

FPCfab

The Fabrication Drawing Generator
Version 1.40

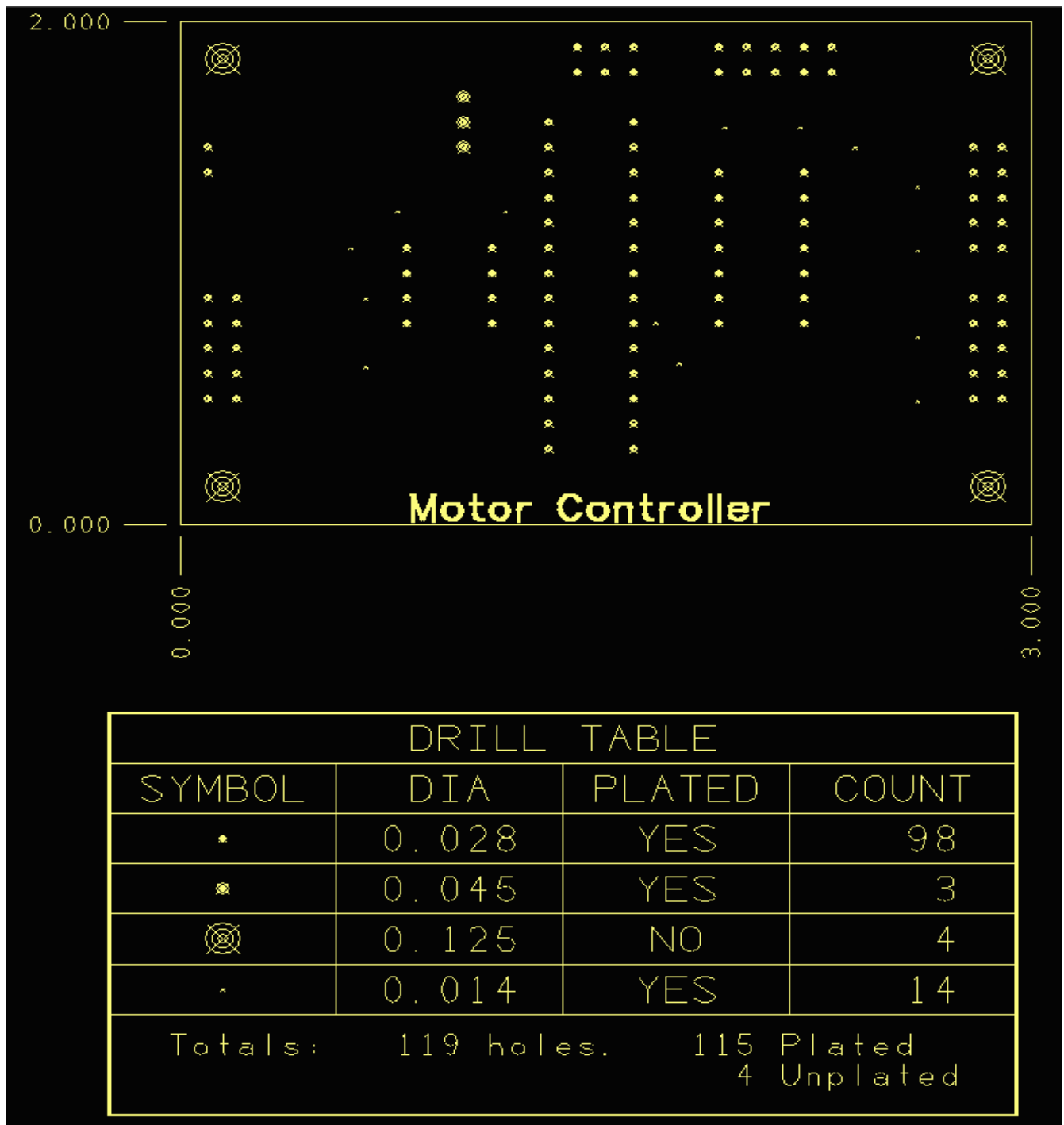


Table of Contents

FPCfab End User License Agreement (EULA).....	5
Introduction.....	7
Installation and setup.....	9
Operation.....	10
Command Line Parameters.....	11
Template Files.....	12
Fabrication Drawing File.....	13
Annotation Script Files.....	16
A – Adjust Text.....	17
B – Box.....	17
C – Circle.....	19
D - Ordinal Dimension.....	20
E – End file.....	20
I - Include.....	21
L - Line.....	22
N - Note.....	22
O - Offset.....	23
P - Polygon.....	24
S - Set Polarity.....	25
T - Text.....	26
U – User Control.....	27
Appendix A: Fonts.....	28
Appendix B: Annotated Fab Drawing.....	29
Adapter.ano:.....	30
B size.ani:.....	32
Pg1 Title Blk.ani:.....	33
Layer Tbl 8.ani:.....	35
Document History:.....	36

FPCfab End User License Agreement (EULA)

General

FPCfab Software is distributed as Freeware. You may use the Software on any number of computers for as long as you like. The Software is NOT Public Domain software. You are allowed to freely distribute the Software, but Bruce Parham retains ownership and copyright of the Software in its entirety. You may use and/or distribute the Software only subject to the following conditions:

1. You may not modify the program or documentation files in any way.
2. You must include all the files that were in the original distribution.
3. You may not reverse engineer, decompile, or otherwise reduce the Software to a human perceivable form.
4. You may not sell the Software or charge a distribution fee, except to recover the media costs.
5. You understand and agree with this license and with the Disclaimer of Warranty and the Limitation of Liability printed below.
6. The Software may be bundled and distributed together with other software products provided the same conditions apply.

Agreement to this license

By using, copying, transmitting, distributing or installing the Software, you agree to all terms of this license. If you do not agree to all of the terms of this License, then do not use, copy, transmit, distribute or install the Software and immediately remove the Software from your storage device(s).

Warranty disclaimers and liability limitations

The Software and related documentation are provided "as is", without warranty of any kind. None whatsoever. Bruce Parham disclaims all warranties, express or implied, including, but not limited to, warranties of design, merchantability, or fitness for a particular purpose or any warranty of title or non-infringement. Bruce Parham does not warrant that the functions contained in the Software or documentation will meet your requirements, or that the operation of the Software will be error-free, complete, or that defects in the Software or documentation will be corrected.

Limitation of Liability

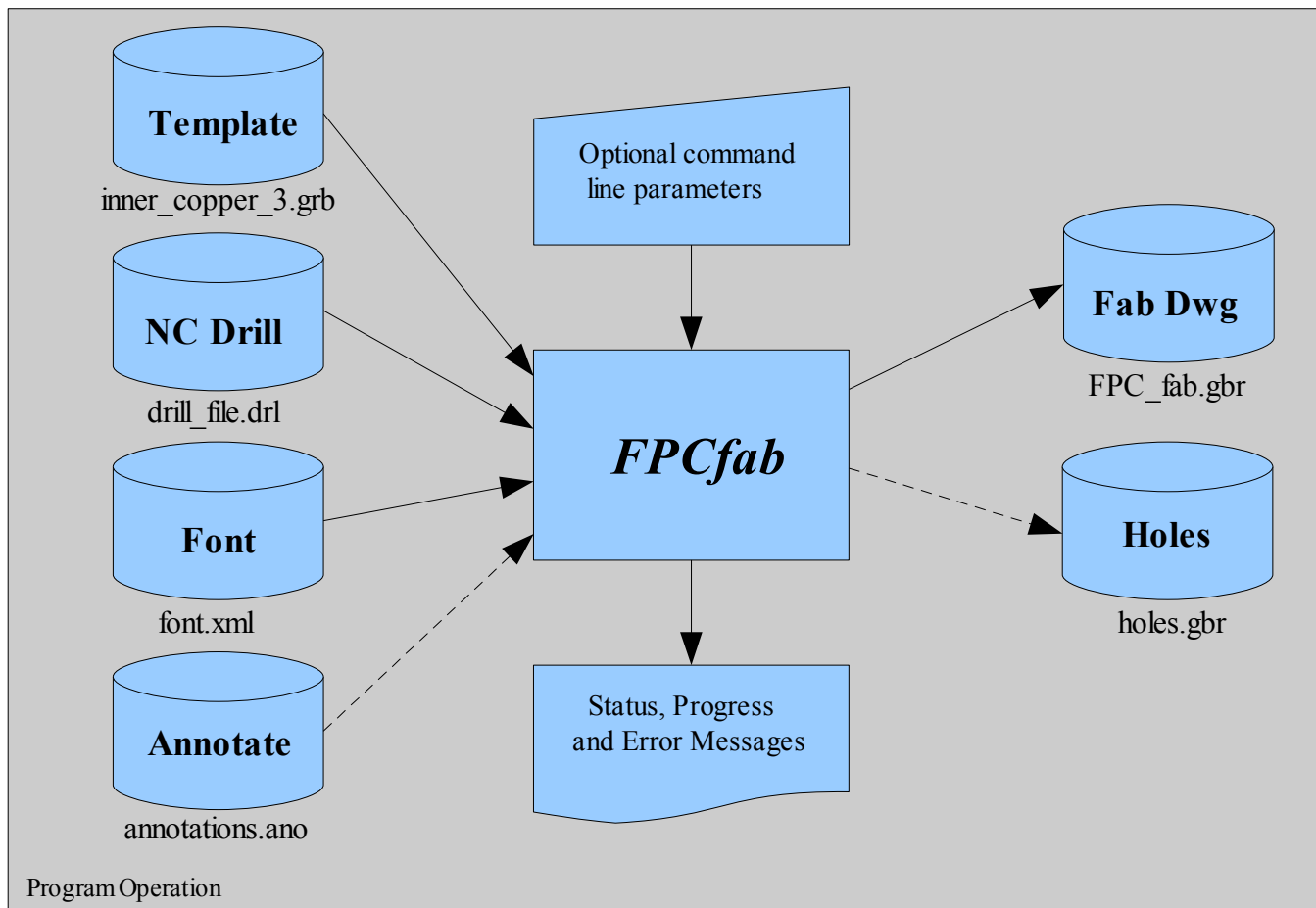
Under no circumstances shall Bruce Parham be liable for any lost revenue or profits or any incidental, indirect, special, or consequential damages that result from the use or inability to use the Software or related products or documentation, no matter what legal theory it is based on.

