**ALTIUM
DESIGNER**

**ALTIUM
365**

Altium Designer

Advanced Training with Altium 365

Differential Pair Routing with Impedance Profile and Constraint Manager

**Altium®
TRAINING**

Software, documentation and related materials:

Table of Contents

ALTIUM
TRAINING

Differential Pair Routing with Impedance Profile and Constraint Manager

1 Purpose

A differential signaling system is one where the signal is transmitted through a pair of tightly coupled carriers, with one carrying the signal and the other carrying an inverted (equal but opposite) image of the signal.

Differential signaling is inherently immune to common-mode electrical noise, which the most common interference artifact present in electronic products. Another major advantage of differential signaling is that it minimizes the electromagnetic interference (EMI) generated from the signal pair.

2 Shortcuts

Shortcuts used when working with Differential Pair Routing with Impedance Profile and Constraint Manager

F1	Help
D » G	Constraint Manager (SCH)
D » R	Constraint Manager (PCB)
CTRL+Q	Change Unit (Constraint Manager)
U » I	Differential Pair Routing
Spacebar	Routing Angle
Shift+Spacebar	Routing Corner Style
CTRL+S	Save Document

Hint: We recommend to do module *Creating Classes from Schematic with Constraint Manager* before continuing with this module. Module *Creating Classes from Schematic with Constraint Manager* includes a general Overview for the Constraint Manager.

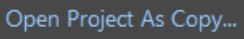Caution: The *Constraint Manager* is only available if you are using an Altium Designer with a Professional Subscription. Note that the Constraint Manager isn't supported with the Altium Designer Standard Subscription. For details, please see our Online Documentation.

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

3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `Differential Pair Routing with Impedance Profile and Constraint Manager`.

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 `Differential Pair Routing with Impedance Profile and 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: `Differential Pair Routing with Impedance Profile and 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*.

> Note: Due to Length Limit limitations, the names for files in the workspace may be different to the names used in this document. We use CM for Constraint Manager and IP for Impedance Profile.

Altium TRAINING

4 Defining the Differential Pair

In this part, you will predominantly use the Active Bar menu, although the same features are available from the Place dropdown menu.

First, you will do the first step to create the differential pair signals by placing Netlabels.

9. Open the `Differential Pair Routing with IP and CM.PcbDoc` document.

10. Open the `Differential Pair Routing with IP and CM Setup.SchDoc` document.

11. If not already the active document, change focus to the schematic document.

12. In the *Active Bar*, right-click on the **Place Wire** button and select **Net Label**, as shown in Figure 1.

13. Press the **Tab** key to access the *Properties* panel while the net label is attached to the cursor, and change the name to `D1_N`.

14. Press **Escape** or select the Pause ⏸ to place the net label on connection between `U1-Pin1` and `L1-Pin2`.

15. Repeat Steps 12 - 14 to place net label `D1_P`. Final placement of net labels should look like Figure 2.
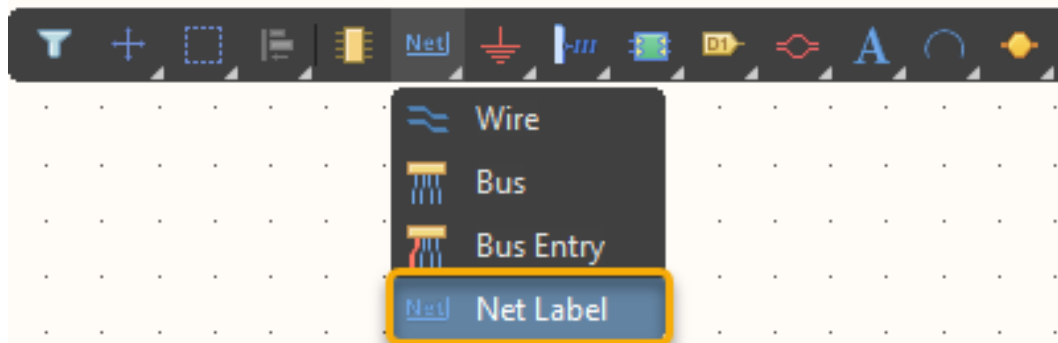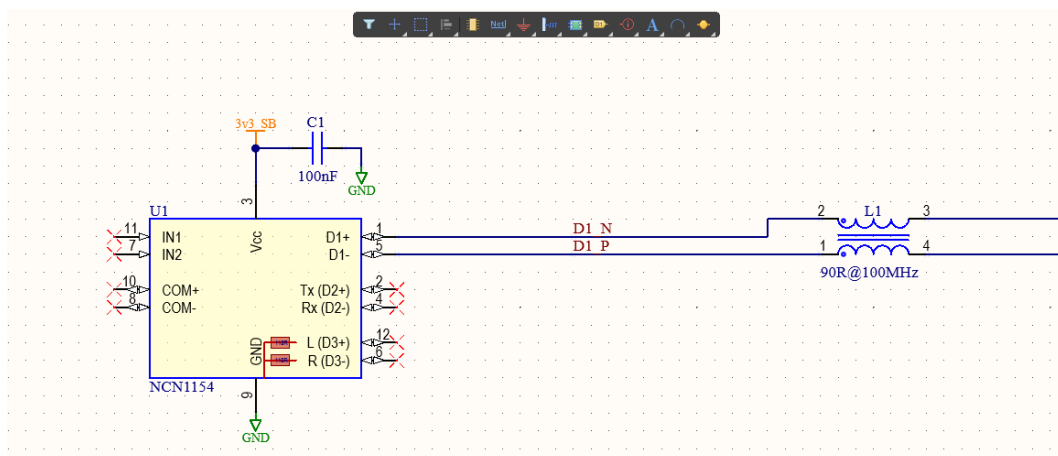

Figure 1. Place Net Label

Figure 2. Differential Signals

16. Notice that other net labels have already been placed to speed up the process.

> Hint: The default suffix for differential pair signals is _P and _N. You can also add custom suffix characters in the *Options* tab of the *Differential Pairs* section within *Project Options*.

Next, you will use the Constraint Manager to create the Differential Pairs and a Differential Pair Class.

17. Open the Constraint Manager, **Design » Constraint Manager...** .

18. Select the *Physical* view | Physical | .

> Hint: For details about Constraint Manager GUI, the difference between Constraint Manager opened from SCH (| Clearances | , | Physical | , | Electrical |) and Constraint Manager opened from PCB (| Clearances | , | Physical | , | Electrical | , | All Rules |), see the Module `Creating Classes from Schematic with Constraint Manager`.

