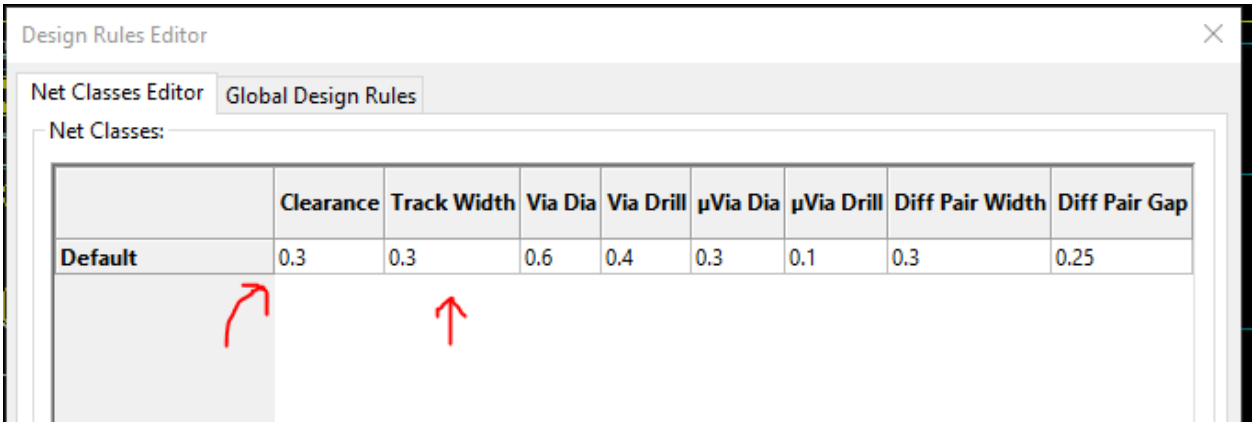
Note : This guide is for Kicad 5.0.1. Previous release version may slightly differ.

Design Rule Editor

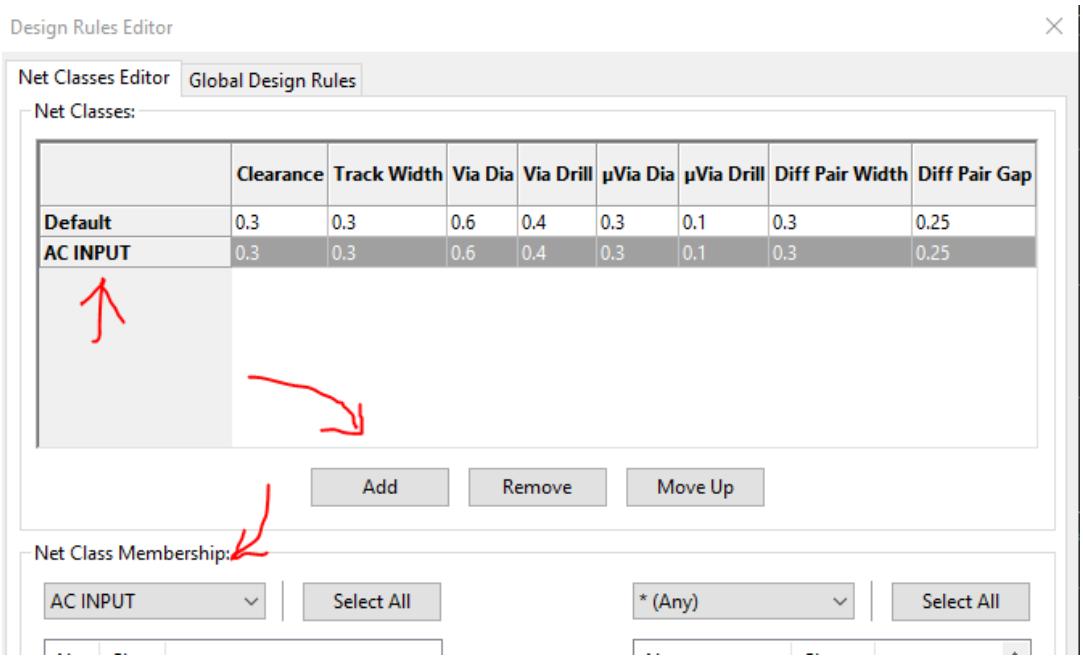
Setting your design perimeter based of limitation set by your fabricator. Any perimeter set here will be used as a guide for your Design Rule Check (DRC) later.

Click **Setup > Design Rule** and set your perimeter based on Notes provided. Ie Clearance, track width etc.

