Module 13: Classes and Rooms

Module 13: Classes and Rooms

Module Seq = 13

13.1 Object classes

13.1.1 Defining Classes

Classes are provided to enable various commands to operate on sub-sets of object types, for example a group of components or a group of nets. Note that an object can belong to more than one class.

Commands will operate on a class if a design rule for that class has been defined.

Classes can be created for:

- nets
- components
- pads
- from-tos
- differential pairs
- layers.
- design channels
- polygons

To create an object class, select **Design » Classes**. This displays the *Object Class Explorer* dialog shown in Figure 1 below.
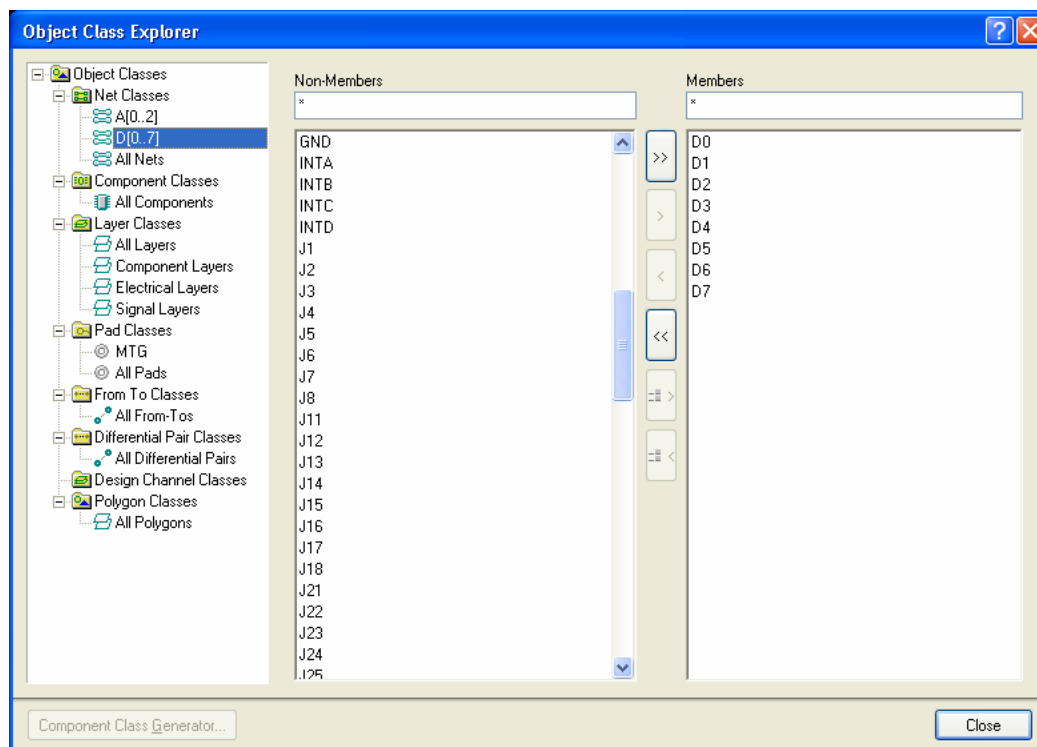

Figure 1. Use the Object Class Explorer to create and manage Object Classes.

- Click on the class type of the class you want to create, right-click and select **Add Class**. A new class will appear in the list with the default name of **New Class.** Click on the class name to edit the class and add the members, right-click on the class name and select **Rename Class** to rename it.

