

Getting Started in Eagle 7.3.0 Professional Schematic Software

**Tyler Borysiak
Team 9 Manager**

Executive Summary

PCBs, or Printed Circuit Boards, are all around us. Almost every single piece of electrical technology contains a printed circuit board that is used to control a specific application. There are numerous modern tools that are used to develop printed circuit boards. Learning how to construct and fabricate a PCB using design software can prove to be a very useful tool for any electrical engineer. PCB design software gives engineers the ability to be creative, innovative, and cost effective when designing any piece of circuitry. One of the most advanced and well-known software programs is EAGLE. EAGLE stands for Easy Applicable Graphical Layout Editor. It is a very powerful design software that is tailored to meet the needs of professional engineers, hobbyists, and students worldwide. It provides a fast learning curve through the existence of online manuals and active user forums. The openness of Eagle's extensive and fully-open component libraries eases the design process for all.

Table of Contents

- 1. Title Page
- 2. Executive Summary
- 4. Introduction
- 4. Objective
- 5. Getting Started
- 8. Selecting or Creating Components
- 15. Building a Schematic
- 17. Testing a Schematic
- 19. References

Introduction

Using Eagle, an engineer or a hobbyist has the capability to create millions of different schematics using the tens of thousands of free component libraries available. Rather than using a large breadboard as a prototype schematic, a PCB layout can be easily created using the Eagle software. Creating a PCB is more robust and compact. It encourages the development of creative layouts and designs.

Developing a PCB on Eagle software can be broken down into a few sections or steps. The first step is to create a schematic. This involves adding components from existing libraries. If a component cannot be found in an existing library, the user can also create a component footprint and add it to the schematic. After creating a schematic file, the next main step is to transform the schematic file into a board file in order to place components and route traces on a PCB of any desired length and width. This board file can then be easily transformed into a file that can be sent out to a PCB fabrication shop to be printed.

Objective

The objective of this application note is to describe and outline the process of developing a schematic using Eagle. The first objective is to become familiar with the general layout of the Eagle user interface and the various available tools. After understanding the layout of the software, the next step can be implemented. This step involves adding or creating components in order to initiate the construction of the schematic. After adding all of the required components to the schematic file, circuit construction can now be set in motion. Movement and rotation of components, routing of wires to specific pins, and the assignment of nets will be the useful skills learned and understood from the circuit construction. After constructing a working circuit, the next step will be to test and verify every connection using the “show” tool and the “electrical rule check” tool. This allows for the transfer of the schematic file to the board file for PCB fabrication. After reviewing this application note, it is expected that basic knowledge and understanding of the Eagle software will be obtained.

Getting Started

In the first section of this application note, basic setup steps will be walked through. Before getting started in the program, it is important to know if the computer to be used is compatible. Figure 1 below is a list of the system requirements in order to use the Eagle software.

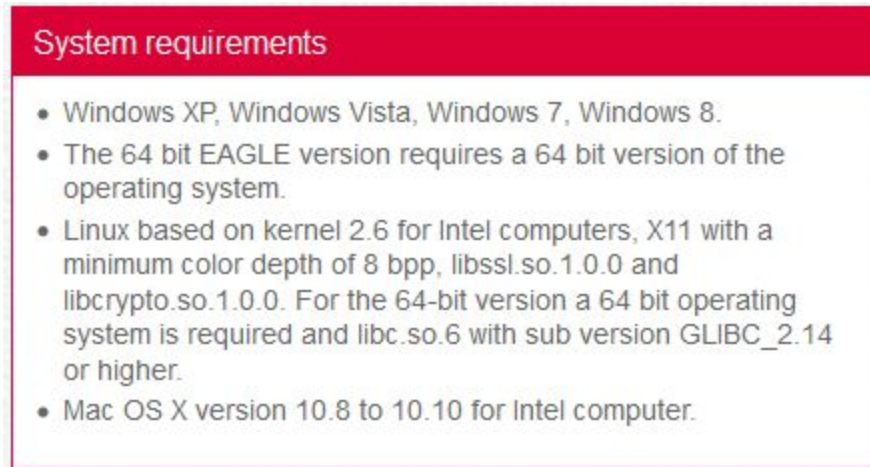


Figure 1: System Requirements

Using a desktop that is compatible with the Eagle software, the first step is to launch the program. The first time the program is launched, a message will pop up saying that a directory for Eagle does not exist. Simply select “Yes” to create a directory in order to work on Eagle projects.