Copyright and trademarks

Copyright © 2006-08 by Bruce Parham. All Rights Reserved.

Introduction

FPCfab is a simple Win2K/XP console application that creates a basic fabrication drawing based on information contained in a template RS-274X Gerber file and Excellon drill file generated by **FreePCB**.



The new fab drawing is an RS-274X Gerber file that includes some or all of the data from the template file and adds a Drill Icon Table, Drill Targets and dimensions to the board outline. In addition to providing useful project documentation, some board houses prefer to use this type of drawing for drill data extraction.

An optional holes file, also RS-274X, can be generated that contains the data from the NC Drill file translated into flashed apertures sized to match the drill sizes. Although ideal for depicting through holes when creating a blank board graphic with **PBC-Render**, its real purpose is that it is the preferred drill data format for some board houses.

An optional annotation file can be used to add graphics and text to the fab drawing. The file uses a simple scripting language to generate graphic primitives and more complex objects many of which can be rendered as line graphics or as polygons.

The template file need only contain the board outline(s) but, with some limitations, may include additional data. The NC drill file is used without modification.

Installation and setup

FPCfab is distributed as a set of files in a zip archive; no install utility is used. To install **FPCfab**, extract the **FPCfab.exe** and **font.xml** files to any handy directory such as C:\Program Files\FPCfab\ or C:\Program Files\FreePCB\bin\.

If the path to **FPCfab.exe** is not already included in the **PATH** variable, it must be added. Additionally, a new environment variable, **FPCfabFont**, must be created and set to point to the newly installed **font.xml** file.

For those unfamiliar with setting environment variables, assume for example that **FPCfab.exe** and **font.xml** is installed in the C:\Program Files\FreePCB\bin directory:

- Open the Control Panel
- Open the System tool
- Click the Advanced tab and open the Environment Variables
- Under the System Variables click new
- Set the variable name to **FPCfabFont**
- The variable value to **C:\Program Files\FreePCB\bin**
- Click on OK to add the new variable
- Scroll the variable list down to and highlight the **path=** entry and click EDIT
- Click in the value field to clear the highlight
- Scroll to the end of the field and add **;C:\Program Files\FreePCB\bin** to the existing value
- Click OK to exit the edit window
- Click OK to save and exit the Environment Variables window

An additional, optional, variable **FPCfabCAMpath** can be used to point to the CAM files directory. If this variable is not set, **FPCfab** will use current working directory for CAM data I/O.

Note: The following example uses the **FreePCB** tutorial motor.fpc which has only two inner layers. To add an additional layer to an existing board for use as a template, make a copy of the file. Rename it as desired and open it with a text editor. Locate the line that begins with *n_copper_layers: 4* and change the four to a five. Save the file and exit the editor; the next time **FreePCB** opens this file, the board will have three inner layers. The color of the new layers may need to be adjusted.

Later versions of **FreePCB** allow the number of layers to be changed, at any time, as needed, from within the Project-Options menu and also provide the option to generate a board outline only Gerber file.

Operation

FPCfab is a 32-bit console application; to use it, open a new command shell after the environment variables are edited. If the shell was opened prior to the environment edits, its copy of the environment will not be updated.

FPCfab uses a number of optional command line parameters to control its behavior. Each parameter is specified by a single character preceded by a dash (-) and may be followed immediately, without any spaces, by one or more text or numeric values. If embedded spaces are needed in the value, the whole field including the dash and parameter character must be enclosed in quotes. The parameter characters are not case sensitive.

Some equivalent parameter entries with and without embedded spaces:

```
C:> fpcfab -i3.5,2.0
```

```
C:> fpcfab "-I3.5 2.0"
```

Within a quoted field, leading spaces are acceptable with numeric¹ values but not for text:

```
C:> fpcfab "-I 3.5 2.0"
```

```
C:> fpcfab "-T inner_copper_3.grb" ← Error! Space after the -T token
```

The results of parameter processing are displayed as they are processed:

```
C:> fpcfab -i-.25,-.75,2 -tinner_copper_3.grb -u.125 -U.08 -v
```

```
Icon table relocated to -0.25000, -0.75000
Template file name changed to "inner_copper_3.grb"
Drill diameter 0.12500 set to unplated
Drill diameter 0.08000 set to unplated
Verbose output enabled
```

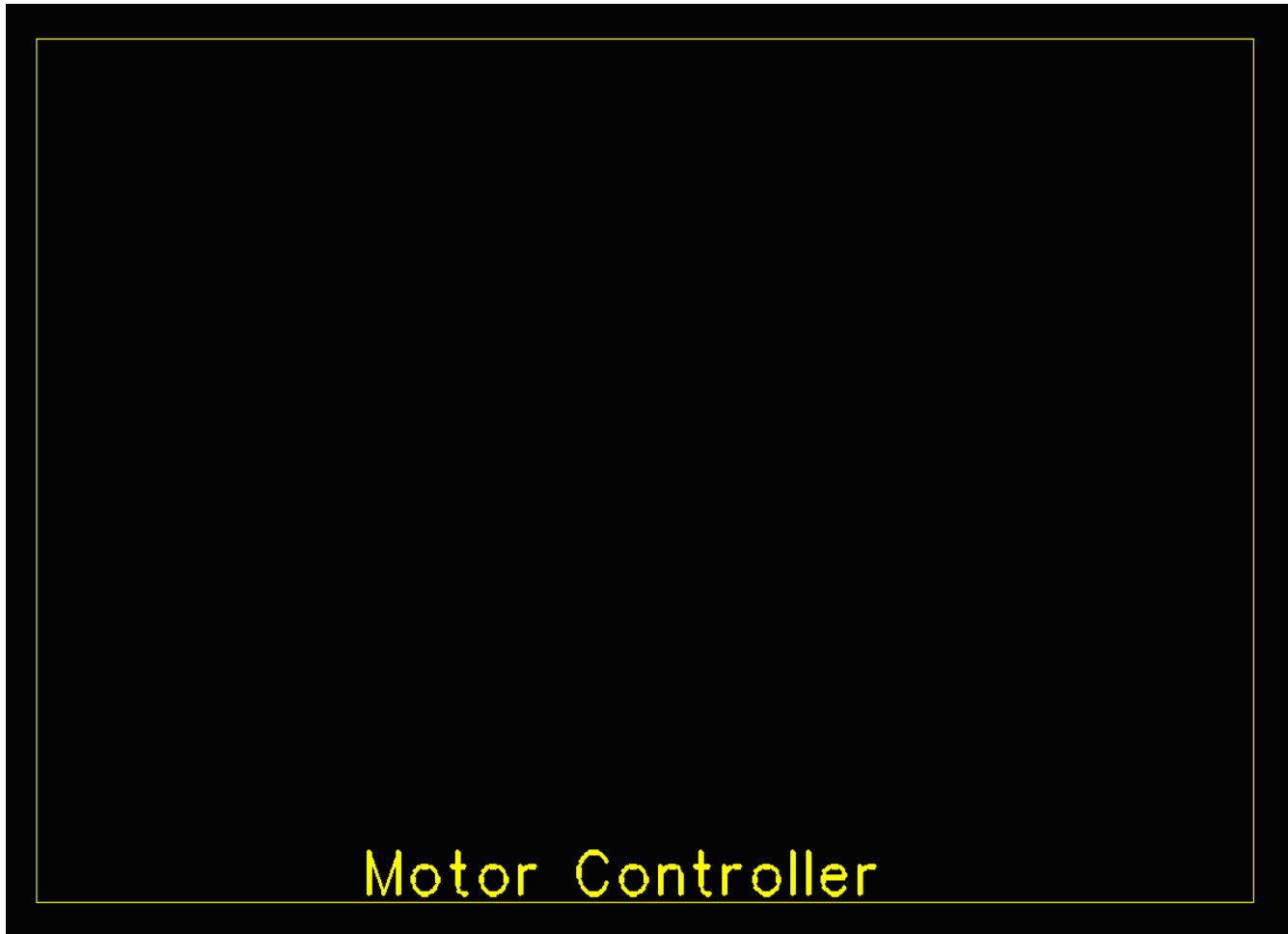
¹ Numeric values are converted using a *scanf* family function that discards one character between fields.

Command Line Parameters

- Afilename Run Annotation file filename to add text and graphic objects to the output.
- Cnewname Replace the default font file name font.xml with newname. Not needed for normal operation, this parameter can be used to switch to or test a new font.
- Dnewname Replace the default drill file name drill_file.drl with newname.
- En Exclude feature n from the final drawing.
- n:**
- 1 – Omit all outline dimensions.
- 2 – Omit the drill icon table.
- 3 – Omit all drill icons.
- 4 – Omit all board outlines
- Fnewname Replace the default fab dwg file name FPC_fab.gbr with newname.
- Gfilename Generate a drill hole Gerber file named filename.
- H | -? Display the following short help screen and exit.
- Usage: FPCfab [options]
- FPCfab command options[1]: (default value)
- | | | |
|-------------|------------------------|-------------------------------|
| -Afilename | Annotate file name | (-empty-) |
| -Cfname | Font file name | ("font.xml") |
| -Dfname | Drill file name | ("drill_file.drl") |
| -En | Exclude feature n | [3] |
| | n: | 1 Outline Dimensions |
| | | 2 Drill table |
| | | 3 Drill icons |
| | | 4 Board outlines |
| -Ffname | Fab dwg file name | ("FPC_fab.gbr") |
| -Gfname | Drill-Gerber name | (-empty-) |
| -H -? | Help - this screen | |
| -Ix.xx,y.yy | Icon table location | (3.000, 3.000) [2] |
| -O | Template Outline only | (Off) |
| -Tfname | Template file name | ("board_outline.grb") |
| -Ud.ddd | Unplated hole size | (-none-) [3] |
| -V | Verbose output | On |
| -X | Font and graphics test | (font test string → test.gbr) |
- 1 - Not case sensitive
- 2 - Alt form with space must be quoted: "-Ix.xxx y.yyy"
- 3 - Multi entries ok: -u0.093 -u0.115 ...
- Ix.xxx, y.yyy Relocate the Drill Table to X.XXX, Y.YYY. This is the location, in inches, of the upper left corner of the table relative to the template file origin.
- O Extract only the board outline from the template file. Allows using any FreePCB Gerber file as a template.
- Tnewname Replace the default template file name board_outline.grb with newname.
- Ud.ddd Assigns any drill with Diameter = d.ddd to be listed as unplated. No error is generated if a drill of the specified diameter is not found. This parameter may be entered multiple times to specify additional sizes.
- V Enable verbose output. Mostly for debug, this parameter enables the listing of data accumulated and generated while scanning and processing the input files.
- X Generate a font and graphics test drawing Gerber file named test.gbr. Another debug option.

