

CadSoft Computer, Inc.

Frequently Asked Questions

By: Eng. Edwin Robledo

Q. Why do I get an ERROR CODE ### when I try to load a design?

A. It is possible that the file you are trying to load was created with a cracked or hacked license of EAGLE.

Q. What is the difference between EAGLE Light, Standard, Professional? What are Modules?

A. The main difference between EAGLE Light, Standard and Professional is found in the board size allowed and amount of routing layers.

Power	Maximum board size	Routing layers
*Light Edition	100 X 80 mm (3.93 X 3.14 inches)	2 layers (top and bottom)
Standard Edition	160 X 100mm (6.29 X 3.93 inches)	4 routing Layers
Professional Edition	Unlimited board size	16 routing layer

* Light edition of EAGLE only allows one schematic sheet per design.

Please contact your dealer for price information.

Modules:

EAGLE modules only work with EAGLE Standard and Professional. The board layout editor of EAGLE can be a Stand Alone module for those designer not interested in the Schematic and/or Autorouter portions of EAGLE. You may also combine the Board layout editor with the Schematic and/or the Autorouter if need to be.

NOTE: Schematic and Autorouter are not stand alone modules.

Q. With EAGLE, can I create my own custom libraries? If yes, How? Can I use already existing libraries to create my own?

A. Yes, EAGLE provides all necessary tools to create your own components. Libraries consist of three (3) parts: the component Package, the Schematic Symbol and finally the Device.

Package

The package will consist of the physical properties of your components. You will need to set the correct grid to allow the correct placement of your pads. To start a library from the EAGLE control panel click on FILE/NEW and select Library. Now select EDIT PACKAGE from the top tool bar, provide a name for your package in the appropriate field and click OK. Use the grid command to set the correct grid for you pad spacing. Use the PAD or SMD command to define your connecting points for this component. If you are placing PADS make sure you are using the correct drill hole size as specified by

manufacturer. Use the NAME command to number your pads according to your manufacturer spec sheets.

Define your component outline (Silk Screen) on the TPlace layer. Create your NAME and VALUE fields. These fields need to be created in the correct layers with the appropriate text syntax. The name field will be define using the TEXT command and typing >NAME, make sure you are working on the TNAME layer. The value field will be define in the same matter but using the text >VALUE, placed on the TValue layer.

Symbol

The symbol will be the logical representation of your package. From the EAGLE library editor mode click on EDIT SYMBOL icon and in the NEW available field give your new symbol a name. Make sure you always work on a .1 inch grid all the time. Use the PIN command to place your pins and use the NAME command to name your pins accordingly.

There are several properties you may change for your pins. Click on the CHANGE icon and select the option Direction, here you may define the direction property of the pin. You options will be:

NC = No Connect	Pas = Passive	HiZ = High Impedance
Pwr = Power	Sup = Supply	In = Input
Out = Output		

Select the correct direction then click on the pin.

Another characteristic you may change for your pin is its function. Your options for the function are:

None = No function	Dot = Not	Clk = Clock	DotClk =NotClock
--------------------	-----------	-------------	------------------

The visual size of the pin may be change in CHAGE Length, the options for this characteristics are:

Long (Default)	Middle	Short	Point
----------------	--------	-------	-------

The last characteristic for you pin you may change is the Visibility. This allows you to determine if you want to be able to see name of the pin or not, once the device is place on a schematic. Your choices are:

Off = Pin and pad name assignment are not visible	Pad = Pad assignment is visible
Both = Pin Name and Pad assignment are visible.	Pin = Only the pin name is visible

After placing all your pins and assigning them all the characteristics, you should create a outline for the component. This outline should be draw using the wire command and placed on the Symbol layer.

Now we will define the Name and Value field. This is done with the help of the TEXT command. Click on the TEXT command and a dialog box will appear prompting for the text, type >NAME, before you place this text make sure the NAME layer is selected. Perform the exact same steps for the value, make sure the >VALUE field is placed on the VALUE layer.