19. Right-click anywhere in the grid area and select **Differential Pairs » Create Differential Pairs From Nets** from the context menu, Figure 3.
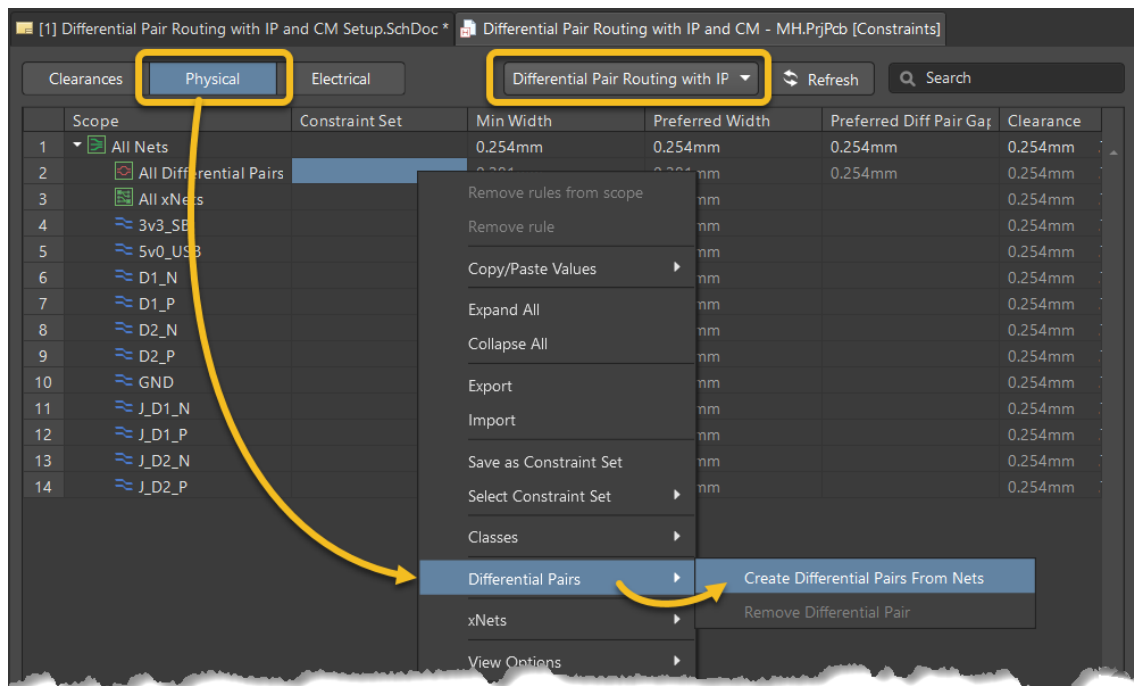

Figure 3. Constraint Editor - Physical View, ready to create Differential Pairs

20. This opens the *Create Differential Pairs From Nets dialog,* were you see the differential pair creation options, Figure 4. Review the default configuration:

   a) Net Name Suffix: `_P`, `_N`

   b) Diff Pair Prefix: `D_`

   c) Create the Differential Pairs by selecting **Execute**.
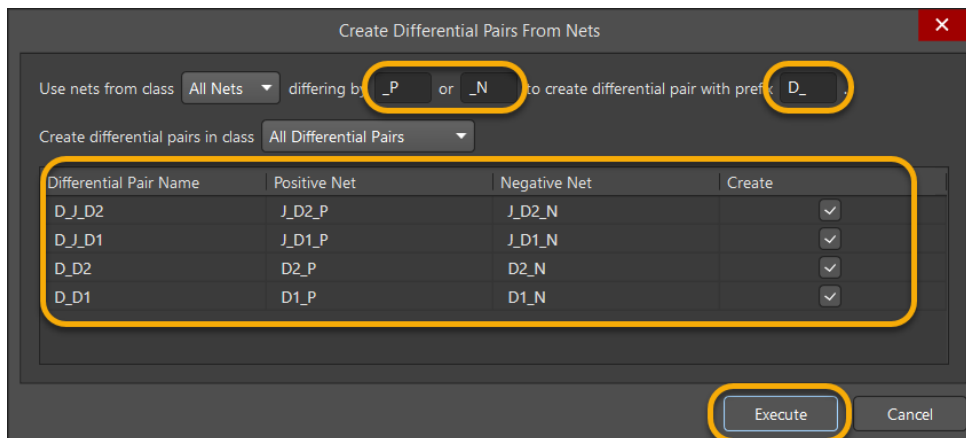

Figure 4. Create Differential Pairs from Net dialog

21. Back in the *Constraint Manager*, the new Differential Pairs are listed in the *All Differential Pairs* category, as seen in Figure 5.

Figure 5. Physical View with Differential Pair

22. Next, you will create a Differential Pair class.

23. Select the four Differential Pairs and the individual signals, Figure 6.

24. Right-click anywhere in the grid area and select **Classes » Add Class » Diff Pairs Class**, Figure 6. This opens the *Add Class* dialog.
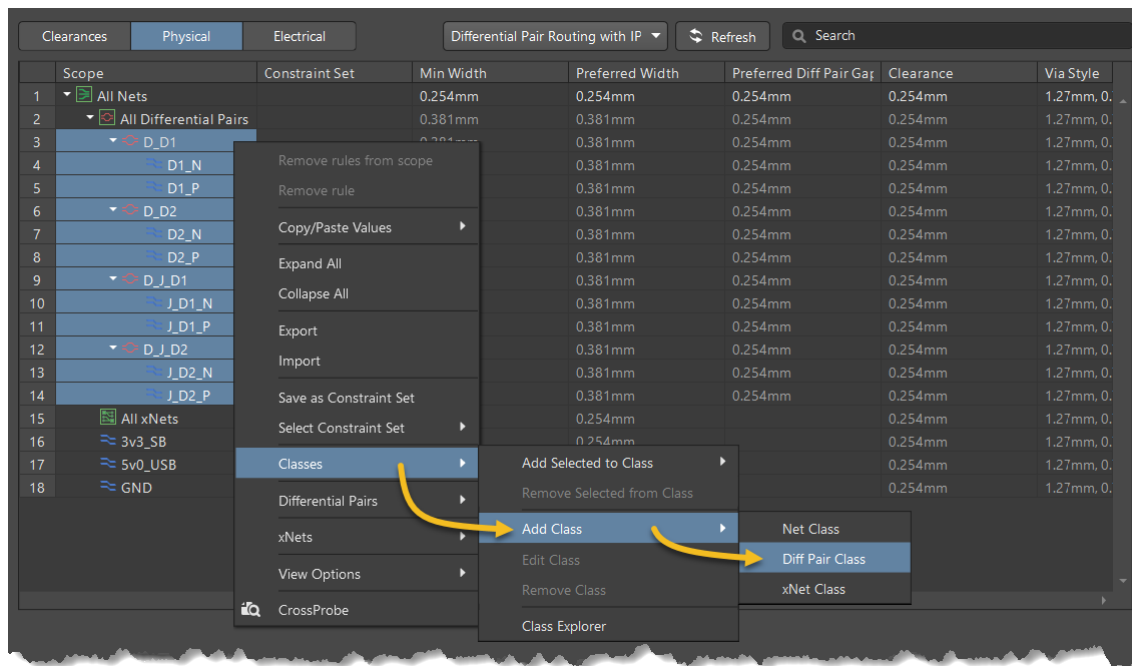

Figure 6. Constraint Manager, Creation of a Differential Pair Class

25. In the *Add Class* dialog, follow the steps below, Figure 7:
    a) Change the name `90R_DIFF`.
    b) Select the four Diff Pairs in the Non-Members area.
    c) Transfer the signals from the Non-Members area to the Members area.
    d) Select **OK** to close the dialog.
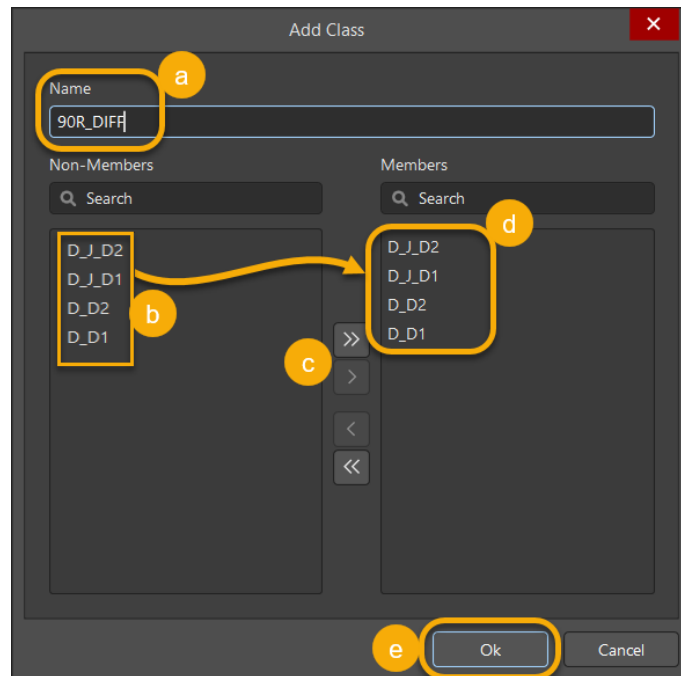
Figure 7. Add the members to the new Diff Pair class 90R_DIFF

