CST MICROWAVE STUDIO®

3D EM FOR HIGH FREQUENCIES

TUTORIALS

CST STUDIO SUITE™ 2006

CST – Computer Simulation Technology
www.cst.com

CST MICROWAVE STUDIO®

Tutorials

Rectangular Waveguide Tutorial 3

Coaxial Structure Tutorial 31

Planar Device Tutorial 77

Antenna Tutorial 115

Resonator Tutorial 165

Filter Tutorial 193

10/04/2005

Rectangular Waveguide Tutorial

Geometric Construction and Solver Settings **4**

Introduction and Model Dimensions 4
Geometric Construction Steps 5

Calculation of Fields and S-Parameters **14**

Transient Solver 14
Transient Solver Results 15
Accuracy Considerations 19
Frequency Domain Solver 22
Frequency Domain Solver Results 25
Accuracy Considerations 29

Getting More Information **30**

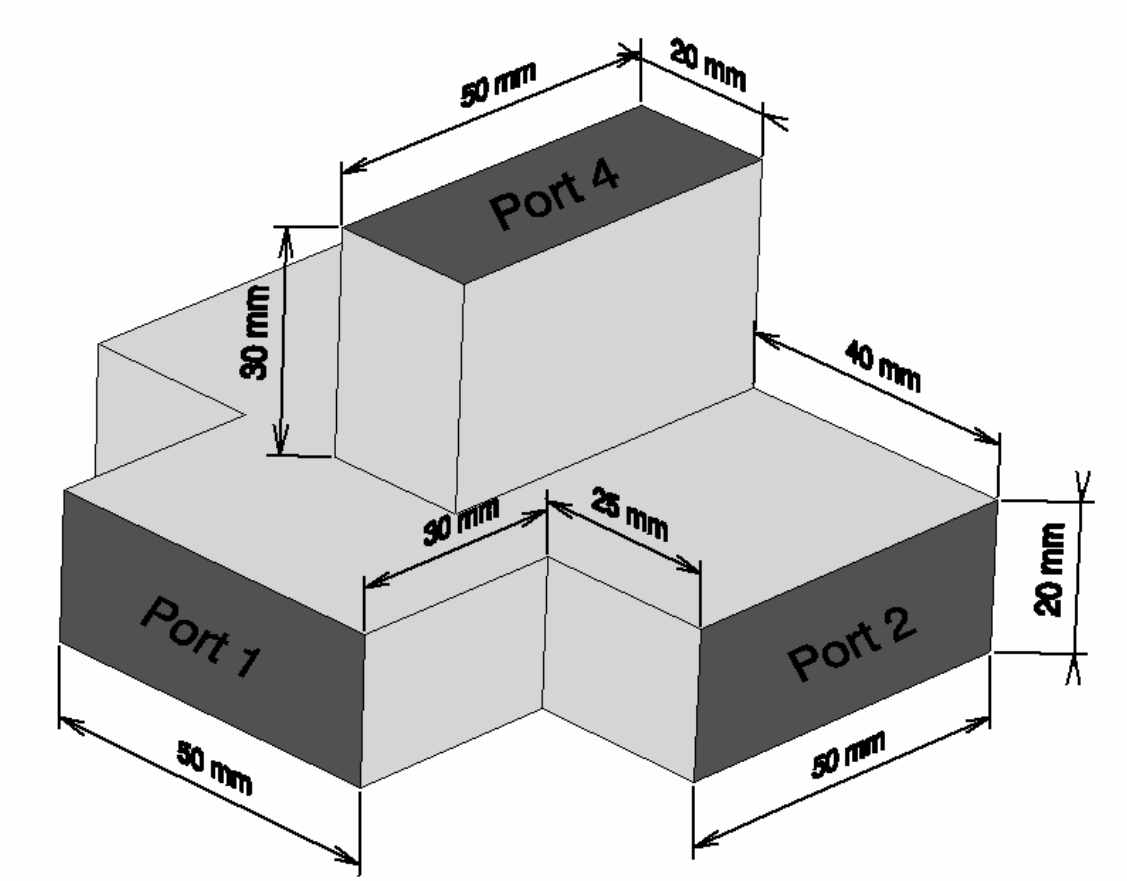
Geometric Construction and Solver Settings

Introduction and Model Dimensions

In this tutorial you will learn how to simulate rectangular waveguide devices. As a typical example for a rectangular waveguide, you will analyze a well-known and commonly used high frequency device: the Magic Tee. The acquired knowledge of how to model and analyze this device can also be applied to other devices containing rectangular waveguides.

The main idea behind the Magic Tee is to combine a TE and a TM waveguide splitter (see the figure below for an illustration and the dimensions). Although CST MICROWAVE STUDIO® can provide a wide variety of results, this tutorial concentrates solely on the S-parameters and electric fields. In this particular case, port 1 and port 4 are de-coupled, so one can expect S14 and S41 to be very small.

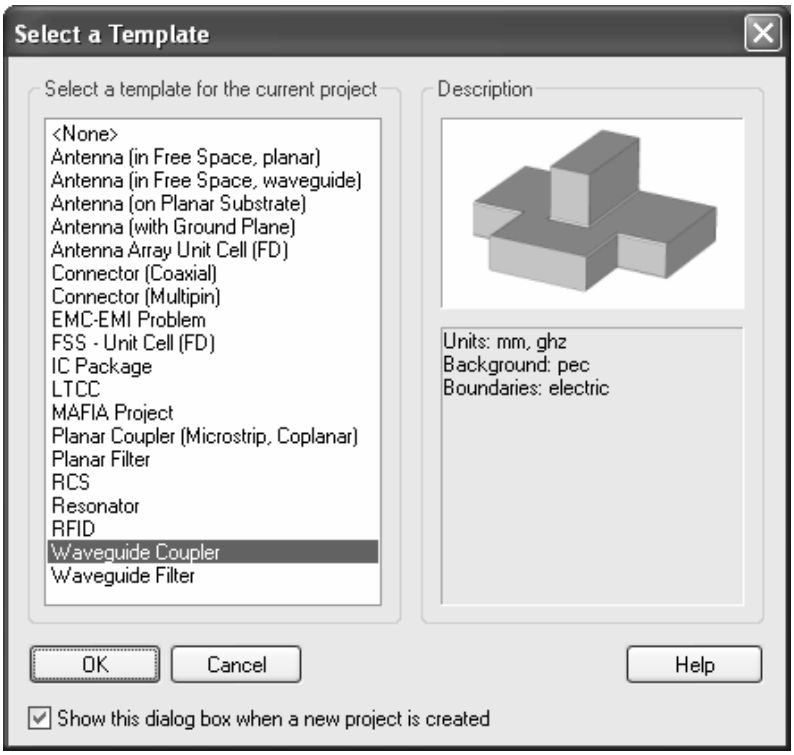
We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.



Geometric Construction Steps

☐ Select a Template

After you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the "Waveguide Coupler" template should be selected.

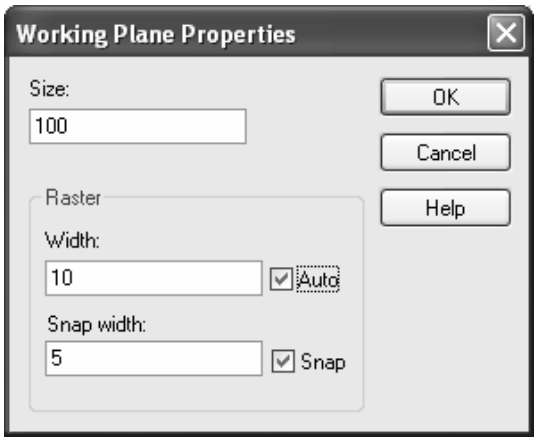


This template automatically sets the units to mm and GHz, the background material to PEC (which is the default) and all boundaries to be perfect electrical conductors.

Because the background material (that will automatically enclose the model) is specified as being a perfect electrical conductor, you only need to model the air-filled parts of the waveguide device. In the case of the Magic Tee, a combination of three bricks is sufficient to describe the entire device.

☐ Define Working Plane Properties

Usually, the next step is to set the working plane properties in order to make the drawing plane large enough for your device. Because the structure has a maximum extension of 100 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 *Edit* ⇨ *Working Plane Properties* from the main menu. Please note that we will use the same document conventions here as introduced in the *Getting Started* manual.



Change the settings in the working plane properties window to the values given above before pressing the *OK* button.

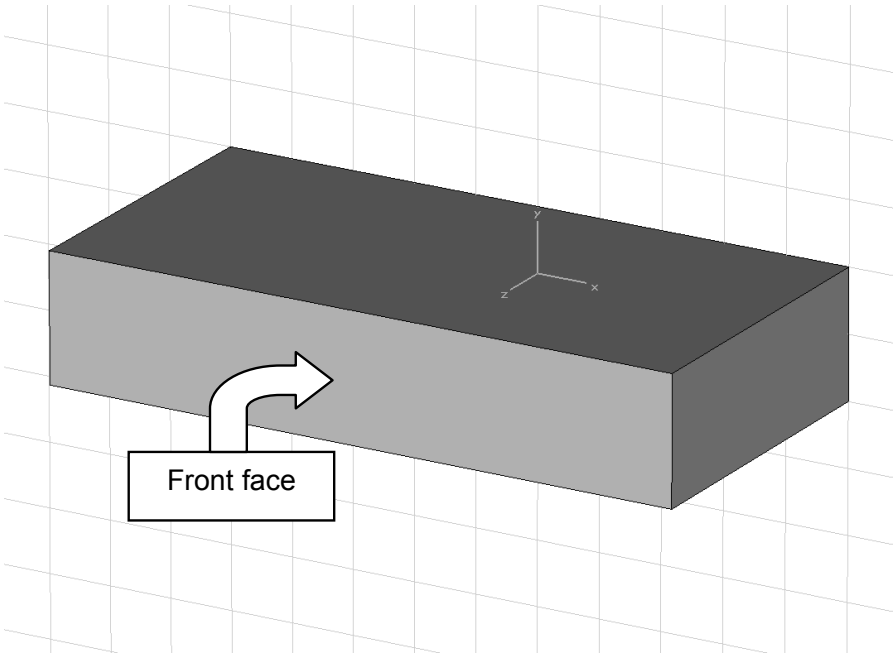
☐ Define the First Brick

Now you can create the first brick:

This is most easily accomplished by clicking the "Create brick" icon  or selecting *Objects ⇨ Basic Shapes ⇨ Brick* from the main menu.

CST MICROWAVE STUDIO® now asks you for the first point of the brick. The current coordinates of the mouse pointer are shown in the bottom right corner of the drawing window in an information box. After you double-click on the point x=50 and y=10, the information box will show the current mouse pointer's coordinates and the distance (DX and DY) to the previously picked position. Drag the rectangle to the size DX=-100 and DY=-20 before double-clicking to fix the dimensions. CST MICROWAVE STUDIO® now switches to the height mode. Drag the height to h=50 and double-click to finish the construction. You should now see both the brick, shown as a transparent model, and a dialog box, where your input parameters are shown. If you have made a mistake during the mouse based input phase, you can correct it by editing the numerical values. Create the brick with the default component and material settings by pressing the *OK* button. Your brick's mouse-based input parameters are summarized in the table below.

Xmin	-50
Xmax	50
Ymin	-10
Ymax	10
Zmin	0
Zmax	50



You have just created the waveguide connecting ports 2 and 3. Adding the waveguide connection to port 1 will introduce another of CST MICROWAVE STUDIO®'s features, the **W**orking **C**oordinate **S**ystem (WCS). It allows you to avoid making calculations during the construction period. Let's continue and discover this tool's advantages.

☐ Align the WCS with the Front Face of the First Brick

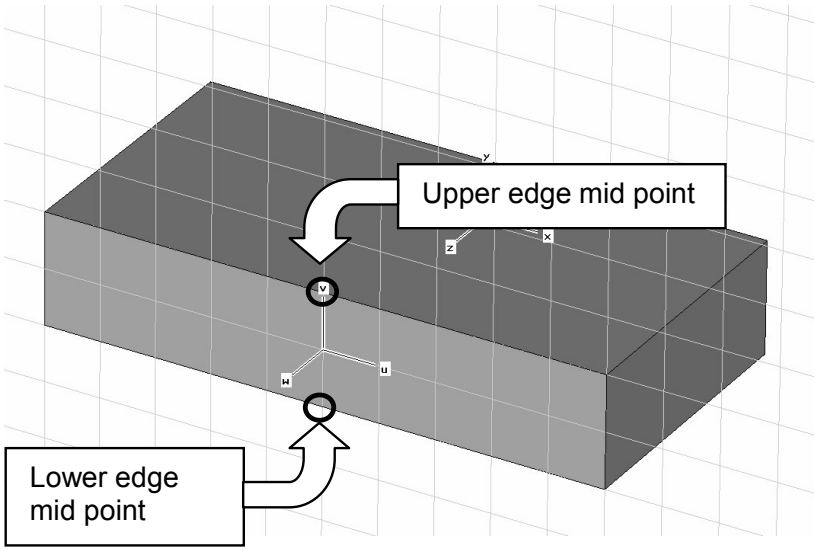
To add the waveguide belonging to port 1 to the front face, as shown in the above picture, activate the "Pick face" tool with one of the following options:

1. "Pick face" tool icon 
2. *Objects ⇨ Pick ⇨ Pick Face*
3. Shortcut: *f*

> **Please note**: The shortcuts only work if the main drawing window is active. You can activate it by single-clicking on it.

Now simply double-click on the front face of the brick to complete the pick operation.

The working plane can now be aligned with the selected face by pressing the "Align the WCS with the most recently selected face" icon  (or by using the shortcut *w*). This action moves and rotates the WCS so that the working plane (uv plane) coincides with the selected face.



☐ Define the Second Brick

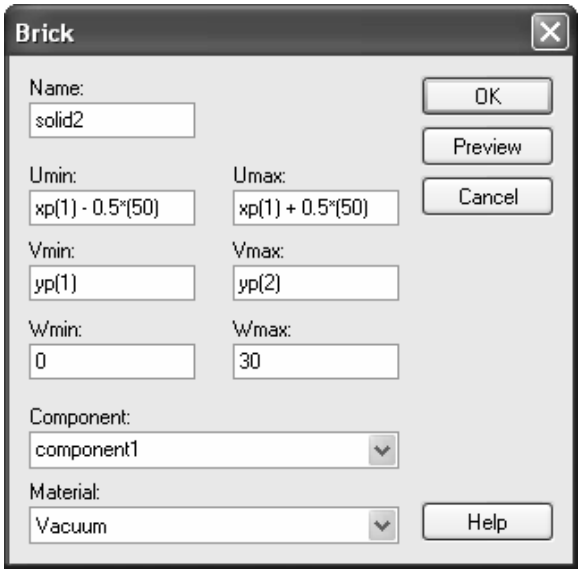
With the WCS in the right location, creating the second brick is quite simple. Start the brick creation mode with either the main menu's *Objects ⇨ Basic Shapes ⇨ Brick* or the corresponding icon  . Please remember that all values used for shape construction are relative to the uvw coordinate system as long as the WCS is active.

