ADS Momentum EM Co-Simulation Tutorial

August 25, 2025

1. Introduction

High-performance circuits are driven by higher component density, complex interconnects, electromagnetic interference and compatibility (EMI/EMC) requirements, signal and power integrity requirements. At high frequencies, the separation between component and traces becomes small, and circuits performance might be affected by its surroundings. The placement and interconnection of these components in a printed circuit board (PCB) is called layout. Electromagnetic simulation considers the effects of the layout, including coupling and radiation effects. These might vary from a circuit simulation which doesn't take into account the layout parasitic. Using numerical methods (FDTD, FEM, MoM, etc.) allows engineers to simulate these effects and tune the circuit before manufacturing and testing.

2. What is Momentum?

Momentum is a 2.5D Planar electromagnetic (EM) simulator from Keysight Technologies, Inc for RF passives, high-frequency interconnects and parasitic modeling. It combines full wave and quasi-static EM solvers to provide insight into MMIC, RFIC, RF PCB, Signal Integrity, and PCB antennas. Momentum is integrated into ADS providing capabilities for:

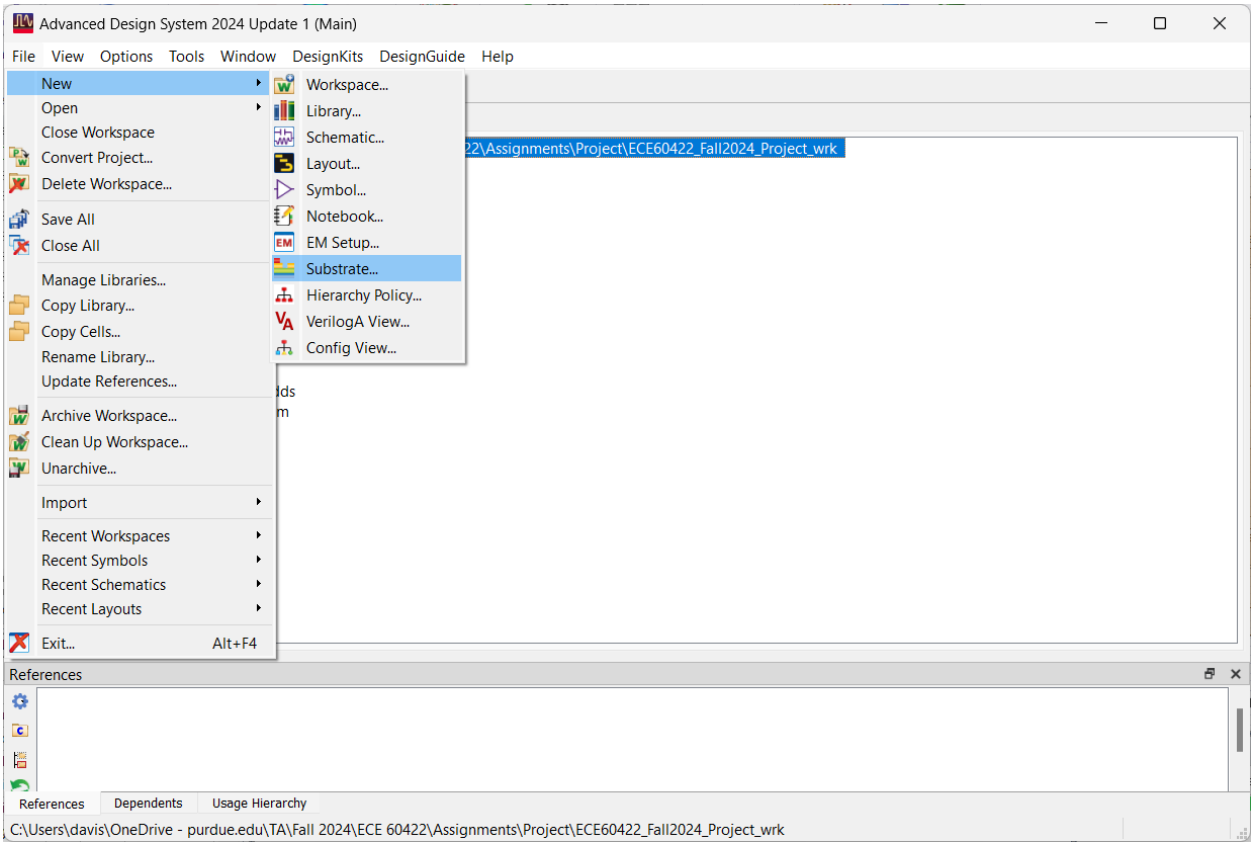
- Method of Moments 3D planar electromagnetic simulator engineered for speed and capacity through advanced NlogN solver technology.
- Multi-threaded simulation for additional speed.
- Visualization of surface currents and planar antenna radiation in 3D space.
- Sophisticated post processing of S-parameters and other data.
- User selectable microwave full-wave or faster RF quasi-static mode EM simulation.
- Adaptive frequency sweep automatically and quickly finds all resonant frequencies across the full simulation frequency band.
- Arbitrary polygonal meshing with adaptive mesh reduction for optimal speed, accuracy, and capacity.
- Efficient bond wire and via models.
- Thick metal analysis.
- Co-simulation with circuit simulators in ADS and Genesys.
- Optimization of parameterized geometries together with circuit and system components.
- Layout look-alike symbols in schematic for error free hookup with circuit or system components.
- SI/PI Analyzer with easy setup of net based connection-oriented simulations for signal and power integrity.
- Post-processing with excitations extracted from circuit simulation

3. Setup

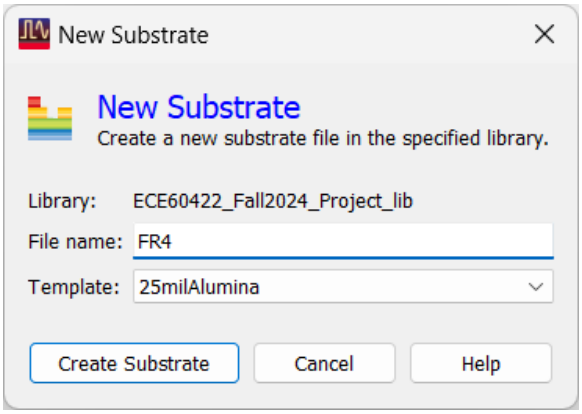
This tutorial simulates a bias tee with a series capacitor and meandered 90° stub at 2.4 GHz in an FR4 substrate with 1oz copper. A meandered line means it is bend to be compact.