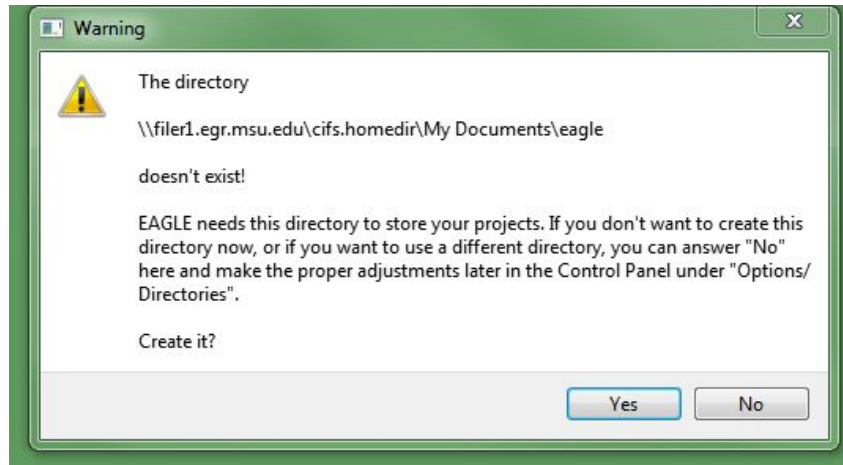


Figure 2: Creation of a Directory

Selecting “Yes” will open up a control panel. From here a new project needs to be made in order to create a design schematic. This can be done by expanding “Projects”, right clicking on the “eagle” directory, and selecting “New Project”.

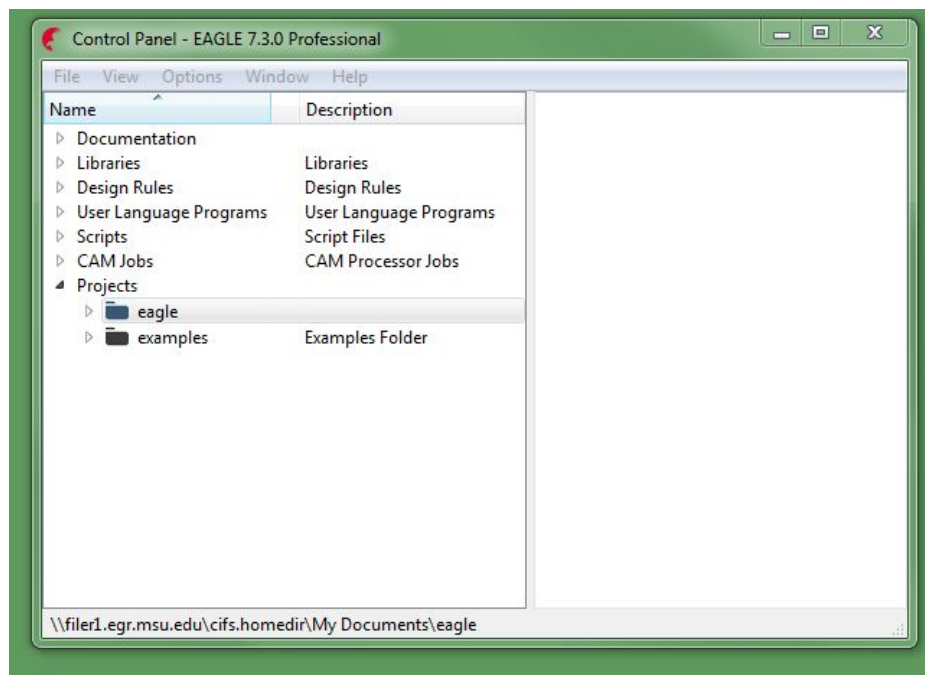


Figure 3: Creating a New Project

Next, the newly created project folder needs a name. For this application note, the folder will be titled MOSFET Circuit, since this is the circuit that will be created. This created project folder contains both the schematic file and the board file to be created.

Since the goal of this application note is to show the development of an Eagle schematic, the next step is to right click on the new project name “MOSFET Circuit” and select “New” and then “Schematic”.

This opens up a blank schematic file like the one shown in figure 4 below.

