

Altium[®]

Getting Started with PCB Design



David Haboud

Product & Persona Marketing Engineer

TABLE OF CONTENTS

INTRODUCTION.....	3
THE DESIGN.....	3
CREATING THE PCB PROJECT.....	3
ADDING A SCHEMATIC TO THE PROJECT.....	4
SETTING SCHEMATIC PREFERENCES.....	5
PLACING THE COMPONENTS ON THE SCHEMATIC.....	6
Placing the Resistors	8
Placing the Capacitors	9
Placing the Connector	10
MAKING CIRCUIT CONNECTIONS.....	11
NETS AND NET LABELS.....	12
COMPILING AND VALIDATING THE SCHEMATIC.....	13
ADDING A PCB TO THE PROJECT.....	13
TRANSFERRING THE SCHEMATIC TO THE PCB.....	14
PLACING COMPONENTS ON THE PCB.....	14
DEFINING AND MANAGING DESIGN RULES.....	15
Manually Creating a New Design Rule	16
Modify a Design Rule	16
Using the Design Rule Wizard	17
CONFIGURING THE DESIGN RULES CHECKER.....	20
INTERACTIVELY ROUTING THE BOARD.....	21
TRACE MODIFICATION AND REROUTING.....	23
AUTOMATICALLY ROUTING THE BOARD.....	25
PCB DESIGN RULE VERIFICATION.....	26
VIEWING THE PCB IN 3D.....	27
GENERATING FABRICATION OUTPUT FILES.....	28
CONCLUSION.....	29

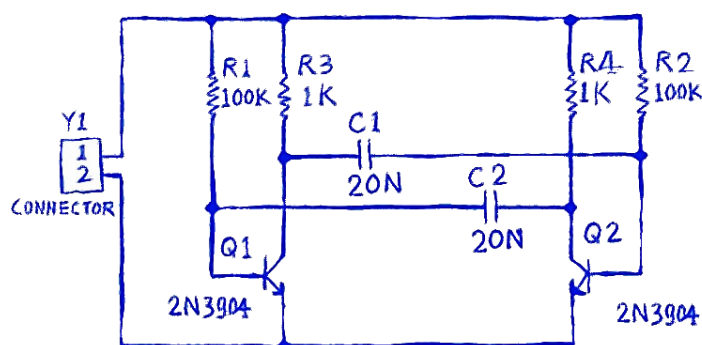
INTRODUCTION

This tutorial will start with a sketch of a circuit and will take you through the process of representing the circuit as a schematic, implementing the circuit as a printed circuit board, and generating the outputs required for board fabrication.

This tutorial is based on an astable multivibrator design. New users of Altium software may find value in reading the article [Exploring Altium Designer](#) for an explanation of the interface, information on how to use panels, and guidelines for managing design documents.

THE DESIGN

The design that will be captured and implemented as a printed circuit board (PCB) is a simple astable multivibrator. The circuit is shown below; it uses two 2N3904 transistors configured as a self running astable multivibrator.



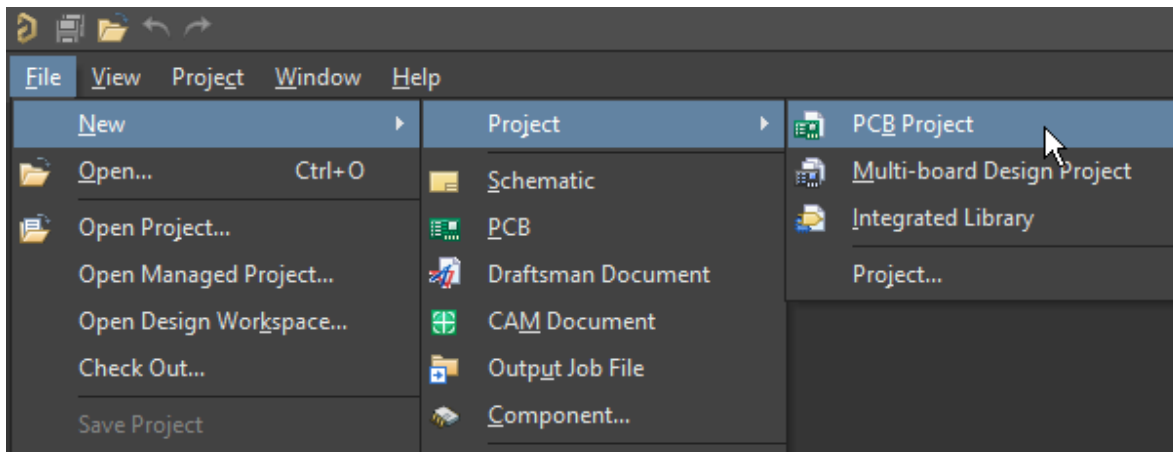
Astable Multivibrator Circuit Sketch

Capturing (drawing) the schematic is the first step of the tutorial, but first a project must be created.

CREATING THE PCB PROJECT

The PCB project is the set of design documents (files) required to specify and manufacture a printed circuit board. More information about managing and creating Altium projects can be found [here](#).

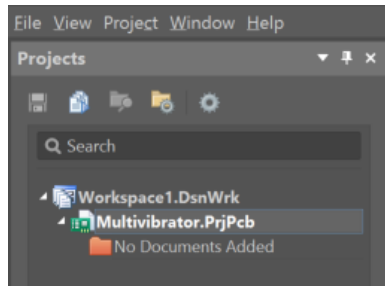
1. Create a New PCB Project. **File » New » Project » PCB Project.**
2. Use **File » Save Project As...** to save the new project. Choose or create the project directory and save the project with the name *Multivibrator*.



Create a New PCB Project

GETTING STARTED WITH PCB DESIGN

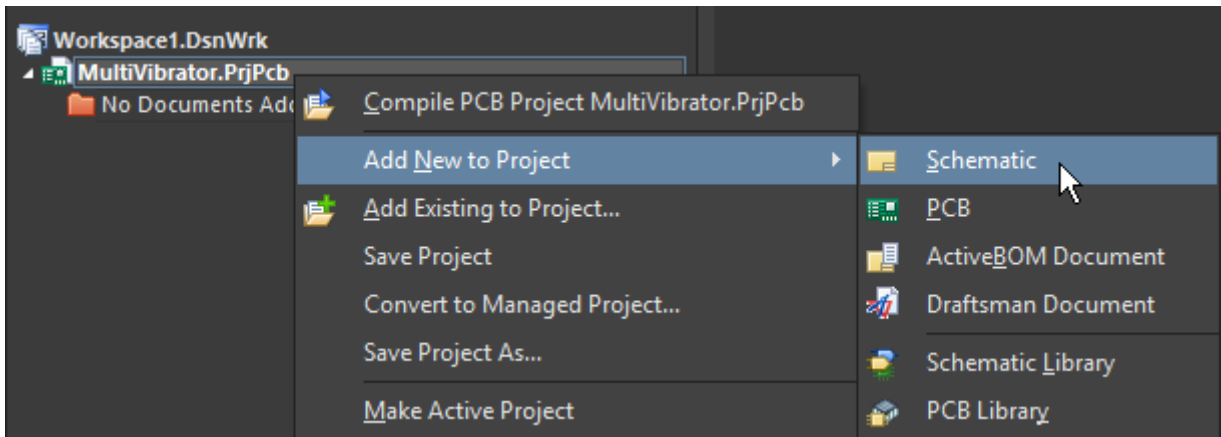
3. The project name will appear in the Projects panel.



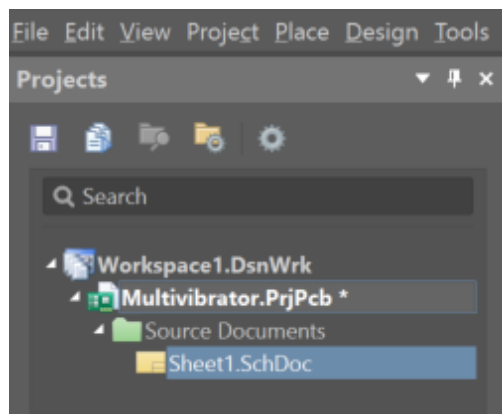
Projects Panel

ADDING A SCHEMATIC TO THE PROJECT


1. **Right-Click** on **Multivibrator.PrjPcb** and select **Add New to Project » Schematic**.



Create a New Schematic Sheet for the Circuit

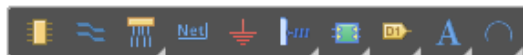


Schematic Sheet in Projects Panel

1. To save the new schematic sheet, select **File » Save As** or select . Type the name **Multivibrator** in the **File Name** field and click **Save**. (The file extension *.SchDoc is automatically added).

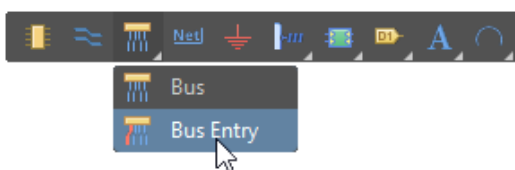
GETTING STARTED WITH PCB DESIGN

When the blank schematic sheet opens, the main menu bar and related buttons change in context for schematic entry displaying the **Active Bar** in the workspace. The **Active Bar** is displayed in all document editors in Altium Designer and provides placement controls for PCB, schematic, Draftsman, and library objects at your fingertips. Also, you can customize many aspects of the workspace including repositioning and customization of panels and toolbars.



Schematic Document Active Bar

Active Bar icons which have a triangle in the lower right allow you to right-click on them to select different objects for placement:

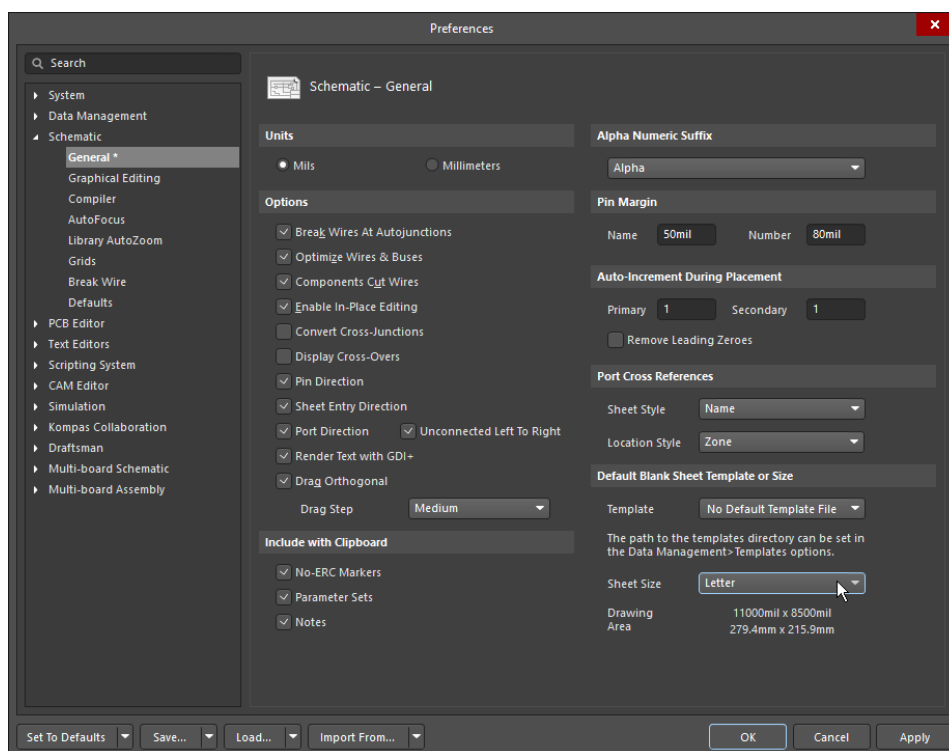


Bus Entry on Schematic Document Active Bar

SETTING SCHEMATIC PREFERENCES

Before composing the circuit, set up the appropriate document options by completing the following steps:

1. Set the sheet size to Letter as shown in the following guide:



Setting Page Size in the Preferences Panel

To make the document fill the viewing area, select **View » Fit Document** (shortcut: V, D).