Template Files

The template file is used to define the board outline and Gerber format. In **FreePCB**, the format is currently fixed at 2.4 but this may change in the future. The template may contain additional data but copper areas within the outline should be avoided because **FPCfab** does not do any voiding prior to adding drill targets.



Sample Template File

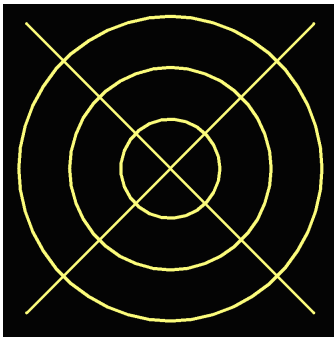
In the sample template shown above, the only thing added to the otherwise empty layer is a copy of the board title text. The copy is scaled and located to match the text on the top silkscreen.

The **-O** parameter limits template data extraction to the board outline only thus allowing any Gerber file containing a board outline to be used as the template.

Later versions of FreePCB can generate an outline only Gerber file named **board_outline.grb** that is ideal for use as a template file and it is now the **FPCfab** default template name.

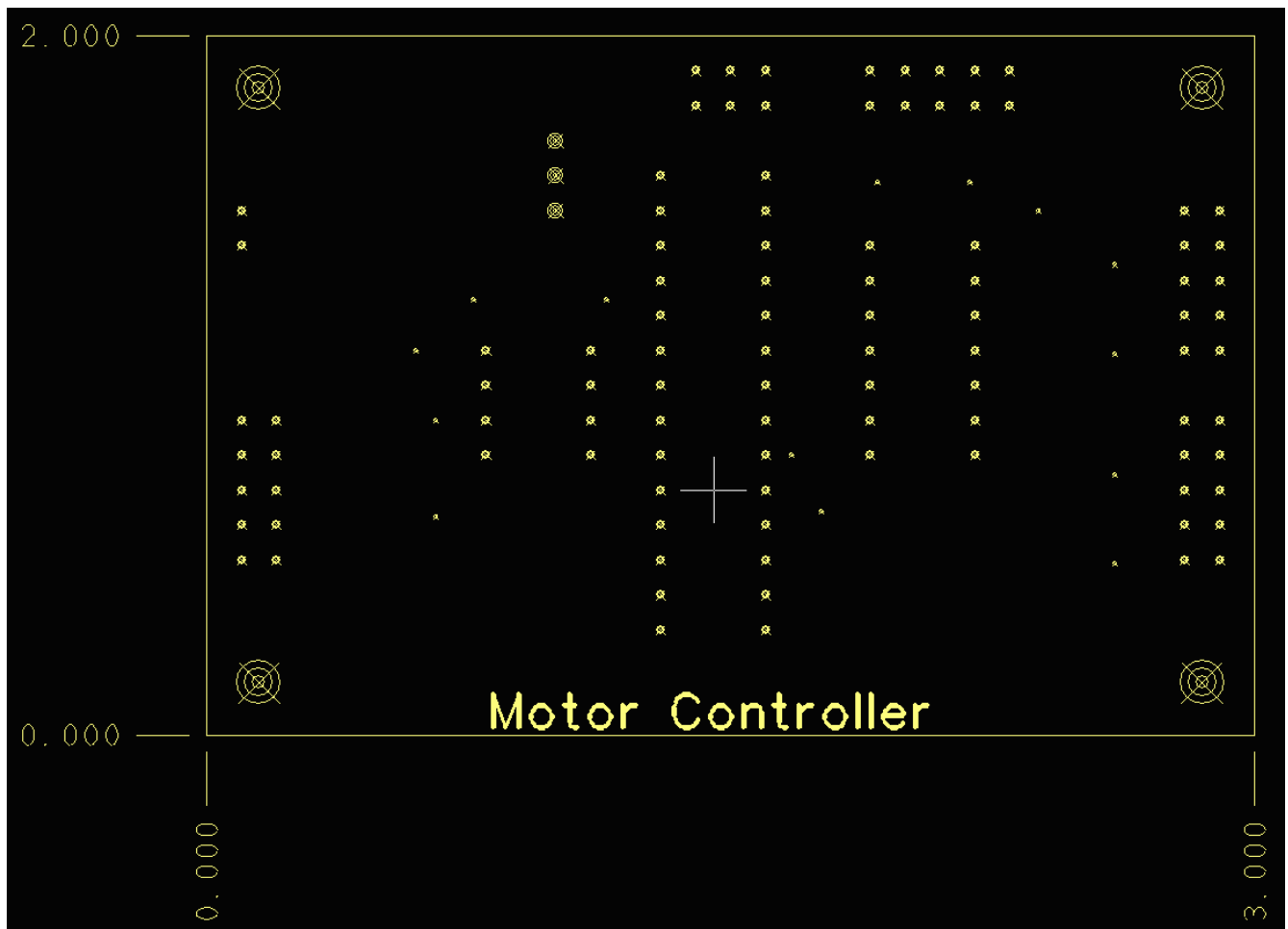
Fabrication Drawing File

FPCfab copies some or all of the data from the template file into the fab file, adds aperture macros and definitions as needed, dimensions the board outline extents and board area, adds drill targets within the outline as defined in the drill file and builds the drill icon table.



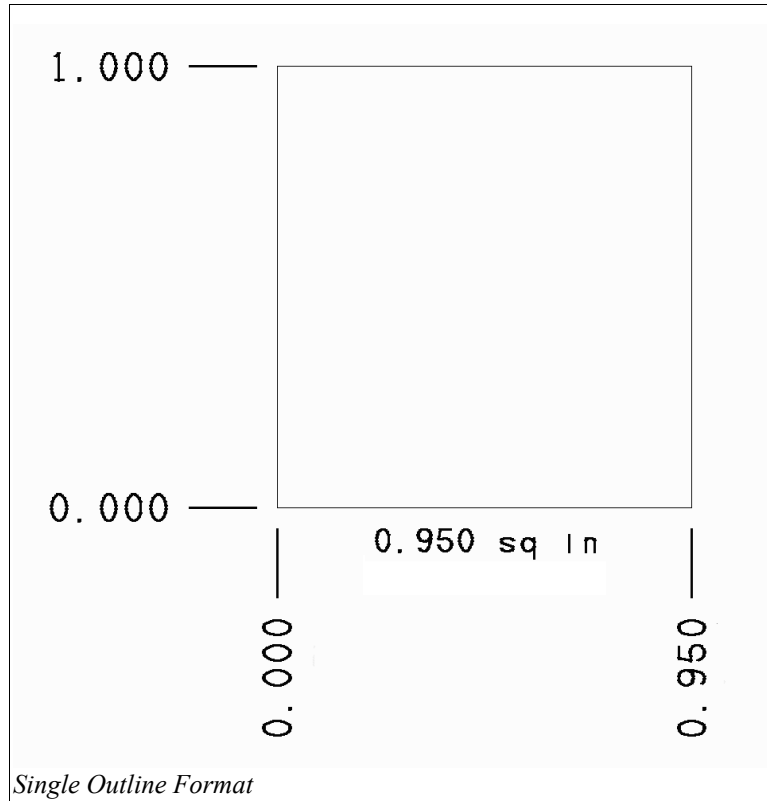
Drill Target Closeup

Each hole size uses a target, as shown at left, consisting of three rings and a set of cross-hairs. The rings are scaled to be equal to $3/3$, $2/3$ and $1/3$ the hole diameter and the lines are scaled to $4/3$. All line widths are scaled to 1% of the diameter. Thus, all targets, when zoomed into, will look alike even though, when viewing the whole board or table, it may appear that detail in smaller targets is lost.

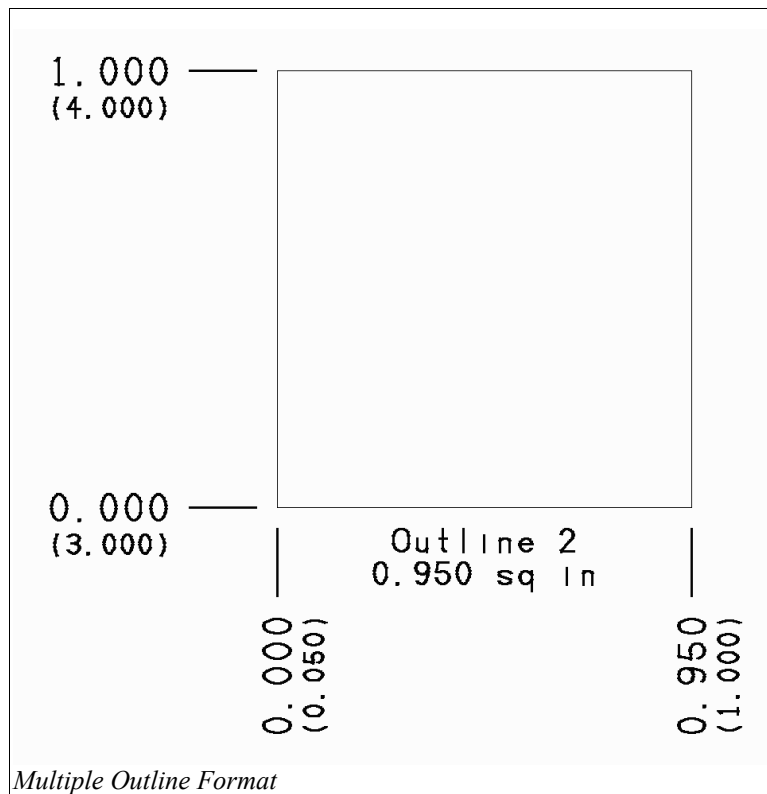


Board Outline with Dimensions and Targets Added

FreePCB ver 1.312 has added the capability of having more than one board outline in a design. **FPCfab** has been updated to list and dimension each outline along with its area.



If only one outline is present, the outline is dimensioned as before.



In the case where two or more outlines are present, each outline is labeled in the order found in the template file. Also, the outline dimensions are expanded to include absolute location values, in parenthesis, as well as board relative values.

