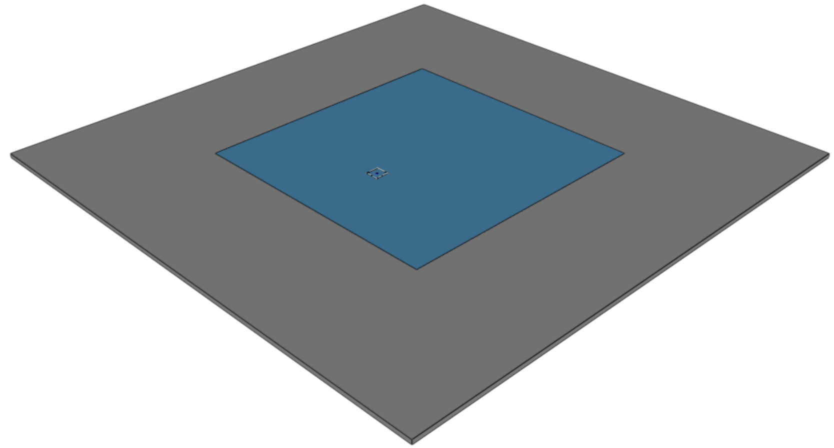
EMPIRE XPU Tutorial

Patch Antenna started from scratch

Overview

- Group creation
- Property setting
- Object creation
- Port definition
- Simulation parameters
- Field recording
- Simulation
- Results



- Target frequency: 2.45 GHz
- Substrate: Rogers, 635um, epsr=2.2
- Patch size ~ 40mm x 40mm (lambda/2)
- Substrate size ~ 80mm x 80mm
- Infinite ground plane

Start

Help:

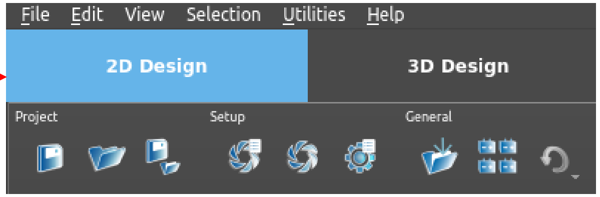
- Methodology and an overview of basic features are explained in “Getting Started.pdf”
- Complete manual is available in EMPIRE-Manual-800.pdf
- Videos are available at www.empire.de - Downloads - Videos
- Send questions to empire.support@imst.de (include input file .gym if applicable)

Start:

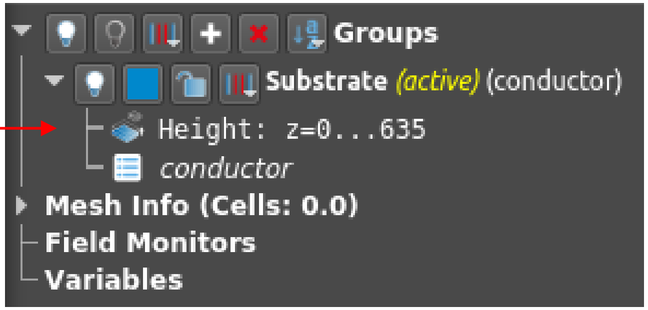
- Start Empire XPU
- Select “New Project”, press “OK"
- Click “File" in the top menu and select → “Save As”
- Create a new folder for each new project as multiple files will be created in the selected directory
- Select the directory, e.g. C:\tutorial0\scratch and click “Save”

Step 1: Group creation

- Switch to "2D Design Tab"



- Open the "Groups" list on the left (select the ▸ sign) if it is not already open
- Open the default group ▸
- Right click on group name #001 *(active)* (conductor) and edit the name to "Substrate" (optional)
- Double click Height: z=0...1000 to change it to z=0...635

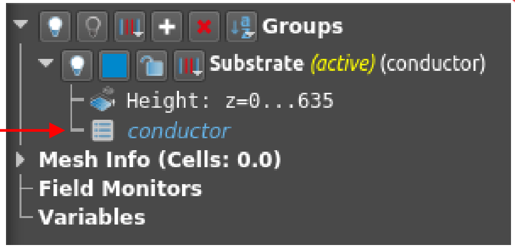


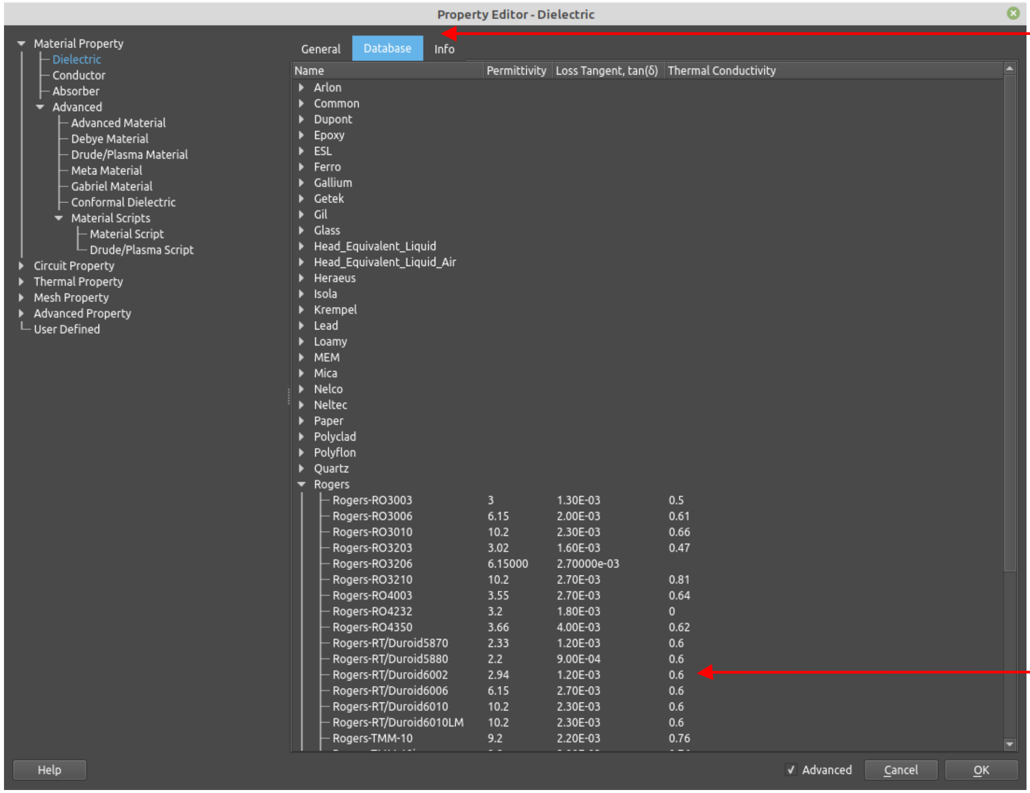
In EMPIRE, the structure is organized in groups. It is recommended to separate objects with different properties on different groups.
Groups are used
- *to group objects with common properties*
- *to define the height of objects, like boxes and polygons*
- *to set the insertion point of library objects, like ports*
- *to color, lock or hide objects*
- *to define their properties*

