

Preparing the PCB

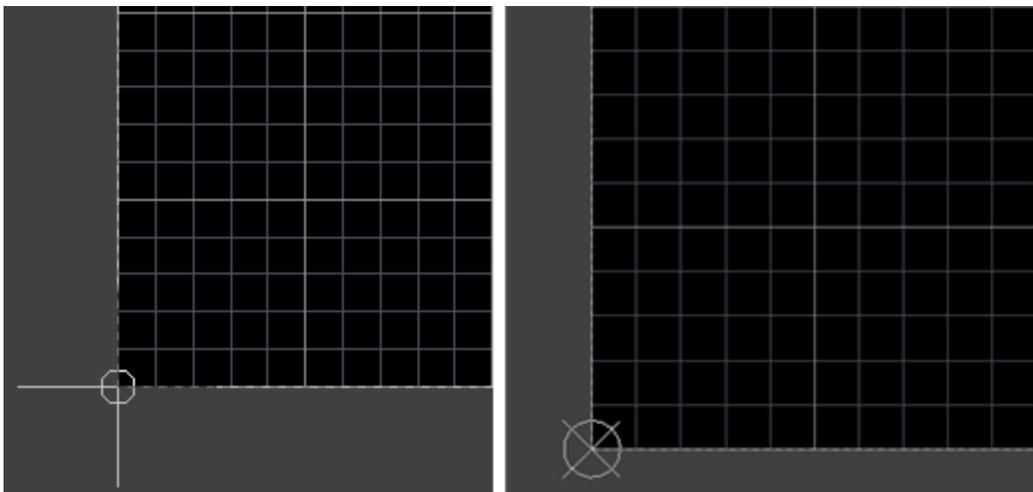
Before we pass our captured design into the PCB for layout, we want to prepare the PCB by defining the board extents. Before we begin component placement we will want to define and modify document options and design rules, and define the layer stack we plan to use.

First, let's get our Board Shape defined. This is a small PCB, so let's just make a 30mm x 30mm square for now. We will start by setting the origin, and defining our snap grid.

Setting the Origin and the Grid:

Exercise

1. There are two origins used in the software, the Absolute Origin, which is the lower left of the workspace, and the user-definable Relative Origin, which is used to determine the current workspace location. Before setting the origin, Keep zooming in to the lower left of the current board shape until you can easily see the grid - to do this position the cursor over the lower-left corner of the board shape and press PgUp until both the Coarse and Fine grids are visible, as shown in the images below.
2. To set the Relative Origin, select Home Tab → Grids and Units | Origin → Set, position the cursor over the bottom left corner of the board shape, then left click to locate it.



Select the command, position the cursor over the lower-left corner of the board shape (left image), then click to define the origin (right image).

3. The next step is to select a suitable snap grid. The grid is set in the Grids and

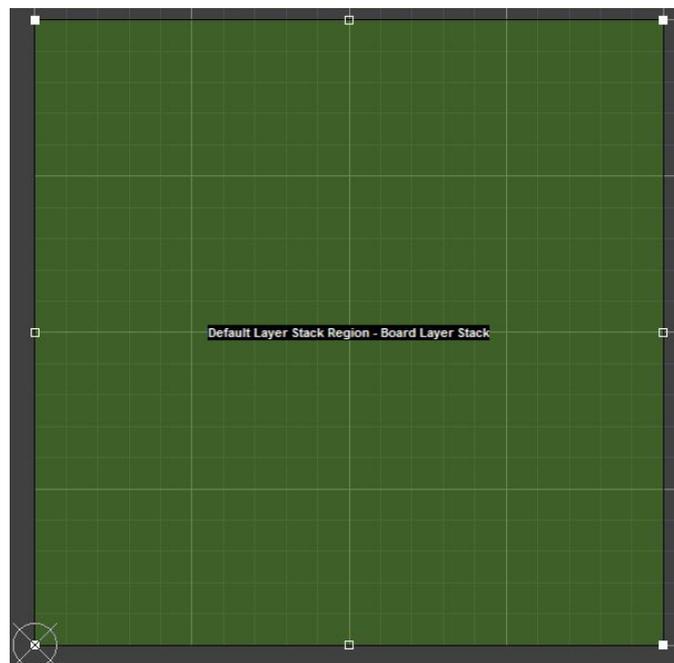
Units section of the Home tab of the Ribbon. During the course of design it is quite common to change grids, for example you might use a coarse grid during component placement, and a finer grid for routing, which can be done by selecting a new value from the dropdown list, or typing in the required value directly - press Enter to have your value accepted.

4. For this tutorial you will be using a Metric grid, so click the  button to switch to Metric.
5. The first grid you will be using is 5mm, enter this value into the Snap Grid field.

Redefining the Board Shape:

6. The default board shape is 4x4 inch, for the tutorial the board size is 30mm x 30mm.
7. To zoom back out and show the all of the workspace currently being used, click View | Zoom | Zoom All (Ctrl+PgDn).
8. The next step is to change the board shape to 30mm x 30mm. Your choice now is to either redefine the board shape (draw it again), or edit the existing board shape. For a simple square or rectangle, it is more efficient to edit the existing board shape, to do this select Home | Board | Board Shape » Edit Board Shape.
9. The board display will change, with the shape shown in green. Editing handles will appear at each corner and the center of each edge.

Note that clicking anywhere other than on an editing handle or an edge of the shape will drop you out of board shape editing mode.



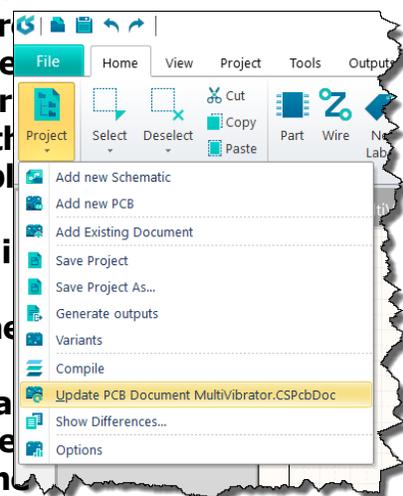
10. The objective is to resize the shape to create a 30mm by 30mm board. The Coarse visible grid is 25mm (5x the snap grid), and the Fine visible grid is 5mm - this will be used as a guide. You can now either: slide the upper and right edges down and in to create the correct size; or move 3 of the corners in, leaving the one that is at the origin in its current location.
11. To slide the top edge down, position the cursor over the edge (but not over a handle), then when the cursor changes to a double-headed arrow click and hold, then drag the edge to the new location, so that the Y cursor location is 30mm on the Status bar.
12. Repeat the process to move the right-hand edge in, positioning it when the X cursor location is 30mm on the Status bar.
13. Click anywhere in the workspace to drop out of board shape editing mode. If you inadvertently click and exit this mode before you are finished, simply re-enable it and continue editing your board shape.
14. Save the board.

Transferring the Design

The process of transferring a design from the capture stage to the board layout stage is launched via the Update command Home Tab → Project | Project → Update PCB Document Multivibrator.CSPcbDoc on the Schematic editor Ribbon (or Home Tab → Project | Project → Import Changes from Multivibrator.PrjPcb from the PCB editor Ribbon).

When you run this command the design is compiled and a set of Engineering Change Orders is created, which:

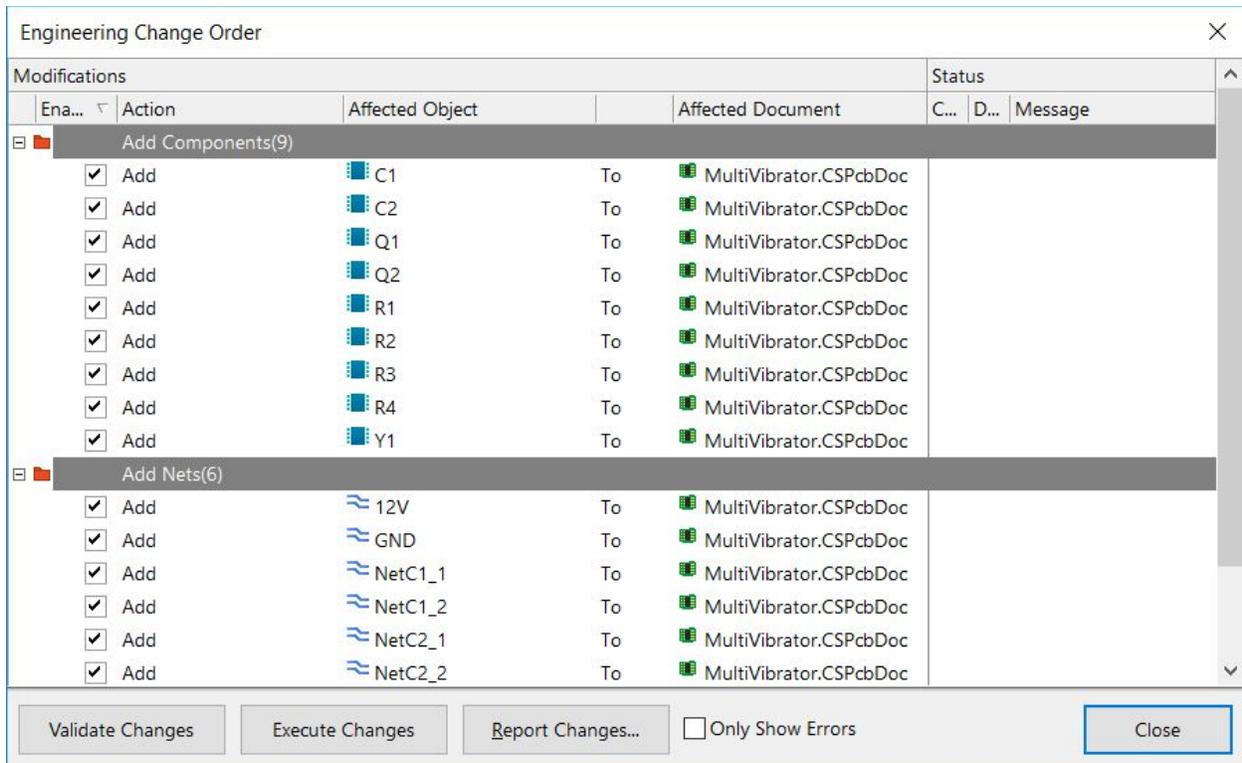
- List all components used in the design, and the footprint required for each. When the ECOs are executed the software will attempt to locate each footprint in the currently available libraries or available Content Vault, and place each into the PCB workspace. If the footprint is not available an error will occur.
- A list of all nets (connected component pins) in the design is created. When the ECOs are executed the software will add each net to the PCB, and then attempt to add the pins that belong to each net. If a pin cannot be added an error will occur - this most often happens when the footprint was not found, or the pads on the footprint do not map to the pins on the symbol.



Exercise

Set the schematic document, **Multivibrator.SchDoc** as the active document in CircuitStudio by clicking on its document tab.

1. Select Home Tab → Project | Project → Update PCB Document **Multivibrator.CSPcbDoc** from the Ribbon. The project will compile and the



Engineering Change Order dialog will open.

2. Click on **Execute Changes** to send the changes to the PCB editor.
3. When completed, the target PCB opens with the **Engineering Change Order** dialog open on top of it, and the **Done** column entries become ticked (as shown in the image below).
4. Click to **Close** the dialog and complete the transfer process.

Engineering Change Order					Status		
Ena...	Action	Affected Object	Affected Document	C...	D...	Message	
Add Components(9)							
<input checked="" type="checkbox"/>	Add	C1	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	C2	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	Q1	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	Q2	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	R1	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	R2	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	R3	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	R4	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	Y1	To	MultiVibrator.CSPcbDoc	✓	✓	
Add Nets(6)							
<input checked="" type="checkbox"/>	Add	12V	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	GND	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	NetC1_1	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	NetC1_2	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	NetC2_1	To	MultiVibrator.CSPcbDoc	✓	✓	
<input checked="" type="checkbox"/>	Add	NetC2_2	To	MultiVibrator.CSPcbDoc	✓	✓	

Only Show Errors

The components will have been positioned outside of the board, ready for placing on the board. There are a few steps before starting the component process, such as configuring the placement grid, the layers and the design rules.

Configuring the Display of Layers

As well as the the layers used to fabricate the board, including: signal, power plane, mask and silkscreen layers, the PCB Editor also supports numerous other non-electrical layers. The layers are grouped in the following way:

- **Electrical layers** - includes the 32 signal layers and 16 internal power plane layers.
- **Mechanical layers** - there are 32 general purpose mechanical layers, used for design tasks such as dimensions, fabrication details, assembly instructions, or special purpose tasks such as glue dot layers. These layers can be selectively included in print and Gerber output generation. They can also be paired, meaning that objects placed on one of the paired layers in the library editor, will flip to the other layer in the pair when the component is flipped to the bottom side of the board.
- **Special layers** - these include the top and bottom silkscreen layers, the solder and paste mask layers, drill layers, the Keep-Out layer (used to define the electrical boundaries), the multilayer (used for multilayer pads and vias), the connection layer, DRC error layer, grid layers, hole layers, and other display-type

layers.