26. Back in the *Constraint Manager*, you see the updated *Physical* view with the `90R_DIFF` Differential Pair class listed, Figure 8.
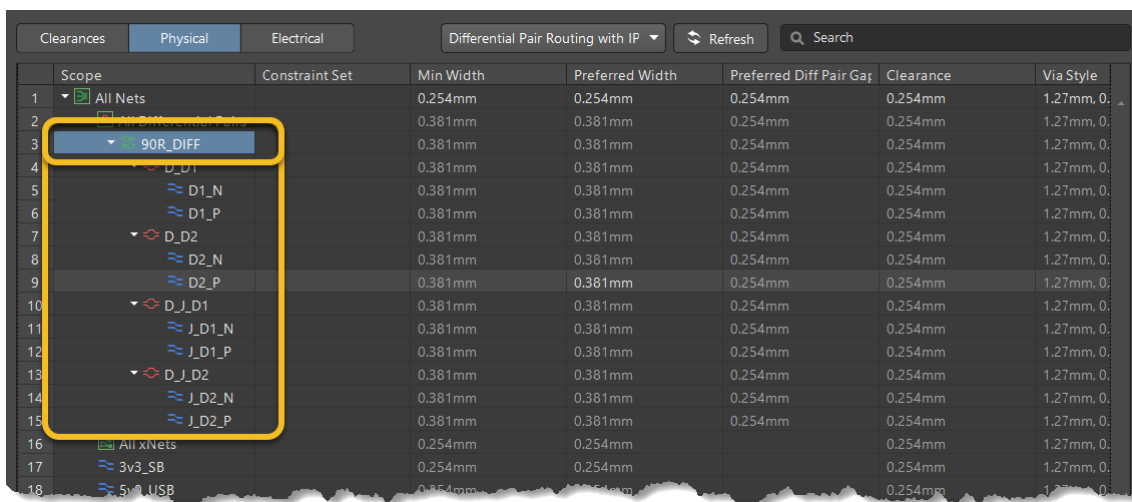
Figure 8. Physical View with 90R_DIFF class

27. Save your modifications, **File » Save** and close the *Constraint Manager*.

> Hint: At this stage, we recommend running the electrical rules check by validating the project.

28. Run validation either by using the right mouse button over the project name in the *Project* panel or the Project dropdown menu.

9

29. Open the *Messages* panel to check for any errors or warnings.

> Hint: The messages panel only pops open if there are errors, and doesn't pop open if there are just warnings. However, we recommend opening the messages panel, even if it doesn't open automatically.

30. Ensure the compile is successful and there are no errors or warnings, as shown in Figure 9.

Figure 9. Project Validation

31. Now transfer the design to the PCB, by either updating the PCB from the Schematic using the **Design » Update PCB…** menu or by opening the PCB and running the Import from the PCB using the **Design » Import Changes from …** menu.

> Hint: It's always a good idea to look at the ECO and understand what information is being transferred.

32. Select **Execute Changes** to transfer the schematic updates to the PCB document, Figure 10.
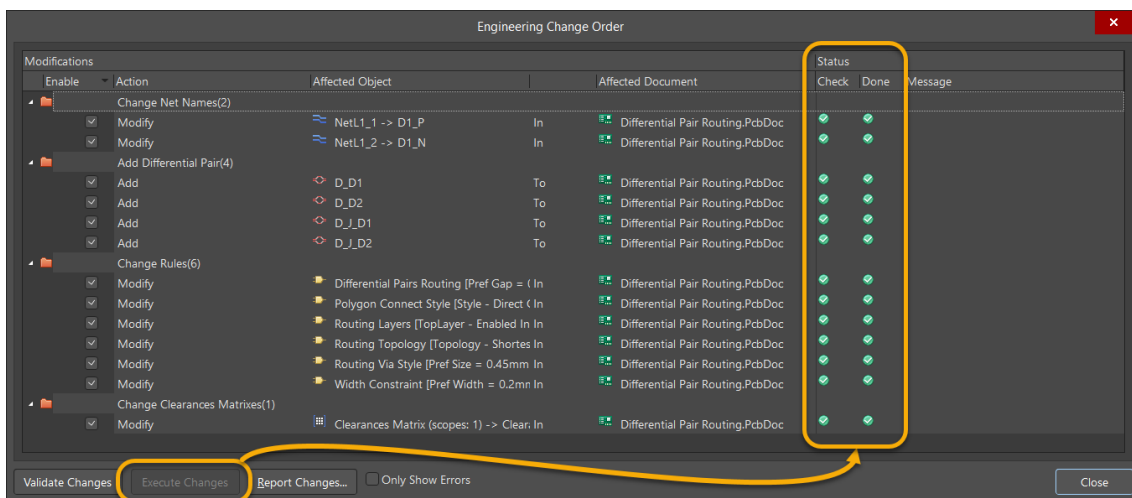
Figure 10. Engineering Change Order

33. Close the *ECO* dialog.

Differential Pair Routing with Impedance Profile
and Constraint Manager

34. Change the focus to the PCB document.

35. Using the *PCB* panel, first let's check what information has been transferred to the PCB. Select the **Differential Pairs Editor** from in the *PCB* panel, and notice four differential pairs in the PCB, Figure 11 and Figure 12.

> Hint: Ignore Online DRC errors for now, the rules for the Differential Pairs aren't configured.

