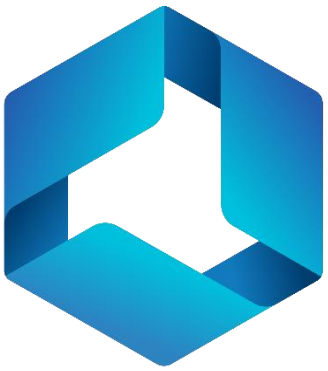

ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 22: PCB Routing

Table of Contents

Module 22: PCB Routing

1.1 Purpose

In this exercise, you will practice routing using various PCB routing modes. Within each mode, routing options and tips will also be discovered.

Routing is essentially the process of making electrical connections through tracks and vias. Altium Designer provides a series of powerful interactive routing tools. The main aspects of Interactive Routing (Clearance, Width, Allowed Layers and Via Style) are driven and defined by your Design Rules.

1.1 Shortcuts

Shortcuts when working with Module 22: PCB Routing

Routing Modes:

U » T or **CTRL+W**:	Interactive Routing
U » I:	Interactive Differential Pair Routing
SHIFT+A:	Active Route
U»M:	Interactive Multi-Routing (Bus-Routing)

During Routing:

F1:	Online Help
~ **(US)** or	
Shift+F1 (other regions):	Shortcuts List
Space:	Elbow Side
Shift+Space:	Corner Style
Shift+R:	Routing mode
Shift+W or **3**:	Track Width
Shift+V or 4:	Via Size
BACKSPACE:	Remove Last Segment
2	Place Via without layer change

Unroute:

U » U » A:	Unroute Board
U » U » N:	Unroute Net

1.2 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 22 PCB Routing`.
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project: `Module 22 PCB Routing (Top\Projects\Altium Designer Essentials Training Course\...)`.
5. Select **Open Project as Copy…** 

.
6. At the new dialog *Create Project Copy*.
   a) Add your name to the project: `Module 22 PCB Routing – [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 22 - [Your Name]`.
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button 

from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`.
      ii) Select **OK.**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.3 Power Planes

During the Layerstack creation, we added two internal plane layers, in this exercise we will assign nets to these layers to enable us to fan out to some of the power nets.

1.3.1 Adding Net Information - Power Planes

8. Make the **GND** layer the active layer from the layer bar. You'll see that this layer is an internal plane layer. If you're not already in *Single Layer Mode*, hit **Shift+S** to easily work on this layer only.
9. Double-click in the board area to open the *Split Plane* dialogue that appears in Figure 1. Assign the net **GND** to the Power Plane and click **OK**.

Check the PCB Filter, set it to All-ON, if you do not see the Split Plane Dialogue.

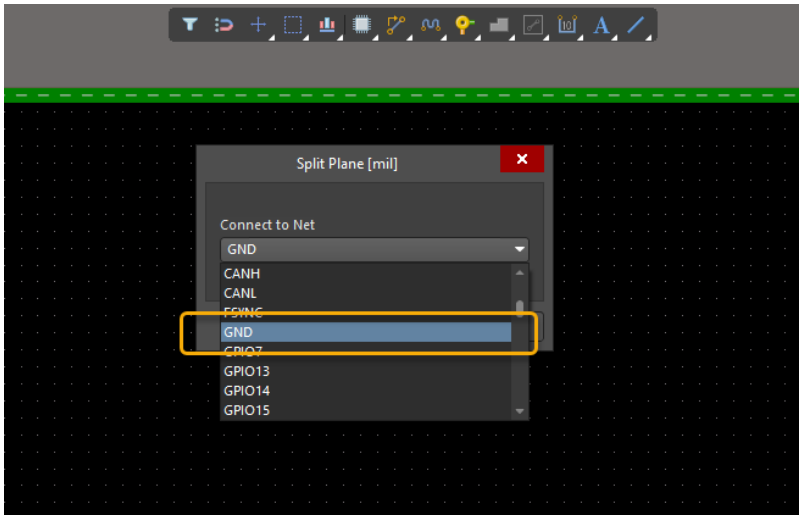

Figure 1. Changing the Power Plane net to GND

10. Now, click on the **5V** layer from the layer bar and repeat the previous step to assign net 5V to the plane as shown in Figure 2.

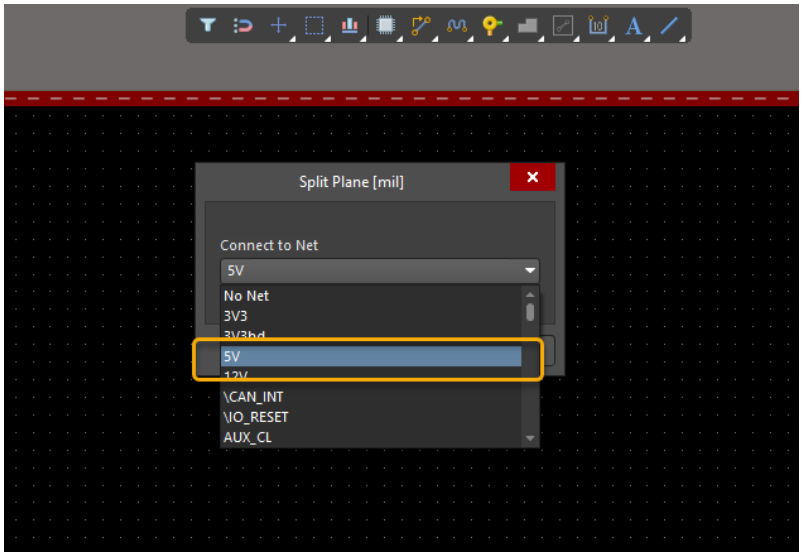

Figure 2. Changing the Power Plane net to 5V

11. Hit **Shift+S** to leave the Single Layer Mode.

1.3.2 **Setting Preferences**

You will notice after opening the PCB document, some of the routing has already been done, this is purely to expedite the exercises. We will further explore different routing options available in Altium Designer to complete some of the remaining connections. First, we will set some of the Routing Preferences.

