

# Configuring and Compiling the Project

Project-specific settings are configured in the Options for PCB Project dialog (**Home Tab → Project → Options**). The project options include the error checking parameters, a connectivity matrix, the Comparator setup, ECO generation, output paths and netlist options, Multi-Channel naming formats, Default Print setups, Search Paths, and project-level Parameters. These settings are used when you compile the project.

Project outputs, such as assembly, fabrication outputs and reports are set up from the **Outputs** tab of the Ribbon. These settings are also stored in the Project file so they are always available for this project.

## Setting Up the Connection Matrix

When the design is compiled a list of the pins in each net is built in 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. Click on a colored square to change the setting, continue to click to move to the next check-level.

### Exercise

Let's make a change to the Connection Matrix:

1. If it is not already open, go to *Project→Project Options* to open your *Options for Project* dialog, and go to the **Connection Matrix** tab.
2. To change one of the settings simply click directly on 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.
3. Our circuit contains only Passive Pins (on resistors, capacitors and the connector) and Input Pins (on the transistors). Let's set the connection matrix to detect unconnected passive pins, every pin in our design should be connected, so this will let us know if we missed anything when wiring. 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.

4. Click on this intersection box until it turns orange, so that an error will be generated for unconnected passive pins when we compile the project. We will purposely create an instance of this error later to check it since I am sure you wired your design together perfectly 😊

## 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. In general, the defaults on this page will always be used. If you need more detailed control, then you can selectively control the comparator using the individual comparison settings.

For this tutorial it is sufficient to confirm that the **Ignore Rules Defined in PCB Only** option is enabled at the bottom of this dialog.

## Compiling the Project to Check for Errors

When we compile our project, it is checked for drafting and electrical errors based on the settings we have defined for our project. Any detected warnings, errors or fatal errors will then be displayed on the *Messages* panel. If we **double-click** on a Compiler generated message, CircuitStudio will quickly navigate to these items within our design to make the process of finding and resolving these errors very fast and efficient.

### Exercise

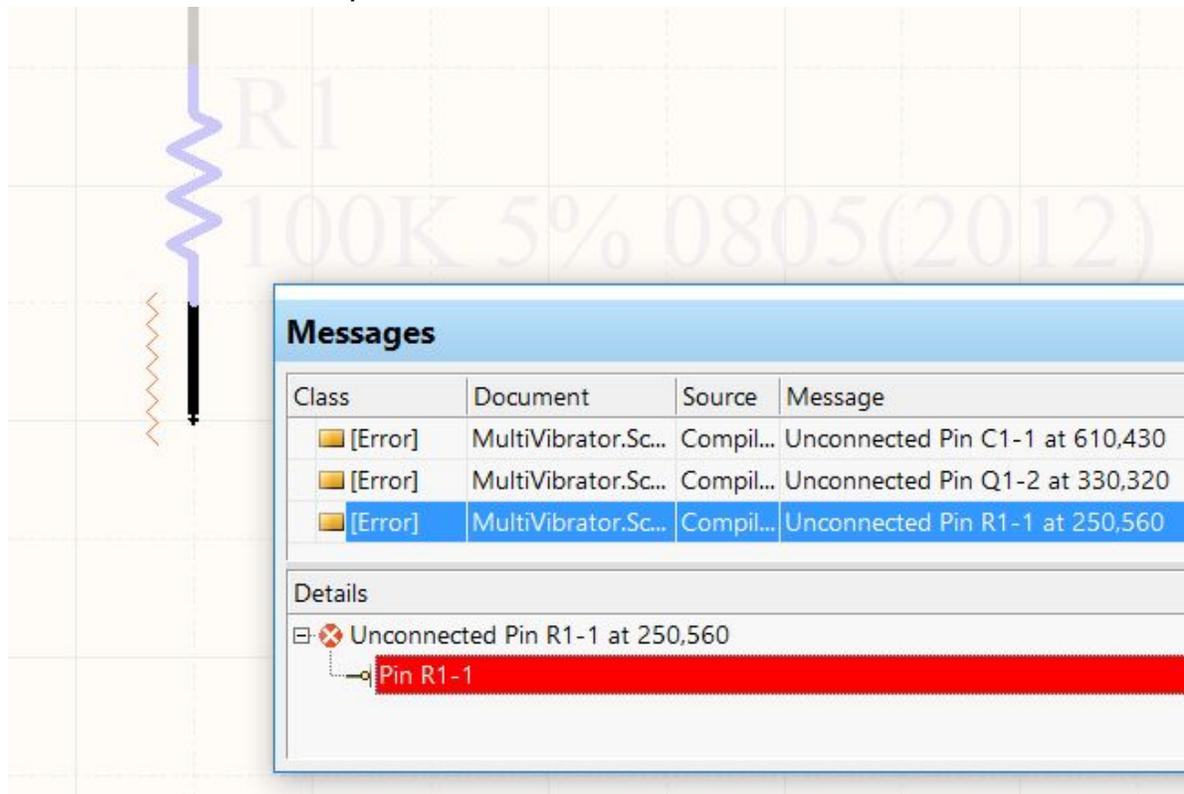
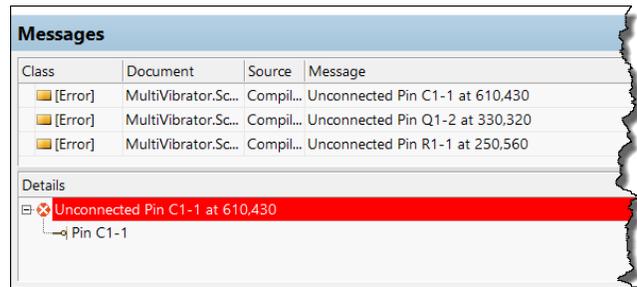
1. Press **OK** to apply and close your Project Options.
2. To compile the Multivibrator project, Right-click on the project within the Projects panel and select *Compile PCB Project*.
3. Once the project is compiled, all warnings and errors will be displayed in the Messages panel. The panel will only appear automatically if there are errors or fatal errors detected, to open it manually click the **View Tab → Messages**. Double-click on an entry in the panel to interrogate an error.
4. Since all of our circuits are drawn correctly 😊 the Messages panel should not contain any errors. If there were errors, you would work through each one, checking your circuit and ensuring that all wiring and connections are correct.

Since we did such a great job on our schematic capture, we are forced to

deliberately introduce an error into the circuit and recompile the project:

## Exercise

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 (**Home Tab** → **Project** → **Compile**) to check for errors. The *Messages* panel will display error messages indicating you have unconnected pins in your circuit. Because they are errors, the *Messages* panel will open automatically. If these were just Warnings, the messages panel will not be forced into view.
4. When you **Double-click** on an element in the Details view of the *Messages* panel, CircuitStudio will navigate to the element, making it easy to find and resolve issues within your schematic



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.

## Exercise

1. Make the schematic sheet the active document.
2. Press the *Undo* button in the upper left of the CircuitStudio window or use the shortcut **[CTRL+Z]**. The wire you deleted previously should now be restored.
3. To check that the *Undo* was successful, recompile the project to check that no errors are found. The Messages panel should show no errors.
4. Save the schematic and the project file as well, *File*→*Save All*.



Now we have completed and checked our schematic, it is time to prepare the blank PCB we created earlier.

---

## COPYRIGHT

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

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

## TRADEMARKS

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