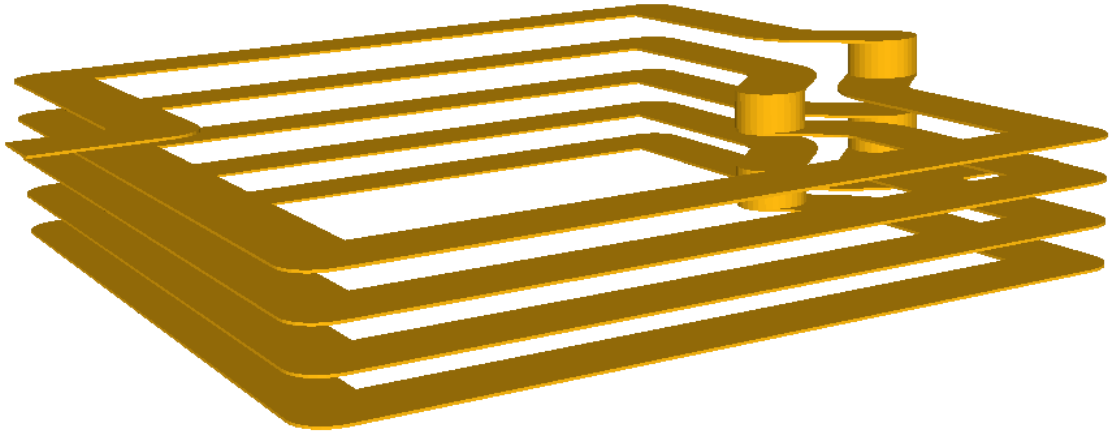
EMPIRE XPU Tutorial

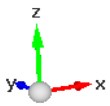
Multi-layer Inductor

Overview: Topics

- 3D geometry import (STL)
- Lumped port
- Loss calculation
- Equations



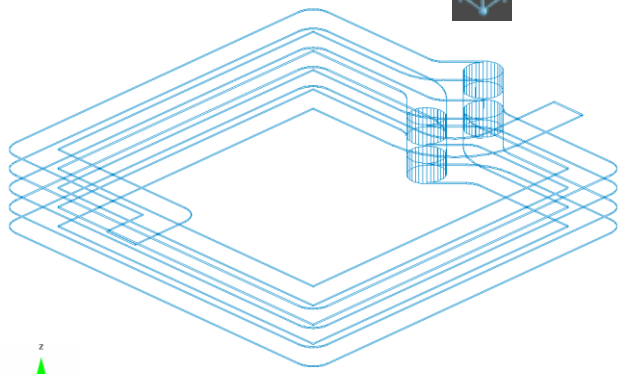
Comments: Here, we import a 3D STL geometry file ml-spiral.stl. This file is available in the subfolder data.