- Note that there are transfer buttons for selected objects; often it is easier to select the objects in the workspace first, then use these transfer selected buttons to build the class. If you have created a selection before hand using *Find Similar objects*, *PCB panel*, *list panel* etc, then the button shown in Figure 2 should enable. Pressing this button shifts the selection from the non-members column to the members column.
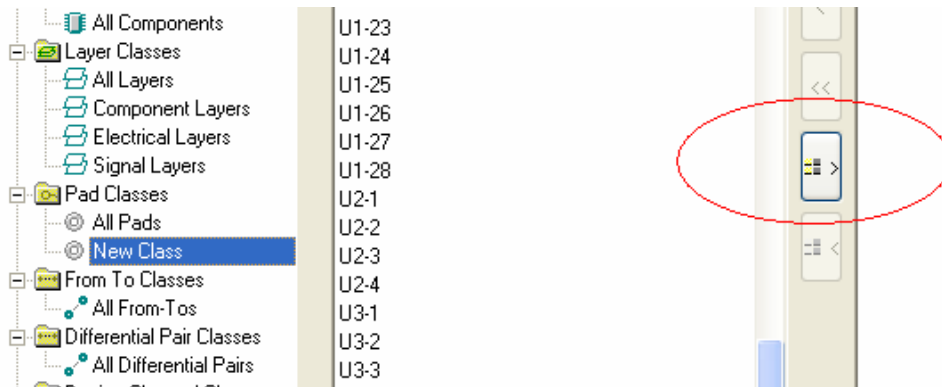

Figure 2. The selection button enabled.

13.1.2 Component Class Generator

The *Edit Component Class* dialog includes the **Class Generator** button, which, when clicked, displays the *Component Class Generator* dialog. This allows you to quickly create a component class containing components based on selected properties.

13.1.3 Exercise – Using classes in design rules

This exercise looks to continue with some of the rule changes we have made in the previous module.

1. Using the Temperature sensor project PCB document, create a Net Class called `Power`, which includes the following nets: 3V3, 5V and GND. To do this:

    - Select **Design » Classes**

    - Right-click on **Net Classes** in the tree on the left and select **Add Class**.

    - Click on the New Class entry that is added to the list, and press **F2** to rename the class to `Power`.

    - Add the class members 3V3, 5V, GND and close the dialog.

2. In the **Design Rules** dialog, add a new routing via style rule that targets the Power net class, with settings of Via diameter = 40 and a hole size of 25. Name this rule `RoutingVias_Power`
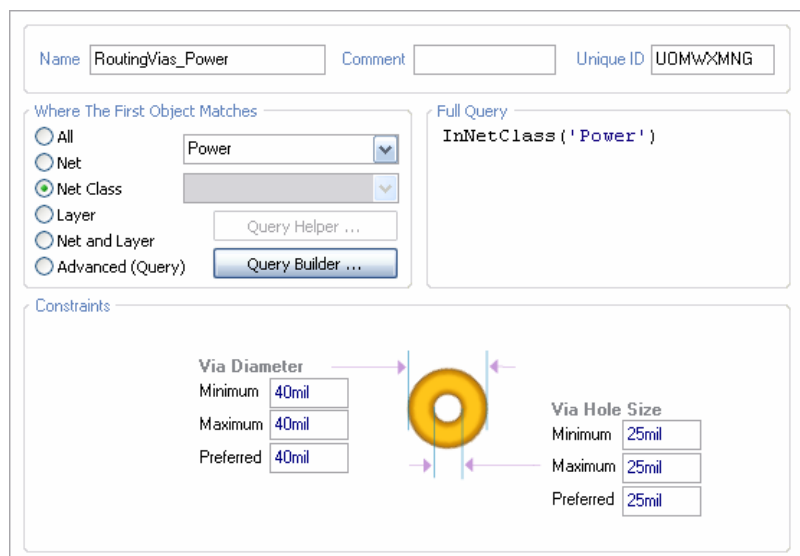

