



Eagle v4.16 Board Convert to Gerber Made Easy!



by Sunstone Circuits, 2006

You have the option to send your Eagle *.brd file using our Convert to Gerber service when you order your circuit boards. Just check the conversion option during the order process and we will produce the files needed for manufacturing.

If you prefer to convert your own files follow these easy steps to produce and preview your own gerber and drill files prior to ordering.

- 1.) Download the Sunstone-EagleCam.zip file from our website: <http://www.pcbexpress.com>
- 2.) Open the zip file and extract the cam files to: C:\Program Files\EAGLE-4.16\cam

If you convert your own files you will need to know the layer extension names to send files for your pcb order. The layer names will be your *[board file name]+Sunstone extension* – we pre-name the extensions for easy layer identity to run with our Cam setup process.

Individual Cam files are custom made to generate required file sets for each pcb service offered on our Sunstone Circuits Internet site: www.sunstone.com

In addition each script produces top & bottom solder-paste files, these are not required by our service. Solder-paste files are needed to order stencils from: www.stencilsunlimited.com

top solder paste = Name.tps
bottom solder paste = Name.bps

See Additional Tips at the End of Document:

File Naming
Uncheck Mirroring
Silkscreen Text Settings
Excellon Drill Errors
Producing BOM File
Eagle Resources
Gerber Viewing using Viewmate

Manufacturing Data Files Produced from CAM Files:

2LBasic-Sunstone.cam: Contains data for basic 2-layer pcb no silkscreen/no solder mask.

You will get 3 gerber extended (RS274x) format files:

- component side = Name.cmp
- solder side = Name.sol
- outline = Name.oln

2LPlus-Sunstone.cam: Contains data for 4-layer pcb with silkscreen and solder mask.

You will get 8 gerber extended (RS274x) format files:

- component side = Name.cmp
- solder side = Name.sol
- silkscreen component side = Name.slk
- solder stop component side = Name.smt
- solder stop solder side = Name.smb
- outline = Name.oln

4LPlus-Sunstone.cam: Contains data for 4-layer pcb with silkscreen and solder mask.

You will get 10 gerber extended (RS274x) format files:

- component side = Name.L1 (top)
- midlayer 1 = Name.L2 (mid 1)
- midlayer 2 = Name.L3 (mid 2)
- solder side = Name.L4 (bottom)
- silkscreen component side = Name.slk
- solder stop component side = Name.smt
- solder stop solder side = Name.smb
- outline = Name.oln

6LPlus-Sunstone.cam: Contains data for 6-layer pcb with silkscreen and solder mask.

You will get 12 gerber extended (RS274x) format files:

- component side = Name.L1 (top)
- midlayer 1 = Name.L2 (mid 1)
- midlayer 2 = Name.L3 (mid 2)
- midlayer 3 = Name.L4 (mid 3)
- midlayer 4 = Name.L5 (mid 4)
- solder side = Name.L6 (bottom)
- silkscreen component side = Name.slk
- solder stop component side = Name.smt
- solder stop solder side = Name.smb
- outline = Name.oln

8LPlus-Sunstone.cam: Contains data for 6-layer pcb with silkscreen and solder mask.

You will get 12 gerber extended (RS274x) format files:

- component side = Name.L1 (top)
- midlayer 1 = Name.L2 (mid 1)
- midlayer 2 = Name.L3 (mid 2)
- midlayer 3 = Name.L4 (mid 3)
- midlayer 4 = Name.L5 (mid 4)
- midlayer 4 = Name.L6 (mid 5)
- midlayer 4 = Name.L7 (mid 6)
- solder side = Name.L8 (bottom)
- silkscreen component side = Name.slk
- solder stop component side = Name.smt
- solder stop solder side = Name.smb
- outline = Name.oln

Excellon Drill file will be named [name]. drd

Tool list file will be named [name]. dri



Eagle Board Conversion Made Easy!

Convert to Gerber Steps by Sunstone Circuits, 2006

This demo is for producing 6-layer gerber files using Eagle v4.16

Open Eagle software by double clicking Eagle Program Icon

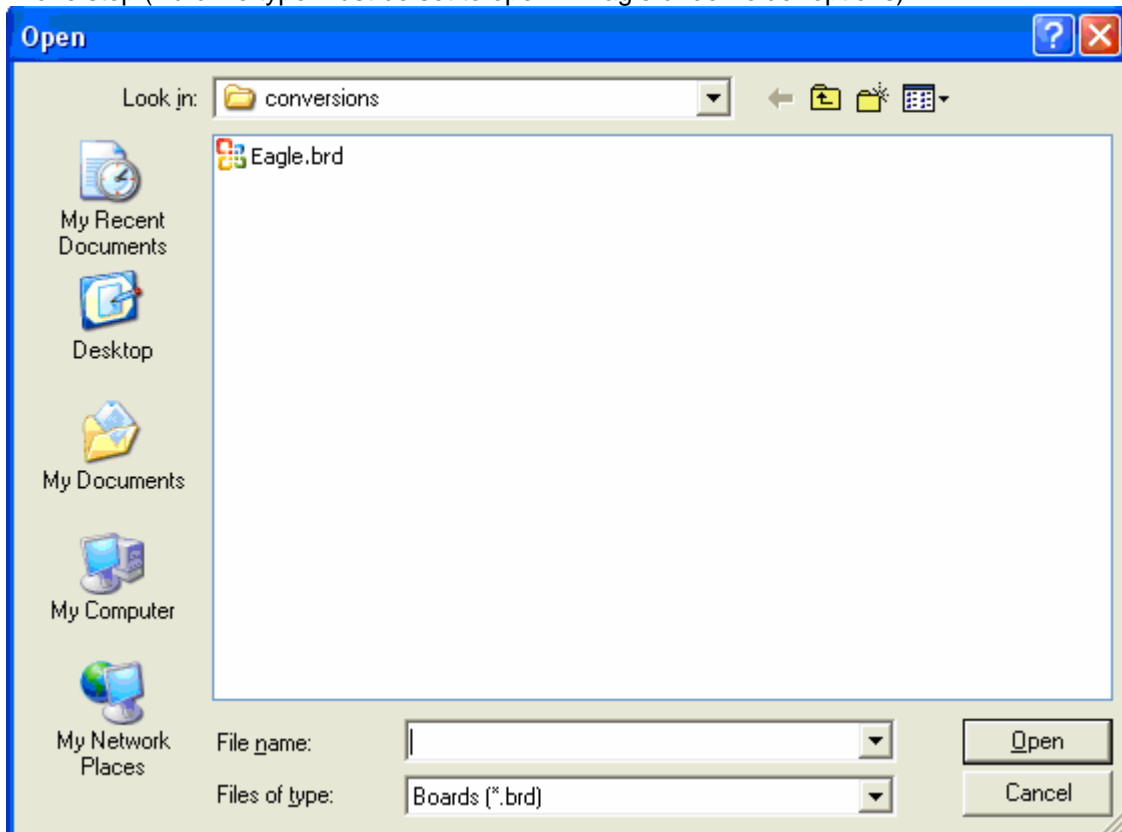


Open board file (e.g. *File/Open/Board*).

Select the folder containing the board file.

