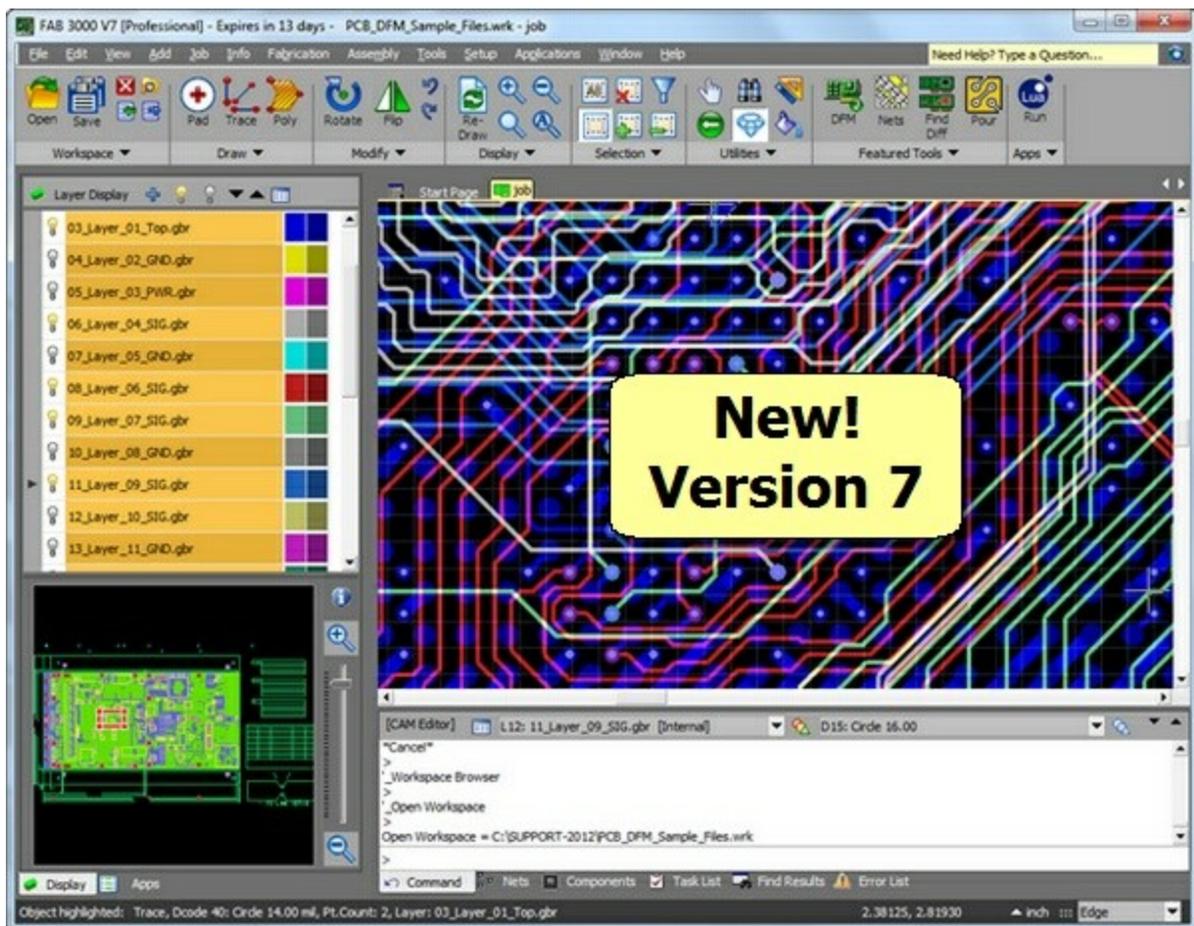


# FAB 3000 V7 - Help Guide



Copyright © 2006-2013 Numerical Innovations (a Division of Caneberra Technologies, Inc.) All Rights Reserved.

# FAB 3000 (Version 7)

**The most unique and easy-to-use CAM/DFM solution**

---

*by Numerical Innovations, a Division of Caneberra Technologies, Inc.*

*Nowadays, it's critical for every PCB designer to use CAM software to analyze and prepare their gerber & drill files for manufacturing. FAB 3000 is a "Full-Featured" CAM software package targeted towards the small-to-mid size company; 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 C/C++ scripts & 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.*

# FAB 3000 V7 - Help Guide

©2006-2013 Numerical Innovations, a Division of Caneberra

## Technologies Inc.

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Printed: November 2013 in San Diego, California USA

### **Publisher**

*Numerical Innovations*

### **Managing Editor**

*Simon Garrison*

### **Technical Editors**

*Steve Geiger*

*Richard Pham*

### **Cover Designer**

*Synergy Communications*

### **Team Coordinator**

*Miranda Yip*

### **Production**

*Steve Geiger*

### **Special thanks to:**

*All the people who contributed to this document, to coffee (with extra cream & sugar), to the pizza service down the street (your daily Capricciosas saved our lives), and to you our customers for giving us an opportunity to satisfy your design & verification requirements.*

*Last not least, we want to thank every engineer and person who has given us timely feedback and suggestions which have enabled us to continue improving this software product.*

*For more credits, please view the About Box.*

# Table of Contents

Foreword	1
<b>Part I FAB 3000 - Help Guide</b>	<b>3</b>
<b>Part II What's New!</b>	<b>5</b>
<b>Part III Getting Started</b>	<b>7</b>
1 System Requirements .....	7
2 Installation .....	7
3 Licensing Options .....	7
4 Node Locked and Demo - License Key Installation .....	8
5 Activation Codes: Single User, Multi-User - License Installation (Windows) .....	10
6 Network - License Installation .....	11
7 Launching FAB 3000 .....	14
Windows OS .....	14
Linux OS .....	14
8 Technical Support .....	15
9 Getting Additional Help .....	15
<b>Part IV Learn Guides</b>	<b>17</b>
1 General .....	17
Viewing & Selection Basics .....	17
Open/Save Workspaces .....	19
Using Measure .....	19
Finding Dcodes .....	21
Using the Properties Dialog .....	21
Plot to PDF .....	23
Using Redline Markup .....	26
2 Import .....	28
Using AutoLoad .....	28
Importing DXF .....	32
Importing ODB++ .....	39
Importing Zip (Compressed) File .....	39
3 Merge Jobs .....	40
Merge Jobs (Automatically) .....	41
Merge Jobs (Manually using Inserts) .....	45
Synchronize Layers .....	46
Using Film Wizard .....	47
4 DRC / DFM .....	50
Using DFM Checker .....	50
Viewing DRC/DFM Violations .....	51
Create a PDF Report for DRC/DFM Violations .....	52
5 Nets .....	52

Extracting a Netlist from Gerber and Drill data .....	52
Extracting with Blind and Buried Vias .....	53
Comparing a CAM Netlist against an External Netlist .....	54
Viewing Netlist Comparison Errors .....	55
Create a PDF report for Net Comparison Errors .....	56
Add/Remove Objects from a Net .....	56
Apply External Net Names to CAM Net .....	57
<b>6 Apertures / Custom Apertures .....</b>	<b>58</b>
Creating Custom Apertures .....	58
<b>7 Composites .....</b>	<b>58</b>
Composites 101 .....	58
Using DeEmbedding to Automatically Create Composites .....	59
<b>8 Assembly / Centroid File .....</b>	<b>60</b>
How to generate a Centroid File .....	60
<b>Part V Working with FAB 3000 .....</b>	<b>65</b>
<b>1 Workbench .....</b>	<b>65</b>
Editor .....	66
Layer View .....	67
Aerial View .....	68
<b>2 Workspace .....</b>	<b>69</b>
Resource .....	70
Job .....	70
Insert .....	70
Panel .....	70
Custom Aperture.....	70
Workspace Browser... .....	70
Library .....	71
<b>Part VI Job Editor Menu .....</b>	<b>73</b>
<b>1 File Menu .....</b>	<b>73</b>
<b>New .....</b>	<b>73</b>
Workspace.....	73
Job .....	73
Panel .....	73
Custom Aperture.....	74
Footprint .....	74
Text File .....	74
Spreadsheet File.....	75
Application File.....	75
<b>Open Workspace .....</b>	<b>76</b>
<b>Close Workspace .....</b>	<b>77</b>
<b>Save Workspace .....</b>	<b>77</b>
<b>Save Workspace As .....</b>	<b>77</b>
<b>Workspace Browser .....</b>	<b>77</b>
<b>Information... .....</b>	<b>78</b>
<b>Open Folder .....</b>	<b>78</b>
<b>Open Compressed file (Zip,Tar,GZ,RAR)...</b>	<b>79</b>
<b>Import .....</b>	<b>79</b>
AutoLoad.....	79
Gerber .....	85

Drill	87
Rout / Mill	88
ODB++	89
ODB++ (Directory)	90
Netlist (IPC-D-356/A)	90
DXF / DWG	91
Image	94
HPGL/2	95
Postscript / PDF	96
Large Bitmap	97
STL (3D)	98
GDSII	99
Part Centroid	99
BOM	100
Aperture List	101
Custom Aperture Library	101
Footprints Library	101
FAB 3000 Matrix	101
<b>Export</b>	<b>102</b>
Gerber	102
Drill	103
Rout / Mill	105
PCB	106
ODB++	107
Netlist (IPC-D-356/A)	107
DXF / DWG	108
GDSII	110
Large Bitmap	111
Centroid File	112
BOM File	112
IGES	112
STL	113
Postscript	113
Custom Aperture Library	114
Footprints Library	114
<b>Plot to PDF (up to E Size)</b>	<b>114</b>
<b>Plot to Image</b>	<b>121</b>
<b>Recent Workspaces</b>	<b>122</b>
<b>Exit</b>	<b>122</b>
<b>2 Edit</b>	<b>122</b>
<b>Undo</b>	<b>122</b>
<b>Redo</b>	<b>123</b>
<b>Clipboard</b>	<b>123</b>
<b>Selection</b>	<b>123</b>
All	123
Display	123
Window	123
Crossing Window	124
Fence	124
Window Polygon	124
Crossing Polygon	124
Clear All	125
Filter	125
Reset Filter	126

<b>Layers</b>	<b>126</b>
Add Layers	126
Delete Layers	127
Merge Layers	128
Scale Layer X/Y	129
Sort Layers	129
<b>Align Layers</b>	<b>129</b>
<b>Delete</b>	<b>130</b>
<b>Move</b>	<b>130</b>
<b>Copy</b>	<b>131</b>
<b>Copy to layers</b>	<b>132</b>
<b>Rotate</b>	<b>133</b>
<b>Mirror / Flip</b>	<b>134</b>
<b>Array</b>	<b>135</b>
<b>Polar Array</b>	<b>137</b>
<b>Scale</b>	<b>138</b>
<b>De-Embed</b>	<b>138</b>
<b>Stretch</b>	<b>139</b>
<b>Split</b>	<b>139</b>
<b>Vertex</b>	<b>139</b>
<b>Trim</b>	<b>139</b>
<b>Boolean</b>	<b>139</b>
<b>Explode</b>	<b>139</b>
All	139
Flashes	139
Traces	140
Step and Repeat	140
Text	140
<b>Properties</b>	<b>141</b>
<b>3 View</b>	<b>143</b>
<b>Redraw</b>	<b>143</b>
<b>Zoom</b>	<b>143</b>
<b>Pan</b>	<b>144</b>
<b>Fill</b>	<b>144</b>
<b>Snap</b>	<b>144</b>
<b>Ortho</b>	<b>145</b>
<b>Object View Filter</b>	<b>145</b>
<b>Layer Display</b>	<b>145</b>
<b>External Centroids</b>	<b>145</b>
<b>Translucent</b>	<b>145</b>
<b>Highlight Active Dcode</b>	<b>145</b>
<b>Origin Icon</b>	<b>145</b>
<b>Film Box</b>	<b>145</b>
<b>Inside Step &amp; Repeat</b>	<b>145</b>
<b>Job Profile</b>	<b>145</b>
<b>Backside</b>	<b>146</b>
<b>Grid</b>	<b>146</b>
<b>Markup</b>	<b>146</b>
<b>Customize Toolbars</b>	<b>146</b>
Toolbar	146
Command Window	146
Design Navigator	146
Status Bar	146
<b>Full Screen</b>	<b>147</b>

<b>Command Console</b> .....	<b>147</b>
<b>4 Add</b> .....	<b>147</b>
<b>Pad</b> .....	<b>147</b>
<b>Trace</b> .....	<b>148</b>
<b>Circle / Arc</b> .....	<b>148</b>
<b>Polygon</b> .....	<b>149</b>
<b>Rectangle</b> .....	<b>150</b>
<b>Ellipse</b> .....	<b>150</b>
<b>Wide Path</b> .....	<b>151</b>
<b>Text</b> .....	<b>152</b>
<b>Multi-Line Text</b> .....	<b>153</b>
<b>Dimension</b> .....	<b>155</b>
<b>Markup</b> .....	<b>155</b>
Markup Line.....	155
Markup Rectangle.....	156
Markup Ellipse.....	156
Markup Polygon.....	156
Markup Text.....	157
Markup Arrow.....	157
Markup Layers.....	157
Markup Settings.....	158
<b>Drill Hit</b> .....	<b>159</b>
<b>5 Job</b> .....	<b>159</b>
<b>Insert Job...</b> .....	<b>159</b>
<b>New Job...</b> .....	<b>160</b>
<b>Delete Job...</b> .....	<b>160</b>
<b>Flatten Job Hierarchy...</b> .....	<b>160</b>
<b>Push-Pop Job</b> .....	<b>160</b>
Push Into.....	160
Pop Out .....	160
Quit PPJ .....	160
<b>Open Selected Job...</b> .....	<b>161</b>
<b>6 Info Menu</b> .....	<b>161</b>
<b>Query</b> .....	<b>161</b>
All .....	161
Dcode .....	161
Net .....	162
<b>Find</b> .....	<b>162</b>
Net .....	162
Dcode .....	163
NC Tool .....	163
Polarity .....	164
Find Text.....	164
Find Insert.....	164
Find Component.....	164
DFM Errors.....	164
Netlist Comparison Errors.....	165
<b>Measure</b> .....	<b>165</b>
Point-Point.....	165
Object-Object.....	166
Net-Net .....	166
<b>Rulers</b> .....	<b>166</b>
<b>Reports</b> .....	<b>166</b>

<b>Generate PDF Error Report</b> .....	167
DRC / DFM Violations .....	167
Compare Net Errors .....	167
Join Errors .....	167
Sales RFQ .....	167
Fabrication Drawing .....	167
<b>Generate Text Error Report</b> .....	167
<b>Generate CSV Error Report</b> .....	167
<b>Generate Error Drawing</b> .....	167
List .....	168
<b>7 Fabrication</b> .....	<b>169</b>
<b>Silkscreen Optimizer</b> .....	169
<b>Copper Balancing</b> .....	170
<b>Grow / Shrink</b> .....	170
<b>Pin Hole Elimination</b> .....	171
<b>Teardrop Creation</b> .....	171
<b>Remove Pads (Isolated / Stacked)</b> .....	172
<b>Remove Covered Data</b> .....	172
<b>Drill</b> .....	172
Gerber to Drill .....	172
Drill to Gerber .....	172
<b>SMT Stencil</b> .....	172
Homebase Pad Conversion .....	172
C-Pad Conversion .....	172
Split Pads .....	172
Scale Pads .....	172
Bow tie Pads .....	172
Pad Pitch Adjustment .....	172
Heel and Toe Edits .....	173
Add Epoxy Bars .....	173
<b>Generate Missing Layers</b> .....	173
Paste .....	173
Mask .....	173
Border .....	174
Create Drill Layer (PTH) .....	174
<b>Copper Area Calculation</b> .....	174
<b>Create Fabrication Drawing</b> .....	174
<b>8 Assembly</b> .....	<b>175</b>
<b>Build Component</b> .....	175
<b>Insert Component</b> .....	176
<b>Find Component</b> .....	176
<b>Component Detection Wizard</b> .....	176
<b>Component Analysis</b> .....	176
<b>View Component Violations</b> .....	177
<b>Apply External Centroid</b> .....	177
<b>Components Table</b> .....	177
<b>BOM Table</b> .....	177
<b>9 Tools</b> .....	<b>177</b>
<b>DFM</b> .....	177
DFM Check (PRO) .....	177
DFM Check (Quick PDF) .....	179
DFM Rules Management .....	179
Find Minimum Gaps Only .....	180

Find Minimum Widths Only.....	181
Drill Checker.....	181
Browse DFM Errors.....	181
<b>Nets</b> .....	<b>181</b>
Netlist Extraction.....	181
Compare Nets.....	181
Find Net Shorts.....	181
View Net Errors.....	182
Apply Nets.....	182
Netlist Table.....	182
<b>Compare Jobs and Layers</b> .....	<b>182</b>
Compare Layers (Graphical FAST).....	182
Compare Jobs (Graphical FAST).....	182
Compare Layers (Physical).....	182
Browse Compare Errors.....	182
<b>Convert to</b> .....	<b>183</b>
Flash (Automatic).....	183
Flash (Interactive).....	183
Flash (Selective).....	184
Flash (Custom Aperture).....	184
Polygon .....	185
Rectangle.....	185
Circle .....	186
Job .....	186
<b>Join Wizard</b> .....	<b>186</b>
Join (Automatic).....	186
Join (Interactive).....	187
View Join Errors.....	187
<b>Polygon Utilities</b> .....	<b>188</b>
Chop Polygons.....	188
Fix Invalid Polygons.....	188
Remove Tiny Segments.....	188
<b>Composite Layers</b> .....	<b>189</b>
Composite Wizard.....	189
Build... ..	189
Flatten... ..	189
Separate.....	189
Open (for Editing).....	189
Close .....	189
<b>Boolean Wizard</b> .....	<b>189</b>
Assign to Group A.....	189
Assign to Group B.....	189
Add .....	190
Subtract .....	190
Intersect .....	190
Difference.....	190
Invert .....	190
Size .....	190
<b>Film Wizard</b> .....	<b>190</b>
<b>Clip Area...</b> .....	<b>195</b>
<b>10 Setup</b> .....	<b>195</b>
<b>Aperture Table</b> .....	<b>195</b>
<b>NC Tools Table</b> .....	<b>196</b>
<b>Layers Table</b> .....	<b>197</b>

Layer Sets.....	199
Detect Layer Types.....	201
Synchronize Layers.....	202
Aperture Rules Editor .....	202
Blind and Buried Via Setup.....	204
Set Grid .....	204
Set Origin.....	204
Set Background Images .....	204
Customize .....	204
Shortcut Keys.....	205
Options .....	205
General .....	205
GUI .....	206
Rendering.....	208
Film Box .....	209
Synchronize Setup.....	210
Restore Defaults.....	210
<b>11 Applications .....</b>	<b>210</b>
New .....	210
Open .....	211
Run .....	211
Add App to Menu .....	211
<b>12 Window .....</b>	<b>211</b>
Workspace Browser.....	211
Custom Aperture Browser.....	211
Close All Pages .....	211
<b>13 Help Menu .....</b>	<b>211</b>
Help Topics .....	211
FAB 3000 - User Support Center .....	211
FAB 3000 - Online Tutorials and Demos .....	211
CAM Features Comparison .....	212
Become a Numerical Reseller or Partner .....	212
Visit Numerical Innovations.com .....	212
Compile TrueType Fonts .....	212
Compile AutoCAD SHX Fonts .....	213
Enter License Key .....	213
Activate .....	213
De-Activate.....	213
FLEXnet Key.....	213
Check for Update .....	215
About .....	216

**Part VII Import Matrix File 218**

1 Example Matrix File #1 .....	220
2 Example Matrix File #2 .....	224
3 Example Matrix File #3 .....	228
4 C++ Source Example for calling Matrix File .....	233
5 Layer Types .....	234
6 NCTOOLS array .....	234
7 DFMCHECKER array .....	235

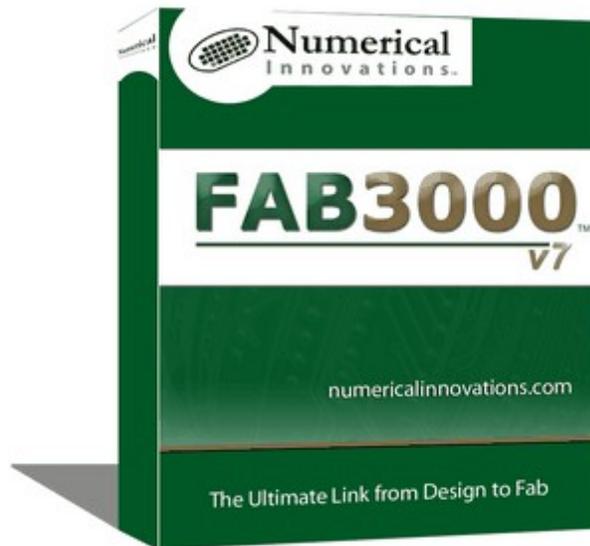
<b>Part VIII Data Formats</b>	<b>239</b>
1 Gerber .....	239
2 ODB++ .....	240
3 DXF .....	240
4 DWG .....	242
5 IGES .....	242
6 Image .....	242
7 GDSII .....	243
8 Postscript .....	243
9 Large Monochrome Bitmap .....	243
10 Drill/Rout .....	243
11 STL .....	243
<b>Part IX Purchase and Support</b>	<b>246</b>
1 How do I place an Order? .....	246
2 Product Pricing .....	247
3 Numerical Maintenance Plan .....	247
4 How do I contact Support? .....	248
5 License Key / Activation Code Problems .....	248
<b>Part X Movie Tutorials</b>	<b>250</b>
<b>Index</b>	<b>251</b>

# Foreword

**Part**



# 1 FAB 3000 - Help Guide



Get Immediate On-Line Help - Visit the FAB Support Forum:  
For the most up-to-date help documentation; Visit our website support forum: [FAB Support Forum](#)



**Part**



---

## 2 What's New!

Click the following link to view the [Revision History](#).



**Part**



## 3 Getting Started

### 3.1 System Requirements

Below are a listing of the minimum required system requirements.

**Windows (32-bit and 64-bit):**

XP/2003/Vista/7/8

**Linux (64-bit):**

kernel 2.6

---

**System Requirements:**

- ▶512+ MB RAM recommended.
- ▶120+ MB of free hard disk space.
- ▶Super VGA (1024 x768) display.
- ▶High Color (16 bit) graphics card.
- ▶4X CD-ROM drive with 32 bit drivers.
- ▶Mouse. Intelli-mouse (wheel mouse) recommended.
- ▶Internet connection recommended.

### 3.2 Installation

For detailed FAB 3000 installation and setup instructions, please view the following link: <http://www.numericalinnovations.com/pages/fab-3000-installation-instructions>

### 3.3 Licensing Options

FAB 3000 offers multiple licensing solutions.

**Single User - Activation Code (Internet Activated):**

Receive one activation code. 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. Visit this link for detailed instructions:

<http://forums.numericalinnovations.com/post/How-do-I-move-my-license-to-another-computer-4903592>

**Multi User - Activation Code (Internet Activated):**

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

**Single User - FLEXnet (Node-Locked):**

A hardware locked license where the software is locked per machine (no internet activation required).

[Node Locked - License Key Installation](#)<sup>[8]</sup>

**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.

Software provided under this option may be installed on and processed by the Server computer located at a site indicated on the invoice, and may be installed on and simultaneously processed by the number of workstation and/or personal computers ("Concurrent Users") indicated on the invoice, which have electronic access to the Server computer located at any site indicated on the invoice and which are accessing the same data concurrently from more than one computer up to the predefined limit. [Network - License Installation](#)<sup>[11]</sup>

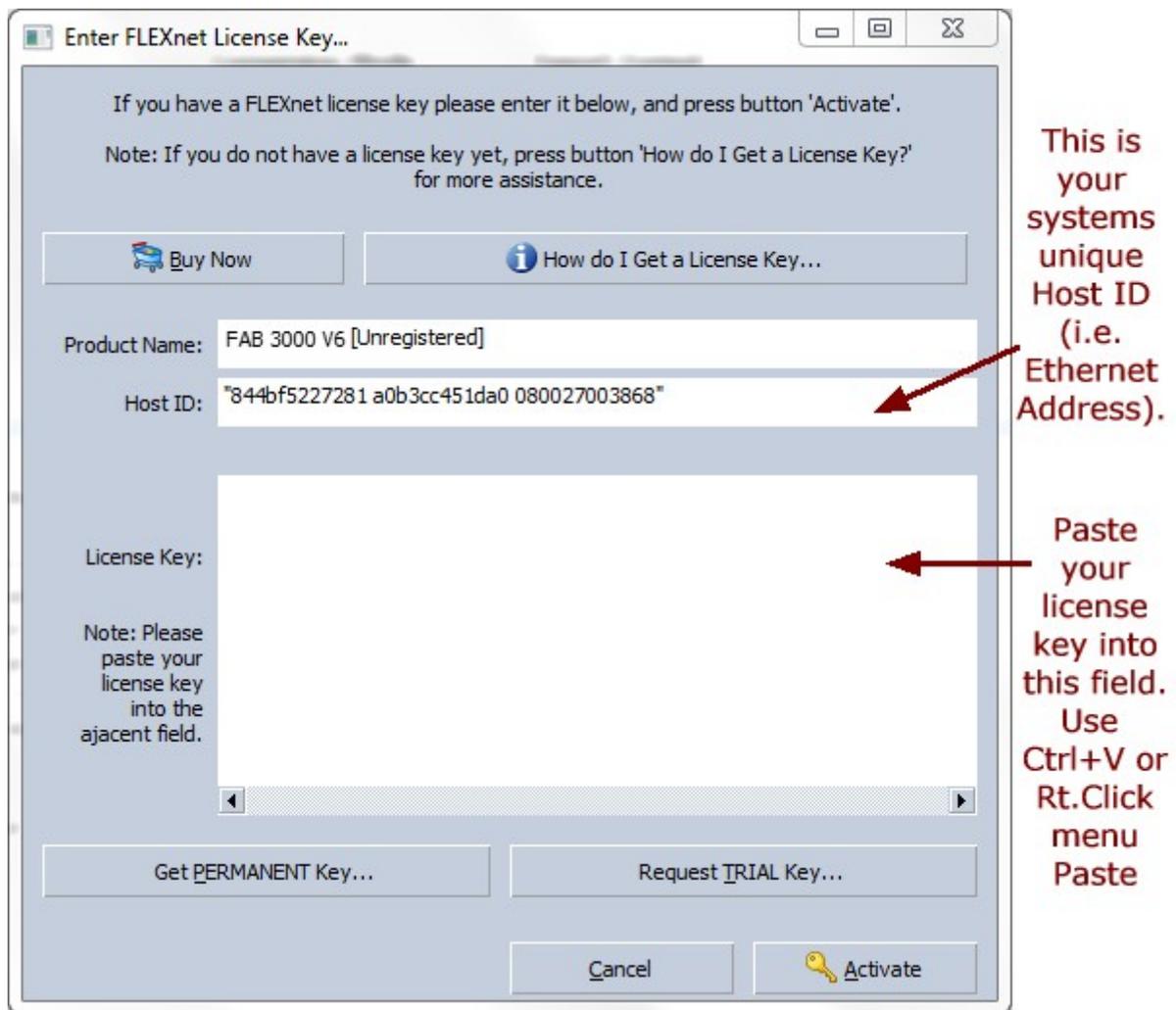
**Enterprise/Site - FLEXnet (Unlimited):**

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.

### 3.4 Node Locked and Demo - License Key Installation

FAB 3000 uses FLEXnet Publisher (formerly FLEXlm) for license management. FAB will accept either a valid License Key or License File (\*.lic). Without a Valid License Key or File (\*.lic), FAB will start in unregistered mode. A Node Locked License is also referred to as a Single User License.

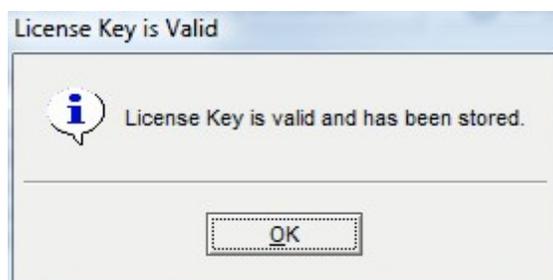
---



### Installing a FAB 3000 License Key:

- A. Start FAB 3000.
- B. Go to FAB 3000 menu: Help | Enter License Key | [Enter Flexnet Key...](#)
- C. Paste your license key into the license window, and press button "**Activate**"

Note: If your license key is valid, you will receive a success message (see below).



Note: If your license key is invalid, you will receive this message (see below). We recommend that if

you are having problems to press 'Yes; to review the error log. If you feel this is an error, please send a screen capture of the error log to [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com) and we'll be happy to resolve the license key issue.



---

### Installing a FAB 3000 License File (\*.lic):

Place your valid license file (\*.lic) into the FAB 3000 license folder:

Windows

ex. C:\Numerical Innovations\FAB3000V6

Linux

ex. /opt/Numerical/FAB3000

Note: FAB 3000 will automatically check the FAB 3000 Installation first for a valid license file.

## 3.5 Activation Codes: Single User, Multi-User - License Installation (Windows)

Follow the instructions below to activate your standard license key.



### Installing a FAB 3000 License Key:

- A. Start FAB 3000.
- B. Go to FAB 3000 menu: Help | Enter License Key | [Activate...](#)<sup>[213]</sup>
- C. Paste your activation code into the edit box (use key 'Ctrl' + 'V').
- D. Create a Password (you make this up - must be at least 4 characters).
- E. Enter your email address.
- F. Press button "**Next**"

FAB 3000 Activation...

NumericalInnovations.com

Activation Information:

Activation Code:

Password:

E-mail:

Please enter your Activation Code, create a password, and provide your email.

Note: You may copy/paste the activation code into field above - using "Ctrl+V".  
If necessary, click button Proxy Settings to enter Proxy Server settings.

Proxy Settings...

Cancel Back Next

**Important Note:** Make sure you retain the Password and Email used during the initial activation. It will be required should you ever want to install FAB 3000 onto another computer.

## 3.6 Network - License Installation

Numerical Innovations, uses FLEXnet Publisher (formerly FLEXlm) for concurrent license management. All new and existing concurrent license customers will use FLEXnet Publisher to authorize each product user within their environment. (This does not apply to individuals who purchase individual user licenses.)

The procedure for installing and using FLEXnet Publisher is outlined completely in the following steps. Follow the steps in the order presented and read through each step before proceeding. You may wish to print this page to use as a guide.

 Note: If you need assistance, contact our Product Support at [support@numericalinnovations.com](mailto:support@numericalinnovations.com)

 Note: Instructions for using FLEXnet Publisher are detailed in the FLEXnet Licensing End User Guide (PDF).

### 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**  
=====

1. You should have received a Numerical Innovations Network Bundle which contains the following files:
  - a. network.txt (this document)
  - b. Imgrd (Server Daemon)
  - c. numinno (vendor Daemon)
  - d. Imtools, Imutils (FLEXnet supplied utilities)
  - e. flexnet\_licensing\_end\_user\_guide.pdf (FLEXnet Publisher instructions)
  - f. vcredist\_x86.exe (for Windows 32-bit only)
2. Extract the files to a temporary folder.
3. Copy all the files from the temporary folder to the designated folder you choose on the FLEXnet server. This folder will be referred to as the FLEXnet folder in subsequent steps.

For Windows platforms, the default location to copy to is C:\FLEXnet. For UNIX and Linux platforms, select a location. Create these folders as needed.

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

=====  
**Generate your License File (\*.lic)**  
=====

4. To generate your License file, requires the following information about your FLEXnet server (Platform, Hostid, Hostname)
5. Select the platform of your FLEXnet server (Windows-32, Windows-64, Linux-32, Linux-64).
6. Determine your hostid. You can use the following procedure:
  - a. Open a command line window.
  - b. Navigate to the FLEXnet folder.
  - c. Run Imutil Imhostid. Note the space between Imutil and Imhostid.
  - d. If the system returns more than one hostid, use the first one that appears on the list.
7. Enter the hostname. If you are not sure of the FLEXnet server's hostname, you can run hostname from a command line on the FLEXnet server.
8. Fill out the Numerical Innovations - "Request License Form":  
[http://www.numericalinnovations.com/license\\_request.html](http://www.numericalinnovations.com/license_request.html)

Note: You will receive your license key (via e-mail) within one business day.

=====  
**Save and Configure your License File (\*.lic)**

- =====
9. Once you receive your License File (\*.lic), save it to the FLEXnet folder.
  10. Go to the FLEXnet folder and open license.lic in a text editor.
  11. Edit the vendor path to match the path for the FLEXnet folder. When you are finished it should look something like:

```
VENDOR numinno C:\FLEXnet\numinno.exe (Windows)
```

where C:\FLEXnet is the path to the FLEXnet folder and numinno.exe is the daemon that checks each installation of the Numerical Innovations, LLC product to make sure it is authorized to run.

The UNIX and Linux path should reflect the path and folder you chose in Step 3.

Leave the rest of this file as is. Save and close it.

=====

### **Start the FLEXnet Server and Install the Client**

=====

12. Start the FLEXnet server by opening a command line on the FLEXnet server while in the FLEXnet folder and typing:

```
lmgrd -c license.lic
```

13. Set the environment variable on each client as follows:

On Windows:

- a. Go to Control Panel > System > Advanced Tab > Environment Variables. A dialog box appears.
- b. Click New under System Variables. Another dialog box appears.
- c. Enter LM\_LICENSE\_FILE in the top field.
- d. Enter port@hostname in the bottom field. The default port is 27000. The hostname is the same one entered in Step 7. (Note: This step may not be required)
- e. Click OK on each open dialog box until they are all closed.

On UNIX or Linux, type `export LM_LICENSE_FILE=port@hostname` on the command line. The default port is 27000. The hostname is the same one entered in Step 7.

=====

### **Install to the Client**

=====

14. Install the Numerical Innovations, LLC product onto each local client machine for which you have a license, by following the installation instructions that came with the product CD (or download).
15. Place a copy of the License File (\*.lic) in the products installation folder ie. "C:\Program Files\Numerical Innovations\..." (or whatever directory you installed the software into).

At this point, you can start the Numerical Innovations, LLC product on each client machine.

```
=====  
Trouble-Shooting  
=====
```

Windows Only - If you receive error message stating "Missing MSVCR80.dll" (or similar) When starting the vendor daemon "numinno.exe", you will need to install the Microsoft Redistribution utility: "vcredist\_x86.exe". This will correctly include all required DLL's that you require.

## 3.7 Launching FAB 3000

Please select your OS (below), for specific information to launch software.

[Windows](#) <sup>[14]</sup>  
[Linux](#) <sup>[14]</sup>

### 3.7.1 Windows OS

Please perform the following to launch for Windows.

#### Tutorial

1. Left Click 'Start' button.
  2. Select: All Programs
  3. Select: Numerical Innovations | FAB 3000
- You will now see FAB 3000 software begin to execute.*

For detailed FAB 3000 installation and setup instructions, please view the following link: <http://www.numericalinnovations.com/pages/fab-3000-installation-instructions>

### 3.7.2 Linux OS

Please perform the following to launch for Linux.

Set the Library environment variable: LD\_LIBRARY\_PATH, to include the path of the Numerical Innovations shared libraries (using a local shell program type the following).

```
▶LD_LIBRARY_PATH=/opt/Numerical/lib  
▶export LD_LIBRARY_PATH  
▶/opt/Fab3000/fab3000
```

 Note: For this example we assume that you have installed Fab3000 into the folder: /opt/Numerical

 Note: You may also create a short startup script for this executable, which first updates 'LD\_LIBRARY\_PATH', and then runs executable. Please consult your platform reference material for more information.

 Note: Kernel 2.4 (or higher) is required for Visual Chip (Linux 32-bit)

 Note: Kernel 2.6 (or higher) is required for Visual Chip (Linux 64-bit)

For detailed FAB 3000 installation and setup instructions, please view the following link: <http://www.numericalinnovations.com/pages/fab-3000-installation-instructions>

## 3.8 Technical Support

Visit our website support: [Technical Support](#)

## 3.9 Getting Additional Help

 For the most up-to-date help documentation visit the [FAB 3000 - Online Help Manual](#)

 Search and Find instant answers from other FAB 3000 Users, visit the [FAB 3000 - User Support Forum](#)

 Contact Numerical Innovations Support Team - [Submit a Support Ticket](#)

**Part**



**IV**

## 4 Learn Guides

Step-by-Step instructions for learning specific features.

---

### 4.1 General

#### 4.1.1 Viewing & Selection Basics

Performing the viewing basics. An introduction to the essential viewing features of the editor including: Editor Modes, Workspace Navigator, and the Layers Table.

---

#### Viewing Basics:

- To turn selective layers **ON** or **OFF**, left-click over the light bulb icon next to each layer name within the Layers Panel. If the light bulb is dark , the layer will be turned **OFF**. If the light bulb is yellow , the layer will be turned **ON**. To adjust layer color, left-click on the adjacent color box. Right-click for more layer options.
- To **Isolate** a single layer (ie. display one layer at a time), right-click in the layer panel, and select 'Isolate' from the menu.

- To **Zoom Window**, set Fab3000 Editor Mode to Zoom Window mode . Left-click anywhere in the editor, and then drag across the editor to cover any area that you need a closer view. If you just left-click, Fab3000 will automatically zoom-in 2X.

You may also activate Zoom Window by either:

1. Menu: View | Zoom | Window
2. Right-clicking in the editor to display the menu, then select menu item: Zoom | Window.

- To **Dynamically Zoom** In or Out, use the **mouse scroll wheel** (if available). Scrolling the mouse wheel forward will automatically Zoom-In. Scrolling the mouse wheel in the other direction will zoom out.
- To **Dynamically Pan**, hold the **mouse middle button** (if available), and drag your mouse in any direction.
- To **View another Job/Panel**, goto the menu: Windows | Workspace Browser..., and double-click any item. A new view will be created in the editor along with a new tab.
- To **Open a Selected Insert**
  - 1) Select any insert you want to view.
  - 2) Goto Menu item: Job | Open Selected Job...

The selected job insert will now be displayed in the editor.
- To **Leave or Reset** any function or selection set, **Press 'ESC' key** at any time. You may also **'Right-Click'** inside the editor, and select menu item **'Cancel'**.

## Selection Toolbar Icons:

- To **Select Objects**, set Editor Mode to Select mode . Move the mouse over any Object and then Left-click. That object will now be highlighted, indicating that it has been selected and added to the **Active Select Set**.
- **Standard Selection Mode**  is the default used by the editor. If activated, each time you select an object, Fab3000 will remove all previous objects in the selection set.
- To **Add Objects to the Active Selection Set**, click on the toolbar button **Add Selection**  and select the objects. Those selected objects will be highlighted and added into the active selection set.
- To **Remove Objects from the Active Selection Set**, click on the toolbar button **Sub Selection**  and re-select the objects. Those re-selected objects will no longer be highlighted and removed from the active selection set.
- To **Clear the Active Selection Set** press click on the toolbar button **Clear Selection** . You can also press the key 'Esc', or right-click and select from menu: 'Cancel'. All objects will be removed from the active selection set.
- To **Add Selection Filter** press click on the toolbar button **Selection Filter** . You can also press the key 'Esc', or right-click and select from menu: 'Cancel'. Selection Filter will be reset, and all objects will be removed from the active selection set. [Click here for more information about Selection Filter](#) <sup>[125]</sup>

## Selection Basics:

- To **Select Window**, goto menu item: Edit | Selection | Window. Left-click anywhere in the editor, and then and then drag across the editor to cover any selection area you require. Any Objects completely contained inside your defined window will be selected.

You may also activate Select Window by either:

- 1) Right-clicking in the editor to display the menu, the select menu item: Selection | Window.
- 2) or Simply pick an empty area in the editor window, and drag the selection window to the right.

- **Dynamic Selection Window** Make sure to set Editor Mode to Select mode . Then, simply Left-Click on any empty area in the editor and drag across the editor (to the Right) to cover any selection area you require. The selection Window will be blue to let you know it is in Selection Window mode.

- To **Select Crossing Window**, goto menu item: View | Selection | Crossing Window. Left-click anywhere in the editor, and then left-click the next point. Any Objects contained inside or intersecting your defined window will be selected.

You may also activate Select Crossing Window by either:

- 1) Right-clicking in the editor to display the menu, the select menu item: Selection | Crossing Window.
- 2) or Simply pick an empty area in the editor window, and drag the crossing window to the left.

- **Dynamic Selection Crossing Window** Make sure to set Editor Mode to Select mode . Then, simply Left-Click on any empty area in the editor and drag across the editor (to the Left) to cover any selection area you require. The crossing Window will be green to let you know it is in Selection Crossing Window mode.
  - To perform **Quick Query**, move the mouse over any Object and then Left-Click. In the status bar, basic information about the selected Object will be displayed. You may repeat this step as many times as necessary.
  - When selecting Polygons and Traces, make sure to select their **Edges**.
- 

### 4.1.2 Open/Save Workspaces

Step-By-Step instructions for Saving and Opening workspaces. A workspace is a collection of the jobs, panels, folders, files that you create with Fab3000 and stored in a single file (\*.wrk). Opening & Saving workspaces is much faster than importing data, and it allows you to save any modifications that you have made.

- This Guide assumes that you have already imported data into the workspace (see [AutoLoad Wizard](#)<sup>[79]</sup> for quick loading).
  - To **Save Workspace**, goto menu item: File | Save Workspace...
  - Choose a **file name (\*.wrk)** to save your workspace, and select '**Accept**'. Fab3000 will now begin to save your workspace.
- 
- To **Open Workspace**, goto menu item: File | Open Workspace...  
Please Note: Open Workspace requires that Fab3000 is empty and does not have any other data or workspaces currently open.
  - Choose a **Workspace file name**, and select '**Accept**'. Fab3000 will now begin to open the workspace.

### 4.1.3 Using Measure

Measure distances inside the editor.

---

### **Measure Point-Point:**

- To begin **Measure** goto menu item: Info | Measure | Point-Point, or select Measure from the toolbar.
- Select anchor point, move mouse over your desired location and left-click.
- Now you may move the mouse location anywhere you desire. A window will display the distance & angle of the mouse location relative to the anchor point.

**Note:** Activating **Snap: Object** is an excellent way to ensure that your measurements are exact. To activate Snap: Object, goto menu item: View | Snap | Snap to Object.

**Note:** To force **Orthogonal (90 Degree)** snapping, goto menu: View | Ortho | 90 Degrees.

- Left-clicking in the editor again will reset Measure, so that you may select another anchor point.

---

### **Measure Object-Object:**

- To begin **Measure** goto menu item: Info | Measure | Object-Object, or select Measure Object-Object from the toolbar.

- To select the base Object, move mouse over your desired Object and left-click. A white boundary will outline the base Object.

Note: The base Object can be nested within an insert.

- Now you may move the mouse over the next Object and left-click. A window will display the distance & angle of the mouse location relative to the anchor point.

**Note:** Measure Object-Object keeps the **Base Object active** until you Escape (pressing 'Esc') or reset (right-click, and select 'Cancel').

- Selecting another Object will now measure the distance between the existing base Object and the new selected Object.

---

### **Measure Net-Net:**

- To begin **Measure** goto menu item: Info | Measure | Net-Net, or select Measure Net-Net from the toolbar.

- To select the base Object, move mouse over your desired Object and left-click. A white boundary will outline the base Object.

Note: The base Object must contain a net. If it does not, see: [Netlist Extraction](#) 

- Now you may move the mouse over the next Object and left-click. A window will display the distance & angle of the mouse location relative to the anchor point.

**Note:** Measure Net-Net keeps the **Base Object active** until you Escape (pressing 'Esc') or reset (right-click, and select 'Cancel').

- Selecting another Object will now measure the distance between the existing base Object and the

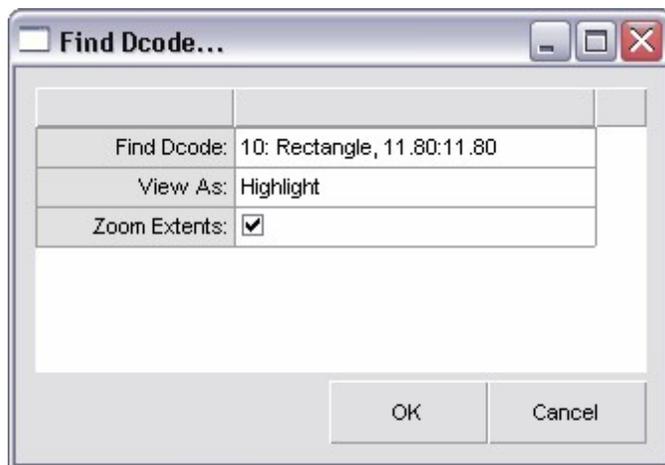
new selected Object.

#### 4.1.4 Finding Dcodes

Step-By-Step instructions for finding Dcodes within an active job. This Guide assumes that you have already imported gerber file(s) into the workspace (see [AutoLoad](#) <sup>79</sup> for quick loading).

##### Command Sequence

1. Activate [Find Dcode](#), by going to menu item: Info | Find... | Dcode
2. Select from the "Find Dcode" drop-down list which dcode you want to view.
3. Determine how to View the results. Choose either Highlight or Isolate.



 Note: All Objects that are using the Dcode (you selected) are now highlighted and zoomed in the editor.

 Note: If you do not see any highlighted objects, make sure you have all layers turned ON that are using the Dcode.

4. Press key 'ESC' (or Right-Click menu: Cancel) to exit Find.

#### 4.1.5 Using the Properties Dialog

Step-by-Step instructions to view & change object properties.

##### Command Sequence

1. Select the object you want to view properties.

 Note: This function requires that you select objects before proceeding.

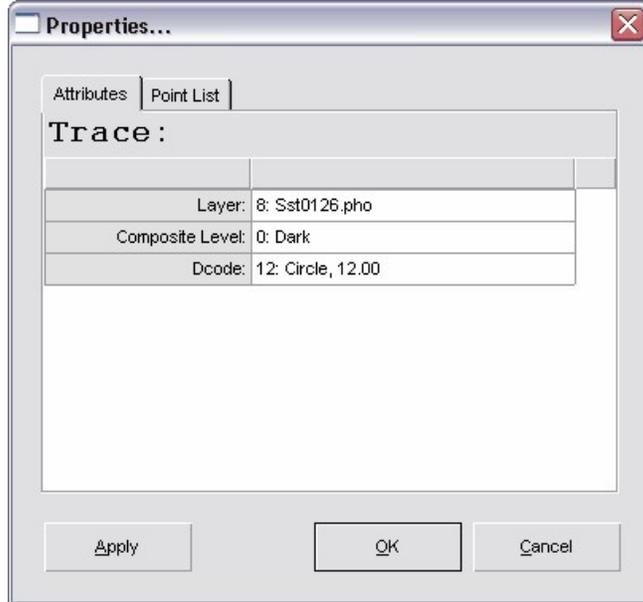
 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>17</sup>

 Note: To select polygons or traces, pick the edges.

2. Select menu: Edit | Properties...

3. All Properties of the selected object will be displayed in the properties dialog.

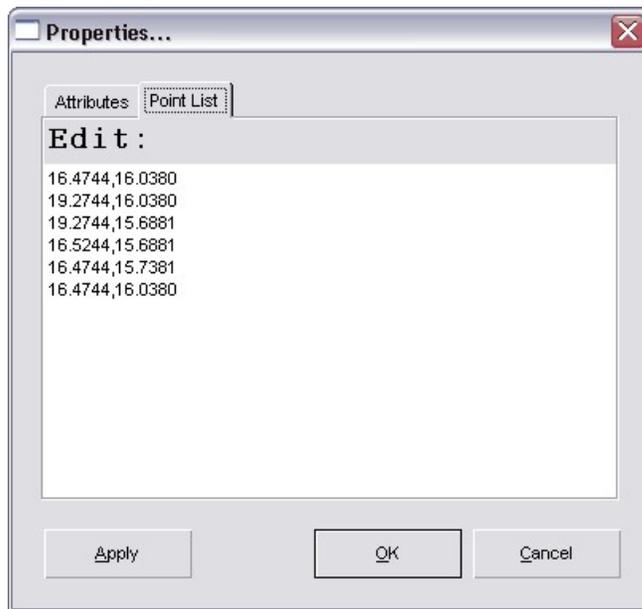
(screen capture of properties dialog for selected trace)



4. You may change any of the properties by clicking in the adjacent column (such as layer).

5. To modify the point list of an object click the tab Point List.

(screen capture of properties dialog "point list")



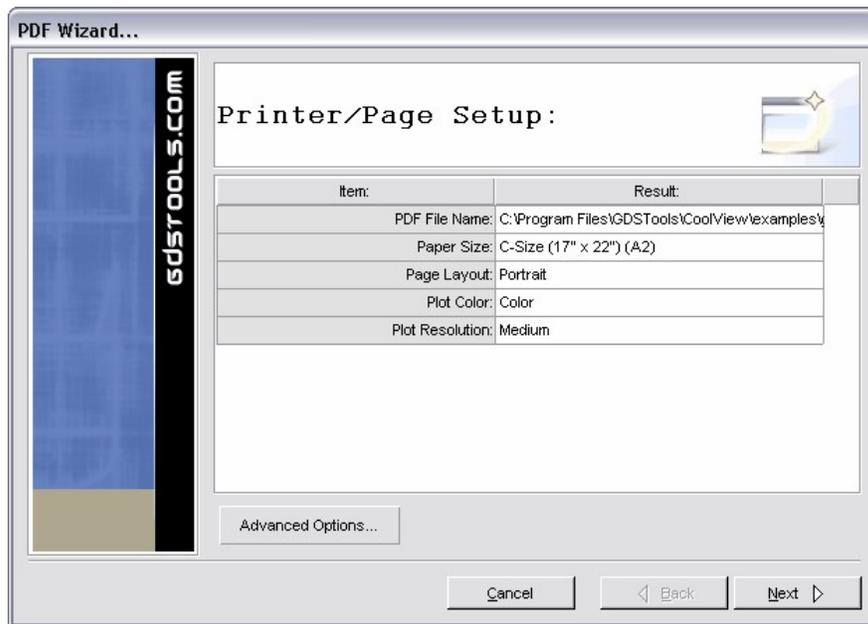
6. You may change, add, or delete any of the points and the object will be updated

 **Note:** To quickly activate the properties dialog, double-click your mouse over any object.

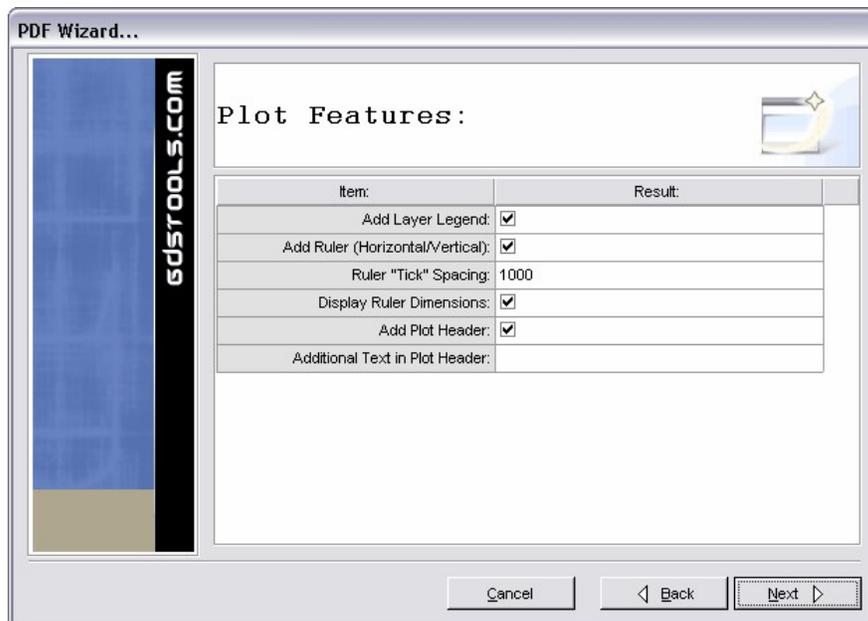
#### 4.1.6 Plot to PDF

Plot active design to an optimized PDF file.

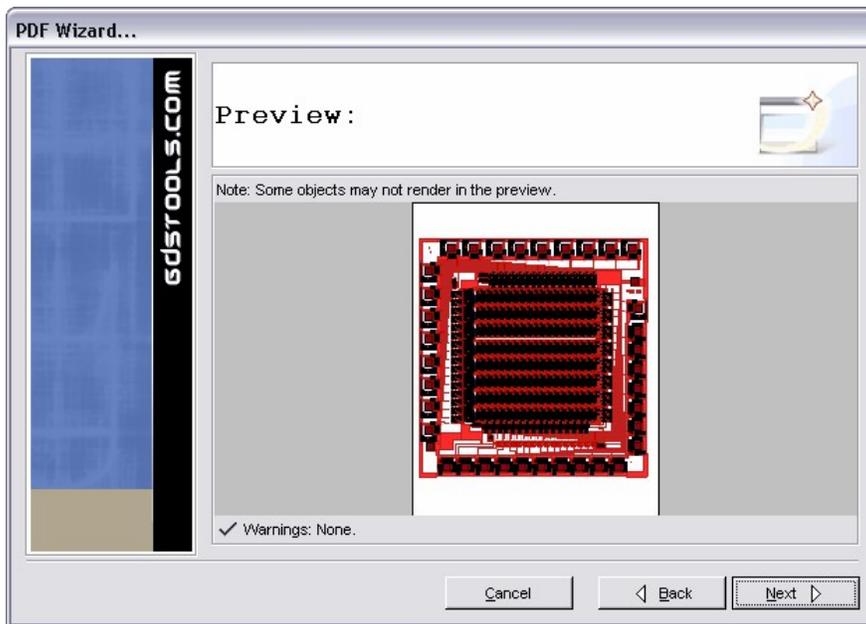
- To begin **Plot to PDF** goto menu item: File | Plot to PDF...
- When the **Printer/Page Setup** options appear, change the **Paper Size** from A-Size to C-Size or any other size you desire. Keep the rest of the default settings.
- Select **Next** to continue.



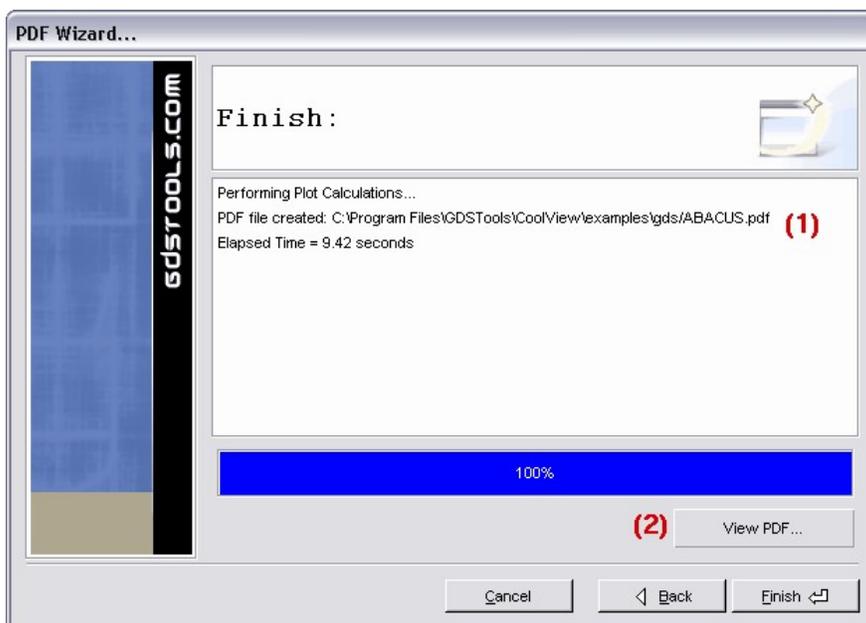
- When the Plot Features page appears, keep the default settings.
- Select **Next** to continue.



