

Simulation of The Capacitive Loaded Monopole Antenna with CST



Content

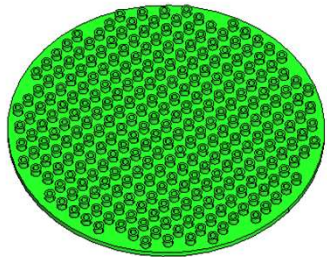


- Capacitive loaded monopole antenna
- Simulation setup in CST
 - Define a project template.
 - Create the model
 - Define the Material
 - Specify boundary
 - Define the port
 - Setup the monitor
 - Set up meshing
 - Run the simulation
- Results

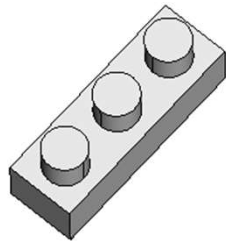


• Capacitive Loaded Monopole Antenna

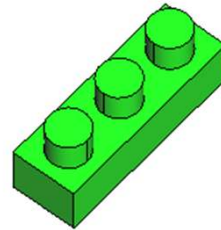
• Dimensions and basic components



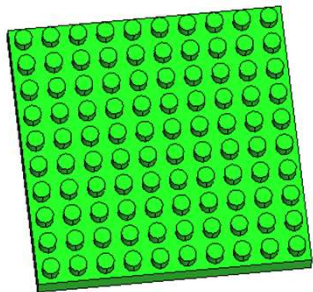
Ground plane



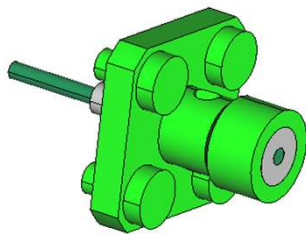
Dielectric cell



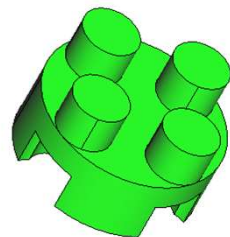
Metal cell



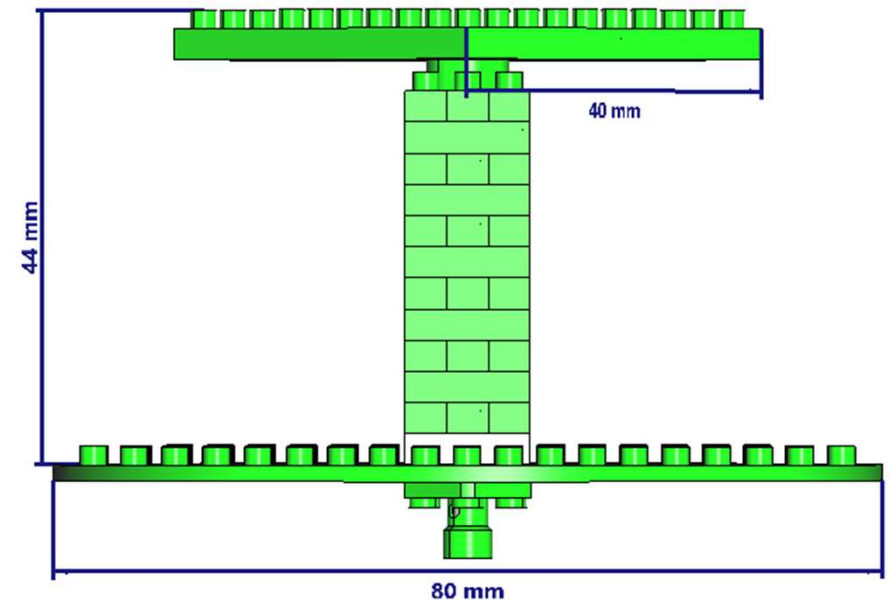
10x10 metal cell



Coaxial connector

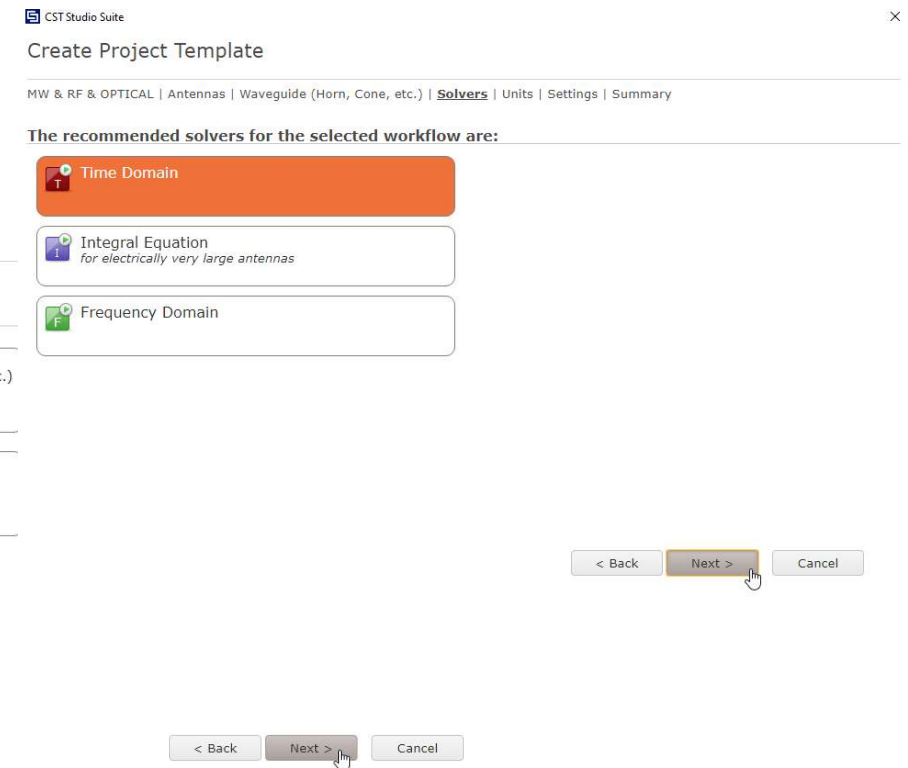
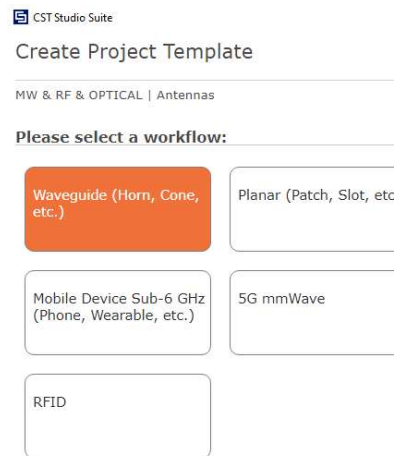
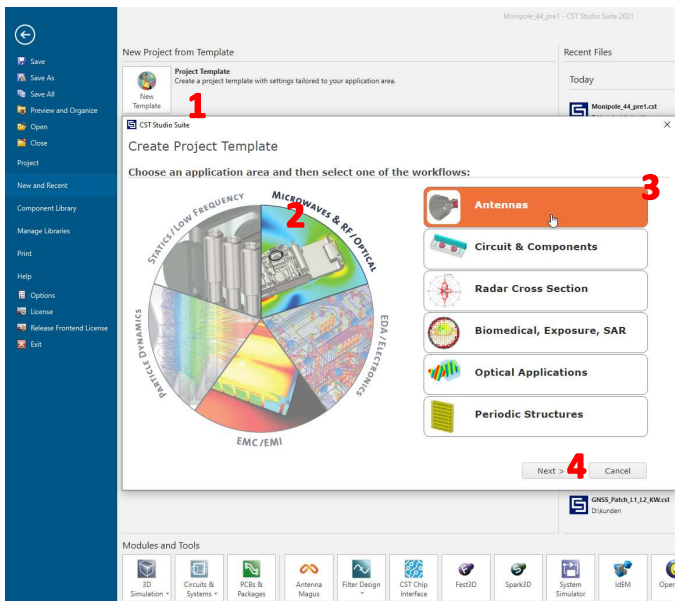


Circular metal cell



• Simulation Setup – Define Template 1

- Choose “New Template” → “MICROWAVES & RF / OPTICAL” → “Antennas” → “Next”
- Choose “Waveguide” → “Next”
- Choose “Time Domain” → “Next”



• Simulation Setup – Define Template 2

- Select the units for the simulation → “ Next”
- Dimensions: mm, Frequency: GHz...
- Define the frequency range → “Next”
- Frequency from 0.5 GHz to 1.9 GHz
- add monitors for “farfield” at 1.2 GHz
- Name the template → “Finish”

CST Studio Suite

Create Project Template

MW & RF & OPTICAL | Antennas | Waveguide (Horn, Cone, etc.)

