

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, 50Ω system.

Use 6mm long 50Ω feed lines to connect the input and output of your filter (for SMA connection)

Verify your design with ADS.

Design Guidelines:

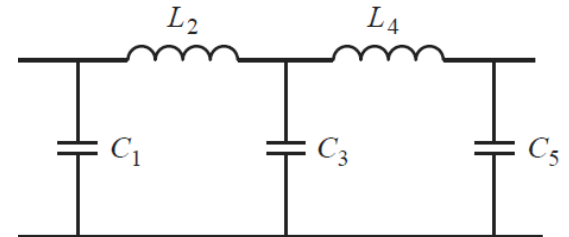
Substrate type: Rogers 4350B ; H (substrate thickness)=1.524 mm

$\epsilon_r=3.66$; $\mu_r=1$

Cond (copper)= $5.88e7$ T(Thickness of metal)=35 μm

$\text{TanD}=0.0032$ $Z_H=120 \Omega$, $Z_L=20 \Omega$

- Design and optimize your filters in ADS schematics and measure S parameters.
- Round all final width and length values to 0.01 mm before generating layout (Ex: 5.878mm \rightarrow 5.88mm)**
- Generate the layout and compare the EM results with schematic.
- Export the gerber files of layout and zip them.**
- 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

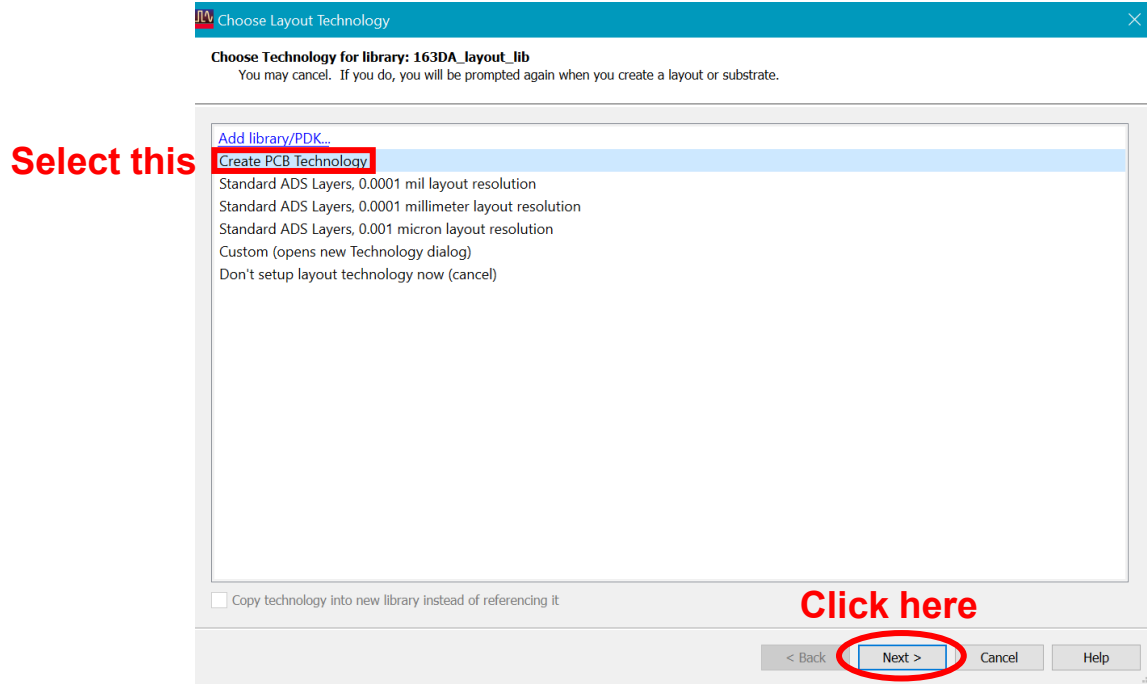
Enter the workspace path

Check this box

Click here

Creating layout in PCB-ready format (2)

- Select Create PCB technology and click on Next



Creating layout in PCB-ready format (3)

- Select the options indicated and click on Next

Select these options

Choose Layout Technology

PCB Layout Technology Setup - Basic
Set up units, resolution, and basic board stack parameters

Layout units: millimet (change resets all values to default)

