Multi-channel Design with a Flat Project

Introduction

Altium Designer offers many methods for repeating circuitry within a single design (usually referred to as a “Multi-channel Design”). Users can set up a Project as a hierarchical design and utilize Sheet Symbols to replicate the circuits within the design. Multiple sheet symbols can reference the same underlying schematic document, or a single sheet symbol can use Altium Designer’s REPEAT keyword to instantiate the circuit as many times as needed. The main advantage here is that any change to the underlying circuit need only be made once, and that change will immediately be seen in every instance. This is a very powerful and efficient method of working with multi-channel designs.

Working with these repeated circuits in the PCB document is also extremely efficient. Altium Designer will automatically create a “Room” for each instantiation of the circuit. The user needs to then place and route just one of the circuits. Using the Copy Room Format feature, the placement and routing data can then be automatically copied to each subsequent circuit. This makes the layout of repeated circuits extremely simple, no matter how many there are!

There are many users, however, who have not worked with hierarchical designs and feel more comfortable using a flat design methodology. Some projects may just be too simple to warrant setting the entire design as hierarchical. Whatever the case, there are legitimate occasions where circuit replication is necessary and the layout of that circuit also needs to be replicated, but the Project is set up as flat.

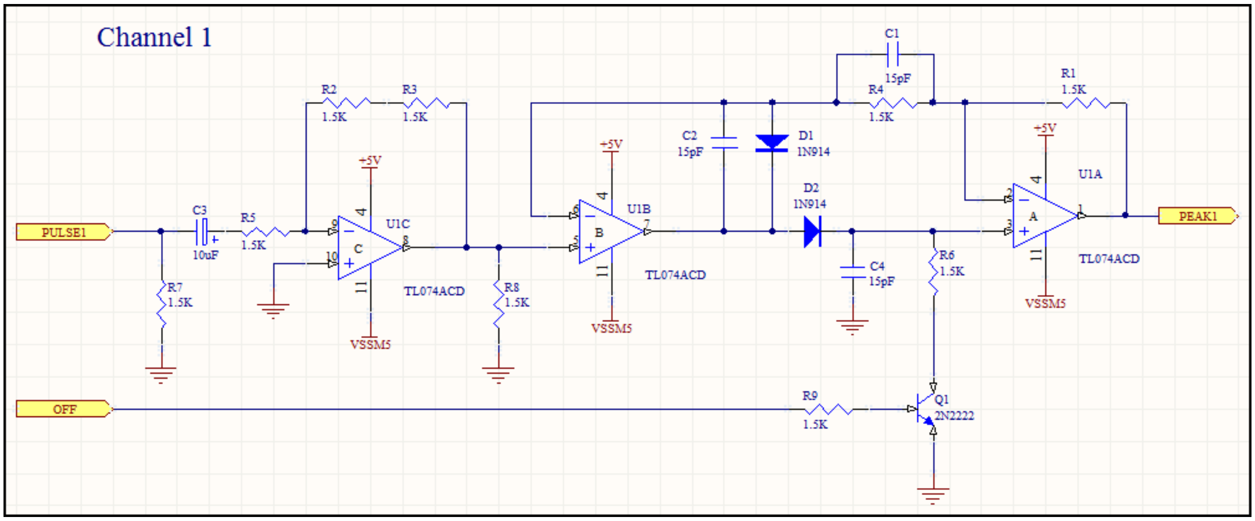
How can this be done?

There are two scenarios that need to be addressed, and each has its own steps in order to accurately set up the PCB document to enable reuse of the placement and routing data. The first possibility is that the each repeated circuit is large enough to take up most, if not all, of a schematic sheet, such that a circuit repeated three times would require three schematic documents. This may be the case where a large power supply for a system requires triple redundancy. A second possibility is that the repeated circuit is small – perhaps just three or four components – but it is used many, many times (maybe a small LED circuit). Creating separate sheets for each circuit in this case is obviously not very efficient. It may be more reasonable to have that circuit repeated multiple times as part of just one schematic document. How each method translates to PCB is surprisingly different. Both methods are covered in this document.

Flat Design Using Multiple Sheets

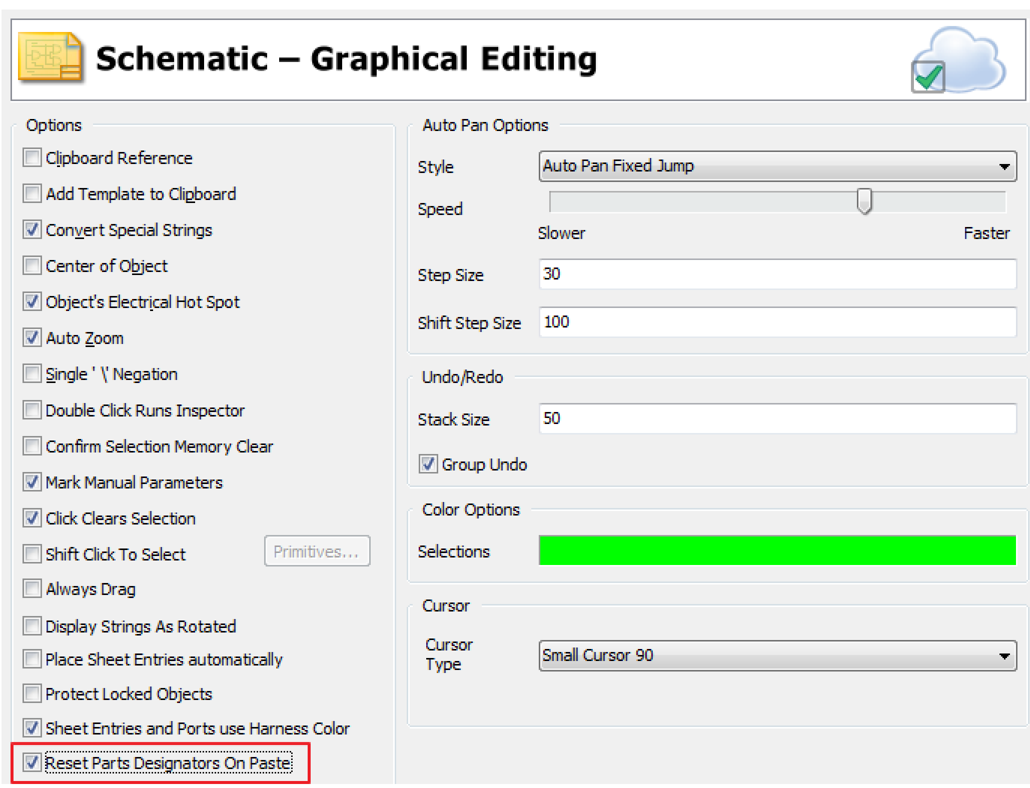
By far the easier of the two methods is the use of a separate sheet for each circuit. The reason for this is that Altium Designer will automate more of the process. In fact, there will be only one bit of manual intervention required by the user in the PCB document during this process.

In the following example, this circuit is to be replicated just once, creating a Channel1 and Channel2:



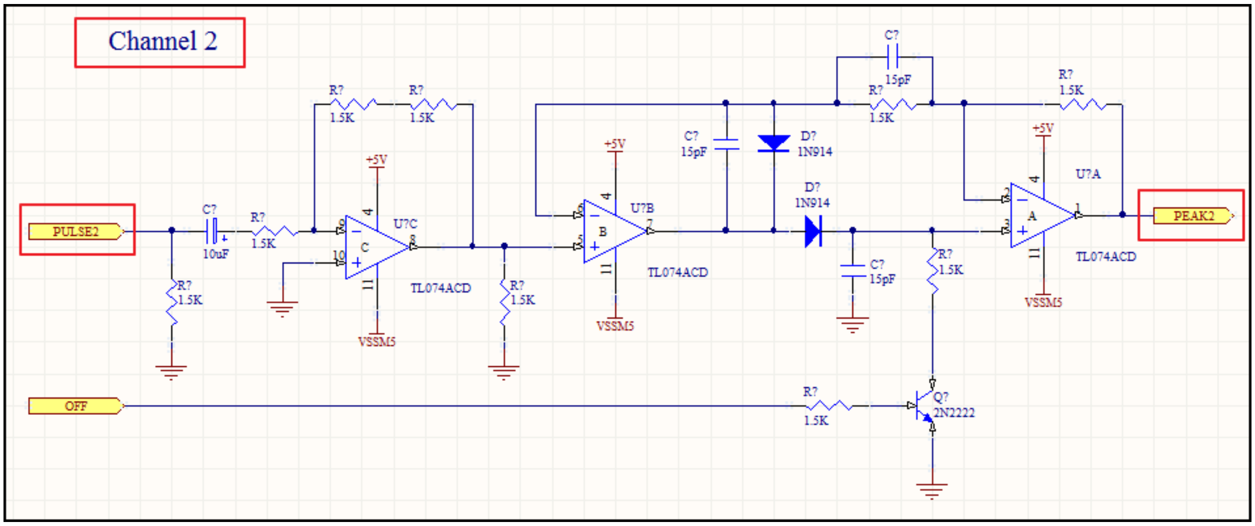
Schematic Creation

Start by creating the initial circuit on the first schematic sheet (named “Channel_1.SchDoc” here) of a PCB Project. Then add a second schematic sheet (“Channel_2.SchDoc”) to the Project. The Channel_1 circuit now needs to be copied and pasted to Channel_2. If the reference designators have already been set for the base circuit, first go to the **DXP** menu and then to **Preferences**. Then expand the Schematic group and select the Graphical Editing section. In the Options area, enable the “Reset Parts Designators On Paste” option.



