Altium Designer

Advanced Training with Altium 365

Creating Classes from Schematic

Altium ®
TRAINING

Altium ®
TRAINING

Software, documentation and related materials:

Table of Contents

ALTIUM
TRAINING

Creating Classes from Schematic

1 Purpose

In this module, you will explore how to define rules for groups of items, like multiple nets, by creating a net class. Also, you will learn how to set up Directives in the schematic, carry them over to the PCB, and use net classes to apply rules that can handle various design situations.

2 Shortcuts
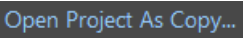
Shortcuts used when working with Creating Classes from Schematic

F1	Help

Schematic:

P » V » ...	Place Directives
P » V » M:	Place Parameter Set
T » R	Parameter Manager

PCB:

D » C	Object Class Explorer
D » R	PCB Rules and Constraint Editor
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `Creating Classes from Schematic.`

3. Select **File » Open Project…** to open the *Open Project* dialog.

4. Enable the folder view button [icon].

5. Navigate to the predefined Training Project `Creating Classes from Schematic (Top\Projects\Altium Designer Advanced Training Course\...).`

6. Select **Open Project as Copy…** `Open Project As Copy…` .

7. In the new dialog *Create Project Copy:*

    a) Add your name to the project name: `Creating Classes from Schematic – [Your Name].`

    b) Add a description: `Altium Advanced Training – [Your name].`

    c) Open the *Advanced* section.

    d) Select the **Ellipsis Button** [...] from the *Folder* configuration to open the *Choose Folder* dialog.

        i) Select the folder with your name: `Project\For Attendees\[Your name].`

        ii) Select **OK**.

    e) Change the **Local Storage** path if needed.

    f) Select **OK** to create the copy.

8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

> Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project*.

4 Investigating Classes

4.1 Checking the Pre-Assigned Classes in the PCB Document

> Note: Although it's not necessary to predefine classes and rules, this will illustrate how to preconfigure classes and rules, and assign members at the schematic level. Initially, the PCB will have no members assigned to these "empty" classes. Also, under *Component Classes*, only the system generated classes are available. Net and component classes will be assigned automatically when you update the PCB from the schematic.

9. In the *Projects* panel, double-click on the file `Classes.PcbDoc` to open it in the PCB editor.

10. Access the command **Design » Classes** to open the *Object Class Explorer*, and click the *All Nets* entry, as seen in Figure 1.

11. The section *Net Classes* shows the predefined AMP_1 to AMP_50 classes and the VOLT_50 to VOLT_240 classes.
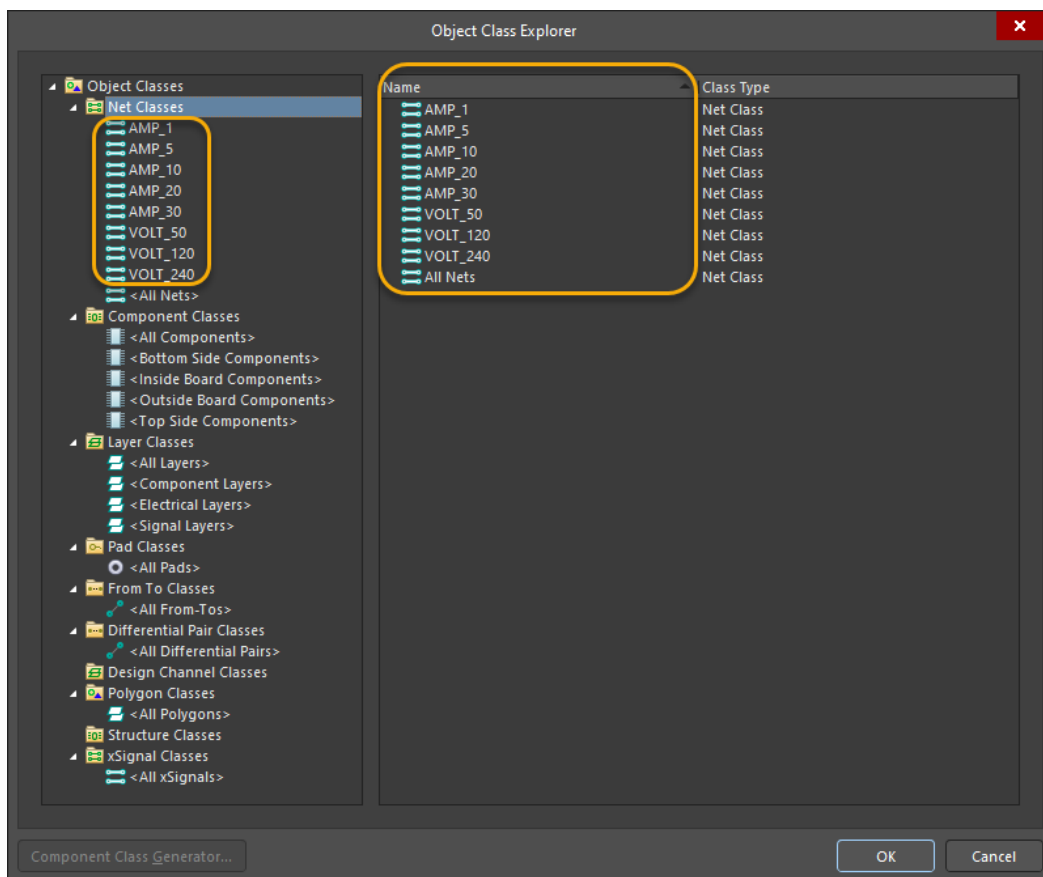