Comments: In this example (cylindrical) objects are created in the xy-plane. The perpendicular coordinates are taken from the group's height. The values entered here represent the thickness of a substrate. The default unit is micron (can be changed in the Simulation Setup).

Step 2: Property setting

- Double click “conductor” to change its properties
- Select “Dielectric”, select the “Database" tab
- Expand “Rogers”, select “Rogers-RT/Duroid 5880”
- Press OK to leave the property editor



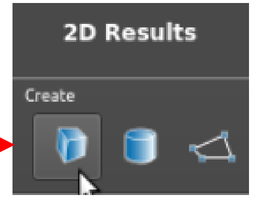


Properties can be divided into:
- *physical, basic, like conductors or dielectric materials*
- *functional, like circuit elements, e.g. resistor*
- *functional, like mesh hints*
- *advanced, for special applications*

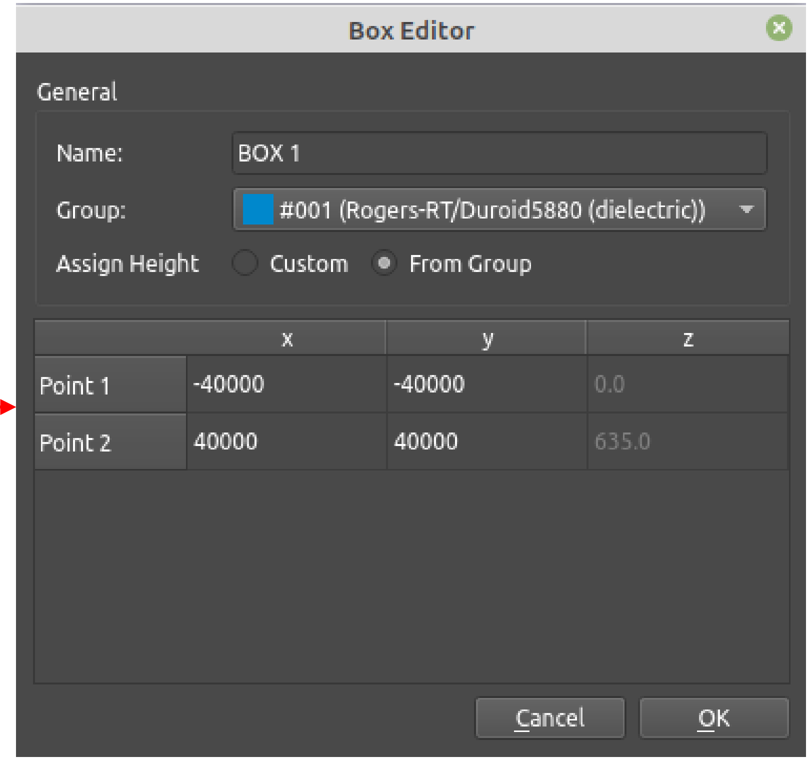
Comments: The default property is conductor (PEC). Here, we want to define the substrate and therefore the property is changed. Groups may have multiple properties, if they are not contradictory.

Step 3: Object Creation

- Select Icon “Create Box”



- Enter xy coordinates of Point 1: (-40000, -40000)
- Enter xy coordinates of Point 2: (40000, 40000)
- Press OK
- Press “Zoom Extents”  or press the Z-key

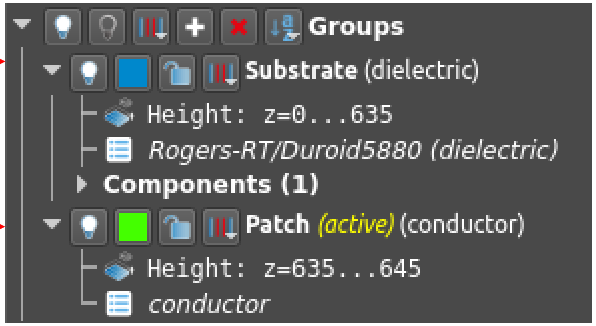


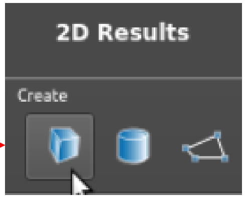
Cylindrical objects can be created in several ways
- *Pressing button “Create Box/Cylinder/Poly” and entering coordinates*
- *Drawing an arrow and pressing “Create Box/Cylinder” afterwards*
- *Entering a set of points and pressing “Create Poly” afterwards*

Comments: The object is created immediately after pressing on Create – Box with the default values. The drawing is updated as soon as values are modified.