12. Before routing, it's good to be aware of some of the preferences which can affect its behavior.
    a) Press **O** » **T** to open the *Preferences* dialog to the *PCB Editor* branch on the *Interactive Routing* page, Figure 3.
    b) Set the following options as follows:
        - *Current Mode* for *Routing Conflict Resolution*: *Walkaround Obstacles*
        - *Automatically Terminate Routing*: **On**
        - *Automatically Remove Loops*: **On**
        - *Track Width Mode*: `Rule Preferred`
        - *Via Size Mode*: `Rule Preferred`

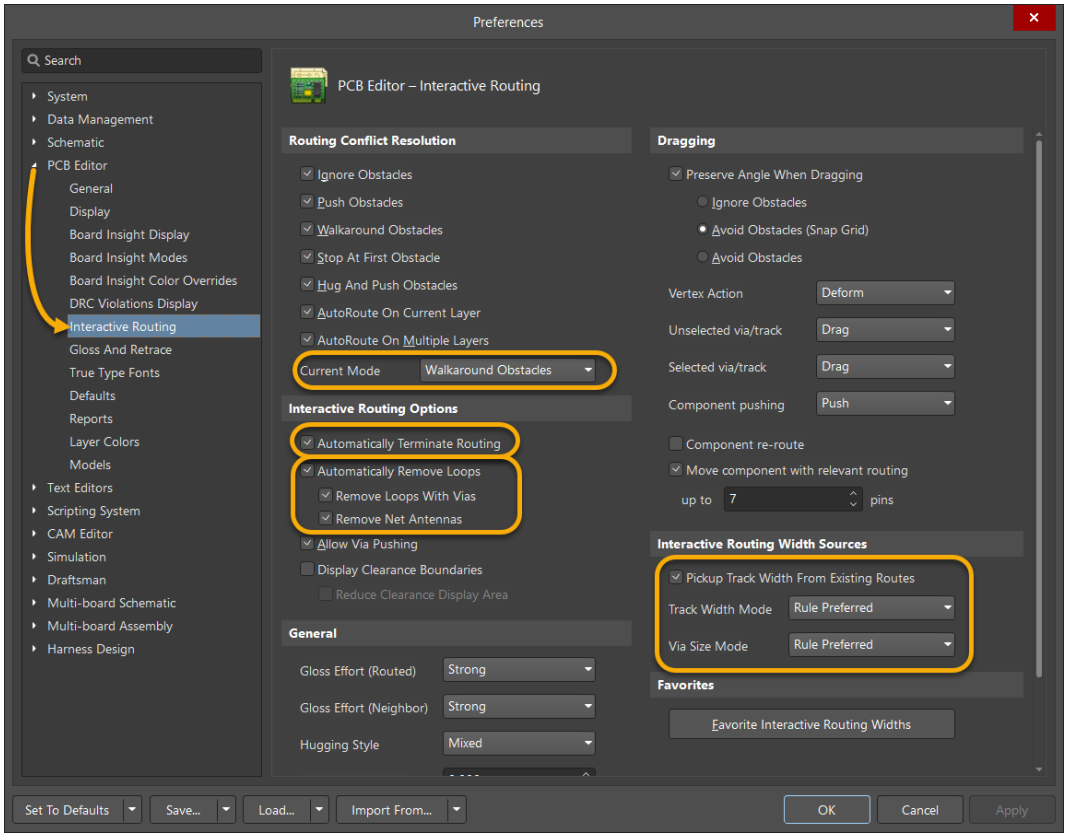

Figure 3. Preferences for Interactive Routing

    c) Click **OK** to apply and close the *Preferences* dialog.
    d) Open the *View Configuration* panel from the **Panels** button, or by hitting the **L** key.
    e) In the *Layers & Colors* tab, we will hide the Mechanical Layers by clicking on the  icon for `Mechanical (M)` .
    f) Feel free to hide to `Overlay Layer` from `Component Layer Pairs (C)` section.

13. Feel free to change the Grid and the Grid Visibility (Dots / Lines), **CTRL+G.**

1.4 Power Nets

In this section we will route some of the Power connections by creating a connection to the Plane Layer.

14. Jump to component `CC2` by hitting **J » C** and type `CC2`.
15. The two CAPs `CC2` and `C3` (zoom out if needed, to see C3 at the right side of CC2) need a connection to the Power Plane GND.
16. Start the Interactive Router, execute **Route » Interactive Routing** menu command or click the Interactive Routing button on the *Active Bar*  or Press **U » T**.
    a) Click on the GND Pad from `CC2`, move your mouse down.
    b) Press **/** (Numeric Keypad- Add fanout Via and Suspend), a via is attached to your moues.
    c) Place the first via with a left click. The current routing is stopped, but routing mode is still active.
    d) Click on the `GND` pad from `C3`, move your mouse to the left.
    e) Press **2** (Add Via No Layer change), a via is attached to your moues.
    f) Place the second via with a left click.
    g) Move your mouse, you will see a new track segment on the same layer starting from the Via (→ Add Via - No Layer change).
    h) Press **ESC** twice to stop routing and to exit the routing mode.
17. For the training example the power nets are not routed on Top / Bottom layer, instead we create connections to Plane layers, as done in the last steps, or the connection is done by Polygon Pours that are placed later in the PCB flow.
18. Jump to component `REG2` by hitting **J » C** and type `REG2`.
19. Start the command **View » Connection » Hide Net** or press **N » H » N.**
    a) Click on `Pad1: 5V` to hide the unrouted 5V Net
    (If you can not select the Pad, change the settings for the PCB Filter).
    b) Left side of to `Reg2` you see `C7` with 12V.
    c) Click on `Pad2: 12V` to hide the unrouted 12V Net.

1.4.1 **Quick Routing for Simple Nets**

In this section, we'll hold the **Ctrl** key and left-click to quickly complete simple, local routes.

20. Jump to component `UI1` by hitting **J » C** and type `UI1`. Without moving your cursor's location, zoom into `UI1`.
21. To start the Interactive Router, press the **U » T** shortcut keys, select the **Route » Interactive Routing** menu command or click the Interactive Routing button on the *Active Bar* .
22. Hold the **Ctrl** key, then left-click on one of the pads, e.g., `Pad 28 of UI1`. If the route is possible on the selected layer, it will be instantly routed.
23. Continue to hold the **Ctrl** key and try another net, e.g., `UI-27`, `UI1-26`, `UI-25`. Right-click or hit the **Escape** key to exit the interactive routing command.

