# Routing the Board

Routing is the process of laying tracks and vias on the board to connect the component pins. The PCB editor makes this job easy by providing sophisticated interactive routing tools.

In this section of the training, we will manually route the entire board single-sided, with all tracks on the top layer. The Interactive Routing tools help maximize routing efficiency and flexibility in an intuitive way, including cursor guidance for track placement, single-click routing of the connection, pushing obstacles, automatically following existing connections, all in accordance with applicable design rules.

Preparing for Interactive Routing

Before starting to route, it is important to configure the Interactive Routing options, found in the **PCB Editor - Interactive Routing** page of the *Preferences* dialog.
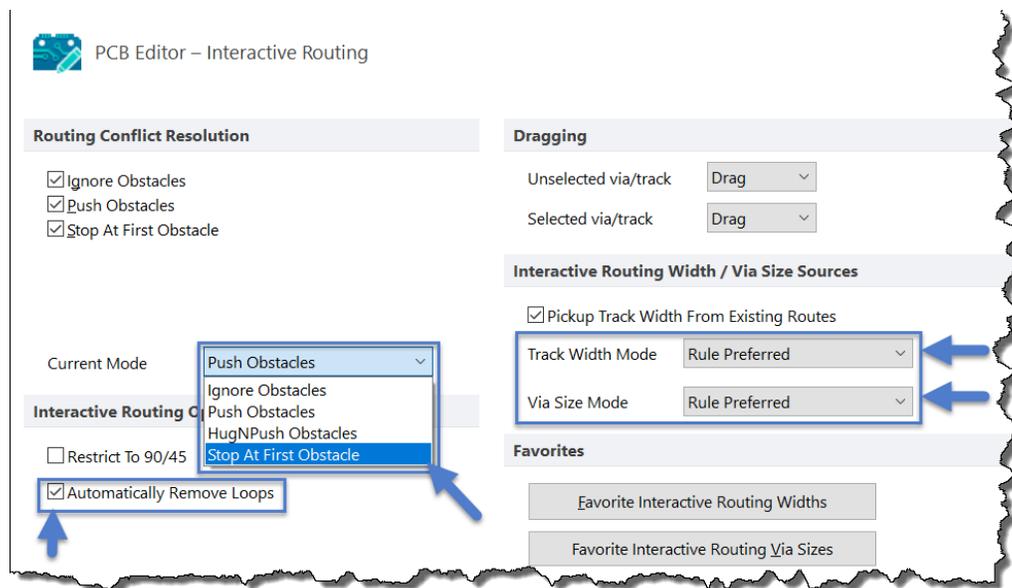

Exercise

1. Set the **Routing Conflict Resolution Current Mode** to *Stop at First Obstacle*. You can cycle through the enabled modes interactively as you route by pressing **Shift+R**.
2. In the **Interactive Routing Options** section of the page, confirm that the **Automatically Remove Loops** option is enabled. This option allows you to change existing routing by simply routing an alternate path - you route a new path until it meets the old path (creating a loop), then right-click to indicate it is

complete - the software then automatically removes the old, redundant part of the routing.

3. Confirm that the **Interactive Routing Width / Via Size Sources** options are both set to **Rule Preferred**. Click **OK** to close the Preferences with these options set

4. Set the Snap Grid to 0.25mm in the **Home | Grids and Units | Snap Grid**.
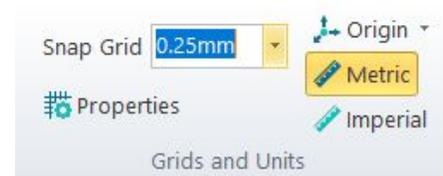
Time to Route

- Interactive routing is launched by clicking the Route button . You only need to use the dropdown menu if you need to select one of the other routing options.
- Since the components are mostly surface mount, the board will be routed on the top layer. As we place tracks on the top layer of the board, we will use the ratsnest (connection lines) to guide us.
- Tracks on a PCB are made from a series of straight segments. Each time there is a change of direction, a new track segment begins. Also, by default the PCB editor constrains tracks to a vertical, horizontal or 45° orientation, allowing you to easily produce professional results. This behavior can be customized to suit your needs, but for this tutorial we will use the default.
- After reaching the target pad, **right-click** or press **Esc** to release that connection - you will remain in *Interactive Routing* mode, ready to click on the next connection line.

