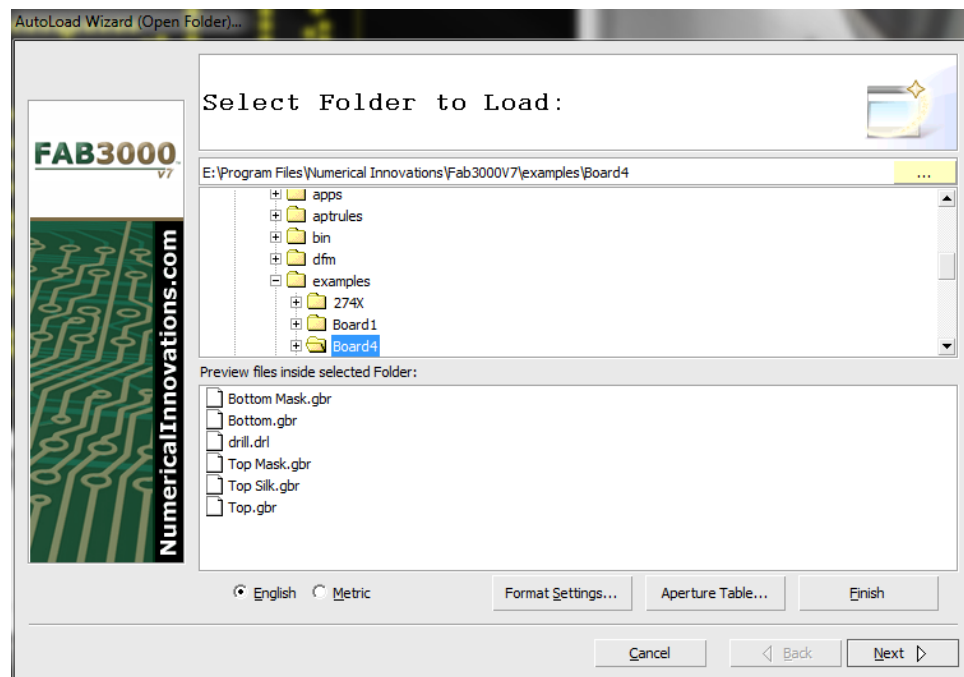
3.8 Importing Files Basics

With FAB 3000, PCB designers can easily import and prepare their Gerber, Drill, DXF, ODB++, and a number of additional file types for analysis and preparation prior to manufacturing. Using the AutoLoad Wizard, multiple files can be imported thereby reducing the time to load individual files. The files available for import are:

Gerber	DXF/DWG	ODB++ & ODB++ Directory	Drill (*.DRL)	Rout/Mill
Netlist	Image	HPGL/2	Postscript/PDF	Large Bitmap
STL (3D Files)	GDS-II (Stream)	Part Centroid Files	BDM Files	Aperture List
Custom Aperture Library	Footprints Library	Text File	Spreadsheet File (*.XLS)	FAB 3000 Matrix

For a complete overview of each file type, see **SECTION 7: FAB 3000 Data Format Overview**.

3.8.1 Using the AutoLoad Wizard



AutoLoad Wizard saves time and increases productivity by allowing you to import multiple Gerber and Excellion Drill files at once. To use the **AutoLoad Wizard**, perform the following:

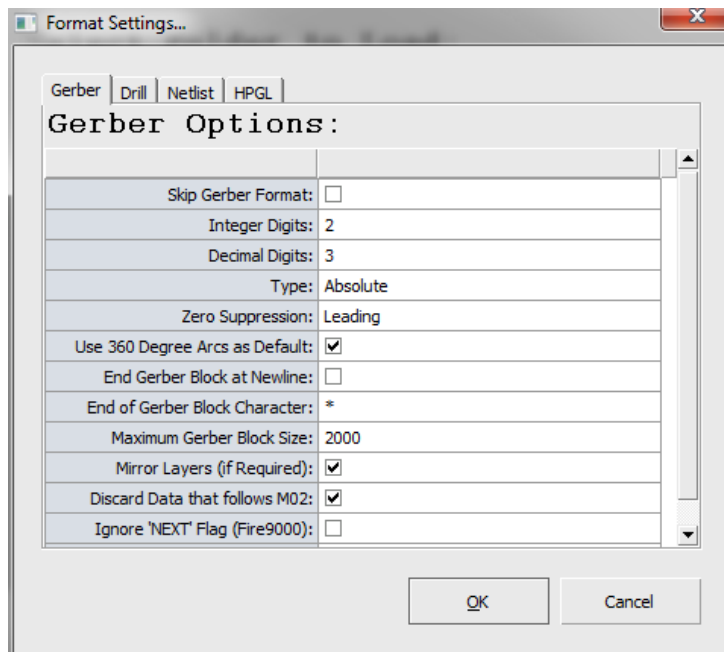
1. **Select File > Import > AutoLoad.** The FAB 3000 AutoLoad Wizard menu will appear.
2. Select a **Folder Path** or **Type** the folder path directly into the path entry box. You can also use the **Folder**

Selection Tree to set the desired folder path for your import.

For a complete overview of the AutoLoad Wizard, see **SECTION 5: FAB 3000 Job Editor Menu**.

3. **Choose** the Folder where your files are located. Below the graphical view of the folder directory path, you'll notice the window: "Files Inside Selected Folder". This reveals the current contents of the file you've chosen.
4. **Choose** English or Metric measurements in the bottom left of the wizard.
4. **Select** Finish. This will bypass any custom import or format settings and immediately proceed to the importation process.

AUTLOAD FORMAT SETTINGS



The format settings button within the AutoLoad Wizard allows you to setup custom import defaults. This option is available to you prior to importing your files. If you do not wish to change the default settings, simply skip this option.

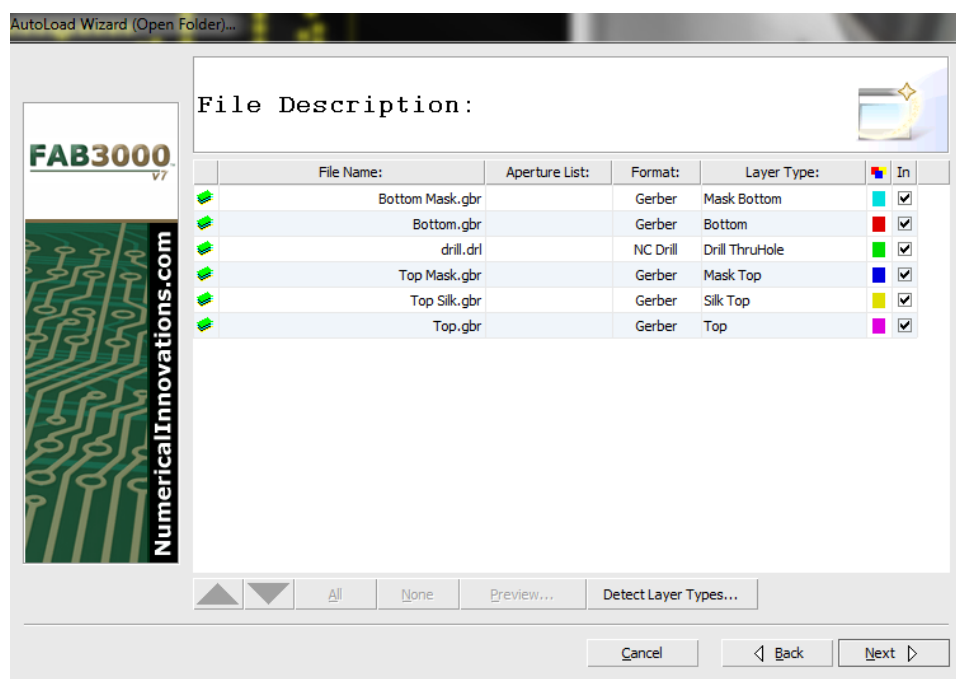


[Forum: How does AutoLoad Work?](#)

[Forum: Load Multiple Aperture Files Along with Gerbers](#)

[Video: FAB 3000 Basics \(Part 1\) – Import Zip Gerber and Drill, Edit Gerber, Align Layers, Query, Measure](#)

[Video: FAB 3000 Basics \(Part 2\) – Import Zip Gerber and Drill, Edit Gerber, Align Layers, Query, Measure](#)



AUTOLOAD FILE DESCRIPTION MENU

If you need to fine-tune what will be imported Via AutoLoad, select **Next** instead of **Finish**. The **File Description Menu** will appear. It allows you to edit the following information:

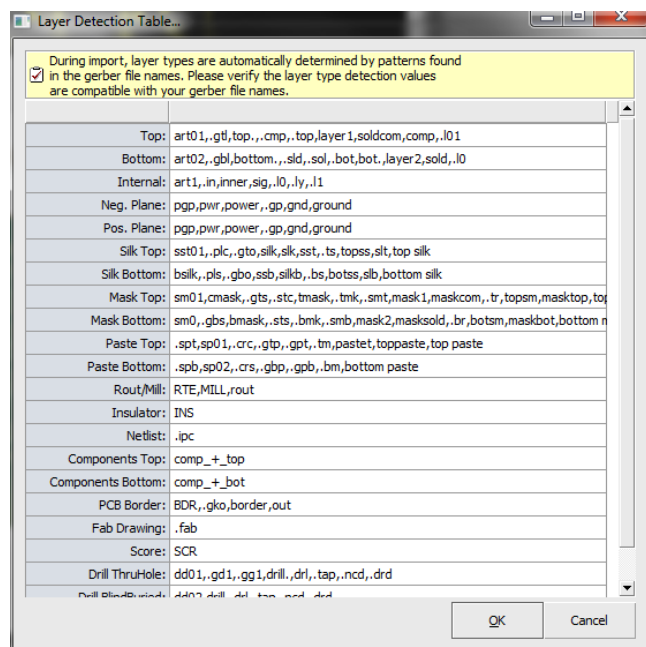
1. File Name: Name of each file to be imported.
2. Aperture List: Allows you to optionally add a corresponding aperture list.

3. Format: Provides you with the detected format of each file.

4. Layer Type: Allows you to assign a layer type to each file.

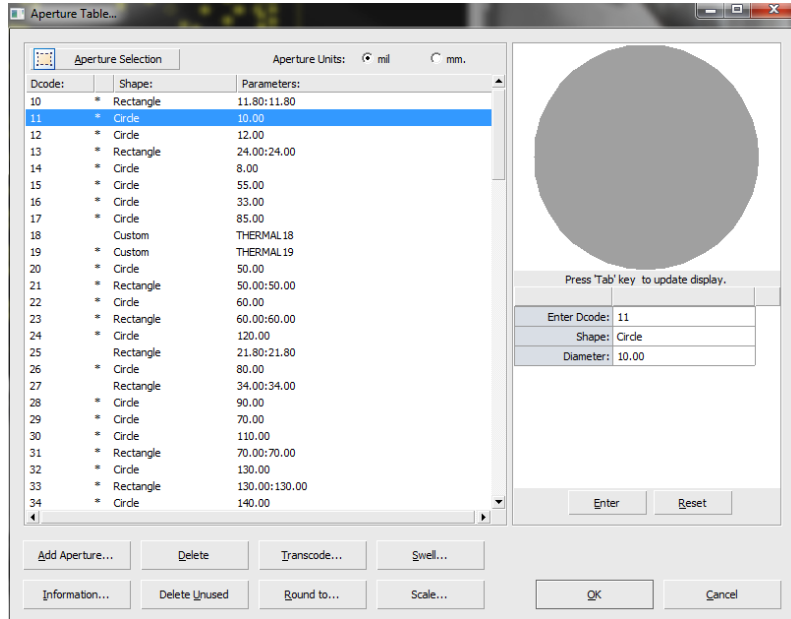
5. Import: Choose which files will be imported by selecting/unselecting each checkbox.

AUTOLOAD DETECT LAYER TYPES



Detect Layer Types is part of the File Description Menu. You can access this table by selecting **Detect Layer Types**. It provides information on each layer type. During import, layer types are automatically determined by patterns found in the Gerber file names. Use this feature to verify that the layer type detection values are compatible with your Gerber files names.

AUTOLOAD APERTURE TABLE



The Aperture Table allows you to individually define the parameters for each polygon defined in the drawing. This includes adding additional apertures, swell, scale, swell, transcode, delete, and information.

AUTOLOAD FINISH PAGE

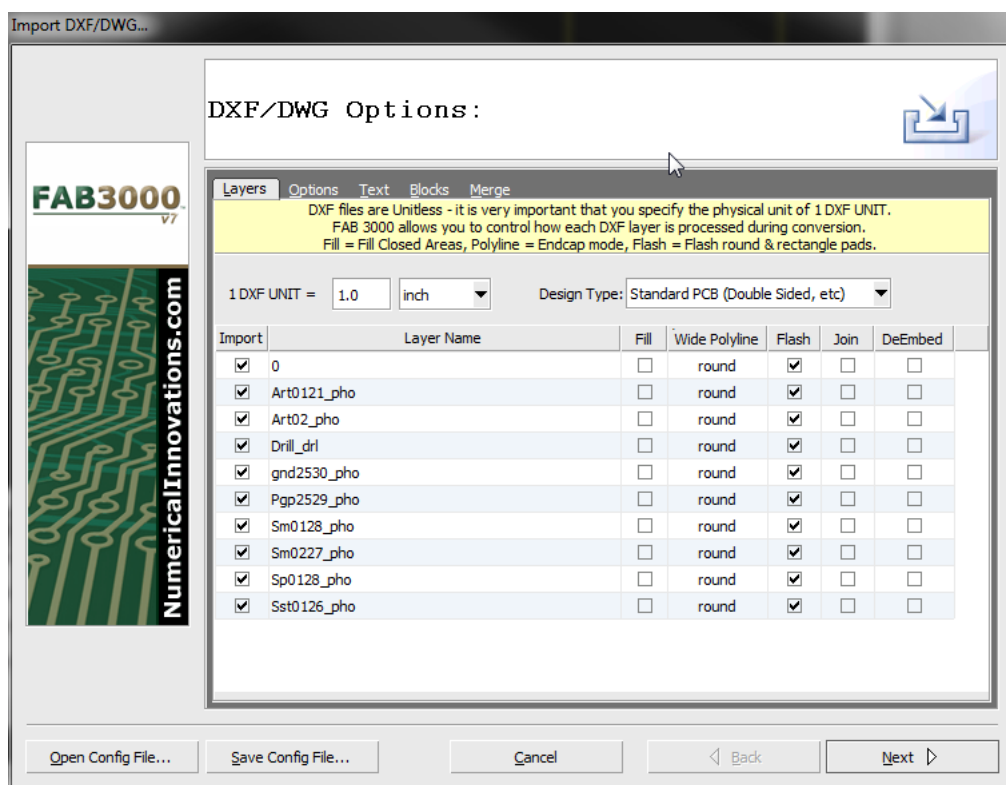
The Finish Page is the final page of the AutoLoad Wizard. It will reveal the progress of each file as it is imported. If no error is found during import the Autoload dialog box will automatically close. If an error is found, the dialog box will remain open in order to view the error log. Alternatively, you may press F2 any time after the import to view the FAB 3000 output log which will also contain information regarding any errors during the import.

3.8.2 DXF/DWG File Import Process

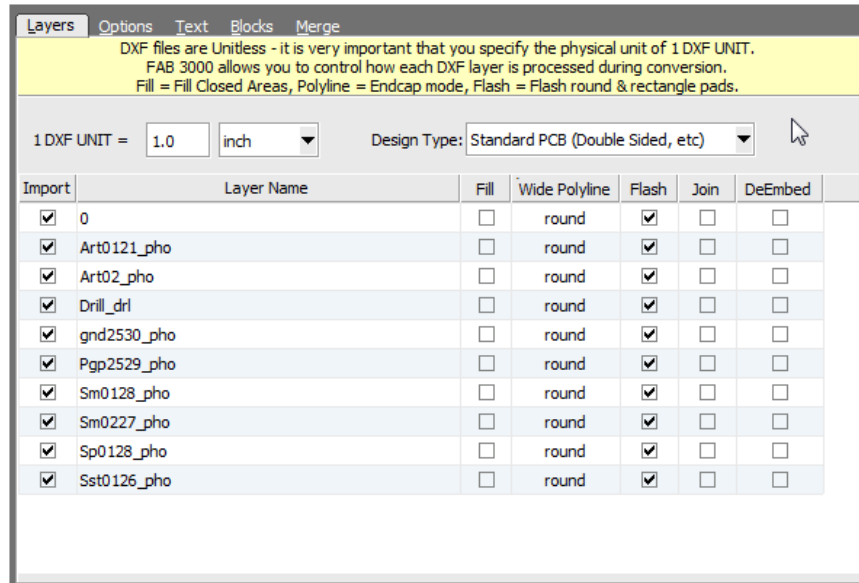
DXF and DWG files can be imported into FAB 3000. Please note that since DXF files lack units, it is important you verify the units & scale factor of your file prior to importing it. Each DXF import is scanned for Layers, Blocks, Styles, and more. The FAB 3000 import menu provides a series of separate tabs to help you control and verify the parameters of your import.

To begin a DXF/DWG file import, perform the following:

1. Select **File > Import > DXF/DWG**.
2. **Choose** a DXF file to import.
3. The **DXF Options Menu** will appear. This is where you'll verify DXF layer options.



The DXF Options menu determines how a file can be imported. Within the DXF Options menu, there are several tabs to help determine import parameters:



1. Layers Tab

Layers allow you to control how DXF layers are handled during import.

- a. Import Column: Check this box to include the layer during import.
- b. Layer Name: Name of layer.
- c. Fill Column: Converts closed boundaries to filled-polygons.
- d. Wide Polyline Column: Adjusts how polylines with width are handled during import. There are three choices – Extend, Truncate, and Round.
- e. Flash Column: Detects circles & rectangles for conversion to Gerber flashes.
- f. Join Column: Check to join.

- g. De-Embed Column: Algorithm used to verify polygons inside of polygons. Please note this check requires additional processing time.



[Video: Converting Gerber to DXF using FAB 3000](#)

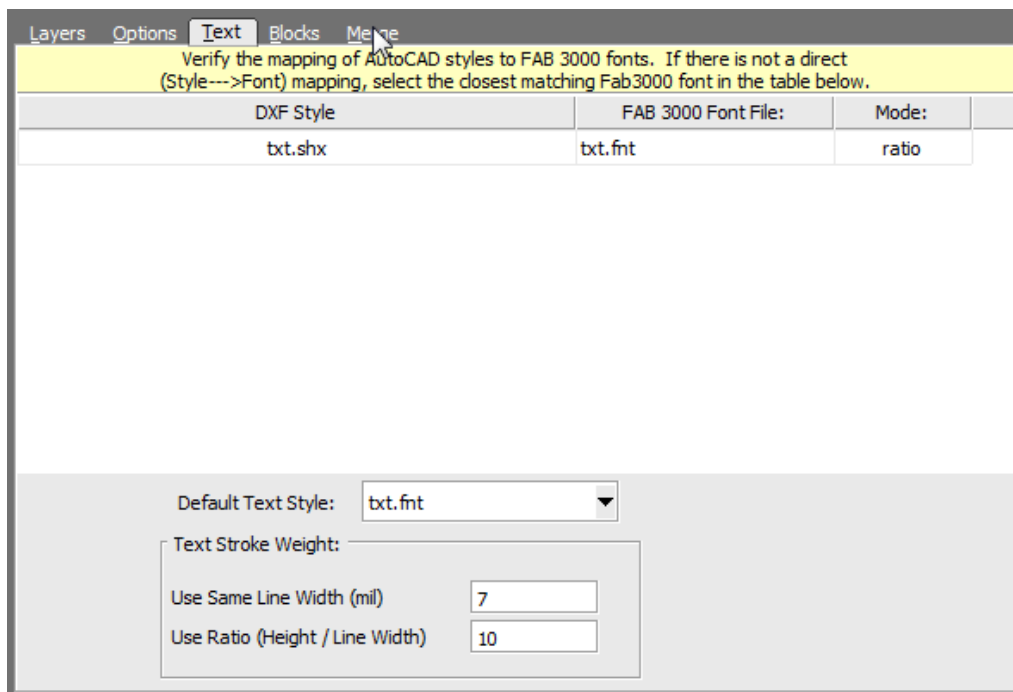
[Video: Converting DXF to Gerber and NC Drill Data](#)

2. Options Tab

Layers Options Text Blocks Merge	
Verify the DXF options you want to use. For high quality curves make sure to adjust option 'Arc Deviation'. Please refer to the "Help Guide" for assistance with any specific option.	
Arc Resolution Mode:	Use Deviation
Arc Degrees:	2.0
Arc Deviation (mil):	0.2
Defalut Line Width (mil):	5
Flash Circle/Rects Less Than (mil):	175
Flash Objects:	Circles/Donuts and Rectangles
Respect Byblock/Bylayer:	<input checked="" type="checkbox"/>
Ignore Hatches:	<input type="checkbox"/>
Ignore 0-Width Entities (Line,Arc,etc.):	<input type="checkbox"/>
Ignore Hidden Entities:	<input checked="" type="checkbox"/>
Explode All Inserts:	<input checked="" type="checkbox"/>
Use AutoCAD Layer Colors:	<input checked="" type="checkbox"/>
Respect Object visibility:	<input checked="" type="checkbox"/>
Force Join to Close All Boundaries:	<input type="checkbox"/>

Set the options which affect how a DXF file is imported.

3. Text Tab



Verify the mapping of AutoCAD styles to FAB 3000 fonts. If there is not a direct (Style--->Font) mapping, select the closest matching Fab3000 font in the table below.

DXF Style	FAB 3000 Font File:	Mode:
txt.shx	txt.fnt	ratio

Default Text Style: txt.fnt

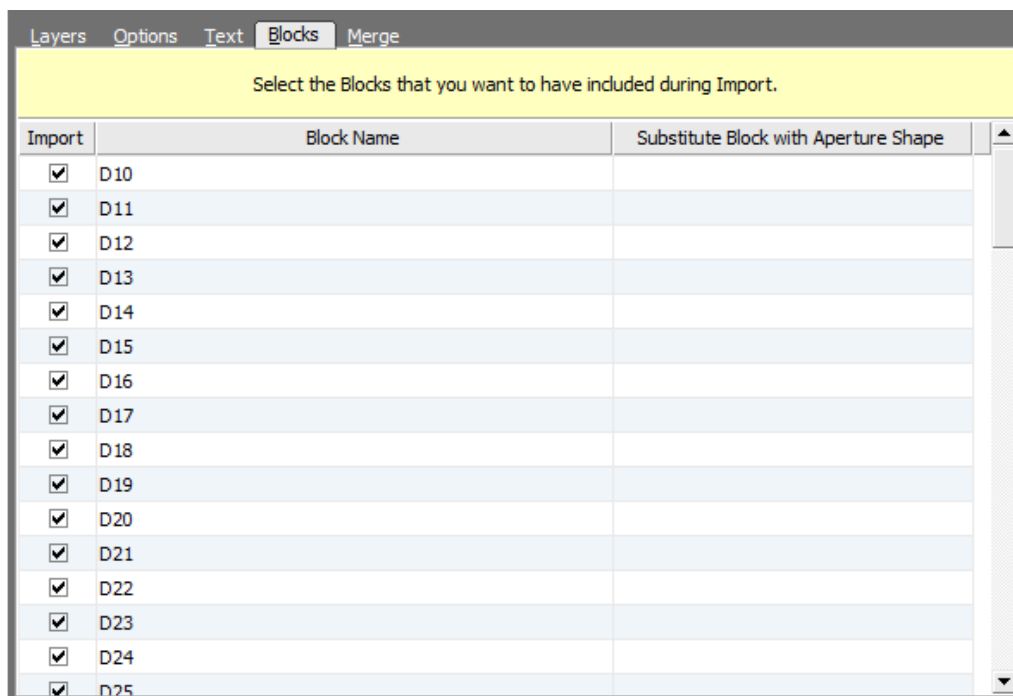
Text Stroke Weight:

Use Same Line Width (mil): 7

Use Ratio (Height / Line Width): 10

Verify the mapping of AutoCAD styles to FAB 3000 fonts.

4. Blocks Tab



Select the Blocks that you want to have included during Import.

Import	Block Name	Substitute Block with Aperture Shape
<input checked="" type="checkbox"/>	D10	
<input checked="" type="checkbox"/>	D11	
<input checked="" type="checkbox"/>	D12	
<input checked="" type="checkbox"/>	D13	
<input checked="" type="checkbox"/>	D14	
<input checked="" type="checkbox"/>	D15	
<input checked="" type="checkbox"/>	D16	
<input checked="" type="checkbox"/>	D17	
<input checked="" type="checkbox"/>	D18	
<input checked="" type="checkbox"/>	D19	
<input checked="" type="checkbox"/>	D20	
<input checked="" type="checkbox"/>	D21	
<input checked="" type="checkbox"/>	D22	
<input checked="" type="checkbox"/>	D23	
<input checked="" type="checkbox"/>	D24	
<input checked="" type="checkbox"/>	D25	

Check each box to include Blocks during import.

5. Merge Tab

Layers Options Text Blocks **Merge**

Merge DXF layers (Optional Feature). To create a Merge, select button 'Add', then enter the Merge Name, and then click on the adjacent row 'Layers' to assign the DXF layers to be included in this merge. Repeat as many times as necessary.

Merge Name	Merge Stackup (Click item to edit):

Add Merge... Delete Merge... Output All Layers (Default) ▼

Merges DXF layers during import.

- Select **Add Merge** to enter a merge name and then click a row under **Layers** to assign the layers to be merged during import. Select **Delete Merge** to remove a merge name and merge stackup.
- Click a row under **Merge Stackup** to edit.

Open/Save Config File

Allows you to open & save an existing DXF configuration file (*.dcf) so that you may assign the exact same settings used previously. Reusing the DXF configuration will save time and reduce errors.

Once these options are set, click **Next** to proceed.

As the import begins, the **Status** window will list any warnings or errors in the log list. If there are no errors, the dialog box will instantly close. If there is an error, you can view the complete error log for correction.

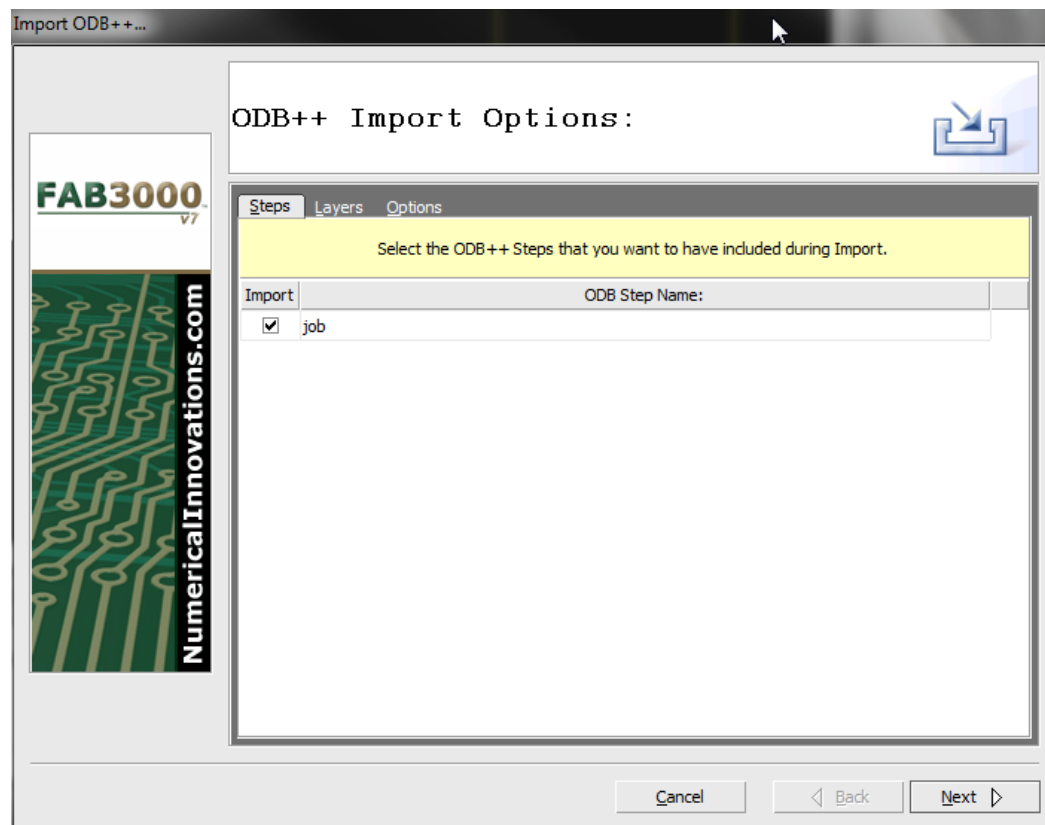
Select **Finish** to close the menu and finalize the import process.



[Video: DXF Import with de-embed and join checked with FAB 3000](#)

3.8.3 ODB++ File Import Process

FAB 3000 allows you to load compressed (*.tgz) ODB++ databases into the workspace. To import an ODB++ file, perform the following:



1. Select **File > Import > ODB++**
2. **Choose** the ODB++ file that you want to import and **Select OK**. The import dialog box will appear.
3. Verify the ODB++ import options that you'll use. This includes Steps (pictured above; also known as Jobs), Layers, and Options.



Forum: Converting Gerber to ODB++

Steps Layers Options

Verify the ODB++ layers you want to import. Please refer to the "Help Guide" for assistance with any specific option. Click 'Next' to begin importing.

Import	ODB Layer Name	Layer Type	Polarity
<input checked="" type="checkbox"/>	art0121.pho	Top	Positive
<input checked="" type="checkbox"/>	art02.pho	Bottom	Positive
<input checked="" type="checkbox"/>	gnd2530.pho	Pos Plane	Positive
<input checked="" type="checkbox"/>	pgp2529.pho	Pos Plane	Positive
<input checked="" type="checkbox"/>	sm0128.pho	Mask Top	Positive
<input checked="" type="checkbox"/>	sm0227.pho	Mask Bottom	Positive
<input checked="" type="checkbox"/>	sp0128.pho	Paste Top	Positive
<input checked="" type="checkbox"/>	sst0126.pho	Silk Top	Positive
<input checked="" type="checkbox"/>	drill.drl	Drill ThruHole	Positive

- a. **Layers:** Verify and edit the ODB ++ layers you would like to import.

Steps Layers Options

Verify the ODB++ options you want to use. Please refer to the "Help Guide" for assistance with any specific option. Click 'Next' to begin importing.

Options	Value
Default Units:	English

- b. **Options:** Set default units for import.

4. **Select Next.**

5. Once the ODB++ loading is complete, a status & import log will be available for review for any errors or warnings.

6. **Select Finish.**

FAB 3000 loads complete ODB++ databases with all steps while maintaining hierarchy. It will search the hierarchy to determine which is known as the "master" step. If it cannot find a master step, a dialog box will appear prompting you to select which step to view. You can also hit F4 to view the import log file.



[Video: How to Skip a Layer during ODB++ Import](#)

3.8.4 Open Zip (Compressed File)

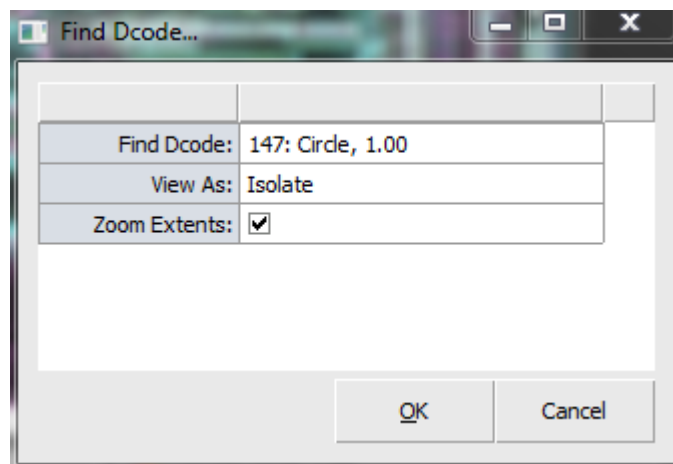
FAB 3000 automatically loads Gerber and Drill data from compressed files. This includes file formats such as *.zip, *.tar, *.gz, *.bz2, and *.7z. To import a compressed file, perform the following:

1. Select **File > Import > Zip (Compressed File)**.
2. Choose the compressed file you want to import and **Select OK**. The import dialog box will appear.

FAB 3000 will automatically determine the correct format and aperture rules necessary to import Gerber files with minimal effort. However, if you are aware of special circumstances regarding your Gerber files (such as 274D Gerbers using an Incremental type, End-Block-Character other than "", etc.) or if you've have previously imported Gerbers loads correctly, you may want to change AutoLoad's default settings by clicking the button **Format Settings**. If you are unsure, please leave the format settings defaulted.*

3. **Select Next.**
4. Once the compressed file loading is completes, a status & import log will be available for review fir any errors or warnings.
5. **Select Finish.**

3.9 Find Dcodes

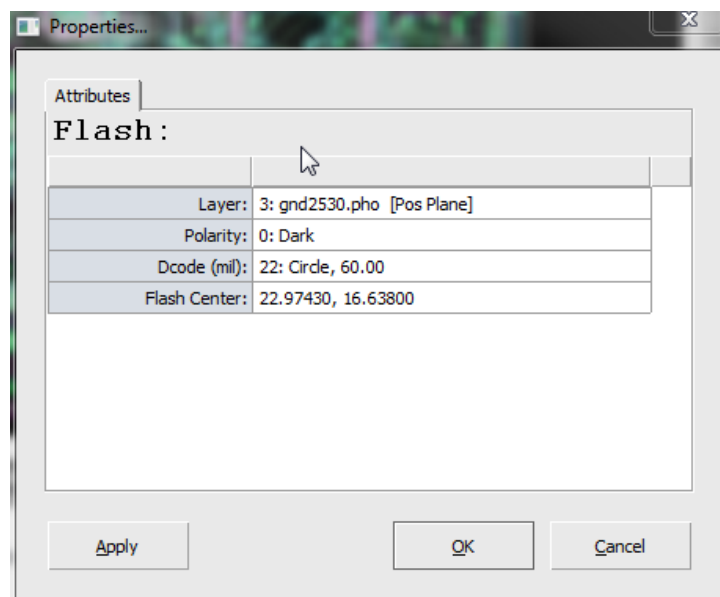


FAB 3000 includes the ability for finding Dcodes within an active Job after importing Gerber files to the workspace. To find Dcodes, perform the following:

1. **Select Info > Find > Find Dcode**
2. In the **Find Dcode** menu, choose which Dcode you want to view by selecting it from the dropdown list.
3. Choose either **Highlight** or **Isolate** to view the results.
4. All objects that using the Dcode which you selected will be highlighted and zoomed in the editor.

Note: Make sure to have all layers turned **ON** while using Find Dcode. If the layers are deactivated you will not see the specified highlighted objects.

3.10 Using the Properties Dialog



For each object in your design, you can view and edit the properties of that object. In FAB 3000, properties of an object are divided into two primary categories: Attributes and Point List.

1. To view the properties of an object, first actively **Select** an object from the editor.
2. Go to **Edit > Properties**. The properties dialog box will appear. This will list the properties of the active object.

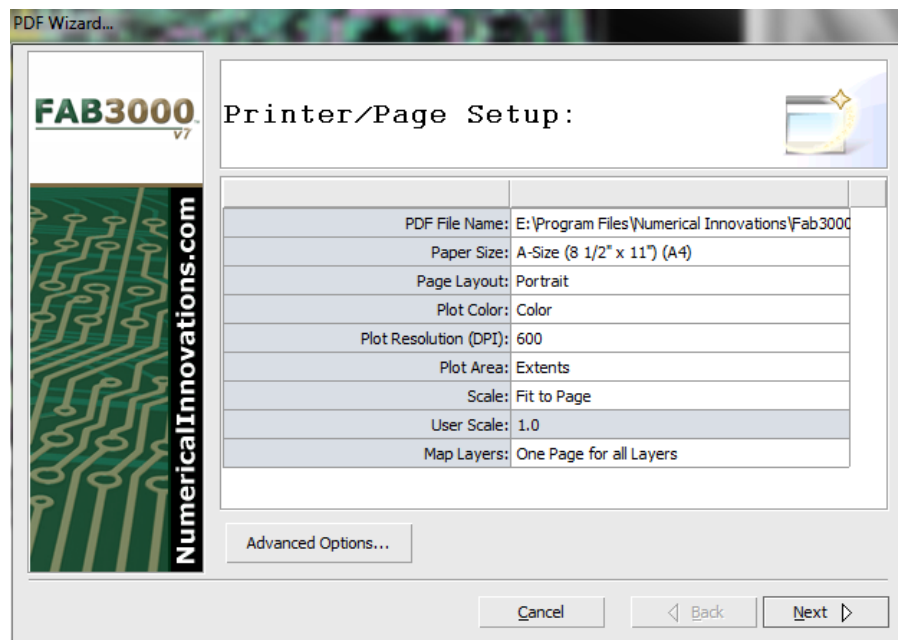
Under the **Attributes Tab** you may select and edit the Layer, Composite Level, Dcode, or Flash Center.

Alternatively, you can view the properties dialog box of an object by **double clicking** (left mouse button) the object directly within the editor.

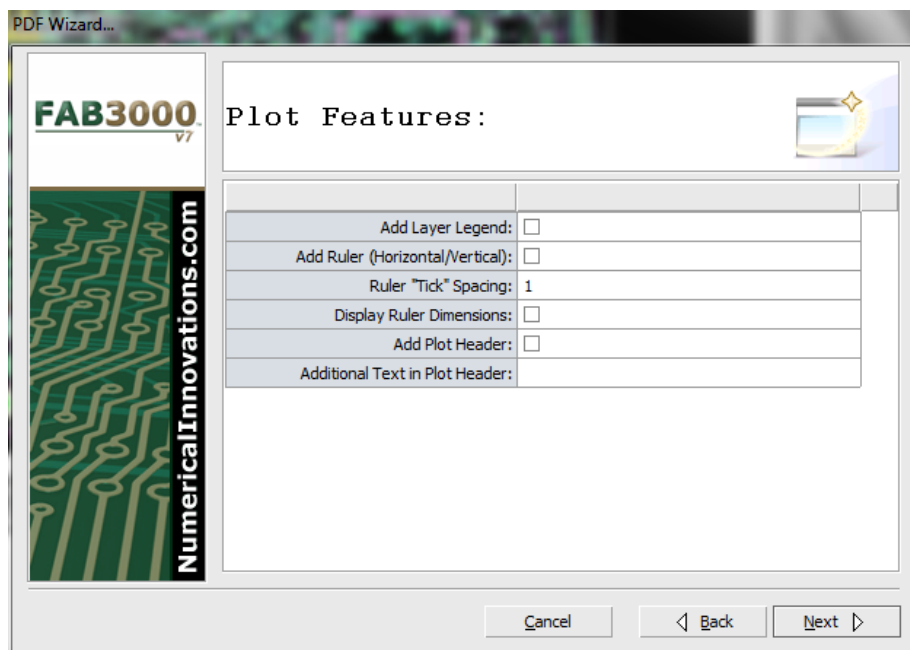
3.11 Plotting to PDF

PCB designs can be quickly plotted to PDF in A-size to C-size as well as user defined size formats. To Plot to PDF, perform the following:

1. Go to **File > Print to PDF**.



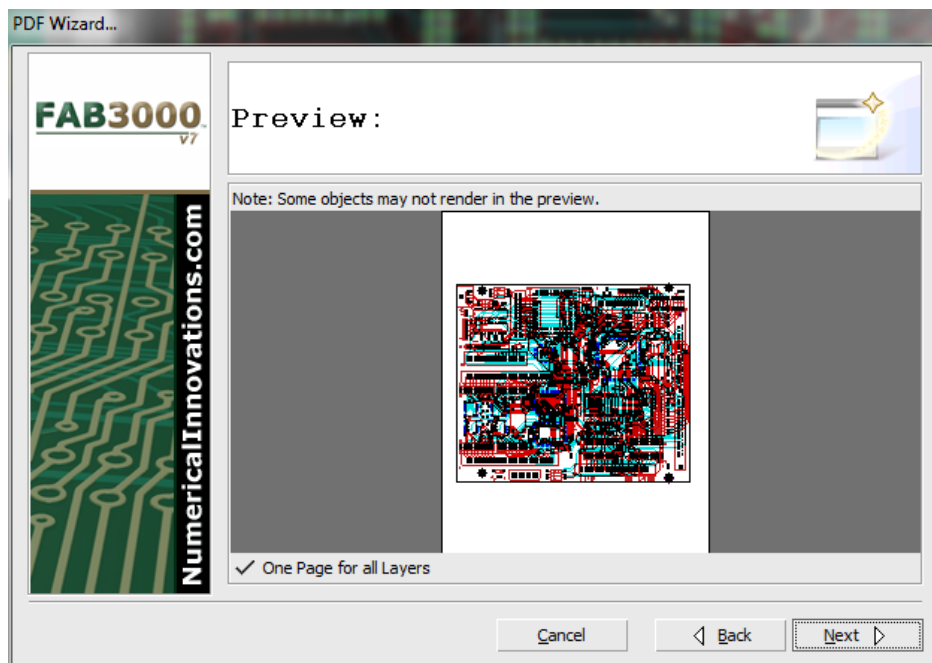
2. In the Printer/Page Setup dialog box, select the applicable plot size by clicking **Paper Size**. All other setting should remain as default.



3. Select **Next**. The **Plot Features** dialog box will appear.

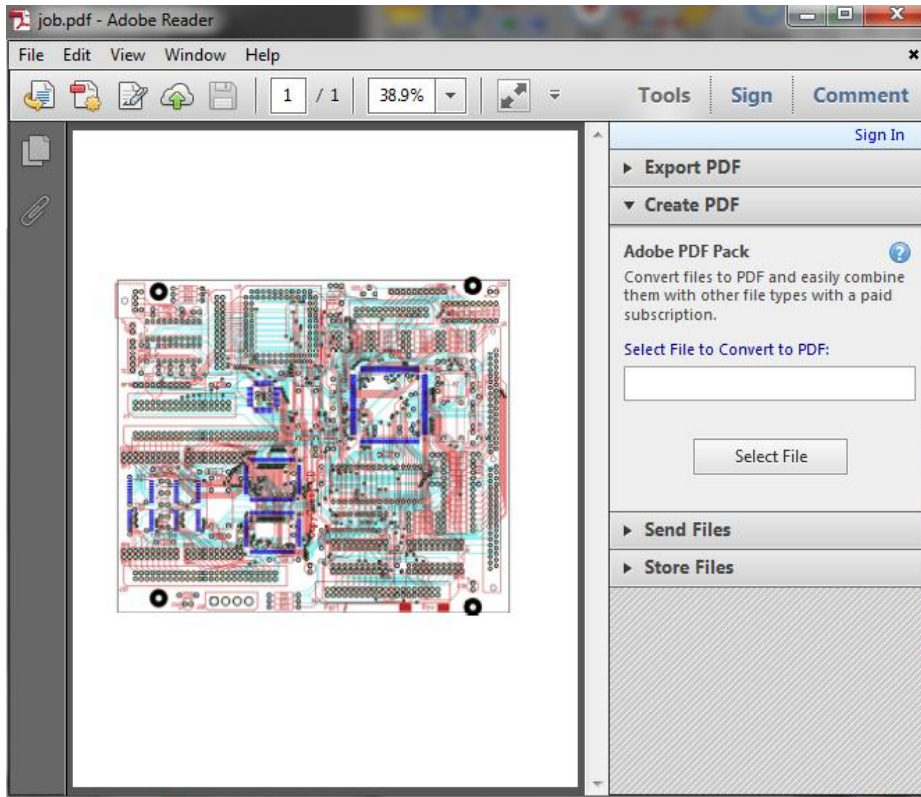
4. Keep all Plot Features settings as default.

5. Select **Next**. The **Preview** dialog box will appear.



6. Review a print preview of the PDF file. If there are no warnings and the preview is acceptable, select **Next**.

7. FAB 3000 will begin to create the PDF file.

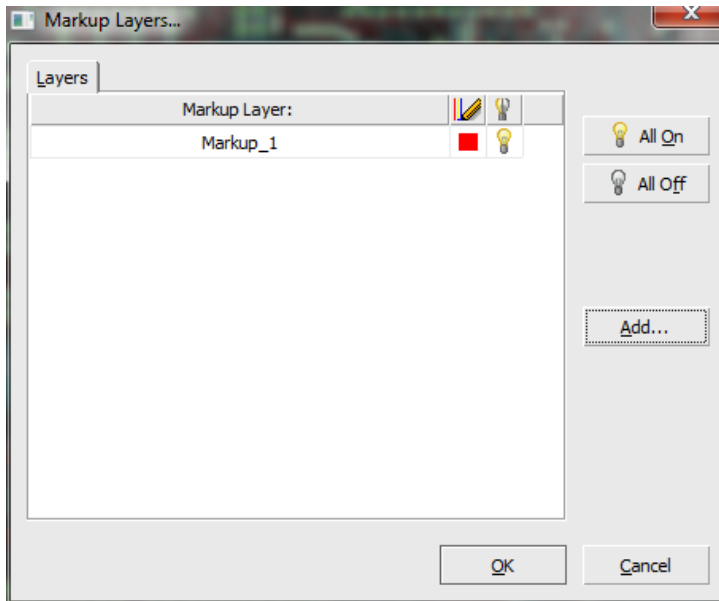


Once the PDF file is created, the **Finish** dialog box will appear and display the location and name of the PDF that was created.

For Windows users, the Finish dialog box will also contain a **View PDF** button that will launch the PDF in Adobe Acrobat Reader.

3.12 Redline Markup

Redline markup allows designers to mark specific areas for review and reference; as well as bringing attention to a particular region of the design. To activate Redline Markup within your design:

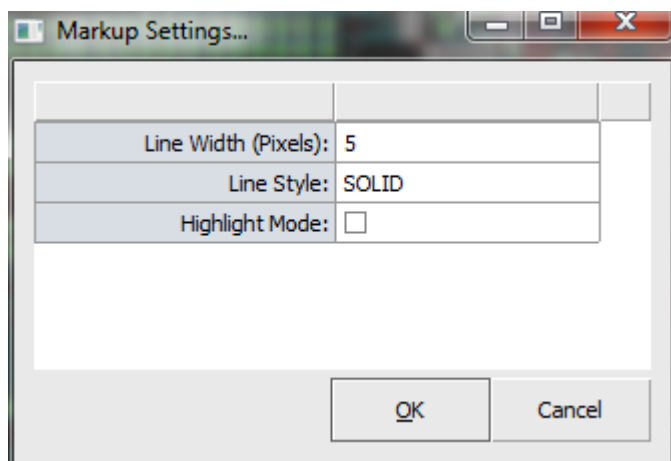


1. First create a **Markup Layer** by selecting **Add > Markup > Markup Layers**.
2. In the Markup Layers dialog box, **Select Add**.
3. Create a layer name to assign to the marked up layer. For example Markup_Layer1.
4. Select **Apply** then select **OK** to close the dialog box.

To change the colors of a markup layer, select the color theme next to the layer.

Markup layers will also be included in PDF and Image exports, saved in the workspace, and are transferable to other users.

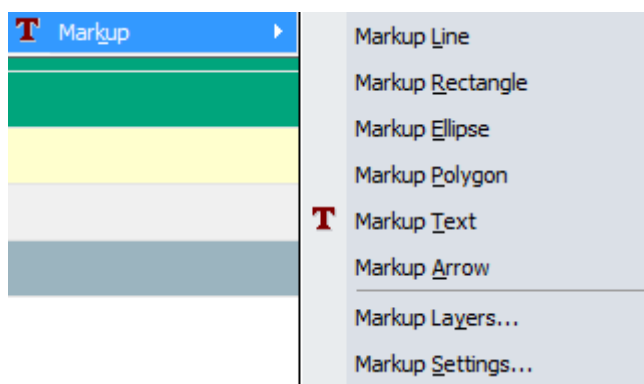
With a markup layer created, it is good practice to set how these markups will appear in a design. For this, go to **Add > Markup > Markup Settings**.



The Markup Settings dialog box provides three custom options:

- Line Width (in Pixels): The width that will be used for the markup objects such as Line, Arrow, Rectangle, Ellipse, etc.
- Line Style: Chooses your markup style between solid or dashed lines. Please note that larger line widths will not be able to display dashes properly.
- Highlight Mode: Allows objects to be marked as solid (unchecked) or highlighted (checked).

3.13 Markup Object Types



There are several choices in Markup Object Types to choose from within the **Add > Markup** menu. They include:

- Markup Line: Select **Add > Markup > Markup Line**

Left click to select the first location of your markup line. Left click again to end the line or continue with additional lines using left click.

- Markup Rectangle: Select **Add > Markup > Markup Rectangle**

Select a corner on the outside of the area you want to box then drag diagonally across to complete the rectangle. Markup rectangle is great for boxing specific regions of a design.

- Markup Ellipse: Select **Add > Markup > Markup Ellipse**

Select a corner on the outside of the area you want to circle then drag diagonally across to complete the ellipse. Markup ellipse is great for circling specific regions of a design.

d. Markup Polygon: **Select Add > Markup > Markup Polygon**

Select a corner on the outside of the area you want using left click. Continue selection until forming the polygon shape you require.

e. Markup Text: **Select Add > Markup > Markup Text**

Select a corner for the text box location then click and drag diagonally to create the box. It will then prompt you to enter text. Markup text is great for instruction or explaining details of a design.

f. Markup Arrow: **Select Add > Markup > Markup Arrow**

Select a starting point on the outside of the area you want to circle. Add additional points as needed. The starting point and second point determine the direction of the arrow. Markup arrow is great for directing to specific points of a design.

SECTION 4: FAB 3000 ADVANCED FEATURES OVERVIEW

In this section, we'll highlight the advanced features of FAB 3000. For a detailed look at each individual menu function in FAB 3000, see **SECTION 5: FAB 3000 Job Editor Menu**.

4.1 Merge Jobs

Merge Jobs is a unique and powerful feature of FAB 3000 that lets so easily merge different PCB designs in order to reduce manufacturing and fabrication costs. Jobs can be merged into a single set of Gerber files for production. FAB 3000 will intelligently perform object transformations, Dcode, and drill tool transcoding in one solution. Using Merge Wizard, designers are able to fuse several PCB designs onto a single panel.

FAB 3000 uses a hierarchal approach to merging Jobs. There are two hierarchal resources that FAB 3000 utilizes: Jobs and Inserts.



[Forum: Merging Gerber Files](#)

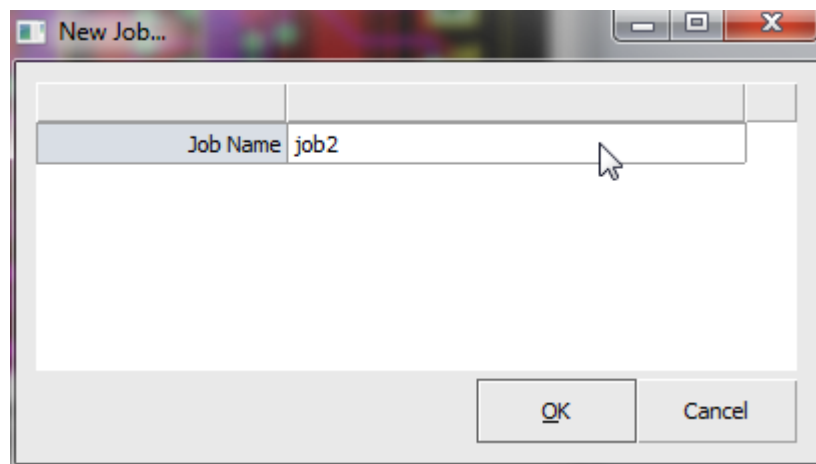
[Video: Merge to Save Money on PCB Manufacturing](#)

4.1.1 Jobs vs. Inserts

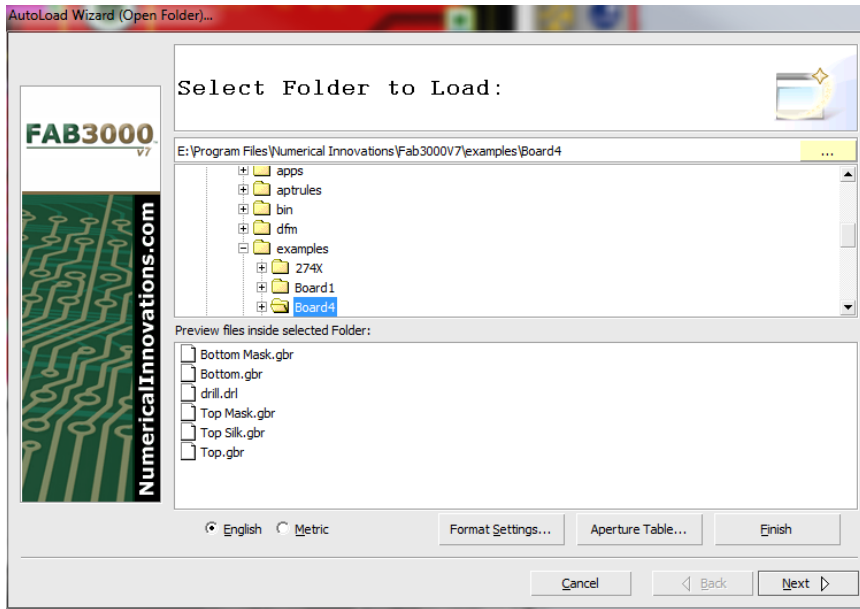
1. Job: A single design (i.e. PCB) with its own unique layers, Dcodes, and other attributes. (All data imported and 90% of editing are handled at the Job level). In FAB 3000 ODB++ terms, a "Step" is defined as the same thing as a "Job"
2. Insert: An Insert represents the inclusion of one Job as a part of another. The Job containing the Insert is the parent Job, and the Job that is inserted into the parent is the child Job. Using this parent/child relationship, you can continue the hierarchy for as many levels as needed to express a complete design.

4.1.2 Merge Jobs Quick Summary

Part 1: Creating a New Job



1. Select **File > New > Job**.
2. Select a name for the new Job. Be sure to create a descriptive name that allows for easy organization. For example, Part1212, Part 1213, etc.
3. Once the Job is created, a blank Job Editor screen will appear.



Part 2: Loading Gerber & Drill Files Automatically into the Active Job

1. Select **File > Import > Autoload**.
2. Select the folder which contains the Gerber and drill files (for a single Job).
3. Click **Finish**.

FAB 3000 will begin to scan the directory and determine whether each file is either a Gerber or Aperture list. There will be a slight delay during scanning – then the

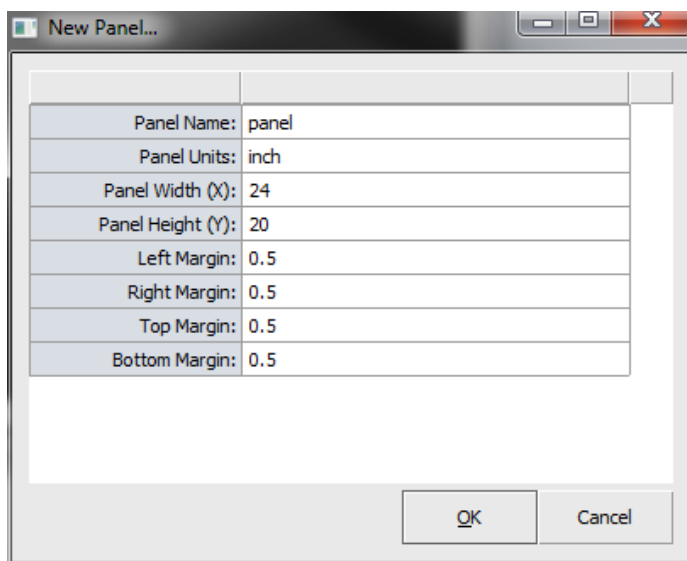
File Description page will appear. The file description page provides the opportunity to define the aperture list to use (RS-274D only), Layer Types, and Import checkbox for each Gerber layer.

When loading is complete, you'll be able to view the imported data as well as be warned of any errors.

Note: For compressed files (*.zip, *.tar, etc.), make sure to use the **Importing Zip (Compressed File)** feature.

For multiple designs, repeat Parts 1 & 2 above. Each of your designs requires a separate Job. For a more detailed description of the Autoload menu, see **Section 5: FAB 3000 Job Editor Menu**.

As mentioned above, all data imported and 90% of editing are handled at the Job level. Merging Jobs requires at least one Job in the current workspace (with data).





4.1.3 Creating a New Panel

1. Select **File > New > Panel**.
2. Choose a name for the new panel.
3. Verify the panel dimensions for Panel Name, Width, Height, Margins, and Units. You can also view and modify panel dimensions by selecting **Tools > Edit Panel**.

Once the new panel is created, you will see a blank panel editor screen with an outlined box representing the panel area.

4.1.4 MERGING JOBS AUTOMATICALLY

Panelize Wizard (Find Smallest Panel Size)...

Job(s) Setup:

Panelize Wizard - Easily merges multiple jobs onto a panel to save on fabrication costs.
 (1) Select the Jobs you want to add.
 (2) Enter the required Count (you need).
 Click 'Next' for Panel Wizard Options.

Select:	Job Name:	Count:
a <input checked="" type="checkbox"/>	job	1



Job Options:
 Job Spacing (Inch):
☒ Center Job(s) in Panel

Panel Options:
 Minimum Panel Efficiency (%)
☐ Increase Panel Size (if needed)

Cancel Back Next

1. Select **Tools > Panelize Wizard**. The panelize wizard dialog box will appear.
2. Check all Jobs under the **Select** column to include in the panel merge.
3. Verify Job Settings Options such as **Job Spacing** and **Panel Options**
5. Select **Next**.

Panelize Wizard (Find Smallest Panel Size)...

Options:

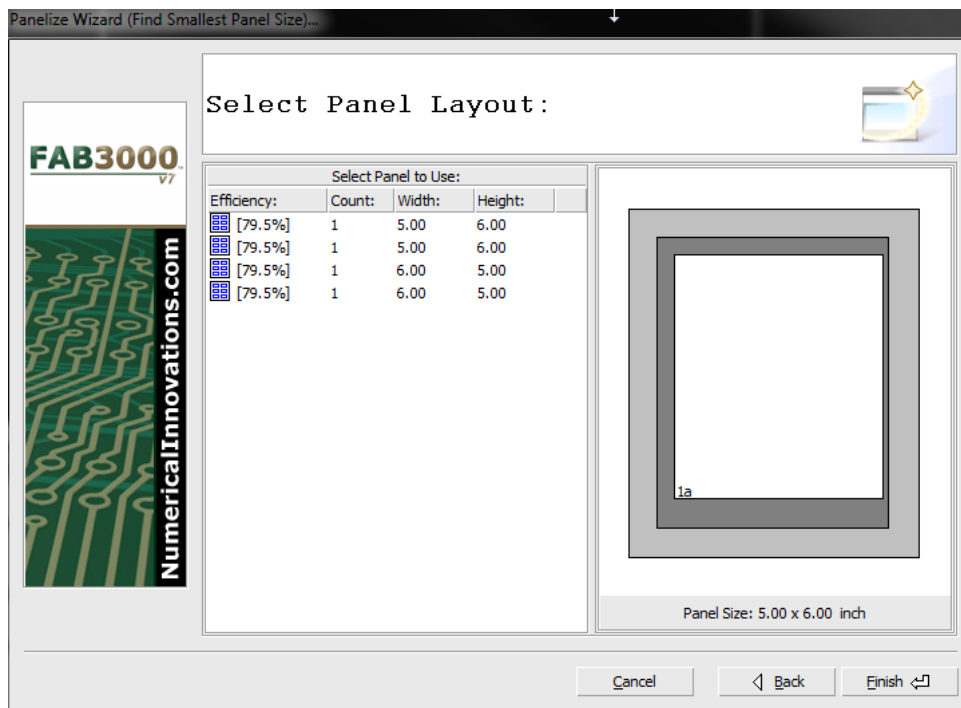
Select the Options that you require.
 Many PCB Fabs require that you provide Outlines, Score Lines, Cropmarks, and Outlines.
 Click 'Next' to Calculate merge.

Add Fiducials:	<input type="checkbox"/>
Add Tooling Holes:	<input type="checkbox"/>
Add Tab-Routs:	<input type="checkbox"/>
Add Scoring Lines:	<input type="checkbox"/>
Add Outline Layer:	<input type="checkbox"/>
Add Crop Marks to Silk Layer:	<input type="checkbox"/>
Add Cut Lines to Silk Layer:	No
Clearance for Cut Lines (in.):	0.0

Cancel Back Next

6. In the next dialog box, choose the panelize options you require.
7. Select **Next** to proceed.

FAB 3000 will instantly calculate the best possible fit for all Jobs. If it finds multiple panel array options, it will allow you to select the panel array of your choice.



Use the Select Panel Layout to verify the panel sizing to use.

8. Select **Next** to begin the merge.

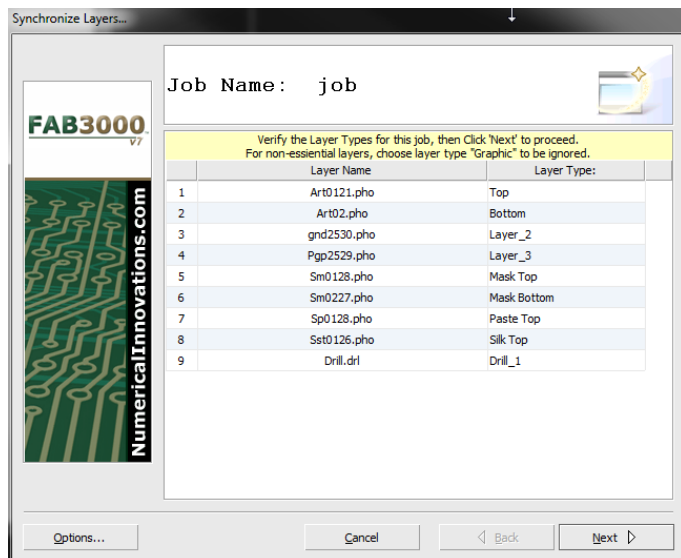
9. Select **Finish**.

You will now see your Jobs merged.

4.2 Synchronizing Layers

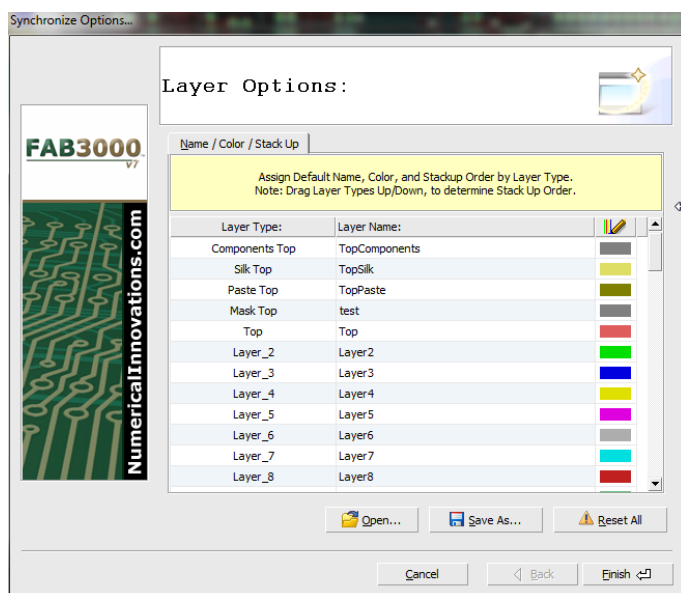
In most cases the final stage of merging Jobs is to perform Synchronize Layers. **Synchronize Layers** merges all Job layers into a common set of layers.

Usually each Job has a different layer names used to describe the same layer types for example, "top02.pho", "topside.gbr", or "art01.ger". Before exporting, all layers of similar layer type must be synchronized into one layer for each type. To synchronize layers:



1. Select **Setup > Synchronize Layers**. A synchronize layers wizard box will appear.
2. Define all layer types of the child Job, and select **Next**.

Synchronize Layers requires that you define the layer types for all Jobs. If you are unsure, choose layer type "Graphic" to be ignored.



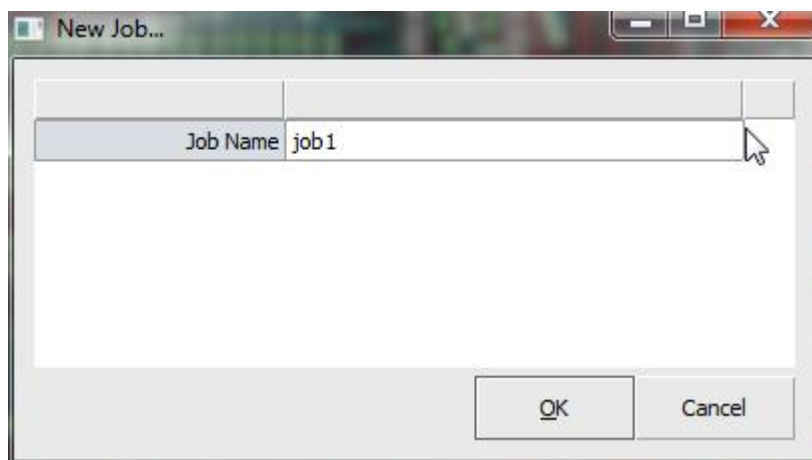
Use the Options button in the bottom right corner to set Layer Options such as Name, Color, and Stackup Order by Layer Type.

3. **Repeat** steps 1-2 for each child Job.

4.3 Manual Job Merge Using Inserts

Once you have created different Jobs for each PCB design, you can merge these Jobs using the power of hierarchy with Inserts. To do this, you must have created at least two or more Jobs.

CREATING A NEW “PARENT” JOB



1. Select **File > New > Job**

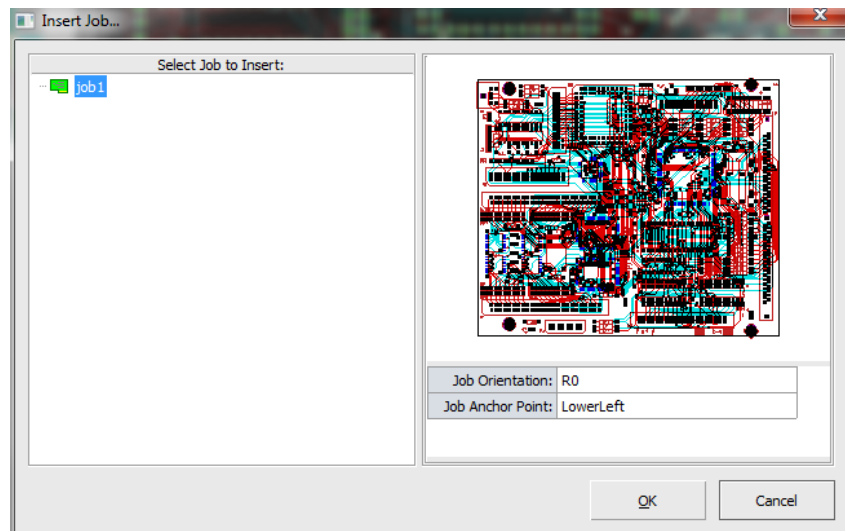
2. Choose a name for the new Job. Be sure that the name is as descriptive as possible.

With the new Job created, a blank Job editor screen will appear. Remember, this new Job is considered the “parent” Job and contain Job Inserts.



[Video: Using Copy to Clipboard and Paste Gerber into New Job](#)

ADDING INSERTS MANUALLY



1. Select **Job > Insert Job**. The Insert Job dialog box will appear.
2. Select the Job to Insert. A preview window will display the selected Job or panel you want to Insert.
3. You may choose to assign **Job Orientations** of *R0 (rotate 0)*, *R90 (rotate 90)*, *R180 (rotate 180)*, *MX (mirror horizontal)*, *MY (mirror vertical)*.

4. You may choose to assign **Job Anchor Points** which controls the Insert origin for the Insert.
Job anchor points are listed as *Lower Left*, *Center*, and *Origin (actual 0,0 location of Inserted Job)*. Choosing the correct anchor point helps when Inserting Jobs.
5. Select **OK** to add the Inserts into the editor.

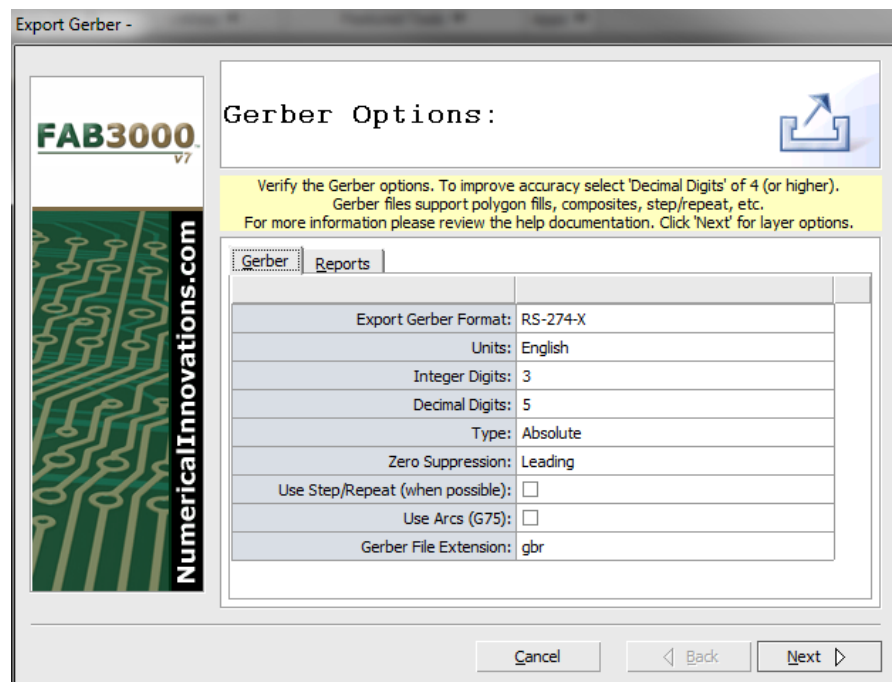
To Insert other Jobs, repeat steps 1-5. Once the other Jobs are added, you may also edit their orientation, location, and make copies. FAB 3000 can handle an unlimited hierarchy.

Once all Jobs have been inserted into the “parent Job”, you must synchronize (merge) layers as explained in **5.2: Merge Layers**.

4.4 Exporting a Parent Job to One Set of Gerber & Drill Files:

Once all Inserts are laid out into an optimal location, you can now export your merged CAM data.

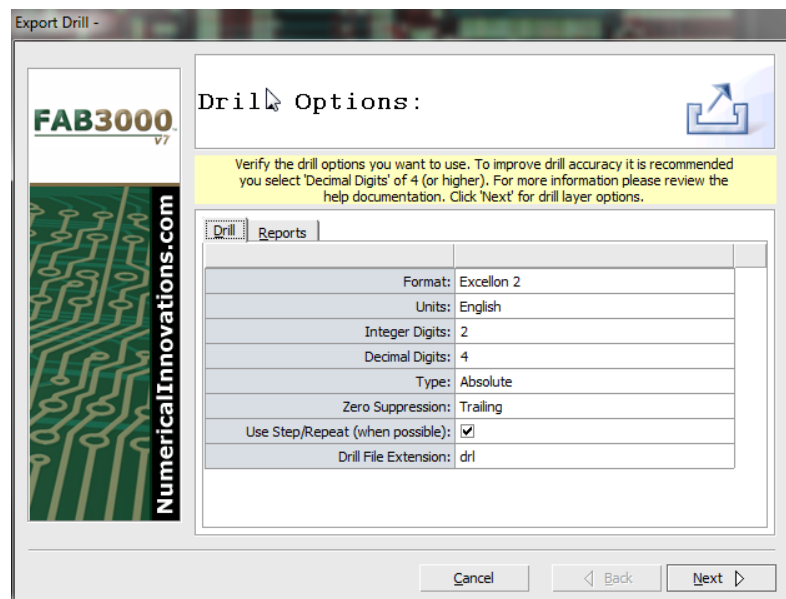
EXPORTING GERBER FILES:



1. Select **File > Export > Gerber**.
2. Verify Gerber settings and select **Next**.
3. Verify Gerber Layer Options and select **Next**.

If you do not want to export certain layers, simply uncheck the corresponding layer. You may also choose to output individual layers with Mirror or Negative polarity attributes.

4. Choose the directory where the Gerber files will be written to. If the directory does not exist, FAB 3000 will create it for you.



EXPORTING DRILL FILES:

1. Select **File > Export > Drill**.
2. Verify Drill settings and select **Next**.
3. Verify Drill Layer Options and select **Next**.

If you do not want to export certain layers, simply uncheck the corresponding layer.

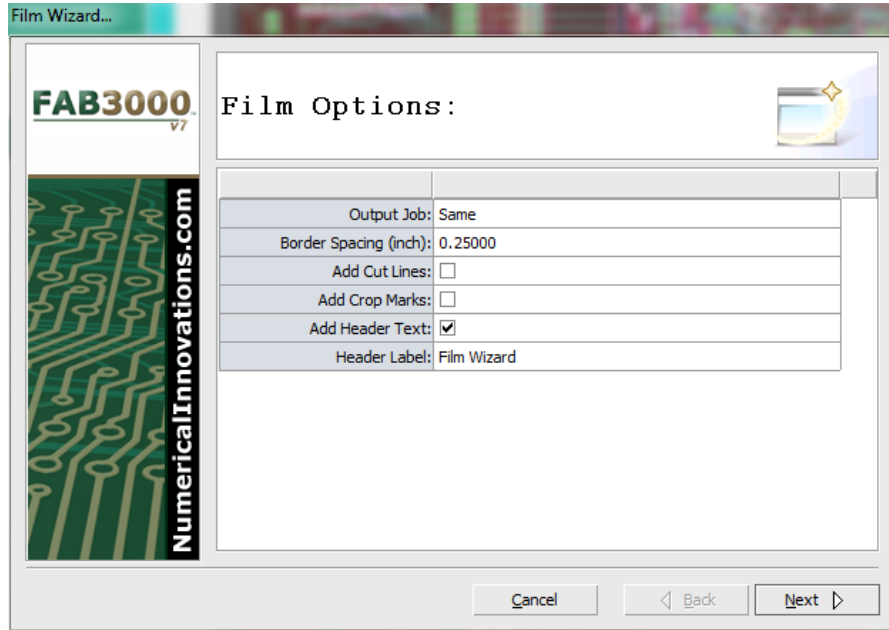
4. Choose the directory where the Drill files will be written to.

If the directory does not exist, FAB 3000 will create it for you.

You have now created merged data from multiple printed circuit board designs into a single set of CAM files.

4.5 Using Film Wizard

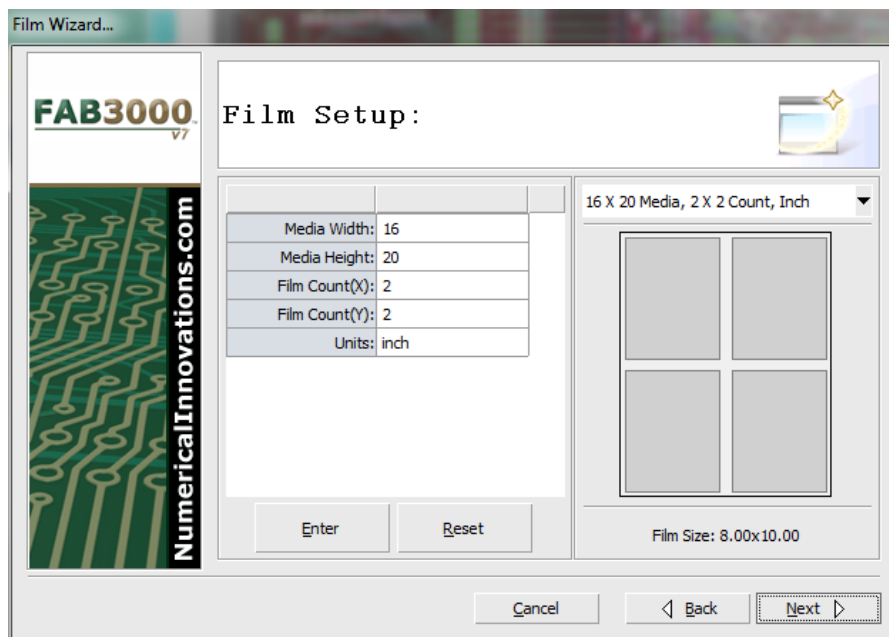
Film wizard can be used to automatically setup Gerber layers for optimal film output making it ideal for anyone requiring Gerber or Postscript photoplots from their artwork. This plug-in is included with FAB 3000 and will **reduce photo plotting expenses by as much as 50%; providing a fast return on investment.** Additionally, Film Wizard will also eliminate operator errors by preventing someone from wrongly mirroring or changing polarity and result in faster turn-around times.



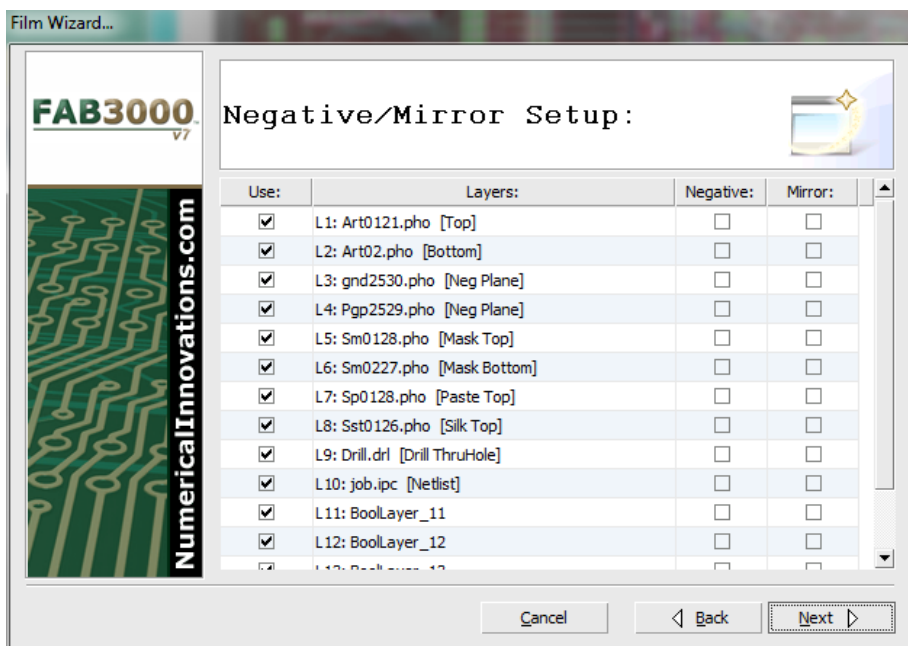
1. Go to **Tools > Film Wizard**.

Assuming you have an 8 layer design, with an area of 5"X7" (120mm X 175mm). Choose a film size that will be able to hold at least 4 design layers. In this example a film size of 16" X 20" will work perfect. (You may

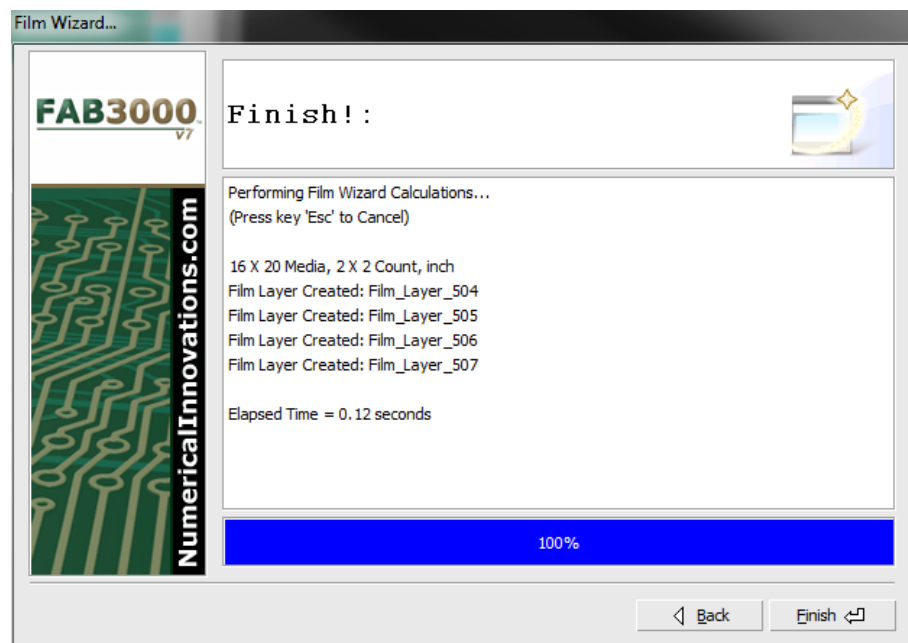
Make sure to contact your photoplotting service to determine what acceptable film sizes to use for their photoplotting process.



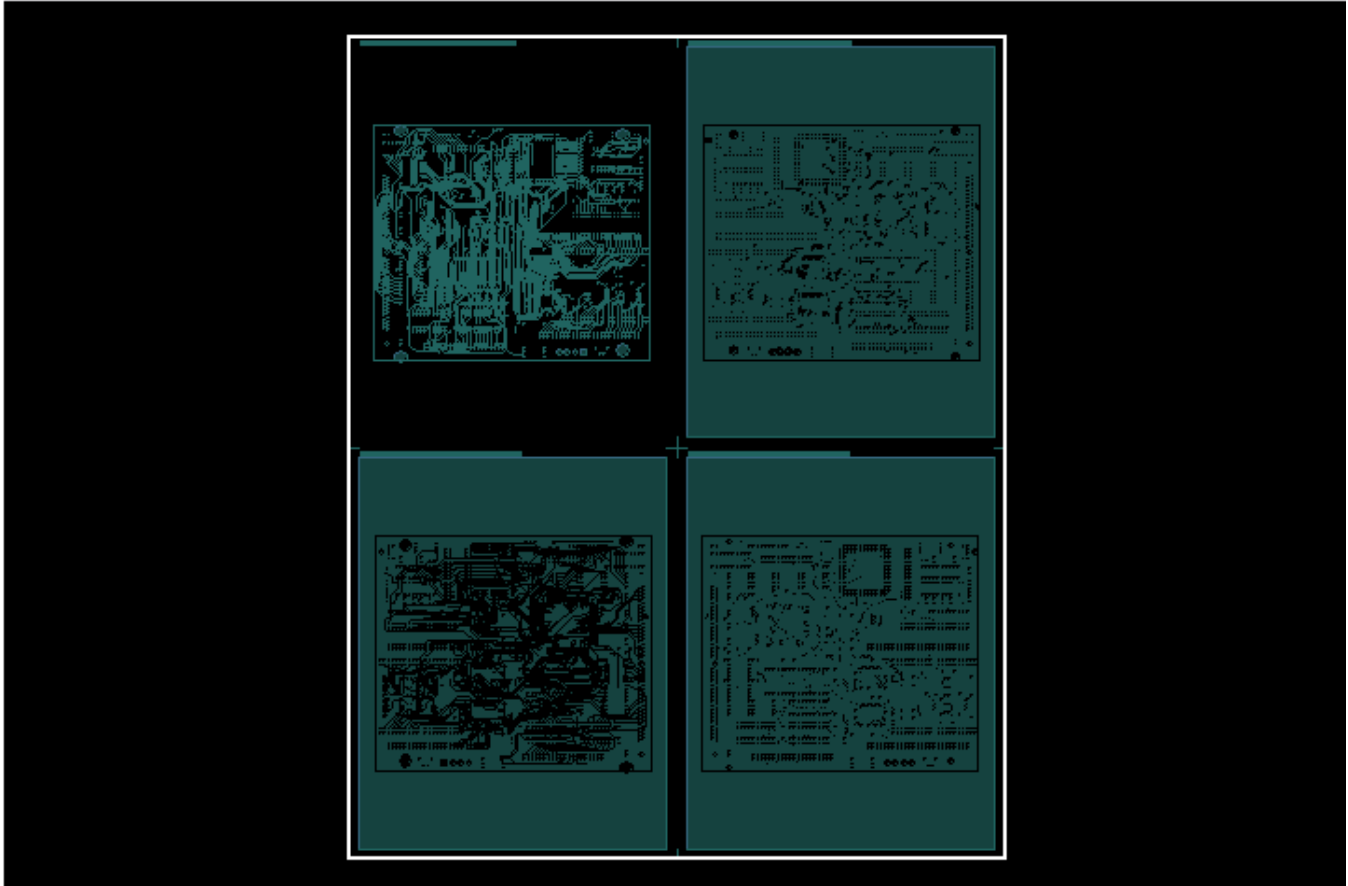
2. Assign options that you want to appear on each film layer.
3. Select Film Setup based on your required layer design and sizing. Select **Next**.



4. Assign polarity & mirror options for each individual layer. Select **Next**.



You're finished! On the next page is an example screen capture of the outputted layer(s) from Film Wizard. Notice that mirror and rotation have been included automatically!



At this point all you need to do now is to export these "Film Wizard" layers to either Gerber or Postscript, and that's it!

You've just saved 50% on your photoplotter service expenses!

4.6 Using DFM Checker

Prior to sending your PCB out to manufacturing, examination of your Gerber design is always required. Each detail should be scrutinized and checked for any manufacturing violations. This saves time, money, and frustration. Using FAB 3000's DRC/DFM Checker, built in analysis tools will check your design for any potential problems. Many CAD systems don't produce a "perfect" representation of your design once outputted. The Gerber/NC files are what is most important to manufacture your boards. FAB 3000 goes beyond basic checks (minimum trace & space) to perform the following:

Silkscreen over Solder Mask - Locate if there are any locations where the silkscreen overlaps with openings in the solder mask layers. **Elevated Risk:** May cause contamination of the pads and make electrical testing plus SMT soldering more difficult.

Non-Functional Internal Pads - Essentially these are unconnected internal pads typically found on plane layers. **Elevated Risk:** May break loose and cause potential drilling problems, risk of shorts, and more.

Isolated Connections - One of the most common problems is to determine whether your PCB CAD system has properly outputted good plane connections to the copper plane. **Critical:** A board will fail if any plane connections is not properly connect the copper plane.

Mask Slivers - Areas in the solder mask where the resist is so narrow that it may cause small pieces of the resist to flake off and present soldering problems later. **Critical:** A board can fail if the resist falls in an area that needs to be soldered later on.

Power/Ground Shorts - Determine any potential shorts between Power and Ground planes. **Critical:** A board will fail if there is a short between the power and ground planes.

Copper Slivers - Narrow areas in the copper where the potential for over-etching is great. This typically happens when trace angles are 30 degrees (or less), when composites are used, and on plane layers. **Critical:** A board can fail if certain copper area are over-etched. Resulting in shorts and decrease in board yield.

Solder Bridges - Typically when the mask layers, contain openings which are too large, and may expose other adjacent conductive areas. **Critical:** During fabrication, copper may get too close and form a bridge to other adjacent conductive areas.

Negative Plane Thermal Conflicts - Determine if any Via or plated hole is connected to more than one negative plane together. **Elevated Risk:** May result in poor board performance.

Starved Thermals - Check whether a thermal connection to the copper plane has been constricted by overlapping or adjacent data that is way too close. **Critical:** May result in poor board performance, short, or other undesired effect.

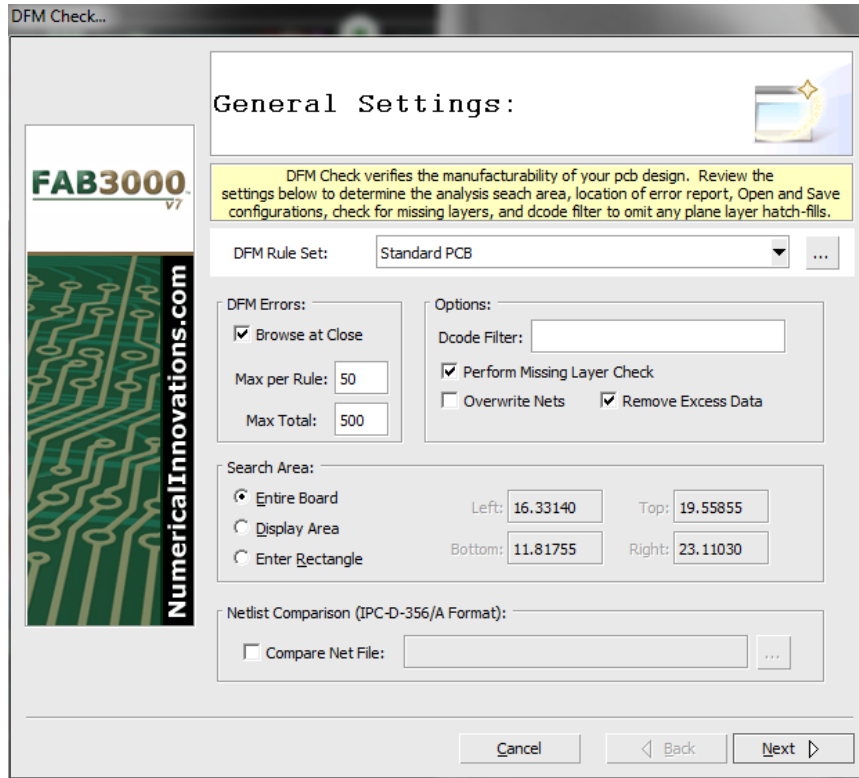
Find Acid Traps - Due to the surface tension during the etching process, larger deposits of acid may get trapped into certain areas - resulting in over-etching. **Critical:** Over-etching will directly affect your board yields.

Antennas / Dangling Traces - Search for Non-terminating traces which may behave as an antenna. **Elevated Risk:** May result in poor board performance, short, or other undesired effect.

There are several conditions to consider prior to using DFM Checker:

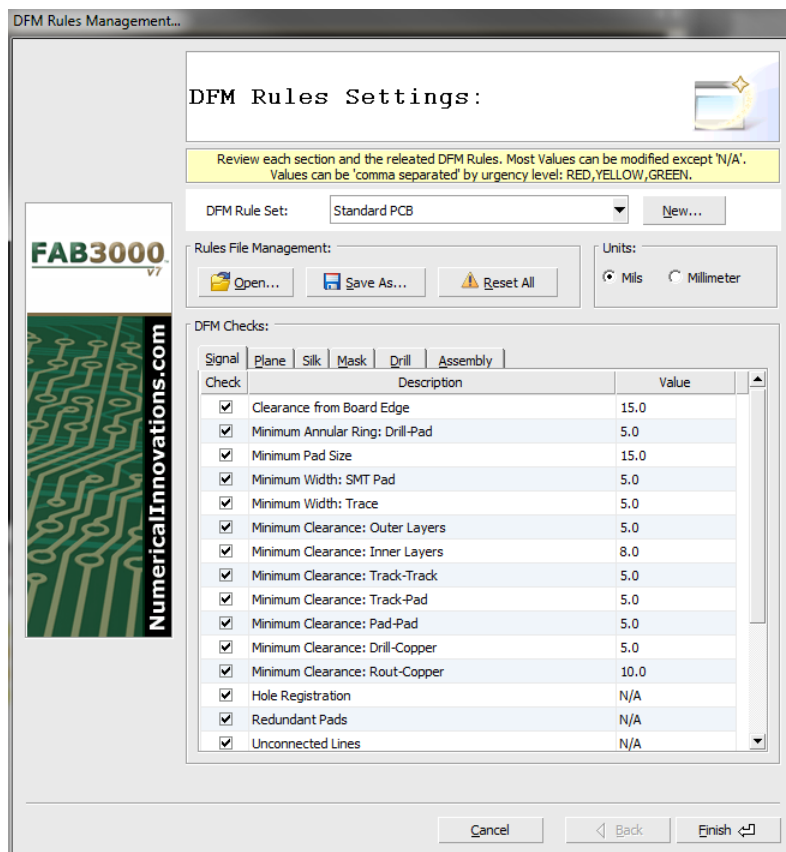
1. DFM Checker requires that you extract at least once Netlist before proceeding. However, if you haven't run a Netlist extraction, DFM Checker will perform it for you first.
2. All layer types must be defined prior to use DFM Checker.
3. You must have at least 1drill file (2 or more for blind/buried Vias).

USING DFM CHECKER



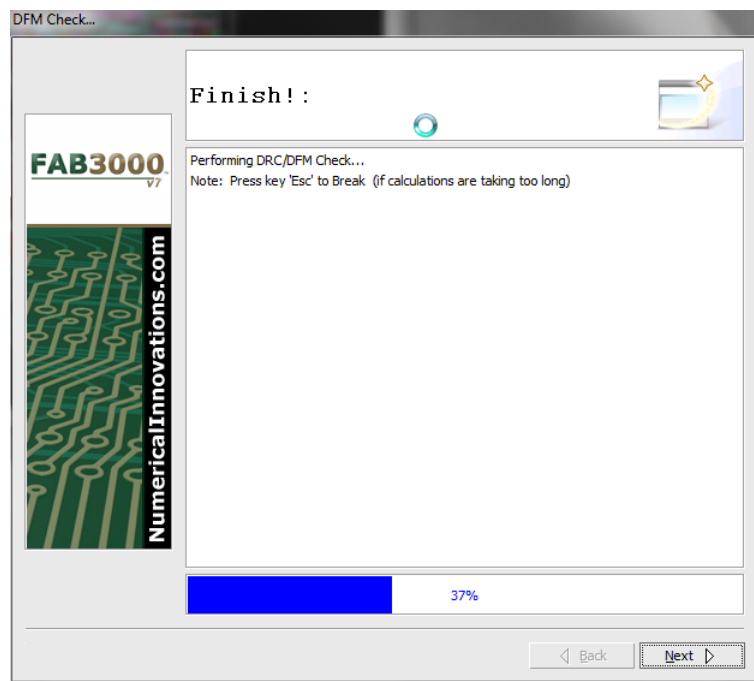
1. Select **Tools > DFM > DFM Check**. The DFM Check dialog box will appear.
2. Verify and edit the DFM rules set. To do this, select the available rules set from the pull down menu under DFM Rule Set.

You may also edit the DFM Rules by pressing the “...” button next to the DFM Rule Set drop down menu (See below, **DFM Rules Settings**).

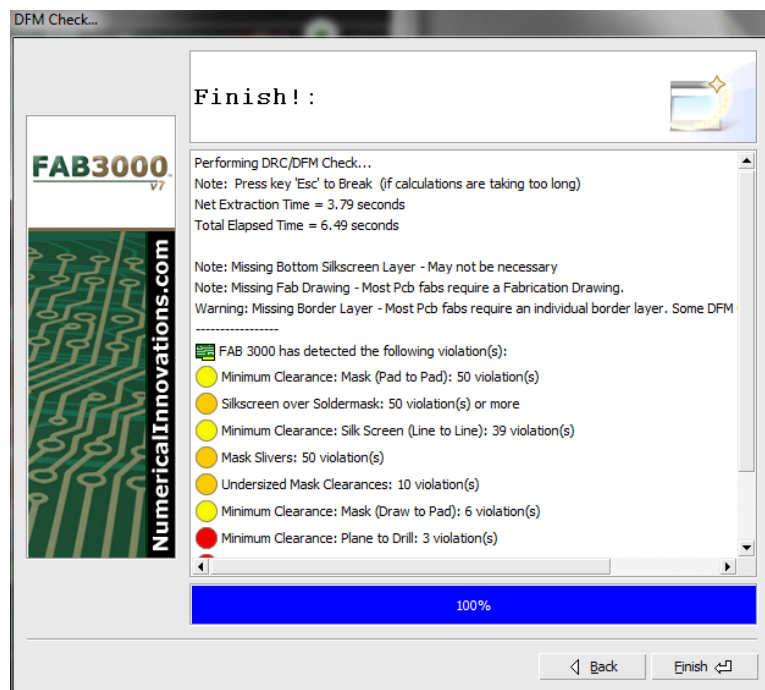


DFM Rules Settings dialog is used to verify or edit rules individually. You can also access DFM Rules Settings by selecting **Tools > DFM > DFM Rules Management**.

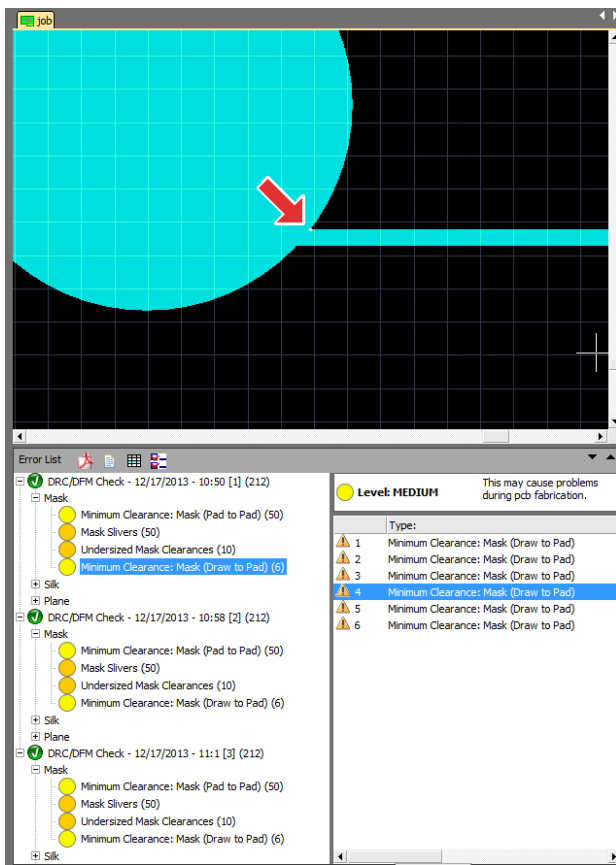
3. Select **Next**.
4. FAB 3000 will now perform an extensive analysis of your PCB design.
 - a. Note that Job Inserts are ignored. If you wish to run a DRC/DFM check on a specific child Job, make sure to open the child Job in a separate window, and then proceed. To open Jobs, go to **Window > Workspace Browser**.