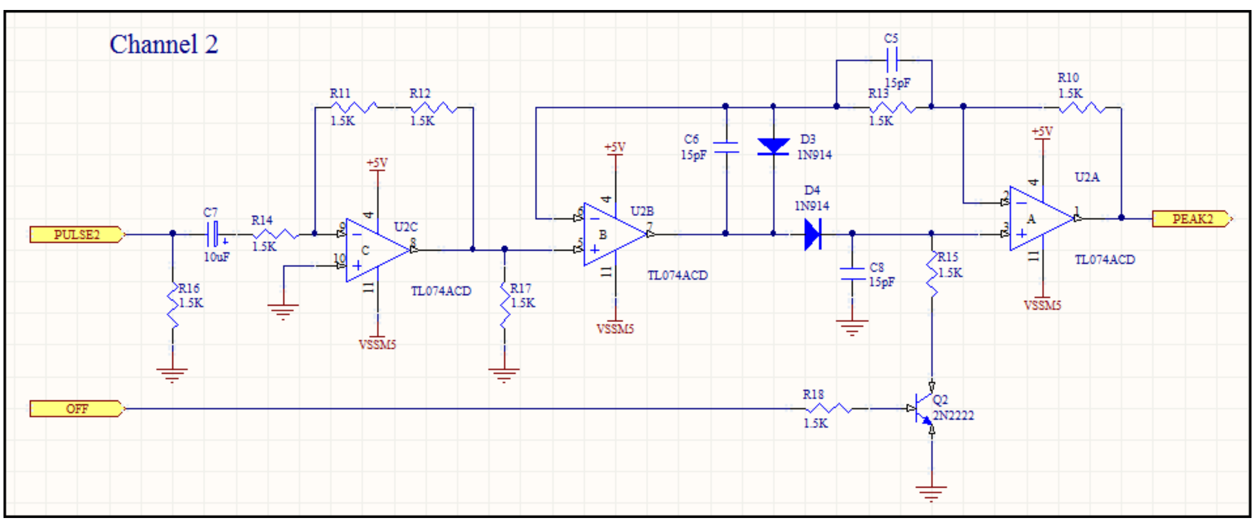
**This option is only available in Altium Designer 10 Build 10.600.22648 or later.

Group-select the base circuit, copy it, and then paste it to Channel_2. Make any edits necessary to the second circuit to ensure proper connectivity with the rest of the design. In this case, the “Pulse1” and “Peak1” ports have been made unique, as has the “Channel 1” text identifier.



Add whatever additional sheets are necessary for the design. *However, it is important that no further additions or changes be made to any of the repeated schematic sheets.* Doing so may cause the Copy Room Formats feature to fail later on. For this project, a third sheet (“Connector.SchDoc”) will be added to include a connector with the design.

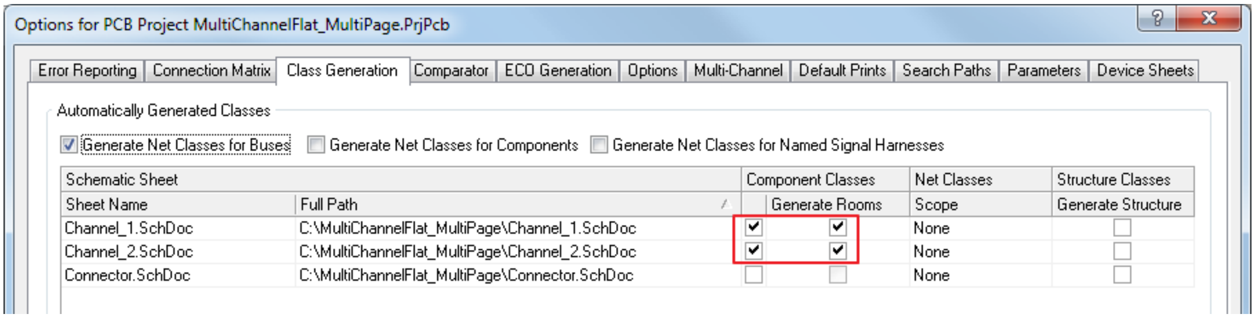
Since the reference designators on Channel_2 have all been reset to ?, run **Tools/Annotate Schematics Quietly** to set the designators.



Another important note has to do with multi-part components. Notice that in this example, only three parts from the op-amp are being used – A, B and C – while part D is not. Make sure that when reference designator annotation is done, unused parts from one circuit are not used in another. There needs to be consistency between each physical circuit so that the routing can match. Here, U1A, U1B, and U1C are used in Channel 1, but U1D does not get used for Channel 2. Instead, U1D is left unused and Channel 2 starts off at U2A.

Project Options Setup

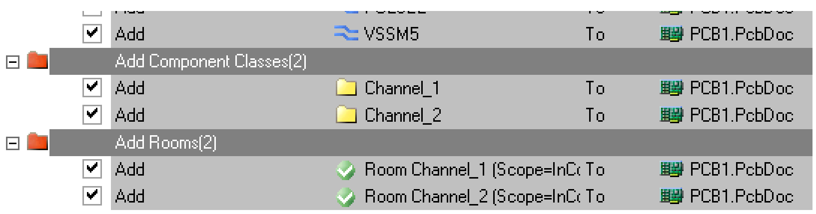
The next step is to set the Project Options to automate the Component Class and Room generation. Go to **Project/Project Options** and switch to the Class Generation tab.



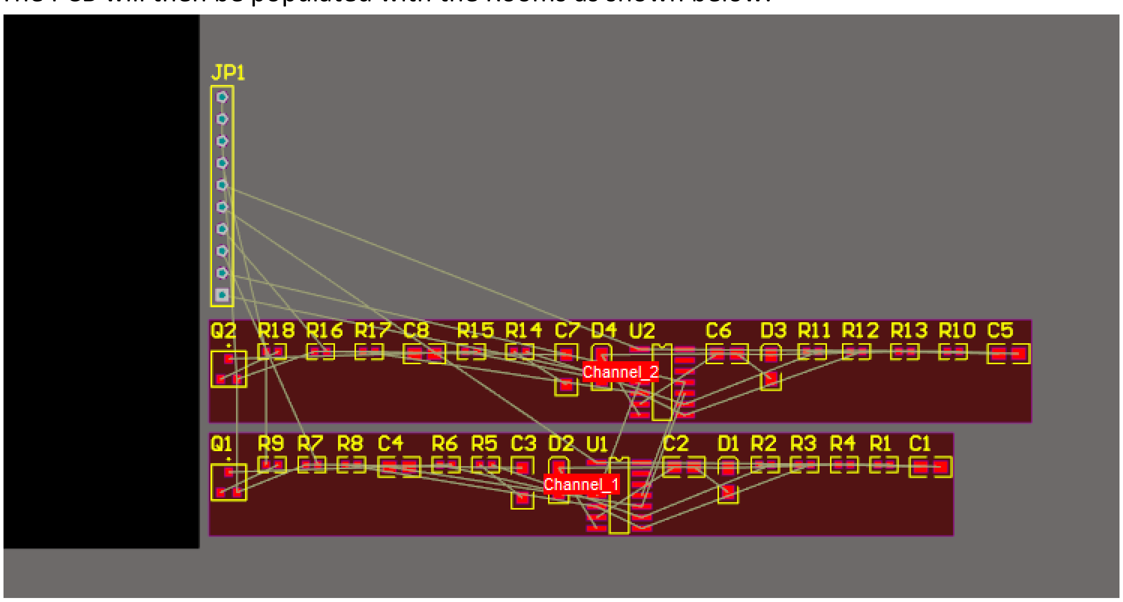
Ensure that the checkboxes for Component Classes and Generate Rooms are enabled for all multi-channel sheets. Any other sheets are optional. Close the Project Options dialog, and then save all schematic documents as well as the Project file.

PCB Layout

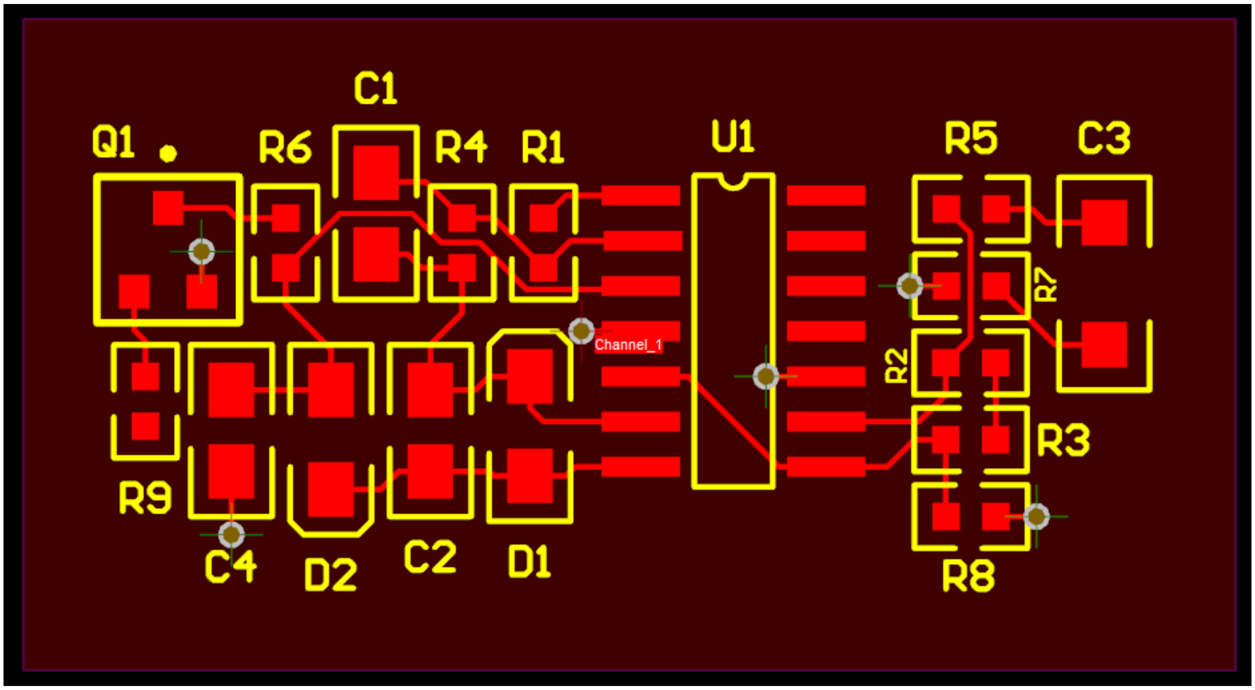
Create and save a new PCB file, then use **Design/Import Changes…** to populate the board. Ensure that the ECO includes the creation of the Component Classes and Rooms. If not, recheck the Project Options setup done previously.



