Altium Designer

Essentials Training with Altium 365

Module 14: Transfer to PCB

Altium
TRAINING

Altium
TRAINING

Software, documentation and related materials:

Table of Contents

ALTIUM
TRAINING

Module 14: Transfer to PCB

1 Purpose

In this exercise, you will be introduced to transferring the design from schematic to PCB. An additional part of this exercise is the check of directives (placed at the schematic), and settings; after the transfer to the PCB is executed.

Using a comparison engine, Altium Designer creates an Engineering Change Order (ECO). This is used to synchronize the schematic to the PCB, as well as update the relevant changes from the PCB back to the schematic. This bi-directional synchronization guarantees a high level of design integrity and helps to ensure the PCB and schematics remain synchronized. If any items fail during the ECO Execution, you should locate and correct the errors before completing the transfer.
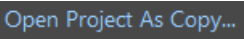
2 Shortcuts

Shortcuts used when working with Module 14: Transfer to PCB

D » U	Update PCB document
Ctrl+S	Save document

3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project `Module 14 Transfer to PCB`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button [ ].
5. Navigate to the predefined Training Project `Module 14 Transfer to PCB` `(Top\Projects\Altium Designer Essentials Training Course\...)`.
6. Select **Open Project as Copy...** `Open Project As Copy...` .
7. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 14 Transfer to PCB - [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 14 - [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 opens the project in the *Projects* panel; this can take up to 1 minute.

> Hint: For details how to Copy / Clone the predefined training project see Module 9 Making the Connection, Step 3 Preparation.

4 Transferring the Design

4.1 PCB Document

9. Since we are using a project template, the project contains a predefined PCB document.

> Hint: In case you may have later a project without a PCB:
> Right-click the project name in the *Projects* panel and select **Add New to Project » PCB**.
> Save the PCB and name it, for example, `[Project Name].PcbDoc`.

4.2 Rename the PCB Document

10. First, open all Schematic Documents.

11. Let's change the name of the PCB by removing 'Blank' from the file name.

   a) In the Project panel right click on the PCB and select **Rename...** .