Interactively routing the board:

Exercise

1. Check which layers are currently visible by looking at the *Layer Tabs* at the bottom of the workspace. If the Bottom Layer is not visible, open the *View Configurations* dialog, and enable the **Bottom Layer**.
2. Click on the **Top layer** tab at the bottom of the workspace to make it the current, or active layer, ready to route on.
3. It is often easier to route in single layer mode, press **Shift+S** to toggle to in and out of single layer mode.
4. Click the **Home Tab | Routing | Route** button on the Ribbon, or right-click and choose **Interactive Routing** from the context menu. The cursor will change to a crosshair, indicating you are in interactive routing mode.
5. Position the cursor over the lower pad on connector Y1. As you move the cursor close to the pad it will automatically snap to the center of the pad - this is the **Snap To Object Hotspot** feature *pulling* the cursor to the center of the nearest

electrical object (configure the **Range** of attraction in the **Board Options** dialog). Sometimes the **Snap To Object Hotspot** feature pulls the cursor when you don't want it to, in this situation press the **Ctrl** key to temporarily inhibit this feature.

6. **Left-Click** or press **Enter** to anchor the first point of the track.

7. Move the cursor towards the bottom pad of the resistor R1, and click to place a vertical segment. Note how track segments are displayed in different ways. During routing, the segments are shown as:
   - **Solid** - the segment has been placed.
     - **Hatched** - hatched segments are proposed but uncommitted, they will be placed when you left-click.
     - **Hollow** - this is referred to as the look-ahead segment, it allows you to work out where the last proposed segment should end. This segment is *not* placed when you click. The look-ahead mode can be toggled on/off using the **[1]** shortcut during routing.
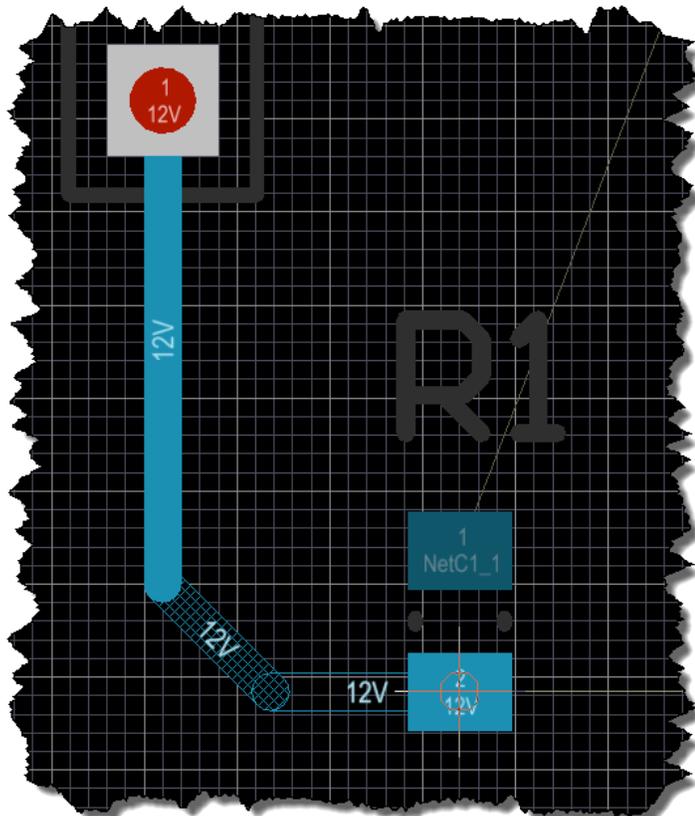
8. Manually route by **Left-Clicking** to commit track segments, finishing on the lower pad of R1. Note how each mouse click places the hatched segment(s). For the connection that you are currently routing, press **Backspace** to rip up the last-placed segment.

9. Rather than routing all the way to the target pad, you can also press **Ctrl+Left Click** to use the *Auto-Complete* function and immediately route the entire connection. Auto-complete behaves in the following way:
     - It takes the shortest path, which may not the best path as you need to always consider paths for other connections yet to be routed.
     - On longer connections, the Auto-Complete path may not always be available as the routing path is mapped section by section and complete mapping between source and target pads may not be possible.
     - You can also Auto-complete directly on a pad or connection line.

10. Let's continue to route all the connections on the board.

11. There is no single solution to routing a board, so it is inevitable that you will want to change the routing. The PCB editor includes features and tools to help with this, they are discussed in the following sections.

