

**Altium**

QUICK GUIDE

# Properties Panel Routing

Via Hole Size  
0.2mm

Via Diameter  
0.4mm

Width  
0.6mm

GP4

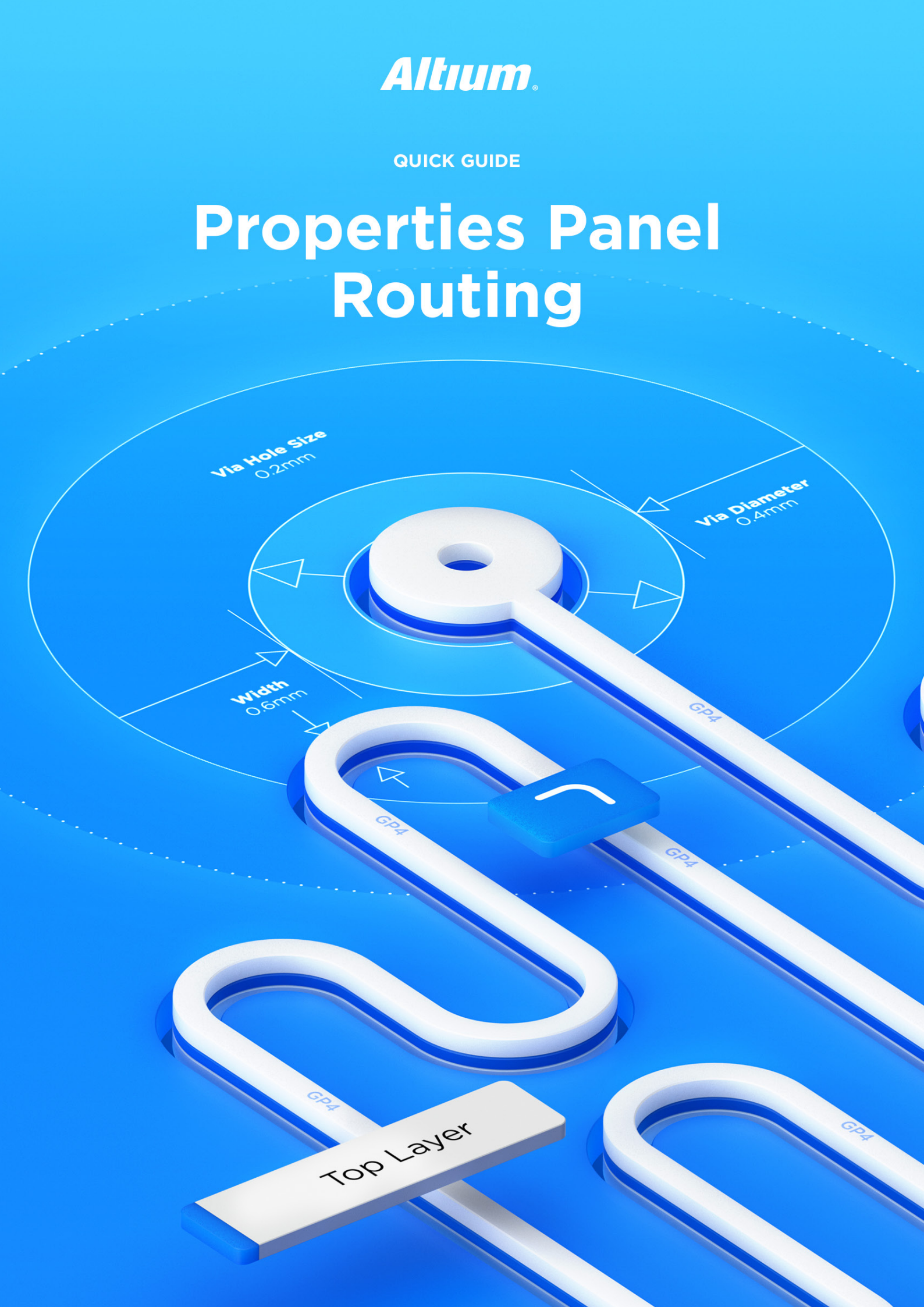
GP4

GP4

GP4

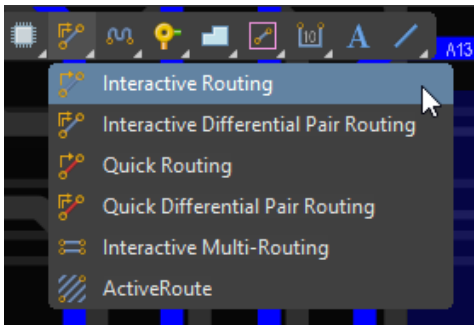
Top Layer

GP4

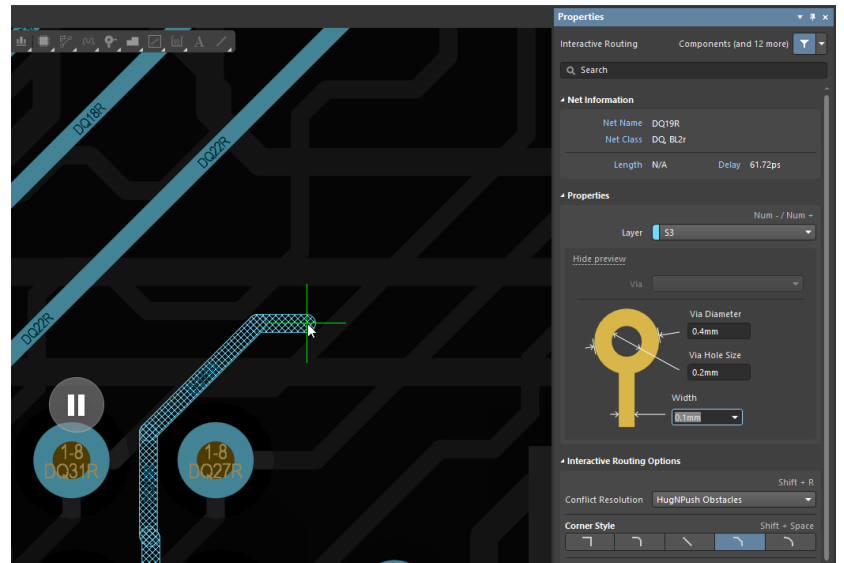


## Interactive routing

The basis of PCB topology development in Altium Designer. By offering powerful routing tools, the system provides comprehensive settings. All groups of settings are arranged in accordance with the logic of the tool, as well as have context-dependent representation.



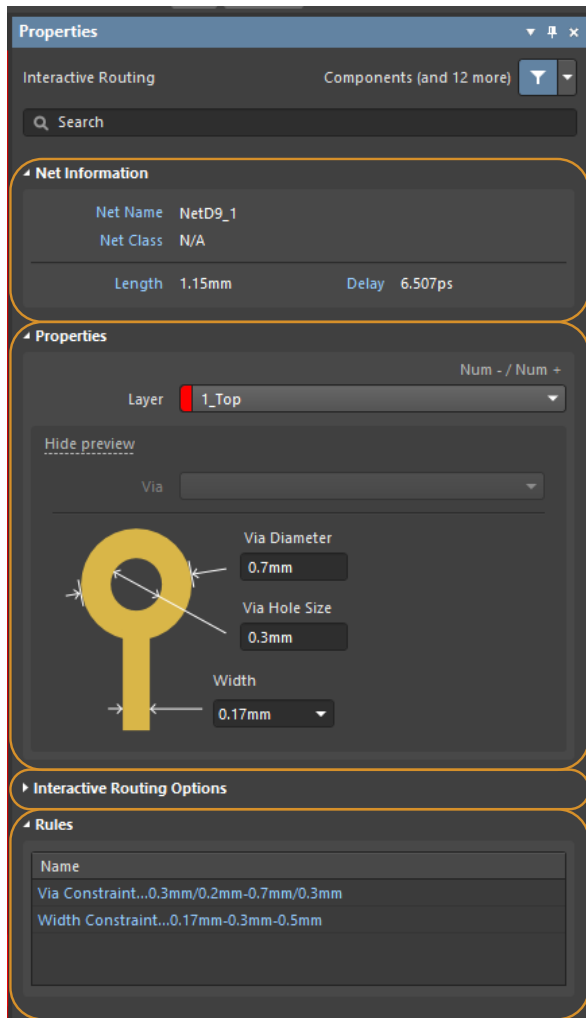
Run interactive routing in the Active Bar



Properties panel becomes active when you press Tab

# Properties panel overview

The properties panel automatically changes when a user activates the Interactive Trace feature to display the appropriate settings. The panel consists of several sections:



**Net Information:**

It will be different for **Single Track** and **Diff Pair**. It is not displayed in **Multi-Routing mode**

**Properties**

Displays parameters of conductors, vias and clearances between tracks. It has three modes for **Single Track**, **Diff Pair**, and **Multi-Routing**

**Interactive Routing Options:**

Displays settings of tracks routing behavior. **Different for Single Track, Diff Pair, and Multi-Routing**

**Rules:**

Quick access to the rules that control current routing

# “Net information” section

## Single-track routing

In the **Net Information** section for **single-track routing**, the following parameters are displayed:

