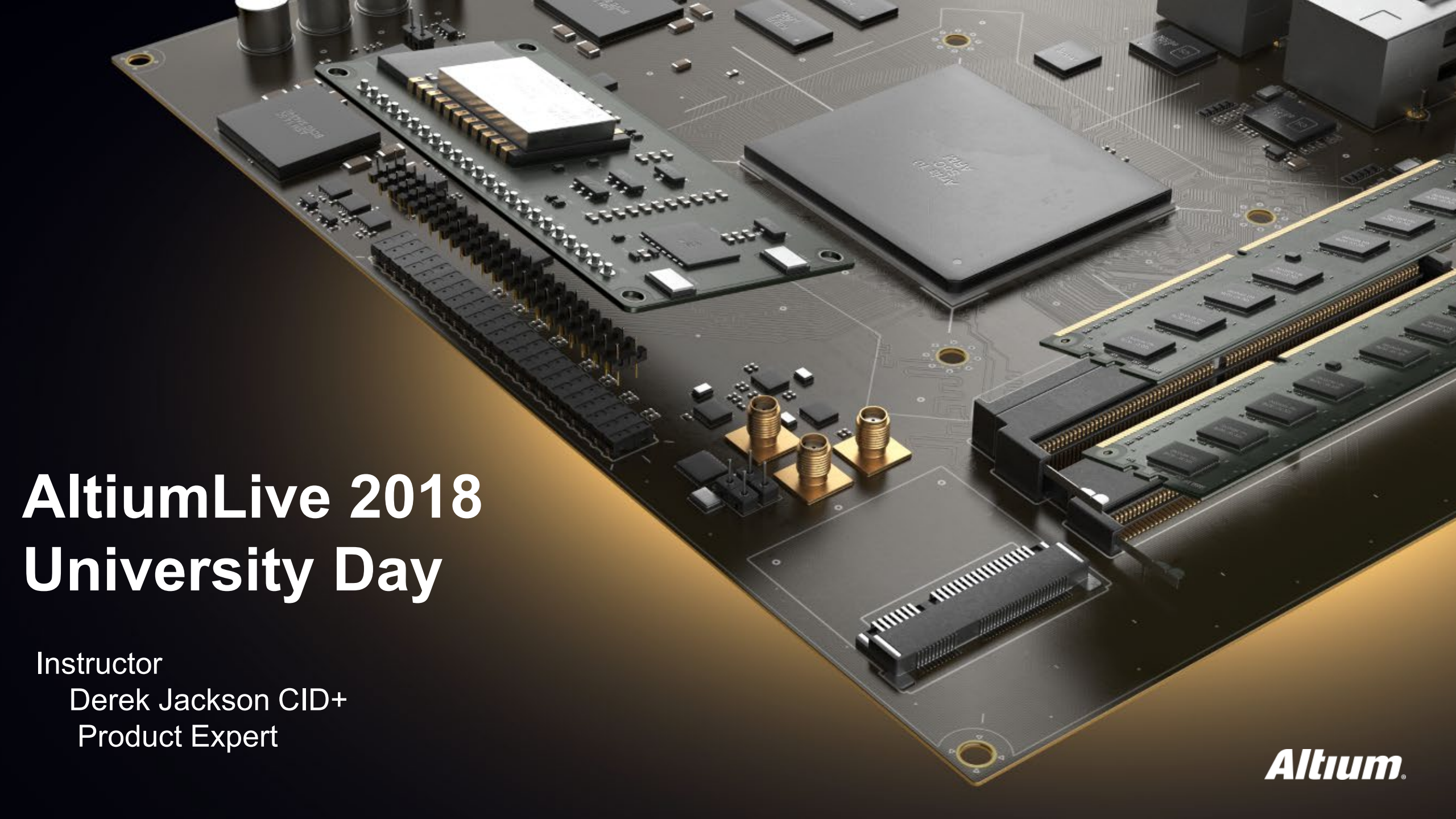
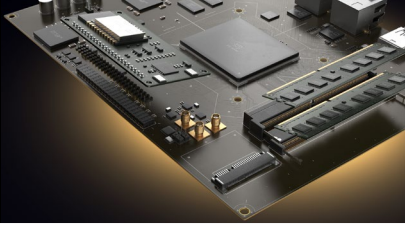


AltiumLive 2018
University Day

Instructor
Derek Jackson CID+
Product Expert

Altium

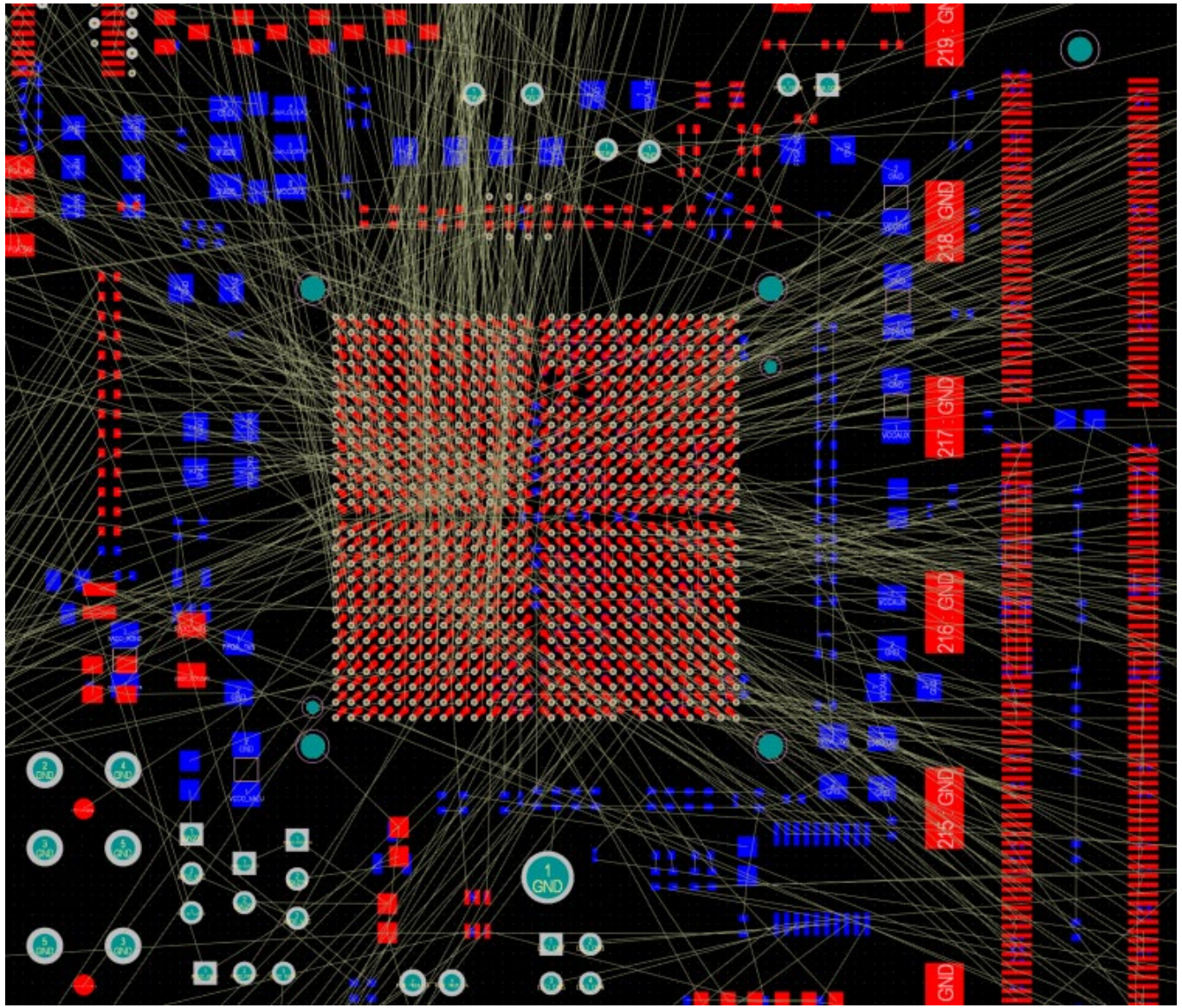


How to Route and tune high speed topologies with xSignals

A look at xSignals in Altium Designer.

Nets (in review)

- Nets are assigned at the schematic level and transferred to the PCB.
- In the Pcb document, without any routing, we see connection lines between unconnected primitives of the same net.

