

Tutorial - Getting Started with PCB Design

Contents

- [Creating a New PCB Project](#)
- [Creating a New Schematic Sheet](#)
- [Setting the Schematic Document Options](#)
- [Drawing the Schematic](#)
 - [Locating the Component and Loading the Libraries](#)
 - [Placing the Components on Your Schematic](#)
- [Wiring up the Circuit](#)
 - [Nets and Net Labels](#)
- [Setting Up Project Options](#)
- [Checking the Electrical Properties of Your Schematic](#)
 - [Setting up the Error Reporting](#)
 - [Setting Up the Connection Matrix](#)
 - [Setting Up the Comparator](#)
- [Compiling the Project to Check for Errors](#)
- [Creating a New PCB](#)
- [Transferring the Design](#)
- [Ready to Start the PCB Design Process](#)
- [Setting Up the PCB Workspace](#)
 - [PCB Workspace Grids](#)
 - [Component Positioning and Placement options](#)
- [Defining the Layer Stack and Other Non-electrical Layers](#)
 - [Physical Layers and the Layer Stack Manager](#)
 - [Configuring the Display of Layers](#)
- [Setting Up the Design Rules](#)
- [Positioning the Components on the PCB](#)
 - [Changing a Footprint](#)
- [Interactively Routing the Board](#)
 - [Tips for Routing](#)
- [Automatically Routing the Board](#)
- [Verifying Your Board Design](#)
- [Viewing Your Board in 3 Dimensions](#)
- [Output Documentation](#)

- [Generating Gerber Files](#)
- [Creating a Bill of Materials](#)
- [Further Explorations](#)
- [See Also](#)

Altium



Welcome to the world of Altium Designer - a complete electronic product development environment. This tutorial will get you started with creating a PCB project based on an astable multivibrator design. If you are new to Altium Designer then you might like read the article [The Altium Designer Environment](#) for an explanation of the interface, information on how to use panels, and managing design documents.

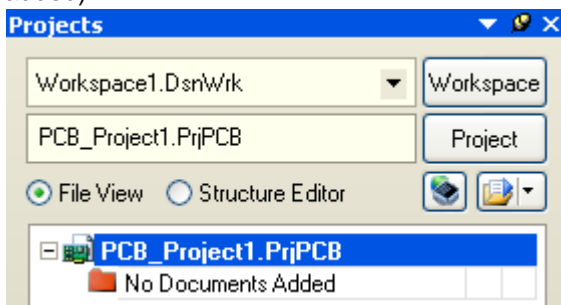
Creating a New PCB Project

A project in Altium Designer consists of links to all documents and setups related to a design. A project file, eg. xxx.PrjPCB, is an ASCII text file that lists which documents are in the project and related output setups, eg. for printing and CAM. Documents that are not associated with a project are called 'free documents'. Links to schematic sheets and a target output, eg. PCB, FPGA, embedded (VHDL) or library package, are added to a project. Once the project is compiled, design verification, synchronization and comparison can take place. Any changes to the original schematics or PCB, for example, are updated in the project when compiled.

The process of creating a new project is the same for all project types. We will use the PCB project as an example. We will create the project file first and then create the blank schematic sheet to add the new empty project. Later in this tutorial we will create a blank PCB and add it to the project as well.

To start the tutorial, create a new PCB project:

1. Select **File»New»Project»PCB Project** from the menus, or click on Blank Project (PCB) in the **New** section of the Files panel. If this panel is not displayed, select **Files** from the **System** button at the bottom right of the main design window.
2. The Projects panel will open, displaying the new project file, PCB_Project1.PrjPCB (with no documents added).



3. Rename the new project file (with a .PrjPCB extension) by selecting **File»Save Project As**. Navigate to a

location where you would like to store the project on your hard disk, type the name Multivibrator.PrjPCB in the **File Name** field and click **Save**.

Creating a New Schematic Sheet

Next we will add a new schematic sheet to the project. It is on this schematic we will capture the astable multivibrator circuit.

Create a new schematic sheet by completing the following steps:

1. **Right-click** on the project file in the Projects panel and select **Add New to Project»Schematic**. A blank schematic sheet named Sheet1.SchDoc will open in the design window and an icon for this schematic will appear linked to the project in the Projects panel, under the Source Documents folder icon.
2. Save the new schematic (with a .SchDoc extension) by selecting **File»Save As**. Navigate to a location where you would like to store the schematic on your hard disk, type the name Multivibrator.SchDoc in the **File Name** field and click on Save. Note that project files stored in the same folder as the project file itself (or in a child/grandchild folder) are linked to the project using relative referencing, whereas files stored in a different location are linked using absolute referencing.
3. Since you have added a schematic to the project, the project file has changed too. **Right-click** on the project filename in the Projects panel, and select **Save** to save the project.

When the blank schematic sheet opens you will notice that the workspace changes. The main toolbar includes a range of new buttons, new toolbars are visible, the menu bar includes new items and the Sheet panel is displayed. You are now in the Schematic Editor. You can customize many aspects of the workspace. For example, you can reposition the panels and toolbars or customize the menu and toolbar commands.

Setting the Schematic Document Options

Tip:

In Altium Designer, 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 submenus, such as the **DeSelect** menu (in the **Edit** menu), can be called directly. For example, to activate the **Edit»DeSelect»All on Current Document** command, you need only press the **X** key (to call up the **DeSelect** menu directly) followed by the **S** key.

The first thing to do before you start drawing your circuit is to set up the appropriate document options. Complete the following steps.

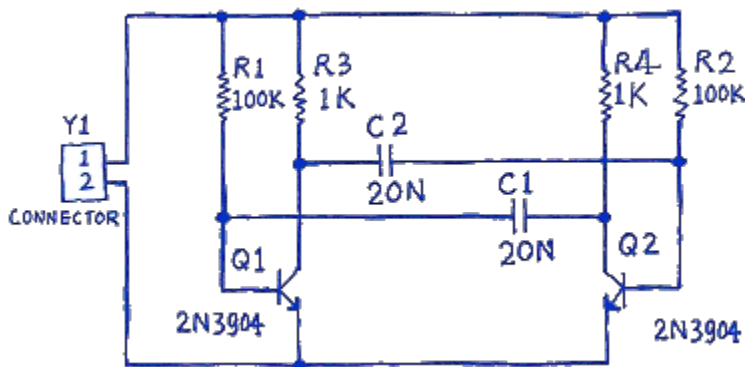
1. From the menus, choose **Design»Document Options** to open the *Document Options* dialog.
2. For this tutorial, the only change we need to make here is to set the sheet size to A4, this is done in the Standard Styles field of the Sheet Options tab of the dialog.
3. Click OK to close the dialog and update the sheet size.
4. To make the document fill the viewing area, select **View»Fit Document**.
5. Save the schematic sheet by selecting **File»Save** (shortcut: **F, S**).

Next we will set the general schematic preferences.

1. Select **Tools»Schematic Preferences** (shortcut: **T, P**) to open the schematic area of the *Preferences* dialog. This dialog allows you to set global preferences that will apply to all schematic sheets you work on.
2. Open the **Schematic - Default Primitives** page of the dialog and enable the Permanent option (on the right hand side of the dialog). Click OK to close the dialog.

Note that Altium Designer has multilevel Undo, allowing you to undo many previous actions. The Undo stack size is user-configurable and limited only by the available memory on your computer, configure it in the **Schematic - Graphical Editing** page of the *Preferences* dialog.

Drawing the Schematic



Circuit for the multivibrator.

You are now ready to begin capturing (drawing) the schematic. For this tutorial, we will use the circuit shown in the figure above. This circuit uses two 2N3904 transistors configured as a self-running astable multivibrator.

Locating the Component and Loading the Libraries

To manage the thousands of schematic symbols included with Altium Designer, the Schematic Editor includes powerful library searching capabilities. Although the components we require are in the default installed libraries, it is useful to know how to search through all libraries to find components. Work through the following steps to locate and add the libraries you will need for the tutorial circuit.

First we will search for the transistors, both of which are type 2N3904.

1. If it is not visible, display the Libraries panel. The easiest way to do that is to click the **System** button down the bottom right of the application, then select **Libraries** from the menu that appears. Refer to the [Working with Panels](#) article to learn more about configuring and controlling panels.
2. Press the **Search** button in the Libraries panel (or select **Tools»Find Component**) to open the *Libraries Search* dialog.
3. Ensure that the dialog options are set as follows:
 - For the first **Filter** row, the **Field** is set to **Name**, the **Operator** set to **contains**, and the **Value** is 3904.
 - The Scope is set to **Search in Components**, and **Libraries on path**.
 - The **Path** is set to point to the installed Altium libraries, which will be something like C:\Program Files\Altium Designer Summer 09\Library.

Libraries Search [?] [X]

Filters [Add Row](#) [Remove Row](#)

	Field	Operator	Value
1.	Name	contains	3904
2.		equals	
3.		equals	

[>> Advanced](#)

Scope

Search in: Components

☐ Available libraries
☒ Libraries on path
☐ Refine last search

Path

Path: ESIGNER SUMMER 09\Library\

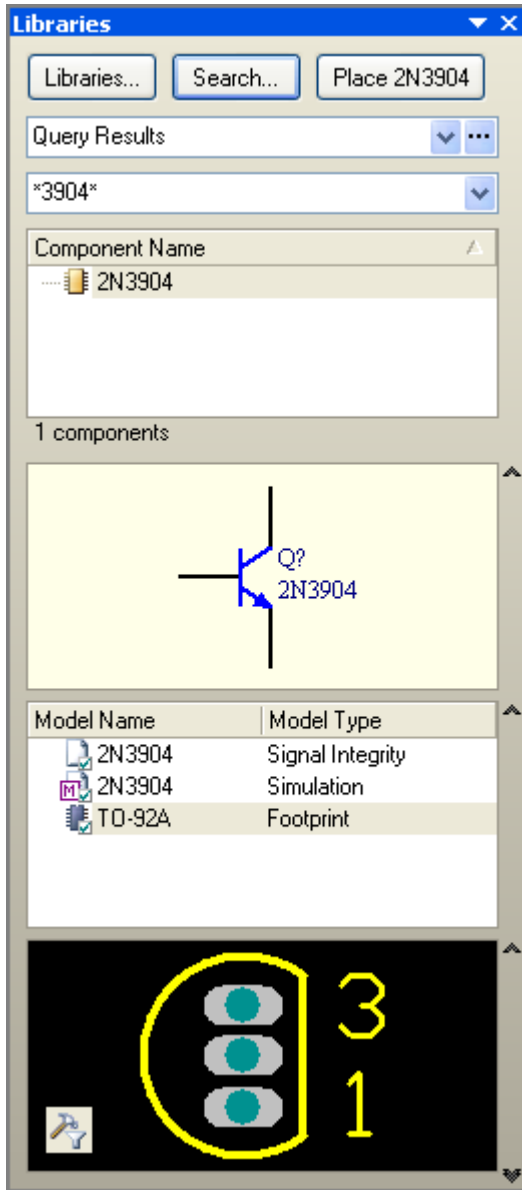
☒ Include Subdirectories

File Mask: *.*

Search installed or all available libraries for components.

- Click the **Search** button to begin the search. The **Query Results** are displayed in the Libraries panel as the search takes place.
- Click on the component name **2N3904** found in the Miscellaneous Devices.IntLib library to select it. This library has symbols for the available simulation-ready BJT transistors.
- If you choose a component that is in a library that is not currently installed, you will be asked to **Confirm the installation** of that library before you can place a component from it. Since the Miscellaneous Devices library is already installed, the component is ready to place. Added libraries appear in the drop down list at the top of

the Libraries panel, as you select a library in the list the components in that library are listed below. Use the component **Filter** in the panel to quickly locate a component within a library.



Search results for components with the string 3904 in their name.

Placing the Components on Your Schematic

The first components we will place on the schematic are the two transistors, Q1 and Q2. Refer to the rough schematic sketch shown above for the general layout of the circuit.

1. Select **View»Fit Document** (shortcut: **V, D**) to ensure your schematic sheet takes up the full window.
2. Display the Libraries panel (by clicking on its tab on the right of the workspace, if it is pop-out mode).
3. Select the Miscellaneous Devices.IntLib library from the Libraries drop-down list at the top of the Libraries panel to make it the active library.
4. Use the filter to quickly locate the component you need. The default is for the filter to be set to the wildcard (*), listing all components found in the library. Type *3904* in the filter field - a list of components which have the text "3904" as part of their Component Name field will be displayed.

- Click on the 2N3904 entry in the list to select it, then click the **Place** button. Alternatively, just double-click on the component name. The cursor will change to a cross hair and you will have an outlined version of the transistor "floating" on your cursor. You are now in part placement mode. If you move the cursor around, the transistor outline will move with it. Do NOT place the transistor yet.
- Before placing the part on the schematic we will edit its properties. While the transistor is still floating on the cursor, press the TAB key to open the *Component Properties* dialog. We will now set up the dialog options to appear as below.

Component Properties

Properties

Designator: ☒ Visible ☐ Locked

Comment: ☒ Visible

Description:

Unique Id:

Type:

Library Link

Design Item ID:

☒ Library Name:

☒ Table Name:

Sub-Design Links

Sub-Project:

Configuration:

Graphical

Location X: Y:

Orientation: ☐ Locked

Mode: ☒ Lock Pins ☐ Mirrored

☐ Show All Pins On Sheet (Even if Hidden)

☐ Local Colors

Parameters for Q? - 2N3904

Visible	Name	Value	Type
<input type="checkbox"/>	Code_JEDEC	T0-92A	STRING
<input type="checkbox"/>	ComponentLink1Descripti	Manufacturer Link	STRING
<input type="checkbox"/>	ComponentLink1URL	http://www.fairchildsemi.com/	STRING
<input type="checkbox"/>	ComponentLink2Descripti	Datasheet	STRING
<input type="checkbox"/>	ComponentLink2URL	http://www.fairchildsemi.com/ds/2	STRING
<input type="checkbox"/>	DatasheetDocument	1997	STRING
<input type="checkbox"/>	LatestRevisionDate	15-Jan-2003	STRING
<input type="checkbox"/>	LatestRevisionNote	PCB Footprint 'BCY'-W3/D4.7 repl	STRING
<input type="checkbox"/>	PackageDocument	Sep-1998	STRING
<input type="checkbox"/>	PackageReference	T0-92A	STRING
<input type="checkbox"/>	Published	8-Jun-2000	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING

Models for Q? - 2N3904

Name	Type	Description
2N3904	Simulation	NPN
2N3904	Signal Integrity	
T0-92A	Footprint	T0, Flat Index; 3 In-Line, Axial Leads; Body Dia. 4

- In the **Properties** section of the dialog, type in the **Designator** Q1.
- Confirm that the Footprint (specified in the Models region of the dialog) is set to **BCY-W3/E4**. Since this is an integrated library each component has a symbol and at least 1 footprint, as well as simulation models for some of the components.
- Leave all other fields at their default values, and click OK to close the dialog.

You are now ready to place the part. When you are in any editing or placement mode (a cross hair cursor is active), moving the cursor to the edge of the document window will automatically pan the document.

If you accidentally pan too far while you are wiring up your circuit, press **V, F (View»Fit All Objects)** to redraw the schematic window, showing all placed objects. This can be done even when you are in the middle of placing an object

Tip:

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.

1. Move the cursor (with the transistor symbol attached) to position the transistor a little left of the middle of the sheet. Once you are happy with the transistor's position, click or press ENTER to place the transistor onto the schematic.
2. Move the cursor and you will find that a copy of the transistor has been placed on the schematic sheet, but you are still in part placement mode with the part outline floating on the cursor. This feature of Altium Designer allows you to place multiple parts of the same type. So let's now place the second transistor. This transistor is the same as the previous one, so there is no need to edit its attributes before we place it. Altium Designer will automatically increment a component's designator when you place a series of parts. In this case, the next transistor we place will automatically be designated Q2.
3. If you refer to the rough schematic diagram shown before, you will 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. This flips the component horizontally (along the X axis).
4. Move the cursor to position the part to the right of Q1. To position the component more accurately, press the PAGE UP key twice to zoom in two steps. You should now be able to see the grid lines.

Tip:

To edit the attributes of an object placed on the schematic, double-click the object to open its *Component Properties* dialog.

1. Once you have positioned the part, click or press ENTER to place Q2. Once again a copy of the transistor you are "holding" will be placed on the schematic, and the next transistor will be floating on the cursor ready to be placed.
2. Since we have now placed all the transistors, we will exit part placement mode by clicking the **Right Mouse Button** or pressing the **ESC** key. The cursor will revert back to a standard arrow.

Next we will place the four resistors.

Tip:

Note, components being simulated may have a number of simulation properties that can be defined (eg, a resistor has 1, a BJT has 5, and a MOSFET has 13) - these properties are defined by using Parameters. If you wanted to simulate this circuit then the resistor value must be defined as a Parameter, whose name is **Value** and whose value is the resistance.

If the circuit being captured is for both simulation and PCB layout, rather than enter the value twice (in the parameter called **Value** and then again in the **Comment** field), Altium Designer supports 'indirection', a feature that maps any parameter's string into the **Comment** field. If you click to display the **Comment** field dropdown list you will see that the software has automatically built a list of all current parameters, in case you want to map the value of one of them into the **Comment** field.

1. In the Libraries panel, make sure the Miscellaneous Devices.IntLib library is active.
2. Set the filter by typing `res1` in the filter field below the Library name.
3. Click on **Res1** in the components list to select it, then click the **Place** button. You will now have a resistor symbol floating on the cursor.
4. Press the **TAB** key to open the *Component Properties* dialog to edit the resistor's attributes. In the **Properties** section of the dialog, set the value for the first component designator by typing `R1` in the **Designator** field.
5. Make sure that footprint name **AXIAL-0.3** is the current footprint in the **Models** list.
6. The contents of **Comment** field of the schematic component maps to the **Comment** field of the PCB component, typically you would enter the value or the resistor here. Enter a value of `100k` into the **Comment** field for R1.
7. Since you will not be simulating ensure that the **Visible** option for the **Value** parameter is disabled, and that the **Comment** field has the correct value (`100K` in this case).
8. Press the **SPACEBAR** to rotate the resistor by 90° so it is in the correct orientation.
9. Position the resistor above the base of Q1 (refer to the schematic diagram shown earlier) and click the **Left Mouse Button** or press **ENTER** to place the part. Don't worry about making the resistor connect to the transistor just yet. We will wire up all the parts later.
10. Next place the other 100k resistor R2 above the base of Q2. The designator will automatically increment when you place the second resistor.
11. The remaining two resistors, R3 and R4, have a value of 1k, so press the **TAB** key to open the *Component Properties* dialog, enter `1k` into the **Comment**, and confirm that the **Visible** option for the **Value** parameter is disabled. Click **OK** to close the dialog.
12. Position and place R3 and R4 as shown in the rough schematic diagram. **Right-click** or press **ESC** to exit part placement mode.

Now place the two capacitors.

Tip:

To reposition any object, place the cursor directly over the object, click-and-hold the left mouse button, drag the object to a new position and then release the mouse button.

Tip:

As with the resistor, if you wanted to simulate this circuit you would need a Value parameter with the value of 20n, in this case you would define the capacitance in the Value parameter and then use the indirection feature to map the contents of the value parameter into the Comment field. Since you will not be simulating ensure that the Visible option for the Value parameter is disabled.