---

**Net Name**      The name of current net

---

**Net Class**      The name of class (or classes) that includes this net

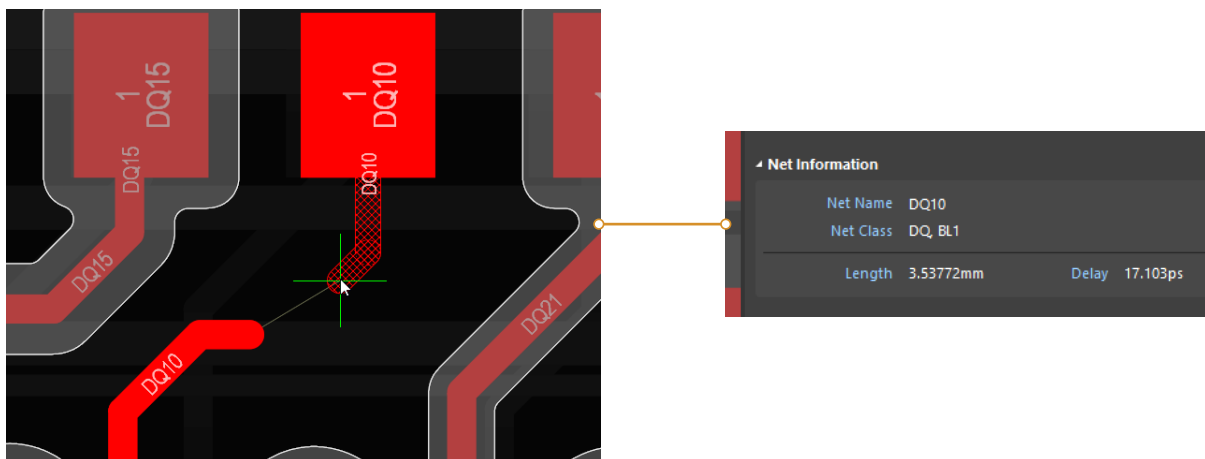
---

**Length**      The length of routing tracks for a given net

---

**Delay**      Means delay in conductors and via

---

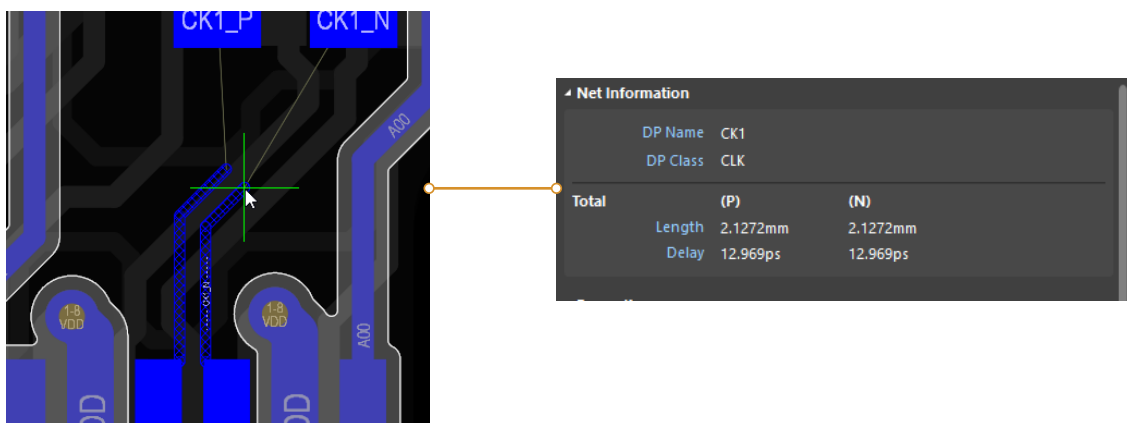


# “Net information” section

## Diff pair routing

For **Diff Pair** routing in the **Net Information** section a user can see the following parameters:

DP Name	The name of current differential pair
DP Class	The name of class (or classes) that includes that pair
Length	The length of positive ( <b>P</b> ) and negative ( <b>N</b> ) tracks of this pair
Delay	The delay in conductors and vias of positive ( <b>P</b> ) and negative ( <b>N</b> ) tracks for a given pair



# “Properties” section

## Single-track routing

The image shows a PCB layout with blue tracks and vias. A yellow via is highlighted in the center. To its right is the 'Properties' panel for the 'BOTTOM' layer, showing 'Via Diameter' (0.4mm), 'Via Hole Size' (0.2mm), and a 'Width' dropdown menu with options 0.09mm (Min), 0.17mm (Pref), and 0.17mm (Max). To the right of the Properties panel is the 'PCB Rules and Constraints Editor [mm]', showing a rule named 'RoutingVias' with a dropdown menu set to 'Template preferred'. Orange lines connect the text annotations to the corresponding UI elements.