The new brick should be aligned with the edge midpoints of the first brick as shown in the picture above. Without leaving the current "Create brick" mode, you should pick the lower edge's midpoint by simply activating the appropriate pick tool  (*Objects ⇨ Pick ⇨ Pick Edge Midpoint* or use the shortcut *m*). Now all edges become highlighted and you can simply double-click on the first brick's lower edge as shown in the picture. Then, continue with the brick creation by repeating the procedure for the brick's upper edge.

Because you have now selected two points that are located on a line, you will be requested to enter the width of the brick. Please note that this step will be skipped if the two previously picked points already form a rectangle (not only a line). Now you should drag the width of the brick to w=50 (watch the coordinate display in the lower right corner of the drawing window) and double-click on this location.

Finally, you must specify the brick's height. Therefore, drag the mouse to the proper height (h=30) and double-click on this location. Please note that instead of specifying coordinates with the mouse (as we have done here), you can also press the TAB key whenever a coordinate is requested. This will open a dialog box where you can specify the coordinates numerically.

After the brick's interactive construction is completed, a dialog box will again appear showing a summary of the brick's parameters.

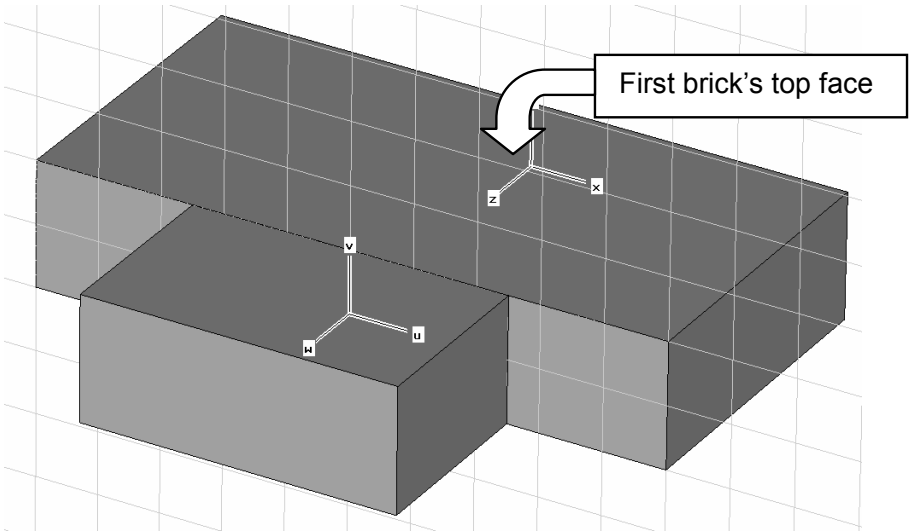


Some of the coordinate fields now contain mathematical expressions because some of the points were entered using the pick tools. Here, the functions xp(1), yp(1) represent the point coordinates of the first picked point (the midpoint of the first brick's lower edge). Analogously, the functions xp(2) and yp(2) correspond to the upper edge's midpoint.

Because you are currently constructing the inner waveguide volume, you can still keep the default "Vacuum" *Material* setting and the same *Component* ("component1") as for the first brick.

> **Please note:** The use of different components allows you to gather several solids into specific groups, independent of their material behavior. For this tutorial, however, it is convenient to construct the complete structure as a single component.

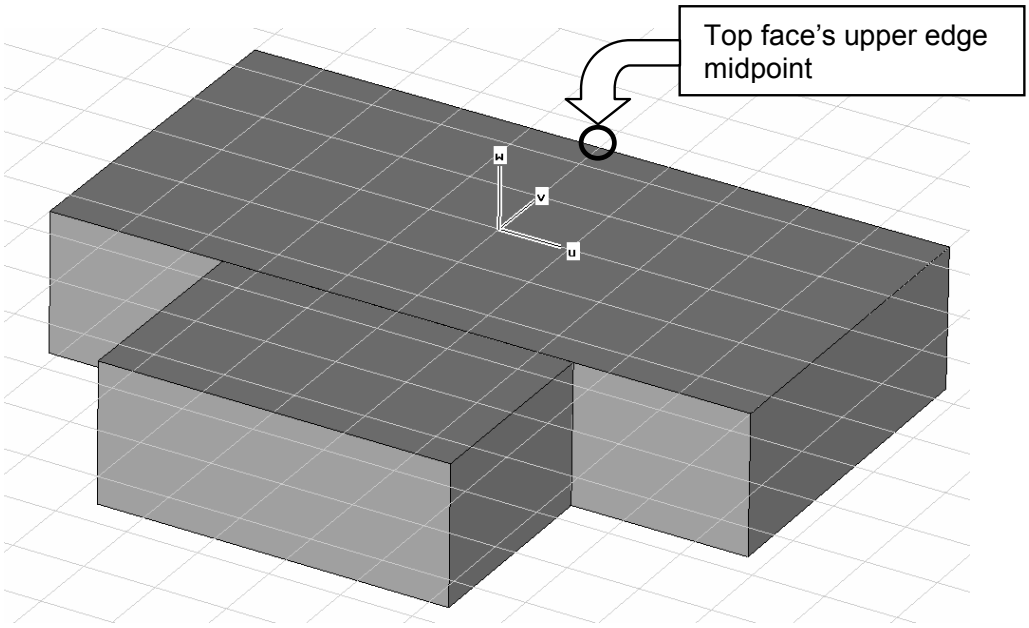
Finally, you should confirm the brick's creation again by pressing the *OK* button. Let's now construct the third brick.



☐ Align the WCS with the First Brick's Top Face

The next brick should be aligned with the top face of the first brick. To align the local coordinate system with this face, you should first activate the *Pick Face* mode (, *Objects ⇨ Pick ⇨ Pick Face* or shortcut *f*) and double-click on the desired face.

Afterwards, you should press the "Align the WCS with the most recently selected face" icon , select *WCS ⇨ Align WCS with Selected Face* from the main menu or use the shortcut *w*.



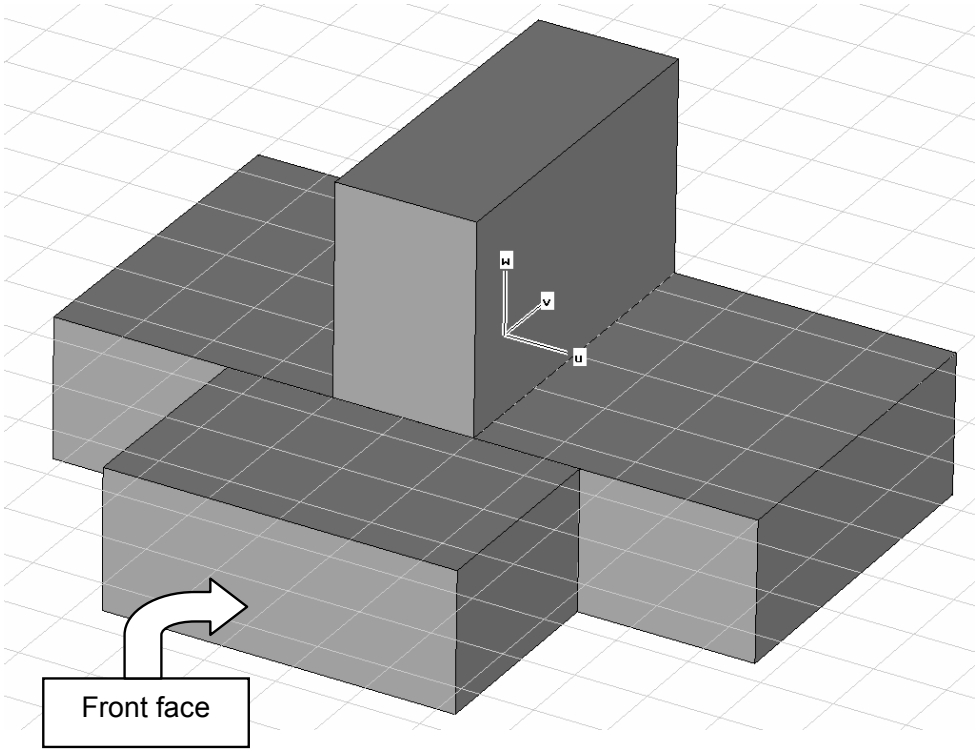
☐ Construct the Third Brick

The brick creation mode for drawing the third brick should now be activated by selecting either *Objects ⇨ Basic Shapes ⇨ Brick* or the "Create a brick" icon .

When you are requested to enter the first point, you should activate the midpoint edge pick tool (shortcut *m*), as you did for the previous brick, and double-click on the top face's upper edge midpoint (see picture above).

The next step is to drag the mouse in order to specify the extension of 50 along the –v direction (hold down the *Shift* key while dragging the mouse to restrict the coordinate movement to the v direction only) and double-click on this location. Afterwards, you should specify the width of the brick as w=20 and the height as h=30 in the same manner, or by entering these values numerically using the *Tab* key.

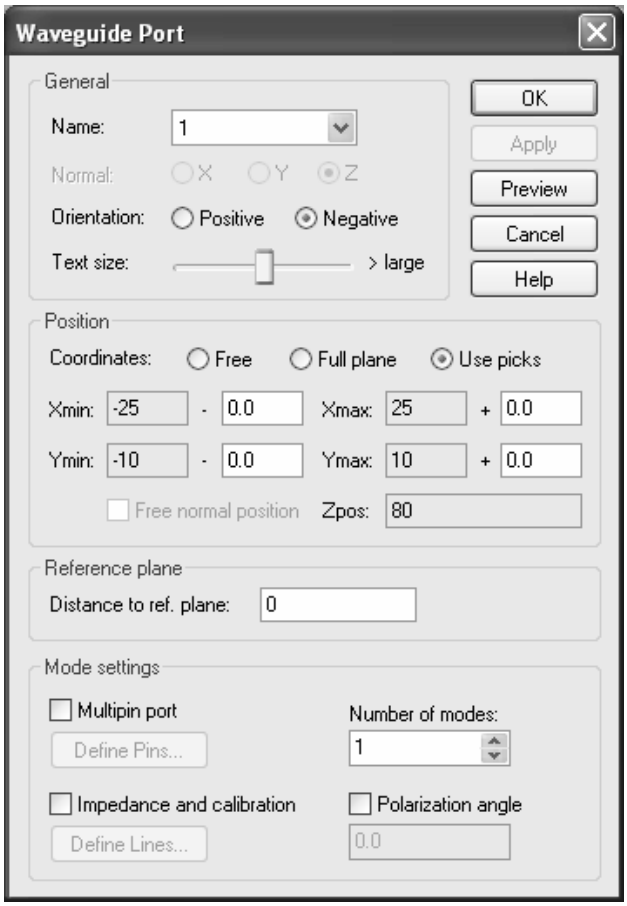
The last brick is also created as a vacuum material and belongs to the component "component1". Finally, confirm these settings in the brick creation dialog box. Now the structure should look as follows:



☐ Define Port 1

In the next step you will assign the first port to the front face of the Magic Tee (see picture above). The easiest way to do this is to pick the port face first by activating the *Pick Face* tool (, *Objects ⇨ Pick ⇨ Pick Face* or shortcut *f*) and then double-click on the desired face.

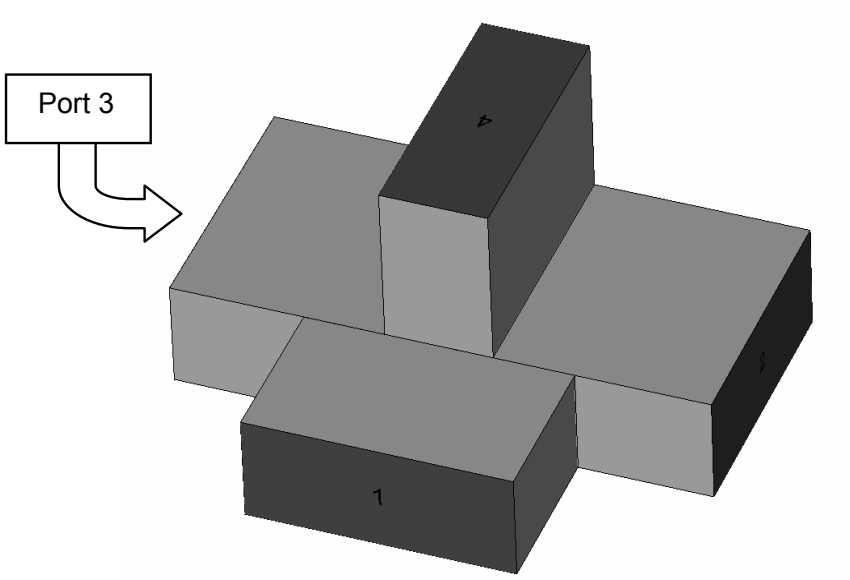
Once the port's face is selected you can open the waveguide port dialog box either by selecting *Solve ⇨ Waveguide Ports* from the main menu or by pressing on the "Define waveguide port" icon . The settings in the waveguide port dialog box will automatically specify the extension and location of the port according to the bounding box of any previously picked elements (faces, edges or points).



In this case, you can simply accept the default settings and press *OK* to create the port. The next step is the definition of ports 2, 3 and 4.

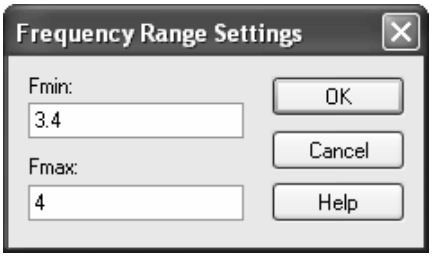
☐ Define Ports 2, 3, 4

Repeat the last steps (pick face and create port) to define port 2, port 3 and port 4. After you have completed this step, your model should look like the below figure. Please double-check your input before proceeding to the solver settings.