   b) Change the Name to `WCTopping PCB.PcbDoc`
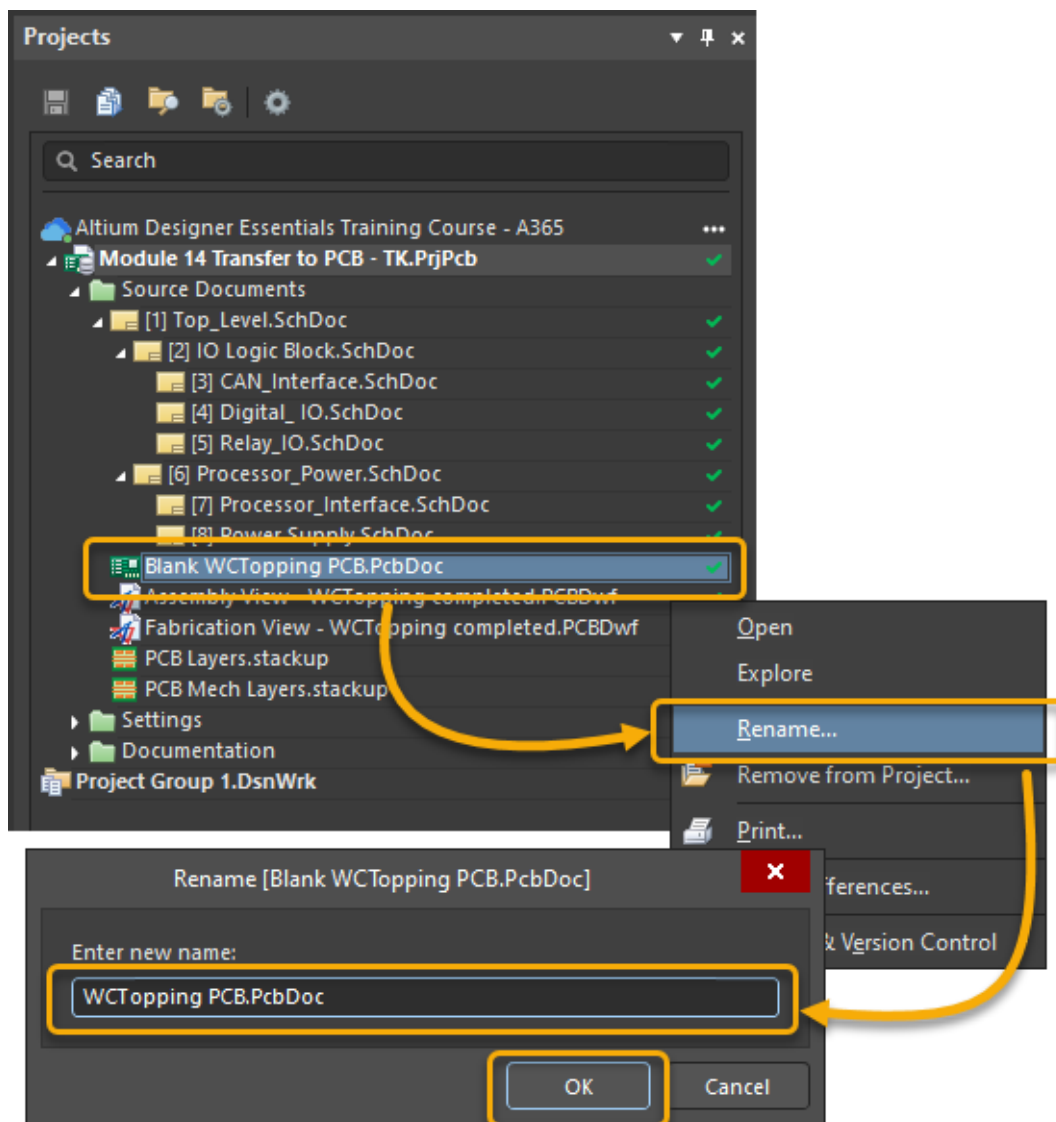
   c) Select **OK** to close the rename dialog.
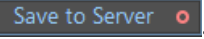
Figure 1. Rename the default PCB

Following the changes to the project, save all updates and modification as shown below.

12. Select **File » Save All** to save all modifications to the local folder.

13. Next, save the modifications to the server:

   a) From the *Project* panel, next to the Project name select **Save to Server** `Save to Server  o`.

   b) At the pop-up *Save* dialog add a comment, for example, `Renamed PCB to WCTopping PCB.PcbDoc,` as seen in Figure 2.

Figure 2. Save to Server

   c) Select **OK** to initiate save to server and close the dialog.