The PCB will then be populated with the Rooms as shown below:



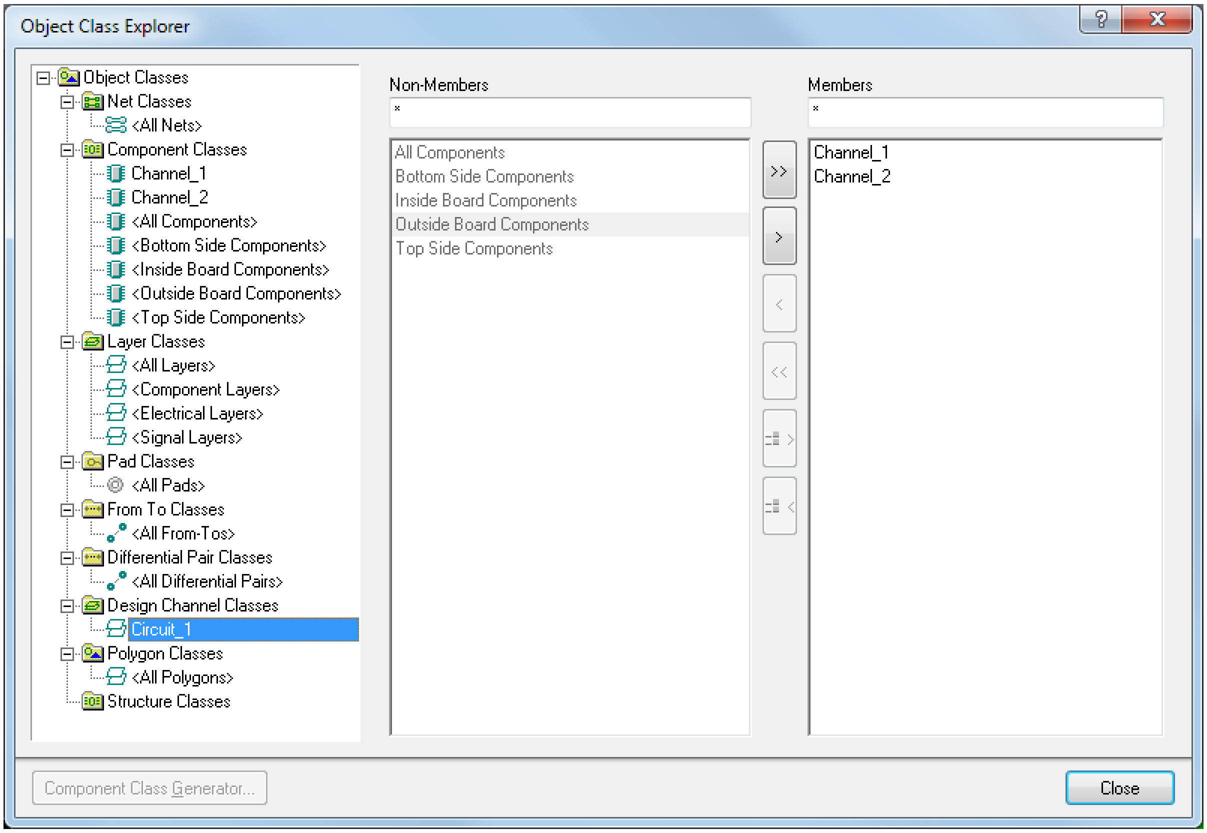
Move the Channel_1 Room into the board area, then place and route it as desired. Resize the Room outline if necessary.



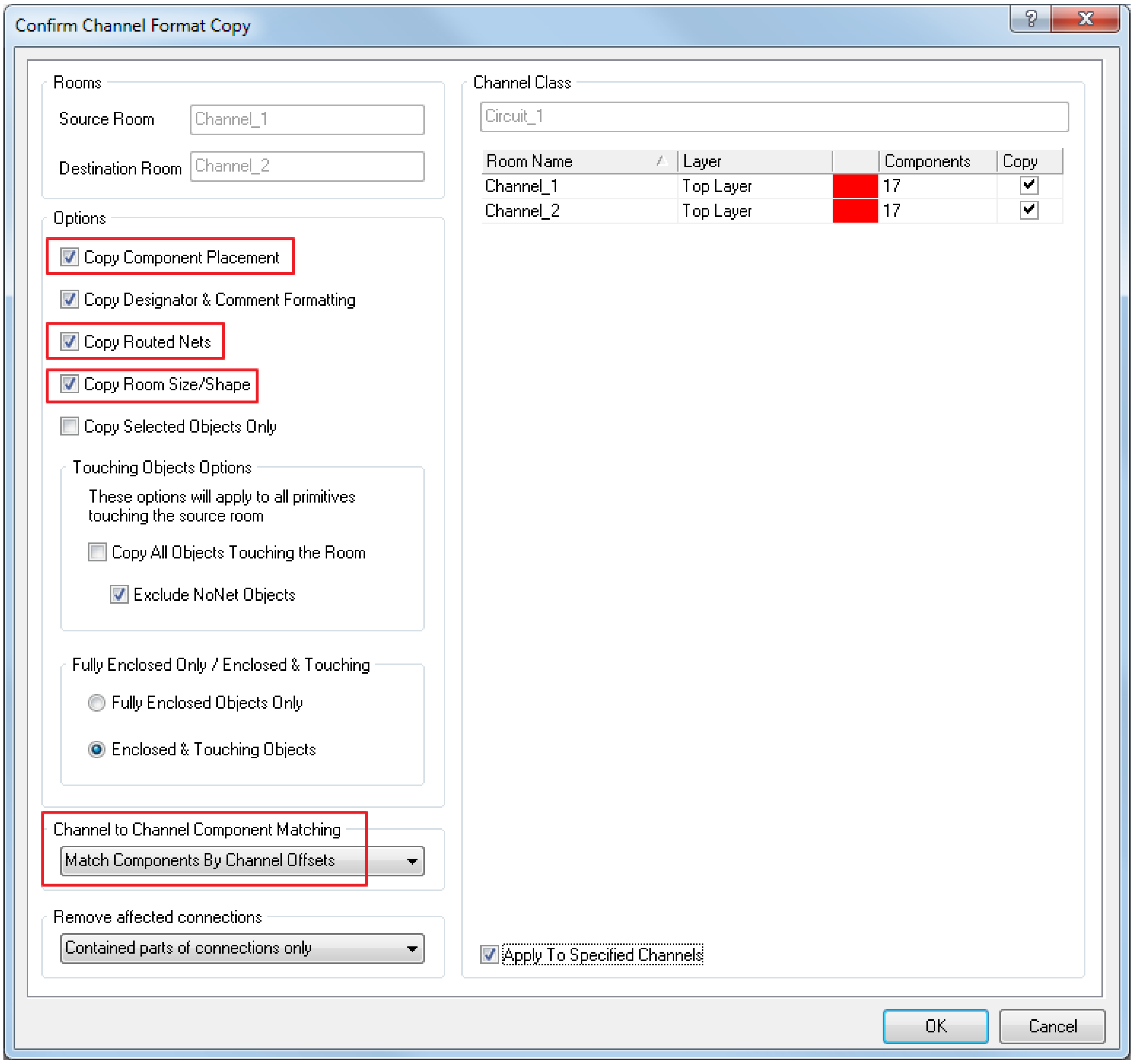
The next part of the process is to use the Copy Room Formats feature to replicate the placement and routing. But before that happens, Altium Designer needs to be told that Channel_1 and Channel_2 are the same circuit type. This is done by creating a Design Channel Class. When a hierarchical structure is used to replicate circuits, the system inherently knows that the circuits are the same based on the fact that the sheet symbol references the same circuit multiple times. Since these circuits were merely copied and pasted, this information is not automatically created. It is possible that one of the circuits was modified by the user so that there is no longer match between them, in which case replication of layout information may not be possible. Since no modifications were made to either circuit in this case, replication can proceed.

Go to the **Design/Classes** menu. Notice that there is a Component Class for each channel. These were automatically created via the Project Options setting, and they are also used to define the contents of each Room. Near the bottom of the Object Classes list is an entry for "Design Channel Classes." Right-click on that group and select **Add Class** which will create "New Class." Right-click the "New Class" name, select **Rename Class**, and change the name to "Circuit_1." This renaming step is optional, but if there is more than one circuit *type* being replicated, it will make it easier to keep track of them.