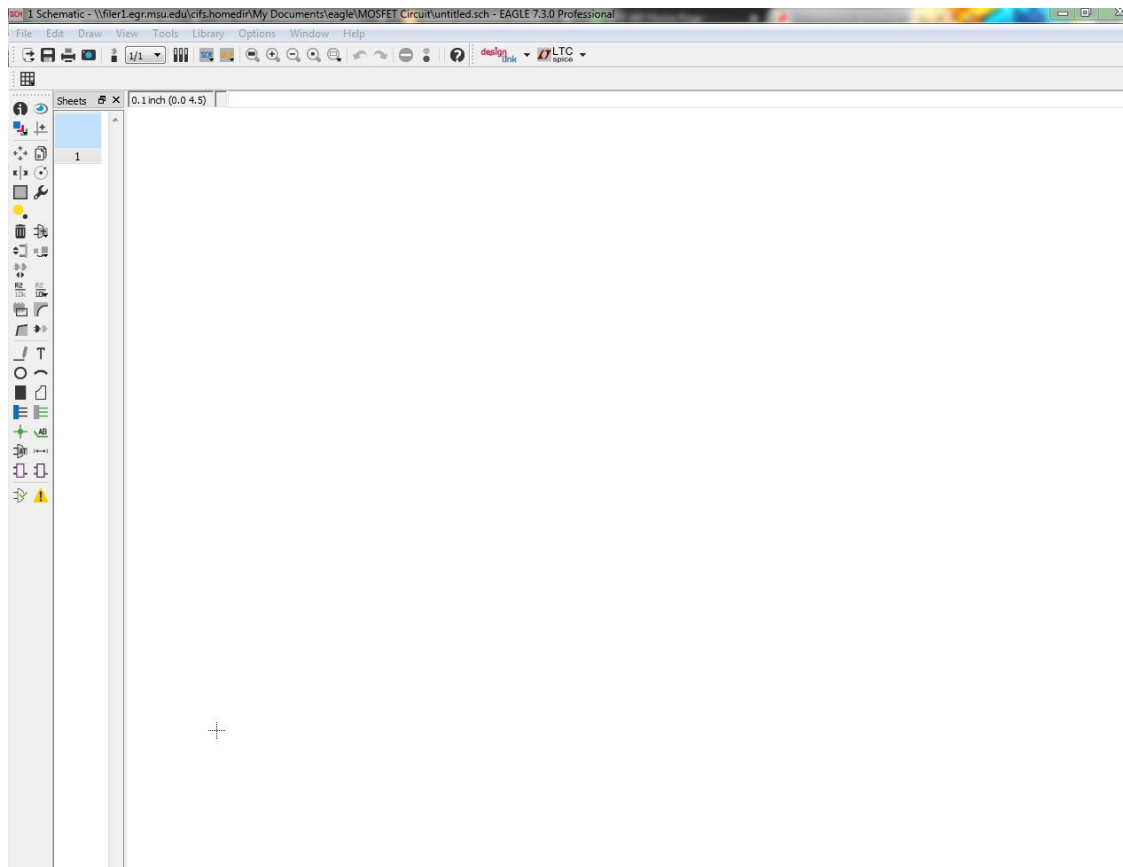


Figure 4: Blank Schematic File

This is where parts will be added and the schematic will be constructed and routed. On the left side of the screen is the tool box. This is where you can change layers, copy components or circuits, move components or circuits, wire pins to each other, and so on. Most of the tools needed to construct the circuit schematic will be in this tool box.

Now that the schematic user interface is understood, the next step is to add or create components for the MOSFET circuit schematic.

Selecting or Creating Components

Before selecting components to add or create in the schematic file, it is important to understand the circuit that is being built. A helpful way of doing this is to hand draw the schematic to be constructed on a piece of paper to help with the layout and design. Once the circuit components and the functionality are understood, the next step is ready to be implemented.

In order to add a component, select the “Add a part” button in the toolbox on the left. This is referenced in Figure 5 below.



Figure 5: Button to Add a Part

The first component to add is a resistor. After selecting the “Add a part” button, move down to the search box and type “resistor”. Numerous items appear after doing this. It is important to know exactly which type of resistor will be used. It is possible to select surface mount resistors or through-hole resistors. For this MOSFET circuit, through-hole resistors are to be used. Since the goal is to manually solder the circuit components onto the board after fabrication, the important thing to consider is the resistor package that corresponds to the correct resistor dimensions. For example, if a simple color code resistor is going to be used, a resistor component with a length of around 7.5mm should be chosen since that is a close approximation to the actual length of a through-hole color coded resistor. This will make it easier to solder the resistor to the PCB leads. In this case, package 0204/7 shown in Figure 6 below is perfect for the MOSFET circuit. On the left hand side is an expanded view of the specific libraries and the parts it holds. On the right hand side is a description and illustrations of the selected

part. The left picture is the schematic component and the right picture is the board view of the component.

