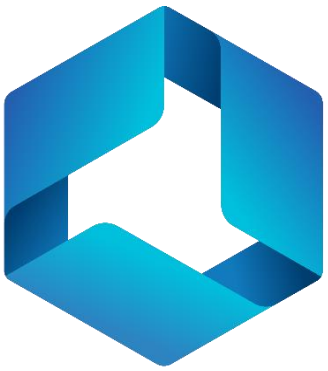

ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 19: PCB Custom Grid Manager

Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.
ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

Module 19: PCB Custom Grid Manager 3

1.1 Purpose 3

1.2 Shortcuts 3

1.3 Preparation 4

1.4 Import Netlist 5

1.5 Predefined Default Grids 5

1.6 Grid Manager 6

1.6.1 Global Board Snap Grid 6

1.6.2 Custom Cartesian Grid 8

1.6.3 Custom Polar Grid 11

Module 19: PCB Custom Grid Manager

1.1 Purpose

In this exercise, the configuration and the functionality for the Cartesian and Polar Grid will be reviewed. Both grids can be used as a local grid within the PCB editor for a specific document.

Designers need to precisely place and align components during PCB design and routing which is extremely difficult without the help of grids systems. Altium Designer supports multiple user-defined grids in both Cartesian and Polar forms, making component placement, routing, … predictable and easy.

1.2 Shortcuts

Shortcuts when working with Grids

G:	Select Snap Grid
CTRL+G:	Open Cartesian Grid Editor dialog
CTRL+Shift+G:	Set Snap Grid
Q:	Change Units (Metric and Imperial)
Ctrl+Q:	Change Units (Metric and Imperial) within a dialog
N » H » A:	To hide all the connections
N » S » A:	To activate all the connections
Spacebar:	Rotate Component counter-clockwise
Shift + Spacebar:	Rotate Component clockwise

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 19 PCB Custom Grid Manager`.
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 19 PCB Custom Grid Manager (Top\Projects\Altium Designer Essentials Training Course\...)`.
5. Select **Open Project as Copy…**

.
6. At the new dialog *Create Project Copy:*
   a) Add your name to the project: `Module 19 PCB Custom Grid Manager – [Your Name]`.
   b) Add a description: `Altium Essential Training – [Your Name]`.
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`.
      ii) Select **OK.**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.4 Import Netlist

8. From the *Projects* panel, open the `Module 19 PCB Custom Grid Manager.PcbDoc`.
9. Go to **Design » Import Changes From [Project Name].PrjPcb** and select **Execute Changes.** The Footprints and unrouted connections are added to the PCB.
10. Close the *Engineering Change Order* dialog once completed.

1.5 Predefined Default Grids

11. Press the key **G** to open a list of predefined grid values.
12. Select one of the predefined values. The visible grid will change in the PCB. The selected value will show in the Status Bar and the Heads-Up Information, as shown in Figure 1 below.
13. Press **Q** to toggle the used Unit between Metric (mm) and Imperial (Mil). You can see the used Unit in the Status Bar and the Heads-Up Information as shown in Figure 1 below. Be aware of changing the Unit does not change the Grid Value.

You can change your units from *mm* to *mil* using the **Q** hotkey for the PCB, or **CTRL+Q** if you're in an action or panel.

