

Lab 4: PCB Design

In this lab exercise students will learn to design a “**Printed Circuit Board**” commonly known as **PCB**. Students will need to learn how to,

1. Design a practical schematic.
2. Choose suitable components with suitable package footprints.
3. Design a PCB
4. Verify PCB design files.

The deliverables for the lab include,

1. The schematic of the designed PCB in pdf format.
2. The GerberFiles files of the PCB.
3. Project files.
4. PCB report from 4PCB.

Recommended software,

1. OrCAD/Allegro
2. Altium Designer
3. KiCAD

Students may use Mosaic lab or Mosaic anywhere to use OrCAD/Allegro. Free student access for the software is also available for both OrCAD and Altium.

Exercise 1: Designing schematic of an instrumentation amplifier circuit.

In the previous lab 2-2 students were taken through the process of establishing closed loop control for an **H-Bridge inverter** in SIMULINK environment. **Figure 1** illustrates the simulation circuit and closed loop control block.

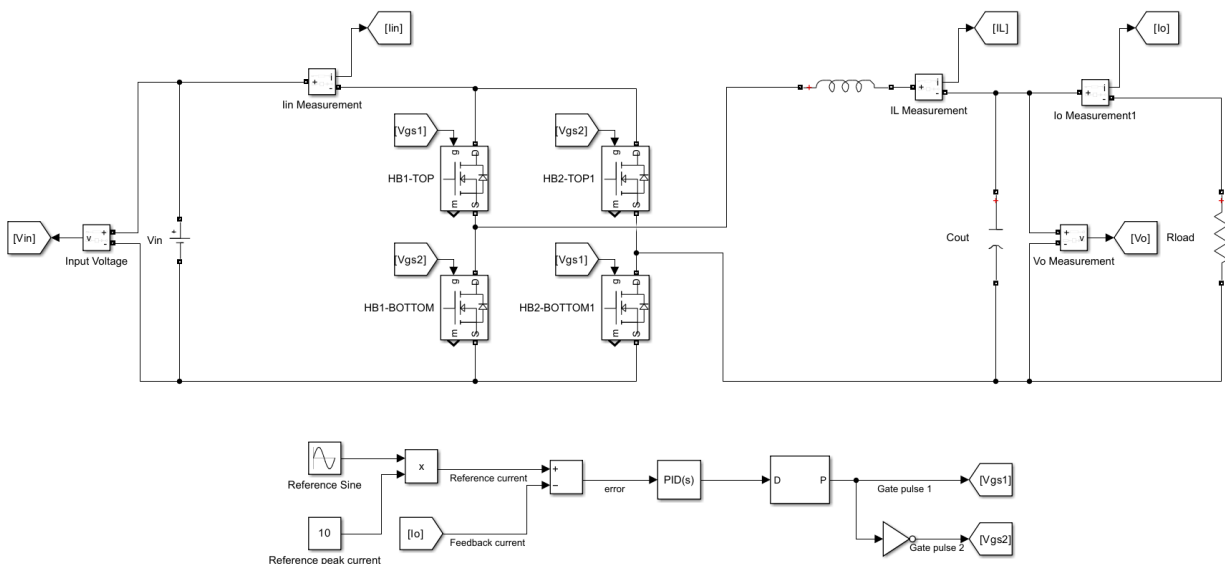


Figure 1: H-bridge simulation

In the simulation we used the measured value of output current designated ‘**Io**’ as feedback. The feedback measurement is compared with a reference to achieve control. The purpose of the control block is to

maintain reference **CURRENT** value at the output. We thus achieve **Current Control** of our **H-Bridge inverter**. Similarly, we can achieve **Voltage Control** by using the output voltage ' V_o ' as feedback. We can understand from the exercise that current and voltage measurements are required to control the output of a converter.

For practical converters we need to design such sensing/ measurement circuits. Instrumentation amplifier circuits are among the circuits that are utilized for such applications. We can use instrumentation amplifier circuit in conjunction with additional off the shelf sensor to design practical sensing units for our own converter circuits. One such example can be seen in **Figure 2**. The complete circuit schematic “**Example schematic 0.3.pdf**” is uploaded with the assignment module. Students are instructed to observe the schematic and understand the application.