1. The capacitor part is also in the Miscellaneous Devices.IntLib library, which should already be selected in the Libraries panel.
2. Type `cap` in the component's filter field in the Libraries panel.
3. Click on CAP in the components list to select it, then click the **Place** button. You will now have a capacitor symbol floating on the cursor.
4. Press the **TAB** key to edit the capacitor's attributes. In the *Component Properties* dialog set the **Designator** to C1, the **Comment** to 20n, disable the **Visible** option for the **Value** parameter, and check the PCB footprint model **RAD-0.3** is selected in the **Models** list. Click **OK**.
5. Position and place the two capacitors in the same way that you placed the previous parts.
6. Right-click or press ESC to exit placement mode.

The last component to be placed is the connector, located in Miscellaneous Connectors.IntLib.

1. Select Miscellaneous Connectors.IntLib from the Libraries list in the Libraries panel. The connector we want is a two-pin socket, so type `*2` into the Libraries panel filter field.
2. Select **Header 2** from the parts list and click the **Place** button. Press **TAB** to edit the attributes and set Designator to Y1 and check that the PCB footprint model is **HDR1X2**. No **Value** parameter is required as you would replace this component with a power source when simulating the circuit. Click **OK** to close the dialog.
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.
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**).

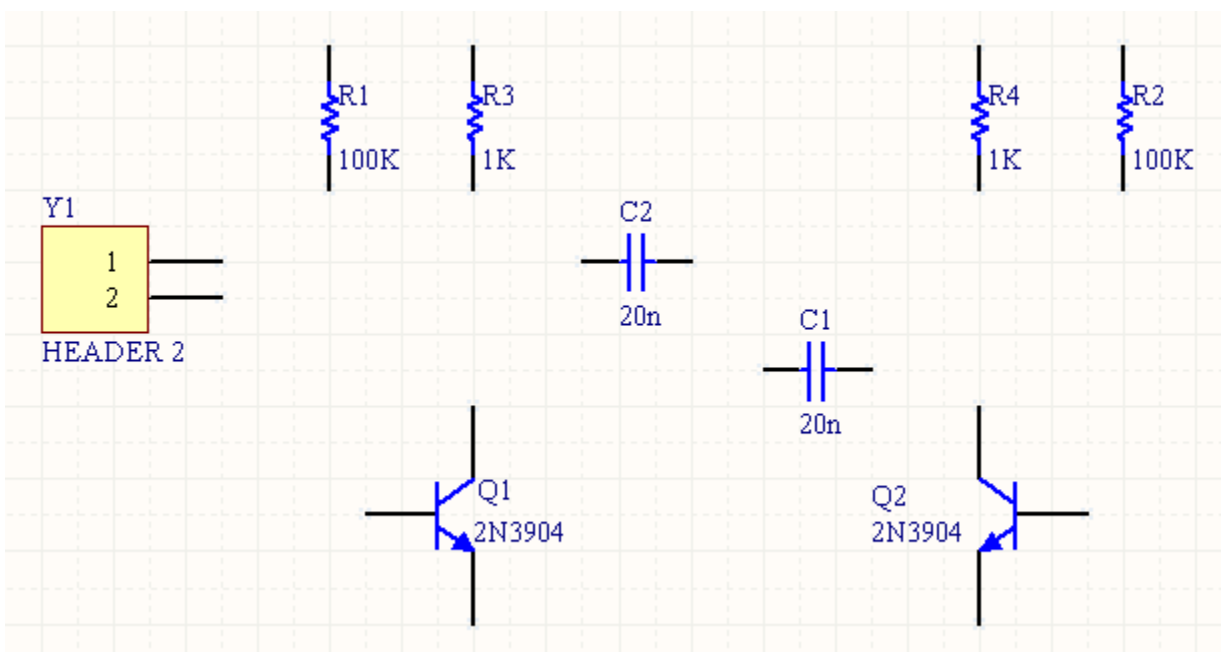
Tip:

You can re-position a group of selected schematic objects using the arrow keys. Selected objects can be 'nudged' by the current snap grid value by pressing an **arrow key** while holding down the **CTRL** key. Hold the **S** **hift** as well to move objects by 10 times the current snap grid.

The movement of selected objects are set according to the current **Snap Grid** setting. Altium Designer supports multiple pre-defined snap grids, press the **G** shortcut at any time to cycle through the current snap grid settings. The current snap grid setting is displayed on the Status bar.

The **Schematic - Grids** page of the *Preferences* dialog (**DXP»Preferences**) is used to edit and add imperial or metric snap grid settings. The **Schematic - Default Units** page of the *Preferences* dialog is used to select the type of units that will be used, select between **DXP Defaults**, **Imperial**, or **Metric**.

You have now placed all the components. Note that the components shown in the figure below are spaced so that there is plenty of room to wire to each component pin. This is important because you can not place a wire across the bottom of a pin to get to a pin beyond it. If you do, both pins will connect to the wire. If you need to move a component, click-and-hold on the body of the component, then drag the mouse to reposition it.




Circuit with all parts placed.

Wiring up the Circuit

Wiring Tips

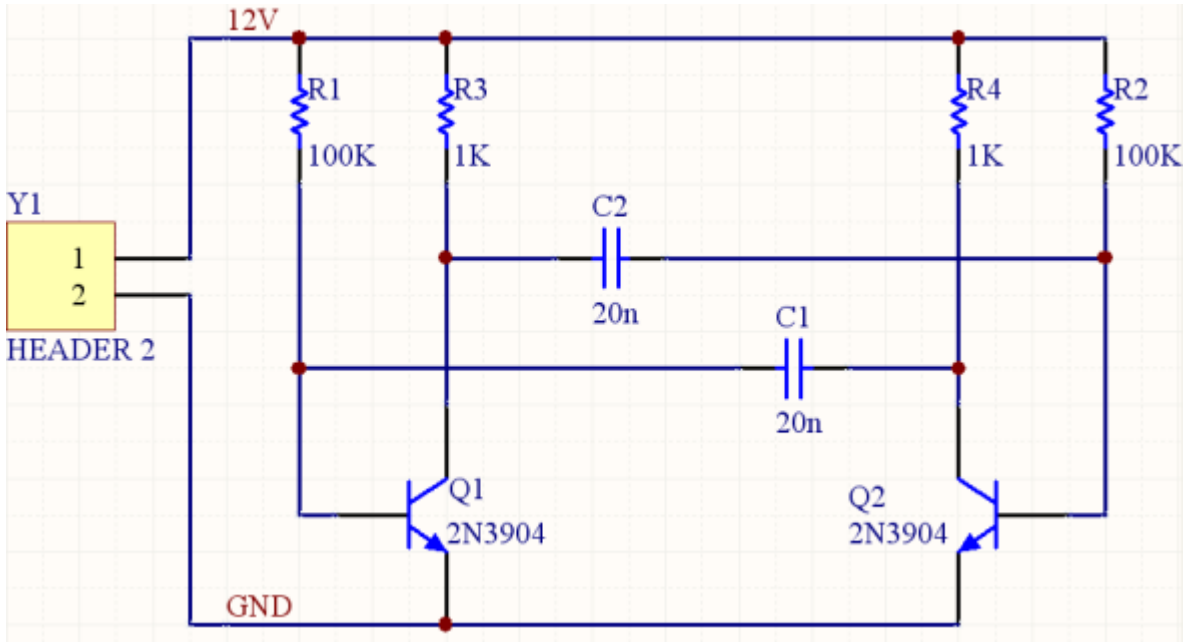
- **Left-click or press ENTER to anchor the wire at the cursor position**
- **press BACKSPACE to remove the last anchor point**
- **press SPACEBAR to toggle the direction of the corner**
- **press SHIFT+SPACEBAR to cycle through all possible corner modes. We will use the right-angle mode for this design.**
- **Right-click or press ESC to exit wire placement mode.**
- **To graphically edit the shape of a wire, Click once to select it first, then Click and hold on a segment or vertex to move it.**
- **Whenever a wire crosses the connection point of a component, or is terminated on another wire, Altium Designer will automatically create a junction.**

Wiring is the process of creating connectivity between the various components of your circuit. To wire up your schematic, refer to the rough schematic diagram and complete the following steps:

1. To make sure you have a good view of the schematic sheet, use the **PAGE UP** key to zoom in or **PAGE DOWN** to zoom out. Alternatively, hold down the **CTRL** key and roll the mouse wheel to zoom in/out, or hold **CTRL + Right Mouse** button down and drag the mouse up/down to zoom in/out.
2. Firstly wire the resistor R1 to the base of transistor Q1 in the following manner. Select  **Place»Wire** (short cut: **P, W**) from the menus or click on the **Wire** tool from the Wiring toolbar to enter the wire placement mode. The cursor will change to a crosshair.
3. Position the cursor over the bottom end of R1. When you are in the right position, a red connection marker (large asterisk) will appear at the cursor location. This indicates that the cursor is over an electrical connection point on the component.
4. Click the **Left Mouse Button** or press **ENTER** to anchor the first wire point. Move the cursor and you will see a wire extend from the cursor position back to the anchor point. The default corner
5. Position the cursor so that it is below R1 and level with the base of Q1. Click or press **ENTER** to anchor the wire at this point. The wire between the first and second anchor points will be placed.
6. 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.

7. Note that the cursor remains a cross hair, indicating that you are ready to place another wire. To exit placement mode completely and go back to the arrow cursor, you would **Right-Click** or press **ESC** again - but don't do this just now.
8. We will now wire C1 to Q1 and R1. Position the cursor over the left connection point of C1 and click or press **ENTER** to start a new wire. Move the cursor horizontally till it is directly over the wire connecting the base of Q1 to R1. A connection marker will appear. Click or press **ENTER** to place the wire segment, then right-click or press **ESC** to indicate that you have finished placing the wire. Note how the two wires are automatically connected.
9. Wire up the rest of your circuit, as shown in the figure below.



The fully wired schematic.

Tip:

A wire that crosses the end of a pin will connect to that pin, even if you delete the junction. Check that your wired circuit looks like the figure shown, before proceeding.

10. When you have finished placing all the wires, right-click or press **ESC** to exit placement mode. The cursor will revert to an arrow.
11. If you wish to move any placed components and drag any connected wires with it, hold down the **CTRL** key while moving the component, or select **Move»Drag**.

Nets and Net Labels

Each set of component pins that you have connected to each other now form what is referred to as a *net*. For example, one net includes the base of Q1, one pin of R1 and one pin of C1.

To make it easy to identify important nets in the design, you can add net labels. To place net labels on the two power nets:

1. Select **Place»Net Label** (shortcut: **P, N**). A net label will appear floating on the cursor.
2. To edit the net label before it is placed, press **TAB** key to open the *Net Label* dialog.

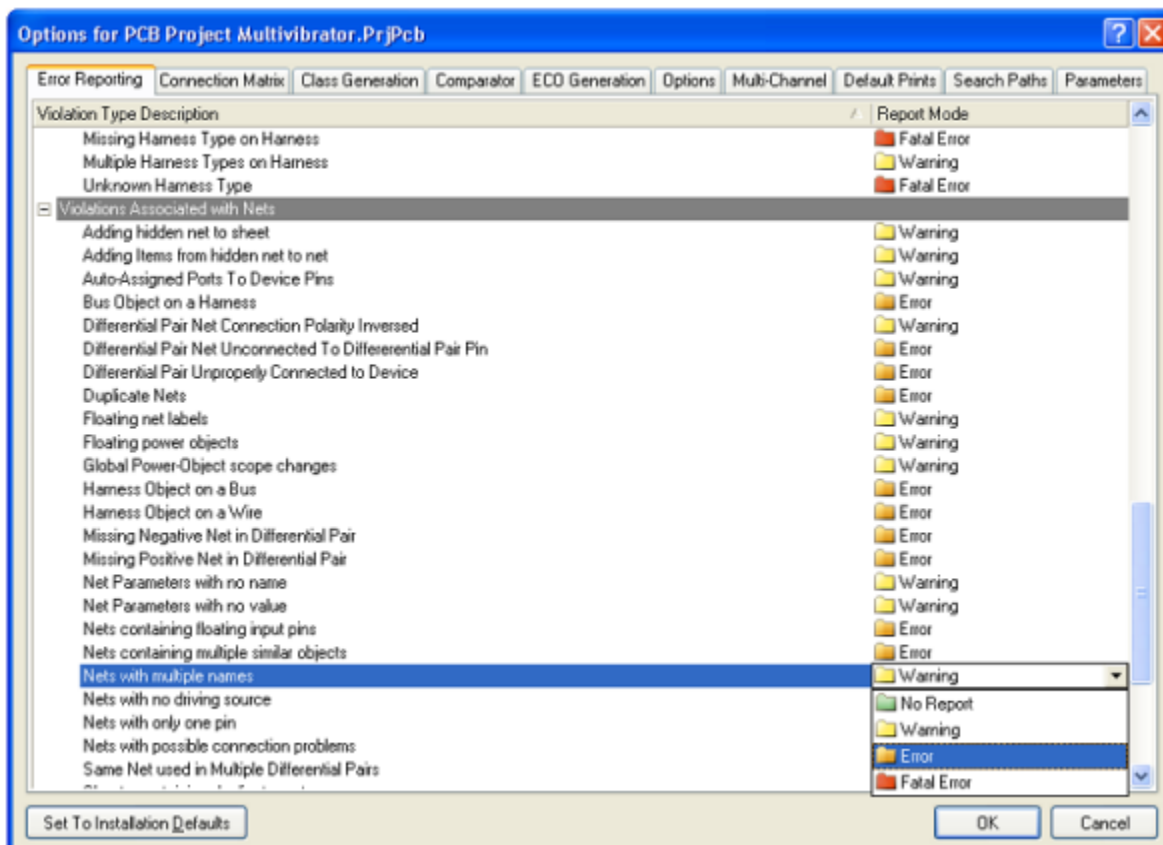
3. Type 12V in the **Net** field, then click **OK** to close the dialog.



4. Place the net label so that the bottom left corner of the net label touches the upper most wire on the schematic. The cursor will change to a red cross when the net label touches the wire. If the cross is light gray, it means you are trying to label a pin instead of a wire.
5. After placing the first net label you will still be in net label placement mode, so press the TAB key again to edit the second net label before placing it.
6. Type GND in the **Net** field, click **OK** to close the dialog and place the net label.
7. Place the net label so that the bottom left of the net label touches the lower most wire on the schematic. Right-click or press **ESC** to exit net label placement mode.
8. Select **File»Save** (shortcut: **F, S**) to save your circuit. Save the project as well.

Congratulations! You have just completed your first schematic capture using Altium Designer. Before we turn the schematic into a circuit board we need to configure the project options, and check the design for errors.

Setting Up Project Options



All project-specific settings are configured in the Options for Project dialog.

All project-specific settings are configured in the *Options for Project* dialog (**Project»Project Options**). The project options include the error checking parameters, a connectivity matrix, Class Generator, the Comparator setup, ECO generation, output paths and netlist options, Multi-Channel naming formats, Default Print setups, Search Paths, and

any project parameters you wish to specify. Altium Designer will use these settings when you compile the project.

When the project is compiled, comprehensive design and electrical rules are applied to verify the design. When all errors are resolved, the compiled schematic design can then be transferred to the target PCB document by generating a series of Engineering Change Orders (ECOs).

Underlying this process is a comparator engine that identifies every difference between the schematic design and the PCB, and generates an ECO to resolve each difference. This approach of using a comparator engine to identify differences means you not only work directly between the schematic and PCB (there is no intermediate netlist file used), it also means the same approach can be used to synchronize the schematic and PCB at any stage during the design process. The comparator engine also allows you to find differences between source and target files and update (synchronize) in both directions.

Project outputs, such as assembly, fabrication outputs and reports can be set up from the **File** and **Reports** menus. These settings are also stored in the Project file so they are always available for this project. Alternatively you can set up output options in an Output Job file (**File»New»Output Job File**). See [Documentation Outputs](#) for more information.

Checking the Electrical Properties of Your Schematic

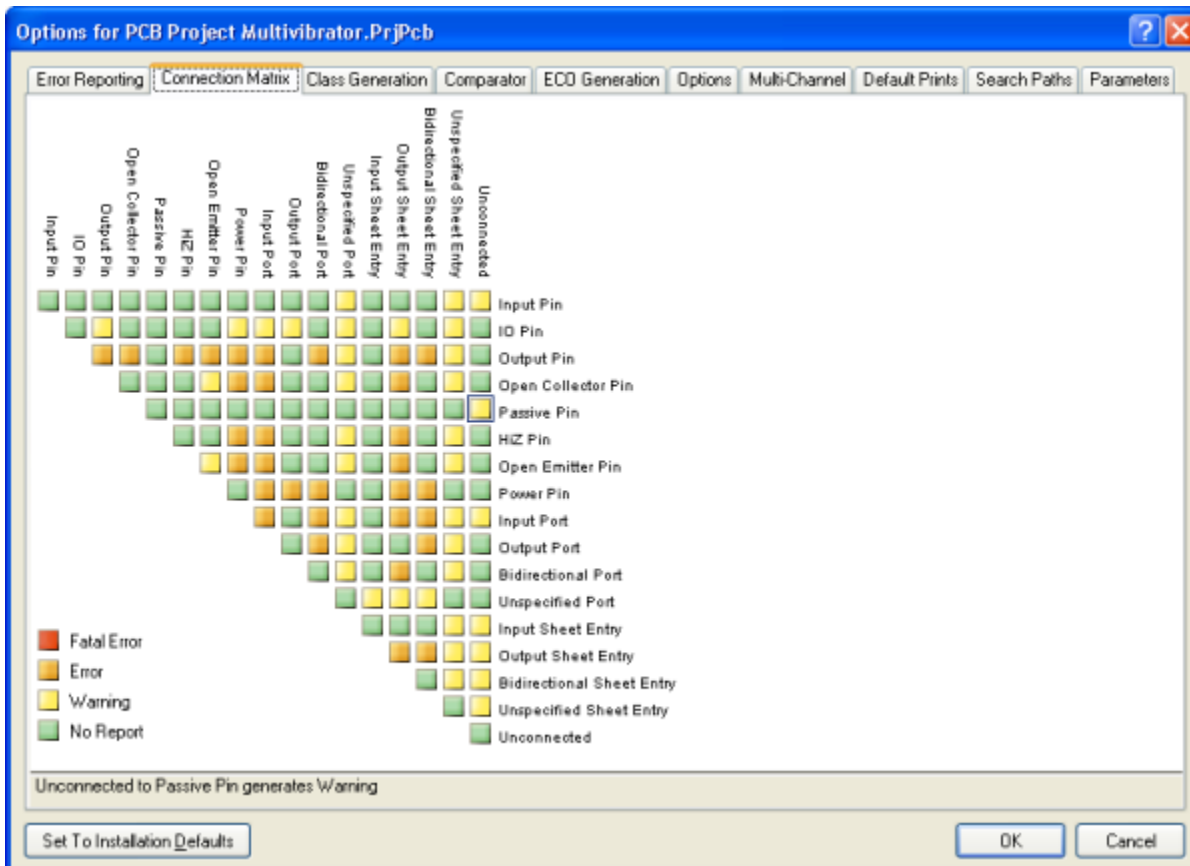
Schematic diagrams in Altium Designer are more than just simple drawings - they contain electrical connectivity information about the circuit. You can use this connectivity awareness to verify your design. When you compile a project, Altium Designer checks for errors according to the rules set up in the **Error Reporting** and **Connection Matrix** tabs of the *Options for Project* dialog. When you compile the project any violations that are detected will display in the Messages panel.

1. Select **Project»Project Options** to open the *Options for Project* dialog (shown in the figure above).
2. Set up any project-related options in this dialog. We will now make some changes to the **Error Reporting**, **Connection Matrix** and **Comparator** tabs.

Setting up the Error Reporting

The **Error Reporting** tab in the *Options for Project* dialog is used to set up design drafting checks. The **Report Mode** settings show the level of severity of a violation. If you wish to change a setting, click on a **Report Mode** next to the violation you wish to change and choose the level of severity from the drop-down list. For this tutorial we will use the default settings in this tab.