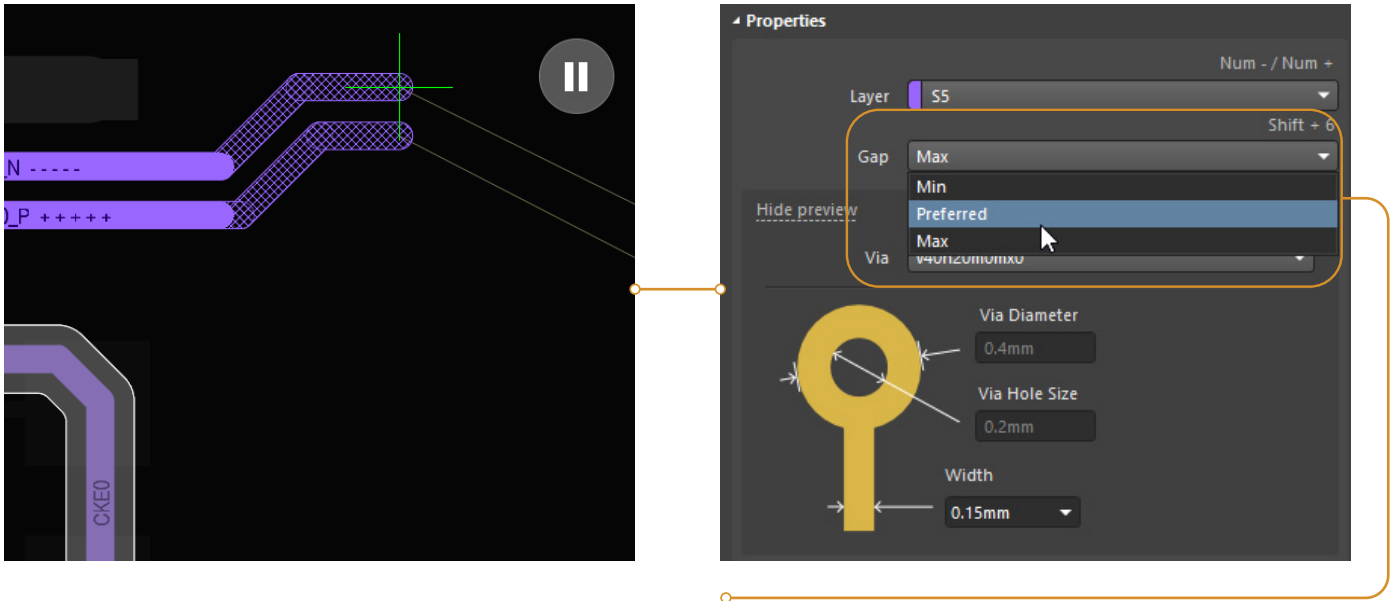
To select via pattern, activate the appropriate mode in the **Routing Via Style** rules

Set via parameters

Enter the desired track width or select one value from drop-down list according to the rules

