ALTIUM DESIGNER

ALTIUM 365

Altium Designer

Advanced Training with Altium 365

Creating Classes from Schematic with Constraint Manager

ALTIUM® TRAINING

Altium® TRAINING

Software, documentation and related materials:

1

Table of Contents

Altium.
TRAINING

Creating Classes from Schematic with Constraint Manager

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 need to set up a net class. Let's see how to define constraints 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.

> Caution: The Constraint Manager is only available if you are using an Altium Designer Pro License. Note that the Constraint Manager is not supported with the Altium Designer Standard Subscription. For details, please see our [Online Documentation](#).

> Caution: Using the Constraint Manager replaces the functionality of defining Rules and Classes with Directives at the schematic for nets.

2 Shortcuts
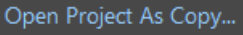
Shortcuts used when working with Creating Classes from Schematic with Constraint Manager

F1	Help – Shortcut Key List
CTRL+S	Save Document
Schematic:	
T » R	Parameter Manager
D » G	Constraint Manager (SCH)
PCB:	
D » C	Object Class Explorer
D » R	Constraint Manager (PCB)
CTRL+Q	Change Unit

3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Creating Classes from Schematic with Constraint Manager`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project `Creating Classes from Schematic with Constraint Manager (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 with Constraint Manager – [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 Check Existing Classes in the PCB Document

Note: When you open the *Object Classes Explorer*, you will notice that 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 `Creating Classes from Schematic with CM.PcbDoc` to open it in the PCB editor.

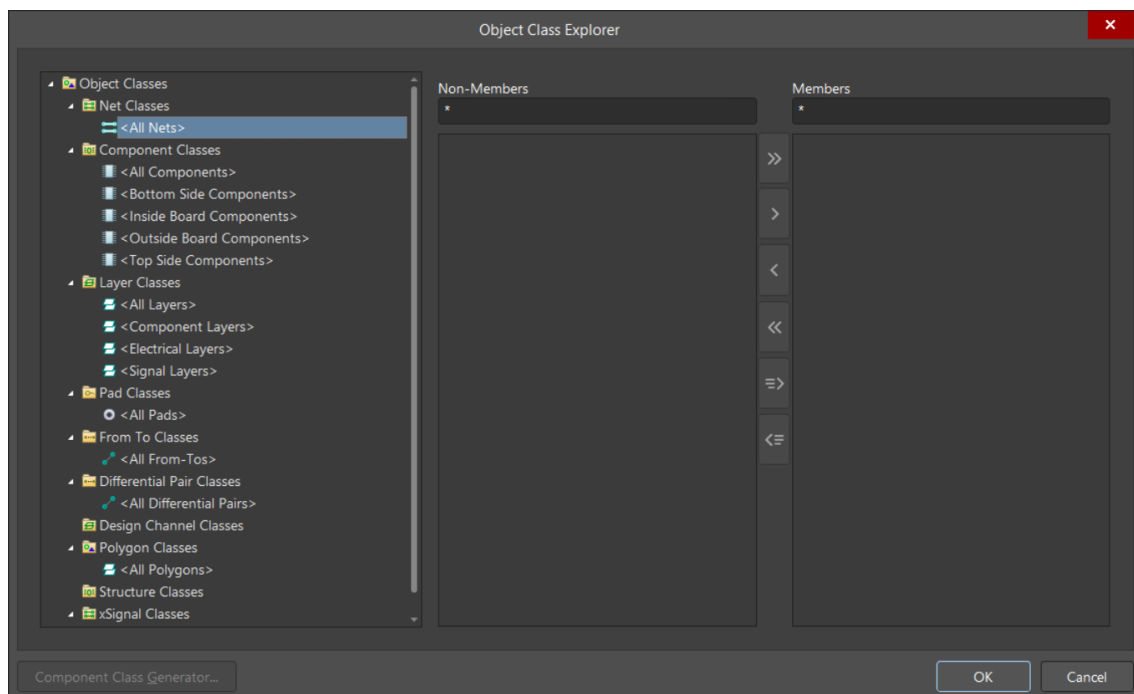10. Go to **Design » Classes** to open the Object Class Explorer, as shown in Figure 1.

Figure 1. Object Class Explorer, note no pre-defined classes Amp and Volt

11. Observe that no user-defined Net or Component classes exist yet.

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

4.2 Constraint Manager – Introduction

The Constraint Manager is a document-based, spreadsheet-like user interface that allows you to view, create, and manage the design constraints used for your PCB designs. By using a document-based presentation interface, rather than a dialog, means that the schematic and PCB editor and its associated functions remain active and accessible. You can access the Constraint Manager from the main menus of the schematic or PCB editor.

4.2.1 Constraint manager: PCB

13. With the PCB as active document: Go to **Design » Constraints Manager (D»R)** to open the *Constraints Manager* dialog, as shown in Figure 2.

   a) The Tab for the PCB Constraint view has the syntax "*.PCBDOC[Constraint]"
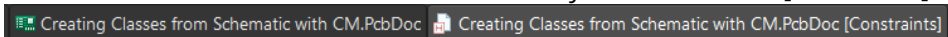
   b) Along the top, there are four constraint types. Select each of the constraint types, starting with Clearance.

     i) `Clearances` A matrix that defines the electrical clearances between net classes and/or differential pairs.

     ii) `Physical` A list of nets, differential pairs, xNets and their classes where you can define physical constraints for the design: widths of conductors, the gap in differential pairs, etc. When the Constraint Manager is accessed from the PCB, rooms currently defined in the PCB document are listed here, and you can define physical constraints for the rooms.

     iii) `Electrical` A list of nets, xNets, xSignals and their classes, where you can define electrical constraints for the design: topology, impedance, etc. xSignals are also listed here on a dedicated tab.

     iv) `All Rules` A list of all rules in the PCB design. From here, you can create custom (or advanced) rules that feature more complex query expressions in their matching scope.

   c) Below, see the views of the four constraint types - PCB View -, showing the rules and constraints currently using the default values for the design.
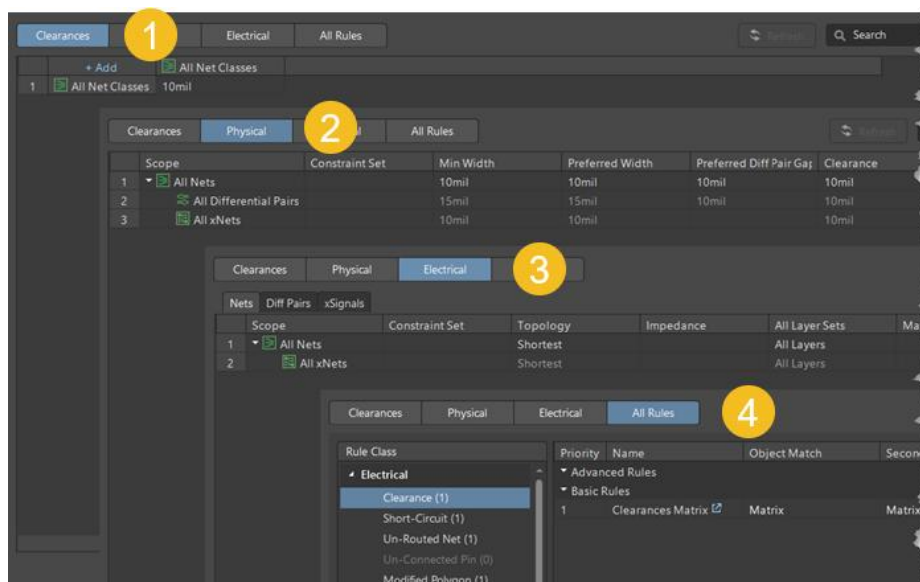
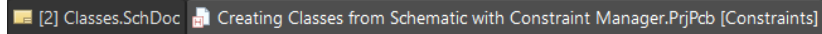*Figure 2. The four Constraint Views for PCB Constraint Manager*

6

4.2.2   Constraint manager: SCH

Similar to the PCB, the Schematic also allows you to open the Constraint Manager. In the view of the Schematic you see three Constraint Types.

14. Open the schematic document `Classes.SchDoc`.

15. With the schematic as active document: Go to **Design » Constraint Manager (D»G)** to open the *Constraints Manager* dialog, as shown in Figure 3.

   a) The tab for the SCH Constraint view has the syntax "*.PrjPcb[Constraint]"

   [2] Classes.SchDoc   Creating Classes from Schematic with Constraint Manager.PrjPcb [Constraints] .

   b) Along the top, there are three constraint types   Clearances ,   Physical ,   Electrical .
   Select each of the constraint types, starting with Clearance to see the three constrain types. The three constraint types have the same functionality as the PCB Constraint seen at step 13b.
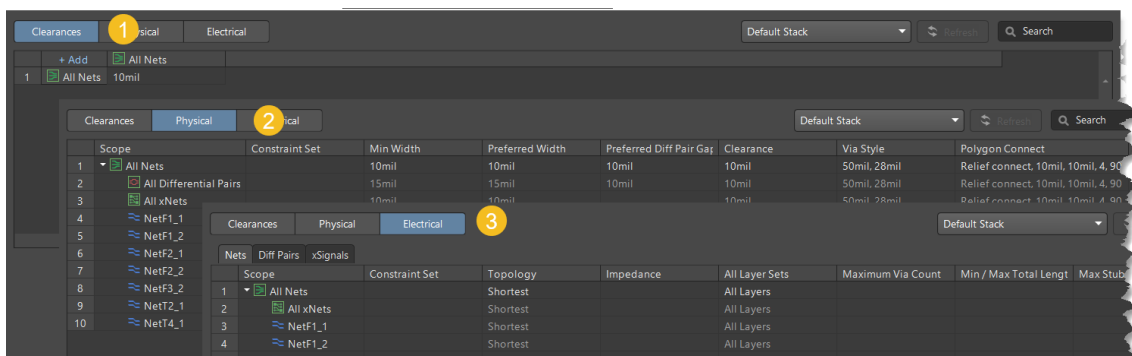

Figure 3. The three Constraint Views for SCH Constraint Manager

4.3 Checking Pre-Assigned Rules and Constraints

When using the Constraint Manager from the schematic, you can set the Width for layers in your selected layer stack.

16. Still in the Constraint Manager from the Schematic, select the *Physical* constraint view, and use the drop-down menu to select a specific PCB document from the design project, Figure 4.
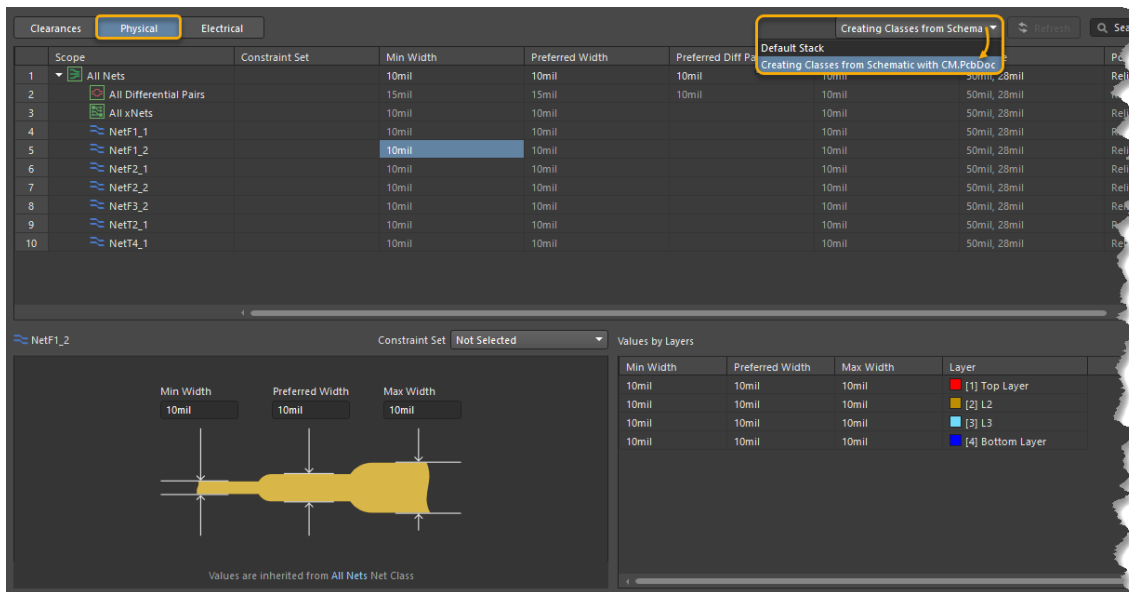

Figure 4. Selecting the required stack

17. Switch to the *Clearances* view, and select the *All Nets* cell to see details for this constraint at the bottom of the window, see Figure 5.
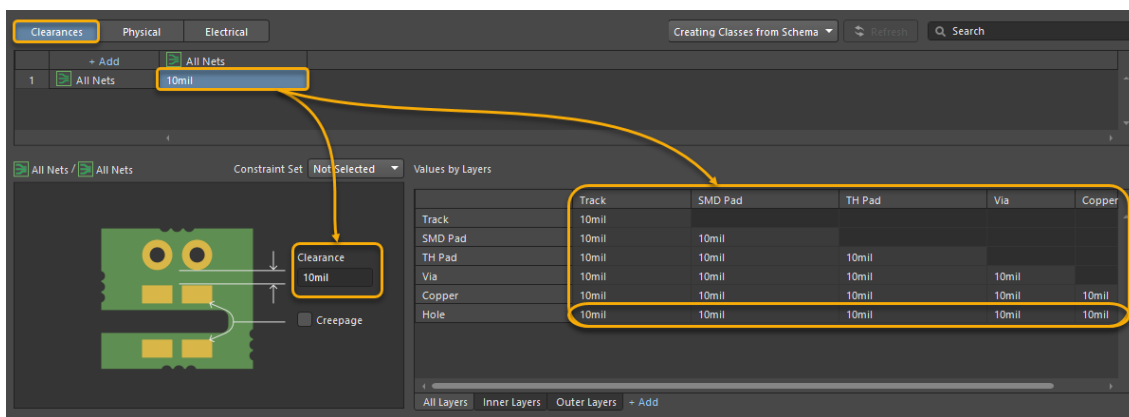

Figure 5. Clearance Constraint View - All versus All

18. Close the *Constraint Manager* opened from the schematic.

19. If the *Unsaved Changes* dialog window pops up, select **Save**.

5 Specifying Constraints in the Schematic Editor

20. Ensure the file `Classes.SchDoc` is open in the Schematic editor.

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.

Next, you will add classes, rules, and constraints for the PCB using the Schematic Constraint Manager.

5.1 Net Classes

Net classes can be assigned in the schematic or created in the PCB. In the next steps, you will use the Constraint Manager, opened from the schematic, to create net classes and constraints.

5.1.1 Constraint Manager

21. Go to **Design » Constraint Manager**.

22. Ensure you have selected the right stack from the drop-down list at the top.

23. The constraint manager opened from the Schematic will show three constraint views: *Clearance, Physical,* and *Electrical*. Select the constraint type *Clearance* Clearances at the top.

24. First, you will create a net-class `AMP_10`. Then, you will define a rule for that net class.

25. Right-click anywhere in the table and select **Classes » Add Class,** Figure 6. The *Add Classes* dialog will appear.
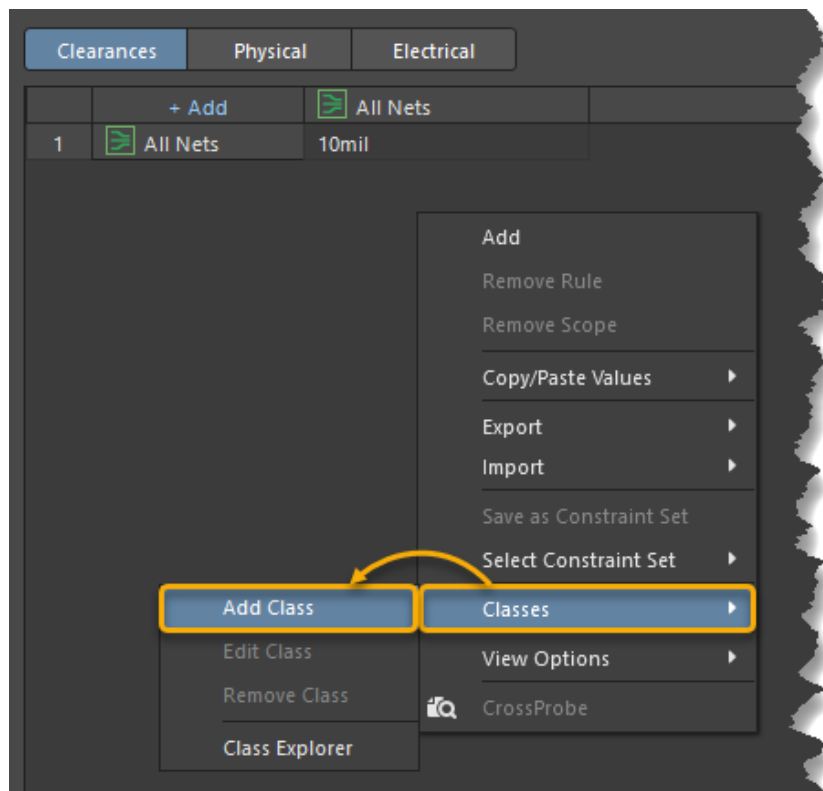

Figure 6. New Class

26. In the *Add Class* dialog*:*

    a) Add the name `AMP_10`.

    b) Select the nets `NetF1_1, NetF1_2, NetF2_2,` and `NetT2_1` from the *Non-Members* list and use the arrow to add them to the *Members* list.

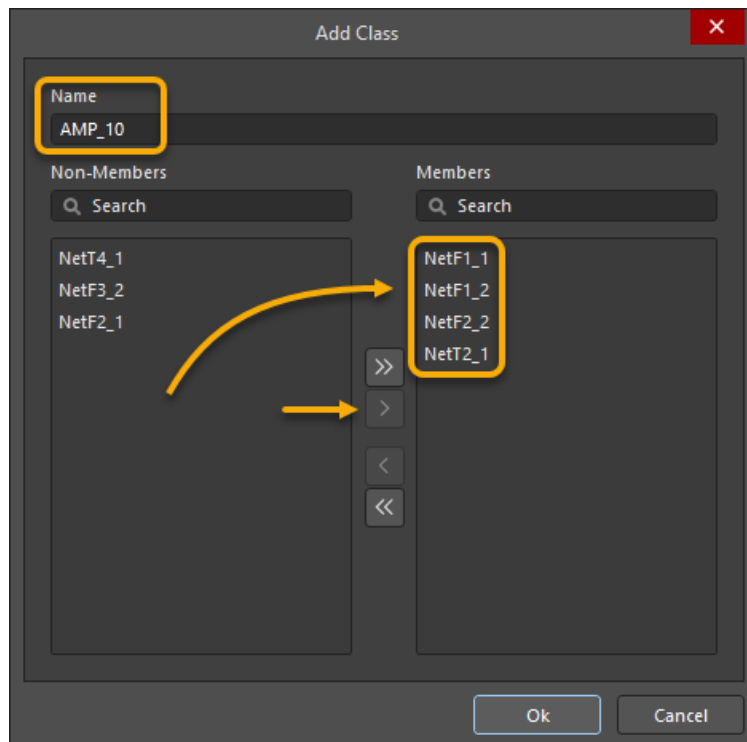   c) Select **OK** to close the dialog, as seen in Figure 7.

Figure 7. Add Class members to a Class

5.1.2 Object Class Explorer

27. Next, you will create the net classes `AMP_20`, `AMP_30`, `Voltage_120` and `Voltage_240`. Instead of adding the net classes one by one, use the *Object Class Explorer*.

28. Right-click anywhere in the table and select **Classes » Class Explorer**. The *Class Explorer* dialog will appear, see Figure 8.

29. Inside the Object Class Explorer, Figure 8:

   a) Select <All Nets> on the left. Then, right-click and select **Add Class** to add a new class.

   b) Add the class name `Voltage_240`.

   c) Select the Nets `NetF2_2`, `NetF2_1`, `NetF3_2` and `NetT4_1` from the *Non-Members* and transfer them to the *Member* area > .

   d) Repeat these steps to add the remaining Classes with the Nets from the following list:

     i) Class: `AMP_20`   Net: `NetF3_2`

     ii) Class: `AMP_30`   Net: `NetF2_1, NetT4_1`

     iii) Class: `Voltage_120` Net: `NetF1_1, NetF1_2, NetT2_1`

   e) Close the *Object Class Explorer* with **OK**.

Figure 8. Object Class Explorer
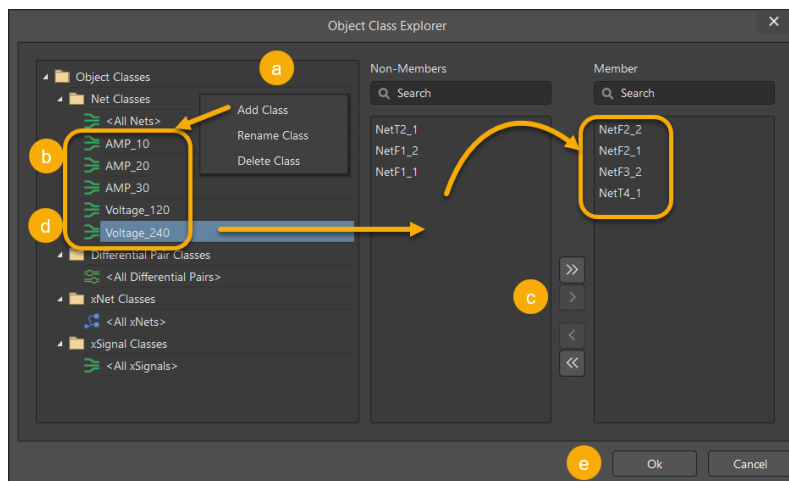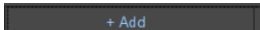
30. To add net classes to the constraint manager matrix, select **+Add** `+ Add` or **Right-click » Add** to open the *Add Net Class* dialog.

31. Select the class `Voltage_120` and `Voltage_240` that you created. Select **Add** to add the classes to the *Constraint manager.* See Figure 9*.*
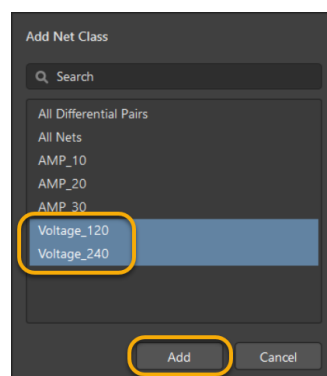
Figure 9. Add Net Classes

32. Select **File » Save** to save all modifications.

5.2 Adding Clearance Values

33. Once the classes are added, clearance values can be applied.

   a) Select the cell for `Voltage_120` x `All Net Classes` and add the value 126mil.

   b) Select the cell for `Voltage_240` x `All Net Classes` and add the value 252mil.

> Hint: Press **CTRL+Q** to change the unit from mil to mm and vice versa.

Figure 10. Adding the Clearance Value

5.3 Adding Net Width Values

34. Select the *Physical* Constraint View.

35. The classes you created in the *Clearances* view are automatically available in the *Physical* view.
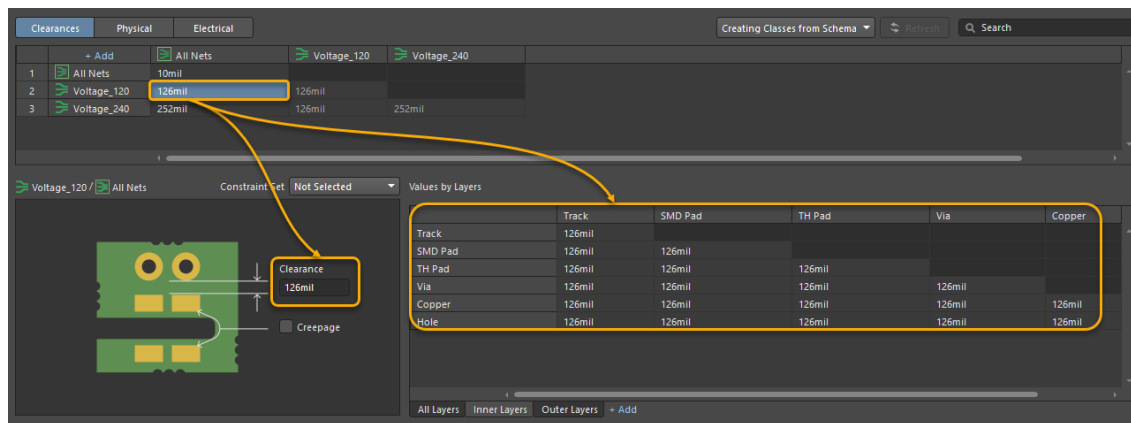
36. From the *Properties* panel, change **Units** to **mm,** alternatively use **CTRL+Q** to switch units.

37. Select the Min Width or Preferred Width cells. Then, add the additional width values at the bottom of the window, as seen in Figure 11.

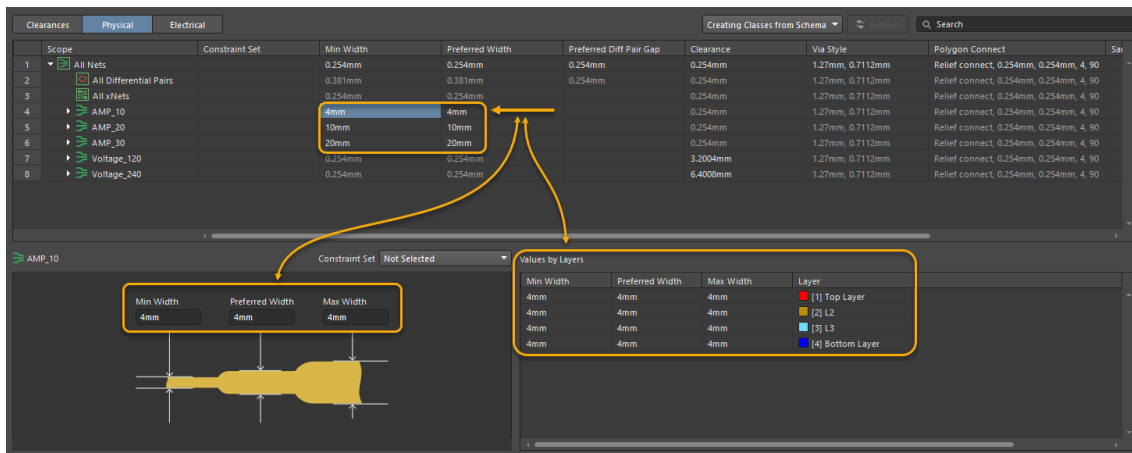   a) AMP_10: 4mm

    b) AMP_20: 10mm

    c) AMP_30: 20mm

Figure 11. Constraint Manager - Physical View adding width values

38. Save the modifications for the Constraint Manager **CTRL+S**.

5.4 Cross Probe

> Hint: If you have two monitors, open the *Constraint Manager* on the first one, and the Schematic on the second one.
> If you have only one monitor, you can create a split view by going to **Window » Tile Horizontally** or **Vertically**.
> To remove the split screen, right-click on one of the document tabs and select **Merge All**.

> Hint: To change the zoom level for the Cross Probe, go into the system preferences under **System » Navigation** and adjust the zoom slider.

39. Let's do a quick check to make sure the classes are correctly defined.

40. Select the class `AMP_10` in the constraint manager, right-click and select **Cross Probe**, Figure 12. View the applicable nets in the schematic page.

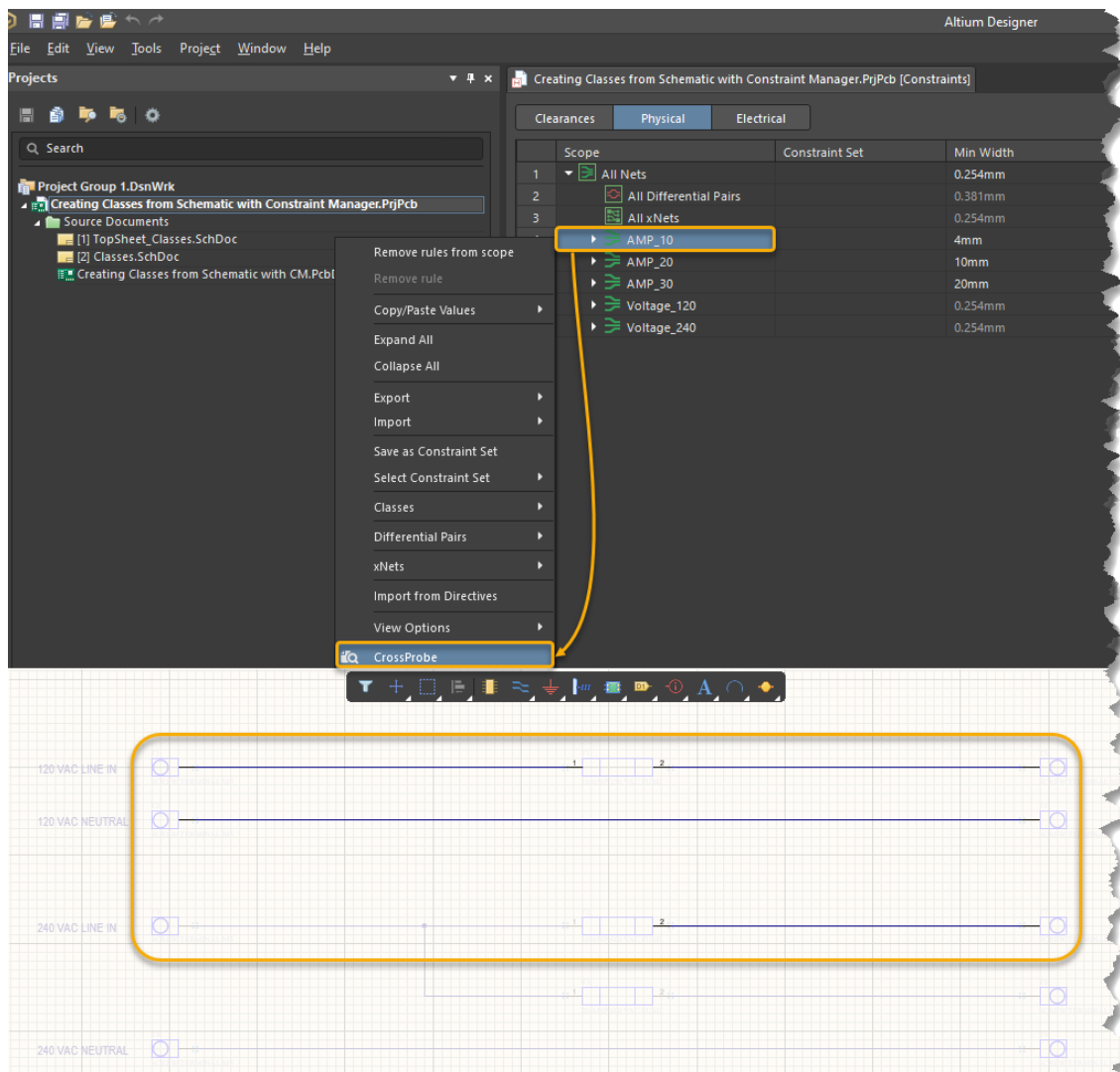41. Repeat the Cross Probe command for the other four classes.

Figure 12. Cross Probe - Constraint Manager to Schematic

42. Save and close the *Constraint Manager*.

5.5 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.

43. If the *Properties* window is not open, you can open it by double clicking on component `F1`, or by clicking the *Panels* button, then Properties.

44. 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 after an update from the schematic to the PCB is done.

45. Repeat Step 43 and Step 44 for components `F2` and `F3`, noting that these components will be added to a new component class named `240_VOLT`.

46. Now, you will add some Component Class parameters. Open the Parameter Manager using **Tools » Parameter Manager**.

47. 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 13.
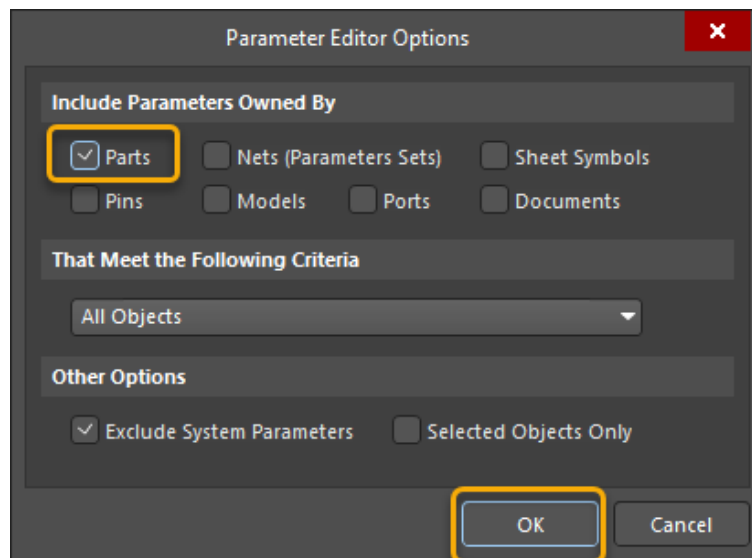

Figure 13. Parameter Editor Options

48. The *Parameter Table Editor* will display all parts used in the project. Add the ClassName to the T1 through T9 components.

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

50. Select the `T1` through `T9` ClassName column entries, right-select and select **Add**. Refer to Figure 14. This will add Plus Symbols for editing, indicated by the arrow and inset image.
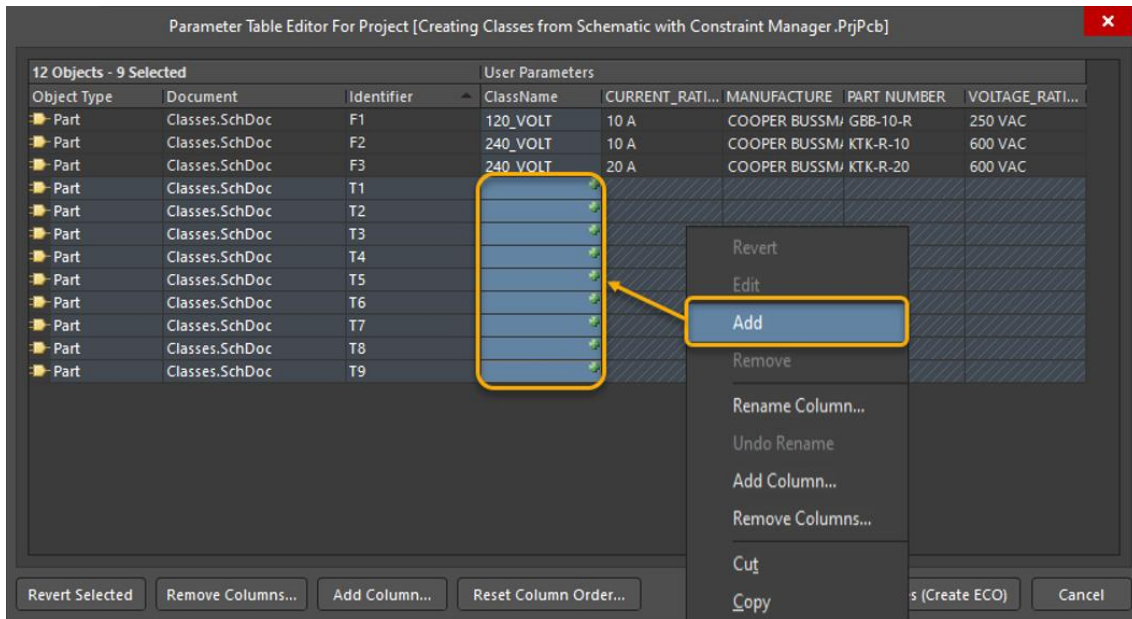

Figure 14. Parameter Table Editor for the Project

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

52. Type in the value `LEFT` and press **Enter**.

53. Repeat for `T5` to `T9` parts, but for the value, enter `RIGHT` instead.

54. Your parameter table should now look similar to what is shown below, Figure 15.
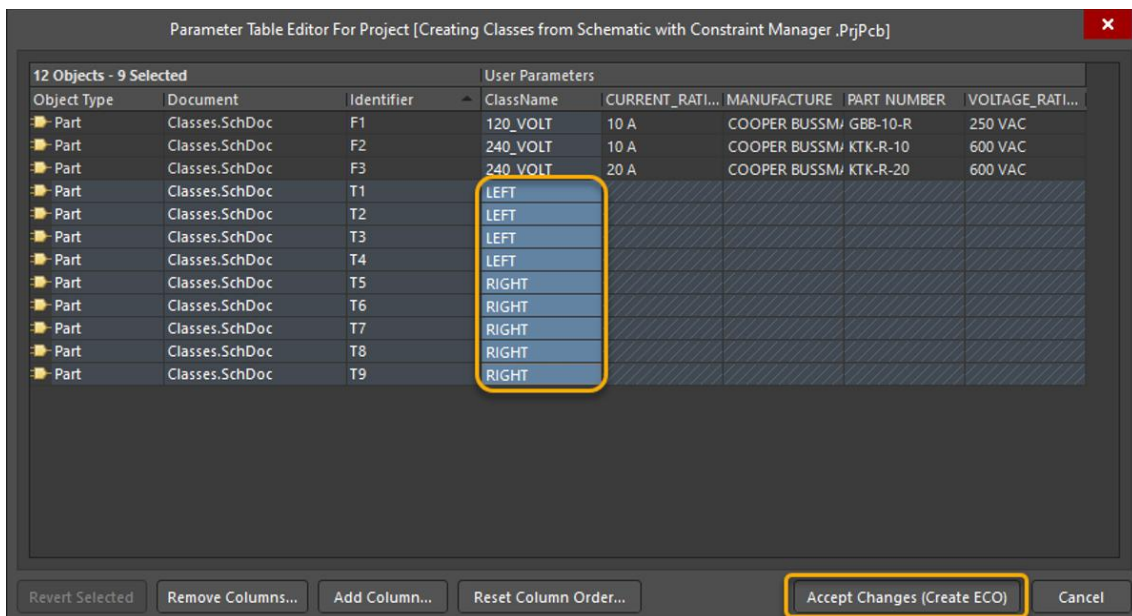

Figure 15. Populated Parameter Table

55. Select **Accept Changes (Create ECO)**, Figure 15, to generate an ECO list, Figure 16.
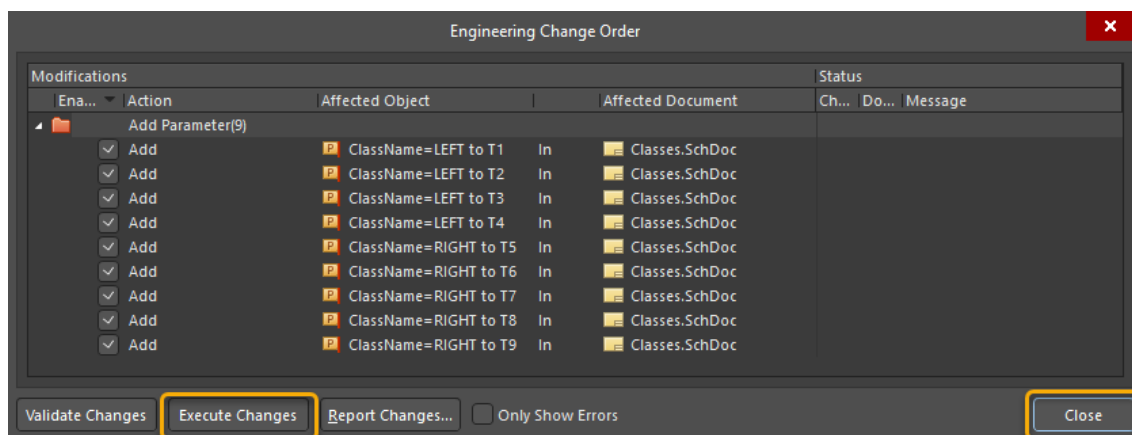