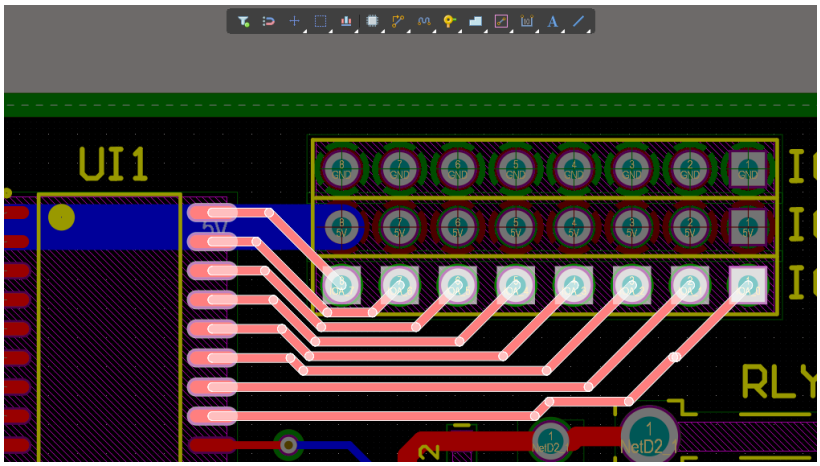

Figure 4. UI1 - Simple Routing with finishing function

1.4.2 **Interactive Routing Nets**

24. Zoom into the area of the board near the upper left portion of the PCB shown in Figure 5 below. We will now route the connections between components `P1, RTERM1, DioCAN1, CH1, CL1, and UC2` through the following procedure:

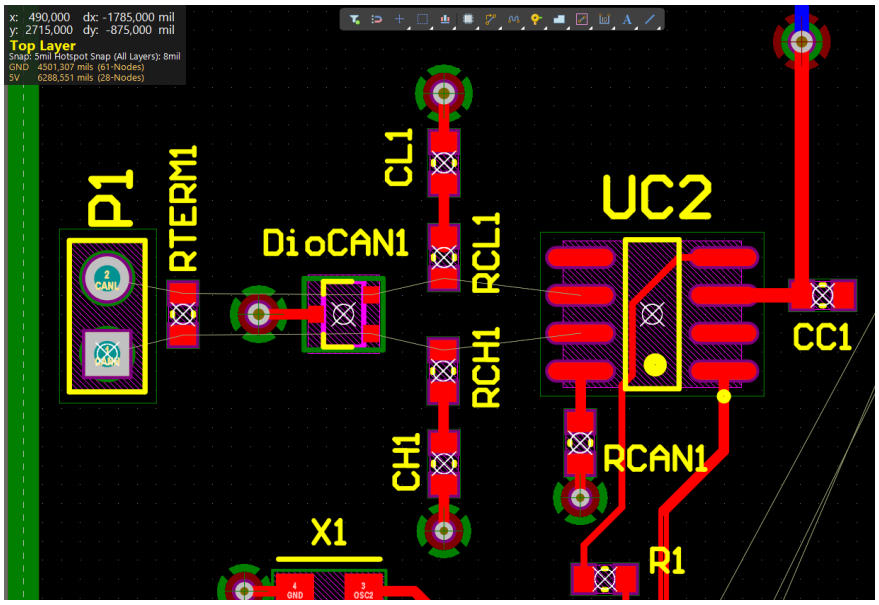

Figure 5. Connection lines (known as a Ratsnest) between un-routed components

a) Start the route command with **Route » Interactive Routing** (or **U » T** hotkeys).

b) The cursor will change to a crosshair, indicating that you are in the routing command.

c) Left-click on `Pad 2` of `P1` to begin the route. Move the mouse away from the pad to see the tracks the Interactive Router is planning to place.

d) Move your mouse to `Pad 1` of `RTERM1` and left-click to terminate the route. Upon terminating the route, we will remain in the interactive routing mode and the editor is ready for the next route, as indicated by the crosshair.

25. While still in the interactive routing mode, left-click on `Pad 1` of `P1`.

a) Hit the **~ tilde** key (US) or **Shift+F1** for other regions, to access all of the available shortcuts and commands while in this mode. Notice that 3 is the hotkey to cycle track-width source of this route as shown in Figure 6.

b) Press **Esc** to exit the shortcuts menu.

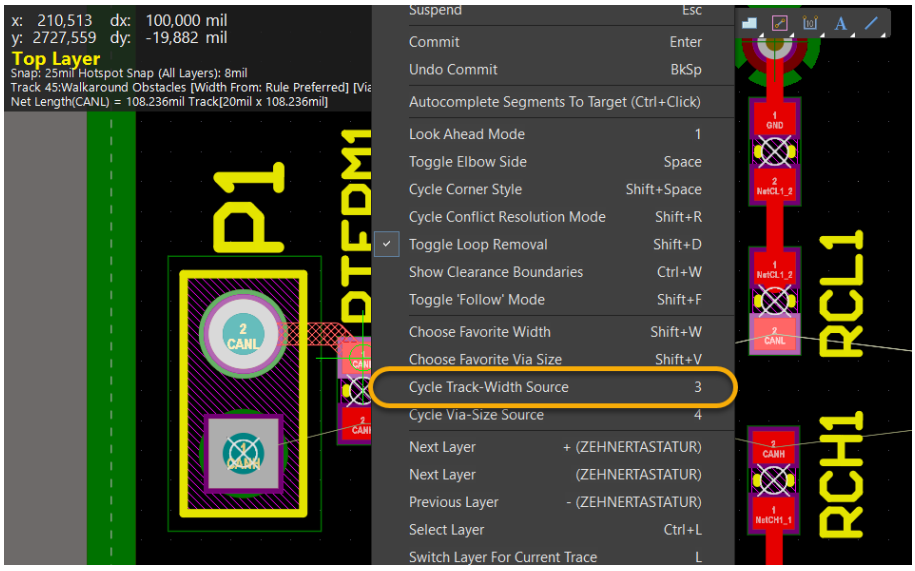

Figure 6. Next available option or shortcut from the Tilde menu

c) Hit the **3** key to cycle through the different trace width values set in the *Trace Width* design rule.

d) You can hit the **Spacebar** key to toggle between different routing angles.

e) Hit **Shift + R** to cycle through the available Conflict Resolution modes.

f) Experiment with a few of the different routing modes, *Walkaround, HugNPush, Push, Ignore Obstacles* and *Stop at First Obstacle*. The current routing resolution mode is displayed in the *Status Bar* and the *Heads-Up Display* as shown in Figure 7.

