

Introduction to CadSoft Eagle

Cadsoft Eagle is printed circuit board (PCB) computer aided design (CAD) software. The software allows users to create schematic circuits, PCBs, and libraries, while also allowing user to generate bill of materials (BOM) and Gerber files. The software is free, easy to use/learn, and is compatible with Windows, Linux, and Mac OS. The free version can be downloaded from <http://www.cadsoftusa.com/download-eagle/>

These tutorials will cover basic navigation of the software's environment, creating libraries and basic components, creating schematics and PCBs, and generating necessary files for fabrication.

1.1 Accessing the Main Control Panel

Each time Eagle is opened, a Control Panel screen will appear. All projects, libraries, processing jobs, and files can be accessed, created, and edited from this screen.

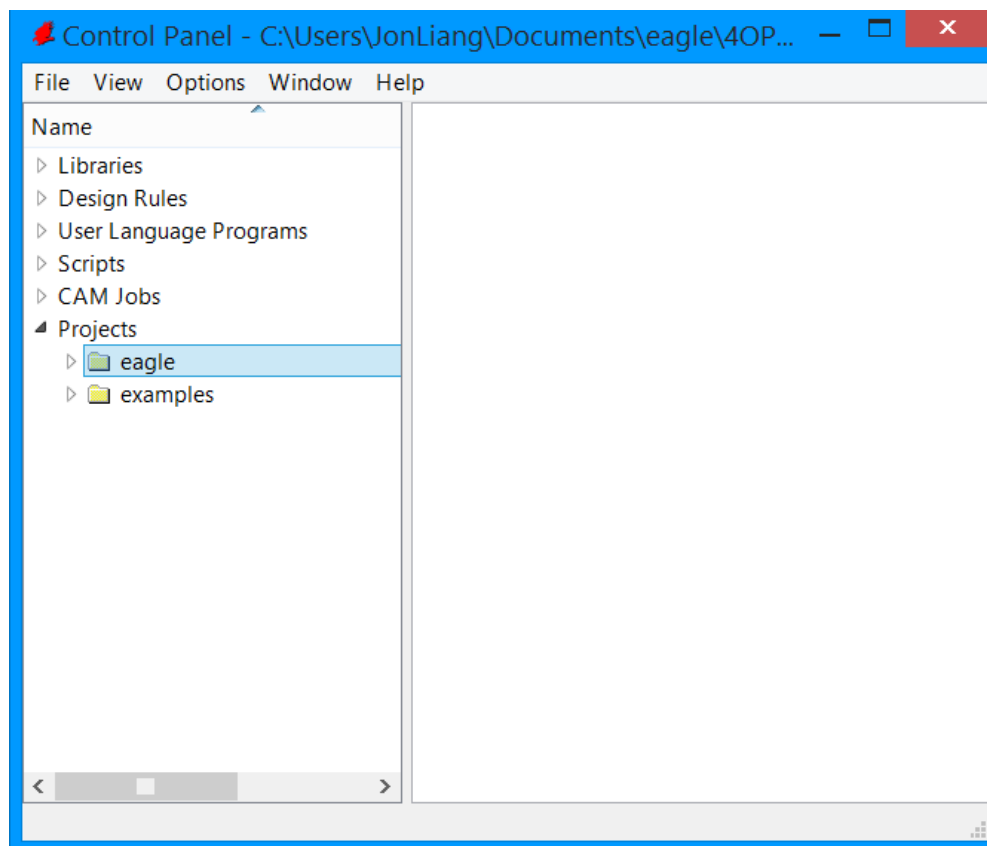


Figure 1. Control Panel

1.2 Starting a New Project

Each time a user wants to create a new PCB, a project folder is used to store all necessary files such as the schematic, layout, and Gerber files. To create a new project, first open the projects folder and click on **Projects > Eagle**.