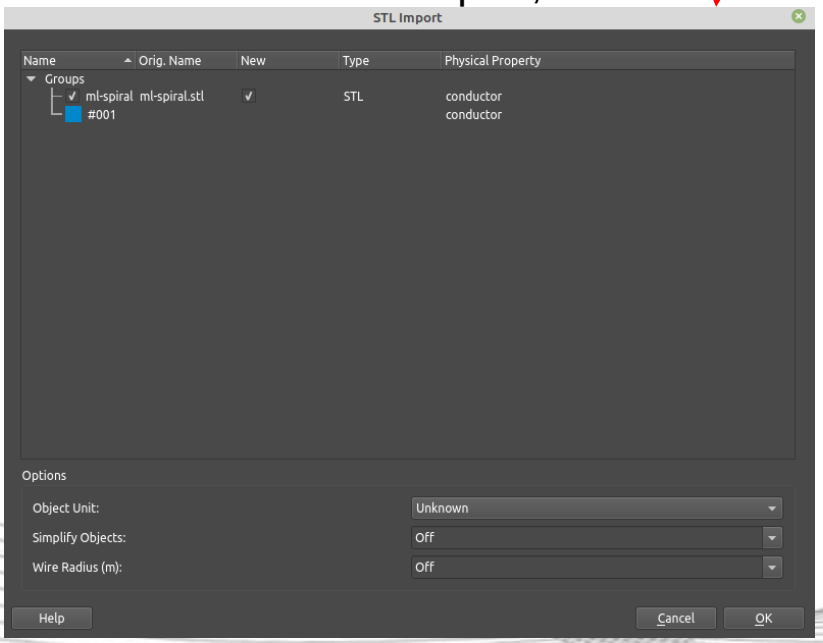


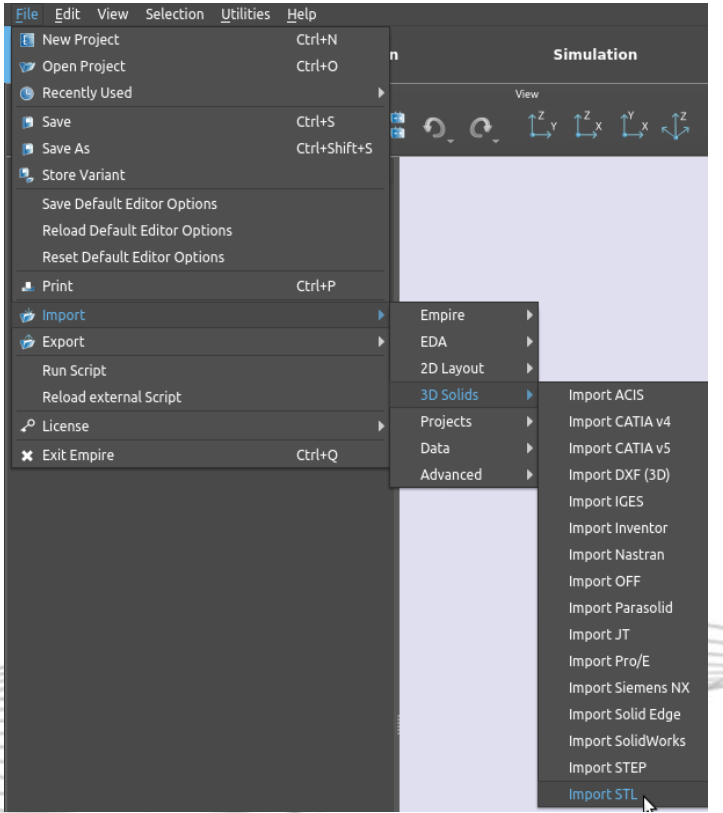
Step 1: Import Data

- Start Empire XPU, Press “New Project”, save in new folder , e.g. tutorial4
- Switch to “2D Design” Tab
- Import Structure
  - File → Import → “3D Solids” → “Import STL”
  - Select ml-spiral.stl
  - Press Import, OK

