

ELE32CMP/ESP/OPP/BMP Laboratory 1

Introduction to Protel

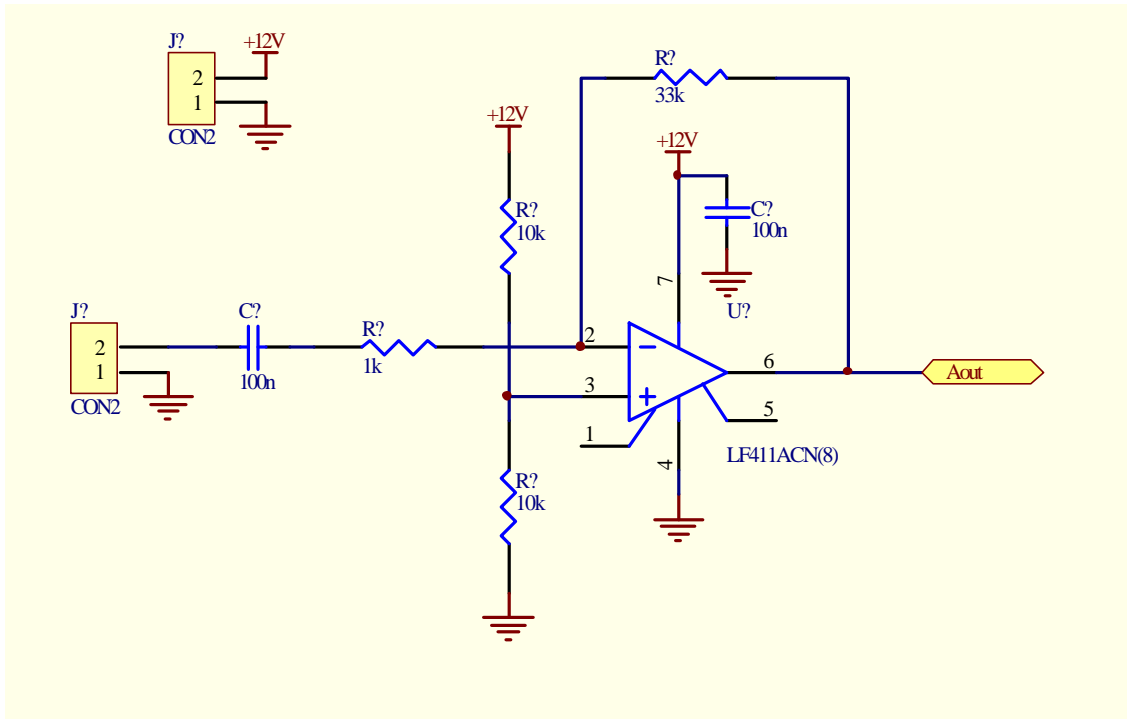
The laboratory is **NOT** required for submission.

Part 1

1. Create a new folder for this lab using Windows Explorer.
2. Start Protel.
3. Create a new project (File||New). Set the Design Storage Type to MS Access Database (this is the default and will result in a single file for your Protel Project). The Database File Name can be anything (eg. ele32prj2003.ddb). Use the Browse button to set the Database location to the folder created in step 1.