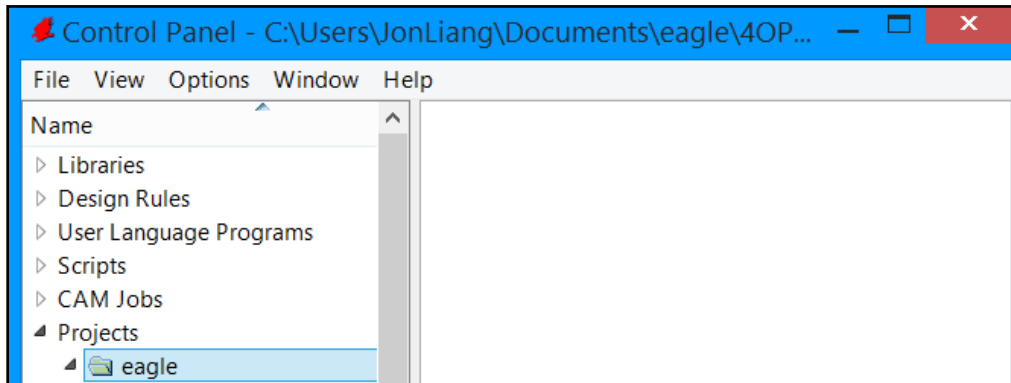


Figure 2. Control Panel – Eagle folder

With the eagle folder opened as shown above in figure 2, click on **File>New>Project** to create a new folder to store all necessary files for a project.

1.3 Creating a New Library

Libraries are folders in the software that contain the files for individual components such as resistors, capacitors, integrated circuits (ICs), etc. The user can create new parts by creating the necessary parameters for a component, storing them in a library, and accessing them in the schematic/layout portion. To create a new library, click on **File>New>Library**. A pop-up window will appear. To create a part like a resistor, there are three components that need to be designed: Symbol, Package, and Device.

The Symbol represents the schematic form of the component. This component is used in the schematic editor which allows the user to connect wires from component to component. When observing datasheets, a user will see a symbolic representation of a component with labels of what each pin does. That is the same case in Eagle.

The Package is the physical representation of the component. This component is used in the layout editor where the user places the part on a printed circuit board to wire. Datasheets provide design specifications of the part, such as pin length and width and various distances. Typically the design specifications can be found at the bottom of datasheets.

The Device is the final step in creating a part. In the device portion, pins are matched between Symbol and Package such that whatever way the user creates the Symbol, it correlates to the correct pin in the Package.

The order in which the component is made does not matter, but the Device will always come last. The user can either make the Symbol first and Device second, vice versa.

All three components can be accessed/created from the pop up window.

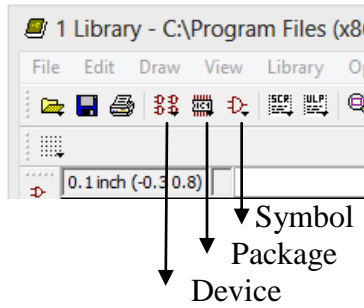


Figure 3. Device, Package, and Symbol Creation

1.4 Creating a Components Symbol

When making a Symbol, a pop-up window will appear requiring the user to label the symbol. When using the schematic layout editor, this label will appear next to the component. An example for naming a symbol is making a certain resistor size (package size). Resistors come in various sizes such as 0402, 0603, 0804, etc. If the user is making a 0603 resistor, a potential name is R0603.

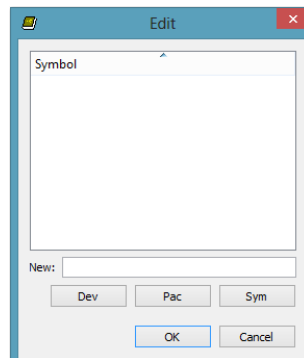


Figure 4. Creating a New Symbol

After naming the Symbol, a Symbol Editor will appear. There are various tools that can be used to create the Symbol, but the most critical are the Name, Text, Rectangle, and Pin tools.

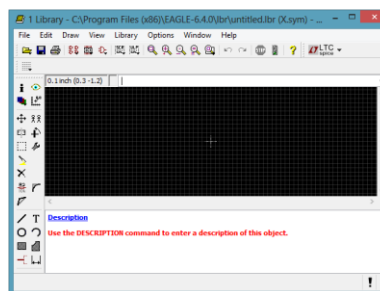


Figure 5. Symbol Editor

The first step adds the necessary number of pins onto the editor. These pins are the symbolic representation of the pins of the component. Arrange the pins such that there is no overlap with the labels. Once the pins are laid out, use the Rectangle tool to draw a box for the pins. This is helpful for documentation purposes. The Name tool can be used to name pins and is helpful for documentation, and for Device creation. An example of a symbol is shown in Figure 6.