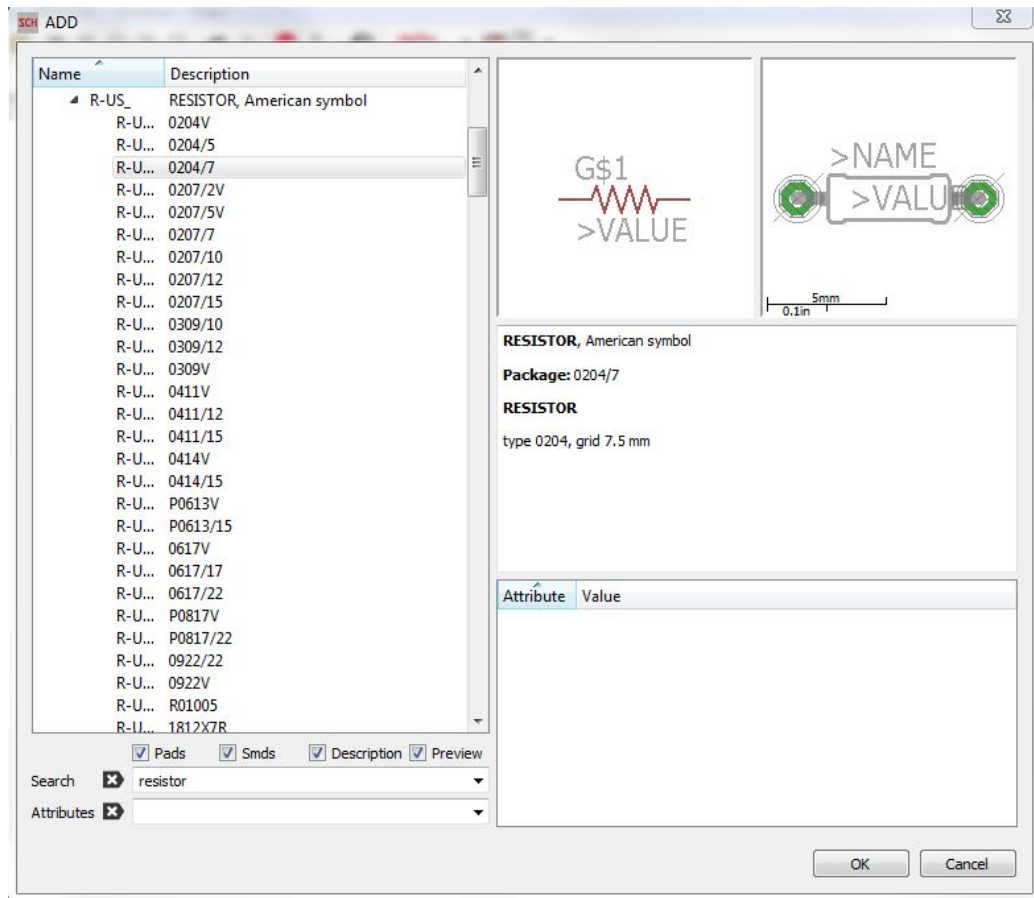


Figure 6: Selecting a Resistor

After selecting this resistor, Eagle will now allow for placement of the resistor anywhere on the schematic file. Since the MOSFET circuit also contains NPN and PNP transistors, a Zener diode, and an N-Channel MOSFET, these will now need to be added to the schematic file in order to move onto the next step in the schematic process. These components are a little more challenging to find. The best way to find these components is to google search the part number in order to find the datasheet. Normally from the datasheet, the package name will be listed. For example, an NPN transistor with a part number of 2N3565 will need to be added to the schematic. From the datasheet, it was found that the package name for a through-hole component is TO92. This is a very useful piece of information. Since the package name and the part number of the NPN transistor are now known, the component can be searched in the library just like how the resistor was searched. This is why it is very important to have a

schematic with all of the components labeled with their respective part numbers prior to developing in Eagle. Figure 7 below shows the correct NPN transistor needed.

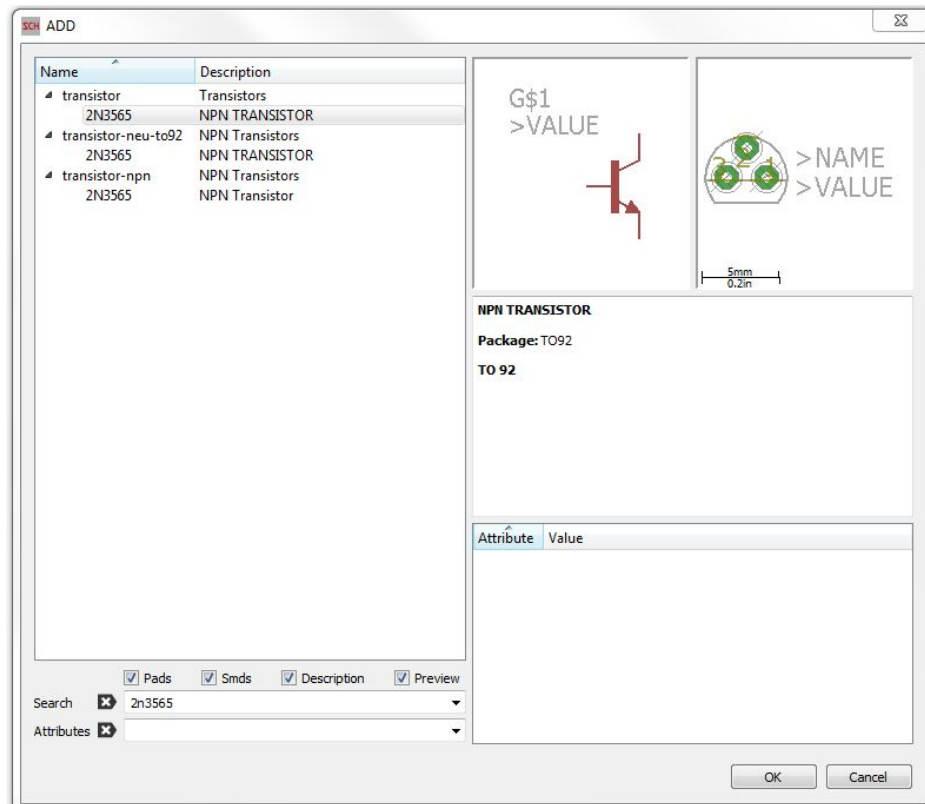


Figure 7: Selecting an NPN Transistor

The same process should be followed for the Zener diode and the N-Channel MOSFET. After adding these components, the final component to add is a terminal block in order to connect the MOSFET circuit to a power supply and ground.