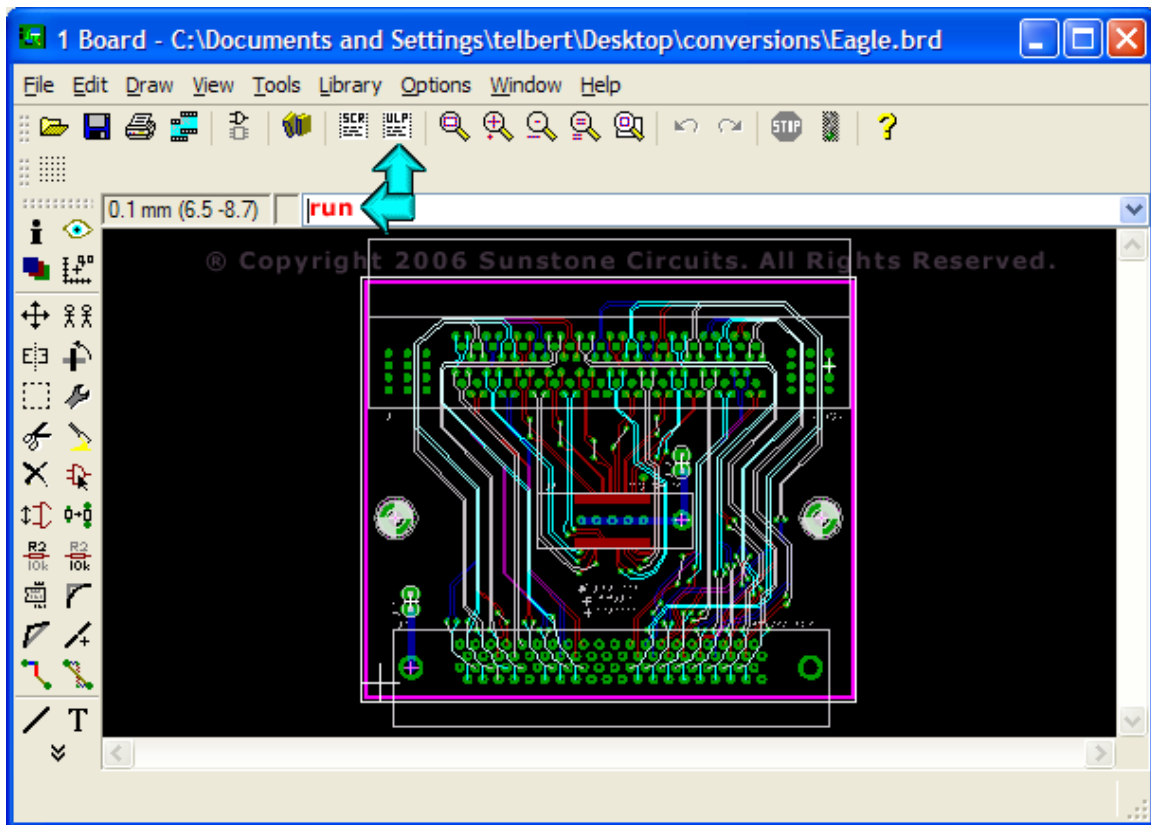
Click on board file, then open.

shortcut: You may double-click the board file directly from your folder to open board file and Eagle in one step (*.brd file type must be set to open in Eagle under folder options).

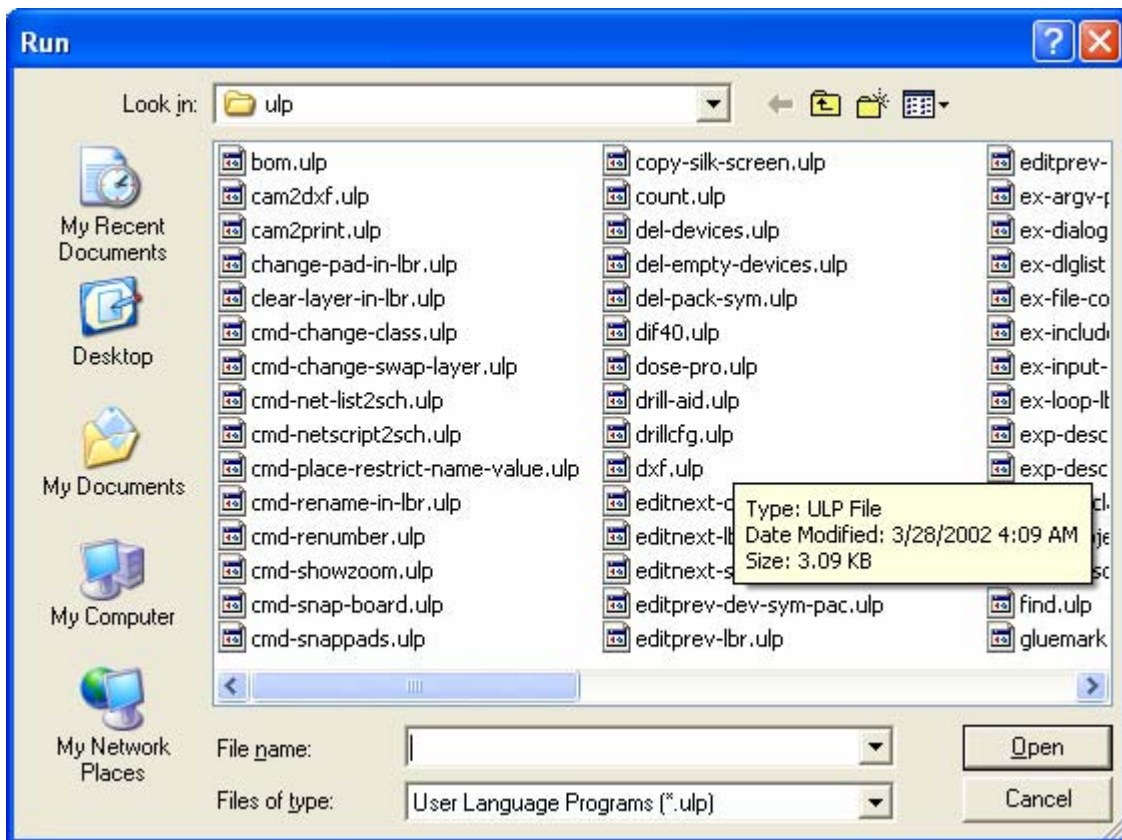


Eagle Board Design File Sample provided with permission from Steven Ellingson, Virginia Tech, Blacksburg, Va. – Thank you Steven!

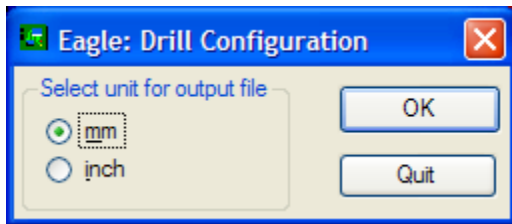
Select ULP button on the menu or type “run” on the command prompt line.