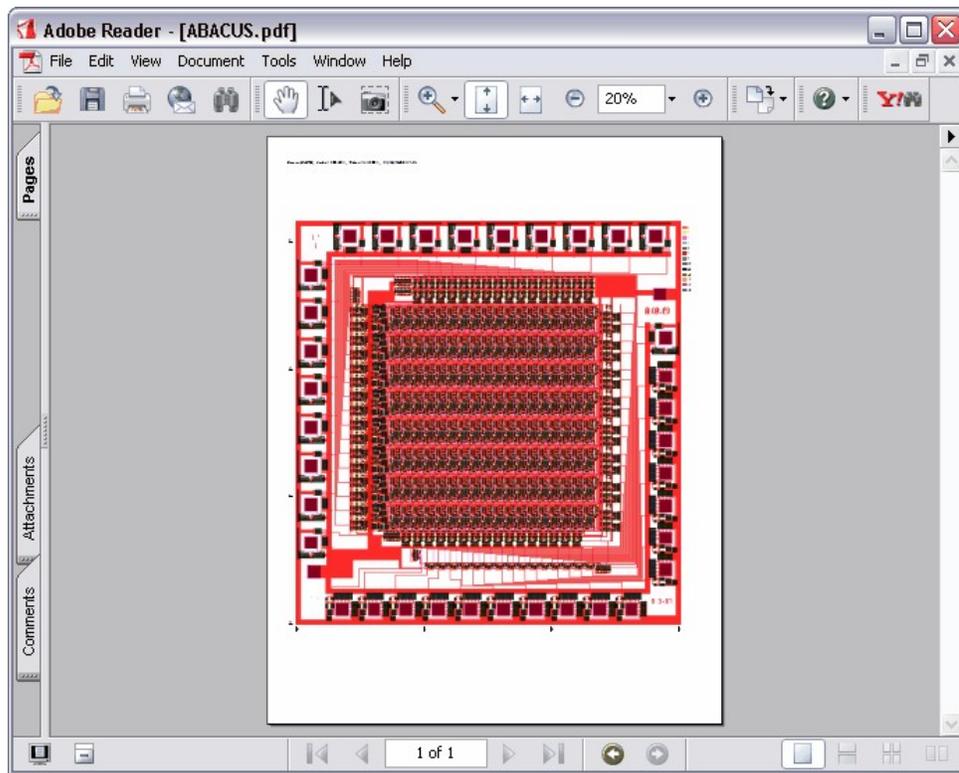
- Verify that Preview appears correctly.
- Select **Next** to continue. Fab3000 will begin to run, and start creating the PDF file.



- Once your PDF file has been created, the finish page will be displayed. The location and name of the PDF file will be displayed (1). You may now send, share, or view this PDF file with anyone
- Select **Finish** to exit.



- **Windows Users Only:** A button named 'View PDF' (2), is available to automatically spawn Adobe's Acrobat Viewer (if available).



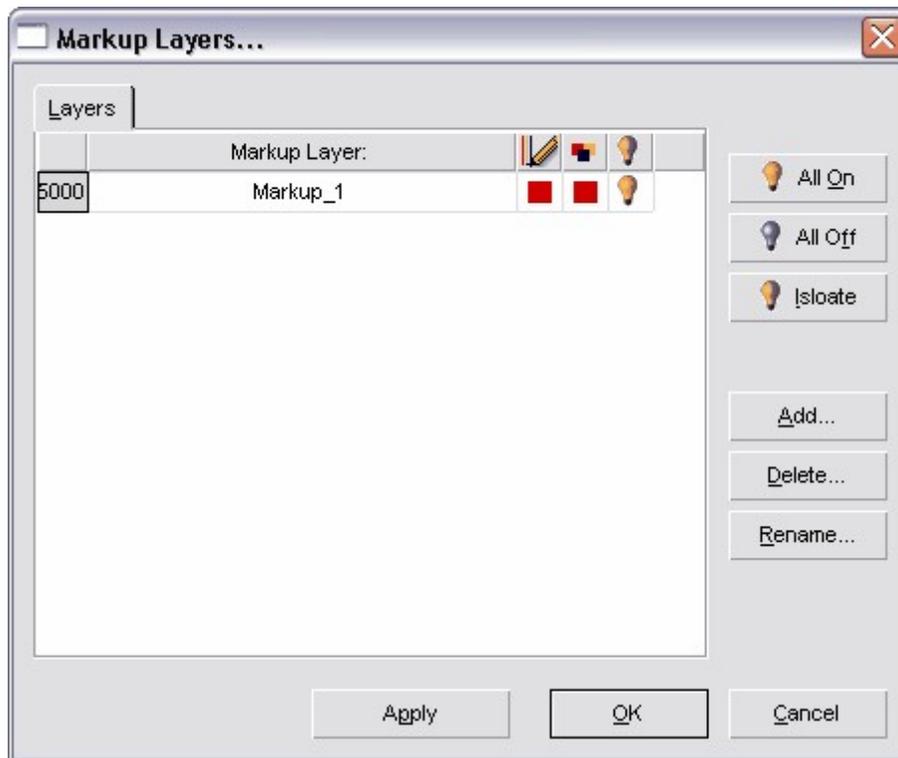
#### 4.1.7 Using Redline Markup

Step-By-Step instructions for using Redline Markup. Redline markup allows you to call-attention-to areas of a design. This feature is very useful for design reviews or when bringing attention to others about a particular region of a design is required.

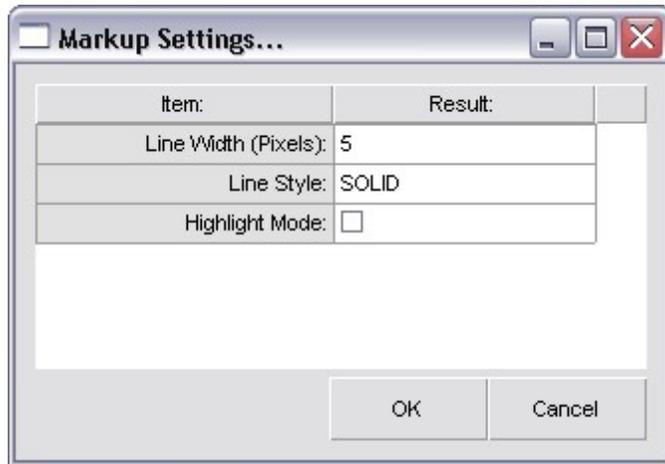
- This Guide assumes that you have already imported a design into the workspace (see [AutoLoad](#) [79] for quick loading).

##### To Start Redline Markup:

- Create a [Markup layer](#). Go to menu: Markup | Markup Layers...
- Click the button '[Add...](#)', and enter the layer name you want assign. In this example 'Markup\_1' was used.



- Once you have created a markup layer, you are now ready to begin adding markup Objects to your design. However before we begin adding markup Objects, it is a good idea to review the settings that effect how these markup Objects will appear in our design. All markup settings are defined in the [Markup Settings...](#) dialog.
- To modify markup settings, go to menu item: **Markup | Markup Settings...**
- The markup settings dialog includes three options:
  - Line Width (Pixels)** - The is the width that will be used (in current display pixels) for markup Objects: Line, Arrow, Rectangle, Ellipse, etc.
  - Line Style** - Choose between Solid or Dashed lines. Markup Objects with a large line width will not be able to display the dashes properly, and it best used with 0-width markup Objects.
  - Highlight Mode** - Allows the markup Objects to either be solid (Unchecked), or Highlighted-See Through (Checked).



- In this example we will be adding markup Objects: Ellipse, Arrow, and Text.
- The **Markup Ellipse** provides an excellent way to circle an entire area. To add, go to menu: Add | Redline Markup | Markup Ellipse, and select a corner on the outside of the area you want to circle, and then drag diagonally across to complete the ellipse.
- The **Markup Arrow** provides an excellent way to point to an area or markup Object. To add, go to menu: Add | Redline Markup | Markup Arrow, and select a start point on the outside of the area you want to circle. Then add any remaining points that you require. The starting point and next point will determine the direction of the arrow.
- The **Markup Text** provides an excellent way to give instruction or explain in detail the markup changes. To add, go to menu: Add | Redline Markup | Markup Text, and select a corner for the text box location, and drag diagonally across to complete the box. You will then be prompted to enter in the text, and text height (in Pixels)
- **Note:** Markup Objects do not always have to be red, and you can change their colors per layer in the **Markup Layers** dialog. Also you may create as many markup layers as you require, and they may be toggled On/Off (just as any other normal layer).
- **Note:** When Markup Objects are visible, they will be included on **PDF Wizard & Image exports** - this makes it easy to send your markups to anyone you require. Markups are also saved in the workspace, and may be transferred to other users as well.

## 4.2 Import

### 4.2.1 Using AutoLoad

Step-by-Step instructions for using the AutoLoad feature. AutoLoad Wizard will automatically import all Gerber and Excellon Drill files found within a selected folder. This makes importing multiple files a snap, and eliminates the time needed to load each file separately.

For more detailed AutoLoad information, please go to

- When starting AutoLoad Wizard, you will be greeted with the Select Folder Page, which requires you to select the folder to load the EDA files from.

- **Selection Folder Page:**

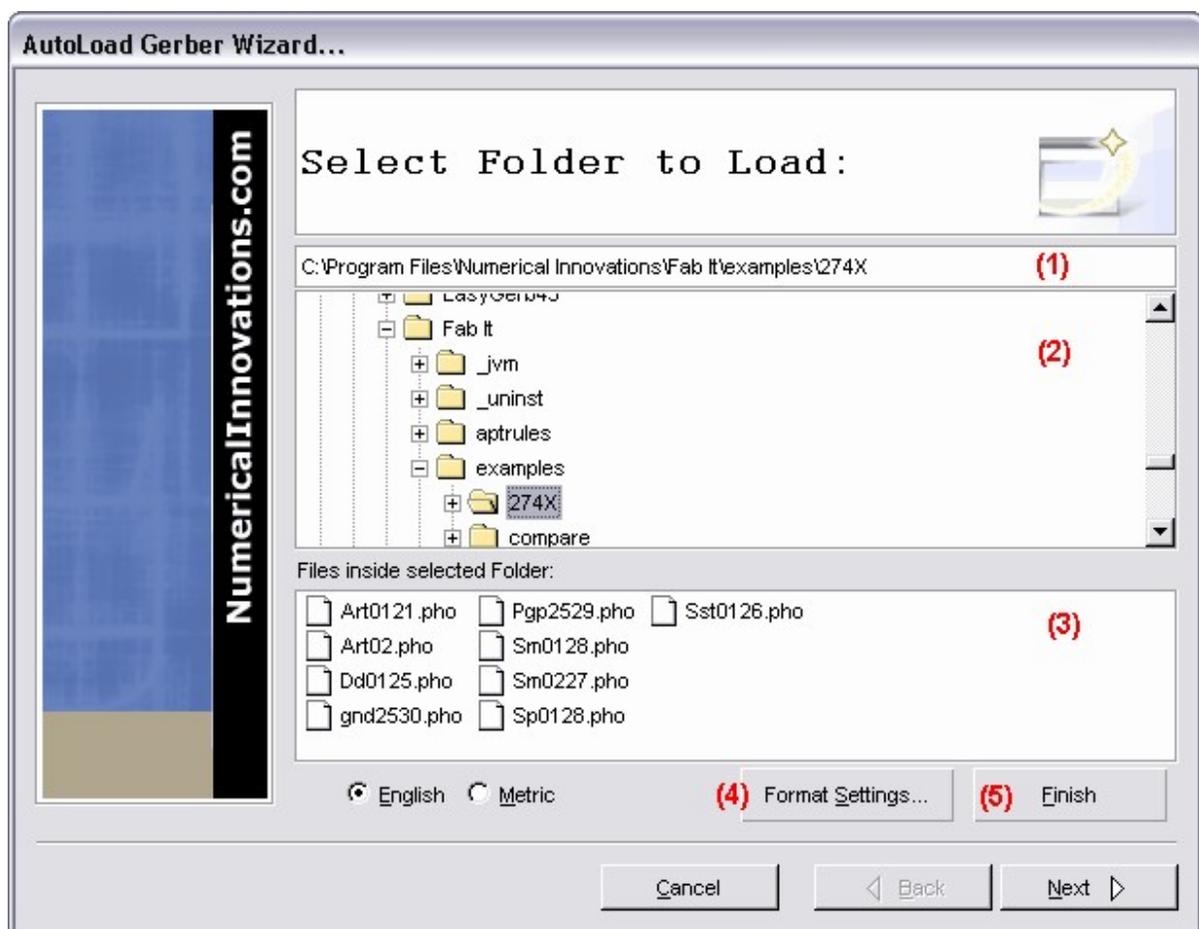
- (1) Selected Folder Path. You may also type the folder path you wish to use in this box also.

- (2) Folder Selection Tree. Use this to navigate throughout your system to find the folder you wish to import.

- (3) File list. This window allows you to see what the contents are inside the currently selected folder. This is used primarily as a reference tool to help determine the correct folder while using the selection tree.

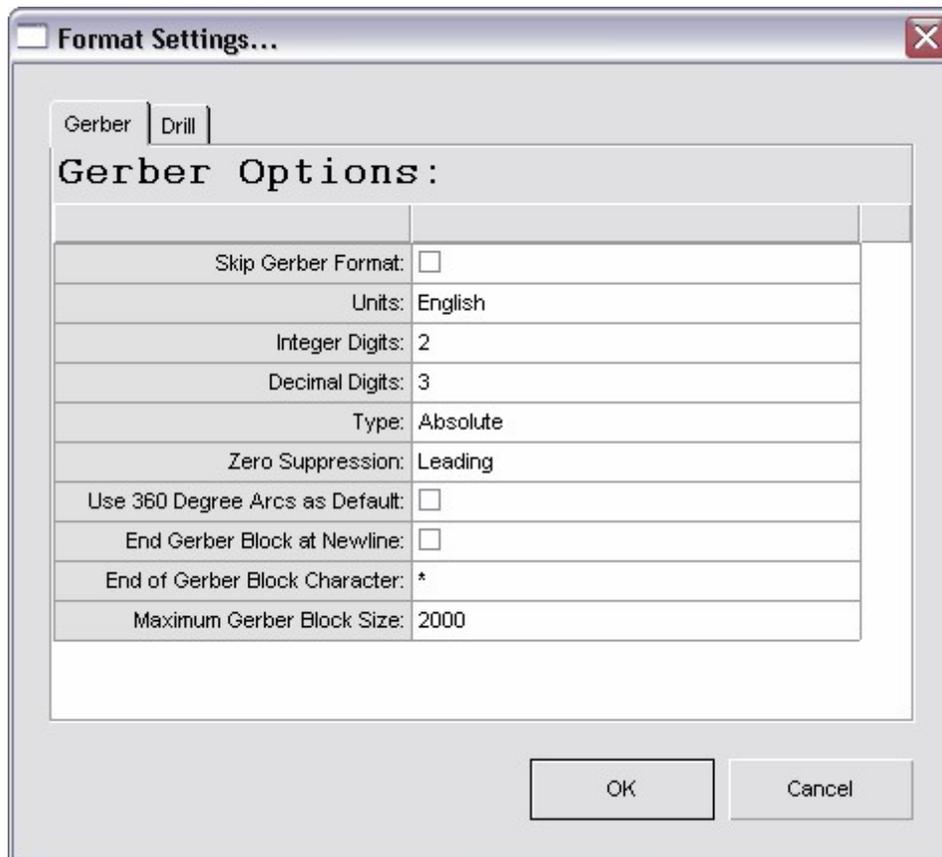
- (4) Format Settings. Lets you customize the default gerber import settings.

- (5) Skip to Finish. Allows you to pass up the 'File Description' page. This is a time-saving device, especially if you're already aware of the contents of the selected folder, and just wish to import everything.



- **Format Settings:**

The Format Settings dialog lets you custom setup the import defaults that AutoLoad Wizard will use while import the files from the selected folder. You also have the option to 'Skip ... Format', if you do not wish to include a particular EDA format type during Auto Import.



#### • The File Description Page:

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

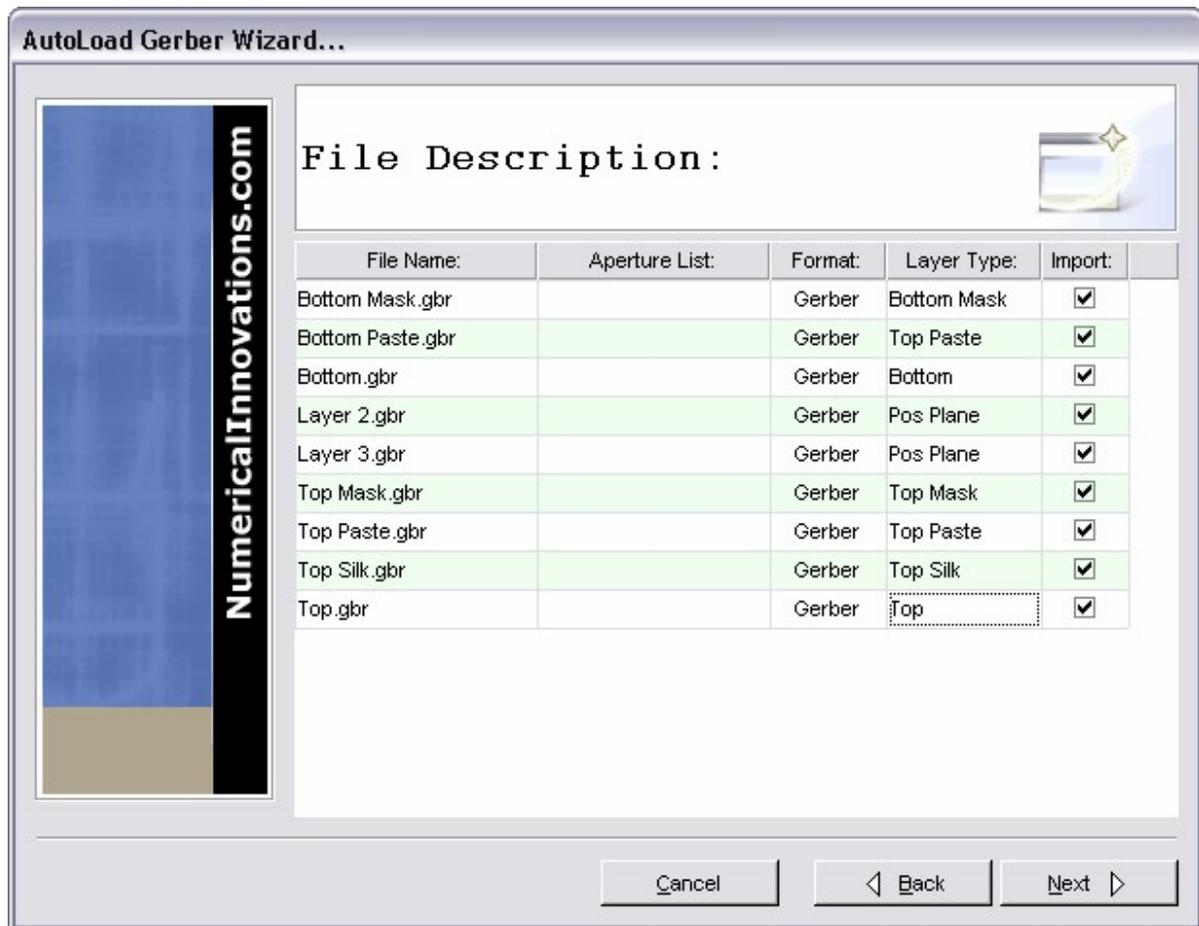
File Name - The name of the file located in the Autoload import directory.

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.

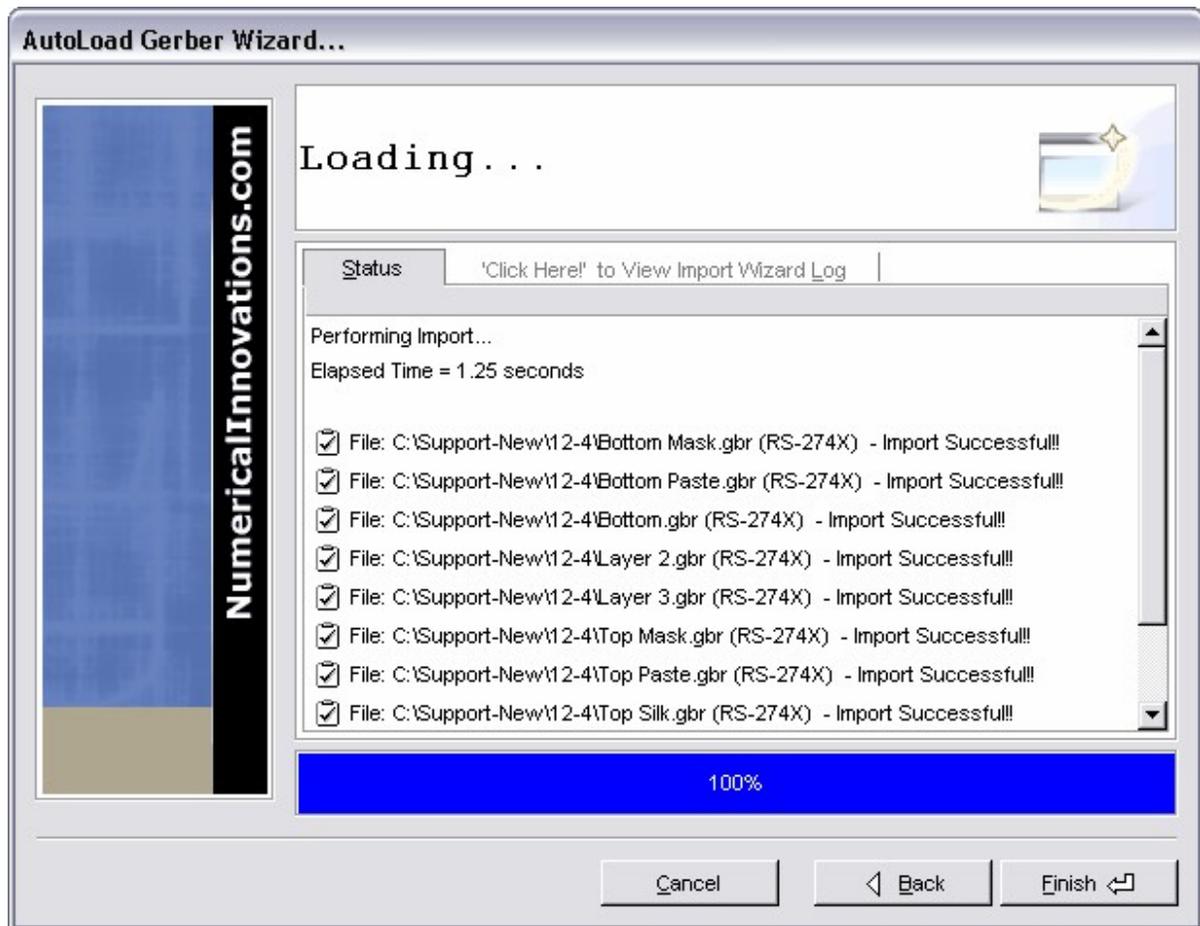
Layer Type - Select the corresponding layer types for each file.

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.



- **Finish Page:**

This is the final page of the AutoLoad Wizard. It will first display a progress bar for each file being imported. Once AutoLoad Wizard has completed, the final results will be outputted in the status tab. Note: For detailed information about each file loaded click on the Import Log Wizard.



#### ● Important Notice about Drill Importing with AutoLoad:

AutoLoad will only import NC drill files that have the tool sizes defined inside the file. AutoLoad scans each drill file beforehand, and if it cannot detect the tool size definitions in the file, it will be ignored; and you will be required to import the drill files separately using [Drill Import](#)<sup>[87]</sup>.

## 4.2.2 Importing DXF

Load the DXF file into the workspace. This Learn Guide will also be helpful for DWG import.

 Online Movie: [DXF import with de-embed and join checked with FAB 3000](#)

1. Select menu item: **File | Import | DXF**
2. You will be prompted to select the DXF file to import.

 **Note:** The DXF file will be scanned for Layers, Blocks, Styles, and more.

 **Note:** In most cases you may only need to verify the DXF Units & Scale Factor.

**Important:** DXF Files are unit-less, so it is important that you verify the DXF Units & Scale Factor, of your DXF file before importing it.

3. Verify DXF Layer Options.

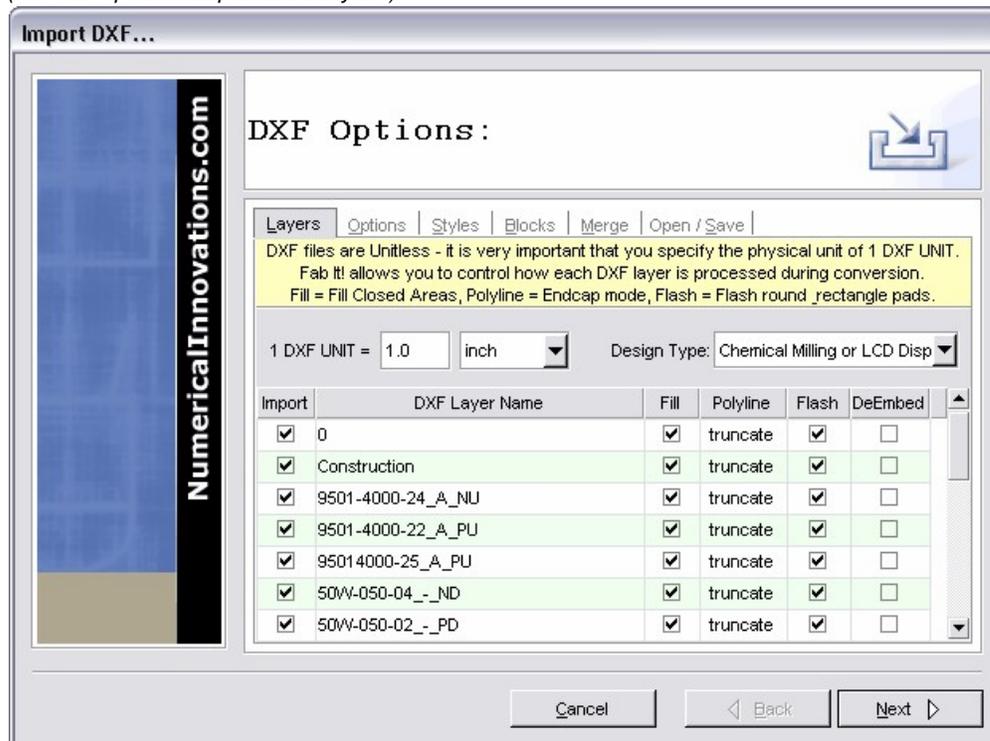
- **Choose "DXF Design Type".** This is an "Optional" feature which adjusts other DXF import settings to accommodate optimal conversion results in most cases.

- **Verify DXF Layers.**

**Note:** This list allows you to control how each individual DXF layer is handled during import.

- Check "Import" to include the layer during import.
- Check "Fill" to convert closed boundaries to filled-polygons.
- Choose "Polyline" mode, to adjust how polylines with width are handled during import. The three choices are: Extend, Truncate, and Round.
- Check "Flash", to detect circles & rectangles, and convert them to gerber flashes.
- Check "DeEmbed" Algorithm to verify polygon-inside-of-polygon (time consuming)

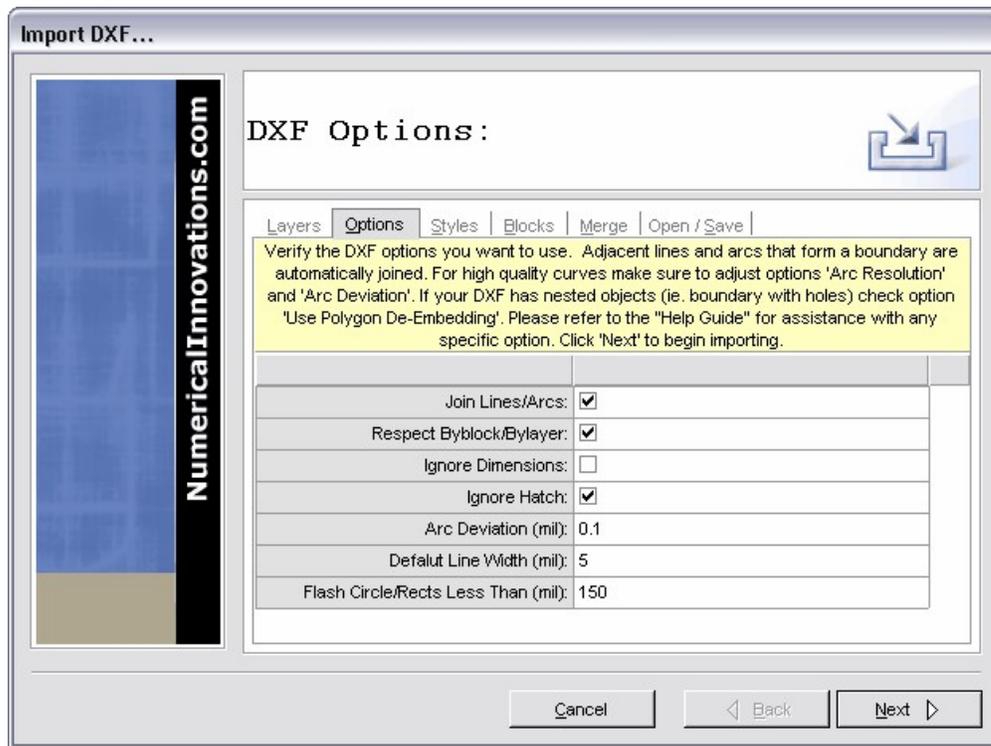
(Screen capture of Import DXF: Layers )



4. Verify DXF Options

**Note:** There are several options that can affect how a DXF file is imported (Join Adjacent Lines/Arcs, Arc-Resolution, Respect Byblock/Bylayer).

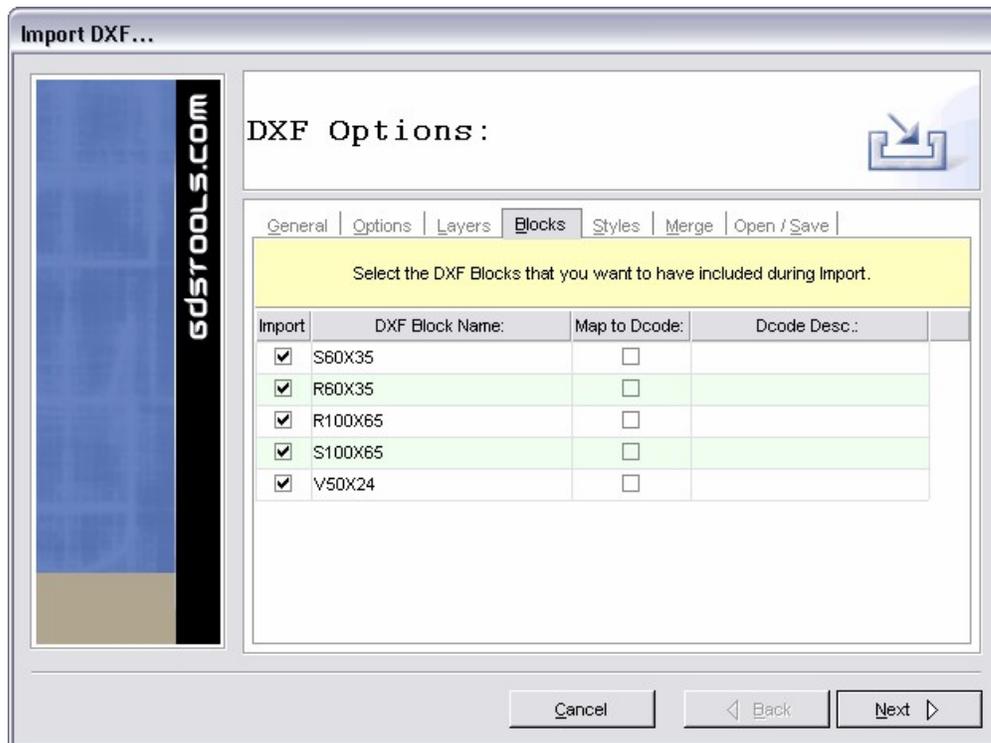
(Screen capture of Import DXF: Options)



#### 5. Verify DXF Blocks

**Note:** This list allows you to control how each individual DXF block is handled during import.  
- Check "Import" to include the block during import.

(Screen capture of Import DXF: Blocks)

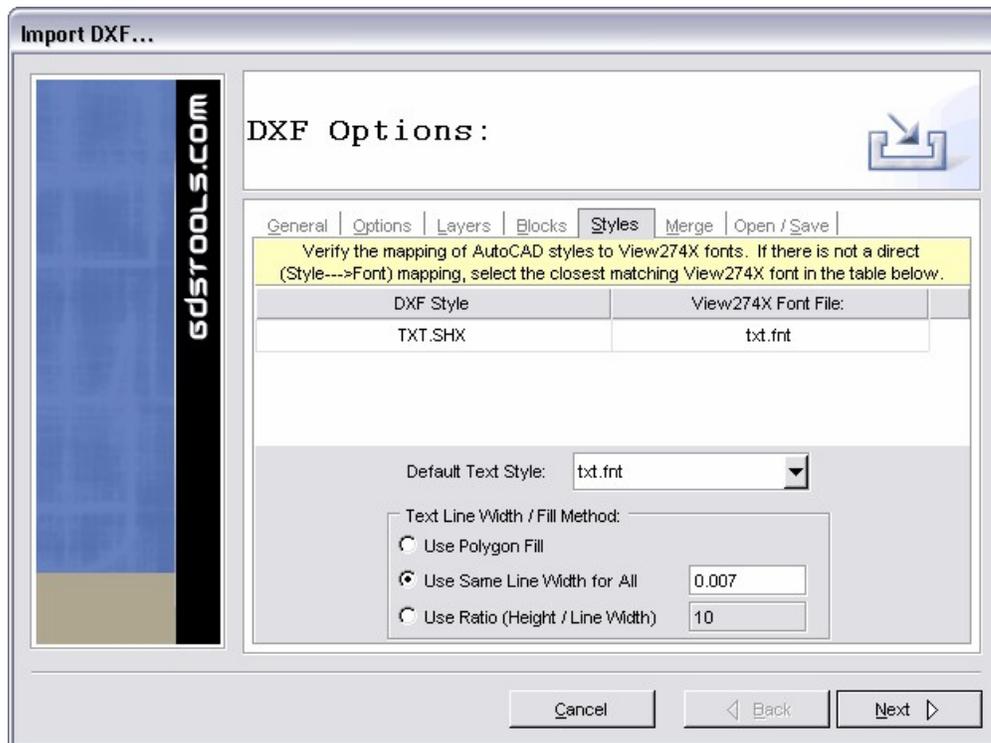


#### 6. Verify DXF Styles

**Note:** This list allows you to map each DXF style to a compiled font file during import.

- Choose a Default Text Style for any unknown dxf style.
- Review and assign the "Text Line Width/Fill method"

(Screen capture of Import DXF: Styles)

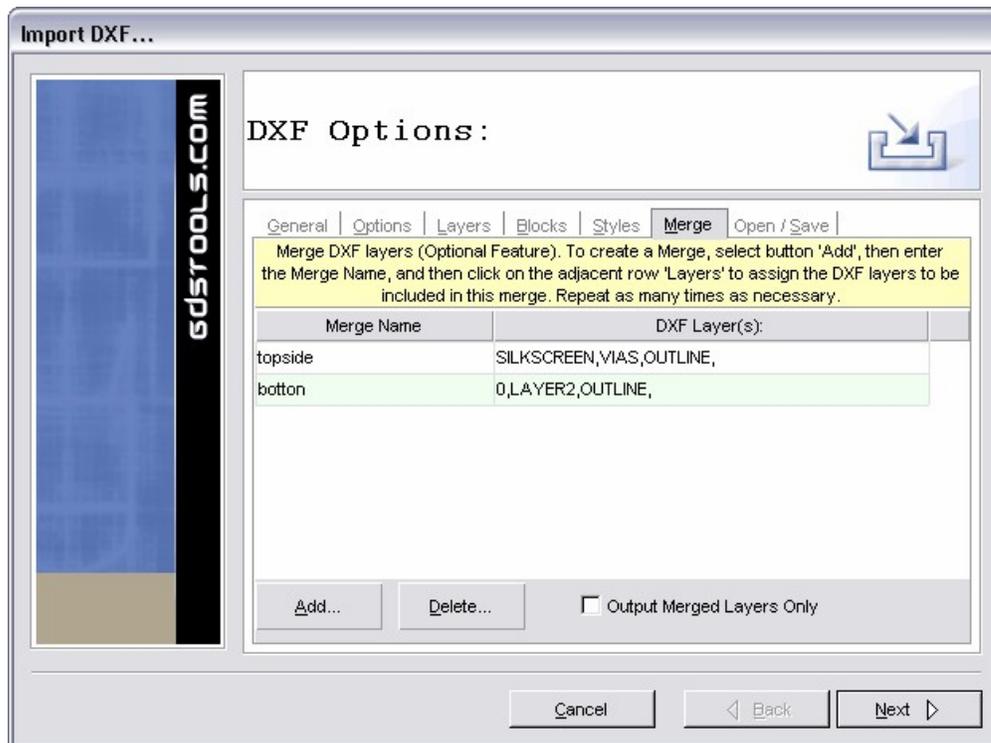


## 7. Merge DXF Styles

**Note:** This list allows you to merge DXF layers during import.

- Check "Output Merged Layers Only" to only write merged layers, and incoher individual DXF layers.

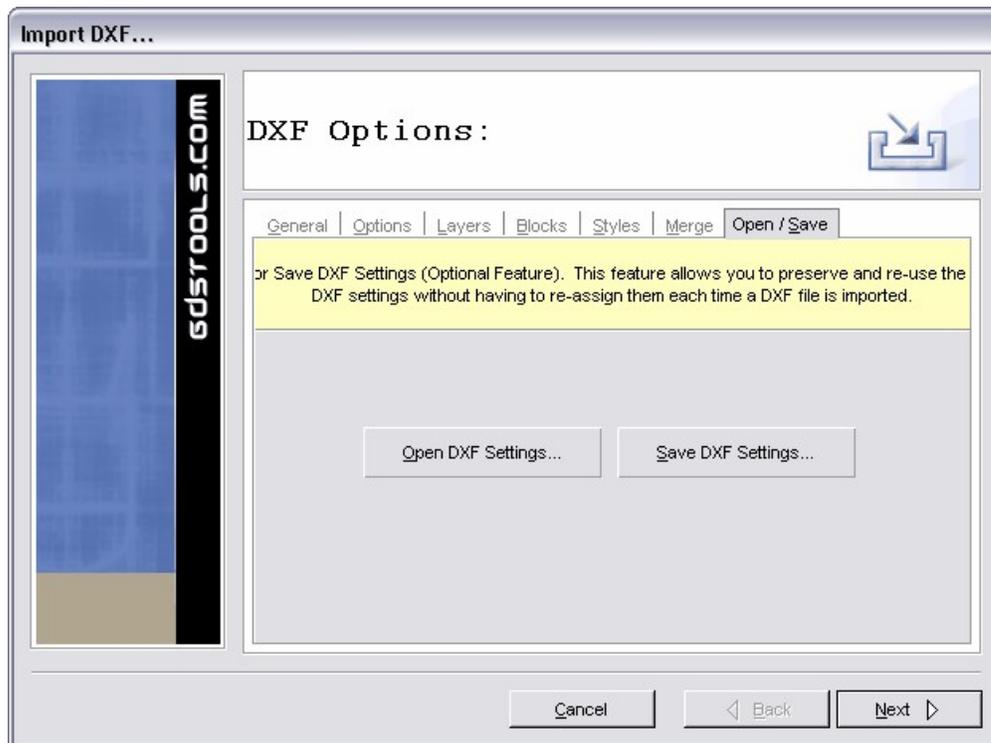
(Screen capture of Import DXF: Merge)



8. Open / Save DXF configuration.

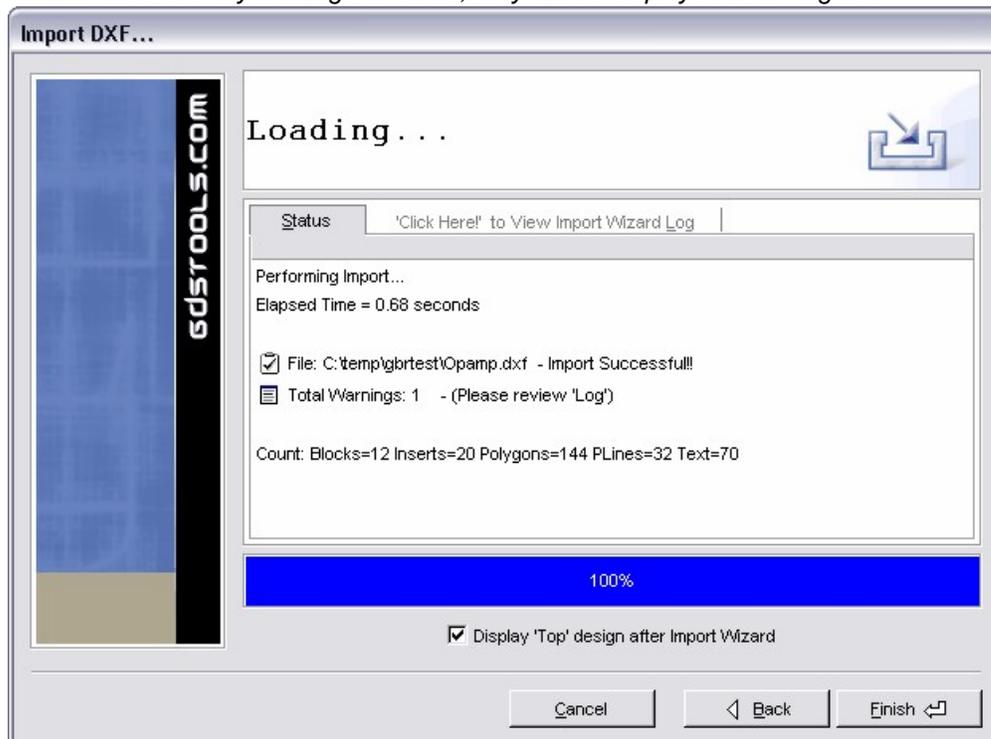
**Note:** This (Optional) feature allows you to open & save an existing DXF configuration file (\*.dxf), so that you may assign the exact same settings used previously. If you regularly perform DXF conversions re-using the DXF configuration will save time and limit mistakes.

(Screen capture of Import DXF: Open/Save)



- Select 'Next' to proceed.

*Note: If there are any warnings or errors, they will be displayed in the log list.*



- Select the **'Finish'** button, to view the DXF entities.

### 4.2.3 Importing ODB++

Load an ODB++ database into the Workspace. This example demonstrates how to load a compressed ODB++ file (normally in a tar ball \*.tgz) into Fab3000

 Online Movie: [How to skip a layer during ODB++ import.](#)

 Note: A "Job" in Fab3000 is the same thing as a "Step" in ODB++

#### Command Sequence

1. Select menu: File | Import | ODB++
2. Select the compressed ODB++ database file you want to import, and press "OK". The Import ODB++ wizard dialog box appears.
3. Verify ODB++ Import Options: Layers, Steps, and Options
4. Click "Next" to begin importing the ODB++ database.
5. Once ODB++ loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

 Note: If there are multiple steps in the ODB++ database, Fab3000 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.

 Note: Unlike other CAM software products, Fab3000 loads the complete ODB++ database including all steps (while maintaining hierarchy). To view all the steps imported into Fab3000, see the [Workspace Brower...](#)<sup>[21]</sup>

 Note: To view the Import log file, press key 'F4'

### 4.2.4 Importing Zip (Compressed) File

Automatically Loads Gerber and Drill data from a compressed file(.zip, .tar, .gz, .bz2, .7z)

## Command Sequence

1. Select menu: File | Import | Zip (Compressed File)..
2. Select the compressed file you want to import, and press "OK". The file is instantly uncompressed and the import wizard dialog box appears.

 **Note:** Fab3000 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. Click "Next" to begin importing the files.
4. Once loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
5. Click button "Finish" button to view the imported data.

 **Note:** To view the Import log file, press key 'F4'

## 4.3 Merge Jobs

The purpose of this section is to introduce you to the unique and powerful merging features of Fab3000

### Hierarchy Concepts:

Fab3000 is one of the only CAM software tools that can natively implement gerber data in a hierarchal fashion. Initially this concept of using hierarchy with Gerber files can seem confusing, but give this "Quick-Start" a chance and hopefully everything will become clearer. Note: If you have any questions, please feel free to contact us anytime: [Numerical Innovations Support Team](#)

#### **Essentially Fab3000 consists of two hierarchal resources: Job, and Insert:**

**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). Note: In ODB++ terms, a Step is the same thing as a Job (in Fab3000)

**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.

---

## Create New Job, and Import Gerbers/Drill:

### Creating a New Job:

1. Goto Menu: File | New | Job...

2. Select a name for the new Job. (Make it a name that is descriptive. Example: Part12212, etc.)  
*Once the new job is created, you will see a blank Job Editor screen.*

#### **Loading Gerber(s) & Drill(s) Automatically into the active Job:**

3. Goto menu: File | Import | AutoLoad...

*Select the folder which contains the gerber and drill files (for a single job).*

4. Click the Next button

*At this point there will be a slight delay while Fab3000 scans the directory to determine whether each file is either a Gerber or Aperture list. Then the "File Description" page is displayed. This page gives you the opportunity to manually define the aperture list to use (for RS-274D only), Layer Types, and import checkbox for each gerber layer.*

5. Once loading is complete, you will be able to view the imported data. If Fab3000 encounters any errors or warnings during AutoLoad, they will be displayed here.

*If Fab3000 encounters any errors or warnings during import, they will be displayed here.*

 Note: Click here for more information about [Using AutoLoad](#)<sup>[28]</sup>

 Note: If you have your Gerbers and Drills inside a compressed file (ie. \*.zip, \*.tar), use the feature [Importing Zip \(Compressed File\)](#)<sup>[39]</sup>

**If you have more than one design, and you would like to eventually merge them, please Repeat the above steps.**

 Note: Keep in mind, that each of your designs requires a separate Job.

 Note: 90% of all typical CAM editing (such as DRC check, Draw 2 Flash, Trim Silkscreen, etc.) is performed here - at the Job level.

### **4.3.1 Merge Jobs (Automatically)**

Automatically Merge multiple jobs for fabrication. This Guide assumes that you have already created your jobs and imported data (ie. Gerber, DXF, etc) into each Job (see [Merge Jobs](#)<sup>[40]</sup> for more information).

 Online Movie: [Merge Different Gerber jobs onto one panel using FAB3000](#)

 Note: You must have at least one job (with data) in the current workspace. This guide assumes that you have 3 jobs loaded into Fab3000 (see [Merge Jobs](#)<sup>[40]</sup> for more information).

#### **Creating a New Panel:**

1. Goto Menu: File | New | Panel...
2. Select a name for the new Panel.
3. Verify the panel dimensions: Width, Height, Margins, and Units.

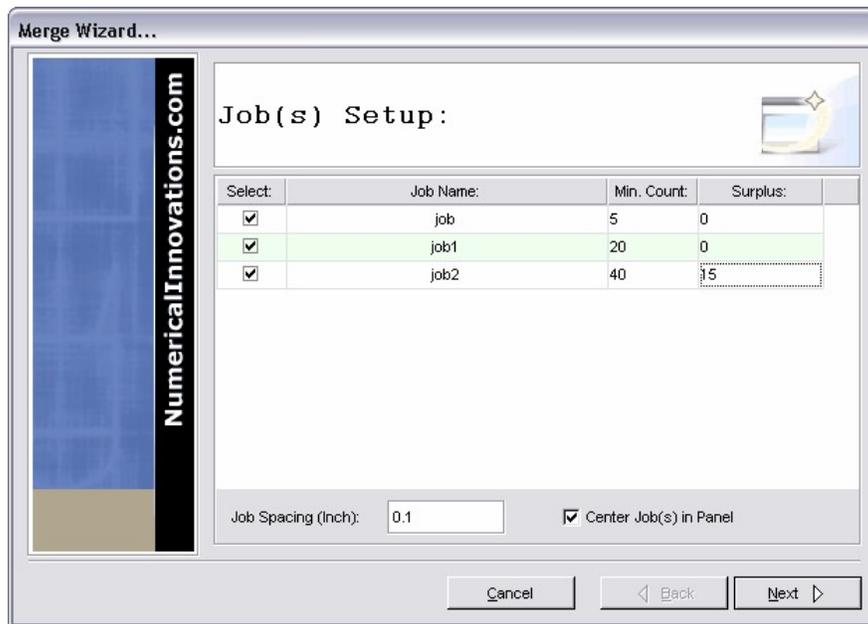
*Once the new panel is created, you will see a blank Panel Editor screen, with a outlined box representing the panel area.*

 Note: To modify the panel dimensions, goto the panel menu: Tools | Edit Panel...

#### **Start Merge Wizard:**

1. Goto menu: Tools | Merge Wizard...

*The Panelize Wizard dialog will now be displayed.*



2. Check all the Jobs that you want to include in the panel merge.

3. Enter the Minimum Count (and surplus) for each job.

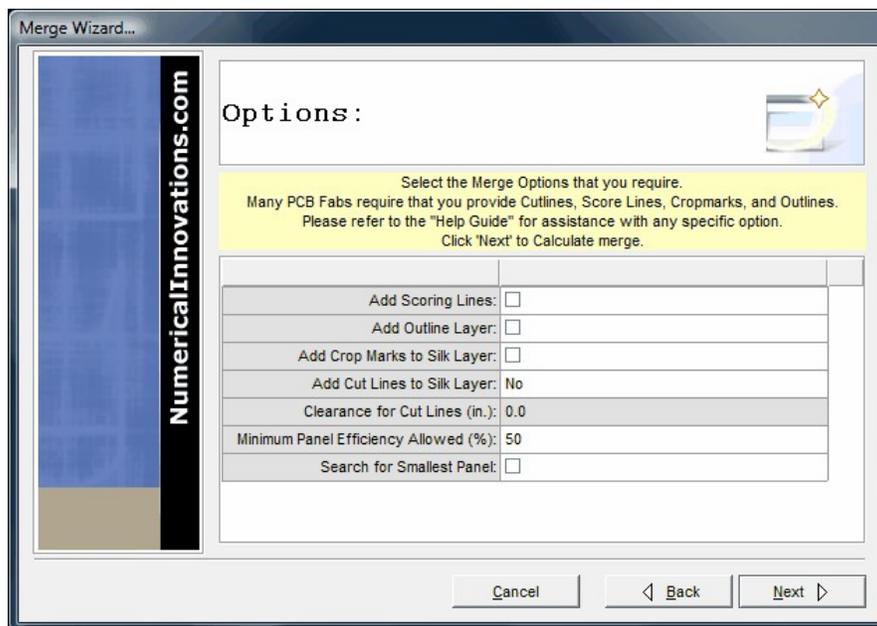
 **Note:** The last number is the surplus. After Fab3000 has included all the Minimum Count requirements, it will go back and look for empty spaces to add surplus jobs. If you enter "0" for the job surplus, then Fab3000 will not bother looking for extra empty spaces to insert the job.

4. Verify Job Setup options: Job Spacing, and Center in Panel. Click Next to proceed.

5. Press Button "Next" to proceed

 **Note:** If it is not physically possible to add all of the Jobs you specified in "Minimum Count", Fab3000 will warn you that the area required exceeds the area of the panel (thus a merge with those quantities is not possible).

6. Choose the Options - such as score lines, etc.

**Add Scoring lines:**

Scoring lines are 1/2 depth routed paths placed between jobs, so that each board may be easily snapped apart from the panel.

**Add Outline layer:**

The outline layer is use more as a reference guide where a new layer is created with all individual board outlines plus the panel outline.

**Add Crop Marks to Silk Layer:**

Crop marks are placed onto the silkscreen layer at the four corners of the final panel. Some board manufacturers may require crop marks.

**Add Cut Lines to Silk Layer:**

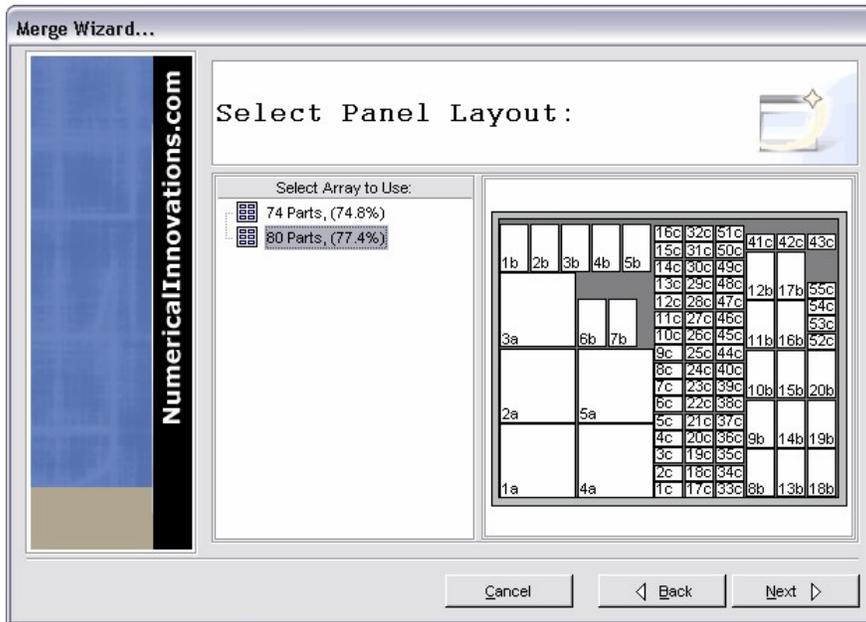
Cut lines are board outlines placed onto the silkscreen layer used to help guide you in cutting out the individual boards from the panel.

**Clearance for cut lines:**

Spacing of cut lines from true boarder edges.

7. Press Button "Next" to proceed

 **Note:** Fab3000 will instantly calculate the best possible fit for all required jobs. In some cases it may find more than one panel array, and it will allow you to select which panel array you prefer.



 Note: If you want to Add or Subtract the number of jobs (after looking at the proposed panel array, you can click button "Back" and reselect the number of jobs and options you wish to use.)

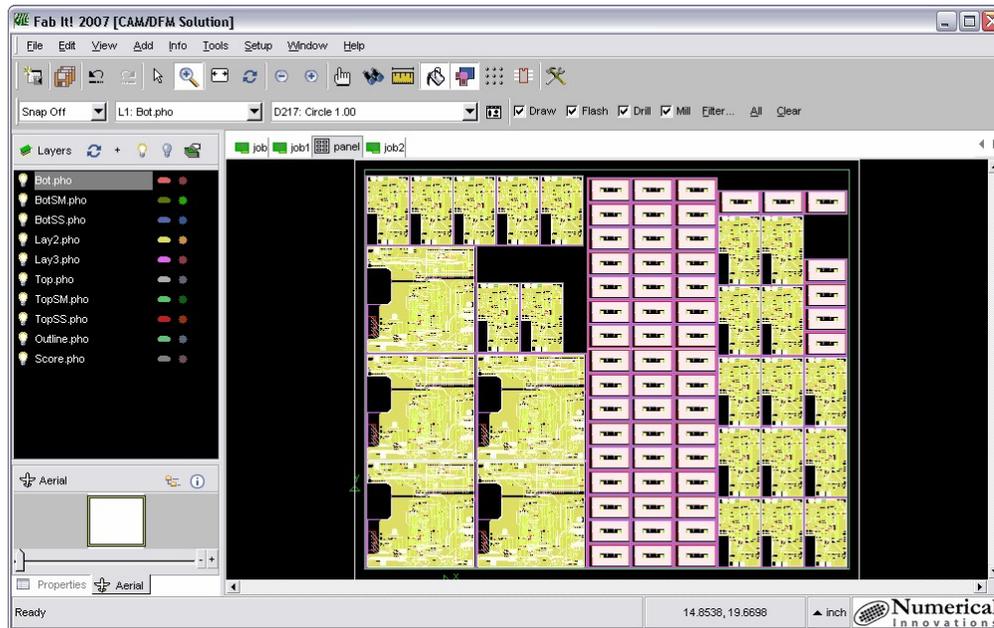
(7) Press Button "Next" to begin the merge

(8) Press Button "Finish", and you will now see your jobs merged!

