Antennas and Radio Links

Computer Exercise 2: Modeling of Microstrip Patch Antenna in ANSYS Electronics Desktop - HFSS

Brno University of Technology
Petr Kaděra, Jaroslav Láčík

2 Modeling of Microstrip Patch Antenna in ANSYS Electronics Desktop - HFSS

2.1 Goal

The goal of this computer exercise is to get experience with the program ANSYS HFSS and design and model in this program a linearly polarized inset-fed microstrip patch antenna.

2.2 Tasks

1) Design a linearly polarized inset-fed microstrip patch antenna for a dielectric substrate ARLON Cuclad217 (relative permittivity $\varepsilon_r = 2.17$; tangent loss $\tan(\delta) = 0.0009$; substrate height $h = 1.524$ mm) working in the ISM ($f_0 = 2.45$ GHz) band. The antenna will be fed by a microstrip with a characteristic impedance $Z_0 = 50\ \Omega$. The dielectric substrate is plated by a copper foil of the thickness t=0.036 mm.
2) Get experience with the program ANSYS HFSS.
3) The designed antenna in the point 1) model in the program ANSYS HFSS. Tune the antenna, so that the reflection coefficient is lower than -10 dB in the whole ISM band. Define the parameters of this antenna (the resonant frequency, the impedance bandwidth for S_{11} < -10 dB, the directivity, the gain, the radiation pattern, the antenna half-power beam width (given by a 3 dB decrease)).

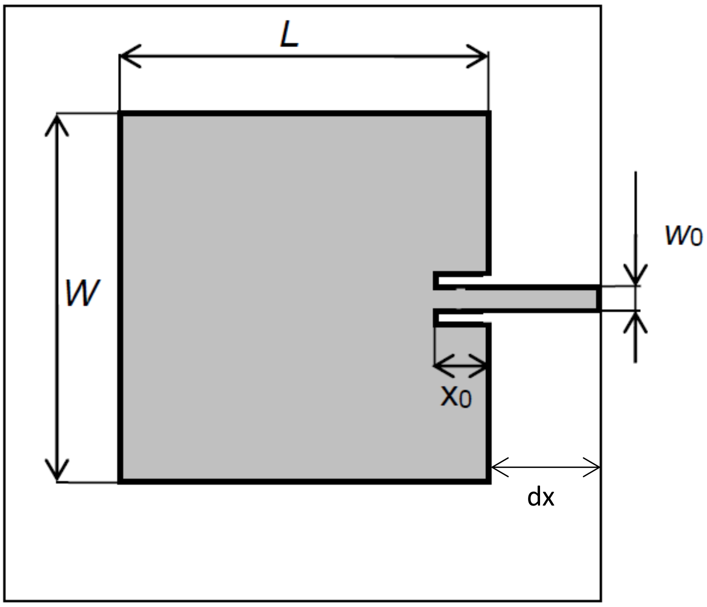


Fig. 2.1: Inset-fed microstrip patch antenna.

2.3 Calculation of microstrip patch antenna

For the input parameters of the patch antenna, we need to calculate its initial dimensions. To do so, we can use either the presentation of the Lecture 3 (1)-(4) or an online calculator (e.g. https://www.emtalk.com/mpacalc.php), where we insert our input parameters and click on Synthetize. The inset feed point of the microstrip line x_0 can be calculated from (4) considering the edge resistance $R_{in}(x=0)$ and desired impedance $R_{in}(x=x_0)$ which is equal to the characteristic impedance of the microstrip line. Further, the microstrip line width w_0 can be calculated again by online calculator (https://www.pasternack.com/t-calculator-microstrip.aspx) to have 50 Ohm feeding line.

$$W = \frac{c}{2f_0\sqrt{\frac{\varepsilon_r+1}{2}}} \tag{1}$$

$$\varepsilon_{r,eff} = \frac{\varepsilon_r+1}{2} + \frac{\varepsilon_r-1}{2}\left(\frac{1}{\sqrt{1+12\left(\frac{h}{W}\right)}}\right) \tag{2}$$

$$L = \frac{c}{2f_0\sqrt{\varepsilon_{r,eff}}} - 2\cdot 0.412h\left(\frac{(\varepsilon_{r,eff}+0.3)\left(\frac{W}{h}+0.264\right)}{(\varepsilon_{r,eff}-0.258)\left(\frac{W}{h}+0.8\right)}\right) \tag{3}$$

$$R_{in}(x = x_0) = R_{in}(x = 0)\,cos^2\left(\frac{\pi}{L}x_0\right) \tag{4}$$

Further, lets choose the distance between the edge of ground and the edge of patch $dx=W/2$.

2.3 ANSYS HFSS

The program ANSYS HFSS is a full-wave electromagnetic (EM) field simulator for arbitrary 3D volumetric passive device modelling. The numerical method used in the HFSS is the Finite Element Method (FEM). At this method, a structure is subdivided into many small subsections called finite elements. In the HFSS these finite elements are in the form of tetrahedrons. The entire collection of the tetrahedrons constitutes the finite element mesh. A solution is found for the fields within these tetrahedrons. These fields are interrelated so that Maxwell's Equations are satisfied across inter-element boundaries yielding a field solution for the entire original structure. Once the field solution is found, the generalized S-matrix solution can be determined.

The HFSS uses the automatic adaptive mesh refinement process, in which the mesh is refined iteratively and is localized to regions where the electric field solution error can be high. This iterative refinement technique increases the solution's accuracy with each adaptive solution. The refinement process continues until the HFSS converges to an accurate solution. The convergence is determined by monitoring a parameter from one adaptive pass to the next one. The most common convergence criterion is to ensure that the difference in the S-parameter value between two consecutive solves is less than the desired value.

2.5 Modeling of microstrip patch antenna

In the first step, run the ANSYS Electronics Desktop and insert a new project (Fig. 2.2).

Project => Insert HFSS Design

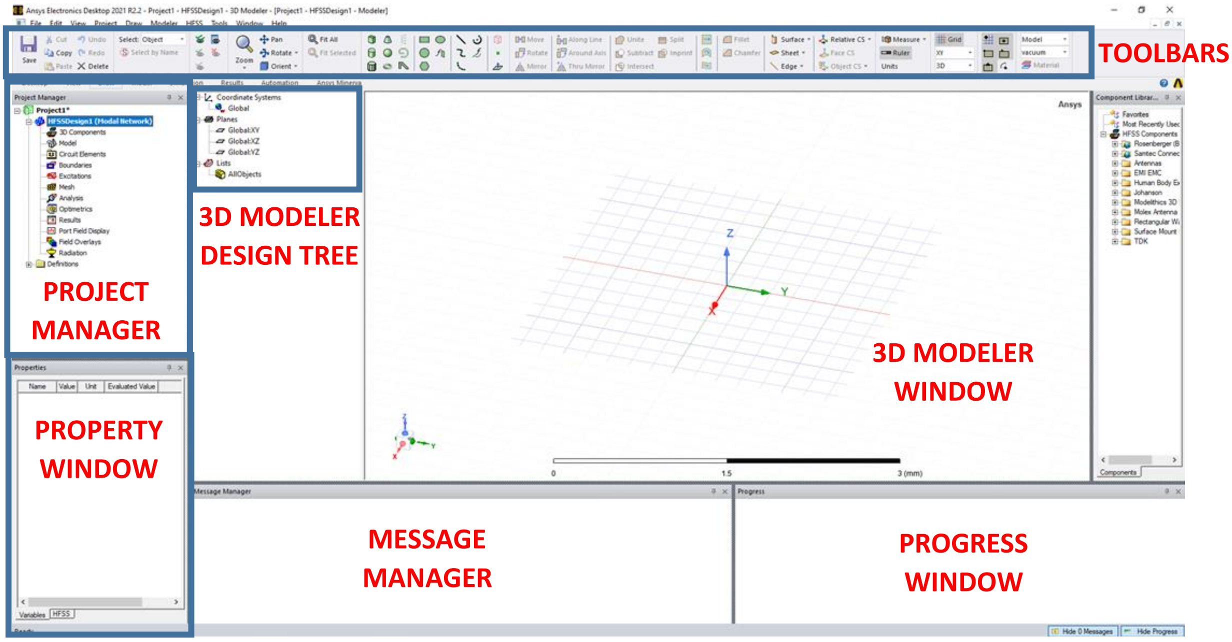
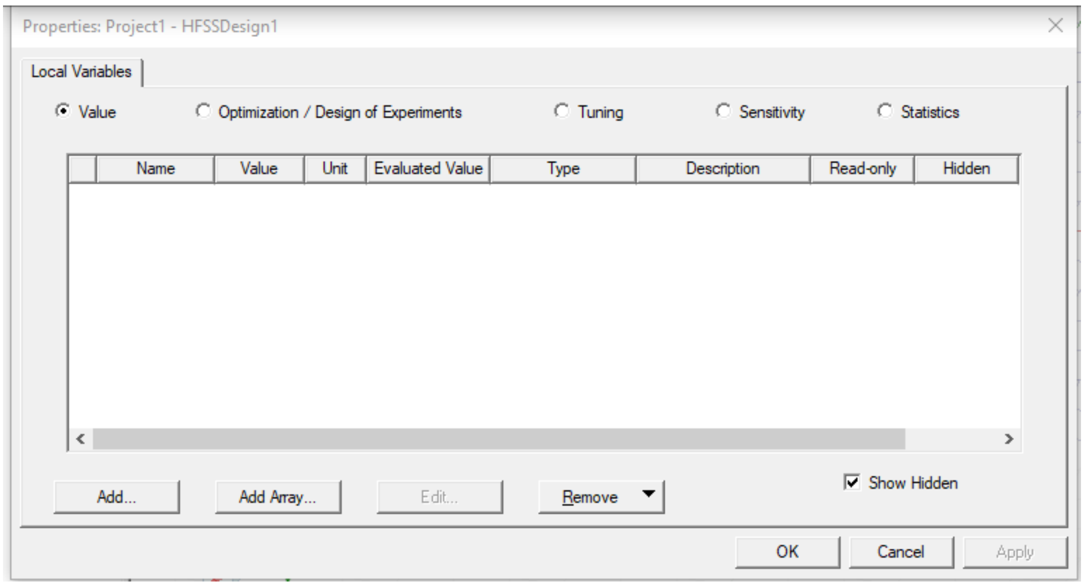


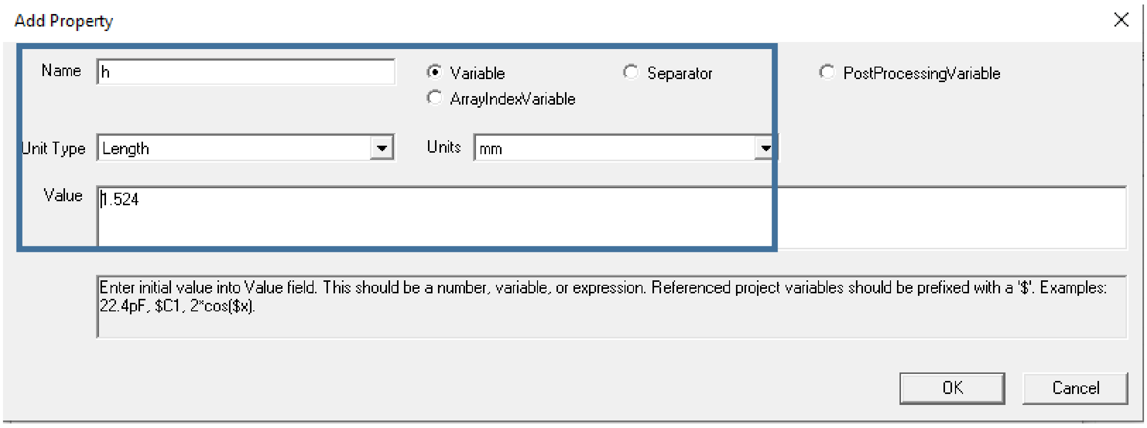
Fig. 2.2: Environment of Ansys HFSS.

In the second step, add all (*h*, *W*, *L*, …) design variables into the current project and fill them with the given or calculated antenna parameters. Detail steps are demonstrated in Fig. 2.3.

a) HFSS => Design Properties => Add



b) *Click the Add button and name the variable, define ist unit type, units and the value.*



Defined variables:

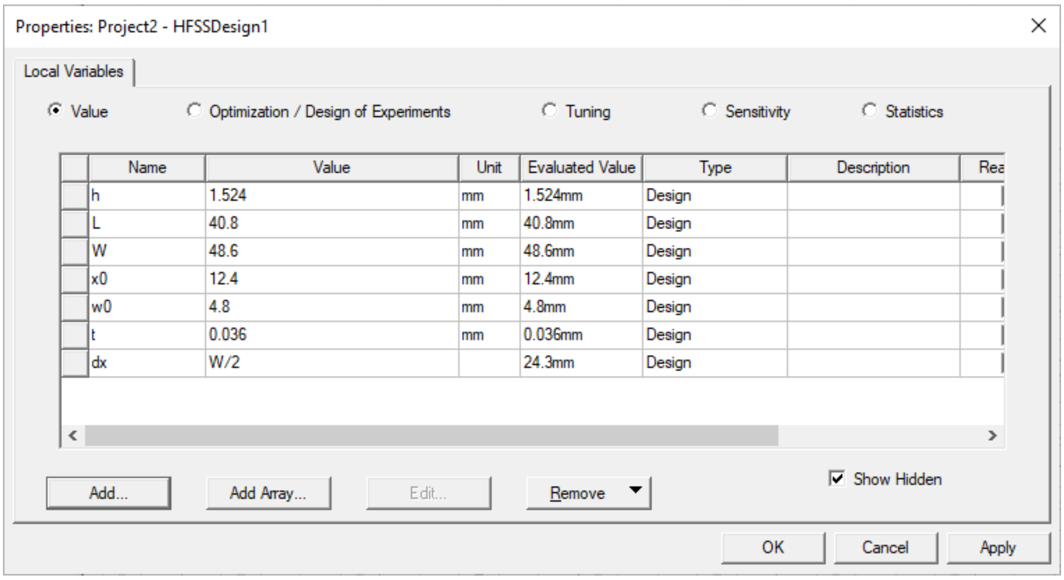


Fig. 2.3: Definition of variables.

For an easier drawing in the xyz coordinate system, change the drawing mode from the Point to the Dialog (Fig. 2.4).

Tools=> Options => General Options => 3D Modeler => Drawing

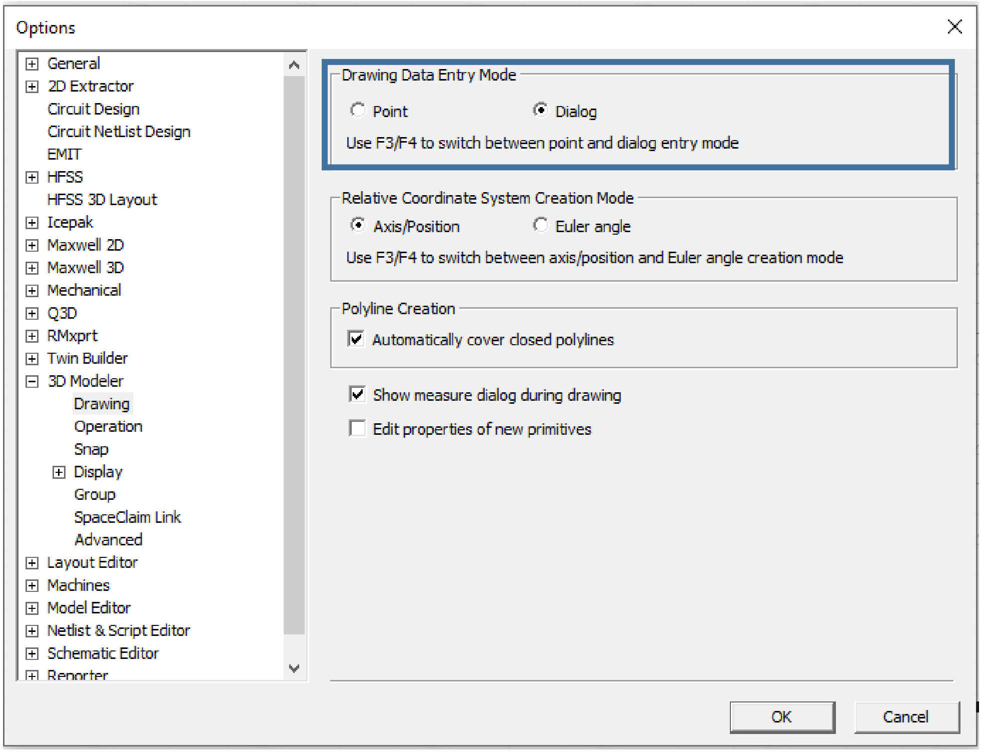
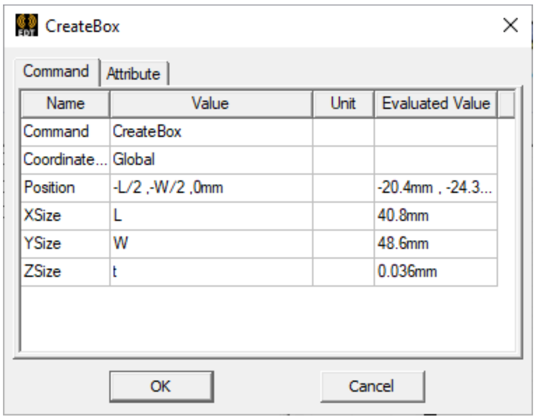


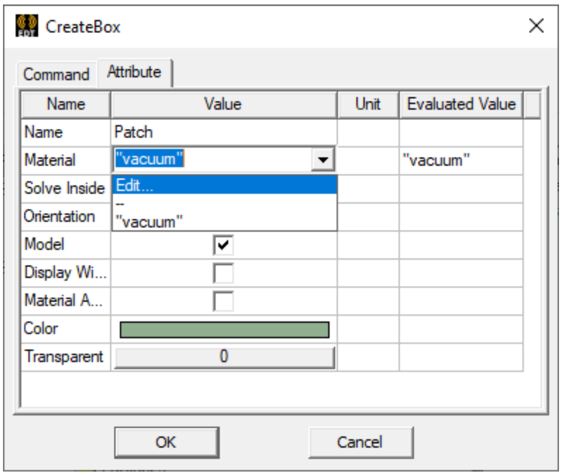
Fig. 2.4: Change of drawing date entry mode.

In the third step, draw/create and define attributes of individual antenna model components (the patch, the dielectric substrate the ground, and the feeding microstrip line). Detail steps related to drawing the antenna patch are presented in Fig. 2.5.

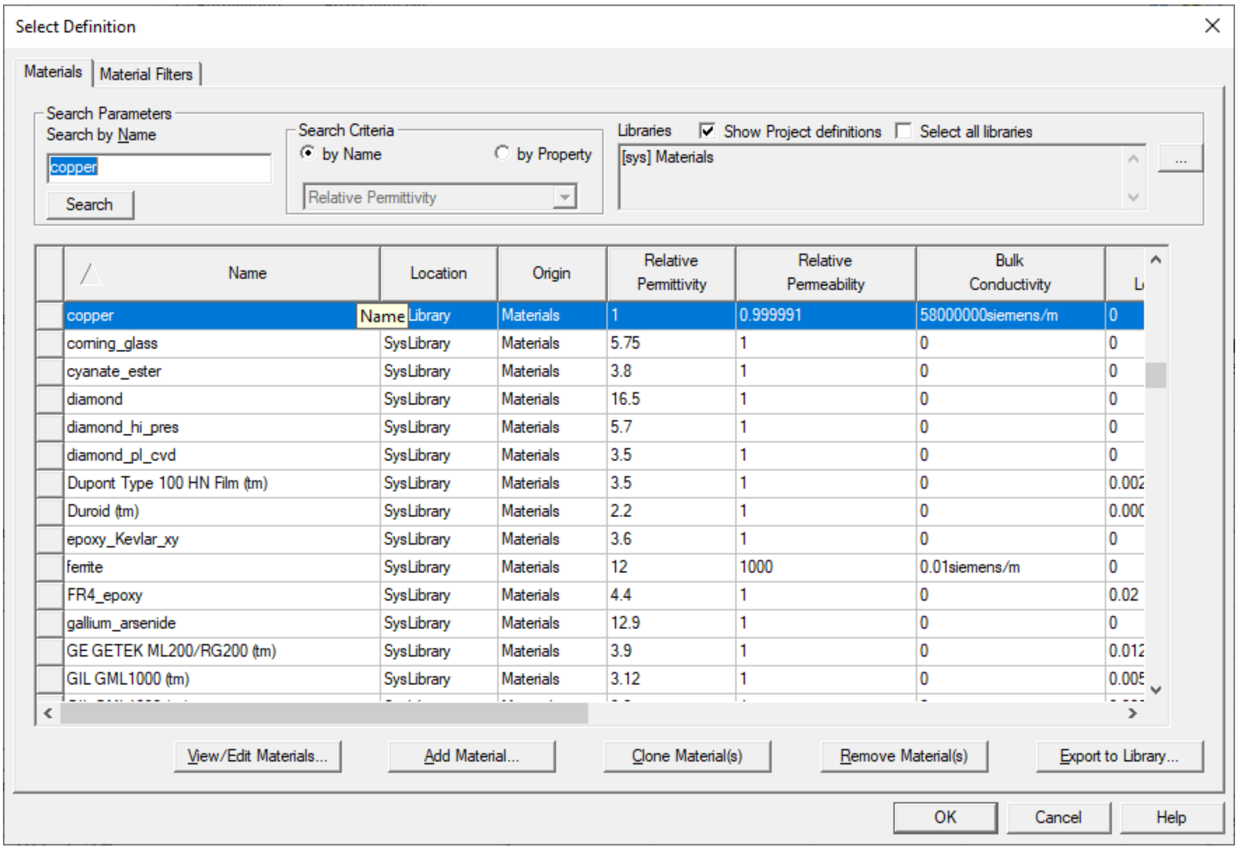
a) *Draw a box for representation of the patch.*
Draw => Box



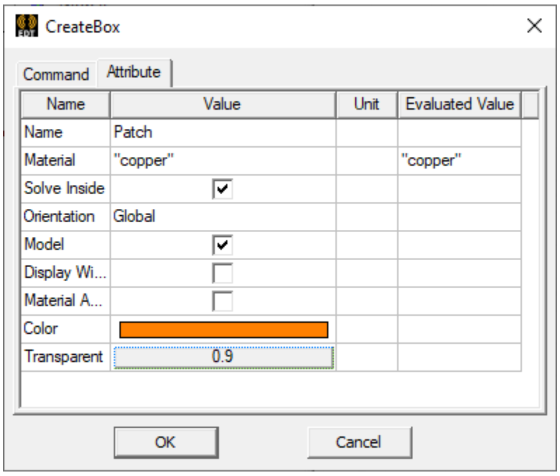
b) *Define box attributes: Name (name of component in 3D modeller tree), Material, Color (color for representation of component in 3D modeller window), Transparent (transparency of draw component).*



The patch is made from copper. If the copper is not presented in the list, select Edit and try to find it in Library of materials. Before selection of material check it properties. For the patch, select "copper".



Note if the material is not presented even in material library, it is possible add your own material to the project (clicking on Add Material button in previous dialog and defining material properties).
After definition of all attributes, the Attribute dialog should look like:



c) Finally, created patch appears in 3D modeller window.

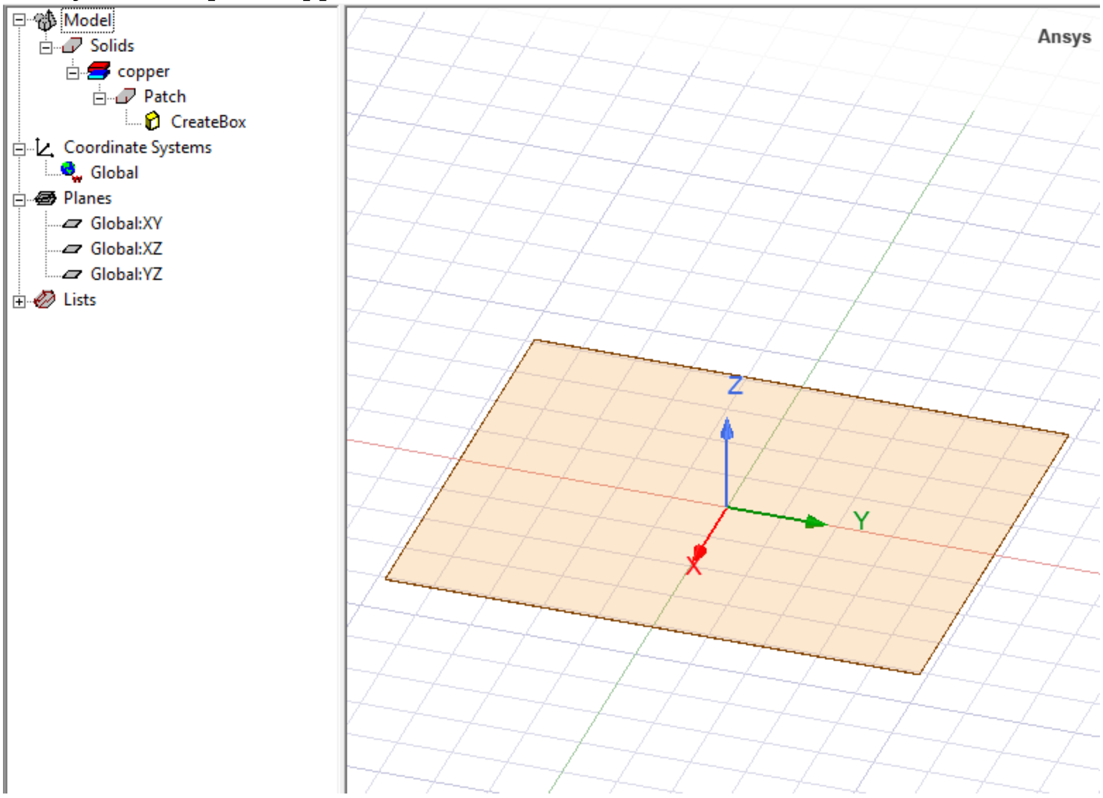
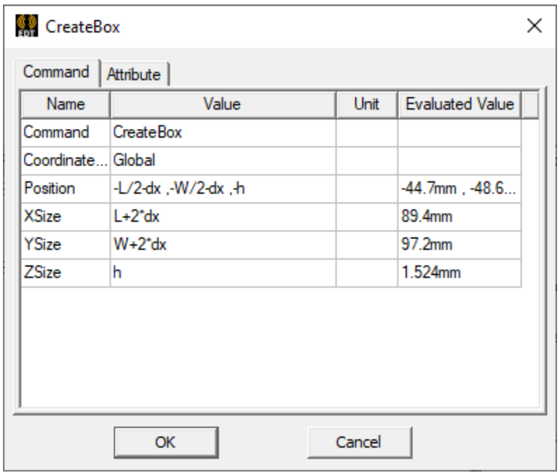


Fig. 2.5: Drawing and defining of patch.

In the same way, define other components of the model on your own. To help with that, the copies of dialogs related to drawing other antenna parts without showing definition of attributes is depicted in Fig. 2.6.

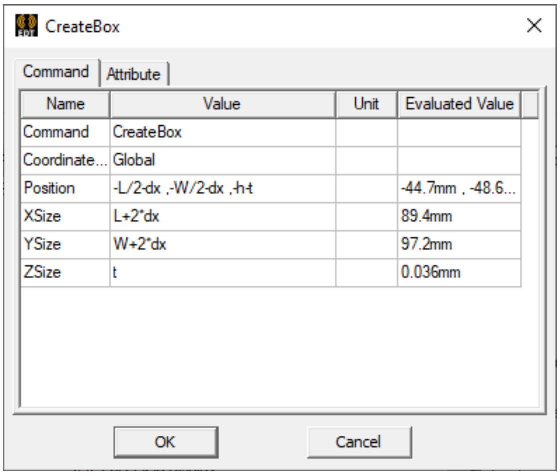
a) Drawing of dielectric substrate.

Draw => Box



b) Drawing of ground.

Draw => Box



c) Drawing of microstrip line.

Draw => Box

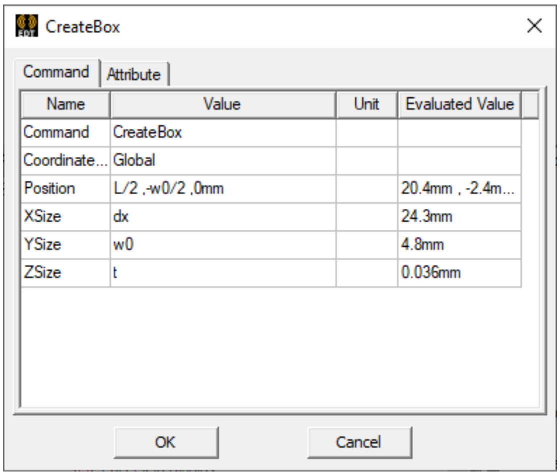
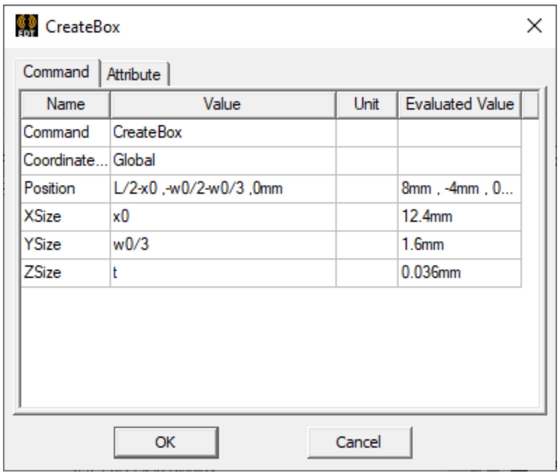


Fig. 2.6: Drawing dielectric susbstrate, ground and microstrip line.

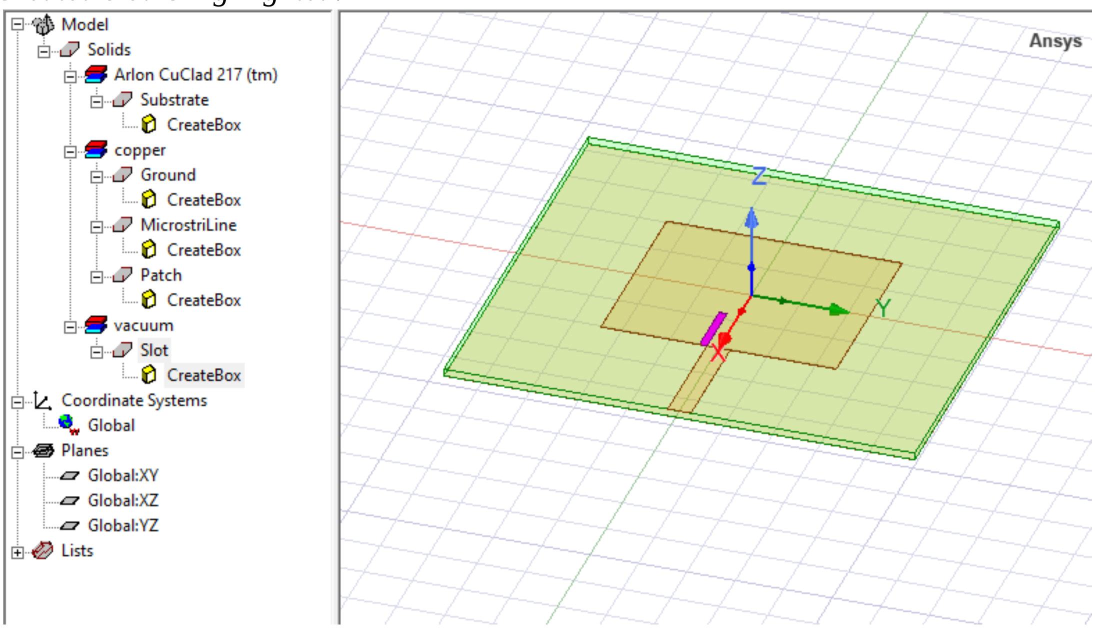
To create inset-fed patch, draw the box representing a slot, duplicate it to obtain another slot, units both slots and subtract them from the patch. Detail steps are presented in Fig. 2.7.

a) *Draw a box for representation of the slot (in the attribute dialog define just the name, the rest leave as it is since this component is finally subtracted from the antenna model to modify the shape of the patch).*

Draw => Box

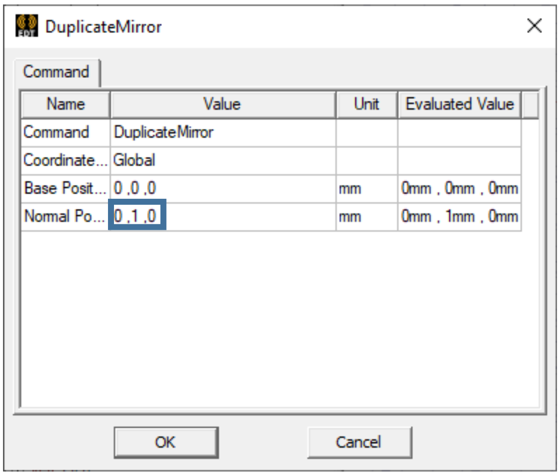


Created slot is highlighted:

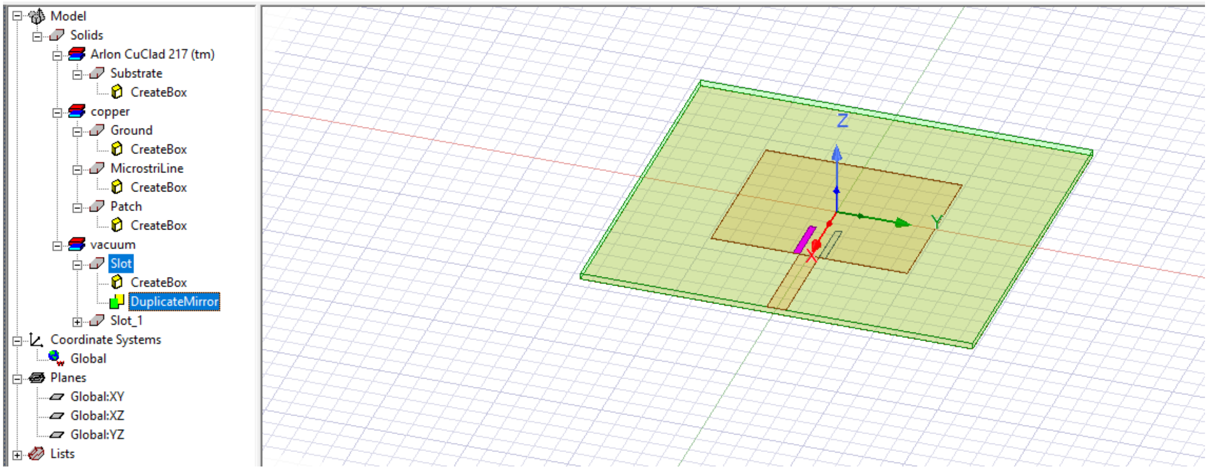


b) *Assure that the slot is selected in 3D modeler design tree (if not, click on it) and duplicate (mirror) it with respect to the y axis.*

Edit => Duplicate =>Mirror



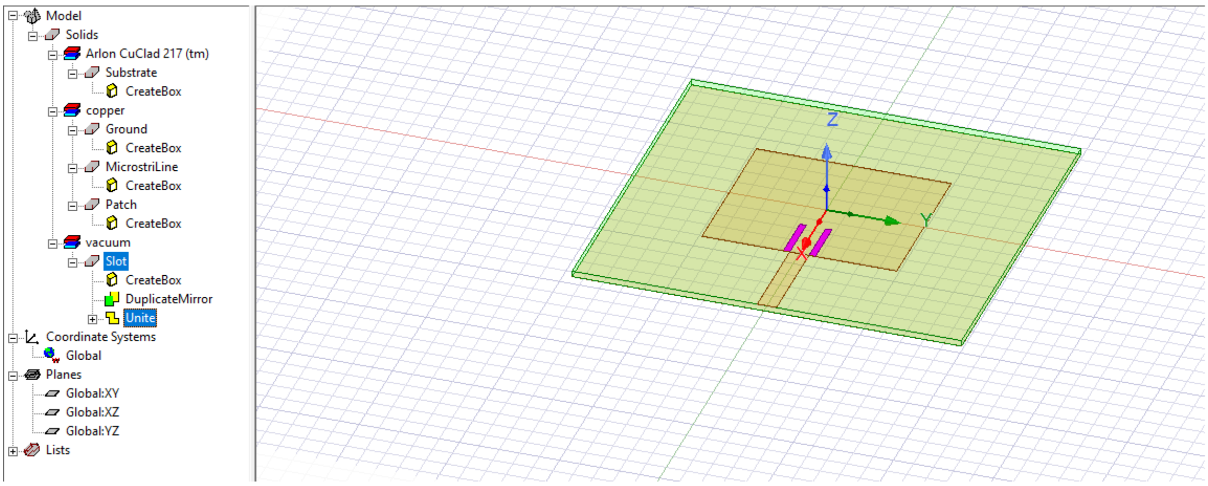
Original and duplicated slots:



c) *Select both slots and unit them.*

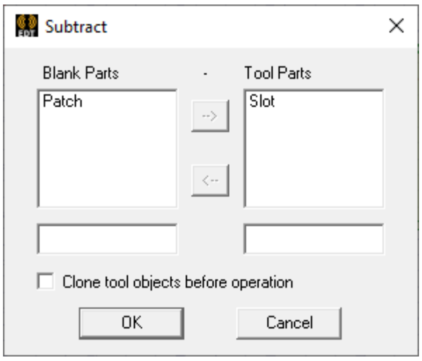
Modeler => Boolean =>Unite

United slots:



d) *Finally, select the patch and united slots and subtract the slots from patch.*

Modeler => Boolean =>Subtract



The final model of the patch is:

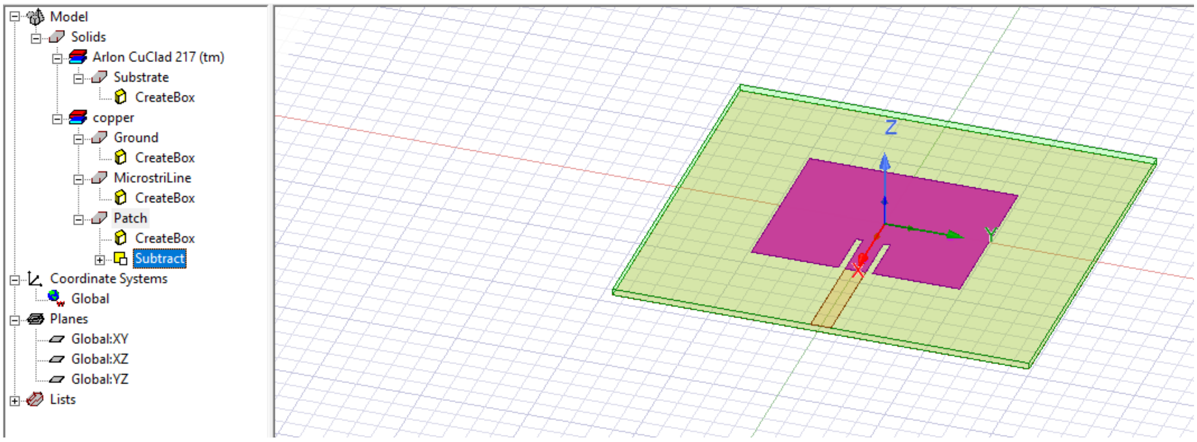
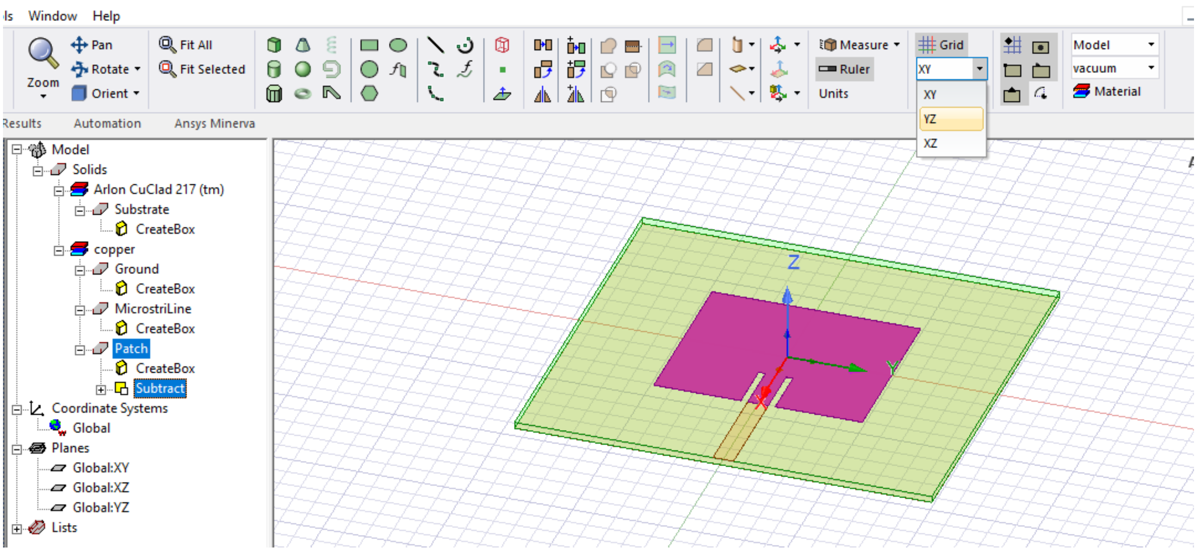


Fig. 2.7: Creating inset-fed patch.

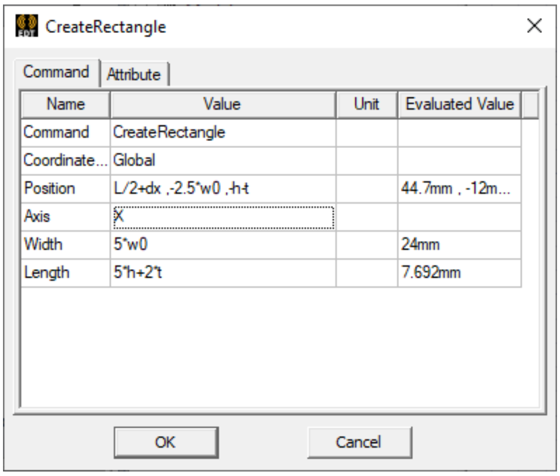
In the fourth step, insert the wave port. Follow the detail steps in Fig. 2.8.

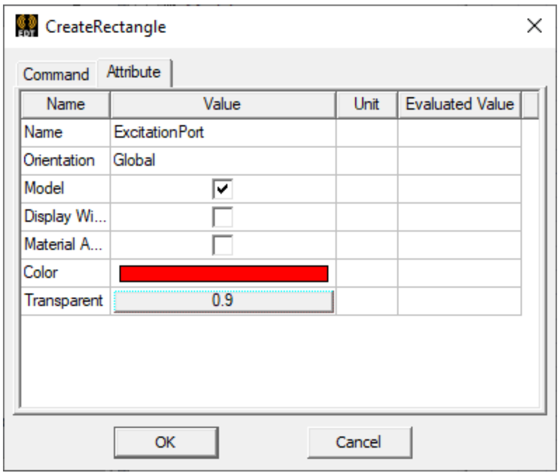
a) *First, switch to the YZ drawing plane.*



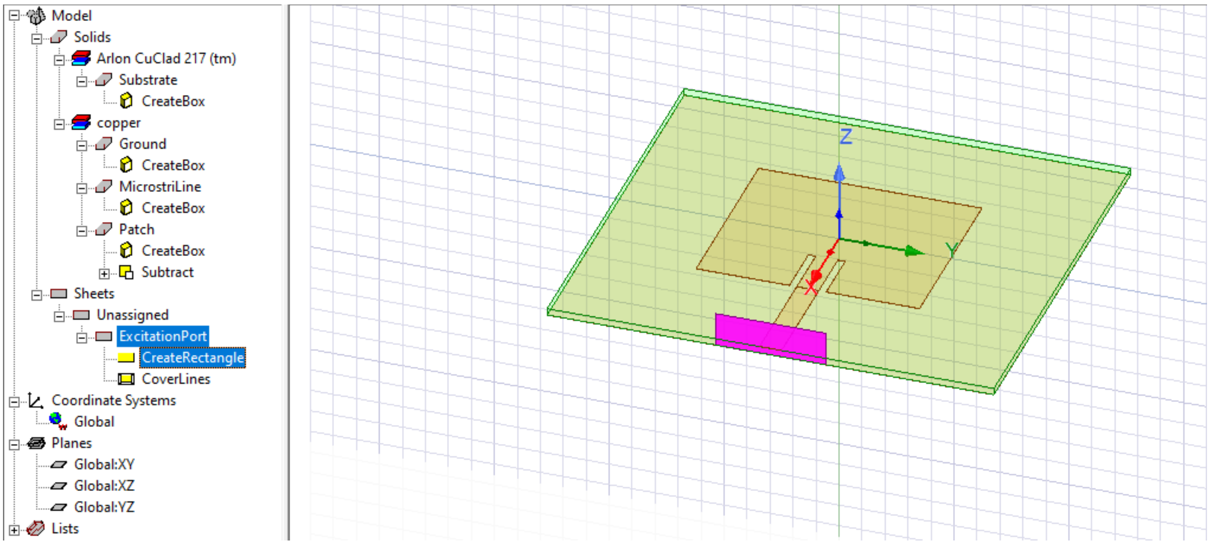
Draw a rectangle and define its attributes. The port can have zero thickness and its dimensions should be sufficient for the proper mode excitation (a rule of thumb: the width of the port should be approximately the five times the width of the feeding line and the three times of the height of the dielectric substrate). The distribution of the field at the port should be checked to ensure that a desired propagation mode is excited and the field has significant decay at the port edges (at least about the 30 dB in comparison to the maximum of the port field).

Draw => Rectangle





Defined rectangle for wave port:

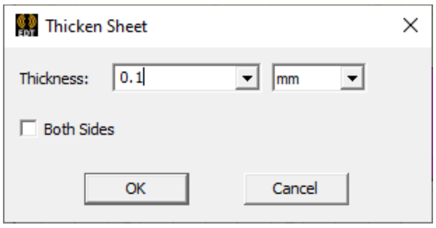


b) *The direction of the wave propagation for the wave port has to be defined. Back the port with PEC. Therefore, select the outer face of the created rectangle (press "F" on keyboard and click on outer face of the rectangle) and create a new object from it.*

Modeler => Surface => Create Object from Face

Then thicken a new surface sheet to the thickness of 0.1 mm.

Modeler => Surface => Thicken Sheet



Make sure that the resulting sheet is going outwards of the antenna. Assign the PEC material to the new solid component.

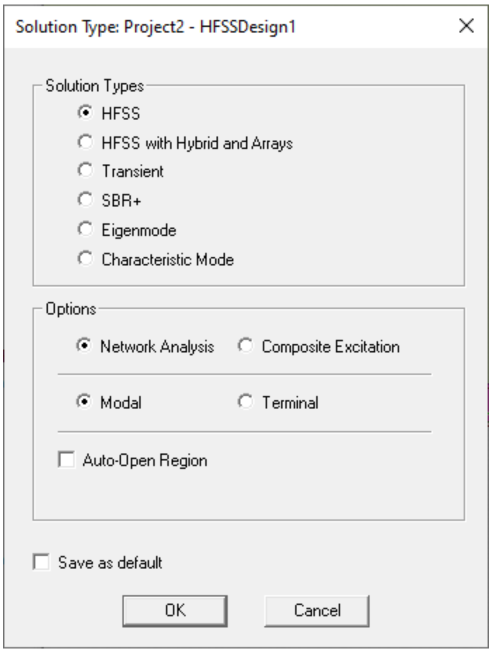
Modeler => Assign Material

Created PEC box from the face:

c) *Place the wave port on the original thin rectangle.*

At first, check/select that the HFSS modal solution type is selected.

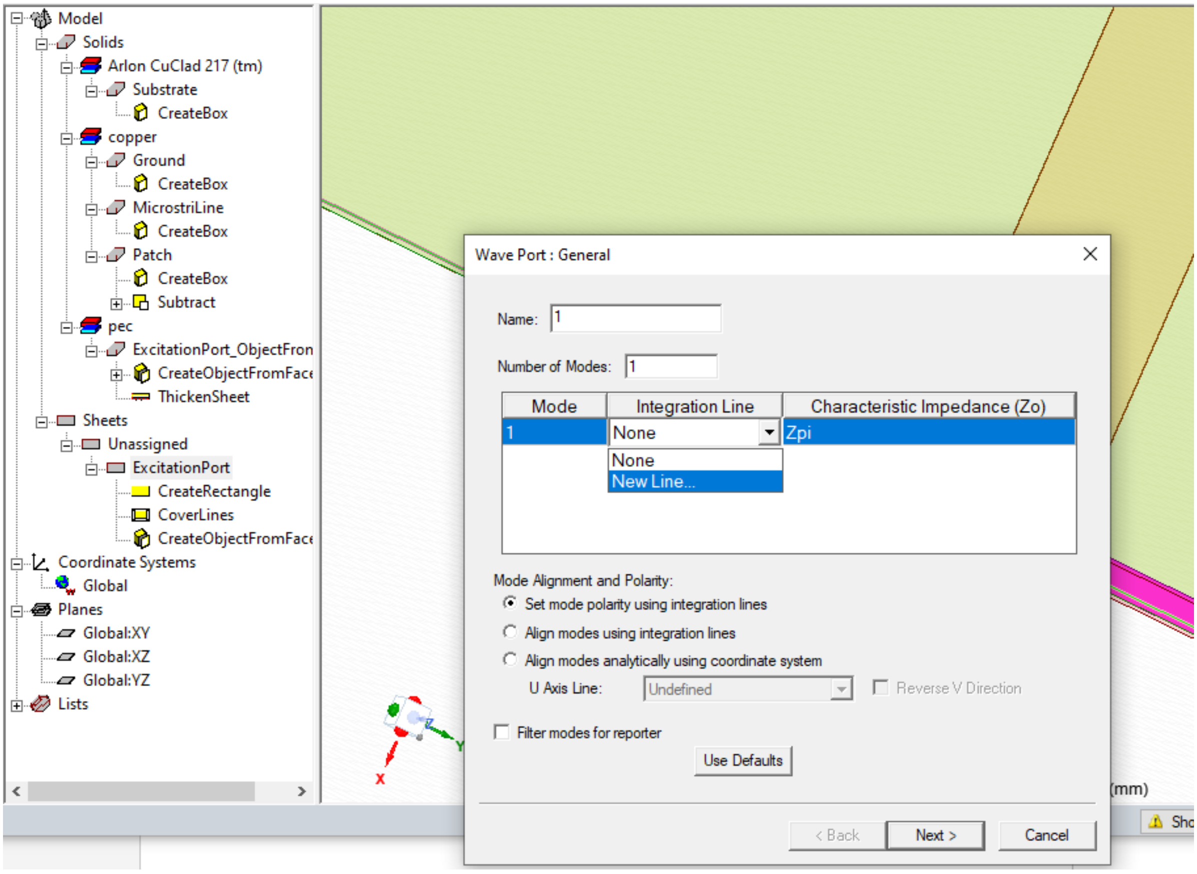
HFSS => Solution type



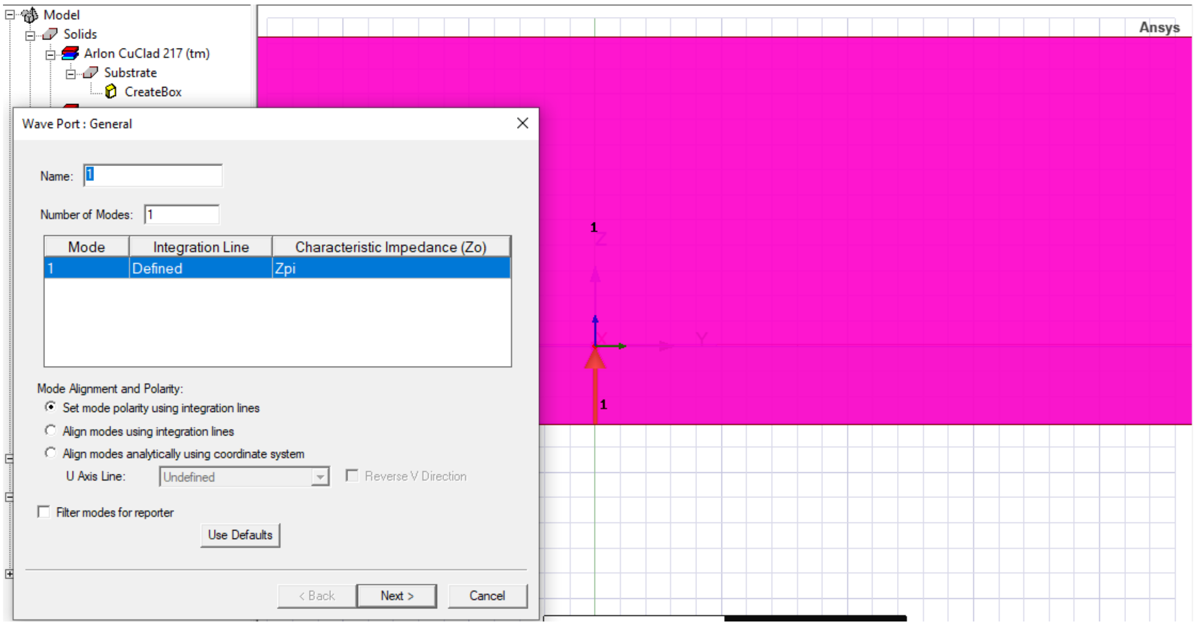
Then select ExcitationPort component in 3D modeller tree and assign wave port

HFSS => Excitation => Assign => Port => Wave Port

Define a name of the port and the integration line between the ground and the microstrip line.



Defined the integration line:



After that, in post-processing tab set "renormalize all modes to full port impedance: 50 Ohm.".

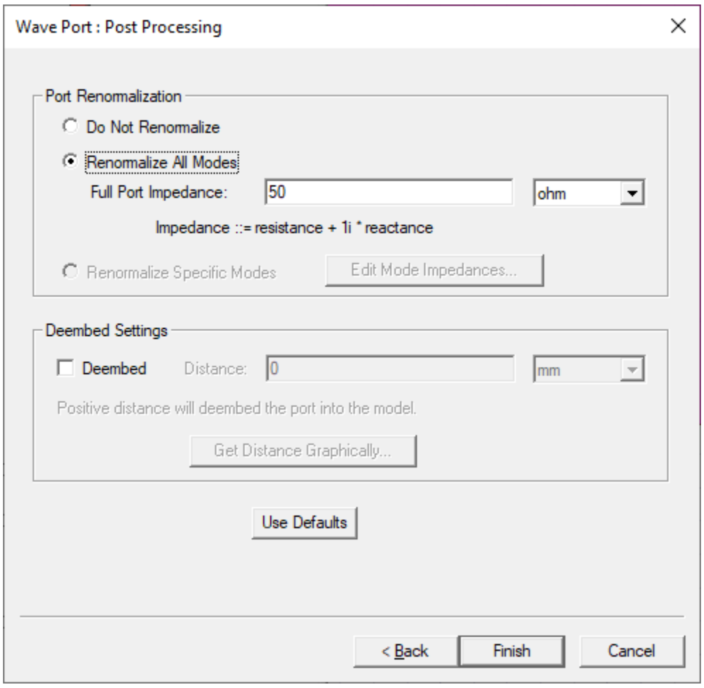
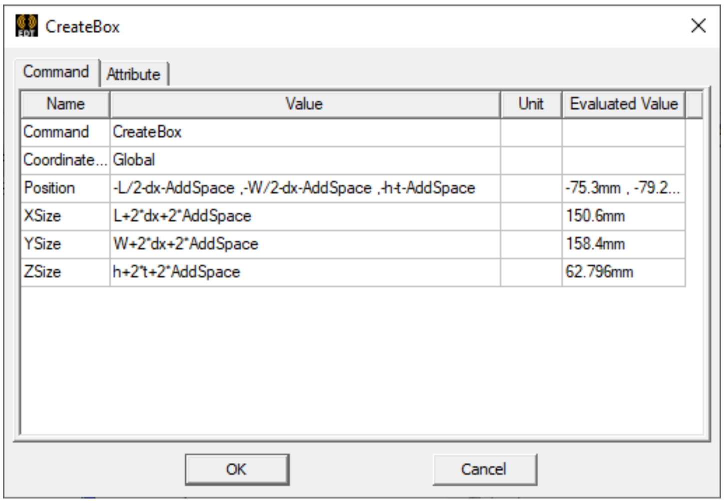


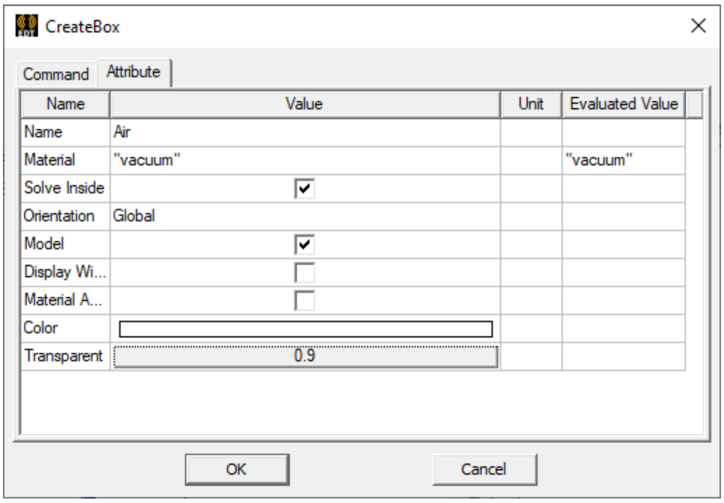
Fig. 2.8: Definition of wave port.

In the fifth step, create the vacuum (air) box to surround the antenna and add radiation boundary on this box to absorb antenna radiation. The size of the box should be large enough to absorb the EM waves and its recommended size should be at least $\lambda_0/4$ (better $\lambda_0/2$). This box should be generally large enough, so the antenna parameters do not change with changing its size. Detail steps are presented in Fig. 2.9.

a) *Draw a vacuum box around the antenna considering that the distance between the antenna and the box is* $\lambda_0/4$. *Let's name this distance in the project by AddSpace (=*$\lambda_0/4$ *=30.6 mm) variable. This variable can be defined e.g. during drawing the box.*

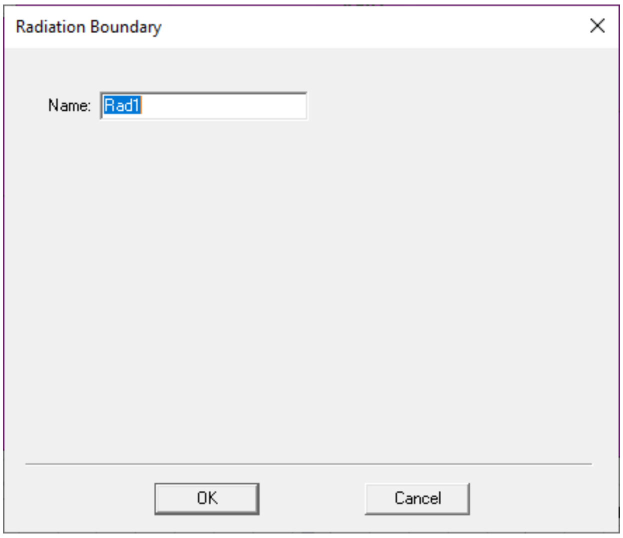
Draw=> Box





b) Select the Air (vacuum) box and add radiation boundary.

HFSS=>Boundary=>Assign=>Radiation



Antenna surrounded by vacuum/air box:

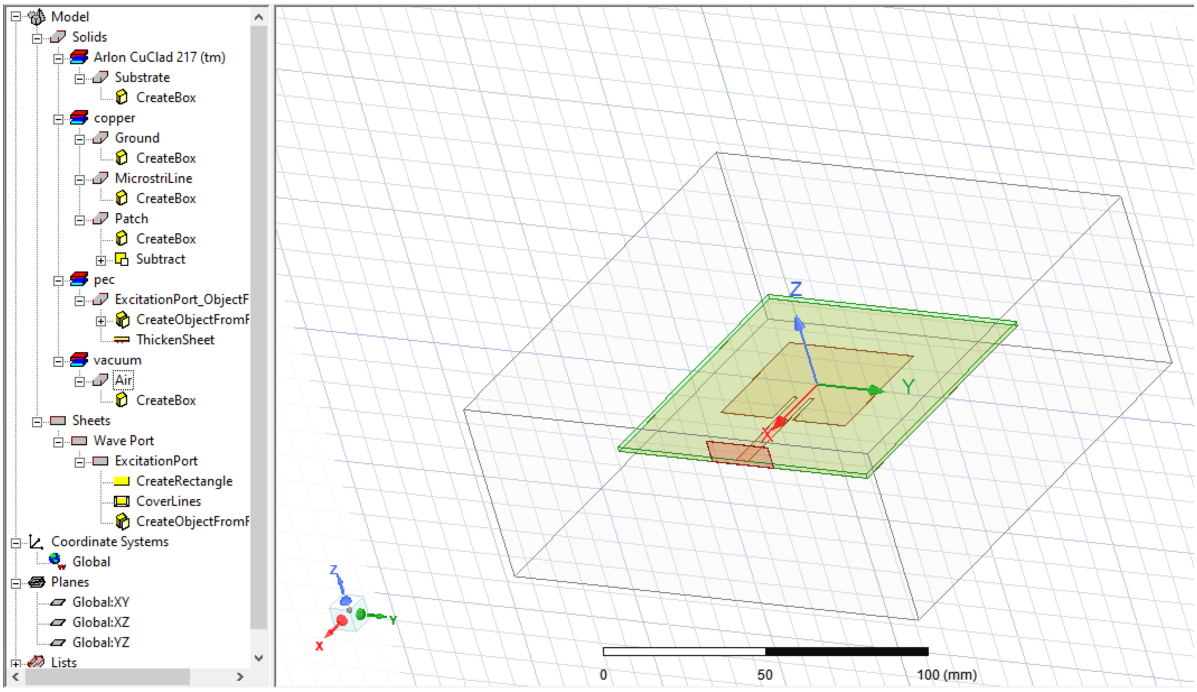


Fig. 2.9: Definition of a vacuum (air) box with radiation boundary.

In the sixth step, define setup for calculation of the antenna far-field. Let's consider that the far-field pattern is recorded in two principal planes: E-plane (XZ plane) and H-plane (YZ plane). Detail steps are presented in Fig. 2.10.

a) Definition of Far-field setup.

HFSS=> Radiation => Insert Far Field Setup => Infinite Sphere

Define angular range and resolution.

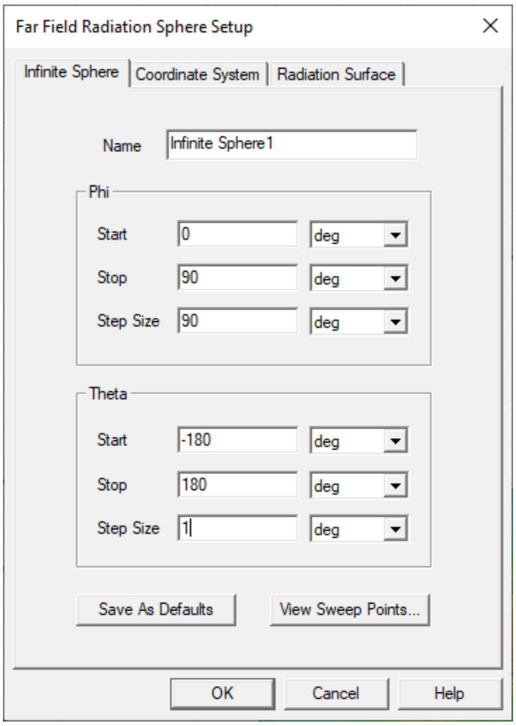
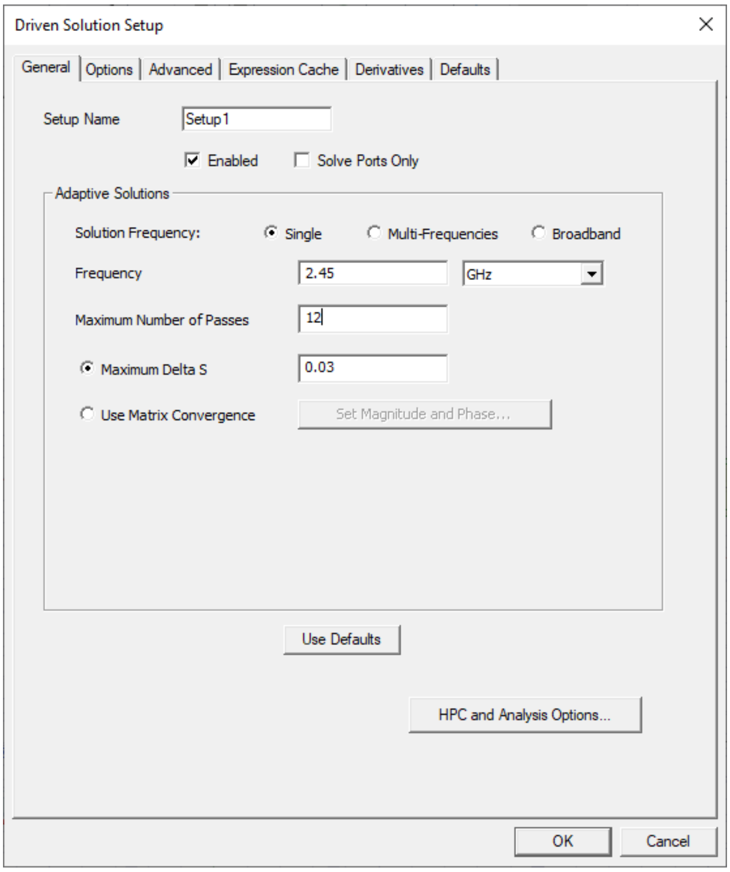


Fig. 2.10: Definition of the far-field radiation simulation setup.

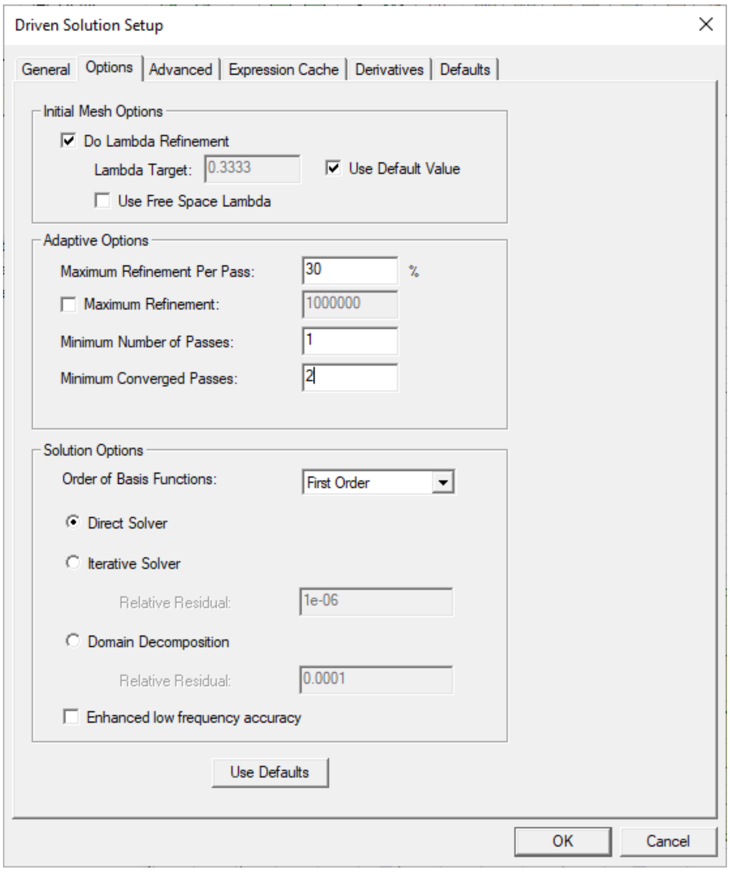
In the seventh step, add analysis setup and choose the frequency range of interest; check the save field and select the infinite sphere for the far-field radiation (Fig. 2.11).

a) *HFSS => Analysis => Add Solution Setup => Advanced Setup*

On General dialog define solution frequency 2.45 GHz, Maximum Number of Passes and Maximum Delta S. This parameters are related to the adaptive meshing (see Lecture 1). Here we consider that the adaptive meshing will be done at the frequency 2.45 GHz (the center frequency of the ISM band), Maximum Number of Passes is 12 and the acceptable difference of S-parameters between consecutive passes is 0.03 (3%) – the Maximum Delta S parameter.



Further, click the Option dialog and set the Minimum Converged Passes to 2. So to have the Converged results, the desired Maximum Delta S has to be met two times.



Then Edit Frequency Sweep dialog is evoked. Select Start and End frequency, calculation method (Interpolating) and sufficient number of points. The interpolating sweep does not simulate every single point and therefore its accuracy is limited, but it will run faster. It is also suitable for wideband frequency sweeps. The discrete sweep simulates every single point as specified in the setup and the simulation reach the highest accuracy but the simulation time will be much longer (depending on the number of points), therefore it is suitable for narrowband simulations.

Finally, tick the save field.

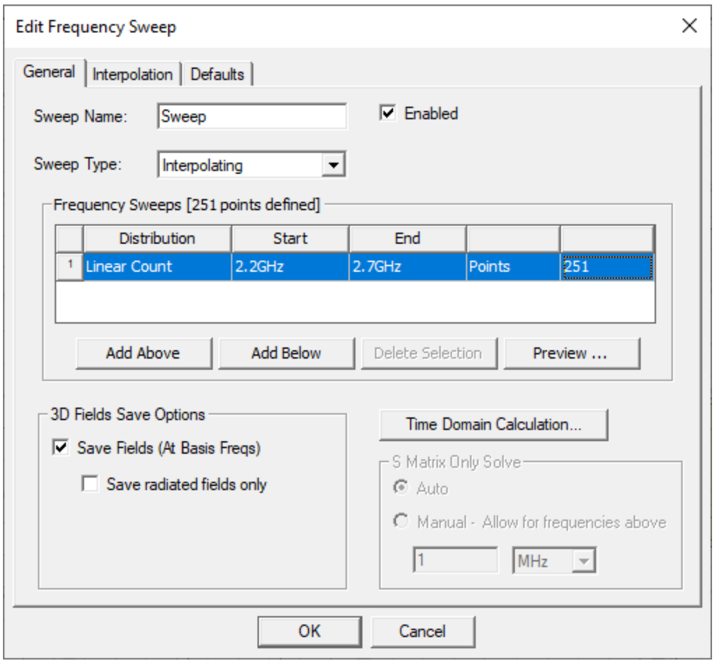


Fig 2.11. Definition of analysis setup.

For proper functionality of the simulation with multiple materials it is recommended to check the Material Override option (Fig. 2.12):

HFSS => Design Settings => Check Enable material override

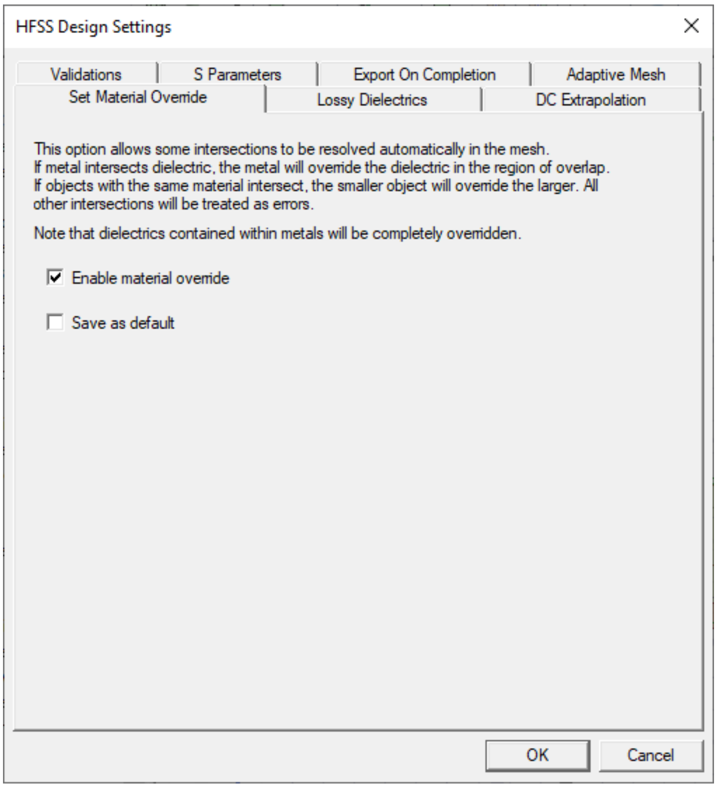


Fig 2.12. Function for materials override.

Now, save the project to your working directory (*File=>Save as*) and check your model (*HFSS=>Validation Check*). If everything is OK (Fig. 2.13), the analysis can be run:

HFSS=> Analysis => Setup1 => Analyze All

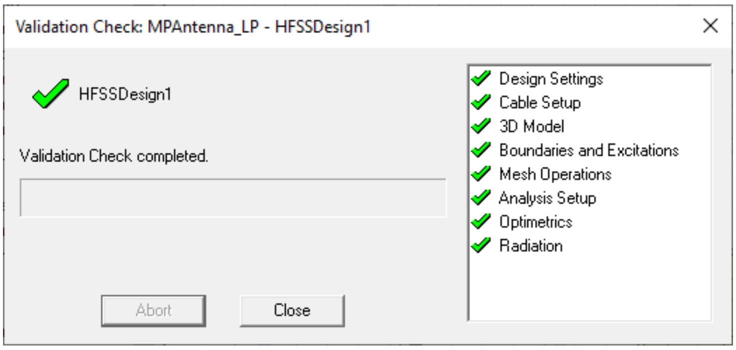


Fig. 2.13: Validation check.

After the analysis, the Solution Data can be observed.

HFSS=>Results=>Solution Data

Click the Convergence table (Fig. 2.14). We can observe that our results have higher delta S then the target. To obtain convergence results, the maximum number of passes should be increased (Fig. 2.11).

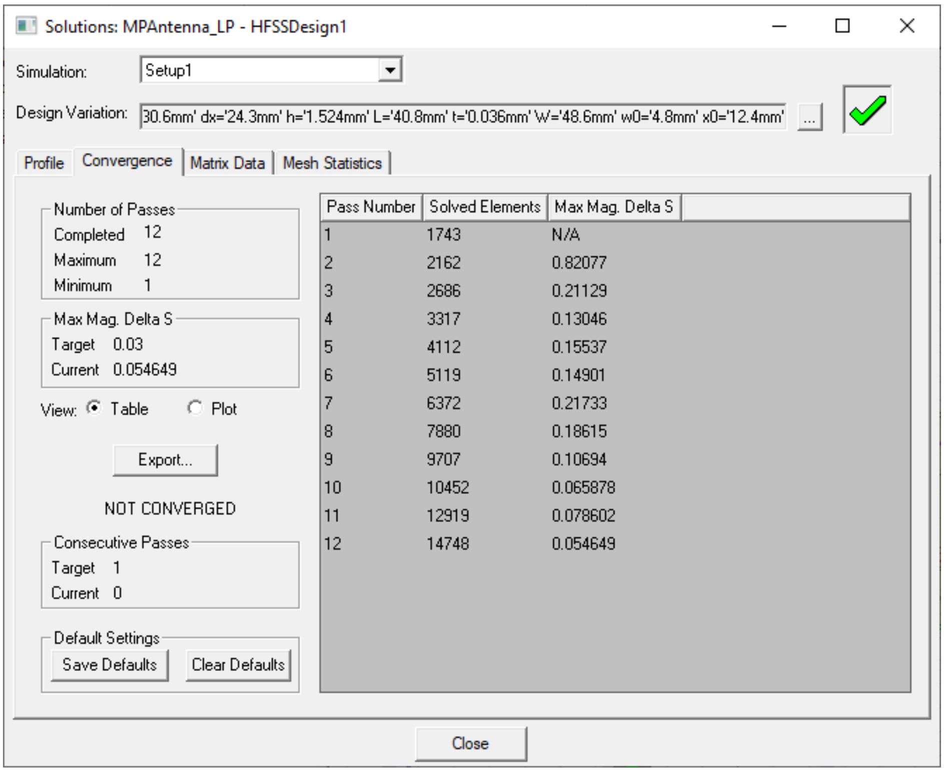


Fig. 2.14: Solution data dialog - Convergence.

At first, check the E-field distribution of the wave port. To do that follow, the steps in Fig. 2.15. Obviously, a desired propagation mode is excited.

