EUROPEAN UNION
European Structural and Investment Funds
Operational Programme Research,
Development and Education

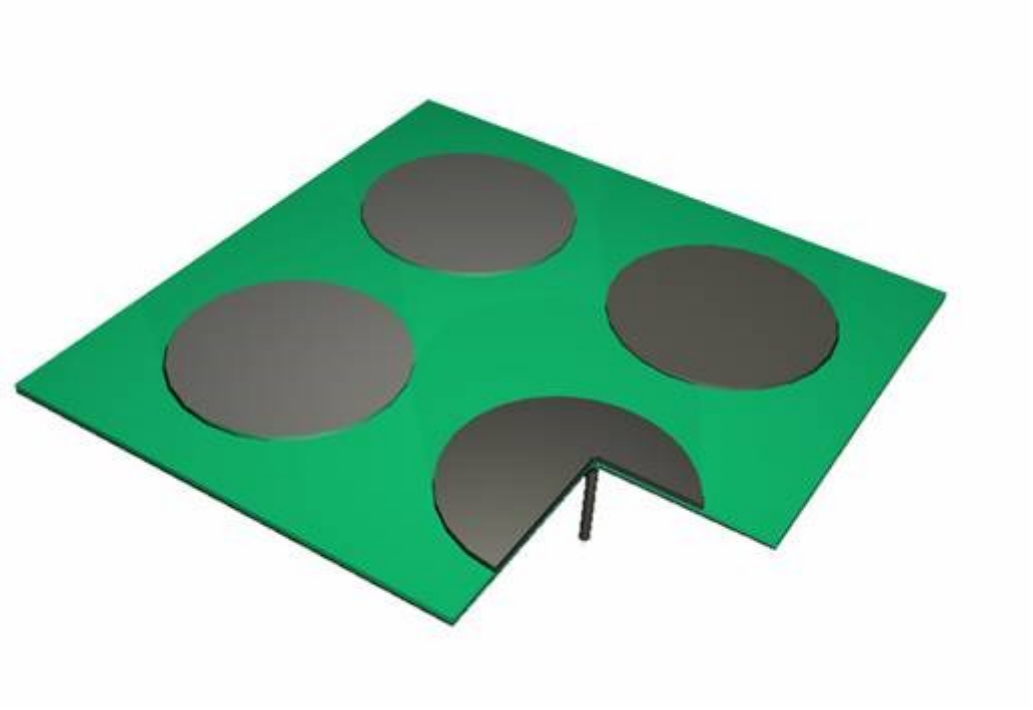
MINISTRY OF EDUCATION,
YOUTH AND SPORTS

Go through documents prepared in CST Studio Suite as portion of Antennas lectures/seminars to verify the theory obtained.

Patch Antenna Document

Please find the corresponding simulation models for this workflow example in the component library:

Single Antenna (Transient Solver):	Patch Antenna (TD)
Single Antenna (Frequency Domain Solver):	Patch Antenna (FD)
Antenna Array (Transient Solver):	Patch Antenna Array
Antenna Array Combine (Transient Solver):	Patch Antenna Array Combine
Antenna Array Simultaneous (Transient Solver):	Patch Antenna Array Simultaneous



EUROPEAN UNION
European Structural and Investment Funds
Operational Programme Research,
Development and Education

MINISTRY OF EDUCATION,
YOUTH AND SPORTS

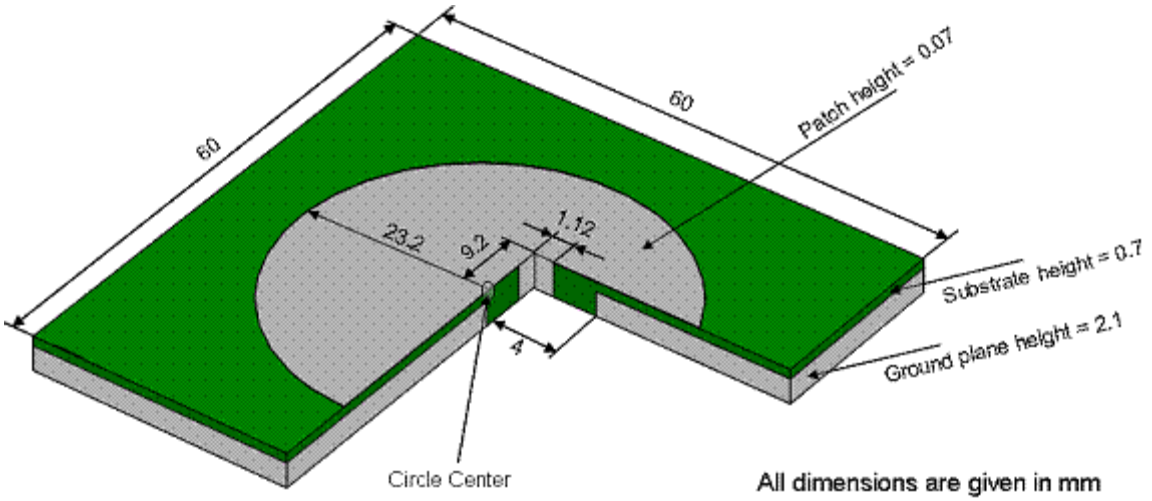
Geometric Constructions

Introduction and Model Dimensions

In this document you will learn how to simulate antenna devices. As a typical example you will analyse a circular patch antenna. The following explanations can be applied to other antennas as well.

Although, CST Microwave Studio can provide a wide variety of results, this document will concentrate mainly on the S-parameters and farfield results.

In addition, the single patch antenna will be extended to a rectangular 2x2 array pattern using three different methods. First, the farfield solution of the single patch antenna is applied to the antenna array feature, superimposing the results with different amplitudes and phase settings. Another possibility expands the patch model to a set of four identical antennas, each excitable with its own coaxial feed. Here, you have the option to calculate the antennas separately one after another and finally combine the results with arbitrary amplitudes and phase values, or to run the excitation simultaneously to produce the farfield result with only one solver cycle. The farfield distributions of all these possibilities will be compared.



The structure depicted above consists of two different materials: the Substrate and the **P**erfect **E**lectric **C**onductor (PEC). There is no need to model the air above because it will be added automatically (according to the current background material setting) when the open boundary conditions are specified. This will be done automatically with an appropriate template. The feeding of the patch is realized with a coaxial line.

Geometric Construction Steps

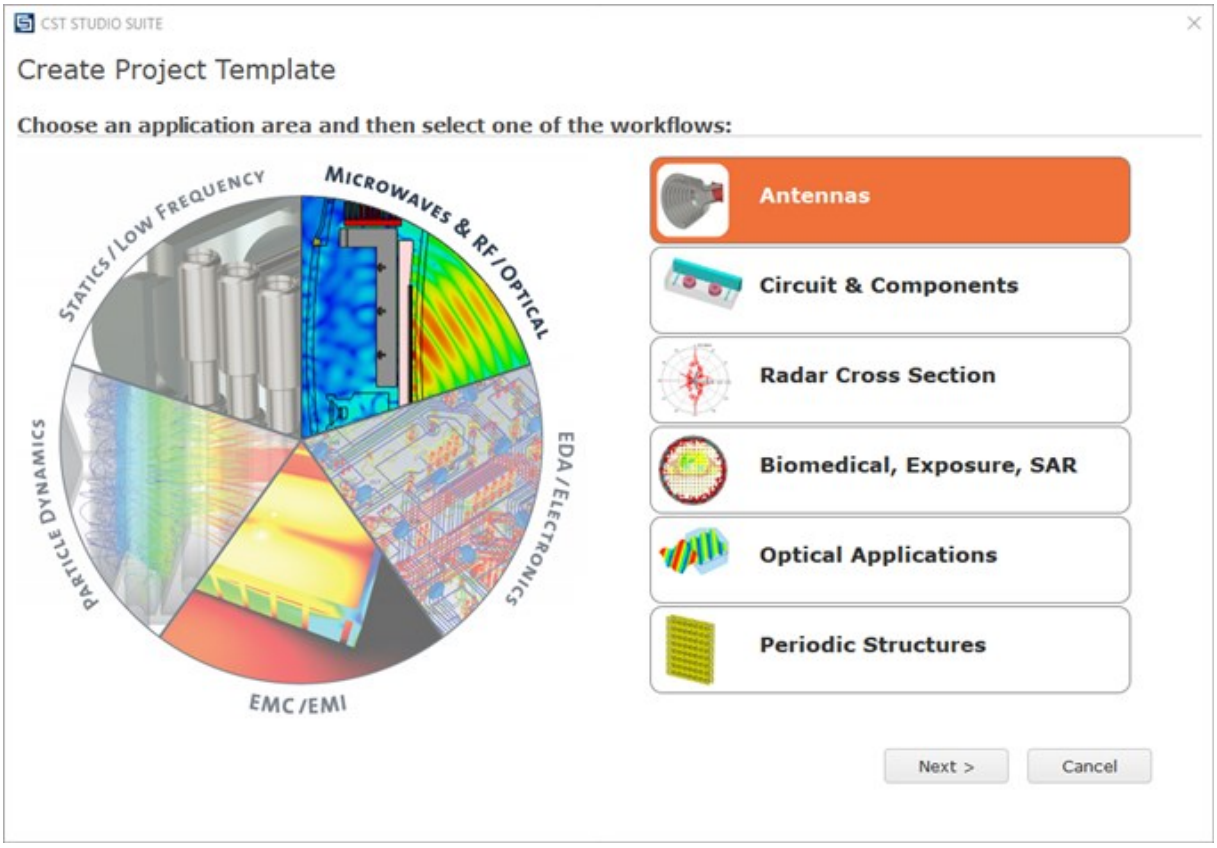
This document will take you step-by-step through the construction of your model, and relevant screen shots will be provided so that you can double-check your entries along the way.

Please remember the ***Undo*** facility in the event that you want to cancel the last construction step.

Create a New Project

After launching the CST Studio Suite, you will enter the start screen showing you a list of recently opened projects and allowing you to specify the application which suits your requirements best. The easiest way to get started is to configure a project template which defines the basic settings that are meaningful for your typical application. Therefore, click on the *New Template* button in the *New Project from Template* section.

Next you should choose the application area, which is *Microwaves & RF* for the example in this document and then select the workflow by double-clicking on the corresponding entry.



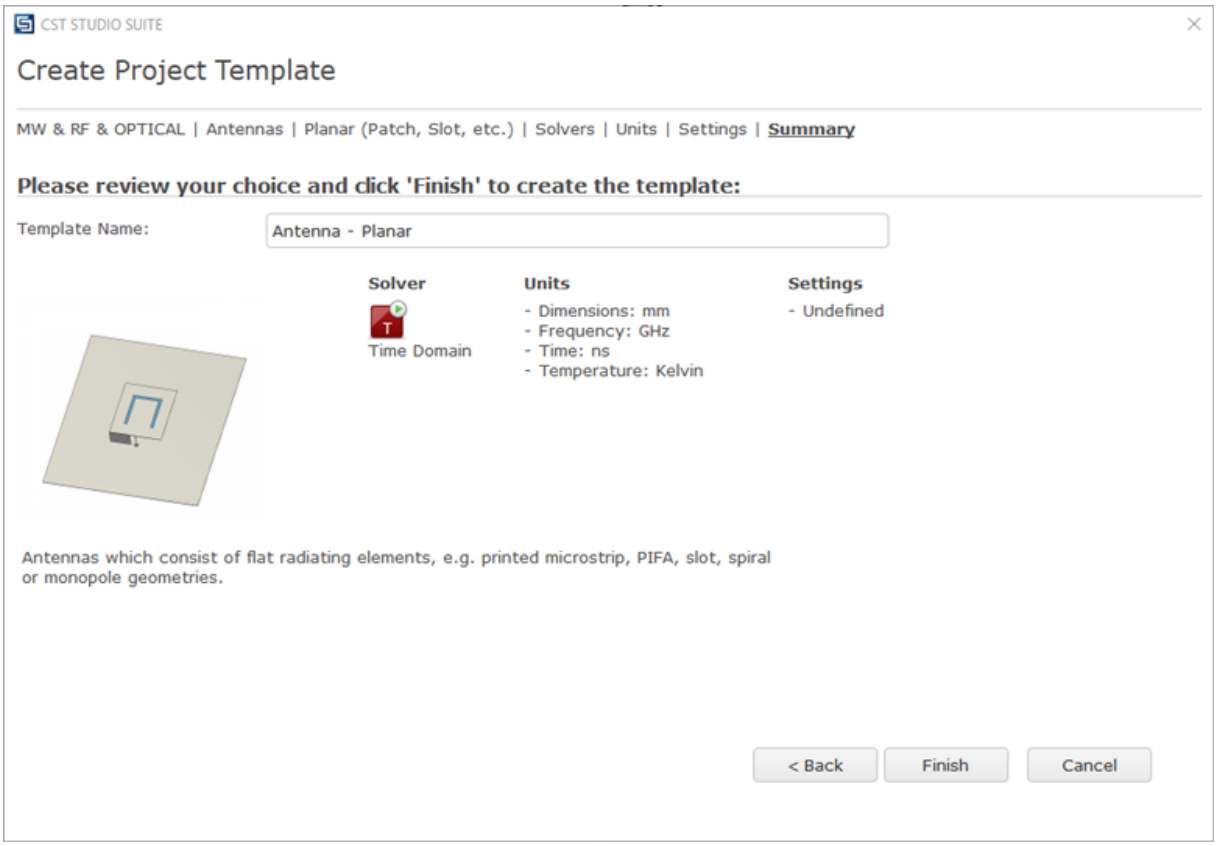
For the Patch Antenna structure, please select **Antennas > Planar (Patch, Slot, etc.) > Time Domain Solver** .

At last, you are requested to select the units which fit your application best. For the Patch Antenna structure, please select the dimensions as follows:

Dimensions:	*mm*
Frequency:	*GHz*
Time:	*ns*

If useful for a larger number of model setups, it is also possible to already store the simulation frequency range together with specific field monitor definitions for this project template. However, in

this document all other settings will be left unchanged. After clicking the Next button, you can give the project template a name and review a summary of your initial settings:

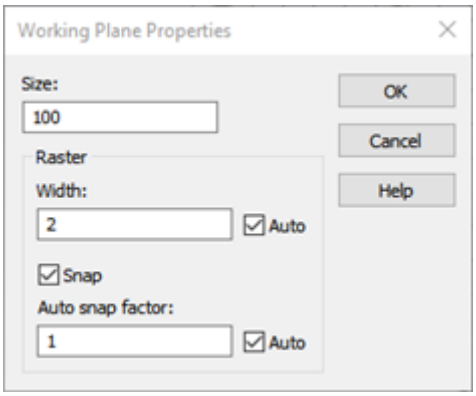


Finally click the *Finish* button to save the project template and to create a new project with appropriate settings. CST Microwave Studio will be launched automatically due to the choice of the application area *Microwaves & RF.*

All settings made for a project template can be modified later on during the construction of your model. For example, the units can be modified in the units dialog box (Home: ***Settings > Units***) and the solver type can be selected in the Home: ***Simulation > Setup Solver*** drop-down list.

Set the Working Planes Properties

The next step will usually be to set the working plane properties to make the drawing plane large enough for your device. Because the structure has a maximum extension of 60 mm along a coordinate direction, the working plane size should be set to at least 100 mm. These settings can be changed in a dialog box that opens after selecting View: ***Visibility > Working Plane***  ***> Working Plane Properties***.

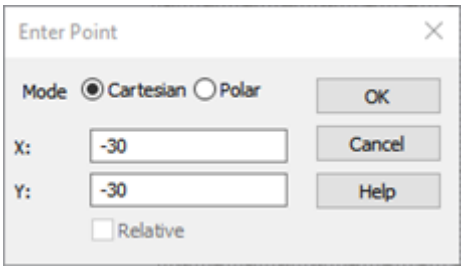


In this dialog box, you should set the *Size* to 100 (the unit which has previously been set to mm is displayed in the status bar), the *Raster width* to 2 and the *Auto snap factor* to 1 to obtain a reasonably spaced grid. Please confirm these settings by pressing the *OK* button.

Draw the Substrate Brick

The first construction step for modelling a planar structure is usually to define the substrate layer. This can be easily achieved by creating a brick made of the substrate's material. Please activate the brick creation mode Modeling: ***Shapes > Brick*** .

When you are prompted to define the first point, enter the coordinates numerically by pressing the *Tab* key that will open the following dialog box:

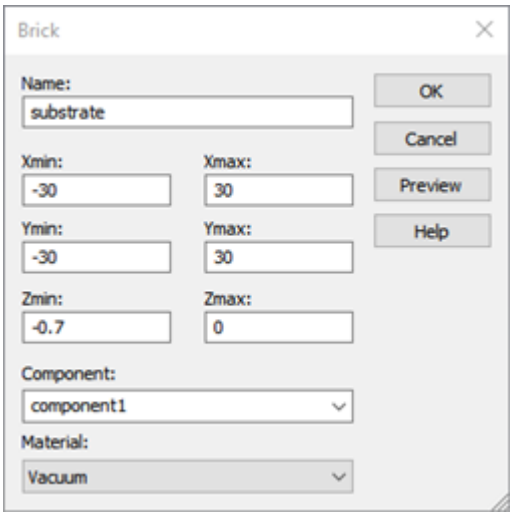


In this example, you should enter a substrate block that has an extension of 60 mm in each of the transversal directions. The transversal coordinates can thus be described by X = -30, Y = -30 for the first corner and X = 30, Y = 30 for the opposite corner, assuming that the brick is modelled symmetrically to the origin. Thus, please enter the first points coordinates *X* = -30 and *Y* = -30 in the dialog box and press the *OK* button.