Figure 3. Set the rule to target the Power net class

3. Save the board.

13.2 Rooms

A room is a region that defines an area where components can either be kept within or kept out.
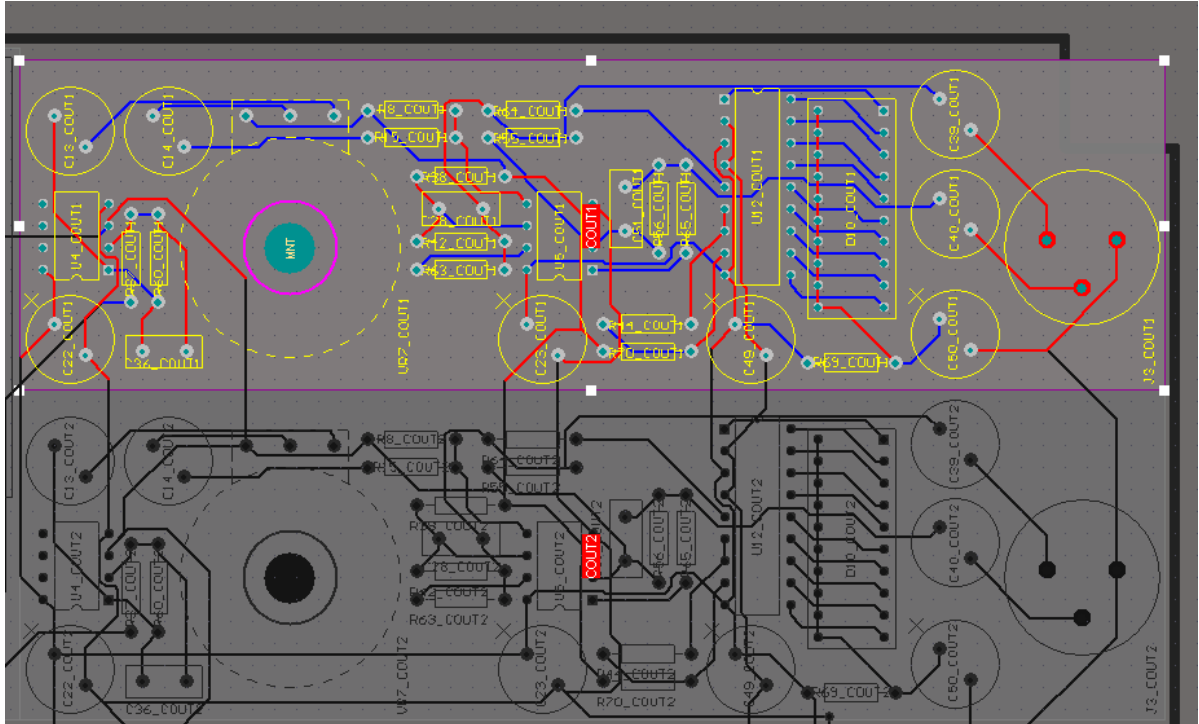

Figure 4. Components and routing in the room COUT1 highlighted, all other components and routing on the board are masked..

13.2.1 Defining Rooms

- Rooms are placed using the commands in the **Design » Rooms** sub-menu, or using the Room tools on the Utilities toolbar.

- A Room Definition design rule is created for each room that is placed.

- Once a room is placed you define the components associated with it (and whether they are to be kept in or kept out), as the scope, or query, for that rule.

- To define the components associated with a room, double-click on the room to display the *Room Definition* dialog. This dialog can also be accessed in the **Placement** region of the *Rules* dialog. Set the scope of the rule to the required component, component class or footprint.

13.2.1.1 Moving components into a room

- Components that have been assigned to a room can be automatically moved into it by selecting the **Tools » Component Placement » Arrange Within Room** command, or clicking the **Arrange Components Within Room** button in the Alignment tools in the Utilities toolbar. You will be prompted to click on the room.

13.2.1.2 Moving rooms

- Once component(s) have been assigned to a room, they move when the room is moved. To move a room without moving the components, temporarily disable the Room Definition rule in the **Placement** section of the *PCB Rules& Constraints* dialog.

- If a component is moved such that it is in violation of the Room Definition rule, it is displayed with a Design Rule Check (DRC) error marker.

13.2.1.3 Using a Room to scope another Rule

- Rooms have a dual nature in that they are defined as a rule themselves, but they can also be used as the scope of other design rules.

- To use a room as the scope of another rule, for example to define a region where you require larger or tighter routing clearances, you first set the Room rule to target nothing by setting its rule Query to something like: **Not IsComponent**.