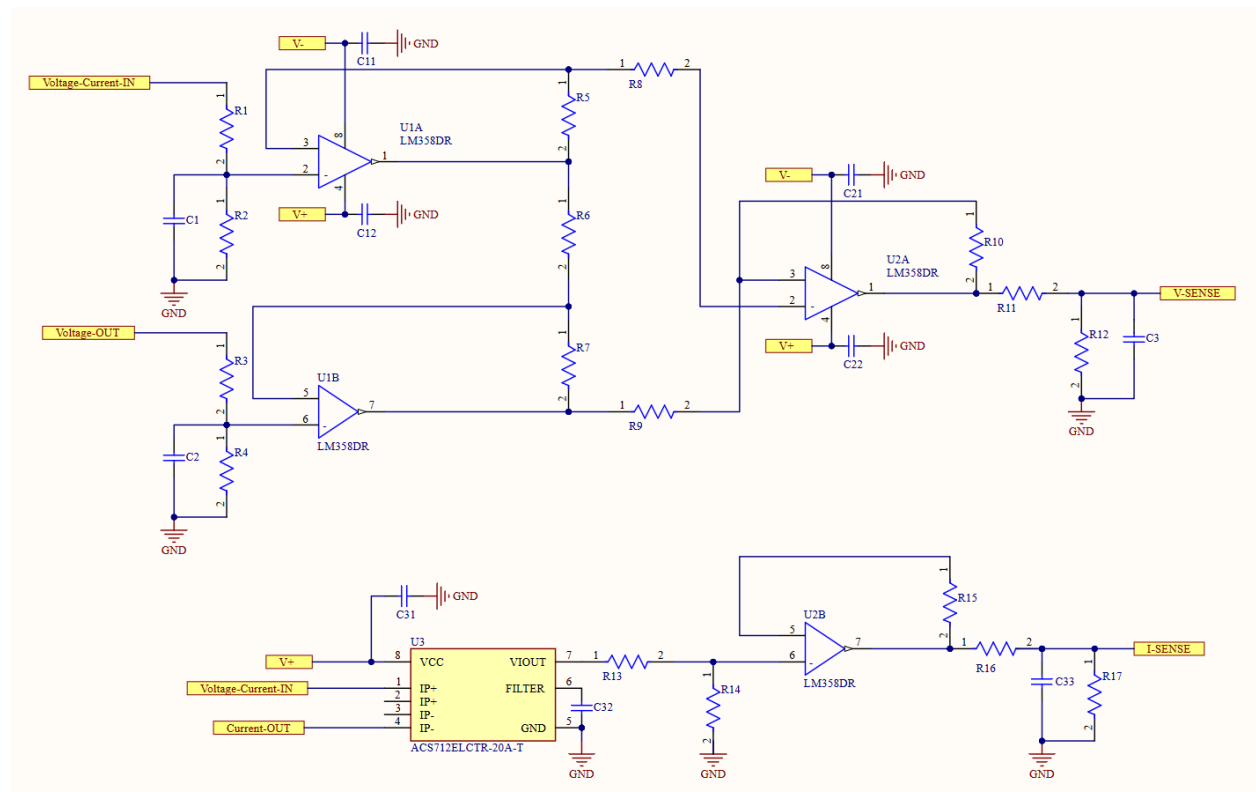


Figure 2: Instrumentation amplifier circuit for voltage and current measurement.

1. Based on the example schematic students are encouraged to design their own current and voltage measurement circuit.
2. Students should design a differential voltage measurement circuit. The top half of the example circuit shows an example of the differential voltage measurement circuit. The op-amp used in this case is an LM358 (8SOIC package).
3. Students must calculate the value of resistors and capacitors for their own application. It is likely that this circuit will be used to interface with a Microcontroller. Most microcontrollers have an input range of 0-3.3V DC. Students must ensure the output of the sensor remains within this range.