Annotation Script Files

Both simple and complex graphic objects as well as multi line text can be added to the fab drawing by including an annotation file. An annotation file is a list of single character commands with parameters that add individual objects.

The simple objects that can be created are lines, boxes, circles and arcs. More complex objects, notes and ordinal dimensions, combine one or more lines and text. Two dimensional objects like boxes and circles can be drawn using a variety of line widths or as solid polygons.

Multiple lines of text can be added by a single command. The text can be of any size, rotated in 90 degree increments and use left, center or right justification.

The command character must be in column one to be recognized. Blank lines and lines with whitespace or semicolons in column one are treated as comments and ignored. Unrecognized command characters are also ignored. Most all commands use one or more parameters separated by whitespace and are NOT case sensitive. See Appendix B for a complete example.

Graphic Object Commands:

Box	B aper fill wide offset x0 y0 x1 y1
Circle	C aper x0 y0 step dia [Y-dia [style start end]]
Dimension	D angle x0 y0 text
Line	L aper x0 y0 x1 y1
Note	N angle length size x0 y0 x1 y1 text
Text	T angle aper size just x0 y0 <i>term</i> {text1 <i>term</i> {multi line text (16K max) <i>term</i> }}

Control Commands:

Adjust Text	A aspect [gap [lspace]]
End file	E
Include File	I filename
Offset	O X-offset Y-offset [Scale]
Polygon	P start/stop
Set Polarity	S clear/dark
User Control	U ParameterSelect Value [Value]

Some commands share common parameters to select aperture size, text angle and text justification. These values are summarized below:

aper:	0 = 0.001	angle:	0 = right	just:	0 = left
	1 = 0.002		1 = up		1 = center
	2 = 0.004		2 = left		2 = right
	3 = 0.008		3 = down		3 ¹ = 1st line right, others left on 1
	4 = 0.016				
	5 = 0.032				
	6 = 0.064				
	7 = 0.128				

¹ Justify code 3 allows a multi line text block to remain left justified while aligning the block on the right end of the first line.

A – Adjust Text

FPCfab uses a fixed pitch font with the text size (height) specified in inches. Character width is set as a function of height and is referred to as the width to height or aspect ratio. Normal text uses an aspect ratio in the range of 0.60 to 0.75 with 0.70 being the default value. The Adjust Text A command allows the aspect ratio to be modified to any value between 0.10 and 4.00 as a means of creating compressed or extended text. The character to character spacing can also be adjusted to optimize spacing and appearance. The gap spacing, normally set 0.15, can be varied from 0.0 to 4.0 again as a function of text size. Line to line spacing is also scaled to text height with normal spacing set to 1.50 to provide a 50% gap between lines. Line spacing can be varied from 0.0 to 4.0.

Format:

A aspect [gap [lspace]]

aspect – aspect ratio as a decimal value.

gap – optional character to character space as a decimal fraction of character height.

lspace – optional line to line spacing as a decimal fraction of character height.

Examples:

A .25	Extra condensed. Gap and line space unchanged.
A .4 .15	Condensed with nominal gap. line spacing unchanged.
A .9 .30 2.5	Extended with doubled character space and expanded line spacing

B – Box

Create a box based on the coordinates of two diagonally opposed corners. Note that the corner ordering is unimportant as long as they are diagonally opposed. A number of patterns are available for box fill as well as border size.

Format:

B aper fill wide offset X0 Y0 X1 Y1

aper – aperture select code (0 – 7)

fill – fill pattern select code

0 – open

1 – diagonal lines (\\)

2 – diagonal lines (//)

3 – crosshatch (both 1 and 2)

4 – horizontal lines

5 – vertical lines

6 – horizontal/vertical hatch (4 and 5)

7 – solid fill (overlapping horizontal lines)¹

wide – BOOLEAN flag. 1 = draw three boxes, one inside and one outside the specified coordinates. 0 = draw a single box.

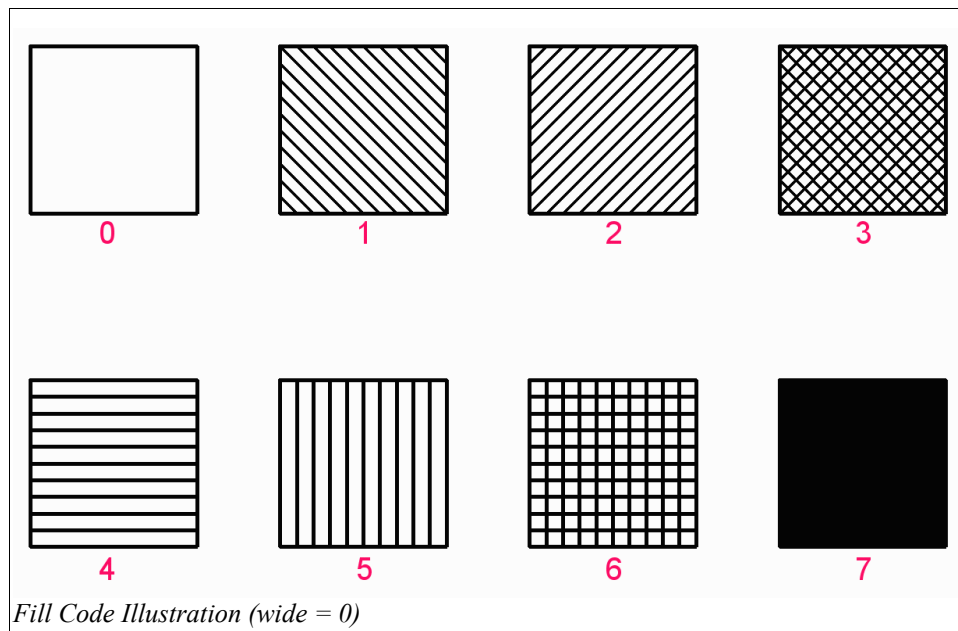
offset – distance between boxes when wide is TRUE as a decimal value.

X0 Y0 – first corner location

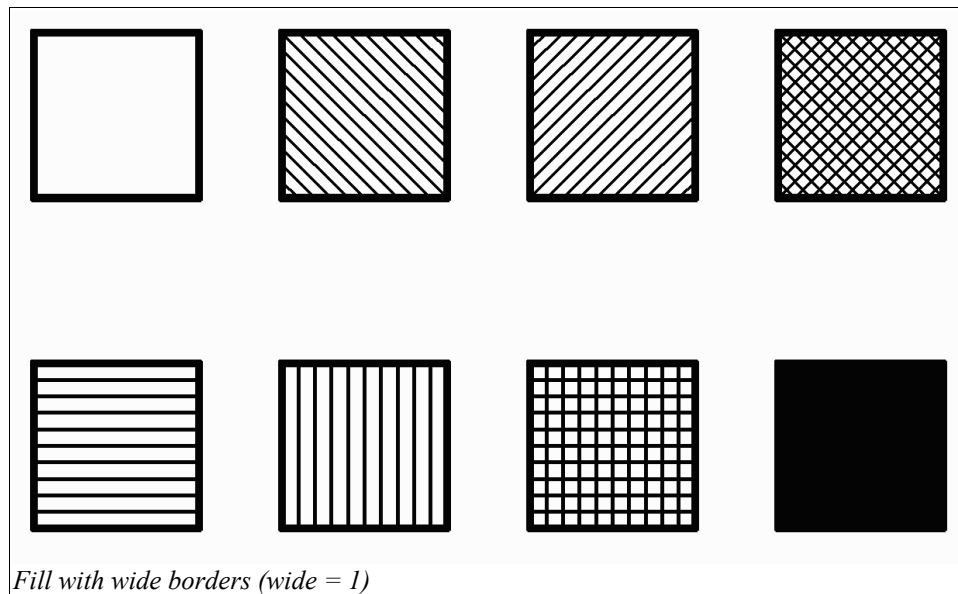
X1 Y1 – second corner location

¹ This fill method is not recommended, see the Polygon command.

Examples:



B 4 0 0 0 0.0 0.0 1.0 1.0
 B 4 1 0 0 0.0 0.0 1.0 1.0
 B 4 2 0 0 0.0 0.0 1.0 1.0
 B 4 3 0 0 0.0 0.0 1.0 1.0
 B 4 4 0 0 0.0 0.0 1.0 1.0
 B 4 5 0 0 0.0 0.0 1.0 1.0
 B 4 6 0 0 0.0 0.0 1.0 1.0
 B 4 7 0 0 0.0 0.0 1.0 1.0



B 4 0 1 .012 0.0 0.0 1.0 1.0
 B 4 1 1 .012 0.0 0.0 1.0 1.0
 B 4 2 1 .012 0.0 0.0 1.0 1.0
 B 4 3 1 .012 0.0 0.0 1.0 1.0
 B 4 4 1 .012 0.0 0.0 1.0 1.0
 B 4 5 1 .012 0.0 0.0 1.0 1.0
 B 4 6 1 .012 0.0 0.0 1.0 1.0
 B 4 7 1 .012 0.0 0.0 1.0 1.0

C – Circle

The circle command can generate not only circles but ellipses, regular polygons, arcs and pie sections. In its simplest form, the center location and diameter define a circle or regular polygon depending on the angular step size selected. Additional optional parameters can be used to set the Y-axis diameter as well as the starting and ending angle and arc style.

Format:

C aper X0 Y0 step dia [Y-dia [style start end]]

aper – draw aperture select code

X0 Y0 – center location

step – angular step size expressed in decimal degrees within the range of -360.0 to -0.1 or +0.1 to +360.0 degrees.

dia – circle diameter in decimal inches

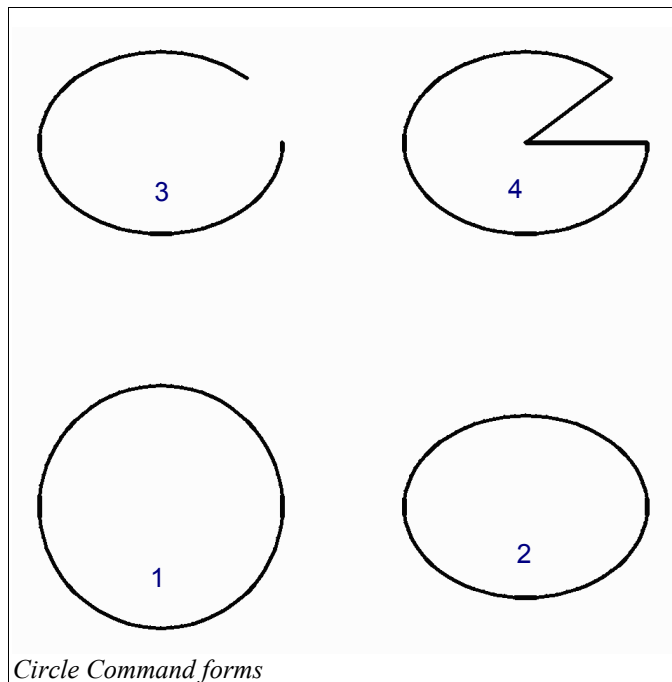
Y-dia – Y-axis diameter in decimal inches

style – BOOLEAN arc termination select. With style FALSE, only the arc segments are drawn. When TRUE, additional line segments are drawn from the center to the starting and ending arc segments forming a pie section.

start – starting angle in decimal degrees within the range of -720 to +720 degrees.

end – ending angle in decimal degrees within the range of -720 to +720 degrees.

Examples:



1: C 2 1.0 1.0 5.0 1.0

2: C 2 1 1 5 1 0.75

3: C 2 1 1 5 1 0.75 0 45.0 360

4: C 2 1 1 5 1 0.75 1 45 360

Not shown:

C 2 1 1 90 1 2

1x2 diamond

C 2 1 1 72 1 1 0 36 396

regular pentagon

D - Ordinal Dimension

The dimension command combines a short line segment and text string to form an ordinal dimension mark. The combination can be rotated in 90 degree steps to 0, 90, 180 and 270 degrees around the X,Y specified point. While the combination can be rotated up to 270 degrees, the text is only rotated 0 or 90 degrees to remain readable. A simple text string without whitespace can be used directly. Text with whitespace, quotes or newlines must be enclosed within quotation marks.

Format:

D angle X0 Y0 text

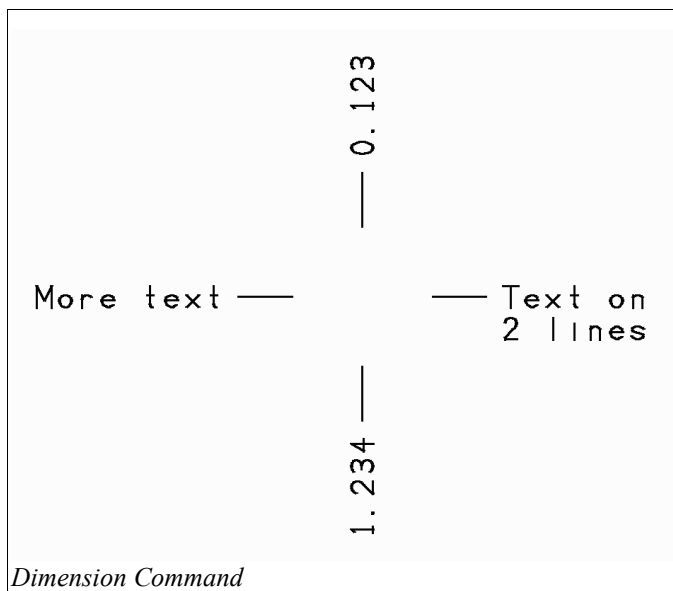
angle – rotation angle code.

X0,Y0 – line segment outer end location in decimal inches.

text – included string. Strings with embedded whitespace, quotes or newlines must be quoted.

Use \" for included quote, \n for newline. Use two quotes "" (empty string) for no text.

Examples:



D 0 1.2 1.0 "Text on\n2 lines"

D 1 1.0 1.2 0.123

D 2 0.8 1.0 "More text"

D 3 1.0 0.8 1.234

E – End file

Normally, annotation file processing continues until the end of file is reached. Use this command to terminate a file prior to EOF.

I - Include

Include a file of annotation commands. All commands are valid in an included file including additional include commands which can be nested up to 32 levels deep.

Any Offset commands in the included file are relative to and scaled by the parent file settings. This applies to scaling as well as offsets and remain in effect only until the end of the included file. When the included file is finished, and the parent file resumes control, the parent files offset and scale settings prior to executing the include command are restored.

Format:

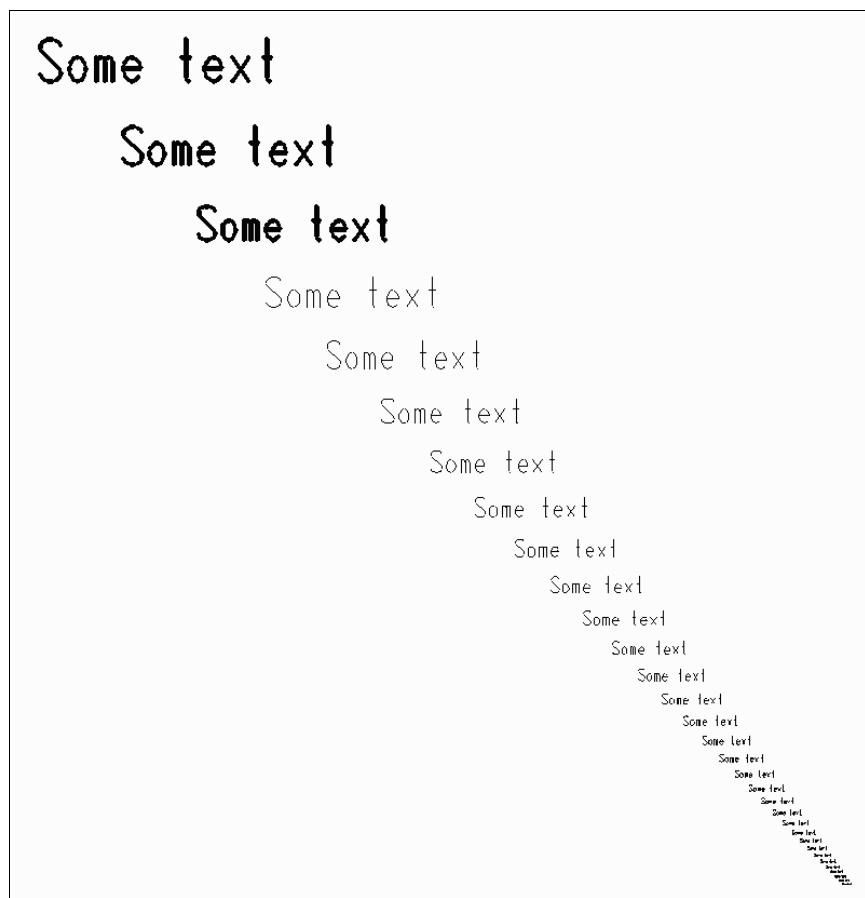
I filename

filename – file to be included. If the file name contains spaces, it must be quoted.

Examples:

```
O -2.5 4.7 0.50      ; place the title block at a specified location
I titleblock.ani      ; add it
```

```
O 4.7 -2 .75         ; locate the table
I "Layer table 8.ani" ; quoted file name with embedded spaces
```



An example of recursion:

File test2.ani, included in the root file, recursively invokes its self until the maximum depth is reached and the include command is ignored.

```
O 0.2 -.2 .9
T 0 3 .1 0 0 0 ^
Some text^
I test2.ani
```

L - Line

Draw a line segment from P{X0,Y0} to P{X1,Y1} with the selected aperture,

Format:

L aper X0 Y0 X1 Y1

Example:

L 1 1.235 2.5 1.34 .5

N - Note

The note command combines the creation of an arrow tipped line segment defined by two points with a adjustable length leader line and a text message. The leader angle can be rotated to 0, 90, 180 and 270 degrees. While the leader can be rotated up to 270 degrees, the text is only rotated 0 or 90 degrees to remain readable. A simple text string without whitespace can be used directly. Text with whitespace, quotes or newlines must be enclosed within quotation marks. Leaders less than 0.5 mil are not drawn to avoid zero length segment issues. Hint: use this command with a zero length leader and empty text string to draw arrow tipped line segments.

Format:

N angle length size X0 Y0 X1 Y1 text

angle – leader segment angle code

length – leader length in decimal inches

size – text height in decimal inches

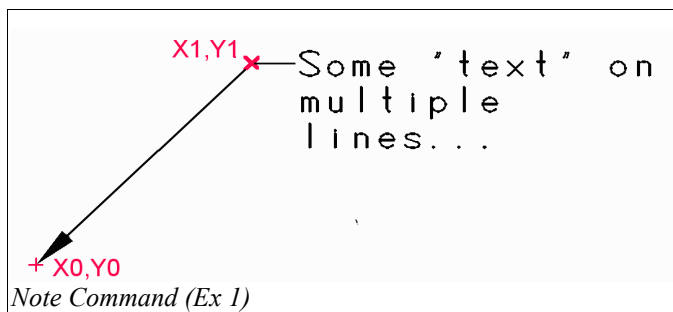
X0 Y0 – arrowhead tip location in decimal inches

X1 Y1 – line-leader junction location in decimal inches

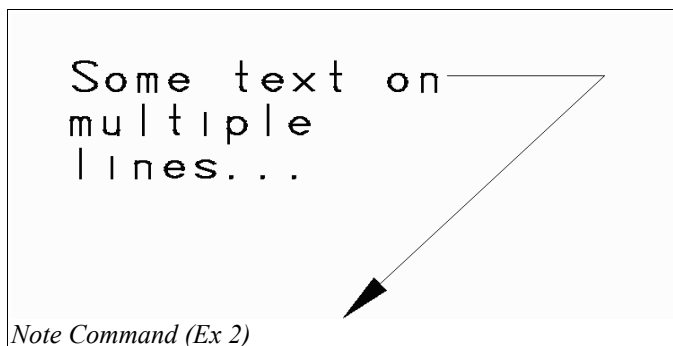
text – included string. Strings with embedded whitespace, quotes or newlines must be quoted.

Use \" for included quote, \n for newline. Use two quotes "" (empty string) for no text.

Examples:



N 0 .15 .1 1 1 2 2 "Some \"text\" on\nmultiple\nlines..."



N 2 .45 .1 1 1 2 2 "Some text on\nmultiple\nlines..."

O - Offset

The offset command relocates the data entry origin relative to the template file data. The optional scale factor can be used to adjust the size of subsequent objects. The offsets are added to all scaled values when generating the Gerber data. Use this command to simplify entry of tables and title blocks or other groups of objects; if a group of objects is preceded by an offset command, the whole group can be resized and relocated easily by editing only the offset command.

Format:

O X-offset Y-offset [Scale]

X-offset Y-offset – offset values in decimal inches

Scale – effects all linear dimensions directly. Apertures are increment or decrement at $n\sqrt{2}$ scale intervals.

The available set of drawing apertures has been expanded and modified. The list is now a regular progression of diameters from 1 to 128 mils. This was done to facilitate scaling and may effect code written for version 1.30. A simple, one time edit may be needed.

Given the old and new sizes:

Ver 1.30	Ver 1.32
Apertures	Apertures
0 = 0.001	0 = 0.001
1 = 0.005	1 = 0.002
2 = 0.015	2 = 0.004
3 = 0.030	3 = 0.008
	4 = 0.016
	5 = 0.032
	6 = 0.064
	7 = 0.128

The following table will give nearly equivalent results:

1.30	1.23
Code	Code
1 →	2
2 →	4
3 →	5

Offset commands in an included file are scaled by and relative to the parent file. All parameters are adjusted by the scale factor in effect in the parent file at the time the file is included and offsets are relative to the parents origin. When an included file completes and the parent file resumes control, the parent files previous settings are restored.

P - Polygon

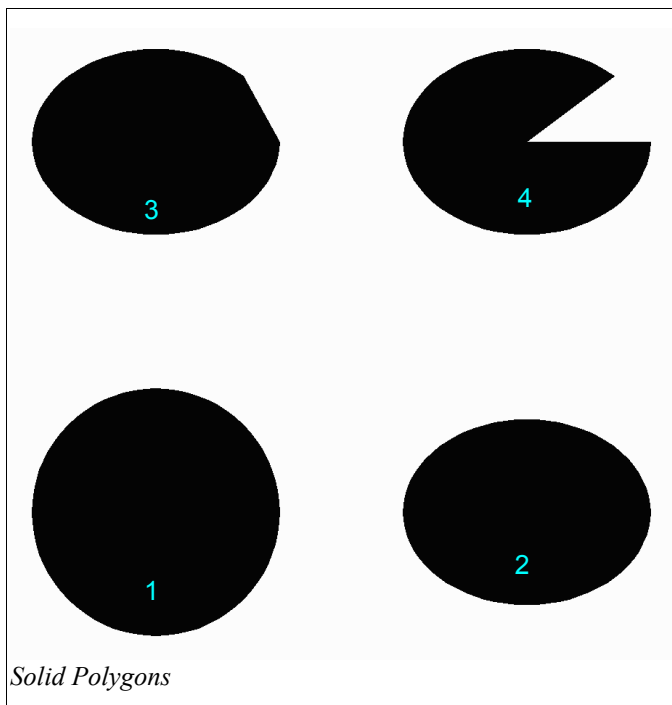
The polygon command inserts a polygon start (G36*) or polygon end (G37*) instruction into the Gerber data. This command will convert an open box or circle into a solid filled figure of the same shape. Apertures are not used when a polygon is drawn so an object drawn as a polygon will be smaller than the same open figure drawn by line segments.

Format:

P sel

sel – BOOLEAN polygon start / stop flag: 1 = start
0 = end

Examples:



```
1: P 1 ; start polygon
   C 1 1.0 1.0 5.0 1.0 ; draw segments
   P 0 ; end polygon

2: P1
   C 1 1 1 5 1 0.75
   P0

3: P1
   C 1 1 1 5 1 0.75 0 45.0 360
   P0

4: P 1
   C 1 1 1 5 1 0.75 1 45 360
   P0
```


S - Set Polarity

Each Gerber file represents a physical layer that consists of one or more data layers. Data layers are classed as dark, marking, or clear where previously marked data is erased. This command starts a new data layer by inserting a Layer Polarity record (%LPx*%) into the Gerber file.

Format:

S sel

sel – polarity select: 1 = clear
0 = dark

Example:



```
P 1  
B 2 0 0 0 0.0 0.0 2.0 2.0 ; dark 2 inch square as a polygon  
P 0  
  
S 1 ; start a new clear data layer  
T 0 2 .15 1 1.0 0.925 ^KNOCKOUT^ ; write text as clear  
S 0 ; start a new dark data layer
```

T - Text

This command can add short single line text strings that do not contain embedded whitespace or long multi line messages. The first character of the inline string is saved as the termination character and not output. If a second occurrence of the termination character is found in the inline string, the intervening sub-string is output and the command is done. If a second occurrence is not found, additional lines are read and loaded (without comment parsing) until the term character is found (or the 16K buffer becomes full).

Any printable character can be used as the string terminator; its only requirement is that it not appear within the text block. Characters like tilde '~', caret '^' and vertical line '|' are unlikely to appear within normal text and so make good terminator candidates.

Format:

T angle aper size just X0 Y0 *term*{text1*term* | {multi line text (16K max)*term*}}

angle, aper and size – per previously definitions

just – text line justification:	0 = Left	left end of line @ X0
	1 = Center	center of line @ X0
	2 = Right	right end of line @ X0
	3 = 1 st line = Right, all others Left	on left end of 1 st line

X0 Y0 – text location (bottom left, center or right of first line)

term – input termination character (first character if inline string)

text1 – optional additional inline text. The inline text can contain \n newline but not \"

multi line text – additional lines as needed up to 16K characters

term – the second occurrence ends input and sends the text to the output

Examples:

T 0 1 0.1 0 3.0 4.0 ^Short_Message^ ; inline string without any whitespace

T 0 1 .1 0 1.0 -0.3 ^ ; multi line message

Top Silkscreen	top_silk.grb
Top Soldermask	top_mask.grb
Top Signal	top_copper.grb

Inner 1 Signal	inner_copper_1.grb
----------------	--------------------

Inner 2 Signal	inner_copper_2.grb
----------------	--------------------

Inner 3 Signal	inner_copper_3.grb
----------------	--------------------

Inner 4 Signal	inner_copper_4.grb
----------------	--------------------

Inner 5 Signal	inner_copper_5.grb
----------------	--------------------

Inner 6 Signal	inner_copper_6.grb
----------------	--------------------

Bottom Signal	bottom_copper.grb
Bottom Soldermask	bottom_mask.grb
Bottom Silkscreen	bottom_silk.grb^

U – User Control

This command uses two or three parameters and provides control of some of the system parameters previously only available as command line options. It also adds a measure of control over the drill icon table text.

Format:

U Psel value [value]

Psel – parameter select (1 – 9)

value – new parameter value

The first four versions of this command duplicate the the functionality of the command line *Exclude* switches to control automatically generated drawing features. If the parameter *e* is equal to zero, the feature is omitted from the drawing; if it is not equal to zero, the feature is drawn. These commands will override their command line equivalents.

U 1 e – Include/Exclude outline dimensions

U 2 e – Include/Exclude the drill table

U 3 e – Include/Exclude drill icons

U 4 e – Include/Exclude board outlines

Ex: **U 2 0** – Do not draw the drill table
U 3 1 – Draw drill icons

The next three versions will increase or decrease the stroke width (aperture size) of various text fields in the drill table. The parameter *n* is limited to the range of -3 to +3 ($-3 \leq n \leq +3$) and, if entered outside this range, will be forced into range. These commands can be entered multiple time to exceed the ± 3 step limit but the resulting aperture selection is still limited to the available 0-7 range.

U 5 n – Increase/Decrease drill table title stroke width

U 6 n – Increase/Decrease drill table column heading stroke width

U 7 n – Increase/Decrease drill table data stroke width

Ex: **U 6 1** – Increase the stroke width (aperture size) of the column heading text by one step
U 7 -1 – Decrease the stroke width of the table data by one step

The next version duplicates the functionality of the command line *-U* switch to define unplated hole sizes. Like its command line equivalent, this command may be entered as many times as needed to define multiple sizes.

U 8 d.ddd – Define drill diameter *d.ddd* as unplated

Ex: **U 8 0.072** – Define 72 mil holes as unplated
U 8 .125 – Define 125 mil holes as unplated

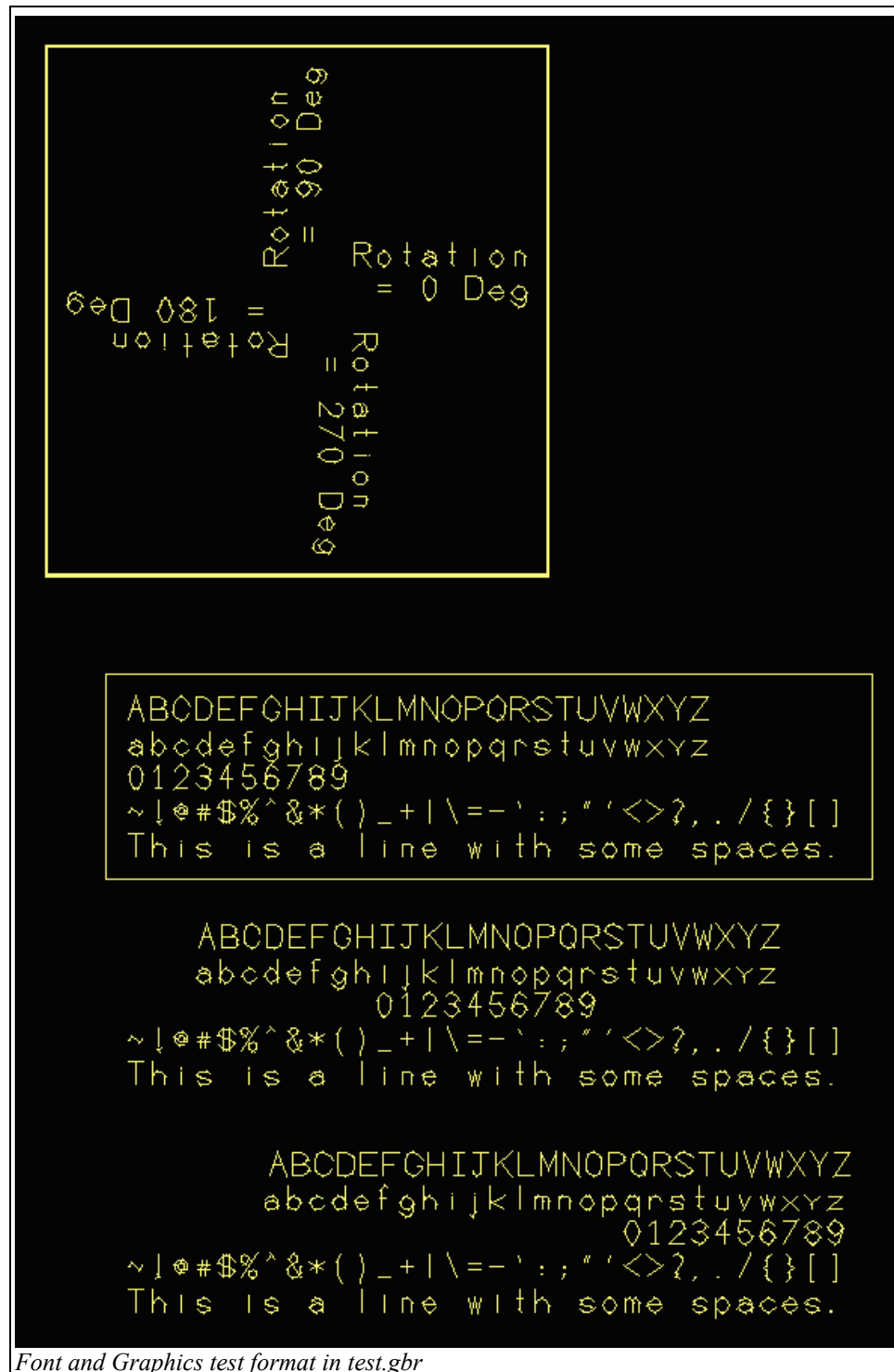
The last version duplicates the functionality of the command line *-I* switch to specify the location of the upper left corner of the drill table. If used, this command will override its command line equivalent.