- You can then define a Routing Clearance design rule that uses a Query like **WithinRoom(MyRoomDefinition)**.

- Examples of where you might use the query WithinRoom include Width, Clearance and Via Style design rules, defining exactly the area of the PCB were that rule is to be applied.

13.2.2 Copying Room Formats

The placement and routing of one room, can be applied to other rooms. Use this capability in multi-channel designs, where the component placement and routing can be identical in all channels.
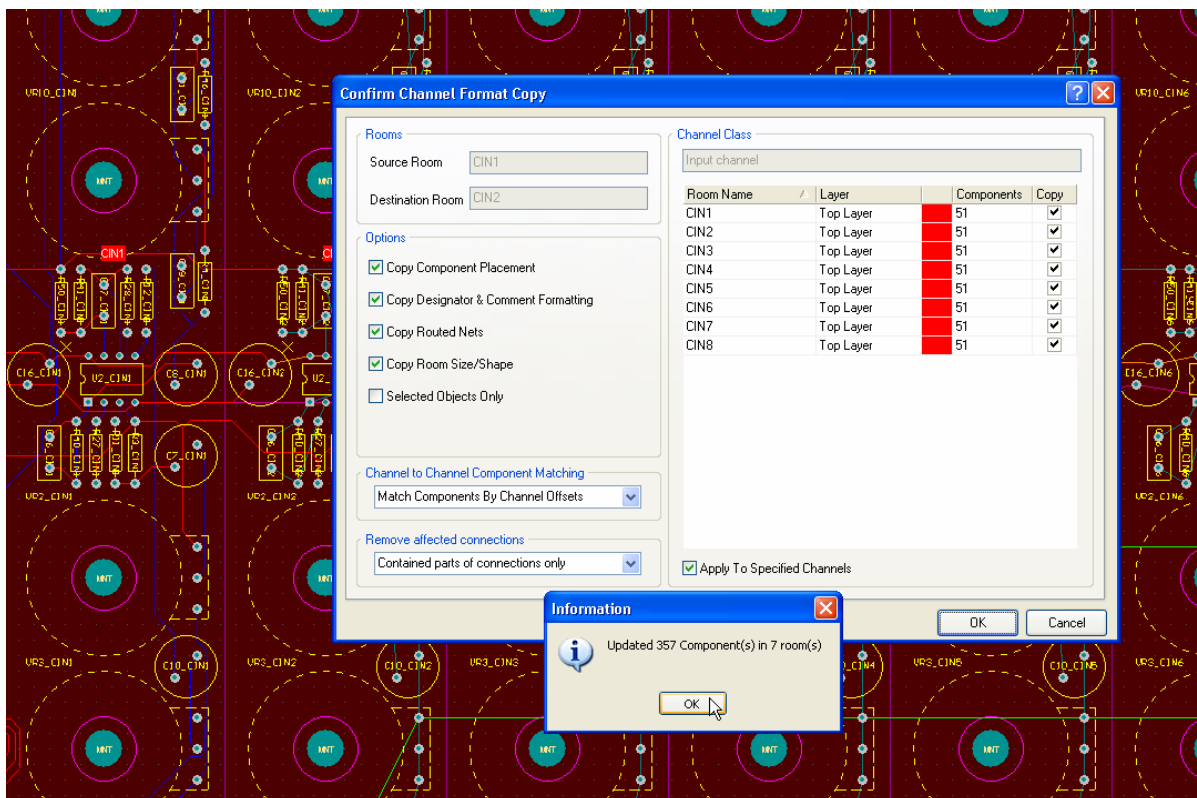
Figure 5. Component placement and routing being copied from one room, to seven other identical rooms. This 'step and repeat' capability is an excellent time-saver for a multi-channel PCB.

- To copy the placement and routing of one room onto others, run the command **Design » Rooms » Copy Room Formats**. You will be prompted to choose the source room, click anywhere in that room. You will then be prompted to choose a destination room, click anywhere within one of the target rooms.