- While selecting components for the circuit students should also pay careful consideration to the device package/ footprints of the components.

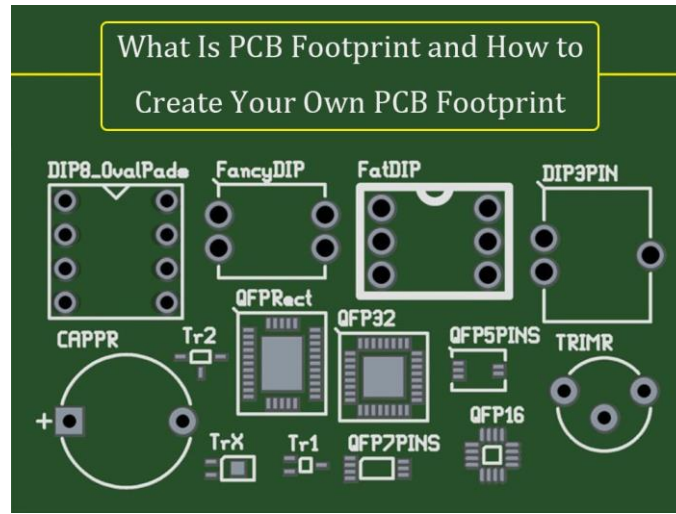
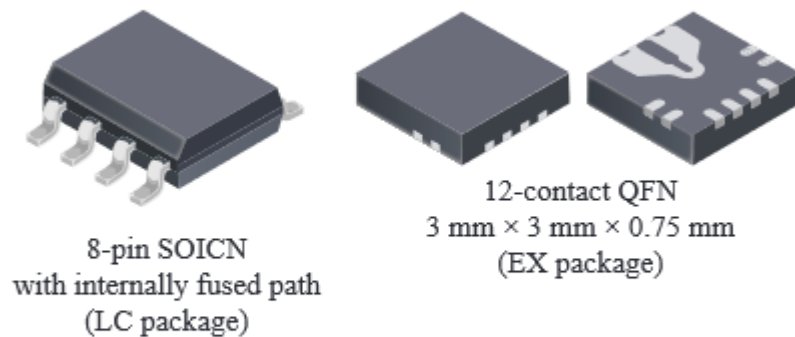


Figure 3: Example of footprints

For our exercise we recommend that students use 0603 package for resistor and capacitors and 8SOIC for the opamp.

- The bottom half of the example schematic displays a current measurement circuit using Allegro ACS712. Students may use current sensors of their own choice.
- Careful consideration should be given to selection of PCB footprint for the current sensing circuit as well.

PACKAGES:



Not to scale

Figure 4: Available package for the ACS711 current sensor

For example, in **Figure 4**, we can see the available footprints for the Allegro current sensor. While the students can use the 12- contact QFN package it is difficult to work with when compared to 8-pin SOICN package. 8-pin SOICN package is thus recommended for this application.

- While designing the circuits students are encouraged to look up the application notes and datasheets of the components. For example, **Figure 5** shows a typical application of the current sensor.