U 9 x.xxx y.yyy – Drill icon table location

Ex: **U 9 6 2.52** – Place the drill table at (6.000, 2.520)

Appendix A: Fonts

The vector font used by **FPCfab** was created with *OpenOffice Draw*. The resulting Font1.odg file is, in fact, a zip archive containing a number of XML and other files. The content.xml file was extracted and renamed font.xml for use with **FPCfab**. The -X command line parameter can be used to verify any font changes by generating a Gerber check file named *test.gbr*.



Text Angle Check

Wide Line Box Check

Full Font

Left Justify Check

Box Check

Center Justify Check

Right Justify Check

[illegible]

FPCfab V1.40

Adapter.ano:

```
=====
;
;
;               FPCfab Ver 1.40 Annotation File template
;
;===== Objects =====
;
; Box           B aper fill wide offset x0 y0 x1 y1
;
;               fill      0 = none
;                       1 = right-down \\\ hatch
;                       2 = right-up   /// hatch
;                       3 = crosshatch (1 + 2)
;                       4 = horizontal lines ---
;                       5 = vertical lines |||
;                       6 = crosshatch (4 + 5)
;                       7 = horizontal w/ solid overlap ====
;                           (use polygon with no fill for smaller files)
;
;               wide:     boolean draw 3 boxes flag
;
;               offset:   delta size of outer and inner box
;
; Circle        C aper x0 y0 step dia [Ydia [style start end]]
;
;               step:     arc segment size in degrees
;
;               dia:      X axis diameter [and Y axis if Ydia not present]
;               Ydia:     optional Y axis diameter
;
;               style     0 = open segment
;                       1 = closed to center (pie circle)
;
; Dimension     D angle x0 y0 text
;               text:     quoted for embedded spaces, \" for quote, \n for newline
;
; Line          L aper x0 y0 x1 y1
;
; Note          N angle length size x0 y0 x1 y1 text
;
;               leader angle and length
;               size is text height
;               text:     quoted for embedded spaces, \" for quote, \n for newline
;
; Text          T angle aper size just x0 y0 term[text1[term]] | {multi line text (16K chars max)}term
;
;               size is text height
;               term is last (unprinted) character
;               text1 is short (64 char max), unquoted text. If term not found, add additional lines.
;               No comment parsing done within additional text line until after final term char.
;
;===== Controls =====
;
; Adjust Text   A aspect [gap [lspace]]
;
;               aspect:   character width/height ratio (normal = 0.7)
;               gap:      character to character space/height ratio (normal = 0.25)
;               lspace:   line to line spacing/height ratio (normal = 1.5)
;
;               limits:  aspect 0.1 - 4.0
;                       gap    0.0 - 4.0
;                       lspace 0.0 - 4.0
;
; End file      E      end file processing prior to EOF.
;
; Include       I filename
;
;               file names with included spaces must be quoted: "file name".
;               32 levels of inclusion nesting maximum.
;
; Offset        O Xoffset Yoffset [Scale]
;
;               limits:  Scale 0.10 - 10.0
;
;
;=====
```

```

; Polygon      P start/stop
;
;              1 = start
;              0 = stop
;
; Set Polarity  S clear/dark
;
;              1 = clear
;              0 = dark
;
; User control  U Psel value1 [value2]    (Psel = 1-4,9 override command line values)
;
;              U 1 e - enable/disable outline dimensions (e=0: disable, e!=0: enable)
;              U 2 e - enable/disable drill table
;              U 3 e - enable/disable drill icons
;              U 4 e - enable/disable board outlines
;
;              U 5 n - increase/decrease drill table title text stroke size (-3 <= n <= +3)
;              U 6 n - increase/decrease drill table column heading text stroke size
;              U 7 n - increase/decrease drill table data text stroke size
;
;              U 8 d.ddd - add drill diameter d.ddd to the unplated list
;
;              U 9 x.xxx y.yyy - locate drill table (upper left corner) at (x.xxx, y.yyy)
;
;===== Common =====
;
; All commands MUST start in column 1 or it will be ignored.
;
; Comments:  whitespace {' '\n\t} or a semicolon (;) in column 1 defines a comment line.
;            semicolon (;) after a command starts inline comment. Really only needed after a command
;            with missing optional parameters but helps readability.
;
; Units:     Linear  = INCHS
;            Angular = DEGREES
;
; ----- shared codes and values -----
;
;      Old V1.30      New V1.32
;      Aper Size      Aper Sizes      Angle      Justify
;      -----
;      0 = 0.001      0 = 0.001      0 = right   0 = left
;      1 = 0.005      1 = 0.002      1 = up     1 = center
;      2 = 0.015      2 = 0.004      2 = left   2 = right
;      3 = 0.030      3 = 0.008      3 = down   3 = 1st line right, others left on 1
;
;      4 = 0.016
;      Old -> New      5 = 0.032
;      0 -> 0          6 = 0.064
;      1 -> 2          7 = 0.128
;      2 -> 4
;      3 -> 5
;
;=====
;
;                        Template End
;=====

N 0 .07 .075 4.265 2.375 4.5775 2.6875 "0.050 Chamfer\n4 PL" ; board relative detail

O -3.5 -6 ; place board org @ 3.5,6 within page

I "B size.ani" ; include page outline

;O 2.05 -5.85 1.0 ; lower left corner (1x, B size -> B size)
O 7.70 -5.85 0.5 ; lower left corner (.5x, B size -> D size)

I "Pg1 Title Blk.ani" ; page 1 title block

;O 6.4 -.5 0.75 ; B size (reduced to allow room for 1x title block)
O 6.4 -.5 1.0 ; D size (full size)

I "Layer Tbl 8.ani" ; Layer Order table

;===== end =====

```

B size.ani:

```
; ===== B Size Sheet Layout Master =====

B 1 0 0 0 0 0 17 11 ; thin outline
B 4 0 0 0 0.15 0.15 16.85 10.85 ; 0.15 inch bold border

L 3 2.125 0 2.125 .15 ; zone marks
L 3 4.250 0 4.250 .15
L 3 6.375 0 6.375 .15
L 3 8.500 0 8.500 .15
L 3 10.625 0 10.625 .15
L 3 12.750 0 12.750 .15
L 3 14.875 0 14.875 .15 ; bottom
P 1
C 0 8.50 0.1125 120 0.075 0.075 0 -270 90
P 0
; Circle C aper x0 y0 step dia [Ydia [style start end]]

L 3 0 2.75 0.15 2.75 ; left
L 3 0 5.50 0.15 5.50
L 3 0 8.25 0.15 8.25
P 1
C 0 0.1125 5.5 120 0.075 0.075 0 0 360
P 0

L 3 2.125 11.00 2.125 10.85 ; top
L 3 4.250 11.00 4.250 10.85
L 3 6.375 11.00 6.375 10.85
L 3 8.500 11.00 8.500 10.85
L 3 10.625 11.00 10.625 10.85
L 3 12.750 11.00 12.750 10.85
L 3 14.875 11.00 14.875 10.85
P 1
C 0 8.50 10.8875 120 0.075 0.075 0 -90 270
P 0

L 3 16.85 2.75 17.00 2.75 ; right
L 3 16.85 5.50 17.00 5.50
L 3 16.85 8.25 17.00 8.25
P 1
C 0 16.8875 5.50 120 0.075 0.075 0 -180 180
P 0

T 0 2 .08 1 1.0625 .035 ^8^ ; bottom
T 0 2 .08 1 3.1875 .035 ^7^
T 0 2 .08 1 5.3125 .035 ^6^
T 0 2 .08 1 7.4375 .035 ^5^
T 0 2 .08 1 9.5625 .035 ^4^
T 0 2 .08 1 11.6875 .035 ^3^
T 0 2 .08 1 13.8125 .035 ^2^
T 0 2 .08 1 15.9375 .035 ^1^

T 0 2 .08 1 1.0625 10.885 ^8^ ; top
T 0 2 .08 1 3.1875 10.885 ^7^
T 0 2 .08 1 5.3125 10.885 ^6^
T 0 2 .08 1 7.4375 10.885 ^5^
T 0 2 .08 1 9.5625 10.885 ^4^
T 0 2 .08 1 11.6875 10.885 ^3^
T 0 2 .08 1 13.8125 10.885 ^2^
T 0 2 .08 1 15.9375 10.885 ^1^

T 1 2 .08 1 .115 9.625 ^D^ ; left
T 1 2 .08 1 .115 4.125 ^B^
T 1 2 .08 1 .115 6.875 ^C^
T 1 2 .08 1 .115 1.375 ^A^

T 1 2 .08 1 16.965 9.625 ^D^ ; right
T 1 2 .08 1 16.965 4.125 ^B^
T 1 2 .08 1 16.965 6.875 ^C^
T 1 2 .08 1 16.965 1.375 ^A^

;===== end =====
```


