AUTODESK

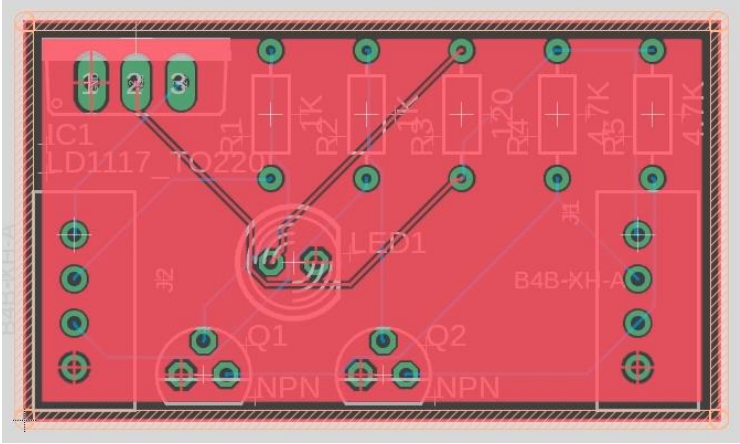
Step-by-step guide

Create an electronics layout

Create a PCB by placing the components and creating the traces.

Learning objectives:

- Create a new PCB.
- Place PCB components.
- Define a ground plane pour.
- Auto-route air wires.
- Review manufacturing constraints.



The completed exercise

---

1. Continue with the PCB module schematic from the previous example.

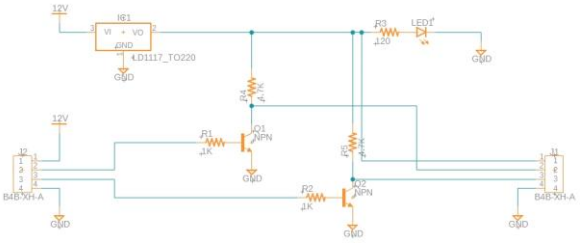


Figure 1. Continue with the file from the previous example

2. Now that the schematic is complete, it can be pushed to a PCB. Click Switch> Switch to PCB document.

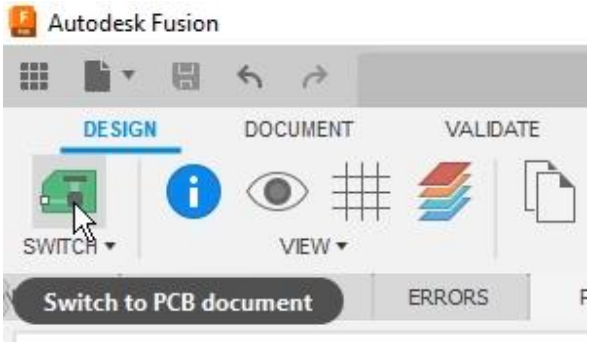


Figure 2. Click Switch to PCB document

3. Notice that all the schematic's components are displayed on the left and a large dark gray PCB area is on the right.

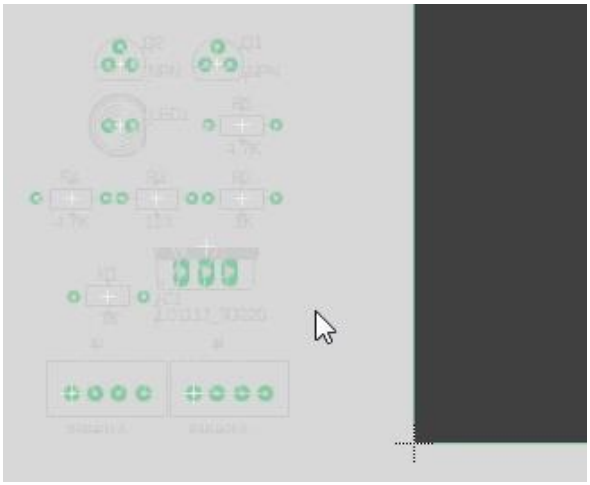


Figure 3. Notice the components and PCB

4. Click and drag the dark gray PCB's edges to redefine its size. You could also bring in a custom 2D PCB shape from Fusion's Design workspace.