4.3 The Initial Transfer

14. From within a schematic, select **Design » Update PCB Document [PCB Name]**
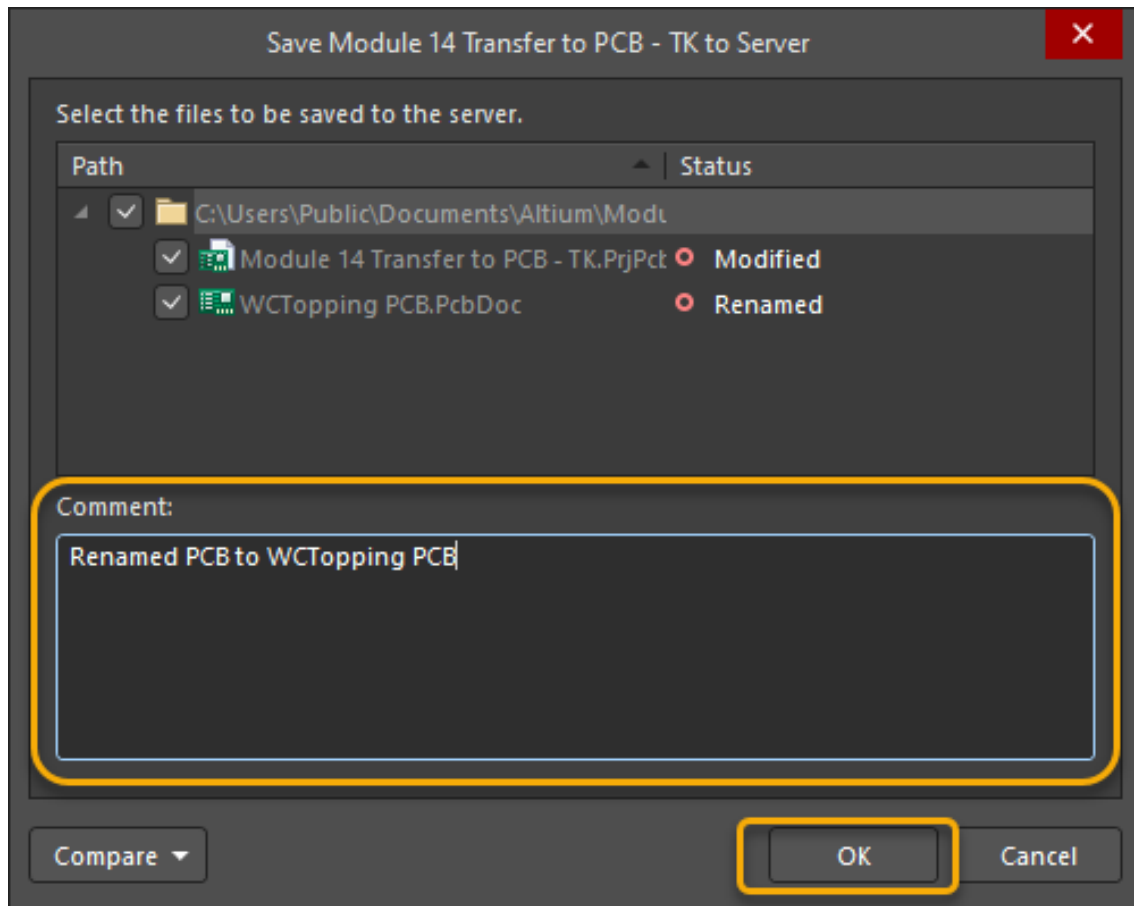
15. At the new dialog *Engineering Change Order* (ECO) select the **Execute Changes** button in the lower left of the dialog, as shown in Figure 3 below.
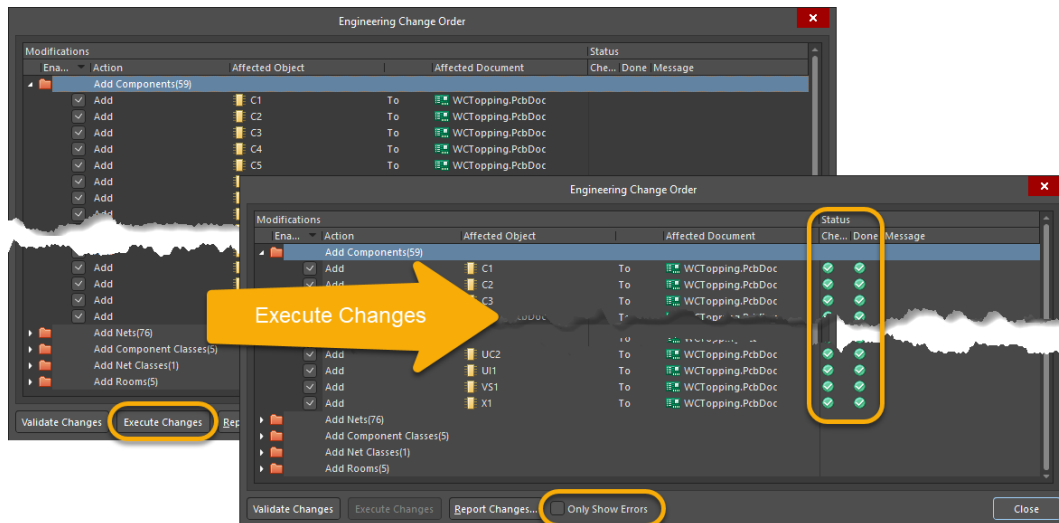
Figure 3. Engineering Change Order dialog

16. After the ECO is executed, enable the **Only Show Errors** checkbox to check for any errors, the training example should be without errors.