Draft View Iso Z

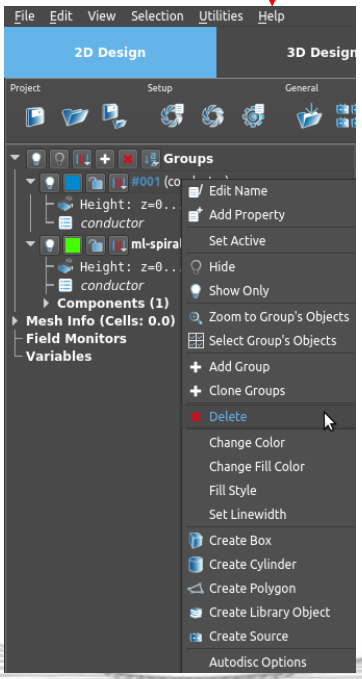


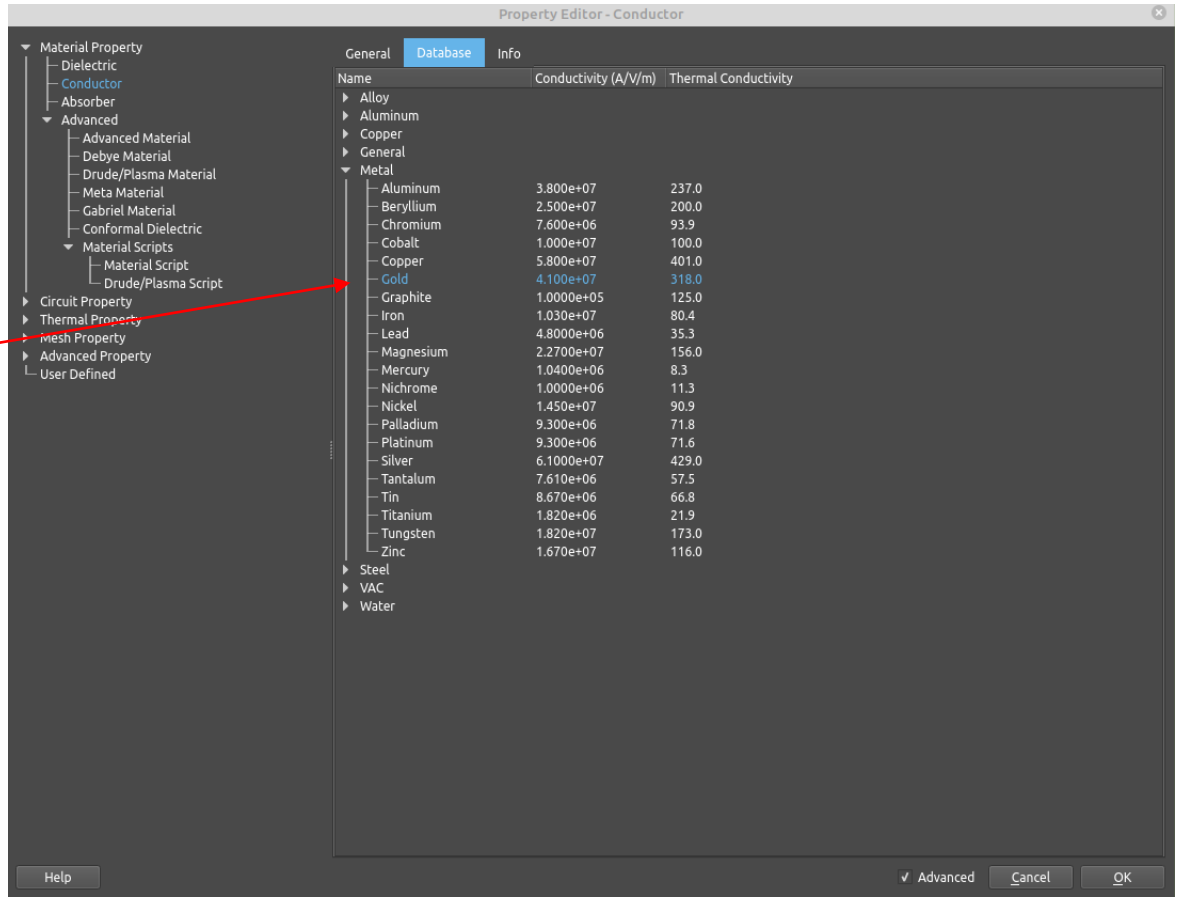




Step 1: Import Data

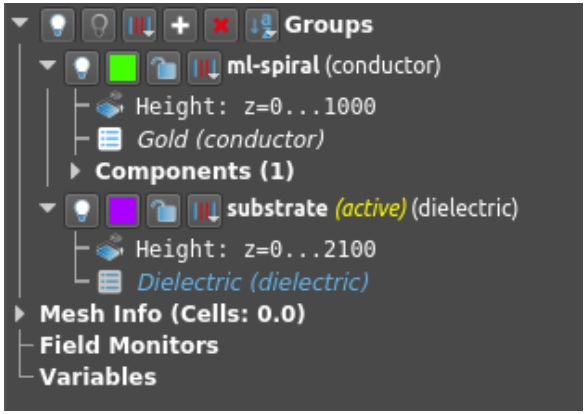
- Open “Groups” List
- Delete group #001
- Open group “ml-spiral”
- Press “Change Property”
- Select Database → Metal
- Set property to Gold
- Exit with OK

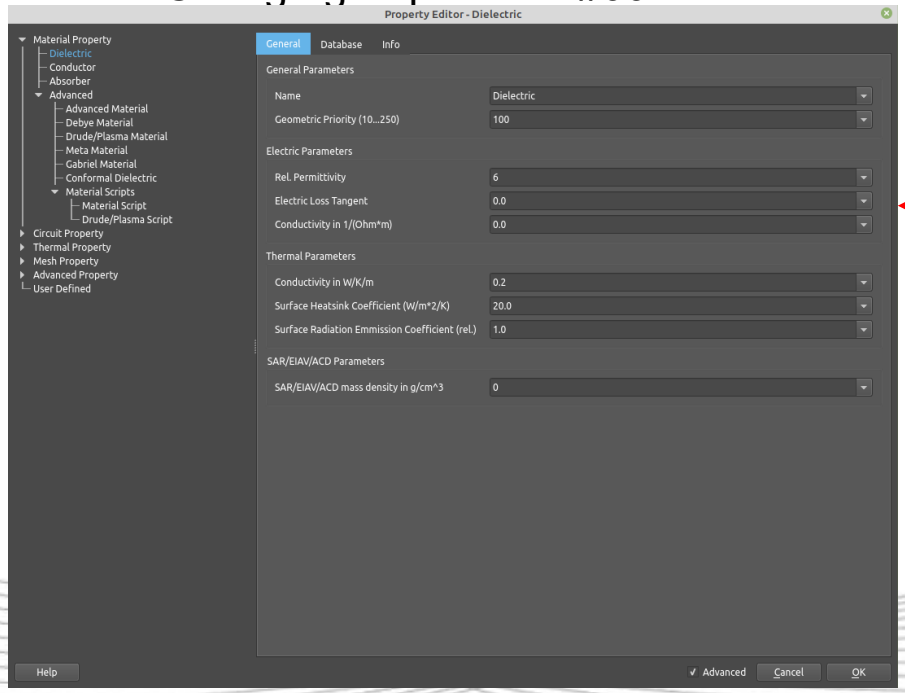




Step 2: Substrate Definition

- Switch to the “Top View” 
- Select “Groups” → “Create Group”
- Adjust group Definition:
  - group height: Start: 0 and End: 2100
  - Change Property from conductor to Dielectric: “Rel. Permittivity”= 6.0, OK
  - Change group name “#001” to “substrate”