 Note: In most cases the final stage of merging jobs is to perform [Synchronize Layers](#)<sup>[46]</sup>. Why? - Usually each job has a different layer name used to describe the same layer types (ie. "top02.pho", "topside.gbr", "art01.ger"). Before exporting, all layers of similar layer type must be synchronized into one layer for each type. (For more information see: [Synchronize Layers](#)<sup>[46]</sup>).

**Congratulations: You have successfully merged multiple jobs into one job, and potentially saved \$\$\$ on PCB Fabrication costs. Simply export the merged data to Gerber/Drill, ODB++, etc! ( For more information on exporting, goto [Gerber Export](#)<sup>[102]</sup>, [Drill Export](#)<sup>[103]</sup>, [ODB++ Export](#)<sup>[107]</sup> )**

(Screen capture of Merged Jobs - inside Fab3000)



### 4.3.2 Merge Jobs (Manually using Inserts)

The purpose of this section is to introduce you to the unique and powerful merging features of Fab3000. This Guide assumes that you have already created your jobs and imported data (ie. Gerber, DXF, etc) into each Job (see [Merge Jobs](#)<sup>[40]</sup> for more information).

**Note:** Insert Jobs is very useful when optimizing space for unusual boards (such as L shaped).

Once you have created different Jobs for each pcb design, it is now time to merge these Jobs using the power of hierarchy with Inserts.

*Note: This example assumes that you have created at least 2 or more Jobs (see [Merge Jobs](#)<sup>[40]</sup> for more information).*

#### Creating a new "Parent" Job:

1. Goto Menu: File | New | Job...
2. Select a name for the new Job. (Make it a name that is descriptive.)

*Once the new job is created, you will see a blank job editor screen.*

*Note: This new job can be considered the "parent" job, because it will contain job inserts (i.e. children).*

#### Adding Inserts Manually:

1. Goto Menu: Job | Insert Job...
2. Select the Job to insert

*A preview window will display the selected Job or Panel you want to insert.*

3. You may also assign the Job Orientation.

*Choose from: R0 (rotate 0), R90 (rotate 90), R180 (rotate 180), MX (mirror horizontal), MY (mirror vertical)*

4. You may also assign the Job Anchor Point. This controls the insert origin for the insert. Choosing the

correct Anchor Point can help when inserting jobs.

*Choose from: LowerLeft, Center, and Origin (actual 0,0 location of inserted job)*

5. Click 'OK' to Add the Inserts into the editor.

*Note: Repeat Steps (1-5) to insert the other Jobs.*

*Note: Once you have Added all required inserts, you may also edit their orientation, location, and even make copies.*

 **Note:** Since Fab3000 can handle an unlimited hierarchy, you can

---

## Synchronizing Layers:

Once you have inserted jobs into the "parent" job, it is now time to synchronize (merge) the layers. (For more information see: [Synchronize Layers](#)<sup>[46]</sup>).

---

## Export Parent Job to One Set of Gerber & Drills:

Once you have layed out all inserts into an optimal location, you may now export you merged CAM data.

### Export Gerber:

1. Goto Menu: File | Export | Gerber...
2. Verify Gerber settings, and select 'Next'
3. Verify Gerber Layer Options, and select 'Next'

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

*Note: You may also choose to output individual layers with Mirror or Negative polarity attributes.*

4. Choose the directory that you want to have the Gerber files written to.

*Note: If the directory does not exist, Fab3000 will create it for you.*

### Export Drill:

1. Goto Menu: File | Export | Drill...
2. Verify Drill settings, and select 'Next'
3. Verify Drill Layer Options, and select 'Next'

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

4. Choose the directory that you want to have the Drill file written to.

*Note: If the directory does not exist, Fab3000 will create it for you.*

**Congratulations: You have created merged gerber data from multiple printed circuit board designs into a single set of CAM files!**

### 4.3.3 Synchronize Layers

Merge all Job Layers into a common set of layers.

 **Note:** When merging different jobs together, chances are that each of those jobs used a different name to describe the same layer types (ie. "top02.pho", "topside.gbr", "art01.ger"). Synchronize

Layers combines those layers with same types into one layer, that can then be easily exported.

## Command Sequence

1. Select menu: Setup | Synchronize Layers...

2. Define all Layer Types for the child Job, and click button 'Next'.

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

3. Repeat Step#2 for each child job.

 **Note:** Once you have defined all layer types for child jobs, Synchronize Layers will begin to merge the layers, into one common set of layers.

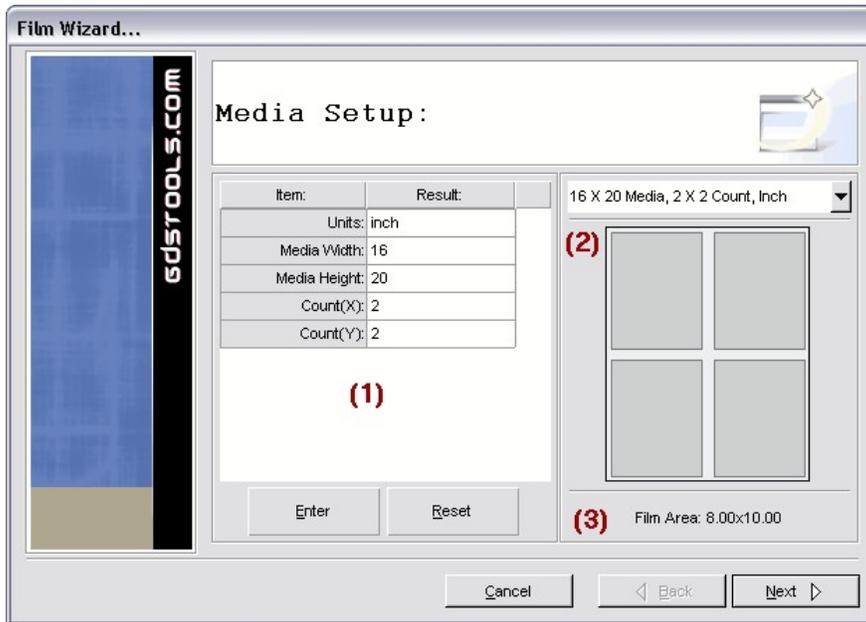
### 4.3.4 Using Film Wizard

Film Wizard will automatically setup gerber layers for optimal film output. Excellent for anyone who needs Gerber or Postscript photoplots from their artwork.

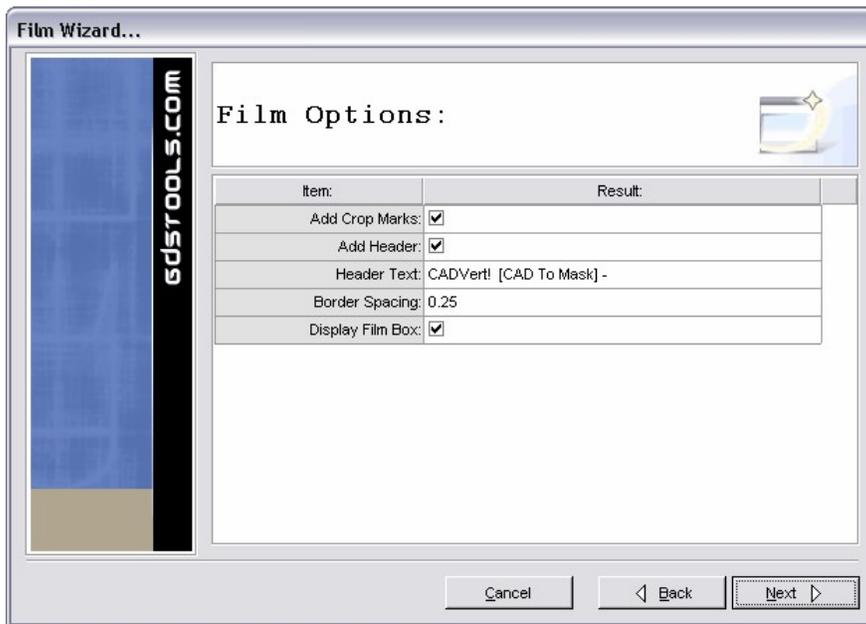
This unique Plug-In is guaranteed to **SAVE YOU UP TO 50% OFF YOUR PHOTOPLOTTING EXPENSES!** Film Wizard will also eliminate any operator errors (due to someone else wrongly mirroring or changing polarity), and get you faster turn-around times. Within a few photoplot jobs Film Wizard will have already paid for itself!

- This Guide assumes that you have already loaded Gerber files into the workspace, and they are visible in the editor . The gerber files used in this Learn Guide are found in the folder examples/274X.
- To begin **Film Wizard** , goto menu: Tools | Film Wizard...
- Verify **Media** size **(1)**. Most photoplotter's group your individual films onto a larger media. This media is sent to the photoplotter and all images are plotted onto the media. Once the media has been developed, it is cut up into individual film sizes when you receive.
- You may refer to the preview area **(2)** to verify that you have chosen an appropriate Media size and film count.
- Select **Next** to proceed.

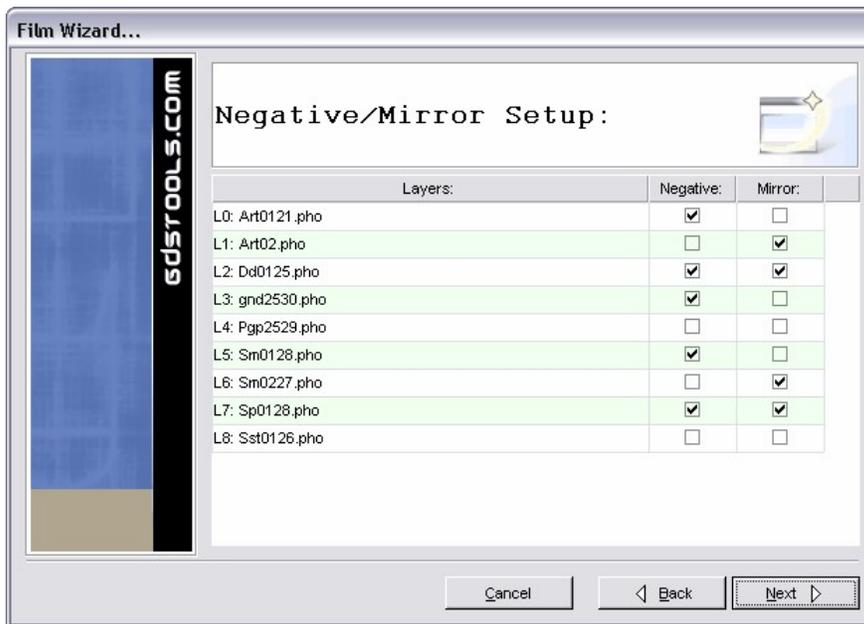
Our example in this guide has 8 gerber layers, with an area of 5"X7" (120mm X 175mm). If you are not using the example, then you must choose a media size that will be large enough to hold at least 2-4 individual films. In this example a media size of 16" X 20" will work perfect, and each final Film Area is 8"X10" **(3)**. (You may contact your photoplotting service to determine what are acceptable media & film sizes to use in their photoplotting process).



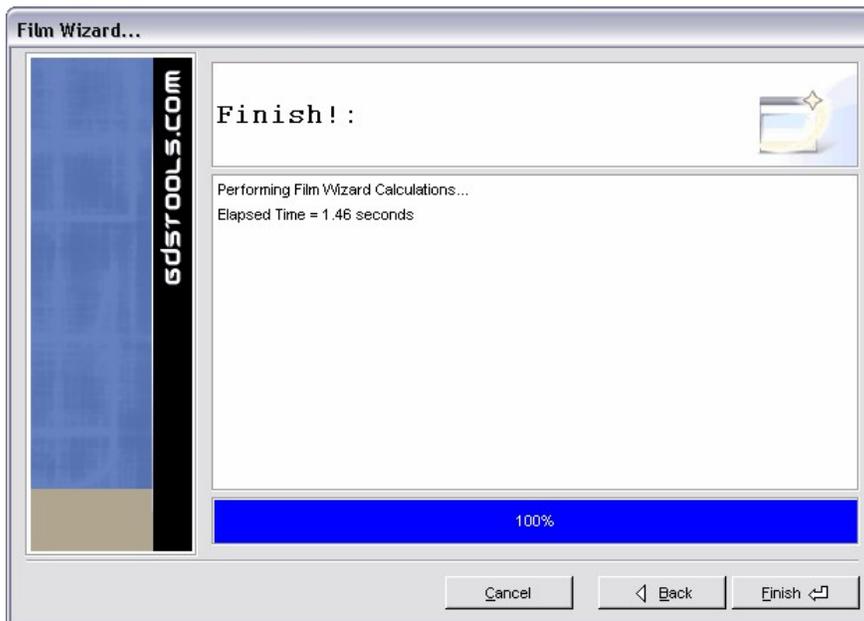
- Review the Film Options, and Select **Next** to proceed.



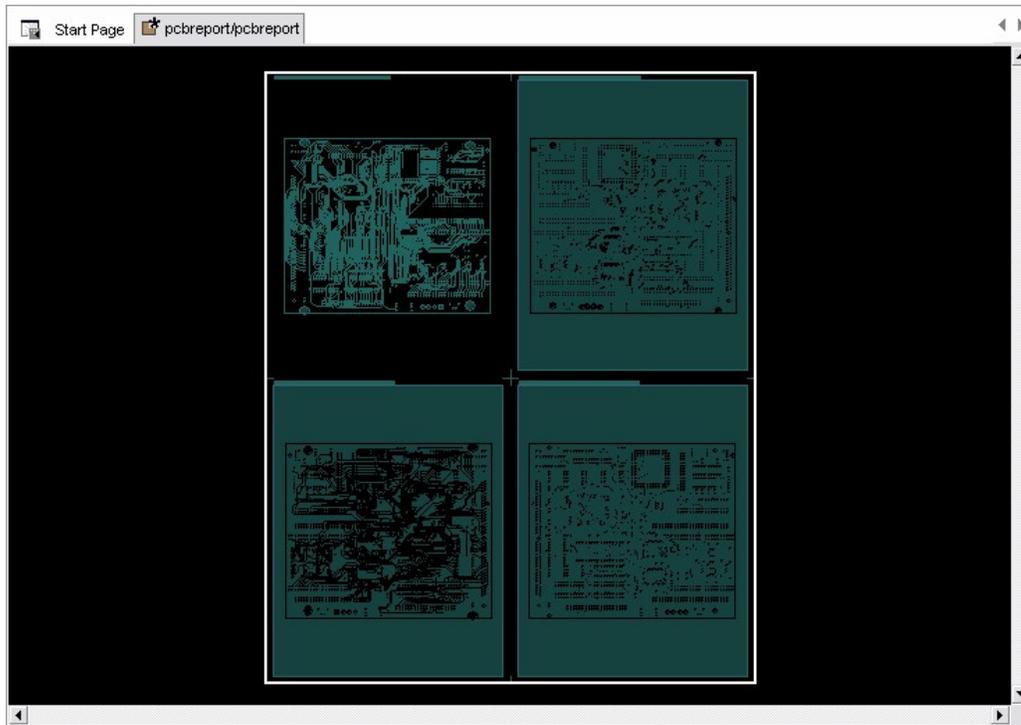
- Assign Polarity & Mirror attributes for each individual layer. In most cases the bottom layers will be mirrored, and plane layers will be negative. You may assign whatever attributes are required for your purposes.
- Select **Next** to proceed. Film Wizard will now begin to run.



- Select **Finish** to complete Film Wizard, and view the new film layers created.



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 export these "Film Wizard" layers to either Gerber or Postscript, and send them to your local photoplotter. You do not need to send your original gerber files.

**You have just saved 50% on your photoplotter service expenses!**

## 4.4 DRC / DFM

### 4.4.1 Using DFM Checker

Find all DRC/DFM violations. This Guide assumes that you have already imported gerber file(s) and drill file(s) into the workspace (see [AutoLoad](#)<sup>[79]</sup> for quick loading).

 **Note:** DFM Checker requires that you extract a netlist before proceeding. If you haven't run netlist extraction, DFM Checker will perform it for you.

 **Note:** DFM Checker requires that all layer types are defined. To define layer types, goto menu: Setup | Layers Table...

 **Note:** DFM Checker requires that you have at least 1 drill file (2 or more for blind/buried vias).

 Online Movie: [Performing simple DRC using FAB 3000](#)

### Command Sequence

1. Select menu: Tools | DRC / DFM | DFM Checker...

2. Verify 'General Settings', and press button Next.

3. Verify the DRC/DFM Checks, and press button Next.

 **Note:** Turn off any DRC/DFM Check, simply uncheck it.

4. Fab3000 will now perform an extensive analysis of your pcb design.

 **Note:** Depending on the complexity of the design and the checks performed, Fab3000 processing time can take anywhere from 30 seconds to 5 minutes.

 **Note:** 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 sperate editor window, and then proceed. To open Jobs, goto menu: Windows | Workspace Browser...

5. Press button 'Finish' when done.

6. If any errors have been detected, Fab3000 will display the [DRC/DFM Violation Browser...](#)<sup>[18]</sup>

 **Note:** The DRC/DFM Violation Browser is an easy-to-use navigation tool used to locate and describe each detected error.

## 4.4.2 Viewing DRC/DFM Violations

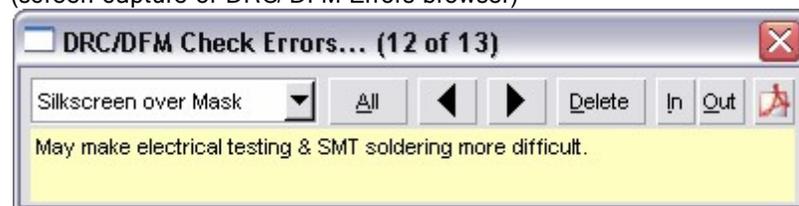
Finding All DRC/DFM errors is a snap. This Guide assumes that you have already run DRC/DFM [DFM Checker](#).<sup>[17]</sup>

### Command Sequence

1. Select menu: Tools | DRC / DFM | View DRC / DFM Violations...

 **Note:** View DRC/DFM violations requires that violations were detected by DFM Checker.

(screen capture of DRC/DFM Errors browser)



Contents of Error Browser Dialog:

Error Category: Combo box which displays all error categories.

All: Zoom to Extents of all errors in a selected category.

Left Arrow: Zoom to Previous Error

Right Arrow: Zoom to Next Error

Delete: Remove active Error marker from database.

In: Zoom In towards active Error marker

Out: Zoom away from active Error marker.

PDF Report: Create a detailed, user-friendly PDF report to share with other colleagues.

Error Description: Text box describing the error.

 **Note:** You may add your own custom statements into the Error Description text box.

2. Click the Next button ("Right Arrow"), to view the first error.
3. Click the In or Out buttons to zoom towards or away from the active error.
4. Change the Error Category, to modify the errors being displayed.

 **Note:** To create a detailed, user-friendly PDF file containing visual references & descriptions for each error, click on the PDF report button.

### 4.4.3 Create a PDF Report for DRC/DFM Violations

Easily organize and display all DRC/DFM violations into a PDF file, that can be shared with colleagues. This Guide assumes that you have already run DRC/DFM [DFM Checker](#).<sup>[177]</sup>

#### Command Sequence

1. Select menu: Info | PDF Report | DRC / DFM Violations...

 **Note:** View DRC/DFM violations requires that violations were detected by DFM Checker.

2. Select the file name to use for your pdf report, and click button 'Next'.
3. When the PDF report has been completed, click the button "View PDF file" to launch Adobe Acrobat (r), and display the report file.

## 4.5 Nets

### 4.5.1 Extracting a Netlist from Gerber and Drill data

One of the primary reasons you want to generate a netlist from your Gerber data is so that you can then do a comparison against a CAD-generated IPC-D-356 netlist. This Guide assumes that you have already imported gerber file(s) and drill file(s) into the workspace (see [AutoLoad](#)<sup>[79]</sup> for quick loading).

 **Note:** Netlist Extraction requires that all layer types are defined. To define layer types, goto menu: Setup | Layers Table...

 **Note:** Netlist Extraction requires that you have at least 1 drill file (2 or more for blind/buried vias).

 **Note:** If your designs contain Blind and/or Buried Vias, please goto Learn Guide: [Extracting with Blind and Buried Vias](#).<sup>[53]</sup>

 **Note:** One of the primary reasons you want to extract a netlist from your Gerber data is so that you can then do a comparison against a CAD-generated IPC-D-356 netlist.

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

#### Command Sequence

1. Select menu: Tools | Nets | Netlist Extraction...
2. Verify Netlist Extraction Options, and press button Next.
3. Press button 'Finish' when done.

 Note: The total number of extracted CAM Nets will be displayed on the Finish page.

 Note: To Compare the extracted Nets against your own CAD generated netlist, goto Learn Guide: [Comparing a CAM Netlist against an External Netlist](#).<sup>[54]</sup>

## 4.5.2 Extracting with Blind and Buried Vias

Extracting a netlist from Gerber and Drill data with Blind & Buried Vias. This Guide assumes that you have already imported gerber file(s) and drill file(s) into the workspace (see [AutoLoad](#)<sup>[79]</sup> for quick loading).

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

 Note: Netlist Extraction requires that all layer types are defined. To define layer types, goto menu: Setup | Layers Table...

 Note: Netlist Extraction requires that you have at least 1 drill file (2 or more for blind/buried vias).

 Note: Thru-Hole Drill layers use the layer type "Primary" while Blind or Buried Drills need to be assigned as layer type "Secondary"

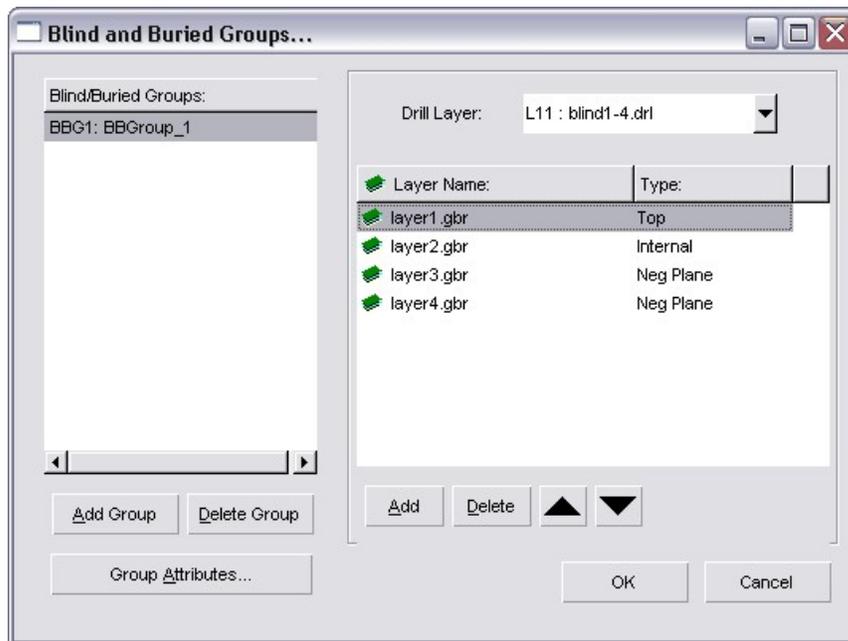
 Note: One of the primary reasons you want to extract a netlist from your Gerber data is so that you can then do a comparison against a CAD-generated IPC-D-356 netlist.

## Command Sequence

### Setup Blind and Buried Drill Relationships

1. Select menu: Setup | Blind and Buried Via Setup...

 Note: All Secondary Drill layers (ie. Not Thru-Hole) must have the layers associated with it defined, using this dialog.



2. Assign Secondary Drill Layer (in combo box)
3. Click Button "Add" and select layer, to be associate with the current secondary drill.
4. Repeat Step#3 until all associated layer for the current secondary drill are defined.
5. If you have more than one secondary drill, click Button "Add Group", and repeat steps#2 - #4 for each additional secondary drill.
6. Press button OK when done.

Proceed with Netlist Extraction

1. Select menu: Tools | Nets | Netlist Extraction...
2. Verify Netlist Extraction Options, and press button Next.
3. Press button 'Finish' when done.

 **Note:** The total number of extracted CAM Nets will be displayed on the Finish page.

 **Note:** To Compare the extracted Nets against your own CAD generated netlist, goto Learn Guide: [Comparing a CAM Netlist against an External Netlist](#).<sup>[54]</sup>

### 4.5.3 Comparing a CAM Netlist against an External Netlist

Netlist comparison is the most efficient and reliable method of ensuring design integrity has been maintained during the manufacturing stages. This Guide assumes that you have already extracted a netlist, and you now want to compare it against an external netlist.

---

 **Note:** The External Netlist file (from your CAD system) must be in standard format: IPC-D-356 or IPC-D-356/A

 **Note:** One of the primary reasons you want to extract a netlist from your Gerber data is so that you can then do a comparison against a CAD-generated IPC-D-356 netlist.

## Command Sequence

### Import External Netlist

1. Select menu: File | Import | Netlist...
2. Select your CAD-generated IPC-D-356 netlist, and press OK.
3. Verify Netlist Format, and press Next.
4. Press Finish.

 **Note:** The external netlist has now been imported into Fab3000

### Running Netlist Comparison

1. Select menu: Tools | Nets | Compare Nets...
2. Fab3000 will now perform an extensive netlist analysis.
3. Press button 'Finish' when done.
4. If any errors have been detected, Fab3000 will display the [View Compare Net Errors](#)<sup>[55]</sup>

 **Note:** The Compare Net Error Browser is an easy-to-use navigation tool used to locate and describe each detected error.

## 4.5.4 Viewing Netlist Comparison Errors

Finding All Compare Nets errors is a snap. This Guide assumes that you have already run [Compare Nets](#)<sup>[18]</sup>.

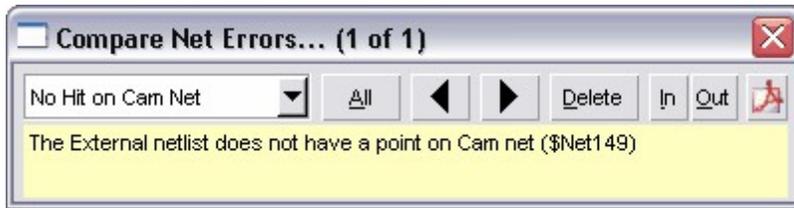
---

## Command Sequence

1. Select menu: Tools | Nets | View Compare Nets Errors...

 **Note:** View Compare Net errors requires that violations were detected by Compare Nets.

(screen capture of Compare Net Errors browser)



#### Contents of Error Browser Dialog:

Error Category: Combo box which displays all error categories.

All: Zoom to Extents of all errors in a selected category.

Left Arrow: Zoom to Previous Error

Right Arrow: Zoom to Next Error

Delete: Remove active Error marker from database.

In: Zoom In towards active Error marker

Out: Zoom away from active Error marker.

PDF Report: Create a detailed, user-friendly PDF report to share with other colleagues.

Error Description: Text box describing the error.

 **Note:** You may add your own custom statements into the Error Description text box.

2. Click the Next button ("Right Arrow"), to view the first error.
3. Click the In or Out buttons to zoom towards or away from the active error.
4. Change the Error Category, to modify the errors being displayed.

 **Note:** To create a detailed, user-friendly PDF file containing visual references & descriptions for each error, click on the PDF report button.

### 4.5.5 Create a PDF report for Net Comparison Errors

Easily organize and display all Compare Net errors into a PDF file, that can be shared with colleagues. This Guide assumes that you have already run [Compare Nets](#) <sup>[18]</sup>.

#### Command Sequence

1. Select menu: Info | PDF Report | Compare Net Errors...

 **Note:** View Compare Net Errors requires that violations were detected by Compare Nets.

2. Select the file name to use for your pdf report, and click button 'Next'.
3. When the PDF report has been completed, click the button "View PDF file" to launch Adobe Acrobat (r), and display the report file.

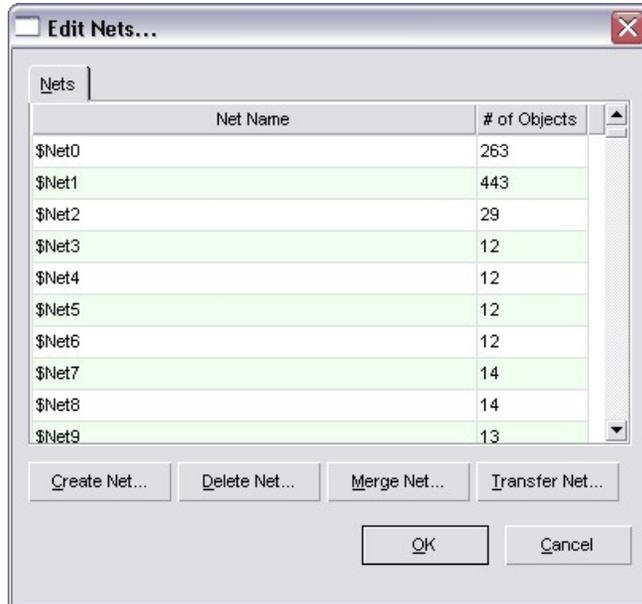
### 4.5.6 Add/Remove Objects from a Net

Editing a netlist.

 **Note:** To edit Nets requires that you extract a netlist before proceeding.

## Command Sequence

1. Select menu: Tools | Nets | Edit Nets...



### Features:

- Create Net - Add a new net to the job.
- Delete Net - Remove Net from job.
- Merge Net - Combine Nets
- Transfer Net - Copy contents from one net to another.

## 4.5.7 Apply External Net Names to CAM Net

When a netlist is extracted from Gerber & Drill data, there is no way for Fab3000 to know the net names used in your CAD design. Using Apply Net allows you to import your CAD-generated netlist, and apply the names to Fab3000. This Guide assumes that you have already extracted a netlist, and imported an external CAD-generated netlist.

### Command Sequence

1. Select menu: Tools | Nets | Apply Nets

 **Note:** Fab3000 will instantly update the existing CAM net names with the external net names.

## 4.6 Apertures / Custom Apertures

### 4.6.1 Creating Custom Apertures

Optimize your gerber data with custom apertures.

-  Online Movie: [Custom Aperture Fun - Mad Max using FAB 3000](#)
-  Online Movie: [Creating Custom Pads using Custom Apertures - FAB3000](#)
-  Online Movie: [Using Convert to Custom Aperture with FAB 3000](#)

1. Select menu: File | New | Custom Aperture...

2. Enter Custom Aperture Name, and click button "Finish".

 Note: You are now in the Custom Aperture Editor.

3. Create your custom aperture.

 Note: You are now in the Custom Aperture Editor. Here is where you perform all drawing & editing steps necessary to create the custom aperture.

 Note: You can use the Composite Number (located in the toolbar) to control dark and clear polarity. If an object is drawn with an "Even" Composite Number, it is opaque. If an object is drawn with an "Odd" Composite Number, it is Clear. Composite Number ranges from 0 to 1000. Objects are drawn in the order of the composite number starting from 0 to 1000. Thus objects with a lower composite number are drawn before objects with a higher composite number.

4. When you are finished click toolbar button: "Save & Update Custom Aperture"

 Note: This will update the custom aperture you created, and allow you to use it in the CAM editor.

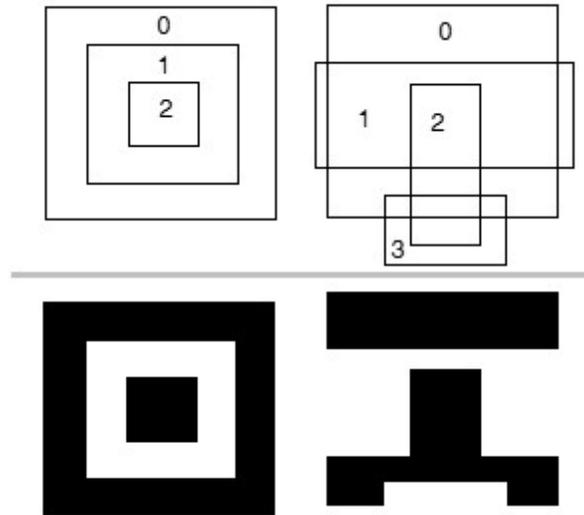
## 4.7 Composites

### 4.7.1 Composites 101

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. Essentially 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 (ie. Paint), while "odd" composite levels are drawn clear (ie. Scratch).

---

[How Composite Level Numbers define Interiors](#)



 Online Movie: [Add Polygons using Composite Logic - FAB 3000](#)

To change the composite level of any object:

### Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>[17]</sup>

2. Select menu: Edit | Properties. The Properties dialog box will appear.

3. Located Item "Composite Level", and change the value from the pull-down list.

4. Click Ok to exit the properties dialog.

 Note: If changes were made that are incorrect, you may [Undo](#) <sup>[122]</sup>.

 Note: To "Automatically" change the composite level of a group of objects, based on their interior location relative to other objects, try using [Polygon De-Embedding](#) <sup>[59]</sup>

## 4.7.2 Using DeEmbedding to Automatically Create Composites

Solving the polygon-inside-of-polygon problem.

### Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>[17]</sup>

2. Select menu: Edit | De-Embed...
3. Object Composite Levels will be automatically defined.

 **Note:** Objects must reside on the same layer.

4. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

## 4.8 Assembly / Centroid File

### 4.8.1 How to generate a Centroid File

Automated equipment is used to place the surface mount components on your pcbs. To rapidly program these machines, a Component Centroid file is required. Some CAD packages will automatically generate this file and some will not. Fab3000 (Professional Version) can easily create this centroid file from your existing gerber files.

 **Note:** 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 using function: [Generate Paste Layer\(s\)...](#)<sup>[173]</sup>

 **Note:** All pads on the Paste Layer must be "Flashed". If there are any pads on the Paste Layer that are not "Flashed", use function: [Draw to Flash](#)<sup>[183]</sup>

 **Note:** To determine if all pads are "Flashes" on the Paste Layer, goto [Layers Table](#)<sup>[67]</sup> 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 may also try using function: [Query](#)<sup>[161]</sup>

 **Online Movie:** [Generate XY Centroid file from Gerber, using FAB 3000](#)

---

## Add Components

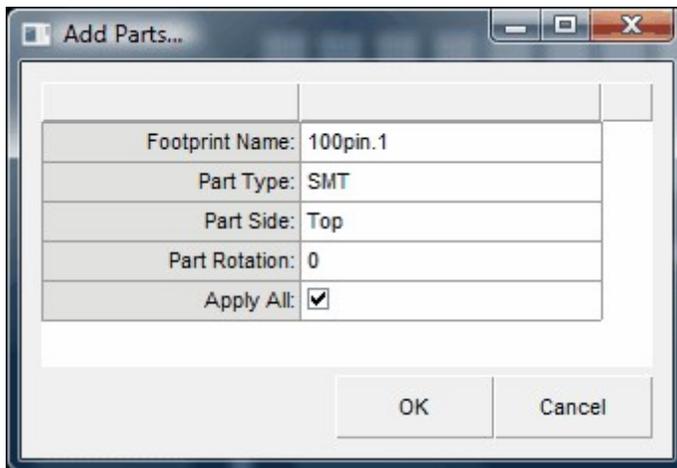
1. Turn-off all layers except for Top Paste & Top Silkscreen.
2. Locate a footprint, and select the entire component.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

 **Note:** Add Components requires flashed footprints and will ignore anything else, so it's fine to select a window around the entire component (any selected Silkscreen objects would be ignored).
3. Goto menu: Tools | Assembly | Add Components...
4. Select Pin One

 **Note:** Usually Pin One is the Top-Leftmost pin for Top side (assuming no rotation), and Top-Rightmost for Bottom side (assuming no rotation).
5. Assign Component Description, and press OK when done.



 Note: 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.

 Note: If Apply All is checked (recommended), Fab3000 will go out and seek all similar footprints.

 Note: 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.

 Note: You can also add components individually by using function: [Insert Single Component](#)<sup>176</sup>

Now its time for the Bottom Side (if required)

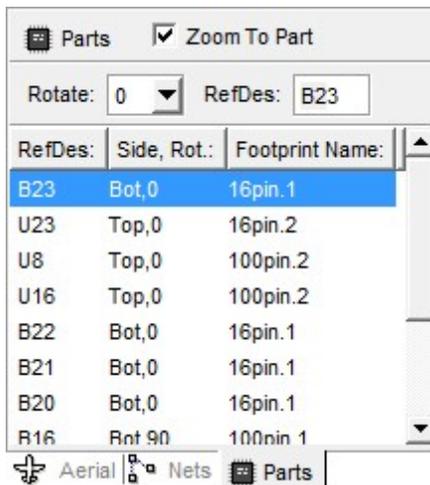
7. Turn off all layers except Bottom Paste & Bottom Silkscreen, and repeat steps 2-6.

---

## Assign Reference Designators to Components

1. Turn-off all layers except for Top Component & Top Silkscreen.

2. Go to the Components Tab Browser, and select the a Component (from the list).



 Note: When a Component is selected from the list, it will be highlighted. If you want to also zoom in towards the selected Component, make sure to check the item "Zoom To Component"

 Note: You can use the arrow keys (up/down) to navigate through the Components list.

3. Click into the edit control "RefDes:", and type-in the correct reference designator. Press Enter Key to update.

 Note: Having the silkscreen visible is 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.

---

## Export Centroid File

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

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

1. Go to menu: File | Export | Component Centroid File...

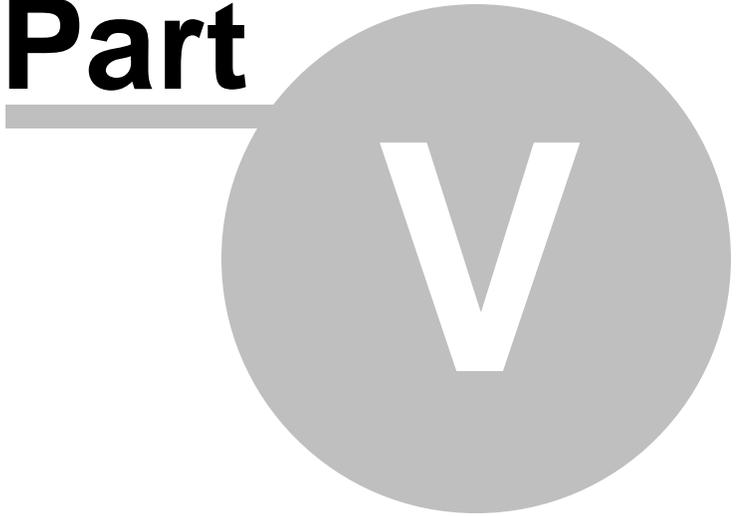
2. Choose from available options and press button 'Next'

3. Select File Name & location.

Congratulations - You have exported a centroid file!



**Part**



## 5 Working with FAB 3000

Fab3000 provides a Multi-Purpose CAM Editing, Translation, and Viewing system - Specifically targeted for the designers of PCB's, MEMs, Microwave and RF circuits, chemically milled parts and optical components; and to the photo mask manufacturers. Accurate viewing of: Gerber, DXF, NC Drill, HPGL and other widely used formats.

▶ [Workbench](#)<sup>[65]</sup>

▶ [Workspace](#)<sup>[69]</sup>

### 5.1 Workbench

The workbench implements the user cockpit for navigating the workspace. By using the workbench, you can navigate resources and you can view and edit the content and properties of these resources. The workbench contains an Editor, Editor Tabs, and 2 Views: Layer View, and Aerial View.

#### (1) [Editor](#)<sup>[66]</sup>

The Editor allows the user to open, browse, edit, and save various types of data. The majority of all user interaction is performed here. Modifications made in the editor follow an open-save-close life cycle model. Multiple instances of editors may exist within a Workbench. The editors can be stacked in the editor area and individual editors can be activated by clicking the tab for the editor. When an editor is active the Workbench menu bar and toolbar contain operations applicable to the editor.

#### (2) [Editor Tabs](#)<sup>[66]</sup>

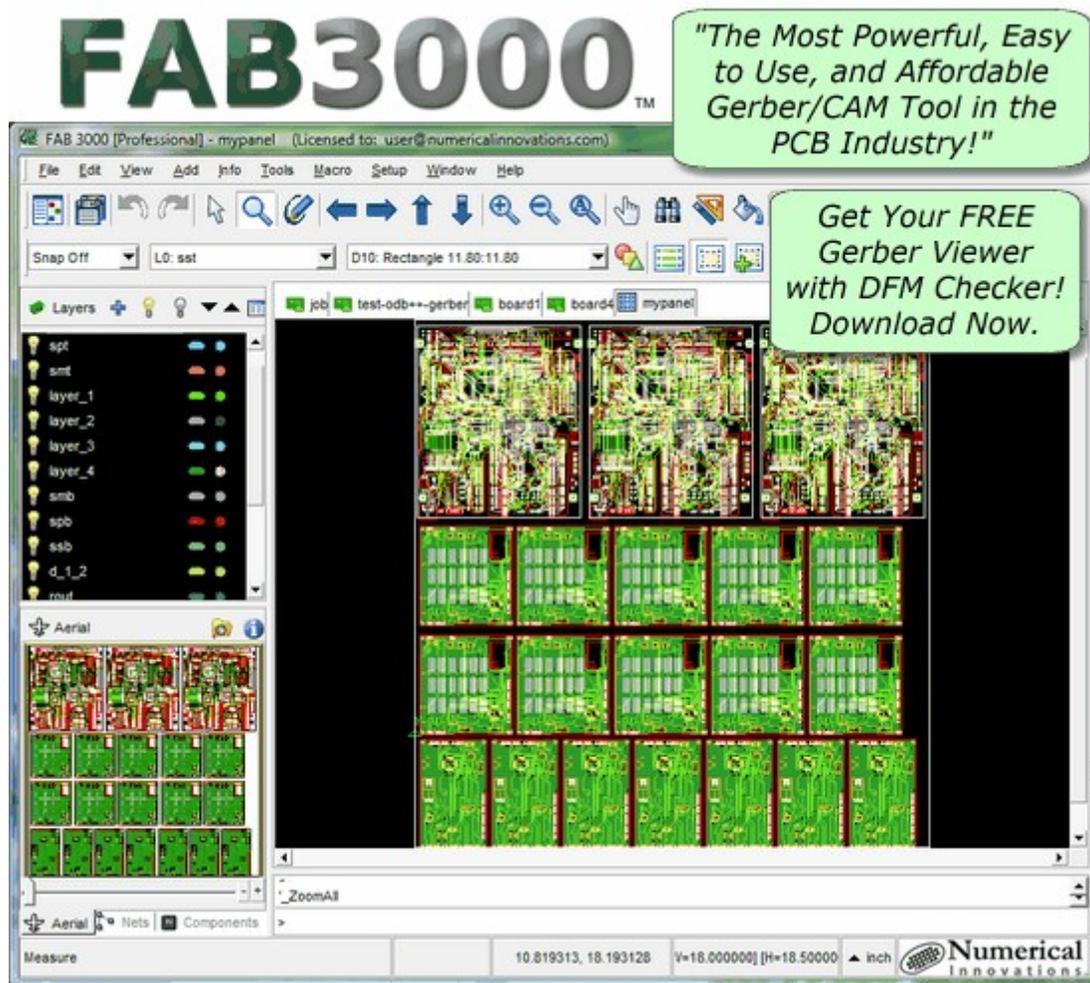
The editors can be stacked in the editor area and individual editors can be activated by clicking the tab for the editor. (See Editor)

#### (3) [Layer View](#)<sup>[67]</sup>

The Layer View is an interactive table that controls the following layer attributes in the editor: Visibility, Draw Color, Flash Color, and Name.

#### (4) [Aerial View](#)<sup>[68]</sup>

The Aerial View displays an overview of the current editor window. It provides an excellent reference in determining the overall location of the current editor view port within the design.



### 5.1.1 Editor

The Editor allows the user to open, browse, edit, and save various types of data. The majority of all user interaction is performed here. Modifications made in the editor follow an open-save-close life cycle model. Multiple instances of editors may exist within a Workbench. The editors can be stacked in the editor area and individual editors can be activated by clicking the tab for the editor. When an editor is active the Workbench menu bar and toolbar contain operations applicable to the editor.

Screen capture of stacked editor tabs:



Screen capture of editor Right-click Menu:

<u>C</u> ancel	Esc
<u>Z</u> oom	▶
<u>R</u> edraw	
<u>U</u> ndo	Ctrl+Z
<u>R</u> edo	Ctrl+Y
<u>S</u> election	▶
<u>D</u> elete	Del
<u>M</u> ove	
<u>C</u> opy	
<u>F</u> lip	▶
<u>R</u> otate	▶
<u>Q</u> uery	
<u>L</u> ist	
<u>P</u> roperties...	

**5.1.2 Layer View**

The Layer View is an interactive table that controls the following layer attributes in the editor: Visibility, Draw Color, Flash Color, and Name.



Screen Capture of Layer Widget:

**Visibility (1):**

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 (2):**

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

**Draw Color (3):**

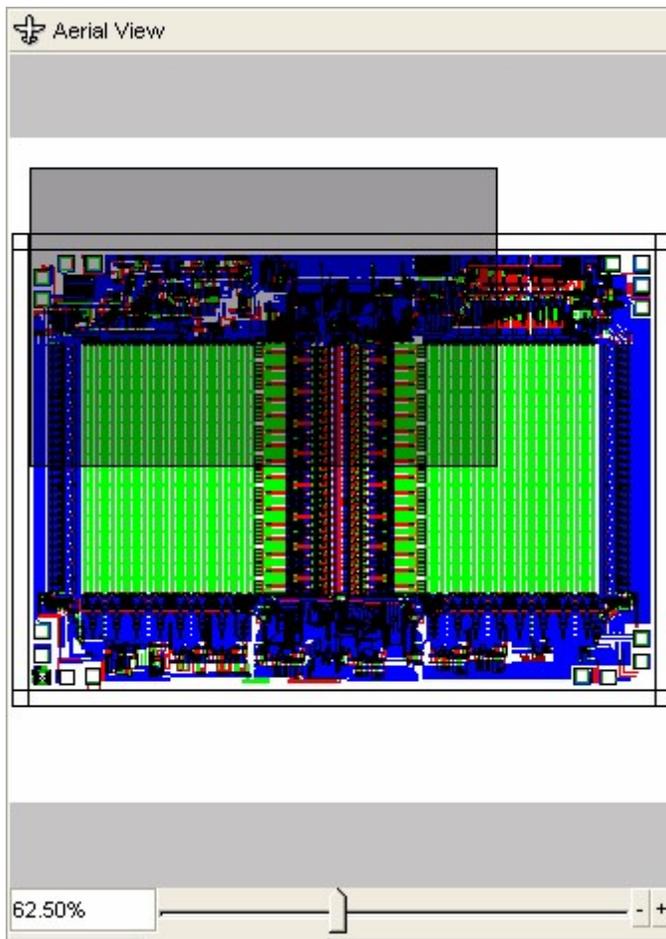
Objects on the associated layer will display that color for all draws (non-flashes)

**Flash Color (4):**

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

### 5.1.3 Aerial View

The Aerial View displays an overview of the current editor window. It provides an excellent reference in determining the overall location of the current editor view port within the design.



#### Zoom Percentage:

Ratio of editors current view port size against the entire design bounds.

#### Inter-Active Panning:

Simply drag the current view port window box with you mouse, and the displacement results will appear in the editor.

#### Notes/Links

You can also pan the aerial widget with the cursor. Place the cursor just inside the aerial view window, and drag the mouse.

## 5.2 Workspace

A workspace is the jobs, panels, custom apertures, folders, files that you create with the Workbench 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.

Note: Fab3000 has a dynamic workspace that can hold both graphical data and documentation.

## 5.2.1 Resource

Resources is a collective term for the designs, files, tables, etc. that exist in the Workspace. The most common resources in Fab3000 are: Jobs, Inserts, Panels, and Custom Apertures.

### 5.2.1.1 Job

The job by default is the base resource of Fab3000. 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.

### 5.2.1.2 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 (ie. rotate, mirror, move, etc) and can be nested into a hierarchy of jobs/panels.

### 5.2.1.3 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.

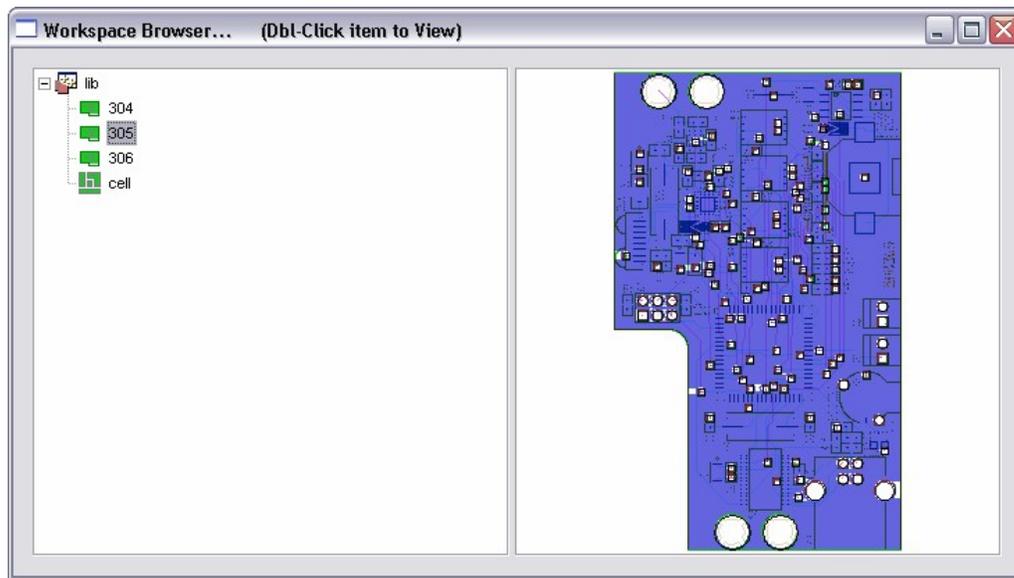
### 5.2.1.4 Custom Aperture

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

## 5.2.2 Workspace Browser...

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.

Screen Capture Of the Workspace Browser:



Tutorial:  
None

### 5.2.3 Library

The 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 (ie. text files,etc.)

**Part**



## 6 Job Editor Menu

Command Reference. All commands are organised in their menu format.

### 6.1 File Menu

#### 6.1.1 New

##### 6.1.1.1 Workspace

Creates a new workspace.

 Notes:

▶If an existing workspace is already active it will first be closed, and you will be given an option to save.

##### 6.1.1.2 Job

Creates a new job.

 Note: Essentially you can think of a Job as a single design with its own unique layers, nets, and other attributes.

### Command Sequence

1. Select menu: File | New | Job
2. Type in Job Name and press "OK". A new Job will be created.

 Note: The job name must be unique within the active workspace.

##### 6.1.1.3 Panel

Creates a new panel.

### Command Sequence

1. Select menu: File | New | Panel
2. Type in Panel Name and other parameters.

 Note: The panel name must be unique within the active workspace.

3. Press "OK". A new Panel will be created.

#### 6.1.1.4 Custom Aperture

Creates a new Custom Aperture.

 **Note:** A custom aperture is a special shapes that may be used as flashes in gerber.

#### Command Sequence

1. Select menu: File | New | Custom Aperture
2. Type-in the Custom Aperture Name and press "OK". The Custom Aperture editor will be displayed.

 **Note:** The custom aperture name must be unique within the active workspace.

 **Note:** The Custom Aperture editor is used to create special shapes that may be used as flashes in the current Fab3000 workspace. Every Fab3000 workspace has its own internal custom aperture library. When importing Gerber files that contain custom apertures, Fab3000 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.

 **Note:** 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).

#### 6.1.1.5 Footprint

Creates a new footprint and opens it up in the FAB 3000 editor.

#### Command Sequence

1. Select menu: File | New | Footprint
2. Type in Footprint name.

 **Note:** The footprint name must be unique within the active workspace.

3. Press "OK". An empty footprint will be created.

#### 6.1.1.6 Text File

Creates a new text file and opens it up in the FAB 3000 editor.

#### Command Sequence

1. Select menu: File | New | Text File...

2. Select file location and name.

 **Note:** The text file name must be unique within your computer.

3. Press "OK". An empty text file will be created.

#### 6.1.1.7 Spreadsheet File

Creates a new spreadsheet file and opens it up in the FAB 3000 editor. A spreadsheet file is outputted in comma separated format (csf) which can be imported by any commercial spreadsheet tool like Microsoft Excel, etc.

#### Command Sequence

1. Select menu: File | New | Spreadsheet File...

2. Select file location and name.

 **Note:** The spreadsheet file name must be unique within your computer.

3. Press "OK". An empty spreadsheet file will be created.

#### 6.1.1.8 Application File

Creates a new Application file (such as Lua) and opens it up in the FAB 3000 editor.

#### Command Sequence

1. Select menu: File | New | Application File...

2. Select file location and name.

 **Note:** The application file name must be unique within your computer.

3. Press "OK".

```
-- This tutorial demonstrates aperture creation
```

```
fab3000.dbSetUserUnits( fab3000.dbcInch )
```

```
-- Create Aperture using Dcode 10
```

```
nDcode = 10
```

```
myApt = fab3000.dbAperture()
```

```
myApt:setType( fab3000.dbApertureTypeCircle )
```

```
myApt:setWidth( 0.015 );
```

```
myApt:setHeight( 0.015 );
```

```
fab3000.dbSetAperture( nDcode, myApt );
```

```
-- Is DCode 10 Empty??
```

```
if fab3000.dbIsDcodeEmpty( nDcode )~=1 then
```

```
    fab3000.princ( "Dcode:" .. nDcode .. "Was Successfully created!" )
```

```

myAptCheck = fab3000.dbAperture()
fab3000.dbGetAperture( nDcode, myAptCheck )

--Output Message
fab3000.princ( "Dcode=" .. nDcode )
fab3000.princ( "Shape=" .. myAptCheck:getType() )
fab3000.princ( "Width=" .. myAptCheck:getWidth() )
fab3000.princ( "Height=" .. myAptCheck:getHeight() )
fab3000.princ( "OD=" .. myAptCheck:getOD() )
fab3000.princ( "ID=" .. myAptCheck:getID() )
else
    fab3000.princ( "Dcode:" .. nDcode .. "Was not created. Nothing to do." )
end

-- Add a few pads (note current dcode is used)
fab3000.uiEditAddLayer( "pads", 1 )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.07000, 0.06000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.09500, 0.04000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.13000, 0.04000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.16000, 0.06500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.08500, 0.11000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.13500, 0.11500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.12500, 0.00500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.16500, 0.02000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.19000, 0.04500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.20000, 0.08000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.19000, 0.12500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.15500, 0.16500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.10000, 0.17500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.06000, 0.16000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.02000, 0.12500 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.00500, 0.07000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.03000, 0.03000 ), nDcode )
fab3000.uiAddFlash( 1, 0, fab3000.dbPoint( 0.07500, 0.01000 ), nDcode )

--Zoom All (view all pads)
fab3000.uiViewZoomAll();

```

## 6.1.2 Open Workspace

Opens an existing FAB 3000 workspace file (\*.wrk).

Tutorial:

1. Select menu: File | Open Workspace...
2. Select the FAB 3000 workspace file from the Open Workspace dialog.
3. Select OK to open workspace.

 Notes:

▶ If an existing workspace is already active it will first be closed, and you will be given an option to save.

### 6.1.3 Close Workspace

Closes an active workspace.

Tutorial:

1. Go to menu: File | Close Workspace...

Note: If the workspace has been modified, you will be prompted to [Save](#) .

### 6.1.4 Save Workspace

Saves a workspace.

Tutorial:

1. Go to menu: File | Save Workspace...

 Note: If you have not saved the workspace before, you will be prompted to assign a workspace file name and location.

### 6.1.5 Save Workspace As

Saves an existing workspace as another file.