Figure 1. Object Class Explorer, note the pre-defined classes Amp and Volt

12. Select **OK** to close the *Object Class Explorer* dialog.

4.2 Checking the Pre-Assigned Rules

13. Go to **Design » Rules** to open the *PCB Rules and Constraints Editor* as shown in Figure 2.

14. In the pane on the left side of the dialog expand the **Electrical - Clearance** rules, and then the **Routing - Width** rules.

15. From the Electrical Clearance Rules select the **240_VOLT** clearance rule, the rule is targeting the VOLT_240 net class, this rule will apply to any nets which will be added to the Net Class **VOLT_240** when the PCB is updated from the schematic. We will investigate how this is accomplished later in the exercise.

16. Select the **Test Queries** button at the upper right of the rule. Note that the expression and applicable number of objects is 0 for the first query, which is targeting Net Class VOLT_240.

17. Close the *Test Queries Result* dialog with **Close**.

18. In the R*outing* rule section, select the *Width* rule `5_AMP`. This is targeting the AMP_5 Net Class. This rule will apply to any nets which will be added to the Net Class AMP_5. Select the **Test Queries** button and note there is no expression result.

19. Close the *Test Queries Result* dialog with **Close** and close the *PCB Rules and Constraints Editor* dialog.
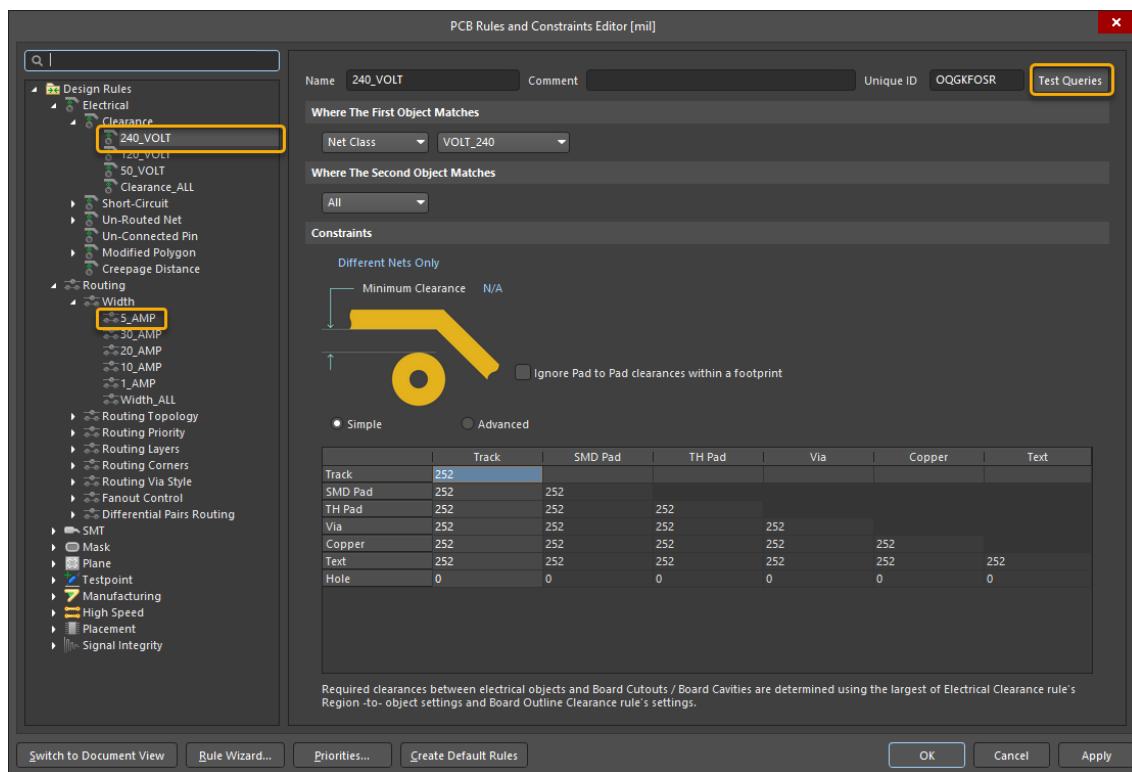

Figure 2. PCB Rules and Constraints Editor

Note: So far, what you have done, could be pre-configured in a template and used for multiple projects.