Click "enter"
A "run" dialog window will open
Select "drillcfg.ulp"
Click open

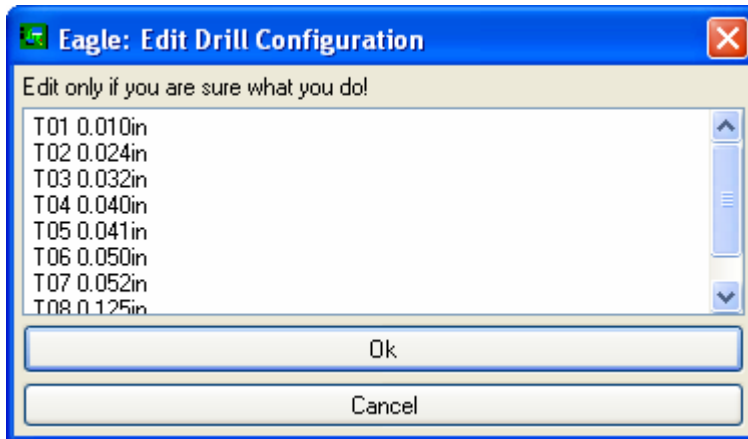


This Drill Configuration dialog box will open
You may leave the default “mm” (or select “inch”)
Click “OK.”



The Edit Drill dialog box will open.
Click “OK”

Note: Sunstone does not edit tool sizes in Eagle keeping customer's default design settings.

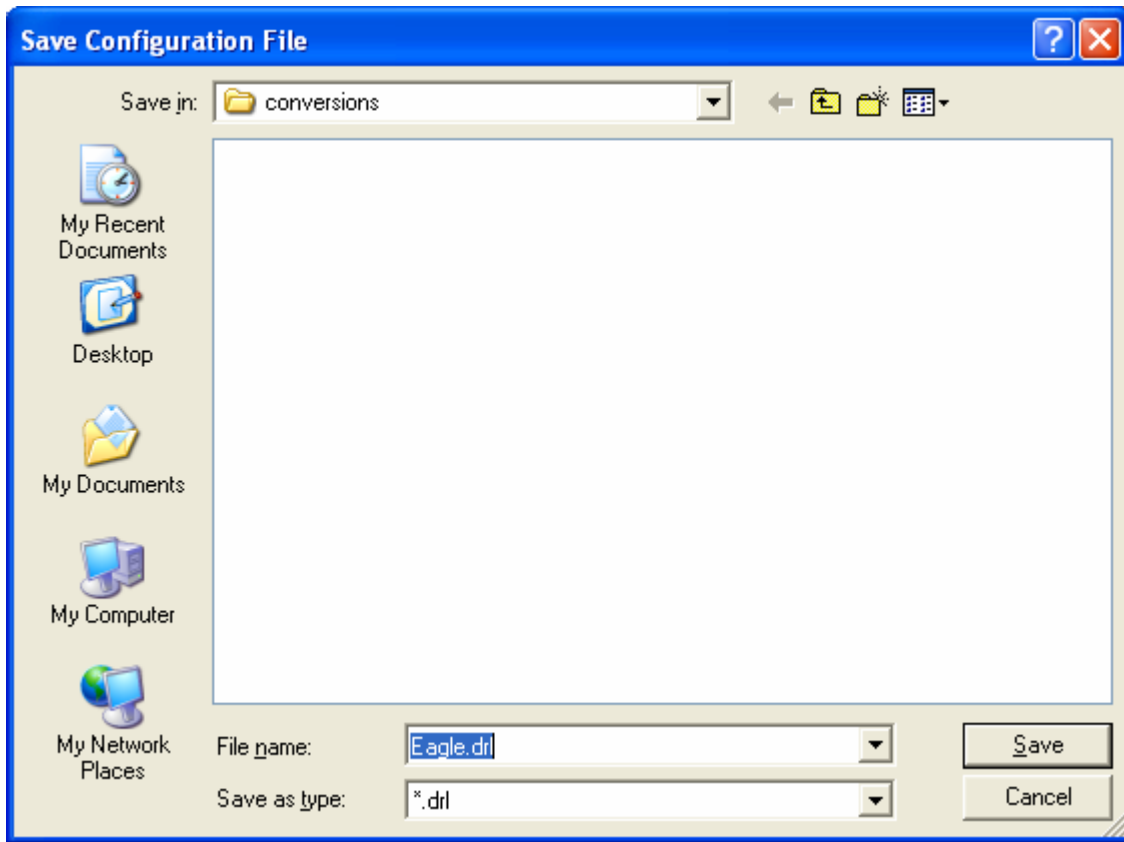


... Save file

Generated files will be named same as = Name.brd file and saved into originating folder.

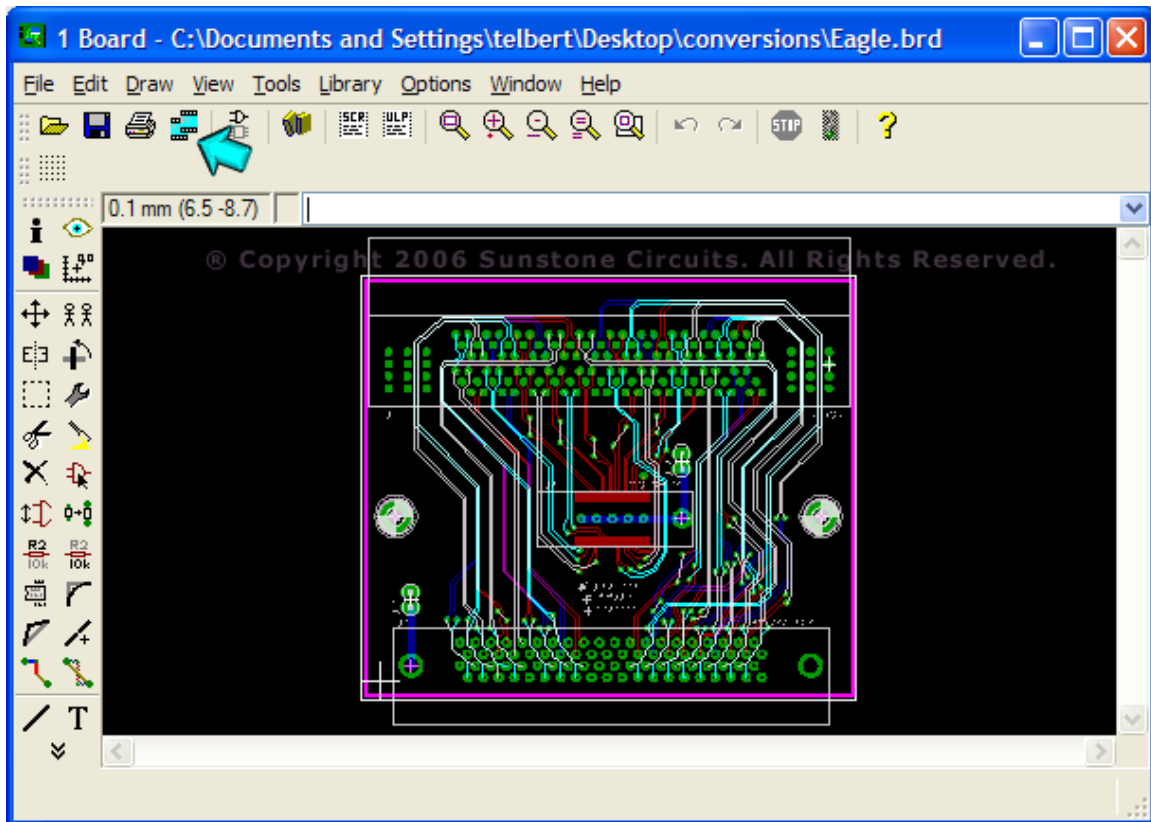
You can leave the file name as is or rename.