3.1. Layout

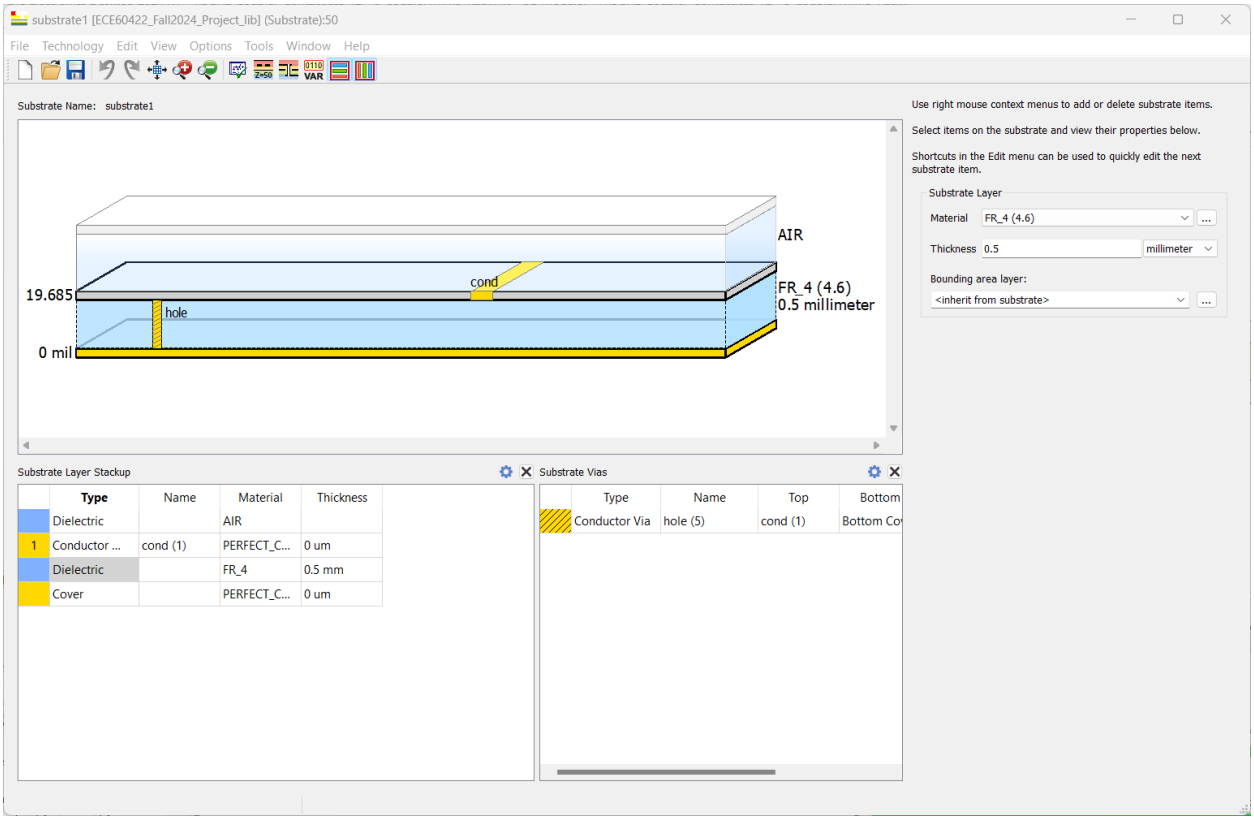
1. Create the PCB stack-up from the main menu, File > New > Substrate



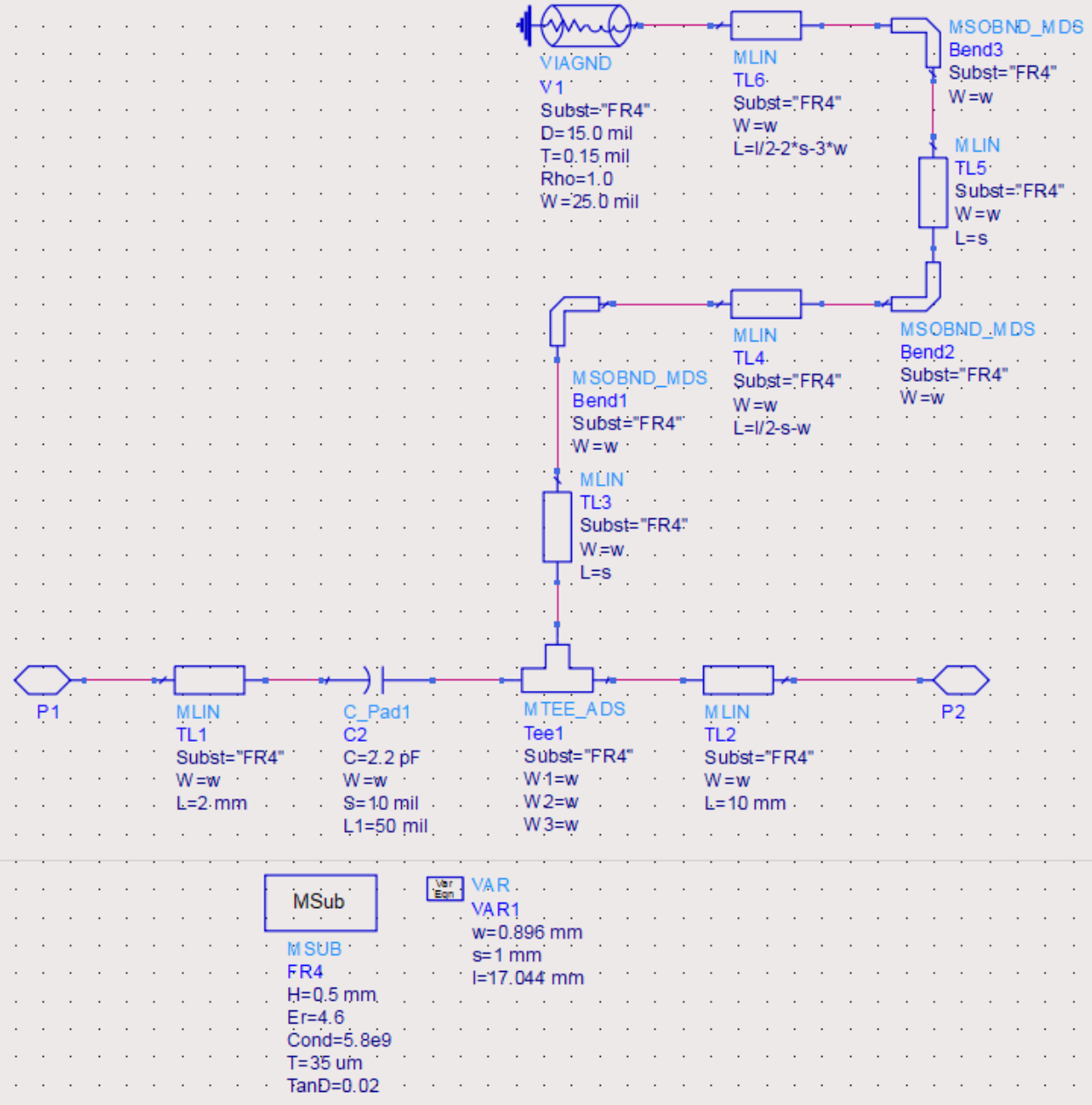
In the substrate tab, create a new substrate using the default template.



