CET246 Electronic Design Automation
David J. Broderick, Ph.D
Laboratory Exercise #1: Generate Gerber, Drill, and Map Files

What to do:

1) Unzip the example project into a directory which you have write access to.
2) Open KiCad
3) Open the example project
4) Open the PCB layout in Pcbnew
5) Plot the following Layers:
    a. Front Copper
    b. Back Copper
    c. Front Silk Screen
    d. Back Silk Screen
    e. Front Soldermask
    f. Back Soldermask
    g. Edge Cuts
6) Generate the drill file
7) Generate the Map file as a PDF
8) Open GerbView
9) Load all gerber files
10) Verify that all PCB layers are present

What to turn in:

1) A single zip file with all gerber, drill, and map files. Name the file according to this format:

    Course_Semester_YourLastName_Lab##.zip

    So I would name my file:

    CET246_Fall2018_Broderick_Lab01.zip