Save the schematic by clicking  or selecting **File » Save** (shortcut: F, S).

GETTING STARTED WITH PCB DESIGN

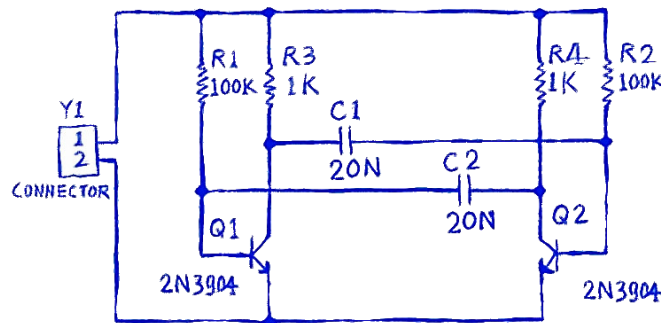
You can activate any menu by pressing the menu accelerator key (the underlined letter in the menu name). Subsequent menu items will also have accelerator keys that you can use to select that item.



For example, the shortcut for selecting the **View » Fit Document** menu item is to press the **V** key followed by the **D** key.

Additionally, many sub-menus, such as the Select menu (in the Edit menu), can be called directly. For example, to activate the **Edit » Select » Touching Line** command, you need only press the **S** key (to call up the Select menu directly) followed by the **L** key.

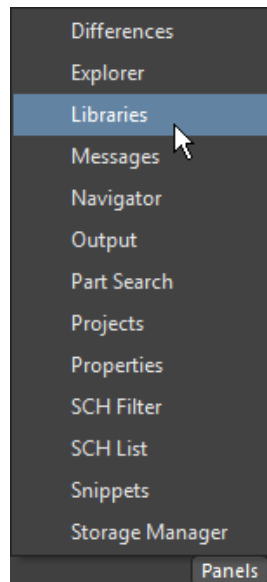
PLACING THE COMPONENTS ON THE SCHEMATIC



Setting Page Size in the Preferences Panel

Refer to the rough schematic sketch above for the general layout of the circuit, and place **2N3904** transistors **Q1** and **Q2**.

1. Display the Libraries panel by clicking the Panels button at the bottom right of the software, and select Libraries.



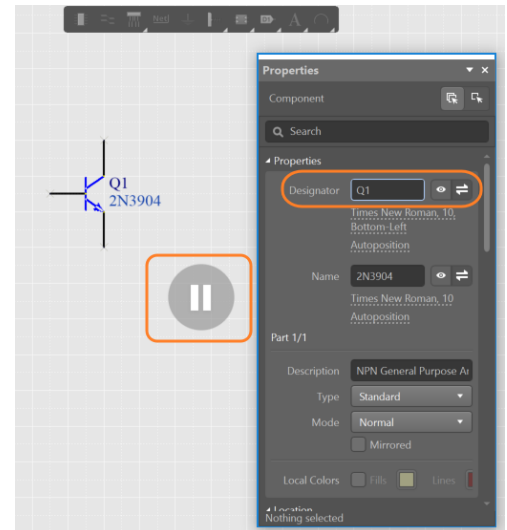
Opening the Libraries Panels

2. Find the **2N3904** transistor in the panel by either scrolling through the component list or by using the search button.
3. Click on the **2N3904** entry, then click the **Place** button. The cursor will change to a cross hair and you will have an outlined version of the transistor floating on your cursor.

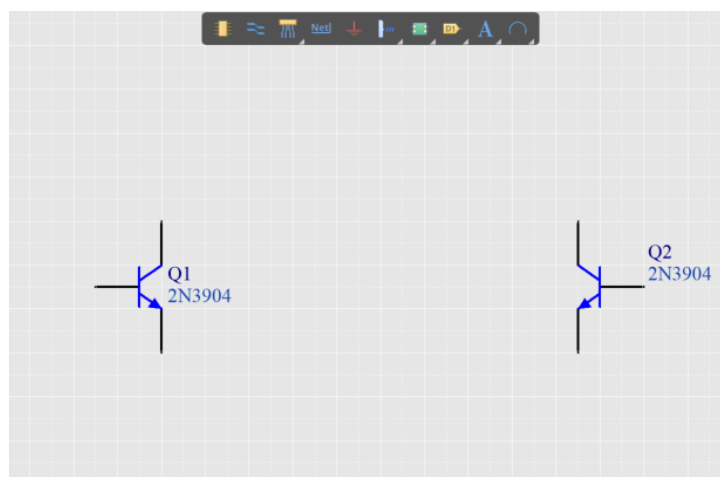
Do not place the transistor yet!

GETTING STARTED WITH PCB DESIGN

- Before placing the part on the schematic, press the **Tab** key to open the properties panel and pause placement. In the **Properties** section of the dialog, type **Q1** into the **Designator** field.
- Continue placement by clicking on the pause button that is displayed in the center of the schematic. Once the transistor is in the desired position, left mouse click or press **Enter** on the keyboard to place the transistor onto the schematic.
- The system remains in component placement mode with the part outline floating on the cursor. Place the second transistor. Because it is the same as the previous one, the software will automatically increment a component's designator.
- Referring to the rough schematic diagram shown before, notice that **Q2** is drawn as a mirror of **Q1**. To flip the orientation of the transistor that is floating on the cursor, press the **X** key on the keyboard. This flips the component horizontally (along the **X**-axis).
- Move the cursor to position the part to the right of **Q1**. To position the component more accurately, press the **PageUp** key twice to zoom in two steps. You should now be able to see the grid lines.
- Once positioned, left mouse click or press **Enter** to place **Q2**. Once again, a copy of the transistor attached to the cursor will be placed on the schematic, and the next transistor will be floating on the cursor ready to be set.
- Exit part placement by clicking the **Right Mouse Button** or pressing the **Esc** key. The cursor will revert to a standard arrow.



Properties Panel During Part Placement



Mirrored Transistors Placed On Schematic Sheet

Use the following keys to manipulate the part floating on the cursor:



- **Y** flips the part vertically
- **X** flips the part horizontally
- **Spacebar** rotates the part by 90° anti-clockwise.
- **Shift+Spacebar** rotates the part by 90° clockwise.

GETTING STARTED WITH PCB DESIGN

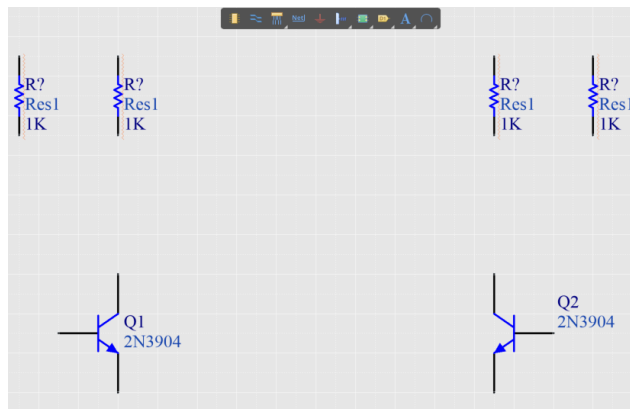
Placing the Resistors

Place the four resistors into the schematic.

1. The **Properties** panel will be used frequently for the remainder of this tutorial. Dock the Properties panel to the left side of the workspace.

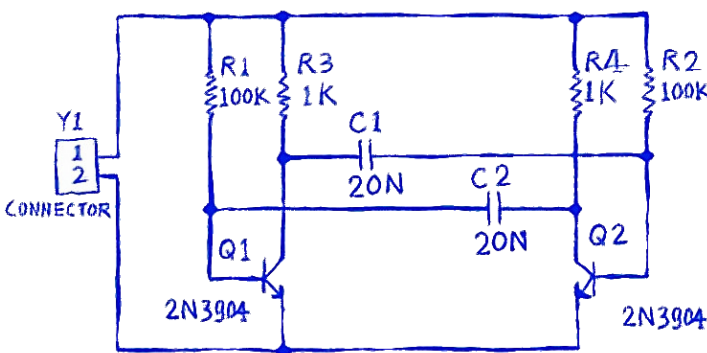
For additional information on panels see: [Working with Panels in Altium Designer](#)

2. In the **Libraries** panel, make sure the **Miscellaneous Devices.IntLib** library is active.
3. Set the filter by typing **res** in the filter field below the library name.
4. Click on **Res1** in the components list to select it, then click the **Place** button. The resistor symbol is floating on the cursor.
5. Without changing any additional properties, place the four resistors as shown below.

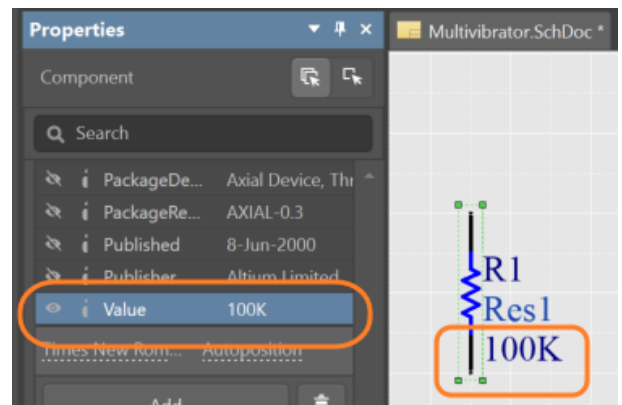


Resistors Placed on Schematic Sheet

6. Using the rough design below as a reference, select each resistor in the schematic and change the **Designator** and **Value** properties of each resistor by clicking on the resistor and updating the fields in the Properties panel.