12. **File→ Save All** when you are finished routing.
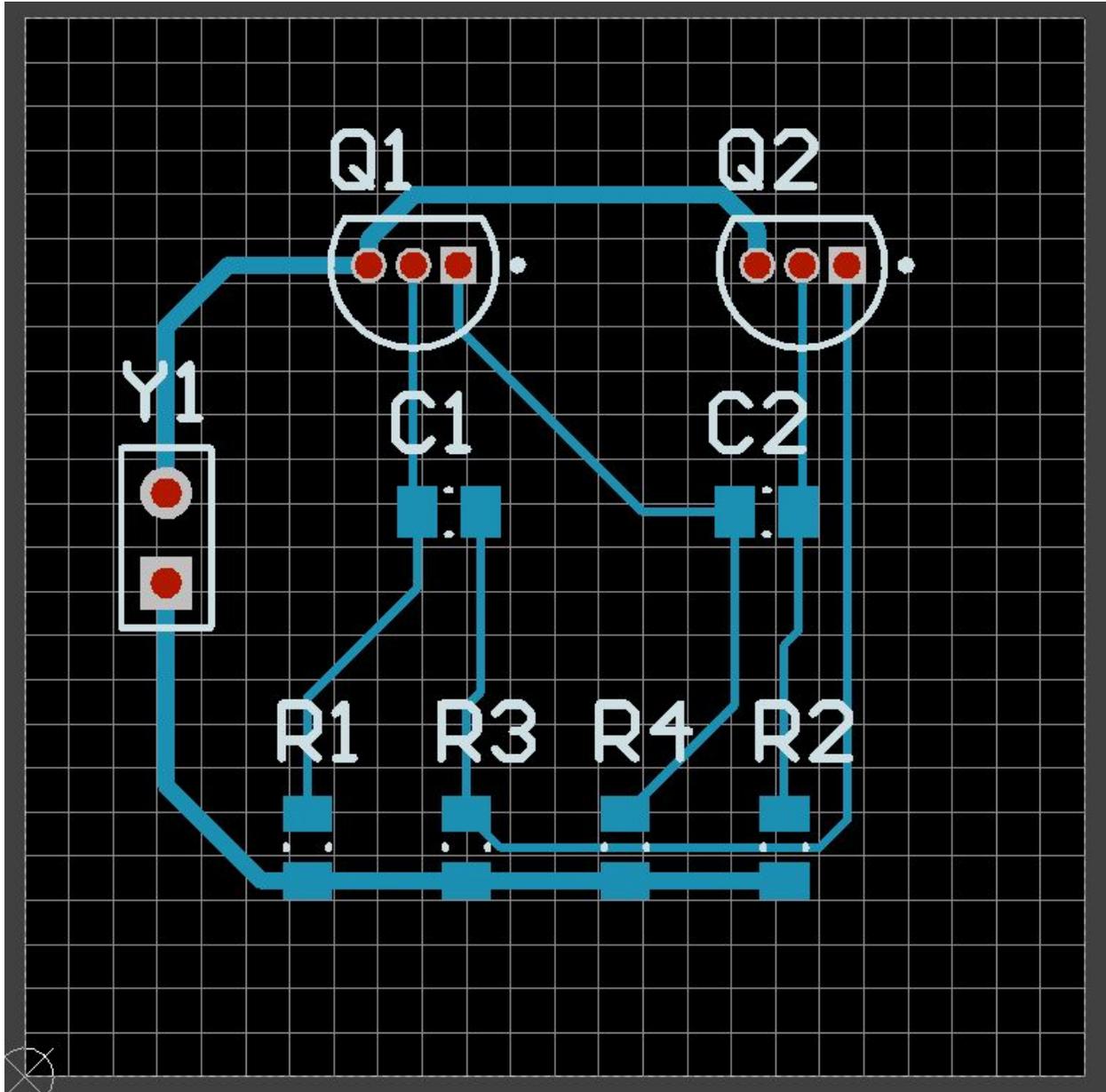
COPYRIGHT

Copyright in the Material on this Website is owned or licensed by Altium LLC and its affiliated companies (collectively, "Altium") **as of 2017**.

Except where necessary for viewing the Material on this website on your browser, or as permitted under the United States Copyright Act or other applicable laws or these Terms of Use, no Material on this website may be reproduced, adapted, uploaded to a third party, linked to, framed, performed in public, distributed or transmitted in any form by any process without the specific written consent of Altium LLC.

TRADEMARKS

ACTIVEBOM®, ActiveRoute®, Altium Designer®, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, TASKING® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.