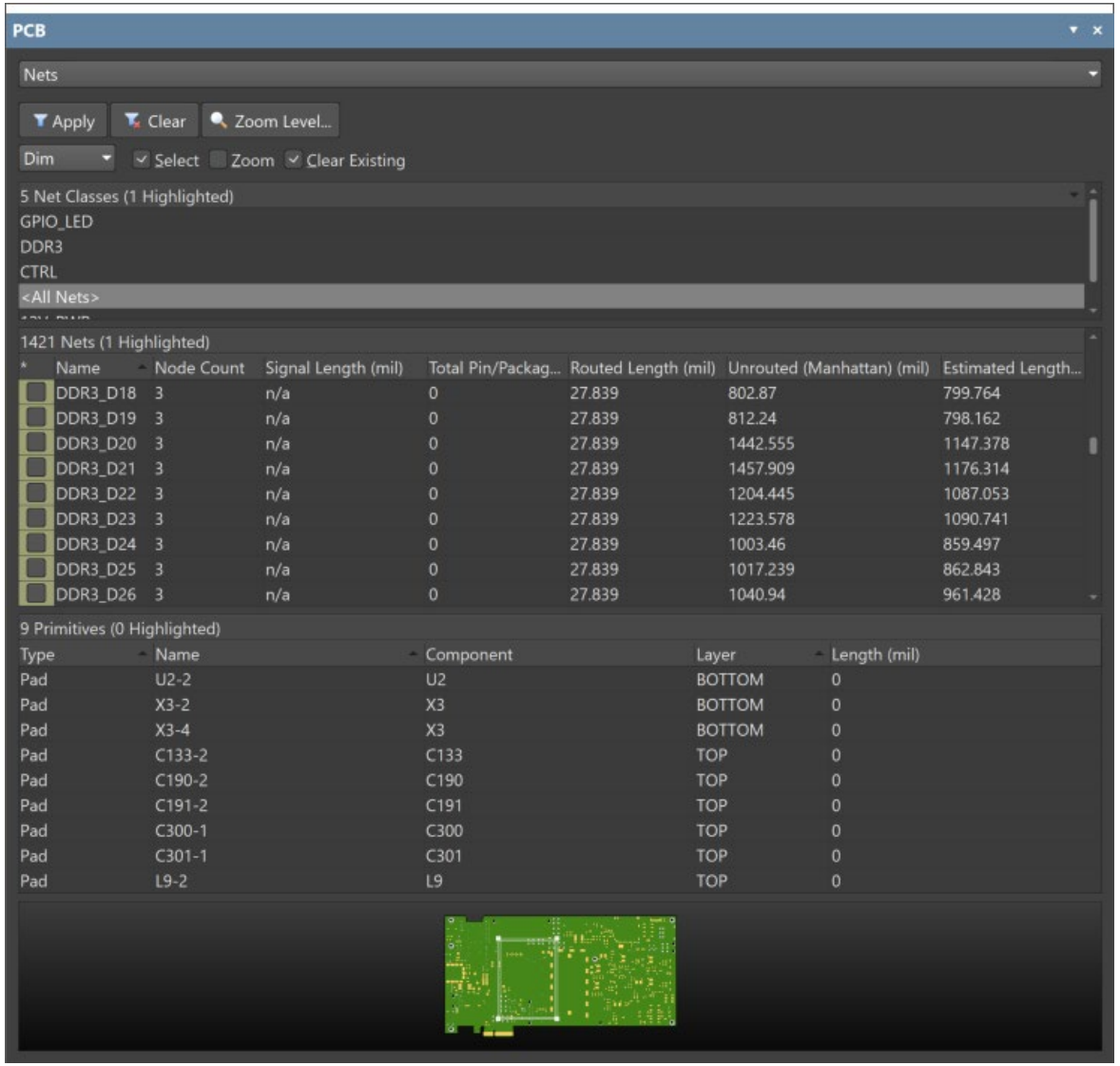


Nets (in review)

In the PCB Panel, under the Nets section, we can see net information , including:

- **Net Name**
- **Routed Net Length**: The routed track length that does not sort out overlapping segments.
- **Signal Length**: A calculated distance that provids an accurate calculation length of routing between two pad objects, excluding track overlaps in the total, and handles fills, via lengths.

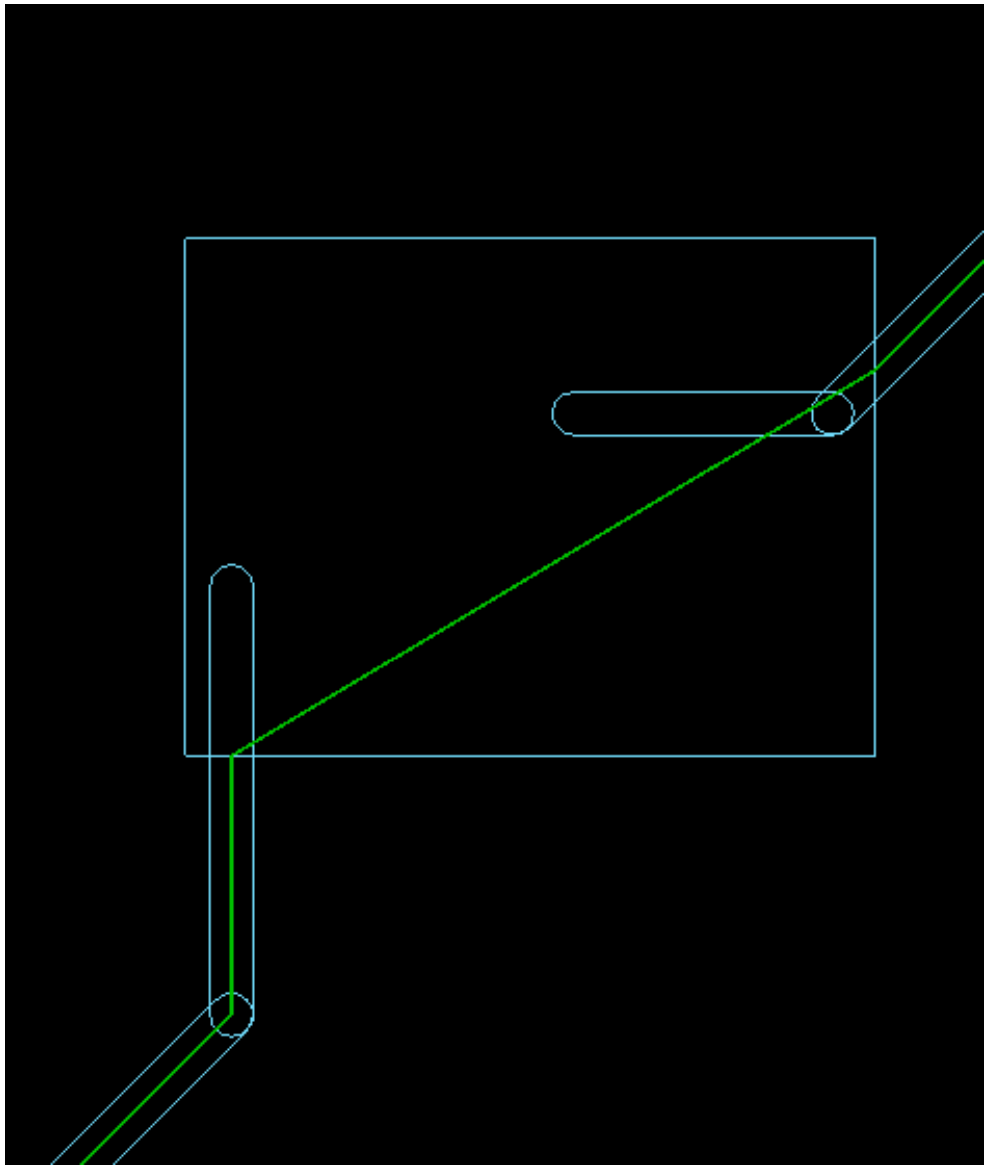
For more than two connected nodes, this will display **n/a**.

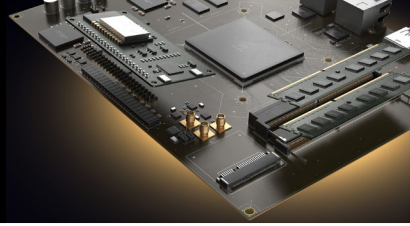


Nets (in review)

- **Signal Length**: The Signal length is automatically generated, and we have no control over it. Well are going to learn how to setup our own “Accurate” length using xSignals in this discussion.

- **More info:** You can read more about Nets in the PCB here: https://www.altium.com/documentation/18.1/display/ADES/PCB_Pnl-PCB((PCB+-+Nets))_AD

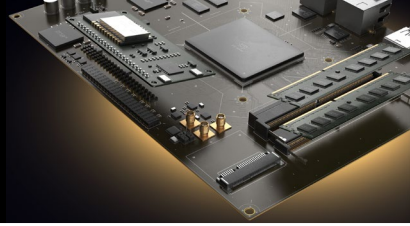




Length Tuning:

- For signals that need to travel from a source component and arrive at different destinations at the same time will need to have their routed tracks the same length. This is done using various Length Matching techniques.

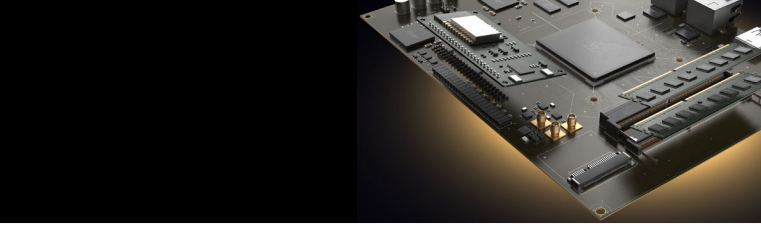
- **More info:** https://www.altium.com/documentation/18.1/display/ADES/((Length+Tuning))_AD



Length Matching Rules

Length Matching Rule:

- In Altium Designer, a Matched Length Rule can be defined which defines the allowable range of the track lengths (**Design » Rules » High Speed » Matched Lengths**) for the specified nets.

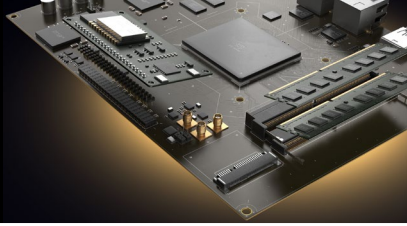


Length Matching Rules

The rule interprets lengths from all the routed nets referenced by the rule as follows:

Matched Net Length's **Min Limit** = Longest Net – Rule Tolerance
Matched Net Length's **Max Limit** = Longest Net
(constrained by the *length rule)*

Valid Range = Highest Min Limit to Lowest Max Limit
(most stringent combination of Length and Matched Length rules)
Target Length = Lowest **Max Limit**



Matched Length Rule:

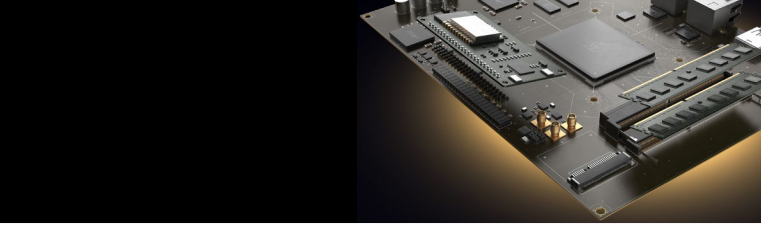
"Group Matched Lengths": Set the target for all nets targeted by the rule, looking at the longest net length to set the maximum.

"Within Differential Pair Length": Matched length is between the legs of a differential pair.

Tolerance: Provides the amount of variance from the total routed length by up to this amount. The target length range is defined by the equation:

(Longest Length - Tolerance) ≤ Current Length ≤ Longest Length

https://www.altium.com/documentation/18.1/display/ADES/PCB_Dlg-MatchedNetLengthsRule_Frame((Matched+Lengths))_AD



Interactive Length Tuning

- When length tuning (**Route » Interactive Length Tuning**), selecting a track to tune, a length gage is displayed showing the Min/Max Length, estimated length, and the target length.

Pressing the Tab key, the **Min** and **Max** Limits will be displayed in the Properties panel.