Layout resolution: 100 database units per millimeter

Number of metal layers: 2

Center slab of board is core (not prepreg)

Include solder mask and paste

Include top cover Height: 25 millimet

Include bottom cover Height: 25 millir

Click here

< Back **Next >** Finish Cancel Help

Creating layout in PCB-ready format (4)

- Select the options indicated and click on Next

Choose Layout Technology

PCB Layout Technology Setup - Materials
Choose materials used in PCB substrate

Dielectric Materials:

Solder mask material: SolderMask ... Thickness: 0.01 millimeter

Core material: Rogers4350B ... Thickness: 1.6 millimeter

Conductor Materials:

Outer metal material: Copper ... Thickness: 35 micron

Solder paste material: Typical_solder ... Thickness: 1.0 mil

Click here

< Back Next > Finish Cancel Help

Creating layout in PCB-ready format (5)

- Select the options indicated and click on Next

View Technology for this Library:

Conductors Dielectrics Semiconductors Surface Roughness

Material		Permittivity (Er)			Permeability (MUr)	
Material Name	Library	Real	Imaginary	TanD	Real	Imaginary
FR_4_Core	Test_lib	4.6		0.01	1	
FR_4_Prepreg	Test_lib	4.6		0.01	1	
Rogers4350B	Test_lib	3.66		0.003	1	
SolderMask	Test_lib	3.3			1	

**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

Choose Layout Technology

PCB Layout Technology Setup - Materials
Choose materials used in PCB substrate

Dielectric Materials:

Solder mask material: SolderMask Thickness: 0.01 millimeter

Core material: Rogers4350B Thickness: 1.524 millimeter

Conductor Materials:

Outer metal material: Copper Thickness: 35 micron

Solder paste material: Typical_solder Thickness: 0.01 mil

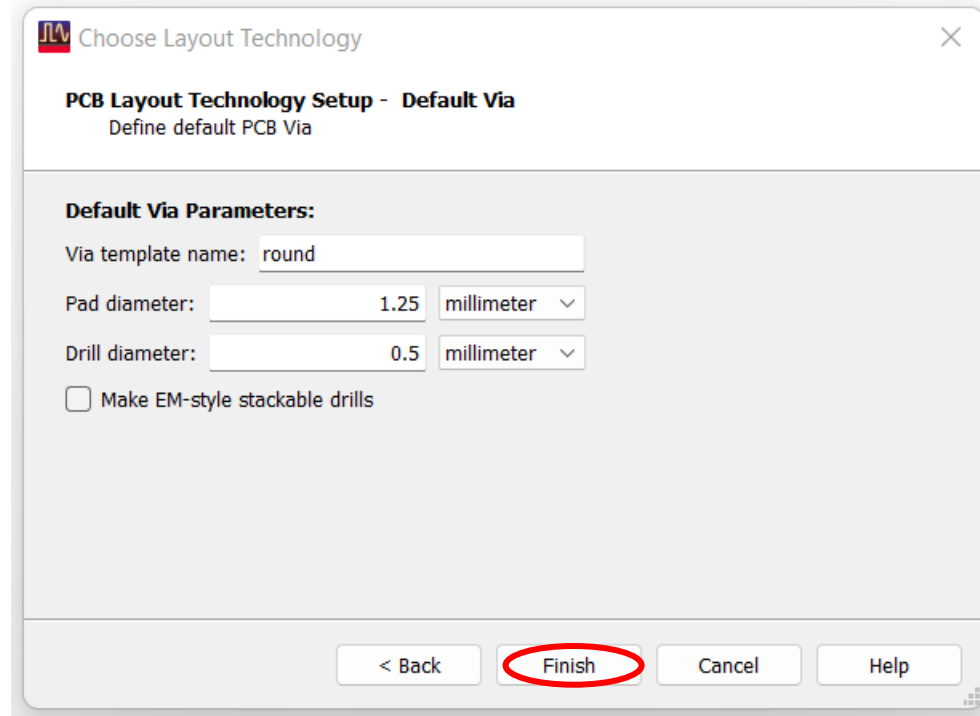
< Back **Next >** Finish Cancel Help

Board
Thickness

Copper
Thickness

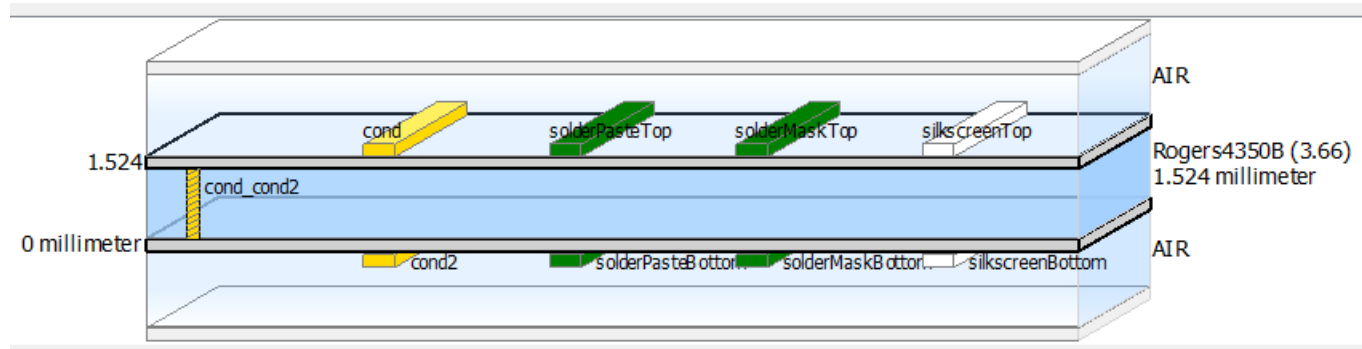
Creating layout in PCB-ready format (7)

- 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**

Layer	Fill	Sel	Vis
silkscreenTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silkscreenBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond_cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
boardBound		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silk_screen		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silk_screen2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
scratch		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
ads_y8		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
ads_drc_error		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Important guidelines about use of layers (1)



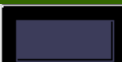




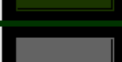



silkscreenTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silkscreenBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond_cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
boardBound		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silk_screen		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

- 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)








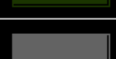



- The entire bottom layer (**cond2** in ADS) should act as the ground plane. This can be done by drawing a rectangle using the **cond2** layer covering the entire PCB layout

Important guidelines about use of layers (2)

silkscreenTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silkscreenBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond_cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
boardBound		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silk_screen		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

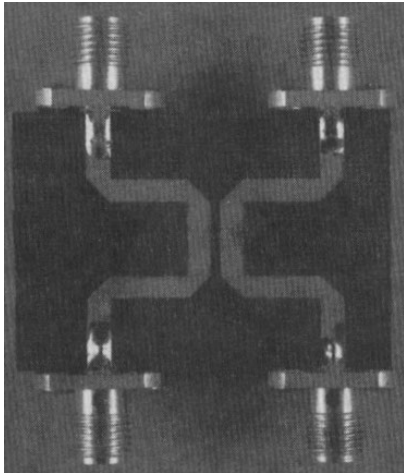
- The solderMaskTop(Bottom) layer covers the parts of the top(bottom) layer where the conductor is to be exposed
- The **SolderMaskTop** layer should cover your entire cond layer
- **solderMaskBottom** layer should cover the entire cond2 layer of the PCB
- **In summary, cond2, solderMaskTop, and solderMaskBottom 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)

silkscreenTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteTop		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderPasteBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
solderMaskBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silkscreenBottom		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
cond_cond2		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
boardBound		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
silk_screen		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

- 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.