Setting Up the Connection Matrix



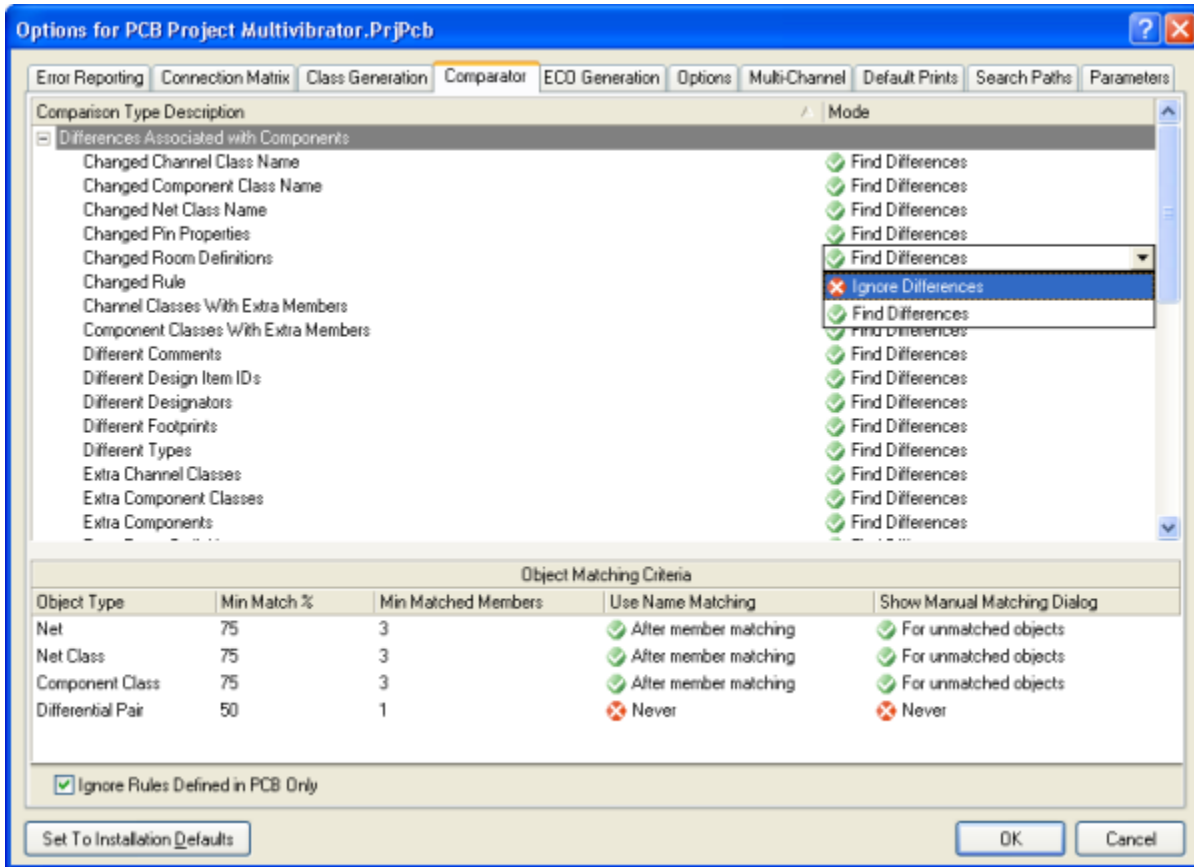
The Connection Matrix defines what electrical conditions are checked for on the schematic.

When the design is compiled a list of the pins in each net is built internally in Altium Designer's memory. The type of each pin is detected (eg: input, output, passive, etc), and then each net is checked to see if there are pin types that should not be connected to each other, for example an output pin connected to another output pin. The Connection Matrix tab of the **Options for Project** dialog is where you configure what pin types are allowed to connect to each other. For example, look down the entries on the right side of the matrix diagram and find **Output Pin**. Read across this row of the matrix till you get to the **Open Collector Pin** column. The square where they intersect is orange, indicating that an Output Pin connected to an Open Collector Pin on your schematic will generate an error condition when the project is compiled.

You can set each error type with a separate error level, eg. from no report, through to a fatal error. To make changes to the Connection Matrix:

1. To change one of the settings click the colored box, it will cycle through the 4 possible settings. Note that you can **right-click** on the dialog face to display a menu that lets you toggle all settings simultaneously, including an option to restore them all to their **Default** state.
2. Our circuit contains only Passive Pins (on resistors, capacitors and the connector) and Input Pins (on the transistors). Let's check to see if the connection matrix will detect unconnected passive pins. Look down the row labels to find **Passive Pin**. Look across the column labels to find **Unconnected**. The square where these entries intersect indicates the error condition when a *passive pin* is found to be *unconnected* in the schematic. The default setting is green, indicating that no report will be generated.
3. Click on this intersection box until it turns yellow, so that a warning will be generated for unconnected passive pins when we compile the project. We will purposely create an instance of this error to check it later in this tutorial.

Setting Up the Comparator



The Comparator tab in the *Options for Project* dialog sets which differences between files will be reported or ignored when a project is compiled. For this tutorial, we do not need to show differences between some features that refer to hierarchical schematic designs only, such as rooms.

1. Click the Comparator tab and find **Changed Room Definitions**, **Extra Room Definitions** and **Extra Component Classes** in the **Differences Associated with Components** section.
2. Select Ignore Differences from the dropdown list in the **Mode** column to the right for each of the above mentioned options. Make sure you do not accidentally ignore components when you meant to ignore component classes!

We are now ready to compile the project and check for any errors.

Compiling the Project to Check for Errors

Compiling a project checks for drafting and electrical rules errors in the design documents and details all warnings and errors in the messages panel, and gives detailed information in the Compiled Errors panel. We have already set up the rules in the **Error Checking** and **Connection Matrix** tabs of the *Options for Project* dialog, so we are ready to check the design.

1. To compile the Multivibrator project, select **Project»Compile PCB Project**.
2. When the project is compiled, all warnings and errors will displayed in the Messages panel. The panel will only appear automatically if there are errors detected, to open it manually click the **System** button down the bottom right of the workspace, and select **Messages** from the menu.
Double-click on an entry in the panel to examine an error. The compiled documents will also be detailed in

the Navigator panel, together with a flattened hierarchy, components and nets listed and a connection model that can be browsed.

3. If your circuit is drawn correctly, the Messages panel should not contain any errors. If there are errors, work through each one, checking your circuit and ensuring that all wiring and connections are correct.

We will now deliberately introduce an error into the circuit and recompile the project:

1. Click on the Multivibrator.SchDoc tab at the top of the design window to make the schematic sheet the active document.
2. Click in the middle of the wire that connects R1 to the base wire of Q1. Small, square editing handles will appear at each end of the wire and the selection color will display as a dotted line along the wire to indicate that it is selected. Press the **DELETE** key to delete the wire.
3. Recompile the project (**Project»Compile PCB Project**) to check for errors. The Messages panel will display warning messages indicating you have unconnected pins in your circuit.
4. When you double-click on an error or warning in the Messages panel, the Compile Errors panel will also open and give more details about the violation. From this panel you can click on an error and jump to the violating object in a schematic to check or correct the error.

Tip:

To clear all messages from the Messages panel, right-click in the panel and select Clear All.

Before we finish this section of the tutorial, let's fix the error in our schematic.

1. Make the schematic sheet the active document.
2. Select **Edit»Undo** from the menus (shortcut: **CTRL + Z**). The wire you deleted previously should now be restored.
3. To check that the undo was successful, recompile the project (**Project»Compile PCB Project**) to check that no errors are found. The Messages panel should show no errors.
4. Select **View»Fit All Objects** (shortcut: **V, F**) from the menus to display the entire schematic.
5. Save the schematic and the project file as well.

Now we have completed and checked our schematic, it is time to create the PCB.

Creating a New PCB

Main article: [Preparing the Board for Design Transfer](#)

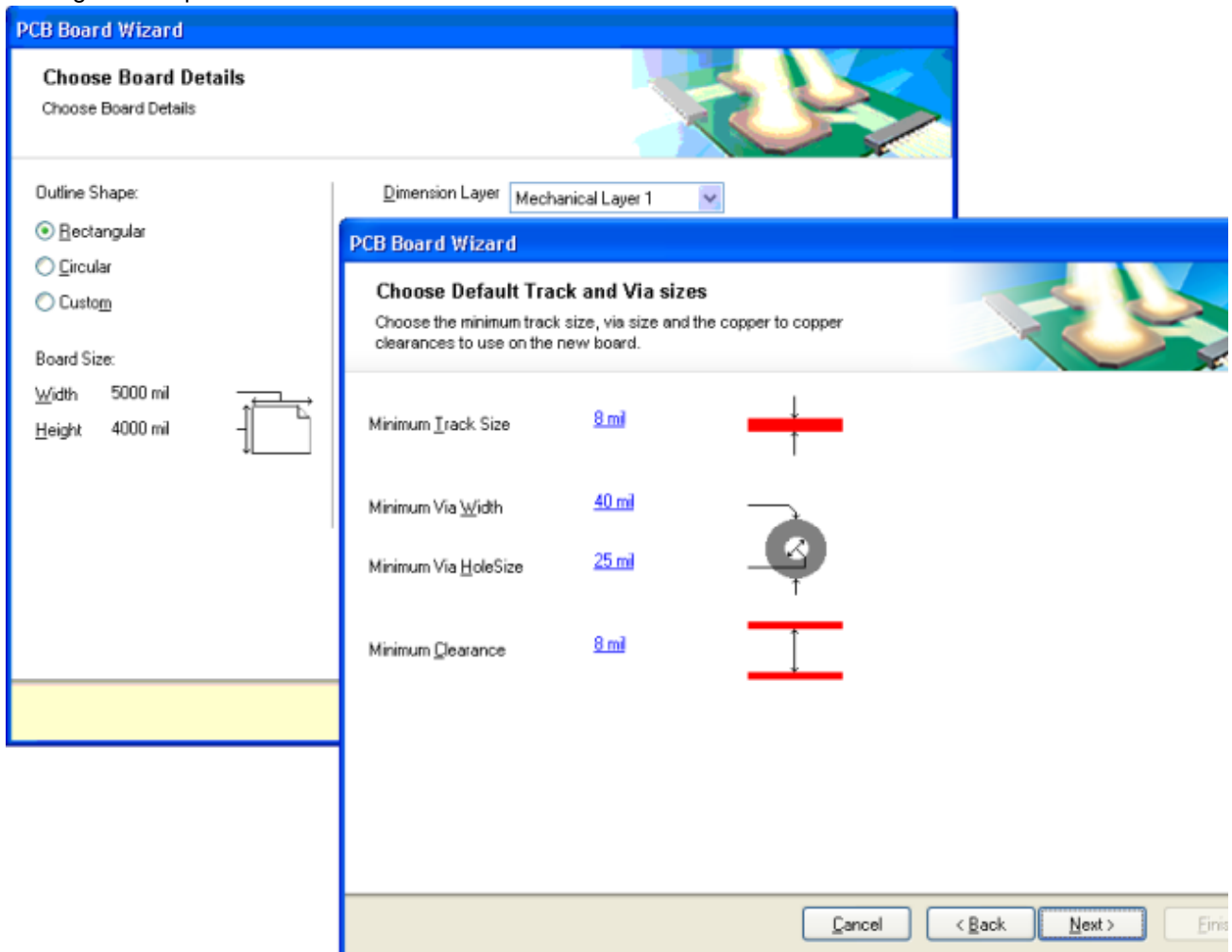
Before you transfer the design from the Schematic Editor to the PCB Editor, you need to create the blank PCB with at least a board outline. The easiest way to create a new PCB design in Altium Designer is to use the **PCB Board Wizard**, which allows you to choose from industry-standard board outlines as well as create your own custom board sizes. At any stage you can use the Back button to check or modify previous pages in the wizard.

To create a new PCB using the PCB Wizard, complete the following steps:

1. Display the Files panel. The default location for this panel is docked on the left side of Altium Designer. If the Files panel is not available, click the **System** button down the bottom right of the workspace and select **Files** f

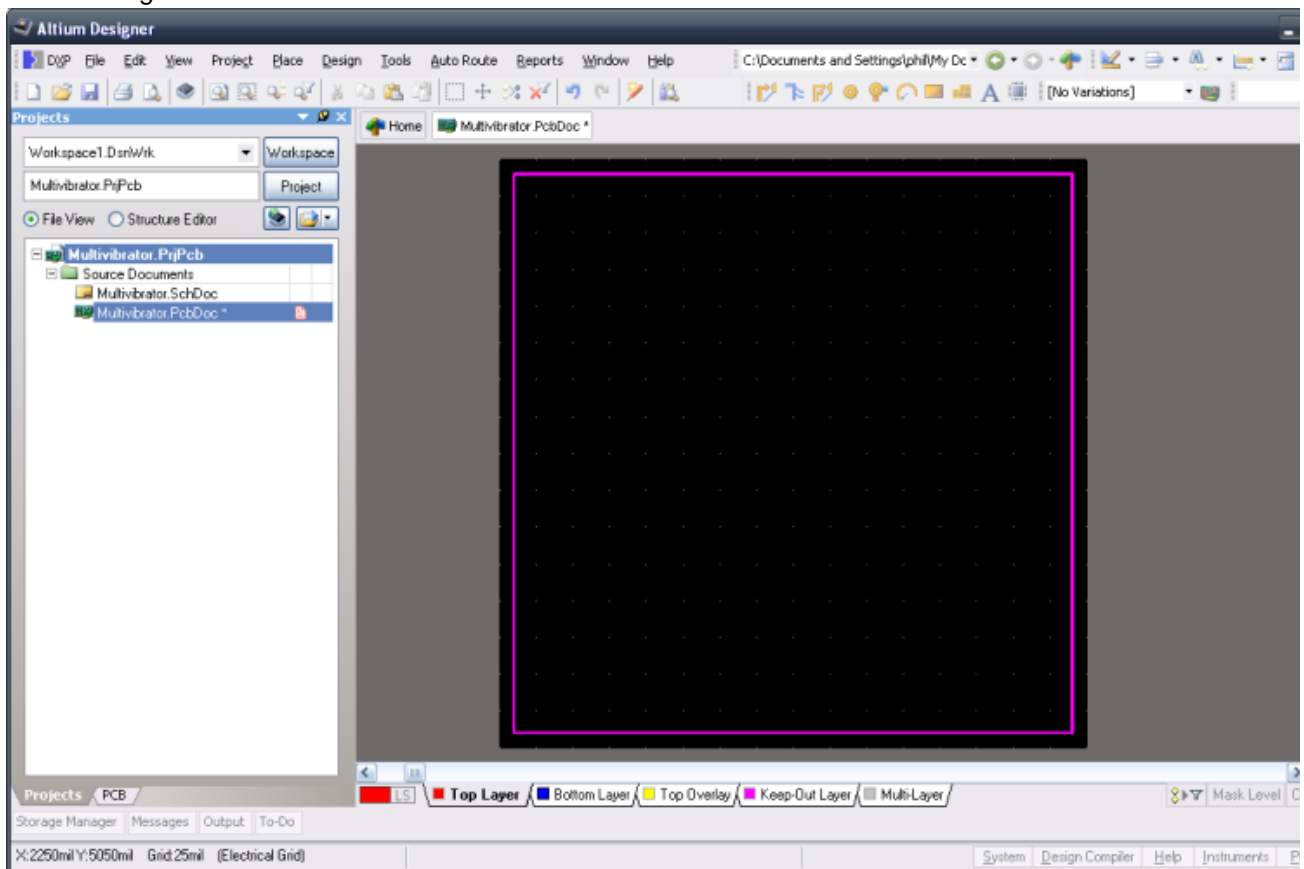
rom the menu that appears.

2. Create a new PCB by clicking on **PCB Board Wizard** in the **New from Template** section at the bottom of the Files panel. If this option is not visible on your screen, close some of the upper sections of the Files panel by clicking on the up arrow icons.



3. The **PCB Board Wizard** opens with an introduction page. Click **Next** to continue.
4. Set the measure units to **Imperial**, i.e. 1000mil = 1 inch.
5. On the third page of the wizard you select the board outline you wish to use. For this tutorial we will enter our own board size. Select **Custom** from the list of board outlines, then click **Next**.
6. In the next page you enter custom board options. For the tutorial circuit, a 2 x 2 inch board will give us plenty of room. Select **Rectangular** for the **Outline Shape** and type 2000 in both the **Width** and **Height** fields. Deselect **Title Block and Scale**, **Legend String** and **Dimension Lines**. Click **Next** to continue.
7. The next page allows you to select the number of layers in the board. We will need **2** signal layers and no power planes. Click **Next** to continue.
8. For the via style, select Thruhole Vias only, then click **Next**.
9. The next page allows you to set the component/track technology (routing) options. Select the **Through-hole components** option and set the number of tracks between adjacent pads to **One Track**. Click **Next**.
10. The next page allows you to set up some of the design rules for track width and via sizes that apply to your board. Leave the options on this screen set to their defaults. Click **Next**.
11. The **PCB Board Wizard** has now collected all the information it needs to create your new board, click **Finish**. The PCB Editor will now display a new PCB file named PCB1.PcbDoc.
12. The PCB document displays with a default sized white sheet and a blank board shape (black area with grid). To turn off the white sheet, select **Design»Board Options** and deselect **Display Sheet** in the *Board Options*

dialog. You can add your own border, grid reference and title block from other PCB templates supplied with Altium Designer.



Adding an Existing PCB

If the PCB you want to add to a project file already exists, you can add it to your project by right-clicking on the project file in the Projects panel (right-click on the project file, not the PCB file) and selecting **Add Existing to Project**. Choose the new PCB file name and click on **Open**. The PCB will now be listed under **Source Documents** beneath the project in the Projects panel and be linked to the project file. Alternatively you can use the drag and drop technique described in the numbered steps to the left.

13. Now the sheet has been turned off, zoom in to display the board shape only by selecting **View»Fit Board** (shortcut: **V, F**).
14. If it is not currently visible, display the Projects panel (use the **System** button down the bottom right of Altium Designer).
15. If the new PCB has not been automatically added (linked) to the Multivibrator project, click and hold on the PCB file in the Projects panel, and drag and drop it on to the Multivibrator project.
16. **Right-click** on the new PCB in the Projects panel, and select **Save As** from the menu that appears. Check that the PCB is being saved into the same folder as the schematic and project files, then save it with the name `Multivibrator.PcbDoc`.

Transferring the Design

The process of transferring a design from the capture stage to the board layout stage is launched by selecting **Design»Update PCB Document** from the menus - when you do the design is compiled and a set of Engineering Change

Orders is created, that will perform the following steps:

- A list of all components used in the design is built, and the footprint required for each. When the ECOs are executed Altium Designer will attempt to locate each footprint in the [currently available libraries](#), and place each into the PCB workspace. If the footprint is not available, an error will occur.
- A list of all nets (connected component pins) in the design is created. When the ECOs are executed Altium Designer will add each net to the PCB, and then attempt to add the pins that belong to each net. If a pin cannot be added an error will occur - this happens when the footprint was not found, or the pads on the footprint do not map to the pins on the symbol.
- Addition design data is then transferred, including placement rooms, net and component classes, and PCB design rules.