Since the specific terminal block required for the circuit is not in any of the Eagle libraries, a footprint will have to be manually created. This process is a little more challenging than just adding a component from an Eagle library.

The first step in creating a component is revert back to the control panel window shown in Figure 3. Right click on the MOSFET Circuit project folder and select "New" and then "Library". This will open up a file that contains three separate sections: Device, Package, and Symbol. This is where the terminal block will be created.

The first step is to create a package. At this point in the process, it is very important to have a copy of the component datasheet open in order to locate and understand the component dimensions. Figure 8 below shows the dimensions of the terminal block from the datasheet.

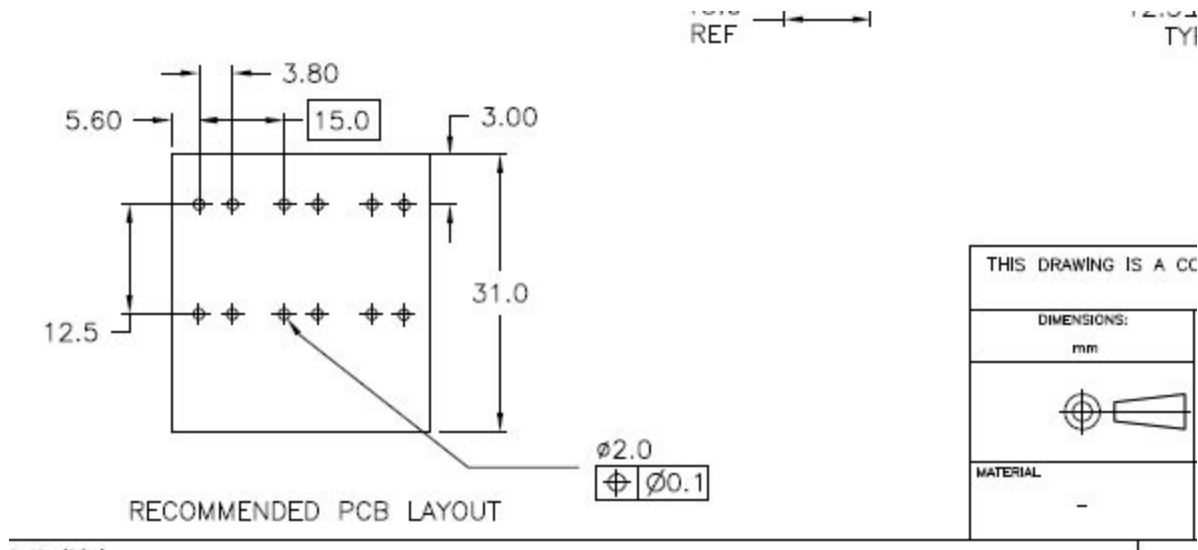


Figure 8: Terminal Block Dimensions

It is also important to know the units used in the datasheet. Next, select the package symbol as shown in Figure 9 below.

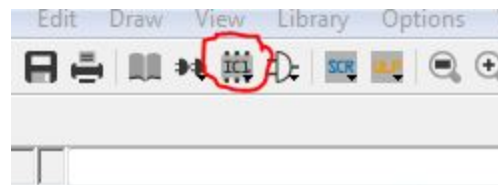


Figure 9: Package Symbol

In the next page, it will ask to enter a name for the package. Enter any name that makes sense. For this schematic, “TERM_BLOCK” was entered. Then select “Yes”. This will open up a board layout where the package will be constructed according to the dimensions. Before building the component, it is important to change the units so that they correspond with the units used in the datasheet. This will make it a lot easier to create the component. Since the dimensions are in “mm” on the terminal block datasheet, the units were switched to “mm”. How this is done is shown in Figure 10 below.