# “Properties” section

## Diff pair routing

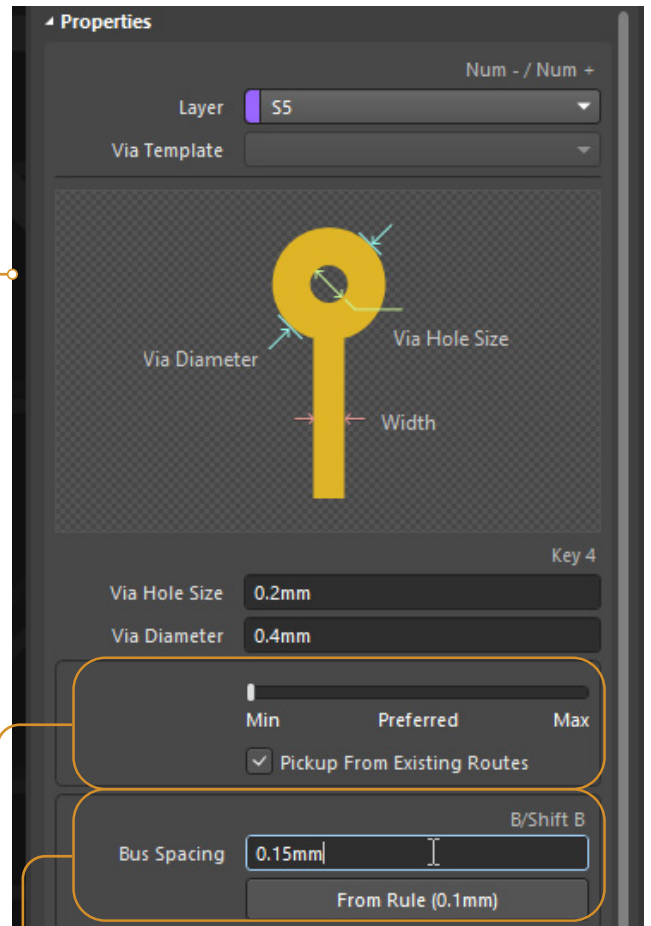
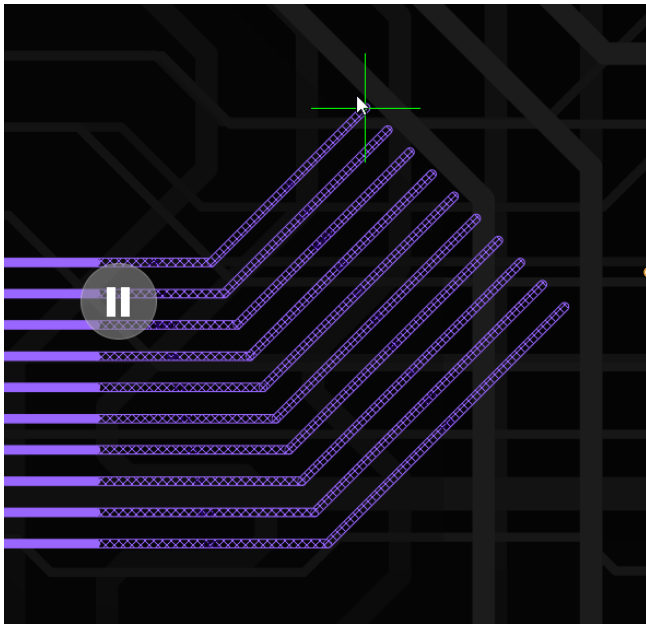


Additional **Gap** drop-down list appears in the section (the gap between conductors of differential pair) when routing **Diff Pair**.

**Min - Preferred - Max** value is taken from the corresponding **Differential Pairs Routing rule**.

# “Properties” section

## Multi-routing



The track width is adjusted with the **min-preferred-max** slider, and values are taken from the design rules for each track individually. Activate the checkbox to use the widths from already existing tracks.

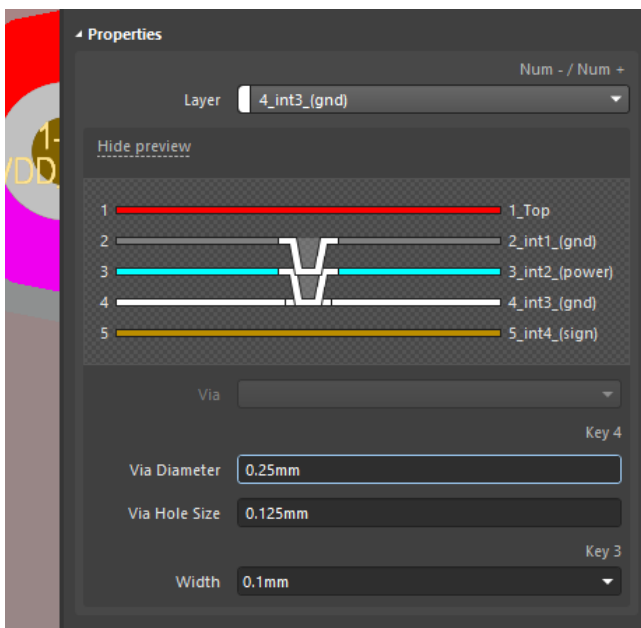
Enter the desired bus spacing between tracks or click **From Rule** to extract the minimum allowable value from the **Clearance** rule