Tutorial:

1. Go to menu: File | Save Workspace As...
2. Select a workspace file name and location.

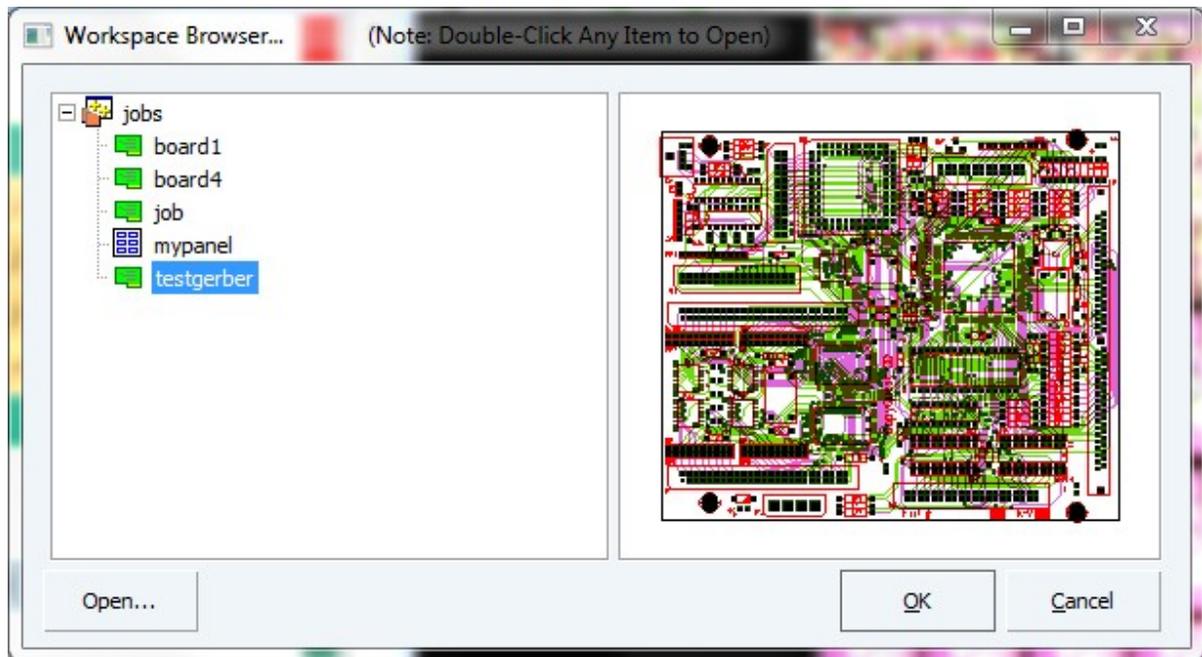
 Note: This can be used as an excellent backup tool allowing you save your current progress in a workspace before attempting to try something else. If you ever want to go back, you can simply open the save workspace file.

### 6.1.6 Workspace Browser

Quickly preview all loaded Job and Panels within the active workspace.

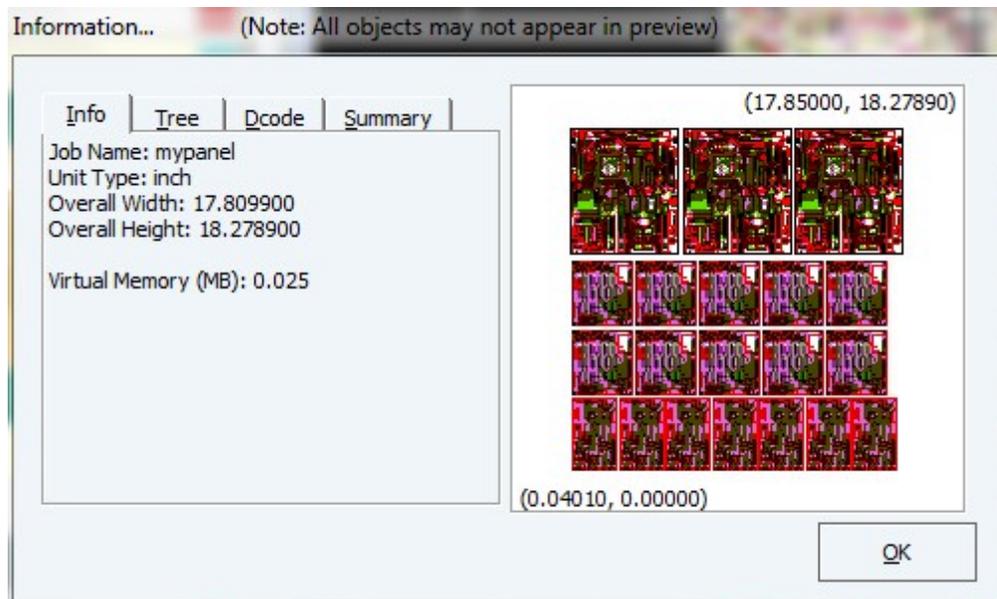
 Single Click on a Job or Panel to Preview it.

 Double click on a Job or Panel to Open it.



### 6.1.7 Information...

Display all job information.



### 6.1.8 Open Folder

Automatically Loads Gerber and Drill data from a folder.

#### Command Sequence

1. Select menu: File | Open Folder

2. Select the folder you want to import, and press "OK". The folder is instantly scanned and the all the contained Gerber, Drill, Netlist files are imported.

 Note: Fab3000 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 use [AutoLoad](#)<sup>[79]</sup>.

 Note: To view the Import log file, press key 'F2'

### 6.1.9 Open Compressed file (Zip,Tar,GZ,RAR)...

Automatically Loads Gerber and Drill data from a compressed file(.zip, .tar, .gz, .bz2, .rar).

#### Command Sequence

1. Select menu: File | Open Compressed file (Zip, Tar, etc)..

2. Select the compressed file you want to import, and press "OK". The file is instantly uncompressed and the import wizard dialog box appears.

 Note: Fab3000 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. Click "Next" to begin importing the files.

4. Once loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.

5. Click button "Finish" button to view the imported data.

 Note: To view the Import log file, press key 'F2'

### 6.1.10 Import

#### 6.1.10.1 AutoLoad...

AutoLoad Wizard will automatically import all Gerber and Drill files found within a selected folder. This makes importing your pcb design a snap, and eliminates the time needed to load each file separately.

#### To import Gerbers & Drills using AutoLoad

1. Put ALL the files pertaining to a job in the same directory.

2. Select menu: File | Import | AutoLoad. The AutoLoad Wizard dialog box appears.
3. Select whether your files are in English or Metric units.

 **Note:** Fab3000 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.

4. Select the Folder which contains the gerber files you wish to load. Once the directory is selected, you have two options:

Click the Finish button. (Fastest Way)

This will scan and load all the files in your directory, and you do not need to continue with any further steps in this procedure. This options is recommended if you know you are loading extended gerber data (Gerber format & apertures are already defined in the file: ie. 274X, Fire9000).

OR

Click the Next button. (Recommended)

At this point there will be a slight delay while Fab3000 scans the directory to determine whether each file is either a Gerber or Aperture list. Then the "File Description" page is displayed. This page gives you the opportunity to manually define the aperture list to use (for RS-274D only), Layer Types, and import checkbox for each gerber layer. After reviewing, click Next button to begin loading Gerbers.

5. Once loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during AutoLoad, they will be displayed here.
6. Click button "Finish" button to view the imported data.

---

## Common Problems

Most major issues with importing Gerber/Drill data fall under these categories:

### Incorrect Apertures

While importing 274D files, the most common cause for incorrect apertures is an incorrect or missing .RUL file. Fab3000 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.

 **Note:** 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, Fab3000 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, Fab3000 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 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 your 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"

#### Other Loading Errors...

Gerber files which contain numerous errors during import most likely are not be 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](mailto: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).

#### 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](#).<sup>[87]</sup>

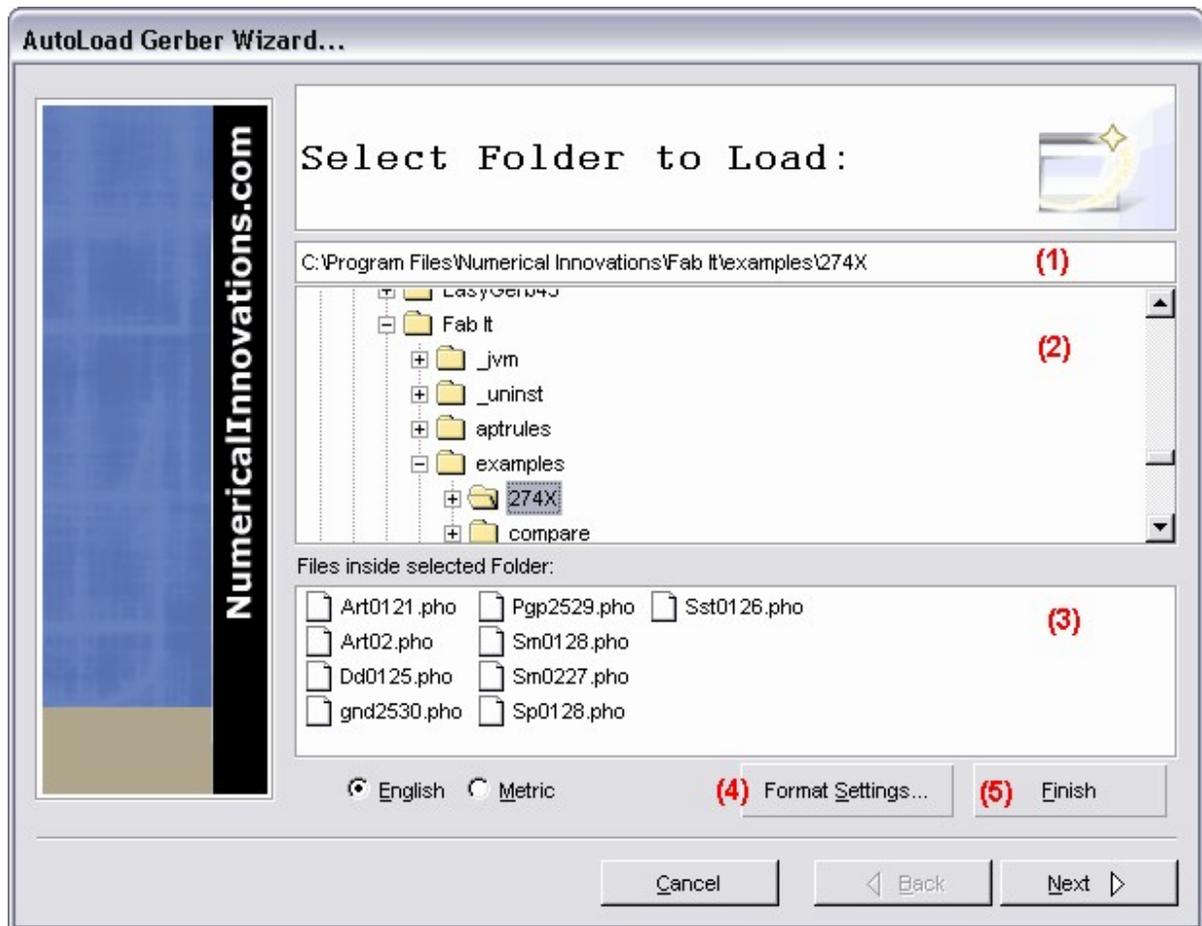
 Note: Fab3000 currently imports Excellon format only.

 Note: If more than one drill files is being imported simultaneously or an existing Tool table is already defined, Fab3000 will automatically map different drill tools with the same number but different diameters.

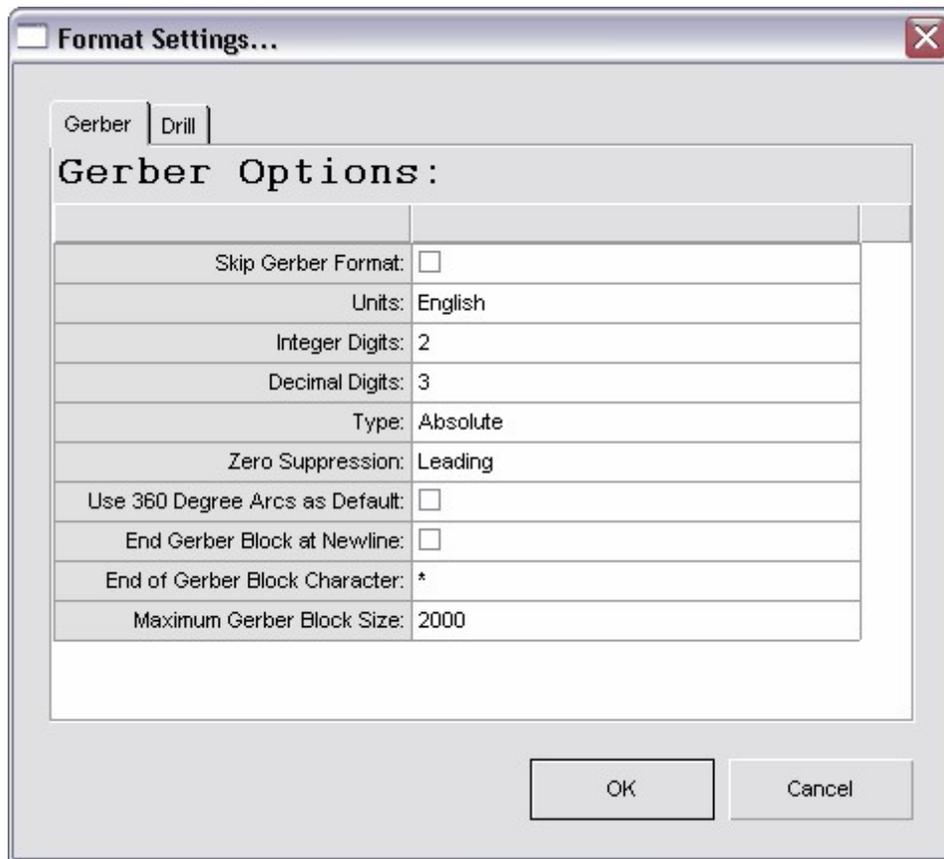
---

## Dialog Interface

AutoLoad main dialog. Used to select the folder for AutoLoad, and verify format settings.



The Format Settings dialog lets you custom setup the defaults that AutoLoad Wizard will use while import the files from the selected folder. You also have the option to 'Skip ... Format', if you do not wish to include a particular EDA format type during AutoLoad



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

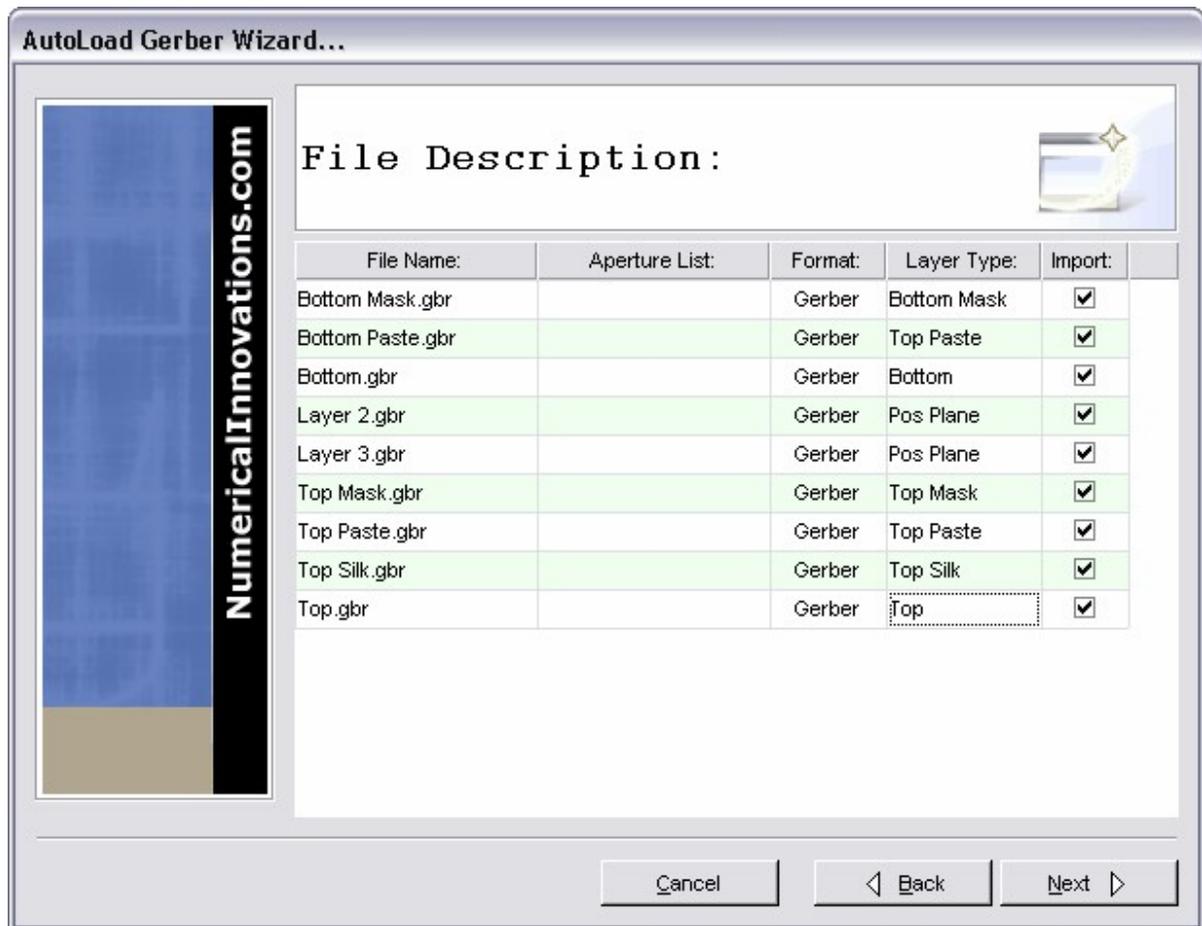
File Name - The name of the file located in the Autoload import directory.

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

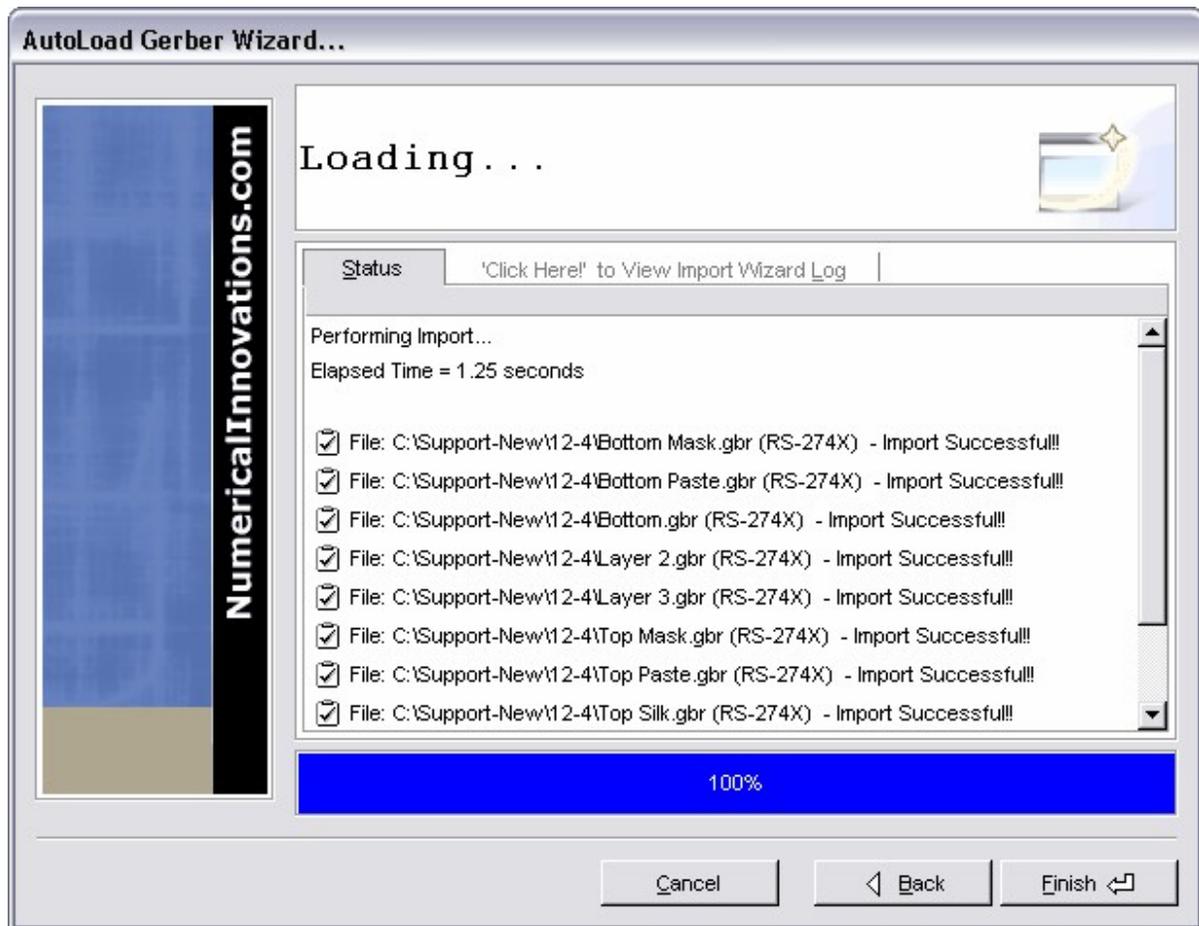
Format - Lets you know the format that AutoLoad has detected this file as.

Layer Type - Select the corresponding layer types for each file.

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.



This is the final page of the AutoLoad Wizard. It will first display a progress bar for each file being imported. Once AutoLoad Wizard has completed, the final results will be outputted in the status tab. Note: For detailed information about each file loaded click on the Import Log Wizard.



### 6.1.10.2 Gerber

Fab3000 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.

The easiest way to import Gerber files is to use the [AutoLoad Gerber Wizard](#).<sup>79</sup> It reads in all the information about your Gerber files automatically, and provides aperture file conversions, etc.

 **Note:** Fab3000 correctly displays Step & Repeat data. Most other CAM software will Crash, Ignore, or draw imaginary boxes. Fab3000 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.

## Command Sequence

1. Select menu: File | Import | Gerber.
2. Select the Gerber files you want to import, and press "OK". The Import Gerber wizard dialog box appears.

 **Note:** To select more than one gerber file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. To modify Gerber Import settings, click button "Format Options...". (For 274D Gerbers Only)

 **Note:** In most cases, Fab3000 will automatically detect the gerber format settings and aperture lists; do not adjust these settings if you are unsure about your gerber file format.

4. Select whether your files are in English or Metric units. (For 274D Gerbers Only)
5. Click the OK button. You are returned to the Import Gerber Wizard dialog box.
6. Select the corresponding layer types for each file. (Optional)
7. Click "Next" to begin importing gerbers.
8. Once Gerber loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
9. Click button "Finish" button to view the imported data.

 **Note:** If you already had existing data in the current job, any imported 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 Fab3000

---

## Import Gerber Format Settings

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.

### 6.1.10.3 Drill

Imports one or more NC drill (.DRL) files.



Note: Fab3000 currently imports Excellon format only.



Note: If more than one drill files is being imported simultaneously or an existing Tool table is already defined, Fab3000 will automatically map different drill tools with the same number but different diameters.

## Command Sequence

1. Select menu: File | Import | Drill
2. Select the Drill files you want to import, and press "OK". The Import Drill wizard dialog box appears.



Note: To select more than one drill file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. Select whether your files are in English or Metric units.
4. Select Integer Digits (digits to the left of the decimal point) you wish to use.
5. Select Decimal Digits (digits to the right of the decimal point) you wish to use.
6. Select the Type of format, Absolute or Incremental.
7. Select the Zero Suppression format required.
8. Click "Next" to begin importing drill files.



Note: Fab3000 will automatically scan your drill files to determine if there are any missing tool definitions.

If a Missing Tool is detected, you will receive a warning, stating that you must define all tools before proceeding. Click on button "Assign Tool Sizes...", to display the [NC Tools Table](#)<sup>[196]</sup>. 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 Drill loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.

10. Click button "Finish" button to view the imported data.

---

Import NC Drill Options:

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

#### 6.1.10.4 Rout / Mill

Imports one or more NC Rout (.rou) files.



Note: Fab3000 currently imports Excellon format only.



Note: If more than one rout file is being imported simultaneously or an existing Tool table is already defined, Fab3000 will automatically map different tools with the same number but different diameters.

#### Command Sequence

1. Select menu: File | Import | Rout
2. Select the Rout files you want to import, and press "OK". The Import Rout wizard dialog box appears.



Note: To select more than one rout file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. Select whether your files are in English or Metric units.
4. Select Integer Digits (digits to the left of the decimal point) you wish to use.
5. Select Decimal Digits (digits to the right of the decimal point) you wish to use.
6. Select the Type of format, Absolute or Incremental.
7. Select the Zero Suppression format required.
8. Click "Next" to begin importing rout files.



Note: Fab3000 will automatically scan your rout files to determine if there are any missing tool definitions.

If a Missing Tool is detected, you will receive a warning, stating that you must define all tools before proceeding. Click on button "Assign Tool Sizes...", to display the [NC Tools Table](#) (196). 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 Wizard and, import each rout file one at a time.

9. Once Rout loading is complete, a Status & Import log are available for your review. If Fab3000

encounters any errors or warnings during import, they will be displayed here.

10. Click button "Finish" button to view the imported data.

Import NC Rout Options:

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

### 6.1.10.5 ODB++

Loads a compressed ODB++ database file. The most common compressed file format for ODB++ data is ".tgz" (i.e tar ball).

 Note: A "Job" in Fab3000 is the same thing as a "Step" in ODB++

#### Command Sequence

1. Select menu: File | Import | ODB++
2. Select the compressed ODB++ database file you want to import, and press "OK". The Import ODB++ wizard dialog box appears.
3. Verify ODB++ Import Options: Layers, Steps, and Options
4. Click "Next" to begin importing the ODB++ database.
5. Once ODB++ loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

 Note: If there are multiple steps in the ODB++ database, Fab3000 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.

 Note: Unlike other CAM software products, Fab3000 loads the complete ODB++ database including all steps (while maintaining hierarchy). To view all the steps imported into Fab3000, see the [Workspace Brower...](#)<sup>[21]</sup>

 Note: To view the Import log file, press key 'F2'

#### 6.1.10.6 ODB++ (Directory)

Loads an ODB++ database from a selected directory.

 Note: A "Job" in Fab3000 is the same thing as a "Step" in ODB++

#### Command Sequence

1. Select menu: File | Import | ODB++
2. Select the folder where the ODB++ database is located, and press "OK". The Import ODB++ wizard dialog box appears.
3. Verify ODB++ Import Options: Layers, Steps, and Options
4. Click "Next" to begin importing the ODB++ database.
5. Once ODB++ loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

 Note: If there are multiple steps in the ODB++ database, Fab3000 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.

 Note: Unlike other CAM software products, Fab3000 loads the complete ODB++ database including all steps (while maintaining hierarchy). To view all the steps imported into Fab3000, see the [Workspace Browser...](#)<sup>[211]</sup>

 Note: To view the Import log file, press key 'F2'

#### 6.1.10.7 Netlist (IPC-D-356/A)

Loads an IPC Netlist (IPC-D-356 and IPC-D-356A) into Fab3000

 Note: IPC-D-356 and IPC-D-356A are generic netlist formats used as a standard for the Bare-Board Testing environment.

#### Command Sequence

1. Select menu: File | Import | IPC Netlist
2. Select the netlist file you want to import, and press "OK". The Import Netlist wizard dialog box appears.
3. Chose the Netlist Format.

4. Click "Next" to begin importing the netlist.
5. Once Netlist loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to exit.

 Note: The data is loaded as the External Netlist for the active Job.

#### 6.1.10.8 DXF / DWG

Load the AutoCAD DWG or DXF file into the workspace.

For more detailed instructions, please goto the Learn Guide: [Importing DXF](#) <sup>32</sup>

#### Command Sequence

1. Select menu: File | Import | DXF
2. Select the DXF file to import.

 Note: The DXF file will be scanned for Layers, Blocks, Styles, and more..

 Note: In most cases you will need to verify the DXF Units & Scale Factor.

3. Verify the DXF Import options.

Important: DXF Files are unit-less, so it is important that you verify the DXF Units & Scale Factor, of your DXF file before importing it.

4. Choose "DXF Design Type". This is an "Optional" feature which adjusts other DXF import settings to accommodate optimal conversion results in most cases.
5. Verify the options.
6. Verify DXF Layers.

 Note: This list allows you to control how each individual DXF layer is handled during import.

- Check "Import" to include the layer during import.
- Check "Fill" to convert closed boundaries to filled-polygons.
- Choose "Polyline" mode, to adjust how polylines with width are handled during import. The three choices are: Extend, Truncate, and Round.
- Check "Auto-Flash", to detect circles & rectangles, and convert them to gerber flashes.
- Enter "Comp" (offset compensation) for an entire layer.

7. Verify DXF Blocks.

 Note: This list allows you to control how each individual DXF block is handled during import.

- Check "Import" to include the block during import.

8. Map DXF Styles.



Note: This list allows you to map each DXF style to a compiled font file during import.

- Choose a Default Text Style for any unknown dxf style.
- Review and assign the "Text Line Width/Fill method"

9. Merge DXF Layers. (Optional)



Note: This list allows you to merge DXF layers during import.

- Check "Output Merged Layers Only" to only write merged layers, and incoher individual DXF layers.

10. Open / Save. (Optional)



Note: This (Optional) 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.

11. Click "Next" to begin importing DXF file.

12. Once DXF loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.

13. Click button "Finish" button to view the imported data.

#### Import DXF Units:

<b>DXF Units/Scale:</b>	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
<b>DXF Design Type:</b>	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:

<b>Use Polygon DeEmbedding:</b>	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.
<b>Respect ByBlock/ ByLayer:</b>	Block entities that are on designed layer '0', will be changed to the layer of the respective insert.
<b>Ignore Dimensions:</b>	If this setting is 'checked', dimensions will be omitted.
<b>Hatch Mode:</b>	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.
<b>Arc Resolution (Degrees):</b>	Accuracy used when mapping points to recreate arcs & circles.
<b>Default Line Width:</b>	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).
<b>Flash Circle/Rects Less Than:</b>	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:

<b>Import:</b>	If "checked" this layer will be imported. If "unchecked", this layer will be omitted.
<b>DXF Layer Name:</b>	The DXF layer name. This is merely for reference, and cannot be changed.
<b>Fill:</b>	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 <a href="#">Default Line Width</a> .
<b>Polyline:</b>	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.
<b>Auto-Flash:</b>	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)
<b>Comp:</b>	If non-zero, apply the offset-compensation to all resulting boundaries on this layer.

## Import DXF Blocks:

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

## Import DXF Styles:

<b>DXF Style:</b>	The DXF style name. This is merely for reference, and cannot be changed.
<b>Fab3000 Font File:</b>	Choose which Pre-Compiled Fab3000 Font File to use from the pull-down. All Fab3000 Font files are derived from TrueType fonts respectively. You may compile a Fab3000 Font file from any

	TrueType font ( <a href="#">see: Font Table for more information</a> <sup>[212]</sup> )
<b>Default Text Style:</b>	Assign the default Fab3000 Font File to use when an associated DXF font file cannot not be determined or located.
<b>Text Line Width/Fill Method:</b>	The setting controls the DXF Text Line Width & Filling options: Use Polygon Fill: All Text will be filled as polygons. Their widths are already determined in the Fab3000 font file itself. Use Same Line Width for All: Determine a standard line width to be used for all DXF text. Use Ration: The text line width will be determined by a ratio of the text height divided by your supplied value.

## Import DXF Merge:

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

## Import DXF Open/Save:

<b>Open DXF Settings:</b>	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.
<b>Save DXF Settings:</b>	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.

## 6.1.10.9 Image

Load the Image (TIFF, JPEG, BMP, GIF) file into the workspace.

## Command Sequence

1. Select menu: File | Import | Image
2. Select the Image files you want to import, and press "OK". The Import Image wizard dialog box

appears.

 **Note:** To select more than one Image file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. Select Pixel Size.

4. Select Pixel Units. Choose from Mil, Inch, Millimeter, Centimeter, and more.

 **Note:** The Logical Pixel Size = Pixel Size \* Pixel Units

5. Choose Image Background Color (Black or White).

6. Choose Background Intensity (1-100).

 **Note:** For best results, you may need to adjust this setting to determine which value works best for your purposes.

7. Click "Next" to begin importing the files.

8. Once Image loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.

9. Click button "Finish" button to view the imported data.

---

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

<b>Pixel Unit:</b>	Specify the proposed units for each pixels physical size. Choose from Micron, Inch, Millimeter, Centimeter, and more.
<b>Pixel Size:</b>	Logical pixel size defined by Pixel Units * Pixel Size
<b>Image Background Color:</b>	Select the image's transparent color, that will be ignored during importing. Choose either Black or White.
<b>Background Intensity (1-100):</b>	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.
<b>Merge Adjacent Pixels:</b>	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.

#### 6.1.10.10 HPGL/2

Imports one or more HPGL/2 files.

 **Note:** Dcodes are assigned automatically, based upon the HPGL/2 data.

## Command Sequence

1. Select menu: File | Import | HPGL/2
2. Select the HPGL/2 files you want to import, and press "OK". The Import HPGL wizard dialog box appears.

 **Note:** To select more than one Hpgl/2 file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. Select whether you want to scale the entire file. (optional)
4. Click "Next" to begin importing HPGL/2 files.
5. Once HPGL/2 loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

---

Import HPGL Options:

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

### 6.1.10.11 Postscript / PDF

Loads a Postscript or PDF file into the active workspace.

 **Note: 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/>

## Command Sequence

1. Select menu: File | Import | Postscript/PDF
2. Select the Postscript or PDF file you want to import, and press "OK". The Import Postscript/PDF wizard dialog box appears.
3. Verify the Page number to use for import (usually 1), and other import options.
4. Click "Next" to begin importing the file.
5. Once Image loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.

6. Click button "Finish" button to view the imported data.

Import Postscript/PDF Options:

<b>Page Number to Convert:</b>	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.
<b>Spline/Arc Deviation (mil):</b>	Approximation accuracy to use for Arcs and Spline objects.
<b>Ghostscript Additional Options:</b>	Assign specific Ghostscript options. Requires knowledge of the Ghostscript interpreter.
<b>Flash Round &amp; Rect Shapes:</b>	Convert round and rectangular objects to flashes. This results in a more optimized data.

#### 6.1.10.12 Large Bitmap

Loads large bitmap (any size) into the active workspace.

#### Command Sequence

1. Select menu: File | Import | Large Bitmap

2. Select the Image files you want to import, and press "OK". The Import Image wizard dialog box appears.

 **Note:** To select more than one Image file, you may use dynamic selection by holding "Shift" or "Ctrl" keys.

3. Select Pixel Size.

4. Select Pixel Units. Choose from Mil, Inch, Millimeter, Centimeter, and more.

 **Note:** The Logical Pixel Size = Pixel Scale \* Pixel Units

5. Choose Image Background Color (Black or White).

6. Choose Background Intensity (1-100).

 **Note:** For best results, you may need to adjust this setting to determine which value works best for your purposes.

7. Click "Next" to begin importing the files.

8. Once Image loading is complete, a Status & Import log are available for your review. If any errors or warnings are encountered during import, they will be displayed.

9. Click button "Finish" button to view the imported data.

---

Import Large Bitmap Options:

<b>Pixel Unit:</b>	Specify the proposed units for each pixels physical size. Choose from Micron, Inch, Millimeter, Centimeter, and more.
<b>Pixel Size:</b>	Logical pixel size defined by Pixel Units * Pixel Size
<b>Image Background Color:</b>	Select the image's transparent color, that will be ignored during importing. Choose either Black or White.
<b>Transparency Factor:</b>	Many images incorporate a type of grayscale 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 then want these photos converted back for comparison purposes.
<b>Merge Adjacent Pixels:</b>	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.

#### 6.1.10.13 STL (3D)

Loads STL (stereolithography) binary format into the active workspace.

#### Command Sequence

1. Select menu: File | Import | STL
2. Select the STL file you want to import, and press "OK". The Import STL wizard dialog box appears.
3. Verify the Units, Scale Factor, Output Type, and other import options.
4. Click "Next" to begin importing the file.
5. Once Image loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

---

Import Postscript/PDF Options:

<b>Units:</b>	Choose the units for the imported STL.
<b>Scale Factor:</b>	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.

#### 6.1.10.14 GDSII

Loads GDSII binary format into the active workspace.

#### 6.1.10.15 Part Centroid

Import Part Centroid File. Loads an Ascii file that contains centroid data.

 Note: A centroid file is commonly a comma separated (or tab separated) file.

### Command Sequence

1. Select menu: File | Import | Part Centroid
2. Select the Centroid file you want to import, and press "OK".
3. Choose the separation character (comma, tab, etc.), and press OK.

 Note: 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"

 Note: You will now see at the top of the column a check mark and the title REFDES. This lets you know that the RefDes 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"

 Note: 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"

 Note: 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"

 Note: 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"

 Note: 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"

 Note: 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"

 Note: 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.

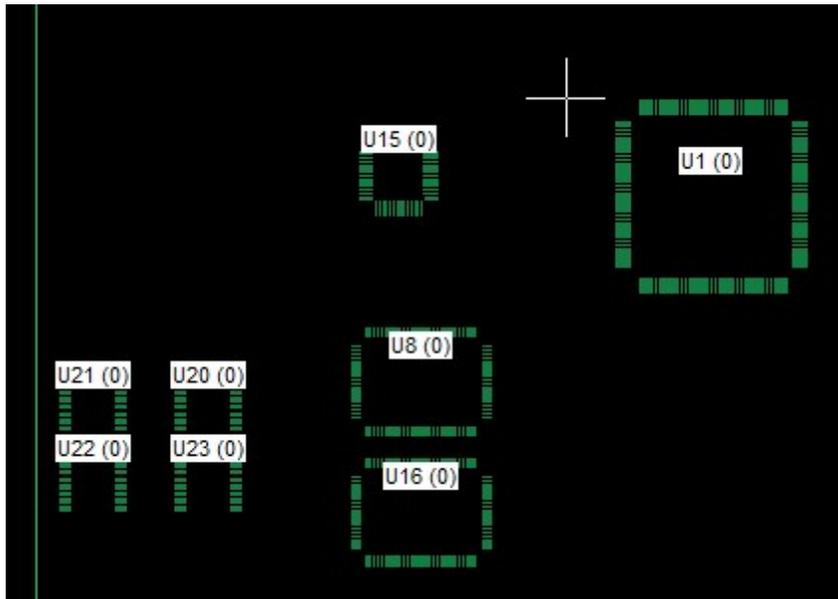
11. Verify the Import Units (located in the Lower/Left) used for the X,Y Coordinates.

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.

 Note: You will now see the external centroid data displayed in the FAB 3000 screen. To control the display of the external centroid files, goto menu: [View | External Centroids](#).<sup>145</sup>

(screen capture with external centroid display)



#### 6.1.10.16 BOM

Import a Bill of Materials File.

### 6.1.10.17 Aperture List

Imports an aperture list in various formats.

 Note: 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](#)<sup>[79]</sup>.

 Note: Fab3000 does not detect the aperture list format using this command, so you must also define which Aperture Wizard Rule to use during import.

 Note: Import Aperture List provides an excellent way to debug Aperture Wizard Rules that you have created with the [Aperture Rules Editor](#)<sup>[202]</sup>. To verify results of a newly created Aperture Wizard Rule: Create a new job, Import the aperture list, and review the [Aperture Table](#)<sup>[195]</sup>.

### Command Sequence

1. Select menu: File | Import | Aperture List
2. Select the Aperture files you want to import, and press "OK". The Import Aperture dialog box appears.
3. Select the Aperture Wizard Rule to use while importing the aperture list.
4. Click "OK" to begin importing the aperture file.
5. Once HPGL/2 loading is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during import, they will be displayed here.
6. Click button "Finish" button to view the imported data.

### 6.1.10.18 Custom Aperture Library

Loads a custom Aperture Library into the active workspace.

### 6.1.10.19 Footprints Library

Enter topic text here.

### 6.1.10.20 FAB 3000 Matrix

Loads a FAB 3000 Import Matrix file. For more information about the FAB 3000 Matrix file, [please click here](#)<sup>[218]</sup>.

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 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 - saving \$\$\$ on fabrication costs. In addition, FAB 3000 will automatically performs all necessary object transformations, dcode & drill tool transcoding.

**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.

## 6.1.11 Export

### 6.1.11.1 Gerber

This command is used to export your data in a Gerber format.

 **Note:** Gerber files exported from Fab3000 are fully compatible with the Gerber specification and support polygon fills, composite images, step and repeat codes, and more.

### Command Sequence

1. Select menu: File | Export | Gerber. The Export Gerber Wizard dialog box appears.
2. Verify the Export Gerber file options you want to use, and press "Next". The Layer Options appears.

 **Note:** To improve gerber file accuracy you should select decimal digits of 4 (or higher).

 **Note:** For description of each option, see below: [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 the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, 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 wish to mirror any of your layers upon export, select the appropriate check boxes in the Mirror column.

 **Note:** Polarity and Mirroring options apply only to 274X, Fire 9xxx, Barco & Gerber files. Fab3000 will display Negative Layers but not the Mirroring option on files re-imported using [AutoLoad Gerber](#)<sup>[79]</sup> or [Gerber Import](#)<sup>[85]</sup> will display as normal, but retain their option codes in the Layers table.

 **Note:** If your layers previously contain Mirror or Polarity options, they will automatically checked in the Layer Options.

 **Note:** Each selected layer will be exported as an individual gerber file.

7. Select the folder to export the gerber files into.

 Note: 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 (ie. gerbers, drill, netlist, notes, fab drawing, etc.)

 Note: If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).<sup>[205]</sup>

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 Fab3000 encounters any errors or warnings during export, they will be displayed here.

10. Click button "Finish" button to exit.

### Export Gerber Options:

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

#### 6.1.11.2 Drill

This command is used to export layers into a Drill file.

## Command Sequence

1. Select menu: File | Export | Drill. The Export Drill Wizard dialog box appears.
2. Verify the Export Drill file options you want to use, and press "Next". The Layer Options appears.

 **Note:** To improve drill file accuracy you should select decimal digits of 4 (or higher).

 **Note:** For description of each option, see below: [Export Drill 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 the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, right-click within the Export column.

 **Note:** Only Drill layers will have their Export column checked.

5. Select the folder to export the drill files into.

 **Note:** 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 (ie. gerbers, drill, netlist, notes, fab drawing, etc.)

 **Note:** If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).

8. Click "Ok" to begin exporting your drill file(s).
9. Once export drill is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.
10. Click button "Finish" button to exit.

---

## Export Drill Options

<b>Units:</b>	Determine exported NC Drill units of English or Metric.
<b>Integer Digits:</b>	Determine exported NC Drill integer digits between (0-6).
<b>Decimal Digits:</b>	Determine exported NC Drill integer digits between (0-6).
<b>Type:</b>	Determine exported NC Drill type of Absolute or Incremental.
<b>Zero Suppression:</b>	Determine exported NC Drill zero suppression of Leading, Trailing, or None.
<b>Use Step/Repeat:</b>	Use Step & Repeat codes if possible.

<b>NC Drill File Extension:</b>	Default extension to use for exported NC Drill file names. (*.drl) is the default.
---------------------------------	--

### 6.1.11.3 Rout / Mill

This command is used to export layers into a Rout file.

**New! Version 5 can automatically detect Arcs.**

#### Command Sequence

1. Select menu: File | Export | Rout. The Export Rout Wizard dialog box appears.
2. Verify the Export Rout file options you want to use, and press "Next". The Layer Options appears.

 **Note:** To improve rout file accuracy you should select decimal digits of 4 (or higher).

 **Note:** For description of each option, see below: [Export Rout 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 the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, right-click within the Export column.

 **Note:** Only Rout layers will have their Export column checked.

5. Select the folder to export the rout files into.

 **Note:** 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 (ie. gerbers, drill, netlist, notes, fab drawing, etc.)

 **Note:** If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).<sup>[205]</sup>

8. Click "Ok" to begin exporting your rout file(s).
9. Once export rout is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.
10. Click button "Finish" button to exit.

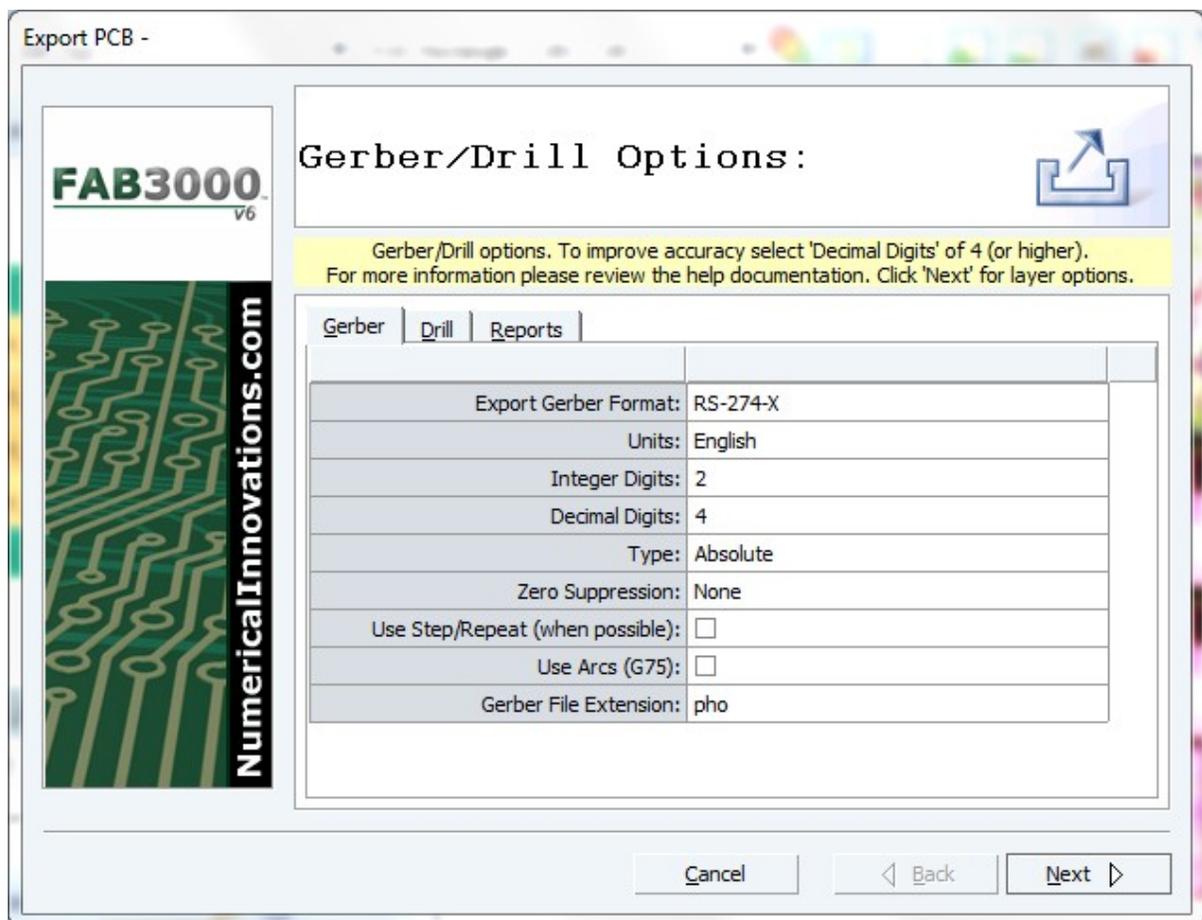
#### Export Rout Options

<b>Units:</b>	Determine exported NC units of English or Metric.
---------------	---

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

#### 6.1.11.4 PCB

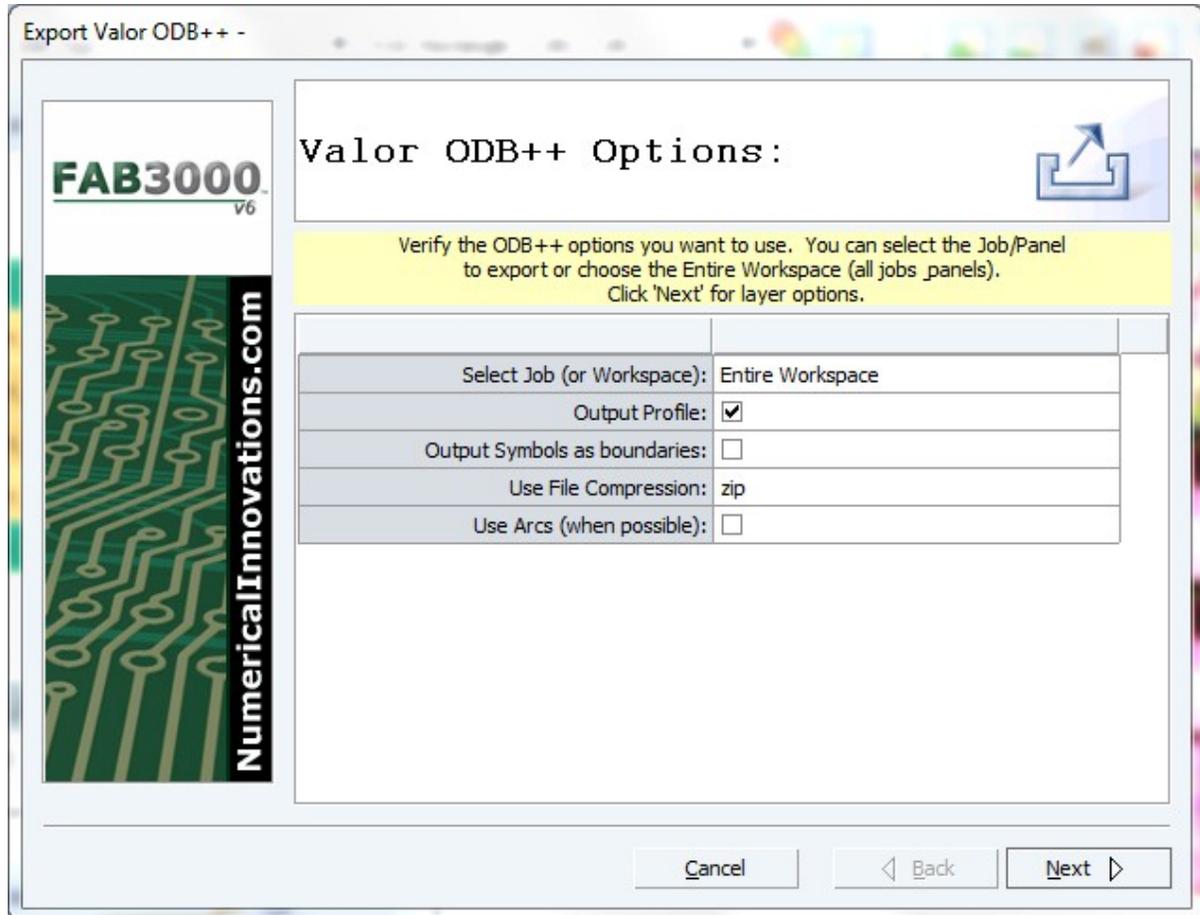
This command is used to export your data in both Gerber and Drill formats simultaneously.



### 6.1.11.5 ODB++

Writes an ODB++ database to an external file system.

 **Note:** The ODB++ Matrix requires layers to have their types (ie. Top, Bottom, Silkscreen, etc.) and sequence order (ie. layer stackup) defined before exporting. To define both Layer Types & Stackup goto [Layers Table...](#)<sup>[197]</sup>, before exporting ODB++ data



### 6.1.11.6 Netlist (IPC-D-356/A)

This command is used to export a netlist.

#### Command Sequence

1. Select menu: File | Export | IPC Netlist. The Export Netlist Wizard dialog box appears.
2. Verify the Export Netlist options you want to use, and press "Next".

 **Note:** For description of each option, see below: [Export Netlist Options](#)

3. Select the netlist file to create.

 Note: If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).

4. Click "Ok" to begin exporting your netlist file.
5. Once export netlist is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.
6. Click button "Finish" button to exit.

---

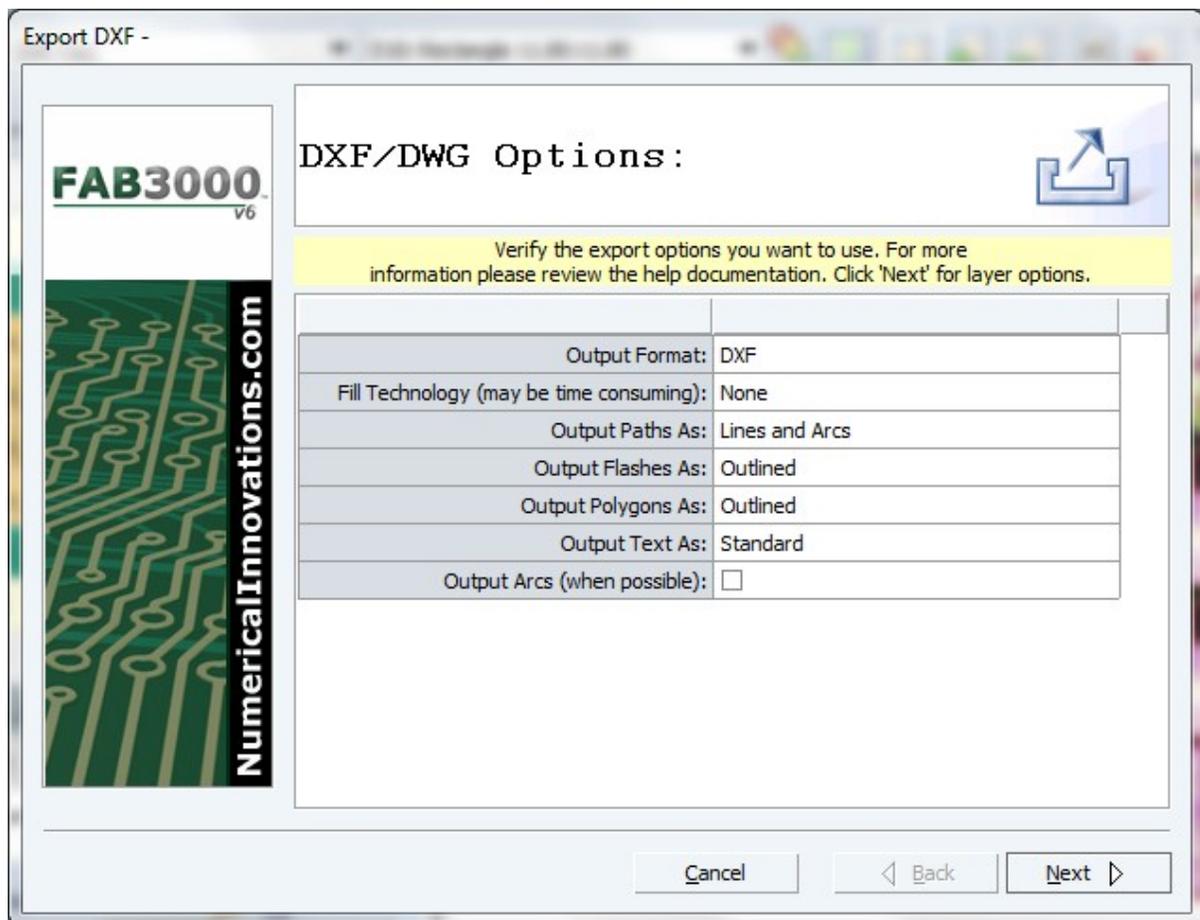
### Export Netlist Options

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

#### 6.1.11.7 DXF / DWG

This command is used to export data into a DXF or AutoCAD DWG file. Note: DWG is essentially the binary equivalent of DXF.

 Online Movie: [How to export a DXF file with outlined objects using FAB 3000.](#)



## Command Sequence

1. Select menu: File | Export | DXF. The Export DXF Wizard dialog box appears.
2. Verify the Export DXF file options you want to use, and press "Next". The Layer Options appears.

 **Note:** For description of each option, see below: [Export DXF Options](#)

 **Note:** 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 the DXF version: "AutoCAD 12 (older)"

3. If the layer names need to be changed, type-in the desired filename in the text boxes.
4. Using the check boxes in the Export column, select the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, right-click within the Export column.
5. Select the DXF file to create.

 **Note:** If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).<sup>2051</sup>

6. Click "Ok" to begin exporting the DXF file.
7. Once export DXF is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.
8. Click button "Finish" button to exit.