In the PCB Panel (Nets section) the net length is color coded for easy viewing.

- **No highlight** color = length is within tolerance.
- **Yellow** highlight color = the route length is less than the rule minimum.
- **Red** highlight color = the route length is greater than the rule maximum.

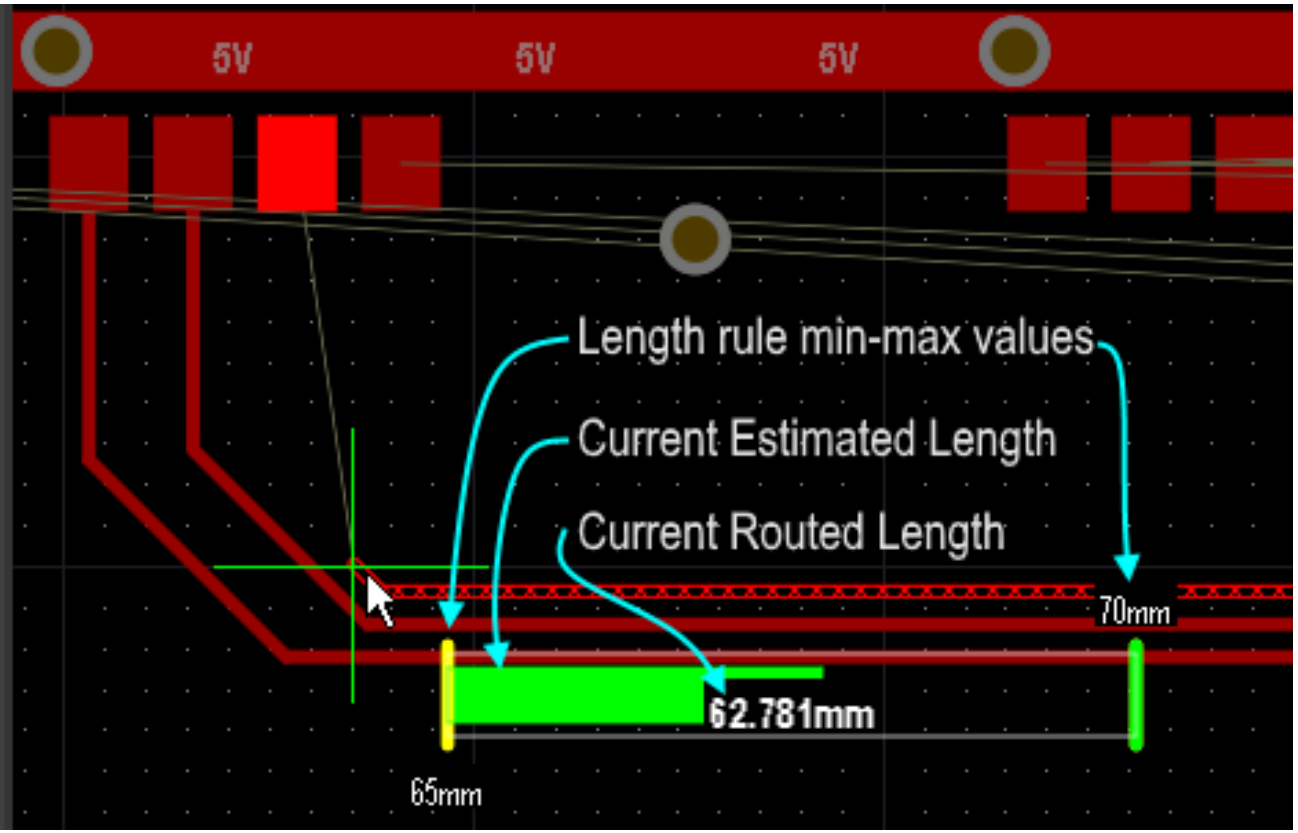
Interactive Length Tuning

8 Nets (1 Highlighted)

*	Name	Routed Le...	Estimated L...	Signal Length (	Unrouted (Ma...
	LCD_DB0	70.207	70.207	70.207	0
	LCD_DB1	68.944	68.944	68.944	0
	LCD_DB2	62.781	67.723	n/a	5.59
	LCD_DB3	0	60.45	61.245	61.245
	LCD_DB4	0	52.556	52.855	52.855
	LCD_DB5	0	51.313	52.985	52.985
	LCD_DB6	0	50.063	52.215	52.215
	LCD_DB7	0	48.82	51.445	51.445

4 Primitives (0 Highlighted)

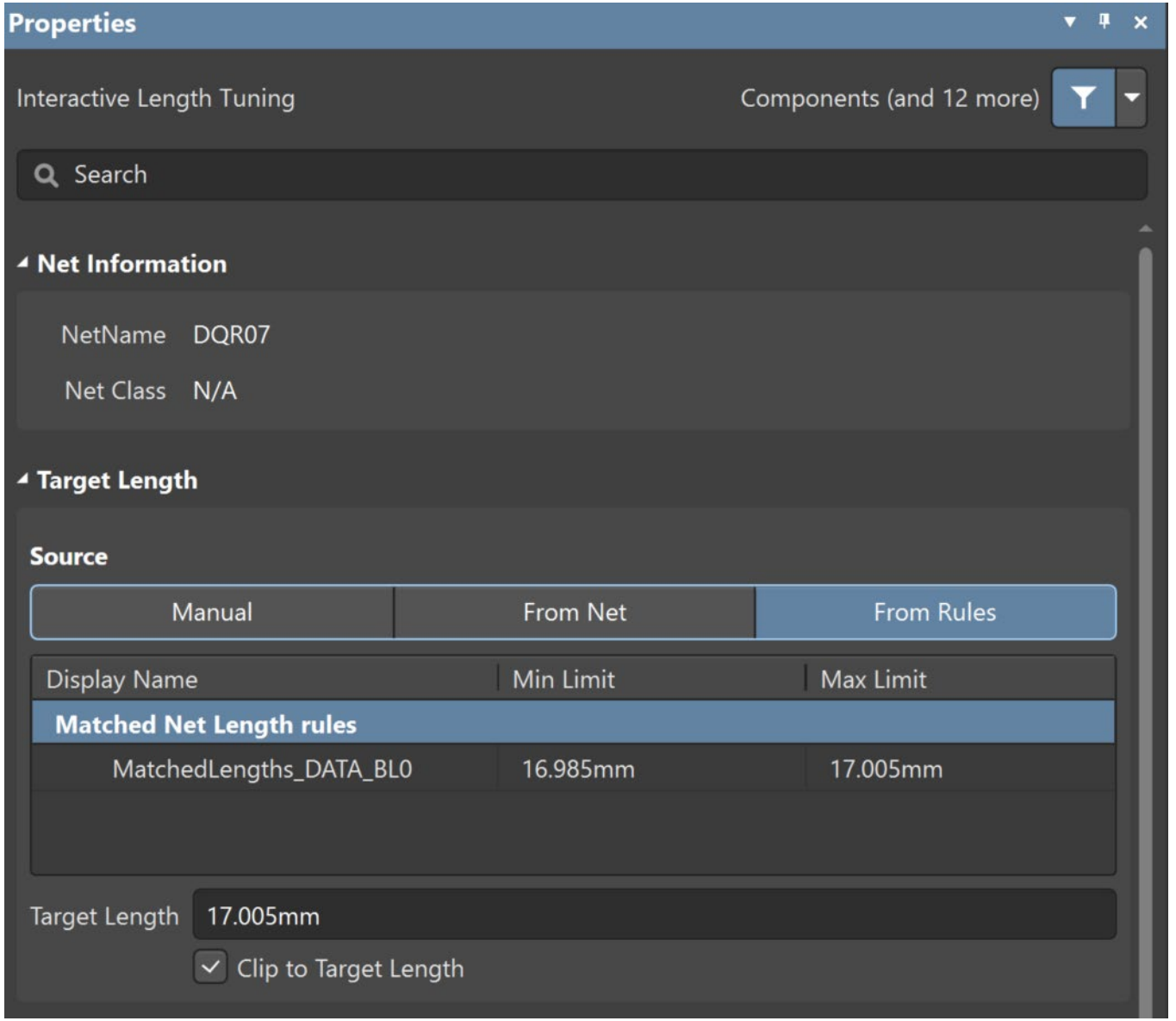
Type	Name	Component	Layer	Length (mm)
Pad	RA2-7	RA2	Top Layer	0
Pad	U1-17	U1	Top Layer	0
Track	Width=0.2mm (190		Top Layer	5.551
Track	Width=0.2mm (193		Top Layer	5.556



Interactive Length Tuning

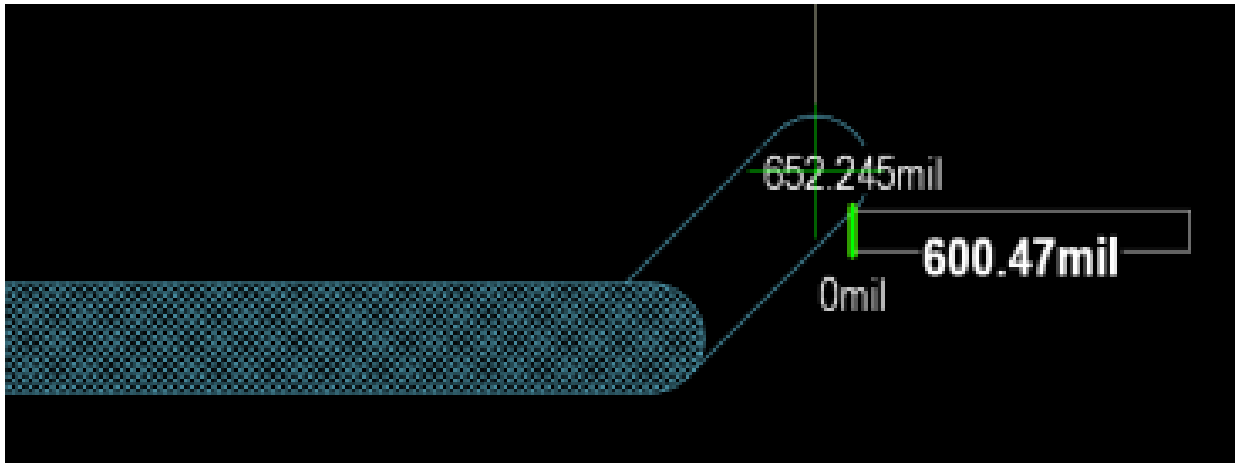
There are three ways to define the length limits when tuning:

- Manual
- From Net
- From Rules

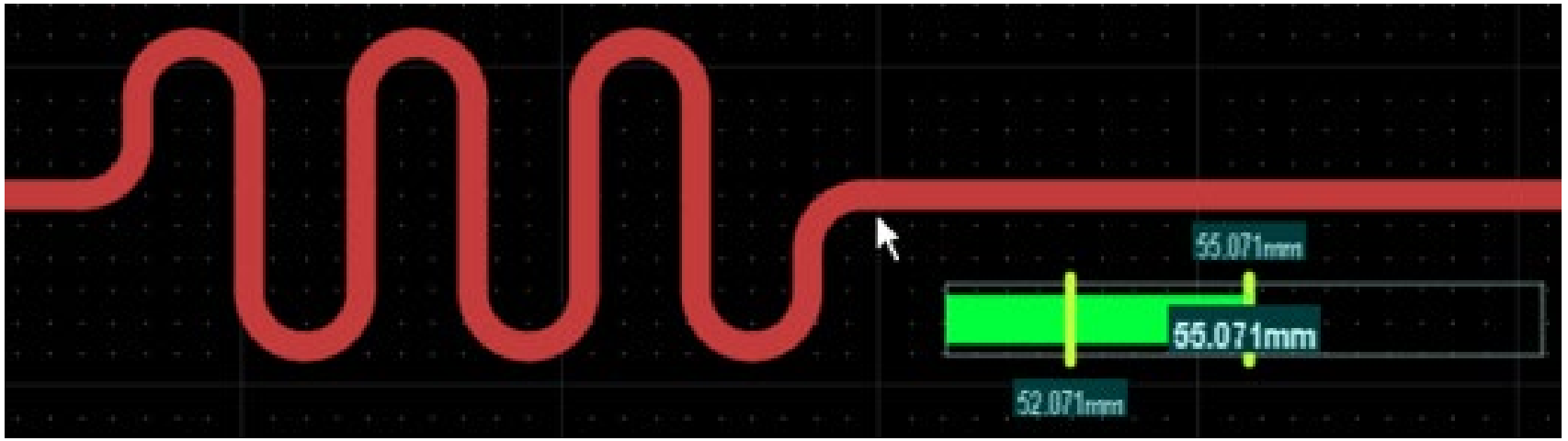


Interactive Length Tuning

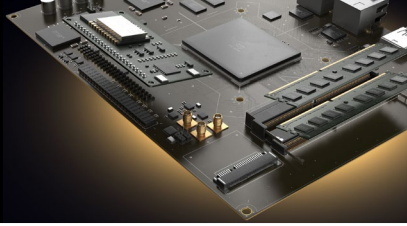
- During routing, display the length gage by pressing **Shift+G**
- If no Length Matching rule applies, the min/max will display 0/Current length or 0/0.



- During Length Tuning, the Min, Max, and target lengths are shown.



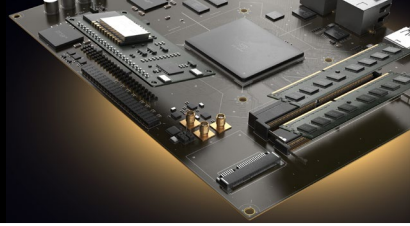
A very old Length tuning video, but great information: http://videos.altium.com/trainingcenter/player.html?ep=1047



Accordions:

For length tuning, accordions are the length of track added to the route to increase the length.

- Style: Mitered Lines/Arcs or Rounded. (**Space Bar** to cycle)
- Interactive Routing: When routing, accordions can be added by pressing **Shift+A**.
- Tab: pressing Tab during a tuning / Routing command, will display its properties in the Pattern section, allowing adjustments. amplitude, gap, and miter. Shortcuts are listed in the Panel. During the command, you can press the **~** to get a list of shortcuts.



Accordion Object:

For length tuned tracks, an accordion control object gets added with the length tuning. is control appears as a bounding box around the tuned section. Modifying an accordion:

- Double-clicking: Select an accordion to edit its properties in the Properties Panel.
- Dragging: Dragging the sides or vertices of the accordion object will adjust the added tuning and scale accordingly.

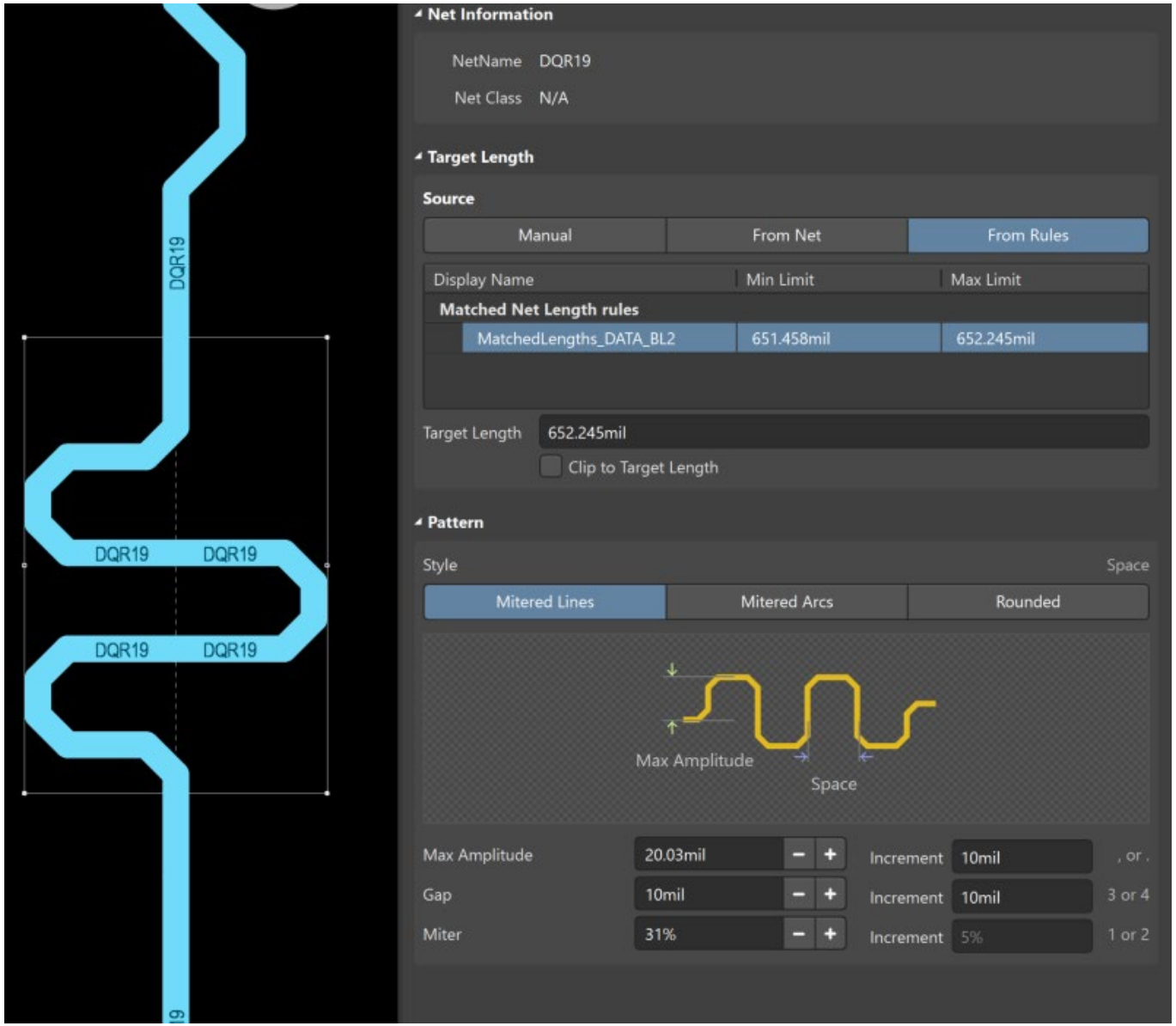
Currently, accordion control objects are not added when tuning Differential Pairs: (https://bugcrunch.live.altium.com/#Idea/8016)

More Info: https://www.altium.com/documentation/18.1/display/ADES/PCB_Obj-Accordion((Accordion))_AD

Altium Designer xSignals

Accordion Object:

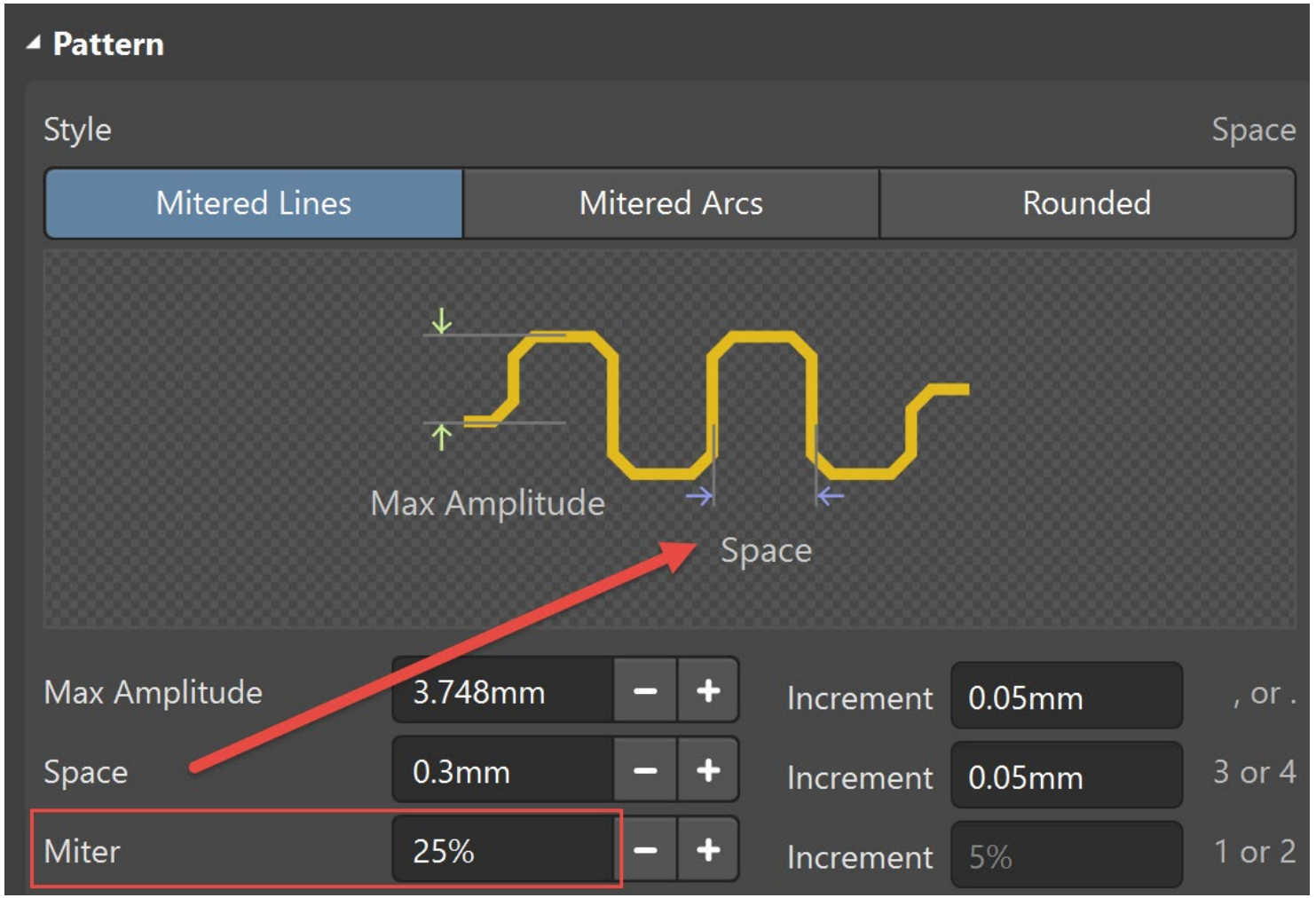
Shown selected with properties showing in Properties Panel.