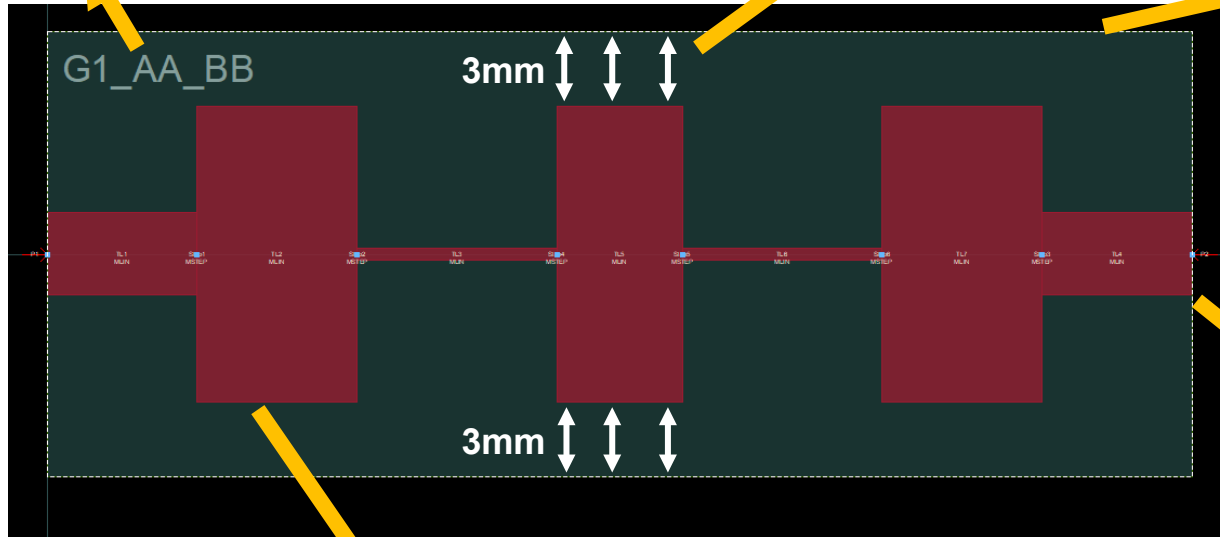
Sample Layout (Important)

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



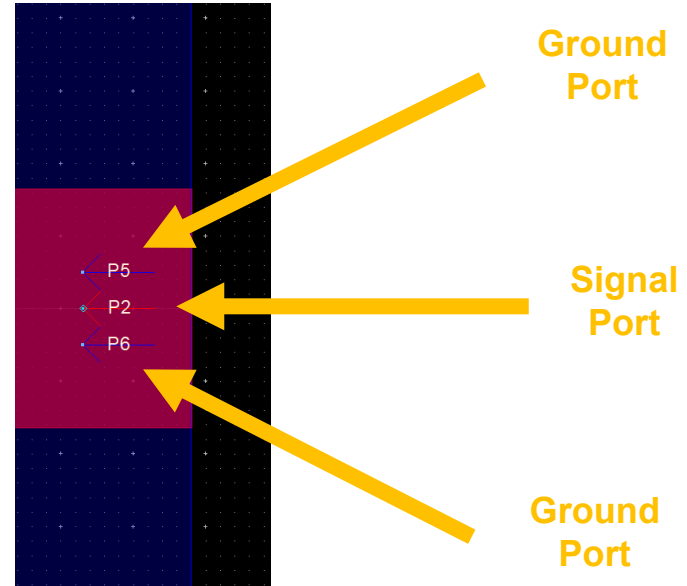
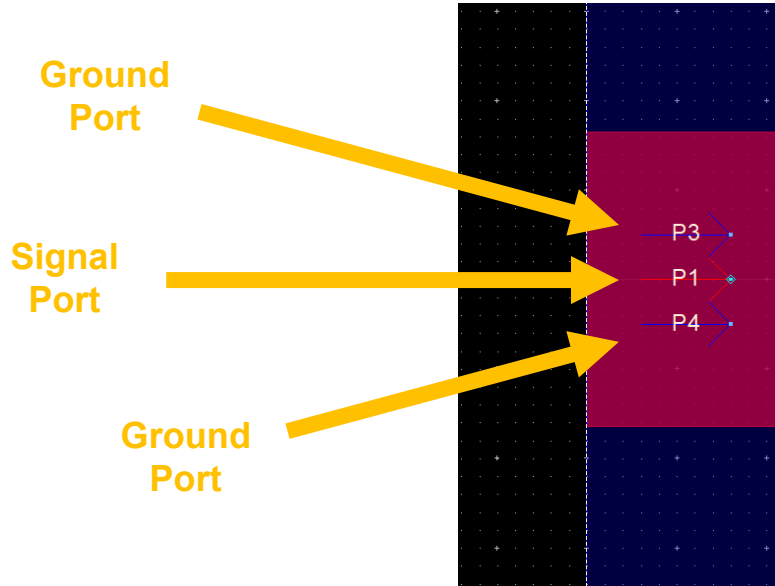
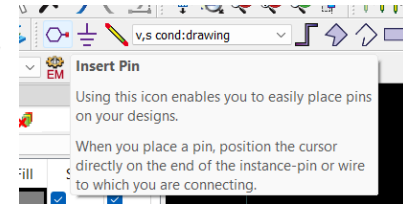
Main filter design on cond layer