The display attributes of all layers are configured in the *View Configurations* dialog. To open the dialog:

- Select the View Tab → View | Switch to 3D → View Configurations → View Configuration menu entry, or
- Click the current layer  color box down the bottom-left of the workspace.

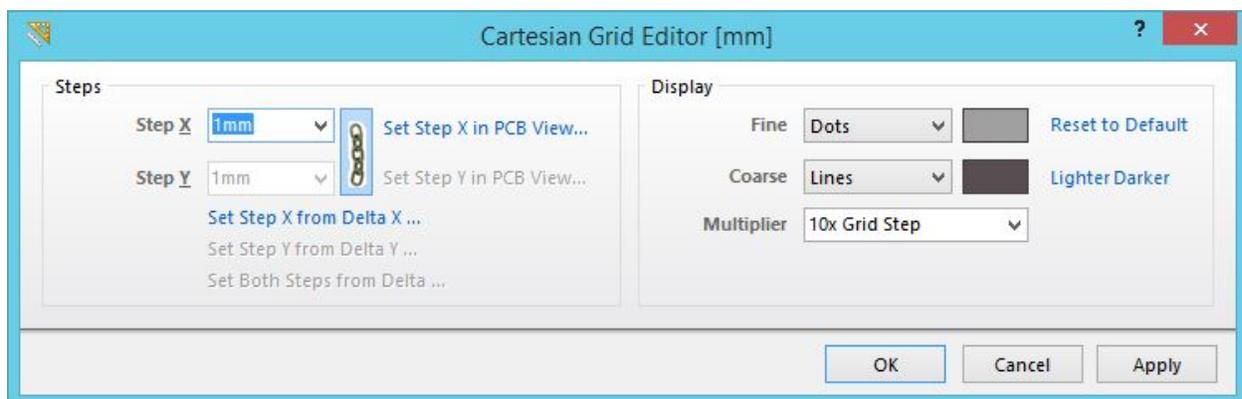
Configuring the Layer Visibility:

Exercise

1. Open the *View Configurations* dialog.
2. In the Board Layers and Colors tab, confirm that the 2 signal layers are visible.
3. Note that this dialog is where you control the display of the mask layers, the silkscreen layers and the system layers, such as DRC and grids.
4. To have less visual "clutter" during placement and routing, disable the display of the Mechanical Layers, all of the Mask Layers, and the Drill Guide and Drill Drawing layers.
5. Switch to the View Options tab.
6. Confirm that the Show Pad Nets option is enabled, and the Net Names on Tracks Display is set to Single and Centered.
7. Click OK to accept the settings and close the dialog.

Setting the Snap Grid:

The value of the snap grid can be configured directly in the Home tab of the Ribbon, or it can be configured in the *Cartesian Grid Editor* dialog (Home Tab → Grids and Units | Properties).



Defining the component placement snap grid:

Exercise

1. Click Home | Grids and Units | Properties to open the *Cartesian Grid Editor* dialog. You can also access the *Cartesian Grid Editor* dialog directly using the **Ctrl+G** shortcut keys.
 2. Type the value **1mm** into the **Step X** field. Because the **X** and **Y** fields are linked, there is no need to define the **Step Y** value.
 3. To make the grid visible at lower zoom levels set the **Multiplier** to **5x Grid Step**, and to make it easier to distinguish between the two grids, set the **Fine grid** to display as lighter colored **Dots**.
 4. Click **OK** to close the dialog.
-

COPYRIGHT

Copyright in the Material on this Website is owned or licensed by Altium LLC and its affiliated companies (collectively, "Altium") as of 2017.

Except where necessary for viewing the Material on this website on your browser, or as permitted under the United States Copyright Act or other applicable laws or these Terms of Use, no Material on this website may be reproduced, adapted, uploaded to a third party, linked to, framed, performed in public, distributed or transmitted in any form by any process without the specific written consent of Altium LLC.

TRADEMARKS

ACTIVEBOM®, ActiveRoute®, Altium Designer®, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, TASKING® 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.