Figure 16. Engineering Change Order

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

57. Select **File » Save All** to save all modifications.

6 Updating the PCB

6.1 Automatically Adding Classes to the PCB Design

58. Select the menu option **Project » Project Options** to open the *Options for PCB Project* dialog and select the *Class Generation* tab. In order for the Component 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** option is enabled. See Figure 17.
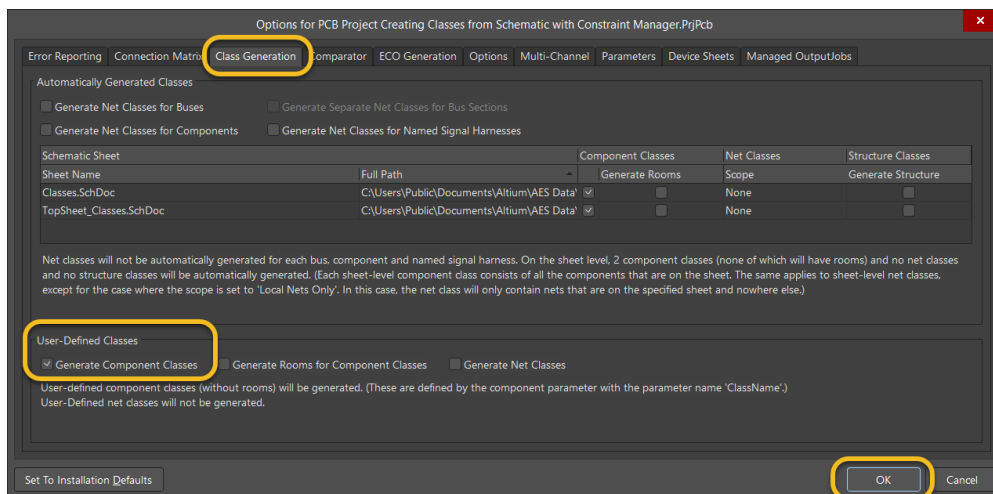

Figure 17. Project Options for User Class Configuration

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

60. From the schematic editor, select **Design » Update PCB Document Creating Classes from Schematic with CM.PcbDoc** command. Alternatively, from the PCB editor, you can use **Design » Import Changes From Creating Classes from Schematic with Constraint Manager – [Your Name].PrjPcb**.

61. When the *Engineering Change Order* dialog appears, Figure 18, 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**.
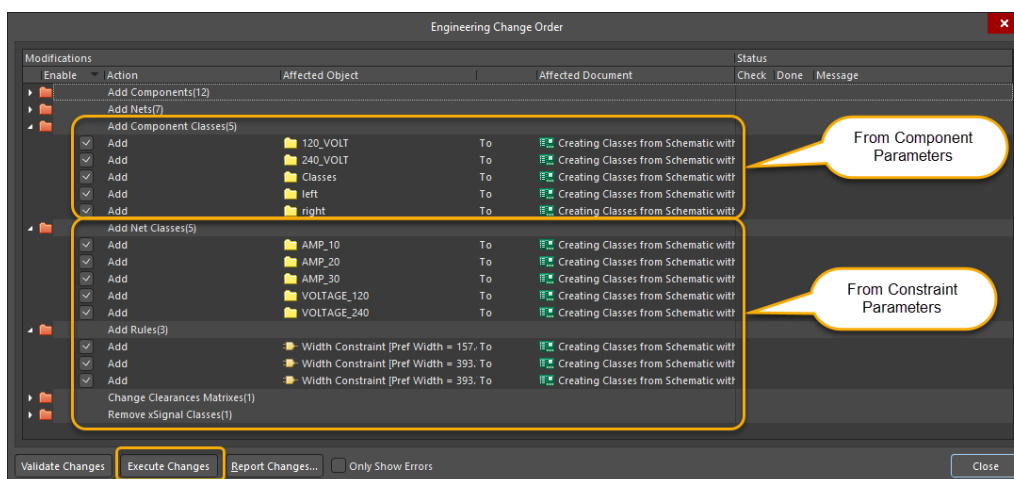

Figure 18. Engineering Change Order

6.2 Investigating the Classes in the PCB Editor

62. Now that the PCB has been updated from the schematic, open the *Object Class Explorer* dialog by accessing the command **Design » Classes**. See Figure 19.

63. 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` being added to the schematic symbols.

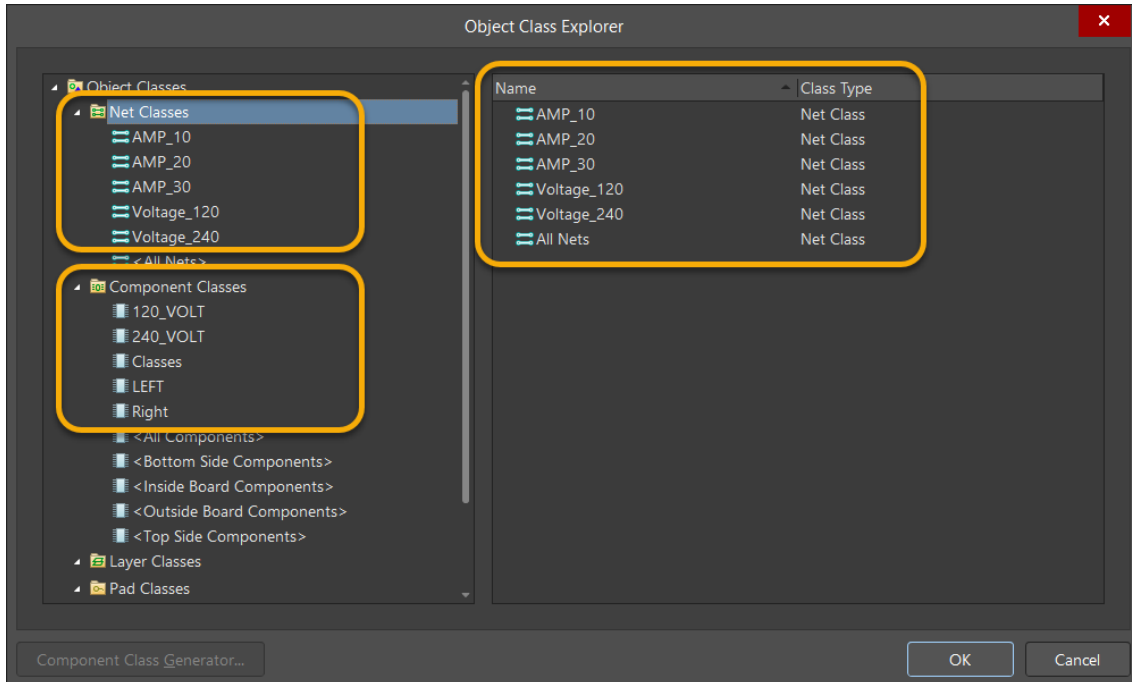64. Close the *Object Class Explorer* dialog.