Astable Multivibrator Circuit Sketch

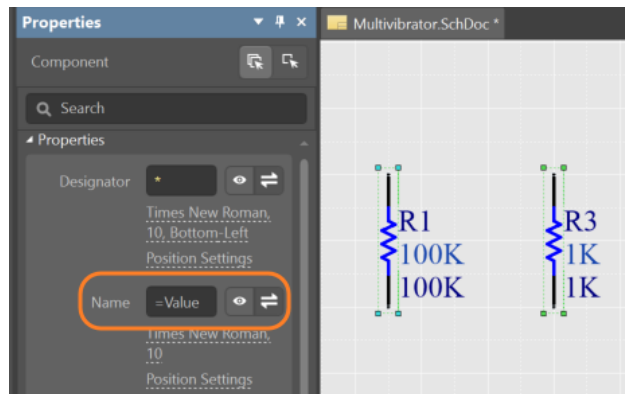


Value of R1 Set in Properties Panel


7. The **Value** property updated in the above step is used for circuit simulation. The **Name** property is typically used to identify the component on the schematic and in downstream reports such as a Bill of Materials. By entering `=va1ue` in the **Name** property, the resistance set in the **Value** property is echoed to the **Name** property. Select all four resistors by clicking with the **Shift** key down.

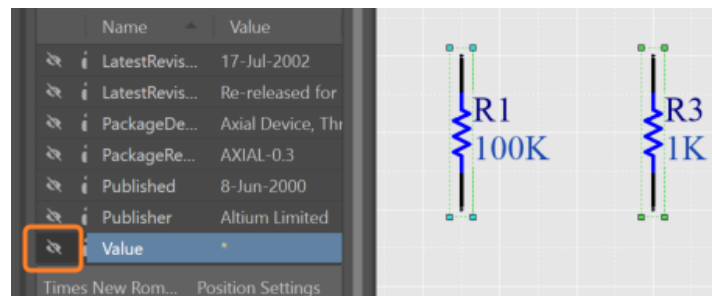
GETTING STARTED WITH PCB DESIGN

- The **Properties** panel now reflects the parameters that are all common across the four resistors. Change the **Name** property to `=value` and the **Name** property will reflect the resistance set in **Value** across all selected resistors as shown below.



Name Set to Value Parameter of Resistors in Properties Panel

- Finally, with all four resistors remaining selected, scroll down to the **Value** property and click on  to turn visibility off. The four resistors will reflect the change.

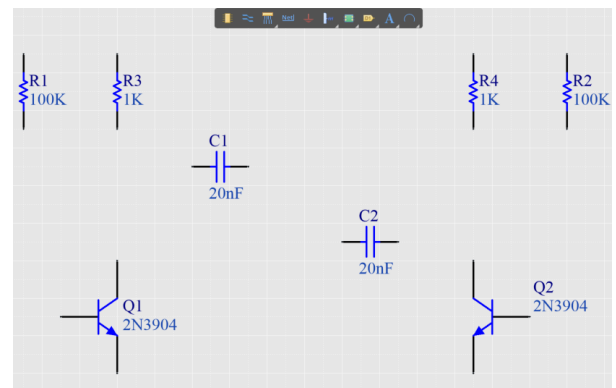


Resistor Value Parameter Visibility Turned Off

Placing the Capacitors

Place the two capacitors.

- The capacitor part is also in the **Miscellaneous Devices. IntLib library**, which should already be selected in the **Libraries** panel.
- Type `cap` in the component's filter field (located below the selected library) in the **Libraries** panel.
- Click on **Cap** in the components list to select it, then click the **Place** button. The capacitor symbol is floating on the cursor.
- Press the `Tab` key to edit the capacitor's attributes. In the Component Properties dialog set the **Designator** to **C1**, the **Name** to `=value`, set the **Value** property to `20nF`, and disable the visibility of the **Value** parameter.



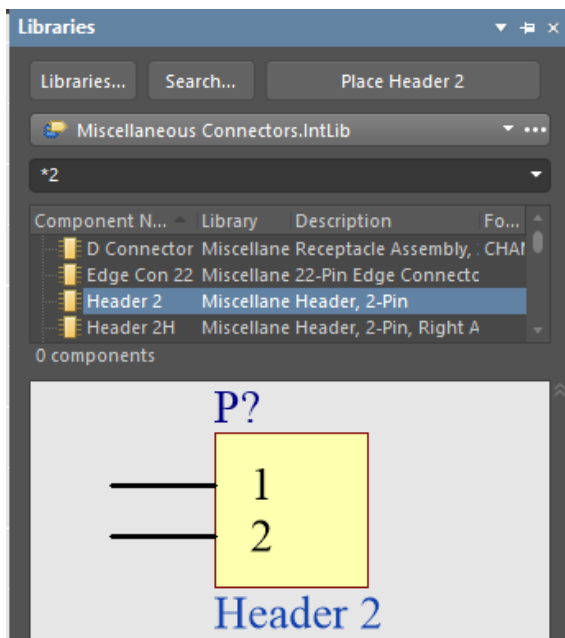
Capacitors Placed on Schematic Sheet

- Using the image below as a guide, position and place the two capacitors in the same way that you placed the previous parts. **Right-click** or press `Esc` to exit placement mode.


Placing the Connector

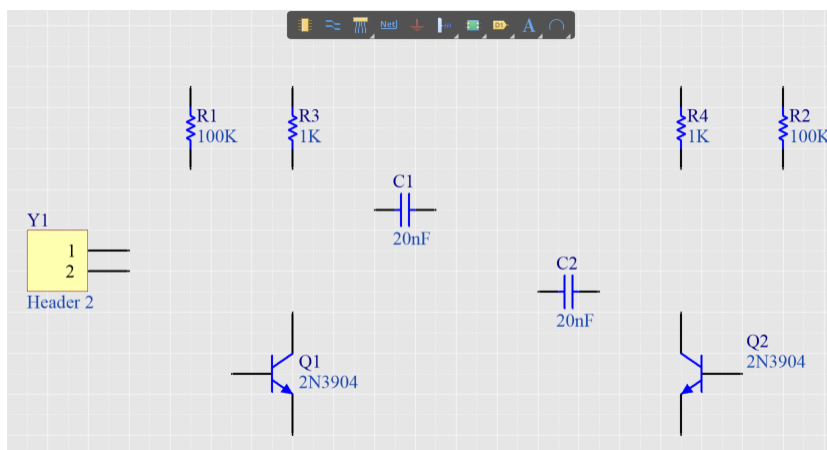
Place the two capacitors.

1. Select **Miscellaneous Connectors.IntLib** in the **Libraries** panel. The required connector is a two-pin socket; type ***2** into the Libraries panel filter field.



2-Pin Header in Libraries Panel

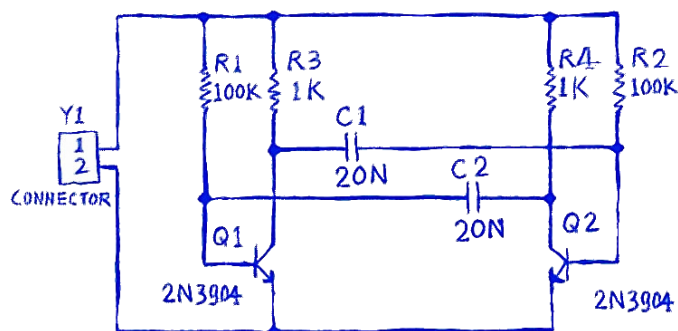
2. Select **Header 2** from the parts list and click the **Place** button. Press Tab to edit the properties and set **Designator** to Y1.
3. Before placing the connector, press x to flip it horizontally so that it is in the correct orientation. Click to place the connector on the schematic, as shown in the image below.
4. **Right-click** or press **ESC** to exit part placement mode.
5. Save your schematic by selecting **File » Save** from the menus (shortcut: F, S) or clicking  .



2-Pin Header in Libraries Panel

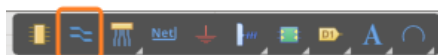
MAKING CIRCUIT CONNECTIONS

Referring to the sketch of the circuit below, use the place wire command to create connectivity between circuit elements.



Astable Multivibrator Circuit Sketch

1. To provide a good view of the schematic sheet, press the `PageUp` key to zoom in or `PageDown` to zoom out. Alternatively, hold down the `Ctrl` key and roll the mouse wheel to zoom in/out, or hold the `Ctrl` + **Right Mouse** button down and **drag the mouse up/down** to zoom in/out.
2. Choose the **place wire** command from the **Active Bar** (highlighted below) and wire the lower pin of resistor **R1** to the base of transistor **Q1** in the following manner by positioning the cursor over the bottom end of **R1**. A red connection marker (large cross) will appear at the cursor location indicating that the cursor is over a valid electrical connection point on the component.

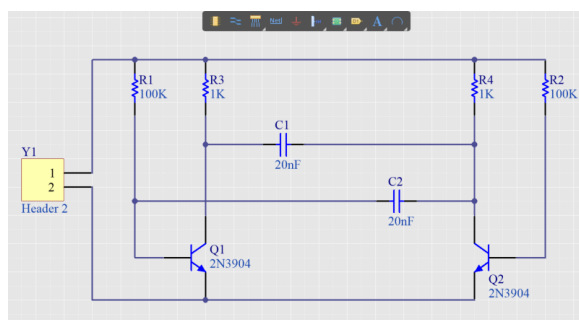


Place Wire Command from the Active Bar

3. Click the **Left Mouse** Button or press `Enter` to anchor the first wire point. Move the cursor and a wire will extend from the cursor position back to the anchor point. The default corner mode is a right angle, the tip box below explains how to change the corner mode. Right angle is the best choice for this circuit.
4. Position the cursor over the base of **Q1** until you see the cursor change to a red connection marker. Click or press `Enter` to connect the wire to the base of **Q1**. The cursor will release from that wire.

Note: The cursor remains a cross hair, indicating that the system is ready to place another wire. To exit placement mode completely and go back to the arrow cursor, **Right-Click** or press `ESC` again. Do not exit at this time.

5. Add connections for the remaining elements of the circuit, as shown in the figure below. When finished placing all connections, right-click or press `ESC` to exit placement mode. The cursor will revert to an arrow.



Complete Schematic Sheet Wiring

NETS AND NET LABELS

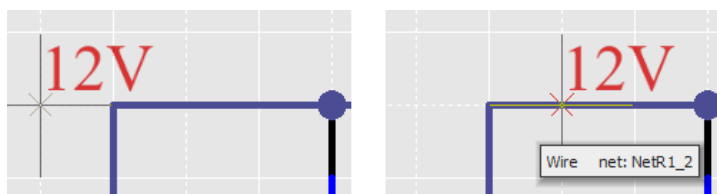
Complete the following steps to place net labels on the two power nets.

1. Select **Place Net Label** from the **Active Bar** (highlighted below). A net label will appear floating on the cursor.



Place Net Label from Active Bar

2. To edit the net label before it is placed, press the **Tab** key to pause label placement and go to the **Properties** panel and enter **12V** in the **Net Name** property and continue placement of the label.
3. Place the net label so that the bottom left corner of the net label touches the uppermost wire on the schematic, as shown in the image below. The cursor will change to a red cross when the net label is correctly positioned to connect to the wire. If the cross is light grey, it means there will not be a valid connection made. See the net label in free space (left image) and positioned over a wire (right image) below.