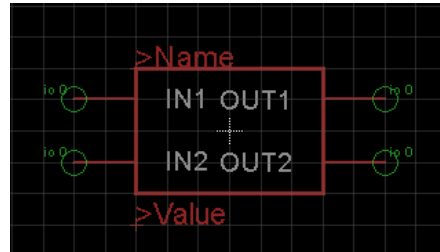


Figure 6. Symbol Example

A general rule of thumb when creating Symbols is to add a name and value to the component. These labels are used to document the specific component and its value (capacitor, resistors, and inductors). The software uses the “>” symbol as a reference to the Name and Value tools in the toolbar. To add a Name and Value, use the Text tool in the toolbar and type “>Name” or “>Value”.

1.5 Package Creation

Now select Package symbol in the main window. Again, a name editor will appear asking for a Package name. Components come in all shapes and sizes, but there are general package types available.

The process of creating a Package varies from that of the Symbol. In the symbol portion, the size and units do not affect the use of the component. In the Package creation step, the component being made must match the design specification of the datasheet. The first step in creating a components Package is identifying the type of unit used. Depending on the country of origin, different units are provided by the manufacturer (typically millimeters or inches). To change the units, the user can access the Grid editor on the top left. The Size is the most important parameter because it affects the scaling of the grid of the Package editor. The Multiple and Alt parameters will scale with the grids units.

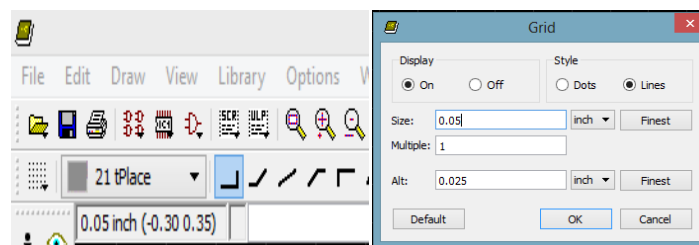


Figure 7. Grid Editor

Similar to the Symbol creation, the most important tools that are used are the Name, Info, Wire, Text, Rectangle, and Pad tools.

The Info tool provides necessary information on anything on the screen such as a wire, a pad, or a text. Parameters can be set from editing the Info window. For example, the user wants to draw a box that is 2x5 mm. The user could draw an arbitrary box using the Rectangle tool, and then edit the dimensions of the box using the Info tool to meet the 2x5mm criteria.

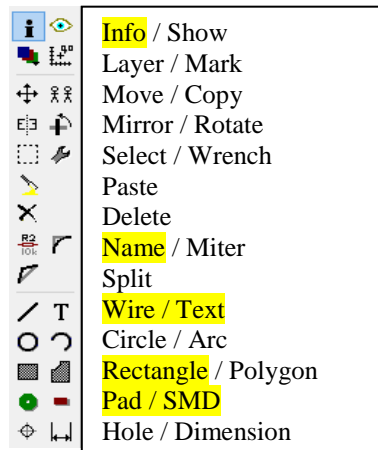


Figure 8. Package Toolbox

There are two types of Pads that are used in the Package creation. Pads represent the contact points for which components, such as resistors and capacitors, connect to the PCB layer. The two types of Pads are either thru hole or surface mount device (SMD). Use the name tool to label the pad so that it matches the datasheet.

To create a Package, first place down the necessary number of Pads for the component. Use the Info tool to edit the dimension of the Pad, such that the Pad properly fits the specification of the datasheet. Then use the Info tool to adjust the coordinates* of Pad to its designated location.

*Eagle references a component by the center of part. For example, there is a circular part centered at (0,0) with a radius of 2mm. Eagle will refer to its location at (0,0), and therefore if the part is moved it will correlate to the position where the part was previously.

After placing the Pads onto the editor, use the Wire tool to draw the dimension of the component. Switch to layer 21 tPlace, which is specifically used for documentation. Then, add the >Name and >Value labels.