4.3 Specifying Directives in the Schematic Editor

20. In the *Projects* panel, double click the file `TopSheet_Classes.SchDoc` to open it in the Schematic editor.

21. Note the Directives in the lower left corner of the schematic that are covered by a Compile Mask, as shown in Figure 3. This is done to remove them from the validation process (Netlist).
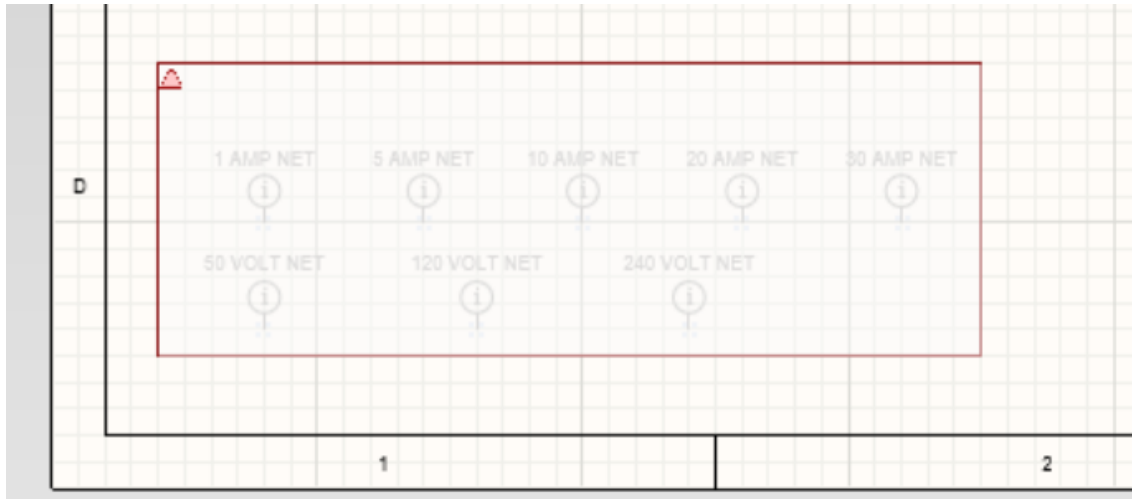

Figure 3. Directives Enclosed by Compile Mask

22. The Directives were predefined on the top-level schematic template and are configured to add any net they are attached to into the corresponding Net Class in the PCB template.

23. Click on the **RED triangle** shaped control in the upper left corner of the Compile Mask to collapse it.

24. Select the **30 AMP Net** directive to select it, and take a look at the *Properties* panel. The directive has a label `30 AMP NET` which is displayed in the schematic. Under the *Parameters section*, the `AMP_30` ClassName parameter defines the Class Name that will be assigned to the connected directive. This name corresponds to the empty class defined in the PCB template document. These predefined directives can be copied and pasted on to various nets in the schematic design to make the class assignments. When the PCB is updated from the schematic, these nets will be added to the class the directive specifies. The Compile Mask has been placed around the predefined directives to remove them from the Compile (Netlist).

25. Re-expand the compile mask by selecting again on the triangle.

Hint: To place your own Net Class directive, use the **Place » Directives » Parameter Set**. Alternatively, Directives can also be placed from the Active Menu.

26. In the *Projects* panel double-click on the file `Classes.SchDoc` to open it in the Schematic editor.

27. This schematic represents the circuitry for a simple fuse board which will need to meet agency compliance for trace, width, and space for branch circuit ampacity and voltage standoff. The directives have already been copied and pasted from the `TopSheet_Classes.SchDoc` to the various nets, where the predefined rules in the PCB document will apply.

7

Altium. TRAINING

5 Component Classes

Component classes can be assigned in the schematic, at the library level, or created in the PCB. In the next steps, you will use the Parameter Manager to add the ClassName parameter to components in the schematic, which will in turn, add them to the specified class. First, you will examine a couple of components that already have a class assignment.

28. Double-click on component `F1` to select it and populate the *Properties* panel.

29. Select the *Parameters* section to see the component parameters. Note the `ClassName` with a value of `120_VOLT`. This will create a new Component Class named `120_VOLT` in the PCB document which the component `F1` will be added to when an update from the schematic to the PCB is performed.

30. Repeat Step 28 and Step 29 for components `F2` and `F3`, noting that these components will be added to a new component class named `240_VOLT`.