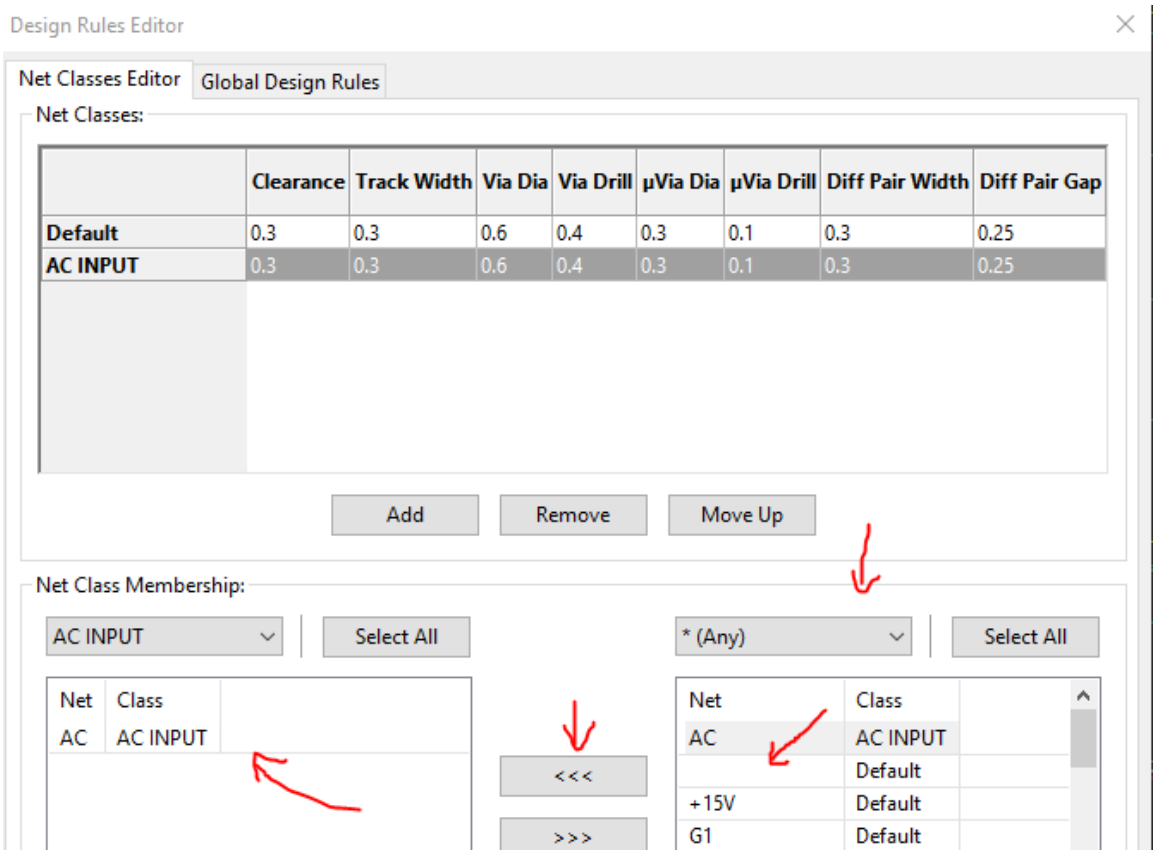


If you have a specific line rails that you wanted to set its own perimeter, you can set it here through setting your Net Classes. Normally, this is set for tracks that handles high current.

To create your Net Classes, inside the **Design Rule Editor** window, click Add and after you named it it will appear on the Net Classes Editor. Under Net Class Membership, find your newly named net class.



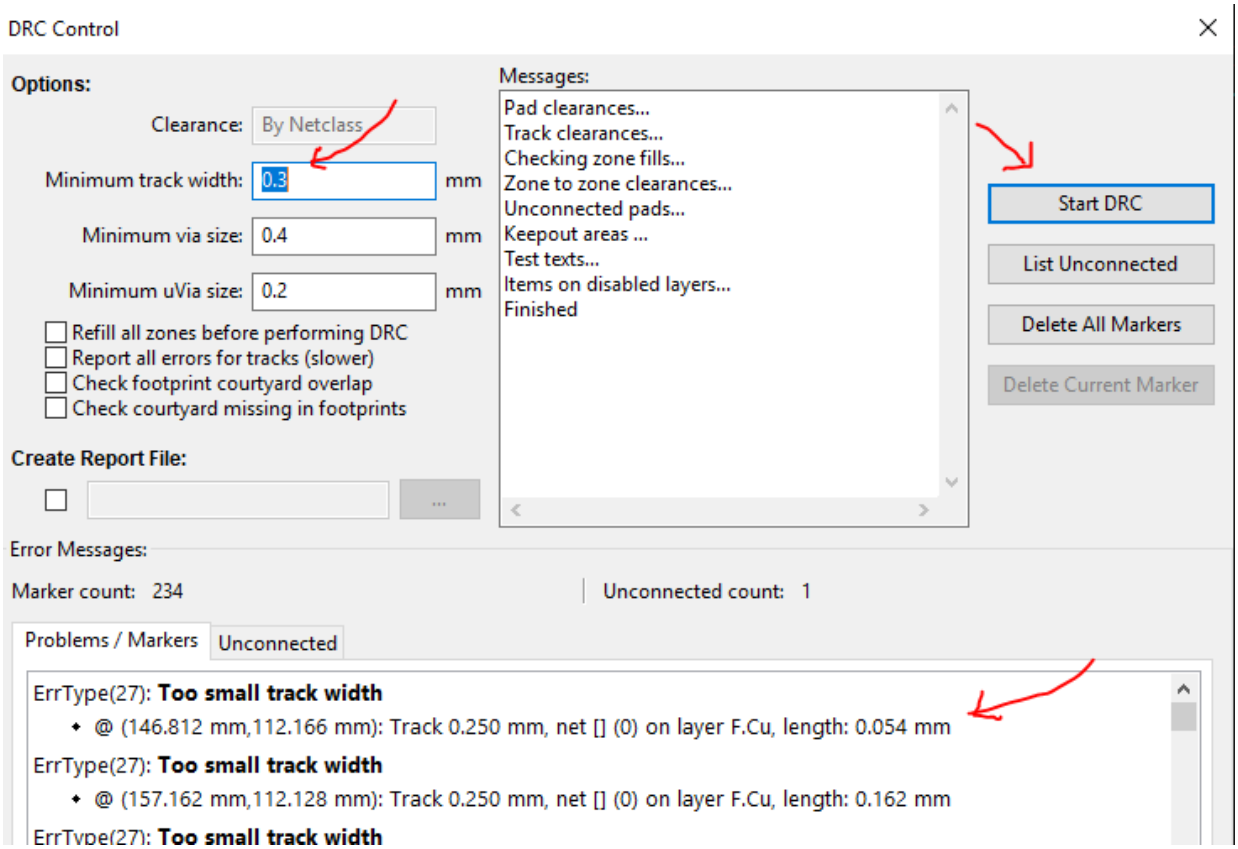
Find the rails that you wanted to set and then click **the arrow button** to add it to your net class. Click **OK** when you finised editing.