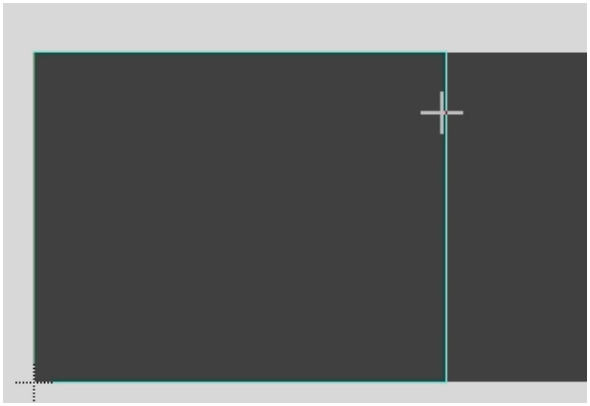


Figure 4. Redefine the PCB's size

5. Expand the Toolbar's menu to explore the PCB's layers.

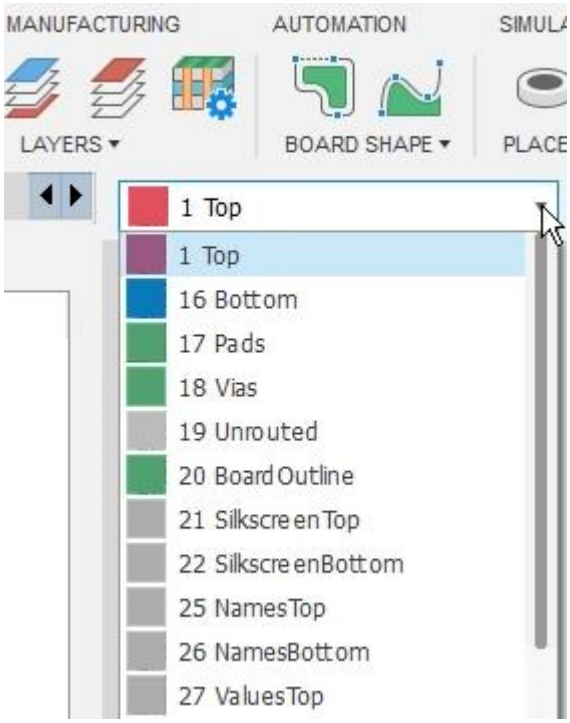


Figure 5. Explore the PCB's layers

6. Click Layers> Layer Stack Manager.

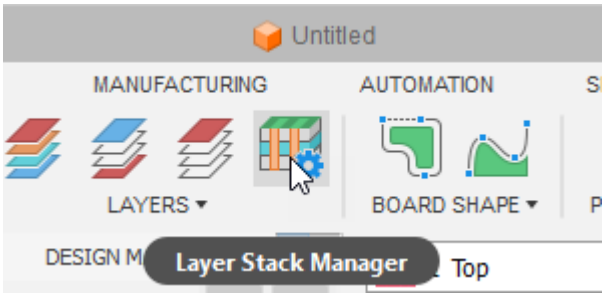


Figure 6. Open the Layer Stack Manager tool

7. Notice that you can use the Layer Stack Manager dialog's drop-down menu to change the PCB's number of stacks. Fusion's Hobby license will only allow you to create a two-layer Stack. Explore the many other ways you can use the dialog's tabs to configure your layer stack. OK the dialog after you finish exploring the options. This documentation will use the default settings.

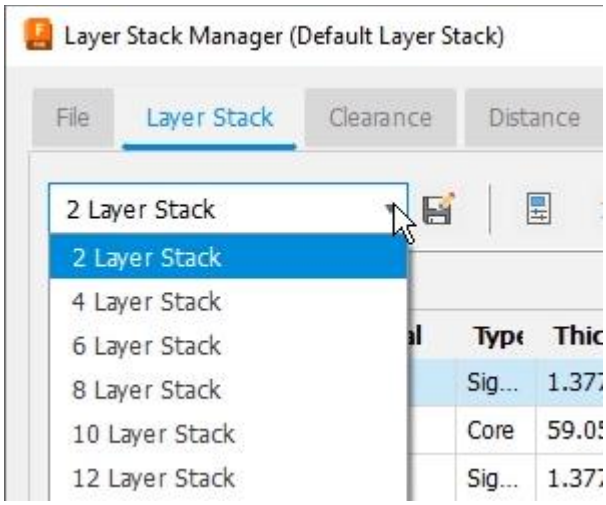


Figure 7. Notice that you can change the number of layers

8. Click and drag one of the connector components onto the PCB. Notice that the component snaps to specific locations. These snaps are configured by Fusion's rules to avoid things like holes being too close to the edge of a board.

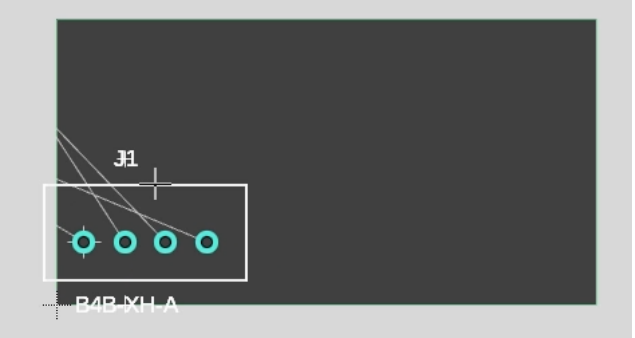


Figure 8. Drag a component onto the board

9. Fusion's rules can be customized if needed. Activate the Toolbar's Rules DRC/ERC tab

Figure 9. Activate the Rules DRC/ERC tab

10. Click DRC> DRC.

Figure 10. Open the DRC tool

11. Notice that the DRC dialog's tabs can be used to configure the same types of information you previously noted in the Layer Stack Manager. Cancel the DRC dialog.