31. Now, let's add some Component Class parameters. Open the Parameter Manager using **Tools » Parameter Manager**.

32. In the *Parameter Editor Options* dialog, disable all options in the *Include Parameters Owned By* section, except for **Parts**, and then select **OK**. See Figure 4.
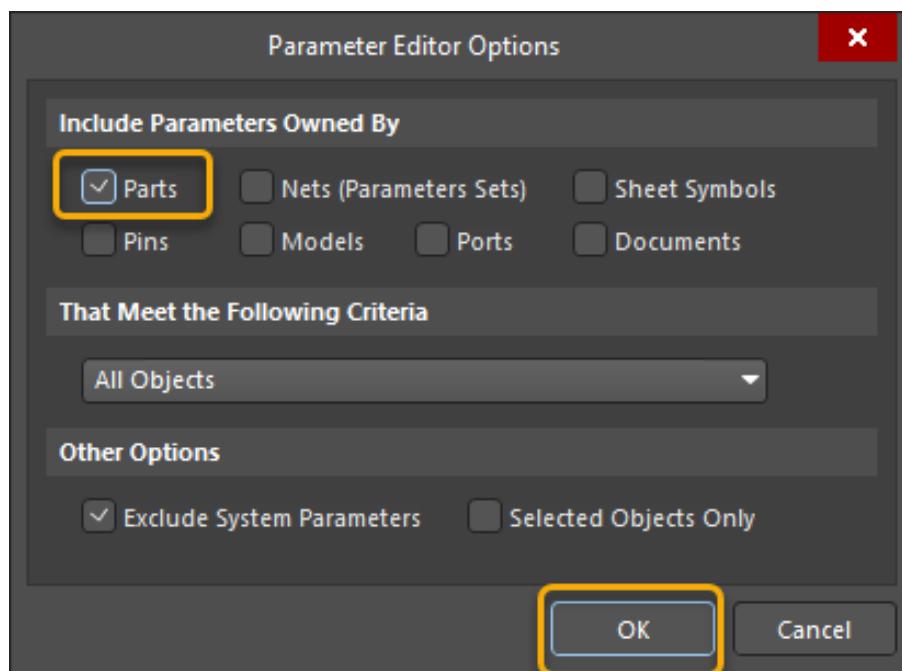

Figure 4. Parameter Editor Options

> Hint: If components are selected before executing the command, the **Selected Objects Only** checkbox can be enabled.

33. The *Parameter Table Editor* will display all parts used in the project. Let's add the ClassName to the T1 through T9 components.

34. Scroll to the right to see the *ClassName* column.

Altium.
TRAINING

35. Select the `T1` through `T9` ClassName column entries, Right-Select and select **Add**. Refer to Figure 5. This will add Plus Symbols for editing, indicated by the arrow and insert image.
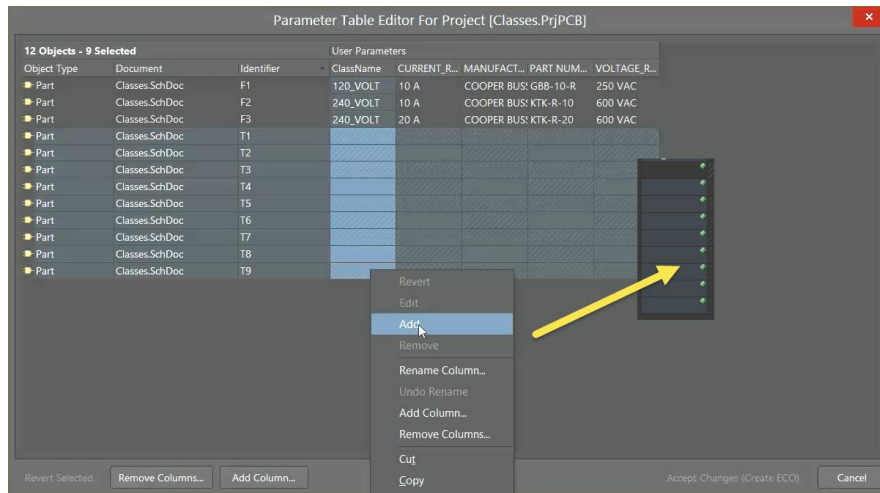

Figure 5. Parameter Table Editor for the Project

36. Locate the column *ClassName* and then select cells for the `T1` through `T4` parts. Right-click on one of the selected ClassName cells and select **Edit**.

37. Type in the value `Left` and press **Enter**.

38. Repeat for the `T5` to `T9` parts, but for the value, enter `Right` instead.

39. Your parameter table should now look similar to what is shown below, Figure 6.
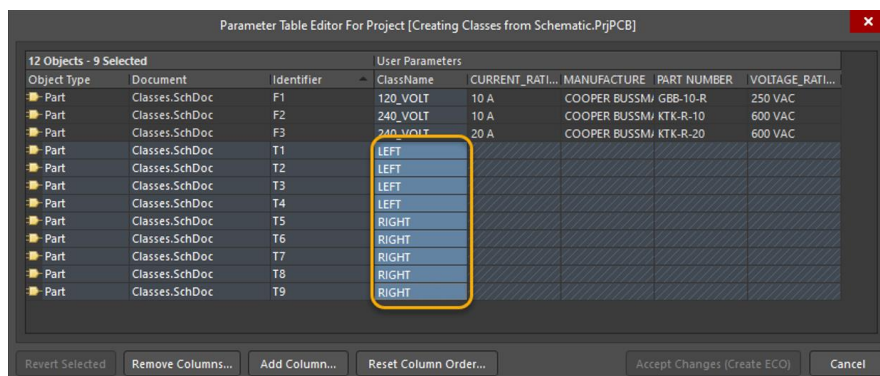