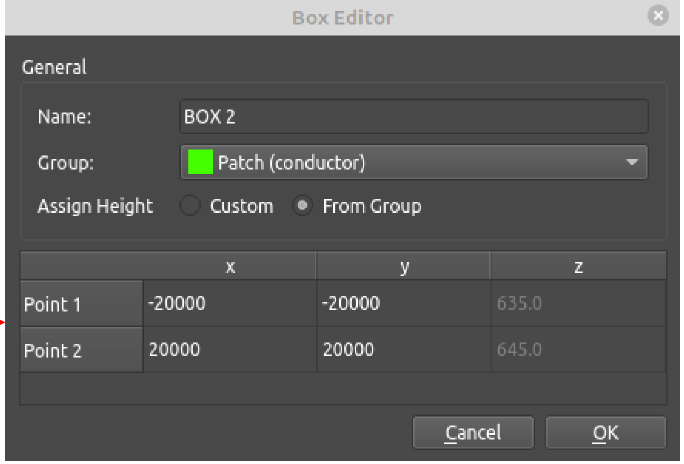
Step 4: More objects

- Create a new group and open it
- Set group height to z=635...645
- Change group name to "Patch”
- Keep conductor as property





- Create patch:
  - Select “Create Box”
  - Enter xy coordinates of: Point 1 (-20000,-20000), Point 2 (20000,20000)

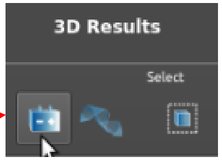


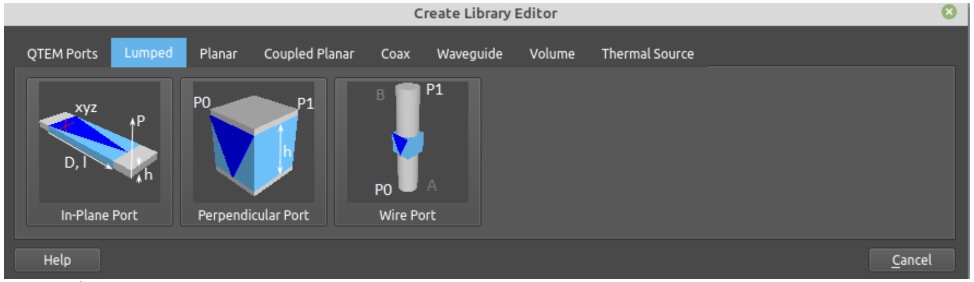
Object modifications can be created in several ways:

- *Select object, press button move, assign, stretch, rotate, mirror, … and enter operator (arrow, point, number,…)*
- *Select object, activate handle, move to handle*
  - *Drag middle mouse to copy object*
  - *Drag right mouse button to move object*
  - *Drag left mouse button to stretch handle*
- *Select two or more objects and apply a Boolean operation merge, intersect, subtract, drill, e.g. to cut out a hole*

Step 5: Port definition

- Open Group list, click “Create group”*
- Repeat Step 1, rename and recolor the group, set height to z=0...635, keep or delete property**
- Click on the icon “Create Source”

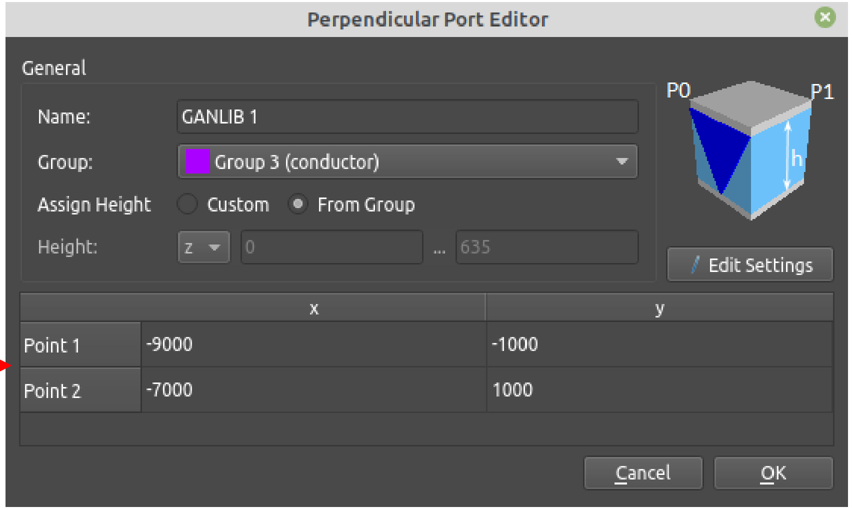




- Select the “Lumped“ tab and

then “Perpendicular Port”

- Enter xy coordinates of Point 1: (-9000, -1000)***
- Enter xy coordinates of Point 2: (-7000, 1000)
- Press OK



Comments:

** A new group is not strictly necessary, the port could also be created on the substrate group which has the same height*

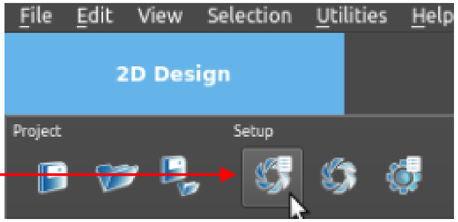
*** This port type does not make use of the property definition*

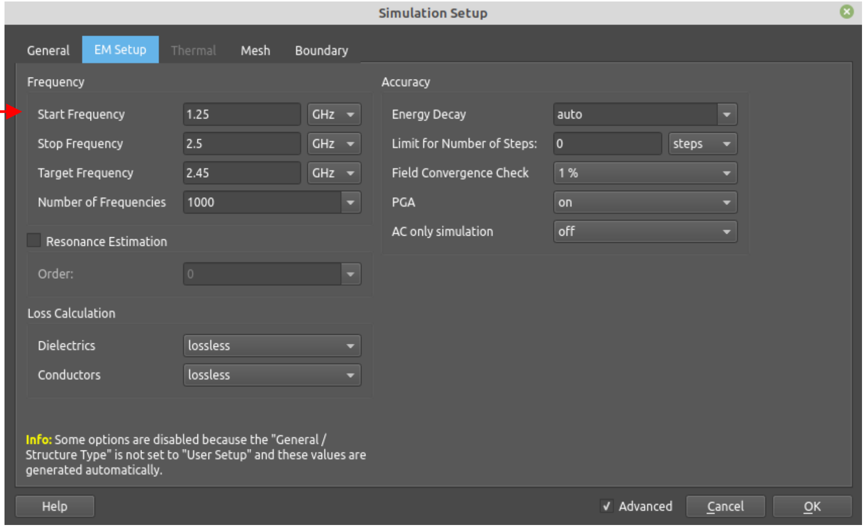
**** The port size should be chosen according to an excitation in reality, e.g. the inner diameter of a coaxial feed cable*