Figure 7. Conflict Resolution modes displayed in the HUD

You can control which Conflict Resolution Modes are available by changing the preferences in the *PCB Editor* branch under the *Interactive Routing Page* and altering the preferences in the *Routing Conflict Resolution* section or by pressing **Tab** during a route.

26. Hit the **Backspace** key to remove the last placed vertex while still in the routing command if you've placed one.
27. During a route:
    a) Right-click once (or press **Esc**) to exit current route placement.
    b) Right-click again while no route is active (or press **Esc**) to exit the *Interactive Routing* command completely.

1.4.2.1 Adding Vias

28. Start the interactive routing command by pressing **U » T**.
    a) Find component `UC1` and start the route on `Pad 13`.
    b) Start routing and hit the **+ key** on the NumPad or press and hold **Ctrl+Shift** and spin the Mouse Weel to have a Via on your cursor. If you do not have a NumPad, hit the **~ tilde** key (**~ tilde** key (US) or **Shift+F1** for other regions) and select **Next Layer**.
    c) The un-routed connection should go up, if not press **7** (Next Routing Target).
    d) Before placing the via, hit the **4** key to toggle through the three via sizes that you added in the design rule earlier. Left-click to place the via size of your choice as shown in Figure 8.
    e) Finish the route to `Pad 12` of `UI1` using the method mentioned above.
29. Once completed, hit **ESC** or right-click to end the command.
30. Continue your routing with Pad 15,16,17.

To help understand where the connection is going, you hit can the **7** key while routing. This will redirect the connection line to the *Next Routing Target*, which is the closest connection.

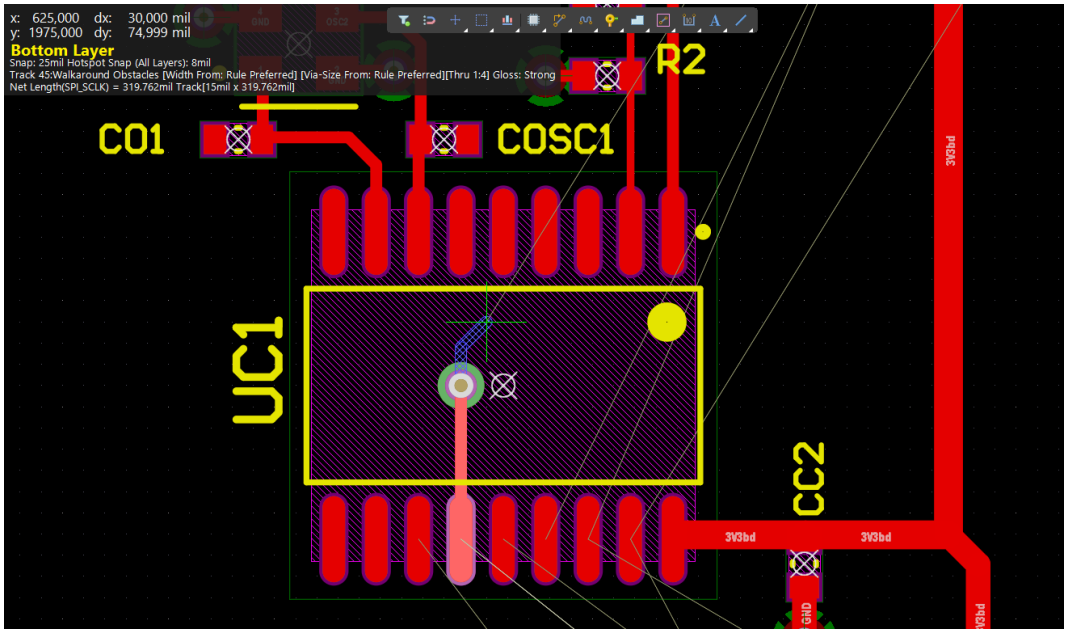

Figure 8. Changing Via Size while routing

It is not necessary to memorize all the interactive routing options. Instead, hit the tilde (~) (US Keyboard layout only) key located below the **Esc** button or the **Shift+F1** key to display all available options. This key works to display all possible command options within the PCB and Schematic Editors.

1.4.3 **Multi-Trace Routing**

To improve the routing efficiency for grouped signals, especially buses, you can use the Interactive Multi-Routing command.

31. Find `UC1` in the PCB document using the command **Jump » Component** (shortcut **J** » **C**).
32. Hold **Shift** and click on `Pads 12, 13, 14, 16` to select multiple pads as shown in Figure 9.