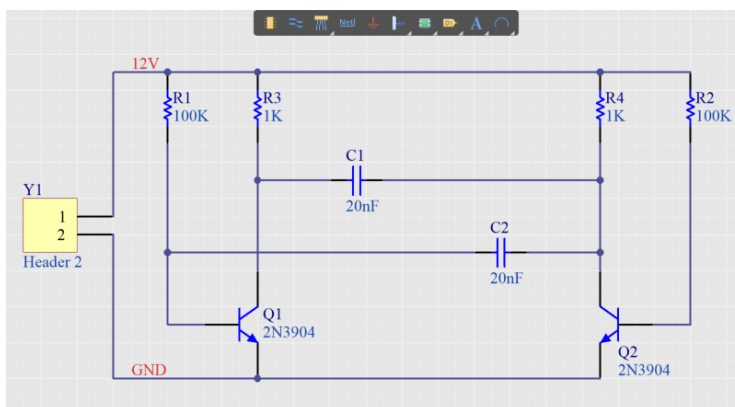


Incorrect Vs. Correct Placement of Net Labels

4. After placing the first net label, net label placement mode will remain active. Press the **Tab** key again to edit the second net label before placing it. Type **GND** in the **Net Name** property.

Place the net label so that the bottom left of the net label touches the lower most wire on the schematic (as shown in the image below). **Right-click** or press **ESC** to exit net label placement mode.

5. Select **File » Save** (shortcut: F, S) to save your circuit. Save the project as well.



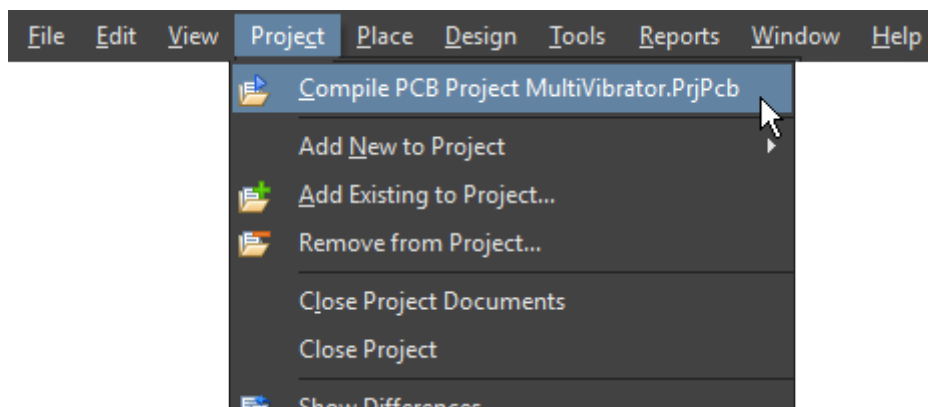
Saved Completed Schematic Sheet

COMPILING AND VALIDATING THE SCHEMATIC

Compiling a project checks for drafting and electrical rules errors in the design documents and details all warnings and errors in the Messages panel. The process also provides detailed information in the Compiled Errors panel. The rules have been previously configured for this tutorial and the design is ready to be validated.

1. Select **Project » Compile PCB Project Multivibrator.PrjPcb** To compile the project. If you have already compiled, select **Project » Recompile PCB Project Multivibrator.PrjPcb**.

When the project is compiled, warnings and errors are displayed in the **Messages** panel. The panel will only appear automatically if there are errors detected.



Compile PCB Project

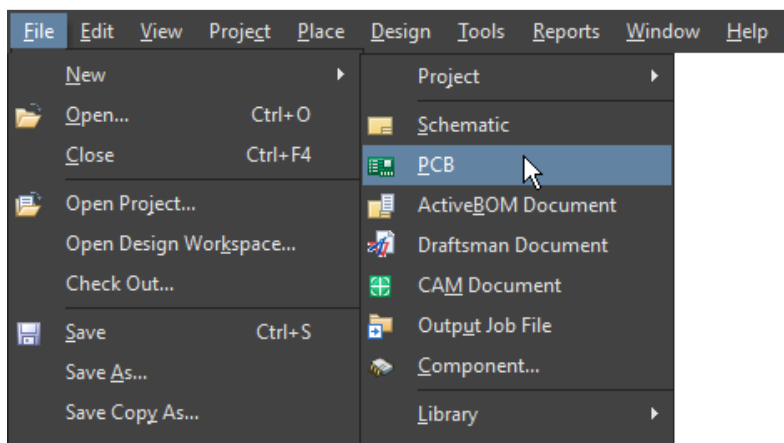
2. Select **View » Fit All Objects** (shortcut: V, F) from the menus to display the entire schematic, then save the schematic and the project.

The schematic is complete. The next phase of the tutorial is to implement the design as a PCB. Additional information about schematic design can be found [here](#).

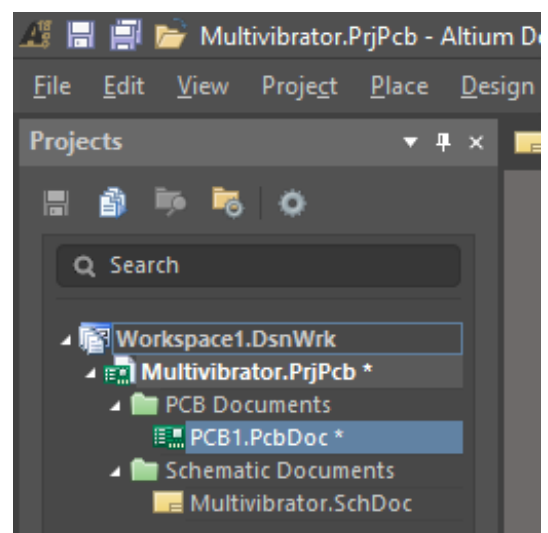
Additional information on design compiling and verification can be found [here](#).

ADDING A PCB TO THE PROJECT

1. Select **File » New » PCB**.




Create a New Pcb Design File for the Project



New Pcb Design File in Projects Panel

GETTING STARTED WITH PCB DESIGN

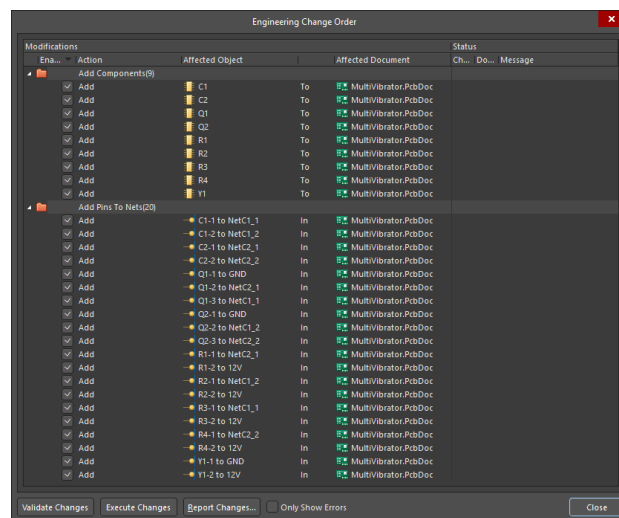
- To save the new PCB file, select **File » Save As** or select . Type the name `Multivibrator` in the **File Name** field and click **Save**.
- When the PCB design file opens, the main menu bar and related buttons change context for schematic entry and the **Active Bar** is displayed in the workspace.



PCB Design File Active Bar

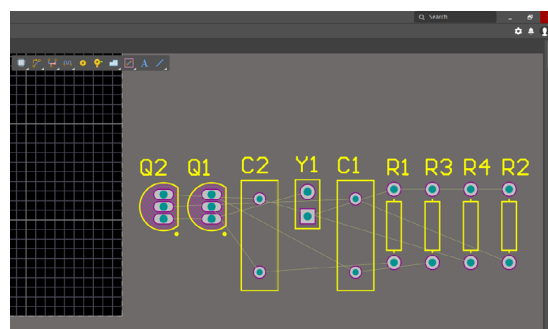
TRANSFERRING THE SCHEMATIC TO THE PCB

- While editing the PCB design file, initiate the process of transferring the schematic design to the PCB by selecting **Design » Import Changes from Multivibrator.PRjPcb** the PCB Editor.



Engineering Change Order

- The PCB footprints of the components are placed to the right of the PCB outline, as shown below.



Components Placed Outside PCB Outline

PLACING COMPONENTS ON THE PCB

In this section the components will be placed on the PCB. Connection lines are automatically re-optimized as you move a component. In this manner the connection lines act as a guide to the optimum position and orientation of the component as it is placed.

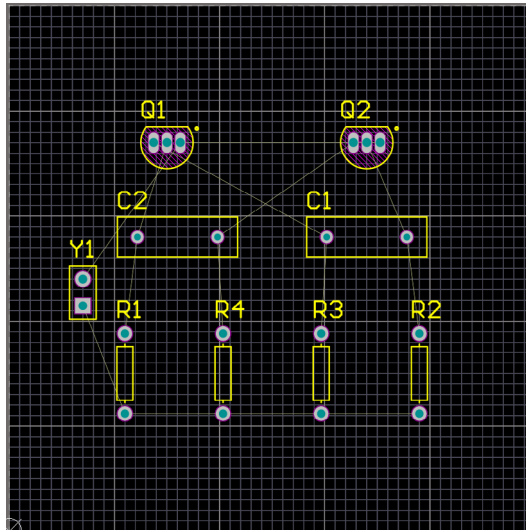
- Position the board in the editor by moving the mouse while holding the `Ctrl` key and right mouse button down. If the mouse has a wheel, press and hold it while moving the mouse to zoom in and out.

GETTING STARTED WITH PCB DESIGN

2. To pan the screen, move the mouse while holding the right mouse button down. All zooming is relative to the current cursor location; position the cursor before zooming.

See [Getting to know your Editor](#) for additional information on zooming and panning.

3. The components in the tutorial will be placed as shown in the image below. To place the components, position the cursor over the middle of the component and click-and-hold the left mouse button. The cursor will change to a cross hair and jump to the reference point for the part. While continuing to hold down the mouse button, move the mouse to drag the component.