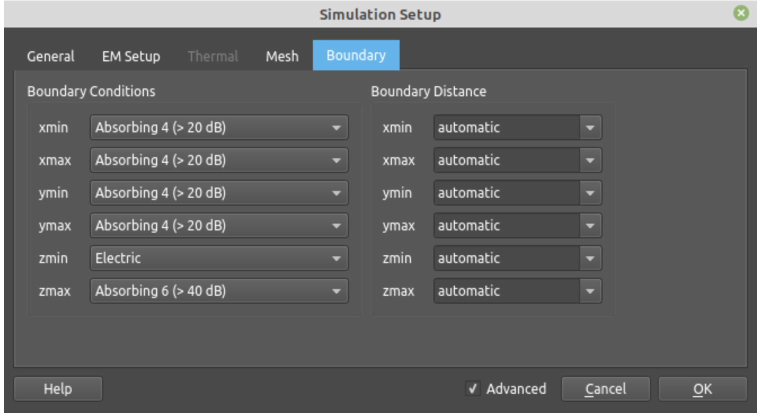
- *Perpendicular ports connect top and bottom conductors, height should be the distance between patch and ground*
- *(In-plane ports connect either between left and right or up and down conductors, height should be equal to metal thickness)*
- *Port numbers should be unique unless simultaneous excitation is desired*
- *Port impedances are 50 Ohm by default*
- *Lumped ports and concentrated ports may not be placed at the boundaries*

Step 6: Simulation setup

- Click on “Simulation Setup”
- Enter Start Frequency: 1.25 GHz
- Enter Stop Frequency: 2.50 GHz
- Enter Target Frequency: 2.45 GHz
- Boundary Conditions
  - Set zmin to Electric, keep zmax, change the others to “Absorbing 4 (>20dB)”
- Click “OK”





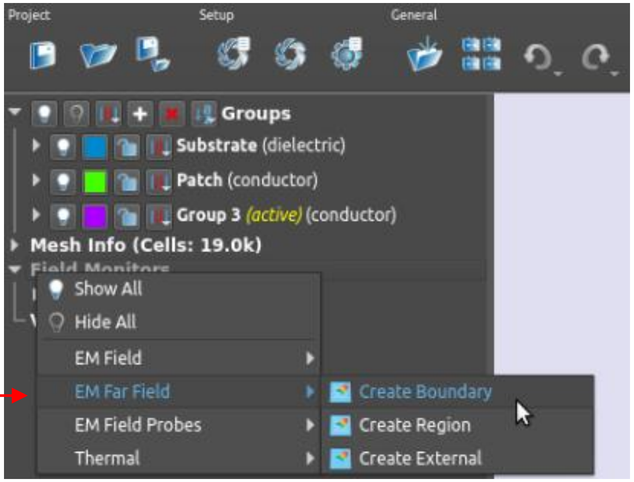


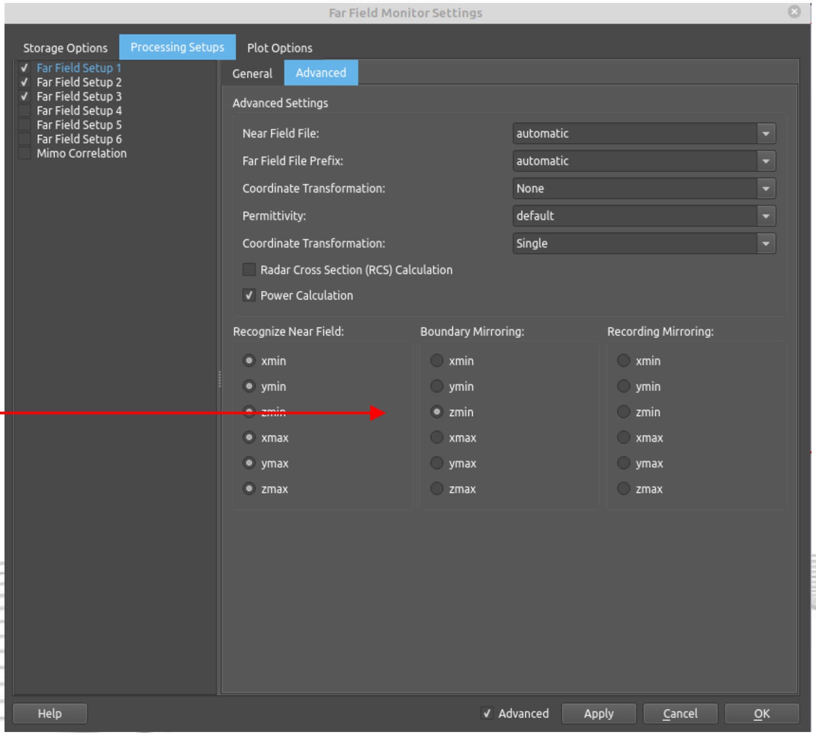
Simulation setup:

- *Geometry: 1 unit in the drawing equals 1 micron, here*
- *Structure Type: Information about the structure for automatic meshing and end criteria*
- *Frequency: Determines the range of the DFT, the pulse width used is derived by maximum cell size*
- *Mesh Resolution: Medium (15/4): Maximum cell size determined by 15 cells per wavelength at Stop Frequency, using at least 4 cells per object or gap*
- *Loss Calculation: Model used for loss calculation, default is lossless*
- *Boundary conditions:*
  - *electric defines infinite ground plane, Et=0, (magnetic Ht=0)*
  - *Absorbing N emulates open space (N should be larger in the main radiation direction)*

Step 7: Far Field recording

- Right click on “Field Monitors” on the left side and select
- “EM Far Field” and then “Create Boundary”, click OK
- Set „Display Origin” as x=0, y=0, z=635*.
- Right click on the boundary and select „Edit” and keep the default „Frequency for storage” unchanged**
- In the „Processing Setups” tab Far Field Setup 1-3, “Advanced” tab check the „Boundary Mirroring” for zmin
- Exit with OK