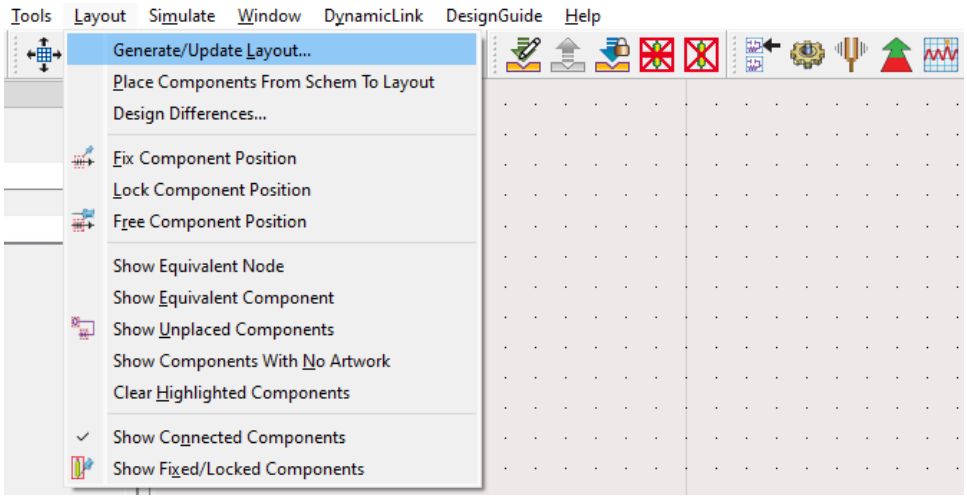
Change the substrate stack-up to the thickness and dielectric to achieve the following substrate stack-up. You can right-click in the layer to add and modify the stack-up. Save and close the window.



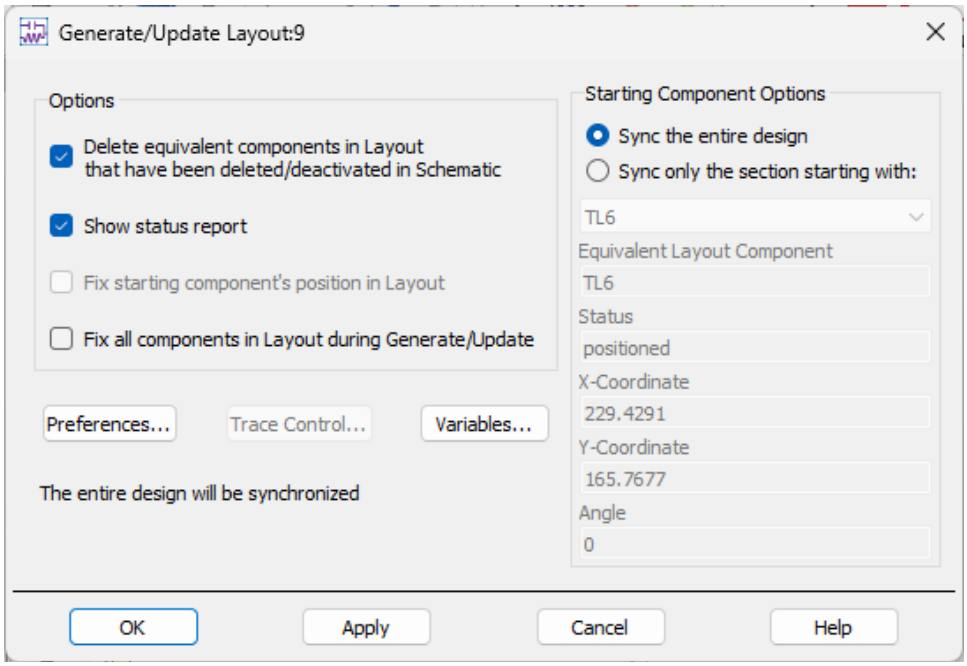
2. Create a schematic for layout using transmission line components for each trace and a corresponding substrate definition. Every corner and junction must be modeled to achieve proper layout geometry. Use pins for the ports and do not setup the simulation in this schematic. The following microstrip components are commonly used (each type of transmission lines has their respective components):
    - Microstrip line (MLIN)
    - Microstrip coupled lines (MCLIN)
    - Microstrip stubs (MLEF, MLOC, MLSC, MRSTUB, MBSTUB, etc.)
    - Microstrip tee (MTEE)
    - Microstrip bend (MSABND, MSOBND, MCORN, etc.)
    - Microstrip taper (MSTEP, MTAPER)
    - Vias (VIA, VIAGND, etc.)
    - Microstrip substrate (MSUB)