Afterwards, you can repeat these steps for the second point:

1. Press the *Tab* key
2. Enter *X* = 30, *Y* = 30 in the dialog box and press *OK*.

Now you will be requested to enter the height of the brick. This can also be numerically specified by pressing the *Tab* key again, entering the *Height* of -0.7 and pressing the *OK* button (it is convenient to define the substrate in the negative z-direction). Now the following dialog box will appear, displaying a summary of your previous input:

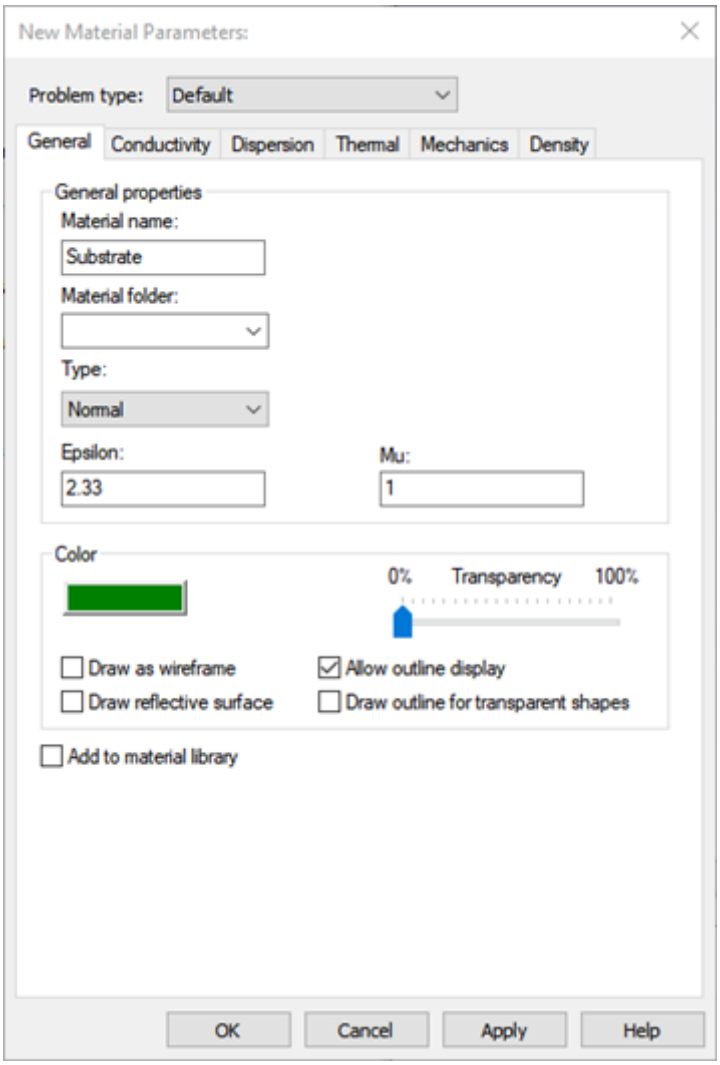


Please check all these settings carefully. If you encounter any mistakes, please change the value in the corresponding entry field.

You should now assign a meaningful name to the brick by entering e.g. substrate in the *Name* field, keep the *Component* default setting (component1).

The use of different components allows you to combine several solids into specific groups, independent of their material behaviour. However, in this document, it is convenient to construct the single patch antenna as a representation of one component that can then easily be extended into a patch antenna array.

Finally, you need to define the substrate material. Because no material has yet been defined for the substrate, you should open the New Material Parameters dialog box by selecting [New Material...] from the *Material* dropdown list:

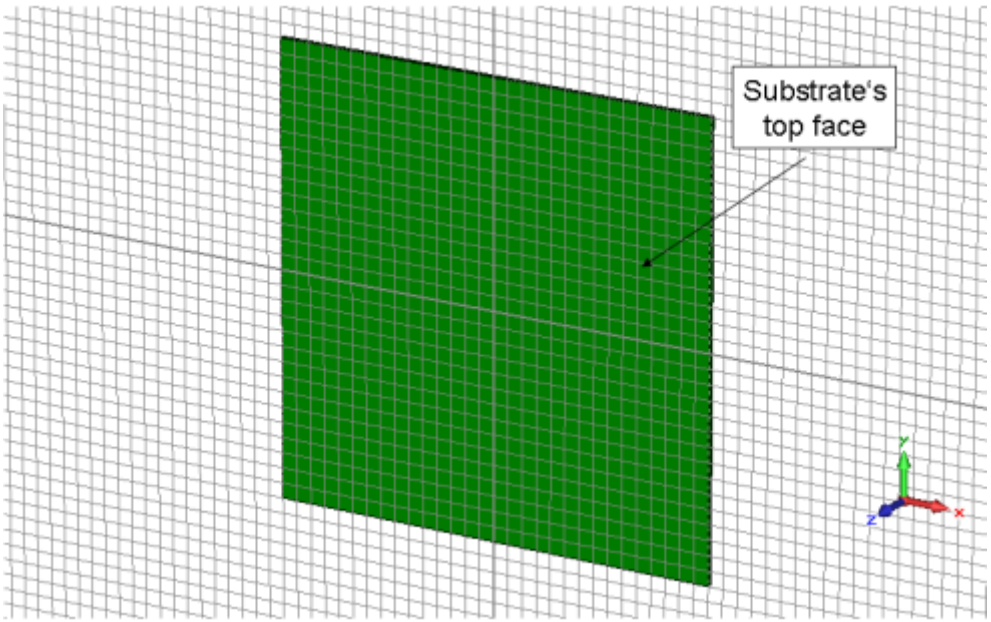


In this dialog box, define a new *Material name* (e.g. Substrate) and set the *Type* to a Normal dielectric material. Afterwards, specify the material properties in the *Epsilon* and *Mu* fields. Here, you only need to change the dielectric constant *Epsilon* to 2.33. Finally, choose a colour for the layer by pressing the coloured button. Your dialog box should now look similar to the above picture before you press the *OK* button.

The defined material Substrate will now be available inside the current project for the further creation of other solids. However, if you want to also save this specific material definition for other projects,

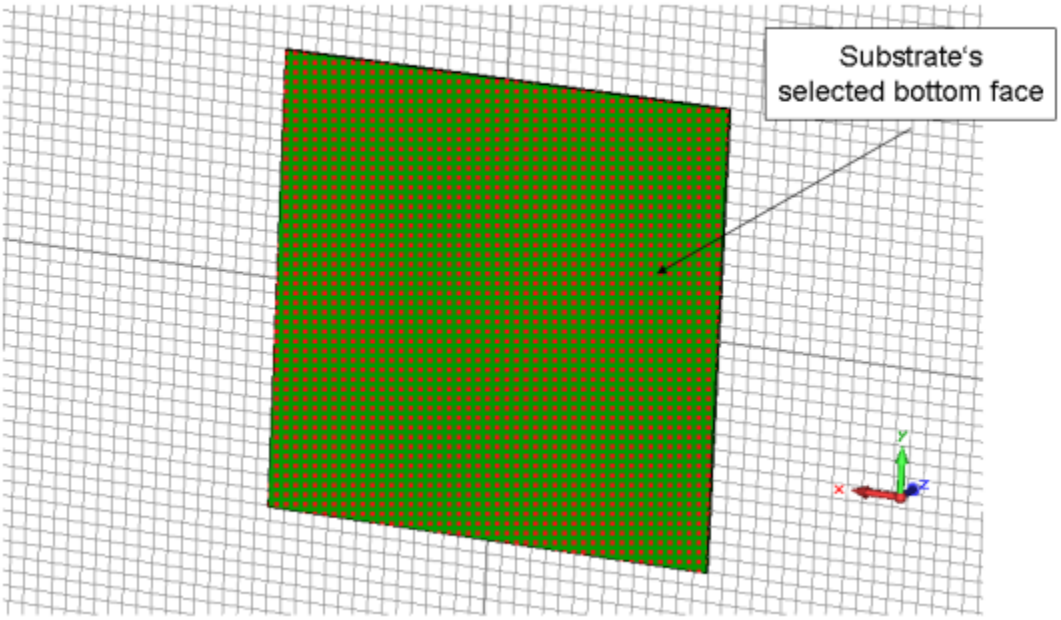
you may check the button *Add to material library*. You will have access to this material database by clicking on Modeling: ***Materials > Material Library***  ***> Load from Library*** .

Back in the brick creation dialog box you can also press the *OK* button to finally create the substrate brick. Your screen should now look as follows (you can press the *Space* bar in order to zoom the structure to the maximum possible extent):

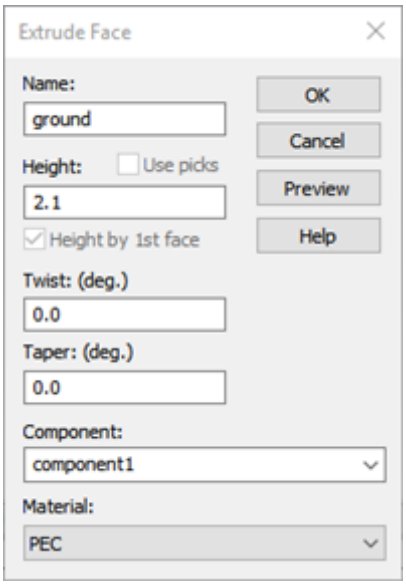


Model the Ground Plane

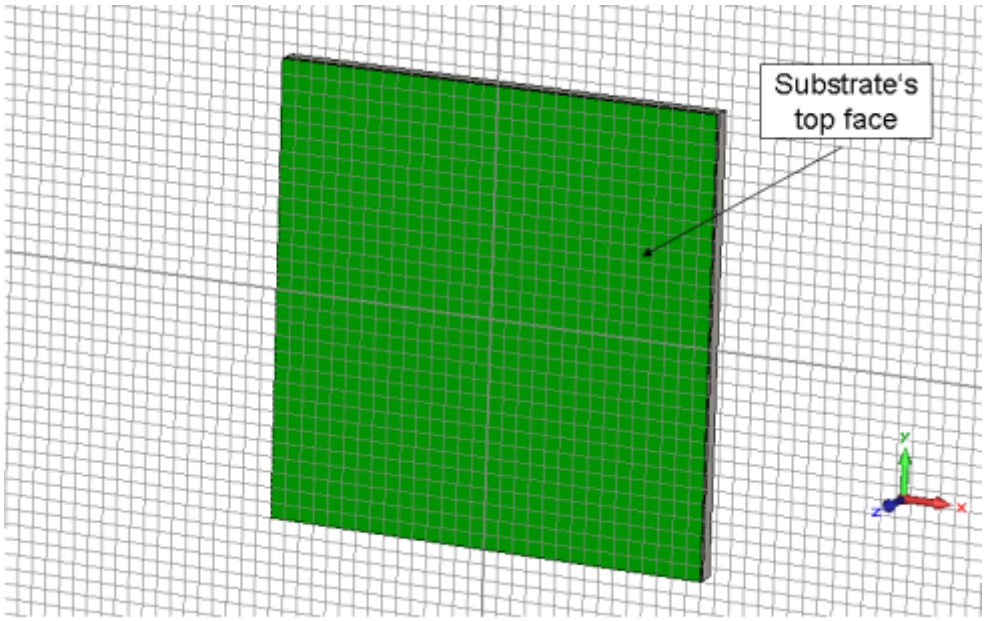
The next step is to model the ground plane of the patch antenna. Because the antenna will be excited by a coaxial feed at the bottom face, the electric boundary at *Zmin* defined by the previously chosen template is not suitable as a ground plane. Consequently, a metallic brick must be additionally defined. First, the model has to be rotated by activating the rotation mode View: ***Mouse Control > Rotate*** . Then the bottom face has to be activated by the face pick tool Modeling: ***Picks > Picks***  ***> Pick Points, Edges or Faces*** double-clicking on the substrates bottom face. The face selection should then be visualized as follows:



You can now extrude the selected face with the Extrude tool selecting Modeling: ***Shapes > Extrude Face*** . Here, you must enter the height and the material of the new shape to be created. In this example, the ground plane must have a non-zero thickness because of the coaxial feed that will be modelled later. In CST Microwave Studio a port region must be homogeneous for at least three mesh lines in longitudinal direction. You can therefore choose a *Height* of 2.1 mm, representing three times the substrate thickness, as a sufficient dimension. Enter this value in the following dialog box and select PEC from the *Material* dropdown list as the metallic material property:

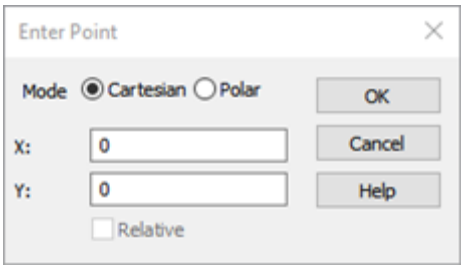


After entering a suitable name (e.g. ground) in the *Name* field and confirming your settings with the *OK* button, the current structure will look like this (rotated again to the see the top face):

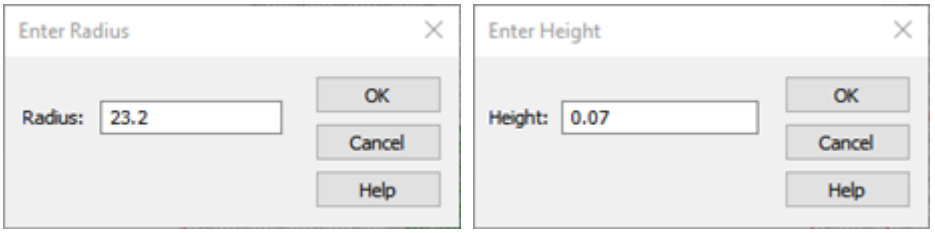


Model the Patch Antenna

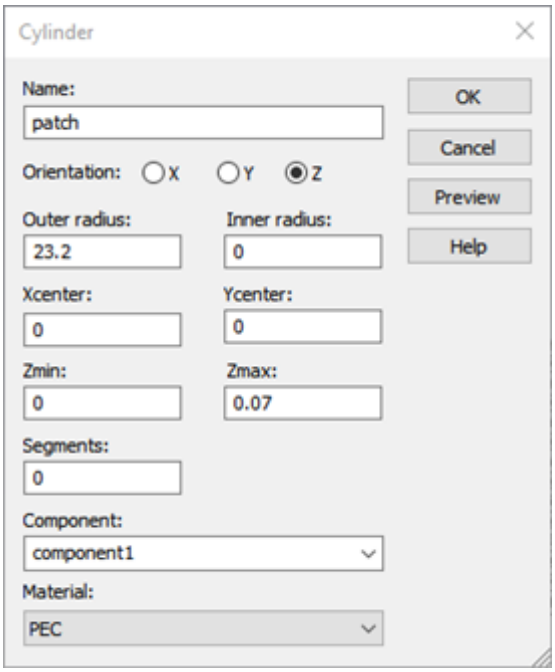
After the ground plane has been defined, the patch antenna must be modelled as a cylindrical shape on the substrates top face. Please activate the cylinder creation mode Modeling: ***Shapes > Cylinder*** . Similar to the construction of the substrates brick, enter the coordinates numerically by pressing the Tab key to open the following dialog box:



Here, enter the centre point of the cylinder with X = 0 and Y = 0 because the patch is located symmetrically on the substrate. Afterwards, please define the *Radius* with 23.2 mm and the *Height* with 0.07 mm in the shown dialog boxes that appear after you have pressed the *Tab* button:

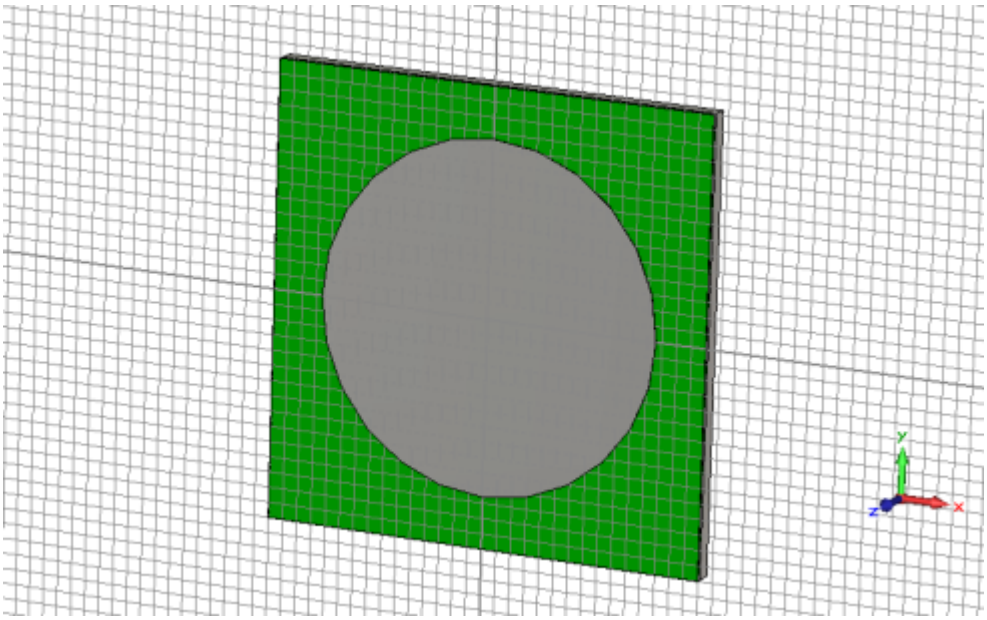


Skipping the entry dialog for the inner radius by pressing the *Esc* button will lead to the following dialog box that provides a summary of your entered parameters:



Select PEC as the *Material* setting for the patch and assign a meaningful name to the brick by entering e.g. patch in the *Name* input field.

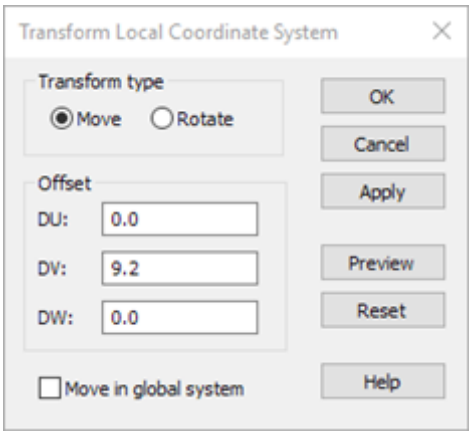
Again, please check the settings carefully and change any mistakes in the corresponding entry fields. After you apply the settings with the *OK* button, your screen should show the following structure:



Model the Coaxial Feed

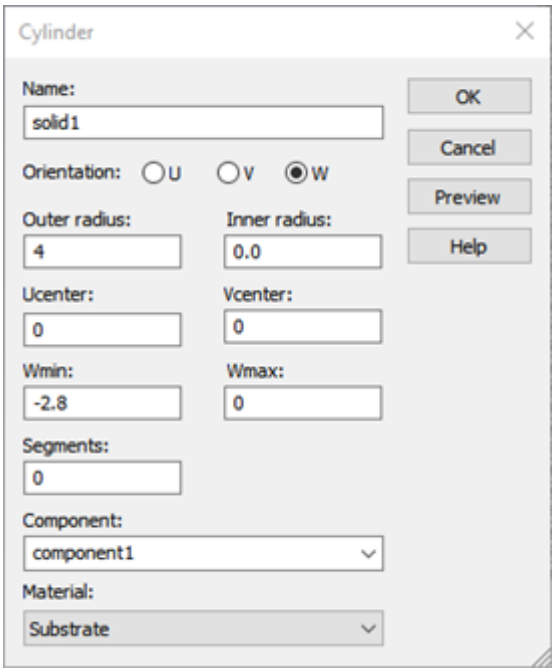
The last modelling step is the construction of the coaxial feed as the excitation source for the patch antenna. This action introduces the working coordinate system (WCS). Because the feeding point is located asymmetrical to the circular patch it is advisable to activate the local coordinate system Modeling: ***WCS > Local WCS***.

