

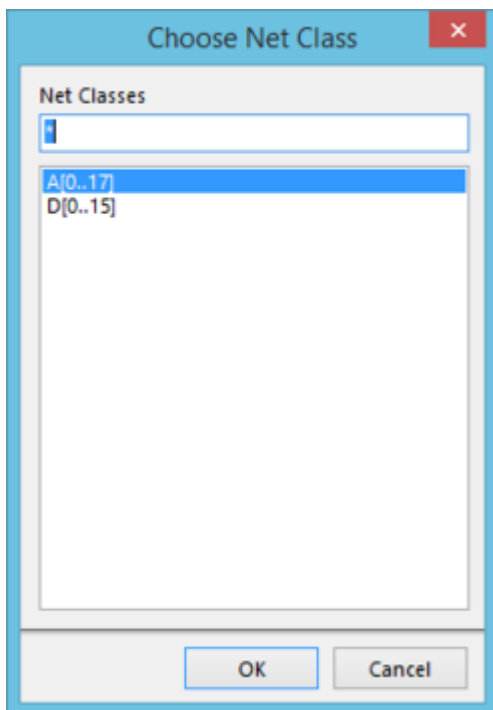
Choose Net Class

Modified by Jason Howie on Oct 21, 2014

Other Related Resources

[Object Class Explorer \(Dialog\)](#)

Parent page: [PCB Dialogs](#)



The *Choose Net Class* dialog.

Summary

This dialog allows the designer to quickly add the associated net(s), for one or more selected net design objects (pad, via, region, fill) in the design workspace, to an existing Net Class - becoming members of that class.

Access

In the PCB Editor, select the design object(s) whose associated net(s) you wish to add to a target net class, then right-click and choose the **Net Actions » Add Selected Net(s) to NetClass** command from the context menu.

Options/Controls

- **Net Class Mask** - this field, at the top of the dialog, allows you to enter a string-based search

mask, to quickly target the net class required. This is particularly useful if there are many net classes defined for the design. As you type, the list of net classes below is filtered to only show matching strings. You can use the wildcard characters; ? (any single character) and * (any characters).



Clear the mask, or enter the * wildcard to list all net classes defined for the board.

- **Net Class List** - this area lists all net classes currently defined for the board (or all net classes passing the current mask), by class name. Select the entry of the net class you wish to add the net(s) to.

The result of the action can be verified using the *Object Class Explorer* dialog. The affected net(s) should be visible as members of the targeted net class.

Source URL:

[http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-ChooseNetClassDlg\(\(Choose+Net+Class\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-ChooseNetClassDlg((Choose+Net+Class))_CM)