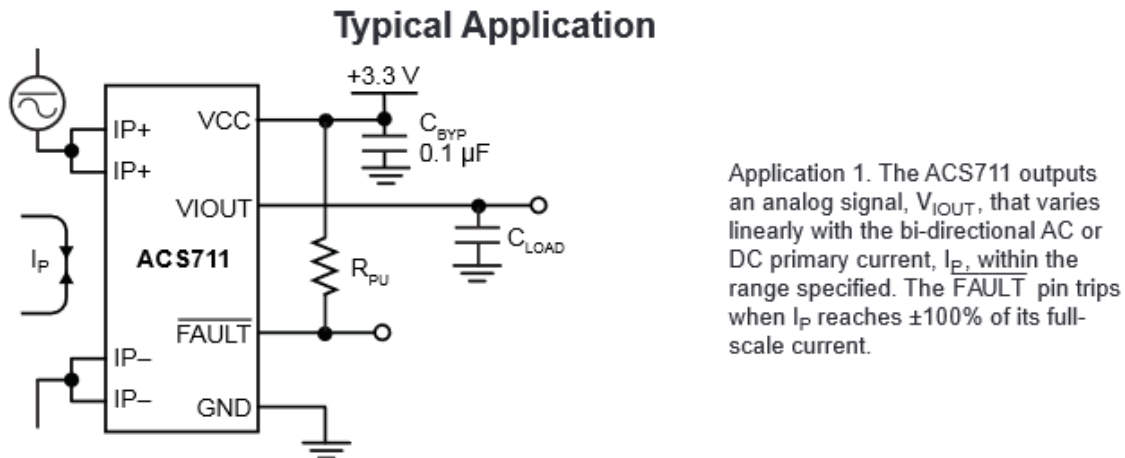


Figure 5: Application example of the current sensor.

- After the circuits have been designed students must add appropriate headers and connectors to the design. The designed PCB will need to interface with a converter power circuit and a microcontroller input. **Figure 1** shows the placement of current and voltage sensors on the converter circuit. Based on that students can observe the “**Example schematic 0.3.pdf**” to see the use of headers and connectors for both power and signal connections. **Figure 6** illustrates the section of the schematic in question.

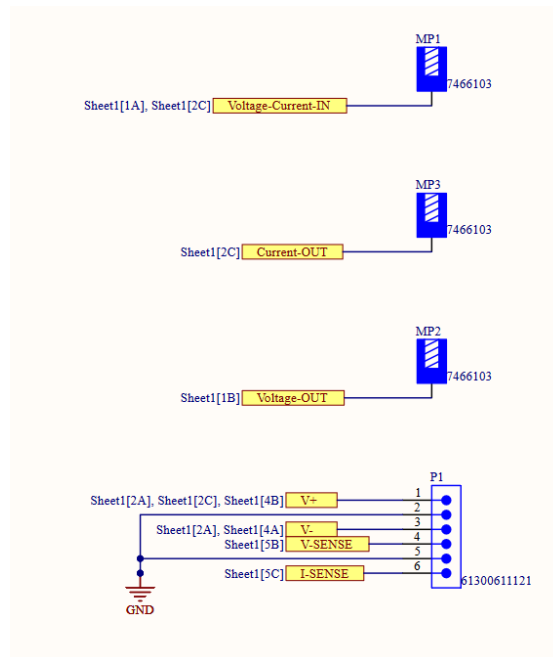


Figure 6: Use of connectors and headers in the circuit for interface.

- After the schematic is completed based on the software students will need to generate the netlist of the circuit. If there are no errors or warnings the students may proceed to the next step.

10. Careful consideration should be given to the warnings. Netlist will be generated even with warnings, but the same netlist may not be imported into the PCB tool.
11. Finally, the viability of the circuit should be manually checked by the students themselves. The software will perform only electrical checks. The software will not ensure the expected function of the circuit. The desired output will be achieved if the correct values and connections are made. This check can only be performed by the user, in this case the students.

Exercise 2: Designing PCB of an instrumentation amplifier circuit

1. Once the schematic is verified and checked for all errors and warnings, the latest generated netlist may be imported to the PCB environment.
2. It is recommended that some environment setups be done before starting work on the actual PCB.
 - a. Clearance; set minimum clearance distances.

	Track	SMD Pad	TH Pad	Via	Copper	Text
Track	20					
SMD Pad	20	20				
TH Pad	20	20	20			
Via	20	20	20	20		
Copper	20	20	20	20	20	
Text	20	20	20	20	20	20
Hole	20	20	20	20	20	20

Figure 7: Clearance setting example, unit in mills

- b. Trace width; minimum, maximum and preferred trace width should be set beforehand. These values can be calculated based on application and board power requirements. To calculate proper and recommended trace widths students may use online tools.