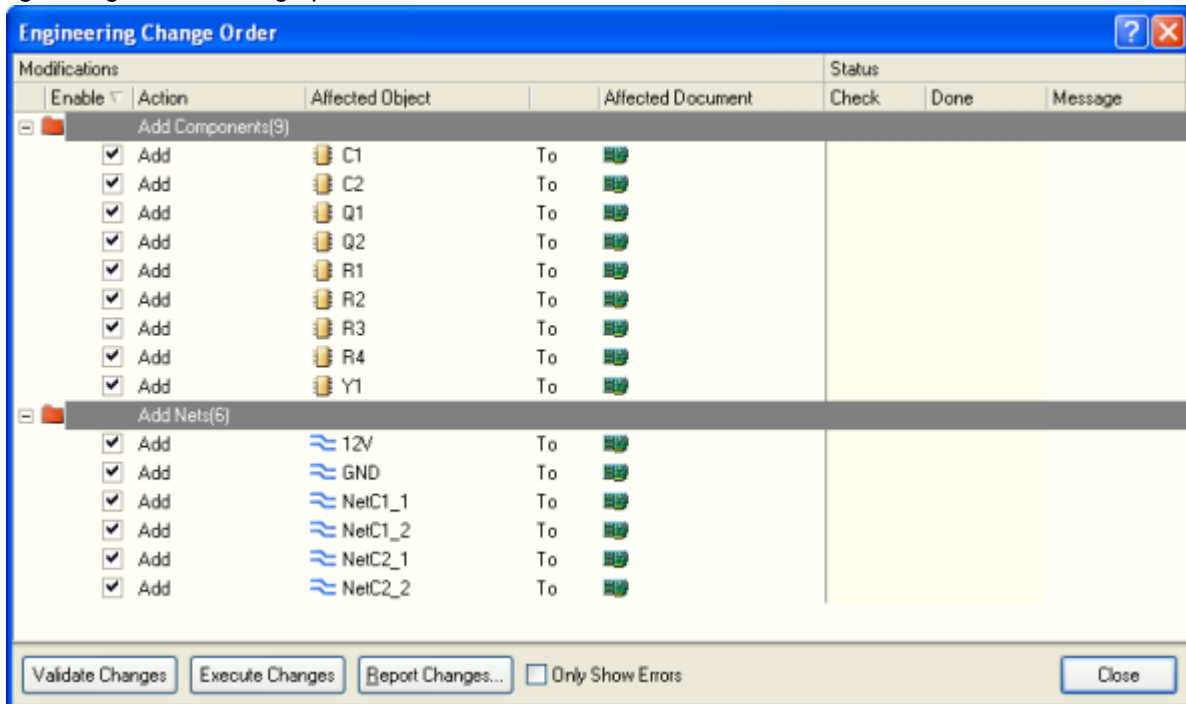
Before transferring the schematic information to the new blank PCB, you should always make sure all the related libraries for both schematic and PCB are available. Since only the default installed integrated libraries are used in this tutorial the required libraries are already available, which means the footprints are available. You are now ready to transfer the design from schematic capture to PCB layout.

Tip:

You can create a report of ECOs to print out by clicking the Report Changes button.

To transfer the schematic information to the target PCB:

1. Open the schematic document, Multivibrator.SchDoc.
2. Select **Design»Update PCB Document (Multivibrator.PcbDoc)**. The project will compile and the *Engineering Change Order* dialog open.



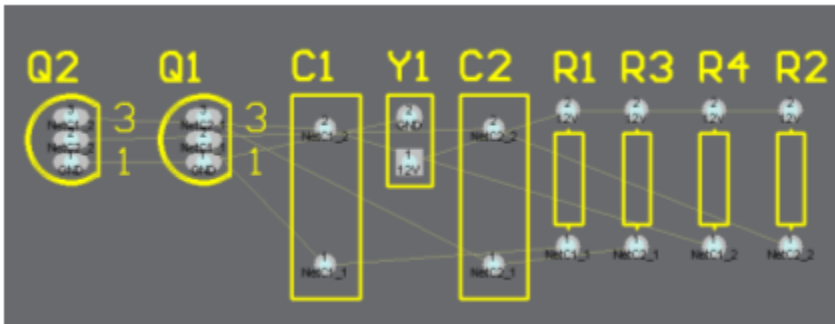
An ECO is created for each change needed to be made to the PCB, so that it matches the schematic.

3. Click on **Validate Changes**. If all changes are validated, a green tick will appear next to each change in the **Status** list. If the changes are not validated, close the dialog, check the Messages panel and clear any errors.
4. Click on **Execute Changes** to send the changes to the PCB. When completed, the **Done** column entries become ticked.
5. Click Close and the target PCB opens with components positioned ready for placing on the board. Use the shortcut **V, D (View»Document)** if you cannot see the components in your current view.

Ready to Start the PCB Design Process

Once all of the ECOs have been executed, the components and nets will appear in the PCB workspace, just to the right of the board outline. Note that some of the component pads will be highlighted in green - that indicates a design rule violation. We'll identify and resolve these shortly.

Note that the PCB Editor is capable of rendering the PCB design in both 2 Dimensional and 3 Dimensional modes (3D requires a graphics card that supports DirectX 9.0C and Shader Model 3 or better, check the [Performance comparison of graphics cards](#) article for more information about suitable cards). 2D mode is a multi-layered environment that is ideal for normal PCB design tasks, such as placing components, routing and connecting. 3D mode is useful for examining your design both inside and out as a full 3D model (3D mode does not provide the full range of functionality available in 2D mode). You can switch between 2D and 3D modes through **View»Switch To 3D** or **View»Switch To 2D** (shortcuts: **2** (2D), **3** (3D)).



The components and nets needed for the design, placed in the PCB workspace.

Setting Up the PCB Workspace

Before we start positioning the components on the board we need to configure certain PCB workspace settings, such as the grids, and also configure board settings, such as layers and design rules.

PCB Workspace Grids

Tip:

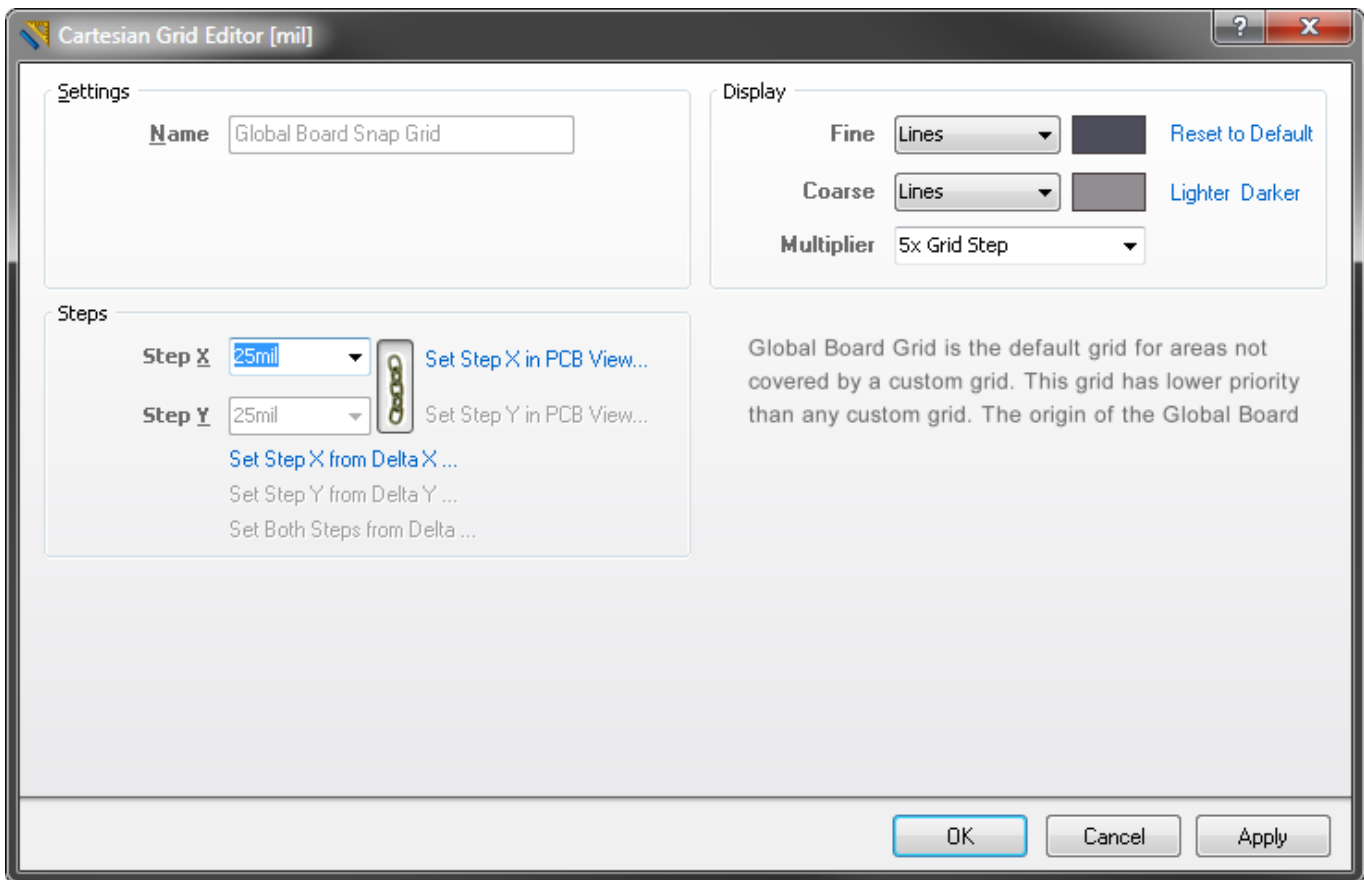
The PCB Editor supports imperial and metric units. Select **View»Toggle Units** to switch (or press the Q shortcut key). Regardless of the current setting for the units, you can include the units when entering a value in a dialog to force that value to be used, or press the **Ctrl+Q** shortcuts to toggle units when a dialog is open.

We need to ensure that our placement grid is set correctly before we start positioning the components. All the objects placed in the PCB workspace are aligned on a grid called the snap grid. This grid needs to be set to suit the routing technology that we plan to use.

Our tutorial circuit uses standard imperial components that have a minimum pin pitch of 100mil. We will set the snap grid to an even fraction of this, say 50mil or 25mil, so that all component pins will fall on a grid point when placed. Also, the track width and clearance for our board are 12mil and 13mil respectively (the default values used by the **PCB Board Wizard**), allowing a minimum of 25mil between parallel track centers. The most suitable snap grid setting would, therefore, be 25mil.

To set the snap grid, complete the following steps:

1. Select **Design»Board Options** (shortcut: **D, O**) to open the *Board Options* dialog.
2. Click the **Grids** button down the bottom right to open the *Grid Manager* (shortcut **G, M**).
3. Altium Designer supports multiple user-defined grids, in both Cartesian and polar forms. For this tutorial only the **Default** grid is used, double-click on it to edit the settings in the *Cartesian Grid Editor*. Set the grid **Steps** value to 25mils.
4. Click **OK** to close the dialogs.

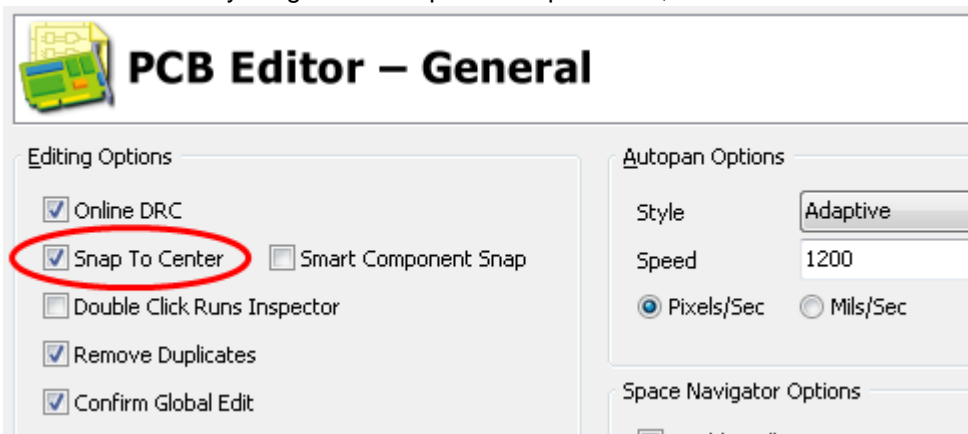


Set the Snap Grid to 25 mils.

Component Positioning and Placement options

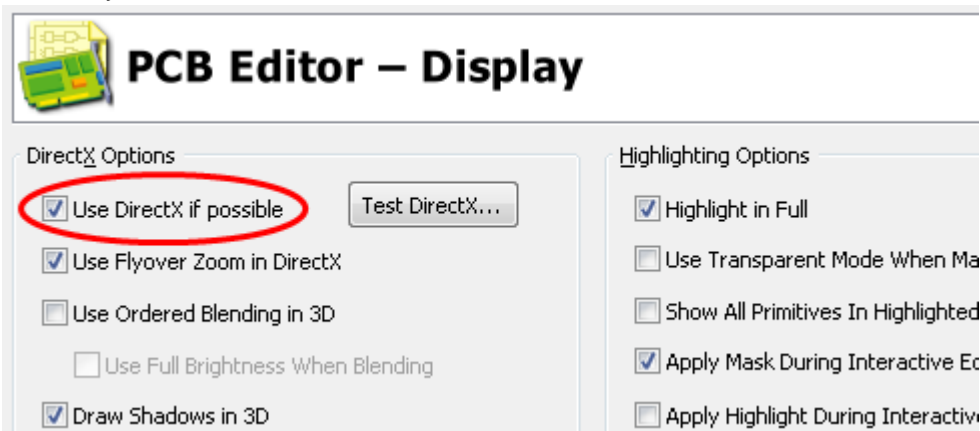
Let's set some other options that will make positioning components easier.

1. Select **Tools»Preferences** (shortcut: **T, P**) to open the *Preferences* dialog. Open the **PCB Editor - General** page of the dialog, in the **Editing Options** section, make sure the **Snap To Center** option is enabled. This ensures that when you "grab" a component to position it, the cursor is set to the component's reference point.

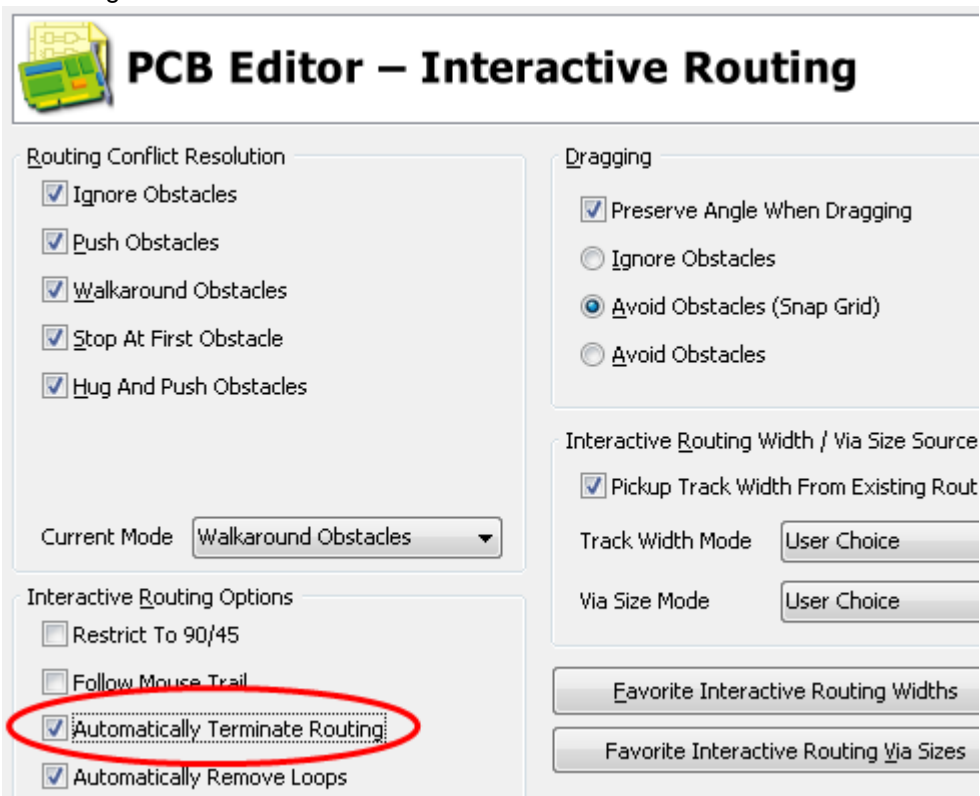


2. Switch to the **PCB Editor - Display** page of the *Preferences* dialog. In the DirectX Options section of this page, enable the Use DirectX if possible option. This will allow us to utilize the 3D view mode. Click **OK** to close the *Preferences* dialog. If you cannot run the DirectX mode you will be limited to using the legacy 3D

viewer if you wish to view the board in 3D.



3. Switch to the **PCB Editor - Interactive Routing** page of the *Preferences* dialog. In the **Interactive Routing Options** section of the page, enable the **Automatically Terminate Routing** option. With this enabled, when a route reaches the target pad the cursor is automatically “released” from that net, ready to select another net for routing.



Defining the Layer Stack and Other Non-electrical Layers

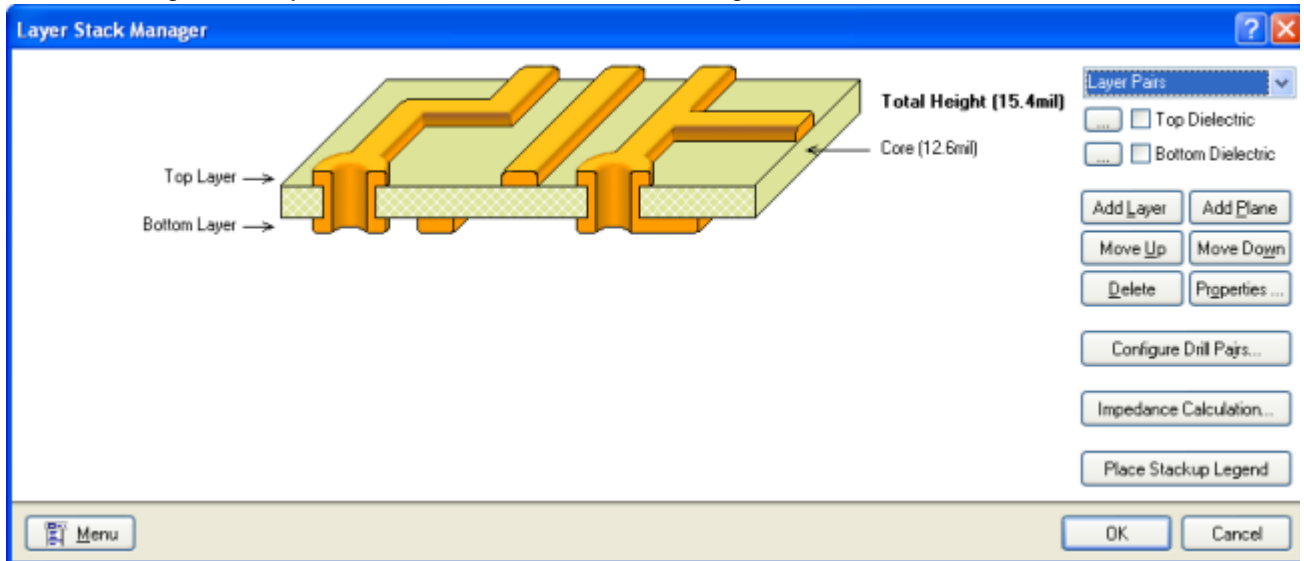
Now that the workspace settings have been configured, the next step is to configure the electrical and non-electrical layers needed for the design.

Physical Layers and the Layer Stack Manager

Altium Designer's PCB Editor supports up to 32 signal and 16 power plane (solid copper) layers. The tutorial PCB is a simple design and can be routed as a single-sided or double-sided board. If the design was more complex, you

would add more layers through the *Layer Stack Manager* dialog.