To define the new centre point for the coaxial feed the local coordinate system is moved along the positive v-direction Modeling: ***WCS > Transform WCS***. Therefore, please enter a value of 9.2 mm in the following dialog box:

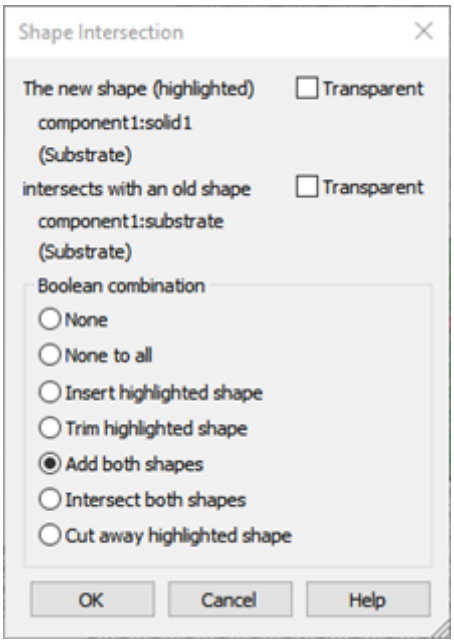


Now it is possible to design the coaxial feed by constructing two cylindrical shapes, similar to the previously defined circular patch.

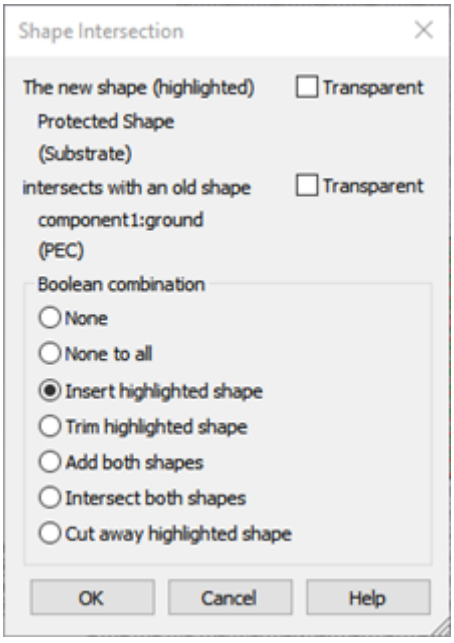
Please activate the cylinder creation mode again (tool Modeling: ***Shapes > Cylinder***). First, enter the values for the coaxial substrate cylinder using the *Tab* key, again skipping the input of the inner radius. The cylinder has an outer radius of 4 mm and an extension in the negative w-direction of 2.1+0.7=2.8 mm. Select the previously defined Substrate material from the *Material* dropdown list. Please check your settings against the following dialog box and then create the cylinder with the *OK* button:



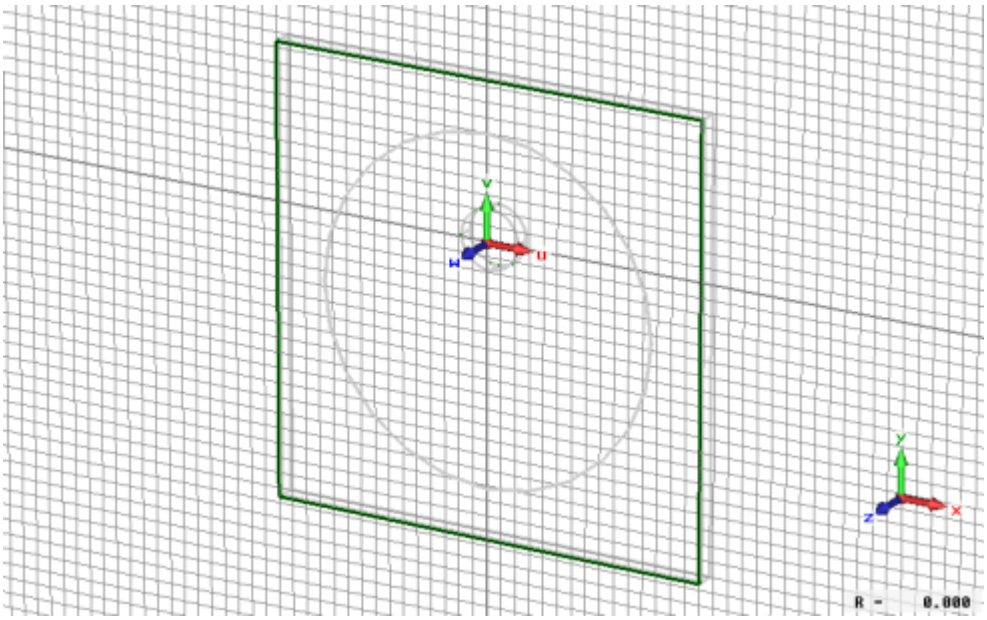
As a result, the cylinder shape component1:solid1 intersects with two already existing shapes, the solid component1:substrate and the ground plane component1:ground. Here it is necessary to determine the type of intersection for the shapes. It is more convenient to combine the two substrate materials into a single shape, so please mark the radio button *Add both shapes* in the Shape Intersection dialog box as presented below and confirm with *OK*:



In the second case, the substrate cylinder must be inserted into the PEC material of the ground plane. Please mark the radio button *Insert highlighted shape* from the Shape Intersection dialog window as presented below and confirm again with *OK*:



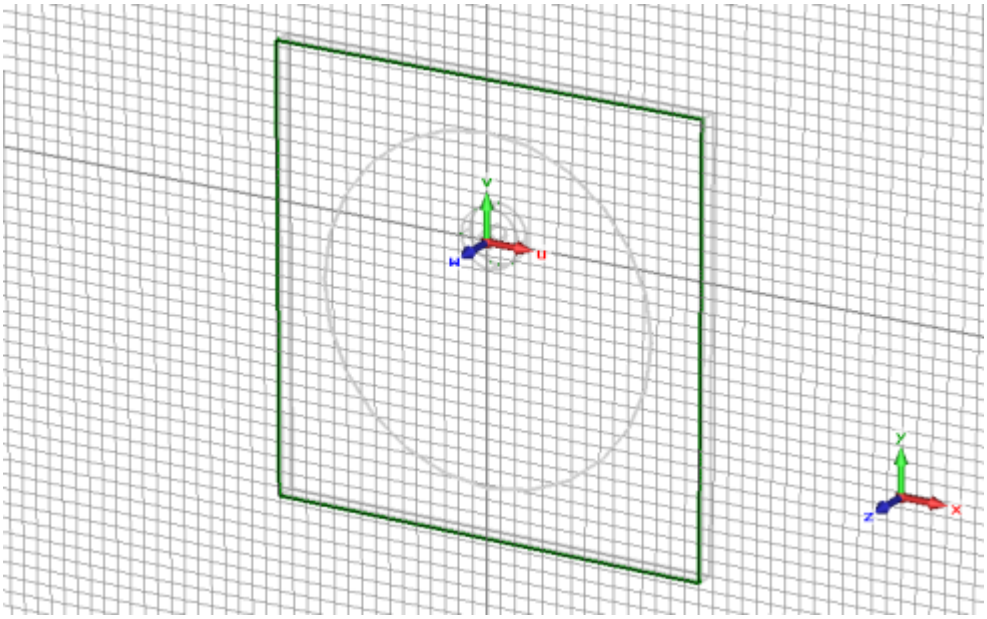
The following screenshot allows you to double-check your model (please use *Ctrl+W* or View: ***Visibility > Wire Frame*** to toggle the wireframe visualization mode on and off):



The inner conductor is constructed by defining another cylinder made of PEC material. Please define the cylinder with an outer radius of 1.12 mm and again an extension of 2.8 mm in the negative w-direction in a similar way as before. This time, select PEC from the *Material* dropdown list and define again a suitable name (e.g. feed) for the cylinder shape. Create the cylinder by pressing the *OK* button.

In this case no Shape Intersection dialog window will appear, because the PEC shape is defined ***after*** the normal material shape (here: Substrate). This implies that the PEC shape is *automatically inserted* into the intersected shape.

After applying the *OK* button the final model will look like the below figure (again use *Ctrl+W* or View: ***Visibility > Wire Frame*** to toggle the wireframe visualization mode on and off):



Common Solver Settings

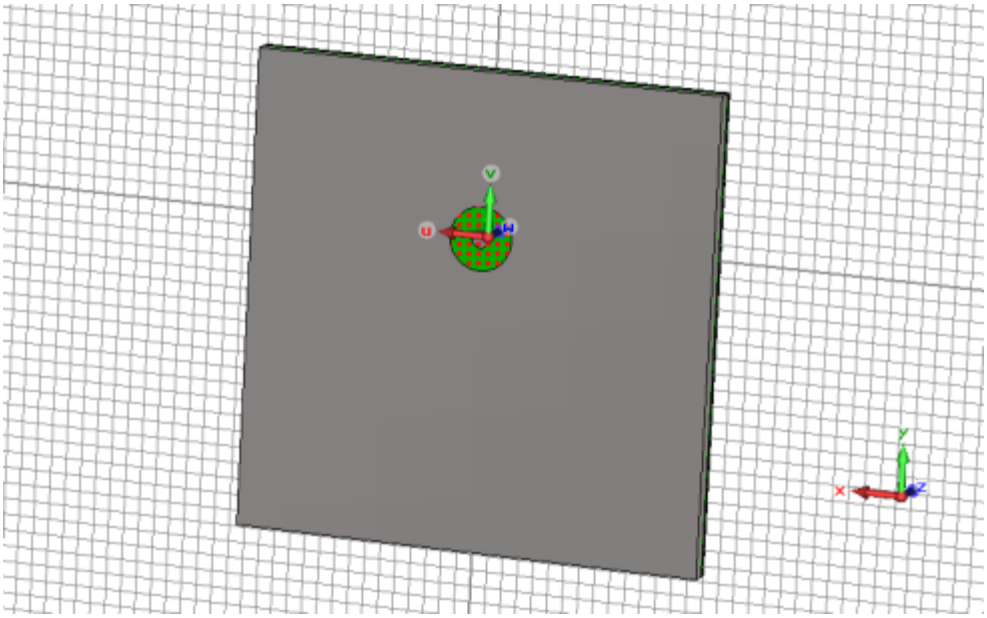
To this point, only the structure itself has been modelled. Now it is necessary to define some solver-specific elements. For an S-parameter calculation you must define input and output ports. Furthermore, the simulation needs to know how the calculation domain should be terminated at its bounds.

Define the Waveguide Port

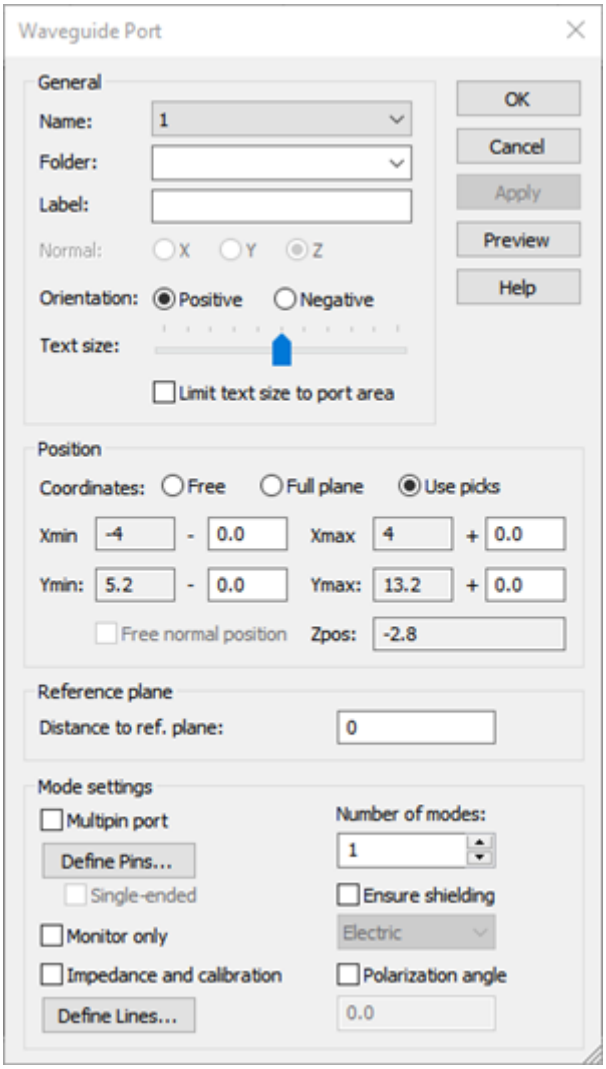
The next step is to add the excitation port to the patch antenna device, for which the reflection parameter will later be calculated. The port simulates an infinitely long coaxial waveguide structure that is connected to the structure at the ports plane.

A waveguide port extends the structure to infinity. Its transversal extension must be large enough to sufficiently cover the corresponding modes. In contrast to open port structures, the port range in this case is clearly defined by the outer shielding conductor of the coaxial waveguide.

Consequently, the easiest way to define the port range is to pick the face (pick tool Modeling: ***Picks > Picks > Pick Points, Edges or Faces***) of the coaxial feed (Substrate material) as demonstrated below (the model is rotated again to the bottom side first):

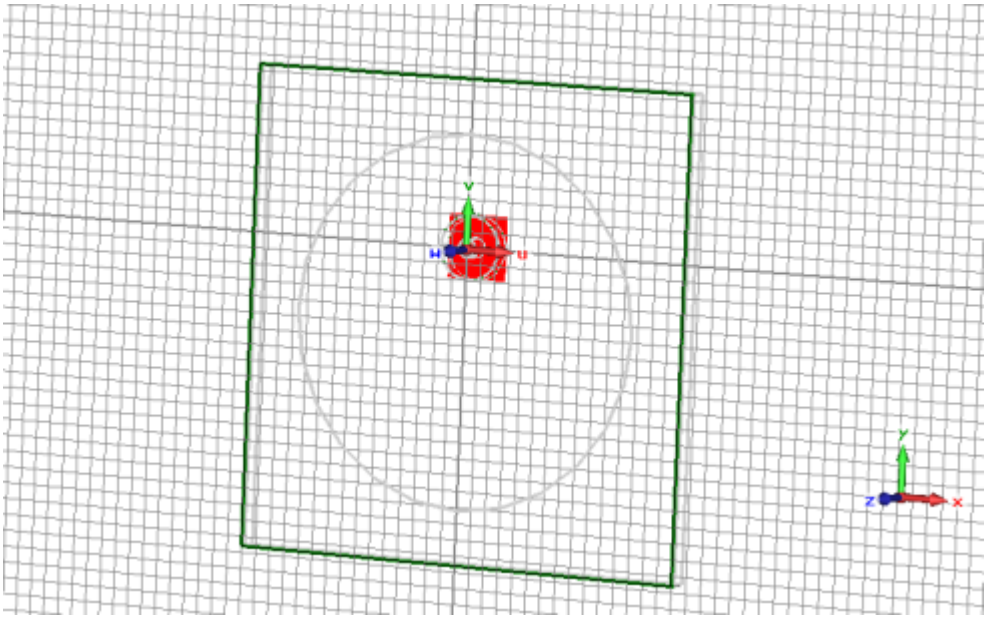


Please now open the waveguide dialog box (Simulation: ***Sources and Loads > Waveguide Port***) to define the port:



Here, you have to choose how many modes should be considered by the port. For a simple coax port with only one inner conductor, usually only the fundamental TEM mode is of interest. Thus, you should simply keep the default setting of one mode.

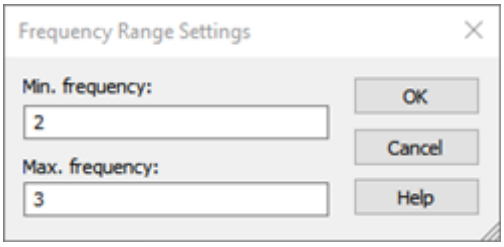
Please confirm your port settings with the *OK* button to finally create the port. After rotating the model again to the top face, your model should now look as follows (please use again *Ctrl+W* to toggle the wireframe visualization mode on and off):



Define the Frequency Range

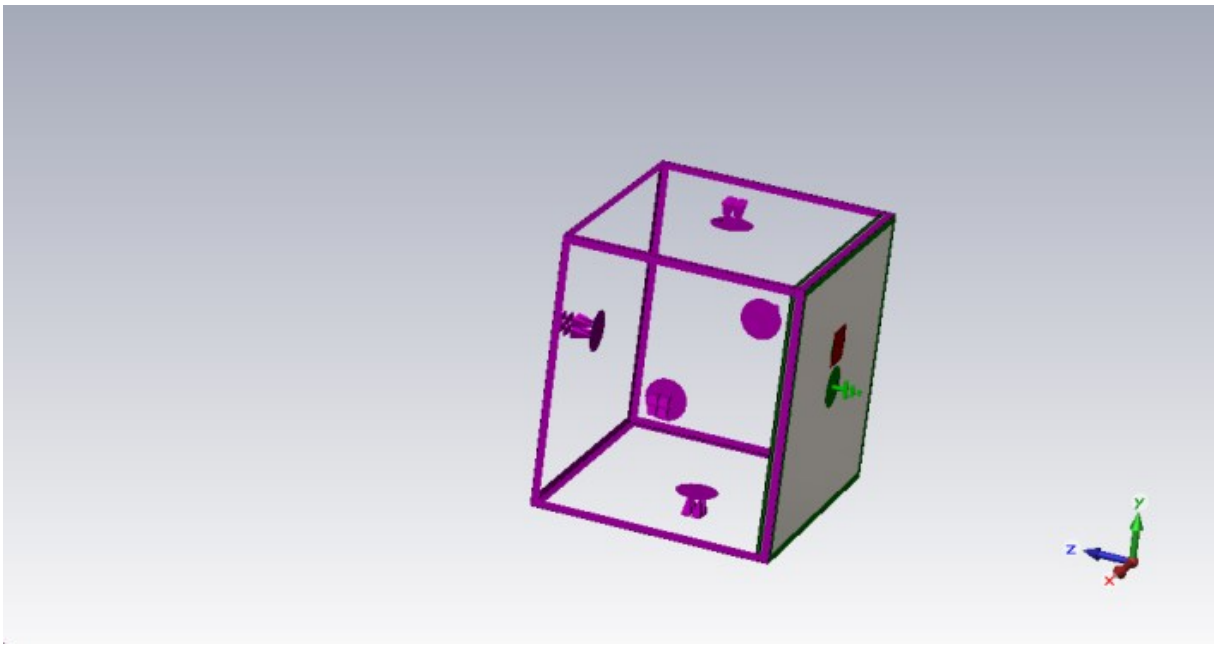
The frequency range for the simulation should be chosen with care. Different considerations must be made when using a transient solver or a frequency domain solver (see next chapter for details).

