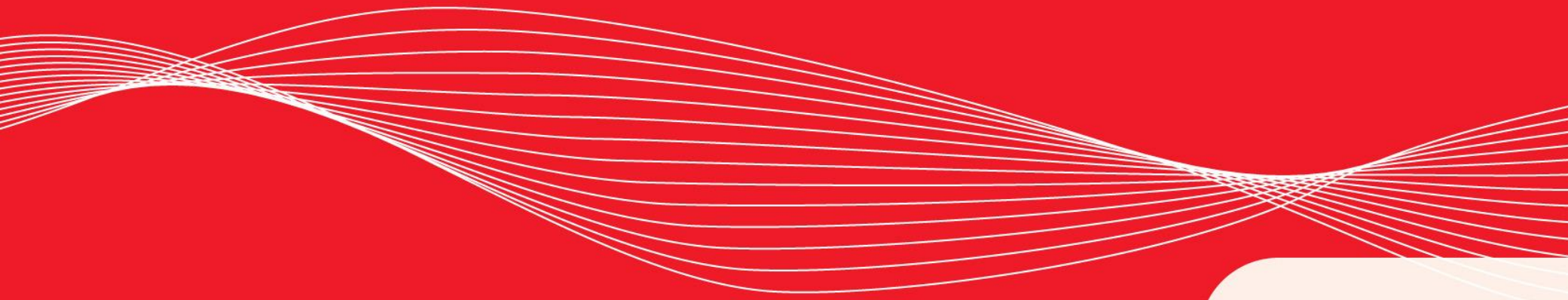
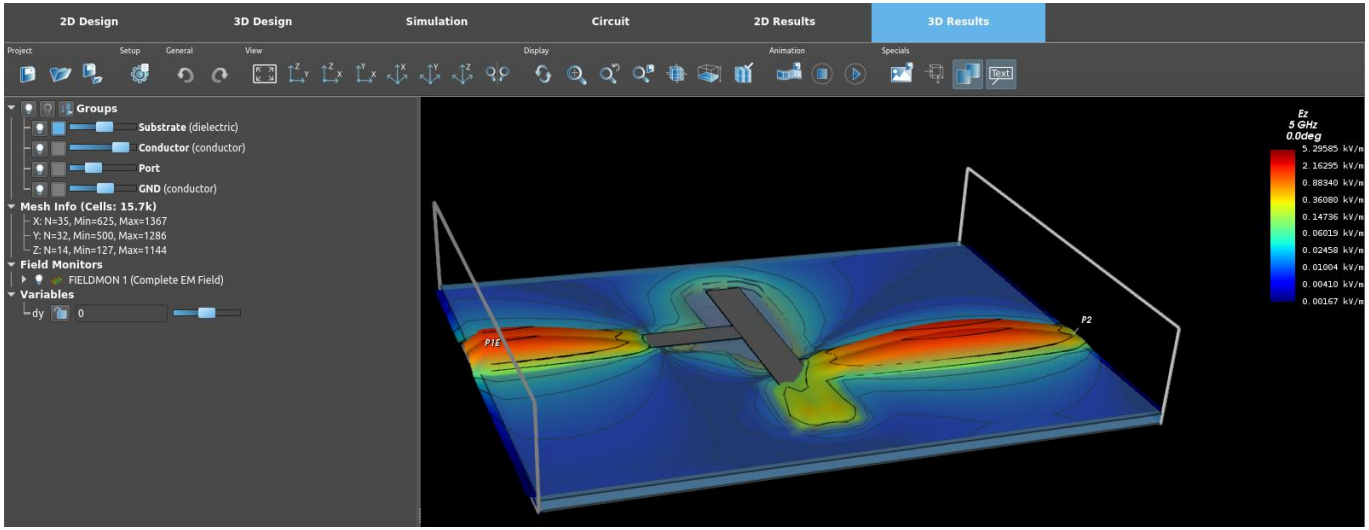
EMPIRE XPU Tutorial