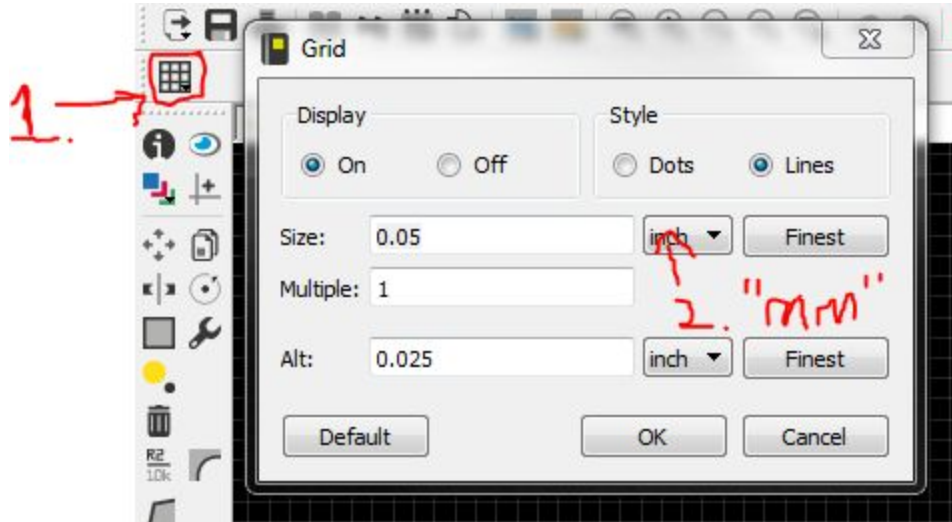


Figure 10: Changing the Grid Units

The “size” is the size of the squares on the board layout. Right now, the squares are set to 0.05 inch x 0.05 inch. These were changed to 0.01 mm x 0.01 mm.

The next step is to draw the package outline. This is done using the “wire” button in the toolbox on the left. The outline of the terminal block component should be identical to the dimensions in the datasheet. The board file is laid out using a coordinate system. Therefore, the outline can either be drawn using the wire tool or the vertices of the box can be entered into the formula bar on the top of the board file.

It is easy to understand from the data sheet that in order to make a four circuit terminal block (one more circuit than the one shown on the data sheet), the width of the package needs to be 60 mm and the length needs to be 31 mm. The wire tool is utilized to draw a box with these dimensions as shown below in Figure 11.

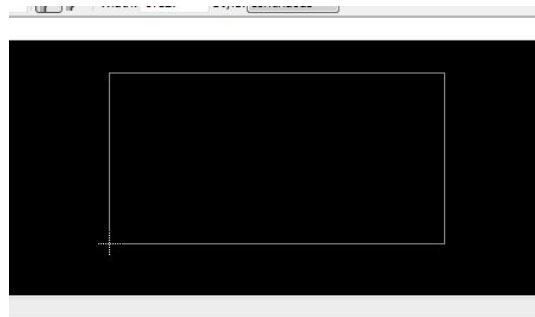


Figure 11: Outline of Terminal Block

The next step is to add pads to the terminal block package. To do this, you need to select the “pad” button in the toolbox on the left. From the datasheet, you can find where each pad should be placed based upon the location of each pin. Figure 12 below shows the terminal block pad placement using the information from the datasheet referenced earlier.



Figure 12: Terminal Block with Pins

This is the end of the package creation. The user also has the ability to add text to the package using the “T” button in the toolbox.

Now that the package is created, it is time to create the symbol. Click on the button that says “symbol”. This will open up a box where you can enter in a name. Choose the same name that was used for the package.

Next, select the “pin” button in the left toolbox. Add the same amount of pins to the schematic as there are pads in the package. For the terminal block, the 4 left-most pins are all shorted together, therefore only one pin is needed for that group of 4. The same is true for the 3 groups of pins to the right. This means that there will be 4 total pins in the symbol for the terminal block. After adding the 4 pins to the diagram, the next step is to draw a rectangle. This is done with the wire tool. Figure 13 below shows the finished symbol.

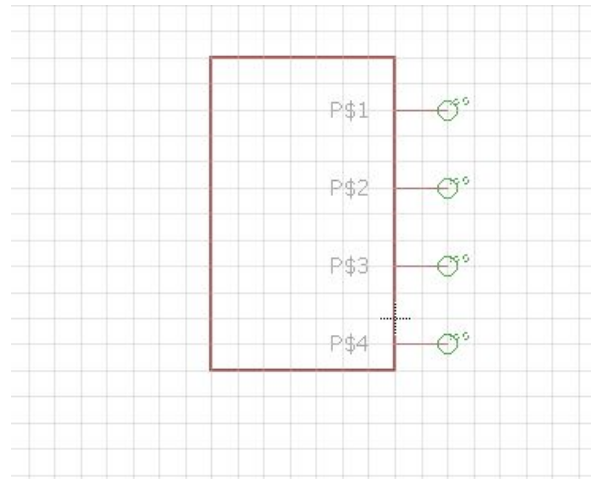


Figure 13: Terminal Block Symbol

Now it is time to create a device. Click on the “device” button in the top left corner. Give it the same name as the symbol and package. The screen will now look like this.