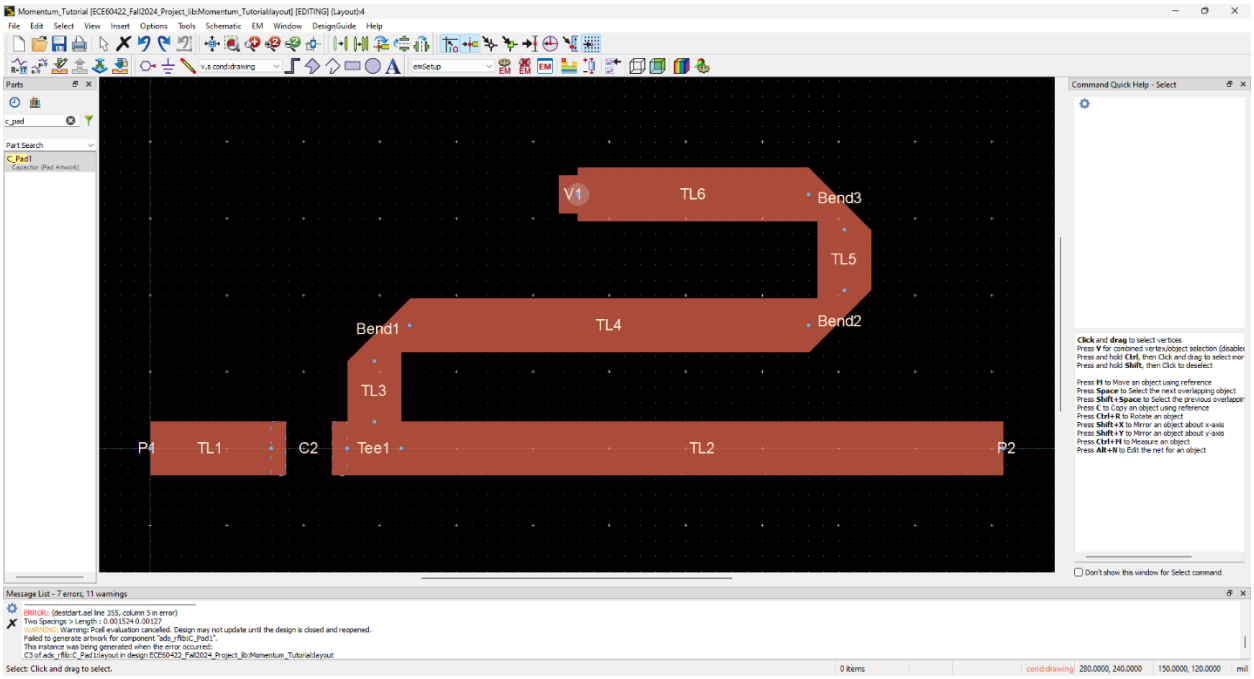

VIAGND
V1
Subst="FR4"
D=15.0 mil
T=0.15 mil
Rho=1.0
W=25.0 mil
MLIN
TL6
Subst="FR4"
W=w
L=l/2-2*s-3*w
MSOBND_MDS
Bend3
Subst="FR4"
W=w
MLIN
TL5
Subst="FR4"
W=w
L=s
MSOBND_MDS
Bend2
Subst="FR4"
W=w
MLIN
TL4
Subst="FR4"
W=w
L=l/2-s-w
MSOBND_MDS
Bend1
Subst="FR4"
W=w
MLIN
TL3
Subst="FR4"
W=w
L=s
P1
MLIN
TL1
Subst="FR4"
W=w
L=2 mm
C_Pad1
C2
C=2.2 pF
W=w
S=10 mil
L1=50 mil
MTEE_ADS
Tee1
Subst="FR4"
W1=w
W2=w
W3=w
MLIN
TL2
Subst="FR4"
W=w
L=10 mm
P2
MSub
MSUB
FR4
H=0.5 mm
Er=4.6
Cond=5.8e9
T=35 um
TanD=0.02
VAR
VAR1
w=0.896 mm
s=1 mm
l=17.044 mm

3. After the schematic is completed, the layout can be generated from the top menu by clicking on Layout > Generate/Update Layout.

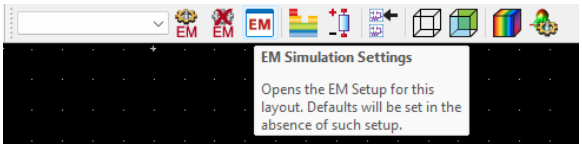


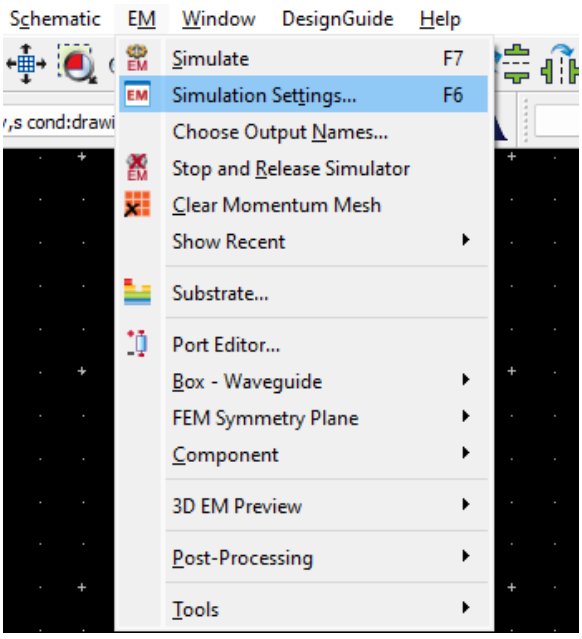
Enable the option to delete equivalent components and show status report, sync the entire design and click on OK to generate the layout. Close the status window. If the schematic changes, the same process can be followed to update the layout.



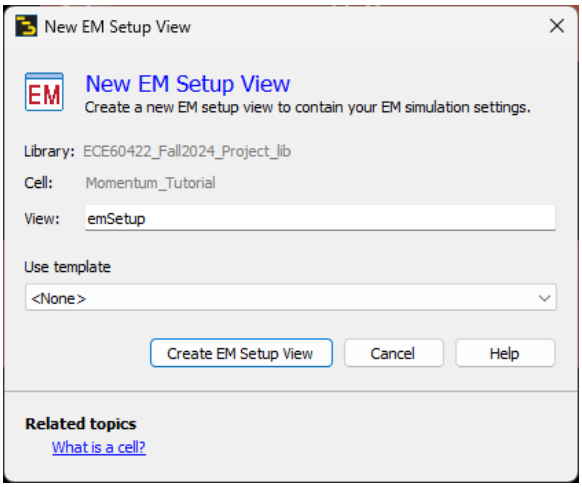


4. Open the EM Simulation Settings with the toolbar icon, by clicking on EM > Simulation Settings, or with the shortcut key F6