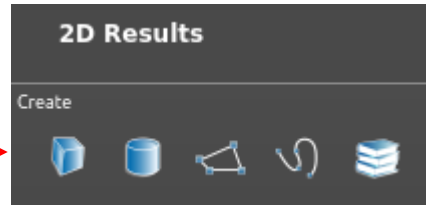


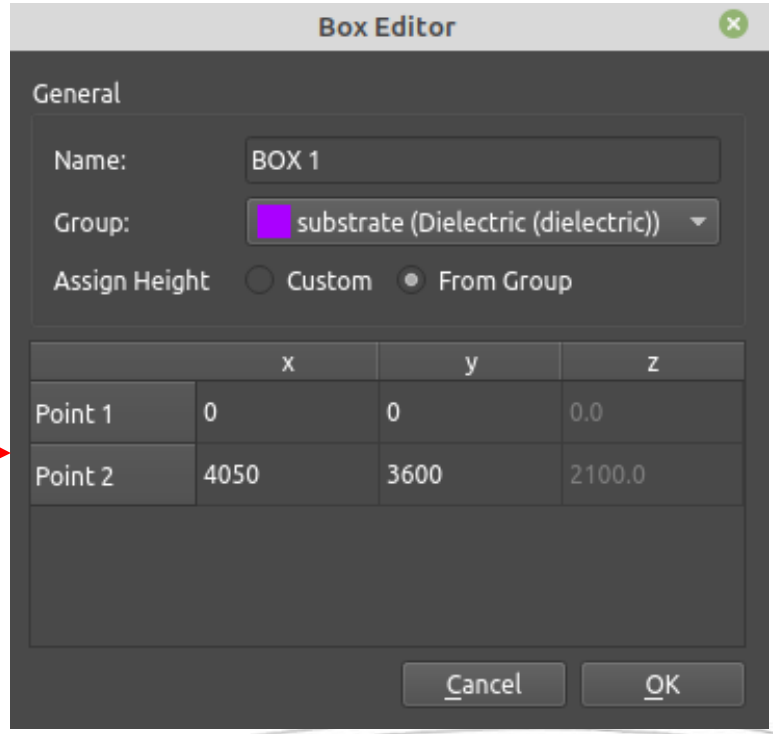


Comments: Default priority number of a dielectric is 100. So the substrate can be simply defined as a brick (box).

Step 2: Substrate Definition

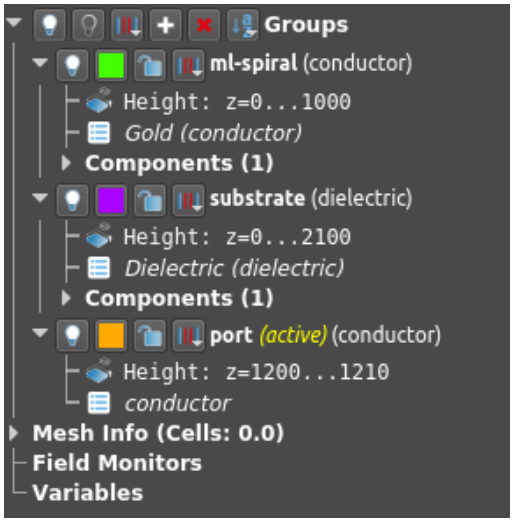
- Draw substrate:
  - Select Icon Create Box
  - Point 0: x=0, y=0
  - Point 1: x= 4050, y= 3600
  - Press OK
- Close group