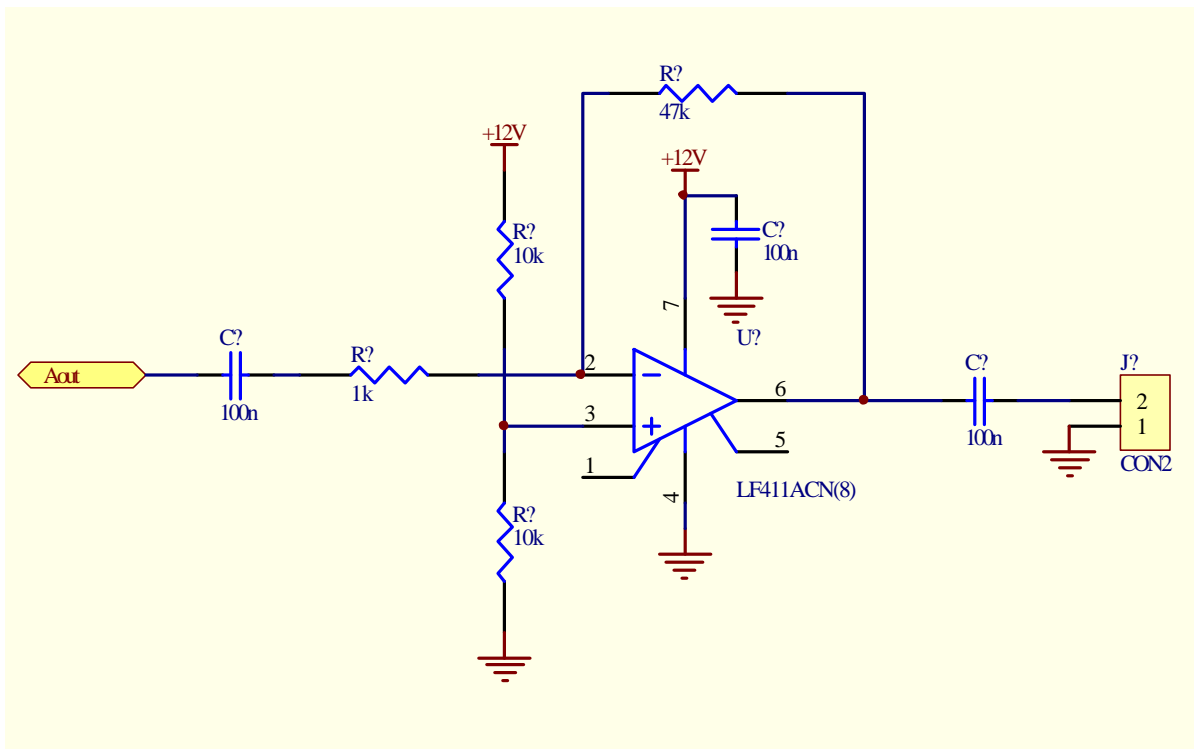
Part 2

1. Create a new schematic page (File||New... and choose Schematic Document). Name this (eg. Amp1.sch). Double click this documents icon to open it.
2. Change the sheet size to A4 (Design||Options... and choose A4).
3. Add the NSC Analog.ddb schematic library (Add/Remove... button on the top left pane). Choose the library click Add then Close.
4. Choose the NSC Operation Amplifier.lib component. A list of op-amps should now be displayed. Choose a LF411ACN(8) (Click on the part then click Place).
5. Drag the component into the middle of the page. Use Page Up to zoom in (Page Down to zoom out). End will refresh the screen and Home will center the page at the tip of the mouse pointer.
6. Add the following components to the schematic. The rest of the components can be found in the Miscellaneous Devices.lib Use the space bar to rotate a component and x and y to flip a component in the x and y direction.
7. Double click on each component and set its reference designator if it requires it (resistors should be R?, op-amps U?, connectors J?)
8. Double click on each component and set its footprint. All resistors and capacitors have a 1206 footprint. The op-amp has a DIP8 footprint (not DIP-8). The connectors have a SIP2 footprint.
9. Place power ports (Place||Power Ports). The default symbol is VCC. Double click each power port to change its label and symbol. Use GND for the ground label (net).
10. Place wires (Place||Wire). Left click to start and change direction. Right click to finish.
11. Add a global port (Place||Port). Double click on it and rename it (Aout).



Part 3

1. Create a new schematic page (File||New... and choose Schematic Document). Name this (eg. Amp2.sch). Double click this documents icon to open it.
2. Change the sheet size to A4 (Design||Options... and choose A4).
3. Add the following components to the schematic.