Figure 11. Differential Pairs Editor

Figure 12. Differential Pair – 90 Ohm

5 Defining the Impedance Profile

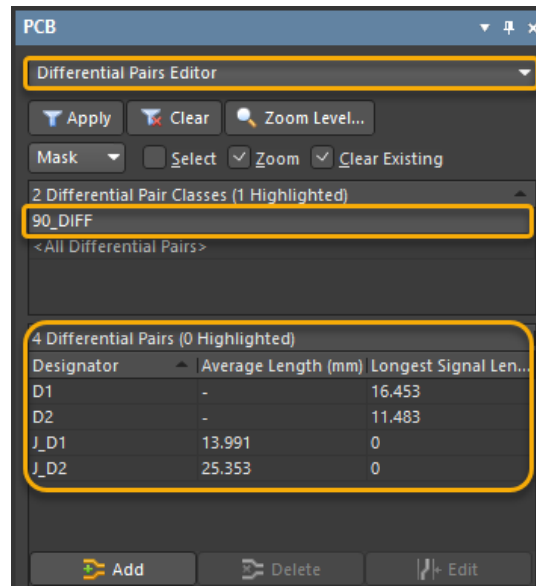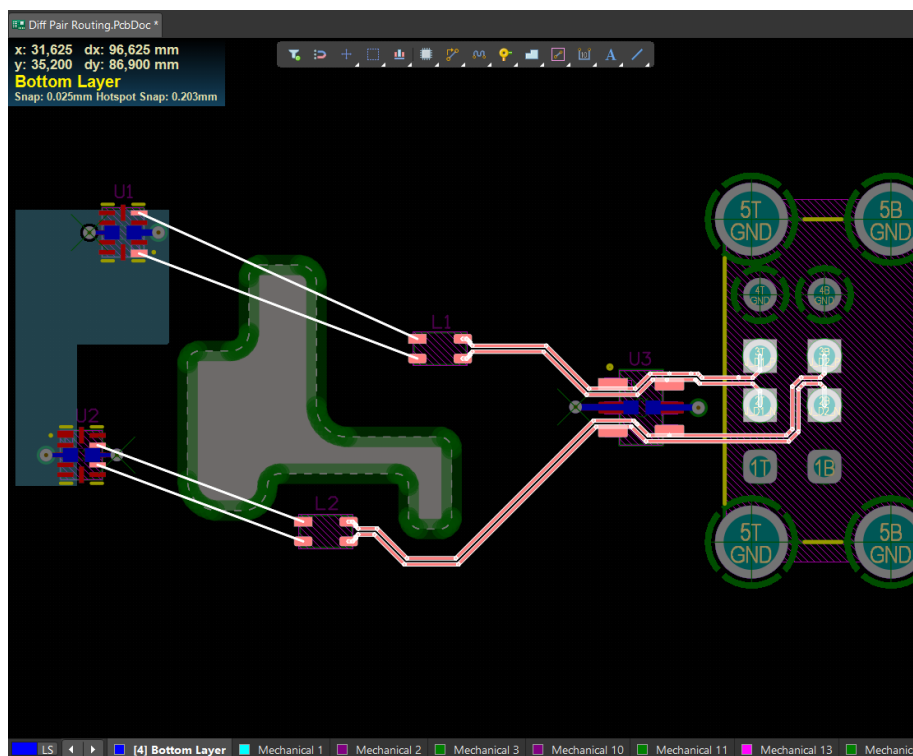Before routing the differential pair signals, we need to create the design rule for track width and gap between the two traces within the differential pair. The track width and gap of differential pairs are determined by certain key factors, some of these are the layer stack, and the required target impedance. One of the ways of obtaining the track width and gap information is by consulting your board manufacturer and providing details of your requirements. Another method is to use Altium Layer Stack Manager to calculate the required Impedance Profile.

36. Open the *Layer Stack Manager* (LSM) **Design » Layer Stack Manager…** and select the *Impedance* tab at the bottom left, Figure 13.

37. Select the **+ Add…** button at the upper left to create a new Impedance profile, Figure 13.
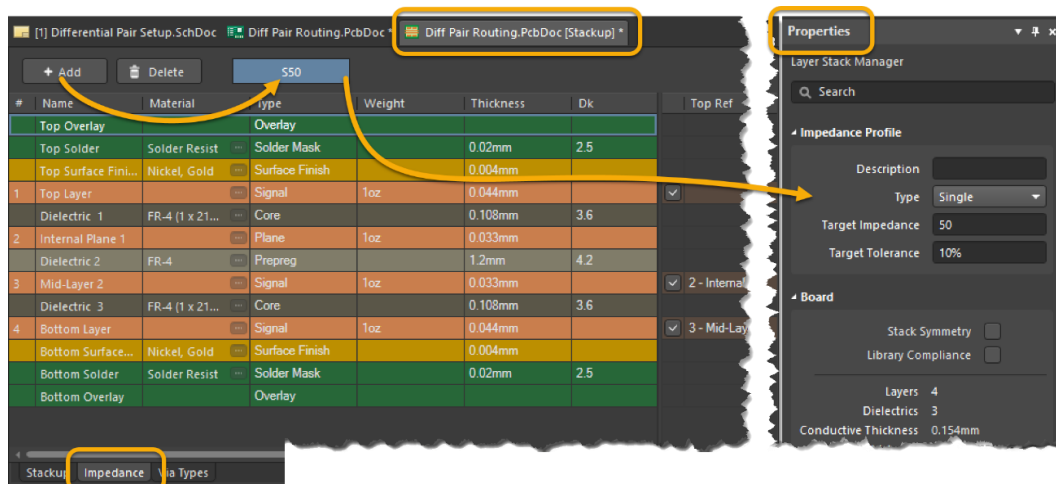

Figure 13. New Impedance profile with default 50 Ohm

38. Dock the LSM *Properties* panel on the left, if not already docked, and enter the following information, Figure 14:

a) *Description*: `USB Differential Signals`

b) *Type*: **Differential**

c) *Target Impedance*: `90`

d) *Target Tolerance*: `10%`
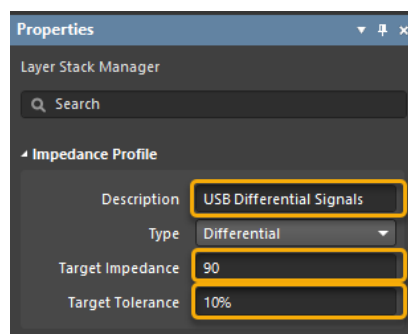
Figure 14. Target Impedance

Next, you need to calculate the track width and gap for the differential pairs. As there are already some routed differential pairs on the PCB, you need to use the same values for gap and trace width for the remaining differential pairs. When starting with a new PCB, the Impedance Profile will already start with default values for track width and gap. These values may change as you modify the layer stack and required target impedance.

12

Altium
TRAINING

39. Select the individual routing layers from the layer stack table (one layer at a time), to see the track width and gap calculations in the *Properties* panel, as shown in Figure 15 and Figure 16:

   a) Select the **Top Layer**.

   b) Add `0.102mm` as *Trace Gap (G)* in the *Properties* panel and press **Enter** to save the changes.

   c) Make sure the **Calculate** button fx is enabled next to *Width (W2)* to update the width value automatically.

   d) The trace width is updated (0.142…).

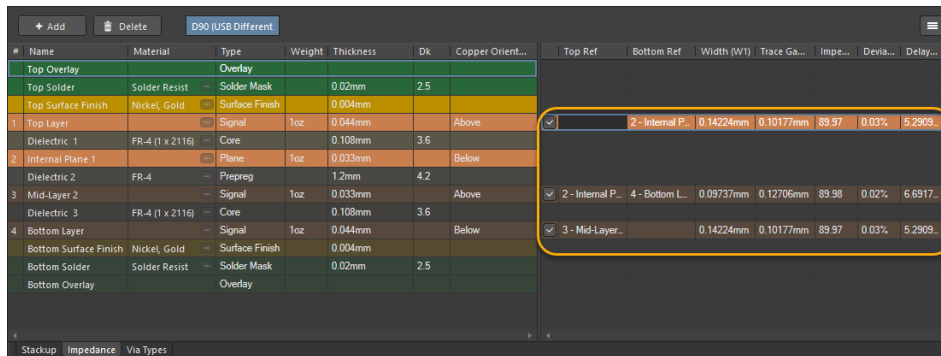   e) Repeat the steps a) to d) for the Bottom and Inner Layers.
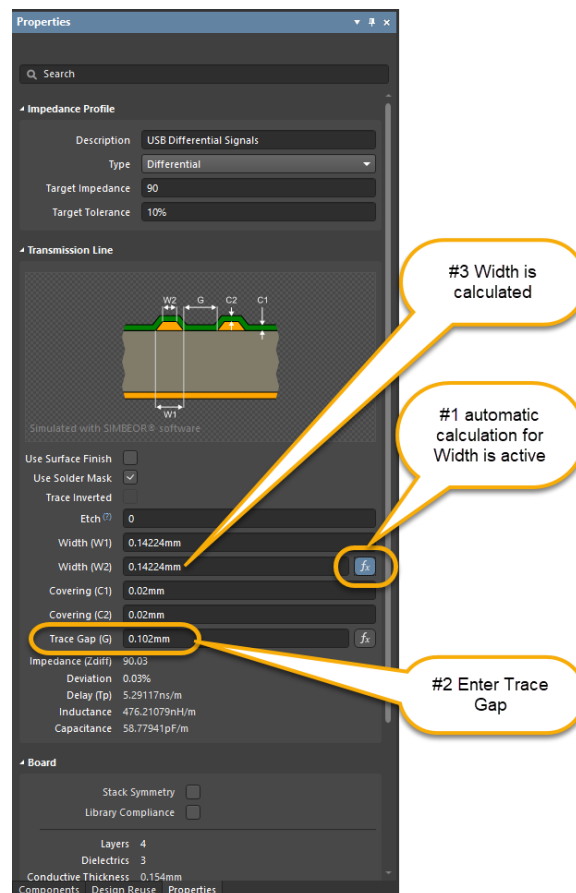