1. Select **Design»Layer Stack Manager** (shortcut: **D, K**) to display the *Layer Stack Manager* dialog.
2. New layers and planes are added below the currently selected layer. Layer properties, such as copper thickness and dielectric properties are used for signal integrity analysis, and can be configured by double-clicking on the layer name. Click **OK** to close the dialog.



Configuring the Display of Layers

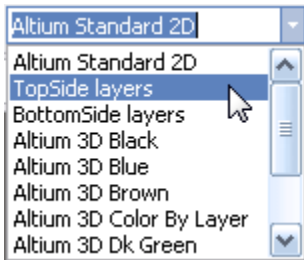
As well as the electrical (signal and power plane) layers, the PCB Editor also supports numerous other non-electrical layers. There are three types of layers available in the PCB Editor:

- **Electrical layers** - includes the 32 signal layers and 16 internal power plane layers.
- **Mechanical layers** - there are 32 general purpose mechanical layers, used for design tasks such as dimensions, fabrication details, assembly instructions, or special purpose tasks such as glue dot layers. These layers can be selectively included in print and Gerber output generation. They can also be paired, meaning that objects placed on one of the paired layers in the library editor, will flip to the other layer in the pair when the component is flipped to the bottom side of the board.
- **Special layers** - these include the top and bottom silkscreen layers, the solder and paste mask layers, drill layers, the Keep-Out layer (used to define the electrical boundaries), the multilayer (used for multilayer pads and vias), the connection layer, DRC error layer, grid layers, hole layers, and other display-type layers.

The display attributes of all layers are configured in the *View Configurations* dialog (**Design»Board Layers and Colors**, or press the **L** shortcut). Since there are so many layers, and since during the design process you will work with many different settings of layers turned on and off, the current settings in the *View Configurations* dialog can be saved as a **View Configuration**. You can easily switch between available **View Configurations** via the menu in the main toolbar, as shown below.

Tip:

The currently enabled layers are shown as a series of Tabs across the bottom of the PCB workspace. Right-click on a Tab to access frequently used layer display commands.



Use the dropdown to quickly switch between view configurations.

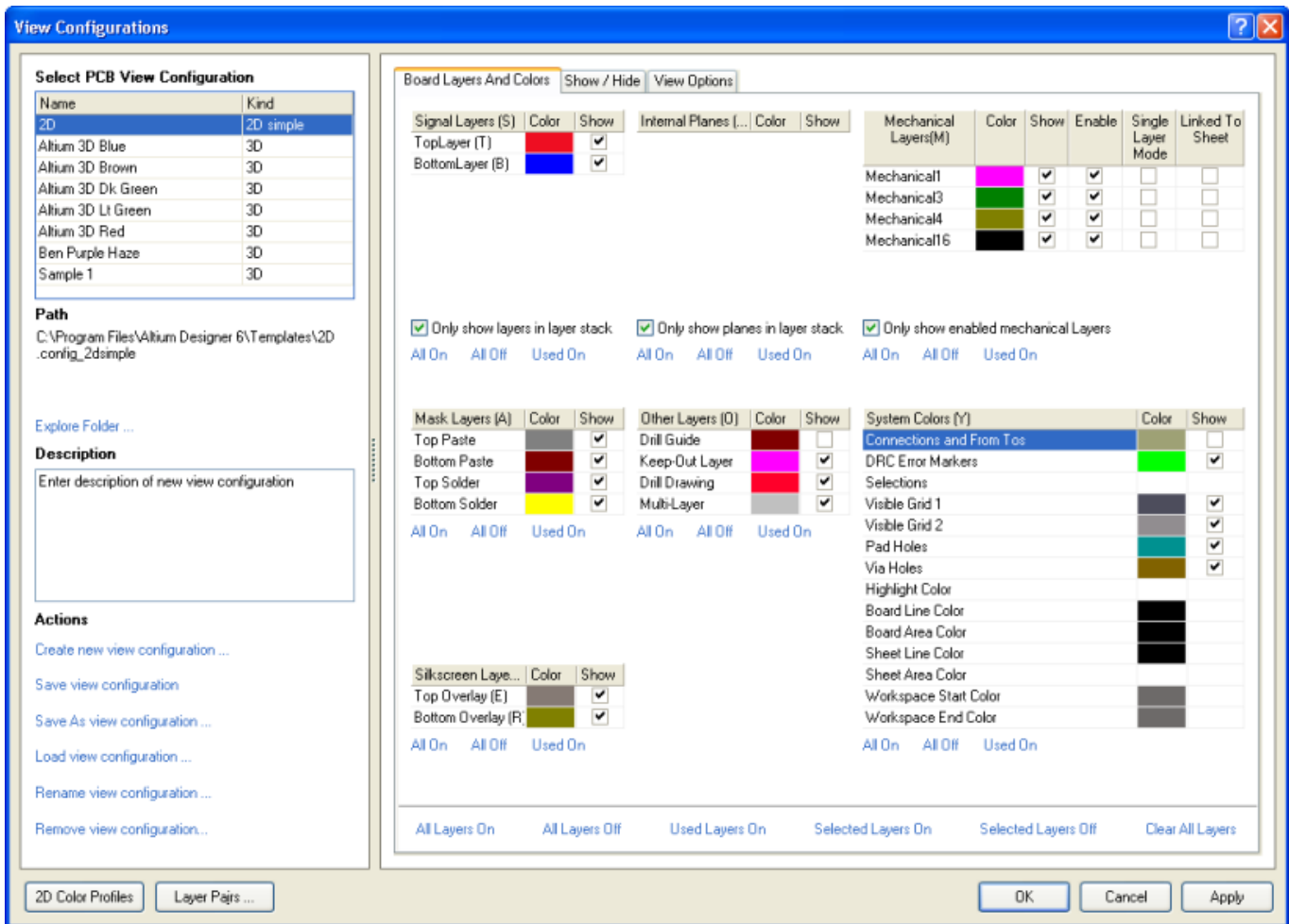
View configurations are settings that control numerous PCB workspace display options for both 2D and 3D display modes (and apply to both the PCB and PCB Library Editors). The view configuration last used when saving a PCB document is also saved with the file itself. This enables it to be viewed on another instance of Altium Designer using its associated view configuration. View configurations can also be saved locally and be used and applied at any time to any PCB document. If you open a PCB file does not have an associated view configuration, a system default one is used. View Configurations are created and saved using the options on the left hand side of the *View Configurations* dialog.



As well as layer visibility, the *View Configurations* dialog provides access to 2D color settings for layers and other system-based color settings - note that since these are *system settings* they will apply to all PCB documents and are not part of a view configuration. Color profiles for the 2D workspace can also be created and saved, similarly to view configurations, and can be applied at any time.

As well as the layer display state and color settings, the *View Configurations* dialog also gives access to other display settings, including:

- How each type of object is displayed (solid, draft or hidden), in the **Show/Hide** tab of the dialog.
- Various view options, such as if **Pad Net** names and **Pad Numbers** are to be displayed, the **Origin Marker**, if **Special Strings** should be converted, and so on. These are configured in the **View Options** tab of the dialog.



Press the L shortcut to open the View Configurations dialog

Let's create a simple 2D view configuration for this tutorial.

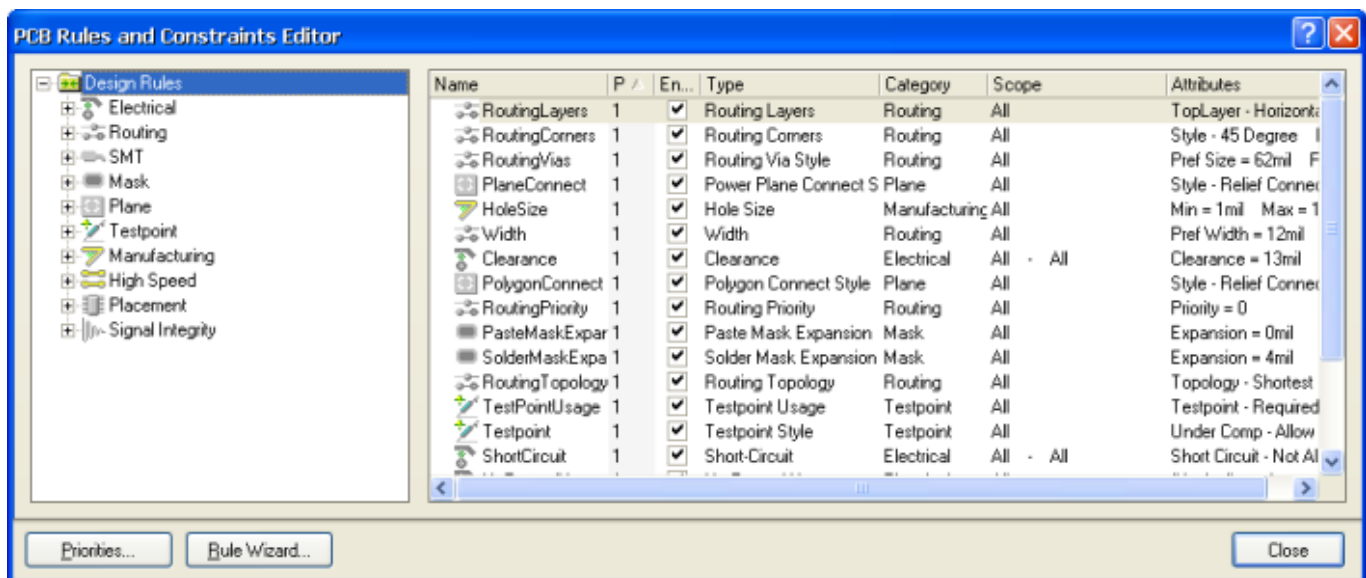
1. Open the *View Configurations* dialog (**Design»Board Layers & Colors**). The dialog opens with the active configuration selected in the **Select PCB View Configuration** area on the top left. If you were in 3D mode, click on a 2D configuration.
2. In the **Board Layers And Colors** tab, ensure that the **Only show layers in layer stack** and **Only show enabled mechanical layers** options are enabled. These settings will display only the layers in the stack.
3. Click the **Used Layers On** control at the bottom of the page. This will display only layers that are currently being used, that is, they have design objects on them.
4. Click on the color next to **Top Layer** to display the *2D System Colors* dialog. Note that you can easily change the color of a layer, remembering that changes made to layer colors are system settings, so will apply to each board you open. Note also the dialog includes a **Previous** option, allowing you to easily restore a color setting if you decide you do not like one you have selected. Click **Cancel** to close the dialog without applying a change.
5. Disable the display of the four Mask layers, the Drill Guide and Drill Drawing layers.
6. In the **Actions** section, click **Save As view configuration** and save the file as *Tutorial*. Note that you do not need to type in the file extension, this is always added automatically in Altium Designer.
7. Click **OK** when you return to the *View Configurations* dialog to apply the changes and close it.
8. Note that your new View Configuration will be active, you can confirm this by checking in the drop down View Configuration list in the main toolbar.

Note : Remember that 2D layer color settings are system-based, affecting all PCB documents, and are not part associated with any view configurations. You can create, edit and save 2D color profiles from the *2D System Colors* dialog.

Setting Up the Design Rules

The PCB Editor is a rules-driven environment, meaning that as you perform actions that change the design, such as placing tracks, moving components, or autorouting the board, Altium Designer monitors each action and checks to see if the design still complies with the design rules. If it does not, then the error is immediately highlighted as a violation. Setting up the design rules before you start working on the board allows you to remain focused on the task of designing, confident in the knowledge that any design errors will immediately be flagged for your attention.

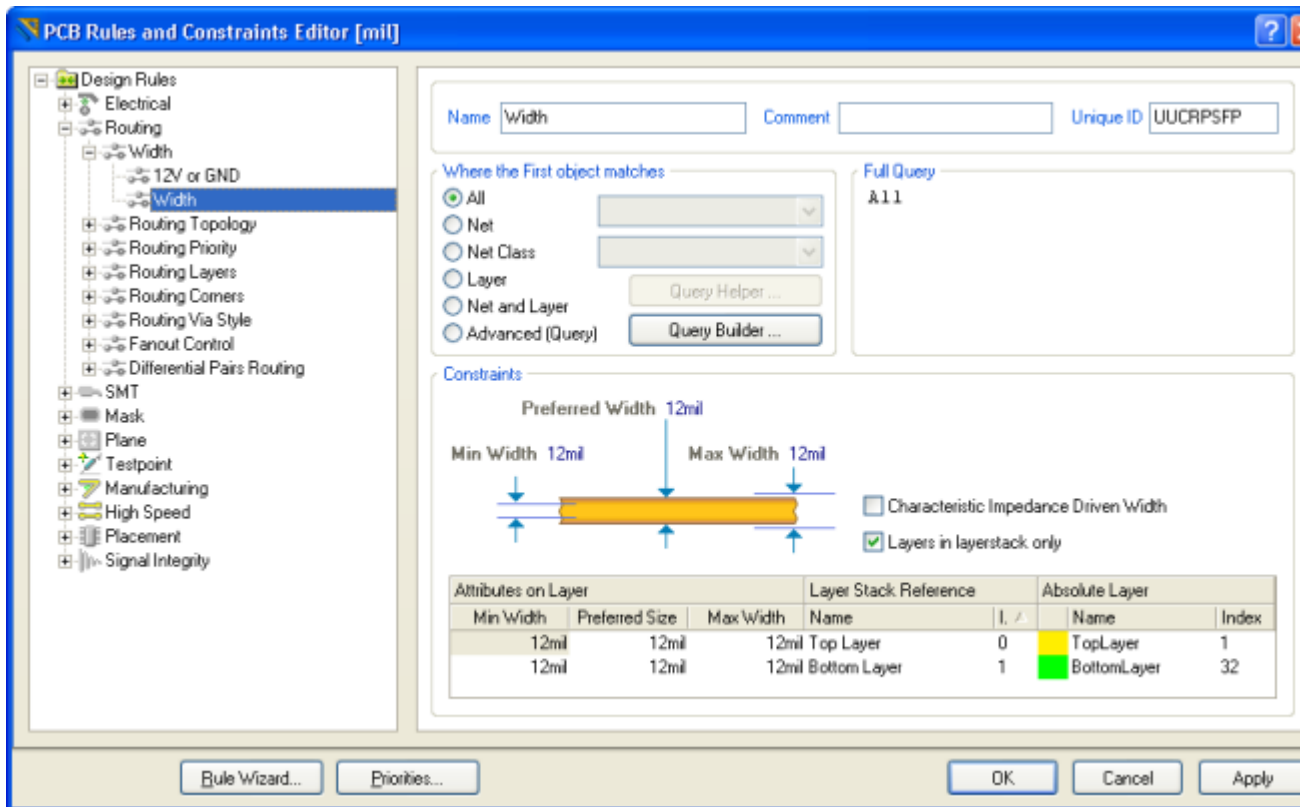
The design rules fall into 10 categories, which can then be further divided into design rule types. The design rules cover electrical, routing, manufacturing, placement and signal integrity requirements.



All PCB design requirements are configured as rules/constraints, in the Rules and Constraints Editor dialog.

We will now set up new design rules to specify the width that the power nets must be routed. To set up these rules, complete the following steps:

1. With the PCB as the active document, select **Design»Rules** from the menus.
2. The PCB Rules and Constraints Editor dialog will appear. Each rules category is displayed under the **Design Rules** folder (left hand side) of the dialog. Double-click on the Routing category to expand the category and see the related routing rules. Then double-click on Width to display the currently defined width rules.
3. Click once on each rule to select it. As you click on each rule, the right hand side of the dialog displays the settings for the rule, including: the rule's scope (what you want this rule to target) in the top section, and the rule's constraints in the bottom section. These rules are either defaults, or have been set up by the **PCB Board Wizard** when the new PCB document was created.
4. Click on the Width rule to display its scope and constraints. This rule applies to all nets on the entire board, because the **Scope** is set to **All**.

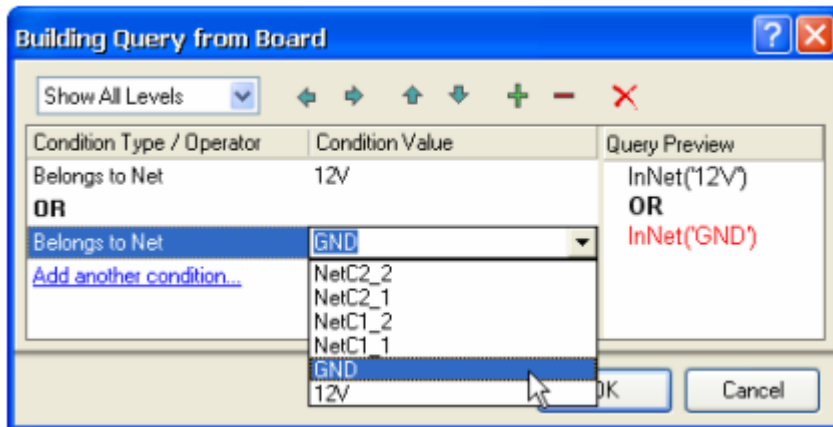


One of the powerful features of Altium Designer's design rule system is that multiple rules of the same type can be defined, each targeting different objects, or a subset of objects already targeted by another rule. The exact set of objects that each rule targets is defined by that rule's **Scope**. The order that rules of the same type are applied is determined by the **Rule Priority**. For example, you could have a width constraint rule for the whole board (meaning all nets must be routed this width), a second width constraint rule for the ground net (this rule would have a higher priority, overriding the previous rule), and a third width constraint rule for a particular connection on the ground net (which has the highest priority, overriding both of the previous rules). The Rule priority is displayed when you click on the rule type in the tree on the left of the dialog, in this example you would click on **Width** to display a summary of all width-type rules, including their Priority setting.