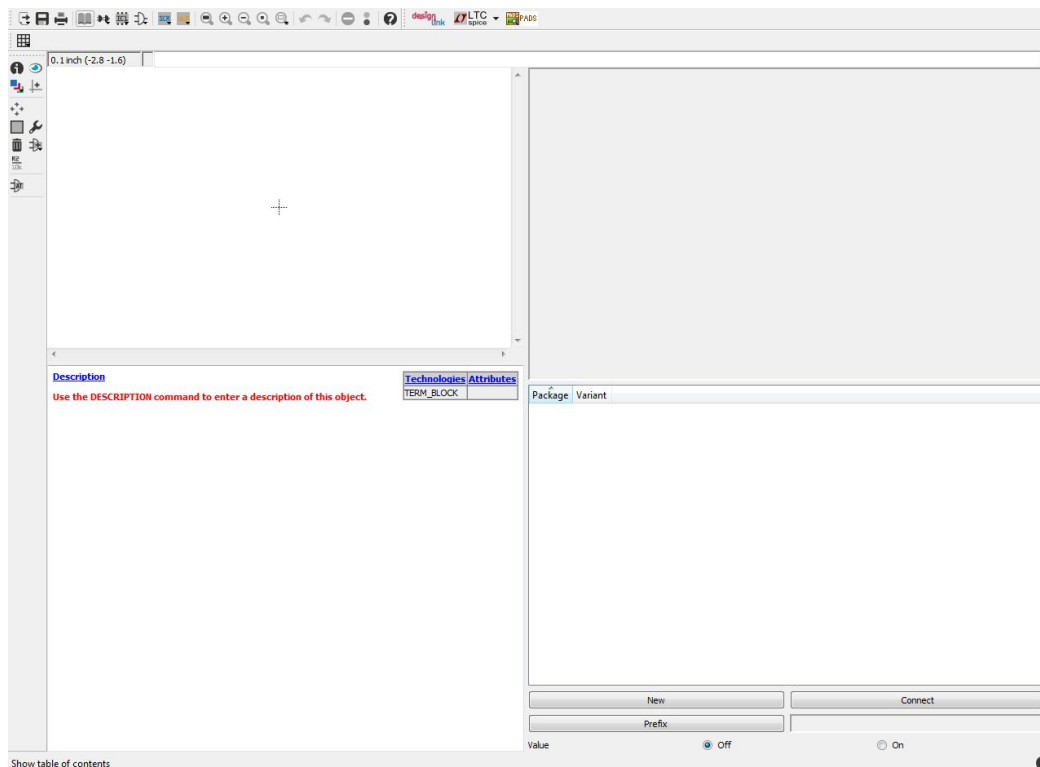


Figure 14: Device Creation Screen

In this screen, the next step is to add a part. From the library, add the part. In this case, the 4 circuit terminal block. The next step is to add a package. Select “New” on

the bottom right and add the terminal block package and give the package variant a name. Now the pins are ready to be connected. Click “connect on the bottom right. This screen will pop up. This is where the pins will be connected.

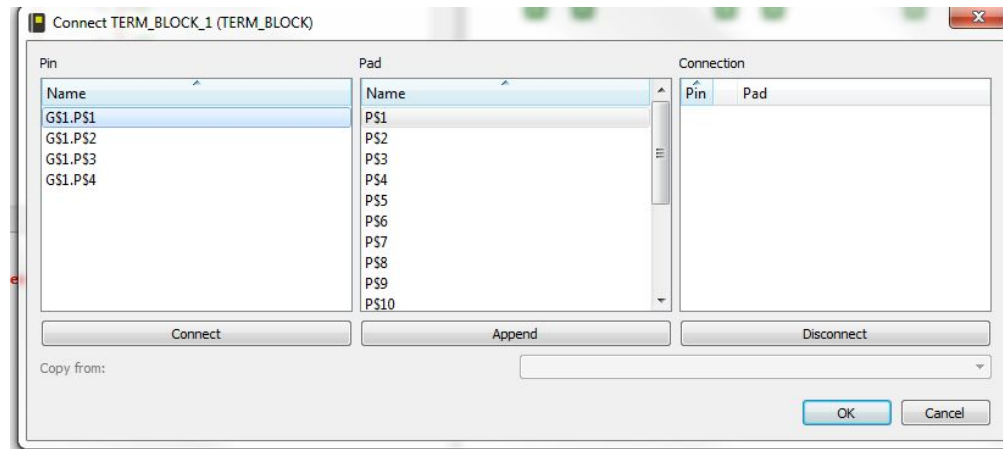


Figure 15: Connecting Pins

Select the pin in the left box and then select the corresponding pad in the middle box. After both are highlighted, select “Connect”. This will connect the pin to the pad. Do the same for the rest of the pins.

The last step is to save the new device under the same directory that the project you are working on is in. The library will now be visible in the control panel.

Building a Schematic

Now that all of the correct components are in the schematic file, the next step in the design process is to wire the schematic. This is done using the “net” tool, NOT the “wire” tool. Figure 16 below shows the location of the “net” tool.



Figure 16: Net Tool

Now it is time to connect the circuit. After looking at the schematic on paper, wire each component to the correct location. Make sure not to overlap the wires on the pins.

A net junction, a little green dot in the schematic file, will show up when a successful connection is made in the schematic. The MOSFET Circuit schematic is shown in Figure 17 below.