---

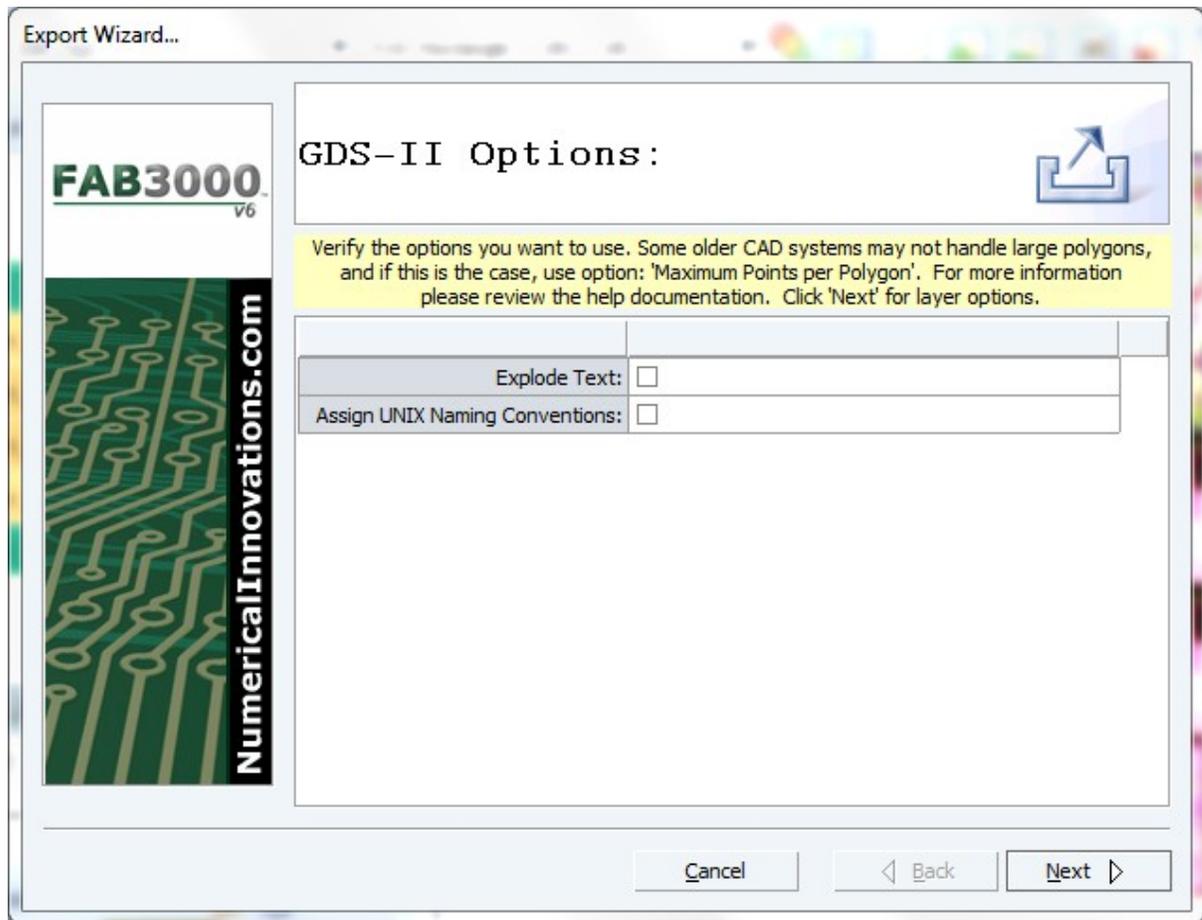
Export DXF Options:

<b>Text Output:</b>	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.
<b>Explode Array Instances:</b>	Convert Array instances to individual instances.
<b>Output Paths as Boundaries:</b>	Create polygon outlines for all paths.
<b>Fill Option:</b>	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 Using Solid Hatch (Only): Export No Geometries, Fill using Solid Hatch Using Multiple Solids: Export Geometries, Fill using solid triangles Using Multiple Solids (Only): Export No Geometries, Fill using solid triangles

#### 6.1.11.8 GDSII

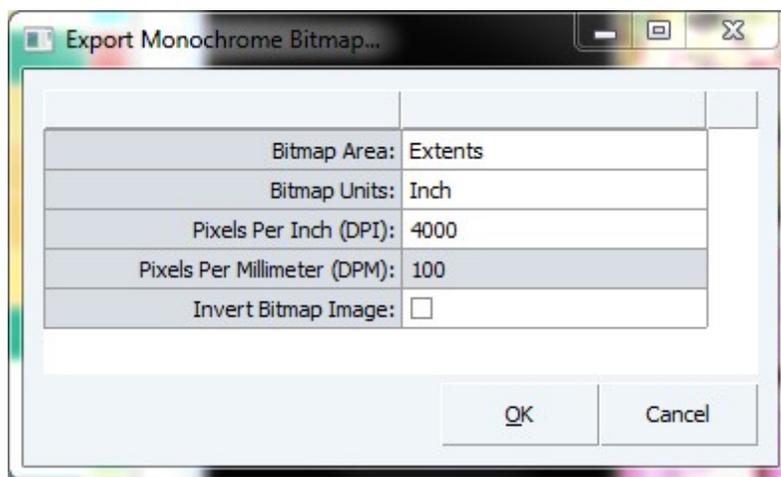
Writes a GDS-II stream file to an external file system.

---



#### 6.1.11.9 Large Bitmap

Writes large 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 Fab3000 requires the editor to render the images that will be exported.



#### 6.1.11.10 Centroid File

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

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

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

#### Command Sequence

1. Go to menu: File | Export | Component Centroid File...
2. Choose from available options and press button 'Next'
3. Select File Name & location.

Congratulations - You have exported a centroid file!

#### 6.1.11.11 BOM File

Export a Bill of Materials File.

#### 6.1.11.12 IGES

This command is used to export data into an IGES file.

#### Command Sequence

1. Select menu: File | Export | IGES. The Export IGES Wizard dialog box appears.
2. Verify the Export IGES file options you want to use, and press "Next". The Layer Options appears.

 **Note:** For description of each option, see below: [Export IGES Options](#)

 **Note:** The IGES format has undergone many version changes in the past 5-10 years, and some older systems may not be able to read newer IGES files. If this is the case, try selecting the IGES version (5.0, 5.1, 5.2, 5.3) compatible with your system.

3. If the layer names need to be changed, type-in the desired filename in the text boxes.
4. Using the check boxes in the Export column, select the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, right-click within the Export column.
5. Map layers to IGES levels.
6. Select the IGES file to create.

 **Note:** If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).

7. Click "Ok" to begin exporting the IGES file.
8. Once export IGES is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.
9. Click button "Finish" button to exit.

---

#### Export IGES Options

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

#### 6.1.11.13 STL

Writes a STL (stereolithography) binary to an external file system.

#### 6.1.11.14 Postscript

This command is used to export layers into PostScript files.

#### Command Sequence

1. Select menu: File | Export | PostScript. The Export PostScript Wizard dialog box appears.
2. Verify the Export PostScript options you want to use, and press "Next". The Layer Options appears.

 **Note:** For description of each option, see below: [Export PostScript 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 the layers you want to export, or deselect the layers you do not want exported. To Set All/Clr All, right-click within the Export column.
5. Select the folder to export the postscript files into.

 Note: 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 (ie. gerbers, drill, netlist, notes, fab drawing, etc.)

 Note: If you would like to always export your files to a specific folder, you may define the "Export Folder" in the [Fab3000 Settings Dialog](#).<sup>[205]</sup>

8. Click "Ok" to begin exporting your postscript file(s).

9. Once export postscript is complete, a Status & Import log are available for your review. If Fab3000 encounters any errors or warnings during export, they will be displayed here.

10. Click button "Finish" button to exit.

---

### Export Postscript Options

<b>Scale Factor:</b>	Determine scale to use for Postscript output. (In most cases for MEMs/EDA masks a factor of 10X is used.)
<b>Postscript Type:</b>	Select Postscript output type either Standard or Encapsulated. If you are unsure, contact your local graphic expert - in most cases we recommend "Encapsulated".
<b>Line Width (Points):</b>	Default line width to assign for 0-width objects such as lines, arcs, etc.
<b>Text Fill:</b>	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.

#### 6.1.11.15 Custom Aperture Library

Writes a custom Aperture Library to an external file system.

---

#### 6.1.11.16 Footprints Library

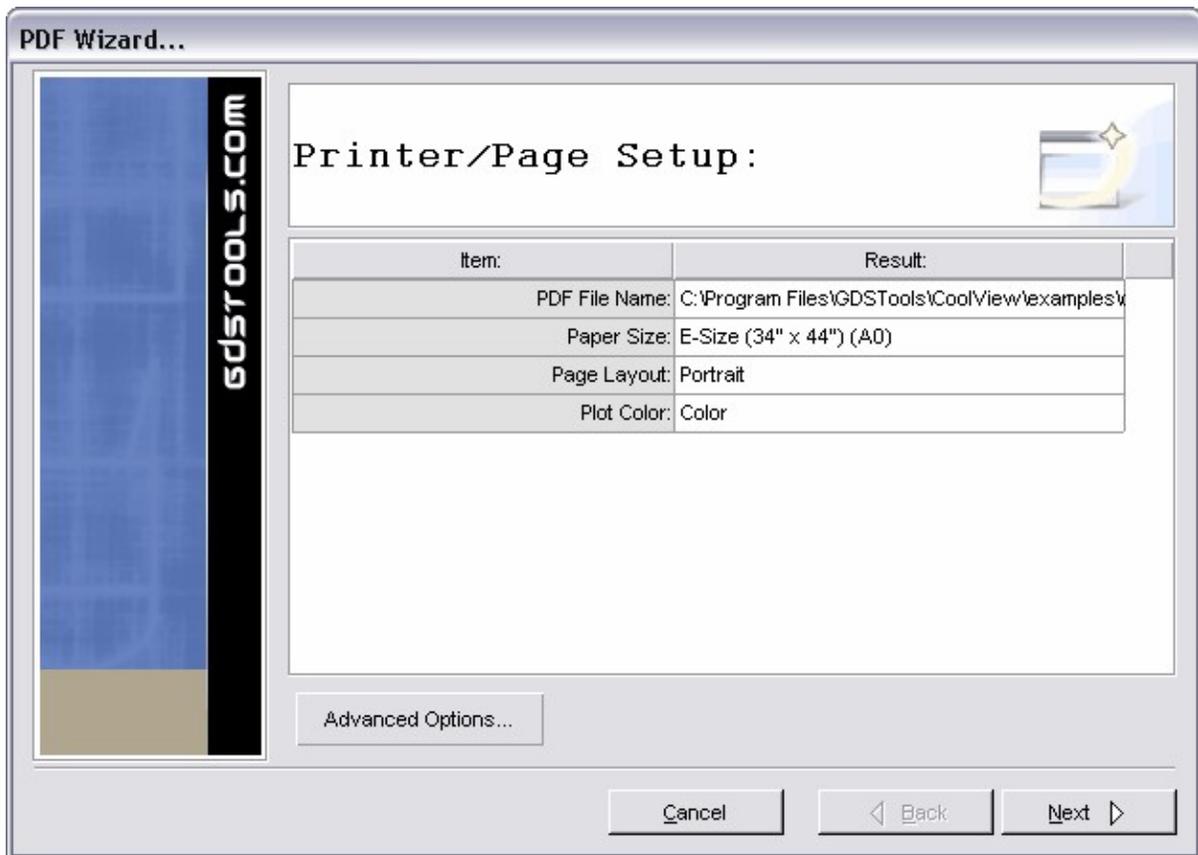
Writes a custom Aperture Library to an external file system.

---

#### 6.1.12 Plot to PDF (up to E Size)...

Create professional EDA plots in PDF format.

---

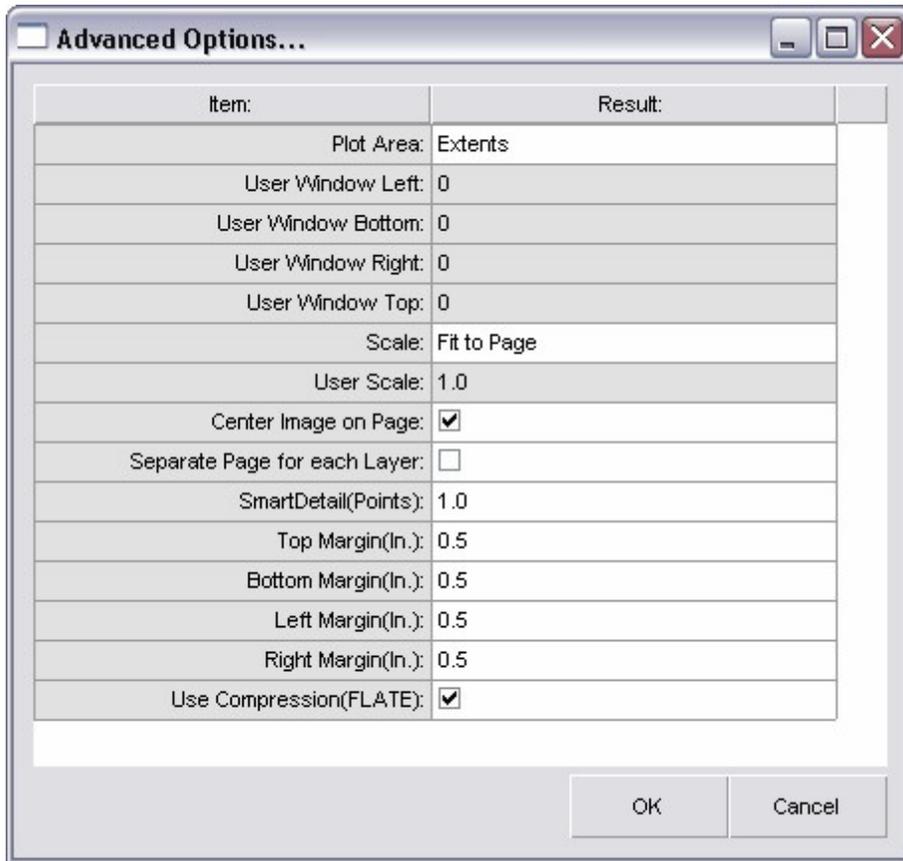


► **PDF File Name:** \_\_\_\_\_  
Location of outputted PDF file

► **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:**  
Determine page orientation:  
Portrait - Vertical page orientation  
Landscape - Horizontal page orientation

► **Plot Color:**  
Choose whether plot color will be in Black & White or Color.



►**Plot Area:**

Determine the workspace area to plot:

Extents - The smallest area that contains all geometries for the active design.

User Window - Select a specific area to plot, using the User window coordinates defined below.

►**User Window Left:**

User selected area - left coordinate.

►**User Window Bottom:**

User selected area - bottom coordinate.

►**User Window Right:**

User selected area - right coordinate.

►**User Window Top:**

User selected area - top coordinate.

►**Scale:**

Determine the image scale to use on the plot:

Fit to Page - Automatically scale up to the maximum extents.

User - Enter in a specific user scale.

►**User Scale:**

Enter in a specific image scale to use on the plot.

▶Center Image on Page:

If unchecked, image will be placed using Left, and Bottom margins.  
If checked, image is centered.

▶Separate Page for Each Layer:

If checked, a sperate page will be generated for each layer.

▶SmartDetail(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).

▶Top Margin (In.):

Spacing on the top of the page.

▶Bottom Margin (In.):

Spacing on the bottom of the page.

▶Left Margin (In.):

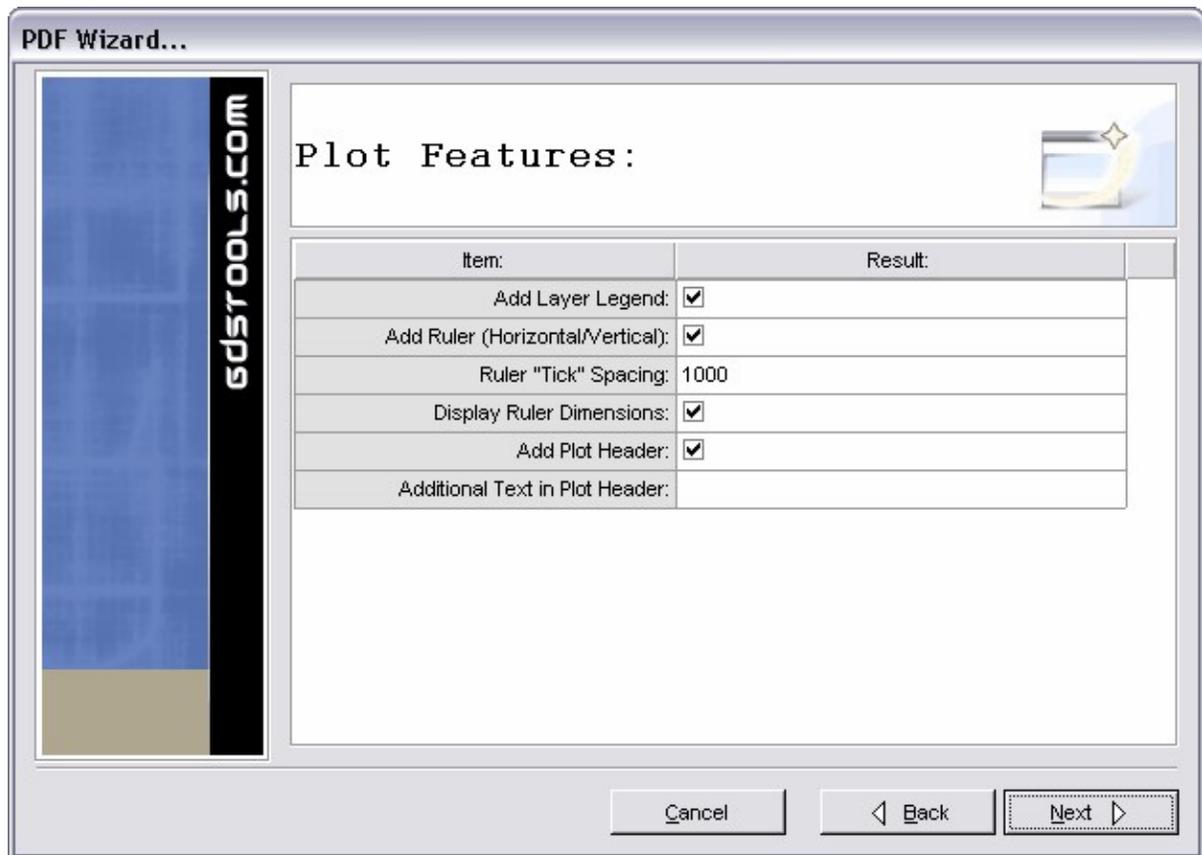
Spacing on the left of the page.

▶Right Margin (In.):

Spacing on the right of the page.

▶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 uncompress 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.



▶**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:**

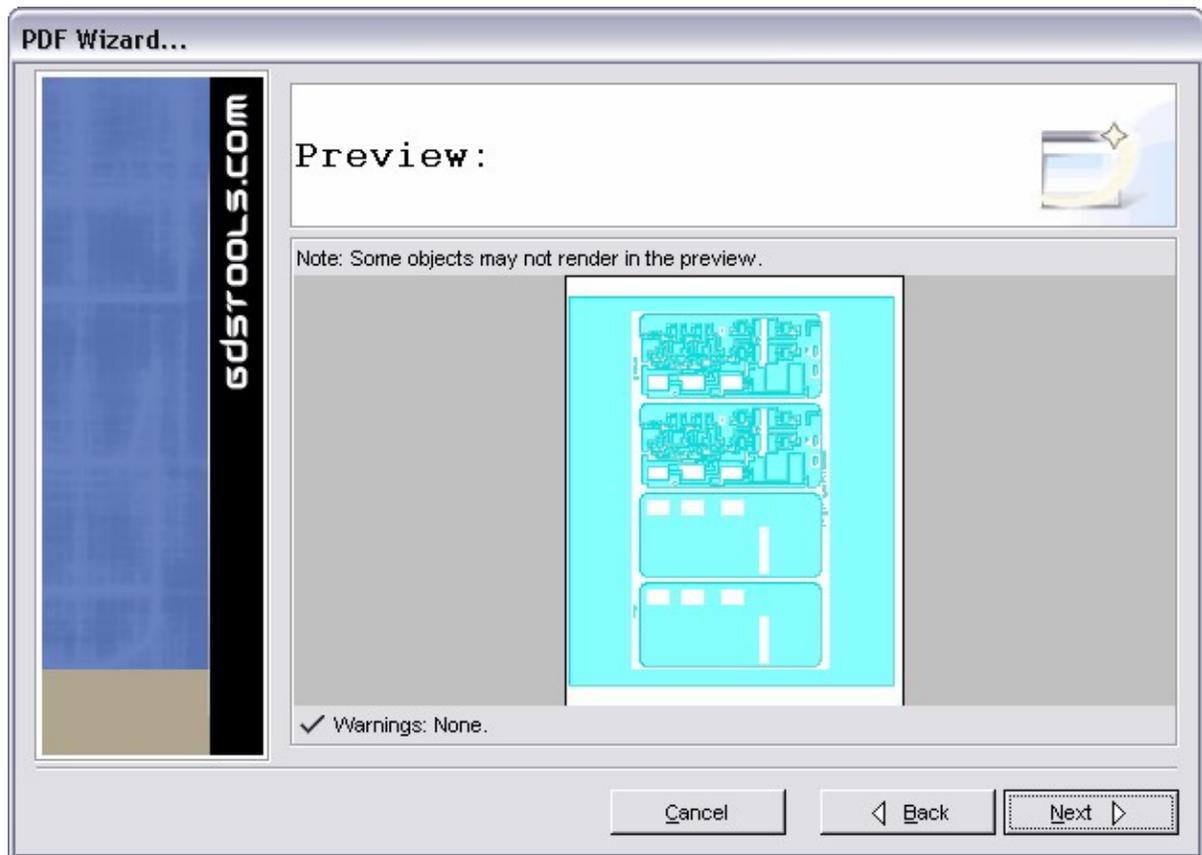
Determine whether to number each tick on the ruler.

▶**Add Plot Header:**

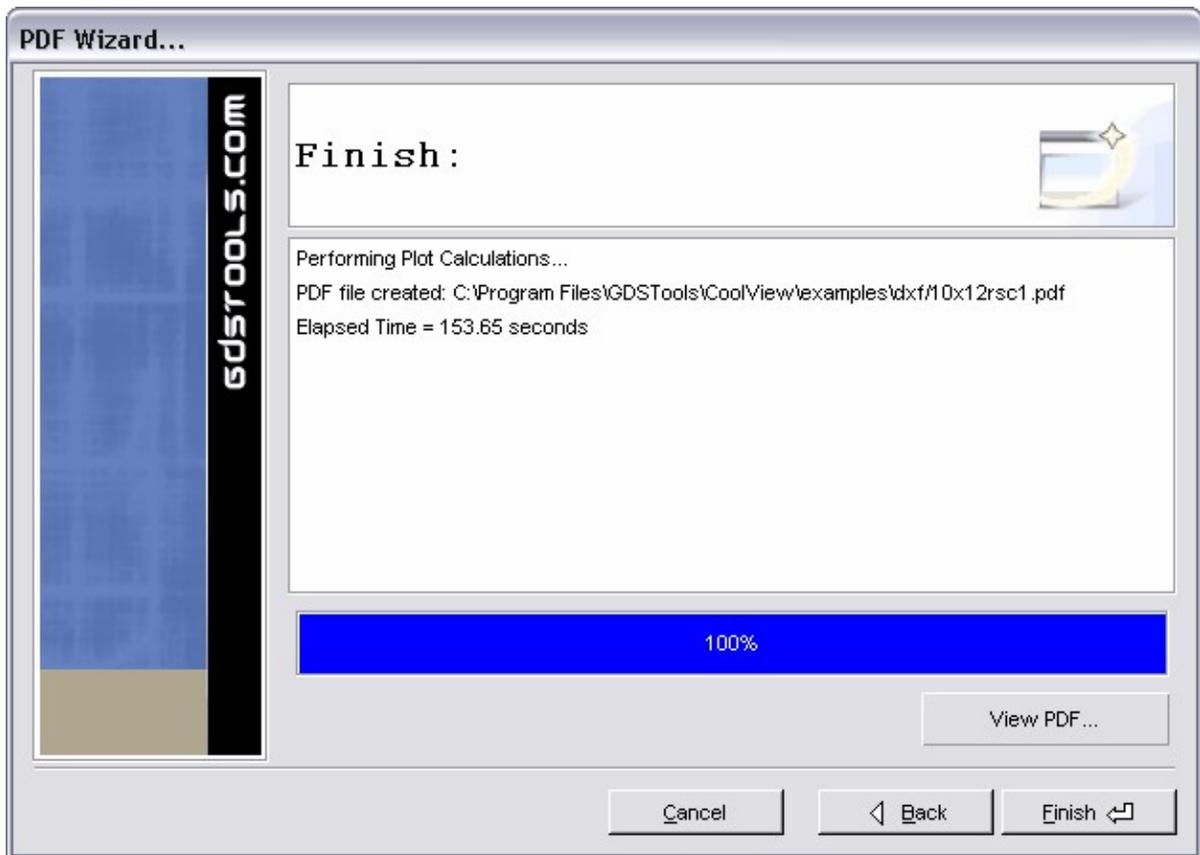
Include standard information regarding the plot (Library/design, Scale, Plot Time, etc.)

▶**Additional Text in Plot Header:**

Include specific or custom plot information.



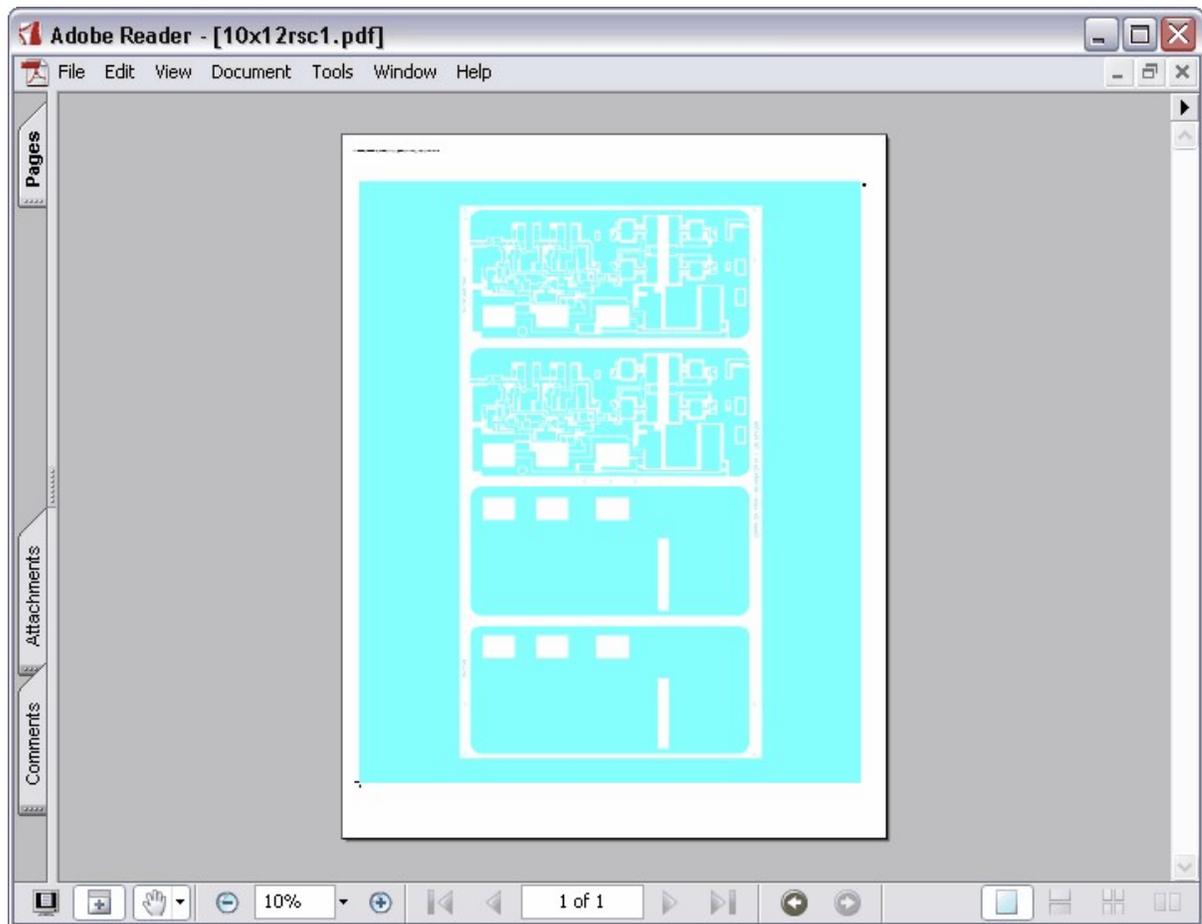
The Plot Preview gives the user the ability to preview the plot image & paper. If there are any potential problems, and warning will be placed under the preview image. If everything looks fine, selecting next will begin the creation of the PDF file.



► **View PDF (Windows Only):**

Automatically views your created PDF file in Adobe Reader.

(Screen capture of 'E' size plot in Adobe Reader)



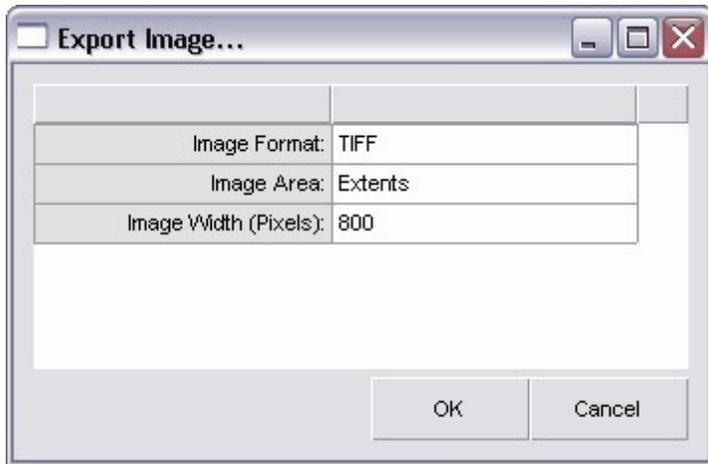
#### Notes/Links:

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

### 6.1.13 Plot to Image

Writes an Image (TIFF, PNG, BMP, GIF, etc.) file to an external file system. To export an image, the active job or panel needs to be displayed in the "editor", because Fab3000 requires the editor to render the images that will be exported.

 A maximum image width is 3000 pixels. To export a Larger Bitmap, try [Export Large Bitmap](#)<sup>[11]</sup>.

**To export an image, you must perform the following:**

- 1) Activate the "Editor", and display the job you want to export to an image.
- 2) Goto menu: File | Export Image...
- 3) Choose the Image Format (TIFF, JPEG, BMP, etc.)
- 4) Verify the image area (either Extents, or Display)
- 5) Verify the image width size in pixels.
- 6) Select the file name & path for the image.

### 6.1.14 Recent Workspaces

Creates a drop down list of the most recently accessed workspaces.

---

### 6.1.15 Exit

Exits the program.

---

## 6.2 Edit

### 6.2.1 Undo

Undo the last command in the editor.

---

## 6.2.2 Redo

Redo the last undone command in the editor.

## 6.2.3 Clipboard

Cut, Copy, and Paste Objects from the clipboard. This clipboard is different from an OS clipboard (ie. Windows clipboard), because it allows you to cut, copy, and paste objects between different jobs in Fab3000. This feature is very useful for merging parts of different jobs together to build new jobs.

## 6.2.4 Selection

### 6.2.4.1 All

Select All Objects in the editor.



Note: This command will automatically place the editor in Select Mode.

### 6.2.4.2 Display

Select Objects currently displayed.



Note: This command will automatically place the editor in Select Mode.

### 6.2.4.3 Window

Select Objects inside a user defined window.

Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) <sup>17</sup>

## Command Sequence

1. Select menu: Edit | Selection | Window.
2. Pick first point. Press left mouse button down and hold.

3. Drag mouse to opposite corner of the selection window. Release left mouse button.

 **Note:** 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.

 **Note:** This command will automatically place the CAM editor in Select Mode.

#### 6.2.4.4 Crossing Window

Select Objects inside and intersecting a user defined window.



Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) 

#### Command Sequence

1. Select menu: Edit | Selection | Crossing Window.
2. Pick first point. Press left mouse button down and hold.
3. Drag mouse to opposite corner of the selection window. Release left mouse button.

 **Note:** 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 fit inside or intersect the defined window are selected.

 **Note:** This command will automatically place the CAM editor in Select Mode.

#### 6.2.4.5 Fence

Select objects using a polyline.



Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) 

#### 6.2.4.6 Window Polygon

Select objects inside a user defined polygon.



Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) 

#### 6.2.4.7 Crossing Polygon

Select objects inside and intersecting a user defined polygon.



Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) 

#### 6.2.4.8 Clear All

Remove selection.

#### 6.2.4.9 Filter

The selection filter is an extremely useful tool used for controlling the content of Objects that can be selected prior to performing any modifications or queries. The selection filter can be a major time saver as it parses out Objects that do not pass certain requirement that you set.

Note: For more information about object selection see Learn Guide: [Viewing & Selection Basics](#) <sup>17</sup>

### Command Sequence

1. Select menu: Edit | Selection | Filter... The Selection Filter dialog will appear.

 Note: Items here allow you to include or exclude specific Dcodes, Layers, Composite Levels, NC Tools, Polygons, or Text from being selected. For the items with check boxes, a check means the item can be selected, removing a check means the item cannot be selected.

2. Verify all Filter items. Select OK when done

 Note: Enter numbers and/or ranges separated by commas.

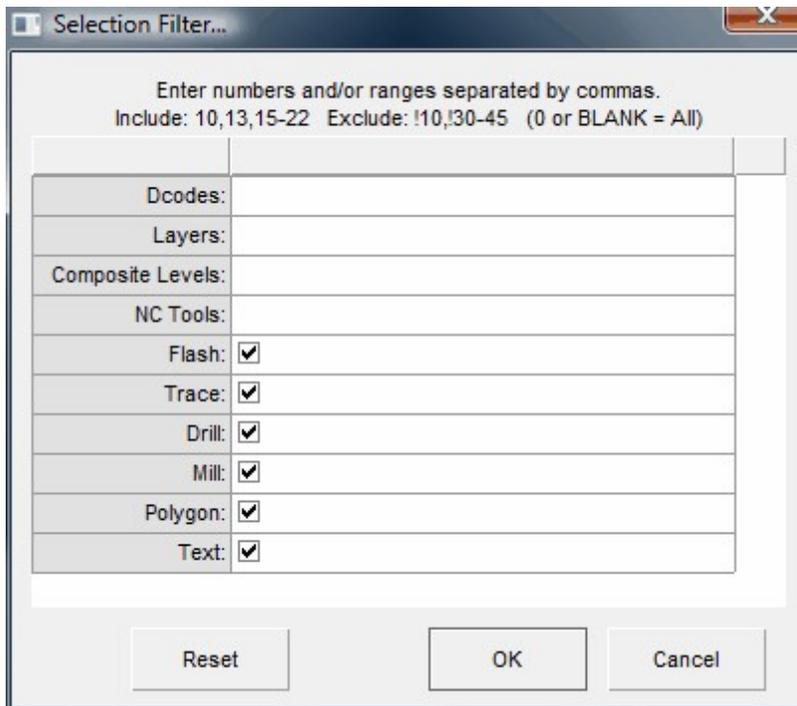
Include: 10,13

Ranges: 15-22,40-42

Exclude: !10,!30-45

Select All: 0 or BLANK

 Note: The Selection Filter is only good for on selection command (ie. Window, Crossing, All, etc.). If you wish to select another Crossing Window, you will need to go back to step 1, and repeat.



#### 6.2.4.10 Reset Filter

Clears the selection filter.

### 6.2.5 Layers

#### 6.2.5.1 Add Layers

Adds a layer.

#### Command Sequence

1. Select menu: Edit | Layers | Add Layer. The Add Layers dialog box will appear.
2. Check the layers that you want to delete to (see below).

3. Enter in the "Layer Name" you want to use.

 Note: If this layer name already exists, no new layer will be created.

4. Enter in the "Layer Number" you want to use. (Optional)

 Note: Fab3000 will assign the next available layer number, and in most cases you do not need to modify this. If this layer number already exists, no new layer will be created.

 Note: This command cannot be undone.

### 6.2.5.2 Delete Layers

Delete selected layers.

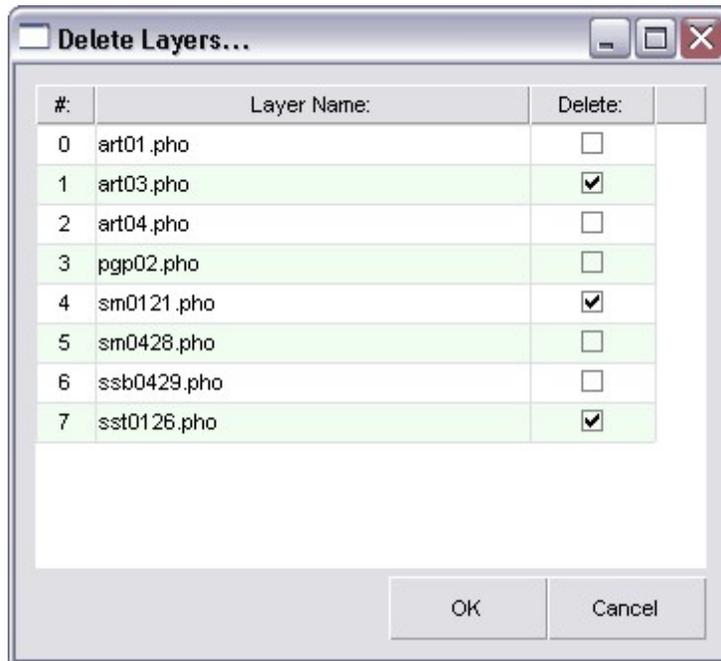
#### Command Sequence

1. Select menu: Edit | Layers | Delete Layers. The Delete Layers dialog box will appear.

2. Check the layers that you want to delete to (see below).

3. Press OK button to begin delete.

 Note: This command cannot be undone.



### 6.2.5.3 Merge Layers

Merge multiple layers simultaneously.

 **Note:** This merges layers without polarity (ie. composites). If you want to merge layers to create composites(dark/clear) use [Composite Layer | Build](#)<sup>189</sup>.

#### Command Sequence

1. Select menu: Edit | Layers | Merge Layers. The Merge Layers dialog box will appear.
2. Enter the "Merge Layer Name" (see below). This is the name for the new merged layer.
3. To select layers for the merge, click in the column box to the right of "From Layer(s):". The Select Layers dialog box will appear.
4. Highlight the layers you want to include for the merge, and press OK button.

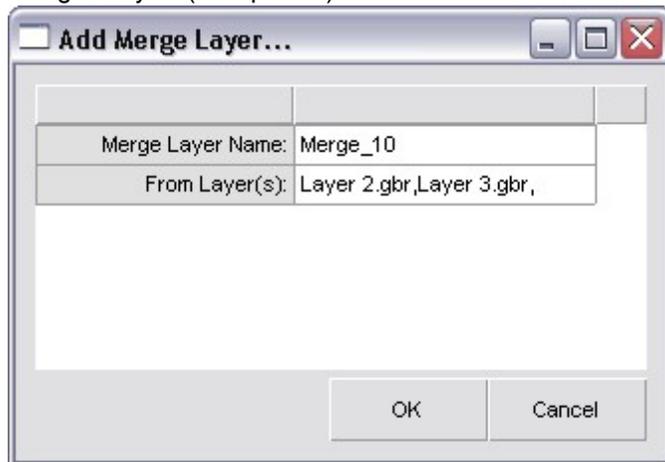
 **Note:** If you want to add another layer to the merge, or you made an error on layer selection, simply re-click in the column box to the right of "From Layer(s):", and reselect the layers to merge.

5. Press OK button to begin layer merge.

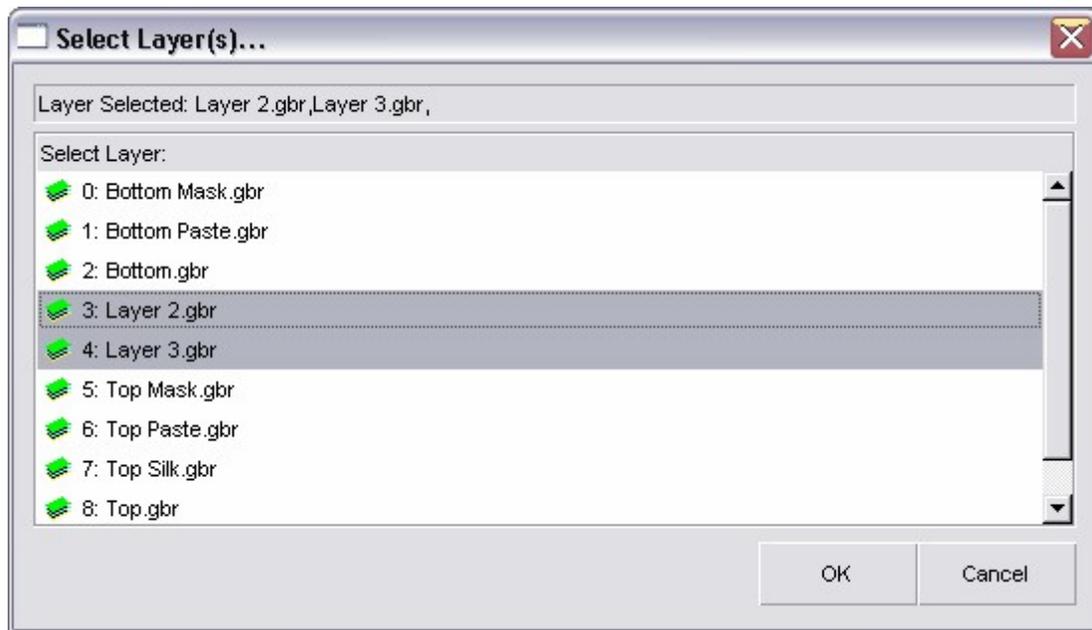
 **Note:** This command cannot be undone.

---

#### Merge Layer (Snapshot)



#### Merge Layer | Select Layers (Snapshot)



#### 6.2.5.4 Scale Layer X/Y

This command scales a layer in X and Y directions respectively.

#### 6.2.5.5 Sort Layers

Automatically sort Layers per the Recommend PCB Stackup.

[TOP SILK]  
[TOP PASTE]  
[TOP MASK]  
[TOP]  
[...INTERNAL METAL LAYERS...]  
[BOTTOM]  
[BOTTOM MASK]  
[BOTTOM PASTE]  
[BOTTOM SILK]

### 6.2.6 Align Layers

Adjusts a layer origin to become aligned with another layer based off a similar object (ie. target, pad, border line, etc.)

 **Note:** 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.

 Online Movie: [FAB3000 Align gerber and drill layers](#)

## Command Sequence

1. Select menu: Edit | Layers | Align Layers.
2. Select the Base Object.

 **Note:** This object must reside on the the master (ie. base) layer, and there must be a similar object on the next (ie. to be aligned) layer.

3. Select the Next Object to Align

 **Note:** This object must be of the same object type (as the Base Object) and reside on the next (ie. to be aligned) layer.

4. Fab3000 will now instantly align the Next layer to be directly on top of the base layer, using those two objects as reference points.

## 6.2.7 Delete

Remove selected Objects from the active design.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

## Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

 **Note:** To select polygons or traces, pick the edges.

2. Select menu: Edit | Delete

3. If the incorrect object is deleted, you may [Undo](#)<sup>[122]</sup>.

## 6.2.8 Move

This command moves objects to a new location.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

## Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>[17]</sup>

 Note: To select polygons or traces, pick the edges.

2. Select menu: Edit | Move

3. Enter Base point.

4. Enter Second point of displacement. The objects will be moved by the displacement amount

5. If the move is incorrect, you may [Undo](#) <sup>[12]</sup>.

Repeat Step 4

OR

Right-Click or Press 'Esc' to exit this command.

 Note: To quickly move a selection, Left-click over a selected object and drag the entire selection to any location in the editor.

## 6.2.9 Copy

This command creates multiple copies of the selection to new locations.

 Note: This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

## Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>[17]</sup>

 Note: To select polygons or traces, pick the edges.

2. Select menu: Edit | Copy
3. Enter Base point.
4. Enter Second point of displacement. The objects will be copied and moved by the displacement amount
5. If the command is incorrect, you may [Undo](#) .

Repeat Step 4

OR

Right-Click or Press 'Esc' to exit this command.

### 6.2.10 Copy to layers

Copy selected objects to other layers.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

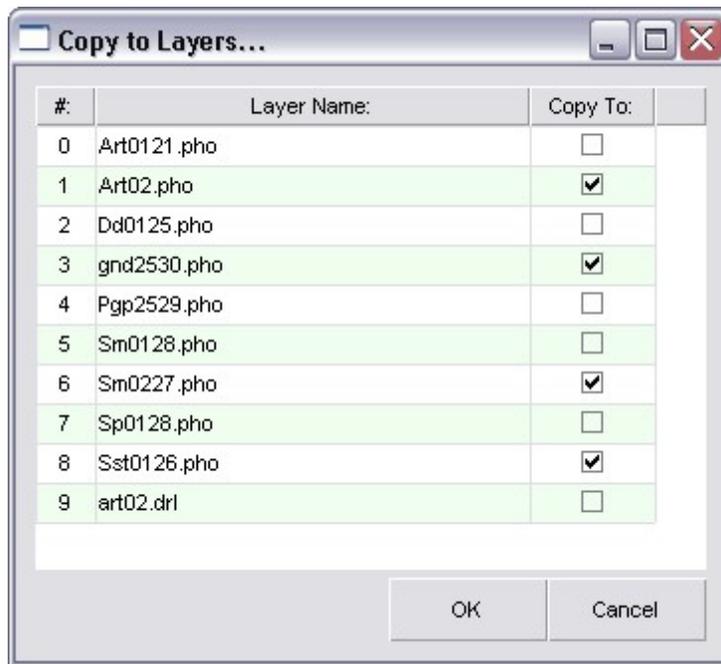
#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) .

 **Note:** To select polygons or traces, pick the edges.

2. Select menu: Edit | Copy to Layers. The Copy to Layer dialog box will appear.
3. Check the layers that you want to copy objects to (see below). Press OK button to begin copy.
4. If the command is incorrect, you may [Undo](#) .

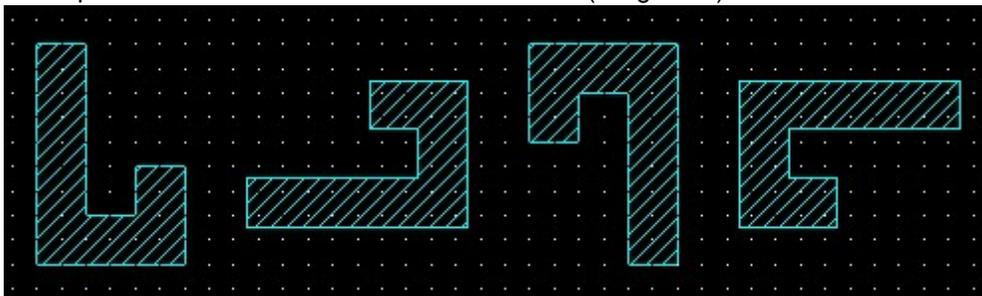


### 6.2.11 Rotate

Rotate selected Objects orthogonally by either 90, 180, 270 degrees.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

Example of Rotation: 0 - 90 - 180 - 270 (Degrees):



#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) <sup>17</sup>

 **Note:** To select polygons or traces, pick the edges.

2. Select menu: Edit | Rotate | (use either 90, 180, 270, or set Angle)

Selected objects will now be rotated.

3. If the command is incorrect, you may [Undo](#) .

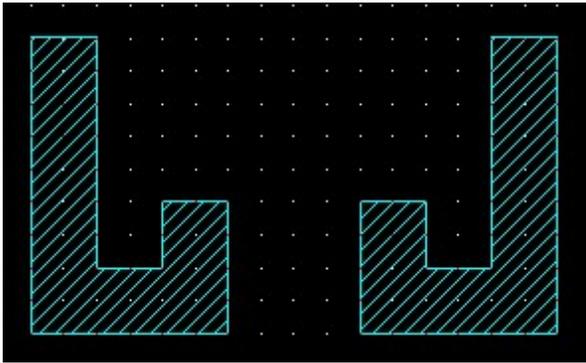
## 6.2.12 Mirror / Flip

Mirror selected Objects either horizontally or vertically.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

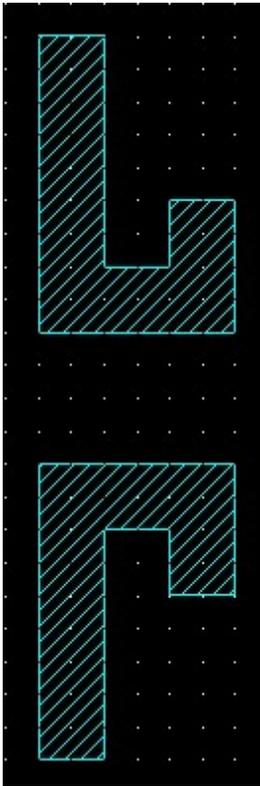
Horizontal:

Flips selection about the Y-axis.



Vertical:

Flips selection about the X-axis.



## Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

 Note: To select polygons or traces, pick the edges.

2. Select menu: Edit | Mirror | Horizontally or Flip | Vertically

3. Specify Mirror Line location for objects.

4. Determine if you want to delete the source (original objects).

Note: Selected objects will now be flipped.

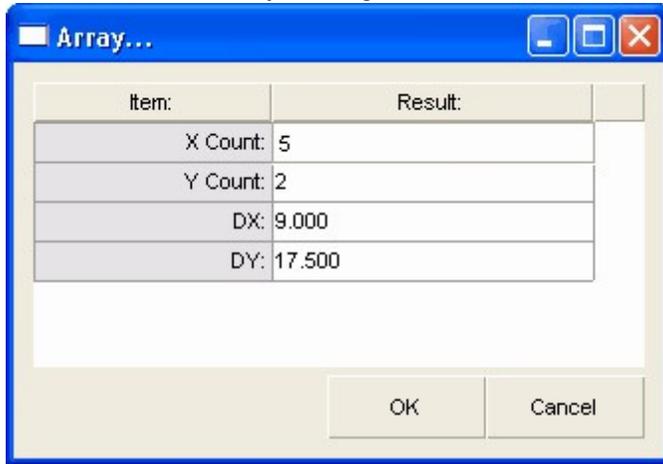
Note: If the command is incorrect, you may [Undo](#)<sup>[12]</sup>.

### 6.2.13 Array

This command arrays the selection by a specified number and distance in both X and Y directions.  
Aka: Rectangular Array

 Note: This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

Screen shot of Array dialog:



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.

## Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) <sup>[17]</sup>

2. Select menu: Edit | Array | Rectangular. The Array dialog box will appear.

3. Enter in the "X Count" you want to use.

4. Enter in the "Y Count" you want to use.

5. Enter in the "DX" you want to use.

 Note: The DX is automatically filled in as the overall horizontal extents of the selected objects.

6. Enter in the "DY" you want to use.

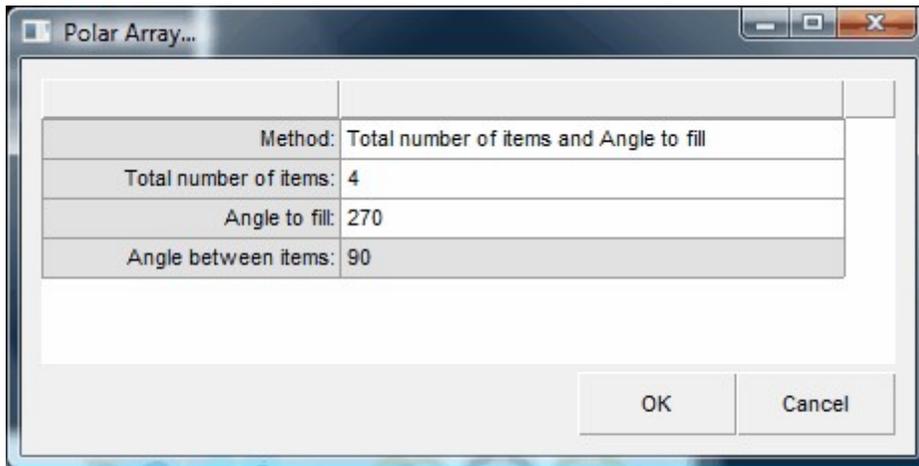
 Note: The DY is automatically filled in as the overall vertical extents of the selected objects.

7. Select OK to begin.

8. If the command is incorrect, you may [Undo](#) <sup>[122]</sup>.

## 6.2.14 Polar Array

This command arrays the selection by a specified center point and angle to fill.



Polar Array Options:

Method: Total number of items and Angle to fill  
Total number of items and Angle between items  
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.

### Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Array | Polar.

3. Specify Center Point.

 Note: Pick a point in the editor, or enter coordinates in command window

The Polar Array dialog box will appear.

4. Choose the Method you want to use.

5. Enter in the appropriate options.

6. Select OK to begin.

7. If the command is incorrect, you may [Undo](#) .

### 6.2.15 Scale

Scale selected objects.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

 **Note:** If you would like to scale an entire layer by different X & Y values, use: [Scale X/Y](#) .

#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) .

2. Select menu: Edit | Scale. The Scale dialog box will appear.

3. Enter in the Scale Factor you want to use.

4. Check if you want to scale apertures or Uncheck to leave the same.

5. If the command is incorrect, you may [Undo](#) .

### 6.2.16 De-Embed

Automatically solves the polygon-inside-polygon problem for objects selected. The feature will generally result



#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) .

2. Select menu: Edit | De-Embed...

3. Object Composite Levels will be automatically defined.

 **Note:** Objects must reside on the same layer.

4. If the command is incorrect, you may [Undo](#) .

### 6.2.17 Stretch

This command stretches objects crossing a defined rectangular area.

### 6.2.18 Split

This command splits a path into two segments and the selected location.

### 6.2.19 Vertex

Vertex modification tools. Move, Insert, Delete.

### 6.2.20 Trim

Trim objects using a reference line or circle.

### 6.2.21 Boolean

Perform 2D boolean operations on two objects: AND,OR,XOR,Subtract.

### 6.2.22 Explode

#### 6.2.22.1 All

Convert selected Objects into base objects.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Explode | All

#### 6.2.22.2 Flashes

Convert Flashes into polygons.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Explode | Flashes

### 6.2.22.3 Traces

Converts traces and Polygons to segments.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Explode | Traces.

 **Note:** Polygons will be exploded into individual segments with 1-mil width.

### 6.2.22.4 Step and Repeat

Flattens Step and Repeat Objects.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Explode | Step and Repeat.

 **Note:** After exploding a Step and Repeat, your database sizes can increase in size exponentially!

### 6.2.22.5 Text

Convert Text into polygons.



 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Explode | Text.

## 6.2.23 Properties

View and modify object properties.

 Note: For a detailed description on how to use this function goto [Learn Guide: Using the Properties Dialog](#) 

 Note: This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

 Note: Not only does this command allow you to view all object properties, you may also edit them.

 Note: Double-Click any object to display this properties dialog.

 Note: When multiple objects are selected, the "Multiple Objects" version of the Properties dialog box will be displayed.

### Command Sequence

1. Select Objects.

 Note: For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Edit | Properties. The Properties dialog box will appear.

