Altium Introduction (II)

Henry Lau (2021)

V1.0

M2 ROBOCON STUDIO

Change log / bug fixes

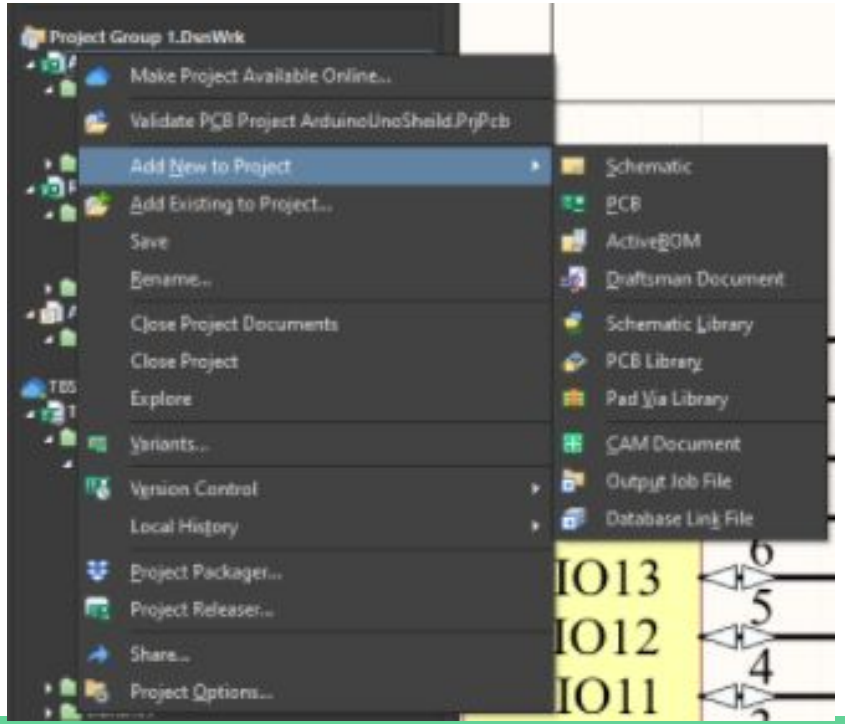
Version	Description
V1.0	Initial

PCB

Add New PCB (if there is no PCB in the project)

Click RMB on the project and navigate to PCB under Add New to Project

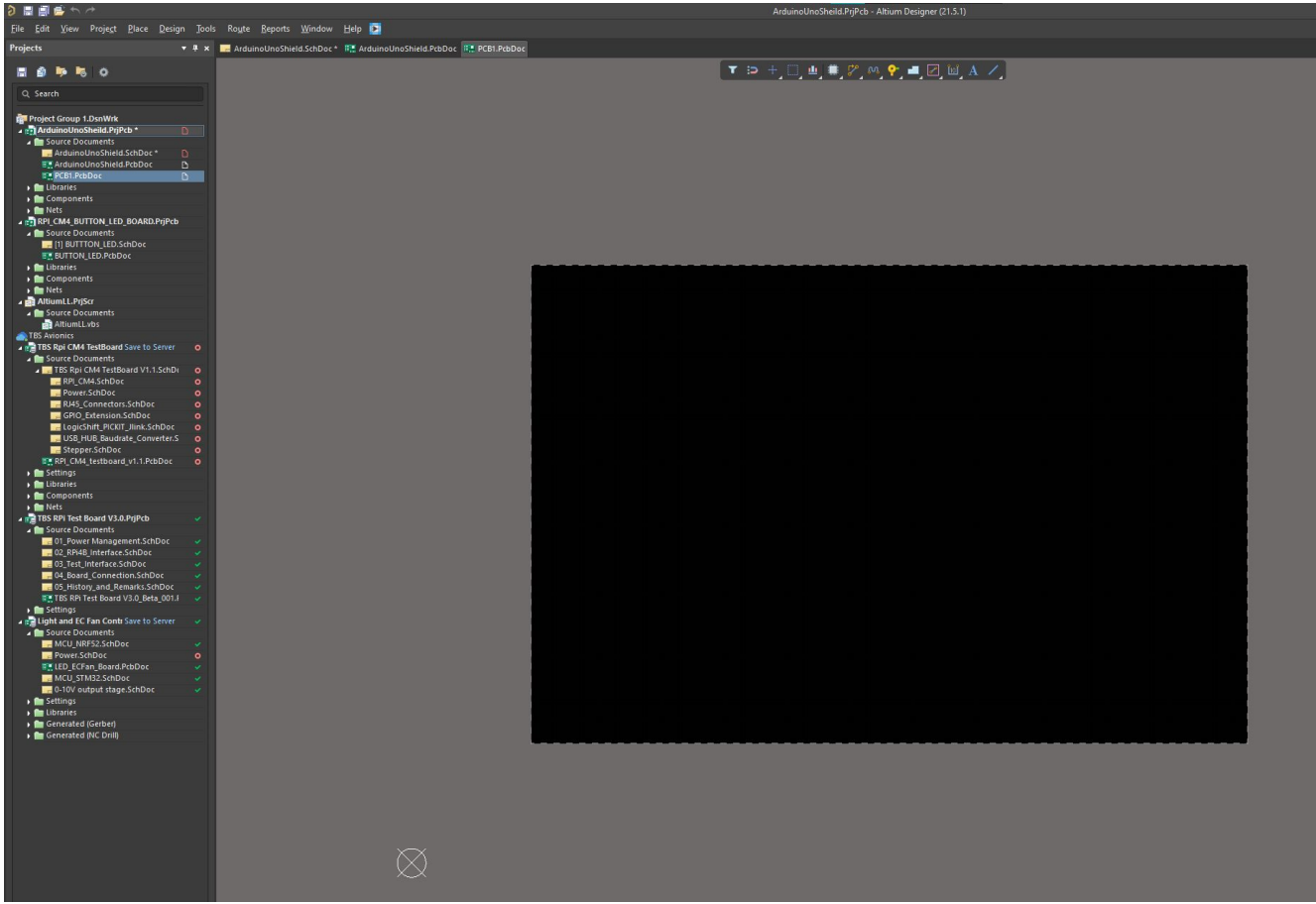
Add Existing PCB should be use when you want to reuse PCB from other projects, don’t forget to clone it beforehand



New PCB

You will be greeted with a blank PCB after you create a new PCB

Save it before progressing



PCB

Unit, Property, Grid size of PCB

Unit in the PCB

HUD will display on the top left of the PCB, showing the position of the cursor.

The unit of the PCB could be changed by press (Q), it could be changed between mm and mil

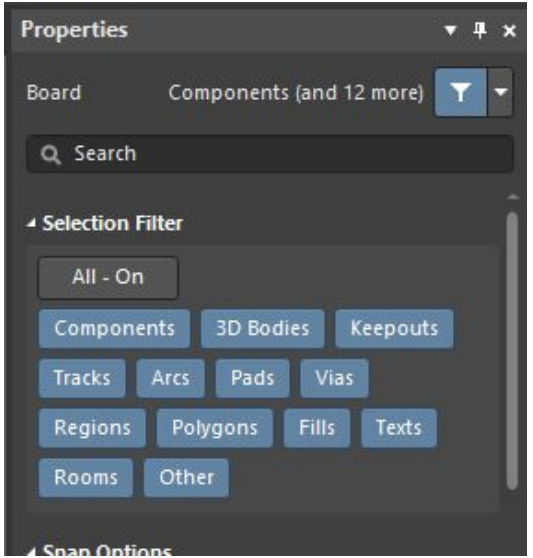
Using either mil or mm is just personnel preference

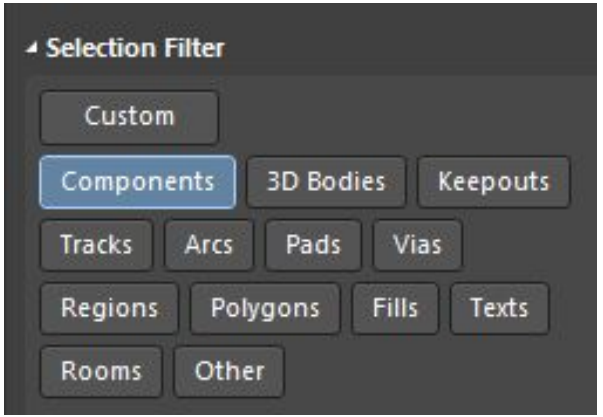
PCB Property (Selection Filter)

The Property of the PCB could be located in the “Properties" tab

Selection Filter allows the user to select the objects that he would like to be able to select

For example, the bottom right picture shows the selection filter changed to selecting “Components” only (You cannot select other things like Wires and Texts)



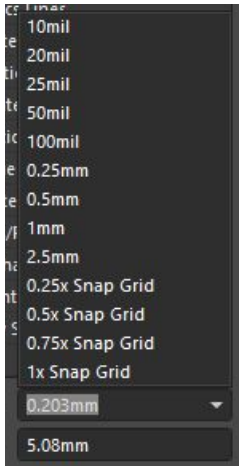


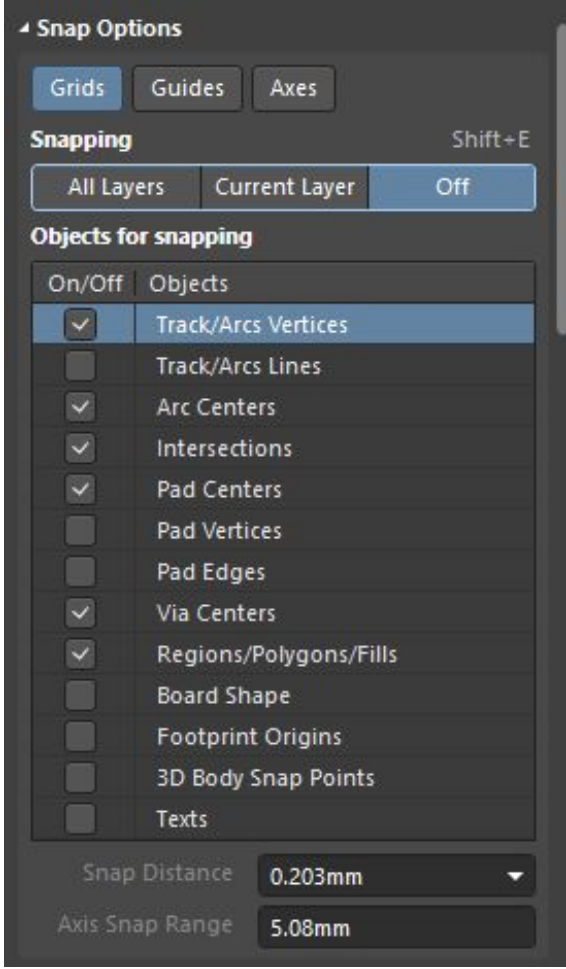
PCB Property (Snap Options)

Scrolling down Property tab to see Snap Options

The Snap distance and Axis Snap Range determines how far away your component will snap to the grid

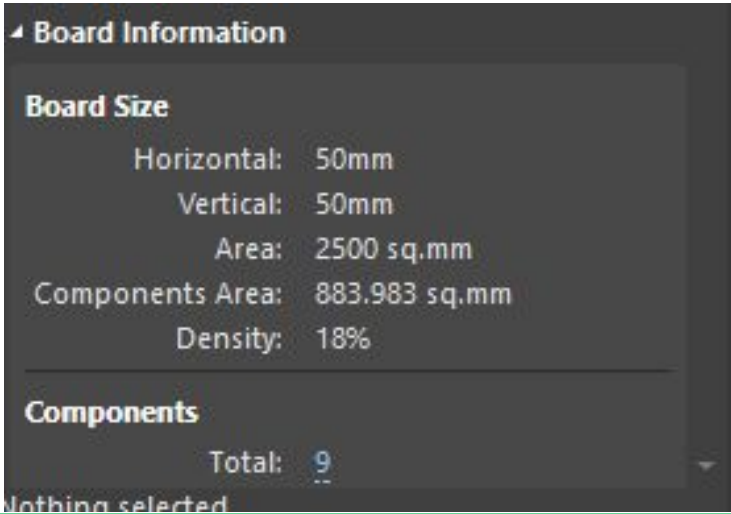
Objects of Snapping determines what the object being moved would snap to. When the object being moved is not snapping to your liking, select and unselect some of the options here



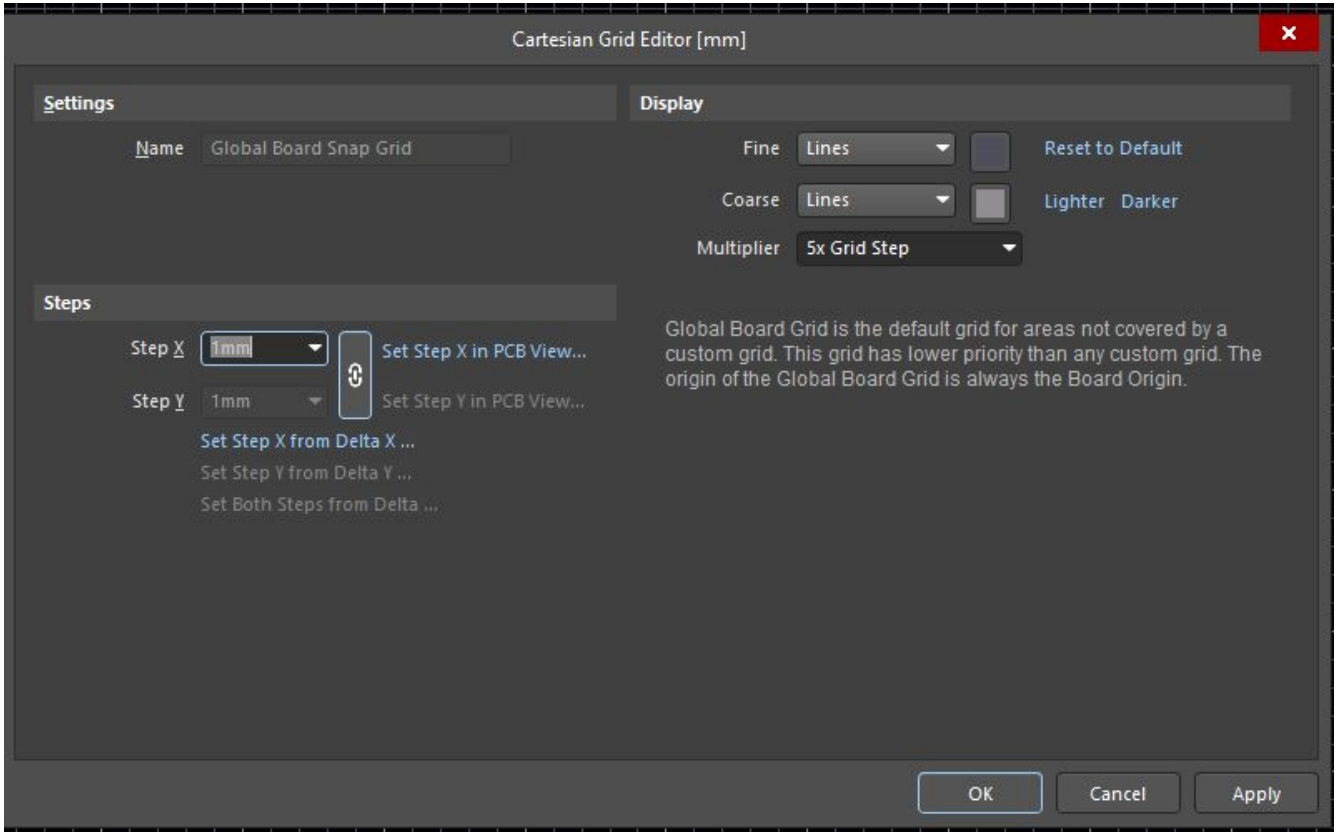


PCB Property (Board Information)

The Board Information shows basic info about the board, like its size, number of components and the density of the board



Grid Editor


Cartesian Grid Editor [mm]
Settings
Name
Global Board Snap Grid
Steps
Step X
1mm
Step Y
1mm
Set Step X in PCB View...
Set Step Y in PCB View...
Set Step X from Delta X ...
Set Step Y from Delta Y ...
Set Both Steps from Delta ...
Display
Fine
Lines
Coarse
Lines
Multiplier
5x Grid Step
Reset to Default
Lighter
Darker
Global Board Grid is the default grid for areas not covered by a custom grid. This grid has lower priority than any custom grid. The origin of the Global Board Grid is always the Board Origin.
OK
Cancel
Apply

Grid Editor (Ctrl + G), Grid Size

Pressing (Ctrl + G) will open a window for editing the grid size

Depend on the complexity of the project the common grid size that we preferred are

- 0.1 or 0.2mm for dense PCB
- 1, 5 or 10mm when editing board shape and placing large components like connectors

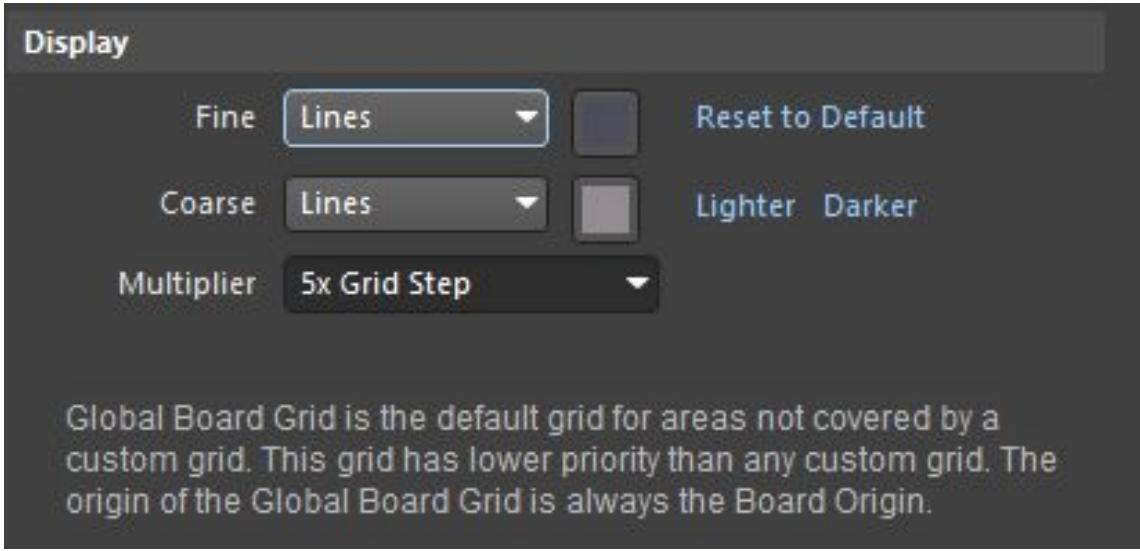
Some prefer to use imperial unit, some prefer to use SI unit

Grid Editor (Ctrl + G), Display

Display of the Grid could be configured here.

Choose between Lines or Dots, 5x or 10x grid size

The setting shown is just my personal preference



PCB

Import schematic to PCB

Import Changes to PCB / Update Changes to SCH

To get the components from schematic to PCB document, use Import command.

To apply the modification done in PCB to schematic, use Update command.

Since it is NOT recommended to use the Update function, only the import function would be mentioned

See next >>

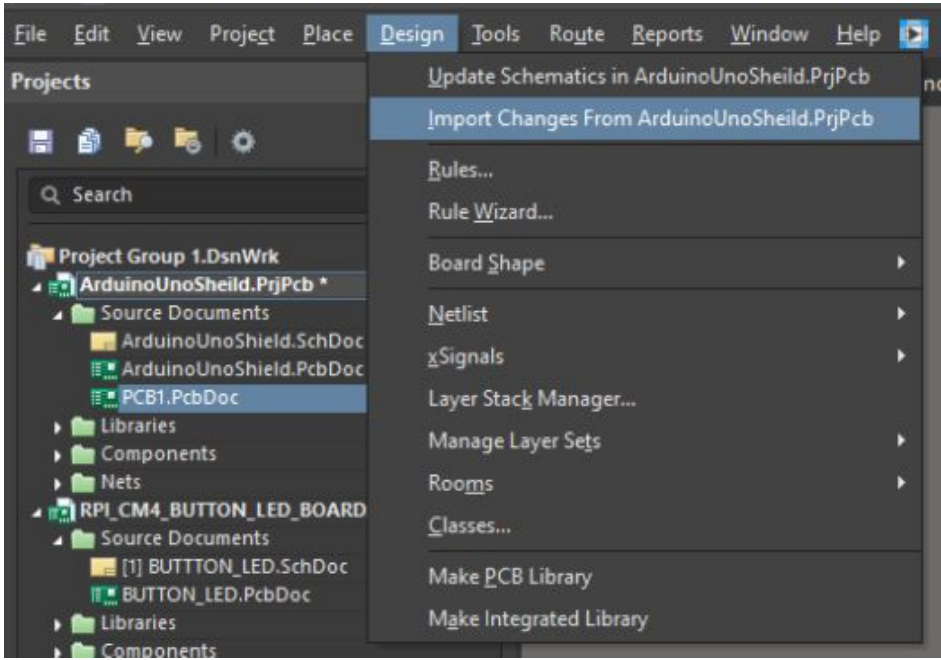
Import changes to PCB

You could either

- click Design -> Import Changes……

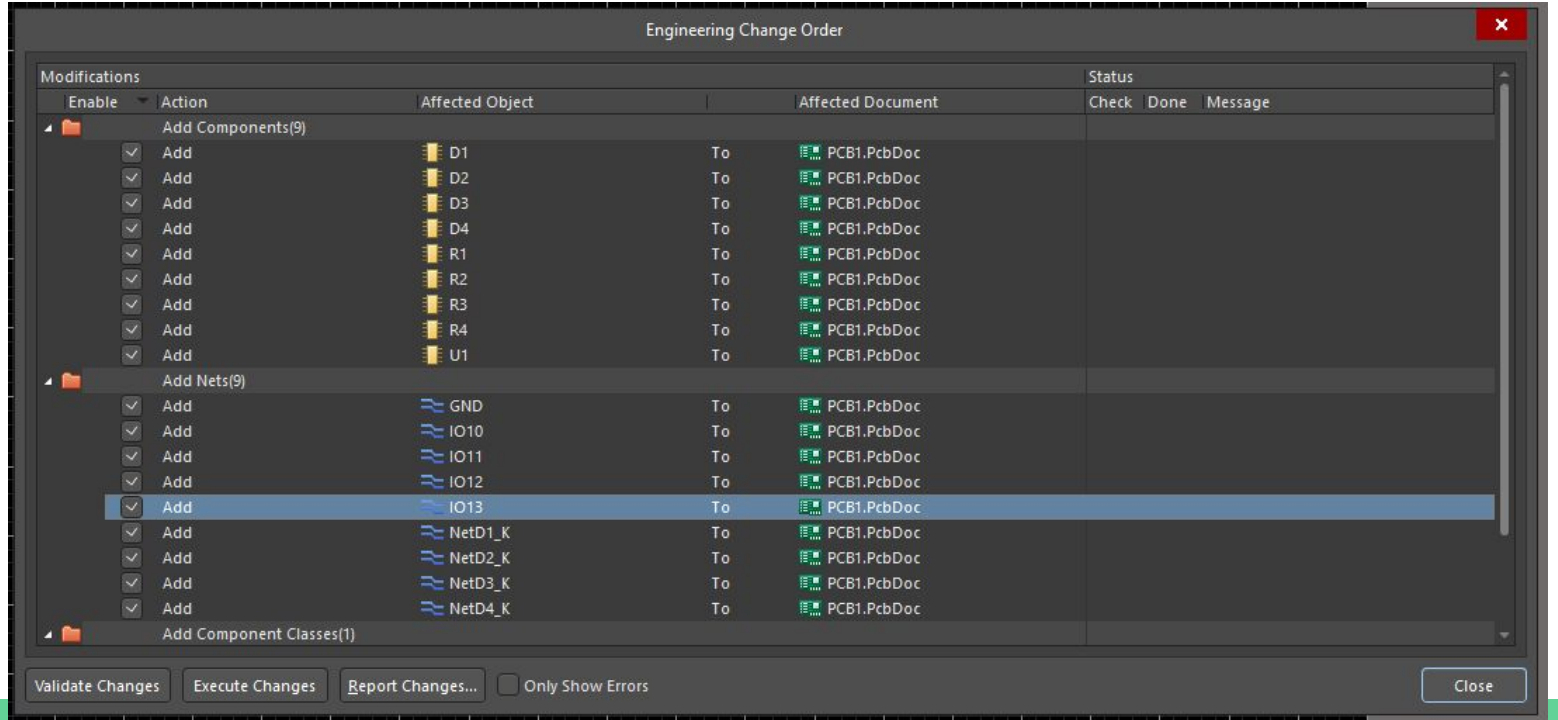
OR

- Click (D, I) on keyboard



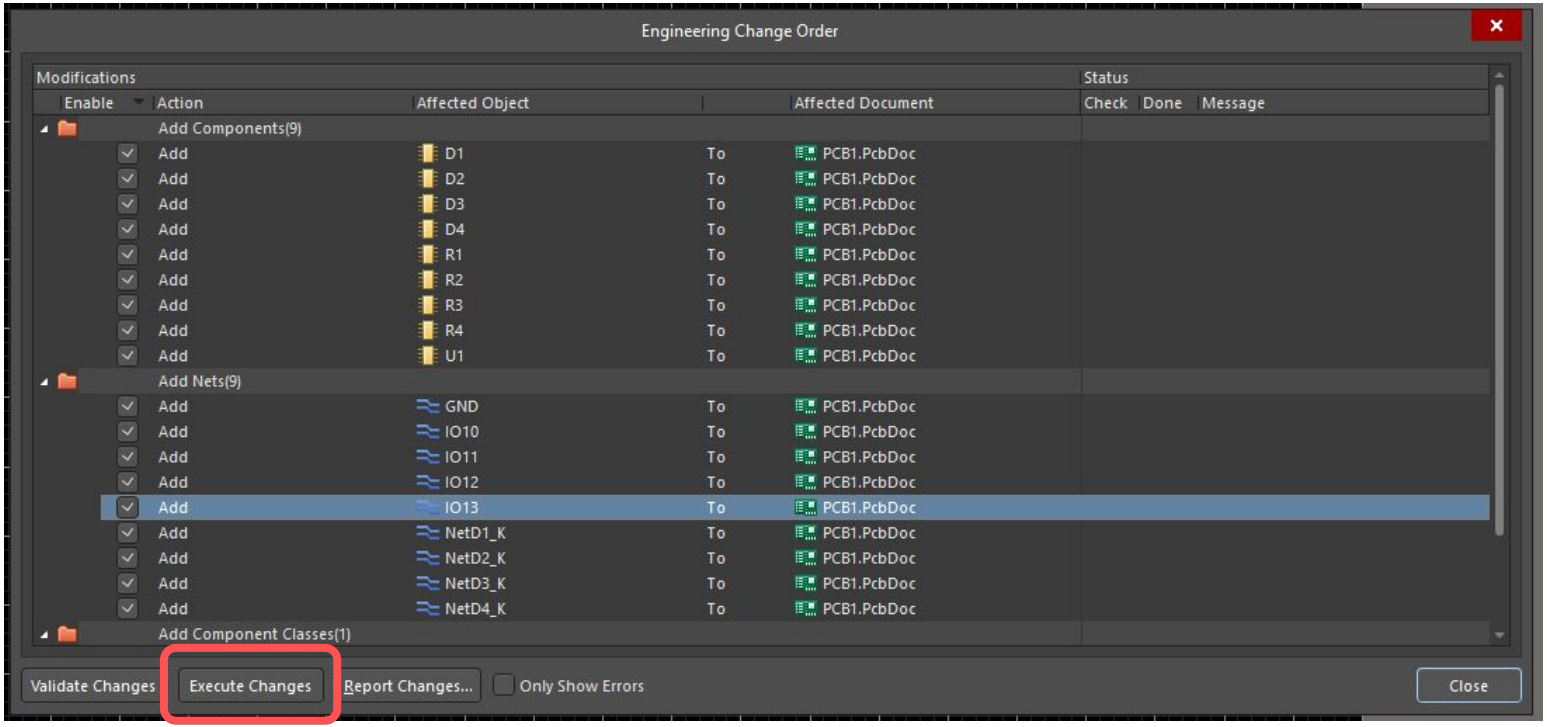
ECO (Engineering Change Order)

ECO will pop up when trying to import from schematic



ECO (Engineering Change Order)

Select “Execute Changes” to start importing the schematic to the PCB

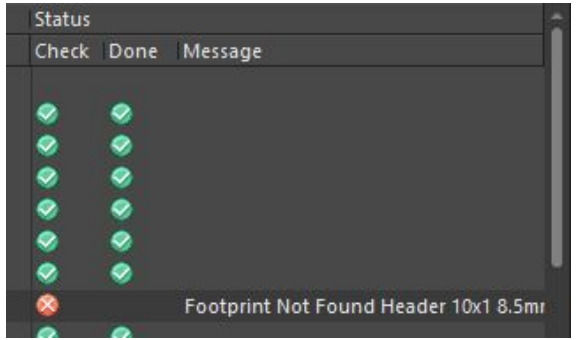


ECO

A successful import will show all changes as done

Having cross means there are some issue with the import, the message will provide details on the source of the error to ease debug process

After closing the windows, you should see a bunch of components on the PCB



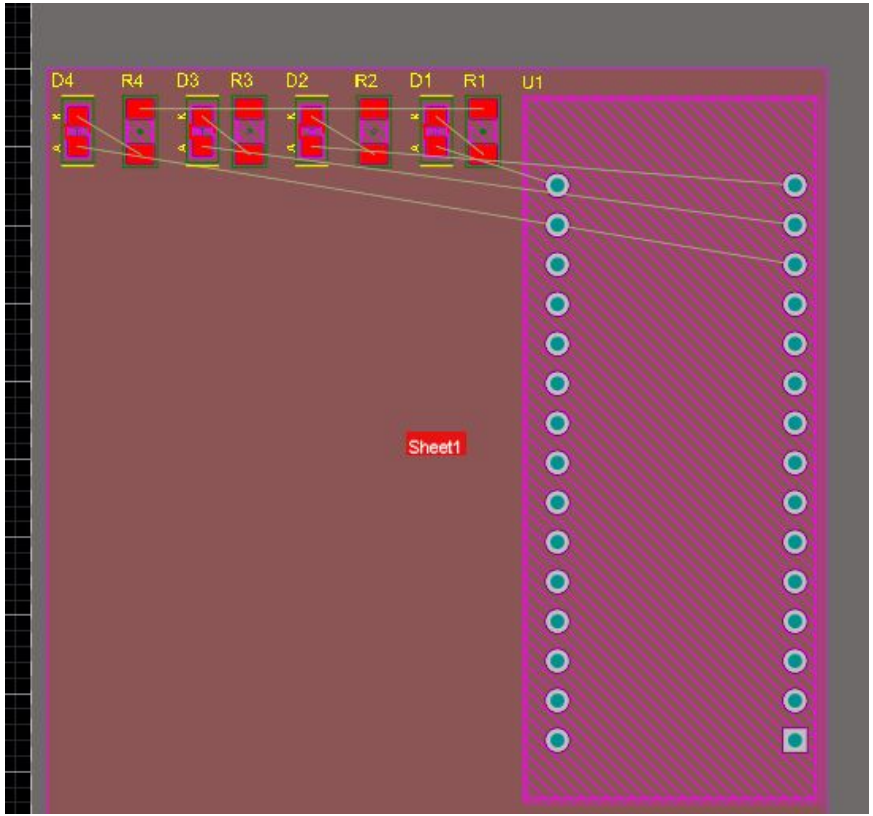
Imported Components

The imported components will show up within a red area called the room.

You could delete the room

The Room is not that useful for small project

Stop importing Room by following Embedded Ninja Altium guide



PCB

The 3 Viewing modes

Different viewing / editing mode for Altium

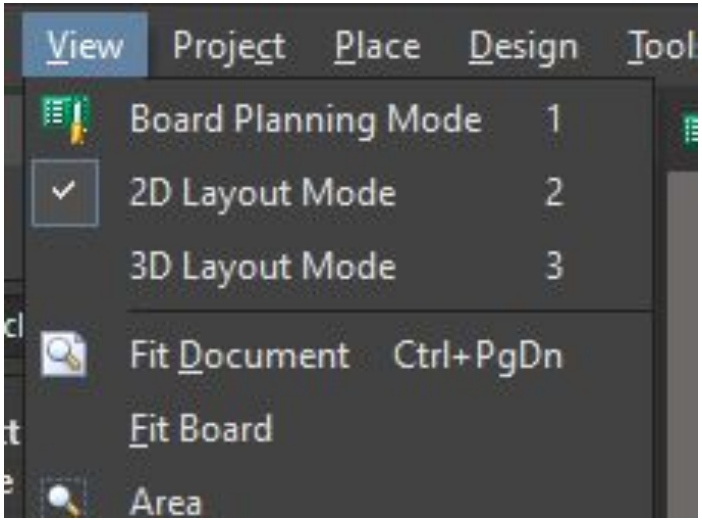
You can press (1) / (2) / (3) to switch between modes :

Press (1) on the keyboard to edit the board shape. The components should be transparent and the board should be turned to green;

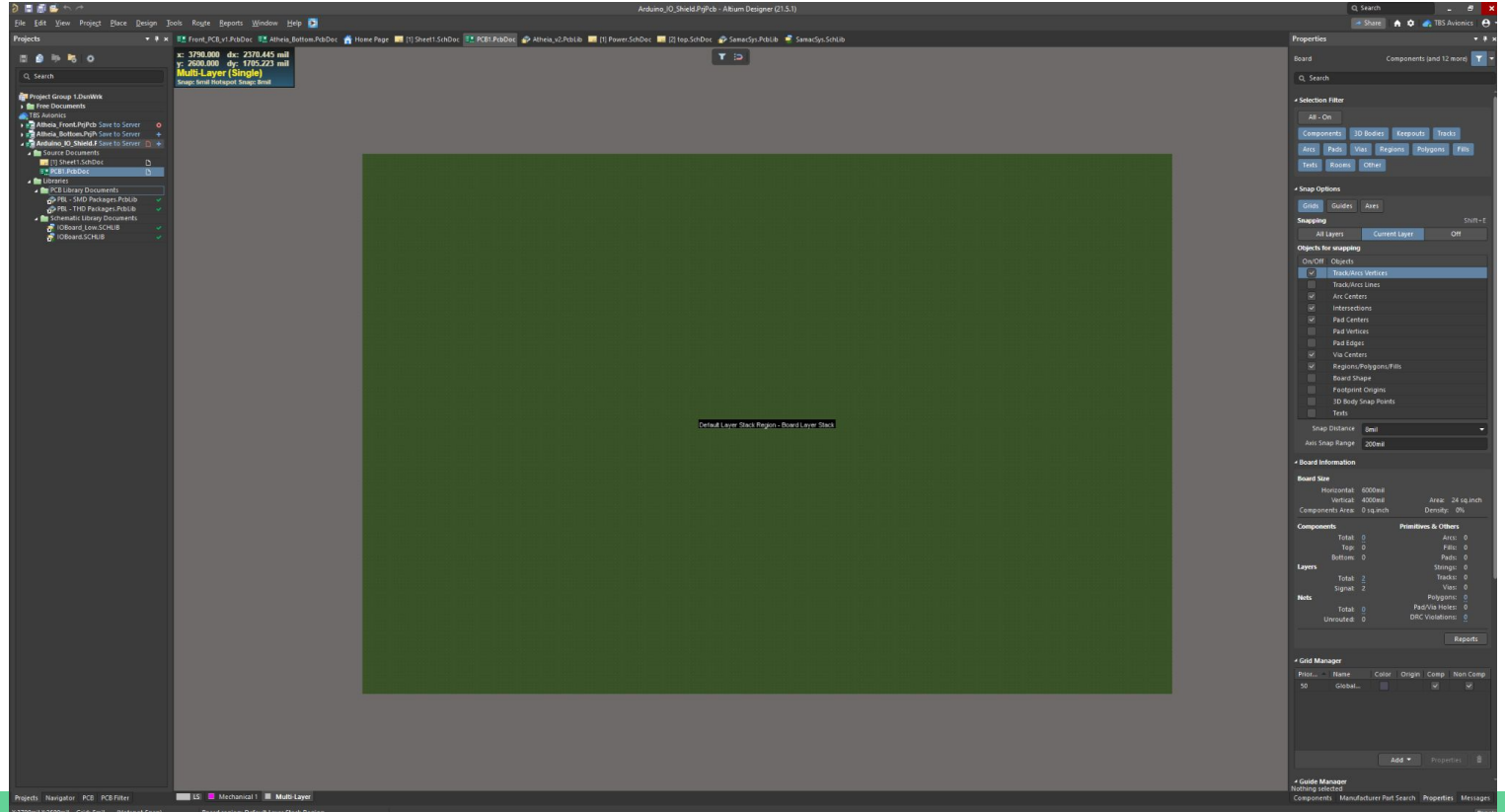
Press (2) on the keyboard to go back to editing the position of components and wiring mode;

Press (3) on the keyboard to enter 3D mode, usually in this mode, you would not be editing anything.

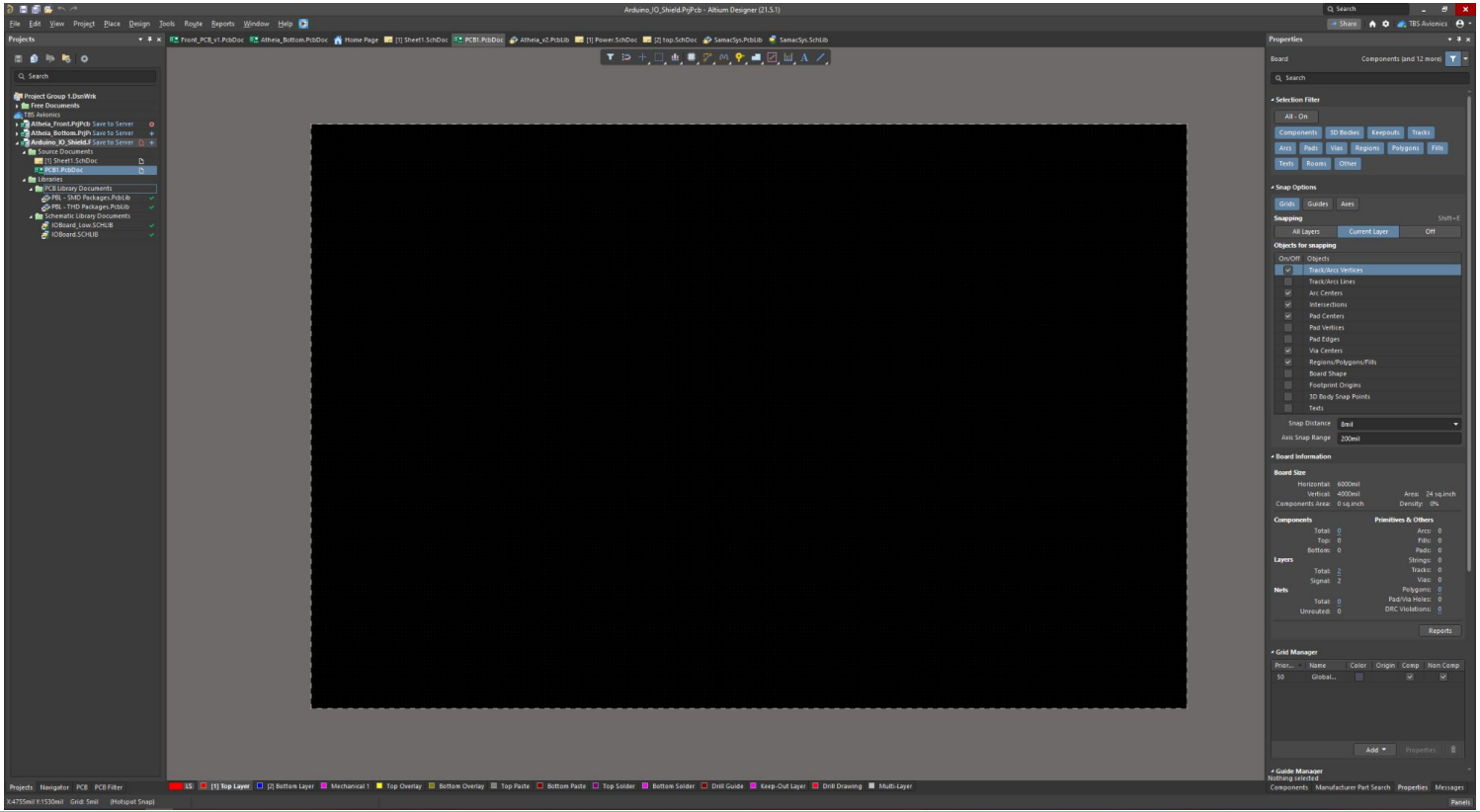
You could also switch the mode from “View”



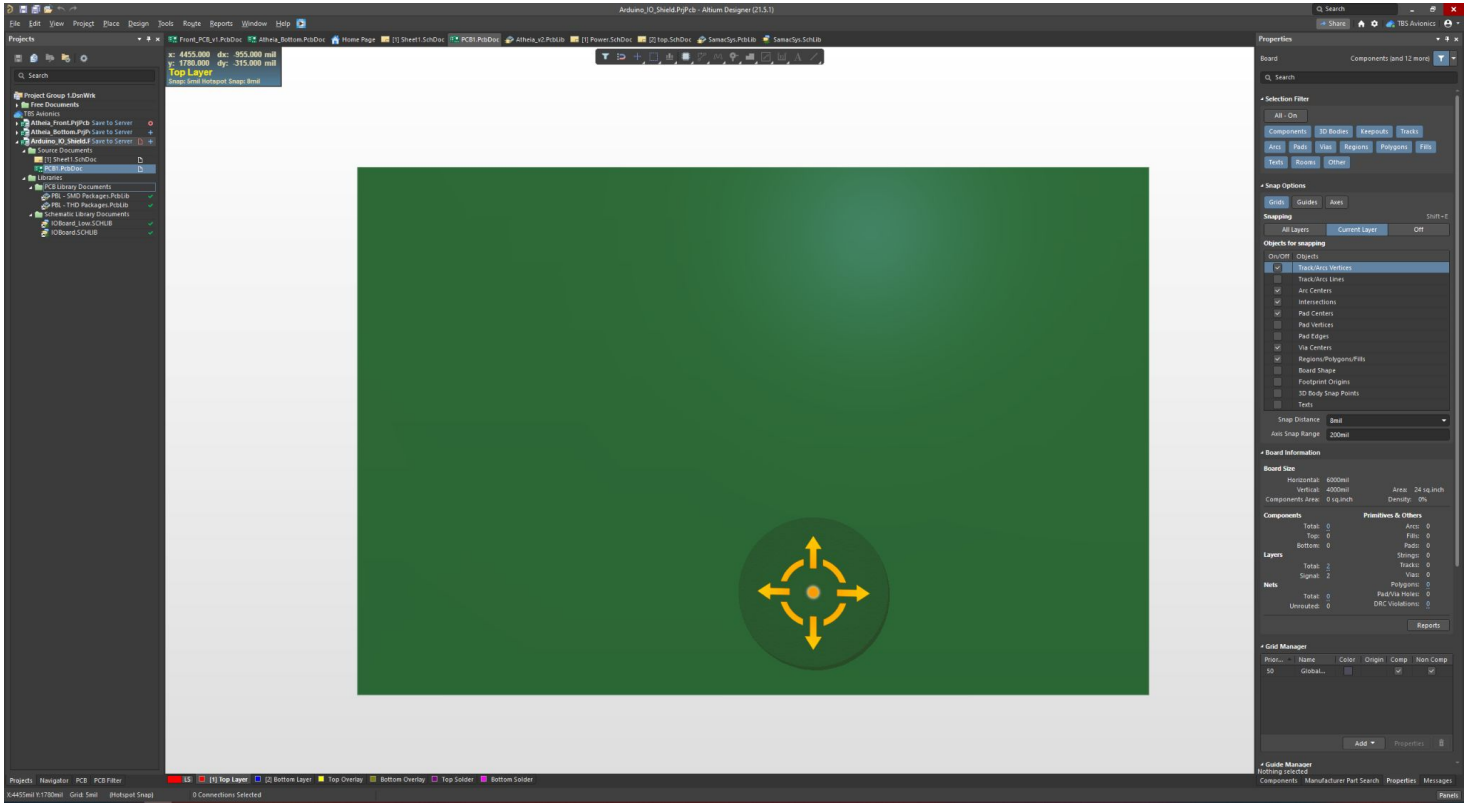
After Pressing 1 (Board Layout Mode)



After Pressing 2 (2D Layout Mode)



After Pressing 3 (3D Layout Mode)



PCB

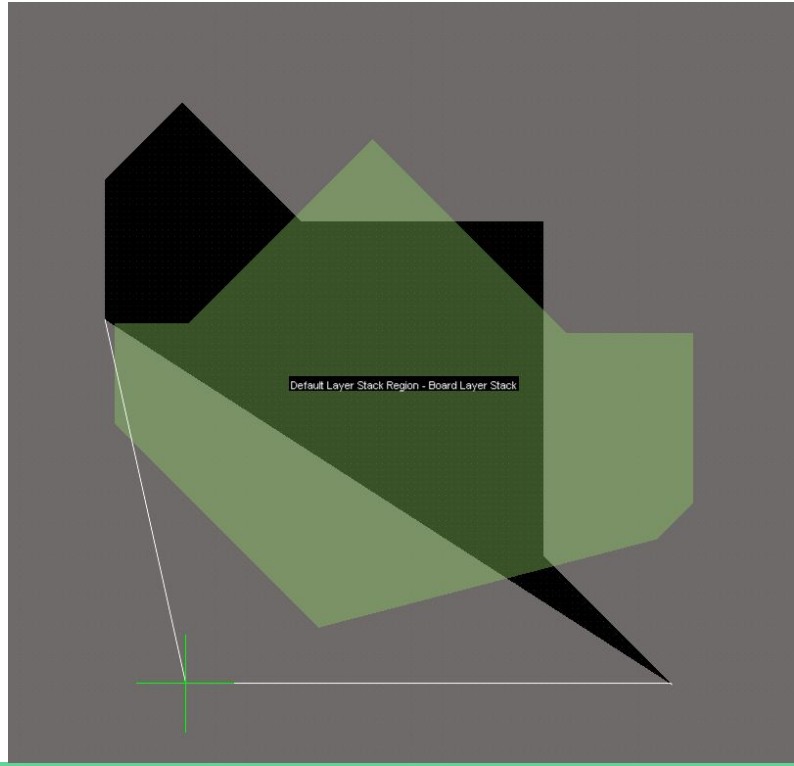
Define the Board in Board Layout Mode (1)

Redefine Board Shape (1, D, R)

The command is used to redefining the shape of the PCB

Click LMB to select vertices for the polygon. After defining the shape, press RMB to escape the redefine mode

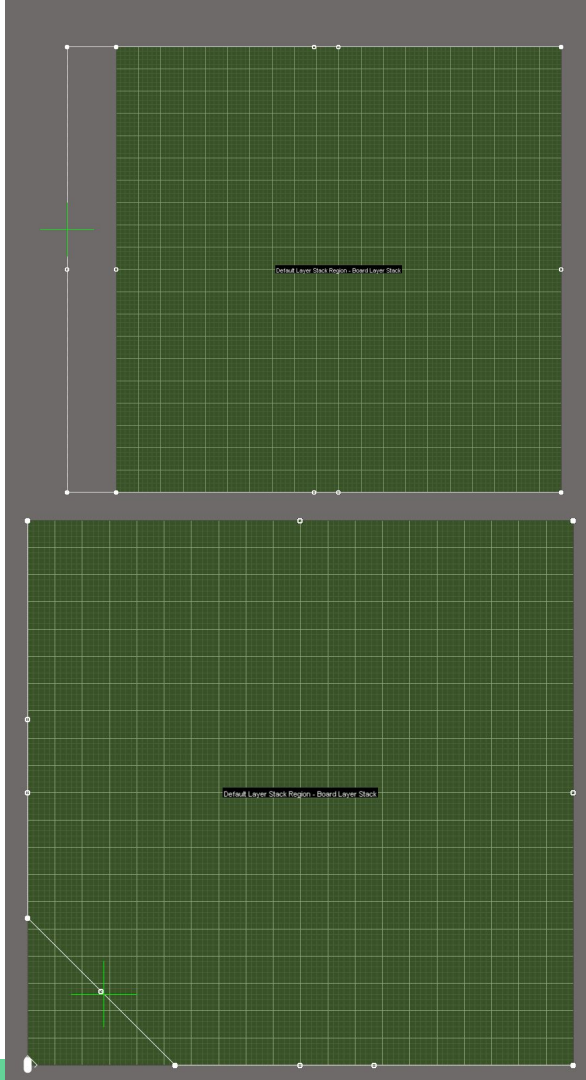
The black colored area is the original PCB, the light green area will become the new PCB



Edit Board Shape (1, D, D)

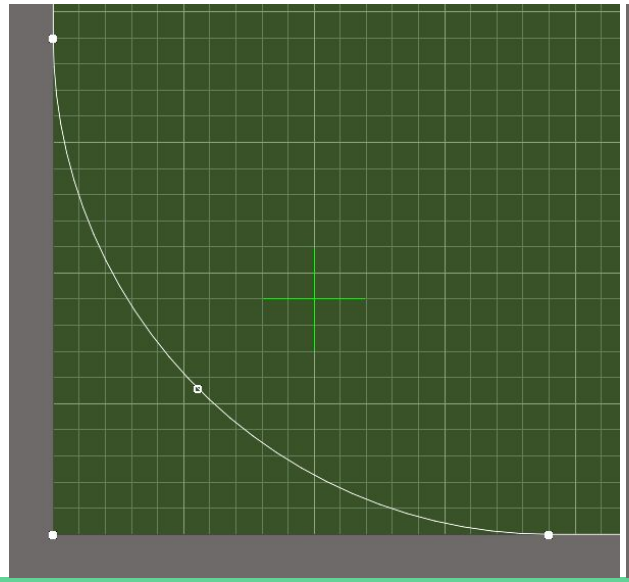
Edit the board shape with Design -> Edit Board Shape

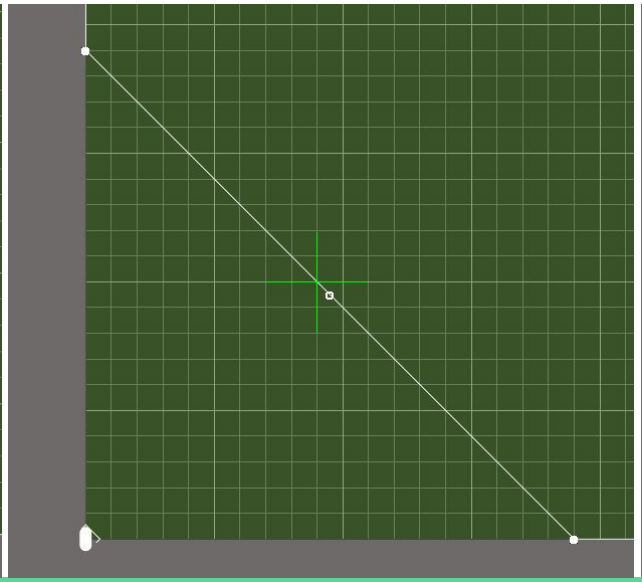
The board shape could be edited by dragging the edge and vertex around

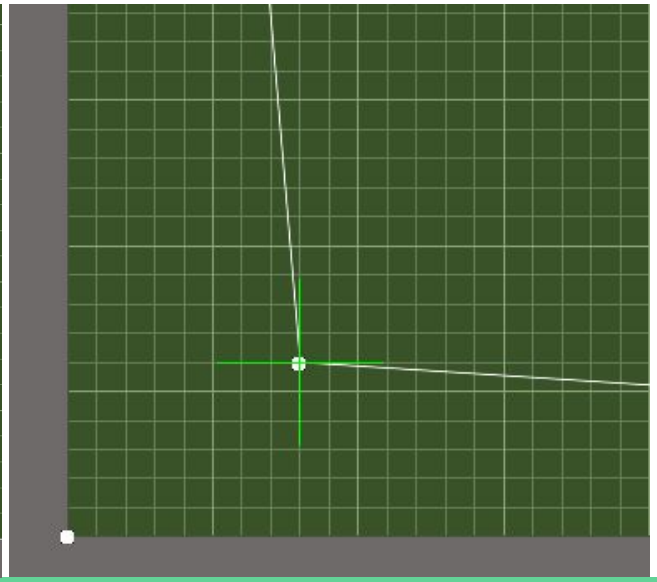


Edit Board Shape (1, D, D) (Vertex)

When dragging the vertex, press (Shift + Space) to transit from different modes.
There are 3 different modes as shown in pictures :

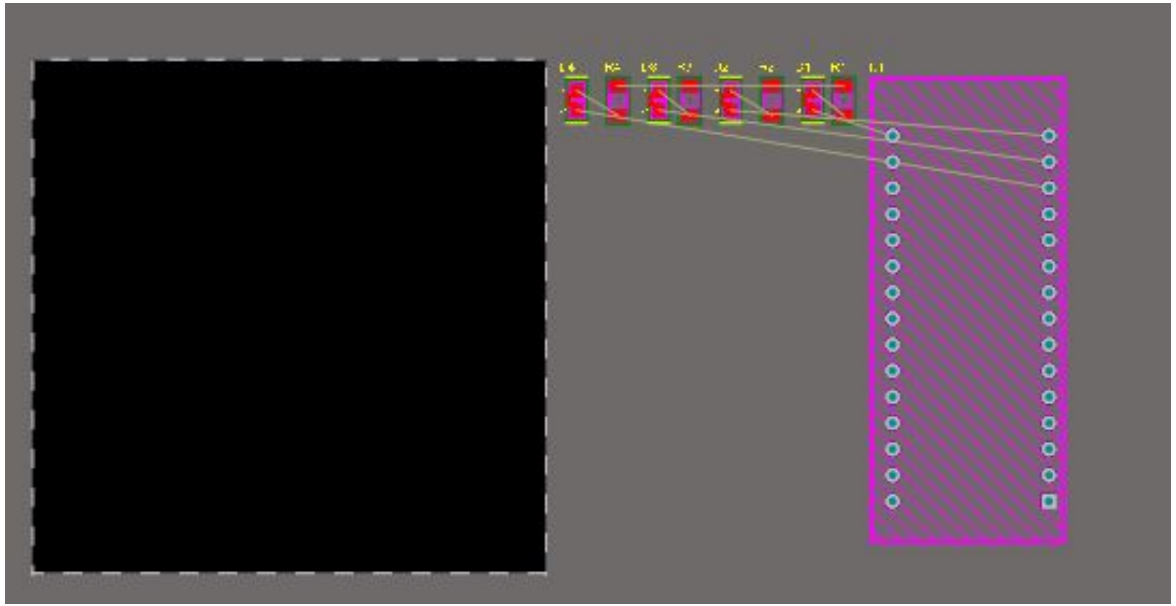






Create a 50mm x 50mm Board

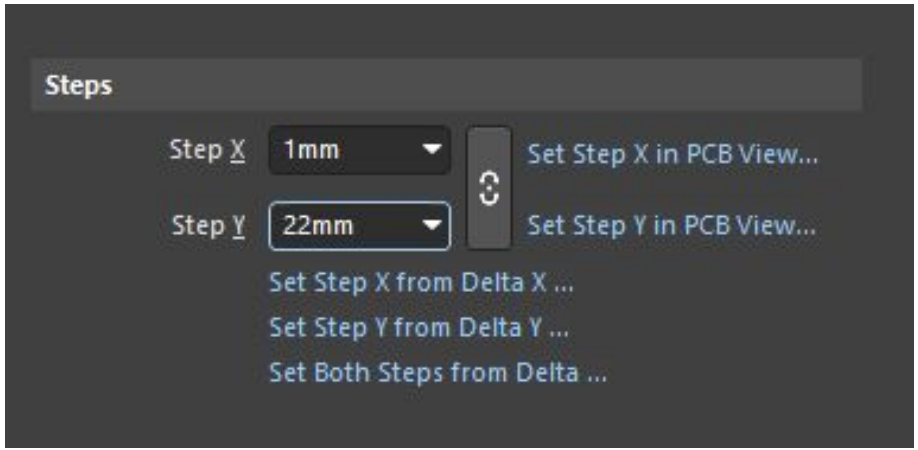
For simplicity, first set the grid size to 50mm x 50mm in 2D Layout Mode. Then enter Board Planning Mode (1). Use Redefine or Edit Board Shape command to resize the PCB to 50mm x 50mm shape



Grid Size (Ctrl + G)

The grid could be asymmetric when editing be unselecting the link icon

Sometimes used to define a PCB with specific shape (for example you could set x to 35mm, y to 111mm when the requires you to make a PCB with specific dimension)



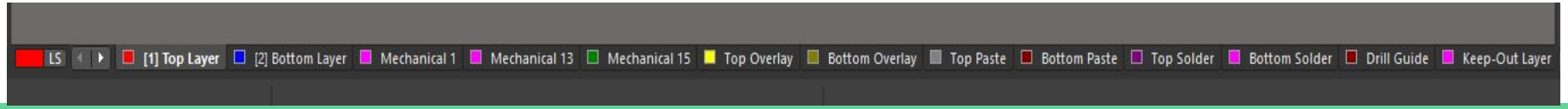
PCB

PCB Layers and their Usage
View Configuration

PCB Layer

There are multiple layers inside a PCB file, Check the tabs at the bottom to see all the layers.

Change between layers by clicking LMB on the tab or use (Ctrl + Shift + Scroll Wheel) to scroll through the layers



PCB Layers (Function)

In this example, we are going to make a 2-layer board. Layers that we mainly focus on are:

Top Layer, Top Solder, Top Paste, Top Silkscreen, which are related to objects on top

Bottom Layer, Bottom Solder, Bottom Paste, Bottom Silkscreen , which are related to objects on bottom

Keepout Layer: For defining the board shape

Top / Bottom Layer

Those are the copper layers in the PCB, used to connect the components together.

In the manufactured PCB, the coppers in top layer are usually covered by solder mask (the green paint in the picture)

Top / Bottom Solder

The solder layers is used to define the solder mask, which is used to indicate where the solder mask should not be exposed.

Where there is Solder on top of the copper, that area of copper is exposed in the final PCB

Usually the solder mask of a pad is larger than the copper

Top / Bottom Paste

The paste mask defines the stencil for that PCB, which is used when applying solder paste onto the PCB

The paste pad is usually smaller than the copper pad.

Don’t really need to pay attention to unless you plan to use stencil + solder paste to mass produce your PCB

Top / Bottom Silkscreen

Silkscreen are the white words on the PCB, which serves several functions

- Designator
- Information about the PCB (Name and designer)
- Debug Information
- Logos
- etc.

Keepout Layer

Define the outline of the PCB

Remember to add keepout layer along the side of the PCB

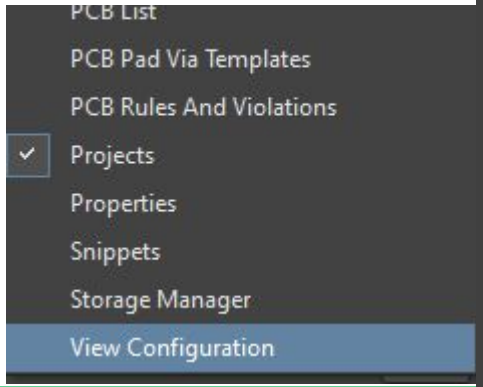
View Configuration

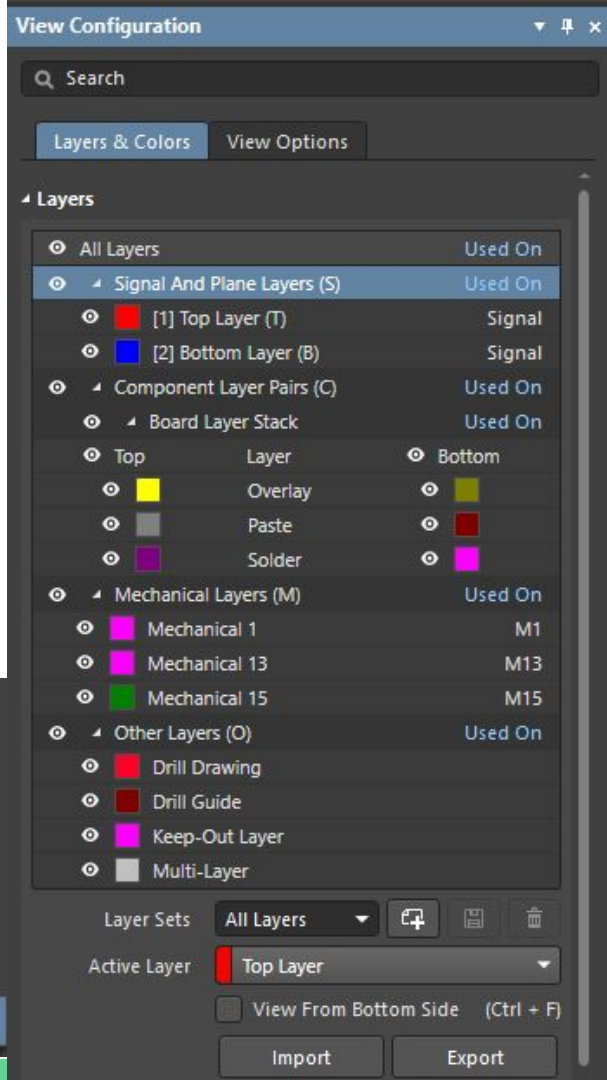
Open the tab from “Panels”

The visibility of different layer could be changed by pressing the “eye” icon.

Enable and disable their visibility at your own choice

Use (Shift+S) to toggle single layer mode





PCB

Traversing the PCB
Select, Drag, Move, Copy/Paste objects
Change Location of Component

Traversing the PCB

Same as Schematic

Zoom:

- (Ctrl + mouse wheel)
- Pushing the mouse forward or backward when holding middle mouse button

Pan

- Holding RMB while moving the mouse
- Using mouse wheel to move up and down
- Using (Shift + mouse wheel)) to move left and right

Drag/Move of Objects in PCB

The Drag, Move and Selection of Objects in PCB is same as schematic

The shortcuts are also the same

When moving the the objects around, it will try to snap to the closest grid

Set the grid size larger when doing rough placement of PCB (something like 1mm or 2mm), and reduce it back to 0.1mm to 0.2mm for fine adjustments

Drag/Move of Objects in PCB

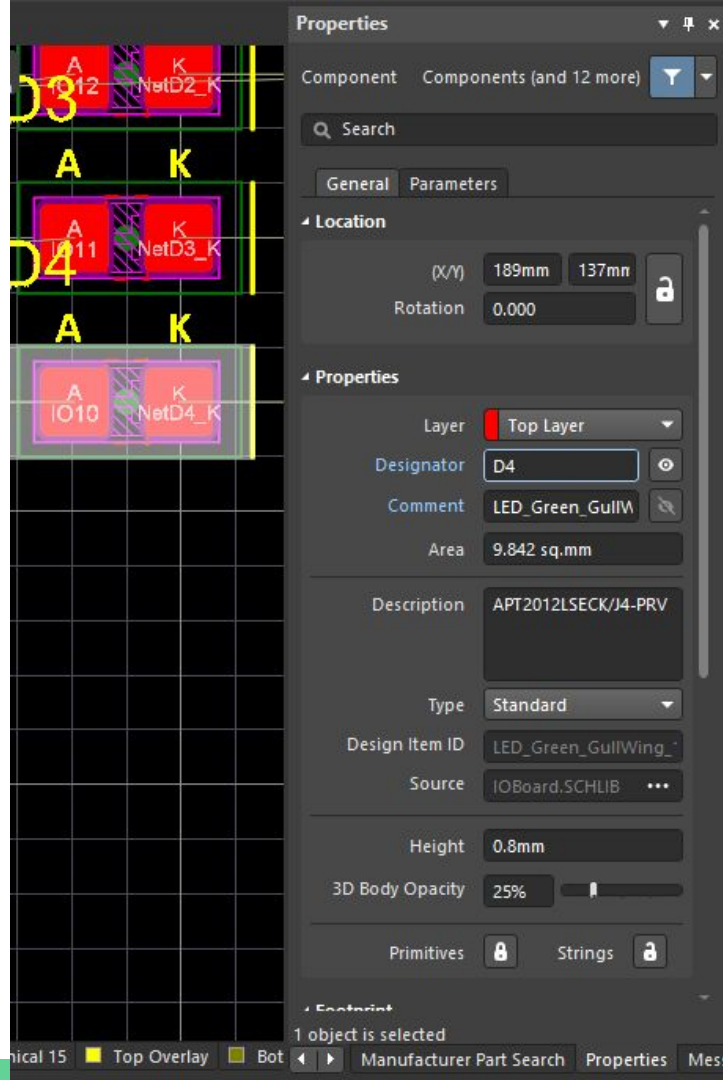
Currently most components should be outside of the PCb

Component Property

Double click LMB on the component, the property tab will display properties of that component

Most parameter SHOULD NOT be modified in the PCB.

The location parameter could be edit to move the components directly to that coordinate.



PCB

Design Rules

Design Rules

Design rules are rules that govern the PCB

A good designer should always define the rules correctly, and ensure that the PCB design follows the rules.

Refer to the manufacturing capability of the manufacturer when defining the rules

https://jlcpcb.com/capabilities/Capabilities

Design Rules

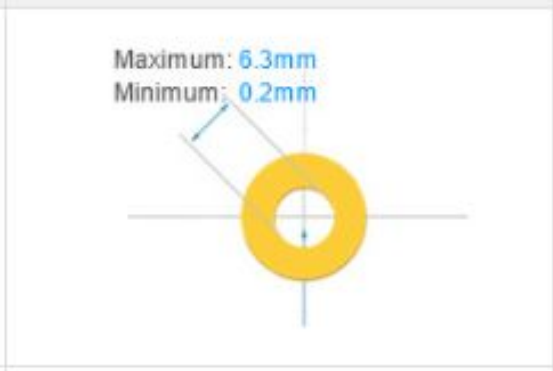
If the rules is not what you want, it could be edited from Design -> Rule (D, R)

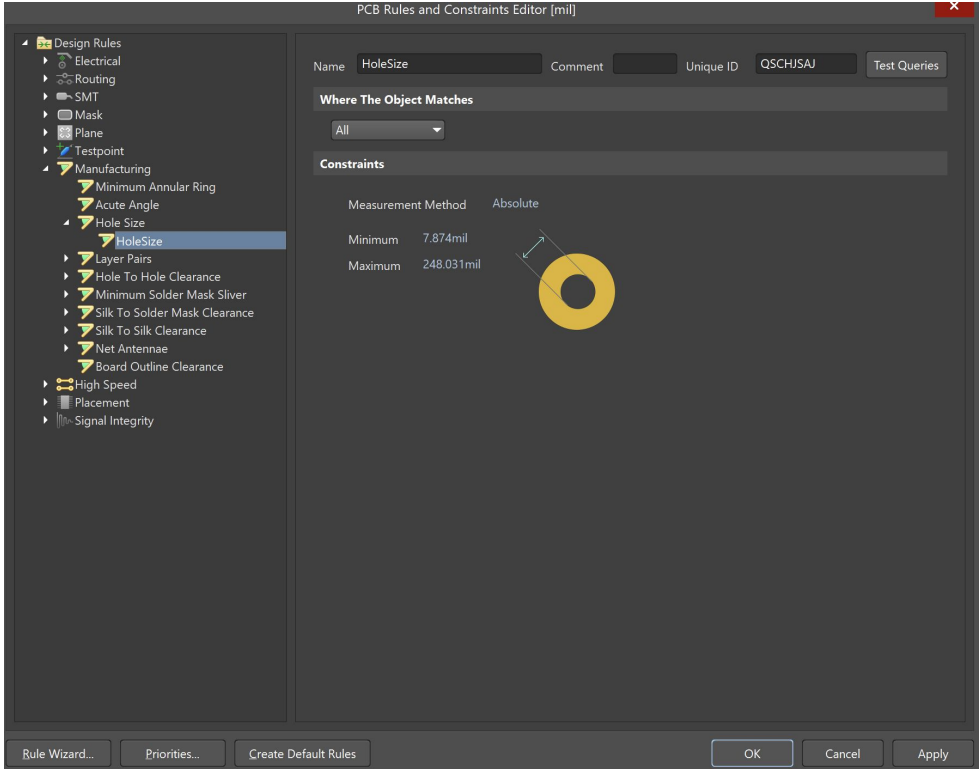
This powerpoint will show how to edit Hole Size, Wire Width and Polygon Connection Style in Design Rules.

Design Rules (Hole Size Constraints)

The default Hole Size is 2.54mm

Change it to max to 6.3mm and min to 0.2mm because of JLC's limit



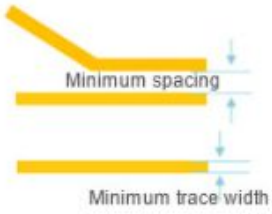


Design Rules (Wire Width and minimum clearance)

The minimum is 0.127mm, but it is recommended to use thicker wires like 0.2mm

For the max width, it could be set to any value you like, but if you need to use width like 1mm or 2mm for high current, please consider using polygon to connect them instead

Minimum trace width and spacing –

Copper weight	Min. Trace width	Min. Spacing	Patterns
H/HOZ (Inner layer)	5mil (0.127mm)	5mil (0.127mm)	
1oz (Outer layer)	1/2 layers: 5mil (0.127mm) 4/6 layers: 3.5mil(0.09mm)	1/2 layers: 5mil (0.127mm) 4/6 layers: 3.5mil(0.09mm)	
2oz (Outer layer)	8mil (0.2mm)	8mil (0.2mm)	

PCB Rules and Constraints Editor [mm]

Search

- Design Rules
  - Electrical
  - Routing
    - Width
      - **Width***
    - Routing Topology
    - Routing Priority
    - Routing Layers
    - Routing Corners
    - Routing Via Style
      - RoutingVias
    - Fanout Control
    - Differential Pairs Routing
  - SMT
  - Mask
  - Plane
  - Testpoint
    - Fabrication Testpoint Style
    - Fabrication Testpoint Usage
    - Assembly Testpoint Style
    - Assembly Testpoint Usage
  - Manufacturing
    - Minimum Annular Ring
    - Acute Angle
    - Hole Size
    - Layer Pairs
    - Hole To Hole Clearance
    - Minimum Solder Mask Sliver
    - Silk To Solder Mask Clearance
    - Silk To Silk Clearance
    - Net Antennae
    - Board Outline Clearance
  - High Speed

Name: Width | Comment: | Unique ID: CWUBPODO | Test Queries

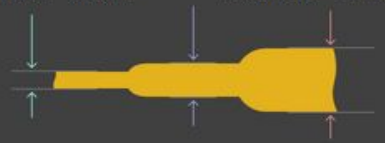
Where The Object Matches

All

Constraints

Preferred Width 0.2mm

Min Width 0.2mm Max Width 2mm



- (•) Check Tracks/Arcs Min/Max Width Individually
- ( ) Check Min/Max Width for Physically Connecte
- [ ] Use Impedance Profile

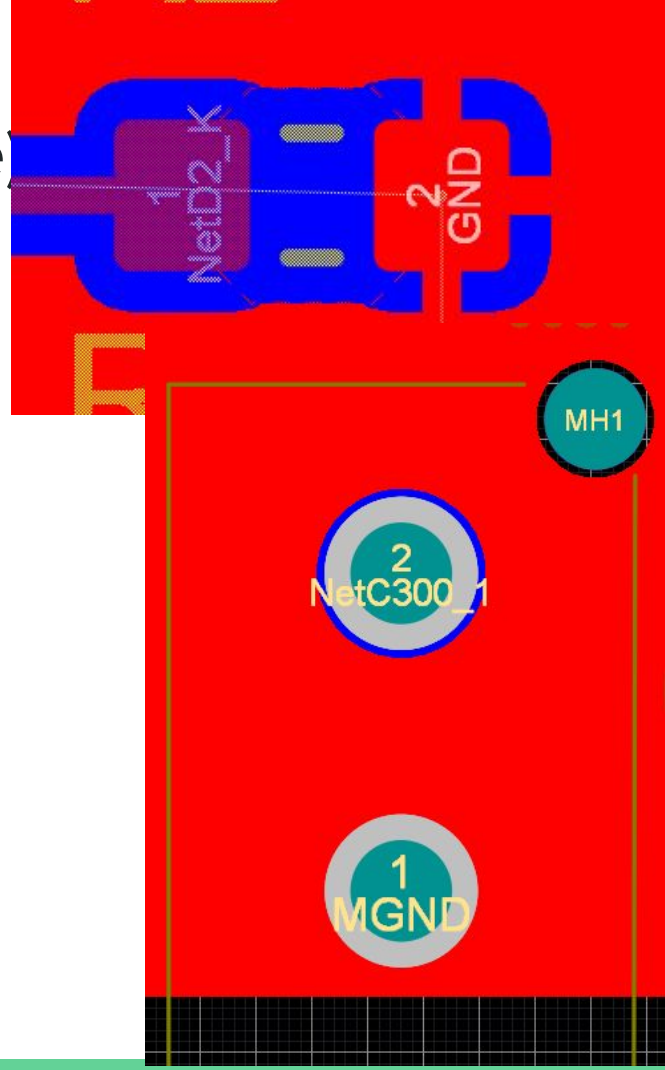
Min Width	Preferred Width	Max Width	Layer Name
0.2mm	0.2mm	0.254mm	1 - Top Layer
0.2mm	0.2mm	0.254mm	2 - Bottom Layer

Switch to Document View | Rule Wizard... | Priorities... | Create Default Rules | OK | Cancel | Apply

Design Rules (Polygon Connection Style)

The polygon in the PCB is connected to the pad of the component with 4 thinner wires. It is called the thermal relief connection. (Top Picture)

For polygons that you expect it will carry more current, the connection style should be changed to direct connection in the rules (Bottom Picture) This is not used in this lab



Design Rules

There are tons of different types of rule in

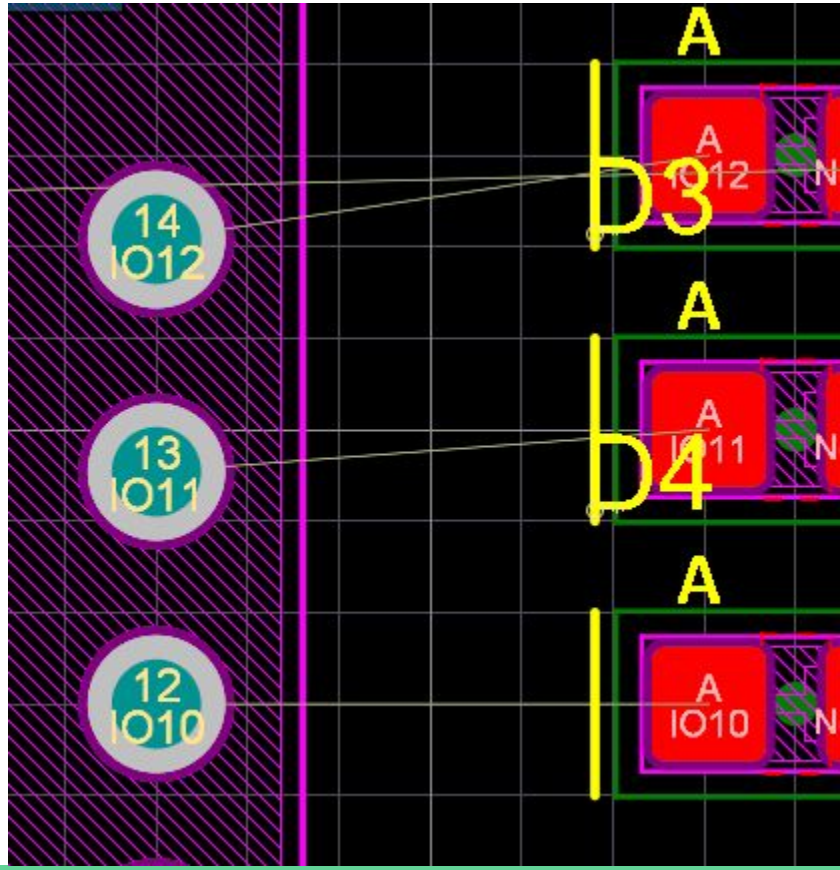
PCB

Grey Lines and Routing
Wire
Polygon
Mounting Holes

Grey Lines

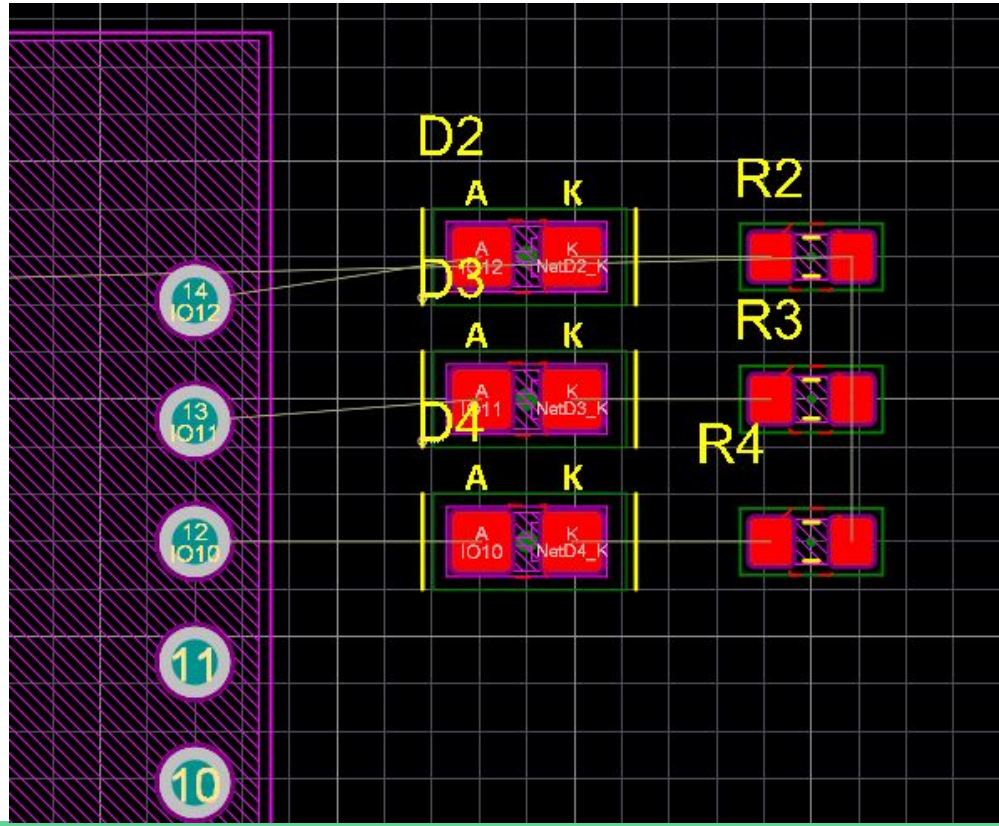
The grey lines shows where the pad connect to, it helps to show which components should be grouped together.

Place the component with the guidance from the grey lines



Grey Lines

The more tidy the grey line is, the more easier the routing will be, since when you start routing your PCB, most of the real wires will be drawn close to the grey wire



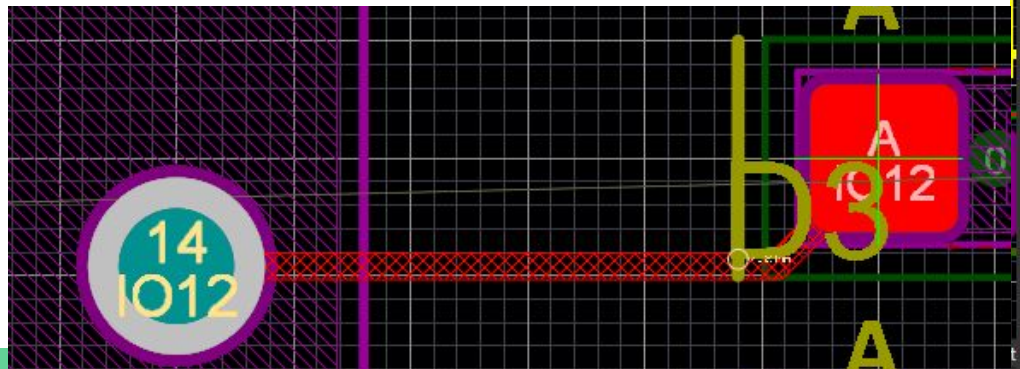
Add an overview of the components placement here!!

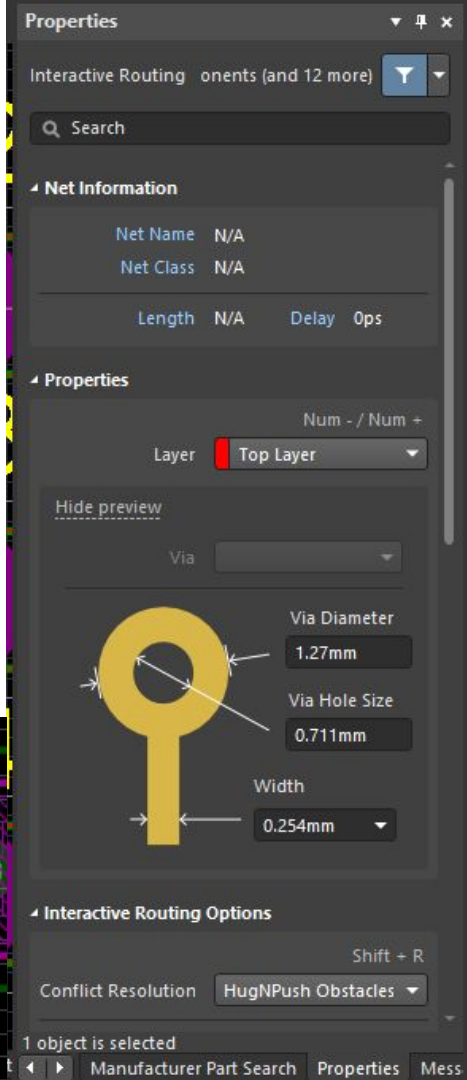
Routing the PCB (Ctrl + W)

Use (Ctrl + W) to enter the Interactive Routing Command

Press (Tab) to pause to edit property before placing wire.

Click LMB on the start point (Must start on a PAD of a component or Vias). In this example, click on the Arduino nano's IO12 pad first



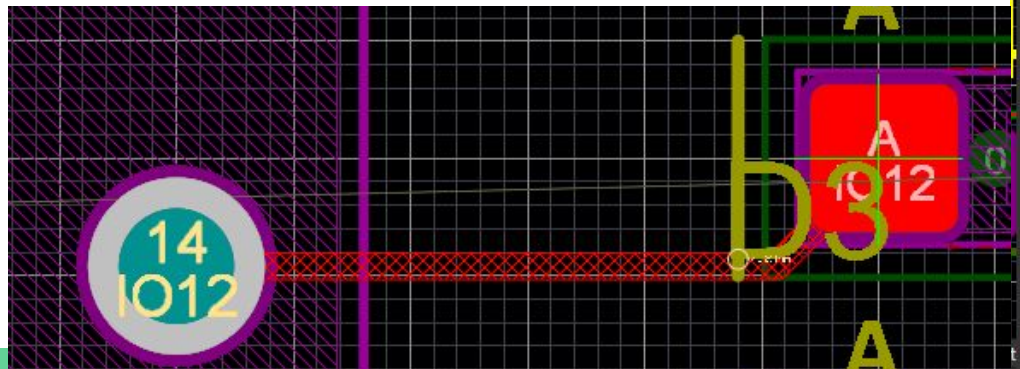


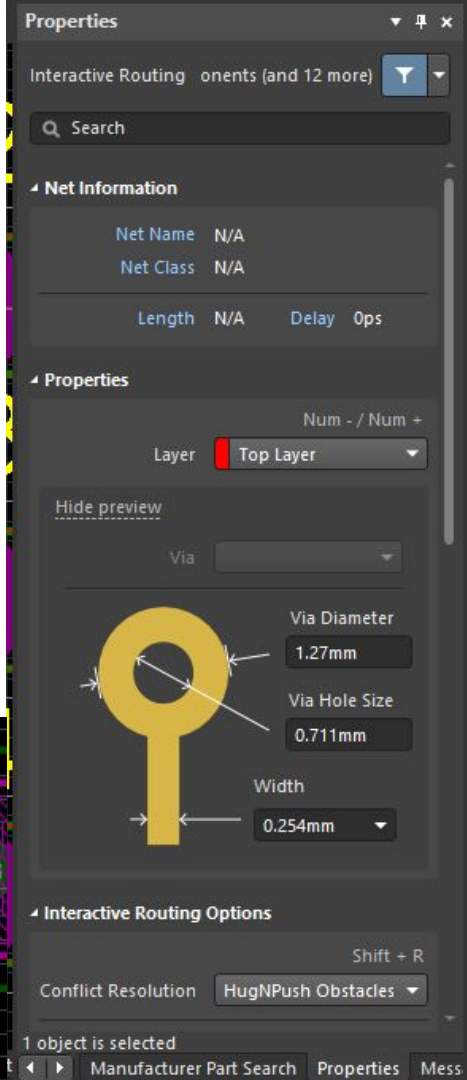
Routing the PCB (Ctrl + W)

You will see a semi-transparent line when you move the cursor around.

You would click LMB to set way points, Click RMB to escape Interactive Routing Mode

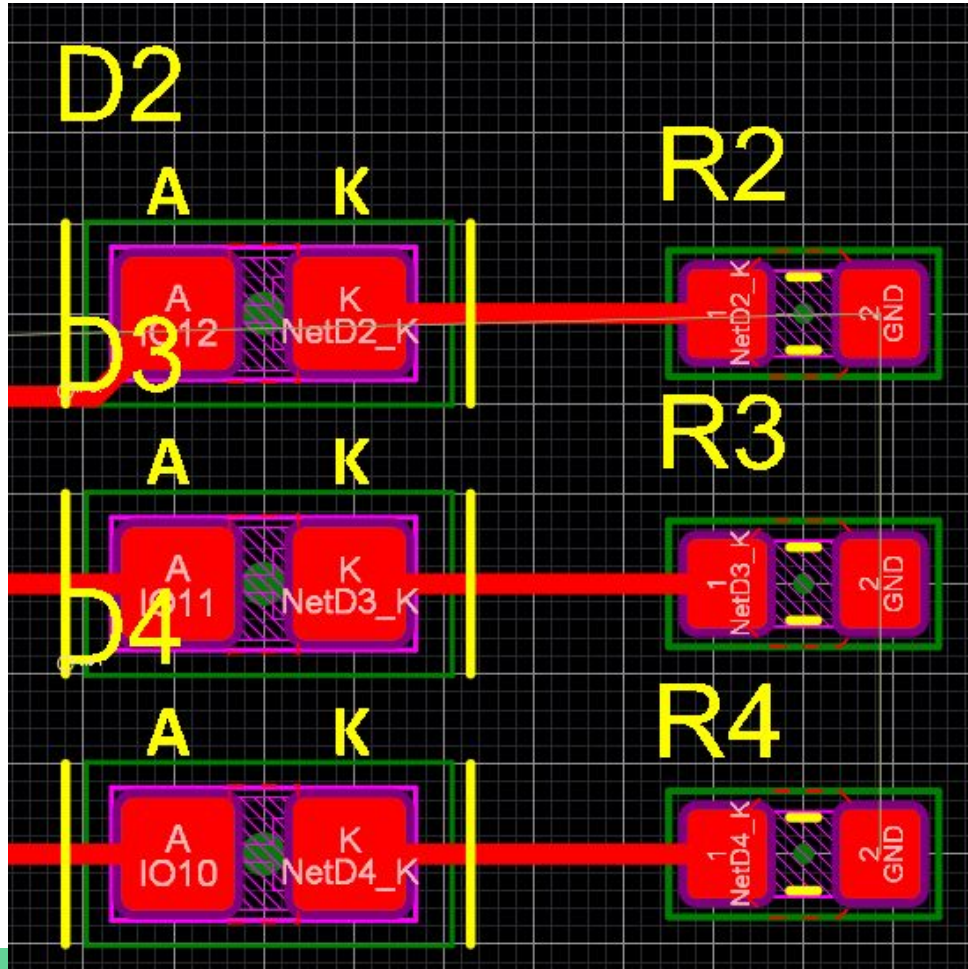
In this example, move your cursor to the pad of the diode, named IO12 and click LMB





Routing the PCB (Ctrl + W)

Route the PCB except for the pads with Net “GND”



Routing the PCB with Auto Route (Advance)

Autoroute could be performed when the trace is short

(Ctrl + LMB) on the pad of the component that you want autoroute to be performed

If Altium fails to autoroute the connection, you would hear the system notification sound.

Only use autoroute when the trace is short and direct

GND Plane (Polygon)

For connecting GND, a GND plane is usually used

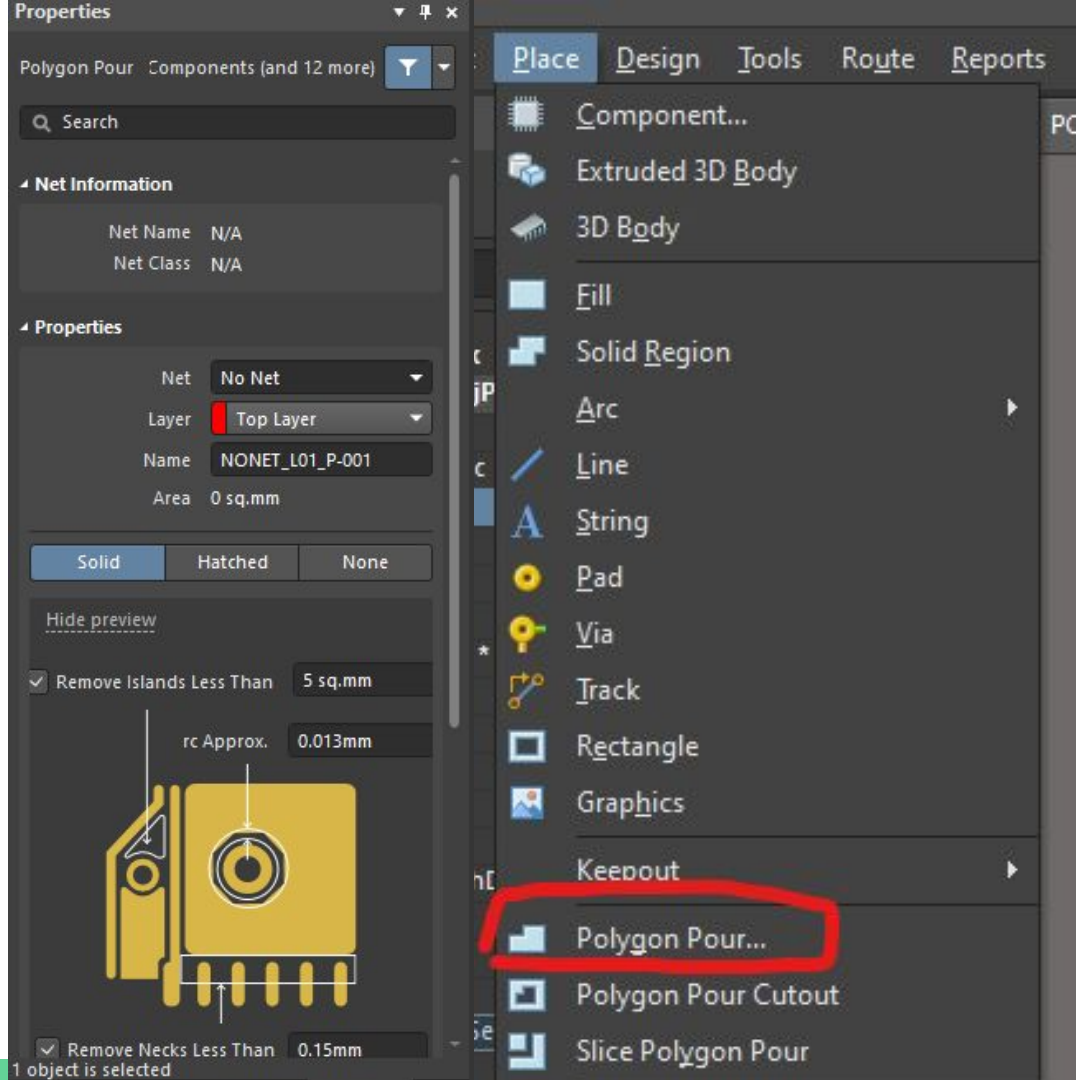
GND plane could be created with Polygon Function

Place Polygon (P, G)

Make sure you are on Top Layer to create a polygon on Top Layer

Select the command from the dropdown menu or click (P, G)

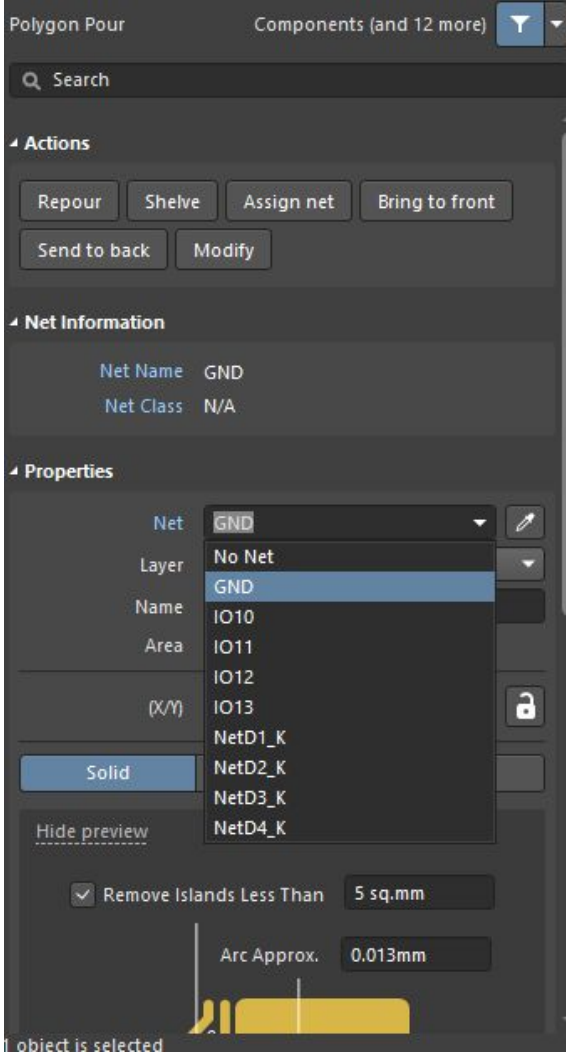
You will see the property panel's content changed into this



Place Polygon (Edit Property)

Default, the polygon is not associate to any Net

Edit the Net from “No Net” to “GND”

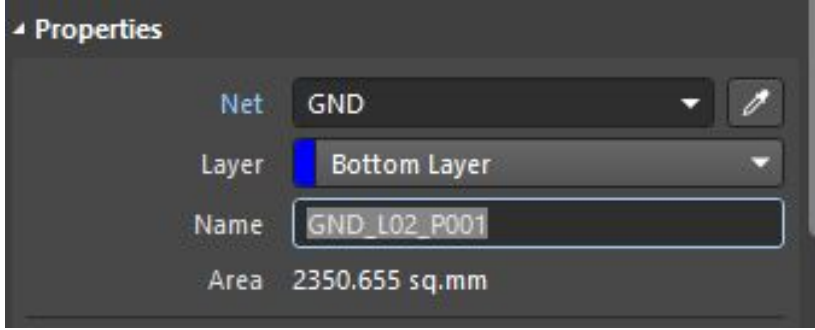


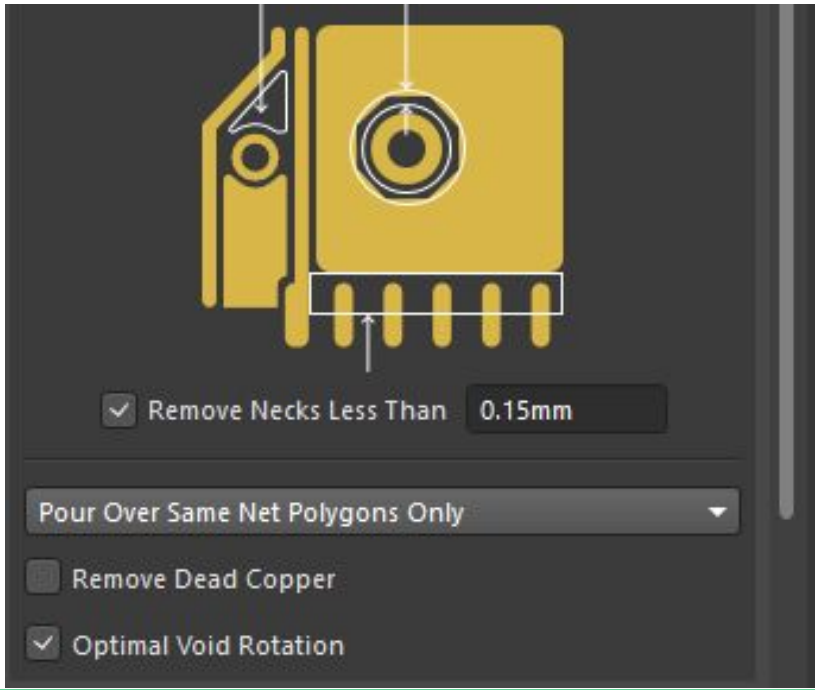
Place Polygon (Edit Property)

You could assign a unique name in “Name” to the polygon or let it keep its default name

You could change the Layer of the Polygon by changing the “Layer”, but for this one, we should keep it at Top Layer

Ensure the “Pour Over Same Net Polygon Only” is selected, other options are usually not used in simple projects





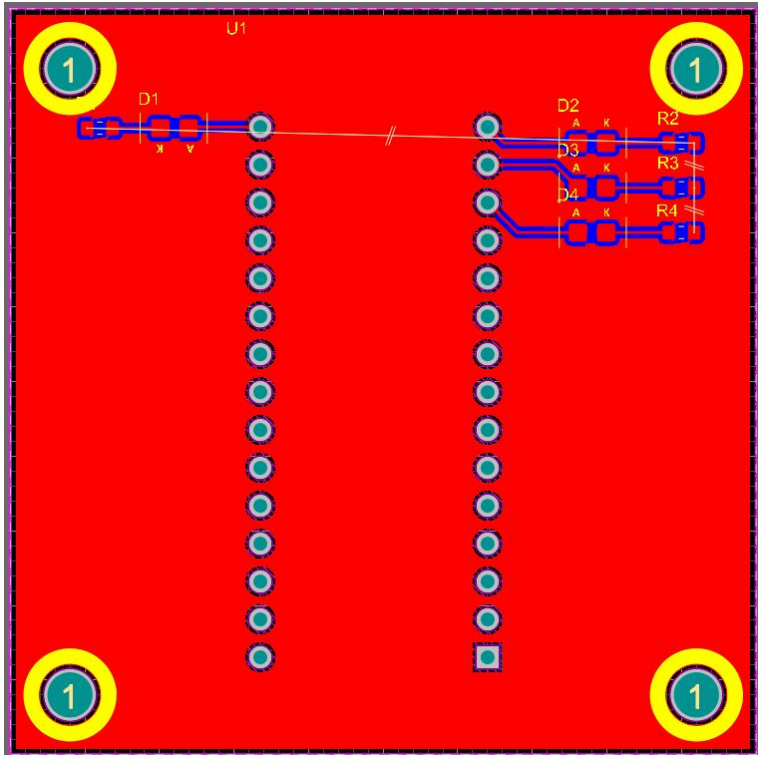
Place Polygon

Click LMB to form the vertex of the Polygon

Create a Polygon to cover the entire PCB

The polygon will connect to pads with the same Net, while bypassing the others

When you are finished, you should see a big red area on the PCB

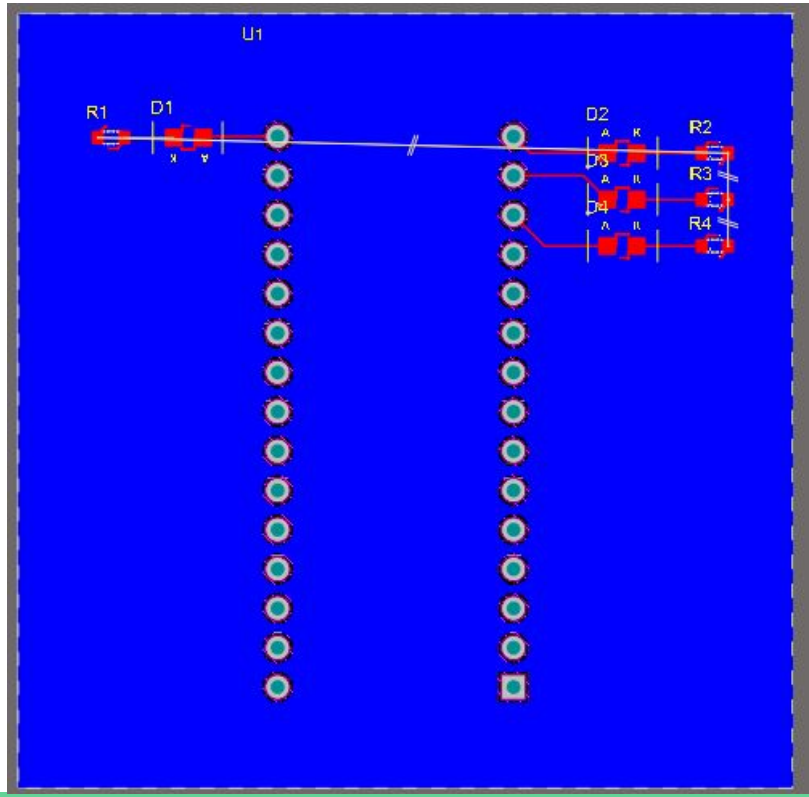


Add polygon

Repeat the step, but this time create a Polygon with Net GND to the bottom of the PCB.

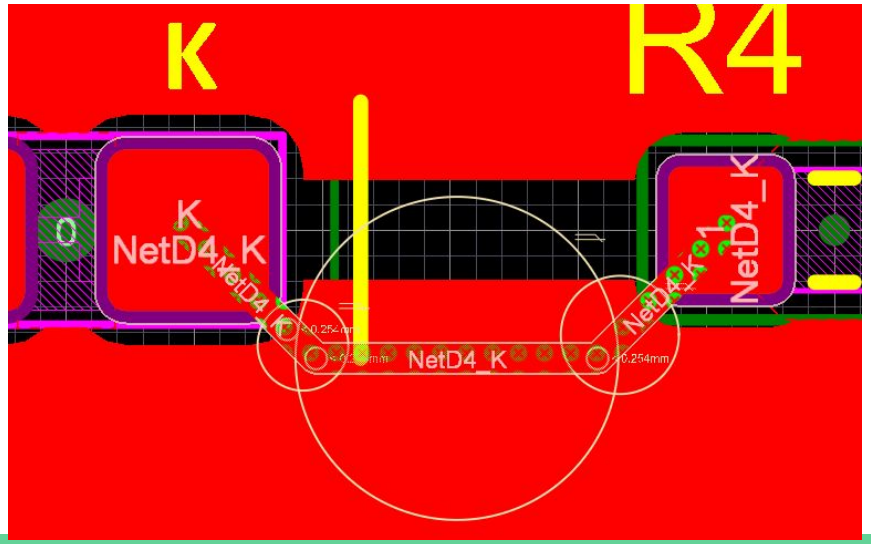
A Blue Polygon should appear if it is added to the bottom layer

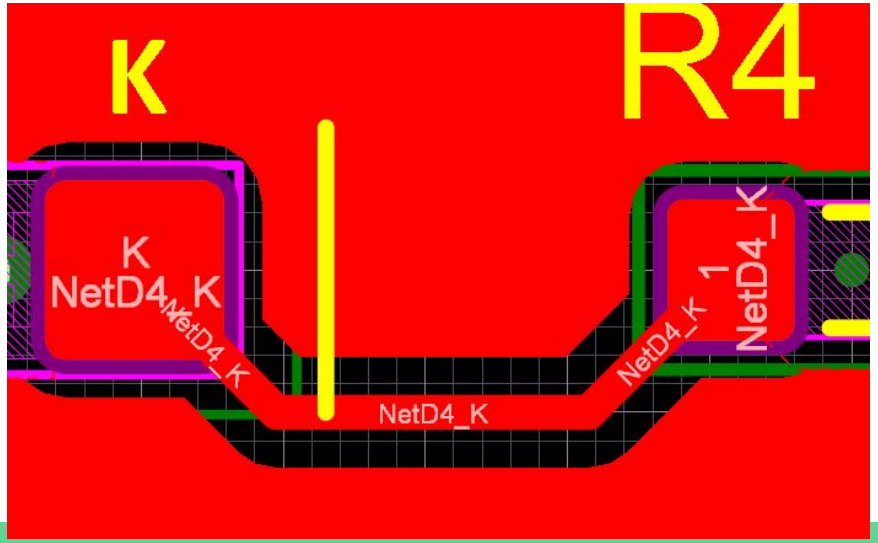
Now you should have 2 polygons, one at top layer, one at bottom layer, both with GND



Polygon repour

After modifying the connection after adding the polygon, the wire could be be contact with the polygon, use Polygon repour to regenerate the polygon according to the new placements.



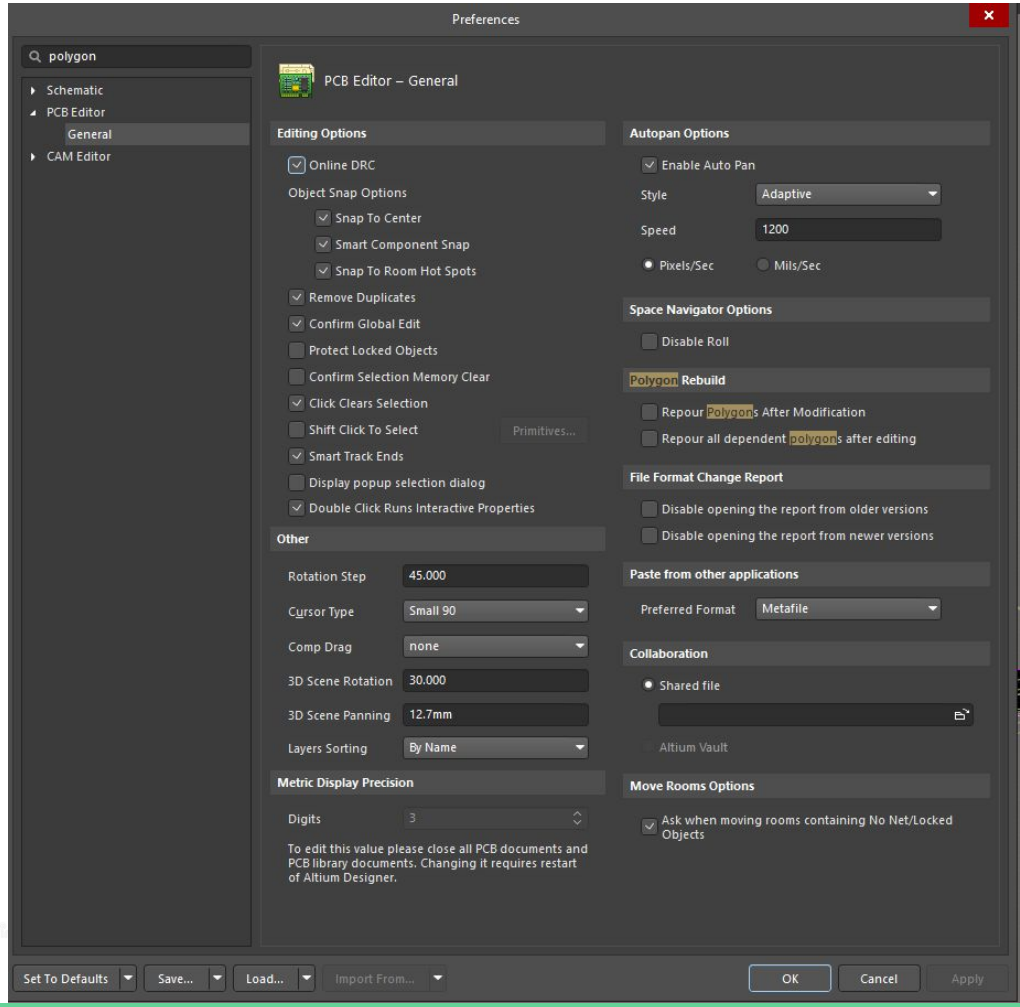


Repour Polygon

Moving the components / connections will usually lead to them overlapping with the poured polygon

Click (RMB, Y, A)) to use the “repour all” command

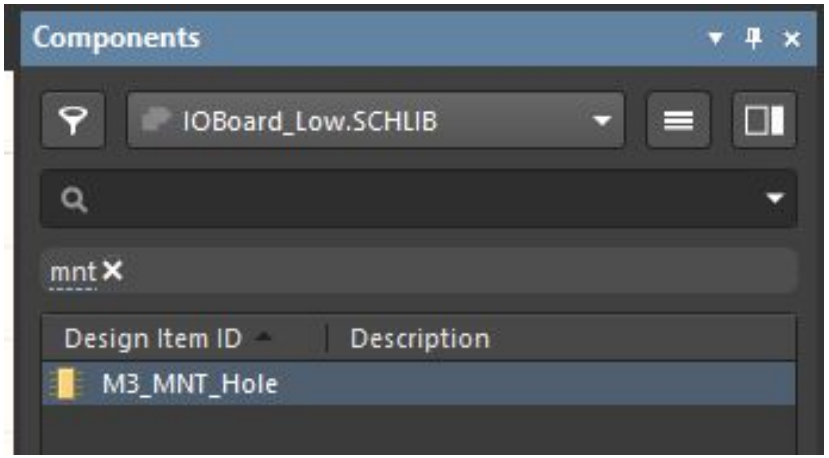
You could also change the Preference (The Gear Icon on top left corner) to Auto Repour when something is edited in the PCB, but it would use more CPU resource

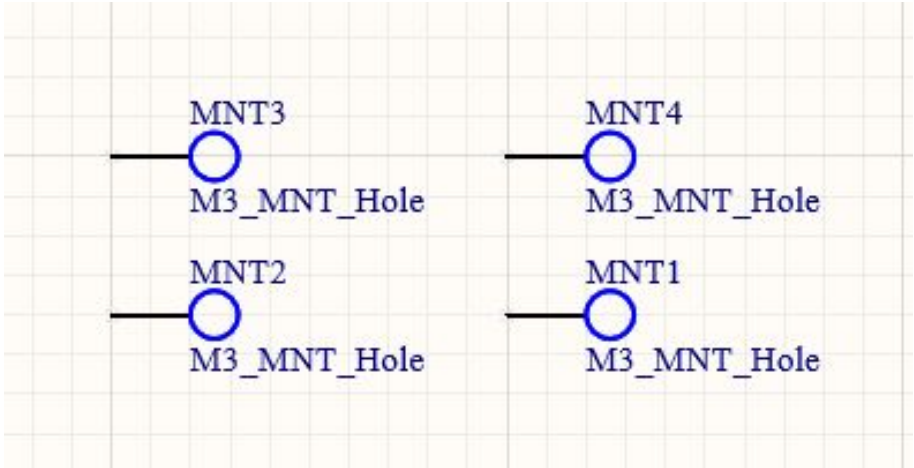


Mounting Holes

At the end, remember to add mounting holes to the PCB to provide spaces

Go back to the schematic file and search “mnt” in IOBoard_Low schematic library to find the the mounting hole and place 4 mounting hole onto the schematic. Remember to annotate them.



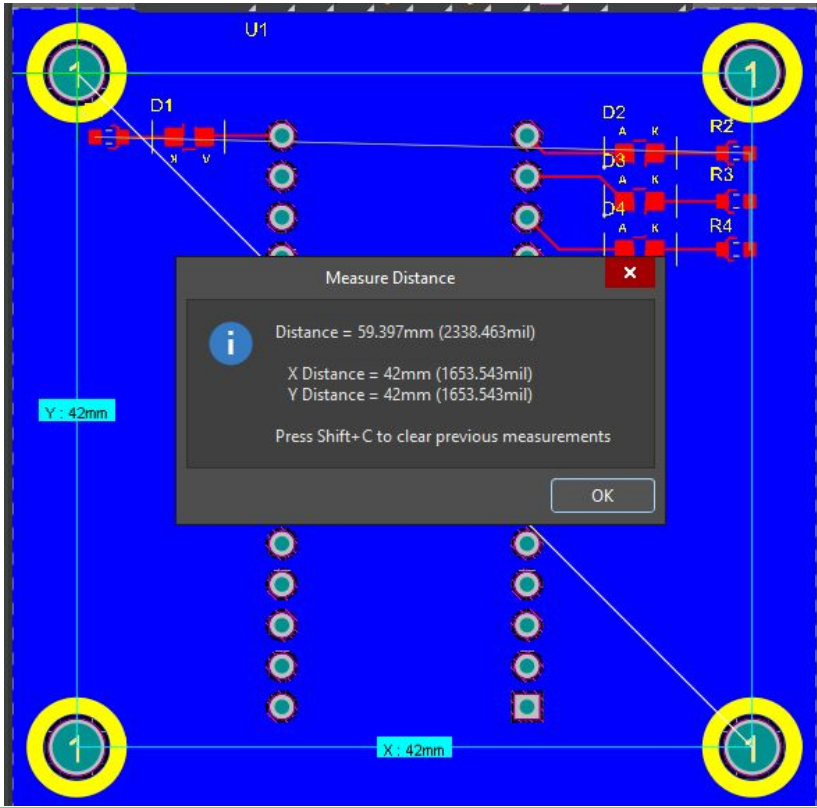


Mounting Holes

Go back to the PCB, Place the mounting hole on the four corners of the PCB

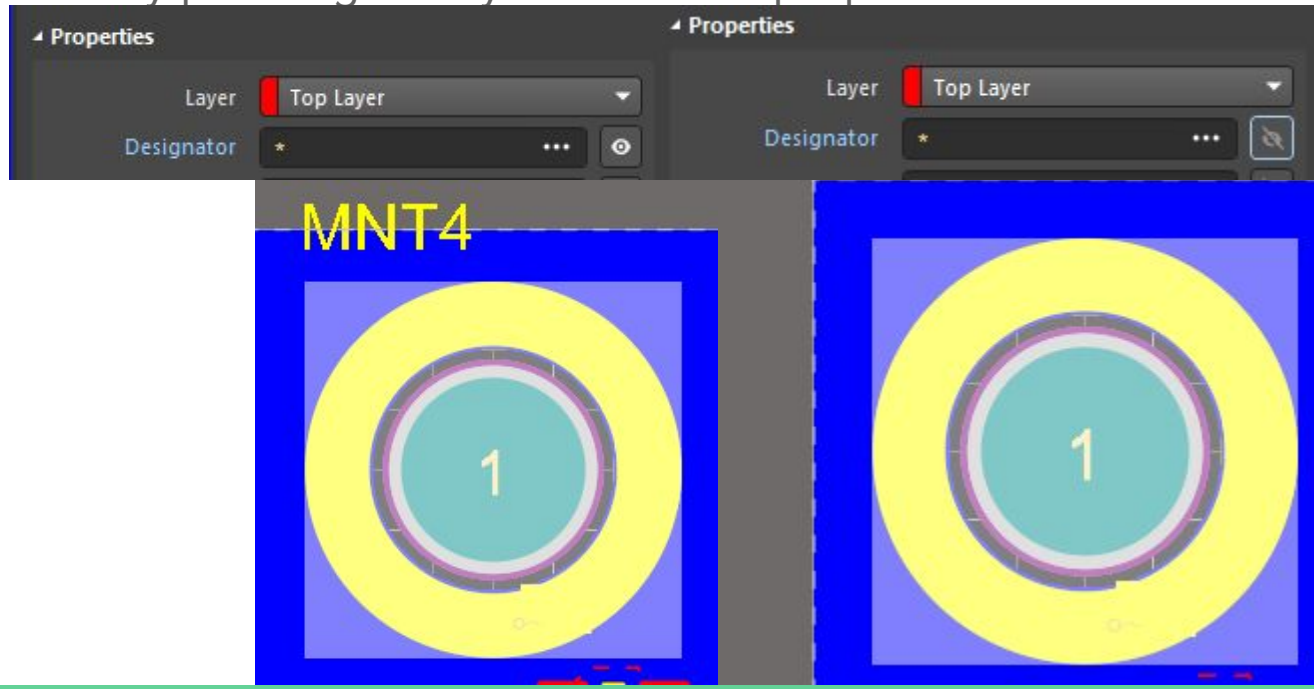
To save mechanical people from trouble, please use unit in millimeters instead of mil when placing the corners

For HKU, we used unit of 10mm when planning mounting hole to facilitate the mounting process (just need a acrylic board with holes in 10mm interval to mount everything)



Hiding Designator of Mounting Holes

Since you don’t really need to know what components to place onto mounting hole, you could hide the designator by pressing the eye icon in the properties.



Place Via (P, V)

You could use Via to connect components / wires / polygons between the layers

After using Place Via command, you could press (Tab) to pause it and edit

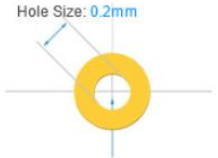
Place vias with certain interval around the PCB to ensure the two polygon is connected at multiple place.

Extra Info: have 2 ground planes that are connected by multiple via will enhance signal integrity

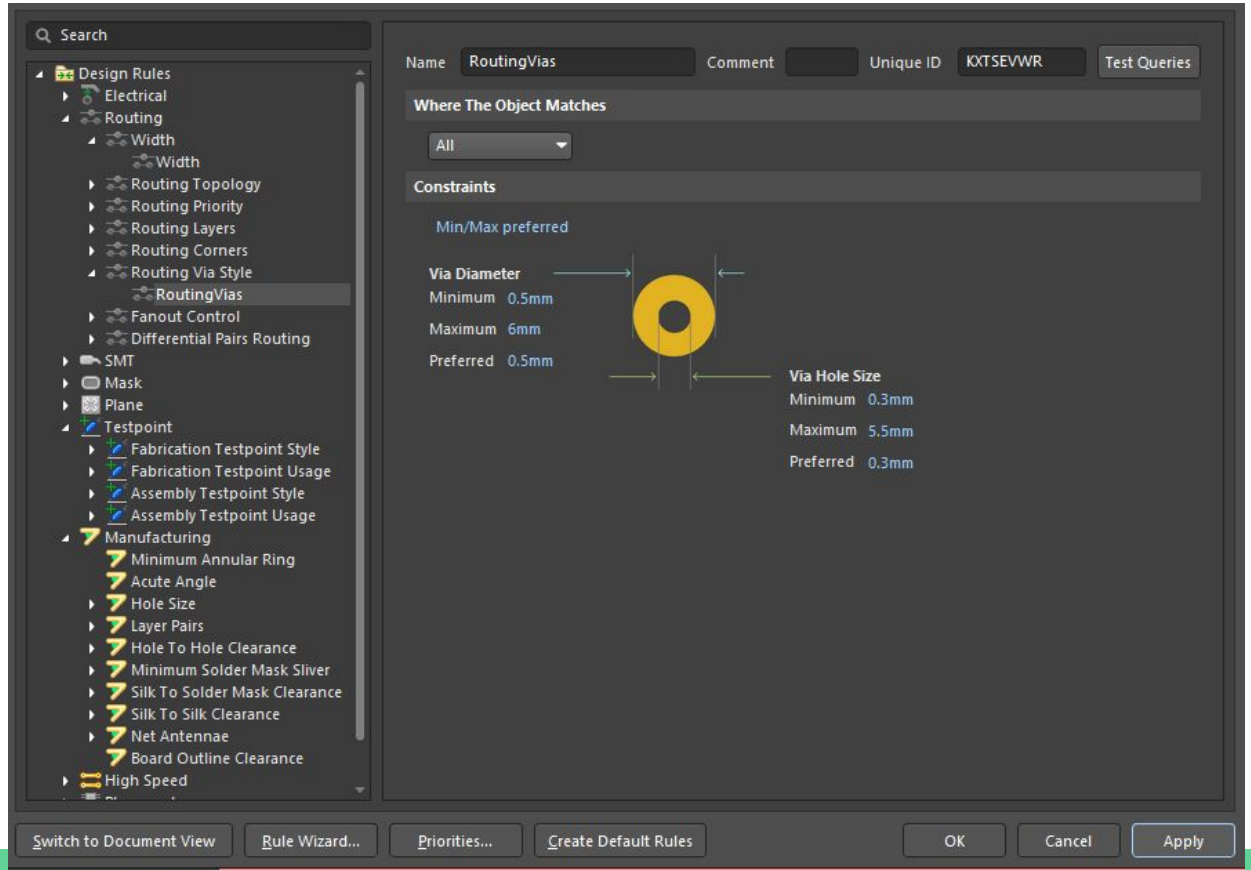
Design rules of Via

The smallest dimension for a via is list as 0.3mm for the hole and 0.5mm for the diameter for 2-layer PCB

Change it in design rules accordingly

Min. Via hole size	0.2mm	For Single&Double Layer PCB, the minimum via hole size is 0.3mm;For Multi Layer PCB, the minimum via hole size is 0.2mm	
Min. Via diameter	0.4mm	For Single&Double Layer PCB, the minimum Via diameter is 0.6mm;For Multi Layer PCB, the minimum via diameter is 0.4mm.	

Design rules for Via, open menu with (D, R) command

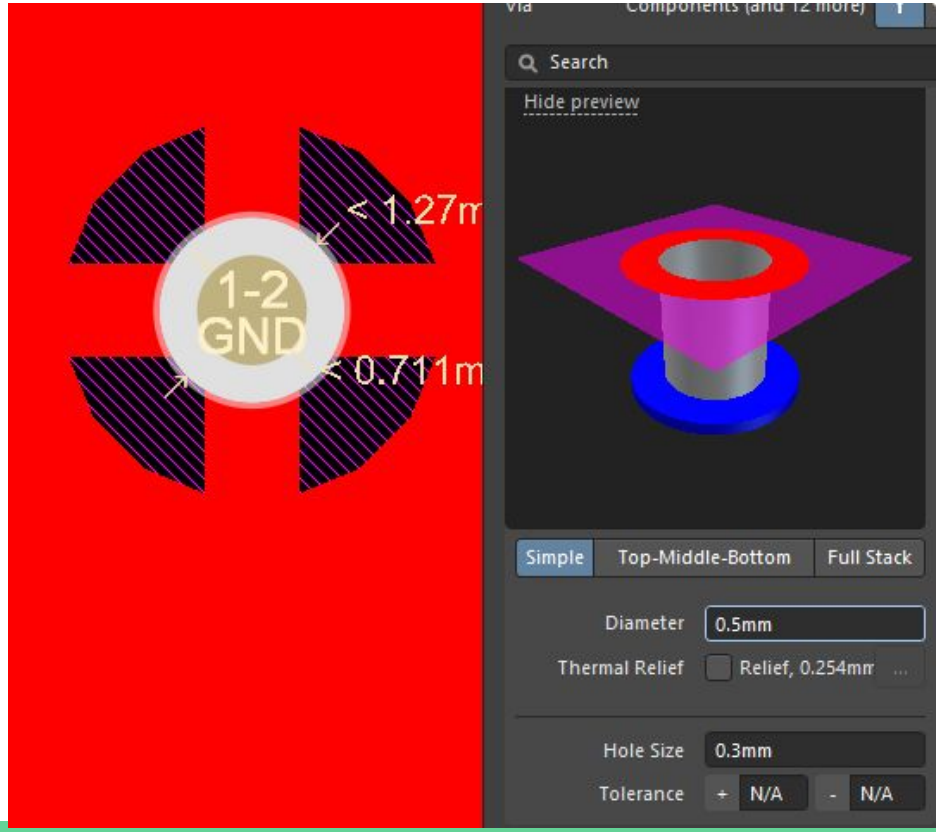


Via Properties

Diameter is the diameter of the entire via

Hole size is the diameter of the hole in the middle, set it to 0.3mm according to the rule

The diameter and hole size of the Via could be increase to allow more current to flow through

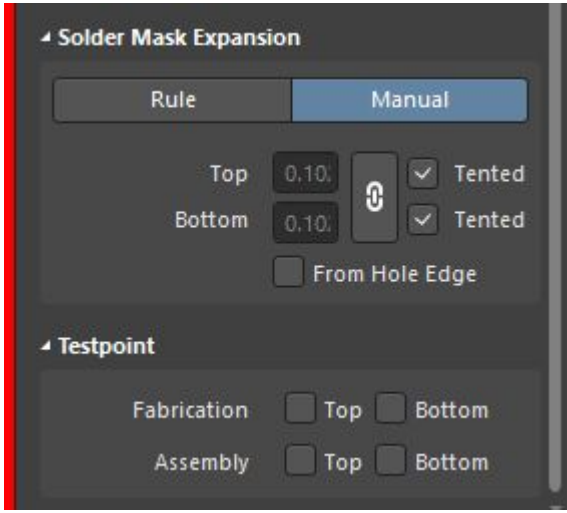


Via Properties

When you stroll down, you will see the solder mask expansion rule.

Select the tented option for both box

After adding tenting, the via will be covered by solder resist (picture on the right). Not tenting the via may lead to short circuit (picture on the left)



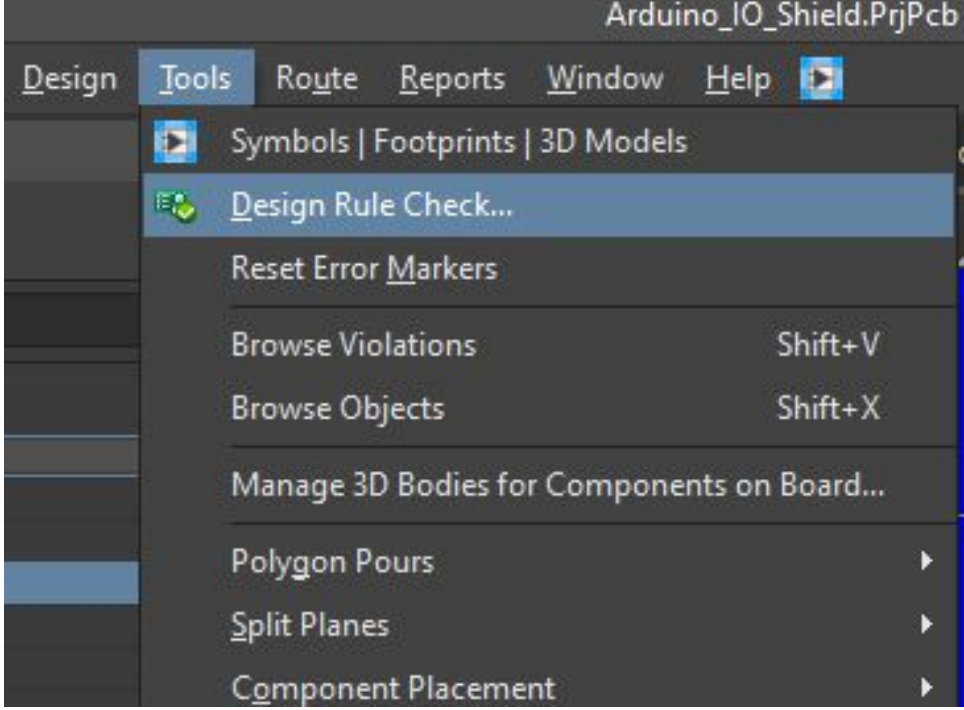
PCB

Before sending out final PCB, make sure to run DRC and place KeepOut

Design Rules Check (DRC)

Everytime when you think the PCB design is ready, run DRC to check if there is any error on the PCB, the shortcut is (T, D, R)

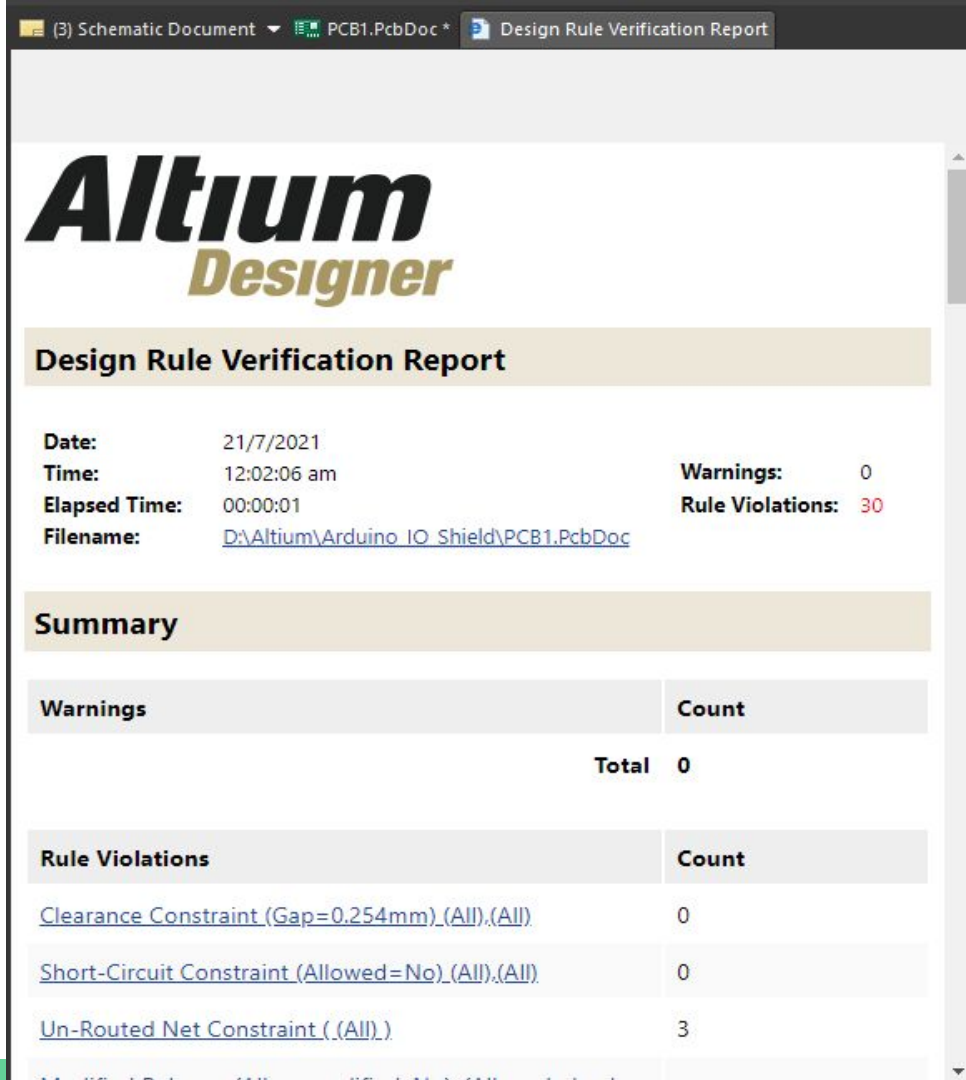
A report will be generated, look into the report and fix the errors / warnings



DRC Report

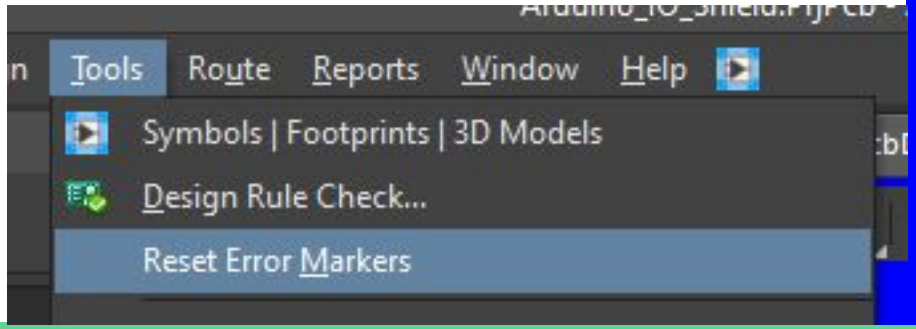
Some warnings are not critical (silkscreen clearance issue) while some are more critical errors (unrouted nets, short circuits, copper clearance issue)

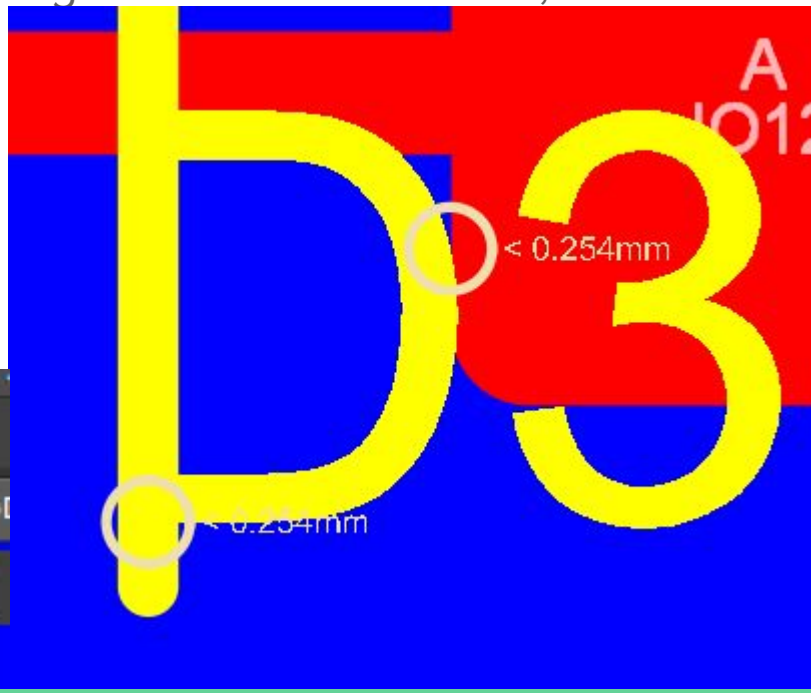
Click on the violation and it will zoom to the violation



DRC Error Marking

The error marking will show on the PCB, indicating that object has violated design rule(s). The error markings could be cleared by using “Reset Error Markers”, shortcut (T, M)





DRC

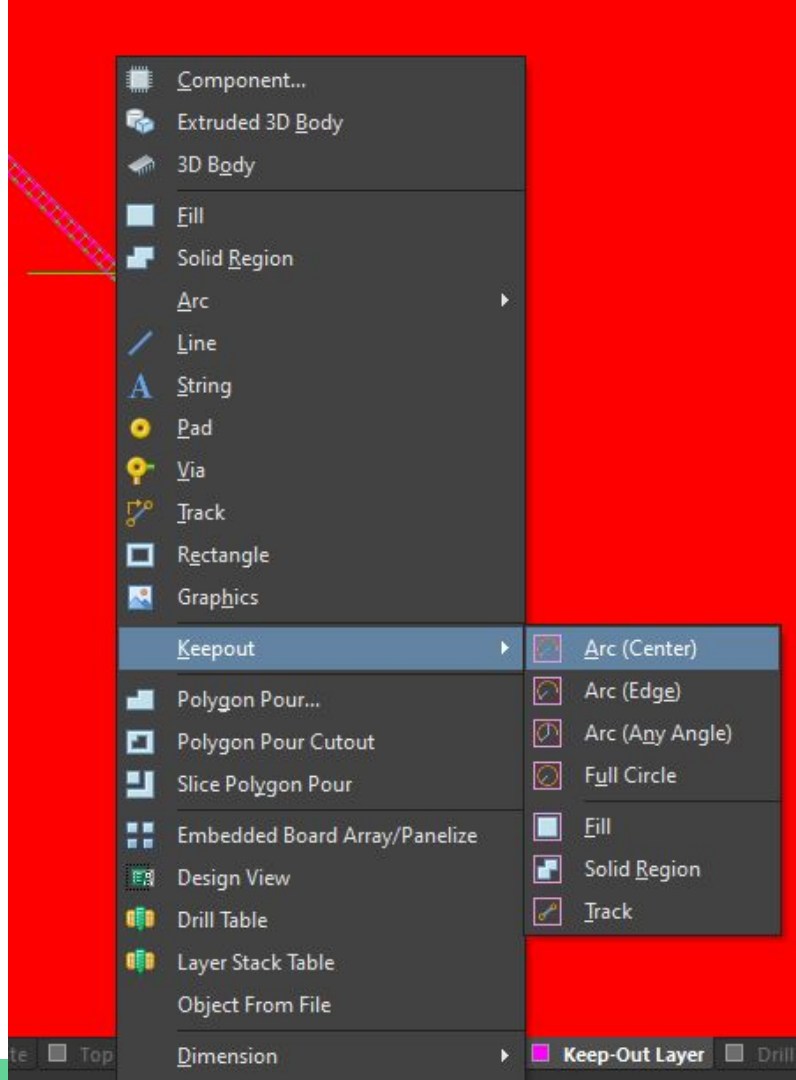
Run DRC until all the errors are gone or the errors left are violations that are minor (For example clearance issue between silkscreens, which will not affect the performance of the PCB)

Place Keepout Track

Select “Keep Out Layer”

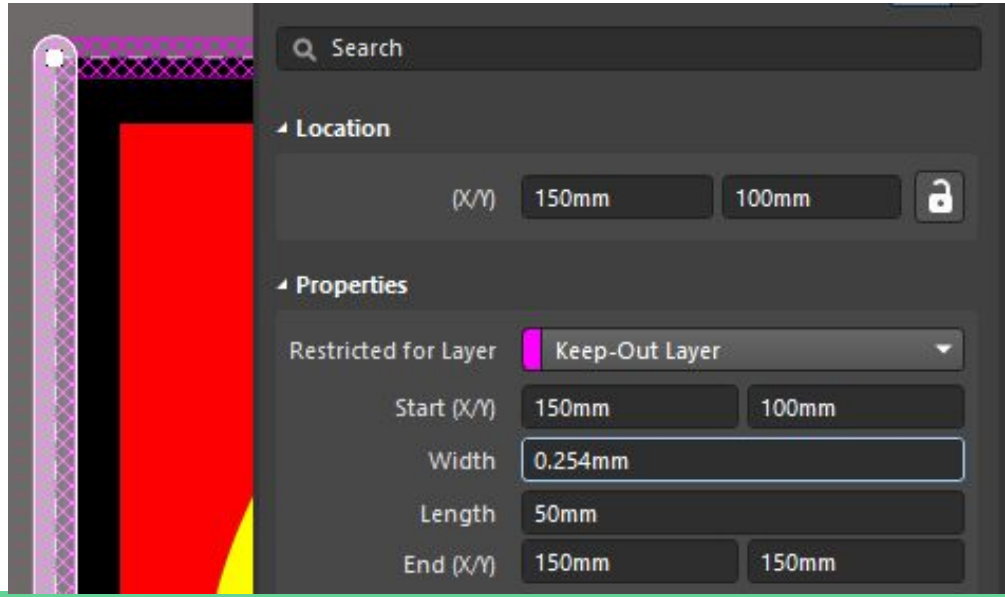
Use (P, K, T) command to place tracks on keep out layer.

Place the keepout layer around the edge of the PCB and then repour the polygon



Keep Out Layer

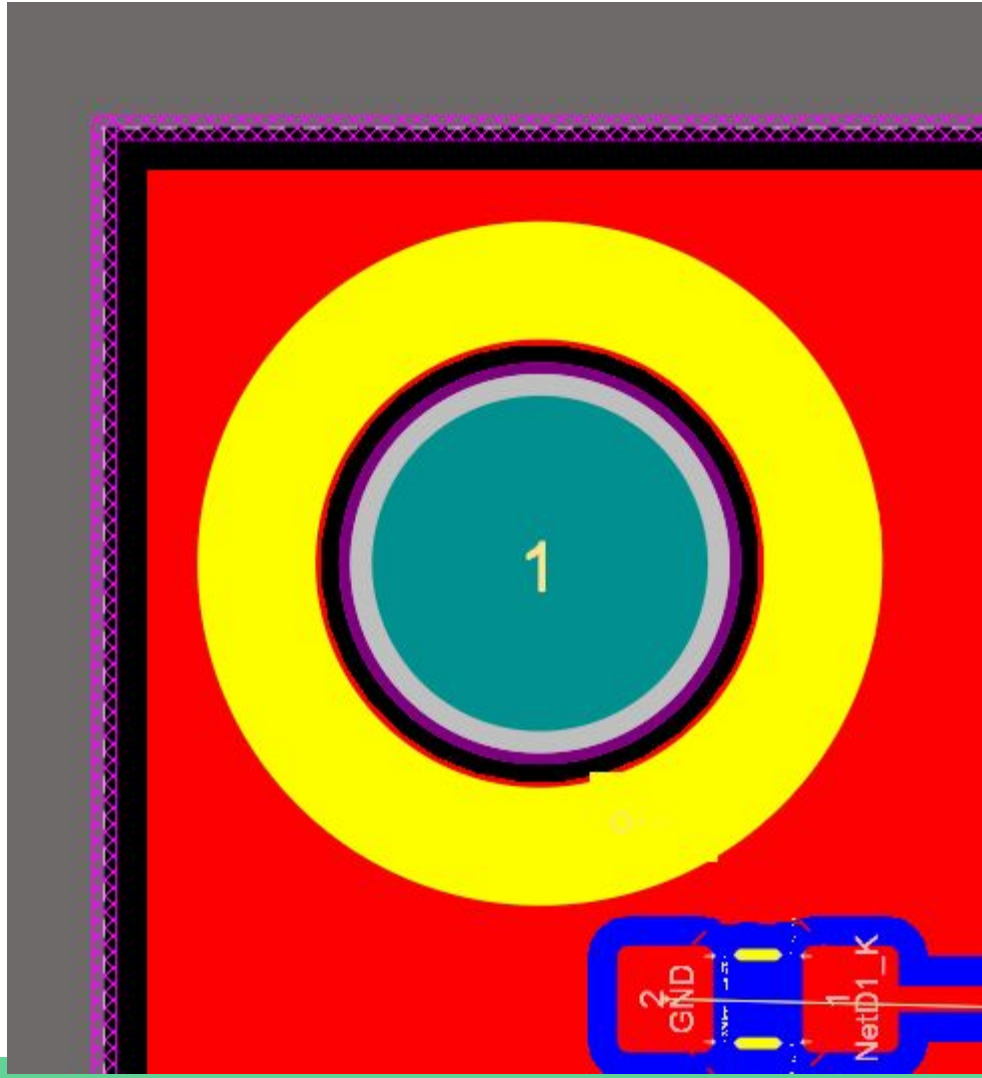
Make sure the Restricted for Layer is set to “KeepOut Layer”



Keep Out Layer

Confirm the Keep-out Layer is placed around the edge of the PCB

(The purple lines)



PCB

Generate GERBER and NCDrill files for manufacture

Don’t really need to do now, just for reference when you want to manufacture it

GERBER and NCDrill files

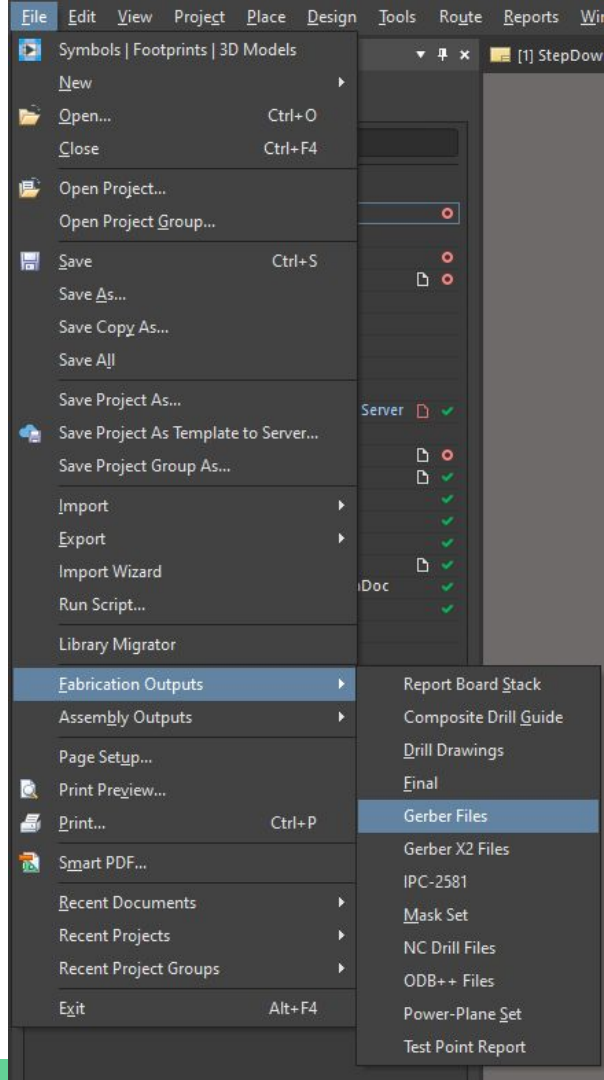
Those are the files that the PCB Manufacture needs in order to manufacture the PCB

GERBER

GERBER is de-facto standard format used in PCB fabrication

It is under File -> Fabrication Outputs -> GERBER

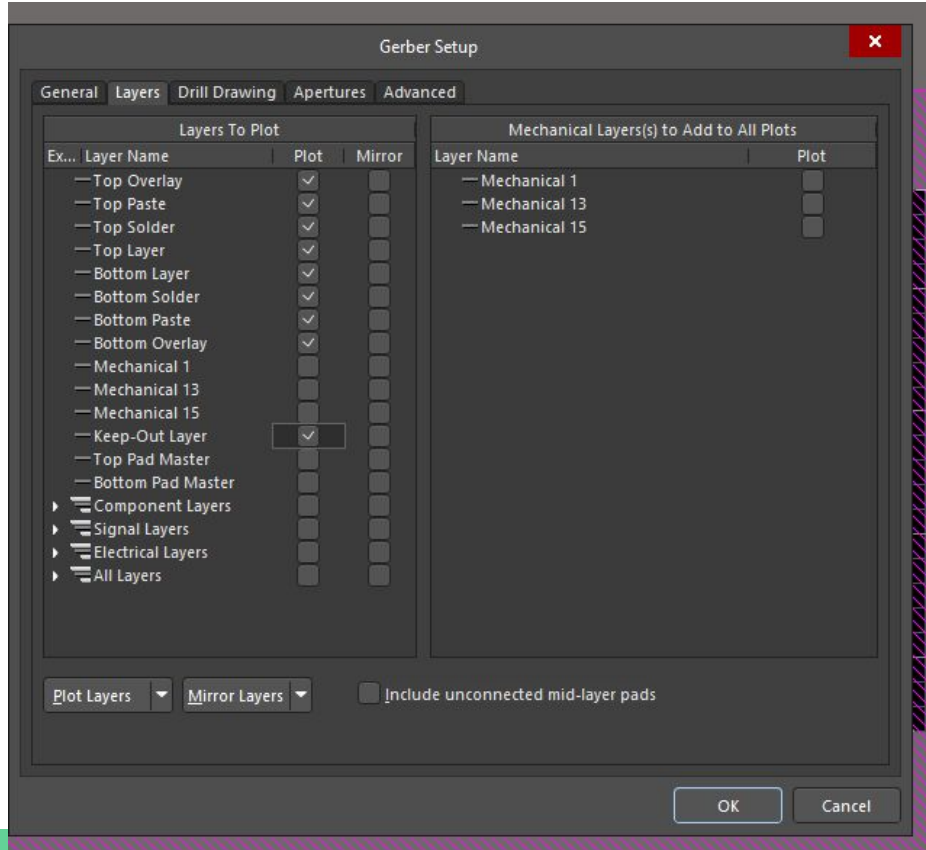
Shortcut is (F, F) to open the menu



GERBER

For a 2-layer board, usually you just need these 9 layers

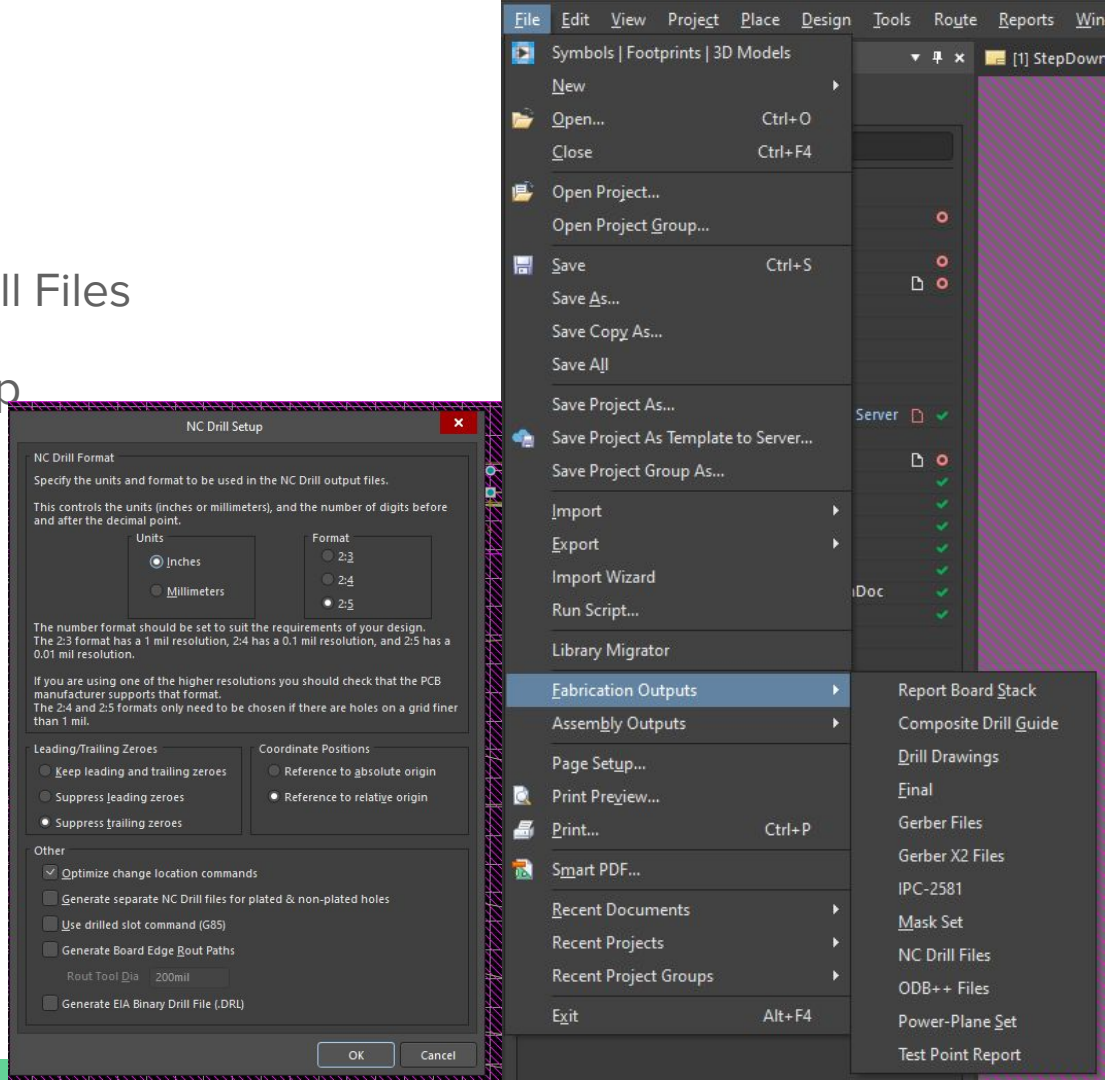
For 4 layer boards, 2 middle layer will need to be included as well (11 in total)



NCDrill

File -> Fabrication Output -> NC Drill Files

Press “OK” when the menu pops up



Submitting the PCB for manufacturing

Put both of the NC Drill and GERBER files generated into a zip folder and follow the instructions on the PCB manufacturer’s website.

It is advised to double check if you have submit the correct files and/or submit all the required files