The members of a Design Channel Class are Component Classes. Notice that the "Channel_1" and "Channel_2" Component Classes are listed in the Non-Members list. Select them both, and then click the arrow to move them to the Members list:

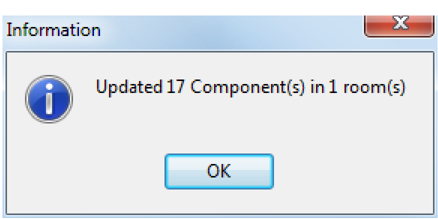


Close the dialog. Set the display of the board such that both Rooms are visible. Go to **Design/Rooms/Copy Room Formats**. The cursor will now change to a large cross-hair and the Status Bar instructs the user to choose the Source Room. Click anywhere inside the “Channel_1” Room. The Status Bar now instructs the user to choose the Destination Room. Click anywhere inside the “Channel_2” Room. The “Confirm Channel Format Copy” dialog opens presenting several copying options, as well as a list of all Rooms in the Design Channel Class that are available to copy to. Ensure that Channel_2 has the Copy checkbox enabled. If not, the “Apply to Specified Channels” checkbox may need to be enabled to access the Copy checkbox.

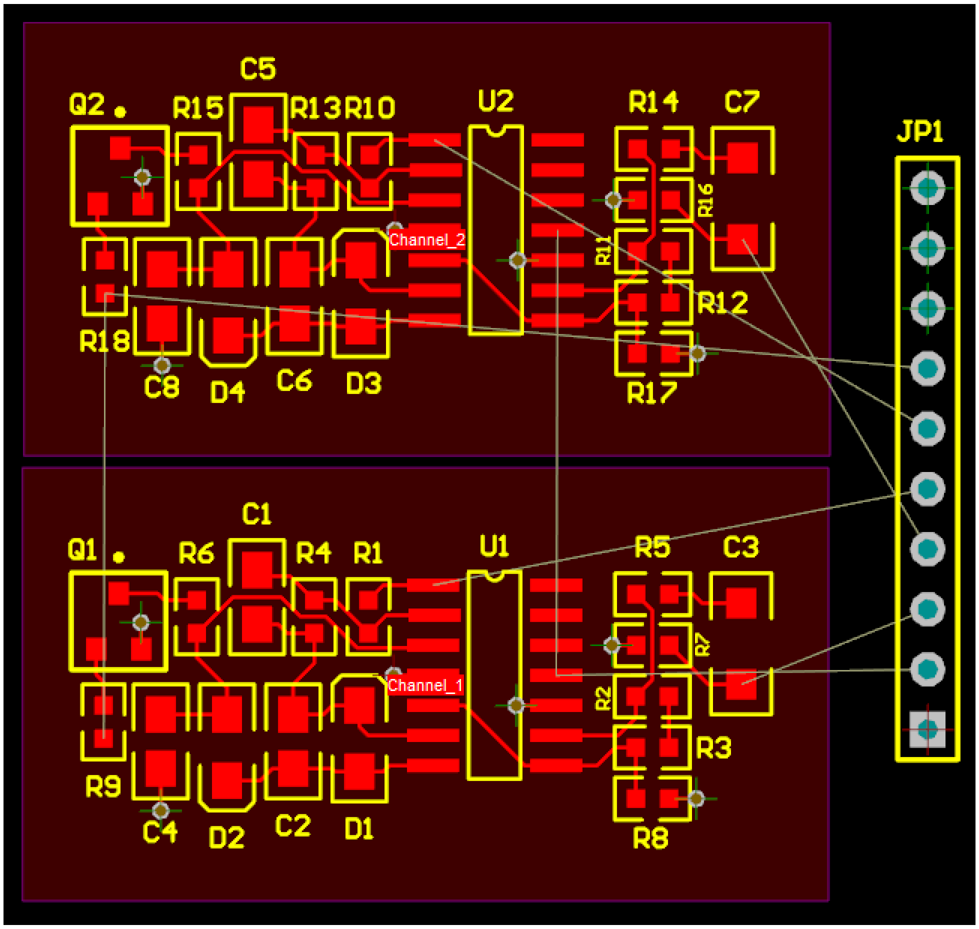


In the Options area, ensure that Copy Component Placement, Copy Routed Nets, and Copy Room Size/Shape are enabled. Also ensure that Channel to Channel Component Matching is set to “Match Components By Channel Offsets.” Channel Offsets will be discussed in more detail for the second replication method.

Clicking **OK** will run the copy routine. The system will look for matching components and connections and duplicate the placement, routing, and Room shape as best as possible.



The Channel_2 Room outline now has the identical shape, and the placement and routing from Channel_1 has been copied to Channel_2. It can now be moved to the desired location on the board.

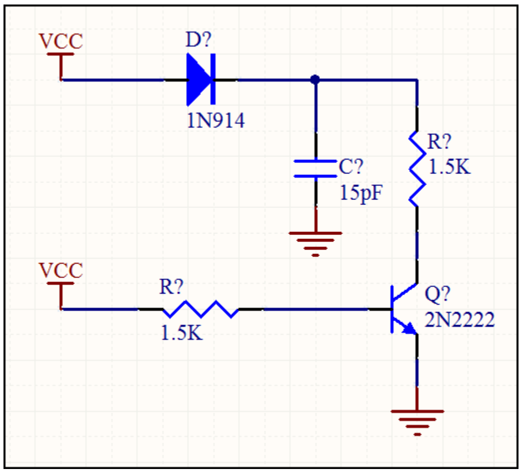


The final connections between Rooms and from the Rooms to the rest of the design can now be completed.

Flat Design Using a Single Sheet

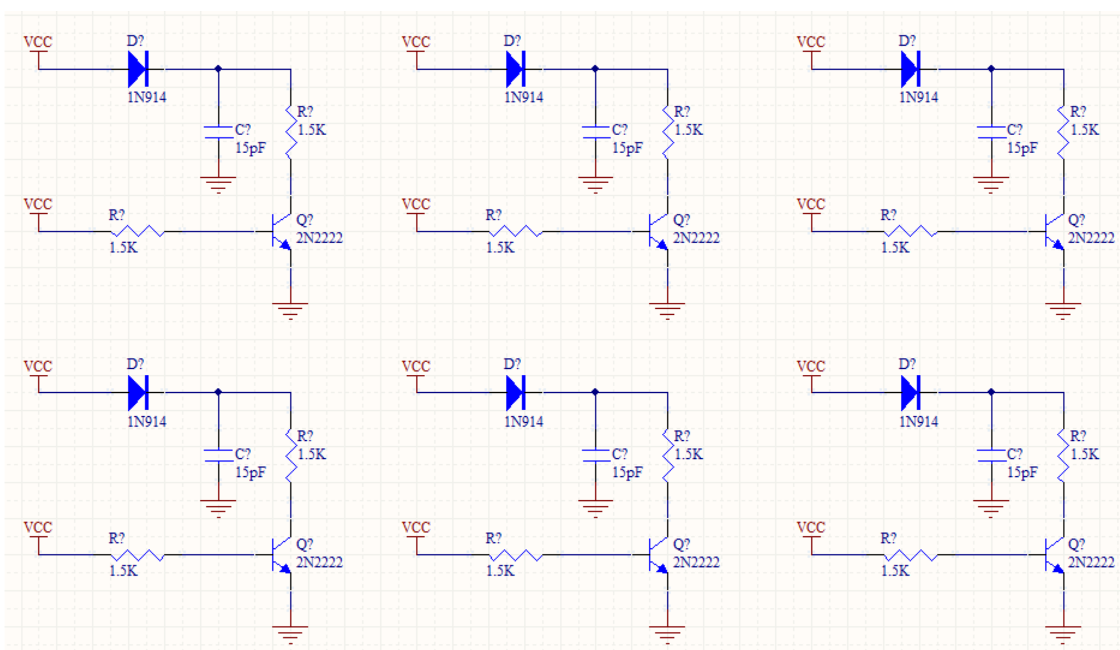
The second multi-channel situation will make use of a much smaller circuit copied and pasted many times within the same sheet. In this case, it would not be very efficient to create a separate sheet for each circuit as in the previous example. As mentioned, though, this method requires a few more manual steps in order for Copy Room Formats to function correctly.