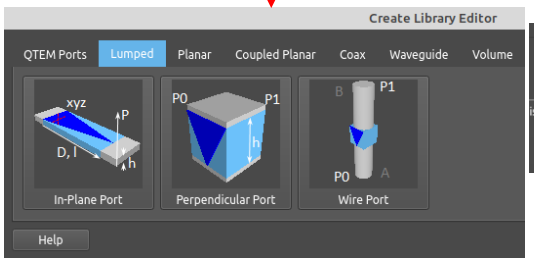


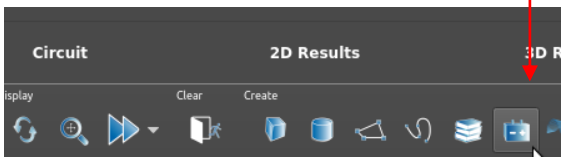


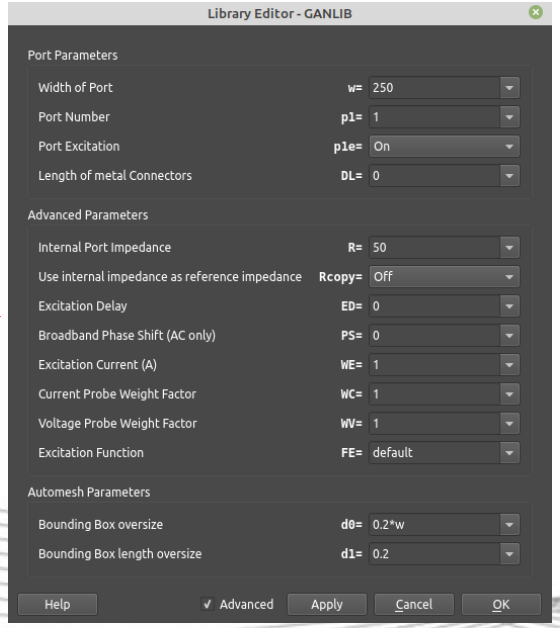
Step 3: Port Definition

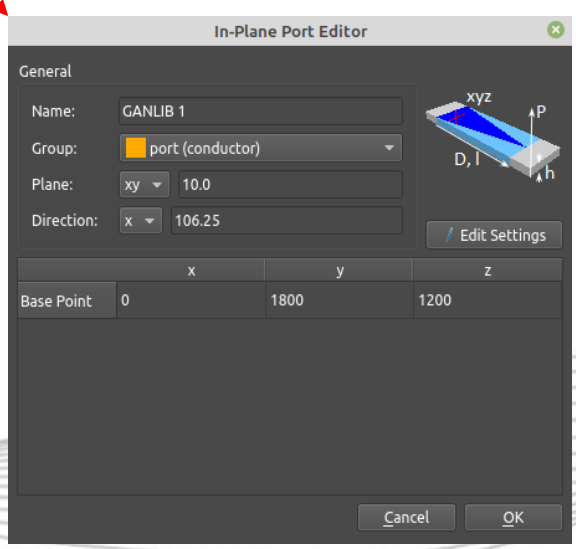
- Create a new group “port”
- Set height to 1200 to 1210
- Select “Create Source” → Lumped → “In-Plane Port”
- Enter Start Coordinates (0,1800,1200), Direction: x, length =106.25
- Press Parameters
  - Adjust “Width of Port” to w=250
  - Press “OK”



