INVENTSIM

Electromagnetic field simulator for RF, microwave and sub-THz passive component design

Patch antenna design

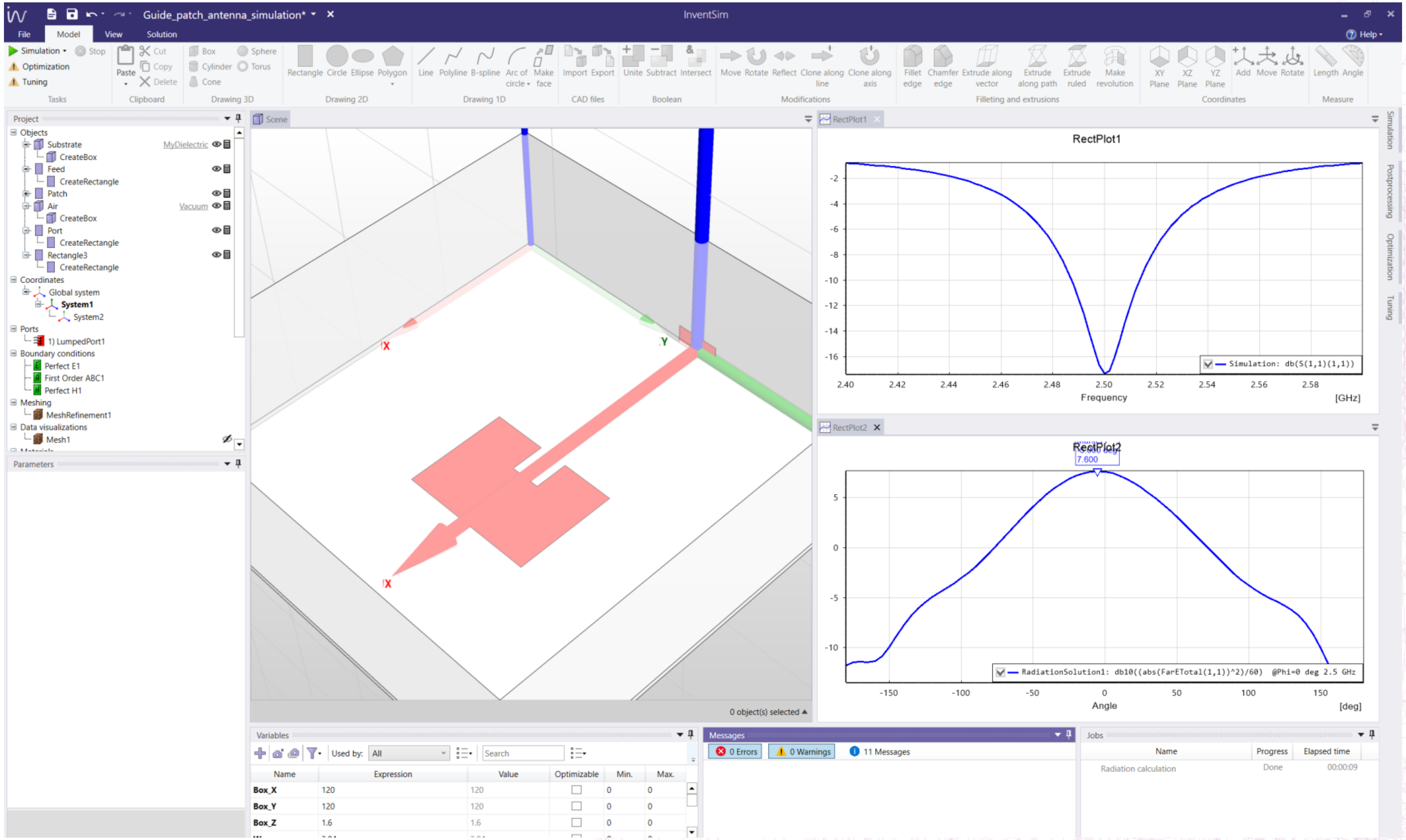
www.inventsim.com

Outline

- The presentation shows step-by-step guidelines to set up a simulation of basic microstrip patch antenna in **InventSim** simulator:
    - Drawing the structure with design parameterization
    - Setting boundary conditions
    - Adding a port
    - Setting meshing parameters
    - Setting frequency sweep
    - Running simulation
    - Adding postprocessing tasks to calculate radiation pattern and antenna parameters

EM Invent sp. z o.o., Trzy Lipy 3, 80-172 Gdansk, Poland. Company registration number 0000277738

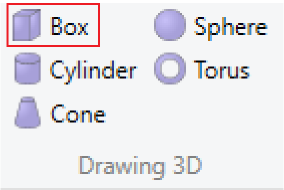
Final project



Goal: Draw a patch antenna and solve for S-parameters and gain.

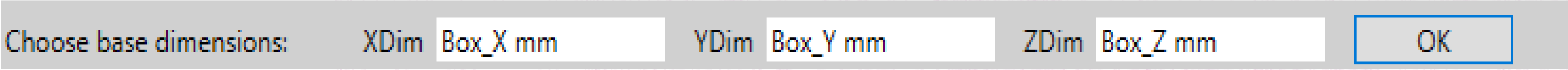
Drawing the substrate

- Run the InventSim simulator and create a new project.
- In the first step we draw a box to model substrate size of 140 mm x 120 mm x 1.6 mm. Select “Box” tool from the Drawing ribbon:



- Then choose the first corner as point (0,0,0) and accept it:

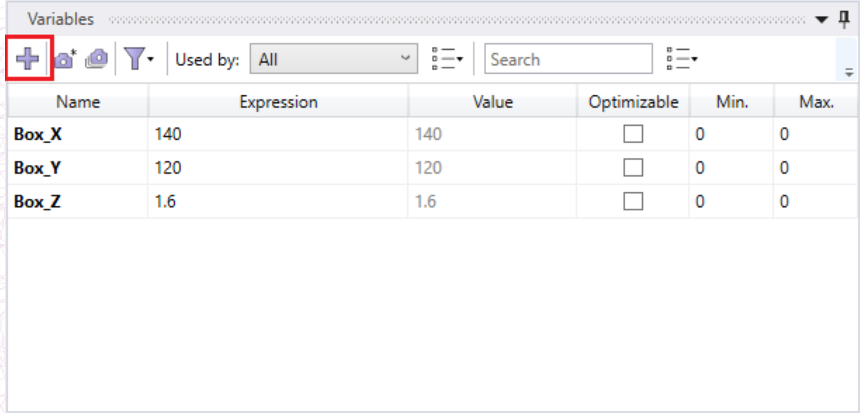
- And box size of 140 mm x 120 mm x 1.6mm. We can parameterize the design on the fly, entering box dimensions as “Box_X=120mm”, “Box_Y=120mm”, “Box_Z=1.6mm” and accept with Enter key:



Drawing the substrate

- When we use the expression “*Name*=*value*” at the input, the design variable *Name* is instantly added to the list of parameters.
    - You can either add variables upfront or define them on the fly!

Select “+” icon to add the variable to the project



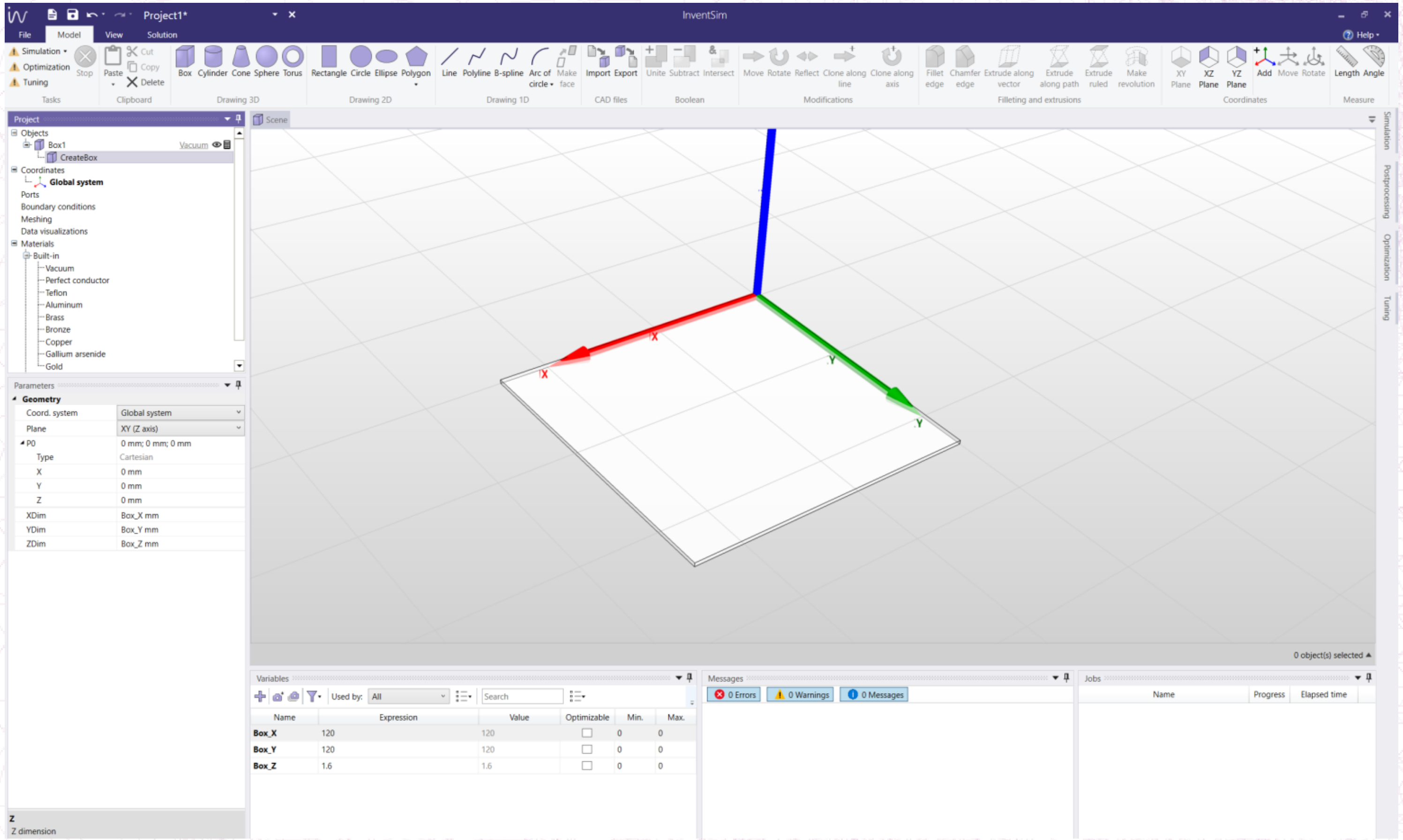
- In our case three variables are added to the variable list: Box_X, Box_Y and Box_Z

Drawing the substrate

- In the 3D view we will see the box representing the substrate, and the item is also added to the project tree

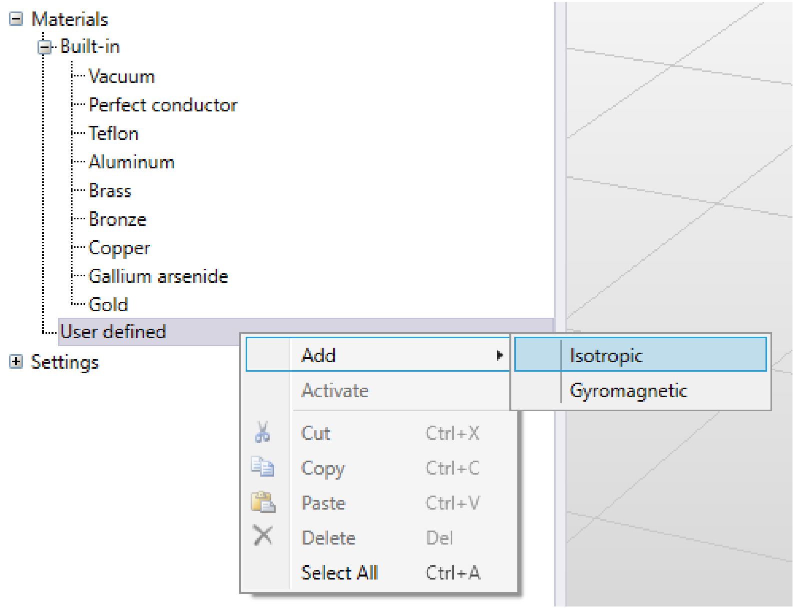
List of objects on project tree

Editable properties of selected item

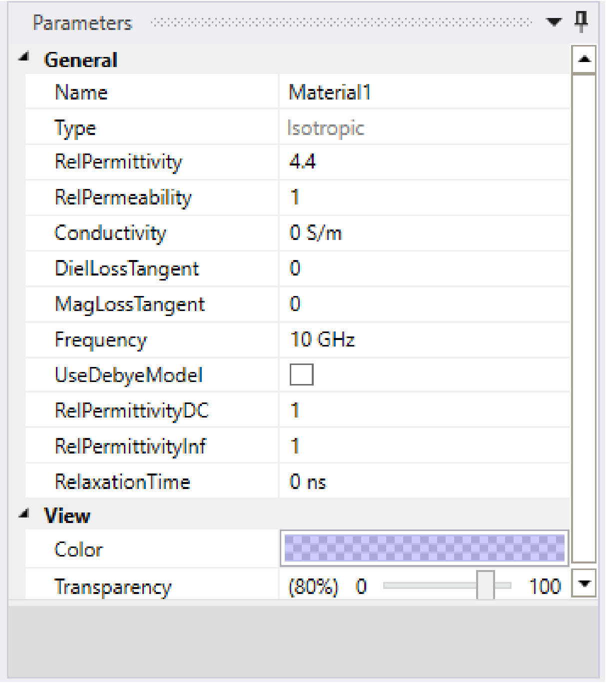


Drawing the substrate

- Next, we add to our project a new isotropic dielectric material with dielectric relative permittivity $\epsilon_r = 4.4$
- Go to the “Project tree → Materials → User defined” and invoke context menu with right mouse button click. Add new isotropic material.

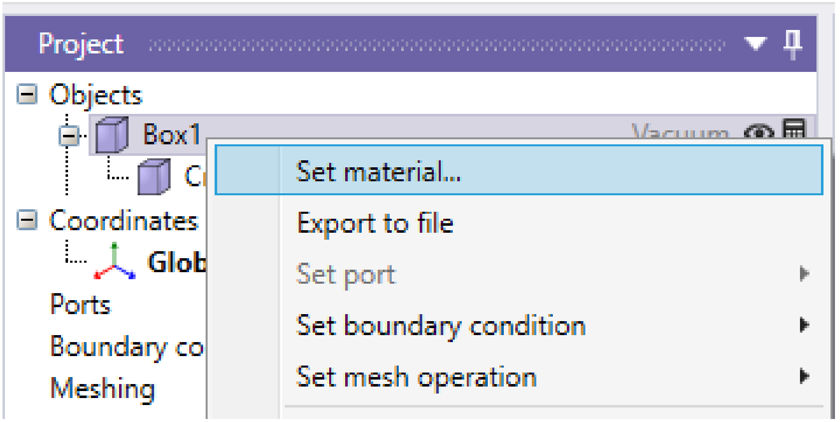


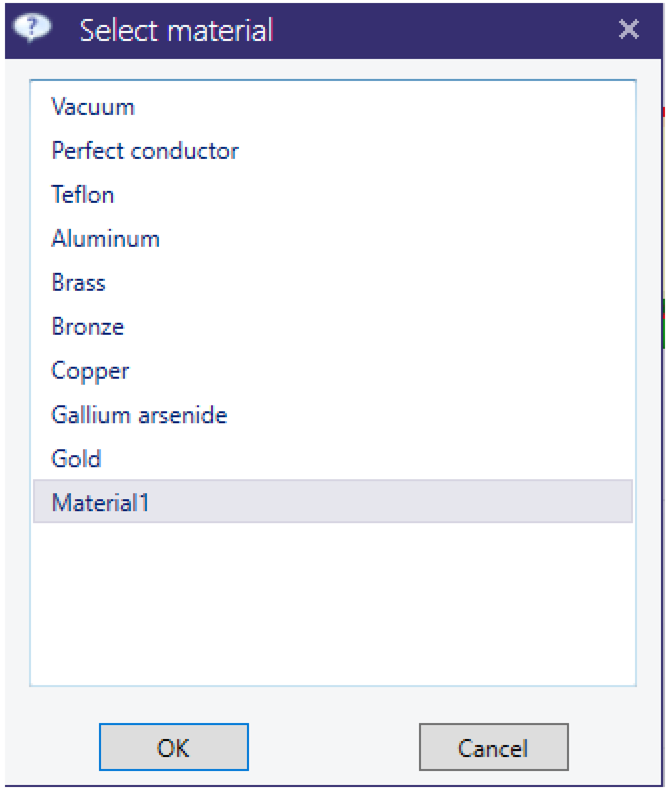
New material “Material 1” is added to the library. You can edit its properties, including name, value of permittivity or loss tangent



Drawing the substrate

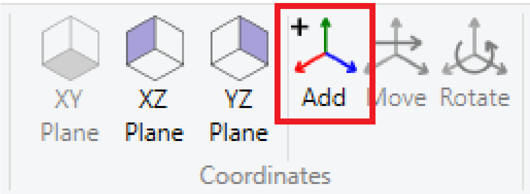
- To assign the material to the box, select the box from the objects list, click right mouse button, select „Set material…” from the context menu and select „Material1” from the list of available materials:



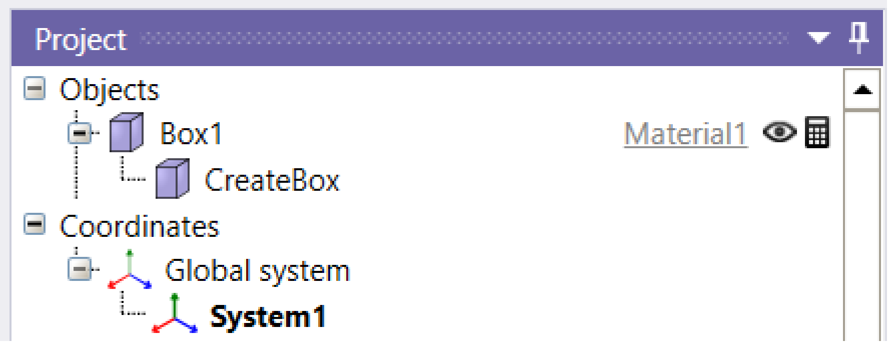


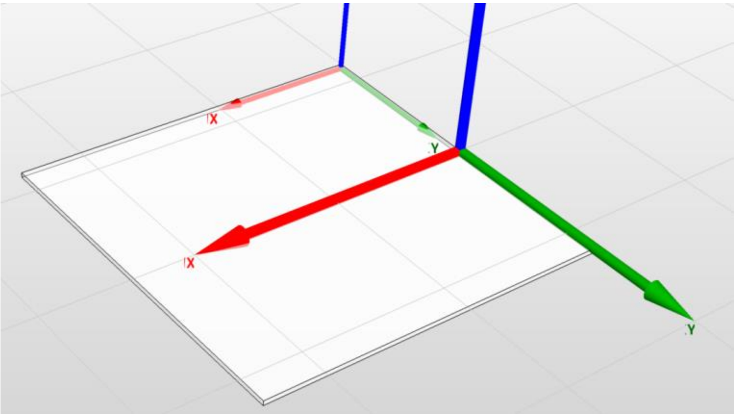
Drawing the feed line

- To draw the 2D object on the top of the substrate we need to add a new local coordinate system
- Select the „Add coordinate system” icon and enter the location:



- The new local coordinate system “System 1” becomes active.

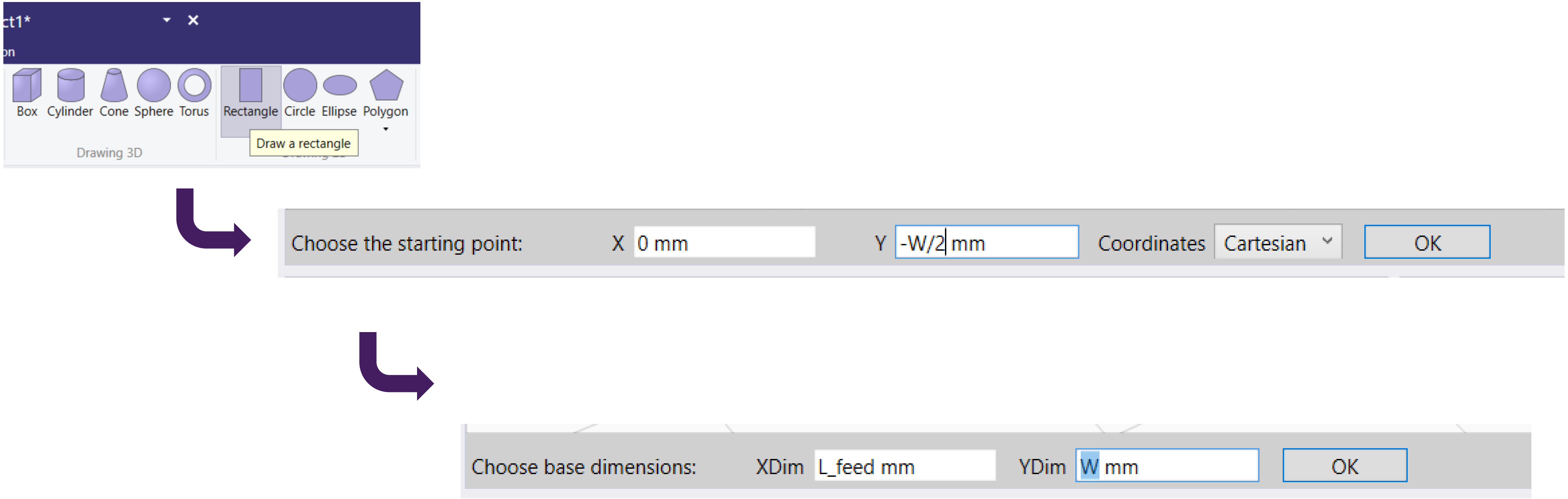




You can easily switch between coordinate systems defined in the project

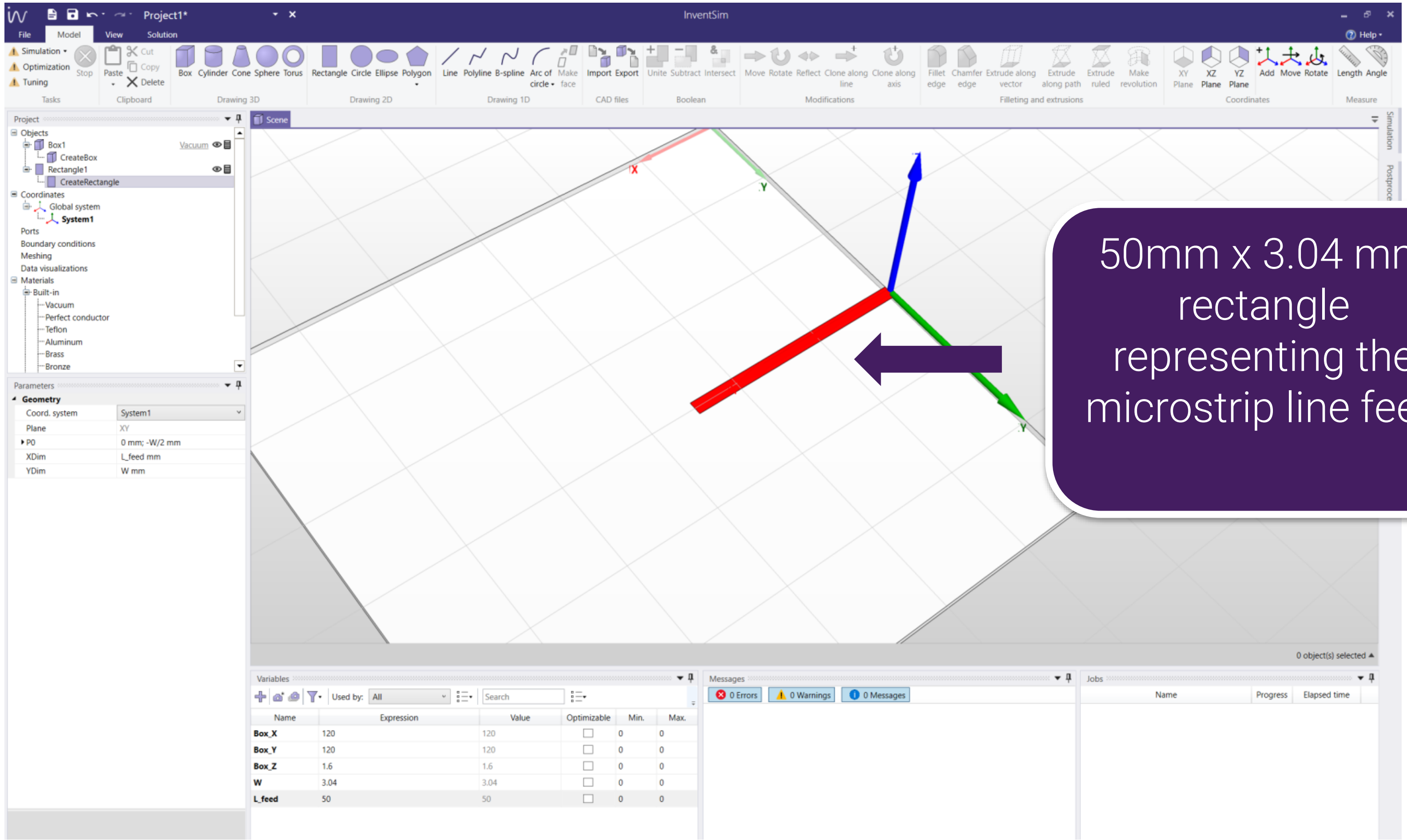
Drawing the feed line

- Now we create two new design variables (L_feed=50, W=3.04) and draw microstrip line feed as a 2D rectangle object on XY plane starting from point (X=0; Y=-W/2) with dimensions XDim=L_feed, YDim=W.



Drawing the feed line

- We should see the rectangle on the top of substrate, as shown here:

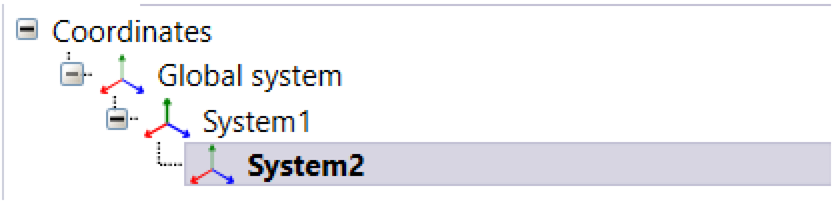


50mm x 3.04 mm rectangle representing the microstrip line feed

Width W=3.04mm corresponds to 50 Ohm microstrip line on this substrate

Drawing the patch

- To draw the radiating element (patch) we add a new coordinate system at (X=L_feed, Y=0, Z=0),



Parameters	
General	
Name	System2
Geometry	
Relative to	System1
Plane	XY
Origin	L_feed mm; 0 mm; 0 mm
XY_Angle	0 deg
XZ_Angle	0 deg
YZ_Angle	0 deg

- Then we add two new variables (W2=37.23, L2=28.05), defining the size of the patch

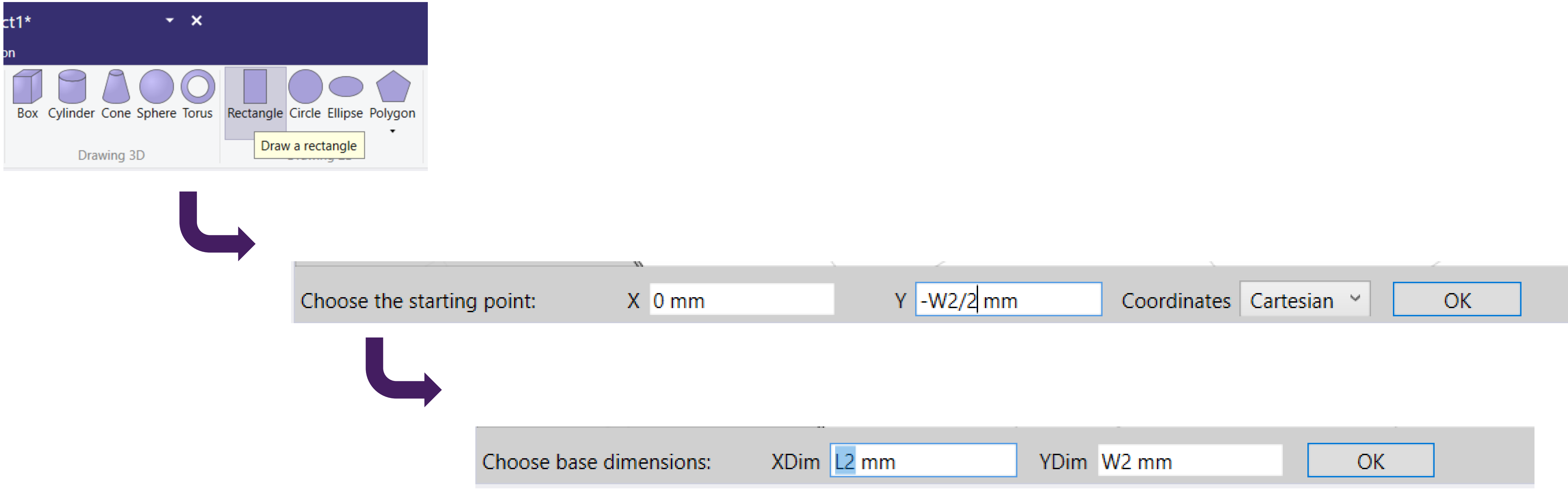
Variables — Used by: All — Search

Name	Expression	Value	Optimizable	Min.	Max.
Box_Y	120	120	☐	0	0
Box_Z	1.6	1.6	☐	0	0
W	3.04	3.04	☐	0	0
L_feed	50	50	☐	0	0
W2	37.23	37.23	☐	0	0
L2	28.05	28.05	☐	0	0

Note: One can define the variables on the fly or skip the design parametrization entering the numeric values when drawing objects.

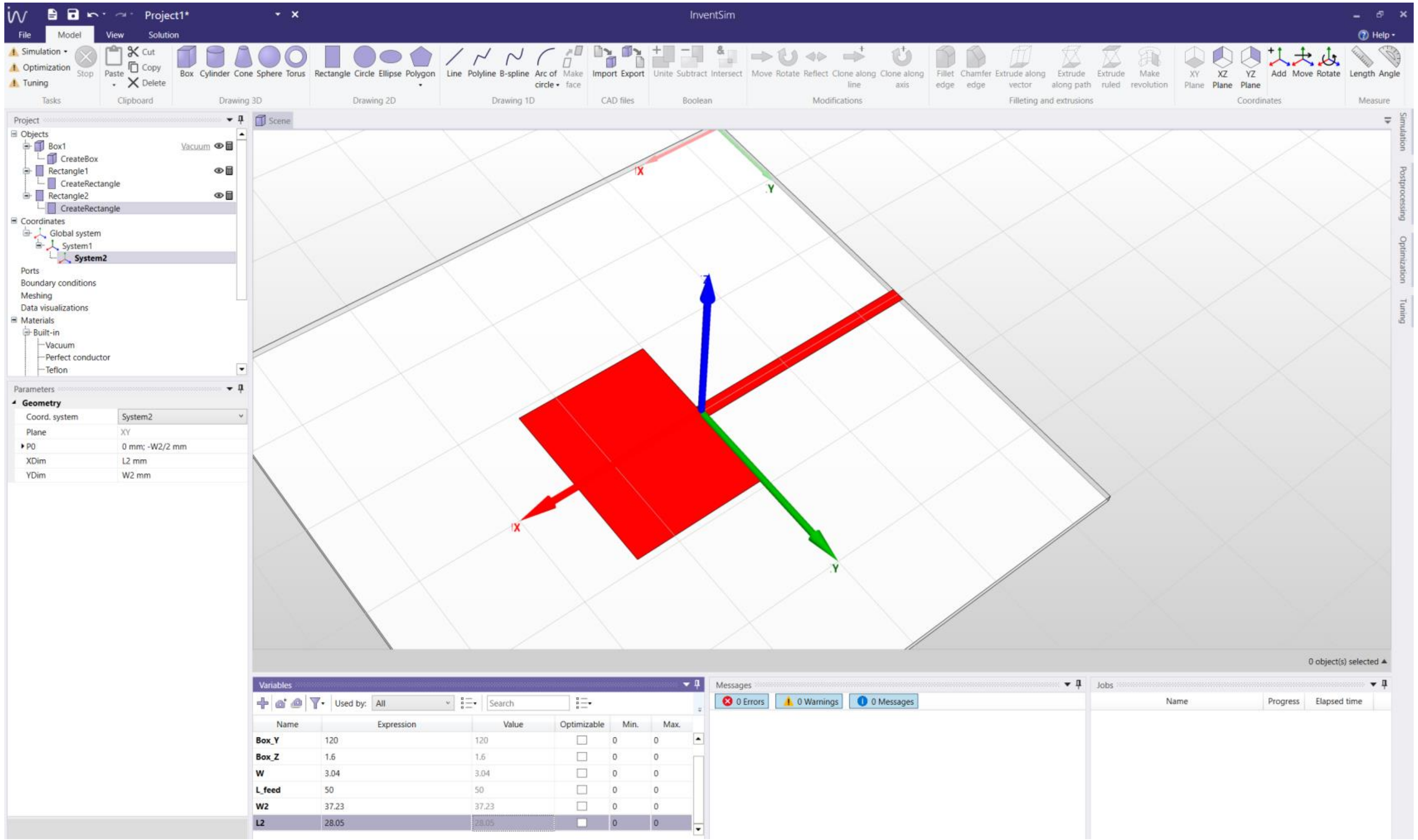
Drawing the patch

- Now we can draw a second rectangle shape starting from point X=0; Y=-W2/2 with dimensions XDim=L2, YDim=W2



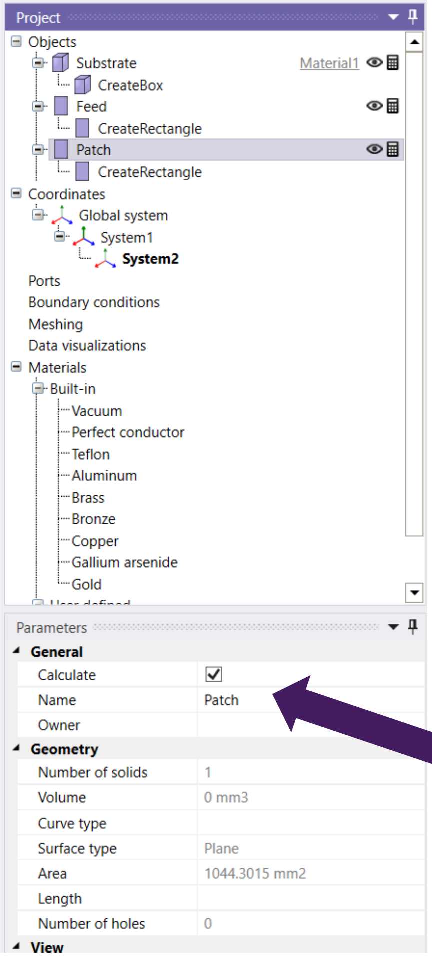
Drawing the patch

- We should see the patch in the 3D view window:



Drawing the patch

- **TIP:** It is good practice to rename the objects to give them clear, self explaining labels



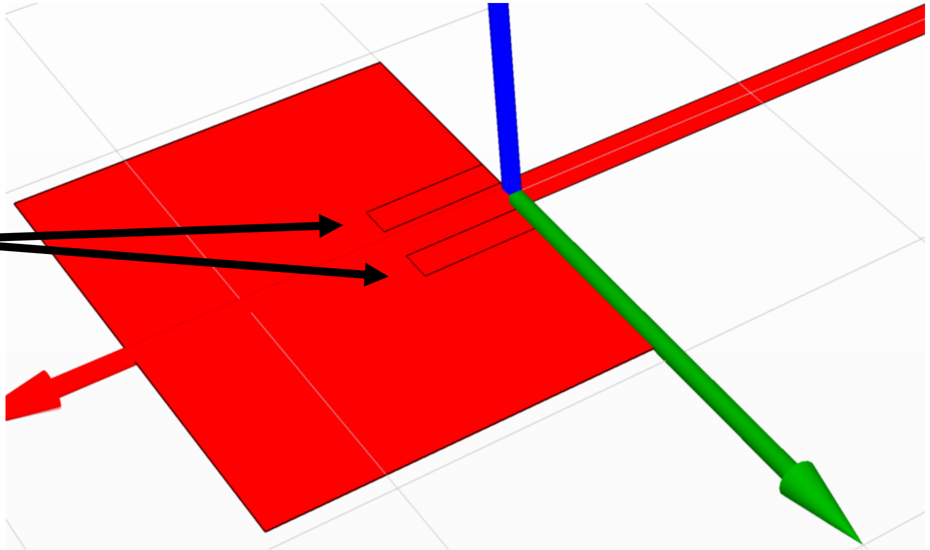
Rename the drawn objects to “Substrate”, “Feed” and “Patch”, respectively.

Name of the object can be changed in object properties list

Drawing the patch

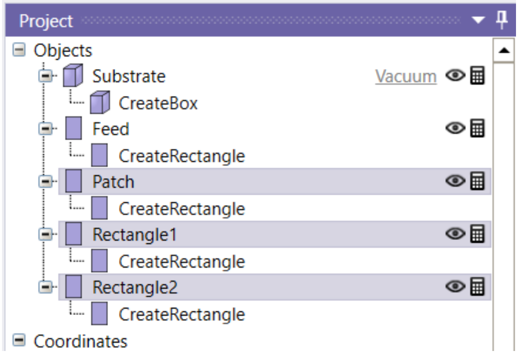
- A similar procedure is applied to draw two rectangular objects that will be subtracted from the patch to create matching slots
  - Add two new variables (W3=2.5, L3=9.075) and draw a rectangle. Starting from point X=0; Y=W/2 and dimensions XDim=L3, YDim=W3.
  - Draw second rectangle on the opposite side: XDim=L3, Ydim=-W3; Starting Point X=0, Y=-W/2.

Newly added “Rectange1” and “Rectangle2”

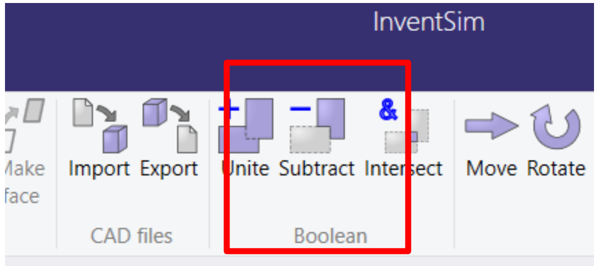


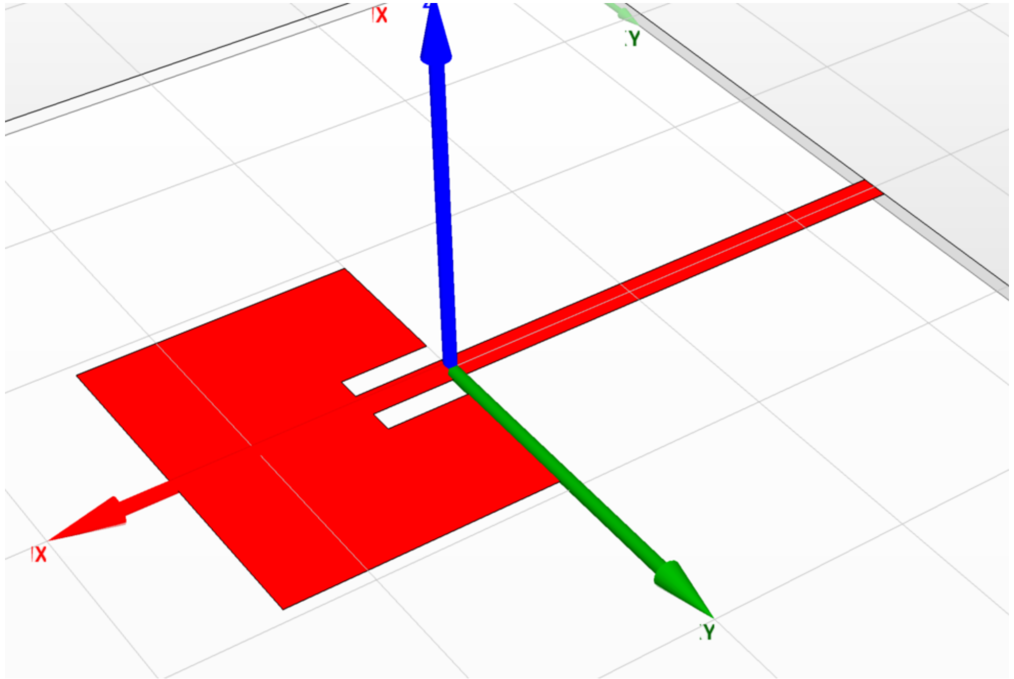
Drawing the patch

- Select objects to subtract. The order of your selection is important. In the project tree and select “Patch,” hold SHIFT key and **then select two newly drawn objects** “Rectangle1” and “Rectangle2”.



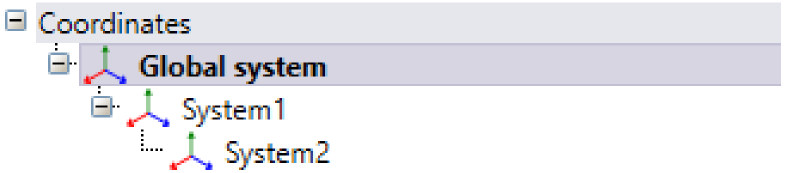
- Execute “Subtract” operation



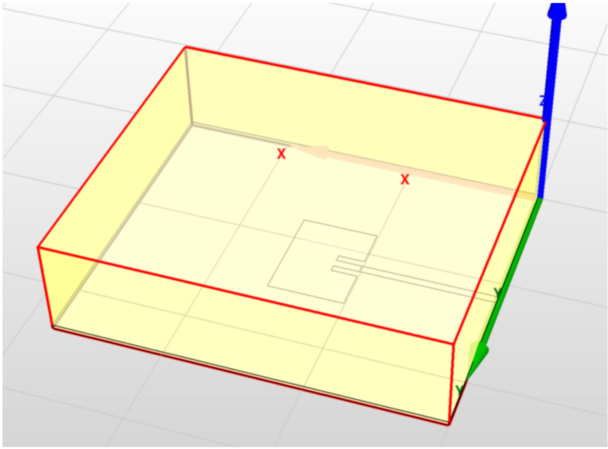


Drawing the air above the antenna

- Activate “Global Systems” as an active coordinate system
    - double click or right click and “Activate”



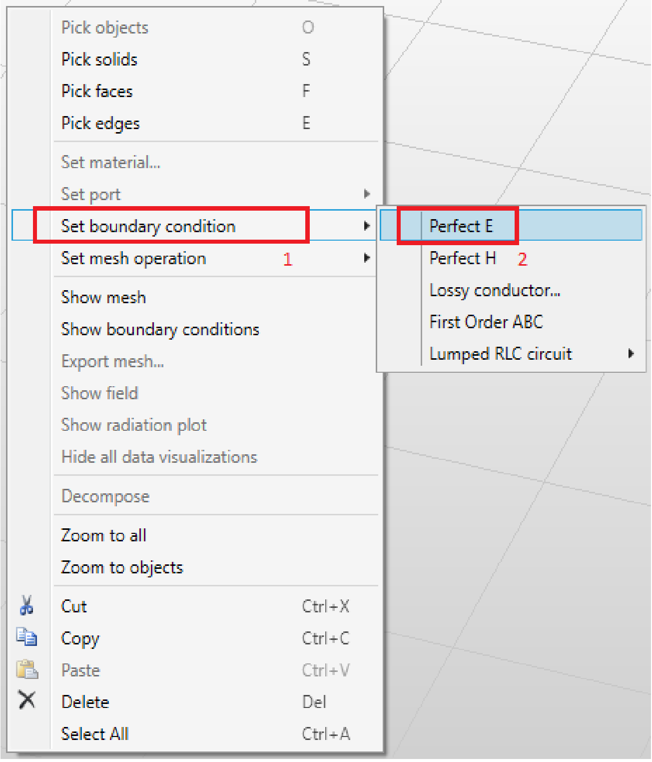
- Add new variable VacuumH = 38
- Draw a box. Starting point set as X=0, Y=0, Z=0 and dimensions as XDim=Box_X, YDim= Box_Y, ZDim= VaccumH



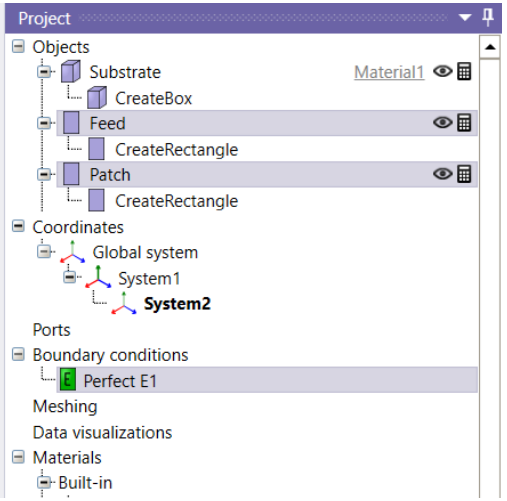
- Rename the object to “Air”

Setting the PEC boundary condition (perfect conductor)

- Select objects "Patch" and "Feed"
- Click the right mouse button on Scene and select „Set boundary condition"→"Perfect E"



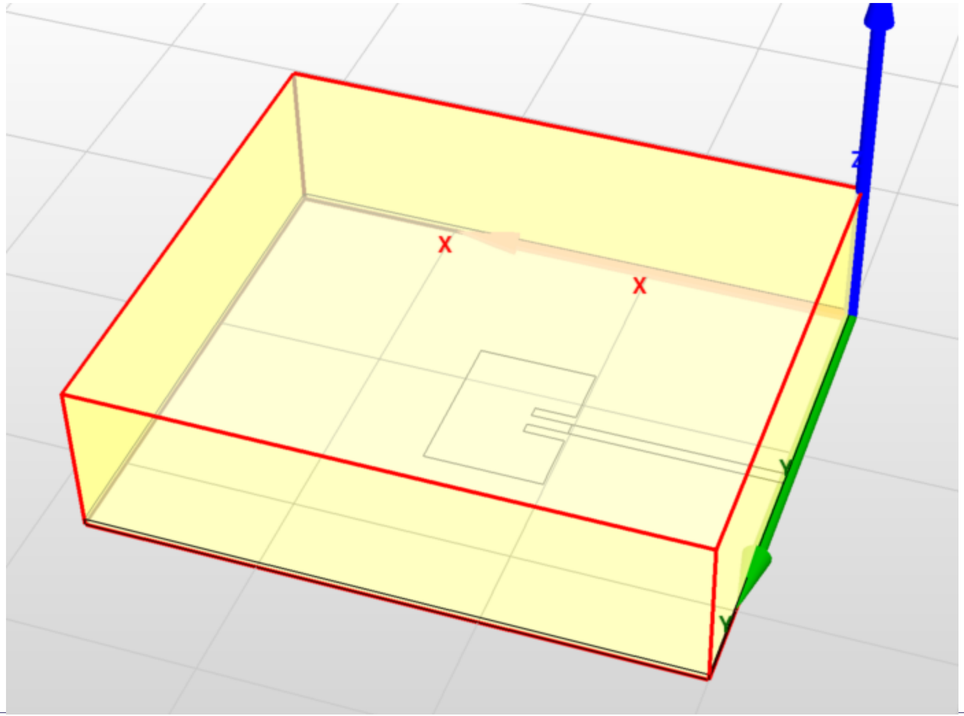
New boundary condition is listed in the project tree



Alternatively, we can enforce lossy conductor surface (like copper)

Setting the ABC boundary condition (radiation)

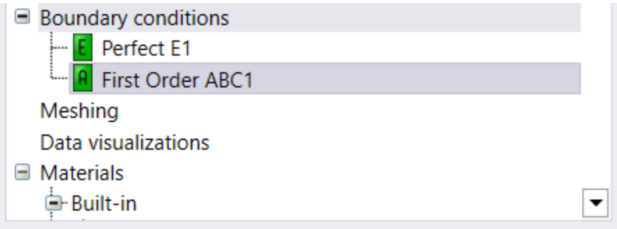
- **The default boundary condition on the outer surfaces of the domain is PEC** (perfect conductor)
  - If we draw a vacuum box, its walls are treated as PEC
  - **We must change this condition for antenna simulation**



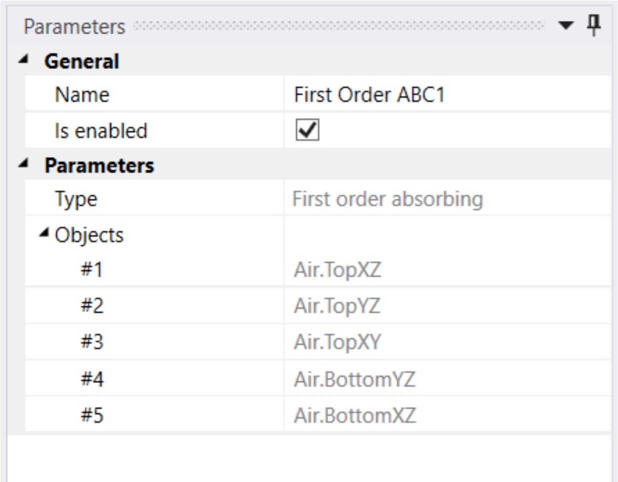
Setting the ABC boundary condition (radiation)

- **Absorbing boundary condition (ABC) simulates open space.**

- Click the right mouse button on Scene and select „Pick faces"
- Select the top surface and 4 sides of the "Air" object and define an absorbing boundary condition on them (Context menu → „Set boundary conditions" → „First order ABC").
- New boundary condition as added to the list:



Selecting a boundary condition displays the affected surfaces

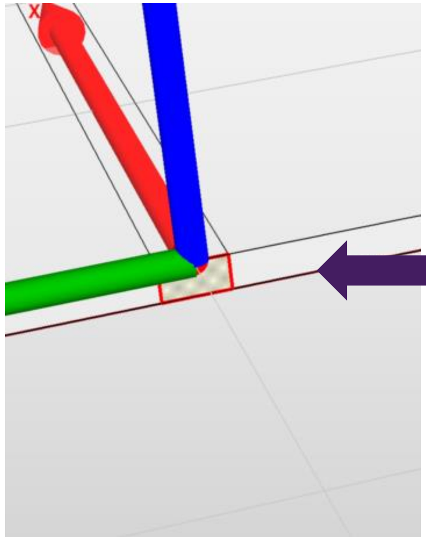


Adding a lumped port

- Set “System1” as active coordinate system.
- Select YZ Plane in Coordinates panel
  - → We will be drawing on the YZ plane

- Draw a rectangle representing port. Starting Point: Y=-W/2, Z=0; YDim = W, Zdim= -Box_Z

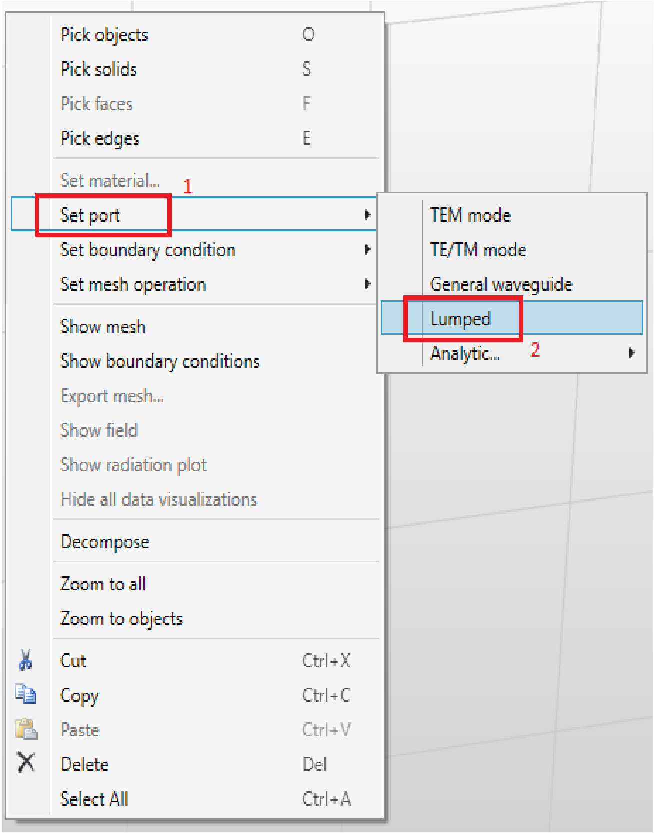
- Rename the object to “Port”



Result: a rectangle placed between the feed and the ground plane

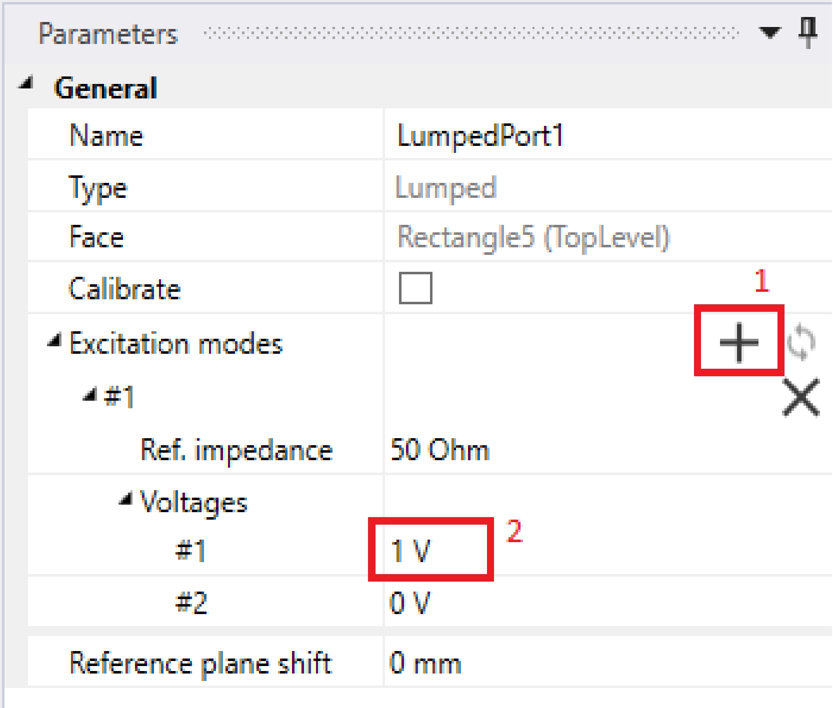
Adding a lumped port

- Select an object representing port in the project tree and click right mouse button on Scene. Click “Set port” and select “Lumped”



Adding a lumped port

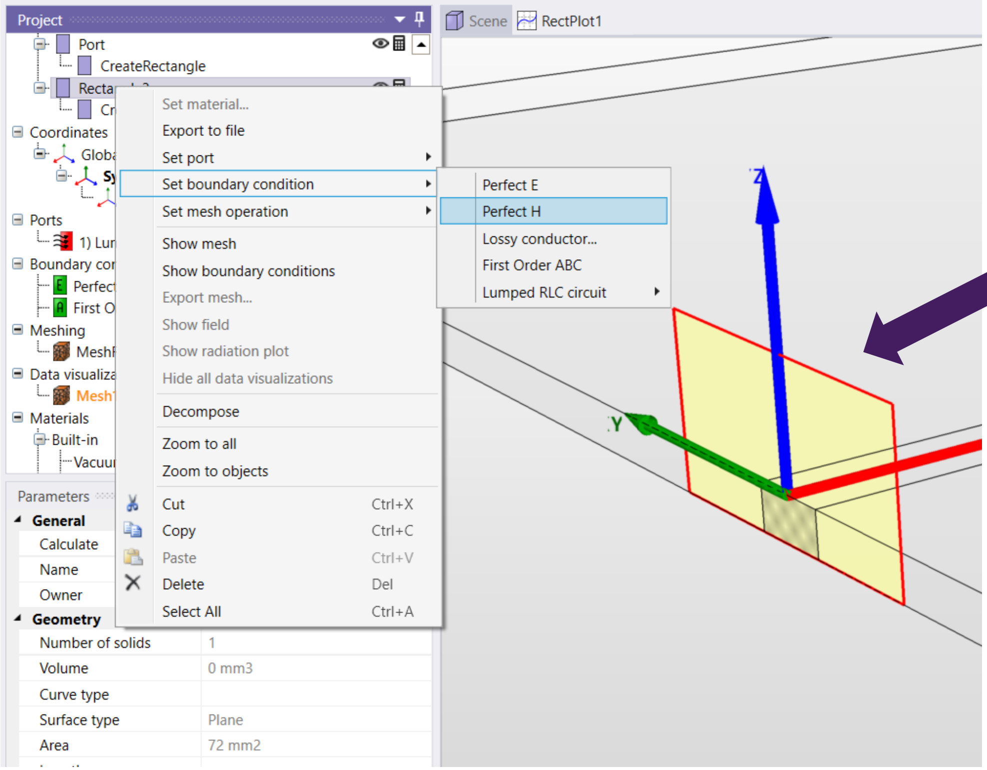
- On the left in the port "Parameters" window we must add mode and assign voltage for each conductor:



Lumped port acts as a voltage gap, it means that the port imposes a specific voltage difference (gap) between two conductors in the simulation

Isolating port from the ABC

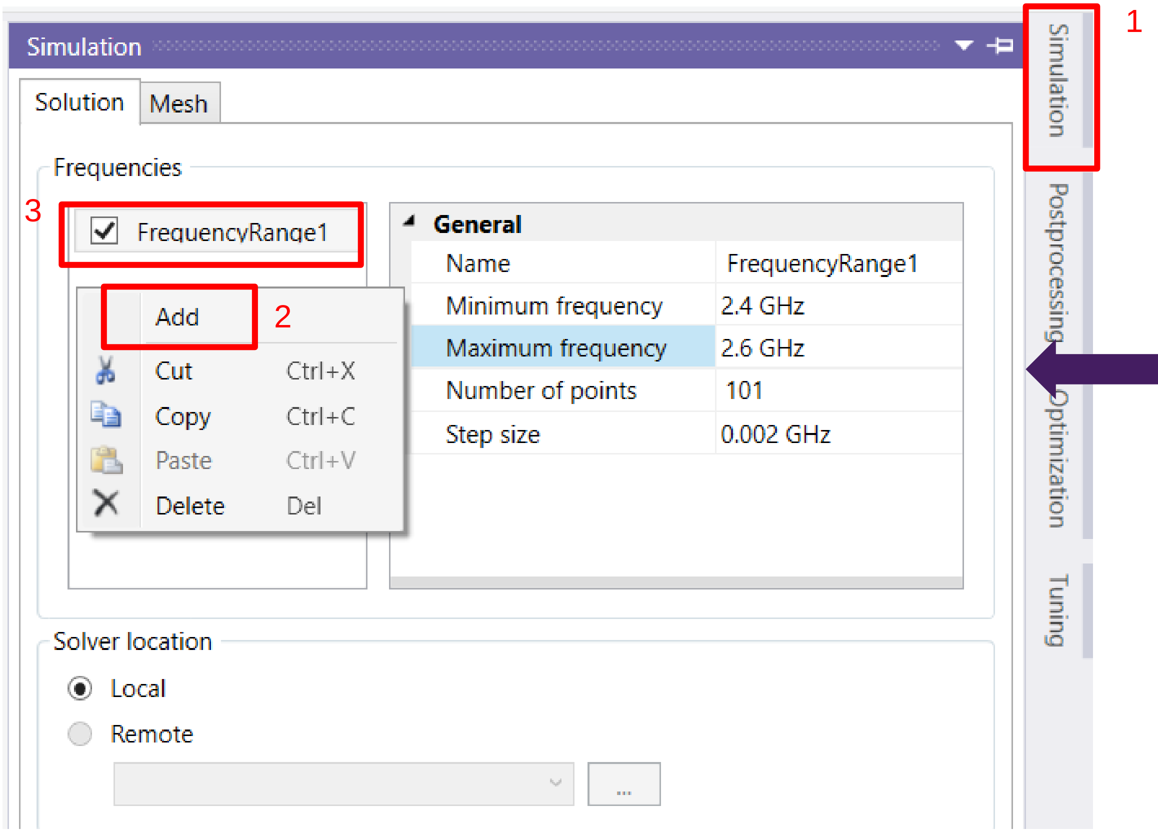
- **The surface of lumped port collide with the radiation boundary condition (ABC)!**
- To isolate the port, we draw an additional rectangle surrounding the port, and set the ‘Perfect H’ boundary condition on it (Perfect Magnetic Wall, PMW)



A rectangle size of 12mm x 6mm is added in the port plane and assigned the ‘Perfect H’ boundary

Setting a frequency sweep

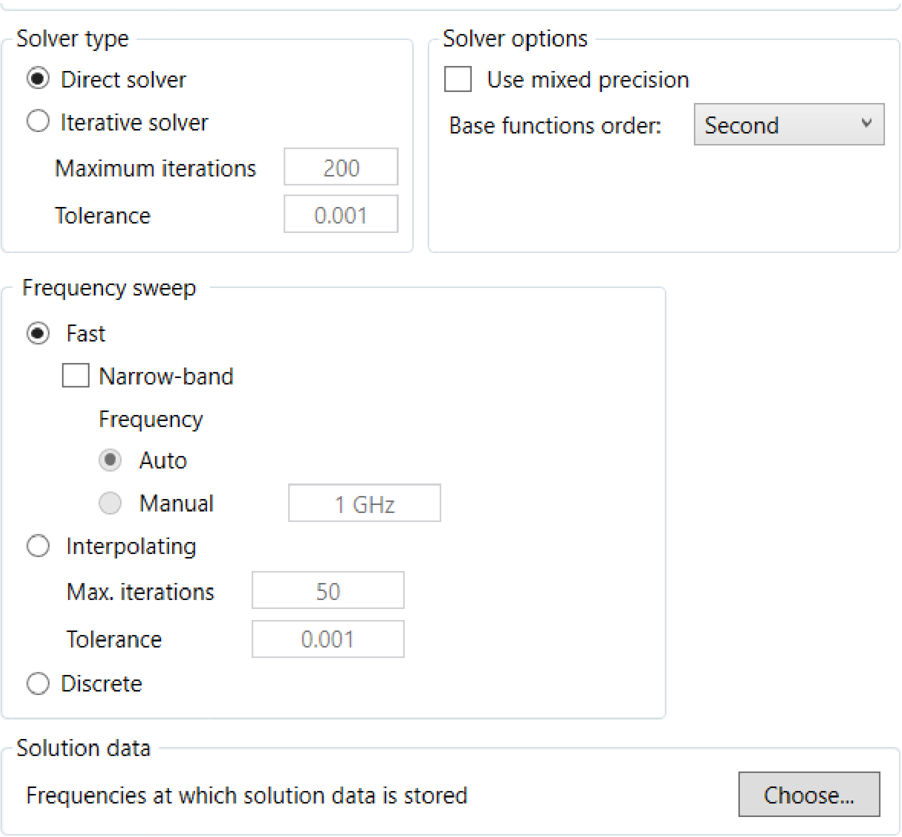
- Open "Simulation" panel on the right (1) and right click on the "Frequencies" part (2). Then click Add (3) to add a new frequency plan



We add a frequency sweep from 2.4GHz to 2.6GHz and 101 frequency points

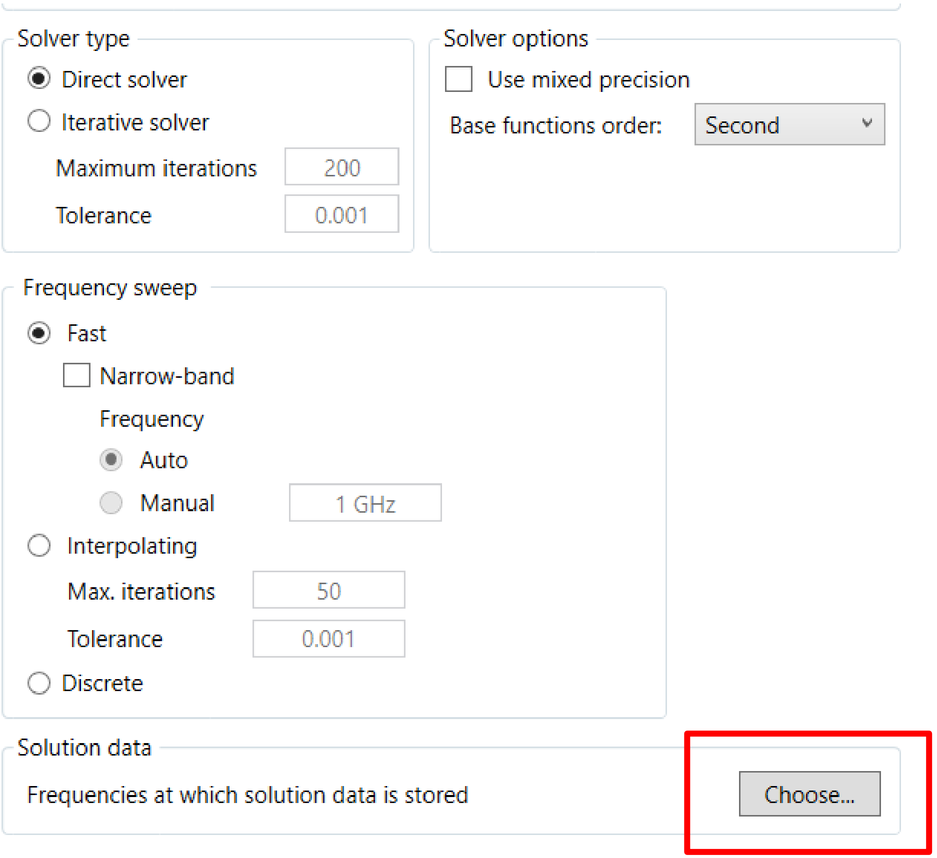
Setting a frequency sweep

- The rest of frequency sweep options remain unchanged, as we will use the Direct solver and Fast Frequency Sweep



Setting a frequency sweep

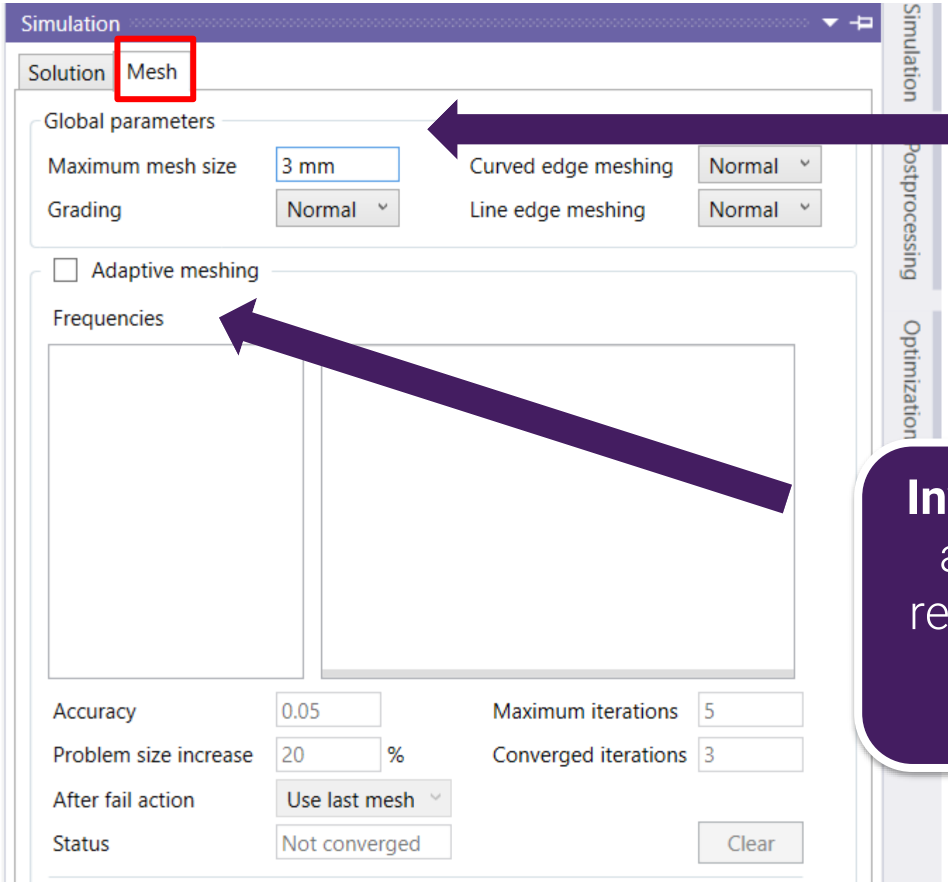
- We can select one or more of frequency points to save the fields for further postprocessing (for example radiation pattern calculation)