Low Pass Filter



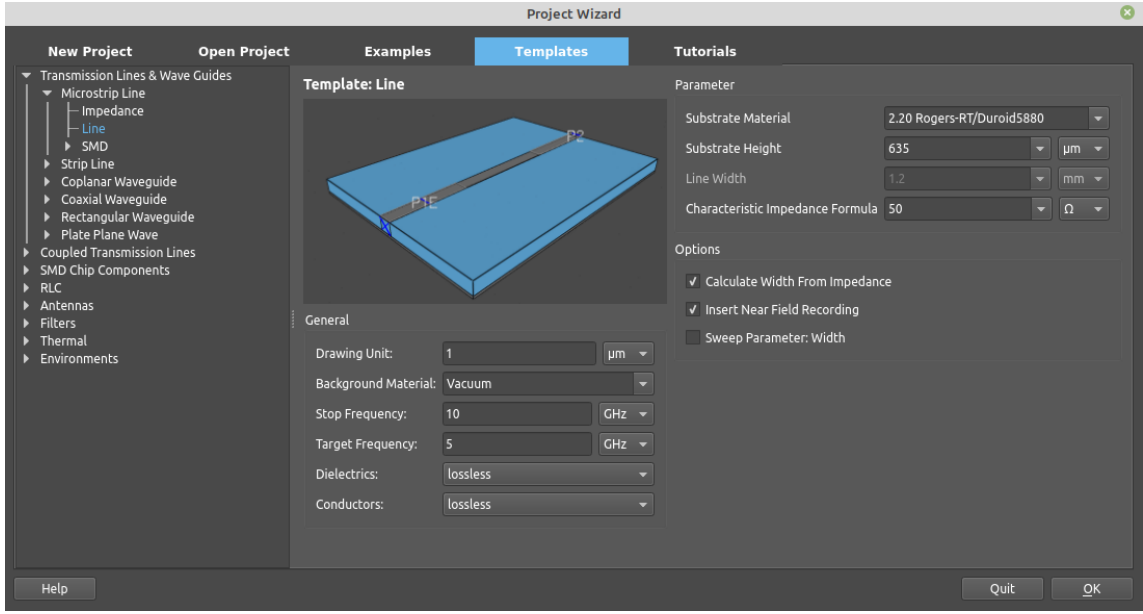
Overview: Topics

- Start using a template
- Basic features
- Mouse control
- Simulation flow
- QTEM ports
- Modify objects
- Animation



Comments: A three-button mouse with wheel is recommended for EMPIRE.

Step 1: Start



- Start EMPIRE XPU from Desktop
- Enter “10 GHz” in End Frequency and “5 GHz” in Target Frequency
- Choose “Rogers-RT/Duroid 5880” as Substrate Material
- Choose “635 µm” as Substrate Height*
- Press “Open Selection”

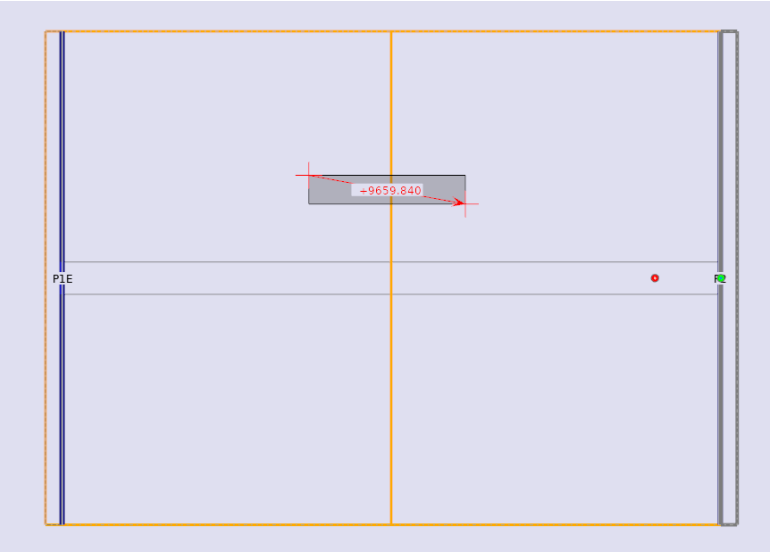
- Select File → Save As
- Enter, e.g. “C:\tutorial1\msl”

Comments:

- *Here, we want to adapt a template for the filter definition thus we can take advantage of the predefined groups and settings.*
- ** Please note the current unit is micron*

Step 2: Draft Mode, basic features

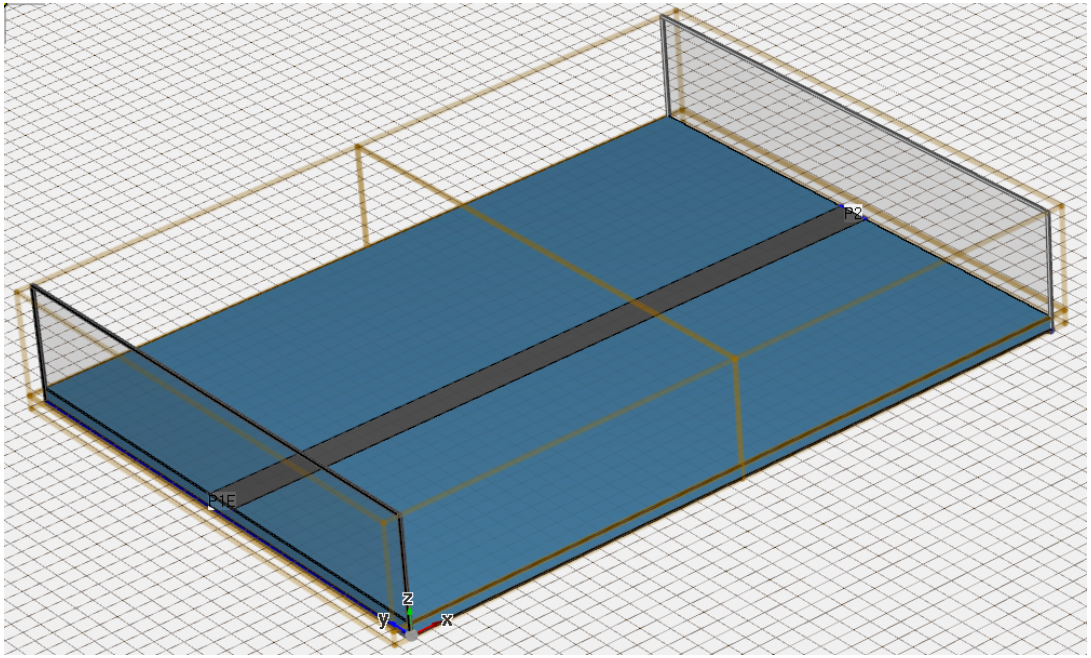
- Switch to “2D Design” Tab
- Move cursor in the drawing area
- Press and drag left mouse button to enter an arrow
- Press left mouse button on an object edge for selection
- Scroll wheel forward/backward or press Page up/down to zoom in/out
- Press up/down/left/right arrow on keyboard or scrollbars to pan
- Press Escape key to erase arrows and deselect objects



Comments: The 2D Design mode displays the structure in a wire frame model and is well suited for editing planar geometry.