Please select the units:

Dimensions:

Frequency:

Time:

Temperature:

Voltage:

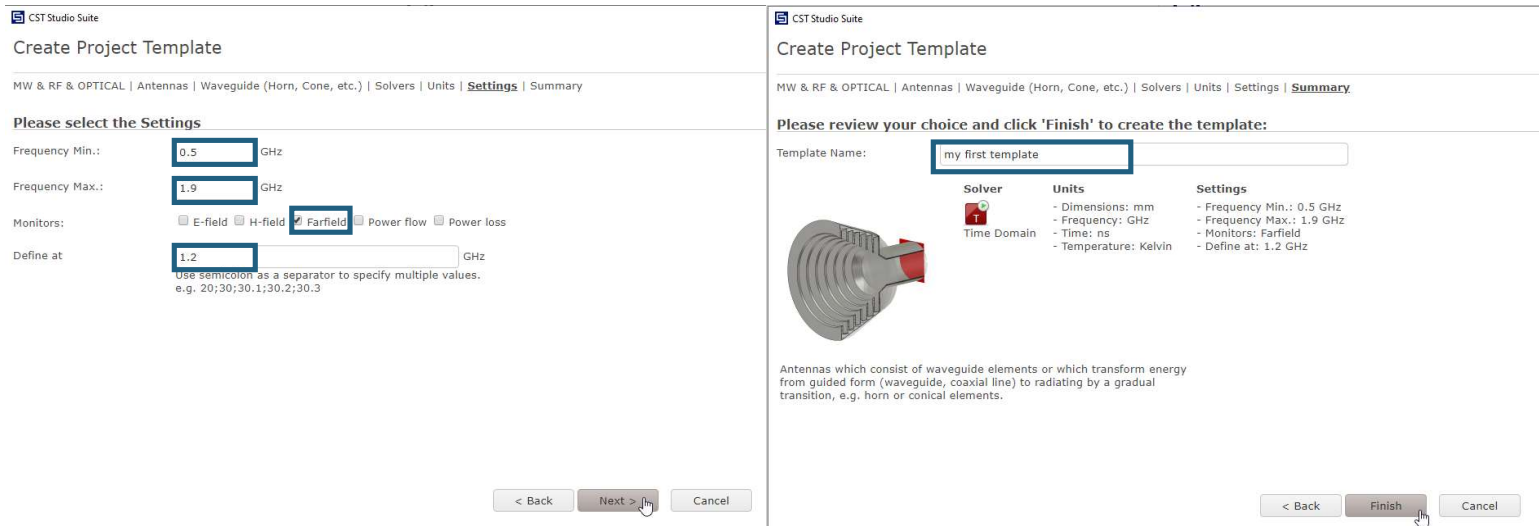
Current:

Resistance:

Conductance:

Inductance:

Capacitance:

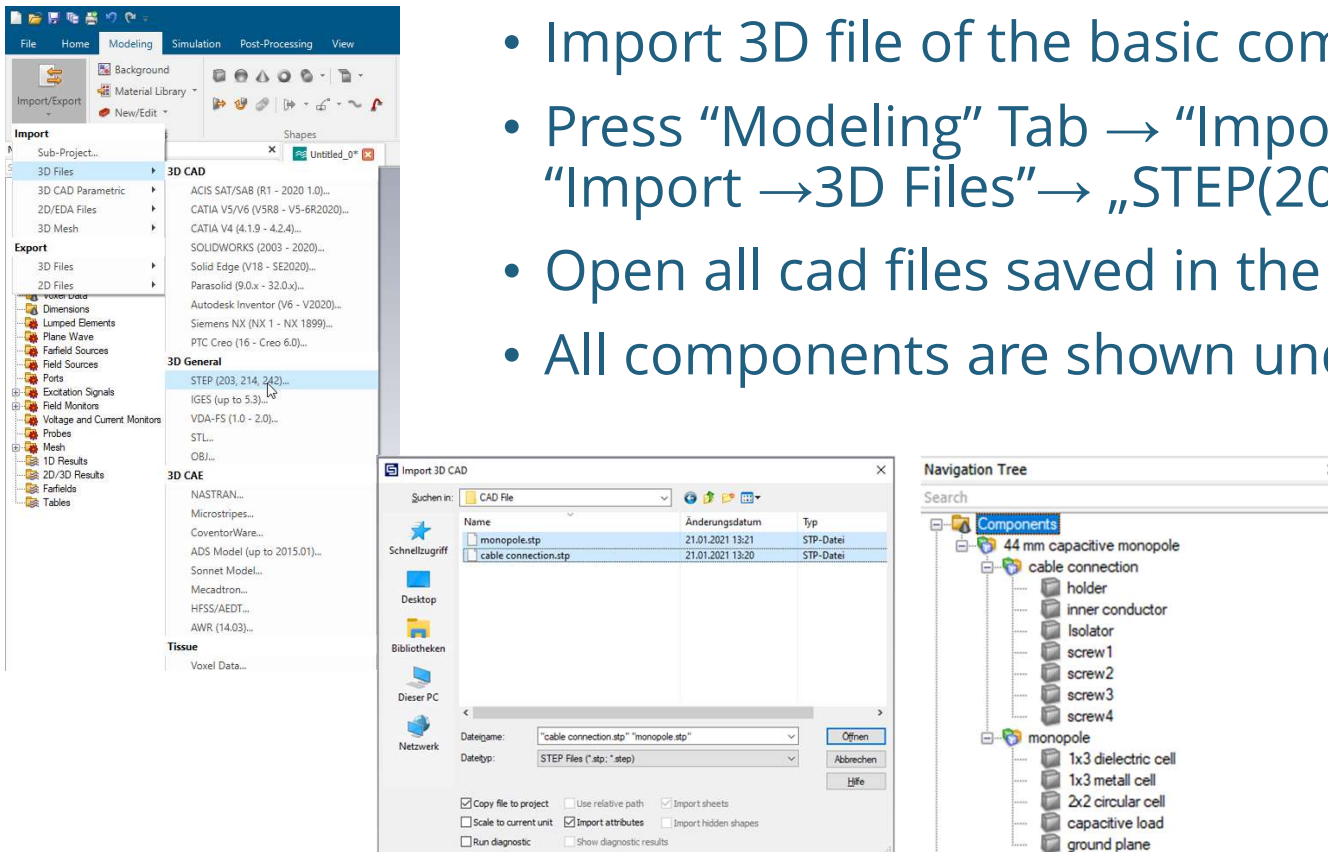


Solver	Units	Settings
Time Domain	- Dimensions: mm - Frequency: GHz - Time: ns - Temperature: Kelvin	- Frequency Min.: 0.5 GHz - Frequency Max.: 1.9 GHz - Monitors: Farfield - Define at: 1.2 GHz

Antennas which consist of waveguide elements or which transform energy from guided form (waveguide, coaxial line) to radiating by a gradual transition, e.g. horn or conical elements.

• Create the model: import the cad files

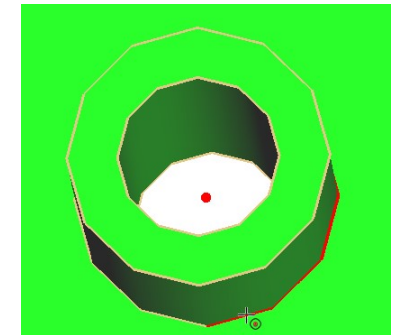
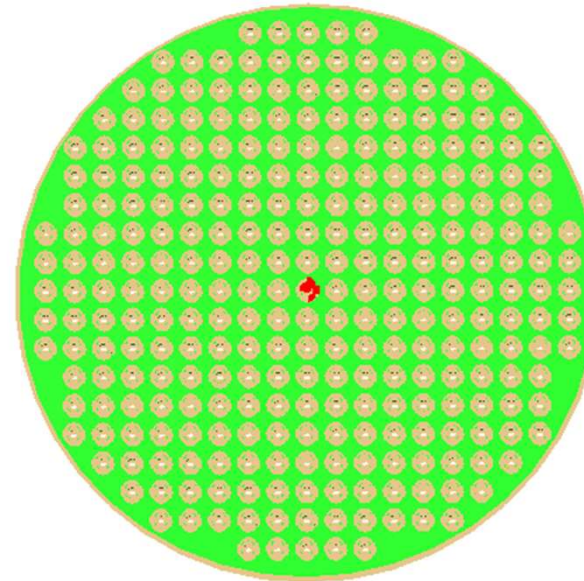
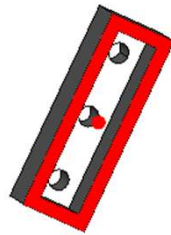
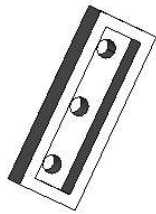
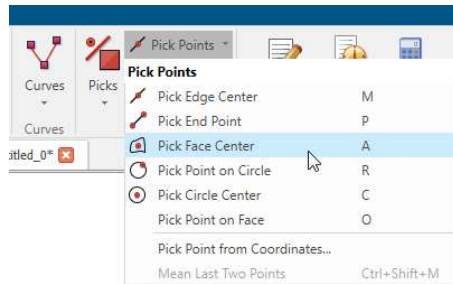
- Import 3D file of the basic component
- Press “Modeling” Tab → “Import/Export” → click under “Import → 3D Files” → „STEP(203,214,242)...”
- Open all cad files saved in the local folder
- All components are shown under „Component” in CST



The screenshot displays the CST Studio Suite interface. The 'Modeling' tab is active, and the 'Import/Export' menu is open, showing the 'Import' sub-menu with '3D Files' selected. The '3D CAD' section lists various file formats, with 'STEP (203, 214, 242)...' highlighted. Below this, the 'Import 3D CAD' dialog box is open, showing a list of files: 'monopole.stp' and 'cable connection.stp'. The 'Dategame' field is set to 'cable connection.stp' and 'monopole.stp', and the 'Datebyp' is set to 'STEP Files (*.stp;*.step)'. The 'Navigation Tree' on the right shows the imported components under 'Components', including '44 mm capacitive monopole', 'cable connection', 'holder', 'inner conductor', 'isolator', 'screw1', 'screw2', 'screw3', 'screw4', 'monopole', '1x3 dielectric cell', '1x3 metall cell', '2x2 circular cell', 'capacitive load', and 'ground plane'.