17. When ready, close the *Engineering Change Order* dialog with **Close**

18. After the ECO is executed the components and unrouted connections can be seen on the right side of the PCB, Figure 4.

Figure 4. PCB after initial Assigning Differential Pair

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

7

ALTIUM
TRAINING

4.4 General

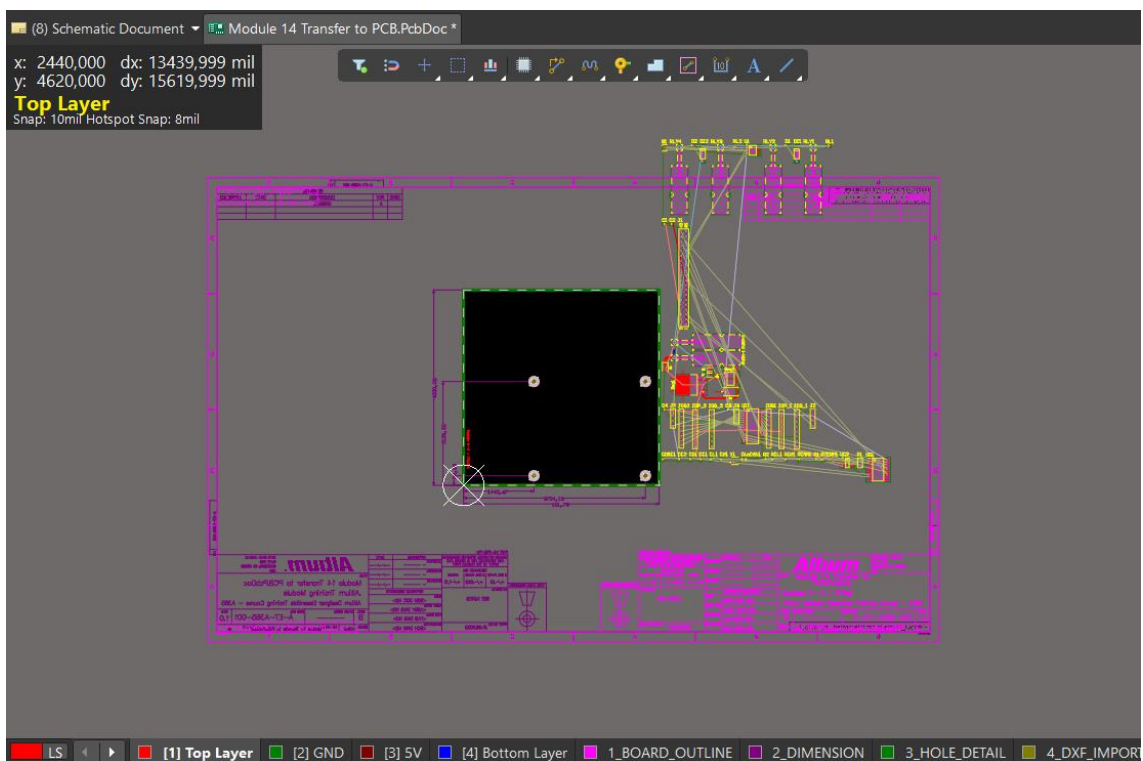20. To control the visibility of the used layers, from the lower left select **Layer Set (LS)** ![LS icon], from the pop-up menu select **Default**. The **Default** Layer-Set deactivates the layers: Title Block information and Dimensions, see Figure 5.