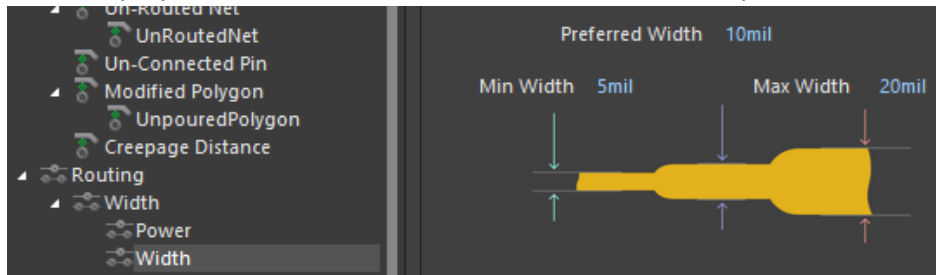


Figure 8: Default PCB trace setting

- c. Board layers: Required number of board layers can be set up at this stage. It is common to setup the PCB with 4 copper layers.

#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.4mil	3.5	
1	Top Layer		Signal	2oz	2.756mil		
	Dielectric 2	PP-006	Prepreg		2.8mil	4.1	0.02
2	Layer 1	CF-004	Signal	1oz	1.378mil		
	Dielectric 1	FR-4	Dielectric		12.6mil	4.8	
3	Layer 2	CF-004	Signal	1oz	1.378mil		
	Dielectric 3	PP-006	Prepreg		2.8mil	4.1	0.02
4	Bottom Layer		Signal	2oz	2.756mil		
	Bottom Solder	Solder Resist	Solder Mask		0.4mil	3.5	
	Bottom Overlay		Overlay				

Figure 9: Default layer setup.

3. Students may start placing components on PCB now.
4. Components may need to be moved around and rotated a couple of times to achieve the most optimum and streamlined connection. It is recommended to go through a few trials to achieve the best possible results.
5. Components can be clustered together based on their interconnections and purpose. For example, capacitor *C11* and *C12* serve as the decoupling capacitor for the Op-Amp *U1*. These capacitors should be placed close to the capacitor voltage rail pins.

