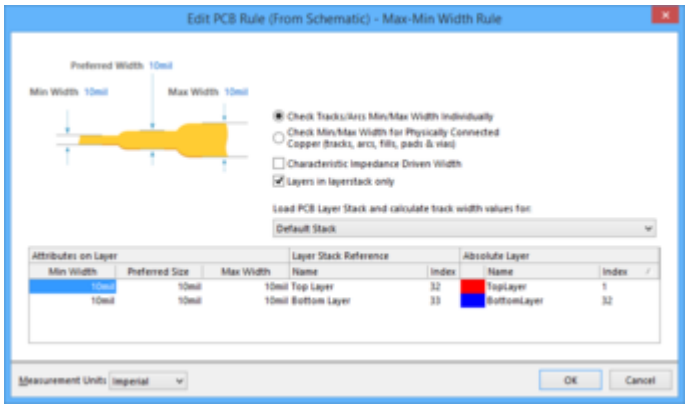
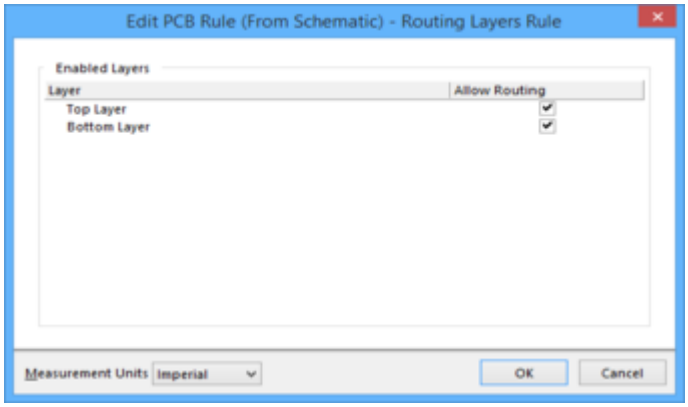
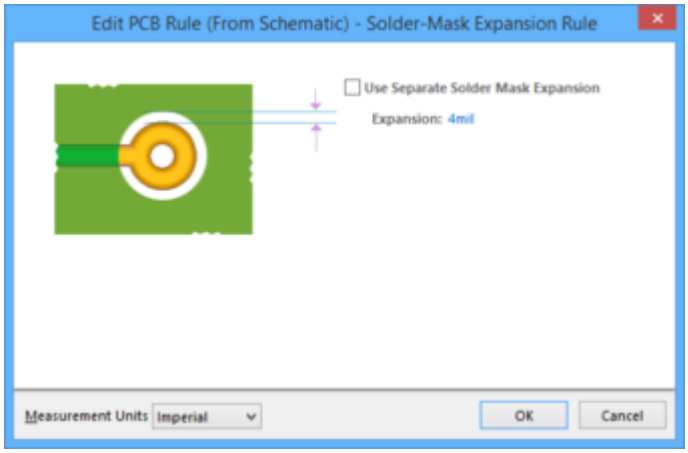
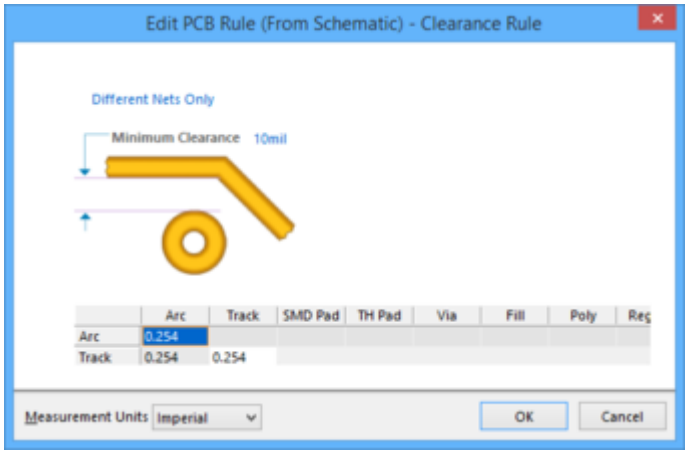
Using Altium Documentation
Modified by Phil Loughhead on Jun 16, 2017









A few variations of the *Edit PCB Rule (From Schematic)* dialog: Max-Min Width, Routing Layers, Solder-Mask Expansion, and Clearance Rules.

Summary

The *Edit PCB Rule (From Schematic)* dialog allows you to define the constraints for a PCB rule from a schematic document.

Access

The dialog is accessed by completing the following steps:

1. In a schematic, click **Place » Directives » PCB Layout**.
2. Place the directive where desired on the schematic using normal placement techniques.
3. Double-click the placed directive, then double-click the desired rule in the resulting *Parameters* dialog.
4. Click the **Edit Rule Values** button in the resulting *Parameter Properties* dialog, then double-click the desired rule type in the resulting *Choose Design Rule Type* dialog.

Options/Controls

> The specific controls of this dialog are dependent upon the selected design rule. For detailed information regarding PCB Design Rules and constraints, click here.

Image Region (If Displayed)

- Click on the available controls in the image (editable measurements are in blue text) to edit the desired areas. If a table also is displayed in the dialog, the table will automatically update after making changes in the image.
- Select the desired options and use the associated drop-downs adjacent to the image to set the desired contraint(s).

Table Region (If Displayed)

- Click on the data you need to edit then enter the desired data directly. The image above (if displayed) will automatically update.

Additional Controls

- **Measurement Units** - use the drop-down to select **Metric** or **Imperial** units.