Click 'Choose' button to add a frequency 2.5GHz to the list of saved field solutions

Meshing options

- The global meshing options can be found in the “Mesh” tab of “Simulation” panel

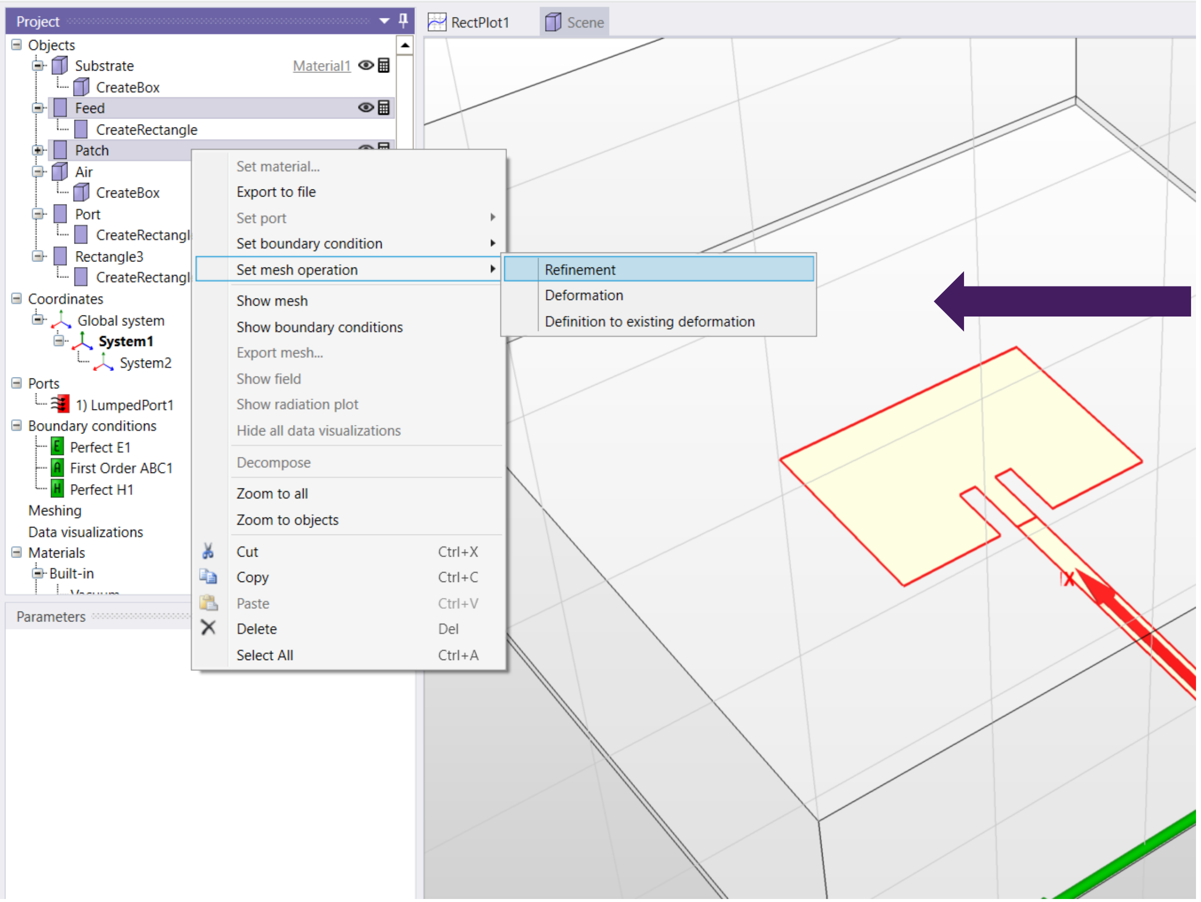


- Here we define the maximum mesh size → enter 3mm

InventSim offers an option to activate adaptive meshing, which iteratively refines the mesh while monitoring the convergence of the results at a selected frequency

Meshing options

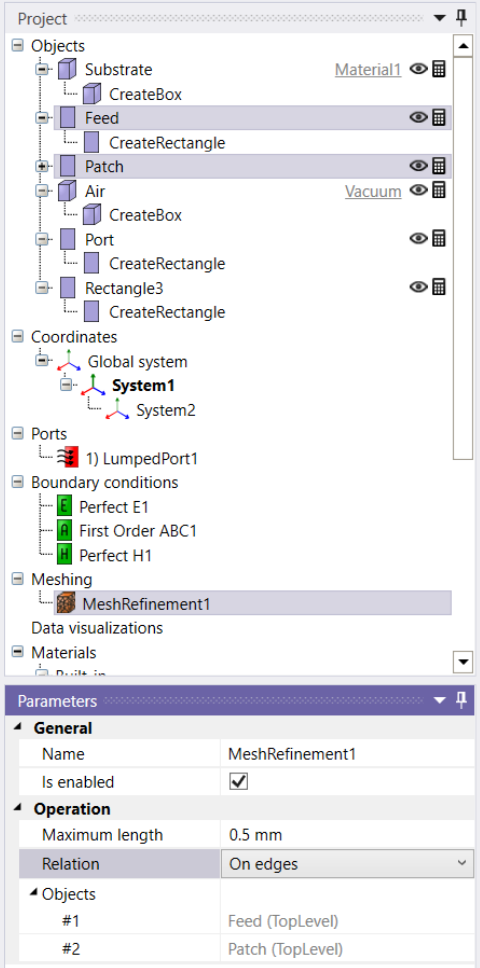
- To improve the accuracy of the results, one can define a local mesh refinement in the critical areas, where the high field variation is expected.

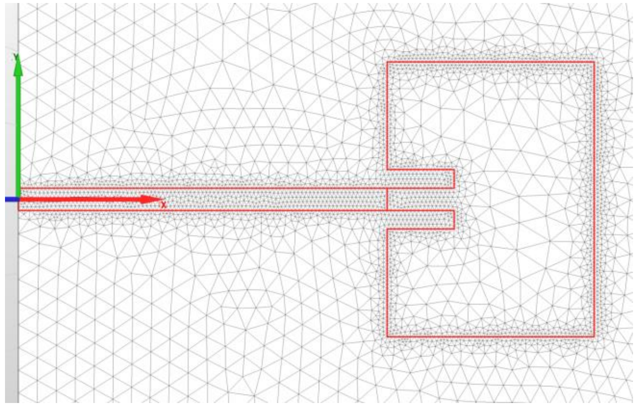


To additionally refine the mesh on the patch and feeding microstrip line, select both objects, invoke the context menu, and select **'Set mesh operation → Refinement'**

Meshing options

- A new item “MeshRefinement1” is added to the project tree in ‘Meshing’ section.





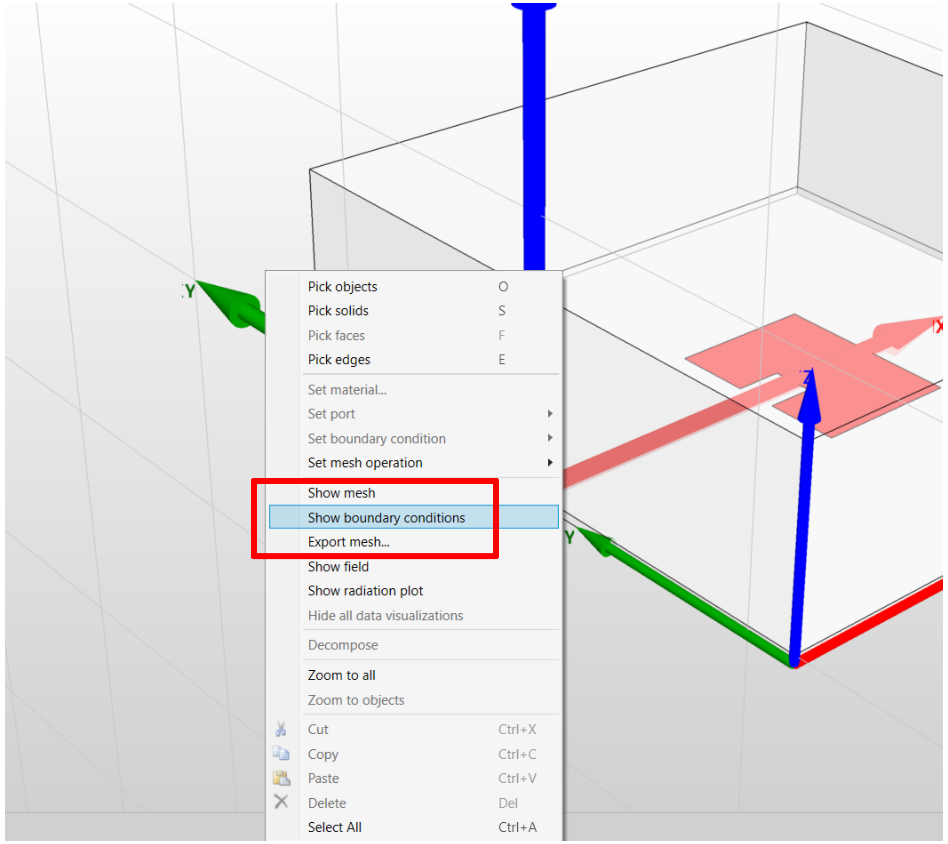
Resulting non-uniform mesh

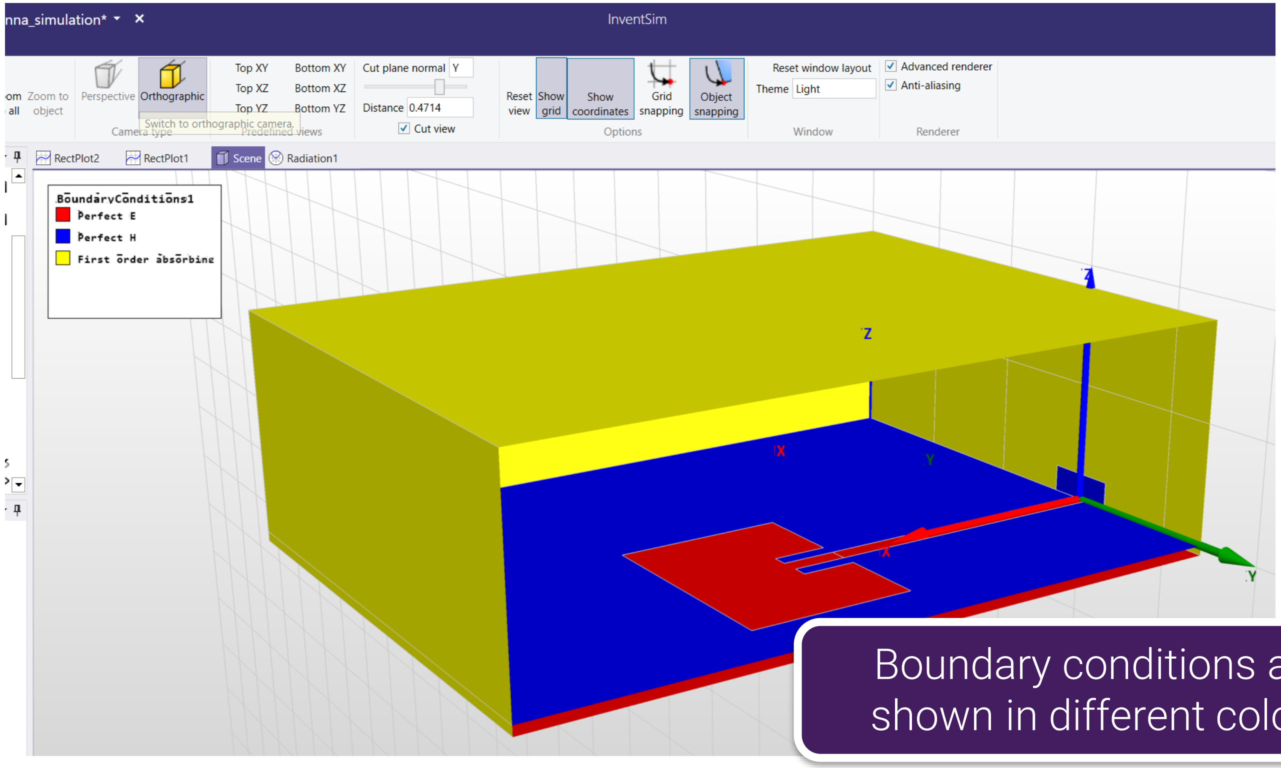
Here we set the mesh refinement to 0.5mm on the edges of the objects

Verifying boundary conditions

- Before we start simulation, it is recommended to verify the boundary conditions defined in the project:

Use 'Cut View' option to look inside

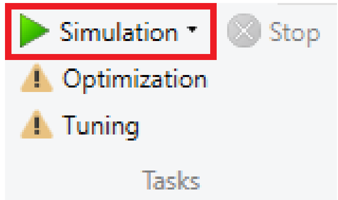




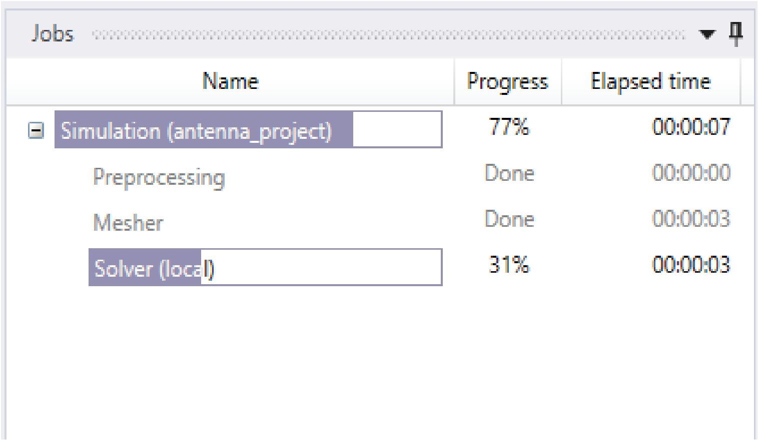
Boundary conditions are shown in different colors

First simulation

- **The project is ready to be simulated!**
- To start the simulation, we click the “Simulation” button.

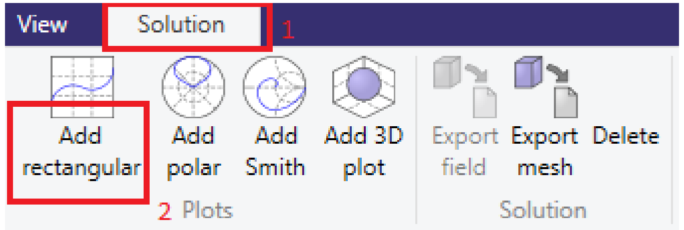


- The simulation progress is displayed in the “Jobs” window on the bottom

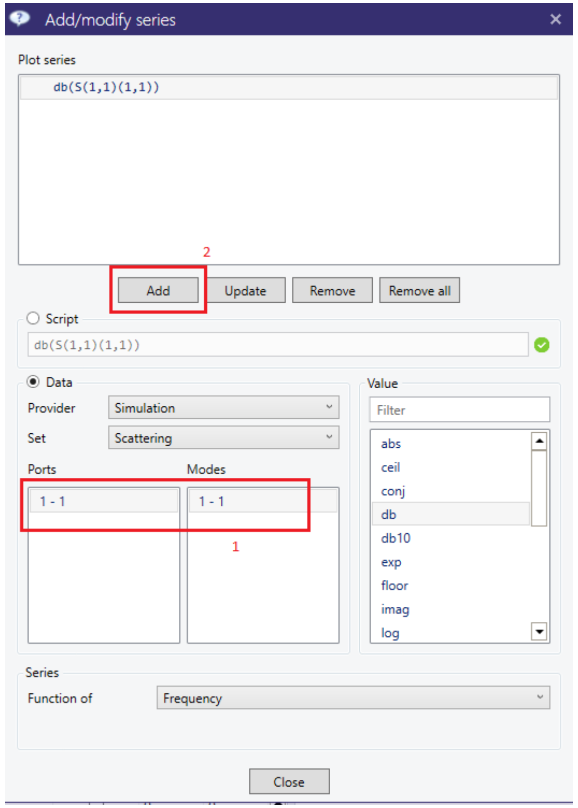


First simulation

- When the simulation finishes, we add a plot to the display results
  - → The plots are accessible in the „Solution" tab on the top.
  - → We add a new rectangular plot

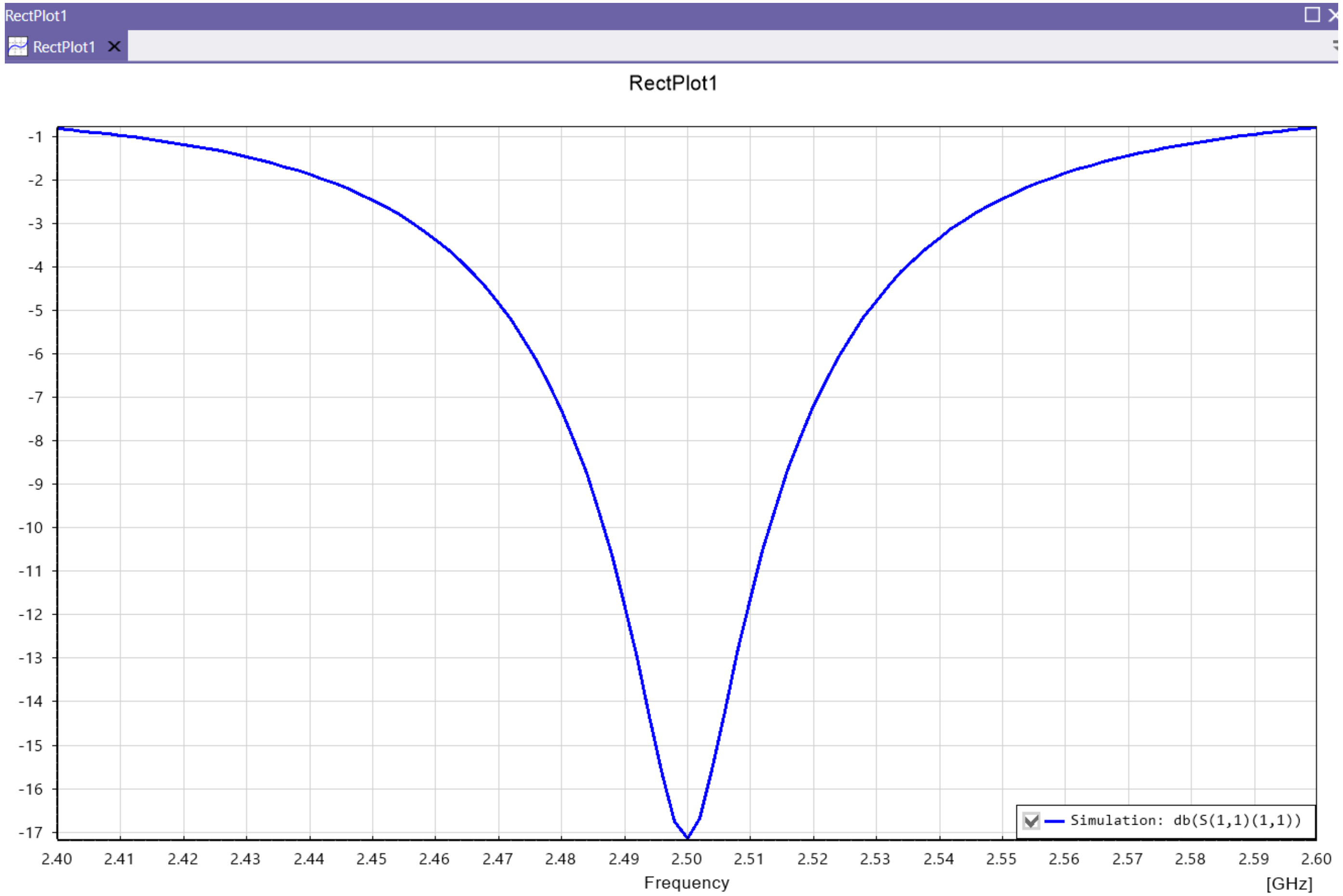


  - → Then we add data series to display S_{11} vs. frequency (dB)



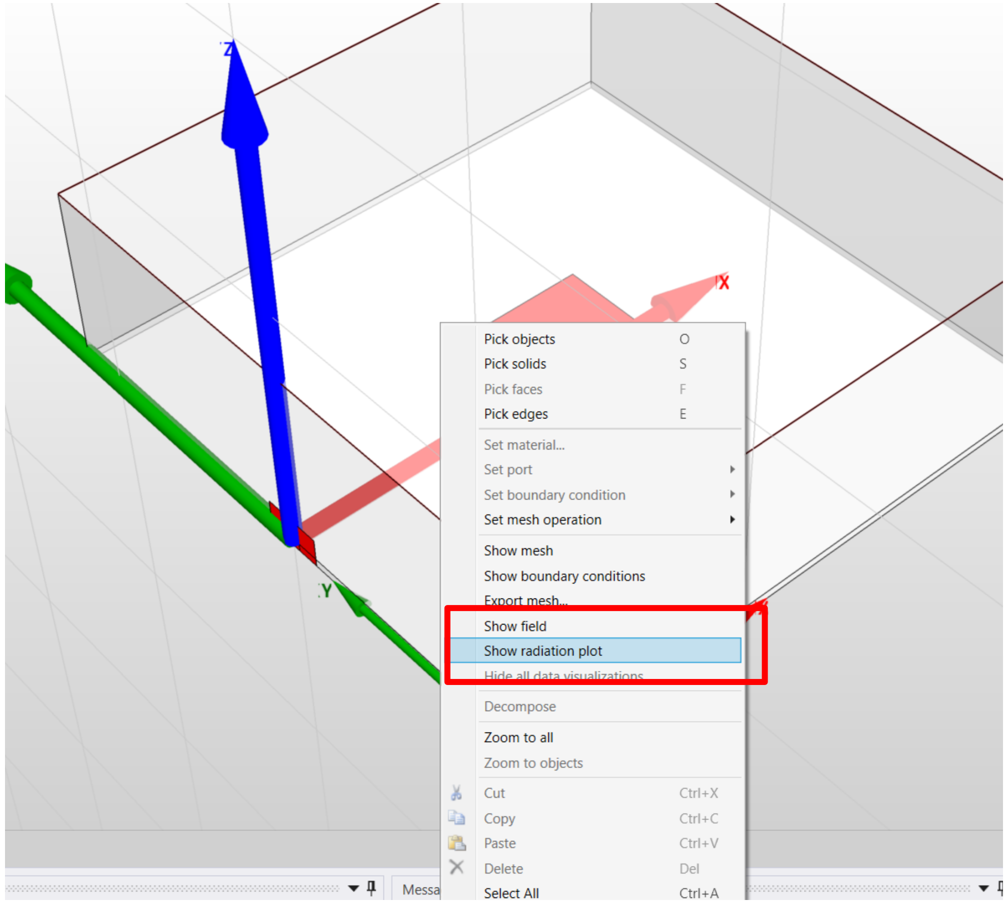
First simulation

- Simulation results show antenna is matched at 2.5 GHz:



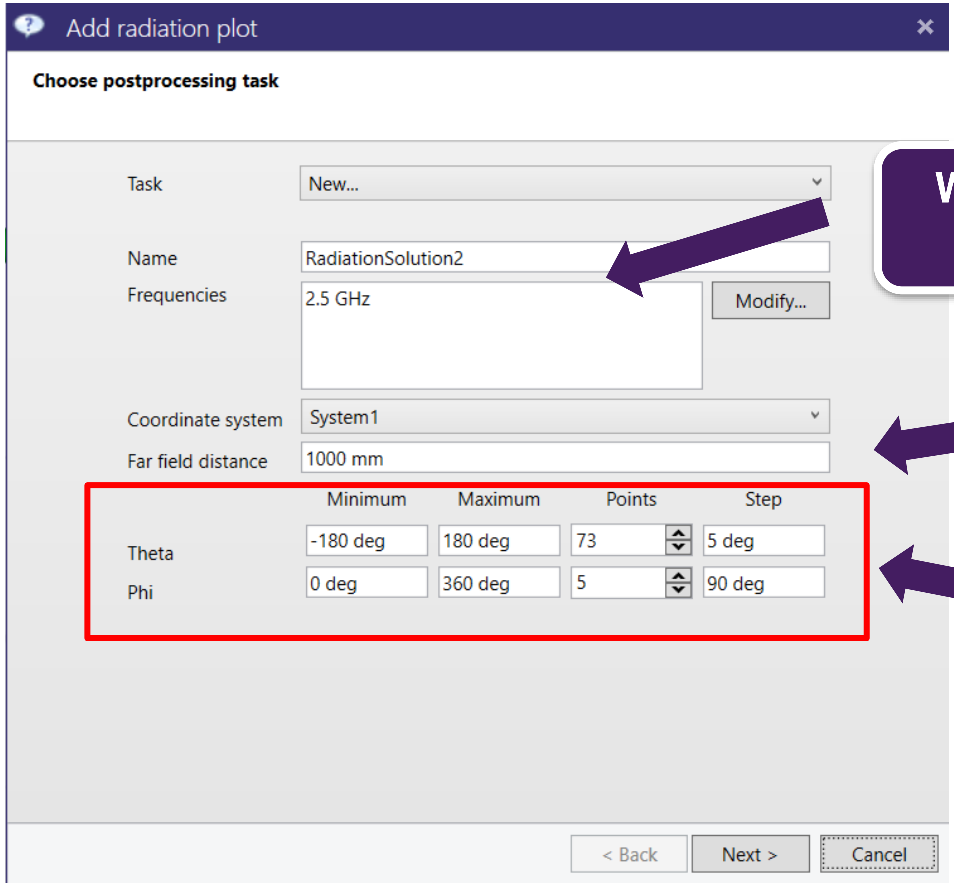
Calculating far fields

- The calculate the radiation parameters, we add a new postprocessing using 'Show Radiation from the context menu:



Calculating far fields

- The calculate the radiation parameters, we add a new postprocessing using 'Show Radiation from the context menu:



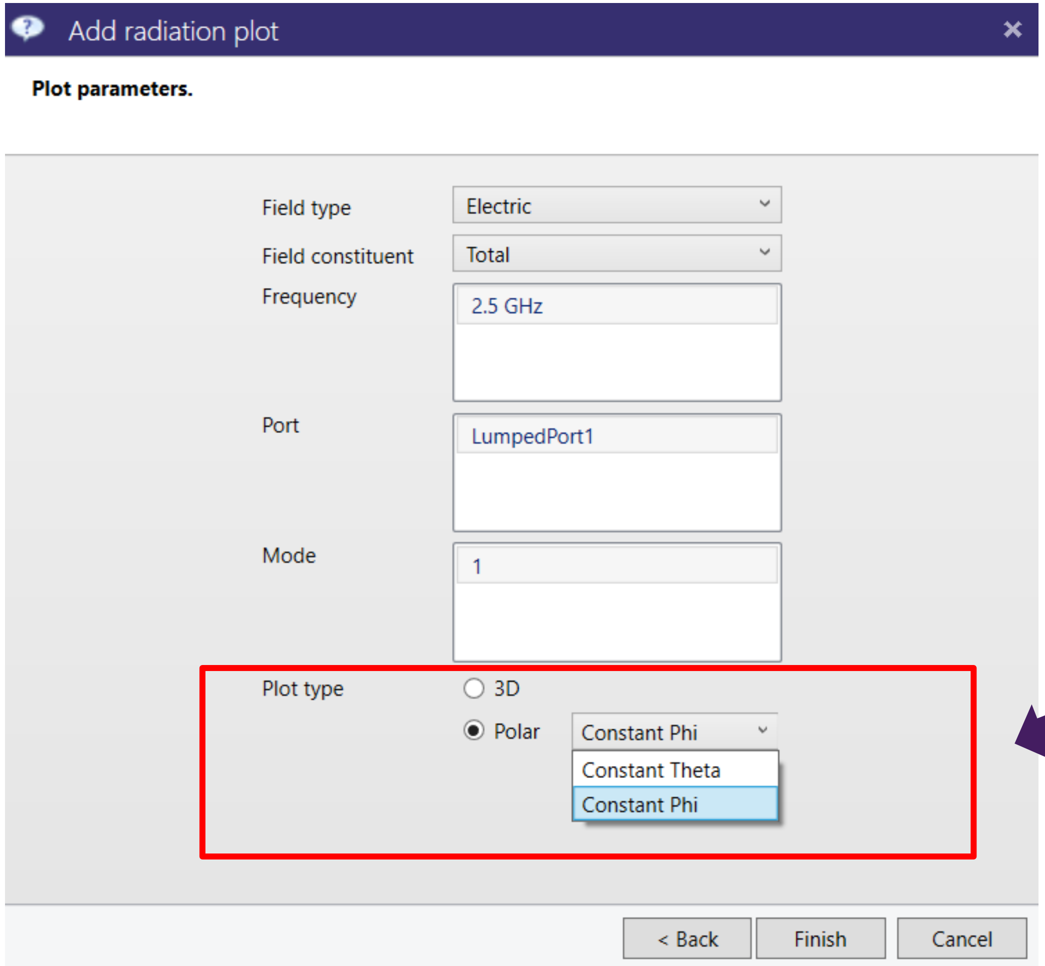
We saved the fields at 2.5 GHz

Far field zone is set as 1m

We set the angular resolution in θ, ϕ directions

Calculating far fields

- Next, we define the plot type:

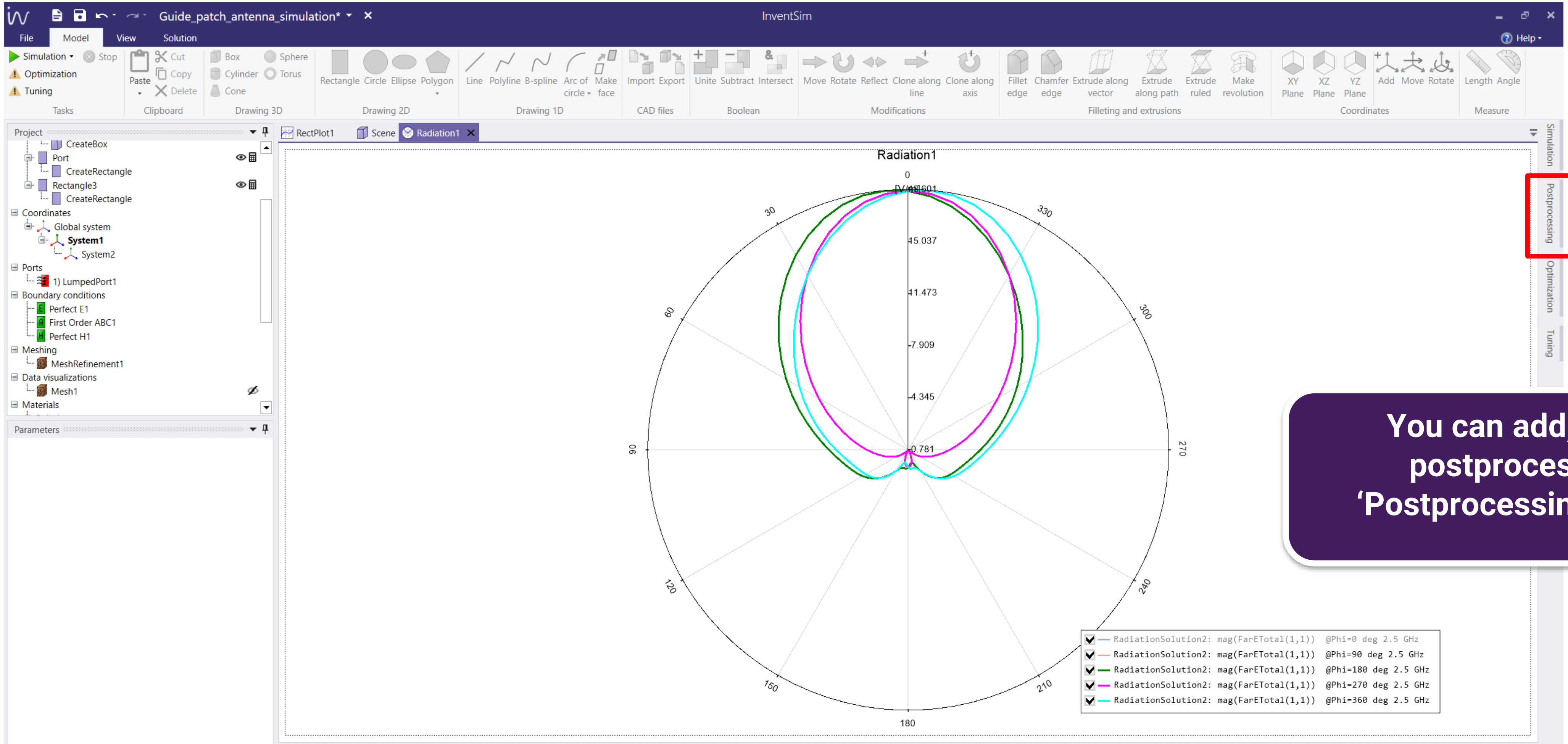


Create a 2D polar plot on a plane of constat ϕ

When we click finish, the fields will be calculated.

Calculating far fields

- When the far fields are calculated, a new polar plot is created:



You can add, remove and edit postprocessing task in the 'Postprocessing' tab on the right.

Calculating antenna gain

- The gain of the antenna is related with electric field $\mathbf{E}(\theta,\phi)$ at given distance R with formula:

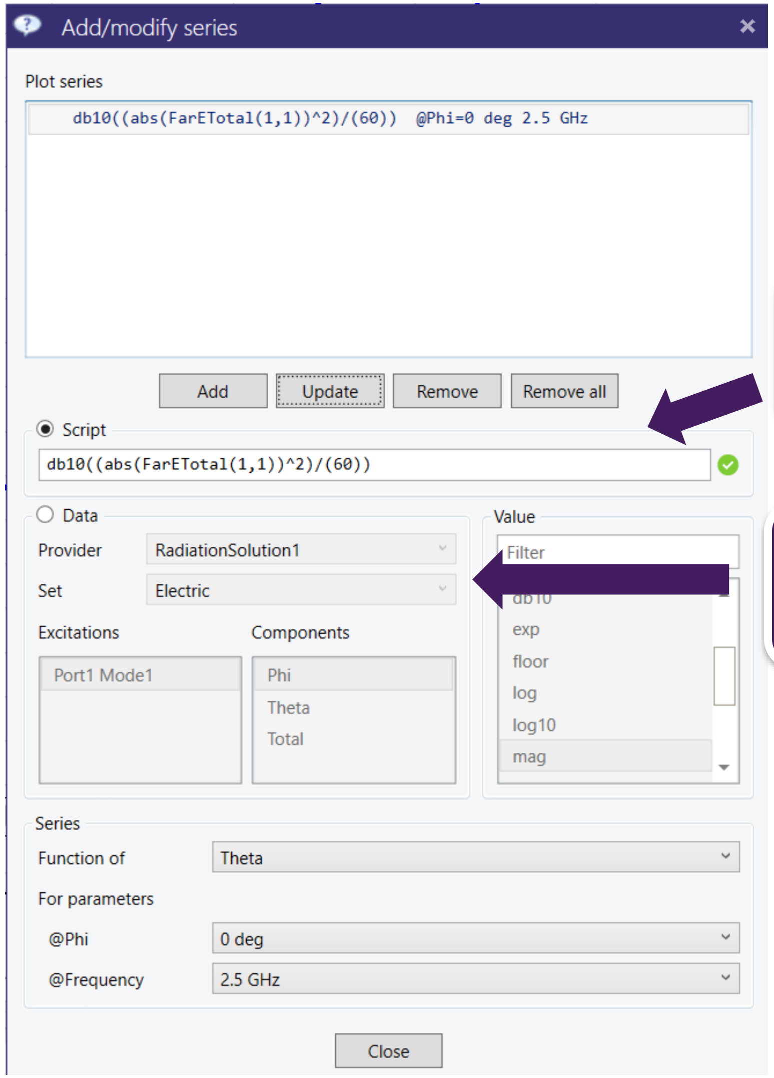
$$G(\theta,\phi) = 4\pi \frac{R^2 \,|\, E(\theta,\phi)|^2}{2 \cdot 120 \cdot \pi \cdot P_{in}} = \frac{R^2 \,|\, E(\theta,\phi)|^2}{60 \cdot P_{in}}$$

- This can be implemented in InventSim as an equation defined by script when adding a data series to the plot:
  - Radius R=1m (as defined in the postprocessing task)
  - Pin = 1W

Script

```
db10((abs(FarETotal(1,1))^2)/(60))
```

Calculating antenna gain

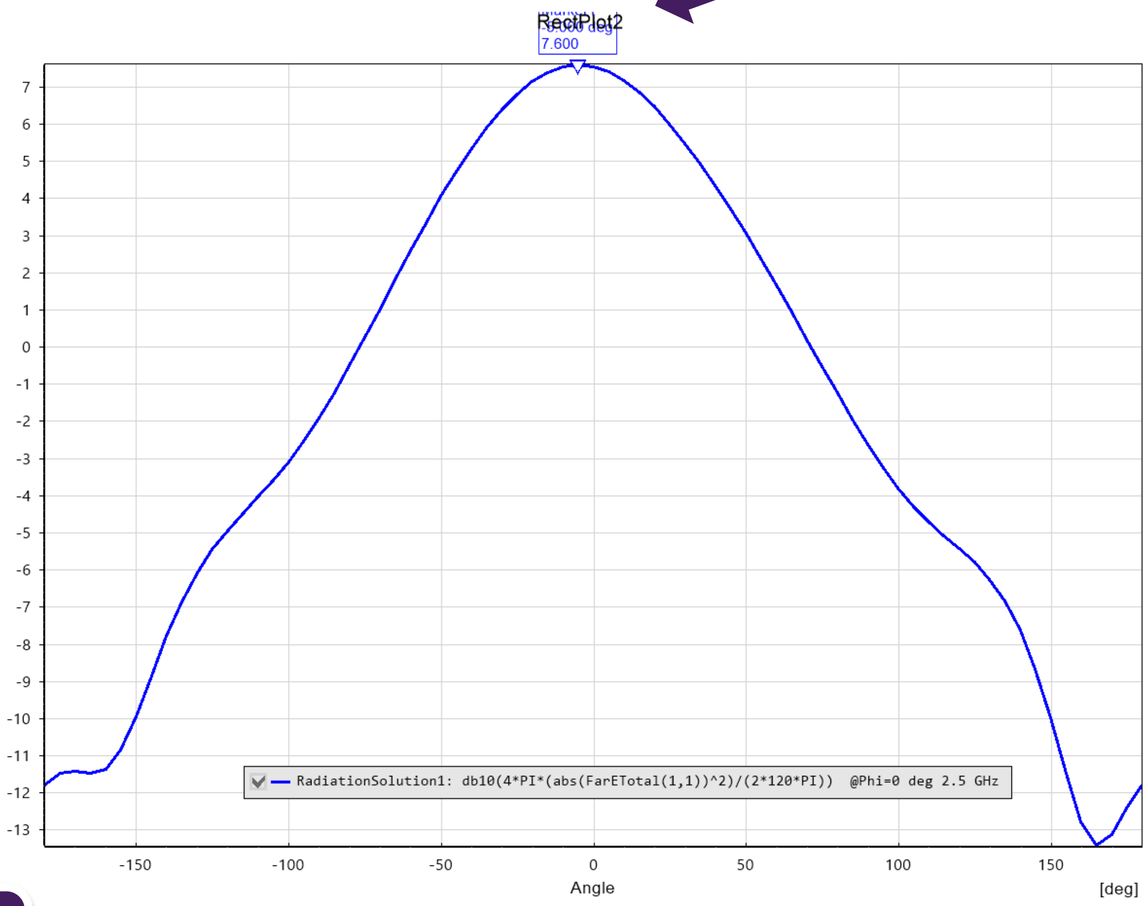


Equation to evaluate gain

Data provider is ‘RadiationSolution1’

Plot gain vs. θ at plane $\phi = 0$

Maximum gain 7.6dBi



INVENTSIM

www.inventsim.com

e-mail us: contact@eminvent.com

EM Invent Sp. z o.o. (Ltd.)

Trzy Lipy 3, 80-172 Gdańsk, Poland