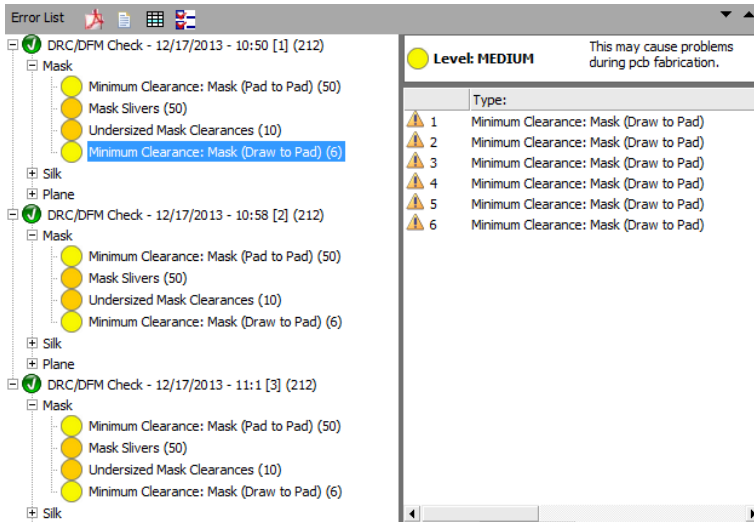
- b. Depending on the complexity of the design and the checks performed, processing time can take anywhere from 30 seconds to 5 minutes.



5. If any errors are detected, FAB 3000 will display them in the dialog box. If there are no errors, the process is complete.
6. Select **Finish**. Any errors listed will now appear in the DFM Violation Browser. The browser will appear just below the editor and provide you with an easy to use navigation tool to locate and describe each detected error. For a more complete overview of the DFM Violation Browser, see section **4.7: Viewing DFM Violations with the Error Browser**.



7. The browser contains a complete list of errors organized in a hierarchy format. Selecting and highlighting each error will provide a description of the error and severity level for PCB fabrication.
8. For each error selected, you'll be taken to the location of that error within the drawing for closer viewing and editing.



9. Create reports in PDF, Text, or CSV format by selecting the appropriate icon next to Error List.
10. You can also generate an error drawing which may be exported to various formats such as Gerber, DXF, or ODB++ and then imported back into your PCB layout tool for verification or to share with colleagues.



[Video: Performing Simple DFM/DRC Using FAB 3000](#)

4.7 Saving DRC/DFM Violations into PDF Format

To compile and save a list of DRC/DFM violations into PDF format after a DRC/DFM Check has been run, perform the following:

1. Select **Info > Generate PDF Error Report**.
2. Assign a filename to the report and Select **Next**.
3. Once the PDF is ready, select **View PDF File**.

A PDF of all warnings will launch in Adobe Acrobat.

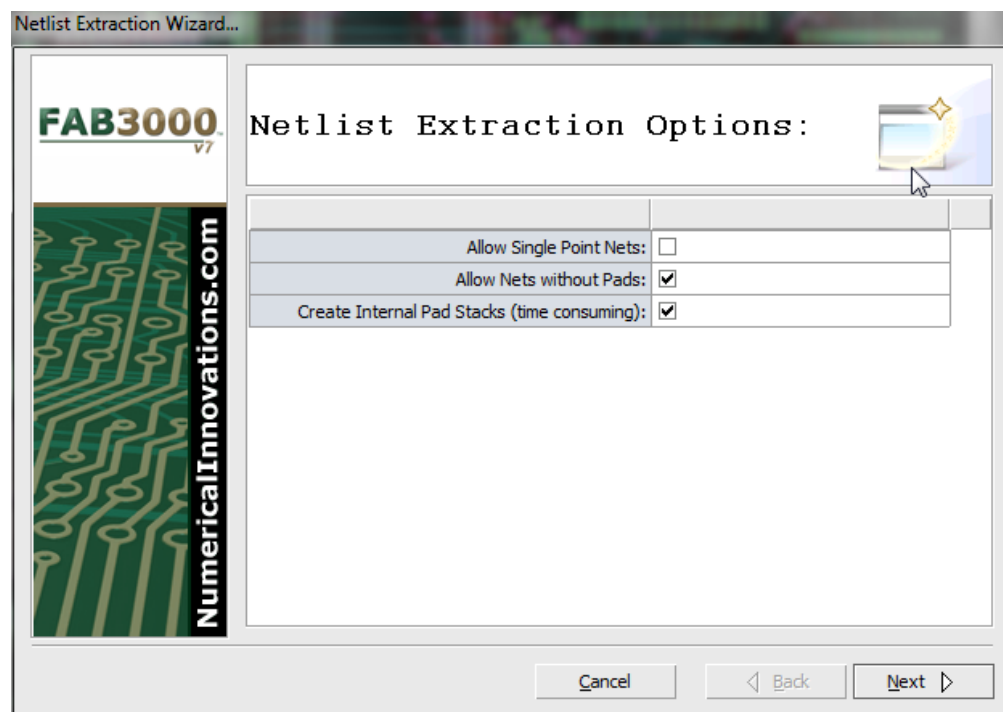
4.8 Extracting Netlists from Gerber and/or Drill Data

After importing Gerber and/or Drill files into a workspace, a Netlist should be extracted in order to do a comparison against a CAD-generated IPC-D-356 Netlist. There are several requirements for extracting Netlists:

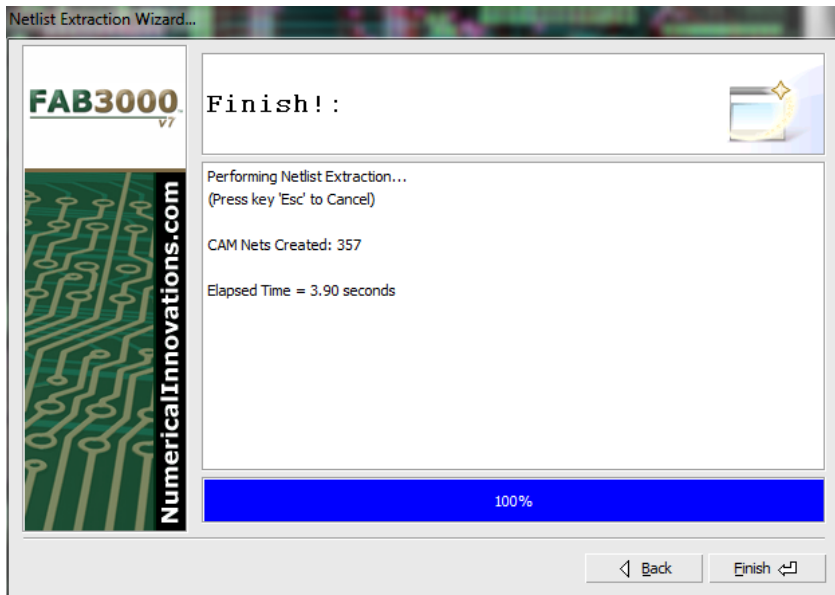
1. All layer types should be defined before extracting a Netlist.
2. At least one drill file (2 or more for blind/buried Vias) is required before extracting a Netlist.
3. If your design contains blind or buried Vias, see **3.10, Extracting with Blind and Buried Vias**.

To extract a Netlist from Gerber and/or Drill files, perform the following:

Extract a true interconnectivity map of any PCB design.



1. Select **Tools > Nets > Netlist Extraction**.
2. Verify and select extraction options.
3. Select **Next**. The extraction will be performed.



4. In the **Finish** dialog box, information will be included regarding CAM Nets created and elapsed time.

The total number of extracted CAM Nets will be displayed in the **Finish** section.

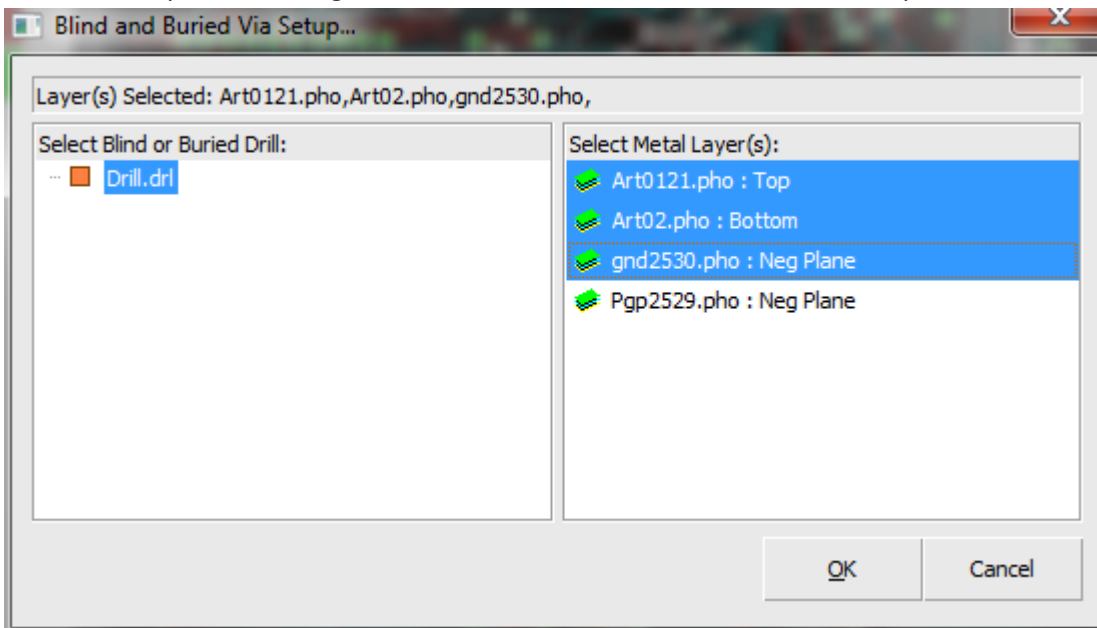


[Video: Extract Netlist from Gerbers and View Nets using FAB 3000](#)

4.10 Extracting Netlists from Blind and Buried Vias

Assuming all prerequisites are complete prior to extraction (all layers types must be defined and you must have at least 2 or more drill files), you may also extract a Netlist from designs which contain blind and buried Vias.

The first step in extracting Netlists from blind and buried Vias is to setup **Blind and Buried Drill**



Relationships.

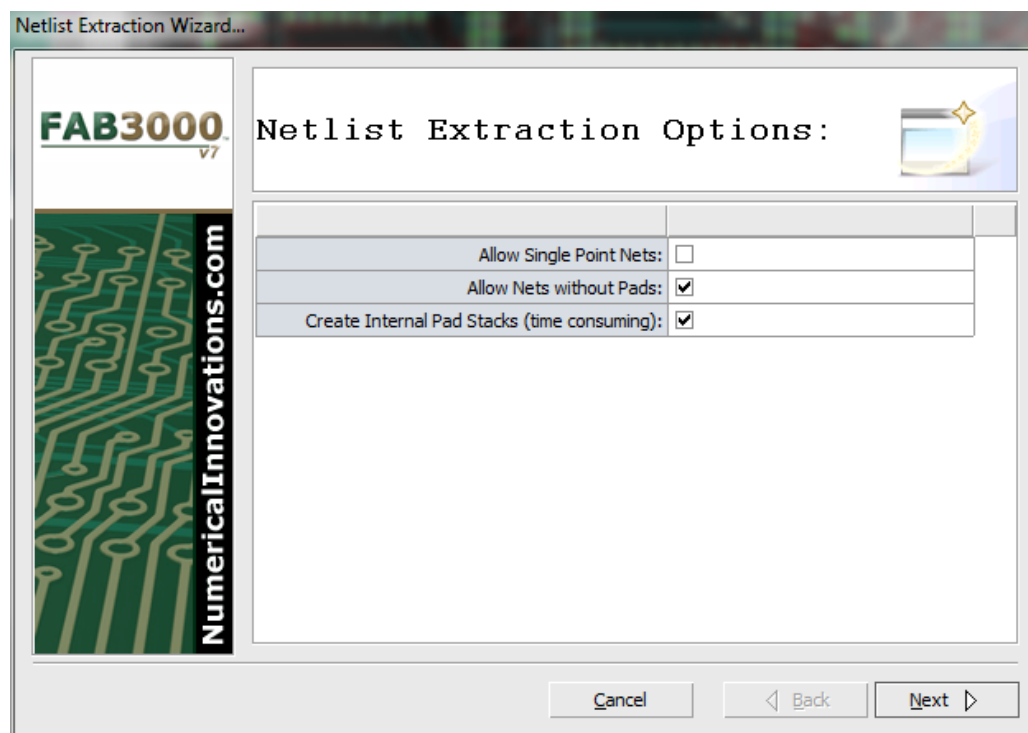
To do this, perform the following:

1. Select **Setup > Blind and Buried Via Setup**.
2. Select and highlight the associated layers to form relationships.
3. Select **OK**.

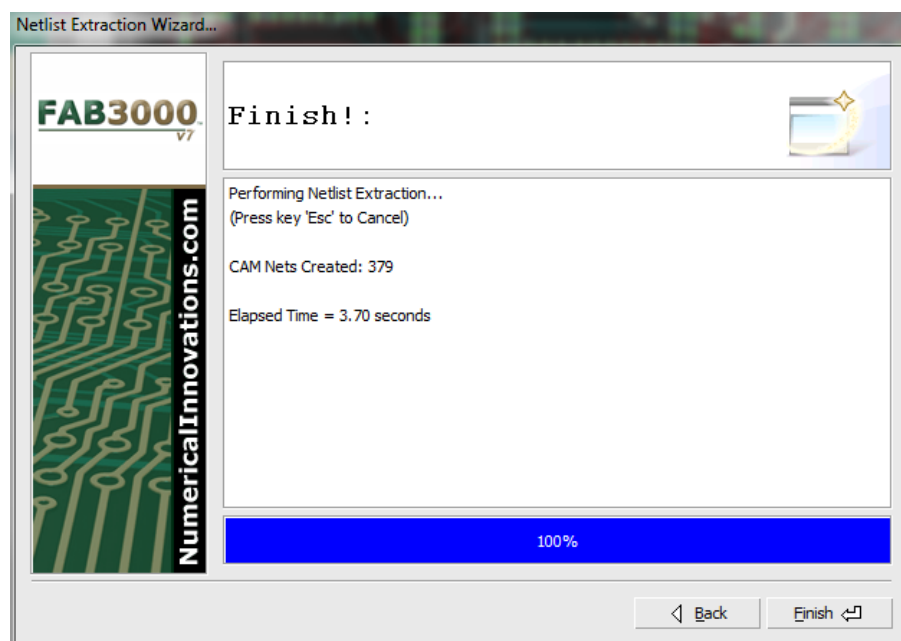
4. You may now proceed with Netlist extraction.

PROCEED WITH NETLIST EXTRACTION FROM BLIND AND BURIED VIAS

1. Select **Tools > Nets > Netlist Extraction**. The Netlist Extraction wizard will appear.



2. Verify Netlist Extraction Options.
3. Select **Next**. FAB 3000 will perform the Netlist Extraction.



4. The total number of CAM Nets will be displayed.
5. Select **Finish**.

To compare the extracted Nets against your own CAD generated Netlist, see **4.11: Comparing a CAM Netlist against an External Netlist**.

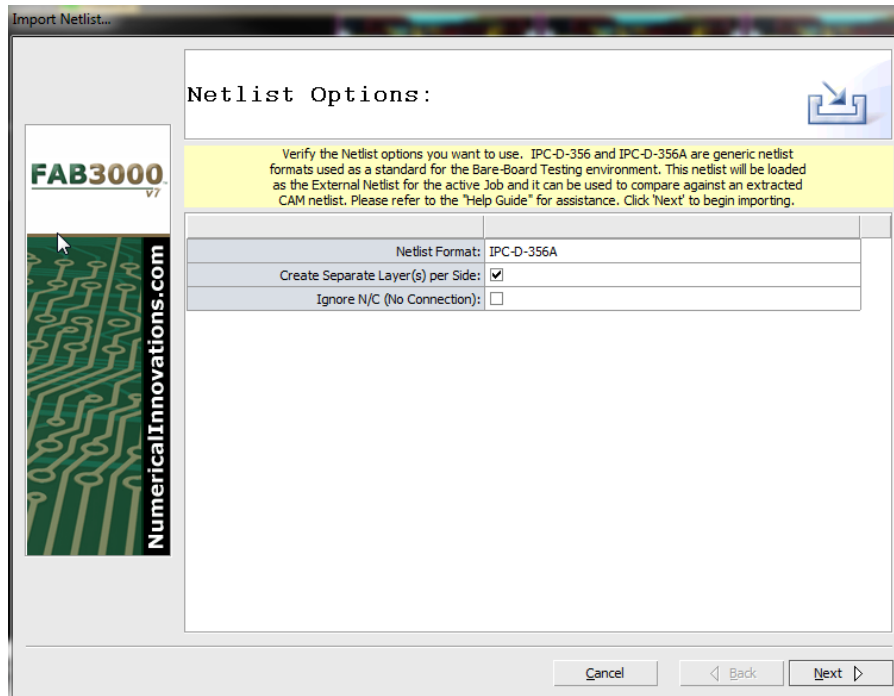


[Video: Assign Layer Types & Stackup, Assign Blind & Buried Drills](#)

[Video: Sort Layers and Assign Blind/Buried Vias using FAB 3000](#)

4.11 Comparing a CAM Netlist against an External Netlist

In order to compare a CAM Netlist against your external Netlist, the external Netlist (from your CAD system) must be in a standard format such as IPC-D-356 or IPC-D-356/A.

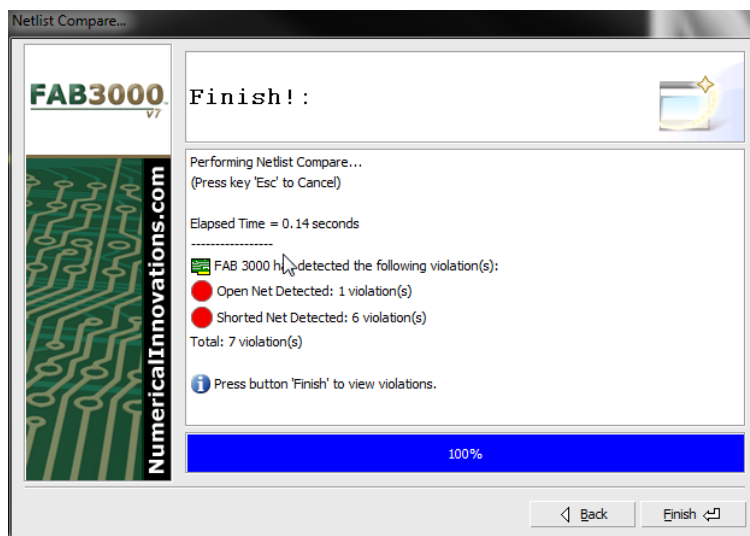


IMPORTING AN EXTERNAL NETLIST

FAB 3000 can import IPC Netlists which are generic Netlist formats used as a standard for the bare-board testing environment. The Netlist will be loaded as the External Netlist for the active Job and it can be used to compare against an extracted CAM list. To import a Netlist:

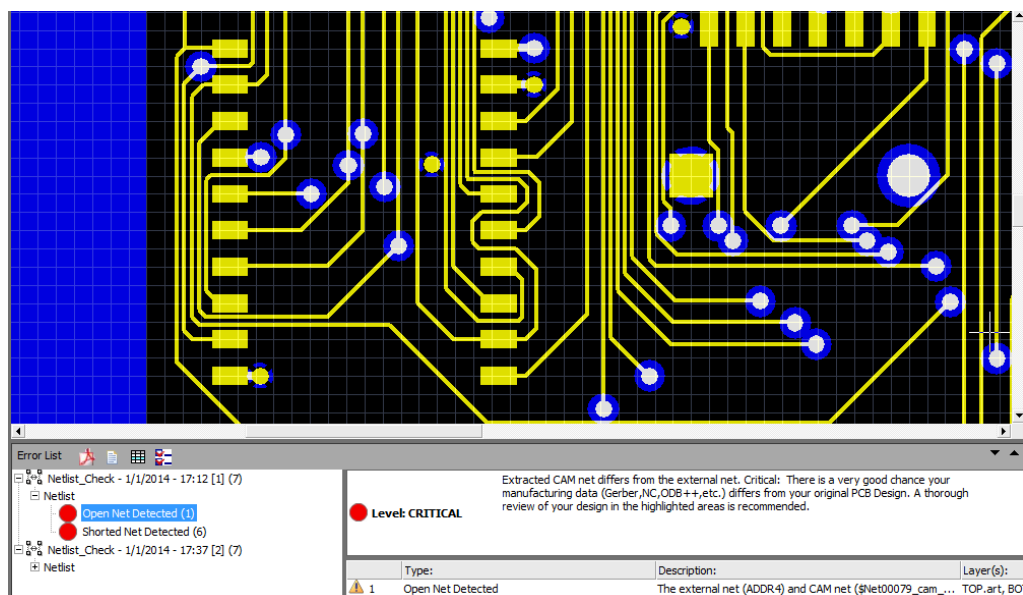
1. Select **File > Import > Netlist (IPC-D-356/A)**.
2. Select the Netlist file you want to import. Select **OK**.

3. Choose the Netlist Format (IPC-D-356 or IPC-D-356A) you wish to import from the Import Netlist Wizard dialog box.



4. Select **Next** to begin the import.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. An example of a Netlist comparison error is pictured here. Upon selecting Finish, FAB 3000 will immediately launch the Netlist error browser for further investigation.

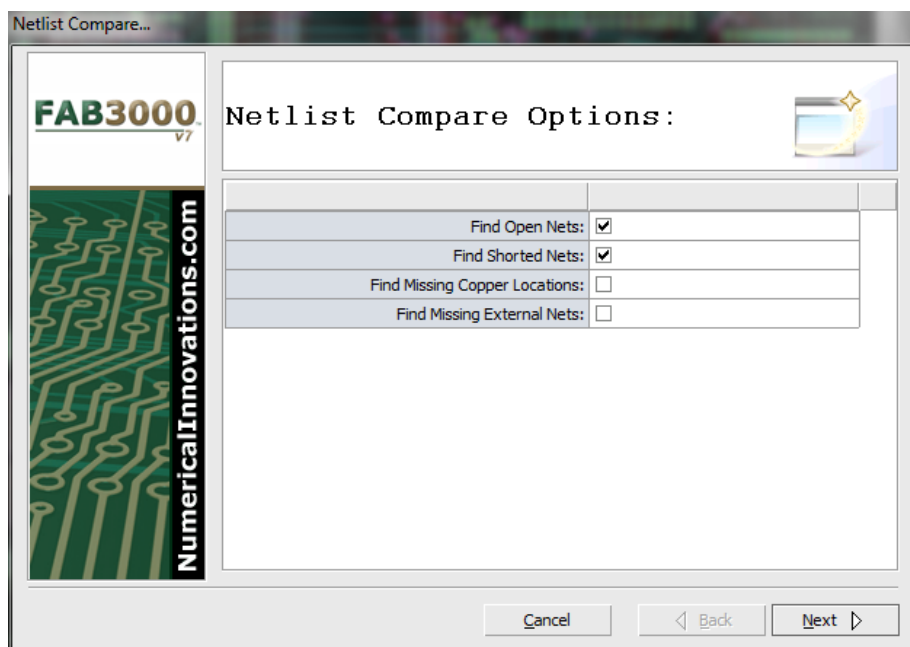


Example of Netlist error browser.

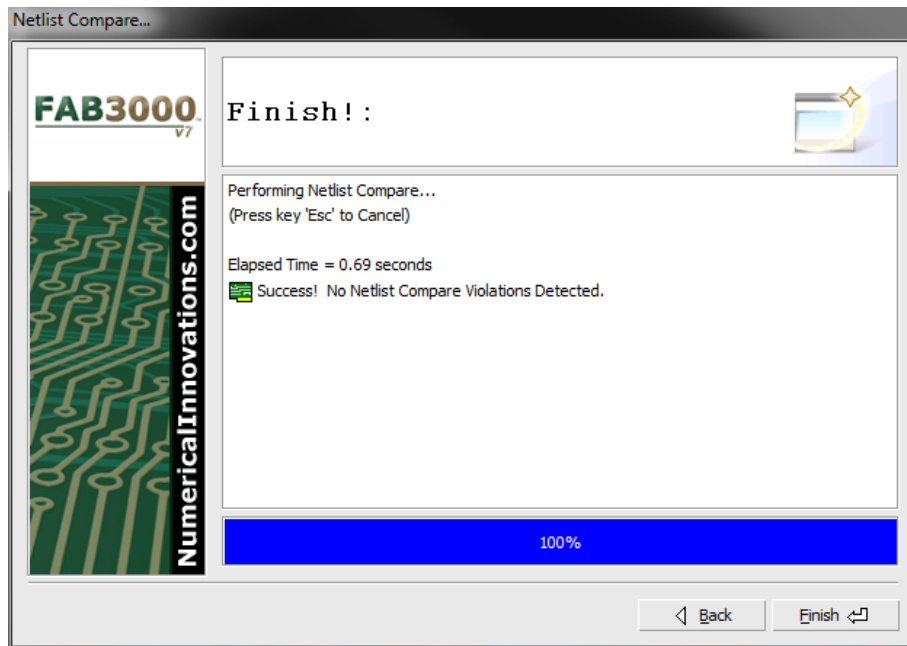
RUNNING THE NETLIST COMPARISON

Compares extracted nets against an imported Netlist.

The purpose of Net Compare check is to verify whether the PCB design has been constructed correctly (per the Netlist definition) and to verify the PCB layout tool has generated the Gerber files correctly. Netlist compare is used to verify that the Gerber files have been successfully outputted from PCB Layout and imported into CAM software.



1. Select **Tools > Nets > Netlist Compare**.
2. Verify and select compare options.
3. Select **Next**. The comparison will be performed.



4. In the **Finish** dialog box, information will be included regarding comparison time and any violations that were detected during the comparison. If there were errors, you will be prompted to select Finish. FAB 3000 will automatically launch the Netlist error browser to easily identify and fix Netlist errors.

4.12 Creating a PDF Report for Net Comparison Errors

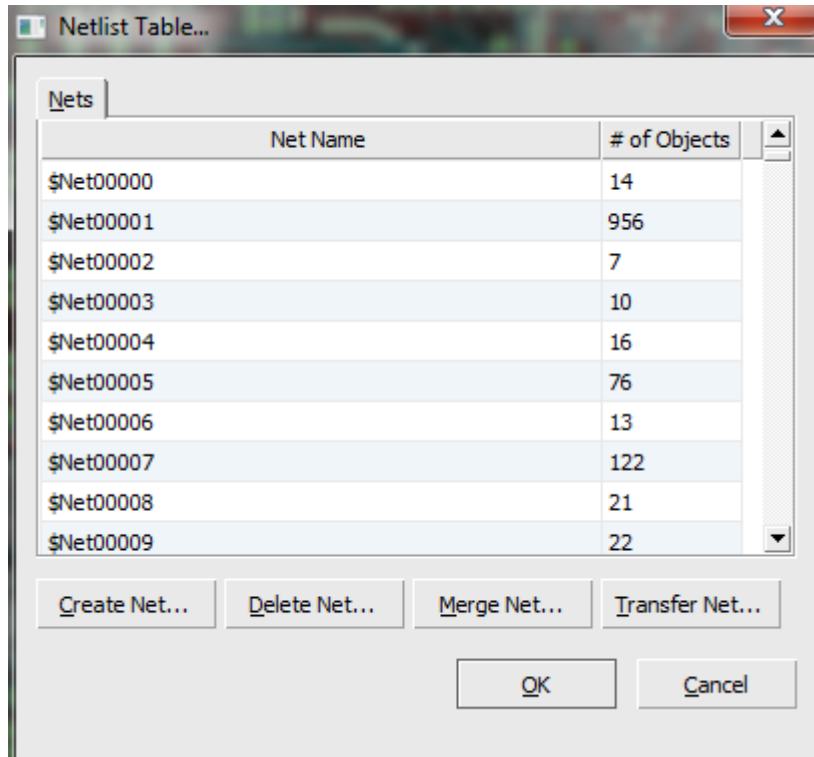
All Netlist comparison errors can be documented and organized into a PDF file after running the comparison. To document in PDF, perform the following:

1. Select **Info > Generate PDF Error Report**.
2. Assign a filename to the report and Select **Next**.
3. Once the PDF is ready, select **View PDF File**.

A PDF of all errors will launch in Adobe Acrobat.

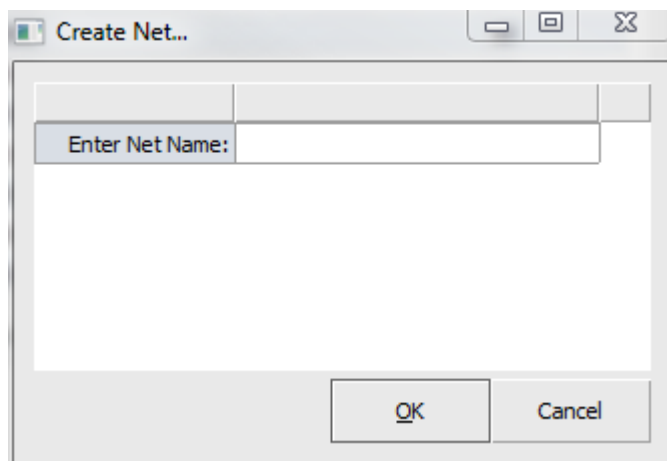
4.13 Add/Remove Objects from a Net

After extracting a Netlist, you can edit objects from a Net. This can be done within the Netlist Table.



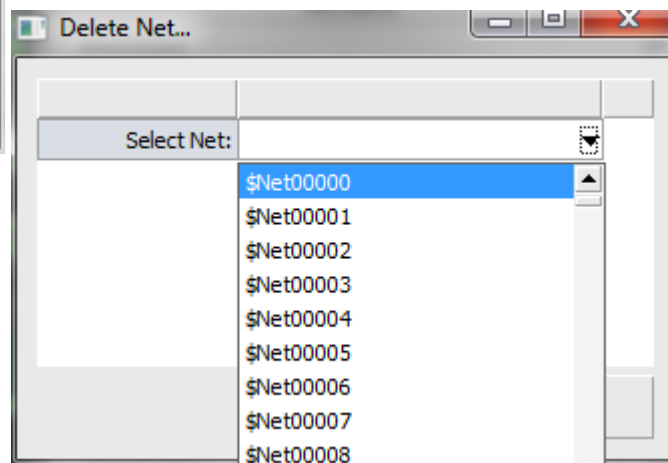
1. Select **Tools > Nets > Edit Nets**.

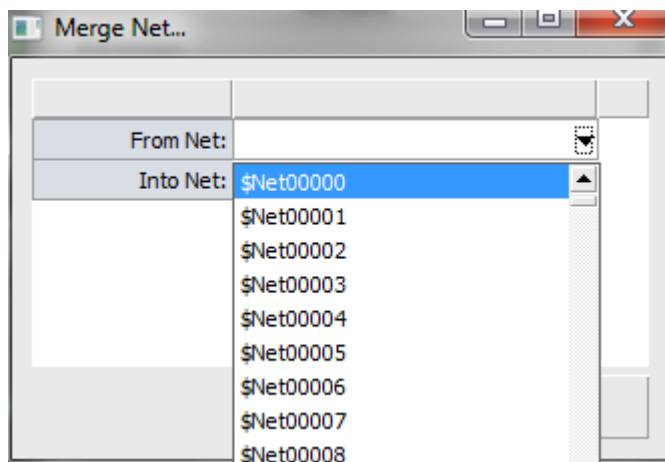
2. There are several features used to edit objects from a Net. This includes **Create Net, Delete Net, Merge Net, and Transfer Net**.



Create Net - Add a new net to the Job. Select Create Net to enter a Net Name.

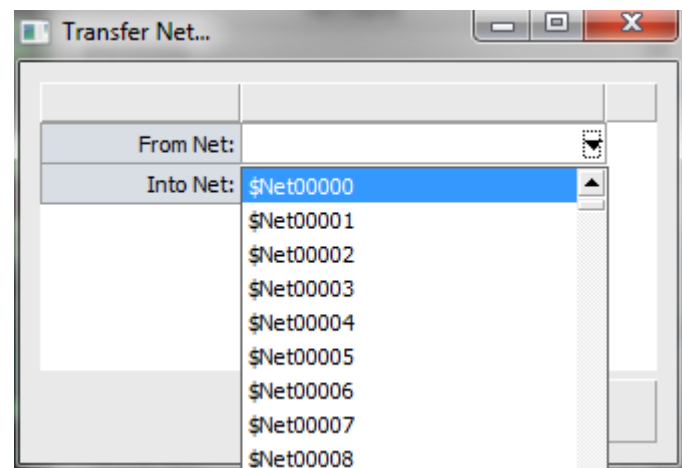
Delete Net - Remove Net from Job. In the dropdown menu, select a Net to delete.





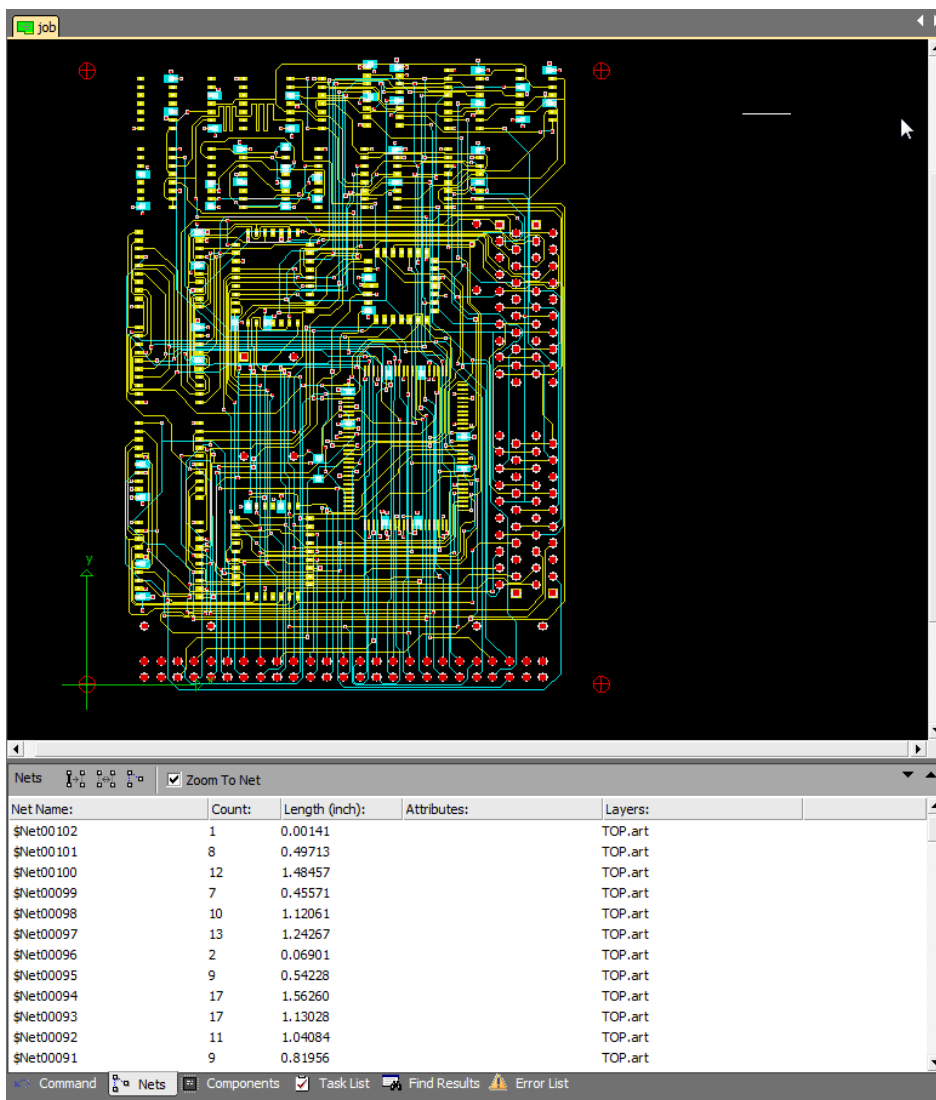
Merge Net - Combine Nets. Select the From Net and Into Net via dropdown menus.

Transfer Net - Copy contents from one net to another.



4.14 Applying External Net Names to CAM Net

After a Netlist is extracted from Gerber & Drill data, there is no way for FAB 3000 to know the net names used in your CAD design. This allows you to import your CAD-generated Netlist, and apply the names to FAB 3000. After importing a Netlist into FAB 3000, perform the following:

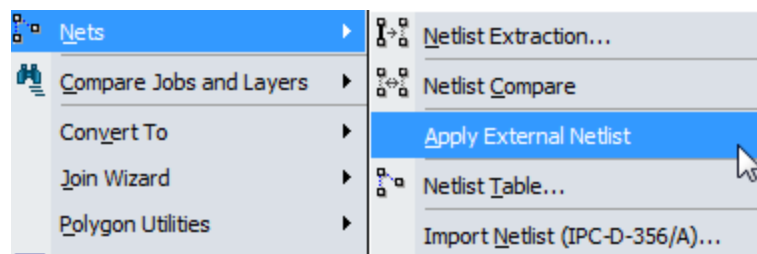


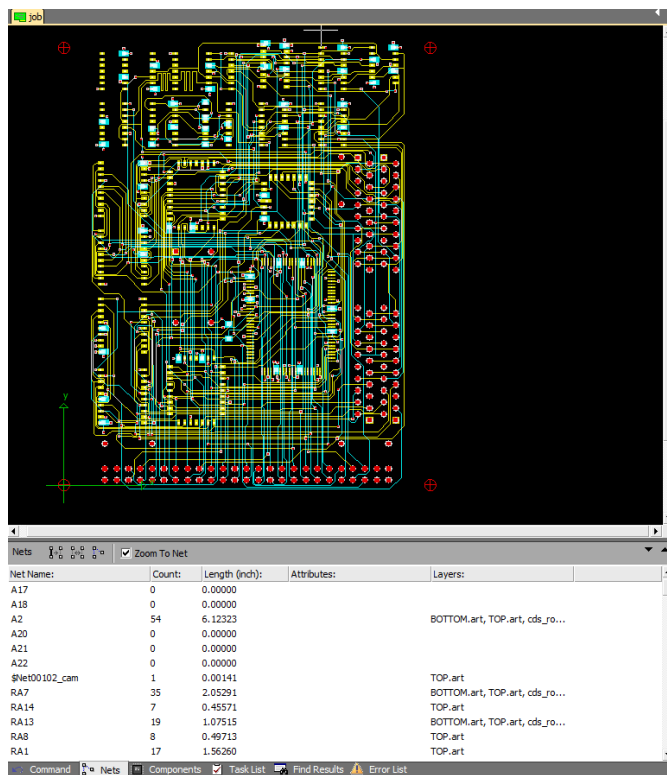
1. Extract a Netlist using **Tools > Nets > Netlist Extraction** (see 4.8: **Extracting Netlists from Gerber and/or Drill Data**). The extracted Netlist is pictured here. The Netlist information can be found at the bottom browser tab (below the editor) under “Nets”.

You’ll notice FAB 3000 generates a generic Net name beginning with “\$Net#”. Usually, designers will want to use the actual design net names in FAB 3000. In this case, they will use **Tools > Nets > Apply External Netlist**.

How to Apply External Net Names to CAM Net

1. Select **Tools > Nets > Apply External Netlist**.



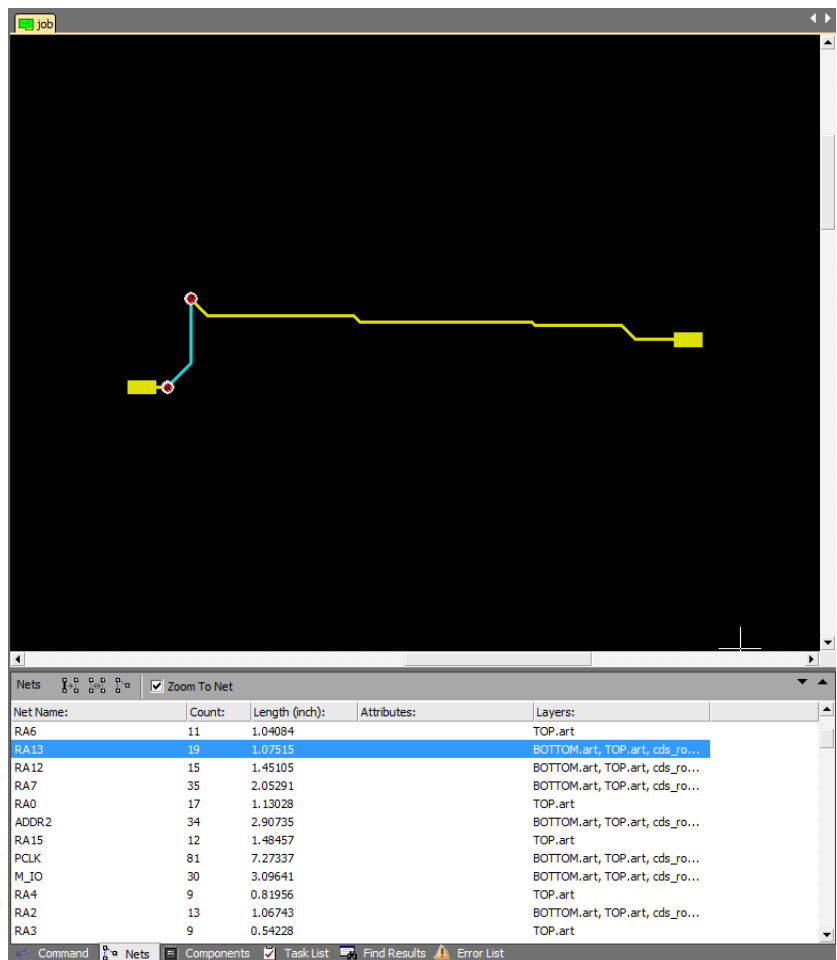


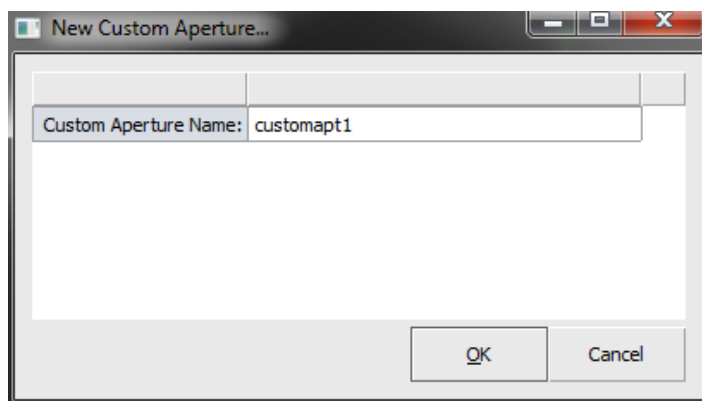
FAB 3000 will automatically update the existing CAM net names with the external net names.

You'll notice now in the Nets section tab that all Net names are no longer generic and refer back to the original names used in the CAD design.

On some occasions you may find Nets in this list which may not have changed after running **Apply External Netlist**. This will most likely be the result of a Net flaw in the design.

To identify this flaw, run **Tools > Nets > Netlist Compare** (see Section 4.11).





4.15 Creating Custom Apertures

Custom apertures are used to optimize Gerber data.

1. Select **File > New > Custom Aperture**
2. Enter Custom Aperture Name and select **Finish**.

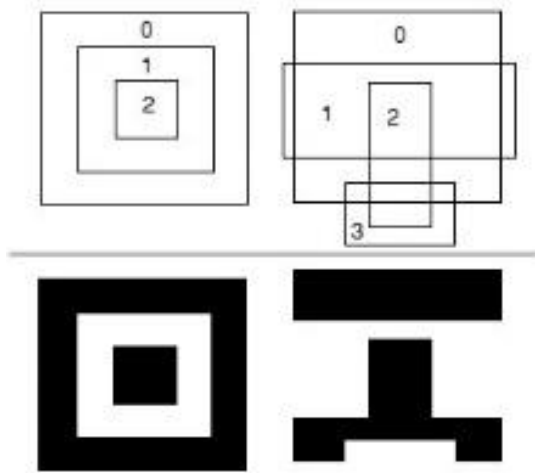
The Custom Aperture Editor will now appear as a separate tab. This is where you'll perform all required drawing and editing steps in order to create a custom aperture.

- a. The custom aperture name must be unique within the active workspace.
- b. The Custom Aperture editor is used to create special shapes that may be used as flashes in the current FAB 3000 workspace. Every FAB 3000 workspace has its own internal custom aperture library. When importing Gerber files that contain custom apertures, FAB 3000 adds them to the internal custom aperture library automatically. Custom aperture libraries may also be maintained as independent files for use from design to design.
- c. The Custom Aperture editor differs from the CAM Editor in a few ways. There is only one generic layer. Custom apertures may contain both positive (dark) and negative (clear) data. For example, a donut is created by having a positive outer circle with a negative inner circle. Positive and negative data can be stacked in any order, as long as you assign the correct composite level (located in the toolbar).



[Video: Creating Custom Pads using Custom Apertures – FAB 3000](#)

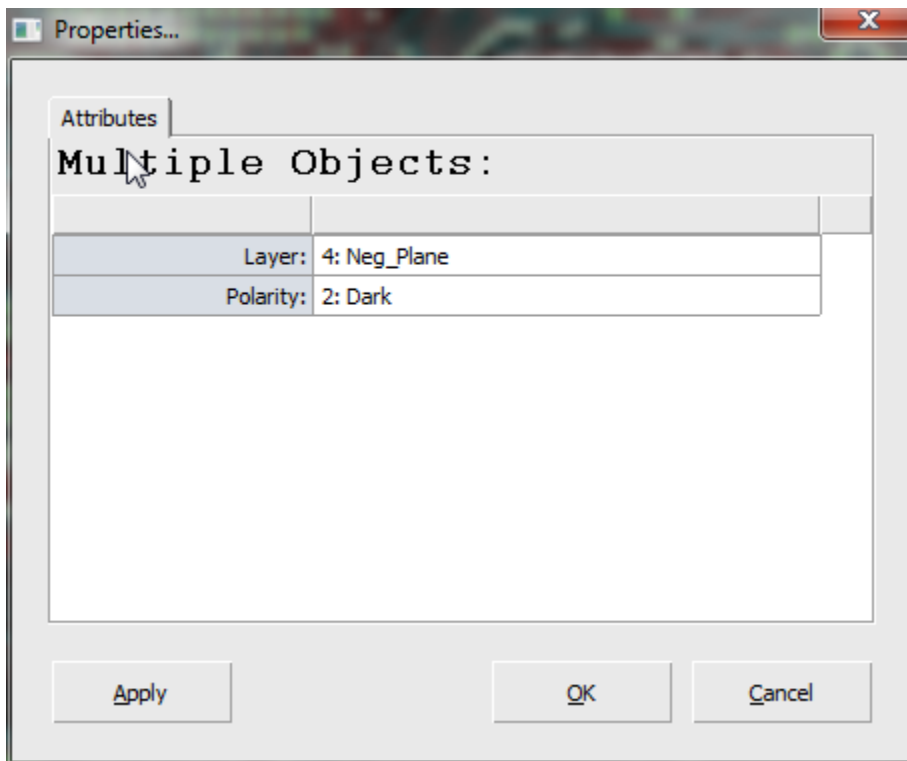
4.16 Understanding Composites



Layers that contain one or more objects with a defined "Composite Level" greater than zero are considered **composite layers**. Composite levels make it easy to define the fill rules for objects without the construction of complex polygons or geometries. Objects are drawn per layer in the order of their composite level, starting with 0 and increasing to a maximum composite level of 1000. All "even" (and zero) composite levels are filled dark (i.e. Paint), while "odd" composite levels are drawn clear (i.e. Scratch).

To change the composite level of an object:

1. Select the **Object**.
2. Select **Edit > Properties**.



The properties dialog box will appear.

3. Locate the item "Polarity:" Change the composite level by changing the value from the pull down list.

4. Select **OK** to exit the dialog box.

You may undo any changes by selecting Undo using **Edit > Undo**.

To automatically change the composite level of a group of objects based on their interior location relative to other objects, see **4.18 Polygon De-Embedding**.

4.17 Polygon De-Embedding

Polygon De-Embedding solves the dilemma of changing polygons inside other polygons. This uses de-embedding to automatically create composites. To perform Polygon De-Embedding:

1. Select the objects for de-embedding.
2. Select **Edit > De-Embed**.
3. Object composite levels will become defined automatically in FAB 3000.

In order for composite levels to become defined, the objects must reside on the same layer.

You may undo any changes by selecting Undo using **Edit > Undo**.

4.18 Centroid Files

WHAT ARE CENTROID FILES?

Automated equipment is used to place the surface mount components on your PCB. To rapidly program these machines, a Component Centroid file is required. Some CAD packages will automatically generate this file and some will not. FAB 3000 (Professional Version) can easily create this centroid file from your existing Gerber files.

IMPORTANT CONSIDERATIONS WHEN GENERATING CENTROID FILES:

Centroid files are primarily used for placement of SMT components. For best results when generating a Centroid File from Gerber, use the paste layer. If you do not have a Solder Paste Layer, please create one by selecting **Fabrication > Generate Missing Layers > Paste Layer(s)**.

All pads on the Paste Layer must be "Flashed". If there are any pads on the Paste Layer that are not "Flashed" please do so by selecting **Tools > Convert To > Flash (Automatic)**.

To determine if all pads are "Flashes" on the Paste Layer, go to the **Layers Table** and try assigning a different color for the Flashed objects. Then when you view the Paste layer, all Flashed Pads will use the different color you have assigned. You can also select **Info > Query > Query All**.

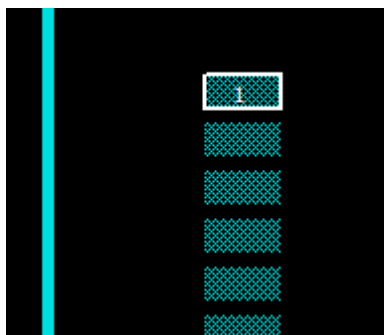
ADDING COMPONENTS

1. Turn on paste or mask layer (paste preferred).
 - a. If paste layer is not defined you must instruct FAB 3000 by going to the **Setup > Layer Table**.
2. Locate a pad within the editor and select it.
 - a. FAB 3000 will try to guess Pin #1.

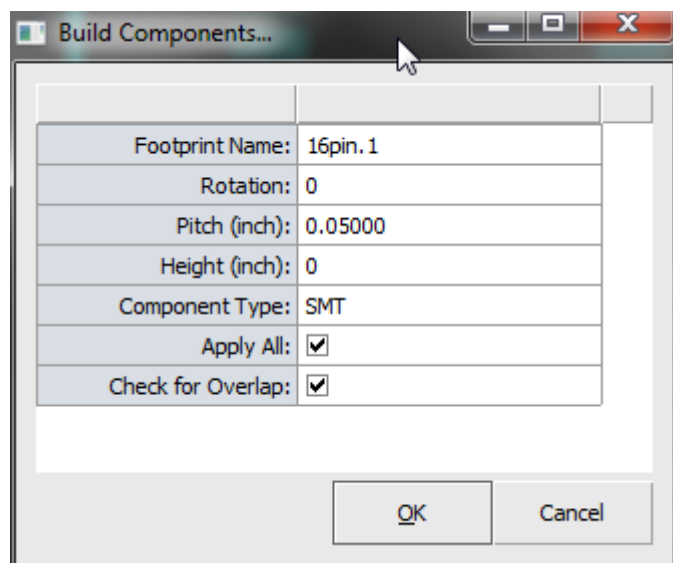
- b. You may also select a different pad for Pin #1.

3. Select Assembly > Build Component

4. Select Pin One.

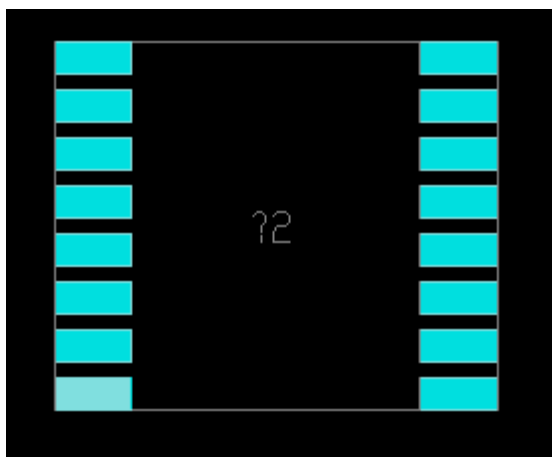


- a. Usually pin one is the top-leftmost pin for top side (assuming no rotation) and top right most for bottom side (assuming no rotation). Pin #1 is needed as it helps orient the placement of the component.
- b. Once Pin #1 is chosen, right click to accept.



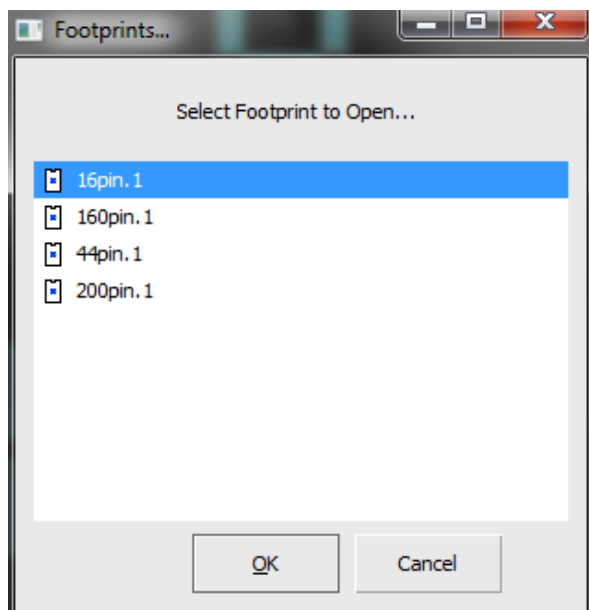
5. Assign component description. A **Build Components** dialog box will appear. Select OK when done.

- a. Enter component rotation as it appears on the board. If you have selected a component that is rotated 90 degrees on the board, then make sure to enter that here.
- b. If apply all is check (recommended), FAB 3000 will go out and seek all similar footprints.



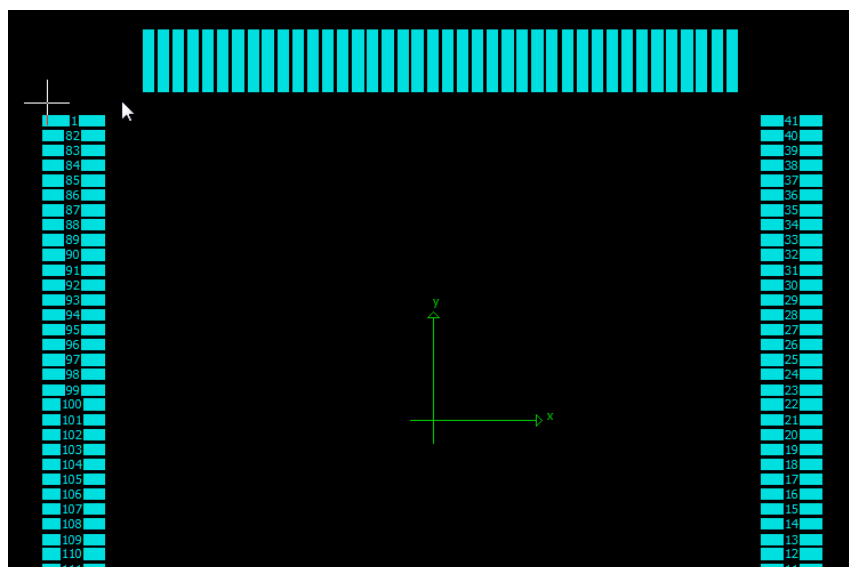
You will notice (if apply all is checked) that all similar footprints have been detected and assigned a generic reference designator value (for example "?21").

6. Repeat steps 2 thru 5, until all footprints on the paste layer have been completed.



7. Review the Footprint browser and verify remaining components. Select **Window > Footprint Browser**.

8. Select the Footprint to open by clicking it. Select **OK**.



8. In the Footprint browser you may load external libraries (containing footprints) or make modifications here.

Note: To perform these actions for the bottom side (if required), turn off all layers except **Bottom Paste &**

Bottom Silkscreen. Repeat steps 2-6.

ASSIGN REFERENCE DESIGNATORS TO COMPONENTS

1. Disable all layers except for **Top Component & Top Silkscreen**.
2. In the **Components Tab Browser**, select a component from the list.

Components selected from this list will be highlighted. To zoom into the selected component, enable **Zoom To Component** by checking the corresponding box.

Arrow Keys are helpful in navigating up and down through the Components list.

3. Click into the edit control "RefDes:", and type-in the correct reference designator. Select **Enter**.

Keep the silkscreen visible. It's a great way to determine which reference designator to assign the component.

4. Repeat Step 3 until all Components have been assigned their reference designators.

EXPORTING A CENTROID FILE

In order to export a centroid file, you must have at least one or more components defined before exporting.

Centroid files are comma separated files; you can load them into almost any spreadsheet program or text editor.

To export a Centroid File:

1. Select **File > Export > Centroid File**.
2. View and select the available options.
3. Select File Name & Location. The centroid file will now export.

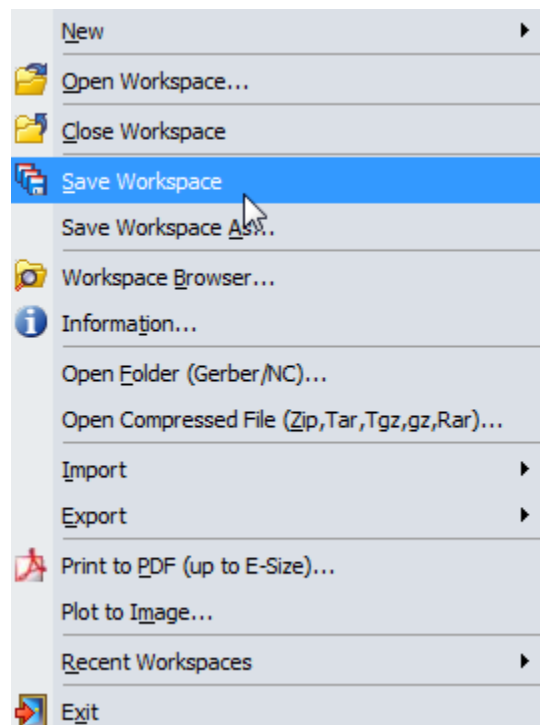


[Video: Generate X/Y Centroid File from Gerber, using FAB 3000](#)

[Video: Generate PCB Centroid File \(used in PCB Assembly\) from Gerber](#)

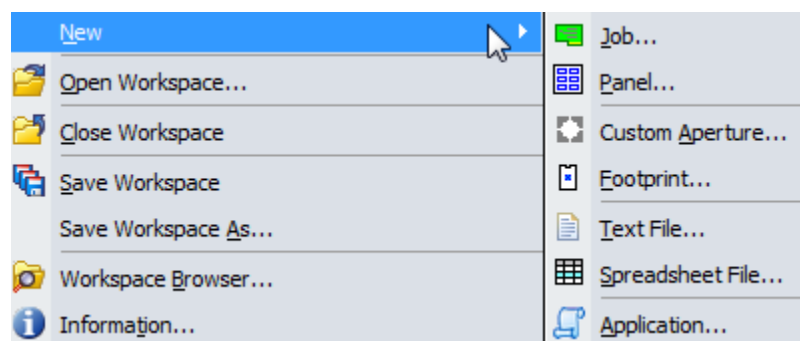
SECTION 5: FAB 3000 JOB EDITOR MENU SUMMARY

5.1 File Menu



The file menu is the backbone for opening, closing, importing, and exporting designs. It also contains features such as printing and plotting as well as the workspace browser; a tool used to navigate resources (from the active workspace) in a hierarchy of libraries, designs, folders and files. From here, you can open resources for editing or select resources for operations such as exporting.

File > New



When selecting **File > New**, you're presented with a sub menu of options for new projects.

Job: Creates a new Job. Enter a new Job name and select OK. A new Job will be created and also appear as an Editor Tab.

Panel: Creates a new Panel. Enter a new Panel name as well as additional

parameters such as Panel Units, Panel Width/Height, & Panel Margins.

Custom Aperture: Creates a new Custom Aperture. Enter a Custom Aperture name and select OK. A new Custom Aperture will appear in the Editor Tab.

- a. The custom aperture name must be unique within the active workspace.
- b. The Custom Aperture editor is used to create special shapes that may be used as flashes in the current FAB 3000 workspace. Every FAB 3000 workspace has its own internal custom aperture library. When importing Gerber files that contain custom

apertures, FAB 3000 adds them to the internal custom aperture library automatically. Custom aperture libraries may also be maintained as independent files for use from design to design.

- c. The Custom Aperture editor differs from the CAM Editor in a few ways. There is only one generic layer. Custom apertures may contain both positive (dark) and negative (clear) data. For example, a donut is created by having a positive outer circle with a negative inner circle. Positive and negative data can be stacked in any order, as long as you assign the correct composite level (located in the toolbar).

Footprint: Creates a new footprint and opens it up in the FAB 3000 editor. Enter a Footprint name and select OK. A new Footprint will also appear in the Editor Tab.

- a. A footprint name must be unique within the active workspace.

Text File: Opens a new Text File. Choose a Text File and select OK. A new Text File will also appear in the Editor Tab.

- a. A new Text File must be unique within your computer.

Spreadsheet File: Opens a new Spreadsheet File. Choose a Spreadsheet File and select OK. A new Spreadsheet File will also appear in the Editor Tab.

- a. A Spreadsheet File must be unique within your computer.

Application File: Opens a new Application File (such as LUA Files). Choose an Application File and select OK. A new Spreadsheet File will also appear in the Editor Tab.

- a. An Application File must be unique within your computer.

File > Open Workspace

Opens an existing FAB 3000 Workspace using *.**wrk** file format. Choose a Workspace and select OK.

File > Close Workspace

Closes an active Workspace. If the Workspace has been modified, you'll be prompted to **Save**.

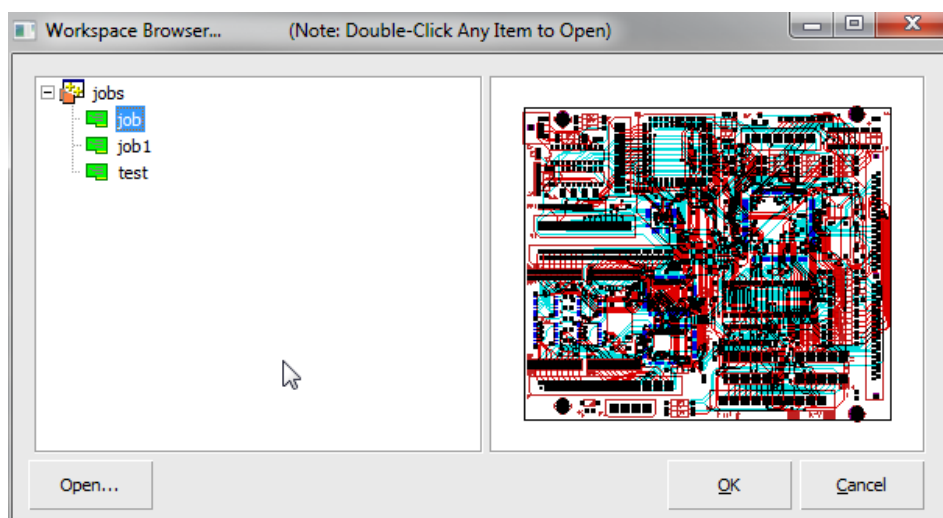
File > Save Workspace

Saves a Workspace.

- a. If you have not saved your workspace before, you'll be prompted to assign a Workspace file name and location.

File > Save Workspace As

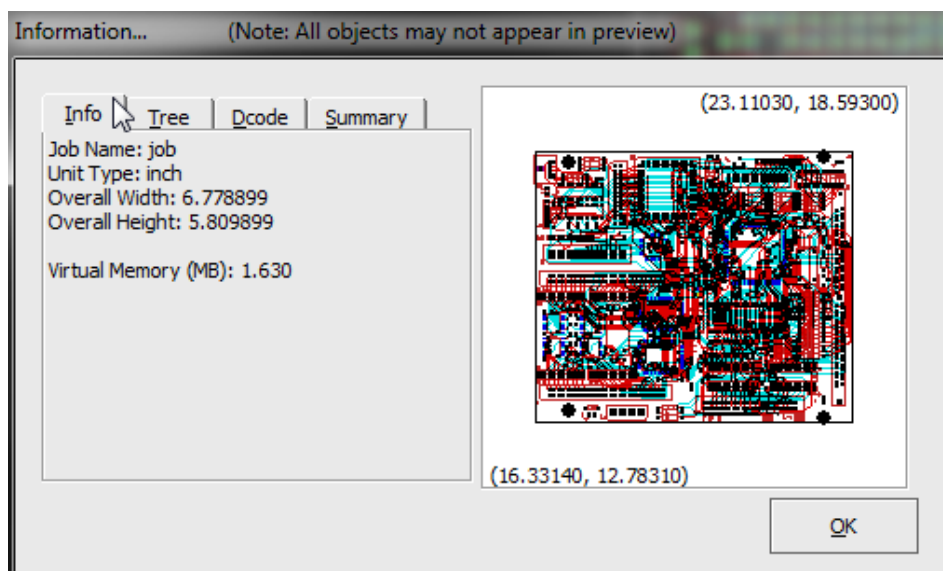
Saves an existing Workspace as another file. Select or modify a Workspace File name and location then select OK.



File > Workspace Browser

Allows you to quickly preview all loaded Jobs and Panels within an active Workspace.

- a. Single Click: Preview
- b. Double Click: Open



File > Information

Displays all Job information. Includes tabs for Tree Hierarchy, Dcode, & Summary.

File > Open Folder (Gerber / NC)

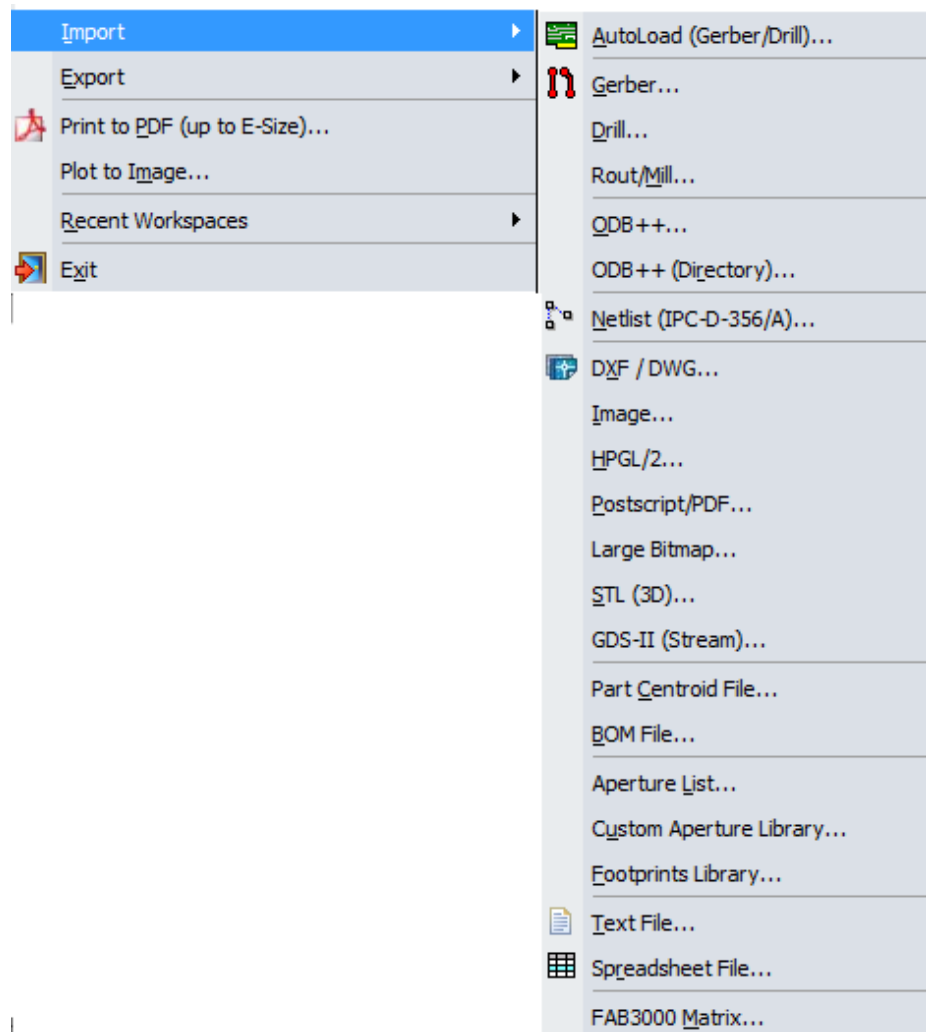
This command automatically loads Gerber and Drill data from a specified folder. Choose the folder you want to import and then select OK. The entire folder is scanned – all Gerber, Drill, & Netlist files found are quickly imported.

- a. FAB 3000 will automatically determine the correct format and aperture rules necessary to bring in your Gerber files with minimal effort. However, if you are aware of special circumstances regarding your Gerber files (such as 274D Gerbers using an Incremental type, End-Block-Character other than "*", etc.) or you've had previously imported Gerbers load correctly, you may want to use **AutoLoad**.
- b. To view the log file of the import you performed, select the **F2** key.

File > Open Compressed File (Zip, Tar, GZ, RAR)

This command automatically loads Gerber and Drill data from a specified compressed folder. FAB 3000 will automatically uncompress the file while a wizard dialog box will appear. To open a compressed file, perform the following:

1. **Select File > Open Compressed File (Zip, Tar, GZ, RAR).**
2. Choose the compressed file to import.
 - a. FAB 3000 will automatically determine the correct format and aperture rules necessary to bring in your Gerbers with minimal effort. However, if you are aware of special circumstances regarding your Gerber files (such as 274D Gerbers using an Incremental type, End-Block-Character other than "*", etc.) or you've had previously imported Gerbers load correctly, you may want to change AutoLoad's default settings by clicking the button **Format Settings**. If you are unsure, please leave the format settings defaulted.
3. Select OK
4. FAB 3000 will begin the process of uncompressing the file. At the same time, an import wizard dialog box will appear.
5. Select Next to begin the import.
6. Upon completion review the status & import log. Any errors or warnings will be displayed here.
 - b. To view the log file of the import you performed, select the **F2** key.

File >Import

The following file types are possible using File > Import:

AutoLoad (Gerber/Drill)	Image	Aperture List
Gerber	HPGL/2	Custom Aperture List
Drill	Postscript/PDF	Footprints Library
Rout/Mill	Large Bitmap	Tex File
ODB++	STL (3D)	Spreadsheet File
ODB++ (Directory)	GDS-II (Stream)	FAB 3000 Matrix
Netlist	Part Centroid File	
DXF/DWG	BOM File	

File > Import > Autoload

AutoLoad Summary

Autoload Wizard is an automatic import tool for Gerber and Drill files from a selected folder. It's designed to make importing PCB designs an easy and time efficient by eliminating individual loading of each file. To perform Autoload:

1. Put all pertaining files to a Job in the same directory.
2. Select **File > Import > Autoload**.
 - a. The Autoload Wizard will appear.
3. Choose either English or Metric Units.
4. Choose the folder which contains the Gerber files you wish to load. There are two selection options:
 - a. Immediately select **Finish** (fastest). This scans and loads all files in your directory without any further steps. This option works for loading extended Gerber data (format & apertures are already defined in the file).
 - b. Select **Next** (recommended). FAB 3000 will scan the directory to determine if each file is either a Gerber or Aperture list. This will display a File Description page which allows you to manually define the aperture list to use, layer types, and individually import each Gerber layer. Select **Next** to begin loading.
5. Once loading is complete, a status & import log will be available for your review. If any errors or warnings are encountered they will be displayed here.
6. Select **Finish**.

Common Issues with Gerber/Drill Importation

Incorrect Apertures

While importing 274D files, the most common cause for incorrect apertures is an incorrect or missing **.RUL** file. FAB 3000 tests each aperture file against all the **.RUL** files that are present in the **aptrules** directory. The **.RUL** file must be correct (less than 20% read errors) and located in the **aptrules** directory for this process to pass.

- a. Some aperture files define custom shapes and thermals in an inconsistent manner that cannot be read. These apertures contribute to the 20% error limit.

Incorrect Format (For 274D only)

Gerber files contain only numbers with no decimal point. Thus, a number such as X12345 could mean 1.2345 inches, 12.345 millimeters, or 123.45 inches. To limit the possibilities of loading errors, FAB 3000 assumes that the size of the data will probably be greater than 2 inches and less than 20. If your imported Gerber files appear to have imported with the wrong scale, you may want to change AutoLoad's default format settings by clicking the button **Format Settings**.

Incorrect Type (For 274D only)

Gerber files contain only numbers with no decimal point. Thus, a number such as X12345 could mean 1.2345 inches, 12.345 inches, or 123.45 inches. To limit the possibilities of loading errors, FAB 3000 assumes that the size of the data will probably be greater than 2 inches and less than 20. If the imported Gerber files appear to have incorrect lines stretching across the screen (like a rats-net), you may want to change AutoLoad's default settings by clicking the button **Format Settings**.

Incorrect Arcs (For 274D only)

Some Gerber generating products (like PCB layout software) assume all Gerber arcs to be of the "360-degree type", and do not follow the proper procedure to define these arcs. If the imported Gerber files appear to have incorrect arcs, you may want to change AutoLoad's default settings by clicking the button "Format Settings...", and check "Use 360-degree arcs as default"

Drill Data missing Tool Definition...

AutoLoad will automatically scan your drill files to determine if there are any missing tool definitions. If a Missing Tool is detected, AutoLoad will ignore the drill file, and you will be required to load the drill manually Import Drill.

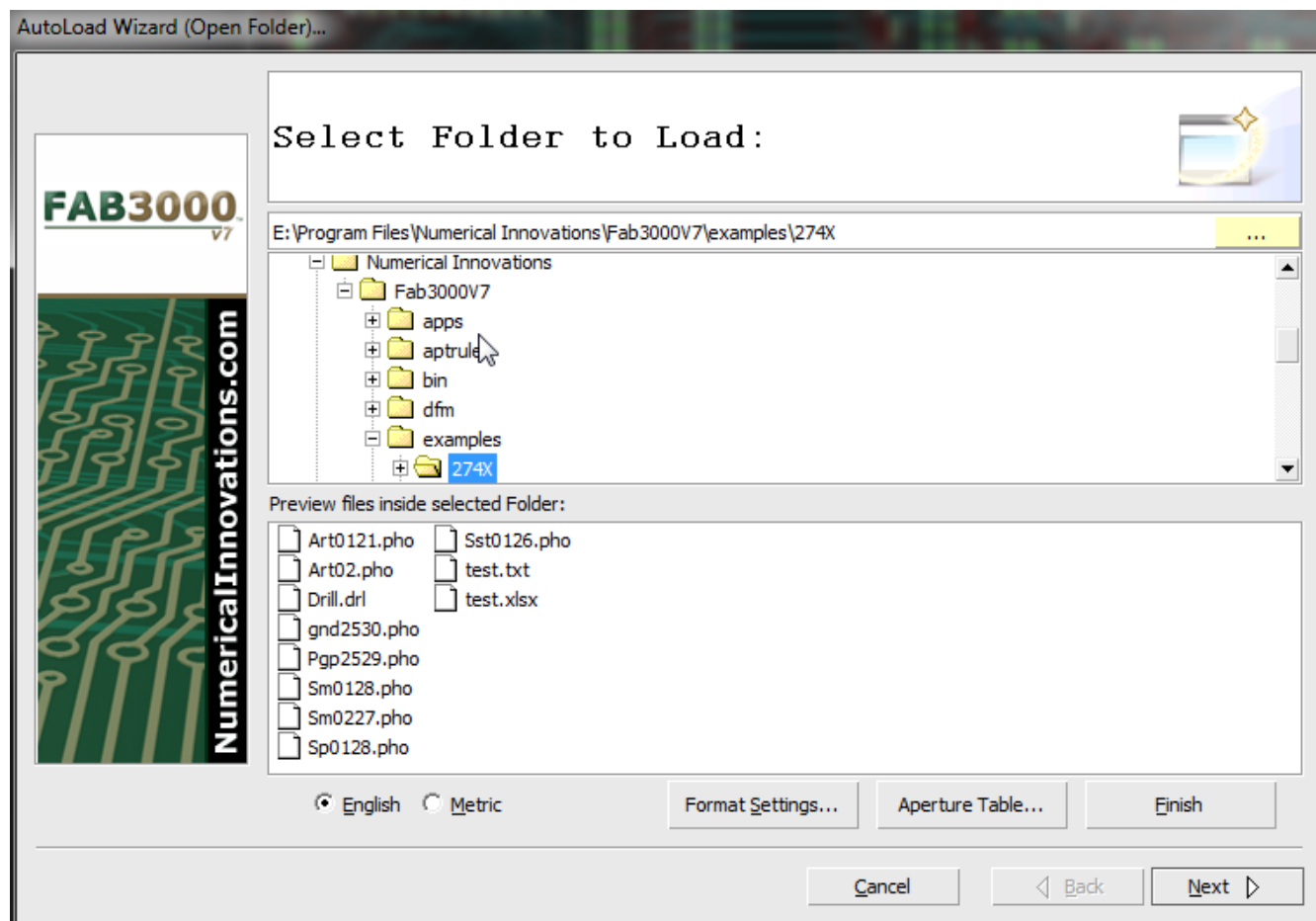
- a. FAB 3000 currently imports Excellon format only.
- b. If more than one drill files is being imported simultaneously or an existing Tool table is already defined, FAB 3000 will automatically map different drill tools with the same number but different diameters.

Other Loading Errors...

Gerber files which contain numerous errors during import most likely are not properly constructed. We have spent years refining our Gerber importer to accept non-standard Gerber files from all different software products throughout the world. If you are having problems loading any Gerber files into FAB 3000, please contact our support team for further assistance: support@numericalinnovations.com (be sure to provide us with an accurate explanation of the problem, along with the Gerber files you are trying to load).

AutoLoad Dialog Wizard Picture Summary

Step 1: AutoLoad Main Dialog



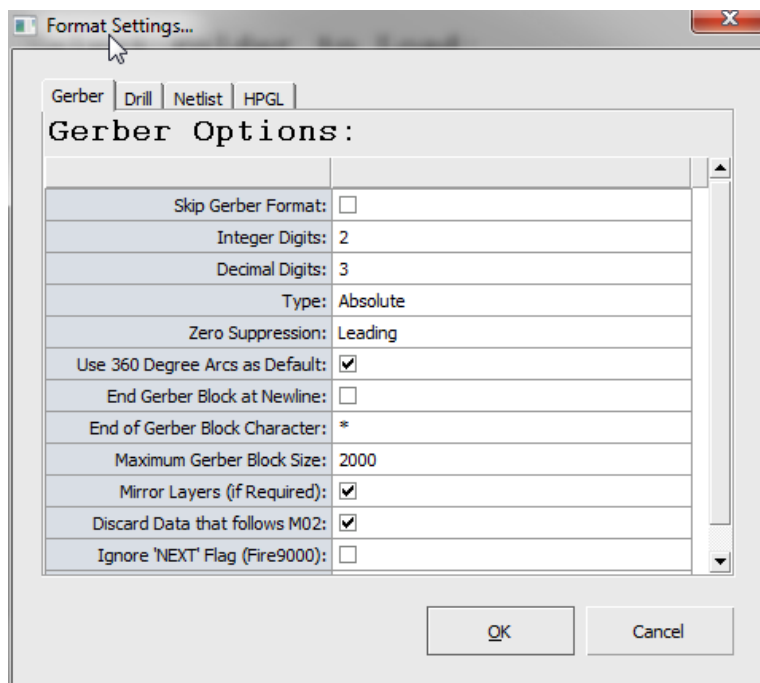
In the AutoLoad Wizard, first choose the folder from which your files will be imported.

Select units in either English or Metric.

Choose to **Format Settings** or view & edit the **Aperture Table**. (See *Format Settings & Aperture Table*)

Choose either **Finish** to immediately begin import or **Next** to individually define the aperture list, layer types, and individual layers to be imported (See *Step 2: File Description Box*).

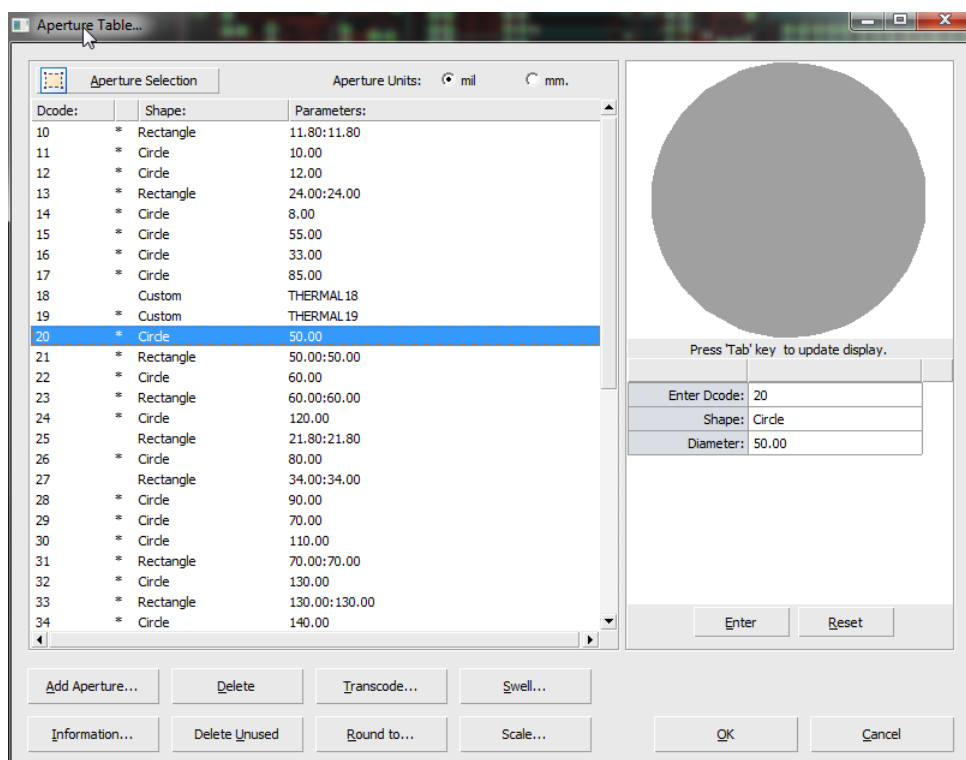
Format Settings Button



The Format Settings dialog box lets you setup custom defaults that AutoLoad Wizard will use during import. You also have the option to skip this format if you do not wish to include a particular EDA format type during AutoLoad.

You can set custom defaults for Gerber, Drill, Netlist, and HPGL options.

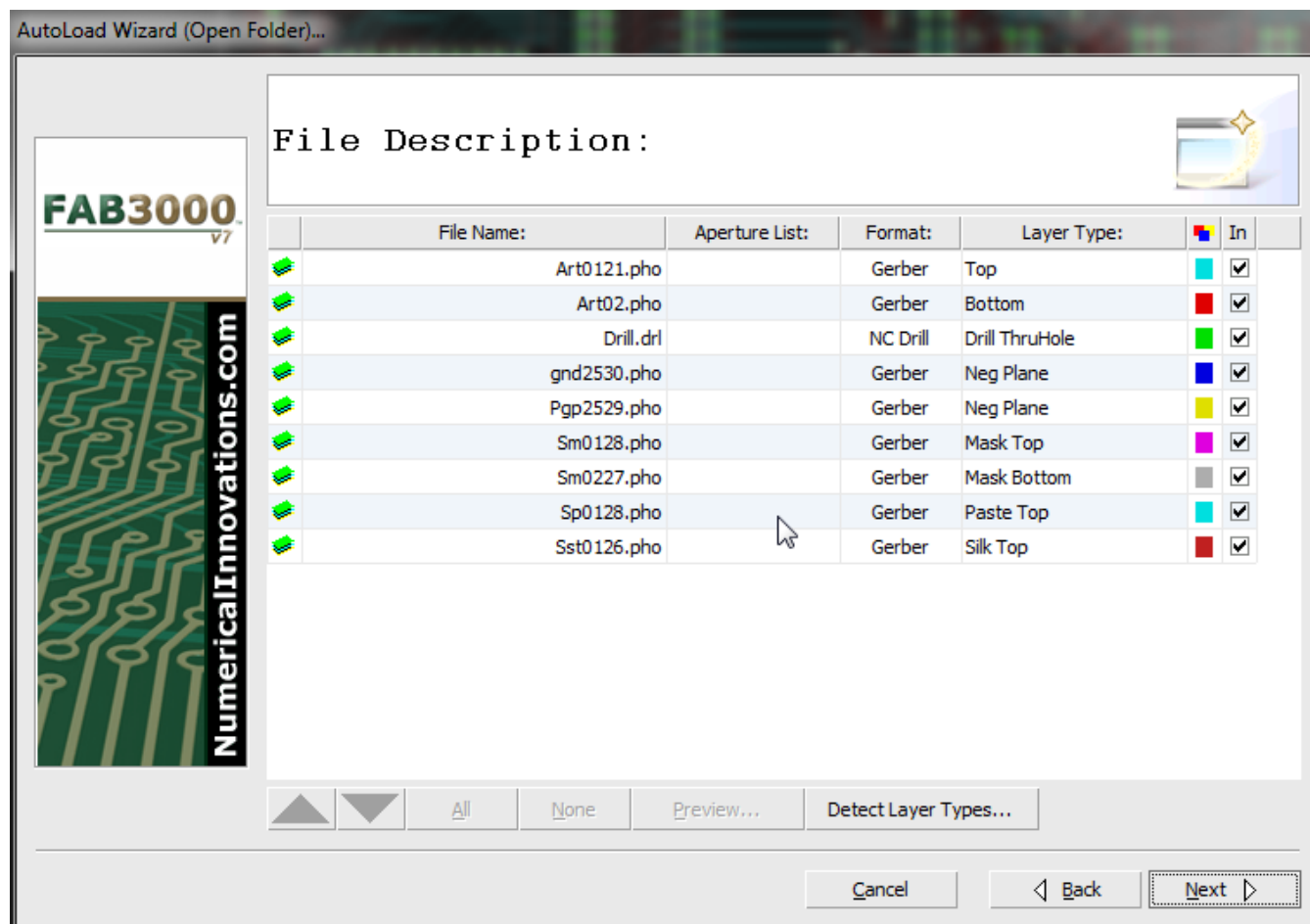
Aperture Table



The Aperture Table is available as an option for adding/deleting apertures, transcoding, swell, scaling, and information.

EDIT THIS DESCRIPTION

Step 2: File Description Box



The **File Description** Page lets you fine-tune exactly what will be imported with AutoLoad. Beside each file name is the following information:

File Name: The name of the file located in the Autoload import directory.

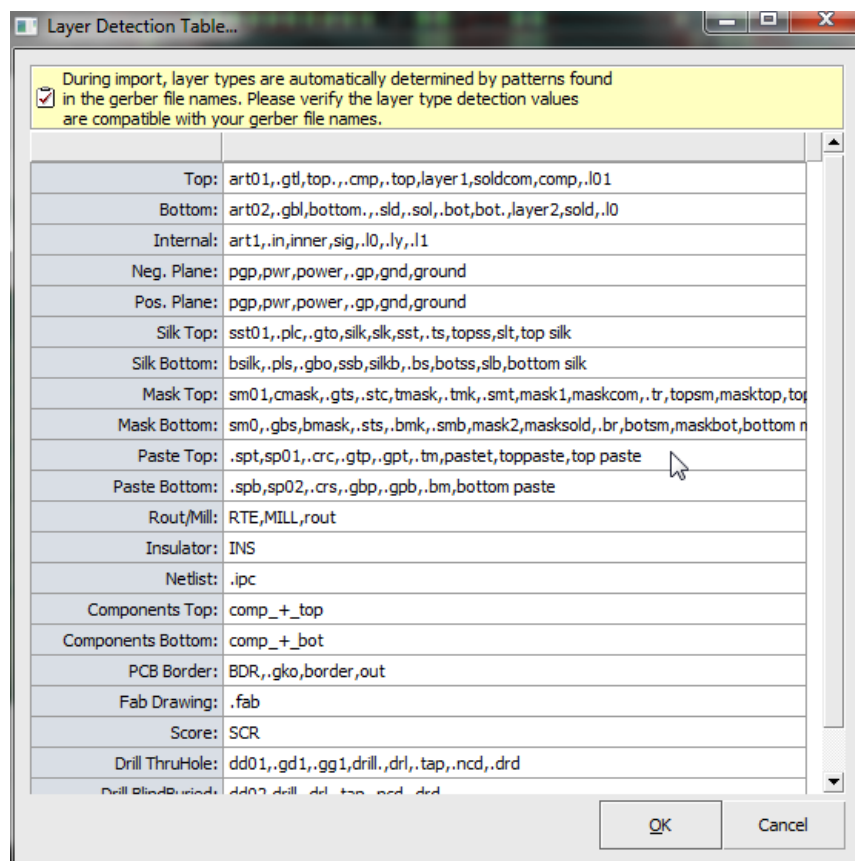
Aperture List: (Optional) Select corresponding aperture list. (For RS-274D with separate aperture lists per layer)

Format: Lets you know the format that AutoLoad has detected this file as.

Layer Type: Select the corresponding layer types for each file. You may also assign layer color by choosing the corresponding color box.

Import: A Check box to determine if you want to load this file. If you de-select a previously checked item, it will not be included during AutoLoad.

Detect Layer Types: During import, layer types are automatically determined by patterns found in the Gerber file names. The Detect Layer Types button summarizes all layer type detection values are compatible with your Gerber file names (see below).



Step 3: Finalizing Import

Upon selecting either Finish (faster way) or Next (recommended way using File Description), the AutoLoad Wizard will immediately begin the import process. A progress bar will appear for each file imported. If the load has a problem, you will be notified of any warnings or errors which occur. If the load is successful, the box will immediately disappear and import your files into the workspace and editor.

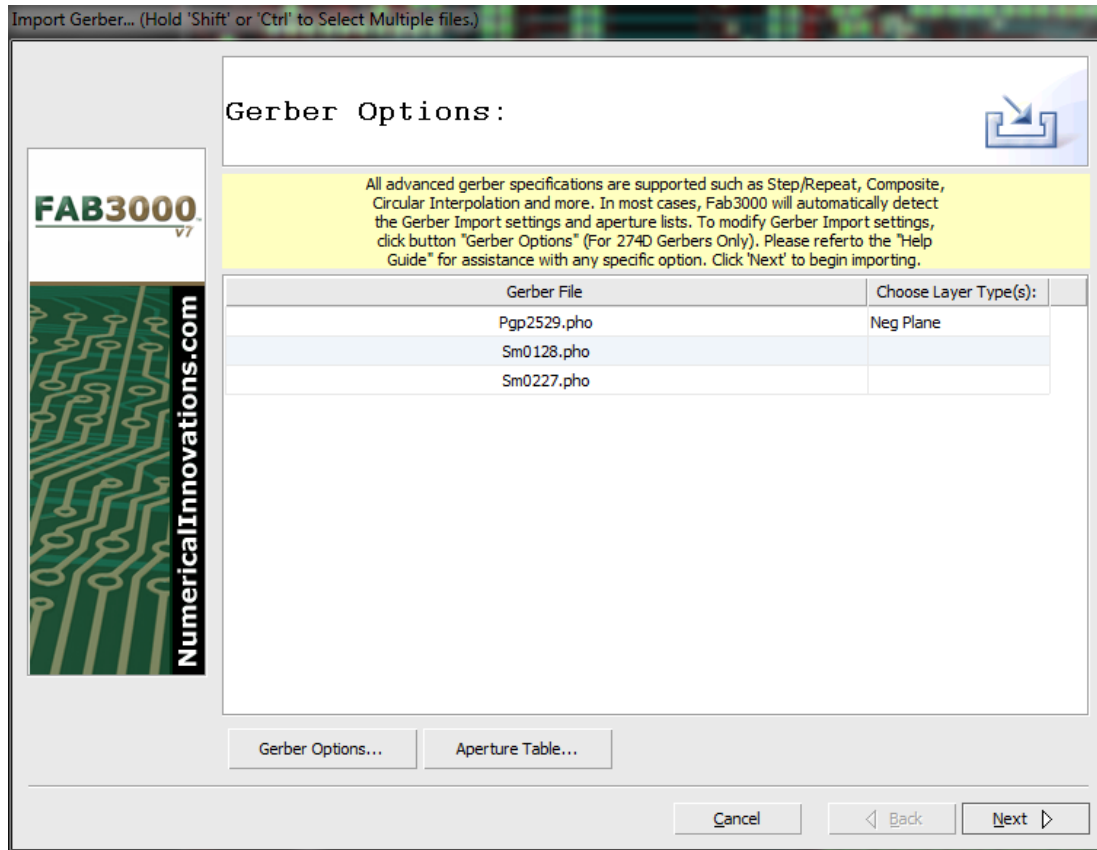
File > Import > Gerber

FAB 3000 reads Gerber data in RS-274D, RS-274X, and Fire 9000 formats. Multiple files are read into multiple layers, allowing simultaneous viewing and editing of the Gerber data. The system is fully compatible with polygon fills, composite images, and step and repeat codes.

- a. FAB 3000 correctly displays Step & Repeat data. Most other CAM software will Crash, Ignore, or draw imaginary boxes. FAB 3000 will create a separate Job for each Step & Repeat block which is then Inserted into the parent Job; maintaining correct Step & Repeat hierarchy, and improving the ability to edit and modify existing Step & Repeats.

To import a Gerber file, perform the following:

1. Select **File > Import > Gerber**
2. Choose the Gerber files you wish to import (select multiple files using the **Shift** or **Ctrl** key) Select **OK**.
3. The Import Gerber wizard dialog box will appear with your chosen files.



4. To modify Gerber import settings, select **Gerber Options**. Otherwise, skip this step.

[Forum: During Gerber import, I get the warning message "Invalid Gerber Quadrant Arc"](#)

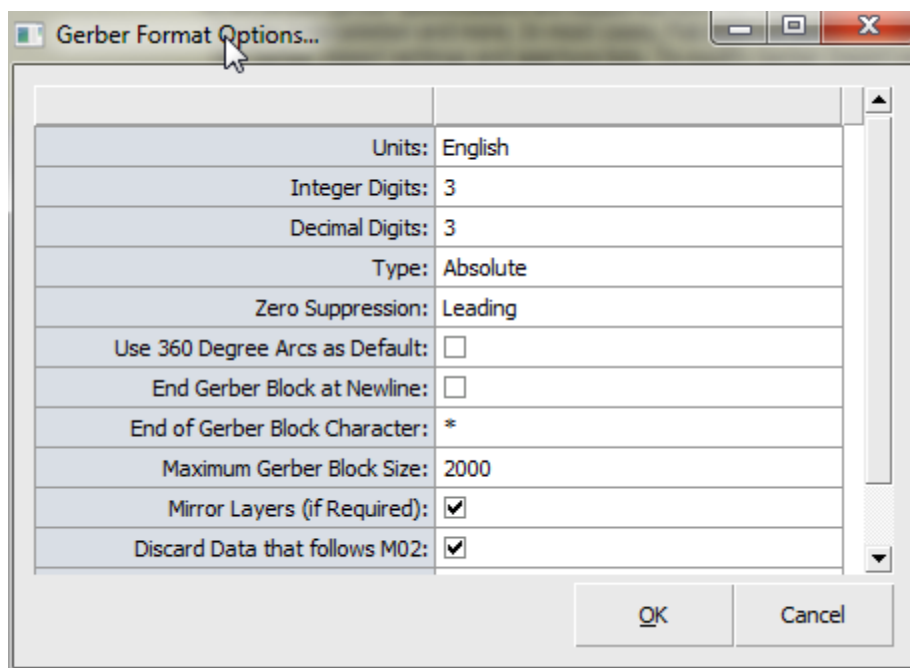


[Video: Automatically Import Gerber and Panelize](#)

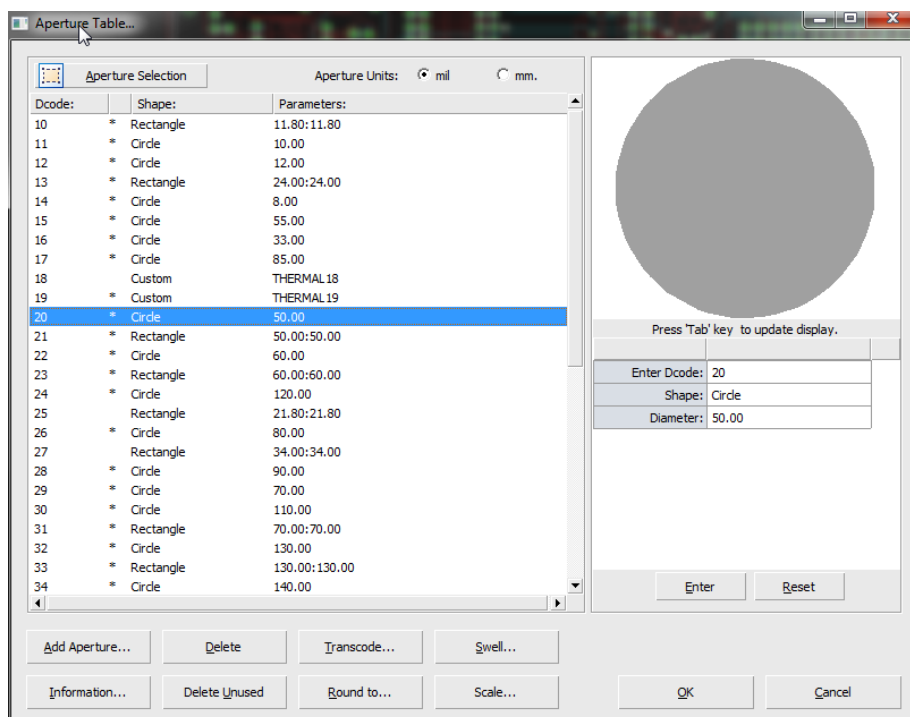
[Video: Import Gerber files, detect layer types, assign stackup, and more](#)

[Video: Gerber Import with 90 degree arcs using FAB 3000](#)

[Video: Step and Repeat Gerber Files](#)

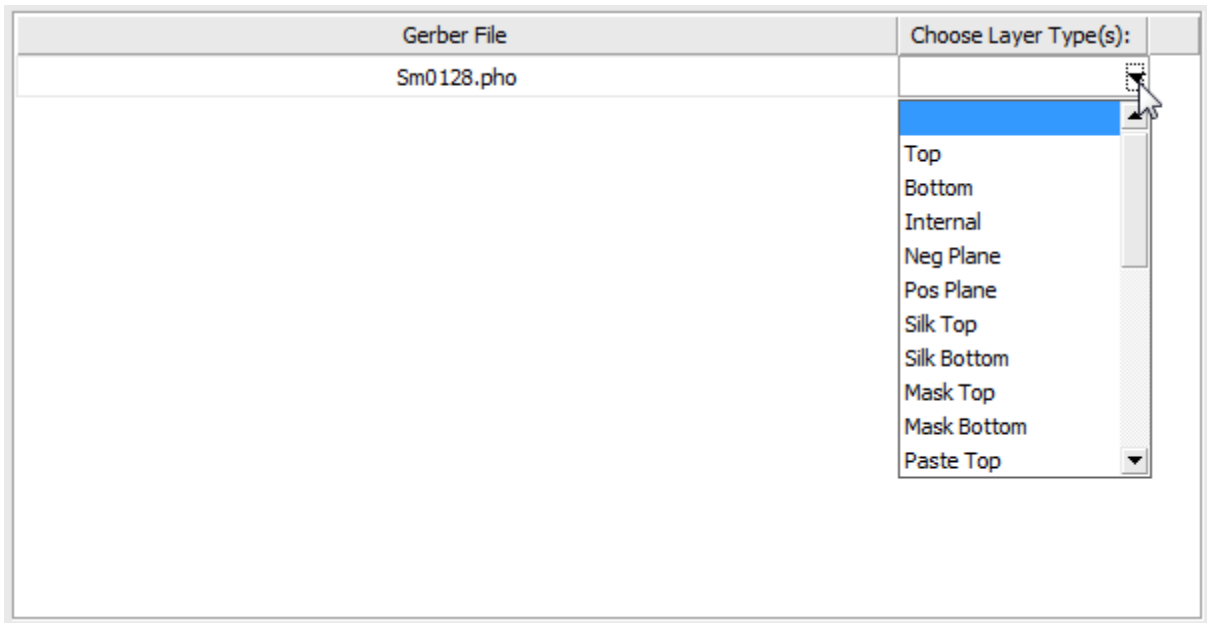


Gerber options allows you to edit the format for Gerbers. FAB 3000 will automatically detect Gerber formatted settings and aperture lists. If you are unsure about these settings, use the recommended defaults.