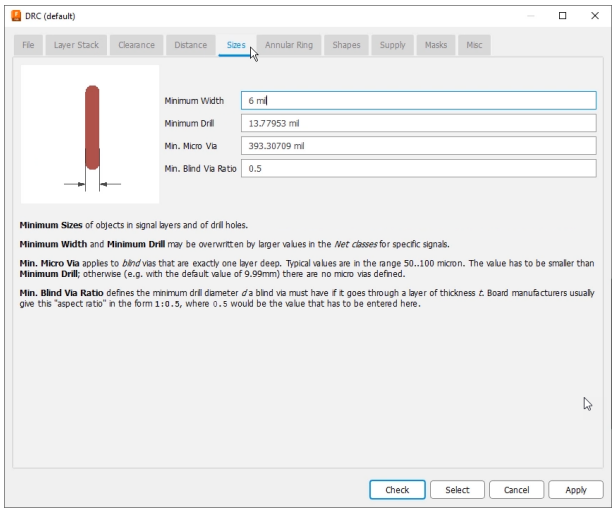


Figure 11. Explore the DRC dialog

12. Expand the Push Violators menu at the bottom of the screen and notice that you can customize what happens when the rules are violated.

Figure 12. Explore the Push Violations options

13. Make sure that the connector's first pin is labeled as N$8 to indicate that it is the output side's 5V pin.

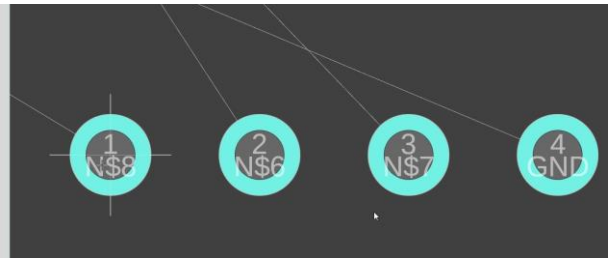


Figure 13. Inspect the connector's labels

---

14. Drag the connector to the right side of the board.

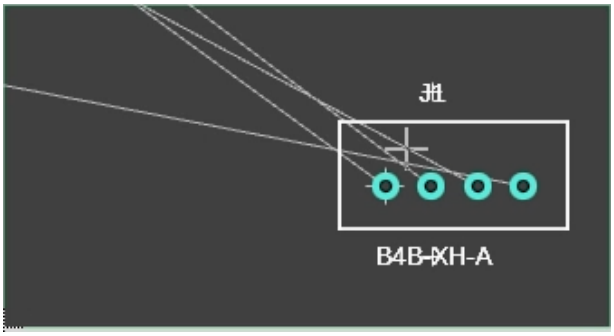


Figure 14. Move the connector

---

15. Activate the Toolbar's Design tab, then click Modify> Rotate.

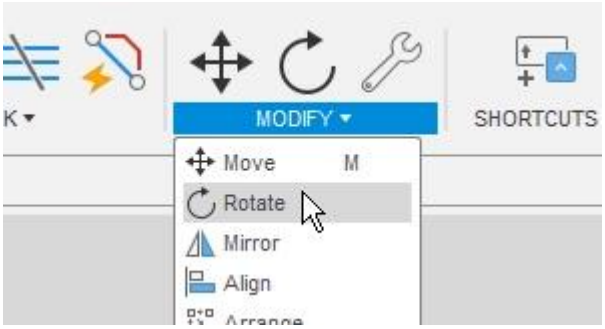


Figure 15. Open the Rotate tool

16. Click the connector to rotate it 90° and make sure the first pin is on top. Click the Rotate dialog's Done.

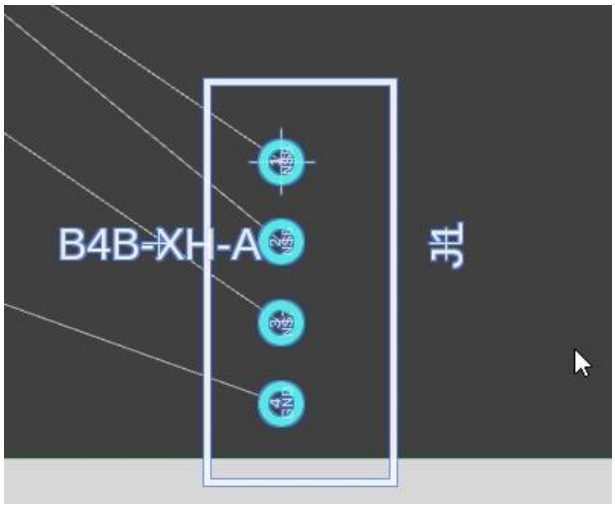


Figure 16. Rotate the connector

17. Reposition the connector to the board's bottom right corner. If any of the connector's notes need to be repositioned, click and drag them to a new location.

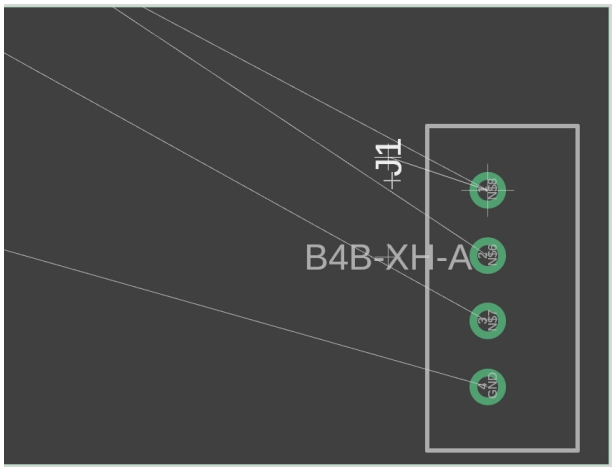


Figure 17. Move the connector

18. Repeat Steps 14 through 18 to place the input connector onto the PCB. Rotate it so the first pin is at the top, then place the connector to the board's left side.