DEVICE

In the device is where the Package and the Symbol come together. In the library editor mode you may click on the EDIT DEVICE icon. In the NEW field you may provide the name of the device.

Click on the ADD command and add the schematic symbol. You may continue clicking on the ADD command if your component consist of more than one symbol.

Click on the NEW button to find the respective Package that will used with this schematic symbol. In the NEW dialog box you will notice a field for variant. If this device will only contain one Package or this is the first package being used you may place tow Single Quotes in the variant field.

Place a value in the variant field that will relate to the particular package being used.

Use the connect command to assign the proper Pad to correct PIN. After all your connects are define you may click on the NEW button again to assign another variant (package) to this symbol and use the connect command to assign the pin to pad relation.

Use the PREFIX command to assign a prefix to your device. If the device is a multi gated element you may use the NAME command to name your gates.

Cutting and pasting between libraries

Packages and Symbols may be shared among other libraries. At this time it is not possible to share the device between libraries. To use a package or a symbol in another library simply Group/Cut the package or symbol. Open up the new library and create the appropriate element (Package or Symbol) then do a paste.

Q. Does EAGLE generate Gerber files? How?

A. Yes, EAGLE does output Gerber files. In fact we have two (2) formats of Gerber files available. We have device drivers for Gerber 274 and Gerber RS 274X format.

The main difference between these two formats is that Gerber 274X has the aperture wheel file is imbedded in the Gerber file. The aperture wheel file contains shapes and sizes that will be used to create your board.

To drill the holes on your board, you will need to be generate EXCELLON files. The EXCELLON files actually consist of two files, one being the DRILL RACK file and the other being the actual EXCELLON output file. The Drill Rack file

contains the Drill sizes that are going to be used on your board and assigns them a T Code. Drill files can be easily generated with EAGLE.

EAGLE comes with easy provisions to make Gerber files for a two sided board and the drill files.

From the Board file click on ULP icon and select the ULP called DRILCFG.ULP. This will generate a Drill RACK file.

Click on the CAM processor icon to bring up the CAM processor tool. Click on File/Open/Job and select the CAM job called EXCELLON.CAM. Click on Process Job button to generate the EXCELLON output file.

Now to generate the Gerber files. From the CAM processor click on FILE/OPEN/JOB and select the CAM Job called GERBER274X.CAM. You will notice this Job file contains multiple Tabs. Each tab pertains to a set of layers used to create the Gerber for that layer of your board. For example, the component side will be made up of the layers TOP, PADS and VIA. This will build all of the copper for the top side of you board.

Click on process job to process your Gerber files. Now click on FILE/OPEN/Job and select the CAM job called EXCELLON.CAM. Now click on Process Job.

All of the Drill files and Gerber files will have the exact same name of your board with different extensions files will be located in the same directory of you board. Below is a brief explanation of the files created and their meaning.

File Extension	Description	Layers
.CMP	Copper Component side	Top, Pads Vias
.SOL	Copper Solder Side	Bottom Pads Via
.PLC	Silk Screen Component side	Dimension, TPlace and TName
.STC	Solder Stop Mask Top	TStop
.STS	Solder Stop Mask Bottom	BStop
.DRL	Drill Rack File	DRILCFG.ULP
.DRD	Drill Output or Excellon	Drill and Holes

Q. What is a ULP?

A. ULP stands for User Language Programming, this system allows user to program EAGLE to output and import different style of ASCII information.

Q. Can I import files from other products (PADs, Protel, OrCAD)? How?

A. EAGLE can import Netlist from OrCAD Multiwire, TANGO, PADs, Schema and Susie. EAGLE comes with a netlist converter called XCONVERT. This will convert the above mentioned netlist into a EAGLE script format. Then you must manually place all components on the board and name them correctly. Now you may use the SCRIPT command to import the converted netlist.

Q. Can I place components on the back side of my board?

A. Sure you can, use the MIRROR command to flip components onto either side of the board.

Q. For the autorouter, How do I define my default wire width? Can I have multiple wire width for the autorouter?

A. The default wire width and via drill size is set in the DRC/Sizes dialog box. To have multiple wire width you have to define NET CLASSES. The net class may then be assigned to your schematic or board connections.