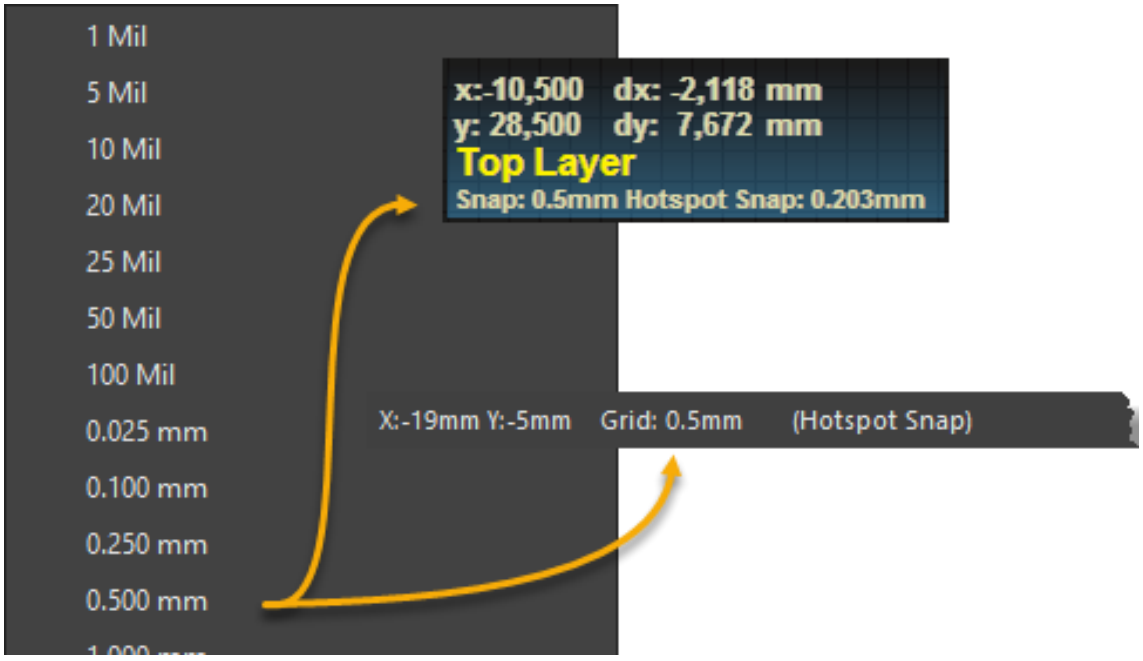

Figure 1. Grid selection dialog

1.6 Grid Manager

Grid manager enables configuration of user defined grids, to suit both personal placement preferences and design requirements.

1.6.1 Global Board Snap Grid

14. Open the *Properties* panel by clicking **View » Panels » Properties.** The grid configuration can be found in the *Grid Manager* section of the panel, as shown inFigure 2 below. You can minimize the other sections of the panel or scroll down to see the relevant information.

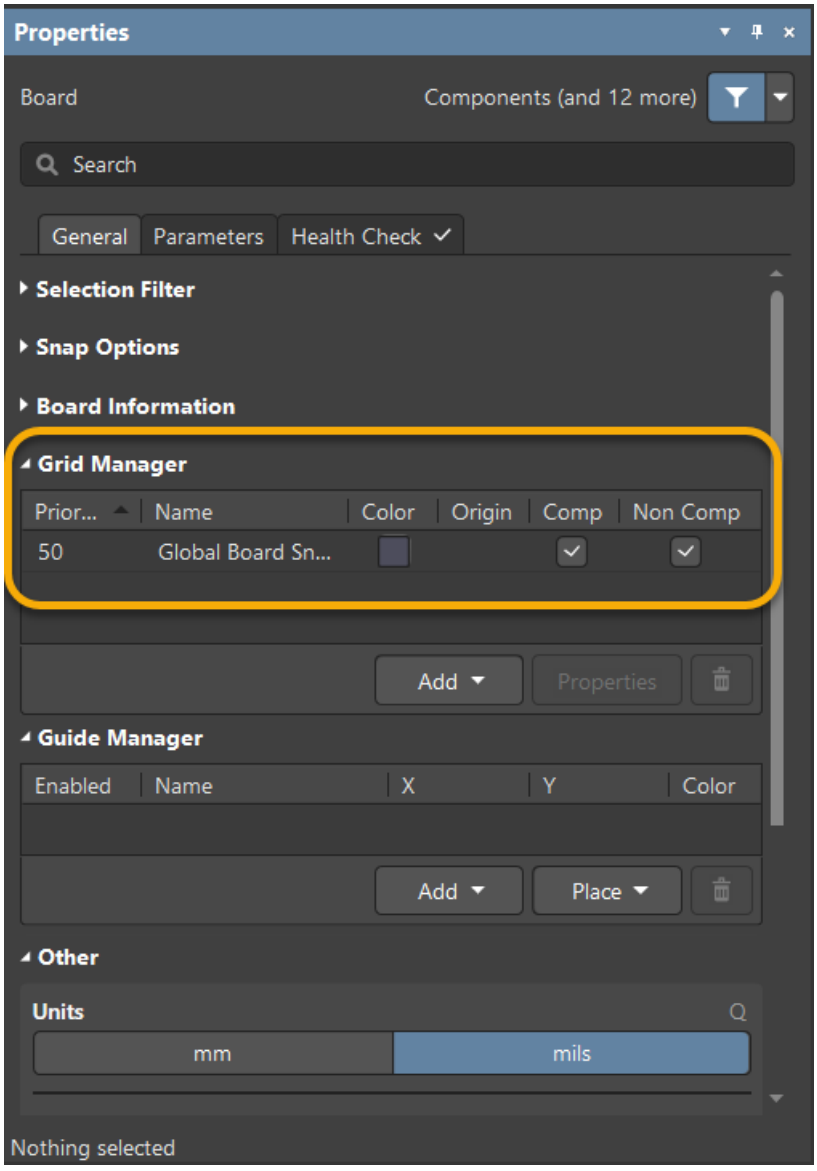

Figure 2. Properties panel with the Grid Manager

15. Double-click on the *Global Board Snap Grid* name in the *Grid Manager* pane to access the default grid as shown in Figure 3 below. You can also access the Cartesian Grid Editor using the **CTRL+G** shortcut keys.