Figure 15. Routing Layers

Figure 16. Trace Width and Trace Gap Calculations

13

6 Interactive Differential Pair Routing

You will use the new impedance profile for creating the design rules required for routing the differential pairs.

6.1 Design Rules

40. Save the layer stack in order to update the PCB.

41. Change the focus back to the PCB document.

42. Save the PCB Document.

43. Using the **Design » Constraints Manager...** menu, open the *Constraints Manager* (PCB).

> Check that you see the Constraint Manager from the PCB with the four Views (Clearances , Physical , Electrical , All Rules).

44. Select the *Physical* view. The Differential Pairs and the Differential Pair Class defined in the schematic are now also available in the PCB, as seen in Figure 17.
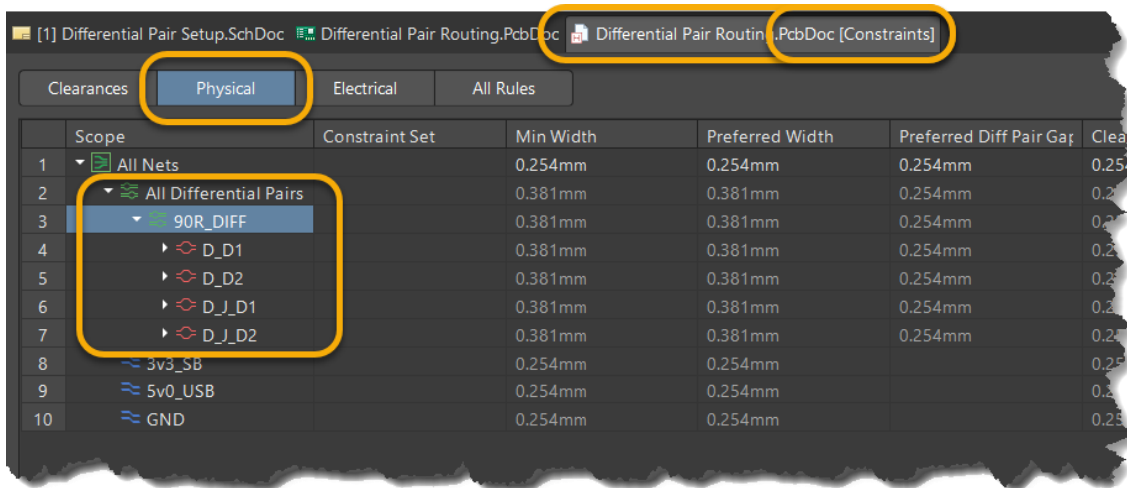

Figure 17. Constraint Manager opened from PCB with Physical view

6.2 Routing Rule - Differential Pairs Routing

45. Select the *All Rules* [All Rules] view.

46. On the left side, browse to the Routing category and select the Differential Pairs Routing rule.

47. Next, configure a new routing rule for the 90R_DIFF Class, Figure 18:

   a) Right-click anywhere in the grid area and select the **Add Advanced Rule** command from the context menu.

   b) Rename the new Rule to `DifferentialPairsRouting_90R_DIFF`.

   c) At the bottom, select the Impedance Profile `D90 (USB Differential Signals)` from the drop-down list.

   d) Select the **Ellipsis** button and choose **Open Query Builder** (Figure 19).

      i) Select as Condition Type / Operator *Belongs to Differential Pair Class*.

      ii) Select as Value *90R_DIFF*.

      iii) Close the *Building Query from Board* dialog with **OK**.

   e) Back to *the Constraint Manager,* review the values in the design rule section, these have been pulled in from the Impedance profile.

      Check the Min-, Preferred-, Max Gap value (`0.102mm`).

   f) Change the Max Uncoupled Length to `15mm`.

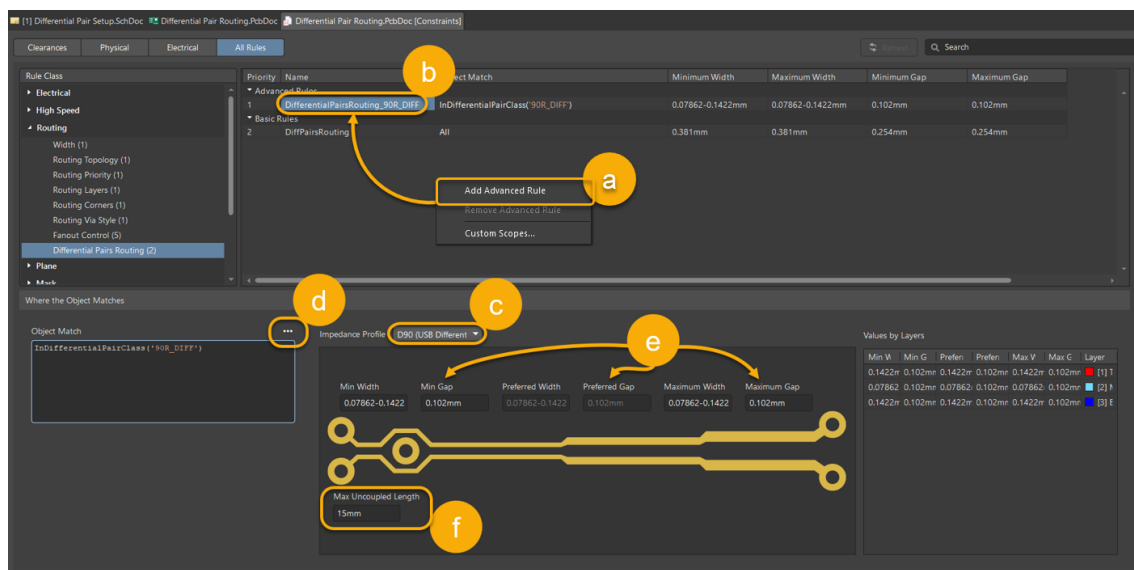48. Close the *Constraint Manager* and save the modifications with **Save All** 🖫.

