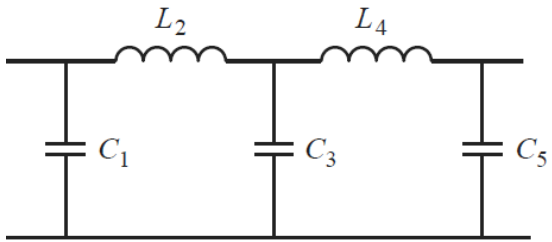
Project Description

Design a 5-section (N=5) stepped-impedance **maximally-flat** microstrip low-pass filter with capacitor input, cutoff frequency assigned to your group, 50 Ω system.
Use 6mm long 50 Ω feed lines to connect the input and output of your filter (for SMA connection)
Verify your design with ADS.
Design Guidelines:
Substrate type: Rogers 4350B ; H (substrate thickness)=1.524 mm
E_r=3.66 ; Mu_r=1
Cond (copper)=5.88e7 T(Thickness of metal)=35 um
TanD=0.0032 Z_H=120 Ω , Z_L=20 Ω

a) Design and optimize your filters in ADS schematics and measure S parameters.
b) Round all final width and length values to 0.01 mm before generating layout (Ex: 5.878mm → 5.88mm)
c) Generate the layout and compare the EM results with schematic.
d) **Export the gerber files of layout and zip them.**
e) Zip file name: Same as labeling (slide 4)



Labeling

Write Group Number and participants initials on the top left corner of board with “silkscreenTop” layer and font size of 1.5 (or the largest that fits)

Example:
If Group #1 participants are Ronald Reagan and John Kennedy, they should label their PCB as:
G1_RR_JK

Make sure the label text does not touch any of the metal traces and lies inside the board shape.

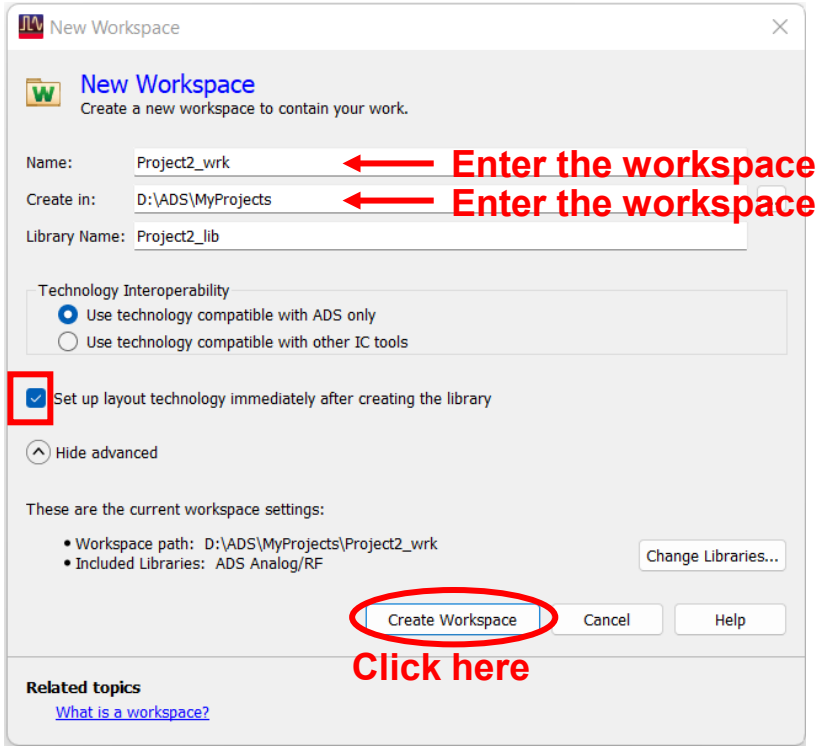
Your board shape is defined by the area covered by cond2 (bottom ground plane) rectangle.

Layers Configuration and Use (Important)

Follow these steps when you create a new workspace to make sure all required fabrication layers are present in the design and ready to export

Creating layout in PCB-ready format (1)