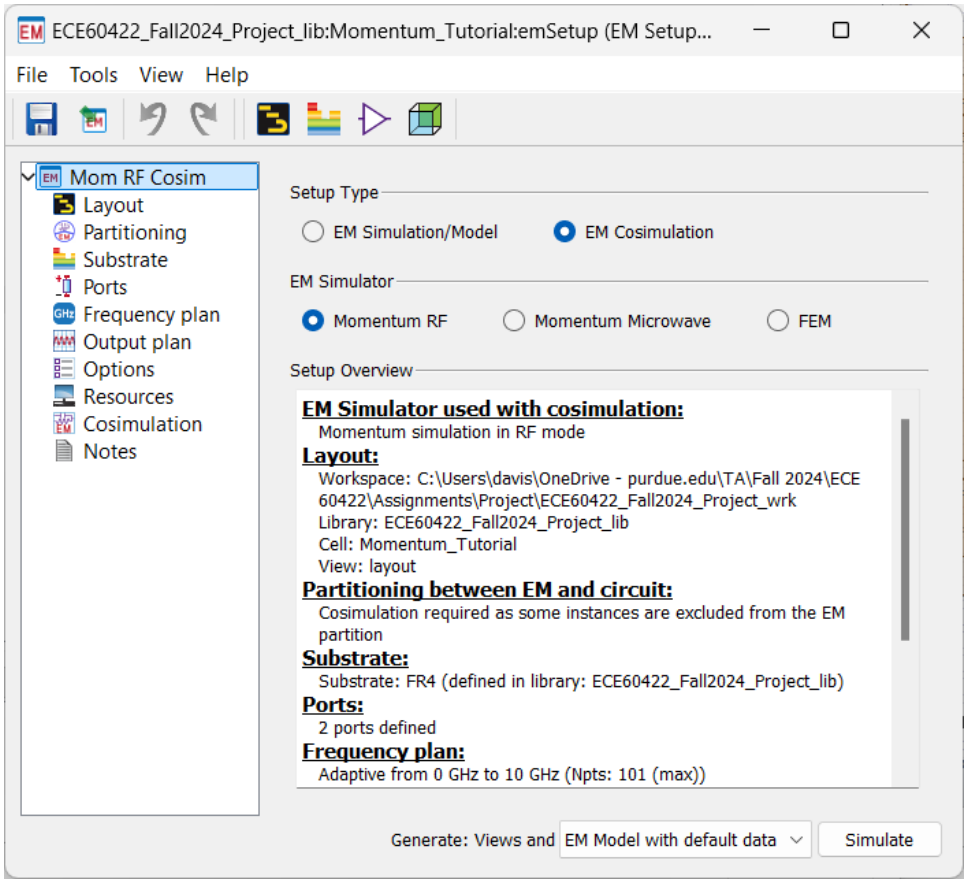




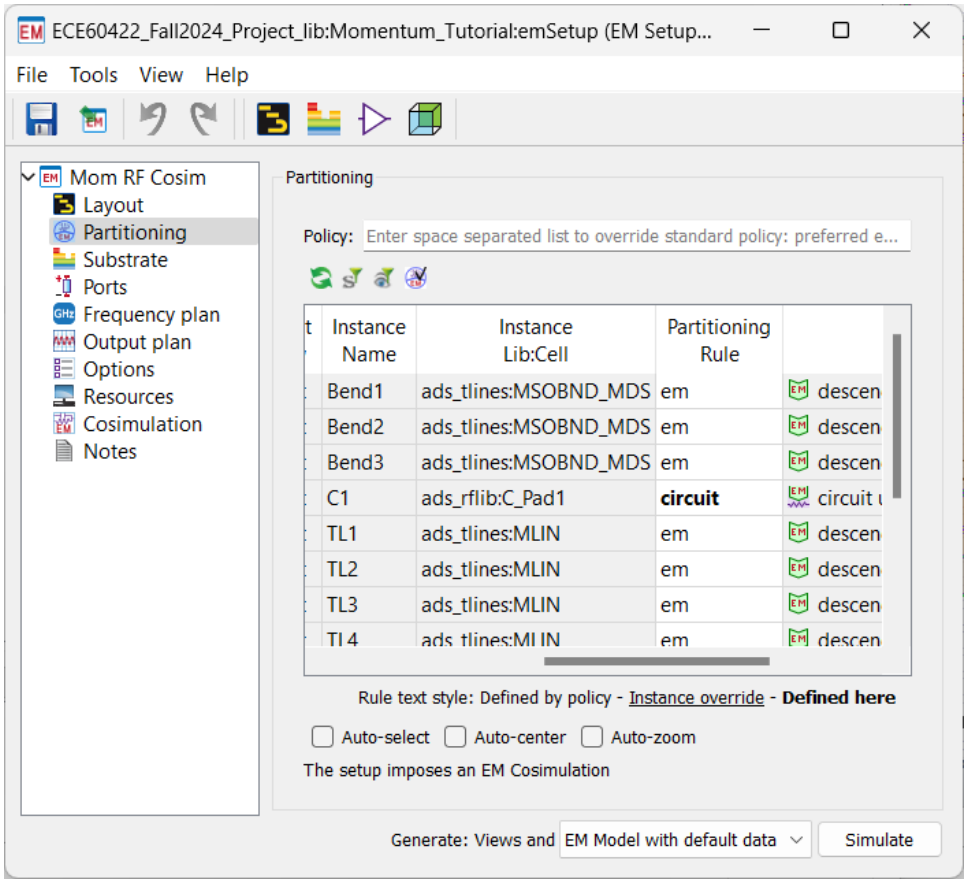
Create a new EM Setup View, the name of the view can be changed. Do not select any template for this tutorial, all the settings will be configured from scratch to learn how to use Momentum.



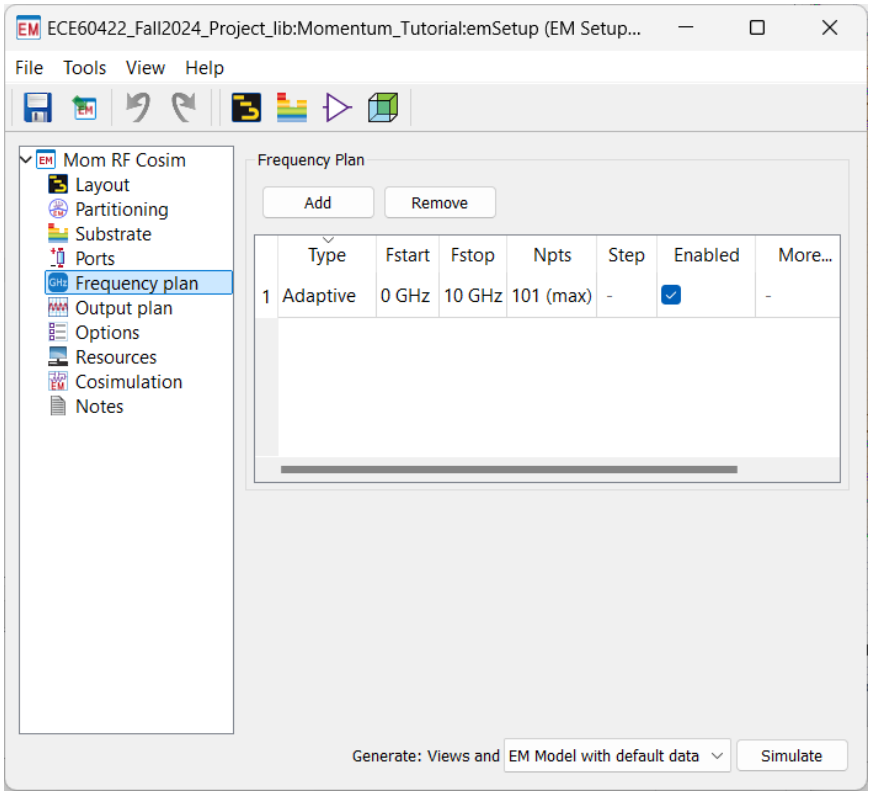
In the EM Setup View, select EM co-simulation. This enables the use of schematic component symbols that don't have a corresponding footprint such as lumped elements. Momentum RF can be used when the circuit is electrically small, is geometrically complex, and does not radiate. Use Momentum Microwave for problems that involve radiation effects. FEM is used for 3D finite element electromagnetic simulations.



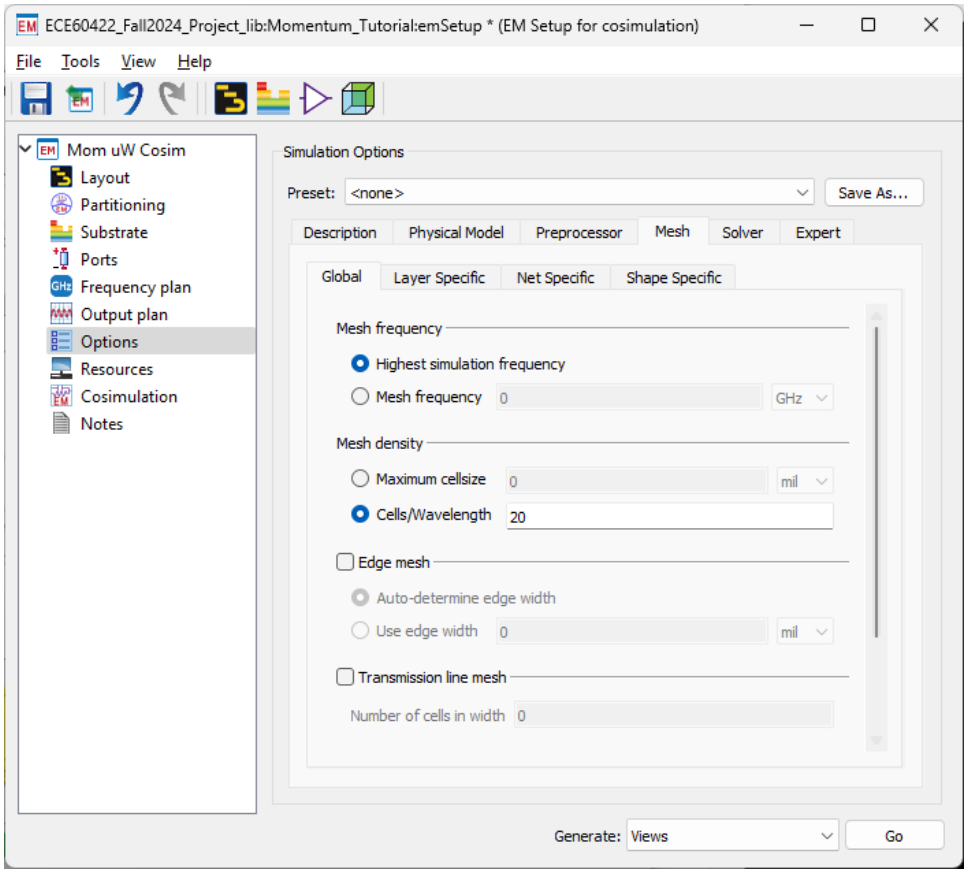
5. The partitioning window allows to choose between EM and circuit simulation. Lumped elements (resistors, inductors, capacitors, etc.) should not be included in the EM partitioning.



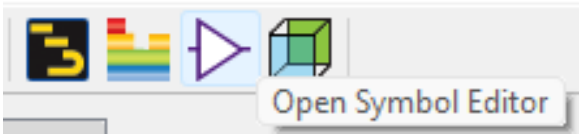
6. Adjust the frequency plan tab to simulate 101 points from DC to 10 GHz with adaptive steps.

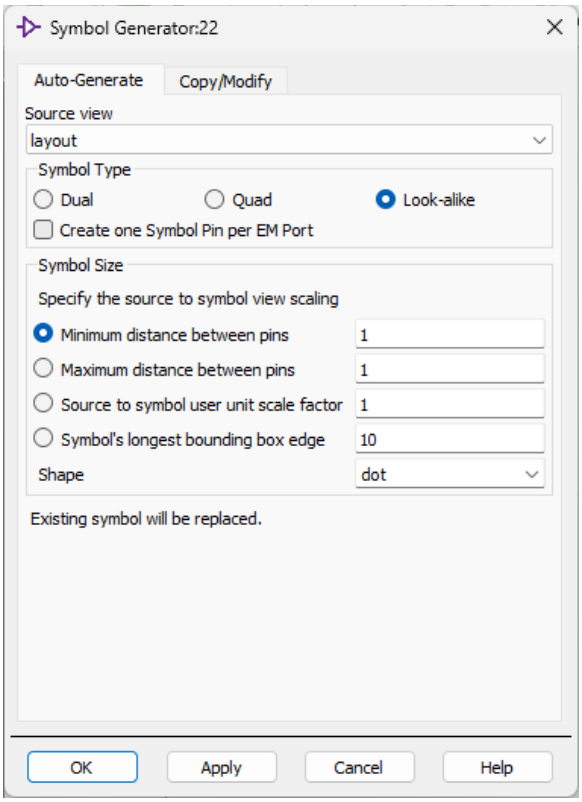


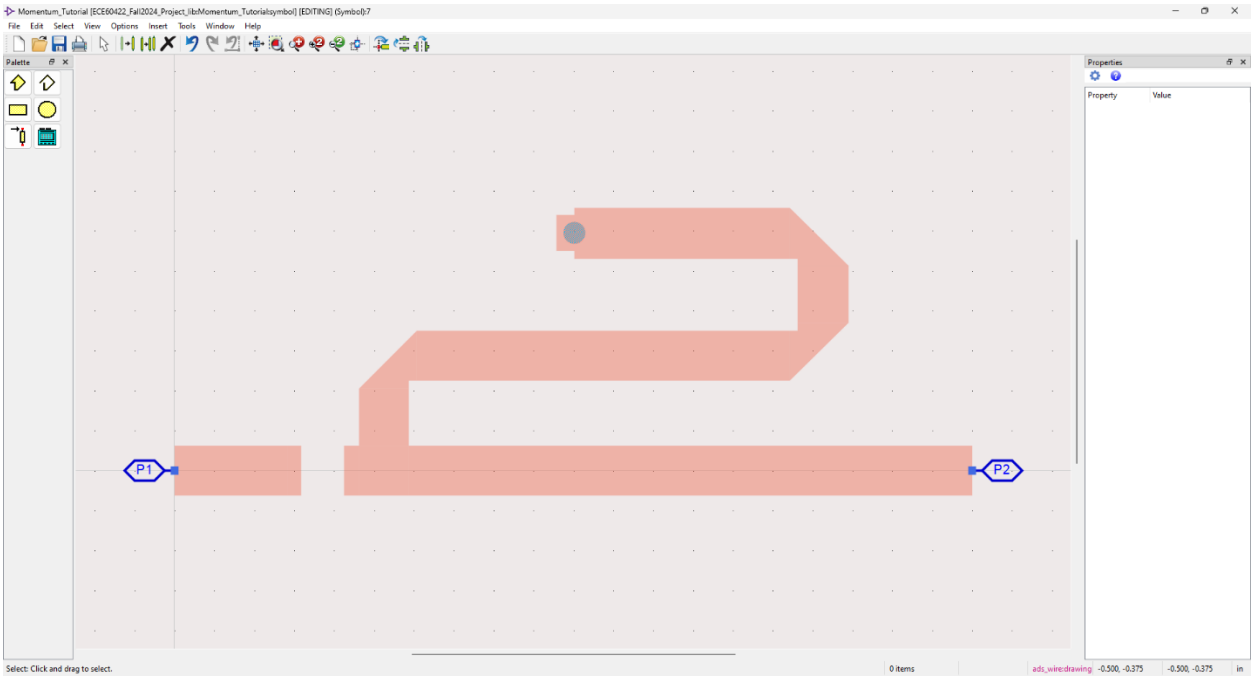
7. The mesh settings can be changed in the Options > Mesh tab. Default values work well but can be configured to achieve desired accuracy and solution complexity (computing time, memory, etc.).