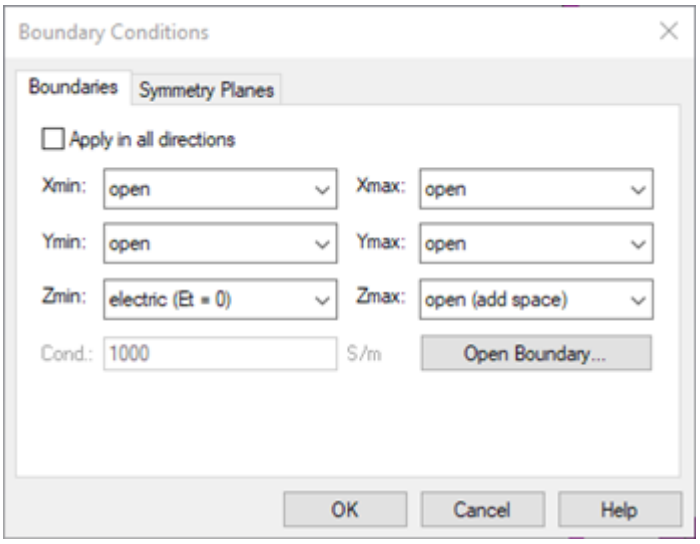
In this example, the S-parameters are to be calculated for a frequency range between 2 and 3 GHz. Open the frequency range dialog box (Simulation: ***Settings > Frequency***) and enter the range from 2 to 3 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz and is displayed in the status bar):



Boundary Conditions

Because the calculation domain is only a limited volume it is necessary to define boundary conditions that incorporate the influences of the outer space. Please open the Boundary Conditions dialog box by selecting (Simulation: ***Settings > Boundaries*** *),* where all currently selected boundary conditions are simultaneously displayed in the main view*:*





At the ground plane, an *electric* boundary condition has to be set that behaves like an infinite solid PEC brick. All other boundary planes are set to *open* or *open (add space)*; they realize free space behind their boundary planes. Free space means that the electromagnetic fields are absorbed at these boundaries with virtually no reflections as if they propagate in infinite empty space.

As a general rule, the open boundary conditions work best if they are at least 1/8 wavelength apart from the field source. Open (add space) already incorporates this rule and automatically adds the correct amount of background space to the structure.

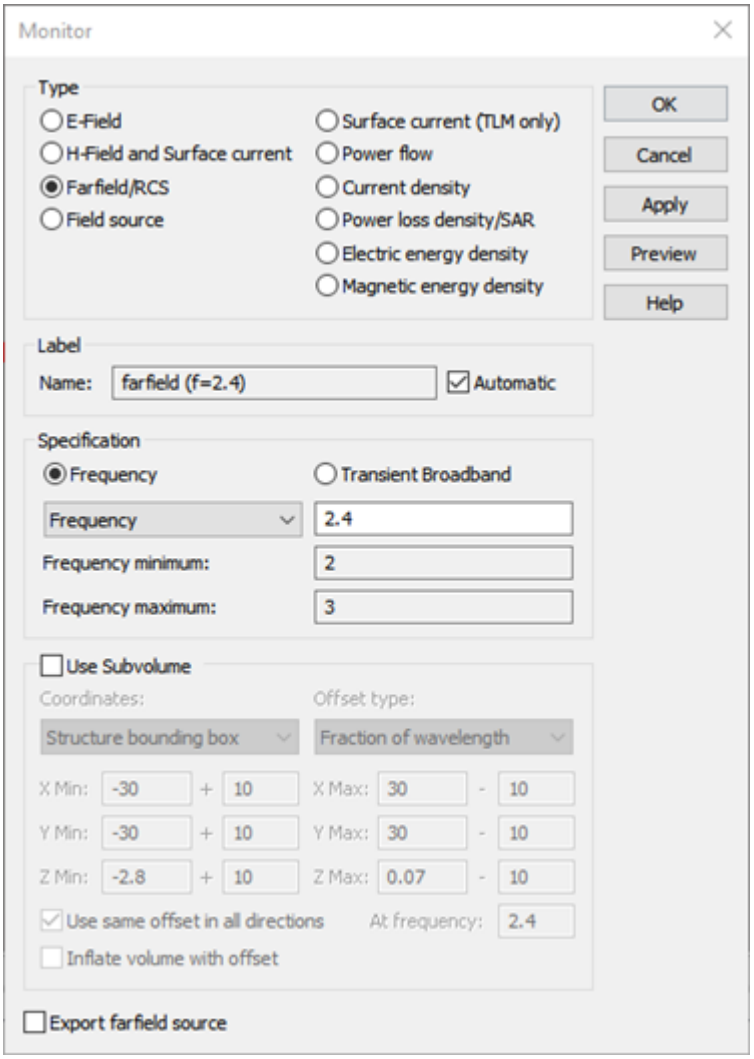
Because the open (add space) boundary condition only adds background material to the structure, it should not be used if there is material that crosses the boundary plane and should practically extend to infinity (such as the substrate and ground solids in this example). In these cases, the open boundary condition must be invoked.

Please close this dialog box without any further changes.

Define Farfield Monitor

Besides the S-parameters, the main result of interest for antenna devices is the farfield distribution at a given frequency. The solvers in CST Microwave Studio offer the possibility of defining several field monitors to specify at which frequencies the field data shall be stored.

Please open the monitor definition dialog box by selecting Simulation: ***Monitors > Field Monitor*** :



In this dialog box, you should first select the *Type* Farfield/RCS before specifying the frequency for this monitor in the *Frequency* field. Afterwards, press the *Apply* button to store the monitors data. Please

define a monitor at the frequency of 2.4 (with GHz being the currently active frequency unit). However, you may define additional monitors at other frequencies, each time pressing the *Apply* button to confirm the setting and add the monitor in the *Field Monitors* folder in the navigation tree. After the monitor definition is complete, please close this dialog box by pressing the *OK* button.

S-Parameter and Farfield Calculation

A key feature of CST Microwave Studio is the *Method on Demand* approach that allows specification of a simulator or mesh type that is best suited to a particular problem. Another benefit is the ability to compare the results from completely independent approaches. This strength is demonstrated in the following two paragraphs by calculating the S-parameters and the farfield of the constructed antenna device with both the transient and frequency domain solvers. The transient simulation uses a hexahedral mesh while the frequency domain calculation is performed with a tetrahedral mesh in this case. However, because both methods are self-contained, it is sufficient to work through only one of them. The chapter ends with a comparison of the two methods.

Transient Solver

Frequency Range Considerations for the Transient Solver

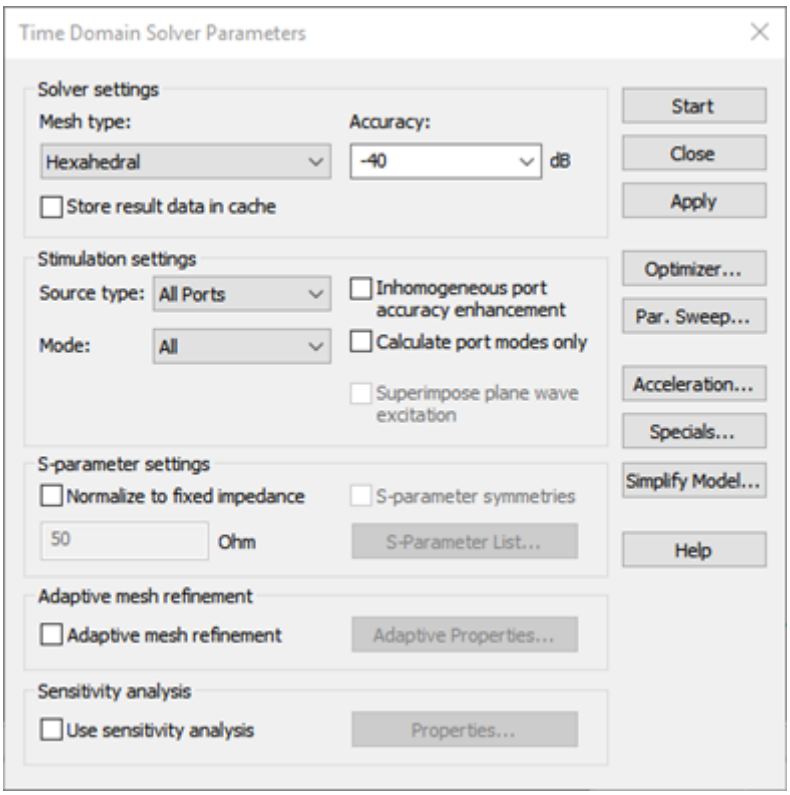
Using reasonably large bandwidths of 20% to 100% for the transient simulation is recommended. In this example, the S-parameters are to be calculated for a frequency range between 2 and 3 GHz. With a centre frequency of 2.5 GHz, the bandwidth (3 GHz - 2 GHz = 1 GHz) is 40% of the centre frequency, which is inside the recommended interval. Thus, you can simply choose the frequency range as desired between 2 and 3 GHz.

In a case where you just cover a bandwidth of less than 20%, you can increase the frequency range without losing accuracy. This extension of the frequency range could speed up your simulation by more than a factor of three!

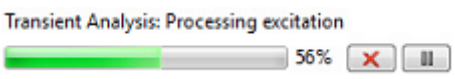
In contrast to frequency domain solvers, the lower frequency can be set to zero without any problems! The calculation time can often be reduced by half if the lower frequency is set to zero rather than e.g. 0.01 GHz.

Transient Solver Settings

The solver parameters are specified in the Transient Solver Parameters dialog box that can be opened by selecting Home: ***Simulation > Setup Solver > Time Domain Solver*** :



You can accept the default settings and press the *Start* button to run the calculation. A progress bar appears at the bottom of the main window, displaying information about the calculation's status. This progress window disappears when the solver has successfully finished.



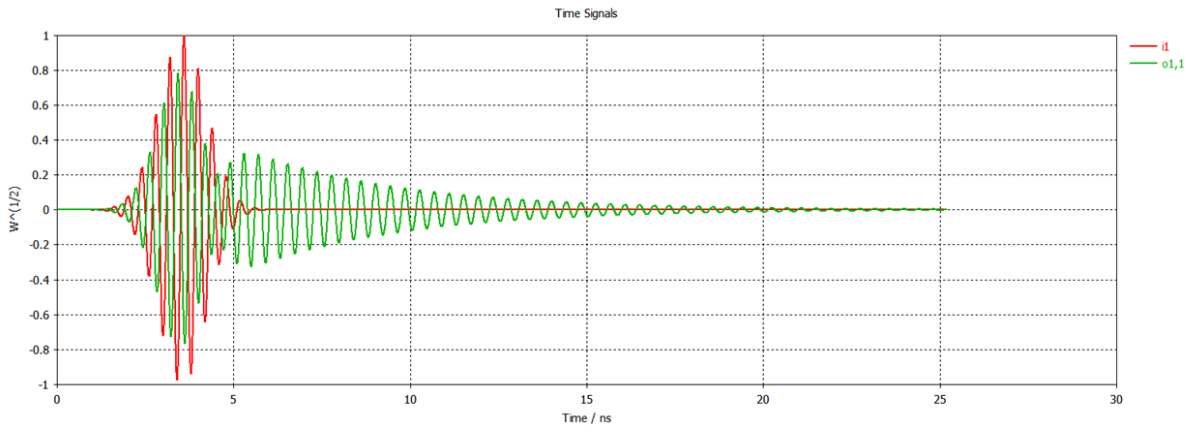
During the simulation, the Message Window will show some details about the performed simulation.

Transient Solver Results

1D Results (Port Signals, S-Parameters)

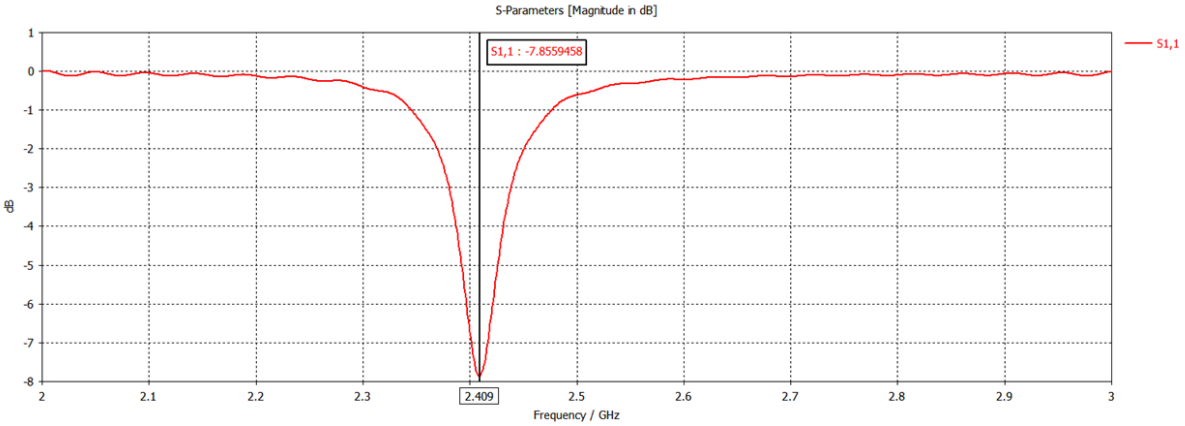
First, observe the port signals. Open the *1D Results* folder in the navigation tree and click on the *Port signals* folder.

It is possible to observe the progress of the results during the computation. In order to get the complete information, however, wait until the solver has finished.



This plot shows the incident and reflected wave amplitudes at the waveguide port versus time. The incident wave amplitude is called i1 (referring to the port name: 1) and the reflected wave amplitude is o1,1. As evident from the above time-signal plot, the patch antenna array has a strong resonance that leads to a slowly decreasing output signal.

A primary result for the antenna is the S11 parameter that will appear if you click on the *1D Results* > *S Parameters* folder from the navigation tree and selecting 1D Plot: ***Plot Type > dB*** . The following screenshot shows the reflection parameter:

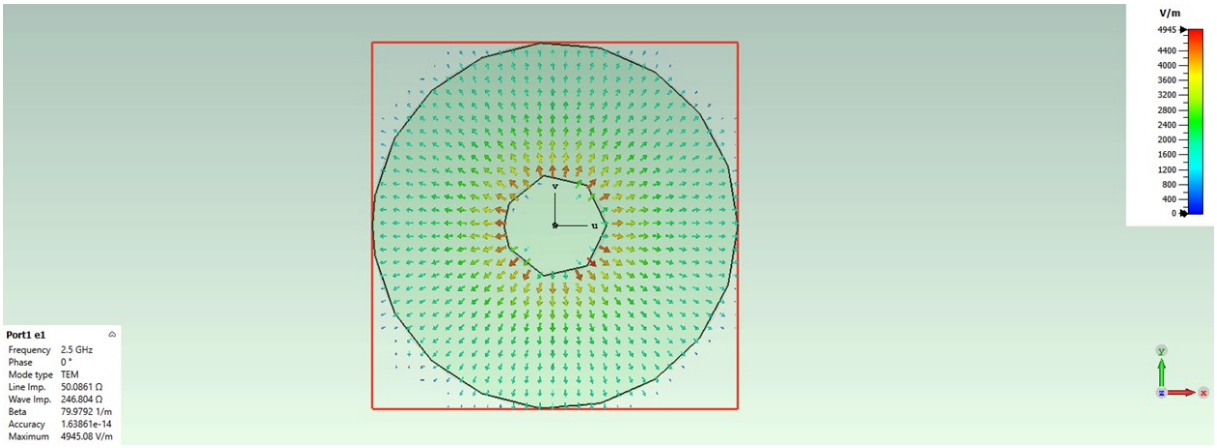


It is possible to precisely determine the operational frequency for the patch antenna. Activate the axis marker by pressing 1D Plot: ***Markers > Axis Marker*** or by pressing the right mouse button and selecting the *Show Axis Marker* option from the context menu. Now you can move the marker to the S11 minimum and pinpoint a resonance frequency for the patch antenna of about 2.4 GHz.

The ripples that appear in the reflection parameters result from the time signal not sufficiently decaying (review again at the time signal plot). The amplitude of the ripples increases with the signal amplitude remaining at the end of the transient solver run. However, these ripples do not affect the location of the resonance frequency and therefore can be ignored for this example.

2D and 3D Results (Port Modes and Farfield Monitors)

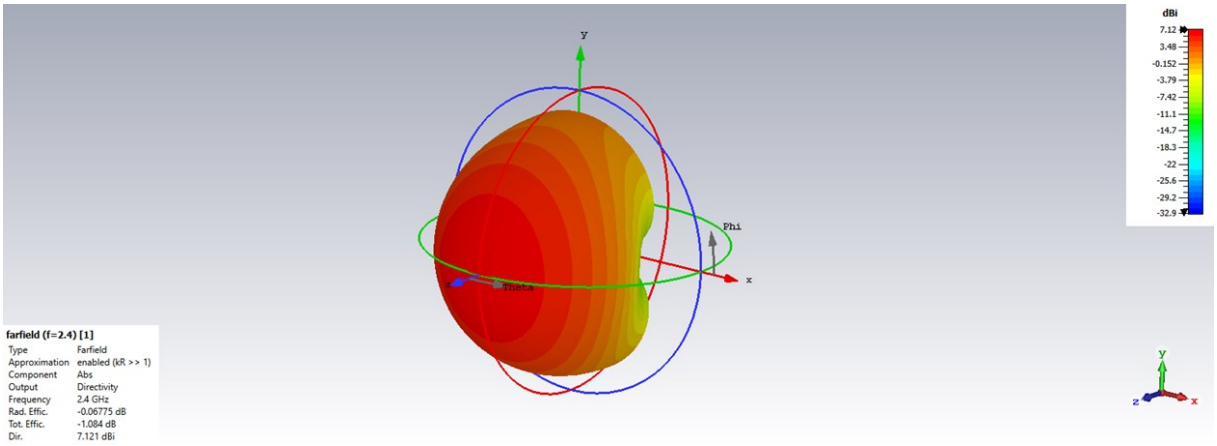
You should first inspect the port modes that can be easily displayed by opening the *2D/3D Results* > *Port Modes* > *Port1* folder from the navigation tree. To visualize the electric field of the fundamental port mode, click on the *e1* folder. After properly rotating the view and tuning some settings in the plot properties dialog box, you should obtain a plot similar to the following picture:



The plot also shows some important properties of the coaxial mode such as TEM mode type, propagation constant, line impedance, etc.

In addition to the resonance frequency, the farfield is another important parameter in antenna design.