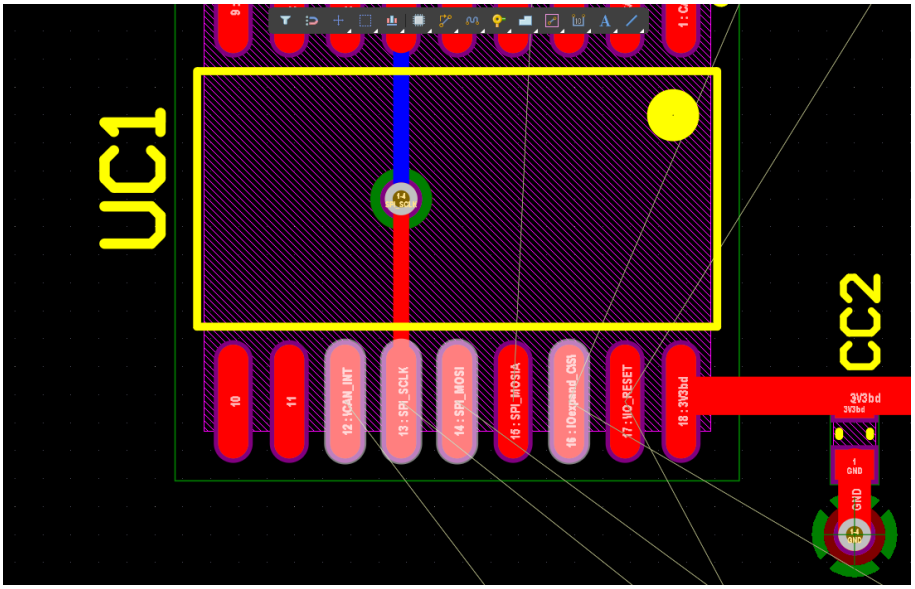

Figure 9. UC1 pads to be used for Multi-Trace routing

33. Start the multi-route from these pads using the command **Route » Interactive Multi-Routing** (**U » M**). The command crosshair will appear to indicate you are in an interactive command.
    a) Left-click either of the four pads to set the base track for the multi-route.
    b) Press **Tab** to open the *Properties* panel.
    c) Change the Bus Spacing to 10mil by pressing **From Rule (10mil)** as shown in Figure 10.

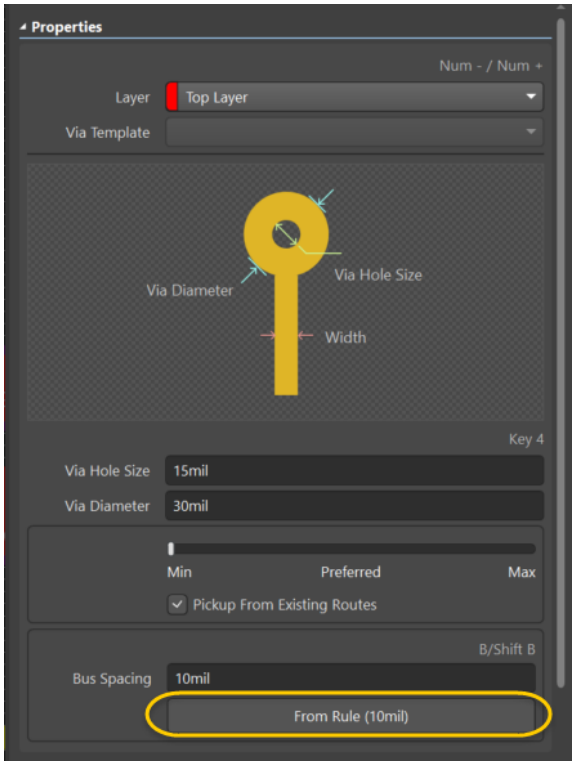

Figure 10. Bus Spacing from Rule

d) Also, whilst in the *Properties* panel, in the *Interactive Routing Options*, change the routing mode to *Walkaround Obstacles* mode. Hit the **Pause** button to resume routing.

e) Move the cursor down towards `J4`, and you will see four tracks are routing simultaneously. If the traces do not appear, press the **3** key to cycle the trace width until it changes to 10mil.

f) Left-click to place the first set of vertical routes. Then, move the cursor and hover on the furthest destination pad, which is `Pin 7` of `J4` as shown in Figure 11. If you're satisfied with the routes, left-click to place them.

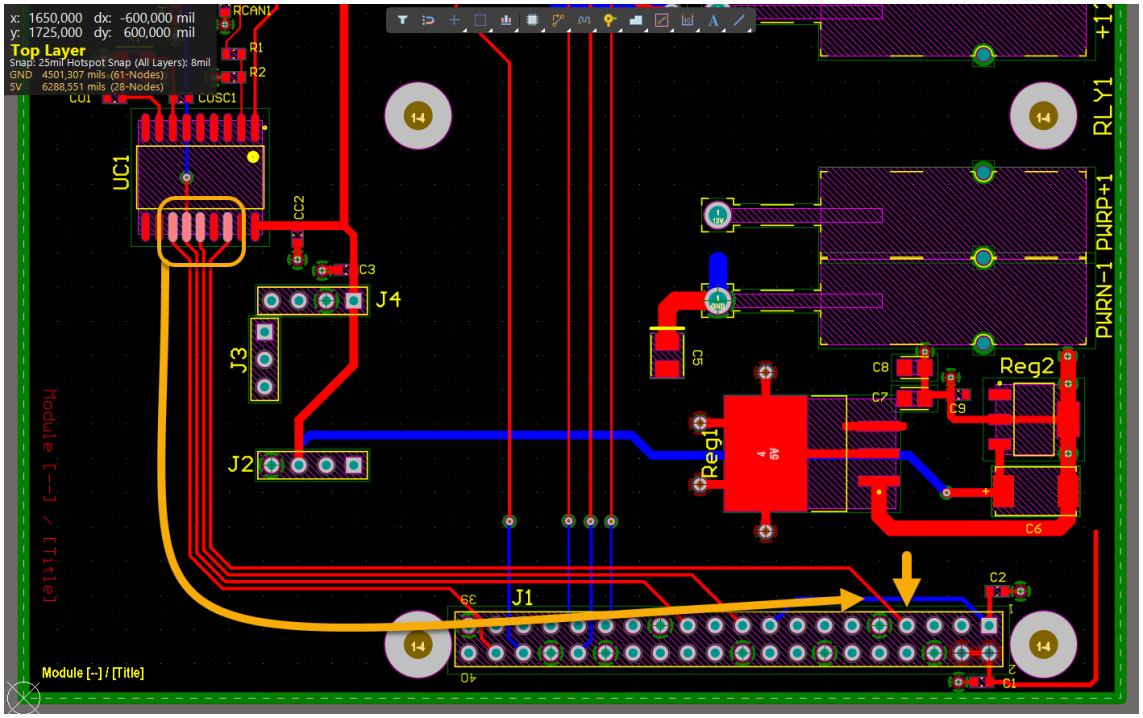

Figure 11. Multi-Trace Routing from UC1 to J4

During the Interactive Multi-Routing process, hit **Tab** and you can set the gap between the tracks. Or simply press shortcut **B** to reduce the gap, or **Shift + B** to increase the gap as you route. If there is not enough room, the multi-router will not complete the route.