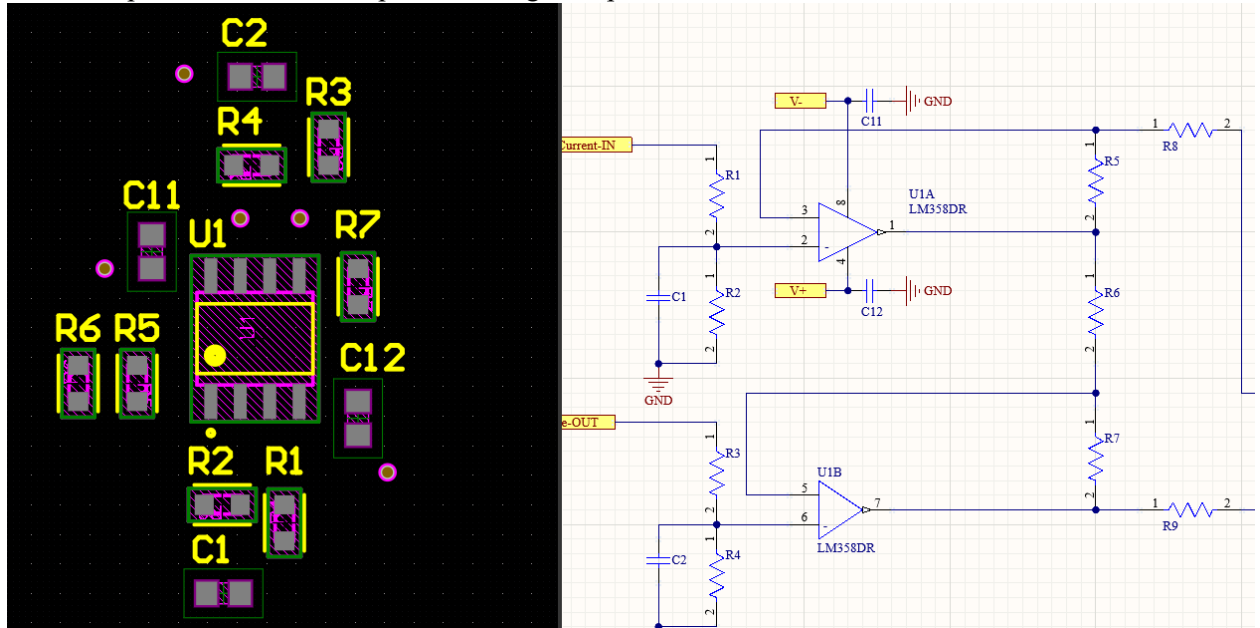


Figure 10: Components schematic and placement

6. Once the components are all placed the traces can be connected. Careful consideration should be given to trace width. Signals traces may have smaller width, but power traces require wider traces. They may even require higher clearance depending on the voltage and current requirements.

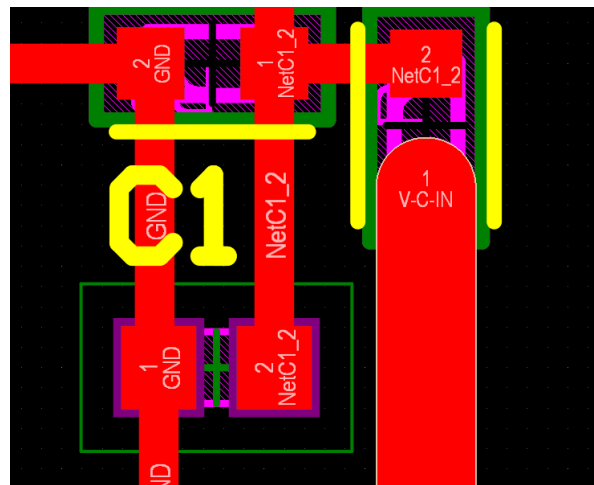


Figure 11: V-C-IN power trace is wider than NetC1_2 which is a signal trace.

7. To handle higher power copper pour or power planed may be used. For component pads being connected to these planes/pours thermal relief should be considered. Otherwise, it may prove difficult to solder these components. Based on the design software thermal relief settings may be adjusted up-to the designers' requirements.

