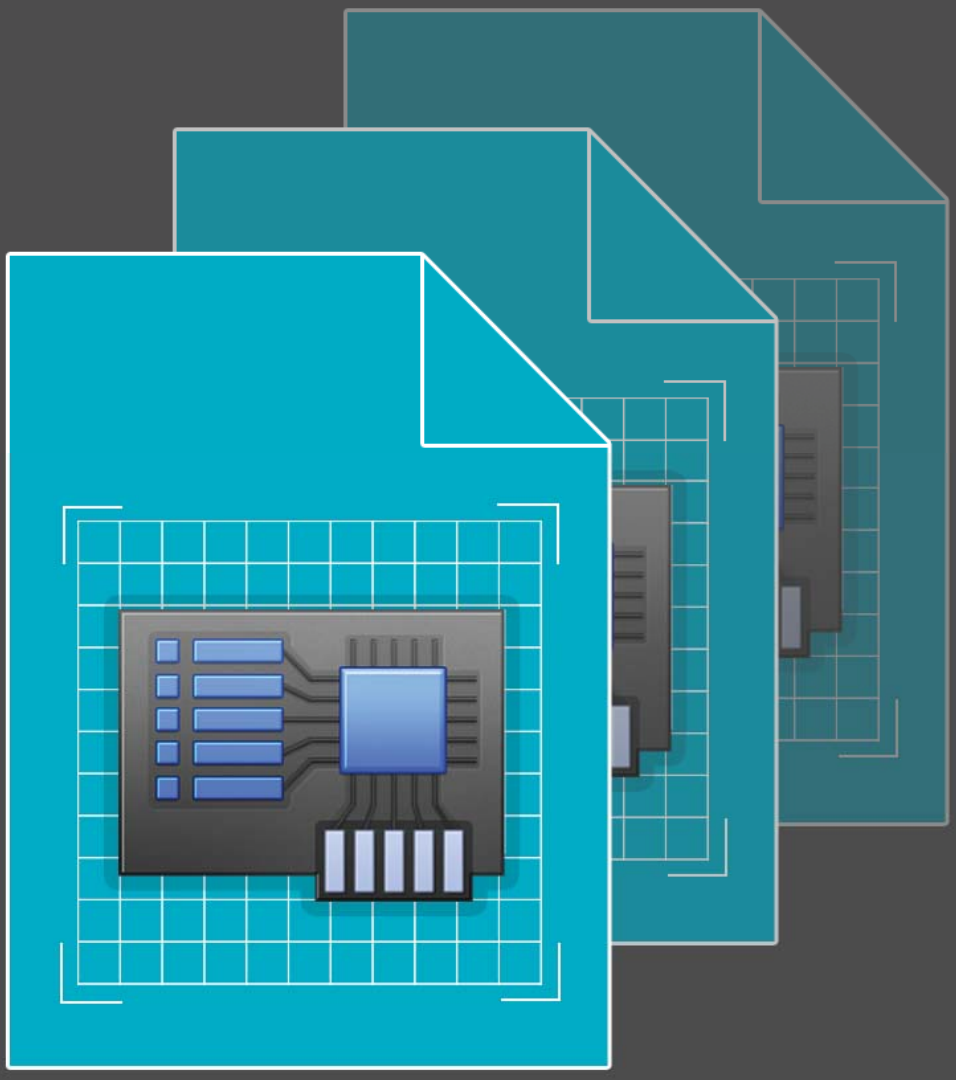
Altium®

Multi-Channel Design with a Flat Project



Dave Cousineau

Sr. Field Applications Engineer

MULTI-CHANNEL DESIGN WITH A FLAT PROJECT

Multi-channel designs have identical or nearly-identical circuitry reproduced for each channel. Replicating circuitry on a PCB design is a simple process in Altium Designer® when true hierarchical schematics are employed. However, flat design offers unique challenges that make PCB circuit replication a bit more complex. This paper will instruct the user on overcoming these challenges. At the end of the paper, you will find handy checklists to aid you with using the concepts presented.

INTRODUCTION

Altium Designer offers many methods for multi-channel design (i.e. repeating circuitry within a single design). For example, 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 the **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 within the PCB document is also extremely efficient. Altium Designer will automatically create a "room" for each iteration of the circuit. Then, you only need to place and route just one of the circuits. Using the **Copy Room Formats** feature, the placement and routing data can be automatically copied to each subsequent circuit. This makes the layout of repeated circuits extremely simple, no matter how many there are!

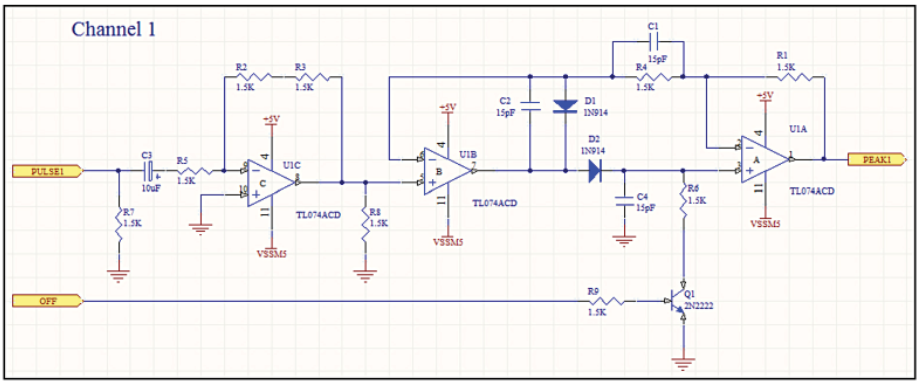
There are many users, however, who have not worked with hierarchical designs and feel more comfortable using a flat design methodology. Or, some projects may just be too simple to warrant setting the entire design as hierarchical. Whatever the case, there are a number of legitimate occasions wherein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

How Can This be Done?

There are two scenarios that need to be addressed. Each has its own steps to set up the PCB document accurately, to enable reuse of the placement and routing data. The first possibility is that each repeated circuit is large enough to take up most, if not all, of a schematic sheet. Thus, a circuit repeated three times would require three schematic documents—for example, if a system has a large power supply that 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, such as in a small LED circuit. In this case, creating separate sheets for each circuit is obviously not very efficient. It may be more reasonable to have that circuit repeated multiple times as part of just one schematic document. Each of these methods translates to the PCB in a surprisingly different way. Both methods are covered here.

FLAT DESIGN USING MULTIPLE SHEETS

By far the easier of the two methods is the use of a separate sheet for each circuit. This is because Altium Designer will automate more of the process in this case. In fact, there will be only one bit of manual intervention required by the user in the PCB document during the process. In the following example, the circuit below is to be replicated just once, creating a Channel1 and Channel2:



Schematic Creation

Start by creating the initial circuit on the first schematic sheet of a PCB project (named **Channel_1.SchDoc** in this example). Then add a second, empty 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, go to the **DXP** menu and then to **Preferences**. Expand the **Schematic** group and select **Graphical Editing** as illustrated in Figure 1. In the **Options** area, enable the **Reset Parts Designators on Paste** option.

