

MICHIGAN STATE UNIVERSITY COLLEGE OF ENGINEERING

Eagle use in Prototyping

Eleazar Gutierrez

4/3/2015

Facilitator: Dr. Binseng Wang

Design Team 12

Sponsor: MSU Solar Car Team

Abstract

In the development of any product it is necessary to develop prototypes to test key ideas and concepts. The design for team 12 is no different requiring several iterations of prototypes to ensure the design is on track to work properly. The team decided to use CADSoft Eagle software to develop breakout boards for parts because the components are surface mount devices. This makes working with components easier because the leads do not

Objective

This application notes is written as a way to help students develop their prototypes using CADSoft Eagle. Eagle is a powerful tool that allows students to quickly and economically develop their prototypes. Additionally, Eagle is the software that is recommended by the Electrical and Computer Engineering Shop to develop prototypes. There are a number of sources for students to use but this aims to consolidate some of those sources into a more compact document.

Getting a Version of EAGLE

Eagle is a commercial product sold by CADSoft Eagle. However, students can acquire a light version of the software for educational purposes. The light version has some limitations that include only being able to develop boards that are 100 mm by 80 mm or 4 by 3.2 inches [1]. Additionally only two signal layers can be used a top and a bottom, and the schematic editor can only create one sheet.

Selecting components in Eagle

The student version of Eagle comes with a pre-determined library. However, this library is not an exhaustive list of all known parts. Sometimes the desired components are not in the default library and a student can either make their own library file or can find their necessary library file on the internet. In particular element 14 offers extensive libraries for different companies ranging from Texas Instruments, Microchip, Freescale and many more.

In order to begin choosing parts the user needs to click on the “ADD” button on the left sidebar. This will bring up the menu shown in figure 1. Parts are listed in alphabetical order but also allow users to type and search for a specific part.

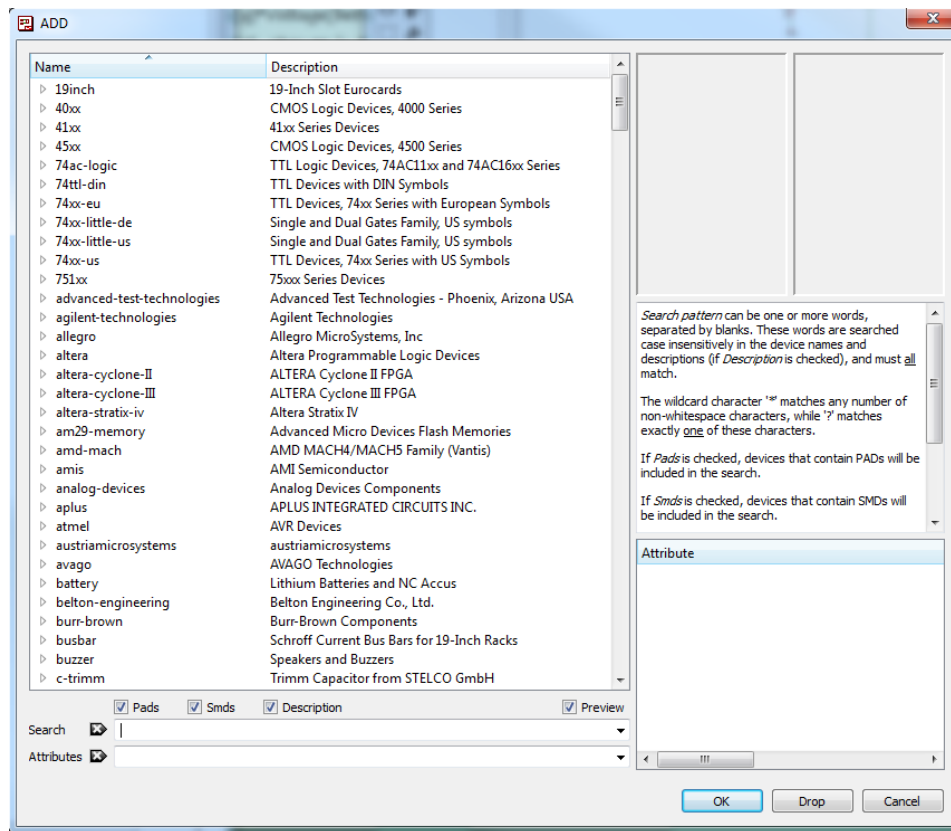


Figure1: ADD Menu

Once the correct parts have been chosen the devices can be wired using the “Wire” command on the left hand menu bar. With all the parts selected and wired the user now wants to go to the “switch to board” option in the left hand corner of the screen. This takes the user to the board layout and allows the board to be arranged and routed as necessary.