Left and right edges of rectangles must be aligned with cond

Port Assignment (Important)

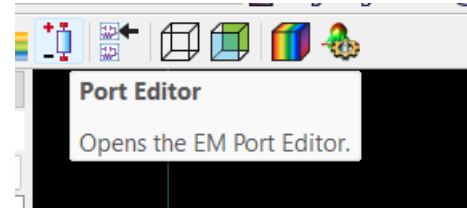
More Accurate EM Simulation

- 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

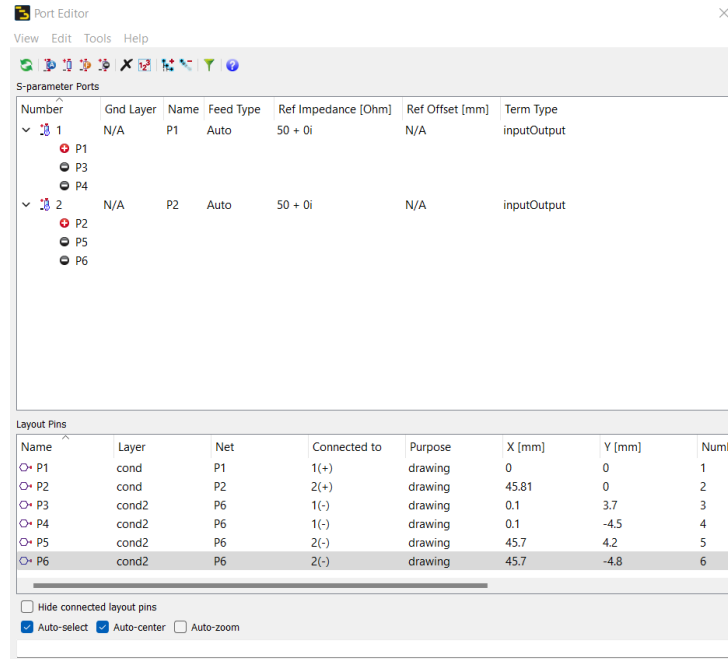


- Drag ground ports from the bottom and drop them on the Gnd of your signal ports at top

- 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)

