

CET246 Electronic Design Automation
David J. Broderick, Ph.D
Laboratory Exercise #4: Footprint Creation

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 Schematic in Eeschema (within KiCad)
- 5) Run Pcbnew from the toolbar in EESchema
- 6) Create a new footprint library
- 7) Create footprints for the three missing parts in your new library
- 8) From Eeschema run CvPcb and associate the new footprints with the appropriate part designation
- 9) From Eeschema re-generate the netlist in the 'Pcbnew' format
- 10) Run Pcbnew from the toolbar in EESchema
- 11) Import the netlist in Pcbnew from the toolbar
- 12) Arrange the components
- 13) Make all traces necessary to remove the ratnest. Use the appropriate layers
- 14) Draw the board edge on the **Edge Cuts** layer. The board shall be no larger than 100mmx 70mm.
- 15) Include mounting holes
- 16) Run DRC (bug) check with default design rules and resolve all errors
- 17) Generate all gerber, drill, and map files as done in the previous lab

What to turn in:

- 1) A single zip file with all PCB, library, 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_Lab04.zip