Currently there is one width constraint rule for your design, which applies to the whole board (width = 12mil). We will now add a new width constraint rule for the 12V and GND nets (width = 25mil). To add new width constraint rules, complete the following steps:

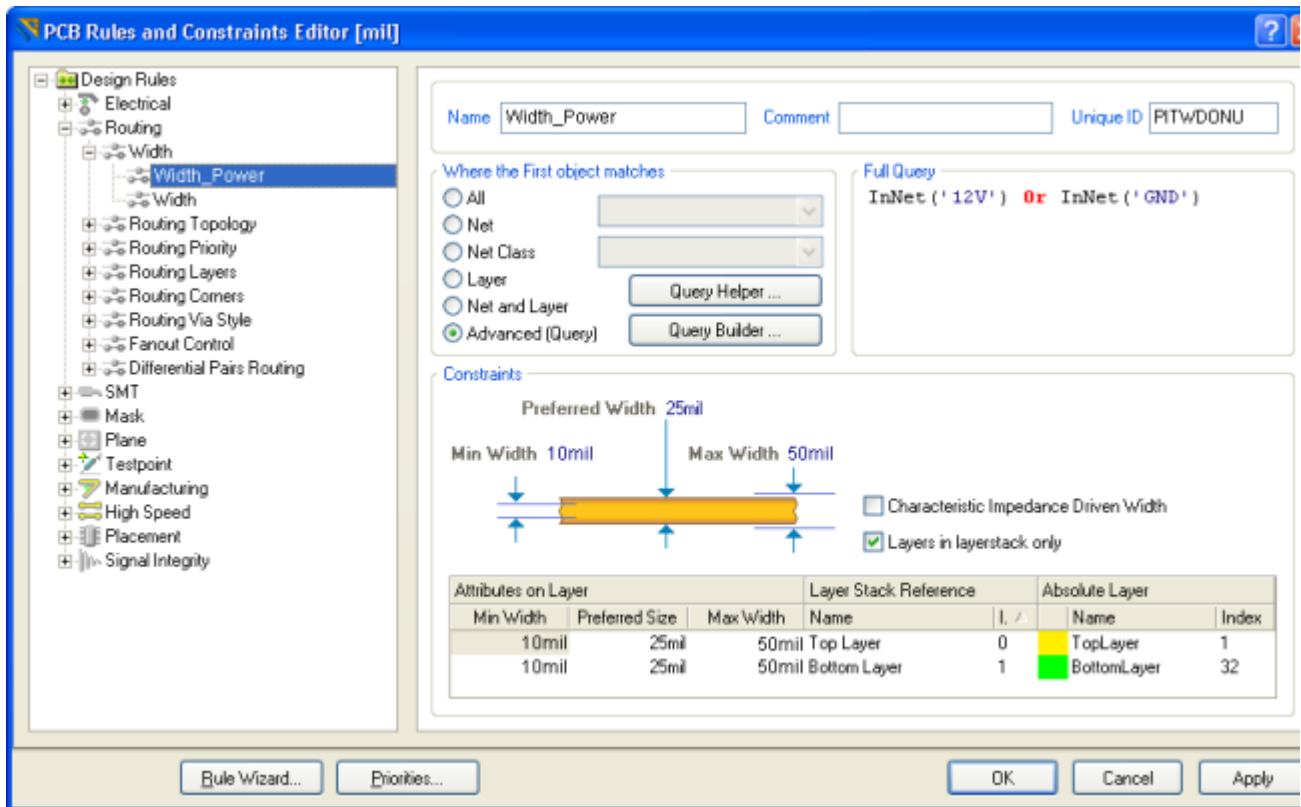
1. With the **Width** rule-type selected in the Design Rules tree on the left of the *PCB Rules and Constraint Editor* dialog, right-click and select **New Rule** to add a new width constraint rule.
A new rule named Width_1 appears. Click on the new rule in the Design Rules folder to modify the scope and constraints.
2. Type Width_Power in the **Name** field.
3. Next we set the rule's scope using the **Query Builder**, to access this select the **Advanced (Query)** option.
Note that you can always type in the scope directly if you know the correct syntax. Alternatively, if your query

is more complicated you could select the Advanced option, then click the **Query Helper** button to use the *Query Helper* dialog.



4. Click on the **Query Builder** button to open the *Building Query from Board* dialog.
5. Click on **Add first condition** and select **Belongs to Net** from the drop-down list. In the **Condition Value** field, click and select the net **12V** from the list. The **Query Preview** now reads **InNet ('12V')**.
6. Click on **Add another condition** to widen the scope to include the GND net. Select **Belongs to Net** and **GND** as the **Condition Value**.
7. Change the operator on the left of the dialog by clicking on the operator **AND**, and then selecting **OR** from the dropdown list. Check that the preview reads **InNet('12V') OR InNet('GND')**, then click **OK** to close the dialog.
8. Click **OK** to close the *Building Query from Board* dialog. The scope in the **Full Query** section of the new design rule has now been updated with the new query.
9. In the bottom section of the *PCB Rules and Constraints Editor* dialog, set the Width settings to the following values:
 - **Min Width** = 10mil
 - **Preferred Width** = 25mil
 - **Max Width** = 50mil

The new rule is now set up and will save when you select another rule or close the dialog.
10. Finally, click to edit the original rule named **Width** (**Scope** set to **All**) and confirm that the **Min Width**, **Max Width** and **Preferred Width** fields are all set to 12mil.
11. Click **OK** to close the dialog.



When you route the board manually or using the autorouter, all tracks will be 12mil wide, except the GND and 12V tracks which will be 25mil.

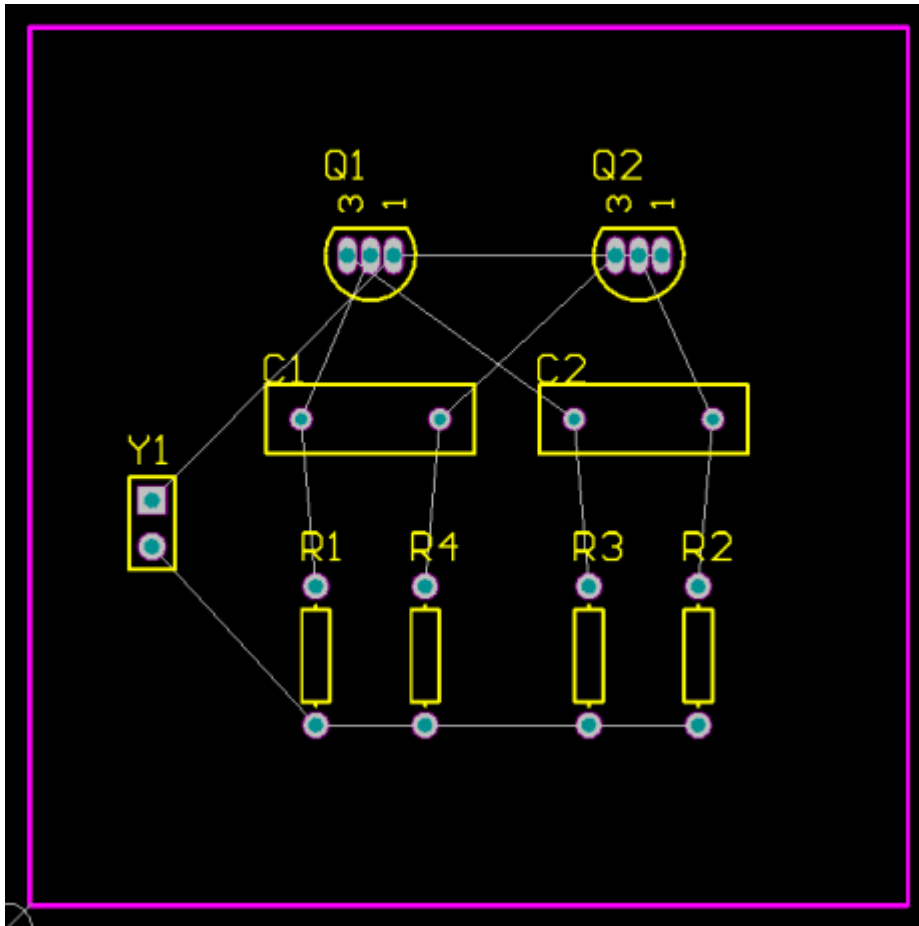
Positioning the Components on the PCB

Now we can start to place the components in their right positions.

Tip:

The connection lines are automatically re-optimized as you move a component. In this way you can use the connection lines as a guide to the optimum position and orientation of the component as you place it.

1. Press the **V, D** shortcut keys to zoom in on the board and components.
2. To place connector Y1, position the cursor over the middle of the outline of the connector, 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.
3. Position the footprint towards the left-hand side of the board (ensuring that the whole of the component stays within the board boundary), as shown in the figure below.

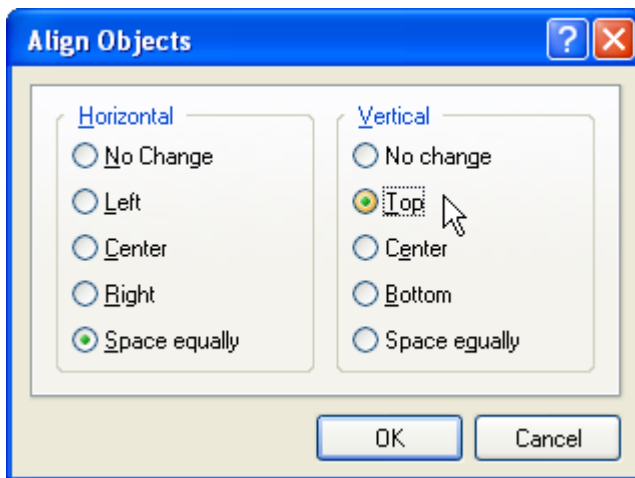


Components positioned on the board.

4. When the connector component is in position, release the mouse button to drop it into place. Note how the connection lines drag with the component.
5. Reposition the remaining components, using the figure above as a guide. Use the **SPACEBAR** to rotate (increments of 90° anti-clockwise) components as you drag them, so that the connection lines are as shown in the figure.
6. Component text can be repositioned in a similar fashion - click-and-drag the text and press the **SPACEBAR** to rotate it.
Altium Designer also includes powerful interactive placement tools. Let's use these to ensure that the four resistors are correctly aligned and spaced.
7. Holding the **SHIFT** key, click on each of the four resistors to select them, or click and drag the selection box around them. A shaded selection box will display around each of the selected components in the color set for the system color called **Selections**. You can change this selection color in the *View Configurations* dialog (**Design»Board Layers & Colors**, or the **L**shortcut).

Tip:

Selected objects can be moved in increments of the current snap grid by pressing one of the **Arrow** keys while holding down the **CTRL** key. Include the **SHIFT** key to move selected objects in 10xSnap Grid steps.



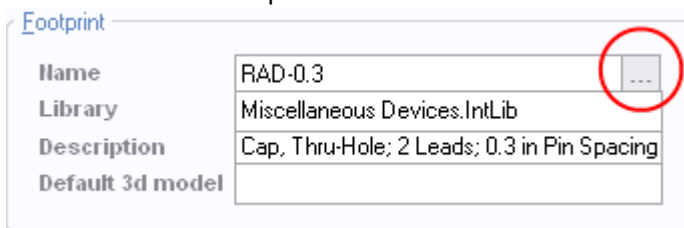
Align and space the resistors.

8. Right-click and select **Align»Align** (shortcut: **A, A**). In the *Align Objects* dialog, click on **Space Equally** in the **Horizontal** section and click on **Top** in the **Vertical** section. The four resistors are now aligned and equally spaced.
9. Click elsewhere in the design window to de-select all the resistors.

Changing a Footprint

Now that we have positioned the footprints, we can see the capacitor footprint is too big for our requirements! Let's change the capacitor footprint to a smaller one.

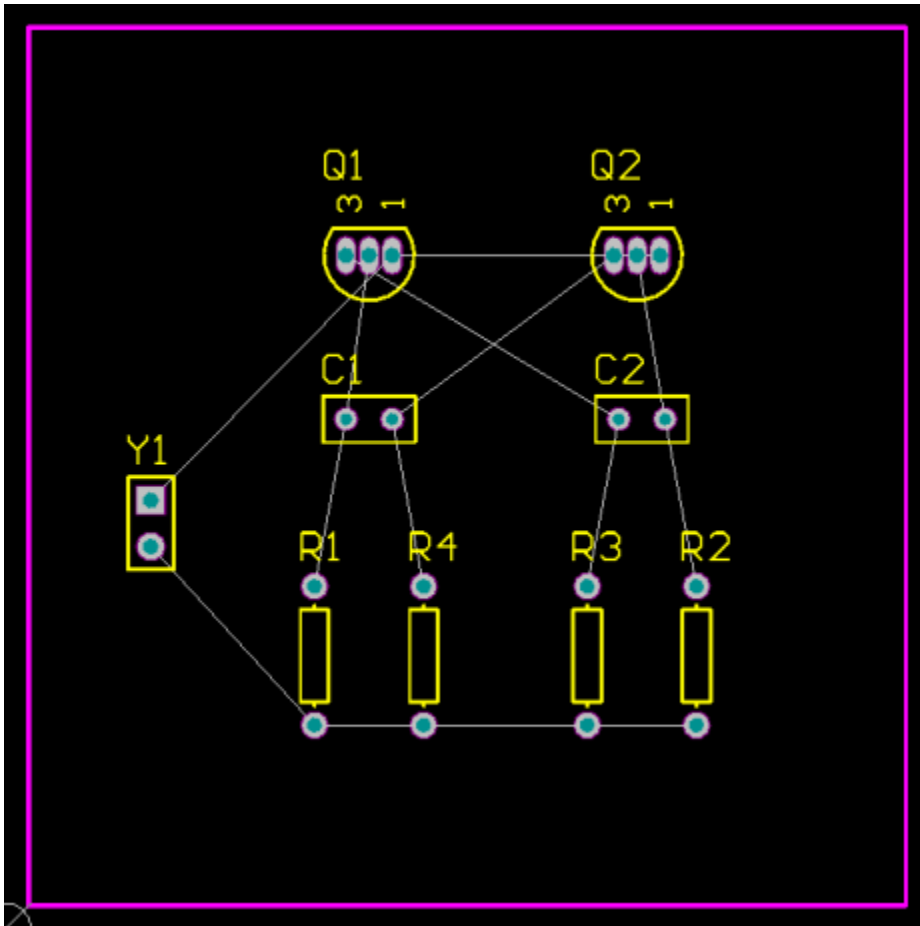
1. Double-click on one of the capacitors to open the *Component* dialog.
2. In the **Footprint** region of the dialog, you will see that the current footprint **Name** is RAD-0.3. To choose another footprint, click the ... button, as shown in the figure below, to open the *Browse Libraries* dialog and choose a different footprint.



Browse to select a different footprint.

3. In the *Browse Libraries* dialog, click the dropdown arrow to display the list of currently installed libraries, ensure that the **Miscellaneous Devices.IntLib** is selected.
4. We want a smaller radial type footprint, so type `rad` in the **Mask** field of the dialog to display only the radial style footprints.
5. RAD-0.1 will be suitable, so select it and click **OK** to close the *Browse Libraries* dialog, then click OK again to close the *Component* dialog. The capacitor should show the new smaller footprint.
6. Repeat the process for the other capacitor.
7. Reposition the designators as required.
8. Save the PCB file.

Your board should now look something like the figure below.



Components placed on the board with new footprints.

With everything positioned, it's time to do some routing!

Interactively Routing the Board

Main articles: [Getting ready to route](#), [Interactively Routing a Net](#), [Modifying Existing Routing](#)

Routing is the process of laying tracks and vias on the board to connect the components. Altium Designer makes this job easy by providing sophisticated interactive routing tools as well as the Situs topological autorouter, which optimally routes the whole or part of a board at the touch of a button.

While autorouting provides an easy and powerful way to route a board, there will be situations where you will need exact control over the placement of tracks - or you may want to route the board manually just for the fun of it! In these situations you can manually route part or all of your board. In this section of the tutorial, we will manually route the entire board single-sided, with all tracks on the bottom 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 or walking around obstacles, automatically following existing connections, in accordance with applicable design rules.


We will now place tracks on the bottom layer of the board, using 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

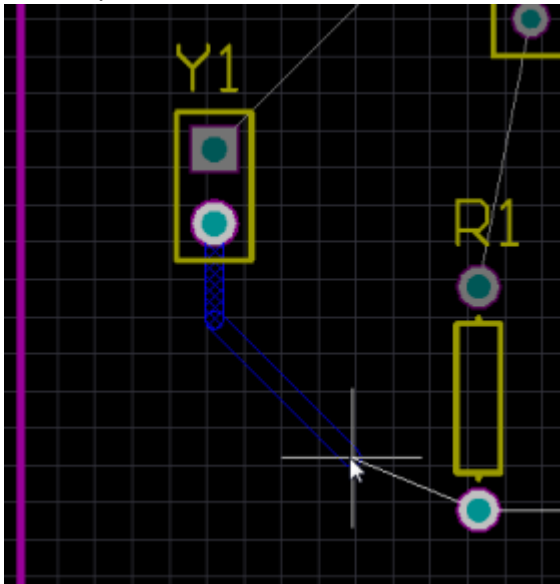
segment begins. Also, by default Altium Designer constrains tracks to a vertical, horizontal or 45° orientation, allowing you to easily produce professional results. This behavior can be customized to suit your needs, but for this tutorial we will use the default.

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, press the **L** shortcut to open the *View Configurations* dialog, and enable the **Bottom Layer**.
2. Click on the **Bottom layer** tab at the bottom of the workspace to make it the current, or active layer.

Tip:

While routing, you can cycle through currently displayed signal layers by pressing the * shortcut on the numeric keypad.

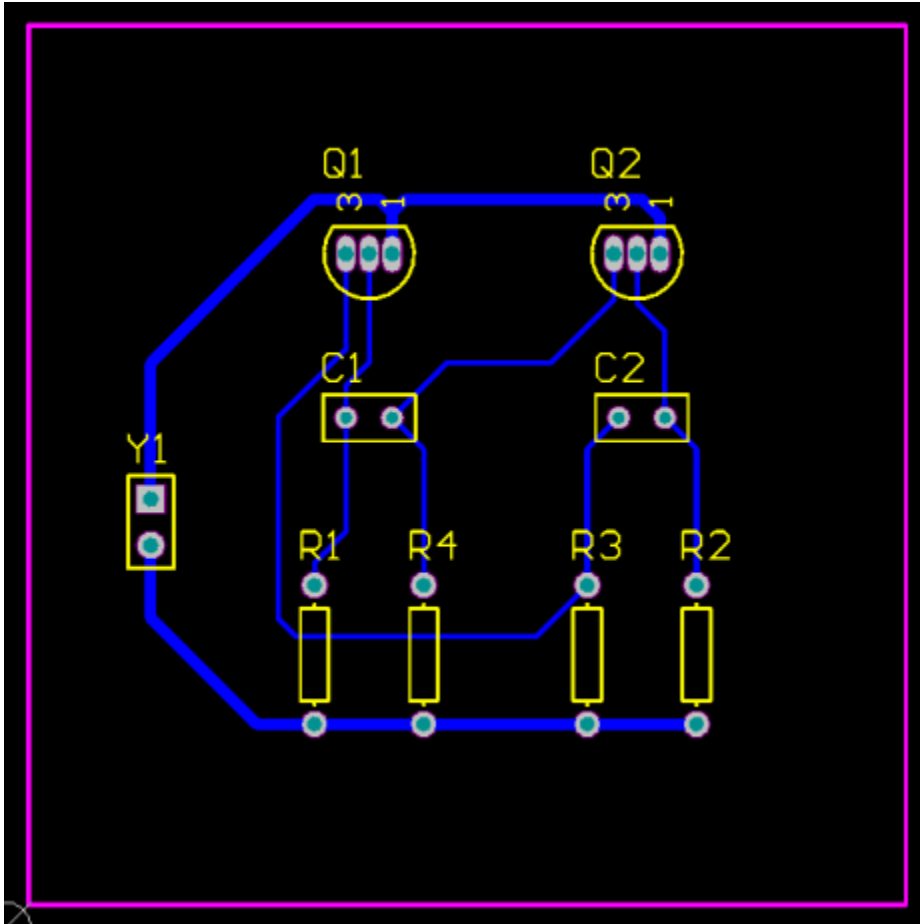
3. Select **Place»Interactive Routing** from the menus (shortcut: **P, T**) or click the Interactive Routing button . The cursor will change to a crosshair indicating you are in track placement mode.
4. Position the cursor over the lower pad on connector Y1. As you move the cursor close to the pad it will automatically snap to the center of the pad - this is the electrical grid feature *pulling* the cursor to the center of the nearest electrical object. **Left-Click** or press **ENTER** to anchor the first point of the track.
5. Move the cursor towards the bottom pad of the resistor R1. Note how track segments are displayed in a check pattern or are displayed hollow, following your cursor path (the figure below). The check pattern/hollow indicates that they have not been committed (placed). If you pull your cursor back along the path, the uncommitted routing *unwinds* also. You have two choices with routing here:
 - Press **CTRL + Left Click** to use the *Auto-Complete* function and immediately route the connection (you can also use this technique directly on a pad or connection line). The routing has to be valid in terms of any obstacles on the board for Auto-Complete to work. On large boards, 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.
 - Manually route by **Left-Clicking** to commit track segments, finishing on the lower pad of R1. This method provides control over the route and still minimizes the number of user actions required.