[All files extracted will automatically save into the folder that your Eagle = Name.brd originated from]

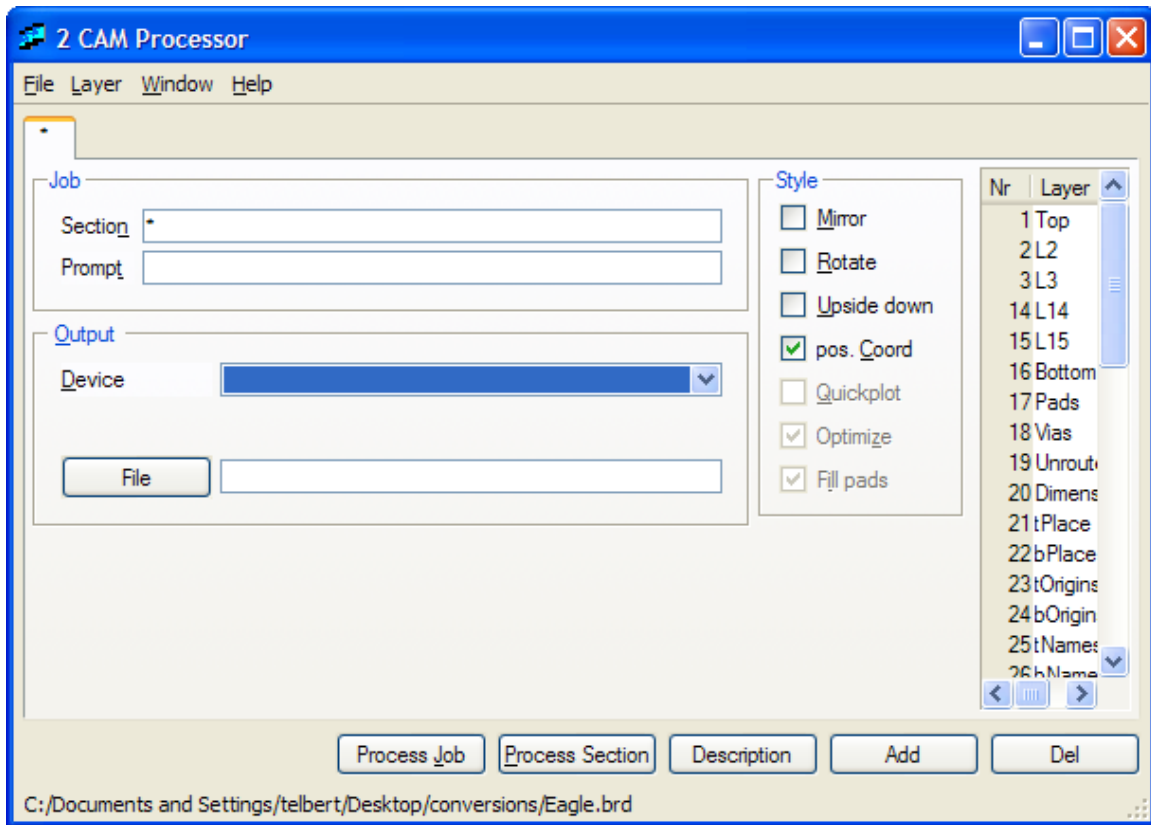


The save window will close.

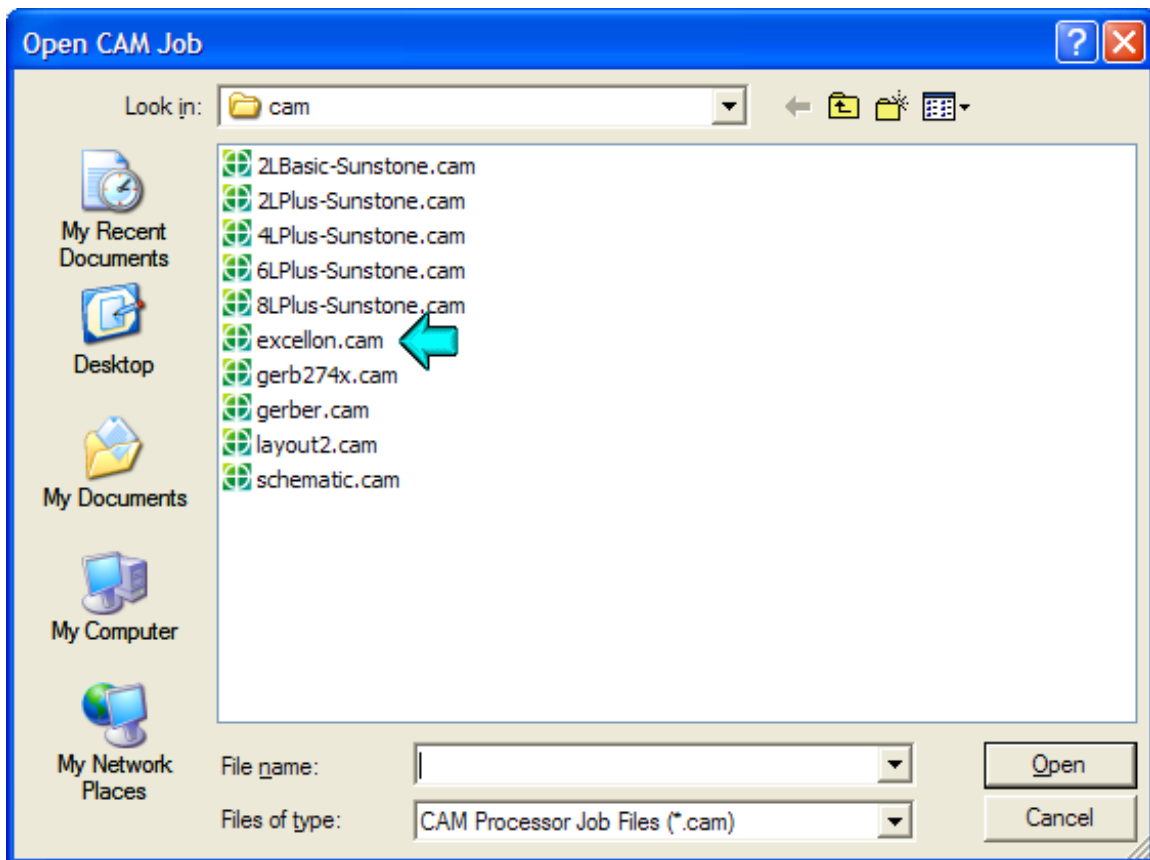
Click on the CAM Processor Icon (4th icon from the left, Green double film menu icon).