Pg1 Title Blk.ani:

```

===== Std Pg 1 Title Block =====

B 4 0 0 0 0.6 0.0 11.3 2.4      ; big box
B 4 0 0 0 0.0 0.0 0.6 0.5      ; small box
L 4 2.6 0.0 2.6 2.4            ; heavy lines
L 4 4.8 0.0 4.8 2.4
L 4 7.0 0.0 7.0 2.4
L 4 0.6 0.2 2.6 0.2
L 4 2.6 0.9 4.8 0.9
L 4 4.8 1.60 7.0 1.6
L 4 7.0 1.7 11.3 1.7
L 4 7.0 0.75 11.3 0.75

L 2 1.6 0.2 1.6 2.4            ; fine lines
L 2 0.6 0.4 2.6 0.4
L 2 0.6 0.6 2.6 0.6
L 2 0.6 0.8 2.6 0.8
L 2 0.6 1.0 2.6 1.0
L 2 0.6 1.2 2.6 1.2
L 2 0.6 1.4 2.6 1.4
L 2 0.6 1.6 2.6 1.6
L 2 0.6 1.8 2.6 1.8
L 2 0.6 2.0 2.6 2.0
L 2 0.6 2.2 2.6 2.2
L 2 2.6 0.0 4.8 0.9
L 2 4.8 0.2 6.3 0.2
L 2 4.8 0.4 6.3 0.4
L 2 4.8 0.6 6.3 0.6
L 2 4.8 0.8 6.3 0.8
L 2 4.8 1.0 6.3 1.0
L 2 4.8 1.2 6.3 1.2
L 2 4.8 1.4 6.3 1.4
L 2 6.4 0.2 7.0 0.2
L 2 6.4 0.4 7.0 0.4
L 2 6.4 0.6 7.0 0.6
L 2 6.4 0.8 7.0 0.8
L 2 6.4 1.0 7.0 1.0
L 2 6.4 1.2 7.0 1.2
L 2 6.4 1.4 7.0 1.4
L 2 7.0 0.24 11.3 0.24
L 2 7.43 0.24 7.43 0.75
L 2 8.44 0.24 8.44 0.75
L 2 10.98 0.24 10.98 0.75
L 2 8.2 0.0 8.2 0.24
L 2 9.9 0.0 9.9 0.24

A 0.5 0.15
T 0 2 0.1 1 0.3 0.275 ^
XYZ USER
DOCUMENT^
A 0.5 0.2
T 0 2 0.08 1 1.6 0.06 ^APPLICATION^
T 0 2 0.08 1 1.1 0.26 ^
NEXT ASSEMBLY^
T 0 2 0.08 1 2.1 0.26 ^
USED ON^
A .5 .2 1.8
T 0 2 0.10 1 3.7 2.25 ^
UNLESS OTHERWISE SPECIFIED^
T 0 2 0.075 1 3.7 2.11 ^
DIMENSIONS ARE IN INCHES
TOLERANCES^
T 0 2 0.075 1 3.7 1.8 ^
DECIMALS          ANGLES^
T 0 2 0.075 1 3.7 1.6 ^
.XX .03 .XXX .010      1/2      ^
T 0 2 0.075 0 3.0 1.35 ^
MACHINE FINISH^
T 0 2 0.075 1 3.7 1.10 ^
DO NOT SCALE DRAWING
INTERPRET DRAWING PER ASME Y14.100M^
T 0 2 0.075 0 2.7 .75 ^MATERIAL^

L 2 2.91 1.645 2.97 1.645      ; +/-
L 2 2.94 1.615 2.94 1.675
L 2 2.91 1.600 2.97 1.600

```

```

L 2 3.49 1.645 3.55 1.645 ; +/-
L 2 3.52 1.615 3.52 1.675
L 2 3.49 1.600 3.55 1.600

L 2 4.13 1.645 4.18 1.645 ; +/-
L 2 4.15 1.615 4.15 1.675
L 2 4.13 1.600 4.18 1.600

C 2 4.38 1.66 20 0.03 ; degree sym

L 2 4.18 1.3875 4.20 1.3875 ; finish sym
L 2 4.20 1.3875 4.22 1.3575
L 2 4.22 1.3575 4.26 1.4175

A .7 .2
T 0 2 0.075 0 4.85 2.25 ^
CONTRACT NO^
L 2 5.65 2.25 6.90 2.25
A .6 .2 1.5
T 0 2 0.075 0 4.85 2.07 ^
PREPARED BY MTC IN COMPLIANCE WITH
XYZ D-20478 VOLUME 1;
DSMS 813-203, VOLUME 1^
T 0 2 0.075 0 4.85 1.65 ^
APVD DATE^
L 2 5.155 1.65 6.09 1.65
L 2 6.49 1.65 6.93 1.65
A .6 .2 2.6666
T 0 2 0.075 0 4.85 1.463 ^
DWN
CHK
STRU
RF
SAF
PKG
ENGR
DSGN SUPV^
T 0 4 0.18 1 9.15 2.17 ^
The XYZ COMMUNICATIONS CO.^
A 0.65 0.2 1.5
T 0 2 0.07 1 9.15 2.050 ^
A DIVISION OF ABC BIG COMPANY
PASADENA, CA 91234^
T 0 2 0.1 1 9.15 1.75 ^
RELEASED THROUGH EDMG^
A 0.55 0.15
T 0 4 .2 1 9.15 1.30 ^ ; Title
C3I Comm Adapter
Fabrication Drawing^
T 0 2 .06 1 7.215 .65 ^SIZE^
T 0 2 .06 1 7.935 .65 ^
CAGE NO^
T 0 2 .06 1 11.14 .65 ^REV^
T 0 2 .06 0 7.05 .09 ^SCALE^
T 0 2 .06 0 9.95 .09 ^
SHEET OF^
A 0.8 .2
;T 0 5 .18 1 7.215 .365 ^B^ ; page size
T 0 5 .18 1 7.215 .365 ^D^
T 0 5 .18 1 11.14 .365 ^B^ ; Rev letter
A 0.7 .15
T 0 4 .12 1 7.935 .385 ^12345^
T 0 4 .15 1 9.71 0.37 ^9XXXXXX^ ; dwg #
A 0.6 .15

;T 0 3 .15 1 7.715 .053 ^1:1^ ; B size scale value
T 0 3 .15 1 7.715 .053 ^2:1^ ; D size scale value

T 0 3 .15 1 9.05 .053 ^UNCLASSIFIED^
T 0 3 .15 1 10.65 .053 ^
1 1^

T 0 3 .1 0 5.15 1.45 ^
B PARHAM 18 AUG 06^

;===== end =====

```

Layer Tbl 8.ani:

```
;===== Layer Order table (8) =====

B 4 0 0 0 -0.4 0.2 3.65 -3.22
L 2 -.4 0 3.65 0

A .7 .25
T 0 3 .10 1 1.625 .05 ^
LAYER ORDER^

L 0 -.2 -.25 0 -.25
L 0 -.2 -2.95 0 -2.95
N 3 .1 .1 -.1 -.25 -.1 -1.4 ""
N 1 .1 .1 -.1 -2.95 -.1 -1.8 ""
T 0 1 .06 1 -.1 -1.63 ^0.060^

A .5 .15 ; condensed text

T 0 2 .1 0 1.0 -0.3 ^
Top Silkscreen top_silk.grb
Top Soldermask top_mask.grb
Top Signal top_copper.grb

Inner 1 Signal inner_copper_1.grb
Inner 2 Signal inner_copper_2.grb
Inner 3 Signal inner_copper_3.grb
Inner 4 Signal inner_copper_4.grb
Inner 5 Signal inner_copper_5.grb
Inner 6 Signal inner_copper_6.grb

Bottom Signal bottom_copper.grb
Bottom Soldermask bottom_mask.grb
Bottom Silkscreen bottom_silk.grb^

L 4 .20 -.25 .80 -.25 ; layer lines
L 4 .20 -.40 .80 -.40
L 4 .20 -.55 .80 -.55

L 4 .20 -.85 .80 -.85
L 4 .20 -1.15 .80 -1.15
L 4 .20 -1.45 .80 -1.45
L 4 .20 -1.75 .80 -1.75
L 4 .20 -2.05 .80 -2.05
L 4 .20 -2.35 .80 -2.35

L 4 .20 -2.65 .80 -2.65
L 4 .20 -2.80 .80 -2.80
L 4 .20 -2.95 .80 -2.95

B 2 1 0 0 .25 -.60 .75 -.80 ; fiberglass as hatched boxes
B 2 2 0 0 .25 -.90 .75 -1.10
B 2 1 0 0 .25 -1.20 .75 -1.40
B 2 2 0 0 .25 -1.50 .75 -1.70
B 2 1 0 0 .25 -1.80 .75 -2.00
B 2 2 0 0 .25 -2.10 .75 -2.30
B 2 1 0 0 .25 -2.40 .75 -2.60

;===== end =====
```

Document History:

Ver	Date	Changes
1.00	18 May 06	Initial version
1.10	25 May 06	Pg 2: Added EULA Pg 6 & 7: Added -O option Pg 11: Added Doc History Version # added to footer Font test graphic added to Pg 10 Misc. minor edits for readability
1.20	28 May 06	Added -G option and graphic on Pg 3.
1.30	16 Aug 06	Added multiple outline example on Pg 9. Added updated drill table example to Pg 10. Added Annotation file commands Pg's 11-19. Added Appendix B Pg's 21-24. Updated graphic on Pg 3. Updated parameter list on Pg 6.
1.32	17 Sep 06	Added Include command on Pg 16. Added Scale option to the Offset command on Pg 18 Updated Appendix B Pg's 23-29
1.33	25 Sep 06	Version number updated to match software maintenance release.
1.35	26 Aug 08	Added -E option. Changed the default template file name to board_outline.grb . Updated manual.
1.40	26 Dec 08	Added board outlines to -E option. Added annotation file End and User commands. Changed drill table column heading and data text stroke size. Manual updated with numerous minor changes and addition, reformatted for double sided printing and TOC added. Annotation template file template.ano updated.

This document was prepared with *OpenOffice Writer*. If I never use *Word* again, it will be too soon!