- Click on File -> New ->Workspace



Enter the workspace name

Enter the workspace path

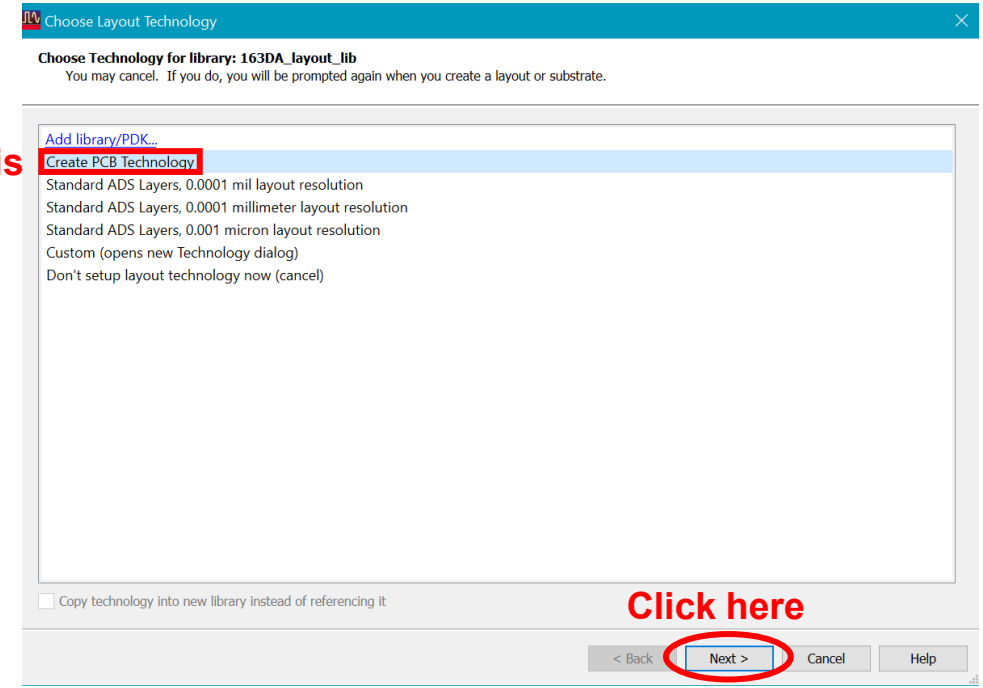
Check this box

Click here

Creating layout in PCB-ready format (2)

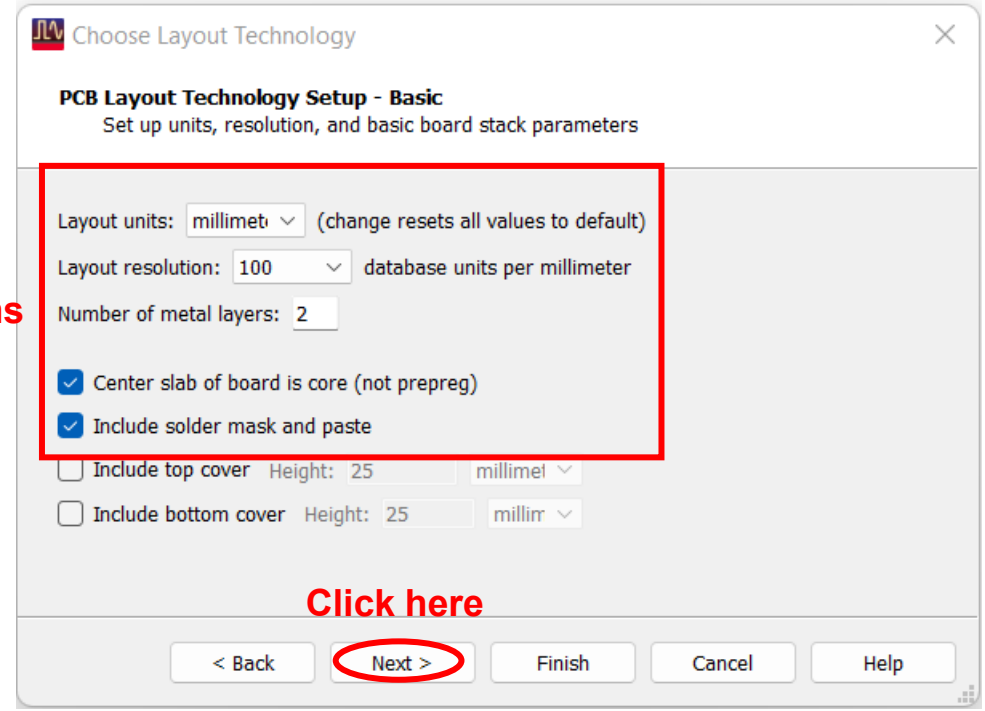
- Select Create PCB technology and click on Next

Select this



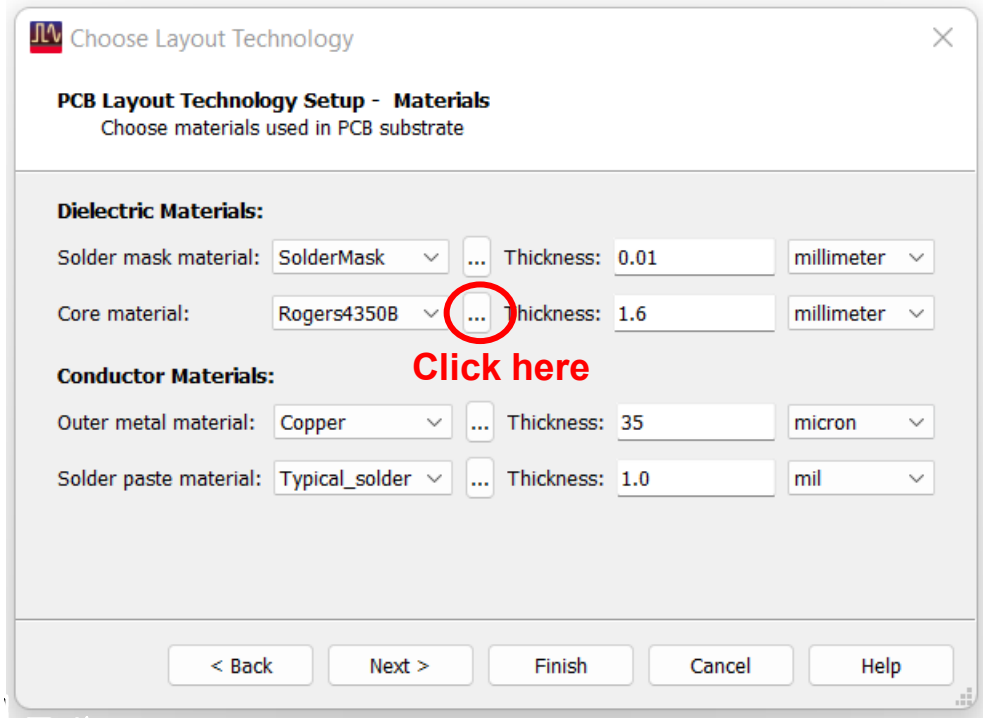
Creating layout in PCB-ready format (3)

