

ECE 491 Lab 4

Altium Designer Introduction

1. Introduction:

The purpose of this lab is to get familiar with the printed circuit board CAD software, Altium Designer. The tutorial is provided on Blackboard as a 46 page PDF titled, "Altium_Designed_Introduction".

The groups will demonstrate the following for checkoff:

- Create the astable multivibrator schematic with correct values and successful compilation.
- Manually route the printed circuit board design, based on the schematic, which complies with the provided DRC file.
- Generate Gerber and NC Drill files of the circuit design.

2. Advanced Circuits (4PCB) Design Rules:

Add these values into your project's design rules.

Design Rule	Value
Clearance	6 mils
Routing Minimum Width	6 mils
Routing Maximum Width	N/A
Minimum Annular Ring	7 mils
Hole Size Minimum	15 mils
Hole Size Maximum	N/A
Minimum Solder Mask Sliver	6 mils
Solder Mask Expansion	0 mils

3. Creating Gerber and NC Drill Files:

a) Gerber Files:

After we have fixed all of our DRC errors, there are two types of files we need to generate for our design. The first are called the **Gerber files**, which are a collection of the different layers and the designs

that make up those layers. The second are the NC Drill Files which contain information regarding holes/vias that are drilled through the board. **Before we create them, it is important you have done a design rule check and verified that your footprints match your real components size .**

To create the Gerber files click **Files->Fabrication Outputs->Gerber Files**.

After selecting the **Gerber Files** option, a Window will pop up. Go ahead and switch to the **layers** tab in the Gerber setup menu. Click all of the layers except **Top Pad Master** and **Bottom Pad Master**. Also **do not click any of the boxes in the Mechanical Layers(s) to add to all plots menu**. You do not want to add any mechanical layers to your Gerber files. Press **OK** and a CAM File is generated then save it in the **Projects Output folder** within your PCB project folder.

b) NC Drill Files:

Now we will generate the NC Drill Files. Click **Files->Fabrication Outputs->NC Drill Files**. This will cause a window to pop up. Click ok and your Drill files will show up. Save them in your projects output folder within your main project folder.

This collection of files will be used to send to the device manufacturer. Once generated, save them in the in the “Project outputs” within your main PCB project.

4. Checkout:

The group has to successfully demonstrate an understanding of printed circuit board design using Altium Designer.