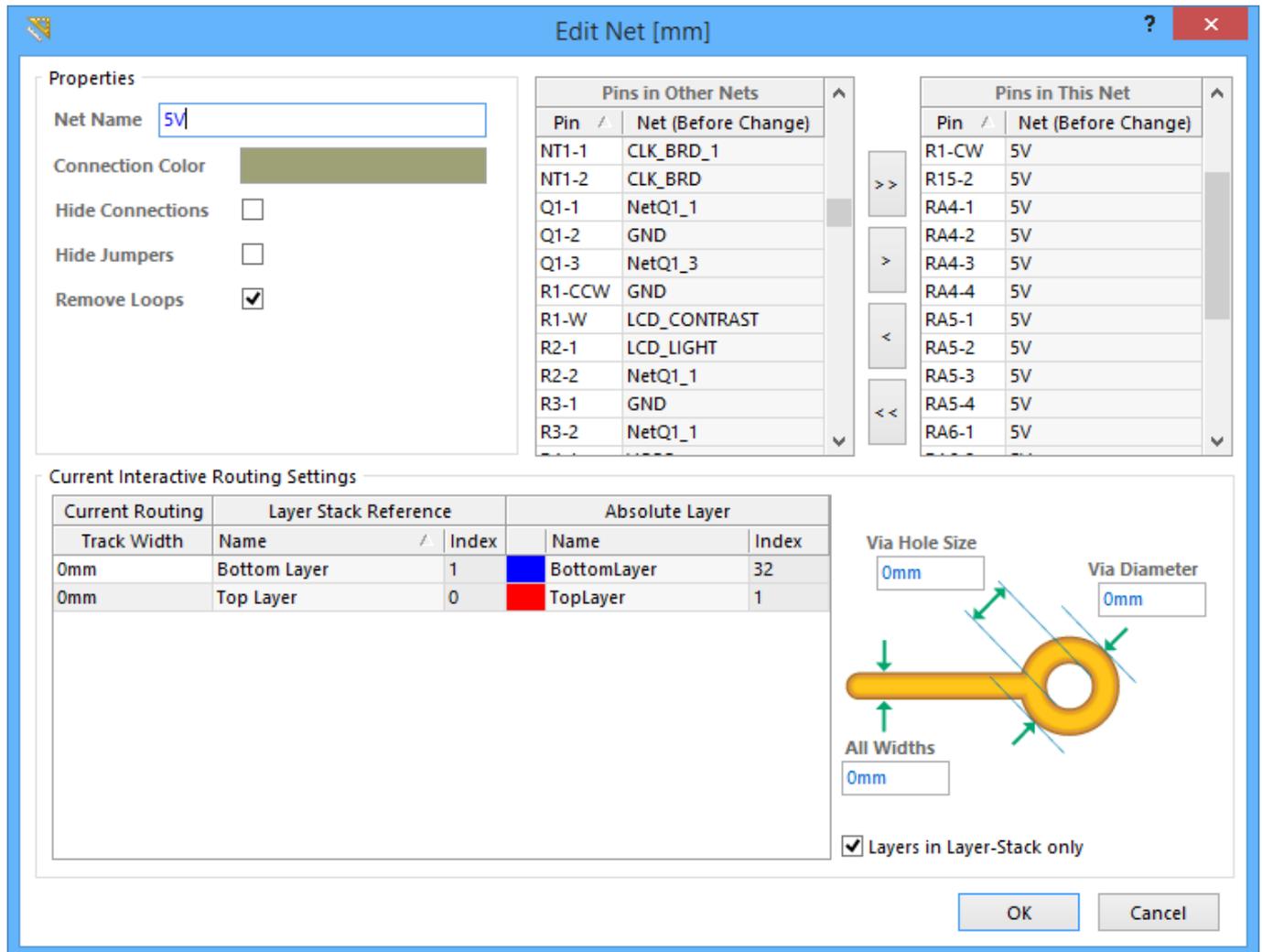


Using Altium Documentation

Modified by Phil Loughhead on Jun 16, 2017



The *Edit Net* dialog.

Summary

This dialog allows designers to edit nets, including changing the net name, adding or removing physical pins for the specified net, and specifying track length for the net.

Access

The dialog can be accessed in the following ways:

- From the *PCB* panel, select **Nets** view. In the **Nets** section, double-click on a net to access this dialog.
- Right-click over a placed design object and select **Net Actions** » **Properties** from the context menu.

Options/Controls

Properties

- **Net Name** - Rename the net if necessary.
- **Connection Color** - Choose a connection color for the specified net.
- **Hide Connections** - Enable to hide connection wires.
- **Hide Jumpers** - Enable to to hide jumpers, or short connections between routed segments.
- **Remove Loops** - Enable to automatically remove any redundant loops that are part of this net.

Pins in Other Nets

This is a list of all the pins on the PCB. Pins that are currently assigned to a net include their net name in brackets. Select the pins you wish to add to the net being edited. Use the **Shift** and **Ctrl** keys to select multiple pins. Use the control buttons to move selected pins into the current net list or right-click to remove selected pins from the list using the context menu.

Pins in This Net

Lists all pins in this net. Select the pins you wish to remove from this net. Use the **Shift** and **Ctrl** keys to select multiple pins. Use the control buttons to move selected pins out of the current net list or right-click and use the context menu to remove selected pins.

Buttons

- **>>** - Add all **Pins in Other Nets** to **Pins in This Net**.
- **>** - Add the selected **Pins in Other Nets** to **Pins in This Net**. Use the **Shift** and **Ctrl** keys to build up a selection.
- **<<** - Move all **Pins in This Net** to **Pins in Other Nets**.
- **<** - Move the selected **Pins in This Net** to **Pins in Other Nets**. Use the **Shift** and **Ctrl** keys to build up a selection.

Current Interactive Routing Settings

Lists any current settings for interactive routing.

- **Track Width** - The track width field is editable and can be changed to your preference and/or design requirements.
- **Via Hole Size** - This via hole size represents the current via hole size's user choice value which is stored in the net. This dialog provides a way to modify the current values for the current interactive routing settings. If the values are zero, the user choice values are not being sourced from this dialog and the last used value for this board will be used.
- **Via Diameter** - This via size represents the current via size's user choice value which is stored in this net. This dialog provides a way to modify the current values for the current interactive routing settings. If the values are zero, the user choice values are not being sourced from this dialog and the last used value for this board will be used.
- **All Widths** - This is a list of current routing and their layer references values that represent the current user choice values that are stored in the net. It provides a way to modify the current values for the current interactive routing settings. If the values are zero, the user choice values are not being sourced from this dialog and the last used value for this board will be used.
- **Layers in Layer-Stack only** - Enable this to apply the via parameters for layers in the layer-stack only.

Source URL: [https://www.altium.com/documentation/display/ADES/PCB_Dlg-EditNet\(\(Edit+Net\)\)_AD](https://www.altium.com/documentation/display/ADES/PCB_Dlg-EditNet((Edit+Net))_AD)