5. To modify Aperture settings, select **Aperture Table**. NEED MORE DEFINITION HERE

6. In the Gerber Options dialog box, assign a layer type using the Choose Layer Type drop down box next to each Gerber file.



7. Select Next to begin the import.
8. Once the importation is complete, FAB 3000 encounters any errors or warnings they will be displayed in the dialog box. Otherwise, the dialog box will disappear and the import is complete.
 - a. If you already had existing data opened in the current Job, any layers with the same name will be merged within that Job. It is recommended that you instead create a new Job for each new "design" that you import into FAB 3000.

Import Gerber Format Options Defined

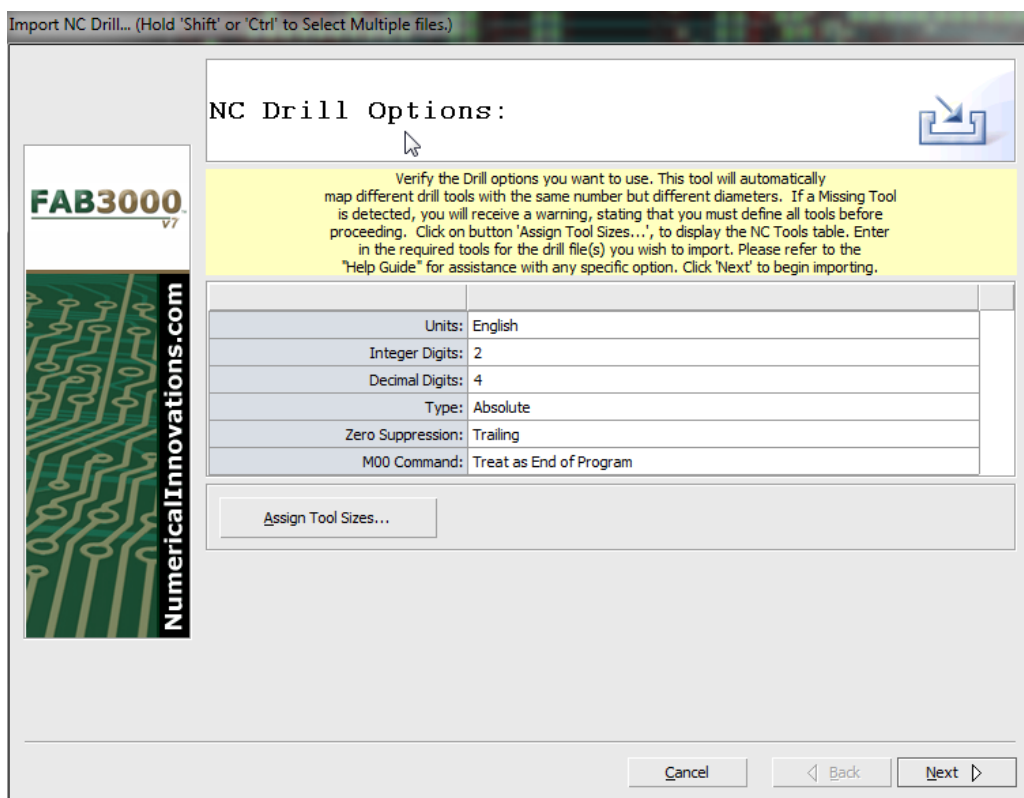
Units:	Determine if imported Gerber's have units of English or Metric.
Integer Digits:	Determine if imported Gerber's have integer digits between (0-6).
Decimal Digits:	Determine if imported Gerber's have decimal digits between (0-6).
Type:	Determine if imported Gerber's have type of Absolute or Incremental.
Zero Suppression:	Determine if imported Gerber's have zero suppression of Leading, Trailing, or None.
Use 360 Degree Arcs as default:	Read Gerbers using 360 degree arcs mode as a default.
End Gerber Block at Newline:	Treat newline characters as an end of Gerber block.
End of Gerber Block Character:	Character to for ending Gerber blocks. Usually an asterisk '*'.
Maximum Gerber Block Size:	Maximum size allowed per Gerber block.

File > Import > Drill

Import > Drill allows you to import one or more NC drill files in *.DRL format. Currently FAB 3000 only imports in Excellon format. If more than one drill file is imported at the same time or an existing Tool table

is already defined, FAB 3000 will automatically map different drill tools with the same number but different diameters. FAB 3000 will also automatically scan your drill files to determine if there are any missing tool definitions. To import drill files, perform the following:

1. Select **File > Import > Drill**.
2. Choose the Drill files you want to import and select **OK**. The Drill wizard dialog box will appear. To select more than one file, use the **Shift** or **Ctrl** keys.



3. Select English or Metric units.
4. Select the Integer digits required. These are digits to the left of the decimal point.
5. Select the Decimal digits required. These are digits to the right of the decimal point.
6. Select either **Absolute** or **Incremental** format type.
7. Select the Zero Suppression format required.
8. Select Next to begin importing drill files.

If a Missing Tool is detected, you will receive a warning stating that you must define all tools before proceeding. In the **Drill Options** menu use the button, **Assign Tool Sizes** to display the NC Tools Table.

#	Size	Plated	Final Size	Drl.Cnt	Mill.Cnt	Comments	Sort
1	0.00	<input checked="" type="checkbox"/>	0.00	0	0		1
2	0.00	<input checked="" type="checkbox"/>	0.00	0	0		2
3	0.00	<input checked="" type="checkbox"/>	0.00	0	0		3
4	0.00	<input checked="" type="checkbox"/>	0.00	0	0		4
5	0.00	<input checked="" type="checkbox"/>	0.00	0	0		5
6	0.00	<input checked="" type="checkbox"/>	0.00	0	0		6
7	0.00	<input checked="" type="checkbox"/>	0.00	0	0		7
8	0.00	<input checked="" type="checkbox"/>	0.00	0	0		8
9	0.00	<input checked="" type="checkbox"/>	0.00	0	0		9
10	0.00	<input checked="" type="checkbox"/>	0.00	0	0		10
11	0.00	<input checked="" type="checkbox"/>	0.00	0	0		11
12	0.00	<input checked="" type="checkbox"/>	0.00	0	0		12
13	0.00	<input checked="" type="checkbox"/>	0.00	0	0		13
14	0.00	<input checked="" type="checkbox"/>	0.00	0	0		14
15	0.00	<input checked="" type="checkbox"/>	0.00	0	0		15
16	0.00	<input checked="" type="checkbox"/>	0.00	0	0		16

Enter in the required tools for the drill file(s) you wish to import. If you are importing more than one drill file, and they use different tools sizes for the same tool number, than you will need to cancel the Import Drill Wizard and, import each drill file one at a time.

9. Once the drill loading is complete, the dialog box will disappear. If there are errors or warnings during import, a status and import log will appear.

Import NC Drill Options Defined

Units:	Determine if imported Drills have units of English or Metric.
Integer Digits:	Determine if imported Drills have integer digits between (0-6).
Decimal Digits:	Determine if imported Drills have decimal digits between (0-6).
Type:	Determine if imported Drills have type of Absolute or Incremental.
Zero Suppression:	Determine if imported Drills have zero suppression of Leading, Trailing, or None.



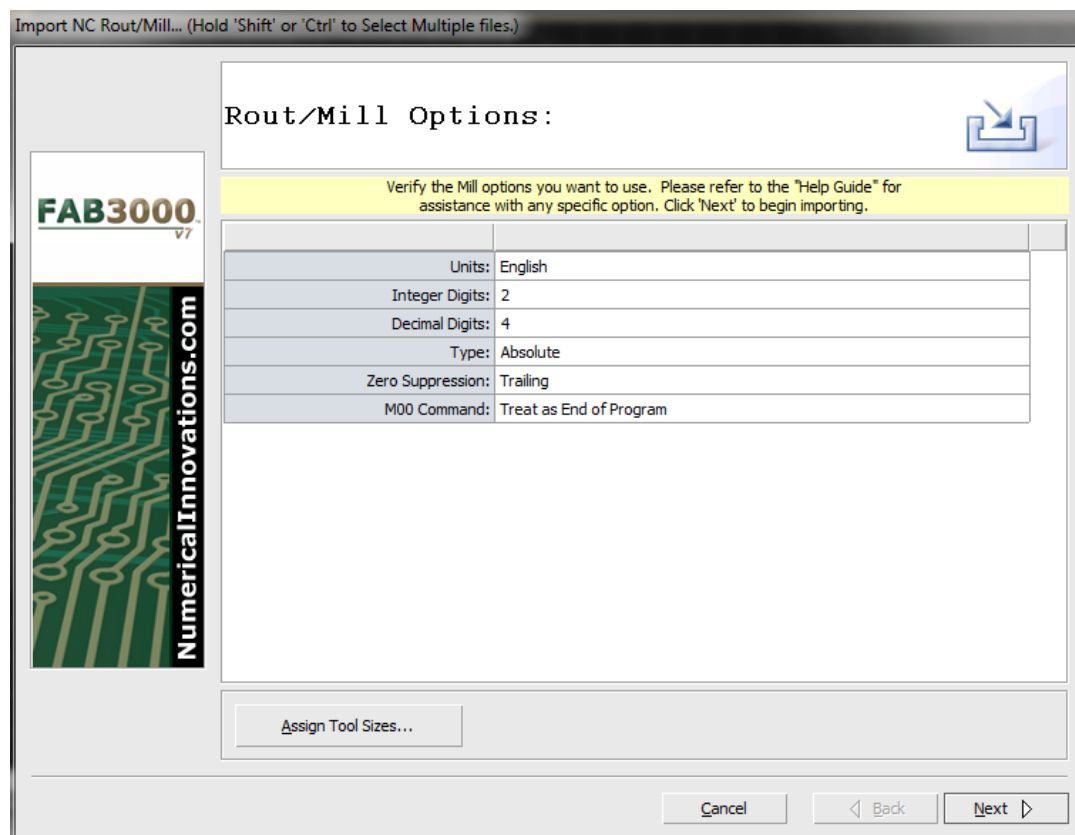
[Forum: Why does FAB 3000 assign generic tools sizes when importing drill files?](#)

[Video: Understanding Import NC Drill Problems using FAB 3000](#)

[Video: Gerber to Drill using FAB 3000](#)

[Forum: How can I change the drill file format without having to re-import it?](#)

File > Import > Rout/Mill



Rout/Mill allows you to import multiple NC Rout (*.ROU) files simultaneously.

FAB 3000 currently imports in Excellon format only.

If more than one rout file is imported at the same time or an existing Tool table is already defined, FAB 3000 will map different tools with the same number but different diameters.

FAB 3000 will also automatically scan your rout files to determine if there are any missing tool definitions.

To import Rout/Mill files perform the following:

1. Select **File > Import > Rout**.
2. Choose the Rout files to import. Select OK.
 - a. Click and hold the Shift or Ctrl key for multiple imports.
3. Choose English or Metric units.
4. Select the Integer digits required. These are digits to the left of the decimal point.
5. Select the Decimal digits required. These are digits to the right of the decimal point.
6. Select either **Absolute** or **Incremental** format type.
7. Select the Zero Suppression format required.
8. Select Next to begin importing rout files.

Once the drill loading is complete, the dialog box will disappear. If there are errors or warnings during import, a status and import log will appear.

If a Missing Tool is detected, you will receive a warning stating that you must define all tools before proceeding. In the **Rout/Mill Options** menu use the button, **Assign Tool Sizes** to display the NC Tools Table.

#	Size	Plated	Final Size	Drl.Cnt	Mill.Cnt	Comments	Sort
1	0.00	<input checked="" type="checkbox"/>	0.00	0	0		1
2	0.00	<input checked="" type="checkbox"/>	0.00	0	0		2
3	0.00	<input checked="" type="checkbox"/>	0.00	0	0		3
4	0.00	<input checked="" type="checkbox"/>	0.00	0	0		4
5	0.00	<input checked="" type="checkbox"/>	0.00	0	0		5
6	0.00	<input checked="" type="checkbox"/>	0.00	0	0		6
7	0.00	<input checked="" type="checkbox"/>	0.00	0	0		7
8	0.00	<input checked="" type="checkbox"/>	0.00	0	0		8
9	0.00	<input checked="" type="checkbox"/>	0.00	0	0		9
10	0.00	<input checked="" type="checkbox"/>	0.00	0	0		10
11	0.00	<input checked="" type="checkbox"/>	0.00	0	0		11
12	0.00	<input checked="" type="checkbox"/>	0.00	0	0		12
13	0.00	<input checked="" type="checkbox"/>	0.00	0	0		13
14	0.00	<input checked="" type="checkbox"/>	0.00	0	0		14
15	0.00	<input checked="" type="checkbox"/>	0.00	0	0		15
16	0.00	<input checked="" type="checkbox"/>	0.00	0	0		16

Enter in the required tools for the rout file(s) you wish to import. If you are importing more than one rout file, and they use different tools sizes for the same tool number, than you will need to cancel the Import Rout/Mill Wizard and, import each rout file one at a time.

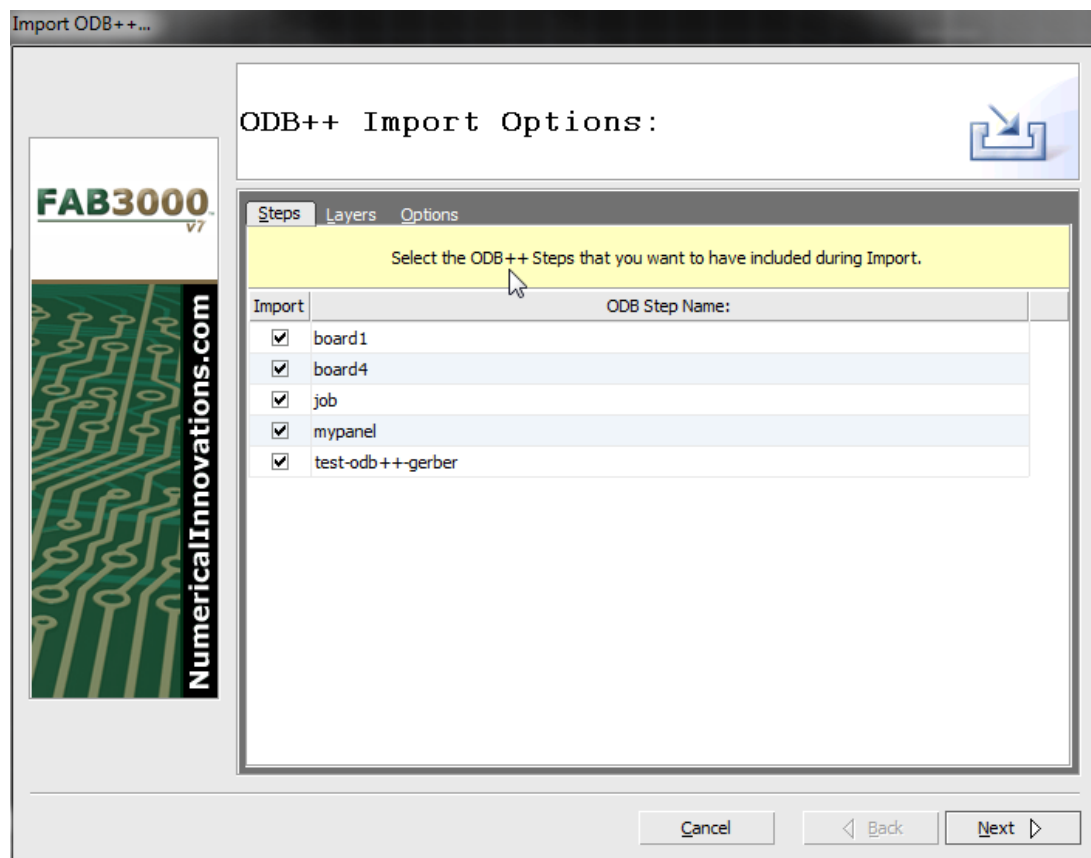
Import NC Rout Options:

Units:	Determine if imported Drills have units of English or Metric.
Integer Digits:	Determine if imported Drills have integer digits between (0-6).
Decimal Digits:	Determine if imported Drills have decimal digits between (0-6).
Type:	Determine if imported Drills have type of Absolute or Incremental.
Zero Suppression:	Determine if imported Drills have zero suppression of Leading, Trailing, or None.



[Video: Construct an NC Rout Path from a PCB Border using FAB 3000](#)

[Video: Create Rout for PCB Border, Slots, Cutout, and add Drill holes](#)

File > Import > ODB++

FAB 3000 allows you to import a compressed ODB++ database file. This file is usually formatted in ***.TGZ** format; otherwise known as a “tar ball”.

Please note that in FAB 3000 a “Job” is the same thing as a “Step” in ODB++.

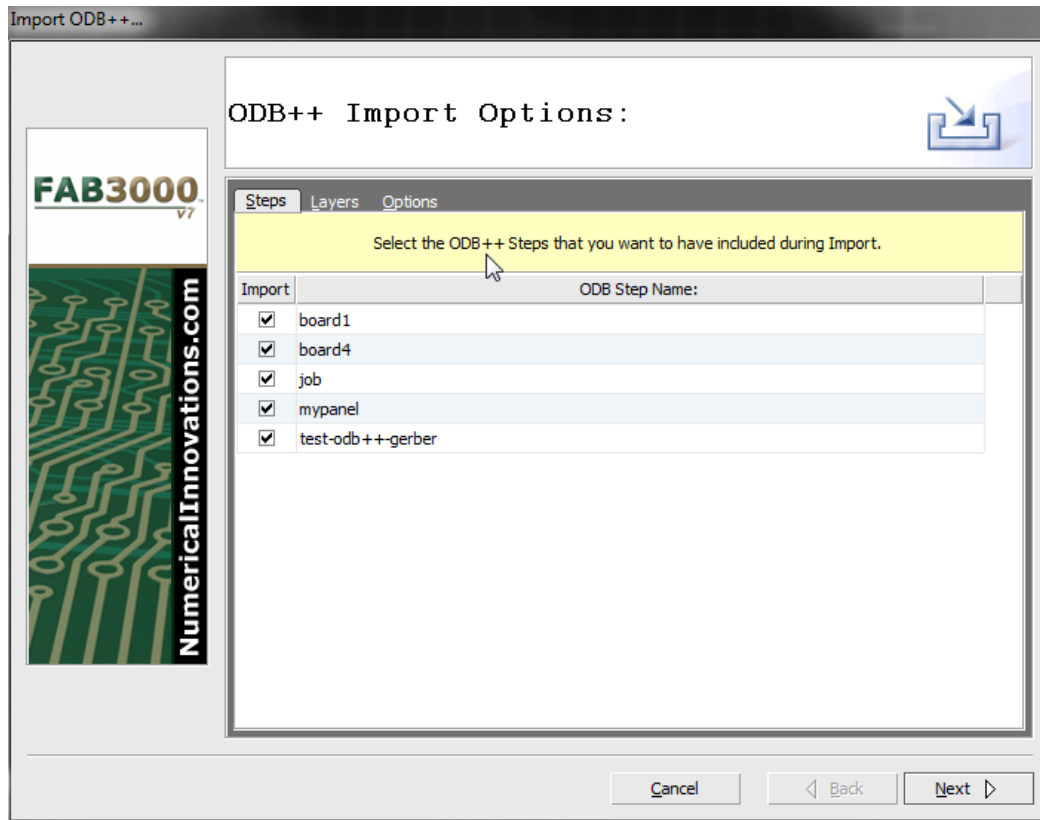
To import a compressed ODB++ database file, perform the following:

1. Select **File > Import > ODB++**.
2. Select the compressed ODB++ file you would like to import. Select **OK**.
3. Use the tabs at the top of the ODB++ import options menu to verify layers, steps, and options.
 - a. Steps: Choose which ODB file names to import by using the corresponding check box.
 - b. Layers: Verify the layers you want to import. Use the drop down menus for layer type and polarity by double clicking each value.
 - c. Options: Choose either English or Metric units.
4. Select Next to begin importing.
5. Once importing is complete, the ODB++ Import Wizard dialog box will disappear. If there are any errors or warnings, they will appear here. You can also view the import log file using F2.

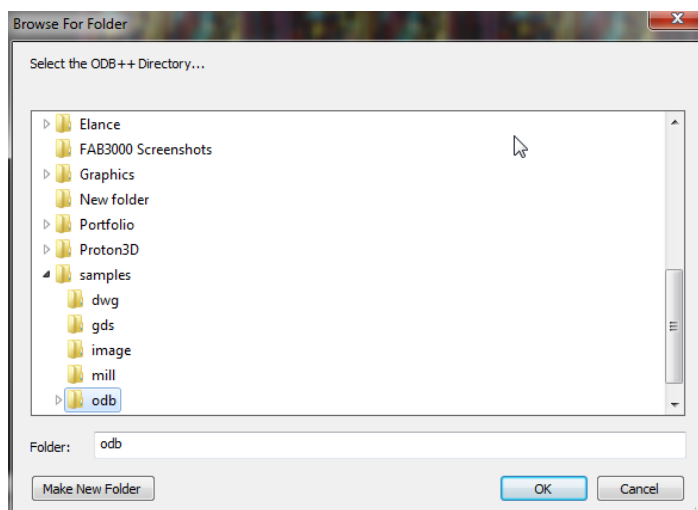
If there are multiple steps in the ODB++ database, FAB 3000 will search the hierarchy to determine which step is the "Master" step. If it cannot find a Master Step, a dialog will appear, prompting you to select which step to view.

Unlike other CAM software products, FAB 3000 loads the complete ODB++ database including all steps (while maintaining hierarchy). To view all the steps imported into FAB 3000, use the Workspace Browser (**File > Workspace Browser**).

File > Import > ODB++ (Directory)



Similar to compressed ODB++ files, FAB 3000 allows you to import ODB++ database directories using the import wizard.



1. Select **File > Import > ODB++ (Directory)**.
2. A menu will appear allowing you to select the ODB++ directory you would like to import. Choose the directory and select **OK**.
3. Use the tabs at the top of the ODB++ import options menu to verify layers, steps, and options.
 - a. **Steps:** Choose which ODB file names to import by using the corresponding check box.

- b. **Layers:** Verify the layers you want to import. Use the drop down menus for layer type and polarity by double clicking each value.

- c. **Options:** Choose either English or Metric units.
4. Select Next to begin importing.
5. Once importing is complete, the ODB++ Import Wizard dialog box will disappear. If there are any errors or warnings, they will appear here. You can also view the import log file using F2.

If there are multiple steps in the ODB++ database, FAB 3000 will search the hierarchy to determine which step is the "Master" step. If it cannot find a Master Step, a dialog will appear, prompting you to select which step to view.

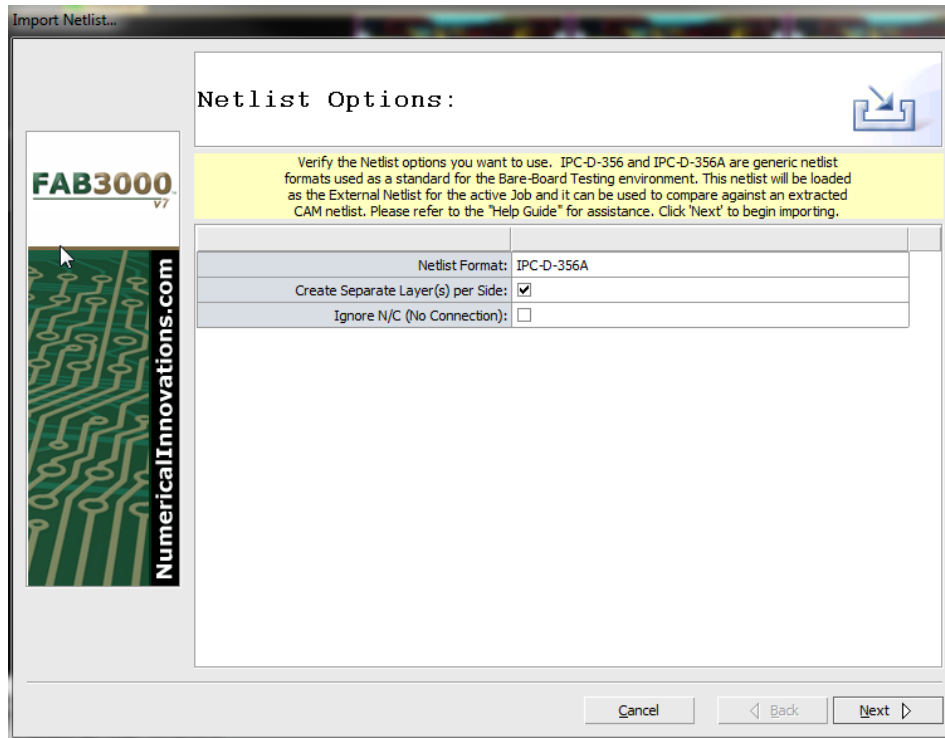
Unlike other CAM software products, FAB 3000 loads the complete ODB++ database including all steps (while maintaining hierarchy). To view all the steps imported into FAB 3000, use the Workspace Browser (**File > Workspace Browser**).



[Video: Convert Gerber to ODB++](#)

File > Import > Netlist (IPC-D-356 and IPC-D-356A)

FAB 3000 can import IPC Netlists which are generic Netlist formats used as a standard for the bare-board testing environment. The Netlist will be loaded as the External Netlist for the active Job and it can be used to compare against an extracted CAM list. To import a Netlist:



1. Select **File > Import > IPC Netlist**.
2. Select the Netlist file you want to import. Select **OK**.
3. Choose the Netlist Format (IPC-D-356 or IPC-D-356A) you wish to import from the Import Netlist Wizard dialog box.
4. Select **Next** to begin the import.

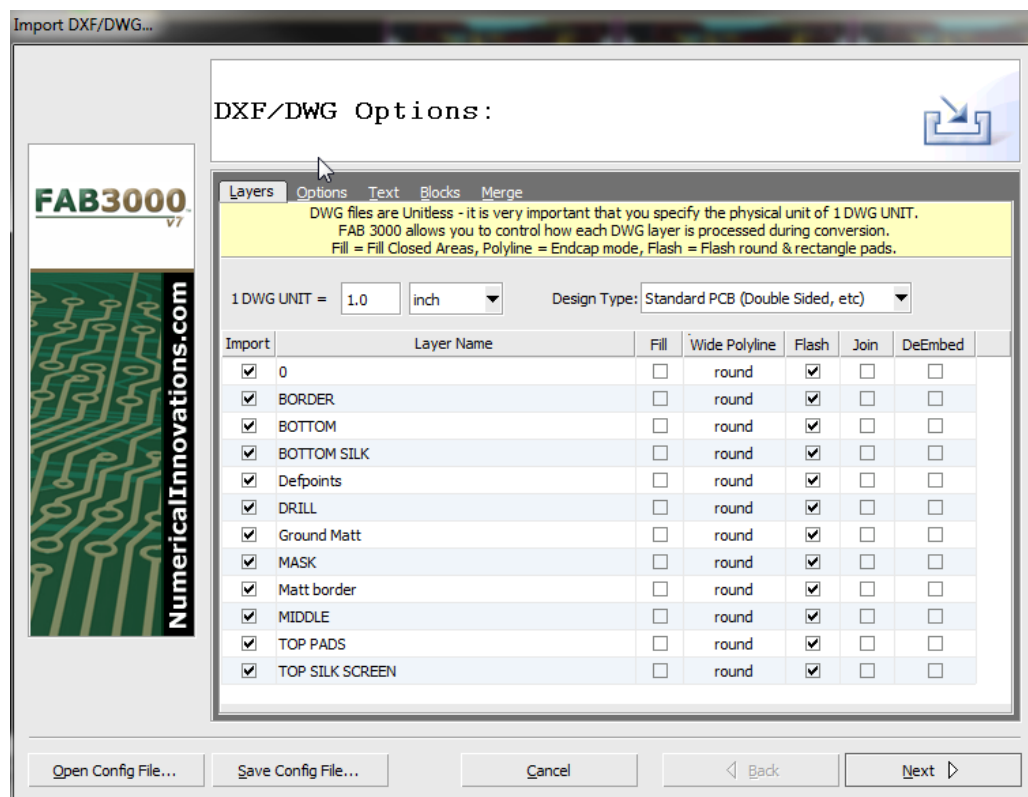
Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import.



[Video: Import Nets, Move Origin, and Perform Netlist](#)

File > Import > DXF/DWG

FAB 3000 can import DXF/DWG files into a workspace.



To import DXF/DWG files, perform the following:

1. Select **File > Import > DXF / DWG**.
2. Choose the DXF file to import.
 - a. DXF files are scanned by layers, blocks, styles, and more.
 - b. DXF files are unit-less. It is import that you verify the DXF units & scale factor of your DXF file before importing it.
3. Choose DXF Design Type from the pull down menu. This feature is optional and adjusts other DXF import settings to optimize conversion results.
4. Verify the DXF Layers in the **Layer Tab** of the import wizard dialog box. The following columns are available for your review & edit:
 - a. Import: Includes the layer during import.
 - b. Fill: Converts closed boundaries to filled-polygons.

- c. Wide Polyline: Adjusts how polyline width is handled during import. Choose from either Extend, Truncate, or Round.
 - d. Flash: Check this box to detect circles & rectangles – then convert them to Gerber flashes.
 - e. Join: Joins layers for import.
 - f. De-Embed: Algorithm used to verify polygon-inside-of-polygon. Please note this is a time consuming feature.
5. Verify DXF Options in the **Options Tab** of the import wizard dialog box. There are several options that can affect how a DXF file is imported (Join Adjacent Lines/Arcs, Arc-Resolution, Respect Byblock/Bylayer).
 6. Verify DXF Text in the **Text Tab** of the import wizard dialog box. This allows you to verify and select the mapping of AutoCAD styles to FAB 3000 fonts.
 7. Verify DXF Blocks in the **Blocks Tab** of the import wizard dialog box. This allows you to control how each individual DXF block is handled during import.
 8. Merge DXF Layers in the **Merge Tab** of the import wizard dialog box. This allows you to merge DXF layers during import. Create a merge and assign DWG layers to be included in this merge.
 9. Select next to begin the import.

Opening and Saving Config Files (Optional): This feature allows you to open & save an existing DXF Settings file (*.ds), so that you may assign the exact same settings used previously. If you regularly perform DXF conversions re-using the DXF settings will save time and limit mistakes.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

Import DXF Units:

DXF Units/Scale:	DXF files are unit-less, so it is vital to specify the proposed units & scale for 1 DXF physical unit. Choose from Micron, Inch, Millimeter, Centimeter, and more. The Logical DXF unit size will be defined by defined by DXF Units * Scale
DXF Design Type:	This is an optional convenience setting, which will optimize other DXF import settings such as Fill, Polyline End cap, and more. Choose whatever description best describes the type of DXF design you are importing.

Import DXF Options:

Use Polygon De-Embedding:	Automatically solves the polygon-inside-of-polygon polarity problem. Closed boundaries (on the same layer) found completely inside another boundary will be made clear (or dark). Due to its complexity, this feature may be time consuming during the conversion.
Respect ByBlock/ByLayer:	Block entities that are on designed layer '0', will be changed to the layer of the respective Insert.
Ignore Dimensions:	If this setting is 'checked', dimensions will be omitted.
Hatch Mode:	Determines how to process DXF "Solid" hatches: Ignore: All Solid Hatches will be omitted. Normal: All Solid Hatches will be included. Use De-Embedding: All holes in Solid Hatches will be separated and formed into a respective composite.
Arc Resolution (Degrees):	Accuracy used when mapping points to recreate arcs & circles.
Default Line Width:	All 0-width DXF entities (such as lines, arcs, 0-width polylines) will be assigned this physical width (if they do not form a filled-polygon).
Flash Circle/Rects Less Than:	All detected Circles & Rectangle shapes will only be flashed in their bounding width & length are less than this value. (For Gerber Export Only)

Import DXF Layers:

Import:	If "checked" this layer will be imported. If "unchecked", this layer will be omitted.
DXF Layer Name:	The DXF layer name. This is merely for reference, and cannot be changed.
Fill:	If "checked", all closed boundaries for this layer will become filled polygons. If "unchecked", all closed boundaries for this layer will be open paths, and use the Default Line Width .
Polyline:	This setting controls the End caps for open-polylines with width on this layer. Truncate: Square End cap, no extension. Extend: Square End cap, 1/2 width extension. Round: Round End cap, 1/2 width extension.

Auto-Flash:	If "checked", all Rectangles and Circles on this layer, which are smaller than the setting "Flash Circle/Rects Less Than", will be converted to Gerber Flashes. (For Gerber Export Only)
Comp:	If non-zero, apply the offset-compensation to all resulting boundaries on this layer.

Import DXF Blocks:

Import:	If "checked" this block will be imported. If "unchecked", this block will be omitted.
DXF Layer Name:	The DXF block name. This is merely for reference, and cannot be changed.

Import DXF Styles:

DXF Style:	The DXF style name. This is merely for reference, and cannot be changed.
FAB 3000 Font File:	Choose which Pre-Compiled FAB 3000 Font File to use from the pull-down. All FAB 3000 Font files are derived from TrueType fonts respectively. You may compile a FAB 3000 Font file from any TrueType font.
Default Text Style:	Assign the default FAB 3000 Font to use when an associated DXF font file cannot not be determined or located.
Text Line Width/Fill Method:	<p>The setting controls the DXF Text Line Width & Filling options:</p> <p>Use Polygon Fill: All Text will be filled as polygons. Their widths are already determined in the FAB 3000 font file itself.</p> <p>Use Same Line Width for All: Determine a standard line width to be used for all DXF text.</p> <p>Use Ration: The text line width will be determined by a ratio of the text height divided by your supplied value.</p>

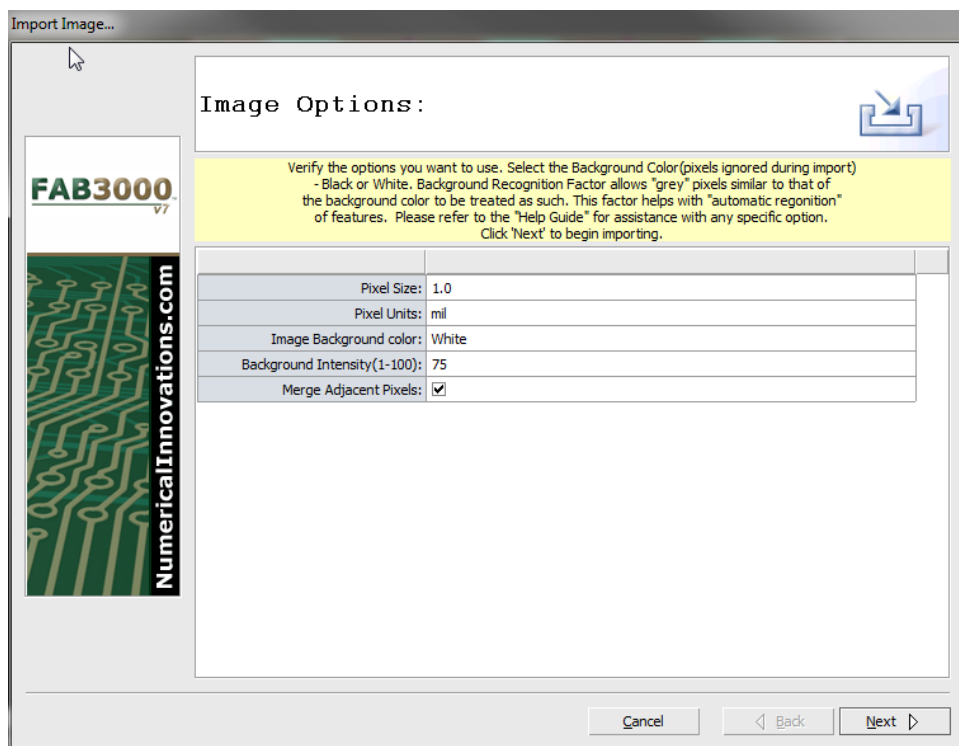
Import DXF Merge:

Merge Name:	The output name to be used for the merged layer.
--------------------	--

Layers (Click to Add):	Select all DXF Layers that will be included in this merged layer.
New:	Create a new "empty" merge layer.
Delete:	Delete an existing merge layer.
Output Merged Layers Only:	Do not output any individual DXF layers, only the merged layers will be created.

Import DXF Open/Save:

Open DXF Settings:	This (Optional) feature allows you to open an existing DXF Settings file (*.ds), so that you may assign the exact same settings used previously. If you regularly perform DXF conversions re-using the DXF settings will save time and limit mistakes.
Save DXF Settings:	This (Optional) feature allows you to save your DXF Settings to a file (*.ds), so that you may re-use these same settings at a later date. If you regularly perform DXF conversions re-using the DXF settings will save time and limit mistakes.



File > Import > Image

FAB 3000 will import images from .TIFF, .JPG, .BMP, & .GIF file types into a workspace.

To import an image file into the workspace, perform the following:

1. Select **File > Import > Image**.
2. Choose the images you would like to import and select OK. The Import Image wizard dialog box will appear.
 - a. To select more than one image file to import, hold either the Shift or Ctrl key.
3. Select the Pixel Size from the dropdown menu.
4. Select the units from the dropdown menu. Units are in Meter, Mil, Inch, Millimeter, Centimeter and DPI.
5. Choose an Image Background Color of either white or black.
 - a. Background color and intensity allow "grey" pixels similar to that of the background color to be treated as such. As a result, this helps with automatic recognition of features.
6. Choose the Background Intensity (1-100).
 - a. Try experimenting by adjusting this setting for optimum results.
7. Select Next to begin import.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

Import Image (TIFF, JPEG, BMP, GIF) Options:

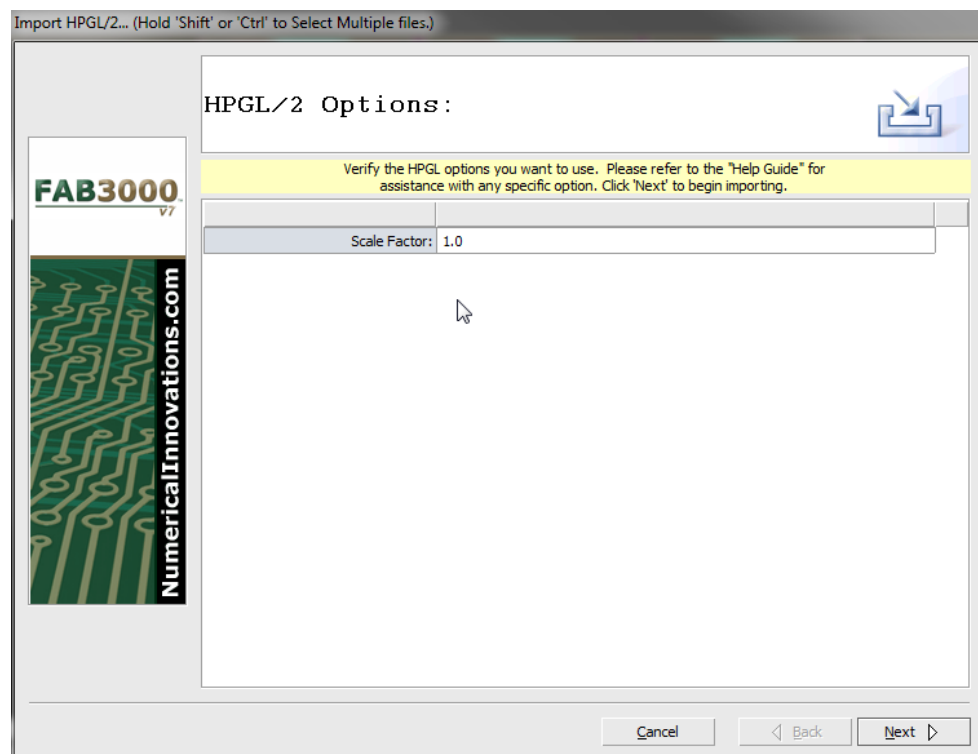
Pixel Unit:	Specify the proposed units for each pixels physical size. Choose from Micron, Inch, Millimeter, Centimeter, and more.
Pixel Size:	Logical pixel size defined by Pixel Units * Pixel Size
Image Background Color:	Select the image's transparent color that will be ignored during importing. Choose either Black or White.
Background Intensity (1-100):	Images such as JPEG's incorporate a type of blending which leads to very few pixels actually being black or white. This setting is a ratio which allows pixels with colors similar to the transparent color to be treated as such. This setting has proved very useful for "inspection tools" which take photos of microscopic devices, and output them as jpegs; and then want these photos converted back to GDSII or Gerber for comparison purposes.

Merge Adjacent Pixels:

For Advanced Users Only. This option will combine adjacent pixels to form polygons. This feature is very time consuming and not recommend for use with complex images.

File > Import > HPGL/2

FAB 3000 is also capable of importing HPGL/2 files.

**To import HPGL/2 files, perform the following:**

1. Select **File > Import > HPGL/2**.
2. Select the HPGL/2 files you want to import. Select **OK**. The Import HPGL/2 wizard dialog box will appear.
 - a. To select more than one HPGL/2 file, hold down the **Shift** or **Ctrl** key.
3. Optional: Select whether you want to scale the entire file. Use **Scale Factor** and enter a value for scale size.
4. Select **Next** to begin importing HPGL/2 files.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

Import HPGL/2 Options:

Scale Factor	Assign HPGL/2 Scale factor (default is 1).
---------------------	--

File > Import > Postscript/PDF

Import Postscript/PDF...

Postscript/PDF Options

Requires GhostScript Version 7.04 or higher (32-bit)

Verify the PostScript options you want to use: Page Number, Map Objects, and Ghostscript Options. To change the overall quality and precision of curves adjust the option 'Spline Deviation'. Please refer to the "Help Guide" for assistance with any specific option. Click 'Next' to begin importing.

Page Number to convert:	1
Spline/Arc Deviation (mil):	0.5
Map Layers:	Merge All to a Single Layer
Convert Paths to:	Auto-Detect
Fill Area Mode:	Treat White Areas as Holes (Auto-Detect)
Flash Round Shapes:	<input type="checkbox"/>
Flash Rectangle Shapes:	<input type="checkbox"/>
Text Output:	Polygon
Default FAB 3000 Font:	arial.fnt
Additional GhostScript Option:	

Select Ghostscript DLL/EXE (Required)

Cancel Back Next

This allows you to import a Postscript or PDF file into an active workspace.

Note: Ghostscript is required on your workstation or server in order to import Postscript. It is freely available and distributed under the GNU General Public License. If you do not have Ghostscript installed you may download it from the following websites (or from other available mirrors):

<http://www.cs.wisc.edu/~ghost/doc/gnu/index.htm>

<http://www.ghostscript.com/>

To import a Postscript or PDF file, perform the following:

1. Select **File > Import > Postscript/PDF**.
2. Choose the **Postscript or PDF** file you want to import. Select **OK**.
3. Edit and verify the options in the dialog box.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

Import Postscript/PDF Options

Page Number to Convert:	Select the page number to convert. Many Postscript and PDF files have multiple pages. The importer will only import one page at a time. Usually the first page.
Spline/Arc DeViation (mil):	Approximation accuracy to use for Arcs and Spline objects.
Map Layers:	
Convert Paths to:	
Fill Area Mode:	
Flash Round & Rect Shapes:	Convert round and rectangular objects to flashes. This results in a more optimized data.
Text Output:	
Default FAB 3000 Font:	
Ghostscript Additional Options:	Assign specific Ghostscript options. Requires knowledge of the Ghostscript interpreter.



[Video: Convert PDF to Gerber & Drill – Part 1](#)

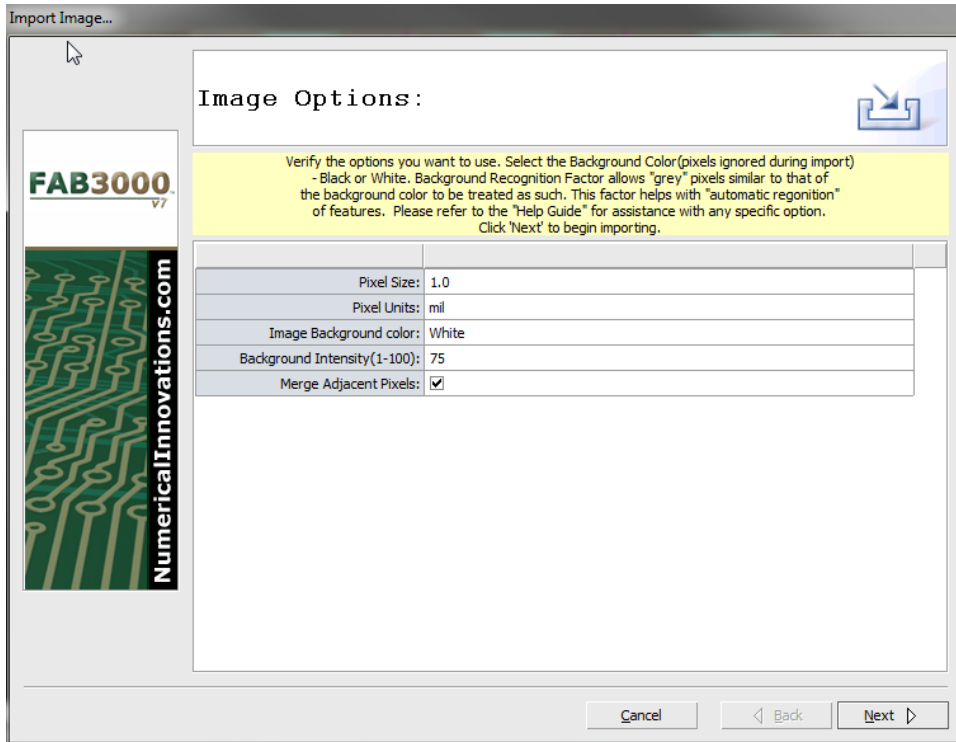
[Video: Convert PDF to Gerber & Drill – Part 2](#)

[Video: Convert PDF to PCB Gerber Data – Avoid Raster PDF Files](#)

[Video: Import PDF and Export to Gerber](#)

[Video: PDF to Gerber Conversion](#)

File > Import > Large Bitmap



To import a Large Bitmap file into the workspace, perform the following:

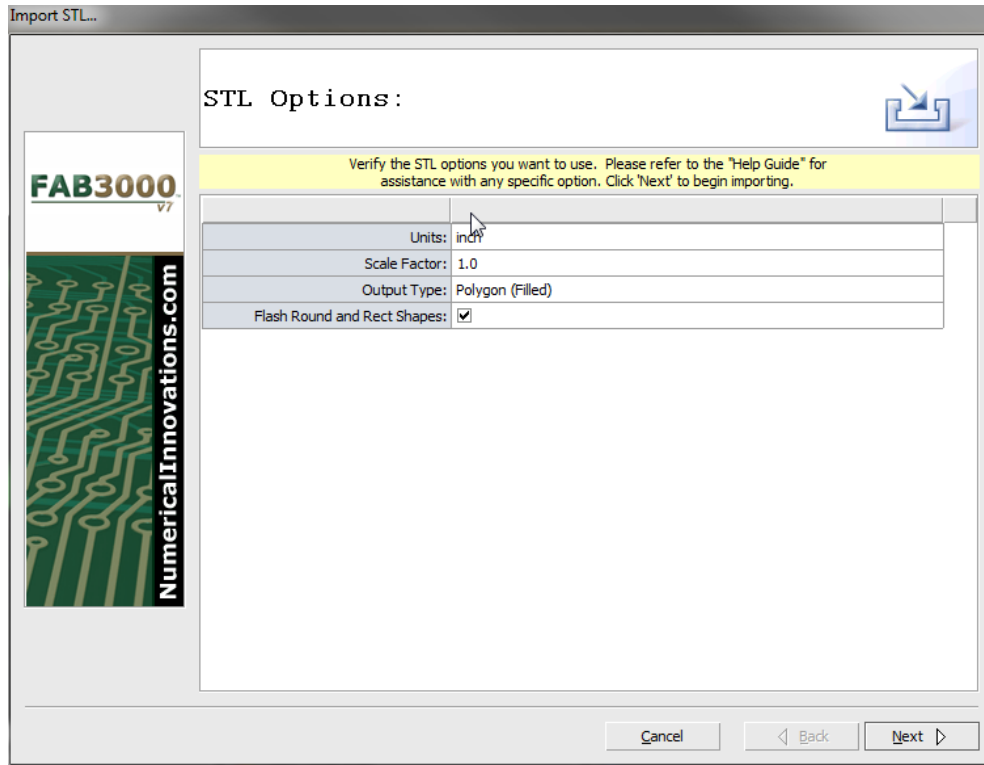
1. Select **File > Import > Large Bitmap**.
2. Choose the images you would like to import and select OK. The Import Image wizard dialog box will appear.
 - a. To select more than one image file to import, hold either the Shift or Ctrl key.
3. Select the Pixel Size from the dropdown menu.
4. Select the units from the dropdown menu. Units are in Meter, Mil, Inch, Millimeter, Centimeter and DPI.
5. Choose an Image Background Color of either white or black.
 - a. Background color and intensity allow "grey" pixels similar to that of the background color to be treated as such. As a result, this helps with automatic recognition of features.
6. Choose the Background Intensity (1-100).
 - a. Try experimenting by adjusting this setting for optimum results.
7. Select Next to begin import.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

Import Image (TIFF, JPEG, BMP, GIF) Options:

Pixel Unit:	Specify the proposed units for each pixels physical size. Choose from Micron, Inch, Millimeter, Centimeter, and more.
Pixel Size:	Logical pixel size defined by Pixel Units * Pixel Size
Image Background Color:	Select the image's transparent color that will be ignored during importing. Choose either Black or White.
Background Intensity (1-100):	Images such as JPEG's incorporate a type of blending which leads to very few pixels actually being black or white. This setting is a ratio which allows pixels with colors similar to the transparent color to be treated as such. This setting has proved very useful for "inspection tools" which take photos of microscopic devices, and output them as jpegs; and then want these photos converted back to GDSII or Gerber for comparison purposes.
Merge Adjacent Pixels:	For Advanced Users Only. This option will combine adjacent pixels to form polygons. This feature is very time consuming and not recommend for use with complex images.

File > Import > STL



FAB 3000 imports stereolithography files with ease. Whether you're a designer, manufacturer, or 3D printer, you can import .STL files into the workspace.

To import .STL files into FAB 3000, perform the following:

1. Select **File > Import > STL (3D)**.
2. Choose the STL file you want to import. Select OK. The Import STL wizard dialog box will appear.
3. Edit and verify the Units, Scale Factor, Output Type, and Flash Round/Rect Shapes.

Once the import is complete, the import wizard dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the wizard dialog box.

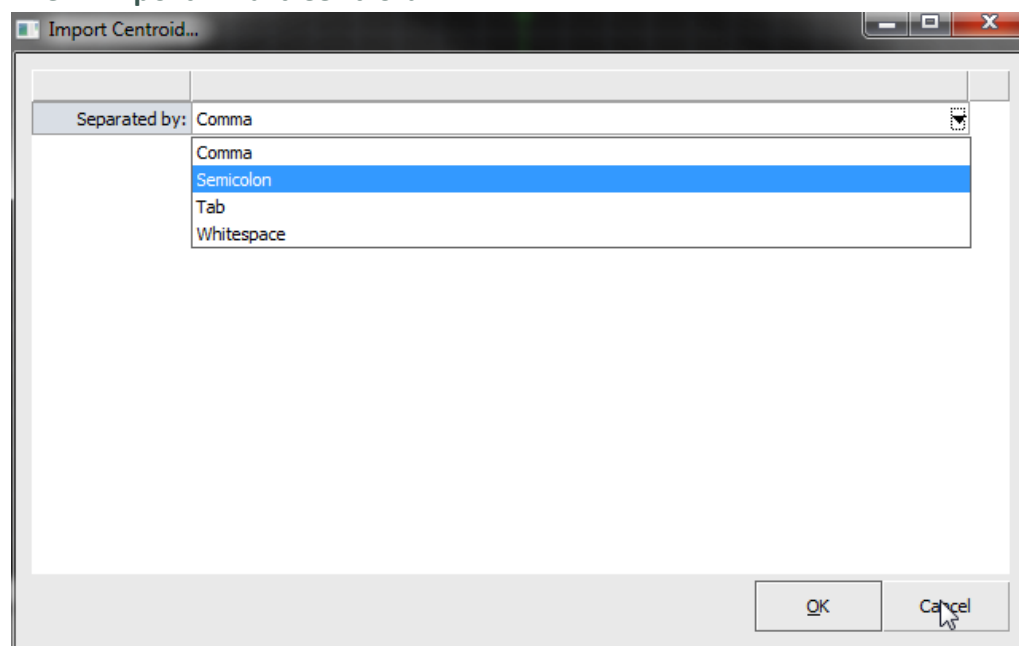
Import STL (3D) Options:

Units:	Choose the units for the imported STL.
Scale Factor:	Logical pixel size defined by Units * Scale Factor
Output Type:	Output STL boundaries as Filled Polygon, or Outlined.
Flash Round & Rect Shapes:	Convert round and rectangular objects to flashes. This results in a more optimized data.

File > Import > GDSII (Stream)

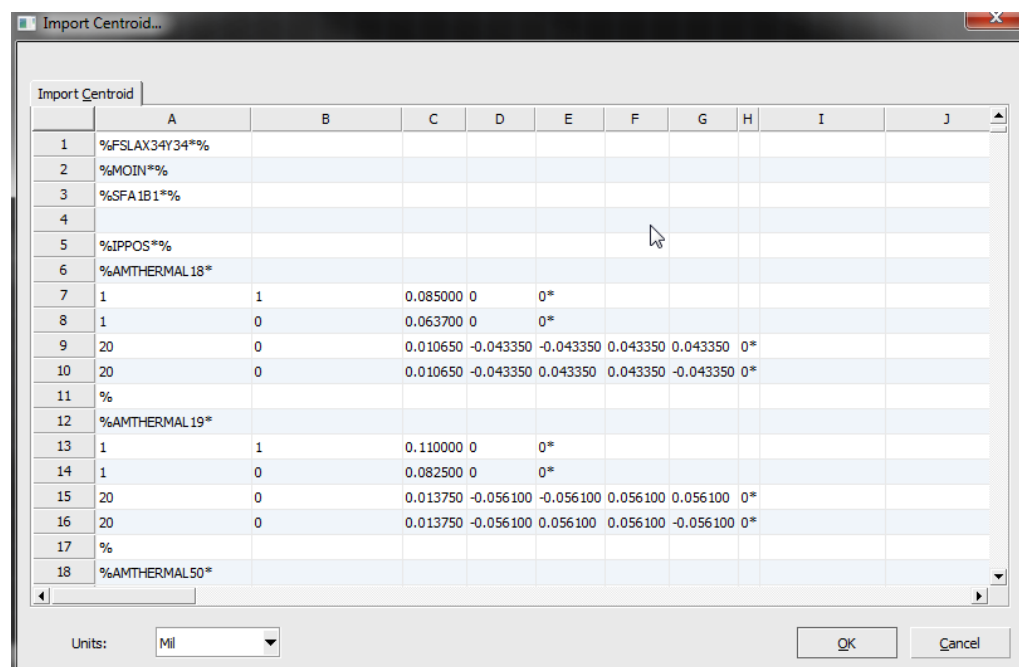
Loads GDSII binary format into the active workspace.

1. Select File > Import > GDSII (Stream).
2. Choose the file you wish to import.
3. Select **OK**.

File > Import > Part Centroid

Part Centroid files are ASCII formatted files that contain centroid data. Centroid files are commonly separated by comma or tab.

FAB 3000 loads Part Centroid files into a spreadsheet for editing.



To import a **Part Centroid file**, perform the following:

1. Select **File > Import > Part Centroid**.
2. Choose the Centroid file you want to import. Select OK.
3. Choose the separation character you will use for the import. This will be either Comma, Semicolon, Tab or Whitespace. Select OK when complete.

The centroid file will now be imported and displayed in a grid table.

4. Assign the "**Ref. Des**" column variable, by clicking into a cell that contains the Reference Designators and choose menu item: "Column: Ref. Des"

You will now see at the top of the column a check mark and the title REFDES. This lets you know that the Ref Des column has been assigned.

5. Assign the "**Layer**" column variable, by clicking into a cell that contains the Layer (usually labeled either: Top or Bottom) and choose menu item: "Column: Layer"

You will now see at the top of the column a check mark and the title LAYER. This lets you know that the Layer column has been assigned.

6. Assign the "**X**" column variable, by clicking into a cell that contains the X coordinate and choose menu item: "Column: X Location"

You will now see at the top of the column a check mark and the title X. This lets you know that the X column has been assigned.

7. Assign the "**Y**" column variable, by clicking into a cell that contains the Y coordinate and choose menu item: "Column: Y Location"

You will now see at the top of the column a check mark and the title Y. This lets you know that the Y column has been assigned.

8. Assign the "**Rotation**" column variable, by clicking into a cell that contains the part rotation and choose menu item: "Column: Rotation"

You will now see at the top of the column a check mark and the title ROT. This lets you know that the rotation column has been assigned.

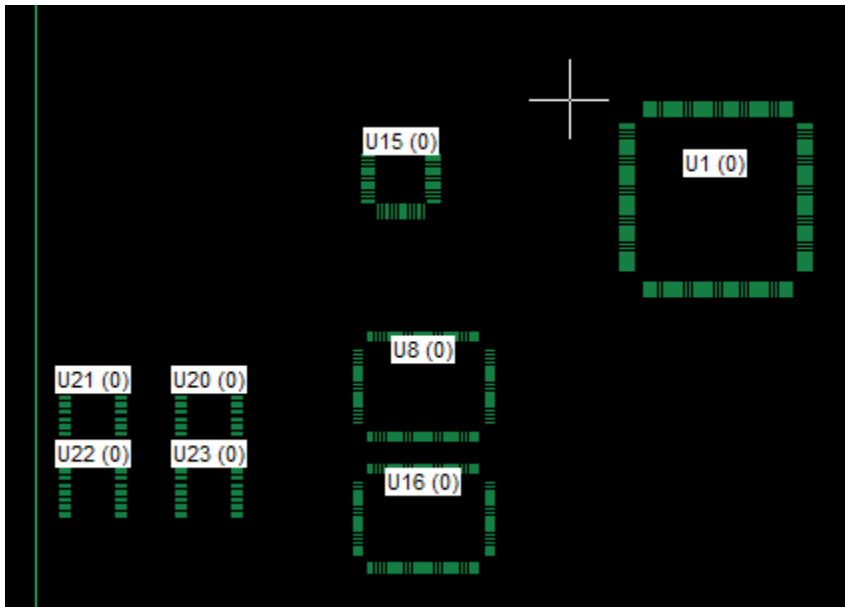
9. Assign the "**Start Row**" column variable, by clicking into a cell that contains the first centroid definition and choose menu item: "Start Row"

You will now see at the left of the row a check mark and the title ST. This lets you know that the start row has been assigned.

10. Assign the "**End Row**" column variable, by scrolling down & clicking into a cell that contains the last centroid definition and choose menu item: "End Row"

You will now see at the left of the row a check mark and the title END. This lets you know that the start row has been assigned.

11. Verify the Import Units (located in the Lower/Left) used for the X & Y Coordinates.
12. Press "OK" to begin importing the centroid file.
13. Once the Centroid loading is complete, a dialog will appear giving you the count & status.

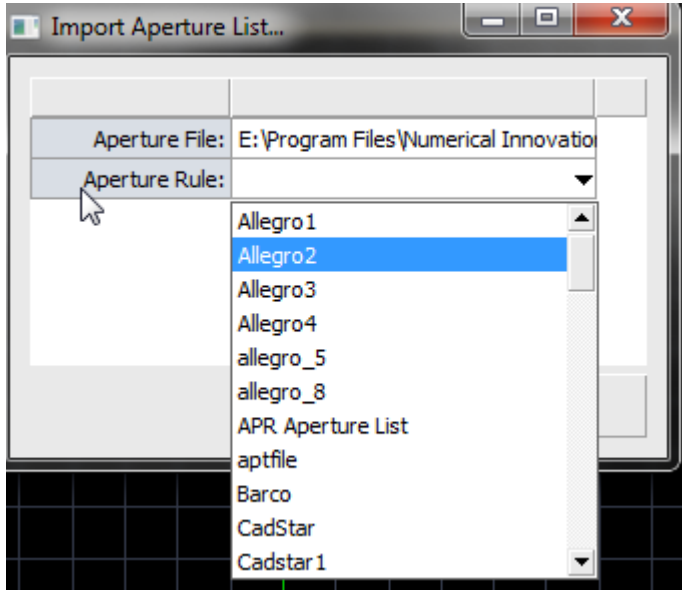


You will now see the external centroid data displayed in the FAB 3000 screen. To control the display of the external centroid files, select **View > External Controls**.

File > Import > BOM File

Loads a Bill of Materials File into the active workspace.

1. Select **File > Import > BOM File**
2. Choose the file you wish to import.
3. Select **OK**.

File > Import > Aperture List

FAB 3000 can import an aperture list in various formats.

The loaded file overwrites the current aperture data, and there is no Dcode translation. In most cases you will want to use this feature for loading specific aperture files that do not load using **Autoload Gerber Wizard**.

FAB 3000 does not detect the aperture list format using this command, so you must also define which **Aperture Wizard Rule** to use during import.

Import Aperture List provides an excellent way to debug Aperture Wizard Rules that you have created with the **Aperture Rules Editor**. To verify

results of a newly created Aperture Wizard Rule: Create a new Job, Import the aperture list, and review the **Aperture Table**. To import an aperture list, perform the following:

1. Select **File > Import > Aperture List**.
2. Choose the Aperture files you want to import. Select OK.

The Import Aperture List dialog box will appear.
3. Select the Aperture Rule to use while importing the Aperture List.
4. Select OK to begin importing the Aperture file.

Once the import is complete, the import dialog box will disappear unless FAB 3000 encounters any errors or warnings during import. If there are any issues, FAB 3000 will display them within the dialog box.

File > Import > Custom Aperture Library

With FAB 3000, Custom Aperture Libraries can be imported into an active workspace.

Select File > Import > Custom Aperture Library.

2. Choose the file you wish to import.
3. Select **OK** to begin importing the Custom Aperture Library file.



[Forum: How do I delete unwanted aperture rules?](#)

[Video: Reading Old Aperture Files](#)

[Video: Create Aperture List Reader for RS-274D Gerber Files](#)

File > Import > Footprints Library

With FAB 3000, Footprints Library Files can be imported into an active workspace.

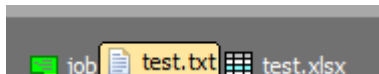
Select File > Import > Footprints Library.

2. Choose the file you wish to import.
3. Select **OK** to begin importing the Footprints Library file.

File > Import > Text File

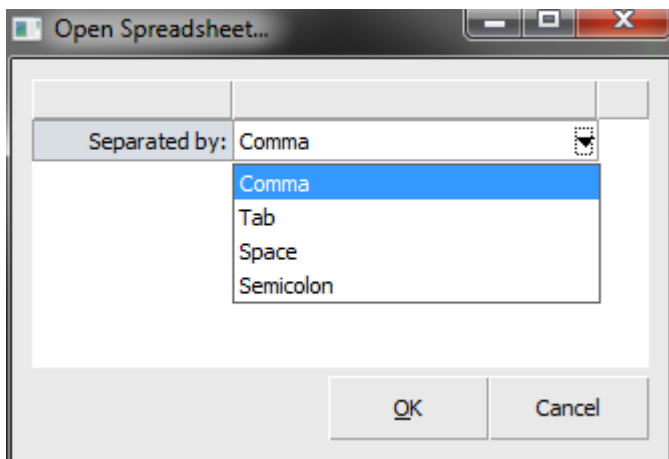
With FAB 3000, Text Files can be imported into an active workspace.

1. Select File > Import > Text File.
2. Choose the file you wish to import.
3. Select **OK** to begin importing the Text File.



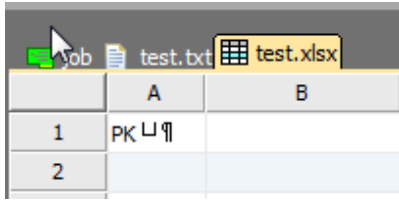
The Text File will be created as a separate tab in the workspace and listed in the Tab Editor.

File > Import > Spreadsheet File



1. Select File > Import > Spreadsheet File.
2. Choose the file you wish to import.
3. Choose the separation character you will use for the import. This will be either Comma, Tab, Space, or Semicolon. Select **OK** when complete.
4. Select **OK** to begin importing the Spreadsheet File.

The Spreadsheet File will be created as a separate tab in the workspace and listed in the Tab Editor.



File > Import > FAB 3000 Matrix

The FAB 3000 Import Matrix File permits external tools (like PCB Design software) to predefine layers and Jobs for intelligent loading into FAB 3000 for instant Gerber Viewing & Verification. This prevents the monotonous steps of loading Gerber & Drill files, assigning layer colors, assigning layer types, defining layer stack ups, defining blind/buried drill layers (if required), etc.; allowing PCB designers to easily use Gerber/CAM software for its primary purpose: to quickly verify their Gerber & Drill files are correct and ready for manufacturing.

The FAB 3000 Import Matrix file also supports PANEL arrays which allow the merging of several PCB Jobs together so they can be manufactured simultaneously - saving \$\$\$ on fabrication costs. In addition, FAB 3000 will automatically performs all necessary object transformations, Dcode & drill tool transcoding.

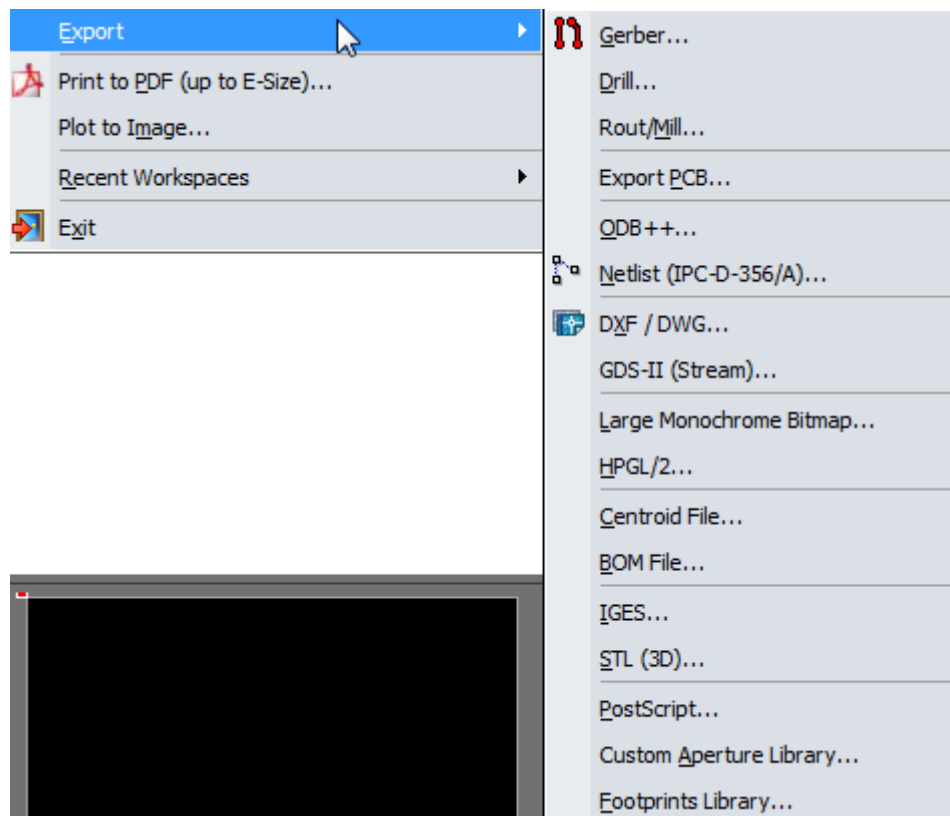
Loads a FAB 3000 Matrix File into the active workspace.

1. Select **File > Import > FAB 3000 Matrix**.
2. Choose the file you wish to import.
3. Select **OK**.

Note: Simply call FAB 3000 and pass the matrix file as the first argument. All FAB 3000 Import Matrix files must end with *.FM6.



[Video: FAB 3000 Transcode Technique when Loading Multiple Drill and Gerber Files](#)

File >Export

The following file types are possible using File > Export:

Gerber	HPGL/2
Drill	Centroid File
Rout/Mill	BOM File
Export PCB	IGES
ODB ++	STL (3D)
Netlist (IPC-D-356/A)	PostScript
DXF/DWG	Custom Aperture Library
GDS-II (Stream)	Footprints Library
Large Monochrome Bitmap	

File > Export > Gerber

Export Gerber -

FAB3000 V7

Gerber Options:

Verify the Gerber options. To improve accuracy select 'Decimal Digits' of 4 (or higher). Gerber files support polygon fills, composites, step/repeat, etc. For more information please review the help documentation. Click 'Next' for layer options.

Gerber Reports

Export Gerber Format:	RS-274-X
Units:	English
Integer Digits:	3
Decimal Digits:	5
Type:	Absolute
Zero Suppression:	Leading
Use Step/Repeat (when possible):	<input type="checkbox"/>
Use Arcs (G75):	<input type="checkbox"/>
Gerber File Extension:	gbr

Cancel Back Next

Gerber files exported from FAB 3000 are fully compatible with the Gerber specification and support polygon fills, composite images, step and repeat codes, and more. To export Gerber files, perform the following:

1. Select **File > Export > Gerber**. The Export Gerber Wizard dialog box appears.
2. Verify the Export Gerber file options you want to use.

Export Gerber -

FAB3000 V7

Layer Options:

Each selected layer will be exported as an individual gerber file. Using the check boxes in the Export column, select the layers you want to export or deselect the layers you don't want exported. To Select All, right-click within any checkbox column. You may Mirror and/or assign Negative polarity to any layer by selecting the appropriate check boxes. Layers may also be renamed. Click 'Next' to select the export folder.

Layer Display: Show All Layers

Layer Name:	Export:	Neg:	Mirror:
Art0121.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Art02.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
gnd2530.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Pgp2529.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sm0128.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sm0227.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sp0128.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Sst0126.pho	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Drill.drl	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cancel Back Next

3. Select **Next** to proceed to the Layer Options dialog box.

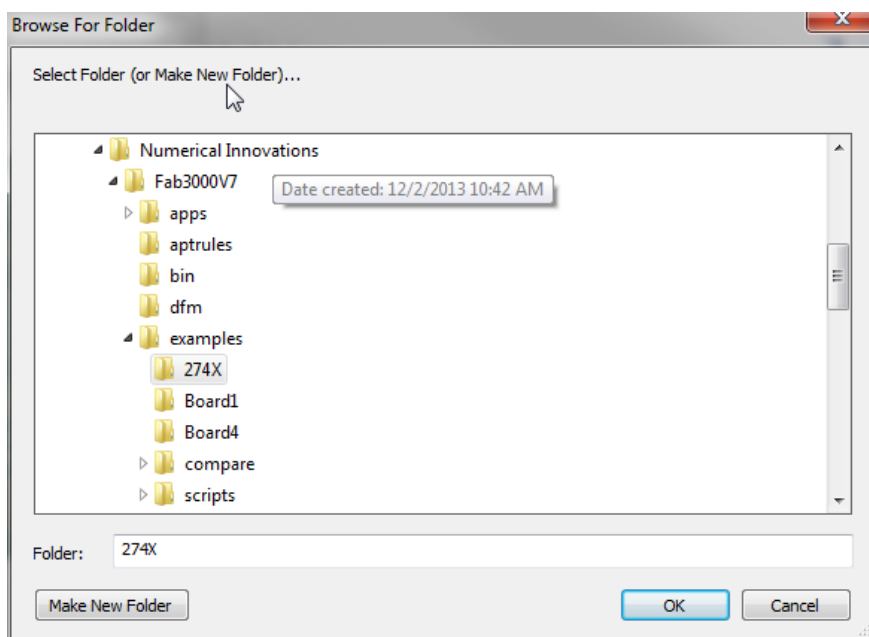
To improve Gerber file accuracy you should select decimal digits of 4 (or higher).

For description of each option, see next page: **Export Gerber Options**

3. If the filenames for the layers need to be changed, type-in the desired filename in the text boxes.

4. Using the check boxes in the Export column, select or unselect the layers you want to export. To Set all or clear all layers, right-click within the Export column.

5. If you want any of the layers to have a negative polarity, select the check box for the appropriate layer(s) in the **Neg** column.
6. If you want to mirror any of your layers upon export, select the appropriate check boxes in the Mirror column.
 - a. Polarity and Mirroring options apply only to 274X, Fire 9xxx, Barco & Gerber files. FAB 3000 will display Negative Layers but not the Mirroring option on files re-imported using **AutoLoad Gerber** or **Gerber Import** will display as normal, but retain their option codes in the Layers table.
 - b. If the layers previously contain Mirror or Polarity options, they will automatically be checked off in the Layer Options.
 - c. Each selected layer will be exported as an individual Gerber file.
7. Select the folder for Gerber export.

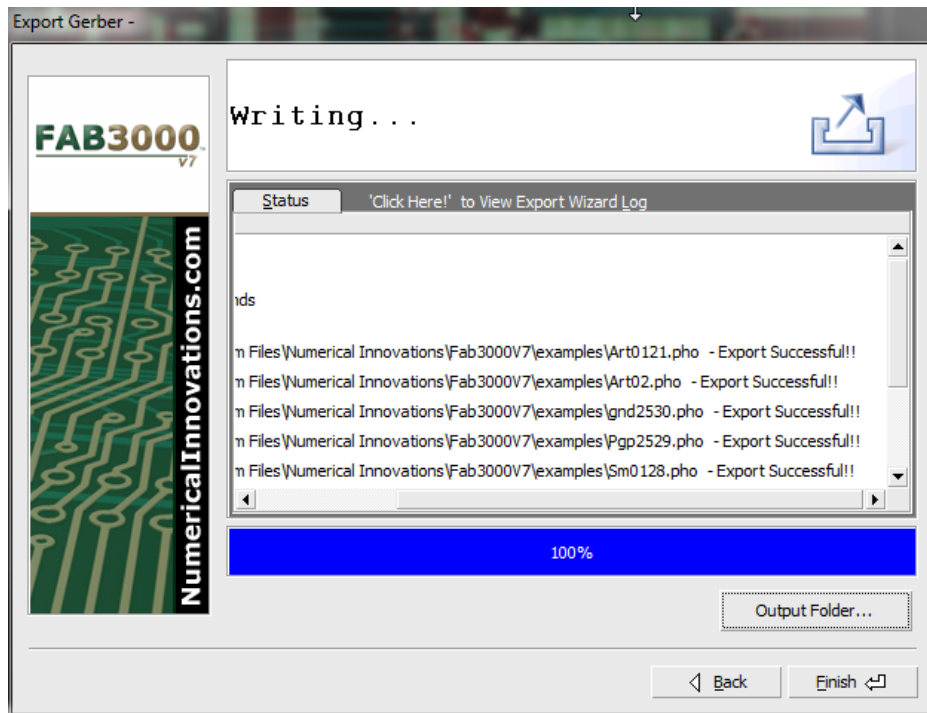


If you select a new folder that does not exist on your system, it will be automatically created. It is recommended that you select a single folder to export all of your files into (i.e. Gerbers, drill, Netlist, notes, fab drawing, etc.)

- a. If you would like to always export your files to a specific folder, you may define the "Export Folder" in the **FAB 3000 Settings Dialog**.



[Video: Easily Convert DXF Files to Gerber](#)



8. Click **OK** to begin exporting your Gerber files.

9. Once export Gerber is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

10. Select **Finish** to exit.

Export Gerber Options:

Export Gerber Format:	Select Gerber format to export. Either RS-274-X or Fire9000.
Units:	Determine exported Gerbers units of English or Metric.
Integer Digits:	Determine exported Gerbers integer digits between (0-6).
Decimal Digits:	Determine exported Gerbers integer digits between (0-6).
Type:	Determine exported Gerbers type of Absolute or Incremental.
Zero Suppression:	Determine exported Gerbers zero suppression of Leading, Trailing, or None.
Use Step/Repeat:	Use Step & Repeat codes if possible.
Use Arcs (G75):	Use arcs when available.
Line Width (mil):	Default line width to assign for 0-width objects such as lines, arcs, etc.
Text Fill:	Specify how to output text (if any): Line: Text will be exploded and outputted as Gerber traces using the Line Width. Solid Fill: Text will be exploded, and output as polygons.
Gerber File Extension:	Default extension to use for exported Gerber file names. (*.gbr) is the default.

File > Export > Drill

Export Drill -

FAB3000 V7

Drill Options:

Verify the drill options you want to use. To improve drill accuracy it is recommended you select 'Decimal Digits' of 4 (or higher). For more information please review the help documentation. Click 'Next' for drill layer options.

Drill Reports

Format:	Excellon 2
Units:	English
Integer Digits:	2
Decimal Digits:	4
Type:	Absolute
Zero Suppression:	Trailing
Use Step/Repeat (when possible):	<input checked="" type="checkbox"/>
Drill File Extension:	drl

Cancel Back Next

To export a Drill file, perform the following:

1. Select **File > Export > Drill**. The Export Drill Wizard dialog box will appear.
2. Verify the Export Drill file options you want to use.
3. Select Next. The **Layer Options** dialog box will appear. To improve drill file accuracy you should select decimal digits of 4 or higher. See the next page for a selection of **Export Drill** options.

Export Drill -

FAB3000 V7

Layer Options:

Using the check boxes in the Export column, select the layers you want to export or deselect the layers you don't want exported. To Select All, right-click within the Export column. Layers may be renamed. Click 'Next' to select the export file location.

Layer Display: Show All Layers

Layer Name:	Export:
Art0121.pho	<input type="checkbox"/>
Art02.pho	<input type="checkbox"/>
gnd2530.pho	<input type="checkbox"/>
Pgp2529.pho	<input type="checkbox"/>
Sm0128.pho	<input type="checkbox"/>
Sm0227.pho	<input type="checkbox"/>
Sp0128.pho	<input type="checkbox"/>
Sst0126.pho	<input type="checkbox"/>
Drill.drl	<input checked="" type="checkbox"/>

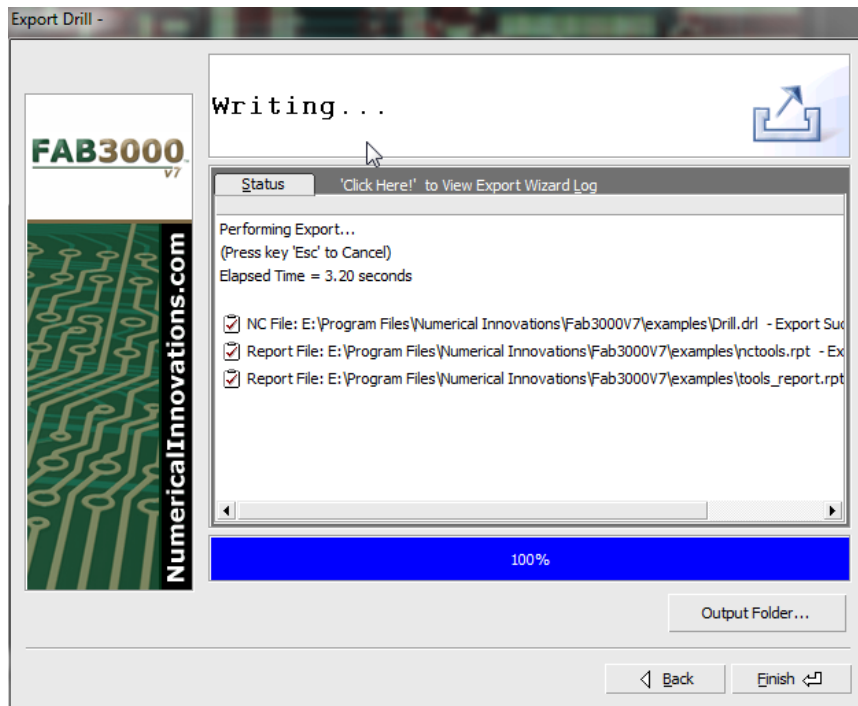
Cancel Back Next

4. Using the check boxes in the Export column, select or deselect the layers you want to export. To set or clear all layers, right click within the Export column.

5. Select Next. Choose a folder to export drill files to.
 - a. If you select a new folder that does not exist on your system, it will be automatically created. It is recommended that you select a single folder to export all your files into (i.e. Gerbers, drill, Netlist, notes, fab drawing, etc.)

If you would like to always export your files to a specific folder, you may define the "Export Folder" in the **FAB 3000 Settings Dialog**.

6. Select Next to begin exporting your drill files.



7. Once export drill is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.


8. Select **Finish** to exit.

Export Drill Options

Units:	Determine exported NC Drill units of English or Metric.
Integer Digits:	Determine exported NC Drill integer digits between (0-6).
Decimal Digits:	Determine exported NC Drill integer digits between (0-6).
Type:	Determine exported NC Drill type of Absolute or Incremental.
Zero Suppression:	Determine exported NC Drill zero suppression of Leading, Trailing, or None.
Use Step/Repeat:	Use Step & Repeat codes if possible.
NC Drill File Extension:	Default extension to use for exported NC Drill file names. (*.drl) is the default.

File > Export > Rout / Mill

Export Wizard...



Rout/Mill Options:

Verify the Rout/Mill options you want to use. To improve accuracy it is recommended you select 'Decimal Digits' of 4 (or higher). For more information please review the help documentation. Click 'Next' for NC Mill layer options.

Mill/Rout | Reports

Format:	Excellon 2
Units:	English
Integer Digits:	2
Decimal Digits:	4
Type:	Absolute
Zero Suppression:	Trailing
Use Step/Repeat (when possible):	<input checked="" type="checkbox"/>
Use Arcs (when possible):	<input checked="" type="checkbox"/>
Mill File Extension:	rou

Cancel
◀ Back
Next ▶


The **Export > Rout/Mill** command is used to export layers into a Rout file.

1. Select **File > Export > Rout**. The Export Rout wizard dialog box will appear.
2. Verify the Rout options you want to use and select **Next**. The Layer options dialog box will appear.

For a description of Export Rout Options, see next page.

To improve accuracy, you should select decimal digits of 4 or higher.

Export Wizard...



Layer Options:

Using the check boxes in the Export column, select the layers you want to export or deselect the layers you don't want exported. To Select All, right-click within the Export column. Layers may be renamed. Click 'Next' to select the export file location.

Layer Display:
Show All Layers

Layer Name:	Export:
Art0121.pho	<input type="checkbox"/>
Art02.pho	<input type="checkbox"/>
gnd2530.pho	<input type="checkbox"/>
Pgp2529.pho	<input type="checkbox"/>
Sm0128.pho	<input type="checkbox"/>
Sm0227.pho	<input type="checkbox"/>
Sp0128.pho	<input type="checkbox"/>
Sst0126.pho	<input type="checkbox"/>
Drill.drl	<input type="checkbox"/>

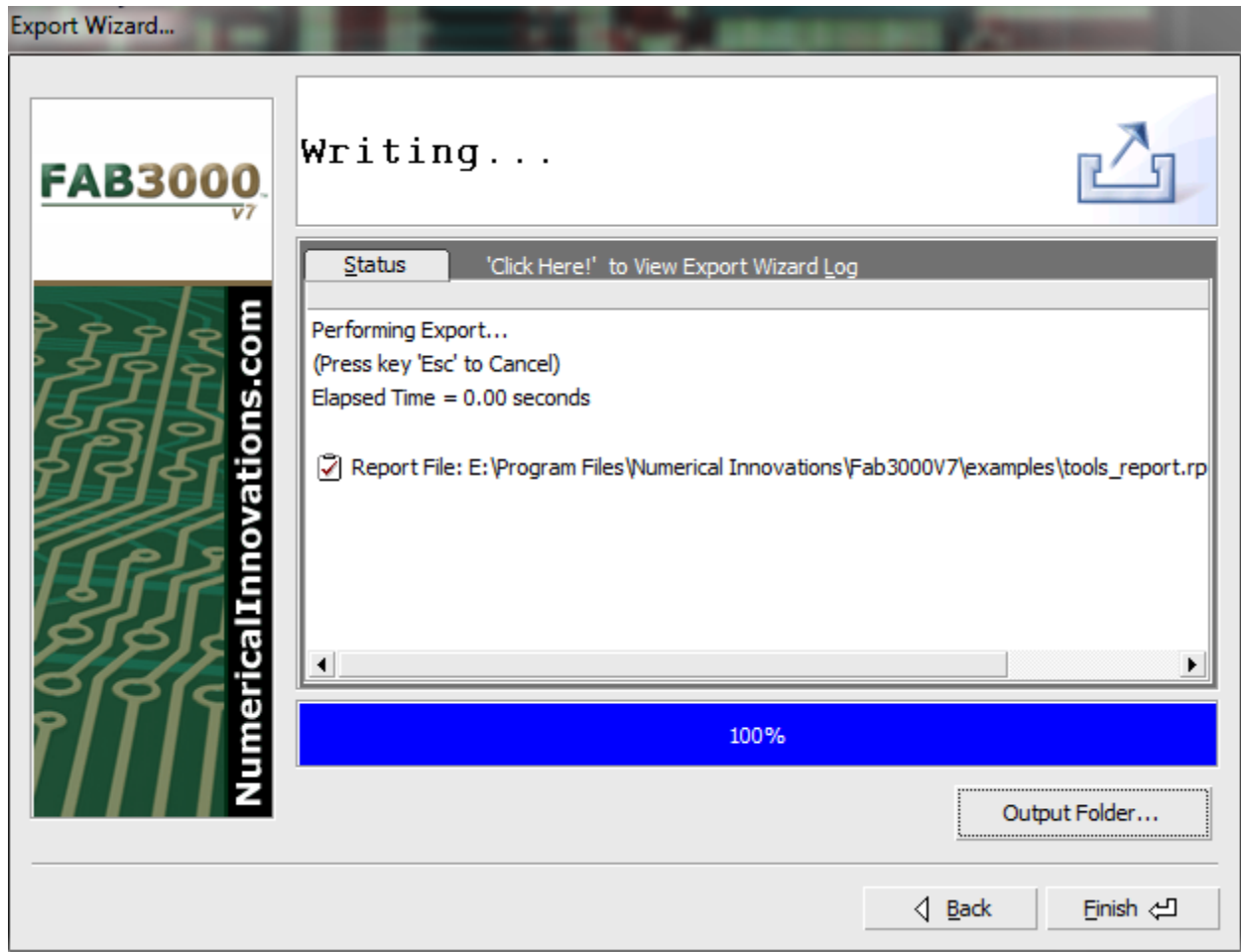
Cancel
◀ Back
Next ▶

3. If the filenames for the layers need to be changed, type-in the desired filename in the text boxes.
4. Using the check boxes in the Export column, select or unselect the layers that you want to export. To set or clear all layers, right click within the Export column.
5. Select **Next**. A folder dialog box will appear. Choose a folder to export rout files to.

If you select a new folder that does not exist on your system, it will be automatically created. It is recommended that you select a single folder to export all your files into (i.e. Gerbers, drill, Netlist, notes, fab drawing, etc.)

If you would like to always export your files to a specific folder, you may define the "Export Folder" in the **FAB 3000 Settings Dialog**.

6. Select **OK** to begin exporting your rout files.

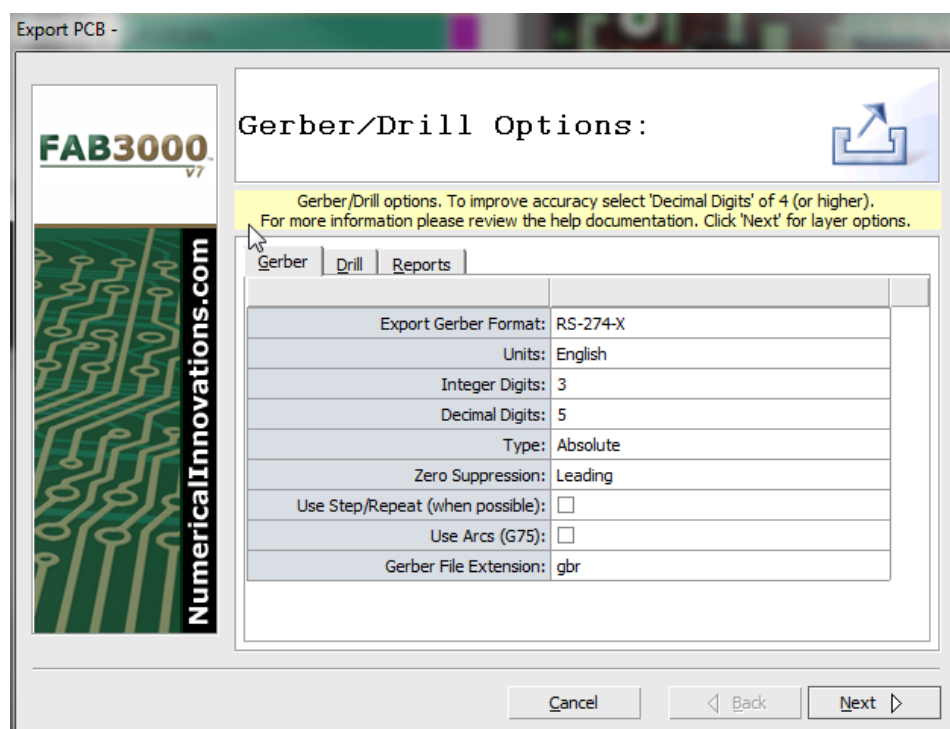


7. Once export rout is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.
8. Select **Finish** to exit.

Export Rout Options

Units:	Determine exported NC units of English or Metric.
Integer Digits:	Determine exported NC integer digits between (0-6).
Decimal Digits:	Determine exported NC integer digits between (0-6).
Type:	Determine exported NC type of Absolute or Incremental.
Zero Suppression:	Determine exported NC zero suppression of Leading, Trailing, or None.
Use Step/Repeat:	Use Step & Repeat codes if possible.
NC Rout File Extension:	Default extension to use for exported NC Rout file names. (*.rou) is the default.

File > Export PCB

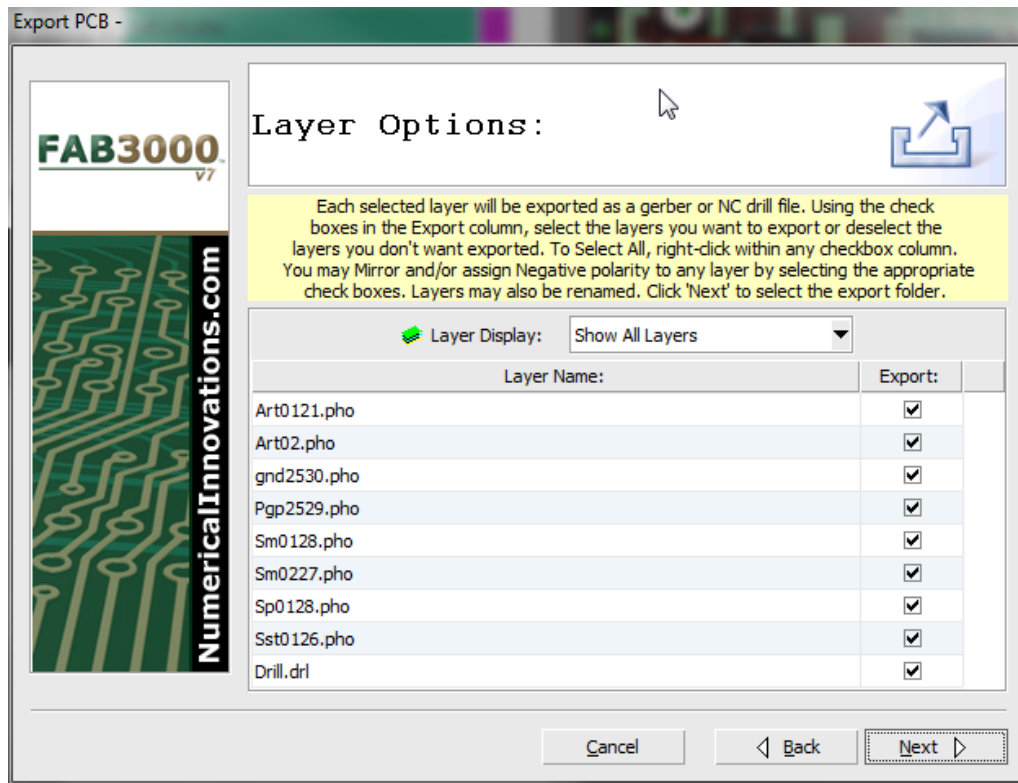


Using **Export > PCB** allows you to export your data in both Gerber and Drill formats simultaneously.

1. Select **File > Export > PCB**
2. Verify Gerber & Drill options using the tabs within the dialog box wizard.
3. Choose which reports will be included in the export using the Reports tab.

To improve accuracy, enter a value of 4 or higher under Decimal Digits.

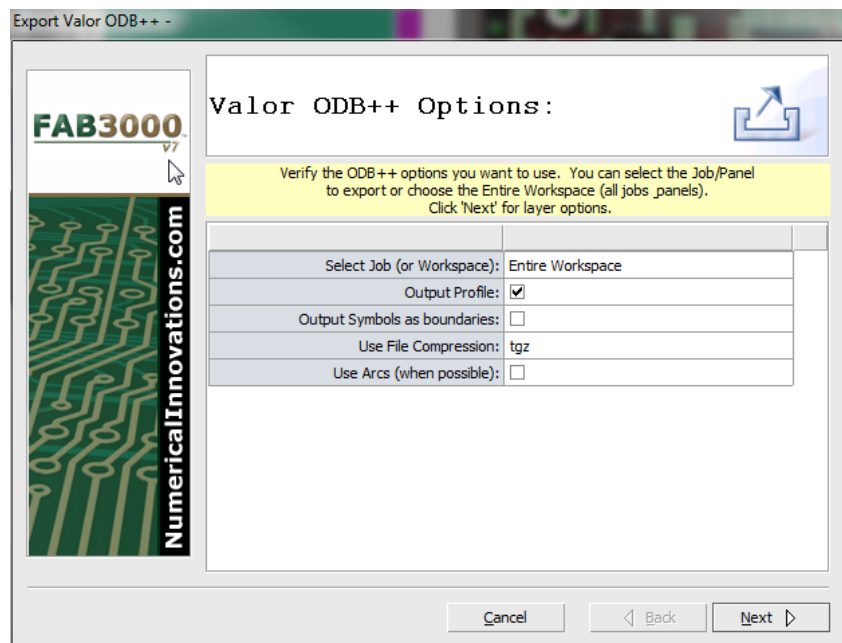
4. Select **Next**. You will then proceed to the Layer Options tab.



5. Assign Layer options. Use the check box to select which layer will be exported as a Gerber or NC Drill file. To select or deselect all checkboxes, right click within the Export column.