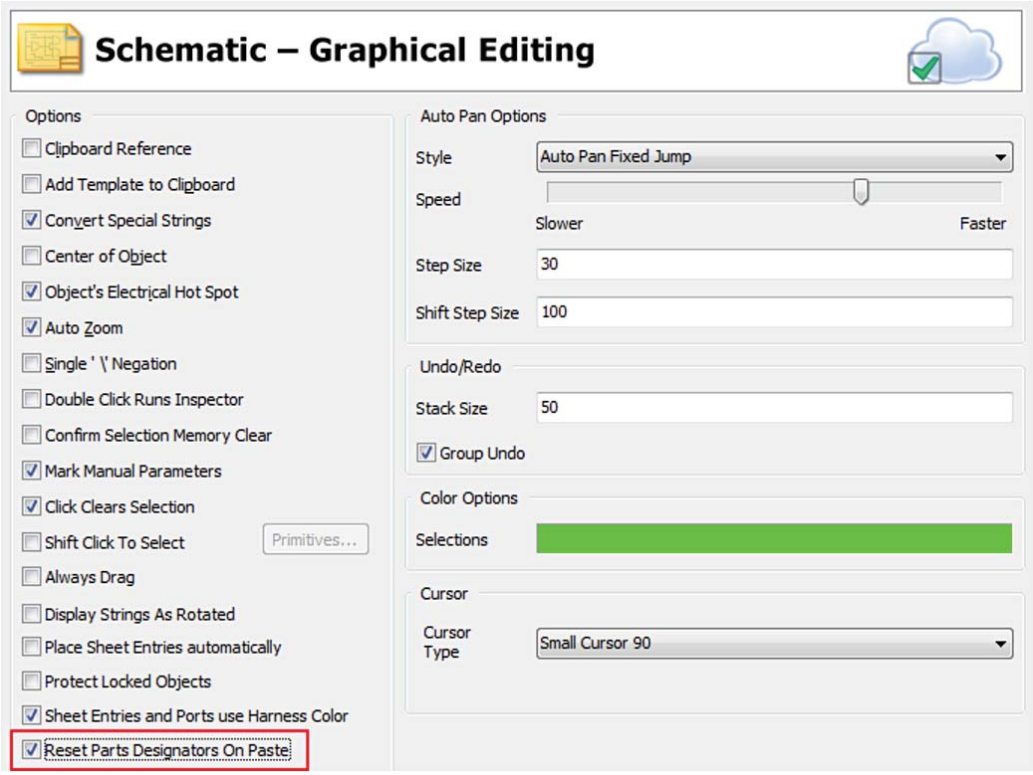


Figure 1: Resetting the reference designators.

Group-select the base circuit, copy it, and paste it to Channel_2 (Figure 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 .

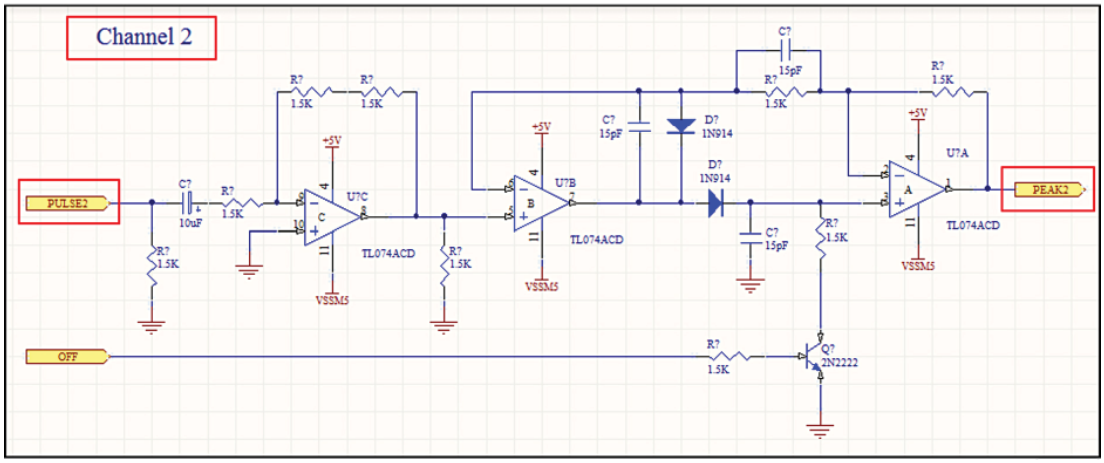


Figure 2: The circuit from Channel 1 has been pasted to create Channel 2. Note the reference designators have been reset to "?" when pasted.

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, run **Tools/Annotate Schematics Quietly** to set the designators (Figure 3).

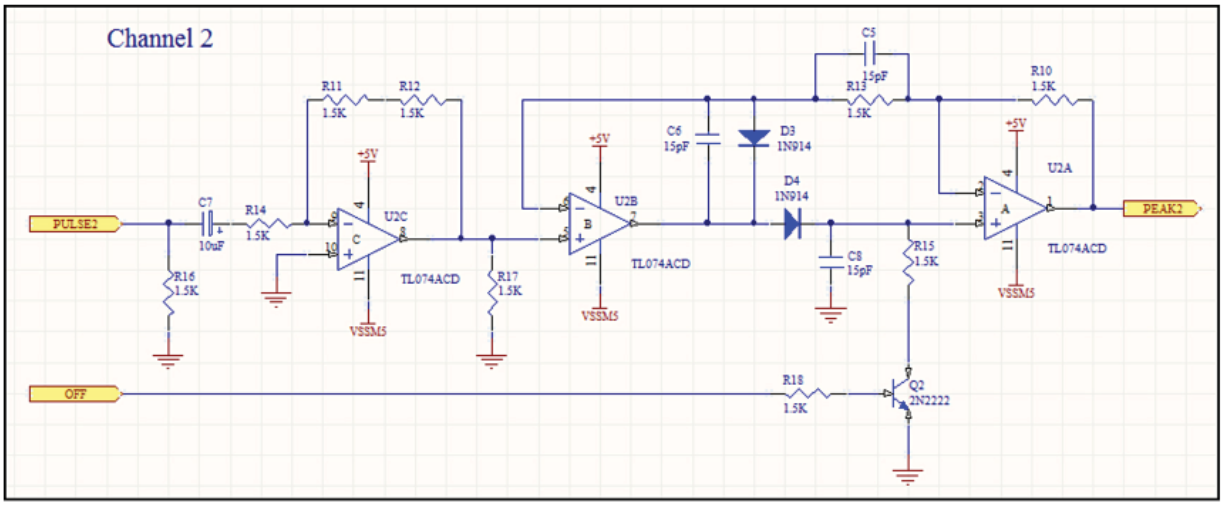


Figure 3: Reference designators have now been reset.

Another important point has to do with multi-part components. In this example, only three of the four op amps in the TL074ACD are being used (A, B, and C), while op amp D is not. Make sure that when the reference designator annotation is done, unused parts from one circuit are not used in another. There must 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, 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, as seen in Figure 4. Any other sheets are optional. Close the Project Options dialog and save all schematic documents, as well as the project file.

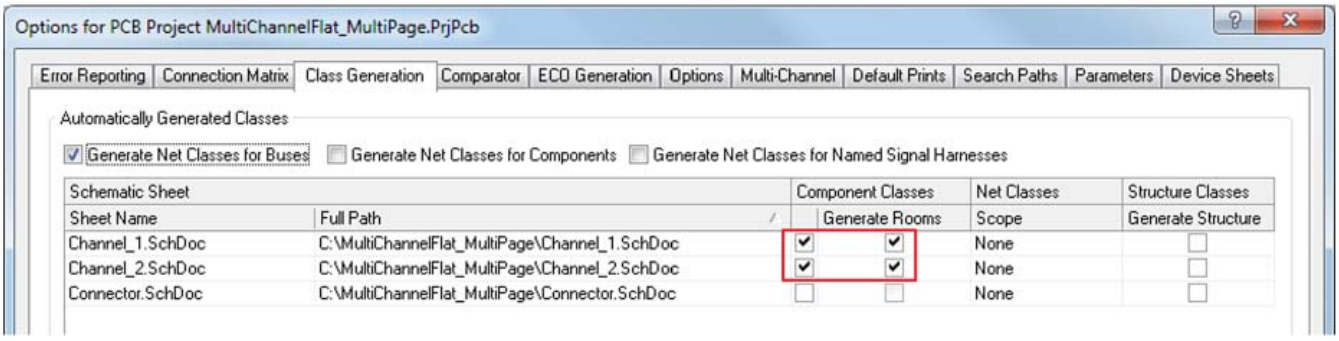


*Figure 4: Check the proper **Component Classes** and **Generate Rooms** boxes.*

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** (Figure 5). If not, recheck the **Project Options** setup done previously.

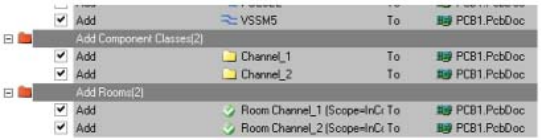


*Figure 5: Create the **Component Classes** and **Rooms**.*

The PCB will then be populated with the **Rooms** as shown in Figure 6.