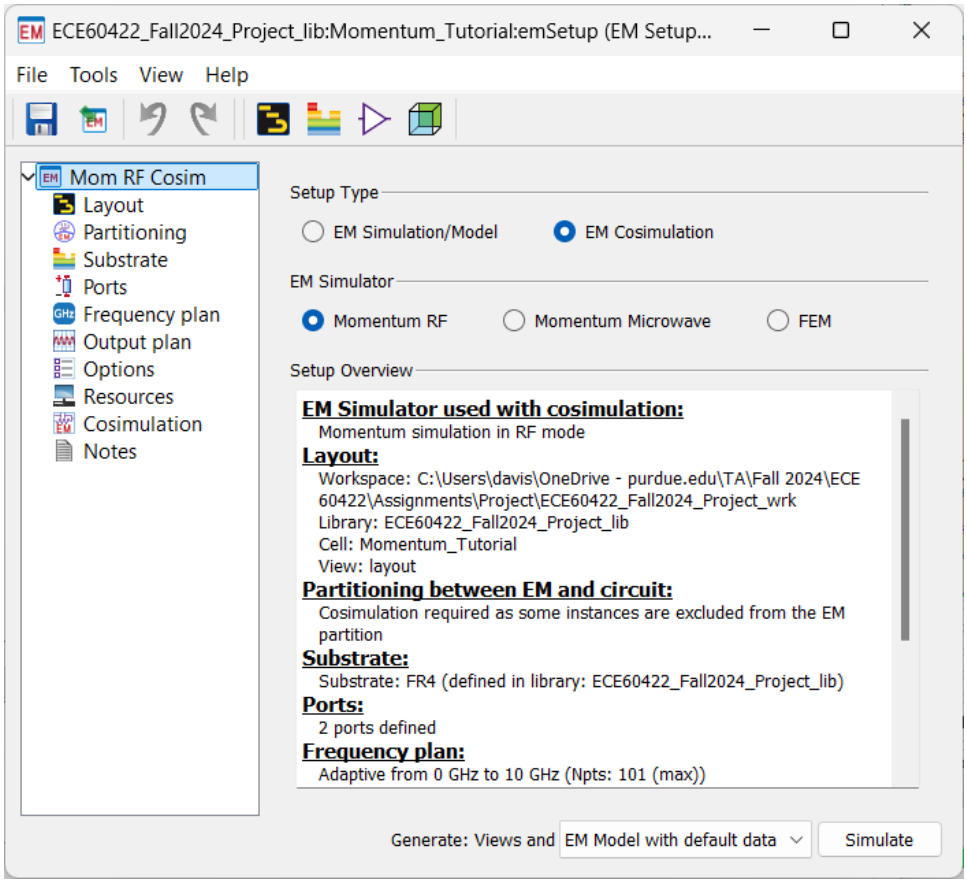
8. Click on the symbol button in the top toolbar to generate a layout schematic symbol. Select *layout* in the source view, and Look-alike in the symbol type. Click OK to generate the symbol.



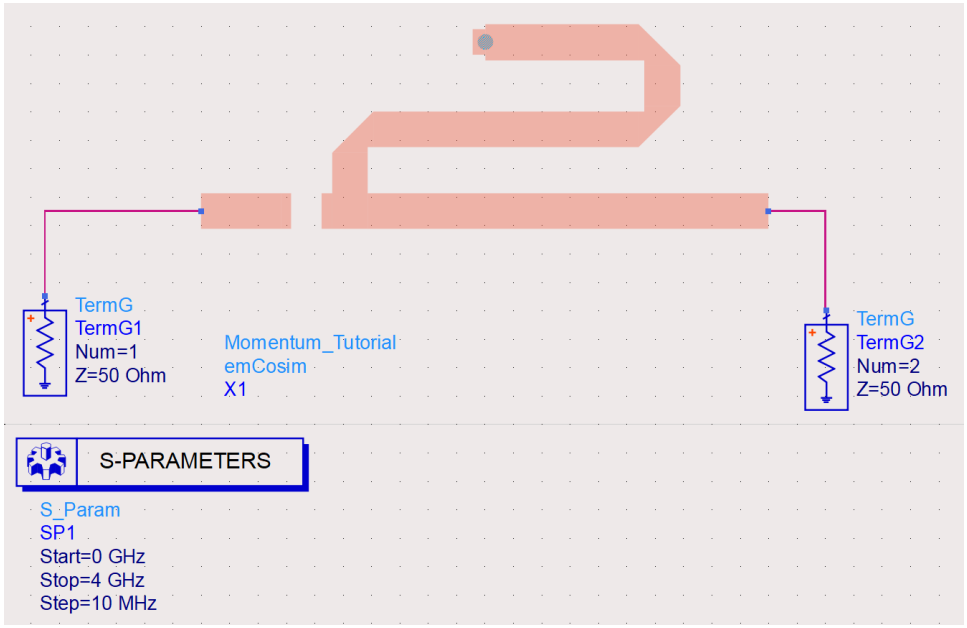

Symbol Generator:22
Auto-Generate
Copy/Modify
Source view
layout
Symbol Type
Dual
Quad
Look-alike
Create one Symbol Pin per EM Port
Symbol Size
Specify the source to symbol view scaling
Minimum distance between pins
1
Maximum distance between pins
1
Source to symbol user unit scale factor
1
Symbol's longest bounding box edge
10
Shape
dot
Existing symbol will be replaced.
OK
Apply
Cancel
Help


P1
P2

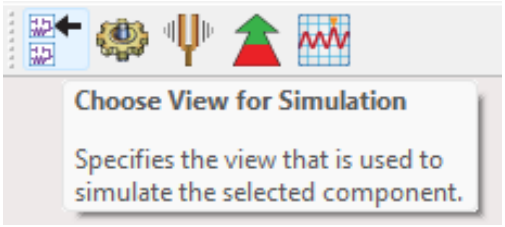
9. Select *EM Model with default data* in the “Generate: Views and” dropdown. Run the simulation by clicking on Simulate. You might need to allow access to the network if prompted by ADS.

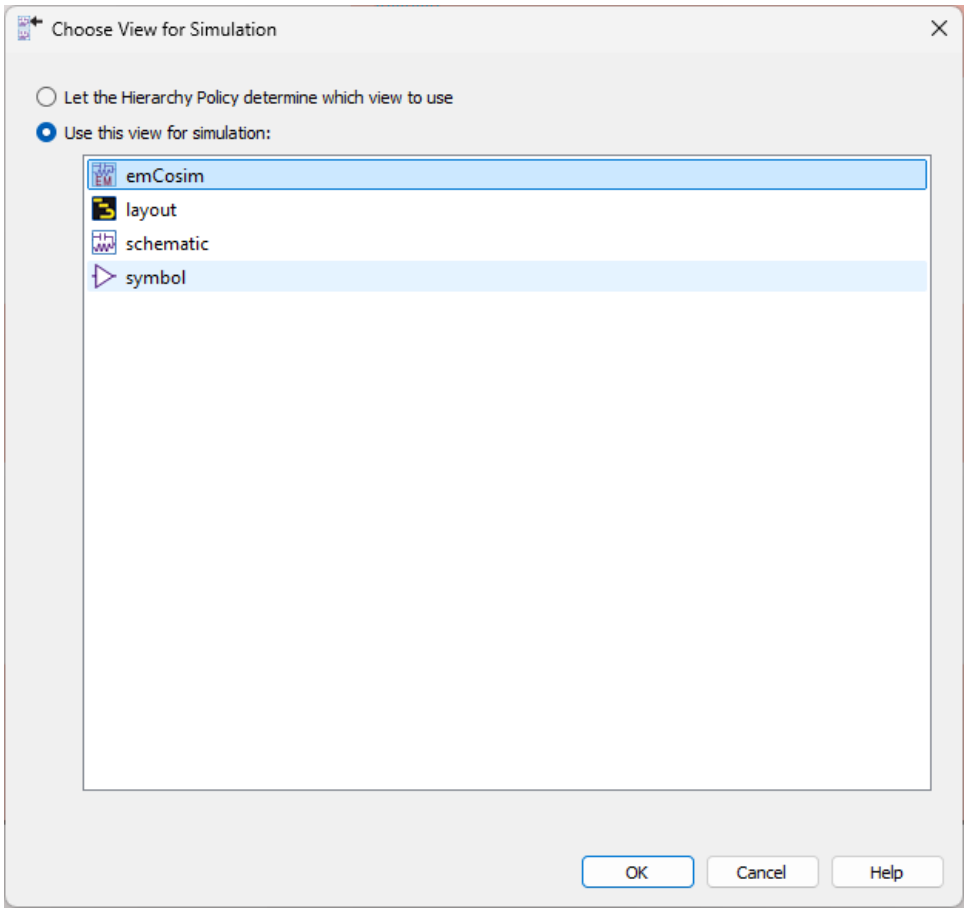


10. Drag and drop the symbol from the main window onto a new schematic. Add the lumped elements and complete the simulation setup.

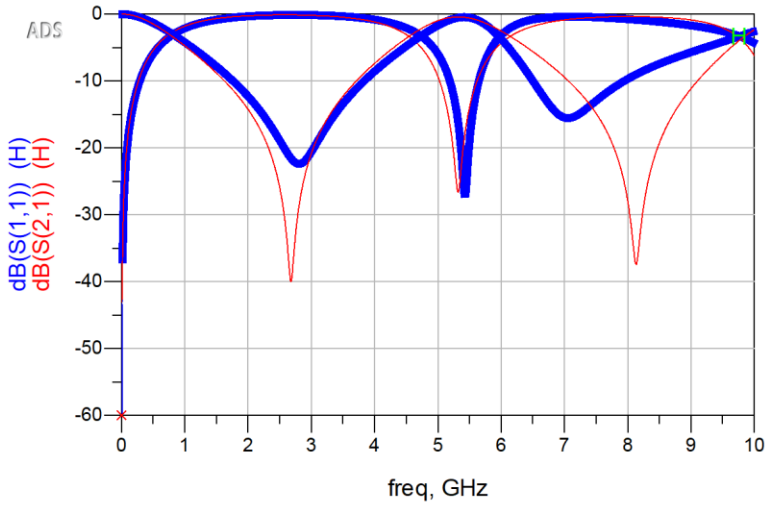


11. Select the layout and click on *Chose View for Simulation*. Select the corresponding view to simulate (emCosim, layout, or schematic).



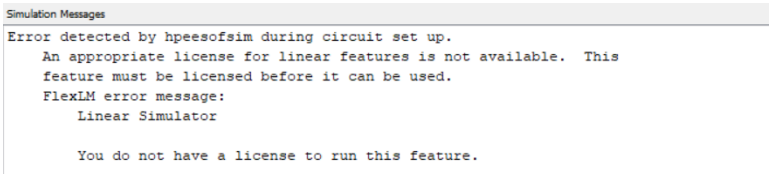


12. Compare the results from schematic and Momentum by changing between the view for simulation as shown before.



4. Troubleshooting of Common Issues

4.1. License



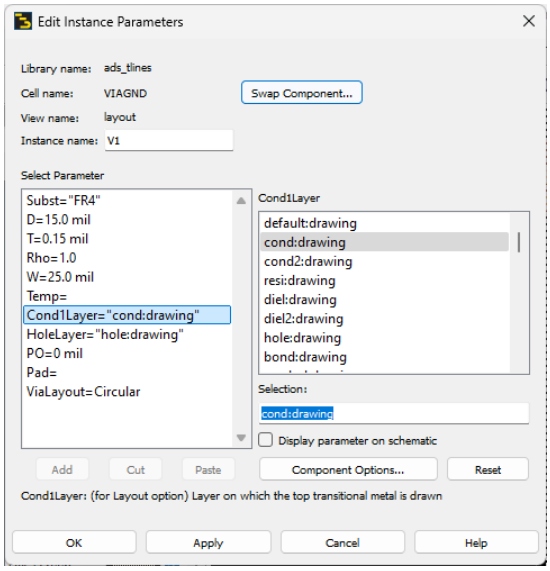
If you have an error that says:

"FlexLM error message: Linear Simulator
You do not have a license to run this feature"

Save all the documents and restart ADS. This is likely caused by the license being inactive and restarting the application should let you open it again.

4.2. Via Layers

If the via is not present, appears disconnected from ground, or disconnected from the trace it might be assigned to incorrect layers in the substrate. Open the via component in the layout and verify the layer selection for Cond1Layer and HoleLayer. They should match the substrate layer names.



4.3. Assign Nets

If you have an error in the schematic that read:

"Error generating the netlist for "schematic_name.sch".
Failed to create netlist"

In the toolbar, select Tools > Assign Nets to Layout.

4.4. Cannot Launch Job Manager

If you have an error when ruing the Momentum simulation that prevents the job manager from opening, check in the EM Setup Window > Resources tab that you're selecting ADS (local/remote) for job management. This can also be caused by installing ADS outside of the main C:\ drive.

5. Conclusion

Now you are able to simulate the effects of layout parasitics and perform electromagnetic simulations in ADS. Note that the circuit present is a DC open and an RF short at 2.4 GHz. Follow this guide for any homework and projects that require EM simulations with Momentum.