The CAM Processor window will open.
To extract the NC Drill
Click: *File/Open/Job*



The CAM Script folder will open...
Select the "excellon.cam" job
Click "Open"



After the excellon cam job opens

Click "process job"

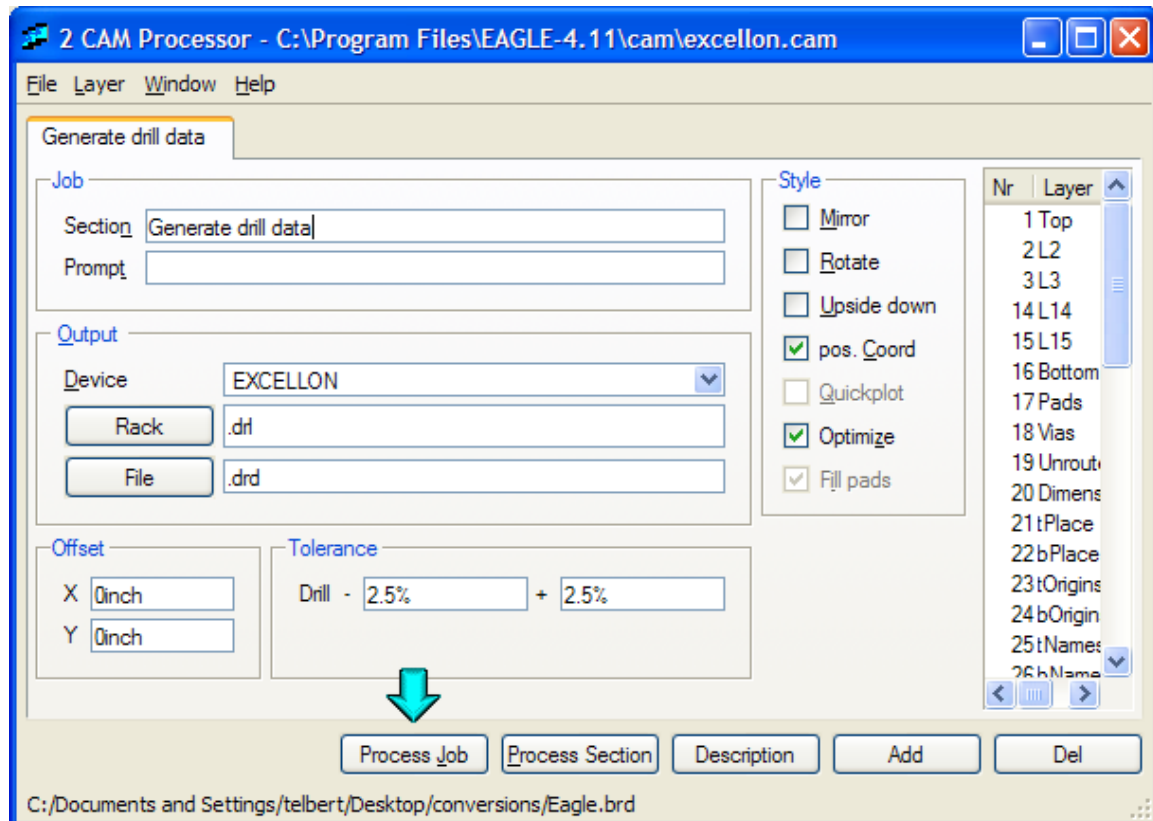
When the processing window stops, extraction of the NC Drill file is complete.

Excellon Drill file will be named [your job]. drc

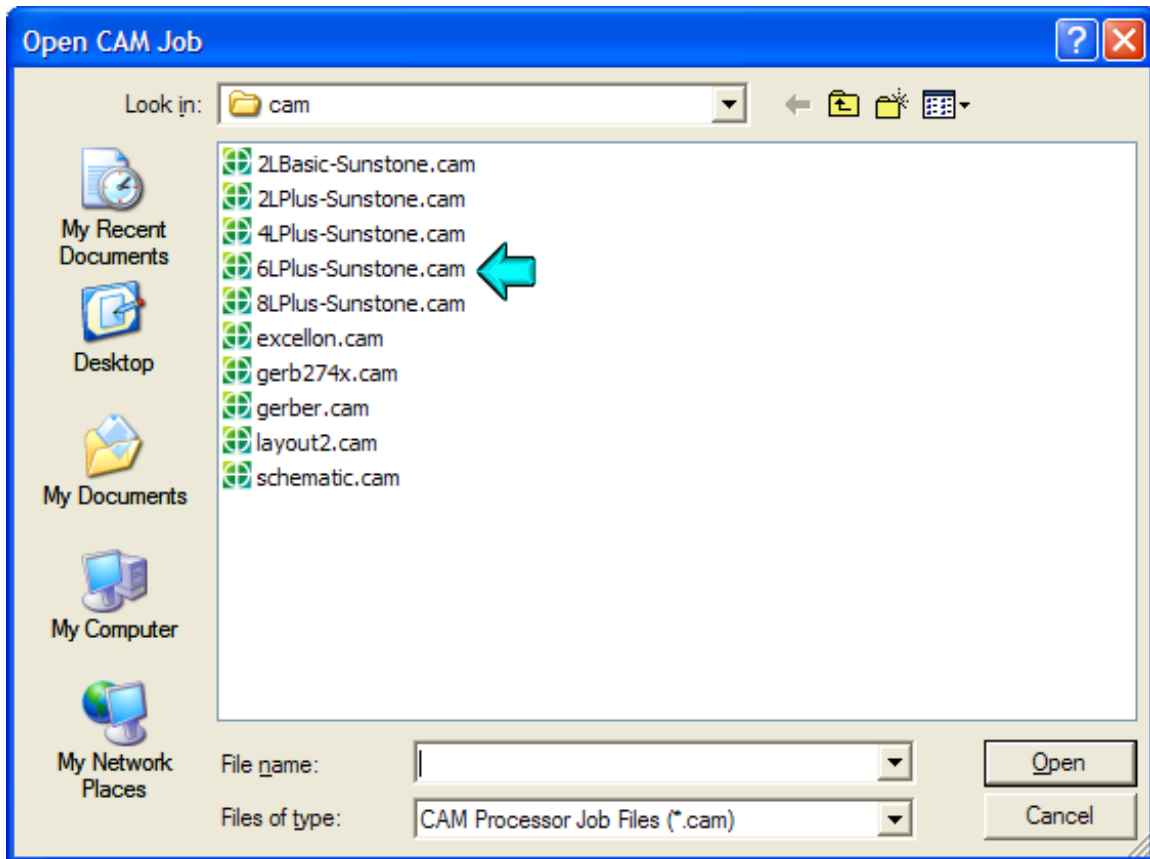
Keep this window open to produce the gerber files.

