Electromagnetic High-Frequency, Full-Wave Tutorials for NX-Magnetics

Dr. Binde Ingenieure

January 2, 2024

THIS SIMULATION SOFTWARE USES FINITE ELEMENT METHODS. USERS SHOULD BE AWARE THAT RESULTS CAN HAVE UNPREDICTABLE ERRORS IF INPUT DATA IS NOT COMPLETELY CORRECT. THEREFORE ANY DESIGN DECISIONS SHOULD NOT BE BASED SOLELY ON THE SIMULATION. USE ADDITIONAL MEASUREMENTS TO ENSURE THE CORRECTNESS.

Contents

1 Tutorial 1 **3**
1.1 Shielding Analysis . . . 3

2 Tutorial 2 **4**
2.1 Waveguide Loaded Cavity (TEAM 18) . . . 4
2.1.1 Basic Model Setup, Fem File . . . 4
2.1.2 Port Definition, Sim File . . . 5
2.1.3 Solve and Postprocessing . . . 7

3 Tutorial 3 **9**
3.1 20 GHz Waveguide Combiner . . . 9
3.1.1 Waveguide Solution . . . 9
3.1.2 Meshing . . . 10
3.1.3 Finite Conductivity Boundary Condition . . . 11
3.1.4 Wave Ports . . . 12
3.1.5 Solve and Postprocessing Field Results . . . 14
3.1.6 S-Parameters and Frequency Sweep . . . 15

4 Tutorial 4 **17**
4.1 Resonance Modes Calculation . . . 17

5 Tutorial 5 **18**
5.1 Dipole Antenna . . . 18
5.1.1 Meshes and Physical Properties . . . 18
5.1.2 Sim File Entities . . . 19

6 Tutorial 6 **25**
6.1 Microstrip Patch Antenna . . . 25

7 Tutorial 7 **27**
7.1 PCB WiFi Antenna . . . 27

8 Tutorial 8 **29**
8.1 Vivaldi Antenna . . . 29

9 Summary of Wavetypes for Rectangular Guides **30**

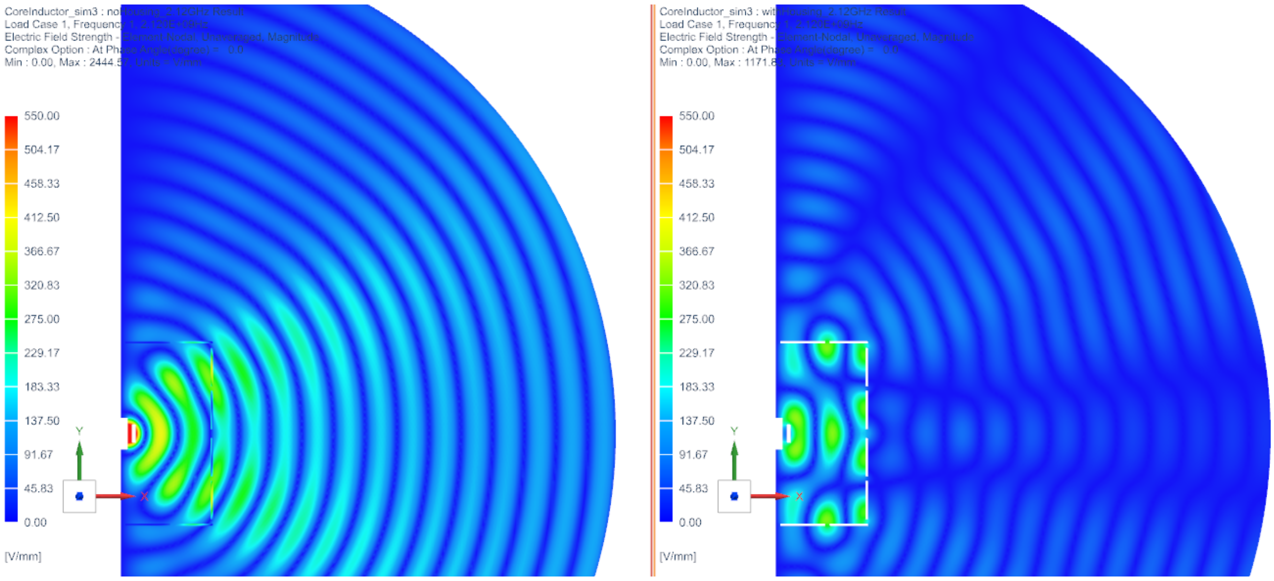
10 Literature **31**

1 Tutorial 1

1.1 Shielding Analysis

This example shows a already complete model of a electronic component with metal-housing that is ready to solve. The interested reader can check the model features and gain insights.

Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.8Shielding.zip`

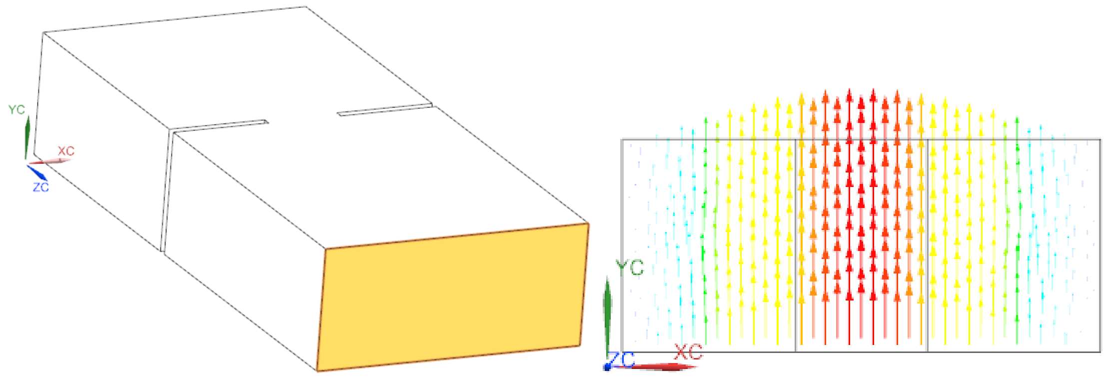


The following figure shows the electric field pattern at frequency 2.12 GHz. Left side: without housing, right side: with metal housing and with gaps in the housing. The shielding effect of the housing can easily be seen: Outside the housing, the field is much smaller.

2 Tutorial 2

2.1 Waveguide Loaded Cavity (TEAM 18)

Goal of this Team benchmark example is to find the resonance frequency of a waveguide cavity. The highlighted face shall be applied by a transversal electric wave mode of type TE10 as shown in the right side of the picture. A full wave solution is necessary to analyze for resonance of such electromagnetic waves. The original task description of Team 18 can be found at [Team18task].



What you learn in this example:

- Performing a 3D Full Wave analysis for Waveguides in frequency domain.
- Learn how to define the basic settings for wave ports.

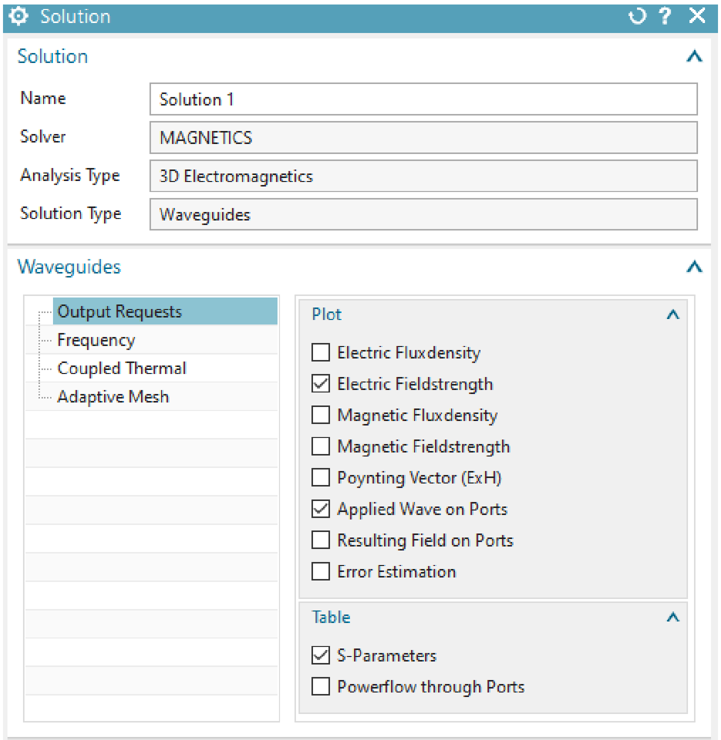
Estimated time for the example: 0.5 h.

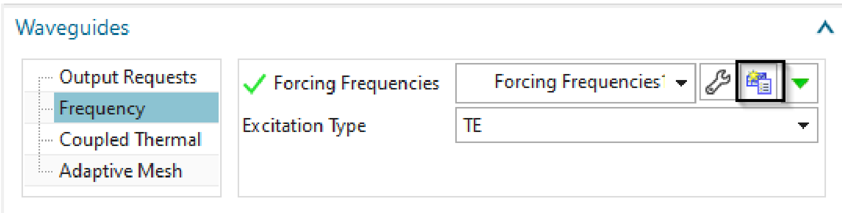
2.1.1 Basic Model Setup, Fem File

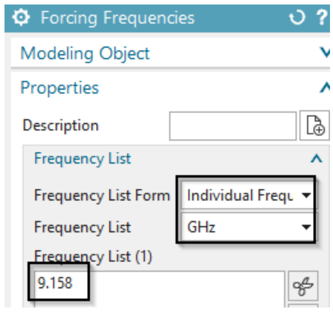
For the setup of this model, follow these steps:

1. Download the model files for this tutorial from the following link:
   `https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.1Team18.zip`
2. Open the part file Team18.prt.
3. Start the Pre/Post application.
4. Create New FEM and Simulation.
5. Choose Solver MAGNETICS and
   - Analysis Type '3D Electromagnetics',
   - switch off idealized part.
   - Select Solution Type 'Waveguides'.
   - In register 'Output Requests', 'Plot' set 'Electric Fieldstrength' and 'Magnetic Fieldstrength' on.
   - In register 'Frequency' at "Forcing Frequency', create a modeling object for the used frequency with 9.158 GHz (This is already the found resonance frequency). Alternatively, we could run a sweep over a frequency range to search for resonances.

- in box ’Table’ activate ’S-Parameters’.







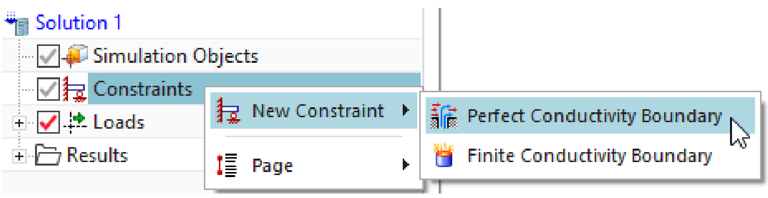
6. Switch to the Fem file

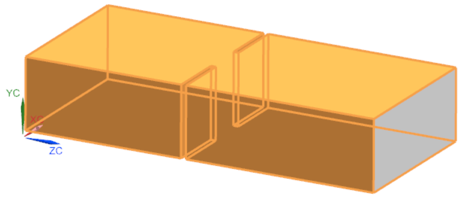
   - Blank the polygon body ‘Housing’. This body is not needed in the analysis.
   - Mesh the air volume using tets. Use the suggested element size divided by 4. (4.63/4 mm)
   - Assign material ’Air’ to it.

2.1.2 Port Definition, Sim File

1. Switch to the Sim file.

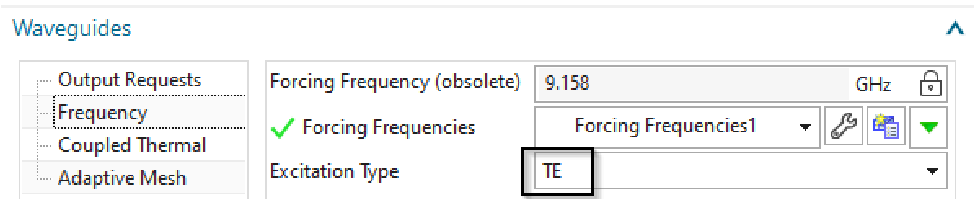
2. Create a constraint of type ‘Perfect Conductivity Boundary’ to all faces but the entrance face for the incoming wave.



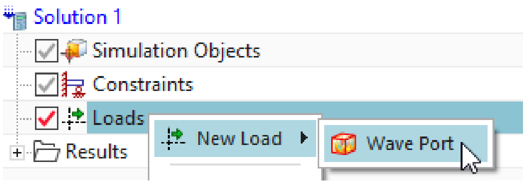


3. Next we are defining the incoming wave. We must define a TE10 wave. TE means that the electric field vector in Z direction must be zero. The opposite is true for TM waves; Here the magnetic field vector is zero in Z. For both TE and TM the first index defines the number of maxima in X direction and the second index those in Y direction. See chapter 'Summary of wavetypes for rectangular guides' for further information.