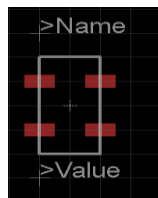


Figure 9. Package Example

1.6 Device Creation

The last step is to connect the pins of the Symbol and Package. Click on the Device symbol and a screen will prompt the user for a name of the device. Typically, users name the Device and Symbol the same.

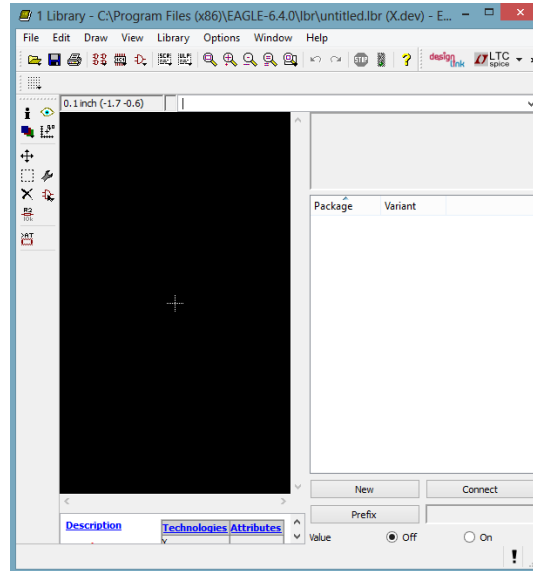


Figure 9. Device Editor

First, add the device by clicking on the Add tool located underneath the Wrench tool. Select the component and center it to the crosshair in the editor. In the bottom right of the screen, click **New > Connect**. In the new window, match the corresponding pin and pad and connect them. After connecting the pin and pad, save the components.

1.7 Schematic Design

A schematic is the file where the symbol of a device is connected with other parts to create a circuit. Click on **File>New>Schematic** to create a new schematic.

The general guide to building a schematic is broken into two processes: adding parts and wiring them. Click on **Edit>Add** to add a new part to the schematic. A window will appear that lists all of the libraries provided by the distributor and the libraries designed by the user. There is a search bar to look for specific components or libraries.

The most necessary components that need to be included in any circuit are the power and ground components. To search for power and ground, type in “VCC” for power and “GND” for ground.

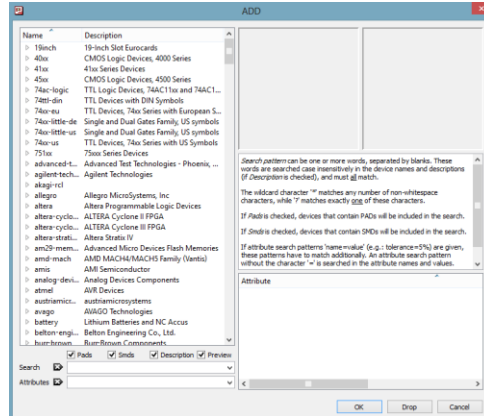


Figure 10. Adding Parts

With the parts placed, use the Wire tool to connect them. The wire tool will draw a green line from where the user designates, and it represents a physical wire that would connect pins on a circuit board.

Use the Name and Value tools to change the name of components or to add values. When the user clicks on a component such as a capacitor with one of the tools selected, a pop-up box will appear.

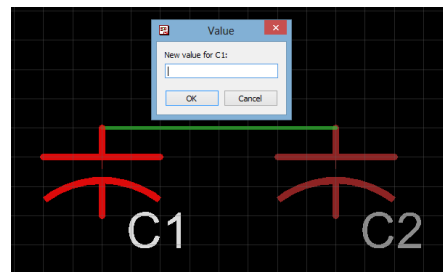


Figure 10. Changing Name and Value

An alternative to connecting components is to use the Label tool. By using the label tool, users can reference where a pin of a component should connect to. The label tool is efficient in reducing the amount of space used by wires, and can help clean up the schematic from being overrun by wires. Use the Name tool after placing a Label to link it to another existing Label.

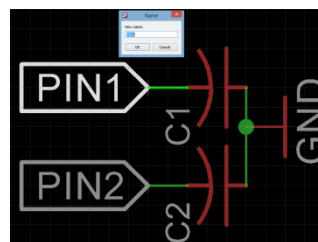


Figure 11. Adding a Label

1.8 Creating a PCB

The PCB is the physical representation of the schematic. To create a PCB from a schematic, click on the 'Board' button. A warning window will appear asking you to 'Create from schematic?'. Click **Yes**.