6. Select Next. A Browse Folder dialog box will appear. Choose a folder to export to.
7. Select OK.
8. Once export is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

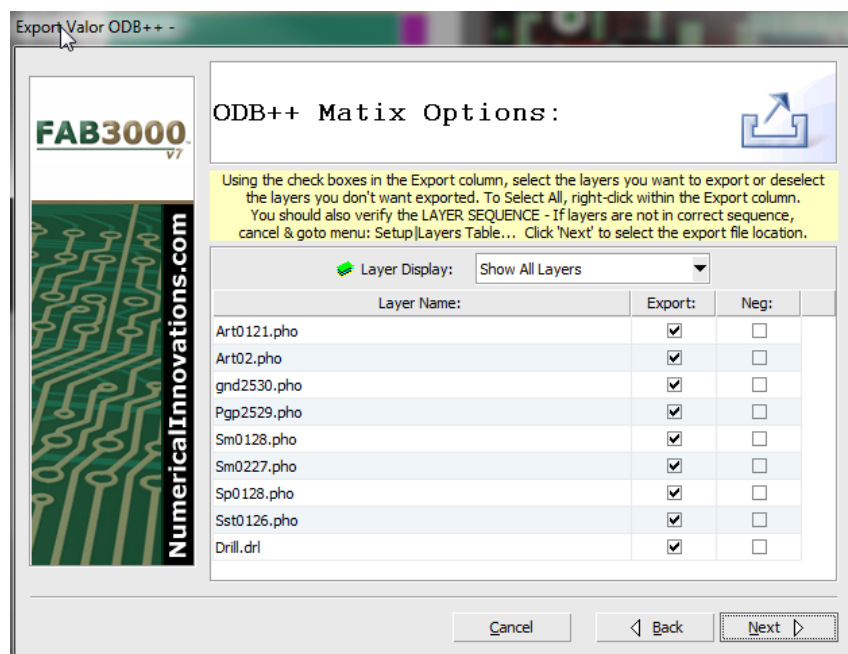
File > Export > ODB++



1. Select **File > Export > ODB++**. This writes an ODB++ database to an external file system.

The ODB++ Matrix requires layers to have their types (i.e. Top, Bottom, Silkscreen, etc.) and sequence order (i.e. layer stackup) defined before exporting. To define both Layer Types & Stackup go to the **Layers Table (File > Setup > Layers Table)** before exporting ODB++ data

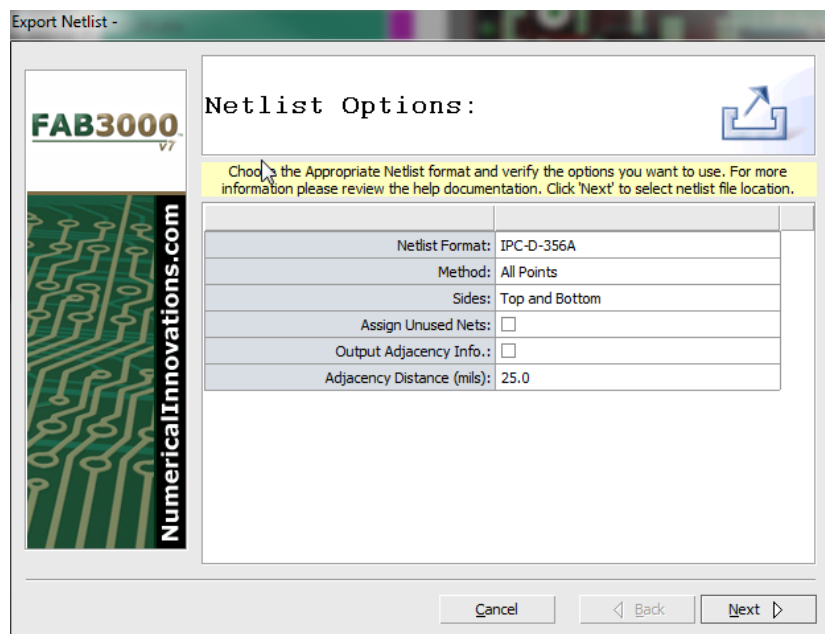
2. Select **Next**.



3. Assign Layer options. Use the check box to select which layers will be exported. To select or deselect all checkboxes, right click within the Export column.
4. Select **Next**. A Browse Folder dialog box will appear. Choose a folder to export to.
5. Select **OK**.

6. Once export is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

File > Export > Netlist (IPC-D-35G/A)



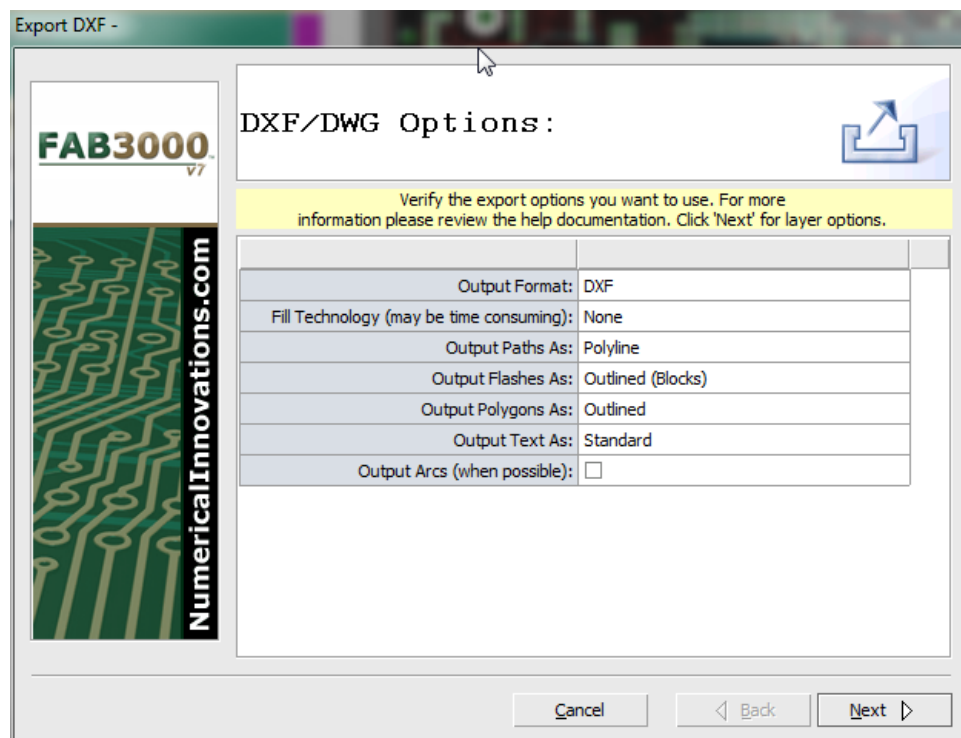
To export a Netlist, perform the following:

1. **File > Export > Netlist (IPC-D-35G/A).** The Export Netlist Wizard dialog box appears.
2. Verify the Export Netlist Options you want to use. For a complete list of options, see Export Netlist Options below.
3. Choose a Netlist file to create. If you would like to always export your files to a specific folder, you may define the **Export Folder** in the FAB 3000 Settings Dialog.
4. Select **OK** to begin exporting your Netlist file.
5. Once export Netlist is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.
6. Click **Finish**.

Export Netlist Options

Netlist Format:	Choose export Netlist format: IPC-D-356, IPC-D-356A formats.
Method:	Choose point method to use: All Points or End Points Only
Sides:	Determine Side to use: Top Only, Bottom Only, Top and Bottom
Assign Unused Nets:	Include Unused nets
Output Adjacency Info:	Include Adjacency Info.
Adjacency Distance:	Choose Adjacency distance.

File > Export > DXF/DWG

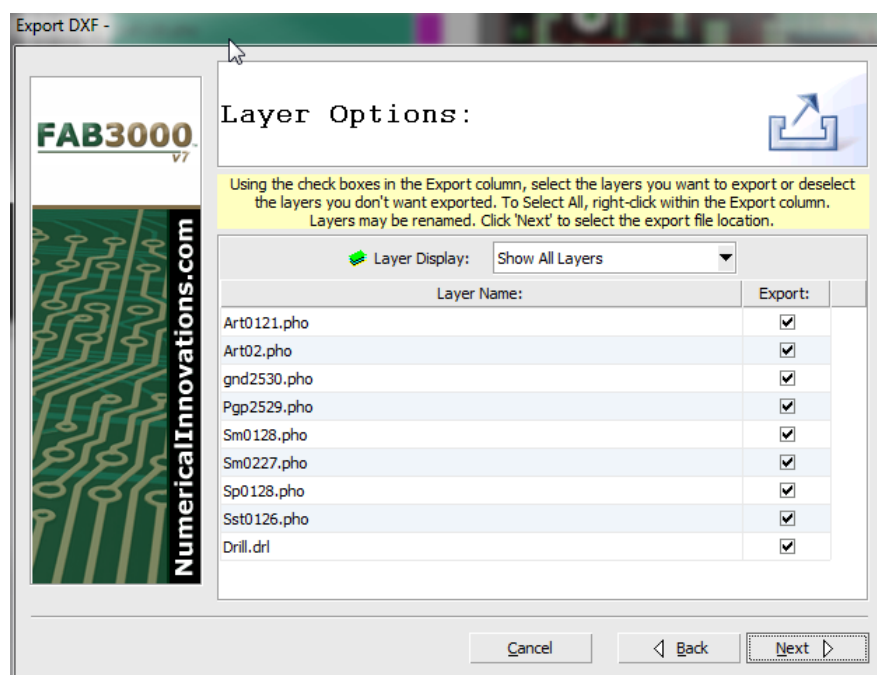


This command is used to export data into a DXF or AutoCAD DWG file. DWG files are the binary equivalent of DXF.

1. Select **File > Export > DXF**. The DXF Wizard dialog box will appear.
2. Verify the Export DXF file options you want to use. For a description of DXF Layer options see next page: Export DXF Options.

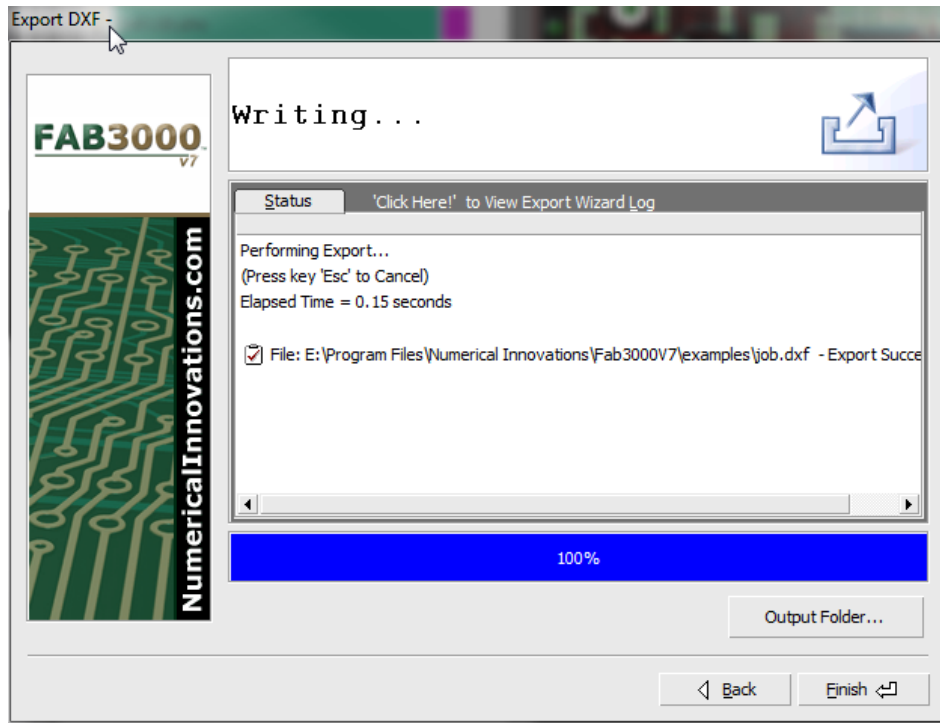
The DXF format has undergone many version changes in the past 5-10 years, and some older systems may not be able to read newer DXF files. If this is the case, try selecting an older DXF format under Output Format.

3. Select **Next**.



4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.
5. Select **Next**. A file dialog box will appear.
6. Choose the DXF file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the **Export Folder** in the FAB 3000 settings dialog under File > Setup > General.

7. Click **Save**.
8. Once export DXF is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.



9. Select **Finish**.

Export DXF Options:

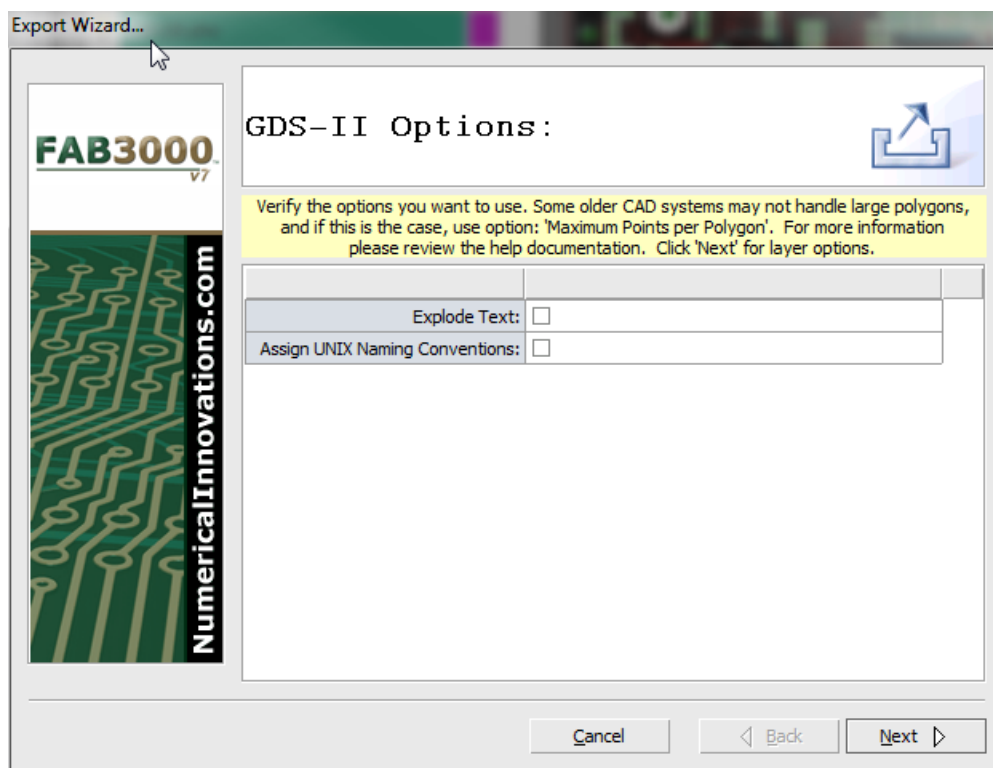
Text Output:	Specify how to output DXF text (if any): Standard: Text will be outputted as DXF text. Ignore: Text will be omitted from the DXF file. Explode: Text will be exploded, and output as polygons.
Explode Array Instances:	Convert Array instances to individual instances.
Output Paths as Boundaries:	Create polygon outlines for all paths.
Fill Option:	For documentation purposes it can be desired to have the DXF file appear exactly the same as a Gerber file (i.e. filled polygons, composites). None: Export with Geometries, No Filling (Default) Using Solid Hatch: Export with Geometries, Fill using Solid Hatch

Fill Options (cont.)	<p>Using Solid Hatch (Only): Export No Geometries, Fill using Solid Hatch</p> <p>Using Multiple Solids: Export Geometries, Fill using solid triangles</p> <p>Using Multiple Solids (Only): Export No Geometries, Fill using solid triangles</p>
-----------------------------	---



Forum: [Export a DXF File from FAB 3000](#)

File > Export > GDS-II

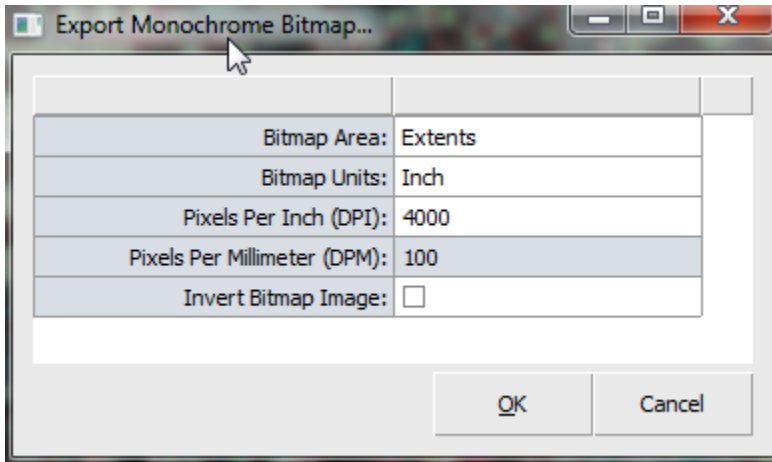


This command writes a GDS-II stream file to an external file system.

1. Select **File > Export > GDS-II**.
2. Verify and Edit GDS-II Options for **Explode Text** and **Assign UNIX Naming Conventions**.
3. Select **Next**.
4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.
5. Select **Next**. A file dialog box will appear.
6. Choose the GDS-II file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the **Export Folder** in the FAB 3000 settings dialog under **File > Setup > General**.
7. Click **Save**.
8. Once export GDS-II is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

9. Select **Finish**.

File > Export > Large Monochrome Bitmap

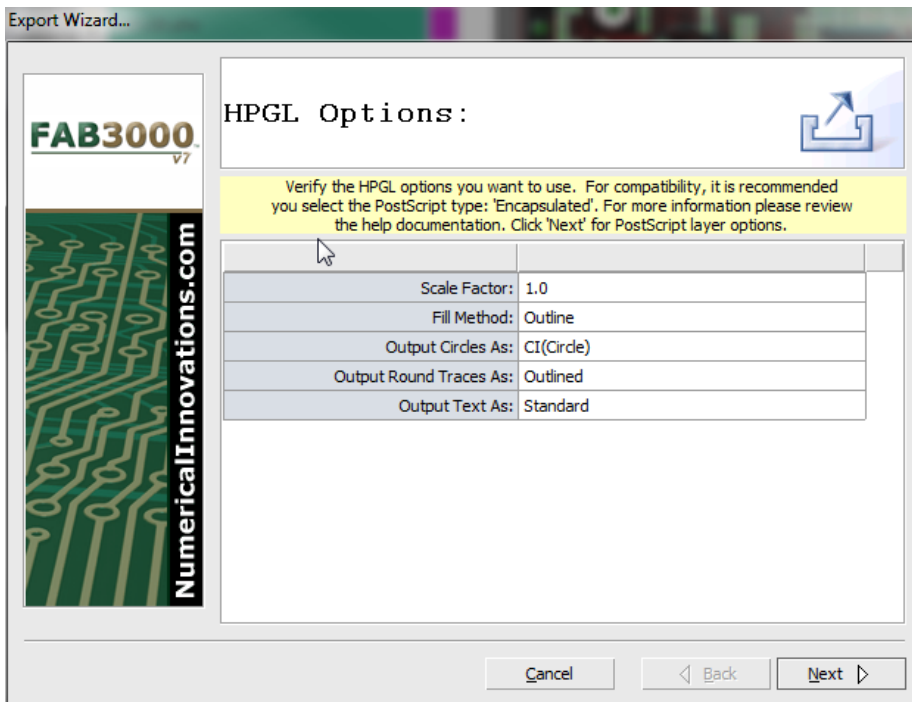


This command writes a large monochrome bitmap (any size) file to an external file system. To export a large bitmap, the active Job or panel needs to be displayed in the "editor", because FAB 3000 requires the editor to render the images that will be exported.

1. Select **File > Export > Large Monochrome Bitmap**.
2. Verify and Edit export options.

3. Select **OK**. An export folder dialog box will appear.
4. Choose the export file name for the bitmap.
5. Select **Save**.

File > Export > HPGL/2



This command exports an HPGL/2 file.

1. Select **File > Export > HPGL/2** an Export Wizard dialog box will appear.
2. Verify and edit HPGL/2 options. For a complete list of options, see **HPGL Options** on next page.
3. Select **Next**. A layer options dialog box will appear.
4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.

5. Select **Next**. A file dialog box will appear.

6. Choose the **HPGL/2** file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the **Export Folder** in the FAB 3000 settings dialog under **File > Setup > General**.
7. Click **Save**.
8. Once export HPGL/2 is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.
9. Select **Finish**.

File > Export > Centroid

Export Part Centroid -

FAB3000 V7

NumericalInnovations.com

Centroid Options:

Verify the Centroid export options you want to use. For more information please review the help documentation. Click 'Next' to select the export file location.

Units:	Inch
Component Types:	Surface Mount Only
Board Side:	Top and Bottom
Component Rotation (from Topside):	Counter-Clockwise
Angle Origin:	3 O-Clock
Include Component Part Number:	<input type="checkbox"/>
Include Component Pitch:	<input type="checkbox"/>
Include Component Height:	<input type="checkbox"/>

Cancel < Back Next >

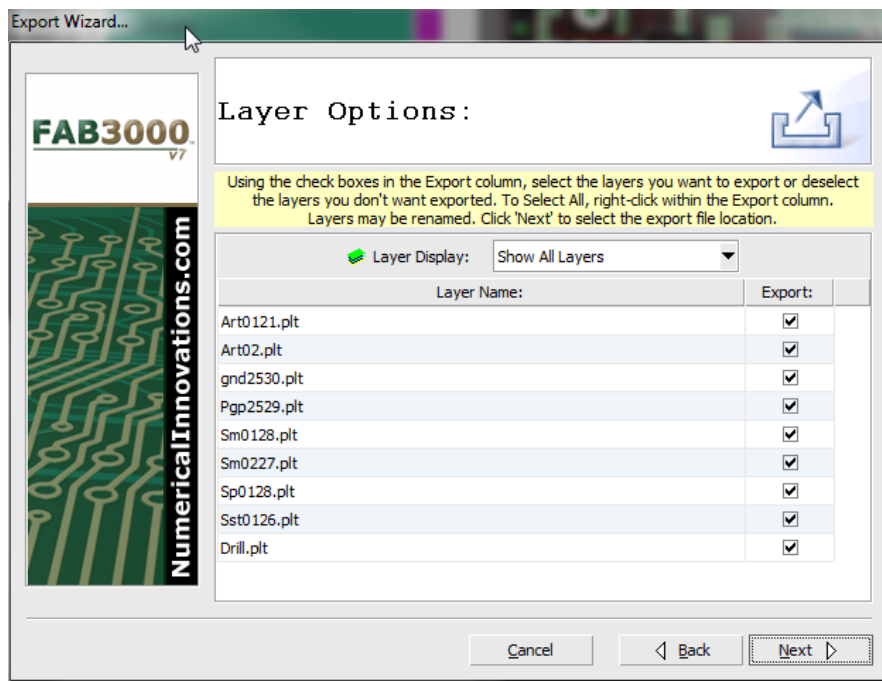
This command is used to export data into a Centroid file (aka Insertion, Pick-and-place or XY file).

You must at least have one or more Components defined before exporting to a centroid file.

The centroid file is a comma separated file that can be loaded into almost any spreadsheet program or text editor.

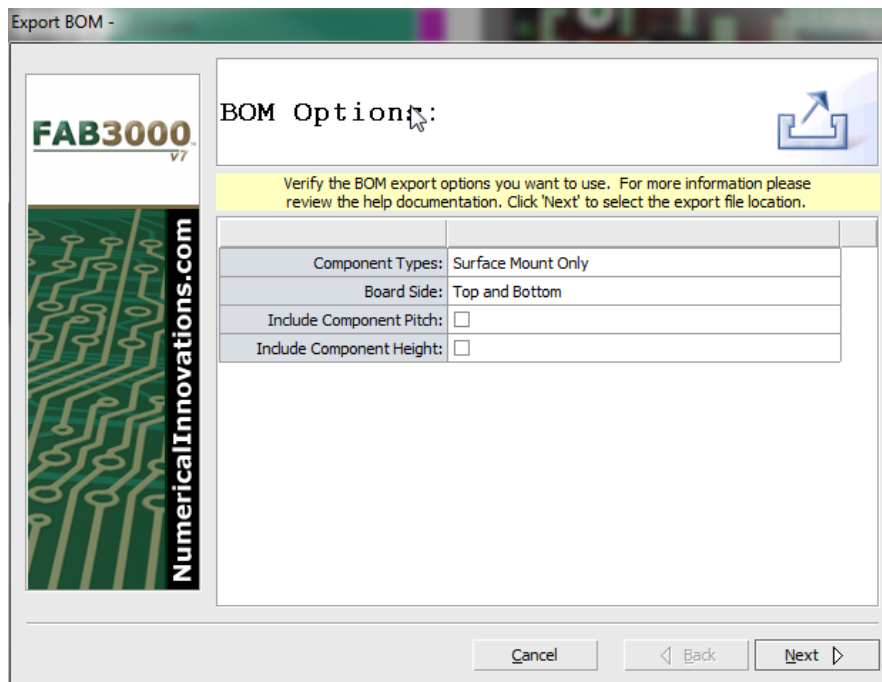
To export a Centroid file, perform the following:

1. Select **File > Export > Centroid File**.
2. Verify and edit Centroid options. For a complete list of options, see **Centroid Options** on next page.
3. Select **Next**. A layer options dialog box will appear.



4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.
5. Select **Next**. A file dialog box will appear.
6. Choose the Centroid file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.

File > Export > BOM File



The Export > BOM File command exports a Bill of Materials file.

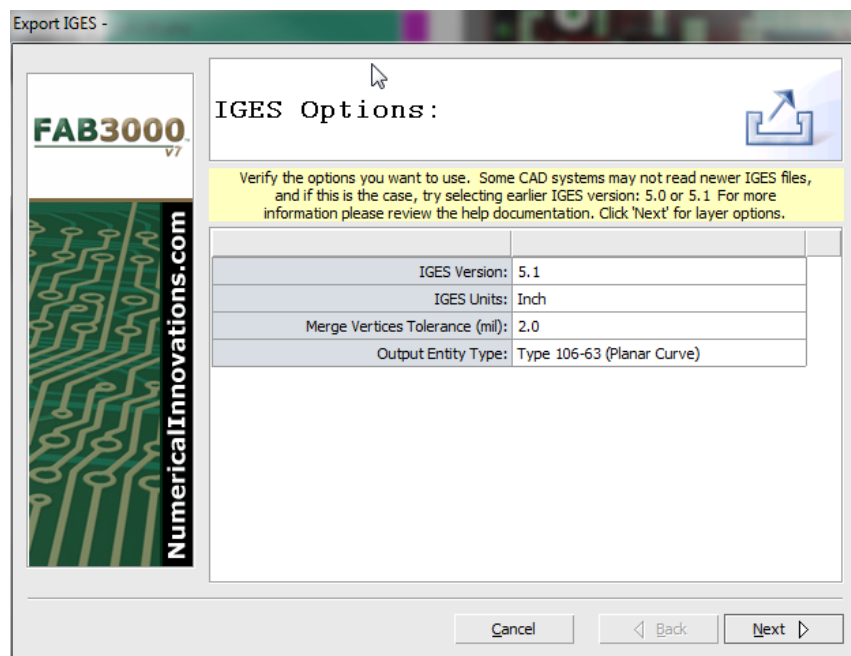
To export a BOM file, perform the following:

1. Select **File > Export > BOM File**.
2. Verify and edit BOM options. For a complete list of options, see **BOM Options** on next page.
3. Select **Next**. A file dialog box will appear.
6. Choose the BOM file to create from the file dialog box. If you

would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.

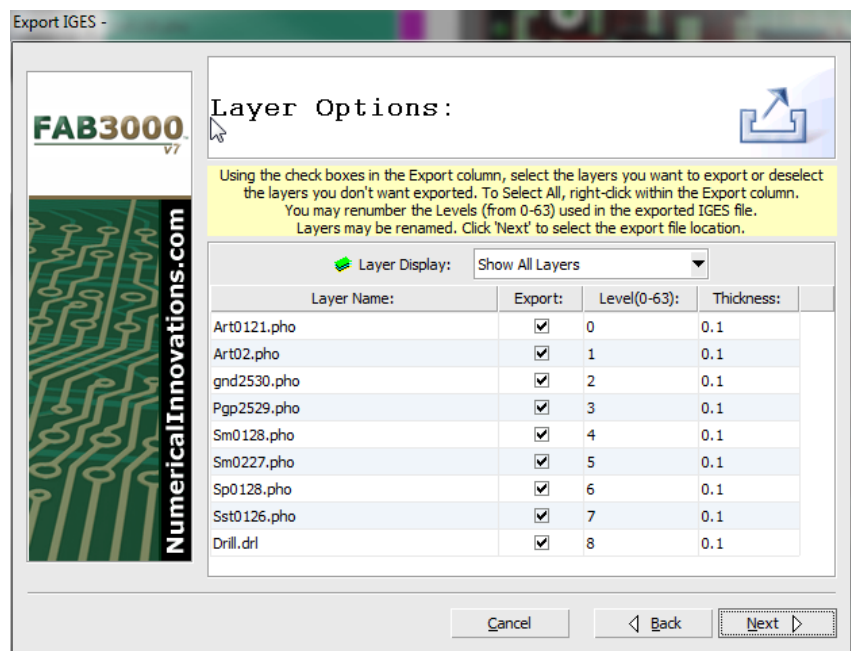
File > Export IGES

The IGES format has undergone many version changes, and some older systems may not be able to read newer IGES files. If this is the case, try selecting the IGES version compatible with your system.



To export an IGES file, perform the following:

1. Select **File > Export > IGES**
2. Verify and Edit IGES Options.
3. Select **Next**.
4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.



5. Select **Next**. A file dialog box will appear.
6. Choose the IGES file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.
7. Click **Save**.

8. Once export **IGES** is complete, a Status & Import log are available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

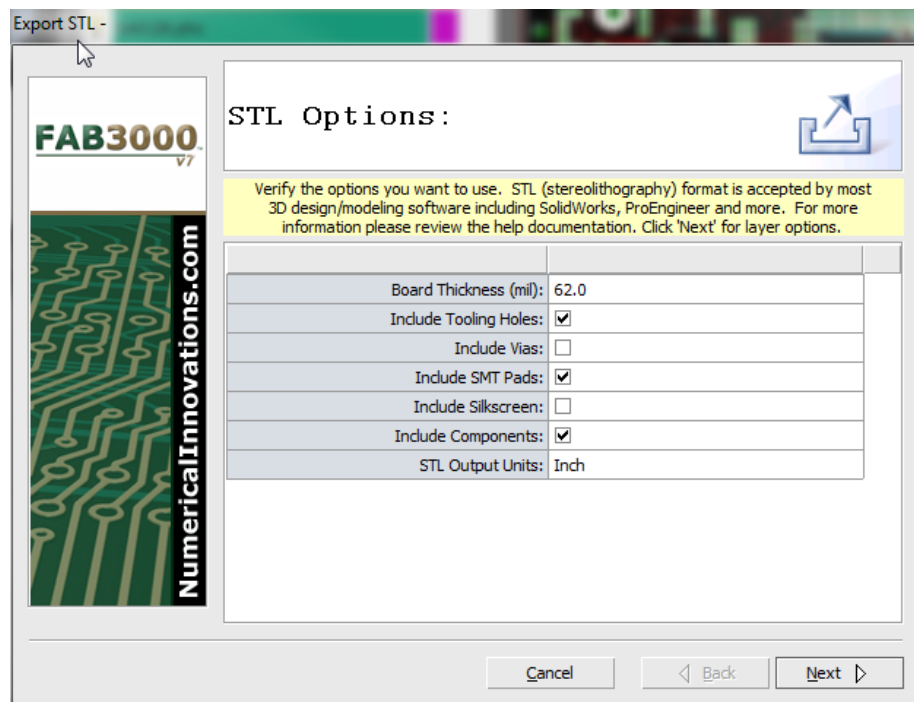
9. Select **Finish**.

Export IGES Options

IGES Units:	Select the units to use for the exported IGES file. Choose from Inch, Millimeter, Micron, etc.
Text Output:	<p>Specify how to output IGES text (if any):</p> <p>Ignore: Text will be omitted from the IGES file.</p> <p>Note (Type 212): Text will be output using IGES type 212.</p> <p>Outline (Type 106-63): Text will be exploded, and output using IGES Type 106-63.</p>
Path Output:	<p>Determine how to output paths with non-zero widths.</p> <p>Line Weight (106-11): Paths will be output using a centerline and single width.</p> <p>Outline (Type 106-63): Paths will converted to polygons using the IGES Type 106-63.</p>

File > Export > STL (3D)

To export an IGES file, perform the following:

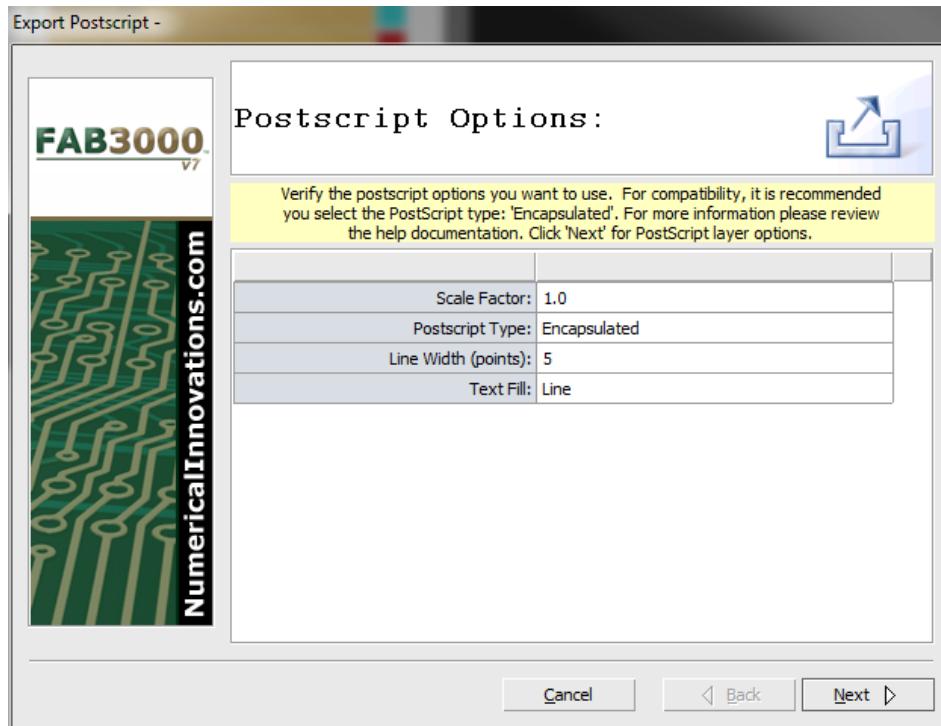


1. Select **File > Export > STL (3D)**
2. Verify and Edit STL options. For a complete list of STL options, see STL Options below.
3. Select **Next**. A file dialog box will appear.
4. Choose the STL file to save from the file dialog box. If you would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.

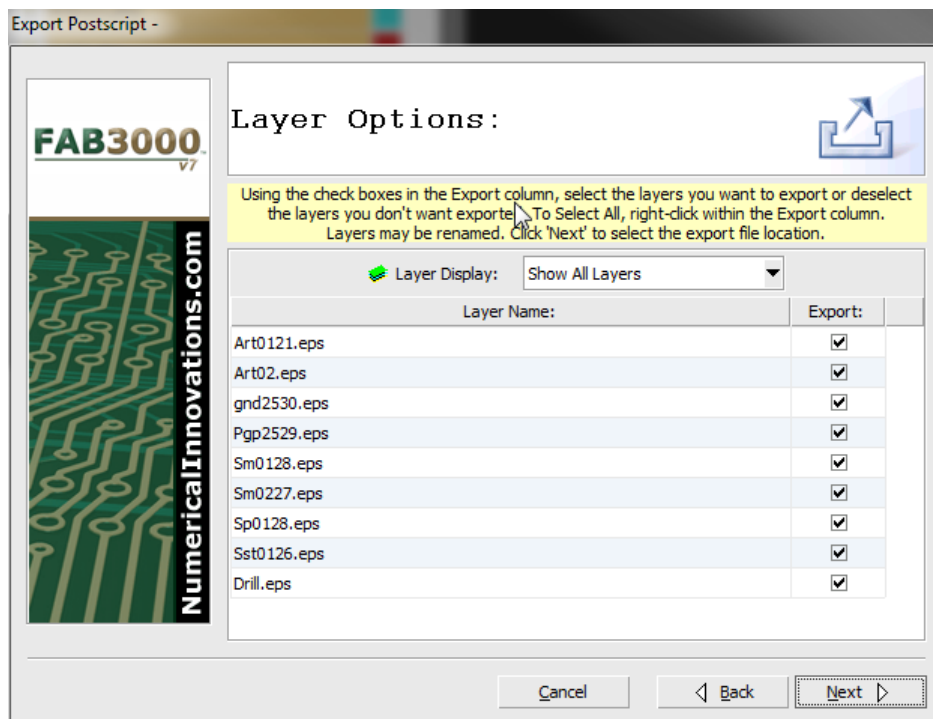
5. Select **Save**.
6. Once export STL is complete, an Export Wizard log is available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

File > Export > Postscript

This command is used to export layers into PostScript files. To export a Postscript file, perform the following:



1. Select **File > Export >**
2. Verify and Edit Postscript Options. For a complete list of options, see **Export Postscript Options** below.
3. Select **Next**.
4. The layer options dialog box will appear. If a layer needs to be changed, type in the desired filename within the text entry boxes.



5. Select **Next**. A file dialog box will appear.
6. Choose the Postscript file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.
7. Click **Save**.

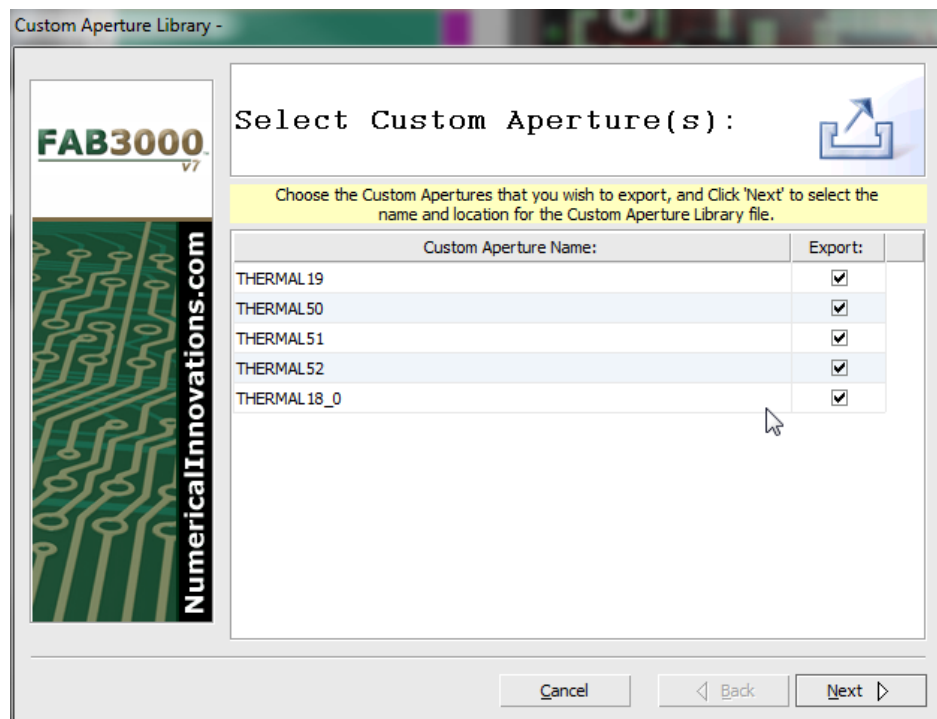
8. Once export Postscript is complete, an Export Wizard log is available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

Export Postscript Options

Scale Factor:	Determine scale to use for Postscript output. (In most cases for MEMs/EDA masks a factor of 10X is used.)
Postscript Type:	Select Postscript output type either Standard or Encapsulated. If you are unsure, contact your local graphic expert - in most cases we recommend "Encapsulated".
Line Width (Points):	Default line width to assign for 0-width objects such as lines, arcs, etc.
Text Fill:	Specify how to output text (if any): Line: Text will be exploded and outputted as Gerber traces using the Line Width. Solid Fill: Text will be exploded, and output as polygons.

File > Export > Custom Aperture Library

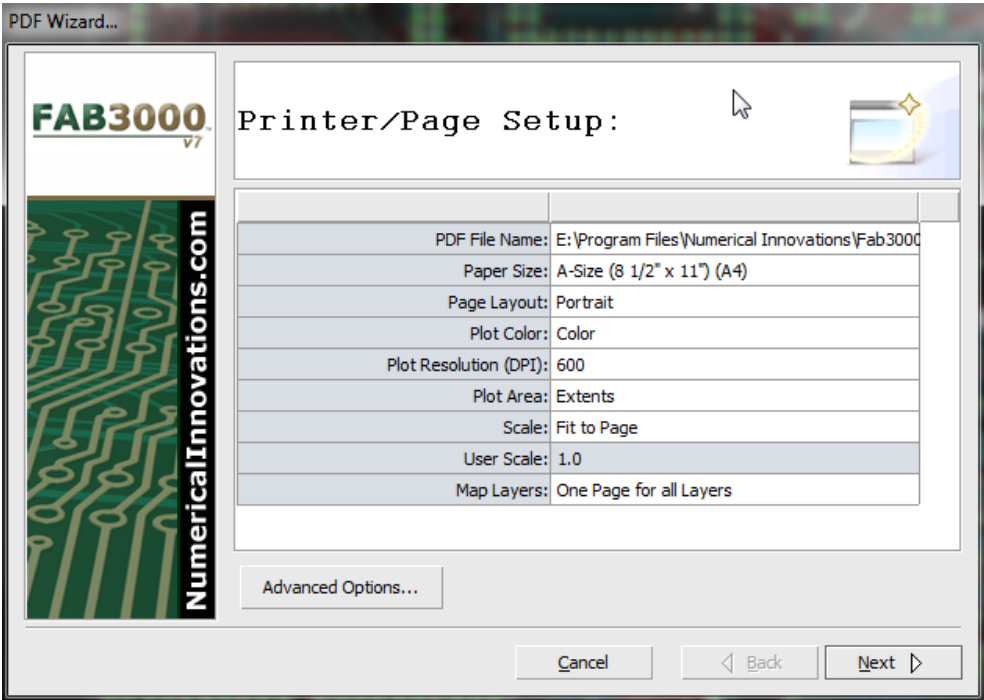
This command writes a custom aperture library to an external file system. To export a custom aperture library, perform the following:



1. Select **File > Export > Custom Aperture Library**.
2. Verify and Edit Postscript Options. For a complete list of options, see **Export Aperture Options** on the next page.
3. Select **Next**. A file dialog box will appear.

- 6. Choose the Custom Aperture Library file to create from the file dialog box. If you would like to always export your files to a specific folder, you may define the Export Folder in the FAB 3000 settings dialog under **File > Setup > General**.
- 7. Click **Save**.
- 8. Once export Custom Aperture export is complete, an Export Wizard log is available for your review. If FAB 3000 encounters any errors or warnings during export, they will be displayed here.

File > Print to PDF (Up to E Size)



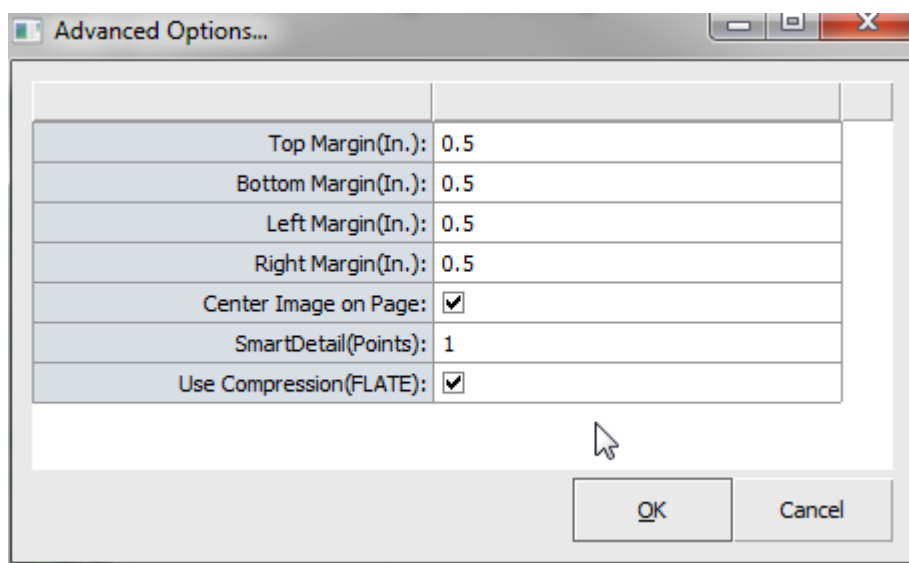
Create professional EDA plots in PDF format.

- 1. Select **File > Print to PDF**.
- 2. Verify and edit the PDF output options. For a definition of each option see **Printer/Page Setup Options** below.

Printer/Page Setup Options

Paper Size	Location of outputted PDF file.
Paper Size	Size of outputted document.
	A-Size (8 1/2" x 11") (A4)
	B-Size (11" x 17") (A3)
	C-Size (17" x 22") (A2)
	D-Size (22" x 34") (A1)
	E-Size (34" x 44") (A0)

Page Layout	Choose either portrait or landscape.
Plot Color	Choose either black and white or color. Available sizes are 300, 600, 900, 1200, or 2400 DPI.
Plot Area	Choose either extents or display.
Scale	Choose either Fit to Page or User Defined Scale.
User Scale	If User Defined Scale is selected, this box will allow you to input a custom scale.
Map Layers	Select one page for all layers, a separate PDF for each layer, or a separate page for each layer.



3. In the same wizard dialog box, you may also choose advanced options by selecting the **Advanced Options** button.

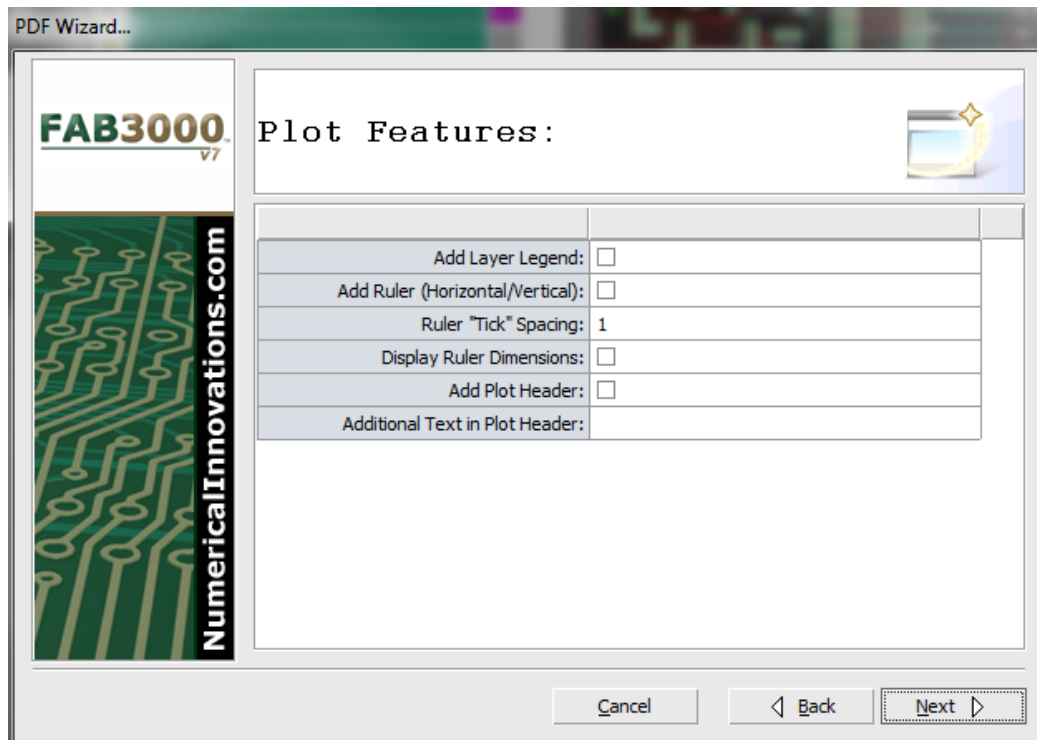
4. For a complete definition of Advanced Options, see below.

Advanced Options

Advanced options are additional options to determine proper dimensions and positioning of your plot. Double click any of these boxes to edit.

- a. Top Margin (in): Spacing on the top of the page.
- b. Bottom Margin (in): Spacing on the bottom of the page.
- c. Left Margin (in): Spacing on the left of the page.
- d. Right Margin (in): Spacing on the right of the page.
- e. Center Image on Page: If unchecked, image will be placed using Left, and Bottom margins. If checked, the image is centered.

- f. **Smart Detail (Points):** This powerful feature optimizes the plotter output, by omitting any objects whose overall size is smaller than this value. In most cases, small details are not visible when they are printed onto paper; so it serves no purpose to include its' data in the plot. The result is a much smaller, easier to manage plot output. The units for Smart Detail are in points (1/72 Inch).
 - g. **Use Compression (FLATE):** Most PDF files that are created use compression to keep the file size small. However compressing a large or detailed PDF file can be very memory intensive, so in some cases it can be easier to work with an uncompressed PDF file. Although the uncompressed pdf file is extremely large (800mb for E-Size plots), Adobe's PDF Viewer actually reads these files faster because it is not required to also decompress these file. These large uncompressed PDF files can also be externally compressed with zip, tar.gz, etc. for backing up, sharing via network or internet.
5. Select **Next**.
 6. The Plot Features dialog box will appear. Edit and verify plot features.



Add Layer Legend: Include a Layer Legend on the plot. A layer legend includes layer number, layer color & fill for each layer used on the plot.

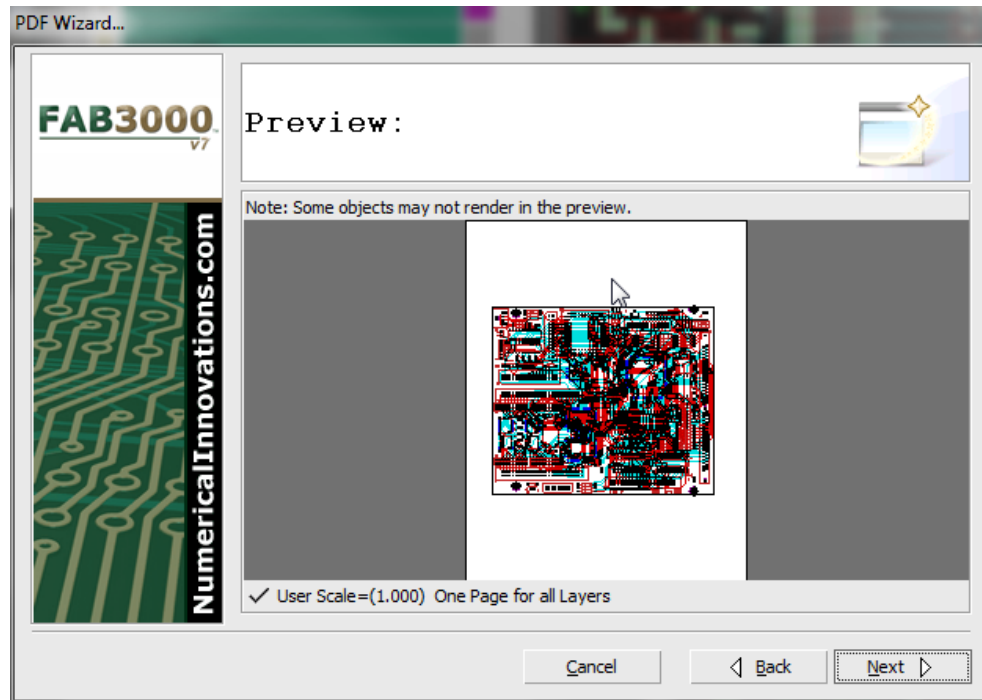
Add Ruler (Horizontal/Vertical): Include a horizontal & vertical ruler on the plot, located on the Left & Bottom sides. The ruler consists of a horizontal or vertical reference line with numbered ticks.

Ruler "Tick" Spacing: Spacing used between ruler Ticks.

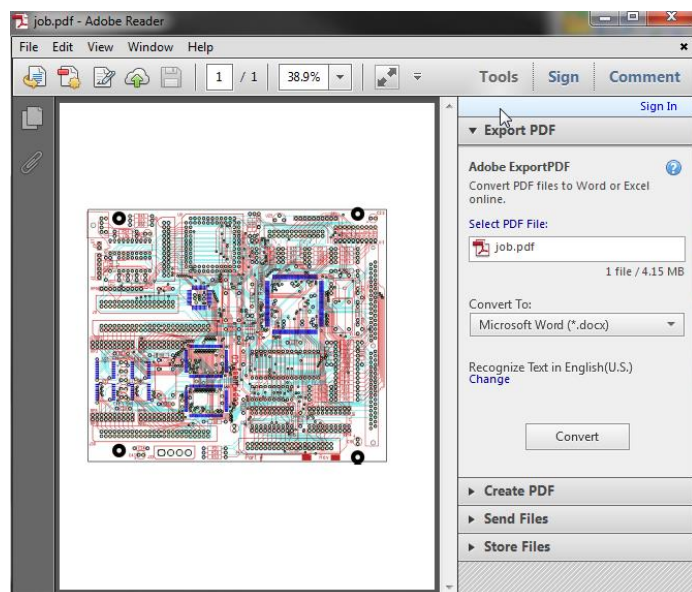
Display Ruler Dimensions: Determine whether to number each tick on the ruler.

Add Plot Header: Include standard information regarding the plot (Library/design, Scale, Plot Time, etc.)

Additional Text in Plot Header: Include specific or custom plot information.



7. Select **Next**. A preview window will appear. Review the PDF output. If there are any potential problems, and warning will be placed under the preview image. If everything looks fine, selecting next will begin the creation of the PDF file.



8. Select **Finish**. FAB 3000 will begin the PDF conversion.
9. To view the PDF output, select **View PDF**. The PDF file will open in Adobe Reader.

When selecting View PDF, the dialog box will automatically close.

PDF creation is very memory intensive (due to image compression). Recommend at least 1-2 GB memory for creating large 'E' Size Plots.

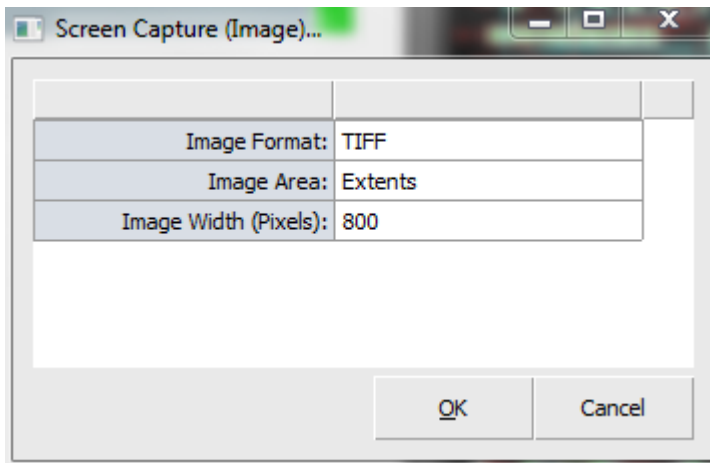


[Forum: Plot to PDF \(up to E size\)](#)

[Forum: Output a PDF file with each layer on a separate page](#)

File > Plot to Image

Plot to Image writes an image file (.TIFF, .PNG, .BMP, .GIF, etc.) to an external file system. To export an image, the active Job or panel needs to be displayed in the editor. FAB 3000 requires the editor to render the images that will be exported. The maximum image width is 3000 pixels. To export a larger bitmap file, use **File > Export > Large Monochrome Bitmap**.



To export an image, perform the following:

1. Activate the editor and display the Job you want to export to an image.
2. Select **File > Export > Plot to Image**.
 - a. Choose the Image Format in the pull down menu.
 - b. Choose the Image Area.
 - c. Choose the Image Width (Pixels).

3. Select **OK**. The file browser dialog box will appear.
4. Choose the file name and & path for the image.

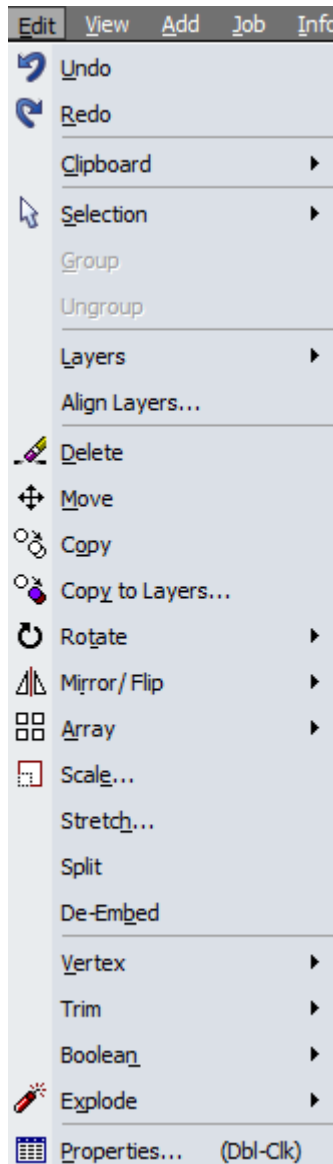
File > Recent Workspaces

When selecting Recent Workspaces, this will provide you with a list of the most recently accessed workspaces.

File > Exit

Closes FAB 3000. If any data is unsaved, a dialog box will prompt you to save your files before closing.

5.2 EDIT MENU



The **Edit Menu** is responsible for any changes, modifications, deletions or copies of your design.

Edit > Undo

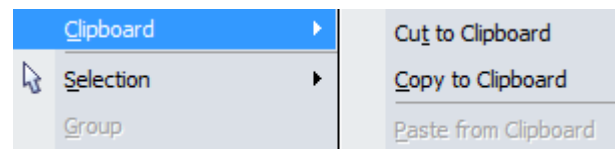
Undo the last command in the editor.

Edit > Redo

Redo the last command in the editor.

Edit > Clipboard

Allows you to Cut, Copy, and Paste objects from the clipboard. This clipboard is different from an OS clipboard (i.e. Windows clipboard), because it allows you to cut, copy, and paste objects between different Jobs in FAB 3000. This feature is very useful for merging parts of different Jobs together to build new Jobs.



Edit > Selection

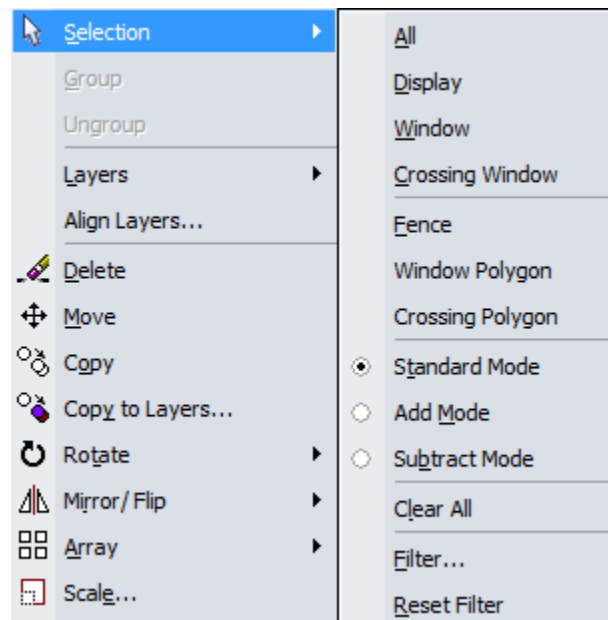
Performs a number of selection actions within the FAB 3000 editor.

Edit > Selection > All

Selects all objects in the editor.

Edit > Selection > Display

Selects objects that are currently displayed. This command will automatically place the editor in Select Mode.



Edit > Selection > Window

Select objects inside a user defined window.

To select via window:

- a. Select **Edit > Selection > Window**.

- b. Pick a first point. Press left mouse button down and hold.
- c. Drag the mouse to the opposite corner of the selection window. Release the left mouse button.

The selection window requires that you pick the first point and drag across the screen, while holding the left-button down. When you release the left-button, all objects that completely fit inside the defined window are selected. This command will automatically place the CAM editor in Select Mode.

Edit > Selection > Crossing Window

Select objects inside and intersecting a user defined window.

- a. Select **Edit > Selection > Crossing Window**.
- b. Pick a first point. Press left mouse button down and hold.
- c. Drag the mouse to the opposite corner of the selection window. Release the left mouse button.

The selection window requires that you pick the first point and drag across the screen, while holding the left-button down. When you release the left-button, all objects that completely fit inside the defined window are selected. This command will automatically place the CAM editor in Select Mode.

Edit > Selection > Fence

Select objects using a polyline.

- 1. Select **Edit > Selection > Fence**.
- 2. Pick a first point, click and hold the left mouse button. A polyline will appear.
- 3. Draw a polyline or series of polylines over the objects you wish to select.
- 4. When finished, right click and select **End** or **Close**. The objects within this fenced area will be selected.

Edit > Selection > Window Polygon

Select objects inside a user defined polygon.

- 1. Select **Edit > Selection > Window Polygon**.
- 2. Pick a first point, click the left mouse button. Guide the first line in the direction of your selection.
- 3. Pick a second point, click the left mouse button. A polygon will appear surrounding your selection.
- 4. Repeat this process and guide the polygon shape around the object area you wish to select.
- 4. When finished, right click to **Close**. The objects within this polygon window area will be selected.

Edit > Selection > Crossing Polygon

1. Select **Edit > Selection > Crossing Polygon**.
2. Pick a first point, click the left mouse button. Guide the first line in the direction of your selection.
3. Pick a second point, click the left mouse button. A polygon will appear surrounding your selection.
4. Repeat this process and guide the polygon shape around the object area you wish to select.
4. When finished, right click to **Close**. The objects within and intersecting this window area will be selected.

Edit Modes

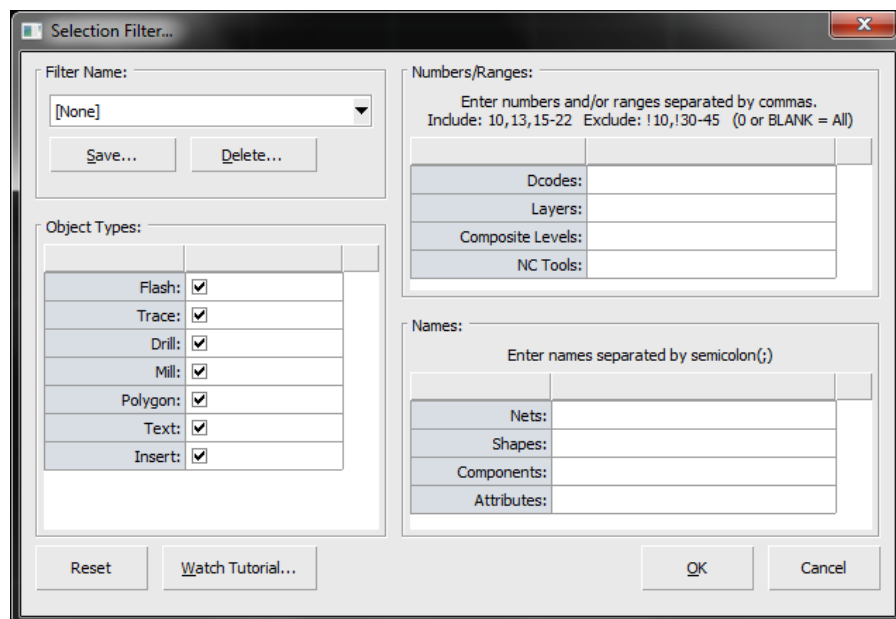
There are three Edit Modes under **Edit > Selection**:

1. Standard Mode
2. Edit Mode
3. Subtract Mode

Edit > Selection > Clear All

Removes a selection.

Edit > Selection > Filter



The selection filter is an extremely useful tool used for controlling the content of Objects that are selected prior to performing any modifications or queries. It can be a major time saver as it parses out Objects that do not pass certain requirement that you set.

1. Select **Edit > Selection > Filter**. A Selection Filter dialog box will appear.

2. Verify or Edit all Filter items. Select **OK** when complete.

Filter Name:

Save and reuse your selection filters. This feature drastically speeds up editing in between sessions.

Object Types:

For items with check boxes, a check means the item can be selected, removing a check means the item cannot be selected. By default all are checked.

Number/Ranges:

Items here allow you to include or exclude specific Dcodes, Layers, Composite Levels, or NC Tools from being selected. Enter numbers and/or ranges separated by commas as directed in the dialog box.

Names:

Items here allow you to include specific objects for selection based from matching names. Use a semicolon ';' to separate names for Nets, Shapes, Components, and Attributes.

Example A: Select only objects with net names "Pwr_2324" and "Pwr_0010", enter in the Nets field:

Pwr_2324; Pwr_0010

Example B: Select all objects with nets that begin with "Pwr_", just enter in the Nets field: ***Pwr_***

Example C: Select all Rectangle Shapes with a height less than 0.01" and all Circle shapes greater than or equal to 0.02" just enter in the Shapes field: ***rectangle,height<0.01;circle>=0.02***

Important: Each shape filter argument can accept the following: ***<shape>,width=x,height=x*** separated by a semicolon if there are multiple shape filter arguments. The units for X is the current units of the FAB 3000 editor.

Important: The Selection Filter is only good for one command (i.e. Window, Crossing, All, etc.). If you wish to select another Crossing Window, you will need to go back to step 1, and repeat.

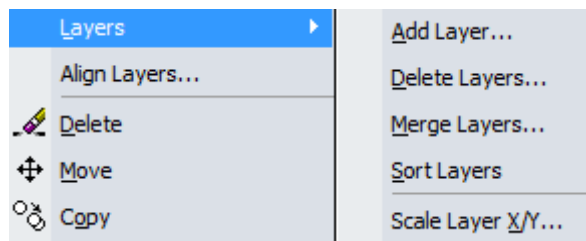
Edit > Selection > Reset Filter

Clears the selection filter.

Edit > Layers > Add Layer

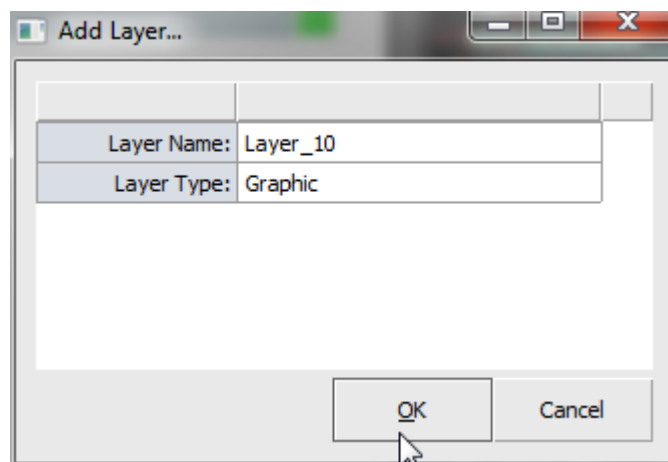
Adds a layer.

- a. Select **Edit > Layers > Add Layer**. A dialog box will appear to edit by **Layer Name** and **Layer Type**.

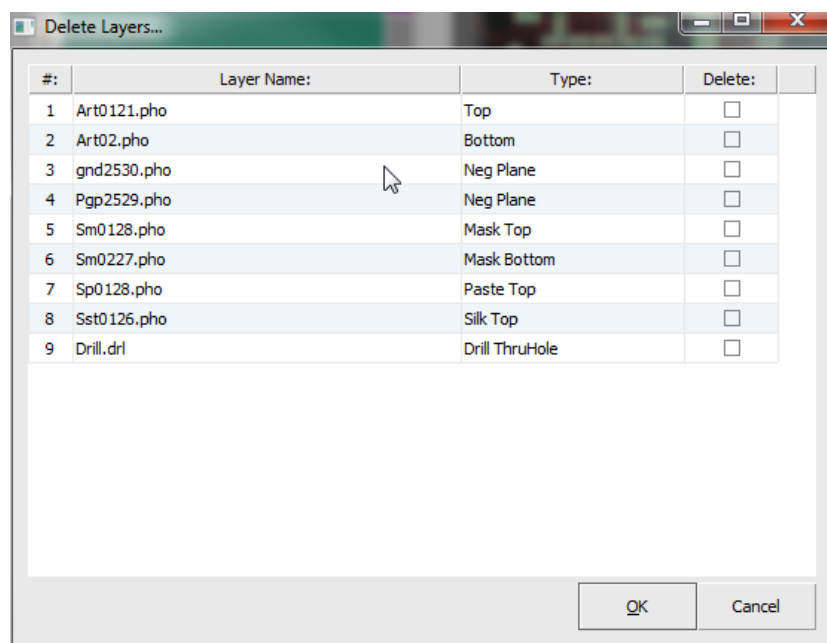


- b. Enter the **Layer Name** and layer type you would like to use. If the layer name already exists, a new layer will be created.
- c. Enter the **Layer Type** you would like to use. FAB 3000 will assign the next available layer type, and in most cases you do not need to modify this. If the layer type already exists, no new layer will be created.

This command cannot be undone.



Edit > Layers > Delete Layers



Deletes selected Layers.

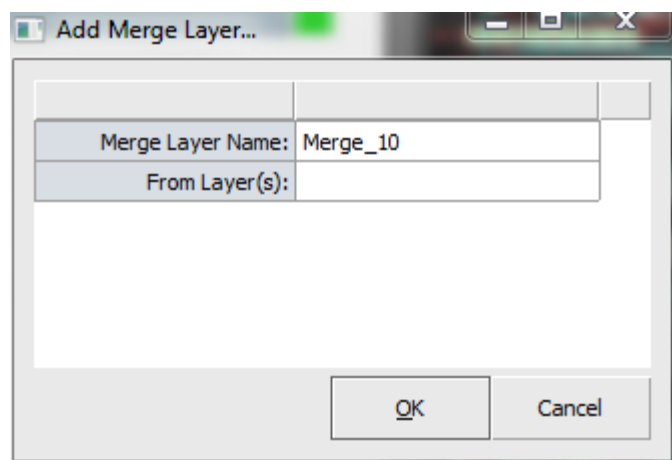
1. **Select Edit > Layers > Delete Layers.** The Delete Layers dialog box will appear.
2. Check the layers that you want to delete.
3. Select **OK** to delete the selected layers.

This command cannot be undone.

Edit > Layers > Merge Layers

Merge multiple layers simultaneously.

This merges layers without polarity (i.e. composites). If you want to merge layers to create composites (dark/clear) use **Tools > Composite Layer > Build**.

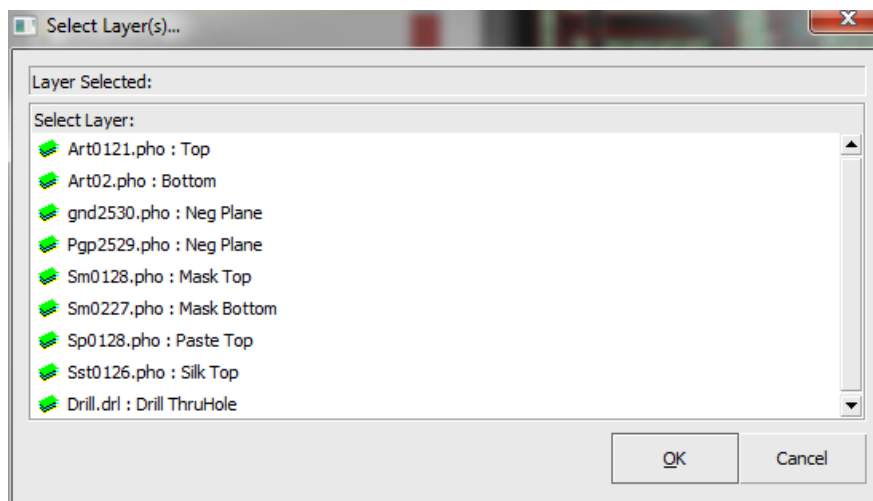


1. Select **Edit > Layers > Merge Layers**. The Merge Layer dialog box will appear.
2. Enter the name of the newly merged layer in the **Merge Layer Name** box.
3. Click the **From Layer(s)** entry box. A **Select Layers** dialog box will appear.

4. Highlight the layers you want to include for the merge.

If you want to add another layer to the merge, or you made a selection error, re-click in the entry box and reselect the layers to merge.

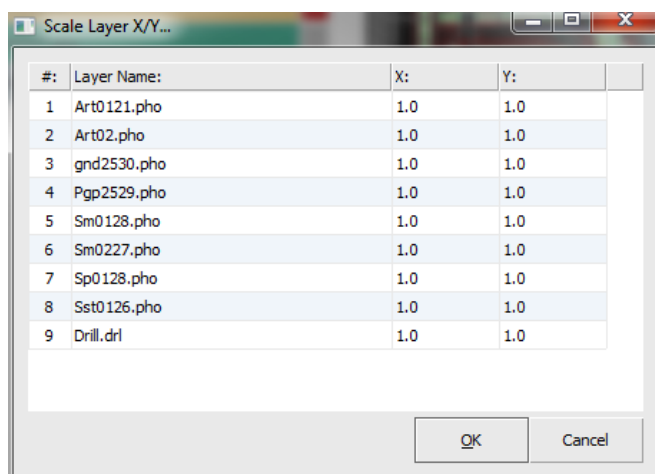
5. Select **OK** to begin the merge.



This command cannot be undone.

Edit > Layers > Sort Layer

Automatically sort Layers per the recommended PCB stackup.



Edit > Layers > Scale Layer X/Y

This command scales a layer in the X and Y directions.

1. Select **Edit > Layers > Scale Layer**.
2. The Scale Layer X/Y dialog box will appear.
3. Assign values for the X/Y directions for each layer.

Edit > Align Layers

Adjusts a layer origin to become aligned with another layer based off a similar object such as a target, pad, border, line, etc. In order to align layers correctly, you will need to locate a similar object that exist on both the base layer and the other layer, such as at Target, Fiducial Point, Pad, Border, etc.

To use Align Layers, perform the following:

1. Select **Edit > Layers > Align Layers**.
2. In the editor, select the base object.
 - a. This object must reside on the master (i.e. base) layer, and there must be a similar object on the next (i.e. to be aligned) layer.
3. In the editor, select the next object to align.
 - a. This object must be of the same object type (as the Base Object) and reside on the next (i.e. to be aligned layer) layer.
4. FAB 3000 will now instantly align the Next layer to be directly on top of the base layer, using those two objects as reference points.



[Video: FAB 3000 Align Gerber and Drill Layers](#)

Edit > Delete

Removes selected objects from the active design. This command requires that you select objects before proceeding. When no objects are selected, this command becomes de-activated in the menu.

1. In the editor, select objects for deletion. To select polygons or traces, pick the edges of an object.
2. Select **Edit > Delete**.
 - a. If an incorrect object is deleted, select **Undo**.

Edit > Move

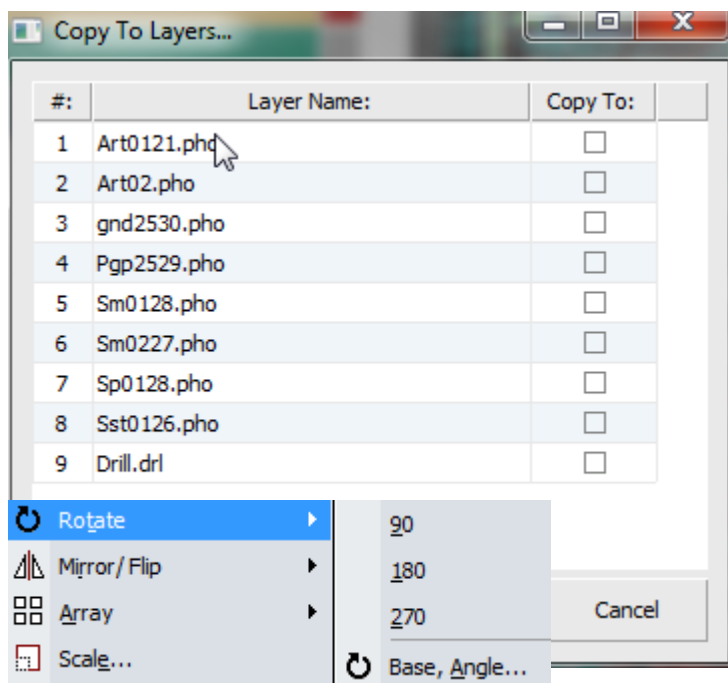
This command moves objects to a new location. This command requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

1. Select objects to move. To select polygons or traces, pick the edges of an object.
2. Select **Edit > Move**.
3. In the editor, select the base point.
4. Enter the second point of displacement. The objects will be moved by the displacement amount.
 - a. For accurate distances, use the command line to enter distance values.
 - b. If the move is incorrect, use the Undo command.
5. When finished, right click or select Esc to exit.
 - a. To quickly move a selection, Left-click over a selected object and drag the entire ` selection to any location in the editor.

Edit > Copy to Layers

Copies selected object to other layers. This command requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

1. Select objects to copy. To select polygons or traces, pick the edges of an object.
2. Select **Edit > Copy to Layers**. The Copy to Layer dialog box will appear.
3. Check the layers that you want to copy objects to.
4. Select **OK** to begin copy.



objects are selected this command in the menu.

- a. If the copy is incorrect, use the Undo command.

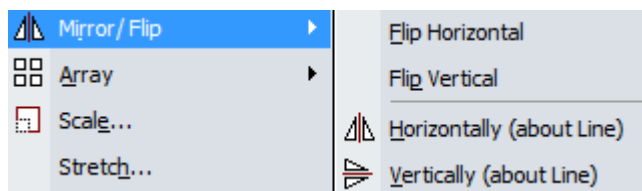
Edit > Rotate

Rotates selected objects by either 90, 180, or 270 degrees. This command requires that you select objects before proceeding. When no objects are selected this command becomes de-activated



Rotation Examples: 0, 90, 180, 270 degrees.

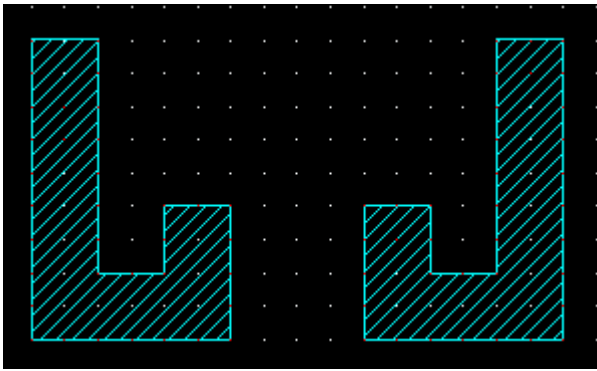
1. Select objects to rotate. To select polygons or traces, pick the edges of an object.
2. Select **Edit > Rotate > (*Use either 90, 180, 270, or Base Angle)**. Base angle refers to user defined value. Enter this value in the command line underneath the editor (Specify Base Point).
3. The select objects will now be rotated.
4. If the rotate is incorrect, use the **Undo** command.



Edit > Mirror / Flip

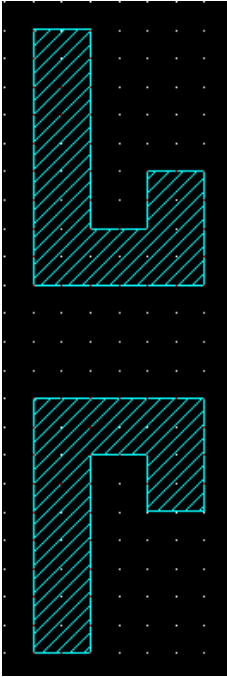
Mirror selected objects either horizontally or vertically. This command requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

There are four flip commands to choose from:



a. **Flip Horizontal:** Flips selection about the Y-axis.

Horizontally (about Line): Flips selection via reference line.



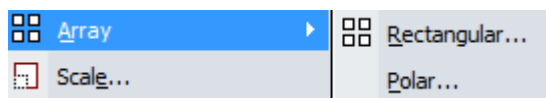
b. **Flip Vertical:** Flips selection about the X-axis.

Vertically (about Line): Flips selection via reference line.

To **Mirror/Flip** an object, perform the following:

1. Select objects to Mirror/Flip. To select polygons or traces, pick the edges of an object.
2. Select **Edit > Mirror > (*choose flip direction – Horizontal or Vertical)**.
3. Specify mirror line location for objects (if using Horizontally or Vertically). Using Flip Vertical or Flip Horizontal will simply flip the object.
4. Determine if you want to delete the source (original object selected).

- a. If the mirror is incorrect, use the Undo command.

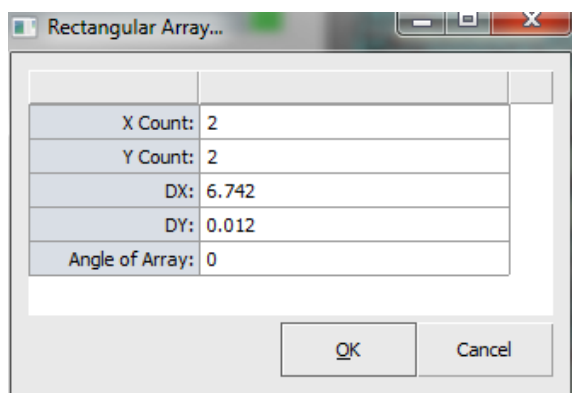


Edit > Array

Displays multiples of the same object in a group. This function requires that you select objects before proceeding. When no objects are selected, this command becomes de-activated in the menu.

There are two types of Arrays, **Rectangular** and **Polar**.

Rectangular Array



X Count: Number of copies in the horizontal direction.

Y Count: Number of copies in the vertical direction.

DX: The offset in the X direction

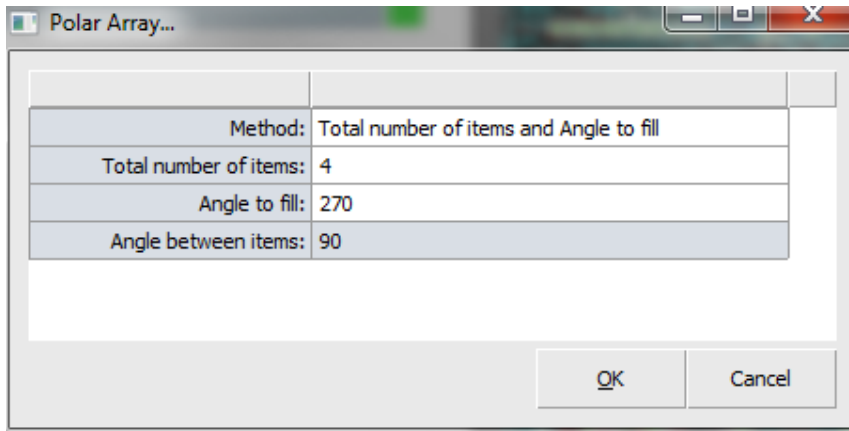
DY: The offset in the Y direction.

Angle of Array: The offset angle of an array.

To perform a Rectangular Array:

1. In the editor, select objects for the array.
 2. Select **Edit > Array > Rectangular**.
 3. Enter the X Count value.
 4. Enter the Y Count value.
 5. Enter the DX you want to use.
 6. Enter the DY you want to use.
 7. Enter the Angle of Array you want to use.
 8. Select **OK** to begin.
- a. If the array is incorrect, use the **Undo** command.

Polar Array



Method:

(Choose from the following)

- a. Total number of items and angle to fill
- b. Total number of items and angle between items
- c. Angle to fill and angle between items

Total # of items: Number of copies.

Angle to fill: Total angle (in degrees) for polar array

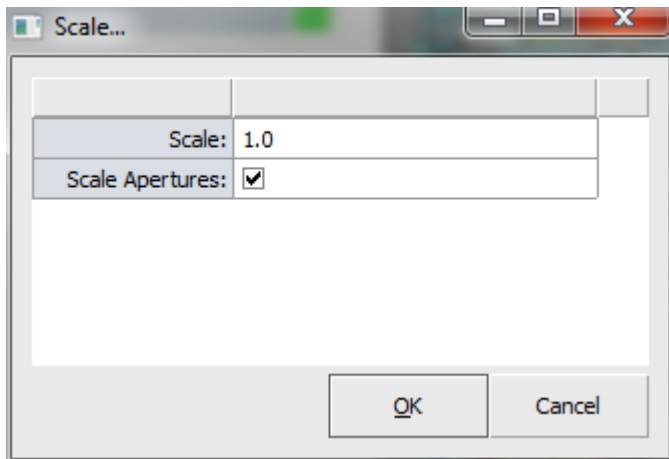
Angle between items: Individual angle between each item.

To perform a Polar Array:

1. In the editor, select objects for the array.
2. Select **Edit > Array > Polar**.
3. Specify a center point by clicking inside your design or enter the coordinates in the command window.
4. The Polar Array dialog box will appear.
5. Enter the Polar Array options as defined above.
6. Select **OK** to begin. If the array is incorrect, use the Undo command.

Edit > Scale

Scales selected objects. This function requires that you select objects before proceeding. When no objects are selected, this command becomes de-activated in the menu. If you would like to scale an entire layer by different X&Y values, use **Edit > Layers > Scale Layers X/Y**.



1. In the editor select the objects you would like to scale.
2. Select **Edit > Scale**. The scale dialog box will appear.
3. Enter the scale factor you would like to use.
4. Check the box if you want to scale apertures.
5. Select **OK**.
 - a. If the array is incorrect, use the Undo command.

Edit > Stretch

Automatically stretches objects crossing a defined rectangular area.

1. Select **Edit > Stretch**.
2. In the editor, specify first corner of an object by clicking it.
3. In the editor, specify opposite corner of an object by clicking it.
4. In the editor, specify a Basepoint.
5. In the editor, specify a second point of displacement.
 - a. Alternatively, use the command line to enter exact coordinates.

Edit > Split

Splits a path into two segments and the selected location.

1. Select **Edit > Split**.
2. In the editor, specify split location by clicking it.
3. In the editor, specify each point.

4. Right click to close the selection process or select **Esc**.

Edit > De-Embed

Automatically solves the polygon-inside-polygon problem for objects selected.

1. In the editor, select the objects for de-embedding.
2. Select **Edit > De-Embed**.
3. Object composite levels will automatically be defined.
 - a. Objects must reside on the same layer.
4. If the De-Embed is incorrect, use the Undo command.

Edit > Vertex



These are tools for modifying the vertex. They include Move, Insert, & Delete.

Move Vertex

1. Select **Edit > Vertex > Move Vertex**.
2. Select the Vertex.
3. Left click and drag to a new location.
4. Press **Esc** to exit.

Insert Vertex

1. Select **Edit > Vertex > Insert**.
2. Select Vertex Insert Location.
3. Press **Esc** to exit.

Delete Vertex

1. Select **Edit > Vertex > Delete**.
2. Select Vertex to delete.

3. Repeat as necessary. Press **Esc** to exit.

Edit > Trim



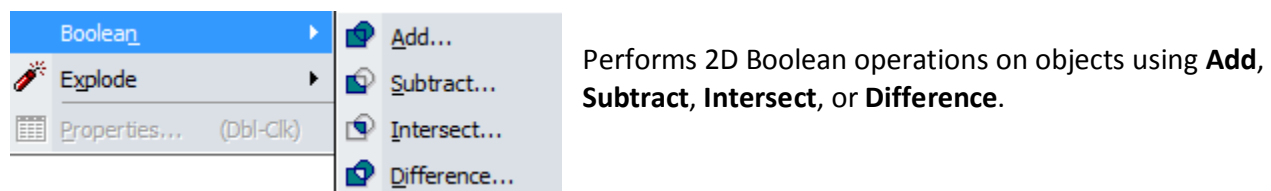
Trim Using Line:

1. Select **Edit > Trim > Using Line**.
2. In the editor, select first point for trim.
3. Select next point.
4. Repeat as necessary. Right click to end.

Trim Using Circle:

1. Select **Edit > Trim > Using Circle**.
2. In the editor, select first point for trim.
3. Select second point in circles circumference.
4. Select trace to trim.
5. Repeat as necessary. Right click to end.

Edit > Boolean



Add Boolean

1. Select **Edit > Boolean > Add**.
2. In the editor, select the base object.
3. In the editor, select the other object for Boolean operation.
4. When finished, press the **Esc** key to complete.

Subtract Boolean

1. Select **Edit > Boolean > Subtract**.
2. In the editor, select the base object.
3. In the editor, select the other object for Boolean operation.
4. When finished, press the **Esc** key to complete.

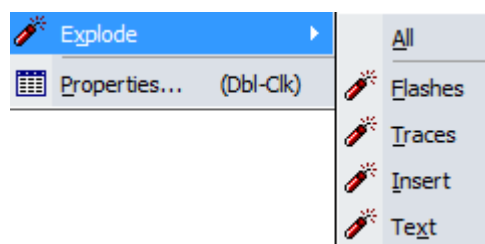
Intersect Boolean

1. Select **Edit > Boolean > Intersect**.
2. In the editor, select the base object.
3. In the editor, select the other object for Boolean operation.
4. When finished, press the **Esc** key to complete.

Difference Boolean

1. Select **Edit > Boolean > Difference**.
2. In the editor, select the base object.
3. In the editor, select the other object for Boolean operation.
4. When finished, press the **Esc** key to complete.

Edit > Explode



Edit > Explode converts selected objects into base objects. This function requires that you select objects before proceeding. When no objects are selected, this command becomes de-activated in the menu. There are five explode options: All, Flashes, Traces, Insert, & Text.

Explode All: Converts all objects into base objects.

1. In the editor, select objects to explode.
2. Select **Edit > Explode > All**.

3. When finished, press the **Esc** key to complete.

Explode Flashes: Converts flashes into polygons.

1. In the editor, select objects to explode.
2. Select **Edit > Explode > Flashes**.
3. When finished, press the **Esc** key to complete.

Explode Traces: Converts traces and polygons into segments.

1. In the editor, select objects to explode.
2. Select **Edit > Explode > Traces**.
 - a. Polygons will be exploded into individual segments with 1-mil width.
3. When finished, press the **Esc** key to complete.

Explode Insert

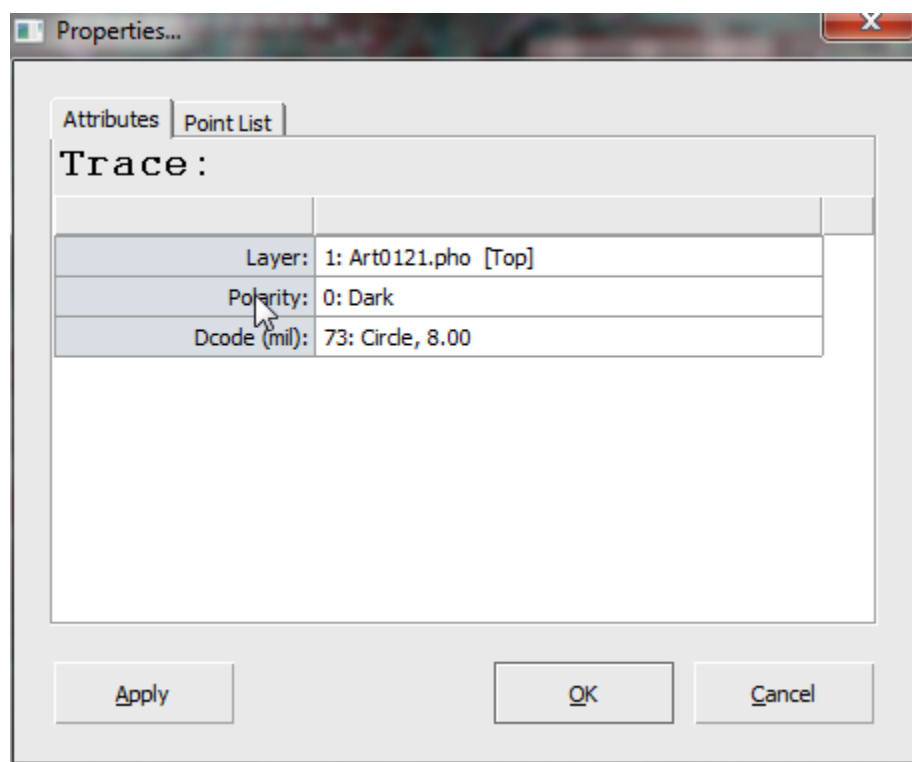
1. In the editor, select objects to explode.
2. Select **Edit > Explode > Traces**.
 - a. Polygons will be exploded into individual segments with 1-mil width.
3. When finished, press the **Esc** key to complete.

Explode Text: Converts text into polygons.

1. In the editor, select objects to explode.
2. Select **Edit > Explode > Text**.
3. When finished, press the **Esc** key to complete.

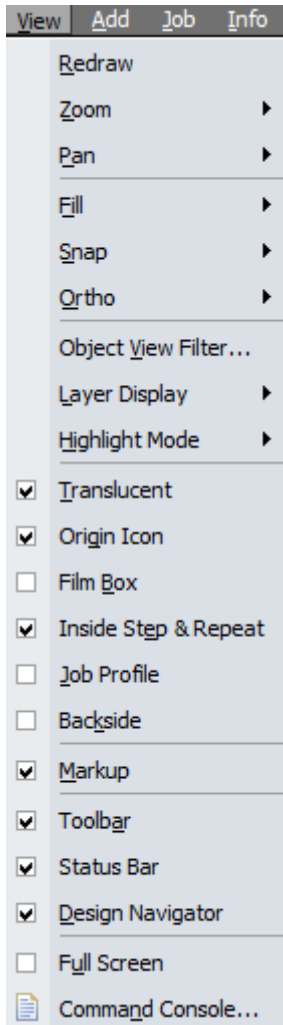
Edit > Properties

Edit properties allows you to view and modify the properties of individual or multiple objects. This function requires that you select objects before proceeding. When no objects are selected, this command becomes de-activated in the menu. Not only does this command allow you to view all object properties, you may also edit them. You can also double click any object to display this properties dialog. When multiple objects are selected, the "Multiple Objects" version of the Properties dialog box will be displayed



1. In the editor, select the object(s) that you want to view and edit properties.
2. Select **Edit > Properties**. The properties dialog box will appear.
3. Review all items displayed. Most items can be changed.
4. Click on the tab Point List to view or edit an object's vertices or center point. This is optional.
5. Click **OK** to exit the properties dialog box.
 - a. If any changes you make are incorrect, select **Undo**.

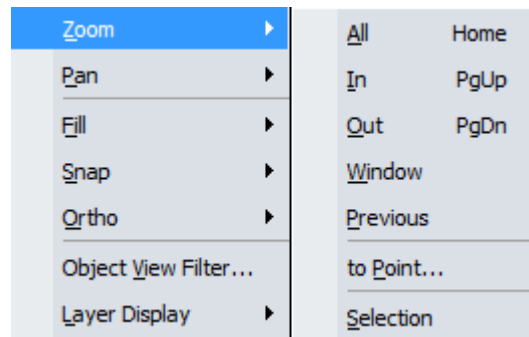
5.3 VIEW MENU



The view menu is responsible for the control and editing of object views, layer displays, workspace controls.

View > Redraw

Refreshes/Redraws the display in the editor window.



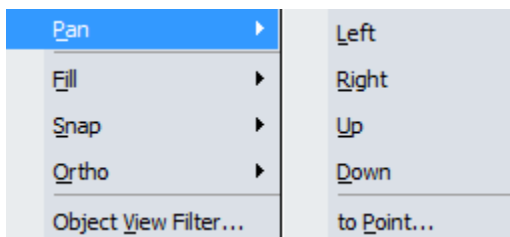
View > Zoom

Increases or decreases object magnification. There are multiple zoom commands available:

- a. **Zoom All:** Shows entire design area in editor.
- b. **Zoom In:** Doubles the magnification.
- c. **Zoom Out:** Halves the magnification.
- d. **Zoom Window:** Allows users to select a window around an area to be zoomed.
- e. **Zoom Previous:** Zooms to previous view.
- f. **Zoom to Point:** Zooms to point selected by user.
- g. **Zoom Selection:** Zooms into the extents of the current selection.

View > Pan

Panning allows the user to scroll about the current view port in the editor. Pan does not change magnification, only movement. There are multiple pan commands available:



Pan Left: Moves the view to the left by one-quarter of the view width.

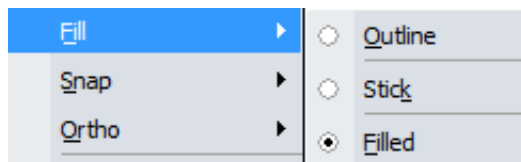
Pan Right: Moves the view to the right by one-quarter of the view width.

Pan Up: Moves the view up by one-quarter of the view width.

Pan Down: Moves the view down by one-quarter of the height of the view.

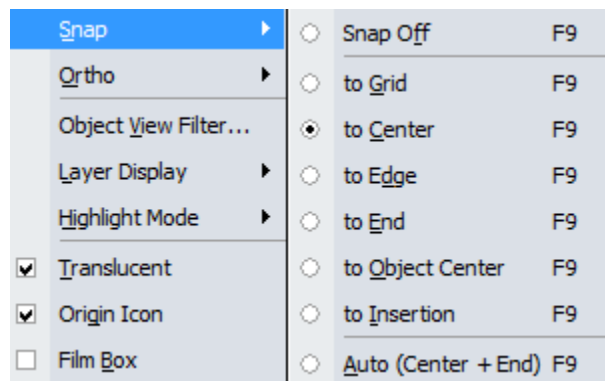
Pan to Point: Pans to a specific point.

View > Fill



View > Fill toggles the fill mode. Fill mode controls the filling of all boundaries using a layer's stipple pattern and color.

View > Snap

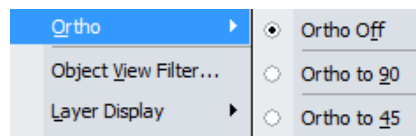


View > Snap Controls the cursor location in the editor. If snap is enabled, it allows the cursor to move onto the nearest grid point or object boundary point. The following snap commands are available: **Snap Off**, **Snap to Grid**, **Snap to Edge**, **Snap to End**, **Snap to Object Center**, **Snap to Insertion**, or **Auto Snap**.

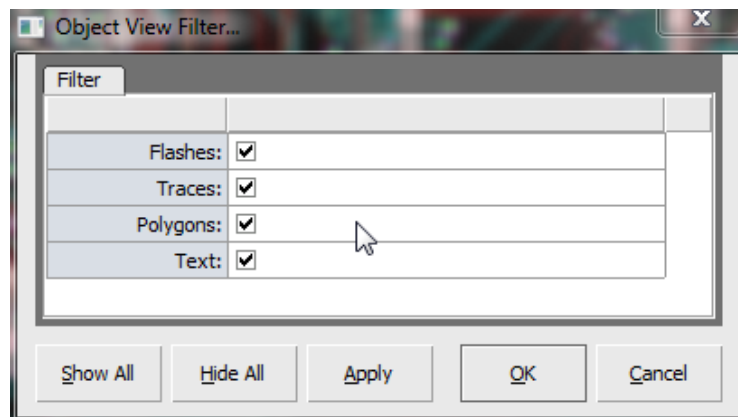


[Video: FAB 3000 Object Snap](#)

View > Ortho



View > Ortho controls the cursor direction in the editor. If **Ortho** is enabled, it allows the cursor to move in directional increments of 90 or 45 degrees.



