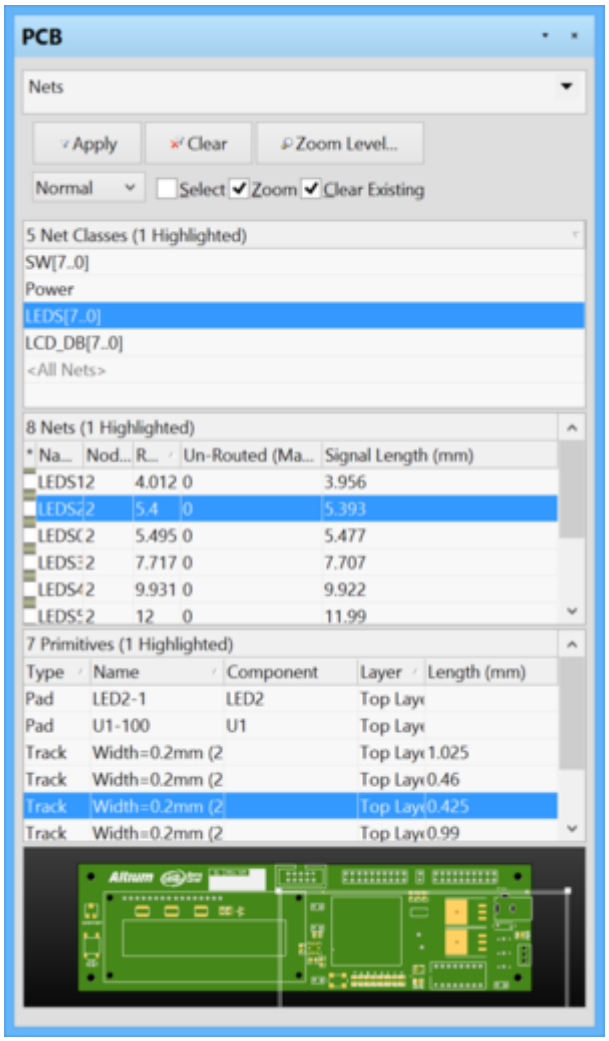
Published on *Online Documentation for Altium Products* (https://www.altium.com/documentation)

Using Altium Documentation
Modified by Phil Loughhead on Oct 24, 2018

Parent page: PCB Panels



Nets mode of the *PCB* panel.

Summary

The *PCB* panel allows you to browse the current PCB design using a range of filter modes to determine which object types or design elements are listed, highlighted or selected. The panel also has editing modes for specific object types or design elements that provide dedicated controls for editing procedures. Note that you can access the properties for any element listed in the panel.

In **Nets** mode, the two main list regions of the panel will change to reflect an object hierarchy, in order from the top:

1. Net Classes, as defined by the board.

2. Individual member Nets within a class.
3. Individual items within a Net (pads, vias, tracks and fills).

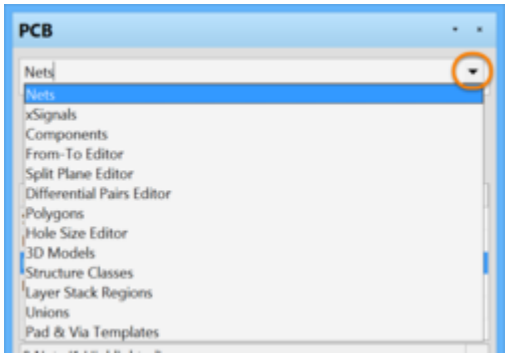
> While the middle sections change between modes, the top and bottom portions of the *PCB* panel remain the same. For a panel overview and information on those sections, please see the *PCB* panel page.

Panel Access

When the PCB Editor is active, click the PCB button at the bottom-right corner of the workspace and select **PCB** from the context menu. Alternatively, you can access the panel through the **View » Workspace Panels » PCB » PCB** sub-menu.

> Panels can be configured to be floating in the editor space or docked to sides of the screen. If the *PCB* panel is currently in a group of panels, use the **PCB** tab located at the bottom of the panels to bring it to the front.

Once the *PCB* panel has been opened, select the **Nets** option from the dropdown menu at the top of the *PCB* panel to enter **Nets** mode.