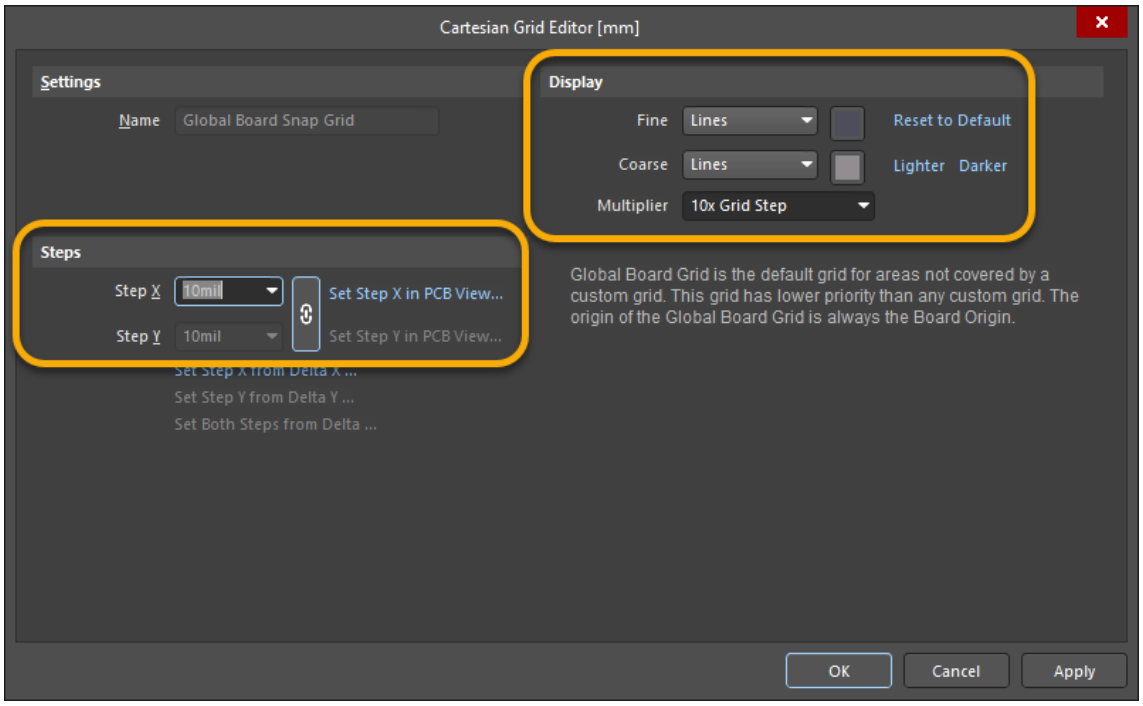

Figure 3. Default Grid Settings

16. Change the *Step X* size to `10mil`, the *Multiplier* to `10x Grid Step` and ensure the *Display* options are set to `Lines`.

The *Multiplier* determines how many *Fine* units will be contained in a *Coarse* grid unit. One *Fine* unit is the value of the *Step* entered in the *Cartesian Grid Editor* dialog.

17. Click **OK** to accept changes.
18. Examine the new grid to see the changes as shown in Figure 4 below. Notice that there are 10 fine grids to each coarse grid and the smallest division is 10mil.

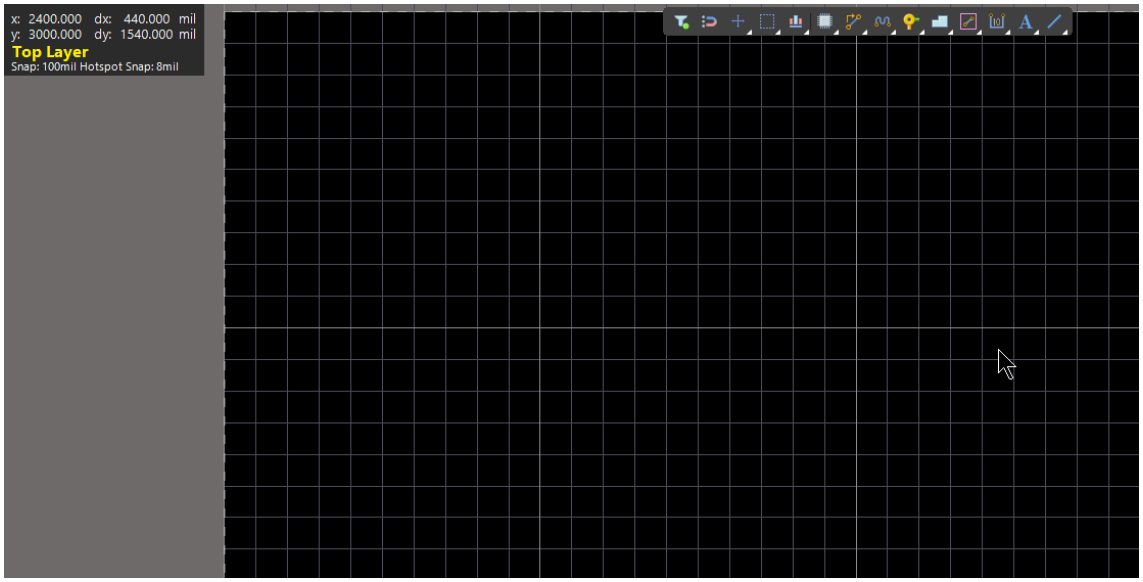

Figure 4. Modified default grid setting

1.6.2 **Custom Cartesian Grid**

19. Open the *Properties* panel.
20. Click the **Add** button in the *Grid Manager* dialog and select **Add Cartesian Grid**.
21. A grid named `New Cartesian Grid` will be added to the list in the *Grid Manager*, as shown in Figure 5 below.