1.4.4 **Adjusting Tracks**

34. If you need to adjust a route rather than deleting it, we can route it again. *Automatic Loop Removal* will remove the old redundant tracks automatically. A good connection to try this on is the portion of tracks on the Bottom Layer between `J3` and `C6`, Figure 12.
    a) Start a route from `J2-Pad3,Bottom Layer`.
    b) Re-route alternative trace to `C6-Pad2`.
    c) Once the route is finished, the *Loop Removal* setting will remove the previously routed trace.
    d) Right-click or hit the **ESC** key to exit the routing command.

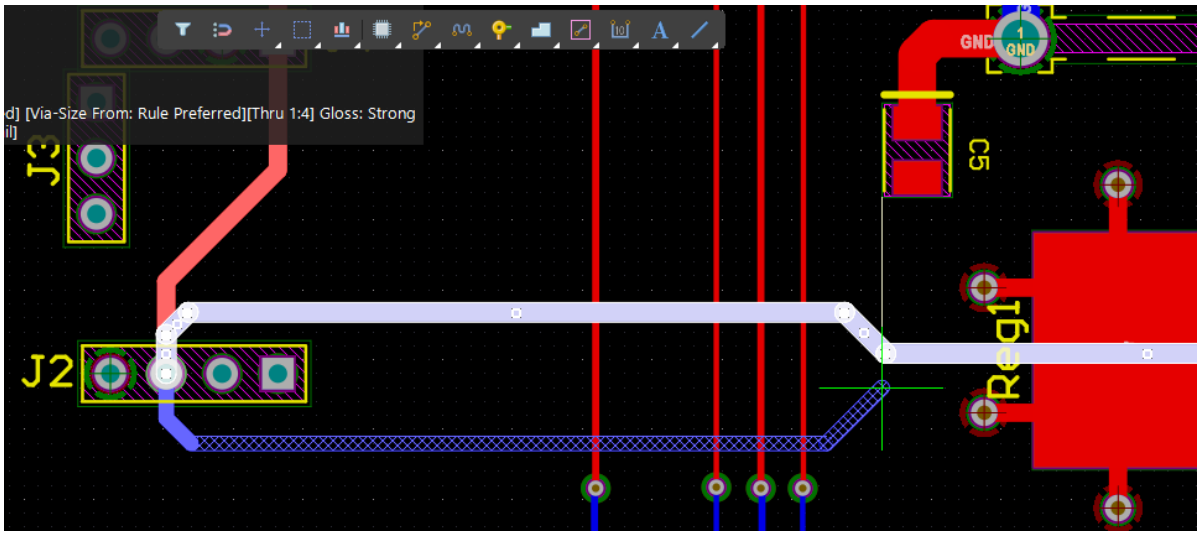

Figure 12. Parallel Routing with remove Loop

35. To adjust a trace, you can left-click and drag it.
36. If you do want to delete an entire trace and start again, you can use the **Route » Un-route » Connection** command or use the **U » U » C** shortcut keys,
    then select the track to remove it.
37. You can also left-click to select a segment of the trace, then press the **TAB** key to select all segments of that track on that layer, followed by the **Delete** key.

When moving or dragging tracks, we can change its behavior in the Preferences, under the *PCB Editor* branch and under the *Interactive Routing Page.*

1.4.5 **Track Glossing**

Using some of the interactive and dynamic routing features may leave some undesired track segments or less effective route paths. We can use *Track Glossing* to clean up route paths after placement.

38. Find the route on the PCB between J2 and J4 that we prepared for some cleanup.
39. To gloss a trace:
    a) Left-click to select a segment of the trace and then press the **TAB** key to select all segments of that track on that layer,
    or **Edit » Select » Physical Connection** or **E » S » C.**
    b) Run the *Track Glossing* command by going to **Route » Gloss Selected** or **CTRL+ALT+G**. See Figure 13 as an example.

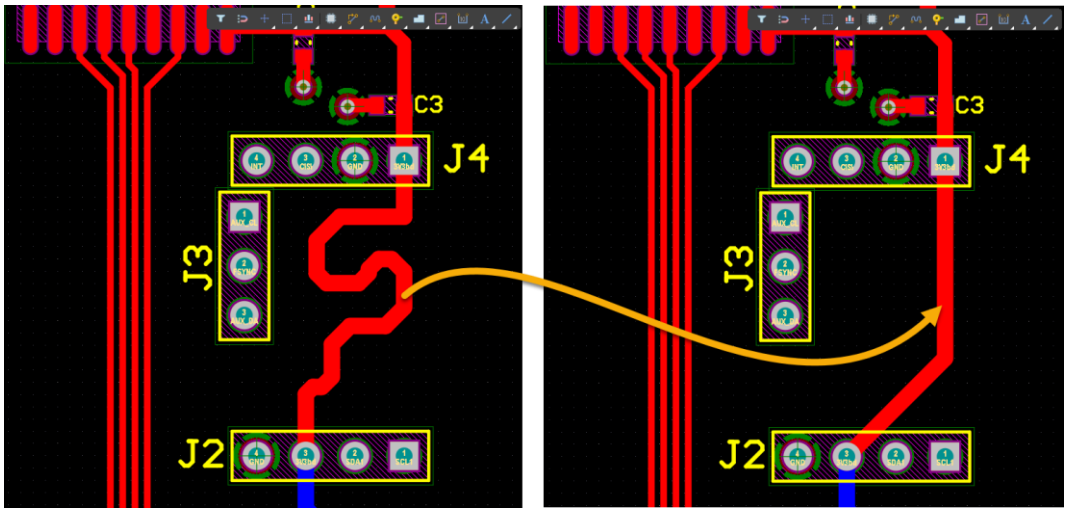

Figure 13. Gloss Tracks, before (left) and after (right)

Routing Tips:
- Use **Ctrl + Click** on a net object to mask everything but the selected net for better visibility of target locations.
- Use HugNPush routing mode for routing close to the bus routes or other traces.
- Press **Shift + C** to clear any applied mask after a route.
- Use the bracket keys to increase **[** and decrease **] (US Keyboard)** the applied mask during routing.
- Holding **Shift + Alt** while selecting connection lines will accumulate the selection.