Part 4

1. Create a new schematic page (File||New... and choose Schematic Document). Name this (egTop.sch). Double click this documents icon to open it.
2. Change the sheet size to A4 (Design||Options... and choose A4).
3. Place two sheet symbols (Place||Sheet Symbol).
4. Double click on each sheet symbol and change the file names to those created above (Amp1.Sch and Amp2.Sch). The name attribute can be anything.
5. Save all (File||Save All).
6. Close and re-open the design. Amp1.Sch and Amp2.Sch should now be below Top.Sch
7. Annotate all the components (Tool||Annotate). Un-check Current Sheet Only.
8. Create a netlist (Design||Create Netlist). Change to Net Identifier Scope to Net Labels and Ports Global.

Part 5

1. Create a new PCB page (File||New... and choose PCB Document). Name this (egTop.pcb). Double click this documents icon to open it.
2. Load the netlist (Design||Load Nets). Click the browse button and find Top.Net. All macros should validate. If not fix the errors (most likely an incorrect or missing footprint). Click Execute.
3. Show all the components (View||Fit Document).
4. Right click on the PCB and change the snap grid (Snap Grid...). Set this to 100mil.
5. Place a Keep Out Layer (Click on KeepOutLayer on the bottom and place a line (Place||Line)). Use a 10mil line (this is the default for Protel). A suggested initial size is 2x2 inch (2000mil x 2000mil).
6. Right click on the PCB and choose Rules... for options such as clearance and track thickness. Click on Routing (first tab at the top). Scroll to the bottom of the associated Rule Classes. This should show Width Constraint. The default is to have the Minimum, Maximum and Preferred widths all set to 10mil. Click on the Properties button (bottom right). Leave the Minimum at 10mil. Increase the Maximum to 100mil and the default to 30mil. This will be the width of all tracks until this property is changed. This may need to be changed several times during the layout of a PCB.
7. Place four pads as physical mounting points. Assume M3 (3mm) screws will be used. Set the pad diameter to 250mil and the hole size to 125mil. Place these pads near the corners of the PCB.
8. Change the snap grid to 50mil.
9. Right Click and component snap grid to 50mil (Right Click||Options||Board Options...)
10. Manually move the components as appropriate. Either click and drag a component or use (Edit||Move||Component and click on either a component or a free area of the PCB in which case a component designator needs to be typed). Think about signal flow and the need to have decoupling capacitors close to the power source they are decoupling. The other components should be moved to minimize the total track length.
11. Change the snap grid and component snap grid to 5mil. This will allow perfect alignment of the surface mount components.
12. Route all but the ground tracks on the top (Click on top layer and use interactive routing (not auto). This is done using Place||Interactive Routing).

There should be no need for vias (Use the legs of leaded components). 30mil tracks should be ok for all routing.

13. Change the snap grid back to 100mil.
14. Place a ground plane on the bottom layer connecting to net GND (Place||Polygon Plane). Check the boxes for Pour Over Same Net and Remove Dead Copper. Use a Grid Size of 20mil and a Track Width of 22mil. Click ok then click on each of the corners of the keep out layer. With the snap grid set to 100mil the first and last corner should line up perfectly.
15. Change layer to top overlay (tab at bottom).
16. Change the snap grid to 20mil.
17. Change the string size of the reference designators (Edit||Change). Set the height to 40mil and width to 7mil. Click Global... then click OK. This will change all the component reference designators. Likewise change the component comments (the components value). Move all the strings so they line up and face in the one direction.
18. Right Click and Rules...||Manufacturing. Edit the second rule (Hole Size Constraint). Change the maximum hole size to 150mil.
19. Check the final design using Tools||Design Rule Check||Run DRC. There should be no errors in routing or clearance.
20. Prepare for printing the final artwork (File||Print/Preview...).
21. Create final artwork (one print per page, not composite in which many layers are printed over top of each other). This is done using Tools||Create Final.
22. Print holes. Right click on the top and bottom layer tabs and choose Properties... Check to Show Holes box.
23. Click on the top layer tab and print current page (File||Print Current). Repeat this for the bottom layer and top silk screen overlay.