Figure 6. Populated Parameter Table

40. Select **Accept Changes (Create ECO)** to generate an ECO list, Figure 7.
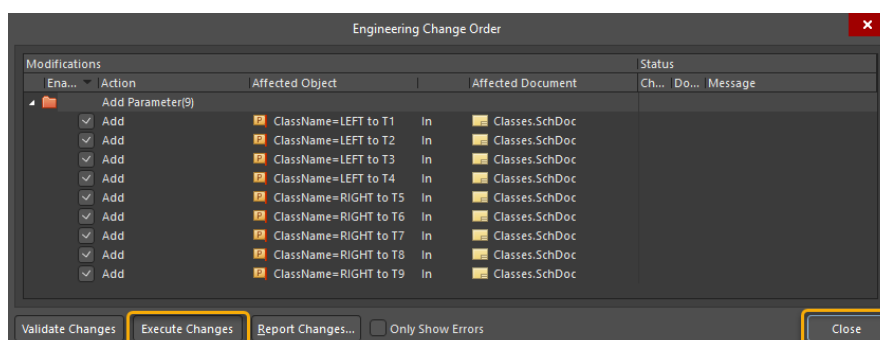

Figure 7. Engineering Change Order

41. Select **Execute Changes** to update your Schematic Documents with the new parameters and then select **Close** to exit the dialog.

6 Updating the PCB

6.1 Automatically Adding Classes to the PCB Design

42. Select the menu option **Project » Project Options** to open the *Options for PCB Project* dialog and select the *Class Generation* tab. The **User-Defined Classes** in the lower portion of the dialog must be enabled, in order for the classes defined in the schematic editor to update to the PCB. Verify if the **Generate Component Classes** and **Generate Net Classes** options are enabled. See Figure 8.
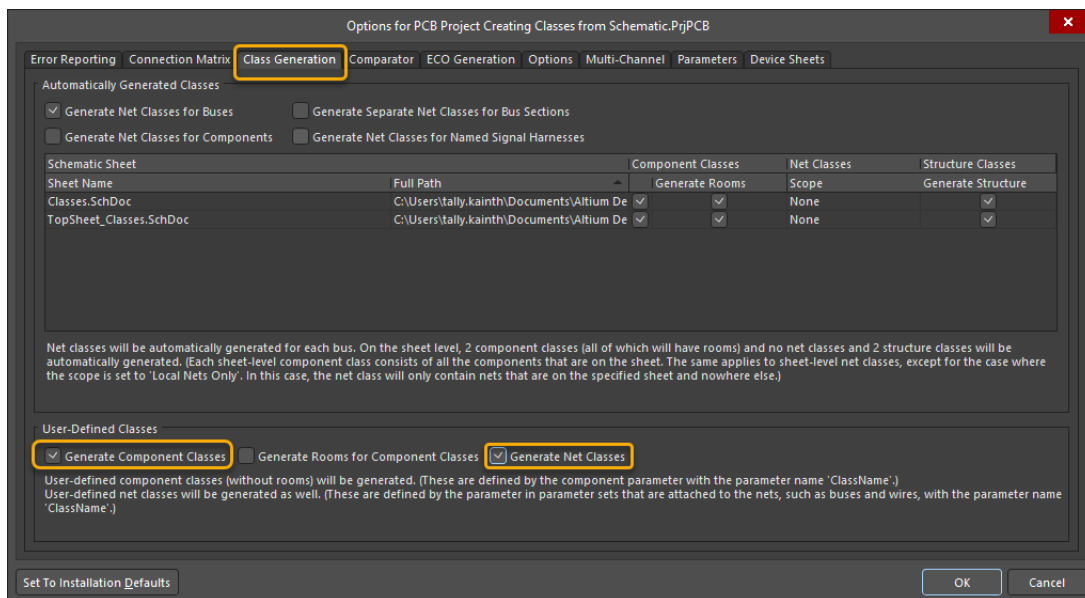