This Project will be a very simple design, consisting of six instantiations of the following circuit (plus one connector):

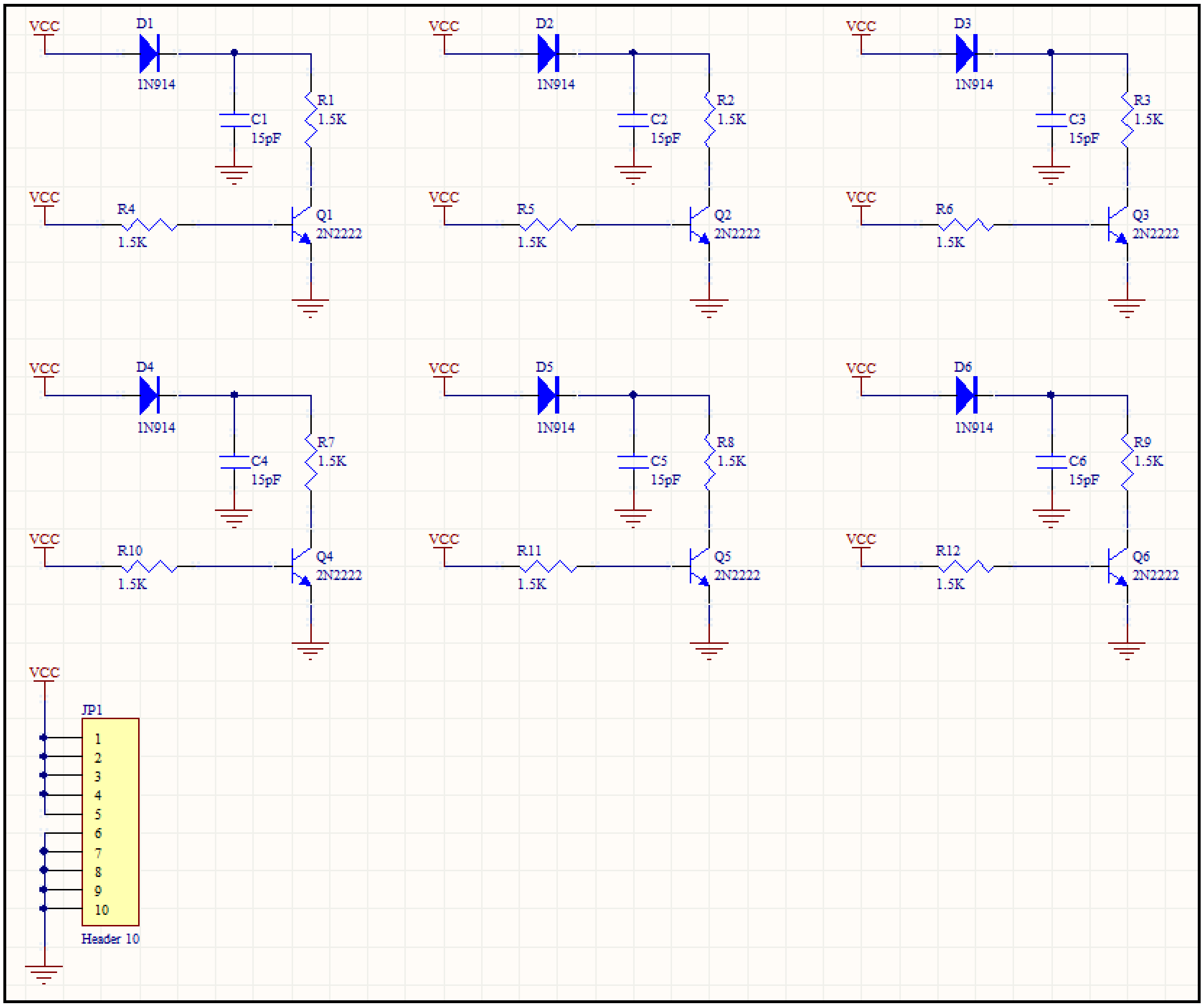


Schematic Creation

Start by creating the base circuit and leave the reference designators at their default ? state. Group-select the circuit, then use the **Edit/Rubber Stamp** function to place five more copies of the circuit:



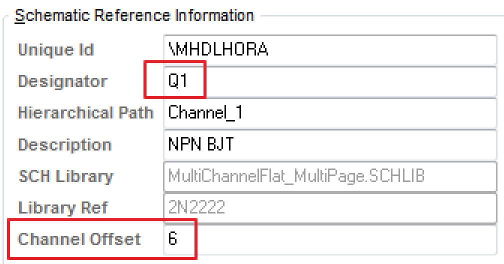
In the previous example, it was important that other components NOT get added to any of the multi-channel sheets. This has to do with Channel Offset values which will be discussed later. In this example, however, it IS acceptable to place other components. The complete design here adds a connector. Use **Tools/Annotate Schematics Quietly** (or any other annotation method) to set the reference designators.



Channel Offset Values

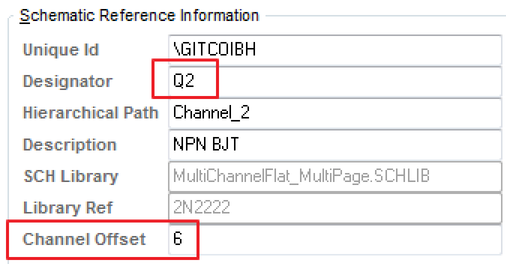
Before continuing with the next step, the concept of a Channel Offset needs to be introduced. The main way the Copy Room Formats function attempts to match components from Room to Room is by looking to see if two components share the same Channel Offset. This is an integer value that Altium Designer places on each component as it is passed to PCB, and it is essentially the component's relative physical position within the schematic sheet.

Looking back at the previous example, the Channel Offset values (accessible in the PCB document, in a component's Properties) for Q1 and Q2 match:



Schematic Reference Information

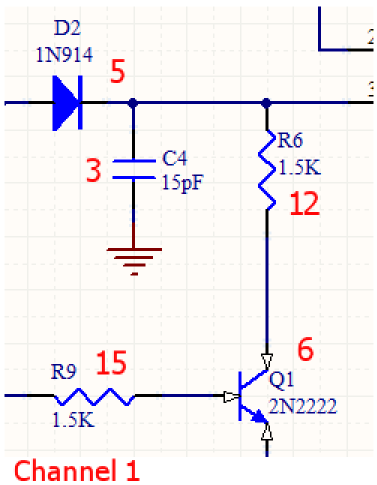
Unique Id	\MHDLHORA
Designator	Q1
Hierarchical Path	Channel_1
Description	NPN BJT
SCH Library	MultiChannelFlat_MultiPage.SCHLIB
Library Ref	2N2222
Channel Offset	6

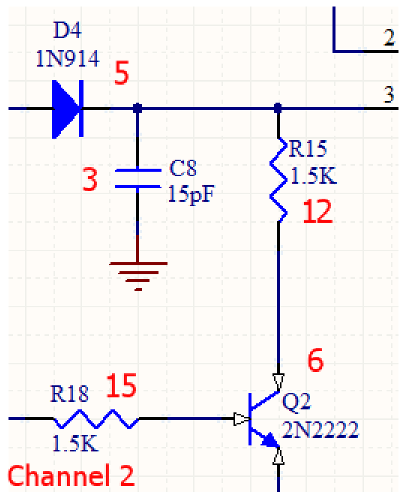


Schematic Reference Information

Unique Id	\GITCOIBH
Designator	Q2
Hierarchical Path	Channel_2
Description	NPN BJT
SCH Library	MultiChannelFlat_MultiPage.SCHLIB
Library Ref	2N2222
Channel Offset	6

This is because the circuits on the Channel_1 and Channel_2 sheets are identical, so the positions of Q1 and Q2 are the same on each sheet. The Channel Offsets match for each like component in Channel_1 and Channel_2 (red text):





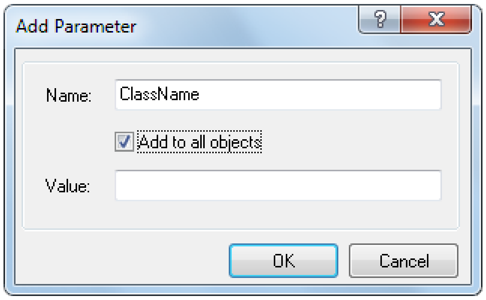
Channel Offset values are sequentially applied to all components on a schematic, so on the single sheet example all components will get unique Channel Offset values. However, that will not allow Copy Room Formats to match components from circuit to circuit. Therefore, the Channel Offset values will need to be manually adjusted in the PCB file. This is easy to do, but it is important to keep the *relative order* of the components within the copied circuits as they were and not make any changes to their placement or reference designators. Channel Offsets will be discussed again later.

Component Class Creation

With the multi-sheet method, Component Classes for the replicated circuits were created automatically because of the settings in Project Options. With a single sheet, the automated class would include *all* of the components on the page, but the Rooms need to be based on just the individual circuits. Therefore, the Component Classes must be manually created on the schematic sheet.

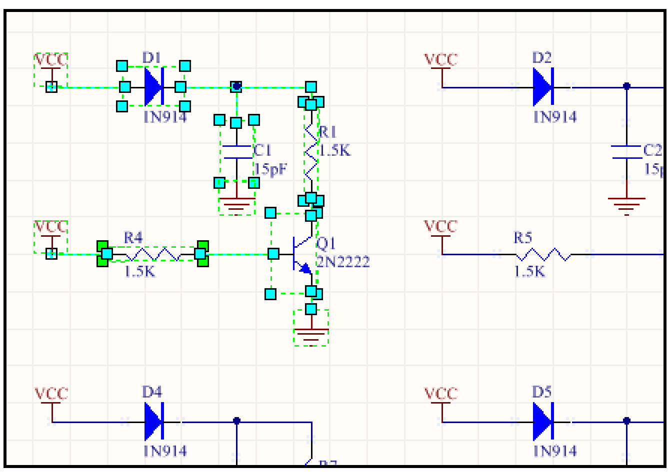
A user-defined Component Class is created by adding a Parameter to each component called “ClassName” with the value being the name of the class as it will appear in the PCB. Of course, editing the Properties of each and every component in the schematic would take some time. Altium Designer has a couple of options to add the “ClassName” parameter information to groups of components, and both will be used here for purposes of demonstration.

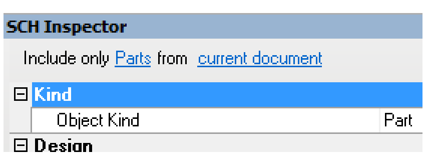
Go to **Tools/Parameter Manager**. Set the Options dialog to include only the Parameters from parts. Click **Add Column…** to add a new Parameter to every component in the design. Enter “ClassName” In the Name field and enable the “Add to all objects” checkbox. Leave the Value field blank.