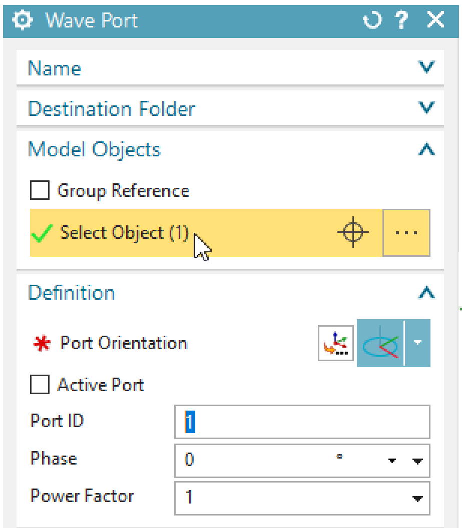
   - To set the type to TE, edit the solution parameters.
   - In register Frequency Domain set the 'Excitation Type' to 'TE'. (Since this is the default, there is nothing to do.)

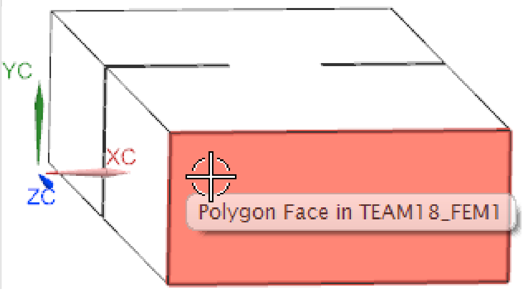
   

   - Create a load of type 'Wave Port'

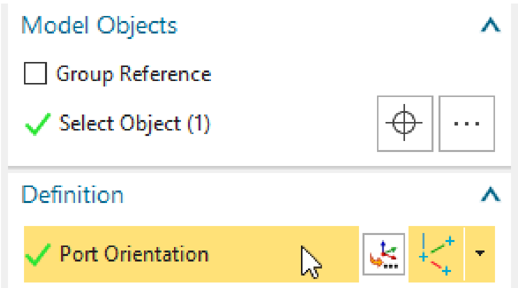
   - At 'Model Objects', select the entrance face,

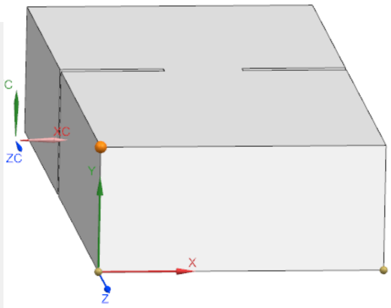
   

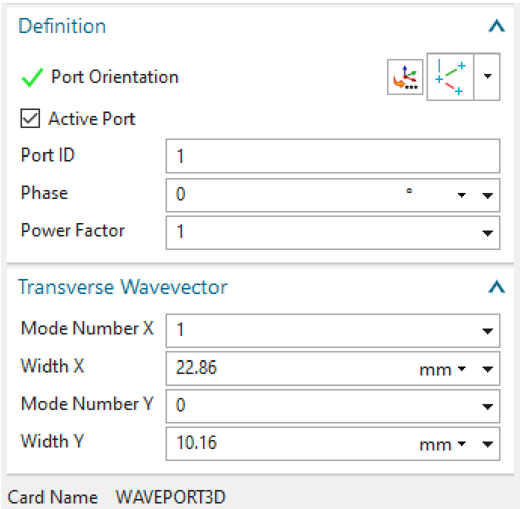
   - At 'Port Orientation', use the 'CSYS Dialog' (or any other) to define a coordinate system. Define the center at one face corner, the x direction along one edge and the y-direction along a perpendicular edge of the face.
     Hints: The CSYS center defines the beginning position of the incoming wave. This

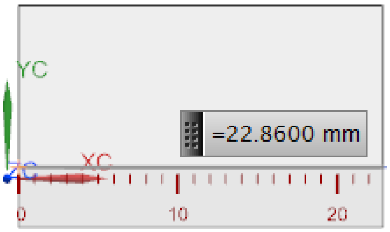
position plus the 'Width X' defines the ending position. Between start and end the wave can have a number of maximums. This number of maximums will be defined by the 'Mode Number X'. The corresponding applies for Y.





- Toggle on 'Active Port', leave 'Port ID' as 1 and 'Phase' as 0.
- At 'Transverse Wavevector': Set 'Mode Number X' to 1 and 'Mode Number Y' to 0. This setting describes the characteristic of the desired TE10 wave. For 'Width X' and 'Width Y' key in the full width of the CAD face: In X direction use 22.86 mm and for Y use 10.16 mm. Of course, its also possible to use the graphical measuring feature.

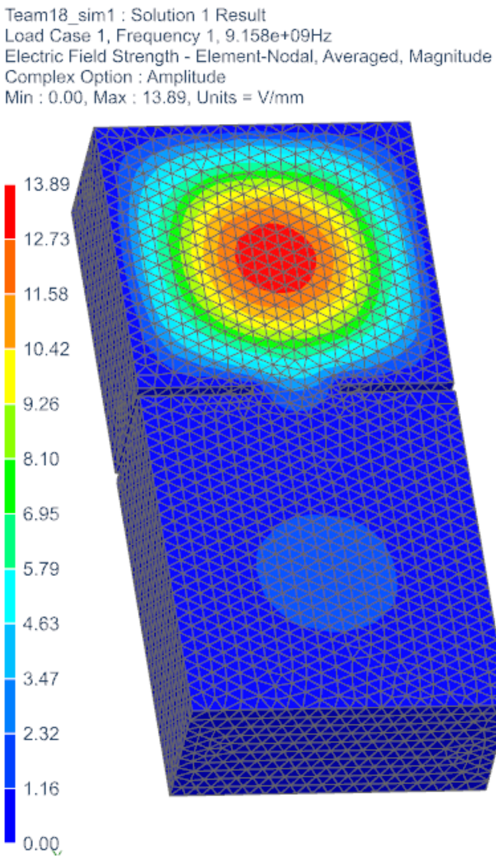


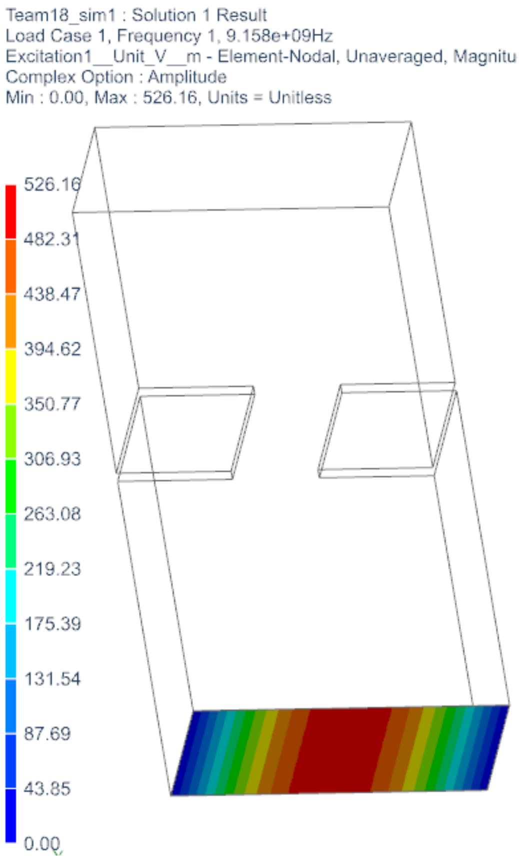


- Click OK to finish the dialogue.

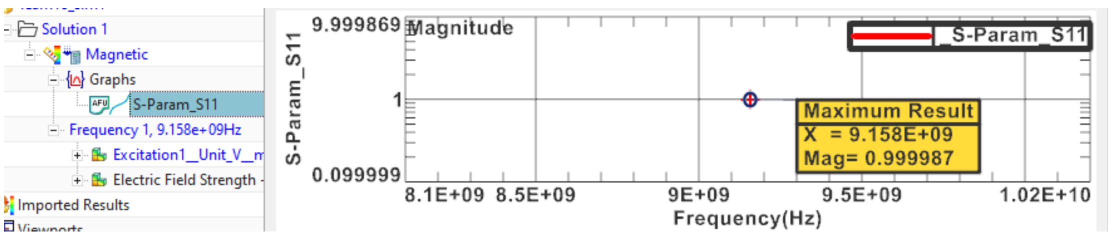
2.1.3 Solve and Postprocessing

1. Solve the solution
2. Postprocessing:
    - Display the electric field strength. Blank the 2D meshes under Post View. Check the pattern of the wave. The resulting field should show a standing wave as seen in the below picture. The maximum value of electric field strength can vary depending on how closely you find the resonance frequency.





- Display the 'Excitation' result. (Picture above right.) This shows the defined TE10 wave at the port face. Check the pattern.
- Check the calculated S-Parameter S11. It is very near to 1.

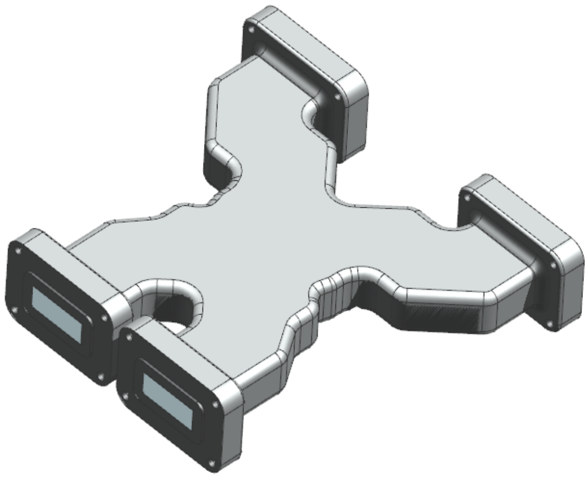


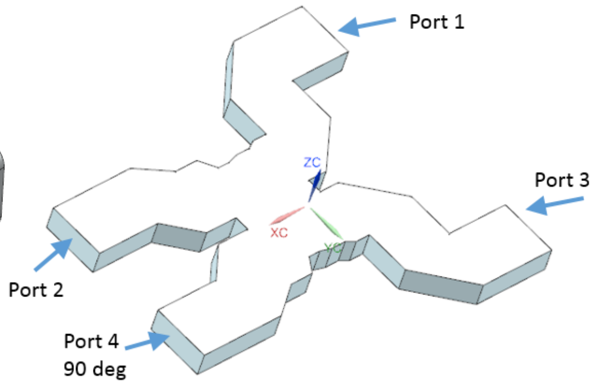
3. The tutorial is finished.

3 Tutorial 3

3.1 20 GHz Waveguide Combiner

The waveguide combiner shown in this exercise is used to combine the output power of two 20 GHz power amplifiers. The two outputs of the amplifiers are fed into ports 2 and 4 of the waveguide with a 90 degree phase change. The two other Ports 1 and 3 are output ports. The phase change between the input ports leads to an absorption of the wave in one of the output ports. The problem is also described and analyzed in [Arcioni].





What you learn in this example:

- Performing a Waveguide analysis in frequency domain,
- Use of Phase changed port signals,
- Use Finite Conductivity Boundary Condition.
- Calculate s-Parameters.

Estimated time for the example: 1 h. Follow the steps:

3.1.1 Waveguide Solution

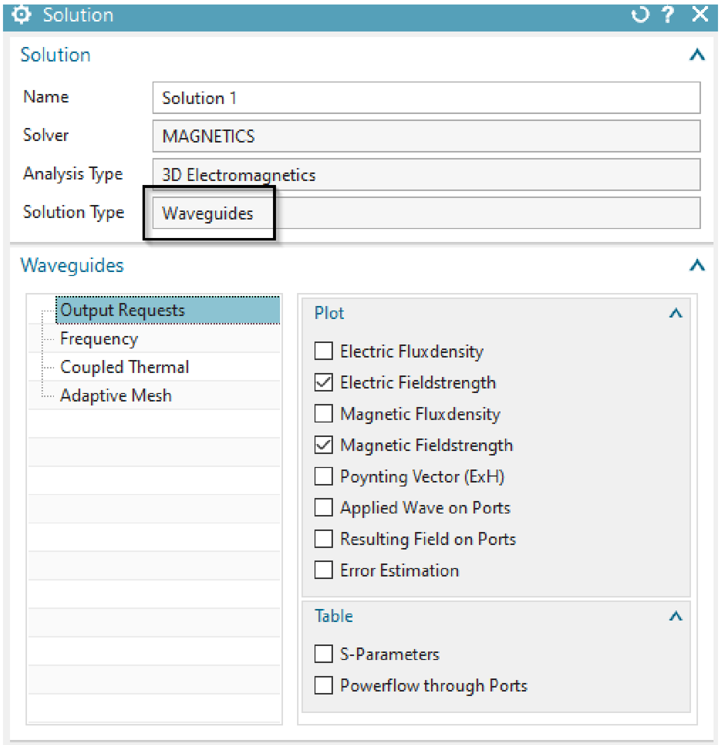
We first create the simulation file structure and a solution for this waveguide problem.