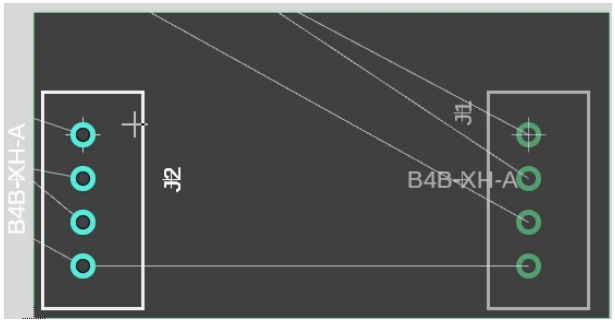


Figure 18. Place the second connector

---

19. Place the voltage regulator onto the board. If the board's perimeter is not large enough to accept the new component, resize the board by clicking and dragging its edges.

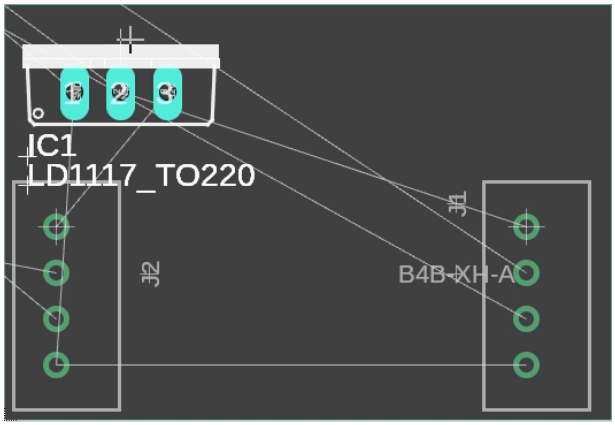


Figure 19. Place the next component onto the board

---

20. Click and drag the two transistors onto the board. The transistors' NPN label will not be silkscreened onto the board, so it does not matter if this label overlaps the connector. The Q1 and Q2 labels will be silkscreened.

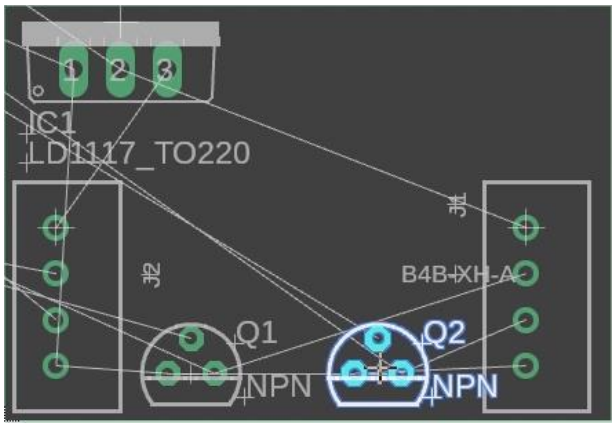


Figure 20. Place the transistors onto the board

21. Before placing the resistors onto the board, open the Rotate tool by clicking Modify> Rotate.

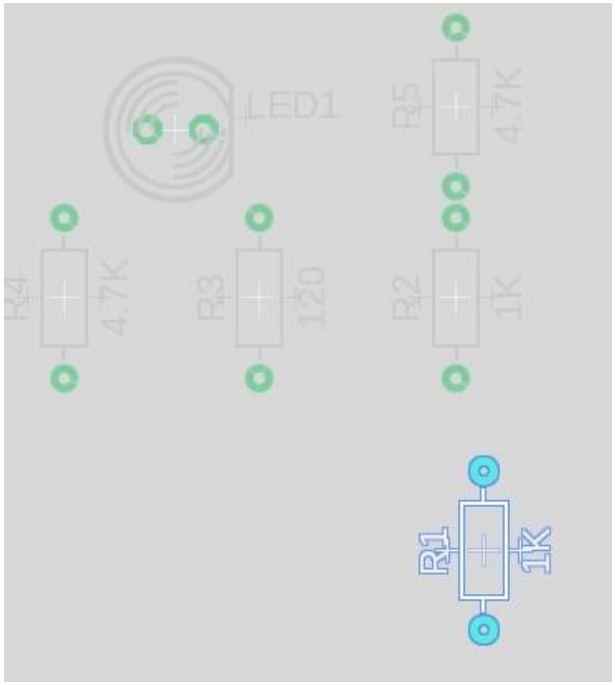


Figure 21. Rotate the resistors

22. Place all five resistors onto the board. Again, click and drag the board's edges to expand the board's perimeter if there isn't enough room to fit all the resistors.