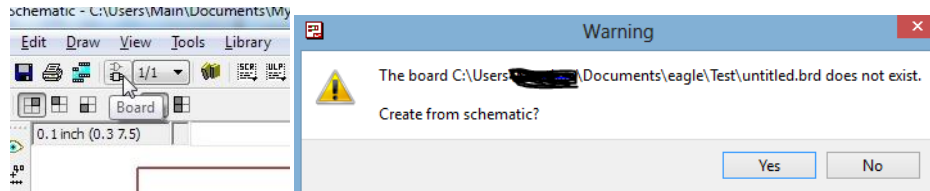


Figure 12. Create a PCB

A new window will appear with the Package representation of the components in the schematic and a white box. The white box represents the amount of space available to the user. For the free version of Eagle, users are allowed 4x3 inches of working space.

First, use the Polygon tool and draw a rectangle around the white box. Polygons are used to define layers and planes such as GND planes and power planes. In the figure below, a GND plane is added in order to sink excess heat of components. Use the Name tool and click on the drawn Polygon to rename it.

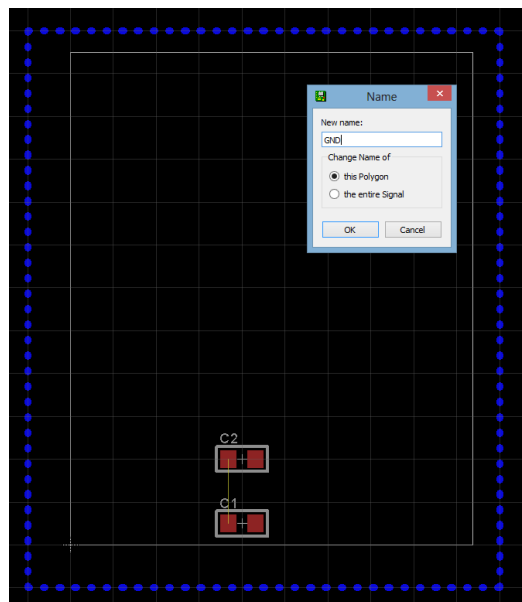


Figure 13. Add a Plane

Place the components onto the PCB using the Move tool. Yellow air wires show components that need to be connected to each other or to a common node. Use the Route tool to connect components with a trace. A trace is a wire, and can be adjusted to meet a certain width, thickness, and length.

When connecting components with traces, there are a few important commands that can be input into the command line to effectively layout a board. Those commands are:

- Grid mm – Switches grid to millimeters
- Grid inch – Switches grid to inches
- Grid mil – Switches grid to mills
- Grid xx – xx refers to the grid unit size
- Ripup @; - used to remove the ratsnest layer, and to shorten unnecessary airwires

Generally, components can be placed in any type of unit on the grid. For drawing traces, it is highly recommended to switch to mil units because most manufacturers limit a users design in mils.

Once the pins of each component are routed to its target location, the user can adjust the board to meet his/her own design style. Depending on the purpose of the board, the user can change the shape and size using the Info tool, and can use the Text tool to label necessary components for documentation.

1.9 Design Rule Check and Gerber Extraction

Eagle provides a Design Rule Check (DRC) that allows the user to input the fabrication limitations of a company that produces PCBs. Machines can only machine PCBs up to certain accuracy, so companies generally state the limits of line thickness and pin separations. To edit the Design Rule Check parameters, click on the DRC button on the bottom of the toolbar.

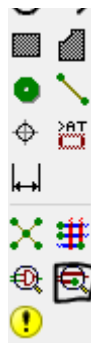


Figure 14. Design Rule Check

A new window will appear and allows the user to input the design parameters for the PCB. Options that can be edited include copper thickness, wire spacing, minimum drill sizes, and isolation sizes between pins. Be sure to thoroughly read the design limitations of a fabrication company and to use the DRC tool to check that the PCB fits within the limits. If the DRC tool returns with inconsistencies after processing, fix the PCB until no errors are returned.