1. Download the model files for this tutorial from the following link:
   `https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.2WaveguideCombiner.zip`
2. Open the part file WaveguideCombiner.prt.
3. Start the application Pre/Post.
4. Create New FEM and Simulation.
5. Select the Solver MAGNETICS and
6. Analysis Type 3D Electromagnetics.

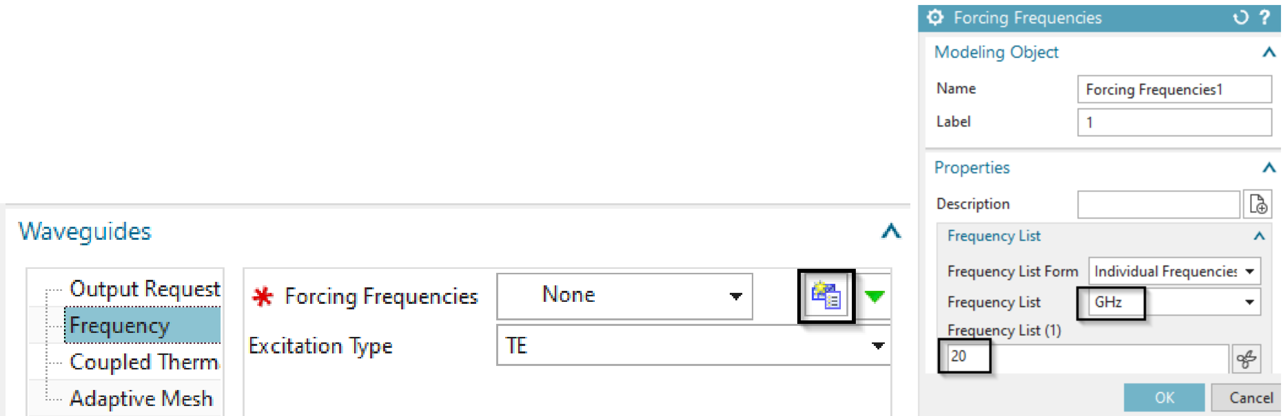
7. Switch off idealized part.

8. Select the Solution Type 'Waveguides'.

9. In register 'Output Requests', 'Plot' set 'Electric Fieldstrength' and 'Magnetic Fieldstrength' on. In a later step we will activate 'S-Parameters'.



10. In register 'Frequency Domain' create a 'Modeling Object' for the forced frequency and set the 'Forcing Frequency' to 20 GHz. Accept the default 'Excitation Type' 'TE' since we want to apply this type of wave. Click 'OK' to finish the solution menu.

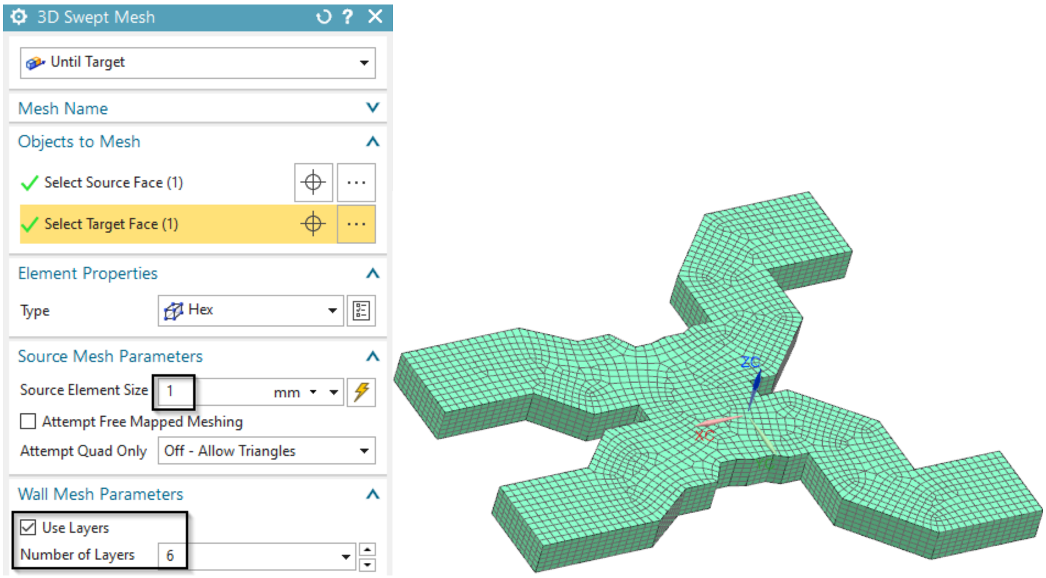


3.1.2 Meshing

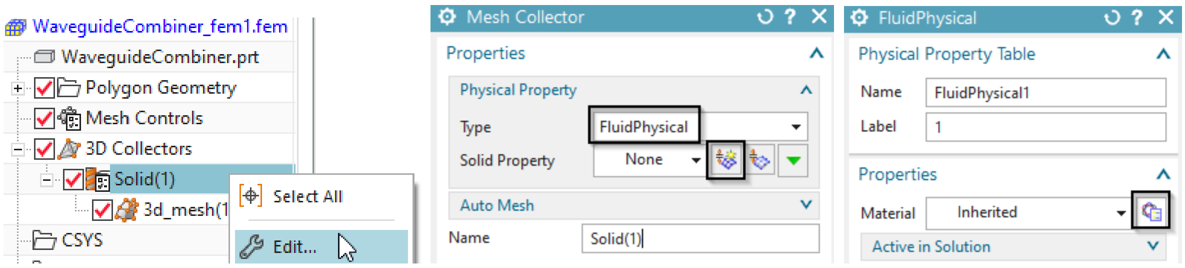
Switch to the Fem file.

1. Blank the housing polygon body.

2. Mesh the air volume using hex elements. Use an element size of 1 mm and 6 layers over the thickness. Of course, tetrahedral elements would also work.



3. Assign a 'Physical Property' of type 'FluidPhysical' and select the material 'Air' from the Magnetics material library.

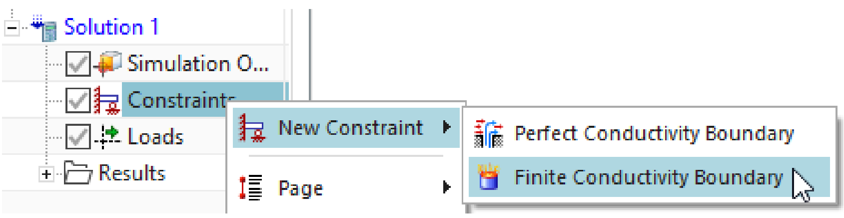


3.1.3 Finite Conductivity Boundary Condition

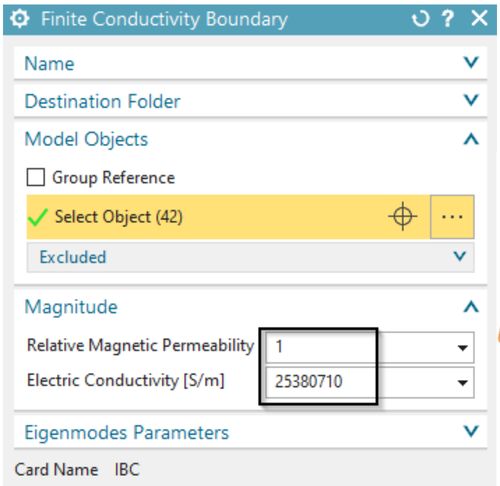
We create a condition on the boundary that represents the electric conductive border material.

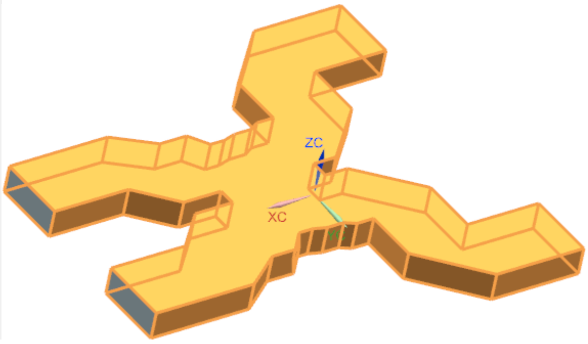
1. Switch to the Sim file.

2. Create a constraint of type 'Finite Conductivity Boundary'. Select all faces but the 4 port faces.



3. Choose the material properties as shown in the picture. These describe a aluminium type material. The thickness of this material layer will be computed by the skin depth. That skin depth results from the given electric conductivity and the applied enforced frequency.





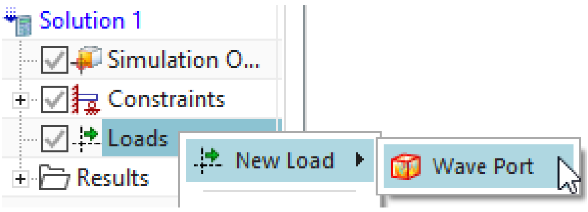
4. Click 'OK' to create the constraint.

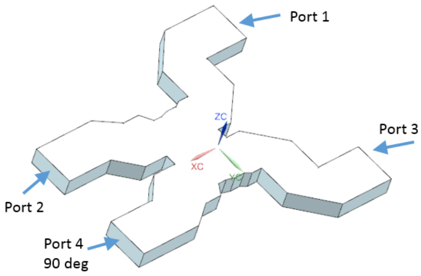
3.1.4 Wave Ports

Now we create the ports at which signals are applied. These correspond to the prior picture, that is again shown here below right.

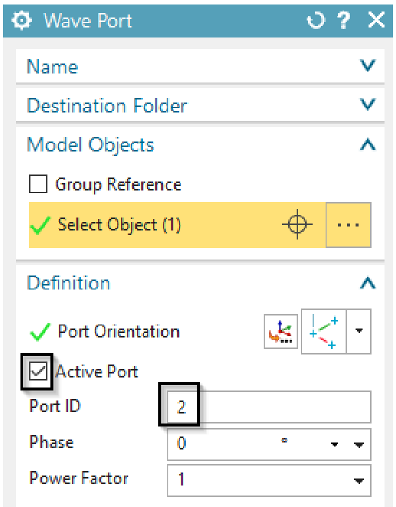
1. Define the active Wave Port number 2:

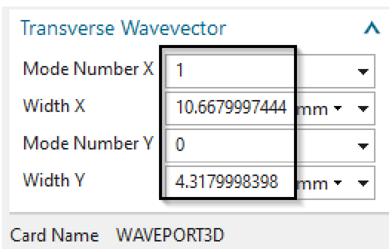
   - Create a load of type 'Wave Port'.

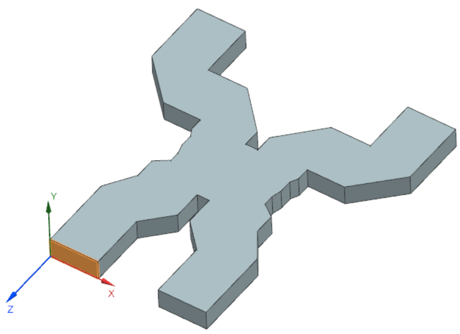




   - Select the face of port 2 and define the Port Orientation as shown in the below picture. Name this load Port(2)_active. Assign the settings as shown in the below dialog.



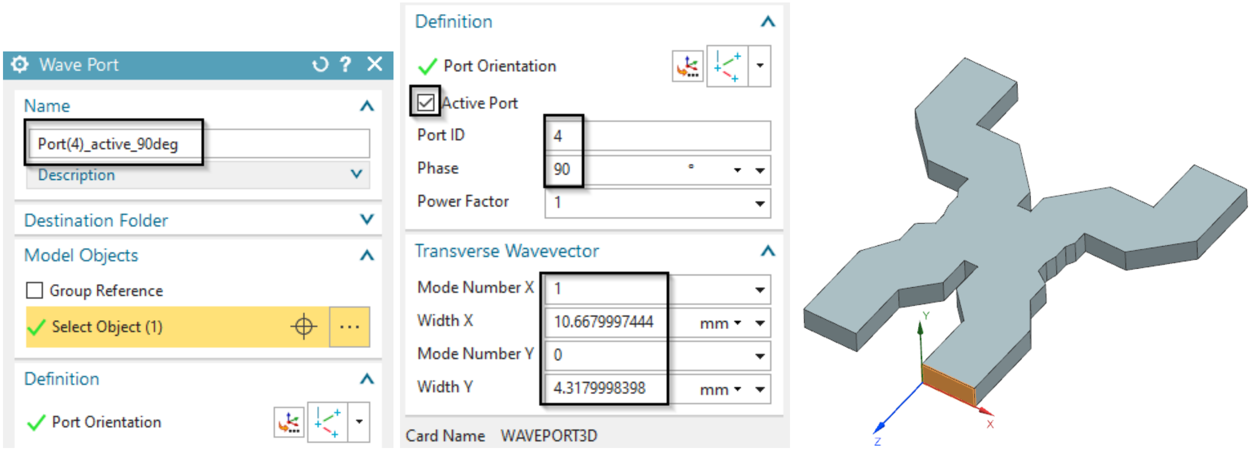




   - Click 'OK'.

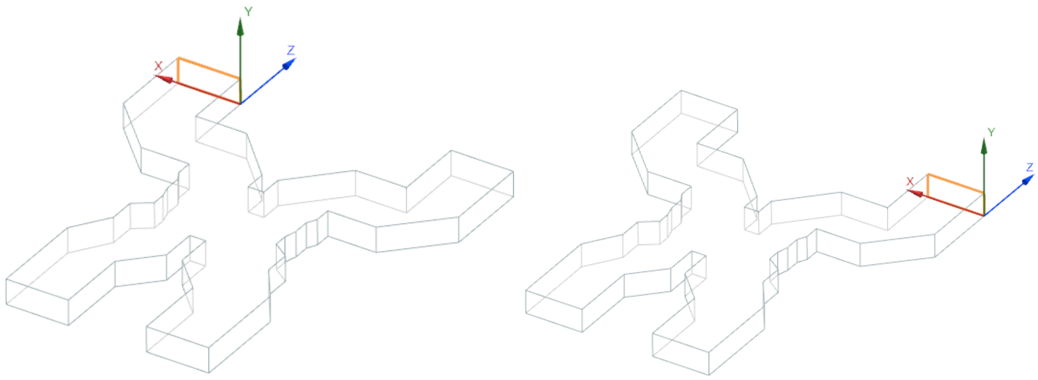
2. Define Wave Port 4

- Create a load of type 'Wave Port'.
- Select the face of port 4 and define the port orientation as shown in the picture.
- Name the load 'Wave Port(4)_active_90deg'
- Assign the similar settings as for port 2 but set the 'Phase' to 90 deg and the 'Port ID' to 4.

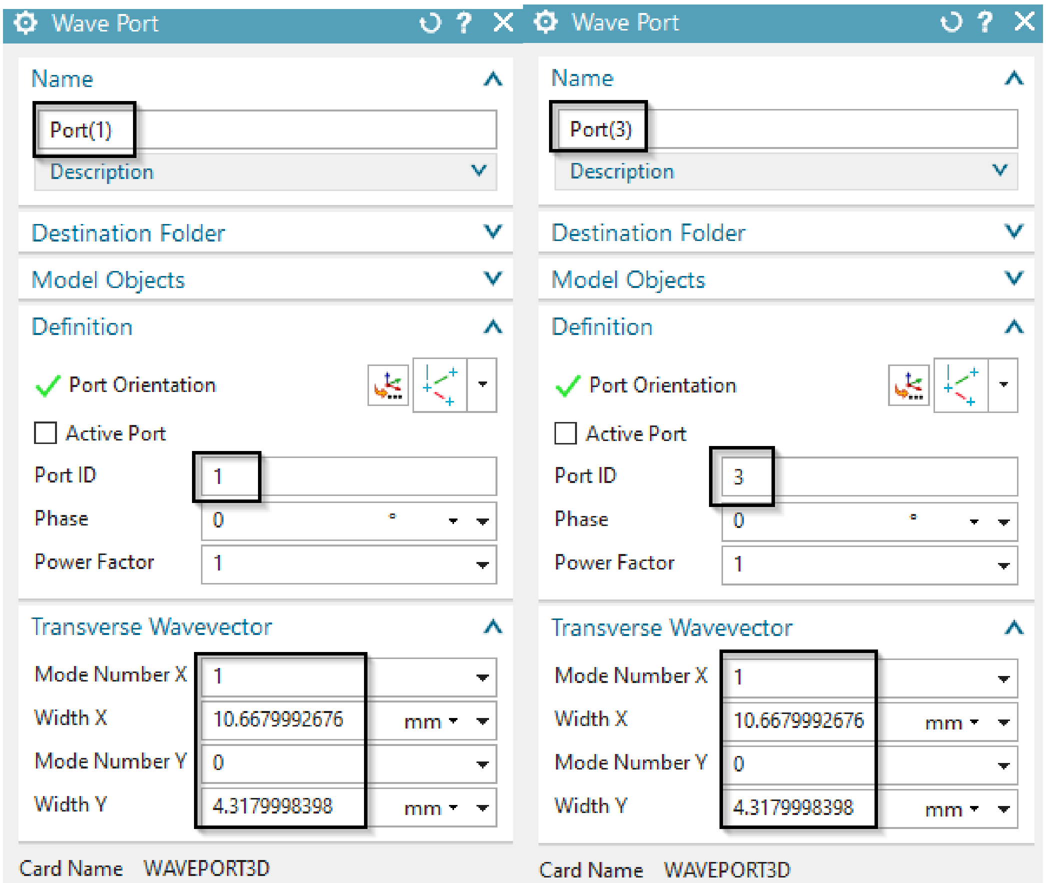


3. Define Wave Ports 1 and 3

- Create again a load of type 'Wave Port' (this is done two times now, for the remaining ports 1 and 3. These two get quite similar input).
- Select the face of port 1 (picture below left) and port orientations as done already before. Do the same for port 3 (picture below right).

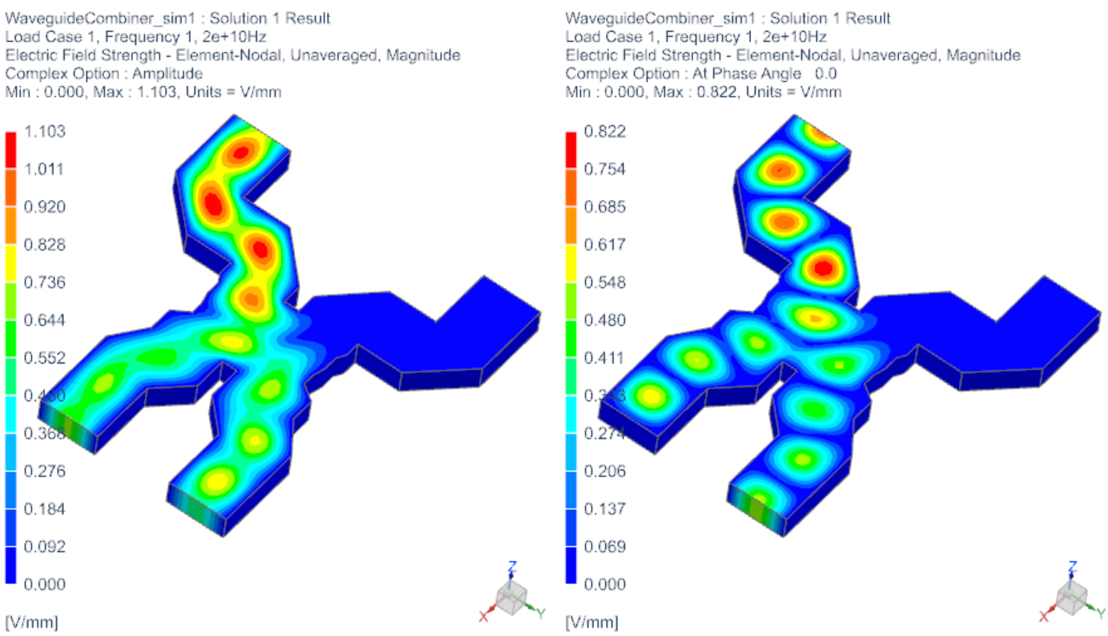


- Name the loads Port(1) and the second one Port(3).
- Assign the shown settings (below picture, left port 1, right port 3).

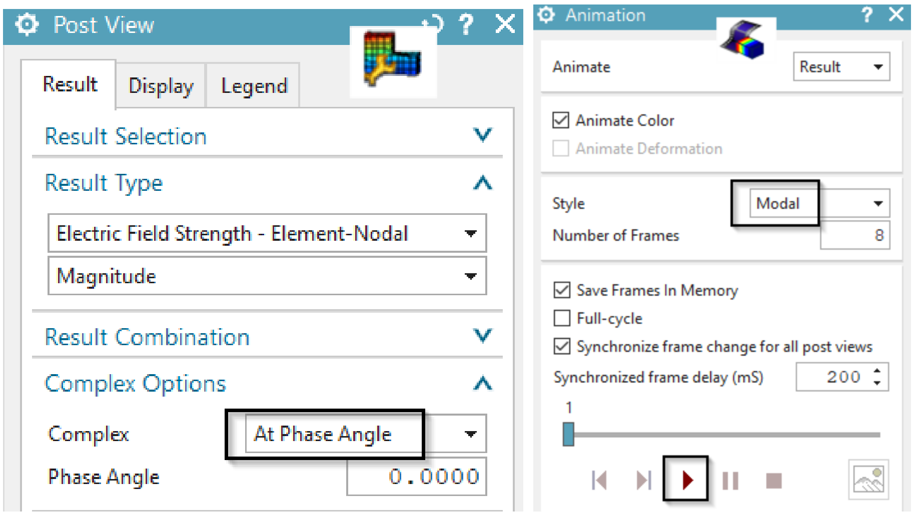


3.1.5 Solve and Postprocessing Field Results

1. Solve the solution. Open the plot results and hide the 2D meshes in 'Post View'.
2. After solve has finished, display the Electric Fieldstrength. The following picture shows left the 'Amplitude' and right the 'Signed Amplitude' at phase 0 deg (explained below) what means the real part.



3. Animations can be used to show the complex result converted to time domain. To do so you click 'Edit Post View', register 'Result'. Set the option 'Complex' to 'At Phase Angle'. Then go to 'Animation' and set the 'Style' to 'Modal' and click 'Play'. The animation runs over the phase angle now and that represents the time domain behaviour of this frequency result.

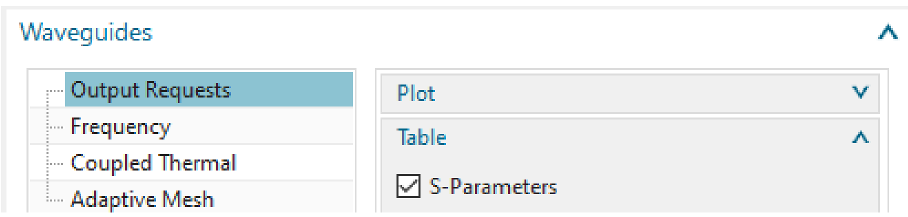


3.1.6 S-Parameters and Frequency Sweep

In a following solution we want to solve for the S-parameters. Also we want to perform a sweep over a frequency range to find out how the S-parameters behave at different excitation frequencies. For S-parameter calculation there must be only one active port in a solution and the results will show how the other ports interact with the active one. To find the complete matrix of all interactions, one had to run several solutions, each with another port being active.

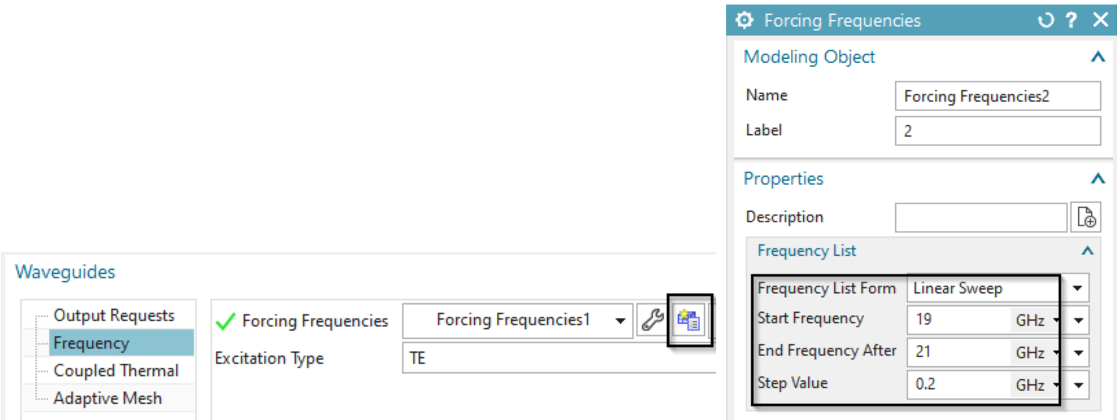
1. Prepare the model for S-parameter calculation:

   - Clone the existing solution and rename the new one to 'SparamP2active'. In this solution we will have only port 2 active.
   - Modify the wave port 4 and set it to inactive. (Maybe create a new wave port for that or clone the old one. Otherwise you will overwrite it.)
   - Edit the solution. In register 'Output Requests', 'Table', activate 'S-Parameters'.

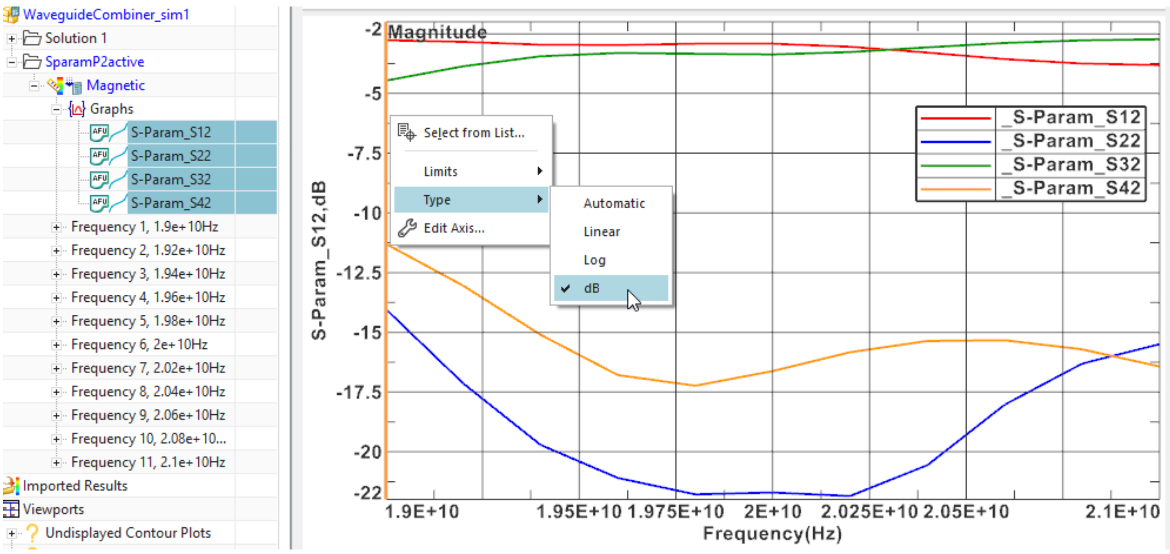


2. Activate Frequency Sweep:

   - Edit the solution. In register 'Frequency', create a new 'Modeling Object' for 'Forcing Frequencies'. Set the list form to 'Linear Sweep' and key in the values for start, end and step as shown in the below picture.



3. Solve the new solution 'SparamP2active'.
4. After solve has finished, open the tabular graph results and display the S-Parameters over Frequency. Set the Y axis type to 'dB' to display the parameters in the usual way.



The tutorial is complete. Save your files and close them.

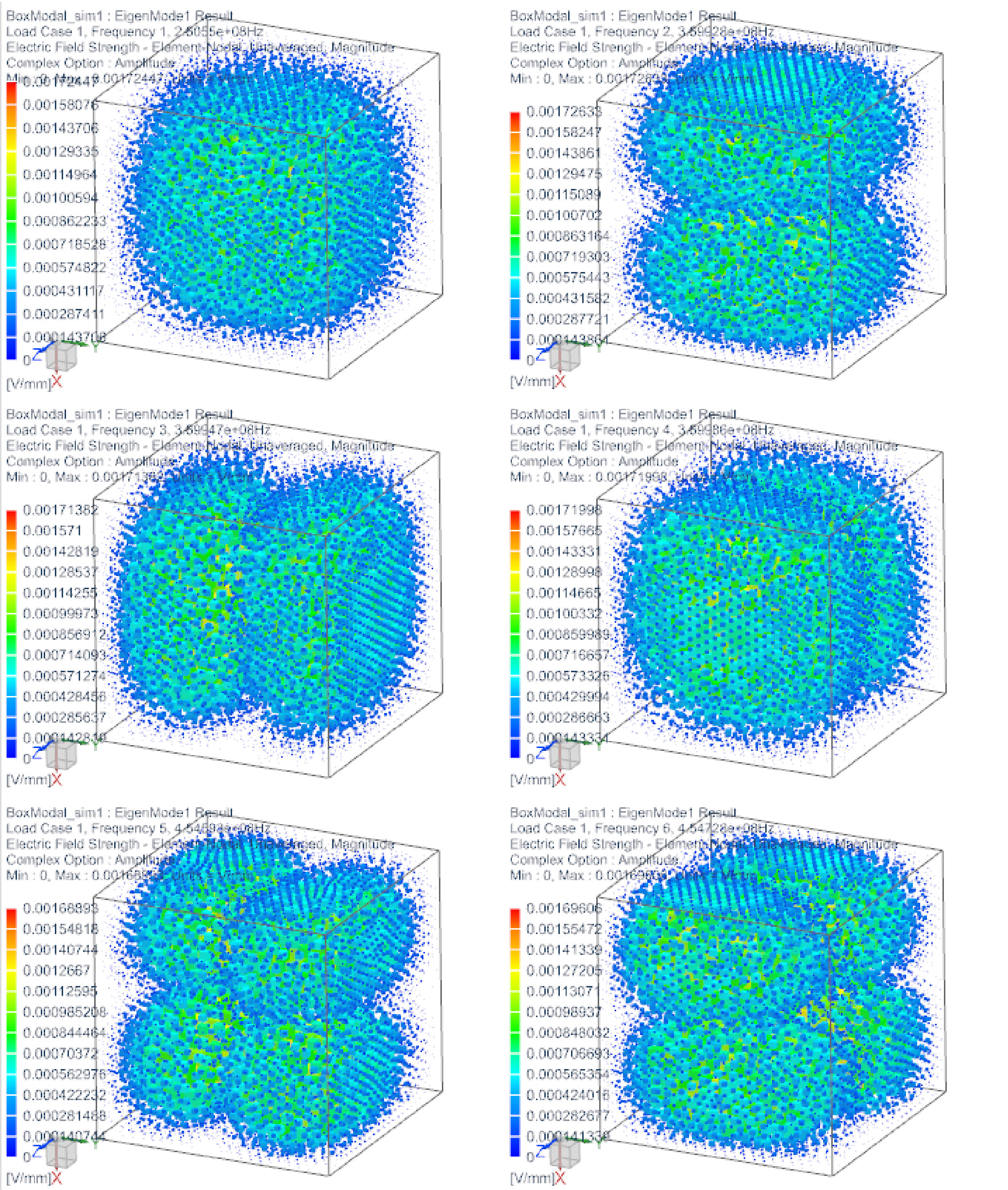
4 Tutorial 4

4.1 Resonance Modes Calculation

In this example it is shown how to calculate the electromagnetic resonance modes (Eigenmodes) of a rectangular box with 1 meter edge length.

Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.4Eigenmodes.zip`

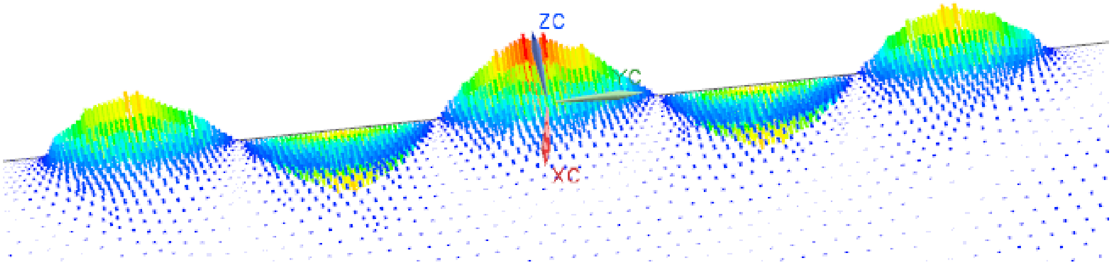
The following figures show the first 9 eigenmodes of the electric field.



5 Tutorial 5

5.1 Dipole Antenna

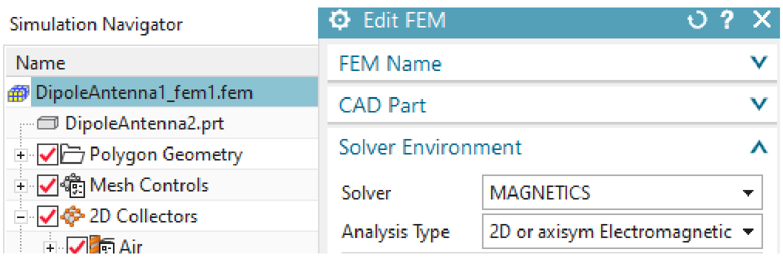
In this tutorial a dipole antenna is analyzed for the field patterns and for impedance and reflection coefficient. This antenna can be modeled in 2D axisymmetric or in 3D. The setup process is very similar. Following the 2D case is shown but differences for 3D there are described as hints. We walk through an existing model to explain and check the included features.



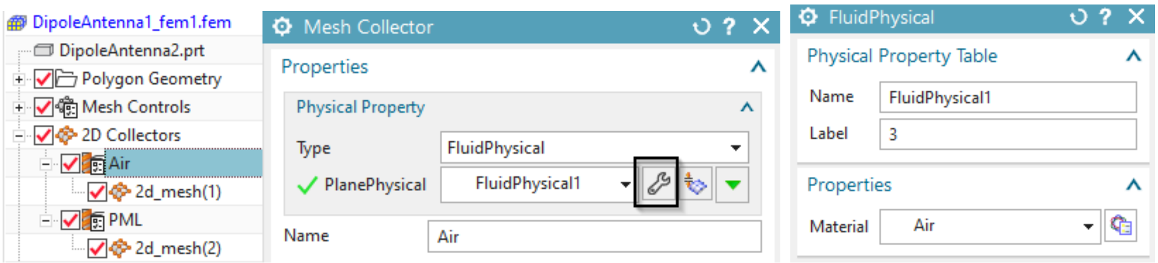
- Download the model files for this tutorial from the following link:
  `https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.3DipoleAntenna.zip`
- Open the file DipoleAntenna1_sim1.sim. (For 3D you would open DipoleAntenna2_sim1.sim). The file already contains a complete mesh and simulation.

5.1.1 Meshes and Physical Properties

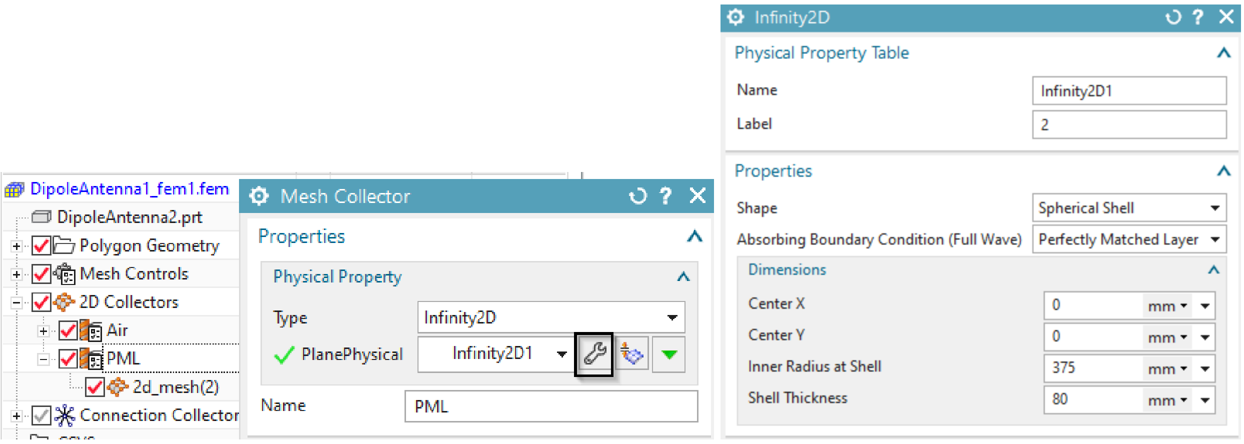
1. Change the displayed part to the Fem file.
2. Check the solver setting (right mouse button on the Fem file, Edit...). Notice, the Solver is set to 'MAGNETICS' and the 'Analysis Type' is '2D or axisym Electromagnetics'. (For the 3D case the 'Analysis Type' is 3D) Click 'OK'.



3. Edit the mesh collector 'Air'. Check that it contains a usual physical of type 'FluidPhysical' with material 'Air'.



4. Next edit the mesh collector named 'PML'. It contains a border mesh around the air and a physical to describes a 'Perfectly Matched Layer' (PML). Such PML technology is used to damp any reflection of EM waves at the outside border of the air mesh. They model a material layer, that is perfectly adapted to a given forcing frequency. If no PML would be used, all outgoing waves would reflect at the border and come back again. For the PML a physical of type 'Infinity2D' is used. (For 3D this is Infinity3D). Check its properties as shown in the following picture.

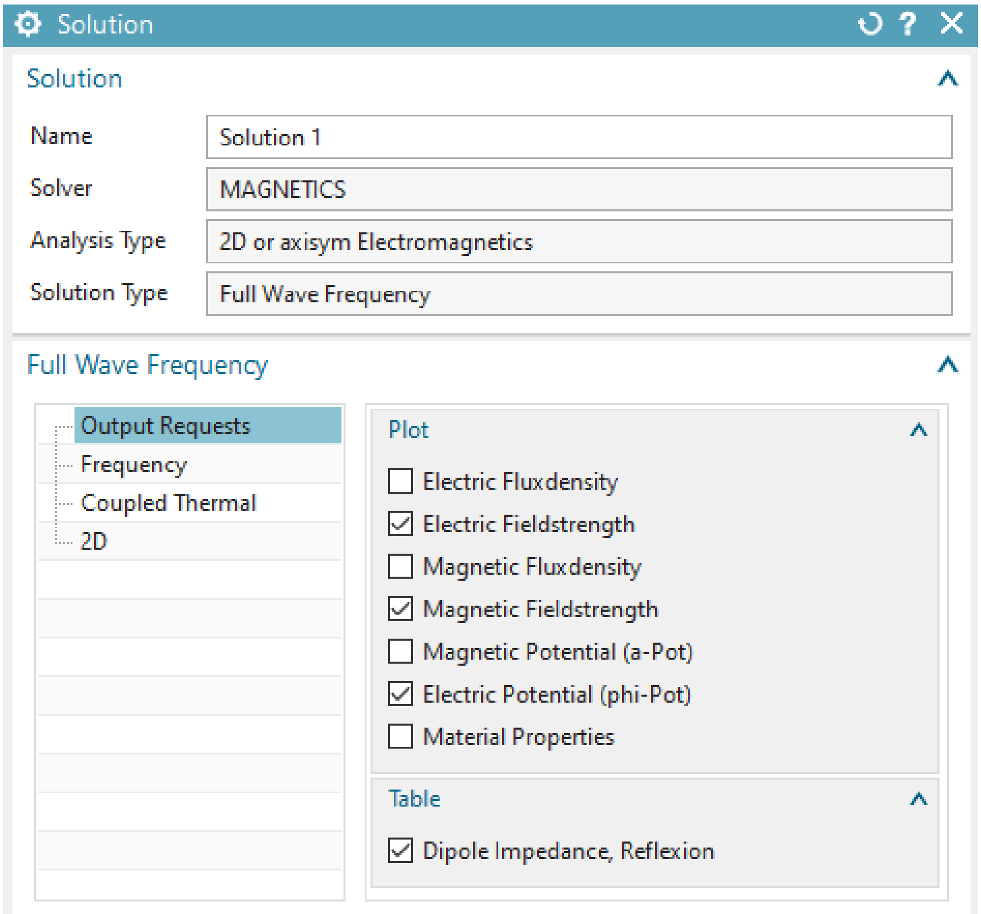


5. The PLM physical contains the following data.

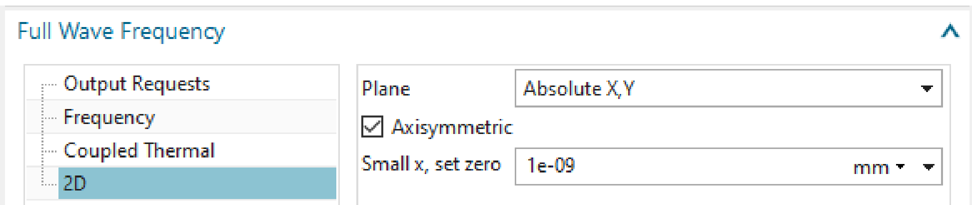
   - 'Shape': This defines the geometry type of the border mesh. It can be 'Spherical Shell' or 'Rectangular Shell'.
   - The 'Center' coordinates of the sphere or rectangle.
   - The 'Inner Radius at Shell' and the 'Shell Thickness'.

5.1.2 Sim File Entities

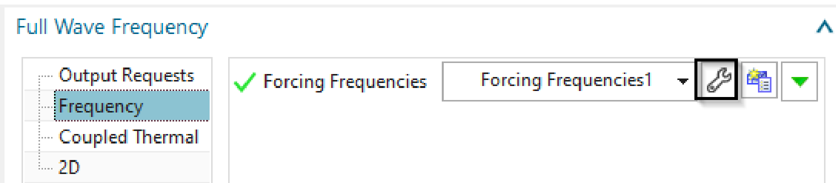
1. Set the displayed part to the Sim file.
2. Edit and check the existing solution 'Solution 1'. Notice, it is of type 'Full Wave Frequency'. Set the settings for the solution as shown in the following pictures.
3. Check register 'Output Requests':

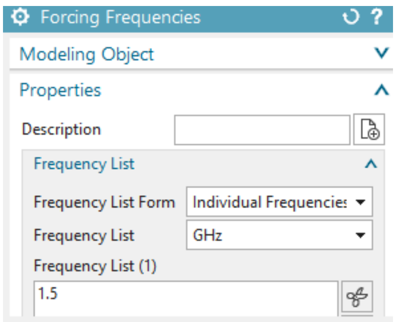


4. And register '2D': (This is not necessary in 3D)



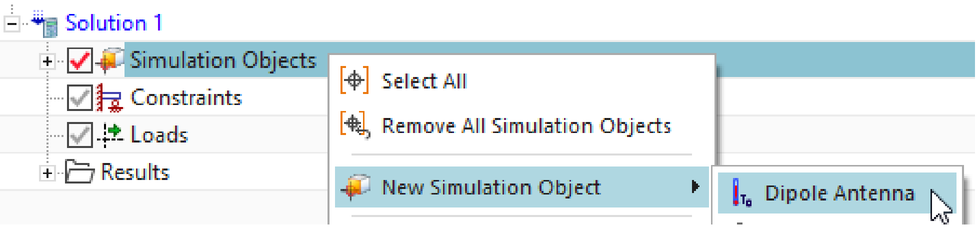
5. At 'Forcing Frequency':



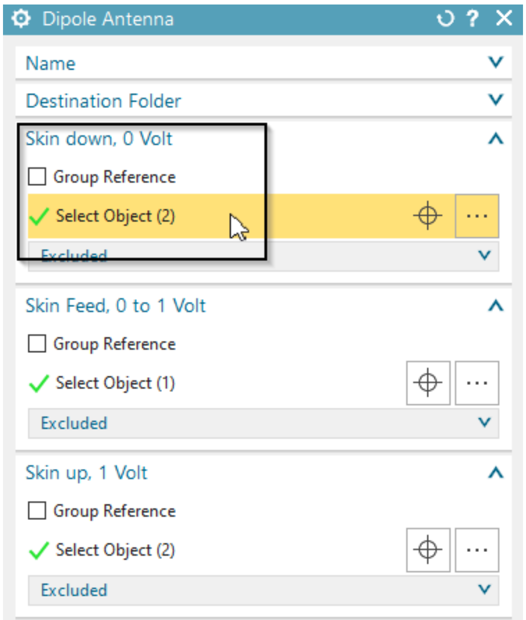


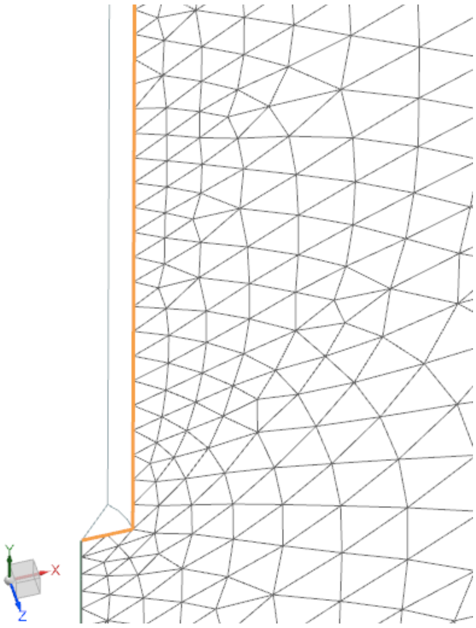
6. Click OK.

7. Edit, Check the simulation object 'DipoleAntenna(1)'. It's of type 'Dipole Antenna'. The meaning of the three edge selections (in 3D faces) and definitions is as follows:

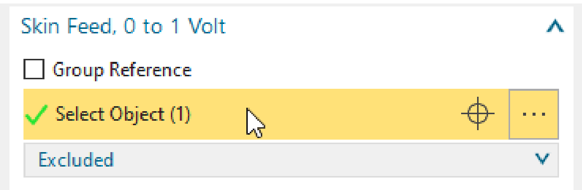


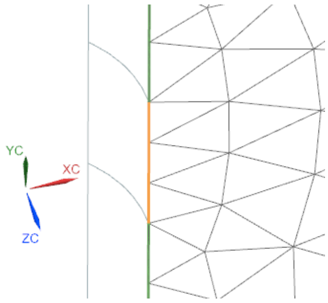
8. The ‘Skin Down, 0 Volt’: Selected are the two edges of the lower part of the dipole. Between this lower part and the following upper part there will be applied a jump in the electric field from zero to one volt. (For 3D: corresponding faces instead of edges)



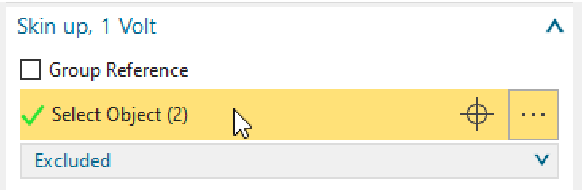


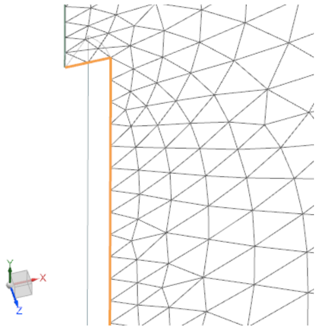
9. The ‘Skin Feed, 0 to 1 Volt’: Selected is the small edge (or corresponding face in 3D) between lower and upper part of the dipole. On this edge there will be a linear transition of the electric field.



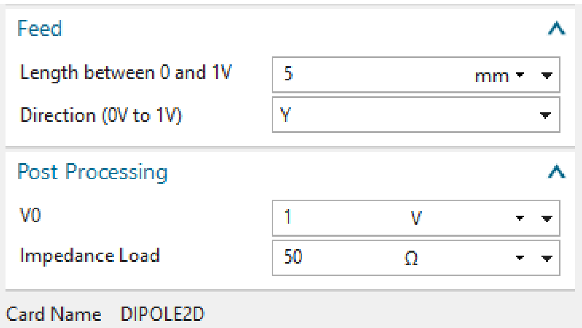


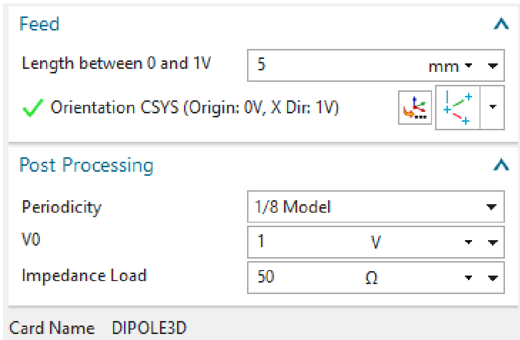
10. Finally, at ‘Skin Up, 1 Volt’, the two edges (faces in 3D) for the upper part of the dipole are selected.





11. Box 'Feed' differs slightly in 2D (below left) and 3D (right). In both cases there must be defined the 'Length between 0 and 1 Volt', the gap distance. In our case this is 5 mm. In 2D there must be given the 'Direction (0V to 1V)'. Instead, in 3D, there is a 'Orientation CSYS (Origin: 0V, X Dir: 1V)' defined, a coordinate system that points into the gap direction.

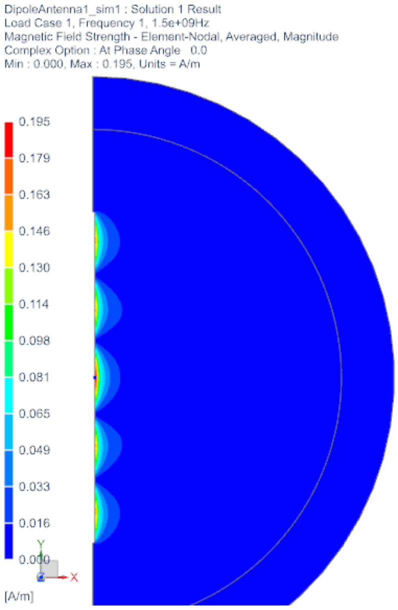


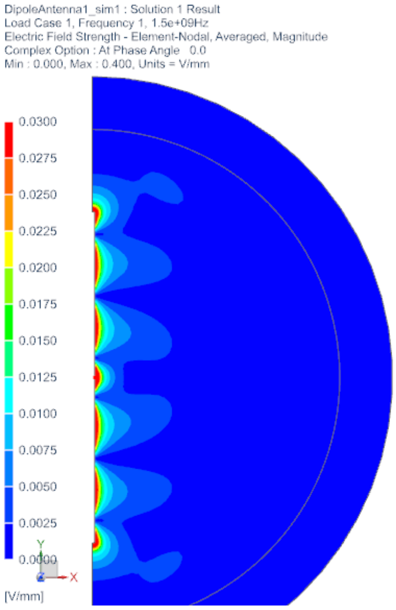


12. In the box 'Post Processing' there is defined 'V0' (the voltage jump), and the 'Impedance Load', values that are used to compute the antenna impedance and reflection coefficient. The defaults here are ok in most cases.
In case of 3D there is an additional option 'Periodicity'. Here we define whether the model is complete ('Full Model') or only a part.

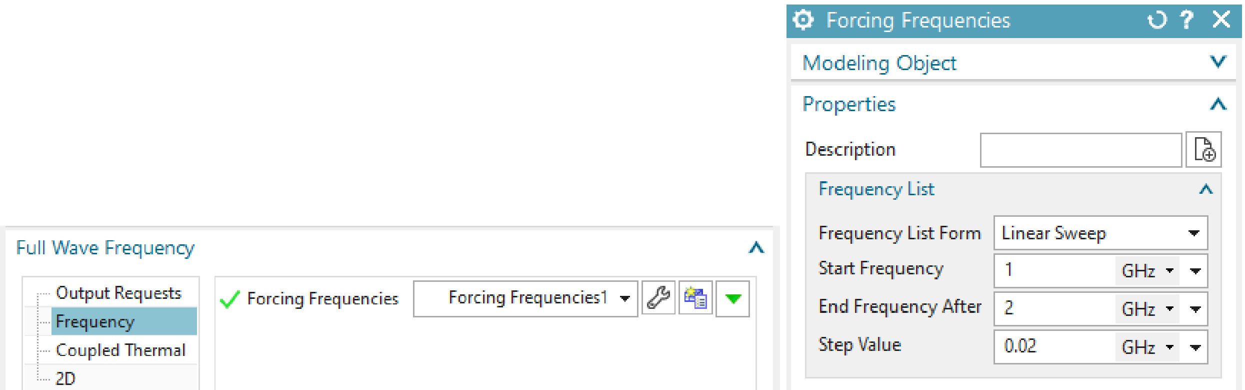
13. Solve the solution.

14. Post process the Results. Next picture left shows the magnetic and right the electric field strength, both with option 'At Phase Angle', so showing the real parts.

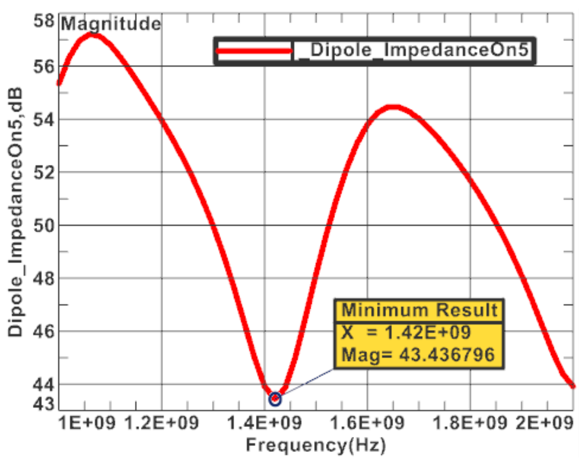


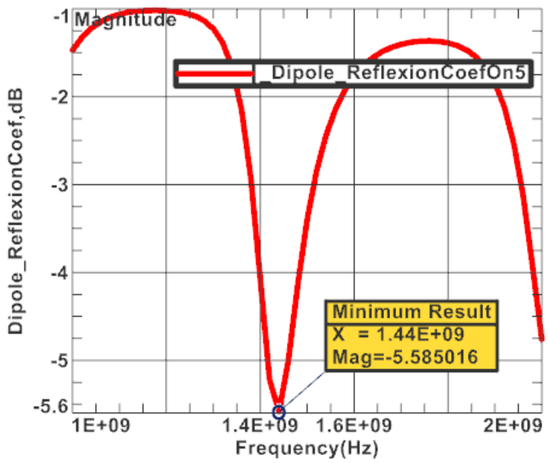


15. Next we modify the forcing frequencies to a sweep, see below picture, and solve again.



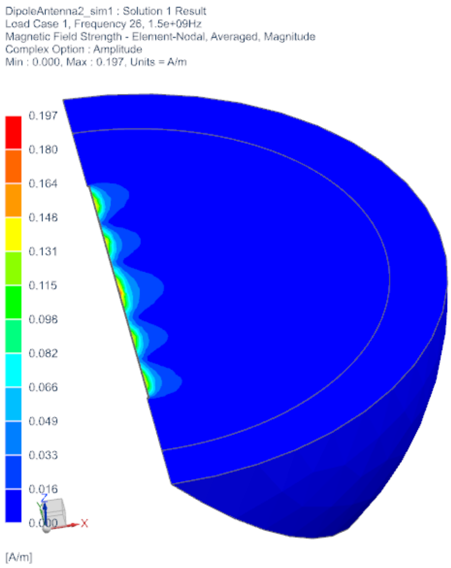
16. the resulting impedance and reflection coefficient are now graphs over the frequency as shown below (y axis set to type 'dB'). They show a minimum at about the same frequency of 1.44 GHz.

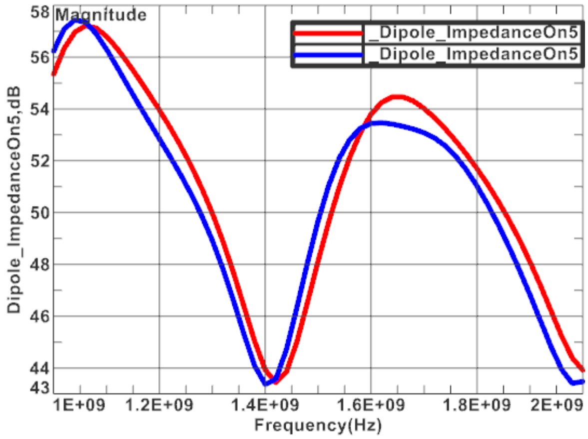




17. the resulting graphs from the 3D model differ slightly. Smaller mesh sizes would further reduce this difference. The following picture below right shows both 2D (red) and 3D

(blue) together.





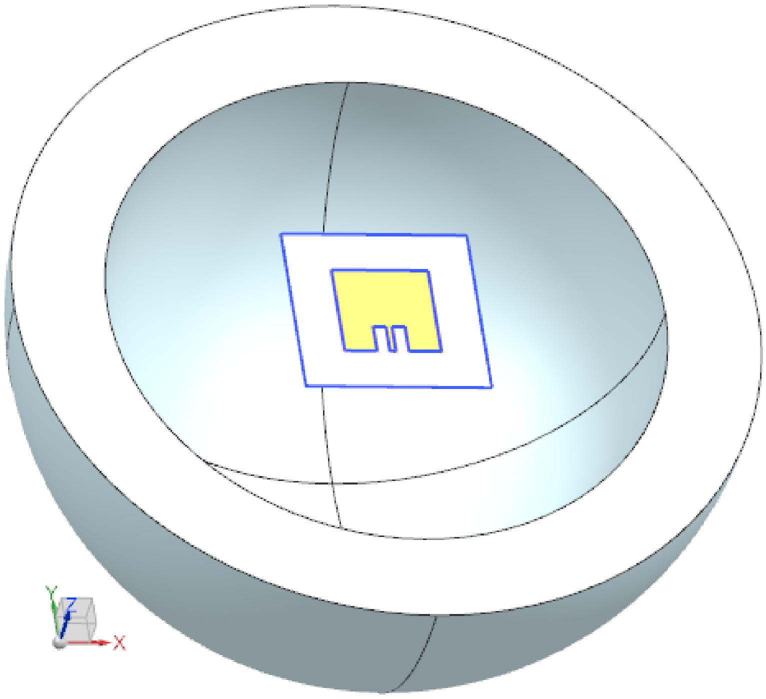
18. The tutorial is complete.

6 Tutorial 6

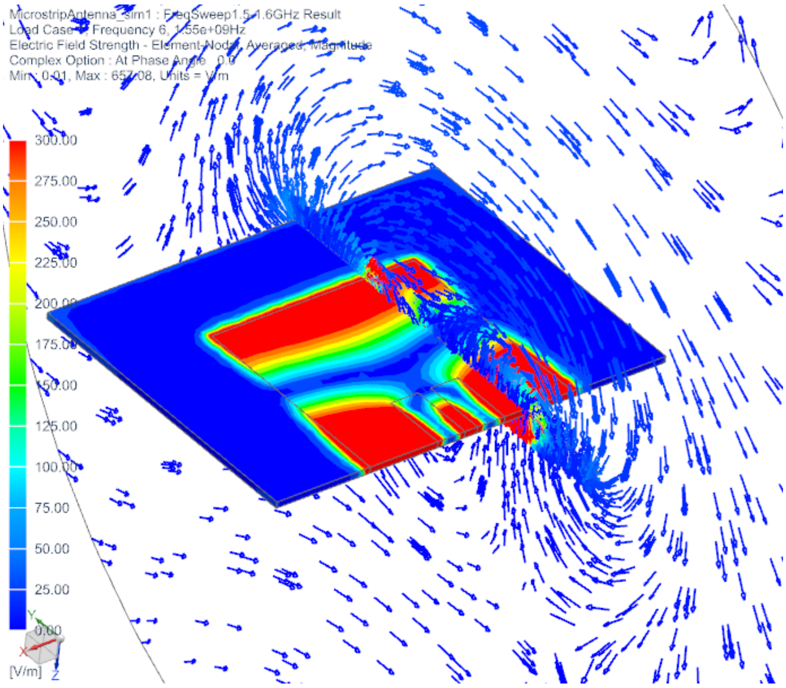
6.1 Microstrip Patch Antenna

This example shows a already complete model of a microstrip patch antenna that is ready to solve. The model setup is very similar to the 'Dipole Antenna' shown in a prior chapter. The interested reader can check the model features and gain insights.

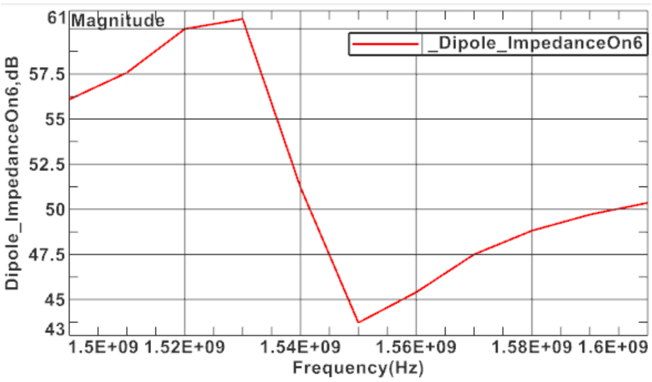
Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.5MicrostripAntenna.zip`

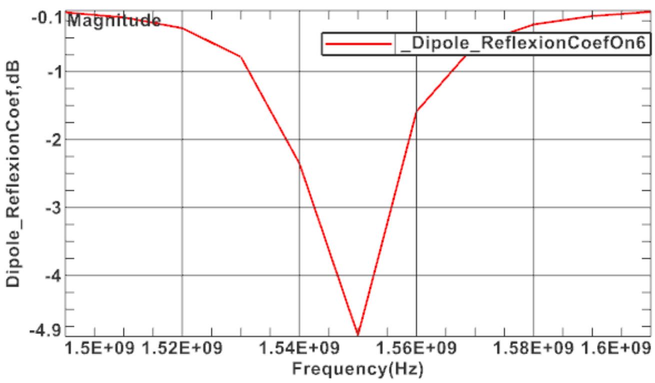


The following figure shows the electric field pattern at frequency 1.55 GHz.