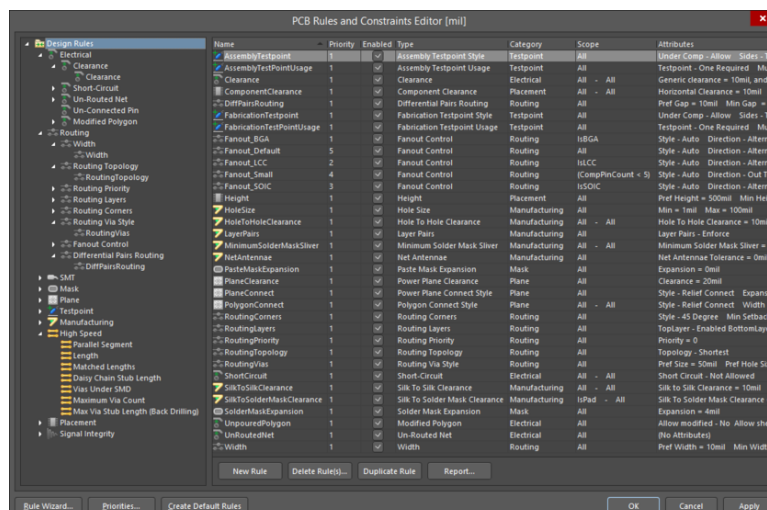


Components Placed on Board Layout

DEFINING AND MANAGING DESIGN RULES

Design rules are defined and managed from within the PCB Rules and Constraints Editor dialog. The dialog has two sections:

- The tree on the left lists the different rule categories. Expand a category to display the individual rule types available. Expand a rule type to display all rules of that type that are currently defined.
- The right-hand side of the dialog presents information in relation to what is currently selected in the tree - either a summary of defined rules of the selected rule type or category, all defined rules for the entire system or, if an actual rule is selected, the constraints for that rule.

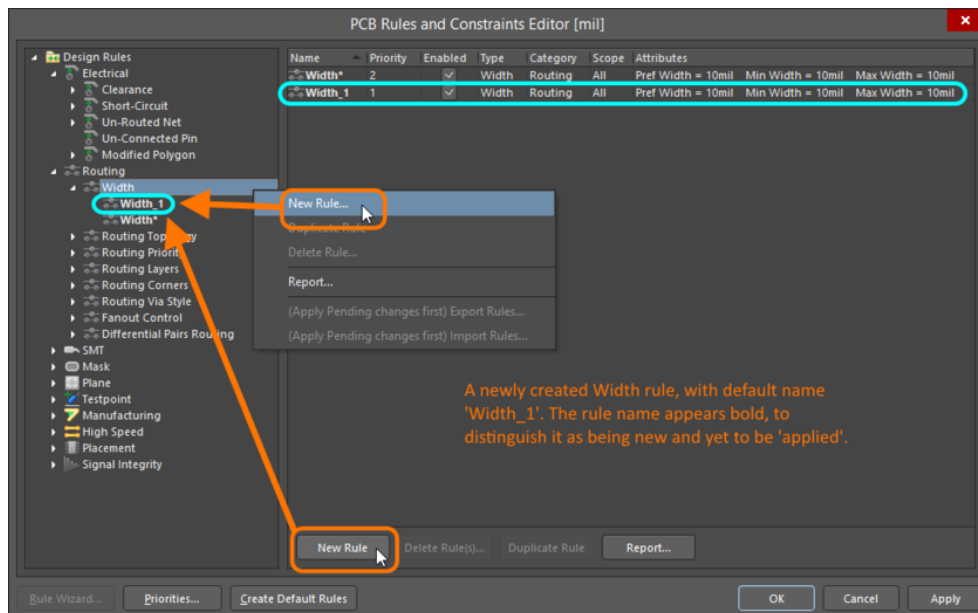


PCB Rules and Constraints Editor

GETTING STARTED WITH PCB DESIGN

Manually Creating a New Design Rule

1. Open the **PCB Rules and Constraints Editor** by selecting **Design » Rules** from the toolbar.
2. Navigate to **Routing** and select the **Width** rule type.
3. Either click the **New Rule** button below the rule summary list or **right-click** and select **New Rule** from the context menu.



Creation of a New PCB Design Rule

Note: Keep track of the rule name for the next section. When a new rule is added, it will initially be given a default name based on the specific type of rule. If a rule with that name already exists, it will simply be given an incremented numerical suffix (i.e. Width_1, Width_2, and so on).

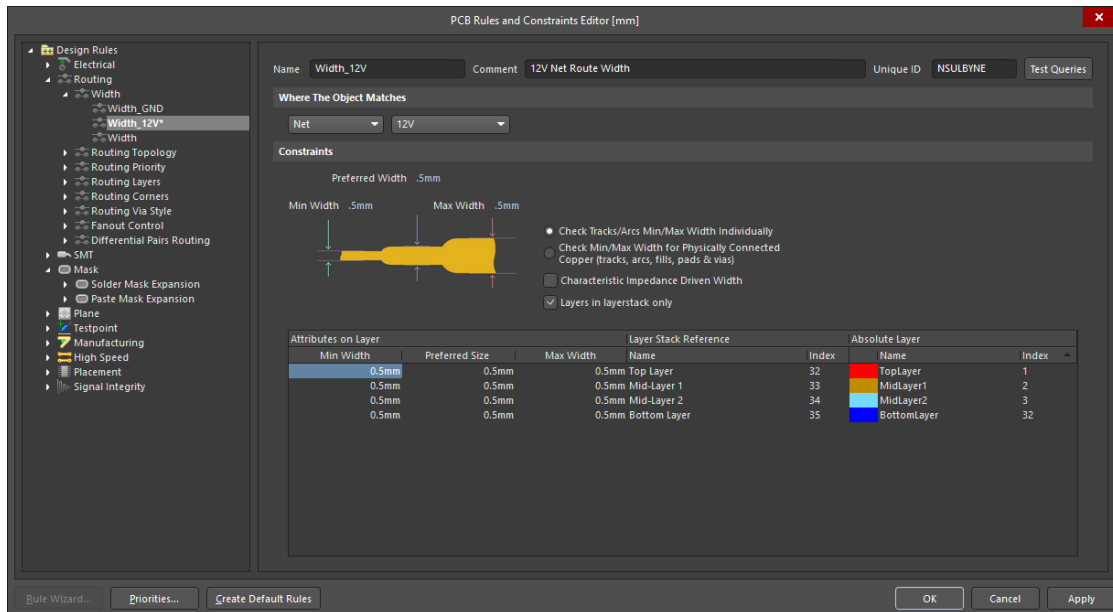
Modify a Design Rule

1. **Click** on the new entry for the rule in the folder-tree pane or **double-click** on its entry in the summary list.

Define the new rule by setting the following values:

2. Set **Name** to width_12V.
3. Define the scope of the rule by setting the first dropdown menu to **Net** and the second dropdown menu to **12V**.
4. Set the constraints of the rule to **Preferred Width** = 0.50mm, **Min Width** = 0.50mm, and **Max Width** = 0.50mm.
5. **Click OK**.

GETTING STARTED WITH PCB DESIGN

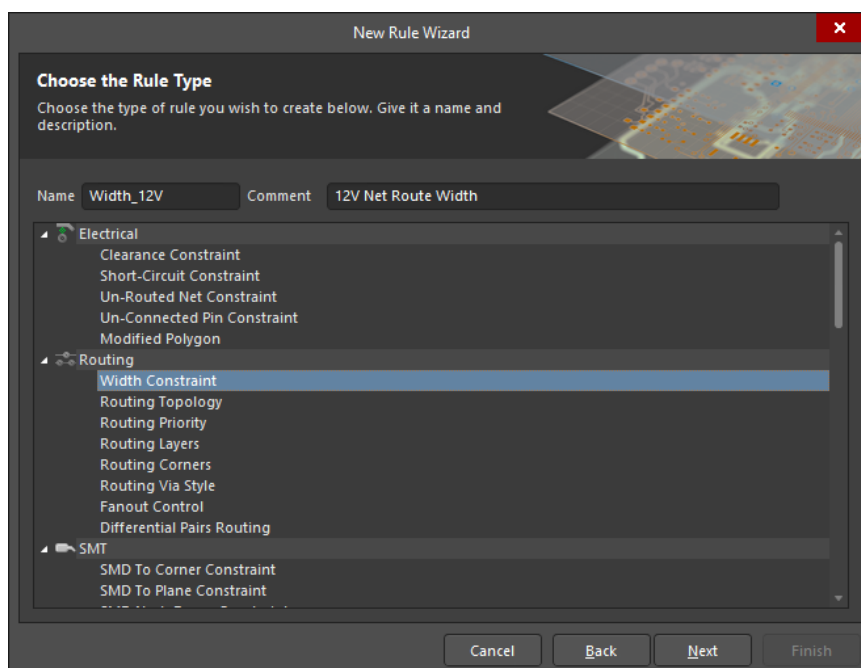


Detailed Design Rules Control

Using the Design Rule Wizard

Design rules can also be created using the Design Rule Wizard. If you click Finish before completing the entire Wizard, the new width rule will be created using the system defaults for the component type you selected.

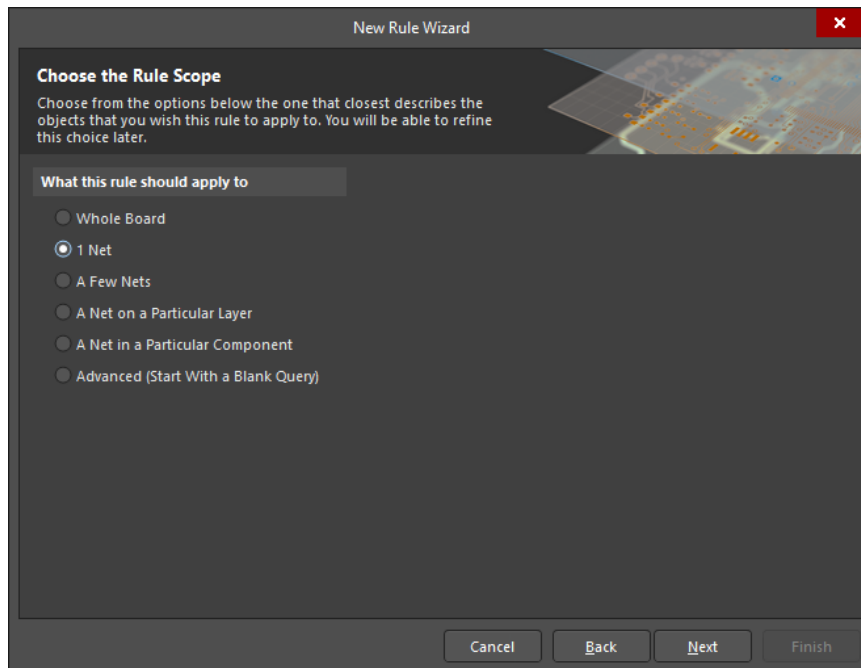
1. Launch the **Design Rule Wizard** from the PCB editor by clicking **Design » Rule Wizard** from the main menus or by clicking the Rule Wizard button in the PCB Rules and Constraints Editor dialog.
2. Choose the **Rule Type**.
3. Type `width_12V` in the **Name** field and `12V Net Route Width` in the **Description** field.
4. Select **Routing » Width Constraint**.
5. **Click Next**.