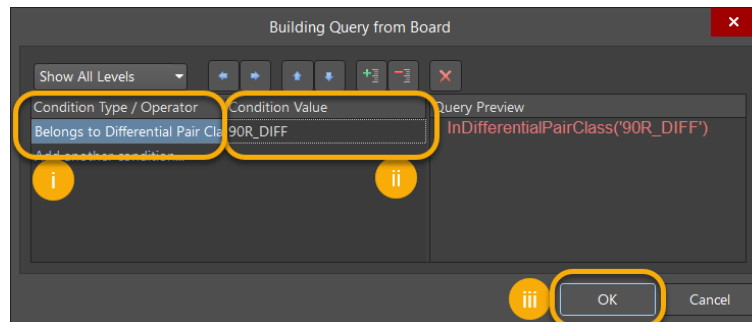*Figure 18. Adding a Diff Pair Width Rule*

Figure 19. Building Query for the 90R_DIFF class

As you will create more rules with the query `InDifferentialPairClass('90R_DIFF')`, save that query now:

a) Select the **Ellipsis** button and choose **Export to Library** (Figure 20) or hover over the rule `DifferentialPairsRouting_90R_DIFF`, right click and select **Export to Library.**

b) The *Scopes Library* dialog will open and a *Custom Scope_1* will be available.

c) Select **Update** to update the system and to close the dialog.

d) By creating the *Custom Scope_1*, the information in the Advanced rule, Object matches cell, is updated to *Custom Scope_1*.

e) Open the *Properties* panel. In the pane *Scopes Library,* the *Custom Scope_1* you've just created is now listed.
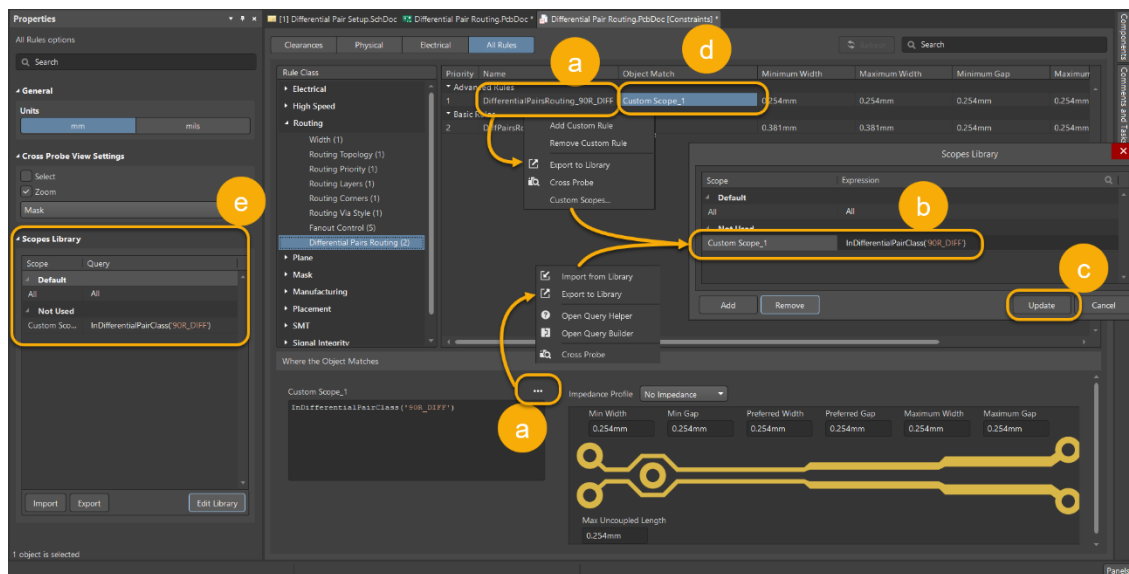

Figure 20. Constraint Manager - All Rules with Export to Library

6.3 High Speed Rule – Matched Length

49. Within the Constraint Manager, browse to the *Matched Lengths* rule in the *High-Speed* section on the left.

50. Next, you will configure a new Matched Lengths rule for the 90R_DIFF class.

51. You can now choose one of the options below:

Option 1: Repeat the steps you have learned in the Section 6.2 Routing Rule - Differential Pairs Routing

52. Right-click anywhere in the grid area and select the **Add Custom Rule** command from the context menu.

    a) Rename the new rule to `MatchedLengths_90R_DIFF`, Figure 21.

    b) At the bottom, select the radio button **Length Unit**.

        i) Add the value `0.5mm` next to tolerance.

        ii) Select the radio button **Within Differential Pair Length** .

    c) Select the **Ellipsis** button and choose **Open Query Builder** (Figure 22).

        i) Select as Condition Type / Operator *Belongs to Differential Pair Class*.

        ii) Select as Value *90R_DIFF*.

        iii) Close the *Building Query from Board* dialog with **OK**.

    d) The cell 'Object Match' is updated to `InDifferentialPairClass('90R_DIFF')`. Altium Designer will show the query in the lower left area of the *Constraints Manager*.

    e) Save the modifications with **Save All** 📑.
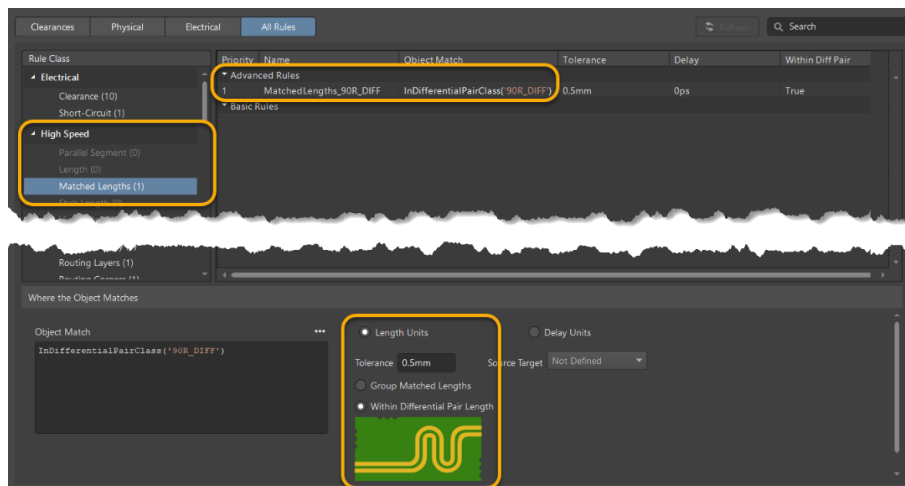
    f) Close the *Constraint Manager*.
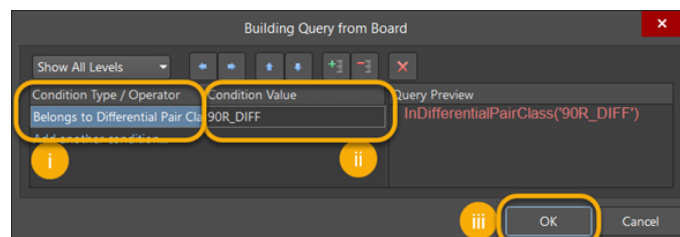

Figure 21. Creating a Matched Length Rule - Option1

Figure 22. Building Query for the 90R_DIFF class

Option 2: Use the Scopes Library to reduce the number of steps for creating a new rule

53. Right-click anywhere in the grid area and select the **Add Custom Rule** command from the context menu.

   a) Rename the Rule to `MatchedLengths_90R_DIFF`.

   b) At the bottom, select the radio button **Length Units**.

      i) Add the value `0.5mm` next to tolerance.

      ii) Select the ratio button **Within Differential Pair Length**.

   c) Select the **Ellipsis** button and choose **Import from Library** ✎ Import from Library , Figure 23.

      i) In the dialog *Scopes Library* select *Custom Scope_1*.

