CHECKLIST | KiCad Workshop

Item	Result
1 Schematic	
• Create mounting holes	☐
• Create connector circuits	☐
• Create buck converter circuit	☐
• Create LED circuits	☐
• Create microcontroller circuit	☐
• Troubleshoot ERC (Electrical Rules Checker)	☐
• **Assign footprints**	☐
• Optional: Organize with text boxes and component values	☐
2 Board Designer	
• Check board setup and set netclasses (assign power netclass to VBUS and +3.3V nets [0.2, 0.6, 0.6, 0.3])	☐
• "Update PCB from Schematic" button in top row (F8)	☐
• Create board outline on Edge Cuts layer **(less than 100mm by 100mm)**	☐
• Optional: Fillet edges	☐
• Place components in the board outline (Consider what placements will make your life easier)	☐
• Use copper fill for ground plane on T.Copper layer and B.Copper layer	☐
• Route each net	☐
• Optional: Optimize routing (Avoid using 90 degree turns, shorter traces = better, minimize # of vias)	☐
• Run DRC and troubleshoot errors/warnings	☐
• Optional: Add silkscreen text with Project Name, your name, and date.	☐
3 Gerber Files	
• Hit "B" to refill any fill zones	☐
• File - > Fabrication Outputs -> Gerbers (.gbr)	☐
• Select and output folder	☐
• Choose layers "F. Cu, F.Silks, F.Mask, B. Cu, B.silks, B.Mask, and Edge.Cuts"	☐
• Select "Use Protel filename extensions" (JLCPCB prefers them)	☐
• Click "Plot"	☐
• Click "Generate Drill Files"	☐
• Select the same output folder as the Plot	☐
• Click "Generate Drill File"	☐
• Optional: Double check files with a Gerber viewer (KiCad has one)	☐
• Compress the Gerber and Drill files into a Zip file and email that to **mss3247@iastate.edu**	☐
• Wait untill the final prduct arrives from the board house	☐