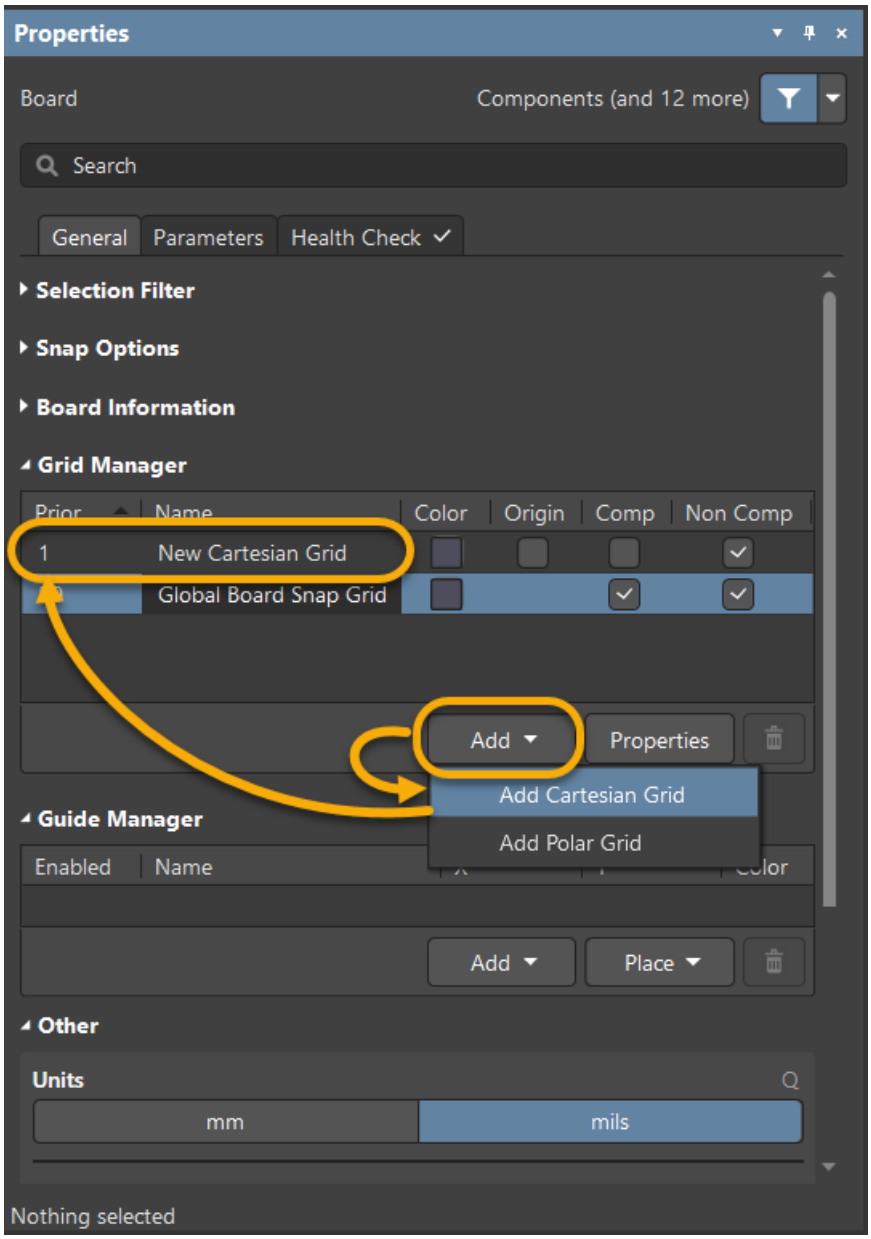

Figure 5. New grid added in the Grid Manager

22. Double click on the *New Cartesian Grid* name to open the *Grid Editor* dialog, Figure 6.
23. Edit the values for each section to match those shown in Figure 6. The *Fine* color is set to `127` while *Coarse* color is set to `123`.
    a) Name: `New Cartesian Grid`
    b) Unit: `Imperial`
    c) Rotation: `0`
    d) Steps: `20mil`
    e) Origin X: `1250mil`
    f) Origin Y: `1250mil`
    g) Width: `1000mil`
    h) Quadrants: `1,2,3 and 4`

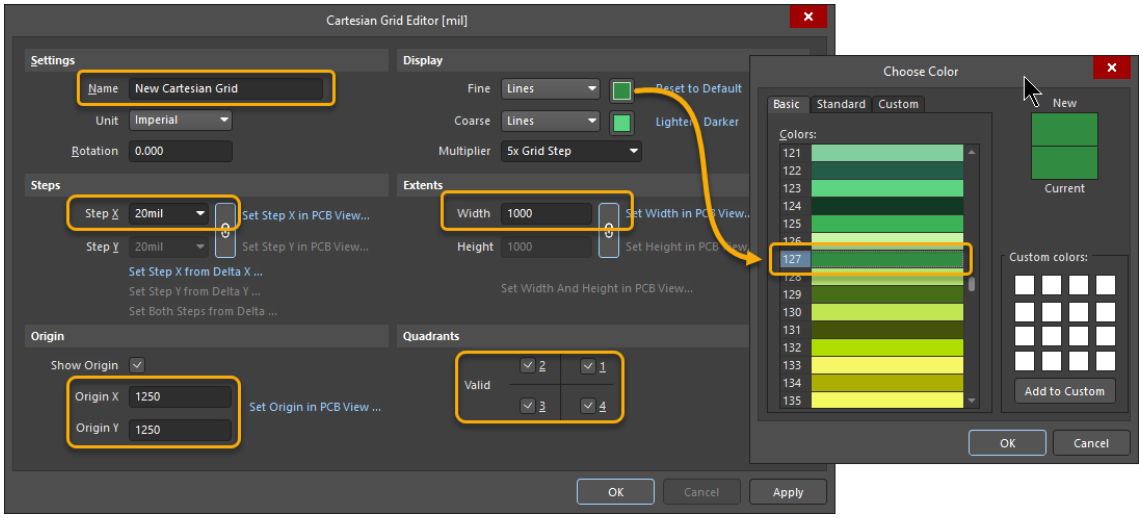

Figure 6. Cartesian Grid Editor

24. Click **OK** to return to the PCB editor and note the presence of the new grid. Zoom accordingly to see the fine grid detail as shown in Figure 7 below.