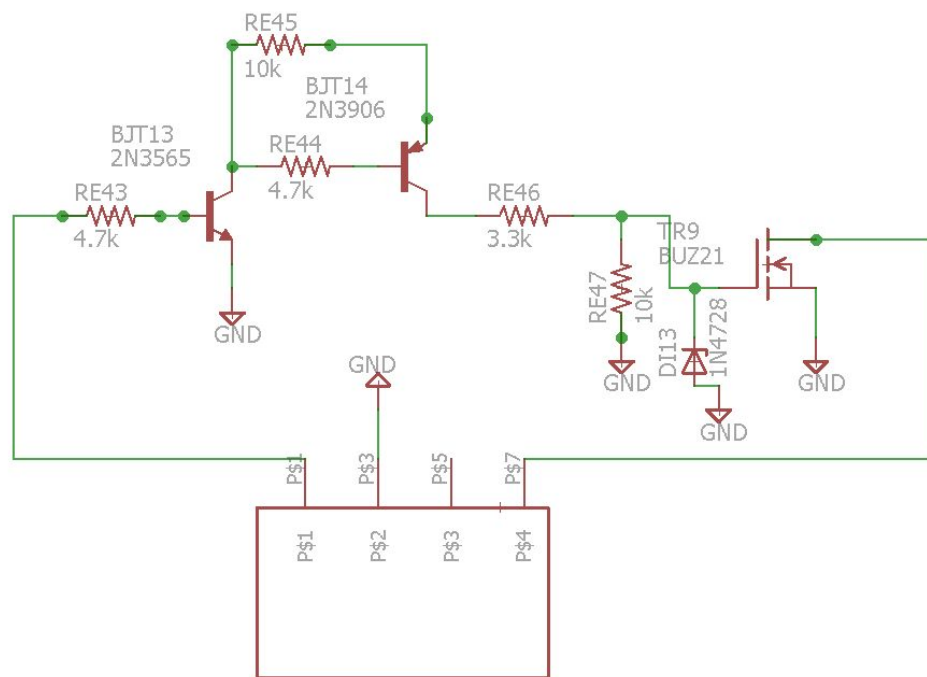


Figure 17: MOSFET Circuit Schematic

Now that the schematic is created, it is possible to add names to each of the components. This is done using the “name” button in the toolbox. This will make placing the devices in the board file much easier.

Now that the schematic is created, it is now time to move to the final part of the schematic assembly, testing the schematic for connection errors.

Testing a Schematic

This section of process is arguably the most crucial step. It makes sure that all of the connections are verified and that there are no errors before the transfer of the schematic file content to the board file. It is important to make sure and double check that the schematic file is 100% error free before routing the board file. This is done using the “Electrical Rule Check” system. The button for this can be found in the toolbox as shown in Figure 18 below.



Figure 18: ERC and Error Buttons

By pressing the left button, the system will run an electrical rule check. If there are any errors, they can be seen by clicking the caution button on the right. This will bring up a window like the one shown in Figure 19 below. This is where any connection errors will be shown, whether it be a wire overlap or an unconnected pin.

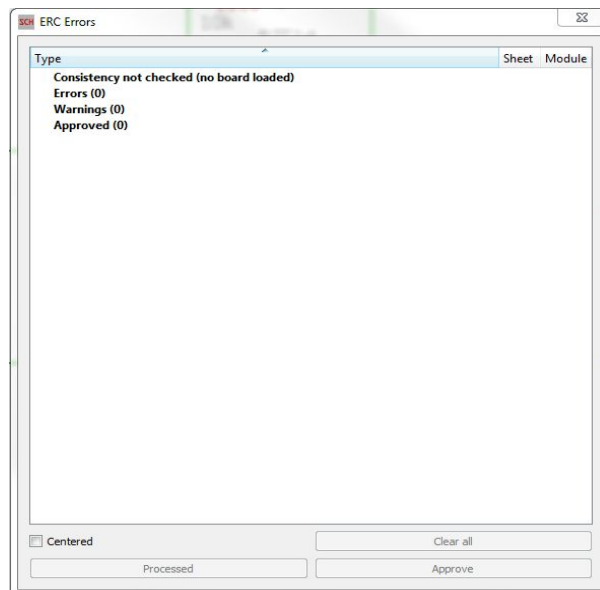


Figure 19: ERC Error Window

Once the ERC Error Window shows zero errors and you know that all of the connections are correct, then you are now ready to transfer the schematic file to the board file.

Wiring a schematic in Eagle is relatively simple if you understand all of the functions that are available to use. It is important to check each and every connection before moving to the board development. Without a correct schematic, the board will not work. Make sure that you pay close attention to detail while wiring the schematic in the schematic file. This will make the entire design process much easier moving forward.

References

1. <http://www.cadsoftusa.com/eagle-pcb-design-software/about-eagle/>
2. https://learn.sparkfun.com/tutorials/using-eagle-schematic?_ga=1.167182974.1166911490.1442352025