- The **Confirm Channel Format Copy** dialog will appear. Use this dialog to define the copy options as required. The options relate to component placement, designator and comment formatting, net routing and room size/shape.

- There is also an option to limit the copy to only those objects that are selected in the source room.

- If the source room is part of a defined channel class, the class and all of its members will be listed on the right-hand side of the dialog. This section is, by default, unavailable until you enable the **Apply To Specified Channels** option. For each room in the class, its name, layer and the amount of components it contains, is listed. There is also a check box to define whether a room should be included in the copy.

- Use the **Channel to Channel Component Matching** field to determine the technique used to match components between the source and destination room(s). Matching can be carried out either by use of the **Channel Offset** for a component - stored in the **Schematic Reference Information** region of the **Component** dialog - or by source designator for the component, i.e. the logical designator, as used in the source schematics.

- After defining the criteria for the copy as required, pressing **OK** will close the dialog and proceed with the copy. An information dialog will appear, listing how many components were updated and in how many rooms.

13.2.3   Polygon Rooms

As well as rectangular shaped rooms, polygonal rooms can also be placed (**Design » Rooms » Place Polygon Room**).

- As with rectangular rooms, the room can either be placed empty and components associated at a later stage, or it can be placed around components in the design, automatically associating them to the room.

- While defining the shape of the room, use the **SPACEBAR** to cycle through various corner modes. Modes available are: 90 Degrees, 45 Degrees and Any Angle.

- Press **TAB** during placement to open the **Edit Room Definition** dialog, from where you can define the scope and constraints of the room.

- Double-click on a placed room to open the **Edit Room Definition** dialog, in order to edit its rule properties. Alternatively, use the **PCB Rules and Violation panel** dialog to gain access to a particular room definition.

- To copy an existing rectangle room format to a polygon shape room format you can use the copy room format command found in Section 13.2.2 Copying Room Formats for more detail.

13.2.4   Exercise – Using a polygon room and the WithinRoom Query with a BGA

1. Open up the Project `\Altium Designer Summer 09\Examples\Reference Designs\ Daughter Boards - 2 Connector\NBP8 Xilinx Virtex-II Pro BGA456 Rev1.01\ NBP8 Xilinx Virtex-II Pro BGA456 Rev1.01.PrjPcb` . Once the project is open, open the PCB in the project.

2. Select **Tools » Polygon Pours » Shelve polygons**. There should be 16 polygons to shelve. The shelving process hides the polygons from view.

3. Select **Tools » Unroute » All**. You can also do this by using the shortcut **UA**. Click **OK** to allow unrouting of locked primitives if asked.

4. Zoom in to clearly show the BGA. Select **Design » Rooms » Place Polygon Room**. Place the room so it goes around the outline (top overlay) of the BGA component. It does not need to be exact for the exercise.

5. Once placed a room definition will be added to the design rules. Select **Design » Rules** to open the rules dialog.

6. Display the **Placement** section, and then the **Room Definition** section. There should be a room called *RoomDefinition,* change its name to *BGAwidth*, and copy that name so it can be used in a moment.

7. Now go to the **Routing** section of the rules dialog, and display all the **Width** rules.

8. To add a new width design rule, right click on the word **Width** and select **New Rule** from the pop up menu.

9. Rename the new rule to *BGA routing width*, and set the Query to **WithinRoom('***BGAwidth')*,

10. In the Constraint section of the rule, set the **Min**, **Preferred** and **Max** width settings to 0.05mm for all three widths. This defines a width rule for routing within the area of the polygonal room.

11. Click on the **Priorities** button at the bottom of the rules dialog, and check that the new *BGA routing width* has the highest priority.

12. Once all the settings are defined click **Apply** to ensure that there are no problems with the rule settings (the rule name will go red if there are), then click **OK** if there are no errors.

13. Run the command **PT** or **Place » Interactive Routing**, and start routing off one of the pads on the BGA. After you place the first corner outside of the room, the routing width will become wider (as per the width specified by the next applicable width design rule).
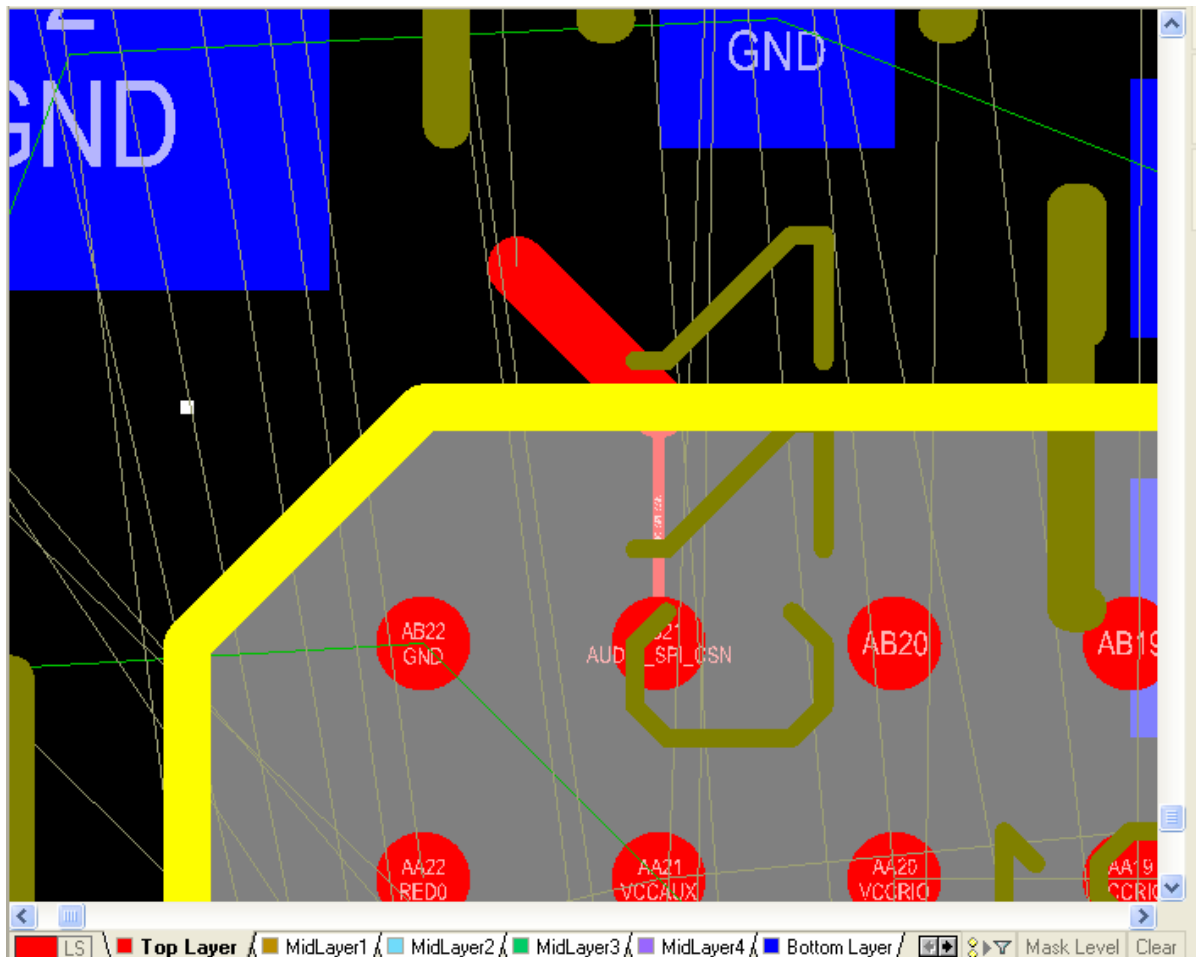

Figure 6. The width change of the track as it leaves the defined polygon room area.

Note: You can also use the WithinRoom query to define rules for clearance and via style for the BGA.