Q. How do I define the PAD and VIA diameter?

A. Pad and Via diameter are controlled by the DRC/RESTRING parameter. The diameter will be calculated according to the Restring % factor that has been setup. The formula works the following way:

$$\text{DIAMETER} = \text{DrillSize} + 2\text{DrillSize}(\% \text{ Restring})$$

You are restricted by a MIN and MAX values. These values may be changed if needed.

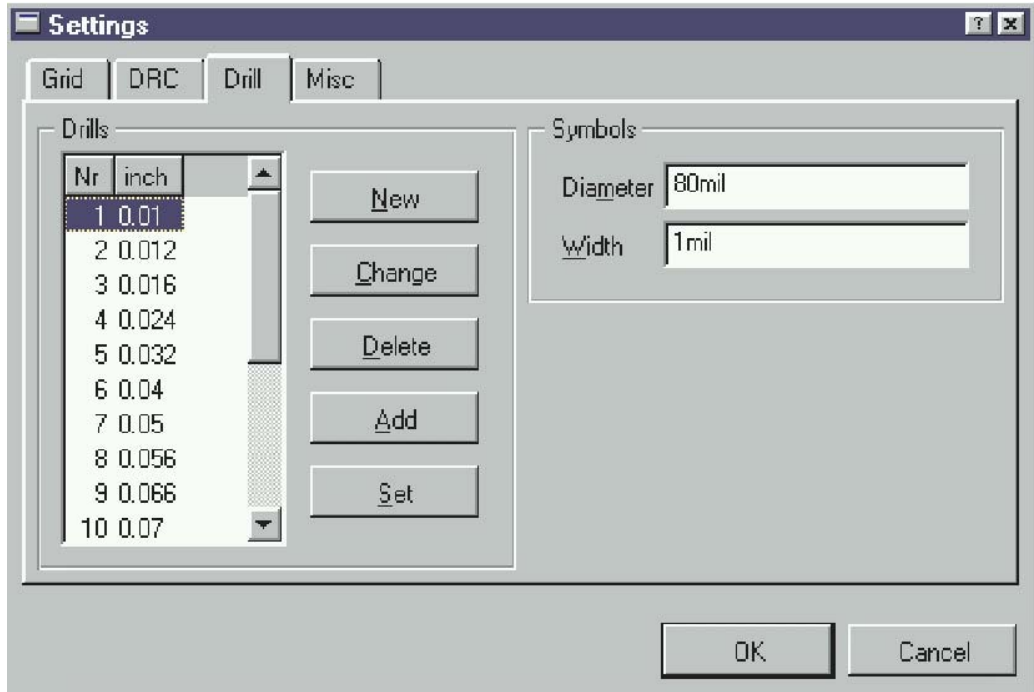
Q. When using the autorouter I get the error “Unreachable SMD Pads ...”. What is wrong?

A. EAGLE default autorouter grid is set to .05 inches which is very large for SMD components. EAGLE autorouter is grid based so it needs to have a grid intersection at the connecting point of the all pads. To solve this, go to the autorouter/General dialog box and lower the grid.

NOTE: Making the grid smaller provides the autorouter with many more options to rout the trace, for then the rout may take more time.

Q. I would like to assign symbols to my different drill sizes on the board, will EAGLE do this for me?

A. Yes, click on Options/Set and select the DRILL tab. In the drill tab dialog box you will notice a SET button. When you press this SET function, it will analyze your board and assign symbols to your drills and holes.



Notice the Nr. column is the symbol number assignment and the inch column refers to the drill size that it is representing. Here is the number representation chart:

1	+	10	✕
2	×	11	▽
3	□	12	△
4	◇	13	◁
5	⊗	14	▷
6	⊗	15	⊕
7	⊕	16	⊕
8	⊕	17	⊕
9	✕	18	⊕

Activate layers 44 and 45 on your board layout to view your drill hole symbols.