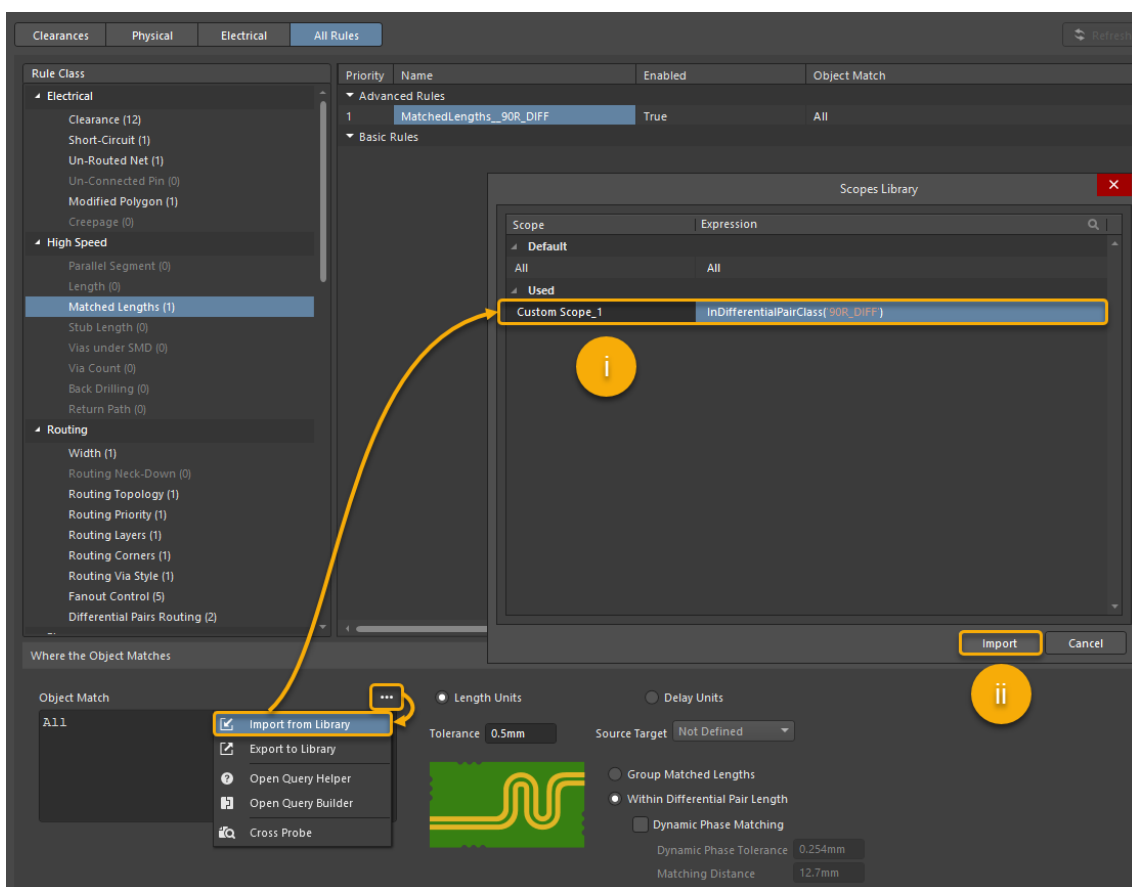   ii) Click **Import** to import the selected Custom Scope and to close the *Scopes Library* dialog.

Figure 23. Import from Library

d) The cell *Object Match* is updated to `Custom Scope_1`, Figure 24. Altium Designer will show the query in the lower left area of the *Constraints Manager*.

e) Save the modifications with **Save All** 💾.
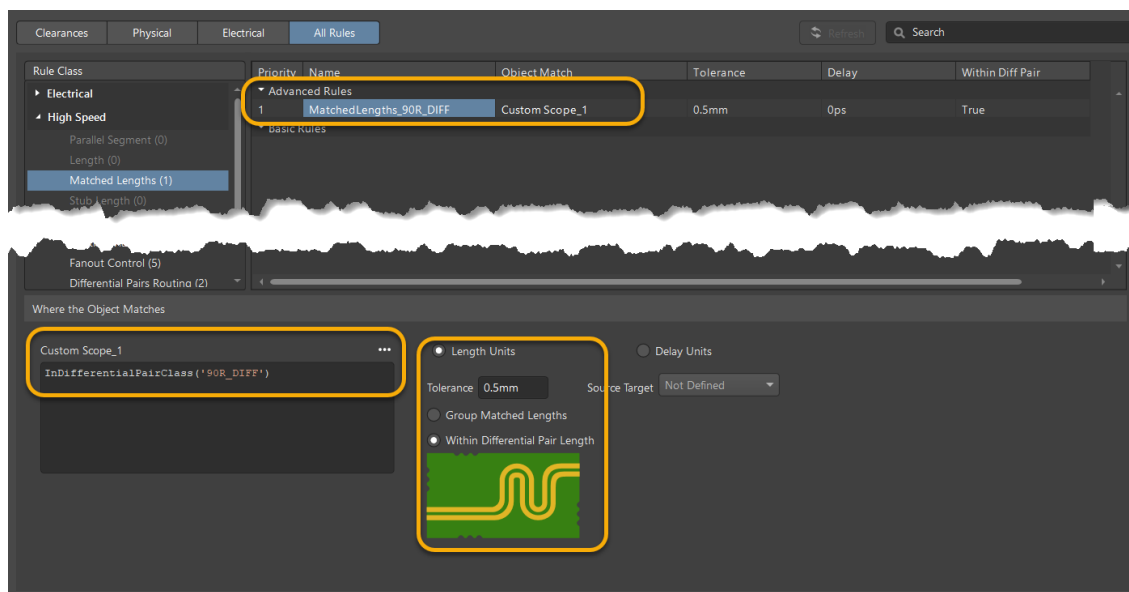
f) Close the *Constraint Manager*.

Figure 24. Creating a Matched Length Rule – Option2

54. With the new PCB Rules added, let's update the SCH and the constraints in the SCH.

a) Run the **Update Schematics in…** command using the **Design » Update Schematics in …** menu.

b) In the *ECO* dialog, select **Execute Changes** to transfer the new PCB rule information to the Schematic document.

c) Close the *ECO* dialog.

6.4 Differential Pair Routing

Prior to routing, ensure the **Smart Track Ends** option is enabled. This option forces connections lines at the end of a track and overrides the net topology. This is preferred for differential pair routing in the case you suspend a route and leave dangling tracks.