Figure 8. Project Options for User Class Configuration

43. Before closing the Project Options, select the *ECO Generation* tab. Find the option **Remove Net Classes** in the *Modifications Associated with Nets* section and set the mode to **Ignore Differences**. If this is not done, Net Classes defined in the PCB that don't have a corresponding schematic directive, will be removed. See Figure 9.
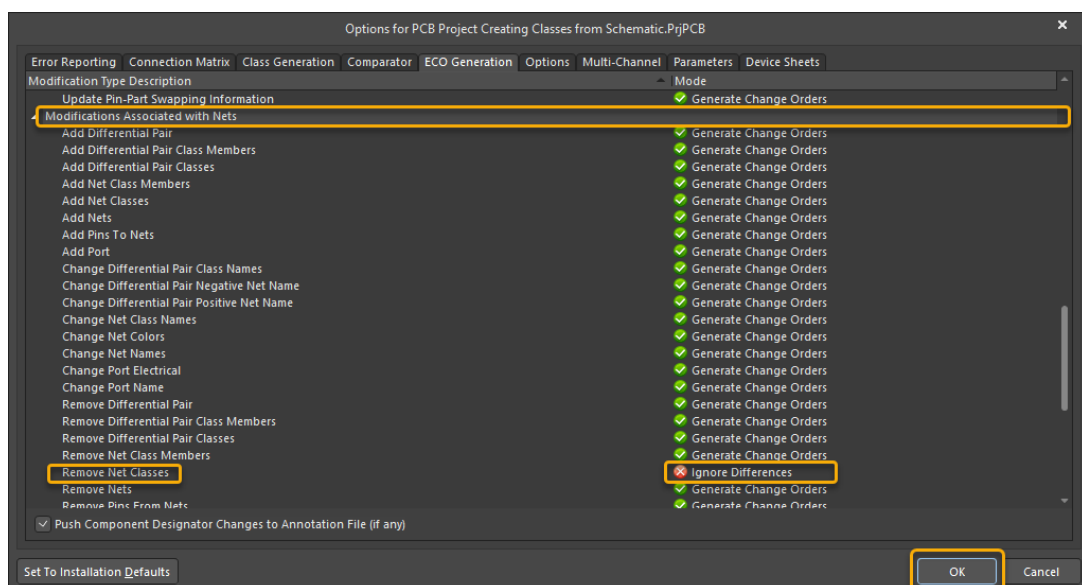

Figure 9. ECO Generation, deactivate Remove Net Classes

44. Select **OK** to close the *Options for PCB Project* dialog.

45. From the schematic editor, select the command **Design » Update PCB Document Classes.PcbDoc**. Alternatively, from the PCB editor, you can use **Design » Import Changes From Classes.PrjPcb**.

46. When the *Engineering Change Order* dialog appears (see Figure 10), examine the items that will be added to the PCB. Note that the component classes are added for the components with the user parameter `ClassName`. Select the **Execute Changes** button to update the PCB document, then select **Close**.
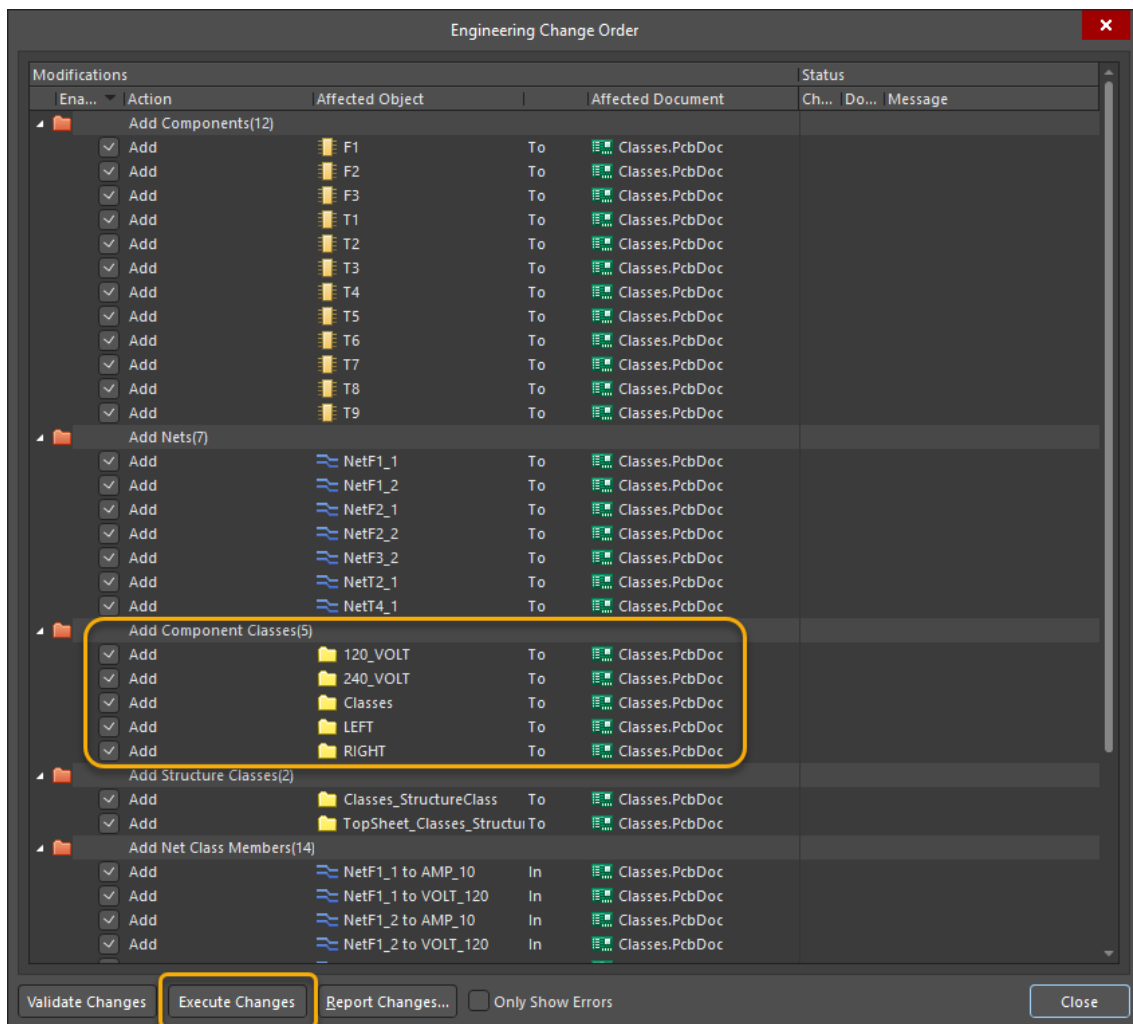