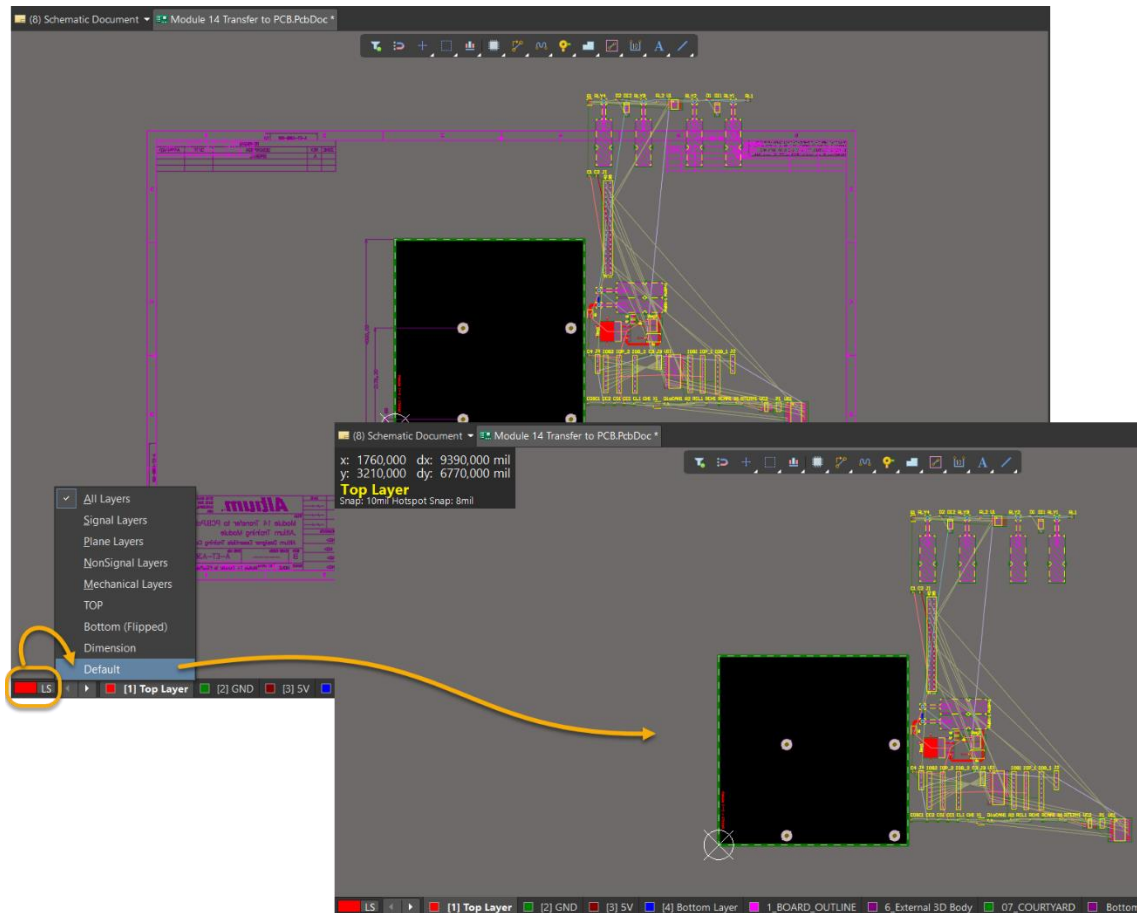
Figure 5. Initial Transfer and using an example Layer Set

4.5 Net Colors

21. Depending on your user settings, some of the unrouted connections will have different colours, the net colors configuration will be same as the colours defined in the schematics during the Module 12 Schematic Updating.

22. Again, depending on your user preferences, component pads may also show net colours, this display can be activated or deactivated using the F5 key.

