

### EXPERIMENT PCB1: PCB Layout Design 2

#### OBJECTIVES:

Learn the basic process of PCB layout design using software tools.

#### EQUIPMENT:

PC with access to PCB design software (e.g. EazyEDA.)

#### INSTRUCTIONS:

1. Load the project from the previous experiment, "PCB Layout Design 1".
2. Convert the schematic into a PCB layout. Refer to "Basic guide to EazyEDA 2" at the end of this manual to learn more about PCB layout design. Note that the circuits given in the guide are just an example and not the actual circuits that you need to design. This is an open-ended experiment, assume your own values (provide justifications) for any parameters that are not specified.
3. Make the necessary modifications to the PCB layout to ensure there are no design rule check errors.
4. Screenshot your final PCB layout to be included in the report.

#### REPORT:

1. Attach the screenshot of the PCB design and provide some justification for the design choice.
2. Conclude the things learned, problems encountered, and how it was overcome.
3. Answer all questions at the Questions section.

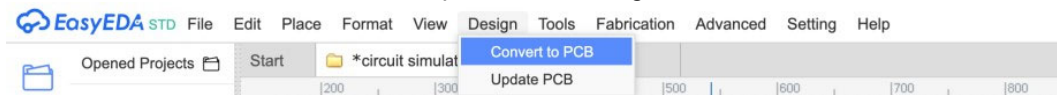
#### QUESTIONS:

1. View the following videos for PCB design in Solidworks, AutoCad/AutoDesk.  
<https://www.youtube.com/watch?v=W1AJOOREcP0>  
<https://www.youtube.com/watch?v=lwpU6uJ-DPU>  
<https://www.youtube.com/watch?v=VZZBEocoYDA>  
How do they compare to EasyEDA? Which one do you prefer and why?
2. What is PCB Design Rule Check and why is it needed?
3. What can you conclude regarding the PCB design process?

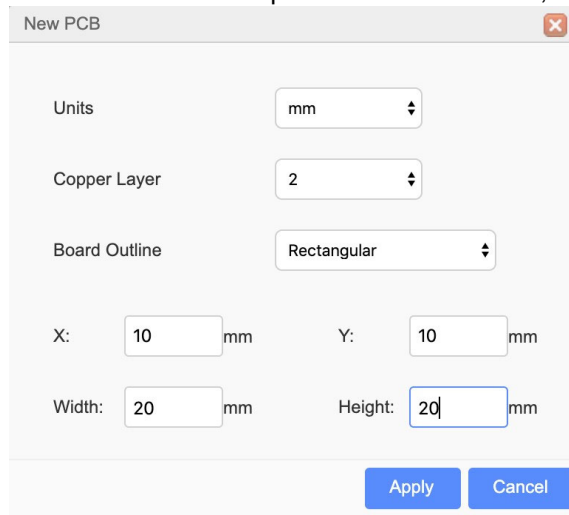
**END OF EXPERIMENT**

### Basic guide to EazyEDA 2:

1. Remove all measurement equipment in the schematic, if any.
2. Save your schematic and switch to “Standard Mode” to begin the PCB design.
3. Convert the schematic to PCB via “Top Menu” > “Design” > “Convert to PCB”.



4. Select the desired shape and size of the PCB, and click the apply button.



5. Rearrange the components manually or use the “Auto Router”. Auto Router is a convenient way to arrange components but it may introduce some errors, so manual checking may be necessary.
6. Check for potential errors regularly during the design process using “Check DRC” via “Top Menu” > “Design” > “Check DRC”. If any error occurs, solve them one by one.
7. Once your PCB design layout is error-free, it is possible to export the file in the desired format. The exported file can be used to fabricate the PCB. For this experiment, it is not required to fabricate the PCB.