Step 3: 3D Design Mode, Basic features

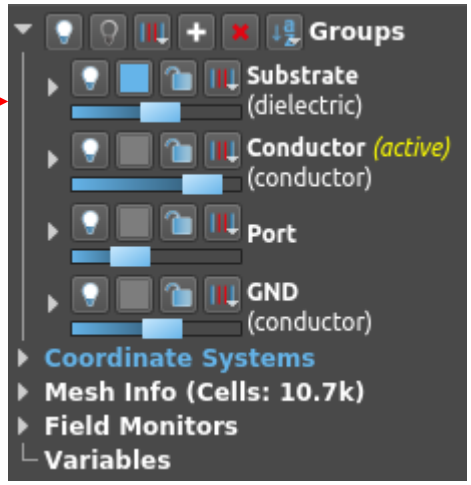
- Press the “3D Design” Tab at the top
- Press and drag middle mouse button to pan
- Turn wheel forward / backward to zoom
- Press and drag right mouse button to rotate
- Open group list on the left and move the slider near “Substrate” to adjust transparency
- Press the “2D Design” Tab at the top



Comments: The 3D Results mode usage is similar. It displays the structure in a rendered model and is used for verifying the geometry as well as displaying field distributions, animations or radiation patterns.

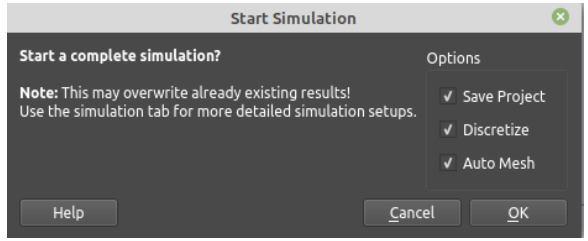
Step 3: Display Mode, Basic features

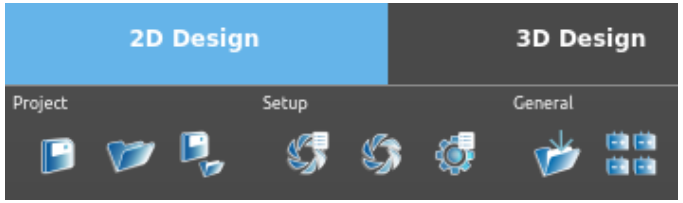
- Open group list on the left and move the slider near “Substrate” to adjust transparency
- Press the “2D Design” Tab at the top



Step 4: Simulation Flow

- Click on “Simulation Control” in the Design Toolbar at the top 
- Press Simulation and OK
- Wait until the simulation is finished and click on the “2D Results tab”