4.6 Object Class Explorer

23. Select **Design » Classes** to open the *Class Explorer* dialog.

24. At the left side you see the different Class types that Altium Designer supports in the PCB.

25. The class type *Net Class* includes the Class `Power`, defined in the Schematic.
    On the right you see the members from the class `Power`.
    This Class was added during Module 12 Schematic Updating.

26. The class type *Component Class* includes predefined Classes <[Name]> as well as classes created during the ECO, based on the Schematic documents in the Project , for example, CAN_Interface, Digital _IO, etc.

> Hint: The Net Class and Component Class generation is controlled by Project setting, **Project » Project Options**, Tab *Classes Generation.*
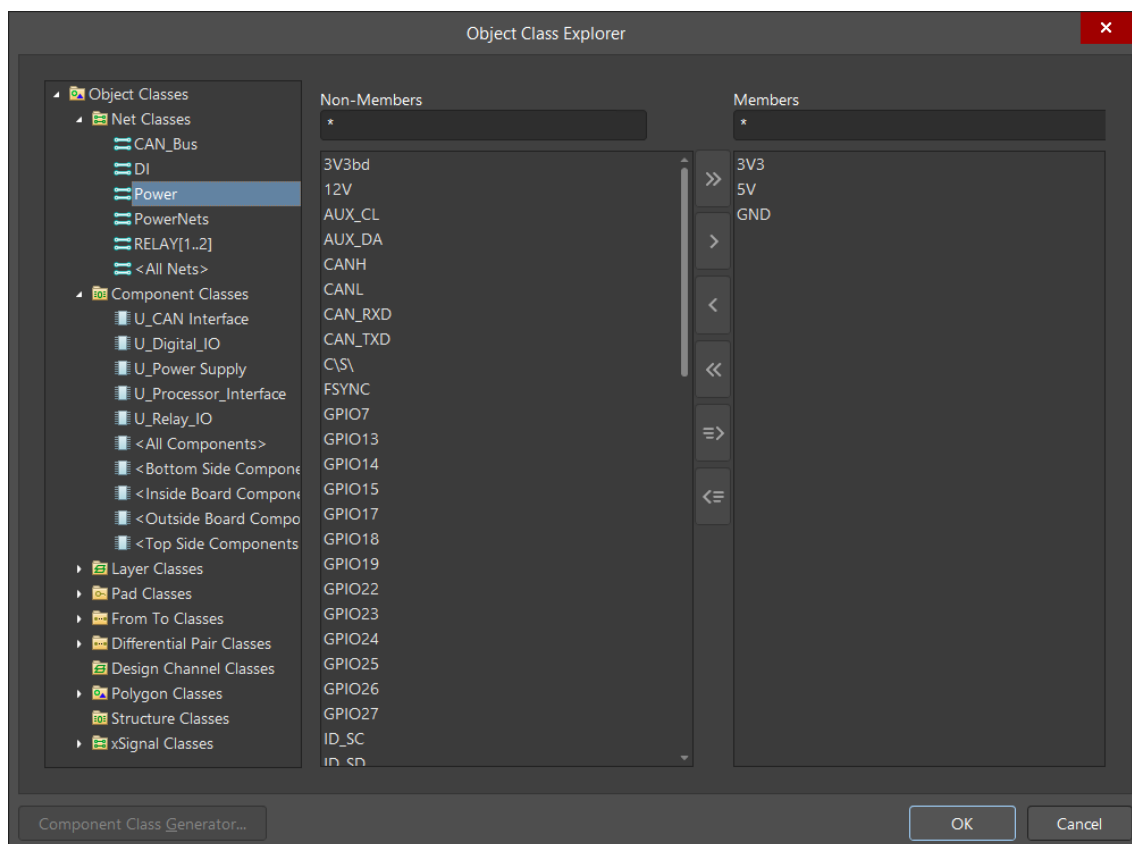

Figure 6. The Object Class Explorer with the Net Class Power

27. Select **Cancel to c**lose the *Class Explorer* dialog**.**

4.7 PCB Rules and Constraints Editor

28. Select **Design » Rules** to open the *PCB Rules and Constraints Editor* dialog.

29. On the left below the Search option are all the supported different design rule categories.

30. The rule category *Routing – Width* includes the rule `Schematic Width Constraint_1`, indicating this rule was defined in the Schematic. On the right side you can see the rule configuration with the values added at the schematic. The Class was added during Module 12: Schematic Updating.