- Select the options indicated and click on Next



Creating layout in PCB-ready format (4)

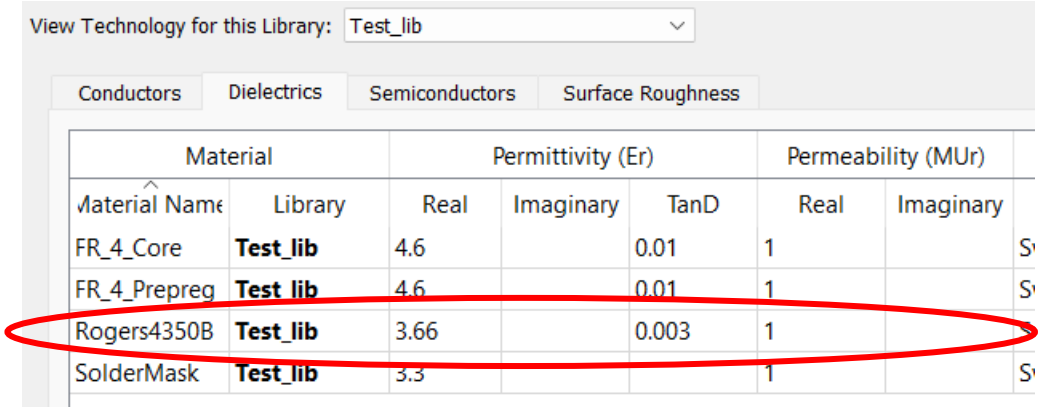
- Select the options indicated and click on Next



Creating layout in PCB-ready format (5)

- Select the options indicated and click on Next

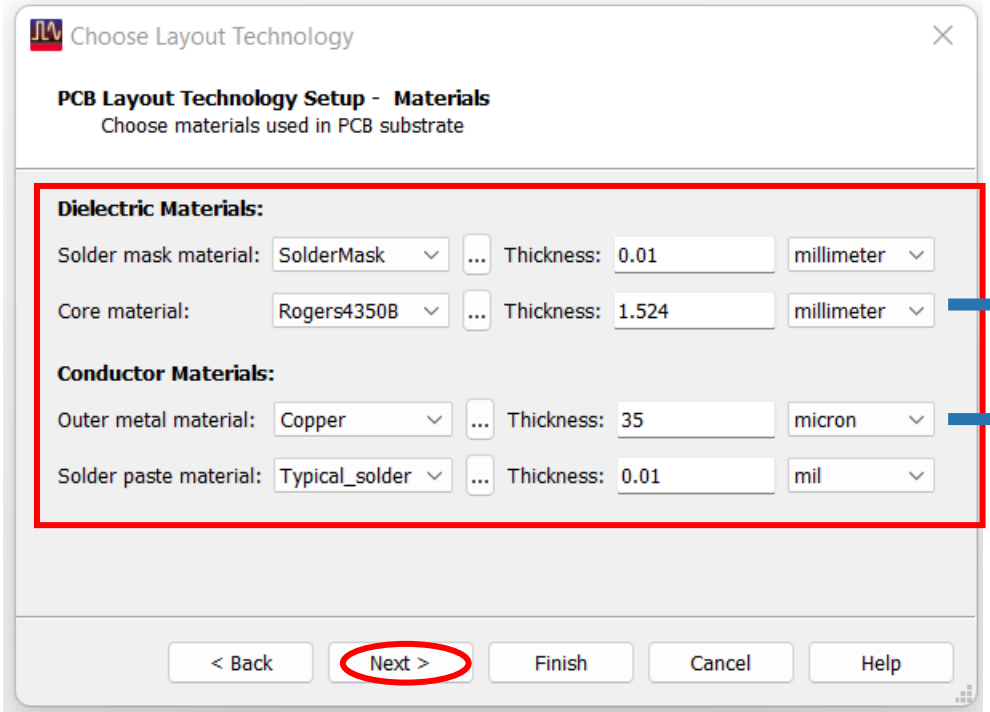
Define Your Desired Substrate



Creating layout in PCB-ready format (6)

- Select the options indicated and click on Next (You can modify later in layout sub stack)