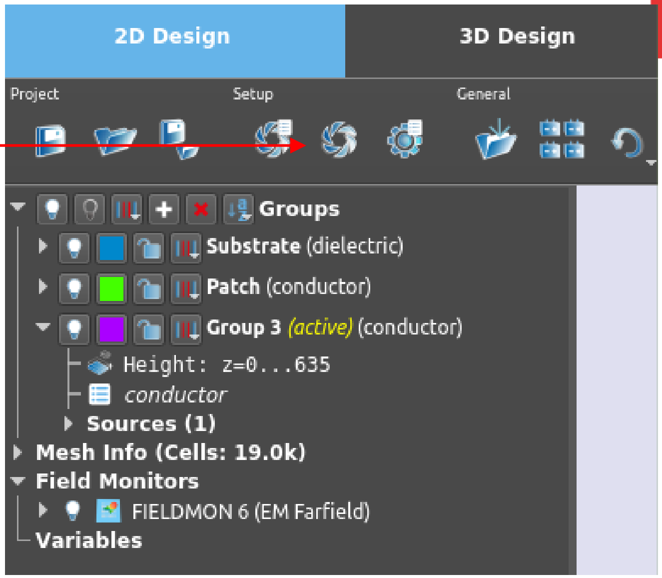




Far fields:
- *Far fields are obtained in post processing by a transformation of the near field recorded on a box surface*
- *The near-field-to-far-field box is created automatically, depending on the frequency settings*
- *Further adjustments can be set after simulation in Far Field-Setups:*
  - *Normalization (Gain, Directivity, maximum, …)*
  - *Sweep mode (2D cuts, 3D pattern,…)*
  - *Rotation*
  - *Far field components (linear, circular, …)*
  - *Mirror planes*
  - *Phase center*

** The entered point acts as 3D display center, only*
*** By default, the target frequency is automatically set*

Step 8: Simulation



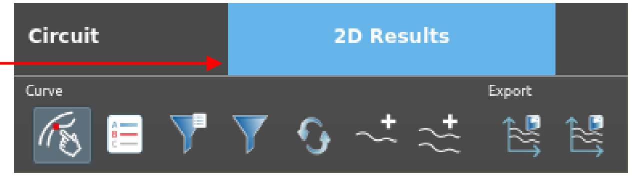
- Press the “Start Simulation” button

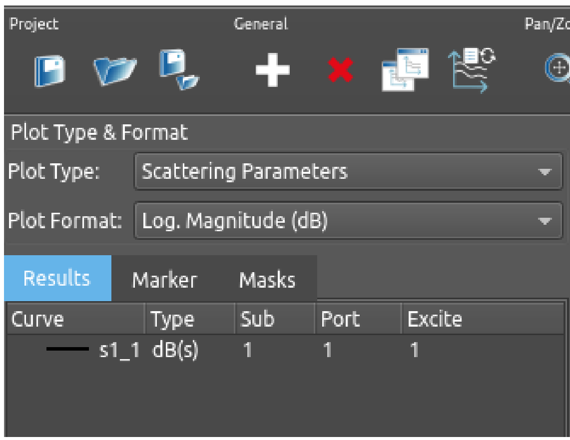
Meshing and simulation:

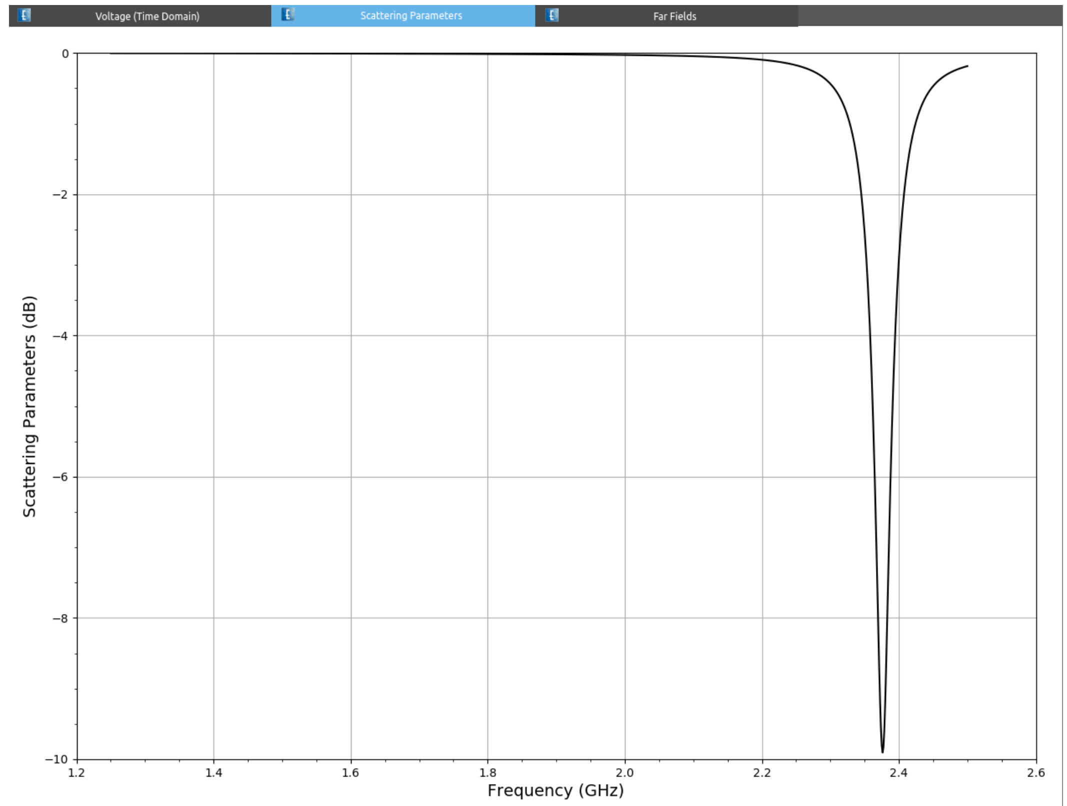
- *The created mesh lines are displayed on the bars at the right and at the bottom*
- *The automatic meshing automatically enlarges the simulation domain to account for the far field transformation*
- *The simulation domain is marked by the dashed lines which indicate open boundary condition*
- *In the front view, the simulation border at the bottom is indicated by a dark red solid line which represents electric (green: magnetic) boundary conditions*
- *With “Start Simulation” the structure is checked, meshed and prepared for simulation*
- *As soon as the energy plot comes up the simulation starts, the evolution of the time signal is shown*
- *When one of the end criteria has been reached, the post processing is triggered and the S-parameters are displayed.*
- *A warning message is written in the Log window (Simulation Tab) because an electric boundary has been detected while the far field mirror option is still disabled*