☐ Define the Frequency Range

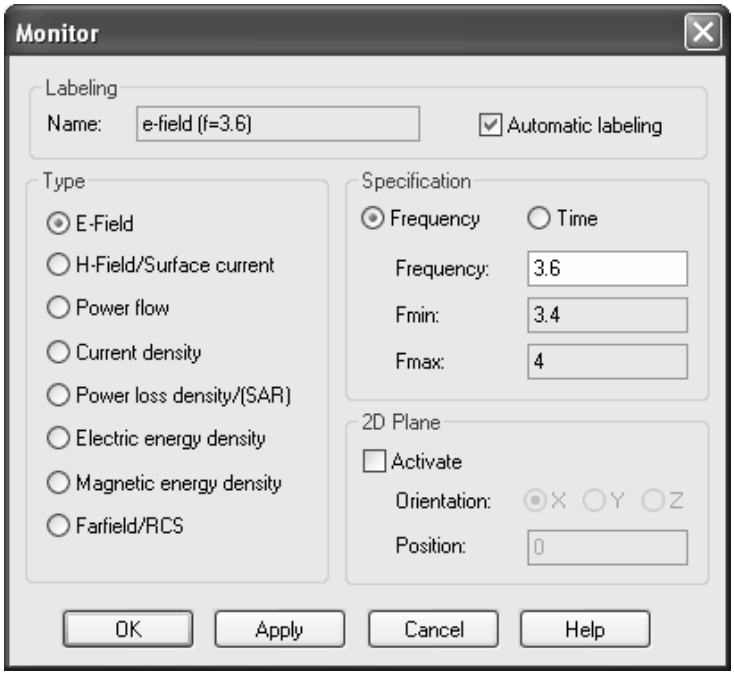
The frequency range for this example extends from 3.4 GHz to 4 GHz. Change *Fmin* and *Fmax* to the desired values in the frequency range settings dialog box (opened by pressing the "Frequency range" icon or choosing *Solve* ⇨ *Frequency*) and store these settings by pressing the *OK* button. Please note that the currently selected units are shown in the status bar.



☐ Define Field Monitors

Because the amount of data generated by a broadband time domain calculation is huge even for relatively small examples, it is necessary to define which field data should be stored before the simulation is started. CST MICROWAVE STUDIO® uses the concept of "monitors" in order to specify which types of field data to store. In addition to the type, you also must specify whether the field should be recorded at a fixed frequency or at a sequence of time samples. You can define as many monitors as necessary to get different field types or fields at various frequencies. Please note that an excessive number of field monitors may significantly increase the memory space required for the simulation.

To add a field monitor, click the "Monitors" icon or select *Solve* ⇨ *Field Monitors* from the main menu.



In this example, you should define an electric field monitor (*Type* = *E-Field*) at a *Frequency* of 3.6 GHz before pressing the *OK* button to store the settings. The green box indicates the volume in which the fields will be recorded.

Calculation of Fields and S-Parameters

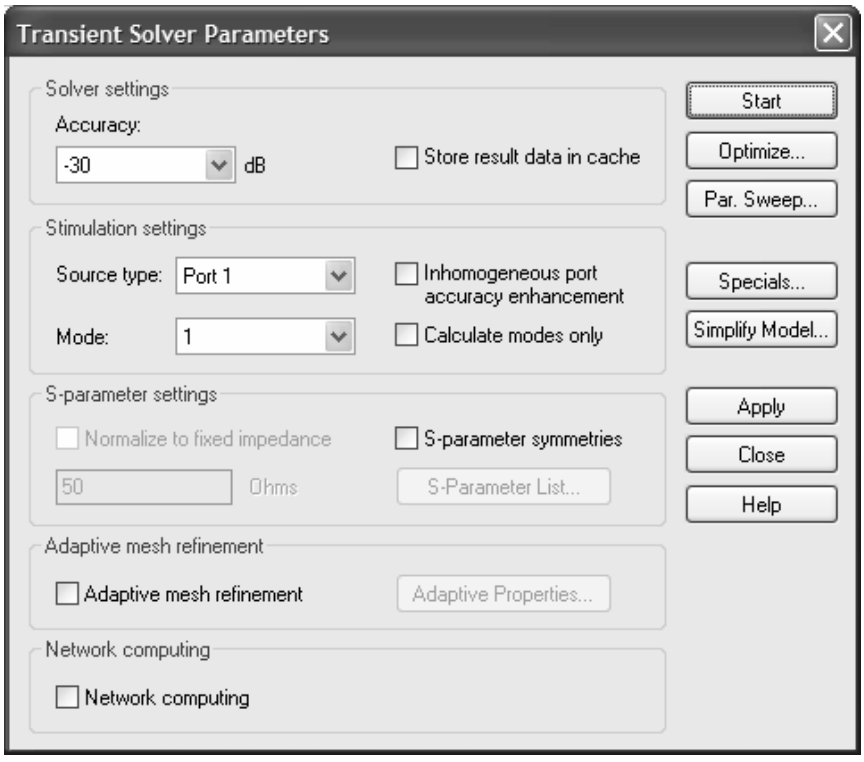
A key feature of CST MICROWAVE STUDIO® is the *Method on Demand* approach that allows a simulator or mesh type that is best suited for a particular problem. Another benefit is the ability to compare the results obtained by completely independent approaches. We demonstrate this strength in the following sections by calculating fields and S-parameters with the transient solver and the frequency domain solver. In this case, the transient simulation uses a hexahedral mesh while the frequency domain calculation is performed with a tetrahedral mesh. Both sections are self-contained and it is sufficient to work through only one of them, depending on which solver you are interested in. The section on the frequency domain solver also provides a comparison with the transient simulation.

Please note that one of the solvers may not be available to you due to license restrictions. Please contact your sales office for more information.

Transient Solver

- Transient Solver Settings

The transient solver parameters are specified in the solver control dialog box that can be opened by selecting *Solve* ⇨ *Transient Solver* from the main menu or by pressing the "Transient solver" icon in the toolbar.



You should now specify whether the full S-matrix should be calculated or if a subset of this matrix is sufficient. For the Magic Tee device we are interested in the input reflection at port 1 and in the transmission from port 1 to the other three ports (2, 3 and 4).

Accordingly, we only need to calculate the S-parameters S1,1, S2,1, S3,1 and S4,1. All of the S-parameters can be derived by an excitation at port 1. Therefore, you should change the *Source type* field in the *Stimulation settings* frame to *Port 1*. If you leave this setting at *All Ports*, the full S-matrix will be calculated.

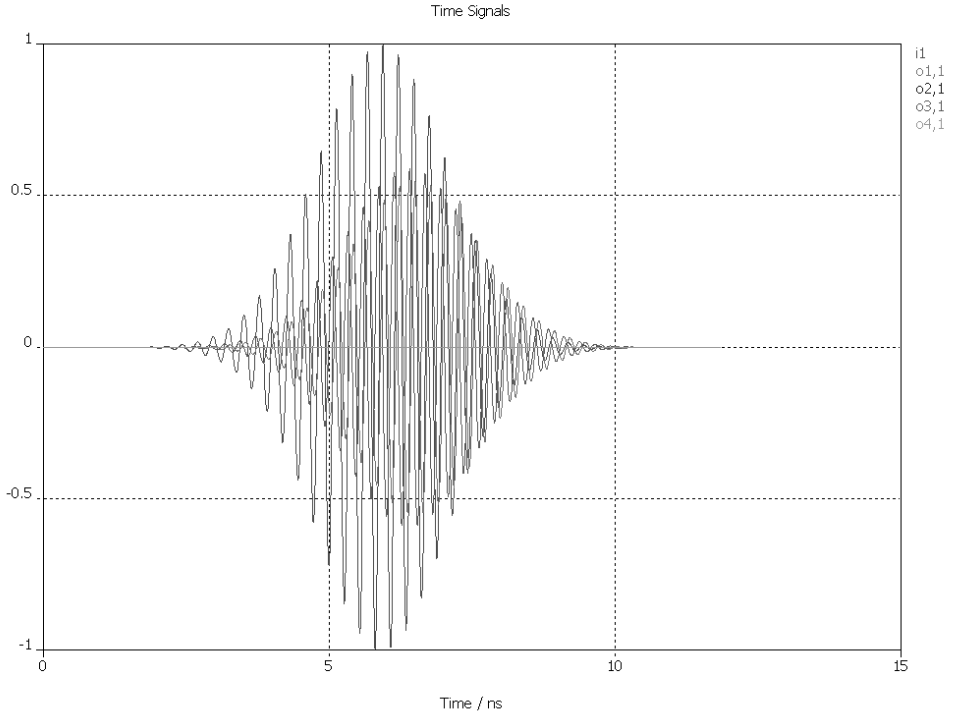
Finally, press the *Start* button to begin the calculation. A progress indicator appears in the status bar displaying some information about the calculation. If any error or warning messages are produced by the solver, they will be displayed in the message window that will be activated automatically, if necessary.

Transient Solver Results

Congratulations, you have simulated the Magic Tee! Let's review the results.

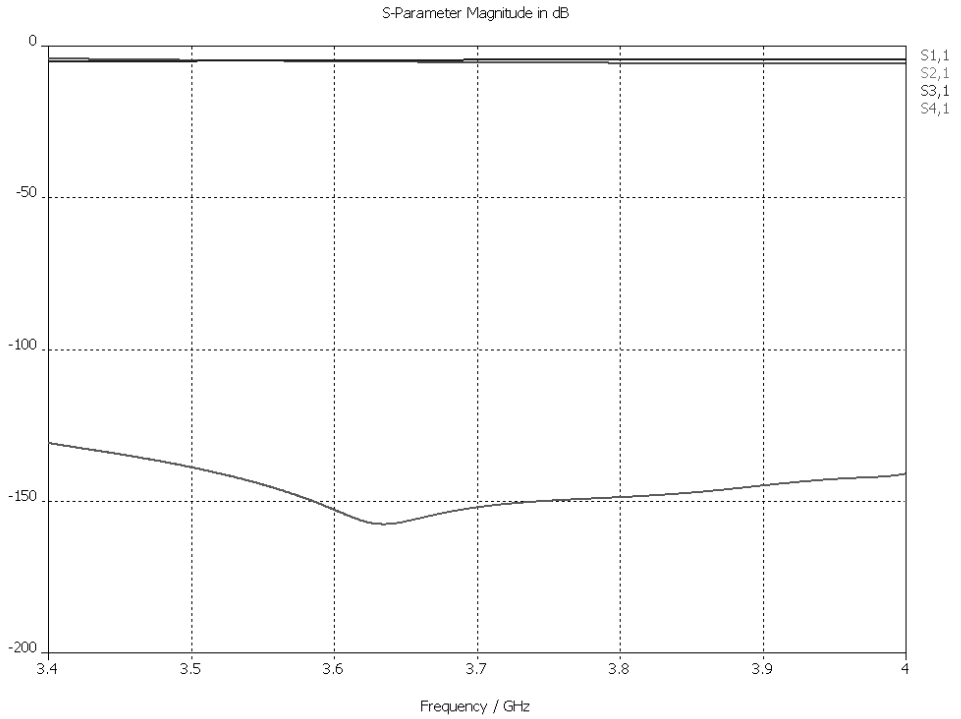
☐ 1D Results (Port Signals, S-Parameters)

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



This plot shows the incident and reflected or transmitted wave amplitudes at the ports versus time. The incident wave amplitude is called i1, the reflected wave amplitude is o1,1 and the transmitted wave amplitudes are o2,1, o3,1 and o4,1. You can see that the transmitted wave amplitudes o2,1 and o3,1 are delayed and distorted (note that o2,1 and o3,1 are identical, so do not be concerned if you only see one curve).

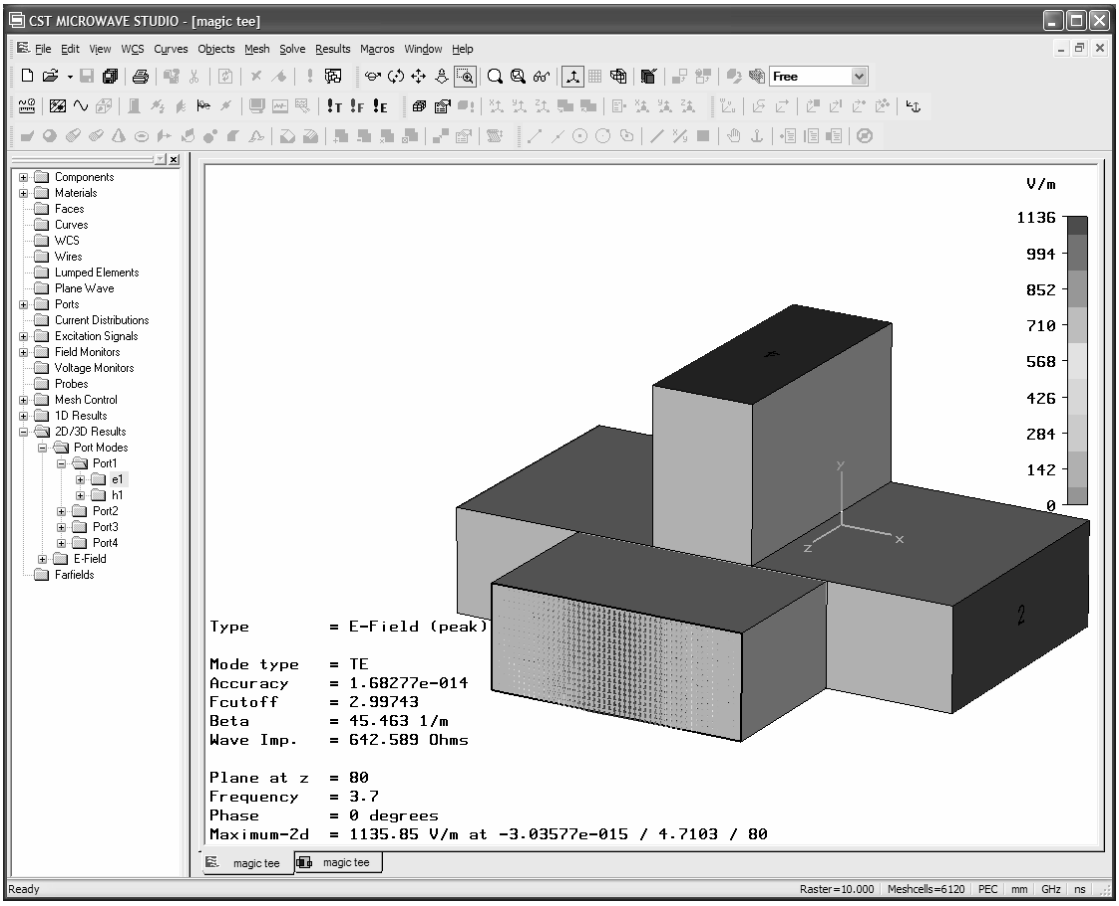
The S-parameters can be plotted in dB by clicking on the *1D Results* ⇨ *S dB* folder.