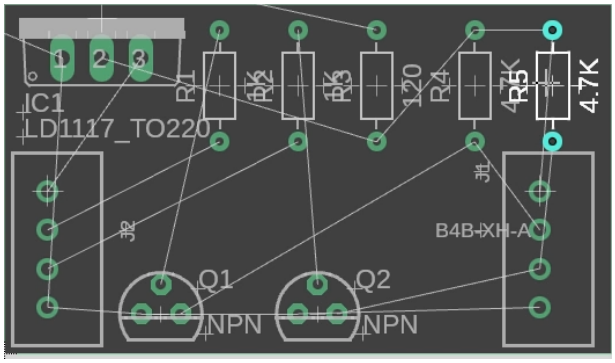


Figure 22. Place the resistors

23. Notice that the PCB's components are snapping to specific locations on the board's grid. You could customize the grid's settings by clicking View> Grid Settings.

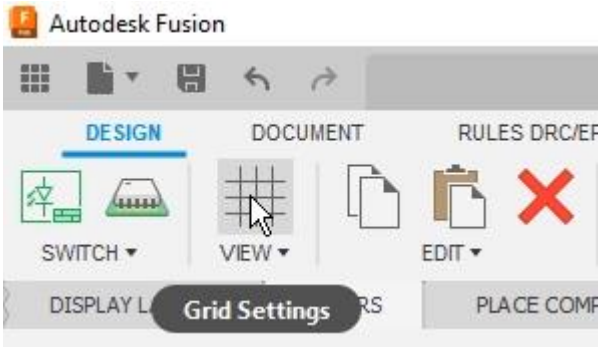


Figure 23. Notice that you could customize the grid settings

---

24. Click and drag the LED component onto the board.

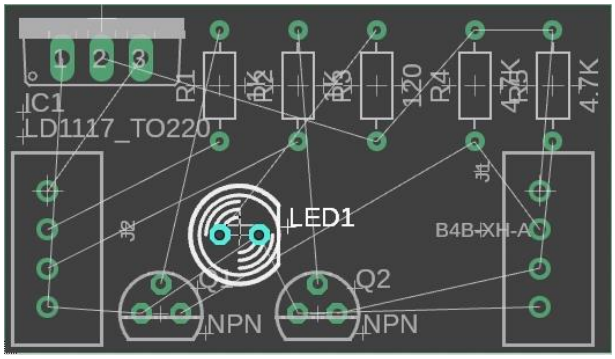


Figure 24. drag the LED onto the board

---

25. The board's component arrangement could be cleaned up; hold Shift and select all five resistors.

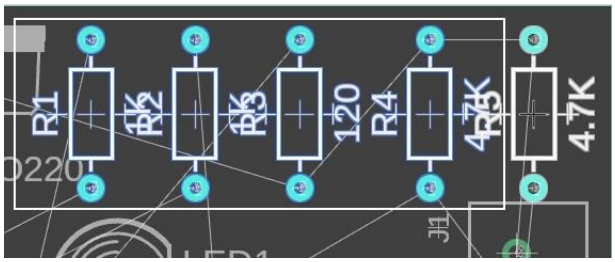


Figure 25. Select all five resistors

26. Click Modify> Align.

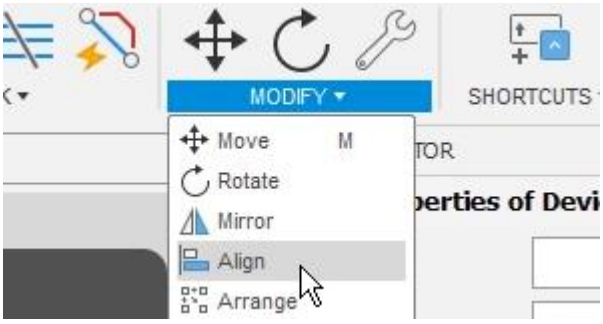


Figure 26. Open the Align tool

27. Explore the Align dialog's options. You can align the components' top, centers, bottom, or sides. You can also evenly distribute the components horizontally or vertically. Click the icon shown in the image on the right to distribute the components centers horizontally. Click the Align dialog's Done.

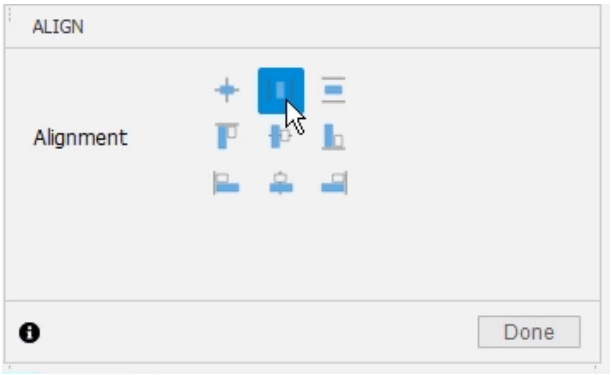


Figure 27. Evenly space the resistors

28. Inspect the result and notice that the resistors are now evenly spaced. The R1 through R5 labels will be silkscreened onto the board but the ohm value labels will not be silkscreened. It is OK if these labels overlap with each other. However, click and drag the labels to a new position if you want to organize the information.