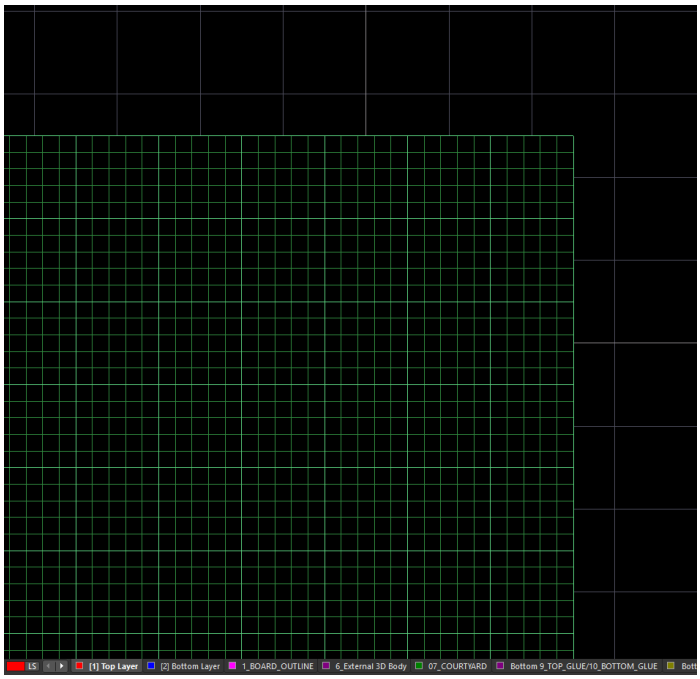

Figure 7. New Cartesian grid shows in Quadrants 1, 2, 3 and 4 in the PCB

25. Find component `DI1` (**J » C**) and drag it into the PCB space, to the right side of the new Cartesian grid. You will notice that the grid is no longer visible when moving the component. It will reappear when the component is not being moved.
26. Open the *Properties* panel and enable the **Comp** checkbox for the `New Cartesian Grid` as highlighted in Figure 8. This will allow us to place components on that grid.

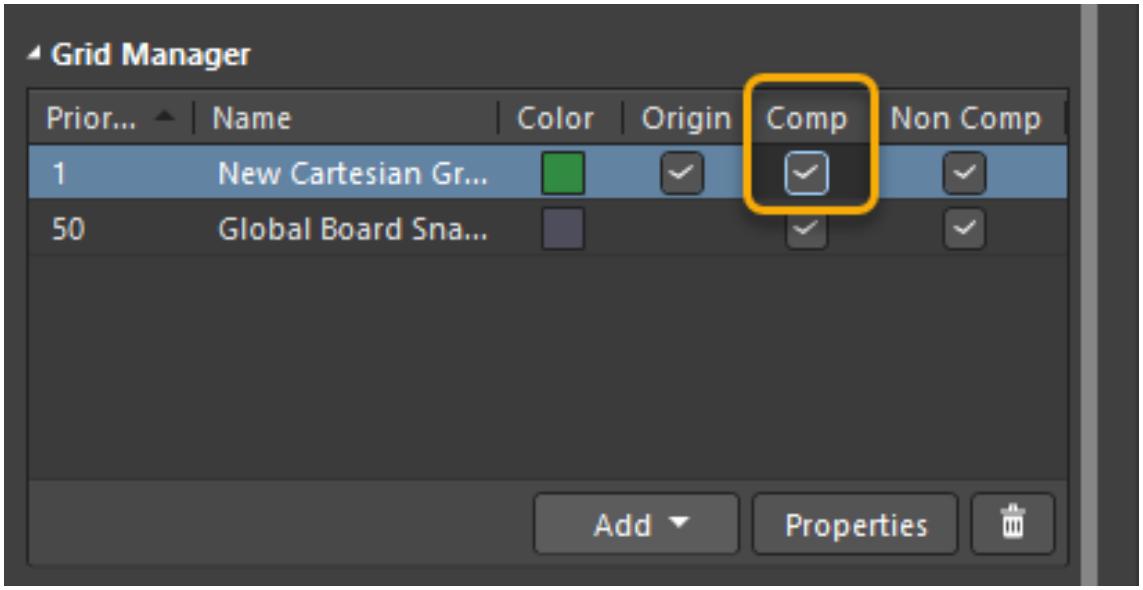

Figure 8. Enabling the Comp checkbox to be able to place components on the new grid

27. Move component `DI1` again to see that the grid now remains visible.

1.6.3 **Custom Polar Grid**

28. Open the *Properties* panel again.
29. Click **Add** in the *Grid Manager* pane and select **Add Polar Grid**.
30. Enable the **Comp** checkbox to make the polar grid available for component placement.
31. Double-click the *New Polar Grid* name to open the *Polar Grid Editor*. Set the values as shown in Figure 9 below. The *Fine* and *Coarse* line colors are assigned to values `106` and `107` respectively.
    a) Name: `Polar Grid`
    b) Unit: `Imperial`
    c) Angular Step: `5`
    d) Radial Step: `50mil`
    e) Origin X: `2750mil`
    f) Origin Y: `2750mil`
    g) Start Angle: `0°`
    h) End Angle: `360°`
    i) Radial Range Min `0mil`
    j) Radial Range Max: `1200mil`
32. Click **OK**.