To extract gerbers,

Click: *File/Open/Job*

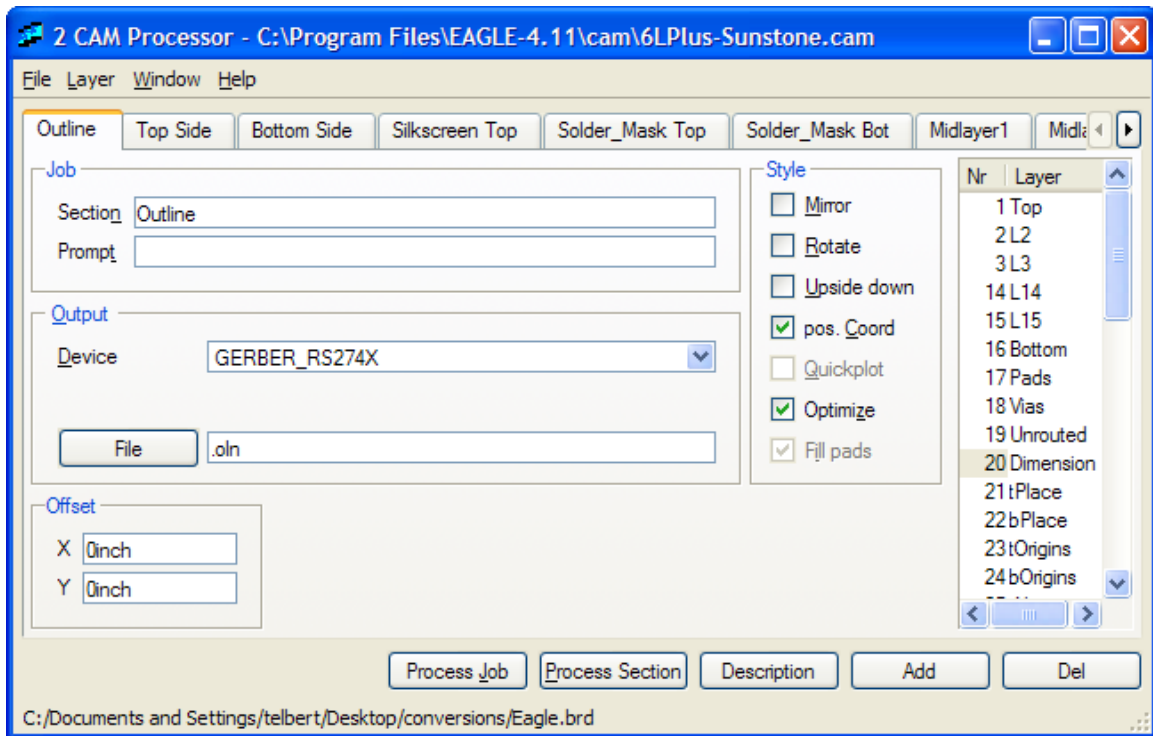


Select the CAM Script for the Sunstone pcb service type you ordering (our demo is for a 6-Layer)
Click "open"



Notice the tabs across the top of the dialog box, these are the layers used to build your design.
Each tab represents a gerber layer that will be generated

Tip: resize window to see all tabs or use the right arrow to scroll over.



Before proceeding you should check the silkscreen and inner layers and for proper layer object setup.

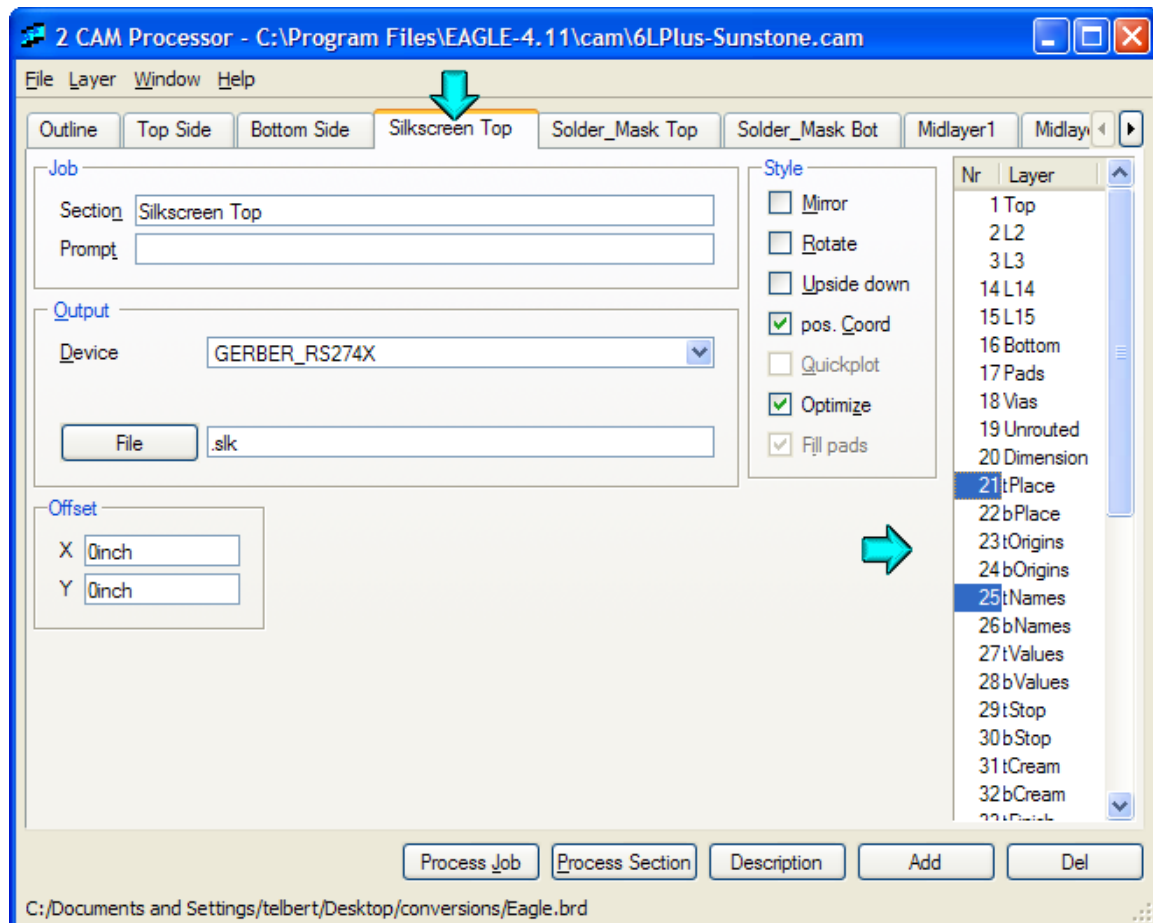
Click on the Silkscreen Tab (not applicable to 2LBasic).

Silkscreen Layer Objects Information:

Click on the tab for that particular layer name.

In the example below, Silkscreen Top has been chosen and the layer object list to the right will show which layers have been turned on. Chosen default layer objects will be shaded when window is opened.

To turn a layer object off/on by clicking on the layer object(s). When you click on a layer, the blue (or shaded) hi-light box will show you it is chosen. The absence of the blue (or shaded) box indicates those layers are turned off.



Silkscreen Layer Objects Information:

If Sunstone is converting your files list the layer objects used to build the silkscreen layer for the "gerberSlk:" when submitting your board file.

Note: We will process using our default layer objects from our generic scripts; we will not include additional layer objects unless customer specifies when submitting files. If no layer objects are listed our script will generate these layers:

For example:

Top silkscreen is L21, L25 (same as tPlace + tNames)

Bottom silkscreen is L22, L26 (same as bPlace + bNames)

e.g: see top silkscreen (gerberslk) as listed on a sample order form:

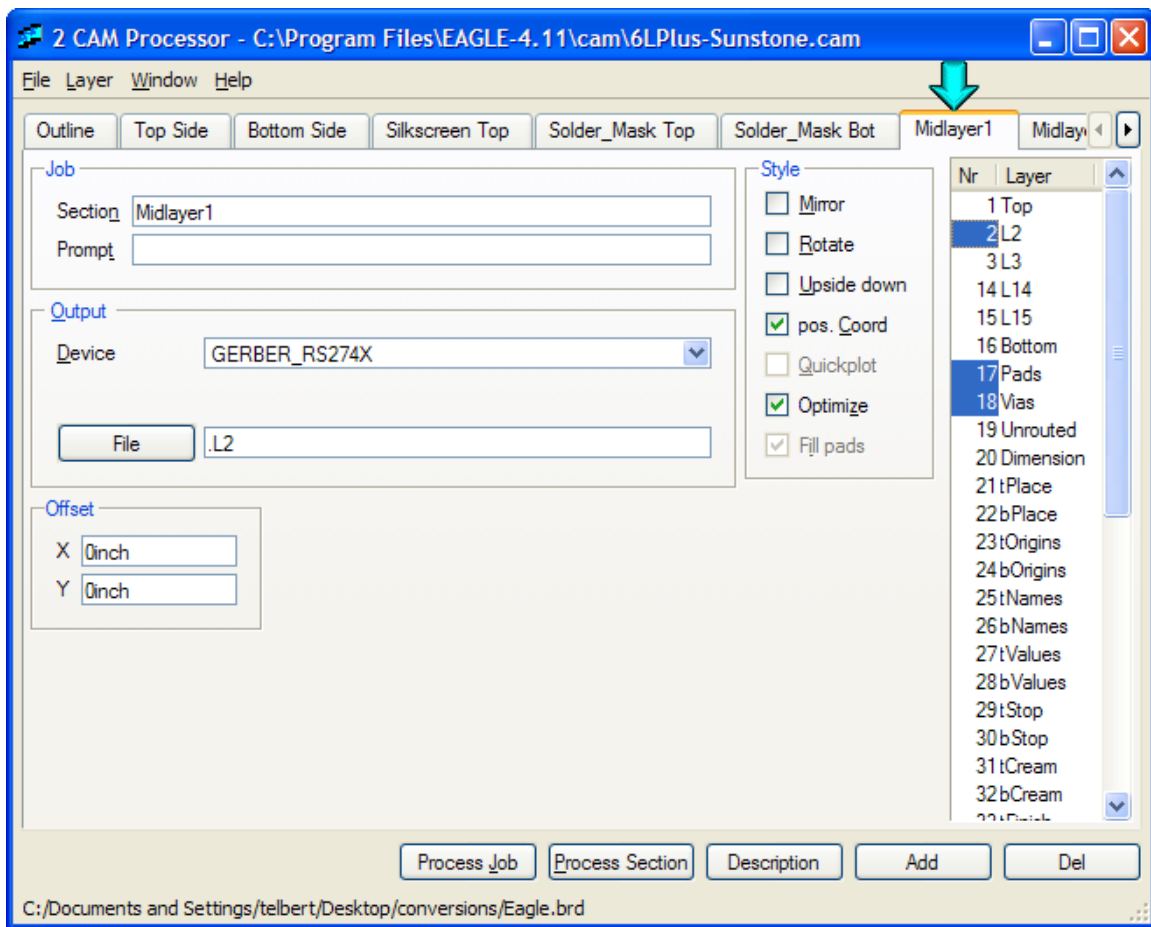
gerberL1:	gerberDri:	gerberSlk: L20,27,28
gerberL2: L2	gerberTol:	gerberBsk:
gerberL3: L3 + pads & vias	gerberApt:	gerberSmt:
gerberEX1: L14 + pads & vias	gerberOln: L20	gerberSmb:
gerberEX2: L15 + pads & vias		
gerberL4:		
Outline Proces OPTION: Use Default Rectangle		

Inner Layer Objects Information:

Click on Midlayer1 (inner layer 1) tab

Selected is layer object L2 + L17, L18 (signal layer + pads and vias).

IF any layer object name has \$ (like \$GND) uncheck L17, L18 (adding pads & vias to a ground layer will create pads stacked on the clearances and cause shorting to ground areas)

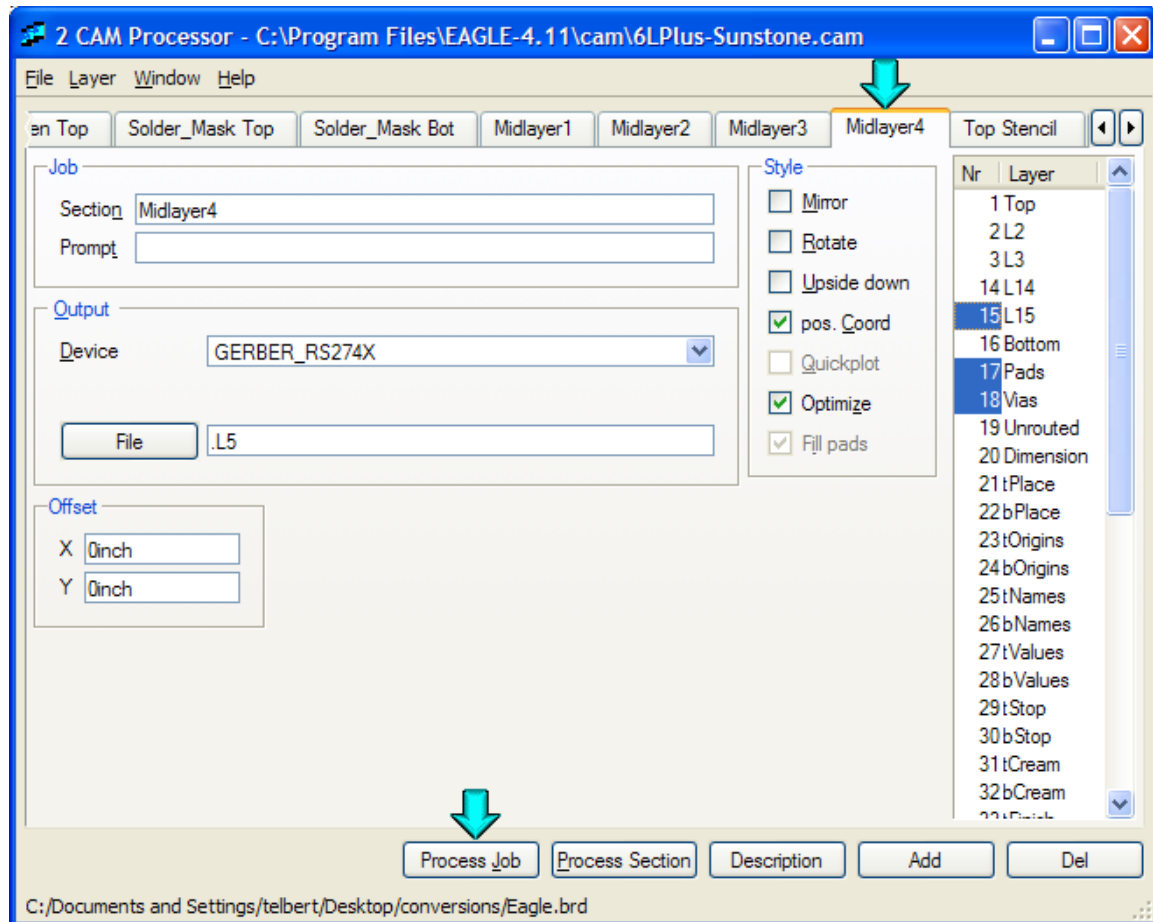


Repeat for all Midlayer Tabs.

(If you are sending your *.brd for conversion note your inner layer and silkscreen objects to put on the order form so we can convert your files for you).

When you are done checking all the layer tabs, generate gerbers.
Click "Process Job"

When conversion is complete, close Eagle, it will prompt you to either save or not save changes. It is recommended not to save changes, it is advised to make changes within each layout than edit the CAM script. (You can download cam scripts to replace when needed).



Saving the files:

Go to folder that you were converting from.

Make a new WinZip file and drop all the files in to the zip file.