Step 9: Results

- Go to the “2D Results tab”
- Select “Scattering Parameters” for the “Plot Type”
- Select “Log. Magnitude (dB)” for “Plot Format”

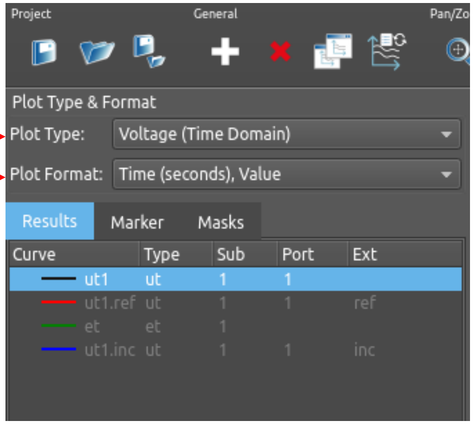


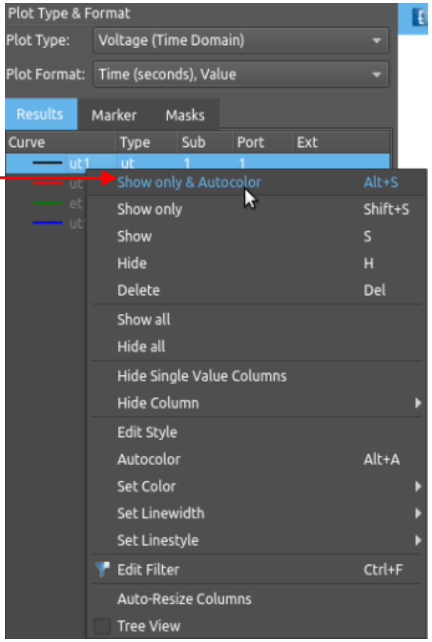


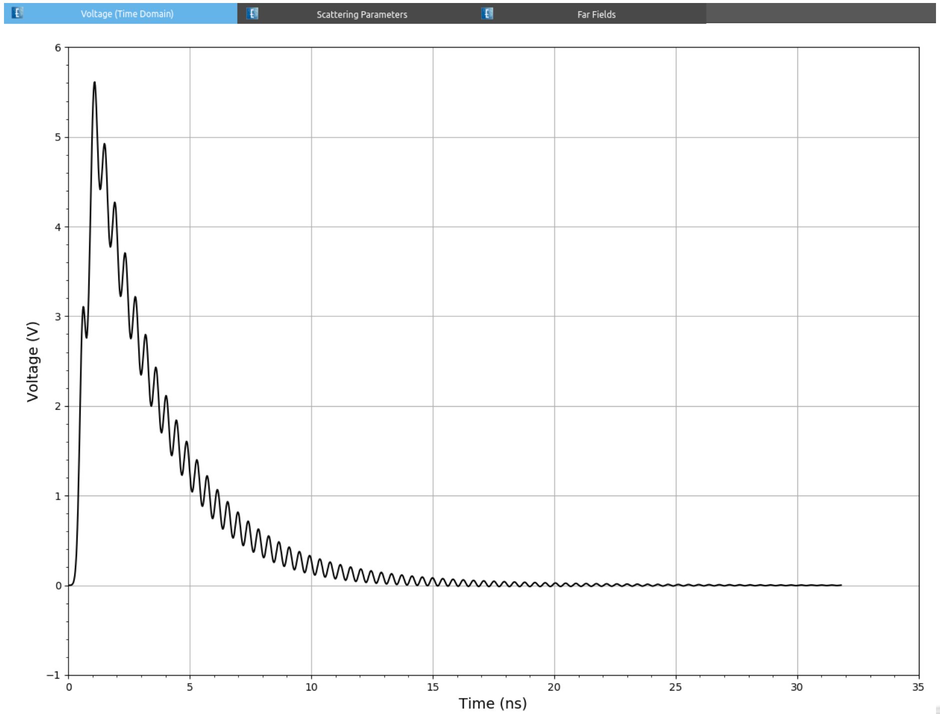


Step 9: Results

- Click on ➕ to create a new plot
- Select “Voltage (Time Domain)” for the “Plot Type”
- Select “Time (seconds), Value” for “Plot Format”
- Right click on the “ut1” curve with “ut” type and select “Show only & Autocolor”