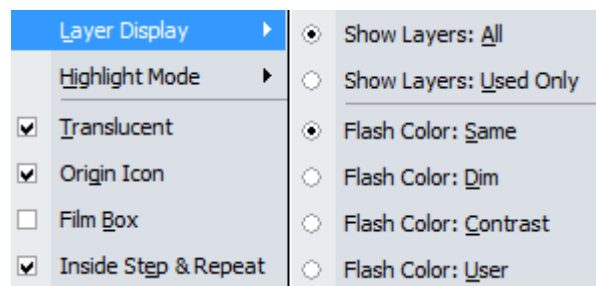
View > Object View Filter

Assigns view filters to an object. Use the check box to filter **Flashes**, **Traces**, **Polygons**, or **Text**.

Show All will automatically check all boxes.
Hide All will automatically uncheck all boxes.

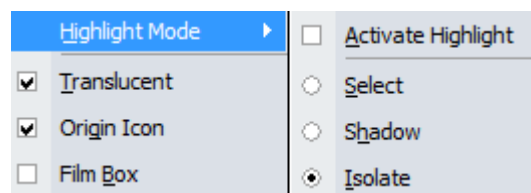
Select **Apply** to assign filters. Select **OK** to exit.

View > Layer Display



Layer Display controls the display of layers in the editor for easier viewing. Choose from several viewing options to make it easier to differentiate between layers.

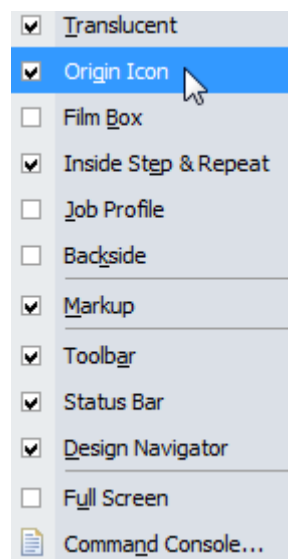
View > Highlight Mode



Highlight Mode enables highlights for object layers.

Additional View Options

Additional view options within the View menu can be activated via checkbox.



View > Translucent

Creates a translucent color over object layers.

View > Origin Icon

Toggles whether to display the origin icon in the editor. The origin icon is used for reference in a layout.

View > Film Box

Toggles whether to display the film box in the editor. The film box is usually used for reference before exporting Gerber or Postscript files.

View > Inside Step & Repeat

Toggles the viewing of objects inside Inserts.

View > Job Profile

Toggles the viewing of the Job profile.

View > Backside

Views the Job from the backside. Helpful when editing needs to be performed from the reverse view of the board.

View > Markup

Toggles whether to display markup (if any).

View > Toolbar

Toggles the visibility of the toolbar.

View > Status Bar

Toggles the visibility of the status bar. The status bar is located at the bottom of the workbench and it displays information about your current command, units, and cursor location.

View > Design Navigator

Increases the size of the editor workspace and command line. Using this feature will hide the layer display and Ariel view.

View > Full Screen

Allows the user to increase the editor window to the full allowable size of the screen.

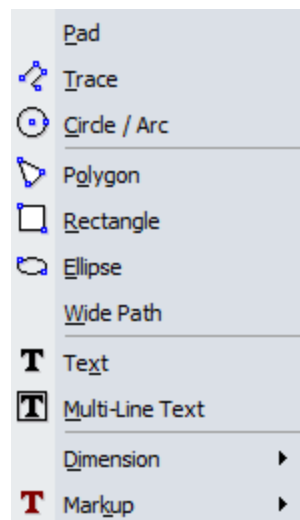
View > Command Console

Displays a pop out widget which lists all command line dialog.



[Video: PCB Backside View and Flip using FAB 3000](#)

5.4 ADD MENU



The add menu adds specific CAD shapes, text, and markups to a design.

Add > Pad

Adds a flash to the active design. At least one aperture must have been previously defined. The flash will be placed on the active layer. To add a pad:

1. Select **Add > Pad**.
2. Assign the active Dcode. The active Dcode combo box is located in the toolbar.
3. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If the active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. Select the center point for the flash.
5. Repeat steps 2-4 for multiple pads.
6. To exit, right click or press Esc.
 - a. If the command is incorrect, use **Undo**.
 - b. To add a flash to a net, use **Tools > Nets > Edit Nets**.
 - c. To add a flash to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Trace

Adds a trace (line) to the active design. At least one aperture must have been previously defined.

1. Select **Add > Trace**.
2. Assign the active Dcode. The active Dcode combo box is located in the toolbar.

3. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. In the editor, select the first point for the trace.
5. In the editor, select the next point for the trace.
 - a. You also have the following additional options when adding Trace.
 1. End: Ends point list. To activate, press E key.
 2. Close: End and close point list. To activate, press C key.
 3. Back: Remove the last point from the current point list. To activate, press B key.
6. Once you've finished your trace, choose End or Close.
7. Repeat steps 2-6 for additional traces.
8. To exit, right click or press **Esc** key.
 - a. If the command is incorrect, use Undo.
 - b. To add a trace to a net, use **Tools > Nets > Edit Nets**.
 - c. To add a trace to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Circle Arc

Adds a circle to the active design.

1. Select **Add > Circle / Arc**.
2. Assign the active Dcode. The active Dcode combo box is located in the toolbar.
3. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. In the editor, select the center point for the circle.
5. In the editor, select the radius, orientation, and degrees for the circle.
6. Repeat steps 2-5 for additional **Circle/Arc** commands.
7. To exit, right click or press **Esc** key.
 - a. If the command is incorrect, use **Undo**.

- b. To add an arc/circle to a net, use **Tools > Nets > Edit Nets**.
- c. To add an arc/circle to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Polygon

Adds a polygon to the active design. No defined aperture is required.

1. Select **Add > Polygon**.
2. Assign the active layer. The active layer combo box is located in the toolbar.
3. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. In the editor, select the first point for the polygon.
5. In the editor, select the next point for the polygon.
 - a. End: Ends point list. To activate, press **E** key.
 - b. Close: End and close point list. To activate, press **C** key.
 - c. Back: Remove the last point from the current point list. To activate, press **B** key.
6. Once you have finished your trace select **Close (C key)** or right click.
7. Repeat steps 2-5 for additional **Polygon** commands.
8. To exit, right click or press **Esc** key.
 - a. If the command is incorrect, use **Undo**.
 - b. To add an arc/circle to a net, use **Tools > Nets > Edit Nets**.
 - c. To add an arc/circle to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Rectangle

Adds a filled rectangle to the active design.

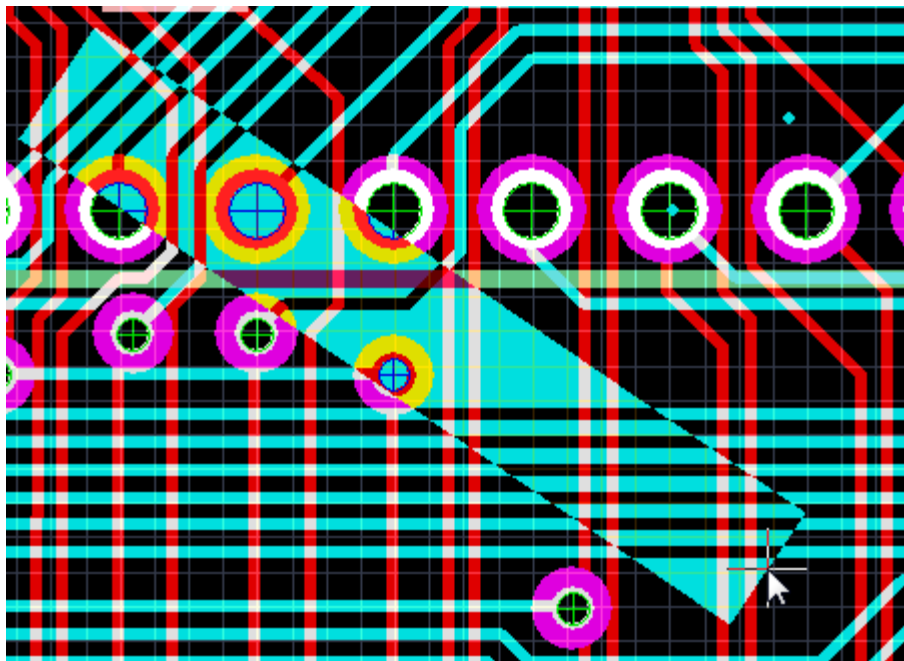
1. Select **Add > Rectangle**.
2. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. In the editor, select the first corner for the rectangle.

5. In the editor, select the opposite corner for the rectangle.
6. Repeat steps 2-5 for additional **Rectangle** commands.
7. To exit, right click or press **Esc** key.
 - a. If the command is incorrect, use **Undo**.
 - b. To add an arc/circle to a net, use **Tools > Nets > Edit Nets**.
 - c. To add an arc/circle to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Ellipse

Adds a filled Ellipse to the active design.

1. Select **Add > Ellipse**.
2. Assign the active layer. The active layer combo box is located in the toolbar.
 - a. New objects will be placed on the active layer. If active layer is turned off, you will not be able to add the new object (until that layer is turned on).
4. In the editor, select the first corner for the ellipse.
5. In the editor, select the opposite corner for the ellipse.
6. Repeat steps 2-5 for additional **Ellipse** commands.
7. To exit, right click or press **Esc** key.
 - a. If the command is incorrect, use **Undo**.
 - b. To add an arc/circle to a net, use **Tools > Nets > Edit Nets**.
 - c. To add an arc/circle to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Wide Path

Adds a wide path (polygon) to the active design. No defined aperture is required.

1. Select **Add > Wide Path**.
2. Select Path Width. Width cannot be 0.
3. Select Path style. Path styles include: Truncate (no path extension) and Extend (extend by ½ path width).
4. Assign the active layer in the Layer Display.

a. New objects will be displayed on the active layer. If the active layer is turned off, you will not be able to add the new object (until the layer is turned on).

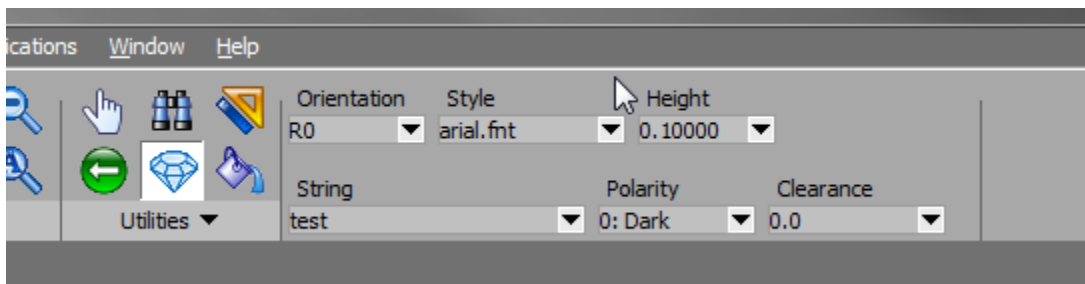
5. Select the first point for the trace.
6. Select the next point for the trace.
 - a. Additional options are available: **End, Close, & Back**.
 1. **End**: Ends point list. Press **E** key.
 2. **Close**: End and close point. Press **C** key.
 3. **Back**: Remove the last point from the current point list. Press **B** key.
7. Once you have finished your trace, select either **End or Close**.
8. Repeat steps 4-7 for additional wide paths.
9. To exit, right click or press **Esc**.
 - a. If the command is incorrect, select **Undo**.
 - b. To add a wide path to a net, use **Tools > Nets > Edit Nets**.

- c. To add a wide path to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Text

Adds text to the active design.

1. Select **Add > Text**.
2. Assign the active layer by selecting a layer in the layer display panel.
 - a. New objects will be displayed on the active layer. If the active layer is turned off, you will not be able to add the new object (until the layer is turned on).
3. Enter Text in the String dropdown located at the top in the workspace menu bar.

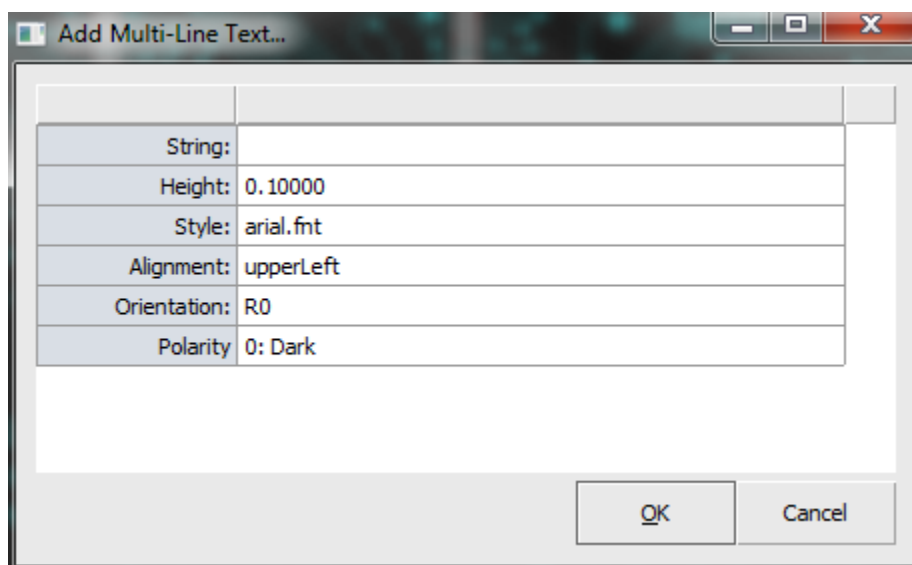


4. Enter Text Height in the dropdown located at the top in the workspace menu bar.
5. Enter Text Style in the dropdown located at the top in the workspace menu bar.
 - a. The text style combo box includes all FAB 3000 compiled fonts located in the FAB 3000 folder **fonts**. If there is not a text style available that you want, you may create your own FAB 3000 compiled font from any TrueType or AutoCAD SHX font. To do this, go to **Help > Compile True Type Fonts**.
6. Enter Orientation in the dropdown located at the top in the workspace menu bar.
7. In the editor, select an origin point for text.
8. Repeat steps 2-7 for multiple text entries.
 - a. If the command is incorrect. Select **Undo**.
9. To add Text to a composite, use **Edit > Properties** and locate **Composite Level**.

Add > Multi-Line

Adds multi-line (paragraph) text to the active design.

1. Select **Add > Multi-Line Text**.
2. Assign the active layer by selecting a layer in the layer display panel.
 - a. New objects will be displayed on the active layer. If the active layer is turned off, you will not be able to add the new object (until the layer is turned on).
3. Select the first corner for multi-line text.
4. Enter the opposite corner for multi-line text. The **Add Multi-Line Text** dialog box will appear.



5. Enter a Multi-Line text String.

6. Enter a text Height.

7. Select a text Style (font).

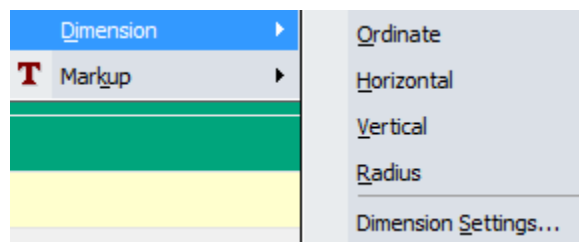
- a. The text style combo box includes all FAB 3000 compiled fonts located in the FAB 3000 folder **fonts**. If there is not a text style available that you want, you may create your own FAB 3000 compiled font from any TrueType or AutoCAD SHX font. To do this, go to **Help > Compile True Type Fonts**.

6. Select the Alignment to be used.
 - a. The alignment options are: **upperLeft, centerLeft, upperCenter, centerCenter, lowerCenter, upperRight, centerRight, lowerRight**.
7. Select the Orientation.

Enter Orientation in the dropdown located at the top in the workspace menu bar.

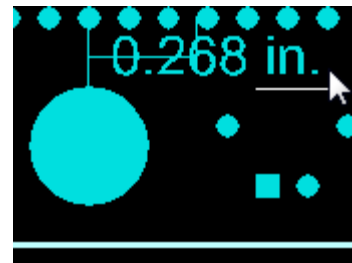
8. Select the Composite Level of the text.
9. Press **OK** to create the text.

- a. If the command is incorrect, select **Undo**.
- b. To add Multi-Line Text to a composite, use **Edit > Properties** and locate **Composite Level**.



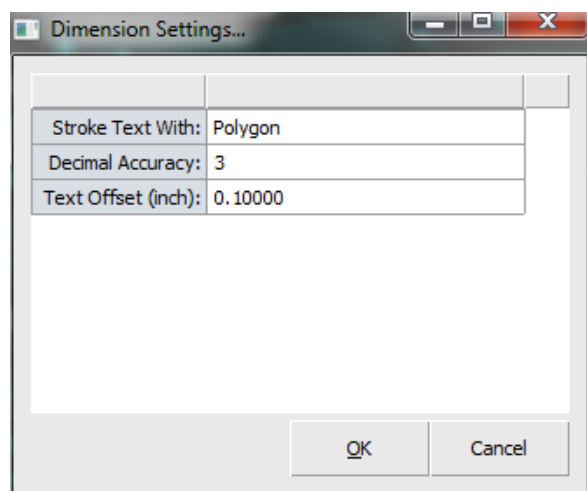
Add > Dimension

This command adds a dimension value to the active design. Select from the following: **Ordinate**, **Horizontal**, **Vertical**, &



Radius. You can also modify the settings of your dimension text by selecting **Dimension Settings**.

Dimension Settings



For dimension settings, choose from the following:

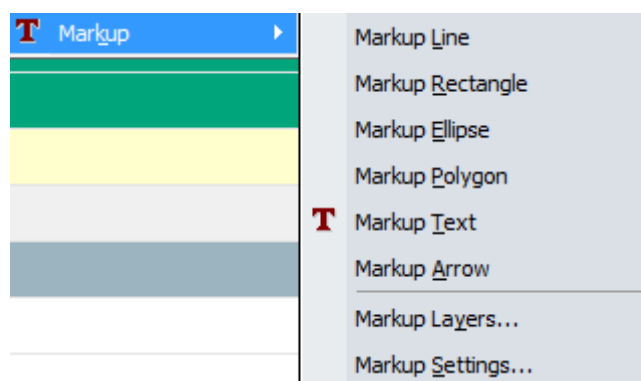
Stroke Text With: Choose either Polygon or Line.

Decimal Accuracy: Choose decimal accuracy to be displayed.

Text Offset (inch): Enter offset value of text.

Add > Markup

Add lines, rectangles, ellipses, polygons, text, and arrows to your design. You can also markup layers.



Markup Line: Adds a markup line to the active design.

1. Select **Add > Markup > Markup Line**.
2. In the editor, pick the first point.
3. In the editor, pick the next point.
4. Repeat the steps until complete. Use **Close** or **End** to deactivate.

The markup line will be placed on the active layer.

Markup Rectangle: Adds a markup rectangle to the active design.

1. Select **Add > Markup > Markup Rectangle**.
2. In the editor, pick the first corner.
3. In the editor, pick the opposite corner.

The markup rectangle will be placed on the active layer.

Markup Ellipse: Adds a markup ellipse to the active design.

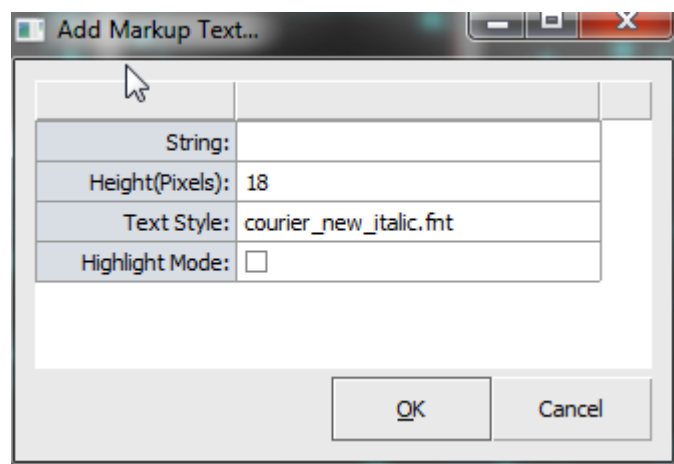
1. Select **Add > Markup > Markup Ellipse**.
2. In the editor, pick the first corner.
3. In the editor, pick the opposite corner.

The markup ellipse will be placed on the active layer.

Markup Polygon: Adds a markup polygon to the active design.

1. Select **Add > Markup > Markup Rectangle**.
2. In the editor, pick the first point.
3. In the editor, pick the next point.
4. Repeat the steps until complete. Use **Close** or **End** to deactivate.

The markup polygon will be placed on the active layer.



Markup Text: Adds markup text to the current markup layer.

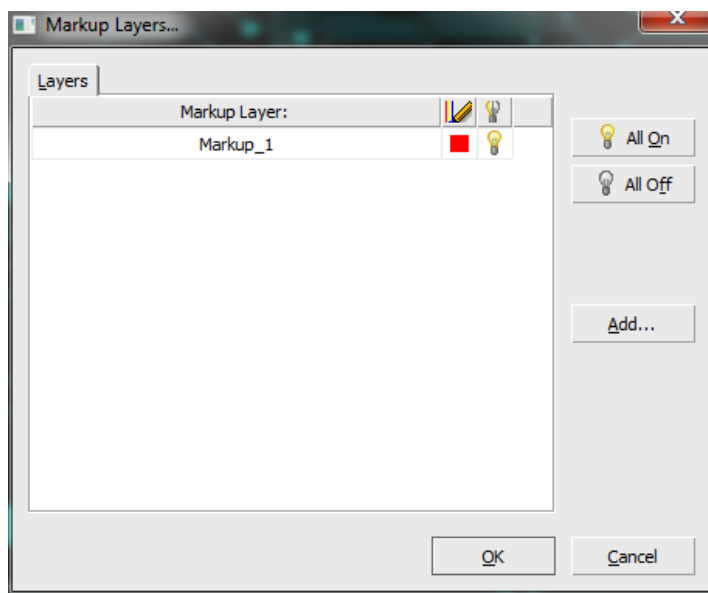
1. Select **Add > Markup > Markup Text**.
2. Select bounding rectangle for markup text.
3. A Markup Text dialog box appears. Edit or verify Text options.
4. Select **OK**.

The markup text will be placed on the active layer.

Markup Arrow: Adds a markup leader with an “arrow” at the first point, to the current markup layer.

1. Select **Add > Markup > Markup Arrow**.
2. In the editor, pick the first point (where the arrow will point to).
3. In the editor, pick the next point.
4. Repeat steps 2-3 or until finished. Right click to end.

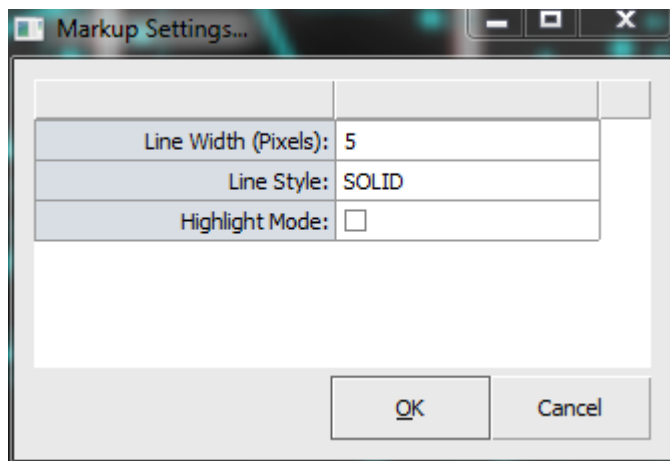
The markup arrow will be placed on the active markup layer.



Markup Layers: Defines all markup layers. These layers only contain markup objects and are not included during export.

Add > Markup > Markup Settings

Defines all markup settings.

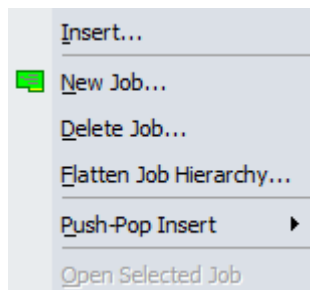


Line Width (Pixels): The default line width to use for any new markup object. The width is not set to a pure physical unit, instead is it assigned by the (Current zoom level) X (# of Pixels).

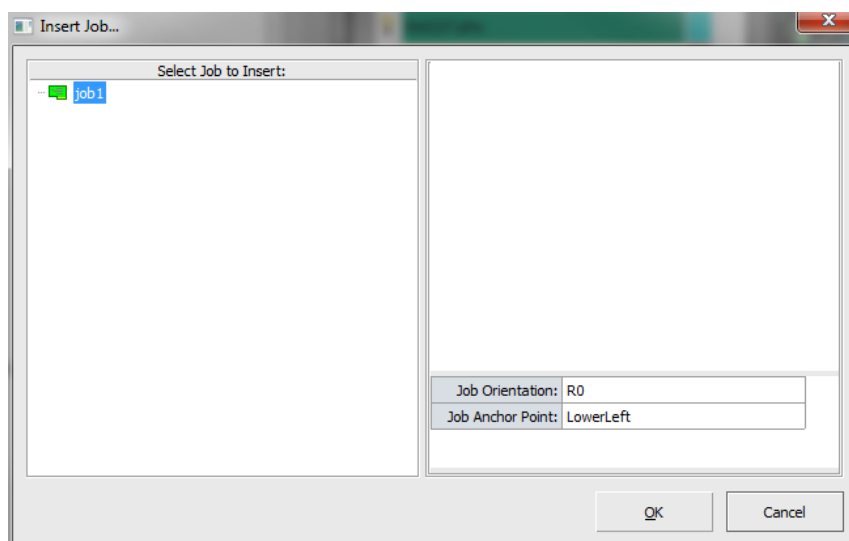
Line Style: Choose the line style, and choose from either: SOLID, ONOFF_DASH, DOUBLE_DASH.

Highlight Mode: Make any new markup object "see-through".

5.5 Job Menu



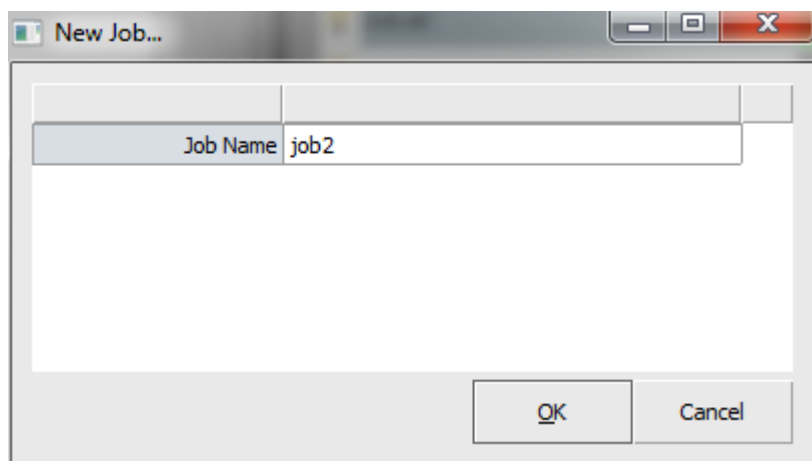
The Job menu allows you to add, edit and delete Jobs.



Job > Insert Job

Adds an Insert in to the active Job. An Insert represents the inclusion of one Job as part of another.

1. Select **Job > Insert Job**.
2. Select the Job to Insert.
3. Select a Job Orientation. Choose from **Lower Left, Center, or Origin**.
4. Select **OK**.



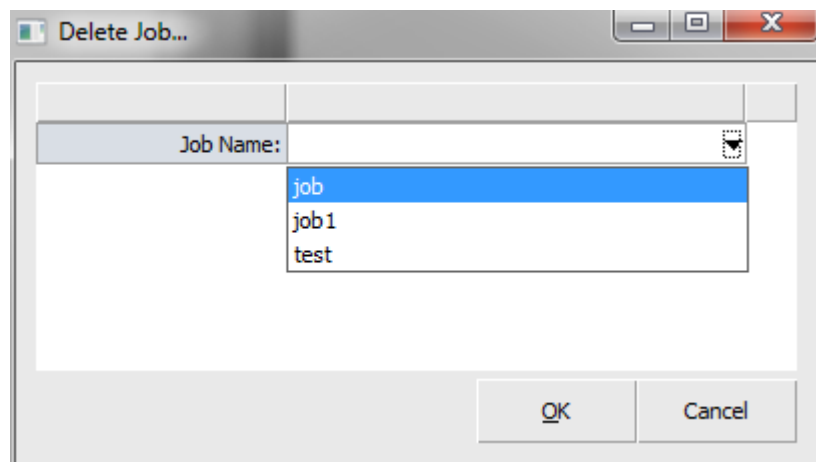
Job > New Job

Creates a new Job. The new Job will appear in the editor tab.

1. Select **Job > New Job**.
2. Enter a name for the Job.
3. Select **OK**.

Job > Delete Job

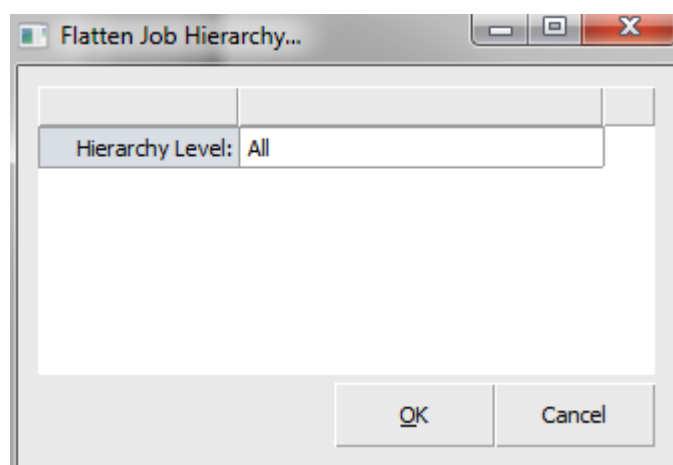
Deletes a Job from the workspace.



1. Select **Job > Delete Job**.
2. Choose which Job to delete from the drop down menu.
3. Select **OK**.

Job > Flatten Job Hierarchy

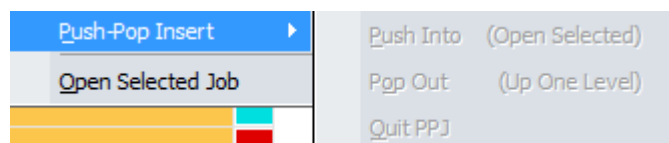
Flattens all Inserts in a Job.



1. Select **Job > Flatten Job Hierarchy**.
2. Select a Hierarchy level from the dropdown menu.
3. Select **OK**.

Job > Push-Pop Insert

Allows the contents inside an Insert to be viewed and edited in place within the parent Job.



Push-Pop Insert is divided into several actions: **Push Into**, **Pop Out**, and **Quit PPJ**.

Push Into: Allows the user to edit/analyze geometries inside a selected Insert.

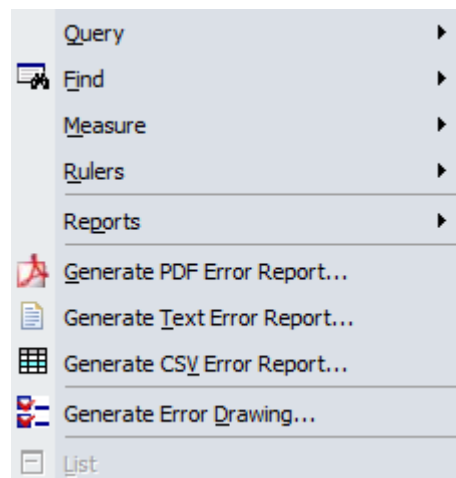
Pop Out: Closes the Insert.

Quit PPI: Completely exits the Push/Pop Job. All changes will be ignored.

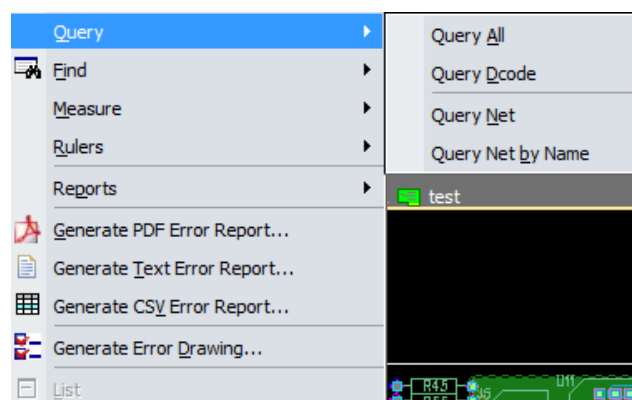
Job > Opens Select Job

Opens selected Insert master Job in a new editor window.

5.6 Info Menu



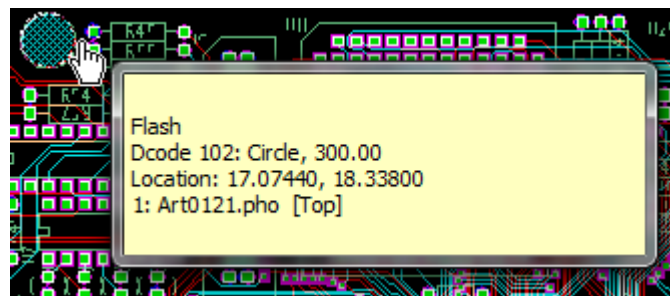
Info menu provides general information, reporting, and error details.



Info > Query

Info > Query actions display information about a selected object.

Info > Query > All

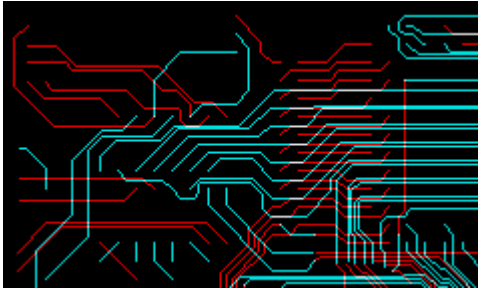


Displays information of a selected object. Depending on the object selected, Query reports different types of information.

1. Select **Info > Query > Query All**.
2. Select the object to query. Move the cursor over the object and left click.
3. A pop up will display the information for the selected object.

4. To exit, right click or press the **Esc** key.

If you have more than one item in the same location and query does not display the object you wanted to view, simply click the left mouse button again and FAB 3000 will cycle through the other objects in the same location until you see the one you want.



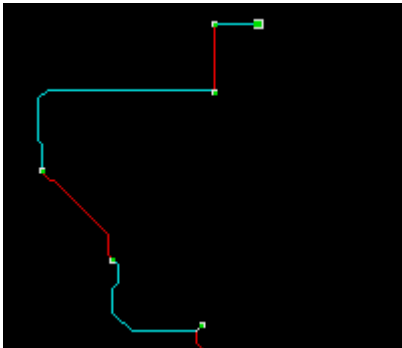
Info > Query > Dcode

Displays the Dcode information only of a selected object.

1. Select **Info > Query > Query Dcode**.
2. Select an object to query Dcode. Move the cursor over the object and left click.

3. The selected object information will be displayed within the editor window.
4. To exit, right click or press the **Esc** key.

If you have more than one item in the same location and query does not display the object you wanted to view, simply click the left mouse button again and FAB 3000 will cycle through the other objects in the same location until you see the one you want.



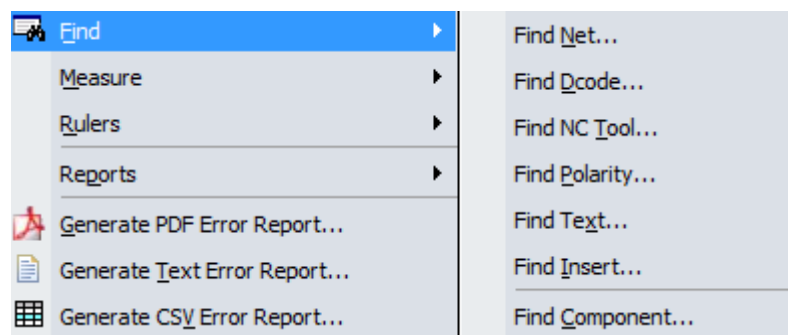
Info > Query > Query Net

Displays net information of a selected object. Depending on the object selected, Query Dcode reports different types of information.

1. Select **Info > Query > Query Net**.
2. Select an object to Query Net. Move the cursor over the object and left click.

3. The selected object information will be displayed within the editor window.
4. To exit, right click or press the **Esc** key.

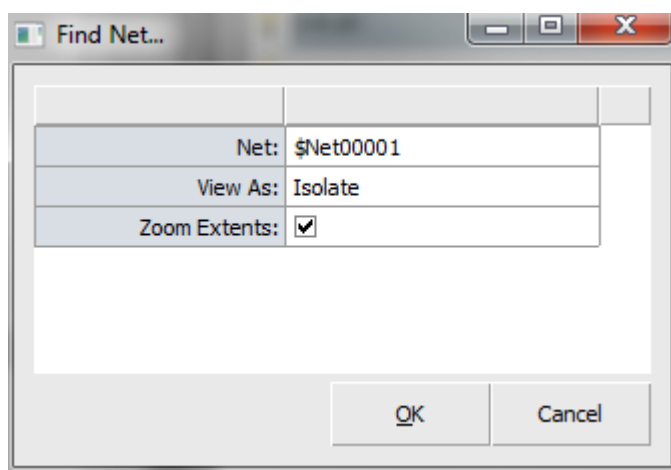
If you have more than one item in the same location and query does not display the object you wanted to view, simply click the left mouse button again and FAB 3000 will cycle through the other objects in the same location until you see the one you want.



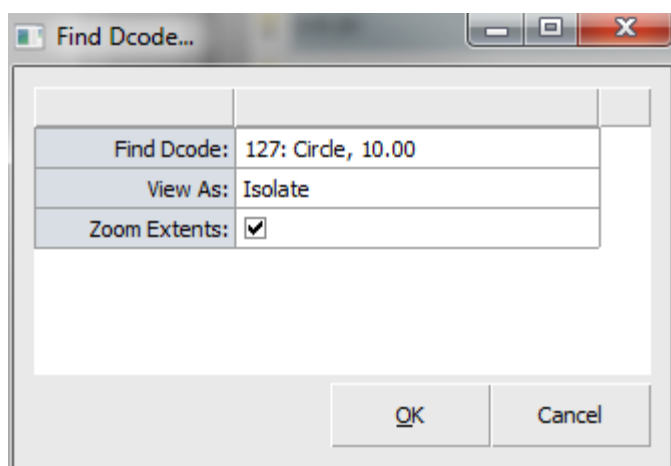
Info > Find > Find Net

Search active Jobs for defined Net.

1. Select **Info > Find > Find Net**. The Find Net dialog box will be displayed.
2. Select the Net to find from the drop down menu.



3. Select View As. Choose from either Highlight or Isolate.
 - a. **Highlight:** All net objects on visible layers highlighted.
 - b. **Isolate:** All net objects are visible, while all other objects are hidden.
4. Select Zoom Extents. If checked, this zooms out to view the entire net.
5. To exit, right click or press **Esc**.

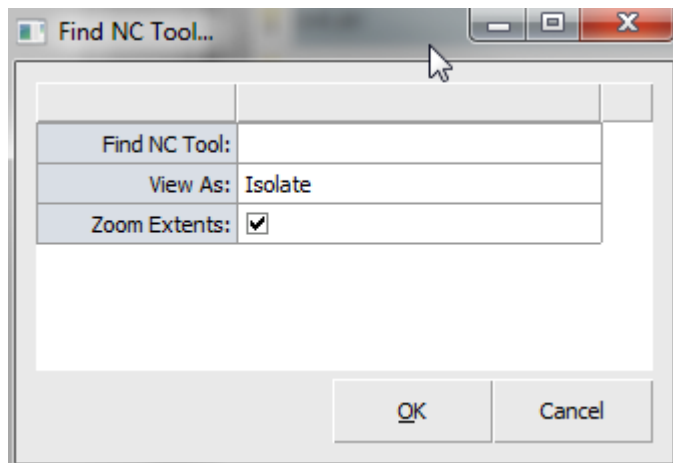


Info > Find > Find Dcode

Searches active Job for defined Dcode.

1. Select **Info > Find > Find Dcode**. The Find Dcode dialog box will be displayed.
2. Select Dcode to find from the dropdown menu.
3. Select View As. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight:** All net objects on visible layers highlighted.
 - b. **Isolate:** All net objects are visible, while all other objects are hidden.
4. Select Zoom Extents. If checked, this zooms out to view all Dcodes.

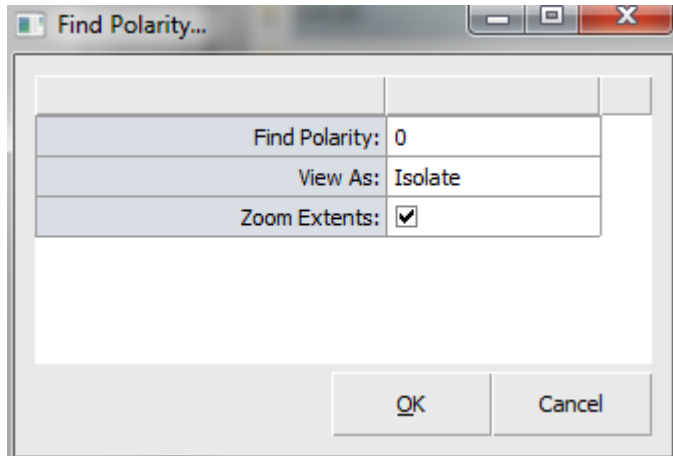
Info > Find > Find NC Tool



Searches active Job for defined NC Tool.

1. Select **Info > Find > Find NC Tool**. The Find NC Tool dialog box will be displayed.
2. Select NC Tool to find from the dropdown menu.
3. Select **View As**. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight**: All net objects on visible layers highlighted.
 - b. **Isolate**: All net objects are visible, while all other objects are hidden.
4. Select **Zoom Extents**. If checked, this zooms out to view all NC Tools.
5. To exit, right click or press **Esc**.

Info > Find > Find Composite Level

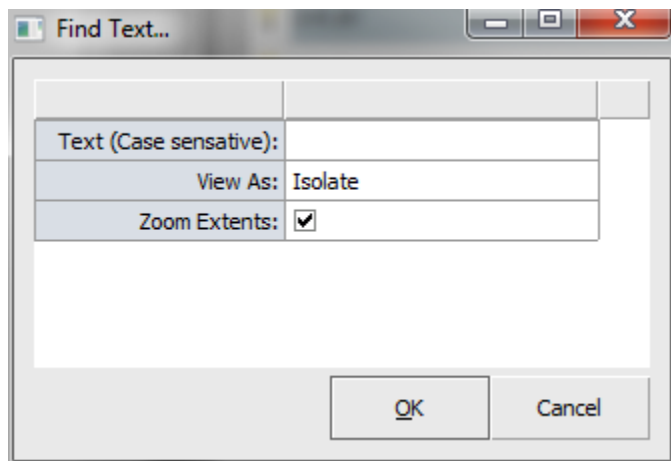


1. Select **Info > Find > Find Composite Level**. The Find Composite Level dialog box will be displayed.
2. Select Composite Level to find from the dropdown menu.
3. Select **View As**. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight**: All net objects on visible layers highlighted.
 - b. **Isolate**: All net objects are visible, while all other objects are hidden.

4. Select **Zoom Extents**. If checked, this zooms out to view all composite level objects.
5. To exit, right click or press **Esc**.

Info > Find > Find Text

Searches active Job for defined Text.



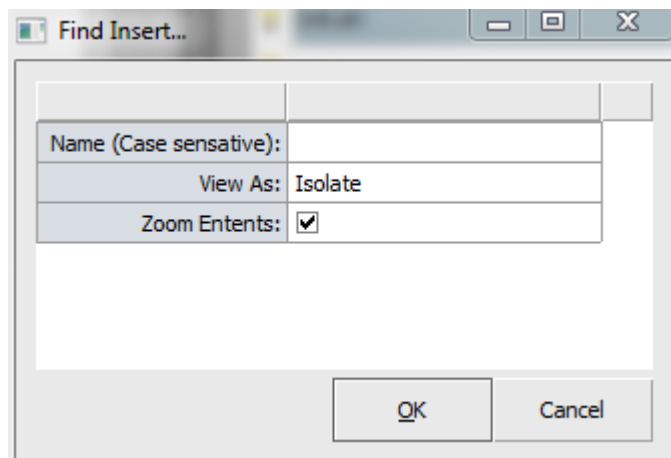
1. Select **Info > Find > Find Text**. The Find Text dialog box will be displayed.
2. Enter Text to find from the dialog box.
3. Select **View As**. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight**: All net objects on visible layers highlighted.
 - b. **Isolate**: All net objects are visible, while all other objects are hidden.

4. Select **Zoom Extents**. If checked, this zooms out to view all composite level objects.

5. To exit, right click or press **Esc**.

Info > Find > Find Insert

Searches active Job for defined Insert.

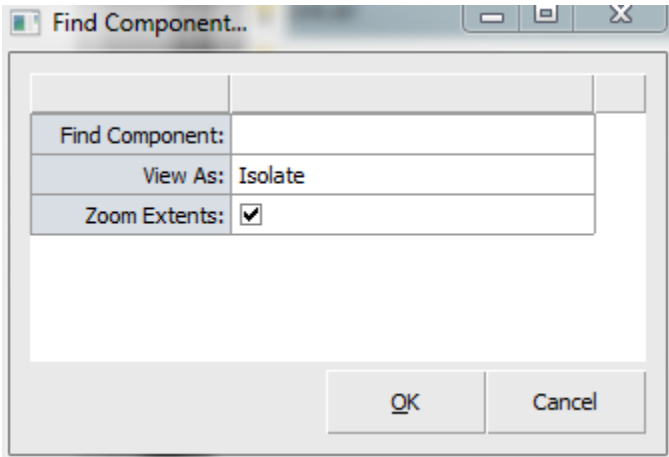


1. Select **Info > Find > Find Insert**. The Find Insert dialog box will be displayed.
2. Enter Name to find from the dialog box.
3. Select **View As**. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight**: All net objects on visible layers highlighted.
 - b. **Isolate**: All net objects are visible, while all other objects are hidden.

- 4. Select **Zoom Extents**. If checked, this zooms out to view all composite level objects.
- 5. To exit, right click or press **Esc**.

Info> Find > Find Component

Search an active Job for defined Components.



- 1. Select **Info > Find > Find Component**. The Find Component dialog box will be displayed.
- 2. Select a Component to find from the drop down menu.
- 3. Select View As. Choose from either **Highlight** or **Isolate**.
 - a. **Highlight**: All net objects on visible layers highlighted.
 - b. **Isolate**: All net objects are visible, while all other objects are hidden.

- 4. Select **Zoom Extents**. If checked, this zooms out to view all composite level objects.
- 5. To exit, right click or press **Esc**.

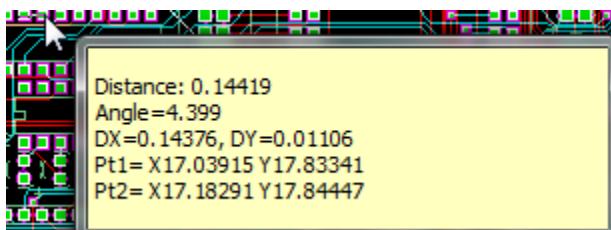
Info > Measure

Provides interactive measurement readouts for selected Points, Objects, and Nets.



Info > Measure > Point-Point

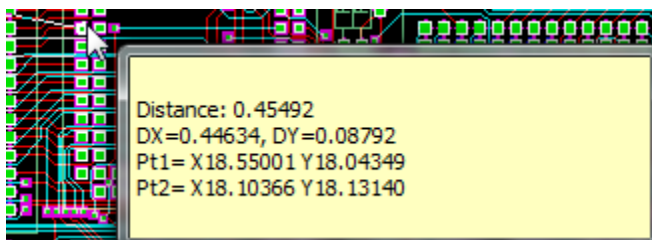
Measurement readout between two selected points within the editor.



1. Select **Info > Measure > Point-Point**.
2. Select the first point of measurement.
3. Select the second point of measurement.

4. A pop up menu will display showing the distance information between these two points.
 - a. Distance: Distance between two selected points.
 - b. DX/DY: Horizontal and vertical displacement between the two selected points.
 - c. Angle: The angle (in degrees) between the two selected points.
5. To exit, right click or press **Esc**.
 - a. After you have selected the first point, an interactive measurement is continuously calculated and displayed in the dynamic window. This helps to measure multiple distances using the same base reference point.
 - b. Use snap for precise measurements between objects and grid points.

Info > Measure > Object-Object



Measurement readout between two objects, taking into account true width.

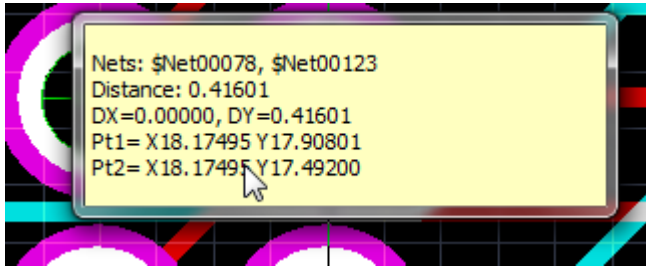
1. Select **Info > Measure > Object-Object**.
2. Select the first point of measurement.

3. Select the second point of measurement.
4. A pop up menu will display showing the distance information between these two points.
 - a. Distance: Distance between two selected points.
 - b. DX/DY: Horizontal and vertical displacement between the two selected points.
 - c. Angle: The angle (in degrees) between the two selected points.
5. To exit, right click or press **Esc**.
 - a. After you have selected the first point, an interactive measurement is continuously calculated and displayed in the dynamic window. This helps to measure multiple distances using the same base reference point.

- b. Use snap for precise measurements between objects and grid points.

Info > Measure > Net-Net

Determines closest adjacent distance between two nets.



1. Select **Info > Measure > Object-Object**.
2. Select the first point of measurement.
3. Select the second point of measurement.
4. A pop up menu will display showing the distance information between these two points.

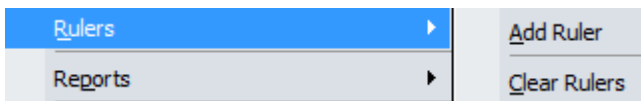
- a. Distance: Distance between two selected points.
- b. DX/DY: Horizontal and vertical displacement between the two selected points.
- c. Angle: The angle (in degrees) between the two selected points.

5. To exit, right click or press **Esc**.

- a. After you have selected the first point, an interactive measurement is continuously calculated and displayed in the dynamic window. This helps to measure multiple distances using the same base reference point.
- b. Use snap for precise measurements between objects and grid points.

Info > Rulers

Applies ruler measurements directly to the design.



Info > Rulers > Add Ruler

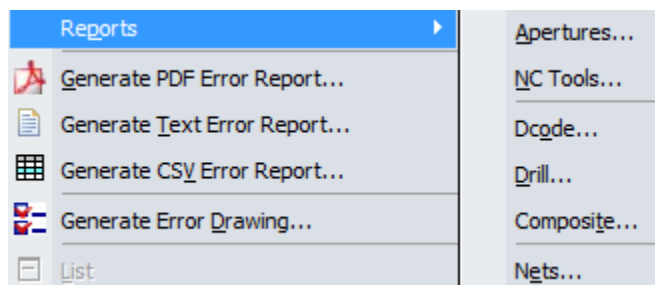
Adds a ruler and measurement information directly to the design by selecting two points.

Info > Rulers > Clear Ruler

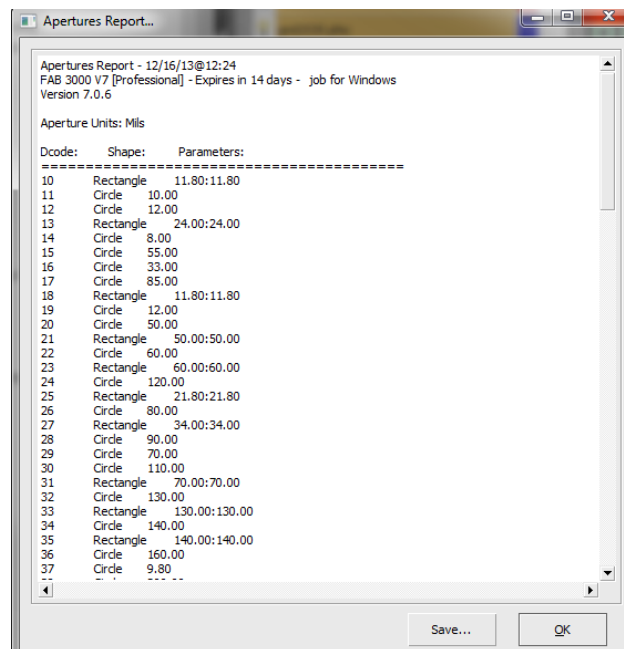
Clears all displayed rulers in the design.

Info > Reports

Generates detailed reports for the active design. This includes reports for: **Apertures, NC Tools, Dcode, Drill, Composites, & Nets.**



These reports are presented in a table file and displayed in the editor.



Info > Generate PDF Error Report

Generates a detailed report with graphics in PDF format. There are several PDF report types to choose from:

- DRC/DFM Violations
- Compare Net Errors
- Join Errors
- Sales RFQ
- Fabrication Drawing

Info > Generate Text Error Report

Generates a detailed report in Text/ASCII format.

Info > Generate CSV Error Report

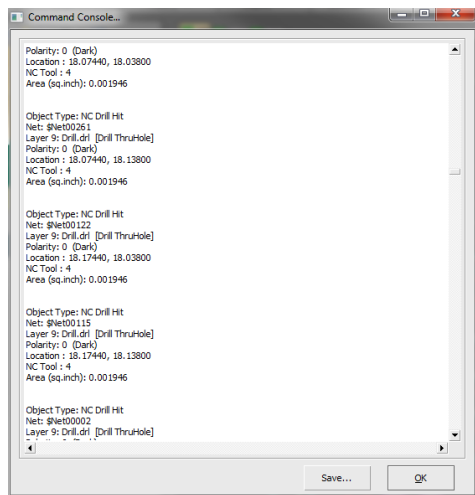
Generates a detailed error report in CSV format which may be imported to various spreadsheet applications.

Info > Generate Error Drawing

Generates marker objects which may be imported into various CAD systems for concurrent verification.

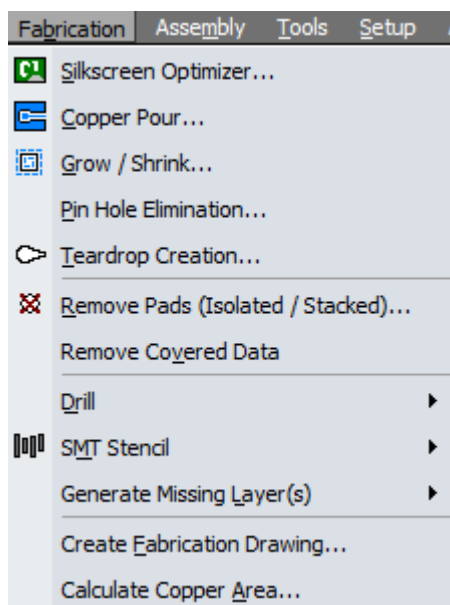
Info > List

Displays all properties of selected objects in a separate command console. The command allows the complete display of multiple object properties simultaneously.



1. Select object(s) to list within the editor.
2. Select **Info > List**. A command console will appear with all object properties listed.
3. If necessary, select **Save** to save the report to a specific directory.
4. Select **OK** to exit.

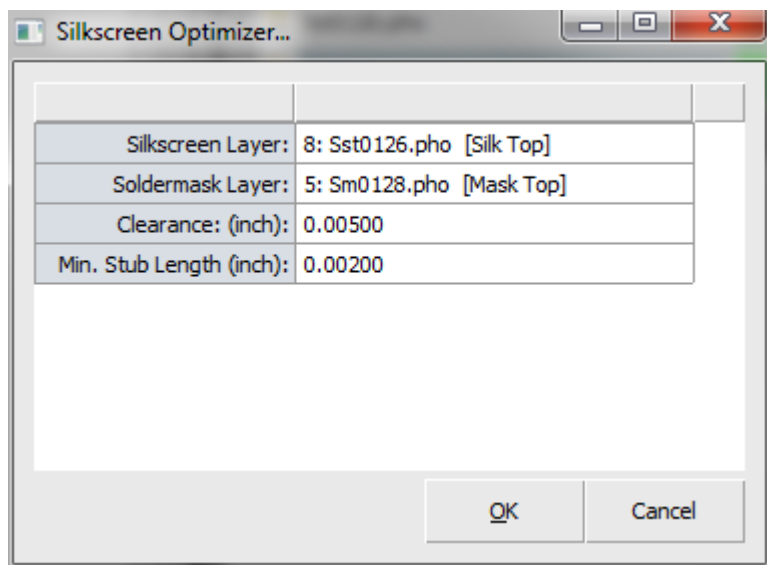
5.7 Fabrication Menu



The fabrication menu provides a number of tools for editing and updating your design prior to manufacturing.

Fabrication > Silkscreen Optimizer

When silkscreen data overlaps soldermask areas, it makes electrical testing and SMT soldering more difficult. Silkscreen optimizer removes the overlapping silkscreen data from soldermask pads.



1. Select **Fabrication > Silkscreen Optimizer**.
2. Select the Silkscreen Layer from the drop down men.
3. Choose the **Soldermask Layer** or "Mask" layer from the drop down menu.
 - a. Make sure to select the appropriate Silkscreen & Mask layers. A common mistake occurs when a user selects a Top Silkscreen against a Bottom Mask.

4. Verify silkscreen **Clearance** from the soldermask area.
5. Verify the Min. Stub Length for the silkscreen layer.
6. Press **OK** to begin Silkscreen Optimizer.
7. A results dialog box will be displayed.

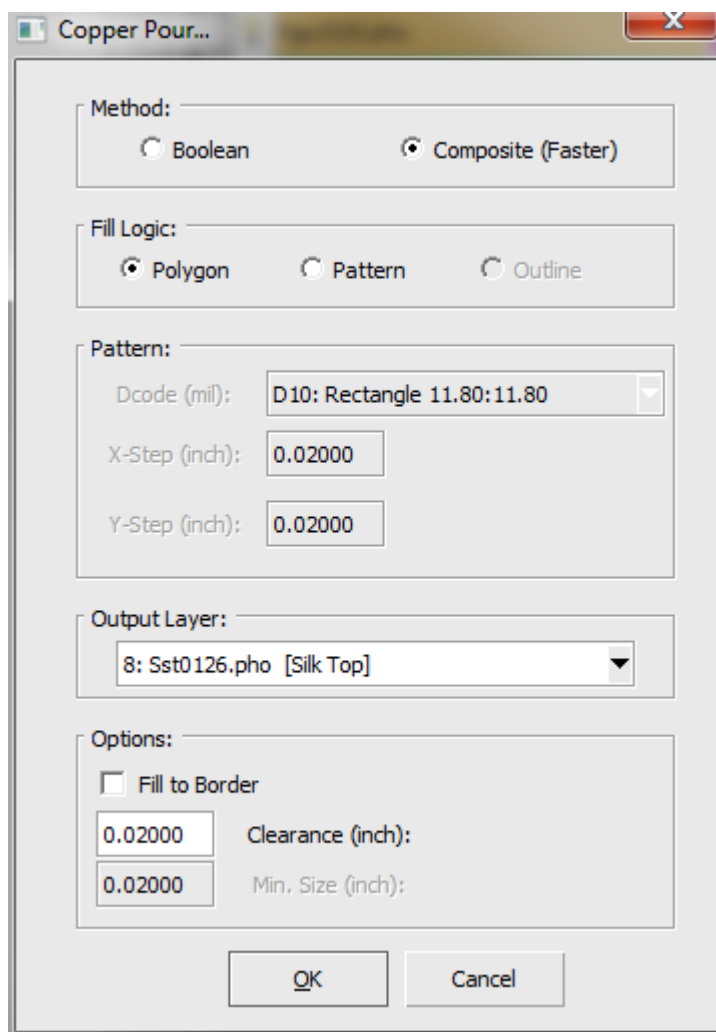


[Video: Add Barcode to PCB Silkscreen using FAB 3000](#)

[Video: Add Text to PCB Silkscreen before manufacturing using FAB 3000](#)

Fabrication > Copper Balancing

Copper pour utility for FAB 3000. To perform this, the active layer must be displayed before activating Copper Balancing. You should only turn on one layer.



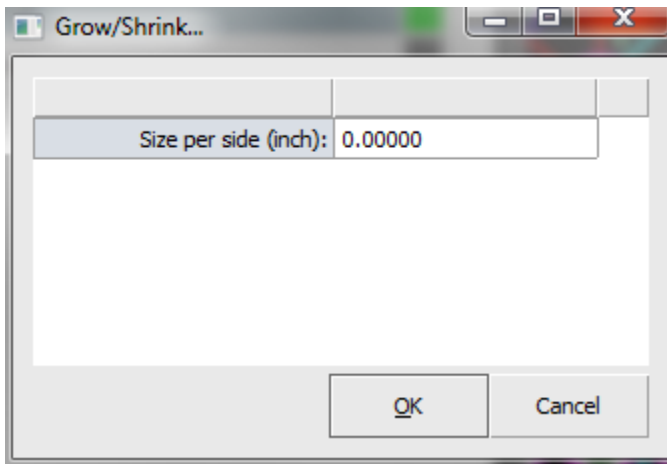
1. Select **Fabrication > Copper Balancing**.
2. Verify the Clearance. This is the distance the copper will keep between itself and other objects.
3. Verify the Minimum Size.
 - a. Make sure to select the appropriate Silkscreen & Mask layers. A common mistake occurs when a user selects a Top Silkscreen against a Bottom Mask.
4. Select **OK** and select the boundary to use for Copper Balancing. All copper will be placed on the active layer.



[Forum: Copper / Pour Balance](#)

Fabrication > Grow/Shrink

Enlarge or reduce object sizes. This feature is excellent for adding compensation to geometries that will be affected during manufacturing.

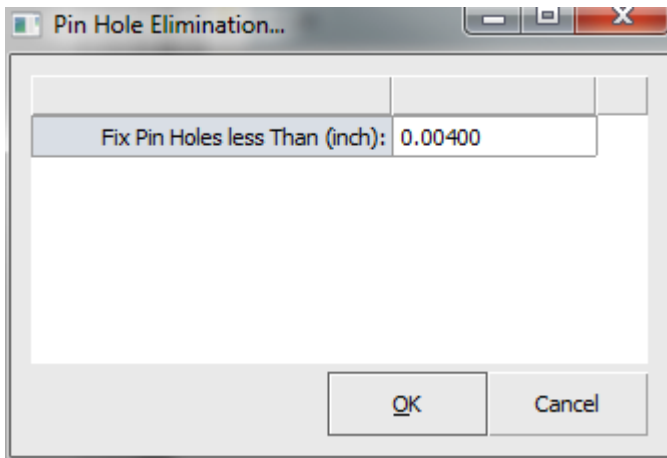


1. Select the object(s) to apply enlargement or reduction.
2. Select **Fabrication > Grow/Shrink**.
3. Verify or edit the compensation size under Size per side (inch).
4. FAB 3000 will now apply to all selected objects.

For traces & flashes, FAB 3000 will replace the existing Dcode with a larger or reduced Dcode size. For polygons, it will offset the vertices.

Fabrication > Pin Hole Elimination

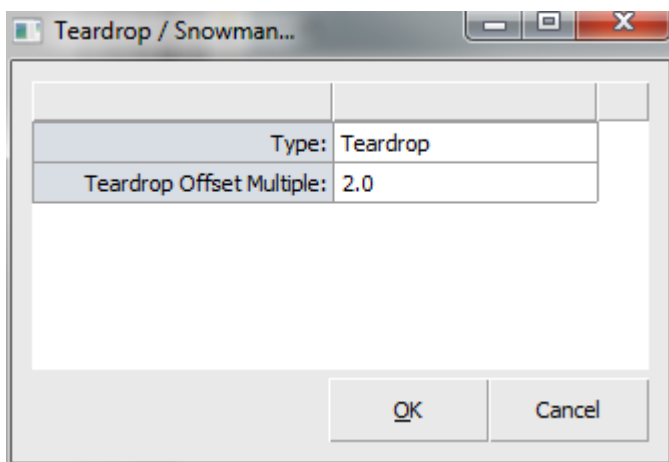
Increase yields by eliminating small openings and conductor-islands on metal layers (non-negative).



1. Select **Fabrication > Pin Hole Elimination**.
2. Enter a value for **Fix Pin Holes less Than (inch)**.
3. Select **OK**.
 - a. Negative plane layers are not processed.

Fabrication > Teardrop Creation

Creates a Teardrop/Snowman to improve Pad-Trace connections.



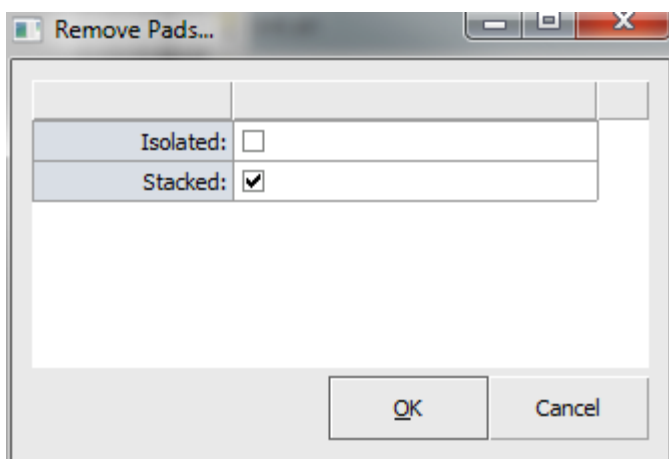
1. Select objects to be used for teardrop creation. Any pads selected will also be used.
2. Select **Fabrication > Teardrop Creation**. A Teardrop/Snowman dialog box will appear.
3. Select type from the dropdown menu. Choose from either Teardrop or Snowman.
 - a. Teardrop is a standard teardrop creation while snowman adds a small flash in the appropriate location.

4. Enter Teardrop Offset Multiple. This controls the length of the teardrop.

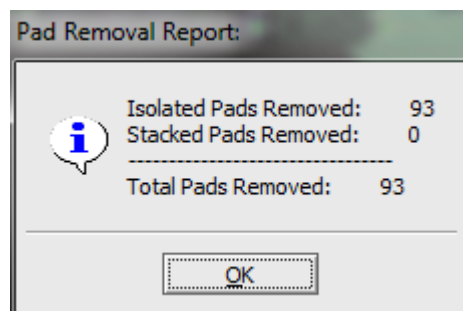
5. Select **OK** and the teardrops will be created.

Fabrication > Remove Pads (Isolated/Stacked)

Removes pads.



1. Select objects for Pad removal.
2. Select **Fabrication > Remove Pads (Isolated/Stacked)**.
3. Use the checkboxes to select Isolated or Stacked.
4. Select **OK**.
5. A Pad Removal Report dialog box will appear highlighting the number of pads removed.



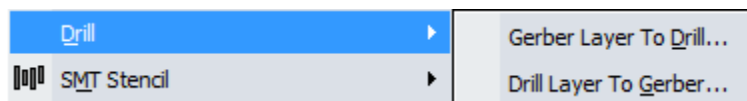
Fabrication > Remove Covered Data

Removes covered or overlapping data.

1. Select objects for removal.
2. Select **Fabrication > Remove Covered Data**.
3. Select **OK**.

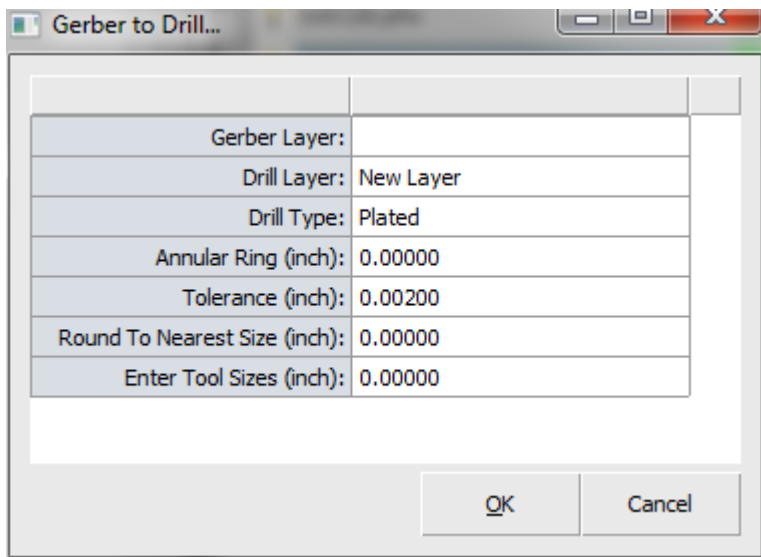
Fabrication > Drill

Conversion for Gerber and Drill Layers.



Fabrication > Drill > Gerber Layer to Drill

Converts Gerber flashed pads to Drill hits.

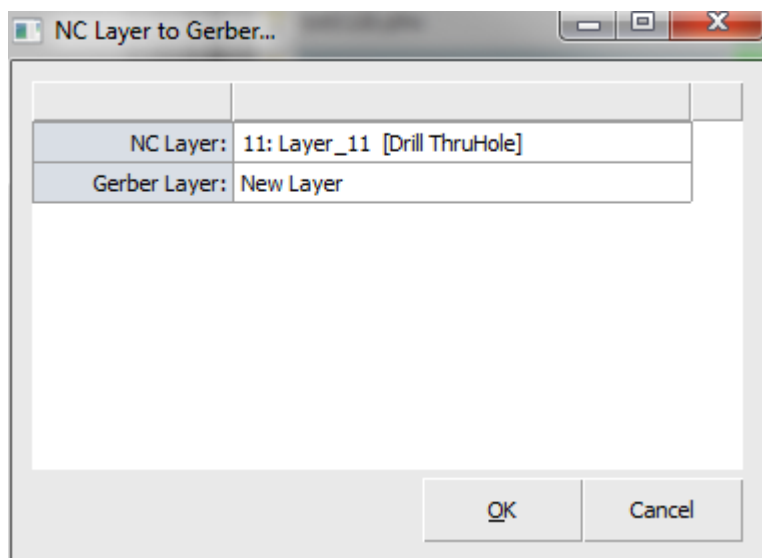


1. Select **Fabrication > Drill > Gerber Layer to Drill**. A Gerber to Drill dialog box will appear.
2. Verify and edit conversion parameters.
3. Select **OK** when finished.



[Forum: I don't have a Drill file, is there a way I can create one?](#)

[Video: Sort Layers, Create Drill from Gerber file using FAB 3000](#)



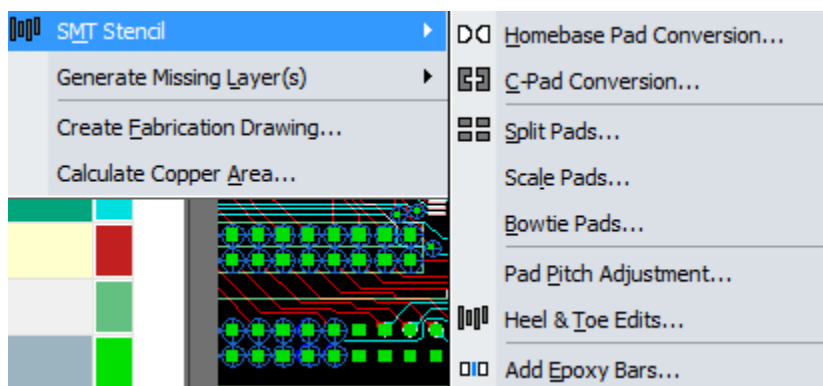
Fabrication > Drill > Drill to Gerber

Converts Drill hits to Gerber flashes.

1. Select **Fabrication > Drill > Drill to Gerber**. An NC to Gerber dialog box will appear.
2. Verify and edit conversion parameters.
3. Select **OK** when finished.

Fabrication > SMT Stencil

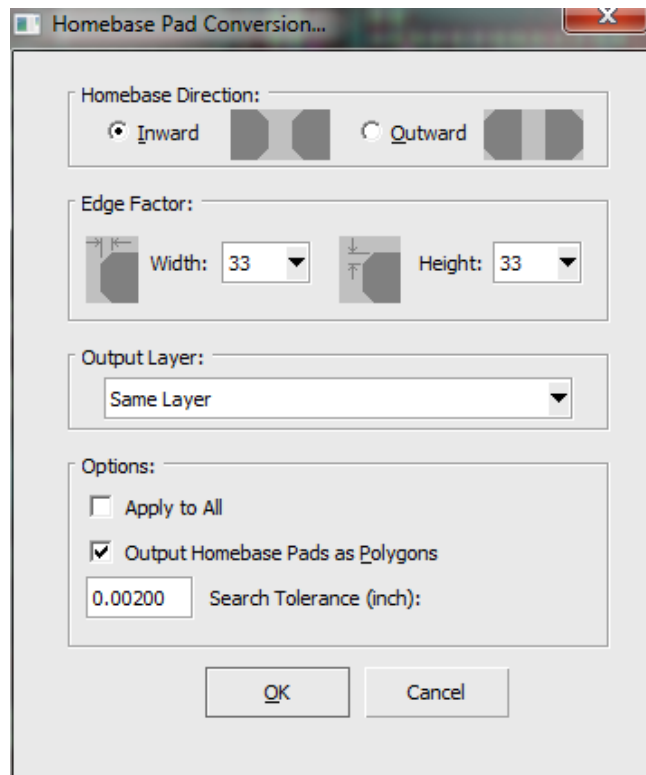
Apply SMT Stenciling for PCB solder based components.



[Video: Using Homebase Pads to Create Groups and then Modify the Custom Aperture](#)

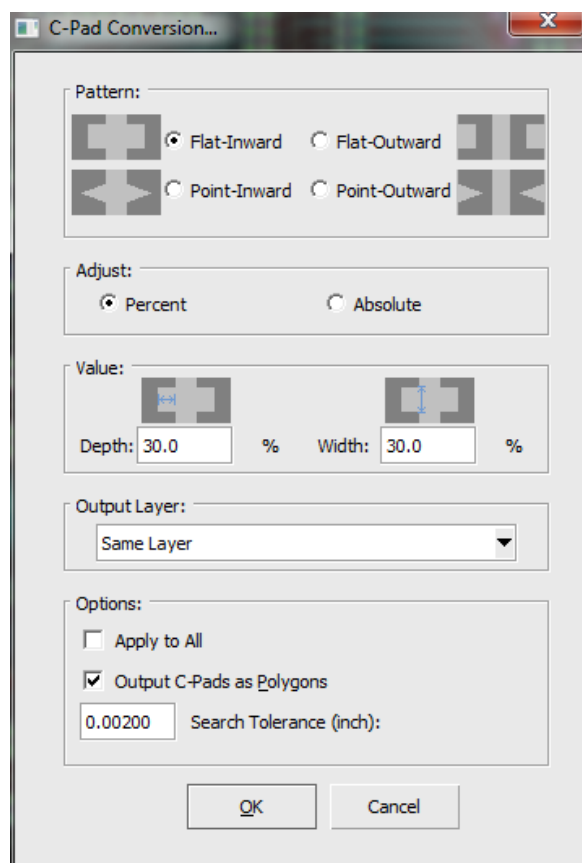
[Video: Modifying SMT Stencil Pads in Gerber Files](#)

Fabrication > SMT Stencil > Homebase Pad Conversion



Convert flashed pads to homebase pads.

1. Select **Fabrication > SMT Stencil > Homebase Pad Conversion**.
2. Verify or edit conversion options.
3. Select **OK**.



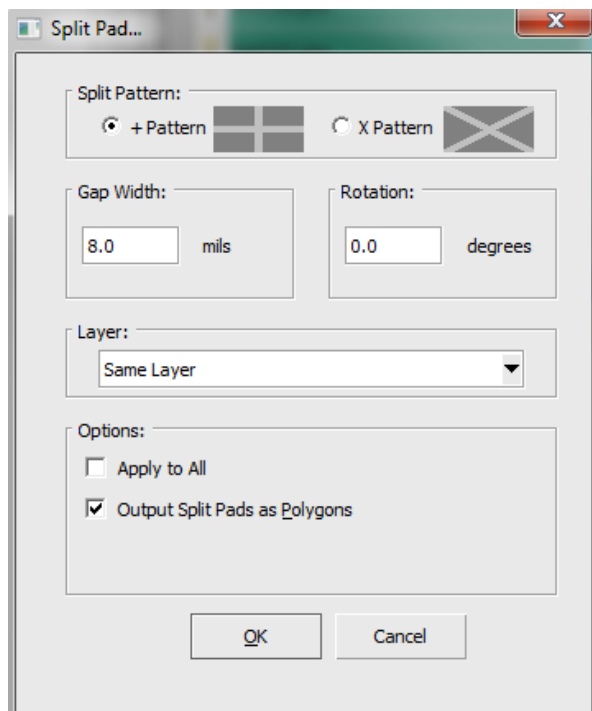
Fabrication > SMT Stencil > C-Pad Conversion

Convert flashed pads to C-Pads.

1. Select **Fabrication > SMT Stencil > C-Pad Conversion**.
2. Verify or edit conversion options.
3. Select **OK**.

Fabrication > SMT Stencil > Split Pads

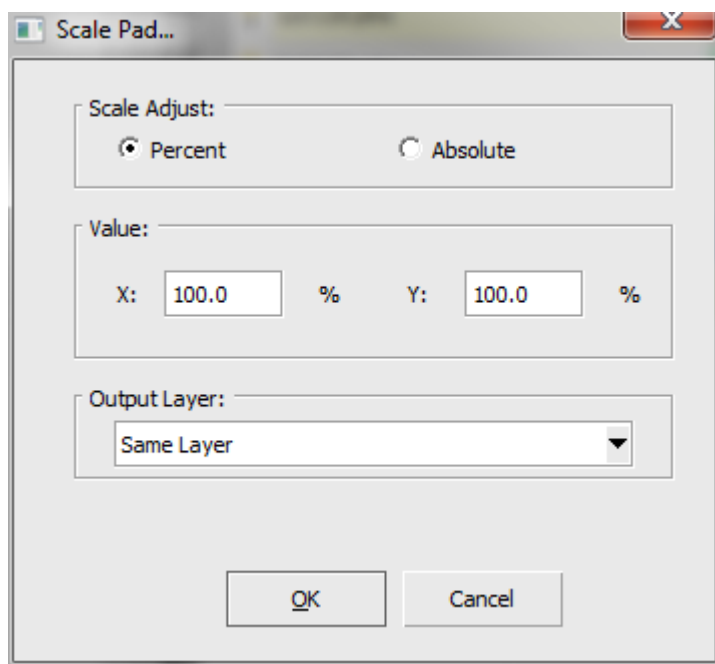
Convert flashed pads to C-Pads.



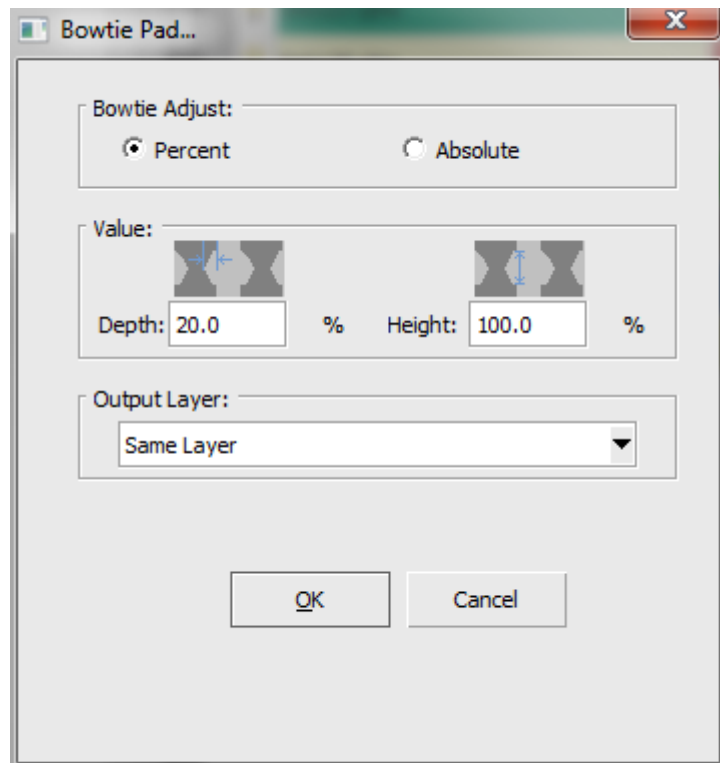
1. Select a pad. The pad must be a flash.
2. Select **Fabrication > SMT Stencil > Split Pads**.
3. The Split Pads dialog box will appear.
4. Edit and verify split pad details.
5. When finished, select **OK**.

Fabrication > SMT Stencil > Scale Pads

Converts flashed pads to scaled pads.



1. Select pads.
2. Select **Fabrication > SMT Stencil > Scale Pads**.
3. The Scale Pads dialog box will appear.
3. Edit and verify scale pad details.
4. When finished, select **OK**.

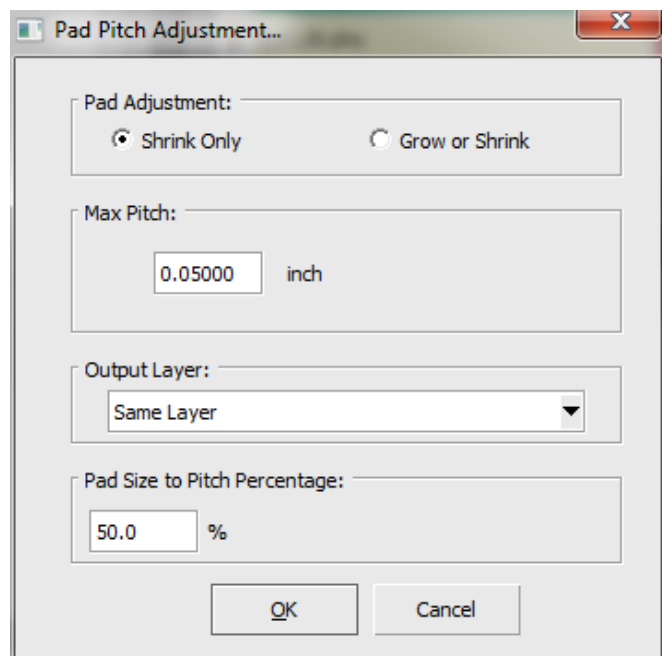
Fabrication > SMT Stencil > Bowtie Pads

Convert flashed pads to bowtie pads.

1. Select **Fabrication > SMT Stencil > Bowtie Pads**.
2. The Bowtie Pad dialog box will appear.
3. Edit and verify bowtie pad details.
4. When finished, select **OK**.

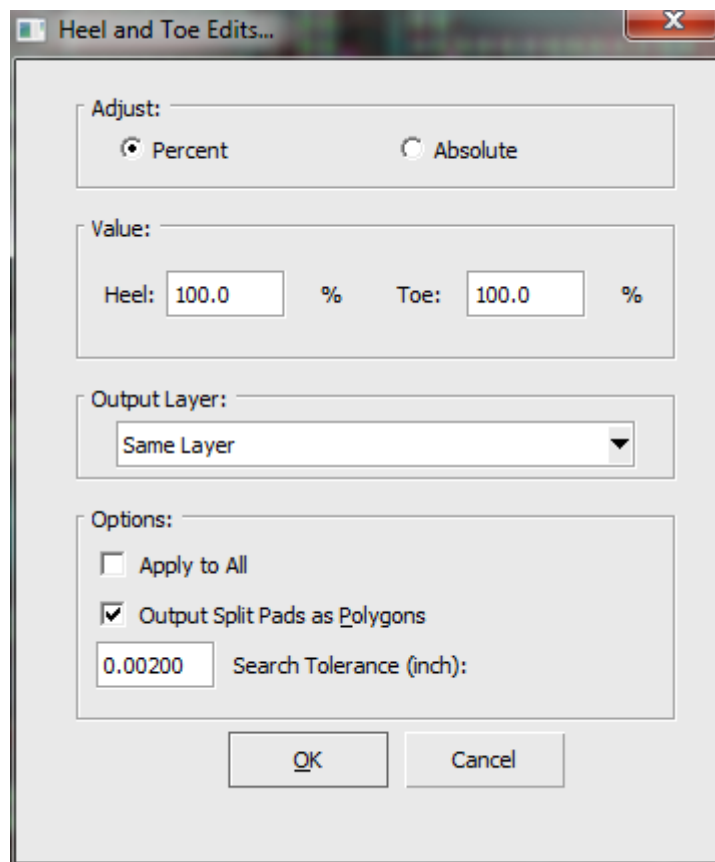
Fabrication > SMT Stencil > Pad Pitch Adjustment

Adjust pitch between flashed pads.



1. Select **Fabrication > SMT Stencil > Pad Pitch Adjustment**.
2. The Pad Pitch Adjustment dialog box will appear.
3. Edit and verify pad pitch adjustment details.
4. When finished, select **OK**.

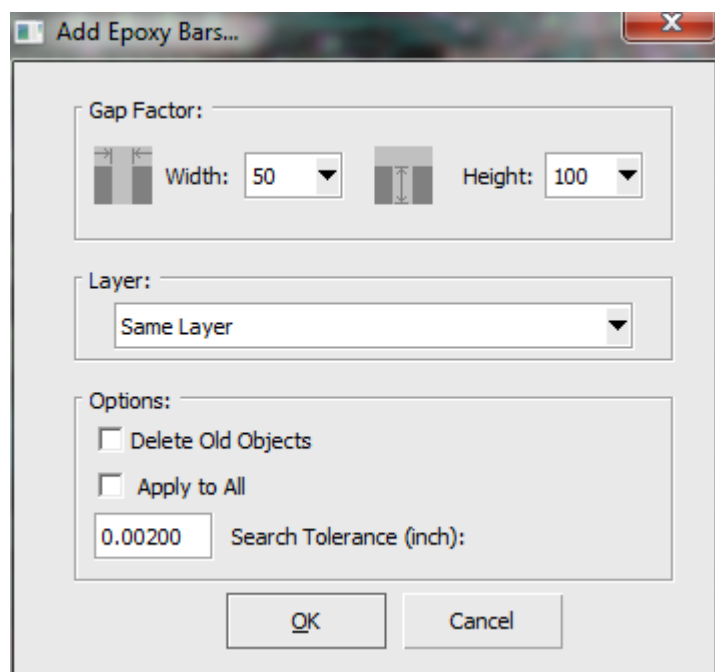
Fabrication > SMT Stencil > Heel and Toe Edits



Adjusts flashed pad lengths. Requires rectangular pads with orthogonal rotations (0, 90, 180, 270). Irregular pad rotations such as 34.2 degrees are not recommended.

1. Select a Pad Pair. A Pad pair must consist of two rectangular flashes.
2. Select **Fabrication > SMT Stencil > Heel and Toe Adjustment**.
3. The Heel and Toe Adjustment dialog box will appear.
4. Edit and verify Heel and Toe adjustment details.
5. When finished, select **OK**.

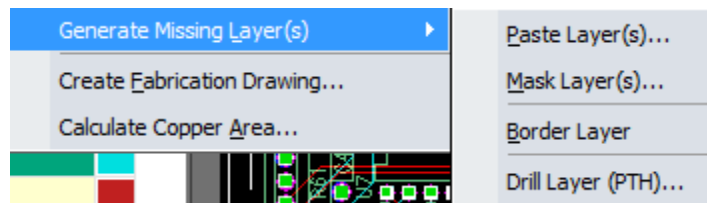
Fabrication > SMT Stencil > Add Epoxy Bars



Adds epoxy bars between selected pads.

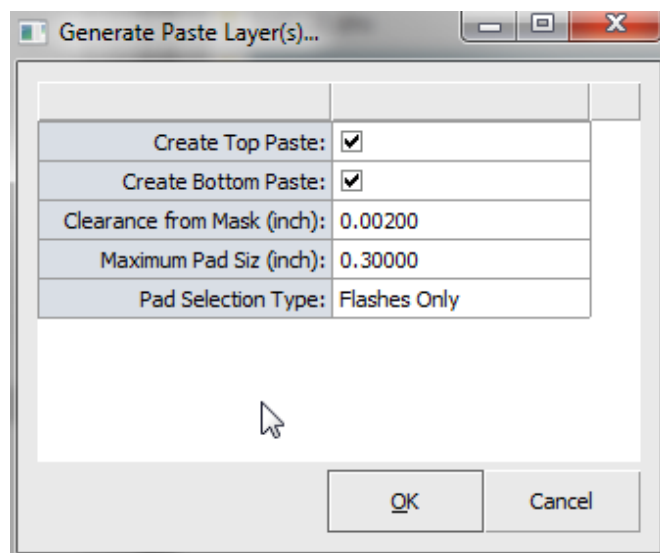
1. Select a Pad Pair. A Pad pair must consist of two rectangular flashes.
2. Select **Fabrication > SMT Stencil > Add Epoxy Bars**.
3. The Add Epoxy Bars dialog box will appear.
4. Edit and verify Add Epoxy Bar adjustment details.
5. When finished, select **OK**.

Fabrication > Generate Missing Layers



Generates missing layers using a variety of functions such as paste, mask, border, and Create Drill Layer (PTH).

Fabrication > Generate Missing Layers > Paste (Layers)

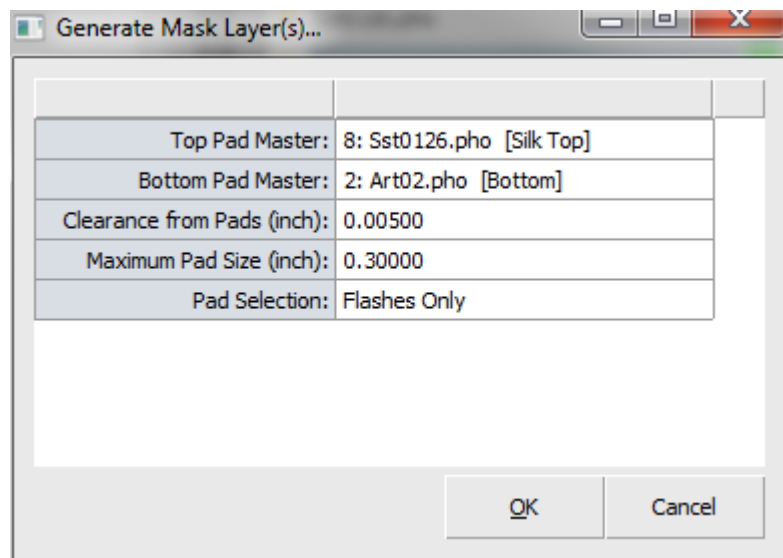


Creates paste layers from existing mask & drill layers.

1. Select **Fabrication > Generate Missing Layers > Paste (Layers)**.
2. Verify and edit paste settings in the dialog box.
3. Select **OK**.

Fabrication > Generate Missing Layers > Mask (Layers)

Creates mask layers from existing top and bottom layers.



1. Select layers for mask.
2. Select **Fabrication > Generate Missing Layers > Mask (Layers)**.
3. Verify and edit mask settings in the dialog box.
4. Select **OK**.

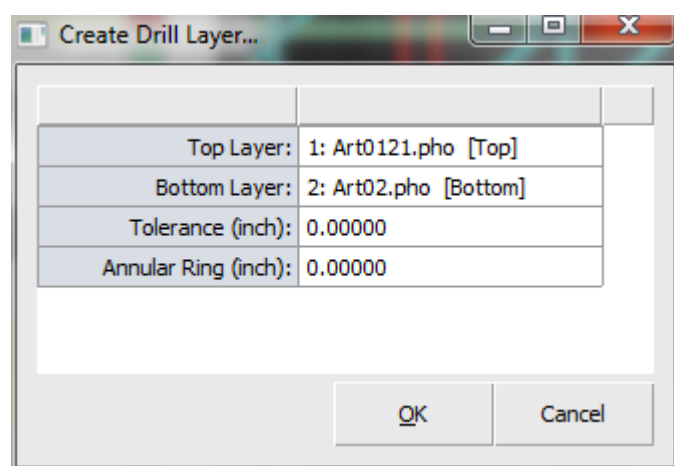
Fabrication > Generate Missing Layers > Border

Creates PCB border layers from existing data.

1. Select **Fabrication > Generate Missing Layers > Border**.
2. Select objects on PCB border.
3. Right click to select End or Close.

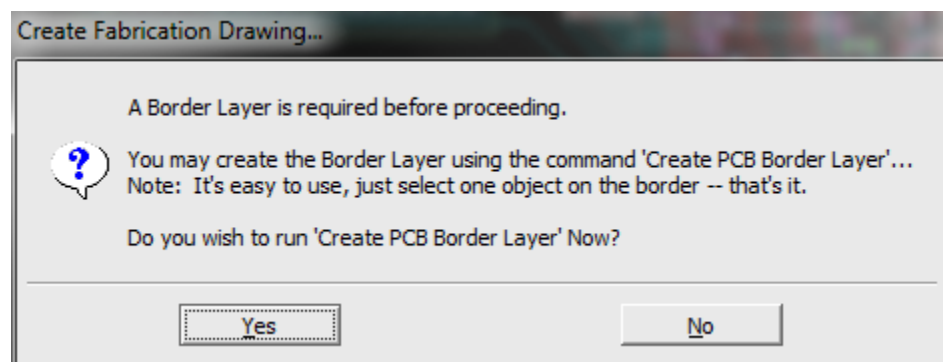
Fabrication > Generate Missing Layers > Create Drill Layer (PTH)

Builds a plated-thru-hole drill template when no drill layer is available.



1. Select **Fabrication > Generate Missing Layers > Create Drill Layer (PTH)**. If a drill layer has already been defined, a dialog box will ask you if you want to proceed anyway.
2. Verify and edit drill layer settings.
3. Select **OK**. A new layer will be created.

Fabrication > Create Fabrication Drawing



Creates a fabrication drawing from a drill layer and board outline layer.

- a. All layer types must be defined.
- b. At least one drill layer must be defined.
- c. A border layer must be defined.

Create Fabrication Drawing...

FAB3000 V7

NumericalInnovations.com

Fab Drawing Settings:

Create Fabrication Drawing easily creates the documentation required by PCB fabricators. Assign the required information for the drawing and include any notes. Click 'Next' to assign and review the Hole Schedule.

Select Drill Layer: L9: Drill.drl [Drill ThruHole]

Company: My Company Here
 Contact: My Name Here
 Part #:
 Revision: REV. A - INITIAL RELEASE
 Units: English

Notes:
 * FR4 Material
 * Board thickness is 0.062 +/- 10% inches.
 * Drills shown as finished sizes.

Options:
☒ Add Dimensions
☒ Add PCB Stack Up
☒ Add Layer Table

Cancel Back Next

1. Select **Fabrication > Create Fabrication Drawing**.
2. The Create Fabrication dialog box will appear.
3. Select a drill layer. For designs with more than one drill layer, repeat Create Fabrication Drawing and select the other drill file(s). Each time this command is run a new Fabrication Drawing is created.
4. Assign the required information for the drawing and include any necessary notes.

4. Select Next.

Create Fabrication Drawing...

FAB3000 V7

NumericalInnovations.com

Hole Schedule:

Review and modify the Hole Schedule. You may change Symbols, Notes, etc. If changes (other than symbols) are required, it is recommended that you make changes in the NC Tools Table first before proceeding.

Hole Schedule

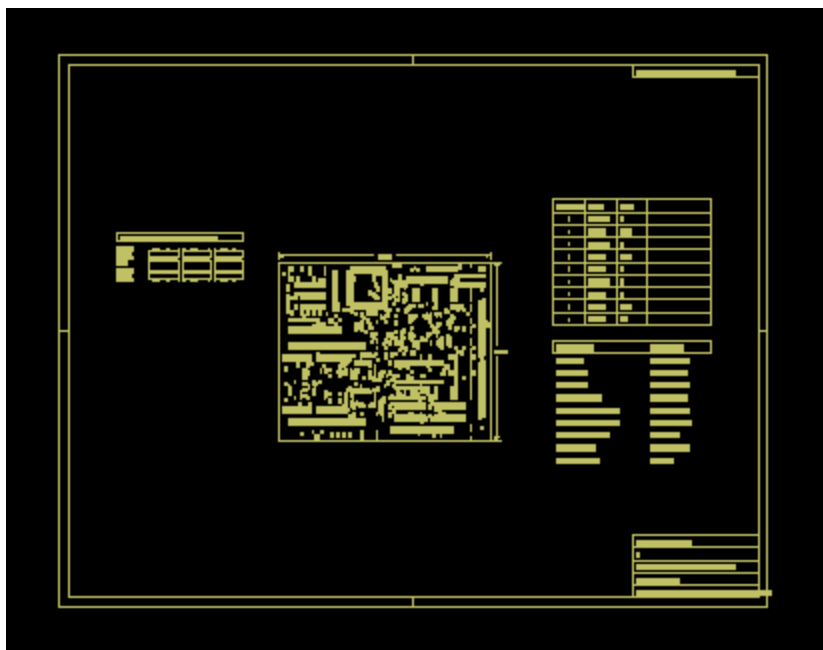
Symbol	Size	Qty	Plated	Tool #
A	110.00	4	<input checked="" type="checkbox"/>	1
B	40.00	284	<input checked="" type="checkbox"/>	2
C	120.00	4	<input checked="" type="checkbox"/>	3
D	50.00	603	<input checked="" type="checkbox"/>	4
E	90.00	2	<input checked="" type="checkbox"/>	5
F	100.00	1	<input checked="" type="checkbox"/>	6
G	70.00	1	<input checked="" type="checkbox"/>	7
H	23.00	433	<input checked="" type="checkbox"/>	8
I	45.00	56	<input checked="" type="checkbox"/>	9

Cancel Back Next

5. Review and modify the Hole Schedule. Change items such as symbols, text, notes, etc. for each drill hole.
6. If changes other than symbols are required, make these changes in the NC Tools Table first before proceeding.



