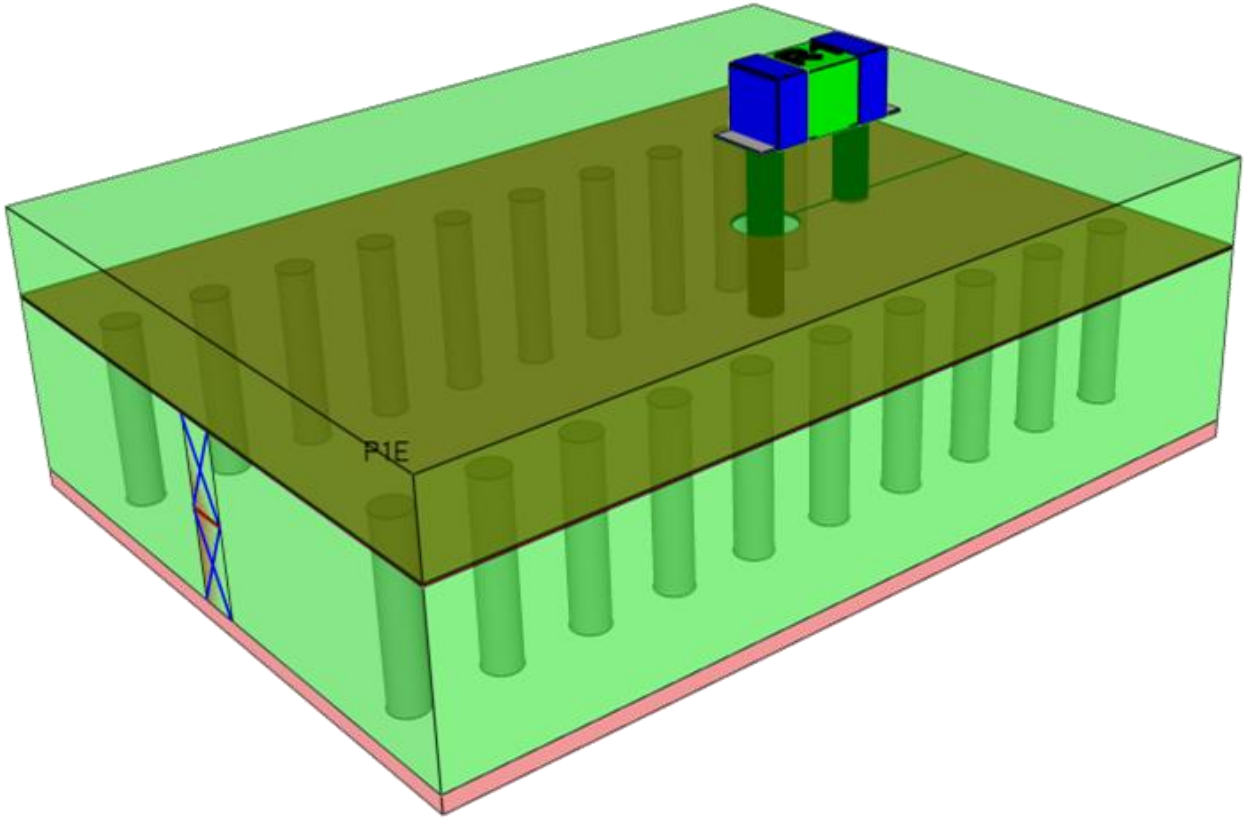
EMPIRE XPU Tutorial

LTCC Transition

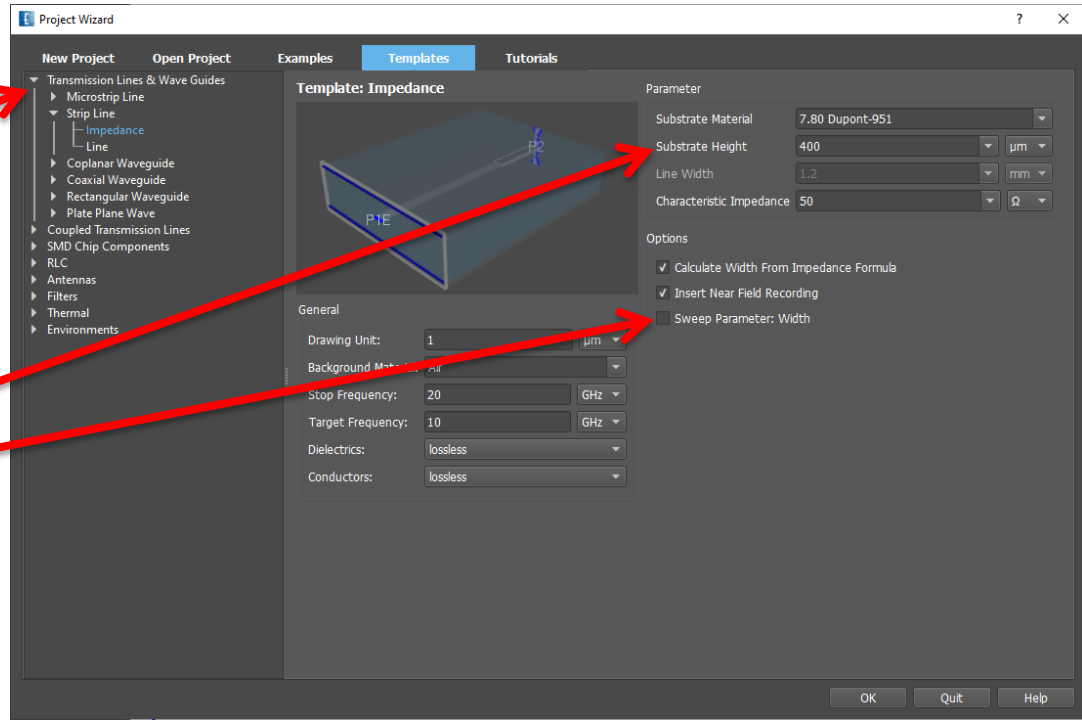
Overview: Topics

- Template Wizard
- Simulation
- Insert SMD Resistor
- Parameter Sweep
- Via Fence



Step 1: Stripline in Dupont 951

- Start Empire XPU
- Select Templates → Transmission… → Strip Line → Impedance
- Keep Substrate Material 7.8 Dupont-951
- Set „Substrate Height“ to 400 um
- Uncheck „Sweep Parameter: Width“

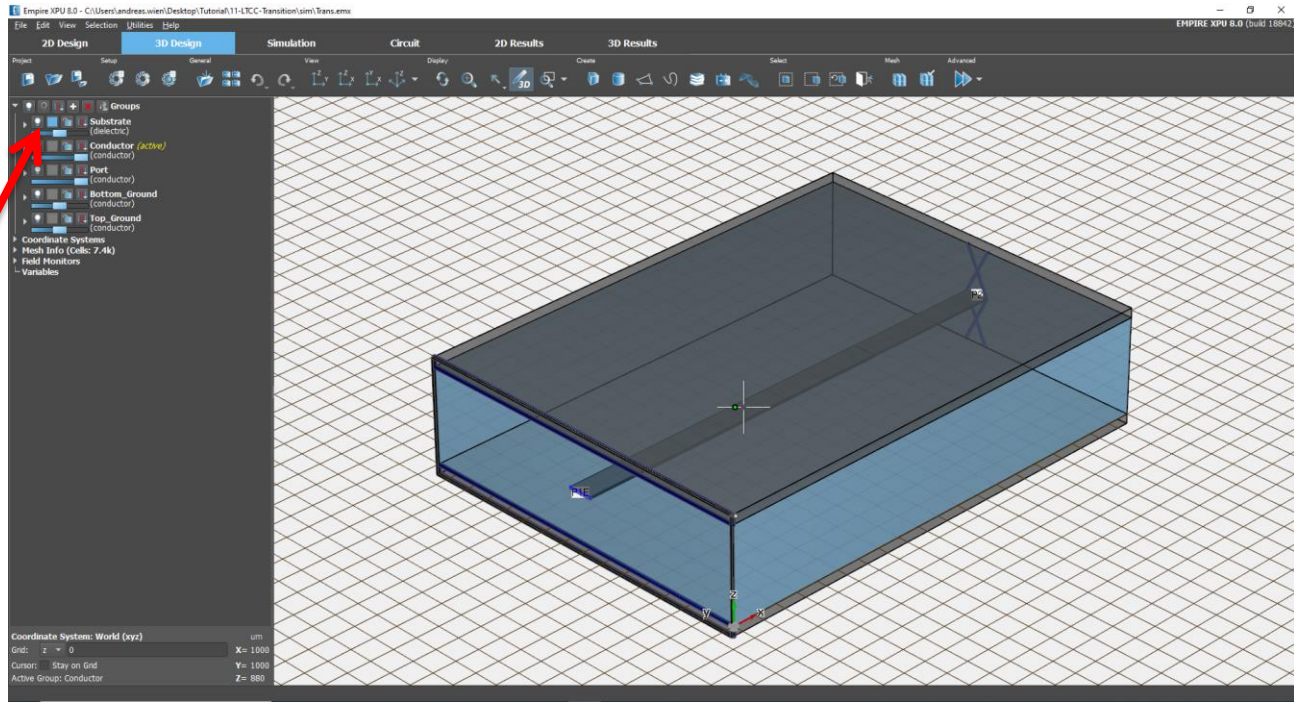


- Click OK
- Select “File -> Save As”
- Select folder and enter name, e.g. Trans.emx

Comments:
Using the template “Strip line…”
• Automatic generation of Dielectric, port and mesh
• QTEM port at start, absorbing port at end
• 50 Ohm line, designed by formula

Step 2: Structure Check

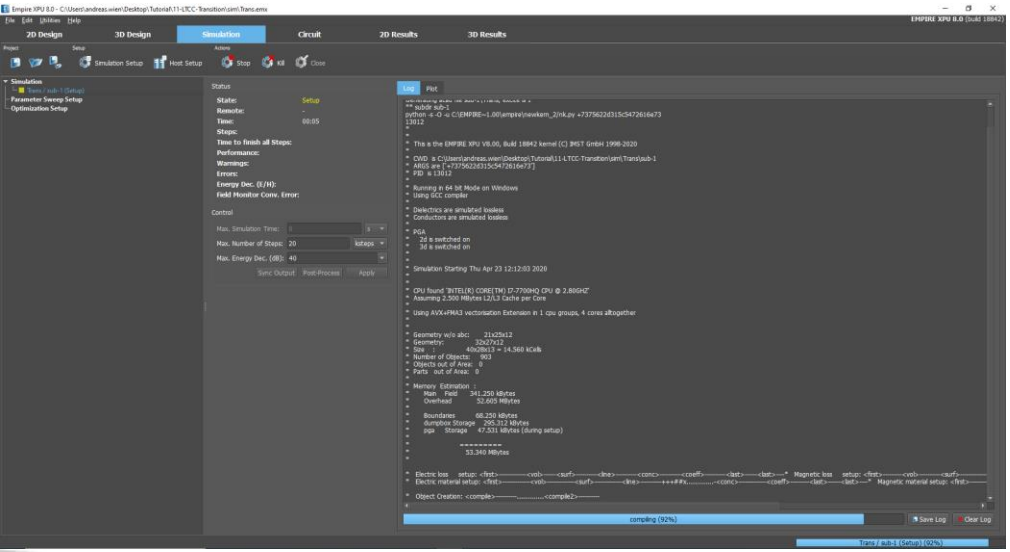
- Check the model
- Increase transparency of groups
- Optionally recolor groups
- Open groups to verify heights and properties



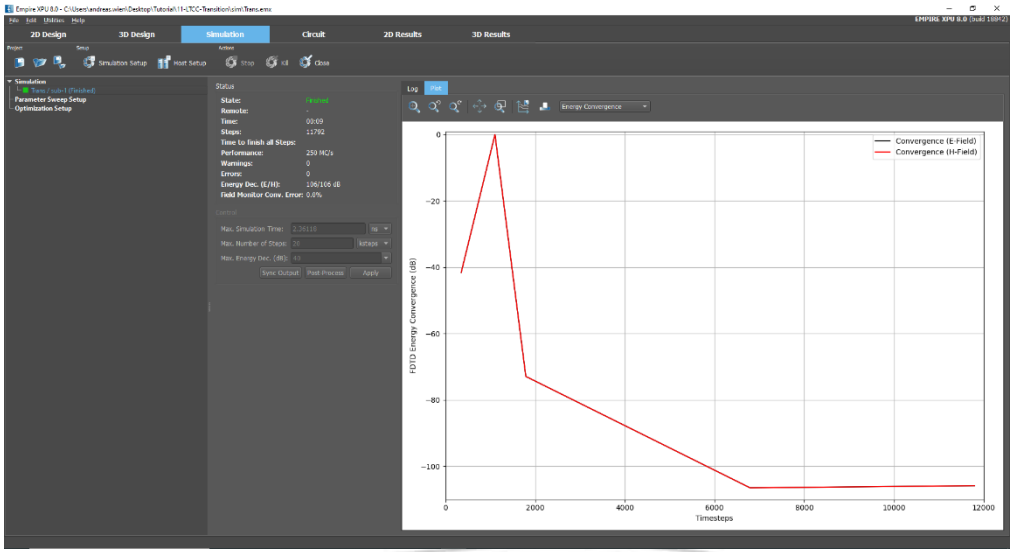
Step 3: Stripline Simulation

- Click “Start Simulation“ → “OK“

Log Tab: Compilation progress

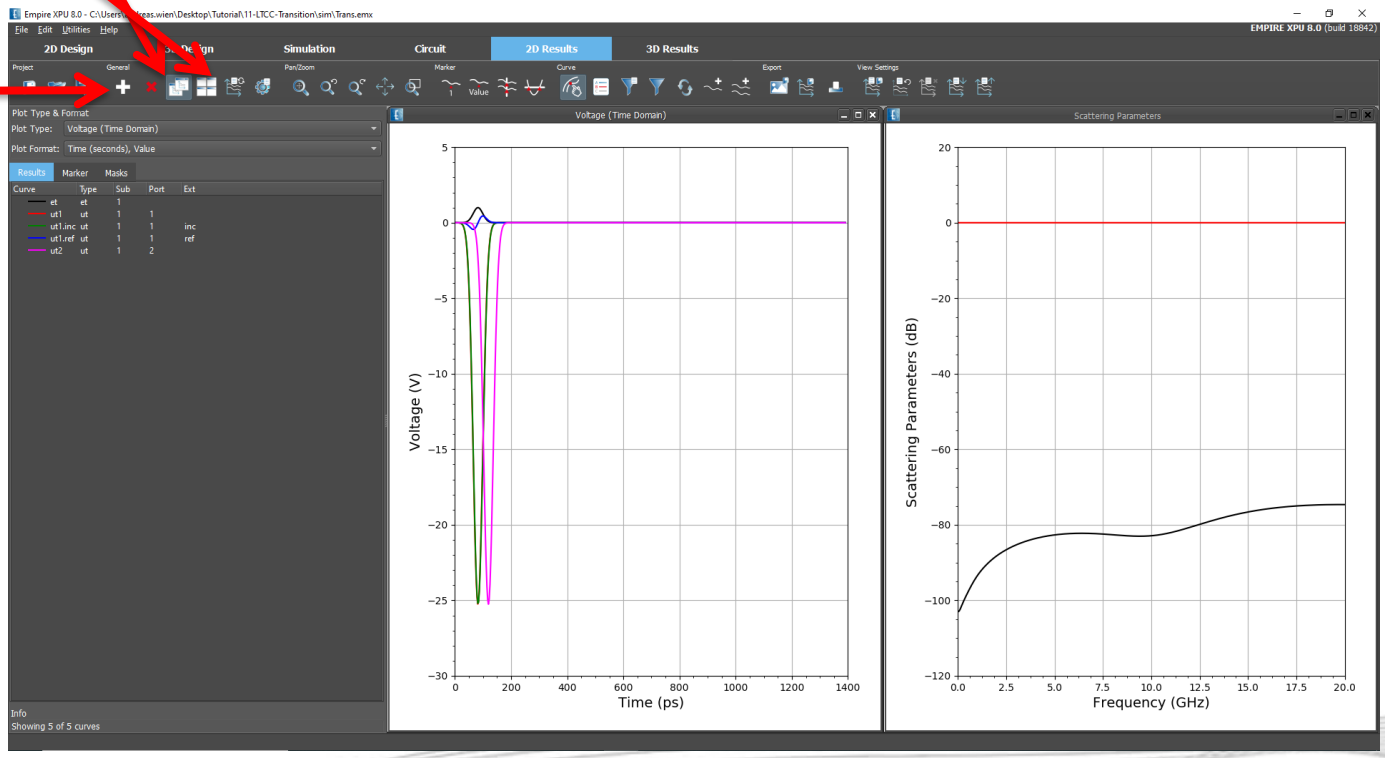


Plot Tab: Energy vs. Timesteps



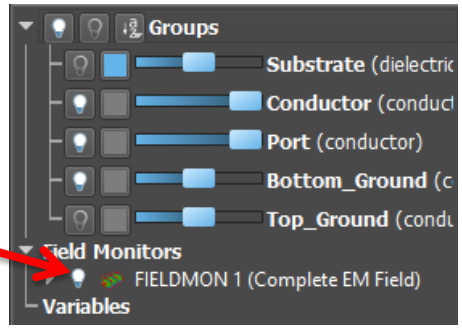
Step 4: 2D Postprocessing Results

- Switch to “2D Results” tab. Plot Type: Scattering Parameters
- Click General: Add Result. Change Plot Type to: Voltage (Time Domain)
- Click Toggle View Mode.
- Click Tile Sub Windows

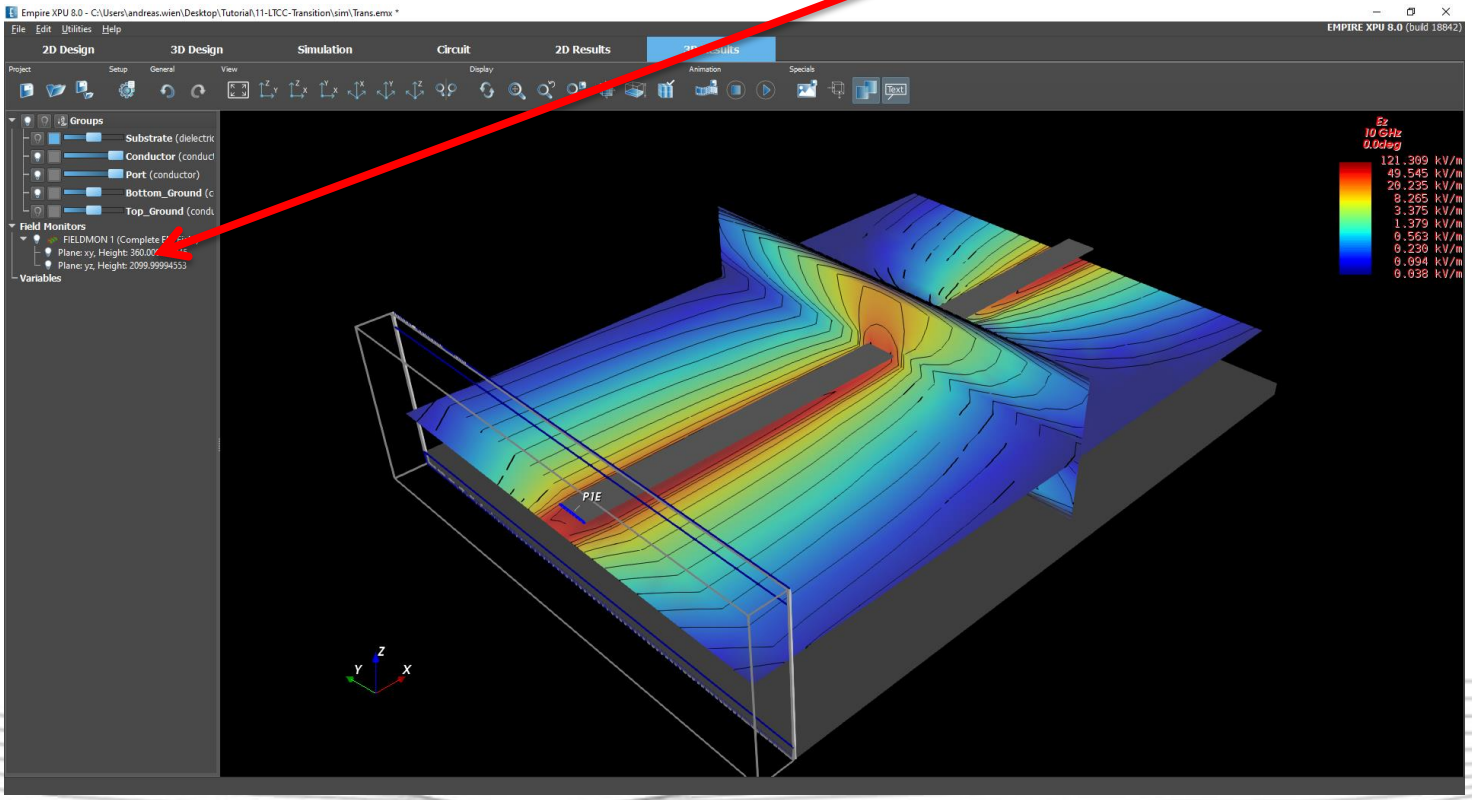


Step 5: Near Field

- Switch to „3D Results“ tab, Click „Iso z View“ 
- Open Field Monitors – Switch On FIELDMON 1
- Turn off visibility of Top_Ground and Substrate groups



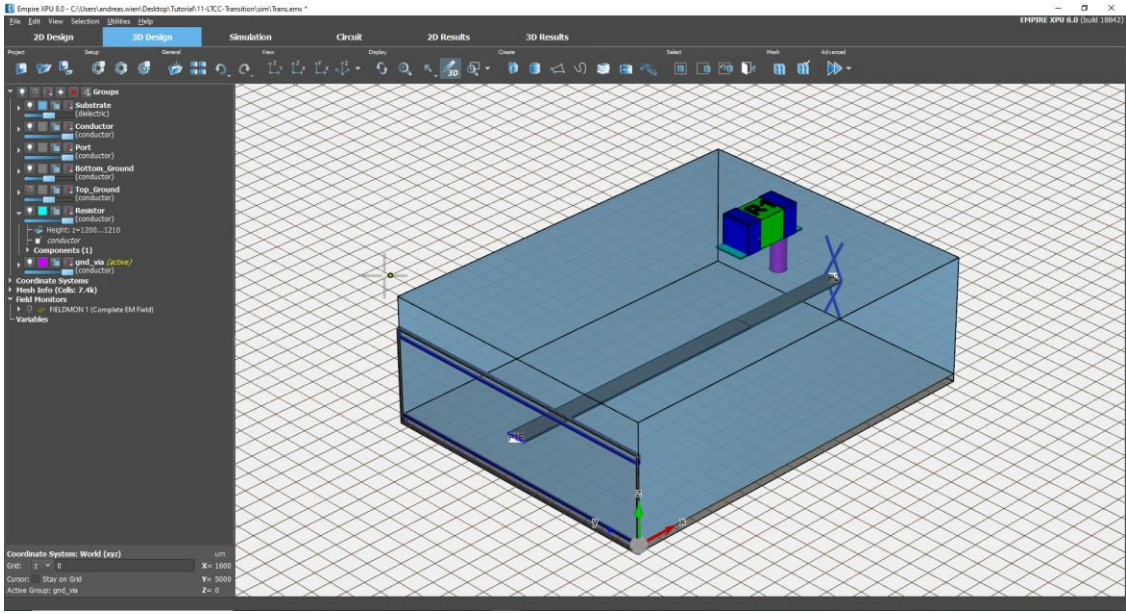
Hint: Move field monitor planes to adjust the field display planes (double click on plane height to adjust the height)



Step 6: Insert SMD Resistor

- Go back to „3D Design“ Tab – Iso Z View, Switch off Field Monitor
- Open group Substrate, turn on group
- Double click „Height“, set 0…1200, confirm „Assign new height …“

- Click „Add Group“  , enter Resistor
- Set height 1200…1210
- Click “Create Library Object“ 
- Tab: “SMD“ → Click “SMD Resistor“
- Left click at Point: x=3000, y=1500
- Click „Edit Settings“
- Set size SMD0201, Press Ok (2x)

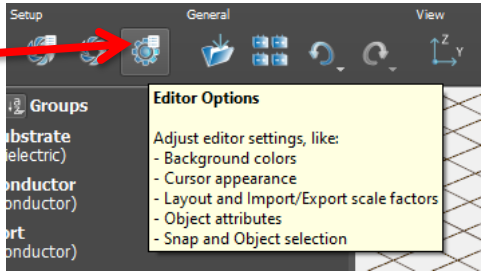


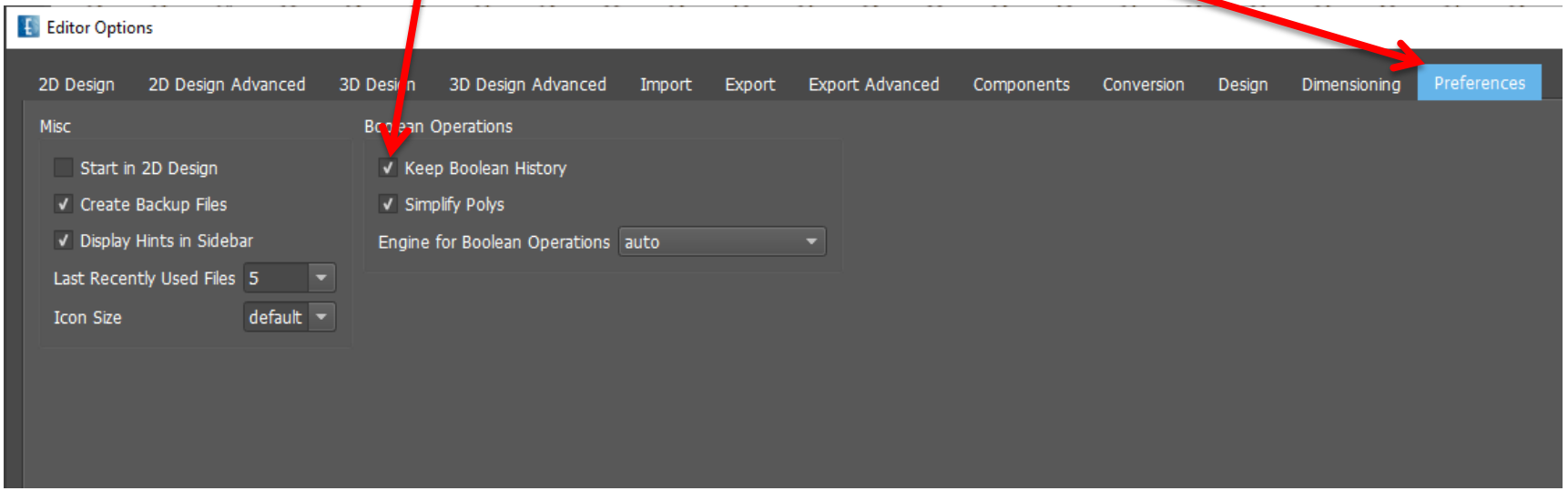
- Click „Add Group“, enter “gnd-via“
- Hide group: Resistor, zoom in
- Click „Create Cylinder“, 
- Click at Point : x=3250, y=1500
- Press Tab and define:
  - height z: 800 … 1200
  - radius r=80
- OK

Step 7: Boolean History

For the next steps we need Boolean History to define a parametric hole

- Go to “Editor Options“
- Switch to “Preferences“ tab
- Check box “Keep Boolean History“
- OK





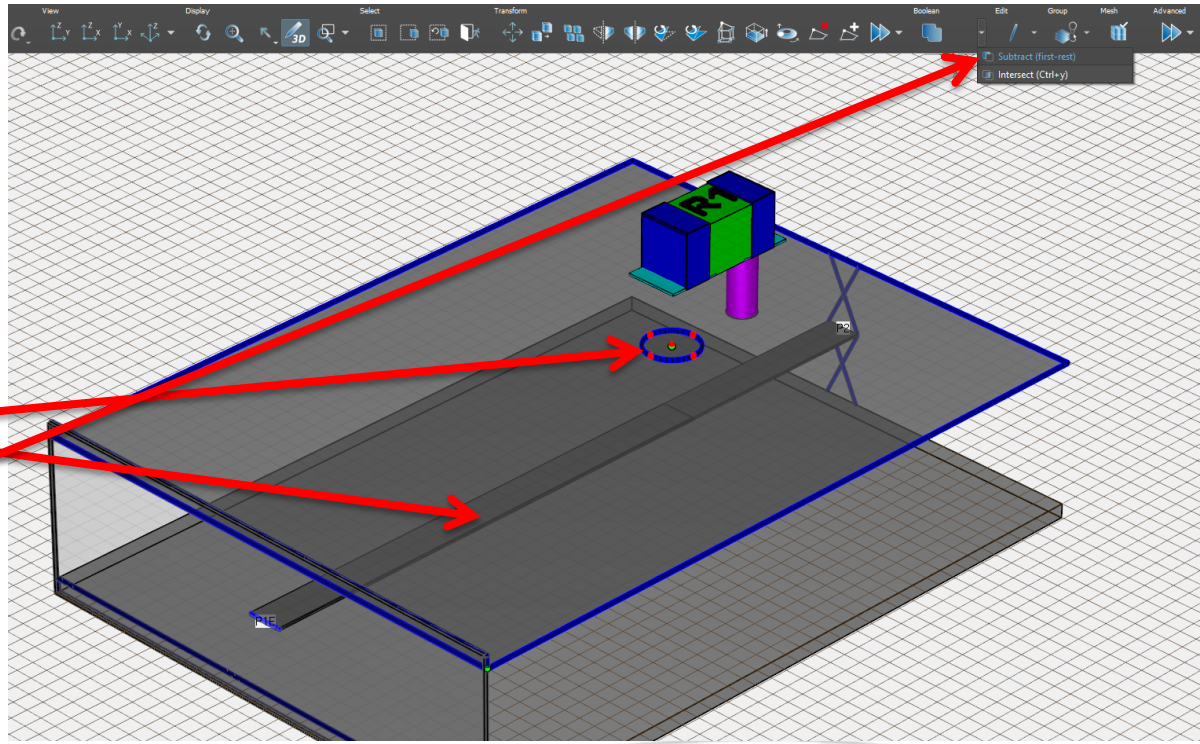
Step 7: Hole creation

- Hide group „Substrate“
- Switch on group „Top_Ground“
- Right click on group name, select „Set Active“
- Open group, double click „Height“, set 800…810, confirm Yes

- Click „Create Cylinder“, 
- Click Point : x=2750, y=1500, r=150
- Set height to dw= -10, OK

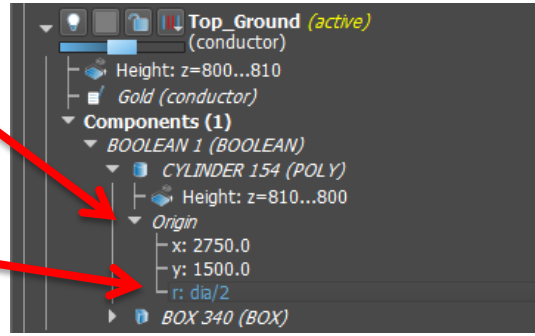
- Click the plane
- Click the cylinder

- In Boolean Menu click 
- Select Subtract (first-rest) 

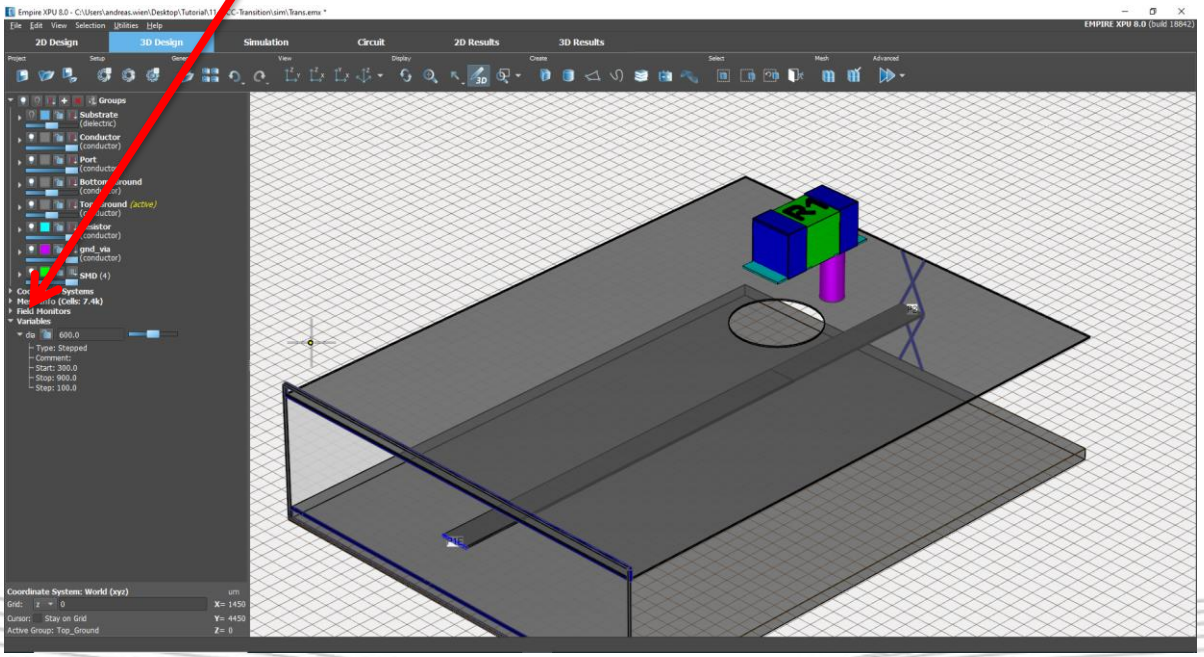


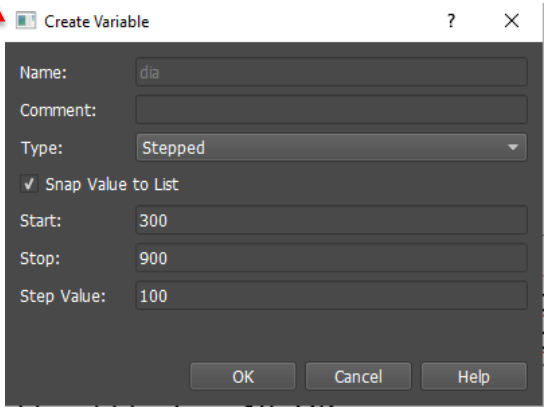
Step 8: Parametric Hole

- In Group „Top_Ground“ open Components – BOOLEAN – Poly - Origin
- Double click „r“, Enter: dia/2



- Set Type: Stepped, Start 300, Stop 900, Step 100, OK
- Check Variable range in side menu

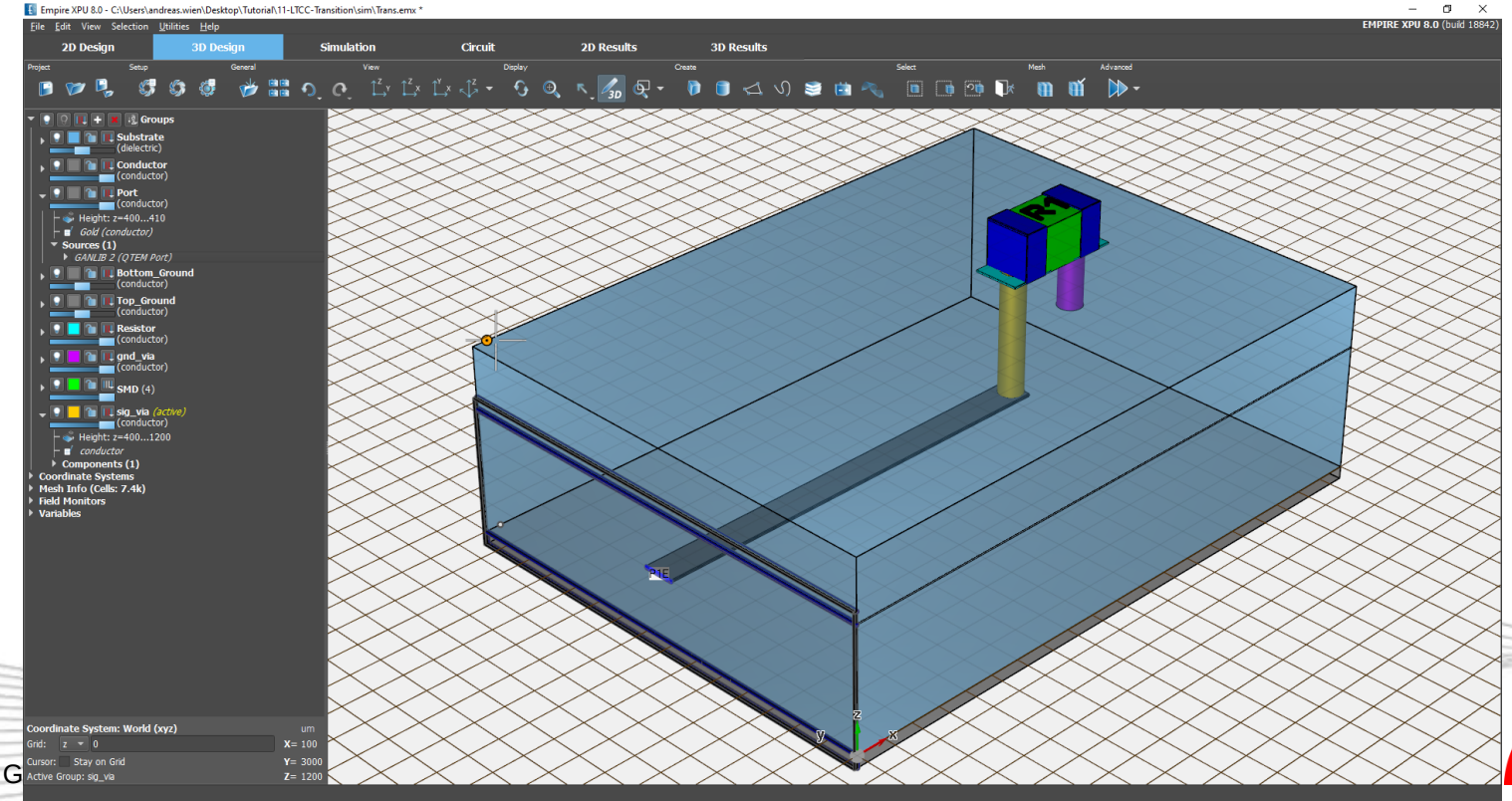




Step 9: Adjust Model, Via Hole

- Create group „sig-via“, set height 400...1200
- Show groups: Conductor & sig-via
- „Create Cylinder“ 
- Click Point: x=2750, y=1500, r=80, long click for group height, OK
- Unhide all groups 

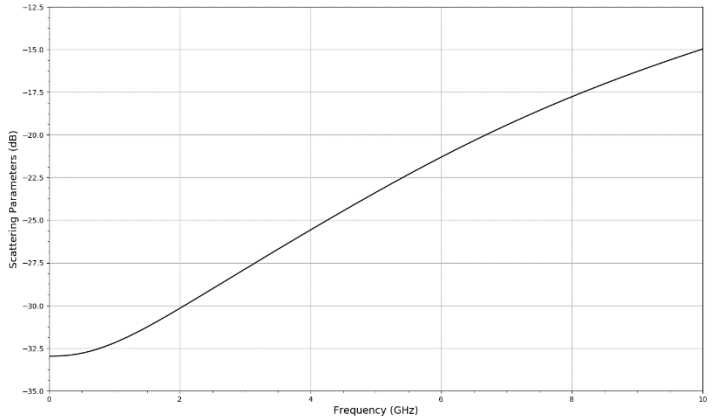
- Open Group Ports, open Sources, Click GANLIB 1 (Stripline), Click „Delete“ 



Step 10: Simulation Setup

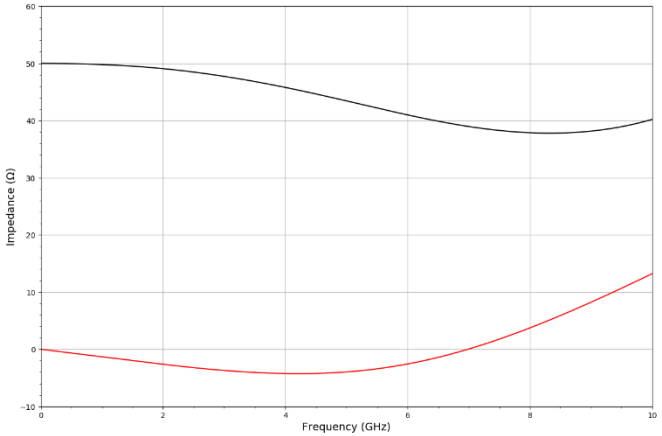
- Open „Simulation Setup“ , select „EM Setup“ Tab
- Set „Stop Frequency“ = 10GHz
- Select „Boundary“ Tab
- „Boundary Conditions“ ymin=Electric, ymax=Electric
- „Boundary Distance“ → Set zmax = 500, OK
- Click „Start Simulation“ , OK, wait for finish
- Select „2D Results“ Tab, Select Plot Type Scattering Parameters, Impedance

S11



20 dB Matching up to 6.5 GHz (dia=300)

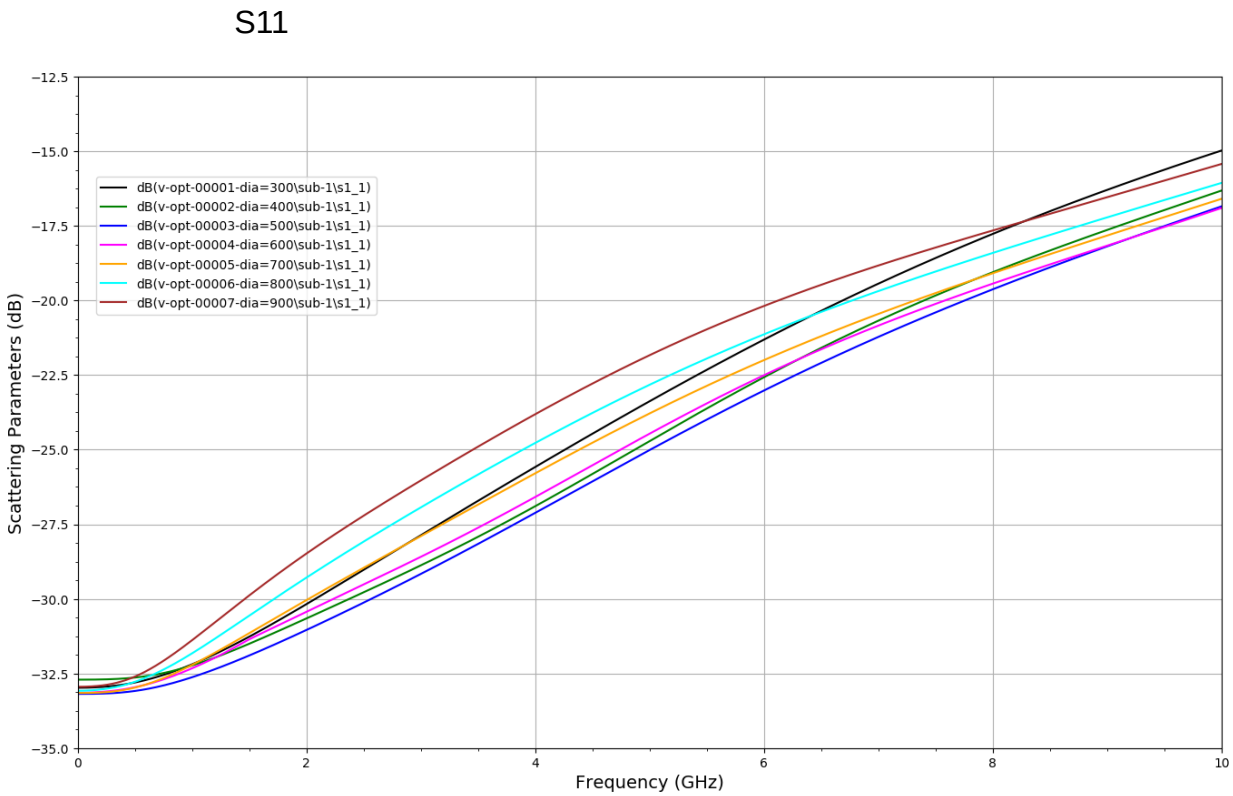
Impedance Z1.in



Capacitive for low frequencies

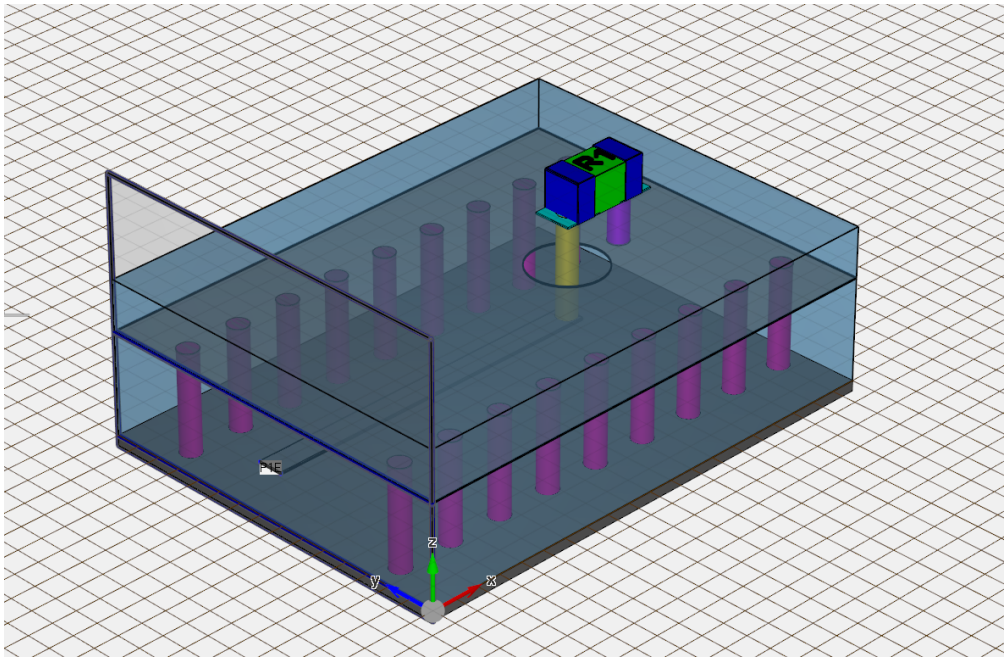
Step 11: Diameter Variation

- Select “Simulation“ Tab
- Click „Parameter Sweep Setup“
- Click „Create Parameter Sweep“
- Click „Start“, confirm
- Wait for Sweep to finish
- Select „2D Results“ Tab
- Plot Type: S-Parameters
- Select all Curves – right click – Show all



20 dB Matching up to 7.8 GHz with dia=500

Step 12: Via Fence



- Select „3D Design“ tab
- Create group „fence“, height 0…800
- Click „Create Library Object“
- Tab : „Layout“ → „Viafence“
- Click at x=0, y=500, (z=810)
- Click at x=4000, y=500
- Long click to finish and use group height
- Click „Edit Settings“
- Set: D1=160, D2=160, dist=450, 2x OK

- Right click on group „fence“, „Select group objects“
- Click „Copy & Mirror “
- Enter Start X=0, Y=1500 Stop X=10, Y=1500, OK

- Click „Simulation Setup“ → Boundary Tab
- Change „Boundary Conditions“: ymin = ymax = Absorbing Sheet, OK
- Click „ Start Simulation“, OK

Step 13: Animation

- „3D Results“ Tab, Switch on Field Monitor, Right click Edit, Use Optimization = Off, OK
- Switch off groups Substrate, Top_ground
- Monitor Planes xz=1500, xy=280