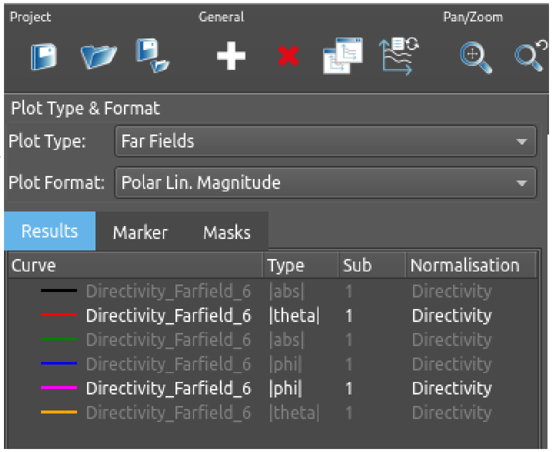


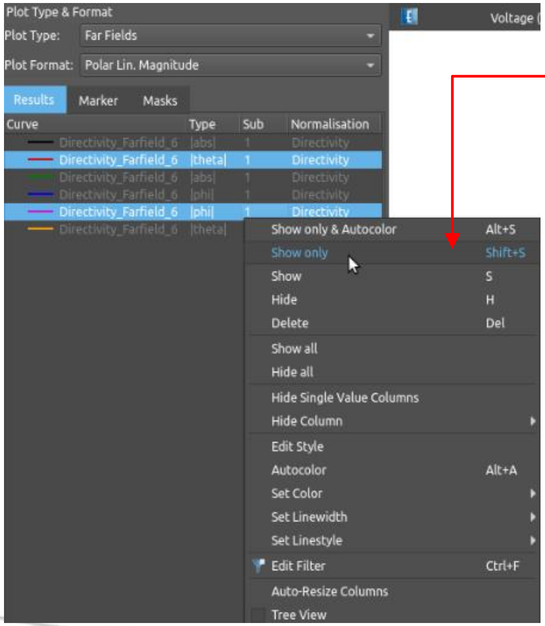


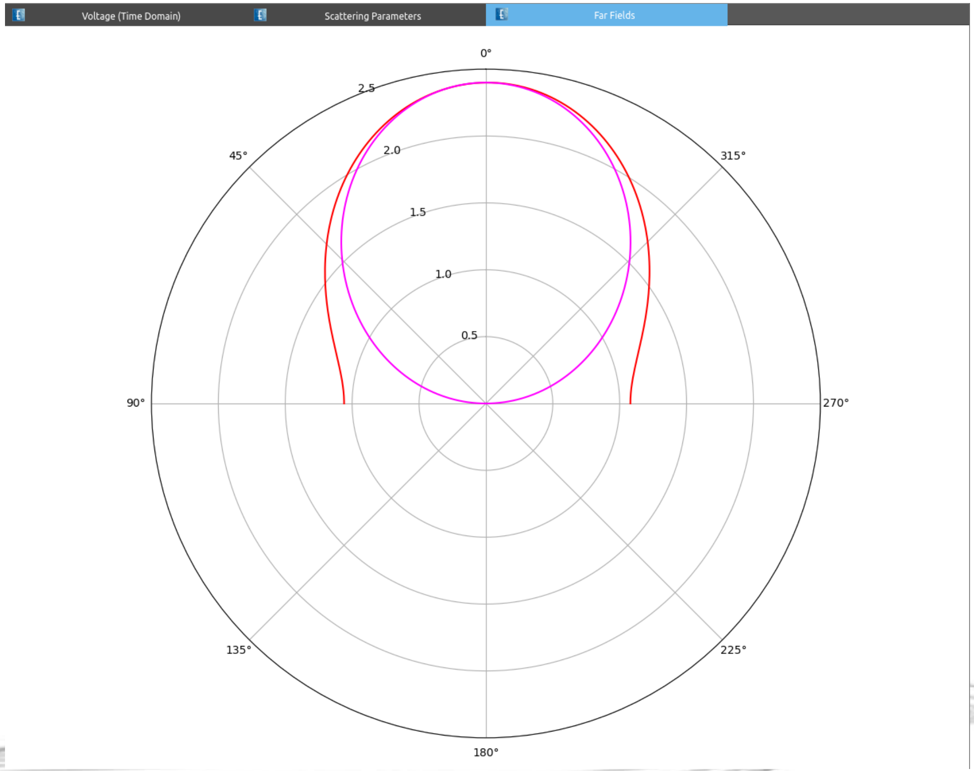


Step 9: Results

- Click on [+] to create a new group
- Select “Far Fields” for the “Plot Type”
- Select “Polar Lin. Magnitude” for “Plot Format”
- Select the “Directivity_Farfield_6” type “theta” and type “phi” and select “Show only”

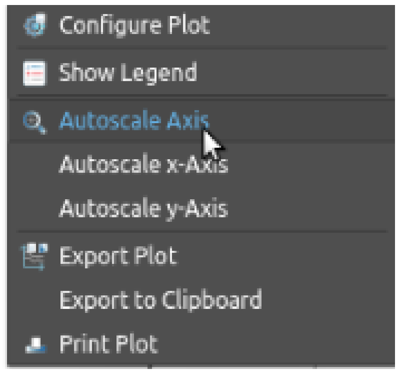






Step 9: Results

- Anytime the plot zooms in or out of the desired view, it can be centered right clicking on the plot and selecting "Autoscale Axis"

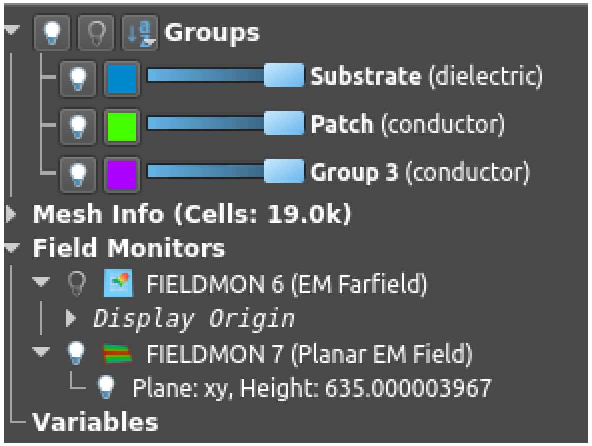


Results:

- *The different results can be viewed by adding more Tabs (Voltage, S-Parameters, Impedance, Farfield) or selecting the Plot Type*
- *Result files are automatically detected in the list using a naming convention. Additional files can be selected from other folder by using the "Add File" button*
- *Click the files with right mouse to obtain a context menu e.g. to show or hide*

Step 10: Near fields

- Return to 2D Design Tab, select “Field Monitors”
- Switch off the Far field monitor
- Create new “Planar EM Field”, “Plane” by right clicking on “EM Field” and selecting “Create Plane”