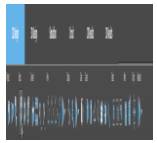


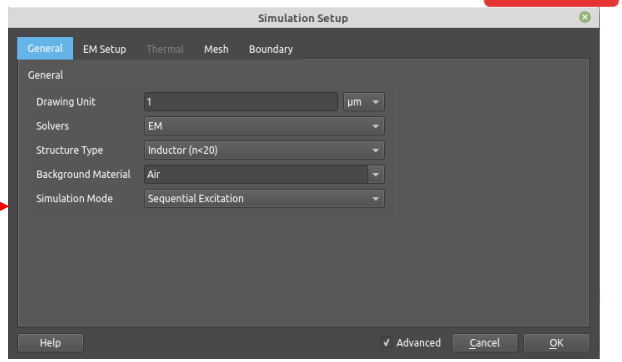


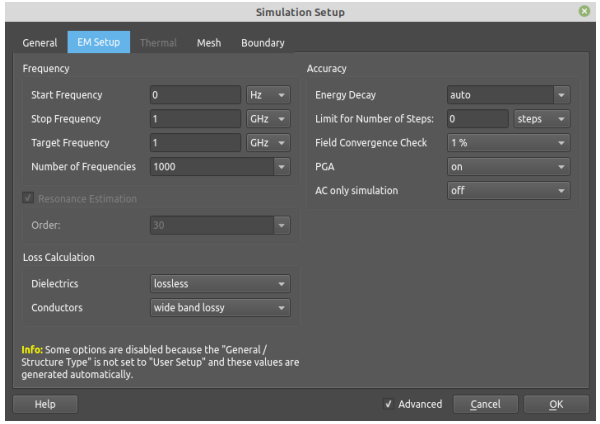


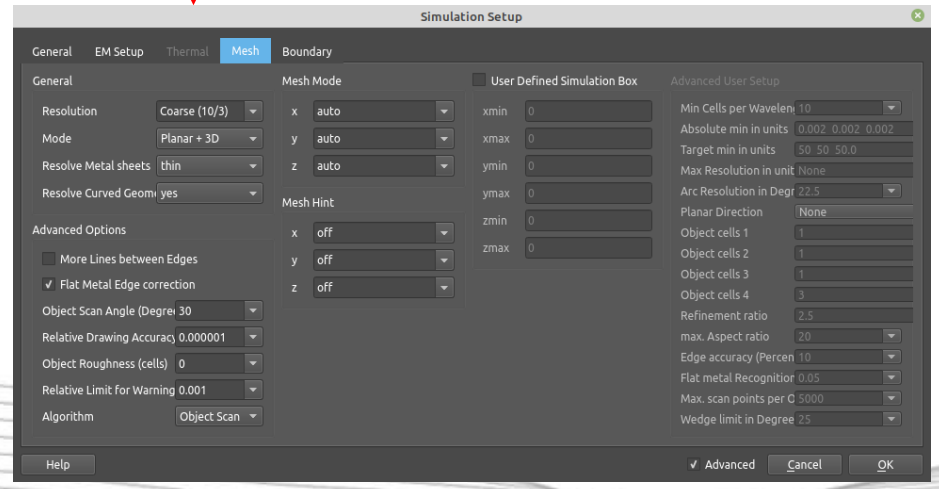


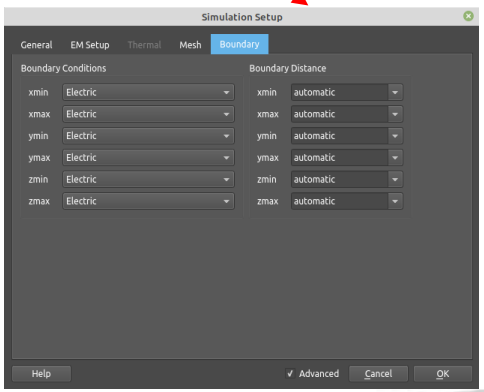
Step 4: Simulation Setup

- Select Icon Simulation Setup 
- Set Structure Type: “Inductor (n<20)”
- Set End frequency and Target Frequency to “1 GHz”
- Set Loss Calculation→ Conductors: “wide band lossy”
- Set Mesh Resolution: “Coarse (10/3)”
- Set Boundary Conditions: All “Electric”
- Press “Simulation” 