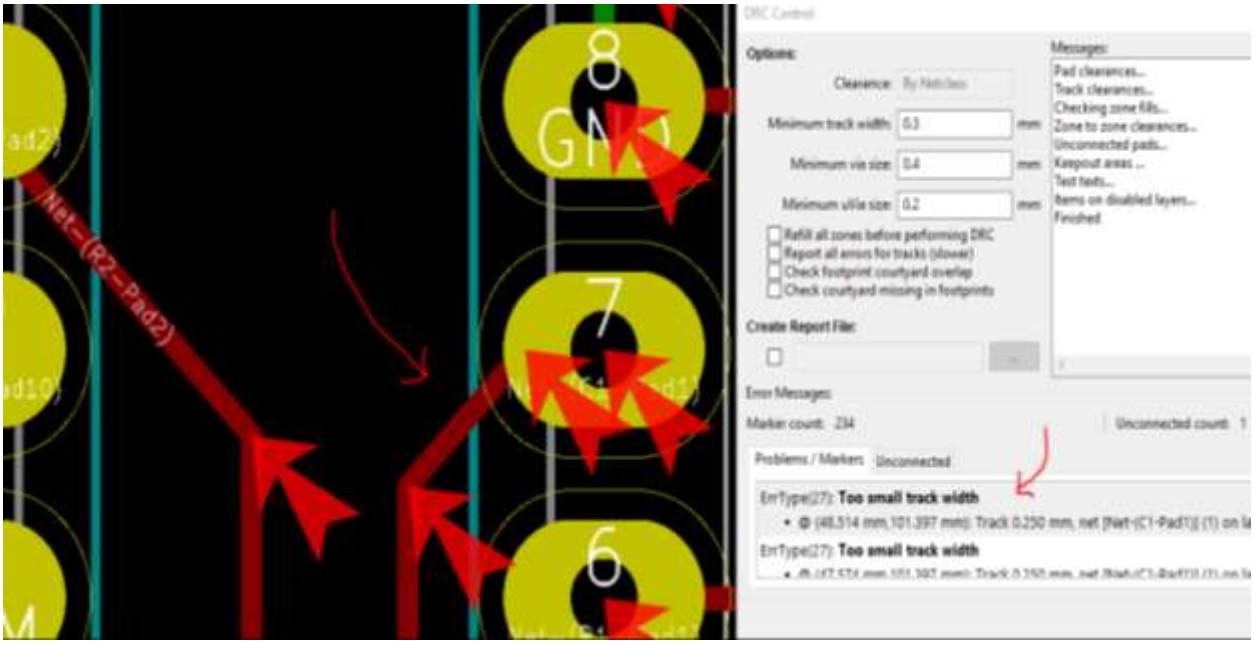
Design Rule Checker (DRC)

This will help you highlight and locate any error that you made on your design. This is set based on the perimeters that you set on Design Rule Editor.

Click Inspect > Design Rule Checker. Inside the DRC window, set your minimum track width and then Click **Start DRC.**



You can locate where your error is by clicking on the error message and the window will automatically moved to it.



For this example, an error reads : Too small track width appear because the designer did not set the proper perimeter on Design Rule Editor and then drawing a track width not according to the fabricator limitation.

You can also check for any unconnected Netlist by clicking on the **Unconnected** tab.