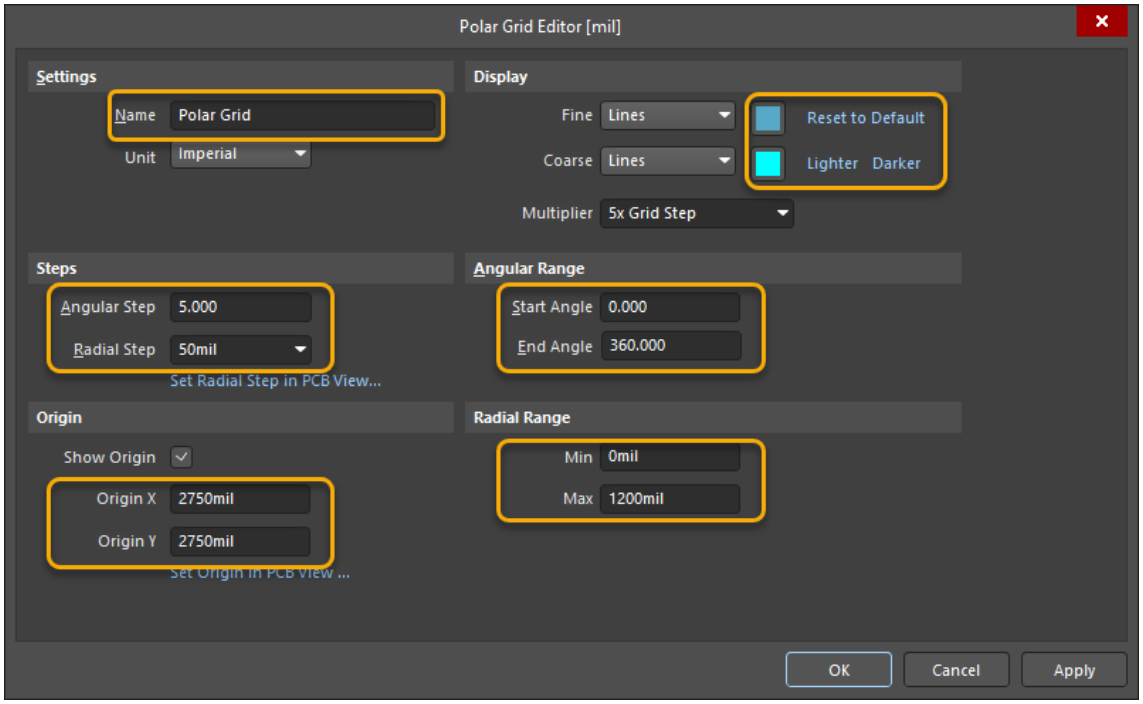

Figure 9. Polar Grid Editor

33. Note the new Polar Grid in the PCB editor was generated over the Cartesian Grid we just generated.
34. The Grid Manager allows you to change the priority for the overlapping grids if needed as shown in Figure 10 below.

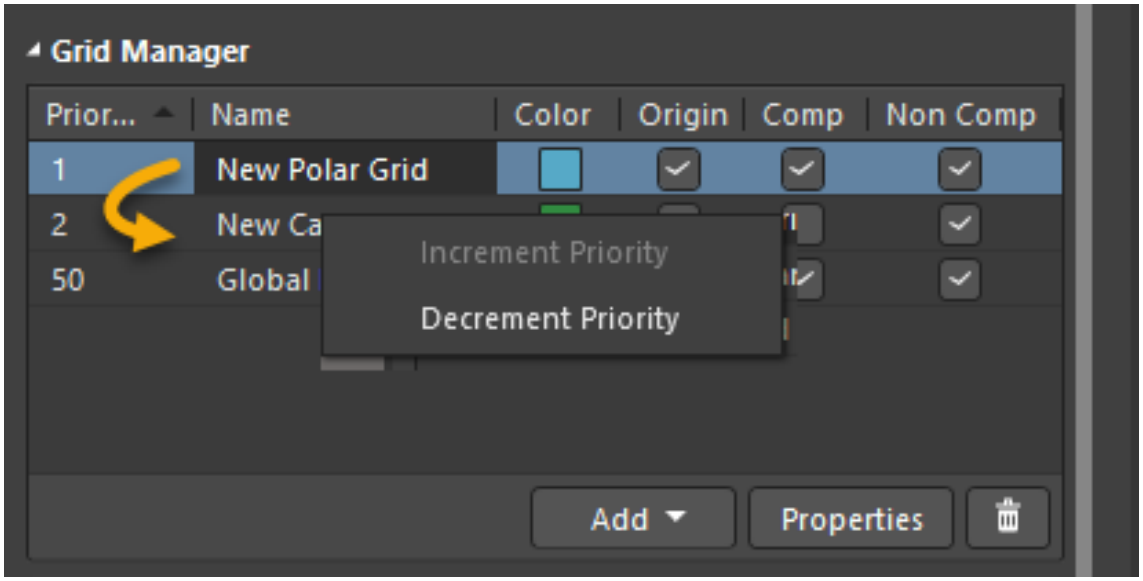

Figure 10. Priority for the Grids

35. Drag component `DI1 and DI2` to the newly added polar grid. Test the rotations of the components by pressing **Spacebar** and **Shift + Spacebar** while moving the component to notice its alignment to the grid as shown in Figure 11 below.