Set These Options

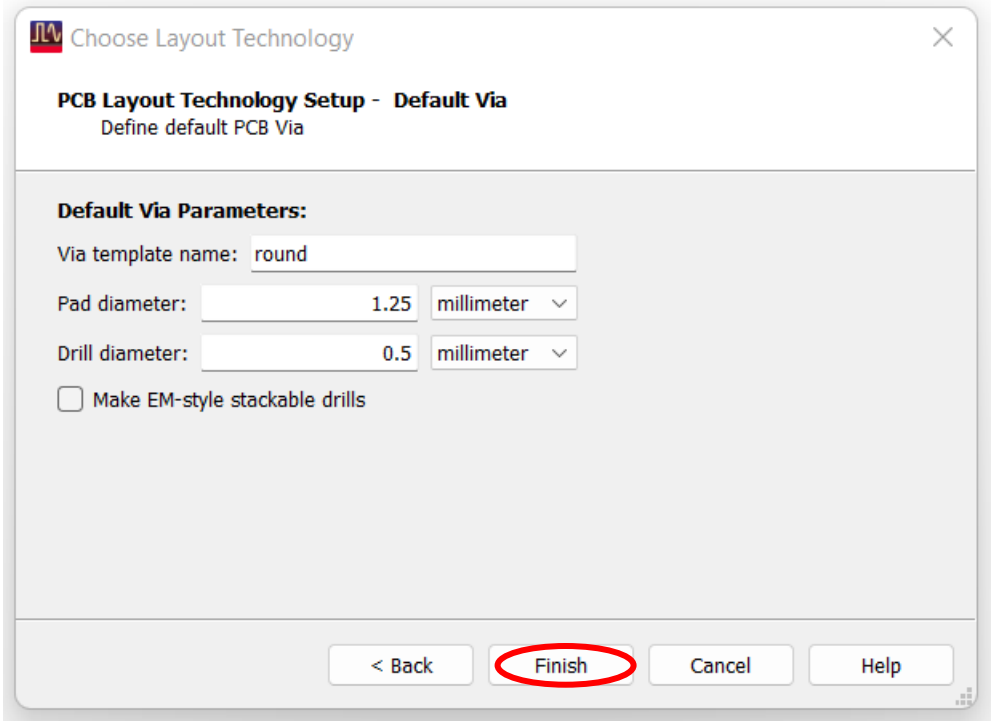


Board Thickness

Copper Thickness

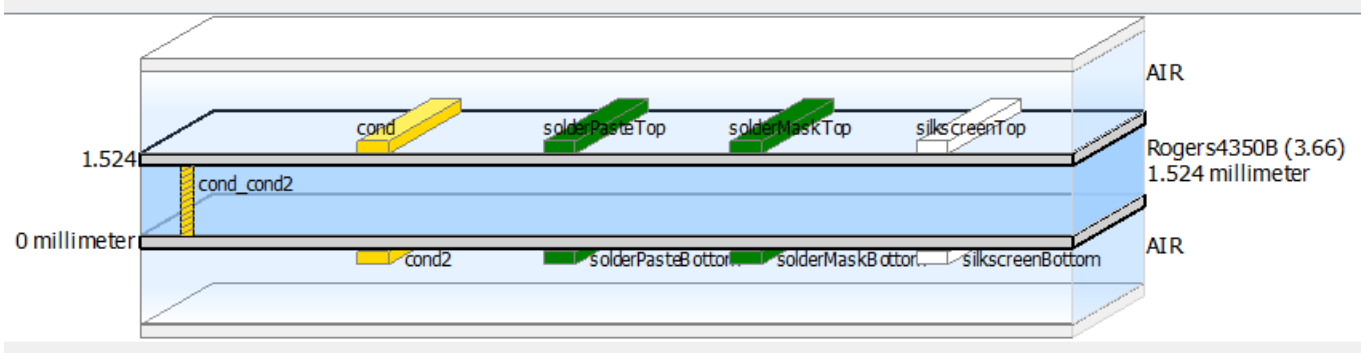
Creating layout in PCB-ready format (7)

- Click on Finish



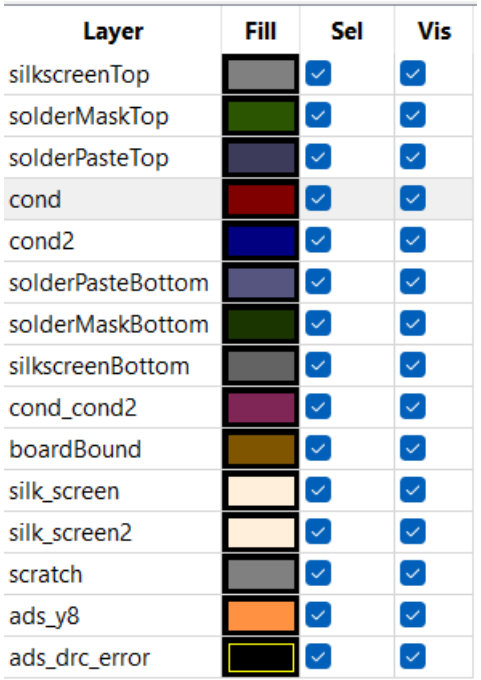
Creating layout in PCB-ready format (8)

- Now you have defined PCB tech, you do not need to choose anything when placing MSUB in schematics.
- Your substrate stack in layout should look like the picture below.
- Modify the conductors thickness, conductors conductivity, dielectric constant, dielectric TanD, and etc., in both layout stack and schematic (according to your design), if necessary.



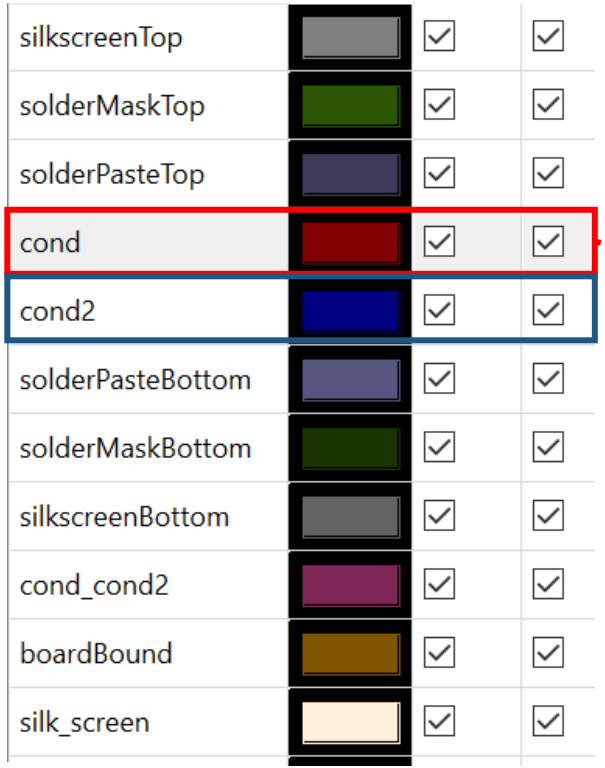
Creating layout in PCB-ready format (9)

- Your available layout layers should look like this:



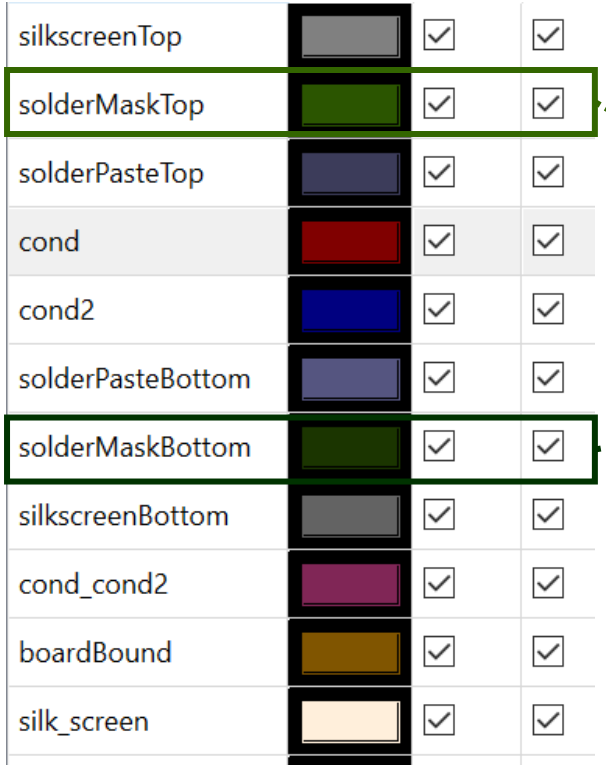
Layers you need to export are listed below
- cond
- cond2
- solderMaskTop
- solderMaskBottom
- silkscreenTop

Important guidelines about use of layers (1)



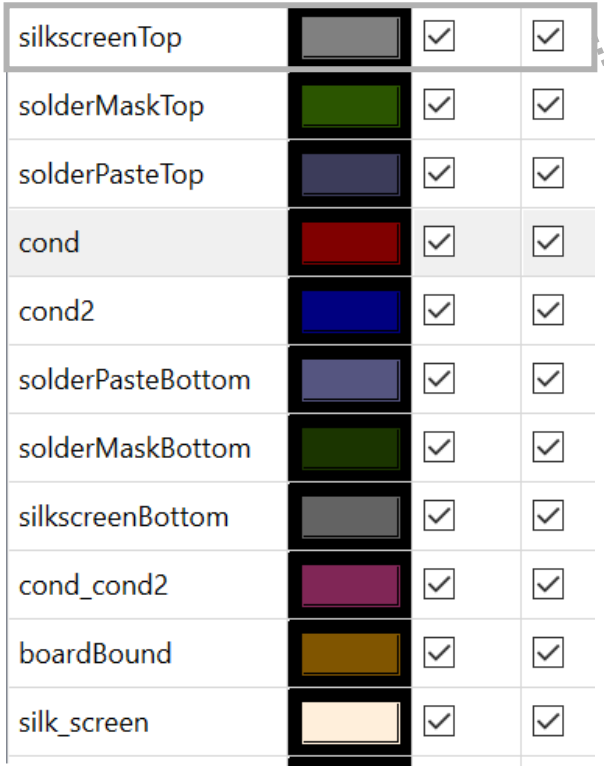
- You are going to fabricate a 2-layer PCB having top and bottom layers
- Your design should be done using the top layer (**cond** in ADS)
- The entire bottom layer (**cond2** in ADS) should act as the ground plane. This can be done by drawing a rectangle using the **cond2** layer covering the entire PCB layout

Important guidelines about use of layers (2)



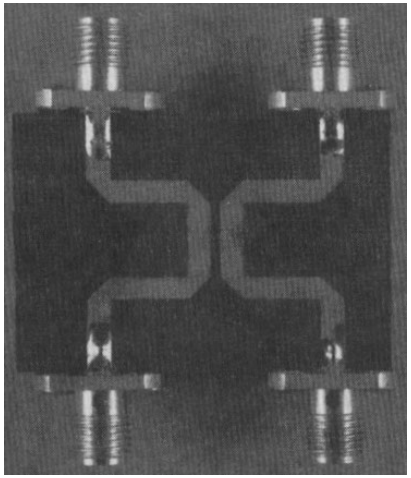
- The solderMaskTop(Bottom) layer covers the parts of the top(bottom) layer where the conductor is to be exposed
- The **solderMaskTop** layer should cover your entire cond layer
- **solderMaskBottom** layer should cover the entire cond2 layer of the PCB
- **In summary, cond2, solderMaskTop, and solderMaskBottom are aligned rectangles of the same dimension that define and cover the board shape. This is to expose all metals and enable soldering SMA connectors.**

Important guidelines about use of layers (3)



- The silkscreenTop(Bottom) layers are non-conductive epoxy ink layers used for writing text or other information on the PCB
- You should write labelling on the top left corner of your PCB using the **silkscreenTop** layer

Planning for use of SMA edge connectors



- SMA edge connectors (top left) will be directly soldered to each of the ports of your PCB for the purpose of measurement. You do not need to add pads to your layout.
- This is the reason metals are left exposed by adding solder mask layers.

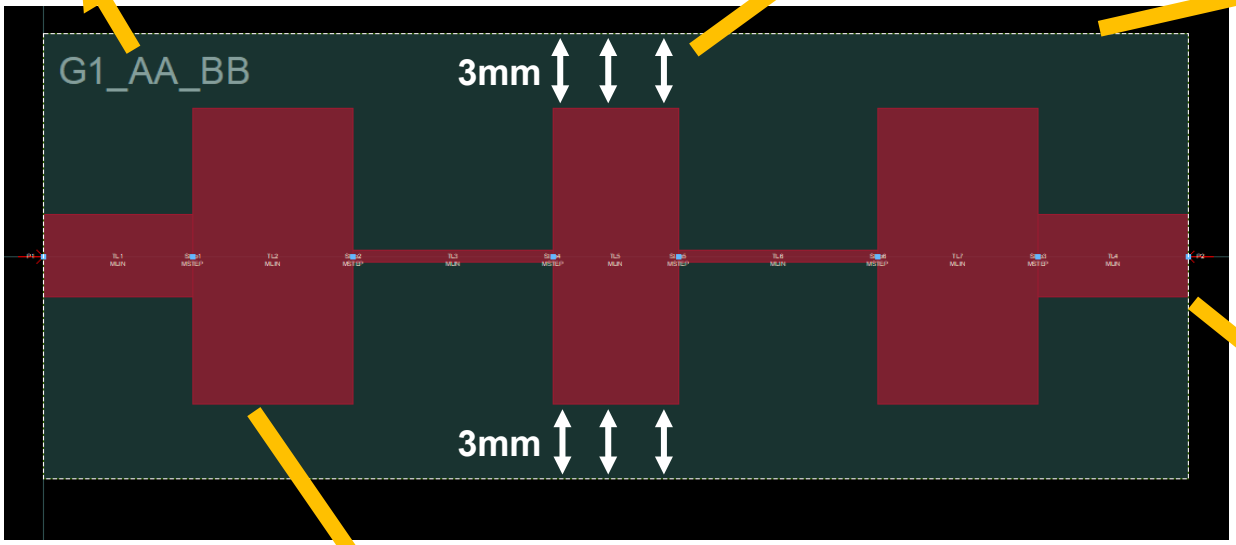
Sample Layout (Important)

Layout Considerations

Labeling on silkscreenTop

Extend the rectangle by roughly 3mm from the top and bottom edge of cond layer

3 aligned rectangles covering the design on cond2, solderMaskTop, and solderMaskBottom



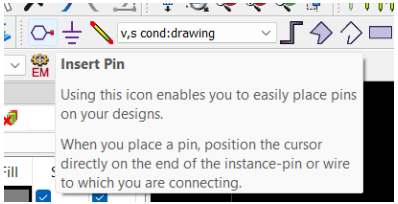
Left and right edges of rectangles must be aligned with cond

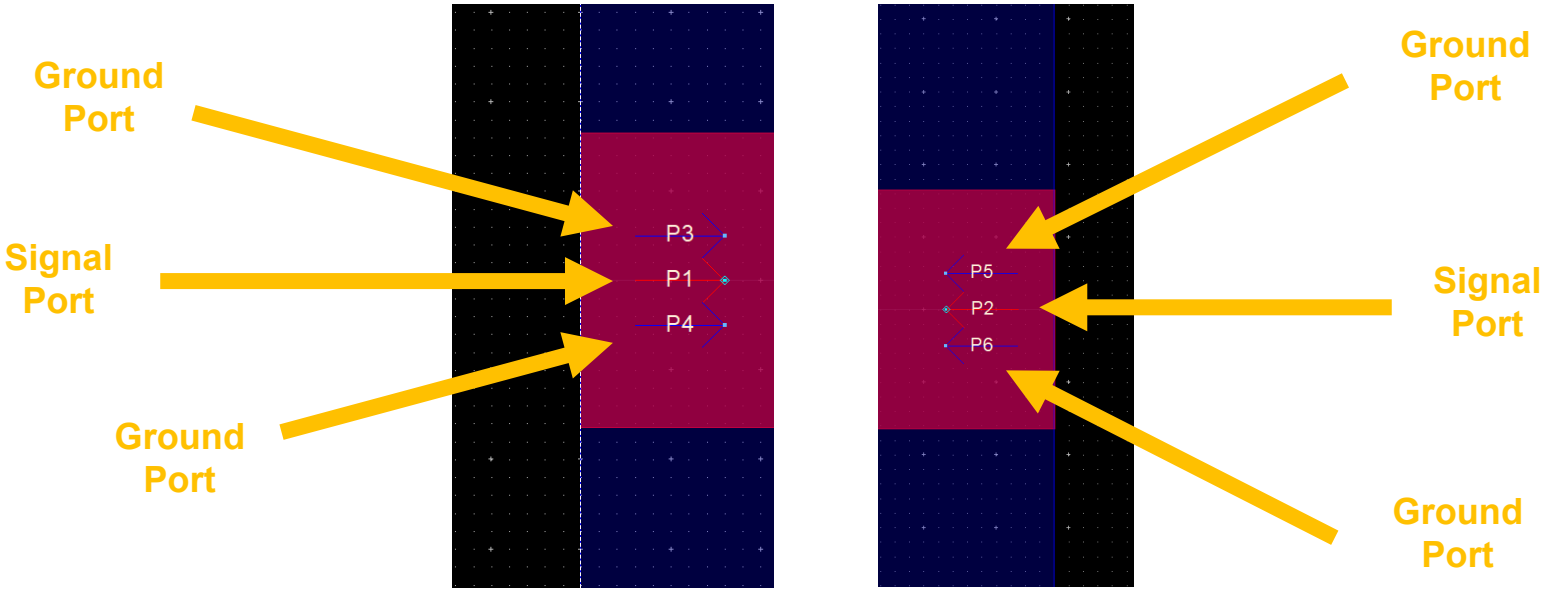
Main filter design on cond layer

Port Assignment (Important)

More Accurate EM Simulation

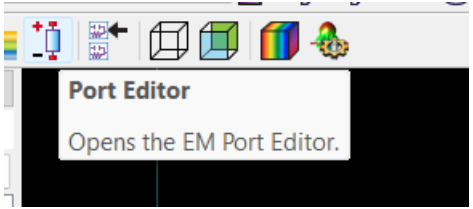
- Place two ground pins in layout on the cond2 layer for each port (both sides of the signal pins, around **0.5mm** far from the signal port)
- Make sure ports are touching the metals (bring them inside) and are in a symmetric fashion
- This is called GSG (Ground-Signal-Ground) Port





Defining GSG ports

- Go to -> Port Editor

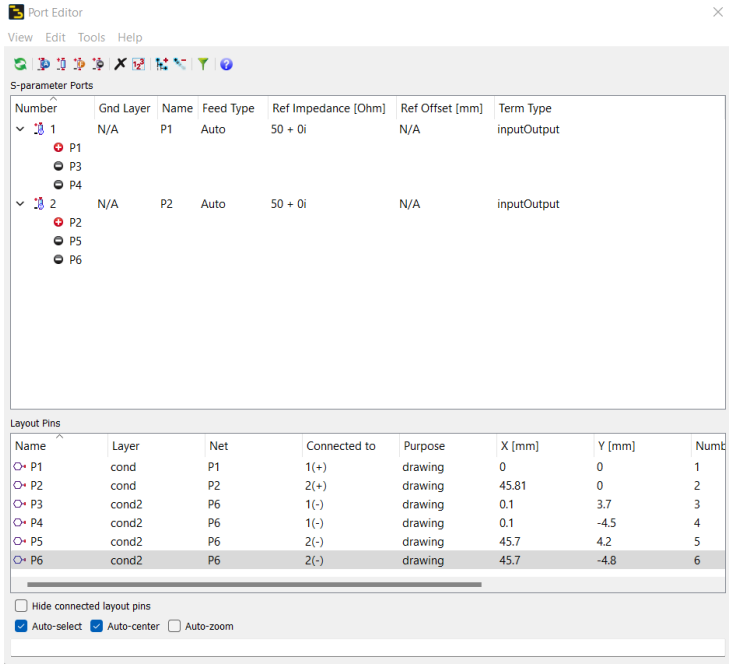


- Drag ground ports from the bottom and drop them on the Gnd of your signal ports at top

- This is how it looks like after (also in EM setup):

- Run EM simulations

- If result does not look smooth and uniform, especially at higher frequencies, try Momentum Microwave and increase Mesh density.



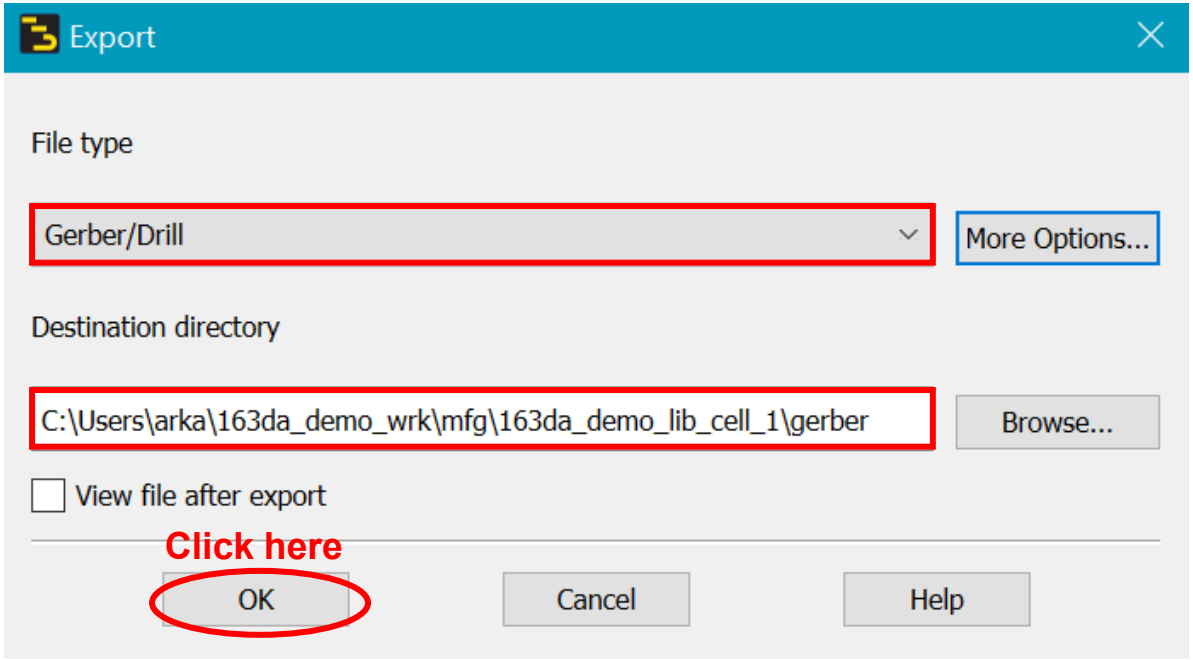
Fabrication Outputs (Important)

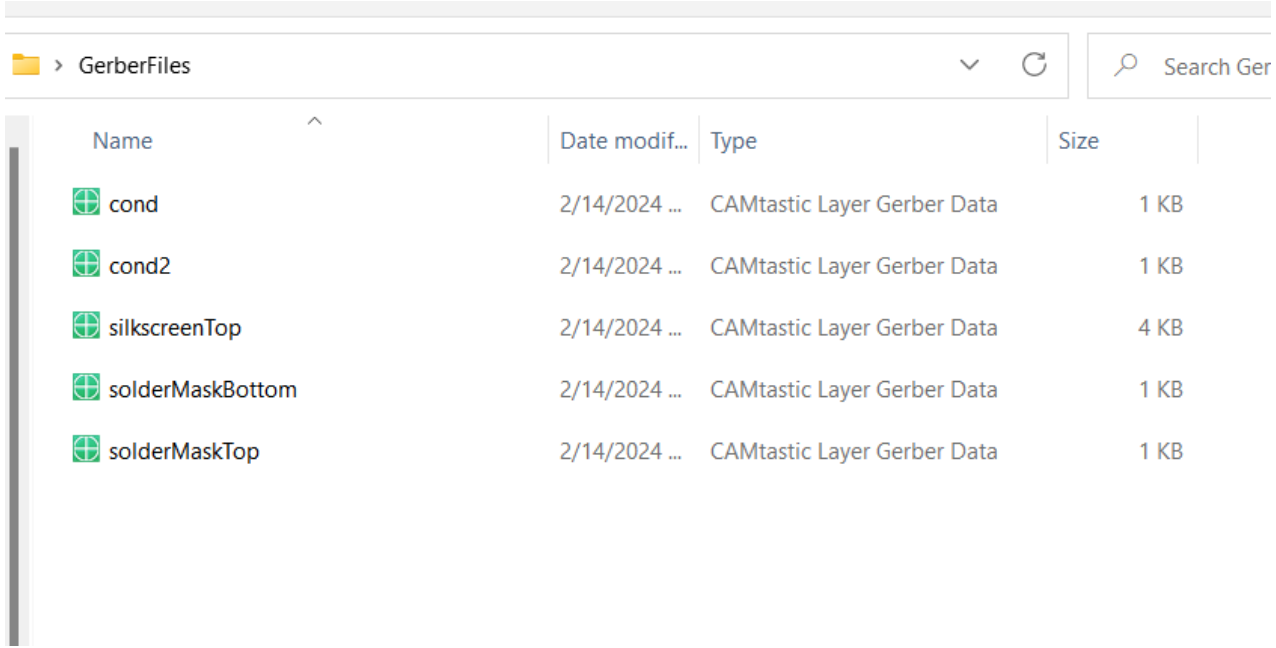
Exporting Gerber files

- Click on File -> Export (Use an empty folder for the directory)

Select this option

Choose the destination directory



Exporting Gerber files

- All files shown below with the exact same name must be present in your zip file