• Create the model: build dielectric cells 1

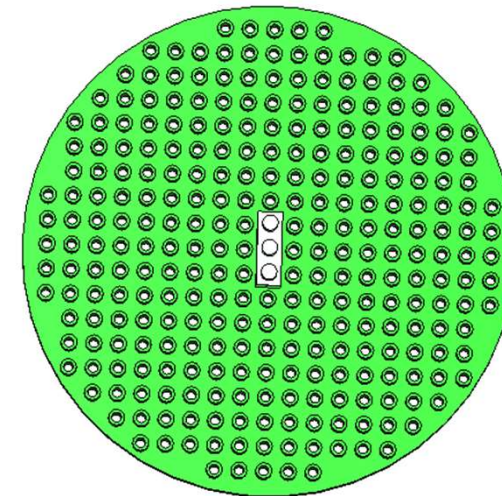
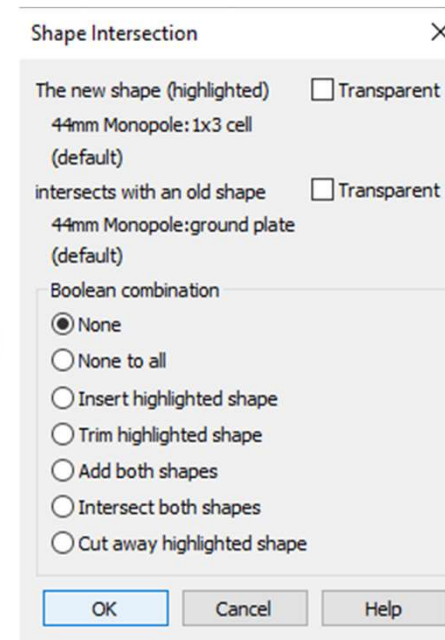
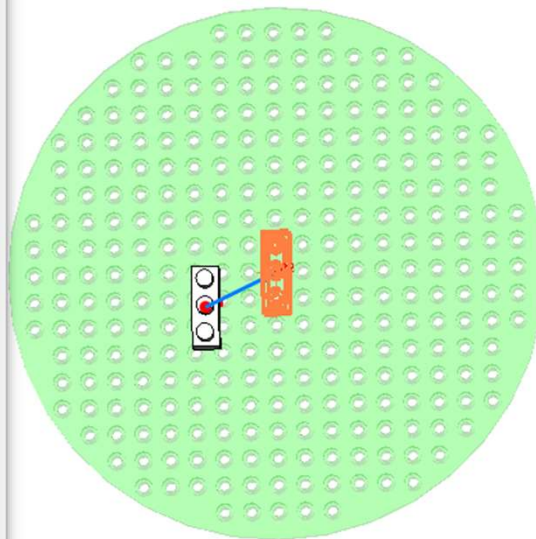
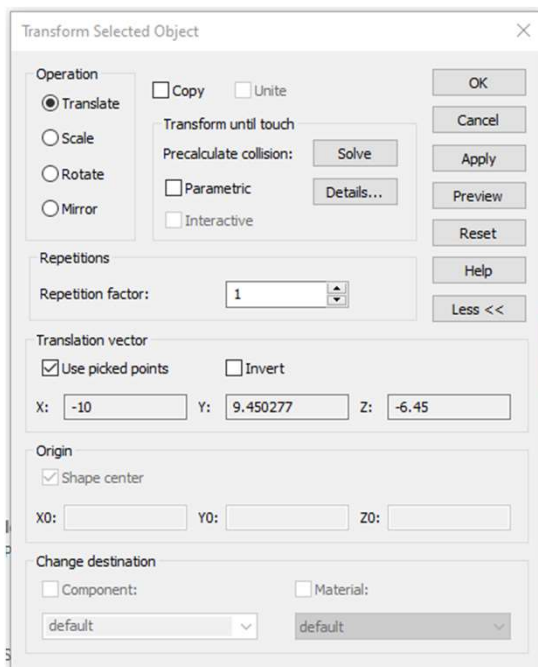
- Select 1x3 cell → click “Pick Points” → “Pick Face Center”
- Double click the red highlighted face shown as below to pick the center point



- Repeat the two steps and pick the center point on the center hole

• Create the model: build dielectric cells 2

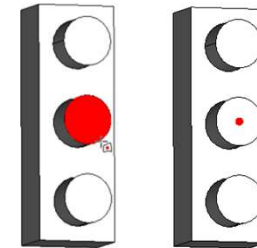
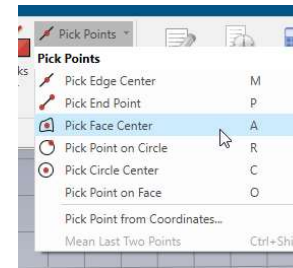
- Select 1x3 cell → click “Transform” → “Translate...” → “Ok”



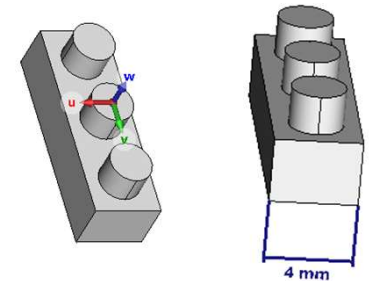
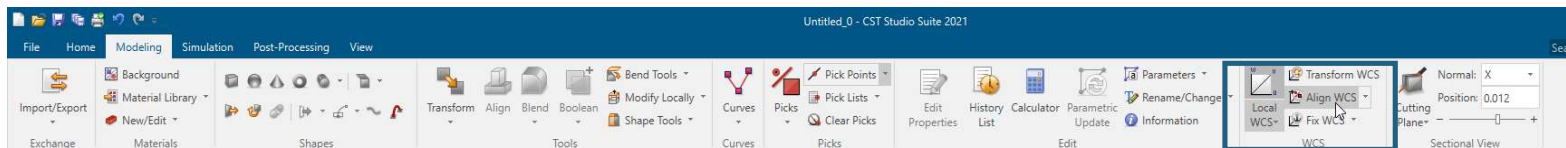
- Choose “None” in the “Shape Intersection” window. Now the first dielectric cell should be located in the middle of the ground plane

• Create the model: build dielectric 3

- Select 1x3 cell → click “Pick Points” → “Pick Face Center”



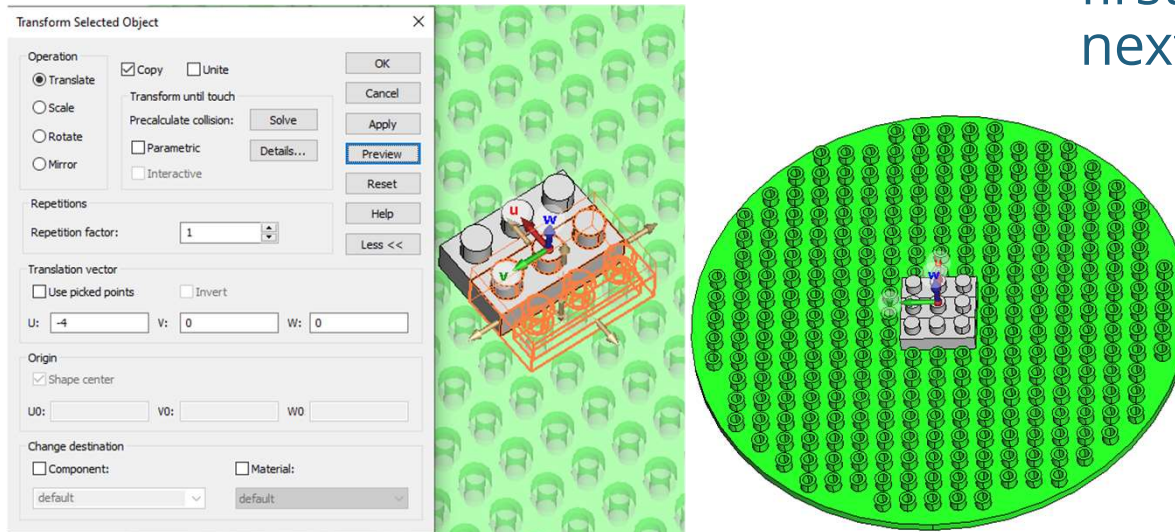
- Click the “Local WCS” → “Align WCS”, the local WCS is at the center of the selected cell.



- Select 1x3 cell → click “Transform” → “Translate...” → check “Copy” → give 4 in direction U → “OK”
 - Since the 1x3 cell has a width of 4 mm, it should be shifted both left and right to generate the other two dielectric cells on the same plane.

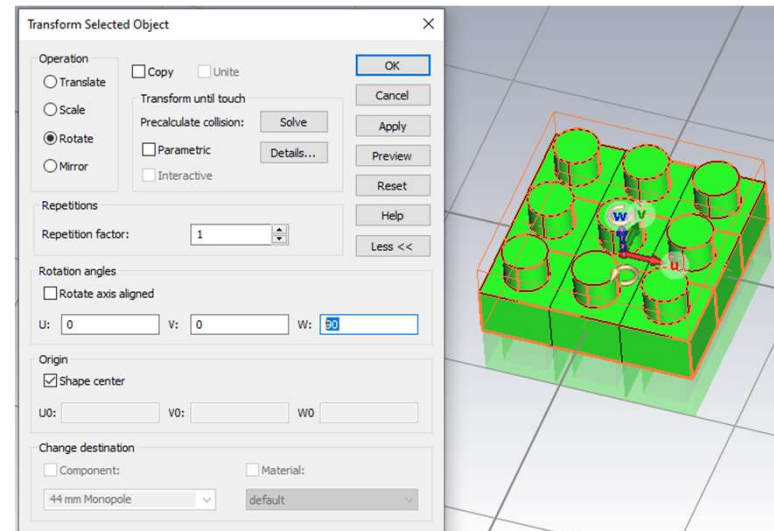
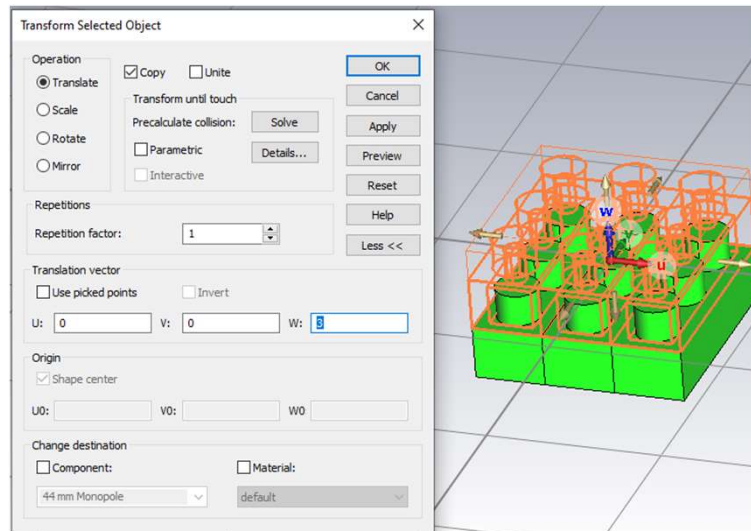
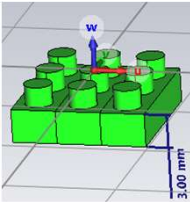
• Create the model: build dielectric cells 4

- Repeat the last step by give “-4” in direction U
- All three dielectric cells are generated on the ground plane
- The same procedure can be repeated with the metal cells to generate the first metal layer
- The other option to generate the first metal layer is shown in the next folie.



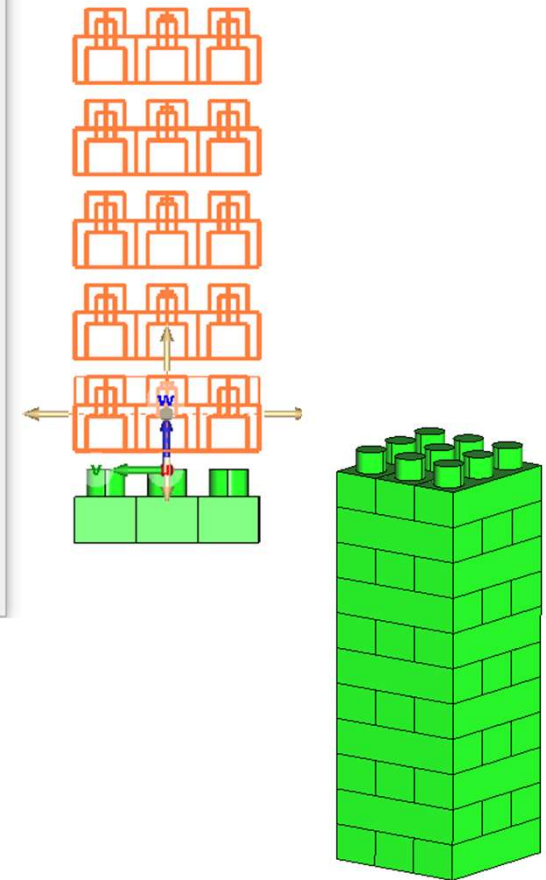
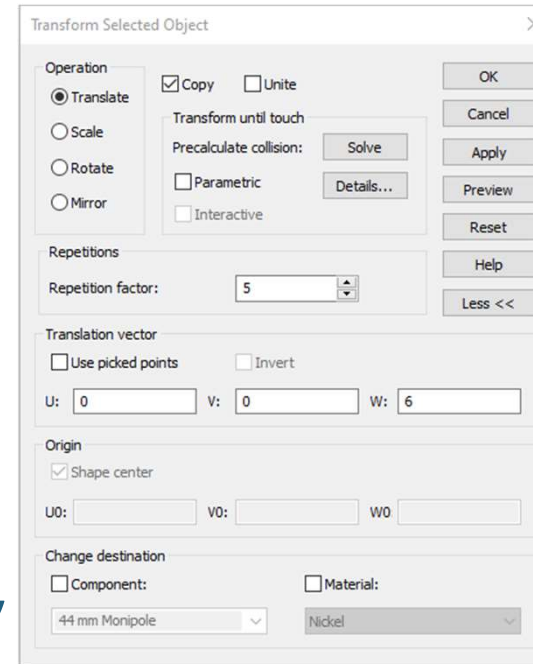
• Create the model: build first layer metal cells

- Since the first layer of metal cells are 90° rotated comparing with the dielectric cells and the cell has a height of 3 mm, therefore all three cells has to be translated first and then rotated.
 - Select all three 1x3 cells → “Transform”→ “Translate...”→ “W:3”→”OK”
 - Select all three new cells → “Transform”→ “Rotate...”→ “W:90”→”OK”



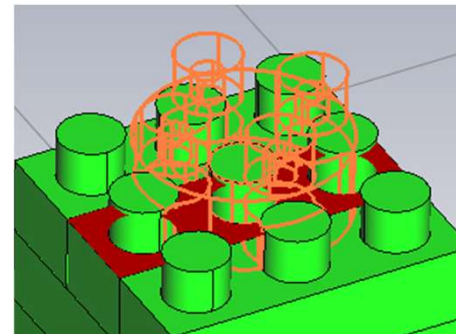
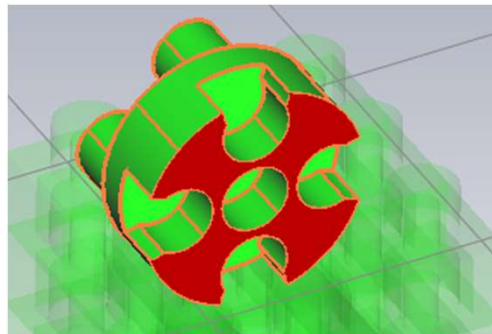
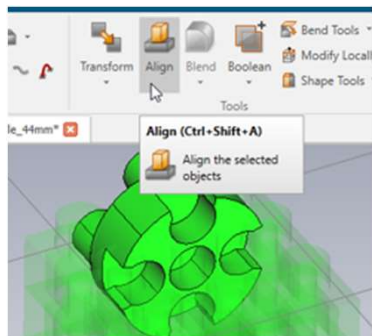
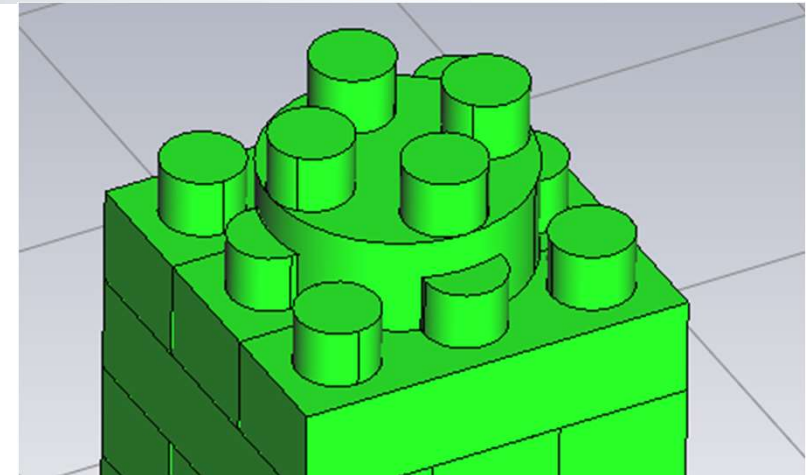
• Create the model: generate all metal cells

- For the 44 mm monopole antenna there are 11 layers which have a shift of 90 degree to the adjacent layer
- Generate all 5 layers in one direction
 - Select the first layer (the lowest) → “Transform” → “Translate...” → “W:6” → “Repetition Factor: 5” → “OK”
 - Repeat the last step by selecting the second layer in order to generate the other 4 layers in the perpendicular direction



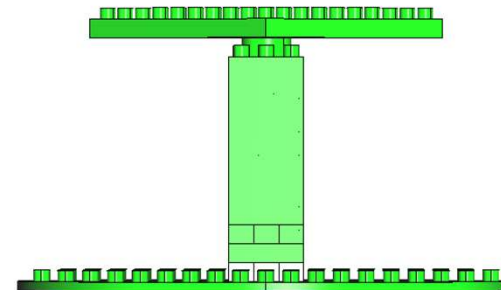
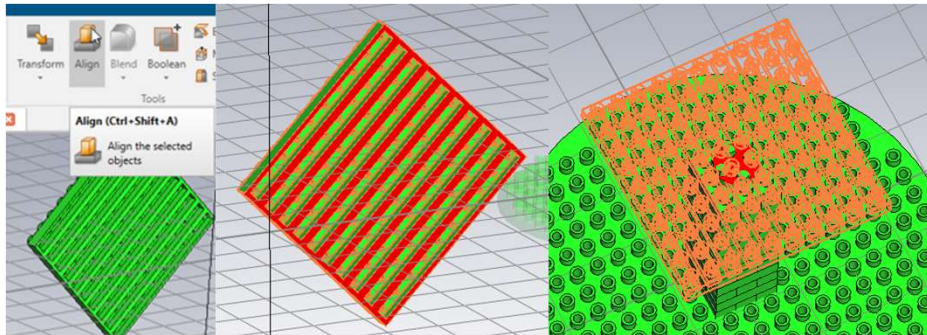
• Create the model: build the circular cell 1

- The circular cell is used to connect the capacitive load.
 - Select the circular cell → press “Align” → select the bottom of the circular cell → select the face of the middle cell on the top layer → press Return on keyboard

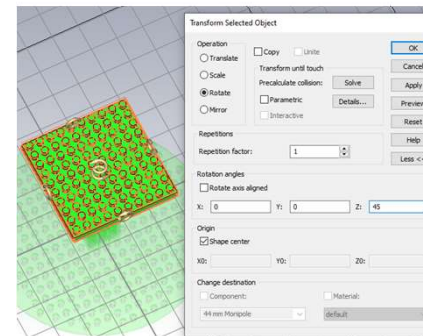


• Create the model: build the capacitive load plane

- The same procedure will be repeated as for the circular cell, select the object and then align two faces so that the capacitive plane will be located on top of the circular cell
 - Select the capacitive load → press “Align” → select the bottom of the capacitive load → select the top face of the circular cell → press Return on keyboard

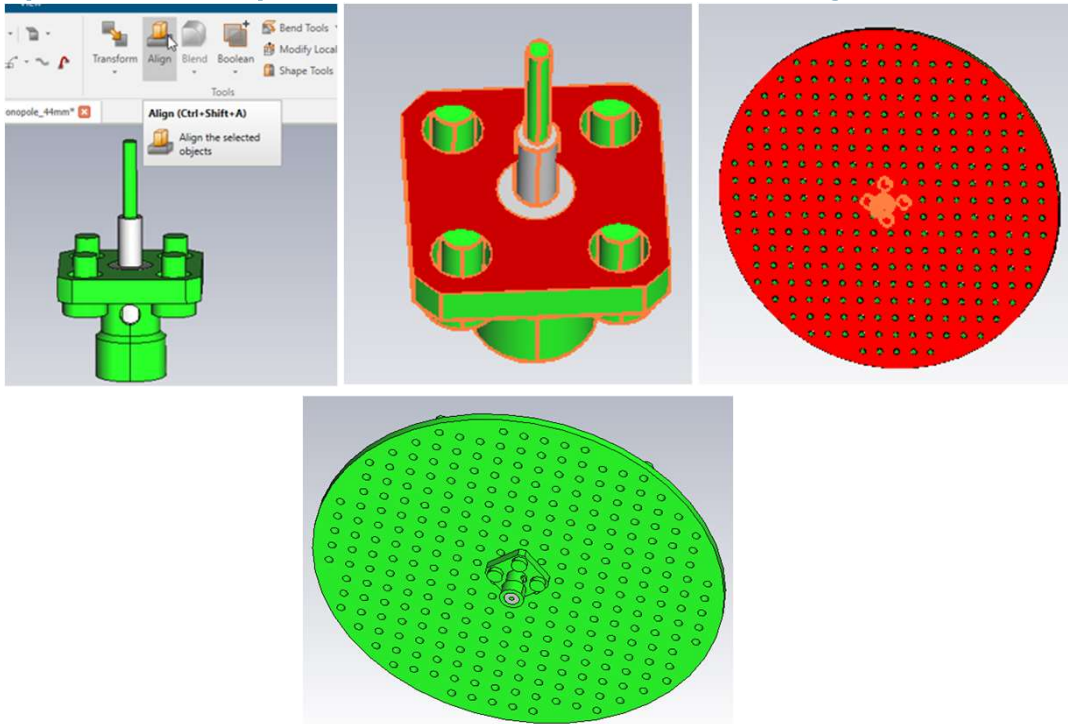


- Select the capacitive load → “Transform” → “Translate...” → “Z:45” → “OK”

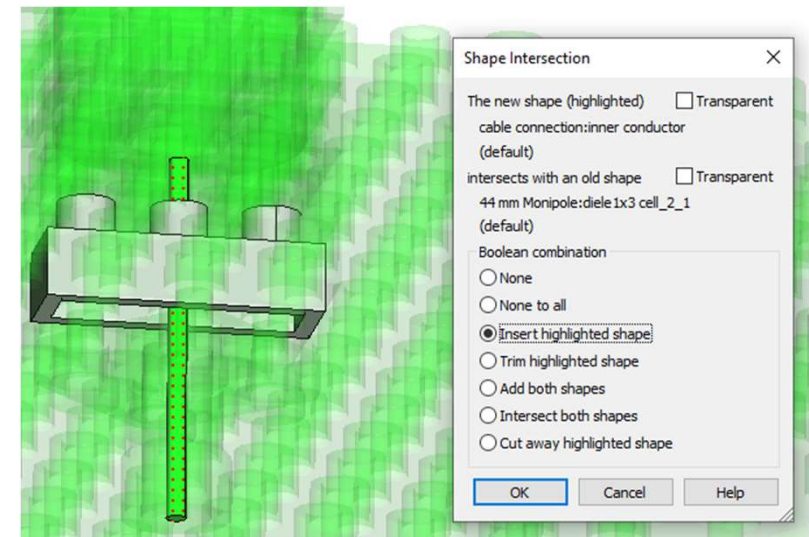


• Create the model: add connector

- Select the cable connector → click “Align” → pick both face of the cable connector and bottom of the ground plane → press return on the keyboard



- The inner conductor of the cable should go through two 1x3 cells at the bottom of the antenna.
- Select here “Insert highlighted shape” so that the inner conductor will go through the cells.

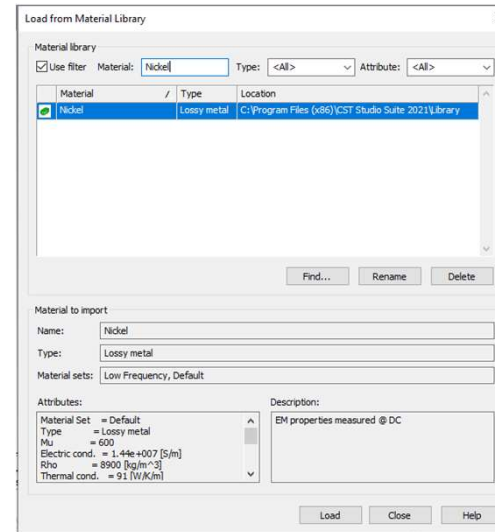
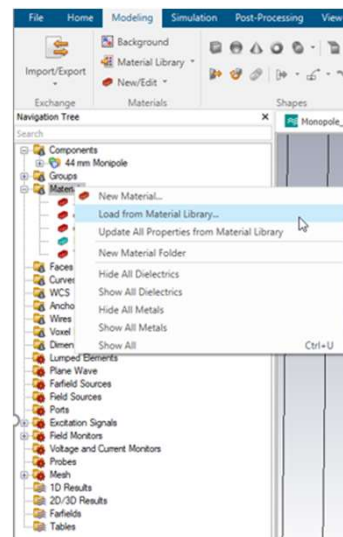
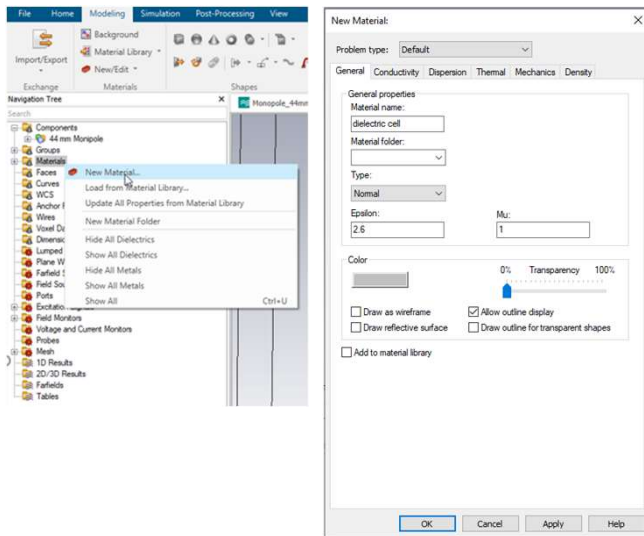


• Define the material property 1

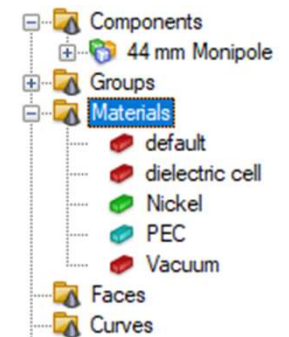
- Applied materials are listed in the table right
 - Step 1: generate the materials by right click of “Materials” → click “New Material...”
 - Step 2: Given in the opening window “Material name: dielectrikum” → “Epsilon: 2.6” → Color grey

Components	Material
Ground plane, Circular cell Capacitive load Cable connector (exclusive isolator)	Nickel
Dielectric cell, isolator (cable)	$\epsilon = 2.6$

- Step 3: Repeat step 1 and select “Load from Material Library...” → check “Use filter” searching “Nickel” → “Load”

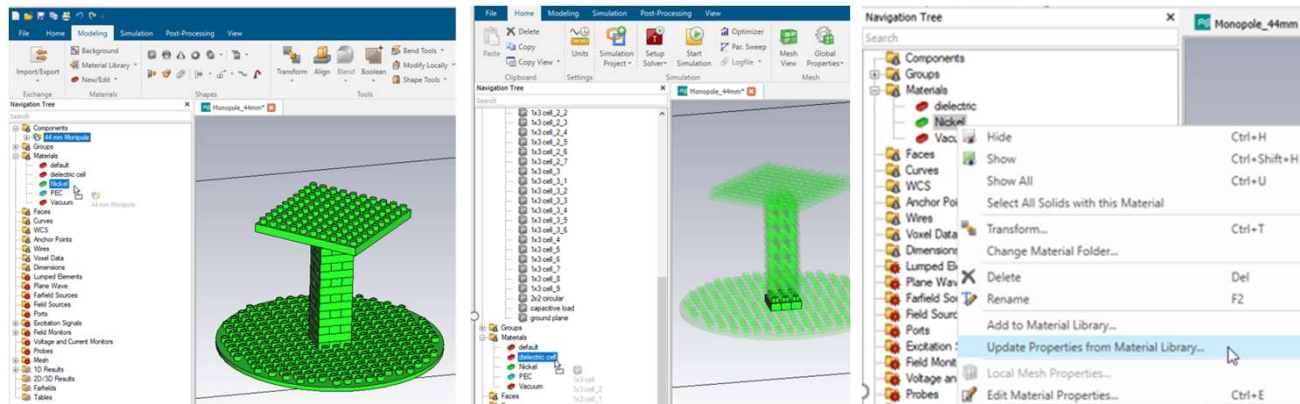


- Defined materials are all shown in menu “Materials”



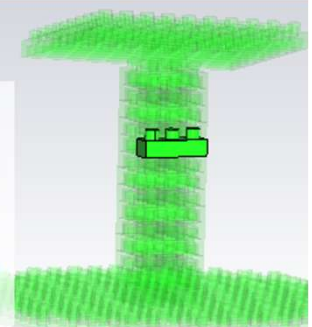
• Define the material property 2

- Drag all the components to “Materials → Nickel”
- Select all three dielectric cells and drag them to “Materials → dielectric cell”
- Since the material properties are already included in the CAD files, by right click on the corresponding materials, the material will be updated.



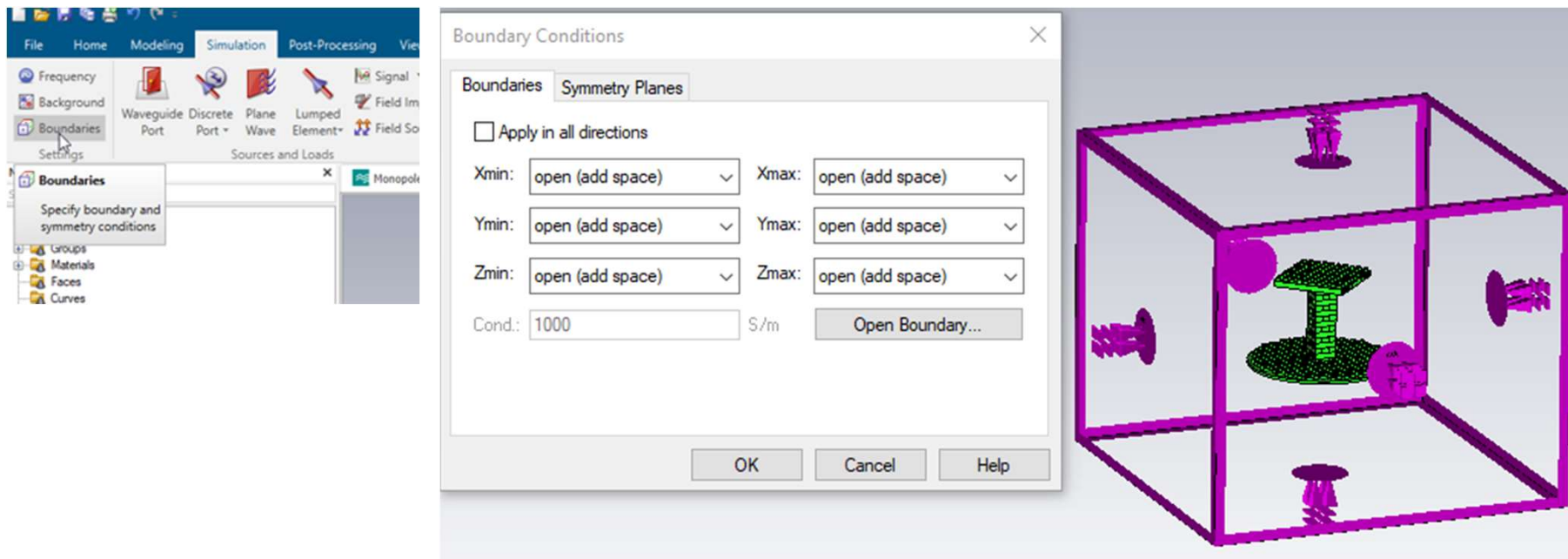
By clicking on the object, the corresponding material properties are shown in the information box.

44 mm Monopole:1x3 cell_1_1_4	
Material	Nickel
Type	Lossy metal
Mu	600
Electric cond.	1.44e+07 [S/m]
Rho	8900 [kg/m ³]
Thermal cond.	91 [W/K/m]
Specific heat	450 [J/K/kg]
Diffusivity	2.27216e-05 [m ² /s]
Young's modulus	207 [kN/mm ²]
Thermal expan.	13.1 [1e-6/K]



• Specify the boundary

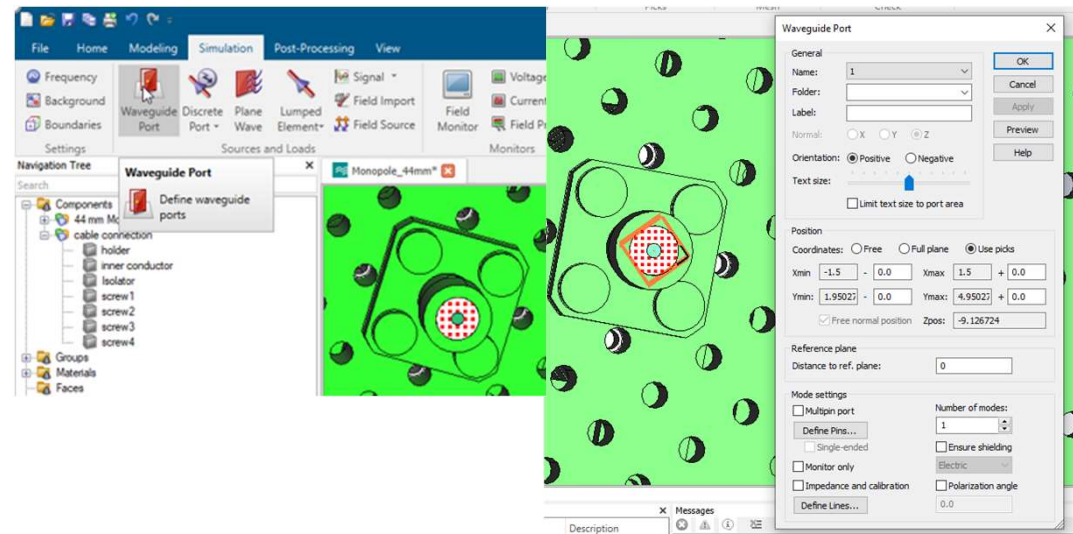
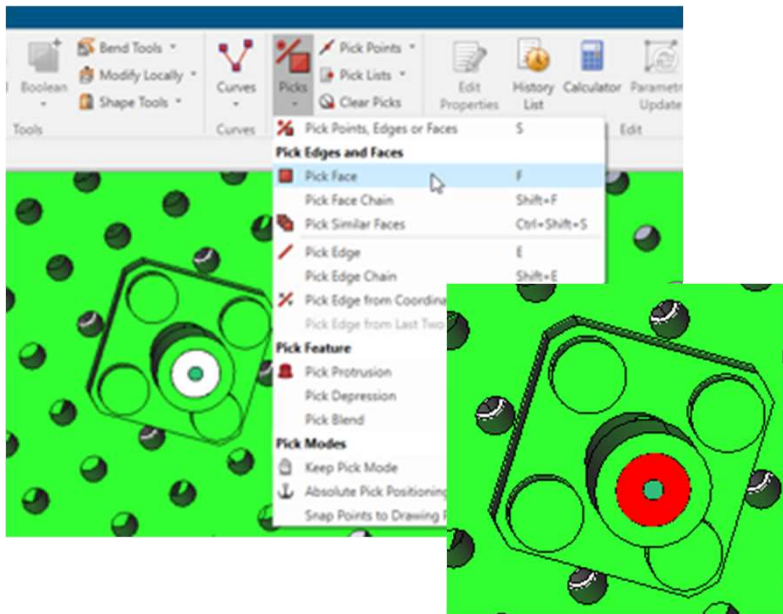
- Go to the Tab “Simulation” → press “Boundaries” → select for all directions “open (add space)” → “OK”
- The boundaries are added around the model



• Define the port

- Go to the Tab “Modeling” → press “Picks” → select “Pick Faces” → select the face on the cable connector shown right

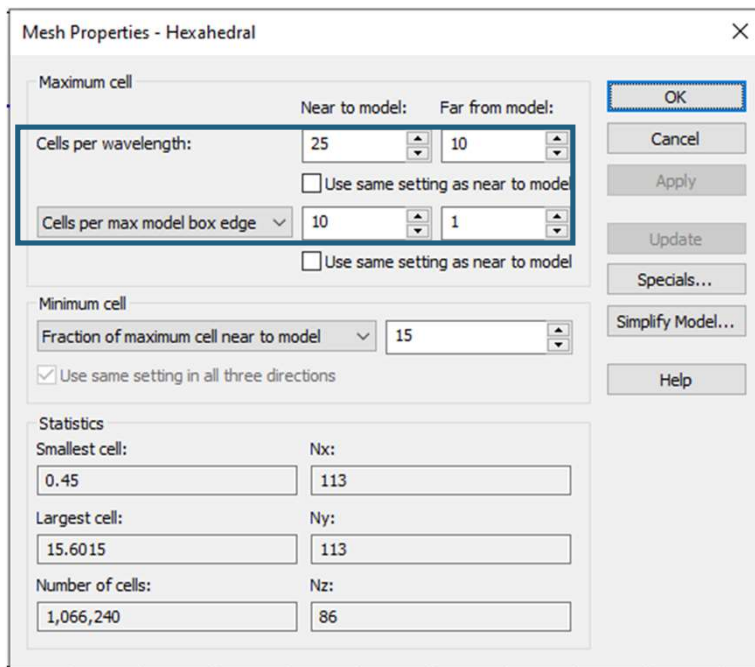
- Go to Tab “Simulation” → click Waveguide Port → an orange frame will be shown up and a window of setting up for waveguide port → default setting is fine, press “OK”



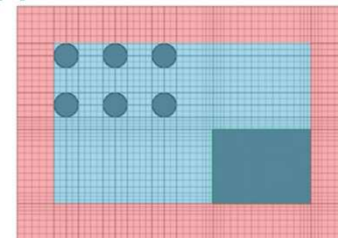
• Setup the mesh : global properties 1



- Global properties define the global meshing by limiting maximum cell and minimum cell.

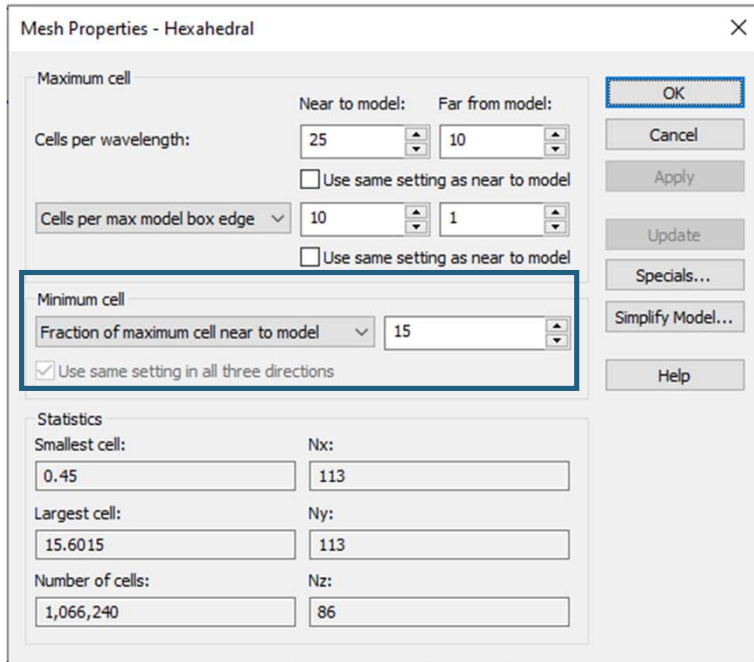


- Maximum cell can be defined by no. of cells either within one wavelength or within the largest box edge. Otherwise, the absolute values can be given for maximum cell.
- The meshing is discussed in two regions, namely “near to model” and “Far from model” which helps to reduce the total no. of meshing cells.
- When the box “use same setting as near to model” is checked, the same dense of mesh will be applied inside the entire computing domain.

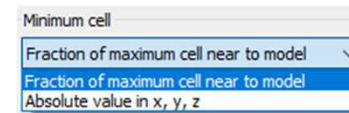


- Give the value as shown in blue box

• Setup the mesh : global properties 1



- Two options are available to define minimal cell

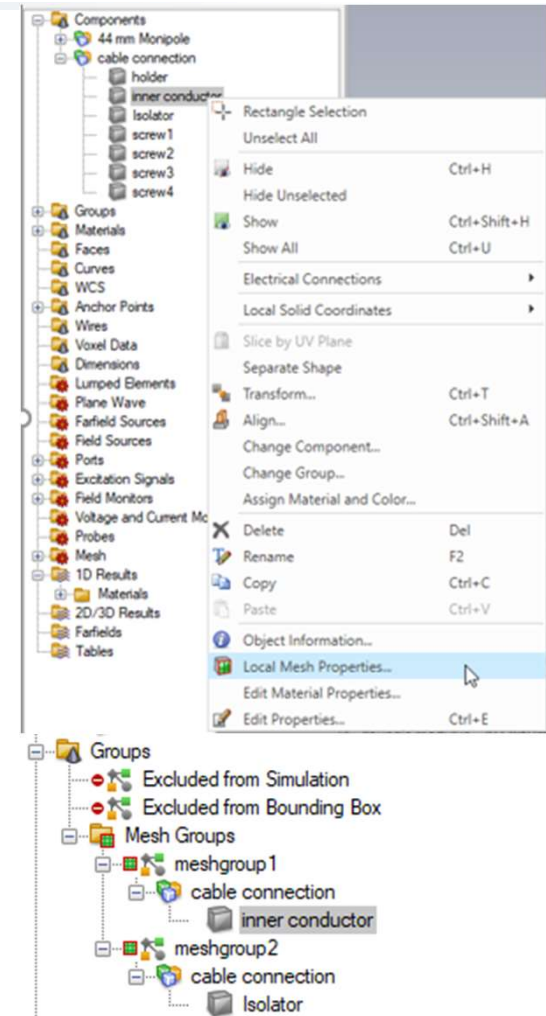


- Instead of using the absolute value, minimal cell is calculated by division of the maximal cell to the given value, larger the given value, smaller the minimum cell.
- Give the value as shown in blue box

- Statistics the overview of the total meshing is summarized, where the size of the smallest cell could be critical in terms of the simulation time in case the simulation is running using the time domain solver.

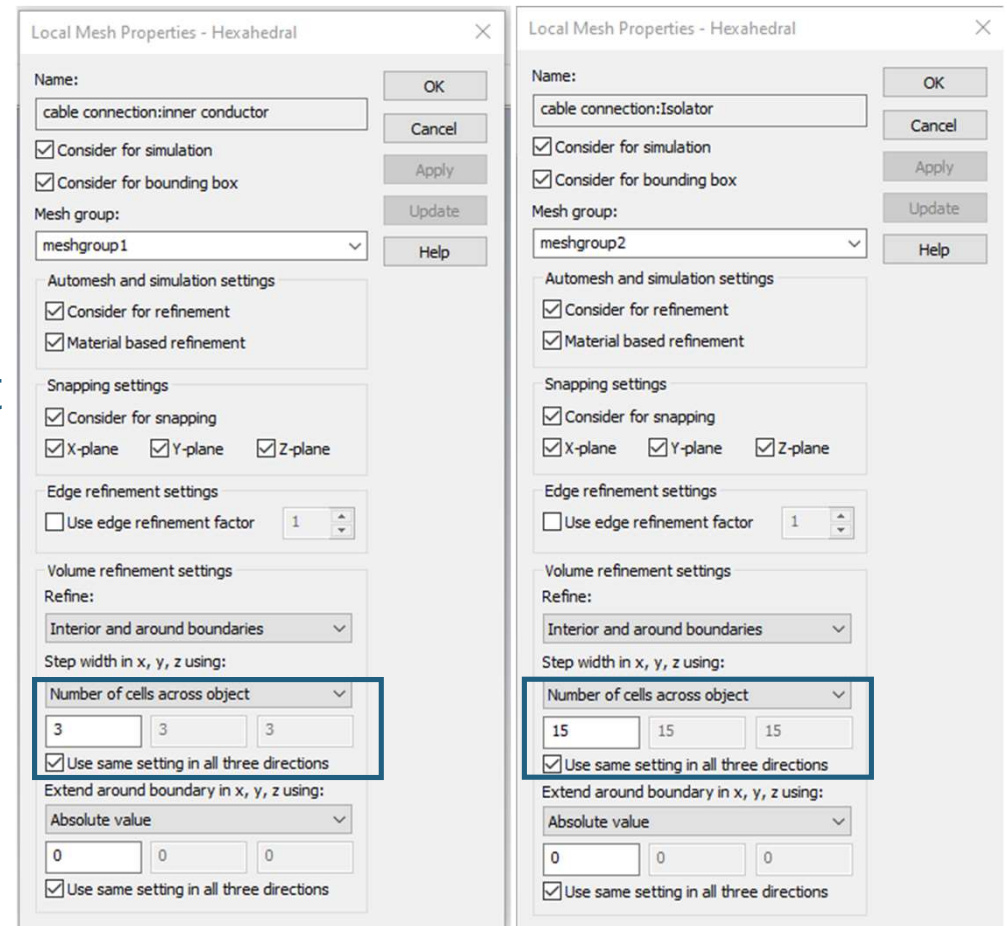
• Setup the mesh : local mesh properties 1

- Local mesh can refine the mesh for relatively small component in the model without increasing the total mesh cells
- By right click on the component the local meshing can be performed to a single component.
- A new folder will be created under “Mesh Groups”. In case the same setting need be applied to other components, simply add the corresponding component under same subfolder of the mesh group by “drag-and drop”.



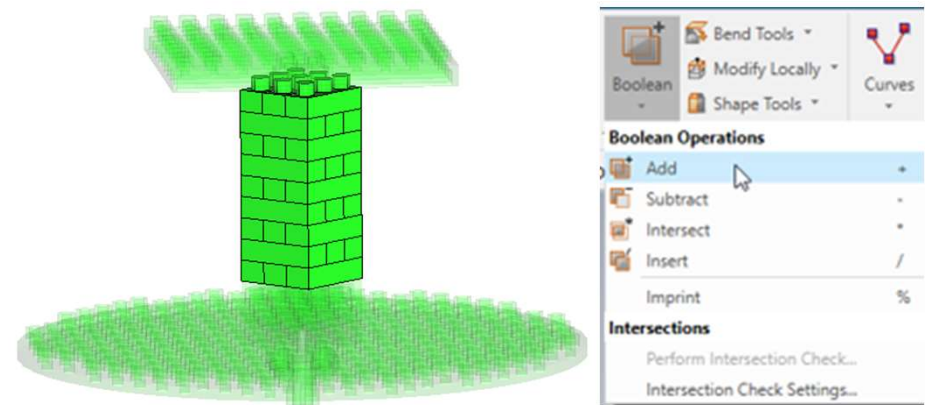
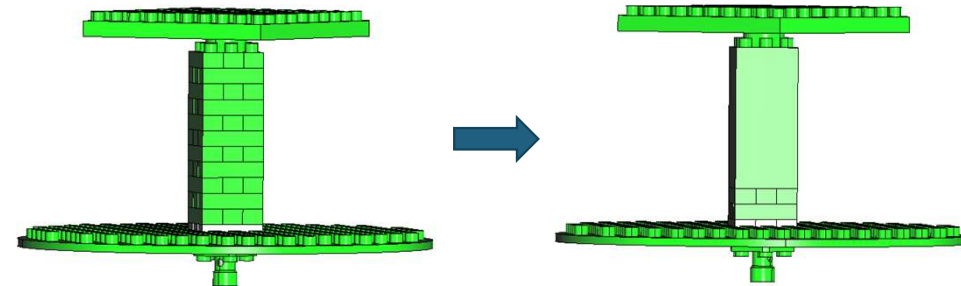
• Setup the mesh : local mesh properties 2

- For this model local meshing is carried out for inner conductor as well as the isolator.
- Right click on the “inner conductor” → select “Local Mesh Properties” → select “number of cells across object ” → give in 3 for all directions → “OK”
- Repeat the step to the “Isolator” giving the no. of cells across object by 15.