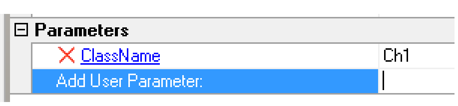
Click **Accept Changes (Create ECO)** then click **Execute Changes** to complete the addition of the Parameters. Click **Close** to dismiss the ECO dialog.

Now each circuit will need to be labeled with a unique ClassName so they each create their own Component Class in the PCB. Group-select the entire first circuit. Open the SCH Inspector panel (**View/Workspace Panels/SCH/SCH Inspector** or F11). Pin the panel in place. Set the filter at the top of the panel to “Include only Parts from current document.”





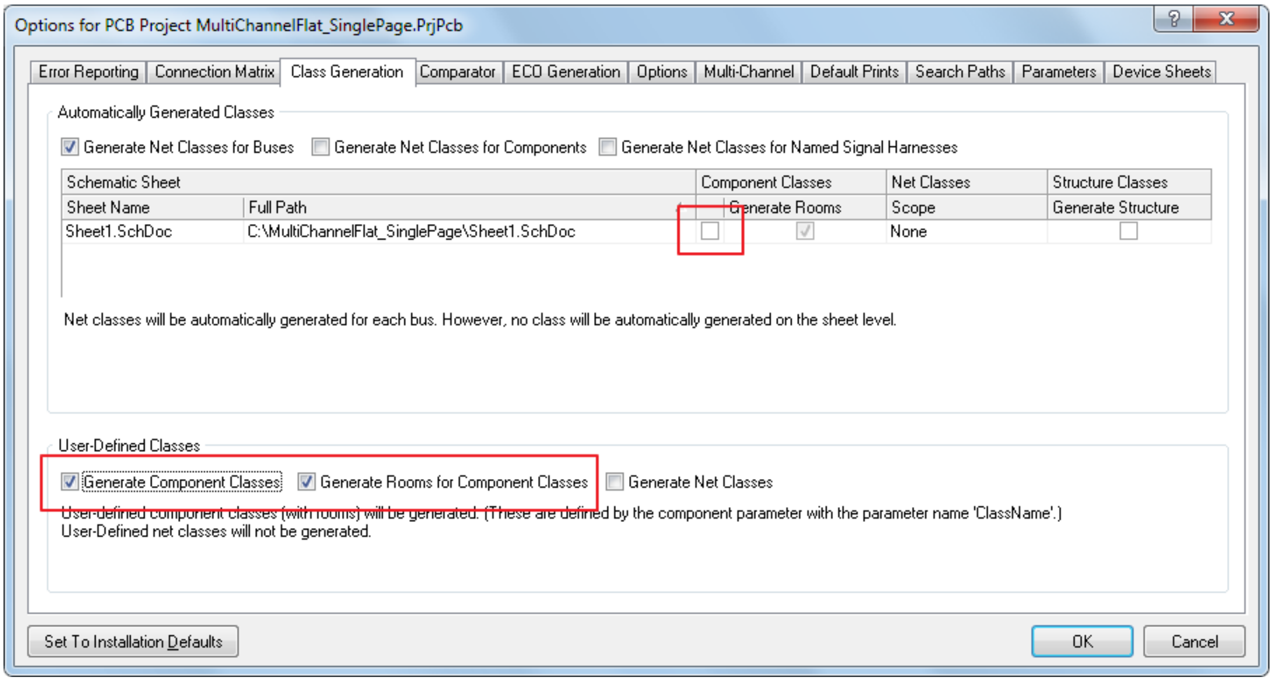
Scroll to the bottom to locate the Parameters section. Set the value of the “ClassName” parameter to “Ch1” and hit the Enter or Tab key.



Group-select the next circuit on the schematic sheet and set the “ClassName” parameter in the Inspector panel to “Ch2.” Repeat for all of the circuits through “Ch6.”

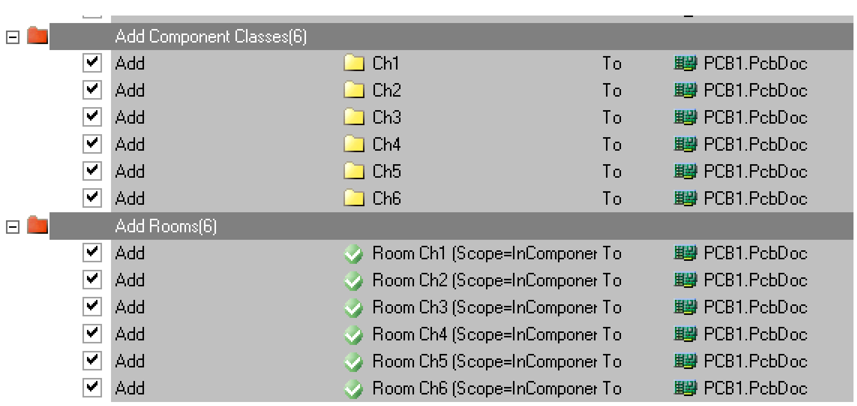
**The “ClassName” parameter and values could have been created entirely using the Inspector panel. However, it required a bit less typing to add “ClassName” once using the Parameter Manager, so that method was used here.

Before transferring the schematic information over to the PCB, there’s one last step in setting the Class Generation tab of the Project Options. In this case, the automatic Component Class generation should be disabled. However, the “Generate Component Classes” and “Generate Rooms for Component Classes” options need to be enabled in the User-Defined Classes section:

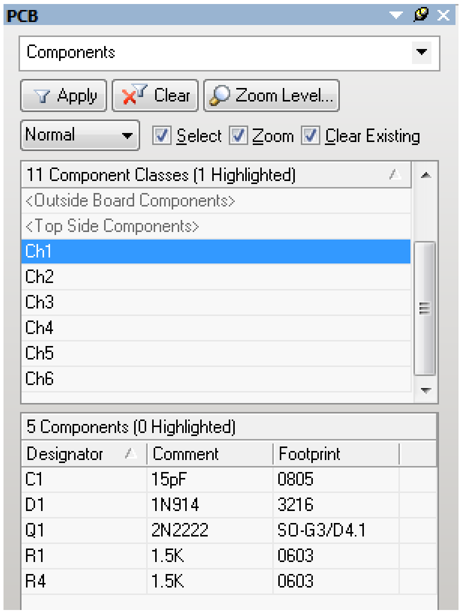


PCB Layout

Create and save a new PCB file, then use **Design/Import Changes…** to populate the board. Ensure that the ECO includes the creation of the Component Classes and Rooms. If not, recheck the existence of the “ClassName” parameters and the Project Options setup done previously.



Open the PCB panel (**View/Workspace Panels/PCB/PCB**), and set the pull-down filter to Components. Enable the “Select” checkbox. The Component Classes area should show the “Ch1” through “Ch6” classes. Select the “Ch1” class and notice that the contents are components from the Ch1 circuit on the schematic (enabling **Tools/Cross Select Mode** in the PCB editor will also select the components in the schematic document, if open).

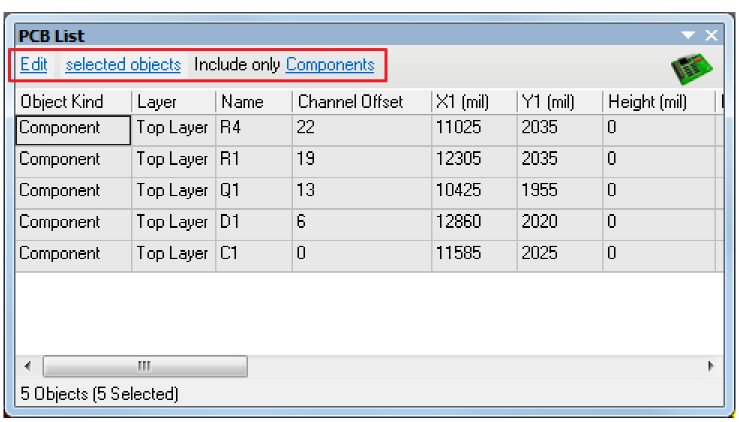


The components and their associated Rooms will be stacked outside the bottom right of the board area. Use **Design/Rooms/Move Room** (or simply click and drag the mouse inside the Room boundary but not on a component body) to spread the Rooms apart.

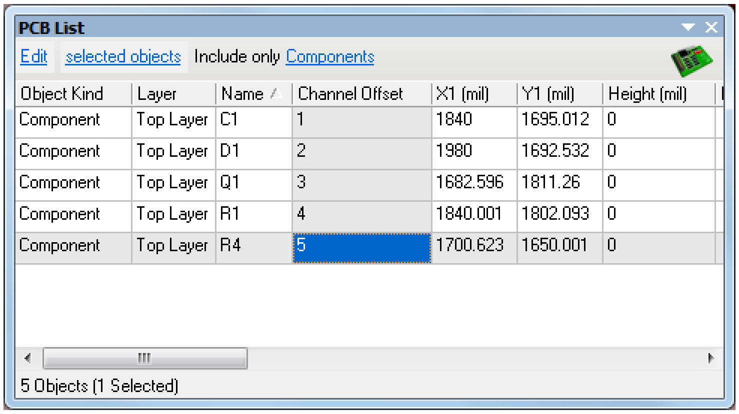
Setting the Channel Offset Values

The next step is the crucial step to this process: setting the Channel Offset values. As previously mentioned, the Copy Room Formats function will only find like components whose Channel Offsets match. This needs to be manually.

Using the PCB panel as mentioned above, select the "Ch1" class (ensuring first that the "Select" checkbox is enabled). Then open the PCB List panel (**View/Workspace Panels/PCB/PCB List)** and set the top filter to "Edit selected objects Include only Components."