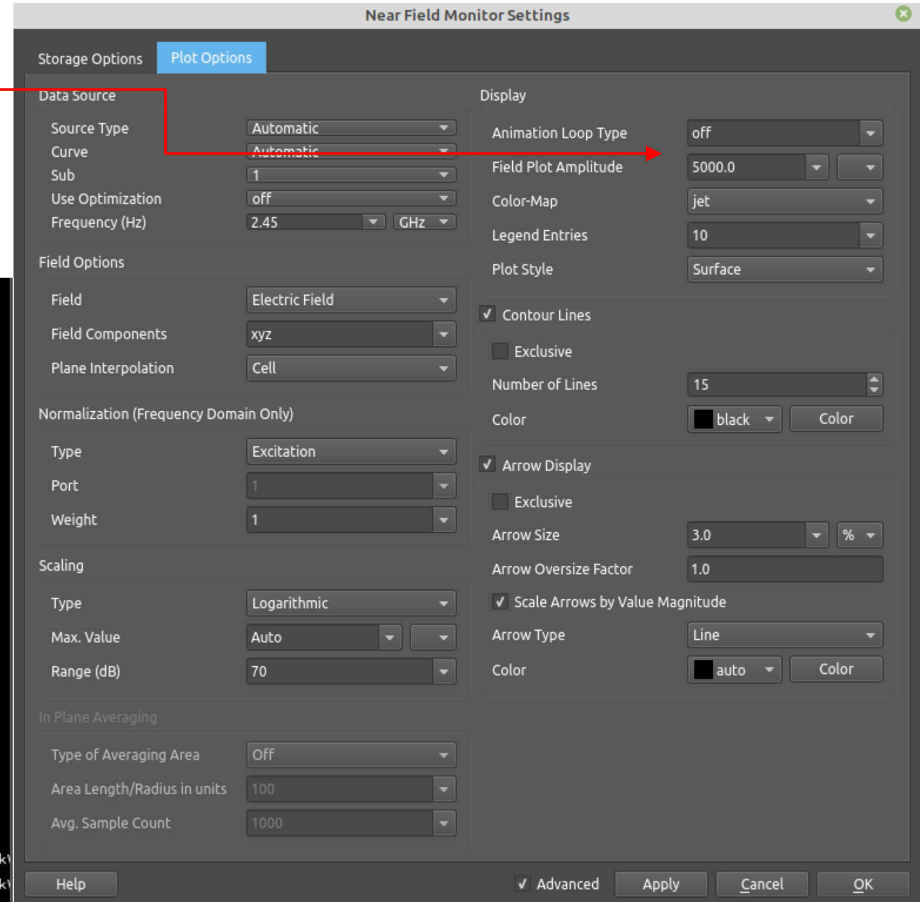


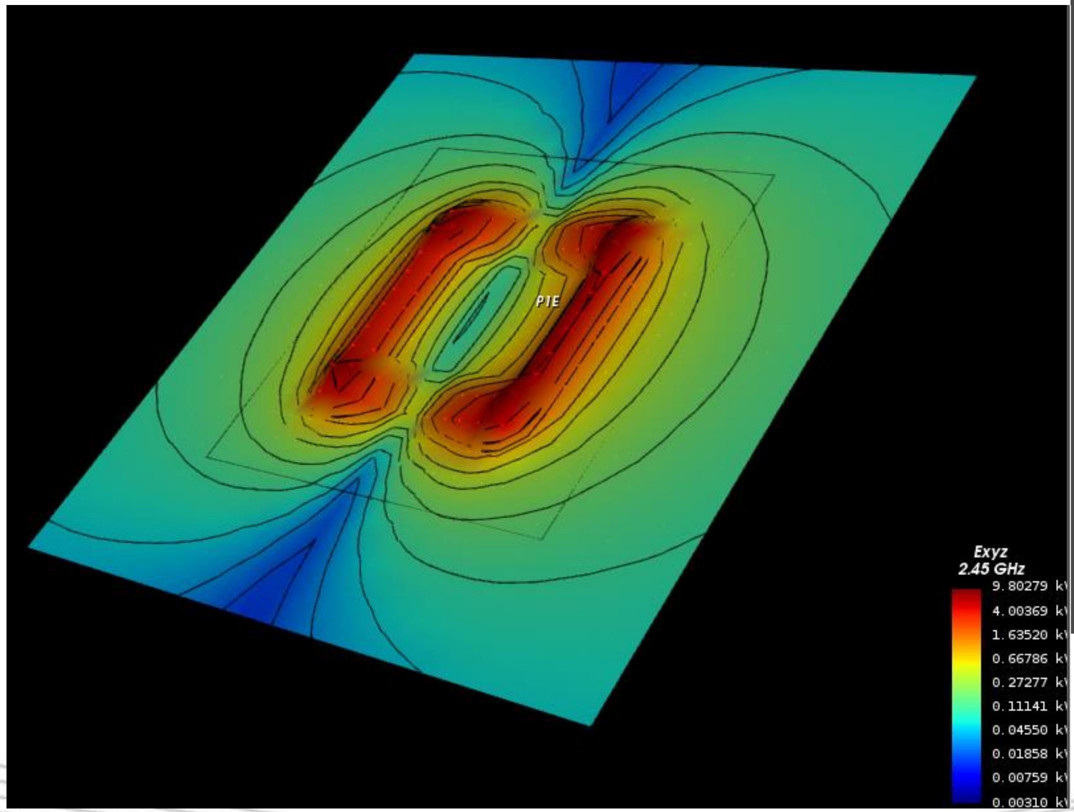
Remarks:

- **Up to now, no* ***near*** *field monitor has been defined, a 2nd simulation is needed*
- *EM Field planes consume less memory than EM Field volumes*
- *The number of frequency points increase the memory usage as well*
- ***The display parameters can be defined after the simulation. The list of frequencies is only available after simulation*

Step 10: Near fields

- Open „Planar EM Field” monitor
- Click on the “Plot options” tab
- Change „Amplitude” to 5000
- Press “Apply”
- Switch off groups to improve field display





Step 10: Near fields

- Open „Planar EM Field” monitor
- Click on the “Plot options” tab
- Change „Amplitude” to 50000
- Customize the “Arrow Display”
- Press “Apply”
- Switch off groups to improve field display