55. Open the *Preferences* from the gear icon ⚙ in the top-right corner of Altium Designer.

56. Under *PCB Editor* section, in the *General* page, verify the **Smart Track Ends** checkbox is enabled, as shown in Figure 25*.*
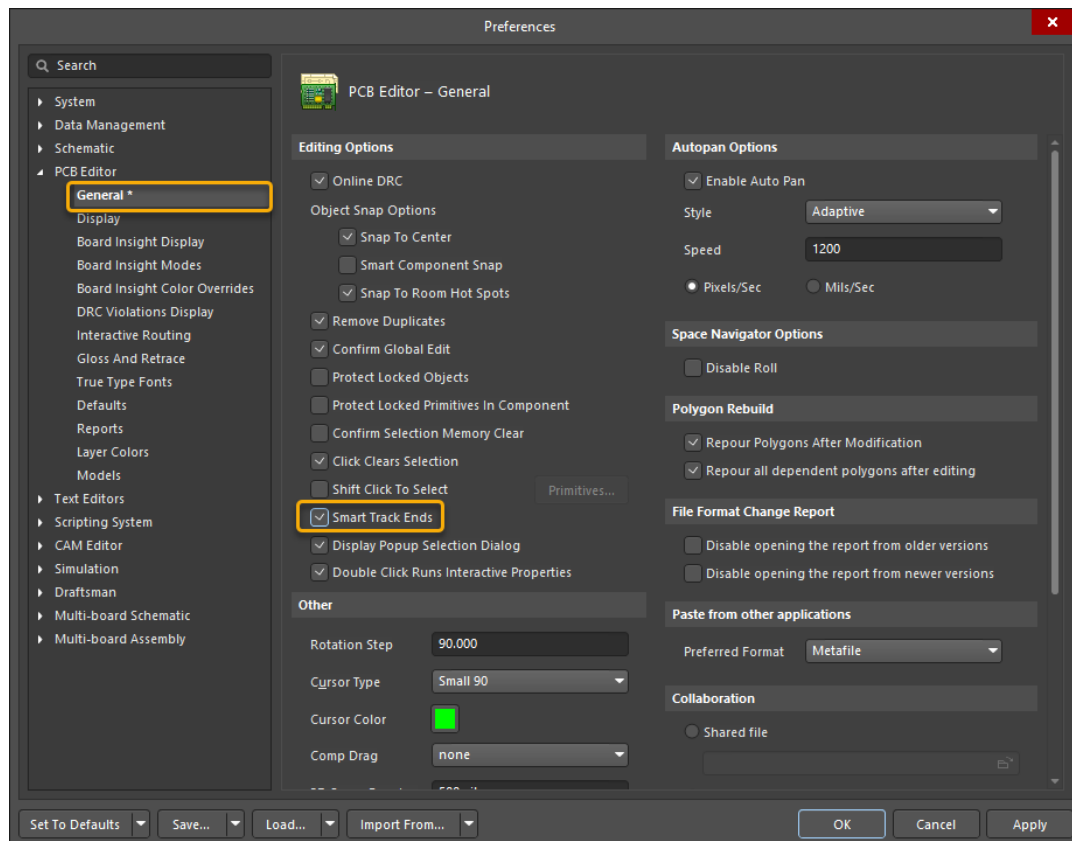
57. Select **OK** to close the Preferences and to continue.

Figure 25. PCB Preferences

58. Back to the PCB, ensure the Snap Grid is set to `0.1mm`. Press **G** to change the grid, if needed.

59. To start the routing select **Interactive Differential Pair Routing** from the Active Bar, Figure 26, or use the shortcut **U»I**.
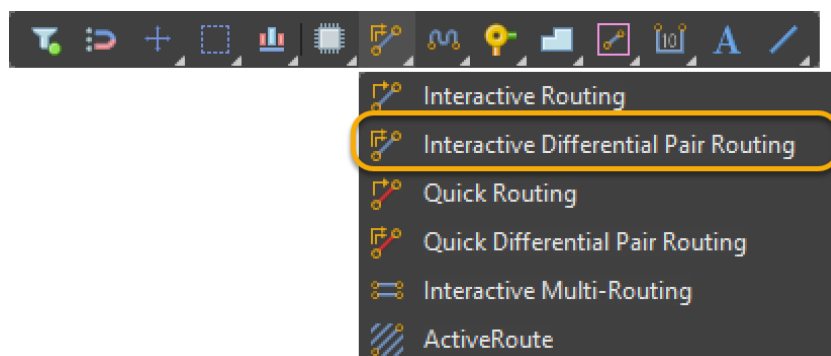

Figure 26. Interactive Differential Pair Routing

60. Select `Pin5` of `U1` to start the routing.

20

61. Press the **Tab** key and set the routing properties, as shown in Figure 27.
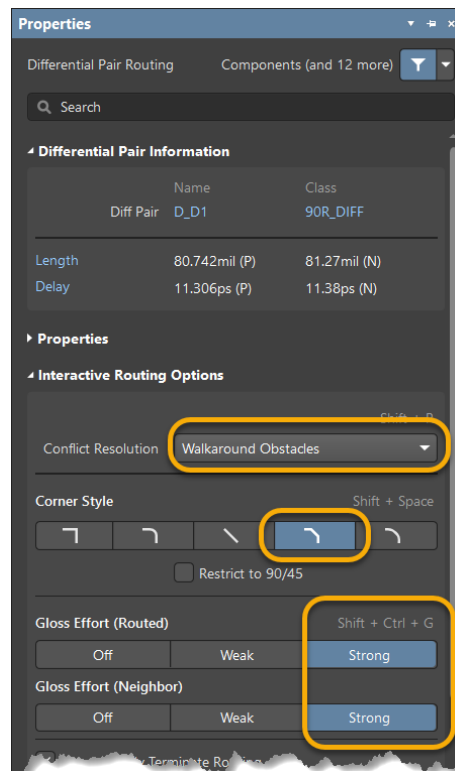

Figure 27. Differential Pair Routing Settings

62. Resume the routing.

63. When the differential pair routing approaches the target, left-click on the target pad or use the Autocomplete Segments To Target feature (**Ctrl+Left-Click**) to finish the routing automatically.

64. Use the same method to route the other differential pair.

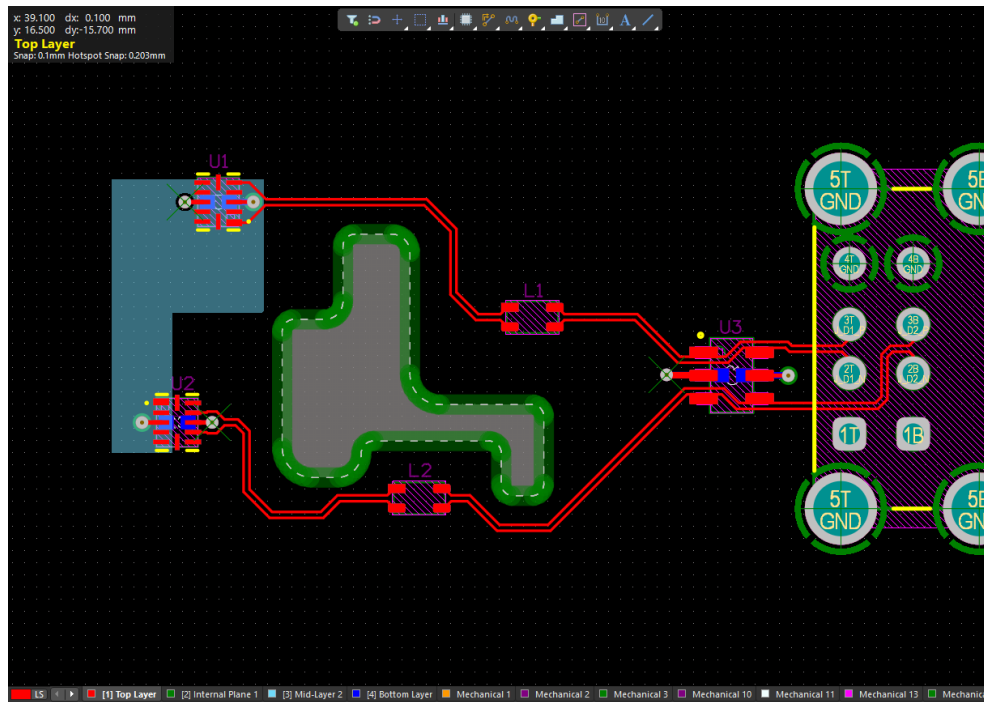65. The final routing should look similar to Figure 28.

Figure 28. Final Routing

> Caution: The two traces should be of equal length, within tolerances of the logic family. The Length Tuning is covered in module *Length Tuning Routing with Constraint Manager*.

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

67. 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 ○`.
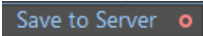
    b) Select **Save to Server**.

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

        i) Add the comment `Differential Pair Routing with Impedance Profile and Constraint Manager - [Add Your Name] - Finished`.

        ii) Select **OK**.

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

Congratulations on completing the Module!

Differential Pair Routing with Impedance
Profile and Constraint Manager

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**