Accordions:

The Accordion style can be 90, mitered with lines or serpentine.

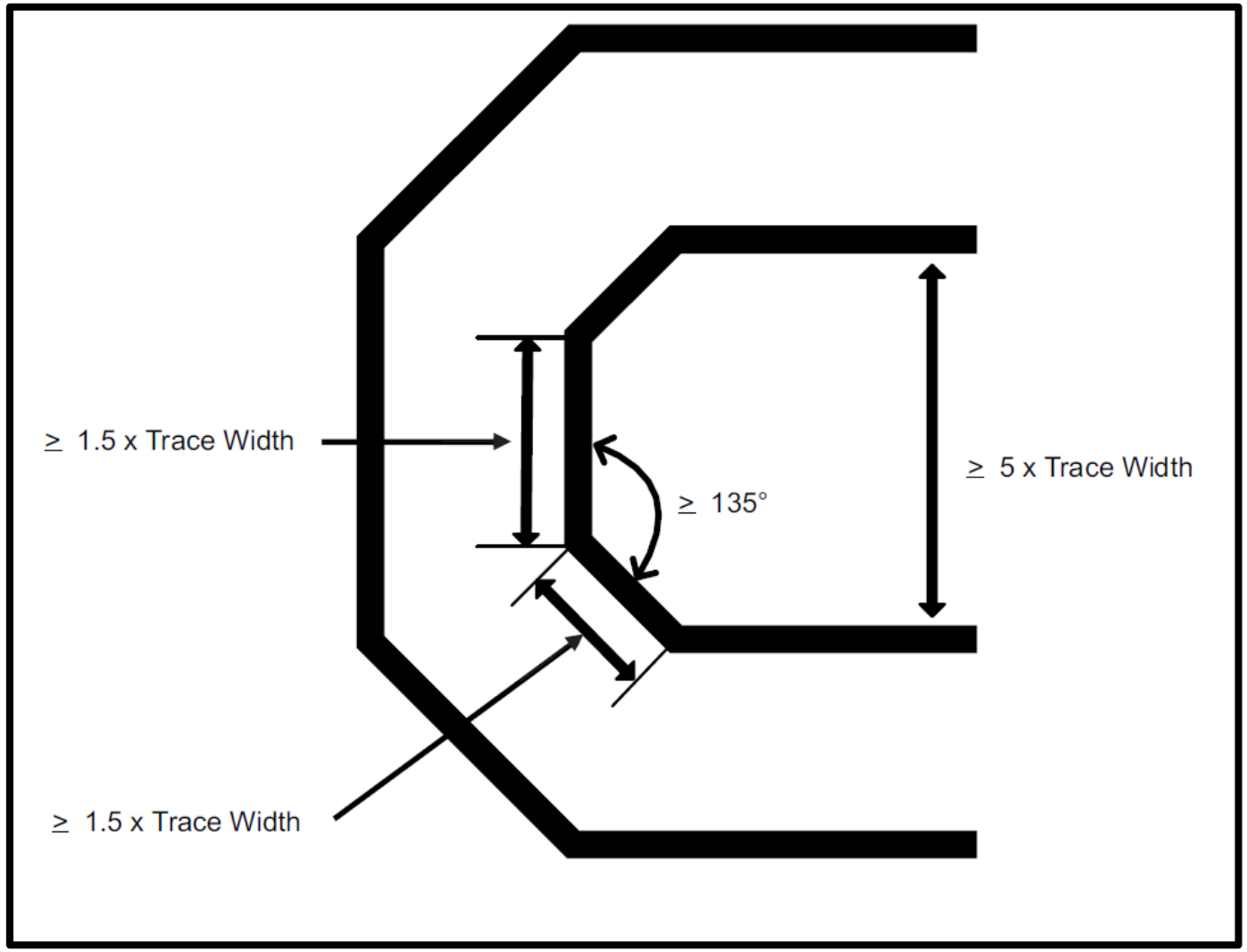
- When using miters, the Space is the inner gap, and the miter is represented as a percentage of the space length value.



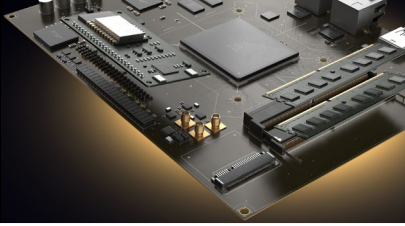
Accordions:

So, what value do you use for the miter percentage?

- Typically Space = 3-5 x Width, Miter >= 1.5 x W and Top line is 1.5 x W.
- The Texas Instrument's USB3 guide recommends that the space is 5x the width, the length of the inner bend is 1.5x the trace width, and the inner segment's top of the bend is 1.5x as shown.



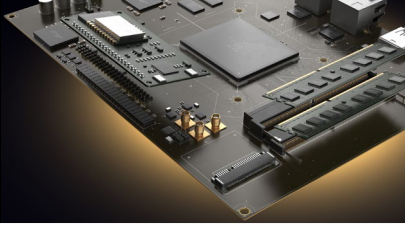
- **Extra reading: https://docs.toradex.com/102492-layout-design-guide.pdf**



xSignals:

An xSignal is defined as a path for a signal to travel between a source and destination (pin pairs), and can include termination components as well as branches (Balanced-T or Fly-By).

- Virtual Net: Similar to the Signal Length calculation, but differs that xSignals can be created and used for accurate route measurements.
- Creation: xSignals can be created with or without routing between the selected pin pairs.
  - xSignals can be created from selected Pads, components or the xSignal wizard.
  - xSignals will follow the routing topology set for the selected nets.

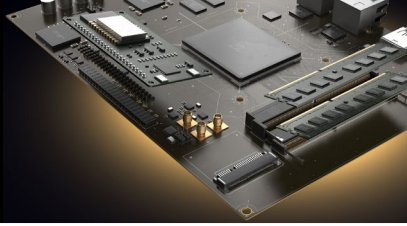


xSignal Creation:

Accessing the **xSignals** menu (**Design » xSignals**), depending on what is selected will affect available menu options…

- From the Menu:
  - **» Run xSignals Wizard**
  - **» Create xSignals**
- With Pads selected (from menu or right-clicking on a selected pad):
  - **» Create xSignal from selected pins**
- With Components selected:
  - **» Create xSignals between components**
  - **» Create xSignals from connected Nets**

More information:
https://www.altium.com/documentation/18.1/display/ADES/((Defining+High+Speed+Signal+Paths+with+xSignals))_AD



xSignal Panel:

There are three regions in the xSignal section of the PCB panel:

- xSignal Classes
- xSignals belonging to the selected class
- Primitives in the xSignal (pads, tracks, arcs, and vias)