For a better visual experience, hide the rat's nests by going to
View » Connections » Hide All, (N » H » A).
To activate all the connections
View » Connections » Show All, (N » S » A).

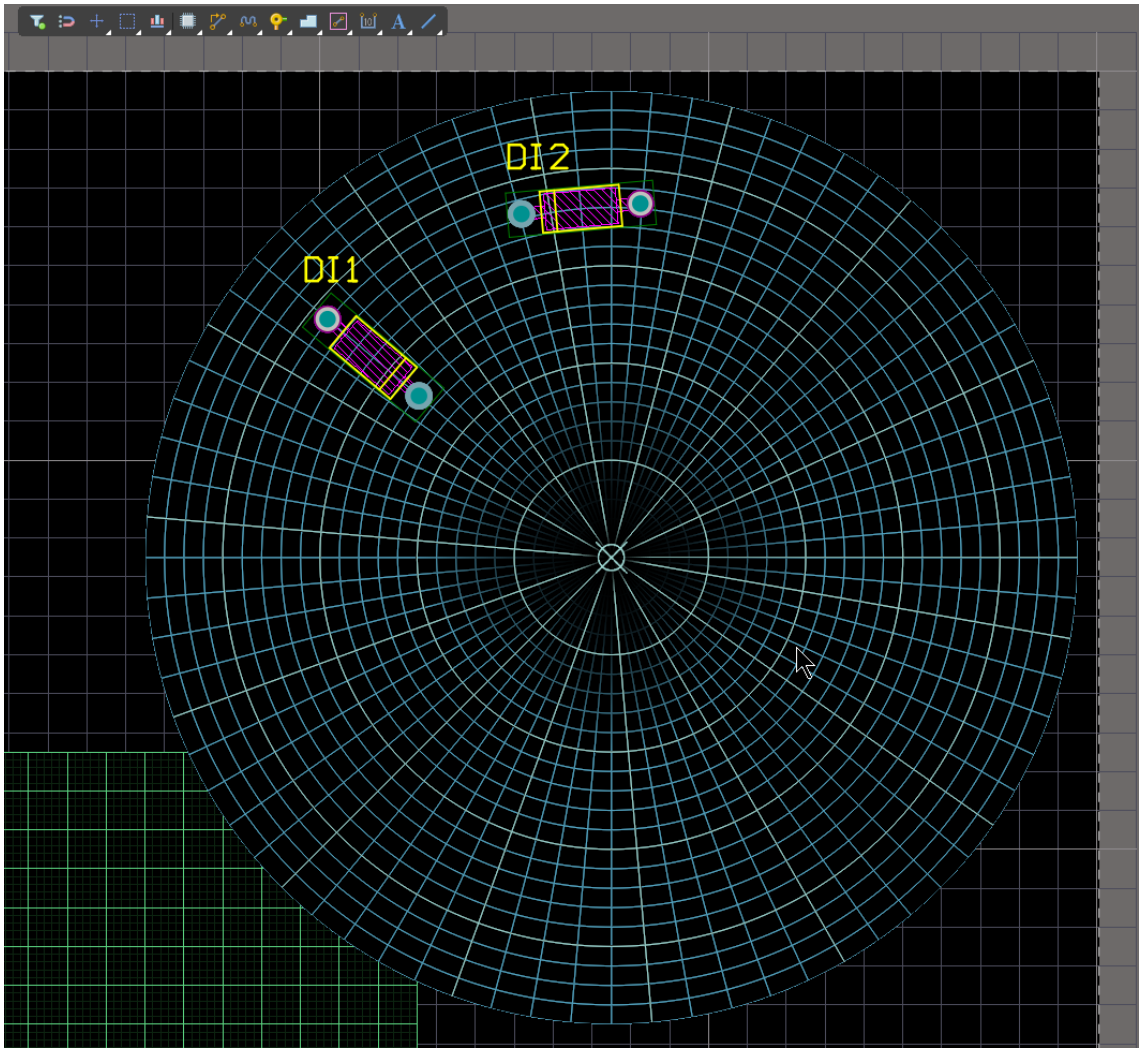

Figure 11. Aligning components to a Polar Grid

36. Save all documents using **File » Save All**.
37. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server**

    .
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 19: PCB Custom Grid Manager - [Add Your Name]- Finished`
        iii) Select **OK**
38. **When ready, close the project and any open documents**.

Congratulations on completing the Module!

Module 19: PCB Custom Grid Manager

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer