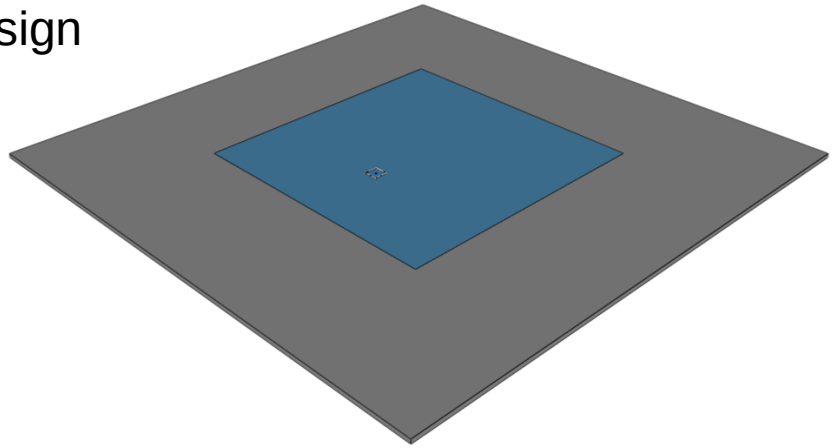
EMPIRE XPU Tutorial

3D Design - Patch Antenna started from Scratch

Overview

- Group creation
- Property setting
- Group height in 3D Design
- 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

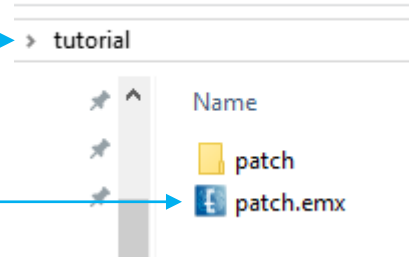
Help:

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

Start:

- Start Empire XPU
- Select “New Project”, OK
- Select File → “Save As”
- Select/create storage folder e.g. „tutorial“
- Enter file name, e.g. „patch“
- Save

Folder structure after save



Subfolder (automatic)
Input file

Step 1: Group creation

- Right click on group name „#001“
- Select „Edit Name“, enter „Substrate“
- Open Group
- Double click on „Height“
- Set Height: z=0...635

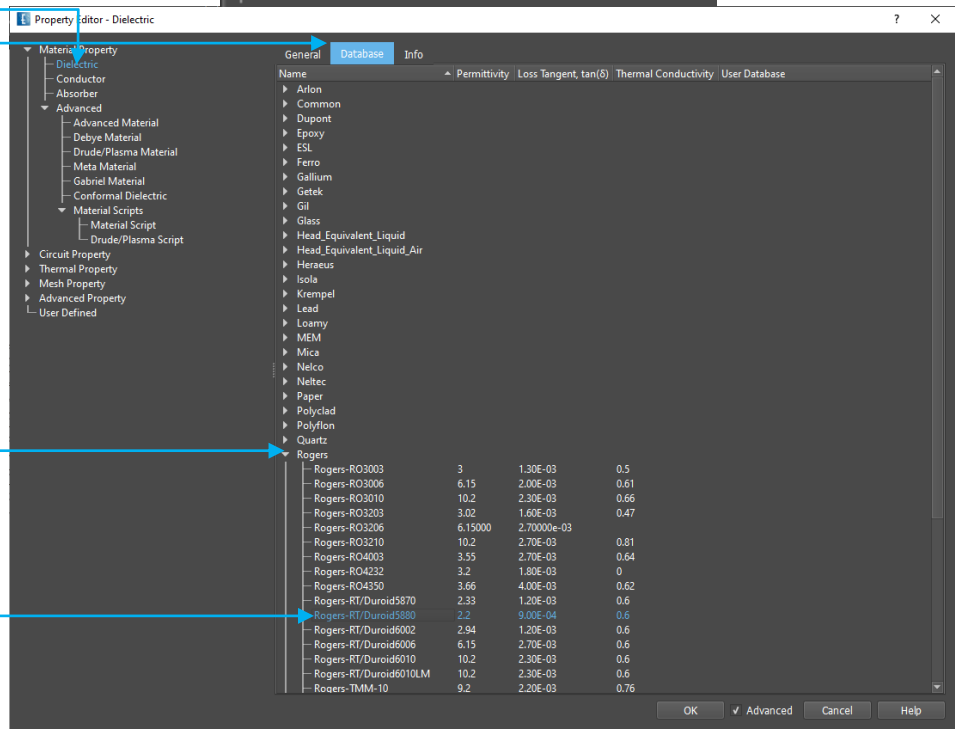
In EMPIRE, the structure is organized in groups. It is recommended to separate objects with different properties on different groups. Groups are used
• to group objects with common properties
• to define the height of objects, like boxes and polygons
• to set the insertion point of library objects, like 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 property
- Select Dielectric
- select Database tab

- Select Rogers
- → Rogers-RT/Duroid5880
- 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*

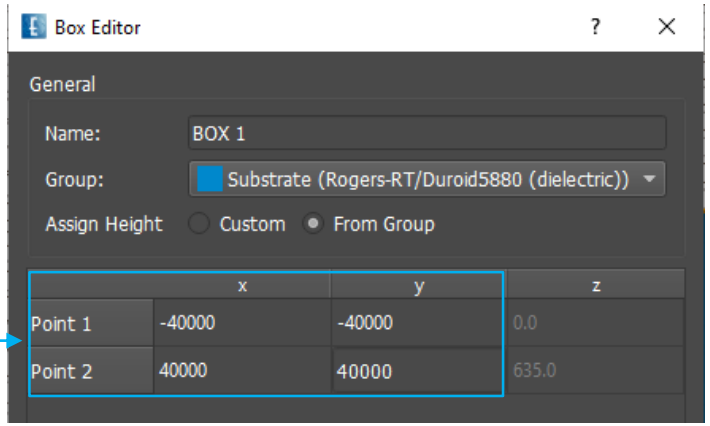
Comments: The default property is conductor (PEC). Here, we want to define the substrate and therefore the property is changed. Groups may have multiple properties, if they are not contradictious.

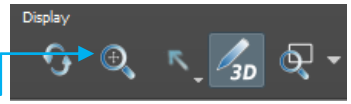
Step 3: Create Box with values

- Click „Create Box“
- 2 clicks at arbitrary points (to be edited later*)

- Long click to use group height

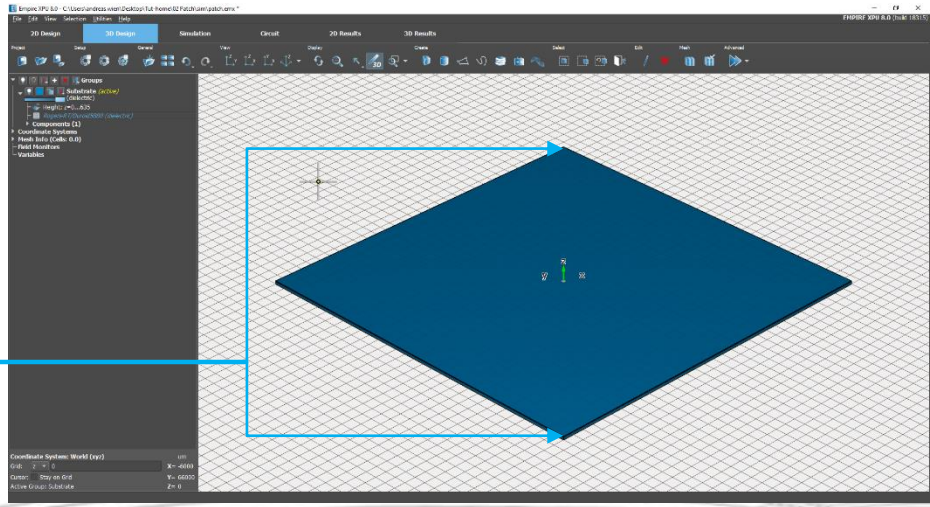
- Adjust coordinates
- OK





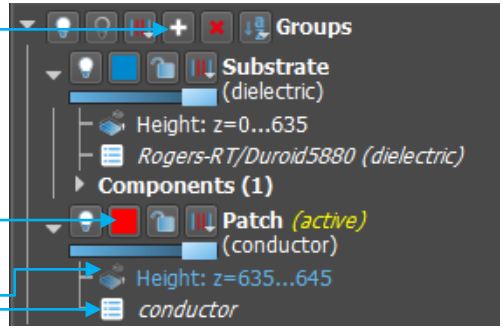
- Zoom extents

Comment:
**It is also possible to click box points at desired location after zooming out (wheel backward)*

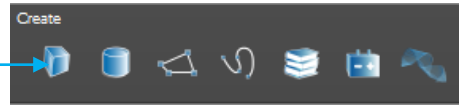


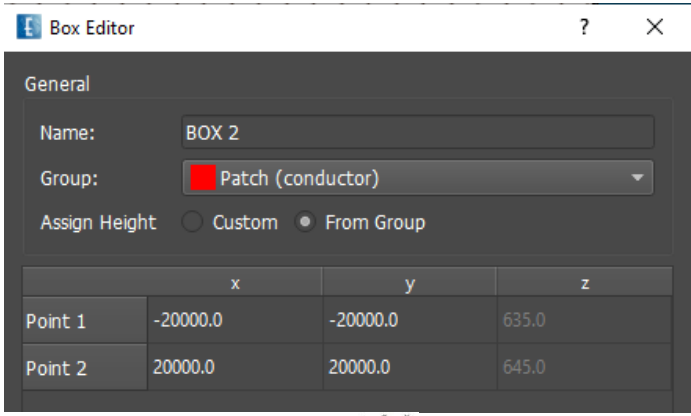
Step 4: Create Box with cursor

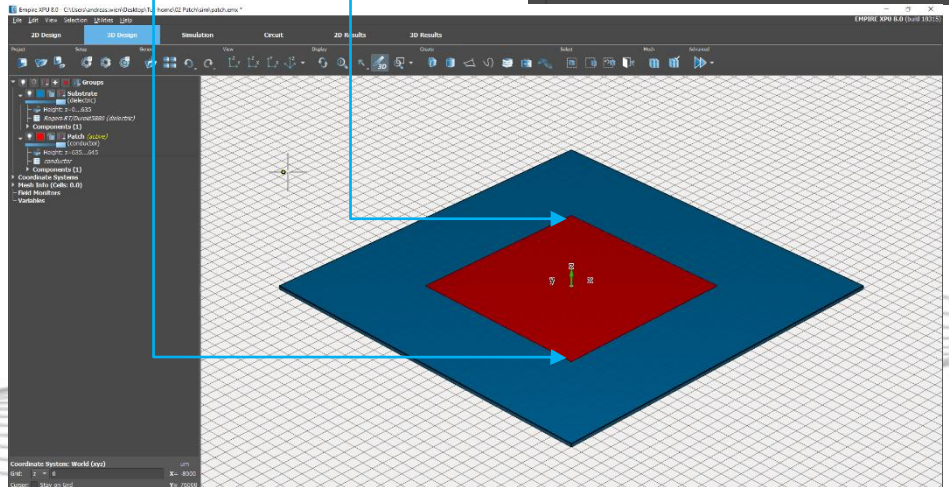
- Click „Add Group“
- Enter „Patch“ as name, recolor
- Double click Height, set z=635...645
- Keep conductor as property



- Create patch:
  - Click ‘Create Box‘
  - Left click at x=20000, y=20000
  - Left click at x=-20000, y=-20000
  - Long click to use group height
  - OK





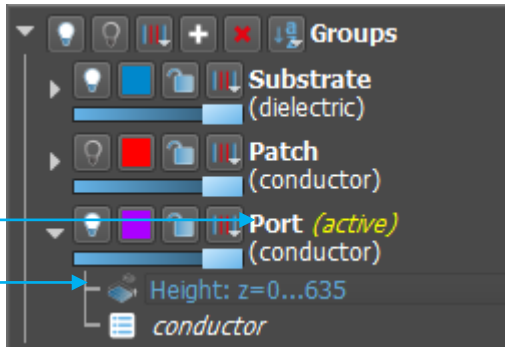


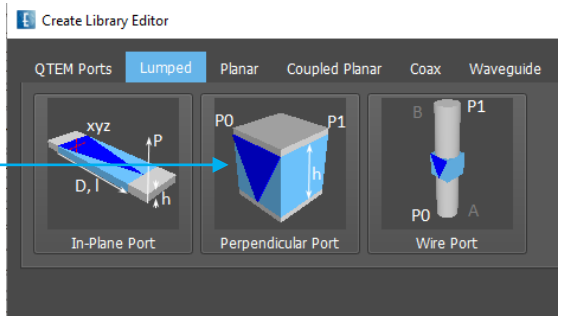
Step 5: Port definition

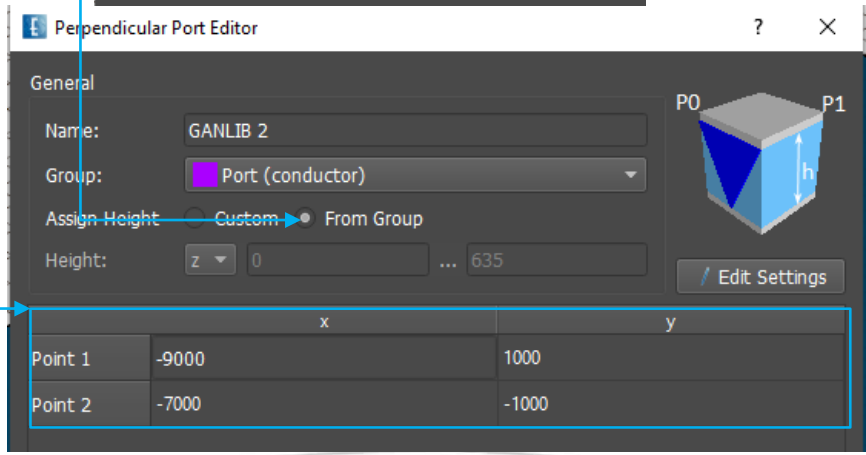
- Create group, rename to „Port“
- Set Height: z=0…635

- Click ‘Create Source‘
- Select ‘Perpendicular Port‘
- 3 clicks at arbitrary points (to be edited later)

- Select Height „From Group“
- Adjust point values
- OK
- Redraw





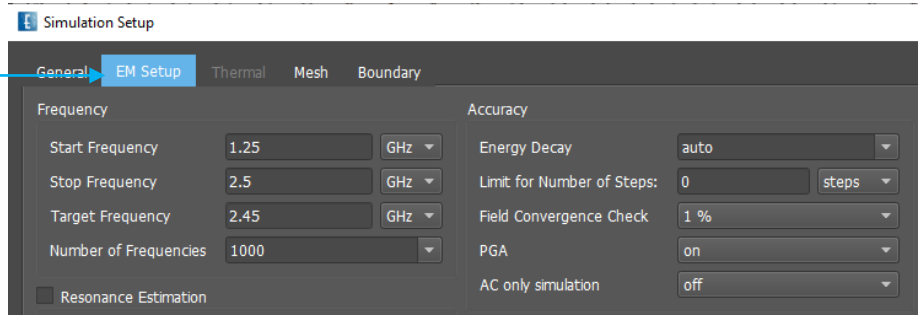


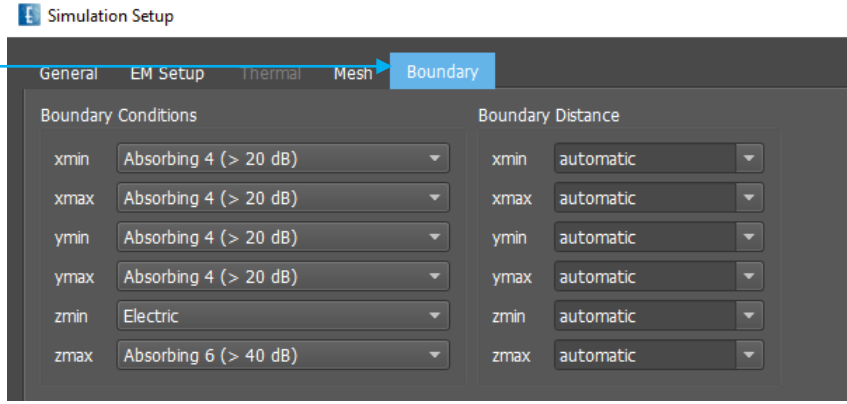
Comments:
** The port size should be chosen according to an excitation in reality, e.g. the inner diameter of a coaxial feed cable*
• Perpendicular ports connect top and bottom conductors, height should be distance between patch and ground
• Port numbers should be unique unless simultaneous excitation 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 „Simulation Setup“

- „EM Setup“ Tab:
  - Start Frequency: 1.25 GHz
  - Stop Frequency: 2.50 GHz
  - Target Frequency: 2.45 GHz
- Boundary Tab:
  - zmin to Electric, keep zmax,
  - change the others to “Absorbing 4 (>20dB)”
- OK

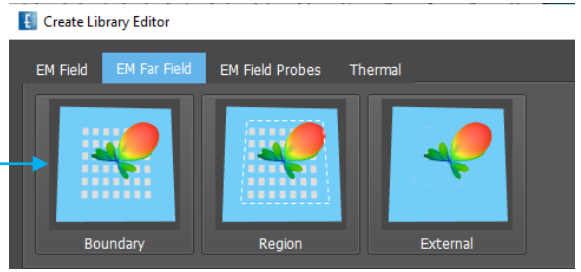




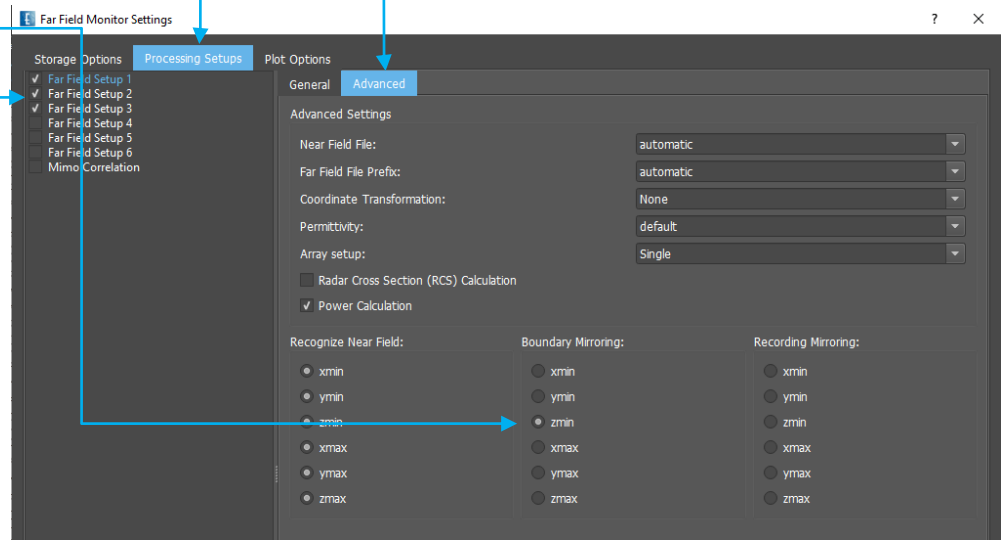
Simulation setup:

- *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 End 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 can be smaller in the non-radiation directions to save simulation time)*

Step 7: Far Field recording



- Click „Create Field Monitor”
- Select “EM Far Field”, Boundary
- Select “Processing Setups”
- ♦ Select Far Field Setup 1, Advanced Tab
- ♦ Enable Boundary Mirroring at zmin
- Repeat ♦ with Far Field Setup 2-3
- Exit with OK



Far fields:

• Far fields are obtained in post processing by a transformation of the near field recorded on a Huygens surface

• 3 Transformation Setups are predefined: 1:phi=0 and 2:phi=90 cuts, 3: 3D Pattern

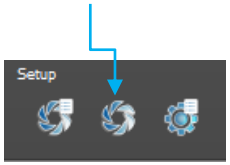
• Further adjustments can be set in the Processing Setups:

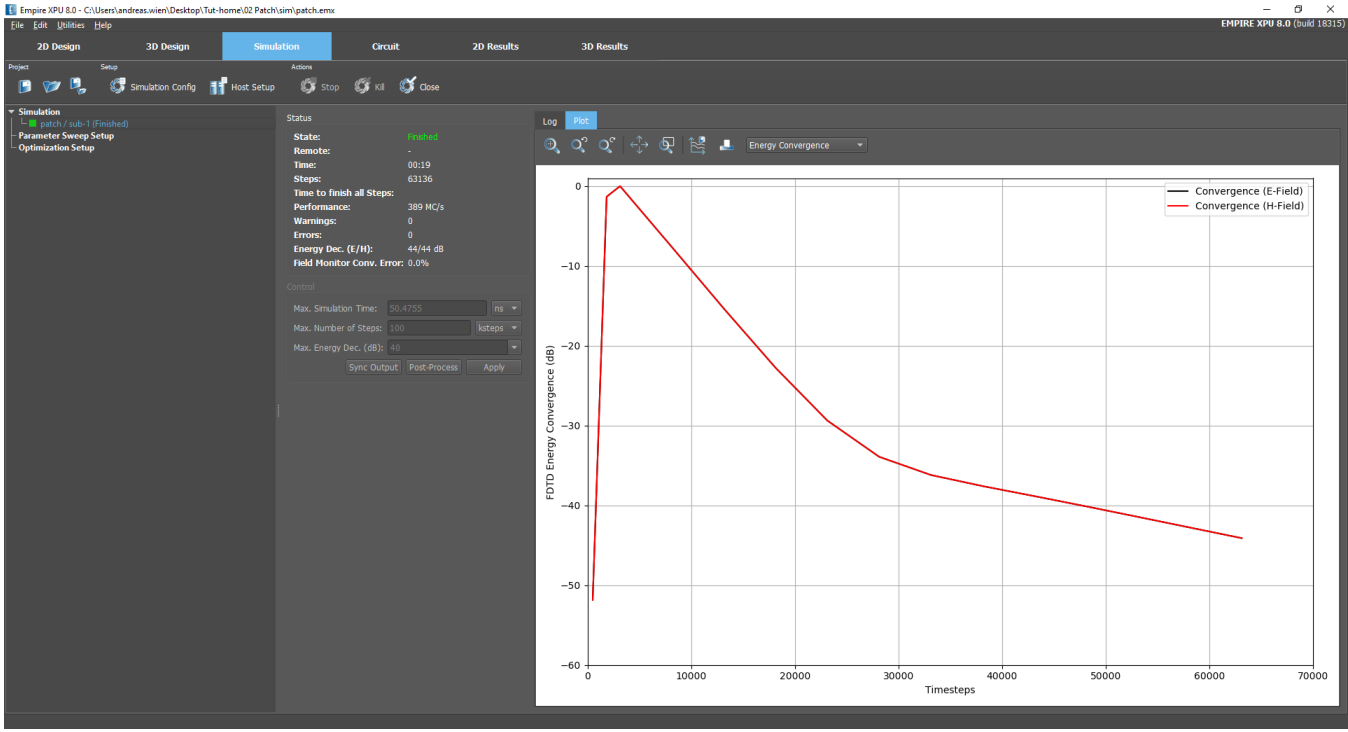
- *Normalization (Gain, Directivity, maximum, …)*
- *Sweep mode (2D cuts, 3D pattern,…)*
- *Rotation*
- *Far field components (linear, circular, …)*
- *Mirror planes*
- *Phase center*

By default, the target frequency is automatically set, more can be added

Step 8: Simulation

- Click “Start Simulation”,
- ‘OK’

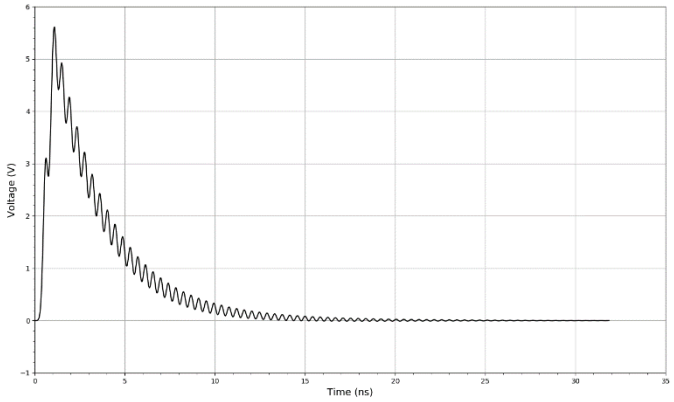




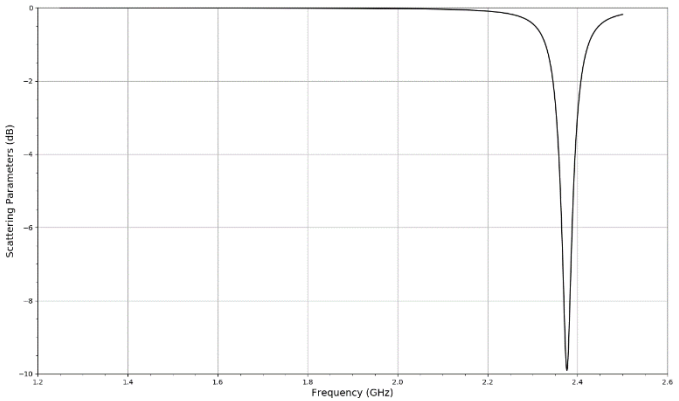
Meshing and simulation:

- *The automatic meshing automatically enlarges the simulation domain to account for the far field transformation*
- *With “Start Simulation” the structure is checked, meshed and prepared for simulation*
- *As soon as the plot comes up the simulation starts, the evolution of the energy in time domain is shown*
- *When the end criteria has been reached, the post processing is triggered and the S-parameters are available*

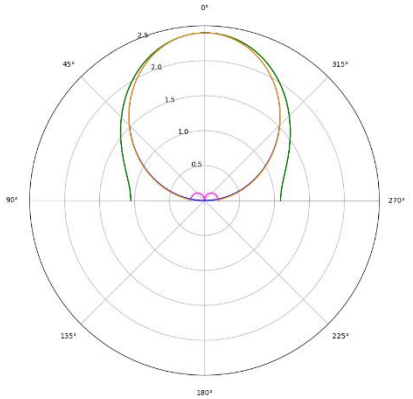
Step 9: Results



Voltage ut1



S-Parameter s11

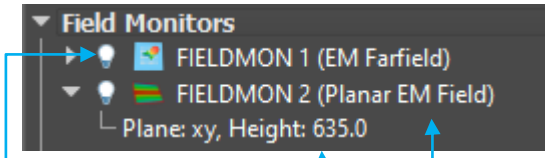


Farfield (Polar Lin)

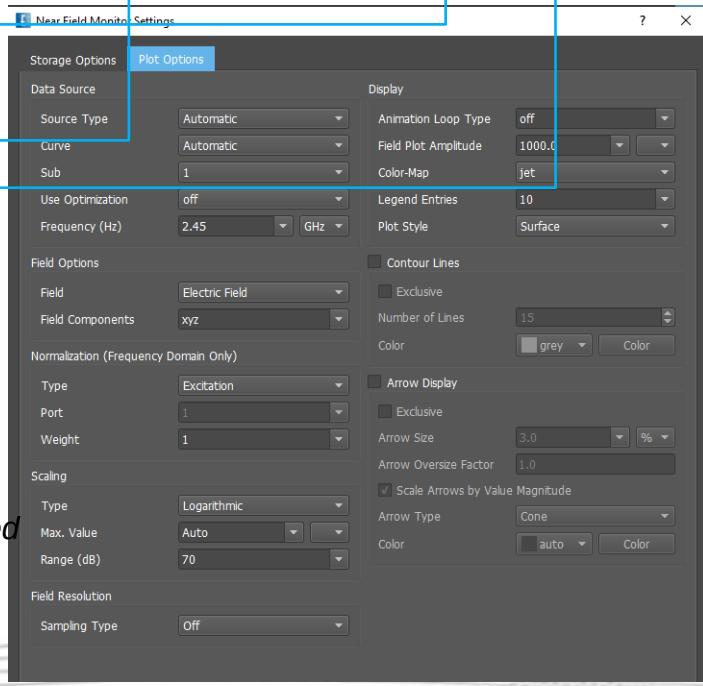
Results:

- *The different results can be viewed by selecting the 2D Results Tab and select Plot Type (Voltage, S-Parameters, Impedance, Farfield, Additional) or use "Add Result" to create multiple result tabs*
- *The Plot Format can be changed on the left (e.g. angular plot for Far Field, select polar)*
- *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 show or hide*

Step 10: Near fields



- Return to 3D Design Tab,
- Click Create Field Monitor
- Select „EM Field”, “Plane”, OK
- Open Field Monitor, Set Height:635
- Repeat simulation * 
- 3D Results Tab, switch off Farfield monitor
- Right click „Planar EM Field“ – Edit
- In “Plot Options”** change settings (see next page)
- Click “Apply”
- Optionally switch off groups to improve field display

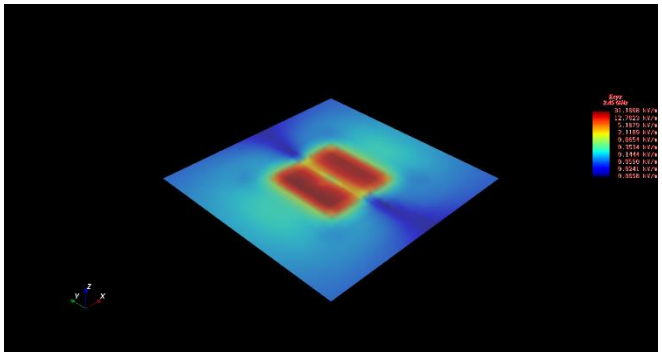


Remarks:

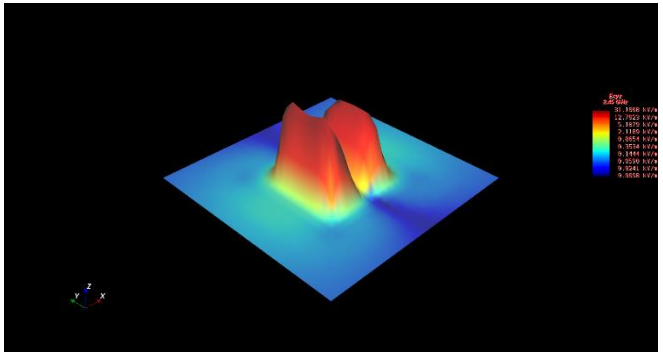
- **Up to now, no **near** field monitor has been defined, a 2nd simulation is needed*
- *EM Field planes consume less memory than EM Field volumes*
- *The number of frequency points increase the memory usage as well*
- *The “Plot Options” can be defined after the simulation.*

Step 11: Plot Options

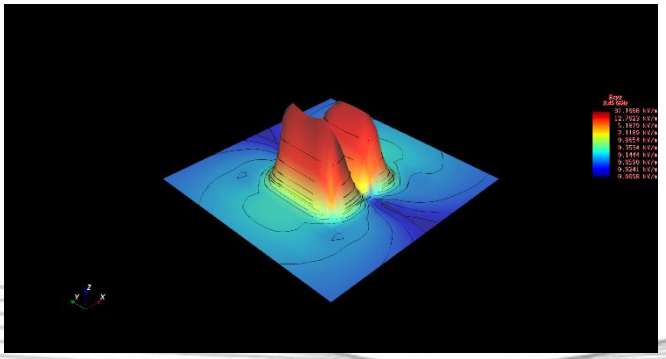
1 *Defaults, electric field*



2 *Defaults*
+ Field Plot Amplitude: 50000



3 *Defaults*
+ Amplitude: 50000 + Contour Lines + Color: Black



4 *Defaults*
Countour lines + Arrow Display - Scale Arrow