As expected, the transmission to port 4 (S4,1) is extremely small (-150 dB is close to the solver's noise floor). It is obvious that this simple device is very poorly matched so that the transmission to ports 2 and 3 is of the same order of magnitude as the input reflection at port 1.

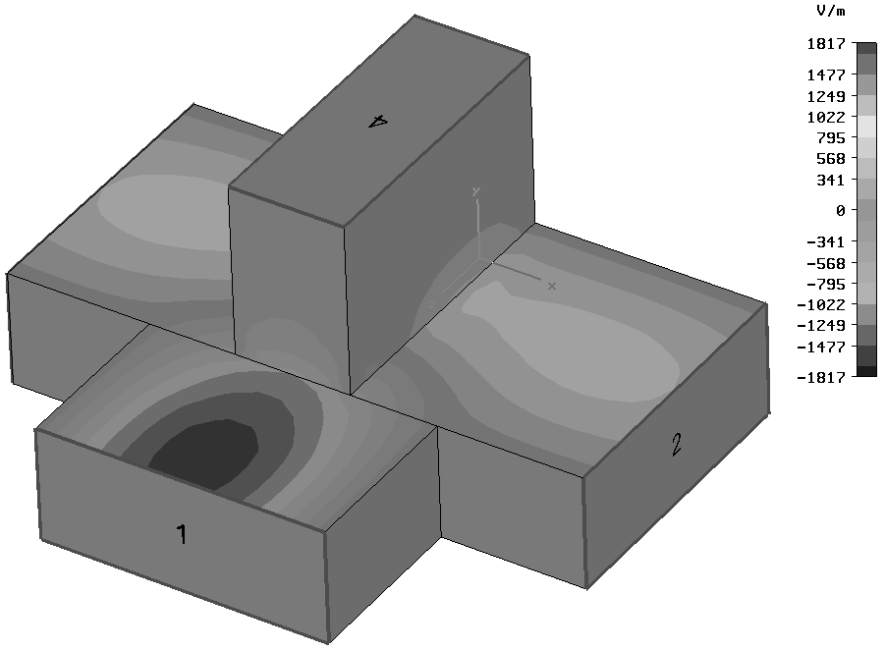
☐ 2D and 3D Results (Port Modes and Field Monitors)

Finally, we will review the 2D and 3D field results. We will 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 you should click on the *e1* subfolder.



Because we have selected the main entry, a 3D vector plot is shown. Selecting either of the subentries will produce a scalar plot. The plot also shows some important properties of the mode such as mode type, cut-off frequency and propagation constant. The port modes at the other ports can be visualized in the same manner.

The full three-dimensional electric field distribution in the Magic Tee can be shown by selecting the *2D/3D Results ⇨ E-Field ⇨ efield (f=3.6)[1]* folder from the navigation tree. If the *Normal* item is clicked, the field plot will show a three dimensional contour plot of the electric field normal to the surface of the structure.




```
Type        = E-Field (peak)
Monitor     = e-field (f=3.6) [1]
Component   = Normal
Maximum-3d  = 1818.79 V/m at -3.03577e-015 / 7.42229 / 74.889
Frequency   = 3.6
Phase       = 0 degrees
```

You can display an animation of the fields by checking the *Animate Fields* option in the context menu (right mouse click in the plot window). The appearance of the plot can be changed in the plot properties dialog box, that can be opened by selecting *Results* ⇨ *Plot Properties* from the main menu or *Plot Properties* from the context menu. Alternatively, you can double-click on the plot to open this dialog box.

Accuracy Considerations

In this case, the transient S-parameter calculation is mainly 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.

In the following section we provide hints on how to minimize these errors and obtain highly accurate results.

- 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 an excitation with a Gaussian pulse at the input port. Thus, the signals at the ports are the fundamental results from which the S-parameters are derived using a Fourier Transform.

Even if the accuracy of the time signals themselves 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 using the *Accuracy* setting in the transient solver control dialog box. The default value of –30 dB will usually give sufficiently accurate results for coupler devices. However, to obtain highly accurate results for waveguide structures it is sometimes necessary to increase the accuracy to –40 dB or –50 dB.

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

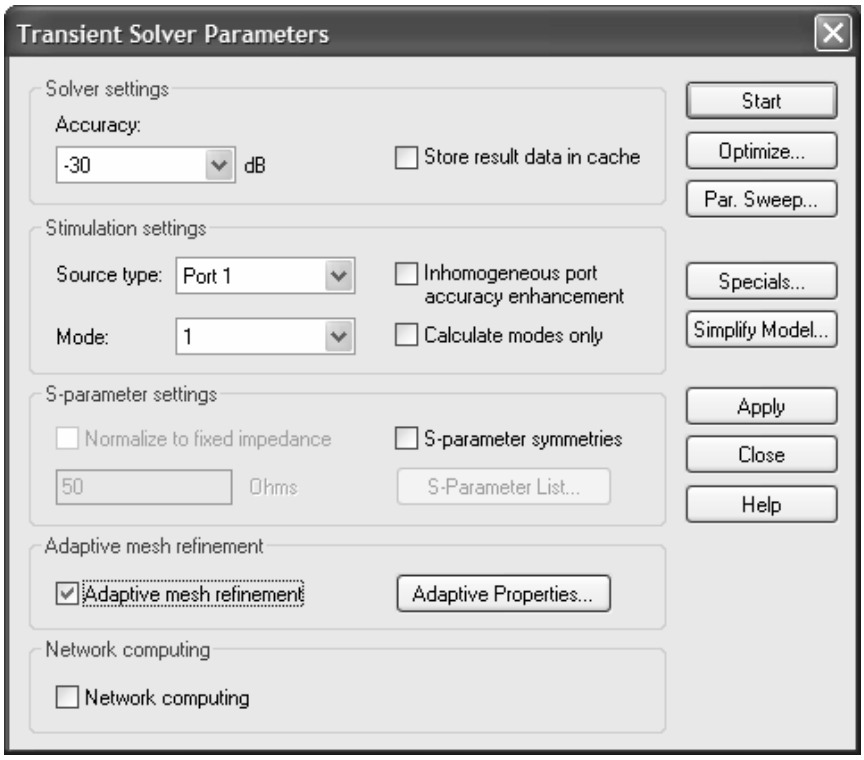
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

If you find a large ripple in the S-parameters, it might be necessary to increase the solver's accuracy setting or use the AR-Filter feature that is explained in the *Advanced Topic* manual and in the online help.

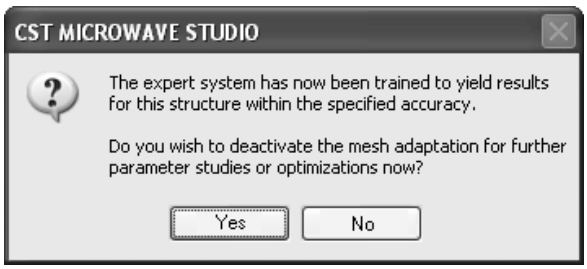
☐ Effect of the Mesh Resolution on the S-parameter's Accuracy

The 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. If these 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 easiest way to prove the accuracy of the results is to use the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (*Solve ⇨ Transient Solver*  *):*



After activating the adaptive mesh refinement tool, you should now start the solver again by pressing the *Start* button. After a couple of minutes (during which the solver is running through mesh adaptation passes), the following dialog box will appear:

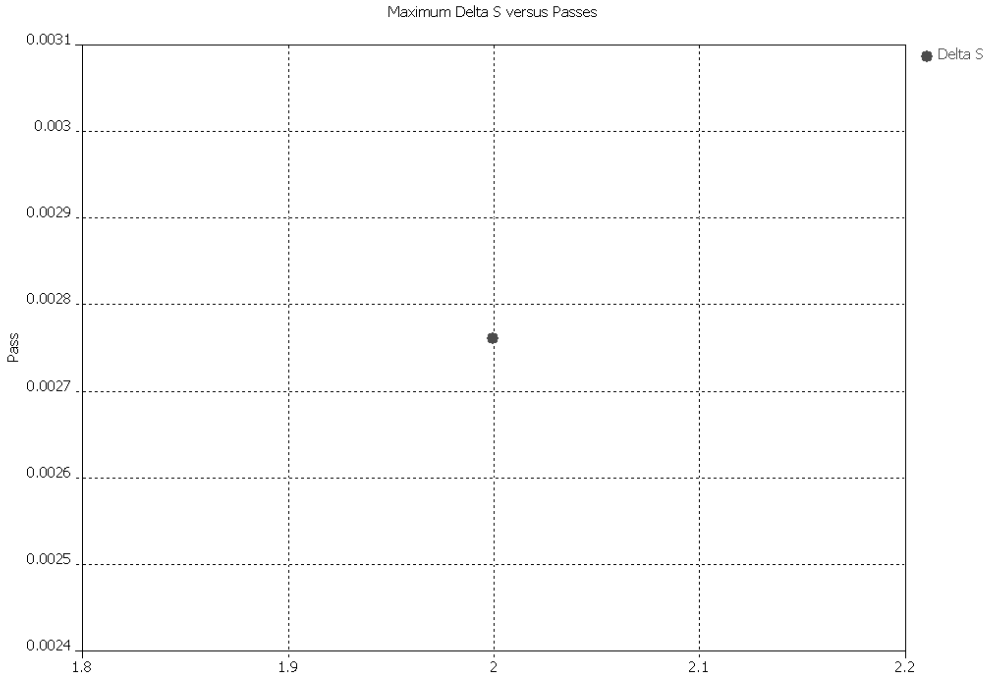


This dialog box informs you that the desired accuracy limit (2% by default) could be met by the adaptive mesh refinement. Because the expert system's settings have now been

adjusted such that this accuracy is achieved, you may switch off the adaptation procedure for subsequent calculations (e.g. parameter sweeps or optimizations).

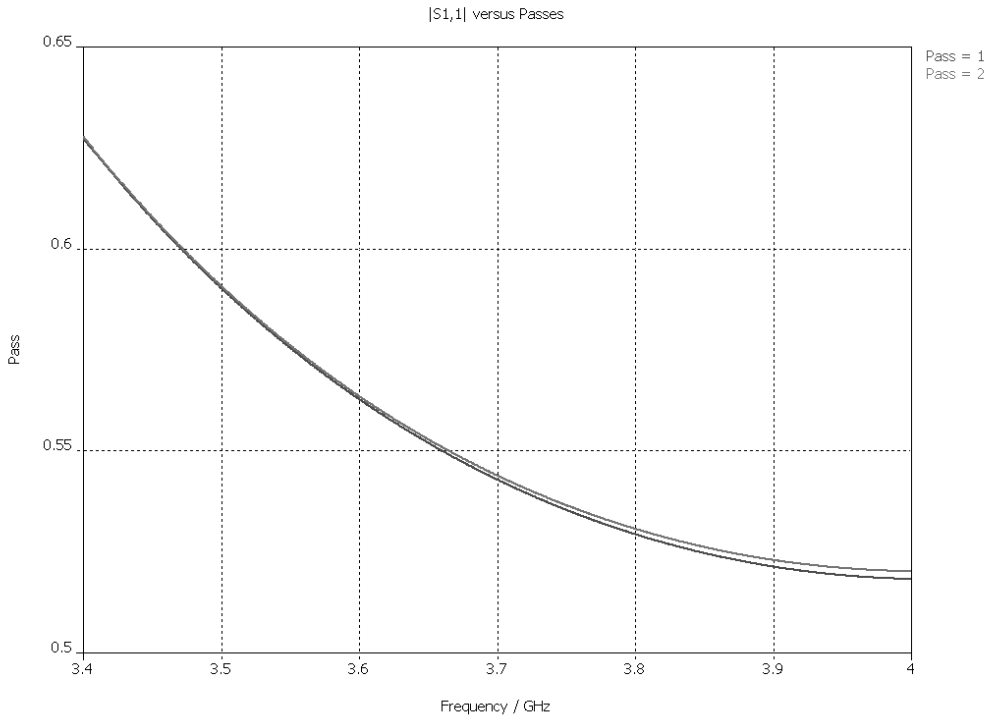
You should now confirm the deactivation of the mesh adaptation by pressing the *Yes* button.

After the mesh adaptation procedure is complete, you can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *Delta S* from the navigation tree:



As you can see, the maximum deviation of the S-parameters is below 0.5%, indicating that the expert system based meshing would have been fine for this example even without running the mesh adaptation procedure.

The convergence process of the input reflection S1,1 during the mesh adaptation can be visualized by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ |*S*|*linear* ⇨ *S1,1* from the navigation tree:



The convergence process of the other S-parameters can be visualized in the same manner. Please note that S4,1 is extremely small (< -120dB) in this example; it's variations are mainly due to the numerical noise and are therefore ignored by the automatic mesh adaptation procedure.

The advantage of this expert system based mesh refinement procedure over traditional adaptive schemes is that the mesh adaptation needs to be carried out only once for each device to determine the optimum settings for the expert system. There is subsequently no need for time consuming mesh adaptation cycles during parameter sweeps or optimizations.

> **Please note:** Refer to the *Getting Started* manual how to use *Template Based Postprocessing* for automated extraction and visualization of arbitrary results from various simulation runs.

Frequency Domain Solver

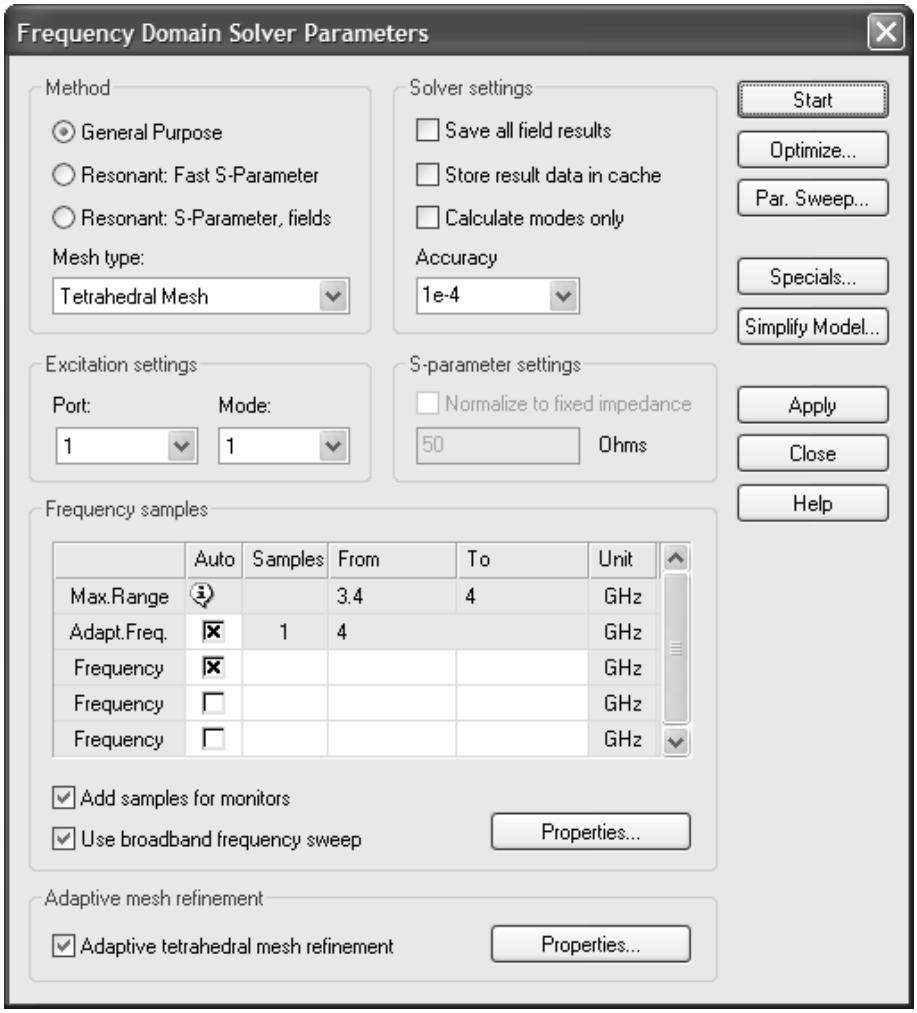
CST MICROWAVE STUDIO® offers a variety of frequency domain solvers specialized for different types of problems. They differ not only by their algorithms but also by the grid type they are based on. The general purpose frequency domain solver is available for hexahedral grids, as well as for tetrahedral grids. In this tutorial we will use a tetrahedral mesh. The availability of a frequency domain solver within the same environment offers a very convenient means of cross-checking results produced by the time domain solver.

☐ Making a Copy of Transient Solver Results

Before performing a simulation with the frequency domain solver, you may want to keep the results of the transient solver in order to compare the two simulations. The copy of the current results is obtained as follows: Select, for example, the |*S*| *dB* folder in *1D Results*, then press *Ctrl+c* and *Ctrl+v.* The copies of the results will be created in the selected folder. The names of the copies will be *S1,1_1, S2,1_1* etc. You may rename them to *S1,1_TD*, *S2,1_TD* and so on with the *Rename* command from the context menu. Use *Add 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.

☐ Frequency Domain Solver Settings

The "Frequency Domain Solver Parameters" dialog box is opened by selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu or by pressing the corresponding icon in the toolbar.



There are three different methods to choose from. For the example here, please choose the *General Purpose* frequency domain solver. In the *Mesh Type* combo box you may choose Hexahedral or Tetrahedral Mesh. Please choose *Tetrahedral Mesh.*

You should now specify whether the full S-matrix should be calculated or if a subset of this matrix is sufficient. For the Magic Tee device we are interested in the input reflection at port 1 and in the transmission from port 1 to the other three ports (2, 3 and 4).

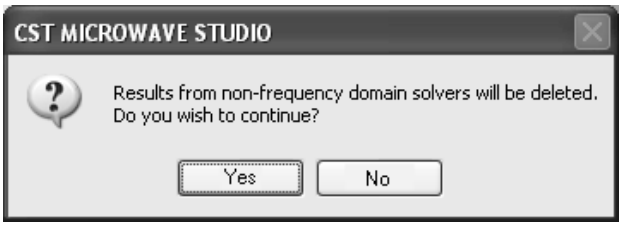
Consequently, we only need to calculate the S-parameters S1,1, S2,1, S3,1 and S4,1. All of the S-parameters can be derived by an excitation at port 1. Therefore, you should change the Source type field in the *Excitation settings* frame to *Port 1* unless already done. If this is set to *All Ports*, the full S-matrix will be calculated.

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 the position of the frequency samples are chosen automatically in order to meet 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 check box is activated by default. All other settings may be left unchanged.

After everything is ready, you may press *Start* to begin the calculation.

Because the old results will be overwritten when starting a different solver, the following warning message appears:



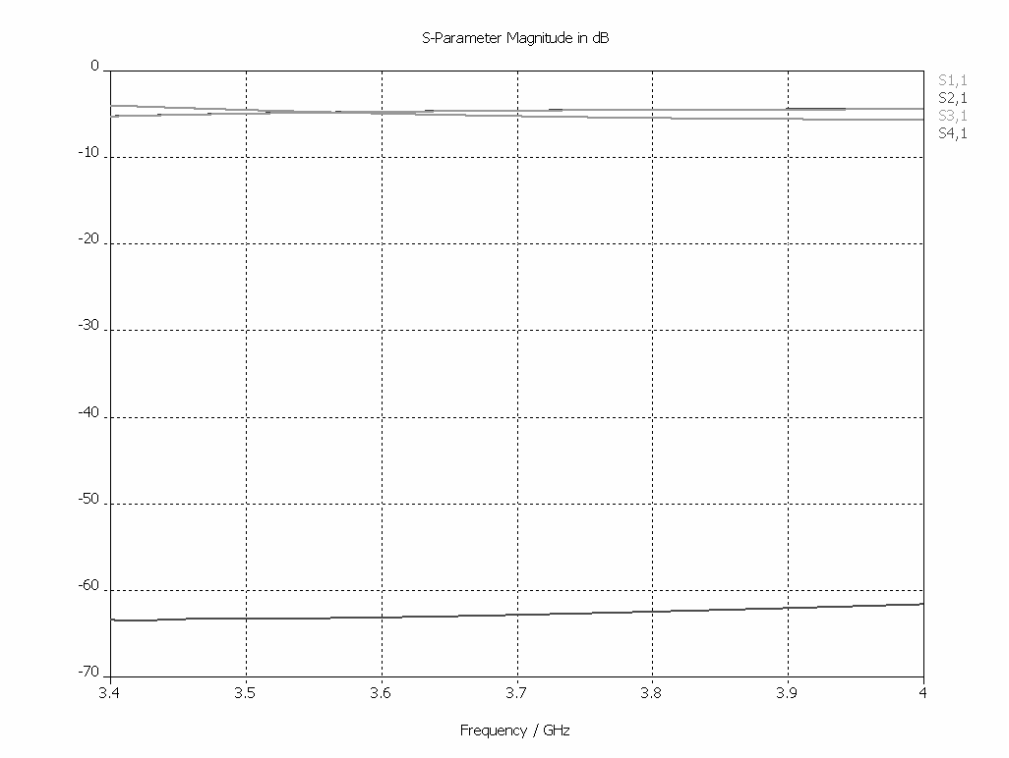
Press *Yes* to acknowledge the deletion. A progress bar will appear at the bottom of the main frame as soon as the solver starts. Additional information about the simulation progress will be shown in the message window that will be activated automatically, if necessary.

Frequency Domain Solver Results

After the desired accuracy for the S-parameter has been reached the simulation stops.

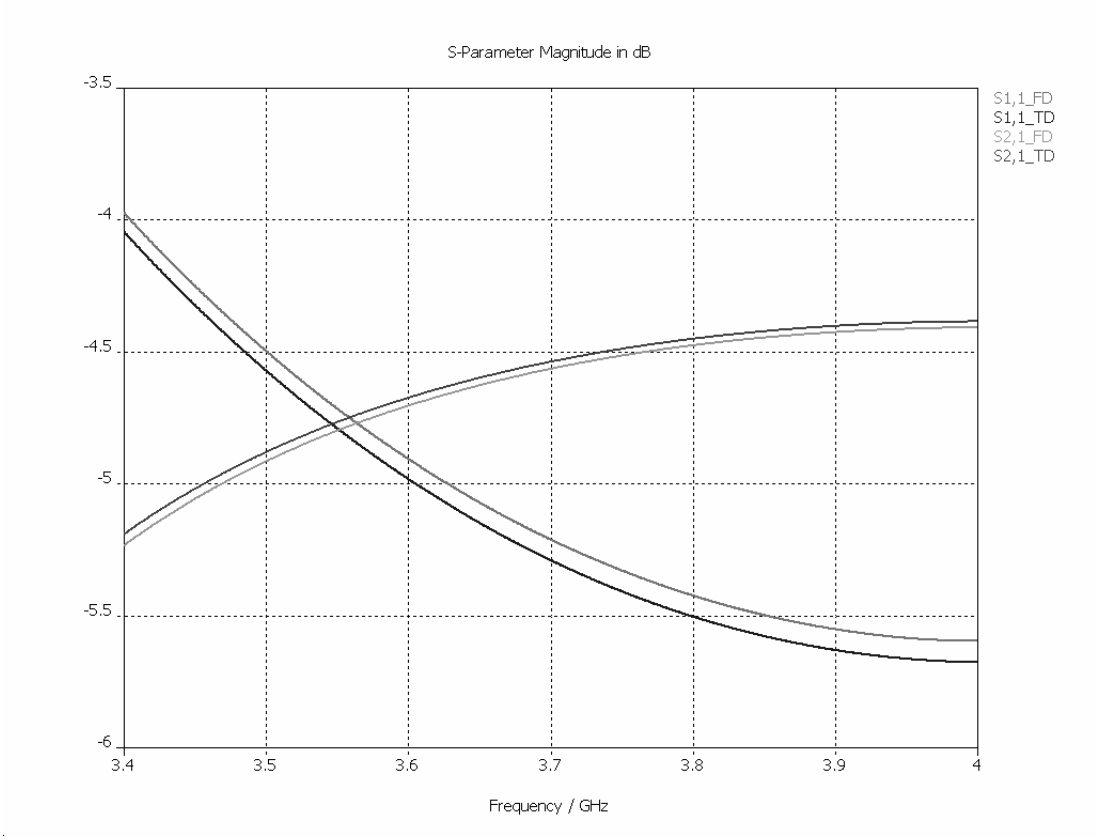
☐ 1D Results (S-Parameters)

As for the transient solver run, you can view the S-parameters by selecting *1D Results* ⇨ |*S*| *dB* in the navigation tree.



Similar to the case of transient solver, an extremely small transmission to port 4 (S4,1) is observed here. In addition, you can conclude that the other S-parameters have at least the same order of magnitude as the S-parameters computed with the transient solver.

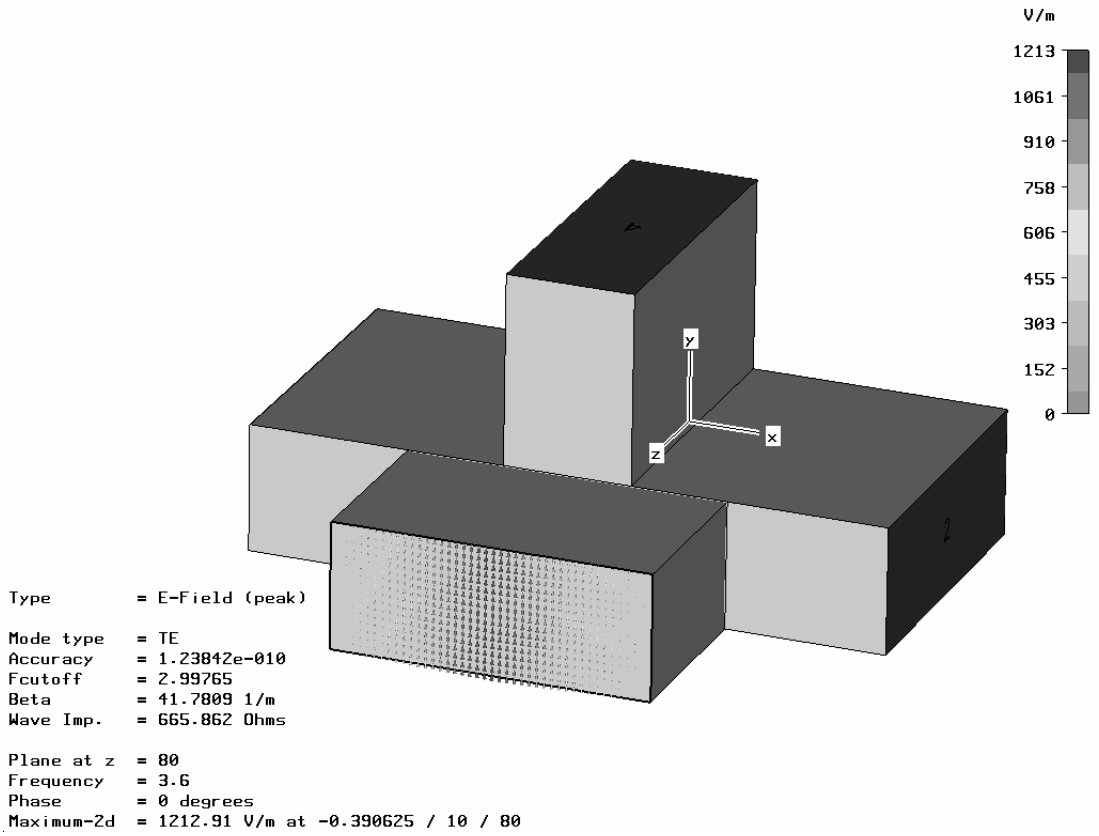
The next figure shows the S-parameters S1,1 and S1,2 for both transient and frequency domain solvers plotted in the same graph. This can be done by copying all these results to an extra folder.



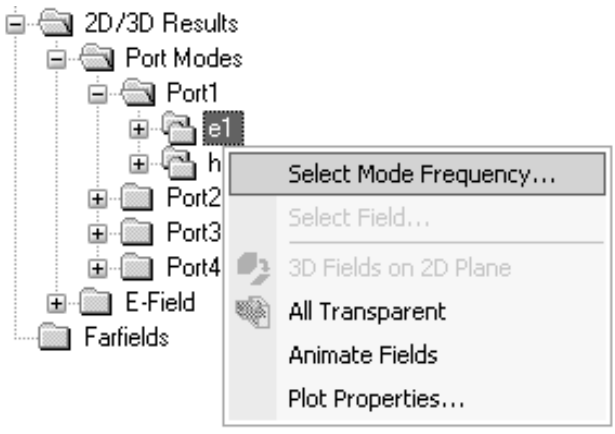
As you can see, the results agree very well. For this specific structure, the transient solver does provide more accurate results by default. The accuracy of the frequency domain simulation can be increased by lowering the accuracy limit for the adaptive mesh refinement.

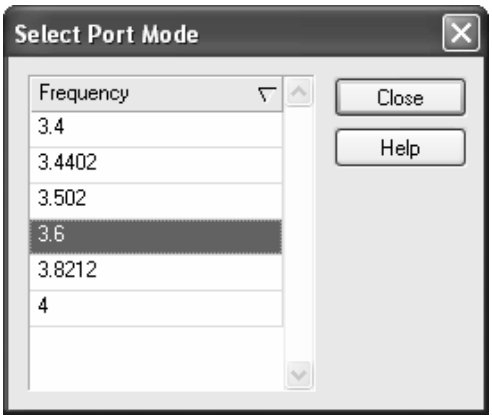
☐ 2D and 3D Results (Port Modes and Field Monitors)

The 2D and 3D field results can be found in the *2D/3D Results* folder of the navigation tree. The electric field of the fundamental mode at port 1 can be visualized by selecting the *Port Modes* ⇨ *Port1* ⇨ *e1* folder. An example of such a visualization is shown in the following figure. Please refer to the *Getting Started* manual for information on how to change the settings of 3D field plots.

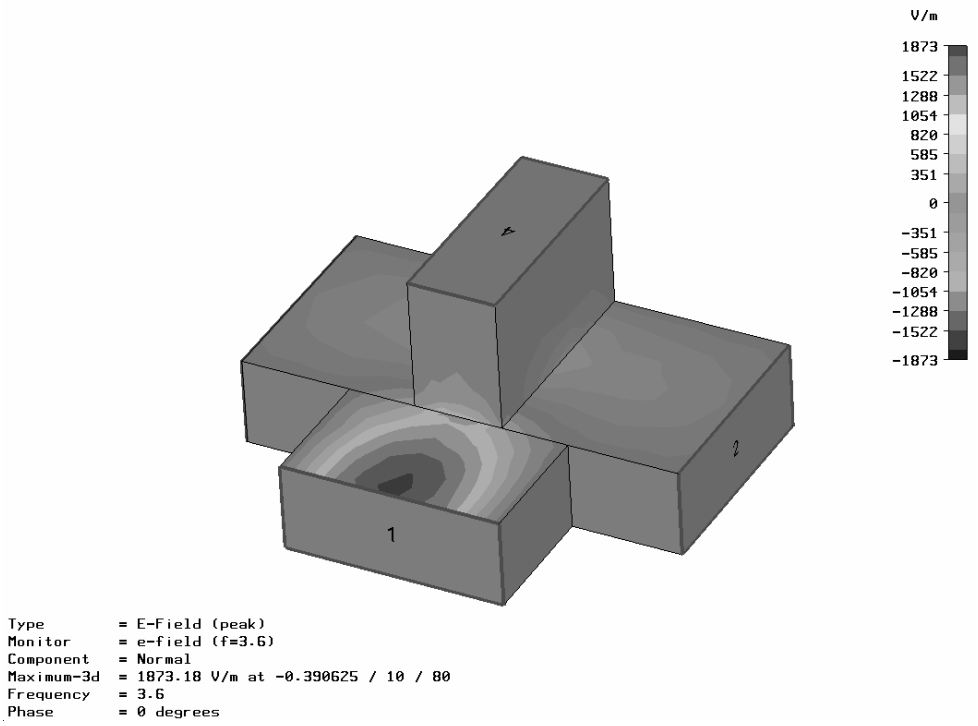


The mode properties shown in the lower left corner of the field plot are close to those computed with the transient solver. Note that the frequency of the mode is not exactly equal to the frequency used by the transient solver. The frequency domain solver calculates the modes for every frequency sample. The frequency of the visualized mode can be selected as follows: Right-click on the current view or on the *Port Modes* ⇨ *Port1* ⇨ *e1* folder in the navigation tree, choose "Select Mode Frequency" from the menu and select the mode frequency in the dialog as shown below:





The three-dimensional electric field distribution in the Magic Tee can be visualized by opening the *2D/3D Results* ⇨ *E-Field* ⇨ *efield (f=3.6)[1]* folder of the navigation tree. After selecting the *Normal* item the field plot will show a three dimensional contour plot of the electric field normal to the surface of the structure.

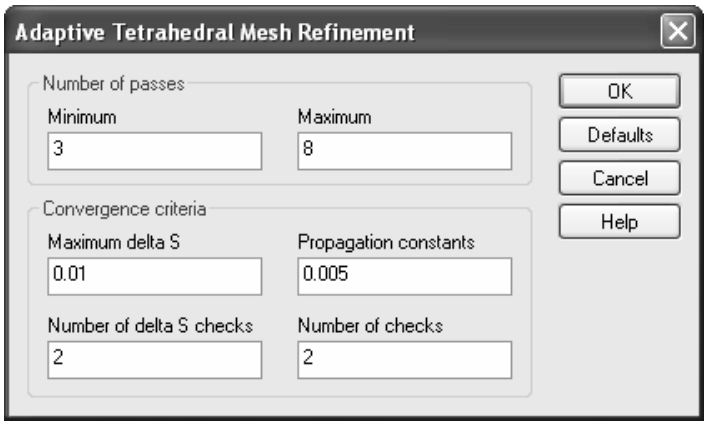


Here, you can observe the effect of the coarse tetrahedral mesh on the computed field: the contours of the computed field distribution are slightly irregular. The next section describes how to influence the mesh refinement and improve the quality of the computed field distribution.

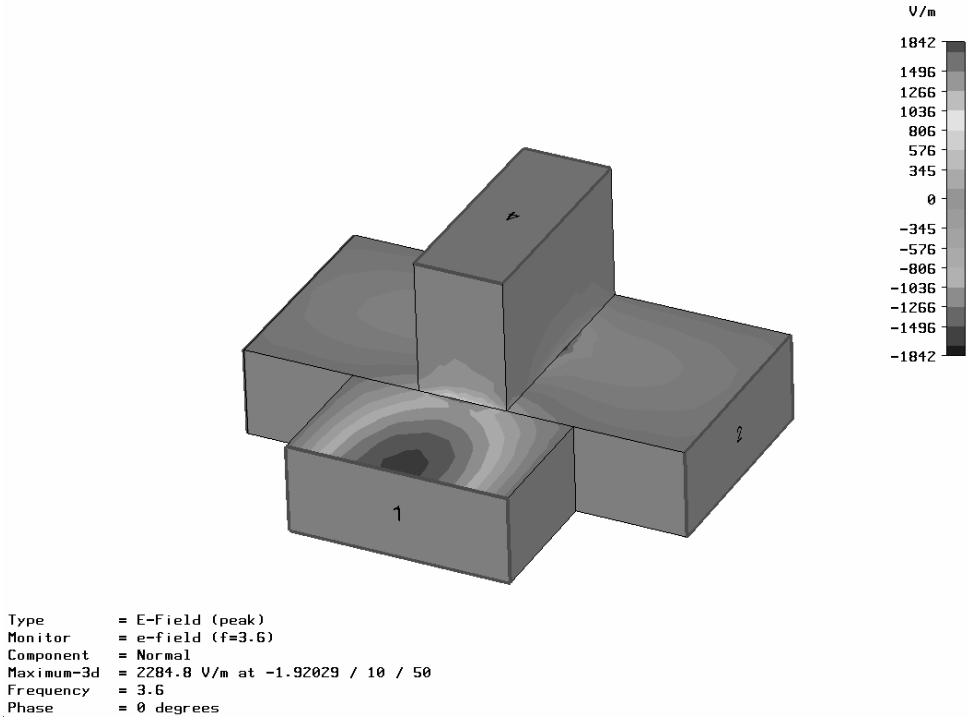
You can display an animation of the fields by checking the *Animate Fields* option in the context menu (right mouse click in the plot window). The appearance of the plot can be changed in the plot properties dialog box that can be opened by selecting *Results* ⇨ *Plot Properties* from the main menu or *Plot Properties* from the context menu. Alternatively, you can double-click on the plot to open this dialog box.

Accuracy Considerations

The results of the frequency domain solver using the tetrahedral mesh are mainly affected by the inaccuracies arising from the finite mesh resolution. In the case of a tetrahedral mesh the adaptive mesh refinement is switched on by default. The mesh adaptation is performed by checking the convergence of the S-parameter values at the highest simulation frequency. The adaptation is oriented towards achieving highly accurate S-parameter calculations. If the quality of the results seems unsatisfactory, additional mesh refinements can be performed. In the following example, three additional mesh adaptation passes were forced by re-starting the frequency domain solver without changing any parameters. Three mesh adaptation passes will be performed according to the *Minimum* number setting in the "Number of passes" frame. This setting can be accessed by pressing *Properties* in the "Adaptive mesh refinement" frame of the "Frequency Domain Solver Parameters" dialog:



The resulting plot of the normal component of the electric field is shown in the next figure. Three additional refinements lead to a noticeably smoother contour plot. The amplitude of the field distribution is very close to the corresponding amplitude delivered by the transient solver.



Getting More Information

Congratulations! You have just completed the Rectangular Waveguide tutorial that should have provided you with a good working knowledge on how to use transient and frequency domain solvers to calculate S-parameters. The following topics have been covered:

1. General modeling considerations, using templates, etc.
2. Use picked points to define objects relatively to each other.
3. Define ports.
4. Define frequency ranges.
5. Define field monitors.
6. Start the transient or the frequency domain solver.
7. Visualize port signals and S-parameters.
8. Visualize port modes and field monitors.
9. Check the truncation error of the time signals.
10. Obtain accurate and converged results using the automatic mesh adaptation.

You can obtain more information for each particular step from the online help system that can be activated either by pressing the *Help* button in each dialog box or by pressing the *F1* key at any time to obtain context sensitive information.

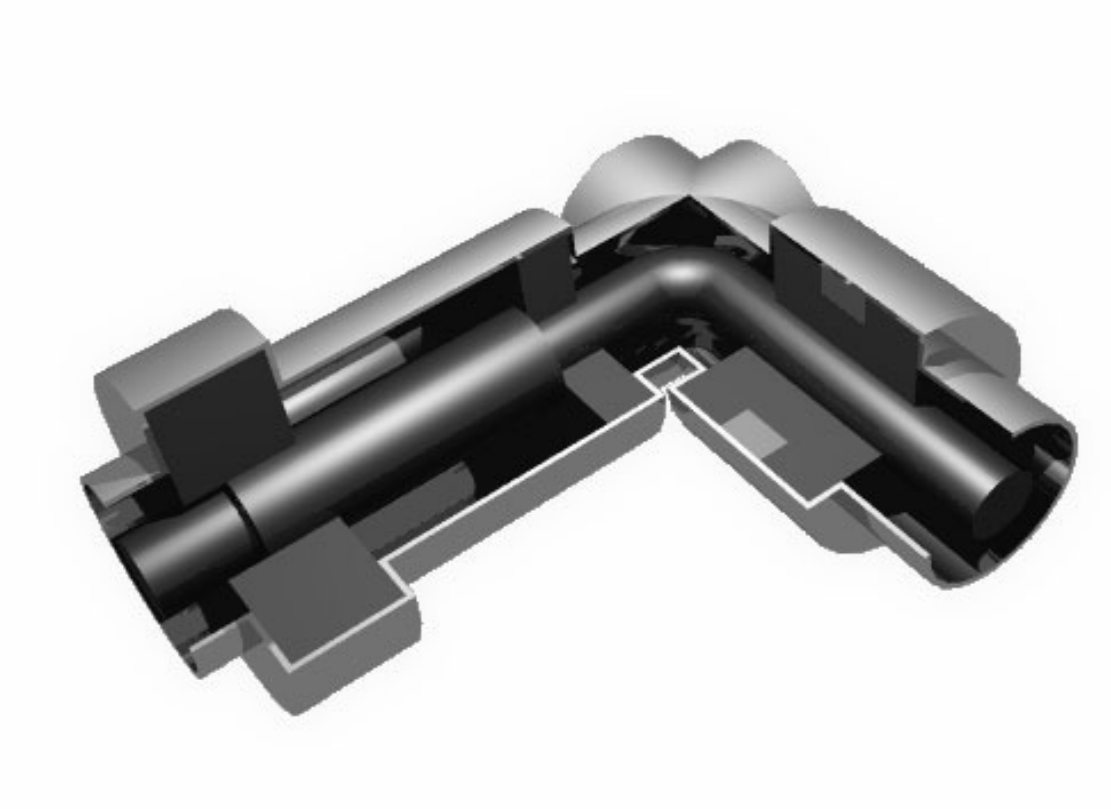
In some cases we have referred to the *Getting Started* manual that is also a good source of information for general topics.

In addition to this tutorial, you can find some more S-parameter calculation examples in the “examples” folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that will give you some more information about the particular device.

Finally, you should refer to the *Advanced Topics* manual for more in-depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please also visit one of the training classes held regularly at a location near you. Thank you for using CST MICROWAVE STUDIO®!

Coaxial Structure Tutorial



Geometric Construction **32**

Introduction and Model Dimensions 32
Geometric Construction Steps 33

Common Solver Settings **53**

S-Parameter Calculation **61**

Transient Solver 61
Transient Solver Results 62
Accuracy Considerations 66
Frequency Domain Solver 69
Results of the Frequency Domain Solver 71
Comparison of the Solver Results 74

Getting More Information **75**

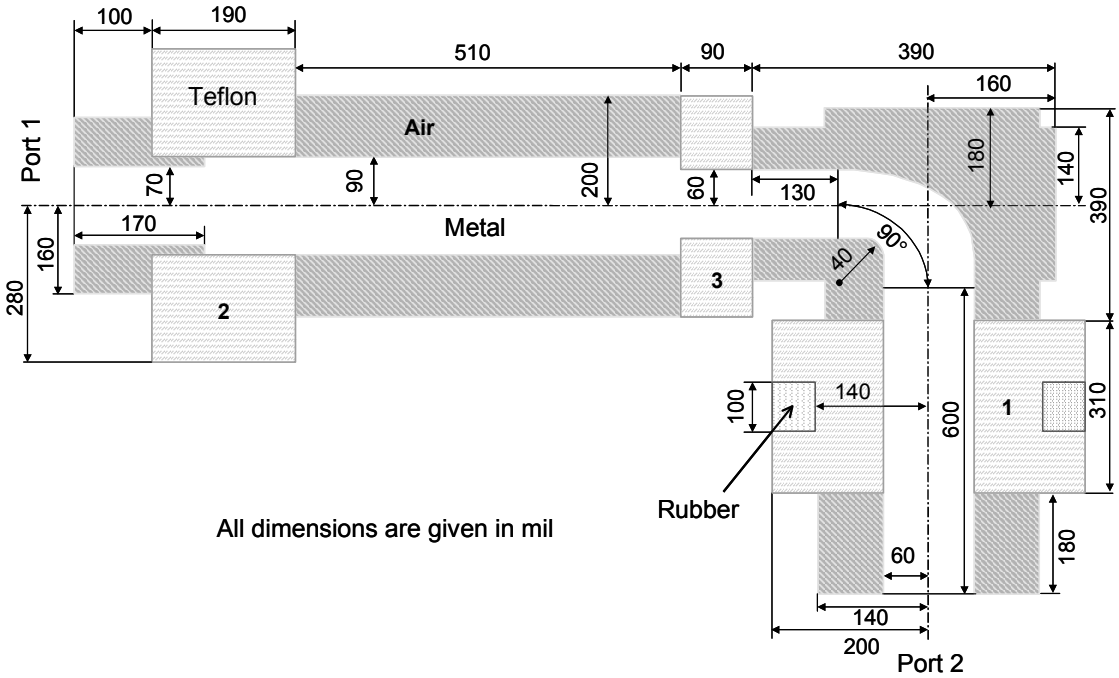
Geometric Construction

Introduction and Model Dimensions

In this tutorial you learn how to simulate coaxial structures. As a typical example for a coaxial structure you will analyze a 90 degree coaxial connector. The following explanations on how to model and analyze this device can also be applied to other devices containing coaxial components.

CST MICROWAVE STUDIO® provides a wide variety of different solvers and results. This tutorial, however, concentrates on S-parameters. The results will be obtained by two alternative techniques. One simulation will be performed in time domain on a hexahedral mesh and the other in frequency domain on a tetrahedral mesh.

We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.



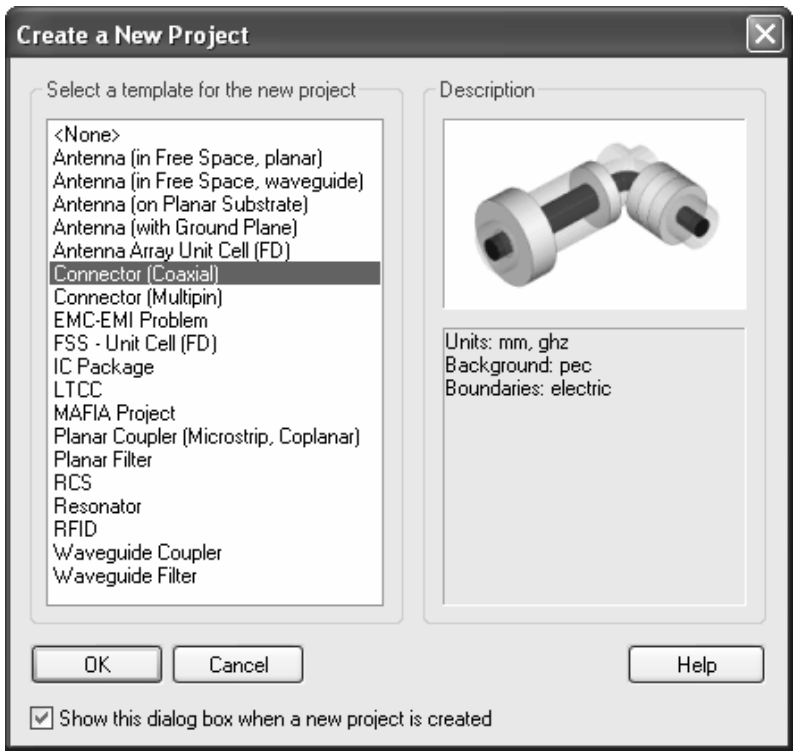
The structure shown above consists of several coaxial sections. The inner conductor of the connector is made from perfect electrically conducting material and is embedded in vacuum. This structure is mounted at three locations with Teflon rings. One of these fixtures additionally contains a rubber ring.

Geometric Construction Steps

This tutorial 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.

☐ Select a Template

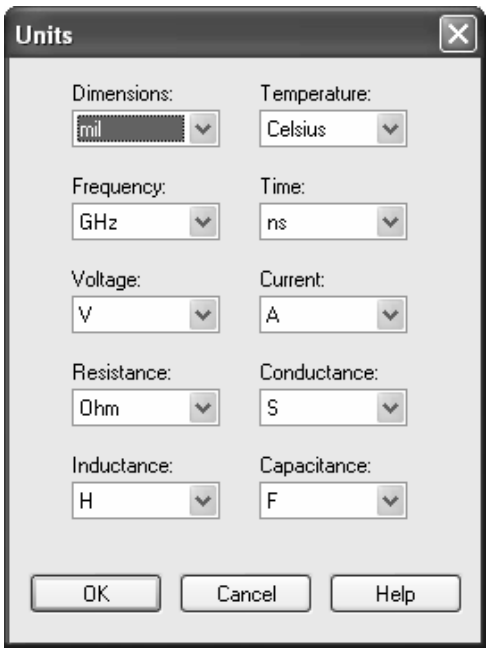
Once you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the "Coaxial Connector" template should be chosen.



This template automatically sets the units to mm and GHz and the background material and all boundaries to be perfect electrical conductors. Because the background material and all boundary conditions have been set to be perfect electrical conductors, you only need to model the interior parts of the connector.

☐ Set the Units

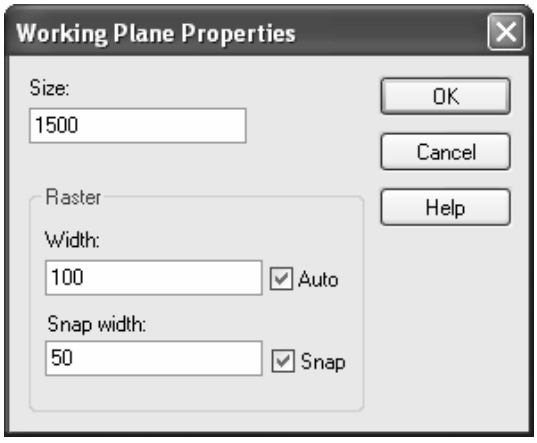
The template has automatically set the geometrical units to mm. Because all geometrical dimensions are given in mil for this example, you should change this setting manually. Therefore, open the units dialog box by selecting *Solve* ⇨ *Units* from the main menu:



Set the *Dimensions* to *mil* and press *OK*.

☐ Set the Working Plane's Properties

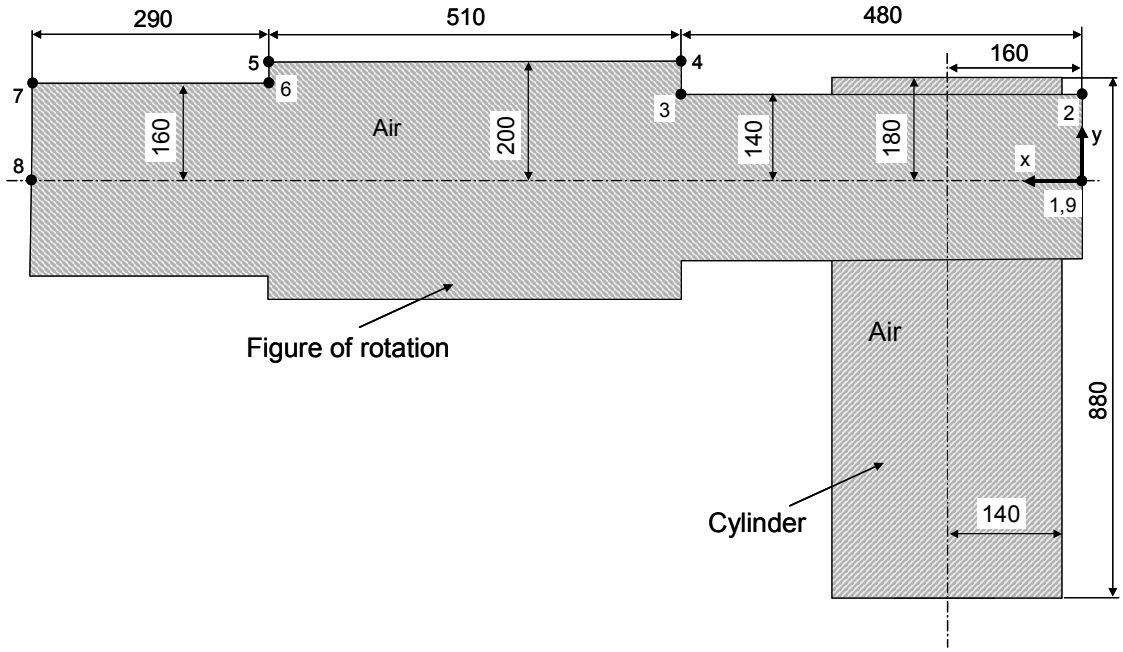
The next step is usually to set the working plane properties in order to make the drawing plane large enough for your device. Since the structure has a maximum extent of 1320 mil along a coordinate direction, the working plane size should be set to at least 1500 mil. These settings can be changed in the dialog box that opens after selecting *Edit* ⇨ *Working Plane Properties* from the main menu. Please note that we will use the same document conventions here as introduced in the *Getting Started* manual.



In this dialog box, you should set the *Size* to 1500 (the unit which has previously been set to mil is displayed in the status bar), the *Raster width* to 100 and the *Snap width* to 50 to obtain a reasonably spaced grid. Please confirm these settings by pressing the *OK* button.

☐ Draw the Air Parts

In this structure, the air can be easily modeled by uniting two rotational symmetric parts (a figure of rotation and a cylinder) as shown in the following diagram:

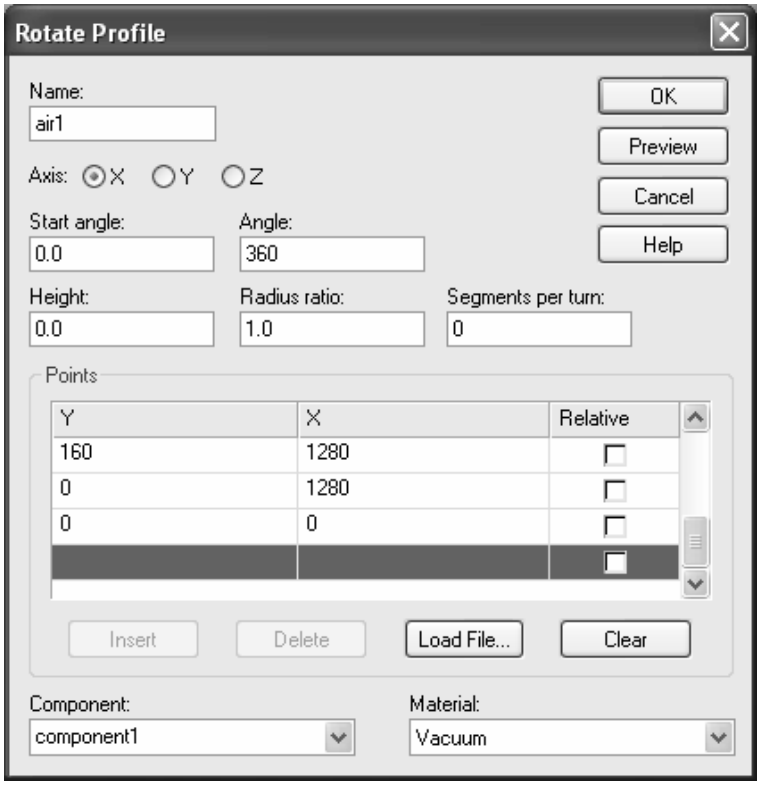


In the first step, you can begin drawing the figure of rotation. Because the cross section profile is a simple polygon, you do not need to use the curve modeling tools here (please refer to the *Getting Started* manual for more information on this advanced functionality). For polygonal cross sections it is more convenient to use the figure of rotation tool, activated by selecting *Objects* ⇨ *Rotate* from the main menu or pressing the "Rotate" toolbar button .

Since no face has been previously picked, the tool will automatically enter a polygon definition mode and request that you enter the polygon's points. You can do this by either double-clicking on each point's coordinates on the drawing plane or entering the values numerically. Since the latter approach may be more convenient, we suggest pressing the *Tab* key and entering the coordinates in the dialog box. All polygon points can thus be entered step-by-step according to the following table. Whenever you make a mistake, you can delete the most recently entered point by pressing the *Backspace* key. Arbitrary changes can also be made at the end of the input process.

Point	X	Y
1	0	0
2	0	140
3	480	140
4	480	200
5	990	200
6	990	160
7	1280	160
8	1280	0
9	0	0

After the last point has been entered, the polygon will then be closed. The *Rotate Profile* dialog box will then automatically appear.

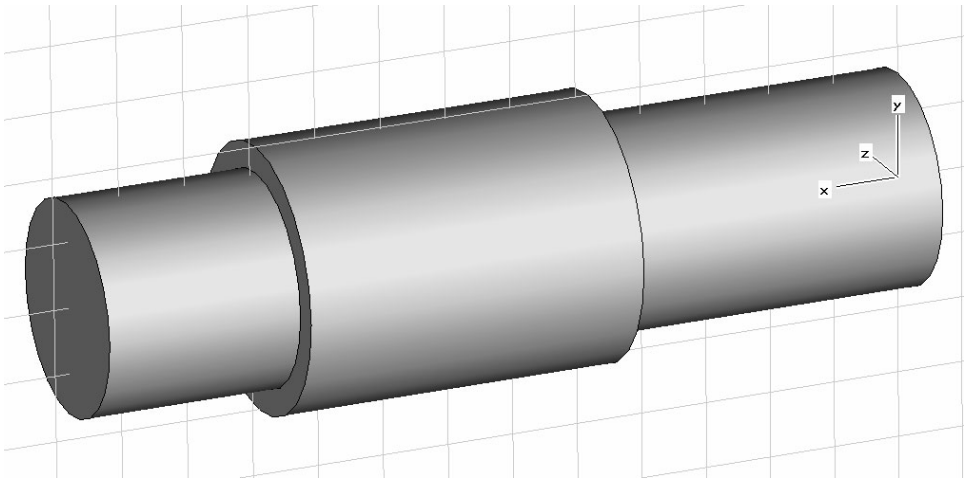


This dialog box allows you to review the coordinate settings in the list. If you encounter any mistakes, you can easily change the values by double-clicking on the incorrect coordinate entry field.