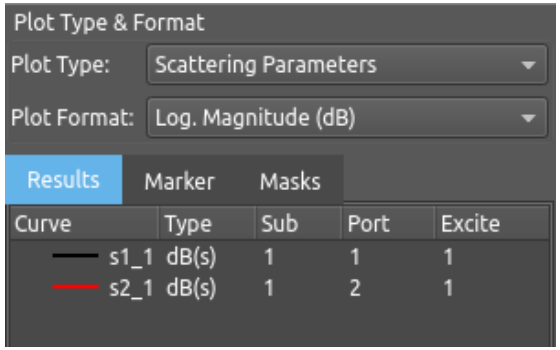


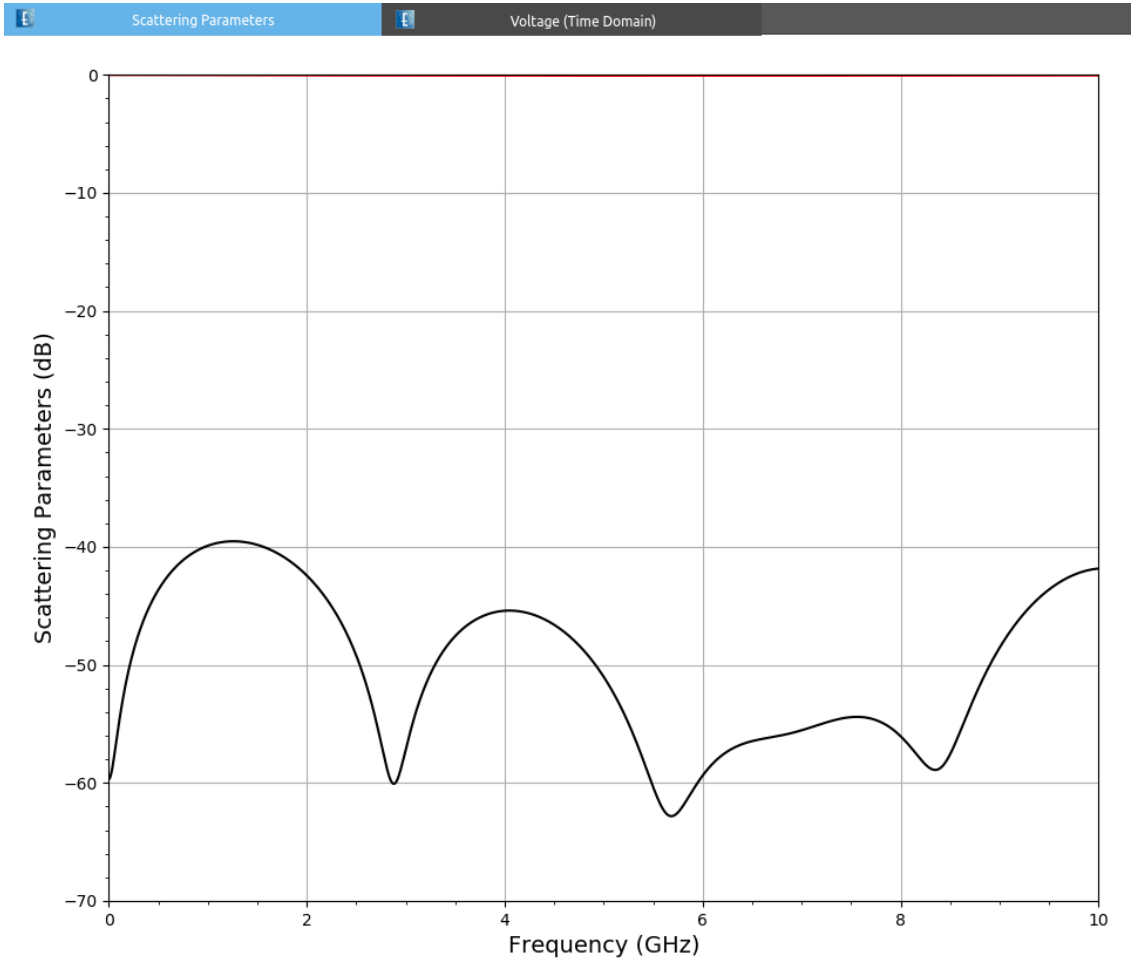


Comments: With Start Simulation, the structure will be meshed and prepared for simulation (creation of a file .acad). Then the simulation engine starts and generates a processor-optimized code for execution. During execution the time domain energy is displayed until steady state is reached. After this the post processing is started (DFT) and S-Parameters are available in 2D Results tab when finished.

Step 4: Simulation Flow

- Select “Scattering Parameters” for the “Plot Type”
- Select “Log. Magnitude (dB)” for “Plot Format”