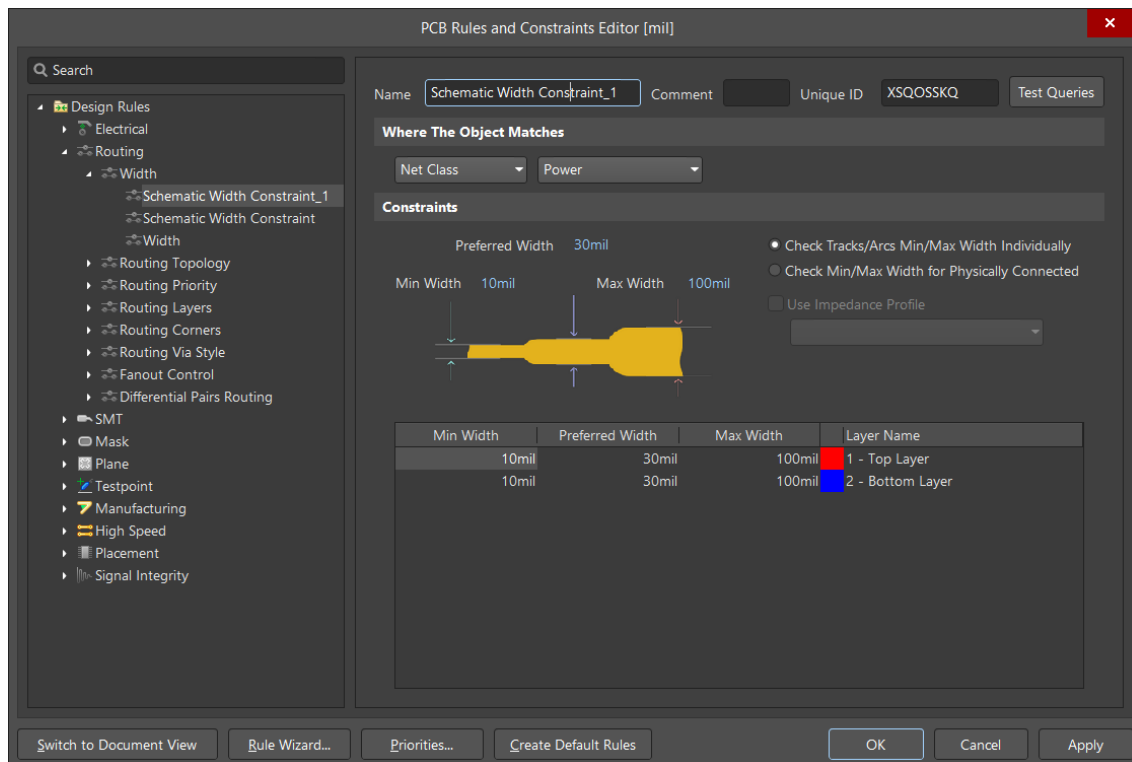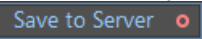31. Select **Cancel to c**lose the *PCB Rules and Constraints Editor* dialog.

Figure 7. The PCB Rules and Constraints Editor with a Width rule for the class Power

32. Save latest project modifications to the server:

   a) From the *Project* panel, next to the Project name select **Save to Server**
   Save to Server.

   b) At the pop-up save dialog as seen in Figure 2.

33. When ready, close the project and any open documents.

Congratulations on completing the Module!

Module 14: Transfer to PCB

from

Altium Designer Essentials Training
with Altium 365

Thank you for choosing **Altium Designer**

Altium
TRAINING