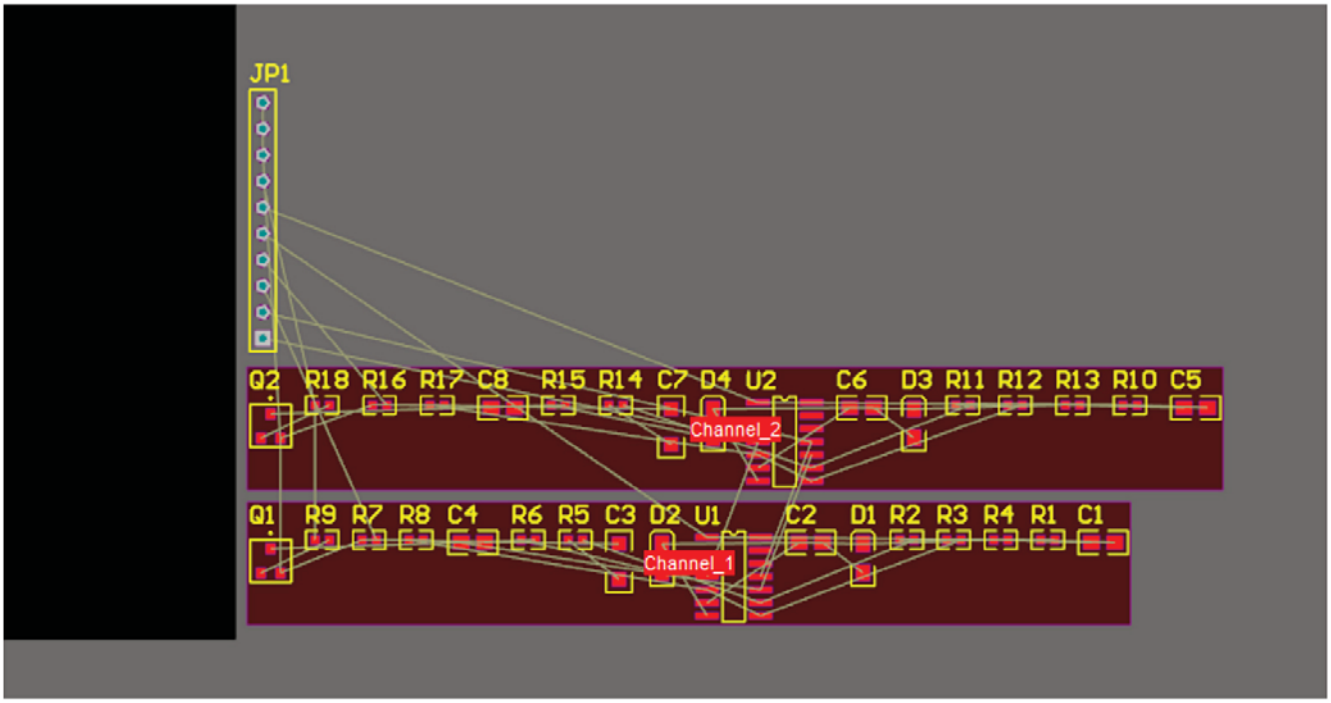


*Figure 6: The PCB has been populated with **Rooms**.*

Next, move the **Channel_1 Room** into the board area, then place and route it as desired. Resize the **Room** outline if necessary (Figure 7).

***Note**: The following section detailing the creation of a **Design Channel Class** is optional when the circuit only needs to be repeated a couple of times. The **Copy Room Formats** command will operate correctly when copying a single **Room** format to another, as is the case in this example. If the **Room's** format needs to be copied to subsequent **Rooms**, however, the command will have to be issued multiple times. It is, therefore, a best-practice to create the **Design Channel Class** as detailed here.*

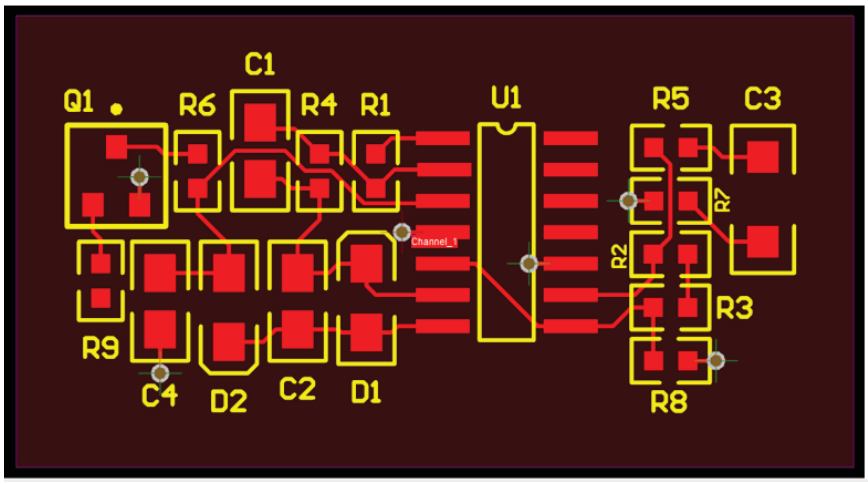


*Figure 7: You can resize the **Room** to see more detail.*

The next part of the process is to use the **Copy Room Formats** feature to replicate the placement and routing. Before that happens, however, you must inform Altium Designer 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, the information is not automatically created. It is possible that one of the circuits was modified by the user so that there is no longer a match between them. If this is the case, the replication of layout information may not be possible. Since no modifications were made to either circuit in this case, the replication can proceed.

Go to the **Design/Classes** menu. Note that there is a **Component Class** for each channel. These were automatically created via the **Project Options** setting and 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**. This creates an item called "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 called **Component Classes**. Note that the "Channel_1" and "Channel_2" **Component Classes** are listed in the **Non-Members** list. Select them both and click the arrow, to move them to the **Members** list, shown in Figure 8.

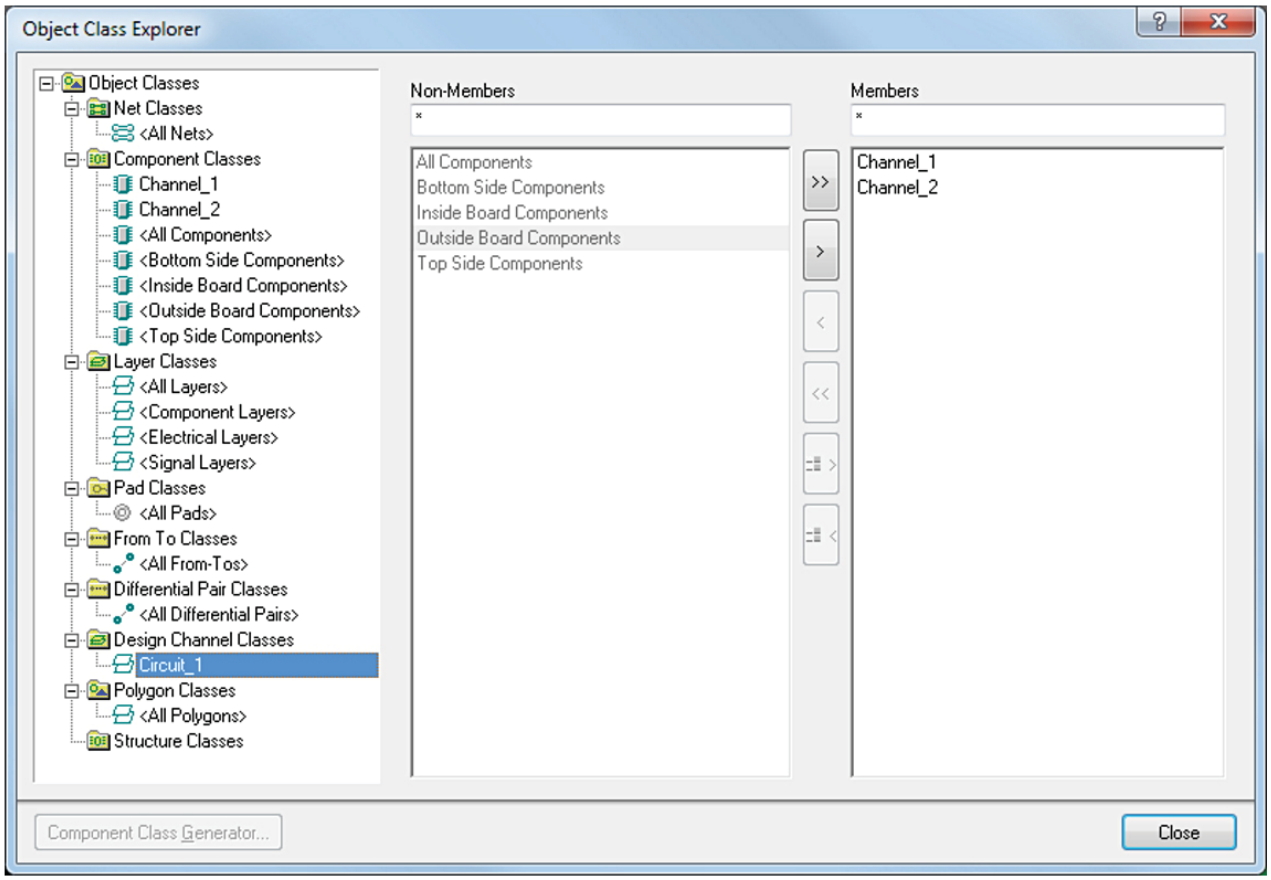


*Figure 8: Move Channel_1 and Channel_2 to the **Members** list.*

Then, close the dialog. Set the board's display so 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 will instruct you to choose the **Source Room**. Click anywhere inside the **Channel_1 Room**. The **Status Bar** then instructs you to choose the **Destination Room**. Click anywhere inside the **Channel_2 Room**. The **Confirm Channel Format Copy** dialog will then open and present 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." We'll discuss **Channel Offsets** in more detail for the second replication method.