Rule Type Selection

6. Choose the **Rule Scope** of **1 Net**.

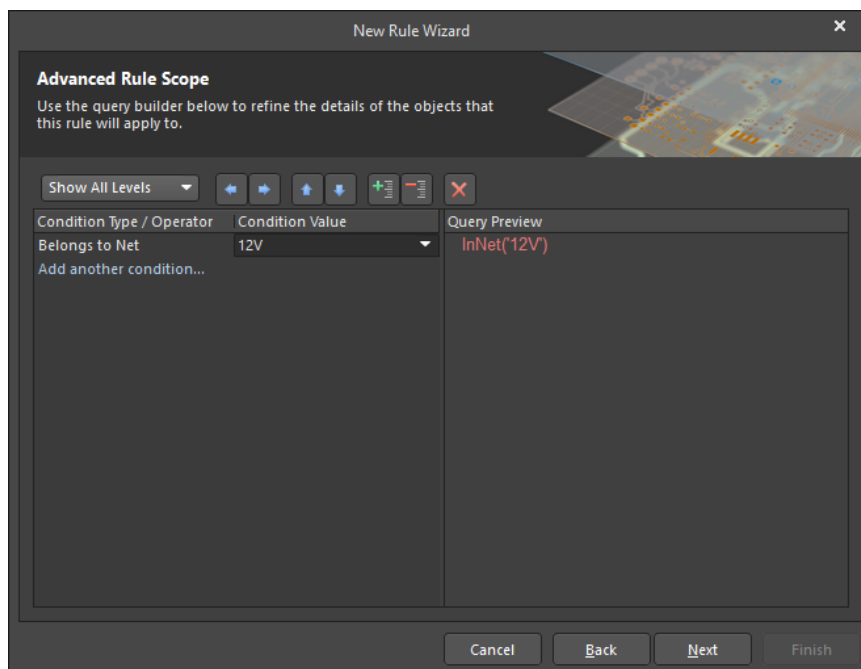
7. **Click Next**.



Rule Scope Selection

8. Set the **Condition Value** to 12v from the dropdown and click **Next**.

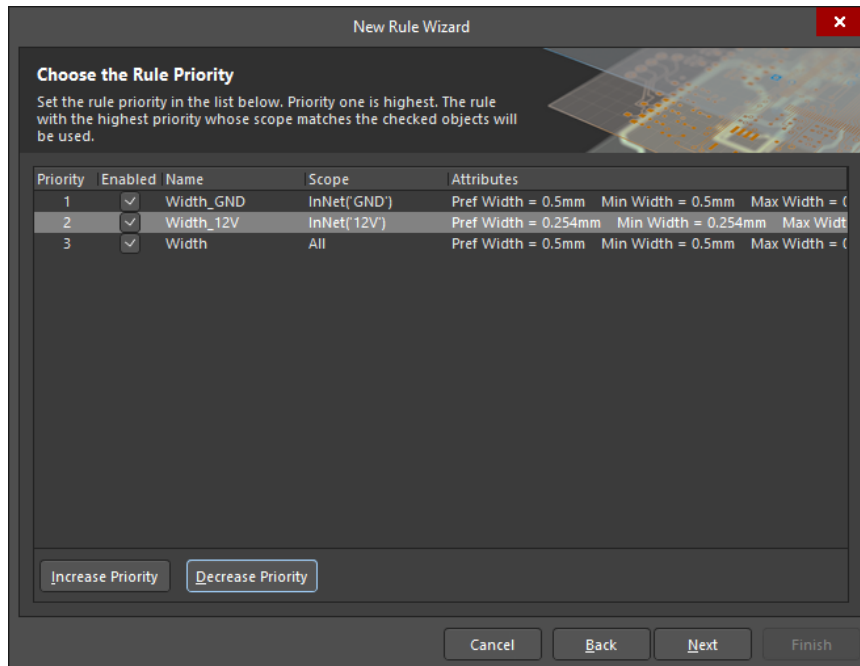
Note: Creating a custom query is a powerful tool for the design process. Visit the [Query Language Reference Page](#) for more information on creating custom queries.



Advanced Rule Scope Selection

GETTING STARTED WITH PCB DESIGN

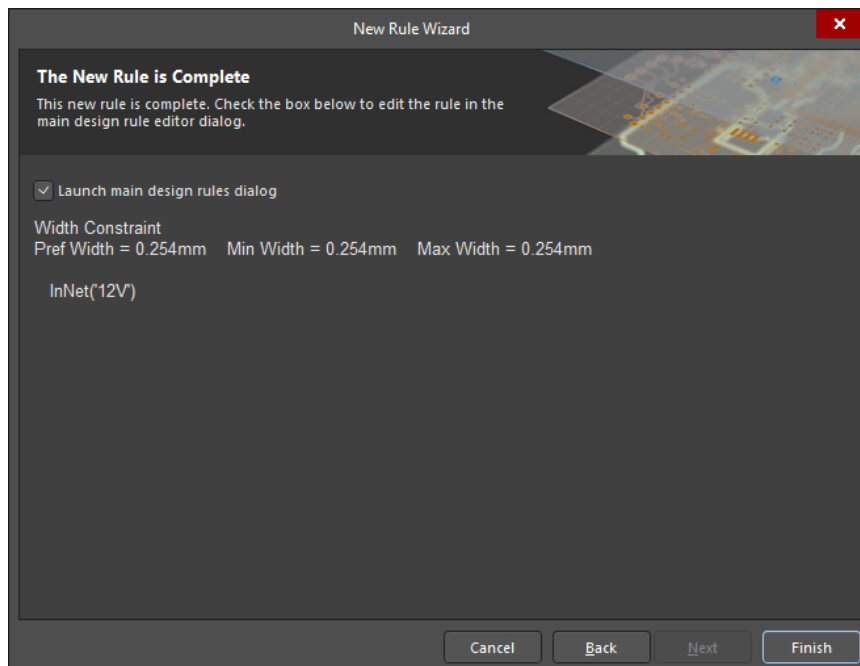
9. Set Rule Priority to **2** by **clicking** on the **Width_12V** and **clicking Decrease Priority**.



Rule Priority Selection

10. Click **Next**.

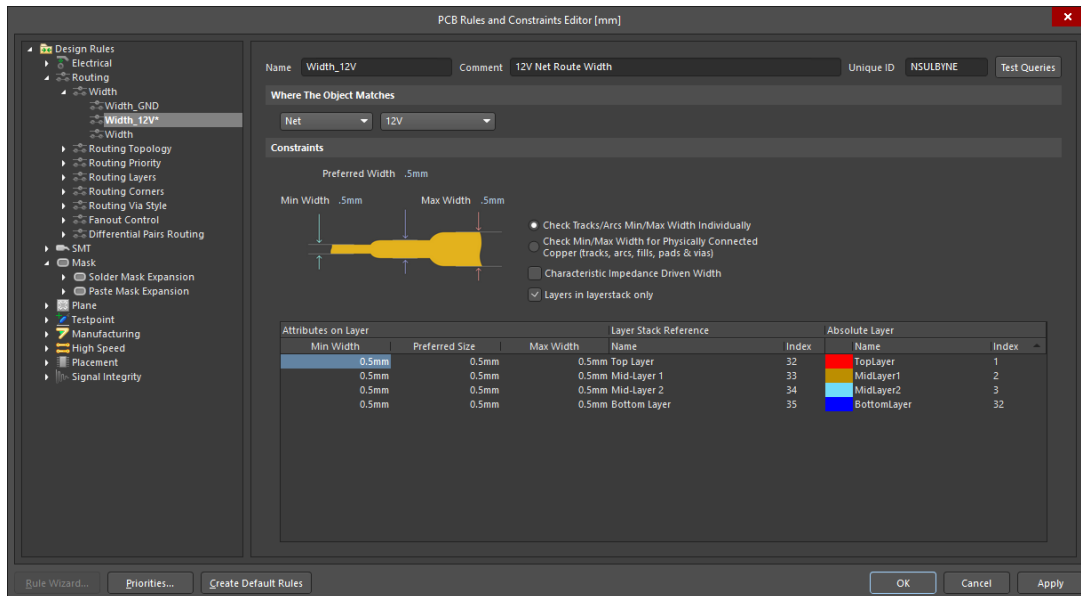
11. Ensure the **Launch main design rules dialog** box is checked and click **Finish**.



Rule Completion

GETTING STARTED WITH PCB DESIGN

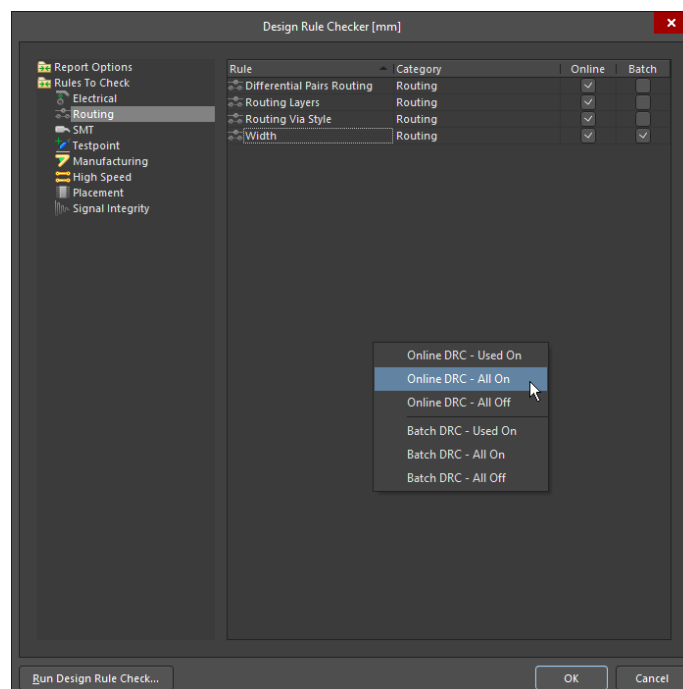
12. Set the constraints of the rule to **Preferred Width** = 0.50mm, **Min Width** = 0.50mm, and **Max Width** = 0.50mm.
13. Click **OK**.



PCB Rules and Constraints Editor


CONFIGURING THE DESIGN RULES CHECKER

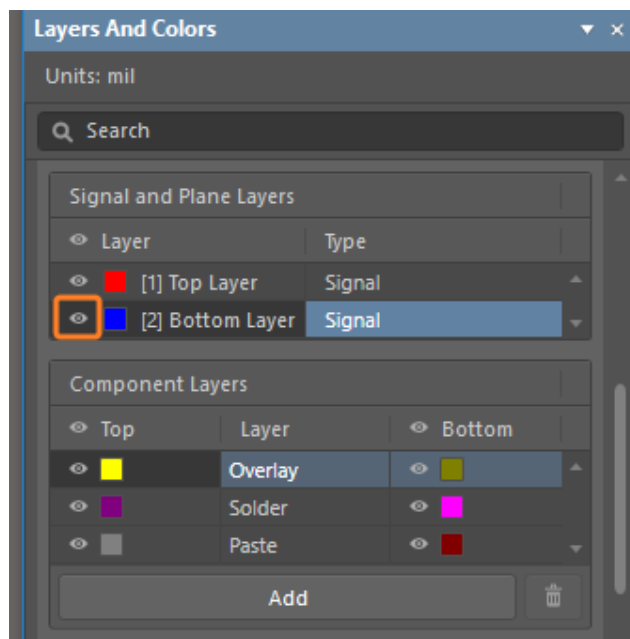
1. Open the Design Rule Checker dialog by selecting **Tools » Design Rule Check**.
2. Select **Rules To Check » Routing**.
3. Right-click in the **Rules** section and select **Online DRC - All On**.
4. Right-click in the **Rules** section and select **Batch DRC - All On**.