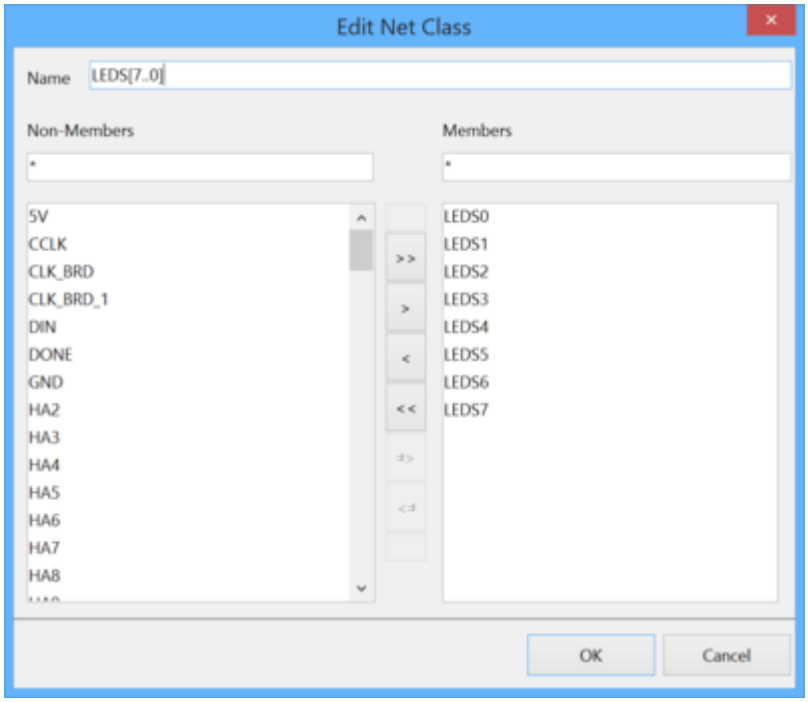


Browsing Nets

Net Classes

In this top region of the panel, right-clicking on a net or net item entry and choosing the **Properties** command from the subsequent pop-up menu (or double-clicking on the entry directly) will provide access to the relevant properties dialog, from which you can view or modify the properties of the net/item as required.

Right-clicking on a net class entry and choosing **Properties** from the menu (or double-clicking on the entry directly) opens the *Edit Net Class* dialog for that class. From this dialog you can view/modify the net membership of the class, rename it, or add additional classes.



Editing a selected net class.

Nets

The middle region of the panel displays nets from a selected Net Class(es) in the above region.

5 Nets (0 Highlighted)

*	Name	Node Count	Routed (mm)	Un-Routed (Manhattan) (mm)	Signal Length (mm)
	AGND	7	18.677	Net is Hidden	0
	2V5	30	101	0	0
	1V2	58	134.685	0	0
	5V0	13	166.013	0	0
	3V3	184	412.55	Net is Hidden	0

The following information is listed with each Net by default:

- **Node Count** - This value reflects the total number of pads and vias that are connected to that net.
- **Routed Length** - The distance traveled if you follow the centerline of the objects, including overlaps or routing wiggles inside pads.
- **Un-Routed (Manhattan) Length** - This length is the vertical or horizontal X+Y distance from here to there.
- **Signal Length** - The distance traveled using the new length calculation engine, which detects and excludes overlaps and wiggles inside pads. It also copes with routing paths that use objects other than tracks or acs (for example, a region or fill). Signal Length also includes the length of any section still to be routed. For the un-routed section, it uses the Manhattan Length.

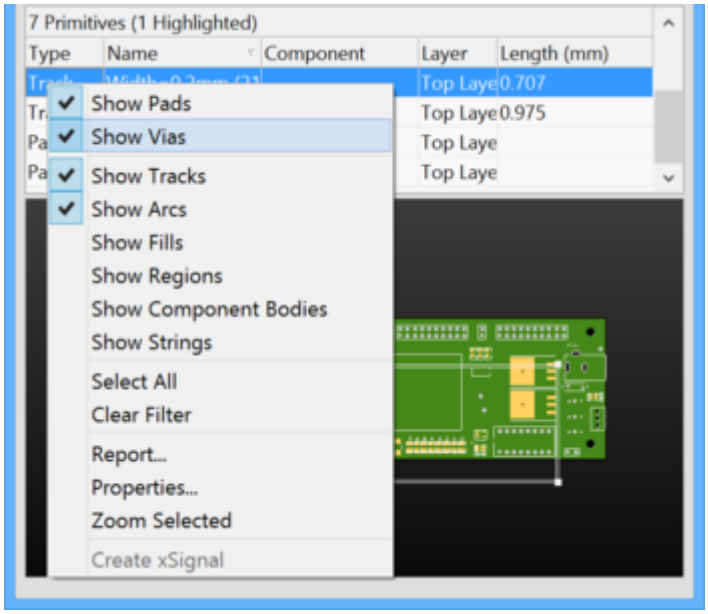
Right-click and navigate through the Columns sub-menu to add the following options:

- **Min/Max** - This distance takes the settings from the applicable Length rule. If there is no applicable rule defined, the internal defaults of `Min=0mil` and `Max=99999mil` are used.
- **Estimated Length** - This distance is the Routed Length plus length of connection lines for any section still to be routed. Estimated Length does not use the Manhattan length for the unrouted portion, it instead uses the direct point to point distance.

The option to change a net's color is also available from the right-click menu in this section by selecting **Change Net Color** to open the *Choose Color* dialog.

Primitives

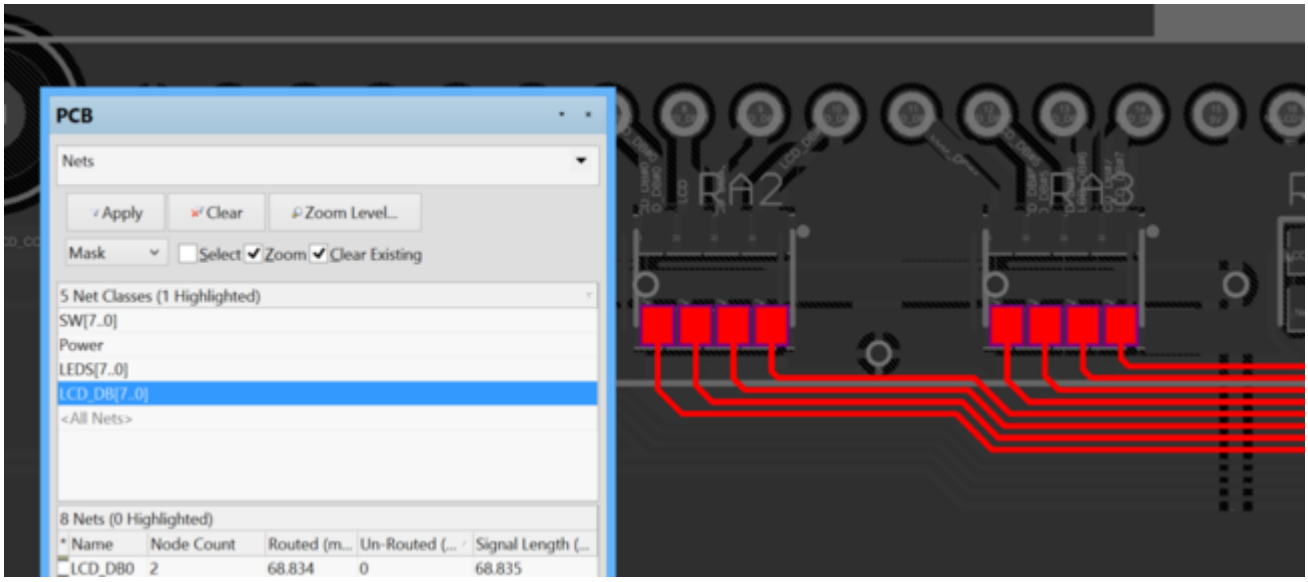
The display/inclusion of each net item type, in the bottom, **Primitives** region of the panel, is dependent on whether the corresponding option for each has been enabled on the right-click menu, accessed from either the **Nets** or **Primitives** regions.



Right-click on a net or net item entry to select the included items.

Note that the Primitive list right-click context menu also offers the option to create an xSignal between two selected items – See the PCB - xSignals panel page for more information.

In summary, as you click on an entry in the panel's list, a filter will be applied based on that entry. The visual result of the selection (in the design editor window) is determined by the highlighting methods enabled (Mask/Dim/Normal, Select, Zoom). Multiple entries can be selected in each region, using standard **Shift**+**Click** and **Ctrl**+**Click** features.



A net class selected with the visual mode set to Mask.

Source URL: https://www.altium.com/documentation/display/ADES/PCB_Pnl-PCB((PCB+-+Nets))_AD