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

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

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

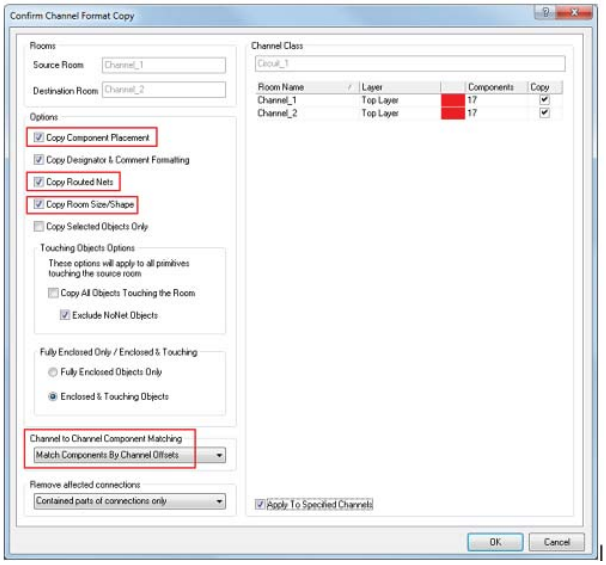


*Figure 9: Be sure the **Copy** boxes are checked.*

FLAT DESIGN USING A SINGLE SHEET

The second multi-channel situation makes 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 for **Copy Room Formats** to correctly function.

For this project example, we'll use a very simple design, consisting of six instantiations of the following circuit plus one connector, shown in Figure 11.

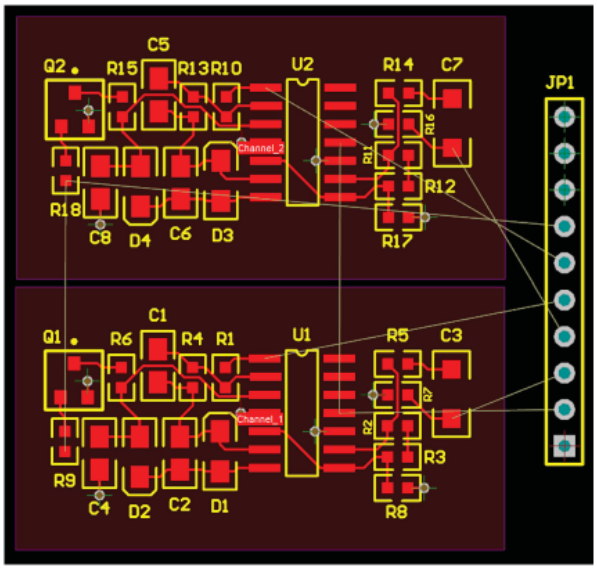


Figure 10: Channel 2 has now been placed on the PCB.

Schematic Creation

Start by creating the base circuit. Leave the reference designators at their default "?" state. Group-select the circuit and use the **Edit/ Rubber Stamp** function to place five more copies of the circuit (Figure 12).

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 we'll discuss in just a moment. In this example, however, it IS acceptable to place other components to multi-channel sheets. The complete design here adds a connector. Use **Tools/Annotate Schematics Quietly** (or any other annotation method) to set the reference designators.

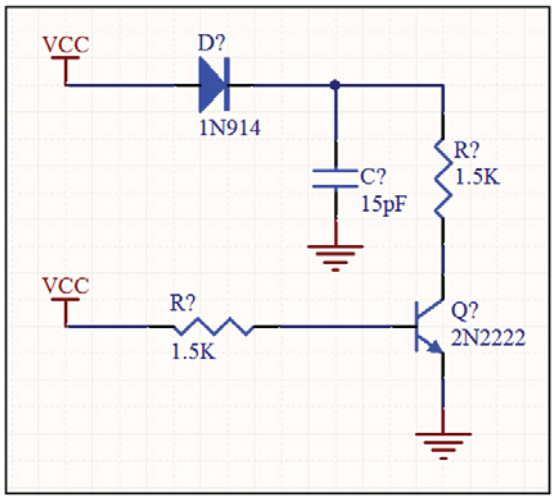


Figure 11: This is the example circuit that will be replicated for six instantiations.

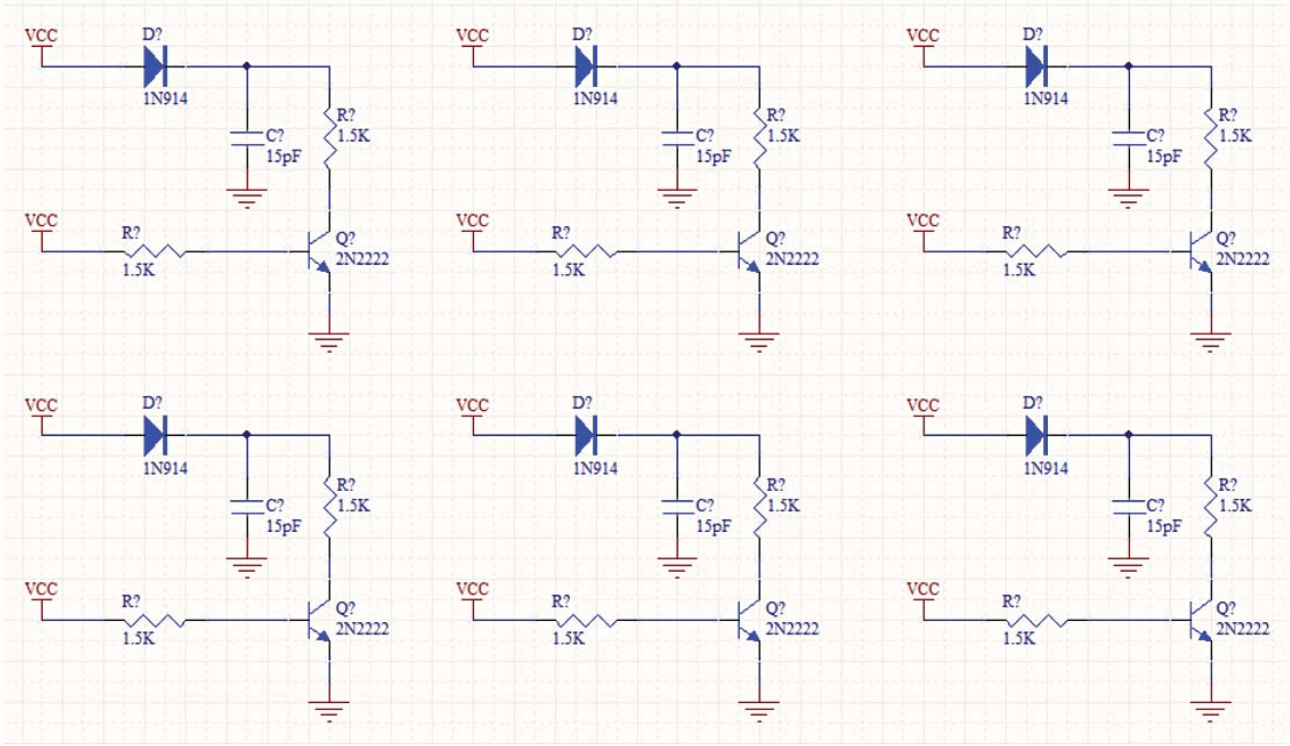


*Figure 12: The **Edit/Rubber Stamp** function reproduces the circuit five times.*

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 checking 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 the PCB, and it is essentially the component's relative physical position within the schematic sheet.