In project manager, click Port Field Display => 1 => Mode 1

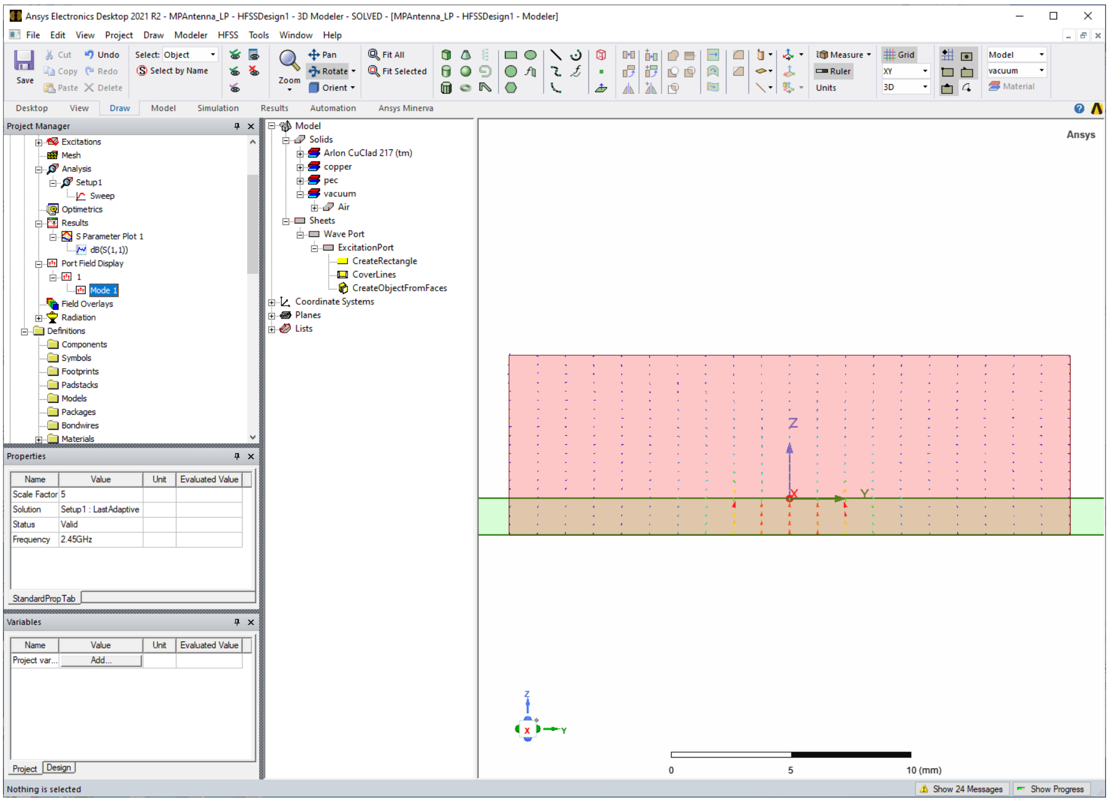


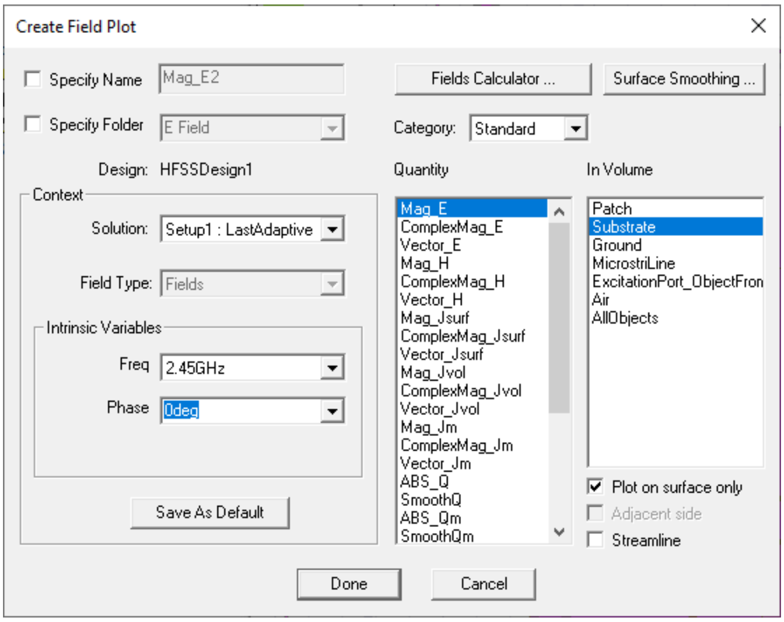
Fig. 2.15: Distribution of E-field on port.

To show the electric field distribution on the antenna, follow the steps in Fig. 2.16. Obviously, the fundamental mode TM_{010} is excited.

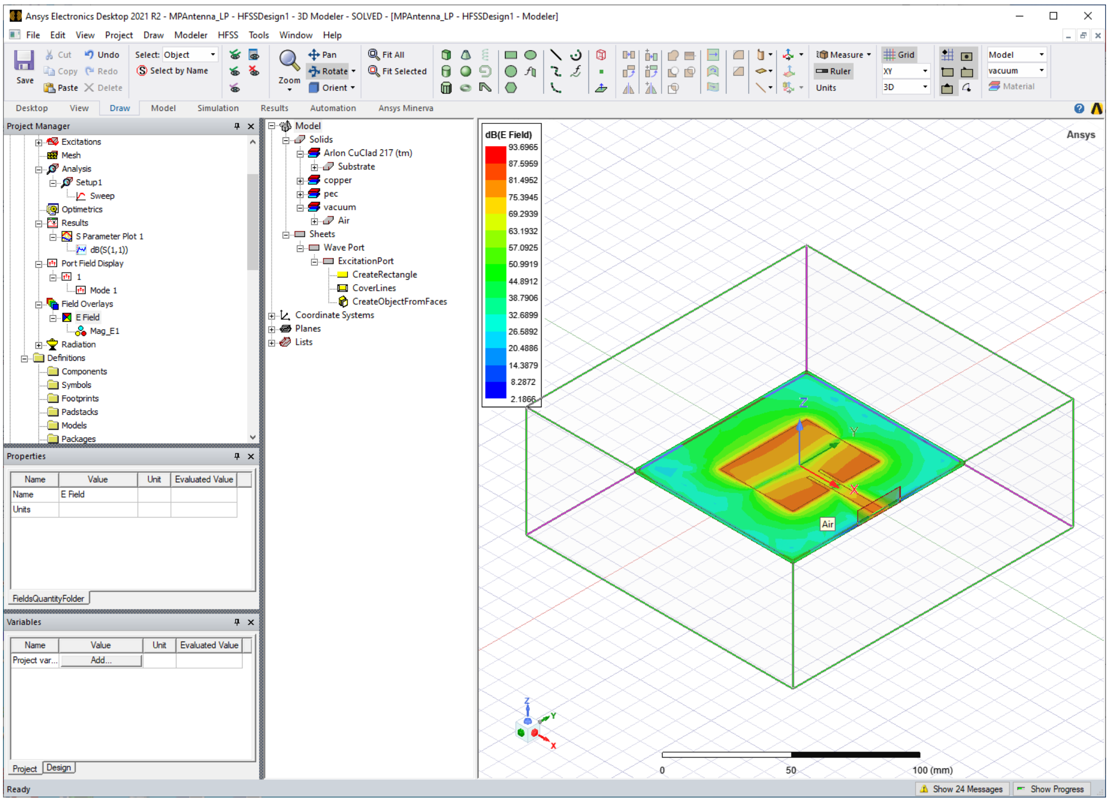
Select a component of the antenna in 3D modeller tree (e.g. a dielectric substrate) where you want to observe the distribution (later it can be changed) and evoke dialog related to field distribution.

HFSS=> Field Overlays => Plot Field => E => Mag_E

Set dialog as depicted below.



After that, the electric field distribution is obtained.



Note that:

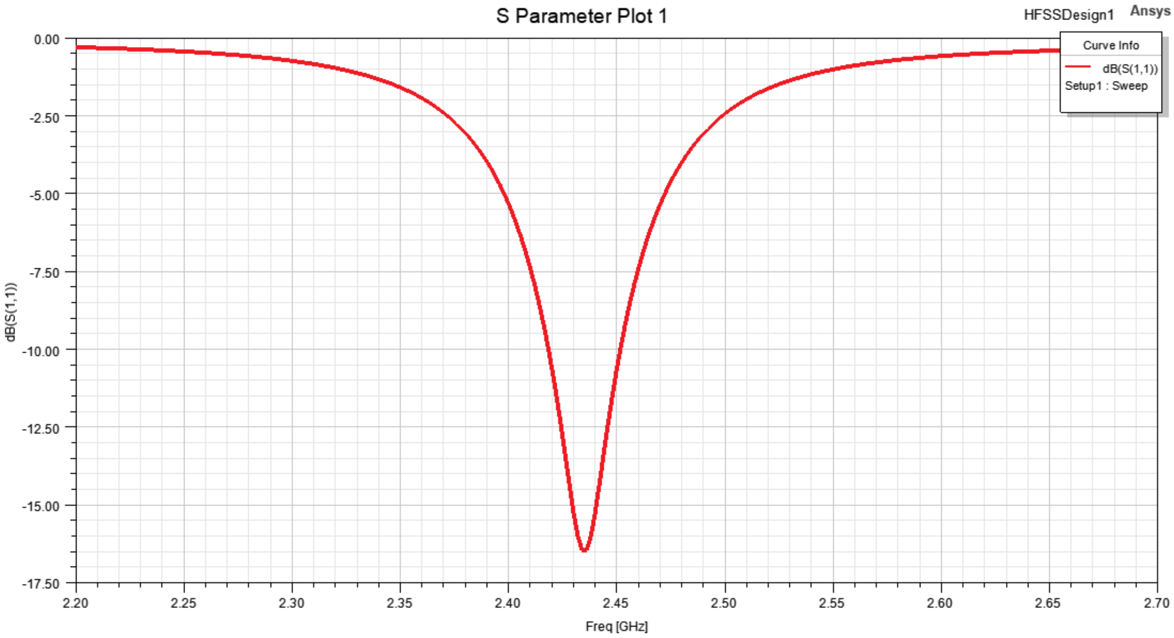
- *the scale of the field can be modified.*
  Right button =>E Field (in Project Manager) => Modify Attributes
- *the animation of the field can be run.*
  Right button =>Mag_E1 (in Project Manager) => Animate
- *the depicted field can be modified.*
  Right button =>Mag_E1 (in Project Manager) => Modify Plot

Fig 2.16. Showing electric field distribution on the antenna.

Further, to see antenna parameters, let's create the reports.

The reflection coefficient at the antenna input can be obtained by steps in Fig. 2.17.

HFSS => Results => Create Modal Solution Data Report => Rectangular Plot



If necessary, the values of the plot can be read by markers available by Right button.

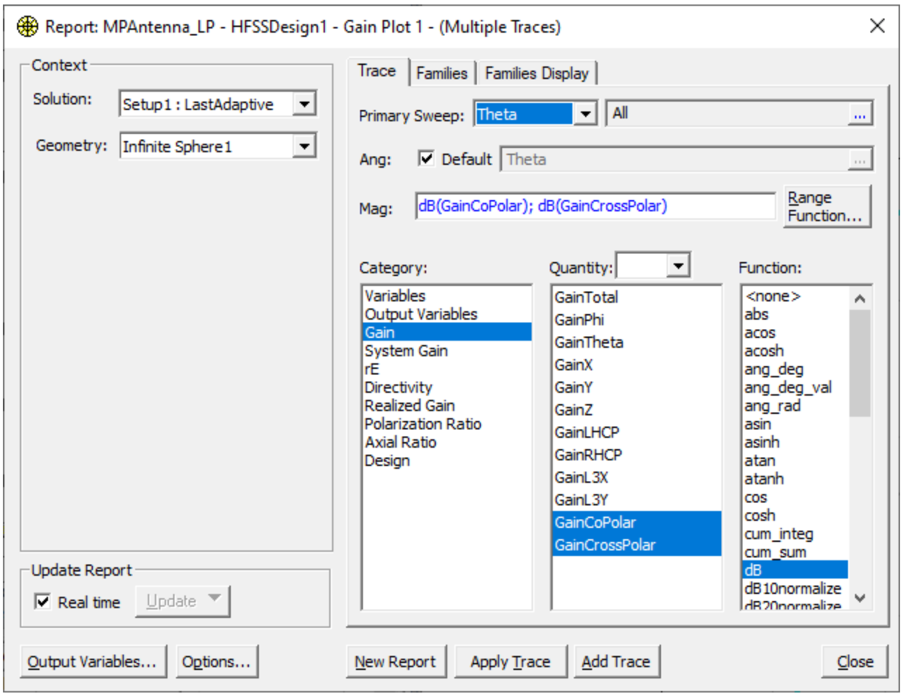
Fig 2.17. Plot of the reflection coefficient.

The radiation pattern in principal planes E/H planes can be obtained by the steps presented in Fig. 2.18. The available data and consequently depicted radiation patterns depend on the setting of the Far field radiation sphere setup (Fig. 2.10). If desired, after modification of that setup also a 3D radiation pattern can be obtained.

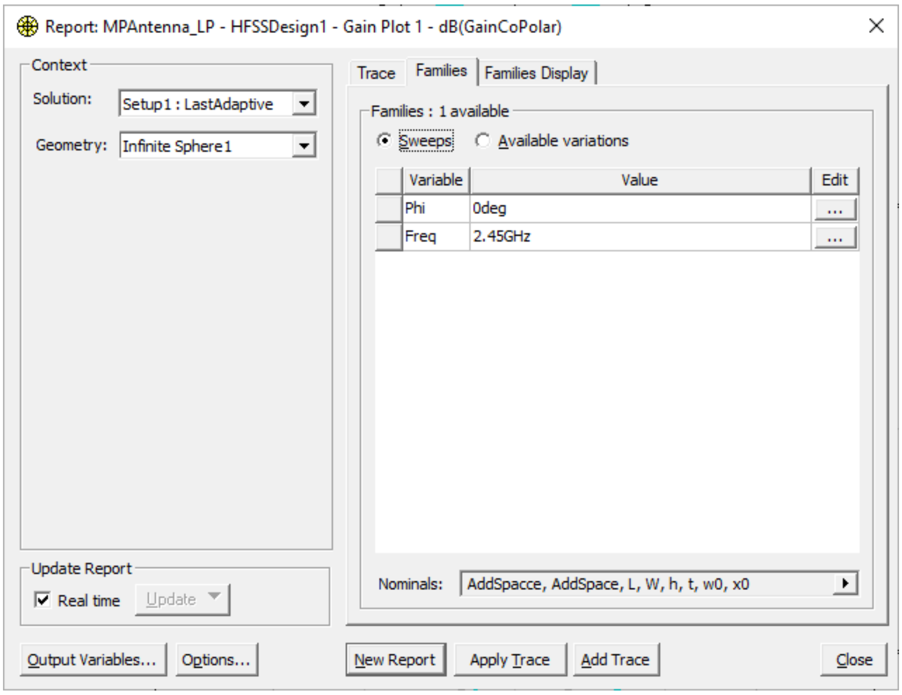
HFSS => Results => Create Far Fields Report => Radiation Pattern

Trace selection:

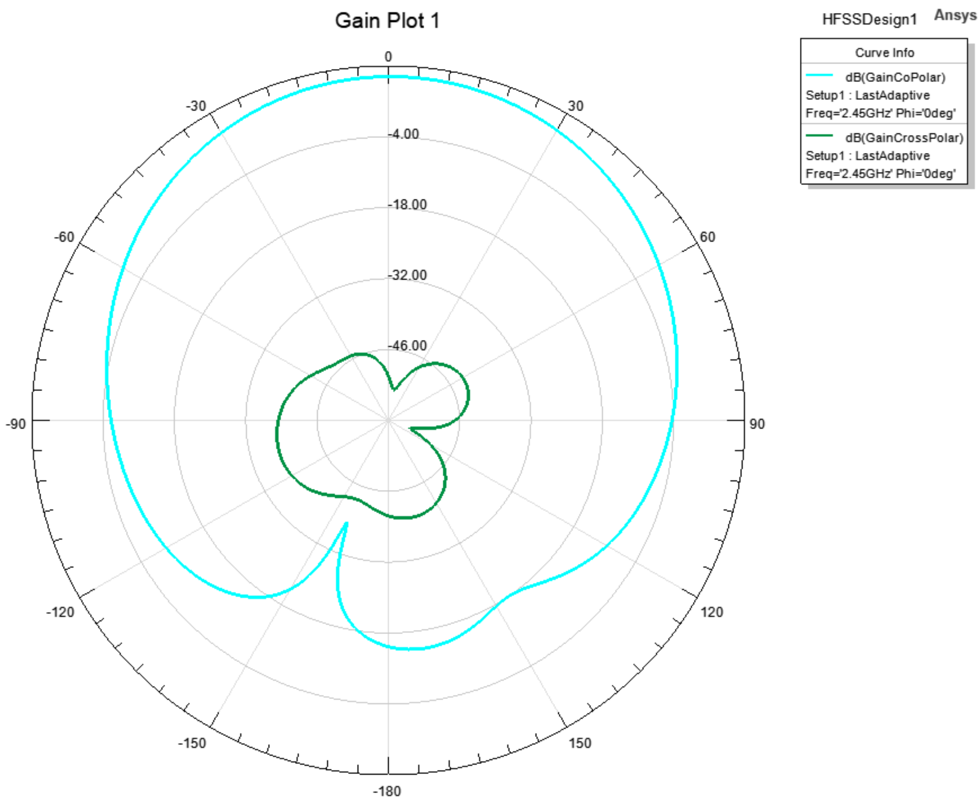
Select category: Directivity/Gain/Realized Gain, component (e.g. GainCoPolar, GainCrossPolar) and angles Theta = all.



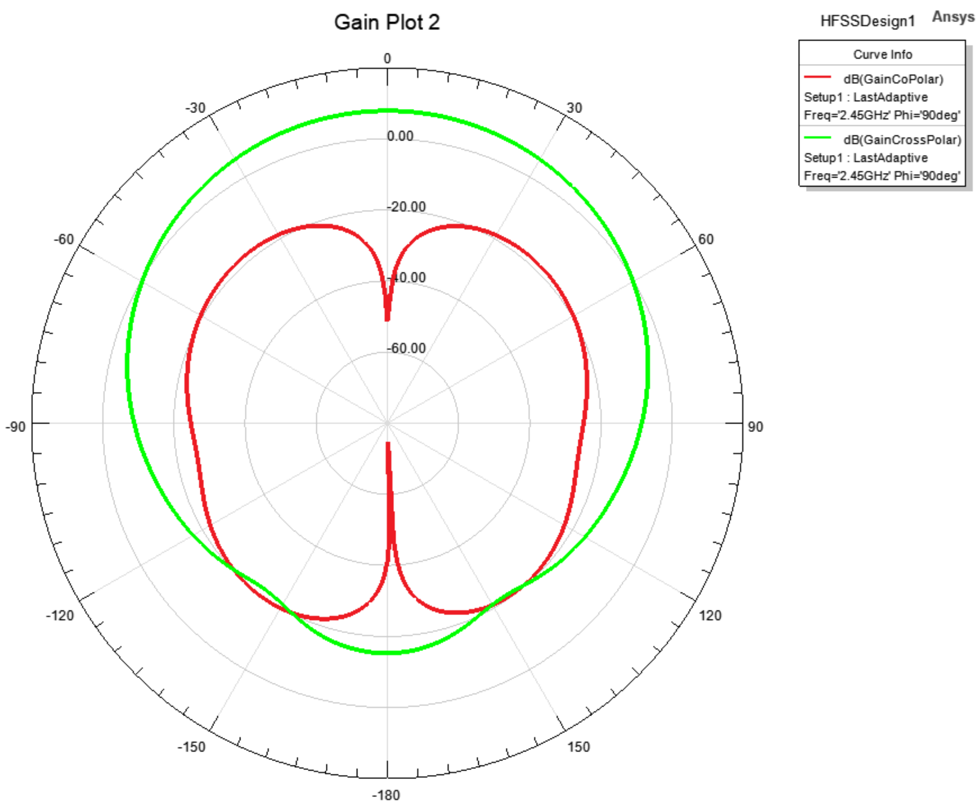
Families selection:



Select Phi=0 deg to obtain radiation pattern in E-Plane.



Select Phi=90 deg to obtain radiation pattern in H-Plane.



Note that if you want to depict a radiation pattern for a different frequency than for the adaptive (e.g. in this case 2.45 GHz), please change Solution to Setup1: Sweep, and than you can select available frequencies from the list Freq.

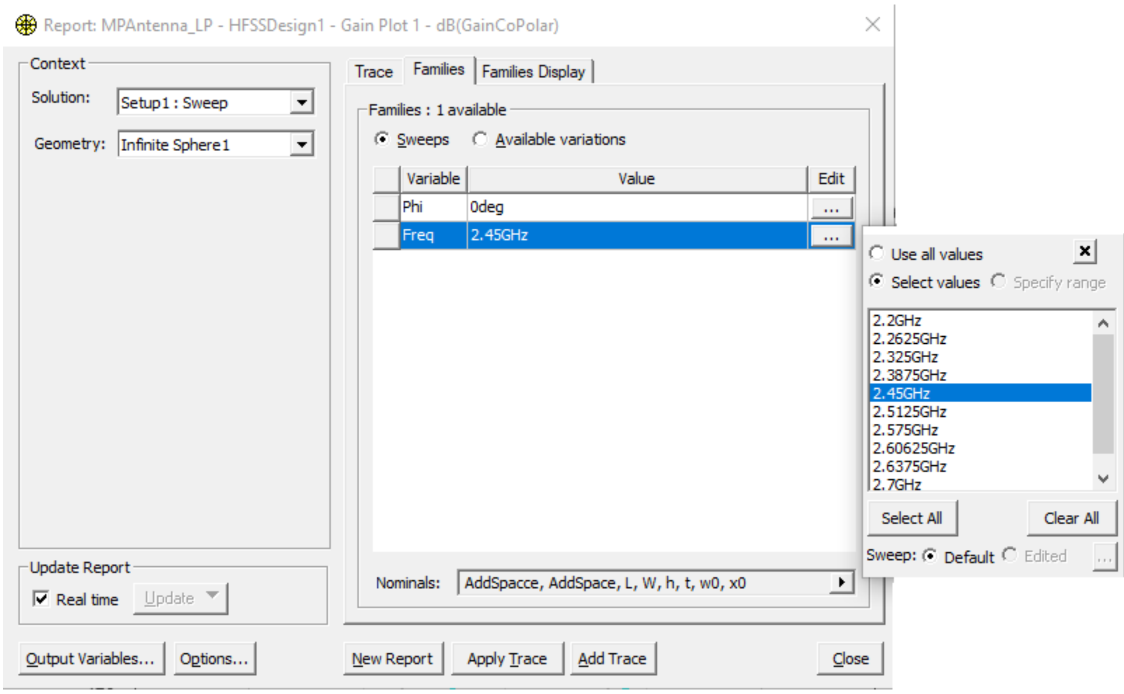


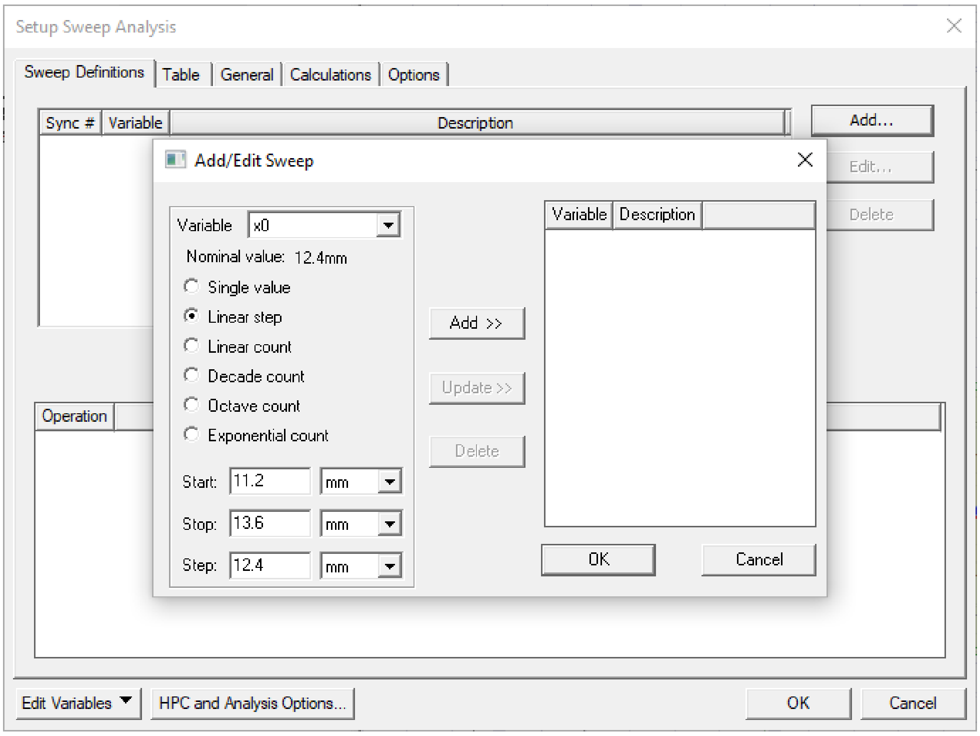
Fig. 2.18: Plotting radiation patterns.

In the rest of exercise, tune the antenna, so that the reflection coefficient is lower than -10 dB in the whole ISM band (if possible). Define the parameters of this antenna (the resonant frequency, the impedance bandwidth for $S_{11} < -10$ dB, the directivity, the gain, the radiation pattern, the antenna half-power beam width (given by a 3 dB decrease)).

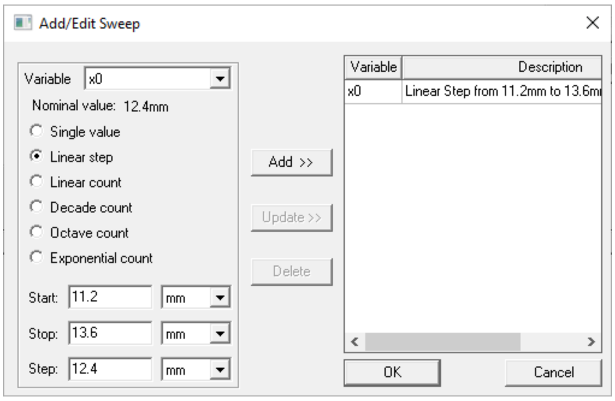
Note that for impedance matching/tuning the antenna, a desired antenna geometrical parameter – variable, can be swept (parametrized). To do that, look at Fig. 2.19.

Right button =>Optimetrics (in Project Manager) => Add => Parametric

Click the Add button and select the variable, and define a kind and range of sweeping.



Click the Add button.



After that, click by the right button on the created ParametricSetup1 (the item below Optimetrics) in the Project Manager and select the Analyse.

The selected calculated s-parameter responses can be depicted.

HFSS => Results => Create Modal Solution Data Report => Rectangular Plot

In the Families dialog, select for the swept variable desired values to plot responses.

Fig. 2.19: Parametric calculation of desired variable.

2.6 References

[1] *Ansys*. An Introduction to HFSS. Accessed on: 22-05-2022. Available at: https://ansyshelp.ansys.com/Views/Secured/Electronics/v212/en/PDFs/An%20Introduction%20to%20HFSS.pdf (Accessible from Ansys Help).