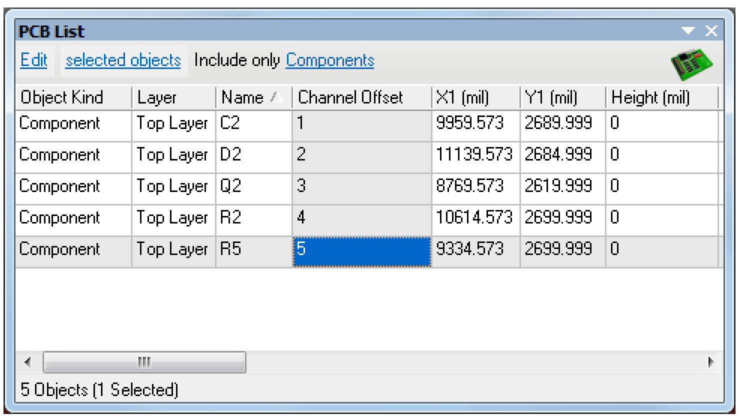


Now sort the list by the reference designator by clicking the "Name" field header. C1 should now appear at the top of the list. Next, click in the "Channel Offset" cell for C1, type the number 1 and hit Enter. This will set C1's Channel Offset value to 1 and bring you down to the next component (D1). Type 2 then Enter and continue down the list until all components are sequentially numbered.



Object Kind	Layer	Name	Channel Offset	X1 (mil)	Y1 (mil)	Height (mil)
Component	Top Layer	C1	1	1840	1695.012	0
Component	Top Layer	D1	2	1980	1692.532	0
Component	Top Layer	Q1	3	1682.596	1811.26	0
Component	Top Layer	R1	4	1840.001	1802.093	0
Component	Top Layer	R4	5	1700.623	1650.001	0

Leave the PCB List panel open and return to the PCB panel and select the "Ch2" class. The components from that class should now populate the List panel. Again, sort the list by reference designator by clicking the "Name" field header. Use the same Channel Offset values as the Ch1 components. Sorting the components this way will ensure that the Channel Offset values will be the same for the matching components in each circuit.

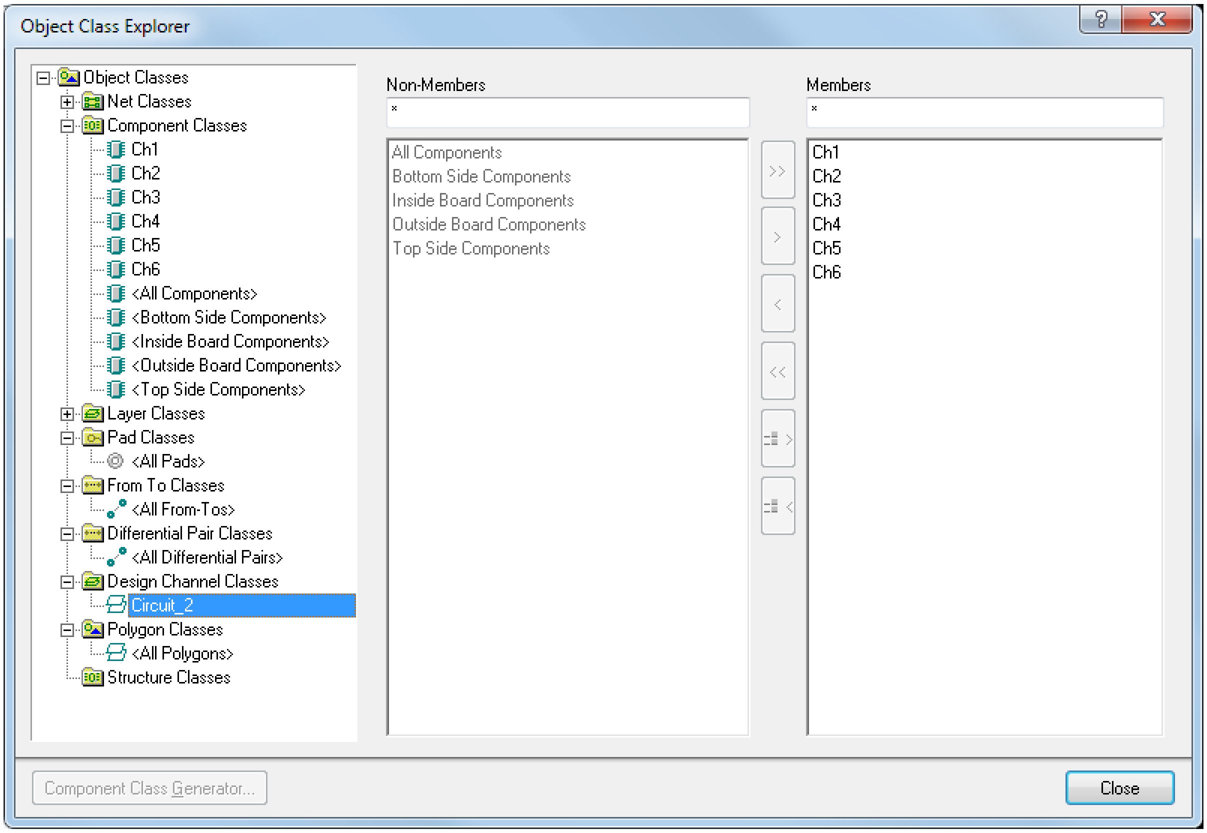


Object Kind	Layer	Name	Channel Offset	X1 (mil)	Y1 (mil)	Height (mil)
Component	Top Layer	C2	1	9959.573	2689.999	0
Component	Top Layer	D2	2	11139.573	2684.999	0
Component	Top Layer	Q2	3	8769.573	2619.999	0
Component	Top Layer	R2	4	10614.573	2699.999	0
Component	Top Layer	R5	5	9334.573	2699.999	0

It is worth noting that the Channel Offset values themselves aren't important as much as making sure that the values are *the same* for the matching components. Setting them in sequential order just makes the process easier.

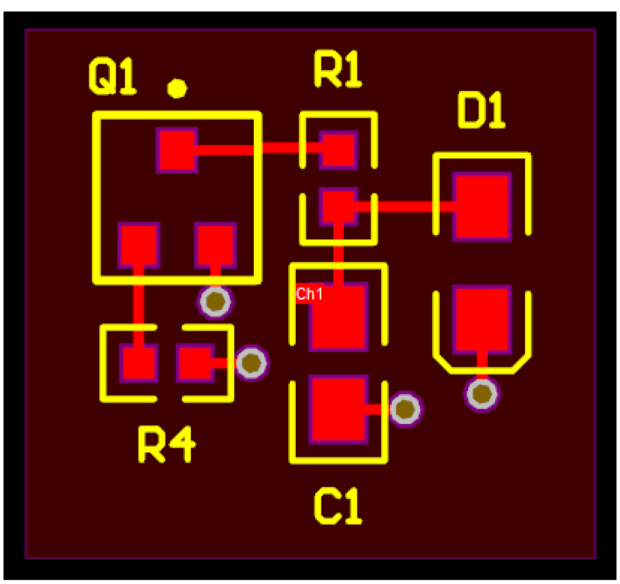
Repeat this process until the Channel Offsets for all 6 groups have been set. The ability to direct-type in the List panel makes this process quick; taking literally seconds to set all the values. For larger circuits, it may be helpful to point out that external data can be pasted to multiple cells at once. This means that, for example, an Excel spreadsheet can be used to quickly create a long column of integers (enter a 1 in a cell then CTRL+drag the corner handle to auto-increment). Copy the cells in Excel, select multiple cells in the Altium Designer List panel, then right-click and select Paste.

The last thing that needs to be done before laying out the circuit is to create the Design Channel class in the same manner that was done for the multi-sheet method (page 5). Go to Design/Classes, right-click the "Design Channel Classes" group, select "Add Class," and rename it "Circuit_2." Select "Ch1" through "Ch6" and move them to the Members List. Close the dialog.