The farfield solution of the antenna device can be shown by selecting the corresponding monitor entry in the *Farfields* folder from the navigation tree. For example, the farfield at the frequency 2.4 GHz can be visualized by clicking on the *Farfields* > *farfield (f=2.4) [1]* entry, showing the directivity over the phi and theta angle:



You have the option to change Farfield Plot: ***Plot Properties > Properties*** angle step to 5 degrees for a better angle accuracy of the plot.

As evident in the above figure, the maximum power is radiated in the positive z-direction. Note that there are several other options available to plot a farfield: the Polar plot, the Cartesian plot and the 2D plot.

Accuracy Considerations

The transient S-parameter calculation is primarily affected by two sources of numerical inaccuracies:

1. Numerical truncation errors introduced by the finite simulation time interval.
2. Inaccuracies arising from the finite mesh resolution.

How to minimize these errors and achieve highly accurate results, follows:

1. Numerical Truncation Errors Due to Finite Simulation Time Intervals

As a primary result, the transient solver calculates the time-varying field distribution that results from excitation with a Gaussian pulse at the input port. Thus, the signals at ports are the fundamental results from which the S-parameters are derived using a Fourier Transform.

Even if the accuracy of the time signals is extremely high, numerical inaccuracies can be introduced by the Fourier Transform that assumes the time signals have completely decayed to zero at the end. If the latter is not the case, a ripple is introduced into the S-parameters that affects the accuracy of the results. The amplitude of the excitation signal at the end of the simulation time interval is called truncation error. The amplitude of the ripple increases with the truncation error.

Please note that this ripple does not move the location of minima or maxima in the S-parameter curves. Therefore, if you are only interested in the location of a peak, a larger truncation error is tolerable.

The level of the truncation error can be controlled with the *Accuracy* setting in the transient solver control dialog box. The default value of 40 dB will usually give sufficiently accurate results. However, to obtain highly accurate results for antenna structures, it is sometimes necessary to increase the accuracy to 50 dB or 60 dB.

Because increasing the accuracy requirement for the simulation limits the truncation error and increases the simulation time, the accuracy requirement should be specified with care. As a general rule, the following table can be used:

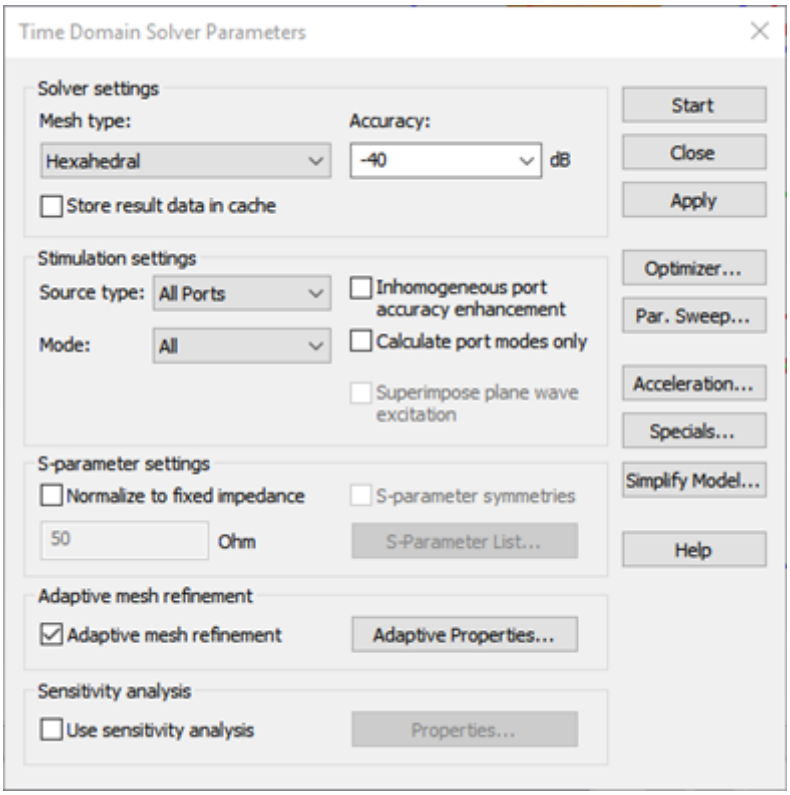
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

The following rule may be useful, as well: If you find a large ripple in the S-parameters, increase the solvers accuracy setting.

2. Effect of the Mesh Resolution on the S-parameters Accuracy

Inaccuracies arising from the finite mesh resolution are usually more difficult to estimate. The only way to ensure the accuracy of the solution is to increase the mesh resolution and recalculate the S-parameters. When the results no longer significantly change when the mesh density is increased, then convergence has been achieved.

In the example above, you have used the default mesh that has been automatically generated by an expert system. The accuracy of the results is most easily tested with the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (Home: ***Simulation > Setup Solver > Time Domain Solver*** *)*:

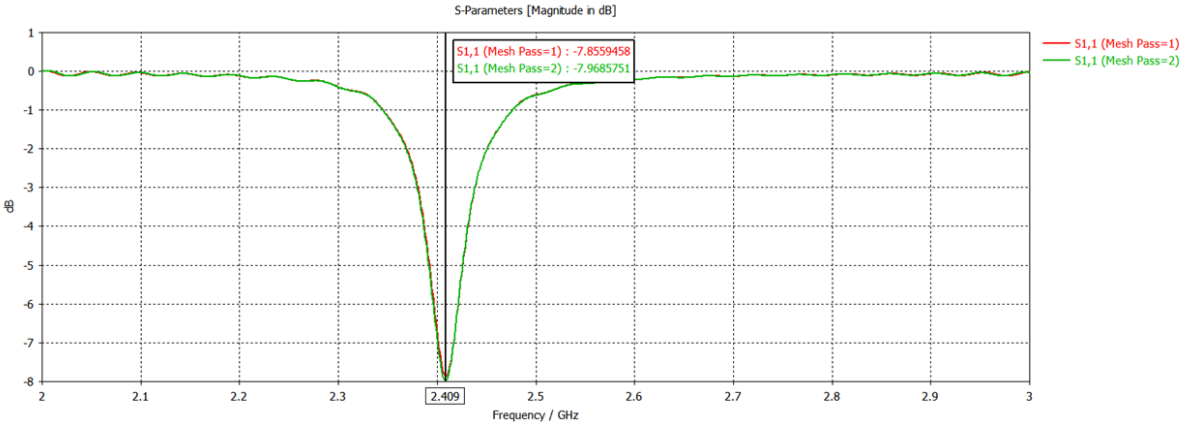


Please note that the previously selected template has changed the default settings to the energy-based adaptive strategy that is more convenient for planar structures.

Start now the solver again by pressing the *Start* button.

In this example, two adaptation passes are necessary to obtain a suitable result. This means that the maximum deviation of the S-parameters between the first and the second run is less than 2%.

The convergence process of the input reflection S1,1 during the mesh adaptation can be visualized by selecting *1D Results* > *Adaptive Meshing* > *S-Parameters* > *S1,1* from the navigation tree and select dB scale:



You have the option of reducing the accuracy limit for the mesh adaptation or starting the adaptation with a finer starting mesh resolution to obtain even more accurate results. However, these options will certainly increase the simulation time and might be more recommendable after the basic design state of the antenna device is finished. Another possibility for obtaining an impression of the reliability of a solution is to perform a second simulation with a completely different solver and mesh type, as will be shown in the following chapter.

Frequency Domain Solver

CST Microwave Studio offers a variety of frequency domain solvers that are specialized for different types of problems. They differ not only by their algorithms, but also by the type of grid on which they are based. The general-purpose frequency domain solvers are available for hexahedral grids as well as tetrahedral grids.

The availability of a frequency domain solver within the same environment offers a very convenient method of cross-checking results produced by the time domain solver with minimal additional effort.

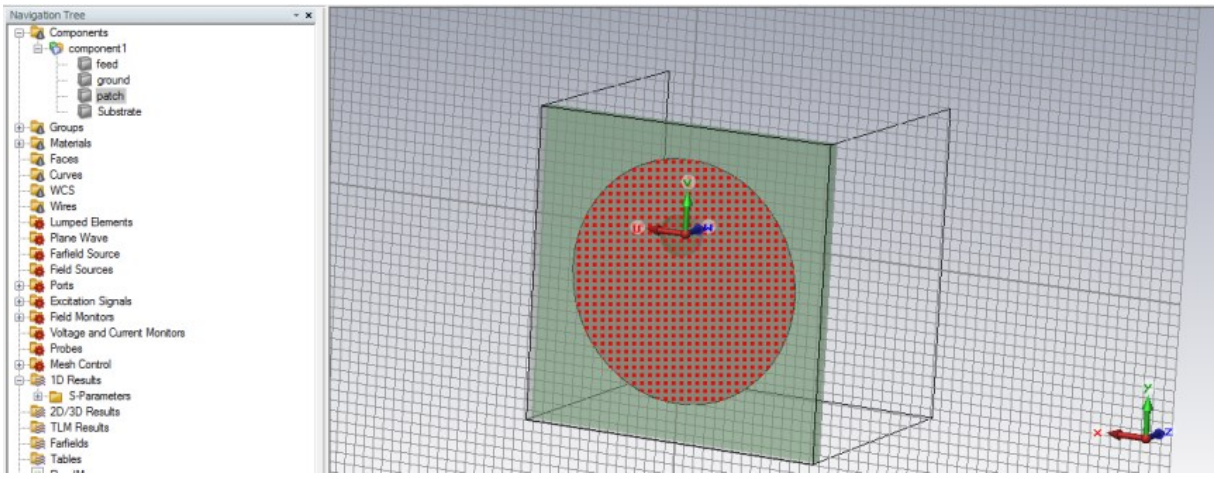
Making a Copy of Transient Solver Results

Before performing a simulation with a frequency domain solver, you may want to keep the results of the transient solver to allow for an easy comparison of the two simulations. To obtain the copy of the current results: select, for example, in the navigation tree *1D Results > S Parameters > S1,1*, then press *Ctrl+C* and *Ctrl+V.* The copy of the result curve will be created in the selected folder. The name of the copy will be *S1,1_1*. You may rename it to *S1,1_TD* with the *Rename* command from the context menu. Use *New Tree Folder* from the context menu to create an extra folder. Please note that at the current time it is not possible to make a copy of 2D or 3D results.

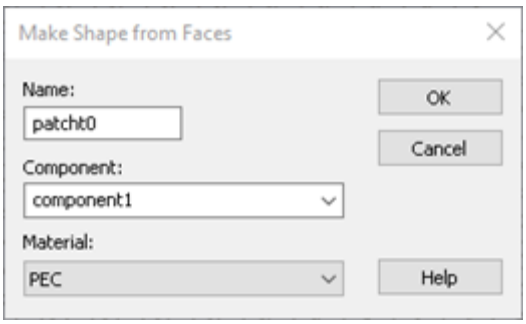
Optimize Structure for Tetrahedral Mesh

In the following section, the general-purpose frequency domain solver is applied to the tetrahedral mesh. This solver is less efficient if there are PEC sheets with very small, but non zero thicknesses, as represented by the antenna patch in our example. Because this thickness has a rather small influence on the results compared to a zero thickness, the patch is rebuilt as a PEC sheet, as shown in the following section.

First, select the patch in the navigation tree and then select the patch's bottom face using the face pick tool (Modeling: ***Picks > Picks > Pick Points, Edges or Faces***). Therefore, rotate the structure as shown in the picture below and double click on the patch.

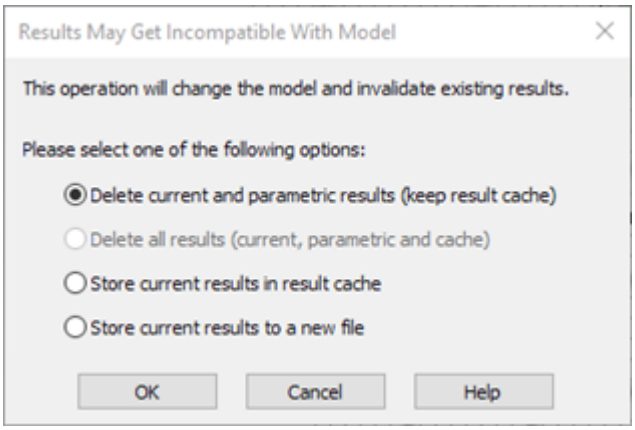


A PEC sheet is easily created from the selected face by applying Modeling: ***Shapes > Faces and Apertures***  ***> Shape from Picked Faces***:



Enter a suitable name for the new shape (*patcht0*) and confirm the creation by pressing the *OK* button. Finally, delete the old patch (*component1* > *patch*) so that only the newly created patch with zero height (*component1* > *patcht0*) remains.

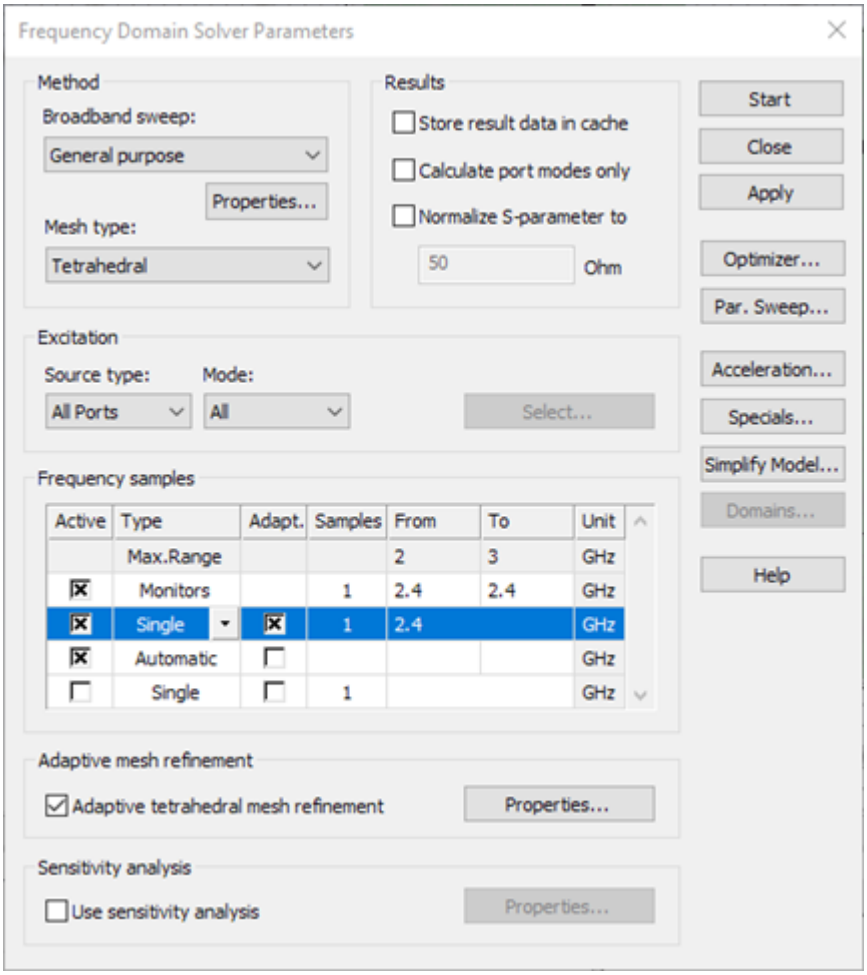
There may be old results present from the previous transient solver run that will be overwritten when changing the model. In this case, the following warning message appears:



Press *OK* to acknowledge deletion of the previous results.

Frequency Domain Solver Settings

In order to now start a frequency domain simulation the currently active solver has to be changed to Home: ***Simulation > Setup Solver > Frequency Domain Solver*** :

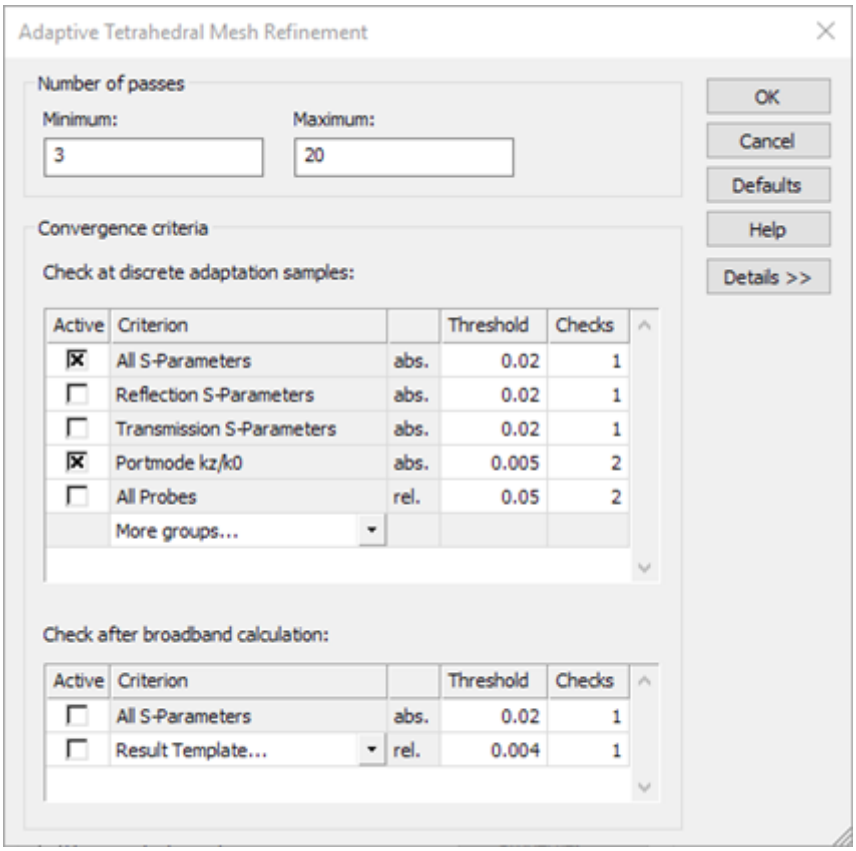


From the above chosen template the *General-Purpose* broadband sweep method and the *Tetrahedral* mesh type are set as default and can remain unchanged. S-parameters in the frequency domain are obtained by solving the field problem at different frequency samples. These single S-parameter values are then used by the broadband frequency sweep to get the continuous S-parameter values. With the

default settings in the *Frequency samples* frame, the number and position of the frequency samples are chosen automatically in order to fit the required accuracy limit throughout the entire frequency band.

Unlike the time domain solver, the tetrahedral frequency domain solver should always be used with the *Adaptive tetrahedral mesh refinement*. Otherwise, the initial mesh may lead to a poor accuracy. Therefore, the corresponding checkbox is activated by default. Please set the mesh adaptation frequency to the centre of the emitting band, to 2.4 GHz.

Furthermore, to ensure that the adaptation will satisfy the desired accuracy limit, the *Maximum number of passes* to 20 in the Adaptive Tetrahedral Mesh Refinement dialog box is increased by pressing the *Properties* button of the mesh refinement:

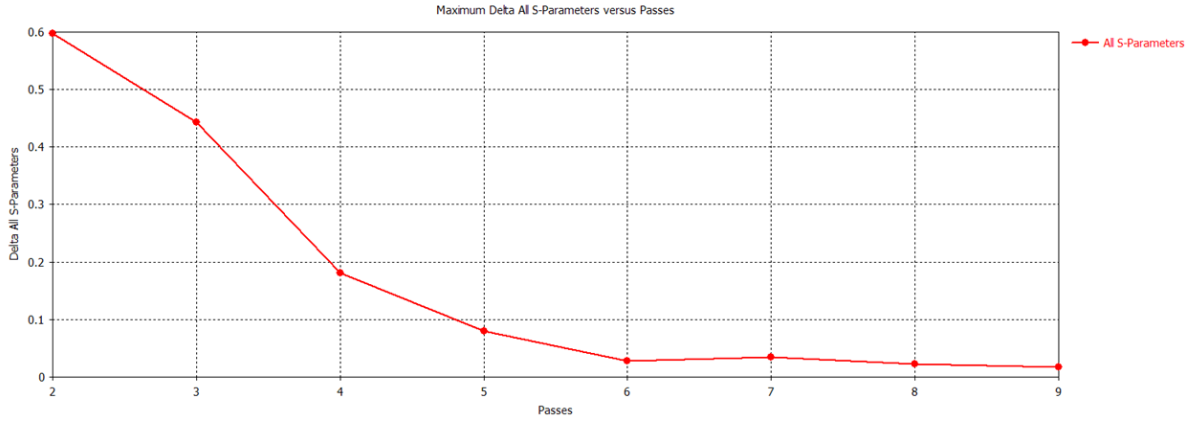


After confirming this setting with the *OK* button, everything is now ready; you may press *Start* to start the calculation. A progress bar and abort button appear in the *Progress* window, displaying some information about the solver stages. After the desired accuracy for the S-parameter has been reached, the simulation stops. When the simulation has finished or has been aborted, both items disappear again. During the simulation, the *Messages* window will display details about the performed simulation.

Frequency Domain Solver Results

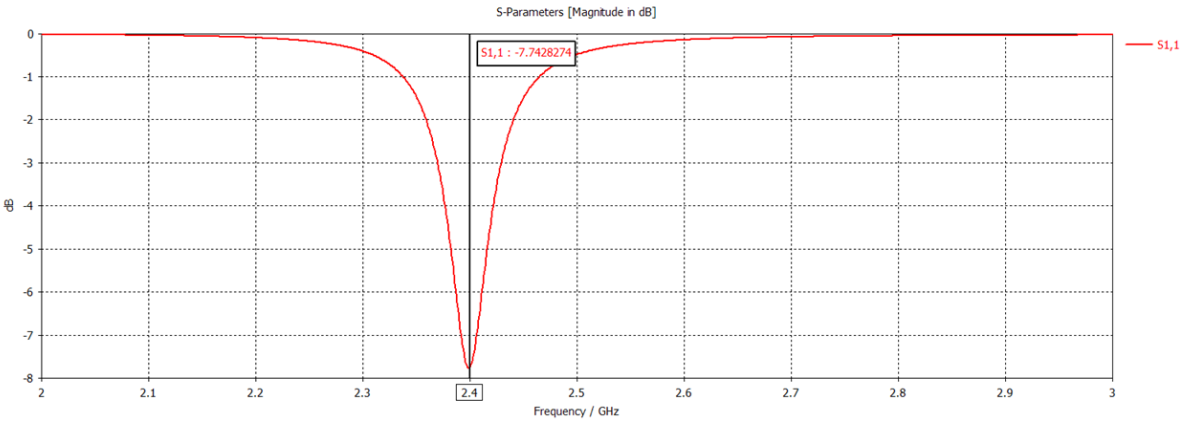
1D Results (S-Parameters)

You can visualize the maximum difference of the S-parameters for two subsequent passes (and for the specific adaptation frequency) by selecting *1D Results* > *Adaptive Meshing* > *f=2.4* > *S-Parameters* > *Delta* > *All S-Parameters* from the navigation tree. In the following picture the adaptation for the 2.4 GHz frequency is shown:



As evident from the above diagram, several passes of the mesh refinement were required to obtain highly accurate results within the given accuracy level.

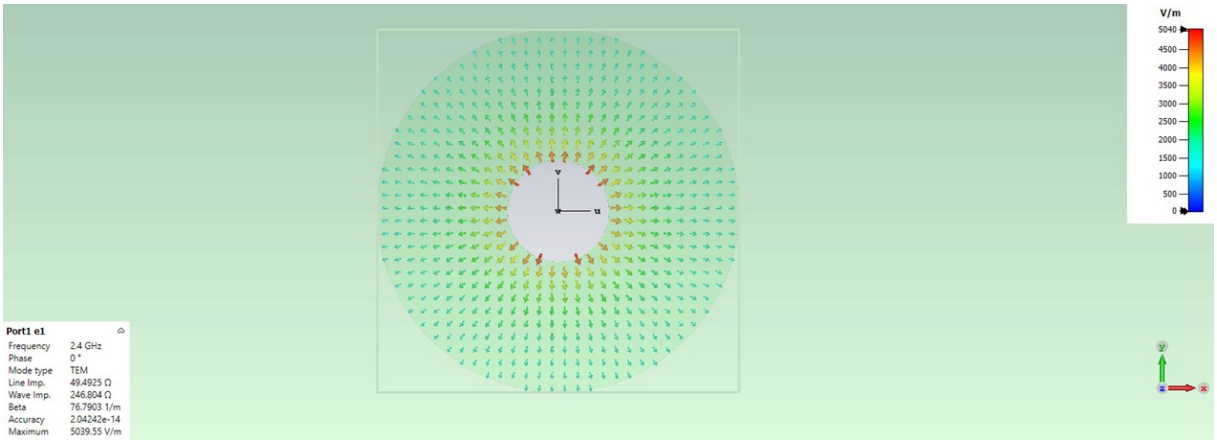
As for the transient solver run, you can view the S-parameters by selecting *1D Results* > *S-Parameters* in the navigation tree and selecting 1D Plot: ***Plot Type > dB*** for dB representation.



It is possible to precisely determine the operational frequency for the patch antenna. Activate the axis marker by pressing 1D Plot: ***Markers > Axis Marker***. Now you can move the marker to the S1,1 minimum and pinpoint a resonance frequency for the patch antenna of about 2.4 GHz.

2D and 3D Results (Port Modes and Farfield Monitors)

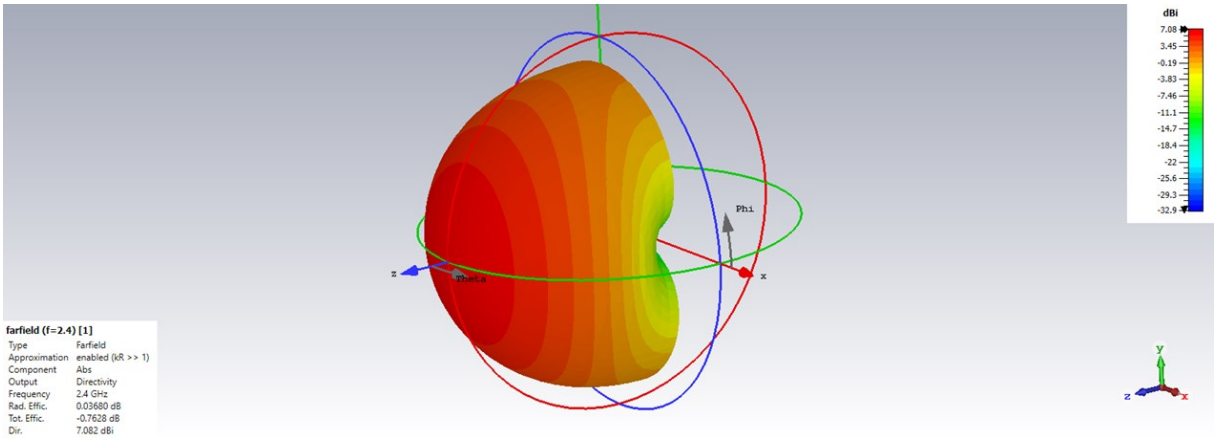
Finally, you can observe the 2D and 3D field results. You should first inspect the port modes that can be easily displayed by opening the *2D/3D Results* > *Port Modes* > *Port1* folder from the navigation tree. To visualize the electric field of the port mode, please click on the *e1* folder. Open the Select Port Mode dialog box by choosing *Select Mode Frequency* via the context menu and ensure that the frequency is 2.4 GHz. After properly rotating the view and tuning some settings in the plot properties dialog box, you should obtain a plot similar to the following picture:



The plot also shows some important properties of the mode such as mode type, propagation constant, and line impedance.

In addition to the resonance frequency, the farfield is another important parameter in antenna design.

The farfield solution of the antenna device can be shown by selecting the corresponding monitor entry in the *Farfields* folder from the navigation tree. For example, the farfield at the frequency 2.4 GHz can be visualized by clicking on the *Farfields* **>** *farfield (f=2.4) [1]* entry, showing the directivity over the phi and theta angle (the plot settings such as colour ramp can be accessed selecting Farfield Plot: ***Plot Properties > Properties*** )

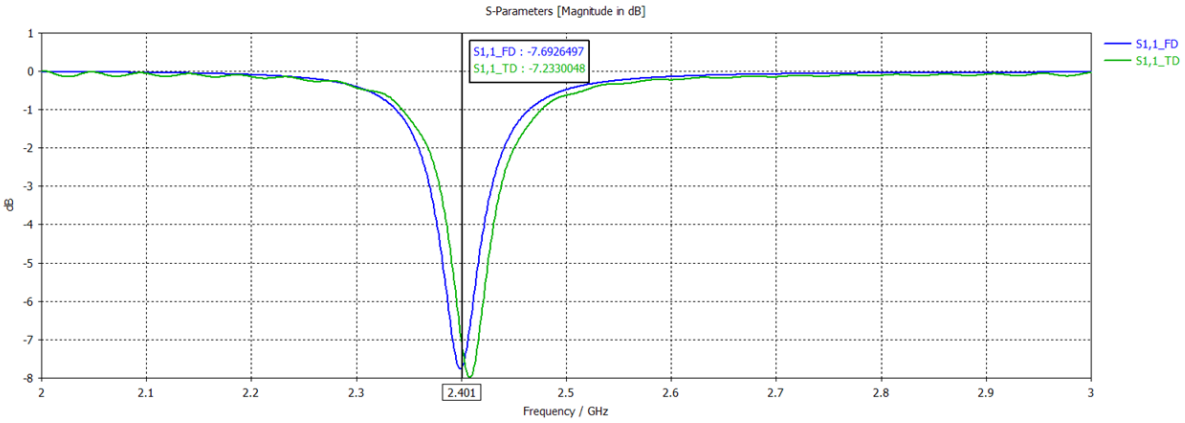


You have the option of changing the Farfield Plot: *Plot Properties* > *Properties* angle step to 5 degrees for a better angle accuracy of the plot.

You can see that the maximum power is radiated in the positive z-direction. Note that there are several other options available to plot a farfield: the Polar plot, the Cartesian plot and the 2D plot.

Comparing the results

The following figure shows the comparison between the S-parameter result of the transient hexahedral and the frequency domain tetrahedral simulation:



Obviously, the result curves are quite similar to each other; they are not, however, precisely identical. When comparing two completely different numerical methods, you should keep in mind that transient analysis is based on a hexahedral grid and frequency domain analysis is based on tetrahedral grid. Consequently, the results show some differences in detail but the same overall qualitative behaviour and therefore provide a satisfying validation. The differences are due to dispersion effects and this example is especially affected by the circular-shaped structural elements. Here, for example, the starting mesh resolution of the tetrahedral grid strongly influences the final accuracy of the simulation.

If more accurate results are needed, both simulations can be calculated with a finer starting mesh resolution that would lead to a better convergence.

Patch Antenna Array

Starting from the single patch antenna constructed in this chapter, the extension to a four-element antenna array will now be demonstrated. Here, purely the farfield calculations will be taken into account. All other results can be analysed in a manner similar to the first part of this document.

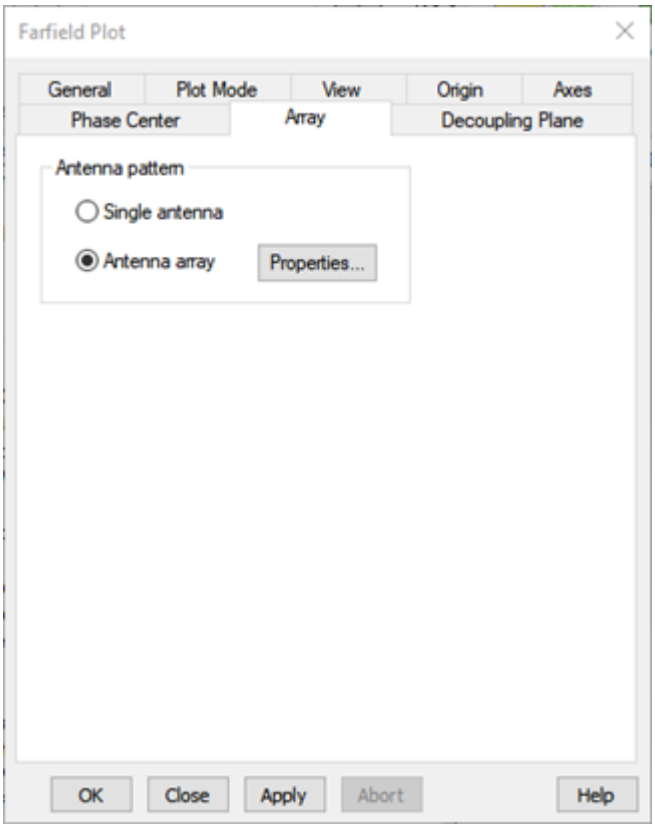
Please note that all following steps are performed using the transient solver. The procedure, however, can be applied to the frequency domain solver and its farfield results presented in the previous chapter.

The array calculation will be done in three steps: First, the antenna array feature is applied to the farfield results of the single patch antenna. Afterwards, the structure is physically expanded to the 2x2 antenna array and a result combination of a sequential excitation as well as a simultaneous excitation to obtain the radiation characteristics is used.

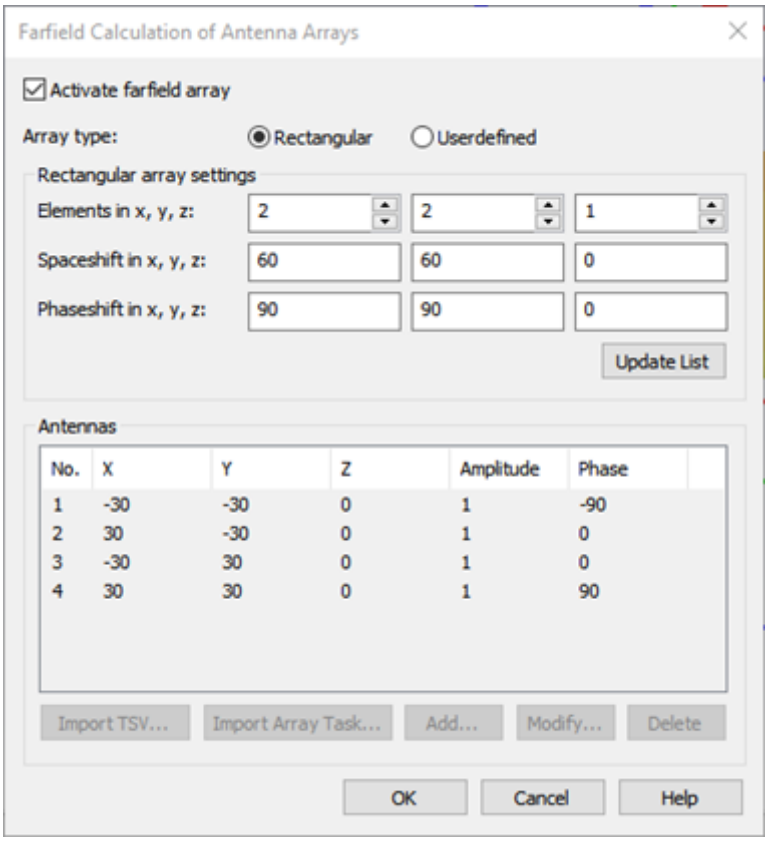
Antenna Array Calculation

Starting from the farfield results of the simulated single patch antenna, it is possible to calculate the farfield distribution for an arbitrary antenna array consisting of identical antenna elements as a post-processing step.

Later, the antenna example by constructing a 2x2 antenna array pattern will be expanded, so it is preferable to apply the mentioned array calculation feature to the same rectangular array dimension. Click on the *Farfields* **>** *farfield (f=2.4) [1]* entry in the navigation tree and open the corresponding dialog box by pressing Farfield Plot: ***Plot Properties > Properties*** , then the tab *Array* and selecting the radio button *Antenna array.*



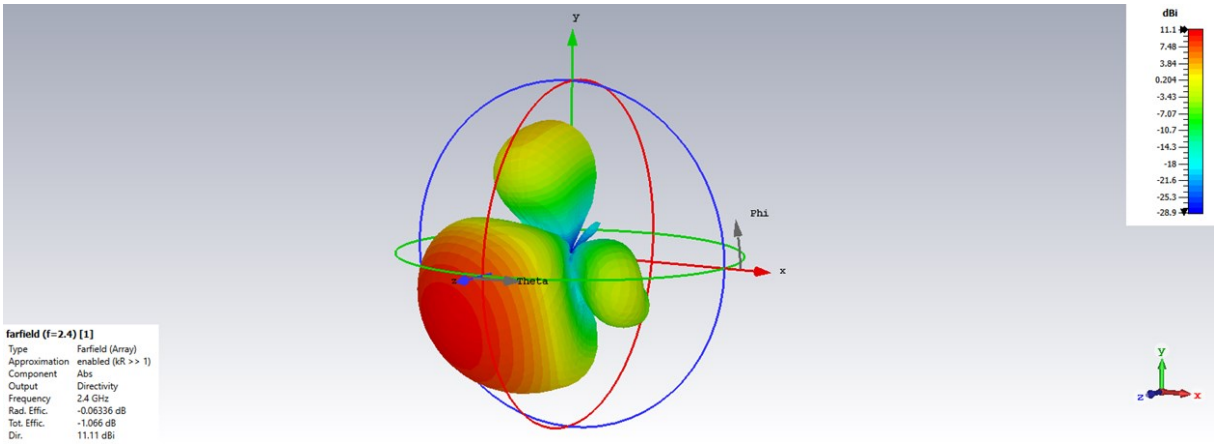
Afterwards, select *Properties* and insert the values to create the rectangular 2x2 array pattern in the XY-plane with a spatial shift of 60 mm (due to the dimension of the antenna substrate) and a phase shift of +90 degrees:



By pressing the *Update Antenna List* button, you can display the coordinates and their respective amplitudes and phase values in the *Antenna list*, as seen in the above dialog box.