Inserting ground planes in EAGLE

Open Eagle and start a new schematic, select the parts and then switch over to the board.

Click on the ‘polygon’ icon located toward the bottom of the toolbar. It is possible to choose what layer the ground plane is placed on by clicking on the toolbar in the left hand side called ‘Layer Settings’.

Route the polygon in the desired shape. Make sure to close the polygon at the starting point or there may be problems generating gerber files later in the process [2]. At this point a

polygon with a dashed line will appear. Once you have drawn the desired polygon click on the “Ratsnest” icon and the polygon will fill with copper [2]. This process is shown in figure 2.

In order to ensure that the placed polygon connects the proper pins of the components to the ground plane it is necessary to name the polygon the same name given to the ground plane in the schematic. This is done either by typing in ‘name’ on the eagle toolbar or right clicking on the polygon and clicking on name. This brings up a small menu and allows the user to change the name. This menu is shown in the figure 3.

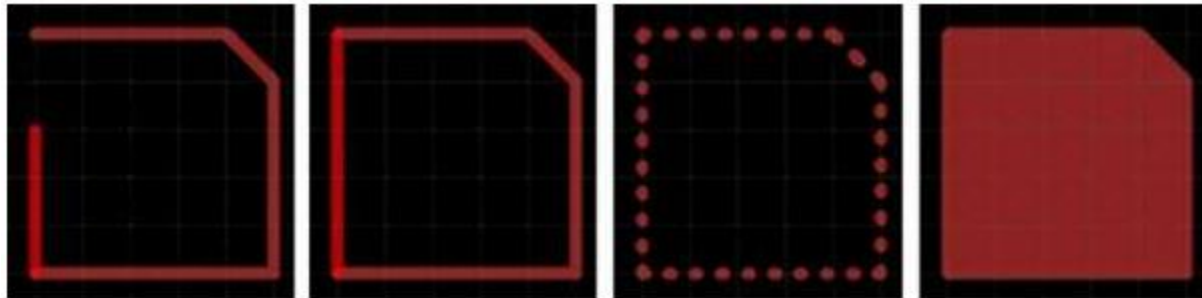


Figure 2: Polygon Construction Process

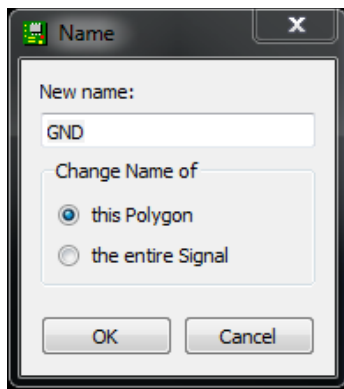


Figure 3: Name Menu

Design Rules in Eagle

Eagle has several built-in functions that allow a user to check and see if they have committed any mistakes. To begin with it has a Design Rule Check command in the board editor that allows the user to investigate the layout of their board. To access this feature the user needs to click on “Tools” in the menu at the very top of the screen and click on “DRC”. This opens a menu as shown in figure 4.