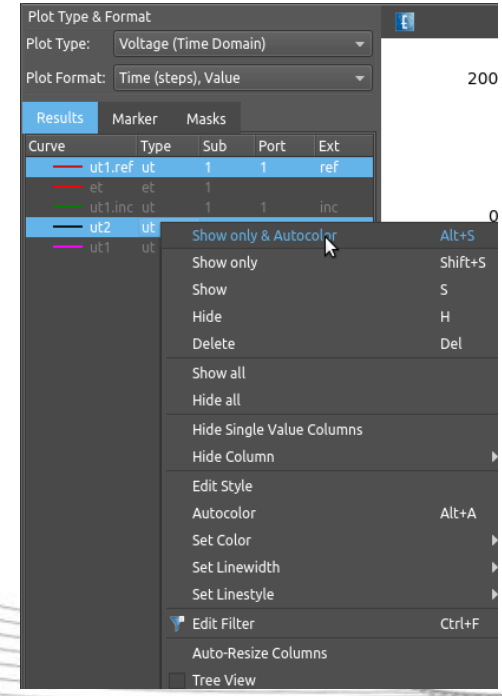


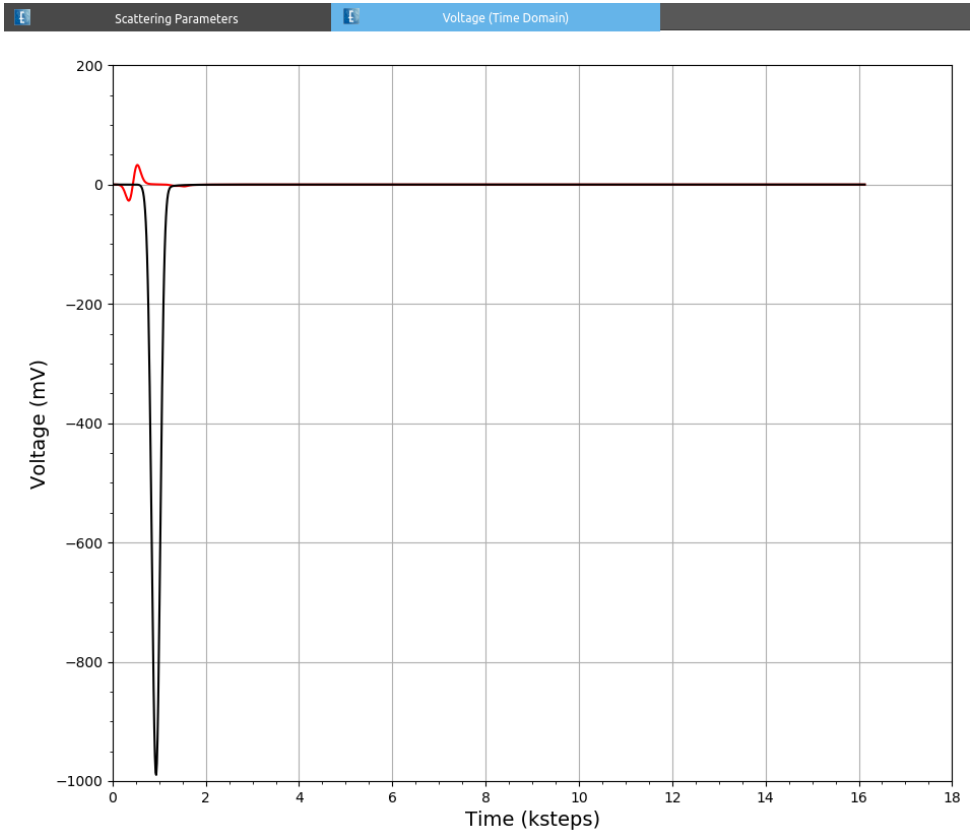


Comments: The simulation of this simple line shows the pulse at the start (black) and at the end (red) of the line with a certain delay.

Step 4: Simulation Flow

- Select “Scattering Parameters” for the “Plot Type”
- Select “Log. Magnitude (dB)” for “Plot Format”
- Right click on “utf1.ref” type “ut” and “ut2” type “ut” and select “Show only & Autocolor”





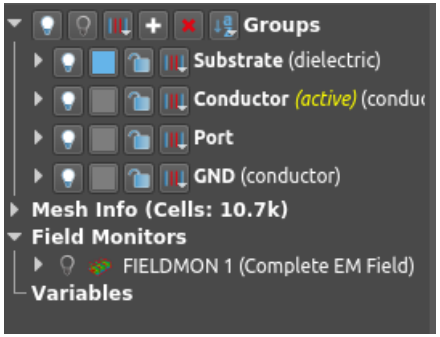
Comments: The transmission (red) and the reflection (black) is shown after the simulation and the post processing has finished.

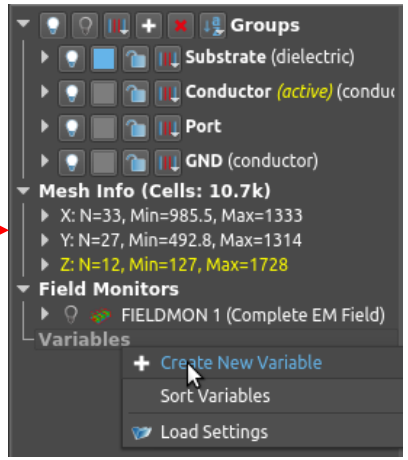
Step 5: Parameter definition

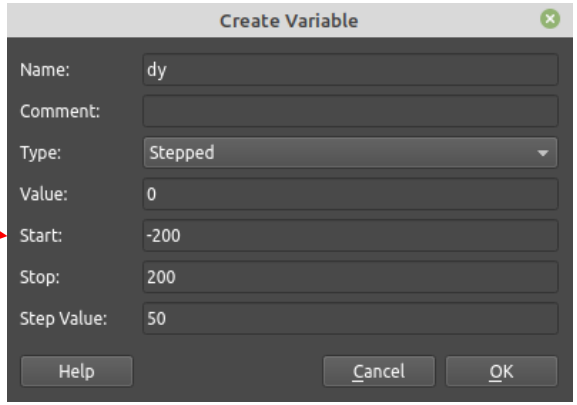
- Return to the 2D Design Tab
- Open Field Monitor, switch off