The Design Rule Checker dialog

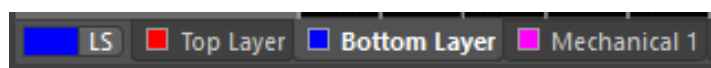
INTERACTIVELY ROUTING THE BOARD

1. Confirm that the grid is appropriate for routing. To change the grid, press **Ctrl+G** to open the **Cartesian Grid Editor**, type the value **0.125mm** into the **Step X** field, and click **OK** to close the dialog.
2. 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, press the **L** shortcut to open the **Layers and Colors** dialog, and enable the **Bottom Layer** by clicking  next to the layer name.



Layers and Colors Panel

3. Click on the Bottom Layer tab at the bottom of the workspace to make it the current, or active layer, for routing.



Bottom Layer Tab

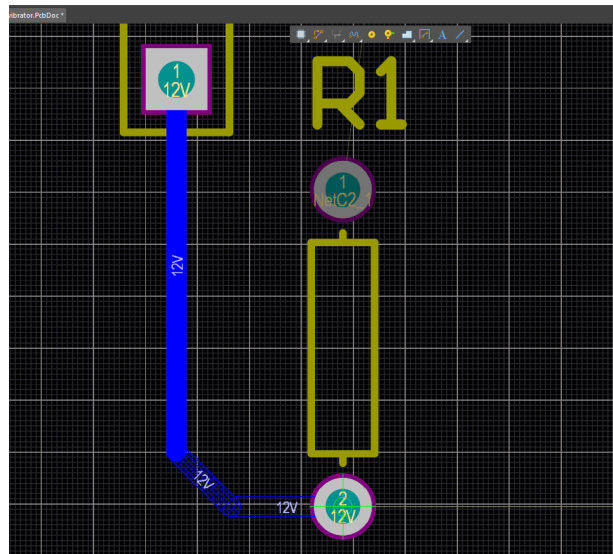
4. Select **Place » Interactive Routing** from the menus (shortcut: P, T), or click the **Interactive Routing** button on the **Active Bar** shown below. The cursor will change to a crosshair, indicating you are in interactive routing mode.



Interactive Route on Active Bar

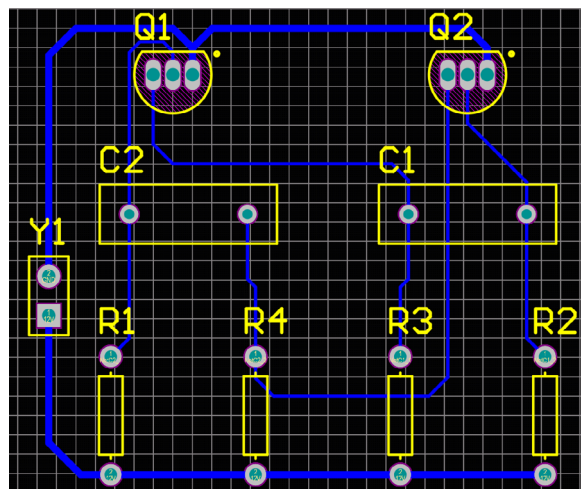
GETTING STARTED WITH PCB DESIGN

8. 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 as illustrated here:



Auto-Complete Route

9. Continue to route all the connections on the board as shown below.



Single Layer Routed PCB

TRACE MODIFICATION AND REROUTING

Routing can use both the top and bottom layers, because the PCB was originally defined as double-sided.

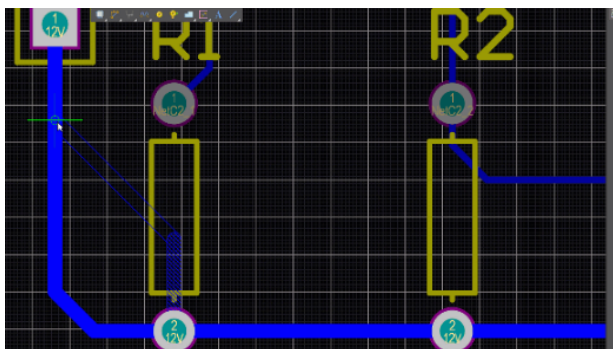
1. First, delete all routing by selecting **Route » Un-Route » All** from the menus.
2. Now, route interactively as before. To switch routing layers press the ***** key on the numeric keypad to toggle between the layers while placing tracks, or use the **Ctrl+Shift+Roll** keys/mouse to move back and forth through the available signal layers as you route.

The PCB editor will automatically insert a via (in accordance with the Routing Via design rule) as necessary when layers are changed.

GETTING STARTED WITH PCB DESIGN

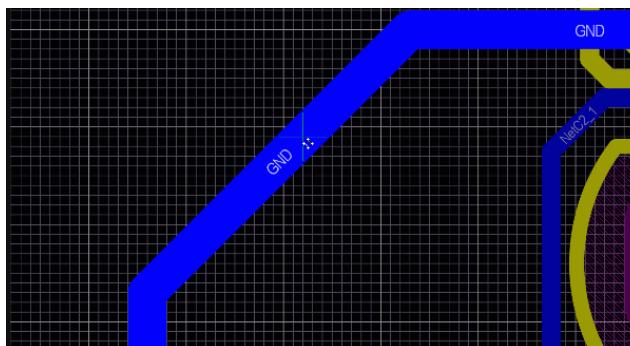
3. Next, modify an existing route. There are two approaches: reroute, or rearrange. When rerouting, there is no need to un-route a connection to redefine its path. Simply initiate interactive routing and start rerouting; the Loop Removal feature will automatically remove any redundant track segments.

The Loop Removal feature will automatically remove any redundant track segments, including vias. The PCB editor also includes a net analyzer that automatically analyses the route path as work progresses and removes redundant segments. Routing can start and end on the new route path at any point, swapping layers as needed.



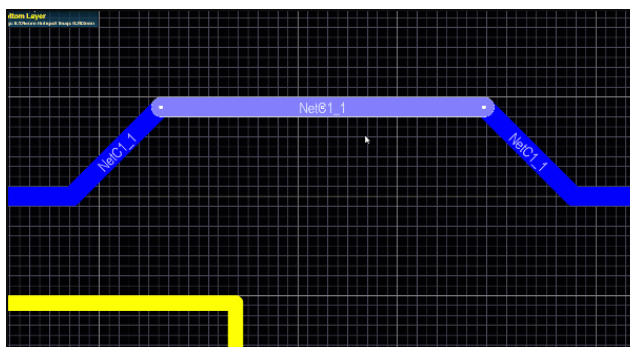
Re-Routing Loop Removal

4. To rearrange an existing route drag the segment by clicking once to select the segment, position the mouse over the segment to display the quad-arrow cursor, then click and hold to start dragging that segment.



Dragging Existing Route

5. Note that the cursor changes if you position it over the center track vertex - this is a different mode, used to break a single segment into 3 segments.

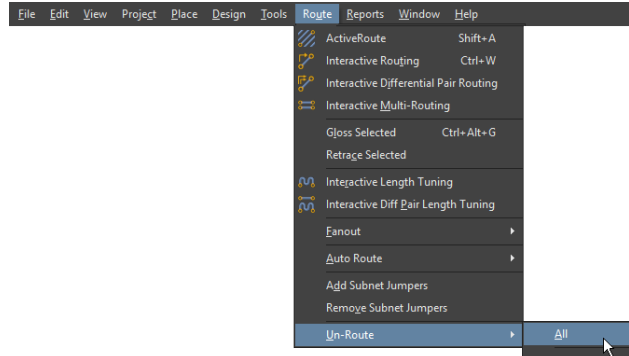


Segmenting Existing Route

AUTOMATICALLY ROUTING THE BOARD

To review the ease of autorouting, complete the following steps:

1. Un-route the board by selecting **Route » Un-Route » All** from the menus (shortcut: U, U, A).



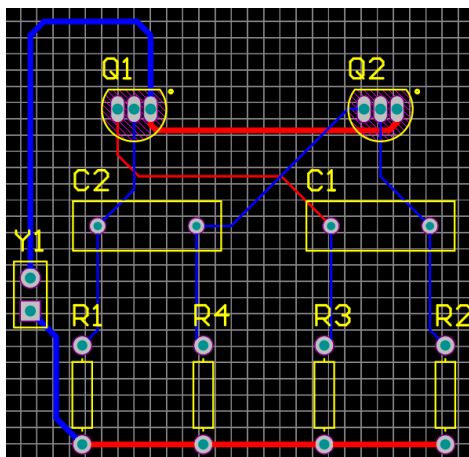
Un-Route PCB

2. Click the **Autoroute Board** icon on the **Active Bar** shown below. The **Situs Routing Strategies** dialog is split into two regions: the top region of the dialog displays the **Routing Setup Report** (remember to always check for warnings/errors that are shown in red), and the lower half of the dialog shows the available **Routing Strategies** (the selected one will be highlighted). For this board, it should default to the **Default Multi Layer Board** strategy.



Auto-Route Board icon on the Active Bar

3. Click on **Route All** in the **Situs Routing Strategies** dialog. The **Messages** panel displays the process of the autorouting.



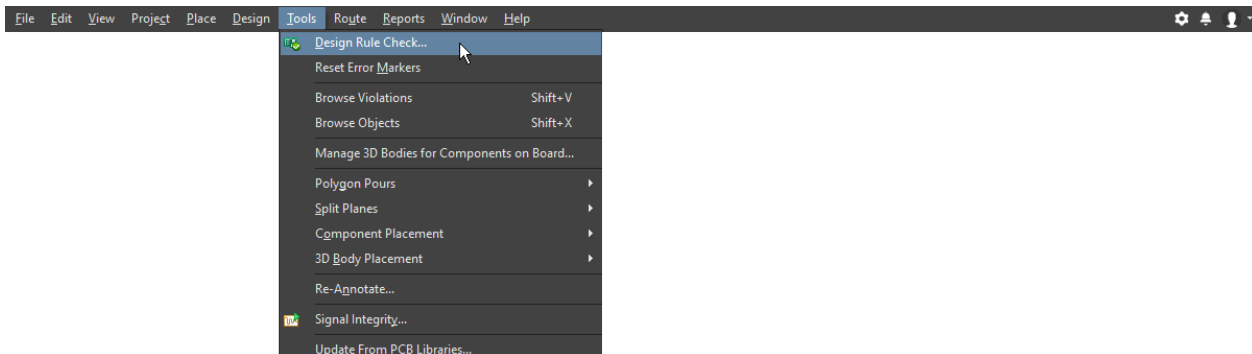
Auto-Routed PCB

4. Do not save the auto-routed board. The tutorial will continue with the manually routed board. Press **Ctrl-Z** twice and see that the auto-routing and **Un-Route All** command can be reversed.