Figure 19. Class Explorer

6.3 PCB Classes and Constraints

65. Make the PCB the active document.

66. Open the Constraint Manager, **Design » Constraint Manager**, Figure 20.

    a) Select the *Physical* constraint.

    b) Select the Net Class `AMP_10` and expand the category to see the four nets that have been added to this class.

    c) Select one of the *Width c*onstraint values. At the bottom, you see the general Width Configuration and the Layer Specific Configuration.
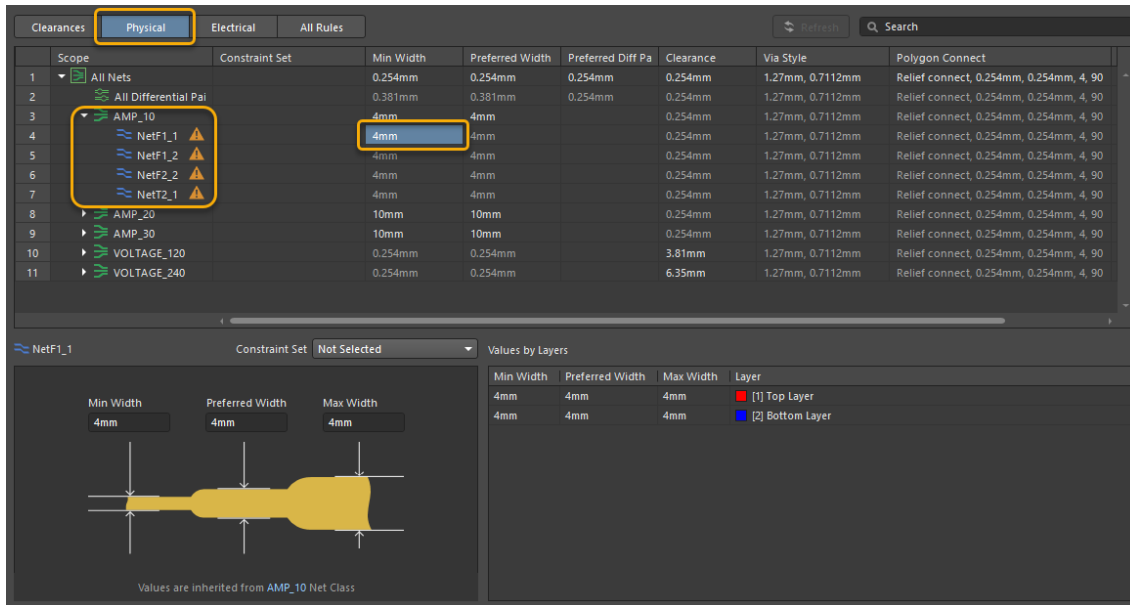

Figure 20. Constraint manager from PCB with Class and Rule definitions

67. Select the constraint types *All Rules*.

    a) The constraint types *All Rules* is only available if you start the *Constraint Manager* from the PCB.

    b) This view shows a list of all the rules present in the PCB design. From here, you can create advanced rules that feature more complex query expressions in their matching scope.

68. Next, you will add a new advanced rule for the Hole Size. See Figure 21.

 a) On the left side, scroll down to the *Manufacturing* section and select the rule category *Hole Size.* On the right side, right-click and select **Add Advanced Rule**. The new rule is added to the top of the list.

 b) Select the Name cell and rename the new rule to `MaxMinHoleSize Screw Terminal`.

 c) Select the rule. At the bottom, select the Ellipsis Button next to the Object Match window.

 d) Choose the command **Open Query Builder.** The *Building Query from Board dialog* will open.

 e) Set up the scope:

 i) Use the Condition Type pull down and select Belongs to Component Class.   Select Left from the condition value pull down: `InComponentClass('LEFT')`

 ii) Click 'Add another condition' and use the pull down to select Belongs to Component Class. Select Right from the condition value pull down: `InComponentClass('Right')`

 iii) Click on **AND**, and use the pull-down menu to change it to **OR**: `InComponentClass('LEFT') or InComponentClass('Right')`

 iv) After finished the query close the *Building Query from Board* dialog with **OK.**

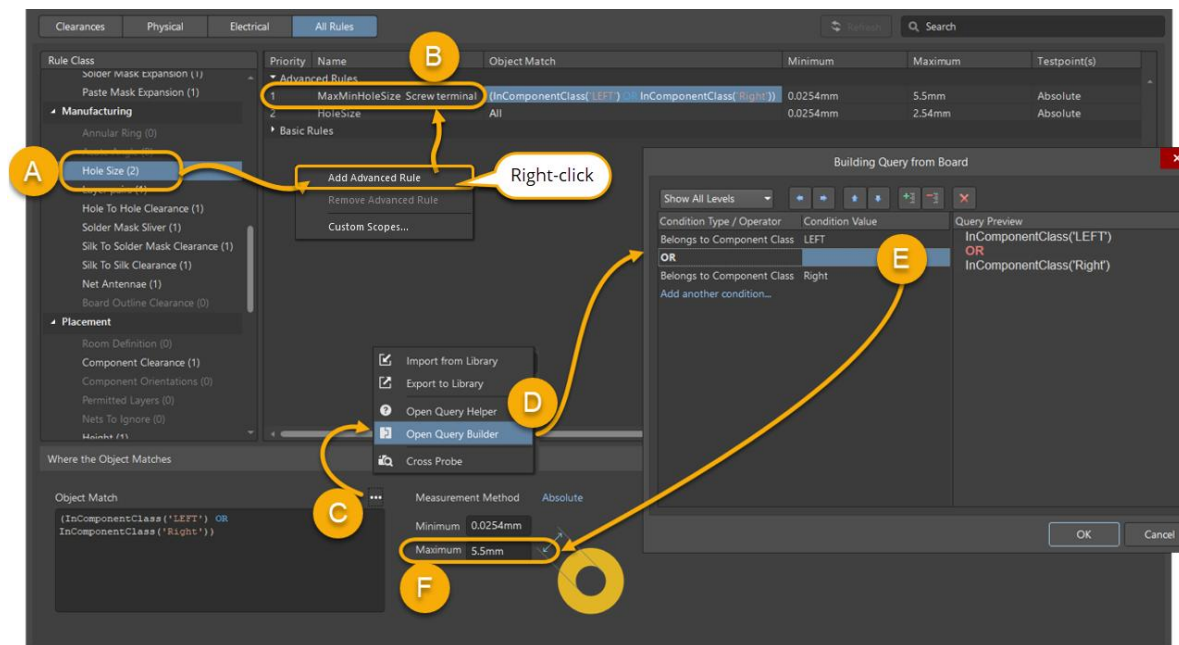 f) Back in the *Constraint Manager*, add `5.5mm` as maximum hole size.

Figure 21. Create Advanced Query Rule

69. Close the *Constraint Manager*.

70. 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 with Constraint Manager - [Add Your Name] - Finished.`

        ii) Select **OK**.

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

Altium.
TRAINING

Congratulations on completing the Module!

Creating Classes from Schematic with
Constraint Manager

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

**Altium®
TRAINING**