1.4.6 **Active Route**

ActiveRoute is an automated interactive routing technology that delivers efficient multi-net routing algorithms, which is applied to the specific nets or connections selected by the designer. *ActiveRoute* also allows the designer to interactively define a route path or Guide, which then defines the river along which the new routes will flow.

40. Select the connection lines between `UI1(Pad 1 to 8)` and `IOS2(Pad 1 to 8)` by holding the **Alt** key followed by dragging your mouse from right to left around the connection lines, then let go of your mouse button to select them. This will create a selection rectangle and will highlight the connection lines as shown in Figure 14.

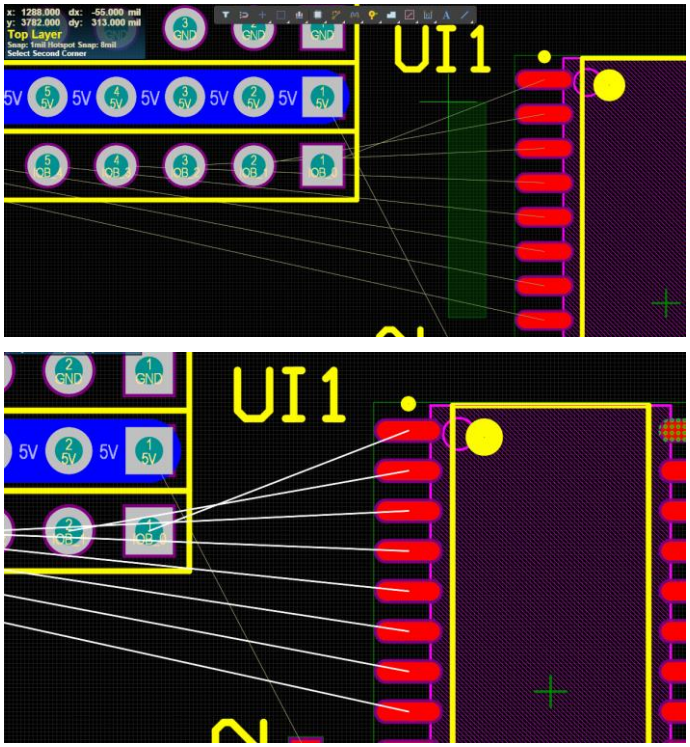

Figure 14. Select the Connection Lines to route the selected traces

41. Open the *PCB ActiveRoute* panel from the **Panels** button.
42. Enable both checkboxes in the *PCB ActiveRoute* panel *Layers* section to allow ActiveRoute to utilize both the Top and Bottom layers when the command is activated, as shown in Figure 15 below. However, in our case since we only have SMD Top-Layer pads on UI1 and ActiveRoute tool does not drop vias, routes will only be placed on the Top Layer.

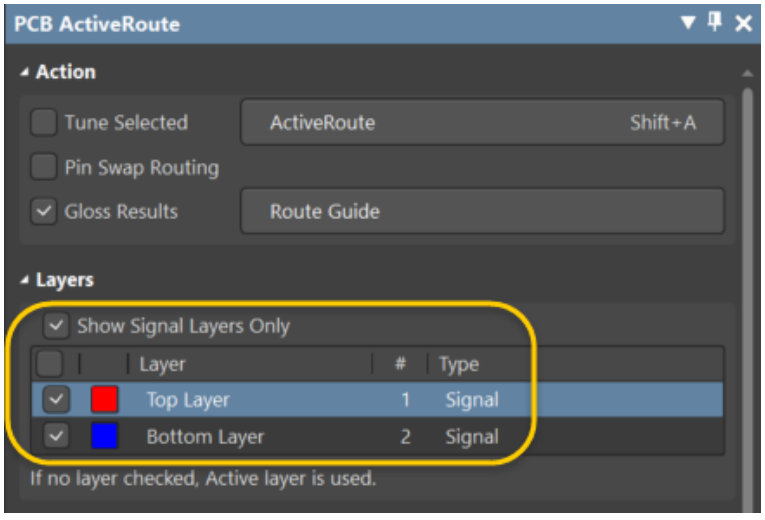

Figure 15. PCB ActiveRoute routing layer setup

43. Click the **ActiveRoute** button in at the top of the panel as shown in Figure 16, or select **Route » ActiveRoute**. The routes will be completed as shown in Figure 17.