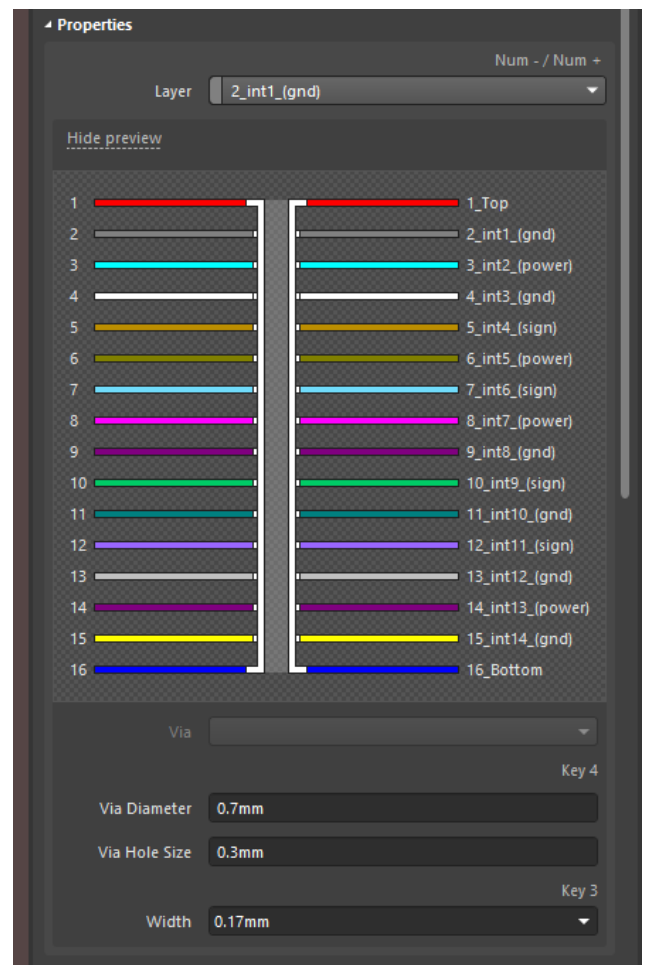
# “Properties” section

## Layer change

When you change layer, the **Properties** section will display detailed information about configuration of current via

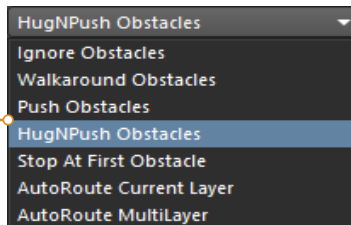
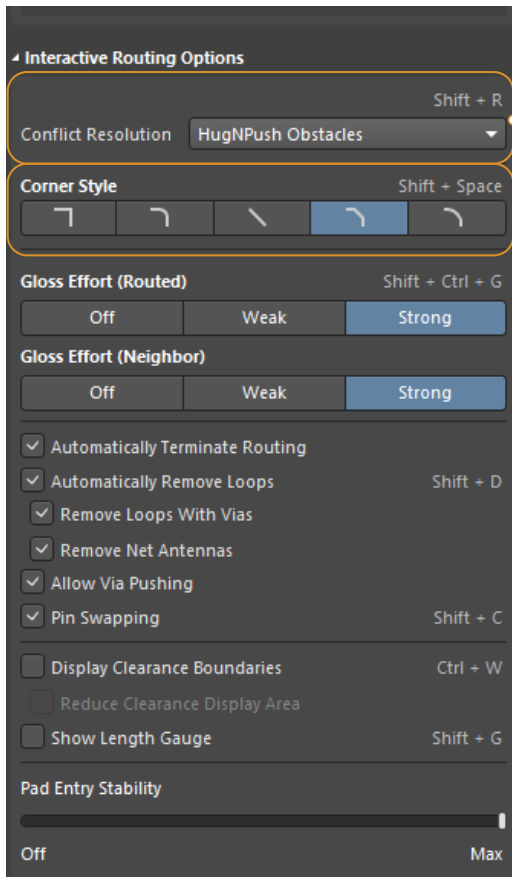


Micro via



Through via

# “Interactive routing options” section

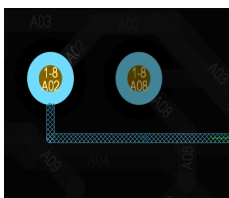


Conflict Resolution mode defines behavior of the routing tool when interacting with already existing topology primitives of other circuits. For example, in **Push** obstacles mode, the routed conductor will push tracks and vias of other circuits if necessary. And **Ignore** mode provides routing even if the rules are violated.

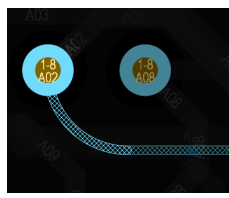
Corner Style (modes are switched by Shift+Spacebar combination)

- Track 90
- Line 90/90 Vertical Start with Arc
- Any Angle
- Track 45
- Line 45/90 with Arc

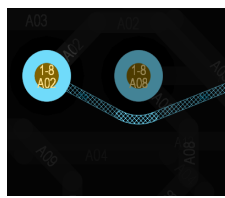
In modes with Arc, the rounding radius is set with the «,» and «.» hotkeys.