The *Antenna list* can be modified if the radio button *Edit antenna list* is activated. This means that not only rectangular arrays with constant space and phase shift can be calculated, but by adding and modifying single antennas any array pattern with arbitrary amplitude and phase values can be defined.

Please confirm with the *OK* button to calculate the resulting farfield of the defined array pattern. The following screenshot demonstrates that the array arrangement together with the constant phase shift of +90 degrees produce not only a constructive superposition with an increased directivity, but also a slight rotation of the main loop in negative x and y directions:



You have the option of changing the Farfield Plot: *Plot Properties* > *Properties* angle step to 5 degrees for a better angle accuracy of the plot.

Please feel free to define some more array patterns to analyse the resulting changes in the farfield distribution. Here, you have a fast and efficient way to design various antenna arrays without necessitating the restart of a calculation.

Regarding the following calculations: the antenna array feature should now be disabled; therefore, please reset the Antenna Array selection in the Farfield Plot dialog box Farfield Plot: ***Plot Properties > Properties*** back to *Single antenna*.

Geometric Construction Steps

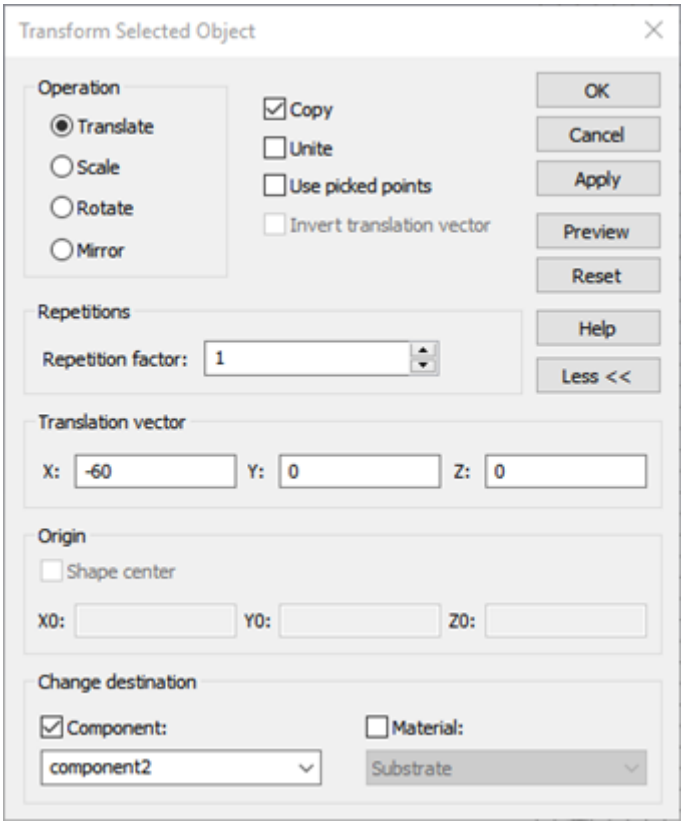
The construction of the array is based on the translation feature for selected objects that will be applied to the complete component of the single patch antenna. This procedure is also applicable to single objects or an arbitrary multiple selection of objects, even when selected from different components.

Before you start, please switch off the local coordinate system, if necessary, by clicking Modeling: ***WCS > Local WCS***

Now, please select component1 from the Components folder in the navigation tree and perform a transformation of the complete component (press the toolbar icon Modeling: ***Tools > Transform > Translate*** *)*. Choose the *Translate* function in the upcoming Transform Selected Object dialog box and enter the value of -60 (referring to the previously defined units of mm) for the x-component of the translation vector, corresponding to the structural dimension.

In order to gather all translated shapes as a new component, please activate the *Copy* and *Component* check boxes and then select [New Component] from the *Component* dropdown menu (it could be necessary to click on the *More* button) to create the destination group component2.

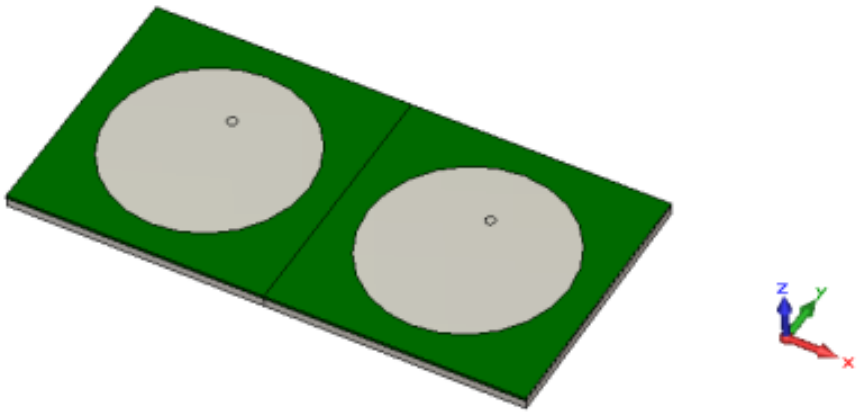
Finally, perform the translation operation with the *OK* button:



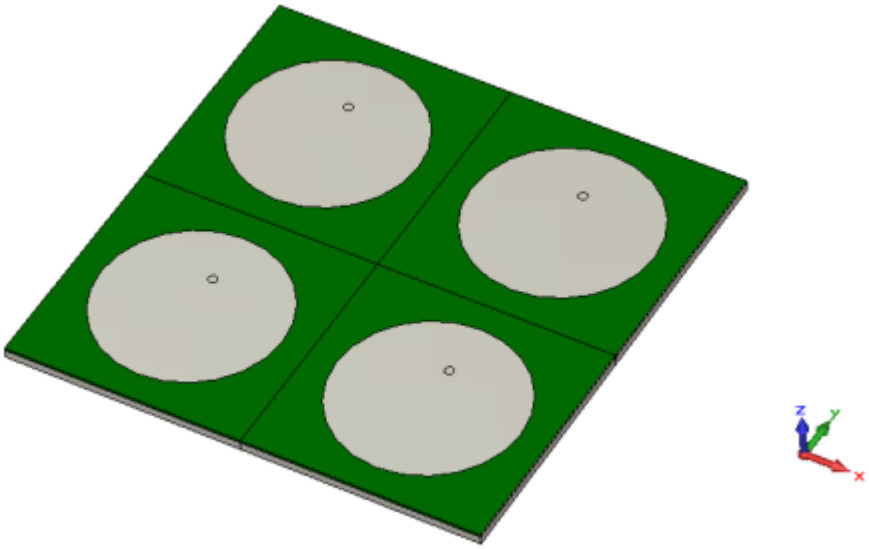
Because the structure will change when this operation is performed, you will be informed that the previously calculated results need to be deleted. Confirm this message by clicking *OK*.

Now, CST Microwave Studio copies the selected component to the coordinates of its translated position and creates a new component, containing all translated single shapes of the patch antenna.

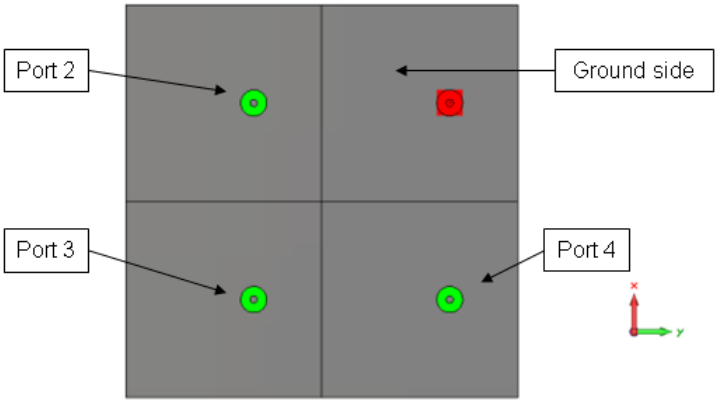
The resulting structure, consisting of two identical patch antennas, will then look as follows:



Please repeat the described transformation in the negative y-direction for both components (component1 and component2), again with a space shift of 60 mm. The occurring intersection dialog box can be solved by activating the *None* button, so that the final array pattern will be created as follows:



To complete the model, the remaining ports of the newly created patch antennas must be created. This is carried out in a similar manner to the definition of the first port by picking the corresponding port area (the bottom face of the respective substrate cylinders) and defining a port, again with only one mode.

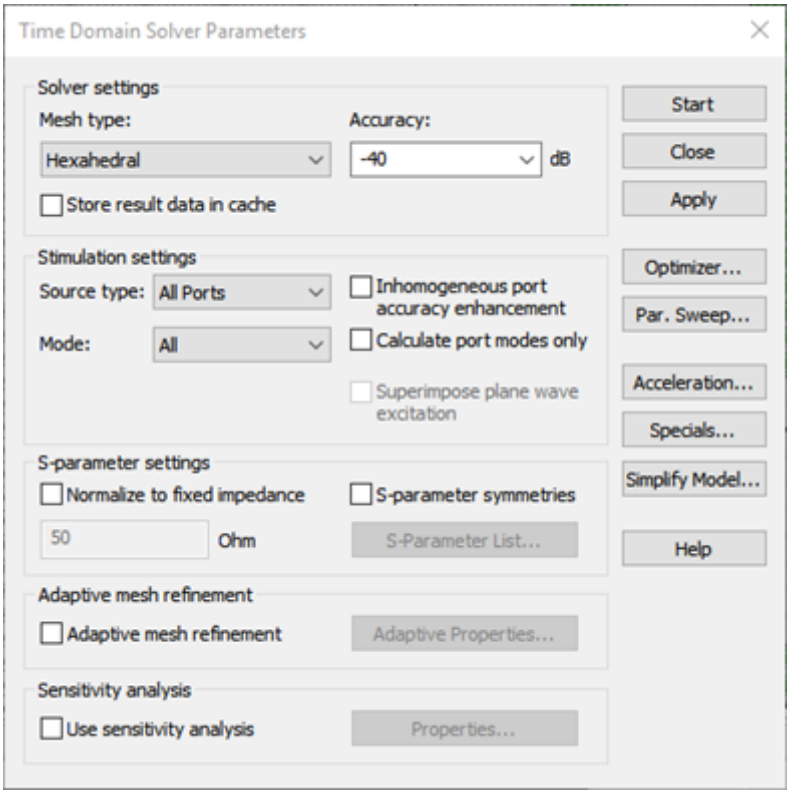


Combine Results

In order to receive farfield results where all patches are driven simultaneously, the results are combined in a post-processing step. This means that, at first, each port is excited individually one after another, after which arbitrary combinations of these excitations can be defined with respect to different amplitude and phase values.

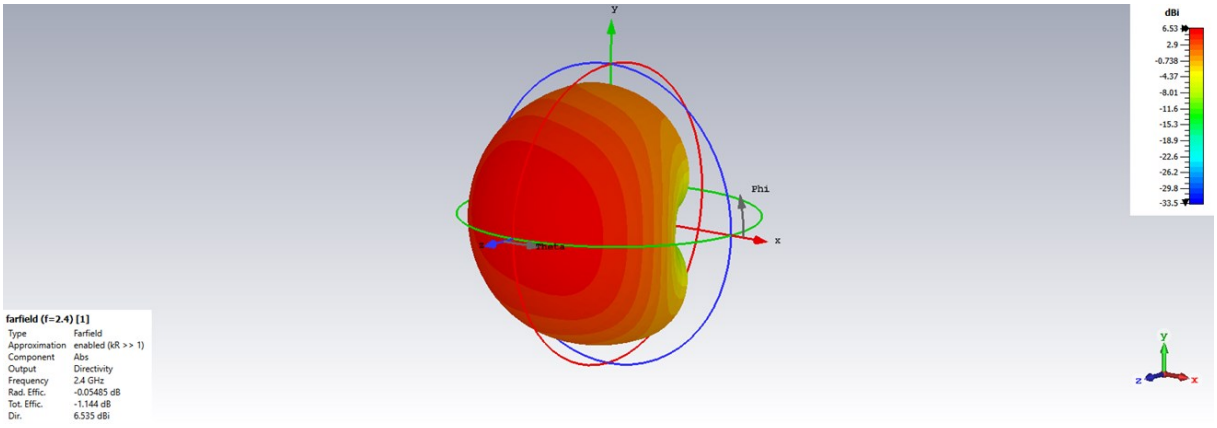
Define the Solvers Parameters and Start the Calculation

As described earlier, the solvers parameters are specified in the Transient Solver Parameters dialog box that can be opened by selecting Home: ***Simulation > Setup Solver > Time Domain Solver*** .



Because all ports should be calculated, you must be certain that *All Ports* is selected in the *Source type* dropdown list. The *Adaptive mesh refinement* checkbox must also be deactivated. Now you can press the *Start* button to run the calculation. Again, the progress bar appears indicating, in addition to the calculation's status, the current solver cycle.

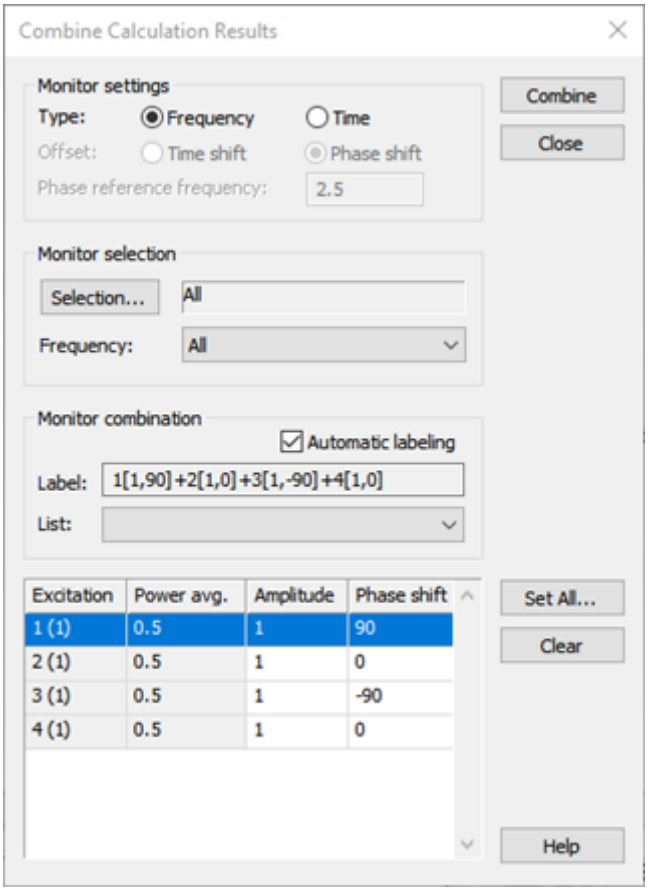
After the transient solver has finished, please observe the farfield of a single patch from the array. It appears quite similar to the farfield distribution of the previously calculated single patch antenna. The following screenshot shows the directivity over the angles theta and phi. Click on the *Farfields* > *farfield (f=2.4) [1]* folder in the navigation tree to bring your plot into view:



Combine Results

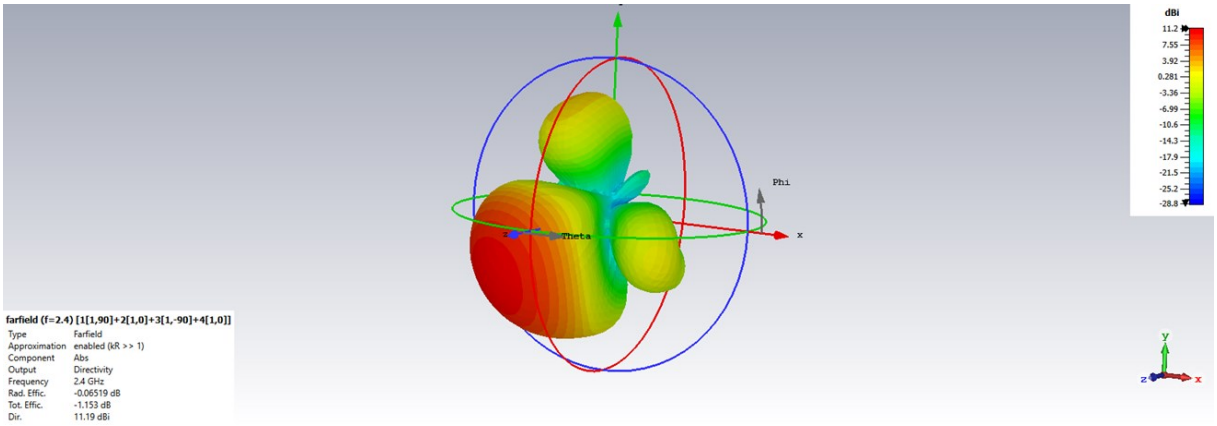
The most interesting results, however, are those where all patches are driven at one time with a pre-determined amplitude and phase variation being taken into account. This can be produced by CST Microwave Studio using the Combine results option. Press Post-Processing: ***Combine Results*** to bring the Combine Calculation Results window into view.

Now you have the option of changing the amplitude and phase excitation in the port mode list. In order to compare the farfield result with the previously demonstrated antenna array feature, identical antenna settings will be defined, i.e. the first antenna receives a phase shift of +90 degrees, the third antenna of -90 degrees and the two remaining antennas 0 degrees:



The corresponding monitor label will be generated automatically due to the settings of the port mode combination; however, by disabling the *Automatic labelling* button you are able to enter a label of your choice.