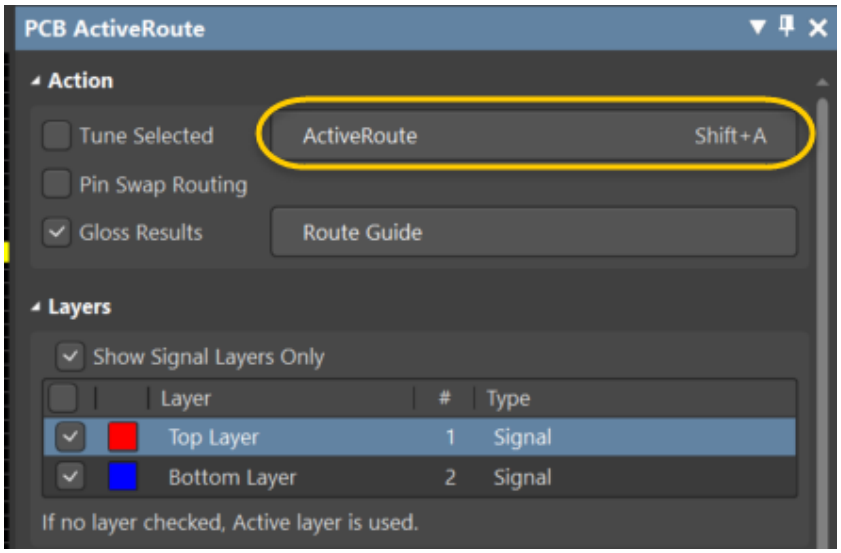

Figure 16. Click the ActiveRoute button to start the routing

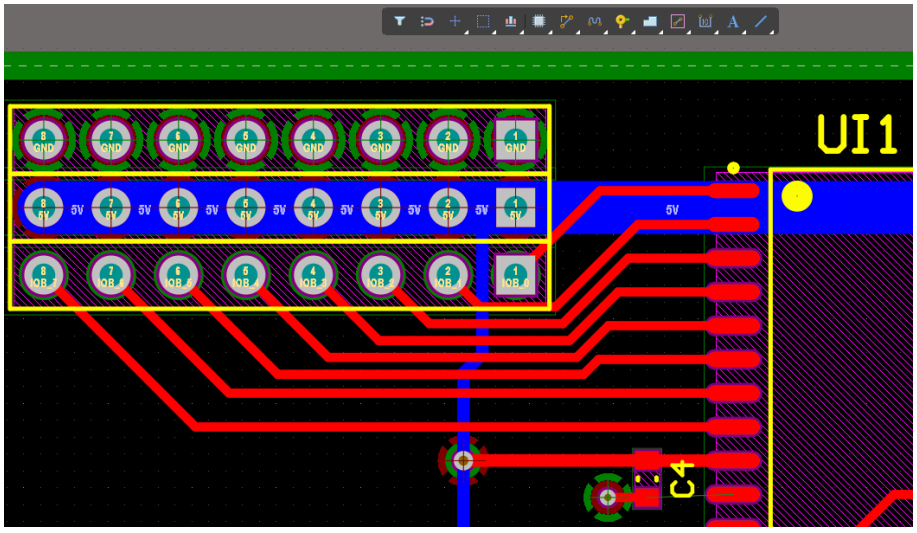

Figure 17. ActiveRoute completed

44. Please continue routing the PCB using the various routing techniques discussed in this exercise. Use Figure 18 below as a guide.
45. Feel free to open the *Properties* panel, section *Board Information – Net: Unrouted / Hidden* to do a first check if you have unrouted connections.
46. **Please bear in mind that the PCB does not have to be completely routed.**

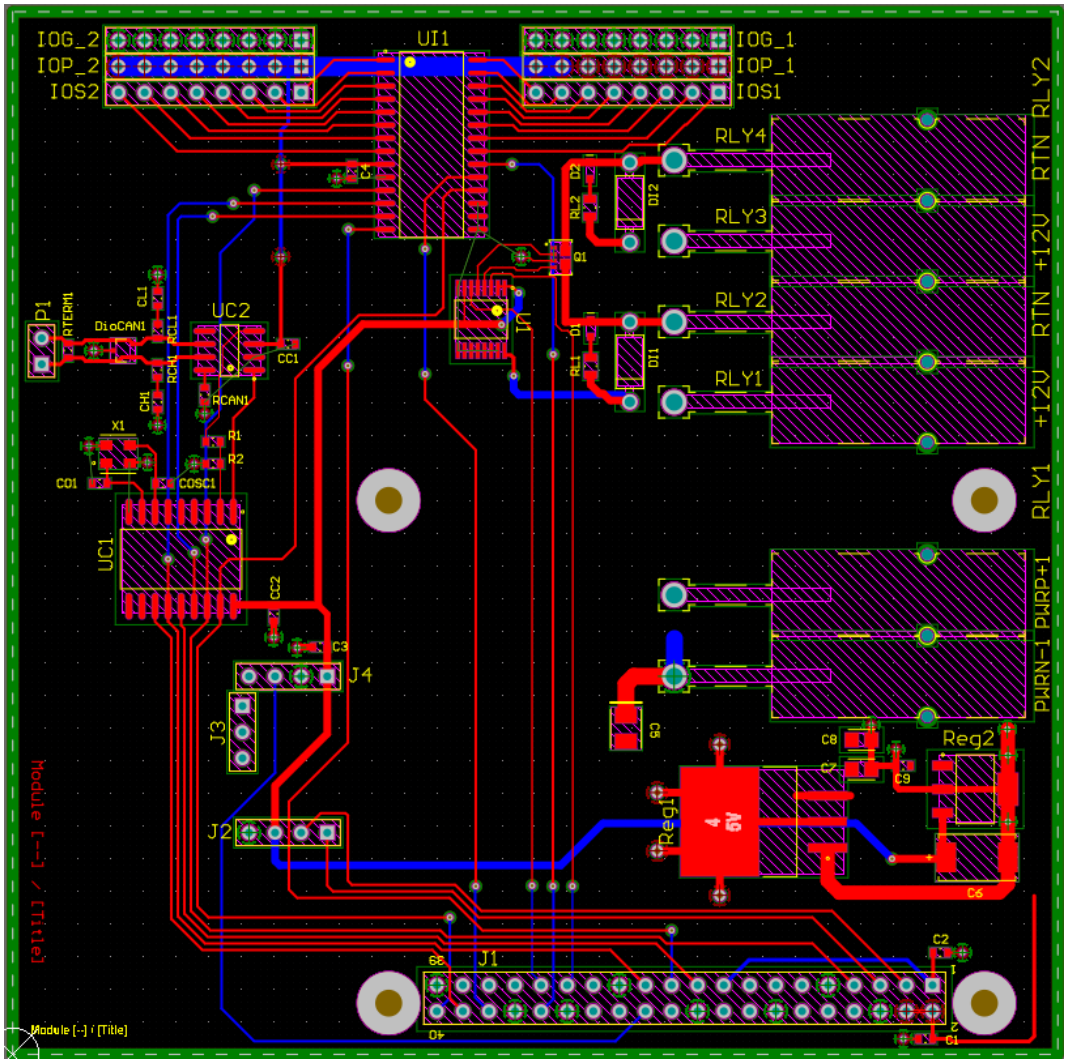

Figure 18. Routed PCB example

47. Save all documents using **File » Save All**.
48. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server** 

.
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 22: PCB Routing - [Add Your Name]- Finished`.
        iii) Select **OK.**
49. **When ready, close the project and any open documents, Window » Close All.**

Congratulations on completing the Module!

Module 22: PCB Routing

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer