

Altium Designer

Advanced Course

Module:
Creating Classes from Schematic

Software, documentation and related materials:

Copyright © 2022 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, Altium 365™, Altium Concord Pro™, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, xSignals®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter™, X2®, xSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Investigating Classes	3
1.4.1 Check the Pre-Assigned Classes in the PCB Document.....	3
1.4.2 Check the Pre-Assigned Rules	4
1.4.3 Specifying Directives in the Schematic Editor	5
1.5 Component Classes	6
1.6 Updating the PCB	8
1.6.1 Automatically adding Classes to the PCB Design	8
1.6.2 Investigate the Classes in the PCB Editor	9
1.7 Optional: Additional Component Class	10
1.8 Conclusion	11

Creating Classes from Schematic

1.1 Purpose

Rules are often defined in relation to specific items, for example a specific net. What if we want to target a group of nets? In that situation we will want to set up a net class. Let us see how to define Directives in the schematic that can then be carried over to the PCB, and how rules using net classes can be leveraged to handle all kinds of situations.

1.2 Shortcuts



Shortcuts when working with Creating Classes from Schematic

F1: Help

PCB

D-C: Object Class Explorer

CTRL+S: Save Document

1.3 Preparation

1. Close all existing projects and documents.
2. Open the `Creating Classes from Schematic.PrjPCB` project found in its respective folder of the Advanced Training.

1.4 Investigating Classes

1.4.1 Check the Pre-Assigned Classes in the PCB Document



Although it is not necessary to predefine classes and rules, this will illustrate how to preconfigure classes and rules and how to 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 we update the PCB from the schematic.

3. In the *Projects* panel double click on the file `Classes.PcbDoc` to open it in the PCB editor.
4. Access the command **Design » Classes** to open the *Object Class Explorer* as shown in Figure 1.
5. The section *Net Class* shows the predefined AMP_1 to AMP_50 classes and the VOLT_50 to VOLT_240 classes.

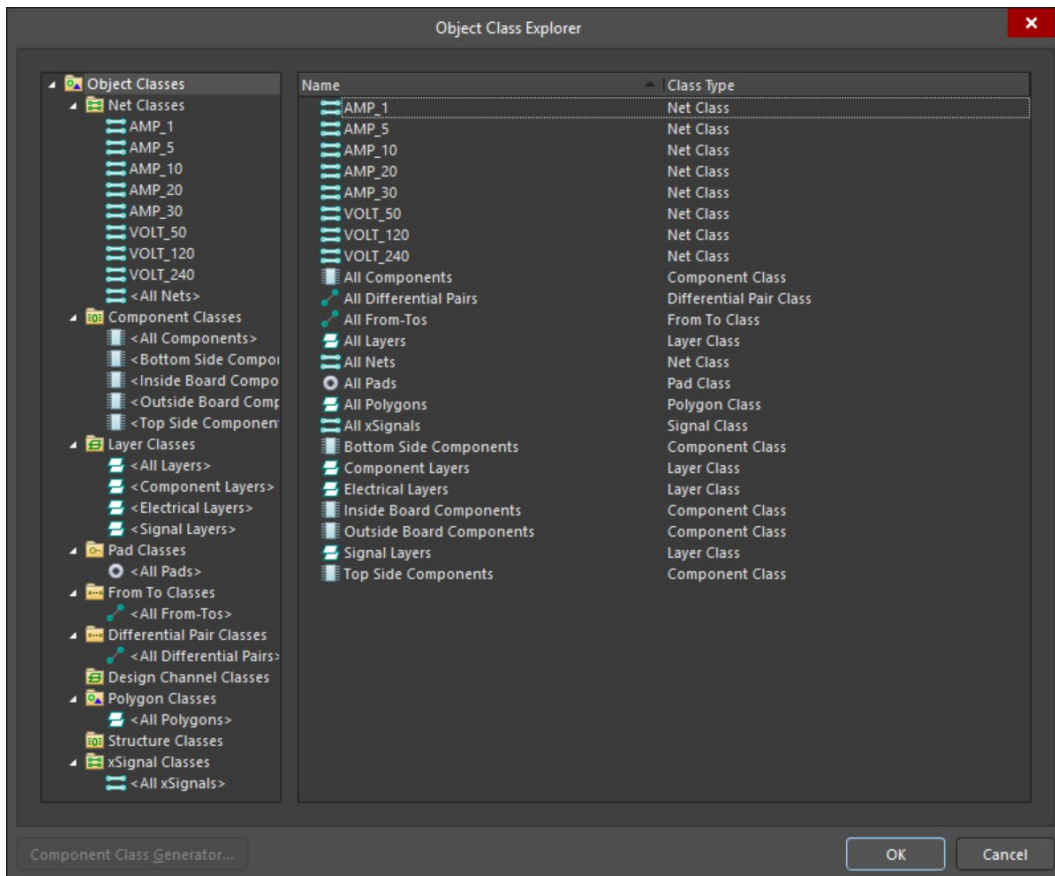


Figure 1. Object Class Explorer, note the pre-defined classes Amp and Volt

6. Click **OK** to close the *Object Class Explorer* dialog.

1.4.2 Check the Pre-Assigned Rules

7. Go to **Design » Rules** to open the *PCB Rules and Constraints Editor* as shown in Figure 2.
8. In the pane on the left side of the dialog expand the **Electrical - Clearance** rules, and then the **Routing - Width** rules.
9. Double click on 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.
10. Click 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.
11. Close the *Test Queries Result* dialog with **Close**.
12. In the *Routing* 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. Click the **Test Queries** button and note there is no expression result.
13. Close the *Test Queries Result* dialog with **Close** and close the *PCB Rules and Constraints Editor* dialog.

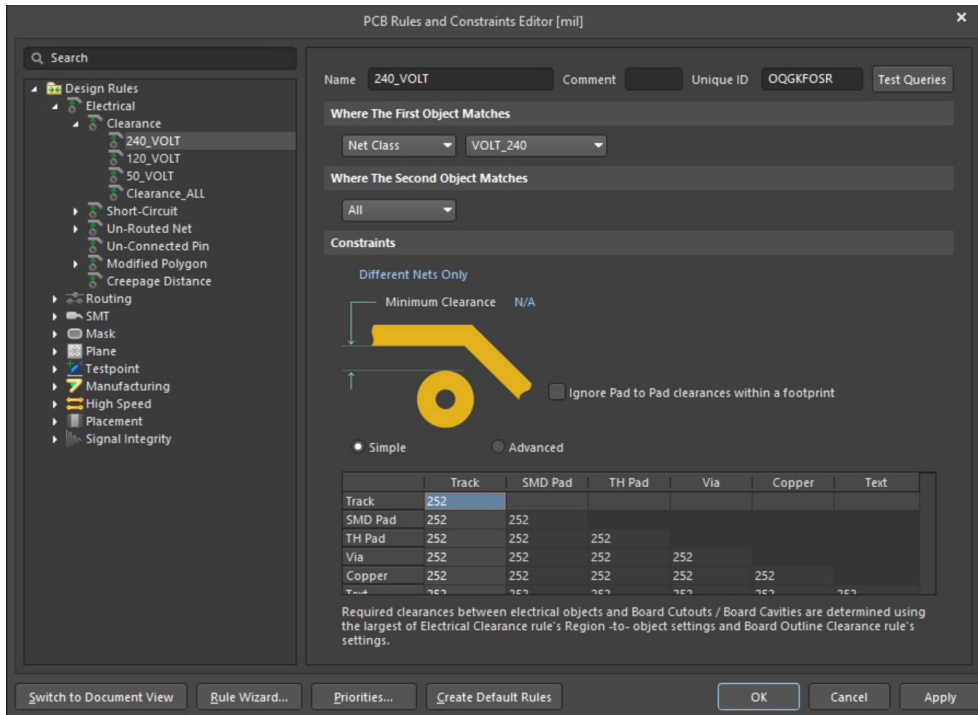


Figure 2. PCB Rules and Constraints Editor



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

1.4.3 Specifying Directives in the Schematic Editor

14. In the *Projects* panel, double click on the file *TopSheet_Classes.SchDoc* to open it in the Schematic editor.
15. 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).