• Setup the mesh : modify the model

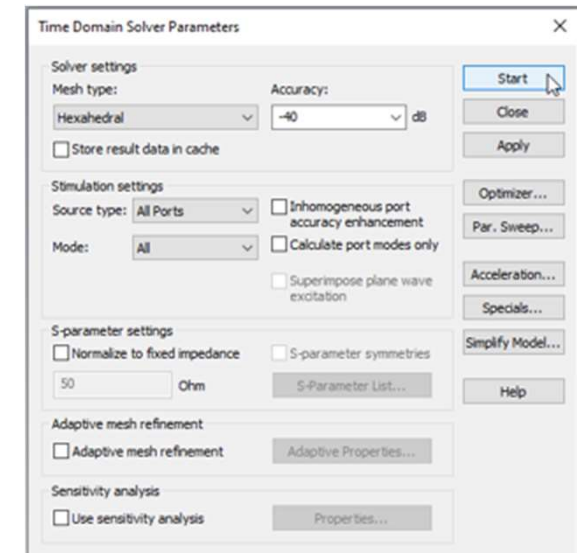
- To reduce the simulation time the model can be modified by merging the middle cells to one piece.
- In this case the inner details of the structure will not be considered during the simulation. Since the current flows through the surface of the model, this wouldn't affect the result.
- Select from the 4th layer until the layer under circular cell → "Modeling", "Boolean" → choose "Add"



• Start the simulation

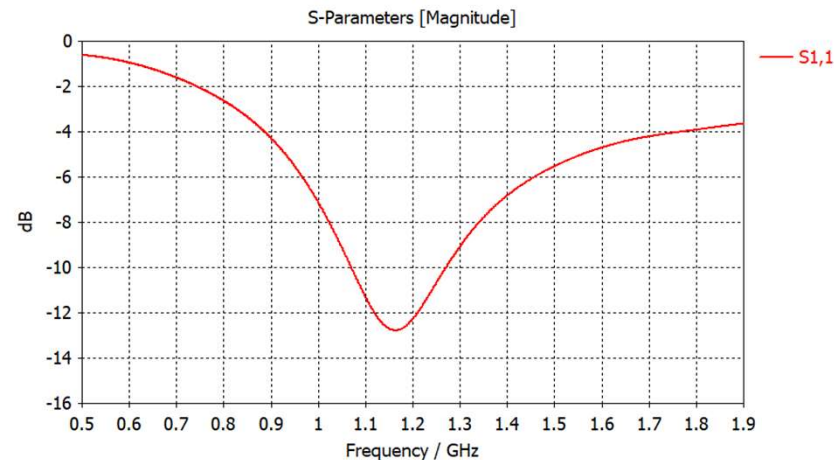
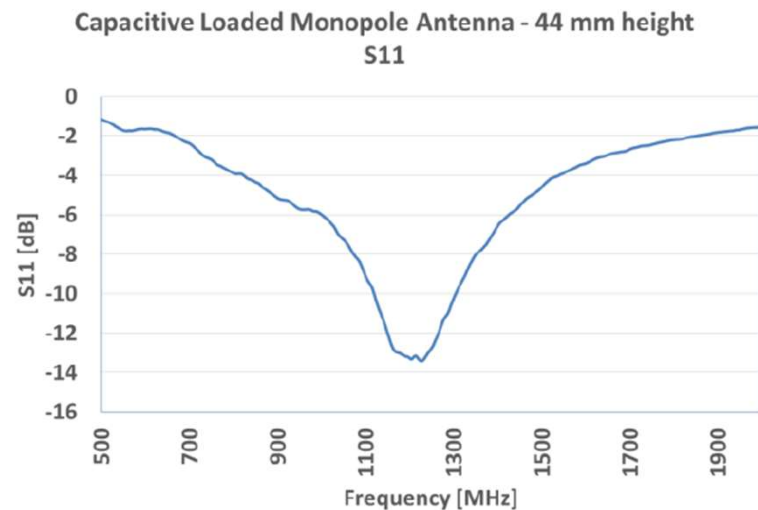
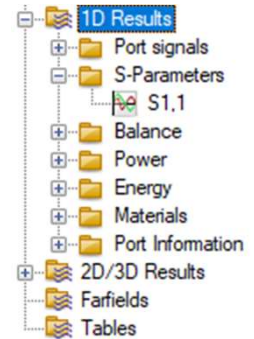


- The time domain solver is already selected by the template.
- Go to the ribbon “Simulation” → “Setup Solver” → “Accuracy”: -40 dB → “Start”
- “Accuracy” gives the stop criterium for the simulation, when the energy decay to -40 dB, the simulation stops with very accurate result.



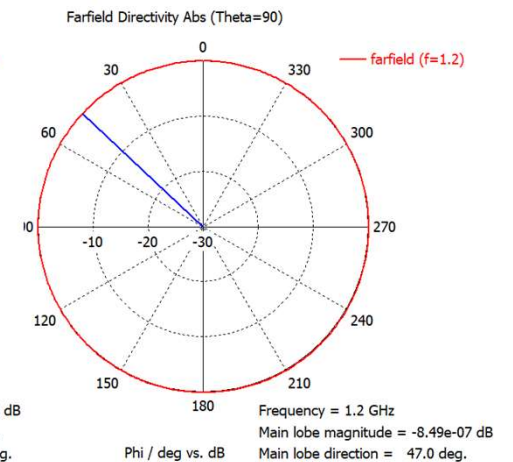
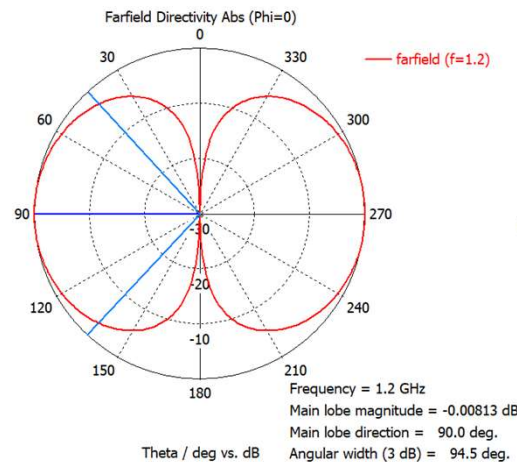
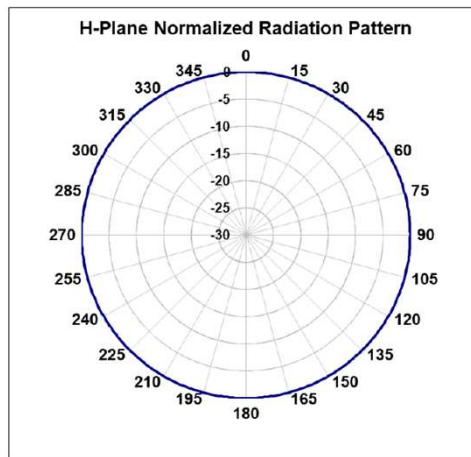
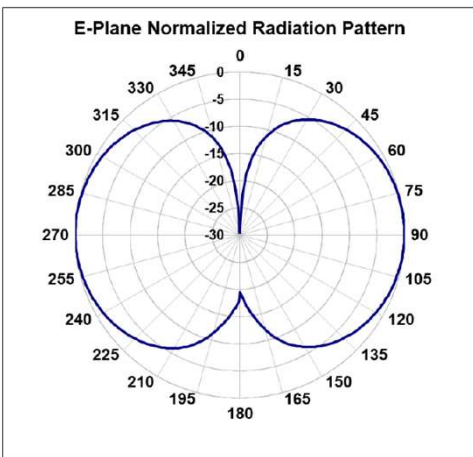
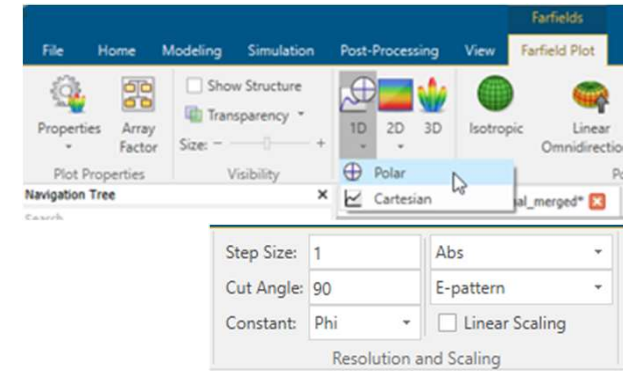
• Results(1D): S parameter

- Open “1D Results” in menu, the reflection s11 is saved in the subfolder “S-Parameters”
- Comparing with the measurement results (left below), s11 from the simulation (right below) shows good agreement.



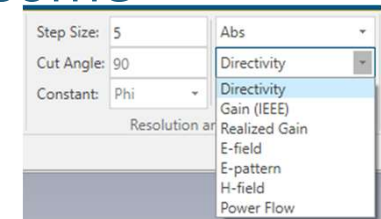
• Results(2D): Far field results

- The radiation pattern in different plane can be calculated and shown in 2D plot.
- Go to ribbon “Farfield Plot” → select “1D”, “Polar”
- The E-Pattern shown on different plane is controlled by selecting the corresponding cut angle on the menu bar



• Results(3D): Far field results

- The far field distribution can be demonstrated also in 3D. As shown below the absolute value of directivity is displayed as well as some other predefined evaluations can be directly selected and demonstrated.



- Another opportunity for postprocessing is using “Result Template” where lots of evaluation are available and achieved in several categories.