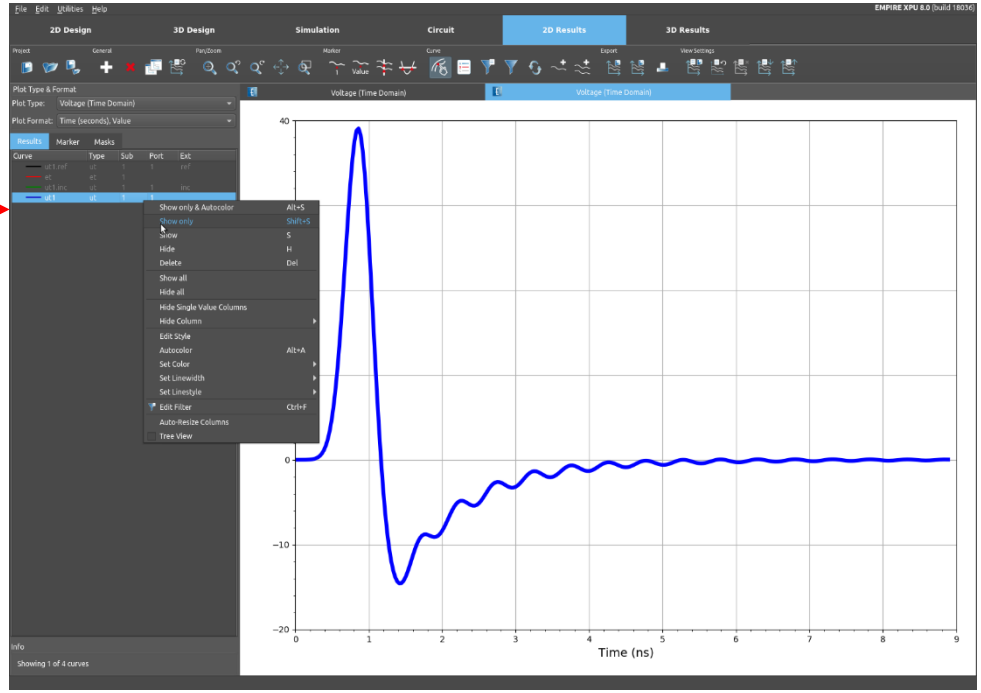


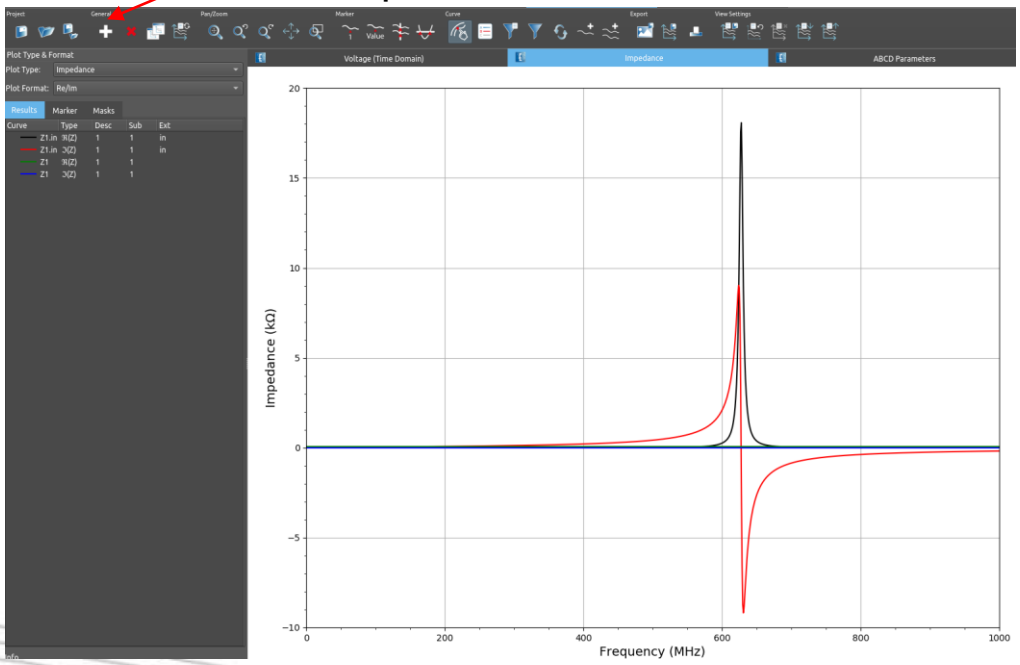


Comments: The “Inductor (n<20)” sets the number of time steps to a large value because the pulse response is quite long for inductances.

Step 5: Results 1

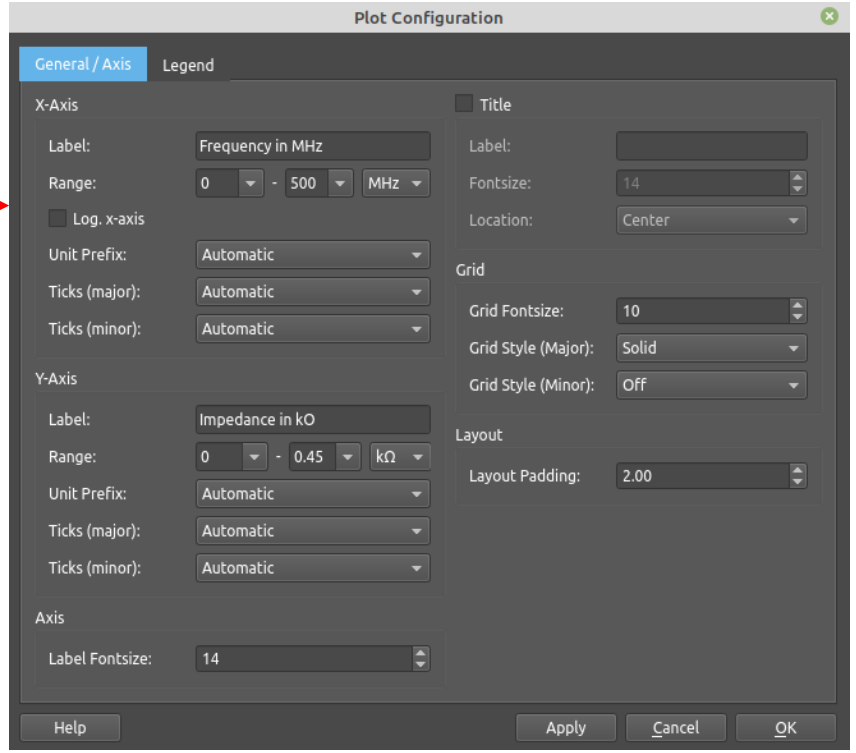
- Go to the 2D Results tab
- Voltage Tab
  - Right-click “ut 1 1” in legend
  - Select “show only”
  - Add a new result plot, select Impedance