Track 90



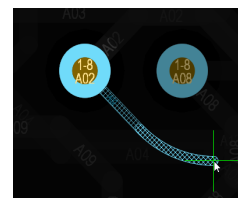
Line 90/90 Vertical Start with Arc



Any Angle

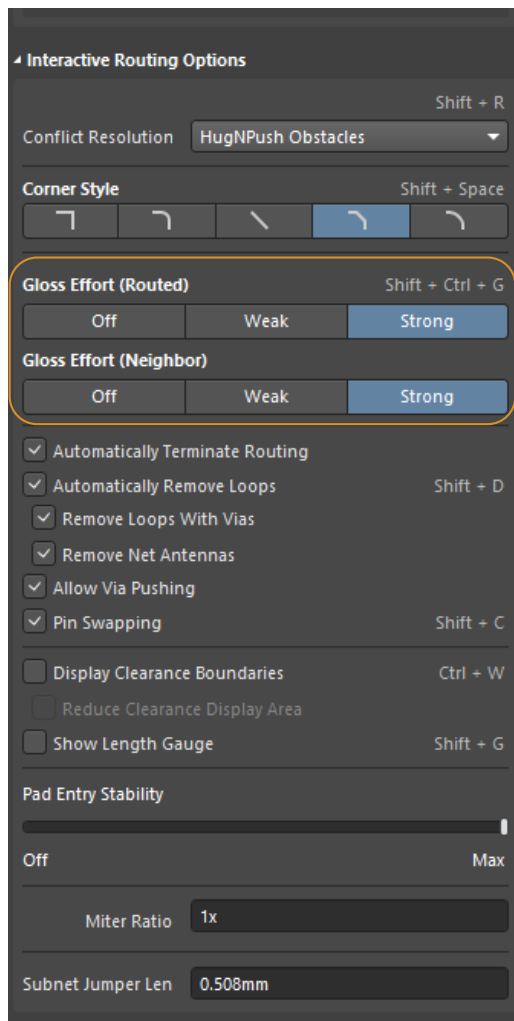


Track 45



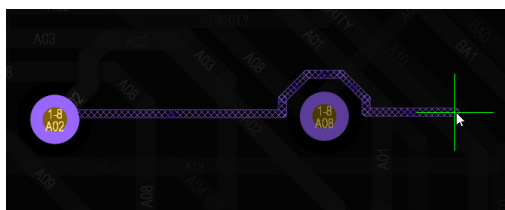
Line 45/90 with Arc

# “Interactive routing options” section

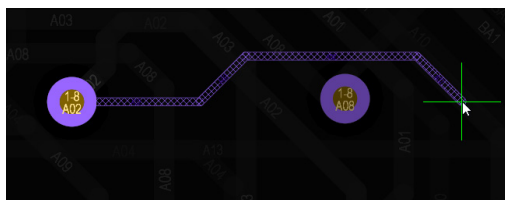


The **Gloss Effort** settings affect the level of changing shape of tracks (routed and already existing) when walkaround obstacles and pushing primitives

