PCB Submission and Ordering Process

1.0 Introduction:

During a semester of ECE477, students are required to design, fabricate, and assemble one or more printed circuit boards (PCBs). At the time of this writing, certain resources are available for students to fabricate and build their first circuit boards. The ordering of printed circuit boards is a time-critical step, and a series of steps must be undertaken to ensure that printed circuit board is ordered, fabricated, and delivered in a timely manner. This process document is intended to describe the course resources available to students for printed circuit boards, as well as the process necessary to order printed circuit boards.

1.1 ECE477 Printed Circuit Board Resources:

At the time of this writing, ECE477 provides the following resources to student teams for printed circuit boards:

- For 2-layer PCBs, the first PCB and related shipping costs are free

- For 4-layer PCBs, the first PCB and related shipping costs are free

The following caveats apply to printed circuit boards:

- If a team requires multiple PCBs or multiple copies of the same PCB, contact course staff

- If a team requires a large PCB (greater than 60 sq. inches), contact course staff

- In the event of very late orders, expedited processing or shipping may be required. Should these circumstances arise, contact course staff

- ECE477 resources apply to the initial iteration of a printed circuit board. Students are responsible for the fabrication and shipping costs of additional printed circuit boards. In the event that an additional iteration is required, please see section 3.

- ECE477 does not cover the costs of circuit boards with internal cutouts (other than drill holes)

- At the time of this writing, ECE477 does not provide resources for other types of circuit fabrication, such as flexible PCBs. If a student design calls for a special type of PCB, students are directed to contact course staff.

2.0 PCB Ordering Process

Students looking to order their initial printed circuit boards should follow the following steps in order:

1. Once PCB is routed, perform an initial check of the board. All of the following sub-items should be accounted for:

    1. The board is fully routed (no airwires).

    2. The team number (ECE477 Group X, where X is the team number) and board revision number, version, or date of last modification is included on ALL printed circuit boards being submitted.

3. The board outline is included in the silkscreen layer.

2. If EagleCAD is being used for PCB design and layout, download the ECE477 Eagle DRC file and run a design rule check on your design. The DRC file is included with this process document on the ECE477 course webpage. Correct any design errors produced through Eagle's DRC.

3. If EagleCAD is being used for PCB design and layout, download the ECE477 Eagle CAM file and run the CAM process on your design. The CAM file is included with this process document on the ECE477 course webpage. (Please note, the CAM file requires the creation of a directory called "Gerbers", in the same directory as the Eagle source files for a given board).

4. In the same directory as the Gerber files output by the CAM processor, create a file called "readme.txt". This file should include the following information:

   1. The dimensions, in inches, of the PCB. Should the PCB not be rectangular, include the maximum dimensions

   2. The total area of an individual board

   3. The number of copies of a given board desired

5. Zip all files produced by the CAM document into a zip archive and submit this archive to the [FreeDFM](#) design rule checker. Any potential showstoppers must be corrected, then repeat steps 2-4 until the returned FreeDFM report contains no showstoppers.

6. Repeat steps 1-5 for each unique board design required by your project (note: create a separate zip archive for each board design; DO NOT combine multiple designs into a single zip file)

7. Send zipped board archives that successfully pass FreeDFM to the ECE Instructional Laboratory submission script (eceil@ecn.purdue.edu). For the zipped board archives being sent to ECEIL, please include the no showstoppers screenshots within the archive. The email should have the following format:

   To: eceil@ecn.purdue.edu
   CC: ALL team members
   Subject: [ECE477 PCB Submission] Team XX PCB Y of Z
   Body:

   1) Attach zips file to the email, one zip file per board requested. Please include all PCBs you intend to order in each email to ECEIL; a newer email will completely overwrite the board contents of older emails.
   2) Include the email and phone number of a primary and secondary contact for the printed circuit board, should any questions concerning the board arise
   Note: Please do not include any non .zip attachments in your emails

8. ECEIL will return a confirmation email within 15 minutes verifying that the board submission was accepted

3.0 PCB Respin Process

Under certain circumstances, it may be necessary for a student team to order a second iteration of one or more printed circuit boards. Under these circumstances, students are directed to follow the PCB respin process, which is described below:

1. Once PCB is routed, perform an initial check of the board. All of the following sub-items should be accounted for:

   1. The board is fully routed (no airwires).

   2. The team number (ECE477 Group X, where X is the team number) and board revision number, version, or date of last modification is included on ALL printed circuit boards being submitted.

   3. The board outline is included in the silkscreen layer.

2. If EagleCAD is being used for PCB design and layout, download the ECE477 Eagle DRC file and run a design rule check on your design. The DRC file is included with this process document on the ECE477 course webpage. Correct any design errors produced through Eagle's DRC.

3. If EagleCAD is being used for PCB design and layout, download the ECE477 Eagle CAM file and run the CAM process on your design. The CAM file is included with this process document on the ECE477 course webpage.

4. Zip all files produced by the CAM document into a zip archive and submit this archive to the FreeDFM design rule checker. Any potential showstoppers must be corrected, then repeat steps 2-4 until the returned FreeDFM report contains no showstoppers.

5. Once the PCB design passes FreeDFM checks without showstoppers, students are directed to submit the PCB design to a PCB vendor. At the time of this writing, students are recommended to use Advanced Circuits or OSH Park for PCB orders. Ordering information for Advanced Circuits is available here. Student discounts are available for Advanced Circuits; information about the Advanced Circuits Engineering Student Program is available here. Information concerning OSH Park can be found here.