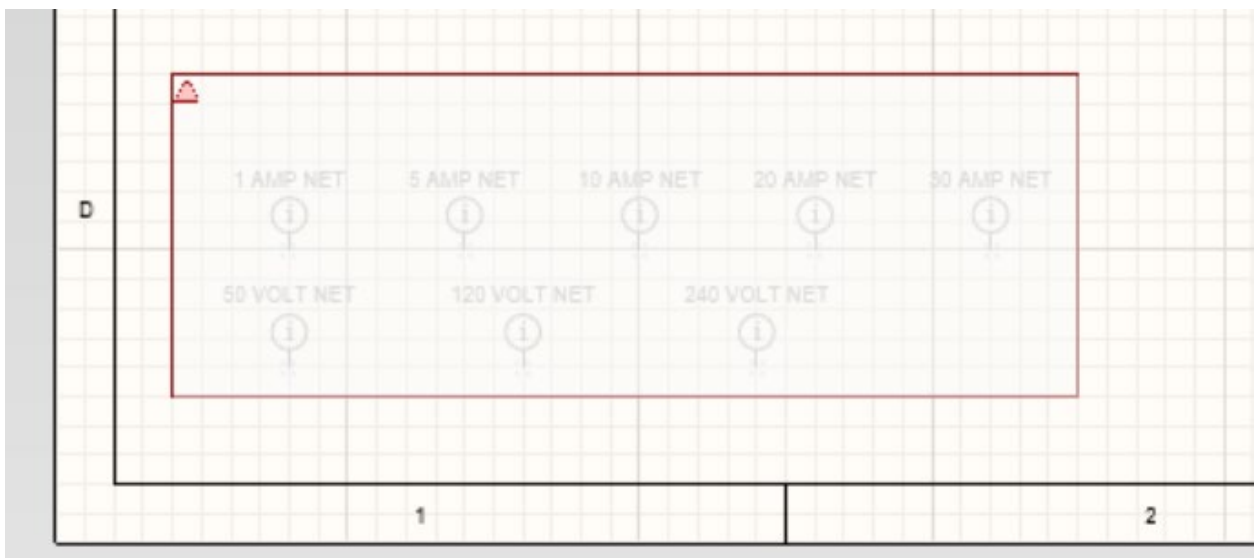


Figure 3. Directives Enclosed by Compile Mask

16. 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.
17. Click on the **RED triangle** shaped control in the upper left corner of the Compile Mask to collapse it.
18. Click on 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).
19. Re-expand the compile mask by clicking again on the triangle.



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



20. In the *Projects* panel double click on the file `Classes.SchDoc` to open it in the Schematic editor.
21. 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 compile mask to the various nets where the predefined rules in the PCB document will apply.

1.5 Component Classes

Component classes can be assigned in the schematic, at the library level, or created in the PCB. In the next steps, we 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, we will examine a couple of components that already have a class assignment.

22. Click on component `F1` to select it and populate the *Properties* panel.
23. Click on the *Parameters* section to see the component's 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.
24. Repeat Step 22 for components `F2` and `F3`, noting that these components will be added to a new component class named `240_VOLT`.
25. Now, let us add some Component Class parameters. Open the Parameter Manager: **Tools » Parameter Manager**.
26. In the *Parameter Editor Options* dialog, disable all options in the *Include Parameters Owned By* section except for **Parts** and then click **OK**. See Figure 4.

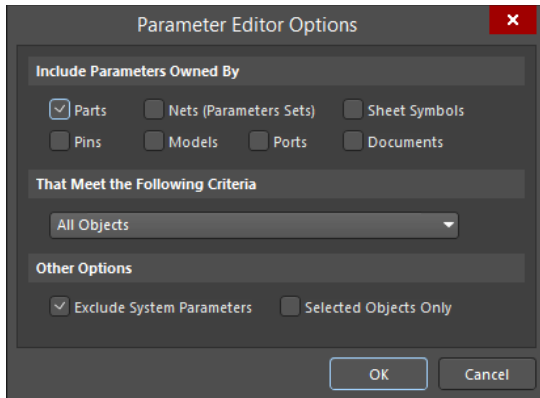


Figure 4. Parameter Editor Options



If the components to be modified are selected prior to invoking the command, the 'Selected Objects Only' checkbox could be enabled.

27. The *Parameter Table Editor* will display all parts used in the project. Let's add the *ClassName* to the T1 through T9 components.
28. Scroll to the right to see the *ClassName* column.
29. Select the T1 through T9 *ClassName* column entries, Right Click and select **Add**. Refer to Figure 5. This will add Plus Symbols for editing, indicated by the arrow and inset image.

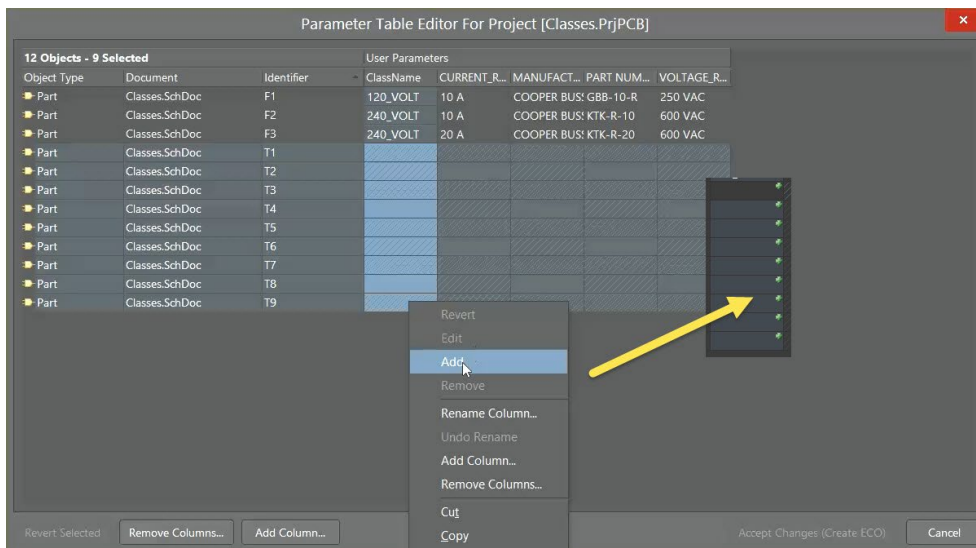


Figure 5. Parameter Table Editor for the Project

30. 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**.
31. Type in the value `Left` and press **Enter**.
32. Repeat for the T5 to T9 parts, but for the value, enter `Right` instead.
33. Your parameter table should now look similar to what is shown below, Figure 6.

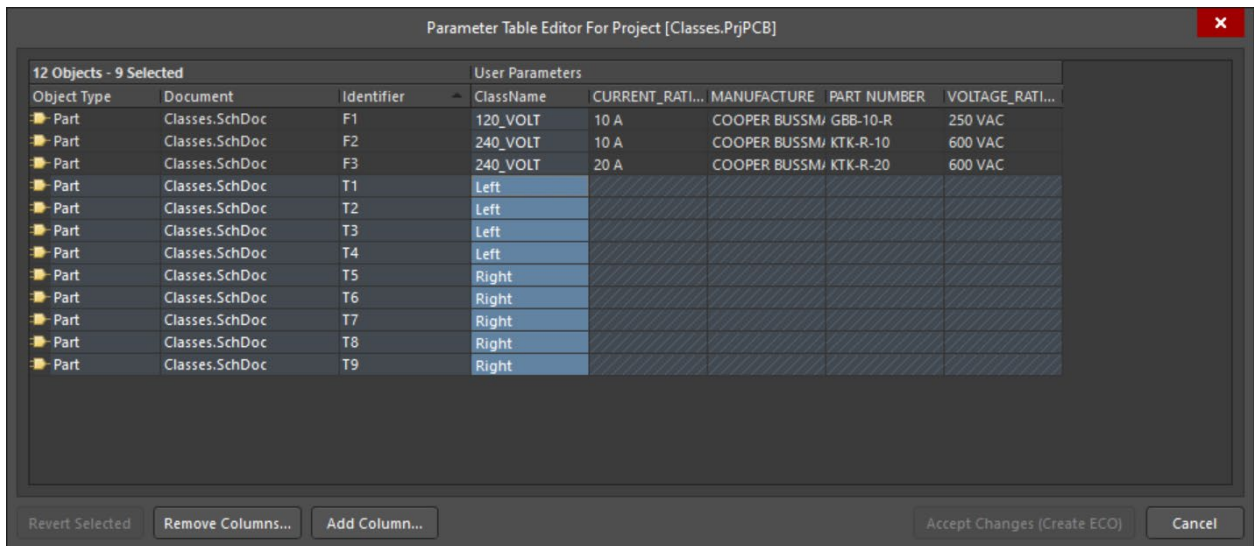


Figure 6: Populated Parameter Table

34. Click the **Accept Changes (Create ECO)** to generate an ECO list (Figure 7).

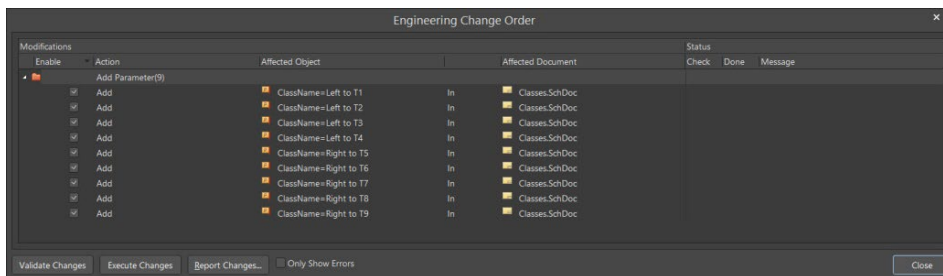


Figure 7. Engineering Change Order

35. Click **Execute Changes** to update your Schematic Documents with the new parameters and then click **Close** to exit the dialog.

1.6 Updating the PCB

1.6.1 Automatically adding Classes to the PCB Design

36. Select the menu option **Project » Project Options** to open the *Options for PCB Project* dialog and click on the *Class Generation* tab. In order for the classes defined in the schematic editor to update to the PCB, the **User-Defined Classes** in the lower portion of the dialog must be enabled. Verify that the **Generate Component Classes** and **Generate Net Classes** options are both enabled.
37. 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 do not have a corresponding schematic directive will be removed.
38. Click **OK** to close the *Options for PCB Project* dialog.
39. From the schematic editor select the command: **Design » Update PCB Document Classes.PcbDoc**. Alternately, from the PCB editor, you can use: **Design » Import Changes From Classes.PrjPcb**.

40. When the *Engineering Change Order* dialog appears (see Figure 8), examine the items that will be added to the PCB. Note that the component classes are being added for the components with the user parameter `ClassName`. Click the **Execute Changes** button to update the PCB document, then click **Close**.

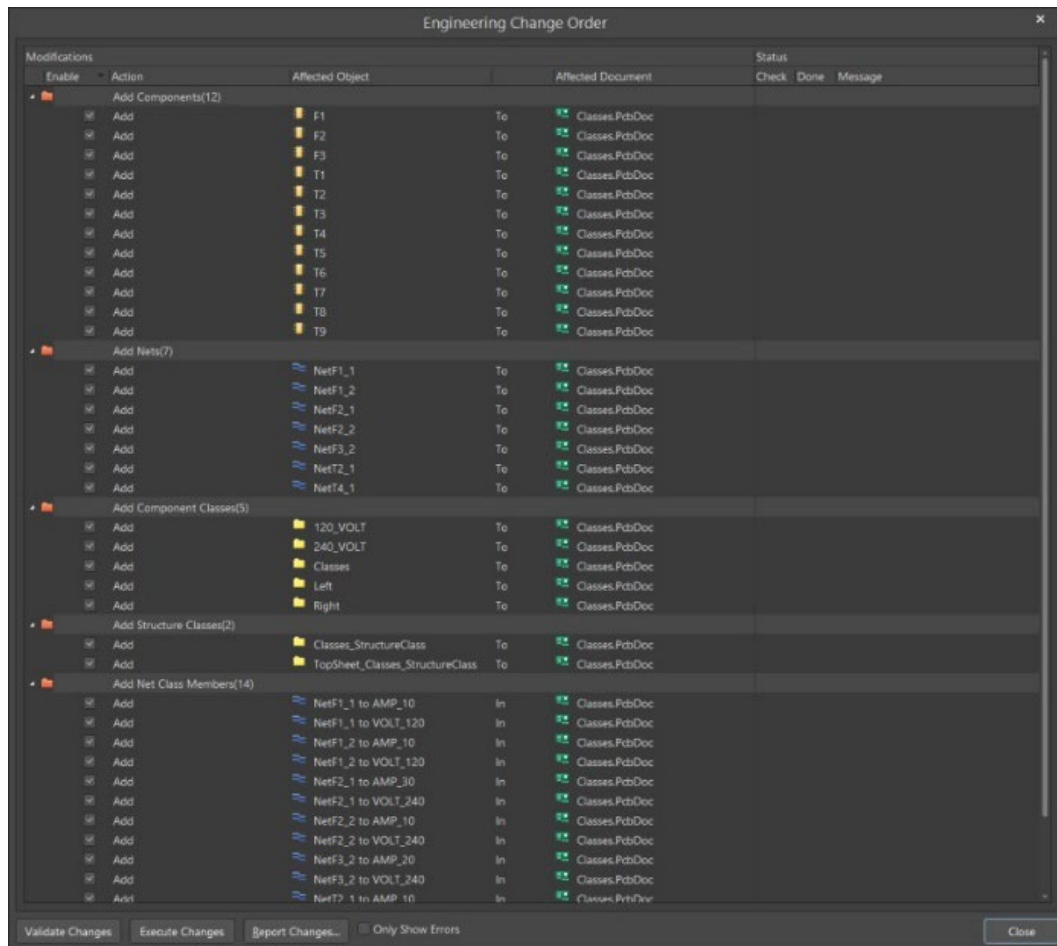


Figure 8. Engineering Change Order

1.6.2 Investigate the Classes in the PCB Editor

41. Now that the PCB has been updated from the schematic open the *Object Class Explorer* dialog by accessing the command **Design » Classes**.
42. 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. Again, this is due to the User Parameter `ClassName` being added to the schematic symbols.
43. Click **OK** or **Cancel** to close the *Object Class Explorer* dialog.
44. Open the *PCB Rules and Constraints Editor* by accessing the command **Design » Rules**.
45. Review the existing Clearance and Width rules remembering that these rules were predefined and scoped to empty classes which have just had members added to them. This was simply accomplished by updating the PCB from the schematic.

1.7 Optional: Additional Component Class

46. Select the menu option **Place » Directives » Blanket** and create a rectangle around the three fuses, use Figure 10 as a reference.
47. Select the menu option **Place » Parameter Set**, press **TAB** to open the *Properties* panel and add a new Component Class, as seen in Figure 9

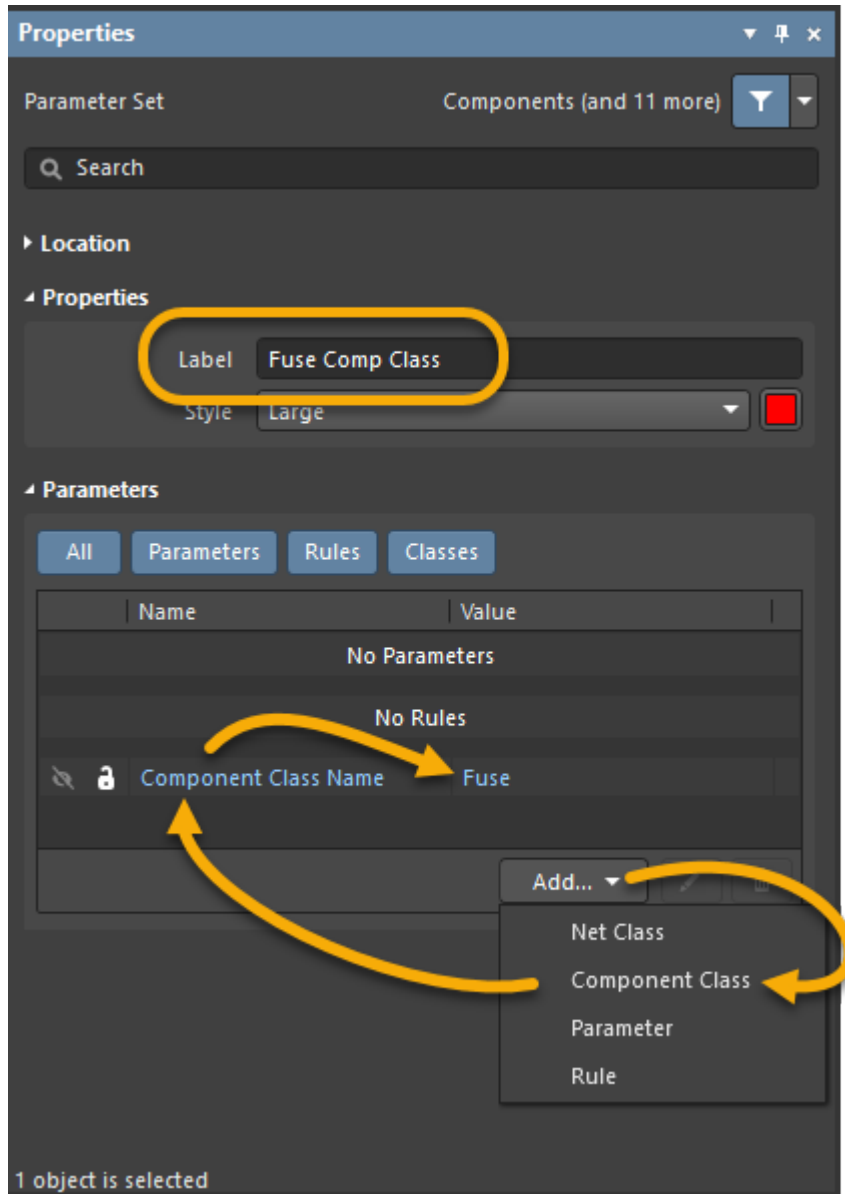


Figure 9. New Component Class based on Blanket

48. Place the Directive as seen in Figure 10.
49. Update the PCB with the new information. After the update was 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 on component class.

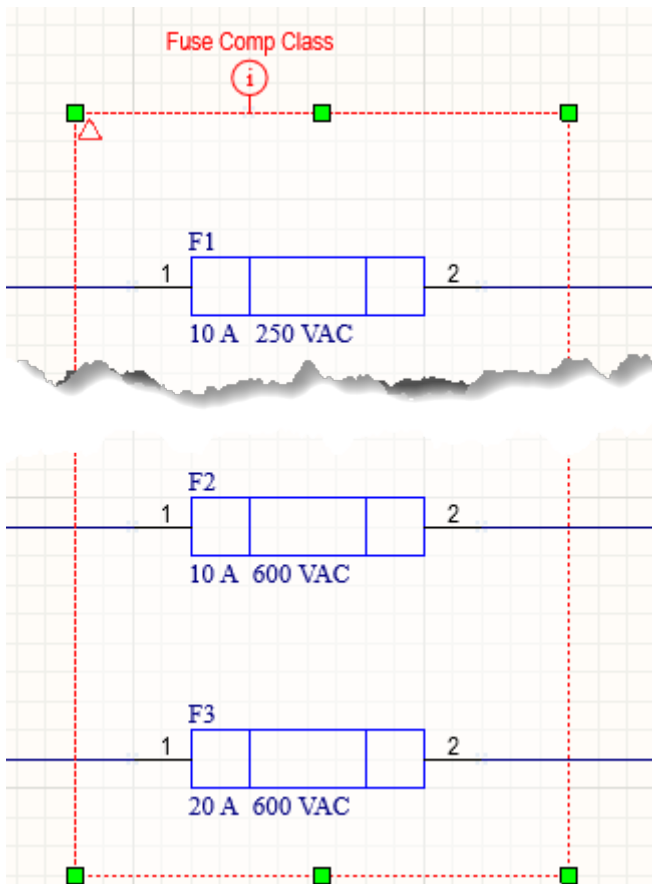


Figure 10. Blanket and Directive for Component Class

1.8 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 need not 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 module:

Creating Classes from Schematic

from the
Altium Designer Advanced Course

Thank you for choosing Altium Designer