PCB Final Design Requirements

- ✓ Create a detailed schematic using Fusion360- Print the Schematic Design >use the Document View> Library>Print> Print the pdf
  Update OneNote-
  Section: HWK4
  Page: Fusion360 Schematic

- ✓ Create a PCB Design using Fusion360- Print the pdf design of the PCB *using the Document tab>Outputs>Print pdf*
  Update OneNote-
  Section: HWK4,
  Page: Fusion 360 PCB

- ✓ Print the Bill of Materials (Parts) list using *the Document tab>Library >Output> (next to printer Icon) Bill of Materials tabs in Fusion 360 Schematic Editor and install in OneNote*
  Update One Note-
  Section: HWK4
  Page: Bill of Materials

- ✓ Save the Gerber file from the PCB Design view using the Manufacturing Tab> under Manufacturing Drop down list>Export Gerber’s or highlight the Export Gerber Icon> Rename the file to include your Group Designator
  Update One Note-
  Section: HWK4
  Page: Gerber’s

When Creating PCB’s these items are to be included for grading requirements.

❖ Your PCB Requirements

- 2.000" x 3.000" (ECE401 only) actual size is whatever you require in ECE405.
- Mounting holes 175 mils in each corner (Diameter)
- Power & Ground Traces: 40mils (required ECE401 only)
- All Other Traces: 20mils (Required ECE401 only)
- Ground plane on the bottom side of PCB
- Silk-Screen designators in correct order
- Board must show the project name /Board name and Vs.#/ Date
- (Font15 recommended for font size)
- If your proud of your work- include your name or Initial it.
- Test points should be available for measurement; example {9V, 5V, ground, Input, Output, and Collector(s)}

- ✓ Save the Gerber's and send via email for order processing to jeffrey.erickson@ndsu.edu.
- ✓ Recommend first sending your files to a Gerber viewer. They are free on the net. If you send your files to JLCPCB.com, there is a Gerber viewer. By layer and by 3d imaging.

Helpful Hints:

Grid settings: Lower Middle **OBJECT Properties Bar**>Grid settings> Size: 100mil, display on/off, style-Dots/Lines-------Suggest leaving in 100mils or 2.54mm spacing, (many parts do snap better in 50mils)

Board Size: How to change board size: In PCB Document View Highlight outer edge of black drawing surface (using the mouse highlight the outer edge of the Design box, change dimensions as needed

Vias: placing vias in PCB Design View >Route>VIA> select and place via or **while Manual routing stop where required and press the spacebar- 2nd press will change top to bottom trace**

Holes: creating mounting holes: in **PCB Design View>PLACE> Hole (NPTH)** Non-plated through hole- or it will short top layer to bottom layer

Top and Bottom traces: *switching from top and bottom traces-**change selection from Layers tab***

Track width: Changing width/size---- **Highlight the track**>Right click>Properties>Change Width size

Silkscreen: Adding Names/ board names >In PCB Design View >Document>Draw>TEXT> Under TEXT Properties page, type in the text> change the Font size> Must *be in correct Layer (21 Silkscreen Top or 22 Silkscreen Bottom)*

Changing Parts> This should be done from the schematic editor

- ✓ Auto Routing >Quick Route>Auto Router>Continue>Start> **Choose Variant that you like**>End Job

Ground Plane>PCB Design Editor> **Change Layer to Bottom** Layer in layers tab> then--Polygon> Polygon Pour > enter GND in the POLYGON POUR Editor page- then Done. Use Mouse to highlight the outer box of your PCB Design Note to edit it Highlight the images and delete Polygon Pour