Cursor following streamlines the manual routing process. Uncommitted tracks are shown hatched/hollow, committed tracks display in solid color.

6. Use either of the above methods to route between the other components on the board. The figure below shows a manually routed board.

7. Note that if you unhappy with a particular route path, simple route the section you want changed again. Altium Designer includes a powerful **loop removal** feature, as soon as your new route re-connects and you right-click to terminate routing, the old routing (loop) is automatically removed.
8. Save the design when you are finished.



Manually routed board, with all tracks placed on the bottom layer.

Altium Designer's Interactive Routing tool features modes that you can use to resolve conflicts with obstacles on the board, such as pads and existing tracks. Press the **SHIFT + R** shortcuts to cycle through these modes as you interactively route - note that the current mode is displayed on the Status bar.

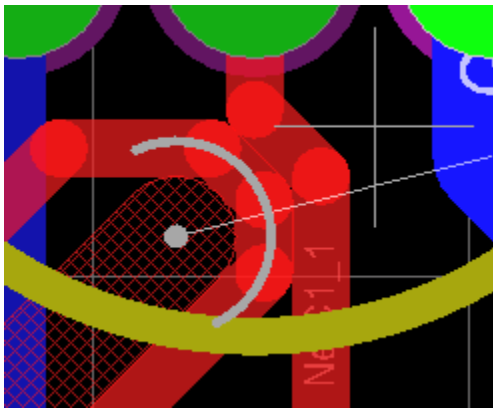
Tip:

You can control which modes are available as you press SHIFT+R, this is configured in the PCB Editor - Interactive Routing page of the *Preferences* dialog (DXP»Preferences).

The available modes are:

- **Walkaround** - This mode will attempt to find a routing path around existing obstacles without attempting to move them.
- **Stop at first obstacle** - In this mode the routing is essentially manual, as soon as an obstacle is encountered the track segment will be clipped to avoid a violation.
- **Push** - This mode will attempt to move objects (tracks and vias), which are capable of being repositioned without violation, to accommodate the new routing.
- **Hug & Push*** - This mode is a combination of Walkaround and Push functionality. It will walkaround obstacles, however, will also take on Push mode against fixed obstacles
- **Ignore** - This mode that lets you place tracks anywhere, over existing objects, ignoring violations.

During interactive routing, if you attempt to route into an area that cannot be resolved using Push or Hug & Push modes, an indicator appears at the end of the permissible tracks so you know immediately that you are blocked, as shown in the figure below.



When Push or Hug & Push modes cannot find a path to the target pad, a blocking indicator appears.

Tips for Routing

Keep in mind the following points as you are placing the tracks:

- **Left-Click** or press **ENTER** to place track segments up to the current cursor position. Check pattern segments represent uncommitted routing, hollow tracks represent the look-ahead segment (this segment is not placed when you click - press the **1** shortcut to disable the look-ahead feature). Committed tracks are shown solid in the layer color.
- Use **CTRL + Left-Click** at any time to Auto-complete the connection. Auto-complete will not succeed if there are unresolvable conflicts with obstacles.
- Use **SHIFT + R** to cycle through conflict resolution modes Push, Walkaround, Hug and Push and Ignore.
- Use **SHIFT + SPACEBAR** to cycle through the various track corner modes. The styles are: any angle, 45°, 45° with arc, 90° and 90° with arc.
- Press **SPACEBAR** to toggle the corner direction for all but any angle mode.
- Press **END** at any time to redraw the screen.
- Use **V**, **F** at any time to redraw the screen to fit all objects.
- Press **PAGE UP** and **PAGE DOWN** keys at any time to zoom in or out, centered on the cursor position. Use the mouse wheel to pan left and right. Hold the **CTRL** key down to zoom in and out with the mouse-wheel.
- Press **BACKSPACE** to "unplace" the last committed track segments.
- Right-click or press **ESC** when you have finished placing a track and want to start a new one.
- You cannot accidentally connect pads that should not be wired together. Altium Designer continually monitors board connectivity and prevents you from making connection mistakes or crossing tracks.
- To delete a track segment, click it to select it. The segment's editing handles will appear (the rest of the track

will be highlighted). Press the **DELETE** key to clear the selected track segment.

- Re-routing is easy - route the new track segments, when you right-click to finish, redundant track segments are automatically removed.
- Alternatively, to slide existing routing while maintaining corner angles, **CTRL + Left-click and drag** on the segment. Use the **SHIFT+R** shortcuts to cycle conflict resolution modes while sliding a track segment.
- When you have finished placing all the tracks on your PCB, right-click or press the **ESC** key to exit placement mode.

Because we originally defined our board as being double-sided in the **PCB Board Wizard**, you could manually route your board "double-sided" using both the top and bottom layers. To do this, un-route the board by selecting **Tools»Un-Route»All** from the menus. Start interactive routing as before, to switch routing layers press the ******* key on the numeric keypad to toggle between the layers while placing tracks. Altium Designer will automatically insert a via (in accordance with the Routing Via design rule) as necessary when you change layers.

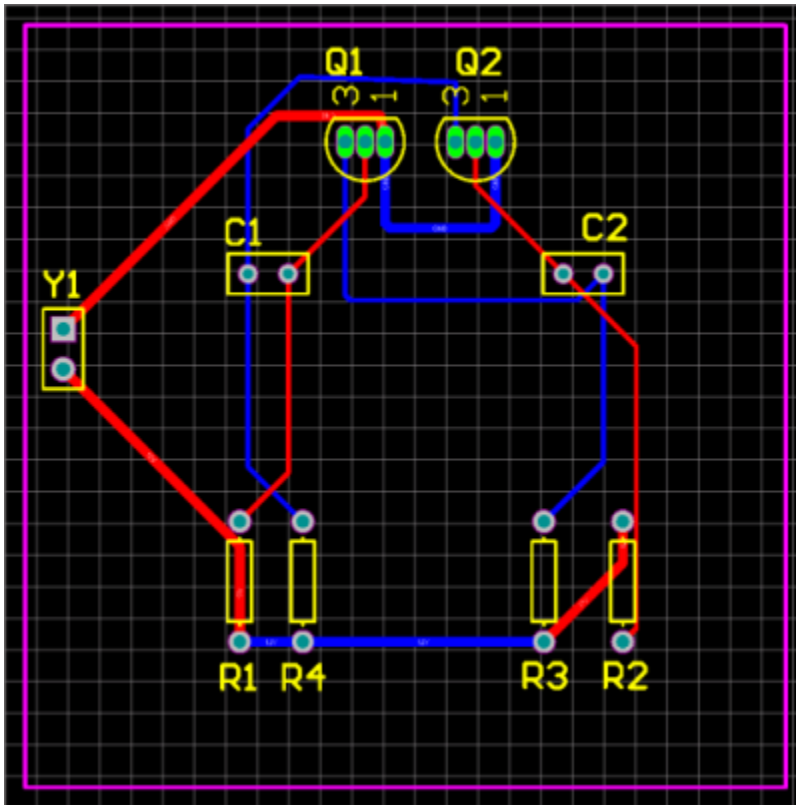
For more information on the various routing tools, refer to the [PCB Routing](#) article.

Automatically Routing the Board

Main article: [Situs Autorouting Essentials](#)

To see how easy it is to autoroute with Altium Designer, complete the following steps:

1. Un-route the board by selecting **Tools»Un-Route»All** from the menus (shortcut: **U, A**).
2. Select **Auto Route»All**. The *Situs Routing Strategies* dialog displays, the top region of the dialog displays the Routing Setup Report, warnings and errors are shown in red, always check for warnings/errors.
3. Click on **Route All** in the *Situs Routing Strategies* dialog. The Messages panel displays the process of the autorouting. The Situs autorouter is a topological autorouter, producing results comparable with that of an experienced board designer. Because it routes your board directly in the PCB editing window, there is no need to wrestle with exporting and importing route files.
4. To route the board single-sided, click the Edit Layer Directions button in the *Situs Routing Strategies* dialog, and modify the **Current Setting** field. Alternatively you can modify the Routing Layers design rule.
5. An interesting point to make, Situs prefers a challenging board, often giving better results on a dense, complex design than on a simple board. To improve the quality of the finished result, select **Auto Route»All** again, except this time select the **Cleanup** routing strategy. You can run the Cleanup strategy multiple times if required.
6. Select **File»Save** (shortcut: **F, S**) to save your board.



Fully autorouted board.

Note : The tracks placed by the autorouter appear in two colors: red indicates that the track is on the top signal layer of the board and blue indicates the bottom signal layer. The layers that are used by the autorouter are specified in the Routing Layers design rule, which was set up in the **PCB Board Wizard** . Also notice the two power net tracks running from the connector are wider, as specified by the two Width design rules you set up. Don't worry if the routing in your design is not exactly the same as shown in the figure above. The component placement will not be exactly the same, so neither will be the routing.

Verifying Your Board Design

Altium Designer is a rules-driven board design environment, in which you can define many types of design rules to ensure the integrity of your board. Typically, you set up the design rules at the start of the design process and then verify that the design complies with the rules as you work through the design, and at the end of the design process.

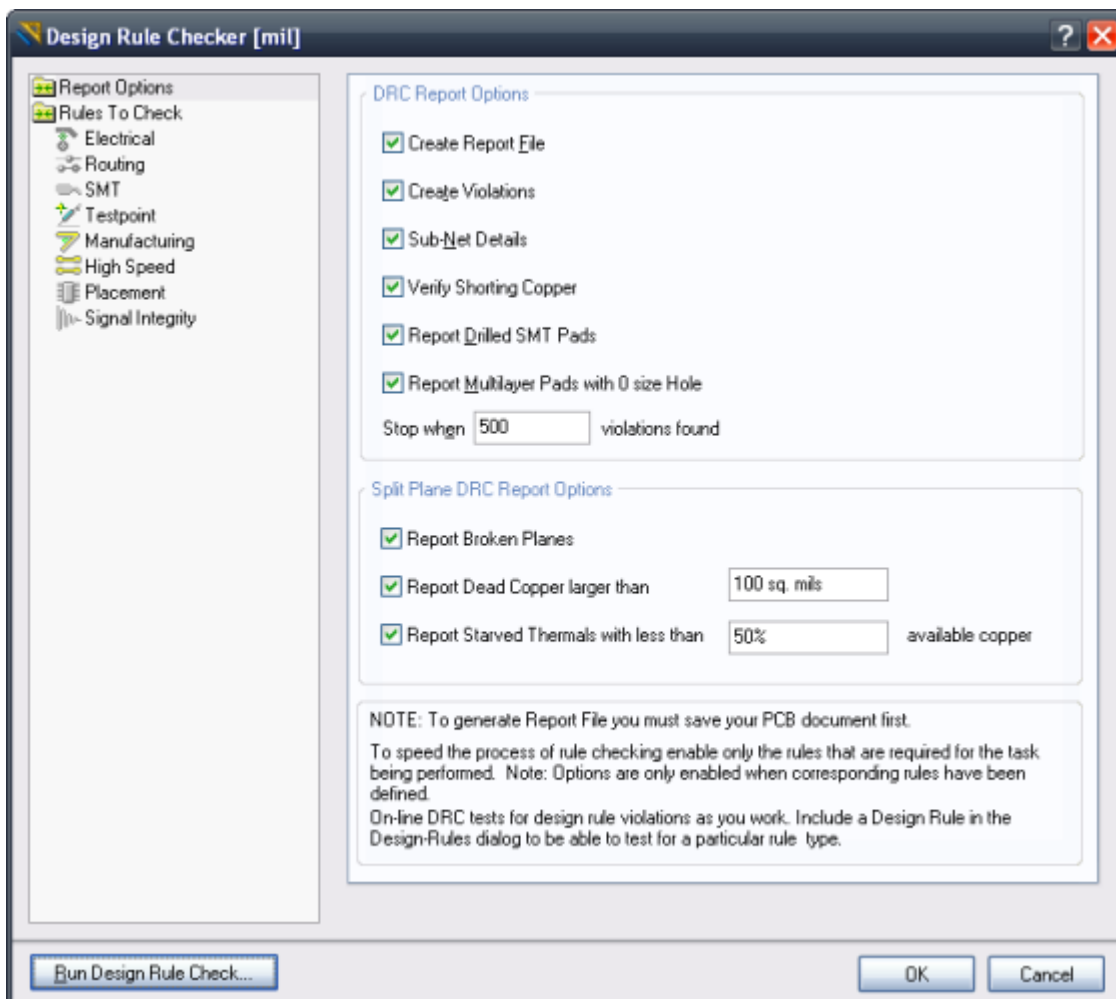
Earlier in the tutorial we examined the routing design rules and added a new width constraint rule. We also noted that there were already a number of rules that had been created by the **PCB Board Wizard**, and that there were some existing design rule violations against these default rules.

Tip:

Altium Designer supports hierarchical design rules. You can set any number of rules of the same class, each with a defined scope. The rule's priority determines the rule's precedence.

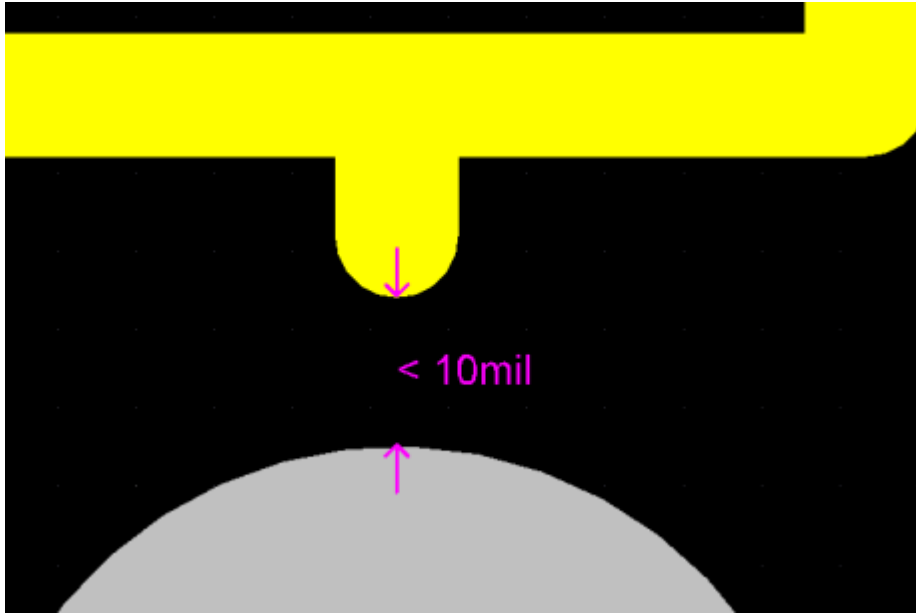
To verify that the routed circuit board conforms to the design rules, we will now run a Design Rule Check (DRC):

1. Select **Design»Board Layers & Colors** (shortcut: **L**) and ensure that **Show** checkbox next to the DRC Error Markers option in the **System Colors** section is enabled (ticked) so that DRC error markers will be displayed.
2. Select **Tools»Design Rule Check** (shortcut: **T, D**). Both the online and batch DRC options are configured in the *Design Rule Checker* dialog. Click on a category, for example **Electrical**, to see all the rules belonging to that category. For each rule type there is an **Online** and **Batch** checkbox, we will leave these set to their defaults.



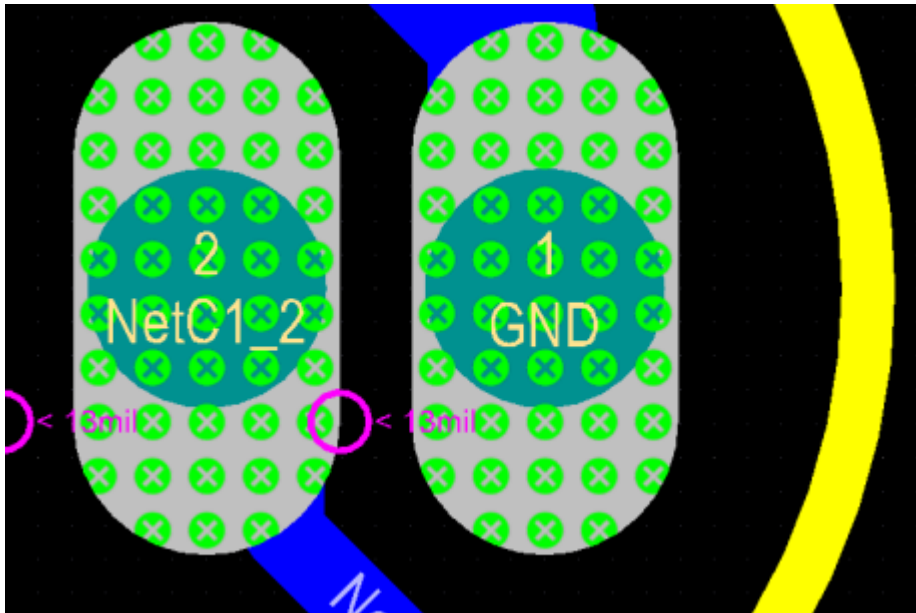
Rule checking, both online and batch, is configured in the Design Rule Checker dialog.

3. Click back on the **Report Options** icon on the left of the dialog. Leave all options in the **Report Options** section at their defaults and click the **Run Design Rule Check** button. The DRC will run and the report file **Design Rule Check - Multivibrator.html** will open. The results will also be displayed in the Messages panel. Scroll down through the report, noting that 2 types of rule violations have been detected, **Silkscreen over Component Pads** and **Clearance Constraint**.
4. Click on the **Silkscreen over Component Pads** violations, you will jump down in the report to where each violation is detailed.
5. Click on one of the **Silkscreen over Component Pads** violations, you will automatically switch to the PCB, zooming in on that specific violation. Note that the zoom level is configured in the **System - Navigation** page of the *Preferences* dialog, experiment to find a setting that suits you.



Each violation is detailed, in this case the silkscreen is less than 10mil away from the pad, the clearance specified in the design rule.

6. Each violation is detailed, for example each of the **Silkscreen over Component Pads** violations will display like the figure above, in this case indicating that the clearance is less than the 10mil specified in the applicable design rule. Note that the color of the **DRC Detail Markers** is configured in the *View Configurations* dialog.
7. To find out what the actual clearance between the silkscreen and the pad is, select **Reports»Measure Primitives** from the menus, click once on the pad, then a second time on the offending silkscreen track. Note that if your current snap grid is coarse you may not be able to click on the track, if that's the case, press **CTR L+G** and enter a smaller grid value, 5mil for example.
8. After clicking the second time, an *Information* dialog will appear, showing the distance between the silkscreen track and the pad to be something like 9.504mil.
9. To resolve this error we can either modify the footprint, increasing the separation, or we can edit the design rule, decreasing the required separation. For this tutorial we will edit the design rule, to do this select **Design»Rules** from the menus to open the *PCB Rules and Constraints Editor* dialog.
10. In the **Manufacturing** category, open the **Silkscreen Over Component Pads** rule type, and click on the existing rule.
11. Edit the **Silkscreen Over Exposed Component Pads Clearance** value, changing it from 10mil to 9mil.
12. The online DRC will run automatically, clearing those violations. We will also run the batch check, to do this select **Tools»Design Rule Check**, and click the **Run Design Rule Check** button.
13. This time the report should only show 4 Clearance Constraint violations. Click on the hyperlink in the report to show the detail for the violations, then click on one of the 4 violations to switch to the PCB and show that violation.



These pads are closer than the 13mil specified in the Clearance Constraint design rule.

- Using the same technique you used to check the gap between the soldermask and a pad, check the clearance between the pads on the transistor. The *Information* dialog should show a distance of 10.63mil.

Switch back to the PCB document and you will see that the transistor pads are highlighted in green, indicating a design rule violation.