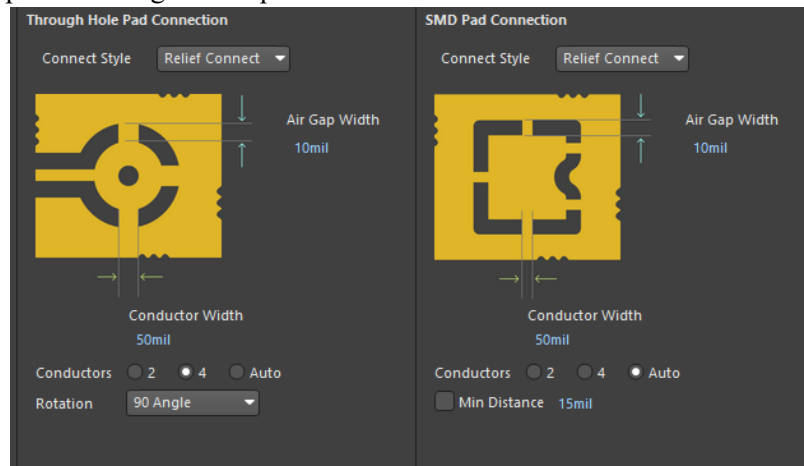


Figure 13 Thermal relief settings

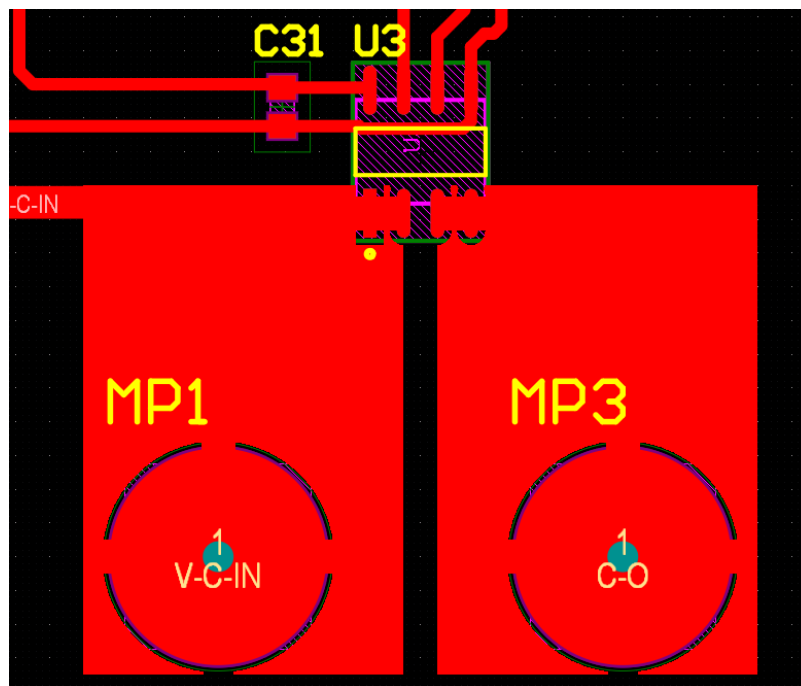


Figure 12 Application of thermal relief and copper plane.

8. Complete all the wiring with these considerations in mind.
9. Run DRC to check for all design rule violations.
10. Address the violations if there are any.
11. Once all the DRC violations are addressed, output **Gerber** and **Drill** files should be generated.

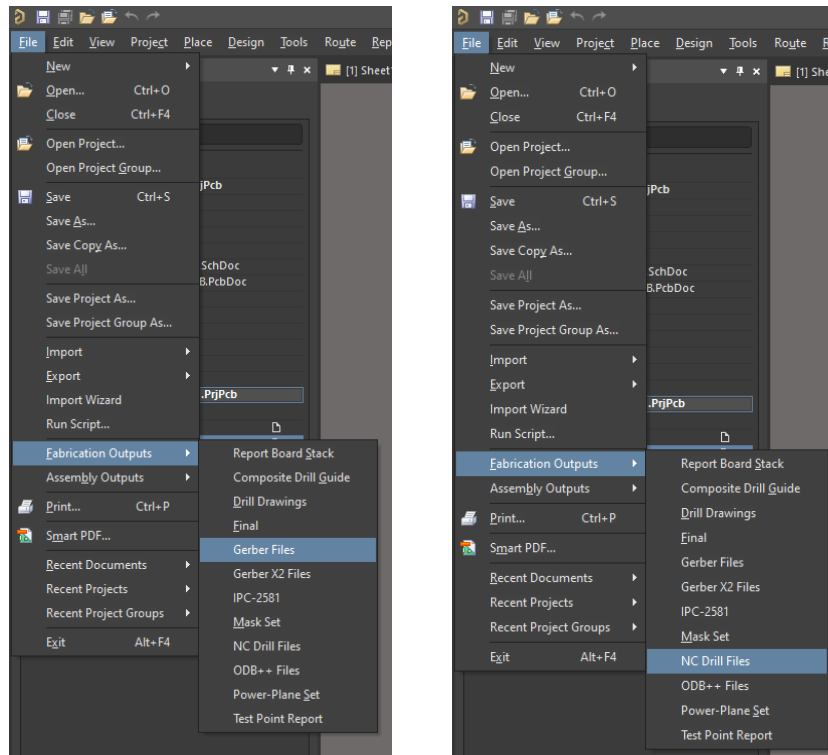


Figure 14 Output menu for 'Altium Designer'

Exercise 3: Upload design files and gather verification reports.

1. Students should now upload their Gerber files for free PCB check on <https://www.4pcb.com/>.
 2. Students may need to open an account to upload files and generate reports.
 3. Based on the report students will have to go back to PCB editor and resolve errors if there are any.
 4. The deliverables for the lab include,
 1. The schematic of the designed PCB in pdf format.
 2. The GerberFiles files of the PCB.
 3. Project files.
 4. PCB report from 4PCB.
- Zip all these in a folder and upload them onto canvas.