The auto-router will route on both the top and bottom layers, red tracks on the top layer, blue on the bottom layer. The layers that are used by the auto-router are specified in the Routing Layers design rule, which defaults to top and bottom layers. Also notice the two power net tracks running from the connector are wider, as specified by the second **Width** design rule you set up.

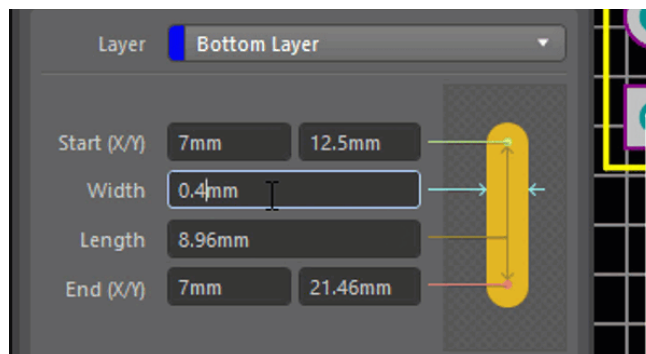
PCB DESIGN RULE VERIFICATION

1. Validate the PCB design using **Tools » Design Rule Check...** If there are no errors, no messages will be displayed in the **Messages** panel and the panel can be dismissed. The **Design Rule Verification Report** is created and appears a tab in the workspace.



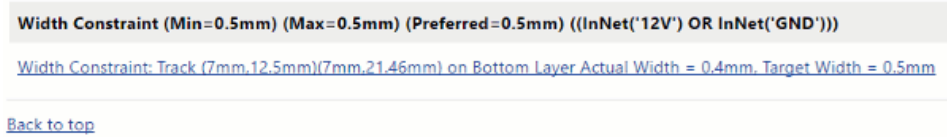
Design Rule Check

2. The design rules for the **GND** net are set such that they are expected to be no more or no less than 0.50mm. Create an intentional DRC error as shown below by clicking a trace segment on the **GND** net, going to the **Properties** panel and changing the **Width** property to 0.4mm.



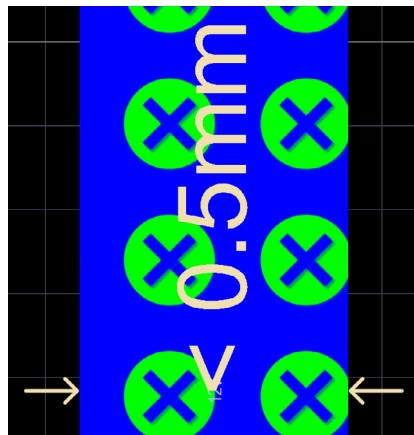
Change Width in Properties Panel

3. Rerun the **Design Rule Check**.
4. The rule checker should display an error in the **Messages** window. Dismiss the window and review the **Design Rule Verification Report**. The top of the report is a summary of any violations. The specific DRC violations are listed below the summary as hyperlinked text. Clicking on the link will bring the area of concern into view in the PCB editor window. Click on the one at the bottom of the report similar to the one shown below:



Clickable Error Messages

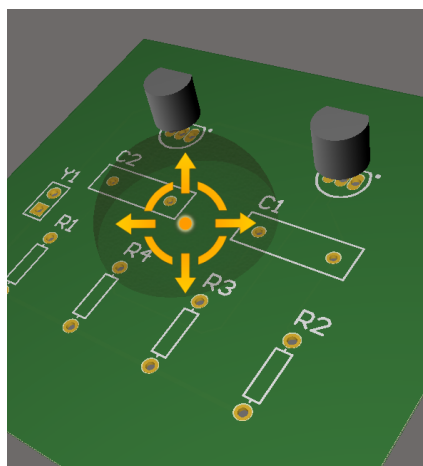
5. The error will be highlighted in a zoomed view similar to the below image. Correct the trace **Width** in the **Properties** panel or by entering `ctrl+z` to undo your width change. Save the file, and assure there are no DRC errors by rechecking again.



Route Marked with Violations

VIEWING THE PCB IN 3D

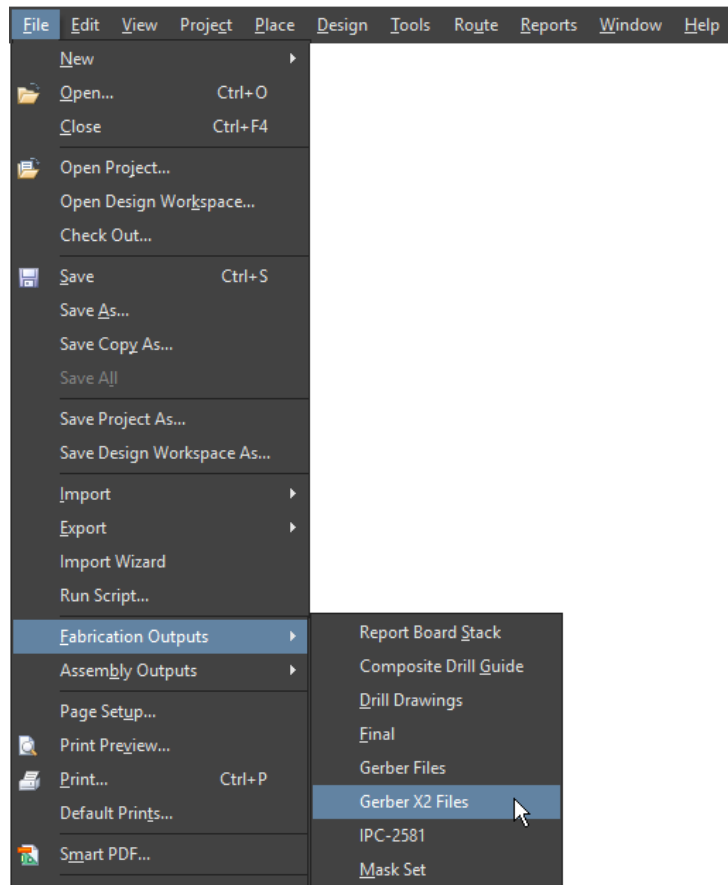
1. Switch to the 3D view by selecting **View » 3D Layout Mode** (shortcut: 3), or select a 3D view configuration from the list on the PCB Standard toolbar. The board will display as a 3 dimensional object.
2. Zoom, using **Ctrl + Right-drag** mouse, or **Ctrl + Roll mouse-wheel**, or the **PageUp / PageDown** keys.
3. Pan by using a **Right-drag** mouse, or the standard Windows mouse-wheel controls.
4. Rotate the board using the **Shift + Right-drag** mouse. Note that a directional sphere appears at the current cursor position, as shown in the figure below. Rotational movement of the model is made about the center of the sphere (position the cursor before pressing **Shift** to position the sphere) using the following controls. Move the mouse around to highlight and select each one:
 5. **Right-drag** sphere when the **Center Dot** is highlighted - rotate in any direction.
 6. **Right-drag** sphere when the **Horizontal Arrow** is highlighted - rotate the view about the **Y-axis**.
 7. **Right-drag** sphere when the **Vertical Arrow** is highlighted - rotate the view about the **X-axis**.
 8. **Right-drag** sphere when the **Circle Segment** is highlighted - rotate the view about the **Z-axis**.



3D Navigation of PCB Layout

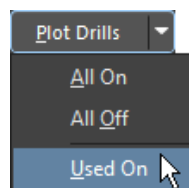
GENERATING FABRICATION OUTPUT FILES

1. Create **Fabrication Output** files by selecting **File » Fabrication Outputs » Gerber X2 Files**.



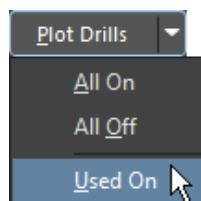
Generating Gerber X2 Files

2. Select **Used On** from the **Plot Layers** dropdown.



Plot Layers

3. Select **Used On** from the **Plot Drills** dropdown.

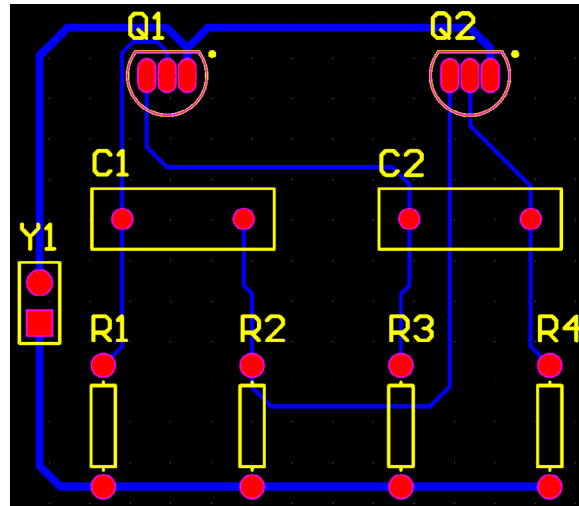


Plot Drills

4. Click **OK**.

GETTING STARTED WITH PCB DESIGN

The **Gerber X2** file will be immediately loaded into the **Altium Designer** viewing tool, **CAMtastic**. Alternatively, batch output files can be rapidly generated, so that the PCB can be fabricated with **Output Job Files**. This dedicated document is a single environment for establishing all settings related to all varieties of output types including **Gerber, NC drill, assembly drawings, pick and place** and more. The file can also be configured for independent or combined project outputs for design variants.



Gerber X2 File in CAMtastic

CONCLUSION

You made it to the end of the tutorial! Thank you for your attention. The tutorial provided just a quick overview into the design process with Altium Designer. Check out other **Getting Started** guides for insight into using robust features like Draftsman® and ActiveRoute®.

Remember to visit [Altium.com/documentation](https://www.altium.com/documentation) to find other useful documentation, and read the article **Exploring Altium Designer** for an explanation of the interface, information on how to use panels, and guidelines for managing design documents. If you want a more in-depth training experience, visit our **Events** page.

ABOUT ALTIUM

Altium LLC (ASX: ALU) is a multinational software corporation headquartered in San Diego, California, that focuses on electronics design systems for 3D PCB design and embedded system development. Altium products are found everywhere from world leading electronic design teams to the grassroots electronic design community.

With a unique range of technologies Altium helps organisations and design communities to innovate, collaborate and create connected products while remaining on-time and on-budget. Products provided are ACTIVEBOM®, ActiveRoute®, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, the TASKING® range of embedded software compilers and Upverter™.

Founded in 1985, Altium has offices worldwide, with US locations in San Diego, Boston and New York City, European locations in Karlsruhe, Amersfoort, Kiev, Munich, Markelo and Zug and Asia Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit www.altium.com. You can also follow and engage with Altium via [Facebook](#), [Twitter](#), [LinkedIn](#) and [YouTube](#).