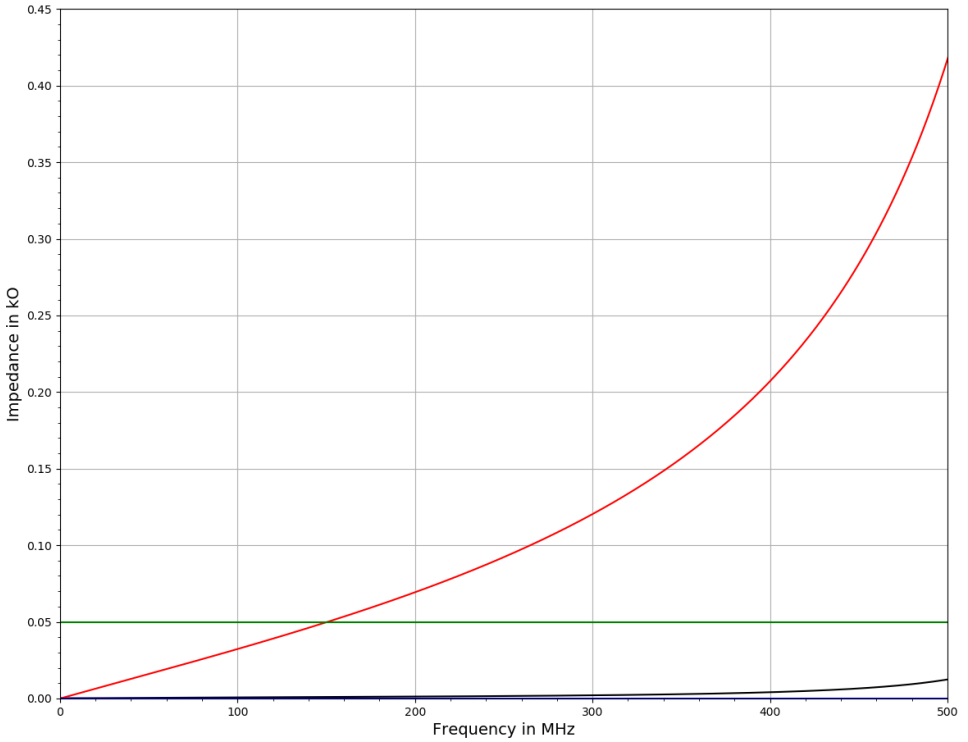




Step 5: Results 1

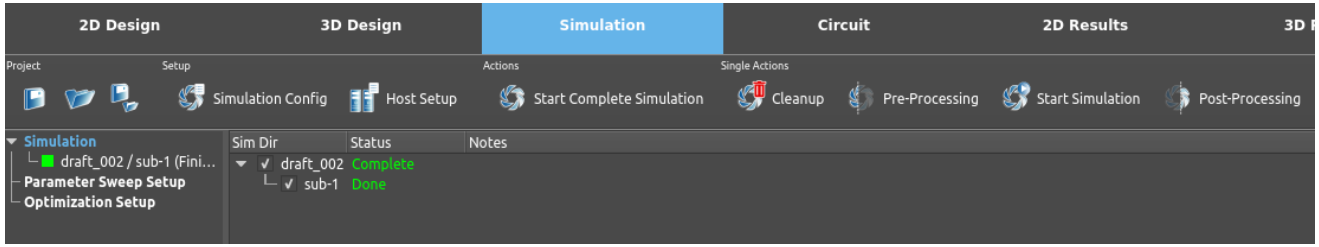
- Impedance Tab
  - Right-click inside the plot
    choose → Configure Plot, enter range
  - From 0 to 500 MHz for the x-axis
  - From 0 to 0.45 kOhm for the y-axis, press OK

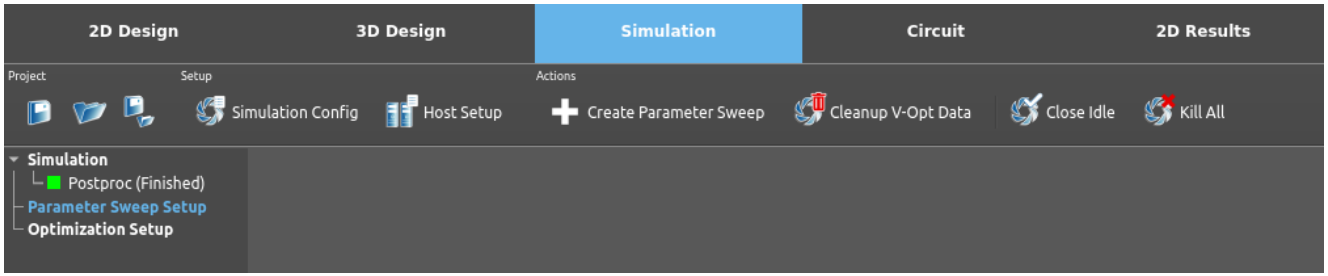




Step 6: Postprocessing

- Switch to Simulation Tab
- Click “Simulation Config”
- In “EM Setup” set “Stop Frequency” to 0.5 GHz
- Switch to “Equations” tab
- Right click on “sub-1” Add Equation - “Symbolic Equation”
- Select Template: Inductivity – ZL= …, OK
- Right click on “sub-1” Add Equation - “Symbolic Equation”
- Select Template: Q-Factor – ZQ= …, OK





Comments: The expressions for the inductance and quality factor is often needed and therefore available in the equation templates.

Step 7: Results 2

- Click “Post-Processing”
- Visualization 2D:
  - Q Factor:
    - Switch to „2D Results“
    - Plot Type to „Custom Curve“
    - Show only “ZQ R(ZQ)”
      (right click on name, select Show Only)
    - Select Autoscale 
  - Inductance:
    - Show only “Re(ZL)”
    - Select Autoscale