CET246 Electronic Design Automation
David J. Broderick, Ph.D
Laboratory Exercise #6: Schematic Capture

What to do:

1) Open KiCad and start a new project
2) Create a new schematic in Eeschema (within KiCad)
3) Redraw the attached motor controller circuit
4) Run CvPCB and associate all components with a footprint.
5) Generate the Netlist from within Eeschema
6) Run Pcbnew from the toolbar in EeSchema
7) Import the netlist in Pcbnew from the toolbar
8) Arrange the components
9) Make all traces necessary to remove the ratnest. Use the appropriate layers
10) Draw the board edge on the **Edge Cuts** layer. The board shall be no larger than 100mmx150mm.
11) Include mounting holes
12) Run DRC (bug) check with default design rules and resolve all errors
13) 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_Lab06.zip