- Right click on Variables
- Create new variable

- Enter Name = dy
- Value = 0
- Min = -200
- Max = 200
- Step = 50
- Ok

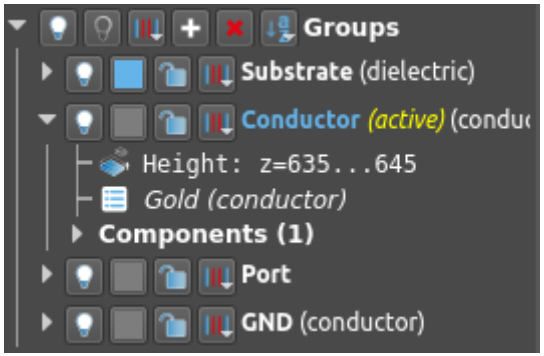


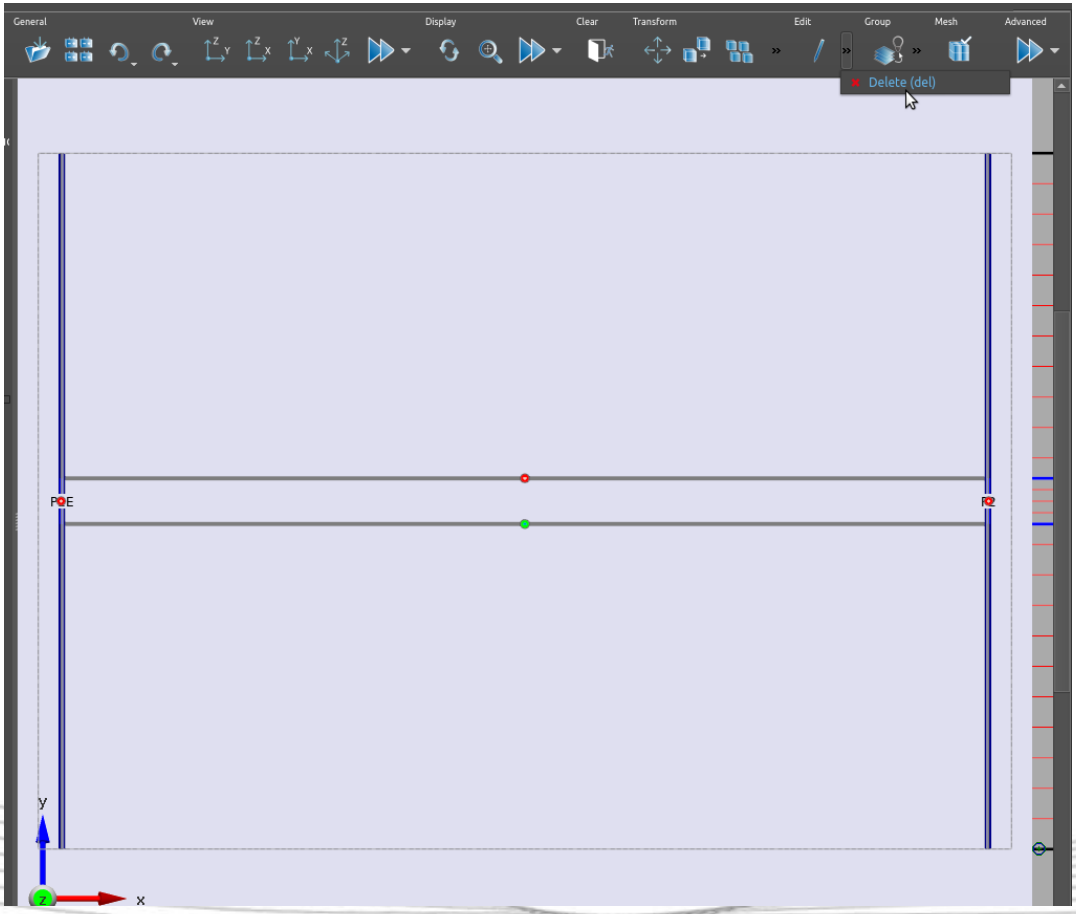




Step 6: Line definition

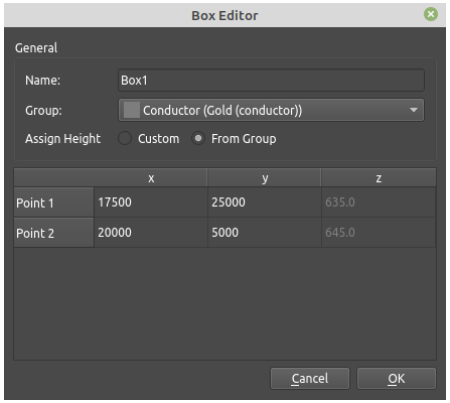
- Open group list, verify Conductor is current group (blue text)
- Select the center line
- Delete

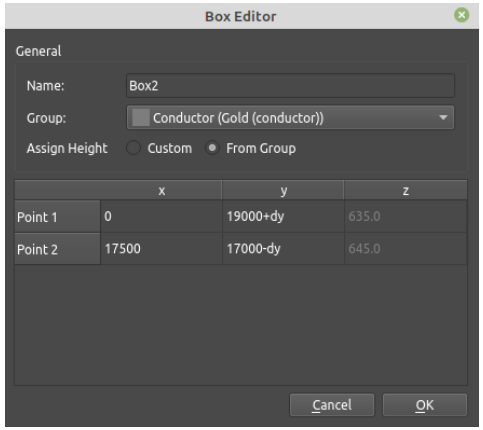




Step 6: Line definition

- Create 3 Boxes (x,y), keep z:

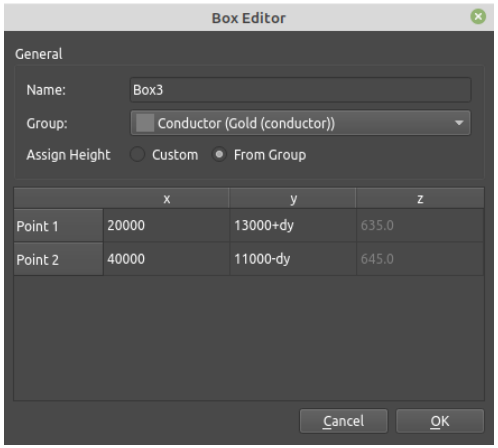




P1=(17500,25000) P2=(20000,5000)

P1=(0,19000+dy) P2=(17500,17000-dy)