1. Look through the errors list in the Messages panel. It lists any violations that occur in the PCB design. Notice that there are four violations listed under the Clearance Constraint rule. The details show that the pads of transistors Q1 and Q2 violate the 13mil clearance rule.
2. Double-click on an error in the Messages panel to jump to its location on the PCB.
Normally you would set up the clearance constraint rules before laying out your board, taking account of routing technologies and the physical properties of the devices. Let's analyze the error then review the current clearance design rules and decide how to resolve this situation.
3. Open the *PCB Rules and Constraints Editor* dialog (**Design»Rules**). Expand the **Electrical**, then the **Clearance** rule type. There will be one Clearance design rule, click on it to display its settings.
4. Note that this rule requires **All** objects to be away from **All** other objects, at least **13mil**. Since the clearance between the transistor pads is less than this, they generate a violation when we run a DRC.
5. We know that the minimum distance between the transistor pads is just over 10mil, so let's set up a design rule that allows the clearance constraint of 10mil for the transistors only.
6. Select the **Clearance** type rule in the **Design Rules** folder on the left of the dialog, right-click on it, then select **New Rule** to add a new clearance constraint rule.
7. Click on the new Clearance rule, Clearance_1. Change the **Name** to Clearance_Transistors, and set the **Minimum Clearance** to 10mil in the **Constraints** section.
8. The final task is to set the Scope, or **Full Query** for the rule. There are a number of ways the rule could be scoped, the most appropriate in this case would be to target the rule to any component that uses the transistor footprint. To do that, select the **Advanced (Query) option (in the upper section of the dialog)**, then click the ***Query Builder** button to open the *Building Query from Board* dialog.
9. Click **Condition/Type Operator** dropdown to **Add first condition**, and select **Associated with Footprint** from the list.
10. Set the **Condition Value** to **BCY-W3/E4** (the footprint type being used by the transistor), then click **OK** to close the dialog. The new design rule should look like the figure shown below.

Name: Comment: Unique ID:

Where The First Object Matches

☐ All
☐ Net
☐ Net Class
☐ Layer
☐ Net and Layer
☒ Advanced (Query)

Query Helper ...
Query Builder ...

Full Query
{HasFootprint ('BCY-W3/E4')}

Where The Second Object Matches

☒ All
☐ Net
☐ Net Class
☐ Layer
☐ Net and Layer
☐ Advanced (Query)


Query Helper ...
Query Builder ...

Full Query
All

Constraints

Different Nets Only

Minimum Clearance 10mil



Design rule to set the clearance for all components using a specific footprint.

11. Click **OK** to close the *PCB Rules and Constraint Editor* dialog. The online DRC will run automatically, clearing the errors.
12. To confirm that the transistor pad clearance violations have been resolved, run the batch design rule check again (**Tools»Design Rule Check**). When the report opens scroll down and confirm that there are no violations.

Design Rule Verification Report

Date : 7/10/2009
Time : 11:41:04 AM
Elapsed Time : 00:00:00
Filename : [C:\Documents and Settings\phil\My Documents\My Designs\Multivibrator\Multivibrator.PcbDoc](#)

Warnings : 0
Rule Violations : 0

Summary

Warnings	Count
Total	0

Rule Violations	Count
Clearance Constraint (Gap=10mil) ((HasFootprint("BCY-W3/E4"))),(All)	0
Short-Circuit Constraint (Allowed=No) (All),(All)	0
Un-Routed Net Constraint ((All))	0
Height Constraint (Min=0mil) (Max=1000mil) (Preferred=500mil) (All)	0
Hole Size Constraint (Min=1mil) (Max=100mil) (All)	0
Hole To Hole Clearance (Gap=10mil) (All),(All)	0
Minimum Solder Mask Sliver (Gap=10mil) (All),(All)	0
Silkscreen Over Component Pads (Clearance=9mil) (All),(All)	0
Silk to Silk (Clearance=10mil) (All),(All)	0
Net Antennae (Tolerance=0mil) (All)	0
Width Constraint (Min=12mil) (Max=12mil) (Preferred=12mil) (All)	0
Clearance Constraint (Gap=13mil) (All),(All)	0
Power Plane Connect Rule(Relief Connect) (Expansion=20mil) (Conductor Width=10mil) (Air Gap=10mil) (Entries=4) (All)	0
Width Constraint (Min=10mil) (Max=50mil) (Preferred=25mil) ((InNet("12V") OR InNet("GND")))	0
Total	0

A clean DRC report, showing that all violations have been resolved.

Well done! You have completed the PCB layout and are ready to produce output documentation. Before doing that, we'll explore Altium Designer's 3D capabilities.

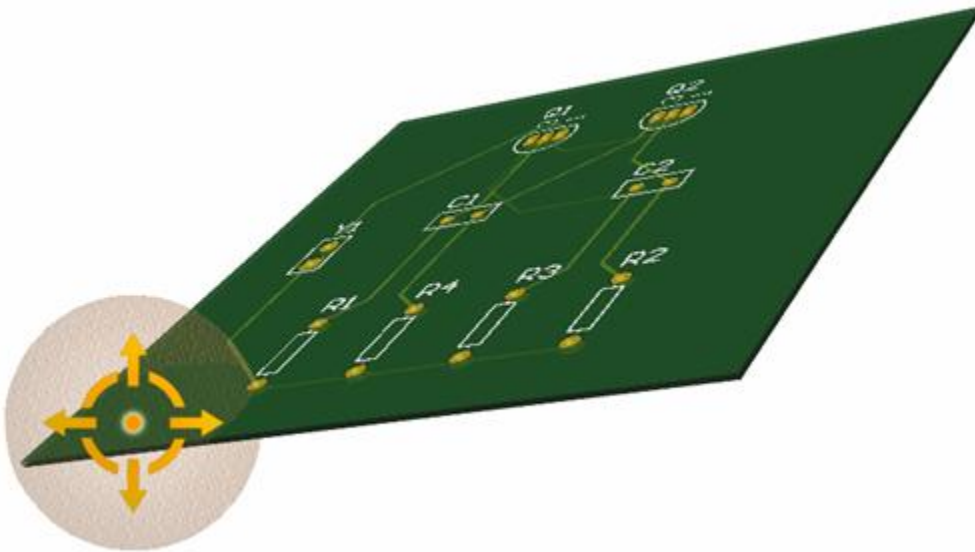
Viewing Your Board in 3 Dimensions

Now that your board design is complete, let's examine it as a 3 dimensional object. Altium Designer's 3D mode allows you to look at your board from any direction as a full 3D model. The Altium Designer 3D environment requires DirectX 9.0c and Shader Model 3, as well as a graphics card that supports them. To enable the 3D mode and to test your system, open the **PCB Editor - Display** page of the *Preferences* dialog (**DXP»Preferences**). Confirm that the Use DirectX if possible option is enabled, then click the **Test DirectX** button.

To switch to 3D, select **View»Switch To 3D** (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.

You can fluidly zoom the view, rotate it and even travel inside the board using the following controls:

- **Zooming** - **CTRL + Right-drag** mouse, or **CTRL + Roll** mouse-wheel*, or the **PAGE UP / PAGE DOWN** keys
- **Panning** - **Right-drag** mouse, or the standard Windows mouse-wheel controls.
- **Rotation** - **SHIFT + Right-drag*** mouse. Note how when you press **SHIFT** 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 using the following controls (move the mouse around to select each one):
 - Right-drag sphere when the **Center Dot** is highlighted - rotate in any direction.
 - Right-drag sphere when the **Horizontal Arrow** is highlighted - rotate the view about the Y-axis.
 - Right-drag sphere when the *** Vertical Arrow*** is highlighted - rotate the view about the X-axis.
 - Right-drag sphere when the **Circle Segment** is highlighted - rotate the view about the Z-plane.



The 3D view rotation sphere.

You can configure 3D workspace display options using the *View Configurations* dialog (shortcut: **L**). There are options to choose various surface and workspace colors as well as vertical scaling, which is handy for examining the PCB internally. Some surfaces have an opacity setting - the greater the opacity, the less 'light' passes through the surface, which makes objects behind less visible. You can also choose to show 3D bodies or render 3D objects in their (2D) layer color.

To display the components in 3D each component needs to have a suitable 3D model. You can import 3D STEP-format models into component footprints, or you can create your own component shape by placing 3D Body Objects in the footprint in the library editor.

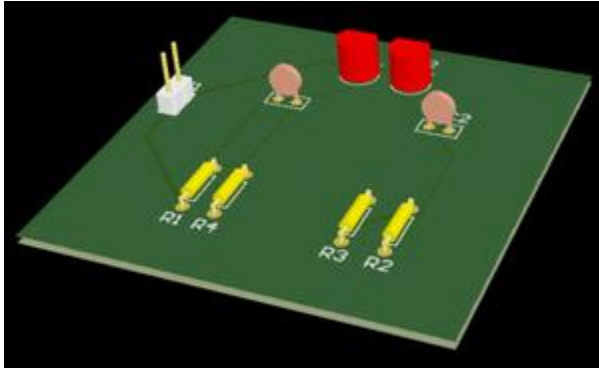
You can also export PCB documents in STEP and DWG/DXF formats for use in other programs (**File»Save Copy As**).

Note: At anytime in 3D mode you can create a clipboard snapshot of the current view, using **CTRL + C**. The image is stored on the Windows clipboard in bitmap format, ready for use in other applications.

For more information on creating 3D bodies for components, refer to the Including Three-Dimensional

Component Detail section of the [Creating Library Components](#) tutorial.

For more information on working with 3D components and integration with MCAD applications, refer to the [Integrating MCAD Objects and PCB Designs](#) tutorial. The tutorial uses this same multivibrator design.



Multivibrator PCB, complete with component 3D bodies.



Multivibrator PCB fully assembled into a two part housing assembly, inside Altium Designer.

Output Documentation

Now that you've completed the design and layout of the PCB, you will want to produce output documentation to get the board reviewed, manufactured and assembled. These documents are generally intended for a board fabricator and, because a variety of technologies and methods exist in PCB manufacture, Altium Designer has the capability to produce numerous outputs for these purposes:

[Assembly Outputs](#)

- Assembly Drawings - component positions and orientations for each side of the board.
- Pick and Place Files - used by robotic component placement machinery to place components onto the board.

Documentation Outputs

- Composite Drawings - the finished board assembly, including components and tracks.
- PCB 3D Prints - views of the board from a three-dimensional view perspective.
- Schematic Prints - schematic drawings used in the design.

Fabrication Outputs

- Composite Drill Drawings - drill positions and sizes (using symbols) for the board in one drawing.
- Drill Drawing/Guides - drill positions and sizes (using symbols) for the board in separate drawings.
- Final Artwork Prints - combines various fabrication outputs together as a single printable output.
- Gerber Files - creates manufacturing information in Gerber format.
- NC Drill Files - creates manufacturing information for use by numerically controlled drilling machines.
- ODB++ - creates manufacturing information in ODB++ database format.
- Power-Plane Prints - creates internal and split plane drawings.
- Solder/Paste Mask Prints - creates solder mask and paste mask drawings.
- Test Point Report - creates test point output for the design in a variety of formats.

Netlist Outputs

Netlists describe the logical connectivity between components in the design and is useful for transporting to other electronics design applications.

Report Outputs

- Bill of Materials - creates a list of parts and quantities (BOM), in various formats, required to manufacture the board.
- Component Cross Reference Report - creates a list of components, based on the schematic drawing in the design.
- Report Project Hierarchy - creates a list of source documents used in the project.
- Report Single Pin Nets- creates a report listing any nets that only have one connection.
- Simple BOM - creates text and CSV (comma separated variables) files of the BOM.

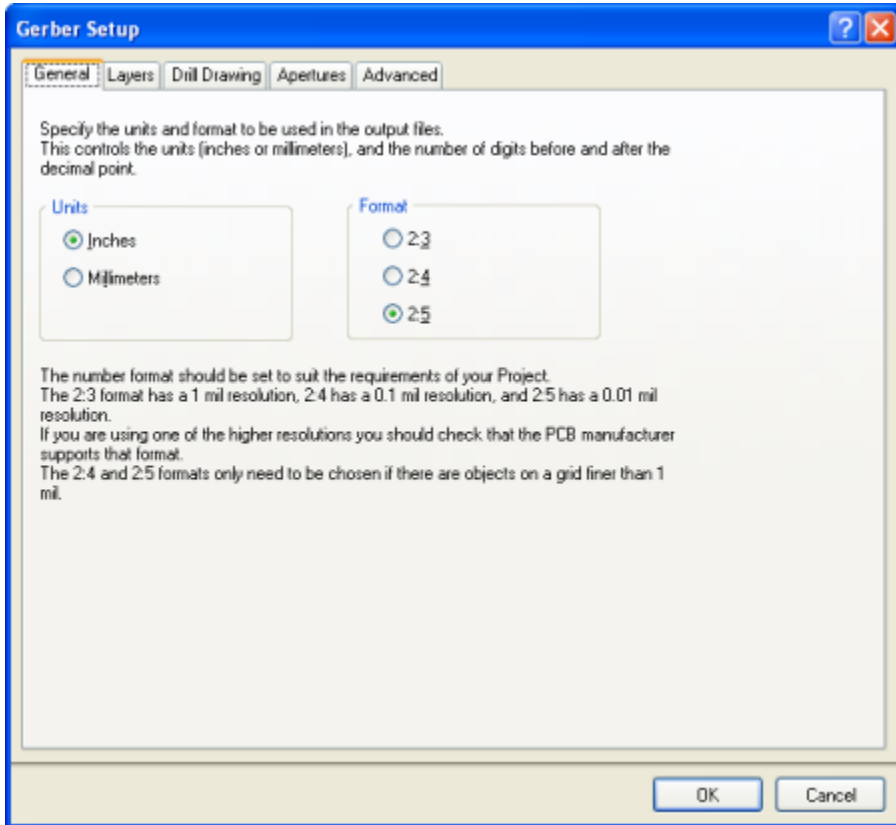
Much of the output documentation is configurable, enabling you to customize the output as necessary. As you complete more designs you may find that you often producing the same or similar output documentation for each. To support this, Altium Designer provides a mechanism called Output Job files, which can be configured for any board design, and then re-used in other designs.

For more information on using the OutputJob Editor, see [Design to Manufacturing](#).

For more information on publishing to PDF, see [Publish to PDF](#).

Generating Gerber Files

Each Gerber file corresponds to one layer in the physical board - the component overlay, top signal layer, bottom signal layer, the solder masking layers and so on. It is advisable to consult with your board fabricator to confirm their requirements before supplying the output documentation required to fabricate your design.



To create the manufacturing files for the tutorial PCB:

1. Select **File»Fabrication Outputs»Gerber Files**. The *Gerber Setup* dialog displays.
2. Click the **Layers** tab, then the **Plot Layers** button and select Used On. Click OK to accept the other default settings.
3. The Gerber files are produced and the CAM Editor opens to display the Gerber files. The Gerber files are stored in the {{\Project Outputs}} folder, which is automatically created in the folder where your project files reside. Each file has the file extension added that corresponds to the layer name, eg. **Multivibrator.GTO** for Gerber Top Overlay. These are added to the Projects panel in the **Generated CAM Documents** folder.

Similarly, use the **File»Fabrication Outputs»NC Drill Files** command to open the *NC Drill Setup* dialog (accepting the defaults for this exercise) to produce the NC drill data.

Tip:

If you do not want output files to automatically open when they are created, select Project»Project Options, click on the Options tab and disable the Open outputs after compile option.

For more information about Fabrication Outputs and configuring your Gerber Outputs, see [Fabrication Outputs](#)

▪

Creating a Bill of Materials

To create a Bill of Materials (BOM) for the tutorial PCB.

1. Select **Reports»Bill of Materials**. The *Bill of Materials for PCB Document* dialog displays.

Grouped Columns	Show	Comment	Description	Designator	Footprint	LibRef	Quantity
Comment	<input checked="" type="checkbox"/>	20nF	Capacitor	C1, C2	RAD-0.1	Cap	
Footprint	<input checked="" type="checkbox"/>	2N3904	NPN General Purpose Amplifier	Q1, Q2	BCY-W3/E4	2N3904	
		100K	Resistor	R1, R2	AQAL-0.3	Res1	
		1K	Resistor	R3, R4	AQAL-0.3	Res1	
		Header 2	Header, 2-Pin	Y1	HDR1X2	Header 2	

Export Options
File Format: Microsoft Excel Worksheet (*.xls)
☐ Add to Project
☒ Open Exported

Excel Options
Template: BOM Manufacturer.XLT
☒ Relative Path to Template File

Supplier Options
Production Quantity: 1
☐ Round up Supplier Order Qty to cheaper price break
☐ Use cached pricing data in parameters if offline

☐ Force Columns to View
☐ Include Parameters From Database
☒ Include Parameters From PCB

Menu Export... OK Cancel

2. Use this dialog to build up your BOM. Enable the Show option for each column you want to include in the report.
3. Select and drag column headings from the **All Columns** list to the **Grouped Columns** list to group components by that data type in the BOM. For example, to group by **Footprint**, select Footprint in the **All Columns** list and drag it into the **Grouped Columns** list. The report will be sorted accordingly.
4. Enable the **Open Exported** option, select **CSV** for the **File Format**, then click the **Export** button to create and immediately open the BOM file in your CSV viewer (eg. Microsoft Excel). There are many options available for BOM and other reports, providing a high degree of flexibility in defining and organizing your reports. Close the dialogs.

For more information about Report Outputs and configuring your Bill of Materials, see [Report Outputs](#)

For more information about Bill of Materials, see [Generating a Custom Bill of Materials](#).

Congratulations! You have completed the PCB design process.

Further Explorations

This tutorial has introduced you to just some of the powerful features of Altium Designer. We've captured a schematic, and designed and routed a PCB, but we've only just scratched the surface of the design power provided by Altium Designer. Once you start exploring Altium Designer, you will find a wealth of features to make your design life easier.

To demonstrate the capabilities of the software, a number of example files are included. You can open these examples in the normal way by selecting **File»Open** from the menus and then navigating to the *Examples* folder of your Altium Designer installation. As well as the board design examples in this folder, there are a number of sub-folders with examples that demonstrate specific features of Altium Designer.

Check out the *Circuit Simulation* sub-folder to explore Altium Designer's analog and digital simulation capabilities. As well as analog examples that demonstrate various circuit designs, such as amplifiers and power supplies, there are mixed-mode examples, a mathematic function example, and an example that includes linear and non-linear dependent sources and even a vacuum tube example.

With faster logic switching and design clock speeds, the quality of the digital signals becomes increasingly important.

Altium Designer includes a sophisticated signal integrity analysis tool that can accurately model and analyze your board layout. The signal integrity requirements such as impedance, overshoot, undershoot, and slope are defined as PCB design rules, and then tested during the standard design rule check. If there are nets that you need to analyze in more detail, you can select **Tools»Signal Integrity** to pass the design to the Signal Integrity Analyzer, where you can perform reflection and cross talk simulations. The results are displayed in an oscilloscope-like waveform analyzer, where you can examine the performance and take measurements directly from the waveforms.

See Also

[**Quickstart - PCB Layout**](#)

[**The Altium Designer Environment**](#)

[**Quickstart - the Environment & Projects**](#)

[**Multi-sheet design**](#)

[**Connectivity and Multi-Sheet Design**](#)

[**Multi-Channel Design Concepts**](#)

[**Component, Model and Library Concepts**](#)

[**Creating Library Components**](#)

[**Design to Manufacturing**](#)

[**Generating a Custom Bill of Materials**](#)

[**Using Components Directly from Your Company Database**](#)

Tutorial - Integrating MCAD Objects and PCB Designs

Shortcut Keys