After confirming your settings with the *Combine* button, you will observe a new farfield subfolder *farfield (f=2.4) [1[1,90]+2[1,0]+3[1,-90]+4[1,0]]* in the navigation tree. By clicking on it you will display the following farfield distribution that is quite similar to the result of the antenna array feature:



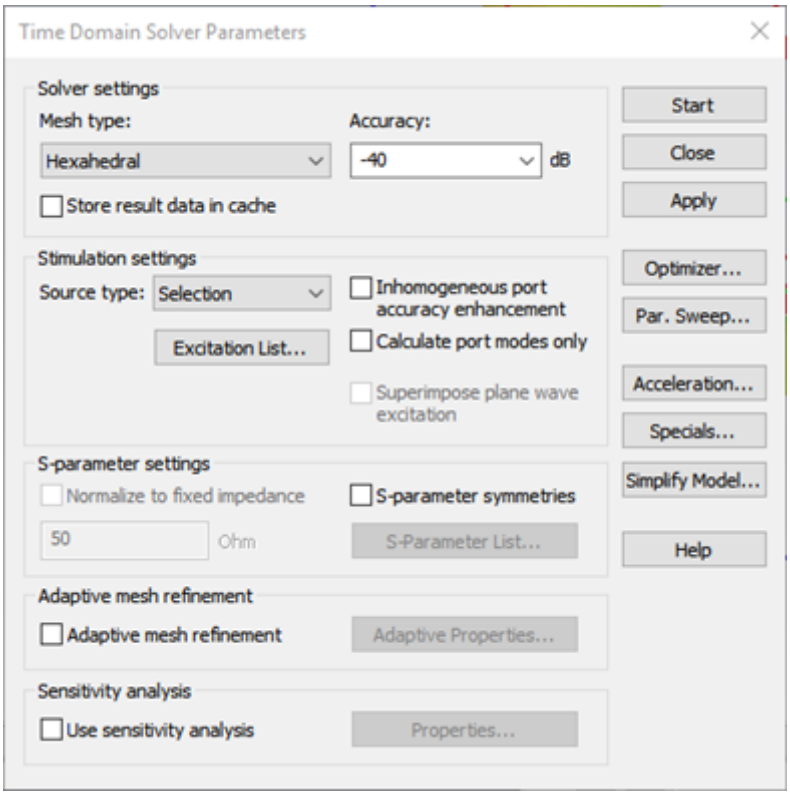
As you can see, the combination of results is time efficient because you do not have to restart the solver; just enter the amplitude and phase and view the combined generated farfield plot.

Simultaneous Excitation

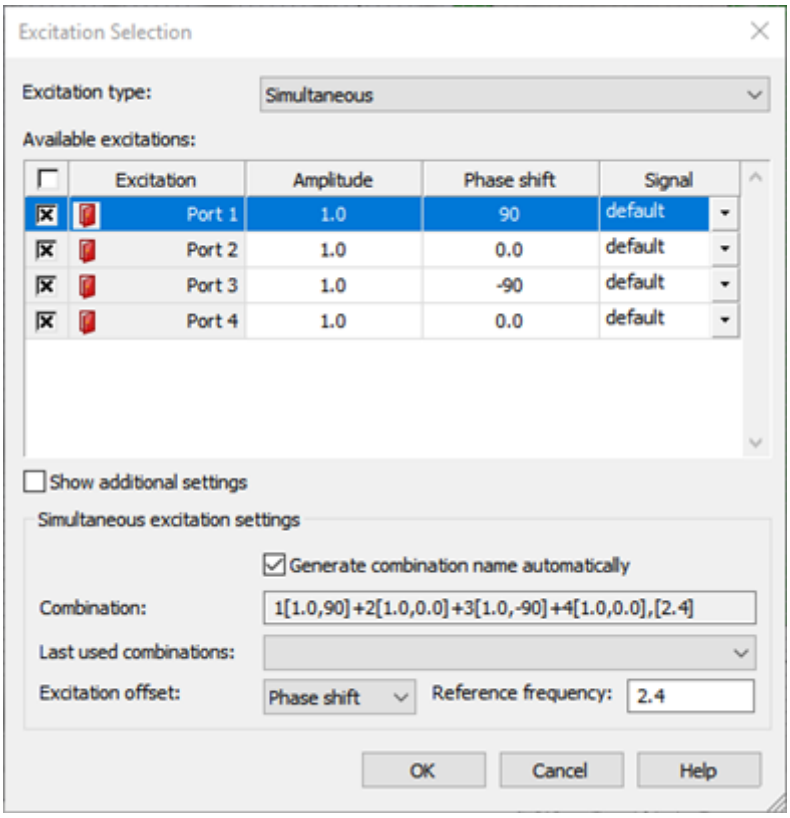
Another possibility for obtaining the farfield result of the array pattern is the excitation of all four ports simultaneously. In contrast to the combine result facility, here only one transient solver run is required. However, the phase shift relationship between the antenna elements has to be known before the solver is started.

The definition of a phase shift in connection with a simultaneous excitation of different ports will be converted, using the defined reference frequency, into a constant time shift between the port excitation signals. Therefore, in order to examine a farfield result, the reference frequency must be identical to the frequency of the farfield monitor.

In order to define the simultaneous excitation you must choose *Selection* in the *Source type* dropdown list in the Transient Solver Parameters dialog box (selecting Home: ***Simulation > Setup Solver > Time Domain Solver***):



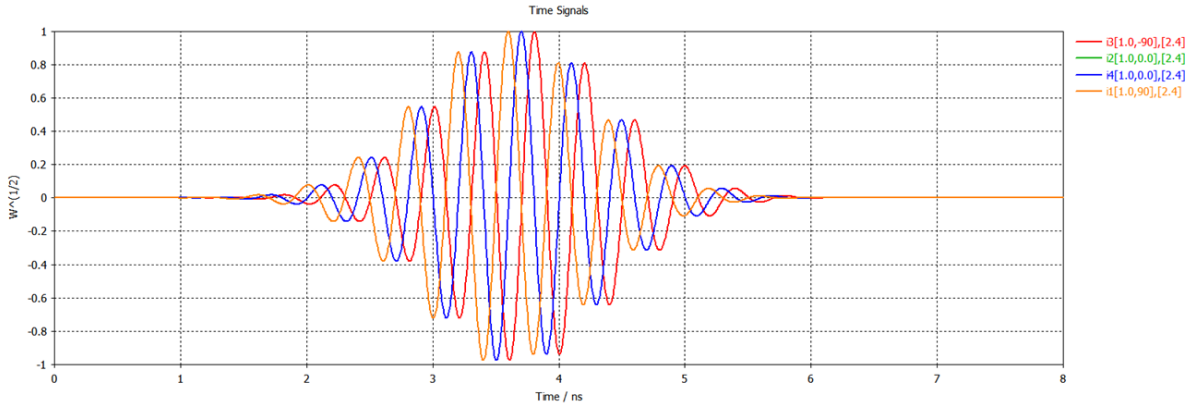
Pressing the *Excitation List* button opens the following dialog box, where the selective and simultaneous port excitation settings can be defined:



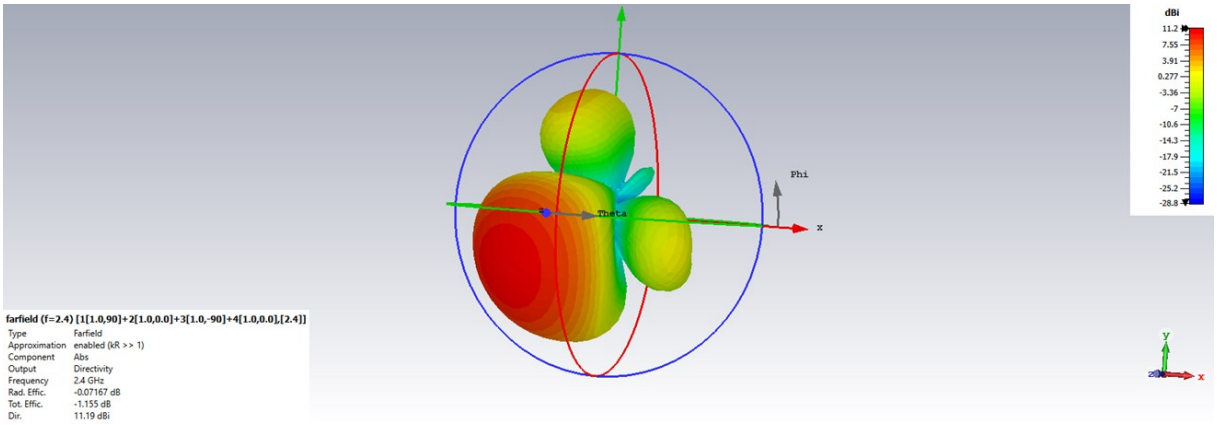
Since all four existing ports will be excited, please activate all available check buttons in the port mode list. In order to enter the amplitude and phase values, you must first activate the *Simultaneous excitation* by checking the corresponding *Activate* button. Please activate the *Phase shift* checkbox and enter a reference frequency of 2.4 GHz so that the result will be compatible to the combined farfield monitor in the previous chapter. Again, all antennas are driven with an amplitude value of one and phase values of 90, 0, -90, 0 degrees. Please double-check your settings against the dialog box above and confirm with the *OK* button.

Finally, you can press the *Start* button in the Transient Solver Parameters dialog box (selecting Home: ***Simulation > Setup Solver > Time Domain Solver***) to start the simultaneous calculation. Again, the progress bar appears indicating, in addition to the calculation's status, the simultaneous solver calculation.

Observe the input time signals (after the calculation has finished, you can select specific curves in your 1D plot view by choosing *Port Signals* > *Select Curves* to open the corresponding dialog box). In the picture below you can see the time delay of the different input signals due to the defined phase shifts of the ports:



After the transient solver has finished, you will find the resulting farfield information in the *Farfields* > *farfield (f=2.4) [1[1.0,90]+2[1.0,0.0]+3[1.0,-90.0]+4[1.0,0.0],[2.4]]* folder. Clicking on it will show the following farfield plot of the directivity over the angles theta and phi. As expected, the result is very similar to that produced by combining the results of the four single port excitations in the previous chapter:



If some ports are stimulated simultaneously the magnitude and phase of the normalized signal spectrum at the ports are recorded (F-Parameters) as well as so-called active S-Parameters for the excited ports.

Getting More Information

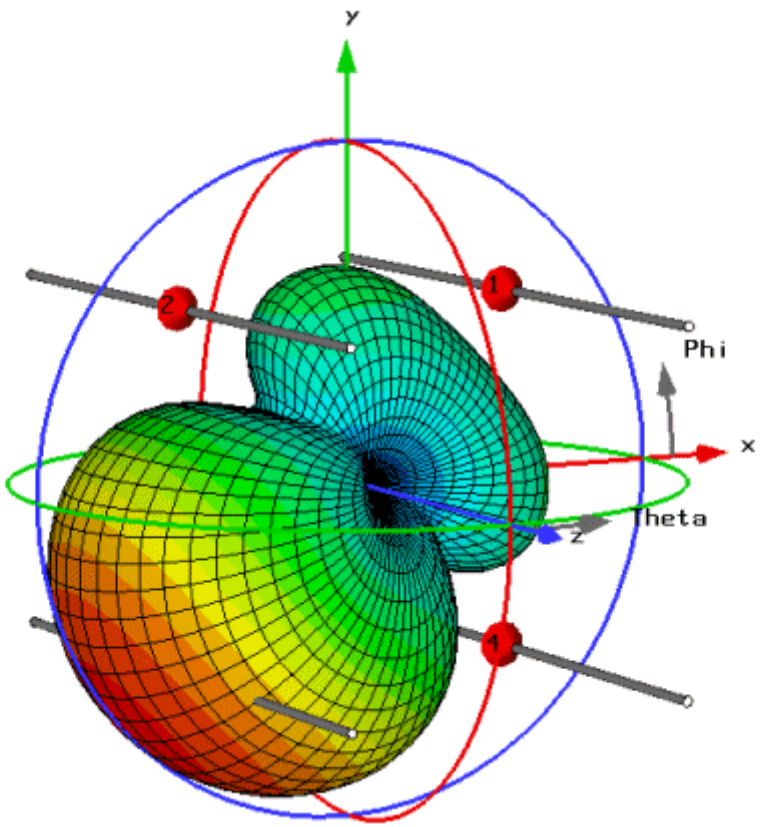
The following topics have been covered:

1. General modelling considerations, using templates, etc.
2. Model a planar structure by using the extrude tool, creating the substrate, circular patch antenna, and a coaxial feed.
3. Define waveguide ports.
4. Define frequency range.
5. Define farfield monitors.
6. Start the transient or the frequency domain solver.
7. Visualize port signals and S-parameters.
8. Visualize port modes and farfield results.
9. Obtain accurate and converged results using the mesh adaptation.
10. Check the truncation error of the time signals.
11. Application of the antenna array feature.
12. Extend the single patch antenna to a four-element antenna array and combine single calculation results as well as perform a simultaneous excitation.

Dipole Antenna Array

Please find the corresponding simulation models for this workflow example in the component library:

Dipole Array, Combine Nearfield Results:	Dipole Antenna Array Nearfield
Dipole Array, Simultaneous Excitation:	Dipole Antenna Array Simultaneous
Dipole Array, Farfield Array Feature:	Dipole Antenna Array
Dipole Array, Mix Farfield Results Template:	Dipole Antenna Array Combine



General Description

This example presents the farfield calculation of a planar 2x2 array pattern, consisting of four identical single electric dipoles, each holding a length of lambda/2 (lambda = 10 mm). Four examples show the behaviour of this antenna pattern:

1. Combine Nearfield Results: The antenna's structure is generated as described below. Compared to the example Dipole Antenna here the complete array is modelled, the antennas separated in space by lambda/4 and showing a phase shift of 90 degree. Each dipole element is excited sequentially in four-time domain simulation runs. In a post-processing step, the superposition of the four results is calculated using the 'Combine Results...' feature considering a phase shift of 90 degree between the single dipoles. Finally, the farfield is calculated from the combined result.

2. Simultaneous Excitation: The structure is the same as above in "Combine Nearfield Results", but the dipole elements are excited simultaneously with a phase shift of 90 degree between the single dipoles in one single time domain simulation run. This has the advantage of being four times faster, but the disadvantage, that the phase shift combination cannot be modified quickly in post-processing. Also, the given phase shift is only valid at one given frequency, because it has to be converted into a time shift for the simulation run.

3. Farfield Array Feature: The antenna is modelled as a single dipole element as described in the Dipole Antenna example. The array is calculated with the law of pattern multiplication, whereas the influence among the single array elements is neglected.

4. Mix Farfield Results Template: The structure is the same as above in "Combine Nearfield Results". Since the structure is rotationally symmetric, only port one is excited and the farfield for this stimulation is calculated. Afterwards, the resulting farfield is rotated and combined with phase shifts using a *mix farfield results* template. This way, the coupling of the array elements is taken into account, and the phase shift can still be modified without needing a time-consuming solver run for recalculation. But still, only one time domain simulation run is necessary.

Structure Generation (of complete array)

One electric dipole is modelled by help of a single very thin cylinder consisting of PEC material, its dimensions are given parametrically. To provide some space for the location of the excitation source a small gap is cut in the antenna afterwards. Since the array pattern consists of identical elements, the single dipole is then multiplied to obtain the final structure using the transform feature with the copy option. Taking advantage of the given symmetry of the structure in z direction a corresponding electric x-y symmetry plane is defined.

Mesh Settings (of complete array)

Due to the structure's very small PEC elements it is necessary to refine the mesh in order to obtain results of sufficient accuracy. Therefore, the number of mesh lines per wavelength is increased as well as the refinement factor at PEC edges. As a consequence, the number of mesh nodes is increased.

Solver Setup (of complete array)

Before starting the simulation, a farfield monitor is set at the respective frequency.

In order to enable a transient calculation, four discrete ports are defined as the excitation sources, feeding each of the electric dipoles. For the examples 1, 2 and 4, the simulation is then performed using the complete structure. For example 1, each antenna is stimulated separately, while for example 2, all antennas are excited simultaneously using an excitation list. For example 4, only port 1 is stimulated.

Post-Processing

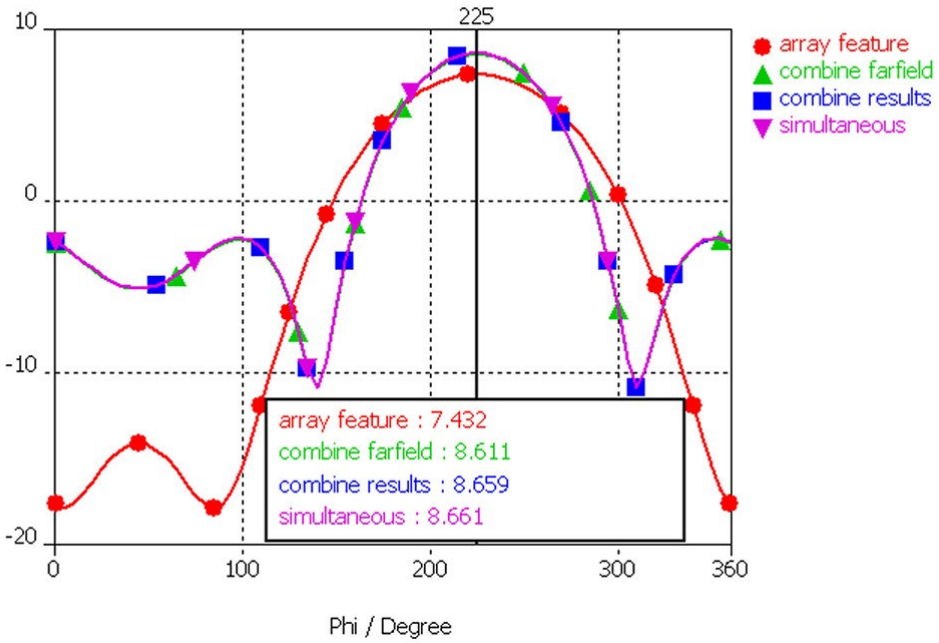
The previously defined farfield monitor offers now the possibility to investigate the radiation characteristics of the antenna array. The farfield results can be found in subfolders of the *Farfields* entry in the navigation tree.

In **example 1** the radiation characteristics of each antenna as well as the behaviour of the complete structure can be investigated. In addition, the nearfield effect of the antennas on each other can be examined regarding the scattering parameters concerning the different ports. The results of the four dipoles are combined to receive the overall farfield considering a phase shift of 90 degree between the antennas [1(1,90)+2(1,0)+3(1,-90)+4(1,0)] with the help of the Post-Processing: ***2D/3D Field Post-Processing*** > ***Combine Results*** option.

Example 2 allows the investigation of the farfield of the complete structure directly since all dipoles are excited simultaneously.

Example 3 offers the investigation of one single dipole as well as the farfield of the array. Here the influence among the single array elements is neglected. The array is calculated with the law of pattern multiplication. A 2x2 planar pattern is chosen perpendicular to the dipole axis, with a constant space shift of lambda/4 and a phase shift of 90 degree between the dipole elements.

For **example 4** a template from Post-Processing: ***Result Templates > Template Based Post-Processing > Mix Farfield Results*** has to be defined. It retrieves the precalculated farfield in 5 degree steps, combines the rotated element patterns and adds the combined pattern to the navigation tree.



The farfield pattern results from all four examples are compared in example 4 under the navigation tree entry *1D Results > Farfield Theta=90* using copy+paste. The plot shows the directivity in the x-y plane (theta=90deg.) over phi. The array behaves as a directional antenna with a main lobe at phi = 225 degree and a directivity of about 8.6 dB. The red curve which represents the array feature result of example 3, differs clearly from the other results. This effect is due to the negligence of the coupling between the array elements. Examples 1, 2 and 4 however are in perfect agreement with each other.