Figure 10. Engineering Change Order

6.2 Investigating the Classes in the PCB Editor

47. Now that the PCB has been updated from the schematic, open the *Object Class Explorer* dialog by accessing the command **Design » Classes**. See Figure 11.
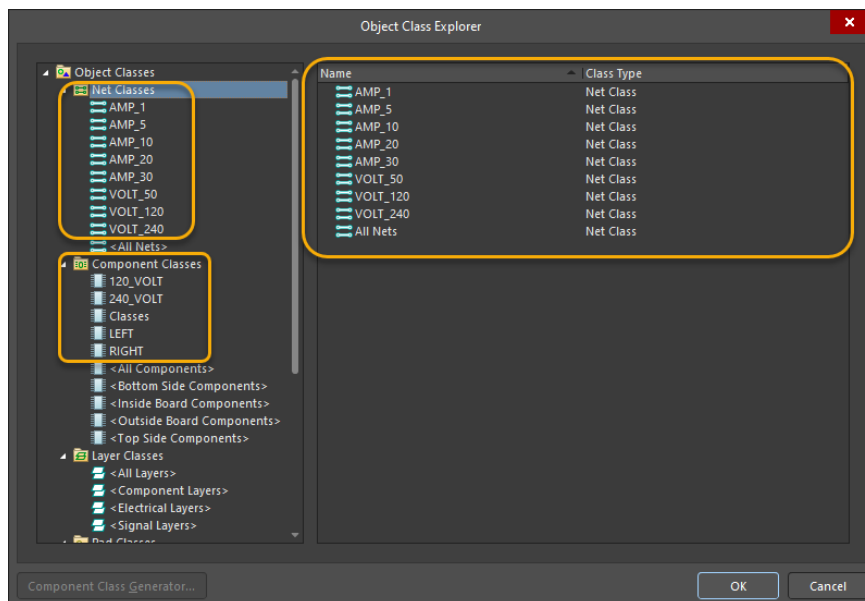

Figure 11. Class Explorer

48. Select the Net Class `AMP_10`. Note that four nets have been added to this class. Select the Component Class `240_VOLT` and note that fuses `F2` and `F3` have been added to this class. This is due to the User Parameter `ClassName` added to the schematic symbols.

49. Select **OK** or **Cancel** to close the *Object Class Explorer* dialog.

50. Open the *PCB Rules and Constraints Editor* by accessing the command **Design » Rules**. See Figure 12.
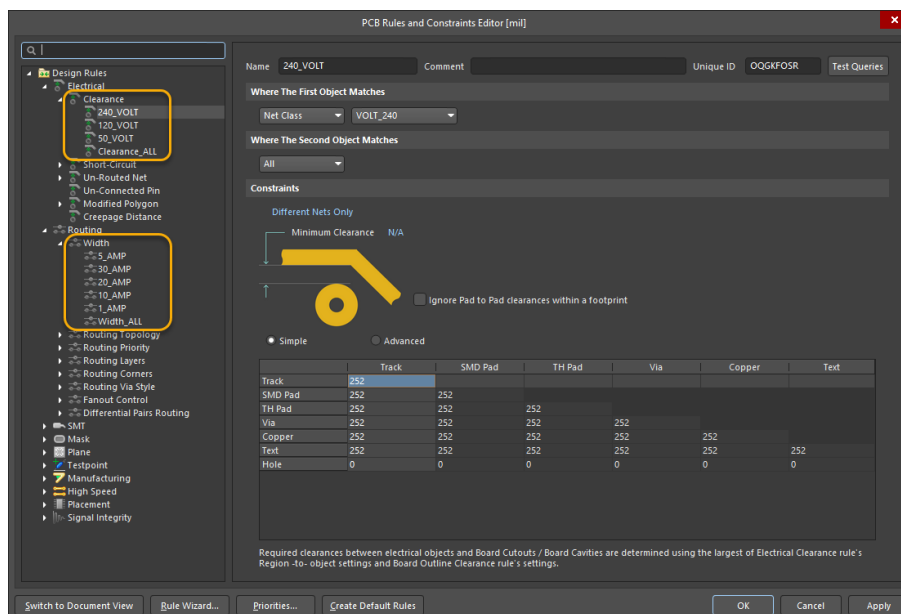