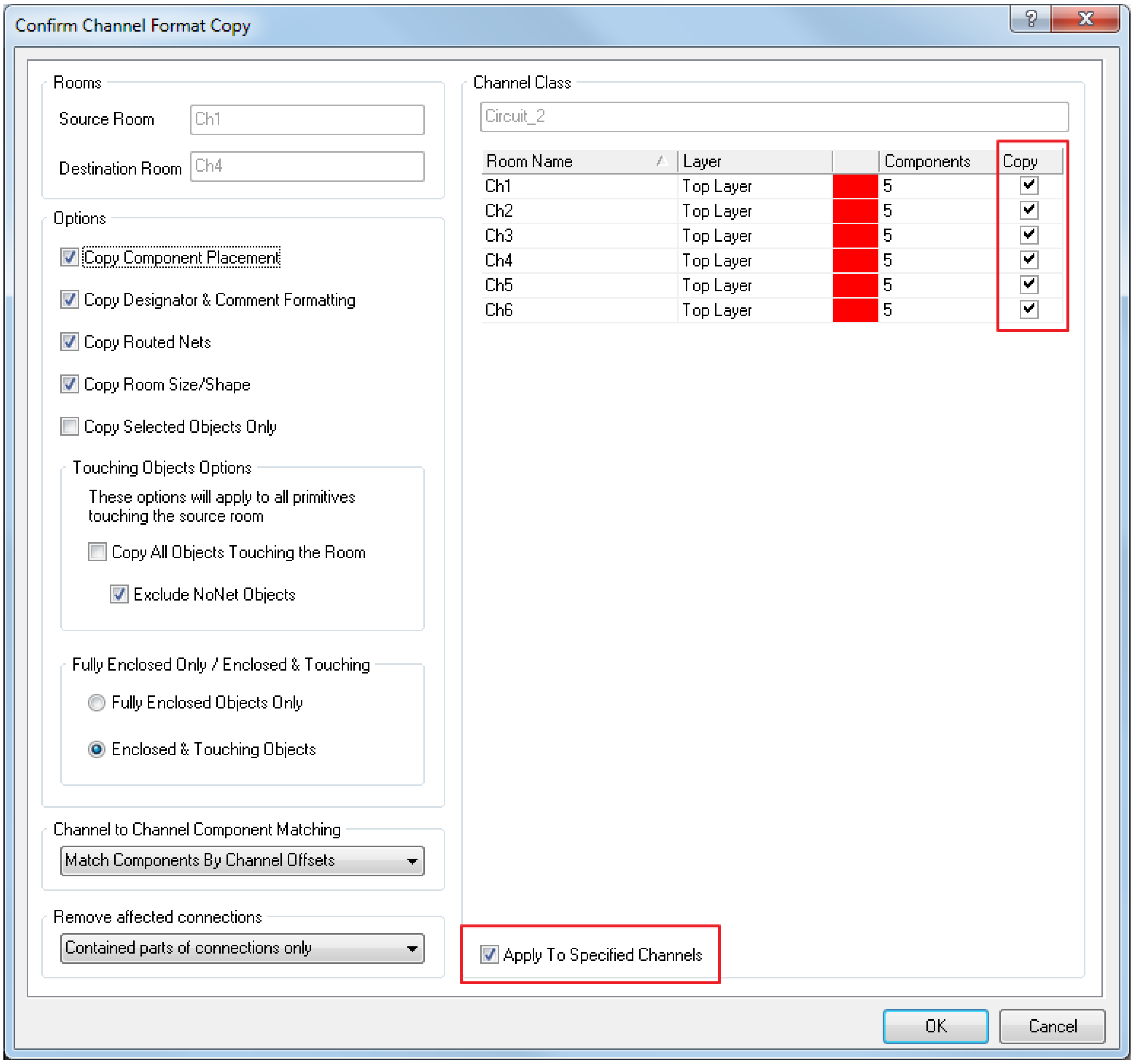


Layout and Copy Rooms

Locate the “Ch1” Room and move it into the board area. Click to select the Room and use the sizing handles to make the Room a small rectangle. Place Ch1’s components inside the Room and route the connections. The VCC ad GND nets were left as fanout vias here.



All that is left is to use the Copy Room Formats exactly as was done the multi-sheet method on page 6 above. Go to Design/Rooms/Copy Room Formats. Click the “Ch1” Room as the Source and then click any of the remaining Rooms as the destination. Since they are all part of the same Channel Class, the system will consider all of them as valid targets for the placement and routing data.

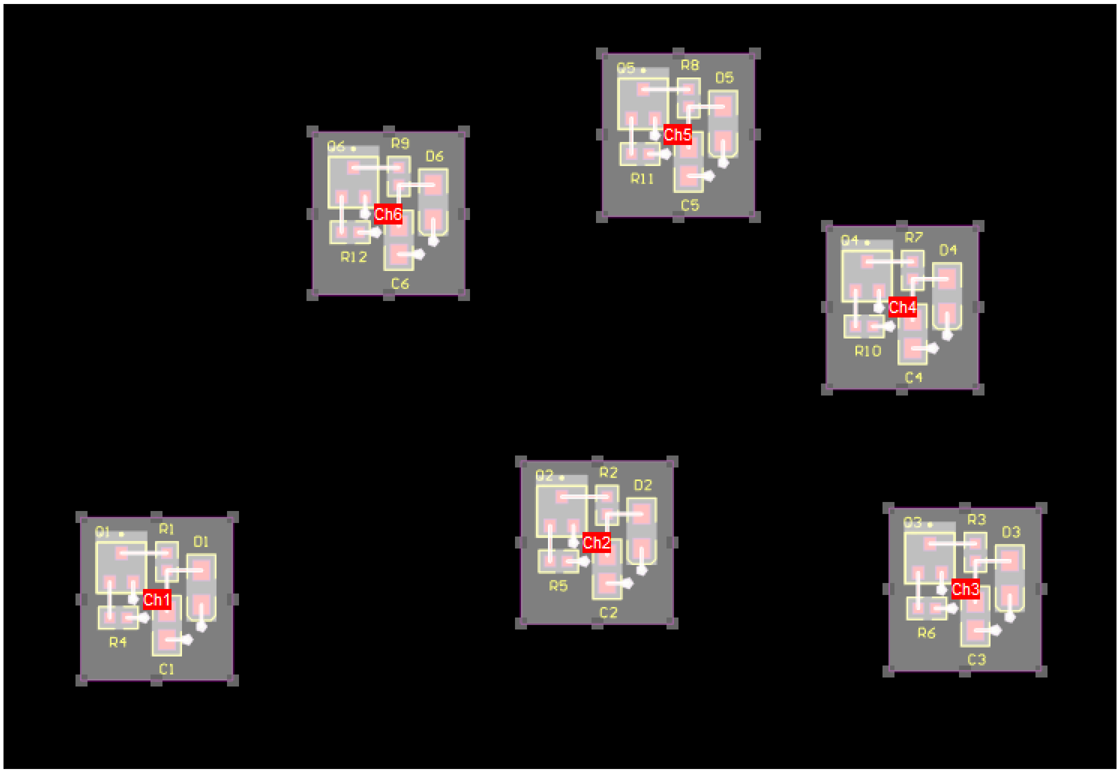


In the Confirm Channel Format Copy dialog, notice that all 6 Rooms are presented. Ensure that the “Apply To Specified Channels” checkbox is enabled and that the “Copy” checkboxes are enabled for all of the Rooms. Click **OK** to run the process. The remaining 5 Rooms should now be placed and routed exactly like the first Room.

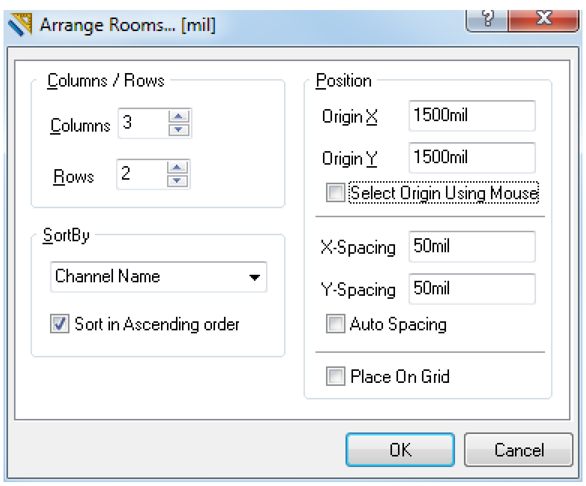
**If a “Channel-Offset Errors” dialog pops up, it is likely because the Offset value changes made in the “Setting the Channel Offset Values” section above were not done correctly and need to be rechecked.

The Rooms can now be moved into place. This can be done manually by dragging the Rooms or using **Design/Rooms/Move Room**. There is also an automated function to arrange them evenly in a grid pattern.

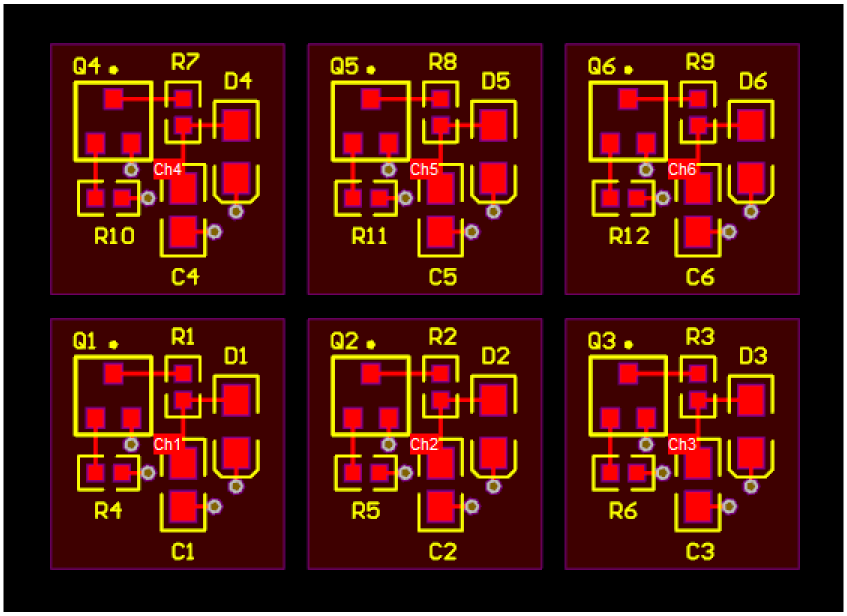
To run this process, first select all of the Rooms:



Now go to the **Design/Rooms/Arrange Rooms** menu. Set the number of columns and rows needed (in this case 2 rows of 3 columns). Other options are available to control Room ordering, location, and spacing:



The result is a neatly spaced grid of Rooms:


Q4
R7
D4
Ch4
R10
C4
Q5
R8
D5
Ch5
R11
C5
Q6
R9
D6
Ch6
R12
C6
Q1
R1
D1
Ch1
R4
C1
Q2
R2
D2
Ch2
R5
C2
Q3
R3
D3
Ch3
R6
C3