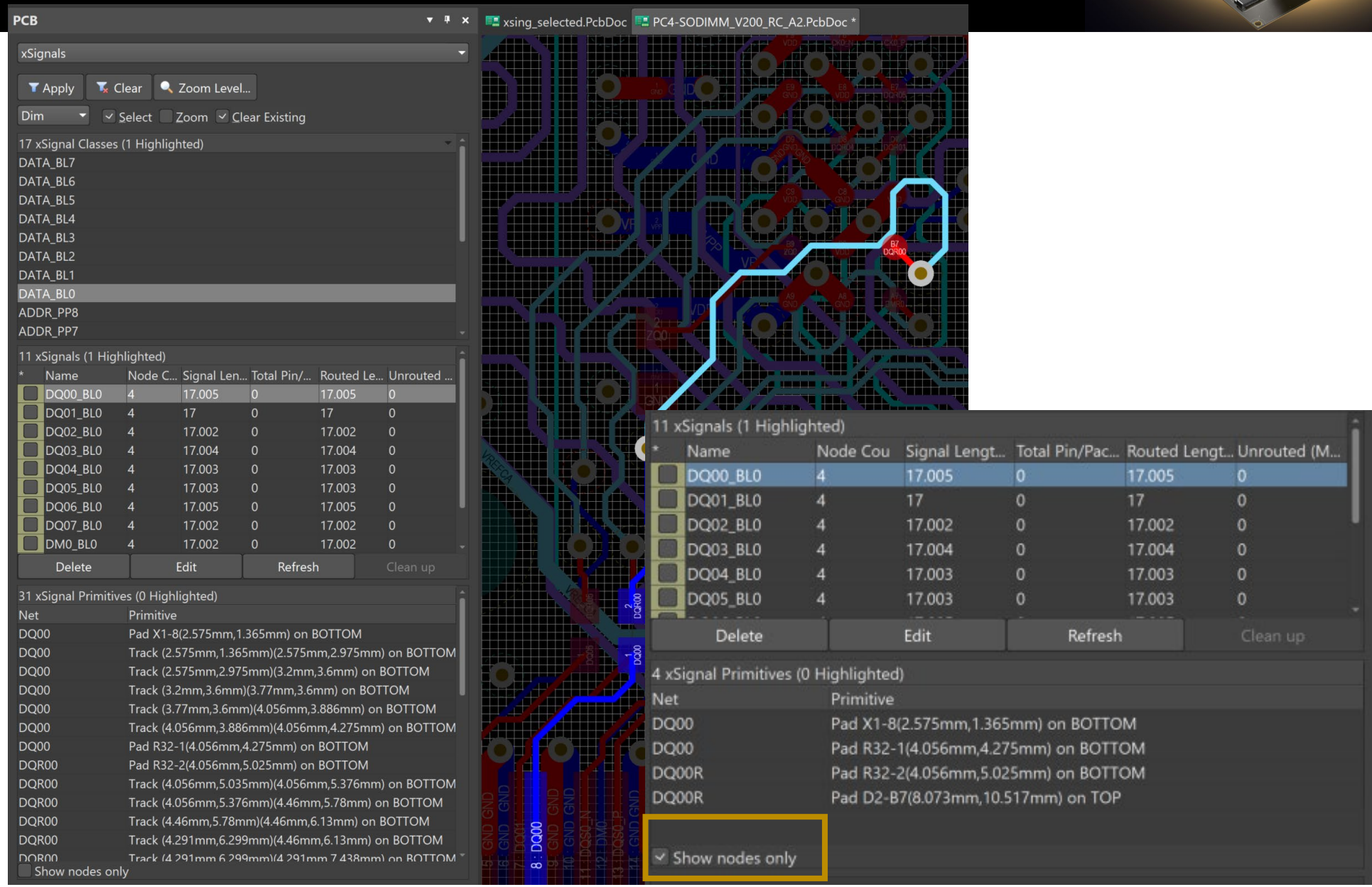
In the xSignal section of the PCB panel, we also see Signal Length, Routed Length, Unrouted Length, and Total Pin/Package length.

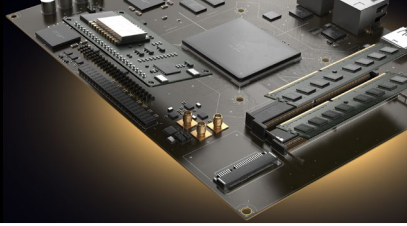
Similarly to the Nets section, for Length Matching rules we will see the nets lengths change color based if they are shorter, longer, or in the target length range as we saw earlier.

Altium Designer xSignals

xSignal Panel:

Shown, for the selected Net signal, all primities are sown. Clicking the Show nodes only will only display connected Pads.





Pin-Package Delays:

- To account for the length added from routing inside the chip package, most manufacturers will supply a pin package length.

- Adding a pin package length to your footprint will add the total package length to the calculated signal length when a single track connects to it (flyby topologies are excluded).

- The length can be define at the schematic level (recommended), or directly in the PCB.

- The Intel Memory Interface Handbook recommends including package delays for speeds greater than 800MHz.

Extra reading Pin Pairs:
https://www.altium.com/documentation/18.0/display/ADES/PCB_Cmd-ManagePinPairs((ManagePinPairs))_AD

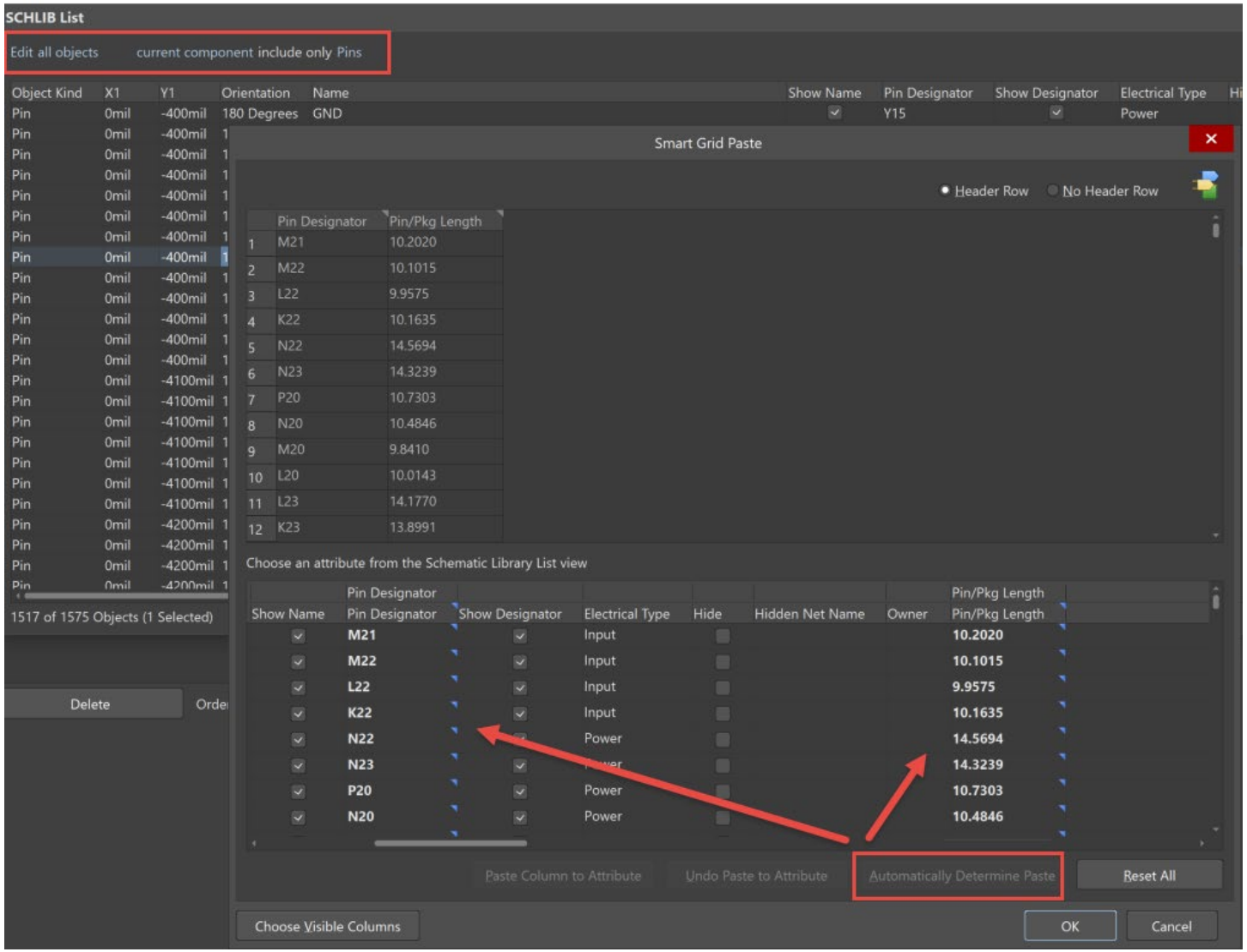
Altium Designer xSignals

Pin-Package Delays:

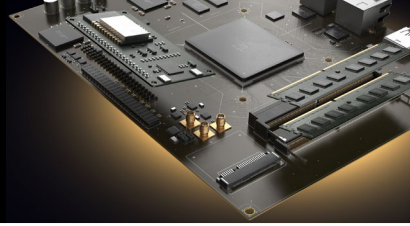
Smart Paste in SchLib List Panel.

	A	B
1	Pin Designator	Pin/Pkg Length
2	M21	10.2020
3	M22	10.1015
4	L22	9.9575
5	K22	10.1635
6	N22	14.5694
7	N23	14.3239
8	P20	10.7303
9	N20	10.4846
10	M20	9.8410
11	L20	10.0143
12	L23	14.1770
13	K23	13.8991
14	H21	9.8334
15	H22	9.9566
16	L19	10.0245
17	K18	9.8406
18	K20	15.5974
19	K21	15.2531
20	J23	12.3338
21	H23	12.0526
22	J19	9.7968
23	J18	9.9681
24	J20	10.4902
25	J21	10.1502
26	AN26	23.8673
27	AP26	23.9324
28	AN23	31.8239
29	AN24	31.5376
30	AK26	25.5254
31	AL26	25.3972
32	AK23	24.0356
33	AL23	23.7827
34	AM24	23.8698
35	AL24	23.8060
36	AL25	24.4561

10SX066NF40



Package download for Arria 10



xSignal Wizard:

The intent of this wizard is to automatically create the xSignals, Matched Length Groups, Diff Pair Matched Lengths.

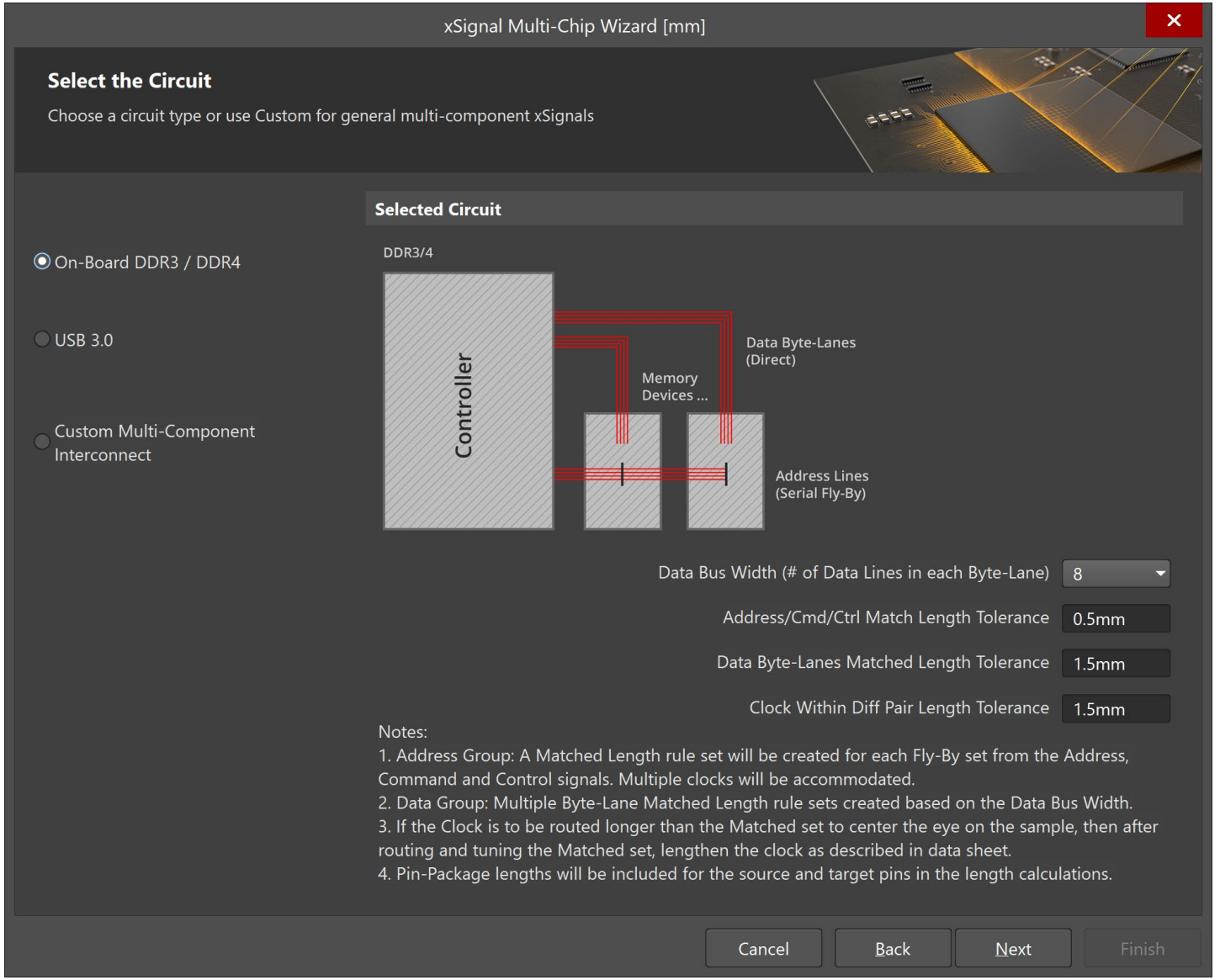
- xSignals Wizard: allows creating:
  - **DDR3/4**
  - **USB 3.0**
  - **Custom Multi-Component Interconnect**

Altium Designer xSignals

xSignal Wizard:

DDR3

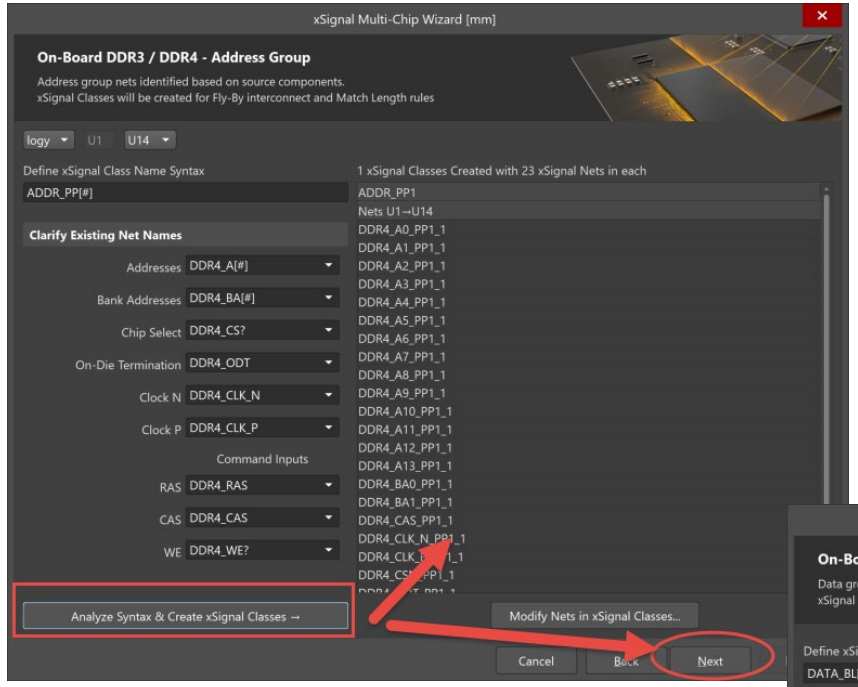
- Supports Flyby or Balanced T.
- For Balanced T topology, at the separation point, signals must use a special single pad component to define the split point.
- Do not use # in net names as this represents the bank number when defining.

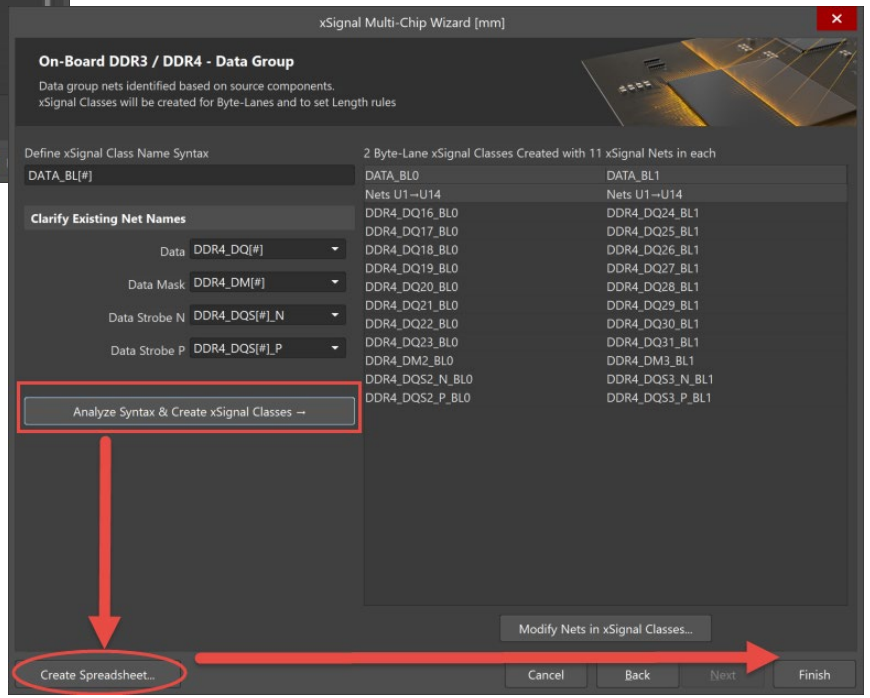


Altium Designer xSignals

DDR3

- Click the Analyze button once net names are filled in.
- Use # to represent the banks.
- Use * and ? Wildcards when necessary.
- Click the Modify Nets to add/remove nets as required.
- On Data source page, click the Create Datasheet to get detailed output of generated xSignals.



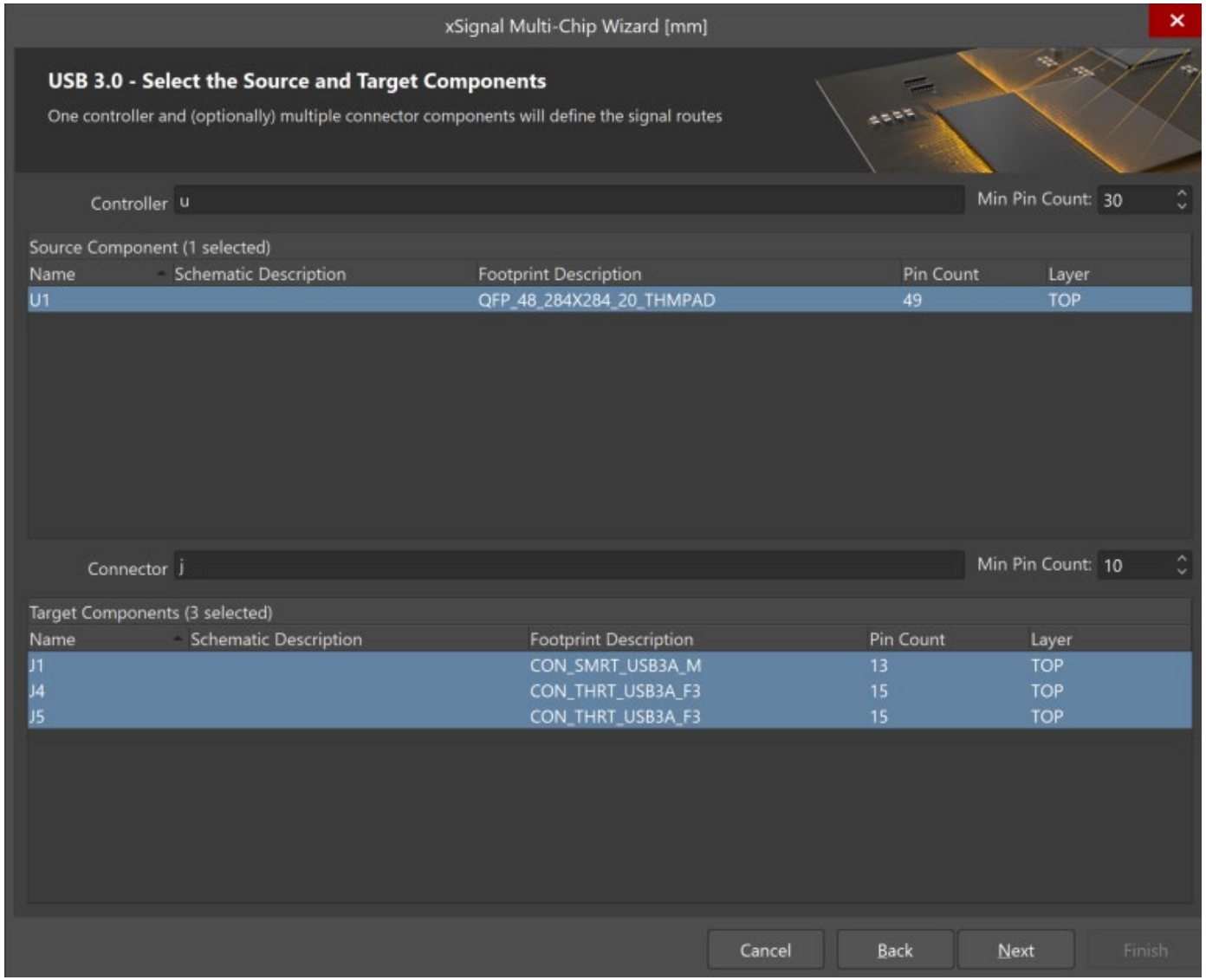


Altium Designer xSignals

xSignal Wizard:

USB 3.0

- The USB Wizard will add length matching for the diff pairs.
- Allows selection of multiple connectors
- TI USB 3 reference: http://www.ti.com/lit/an/spraar7g/spraar7g.pdf

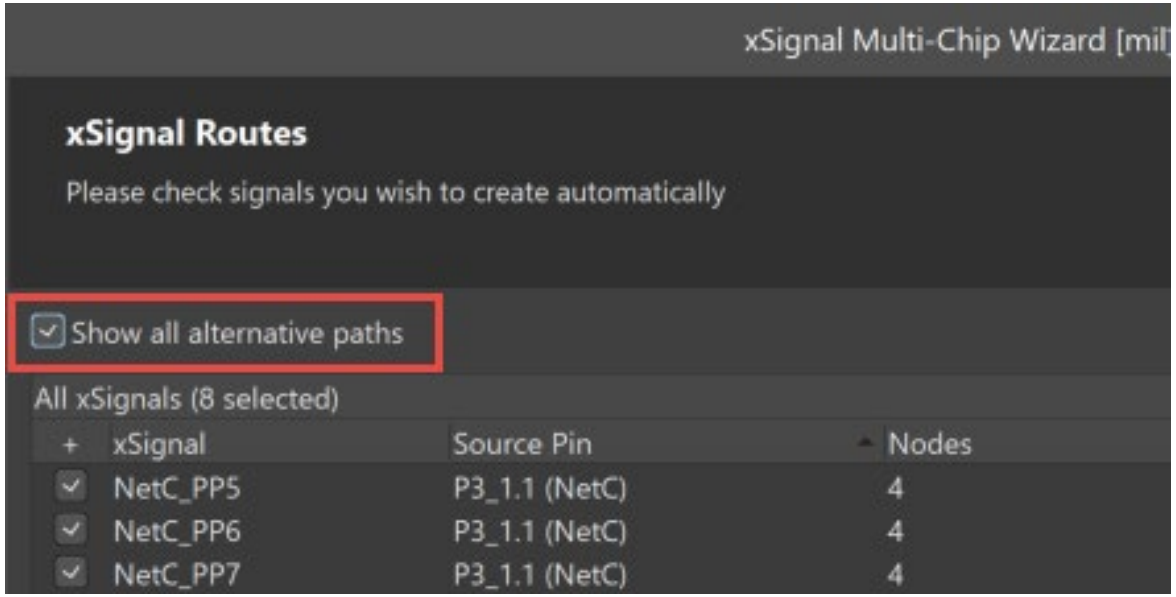


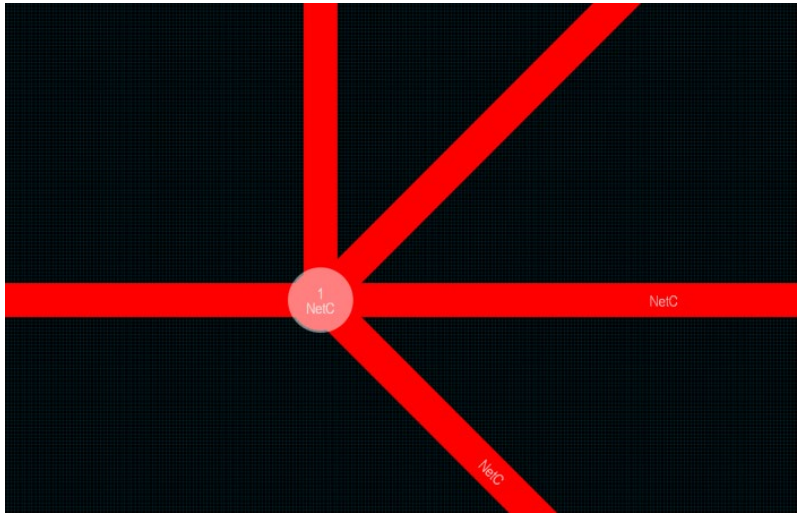
Altium Designer xSignals

xSignal Wizard:

Custom Multi-Component Interconnect

- Looks at all signals between source and target components.
- Allows to select specific nets or use a net class to constrain nets to include.
- Generates Length Matching rules for the generated xSignals.
- For signals that branch, enable the Alternate Path checkbox to see all routes between the source and targets.





Altium Designer xSignals

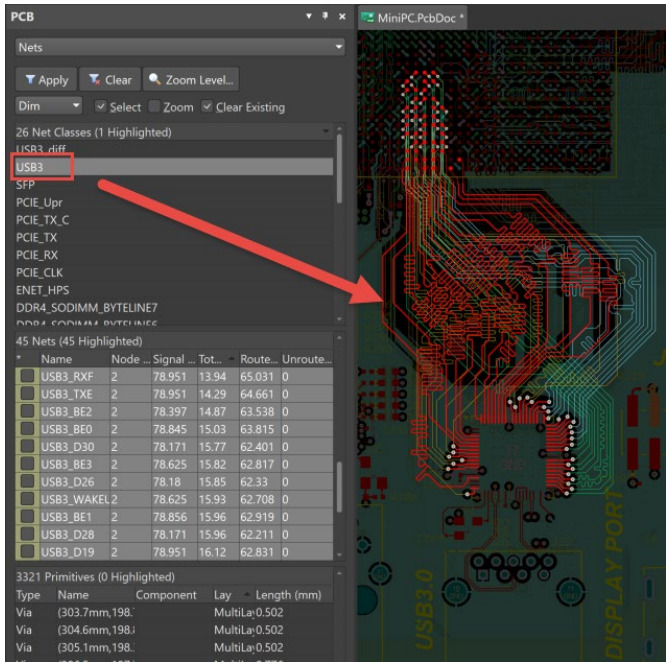
USB 3.0

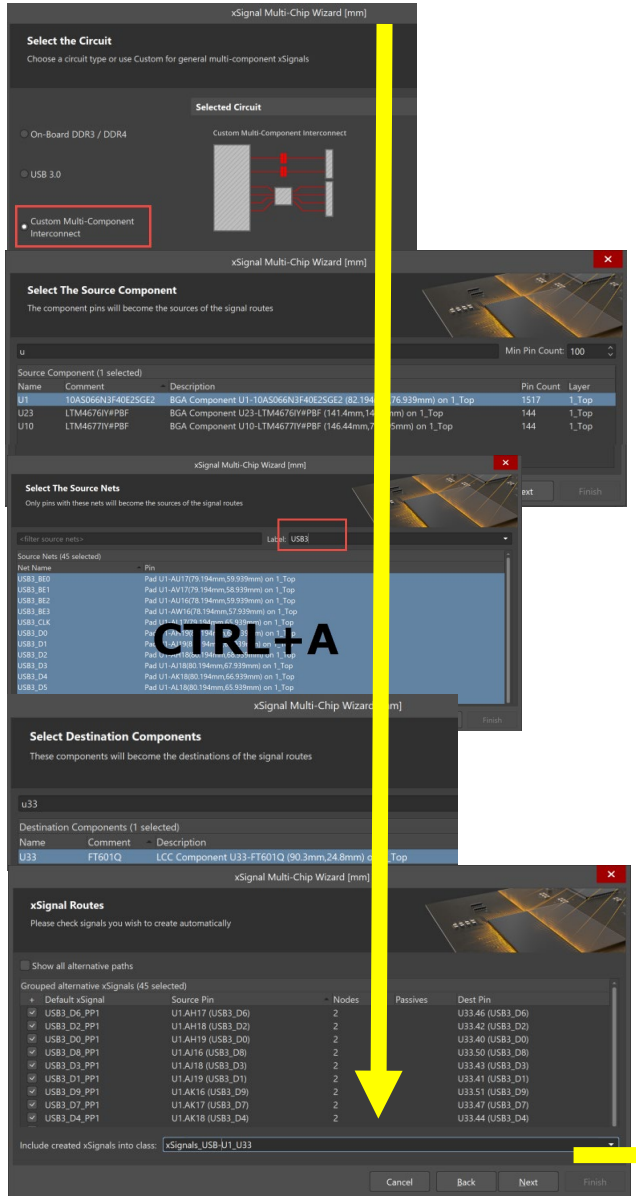
The MiniPC has a multiplexed USB.

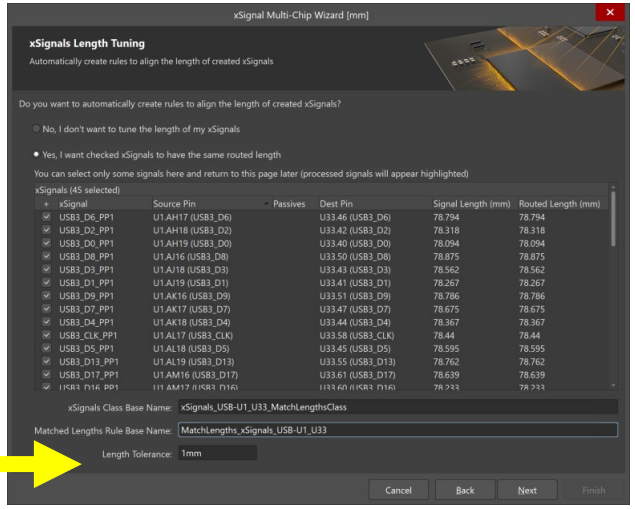
Run xSignal wizard
Custom multi-Component Interconnect.

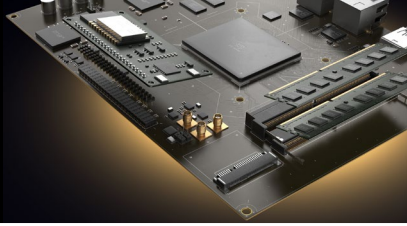
Source: U1
Select USB3 Net Class
Target U33
1mm tolerance

(Add USB in generated names to help differentiate)

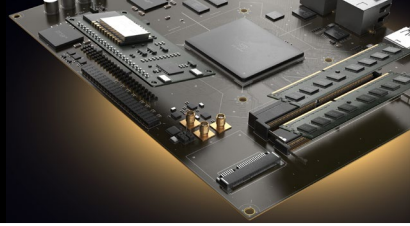








AltiumLive 2018
DEMO



AltiumLive 2018
Questions?