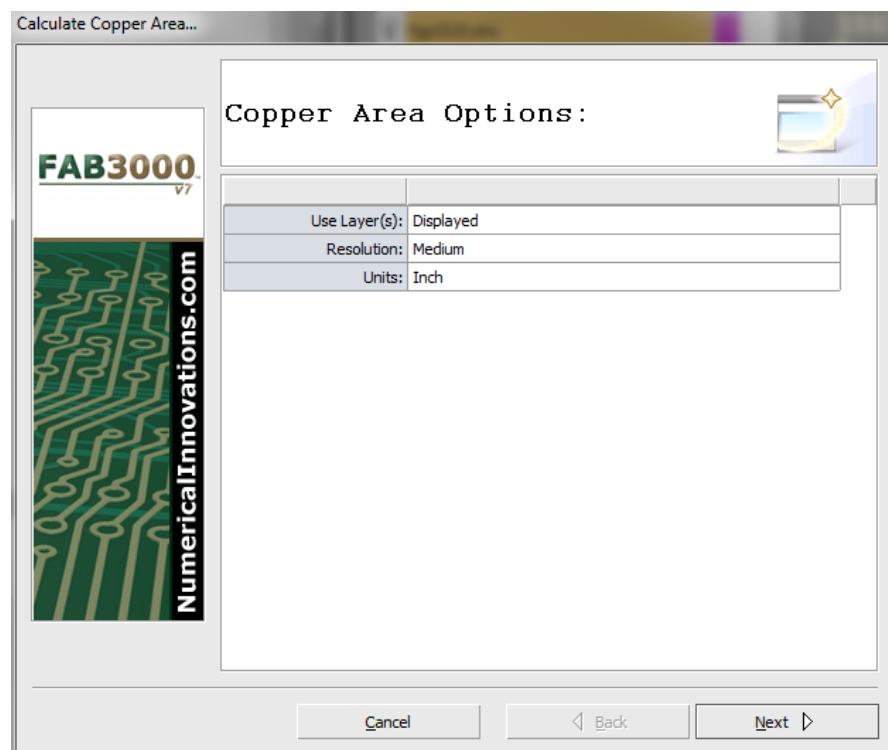
[Video: Create Drill Holes and Slot using FAB 3000](#)



7. Select **Next**. A fabrication drawing will be created.

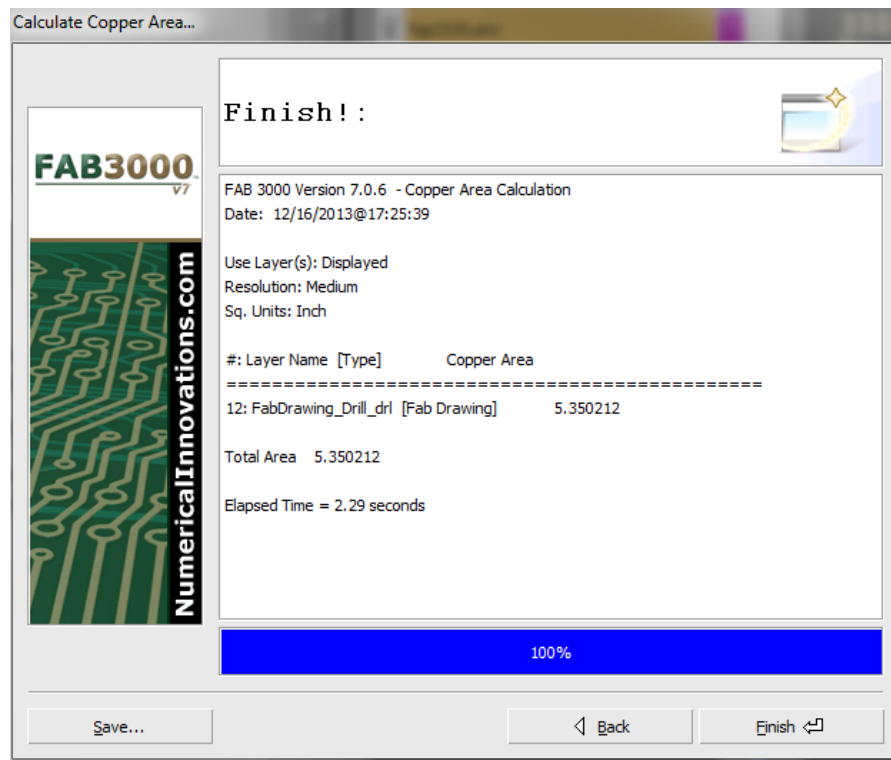
Fabrication > Copper Area Calculation

Calculates copper area per layer.



1. Select **Fabrication > Copper Area Calculation**.
2. Edit and verify area options.
3. Select **OK**. The copper area calculation will begin.

4. Review the copper area calculations. This information can also be saved by selecting **Save**.



5. Copper Area calculations will be highlighted within the Finish page.
6. Select **Finish** to close.



[Video: Perform Copper Area Calculation on Gerber PCB](#)

Fabrication > PCB Fabrication Report

Create detailed Sales/RFQ reports which include pricing, realistic finished photos, DFM and Netlist summaries, plus more. It's a perfect tool for any PCB fabrication sales team.

- a. All layer types must be defined.
- b. At least one drill layer must be defined.
- c. A border layer must be defined.

1. Select **Fabrication > PCB Fabrication Report**.

2. Review 'Select Configuration'. Configuration files are used to store pricing information based on number of layers, area of PCB, quantity, delivery, promotion factors, and more.

3. Enter Quantity and Delivery.

4. Review PCB Dimensions to

ensure they are correct. **Note:** Blank items such as Minimum Trace and Minimum Space will be automatically detected.

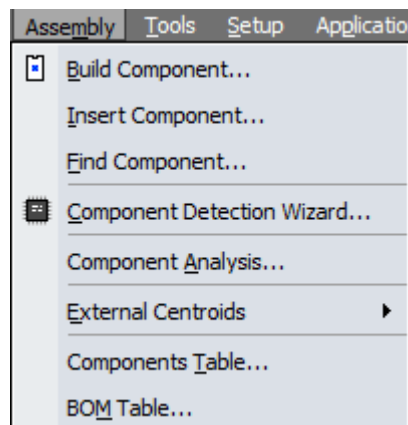
5. Review General Requirements.

6. Select tab 'Options'. You may verify Header options, and report features such as DFM Check, Netlist Check, Find Missing Layers, and more.

7. Enter notes to be included in report.

8. Press 'Next', and select file location to save PDF.

5.8 Assembly Menu

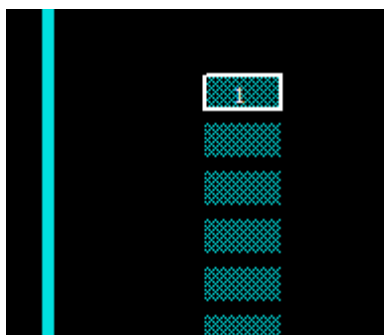


Use this menu to build, Insert, find, and analyze components.

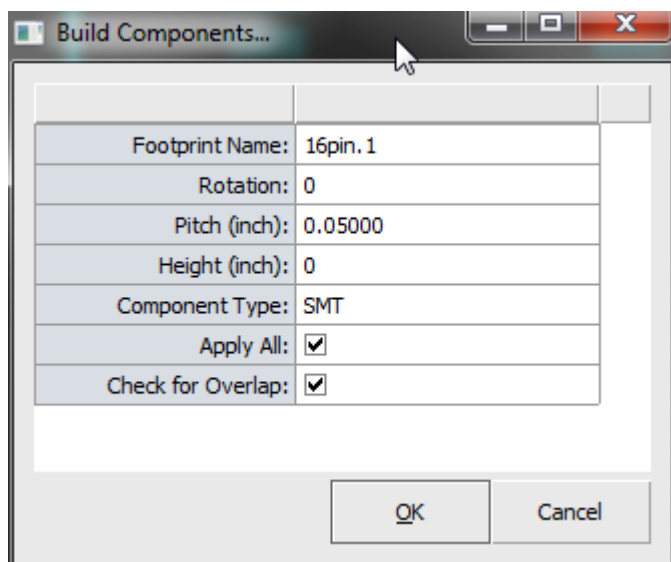
Assembly > Build Component

This adds components to the active Job.

1. Turn on paste or mask layer (paste preferred).
 - a. If paste layer is not defined you must instruct FAB 3000 by going to the **Setup > Layer Table**.
2. Locate a pad within the editor and select it.
 - a. FAB 3000 will try to guess Pin #1.
 - b. You may also select a different pad for Pin #1.
3. **Select Assembly > Build Component**
4. Select Pin One.

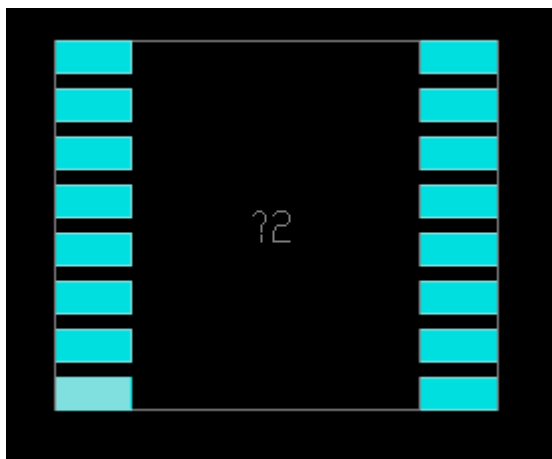


- a. Usually pin one is the top-leftmost pin for top side (assuming no rotation) and top right most for bottom side (assuming no rotation). Pin #1 is needed as it helps orient the placement of the component.
- b. Once Pin #1 is chosen, right click to accept.



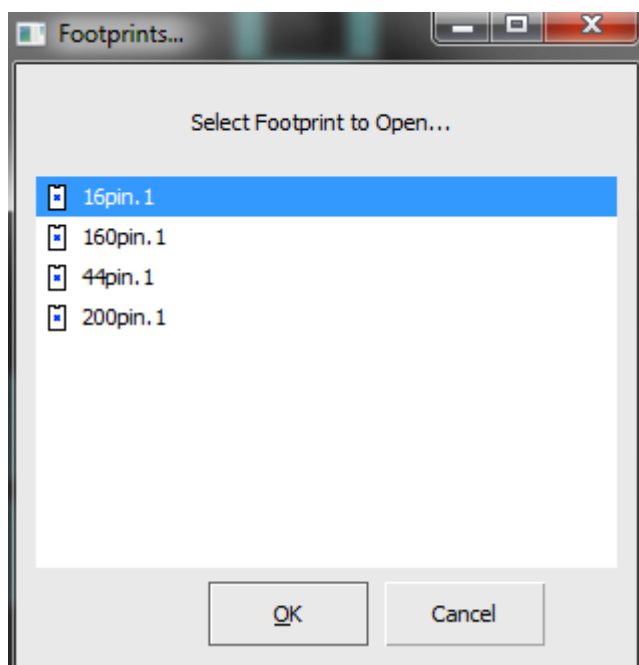
5. Assign component description. A **Build Components** dialog box will appear. Select OK when done.

- a. Enter component rotation as it appears on the board. If you have selected a component that is rotated 90 degrees on the board, than make sure to enter that here.
- b. If apply all is check (recommended), FAB 3000 will go out and seek all similar footprints.



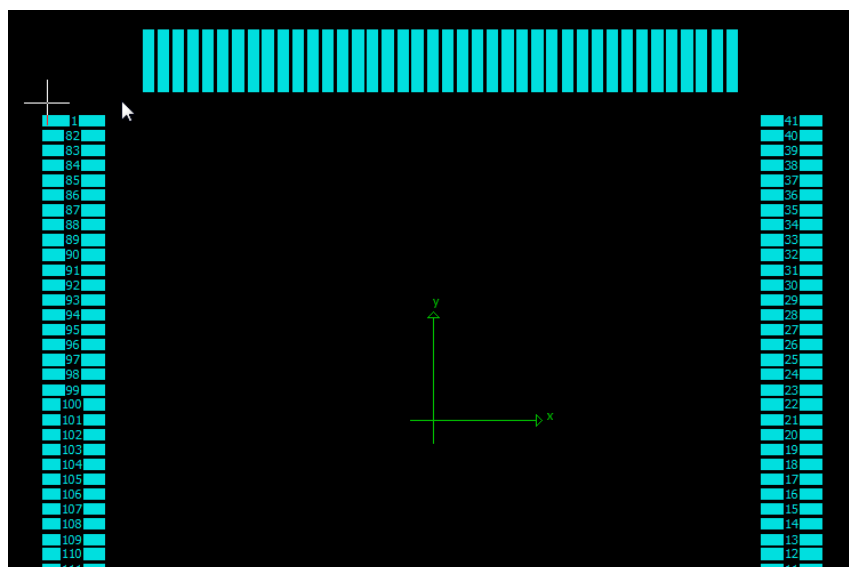
You will notice (if apply all is checked) that all similar footprints have been detected and assigned a generic reference designator value (for example "?21").

6. Repeat steps 2 thru 5, until all footprints on the paste layer have been completed.



7. Review the Footprint browser and verify remaining components. Select **Window > Footprint Browser**.

8. Select the Footprint to open by clicking it. Select **OK**.



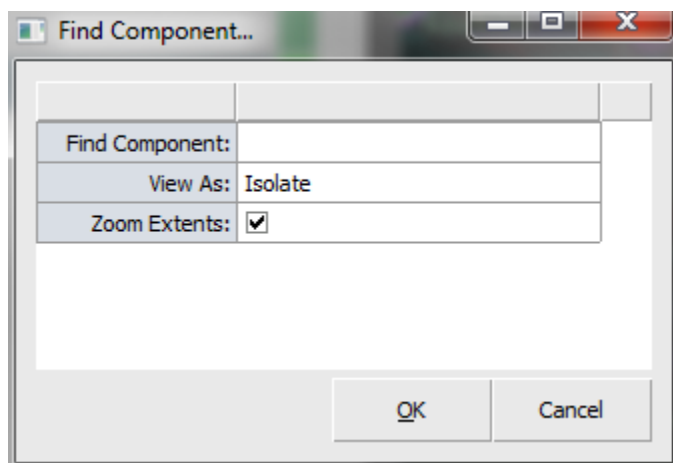
8. In the Footprint browser you may load external libraries (containing footprints) or make modifications here.

Assembly > Insert Component

Inserts a single component into an existing Job.

Assembly > Find Component

Searches the Job for defined components.

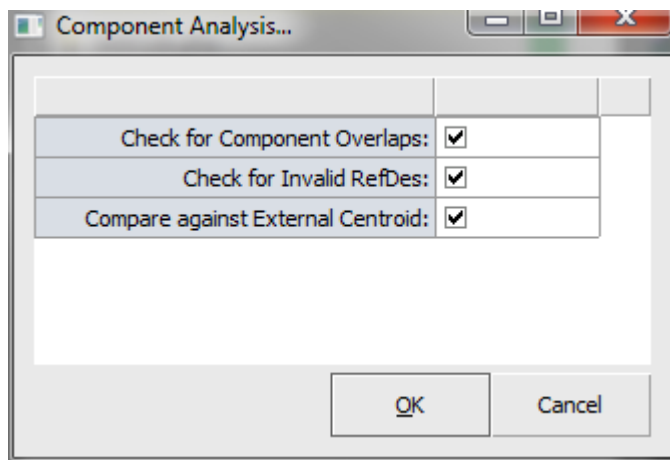


Assembly > Component Detection Wizard

Detects components automatically. Speeds up the process of building components.

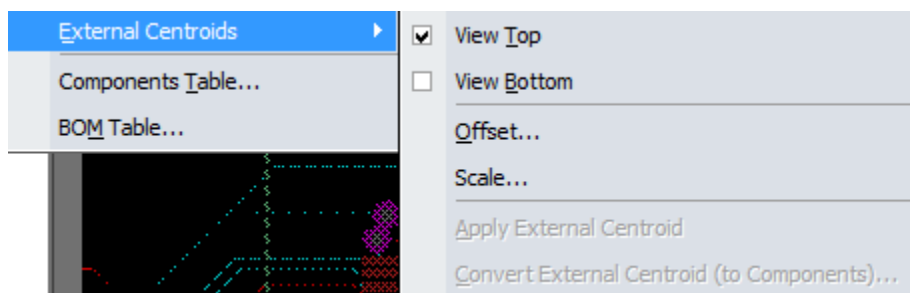
Assembly > Component Analysis

Detects components automatically. Speeds up the process of building components.



Assembly > External Centroids

Apply external centroid attributes to existing components.

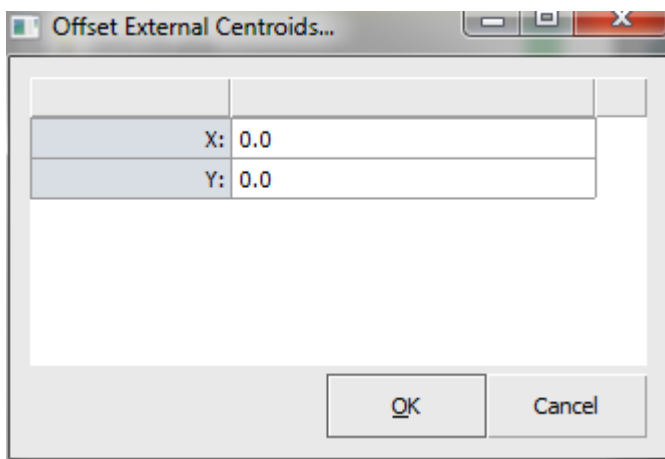


Assembly > External Centroids > View Top

Top view centroid.

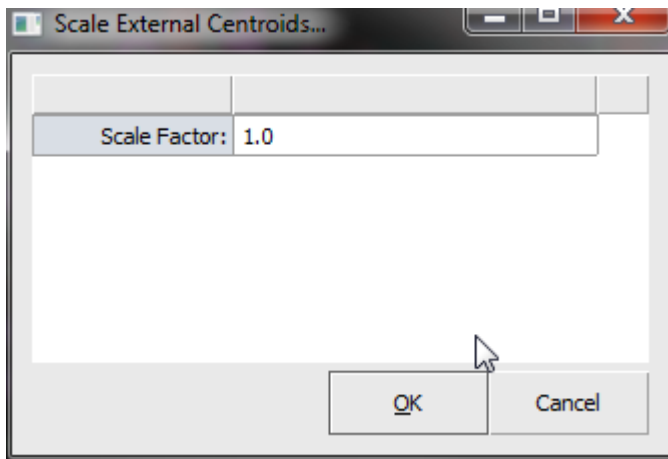
Assembly > External Centroids > View Bottom

Bottom view centroid.



Assembly > External Centroids > Offset

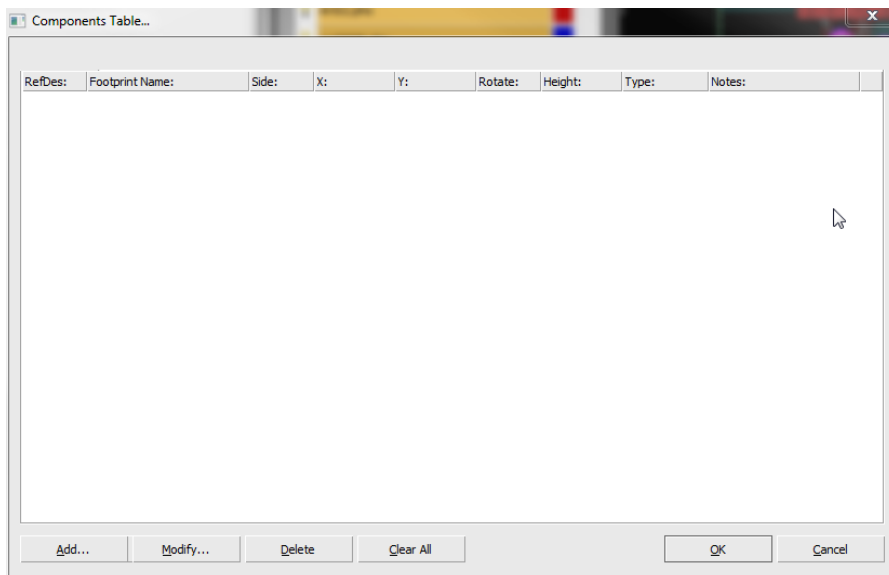
Offset external centroid by entering X and Y values.

Assembly > External Centroids > Scale

Scales external centroids.

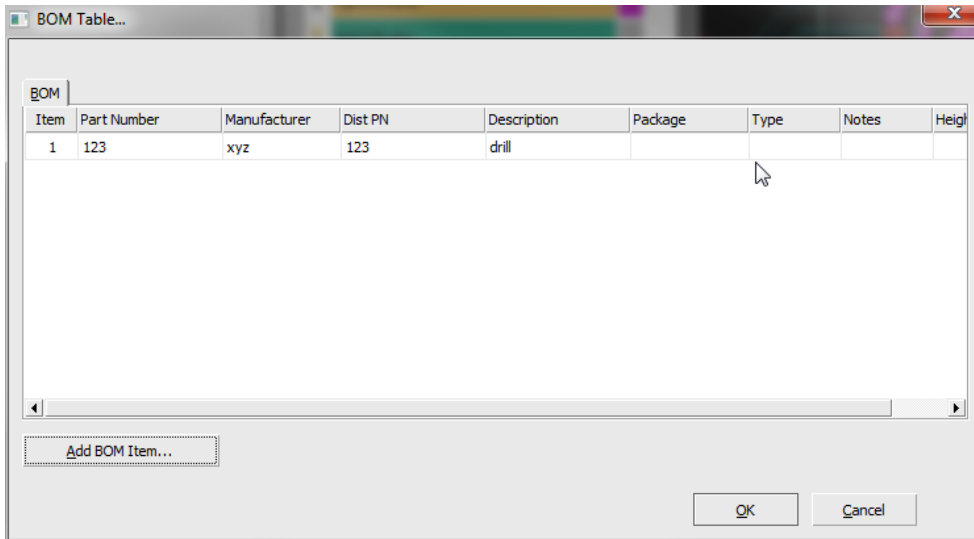
Assembly > Components Table

Display, add, remove, or modify components used in active Job/panel.



Assembly > BOM Table

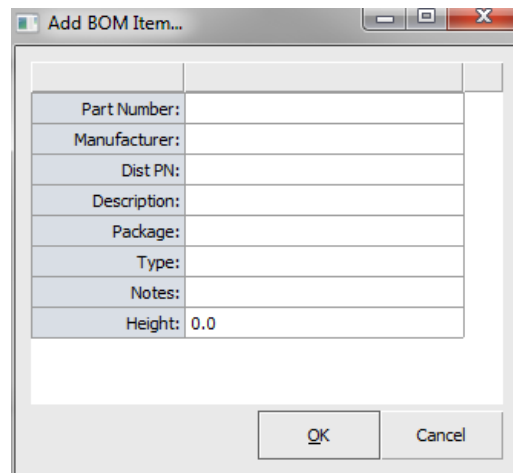
Displays Bill of Materials for active Job/panel.



The screenshot shows a window titled "BOM Table...". Inside, there is a table with the following columns: Item, Part Number, Manufacturer, Dist PN, Description, Package, Type, Notes, and Height. The first row contains the values: 1, 123, xyz, 123, drill. Below the table is a button labeled "Add BOM Item...". At the bottom right of the window are "OK" and "Cancel" buttons.

Item	Part Number	Manufacturer	Dist PN	Description	Package	Type	Notes	Height
1	123	xyz	123	drill				

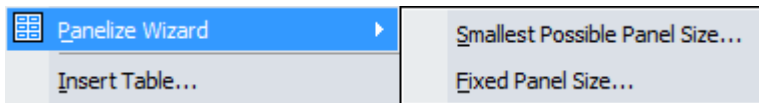
1. To add a BOM item, select **Add BOM Item**.
2. Enter BOM details and select **OK**.
3. The item will be listed in the BOM Table.



The screenshot shows a dialog box titled "Add BOM Item...". It contains several input fields for entering BOM details: Part Number, Manufacturer, Dist PN, Description, Package, Type, Notes, and Height. The Height field is pre-filled with the value 0.0. At the bottom right of the dialog are "OK" and "Cancel" buttons.

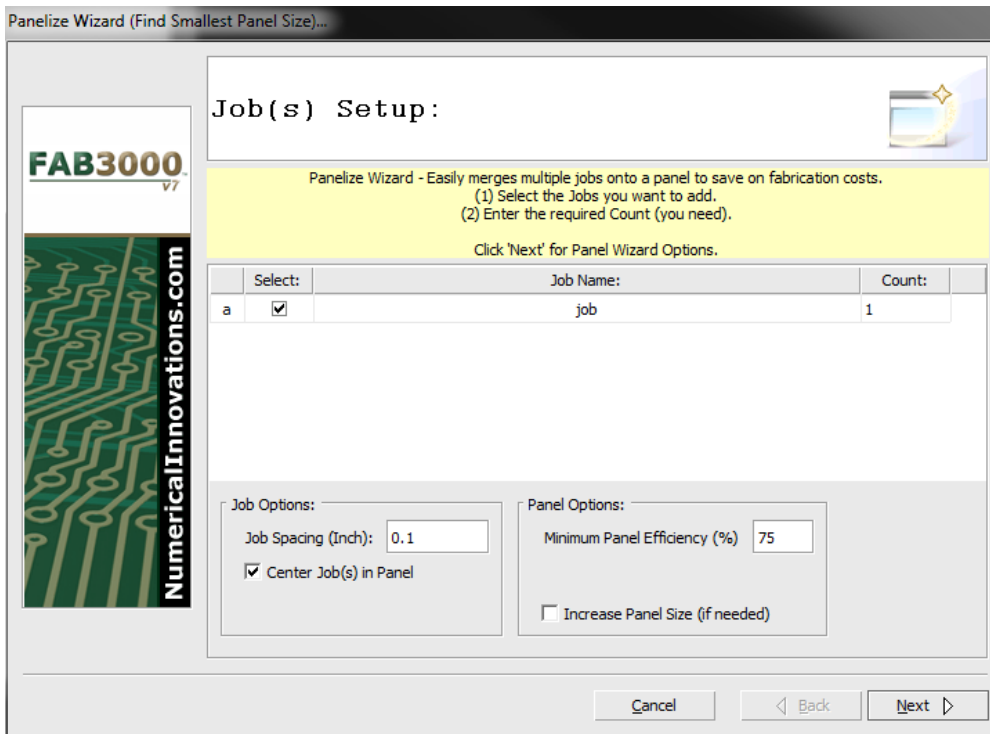
5.9 Tools Menu

Tools > Panelize Wizard



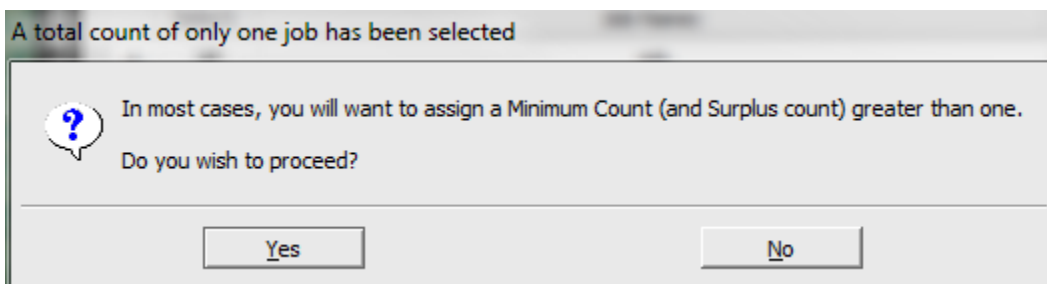
In order to activate the Panelize Wizard, you must first create a New Panel by selecting **File > New > Panel**. Once complete, the Panelize Wizard function will appear under **Tools > Panelize Wizard**. Panelize Wizard merges multiple Jobs onto a Panel to save on fabrication costs.

Tools > Panelize Wizard > Smallest Possible Panel Size

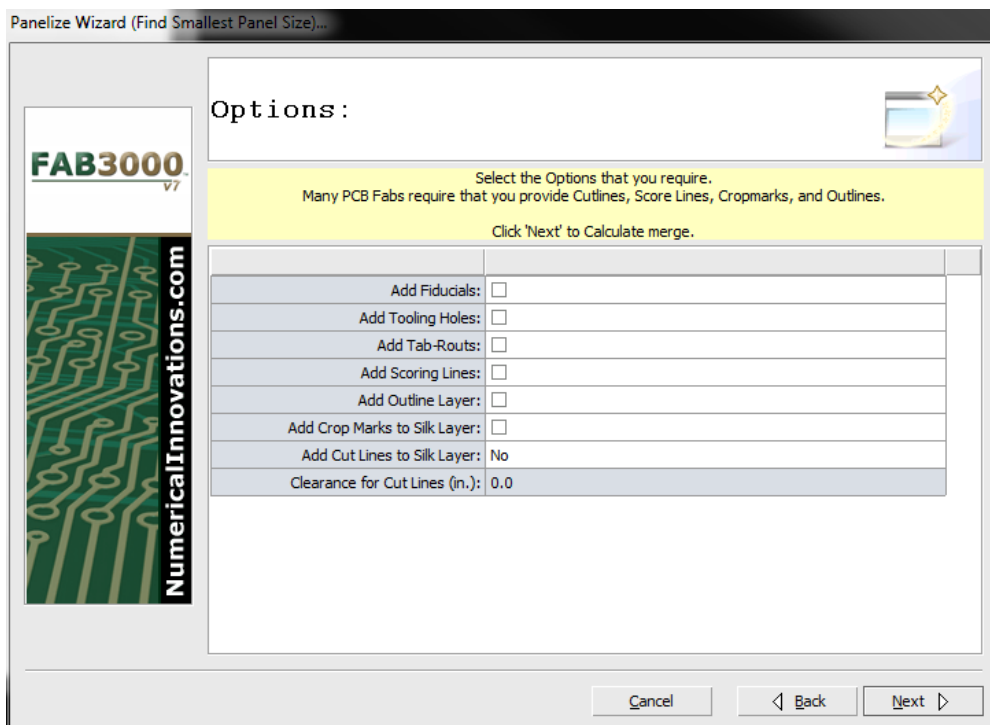


1. Select **Tools > Panelize Wizard > Smallest Possible Panel Size**.
2. In the list, select the Jobs you want to add to the Panel.
3. Enter the required count in the Count column.
4. Edit or verify Job and Panel Options.
5. Select **Next**.

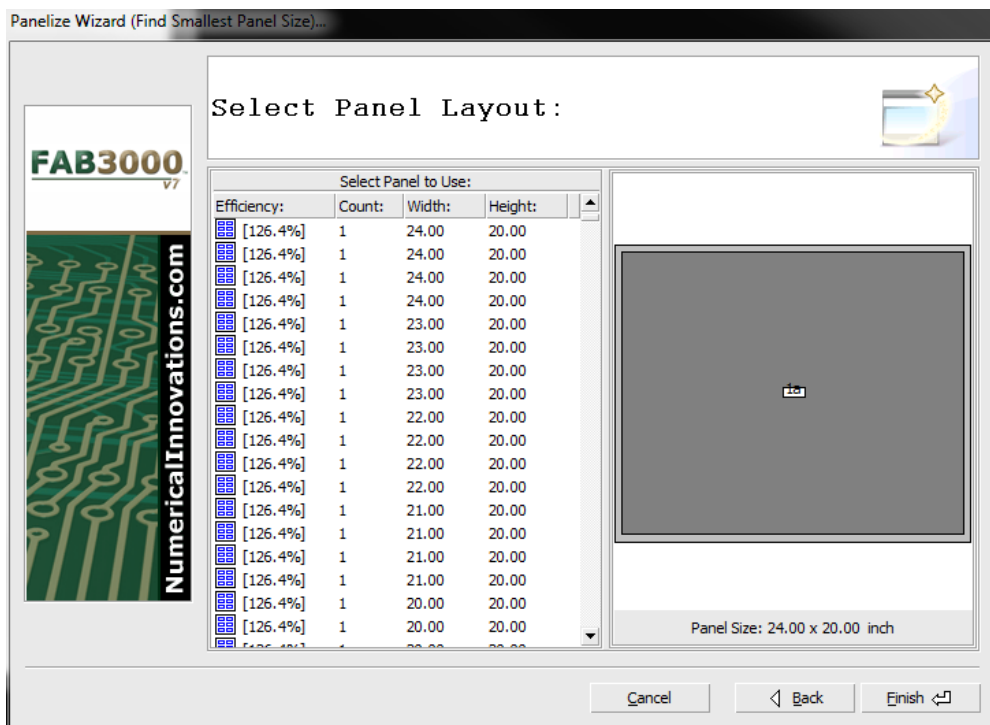
If only one job is selected, a warning will appear with the following message:



6. Select **Yes** to proceed or **No** to cancel.





7. The Panelize Wizard Options dialog box will appear.
8. Verify or edit the options you require.
9. Select **Next** to calculate the merge.



10. The Select Panel Layout dialog box will appear.
11. Select a panel layout.
12. Select **Finish**.

Tools > Panelize Wizard > (Use Fixed Panel Size)

Panelize Wizard (Use Fixed Panel Size)...

Job(s) Setup:

Panelize Wizard - Easily merges multiple jobs onto a panel to save on fabrication costs.
 (1) Select the Jobs you want to add into the panel. (2) Enter the Minimum Count you require.
 (3) Enter the Surplus Count (search any empty spaces to add extra jobs)

Click 'Next' for Panel Wizard Options.

Select:	Job Name:	Min. Count:	Surplus:
a <input checked="" type="checkbox"/>	job	1	0

Job Options:

Job Spacing (Inch):


☒ Center Job(s) in Panel

Panel Options:

1. Select **Tools > Panelize Wizard > (Use Fixed Panel Size)**
2. In the list, select the Jobs you want to add to the Panel.
3. Enter the required count in the Count column.
4. Edit or verify Job and Panel Options.
5. Select **Next**.

If only one job is selected, a warning will appear with the following message:

A total count of only one job has been selected



In most cases, you will want to assign a Minimum Count (and Surplus count) greater than one.
Do you wish to proceed?

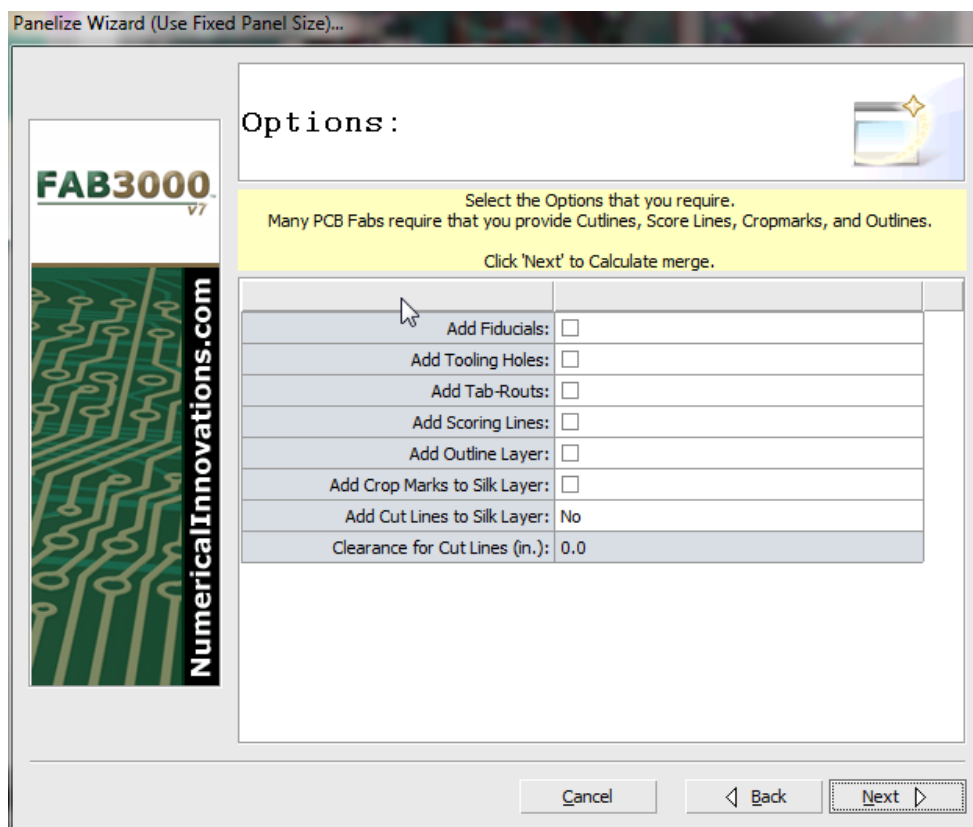
6. Select **Yes** to proceed or **No** to cancel.



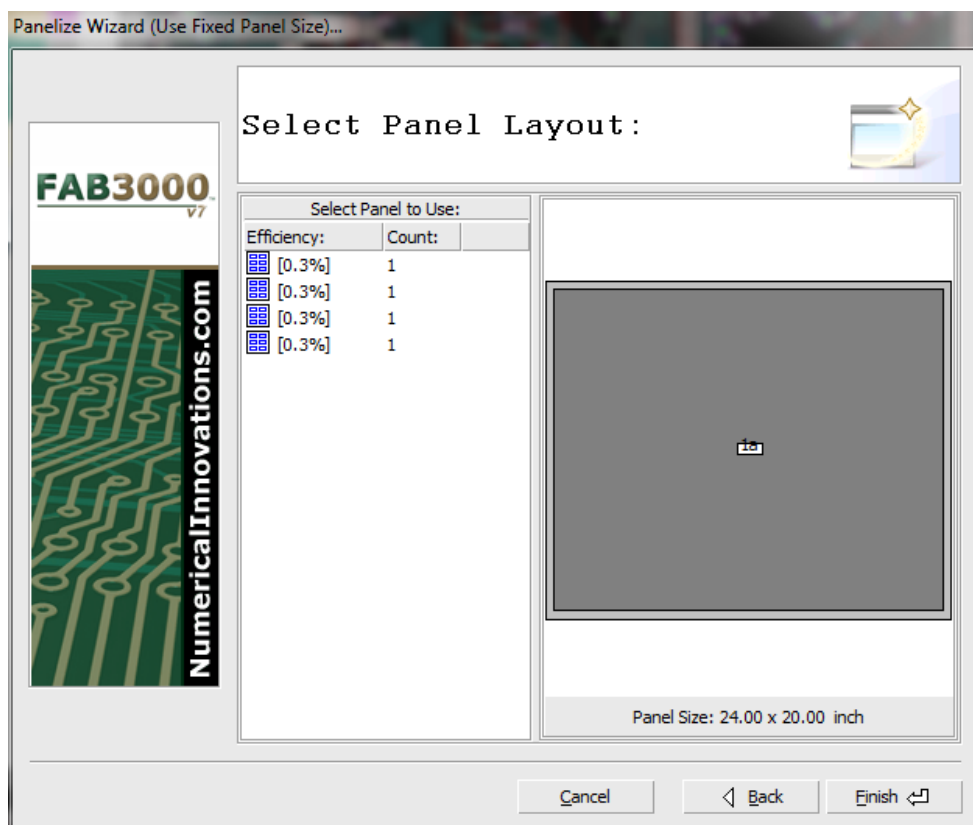
[Forum: FAB 3000 Panelize Feature](#)

[Video: Automatically Panelize PCB Gerbers](#)

[Video: Panelize Gerber Files from Eagle PCB using FAB 3000](#)



7. The Panelize Wizard Options dialog box will appear.
8. Verify or edit the options you require.
9. Select **Next** to calculate the merge.



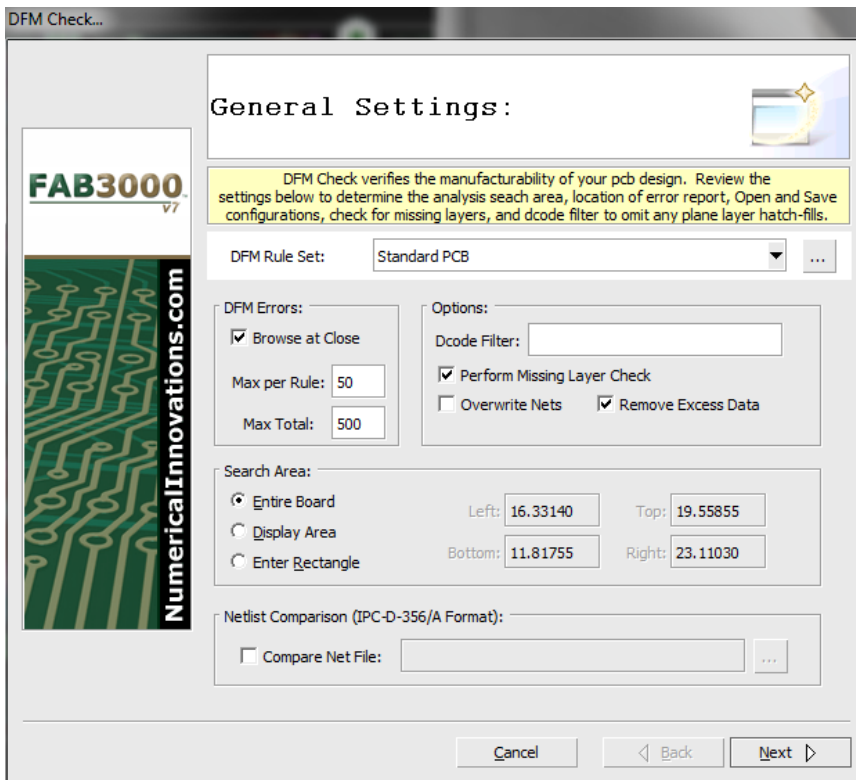
10. The Select Panel Layout dialog box will appear.
11. Select a panel layout.
12. Select **Finish**.

Tools > DFM Check

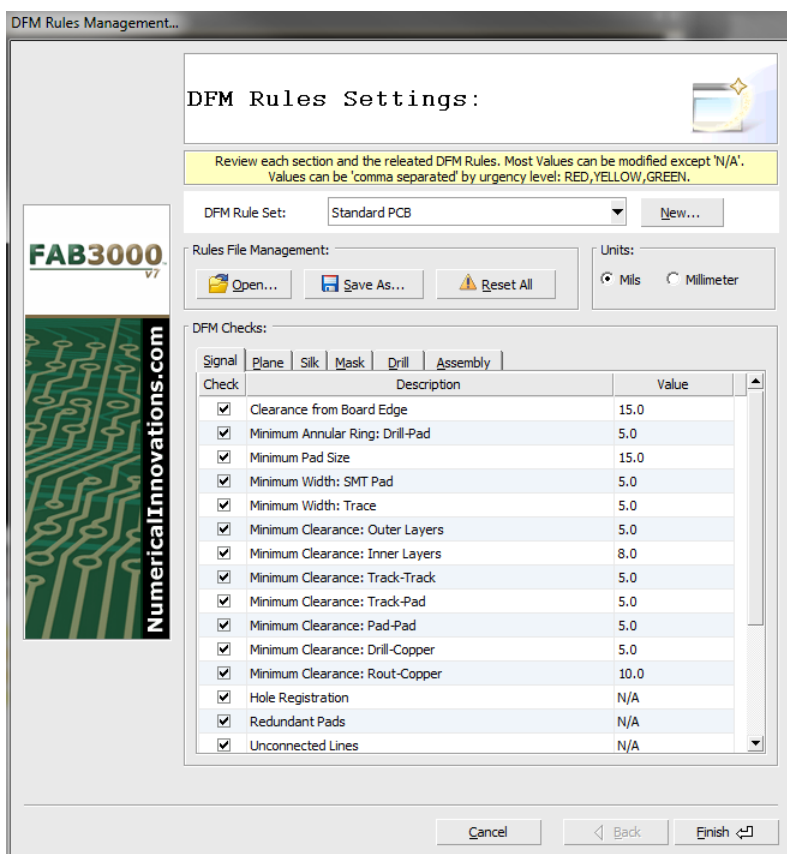


DFM Check finds all DRC/DFM violations. In order to use this feature, you must have imported Gerber and drill file(s) into the workspace. The following is required:

- a. You must extract a Netlist before proceeding. If you haven't run a Netlist extraction, DFM Checker will perform it for you.
- b. All layers must be defined. To define layers, use **Setup > Layer Table**.
- c. You must have at least one drill file (2 or more for blind/buried Vias).



1. Select **Tools > DFM > DFM Check**. The DFM Check dialog box will appear.
2. Verify and edit the DFM rules set. To do this, select the available rules set from the pull down menu under DFM Rule Set.
 - a. You may also edit the DFM Rules by pressing the "..." button next to the DFM Rule Set drop down menu (See Next Page, **DFM Rules Settings**).

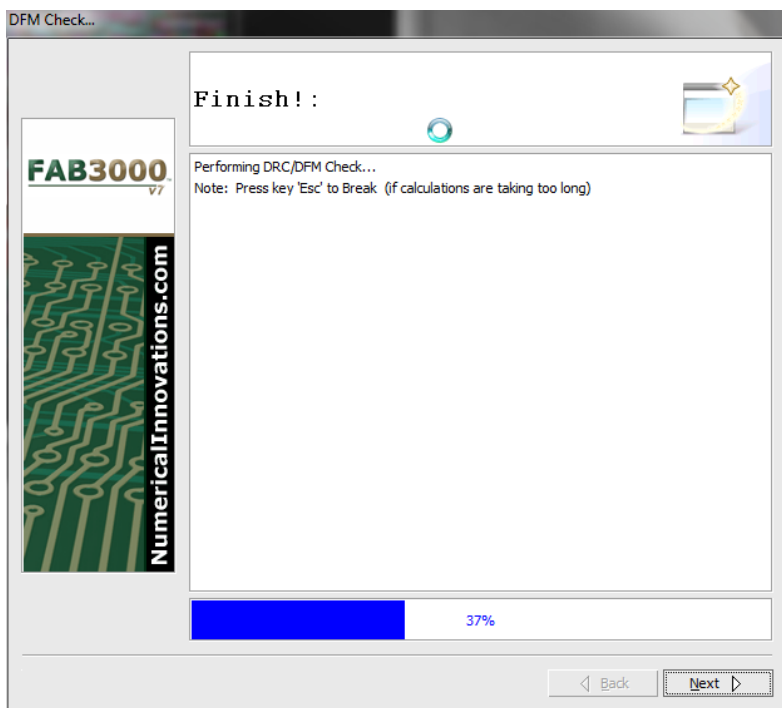


- b. **DFM Rules Settings** dialog is used to verify or edit rules individually.

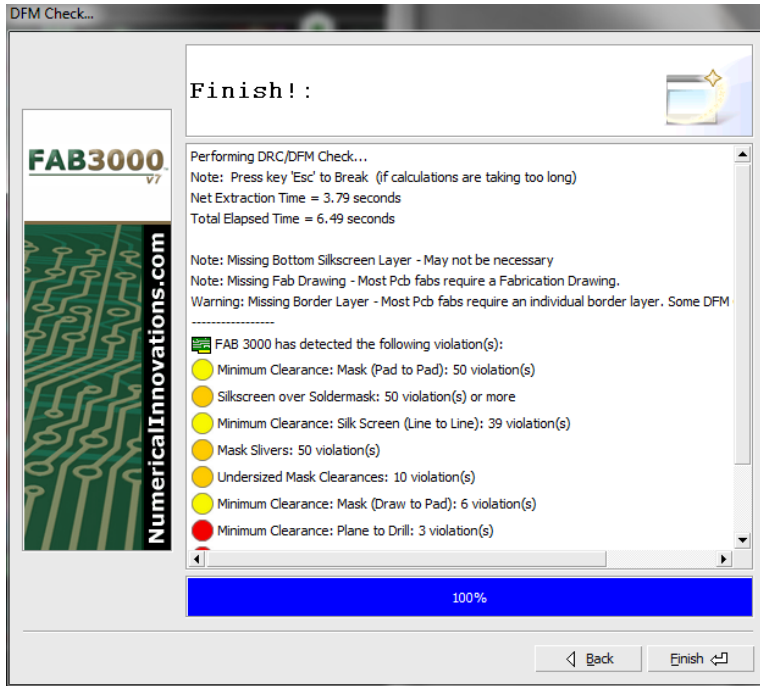
3. Select **Next**.

4. FAB 3000 will now perform an extensive analysis of your PCB design.

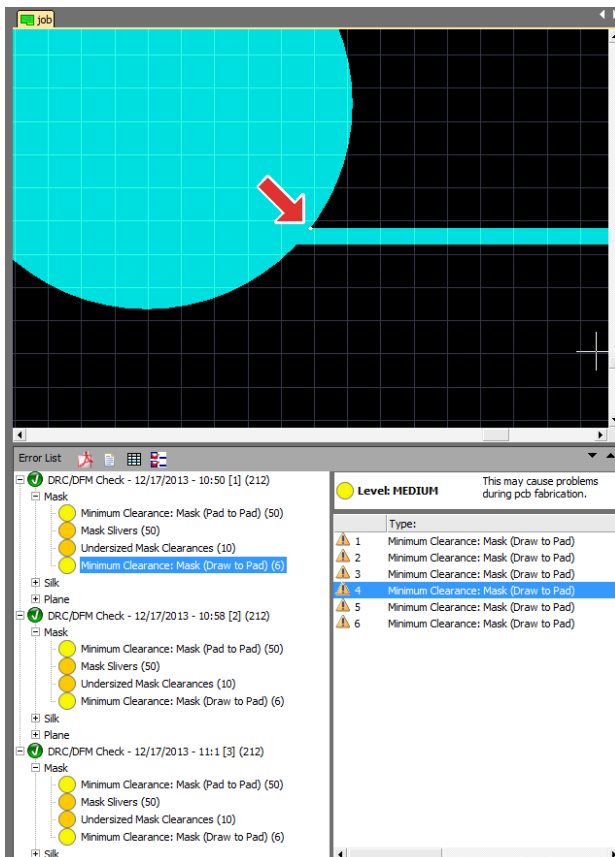
- a. Job Inserts are ignored. If you wish to run a DRC/DFM check on a specific child Job, make sure to open the child Job in a separate window, and then proceed. To open Jobs, go to **Window > Workspace Browser**.



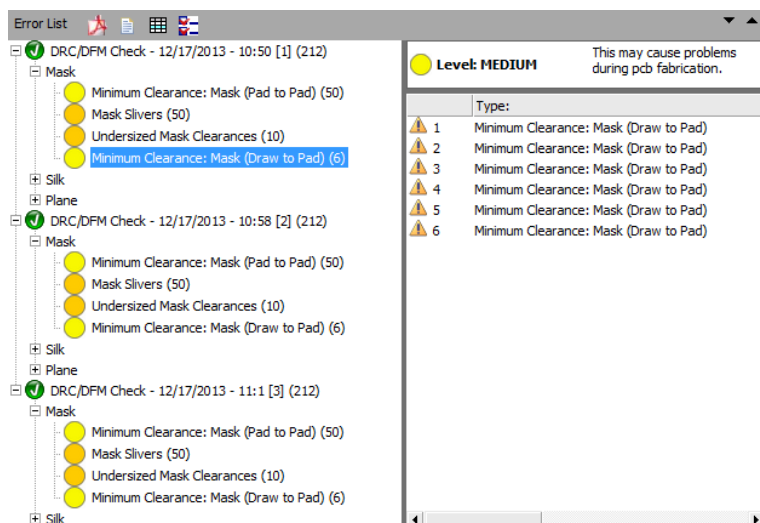
- b. Depending on the complexity of the design and the checks performed, processing time can take anywhere from 30 seconds to 5 minutes.



5. If any errors are detected, FAB 3000 will display them in the dialog box. If there are no errors, the process is complete.
6. Select **Finish**. Any errors listed will now appear in the DRC/DFM Violation Browser. The browser will appear just below the editor and provide you with an easy to use navigation tool to locate and describe each detected error.



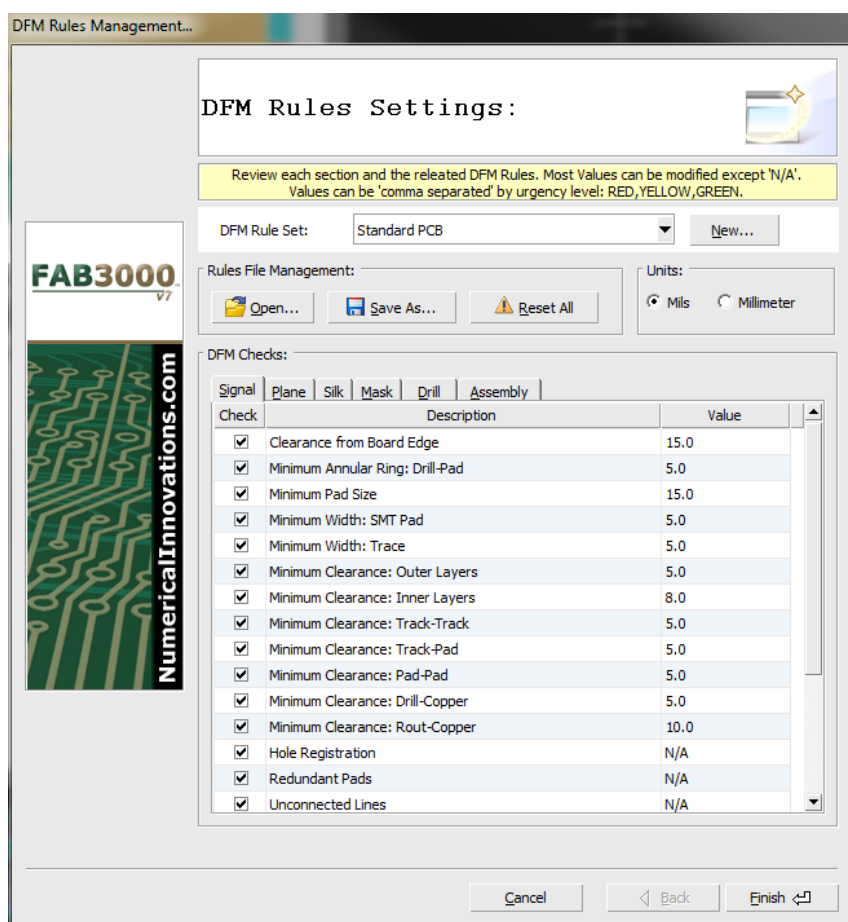
7. The browser contains a complete list of errors organized in a hierarchy format. Selecting and highlighting each error will provide a description of the error and severity level for PCB fabrication.
8. For each error selected, you'll be taken to the location of that error within the drawing for closer viewing and editing.



9. Create reports in PDF, Text, or CSV format by selecting the appropriate icon next to Error List.
10. You can also generate an error drawing which may be exported to various formats such as Gerber, DXF, or ODB++ and then imported back into your PCB layout tool for verification or to share with colleagues.

Tools > DFM Rules Management

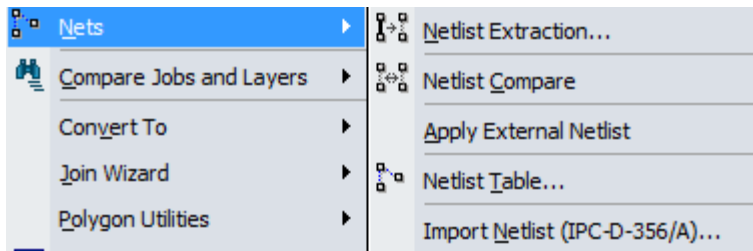
Use this setting to manage your DFM rules.



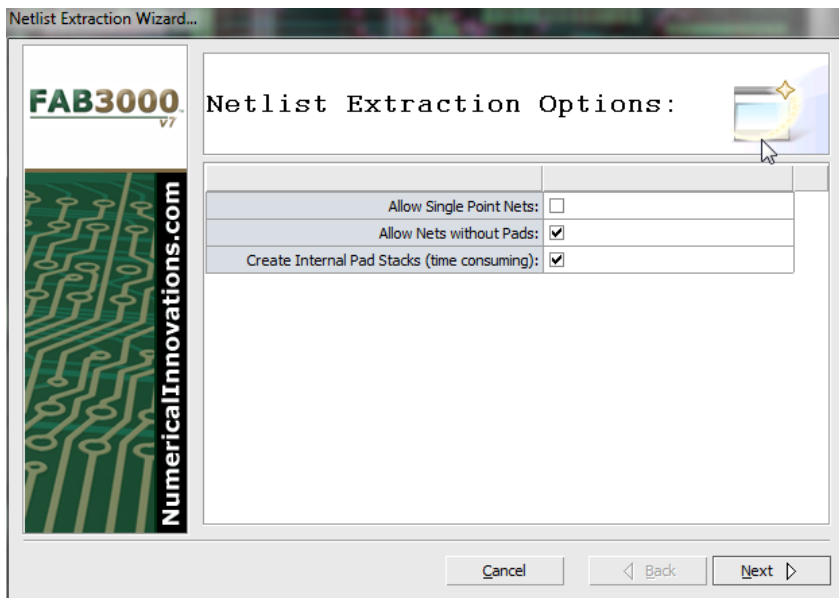
1. Select **Tools > DFM Rules Management**.
2. Select a **DFM Rule Set** from the drop down menu or create a new DFM Rule Set by selecting **New**.
3. Use the DFM Checks tabs to edit individual rules. Tabs include rules for Signal, Plane, Silk, Mask, Drill, and Assembly.
4. Select **Finish**.

Tools > Nets

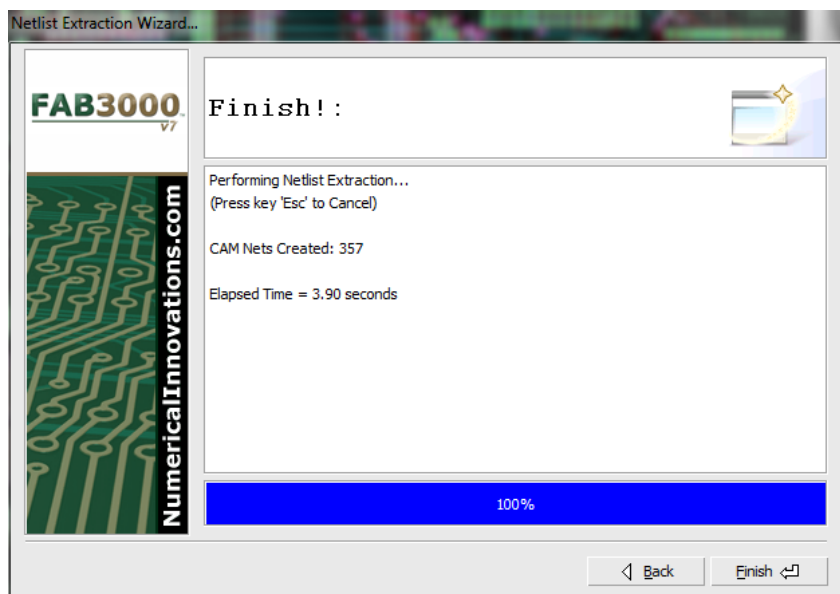
Extract, compare, and apply Netlists to your design.

**Tools > Nets > Netlist Extraction**

Extract a true interconnectivity map of any PCB design.



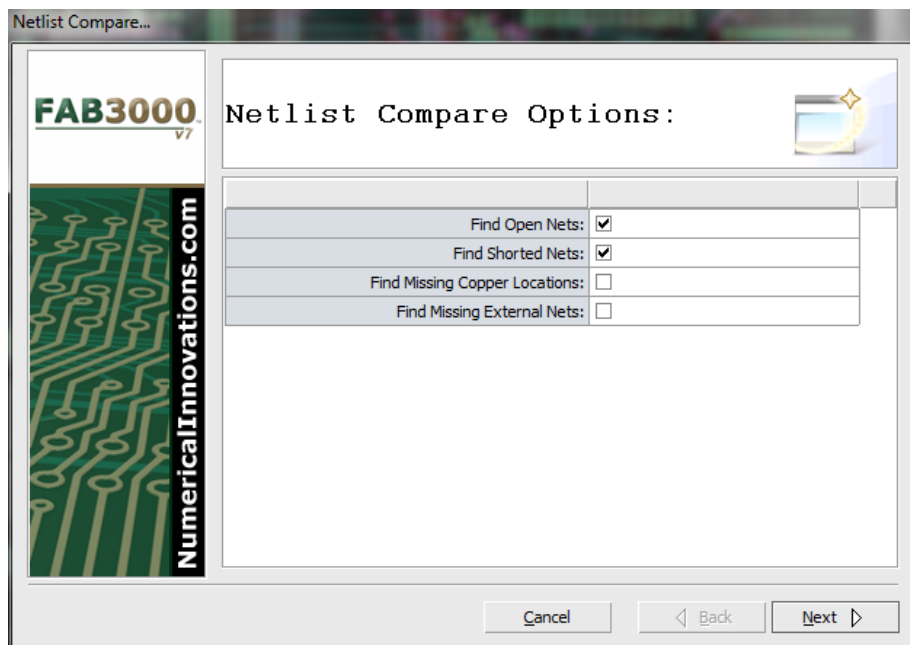
1. Select **Tools > Nets > Netlist Extraction**.
2. Verify and select extraction options.
New! Slice Large Polygons to permit increased performance – sometimes up to 10X faster.
3. Select **Next**. The extraction will be performed.



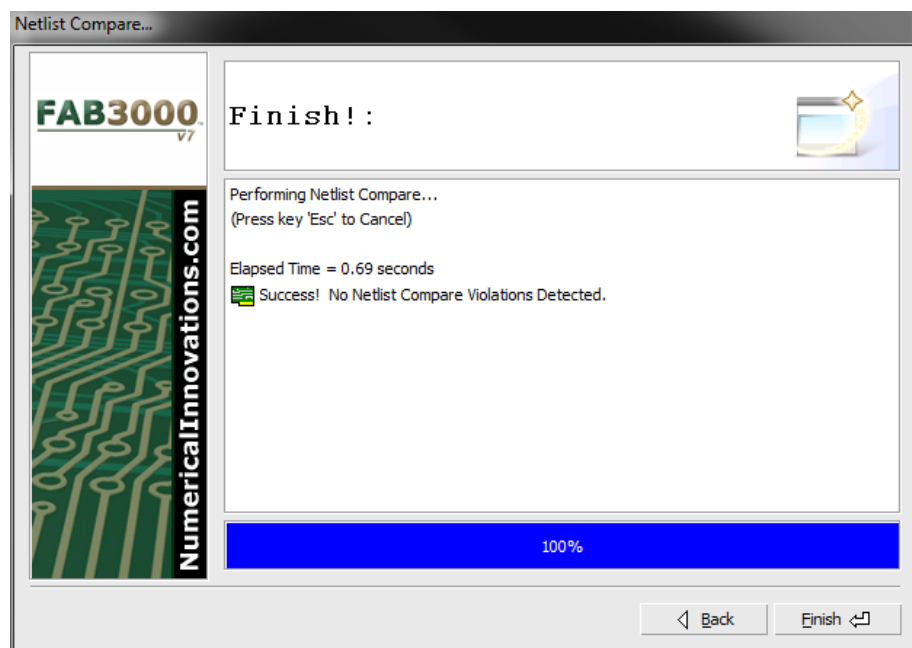
4. In the **Finish** dialog box, information will be included regarding CAM Nets created and elapsed time.

Tools > Nets > Netlist Compare

Compares extracted nets against an imported Netlist.



1. Select **Tools > Nets > Netlist Compare**.
2. Verify and select compare options.
3. Select **Next**. The comparison will be performed.



4. In the **Finish** dialog box, information will be included regarding comparison time and any violations that were detected during the comparison.

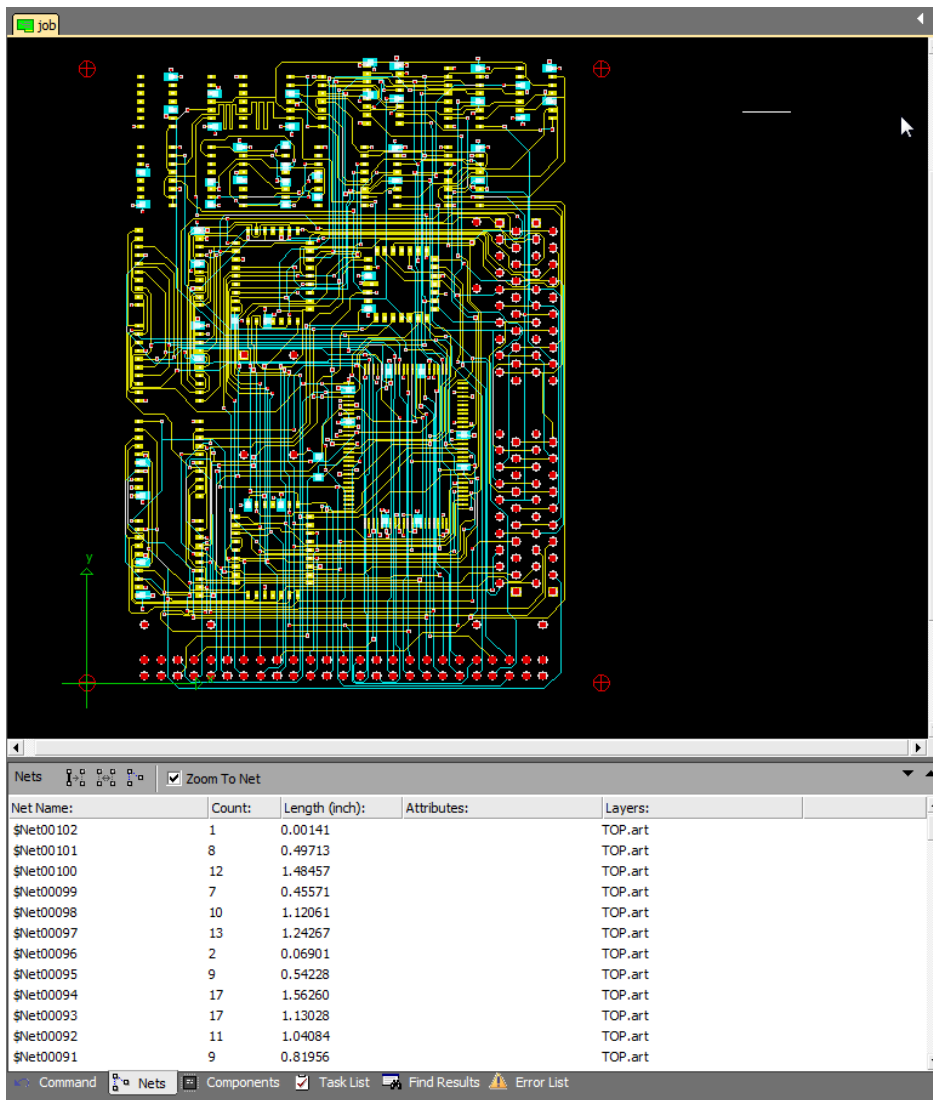


[Video: Check ODB++ Embedded Netlist Against the Artwork](#)

[Video: Performing Netlist Compare on Gerber Files](#)

Tools > Nets > Apply External Netlist

After a Netlist is extracted from Gerber & Drill data, there is no way for FAB 3000 to know the net names used in your CAD design. This allows you to import your CAD-generated Netlist, and apply the names to FAB 3000. After importing a Netlist into FAB 3000, perform the following:

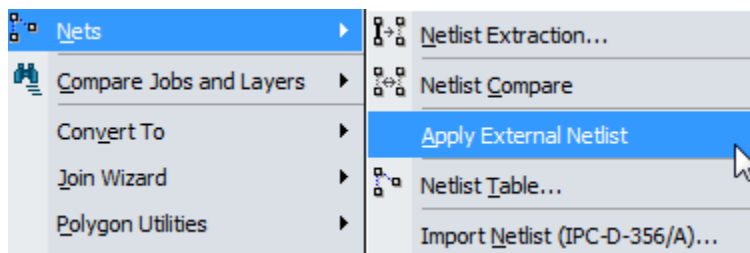


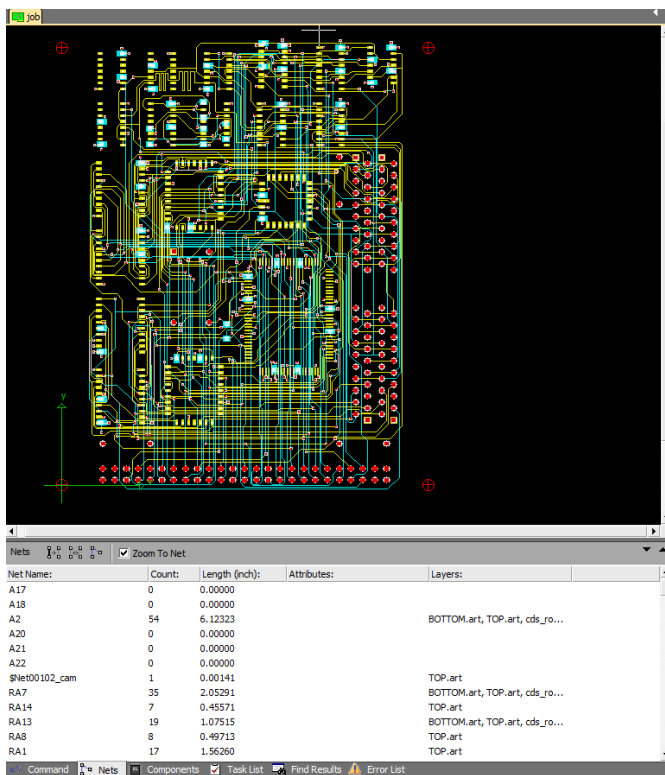
1. Extract a Netlist using **Tools > Nets > Netlist Extraction** (see 4.8: **Extracting Netlists from Gerber and/or Drill Data**). The extracted Netlist is pictured here. The Netlist information can be found at the bottom browser tab (below the editor) under “Nets”.

You’ll notice FAB 3000 generates a generic Net name beginning with “\$Net#”. Usually, designers will want to use the actual design net names in FAB 3000. In this case, they will use **Tools > Nets > Apply External Netlist**.

How to Apply External Net Names to CAM Net

1. Select **Tools > Nets > Apply External Netlist**.



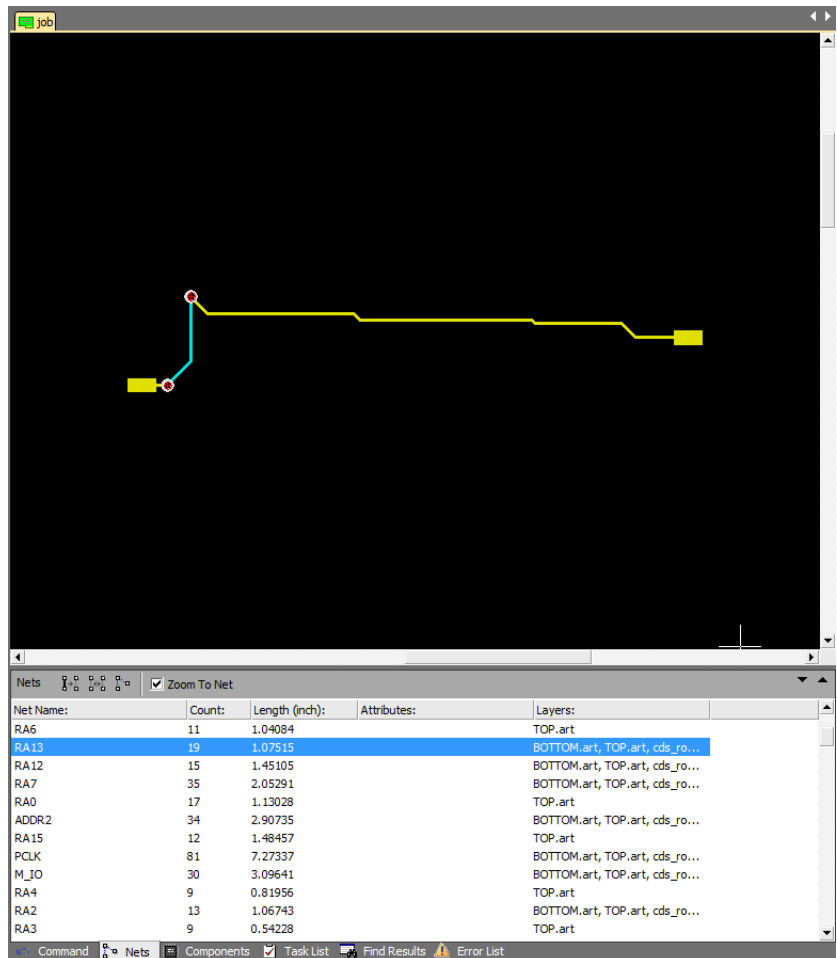


FAB 3000 will automatically update the existing CAM net names with the external net names.

You'll notice now in the Nets section tab that all Net names are no longer generic and refer back to the original names used in the CAD design.

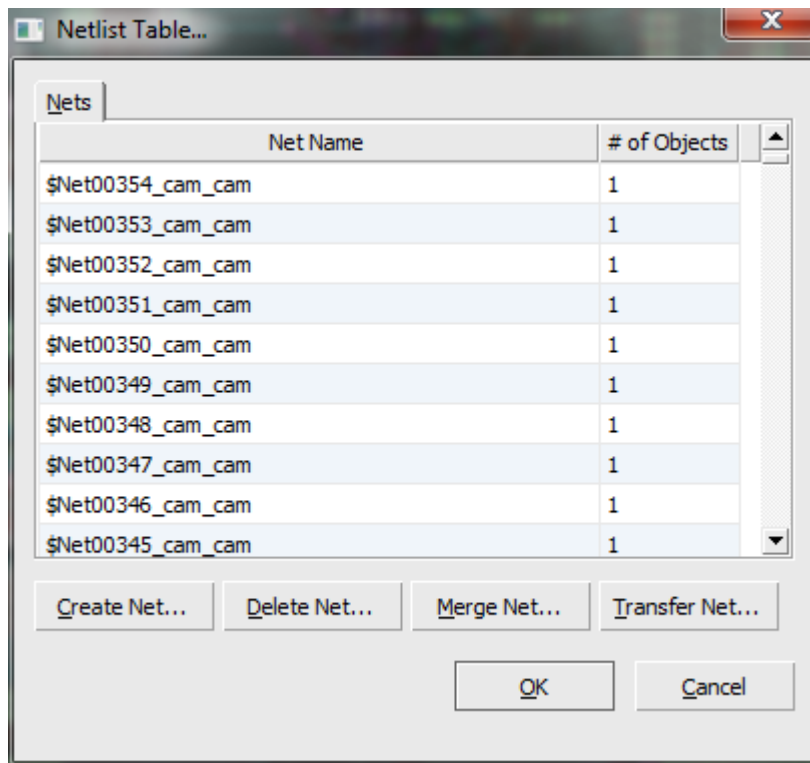
On some occasions you may find Nets in this list which may not have changed after running **Apply External Netlist**. This will most likely be the result of a Net flaw in the design.

To identify this flaw, run **Tools > Nets > Netlist Compare** (see Section 4.11).



Tools > Nets > Netlist Table

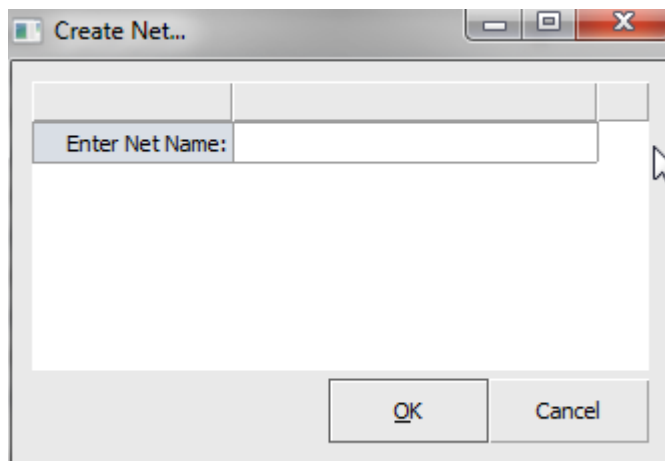
Create, delete, merge, transfer, and edit nets.



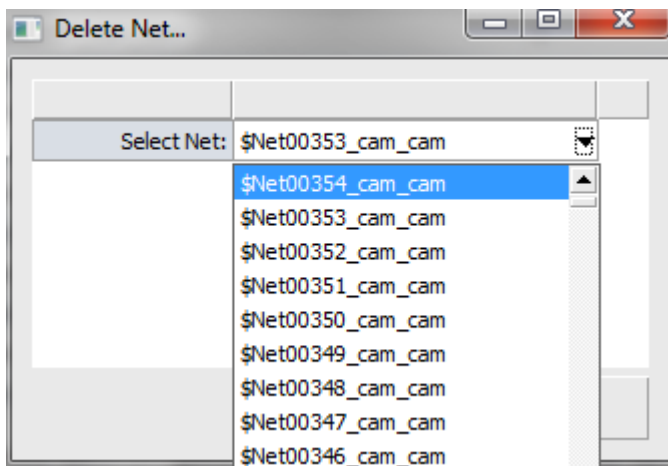
1. Select **Tools > Nets > Netlist Table**.

2. The Netlist Table dialog box will appear.

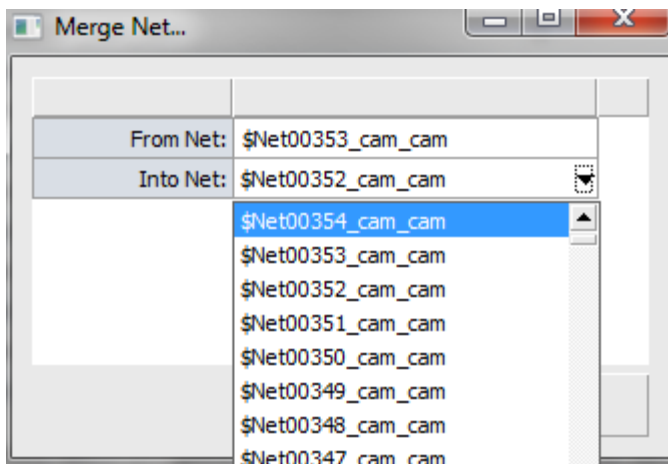
3. There are several options within the table. They are:



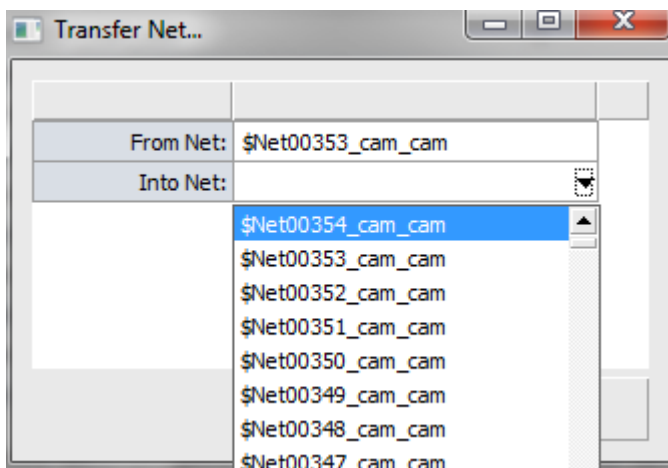
a. **Create Net:** Enter a Net name.



- b. **Delete Net:** Select a Net to delete from the pull down menu.



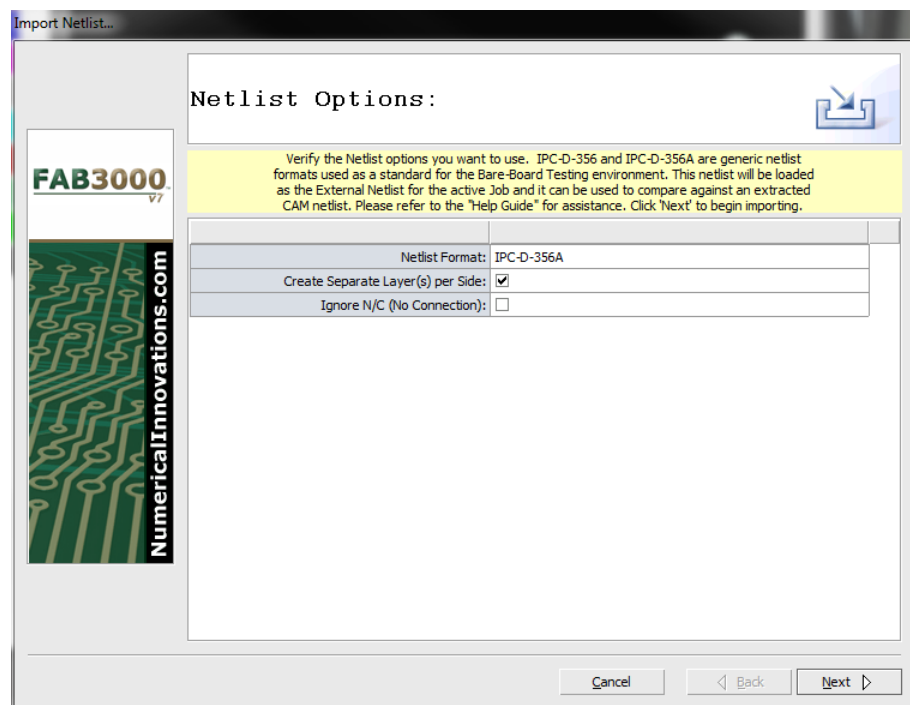
- c. **Merge Net:** Select a Net to merge from the pull down menu.



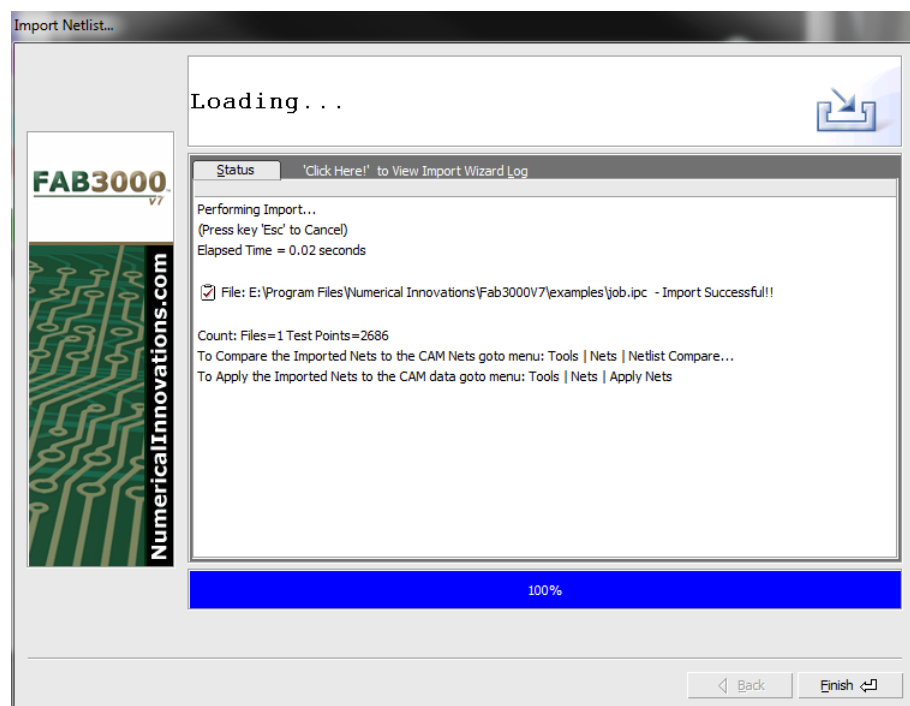
- d. **Transfer Net:** Transfer between Nets.

Tools > Nets > Import Netlist (IPC-D-356/A)

Imports an IPC-D-356/A formatted Netlist. These are generic Netlist formats used as a standard for the Bare-Board Testing environment.



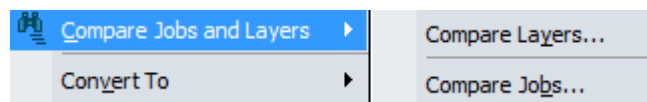
1. Select **Tools > Nets > Import Netlist (IPC-D-356/A)**.
2. Choose a Netlist file to import. Select **OK**.
3. The Import Netlist dialog box will appear.
4. Select Netlist import options.
5. Select **OK**. The Netlist importation process will begin.



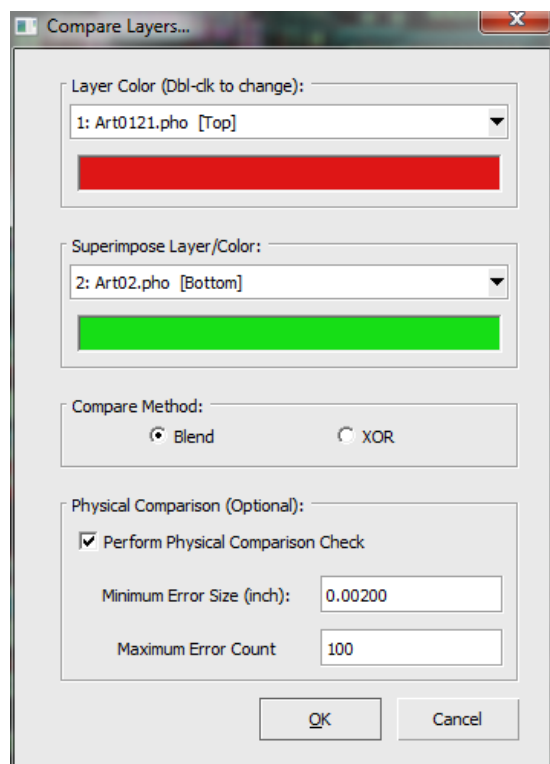
6. If there are any errors, they will be displayed in the dialog box.
7. When complete, select **Finish**.

Tools > Compare Jobs and Layers

Compares differences between layers. Overlays another Job over the active design.



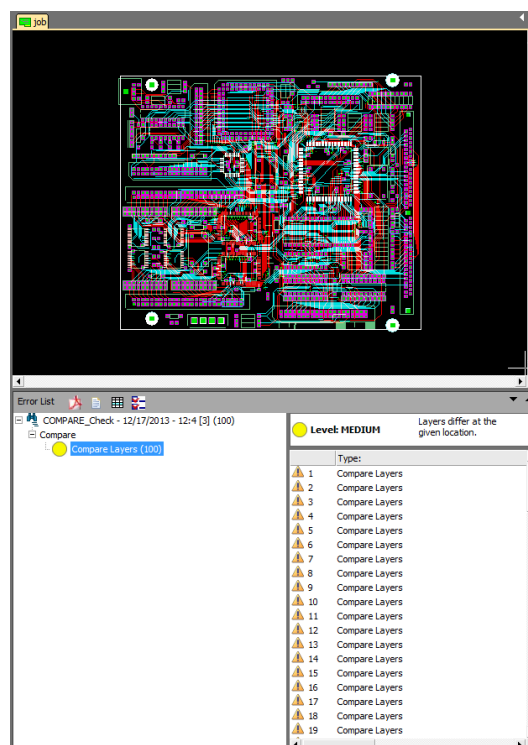
Tools > Compare Jobs and Layers > Compare Layers



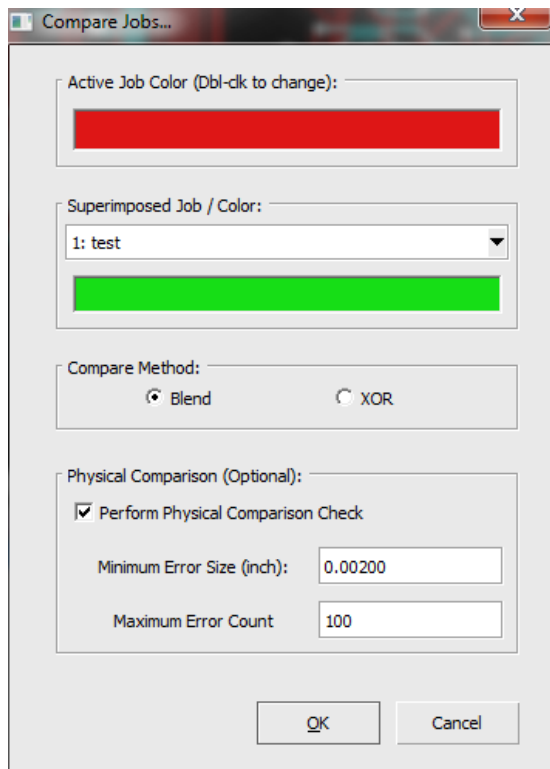
Compares differences between layers.

1. Select **Tools > Compare Jobs and Layers > Compare Layers**.
2. Select layers by choosing the layer name and assigning a layer color.
3. Choose a compare method and physical comparison type (optional).
4. Select **OK**.

5. The Compare Layers error browser will appear just below the editor.
6. Use the browser to compare and analyze each layer.
7. Use the browser to create a PDF, Text, or CSV error list.

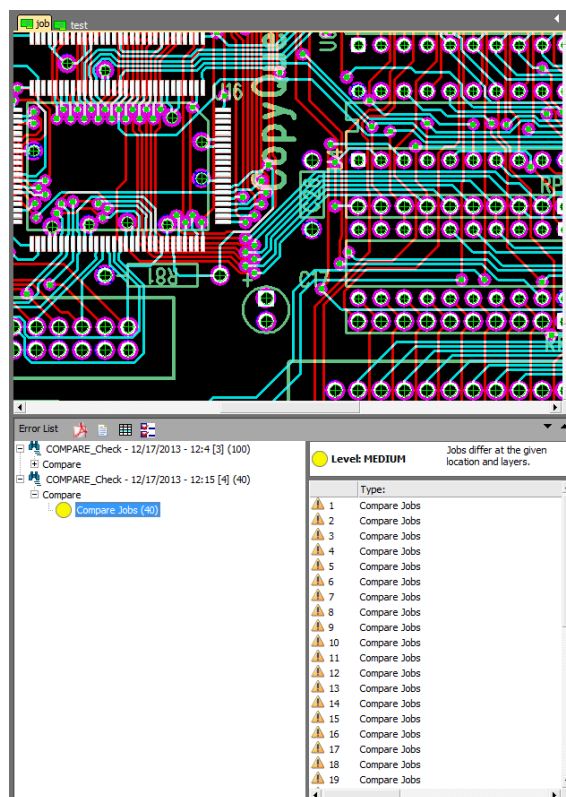


Tools > Compare Jobs and Layers > Compare Jobs



1. Select **Tools > Compare Jobs and Layers > Compare Jobs**.
2. Select Jobs by choosing the name and assigning a color.
3. Choose a compare method and physical comparison type (optional).
4. Select **OK**.

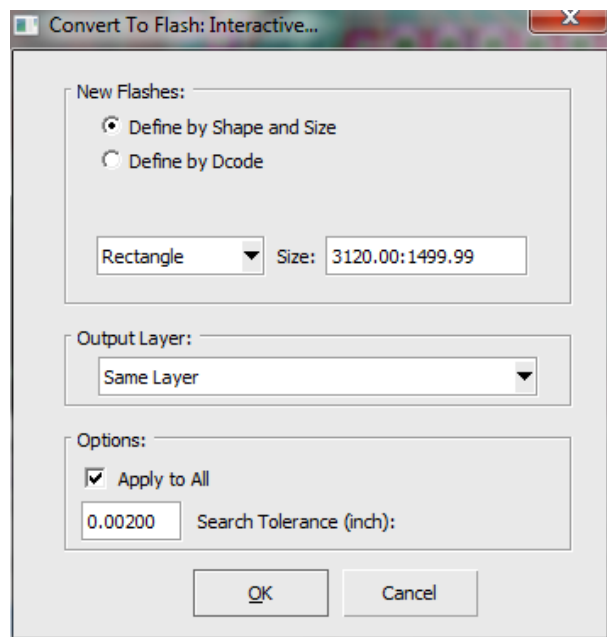
5. The Compare Jobs error browser will appear just below the editor.
6. Use the browser to compare and analyze each Job.
7. Use the browser to create a PDF, Text, or CSV error list.



Tools > Draw To Flash**Tools > Draw To Flash > Interactive**

Converts drawn pads to Gerber flashes.

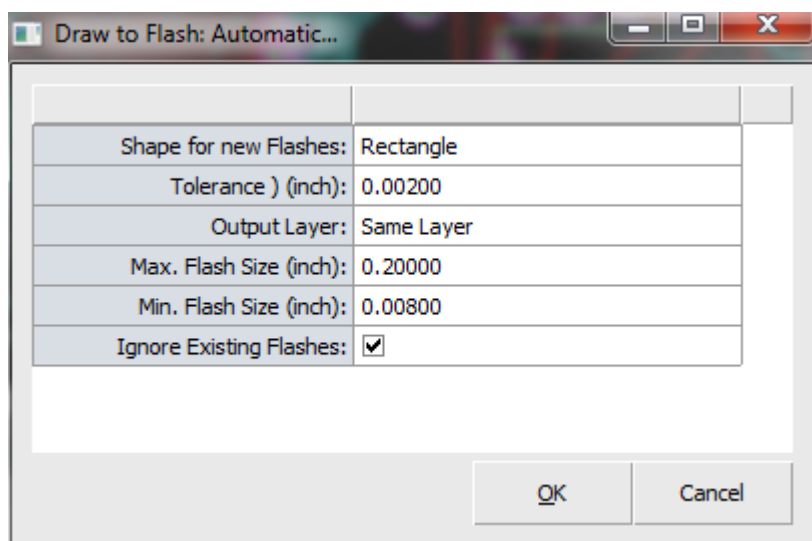
1. Select the drawn pads(s) that you want to convert to a flash.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Draw To Flash > Interactive**.



3. Verify or edit settings for new flashes. Select **OK**. A Convert to Flash dialog box will appear.
4. FAB 3000 will now search for all objects similar to your original selection and convert them to Gerber flashes.
 - a. Turn off any layers that you do not want to convert to flashes.
 - b. It's a good idea to make sure that your draw and flash colors are different colors respectively. This will make it easier to determine which draw pads have not been converted to flashes.

Tools > Draw To Flash > Automatic

Automatically generate Round, Rectangular flashes.

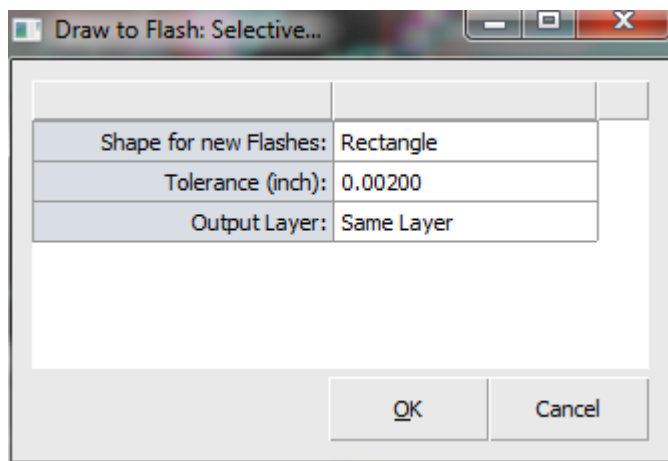


1. Select all objects in an area that you want to have round & rectangular flashes detected. It does not matter if pads are different sizes.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Draw To Flash > Automatic**. A Draw to Flash dialog box will appear.
3. Verify or edit settings for detecting new flashes.

4. Select **OK**. FAB 3000 will now detect round/rectangular pads and convert them to Gerber flashes.
 - a. It's a good idea to make sure Draw and Flash colors are different colors respectively. This will make it easier to determine which draw pads have not been converted to flashes.

Tools > Draw to Flash > Selective

Converts a selected group of objects to Gerber flashes.



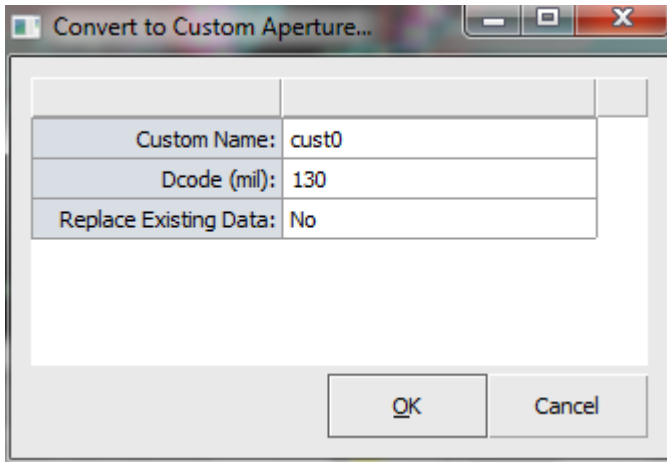
1. Select the drawn pads(s) that you want to convert to a flash.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Draw To Flash > Selective**. A Draw to Flash dialog box will appear.
3. Verify or edit settings.
4. Select **OK**.

It's a good idea to make sure that your draw and flash colors are different colors respectively. This will make it easier to determine which draw pads have not been converted to flashes.

Tools > Draw To Flash > Custom Aperture

Converts selected objects to a custom aperture definition.

1. Select all the objects that you want to convert to a custom aperture.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Draw To Flash > Custom Aperture**.

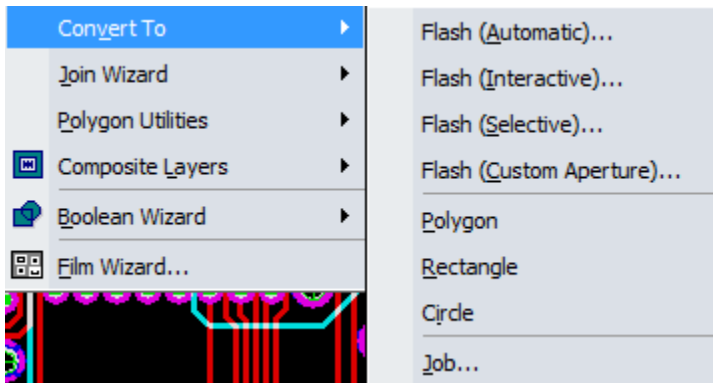


3. Enter the name you want to use as a reference for this custom aperture. You can determine if you want FAB 3000 to replace the selected objects with a new custom aperture or select "All" – FAB 3000 will search for all similar object groups and replace them with custom apertures.
4. FAB 3000 will now create a new custom aperture and add it to the Aperture Table.

- a. Nothing has changed in the active Job. FAB 3000 has simply constructed a new custom aperture based from the selected objects.
- b. The new custom aperture is now current and may be added by using the command **Add > Flash**.
- c. FAB 3000 is intelligent enough to recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

Tools > Convert To

Used for converting objects to flashes and polygons.



Tools > Convert To > Polygon

Converts selected objects to polygons.

1. Select all the objects that you want to convert to a raster polygon.
 - a. You must have at least one object selected before proceeding.

2. Select **Tools > Convert To > Polygon**.
3. FAB 3000 will now calculate all boundaries and construct clean raster polygons.
 - a. This is an excellent tool for optimizing large amounts of drawn data (like a plane layer).
 - b. FAB 3000 is intelligent enough to recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

Tools > Convert To > Rectangle

Converts selected objects to rectangles.

1. Select all the objects that you want to convert to a rectangle.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Convert To > Rectangle**.
3. FAB 3000 will now calculate all boundaries and construct rectangles.
 - a. This is an excellent tool for optimizing large amounts of drawn data (like a plane layer).
 - b. FAB 3000 is intelligent enough to recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

Tools > Convert To > Circle

Converts selected objects to circles.

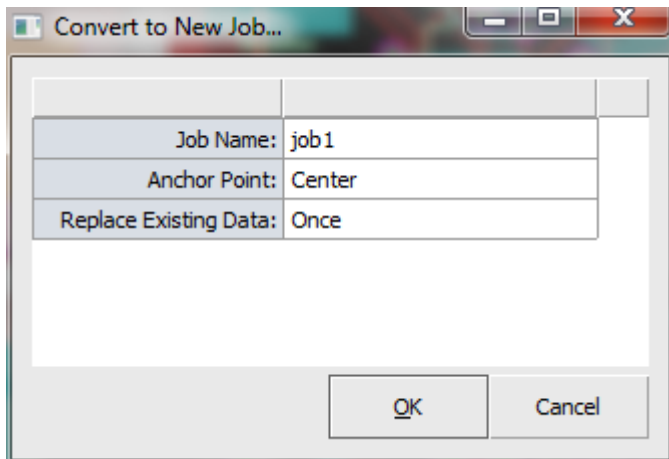
1. Select all the objects that you want to convert to circles.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Convert To > Circle**.
3. FAB 3000 will now calculate all boundaries and construct circle.
 - a. This is an excellent tool for optimizing large amounts of drawn data (like a plane layer).
 - b. FAB 3000 is intelligent enough to recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).



[Video: Move Vias and Parts in a Gerber File](#)

Tools > Convert To > Job

Converts selected objects to a new Job.



1. Select all the objects that you want to convert to a new Job.
 - a. You must have at least one object selected before proceeding.
2. Select **Tools > Convert To > Job**. A Convert to New Job dialog box will appear.
3. Verify or edit settings.
4. FAB 3000 will now create a new Job.

- a. Nothing has changed in the active Job. FAB 3000 has simply constructed a new Job based on the selected objects.
- b. You may now access the new Job from the Workspace Browser or Insert the new Job into another Job (creating hierarchy).
- c. FAB 3000 is intelligent enough to recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

Tools > Join Wizard

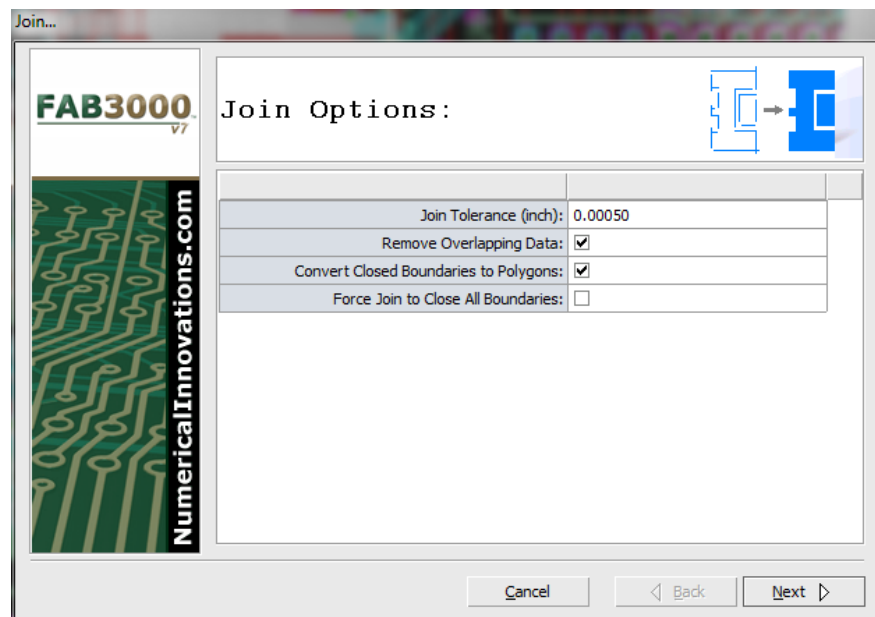
Automatically or interactively fix and close selected open boundaries.



[Video: Join Gerber Traces Lines PCB Border](#)

Tools > Join Wizard > Automatic

Automatically fix and close selected open boundaries.

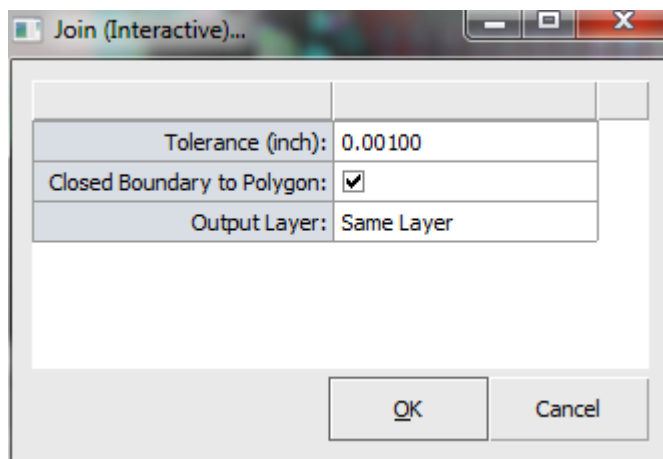


1. Select the boundary objects (lines, traces, arcs, etc.) you want to join.
 - a. This function requires that you select objects before proceeding.
2. Select **Tools > Utilities > Join**. The Join wizard dialog box will appear.
3. Edit or verify settings. Select **Next**.

4. FAB 3000 will now attempt to join all selected objects. When complete, a report will appear within the dialog box describing the number of open or closed boundaries created as well as the elapsed time.
 - a. If there are any open boundaries that could not be joined, FAB 3000 will automatically display their locations using the **Tools > Join Wizard > View Join Errors** command.

Tools > Join Wizard > Join (Interactive)

Selectively fixes and closes open boundaries. In many cases after running Join > **Join Wizard > Join (Automatic)**, you may discover that several troublesome boundaries still cannot be joined and closed, this is where Join Interactive is used.



1. Select **Tools > Join Wizard > Join (Interactive)**.

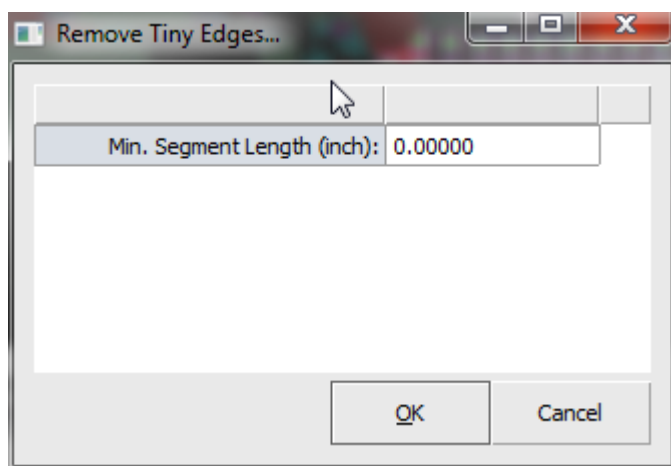
2. Choose the Tolerance and resulting layer for the joined objects. Select **OK**.
 - a. Tolerance is used when join interactive attempts to join adjacent objects.
3. Select an object (lines, arcs, traces, etc.) on the boundary you want to join.
 - a. Join Interactive will now attempt to join adjacent objects to form the closed boundary. When there is a gap or multiple objects, Join Interactive will stop and prompt you to select the next object on the boundary.
 - b. Join Interactive will also accept any coordinate point to select. This is handy when there is a big gap (or missing section) of a boundary.
 - c. Join Interactive will also accept the following commands:
 1. End – Stop and create an open boundary.
 2. Close – Stop and create a closed polygon.
 3. Back – Undo last object added to Join Interactive boundary.
4. Once the boundary has been closed, it will be automatically converted to a polygon and placed on the output layer.
5. Repeat steps 3-4.

Tools > Polygon Utilities

Utilities to edit or modify existing polygons.



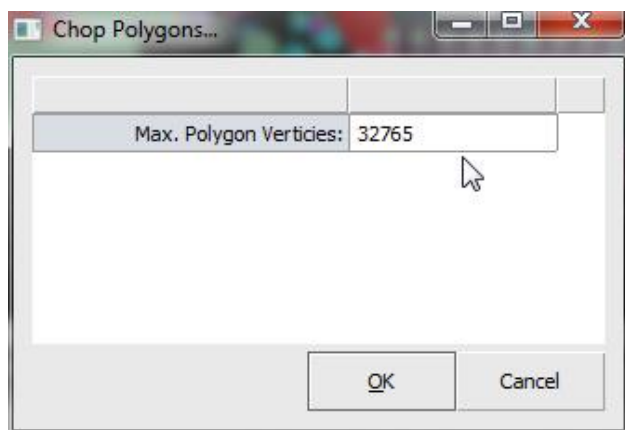
Tools > Polygon Utilities > Remove Tiny Segments: Remove small and self-intersecting segments within a polygon that can cause problems downstream.



1. Select the objects you want to clean.
 - a. This function requires that you select objects before proceeding.
2. Select **Tools > Utilities > Remove Tiny Segments**.
3. Verify or edit the minimum segment length value and select **OK**.
4. FAB 3000 will automatically remove all tiny segments.

Tools > Polygon Utilities > Slice Polygons

Splits large polygons with many vertices into many smaller polygons. This feature is very helpful when exporting data CAD/CAM systems that cannot handle polygons with many vertices. This function requires that you have already run Join beforehand and no error markers were detected.



1. Select the polygon objects you want to chop.
2. Select **Tools > Polygon Utilities > Slice Polygons**.
3. Verify the maximum polygon vertices value and select **OK**.
4. FAB 3000 will automatically chop all selected polygons.

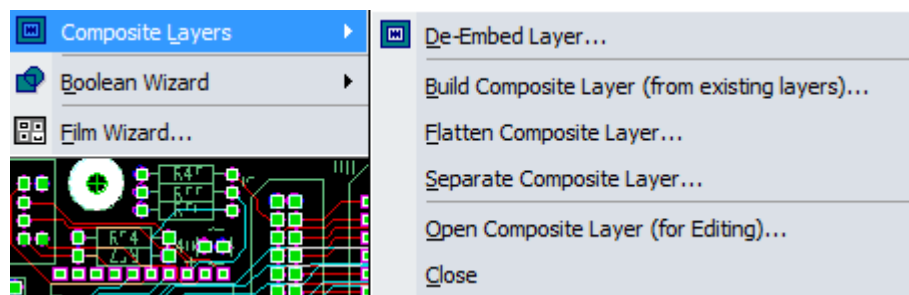
Tools > Polygon Utilities > Fix Invalid Polygons

Automatically finds self-intersecting polygons. These invalid polygons usually go undetected, and can cause unexpected errors & violations that result in mask failure.

1. Select **Tools > Polygon Utilities > Fix Invalid Polygons**.
2. FAB 3000 will automatically fix invalid polygons.
3. A small dialog box will appear showing the number of fixed or invalid polygons.

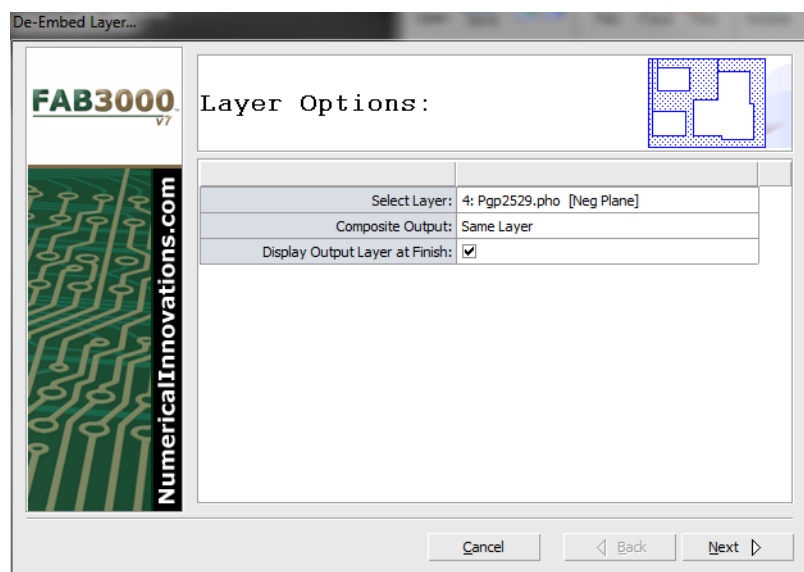
Tools > Composite Layers

Creates, converts, builds, and separates composites from layers.

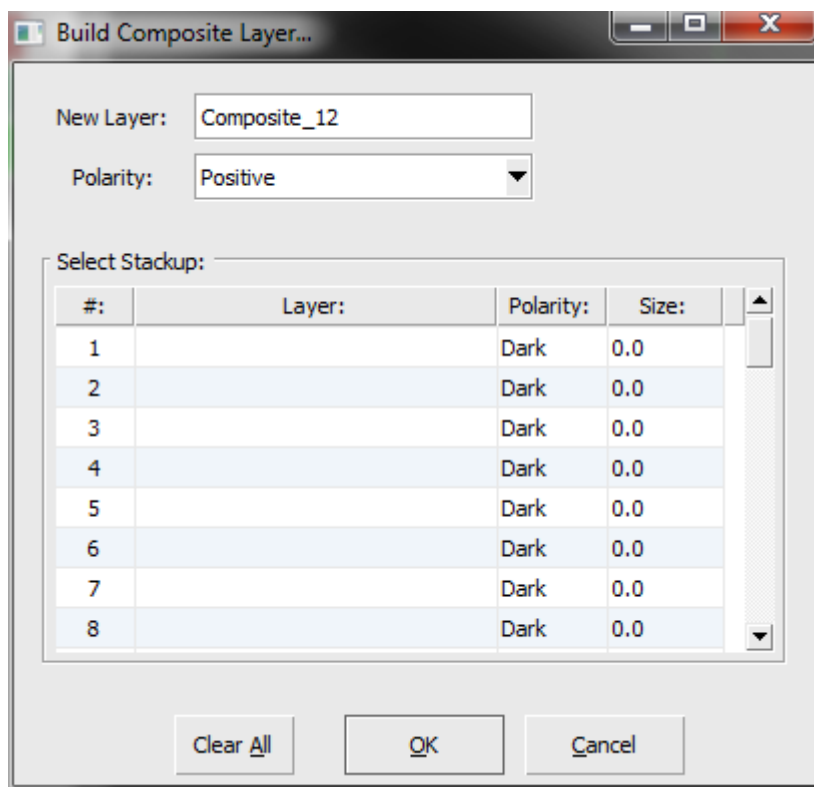


Tools > Composite Layers > De-Embed Layer

Performs polygon de-embedding per layer using the De-Embed Layer Wizard.



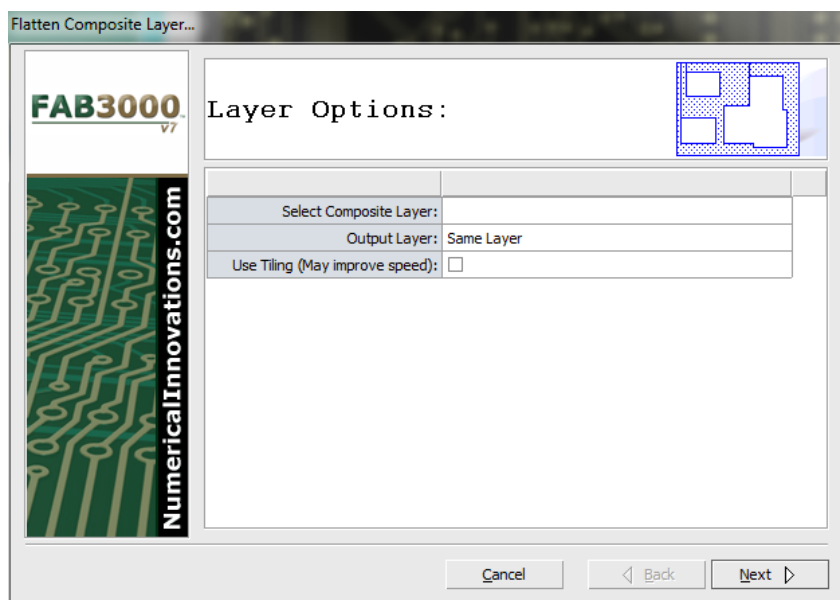
1. Select **Tools > Composite Layers > De-Embed Layer**. The De-Embed Layer wizard dialog box will appear.
2. Verify or edit layer options.
3. Select **Next**.
4. FAB 3000 will automatically perform polygon De-Embedding.



Tools > Composite Layers > Build Composite Layer

Creates a composite from existing layers.

1. Select **Tools > Composite Layers > Build Composite Layer**. The Build Composite Layer dialog box will appear.
2. Verify and edit options.
3. Select **Clear All** to clear selections or **OK** to finish.



Tools > Composite Layers > Flatten Composite Layer

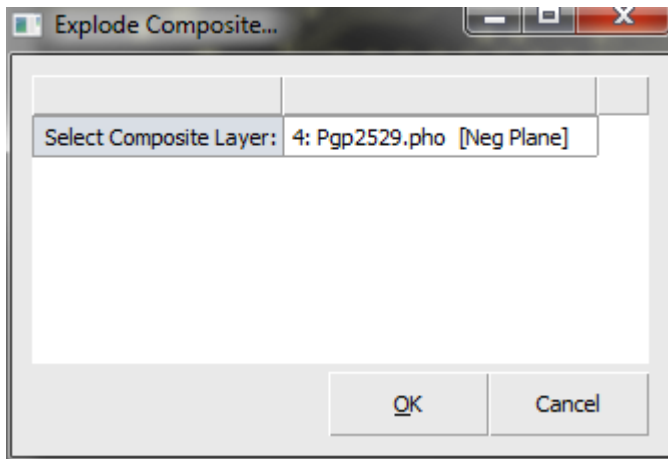
Converts an existing Composite into a single level layer.

1. Select the objects you want to clean.
 - a. This function requires you select a composite layer (a layer that has objects with a composite level greater than 0).

2. Select **Tools > Composite Layers > Flatten Composite Layer**.
3. Edit or verify layer options.
4. Select **Next**.

Tools > Composite Layers > Separate Composite Layer

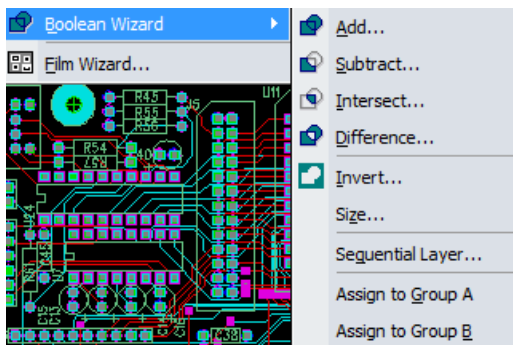
Converts an existing composite into multiple layers. One layer per composite level.

**Tools > Composite Layers > Open Composite Layer for Editing**

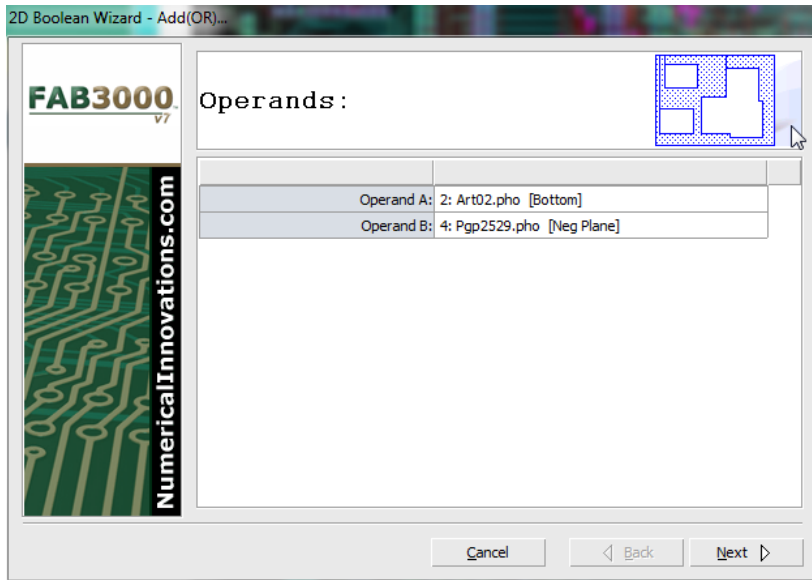
Opens an existing composite layer for editing.

Tools > Composite Layers > Close

Closes a previously opened composite layer for editing.

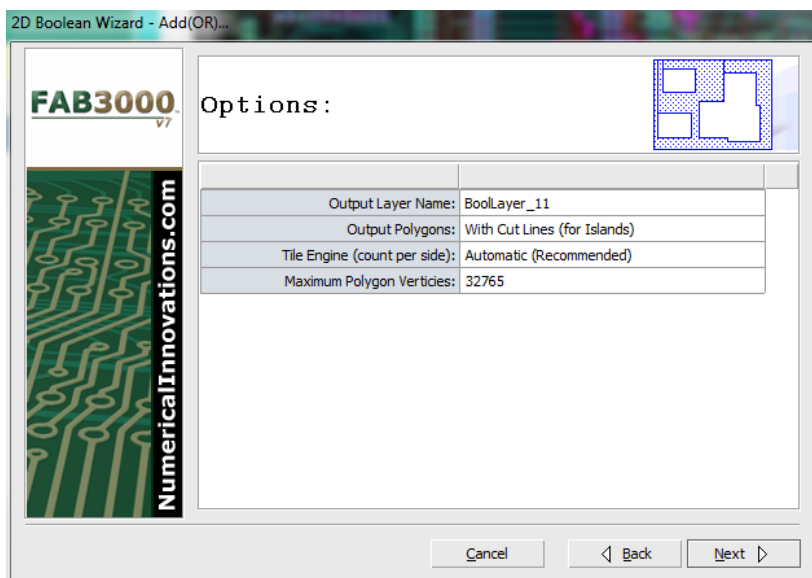
Tools > Boolean Wizard

Tools > Boolean Wizard > Add



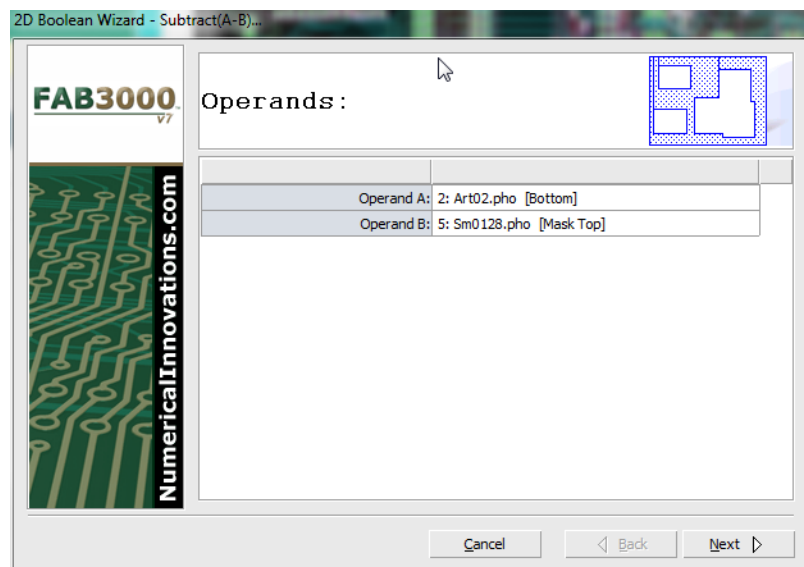
Performs Boolean operation OR.

1. Select **Tools > Boolean Wizard > Add**.
2. Assign Operands A and B.
3. Select **Next**.



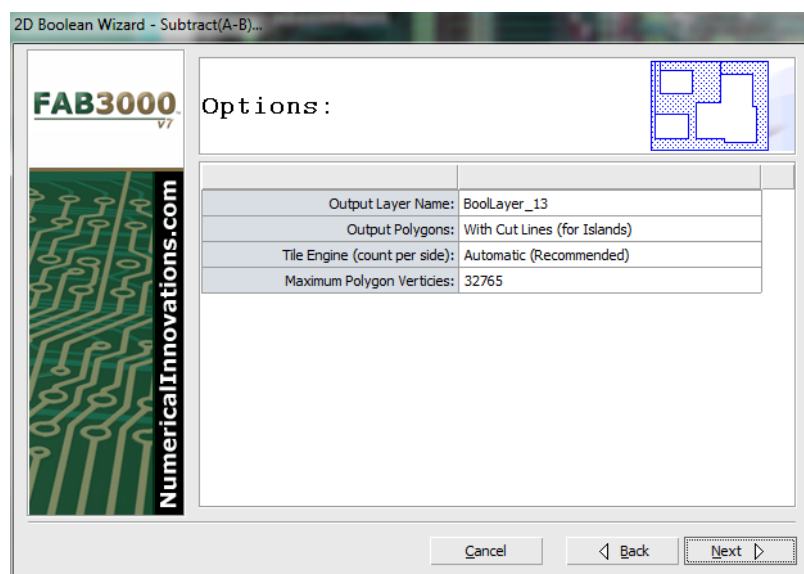
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Add calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Subtract



Performs Boolean operation Subtract.

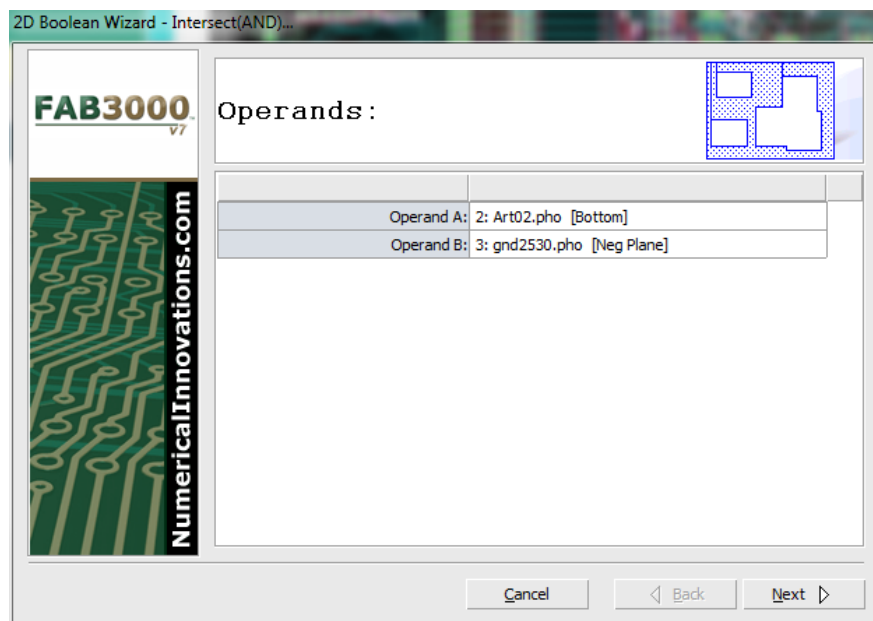
1. Select **Tools > Boolean Wizard > Subtract**.
2. Assign Operands A and B.
3. Select **Next**.



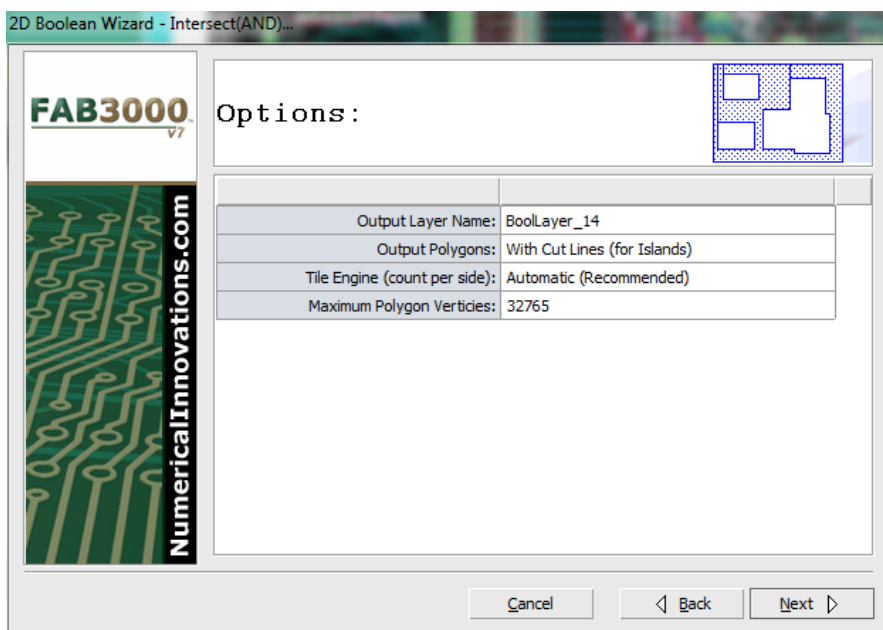
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Subtract calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Intersect

Performs Boolean operation Intersect.

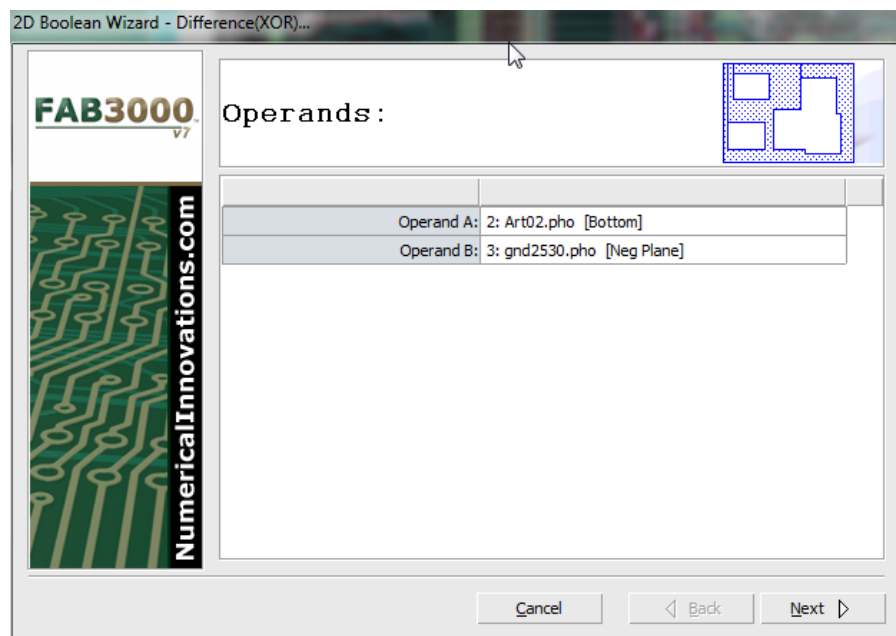


1. Select **Tools > Boolean Wizard > Intersect**.
2. Assign Operands A and B.
3. Select **Next**.



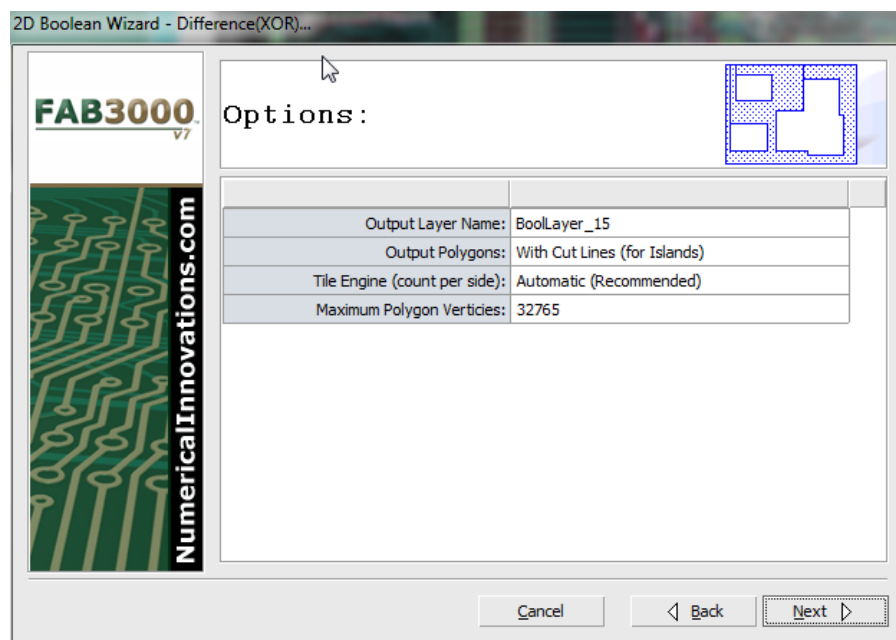
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Intersect calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Intersect



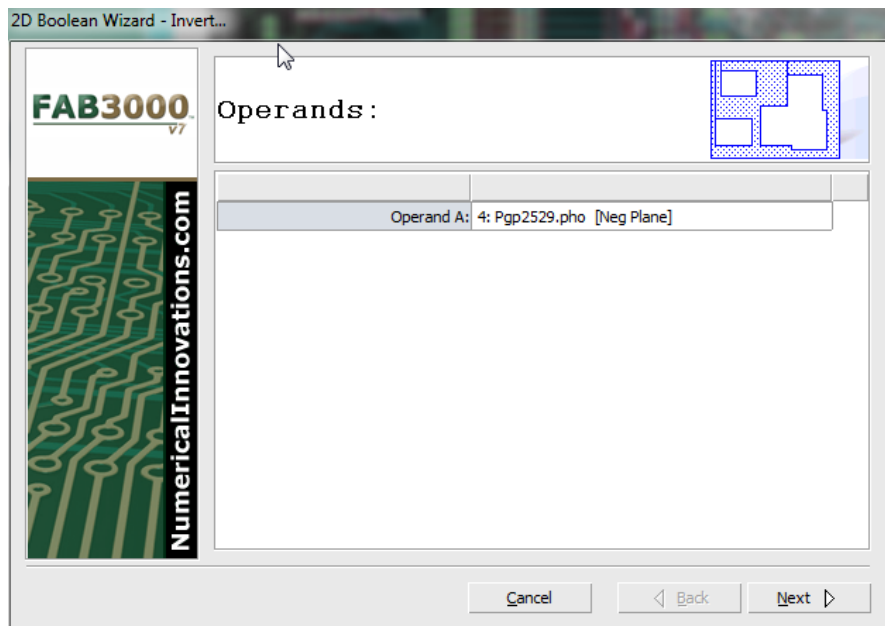
Performs Boolean operation XOR.

1. Select **Tools > Boolean Wizard > Difference**.
2. Assign Operands A and B.
3. Select **Next**.



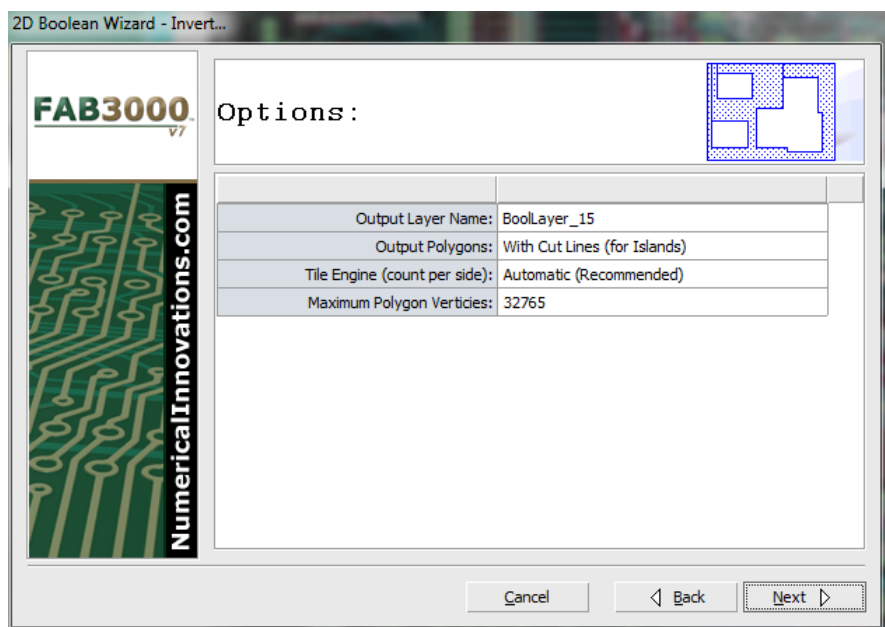
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Difference calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Invert



Performs Boolean operation
Invert.

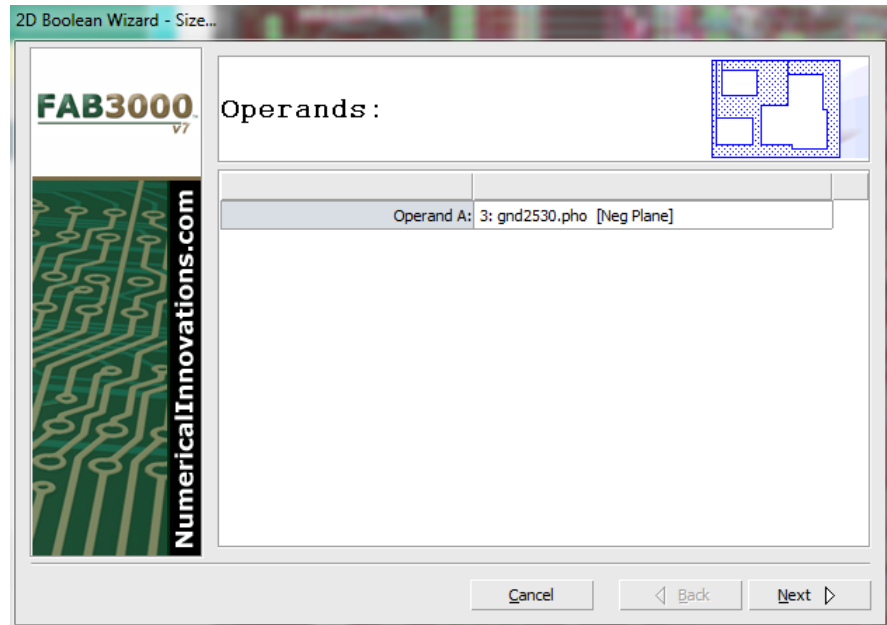
1. Select **Tools > Boolean Wizard > Invert**.
2. Assign Operand A.
3. Select **Next**.



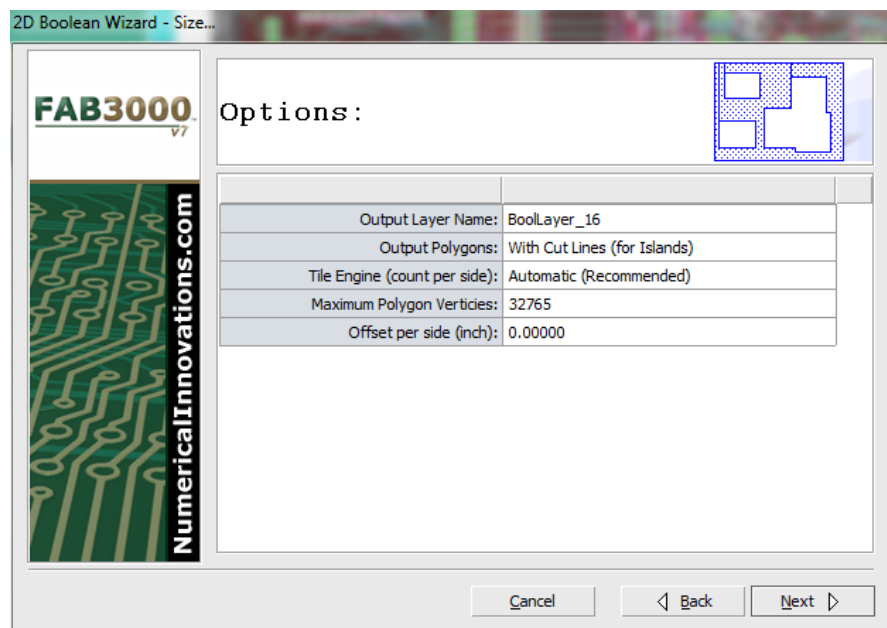
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Invert calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Size

Performs Boolean operation Size.



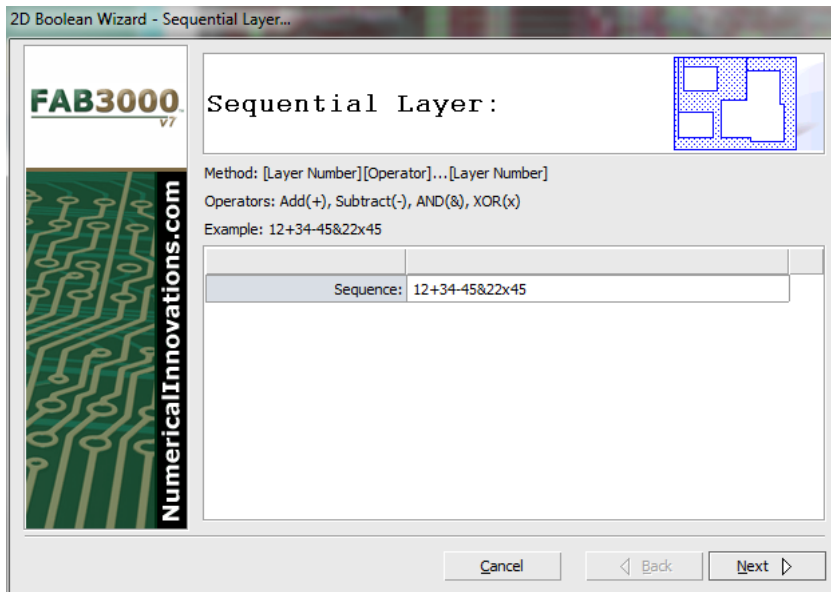
1. Select **Tools > Boolean Wizard > Size**.
2. Assign Operand A.
3. Select **Next**.



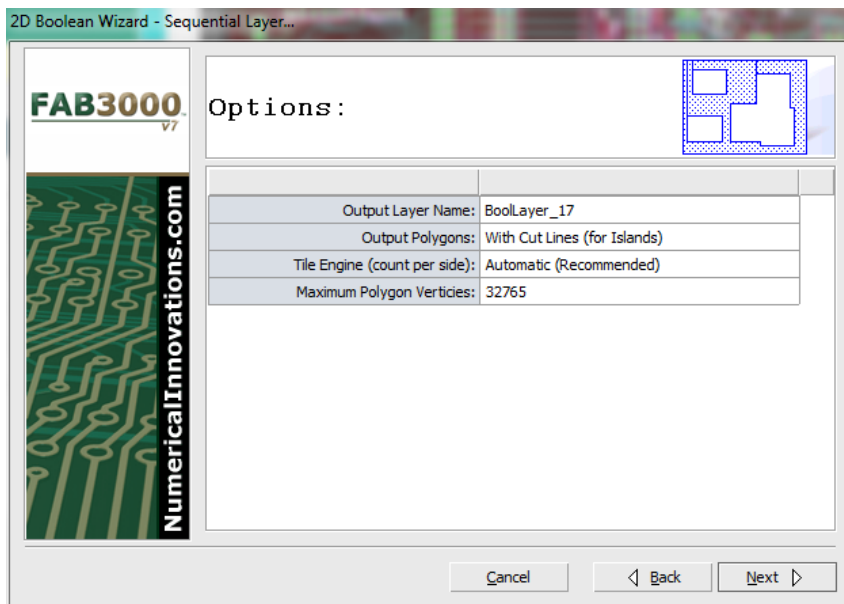
4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Size calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Sequential Layer

Performs Boolean operation Sequential Layer.



1. Select **Tools > Boolean Wizard > Sequential Layer**.
2. Assign a **Sequence**. See the dialog box for method and operator format.
3. Select **Next**.



4. Verify and edit options.
5. Select **Next**.
6. FAB 3000 will perform Boolean Sequential Layer calculations.
7. Select **Finish** when complete.

Tools > Boolean Wizard > Assign to Group A

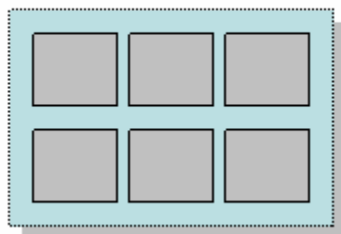
Assigns Boolean objects to a complete group (A).

Tools > Boolean Wizard > Assign to Group B

Assigns Boolean objects to a complete group (B).

Tools > Film Wizard

This unique feature from FAB 3000 is guaranteed to **SAVE YOU UP TO 50% OFF** on your Photoplotting service expenses. It will eliminate any operator errors (due to someone else wrongly mirroring or using the wrong polarity - i.e. Negative/Positive) and gets faster Turn-Around times. Within 4-5 Jobs, FAB 3000 Film Wizard will have already paid for itself.



Film Wizard will automatically setup Gerber layers for optimal film output. This is perfect for anyone who needs to create Gerber or Postscript photoplots from their artwork.

Here's how it works:

You've just finished a design which contains 8 separate layers for photoplotting. Let's assume that your design image area is 5" X 7" (120mm X 175mm). If you were to send this design to your photoplotting service for film, you would be charged about \$160 total (\$20 per film X 8 layers = \$160 total). **Film Wizard** will group this design into 2 larger film layers - each containing 4 of your original layers. The cost of using film wizard will **only be \$80 total** (\$40 per larger film X 2 film layers).

Photoplotting services take your design and edit (mirror/polarity) each layer per customer instructions. They group & setup the individual layers onto one or more larger pieces of film. The larger film is sent directly to the photoplotter, and after plotting has completed, the large film is sliced up into smaller individual films which are then sold to you.

Film Wizard performs all editing (mirror/polarity), grouping, and panelization steps automatically! With just a few clicks you'll save 50%; eliminate any potential operator errors and improve your film turn-around time.



Using FAB 3000 Film Wizard

1. Go to **Tools > Film Wizard**.

Assuming you have an 8 layer design, with an area of 5"X7" (120mm X 175mm). Choose a film size that will be able to hold at least 4 design layers. In this example a film size of 16" X 20" will work perfect. (You may


Make sure to contact your photoplotting service to determine what acceptable film sizes to use for their photoplotting process.

Film Wizard...

Film Options:

Output Job:	Same
Border Spacing (inch):	0.25000
Add Cut Lines:	<input type="checkbox"/>
Add Crop Marks:	<input type="checkbox"/>
Add Header Text:	<input checked="" type="checkbox"/>
Header Label:	Film Wizard



Cancel
◀ Back
Next ▶

2. Assign options that you want to appear on each film layer:

Output Job: Select the output job of either **Same** or **New**.

Border Spacing: Minimum spacing per each individual film.

Add Cut Lines: Check box to add cut lines to film.


Add Crop Marks: Check this box to add cut lines so a photoplotting service can cut individual films.


Add Header Text: Places separate text for each layer. Text includes the layer name and time.

Header Label: Apply a label to the header.

3. Select Film Setup based on your required layer design and sizing. Select **Next**.


Film Wizard...



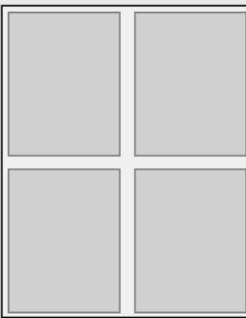


Film Setup:

Media Width:	16
Media Height:	20
Film Count(X):	2
Film Count(Y):	2
Units:	inch




16 X 20 Media, 2 X 2 Count, Inch




Film Size: 8.00x10.00

4. Assign polarity & mirror options for each individual layer. Select **Next**.


Film Wizard...

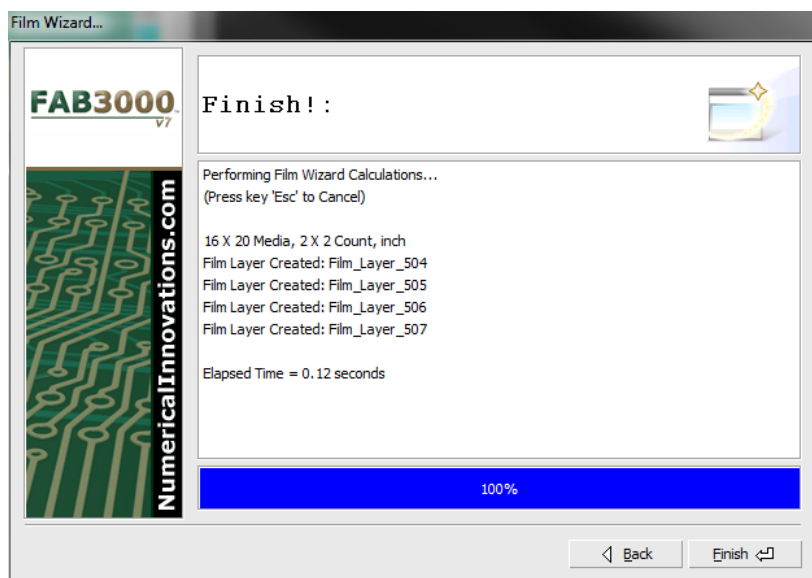




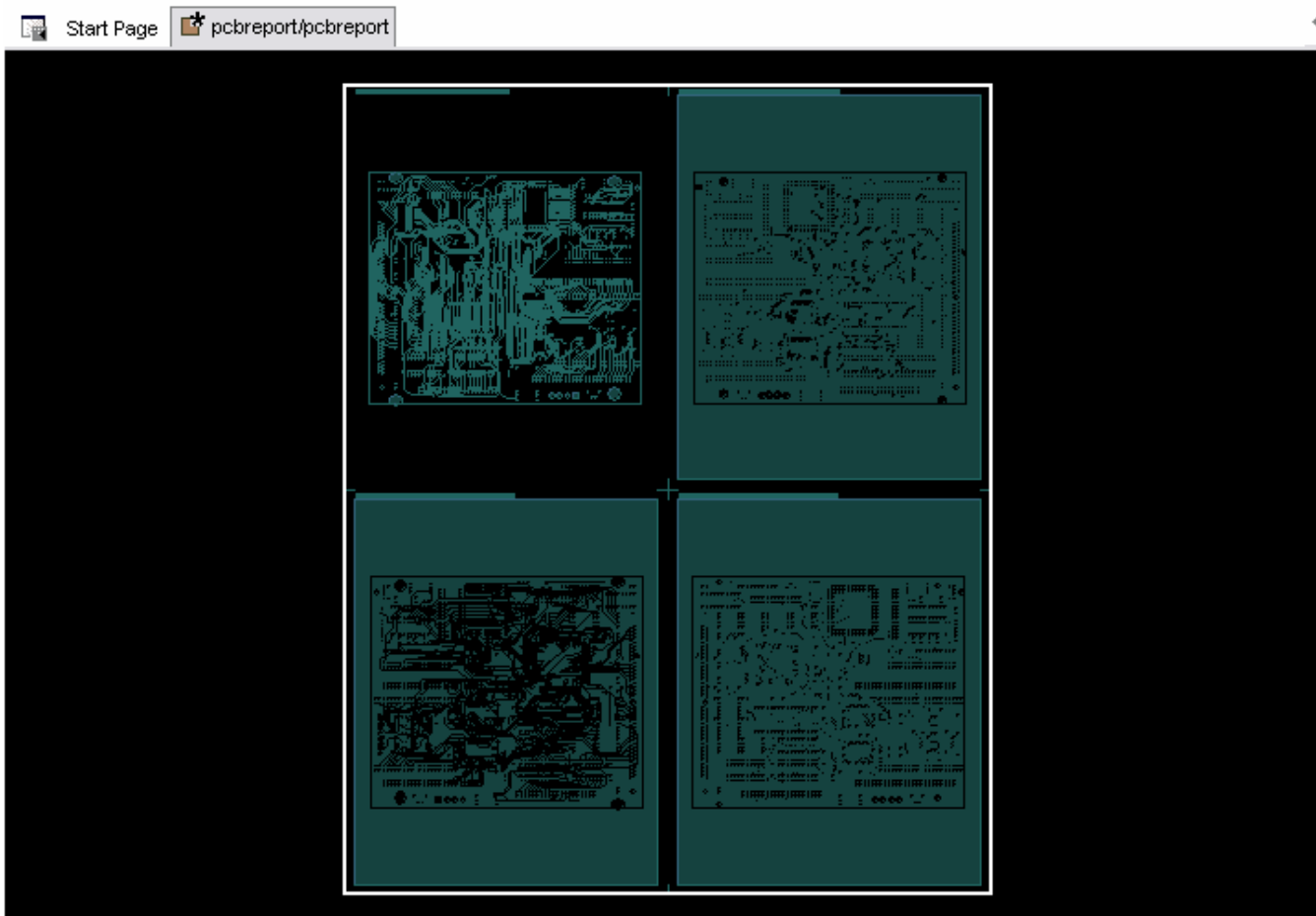
Negative/Mirror Setup:

Use:	Layers:	Negative:	Mirror:
<input checked="" type="checkbox"/>	L1: Art0121.pho [Top]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L2: Art02.pho [Bottom]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L3: gnd2530.pho [Neg Plane]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L4: Pgp2529.pho [Neg Plane]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L5: Sm0128.pho [Mask Top]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L6: Sm0227.pho [Mask Bottom]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L7: Sp0128.pho [Paste Top]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L8: Sst0126.pho [Silk Top]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L9: Drill.drl [Drill ThruHole]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L10: job.ipc [Netlist]	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L11: BoolLayer_11	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L12: BoolLayer_12	<input type="checkbox"/>	<input type="checkbox"/>
<input checked="" type="checkbox"/>	L13: BoolLayer_13	<input type="checkbox"/>	<input type="checkbox"/>





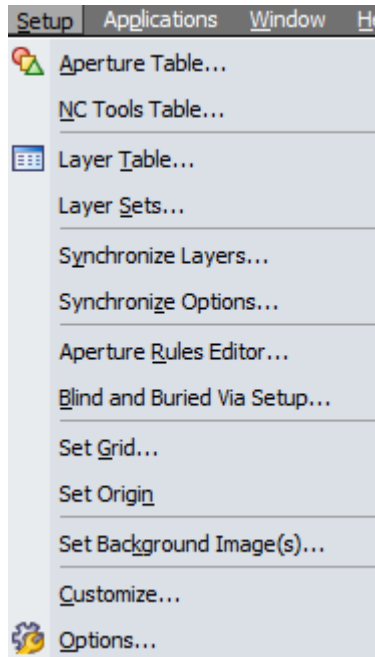
You're finished! Below is the screen capture of the outputted layer(s) from Film Wizard. Notice that mirror and rotation have been included automatically!



At this point all you need to do now is to export these "Film Wizard" layers to either Gerber or Postscript, and that's it!

You've just saved 50% on your photoplotter service expenses!

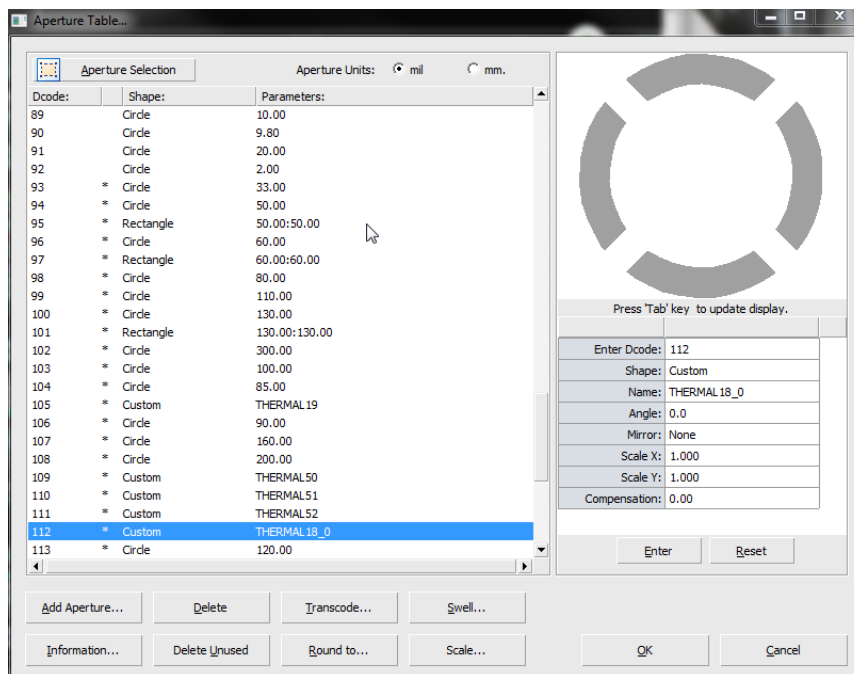
5.10 Setup Menu



The **Setup** menu provides a number of customization tools for apertures, NC tools, layers, and more.

Setup > Aperture Table

Displays and edits all apertures used in the workspace.



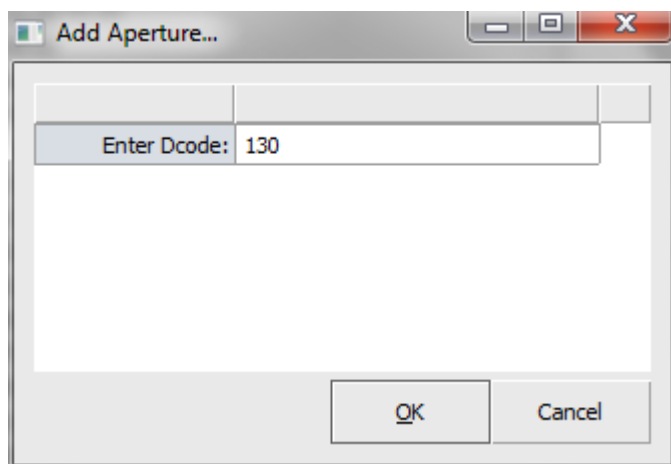
1. Select **Setup > Aperture**. The Aperture Table dialog box will appear.
2. Select an aperture to edit from the list.
3. Edit or verify the aperture details in the preview box on the right. Use the Tab key to cycle through each menu.
4. Select **OK** when complete.



[Video: Create Aperture and then change Dcode of selected objects using FAB 3000](#)

Additional functions included in the aperture table are:

Add Aperture: Add an aperture to the list. Enter a Dcode in the entry box. When finished select **OK**.

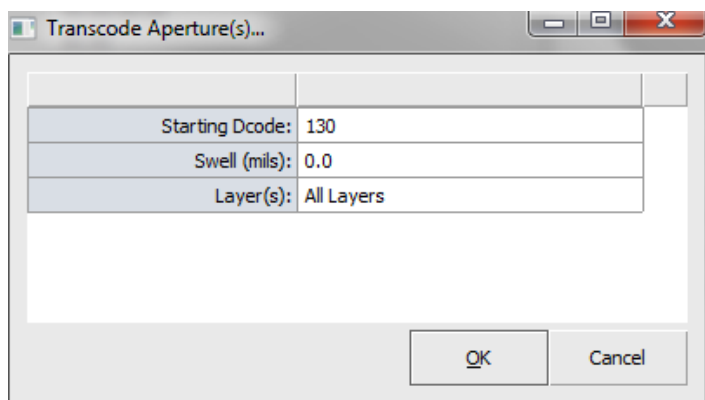


Enter Dcode:	130	

OK Cancel

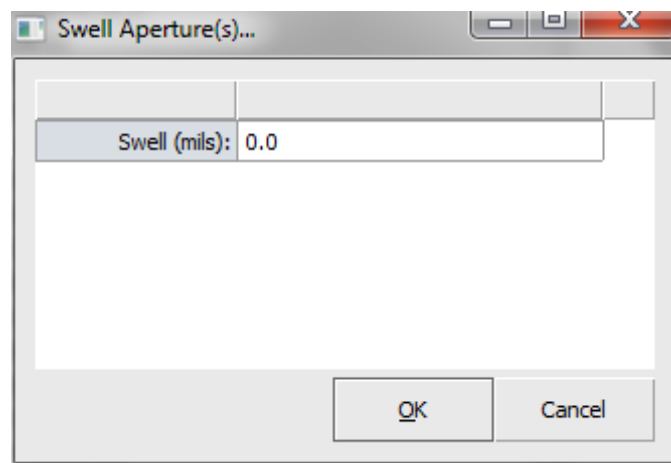
Delete Aperture: Deletes an unused aperture from the list. This operation cannot be undone.

Transcode Aperture(s): Used to transcode apertures by entering starting Dcode, swell, and selecting specific layers.



Starting Dcode:	130	
Swell (mils):	0.0	
Layer(s):	All Layers	

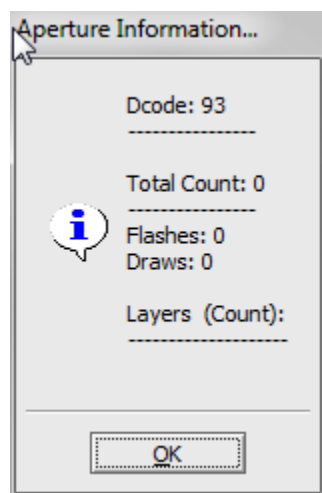
OK Cancel



Swell (mils):	0.0	

OK Cancel

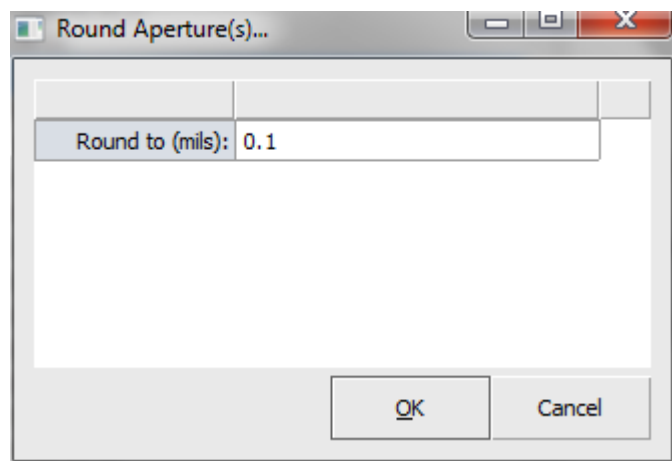
Swell Aperture: Use this entry box to Swell apertures in mils or mm.



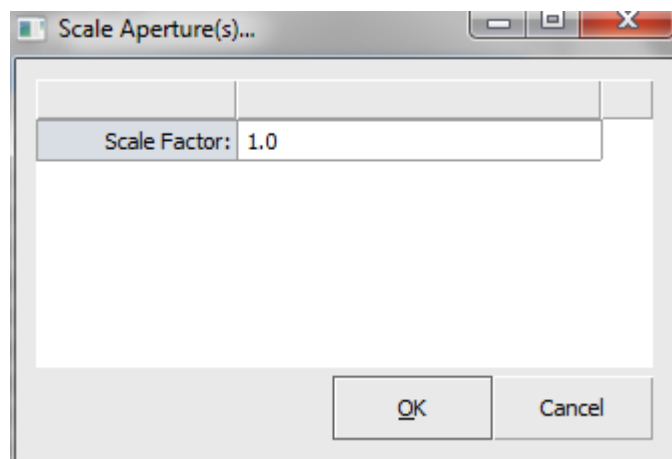
Aperture Information: Provides information on specific apertures. Highlight an aperture first, then select **Information**. Includes information such as Dcode selected, total count, flashes, draws and layer count.

Delete Unused: Deletes unused apertures. This operation cannot be undone.

Round to: Used to Round an aperture in mils or mm.



Scale Aperture: Used to scale an aperture based on scale factor.



Notes:

- a. Use **Aperture Units** to select units in mil and mm.
- b. Select the **Aperture Selection** button to select all apertures in the list. The entire list will highlight in blue.
- c. Under Aperture preview, use the Tab key to update the display after entering or selecting a value. Alternatively, use the **Enter** or **Reset** keys to edit the values.

Setup > NC Tools Table

Displays and edits all NC Tools used in the workspace.

#	Size	Plated	Final Size	Drl.Cnt	Mill.Cnt	Comments	Sort
1	110.00	<input checked="" type="checkbox"/>	110.00	4	0		1
2	40.00	<input checked="" type="checkbox"/>	40.00	284	0		2
3	120.00	<input checked="" type="checkbox"/>	120.00	4	0		3
4	50.00	<input checked="" type="checkbox"/>	50.00	603	0		4
5	90.00	<input checked="" type="checkbox"/>	90.00	2	0		5
6	100.00	<input checked="" type="checkbox"/>	100.00	1	0		6
7	70.00	<input checked="" type="checkbox"/>	70.00	1	0		7
8	23.00	<input checked="" type="checkbox"/>	23.00	433	0		8
9	45.00	<input checked="" type="checkbox"/>	45.00	56	0		9
10	0.00	<input checked="" type="checkbox"/>	0.00	0	0		10
11	0.00	<input checked="" type="checkbox"/>	0.00	0	0		11
12	0.00	<input checked="" type="checkbox"/>	0.00	0	0		12
13	0.00	<input checked="" type="checkbox"/>	0.00	0	0		13
14	0.00	<input checked="" type="checkbox"/>	0.00	0	0		14
15	0.00	<input checked="" type="checkbox"/>	0.00	0	0		15
16	0.00	<input checked="" type="checkbox"/>	0.00	0	0		16

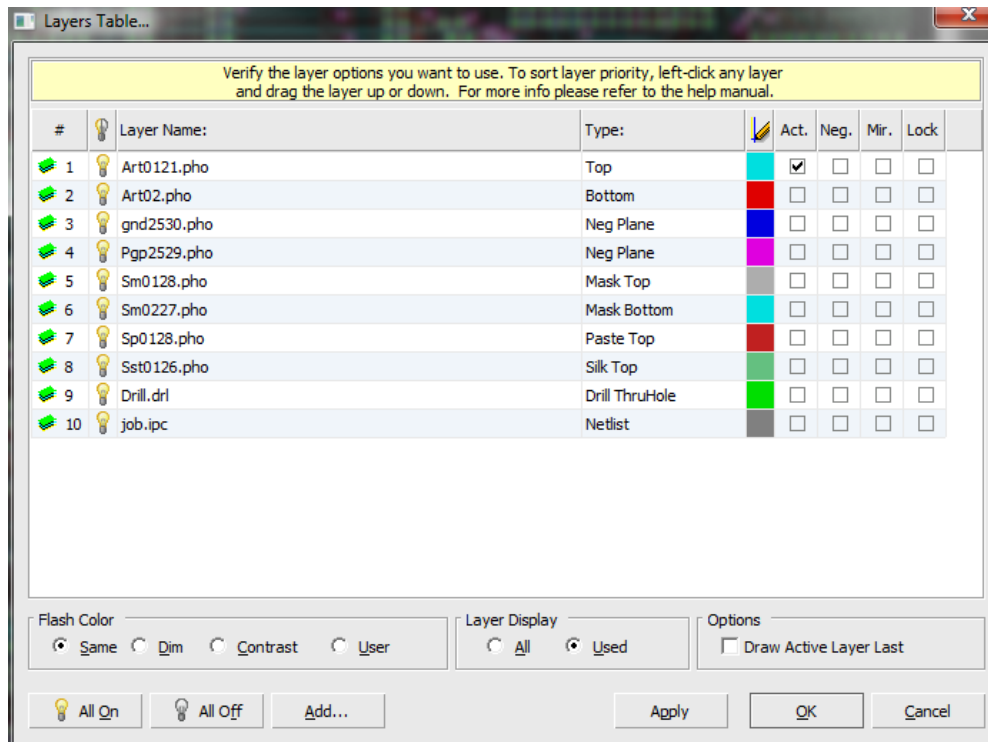
1. Select **Setup > NC Tools Table**.
2. Choose Tool Units (mil or mm) and Tool Display (All or Used Only).
3. Under the Basic and/or Advanced tabs, verify or edit settings for each NC Tool.

Notes:

- a. To edit a row double click the entry box for **Size**, **Final Size**, **Comments** and **Sort** under the **Basic** tab. Use the checkboxes for the **Plated** column.
- b. To edit a row double click the entry box for **Size**, **Comp**, **Plunge (IPM)**, **Retract (IPM)**, **Speed**, **Depth (in)**, and **Max Hits** under the **Advanced** tab.

Setup > Layer Table

Displays all layer properties for the active design. Verify, sort, and edit the layer options you want to use.



1. To activate Layer Table select **Setup > Layer Table**.
2. Verify, edit, and sort layer options.
3. Select Apply to activate edits or **OK** when finished.

Layer Table Overview

#: FAB 3000 Layer Number.

Visibility (Light bulb): Determines if objects on the layer are visible.

- a. If the layer is visible, the bulb icon is on.
- b. If the layer is not visible, the bulb icon is off.

Layer Name: Name of the layer. Double click the entry box to edit the name.

Type: Type of layer. Double click to activate the drop down menu to change the layer type.

Draw Color: Traces and polygons on the associate layer will display that color. Click the color icon to change color.

Act: Active layer. Used when adding objects (i.e. polygons, flashes, etc.). Use the checkbox to activate.

Neg: Negative polarity. Assign a negative polarity on output for this layer. Use the checkbox to activate.

Mir: Mirror. Assign a minor attribute for this layer. Mirror will take effect on output. Use the checkbox to activate.

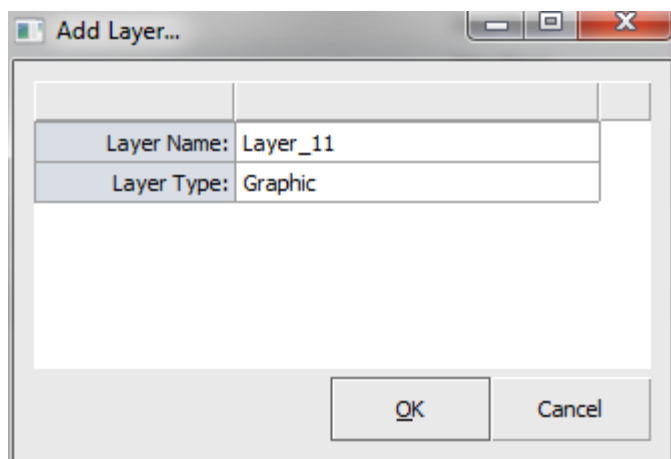
Flash Color: Select flash color. Choose from Same, Dim, Contrast, or User.

Layer Display: Display all or used layers.

Options: Check box to draw active layer last.

All On: Turns all layers on. All light bulb icons will be lit.

All Off: Turns all layers off. All light bulb icons will be unlit.

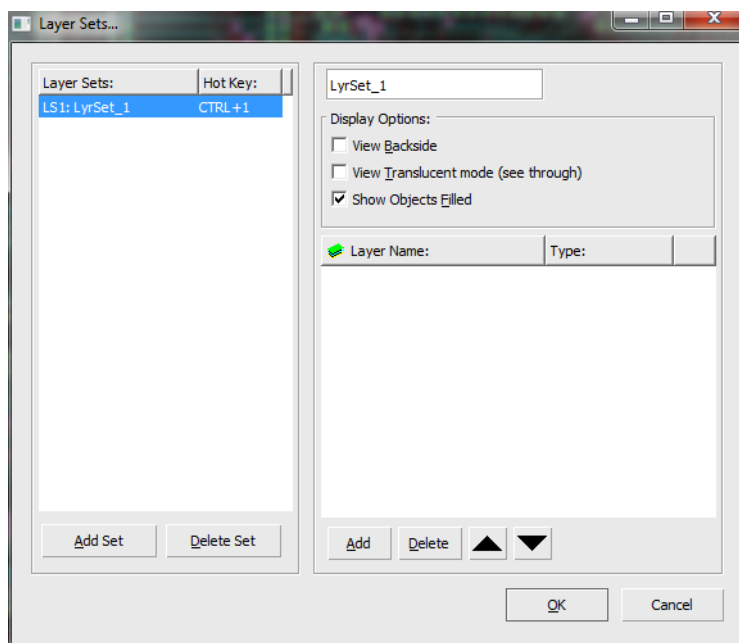


Add Layer: Adds a new layer. A dialog box will appear. Choose a layer name and layer type, then select **OK**.

Apply: Applies changes you make to layer table, but keeps layer table open for additional edits and verification.

OK: Saves changes and closes Layer Table dialog box.

Setup > Layer Sets



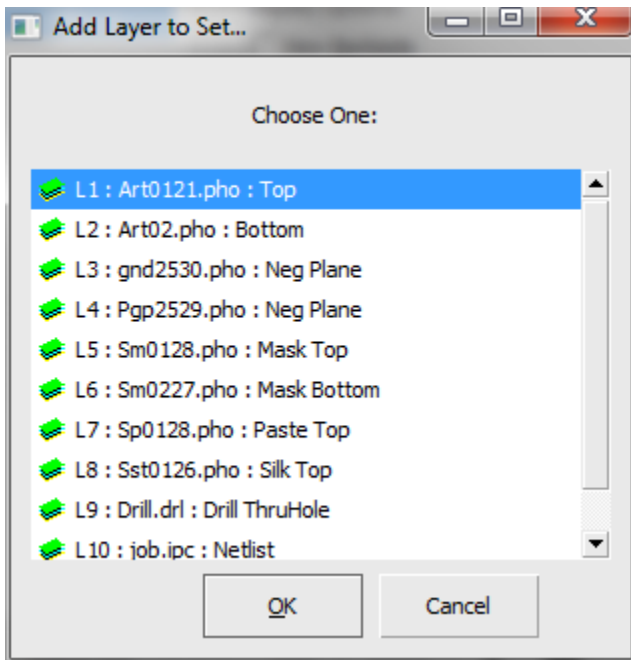
Groups together layers for a purpose. You can assign any layers and define their stack (redraw) order. The system allows you to assign layer groups to each number hot key; starting with 1 (the 0 key represents layer set 10). You can create more than 10 layer groups but only 10 can be assigned as hot keys. This is an excellent feature for quickly references sets of layers.

- a. Layer Sets require you have already imported data into the workspace Via **File > Import** or **File > Import > AutoLoad**.

Layer Sets Overview

The Layer Sets list box shows you the names of each layer set, and what ID or number key they are assigned to. If no layer sets exist, the system displays only an empty, default LyrSet_1. Only layer sets will appear in the list. (If layer sets have already been created the first available number appears.) The name of the highlighted layer set appears in Layer Name text box.

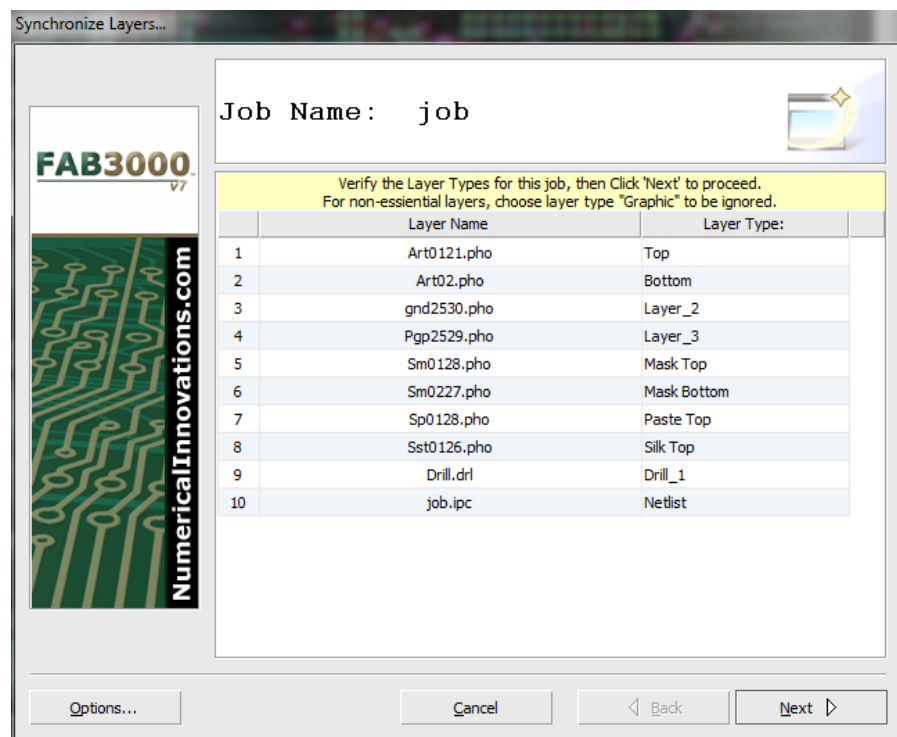
1. To add a layer set, select **Add Set**. To remove a layer set, select **Delete Set**. When adding a new layer set, a default name will appear in the text box. Type the desired layer set into the Layer Name text box or use the default name assigned. To add a layer name, choose **Add** (see #2). To delete a layer name, select **Delete**.
2. Click the '**Add**' button under the Layers list box. This opens the **Add Layer to Set** box.



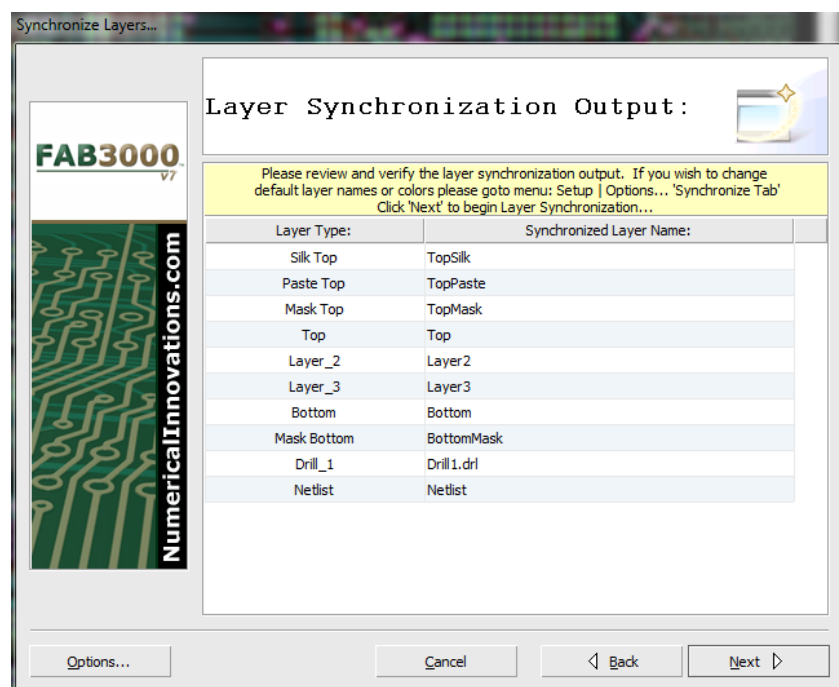
3. Choose a layer that you wish to assign to this layer group, and click OK. The name of the layer now appears in the Layers list box. When you turn ON your layer group later, the last layer in sequence will automatically be the active layer.
4. Choose **Display Options** located underneath the layer set name text box. Options include View Backside, View Translucent mode (see through), or Show Objects Filled.
5. To change the sequence of the layers in a set, select any layer, and use the down or up arrows to change the layer order.
6. Repeat the process until all desired layers and sequences are assigned to the set.
7. Click the OK button when you are finished. **You may turn ON any desired layer set by pressing 'Ctrl' + its associated number key** on your keyboard.
 - a. You may go back into the Layer Sets dialog box and add more user layer sets, or modify existing ones. After you have created layer sets for a particular design, that information is saved in your workspace so that the sets are accessible for future use.

Setup > Synchronize Layers

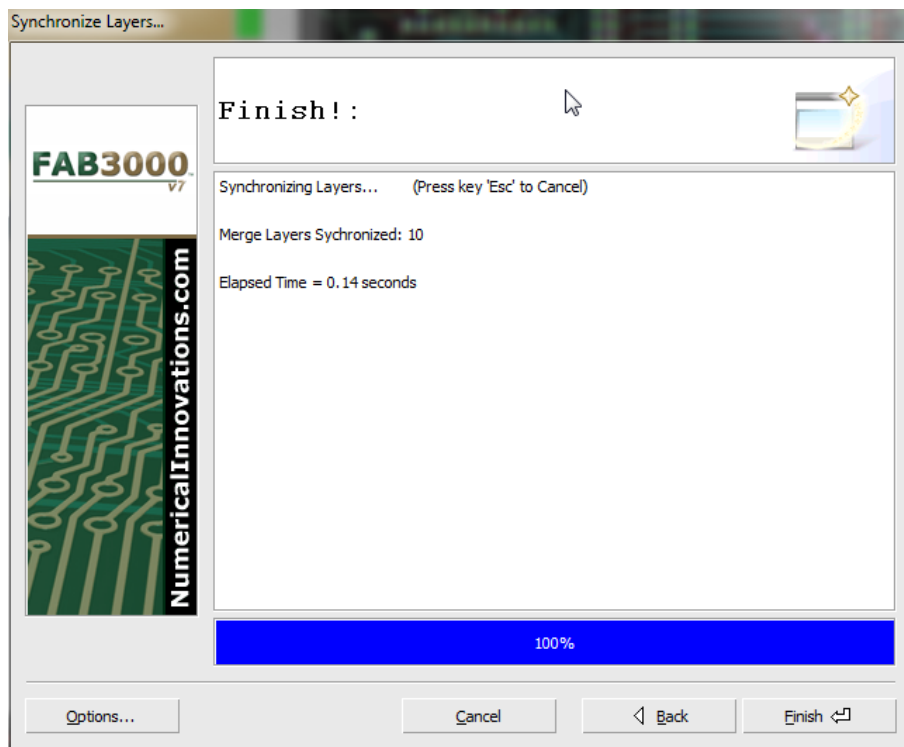
Merges all Job Layers into a common set of layers. When merging different Jobs together, chances are that each of those Jobs used a different name to describe the same layer types (i.e. "top02.pho", "topside.gbr", "art01.ger"). Synchronize Layers combines those layers with same types into one layer that can then be easily exported.



1. To synchronize layers, select **Setup > Synchronize Layers**.
2. Define all layer types for the child Job. Select **Next**.
 - a. Synchronize Layers requires that you define the layer the layer types for all Jobs. If unsure, choose the layer type called "Graphic" to be ignored.



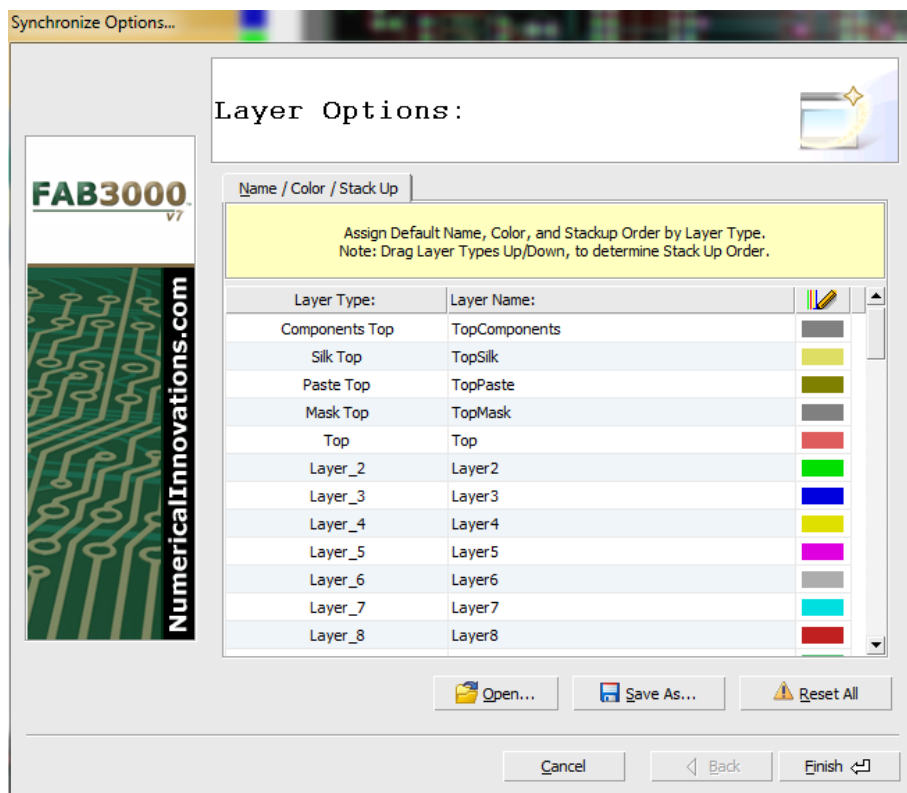
3. Review and verify the layer synchronization output.
4. Select **Next**.
5. Layer Synchronization will begin.



6. Repeat Step #2 for each child Job.
 - a. Once you have defined all layer types for child Jobs, **Synchronize Layers** will begin to merge the layers into one common set of layers.
7. Select **Finish** when complete.

Setup > Synchronize Options

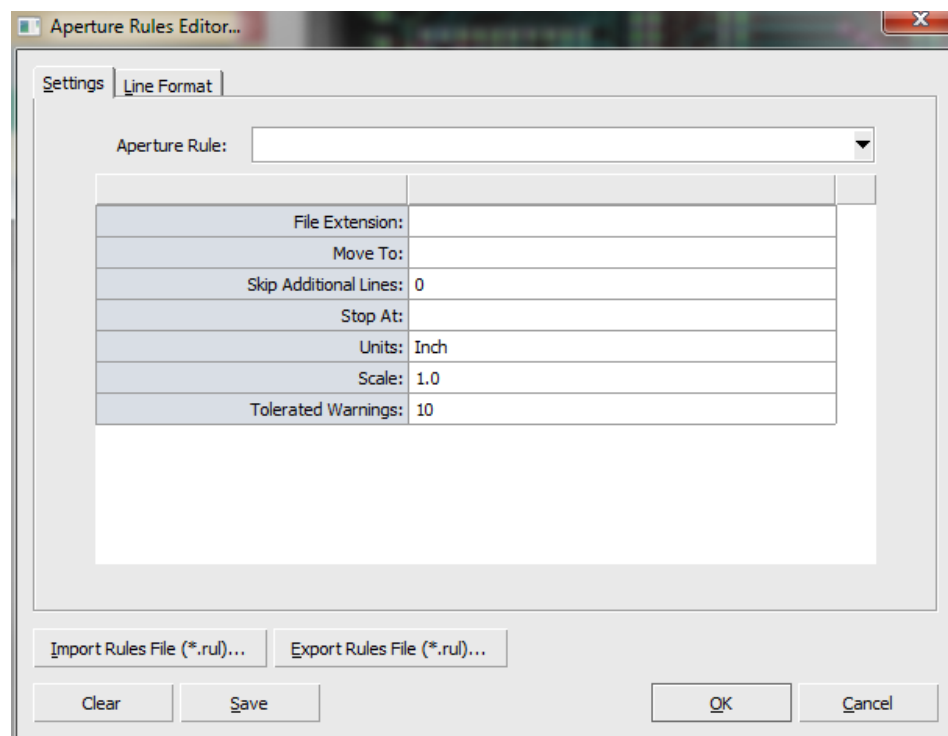
Allows you to assign default name, color, and stack up order (by layer type).



1. Select **Setup > Synchronize Options**.
2. Drag layer types up or down to determine stack up order.
3. Double click **Layer Name** to edit the name.
4. When complete, select **Finish**.
 - a. Use **Open** to open layer technology files from your computer.
 - b. Use **Save As** to save layer technology files to your computer
 - c. Use **Reset All** to reset layer synchronization options to default settings.

Setup > Aperture Rules Editor

Create custom “rules” to load any aperture list file.



1. Select **Setup > Aperture Rules Editor**.
2. Verify and edit rules using the **Settings** and **Line Format** tab (below).

Aperture Rules Editor Settings Tab

Aperture Rule: This drop down contains all available aperture rules. Each aperture rule is stored as a file (*.rul) in the FAB 3000 sub-folder "aptrules".

File Extension: Aperture files generated from a CAD system usually have a defined file extension. This helps FAB 3000 determine which Aperture rule to use during Gerber import.

Move To: Aperture files generally have non-essential text at the beginning (such as part number, time created, etc.). This lets FAB 3000 ignore all text lines until the defined "Move To" text is found. If left empty, FAB 3000 will start reading apertures at the first line.

Skip Additional Lines: This allows FAB 3000 to additionally skip lines after Move To.

Stop At: Tells FAB 3000 to stop reading aperture lines once this text has been detected.

Units: Units for aperture sizes. Choose from: Inch, Mil, or Millimeter.

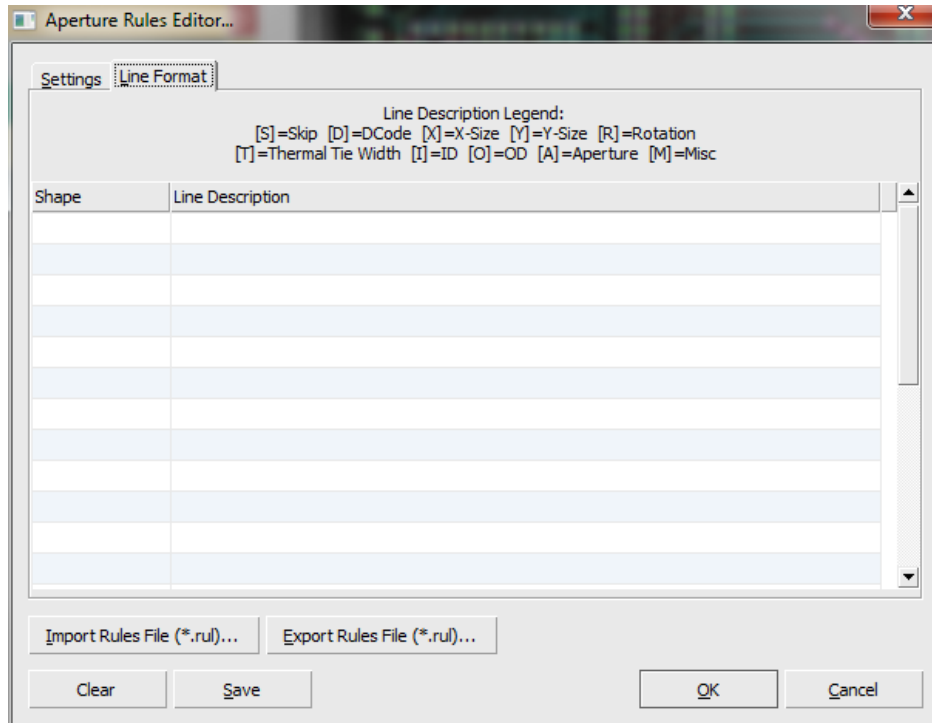
Scale: Scale value used for aperture sizes. 1 is default.

Tolerated Warnings: Determines how many warnings (per Aperture Rule File) FAB 3000 will allow before choosing to use another aperture rules files.

Import Rules File (*.rul): Imports aperture rules file.

Export Rules File (*.rul): Exports aperture rules file.

Aperture Rules Line Format Tab



FAB 3000 includes over 24 default Aperture Rules. Reviewing these default Aperture Rules is a great way to determine how parameters are used to create a successful Aperture Rules File.

Line Parameters:

[S] = Skip [M] = Misc

[D] = Dcode

[X] = X-Size

[Y] = Y-Size

[R] = Rotation

[T] = Thermal Tie Width

[I] = ID

[O] = OD

Line Example #1:

RECTANGLE 100.5 20 Flash D100

"RECTANGLE" [X] [Y] [S] D[D]

Line Example #2:

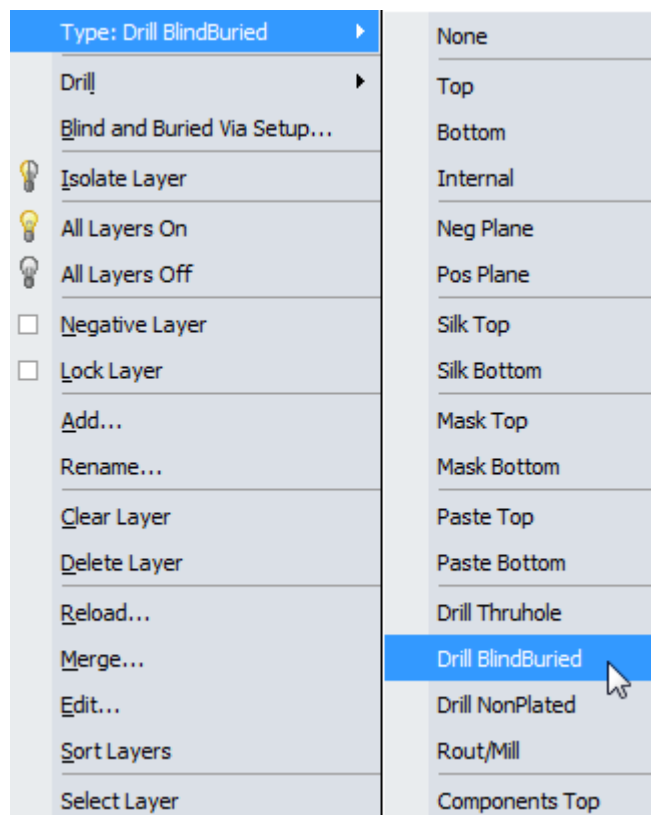
D35 Thermal 80 Yes 55

D[D] "Thermal" [O] [S] [I]

[A] = Aperture

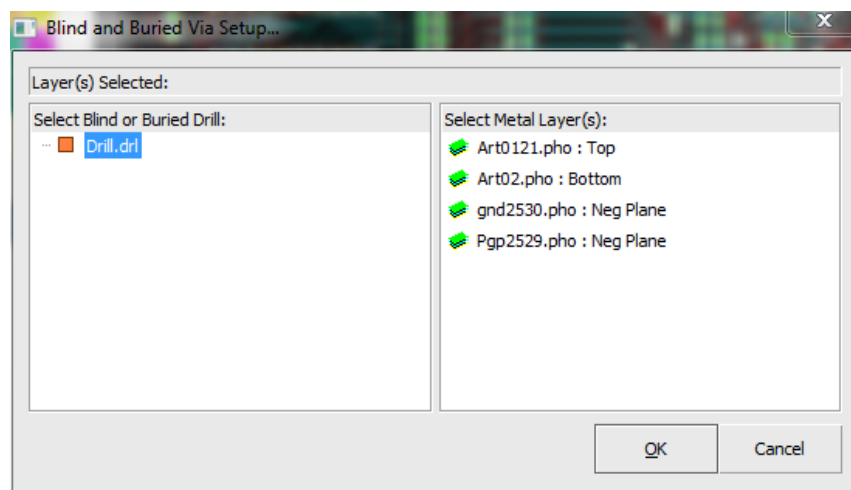
Setup > Blind and Buried Via Setup

Assign blind and buried layer group sets. In order to perform this operation, you must have at least one layer type defined as **Drill BlindBuried**. To define a layer type as Drill BlindBuried, perform the following:



1. Select the Drill layer (*.drl) you wish to define in the Layer Display Panel.

2. Right click and select **Type > Drill BlindBuried**.



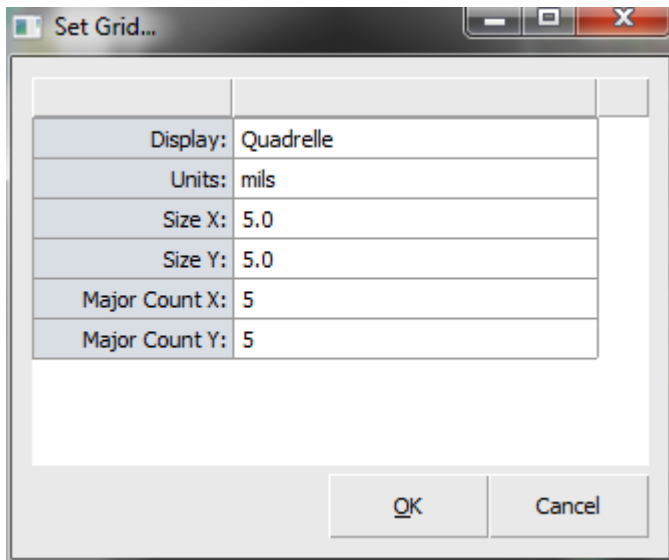
Setup Blind and Buried Via

1. Select **Setup > Blind and Buried Setup**. A dialog box will appear.
2. Select the **Blind or Buried Drill** and select the **Metal Layer(s)**.
3. Select **OK**.



[Forum: Using the Setup > Blind and Buried Via Setup Command](#)

Setup > Set Grid

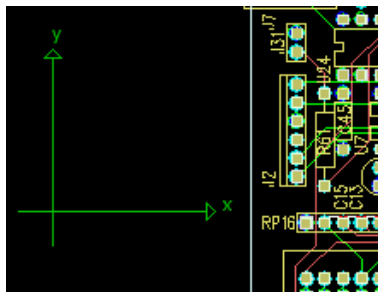


Assign grid settings.

1. Select **Setup > Set Grid**. The Set Grid dialog box will appear.
2. Verify or edit grid settings.
 - a. **Display:** Select from either Grid Off, Quadrelle, or Point using the drop down menu.
 - b. **Units:** Select from either mils or mm.
 - c. Enter **Size** for X and Y.
- d. Enter **Major Count** for X and Y.
3. When finished, select **OK**.

Setup > Set Origin

Change the location of the origin.

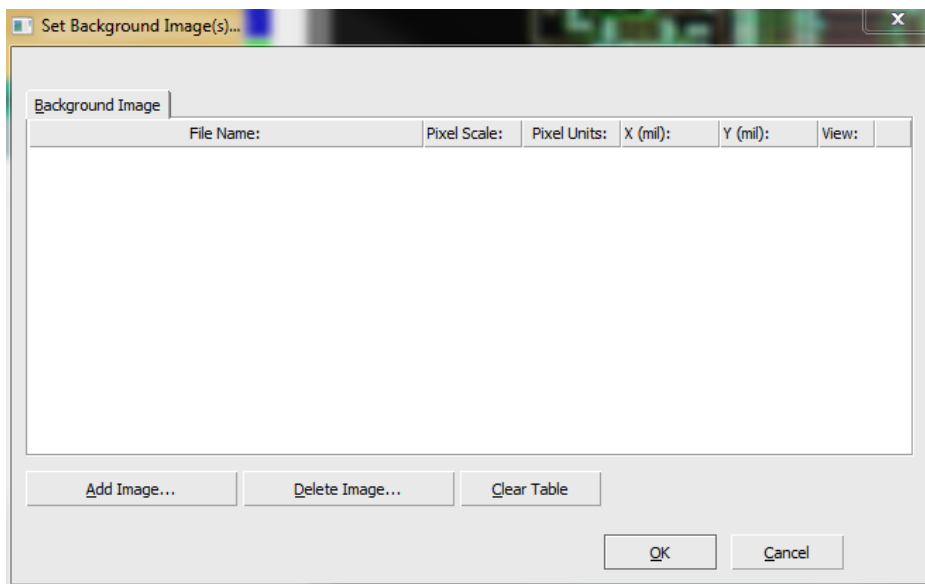


1. Select **Setup > Set Origin**.
2. Specify an origin point by left clicking the mouse within the editor.
3. An X/Y axes origin point will appear in the selected region of the editor.
 - a. You can also specify origin point by entering the coordinates of the origin point in the command line box below the editor using the format X, Y (where X and Y are coordinate values).