In the previous example, the **Channel Offset values** (accessible in the PCB document, in a component's **Properties**) for Q1 and Q2 match (Figure 13).

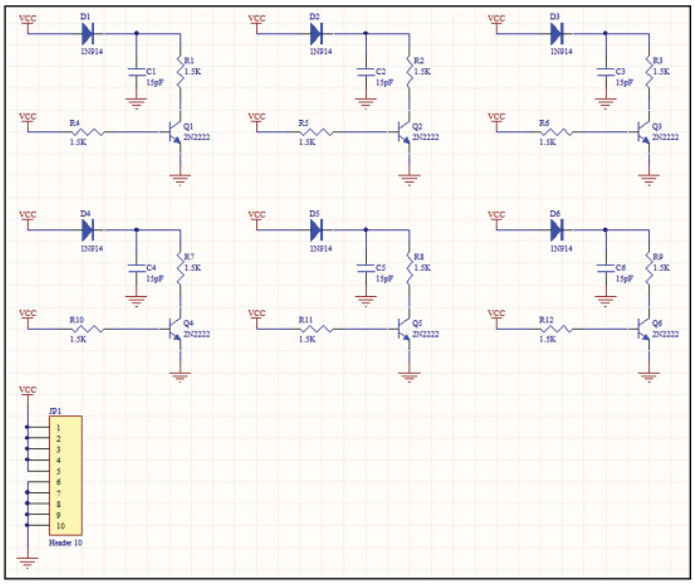


*Figure 13: Note that the **Channel Offset** values are the same for both transistors (Q1 and Q2).*

They match 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. Figure 14 shows the **Channel Offsets** match for each like component in Channel_1 and Channel_2.

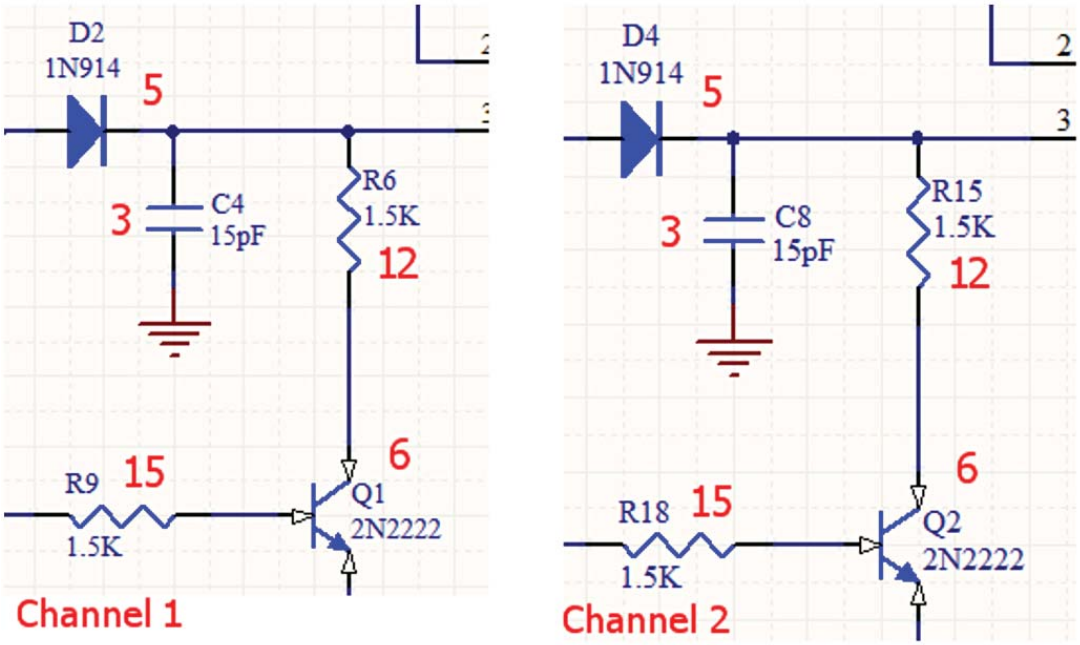


*Figure 14: The **Channel Offset** values are shown next to each component in red. Note how they are identical for Channel 1 and Channel 2.*

Channel Offset values are sequentially applied to all components on a schematic. On the single sheet example, all components will thus 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 within 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. We'll get more into that 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. However, 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 demonstration purposes.

Go to **Tools/Parameter Manager**. Set the **Options** dialog to include only the Parameters owned by "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.

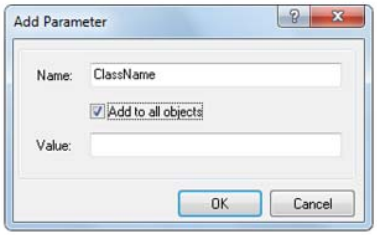


*Figure 15: Name the parameter "ClassName" and check **Add to all objects**.*

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".

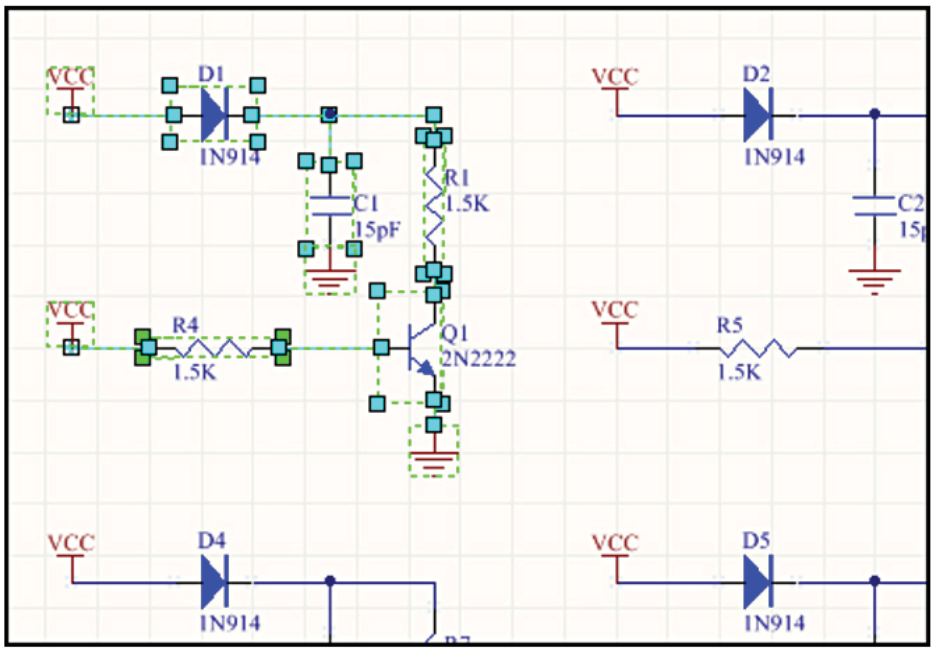


Figure 16: Group-select the components in circuit 1.

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."

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

Before transferring the schematic information over to the PCB, there's one last step in setting the **Class Generation** tab in **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 still need to be enabled in the **User-Defined Classes** section: see Figure 19.

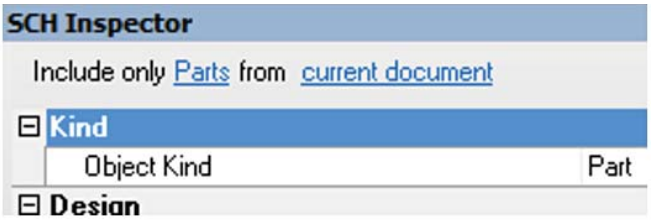


*Figure 17: In the **SCH Inspector** dialog, set the filter to "Include only Parts from current document".*

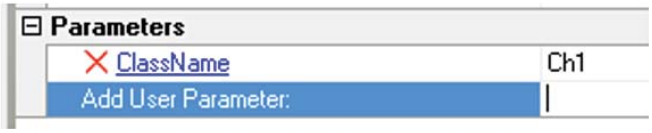


Figure 18: Give the "ClassName" the name "Ch1".

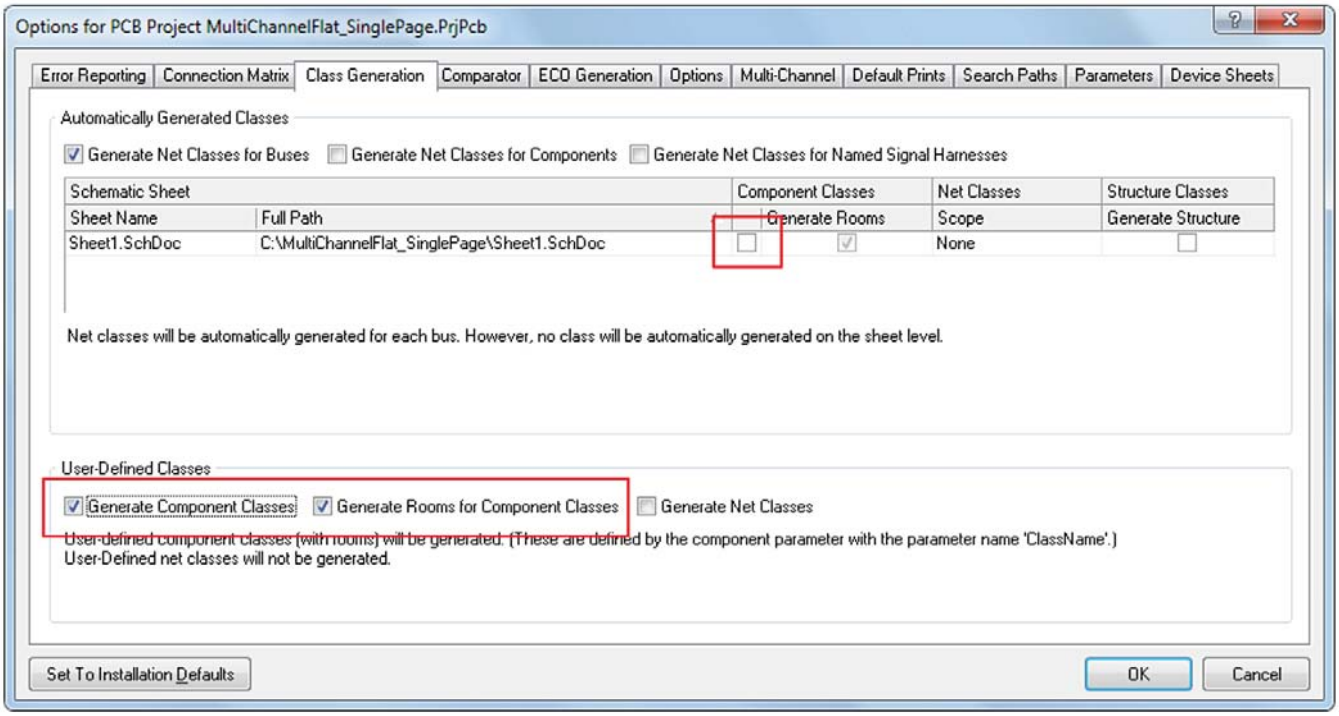


*Figure 19: The **Component Classes** box has been unchecked. Leave the "Generate Component Classes" and "Generate Rooms for Component Classes" enabled (checked).*

PCB Layout

Create and save a new PCB file, then use **Design/Import Changes...** to populate the board. Make sure that the ECO includes the creation of the "Component Classes" and **Rooms**. If it doesn't, close the ECO dialog without executing and recheck the existence of the "ClassName" parameters and the Project Options setup done previously.

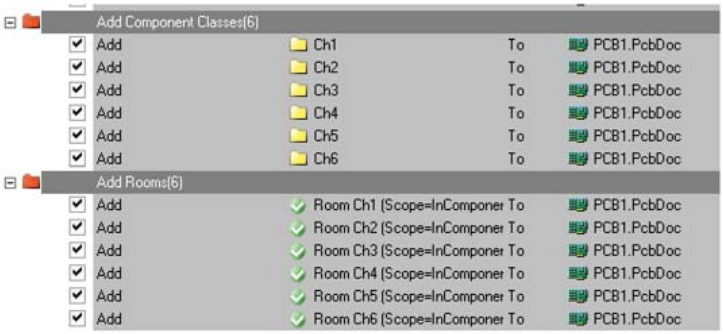


Figure 20: If not already checked, check the "ClassName" parameters.

Next, open the PCB panel (**View/Workspace Panels/PCB/PCB**), and set the pull-down filter to **Components**. Enable the **Select** checkbox.

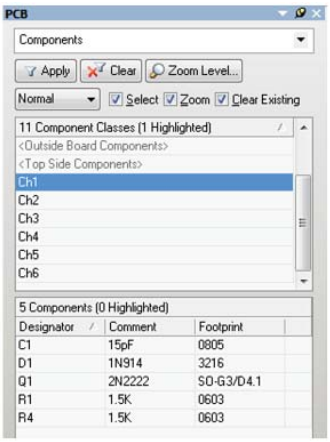


Figure 21: Selecting "Ch1" reveals the components assigned to channel 1.

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 opened).

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

Setting the Channel Offset Values

The next step is crucial 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 done manually.

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

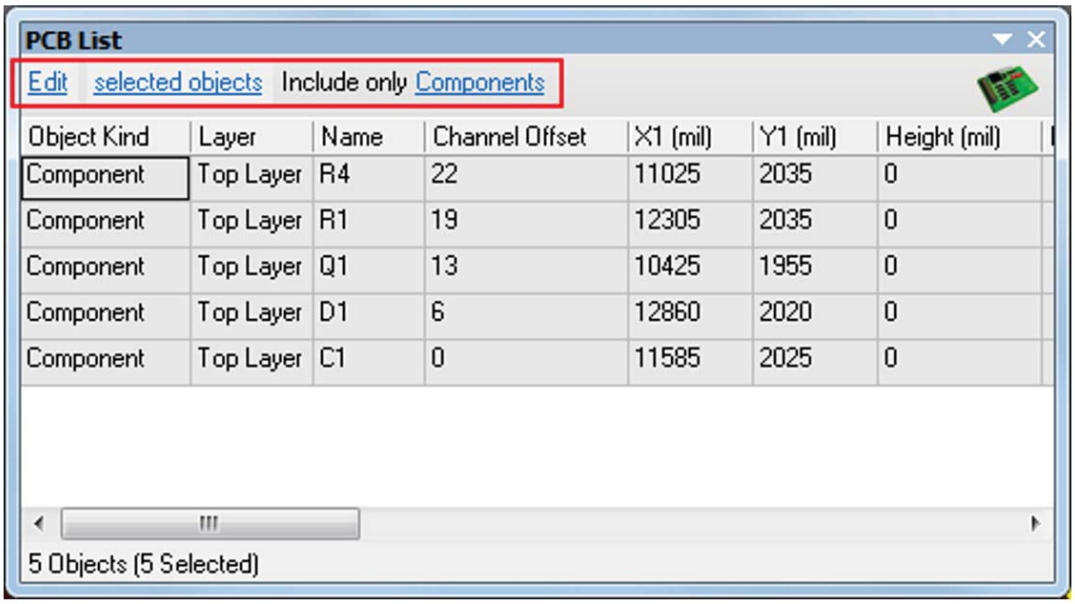


*Figure 22: Set the filter in the **PCB List** to **Edit selected objects Include only Components**.*

Sort the list by the reference designator by clicking the "Name" field header. C1 should now appear at the top of the list. Click in the "Channel Offset" cell for C1, type a 0 (zero) and hit **Enter**. This will set C1's **Channel Offset** value to 0 and bring you down to the next component (D1). Type 1, then hit **Enter** and continue down the list, until all components are sequentially numbered.

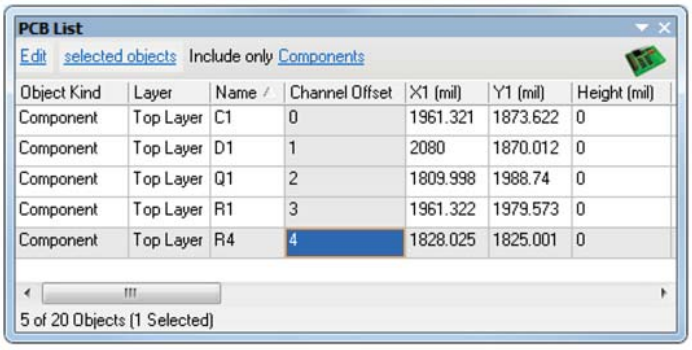


*Figure 23: Set the **Channel Offset** values manually to sequential numbers.*

Leaving the **PCB List** panel open, 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. Sorting the components this way will ensure that the **Channel Offset** values will be the same for the matching components in each circuit. Use the same **Channel Offset** values as for the Ch1 components, as in Figure 24.

Be sure to start numbering at 0 and continue sequentially. **Copy Room Formats** can deal with non-sequential values, but it will display a warning during the process, so it's best just to avoid it in the first place.

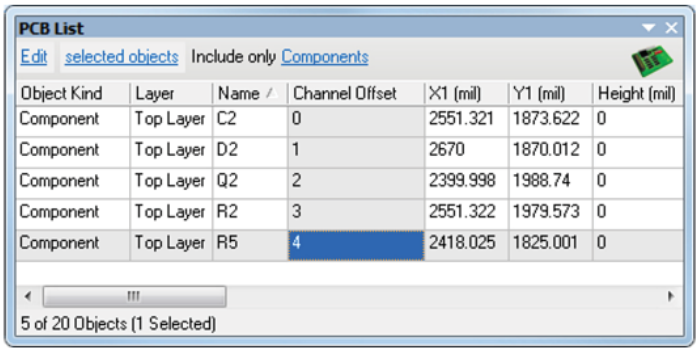


Figure 24: Continue for the rest of the channels.

Repeat this process until the **Channel Offsets** for all 6 groups have been set. Being able to type directly in the **List** panel makes this process quick; it takes just 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, a spreadsheet can be used to quickly create a long column of integers: enter a 0 in a cell then **CTRL+drag** the corner handle to auto-increment. Copy the cells in the spreadsheet, 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. 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.

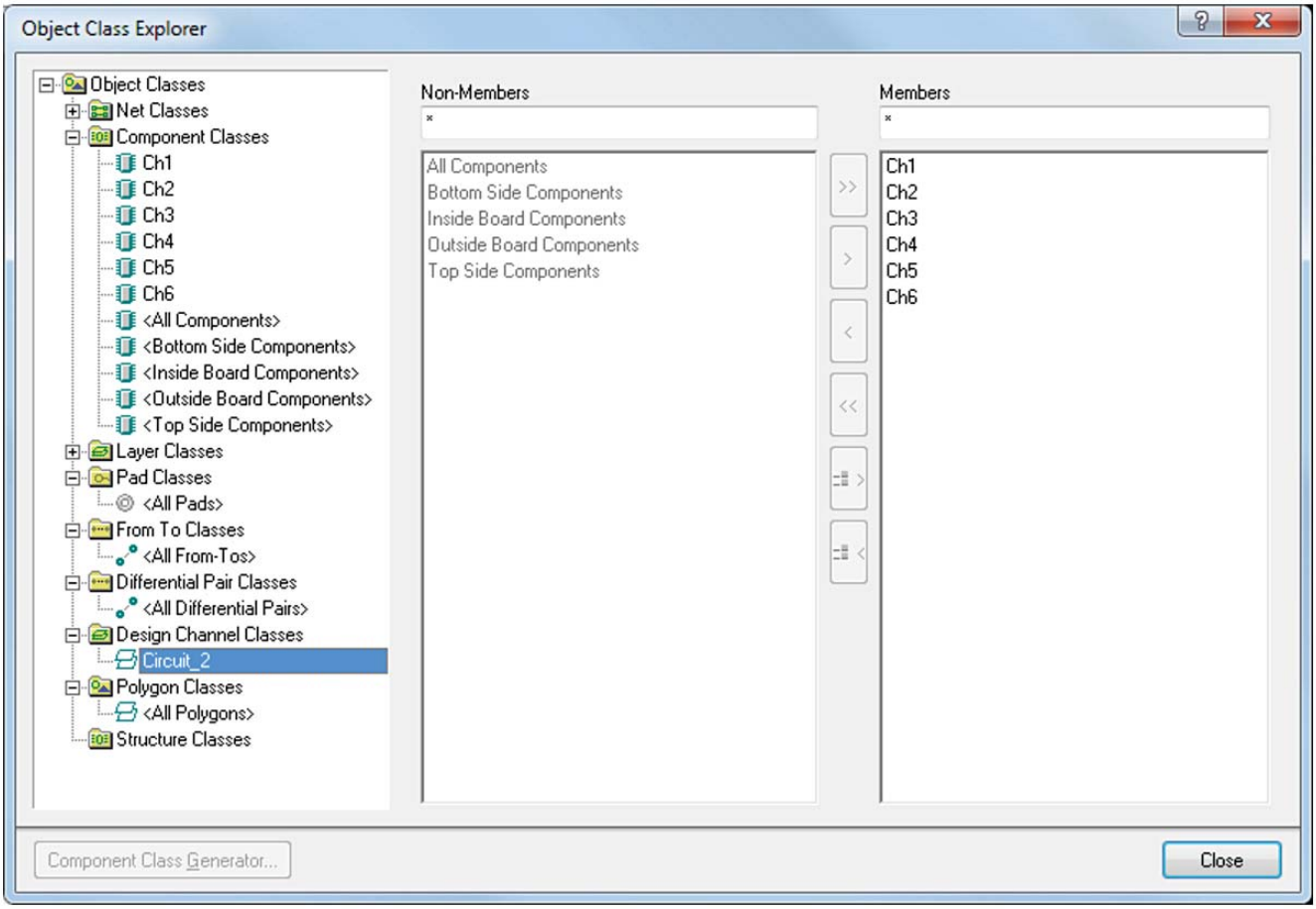


*Figure 25: Add the channels to the **Members** list.*

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 it a small square or rectangle. Place Ch1's components inside the **Room** and route the connections. The VCC and GND nets were left as fanout vias here.

Now, all that's 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**, then click any one 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. See Figure 27.

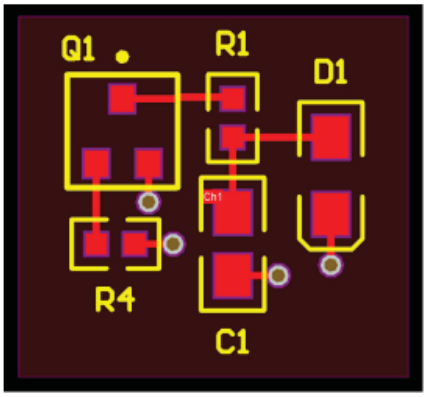


Figure 26: Here is the circuit after moving it to the PCB.

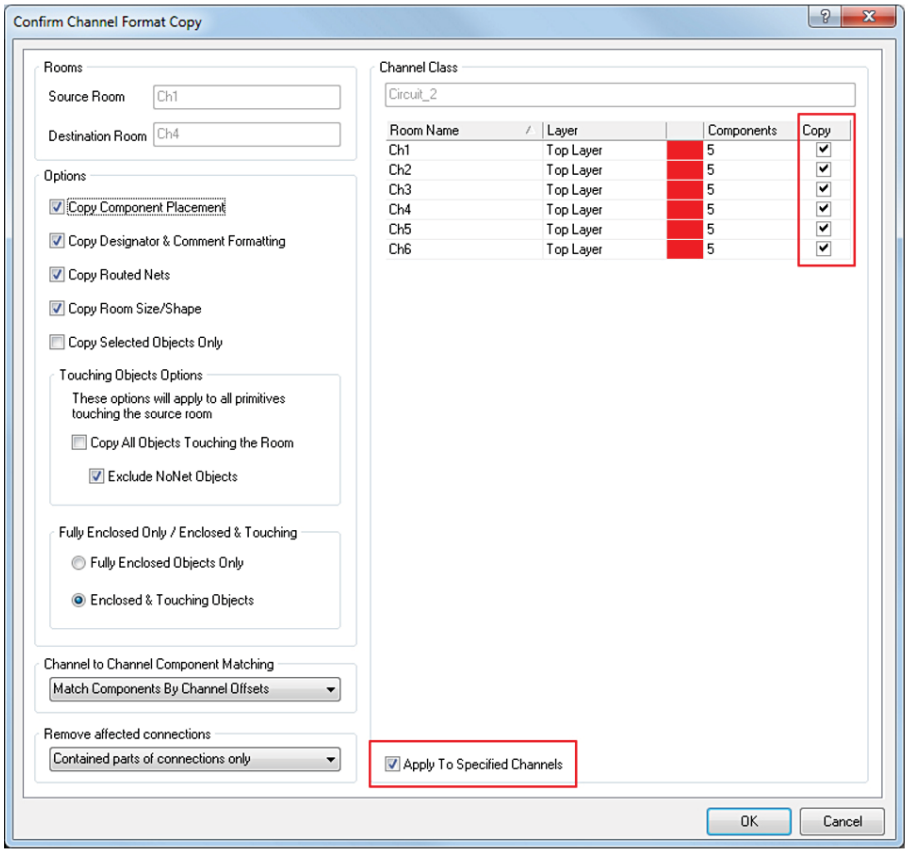


*Figure 27: The final step is to use the **Copy Room Formats**.*

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

Note: If a **Channel-Offset Errors** dialog pops up, it's most likely because the offset value changes made in the **Setting the Channel Offset Values** section above were not done correctly. Check them over again.

The **Rooms** can now be moved into place. This can be done manually by dragging them, or by using **Design/Rooms/Move Room**. Additionally, there's an automated function to arrange them evenly in a grid pattern. To run this process, first select all of the **Rooms** as shown in Figure 28.

Then, as shown in Figure 29, go to the **Design/Rooms/Arrange Rooms** menu and 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:

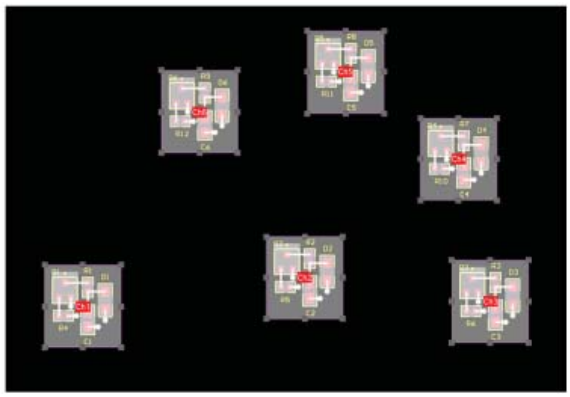


*Figure 28: Before arranging, select all six **Rooms**.*

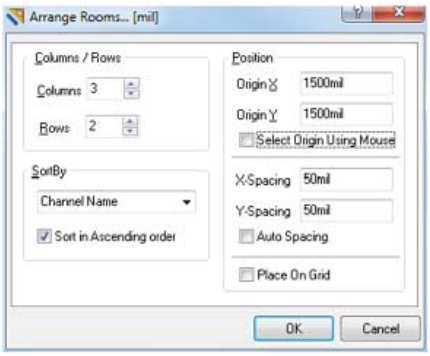


*Figure 29: Next, using the **Design/Rooms/Arrange Rooms** menu, set the number of columns and rows needed (2x3).*

The result is a neatly spaced grid of **Rooms**, illustrated in Figure 30.

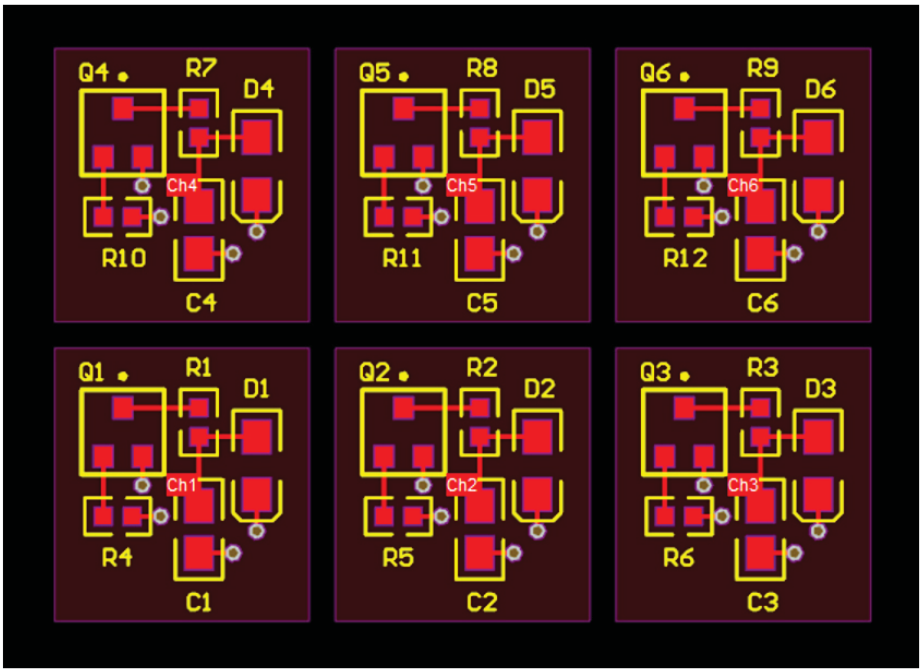


*Figure 30: The **Rooms** have been automatically arranged neatly in a grid.*

MAKING CHANGES TO THE DESIGN

When changes are necessary to the circuits in your schematic, there are a few issues that must be dealt with to keep the synchronization that has been created in the previous examples.

First, when adding components to the repeated circuits using the single-sheet method, make sure that the "ClassName" parameter is added, and that their values are set properly (Ch1, Ch2, etc.). New components will have to have their **Channel Offset** values set via the **PCB List** panel, as was done initially.

Regardless of the method used (single-page or multi-page), passing design changes from the schematic to the PCB may have an effect on the **Channel Offset** values. There are a few different problems that may arise here: repeated offset values for differing components, skipped values in the sequence, etc. In any case, after making any sort of change it's important to load the repeated channel components into the **PCB List** panel and inspect the **Channel Offset** values to make any necessary changes.

Also, regardless of the method used, the synchronization process will attempt to remove the manually-created **Design Channel Class**. That change should be disabled during the ECO, shown in Figure 31.

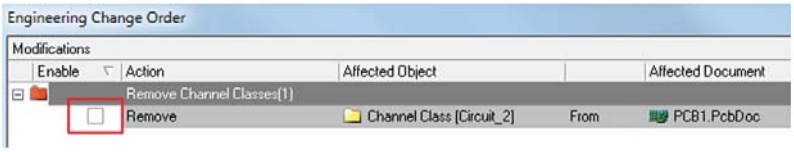


*Figure 31: Uncheck the box for **Remove Channel Classes**.*

Once you've completed the ECO and performed the necessary placement and routing changes to the base **Room**, replicating that change across the other **Rooms** is as simple as rerunning **Copy Room Formats**, as was done initially.

CHECKLIST

Even after learning the above processes, it still may be helpful to have a checklist nearby to ensure that you don't miss any steps.

Flat Design Using Multiple Sheets

- In **SCH Inspector**: Select, copy and paste the circuit from the source sheet to the target sheet(s)
- Annotate reference designators, ensuring that multi-part components are unique to each sheet
- In the **Class Generation** tab of **Project Options**, enable **Component Classes** and **Generate Rooms** for all sheets that have the repeated circuit
- In **PCB**: Import the design as usual. Move one **Room** to the PCB, then place & route the components in that **Room**
- In **Design/Classes**, create a **Design Channel Class** and add to it the **Component Classes** that were auto-generated for each sheet that has the repeated circuit
- Use **Design/Rooms/Copy Room Formats**

Flat Design Using a Single Sheet

- In **SCH Inspector**: Select the circuit and use **Edit/Rubber Stamp** to replicate the circuit
- Annotate reference designators
- Use **Tools/Parameter Manager** to add the "ClassName" parameter to all components
- Select circuits individually and set the unique "ClassName" parameters ("Ch1", "Ch2", etc.) using the **PCB Inspector**
- In the **Class Generation** tab of **Project Options**, disable **Auto-Generated Component Classes**, and enable **User-Defined Component Classes** and **Room** generation
- In **PCB**: Import design as usual
- Open the **PCB** panel in **Components** view and enable **Select** mode
- Select **Component Class** for first channel
- Open the **PCB List** panel in "Edit Selected Objects" mode
- Sort the components list by reference designator, then set the **Channel Offset** values for all components sequentially starting at 0
- Go back to the **PCB** panel, select the next **Component Class** and repeat the **Channel Offset** value step. Do this for all channels.
- In **Design/Classes**, create a "Design Channel Class" and add the **Component Classes** for the repeated channels
- Move one **Room** to the PCB, then place and route the components in that **Room**
- Use **Design/Rooms/Copy Room Formats**