Name the zip file the [your part name].zip

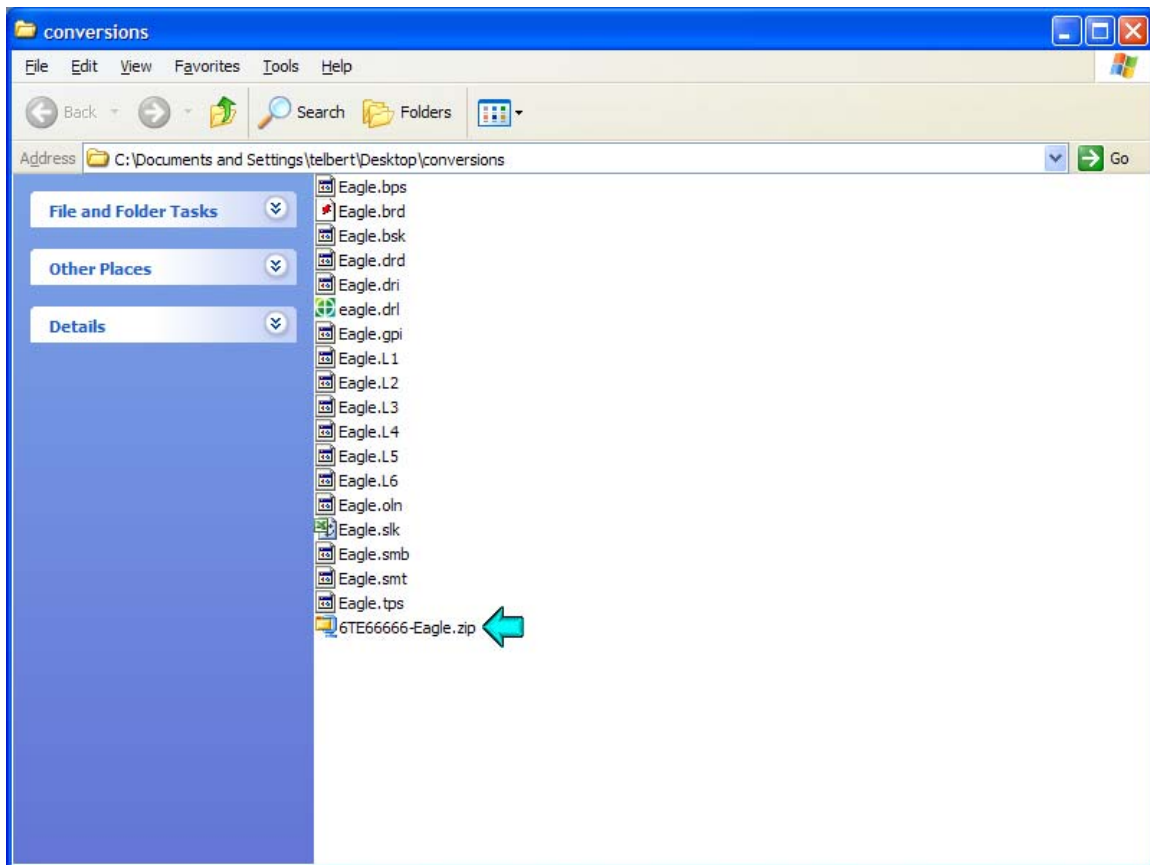
Your files are now ready to preview and ordering!

Pre-tooling (for Sunstone Cam Operators only):

Name the zip file the [customer's order number]–Eagle.zip

Gerber files or import zip file and view in CAMMaster.

Gerber files may be viewed in ViewMate, but they must be extracted from Zip file first.



Additional Tips and Resources:

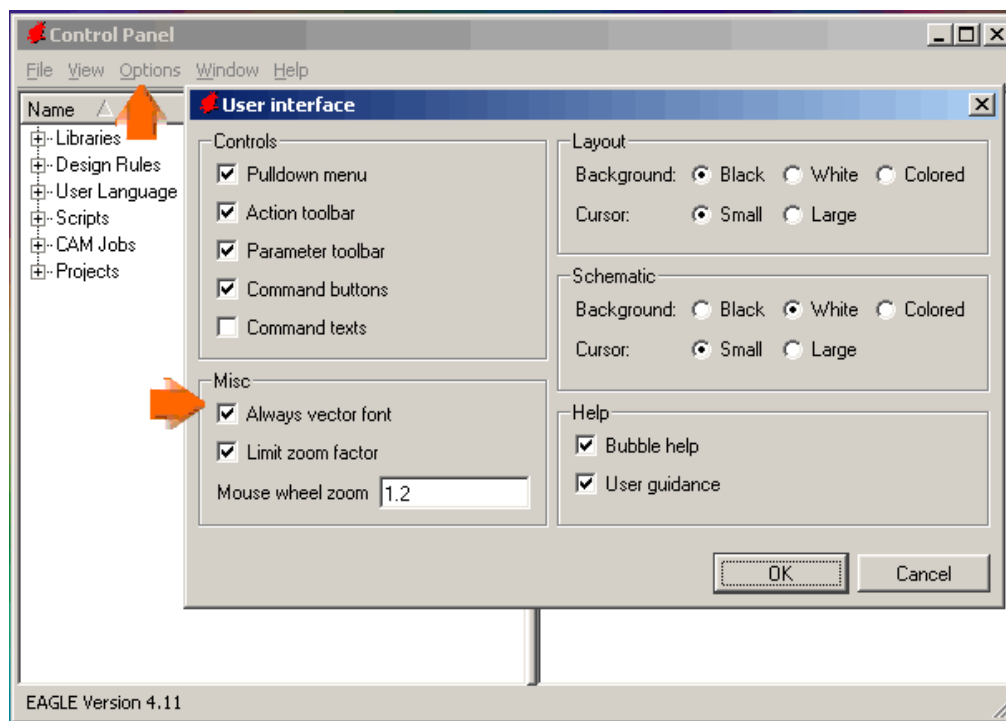
Eagle files are gerber RS274x format (embedded apertures) no aperture file/list needed.

File Naming: Use short file names (6 characters or less) for easier file management and filling out forms.

Uncheck Mirroring: Double check the bottom side layers (copper, soldermask, silkscreen etc.) are not set to mirroring. In the Cam processor click the layer tab and uncheck mirroring

Silkscreen Text Settings: The Eagle Cam processor will automatically output as vector font, other fonts used could cause text to expand. Always use Vector Fonts!

Font fix: Open Eagle, on the menu bar choose Options/User Interface and select the option "ALWAYS VECTOR FONT". Problems may exist if the user is using a different text font. Using the Vector Font only will allow you to see exactly what you are going to get.



Excellon Drill Errors: When running the drillcfg.ulp choose "mm" instead of "inch" then run the Excellon cam job, this should correct "no holes selected" type error.

Excellon Drill file will be named [name]. drd

Drill Tool list file will be named [name]. dri

Drill drawing file named [name].dri not used for prototype orders.

Producing BOM File: From the main command prompt type run, choose bom.ulp or click ULP button on menu, choose bom.ulp

Eagle Resources:

We recommend using the Eagle website for design tips and additional scripts/downloads to help with your board layout. Try their online user forum to find answers.

Eagle user forum: <http://www.cadsoft.de/forum.htm>

Eagle users file sharing & downloads: <http://www.cadsoft.de/download.htm>

Gerber View - Preview using Viewmate: <http://www.pcbexpress.com/technical/faq.php#ref>