3. Review all items displayed. Most items can be changed.

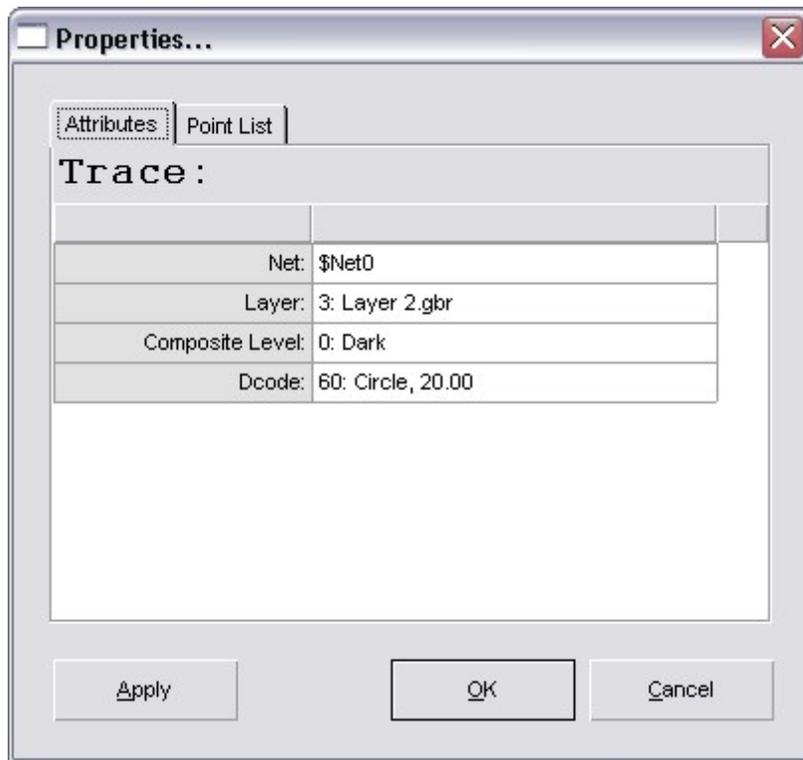
4. Click on the tab "Point List", to view or edit an objects vertices or center point. (Optional)

5. Click Ok to exit the properties dialog.

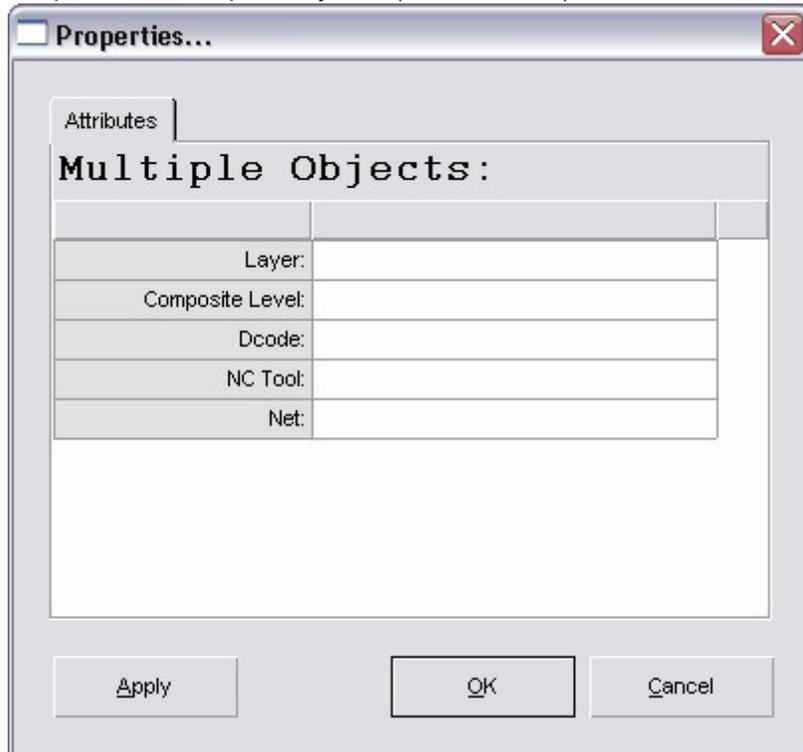
 Note: If changes were made that are incorrect, you may [Undo](#) .

---

Properties... (Screen shot)



Properties: Multiple Objects (screen shot)



## 6.3 View

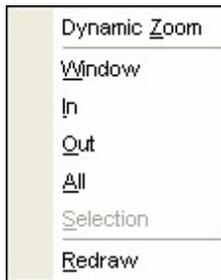
### 6.3.1 Redraw

Refreshes/redraws the display in the Editor window.

### 6.3.2 Zoom

The zooming commands increase or decrease the magnification causes objects to look larger or smaller.

Zoom Menu:



Dynamic Zoom:

Allows user to zoom in or out by dragging the mouse cursor left (for in) and right (for out). A mouse that has a scrolling wheel can also perform this function.

Window:

Allows user to pick a window around area to be zoomed.

In:

Doubles the magnification.

Out:

Halves the magnification.

All:

Shows entire design area in editor.

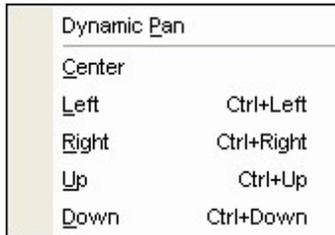
Selection:

Zooms into the extents of the current selection.

### 6.3.3 Pan

The panning commands allows user to scroll about the current view port in the editor. Pan does not change magnification.

Pan Menu:



**Dynamic Pan:** Allows user to drag the pan view and drop in desired location.

**Center:** Allows user to click the center of the desired pan point.

**Left:** Moves the view to the left by one-quarter of the width of the view.

**Right:** Moves the view to the right by one-quarter of the width of the view.

**Up:** Moves the view up by one-quarter of the height of the view.

**Down:** Moves the view down by one-quarter of the height of the view.

### 6.3.4 Fill

Toggles the fill mode. Fill mode controls filling of all boundaries using the layers' stipple pattern and color.

### 6.3.5 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.

Snap Menu



**Snap Off:** Snap is disabled.

**Snap to Grid:** Cursor location is snapped to the nearest grid point.

**Snap to Object:** Cursor location is snapped to the nearest Object vertex.

### 6.3.6 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.

### 6.3.7 Object View Filter

Assign Object View Filter.

### 6.3.8 Layer Display

Controls the display of layers in the editor.

### 6.3.9 External Centroids

Show or set viewing options for an imported part centroid's.

### 6.3.10 Translucent

Toggles whether to display geometries as see-through or opaque.

### 6.3.11 Highlight Active Dcode

Toggles whether to display active Dcode.

### 6.3.12 Origin Icon

Toggles whether to display the origin icon in the editor. The origin icon is usually used for reference in a layout.

### 6.3.13 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.

### 6.3.14 Inside Step & Repeat

Toggles the viewing of objects inside inserts.

### 6.3.15 Job Profile

Toggles the viewing of the Job Profile.

### 6.3.16 Backside

View Job from the Backside. Very helpful when editing needs to be performed from the reverse view of the board.

### 6.3.17 Grid

Toggles whether to display grid.

### 6.3.18 Markup

Toggles whether to display Markup (if any).

### 6.3.19 Customize Toolbars

#### 6.3.19.1 Toolbar

Toggles the visibility of the toolbar.

#### 6.3.19.2 Command Window

Toggles the visibility of the command bar. The Command bar is a row located on the bottom side of the workbench. It contains the Command view.

#### 6.3.19.3 Design Navigator

Toggles the visibility of the navigator bar. The Navigator bar is a column located on the right side of the workbench. It contains both the Navigator and Outline views.

#### 6.3.19.4 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.

### 6.3.20 Full Screen

Allows the user increase the Editor window to the full allowable size of the screen.

### 6.3.21 Command Console

Display the command widget in a dialog.

## 6.4 Add

### 6.4.1 Pad

Adds a Flash the active design.

 **Note:** At least one aperture must have been previously defined.

 **Note:** The Flash will be placed on the active layer.

### Command Sequence

1. Select menu: 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.

 **Note:** 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.
6. To exit Add Flash: Right Click or press 'Esc' .
7. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

 **Note:** To add flash to a net, see Learn Guide: [Add/Remove Objects from a Net](#)<sup>[56]</sup>

 **Note:** To add flash to a composite, see Learn Guide: [Composites 101](#)<sup>[58]</sup>

## 6.4.2 Trace

Adds a Trace (Line) to the active design.



 **Note:** At least one aperture must have been previously defined.

### Command Sequence

1. Select menu: 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.

 **Note:** 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 First point for the trace.
5. Select the Next point for the trace.

 **Note:** You have the following additional options End,Close,Back.

End - Ends point list. (press key 'E')

Close - End and close point list. (press key 'C')

Back - Remove the last point from the current point list. (press key 'B')

Type the first character

6. Once you have finished your trace: End or Close.
7. Repeat Steps 2-6
8. To exit Add Trace: Right Click or press 'Esc' .
9. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

 **Note:** Two coincident and three collinear points are illegal in the point list.

 **Note:** To add traces to a net, see Learn Guide: [Add/Remove Objects from a Net](#)<sup>[56]</sup>

 **Note:** To add traces to a composite, see Learn Guide: [Composites 101](#)<sup>[58]</sup>

## 6.4.3 Circle / Arc

Adds a Circle to the active design.



 Note: At least one aperture must have been previously defined.

## Command Sequence

1. Select menu: 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.

 Note: 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 circle.
5. Select the radius, orientation, and degrees for the circle.
6. Repeat Steps 2-5
7. To exit Add Circle: Right Click or press 'Esc' .
8. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

 Note: To add circle to a net, see Learn Guide: [Add/Remove Objects from a Net](#)<sup>[56]</sup>

 Note: To add circle to a composite, see Learn Guide: [Composites 101](#)<sup>[58]</sup>

### 6.4.4 Polygon

Adds a Polygon to the active design.



 Note: No defined aperture is required.

## Command Sequence

1. Select menu: Add | Polygon
2. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 Note: 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).

3. Select the First point for the polygon.
4. Select the Next point for the polygon.

 Note: You have the following additional options Close,Back.  
Close - End and close point list. (press key 'C')

Back - Remove the last point from the current point list. (press key 'B')  
Type the first character

5. Once you have finished your trace: Close (or Right-Click)
6. Repeat Steps 2-5
7. To exit Add Polygon: Right Click or press 'Esc' .
8. If the command is incorrect, you may [Undo](#) <sup>[122]</sup>.

 Note: Two coincident and three collinear points are illegal in the point list.

 Note: To add Polygon to a net, see Learn Guide: [Add/Remove Objects from a Net](#) <sup>[56]</sup>

 Note: To add Polygon to a composite, see Learn Guide: [Composites 101](#) <sup>[58]</sup>

### 6.4.5 Rectangle

Adds a Filled Rectangle to the active design.



#### Command Sequence

1. Select menu: Add | Rectangle.
2. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 Note: 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).

3. Select the First corner for the rectangle.
4. Select the Opposite corner for the rectangle.
5. Repeat Steps 2-4
6. To exit Add Rectangle: Right Click or press 'Esc' .
7. If the command is incorrect, you may [Undo](#) <sup>[122]</sup>.

 Note: To add rectangle to a net, see Learn Guide: [Add/Remove Objects from a Net](#) <sup>[56]</sup>

 Note: To add rectangle to a composite, see Learn Guide: [Composites 101](#) <sup>[58]</sup>

### 6.4.6 Ellipse

Adds a filled Ellipse to the active design.



## Command Sequence

1. Select menu: Add | Ellipse.
2. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 **Note:** 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).

3. Select the First corner for the ellipse.
4. Select the Opposite corner for the ellipse.
5. Repeat Steps 2-4
6. To exit Add Ellipse: Right Click or press 'Esc' .
7. If the command is incorrect, you may [Undo](#) <sup>[122]</sup>.

 **Note:** To add ellipse to a net, see Learn Guide: [Add/Remove Objects from a Net](#) <sup>[56]</sup>

 **Note:** To add ellipse to a composite, see Learn Guide: [Composites 101](#) <sup>[58]</sup>

### 6.4.7 Wide Path

Adds a Wide Path (polygon) to the active design.



 **Note:** No defined aperture is required.

## Command Sequence

1. Select menu: Add | Wide Path
2. Select Path Width. Width cannot be 0.
3. Select Path Style.

 **Note:** Path Styles include  
Truncate - No Path extension.  
Extend - Extend by 1/2 path width.

4. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 **Note:** 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).

5. Select the First point for the trace.

6. Select the Next point for the trace.

 Note: You have the following additional options End,Close,Back.

End - Ends point list. (press key 'E')

Close - End and close point list. (press key 'C')

Back - Remove the last point from the current point list. (press key 'B')

Type the first character

7. Once you have finished your trace: End or Close.

8. Repeat Steps: 4-7

9. To exit Add Wide Path: Right Click or press 'Esc' .

10. If the command is incorrect, you may [Undo](#) <sup>[122]</sup>.

 Note: Two coincident and three collinear points are illegal in the point list.

 Note: To add Wide Path to a net, see Learn Guide: [Add/Remove Objects from a Net](#) <sup>[56]</sup>

 Note: To add Wide Path to a composite, see Learn Guide: [Composites 101](#) <sup>[58]</sup>

## 6.4.8 Text

Adds Text to the active design.



### Command Sequence

1. Select menu: Add | Text.

2. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 Note: 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).

3. Enter Text String. The text message to be used.

4. Enter Text Height. The text height to be used.

5. Enter Text Style. The text font style to be used.

 Note: The text style combo box includes all Fab3000 compiled fonts located in the Fab3000 folder "/> fonts". If there is not a text style available that you want, you may create your own Fab3000 compiled font from any TrueType or AutoCAD SHX font. See: [Building New Font Definition \(\\*.FNT\)](#) <sup>[212]</sup>

6. Enter Orientation

7. Select the Origin point for text.

8. Repeat Steps 2-7

10. If the command is incorrect, you may [Undo](#) <sup>f122</sup>.

 **Note:** To add text to a composite, see Learn Guide: [Composites 101](#) <sup>f58</sup>

---

#### Add Text Options:



##### String:

The text message to be used.

##### Height:

The text height to be used.

##### Style:

The text font style to be used.

##### Orientation:

The Orientation to be used.

##### Notes/Links:

The Object will be placed on the active layer.

## 6.4.9 Multi-Line Text

Adds Multi-Line(paragraph) text to the active design.

### Command Sequence

1. Select menu: Add | Multi-Line Text.
2. Assign the active layer. The active Layer combo box, is located in the Toolbar.

 **Note:** 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).

3. Select the first corner for multi-line text.
4. Select the opposite corner. The Add Multi-Line Text dialog box will appear.
5. Enter Multi-Line Text String. The text message to be used.
6. Enter Text Height. The text height to be used.

7. Enter Text Style. The text font style to be used.

 Note: The text style combo box includes all Fab3000 compiled fonts located in the Fab3000 folder "/> fonts". If there is not a text style available that you want, you may create your own Fab3000 compiled font from any TrueType or AutoCAD SHX font. See: [Building New Font Definition \(\\*.FNT\)](#)<sup>[212]</sup>

8. Enter Alignment. The alignment to be used.

 Note: The alignment options are upperLeft, centerLeft, lowerLeft, upperCenter, centerCenter, lowerCenter, upperRight, centerRight, lowerRight.

9. Enter Orientation

10. Stroke Width. Determine how to fill text: Choose either Active Dcode or Polygon.

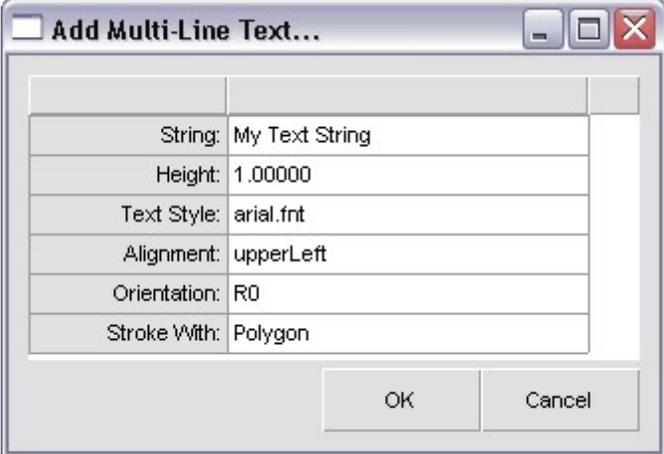
11. Press OK to create Text.

12. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

 Note: To add multi-line to a composite, see Learn Guide: [Composites 101](#)<sup>[58]</sup>

---

#### MText Options:



String:	My Text String
Height:	1.00000
Text Style:	arial.fnt
Alignment:	upperLeft
Orientation:	R0
Stroke With:	Polygon

OK Cancel

**String:**  
The text message to be used.

**Height:**  
The text height to be used.

**Text Style:**  
The text font style to be used.

**Alignment:**  
The alignment to be used.

The alignment options are upperLeft, centerLeft, lowerLeft, upperCenter, centerCenter, lowerCenter, upperRight, centerRight, lowerRight.

Stroke With:

Determine how to fill text. Choose either Active Dcode or Polygon.

Tutorial:

1. Activate command MText
  2. Select Bounding Rectangle for MText
  3. Enter Text string
  4. Verify text options (orientation, text style, etc)
- You will now see the text appear in the editor.



Notes/Links:

The multi-line text will be placed on the active layer.

## 6.4.10 Dimension

Adds Dimension to the active design. Choose from: Ordinate, Horizontal, Vertical, Radius.



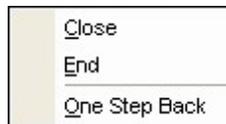
## 6.4.11 Markup

### 6.4.11.1 Markup Line

Adds a Markup Line to the active design.



Standard Point List menu:



Close: End and close point list.

End: Ends point list.

One Step Back: Remove the last point from the current point list.

Tutorial:

1. Activate command: Markup Line
  2. Pick First point
  3. Pick Next point
  4. Repeat step #3, until a Close or End is activated.
- You will now see the markup line appear in the editor.



Notes/Links:

The markup line will be placed on the active layer.

### 6.4.11.2 Markup Rectangle

Adds a Markup Rectangle to the active design.



Tutorial:

1. Activate command: Markup Rectangle
  2. Select First corner.
  3. Select Opposite corner.
- You will now see the markup rectangle appear in the editor.



Notes/Links:

The markup rectangle will be placed on the active layer.

### 6.4.11.3 Markup Ellipse

Adds a Markup Ellipse to the active design.



Tutorial:

1. Activate command: Markup Ellipse
  2. Select First corner.
  3. Select Opposite corner.
- You will now see the markup ellipse appear in the editor.



Notes/Links:

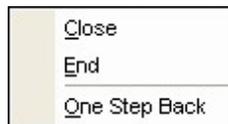
The markup ellipse will be placed on the active layer.

### 6.4.11.4 Markup Polygon

Adds a Markup Polygon to the active design.



Standard Point List menu:



- Close: End and close point list.  
End: Ends point list. If a polygon, the point list will be closed anyways.  
One Step Back: Remove the last point from the current point list.

Tutorial:

1. Activate command: Markup Polygon.
  2. Pick First point
  3. Pick Next point
  4. Repeat step #3, until a Close or End is activated.
- You will now see the markup polygon appear in the editor.



Notes/Links:

The markup text will be placed on the active markup layer.

#### 6.4.11.5 Markup Text

Adds markup text to the current markup layer.



Tutorial:

1. Activate command: Markup Text
  2. Select Bounding Rectangle for Markup Text
  3. Enter Text string
  4. Verify text options (orientation, text style, etc)
- You will now see the text appear in the editor.



Notes/Links:

The markup text will be placed on the active markup layer.

#### 6.4.11.6 Markup Arrow

Adds a markup leader with an "arrow" at the first point, to the current markup layer.



Tutorial:

1. Activate command: Markup Arrow
  2. Pick First point (this is where the arrow will point to)
  3. Pick Next point
  4. Repeat step #3, (Rt. Click to End)
- You will now see the arrow + line appear in the editor.



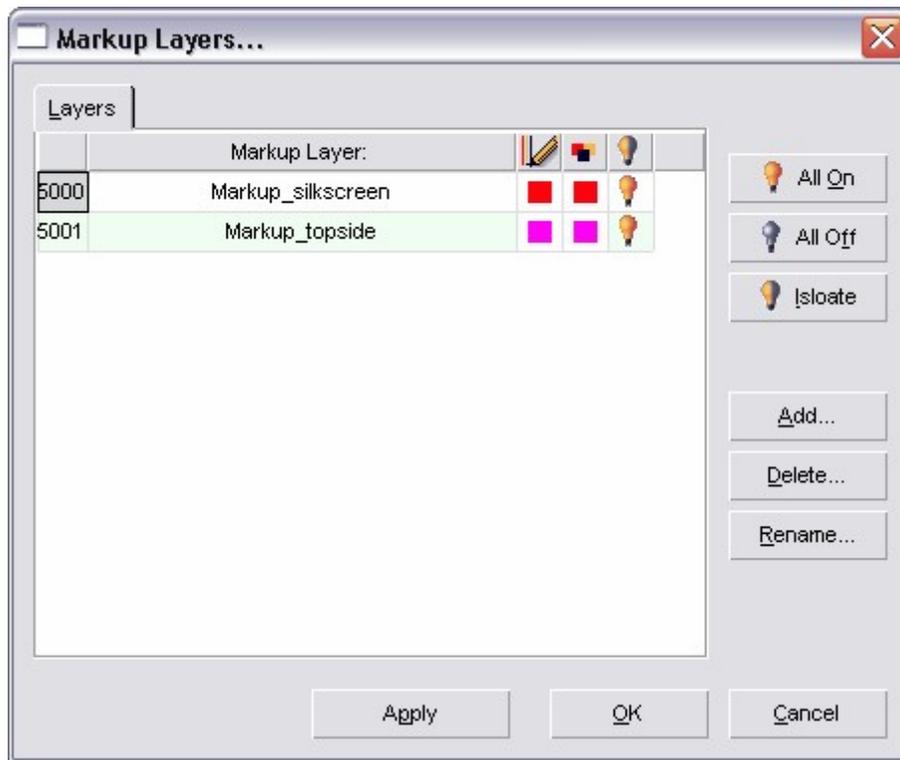
Notes/Links:

The markup arrow will be placed on the active markup layer.

#### 6.4.11.7 Markup Layers...

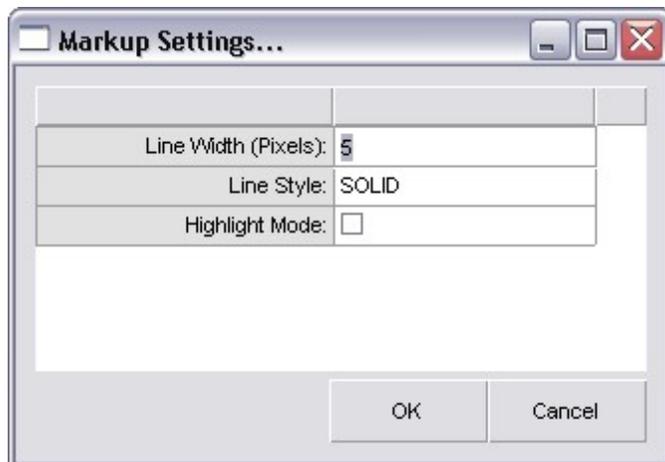
Define all markup layers. These layers only contain markup objects, and are not included during exporting.





#### 6.4.11.8 Markup Settings...

Define 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".

## 6.4.12 Drill Hit

Adds Drill Hit to the active design.

 **Note:** At least one NC Tool must already be defined.

### Command Sequence

1. Select menu: Add | Drill Hit. The Add Drill Hit dialog box will appear.
2. Assign the NC Tool to use. Press OK to start.
3. Assign the active layer. The active Layer combo box, is located in the Toolbar.

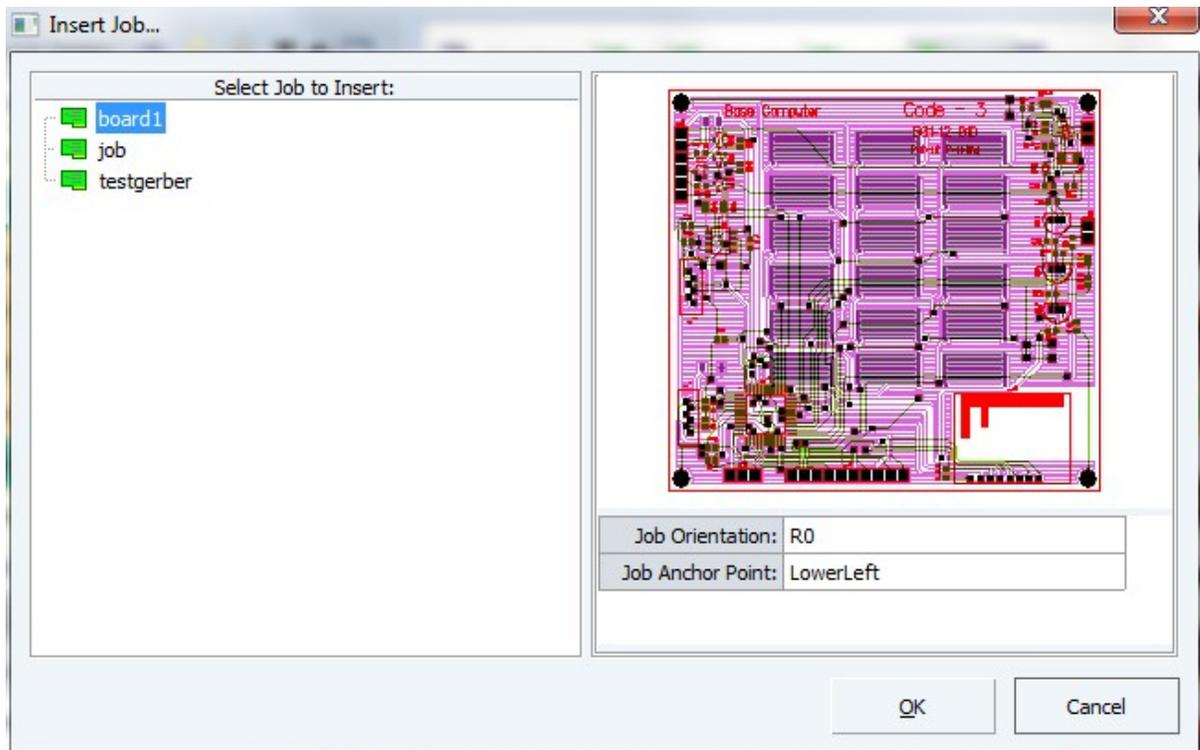
 **Note:** The Drill hit will be placed on the active layer. The active layer must be defined as a type: NC\_PRIMARY or NC\_SECONDARY.

4. Select the center point for the Drill Hit.
5. Repeat Step 4.
6. To exit: Right Click or press 'Esc' .
7. If the command is incorrect, you may [Undo](#)<sup>[122]</sup>.

## 6.5 Job

### 6.5.1 Insert Job...

Adds an Insert into the active job. A Insert represents the inclusion of one job as a part of another.



## 6.5.2 New Job...

Creates a new job.

## 6.5.3 Delete Job...

Deletes a job from the workspace.

## 6.5.4 Flatten Job Hierarchy...

Flattens all inserts in a job.

## 6.5.5 Push-Pop Job

Allows the contents inside an insert to be viewed and edited in place, within the parent job.

### 6.5.5.1 Push Into

Allows the user to edit/analyze geometries inside a selected insert.

### 6.5.5.2 Pop Out

Closes the insert.

### 6.5.5.3 Quit PPJ

Completely exit Push/Pop Job. All changes will be ignored.

### 6.5.6 Open Selected Job...

Opens selected insert master job in a new editor window.

## 6.6 Info Menu

### 6.6.1 Query

#### 6.6.1.1 All

Display information of a selected object.

 Note: Depending on the object selected, Query reports different types of information.

#### Command Sequence

1. Select menu: Info | Query
2. Select Object to Query. Move cursor over the object, and Left-click.

 Note: 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 Fab3000 will cycle through the other objects in the same location until you see the one you want.

3. The Selected Object information will be displayed in the dynamic window on the screen.
4. To exit: Right Click or press 'Esc' .

#### 6.6.1.2 Dcode

Display Dcode information of a selected object.

 Note: Depending on the object selected, Query Dcode reports different types of information.

#### Command Sequence

1. Select menu: Info | Query Dcode
2. Select Object to Query Dcode. Move cursor over the object, and Left-click.

 Note: 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 Fab3000 will cycle through the other objects in the same location until you see the one you want.

3. The Selected Object information will be displayed in the dynamic window on the screen.

4. To exit: Right Click or press 'Esc' .

### 6.6.1.3 Net

Display Net information of a selected object.

 **Note:** Depending on the object selected, Query Dcode reports different types of information.

#### Command Sequence

1. Select menu: Info | Query Net
2. Select Object to Query Net. Move cursor over the object, and Left-click.

 **Note:** 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 Fab3000 will cycle through the other objects in the same location until you see the one you want.

3. The Selected Object information will be displayed in the dynamic window on the screen.

 **Note:** All net objects on visible layers will be highlighted.

4. To exit: Right Click or press 'Esc' .

## 6.6.2 Find

### 6.6.2.1 Net

Search active job for defined Net.

#### Command Sequence

1. Select menu: Info | Find | Net. The Find Net dialog box is displayed.
2. Select Net to find.

 **Note:** The net combo box contains all available nets in the current job.

3. Select "View As". Highlight or Isolate.

 **Note:**

Highlight - All net objects on visible layers will be highlighted.

Isolate - All net objects on visible, while all other objects are hidden. (Excellent for viewing entire nets).

4. Select "Zoom Extents".

 Note: If checked, Zooms out to view the entire net.

5. To exit: Right Click or press 'Esc' .

#### 6.6.2.2 Dcode

Search active job for defined Dcode.

#### Command Sequence

1. Select menu: Info | Find | Dcode. The Find Dcode dialog box is displayed.

2. Select Dcode to find.

 Note: The dcode combo box contains all available dcodes in the current job.

3. Select "View As". Highlight or Isolate.

 Note:

Highlight - All dcode objects on visible layers will be highlighted.

Isolate - All dcode objects on visible, while all other objects are hidden. (Excellent for viewing all dcodes).

4. Select "Zoom Extents".

 Note: If checked, Zooms out to view all dcodes

5. To exit: Right Click or press 'Esc' .

#### 6.6.2.3 NC Tool

Search active job for defined NC Tool.

#### Command Sequence

1. Select menu: Info | Find | NC Tool. The Find NC Tool dialog box is displayed.

2. Select NC Tool to find.

 Note: The nc tool combo box contains all available nc tools in the current job.

3. Select "View As". Highlight or Isolate.

 Note:

Highlight - All nc tool objects on visible layers will be highlighted.

Isolate - All nc tool objects on visible, while all other objects are hidden. (Excellent for viewing all nc tools).

4. Select "Zoom Extents".

 Note: If checked, Zooms out to view the all nc tools.

5. To exit: Right Click or press 'Esc' .

#### 6.6.2.4 Polarity

Search active job for defined Composite Level.

 Note: This is an excellent command to view the make-up of a composite.

 Note: For more information about composites, goto Learn Guide: [Composites 101](#) 

### Command Sequence

1. Select menu: Info | Find | Composite Level. The Find Composite Level dialog box is displayed.

2. Select Composite Level to find.

3. Select "View As". Highlight or Isolate.

 Note:

Highlight - All composite level objects on visible layers will be highlighted.

Isolate - All composite level objects on visible, while all other objects are hidden. (Excellent for viewing entire nets).

4. Select "Zoom Extents".

 Note: If checked, Zooms out to view all composite level objects.

5. To exit: Right Click or press 'Esc' .

#### 6.6.2.5 Find Text

Search active job for defined Text.

#### 6.6.2.6 Find Insert

Search active job for defined Insert.

#### 6.6.2.7 Find Component

Search active job for defined Component.

#### 6.6.2.8 DFM Errors

Search active job for existing DFM/DRC Violations.

 Note: This command requires that you have already run a DFM/DRC command

 Note: This is an excellent command to review errors & warnings found during a DFM/DRC command.

 Note: For more information about DFM/DRC Violations goto Learn Guide: [DFM Checker](#)<sup>50</sup>

 Note: The DRC/DFM Violation Browser requires FAB 3000 Professional (not available with FAB 3000 free DFM)

## Command Sequence

1. Select menu: Info | Find | DFM/DRC Violations. The DFM/DRC error browser dialog is displayed.
2. Select Right or Left Arrow to view each error.
3. To exit: Click red button 'X', at the top right side of the browser dialog box.

### 6.6.2.9 Netlist Comparison Errors

Search active job for existing Netlist Comparison Errors.

## 6.6.3 Measure

### 6.6.3.1 Point-Point

Provides an interactive readout between two selected points within the editor.

## Command Sequence

1. Select menu: Info | Measure | Point-Point
2. Select first point of measurement.
3. Select second point of measurement.
4. The distance between them will be calculated and the final results will be displayed in the dynamic window.
5. To exit: Right Click or press 'Esc' .

---

Measure provides the following results:

Distance:	The total distance between the two selected points.
DX:	The horizontal displacement between the two selected points.
DY:	The vertical displacement between the two selected points.
Angle:	The angle (in degrees) between the two selected points.

 Note: After you have selected the first point of measurement, an interactive measurement is continuously calculated and displayed in the dynamic window. This provides an excellent way to measure multiple distances using the same base reference point.

 Note: Use Snap for precise measurements between Objects and grid points.

### 6.6.3.2 Object-Object

Provides a readout of the minimum distance between two Objects, taking into account their true width.

#### Command Sequence

1. Select menu: Info | Measure | Object-Object
2. Select first object of measurement.
3. Select second object of measurement.
4. The distance between them will be calculated and the final results will be displayed in the dynamic window.
5. To exit: Right Click or press 'Esc' .

---

Measure provides the following results:

Distance:       The total distance between the two selected points.  
DX:             The horizontal displacement between the two selected points.  
DY:             The vertical displacement between the two selected points.  
Angle:          The angle (in degrees) between the two selected points.

 **Note:** This is an excellent command for determining the minimum spacing between two Objects to determine quickly if there are any spacing violations.

### 6.6.3.3 Net-Net

Determine closest adjacent distance between two nets.

### 6.6.4 Rulers

Add and Clear Rulers.

### 6.6.5 Reports

Generate detailed reports for the active design, including: Apertures, NC Tools, Dcode, Drill, Composites, Nets, etc. These reports are presented in a table file, and displayed in the editor.

## 6.6.6 Generate PDF Error Report

Generate detailed reports with Graphics in PDF format.

### 6.6.6.1 DRC / DFM Violations

Generates a detailed DRC violation report with graphics in PDF format.

 **Note: The DRC/DFM Violation PDF Report requires FAB 3000 Professional (not available with FAB 3000 Free Dfm)**

### 6.6.6.2 Compare Net Errors

Generates a detailed Netlist Comparison Error report with graphics in PDF format.

### 6.6.6.3 Join Errors

Generates a detailed Join error report with graphics in PDF format.

### 6.6.6.4 Sales RFQ

Generates a detailed PCB RFQ report with graphics in PDF format - Simplifies the process of getting a quotation.

### 6.6.6.5 Fabrication Drawing

Generates a detailed PCB Fabrication drawing in PDF format.

## 6.6.7 Generate Text Error Report

Generates a detailed Error report in Text/Ascii format.

## 6.6.8 Generate CSV Error Report

Generates a detailed Error report in CSV format which may be imported into various spreadsheet applications .

## 6.6.9 Generate Error Drawing

Generates marker objects which may be imported into various CAD systems for concurrent verification.

### 6.6.10 List

Displays all properties of selected Objects, in the command console. The command allows the complete display of multiple Objects properties simultaneously.

 **Note:** This function requires that you select objects before proceeding. When no objects are selected this command becomes de-activated in the menu.

#### Command Sequence

1. Select Objects.

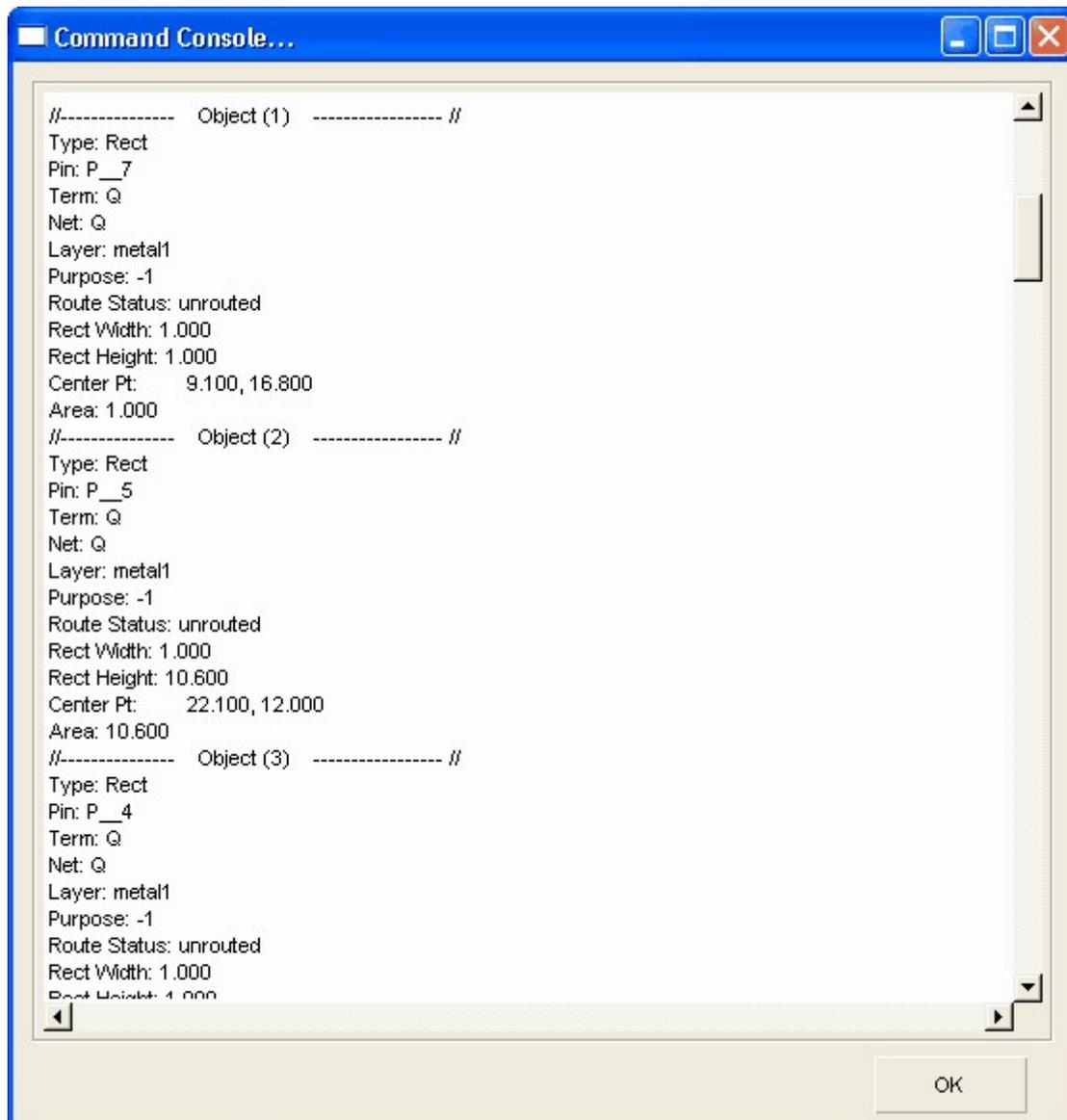
 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Info | List

3. The command console will be populated with all Object properties.

---

Command Console after List Objects



## 6.7 Fabrication

### 6.7.1 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.

#### Command Sequence

1. Select menu: Fabrication | Silkscreen Optimizer....

2. Choose the Silkscreen layer.

3. Choose the Mask layer.

 **Note:** Make sure you 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 soldermask area.

5. Verify the Min. Stub Length for the silkscreen layer.

 **Note:** This is the minimum size silkscreen object to be left behind.

6. Press OK to begin Silkscreen Optimizer.

7. A results dialog will be displayed afterwards.

## 6.7.2 Copper Balancing

Copper Pour.



 **Note:** The Active Layer must be displayed before Copper Balancing can be performed. You should only turn on one layer.

 Online Movie: [Using FAB 3000 to add copper with clearances](#)

### Command Sequence

1. Select menu: 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.

 **Note:** Make sure you select the appropriate Silkscreen & Mask layers. A common mistake occurs when a user selects a Top Silkscreen against a Bottom Mask.

4. Press OK, and select the boundary to use for Copper Balancing. All copper will be placed on the active layer.

## 6.7.3 Grow / Shrink

Enlarge or reduce object sizes. This feature is excellent for adding compensation to geometries that will be affected during manufacturing.



### Command Sequence

1. Select the objects that you want to apply Etch Compensation.

 Note: This function requires that you select objects before proceeding.

 Note: For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Fabrication | Etch Compensation...
3. Verify the compensation size (per side), and press OK.
4. Fab3000 will now apply etch compensation to all selected objects.

 Note: For Traces & Flashes, Fab3000 will replace the existing dcode with a larger or reduced dcode size. For polygons, Fab3000 will offset the vertices.

#### 6.7.4 Pin Hole Elimination

Increase yields by eliminating small openings and conductor-islands on metal layers (non-negative).

##### Command Sequence

1. Select menu: Fabrication | Pin Hole Elimination....
2. Enter the Maximum hole size (in mils) to fix.
3. Press OK, and all pin holes .

 Note: Negative Plane layers are not processed.

#### 6.7.5 Teardrop Creation

Create Teardrop / Snowman to improve Pad-Trace connections.

 Online Movie: [Simple Teardrop Creation using FAB3000](#)

##### Command Sequence

1. Select Objects (any selected pad will be used for teardrop creation).
2. Select menu: Fabrication | Teardrop Creation....
3. Choose the Type: Teardrop or Snowman  
 Note: Teardrop is the standard teardrop creation, whereas Snowman adds a small flash in the appropriate location.
4. Enter Teardrop Offset Multiple.  
 Note: This controls the length of the teardrop.
5. Press OK, and the teardrops will be created.

## 6.7.6 Remove Pads (Isolated / Stacked)

Remove Pads (Isolated/Stacked).



## 6.7.7 Remove Covered Data

Remove Covered or Overlapping Data.



## 6.7.8 Drill

### 6.7.8.1 Gerber to Drill

Convert Gerber flashed pads to Drill hits.



### 6.7.8.2 Drill to Gerber

Convert drill hits to Gerber flashes.



## 6.7.9 SMT Stencil

### 6.7.9.1 Homebase Pad Conversion...

Convert flashed pads to homebase pads.



Online Movie: [Using Homebase Pads to create groups and then modify the Custom Aperture](#)

### 6.7.9.2 C-Pad Conversion

Convert flashed pads to C-Pads.



### 6.7.9.3 Split Pads

Convert flashed pads to Split Pads.



### 6.7.9.4 Scale Pads

Convert flashed pads to Scaled Pads.



### 6.7.9.5 Bowtie Pads

Convert flashed pads to Bowtie Pads.



### 6.7.9.6 Pad Pitch Adjustment

Adjust pitch between flashed pads.



### 6.7.9.7 Heel and Toe Edits

Adjust flashed pads lengths.

 Note: Requires rectangular pads with orthogonal rotations (0, 90, 180, 270). Irregular pad rotations such as 34.2 degrees will result in undesired results.

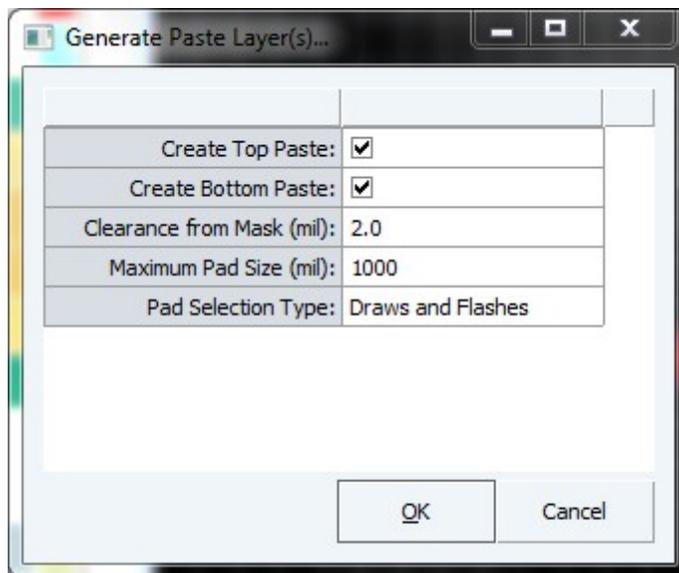
### 6.7.9.8 Add Epoxy Bars...

Add Epoxy bars between selected pads.

## 6.7.10 Generate Missing Layers

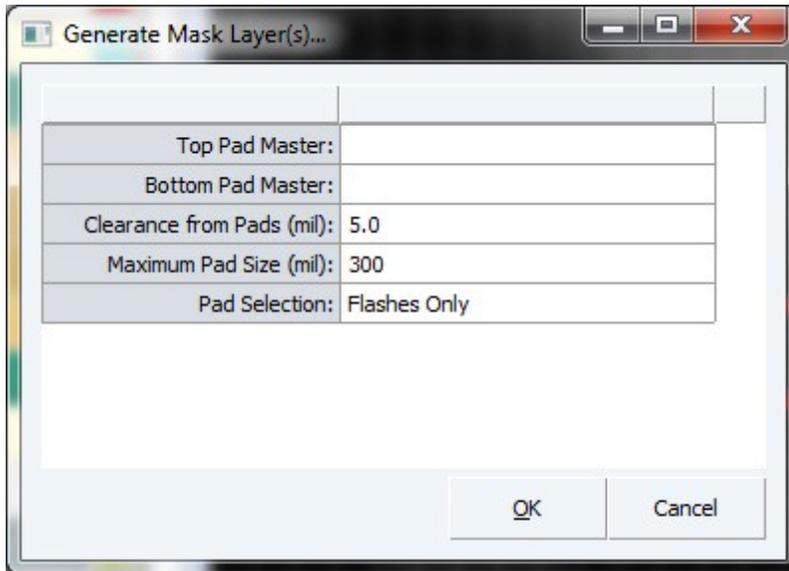
### 6.7.10.1 Paste

Create Paste layers from existing mask & drill layers.



### 6.7.10.2 Mask

Create Mask layers from existing Top and Bottom layers.

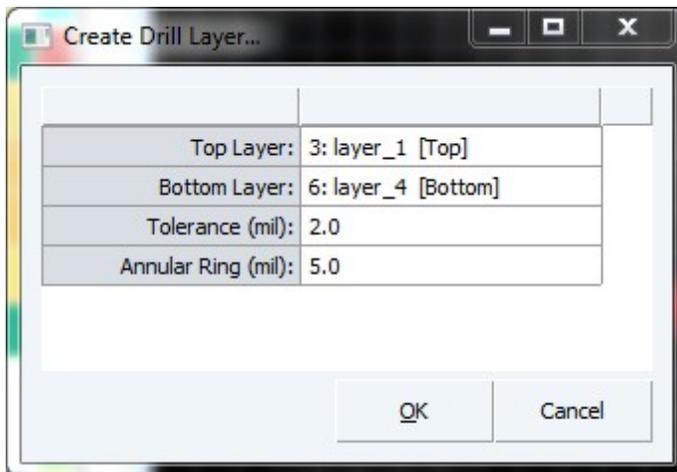


### 6.7.10.3 Border

Create PCB Border layer from existing data.

### 6.7.10.4 Create Drill Layer (PTH)...

Builds a Plated-Thru-Hole drill template when no drill layer is available.



### 6.7.11 Copper Area Calculation

Calculate copper area per layer.

### 6.7.12 Create Fabrication Drawing

Creates a Fabrication drawing from a drill layer and board outline layer.

 **Note:** All Layer Types must be defined. To define layer types see [Layers Table...](#)<sup>[67]</sup> and define all Layer Types before proceeding.

 Note: At least one Drill Layer must be defined. Please [import a drill file](#)<sup>[87]</sup> or [create a Drill Layer](#)<sup>[174]</sup> before proceeding.

 Note: A Border Layer must be defined. Please either import a file containing a PCB border or [create a Border Layer](#)<sup>[174]</sup> before proceeding..

## Command Sequence

1. Goto menu: Fabrication | Create Fabrication Drawing...

2. Select Drill Layer

 Note: If your design has more than one drill layer (for example: Blind/Buried Via's), repeat Create Fabrication Drawing and select the other drill file(s). Each time this command is run a new Fabrication Drawing is created.

3. Verify Units.

4. Enter PCB details such as Part Number, Revision, Notes, etc.

5. Click 'Next' when done.

6. Review and verify the hole chart description.

 Note: You may change the symbols used for each drill hole and edit the text.

7. Click 'Next' when done.

The Fabrication Drawing will be created.

## 6.8 Assembly

### 6.8.1 Build Component

Add Components to the active Job.

Review Learn Guide: [How to Generate a Centroid File](#)<sup>[60]</sup>

 Online Movie: [Generate X/Y Centroid file from Gerber, using FAB 3000](#)

## Command Sequence

1. Turn-off all layers except for Top Paste & Top Silkscreen.

2. Locate a footprint, and select the entire component.

 Note: This function requires that you select objects before proceeding.

 Note: For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

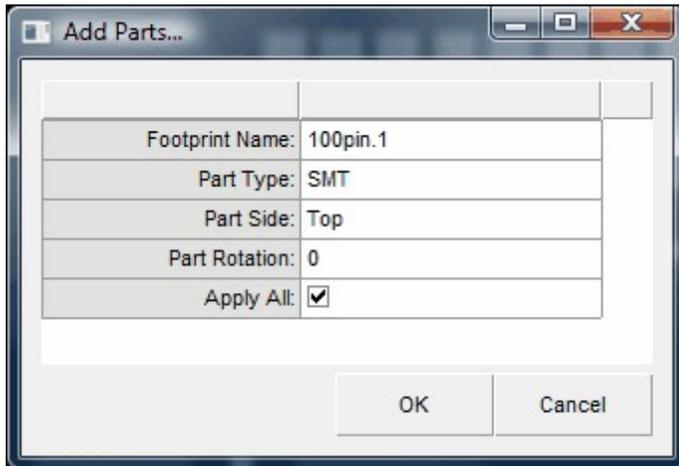
 Note: Add Components requires flashed footprints and will ignore anything else, so it's fine to select a window around the entire Component (any selected Silkscreen objects would be ignored).

3. Goto menu: Tools | Components | Add Components...

#### 4. Select Pin One

 **Note:** Usually Pin One is the Top-Leftmost pin for Top side (assuming no rotation), and Top-Rightmost for Bottom side (assuming no rotation).

#### 5. Assign Component Description, and press OK when done.



 **Note:** 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.

 **Note:** If Apply All is checked (recommended), Fab3000 will go out and seek all similar footprints.

 **Note:** 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.

 **Note:** You can also add Components individually by using function: [Insert Single Component](#)<sup>176</sup>

Now its time for the Bottom Side (if required)

#### 7. Turn off all layers except Bottom Paste & Bottom Silkscreen, and repeat steps 2-6.

### 6.8.2 Insert Component

Inserts a single Component into an existing Job

### 6.8.3 Find Component

Searches the job for defined Component.

### 6.8.4 Component Detection Wizard

Detect Components automatically. Speeds up the process of building Components.

### 6.8.5 Component Analysis

Detect Components automatically. Speeds up the process of building Components.

## 6.8.6 View Component Violations

Searches the job for defined Component Violations.

## 6.8.7 Apply External Centroid

Apply External Centroid attributes to existing components.

## 6.8.8 Components Table

Display Components used in active job/panel.

## 6.8.9 BOM Table

Display BOM for active job/panel.

## 6.9 Tools

### 6.9.1 DFM

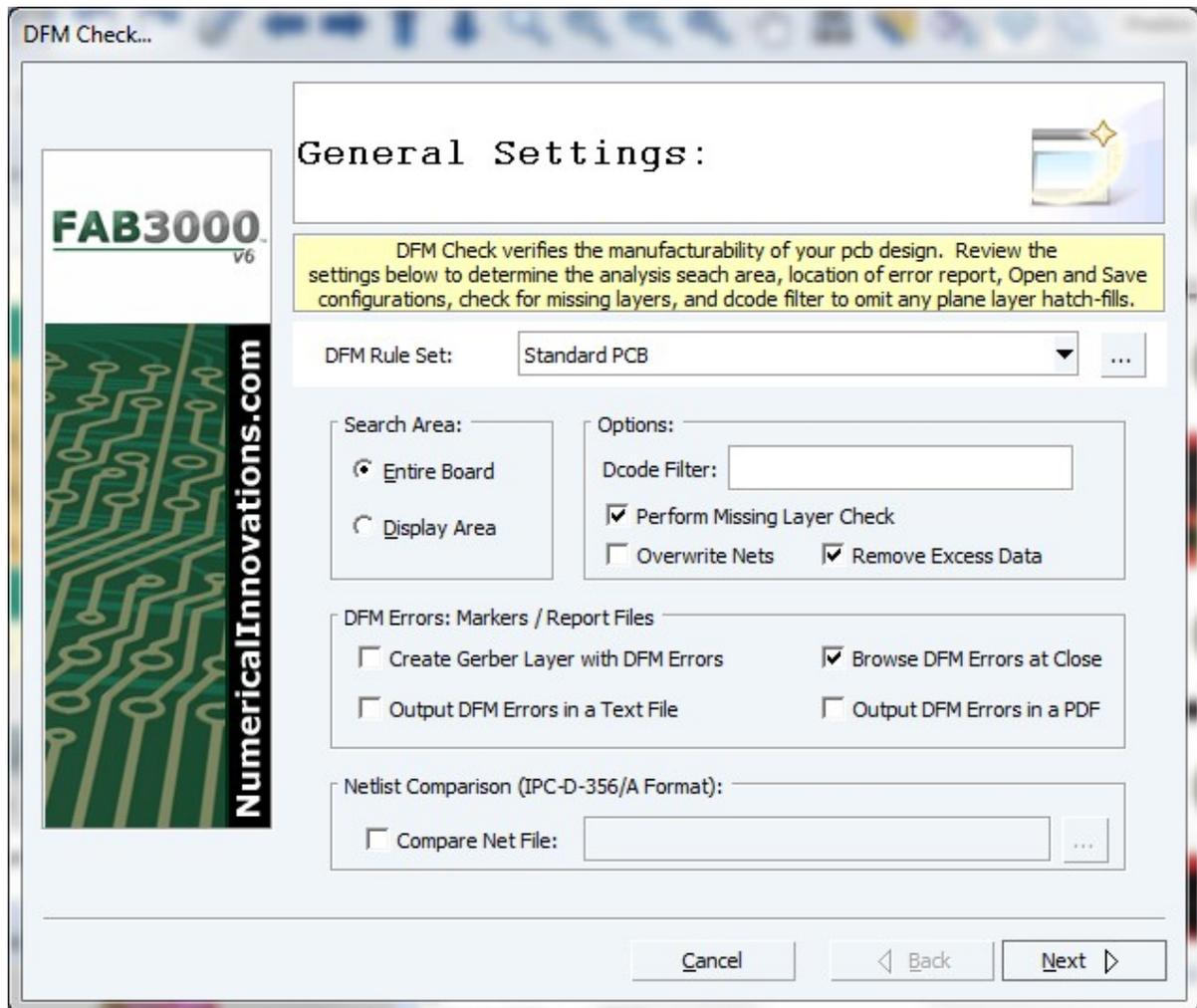
#### 6.9.1.1 DFM Check (PRO)

Find all DRC/DFM violations. Before using this feature, you must have imported gerber file(s) and drill file(s) into the workspace (see [AutoLoad](#)<sup>[79]</sup> for quick loading).

 Note: DFM Checker requires that you extract a netlist before proceeding. If you haven't run netlist extraction, DFM Checker will perform it for you.

 Note: DFM Checker requires that all layer types are defined. To define layer types, goto menu: Setup | Layers Table...

 Note: DFM Checker requires that you have at least 1 drill file (2 or more for blind/buried vias).



## Command Sequence

1. Select menu: Tools | DFM | DFM Check (PRO)...

2. Verify the DFM Rules Set.

 **Note:** Available Rules can be accessed from the pull-down, or you may edit the rules by pressing button "..."

3. Verify 'General Settings', and press button Next.

4. Fab3000 will now perform an extensive analysis of your pcb design.

 **Note:** Depending on the complexity of the design and the checks performed, Fab3000 processing time can take anywhere from 30 seconds to 5 minutes.

 **Note:** 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 sperate editor window, and then proceed. To open Jobs, goto menu: Windows | Workspace Browser...