## ROUTED

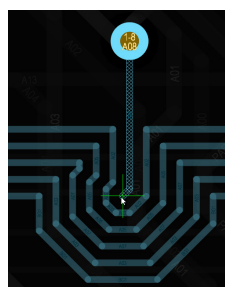


Off

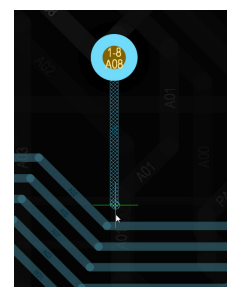


Strong

## NEIGHBOR

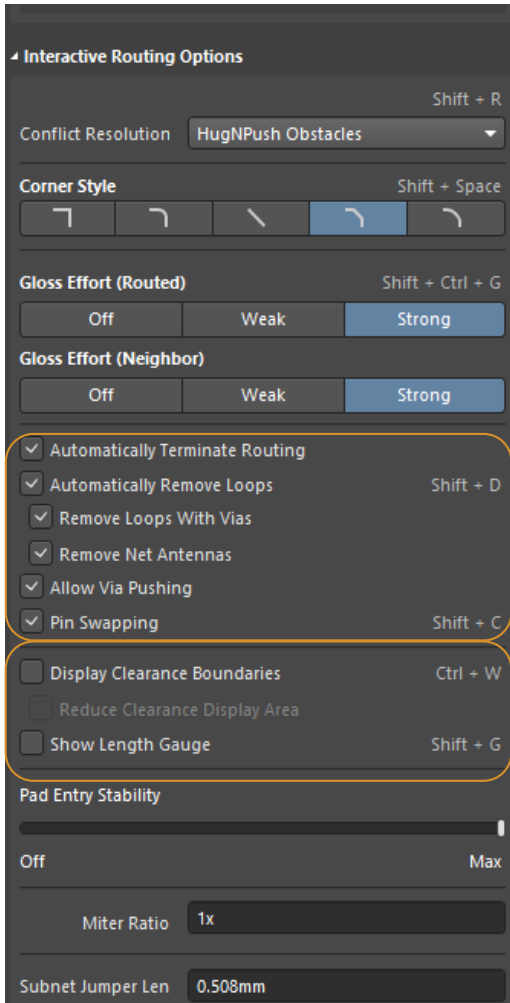


Off



Strong

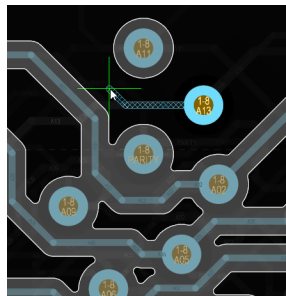
# “Interactive routing options” section



The set of checkboxes defines a variety of routing options:

- terminating track when it reaches pad;
- automatic removal of closed loops (separately - including vias) and antennas;
- activation of via pushing;
- enabling **pin swapping** (in the interactive mode)

## CLEARANCE BOUNDARIES



Full

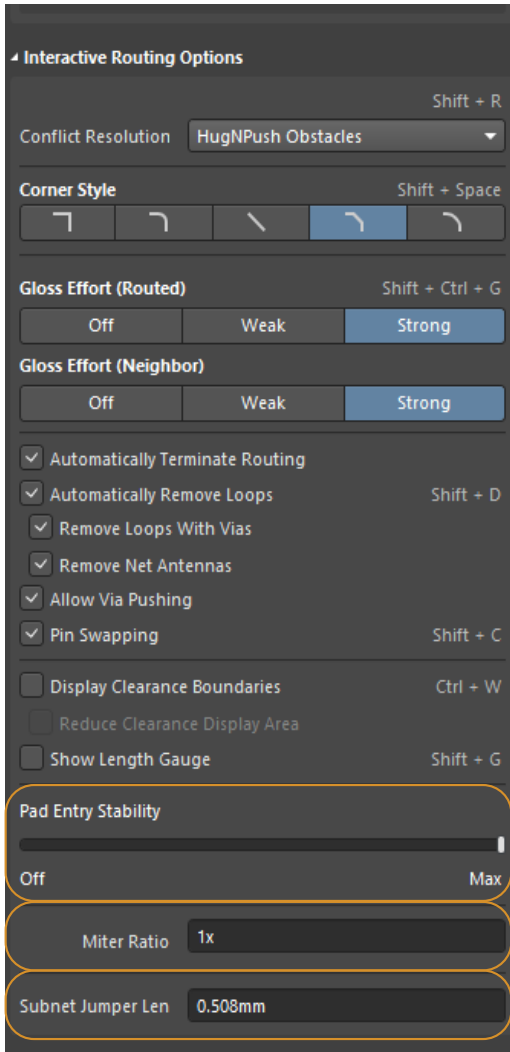


Reduced

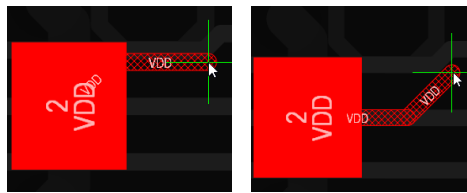


Length Gauge

# “Interactive routing options” section



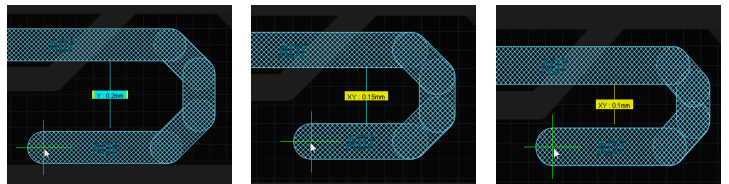
The **Pad Entry Stability** slider sets the degree of protection for pad centered tracks against misalignment. The closer the slider to the **Max** value, the stronger the protection. **Off** function is disabled.



OFF

MAX

Minimum **U-shaped** track width of 0.1mm at various **Miter Ratio**:

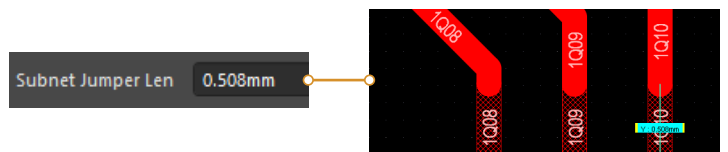


Miter Ratio = 2

Miter Ratio = 1.5

Miter Ratio = 1

**Subnet Jumper Len** sets the jumper length when using interactive **Pin Swapping** (appropriate checkbox must be enabled)



# “Rules” section

The last (bottom) section of panel is the **Rules**

This section displays the current rules which are active during the routing.

Clicking on a rule takes you to the window where you can view or edit it