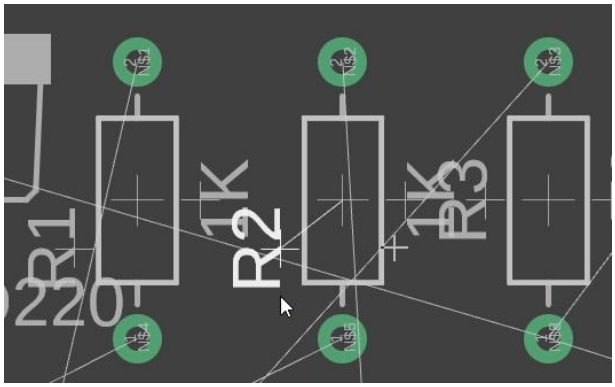


Figure 28. Inspect the result and organize the information

29. Now that all the components are placed onto the board, a polygon pour needs to be created. Click Polygon> Polygon Pour from Outline. This step isn't required, but adding a polygon pour from the outline can be a great way to create a common ground or power plane on your board.

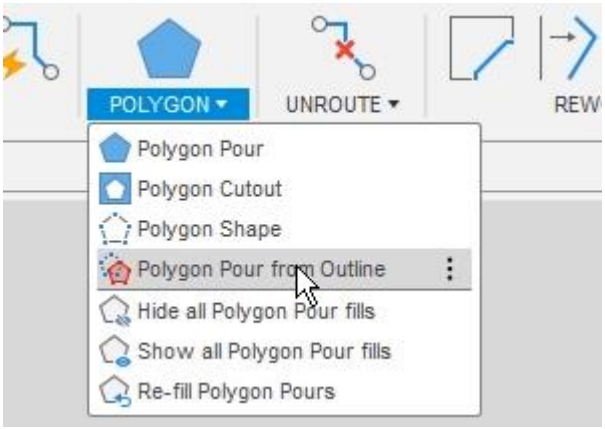


Figure 29. Open the Polygon Pour from Outline tool

30. Click the board's top edge, then choose the Top layer from the Layer dialog. OK the dialog.

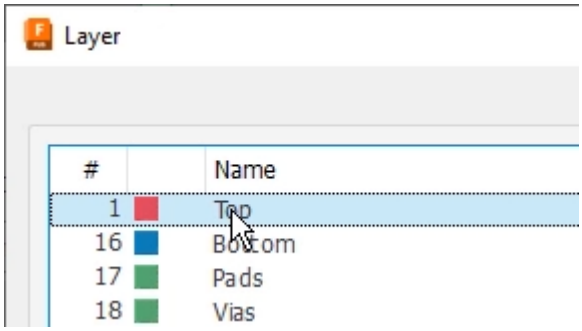


Figure 30. Choose the Top layer

31. Inspect the result. Select the board's far right edge shown in the image on the right.

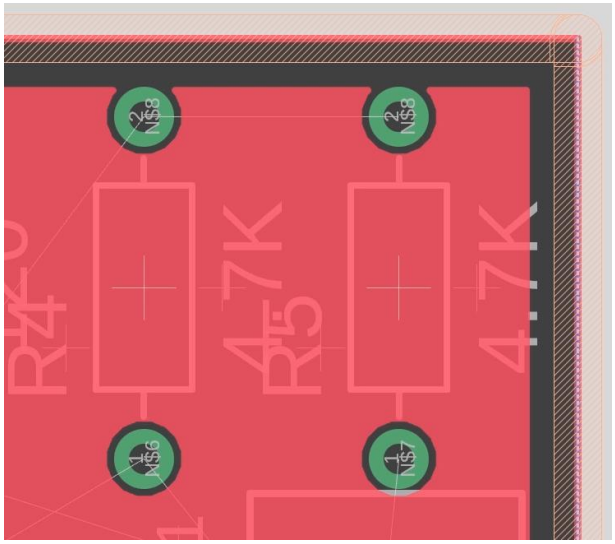


Figure 31. Select the board's edge

32. Click the arrow icons to expand the Inspector on the screen's right side. Enter **GND** into the Name box. Notice that a drop-down menu suggests auto fill options while you are typing; choose the GND option.

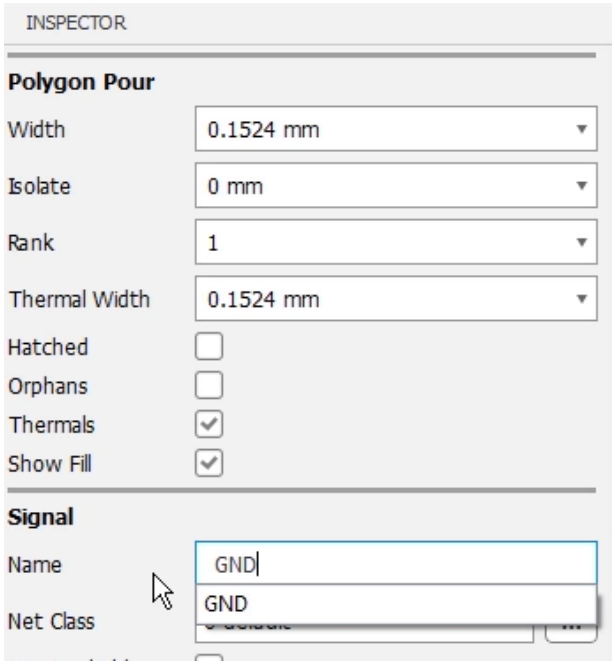


Figure 32. Rename the selection

33. Press Enter to set the value as GND. Click Yes on the dialog to accept the name and assign the polygon to the ground circuit.

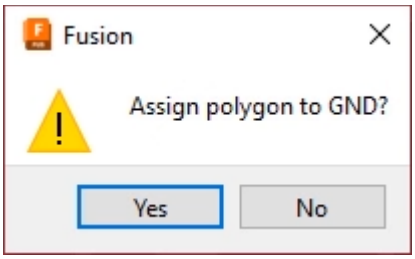


Figure 33. Click Yes

---

34. Now that the polygon pour is assigned as a ground, notice that the voltage regulator's first pin automatically goes to the ground plane instead of requiring a trace to ground it. This simplifies the board's necessary traces.

Figure 34. Notice the grounded pin

---

35. Next, the board's traces need to be created. You could manually route the traces using the options in the Toolbar's Route group. However, Fusion's Autorouter tool could also be used. Click Quick Route> Autorouter.

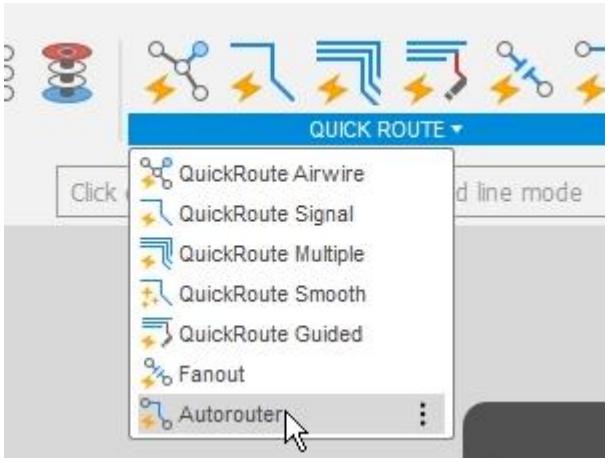


Figure 35. Open the Autorouter tool

36. Explore the Autorouter Main Setup dialog's options, then click Continue.

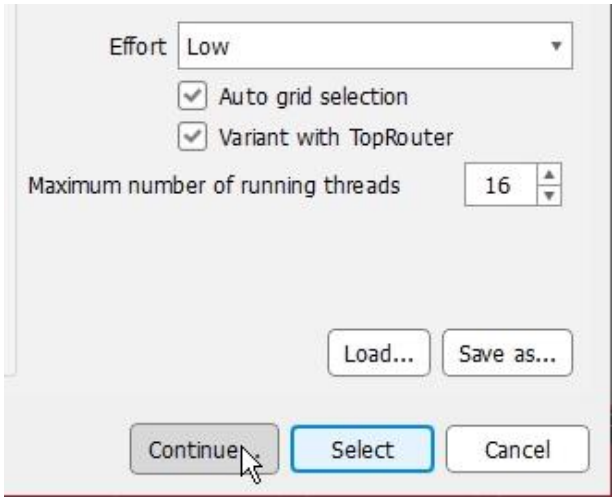


Figure 36. Configure the Autorouter Main Setup dialog

37. Click Start to begin the auto routing.

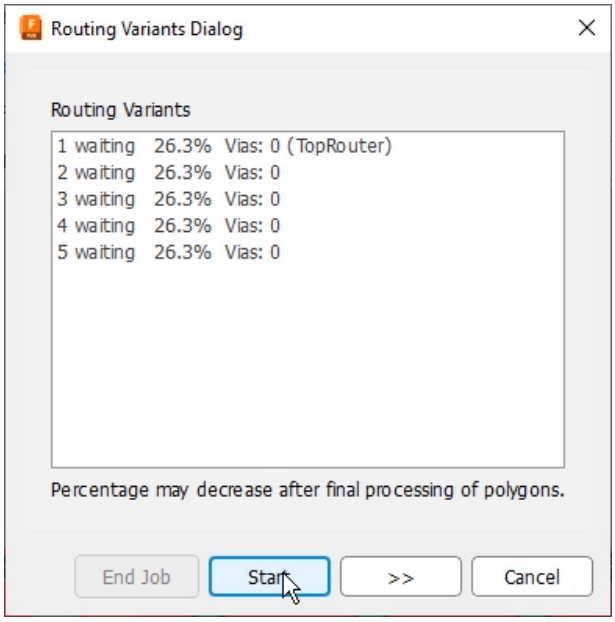


Figure 37. Click Start

38. After Fusion calculates the results, several variants will be displayed in the Routing Variants Dialog. Notice that some variants don't have any vias file, but other variants have one via.