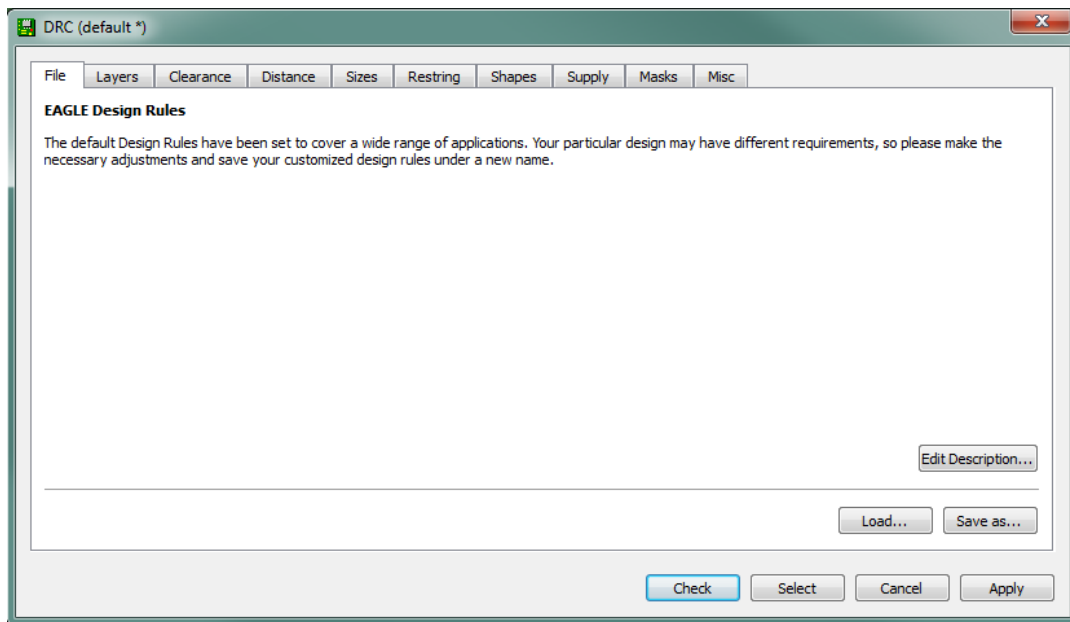


Figure 4: DRC Menu

The “File” tab allows the user to pick which DRC file to use. Different companies or printed circuit board houses allow users to download a DRC file with their specific requirements. These requirements must be met before they fabricate the PCB [3].

The “Clearance” tab tells Eagle how much space the user wants between the different traces on the board. It also defines how wide a cut can be. For all circuits fabricated by the Electrical Engineering shop, cuts must be at least 12 mils [3].

The “Sizes” tab sets the minimum sizes. Minimum width is a number that varies depending on the fabrication house. If the user is etching the board the width should be 24 mils. Minimum Drill is also something that varies depending on the fabrication house. Again, if the user is etching the board they should select the smallest bit they have [3].

The “Masks” tab allows the user to control the size of the openings in the stop mask. The stop mask is the only way to access the pads or vias once the PCB’s have been fabricated. This is because once fabricated the PCB’s have a coating that covers and insulates all of the board [3].

The “Distance” tab allows the user to choose the distance from any routing to the edge of the board. The “Drill/Hole” allows the user to change the distance from any routing to a hole in the board [3].

Eagle also has a function called Electrical Rule Check (ERC). This tool helps users find any problems that may be present in the schematic view of the design. To access this feature the user can click on the “Tools” tab on the top menu and choose ERC. This opens a tab shown in figure 5. The ERC helps detect inconsistencies between the implicit power and supply pins in the schematic [4].