Figure 12. Design Rules

51. Review the existing Clearance and Width rules, remembering that these rules were predefined and scoped to empty classes, which have had members added to them. This was accomplished by updating the PCB from the schematic.

7 Optional: Additional Component Class

52. Change the focus back to the `Classes.SchDoc`.

53. Select the menu option **Place » Directives » Blanket** and create a rectangle around the three fuses. Use Figure 13 as a reference.
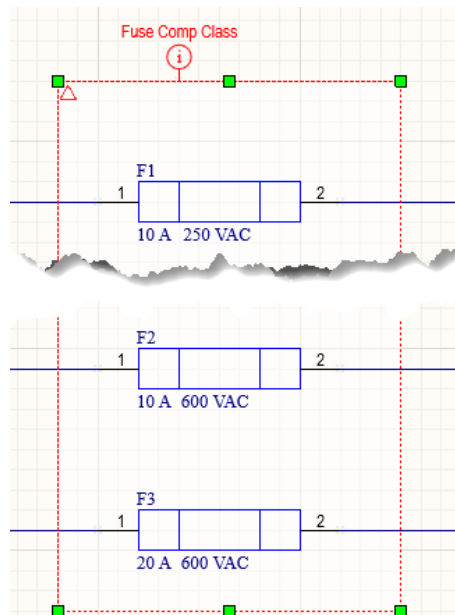

Figure 13. Blanket and Directive for Component Class

54. Select the menu option **Place » Directives » Parameter Set,** press **TAB** to open the P*roperties* panel and add a new Component Class, as seen in Figure 14.
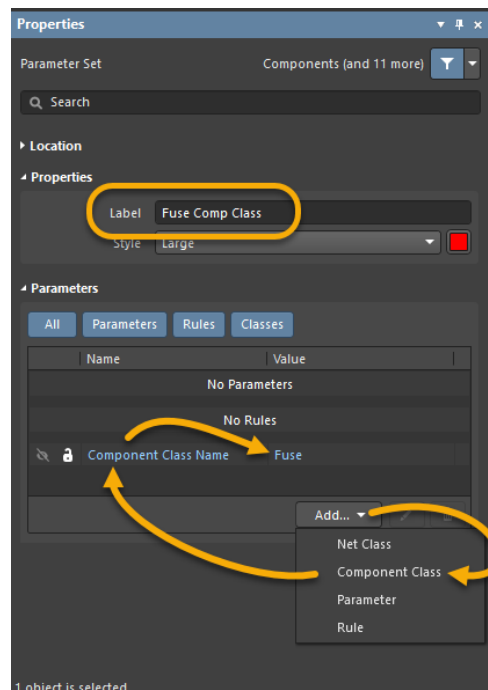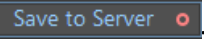

Figure 14. New Component Class based on Blanket

55. Place the Directive as seen in Figure 13.

56. Update the PCB with the new information. After the update is done, you will find the new component class `Fuse` in the PCB (**Design » Classes**). With Blankets and Directives, you can assign components to more than one component class.

57. Save all documents using **File » Save All**.

58. Save the modifications to the server:

    a) In the *Projects* panel, next to the Project name you find the command **Save to Server** Save to Server.

    b) Select **Save to Server**.

    c) In the dialog *Save [Project Name]:*

       i) Add the comment `Creating Classes from Schematic - [Add Your Name] - Finished`.

       ii) Select **OK**.

59. When ready, close the project and any open documents, **Window » Close All**.

Altium.
TRAINING

8 Conclusion

Project templates can be preconfigured to suit a variety of design requirements. In this example, Classes and Design Rules were preconfigured for the PCB by adding directives to the schematic. This was done to facilitate a set of PCB design rules adhering to a corporate or regulatory compliance standards. By using this strategy, the designer doesn't need to have any understanding of these standards, as they have been preconfigured at a project level. All the designer needs to understand, is the practical application, or specifically, the current and voltage requirements of various nets. Everything pertaining to requirements has been preconfigured in the Design Rules by the design engineer from the schematic side.

Congratulations on completing the Module!

Creating Classes from Schematic

from

Altium Designer Advanced Training
with Altium 365

Thank you for choosing **Altium Designer**

Altium®
TRAINING