

## Assignment 6: 555 Design Project Manufacturing Files

---

In this assignment, you will generate all the files needed to manufacture your design. Please follow all the steps listed in each section

### Schematic Entry and Board Layout

The first step is to **create a new Altium project** and create a schematic that represents your design. Make sure that the components you use match up with the ones you plan to order.

- If the components you are going to use don't have Schematic symbols / PCB footprints already included in the default Altium libraries, you will need to either a) download them from the manufacturer or b) if schematics/footprints are not available online, then you will have to create the components yourself. In most cases, you should not have to do this.

After the schematic is complete, you should begin drafting the layout.

### Setting up Board Design Rules

When laying out your board, there are several design rules you will need to consider. One the course website, you will find a rules file named [ECE1895Rules.RUL](#) and instructions on how to import the rules file ([Importing a Rules File](#)). Please do this before starting your layout

- These design rules are configured to be compatible with the following PCB milling machine:
  - [Protomat S64 Datasheet](#)
- The minimum trace width and spacing we suggest is **10 mils**
- Please note, top metal layer to bottom metal layer vias are not filled with metal. Therefore, **you will have to either manually fill each via yourself with solder or use rivets**. In either case, you should make all via holes large enough to do this. The recommended hole (inner) diameter for vias is **1.0mm (39 mils)**. Here is a link showing how to change the hole size <https://www.altium.com/documentation/altium-designer/pcb-dlg-favoriteinteractiveviasizesformchoose-via-sizes-ad>



## Design Guidelines

While the rules file contains minimum requirements for producing a manufacturable board, it does not enforce “best practices”. In class we discussed several rules of thumb for producing reliable boards. Below is additional information. You are encouraged to apply several of these principles in your design, but incorporating all of them is not required. You will be assessed on the quality of your board.

- **Importance of component placement**
  - <https://www.autodesk.com/products/eagle/blog/top-10-pcb-component-placement-tips-pcb-beginner/>
- **Bypass capacitors**
  - <https://www.autodesk.com/products/eagle/blog/what-are-decoupling-capacitors/>
  - <https://www.microcontrollertips.com/why-so-many-bypass-capacitors-faq/>
- **Impact of trace width on signal integrity**
  - <https://www.tempoautomation.com/blog/how-to-use-your-pcb-trace-width-to-improve-signal-integrity/>
- **Through-hole versus Surface mount tradeoffs**
  - <https://resources.altium.com/pcb-design-blog/why-use-through-hole-technology-in-pcb-design>

If you do decide to use surface mount resistors or capacitors, make sure that they are either in an **0805** or **1206** surface mount packages. The other sizes are too small to solder by hand (<http://www.resistorguide.com/resistor-sizes-and-packages/>)
- **Proper Grounding and ground planes**
  - <https://www.allaboutcircuits.com/technical-articles/pcb-layout-tips-and-tricks-use-a-ground-plane-when-ever-possible/>
  - <https://resources.altium.com/pcb-design-blog/understanding-ground-planes-in-your-two-layer-pcb>
  - <https://resources.altium.com/pcb-design-blog/creating-a-ground-plane-for-your-pcb-design>
  - <https://www.autodesk.com/products/eagle/blog/8-pcb-grounding-rules/>

## Finalizing your design

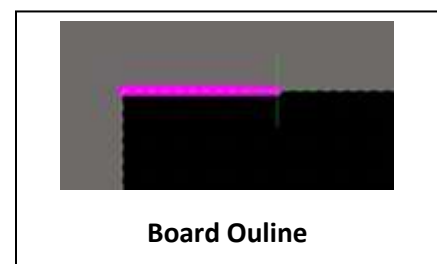
Once your design is complete, you must carry out a few final steps to prepare for manufacturing and file submission.

- **Include your name, or some other identifiable mark, on one of the metal layers.**
  - One way to do that is to simply write your name or initials using wires on the top metal layer.
  - Another possibility is including text on your board as a string and then converting that string to one of the metal layers  
<https://www.altium.com/documentation/altium-designer/pcb-object-string-string-ad>
  - An alternative is to use an image. Here is an article on how to add logos  
<https://www.altium.com/documentation/altium-designer/including-barcodes-and-logos-ad> .

While you are free to include your name and whatever identifying marks you wish, keep in mind that they still have to pass design rule checks since they will be fabricated using one of the metal layers



- **You MUST Include a “Board Outline”.**
  - This outline defines the actual size and shape of the board. The manufacturer will not fabricate your board if this layer is not present. On Canvas there is a guide entitled [Board Outline Guide](#) that explains how to do this. **You must do this step.**
- **Run one final design rule check**
  - To have your board fabricated, you **must** ensure that your PCB passes **all design rule checks** and that no manufacturability or routing errors are present in your design.



## File Submission

- **Export your manufacturing files.**
  - First, create a zip file called **[YOUR\_NAME]\_GERBER\_555\_Project.zip** that contains all of the GERBER files needed to create your design. Online, a guide entitled [\*“Generating Fabrication Outputs”\*](#) shows how to do this.
- **Verify that your design is manufacturable and that the GERBER files were properly exported using an online GERBER viewer.**
  - There are several free GERBER viewers online. One such viewer is found on the frontpage of the website [www.oshpark.com](http://www.oshpark.com). After you Drag and drop the [YOUR\_NAME]\_GERBER\_555\_Project.zip file to the upload window, you should see a preview of your board. If the preview does not generate or looks erroneous, then there is likely an error in your design or with how the design was exported.
- **Upload your files to CANVAS**
  - Create a brief **PDF document** that has screenshots of your Altium schematic and PCB Layout
    - i. Briefly explain your design.
    - ii. You must incorporate at least one PCB design “best practice” and explain how you did so.
  - **[YOUR\_NAME]\_GERBER\_555\_Project.zip** (Project GERBERS in a zip archive)
  - **[DESIGN\_NAME].PcbDoc** (Altium Layout)
    - The is the PCB layout file associated with your project. This file can be found in your Altium project folder. **MAKE SURE THAT YOU UPLOAD THE MOST RECENT PCBDOC FILE, THE ONE THAT CORRESPONDS WITH YOUR LATEST DESIGN REVISION. IF YOU DO NOT, THE WRONG DESIGN WILL BE MANUFACTURED.**
  - **DO NOT ZIP THESE THREE FILES TOGETHER INTO A SINGLE ARCHIVE, UPLOAD THEM AS 3 SEPARATE FILES**
  - **If there were any BOM changes, make sure to notify your instructor so that you have all of the parts needed to assemble your design.**