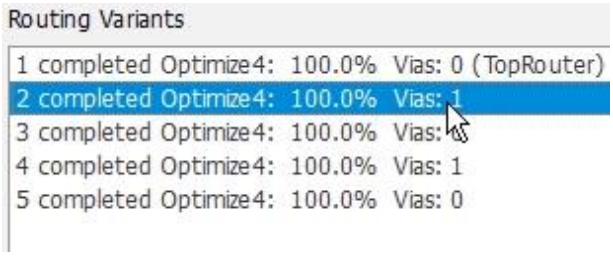


Figure 38. Explore the routing variants

---

39. Select one of the options that does not have any vias, then click the dialog's End Job.

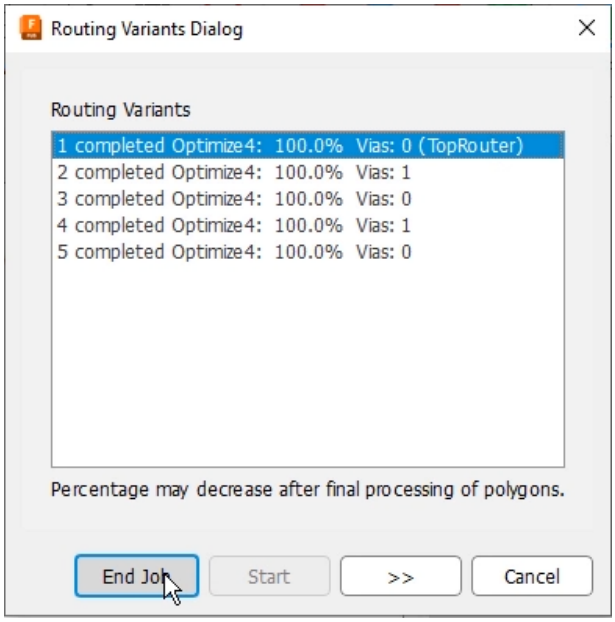


Figure 39. Click End Job

---

40. Explore the result and notice that Fusion removed material from the ground plane to create the traces. The blue lines are traces that are run on the bottom of the PCB.

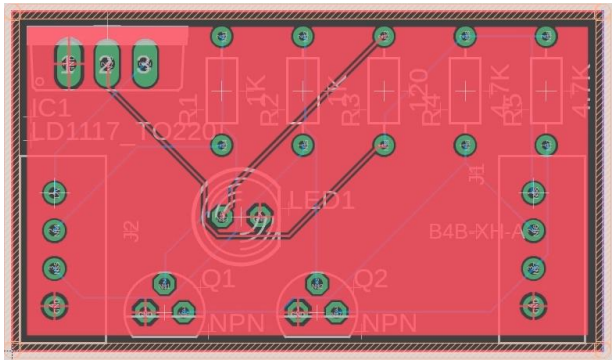


Figure 40. Inspect the result

41. You can view the bottom of the board by selecting it from the Toolbar's drop-down menu.

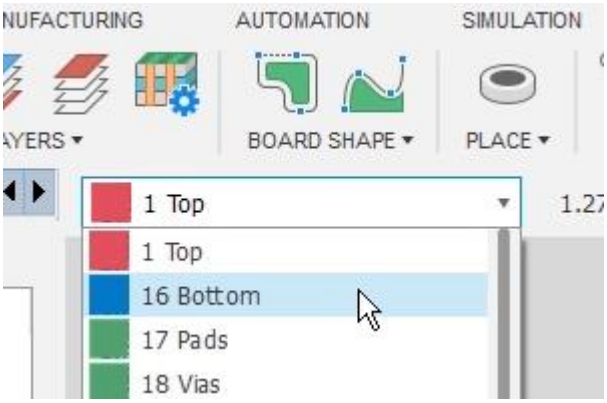


Figure 41. Choose to view the board's bottom layer

42. The board's bottom traces are now more easily visible.

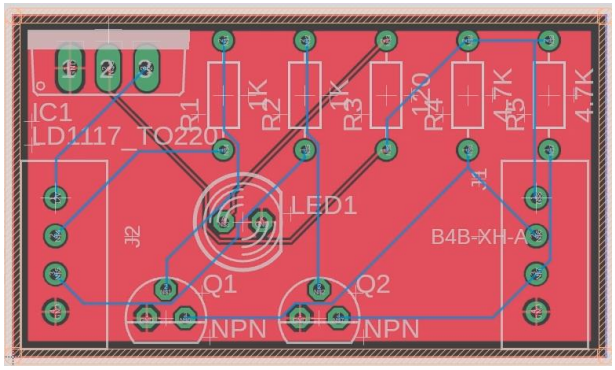


Figure 42. Inspect the bottom layer

43. Inspect the traces and notice that they are labeled. For example, the trace in the image on the right is labeled as 12 Volts.

Figure 43. Inspect the traces

44. Save the file and name it as **PCB Module Board Layout**. Choose the save location, then click the Save dialog's Save. Continue to the next example.

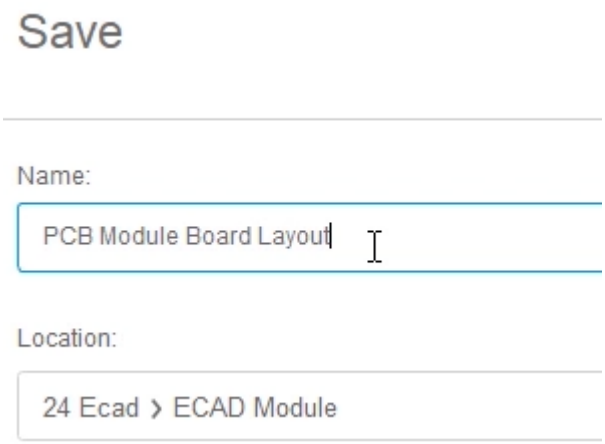


Figure 44. Save the project