

EE313 Schematic Instructions for DipTrace:

A. Atalar

Schematic Entry:

Download diptrace free installer, and install it (PC or Mac versions exist).

Start Diptrace Schematic program.

File → Titles and Sheet Setup → Sheet Template: ISO A4 (Portrait), Bottom-Right → ADT A3,A4, OK

View → Display Titles, Display Sheet

View → Grid (or F11 to turn on Grid Snap)

Zoom In or Out (by mouse wheel)

Right-click on the schematic to pan the schematic in any direction.

Click on “Drawn by” box: Enter your name, click Center

Click on “Date” box: Enter the date, Click Title Box: Enter the title of your design. Click Center.

View → Part Markings... → Show: Select RefDes (reference designator, R1, C2, etc) and Value

Library → Add Library to “User Components” → Add from File → BilkentEE.eli

Objects → Place Component → Libraries: User Components → BilkentEE → Resistor320 → Place (this resistor has 320mils separation between its leads. It allows a trace to be routed between its leads.

Resistor100 has the smallest footprint: the resistor must be mounted vertically. It does not allow a trace to be routed between its leads.

Resistor420 has a larger footprint.

After placing the component, Right click on the component Properties → Value and enter the value of the component if applicable. The values you enter will be visible in the schematic.

You can rotate a component by Space bar after selecting it. Right click on the component → Rotate or Flip the component horizontally or vertically.

In default mode, you can drag a component. If F10 is pressed, the annotations of the component can be moved, without moving the component. Press F10 again, to return to the default mode.

Components → User Components → BilkentEE → Cap, Electrolytic, 150mil (for capacitors with diameter of 150mil=3.8mm, pin-to-pin spacing of 5.08mm), Cap, Electrolytic, 300mil (for capacitors with diameter 300mil=7.62mm), Cap, Electrolytic, 400mil (for capacitors with diameter 400mil=10.16mm).

Components → BilkentEE → LM324 (four identical OPAMPs, with a common power supply connection)

Components → BilkentEE → Power Connector (use it to solder power supply wires to your board)

Components → BilkentEE → Terminal (Screw) (use it to connect the thermocouple leads)

Components → BilkentEE → GND or the appropriate Supply Voltage

If you need other components, you can search for components by Objects → Find Component

You can use the component editor of DipTrace to create new components that are not in the libraries.

Wire the components together using “Place Wire” icon. Correct the ragged wiring to make it nice looking. Make sure that all connections are made correctly. A dot indicating the connection should appear when two wires are joined to form a junction. Otherwise, they are not connected. The program is in Edit Wires and Busses mode if the wire positions are edited. To exit this mode and go to the default mode, press ESC.

Right-click on a wire, and in properties, change the name. Click on Connect Nets by Name. Choose Display Name to display the wire name on the schematic. Click on a pin to be connected to an already existing net → Add to net.. → Select Net, and click Connect without Wire. The wires with the same name are automatically connected to each other. This method can be used to reduce the complexity of the schematic. Use wires to connect components to each other if they are close to each other; otherwise, use the net name connection method.

You may place explanatory text in the schematic using Objects → Place Text.

Verification → Electrical Rule Setup → Check all in Rules to Check.

Verification → Electrical Rule Check (ERC) (to find your possible errors). If there are errors, click on the error and click Localize. Correct and Run ERC until there are no errors.

If you need to move all or part of your schematic, select the components to be moved and use arrow keys to move.

Objects → Bill of Materials (Add RefDes, Value, Name, Quantity for columns). Group Rows by → Name and Value. Export to File → BOM Text File or csv file. You can insert the created file in your PDF report.

Print your schematic as a PDF file to include in your report. Make sure that the schematic is clearly readable in the PDF output.

Note that you will be graded partly on the nicety of the schematic: Input at the left, the output at the right, minimum length of wiring between components, no meandering of wires, and minimum number of crossovers of wiring. The values and reference designators of all components should be visible. The BOM table with correct entries should be in the PDF report.

If you need further help, you can refer to Help of DipTrace.