EE313 PCB Layout Instructions for DipTrace:

PCB Design

View → Units → mil

View → Grid Size → 25mil

Route → Current Autorouter → Grid Router

Route → Autorouter Setup → Number of Layers: 1

Route → Routing Setup → Trace Width: 20mil, Trace Clearance: 20mil, Outer Diameter: 62mil, Hole Diameter: 32mil, Copper to Board Outline: 40mil

Objects → Place Board Outline → (Draw a rectangle with the board dimensions (for example, 2100mil × 2100mil (approx. 53mm x 53mm) then Enter).

View → Grid Size → 12.5mil

File → Update Layout from Schematic → By Components → Choose the Schematic file.

All components are now on the PCB, but outside the Board Outline.

Placement →Placement by List →Click "Arrange Components outside the Board Outline" icon.

Click on the component name in the small window. Place the objects within the board outline appropriately by dragging them while watching the probable routing for the component. You can refer to your schematic for an appropriate location. You may have to rotate some components (SPACE to rotate). You can also swap the parts in a multipart component (an OPAMP component containing 2 or 4 OPAMP parts). Make sure that the components are not overlapping. A good placement is crucial for a good PCB design. Time spent on a good placement is never wasted.

There are three options for resistor pads: Resistor320 has 320mil separation, and Resistor420 has 420mil separation between the legs. If you need to pass more lines underneath the resistor, use the 420mil separation version. Resistor100 is mounted vertically and does not allow any line to pass between the legs.

Double-click on Bottom in the Layer selector box. Route →Manual Routing →Add Trace

Bottom (2) should be selected as the layer to be routed. (This is the only available layer).

Start adding traces to connect the pads on the bottom layer. When you make a connection, the corresponding ratline will disappear. All connections should be in the bottom layer. If you have problems with routing, you may move the components around to generate a possible routing path.

If you cannot finish the routing manually, you may have to add jumper wires to the top layer: Add Trace, J (to add a jumper on the Top Layer), draw the jumper wire, and click to go to the bottom layer. The jumper wires should be as short as possible and in straight lines parallel to the components. Jumper wires should not overlap or cross the components on the top surface. For tighter regions, you may set the trace width to 15 mil for signal traces. You may have to reposition some of the components for a better layout. If you drag a component, the trace connections will not be lost but may also have to be repositioned.

Try to minimize the jumper count. (It is possible to eliminate all jumpers in most cases by properly placing the components.)

To electrically probe your circuit in the testing phase, add a "Connector" component to that node in the schematic. A jumper wire can be soldered between the two holes so that that node can be probed from the top side.

Once the routing is complete, you may add more copper to nodes that need cooling. First, determine the node's name by placing the cursor over it. Objects →Place Copper Pour→ Click to define the region to be covered with copper→ Clearance: 40mil, Thermals: 4 spoke, Spoke Width: 20mil, Connectivity (select the net corresponding to node), Border Clearance: 20mil

If you would like to modify your design, change the schematic first. Then File →Renew Layout from Schematic →By Components (new components will appear next to the board outline). You may place the new components on the board and wire them. If you need to Update the Copper pour: Bring the cursor to the edge of the board, right-click →Update

Select "View \ Move Component Texts" from the main menu to move component markings without moving the component. Select marking and change its position by dragging with the mouse. To rotate the marking by 90 degrees in this mode, press R or Space (when the Move Component Texts tool is active). Move the component markings on the PCB to be oriented in either left-to-right or bottom-to-top directions. This top silk layer will be helpful when mounting the components on the PCB.

Check connectivity: Verification →Check Net Connectivity (you should get "No errors found")

Verification → Design Rules → Change all values of 6 to 20.

Check design rule errors: Verification →Check Design Rules (you should get "No errors found".)

Correct any errors by editing the positions of components or wires if you notice any errors.

Finally, you need to generate Gerber and NC Drill files for PCB production:

File→ Export →Gerber→ Bottom, Use Design Origin, Inches, Enable G54→Export Layer →Save (YourLastnameBottom.gbr) (You may preview the patterns by pressing Preview).

File→ Export →Gerber→ Board Outline, Use Design Origin, Inches, Enable G54→Export Layer →Save (YourLastnameBoard.gbr)

File→ Export →Gerber→ Top Silk, Use Design Origin, Inches, Enable G54→Export Layer →Save (YourLastnameTopSilk.gbr)

File →Export →NC Drill →Auto, Use Design Origin, Inches → Export →Save (YourLastname.drl)

Make a zip file (YourLastname.zip) containing these four files and upload to Moodle.

Check your zip file using

https://www.pcbway.com/project/OnlineGerberViewer.html

Click the layers button in the pcbway gerber viewer to see all layers. Take a screenshot and place it in your report.

Ensure all layers are on top of each other and the drills are at the correct positions. If there are errors in the PCB (like unconnected/missing components, overlapping components, components beyond

the edges of the board, incorrectly generated Gerber/drill files, incorrect PCB size, unnecessary jumper wires, or missing electrical probe holes), you will lose points. You will have to correct the PCB design before fabrication.

Remember that the time you spend on a good PCB design will save many hours when testing your circuit.