Antenna impedance and reflection coefficients over frequency, as they result from the frequency sweep solution, are shown below.



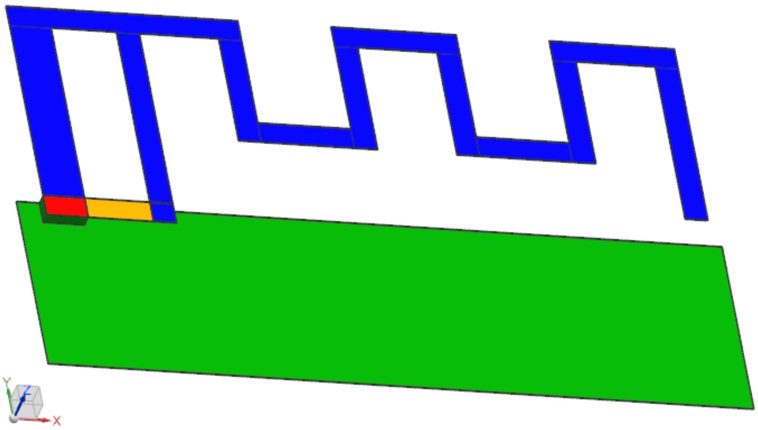


7 Tutorial 7

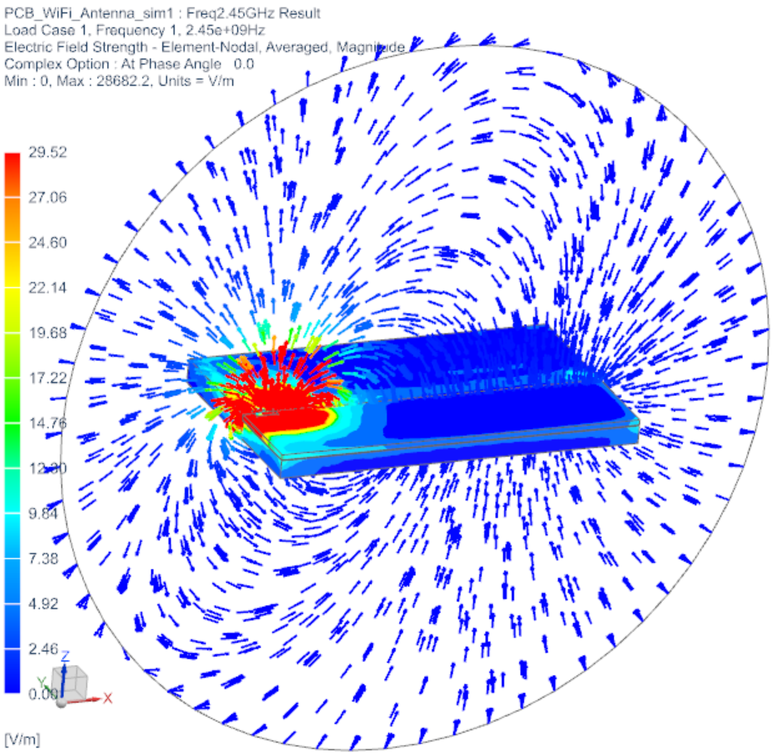
7.1 PCB WiFi Antenna

This example shows a already complete model of a PCB WiFi antenna that is ready to solve. The model setup is very similar to the 'Dipole Antenna' shown in a prior chapter. The interested reader can check the model features and gain insights.

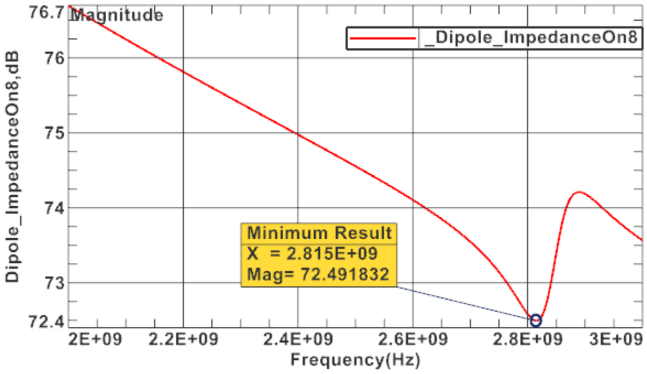
Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.6PCB_WiFiAntenna.zip`

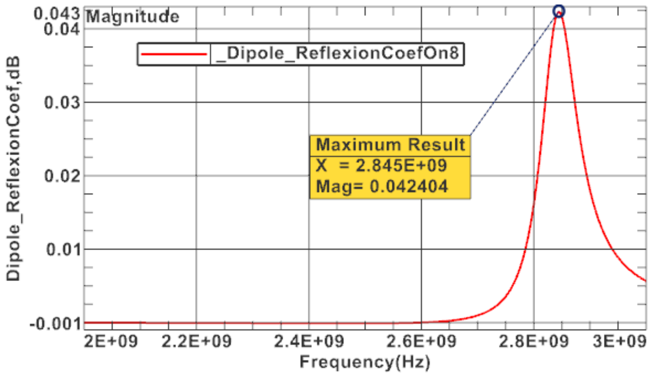


The following figure shows the electric field pattern at frequency 2.45 GHz.



Antenna impedance and reflection coefficients over frequency, as they result from a frequency sweep solution, are shown below.


Magnitude
_Dipole_ImpedanceOn8
Minimum Result
X = 2.815E+09
Mag= 72.491832
Dipole_ImpedanceOn8,dB
Frequency(Hz)

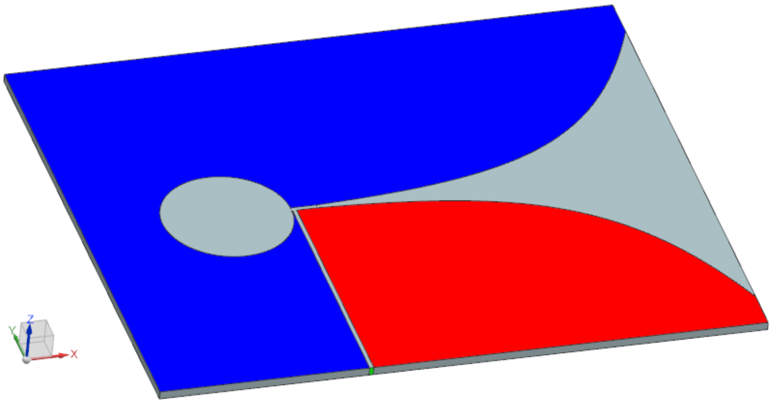

Magnitude
_Dipole_ReflexionCoefOn8
Maximum Result
X = 2.845E+09
Mag= 0.042404
Dipole_ReflexionCoef,dB
Frequency(Hz)

8 Tutorial 8

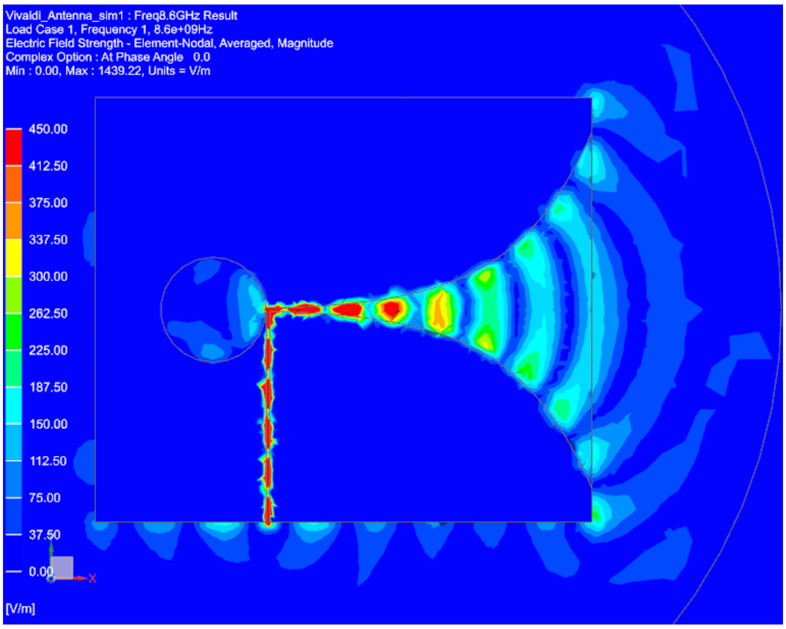
8.1 Vivaldi Antenna

This example shows a already complete model of a Vivaldi antenna that is ready to solve. The model setup is very similar to the 'Dipole Antenna' shown in a prior chapter. The interested reader can check the model features and gain insights.

Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/5.FullWave/5.7VivaldiAntenna.zip`



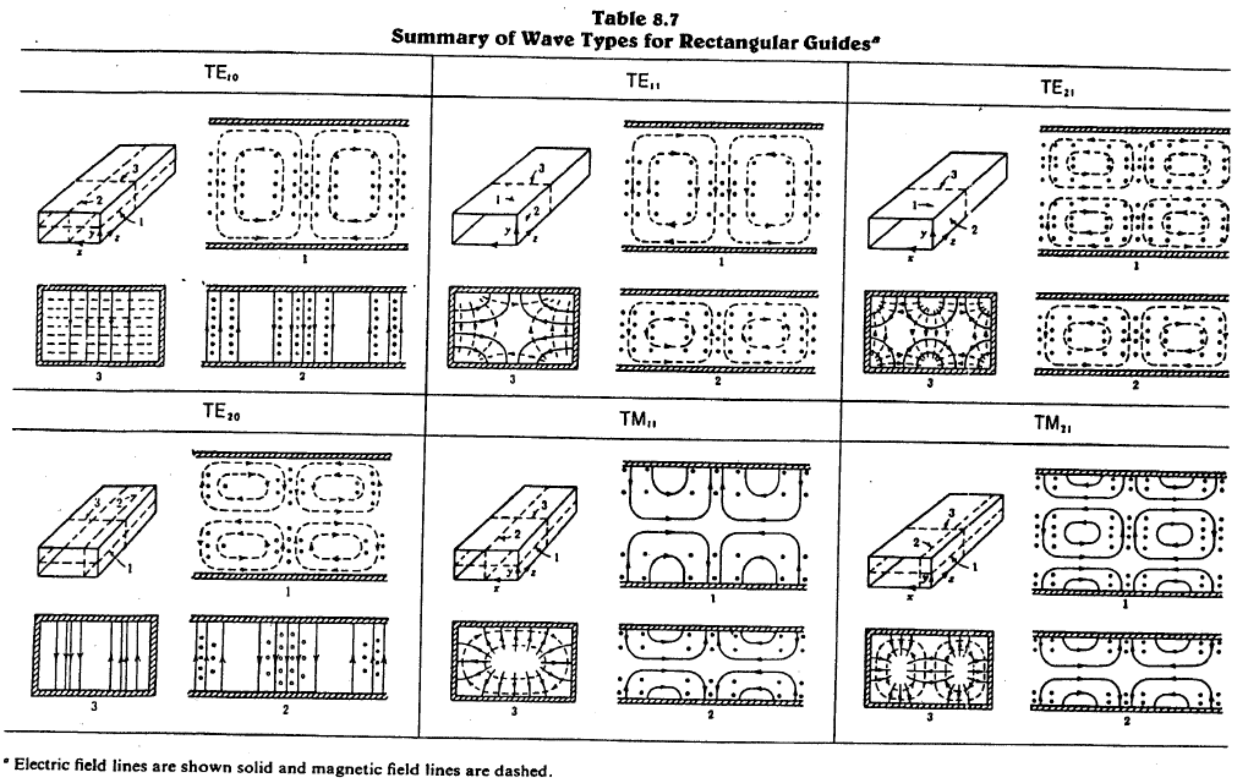
The following figure shows the electric field pattern at frequency 8.6 GHz.



Antenna impedance and reflection coefficients over frequency, as they result from a frequency sweep solution, are shown below.

9 Summary of Wavetypes for Rectangular Guides

The following picture explains how electric and magnetic field vectors behave for the different types of TE/TM modes. It is taken from [Clemen].



10 Literature

Team18task :

Team Workshop Problem 18. Waveguide Loaded Cavity.

http://www.compumag.org/jsite/images/stories/TEAM/problem18.pdf

4th Juni 2015

Clemen :

Manuscript “Elektromagnetische Wellen”. Prof. Dr. Clemen

http://www.hs-augsburg.de/clemen/lehre/Skript_Wellen/8Hohlleiter.PDF

4th Juni 2015

Arcioni :

Arcioni, Paolo, Perregrini, Luca, Bonecchi, Fulvio, “Low-Loss Waveguide Combiners for Multidevice Power Amplifiers,” The Second International Conference on Electromagnetics in Aerospace Applications, September 1991.