5. Press button 'Finish' when done.

6. If any errors have been detected, Fab3000 will display the [DRC/DFM Violation Browser...](#)<sup>[18]</sup>

 Note: The DRC/DFM Violation Browser is an easy-to-use navigation tool used to locate and describe each detected error.

#### 6.9.1.2 DFM Check (Quick PDF)

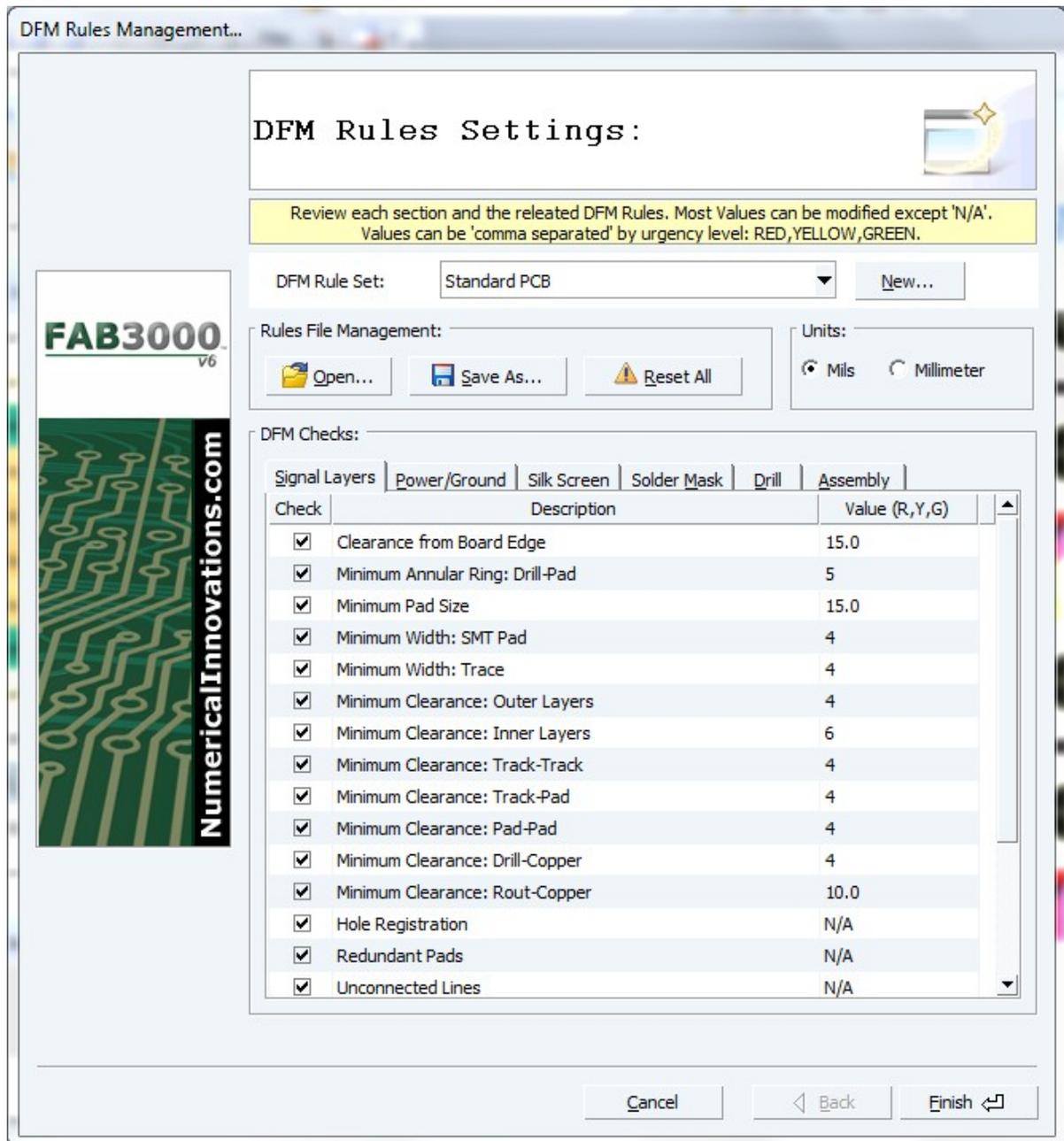
Find all DRC/DFM violations, using the current DFM Rules Set, and output to a PDF. Before using this feature, you must have imported gerber file(s) and drill file(s) into the workspace (see [AutoLoad](#)<sup>[79]</sup> for quick loading).

#### 6.9.1.3 DFM Rules Management

Manage DFM Rules.

 Note: Uncheck any DFM Rule to disable it.

 Note: Units can be toggled between Mils (English) and Millimeter (metric).



 You can assign your required DFM Rules (menu: Tools | DFM | DFM Rules...) and save to a file. All DFM Rule files are actually stored as external files, and located in the local FAB 3000 DFM folder (example: C:\Numerical Innovations\FAB3000v6\DFM ). To share with colleagues, simply give them a copy of the DFM Rules FILE (\*.DRF) and either have them place that DRF file into their DFM folder or have them start FAB 3000 and go to the DFM Rules dialog (menu: Tools | DFM | DFM Rules...), then press button "Open..." select the file and FAB 3000 will automatically place a copy in the DFM folder.

#### 6.9.1.4 Find Minimum Gaps Only

Find minimum gaps.

### 6.9.1.5 Find Minimum Widths Only

Find minimum widths.

### 6.9.1.6 Drill Checker

Find overlapping and duplicate drills.

### 6.9.1.7 Browse DFM Errors

Search active job for existing DRC / DFM Violations.



Note: This command requires that you have already run a DRC\DFM command



Note: This is an excellent command to review errors & warnings found during a DRC\DFM command.



Note: For more information about DRC\DFM Violations goto Learn Guide: [DFM Checker](#)

## Command Sequence

1. Select menu: Tools | DRC/DFM | View DRC/DFM Violations....
2. The DFM/DRC error browser dialog is displayed.
3. Select Right or Left Arrow to view each error.
4. To exit: Click red button 'X', at the top right side of the browser dialog box.



Note: You can "edit" the text inside the yellow message box while browsing errors. Enabling you to add any special notes or comments, that can be passed down to other engineers.

## 6.9.2 Nets

Enter topic text here.

### 6.9.2.1 Netlist Extraction

Extract a true inter connectivity map of any PCB.

### 6.9.2.2 Compare Nets...

Compare extracted nets against an imported netlist.

### 6.9.2.3 Find Net Shorts

Locate shorts in a netlist.

#### 6.9.2.4 View Net Errors...

Search active job for existing Compare Net Errors.

 Note: This command requires that you have already run [Compare Net](#)<sup>[181]</sup>.

 Note: This is an excellent command to review errors found by the [Compare Net](#)<sup>[181]</sup>.

 Note: For more information about Netlist Analyzer goto Learn Guide: [CAD Netlist Verification](#)<sup>[54]</sup>

### Command Sequence

1. Select menu: Tools | Nets | Compare Net Errors. The Compare Net Errors browser dialog is displayed.

2. Select Right or Left Arrow to view each error.

3. To exit: Click red button 'X', at the top right side of the browser dialog box.

 Note: You can "edit" the text inside the yellow message box while browsing errors. Enabling you to add any special notes or comments, that can be passed down to other engineers.

#### 6.9.2.5 Apply Nets

Apply External net names to CAM nets.

#### 6.9.2.6 Netlist Table

Create, delete, and modify nets.

### 6.9.3 Compare Jobs and Layers

#### 6.9.3.1 Compare Layers (Graphical FAST)

Compare differences between layers.

#### 6.9.3.2 Compare Jobs (Graphical FAST)

Overlay another job or merge over the active design.



Online Movie: [Graphical comparison between two similar jobs to quickly find differences - Using FAB 3000](#)

#### 6.9.3.3 Compare Layers (Physical)

Perform physical comparison differences between layers.

#### 6.9.3.4 Browse Compare Errors

Search for comparison errors.

## 6.9.4 Convert to

### 6.9.4.1 Flash (Automatic)

Automatically generate Round, Rectangular flashes.

#### Command Sequence

1. Select all objects in an area that you want to have round & rectangular flashes detected. (It doesn't matter if pads are different sizes).

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Convert To | Flash (Automatic)...

3. Verify settings for Detecting New Flashes, and press OK.



4. Fab3000 will now detect all round / rectangular pads and convert them to gerber flashes.

 **Note:** 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.

### 6.9.4.2 Flash (Interactive)

Converts drawn pads to gerber flashes.

#### Command Sequence

1. Select the drawn pad(s) that you want to convert to a flash.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Convert To | Flash (Interactive)...
3. Verify settings for New Flashes, and press OK.
4. Fab3000 will now search for all objects similar to your original selection, and convert them to gerber flashes.

 **Note:** Turn off any layers that you do not want to convert draws to flashes.

 **Note:** 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.

#### 6.9.4.3 Flash (Selective)

Converts a selected group of objects to gerber Flashes.

##### Command Sequence

1. Select all the drawn pads that you want to convert to a flash. (It doesn't matter if pads are different sizes).

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Convert To | Flash (Selective)...
3. Verify settings for New Flashes, and press OK.
4. Fab3000 will now convert all selected objects to gerber flashes.

 **Note:** 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.

#### 6.9.4.4 Flash (Custom Aperture)

Converts selected objects to a custom aperture definition.

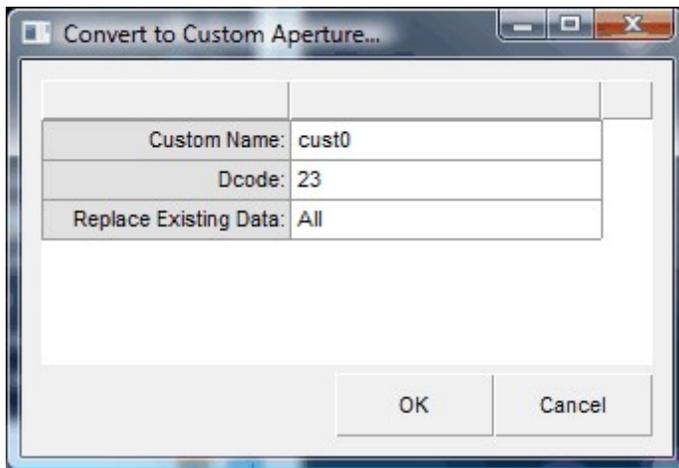
##### Command Sequence

1. Select all the objects that you want to convert to a customer aperture.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Convert To | Flash (Custom Aperture)
3. Enter in the name you want to use as reference for this custom aperture. You can also determine if you want FAB 3000 to replace the selected objects with the new customer aperture; or by selecting "All" FAB 3000 will search for all similar object groups and replace them with custom apertures.



4. Fab3000 will now create a new custom aperture, and adding it to the Aperture Table.

 Note: Nothing has changed in the active job. Fab3000 has simply constructed a new custom aperture based from the selected objects

 Note: The new custom aperture is now current, and may be added, by using the command: Add | Flash

 Note: Fab3000 is very intelligent and will recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

#### 6.9.4.5 Polygon

Converts selected objects to polygons.

#### Command Sequence

1. Select all the objects that you want to convert to a raster polygon.

 Note: This function requires that you select objects before proceeding.

 Note: For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Convert To | Polygon...

3. Fab3000 will now calculate all boundaries, and construct clean raster polygons.

 Note: This is an excellent for optimizing large amounts of drawn data (like a plane layer).

 Note: Fab3000 is very intelligent and will recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

#### 6.9.4.6 Rectangle

Converts selected objects to rectangles.

#### 6.9.4.7 Circle

Converts selected objects to circles.

#### 6.9.4.8 Job

Converts selected objects to a new job.

### Command Sequence

1. Select all the objects that you want to convert to a new job.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

2. Select menu: Tools | Convert To | Job

3. Enter in the name you want to use as reference for this new job.

4. Fab3000 will now create a new job.

 **Note:** Nothing has changed in the active job. Fab3000 has simply constructed a new job based from the selected objects

 **Note:** You may now access the new job from the [Workspace Browser...](#)<sup>[21]</sup>, or [Insert](#)<sup>[159]</sup> the new job into the another job (creating hierarchy).

 **Note:** Fab3000 is very intelligent and will recognize objects with a composite attribute and construct them with dark or clear fields (like a donut).

### 6.9.5 Join Wizard

#### 6.9.5.1 Join (Automatic)

Automatically fix & close selected open boundaries.

### Command Sequence

1. Select the boundary objects (ie. lines, arcs, traces) you want to join.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

2. Select menu: Tools | Utilities | Join...

3. Verify settings for Join and press OK.

4. Fab3000 will now attempt to join all selected objects.

 **Note:** If there are any Open Boundaries that could not be joined, Fab3000 will display their locations using the command [View Join Errors](#)<sup>[187]</sup>.

### 6.9.5.2 Join (Interactive)

Selectively fix & close open boundaries. In many cases after running [Join \(Automatic\)](#)<sup>[186]</sup> you may discover that several troublesome boundaries still cannot be joined and closed, this is where Join Interactive is used.

#### Command Sequence

1. Select menu: Tools | Join Wizard | Join (Interactive)...
2. Choose the Tolerance and resulting layer for the joined objects. Then press OK.

 **Note:** The Tolerance is used when join interactive attempts to join adjacent objects.

3. Select an object (ie. lines, arcs, traces) on the boundary you want to join.

 **Note:** 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.

 **Note:** 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.

 **Note:** Join Interactive will also accept the following options:  
End - Stop and create an open boundary.  
Close - Stop and create closed polygon.  
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.

### 6.9.5.3 View Join Errors

Searches the active job, for any Join errors (ie. boundaries that could not be closed by [Join](#)<sup>[186]</sup>).

 **Note:** This function requires that you have already run [Join](#)<sup>[186]</sup> beforehand, and join error markers were detected.

#### Command Sequence

1. Select the boundary objects (ie. lines, arcs, traces) you want to join.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

2. Select menu: Tools | Utilities | Join...
3. Verify settings for Join and press OK.
4. Fab3000 will now attempt to join all selected objects.

 **Note:** If there are any Open Boundaries that could not be joined, Fab3000 will display their locations using the command [View Join Errors](#)<sup>[187]</sup>.

## 6.9.6 Polygon Utilities

### 6.9.6.1 Chop Polygons...

Split 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.

 **Note:** This function requires that you have already run [Join](#)<sup>[186]</sup> beforehand, and join error markers were detected.

#### Command Sequence

1. Select the polygon objects you want to chop.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

2. Select menu: Tools | Utilities | Chop Polygons...
3. Verify the Maximum polygon vertices value and press OK.
4. Fab3000 will now chop all selected polygons.

### 6.9.6.2 Fix Invalid Polygons

Find and correct self-intersecting polygons. These invalid polygons usually go undetected, and can cause unexpected errors & violations that result in mask failure.

### 6.9.6.3 Remove Tiny Segments...

Remove small and self-intersecting segments within a polygon that can cause problems downstream.

#### Command Sequence

1. Select the objects you want to clean.

 **Note:** This function requires that you select objects before proceeding.

 **Note:** For more information about object selection see [Viewing & Selection Basics](#)<sup>[17]</sup>

2. Select menu: Tools | Utilities | Remove Tiny Edges...

3. Verify the Minimum segment length value and press OK.
4. Fab3000 will now remove all tiny edges.

## 6.9.7 Composite Layers

### 6.9.7.1 Composite Wizard

Perform Polygon De-Embedding per layer.

### 6.9.7.2 Build...

Create a Composite from existing layers.

### 6.9.7.3 Flatten...

Convert an existing Composite into a single level layer.

## Command Sequence

1. Select the objects you want to clean.

 **Note:** This function requires that you select a composite layer (ie. a layer has objects with a composite level greater than 0)

 **Note:** For more information about object selection see [Viewing & Selection Basics](#) 

2. Select menu: Tools | Composite | Flatten...

### 6.9.7.4 Separate

Convert an existing Composite into a multiple layers. One layer per composite level.

### 6.9.7.5 Open (for Editing)

Open an existing composite layer for editing.

### 6.9.7.6 Close

Close a previously opened composite layer.

## 6.9.8 Boolean Wizard

### 6.9.8.1 Assign to Group A

Assign objects for Group A.

### 6.9.8.2 Assign to Group B

Assign objects for Group B.

**6.9.8.3 Add**

Perform boolean operation OR.

**6.9.8.4 Subtract**

Perform boolean operation Subtract.

**6.9.8.5 Intersect**

Perform boolean operation AND.

**6.9.8.6 Difference**

Perform boolean operation XOR.

**6.9.8.7 Invert**

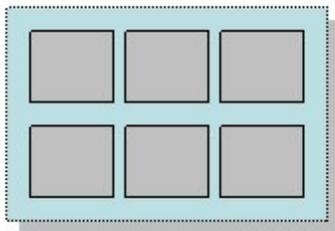
Perform boolean operation Invert.

**6.9.8.8 Size**

Perform boolean operation Size.

**6.9.9 Film Wizard**

This unique feature is guaranteed to **SAVE YOU UP TO 50% OFF** on your Photoplotting service expenses, eliminate any operator errors (due to someone else wrongly mirroring or using wrong polarity - ie. Negative/Positive), and gets faster Turn-Around times. Within 4-5 jobs Fab3000's Film Wizard will have already paid for itself.



Film Wizard will automatically setup gerber layers for optimal film output. Excellent for anyone who needs to make Gerber or Postscript photoplots from their artwork.

Here's how it works:

Say for Instance you have just finished a design which contains 8 sperate layers for photoplotting. We'll also assume that your design image area is about 5"X7" (120mm X 175mm). If you were to send this design to your photoplotting service for film, you would probably 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. Your cost using film wizard will now only be \$80 total (\$40 per larger film X 2 film layers).

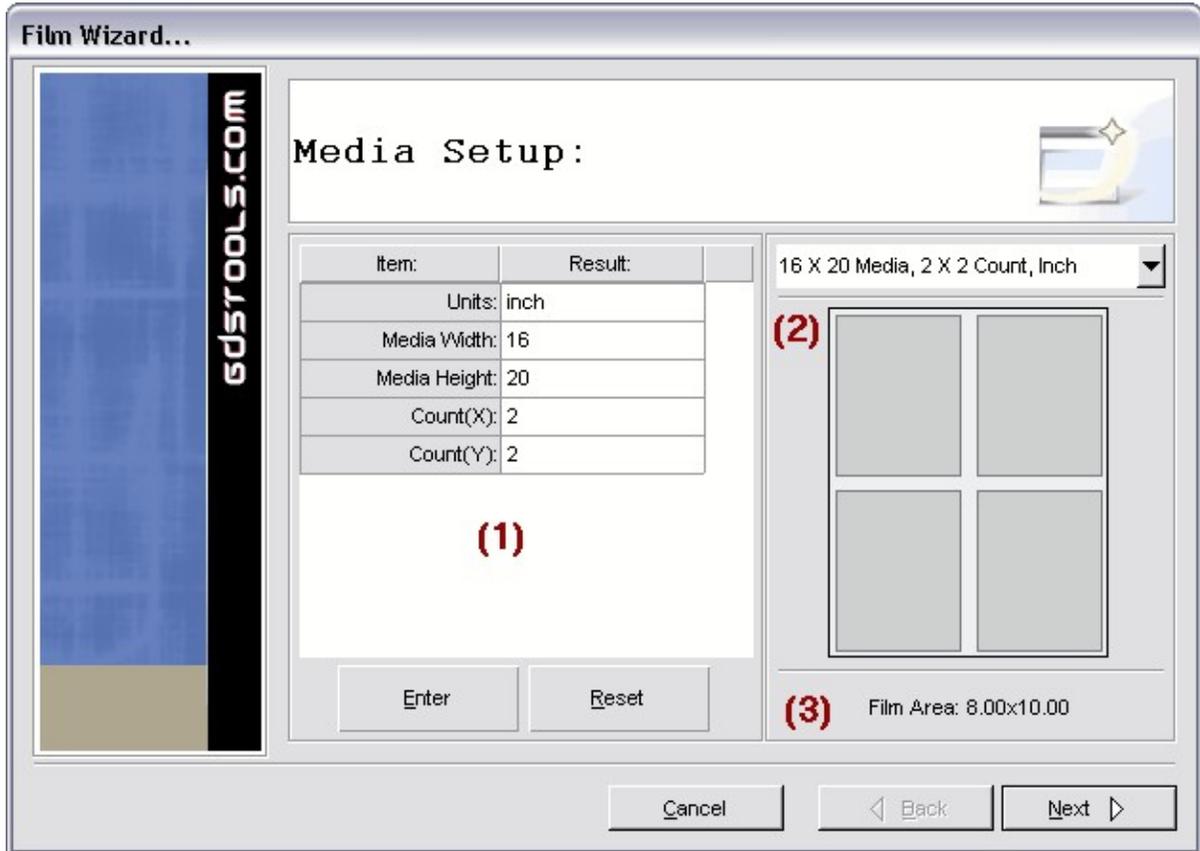
Photoplotting services take your design and edit (mirror/polarity) each layer per customer instructions,

group, & setup the individual layers onto one or more larger pieces of film. Then 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 have saved 50%, eliminated any potential operator errors, and improved your film turn-around time.

Film Wizard Demonstration:

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 contact your photoplotting service to determine what are acceptable film sizes to use for their photoplotting process)



**Assign options that you want to appear on each film layer.**

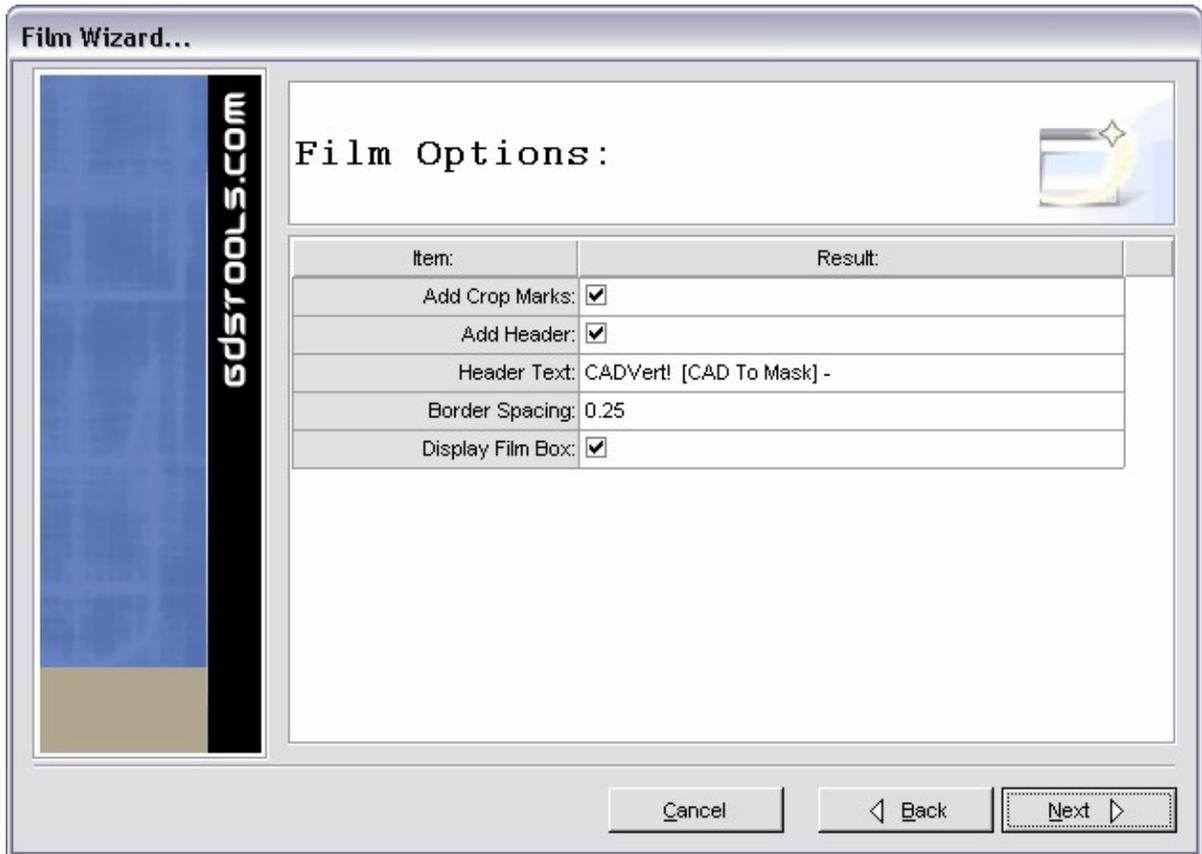
**Crop Marks** - make it easier for the photoplotting service to cut out individual films.

**Add Header** - Place separate text for each layer. Text includes the layer name, and time.

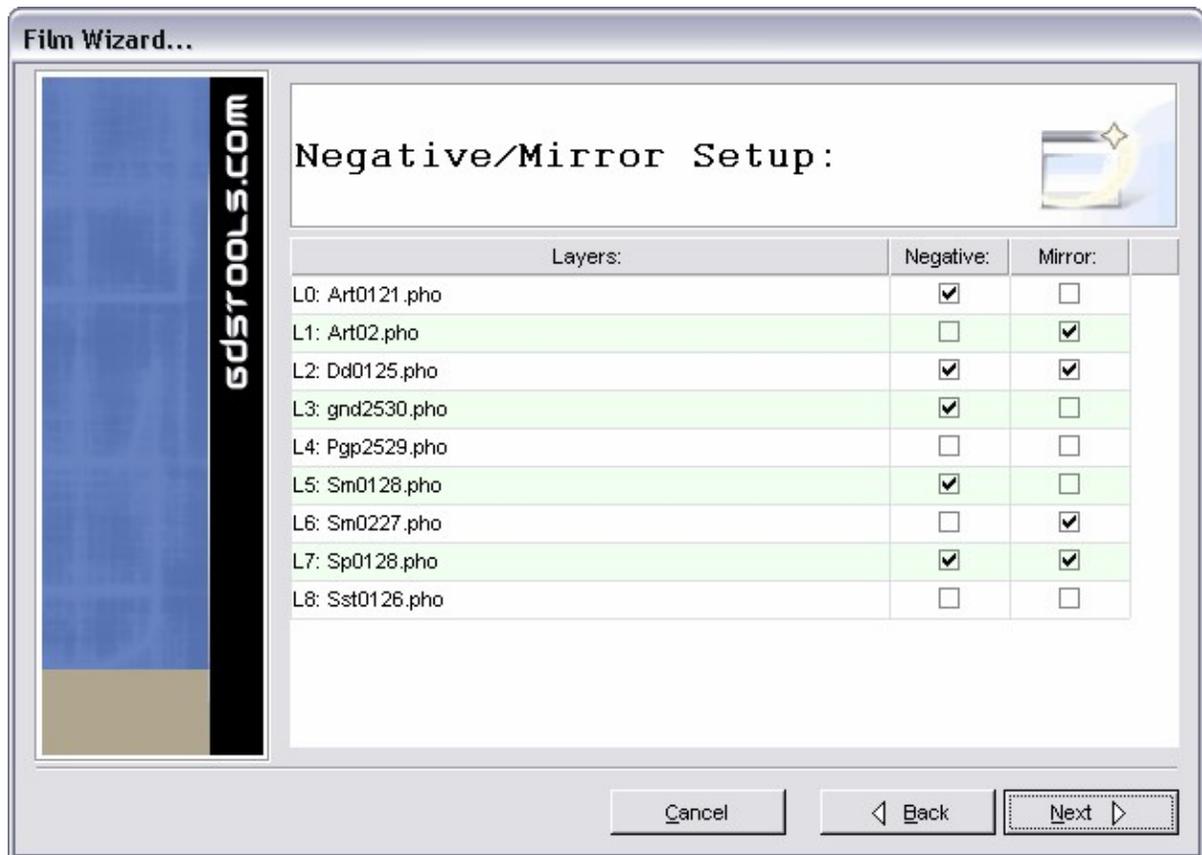
**Header Text** - Additional text to place in the header.

**Border Spacing** - Minimum spacing per each individual film.

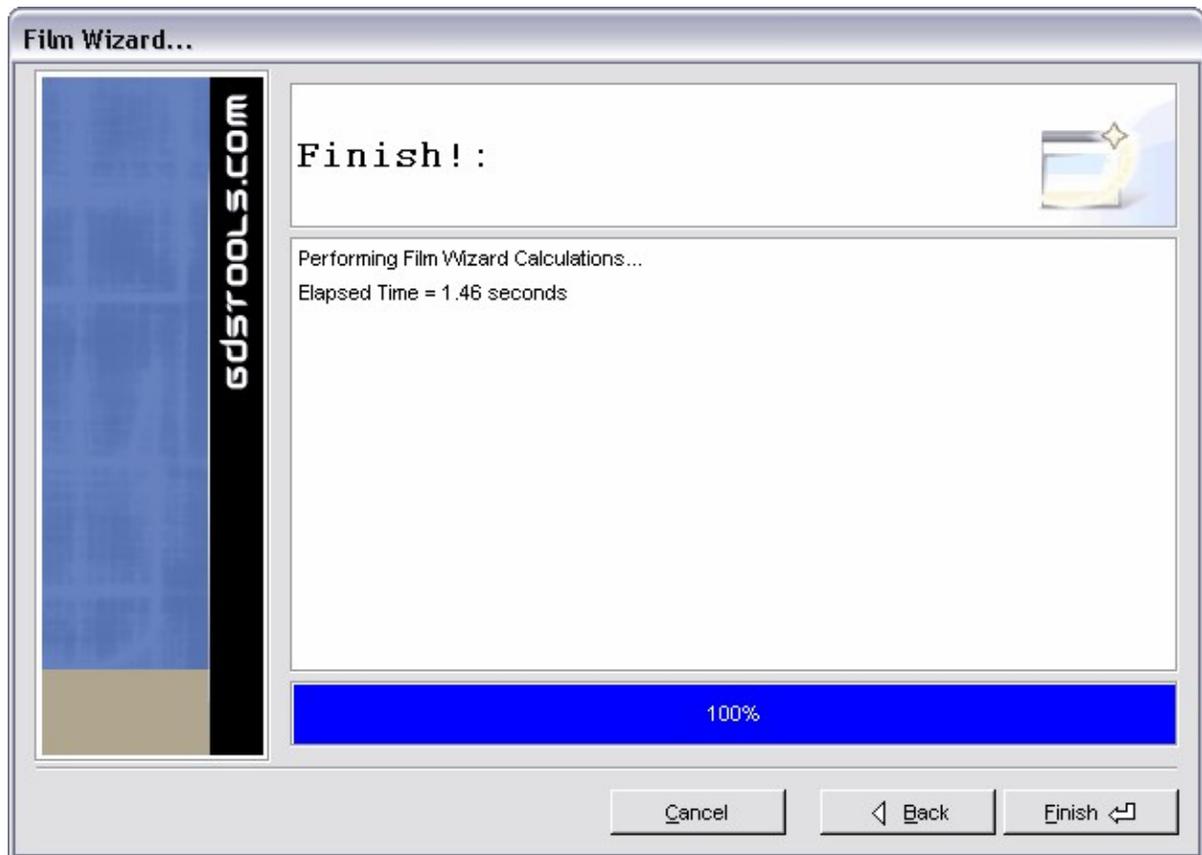
**Display Film Box** - For visual reference only.



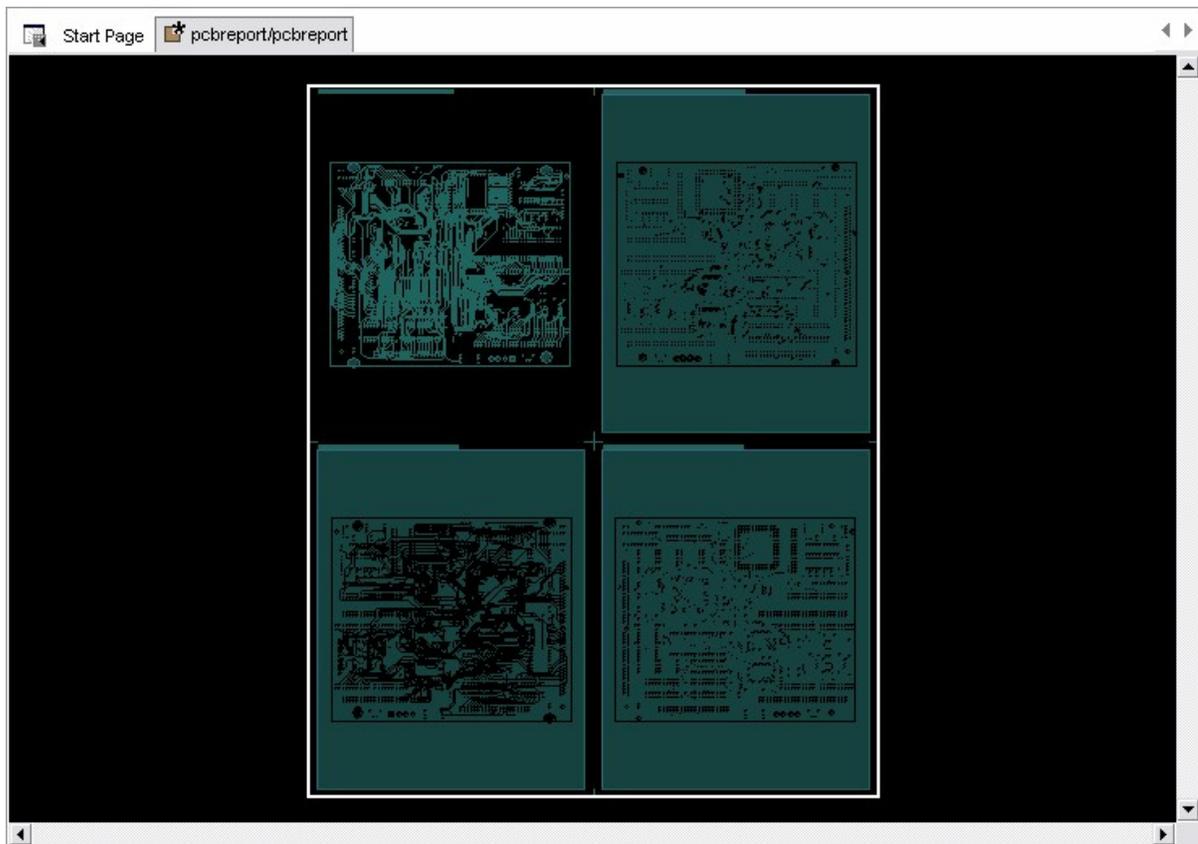
**Assign Polarity & Mirror options for each individual layer.**



Click next to start the Film Wizard...



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 export these "Film Wizard" layers to either Gerber or Postscript, and that's it!

**You have just saved 50% on your photoplotter service expenses!**

## 6.9.10 Clip Area...

Extract an area from a job for sperate analysis, etc.

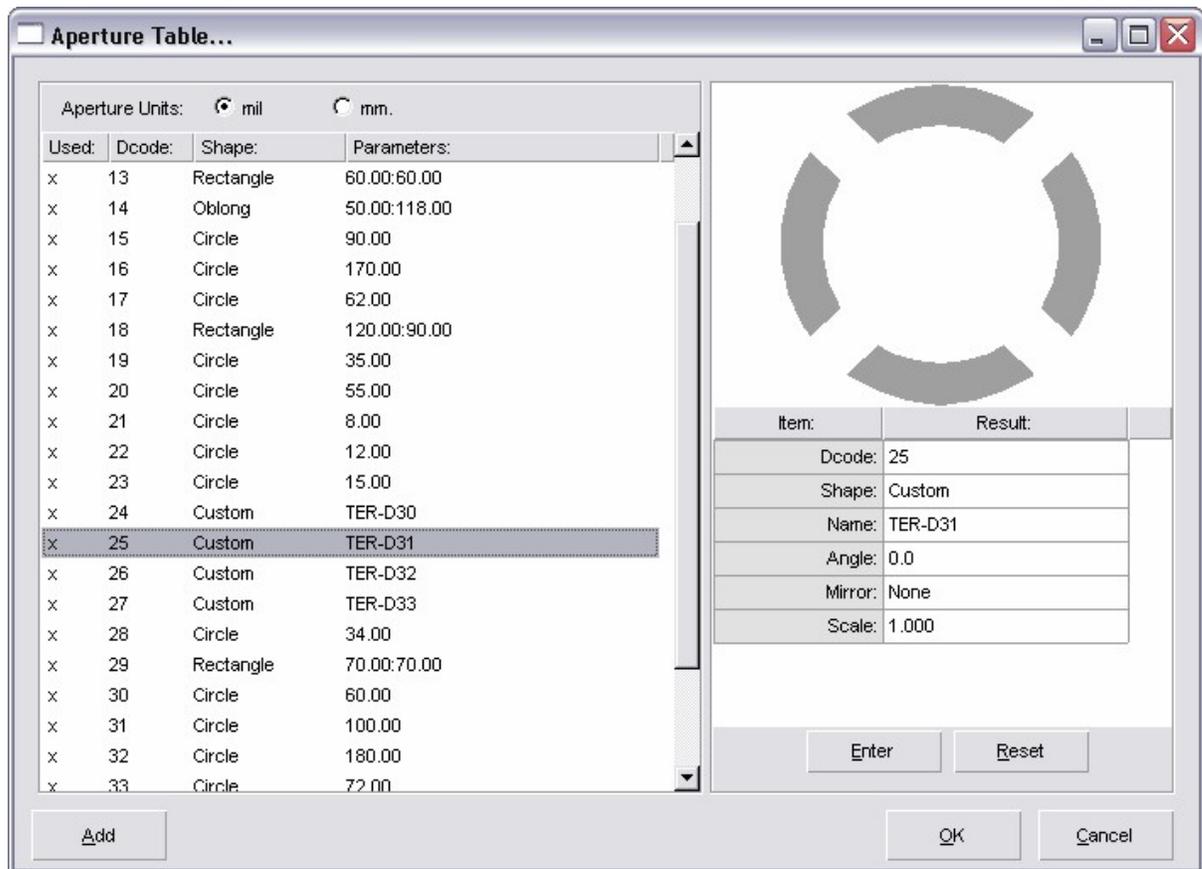
## 6.10 Setup

### 6.10.1 Aperture Table

Display & Edit all Apertures used in the workspace.

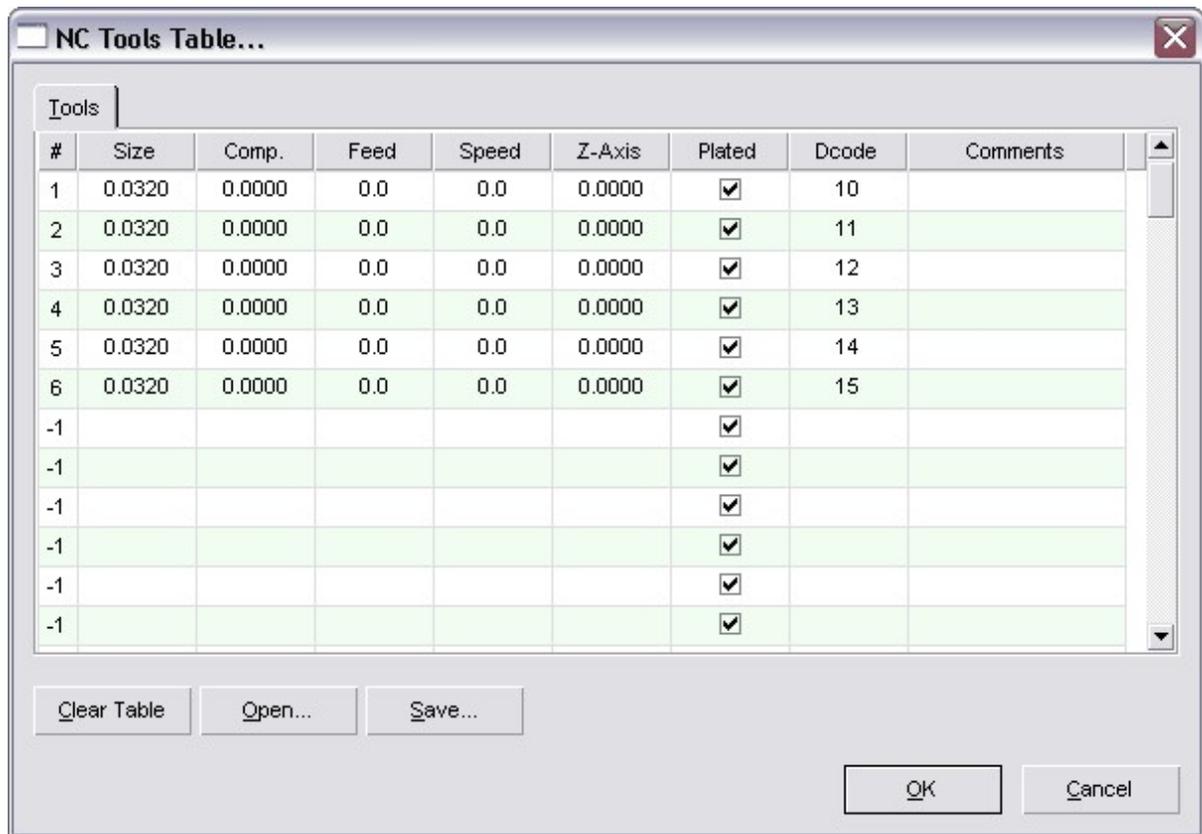
 Online Movie: [Create Aperture and then change Dcode of selected objects using FAB 3000](#)

Sample Screen Capture of Aperture Table.



## 6.10.2 NC Tools Table

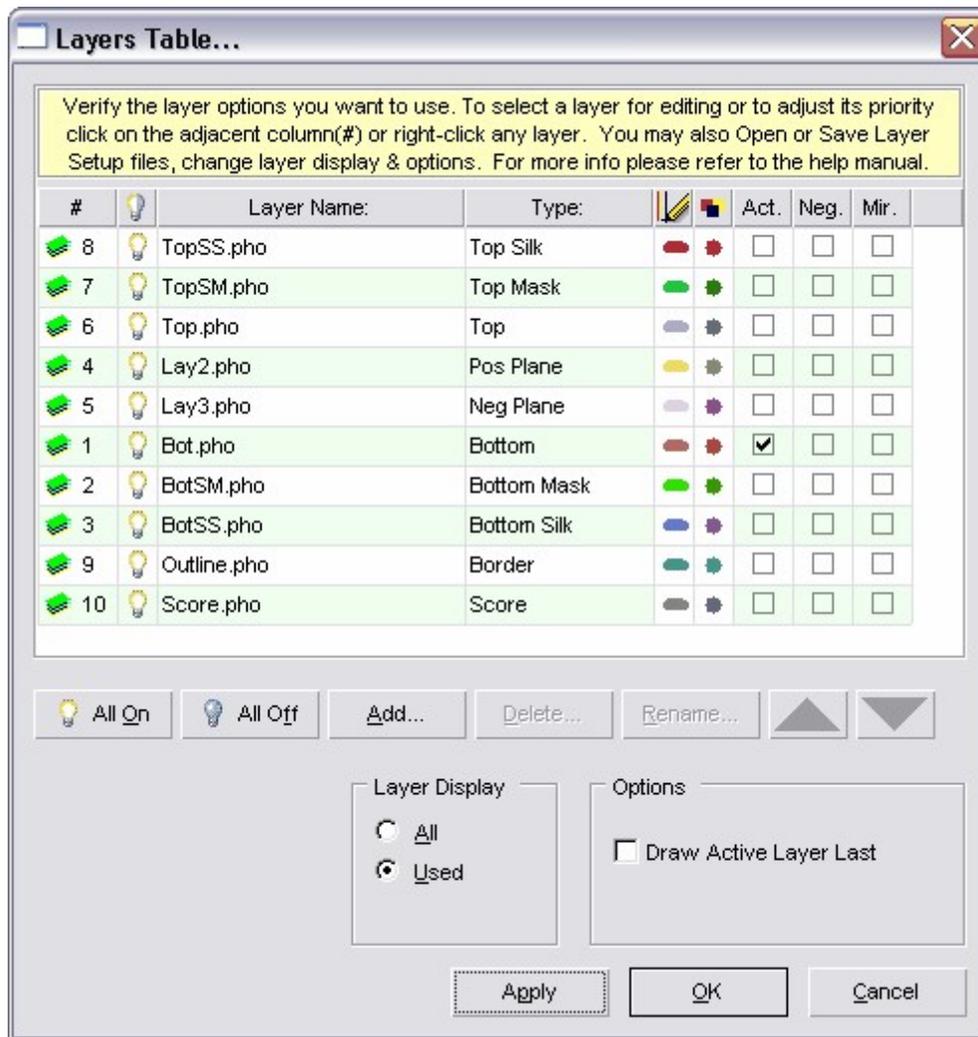
Display & Edit all NC Tools used in the workspace.



### 6.10.3 Layers Table

Displays all layer properties for the active design.

Screen capture of Layers Table



#  
Fab3000 Layer Number

Visibility (bulb icon):  
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.

Layer Type

Draw Color:  
Traces and Polygons on the associated layer will display that color.

Flash Color  
Flashes on the associated layer will display that color.

Act.

Active Layer used when adding objects (ie. Polygons, Flashes, etc)

Neg.

Negative Polarity. Assign a negative polarity on output for this layer.

Mirr.

Mirror. Assign a mirror attribute for this layer. Mirror will take effect on output.

Arrow Up:

Controls Layer Stackup. This feature allows you to control the layer stackup by moving the layer up to the top of the stack. This button remains de-active until a single layer is selected. To select a layer click on the layer icon (located on the leftmost side).

Arrow Down:

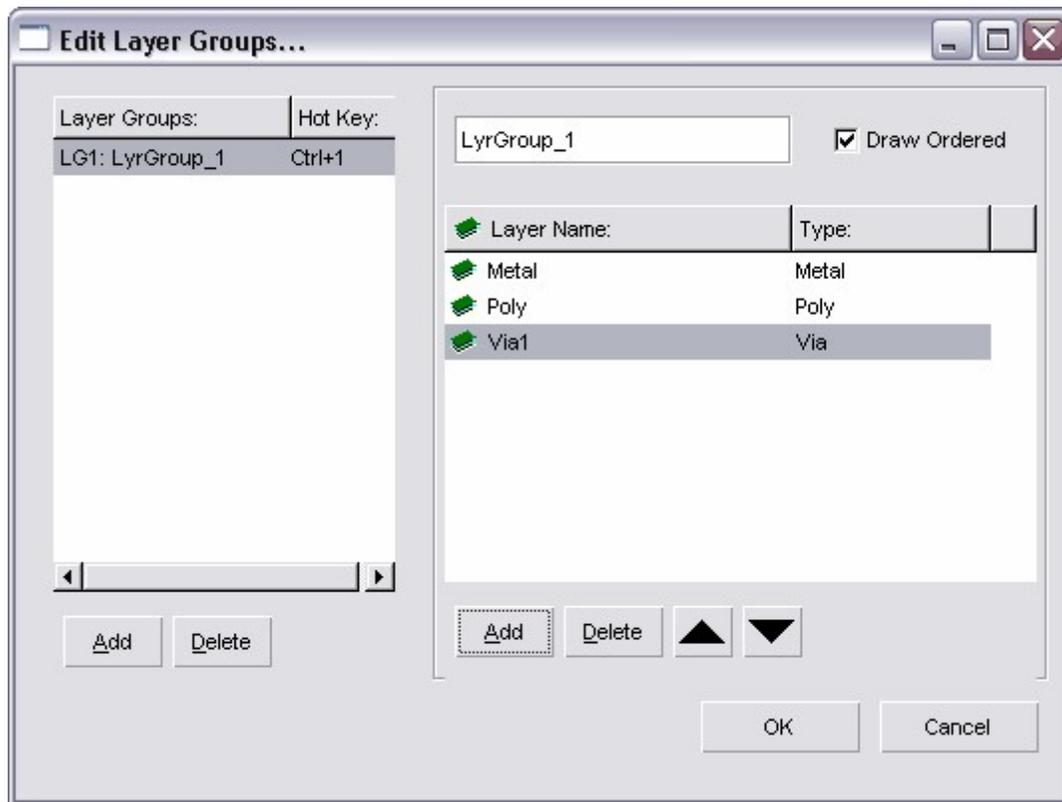
Controls Layer Stackup. This feature allows you to control the layer stackup by moving the layer down to the bottom of the stack. This button remains de-active until a single layer is selected. To select a layer click on the layer icon (located on the leftmost side).

#### 6.10.4 Layer Sets...

Group 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.



- This Guide assumes that you have already imported data into the workspace (see [AutoLoad...](#)<sup>179</sup>).
- To [Assign Layer Groups](#), goto menu item: Setup | Layer Groups...



- The Layer Groups list box shows you the names of each layer group, and what ID or number key they are assigned to. If no layer sets exist, the system displays only an empty, default LyrGroup 1. Only layer groups will appear in the list. (If layer groups have already been created the first available number appears.) The name of the highlighted layer group appears in Name text box.
- Type the desired layer group name in the Name text box, or use the default name.
- Click the 'Add' button under the Layers list box. This opens the Layer Selection box.
- 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.
- Check the option '**Draw Ordered**', to draw layers in a sequenced order from the first layer, down to the last layer of the list. To change the sequence of the layers in a group, select any layer, and use the down or up arrows to change the layer order.
- Repeat the process until you have assigned all desired layers to the group, and their sequence.
- If you wish to delete a layer group, select the layer group name in the Layer Group list, and click the '**Delete Group**' button.
- Click the OK button when you are finished. **You may turn ON any desired layer group by pressing 'Ctrl' + its associated number key** on your keyboard.

You may go back into the Layer Groups dialog box and add more user layer groups, or modify existing ones. After you have created layer groups for a particular design, that information is saved in your

workspace so that the sets are accessible for future use.

### 6.10.5 Detect Layer Types...

Display & edit all layer type detection values. During import, layer types are automatically determined by patterns found in the gerber file names.

Layer Detection Table options:

Layer Detection Table...	
Top:	art01 ,gtl,top , ,cmp ,top,layer1 ,soldcom,comp
Bottom:	art02 ,gbl,bottom , ,sld ,bot,layer2,sold
Internal:	art ,in
Neg. Plane:	pgp,pwr ,power , ,gp,gnd,ground
Pos. Plane:	pgp,pwr ,power , ,gp,gnd,ground
Drill Top:	dd01 ,gd1 ,gg1 ,drill ,drl ,tap
Drill Bottom:	dd02 ,gd ,gg ,drill ,drl ,tap
Drill Internal:	dd ,gd ,gg ,drill , ,drl ,tap
Silk Top:	sst01 ,gto,silk,slk,sst ,ts
Silk Bottom:	sst ,bsilk ,gbo ,ssb,silk ,bs
Mask Top:	sm01 ,cmask ,gts ,tmask ,tmk , ,smt,mask1 ,maskcom ,tr
Mask Bottom:	sm ,gbs ,bmask ,bmk , ,smb,mask2 ,masksold ,br
Paste Top:	sp01 ,gtp ,gpt ,tm ,paste
Paste Bottom:	sp02 ,gpb ,gpb ,bm ,paste
Rout/Mill:	RTE,MILL
Insulator:	INS
Refdes Top:	tref
Refdes Bottom:	bref
PCB Border:	BDR ,gko ,border ,out

OK Cancel

## 6.10.6 Synchronize Layers...

Merge all Job Layers into a common set of layers.

 **Note:** When merging different jobs together, chances are that each of those jobs used a different name to describe the same layer types (ie. "top02.pho", "topside.gbr", "art01.ger"). Synchronize Layers combines those layers with same types into one layer, that can then be easily exported.

### Command Sequence

1. Select menu: Setup | Synchronize Layers...

2. Define all Layer Types for the child Job, and click button 'Next'.

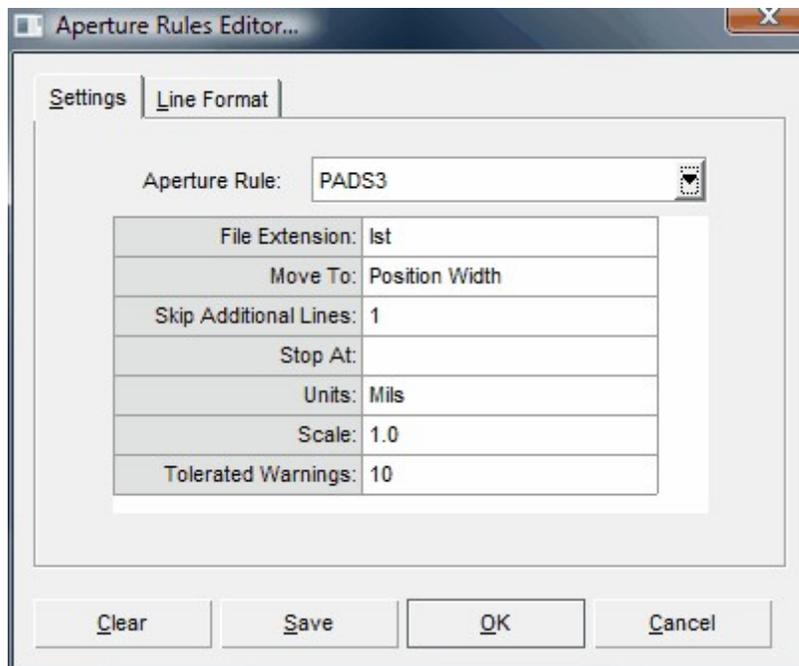
 **Note:** Synchronize Layers requires that you define the layer the layer types for all jobs. If you are unsure, choose layer type "Graphic" to be ignored..

3. Repeat Step#2 for each child job.

 **Note:** Once you have defined all layer types for child jobs, Synchronize Layers will begin to merge the layers, into one common set of layers.

## 6.10.7 Aperture Rules Editor

Create custom "rules" to load-in any aperture list file.



### Aperture Rule:

This drop down contains all available aperture rules. Each aperture rule is stored as a file (\*.rul) in the

Fab3000 sub-folder "aptrules".

**File Extension:**

Aperture files generated from a CAD system usually have a defined file extension. This helps Fab3000 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 Fab3000 ignore all text lines until the defined "Move To" text is found. If left empty, Fab3000 will start reading apertures at the first line.

**Skip Additional Lines:**

This allows Fab3000 to additionally skip lines after Move To.

**Stop At:**

Tells Fab3000 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) Fab3000 will allow before choosing to use another aperture rules files.

---

**Line Parameters:**

[S] = Skip

[D] = Dcode

[X] = X-Size

[Y] = Y-Size

[R] = Rotation

[T] = Thermal Tie Width

[I] = ID

[O] = OD

[A] = Aperture

[M] = Misc

**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]

---

 Note: Fab3000 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.

**6.10.8 Blind and Buried Via Setup...**

Assign blind and buried layer group sets.

---

**6.10.9 Set Grid**

Assign Grid settings.

---

**6.10.10 Set Origin...**

Change the location of the origin.

---

**6.10.11 Set Background Images**

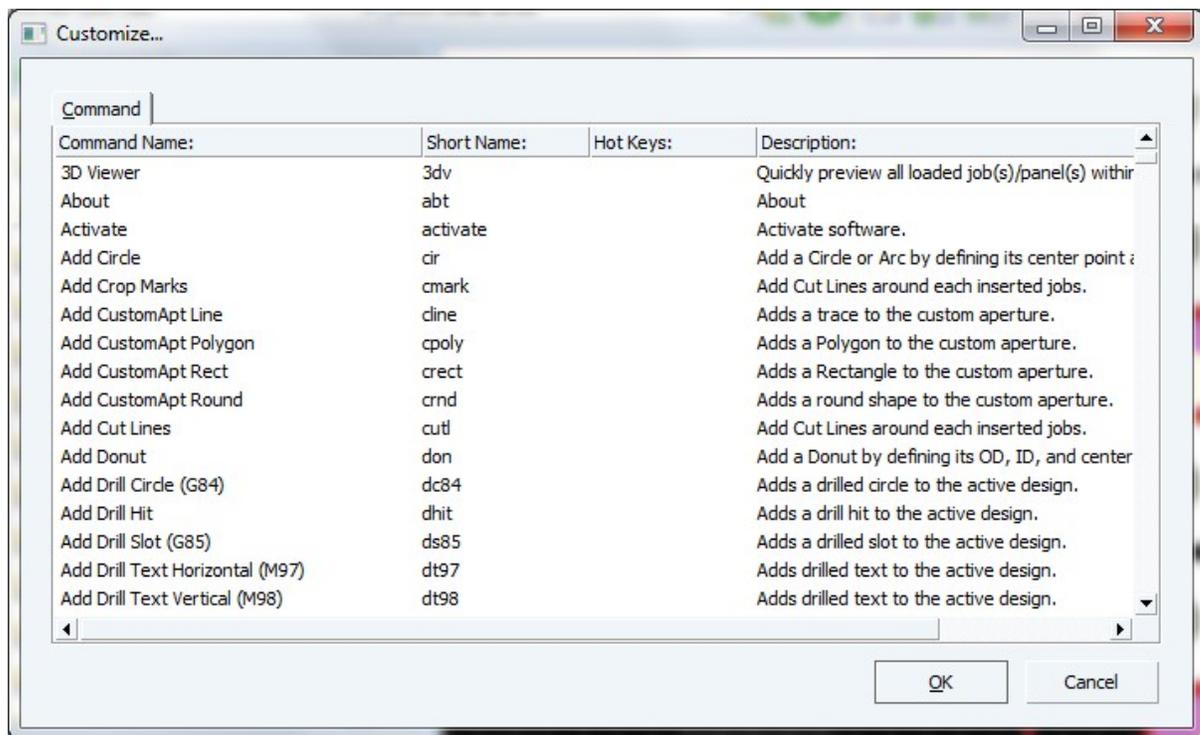
Assign images to be drawn over existing workspace data.