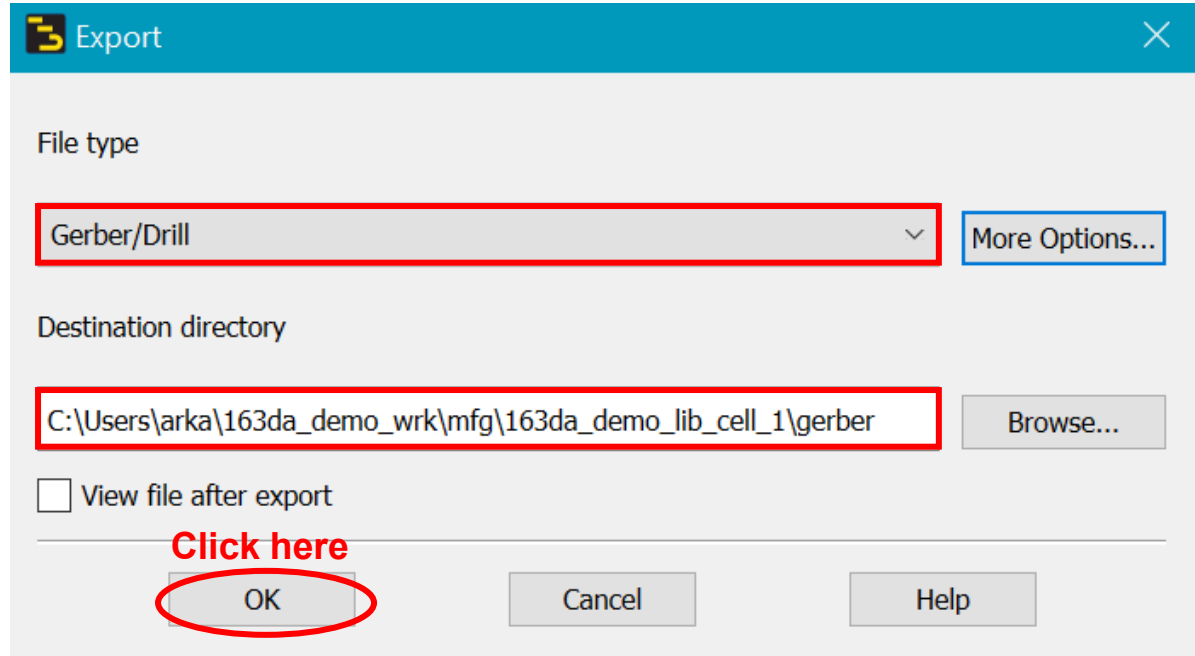
Exporting Gerber files

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

Select this option

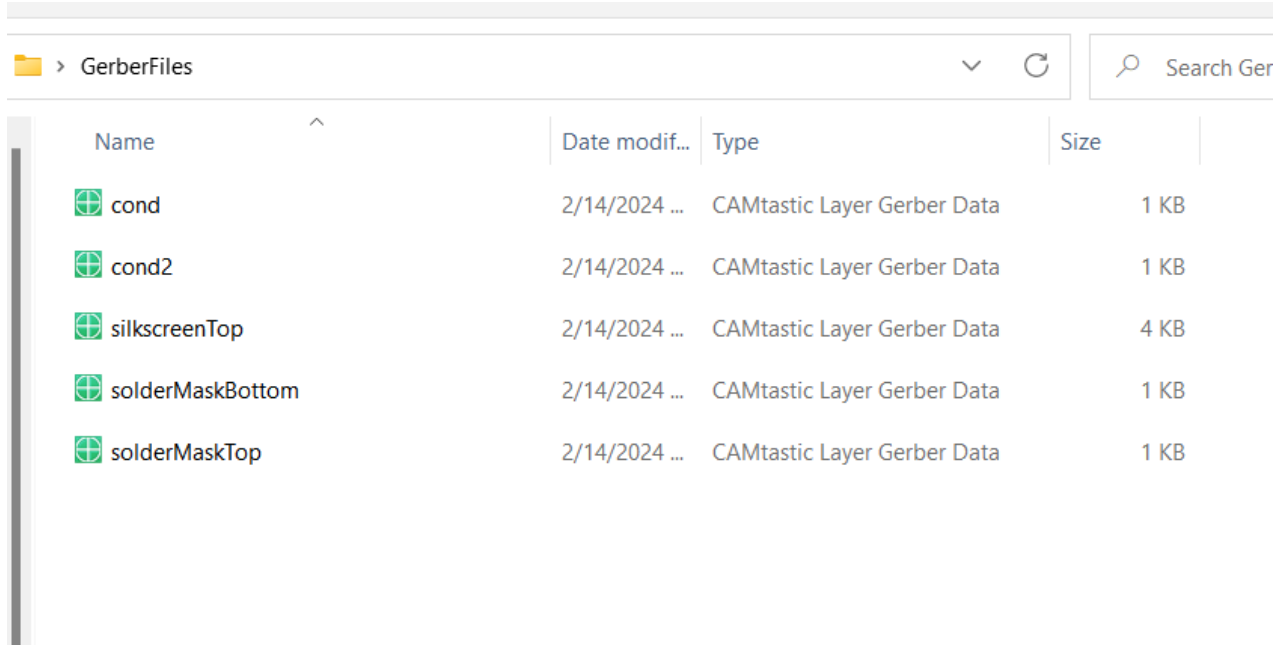
Choose the destination directory

Click here



Exporting Gerber files

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



The screenshot shows a file explorer window with the following details:

- Folder path: GerberFiles
- Search bar: Search Ger
- Table of files:

Name	Date modif...	Type	Size
cond	2/14/2024 ...	CAMtastic Layer Gerber Data	1 KB
cond2	2/14/2024 ...	CAMtastic Layer Gerber Data	1 KB
silkscreenTop	2/14/2024 ...	CAMtastic Layer Gerber Data	4 KB
solderMaskBottom	2/14/2024 ...	CAMtastic Layer Gerber Data	1 KB
solderMaskTop	2/14/2024 ...	CAMtastic Layer Gerber Data	1 KB