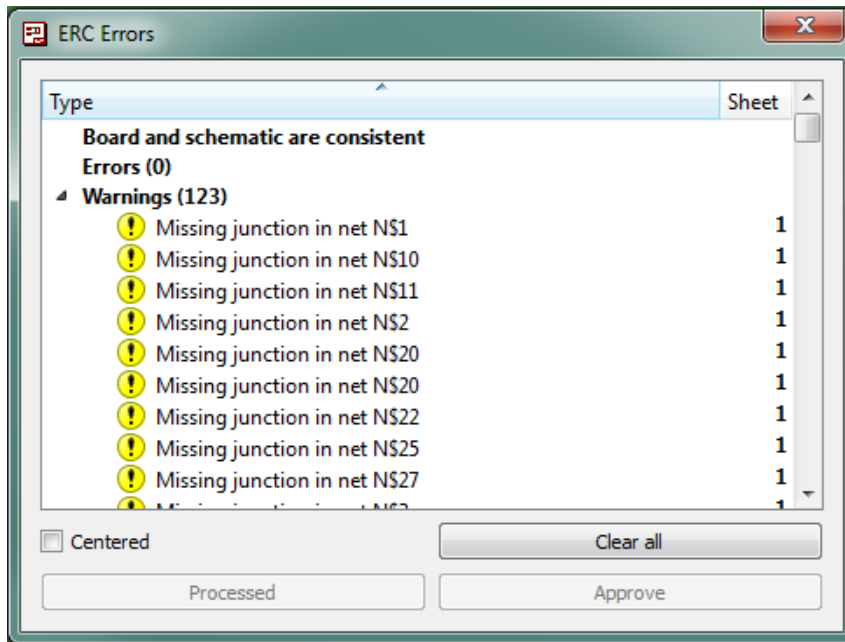


Figure 5: ERC Menu

Generating File for Printed Circuit Boards to be cut by the ECE shop

In order to prototype printed circuit boards it is necessary to download the Eagle CAM job file. Once the files have been downloaded it is necessary to unzip the file into the “Eagle/cam” directory. If it is not possible to unzip in this directory then the user can unzip the files in a directory where they have writing privileges.

Once this has been done the user needs to be on the board editor in order to generate the boards. By clicking on the CAM Processor in the right hand corner there is a menu that will appear as shown in figure 6.

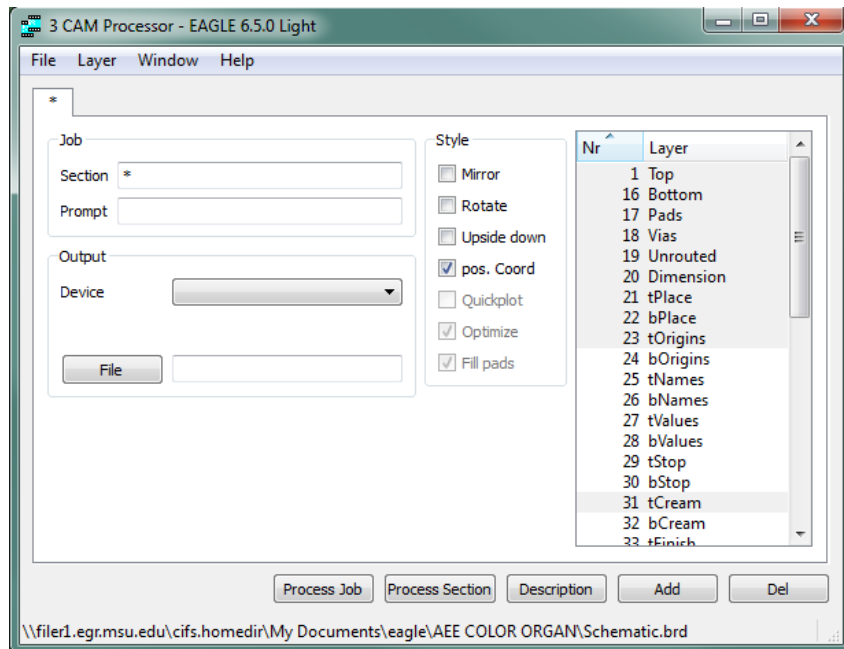


Figure 6 : CAM Process menu

Conclusion

In this application note a wide variety of different subjects were presented. Students were given instructions on how to acquire a student copy of the software at no cost. The selection of components and libraries was also discussed. Next more complicated subjects were covered such as adding a ground plane, generating boards to be cut by the ECE department and design rules within eagle. CADSoft Eagle is a very useful tool that allows students to develop prototypes quickly and economically. Its ease of use allows users who have little to no experience making printed circuit boards to learn and become proficient in developing prototypes.

References

- [1] Cadsoftusa.com, 'Downloads', 2015. [Online]. Available:
<http://www.cadsoftusa.com/download-eagle/>. [Accessed: 03- Apr- 2015].

- [2] Dangerousprototypes.com, 'HOW-TO: Polygons and ground fills for PCBs in Eagle | Dangerous Prototypes', 2015. [Online]. Available:
<http://dangerousprototypes.com/2012/07/18/eagle-polygons/>. [Accessed: 03- Apr- 2015].

- [3] Instructables.com, 'The Design Rule Check', 2015. [Online]. Available:
<http://www.instructables.com/id/PCB-Creation-with-Eagle-for-Beginners/step17/The-Design-Rule-Check/>. [Accessed: 03- Apr- 2015].

- [4] Web.mit.edu, 'EAGLE Help: ERC', 2015. [Online]. Available:
http://web.mit.edu/xavid/arch/i386_rhel4/help/44.htm. [Accessed: 03- Apr- 2015].