---

**6.10.12 Customize**

Customize User Commands

---



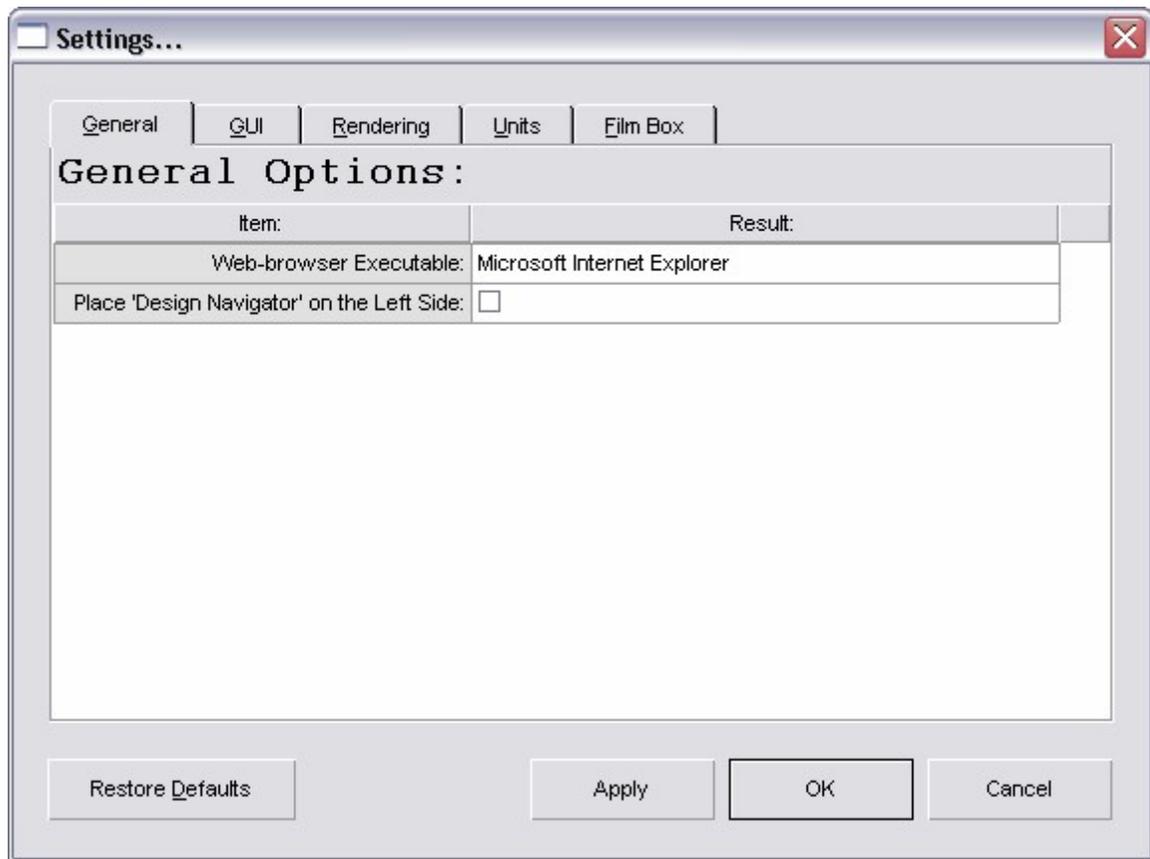
### 6.10.12.1 Shortcut Keys

Shortcut keys available while in the editor. To View Shortcut Keys, start FAB 3000 and go to menu: Setup | Customize..., and press tab "Commands".

## 6.10.13 Options

### 6.10.13.1 General

The following are the general settings and preferences.

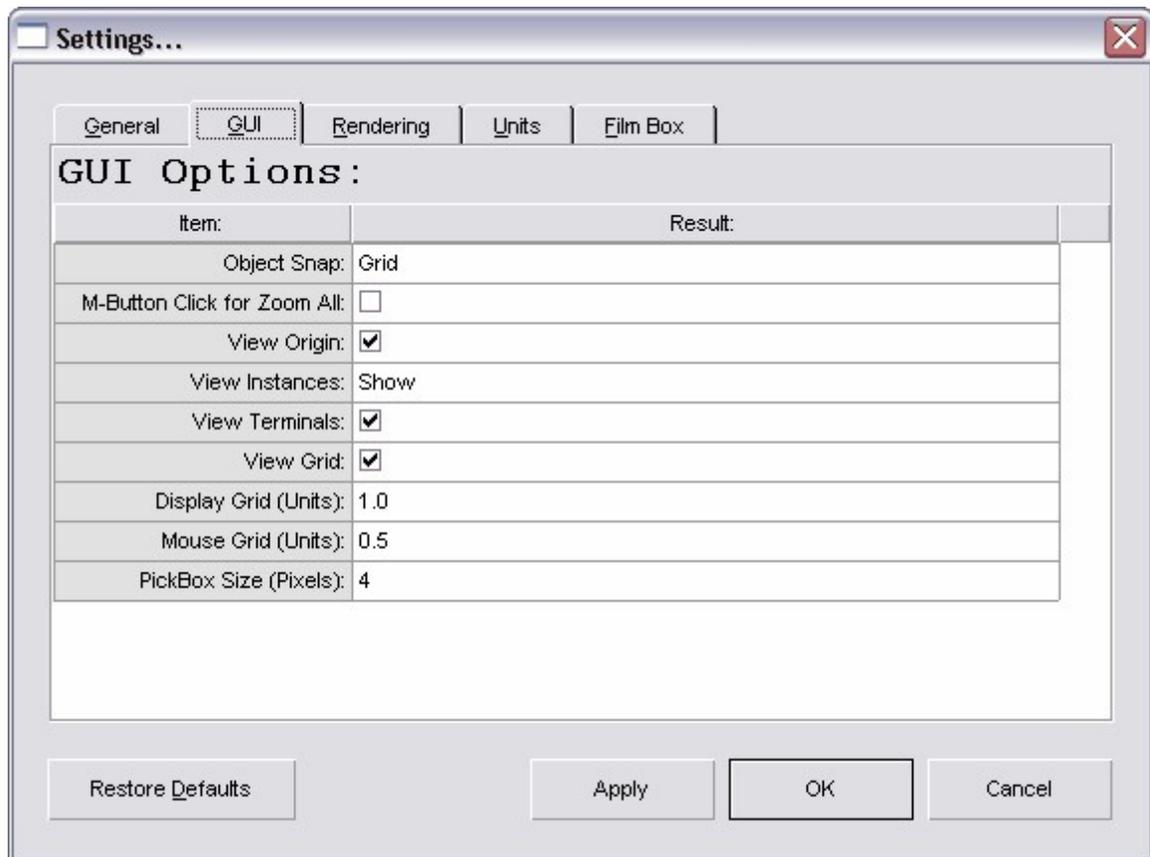


**Web-Browser Executable:**  
Use this browser to display any HTML or XML files.

**Place Design Navigator on the Left Side:**  
Depending on your preference, the Design Navigator window can be set to either the left or right. After making the change, you will need to restart the program.

### 6.10.13.2 GUI

The following are the editor settings and preferences.



#### Object Snap:

None - Turns off all snap modes. With snap modes off, you define points by simply clicking.

Grid - Snaps cursor to the nearest grid point.

Object - Snaps to the nearest vertex on a Object. A vertex can be an endpoint of a line or line segment, corner of a polygon, or endpoint of an arc or curve.

#### M-Button Click for Zoom All:

If you would like to simply click the middle mouse button to zoom all, then check this option.

#### View Origin:

View origin symbol.

#### View Instances:

Show - Display all child Objects within the instance.

Outline - Display only the bounding box of the instance. Used primarily for reference and increasing redraw speed on complex designs.

#### View Terminals:

If checked, display terminal name as text in lower left corner of a Object. It only applies for Objects that have a terminal associated with it.

#### View Grid:

If checked, the grid will be display.

**Display Grid (User Units):**

The visible grid. Grid spacing of points in X/Y direction. The units are defined as user units - this means that a value of 1 could equal 1 micron or 1 inch, it only depends on what current units you are using.

**Mouse Grid (User Units):**

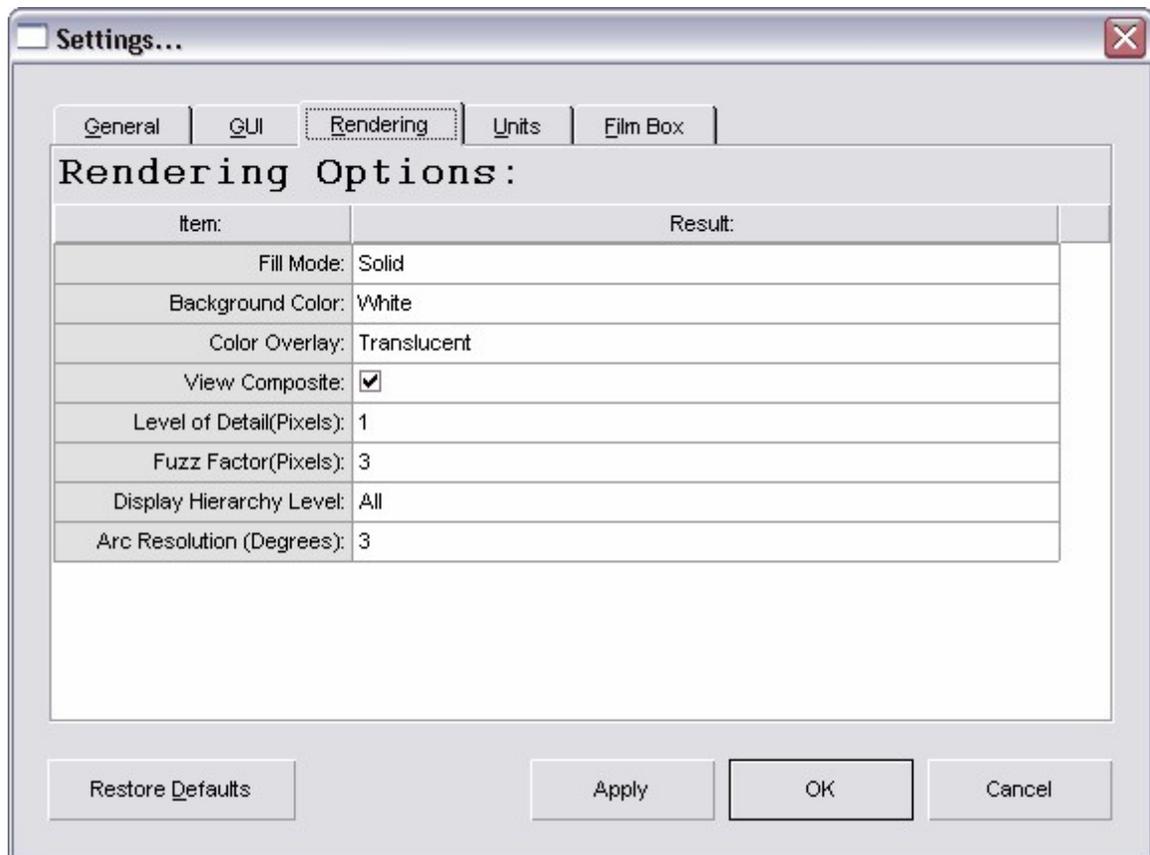
Grid spacing of points in X/Y direction, which the cursor will snap to when Snap to Grid is active. The units are defined as user units - this means that a value of 1 could equal 1 micron or 1 inch, it only depends on what current units you are using.

**PictureBox Size (Pixels):**

The invisible selection box used to determine if your cursor is directly over a Object.

**6.10.13.3 Rendering**

The following are the editor rendering settings and preferences.

**Fill Mode:**

Controls the filling of all Objects: Outline(No fill), Hatch, or Solid.

**Background Color:**

Depending on your preference the background color can be either White or Black.

**Color Overlay:**

Opaque - Object colors are drawn solid.

Translucent - Object colors are drawn with a "see through" appearance.

**View Composite:**

If checked, displays composite layers in their true visual appearance. If uncheck composites are just drawn as normal layers.

**LOD (Pixels):**

Any Objects bounding box that is smaller (in pixels) than this value can be omitted from rendering.

**Fuzz Factor (Pixels):**

Controls the size for geometries that are still to small to be completely rendered, but large enough to be visible on the screen. These objects will be drawn with low resolution to improve overall redraw time.

**Display Hierarchy Level:**

Controls the nested instance depth visible on the screen.

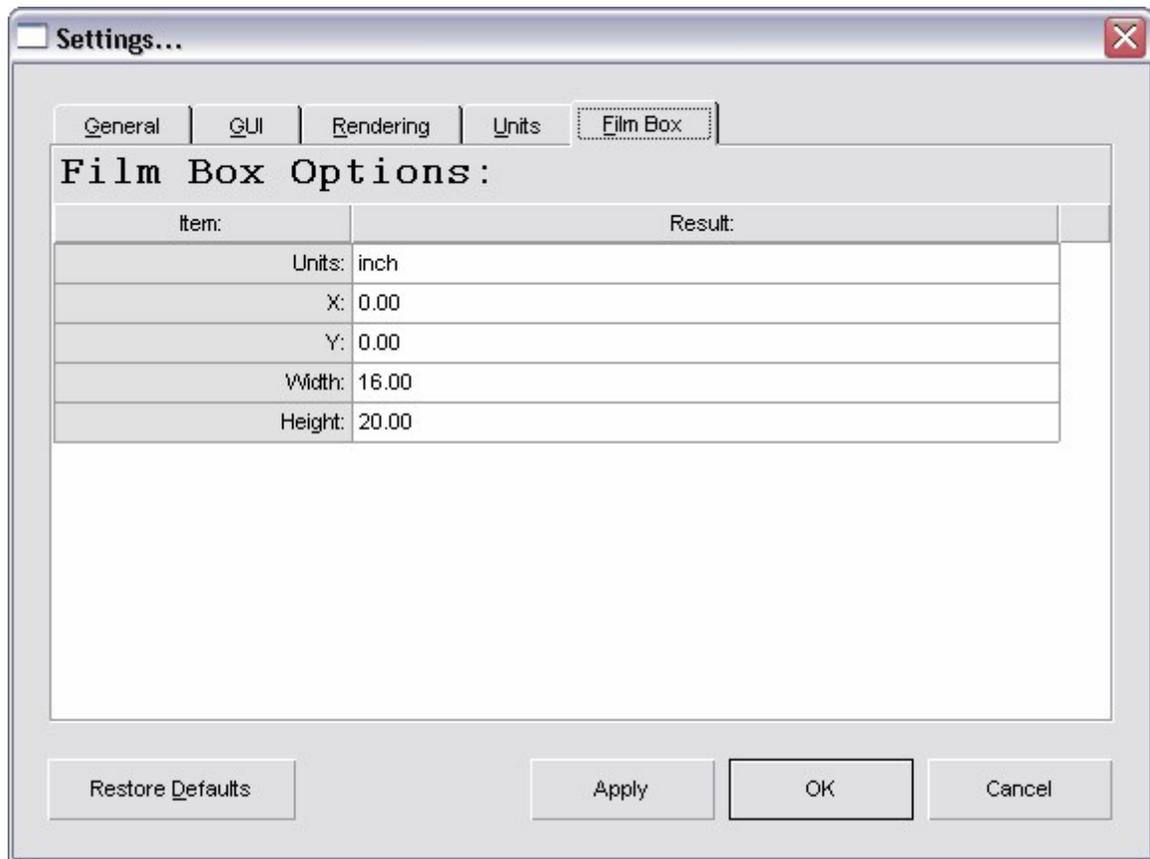
**Arc Resolution:**

Controls the overall resolution factor for vectorized arcs.

**6.10.13.4 Film Box**

The following are the settings for Film Box display.





Controls the dimensioned used when display the film box.

#### 6.10.13.5 Synchronize Setup

Setup Synchronize Layers.

#### 6.10.13.6 Restore Defaults

Restores all Settings to original defaults. This can be a useful feature to reset, in case you have somehow assigned unwanted settings.

## 6.11 Applications

### 6.11.1 New

Create a new Application File.

## 6.11.2 Open

Open an application file.

## 6.11.3 Run

Run an application File

## 6.11.4 Add App to Menu

Customize Menu with your Apps.

## 6.12 Window

### 6.12.1 Workspace Browser...

Quickly preview all loaded job(s)/panel(s) within workspace, without having to open each one into the editor.

### 6.12.2 Custom Aperture Browser...

Quickly preview all loaded custom apertures within workspace, without having to open each one into the editor.

### 6.12.3 Close All Pages

Closes all opened pages in the editor.

## 6.13 Help Menu

### 6.13.1 Help Topics

This help documentation.

### 6.13.2 FAB 3000 - User Support Center

Enter topic text here.

### 6.13.3 FAB 3000 - Online Tutorials and Demos

Enter topic text here.

#### 6.13.4 CAM Features Comparison

Enter topic text here.

#### 6.13.5 Become a Numerical Reseller or Partner

Enter topic text here.

#### 6.13.6 Visit Numerical Innovations.com

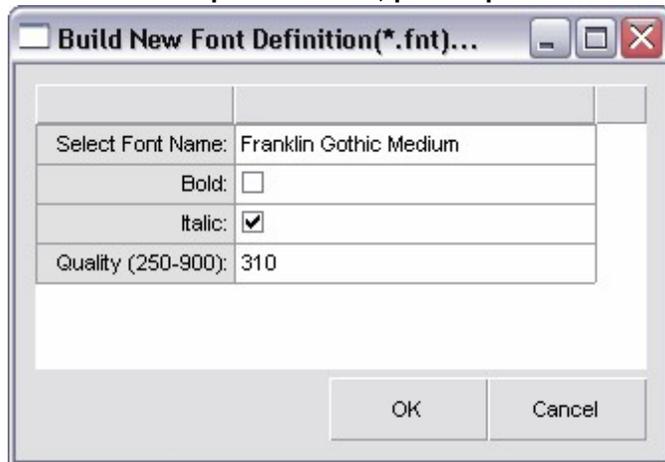
Enter topic text here.

#### 6.13.7 Compile TrueType Fonts

Define all font definitions used in Fab3000.

If you require a font style that is not part of the distribution, you may create and compile your own font files that can be used anywhere in the editor. You may create a compiled font file are based off of any TrueType font currently loaded on your system.

**To create a compiled font file, please perform the following:**



**Select Font Name:**

Choose the TrueType font that you want to create and compile.

**Bold:**

Determine if this font definition will have "Bold" characteristics.

**Italic:**

Determine if this font definition will have "Italic" characteristics.

**Quality (250-900):**

Determine the spline quality used to outline the TrueType fonts. A lower number will product rough edges, however the files will be more compact.

### 6.13.8 Compile AutoCAD SHX Fonts

Converts an AutoCAD SHX file to a binary Fab3000 font file (\*.fnt).

### 6.13.9 Enter License Key

#### 6.13.9.1 Activate

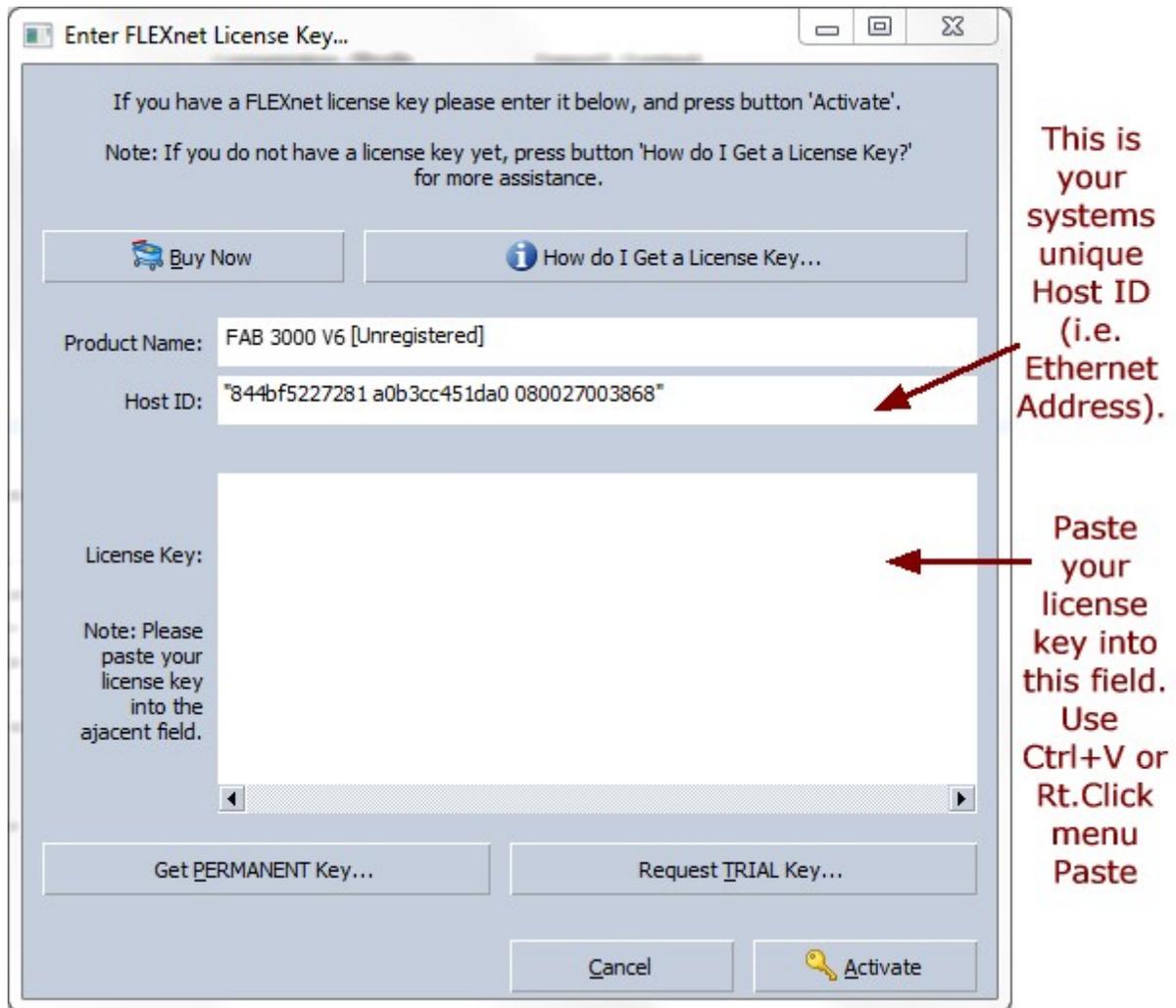
Activate your Activation Code for License Types: Standard, User, and Enterprise. (Windows Only)

#### 6.13.9.2 De-Activate

De-Activate your Activation Code for Standard License Type (Windows Only). Used so you may transfer your existing activation code to another computer.

#### 6.13.9.3 FLEXnet Key...

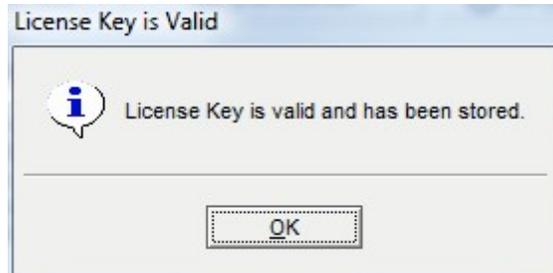
FAB 3000 uses FLEXnet Publisher (formerly FLEXlm) for license management. FAB will accept either a valid License Key or License File (\*.lic). Without a Valid License Key or File (\*.lic), FAB will start in unregistered mode. A Node Locked License is also referred to as a Single User License.



### Installing a FAB 3000 License Key:

- A. Start FAB 3000.
- B. Go to FAB 3000 menu: Help | Enter License Key | [Enter Flexnet Key...](#)
- C. Paste your license key into the license window, and press button "Activate"

Note: If your license key is valid, you will receive a success message (see below).



Note: If your license key is invalid, you will receive this message (see below). We recommend that if

you are having problems to press 'Yes; to review the error log. If you feel this is an error, please send a screen capture of the error log to [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com) and we'll be happy to resolve the license key issue.



---

### Installing a FAB 3000 License File (\*.lic):

Place your valid license file (\*.lic) into the FAB 3000 license folder:

Windows

ex. C:\Numerical Innovations\FAB3000V6

Linux

ex. /opt/Numerical/FAB3000

Note: FAB 3000 will automatically check the FAB 3000 Installation first for a valid license file.

### Buy Now:

Takes you to the [Numerical Innovations Online Store](#) to purchase EasyGerb and get your permanent license key.

### Get PERMANENT Key:

Press button to be taken to the ACE License Request Form.

### Request TRIAL Key:

Press button to be taken to the ACE Trial Request Form.

### Host ID:

Displays the Host ID for your specific machine.

## 6.13.10 Check for Update

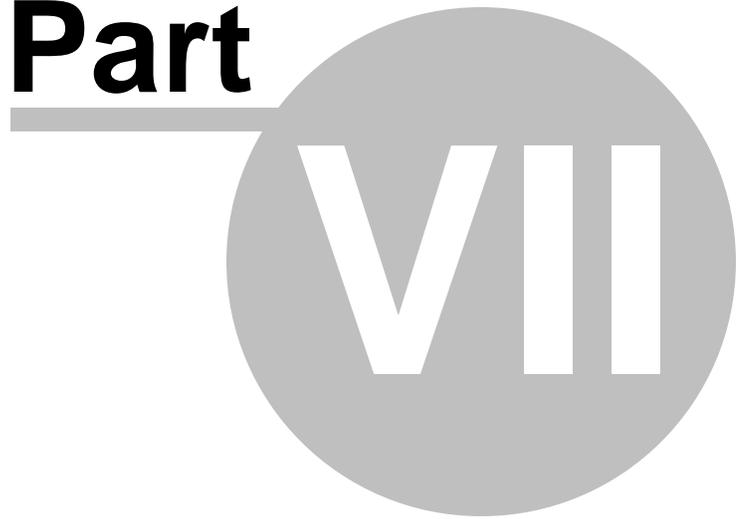
Check the Numerical Innovations website for the latest update.

### 6.13.11 About

Displays About Dialog containing version number, copyrights, and trademarks.



**Part**



## 7 Import Matrix File

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 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 - saving \$\$\$ on fabrication costs. In addition, FAB 3000 will automatically performs all necessary object transformations, dcode & drill tool transcoding.

**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.

**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:\tempexamples\Board1\Bottom Mask.gbr
  LAYER=layer_4,C:\tempexamples\Board1\Bottom.gbr
  LAYER=smt,C:\tempexamples\Board1\Top Mask.gbr
  LAYER=layer_1,C:\tempexamples\Board1\Top.gbr
  LAYER=sst,C:\tempexamples\Board1\Top Silk.gbr
  LAYER=d_1_2,C:\tempexamples\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 <a href="#">Layer Types</a> <sup>[234]</sup>
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	This field contains the layer name and Gerber file location, separated by a comma. <b>Note:</b> The Gerber file location must be an actual location on your system and not localized path (i.e ../gerbers/top.gbr) <b>Note:</b> 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 <a href="#">NCTOOLS array</a> <sup>[234]</sup> in the FAB 3000 Import Matrix before the JOB array.

#### Fields in the PANEL array

<b>NAME</b>	The name of the panel. This will be the name FAB 3000 uses when it creates the panel.
<b>UNITS</b>	Units to use for this panel array. Must be either: <b>INCH</b> or <b>MM</b>
<b>WIDTH</b>	Panel Width. Note: The value is defined in the panel units.
<b>HEIGHT</b>	Panel Height. Note: The value is defined in the panel units.
<b>STEPREPEAT</b>	This field contains the Job Name, X and Y Origin, X and Y Spacing, X and Y Count, Job Angle (must be <b>0, 90, 180, 270</b> ), and Mirror (must be either: <b>YES</b> or <b>NO</b> ) all separated by a commas. <b>Note: Origin and Spacing values are defined in the panel units.</b> <b>Note: If you only require a 1 up (no array) assign X and Y Spacing to 0; X and Y Count to 1.</b>

Note: Jobs must be referenced before Panel

#### Fields in the NCTOOLS array

<b>UNITS</b>	Units to use for this nctools array. Must be either: <b>INCH</b> or <b>MM</b>
<b>INTEGER</b> <b>DECIMAL</b>	Integer digits for drill or rout file between (0-6). Decimal digits for drill or rout file between (0-6).
<b>TYPE</b>	Drill or rout file type. <b>ABSOLUTE</b> -- Use Absolute coordinates during NC Import <b>INCREMENTAL</b> -- Use Incremental coordinates during NC Import
<b>SUPPRESSION</b>	Drill or rout file zero suppression. <b>LEADING</b> -- Suppress Leading Zeros during NC Import <b>TRAILING</b> -- Suppress Trailing Zeros during NC Import <b>NONE</b> -- No Zero Suppression during NC Import
<b>TOOL</b>	This field contains the Tool # (between 1 to 1000), Diameter (in Tool Units), and Plated (either: <b>YES</b> or <b>NO</b> ); 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.

## 7.1 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 FAB3000 - MATRIX FILE
```

```
LAYER {
  TYPE=SILK_TOP
  NAME=ssst
  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  
}
```

## 7.2 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 FAB3000 - 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 - 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
}

```

### 7.3 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 FAB3000 - 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
}
```

```
#####
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
}

```

## 7.4 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\Fab3000"

```

//Start
STARTUPINFO si;
PROCESS_INFORMATION pi;
GetStartupInfo(&si);
SetCurrentDirectory( "C:\\Temp" );

CString CamPath = "C:\\Program Files\\Numerical Innovations\\Fab3000
\\Fab3000.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" );
}

```

## 7.5 Layer Types

Here are a listing of the available layer types which can be used in the LAYER array, TYPE field.

<b>TOP</b>	Top Side Metal
<b>BOTTOM</b>	Bottom Side Metal
<b>INTERNAL</b>	Internal Metal (Positive Polarity)
<b>NEG_PLANE</b>	Plane (Negative Polarity -- Usually contains Thermal Dcodes)
<b>POS_PLANE</b>	Plane (Positive)
<b>DRILL</b>	NC Drill (Should have tools sizes & format embedded inside file)
<b>SILK_TOP</b>	Top Side Silkscreen
<b>SILK_BOTTOM</b>	Bottom Side Silkscreen
<b>MASK_TOP</b>	Top Side SolderMask
<b>MASK_BOTTOM</b>	Bottom Side SolderMask
<b>PASTE_TOP</b>	Top Side Paste
<b>PASTE_BOTTOM</b>	Bottom Side Paste
<b>ROUT</b>	NC Rout (Should have tools sizes & format embedded inside file)
<b>INSULATOR</b>	Insulator Layer
<b>COMPONENTS_TOP</b>	Top side Components (Must be CSV Format Centroid & it will be renamed to "comp_+_top")
<b>COMPONENTS_BOTTOM</b>	Bottom side Components (Must be CSV Format Centroid & it will be renamed to "comp_+_bot")
<b>TEMPORARY</b>	Temporary Layer (Used when the gerber type is unknown)
<b>FAB_DRAWING</b>	Fab Drawing (Used for reference only)
<b>BORDER</b>	PCB Boarder Layer
<b>SCORE</b>	Score Layer used for Panel Snap-Outs
<b>GRAPHIC</b>	Default Layer (Used when the gerber type is unknown)

## 7.6 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

<b>UNITS</b>	Units to use for this nctools array. Must be either: <b>INCH</b> or <b>MM</b>
<b>INTEGER</b>	Integer digits for drill or rout file between (0-6).
<b>DECIMAL</b>	Decimal digits for drill or rout file between (0-6).
<b>TYPE</b>	Drill or rout file type. <b>ABSOLUTE</b> -- Use Absolute coordinates during NC Import <b>INCREMENTAL</b> -- Use Incremental coordinates during NC Import

<b>SUPPRESSION</b>	Drill or rout file zero suppression. <b>LEADING</b> -- Suppress Leading Zeros during NC Import <b>TRAILING</b> -- Suppress Trailing Zeros during NC Import <b>NONE</b> -- No Zero Suppression during NC Import
<b>TOOL</b>	This field contains the Tool # (between 1 to 1000), Diameter (in Tool Units), and Plated (either: <b>YES</b> or <b>NO</b> ); 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
}
```

## 7.7 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.
<b>minimum_padsizes</b>	positive number	Minimum Pad size allowed.
<b>minimum_smtpadwidth</b>	positive number	Minimum SMT pad width allowed.
<b>minimum_tracewidth</b>	positive number	Minimum trace width allowed.
<b>minimum_outer_tracespacing</b>	positive number	Minimum spacing on top or bottom layers
<b>minimum_inner_tracespacing</b>	positive number	Minimum spacing for innerlayers

<b>minimum_drill_pad_annularring</b>	positive number	Minimum drill to pad annular ring
<b>minimum_border_clearance</b>	positive number	Minimum clearance from board edge.
<b>find_acidtraps</b>	positive number	Locate Acid Traps
<b>find_nonfunctional_internalpads</b>		Find non functional pads on internal layers
<b>minimum_drillsize</b>	positive number	Minimum drill hole diameter
<b>find_drill_doublehits</b>		Find drill double hits.
<b>find_drill_missinghits</b>		Find missing drill hits
<b>find_missingpads</b>		Find missing pads
<b>find_touchingholes</b>		Find drill holes that touch
<b>minimum_silkscreen_linewidth</b>	positive number	Find minimum silkscreen line width
<b>find_silkscreen_overmask</b>	positive number	Locate silkscreen over soldermask
<b>silkscreen_border_clearance</b>	positive number	Minimum clearance allowed from Silkscreen to board edge
<b>undersize_mask_clearance</b>	positive number	Locate small mask openings
<b>missing_mask_clearances</b>		Find missing mask clearances
<b>find_mask_slivers</b>		Find mask slivers
<b>find_solder_bridges</b>	positive number	Find solder bridges
<b>inner_clearance</b>	positive number	Inner clearance on plane layers
<b>power_ground_minimum_annularring</b>	positive number	Minimum annular ring on plane layers
<b>find_starved_thermals</b>	positive number	Locate starved thermals
<b>find_power_ground_shorts</b>		Find power/ground shorts
<b>check_negplane_thermal_conflicts</b>		Find thermal conflicts on neg. planes
<b>find_isolated_connections</b>		Find isolated connections
<b>output_drc_pdf_file</b>		Location for resulting PDF file,
<b>ipc_netlist_file</b>		Location of IPC-D-356 netlist file used for netlist comparison
<b>x_netlist_offset</b>		Offset IPC Netlist file in X direction
<b>y_netlist_offset</b>		Offset IPC Netlist file in Y direction
<b>netlist_scale_factor</b>	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_1,274\Art0121.pho
  LAYER=layer_4,274\Art02.pho
  LAYER=layer_2,274\Pgp2529.pho
  LAYER=layer_3,274\gnd2530.pho
  LAYER=sst,274\Sst0126.pho
  LAYER=spt,274\Sp0128.pho
  LAYER=smb,274\Sm0227.pho
  LAYER=smt,274\Sm0128.pho
  LAYER=d_1_2,274\Drill.drl
```

```
}  
  
DFMChecker {  
    units=mil  
    minimum_padsizes=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  
}
```

**Part**



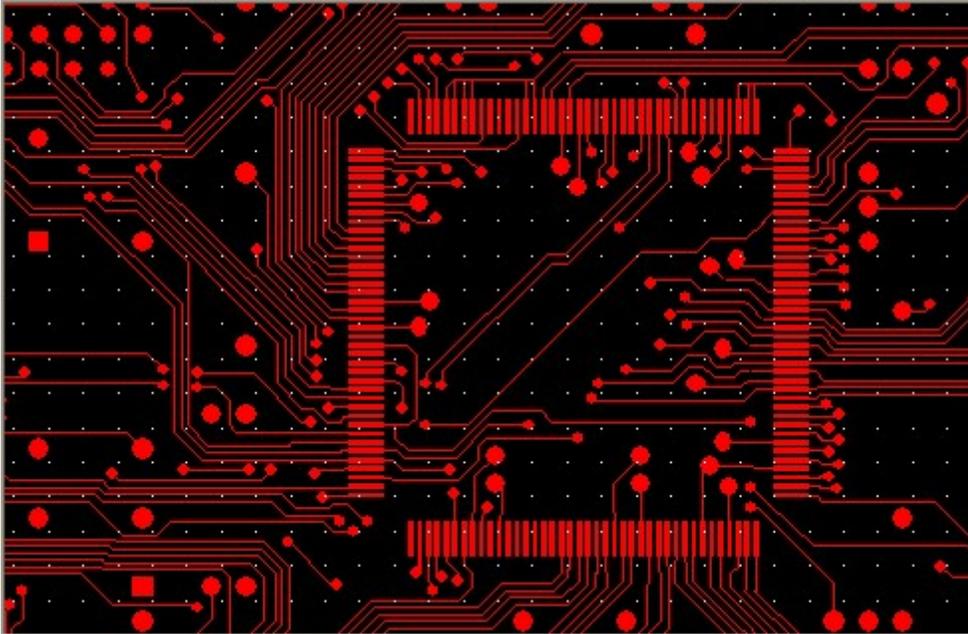
## 8 Data Formats

Fab3000 supports many file formats, such as GDSII, DXF, OASIS, Postscript, Gerber, and more.

### 8.1 Gerber

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.



The Gerber Format:

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 authors 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 zeros.
- ▶Format type: Incremental(G91)
- ▶Most software products have a hard time loading, & rendering gerber files attributes properly (ie. rounded paths, custom flashes, composite layers, and step/repeat).

### **Fab3000 is the Perfect Solution:**

Fab3000 successfully handles all features of the Gerber (RS-274X) format. Custom apertures, composite layers, and step/repeat are all correctly processed.

## **8.2 ODB++**

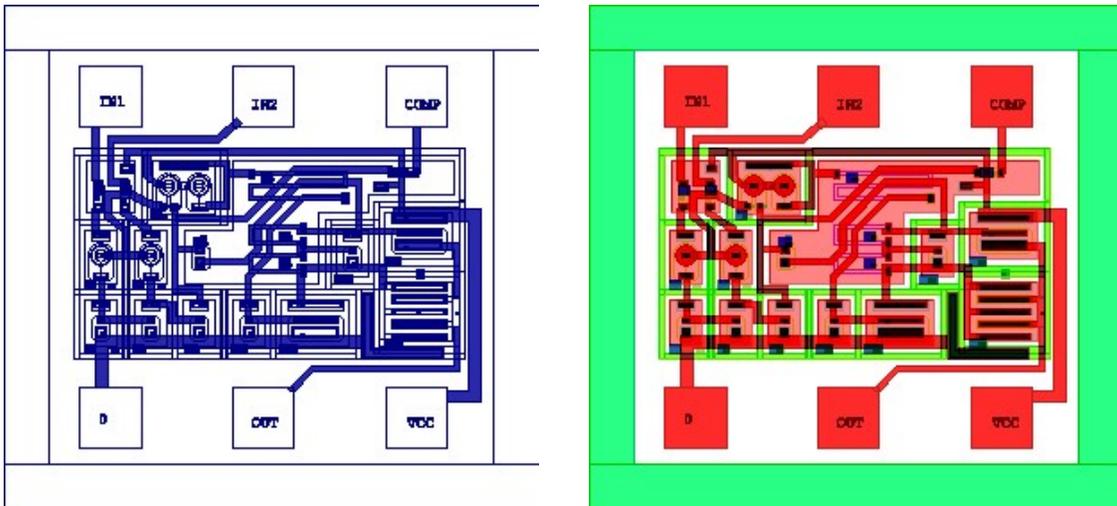
ODB++ is the most intelligent CAD/CAM data exchange format available today, capturing all CAD/EDA, assembly and PCB fabrication knowledge in one single, unified database. Originally developed by Valor Computerized Systems for use in its own PCB CAD/CAM systems, Genesis 2000, Enterprise 3000 and Trilogy , 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.

## **8.3 DXF**

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 Fab3000:*



### The 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.

### 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 block's 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 rectangles.
- ▶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, because 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.

### **Fab3000 is the Perfect Solution:**

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; Fab3000 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 supported 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

Fab3000 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 Fab3000 is for you!



Supports Polygon De-Embedding for Solid Hatches!

## **8.4 DWG**

Bi-directional translation of native AutoCAD DWG format. Similar to the [DXF format](#)<sup>[240]</sup>, except in a binary file.

---

## **8.5 IGES**

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.

---

**Requires Add-On: "3D Formats"**

## **8.6 Image**

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 (ie. change layers, copy, export to GDSII, DXF, etc.). Excellent 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.

## 8.7 GDSII

GDSII is a binary file format which is classified as a "data interchange format", used for transferring mask-design data between the IC designer and the fabrication facility ("Fab").

## 8.8 Postscript

PostScript is the industry standard for sending documents to high-resolution printers.

**Note: 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/>

## 8.9 Large Monochrome Bitmap

Bi-directional translate 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. Note: Image files are unit-less, and a pixel can theoretically represent any physical size. It is important that you specify the proposed units and scale for a pixels physical size. Choose from DPI, Micron, Inch, Millimeter, Centimeter, and more. Logical pixel size defined by Pixel Units \* Image Scale.

**Requires Add-On: "Large Bitmap"**

## 8.10 Drill/Rout

NC code used to control Drill and Rout CNC machines.

## 8.11 STL

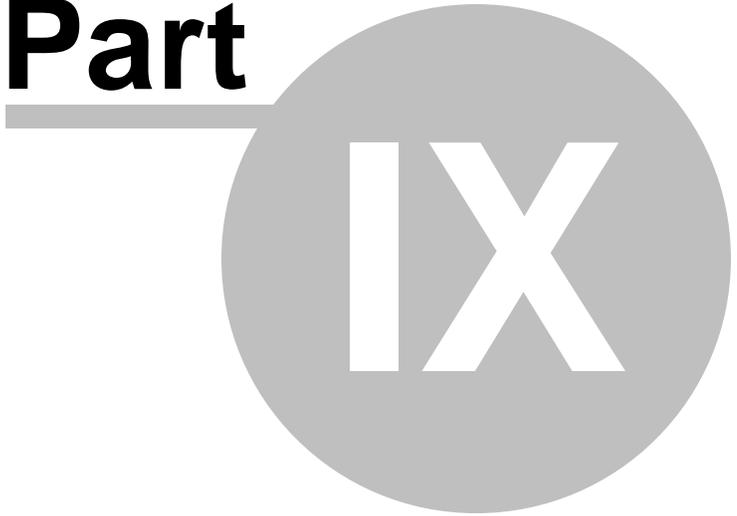
STL is a file format native to the stereolithography CAD software created by 3D Systems. This file format is supported by many other software packages; it is widely used for rapid prototyping and computer-aided manufacturing. STL files describe only the surface geometry of a three dimensional object 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 including SolidWorks, ProEngineer and more. 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.



**Part**



**IX**

## 9 Purchase and Support

### 9.1 How do I place an Order?

All Numerical Innovations software products can be purchased, by contacting our sales team: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com)

---

#### What can I expect after placing an order?

You will immediately receive an e-mail which contains:

- (1) A copy of your paid invoice (to retain for your records).
- (2) An contains a link to download the product (if you haven't already), along with license key setup instructions.

 **Note:** Numerical Innovations, uses FLEXnet Publisher (formerly FLEXlm) for license management.

---

#### Credit Card:



We accept VISA, MasterCard and American Express cards, and this is our preferred method of payment. You may order any product on [our website](#) using your credit card, or FAX your credit card order to our sales department at: 1-866-528-9274 (Monday - Friday 9:00AM to 5:00PM PST). Please make sure you include your name as it appears on the card, the card number and the expiration date. Our Fax line is secure and accessed only by our sales team.

#### Company Purchase Order (PO):

Company Purchase Orders are accepted. We verify company information through D&B, and may request trade references before processing. Terms are: Net 30 days. You may submit your purchase orders to our sales team by fax: (++1) 858-430-2705 or e-mail: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com)

---

#### Shipping:

All software is delivered exclusively by the Internet. After your purchase, we will send you an Activation License file (license.lic), which unlocks the demo version into a registered version. If you have purchased a Network/Floating license, you will also receive the "Numerical Network Bundle" in addition to the activation license file.

 **Note:** Software delivered via CD-ROM is available upon request.

---

#### More Questions:

Please visit our website forum [Numerical Support Center](#) for more assistance.

*We thank you for your business.*

**Paypal, Discover Card, JCB:**

If you would like to use Paypal or another credit card type please contact us 1-(800) 269-5045 or e-mail: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com) and we'll be happy to setup a payment link for you using our PayPal account.

**Company Purchase Order (PO):**

Company Purchase Orders are accepted for orders over \$500 (USD). We verify company information through D&B, and may request trade references before processing. Terms are: Net 30 days. You may submit your purchase orders to our sales team by fax: 1-(800) 269-5045 or e-mail: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com)

**Shipping:**

All software is delivered exclusively by the Internet. After your purchase, we will send you an Activation License file (license.lic), which unlocks the demo version into a registered version. If you have purchased a Network/Floating license, you will also receive the "Numerical Network Bundle" in addition to the activation license file.

 Note: Software delivered via CD-ROM is available upon request.

**More Questions:**

Please visit our website forum [Numerical Support Center](#) for more assistance.

*We thank you for your business.*

## 9.2 Product Pricing

For product pricing, please our sales team: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com)

## 9.3 Numerical Maintenance Plan

The Maintenance plan includes secure login access to receive every upgrade, update, & bug fix; plus get priority technical support from our staff, and license key replacement in the event of a lost license, or computer transfer. The Maintenance plan is good for as long as we continue to develop the software product you have purchased. What this means is that you are eligible to receive every upgrade, update, new release, and bug fix -- there will never be any additional charge for you.

Visit our website for more details:

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

## 9.4 How do I contact Support?

If you need professional assistance, we're here to help.

### **Numerical Support Center:**

Our community message board can be accessed by clicking on the following link:

<http://forums.numericalinnovations.com/mb/numericalinnov>

### **Direct Technical Assistance:**

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

E-mail is preferred for all support issues.

 Note: Please include a detailed explanation of the problem, plus a sample file (if possible).

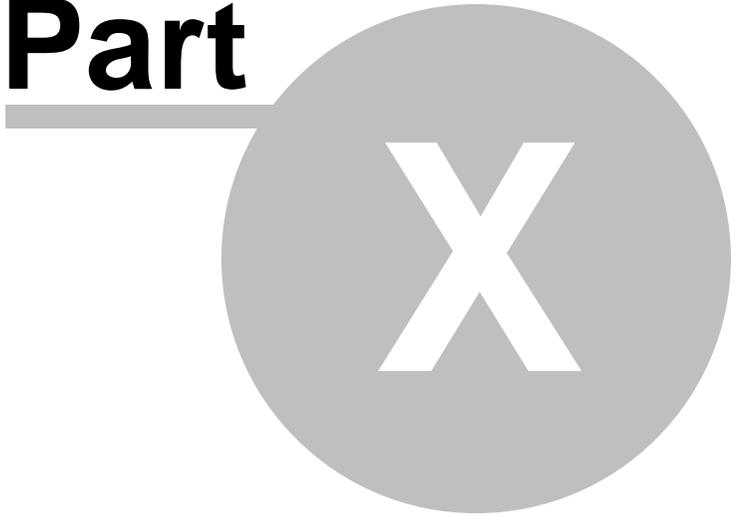
 Note: All support e-mails are responded to within 1-business day or sooner.

## 9.5 License Key / Activation Code Problems

Please contact us if you have an license key problems: [sales@numericalinnovations.com](mailto:sales@numericalinnovations.com)

---

**Part**



## 10 Movie Tutorials

To view hundreds of movie tutorials for FAB 3000, please visit the following link:  
<http://www.youtube.com/user/NumericalInnovations/search?query=fab+3000>

# Index

## - A -

About 216  
Add Circle 148  
Add Drill Hit 159  
Add Ellipse 150  
Add Epoxy Bars... 173  
Add Flash 147  
Add Layers 126  
Add Multi-Line Text 153  
Add Polygon 149  
Add Rectangle 150  
Add Text 152  
Add Trace 148  
Add Wide Path 151  
Add/Remove Objects from a Net 56  
Align Layers 129  
Aperture Rules Editor 202  
Aperture Table 195  
Apply External Net Names to CAM Net 57  
Apply Nets 182  
Array 135

## - B -

Bare-Board Analysis 177  
Blind and Buried Via Setup... 204  
Boolean 139  
Boolean: Add 190  
Boolean: Assign to Group A 189  
Boolean: Assign to Group B 189  
Boolean: Difference 190  
Boolean: Intersect 190  
Boolean: Invert 190  
Boolean: Subtract 190  
Build New Font Definition (\*.FNT) 212

## - C -

Chop Polygons... 188  
Clear All 125  
Clip Area... 195  
Clipboard 123

Close All Pages 211  
Command Bar 146  
Command Console 147  
Compare Net Errors 167  
Compare Nets... 181  
Comparing a CAM Netlist against an External Netlist 54  
Composite Build... 189  
Composite Close 189  
Composite Open (for Editing) 189  
Composites 101 58  
Convert to Custom Aperture 184  
Convert to Job... 186  
Convert to Raster Polygon 185  
Copper Area Calculation 174  
Copper Balancing 170  
Copy 131  
Copy to layers 132  
Create a PDF Report for DRC/DFM Violations 52  
Create a PDF report for Net Comparison Errors 56  
Create Drill Layer (PTH)... 174  
Creating Custom Apertures 58  
Custom Aperture Browser... 211

## - D -

De-Embed 138  
Delete 130  
Delete Job 160  
Delete Layers 127  
Detect Layer Types... 201  
Draw to Flash: Automatic 183  
Draw to Flash: Interactive 183  
DRC / DFM Violations 167  
Drill to Gerber 172  
DXF 240

## - E -

Edit Nets 182  
Etch Compensation 170  
Exit 122  
Exoirt NC Drill 103  
Explode All 139  
Explode Flashes 139  
Explode Step and Repeat 140  
Explode Text 140

Explode Traces 140  
 Export Custom Aperture Library 114  
 Export DXF 108  
 Export Gerber 102  
 Export IGES 112  
 Export Image (TIFF, PNG, etc.) 121  
 Export Netlist (IPC-D-356/A) 107  
 Export ODB++ 107  
 Export Postscript 113  
 Export Rout 105  
 Extracting a Netlist from Gerber and Drill data 52  
 Extracting with Blind and Buried Vias 53

## - F -

Fabrication Drawing 167  
 Fill 144  
 Film Box 145, 209  
 Film Panelize (Film Wizard) 190  
 Find Composite Level 164  
 Find Dcode 163  
 Find DRC/DFM Violations 164  
 Find NC Tool 163  
 Find Net 162  
 Finding Dcodes 21  
 Fix Invalid Polygons 188  
 Flatten Composite To Layer... 189  
 Flatten Job 160  
 Flip 134  
 Full Screen 147

## - G -

Gerber 239  
 Gerber to Drill 172  
 Graphical Comparison: Jobs 182  
 Graphical Comparison: Layers 182  
 Grid 146

## - H -

Help Topics 211  
 Highlight 145  
 Homebase Pad Conversion... 172  
 How do I contact Support? 248  
 How do I place an Order? 246

## - I -

IGES 242  
 Image 242  
 Import Aperture List 101  
 Import AutoLoad 79  
 Import Custom Aperture Library 101  
 Import DXF 91  
 Import Gerber 85  
 Import HPGL2 95  
 Import Image 94  
 Import NC Drill 87  
 Import Netlist (IPC-D-356/A) 90  
 Import ODB++ 89  
 Import ODB++ (Directory) 90  
 Import Rout 88  
 Import Zip (Compressed file) 79  
 Importing DXF 32  
 Importing ODB++ 39  
 Importing Zip (Compressed) File 39  
 Insert Job 159  
 Installation 7

## - J -

Job Information 78  
 Join 186  
 Join Errors 167

## - L -

Launching Fab It! 14  
 Layer Display 145  
 Layer Sets... 199  
 Layers Table 197  
 License Key Problems 248  
 Licensing Options 7  
 Linux OS 14  
 List Objects 168

## - M -

Markup 146  
 Markup Arrow 157  
 Markup Ellipse 156  
 Markup Layers... 157

Markup Line 155  
Markup Polygon 156  
Markup Rectangle 156  
Markup Settings... 158  
Markup Text 157  
Measure (Net-Net) 166  
Measure (Object-Object) 166  
Measure (Point to Point) 165  
Merge Jobs 40  
Merge Jobs (Automatically) 41  
Merge Jobs (Manually using Inserts) 45  
Merge Layers 128  
Minimum Gaps 180  
Minimum Widths 181  
Move 130

## - N -

Navigator Bar 146  
NC Tools Table 196  
Netlist Extraction 181  
Network - License Installation 11  
New Custom Aperture 74  
New Job 73, 160  
New Panel 73  
Numerical Maintenance Plan 247

## - O -

Open Selected Job 161  
Open Workspace 76  
Open/Save Workspaces 19  
Origin 145  
Ortho 145

## - P -

Pan 144  
PDF Reports 167  
Pin Hole Elimination 171  
Plot (up to E Size) 114  
Plot to PDF 23  
Pop Out 160  
Postscript 243  
Product Pricing 247  
Properties 141  
Push Into 160

Push-Pop Job 160

## - Q -

Query All 161  
Query Dcode 161  
Query Net 162  
Quit EIP 160

## - R -

Recent Workspaces 122  
Redo 123  
Redraw 143  
Remove Covered Data 172  
Remove Pads (Isolated / Stacked) 172  
Remove Tiny Edges... 188  
Reports 166  
Rotate 133

## - S -

Sales RFQ 167  
Save Workspace 77  
Save Workspace As 77  
Scale 138  
Scale Layer X/Y 129  
Select All 123  
Select Crossing Window 124  
Select Display 123  
Select Window 123  
Selection Filter 125  
Selection Reset Filter 126  
Set Grid 204  
Set Origin... 204  
Settings General 205  
Settings GUI 206  
Settings Rendering 208  
Settings Restore Defaults 210  
Silkscreen Optimizer 169  
Snap 144  
Status Bar 146  
Stretch 139  
Synchronize Layers 46  
Synchronize Layers... 202  
Synchronize Setup 210  
System Requirements 7

## - T -

Teardrop Creation 171  
Technical Support 15  
Toolbar 146  
Translucent 145  
Trim 139

## - U -

Undo 122  
Using AutoLoad 28  
Using Bare-Board Analysis 50  
Using Film Wizard 47  
Using Measure 19  
Using Polygon De Embedding 59  
Using Redline Markup 26  
Using the Properties Dialog 21

## - V -

View Compare Net Errors... 182  
View DRC / DFM Violations 181  
View Filter 145  
View Join Errors 187  
Viewing & Selection Basics 17  
Viewing DRC/DFM Violations 51  
Viewing Netlist Comparison Errors 55

## - W -

Welcome 3  
Windows OS 14  
Workbench 65  
Working with Fab It! 65  
Workspace 69  
Workspace Browser... 211

## - Z -

Zoom 143