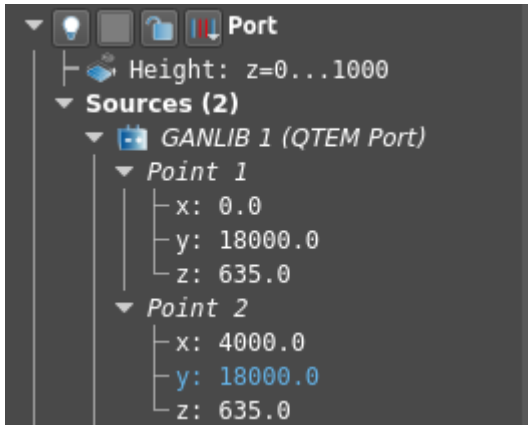
P1=(20000,13000+dy) P2=(40000,11000-dy)

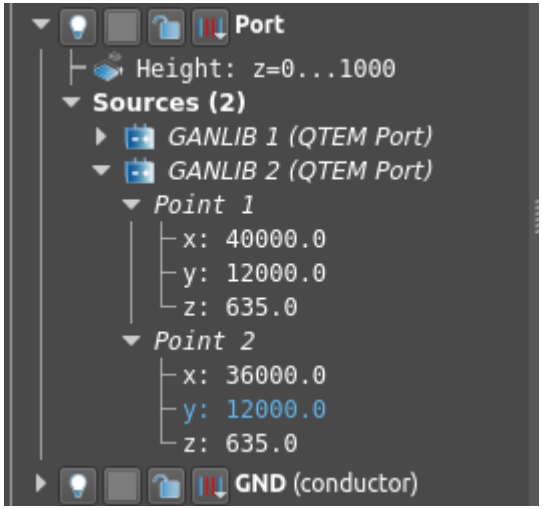


Step 6: Line definition

Step 7: Port Adjustments

- Open Sources - Group Conductor
- Open P1E
- Set y=18000 for P1 and P2
- Open Port P2
- Set y=12000 for P1 and P2
- Close



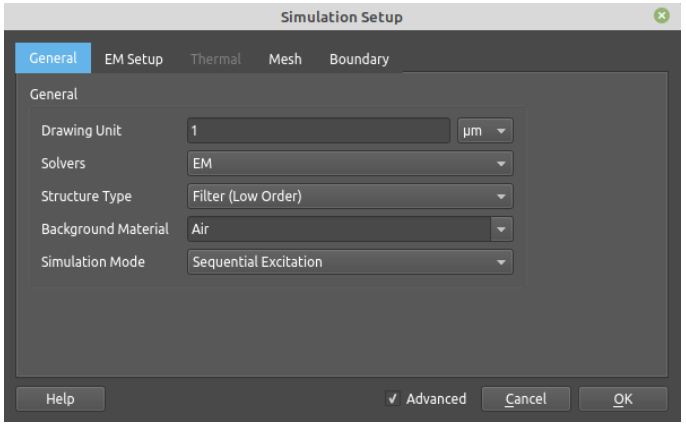


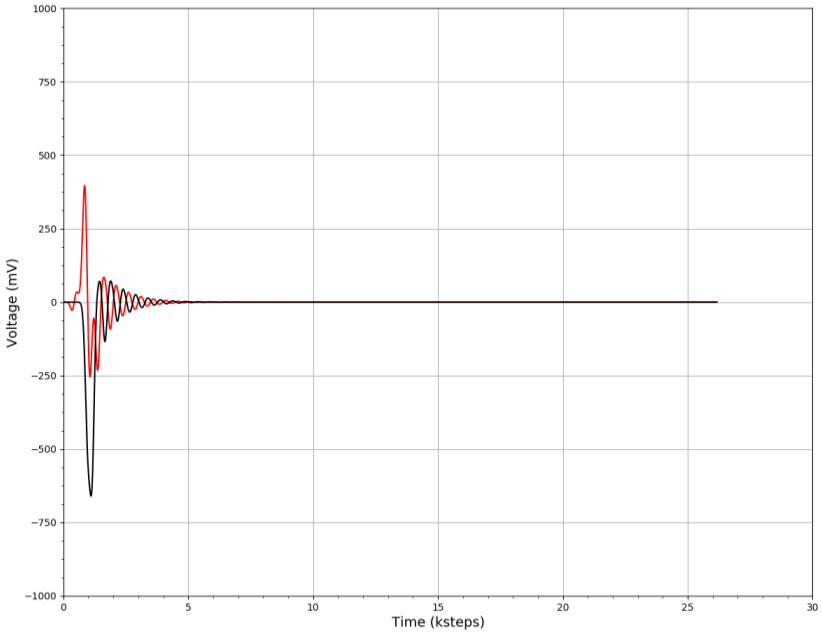
Port Point P1 defines the start of the electrode

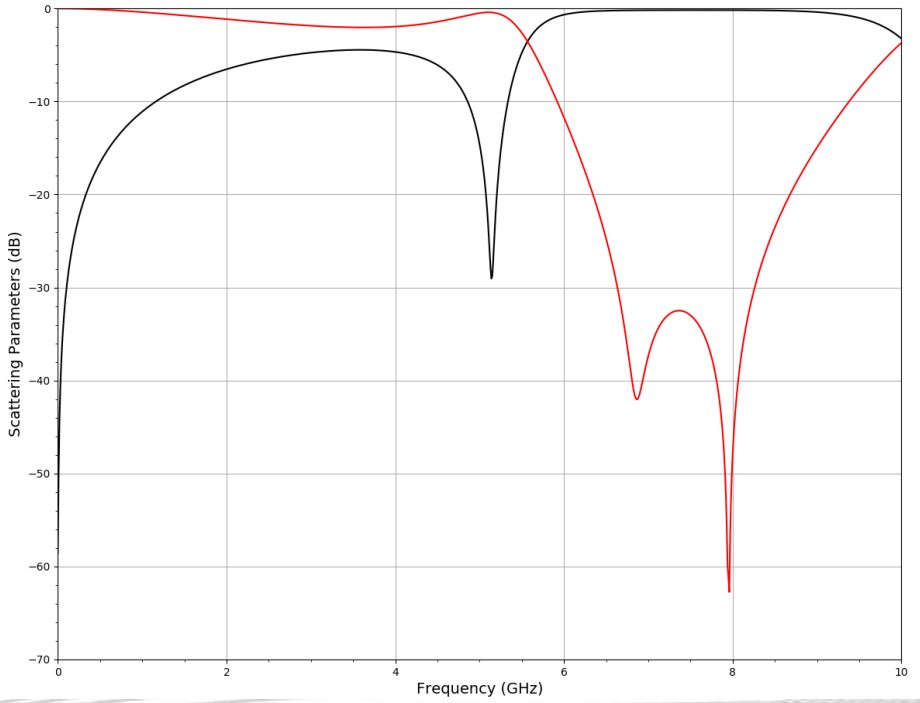
Port Point P2 defines the direction of the wave

Step 8: Start Simulation

- Press the “Simulation Setup” button
- In General → Structure Type select “Filter (Low Order)”





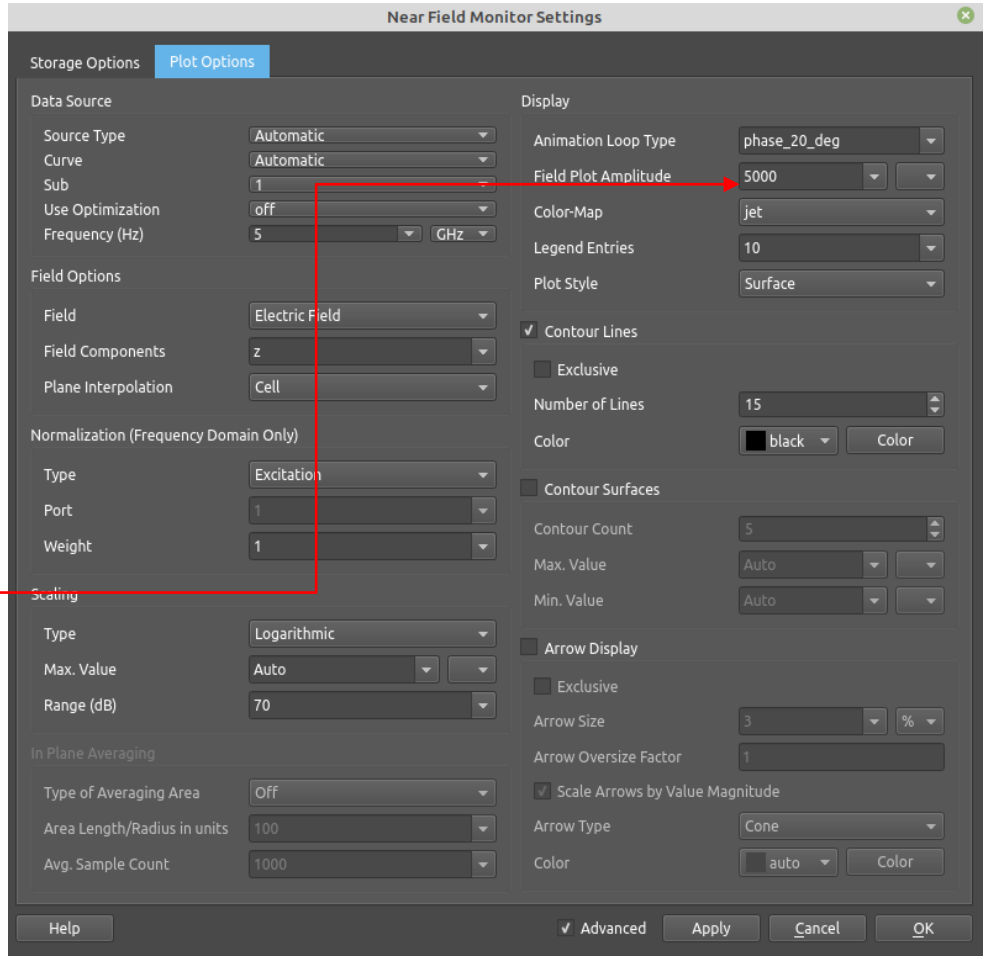


- Close Window
- Click on “Simulation Control” in the Design Toolbar at the top → Simulation

Comments: The Structrure Type parameter defines the maximum number of time steps and energy decay level.

Step 9: Animation

- Switch back to the 2D Design mode
- Switch on the FIELDMON 1 (Light bulb) in the Field Monitor list.
- Right click on FIELDMON 1 and select “Edit”
- In “Plot Options” set “5000” for “Field Plot Amplitude”,OK



Step 10: Animation

- Right click on “Plane: yz” and select “Remove Plane”
- Double click on remaining field plane to set to xy, Height: 635
- Switch to the 3D Results tab