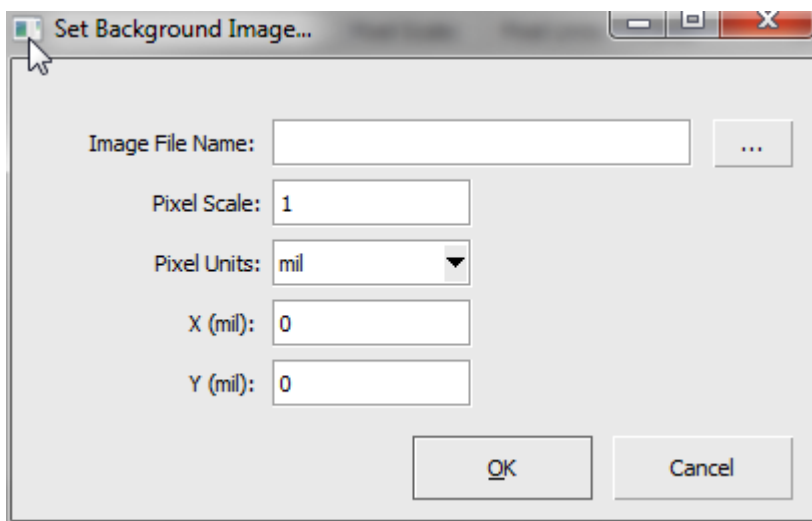
[Video: Set Origin of Gerber Files](#)

Setup > Set Background Image(s)



Assign images to be drawn over existing workspace data.

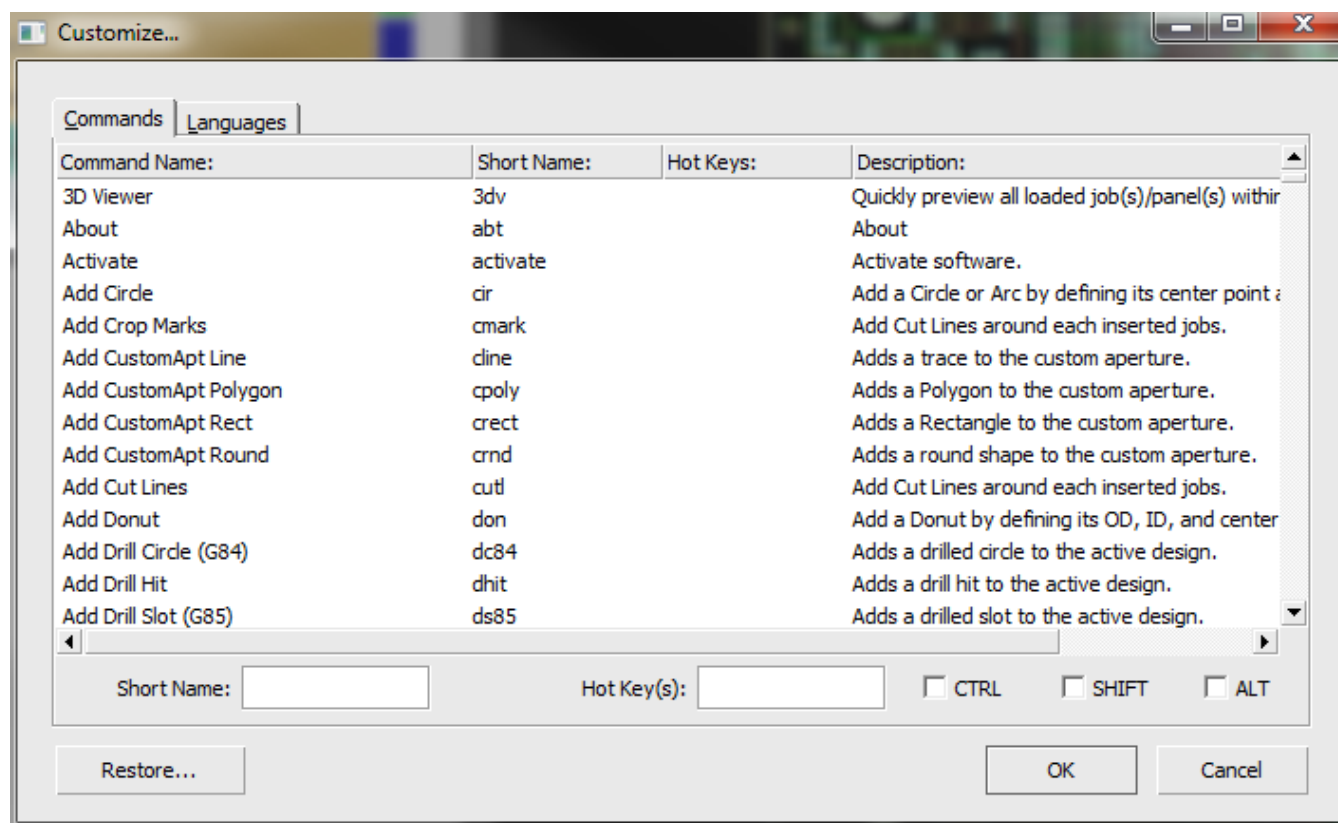
1. Select **Setup > Set Background Image(s)**. The Set Background Image dialog box will appear.
2. Select **Add Image** to insert an image. The **Add Image** dialog box will appear.



3. Choose an image to add by selecting "..." in the **Image File Name** box.
4. Enter a Pixel Scale.
5. Select Pixel Units from either **dpi, micron, millimeter, inch, or mil**.
6. Enter X and Y (mil) values.
7. Select **OK**.
8. The image will be loaded into the Set Background dialog box. To add additional images, repeat steps 2-7.
9. To delete an image, click the image file name and select **Delete Image**.
10. To delete all images, select **Clear Table**.

Setup > Customize

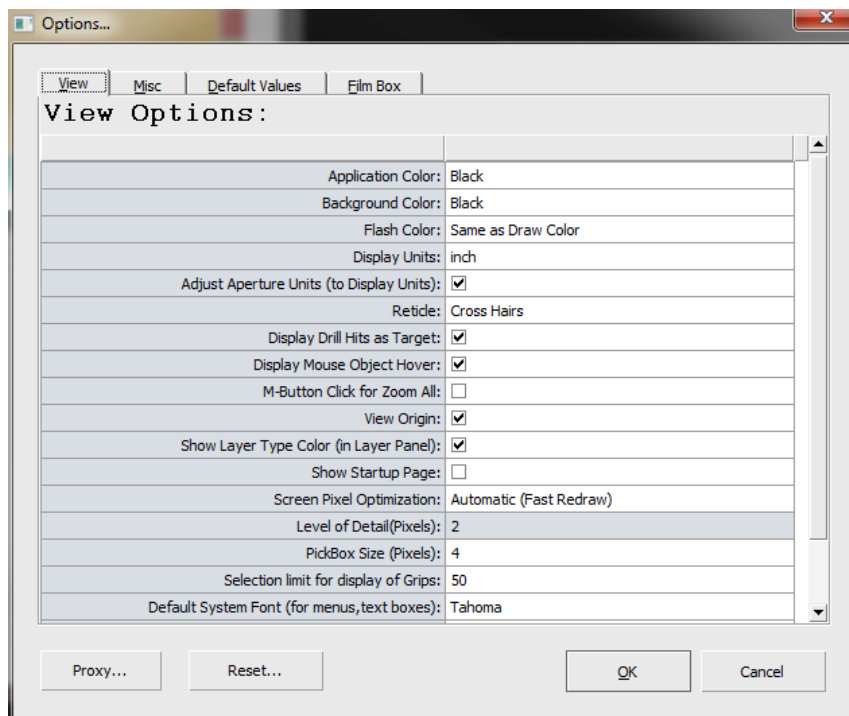
Customize FAB 3000 user commands.



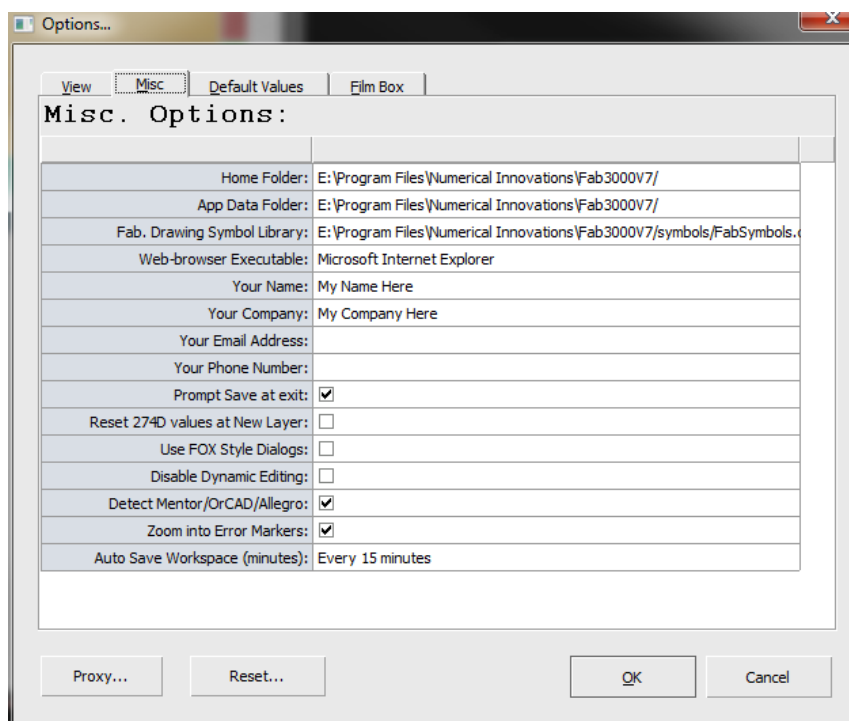
1. Select **Setup > Customize Image**. The Customize dialog box will appear.
2. To edit a command, select the command name.
 - a. You can edit the Command Name by clicking the Short Name text entry box and entering a new command name.
3. Click the Hot Key(s) text entry box to make it active.
4. Select a new Hot Key which will appear in the text entry box.
 - a. Use the **CTRL**, **SHIFT**, or **ALT** checkboxes to add an additional hot key to that command. For example, a new hot key would be the letter A while pressing down on the **SHIFT** key.
5. Select **OK** when finished. The commands entered will be saved for future use.
 - a. To restore to FAB 3000 default user commands, select **Restore**.
 - b. Use the Languages tab to customize the default command language.

Setup > General

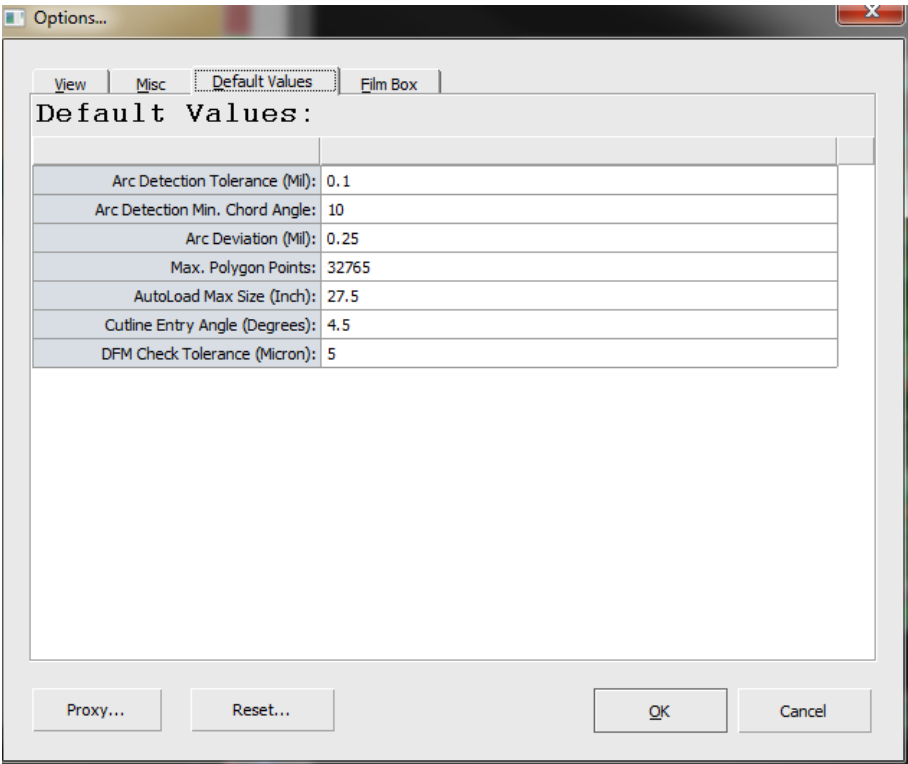
General settings and preferences. There are four tabs: **View**, **Misc**, **Default Values**, and **Film Box**.



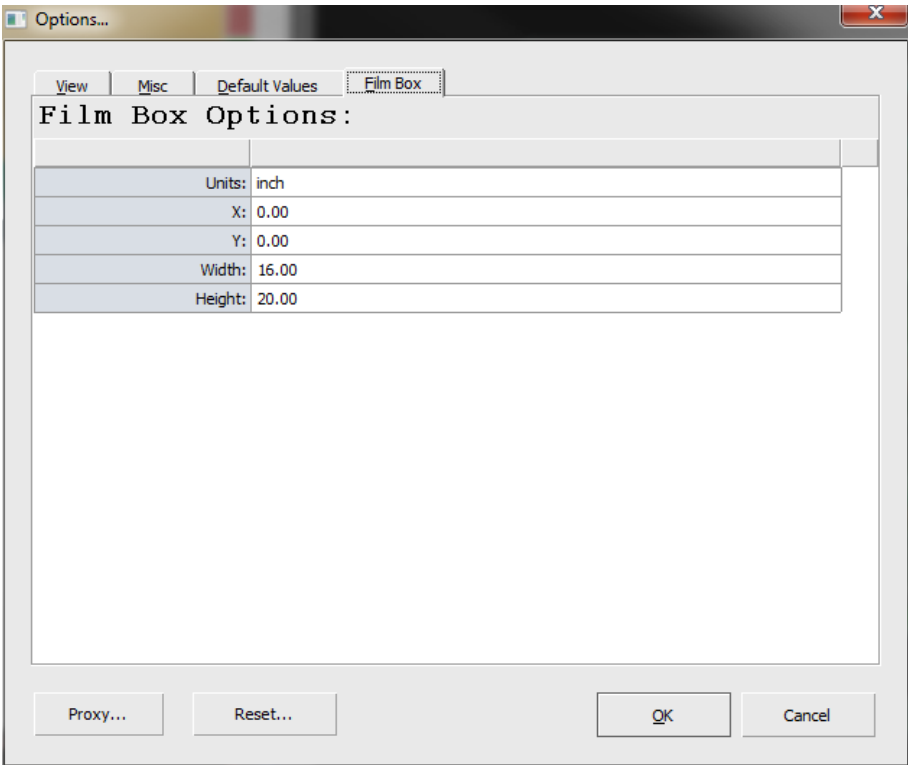
View Options: Used for general view settings such as units, reticle type, reference colors, fonts, and more.



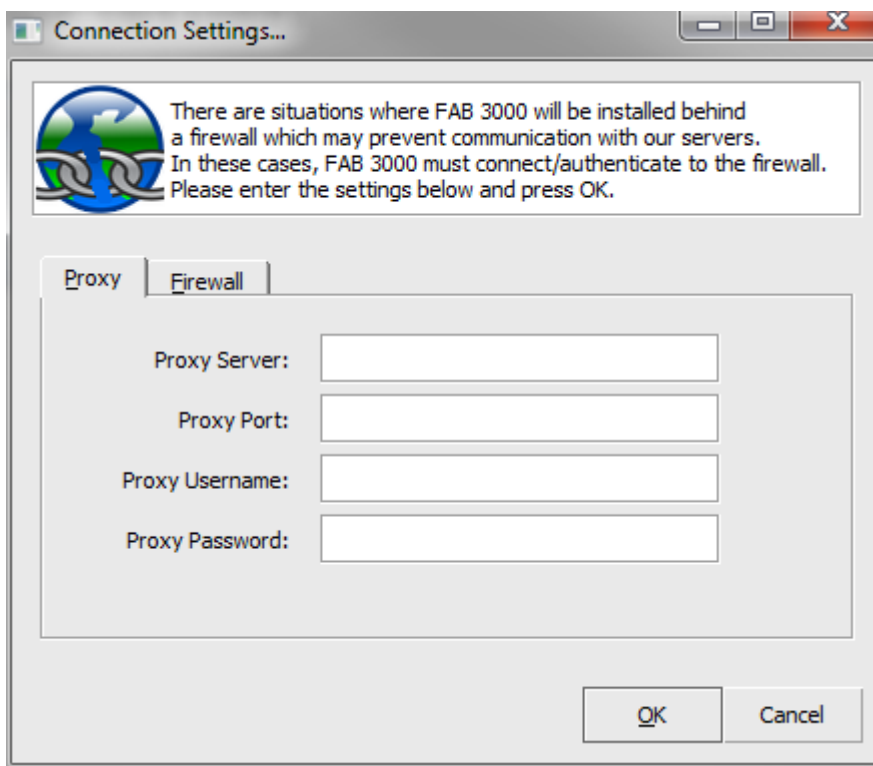
Misc: Miscellaneous options such as default folders, contact information, and auto save settings.



Default Values: Default values settings for tolerance, angles, points, and more.



Film Box: Default film box options.



The image shows a Windows-style dialog box titled "Connection Settings...". It features a message at the top with a globe icon and a warning text. Below the message are two tabs: "Proxy" and "Firewall". The "Proxy" tab is selected, showing four input fields for "Proxy Server:", "Proxy Port:", "Proxy Username:", and "Proxy Password:". At the bottom right are "OK" and "Cancel" buttons.

Connection Settings...

There are situations where FAB 3000 will be installed behind a firewall which may prevent communication with our servers. In these cases, FAB 3000 must connect/authenticate to the firewall. Please enter the settings below and press OK.

Proxy Firewall

Proxy Server:

Proxy Port:

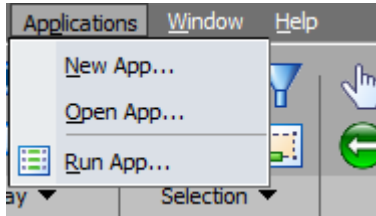
Proxy Username:

Proxy Password:

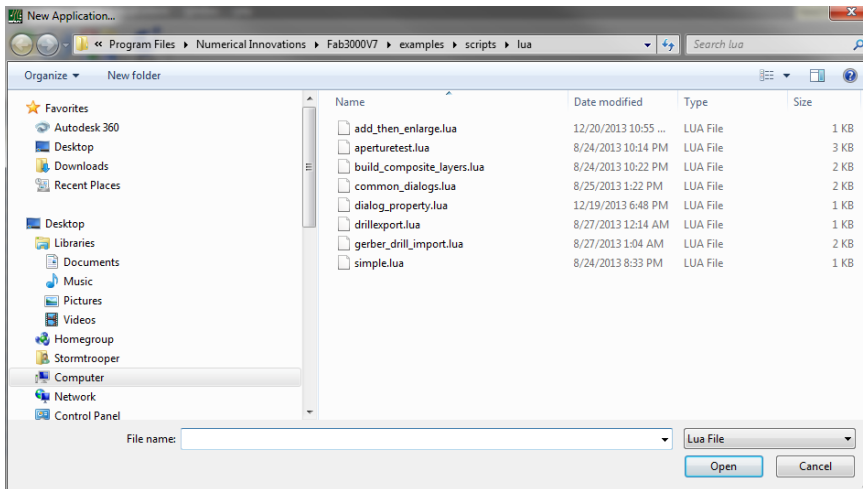
OK Cancel

Proxy (Connection Settings): If FAB 3000 is installed behind a firewall, it may prevent communication with update servers. Use these options to configure, connect, and authenticate internet security settings.

5.11 Applications Menu



Used to create, open, or run custom application files. Apps are created using FAB 3000's LUA Scripting Engine and are in *.lua format.



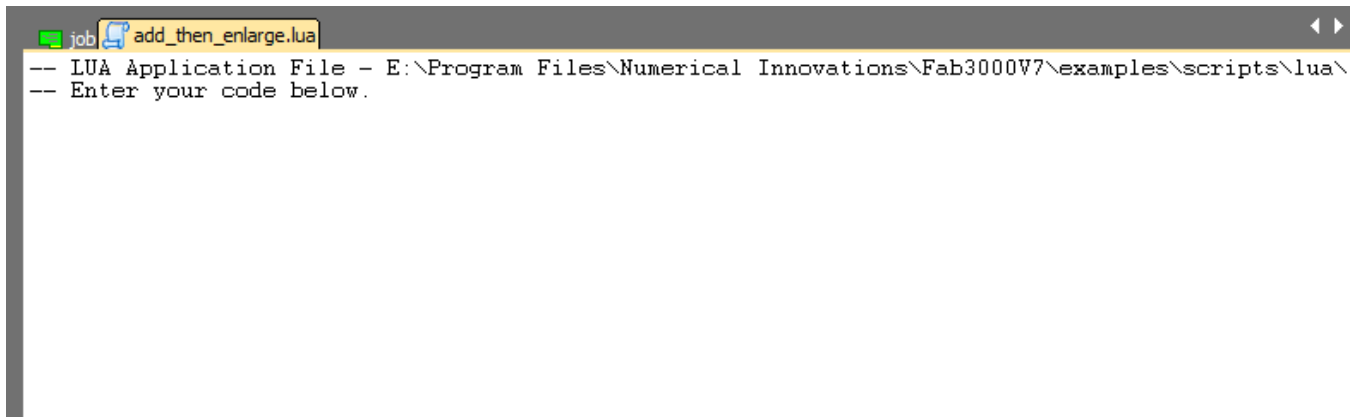
New App

Creates a new application file. This assumes you already have a blank LUA script application file created. Use notepad or text editor and save the file in *.lua file format.

1. Select **Applications > New App**.
2. Select the *.lua file you've created from the windows dialog

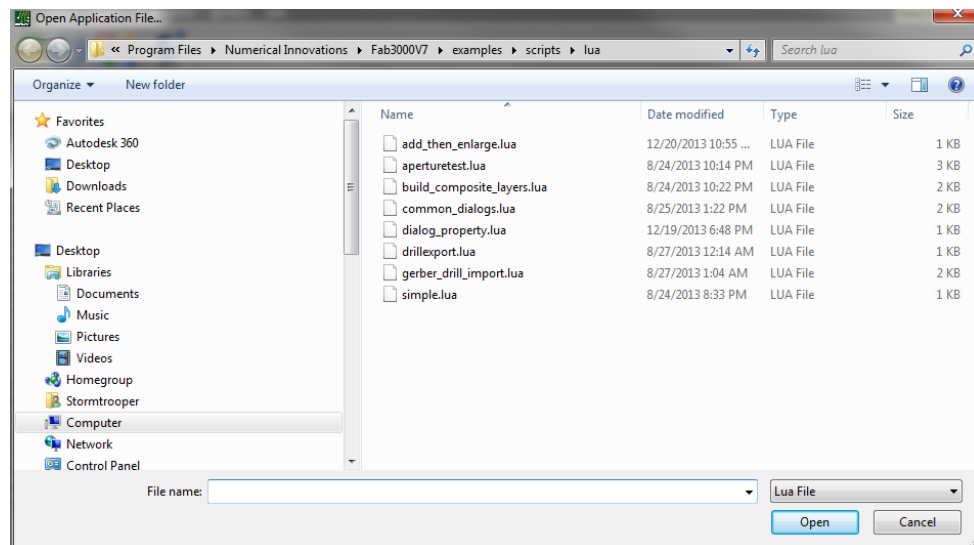
box. Select OK.

3. The LUA application file editor you created will open in a new tab.



4. Enter the code required for your LUA application.

5. Right click on the tab's title then Save your *.lua file.

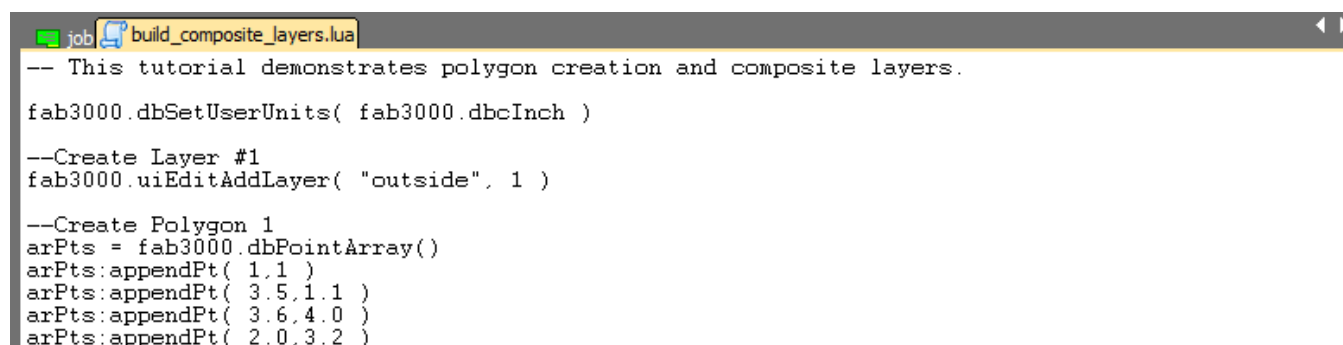


Open App

Opens an existing *.lua application file.

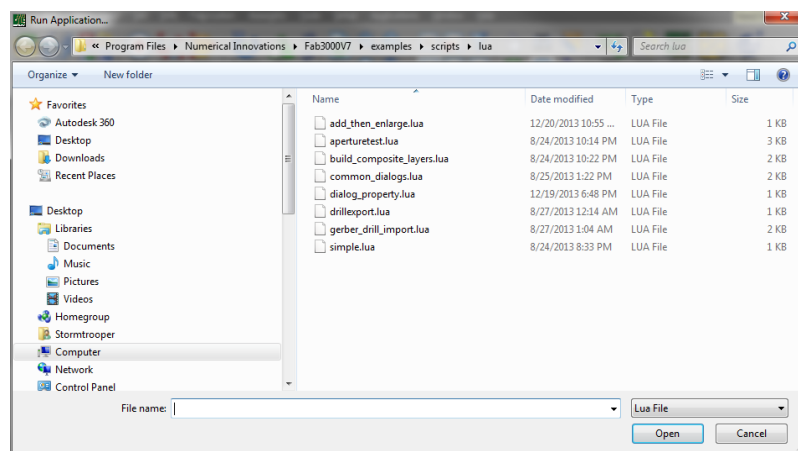
1. Select **Applications > Open App**.
2. Select the *.lua file you've created using notepad or text editor. Select **OK**.

3. The LUA application file editor will open in a new tab with your opened file.



4. Edit the *.lua file which opened in a new tab.
5. Right click on the tab's title, then Save your *.lua file edits.

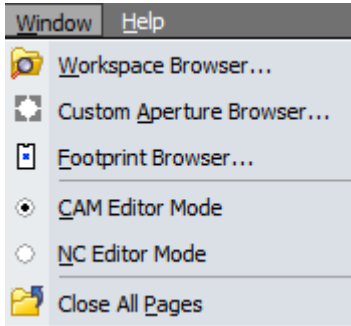
Run App



Runs a *.lua application file.

1. Select **Applications > Run App**.
2. Select the LUA application file you would like to run.
3. Select **Open**.
4. The LUA application will begin.

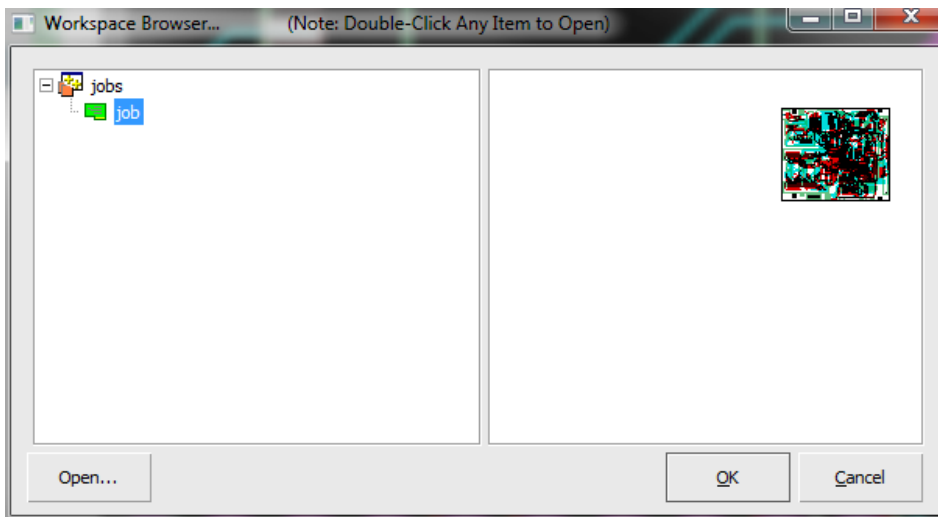
5.12 Window Menu



The Window menu enables various browsers and edit modes.

Window > Workspace Browser

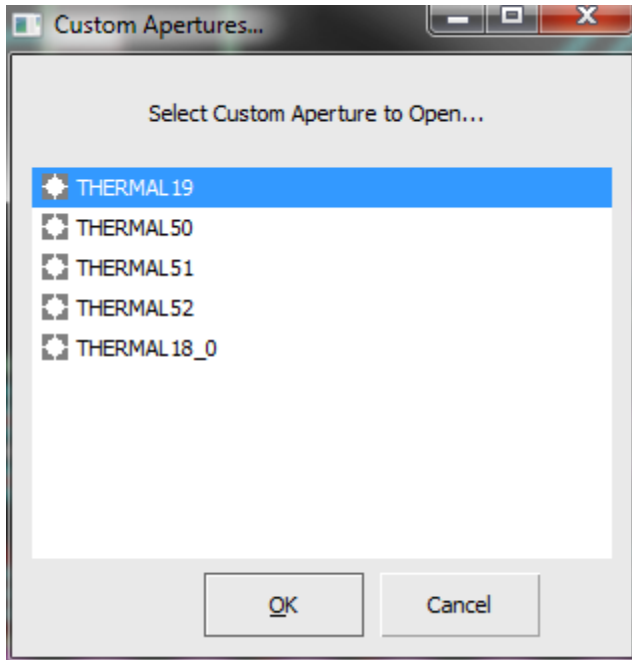
Quickly preview all loaded Job(s)/panel(s) within the workspace without having to open each on into the editor.



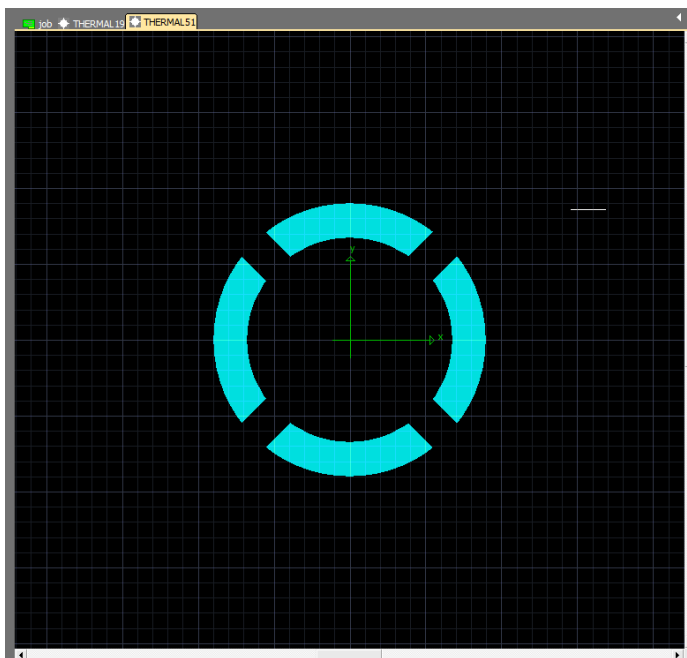
1. Select **Window > Workspace Browser**.
2. The Workspace Browser dialog box will appear with a directory tree of your current Jobs.
 - a. A preview of each Job will appear toward the right side of the dialog box.
3. Double click a Job or select Open to open the Job within the workspace.
4. The selected Job will open as a new tab in the editor.

Window > Custom Aperture Browser

Quickly preview all loaded custom apertures within the workspace without having to open each one into the editor.



1. Select **Window > Custom Aperture Browser**. The Custom Aperture Browser dialog box will appear.
2. Choose the Custom Aperture to open.
3. Select **OK**.



4. The Custom Aperture will open as a new tab in the editor.

Window > Footprint Browser

Requires footprints are defined in the workspace. Allows you to view whatever available footprints are active within FAB 3000 (generated or imported).

Window > CAM Editor Mode

Switches to CAM Editor Mode.

Window > NC Editor Mode

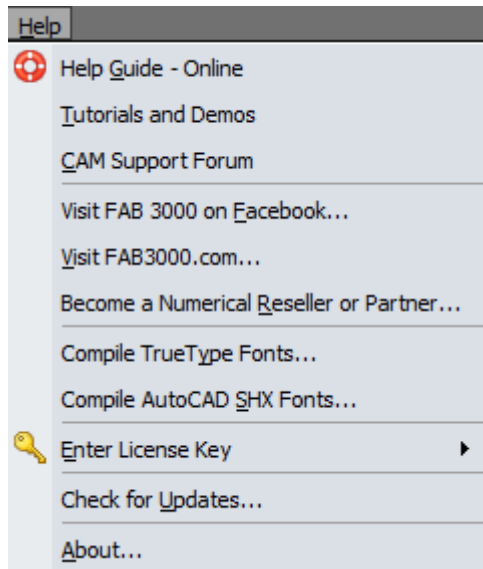
Switches to NC Editor Mode.

Window > Close All Pages

Closes all opened pages in the editor.

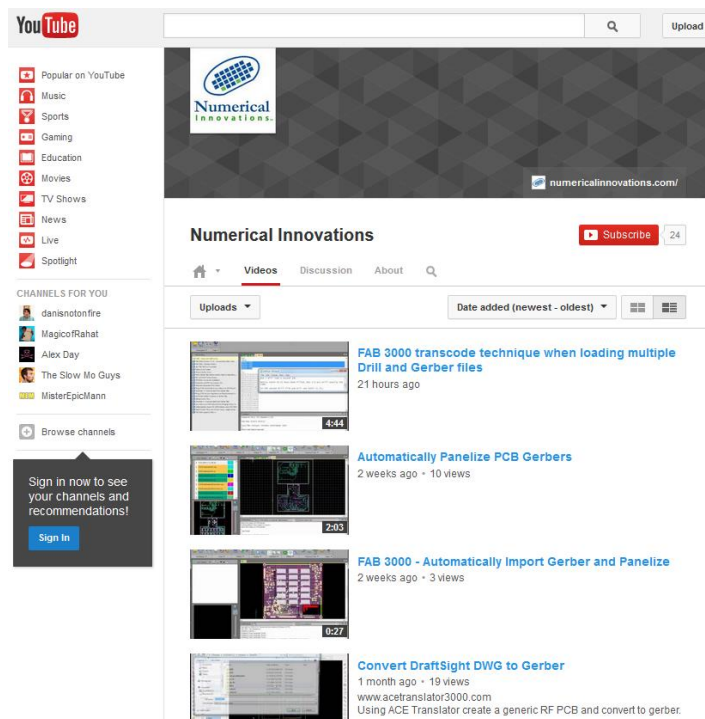
5.13 Help Menu

FAB 3000 offers a number of resources for help, troubleshooting, and additional configuration.



Help > Help Guide – Online

Links to FAB 3000 online help manual.



Help > Tutorials and Demos

Links to the FAB 3000 YouTube Channel with dozens of video tips and resources.

Welcome to the Numerical Innovations Support Forum!

[Register](#)
[Login](#)
[Search](#)
[New Posts](#)
[Chat](#)

Numerical Innovations Website > Forums > **FAB 3000 - Gerber/CAM Software**

Search This Forum

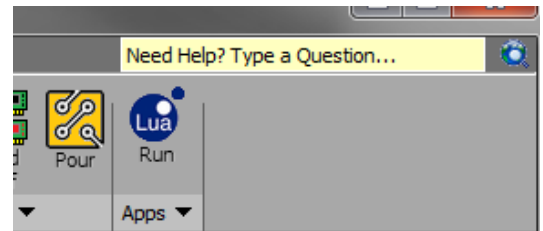
New Topic

Subject	Author	Views	Replies	Last Post
★ Every time when I try paneizing the boards seem to be placed wrongly; could you please help me	Admin	14	0	21 hours ago by Admin
★ When I open more then one gerber at the same time in FAB3000 V7 the first one can not be exported anymore.	Admin	15	0	21 hours ago by Admin
★ I'm trying to panelize two different layouts. The tool numbers in the drill files are not the same for the same drills	Admin	14	0	21 hours ago by Admin
★ Hot Key Zoom to Cursor	BSU	30	1	12/17 by Admin
★ Reg: DFM Now not opening in Allegor 16.3 --- Error message attached	Aadhi	55	0	12/10 by Aadhi
★ How to update from V6 to V7?	maranjo	130	1	12/02 by Admin
★ FAB 3000 (Version 7) has been released!	Admin	124	0	11/25 by Admin
★ Merging Gerber Files	Admin	111	0	11/21 by Admin
★ Can you give me a hint on the best route to start learning the ins and outs of using FAB3000?	Admin	89	0	11/21 by Admin
★ Read in a gerber file in Fabit and some data is reverse tone in Fabit	advancerep	124	1	11/15 by Admin
★ When I configure the unit to inches, I still get readings in mils	Admin	132	0	11/08 by Admin

Help > CAM Support Forum

Links directly to the Numerical Innovations support forum at www.numericalinnovations.com. Use this forum for questions, tips, and support help. The FAB 3000 community users are available to answer beginner and veteran questions.

Need Help? Type a Question in FAB 3000!



Use this question box in the top right corner of the FAB 3000 workspace as a shortcut to the Numerical Innovations support forum.



facebook

Email or Phone Password Log In

Keep me logged in Forget your password?

FAB 3000 is on Facebook.

To connect with FAB 3000, sign up for Facebook today.

Sign Up Log In

FAB 3000

525 likes

Software

Advanced DRC/DFM checks, edit Gerber/ODB++ files, Compare Nets, Panelize PCB's, generate Part Centroid files, and perform many other useful features. For more information visit: <http://www.numericalinnovations.com/collections/fab-3000-gerber-cam>

About Photos Likes Download Free Trial

Highlights

FAB 3000 shared a link.

November 24 near Honolulu, HI

FAB 3000 Version 7 has been released!

<http://www.numericalinnovations.com/collections/fab-3000-gerber-cam>

Gerber Viewer, Editor, Panelize, DFM, Build Centroids & more

www.numericalinnovations.com

Nowadays, it's critical for every PCB designer to use CAM software to

Likes

Numerical Innovations Computers/Technology

ACE Translator 3000 Software

FAB 3000 shared a link.

October 1 near Honolulu, HI

Generate PCB Centroid file (used in PCB assembly) from Gerber using FAB 3000

Download Free Trial: www.fab3000.com

Like Comment Share

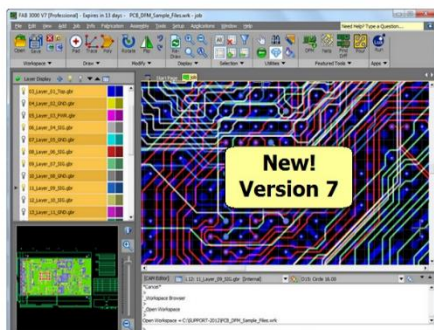
Help > Visit FAB 3000 on Facebook

Stay tuned for important updates, tips, and links on the FAB 3000 Facebook page.

FAB 3000 V7: Gerber Viewing, Editing, Panelization, DRC, DFM, Compare Nets, Build Centroids, DXF, ODB++ and more. Download Trial.

Nowadays, it's critical for every PCB designer to use CAM software to analyze and prepare their Gerber, DRL & ODB++ files for manufacturing. FAB 3000 is a "Full-Featured" CAM software package targeted towards anyone who would like to have "high-end" CAM software to run DRC/DFM verification, edit and output data, compare nets, merge gerber files, write customized LUA applications, and perform many other useful features. FAB 3000 is by far the best value in the PCB industry, and it has ALL the power of those other CAM tools costing thousands of dollars for a fraction of the price.

"The Most Easy to Use Gerber Viewer, Editor & DFM Tool Available."



Download Now

Runs On Windows Linux

- FAB Revision History
- Advanced DFM Checks(PRO)
- Features Comparison Grid
- Help Guide - PDF Version
- FAB 3000 - User Forum
- FAB 3000 Online Tutorials
- Visit FAB 3000 on Facebook

Help > Visit FAB 3000.com

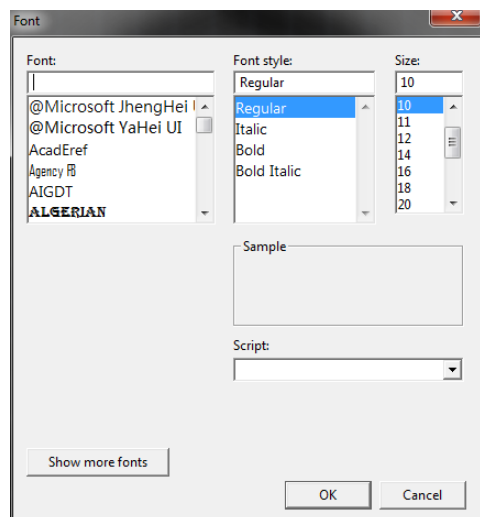
Directly links to the FAB 3000 home page.

From here, you can access:

- Downloads
- Feature Comparison
- PDF Help Guide
- FAB 3000 User Forum
- FAB 3000 YouTube Page
- FAB 3000 Facebook Page

Help > Become a Numerical Reseller or Partner

Want to be a Reseller or Partner with Numerical Innovations? Select this menu option for more information on Numerical Innovations partnerships and alliances.



Help > Compile TrueType Fonts

TrueType Font selector for FAB 3000.

Help > Compile AutoCAD SHX Fonts

Import and add AutoCAD SHX Fonts to the workspace.

Help > Enter License Key

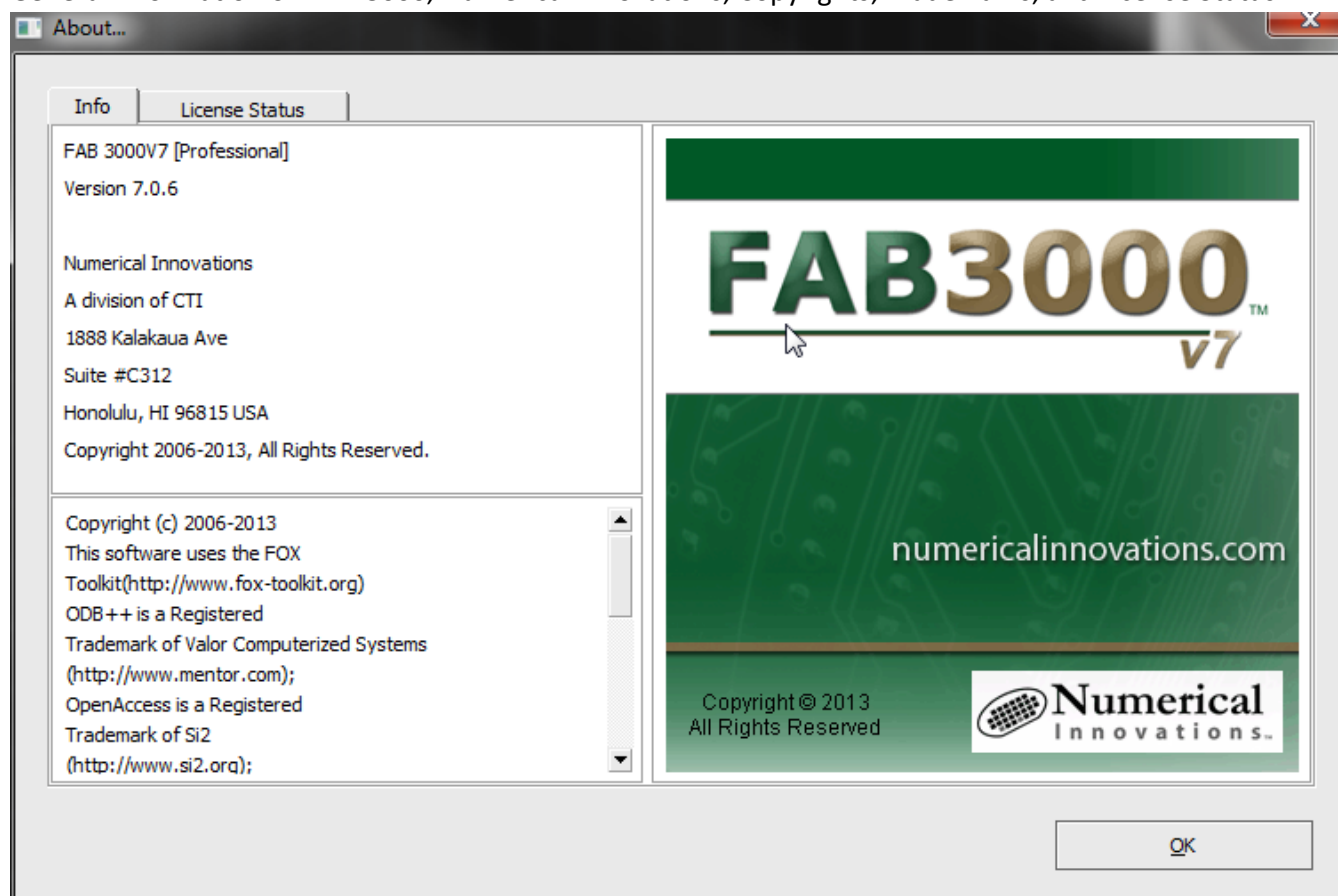
Enter FAB 3000 license keys. For more information on licensing and registration, see section **1.3: Understanding Licensing & Registration**.

Help > Check for Updates

This feature will send you to the Numerical Innovations web page for FAB 3000 revision history.

Help > About

General information on FAB 3000, Numerical Innovations, Copyrights, Trademarks, and License Status.



SECTION 6: IMPORT MATRIX FILE

The FAB 3000 **Import Matrix File** permits external tools (like PCB Design software) to predefine layers and Jobs. This creates intelligent loading with FAB 3000 for instant Gerber Viewing & Verification. Import Matrix File prevents the monotonous steps of loading Gerber & drill files, assigning layer colors & layer types, defining layer stack ups, and defining blind/buried drill layers (if required). It allows PCB designers to easily use Gerber/CAM software for its primary purpose - to quickly verify that Gerbers & drills are correct and ready for manufacturing.

The FAB 3000 Import Matrix file also supports PANEL arrays which allow the merging of several PCB Jobs together so they can be manufactured simultaneously. This creates a significant savings in fabrication costs. In addition, FAB 3000 will automatically performs all necessary object transformations, Dcode, & drill tool transcoding.

6.1 Import Matrix File Overview

Simply call FAB 3000 and pass the matrix file as the first argument.

*All FAB 3000 import matrix files must end with *.fm6.*

RULE #1: LAYER arrays are always defined first.

RULE #2: JOB arrays are defined after LAYER arrays.

RULE #3: PANEL arrays are defined last.

LAYER {

TYPE=MASK_TOP

NAME=smt

START_DRILL=

END_DRILL=

DRAW_COLOR=223,100,100

FLASH_COLOR=223,100,100

}

LAYER {

TYPE=TOP


```
NAME=layer_1

START_DRILL=

END_DRILL=

DRAW_COLOR=100,223,0

FLASH_COLOR=100,223,0

}

...

JOB {

    NAME=test-odb++-Gerber

    LAYER=layer_1,C:\temp\examples\274X\Art0121.pho

    LAYER=layer_2,C:\temp\examples\274X\Art02.pho

    LAYER=layer_4,C:\temp\examples\274X\Pgp2529.pho

    LAYER=layer_3,C:\temp\examples\274X\gnd2530.pho

    LAYER=sst,C:\temp\examples\274X\Sst0126.pho

    LAYER=spt,C:\temp\examples\274X\Sp0128.pho

    LAYER=smb,C:\temp\examples\274X\Sm0227.pho

    LAYER=smt,C:\temp\examples\274X\Sm0128.pho

    LAYER=d_1_2,C:\temp\examples\274X\Drill.drl

}

JOB {

    NAME=board1

    LAYER=smb,C:\temp\examples\Board1\Bottom Mask.gbr

    LAYER=layer_4,C:\temp\examples\Board1\Bottom.gbr
```

```

LAYER=smt,C:\temp\examples\Board1\Top Mask.gbr

LAYER=layer_1,C:\temp\examples\Board1\Top.gbr

LAYER=sst,C:\temp\examples\Board1\Top Silk.gbr

LAYER=d_1_2,C:\temp\examples\Board1\drill.drl
}

...

PANEL {

    NAME=mypanel

    UNITS=INCH

    WIDTH=18

    HEIGHT=18.5

    STEPREPEAT=board1,17.0,4,3.8,3.4,2,5,90,No

    STEPREPEAT=test-odb++-Gerber,17.85,11.5,7,6,1,3,90,No
}

...

```

Fields in the LAYER array

TYPE	Describes the purpose of the layer and must be one of these defined layer types.
NAME	The name of the layer. This will be directly referenced in the JOB array.
START_DRILL END_DRILL	These fields are only used for drill layers. They specify the span of the drill, in case it is a blind or buried Via layers. Each field must be a valid layer name. When the fields are empty, the drill is assumed to be thru hole (i.e. START_DRILL is assumed to be the first board layer and END_DRILL is assumed to be the last board layer).
DRAW_COLOR	The RGB representation of the draw color for display of the layer. Note: RGB values are separated by commas.
FLASH_COLOR	The RGB representation of the flash color for display of the layer. Note: RGB values are separated by commas.

Fields in the JOB array

NAME	The name of the Job. This will be directly referenced in the PANEL array.
LAYER	<p>This field contains the layer name and Gerber file location, separated by a comma.</p> <p>Note: The Gerber file location must be an actual location on your system and not localized path (i.e. .../Gerbers/top.gbr)</p> <p>Note: For DRILL and ROUT layer types, the file location must point to a NC file (not Gerber). The NC file must contain tool number, sizes in the header. If not, we recommend that you define a NCTOOLS array in the FAB 3000 Import Matrix before the JOB array.</p>

Fields in the PANEL array

NAME	The name of the panel. This will be the name FAB 3000 uses when it creates the panel.
UNITS	Units to use for this panel array. Must be either: INCH or MM
WIDTH	Panel Width. Note: The value is defined in the panel units.
HEIGHT	Panel Height. Note: The value is defined in the panel units.
STEPREPEAT	<p>This field contains the Job Name, X and Y Origin, X and Y Spacing, X and Y Count, Job Angle (must be 0, 90, 180, 270), and Mirror (must be either: YES or NO) all separated by a commas.</p> <p>Note: Origin and Spacing values are defined in the panel units.</p> <p>Note: If you only require a 1 up (no array) assign X and Y Spacing to 0; X and Y Count to 1.</p>

Note: Jobs must be referenced before Panel

Fields in the NCTOOLS array

UNITS	Units to use for this nctools array. Must be either: INCH or MM
INTEGER	Integer digits for drill or rout file between (0-6).
DECIMAL	Decimal digits for drill or rout file between (0-6).
TYPE	<p>Drill or rout file type.</p> <p>ABSOLUTE -- Use Absolute coordinates during NC Import</p> <p>INCREMENTAL -- Use Incremental coordinates during NC Import</p>

SUPPRESSION	Drill or rout file zero suppression. LEADING -- Suppress Leading Zeros during NC Import TRAILING -- Suppress Trailing Zeros during NC Import NONE -- No Zero Suppression during NC Import
TOOL	This field contains the Tool # (between 1 to 1000), Diameter (in Tool Units), and Plated (either: YES or NO); all separated by a commas.

Note: NCTOOLS array is optional. It is only intended for NC files that don't include tool definitions in the header. Must be called before Job

- Comment Line (Ignored by FAB 3000). Any line that begins with the pound character (#) will be ignored by FAB 3000.

6.2 Example Matrix File #1

This sample FAB 3000 Matrix file defines several layers and creates a single Job.

Note: All FAB 3000 Import Matrix Files must end with *.fm6.

SAMPLE FAB 3000 - MATRIX FILE

LAYER {

 TYPE=SILK_TOP

 NAME=sst

 START_DRILL=

 END_DRILL=

 DRAW_COLOR=255,0,0

 FLASH_COLOR=255,0,0

}

LAYER {

 TYPE=PASTE_TOP

 NAME=spt

```
START_DRILL=  
  
END_DRILL=  
  
DRAW_COLOR=100,223,223  
  
FLASH_COLOR=100,223,223  
  
}
```

```
LAYER {  
  
    TYPE=MASK_TOP  
  
    NAME=smt  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=223,100,100  
  
    FLASH_COLOR=223,100,100  
  
}
```

```
LAYER {  
  
    TYPE=TOP  
  
    NAME=layer_1  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,223,0  
  
    FLASH_COLOR=100,223,0  
  
}
```

```
LAYER {  
  
    TYPE=neg_plane  
  
    NAME=layer_2  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=50,100,50  
  
    FLASH_COLOR=50,100,50  
  
}
```

```
LAYER {  
  
    TYPE=neg_plane  
  
    NAME=layer_3  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,223,200  
  
    FLASH_COLOR=100,223,200  
  
}
```

```
LAYER {  
  
    TYPE=BOTTOM  
  
    NAME=layer_4  
  
    START_DRILL=
```

```
END_DRILL=  
  
DRAW_COLOR=223,100,223  
  
FLASH_COLOR=223,100,223  
  
}
```

```
LAYER {  
  
    TYPE=MASK_BOTTOM  
  
    NAME=smb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=173,173,173  
  
    FLASH_COLOR=173,173,173  
  
}
```

```
LAYER {  
  
    TYPE=PASTE_BOTTOM  
  
    NAME=spb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=192,32,32  
  
    FLASH_COLOR=192,32,32  
  
}
```

```
LAYER {  
  
  TYPE=SILK_BOTTOM  
  
  NAME=ssb  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=100,192,128  
  
  FLASH_COLOR=100,192,128  
  
}
```

```
LAYER {  
  
  TYPE=DRILL  
  
  NAME=d_1_2  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=192,192,100  
  
  FLASH_COLOR=192,192,100  
  
}
```

```
LAYER {  
  
  TYPE=ROUT  
  
  NAME=rout  
  
  START_DRILL=
```



```
END_DRILL=  
  
DRAW_COLOR=30,128,96  
  
FLASH_COLOR=30,128,96  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=asemt  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=32,96,100  
  
  FLASH_COLOR=32,96,100  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=assemb  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=32,100,96  
  
  FLASH_COLOR=32,100,96  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=ddt  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=128,96,32  
  
  FLASH_COLOR=128,96,32  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=c-nummer  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=64,128,64  
  
  FLASH_COLOR=64,128,64  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=peel_off  
  
  START_DRILL=
```

```
END_DRILL=

DRAW_COLOR=32,192,192

FLASH_COLOR=32,192,192

}


JOB {

NAME=test-odb+-Gerber

LAYER=layer_1,C:\temp\examples\274X\Art0121.pho

LAYER=layer_2,C:\temp\examples\274X\Art02.pho

LAYER=layer_4,C:\temp\examples\274X\Pgp2529.pho

LAYER=layer_3,C:\temp\examples\274X\gnd2530.pho

LAYER=sst,C:\temp\examples\274X\Sst0126.pho

LAYER=spt,C:\temp\examples\274X\Sp0128.pho

LAYER=smb,C:\temp\examples\274X\Sm0227.pho

LAYER=smt,C:\temp\examples\274X\Sm0128.pho

LAYER=d_1_2,C:\temp\examples\274X\Drill.drl

}
```

6.3 Example Matrix File #2

This sample matrix file defines several layers, loads 3 separate Jobs, and creates a panel.

Note: All FAB 3000 Import Matrix Files must end with *.fm6.

```
# SAMPLE FAB 3000 - MATRIX FILE
```

```
LAYER {
```

```
    TYPE=SILK_TOP
```

```
    NAME=sst
```

```
    START_DRILL=
```

```
    END_DRILL=
```

```
    DRAW_COLOR=255,0,0
```

```
    FLASH_COLOR=255,0,0
```

```
}
```

```
LAYER {
```

```
    TYPE=PASTE_TOP
```

```
    NAME=spt
```

```
    START_DRILL=
```

```
    END_DRILL=
```

```
    DRAW_COLOR=100,223,223
```

```
    FLASH_COLOR=100,223,223
```

```
}
```

```
LAYER {
```

```
TYPE=MASK_TOP  
  
NAME=smt  
  
START_DRILL=  
  
END_DRILL=  
  
DRAW_COLOR=223,100,100  
  
FLASH_COLOR=223,100,100  
  
}
```

```
LAYER {  
  
    TYPE=TOP  
  
    NAME=layer_1  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,223,0  
  
    FLASH_COLOR=100,223,0  
  
}
```

```
LAYER {  
  
    TYPE=neg_plane  
  
    NAME=layer_2  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=50,100,50
```

```
FLASH_COLOR=50,100,50  
  
}  
  
LAYER {  
  
    TYPE=neg_plane  
  
    NAME=layer_3  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,223,200  
  
    FLASH_COLOR=100,223,200  
  
}
```

```
LAYER {  
  
    TYPE=BOTTOM  
  
    NAME=layer_4  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=223,100,223  
  
    FLASH_COLOR=223,100,223  
  
}
```

```
LAYER {  
  
    TYPE=MASK_BOTTOM  
  
    NAME=smb
```

```
START_DRILL=  
  
END_DRILL=  
  
DRAW_COLOR=173,173,173  
  
FLASH_COLOR=173,173,173  
  
}
```

```
LAYER {  
  
    TYPE=PASTE_BOTTOM  
  
    NAME=spb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=192,32,32  
  
    FLASH_COLOR=192,32,32  
  
}
```

```
LAYER {  
  
    TYPE=SILK_BOTTOM  
  
    NAME=ssb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,192,128  
  
    FLASH_COLOR=100,192,128  
  
}
```

```
LAYER {  
  
  TYPE=DRILL  
  
  NAME=d_1_2  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=192,192,100  
  
  FLASH_COLOR=192,192,100  
  
}
```

```
LAYER {  
  
  TYPE=ROUT  
  
  NAME=rout  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=30,128,96  
  
  FLASH_COLOR=30,128,96  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=assemnt  
  
  START_DRILL=
```



```
END_DRILL=  
  
DRAW_COLOR=32,96,100  
  
FLASH_COLOR=32,96,100  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=assemb  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=32,100,96  
  
  FLASH_COLOR=32,100,96  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=ddt  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=128,96,32  
  
  FLASH_COLOR=128,96,32  
  
}
```

```
LAYER {  
  
    TYPE=GRAPHIC  
  
    NAME=c-nummer  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=64,128,64  
  
    FLASH_COLOR=64,128,64  
  
}
```

```
LAYER {  
  
    TYPE=GRAPHIC  
  
    NAME=peel_off  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=32,192,192  
  
    FLASH_COLOR=32,192,192  
  
}
```

```
#####
```

```
#Job - 1
```

```
JOB {  
  
    NAME=test-odb++-Gerber  
  
    LAYER=layer_1,C:\temp\examples\274X\Art0121.pho  
  
    LAYER=layer_2,C:\temp\examples\274X\Art02.pho
```

LAYER=layer_4,C:\temp\examples\274X\Pgp2529.pho

LAYER=layer_3,C:\temp\examples\274X\gnd2530.pho

LAYER=sst,C:\temp\examples\274X\Sst0126.pho

LAYER=spt,C:\temp\examples\274X\Sp0128.pho

LAYER=smb,C:\temp\examples\274X\Sm0227.pho

LAYER=smt,C:\temp\examples\274X\Sm0128.pho

LAYER=d_1_2,C:\temp\examples\274X\Drill.drl

}

#Job - 2

JOB {

NAME=board1

LAYER=smb,C:\temp\examples\Board1\Bottom Mask.gbr

LAYER=layer_4,C:\temp\examples\Board1\Bottom.gbr

LAYER=smt,C:\temp\examples\Board1\Top Mask.gbr

LAYER=layer_1,C:\temp\examples\Board1\Top.gbr

LAYER=sst,C:\temp\examples\Board1\Top Silk.gbr

LAYER=d_1_2,C:\temp\examples\Board1\drill.drl

}

#Job - 3

JOB {

NAME=board4

LAYER=smb,C:\temp\examples\Board4\Bottom Mask.gbr

LAYER=layer_4,C:\temp\examples\Board4\Bottom.gbr

```

LAYER=smt,C:\temp\examples\Board4\Top Mask.gbr

LAYER=layer_1,C:\temp\examples\Board4\Top.gbr

LAYER=sst,C:\temp\examples\Board4\Top Silk.gbr

LAYER=d_1_2,C:\temp\examples\Board4\drill.drl

}

#####

#Panel - 1

PANEL {

    NAME=mypanel

    UNITS=INCH

    WIDTH=18

    HEIGHT=18.5

    STEPREPEAT=board4,17.5,0,0,2.5,1,7,90,No

    STEPREPEAT=board1,17.0,4,3.8,3.4,2,5,90,No

    STEPREPEAT=test-odb++-Gerber,17.85,11.5,7,6,1,3,90,No

}

```

6.4 Example Matrix File #3

This sample matrix file defines several layers, loads 3 separate Jobs, and creates a panel. This Import Matrix also makes use of the NCTOOLS array which define tool sizes (since the drill files don't contain tool information inside the file header) before each JOB array.

Note: All FAB 3000 Import Matrix Files must end with *.fm6.

```
# SAMPLE FAB 3000 - MATRIX FILE
```

```

LAYER {

    TYPE=SILK_TOP

```

```
NAME=sst  
  
START_DRILL=  
  
END_DRILL=  
  
DRAW_COLOR=255,0,0  
  
FLASH_COLOR=255,0,0  
  
}
```

```
LAYER {  
  
    TYPE=PASTE_TOP  
  
    NAME=spt  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,223,223  
  
    FLASH_COLOR=100,223,223  
  
}
```

```
LAYER {  
  
    TYPE=MASK_TOP  
  
    NAME=smt  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=223,100,100  
  
    FLASH_COLOR=223,100,100
```

```
}
```

```
LAYER {
```

```
    TYPE=TOP
```

```
    NAME=layer_1
```

```
    START_DRILL=
```

```
    END_DRILL=
```

```
    DRAW_COLOR=100,223,0
```

```
    FLASH_COLOR=100,223,0
```

```
}
```

```
LAYER {
```

```
    TYPE=neg_plane
```

```
    NAME=layer_2
```

```
    START_DRILL=
```

```
    END_DRILL=
```

```
    DRAW_COLOR=50,100,50
```

```
    FLASH_COLOR=50,100,50
```

```
}
```

```
LAYER {
```

```
    TYPE=neg_plane
```

```
    NAME=layer_3
```

```
    START_DRILL=
```

```
END_DRILL=  
  
DRAW_COLOR=100,223,200  
  
FLASH_COLOR=100,223,200  
  
}
```

```
LAYER {  
  
    TYPE=BOTTOM  
  
    NAME=layer_4  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=223,100,223  
  
    FLASH_COLOR=223,100,223  
  
}
```

```
LAYER {  
  
    TYPE=MASK_BOTTOM  
  
    NAME=smb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=173,173,173  
  
    FLASH_COLOR=173,173,173  
  
}
```

```
LAYER {  
  
    TYPE=PASTE_BOTTOM  
  
    NAME=spb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=192,32,32  
  
    FLASH_COLOR=192,32,32  
  
}
```

```
LAYER {  
  
    TYPE=SILK_BOTTOM  
  
    NAME=ssb  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=100,192,128  
  
    FLASH_COLOR=100,192,128  
  
}
```

```
LAYER {  
  
    TYPE=DRILL  
  
    NAME=d_1_2  
  
    START_DRILL=  
  
    END_DRILL=
```


DRAW_COLOR=192,192,100

FLASH_COLOR=192,192,100

}

LAYER {

TYPE=ROUT

NAME=rout

START_DRILL=

END_DRILL=

DRAW_COLOR=30,128,96

FLASH_COLOR=30,128,96

}

LAYER {

TYPE=GRAPHIC

NAME=assemnt

START_DRILL=

END_DRILL=

DRAW_COLOR=32,96,100

FLASH_COLOR=32,96,100

}

LAYER {

```
TYPE=GRAPHIC  
  
NAME=assemb  
  
START_DRILL=  
  
END_DRILL=  
  
DRAW_COLOR=32,100,96  
  
FLASH_COLOR=32,100,96  
  
}
```

```
LAYER {  
  
    TYPE=GRAPHIC  
  
    NAME=ddt  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=128,96,32  
  
    FLASH_COLOR=128,96,32  
  
}
```

```
LAYER {  
  
    TYPE=GRAPHIC  
  
    NAME=c-nummer  
  
    START_DRILL=  
  
    END_DRILL=  
  
    DRAW_COLOR=64,128,64
```

```
FLASH_COLOR=64,128,64  
  
}
```

```
LAYER {  
  
  TYPE=GRAPHIC  
  
  NAME=peel_off  
  
  START_DRILL=  
  
  END_DRILL=  
  
  DRAW_COLOR=32,192,192  
  
  FLASH_COLOR=32,192,192  
  
}
```

```
#####
```

```
NCTOOLS {  
  
  UNITS=INCH  
  
  INTEGER=2  
  
  DECIMAL=4  
  
  TYPE=ABSOLUTE  
  
  SUPPRESSION=TRAILING  
  
  TOOL=1,0.02,YES  
  
  TOOL=2,0.04,YES  
  
  TOOL=4,0.055,YES  
  
  TOOL=5,0.07,YES
```

TOOL=10,0.105,NO

}

JOB {

NAME=test-odb++-Gerber

LAYER=layer_1,C:\temp\examples\274X\Art0121.pho

LAYER=layer_2,C:\temp\examples\274X\Art02.pho

LAYER=layer_4,C:\temp\examples\274X\Pgp2529.pho

LAYER=layer_3,C:\temp\examples\274X\gnd2530.pho

LAYER=sst,C:\temp\examples\274X\Sst0126.pho

LAYER=spt,C:\temp\examples\274X\Sp0128.pho

LAYER=smb,C:\temp\examples\274X\Sm0227.pho

LAYER=smt,C:\temp\examples\274X\Sm0128.pho

LAYER=d_1_2,C:\temp\examples\274X\Drill.drl

}

#####

NCTOOLS {

UNITS=INCH

INTEGER=2

DECIMAL=4

TYPE=ABSOLUTE

SUPPRESSION=TRAILING

```
TOOL=2,0.042,YES
TOOL=3,0.044,YES
TOOL=5,0.045,YES
TOOL=6,0.074,YES
TOOL=10,0.105,NO
}

JOB {
    NAME=board1
    LAYER=smb,C:\temp\examples\Board1\Bottom Mask.gbr
    LAYER=layer_4,C:\temp\examples\Board1\Bottom.gbr
    LAYER=smt,C:\temp\examples\Board1\Top Mask.gbr
    LAYER=layer_1,C:\temp\examples\Board1\Top.gbr
    LAYER=sst,C:\temp\examples\Board1\Top Silk.gbr
    LAYER=d_1_2,C:\temp\examples\Board1\drill.drl
}
```

#####

```
# If the drill file does not contain tool information in the header
# the tool sizes from the previous NCTOOLS array will be used
# for this Job also.
```

```
JOB {
```

```

NAME=board4

LAYER=smb,C:\temp\examples\Board4\Bottom Mask.gbr

LAYER=layer_4,C:\temp\examples\Board4\Bottom.gbr

LAYER=smt,C:\temp\examples\Board4\Top Mask.gbr

LAYER=layer_1,C:\temp\examples\Board4\Top.gbr

LAYER=sst,C:\temp\examples\Board4\Top Silk.gbr

LAYER=d_1_2,C:\temp\examples\Board4\drill.drl

}

```

```
#####
```

```
#Panel - 1
```

```

PANEL {

    NAME=mypanel

    UNITS=INCH

    WIDTH=18

    HEIGHT=18.5

    STEPREPEAT=board4,17.5,0,0,2.5,1,7,90,No

    STEPREPEAT=board1,17.0,4,3.8,3.4,2,5,90,No

    STEPREPEAT=test-odb++-Gerber,17.85,11.5,7,6,1,3,90,No

}

```

6.5 C++ Source Example for calling Matrix File

This sample shows how to call FAB 3000 and then add the matrix file as the first argument. We're assuming the external tool has already created a successful matrix file called "matrix.fm6" located in the C:\Temp folder.

Note: Simply call FAB 3000 and pass the matrix file as the first argument.

Note: All FAB 3000 Import Matrix Files must end with *.fm6.

Note: This sample below is using MFC in Visual C++ 2005.

Note: This sample assumes that you installed FAB 3000 to "C:\Program File\Numerical Innovations\FAB 3000"

```
//Start
```

```
STARTUPINFO si;
```

```
PROCESS_INFORMATION pi;
```

```
GetStartupInfo(&si);
```

```
SetCurrentDirectory( "C:\\Temp" );
```

```
CString CamPath = "C:\\Program Files\\Numerical Innovations\\FAB 3000\\FAB 3000.exe";
```

```
CamPath += " \\C:\\Temp\\matrix.fm6\\";
```

```
if( !CreateProcess( NULL,(LPCTSTR)CamPath,NULL,NULL,FALSE,0,NULL,NULL,&si,&pi))
```

```
{
```

```
    printf( "Unable to Start FAB 3000" );
```

6.6 Layer Types

Below are a listing of the available layer types which can be used in the LAYER array, TYPE field.

TOP	Top Side Metal
BOTTOM	Bottom Side Metal
INTERNAL	Internal Metal (Positive Polarity)
NEG_PLANE	Plane (Negative Polarity -- Usually contains Thermal Dcodes)
POS_PLANE	Plane (Positive)
DRILL	NC Drill (Should have tools sizes & format embedded inside file)
SILK_TOP	Top Side Silkscreen
SILK_BOTTOM	Bottom Side Silkscreen
MASK_TOP	Top Side SolderMask
MASK_BOTTOM	Bottom Side SolderMask

PASTE_TOP	Top Side Paste
PASTE_BOTTOM	Bottom Side Paste
ROUT	NC Rout (Should have tools sizes & format embedded inside file)
INSULATOR	Insulator Layer
COMPONENTS_TOP	Top side Components (Must be CSV Format Centroid & it will be renamed to "comp+_top")
COMPONENTS_BOTTOM	Bottom side Components (Must be CSV Format Centroid & it will be renamed to "comp+_bot")
TEMPORARY	Temporary Layer (Used when the Gerber type is unknown)
FAB_DRAWING	Fab Drawing (Used for reference only)
BORDER	PCB Boarder Layer
SCORE	Score Layer used for Panel Snap-Outs
GRAPHIC	Default Layer (Used when the Gerber type is unknown)

6.7 NCTools Array

NCTools array is optional. It is only intended for NC files that don't include tool definitions in the header. Must be called before Job

Fields in the NCTOOLS array

UNITS	Units to use for this nctools array. Must be either: INCH or MM
INTEGER	Integer digits for drill or rout file between (0-6).
DECIMAL	Decimal digits for drill or rout file between (0-6).
TYPE	Drill or rout file type. ABSOLUTE -- Use Absolute coordinates during NC Import INCREMENTAL -- Use Incremental coordinates during NC Import
SUPPRESSION	Drill or rout file zero suppression. LEADING -- Suppress Leading Zeros during NC Import TRAILING -- Suppress Trailing Zeros during NC Import NONE -- No Zero Suppression during NC Import
TOOL	This field contains the Tool # (between 1 and 1000), Diameter (in Tool Units), and Plated (either: YES or NO); all separated by a commas.

Note: NCTOOLS array is optional. It is only intended for NC files that don't include tool definitions in the header. Must be called before Job

```
NCTOOLS {  
  
    UNITS=INCH  
  
    INTEGER=2  
  
    DECIMAL=4  
  
    TYPE=ABSOLUTE  
  
    SUPPRESSION=TRAILING  
  
    TOOL=1,0.02,YES  
  
    TOOL=2,0.04,YES  
  
    TOOL=4,0.055,YES  
  
    TOOL=5,0.07,YES  
  
    TOOL=10,0.105,YES  
  
}  
  
JOB {  
  
    NAME=board1  
  
    LAYER=smb,C:\temp\examples\Board1\Bottom Mask.gbr  
  
    LAYER=layer_4,C:\temp\examples\Board1\Bottom.gbr  
  
    LAYER=smt,C:\temp\examples\Board1\Top Mask.gbr  
  
    LAYER=layer_1,C:\temp\examples\Board1\Top.gbr  
  
    LAYER=sst,C:\temp\examples\Board1\Top Silk.gbr  
  
    LAYER=d_1_2,C:\temp\examples\Board1\drill.drl  
  
}
```

6.8 DFMChecker Array

DFMCHECKER array is optional. It is used to run the DFM Checker on an imported Job. Must be called after Job and will only run on the previous Job loaded.

Fields in the DFMCHECKER array

UNITS	MIL or MM	Units to use for this dfmchecker array
minimum_padsizesize	positive number	Minimum Pad size allowed
minimum_smtpadwidth	positive number	Minimum SMT pad width allowed
minimum_tracewidth	positive number	Minimum trace width allowed
minimum_outer_tracespacing	positive number	Minimum spacing on top or bottom layers
minimum_inner_tracespacing	positive number	Minimum spacing for innerlayers
minimum_drill_pad_annularring	positive number	Minimum drill to pad annular ring
minimum_border_clearance	positive number	Minimum clearance from board edge
find_acidtraps	positive number	Locate Acid Traps
find_nonfunctional_internalpads		Find non-functional pads on internal layers
minimum_drillsize	positive number	Minimum drill hole diameter
find_drill_doublehits		Find drill double hits
find_drill_missinghits		Find missing drill hits
find_missingpads		Find missing pads
find_touchingholes		Find drill holes that touch
minimum_silkscreen_linewidth	positive number	Find minimum silkscreen line width
find_silkscreen_overmask	positive number	Locate silkscreen over soldermask
silkscreen_border_clearance	positive number	Minimum clearance allowed from Silkscreen to board edge
undersize_mask_clearance	positive number	Locate small mask openings
missing_mask_clearances		Find missing mask clearances

find_mask_slivers		Find mask slivers
find_solder_bridges	positive number	Find solder bridges
inner_clearance	positive number	Inner clearance on plane layers
power_ground_minimum_annularring	positive number	Minimum annular ring on plane layers
find_starved_thermals	positive number	Locate starved thermals
find_power_ground_shorts		Find power/ground shorts
check_negplane_thermal_conflicts		Find thermal conflicts on neg. planes
find_isolated_connections		Find isolated connections
output_drc_pdf_file		Location for resulting PDF file,
ipc_Netlist_file		Location of IPC-D-356 Netlist file used for Netlist comparison
x_Netlist_offset		Offset IPC Netlist file in X direction
y_Netlist_offset		Offset IPC Netlist file in Y direction
Netlist_scale_factor	positive number	Scale IPC Netlist

Note: If you do not wish to check for all of the above violations, simply omit them in the import matrix file.

JOB {

NAME=test-odb++-Gerber

LAYER=layer_1,274X\Art0121.pho

LAYER=layer_4,274X\Art02.pho

LAYER=layer_2,274X\Pgp2529.pho

LAYER=layer_3,274X\gnd2530.pho

LAYER=sst,274X\Sst0126.pho

LAYER=spt,274X\Sp0128.pho

LAYER=smb,274X\Sm0227.pho

LAYER=smt,274X\Sm0128.pho

```
LAYER=d_1_2,274X\Drill.drl  
  
}  
  
DFMCHECKER {  
  
    units=mil  
  
    minimum_padsizesize=15.0  
  
    minimum_smtpadwidth=5.0  
  
    minimum_tracewidth=4.0  
  
    minimum_outer_tracespacing=4.0  
  
    minimum_inner_tracespacing=6.0  
  
    minimum_drill_pad_annularring=4.0  
  
    minimum_border_clearance=10.0  
  
    find_acidtraps=7.0  
  
    find_nonfunctional_internalpads  
  
    minimum_drillsize=7.0  
  
    find_drill_doublehits  
  
    find_drill_missinghits  
  
    find_missingpads  
  
    find_touchingholes  
  
    minimum_silkscreen_linewidth=4.0  
  
    find_silkscreen_overmask=1.0  
  
    silkscreen_border_clearance=10.0  
  
    undersize_mask_clearance=4.0  
  
    missing_mask_clearances
```

```
find_mask_slivers  
  
find_solder_bridges=8.0  
  
inner_clearance=7.0  
  
power_ground_minimum_annularring=4.0  
  
find_starved_thermals=8.0  
  
find_power_ground_shorts  
  
check_negplane_thermal_conflicts  
  
find_isolated_connections  
  
output_drc_pdf_file=C:\output\mydrcfile.pdf  
  
ipc_Netlist_file=C:\Netlist\testNetlist.ipc  
  
x_Netlist_offset=0.0  
  
y_Netlist_offset=15.0  
  
Netlist_scale_factor=1.0  
  
}
```

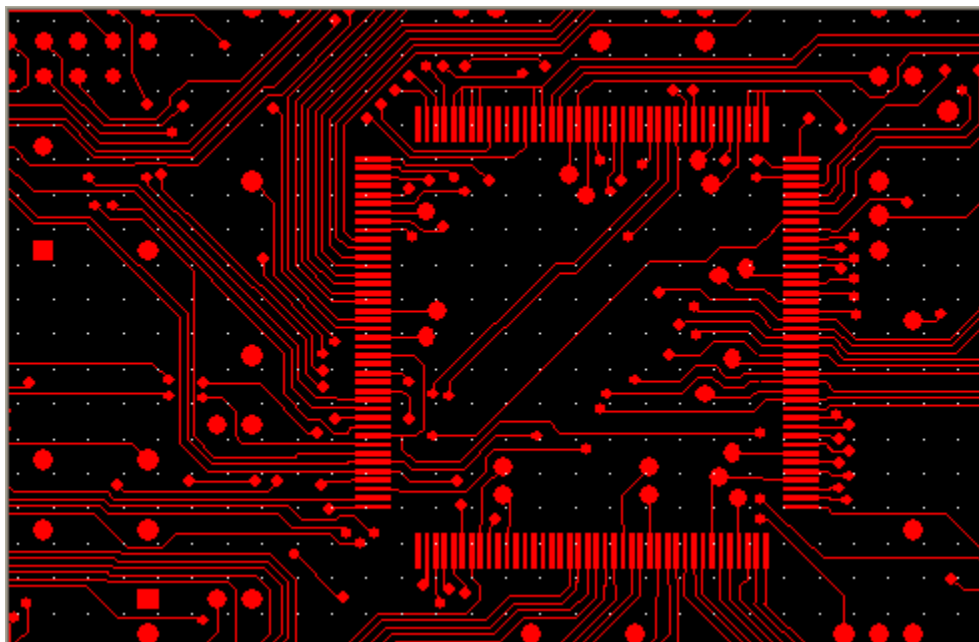
SECTION 7: FAB 3000 DATA FORMAT OVERVIEW

FAB 3000 supports many file formats, such as GDSII, DXF, OASIS, Postscript, Gerber, and more.

7.1 Gerber Format

Bi-directional translation of Gerber format (RS-274D, RS-274X, Fire9000, BarcoDPF) files into the Workspace. Excellent for anyone who needs to make photoplots from their artwork.

Sample Gerber screen capture inside the workspace.



Gerber data is a simple, generic means of transferring printed circuit board information to a wide variety of devices that convert the electronic PCB data to artwork produced by a photoplotter. Virtually every PCB CAD system generates Gerber data because all photoplotters read it. It is a file structure consisting of X,Y coordinates supplemented by commands that define where the PCB image starts, what shape it will take, and where it ends. In addition to the coordinates, Gerber data contains aperture information, which defines the shapes and sizes of lines, holes, and other features.

Gerber RS-274X:

Extended Gerber Format, which is also called RS-274X, provides enhancements that handle polygon fill codes, positive/negative image compositing, and custom apertures, and other features. RS-274X also encapsulates the aperture list in the header of the Gerber data file and therefore allows files to pass from one system to another without the need to re-input the aperture table. RS-274X produces a variety of Gerber data called X data.

The Problem - Gerber files can be Difficult to Handle:

Gerber data is unique in the fact that each file usually represents a single layer, something different from most other EDA formats that may contain multiple layers & designs/blocks all inside a single file. Many Gerber files are not written to specification; thus many "unwritten rules" must be made to properly load in these files to the author's intention.

Here is a list of some common difficulties found inside Gerber files:

- ▶ Transcoding of apertures - when multiple Gerber files define the same aperture Dcode, but require different shapes,
- ▶ Handle complex custom aperture definitions.
- ▶ Handle 'user-defined' custom aperture macros (\$).
- ▶ Circular Interpolation: Arcs (G75), & Quadrant Arcs (G74)
- ▶ Poly-Fill routines (G36, G37)
- ▶ Support composite merging (LPD/LPC)
- ▶ Photo Expose mode (G55)
- ▶ Step/repeat codes (SR)
- ▶ Offsets (OF), Units (G70, G71), Mirror(MI)
- ▶ Zero Suppression with leading or trailing zeroes.
- ▶ Format type: Incremental (G91)
- ▶ Most software products have a hard time loading, & rendering Gerber files attributes properly (i.e. rounded paths, custom flashes, composite layers, and step/repeat).

Why FAB 3000 is the Perfect Solution

FAB 3000 successfully handles all features of the Gerber (RS-274X) format. Custom apertures, composite layers, and step/repeat are all correctly processed.



[Video: Import old 274D Gerber files and Export as Embedded](#)

[Video: Load RS-274D Gerber file. Create Aperture List Template](#)

[Video: How to Easily Change the Field \(i.e. reverse polarity\) for Gerber files](#)

7.2 ODB++ Format

ODB++ is the most intelligent CAD/CAM data exchange format available today. It captures all CAD/EDA, assembly, and PCB fabrication knowledge in one single, unified database. Originally developed by Valor Computerized Systems for use in its own PCB CAD/CAM systems, Genesis 2000, Enterprise 3000 and Trilogy, ODB++ has already become widely accepted as the de facto industry standard providing unprecedented power to PCB design, fabrication, and assembly with the flexibility to expand as required. In parallel, ODB++ is providing most of the technological basis for the new IPC2581 standard for data exchange in the PCB fabrication and assembly industry.



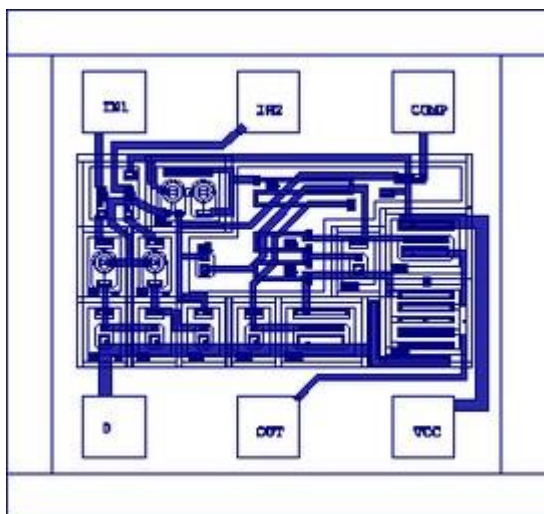
[Video: ODB++ PCB to 3D STL Conversion, view in VisCAM or Solidworks](#)

7.3 DXF Format

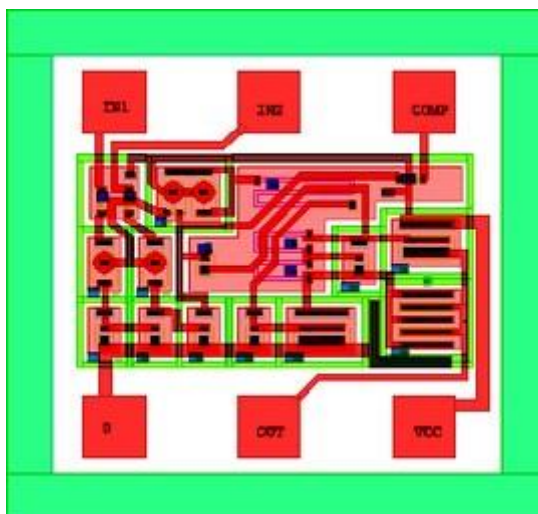
DXF stands for Drawing Exchange Format, and it is a format for transferring drawings between Computer Aided Design systems. DXF is widely used as the de facto standard in the engineering and construction industries. The DXF format is a tagged data representation of all the information contained in an AutoCAD® drawing file. Tagged data means that each data element in the file is preceded by an integer number that is called a group code. A group code's value indicates what type of data element follows. This value also indicates the meaning of a data element for a given object (or record) type. Virtually all user-specified information in a drawing file can be represented in DXF format.

Bi-directional translation of DXF files into the Workspace. Excellent for anyone using software products such as AutoCAD, SolidWorks or ProEngineer to design complex masks, MEMs, RF/Microwave, MCM, Hybrid Packages, etc.

Standard AutoCAD DXF File:



Same DXF file Imported into FAB 3000:



The Problem: DXF files can be tricky to handle

DXF data is probably the most extensive and difficult format to work with; especially when converting to other EDA formats. However it is the de facto standard file format among most engineering CAD tools.

Here is a list of some common difficulties found inside DXF files:

- ▶ Entities found in a Block that are drawn on layer '0', usually need to be moved to the blocks new Inserted layer. If the block is Inserted onto multiple layers, the '0' layer entities from that block must be moved each time.
- ▶ A solid hatch with internal geometries, does not explicitly define the polarity of the internal geometries.
- ▶ Non-Manhattan geometries, angles, and non-uniform scaling are permitted.
- ▶ Invalid, self-intersecting polygons are permitted.
- ▶ Identical, overlapping entities on the same layer can be stacked on top of each other. These identical entities are impossible to see, and can cause problems during conversions, etc.
- ▶ The DXF format specifications change almost every year, with the latest releases of AutoCAD.
- ▶ Nested Non-Manhattan transformations.
- ▶ Tapered polylines with non-uniform widths are permitted.
- ▶ Some boundaries may appear closed, but internally are still open.
- ▶ Filled pads are drawn as outlines of circles, donuts, or rectangular.
- ▶ Extraneous data such as UCS, and 3D object extrusions can be found in many files.
- ▶ AutoCAD will not display filled areas on-screen, such as polygons, and other closed boundaries (only solid hatch). Making it tough to determine the final results.
- ▶ Fonts can be difficult to handle, especially MText (Multi-line text)

Most vendors and mask houses stay away from accepting DXF files as input. They have a hard time dealing with this data and there are only a few decent software products on the market today that can reliably handle DXF.

Why FAB 3000 is the Perfect Solution

Since Software products such as AutoCAD, SolidWorks or ProEngineer provide an excellent way to draw complex masks, MEMs, RF/Microwave, MCM, Hybrid Packages, as well as drafting mechanical information related to the design; FAB 3000 was developed with this in mind.

Here's how DXF Plug-In handles common difficulties found inside DXF files:

- ▶ Block entities drawn on layer '0', are automatically handled by the option: ByBlock/ByLayer.
- ▶ Polygon De-Embedding is used to process entity polarity for solid hatches with internal geometries.
- ▶ All angles, and non-uniform scaling are supported.
- ▶ Invalid, self-intersecting polygons are either automatically corrected or may be fixed using the editor.
- ▶ Identical overlapping entities are either automatically deleted, or may be fixed using the editor.
- ▶ Every year the DXF Plug-In is updated to support the latest versions of DXF.
- ▶ Unlimited nested transforms are supported.
- ▶ Outlines are generated for tapered polylines with non-uniform width.
- ▶ Open boundaries will be automatically closed when it is possible.
- ▶ Recognizes outlined pads: round, donut & rectangular. When requested these pads are automatically filled & flashed.
- ▶ Most extraneous 3D data is ignored.
- ▶ Direct support for TrueType fonts

FAB 3000 is powerful, yet very easy to use, and it operates seamlessly inside of the platform. If you design or work with complex masks, MEMs, RF/Microwave, MCM, Hybrid Packages, as well as mechanical drafting, then FAB 3000 is for you!

FAB 3000 also supports polygon de-embedding for solid hatches.

7.4 DWG Format

FAB 3000 supports bi-directional translation of native AutoCAD DWG format. Similar to the DXF format except in a binary file.

7.5 IGES Format

IGES (Initial Graphics Exchange Specification) is a neutral graphics database format designed primarily for data exchange between mechanical CAD systems. The IGES file format links mechanical CAD systems to the 2D EDA design world. Two-dimensional geometry can be used to interchange layout, or package outline information. The IGES format can represent both mechanical and electrical design data in two and three dimensions.

IGES requires the Add-On: "3D Formats".

7.6 Image Format

FAB 3000 supports bi-directional translation of image (TIFF, PNG, GIF, BMP, etc.) files into the Workspace. Once these image pixels are imported into the workspace, they can be handled just as any other geometry (i.e. change layers, copy, export to GDSII, DXF, etc.). This is perfect for anyone who wishes to produce optical filters, convert company logos into a common EDA format, create grey scales, or plot images to a higher resolution than on an image setter.

7.7 GDSII Format

GDSII is a binary file format which is classified as a "data interchange format" and used for transferring mask-design data between the IC designer and the fabrication facility ("Fab").

7.8 Postscript Format

PostScript is the industry standard for sending documents to high-resolution printers.

Please note that Ghostscript must be installed on your workstation or server to import PostScript!

Ghostscript is freely available and distributed under the GNU General Public License. If you do not have Ghostscript installed you may download it from the following websites (or from other available mirrors):

<http://www.cs.wisc.edu/~ghost/doc/gnu/index.htm>

<http://www.ghostscript.com/>

7.9 Large Monochrome Bitmap Format

FAB 3000 bi-directionally translates large sized Bitmap files (*.bmp) with unlimited pixels. This is very useful when someone wants to convert a high-resolution image into a true vector format.

Image files are unit-less. A pixel can theoretically represent any physical size. It is important that you specify the proposed units and scale for a pixel's physical size. Choose from DPI, Micron, Inch, Millimeter, Centimeter, and more. Logical pixel size is defined by **Pixel Units * Image Scale**.

Large Monochrome Bitmap requires the Add-On: "Large Bitmap".

7.10 Drill/Rout Format

NC code used to control Drill and Rout CNC machines.

7.11 STL Format

STL is a file format native to the stereolithography CAD software created by 3D Systems. This file format is supported by many other software packages and is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object without any representation of color, texture, or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common since they are more compact. ACE only supports the binary formats.

STL (Stereolithography) binary format is outputted by almost every 3D design/modeling software tool. This includes SolidWorks, ProEngineer, and many other design packages. STL files are unit-less and objects can be assigned any scale. It is important that you are aware of the imported STL file units & scale. ACE needs to know what physical unit and scale to use for 1 STL unit.



[Forum: DXF to 3D STL Conversion](#)

SECTION 8: LUA SCRIPTING ENGINE

To visit the FAB 3000 LUA API Guide, please visit:

<http://numerical-help-guide.s3.amazonaws.com/fab3000v7api/HTML/index.html>

SECTION 9: PURCHASE & SUPPORT

9.1 How to order FAB 3000

For product pricing, please contact our sales team below:

Phone: 1-866-528-9274

Hours of Operation: Monday – Friday 9:00 AM to 5:00 PM PST

Email: sales@numericalinnovations.com

Payment Options

Credit Cards: We accept Paypal, VISA, MasterCard, Discover, and American Express cards. This is our preferred method of payment. Additional payment methods accepted include: AMEX, Cirrus, Western Union, Visa (Electron), Switch, Solo, Maestro, Direct Debit, Delta, & Google Checkout.

Purchase Orders: Company Purchase Orders are accepted. We verify company information through D&B, and may request trade references before processing. Terms are: Net 30 days, and the minimum order must exceed \$750 USD. You may submit your purchase orders to our sales team by e-mail: sales@numericalinnovations.com or fax: (+1) 858-430-2705.

Bank Wire Transfer: Contact us, and we will e-mail an invoice with our bank details.

9.2 License Options

Single User License: A permanent license key (or activation code) that is assigned to an individual user.

Floating (Network): Centralized server based licensing using FLEXnet License manager (formerly FLEXlm). The software can be run on as many computers as needed which are connected to a centralized server location. The number of licenses purchased determines the number of concurrent users.

Enterprise License (Site): Receive a master license key that grants your company the right to install UNLIMITED seats of onto your workstations, networks, or on laptops without any restrictions - provided that our software is only used inside your company (at one Site).

Upgrade (Existing Customers only): We offer special pricing for users who have already purchased an older version of our software products and now wish to upgrade to the latest version.

9.3 What is the Numerical Maintenance Plan?

We offer two support/maintenance solutions: **STANDARD** and **LIFETIME**. Standard Maintenance is provided free and LIFETIME Maintenance is optional. The LIFETIME Maintenance Plan is sold at a discounted rate during your initial order placement. Please contact our sales team for more assistance sales@numericalinnovations.com

LIFETIME MAINTENANCE PLAN:

At the time of your order placement, you may optionally purchase our Lifetime Maintenance Plan (LMP) for the following benefits:

- ✔ Secure Login Access to our website.
- ✔ Receive every Upgrade, Update, & Bug-Fixes for life!
- ✔ Unlimited & Priority Technical support from our staff.
- ✔ Unlimited License Assistance in the event of a lost license, computer transfer, crash, etc.
- ✔ Download Access 24/7.
- ✔ Access to Pre-Released & Special Versions of all our software products.
- ✔ Ability to Transfer your LIFETIME Maintenance to another user.

STANDARD MAINTENANCE PLAN:

- ✔ 12-month Email Support for from the date of your purchase.
- ✔ Access to the User Support Forum, and have the ability to post any questions directly on the User Support Forum.
- ✔ A pay-as-you-need [On-Demand Support](#) is also available for additional assistance.

9.4 FAB 3000 Customer Support

There are several ways to get support for FAB 3000:

Numerical Innovations User Forums

Click one of the links below for direct access to the FAB 3000 User Forums.

Forum: Sales and General Questions

Sales and pre-purchase queries.

Forum: FAB 3000 Questions & Answers

Gerber and CAM software related queries.

Forum: ACE 3000 Questions & Answers

Online Support and FAQ for CAD conversions.

Forum: EasyGerb Questions & Answers

AutoCAD ARX Gerber conversion queries.

Submit a support ticket

Can't find an answer you're looking for? Fill out a support ticket to receive personal assistance from our team.

<http://www.numericalinnovations.com/pages/customer-support>

Please include a detailed explanation of the problem. Include a sample file if possible.

Response time is within one business day or sooner.

License Key / Activation Code Problems

Having issues with your license? Contact sales@numericalinnovations.com

General Questions

We welcome your questions, comments, and suggestions as one critical way to continuously improve our services to you. For general information, use our contact form at:

<http://www.numericalinnovations.com/pages/contact-us>

9.5 Numerical Innovations 30 Day Guarantee

Numerical Innovations (a Division of Caneberra Technologies, Inc.), offers you a risk free for 30 day money back guarantee on all of our software products. We are sure you will be pleased with its features, ease of use and reliability. If you are not completely satisfied, let us know within 30 days and if we cannot correct the problem, we will refund your money. Guaranteed.

9.6 Additional Contact Information



A division of Caneberra Technologies Inc.

1888 Kalakaua Ave, Suite #C312

Honolulu, Hawaii 96815

Phone: 1-866-528-9274

Fax: 1-858-430-2705

<http://www.numericalinnovations.com>

Sales Department: sales@numericalinnovations.com

Tech. Support: support@numericalinnovations.com

Career Opportunities: careers@numericalinnovations.com

Partnerships / Alliances: partners@numericalinnovations.com

9.7 Company History



Numerical Innovations (a division of Caneberra Technologies, Inc.) was founded by Simon Garrison and Steve Geiger, two veterans of the PCB industry with nearly 20 years of software development experience.

Back in 1993, Simon started in the PCB industry doing CAD conversions for a small PCB photoplotting company in Santa Clara, California. Steve met Simon from a mutual friend while they were both attending San Jose State University – Simon in the School of Engineering, and Steve in the Business (Marketing) School. They immediately hit it off and became friends and business partners. Simon had been running his company Innovative CAD (Computations and Design) doing his conversion work, and Steve was helping with the marketing. When he explained to Steve that he was disappointed in how cumbersome the translation software in the market was, they decided to join forces to create Innovative CAD Software, Inc.

(ICD) and they rest is history!

Innovative CAD grew from a two man operation to one of the most recognized companies in the PCB CAM market with its tools AutoGERB (world's first exporting of Gerber data directly from AutoCAD) and CAMtastic! (CAM viewer which morphed into the first PCB DFM tool of its kind). They were the first in the industry to do some “cutting-edge” marketing by offering their products to be downloaded directly from the Web with a full 45 day trial, as well as offering complete online support. This and other sound business strategies positioned them for explosive growth and success. The ease of use, comprehensiveness and accuracy of their software tools started attracting big industry attention. ICD did a promotion with Altium, in which a seat of CAMtastic! was included with the purchase of their Protel software; up to that date it was the most successful promotion 'ever' for Altium, and subsequently ICD was acquired by Altium in 2000 for its remarkable technology.

Simon (now one of the leading experts in PCB CAM software development, as well as CAD conversion software) went on to work with Altium for several years developing its CAM division, then started a new firm developing custom OEM solutions and tools for the IC/MEMs industry. Steve started a successful advertising and public relations firm, which he eventually sold as well. Missing that spark and passion for the PCB market they joined forces once again to start NI with a vision to change the landscape of the PCB marketplace.

With tools like DFM Now! and FAB 3000 they are revolutionizing the way CAM and DFM are done for PCB – again with the philosophy of being easy to use, comprehensive and accurate, while providing exceptional value for their customers.

Mission Statement: We have become a leader in the EDA software industry by empowering electronic designers and engineers with clever, high quality software tools which facilitate their designs for manufacturing.