PCB 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 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.

2.0 PCB Ordering Process

Students looking to order their initial printed circuit boards should follow these 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 (additional, more thorough checks are provided in the PCB Verification checklist, distributed separately):
    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).

    Changes in the backend code of Eagle have necessitated the creation of a separate CAM file to be used for students using Eagle version 7.2 and newer. If your design was done in Eagle 7.2 or newer, please make use of the alternate CAM file.
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. Complete Assignment D1: Design Submission Checklist (TA signoff is required to receive credit for PCB completion).
8. Once zipped board files have been verified and generated, they should be ordered from the PCB service of the students' choice. Recommended PCB fabrication services include OSHPark and Advanced Circuits. Alternatively, PCBShopper is an excellent tool which students can use to compare different PCB services and choose viable options. Students should order circuit boards which can be in-hand within 10 business days.
9. Following PCB order, students should receive confirmation emails that their order has been received. These emails should be forwarded to the ECE477 account (ece477@ecn.purdue.edu) with the subject heading:

    ```
    [ECE477] Team x Completed PCB Order for PCB y of z
    ```

    Where x is your team number, y is a numerical designator for your board, and z is the total number of unique PCB designs utilized by your project.