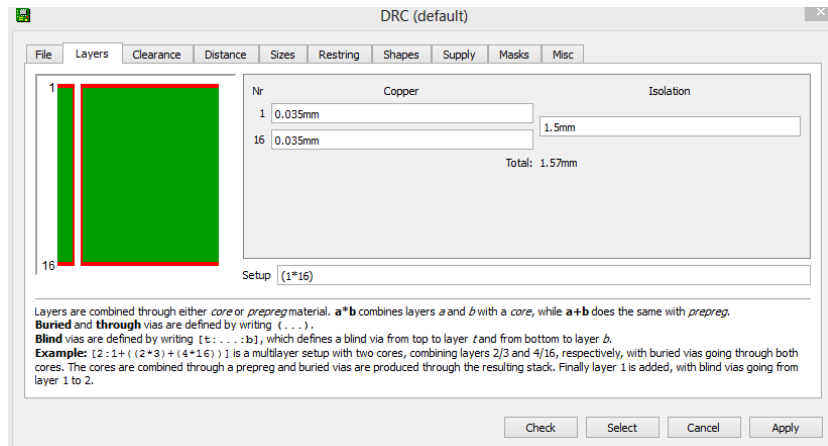


Figure 14. Design Rule Check Tool Editor

With the DRC completed, the last step is to generate the files used to fabricate PCBs. All fabrication companies accept Gerber files to print PCBs. Gerber files are essentially pictures of all the layers of the PCB such as the drill layer, copper layer, solder mask layer, silkscreen layer, etc. Eagle provides a Gerber extraction tool which conveniently generates the necessary files for fabrication.

To start a Gerber extraction, click on **File>CAM Processor**.

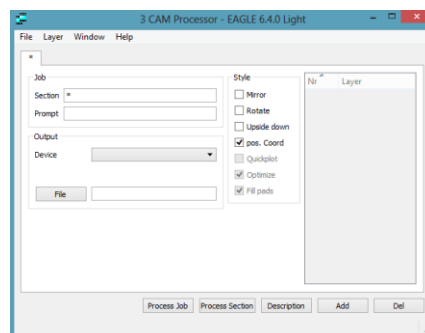


Figure 15. CAM Processor

The CAM processor is a scripting tool which takes a build-in extraction processor to generate Gerber files. Different CAM processors can be downloaded to extract necessary files, but Eagles default files can do the extraction. To start an extraction, click on **File>Open>Job**. Select the CAM script and click 'OK', and then click on 'Process Job'. The files will then be extracted to the Control Panel in the project folder.

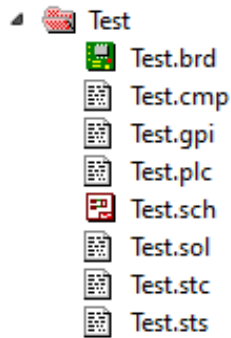


Figure 16. Gerber Files

Review which files are needed for fabrication (different companies require certain files), and copy those files into a new folder. Zip the folder with the Gerber files and they are ready to be sent.

2.0 Tips and Suggestions for Eagle/Fabrication

Fabrication:

There are many companies that fabricate PCBs for large commercial operations and for small groups such as students. Some of the companies that can be used are:

- ITEAD Studio (\$)
- Bay Area Circuits (\$\$)
- Advanced Circuits (\$\$\$)

Vendors have different quality levels and turnaround rates. ITEAD is a company from China, so the turnaround rate is long. The bonus is that the user can get a lot of copies for a cheaper price. Other companies such as Bay Area Circuits and Advanced Circuits are U.S. companies so the turnaround speed is much faster, but can cost more money.

Soldering/Safety:

Soldering uses high heat and lead to weld components to the PCB. Most solder is of a 40/60 mix of lead and other metals. When soldering, be sure to have proper ventilation such as a fan or fume vacuum on to reduce the amount of contact with dangerous chemicals. Do not grab the soldering iron anywhere below the indicated grip as the heat will cause burns. Rack the soldering iron back into resting position and turn off the machine when not in use. When finished soldering, always remember to tin the solder tip with a coat of solder to prevent the tip from burning out. The user should always wash their hands after soldering, and should clean up the soldering area of any rubbish generated during the soldering process. The rule of thumb with working in the lab is to not use machines that the user is unfamiliar with, as safety is priority.