The next step is to assign a specific *Component* and a *Material* to the shape. In this case, the default settings with “component1” and “Vacuum” are practically appropriate.

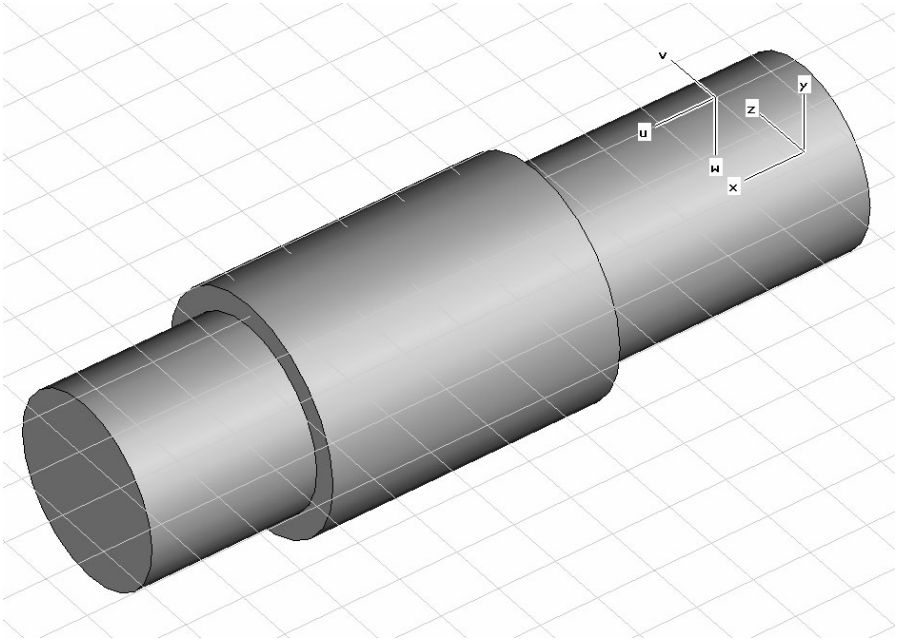
> **Please note:** The use of different components allows you to collect several solids into specific groups, independent of their material behavior. However, here it is convenient to construct the complete connector as a representation of one component.

Finally, assign a proper *Name* (e.g. “air1”) to the shape and press the *OK* button to finish the creation of the solid. The picture below shows how your structure should appear (you may need to rotate the view as explained in the *Getting Started* manual in order to obtain this plot).



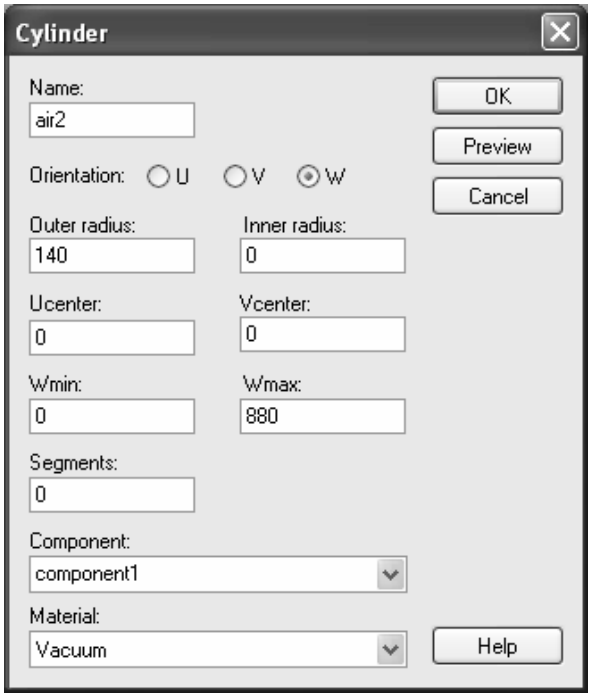
Construction of the second air part by creating a cylinder can be simplified if a local coordinate system is first introduced. You can activate the local coordinate system by selecting *WCS ⇨ Local Coordinate System* or pressing the corresponding toolbar button . Afterwards, the origin of this coordinate system should be moved by selecting *WCS ⇨ Move Local Coordinates* (). The following dialog box allows you to enter a vector along which the origin of the working coordinate system (WCS) will be moved.

You should shift the origin by 160 mil along the u direction and by 180 mil along the v direction in order to position the WCS at the center of the cylinder's base. Afterwards, rotate the WCS along its u-axis by 90 degrees by selecting *WCS ⇨ Rotate +90° around U axis* or using the shortcut *Shift+U*. The model should then look as follows:

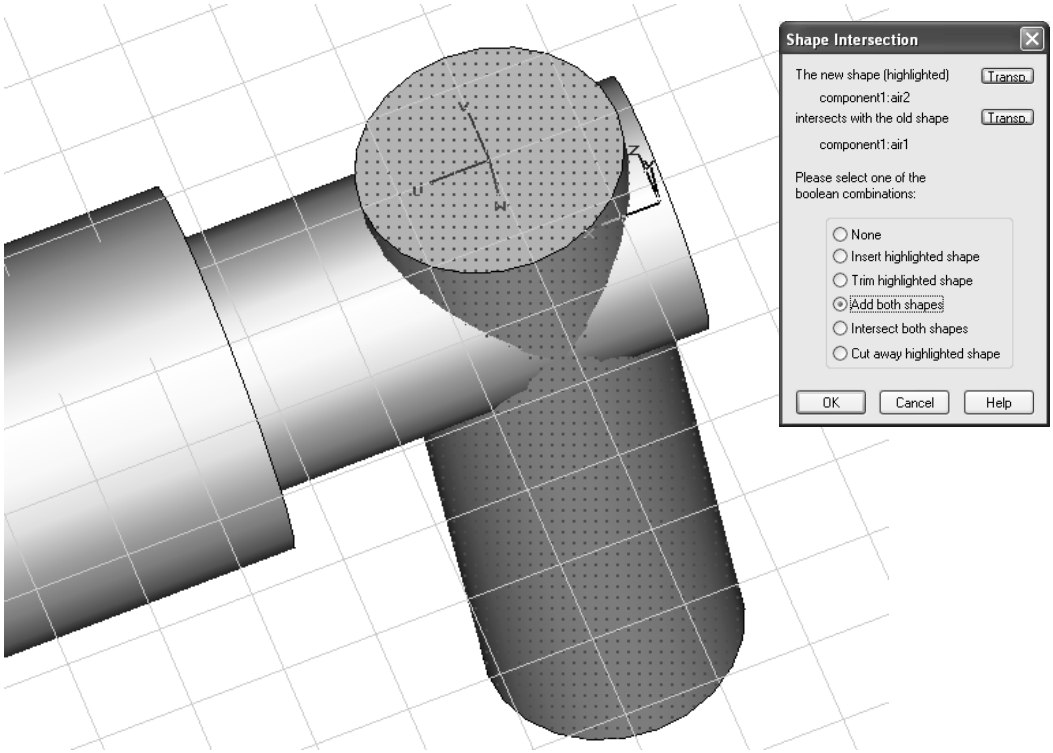


The second air part can now be created using the cylinder tool: *Objects* ⇨ *Basic Shapes* ⇨ *Cylinder* ( ). Once the cylinder creation mode is active, you are requested to pick the center of the cylinder. Because this is now the origin of the working coordinate system, you can simply press *Shift+Tab* to open the dialog box for numerically entering the coordinates and confirm the settings by pressing *OK* (please note that holding down the *Shift* key while pressing the *Tab* key opens the dialog box with the coordinate values initially set to zero rather than the current mouse pointer's location).

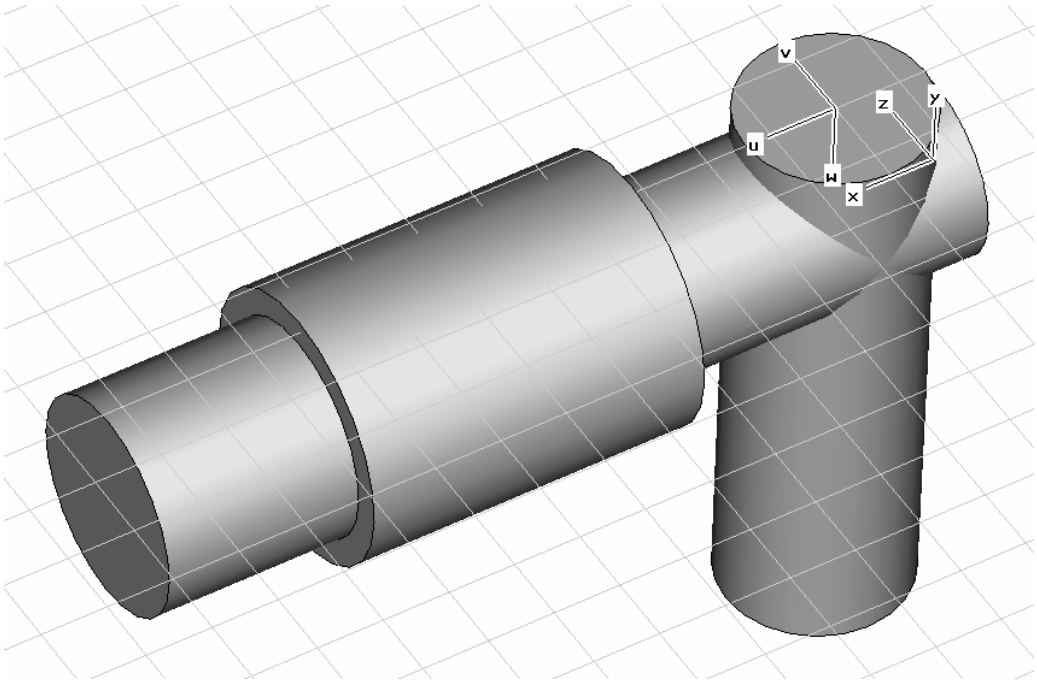
You are now requested to enter the outer radius of the cylinder. Please press the *Tab* key again and set the *Radius* to 140 before pressing the *OK* button. The *Height* of the cylinder can then be set to 880 in the same manner. Skip the definition of the inner radius by pressing the *Esc* key (the air should be modeled as solid cylinder here) and check your settings in the following dialog box:



Finally, set the *Name* of the cylinder to “air2” and verify that the solid again is associated with the vacuum *Material*. Confirm your settings by pressing *OK*. Since the two air parts overlap each other, the shape intersection dialog box will open automatically, asking you to select a Boolean operation to combine the shapes:



Please select the operation *Add both shapes* to unite both parts and press the *OK* button. Your model should then finally look as follows:



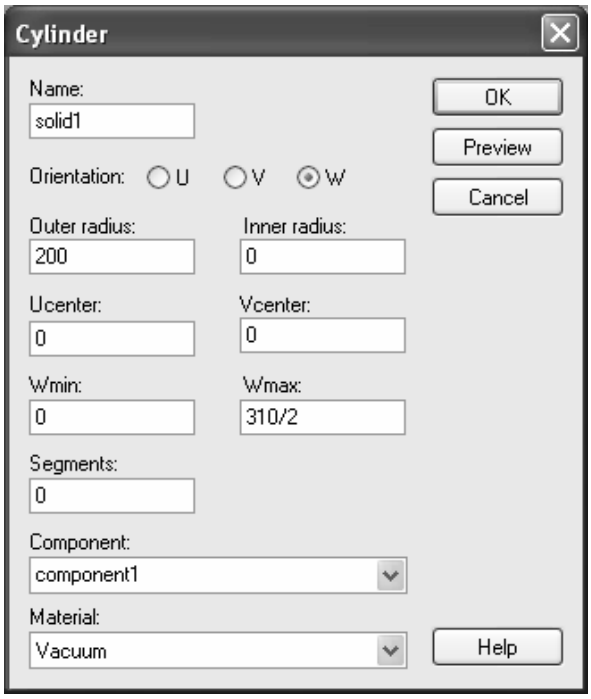
☐ Model the Teflon and Rubber Cylinders

After successfully modeling the air parts, you can now create the first Teflon cylinder. It is advantageous to move the WCS to the middle of the Teflon cylinder by selecting *WCS* ⇨ *Move Local Coordinates* ( ):

In this dialog box, enter the expression "390 + 310 / 2" in the *DW* field to move the WCS along the w-axis by this amount. Please refer to the structure's schematic drawing earlier in this tutorial to confirm that the new origin of the WCS is located in the center of the first Teflon cylinder.

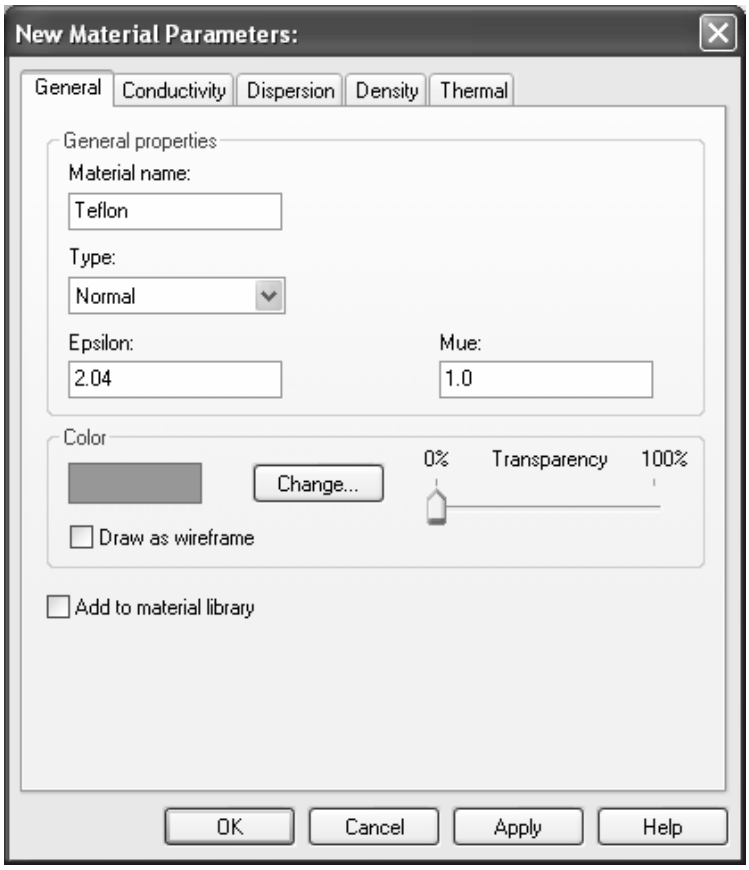
Once the coordinate system is properly located, you can now easily model the Teflon cylinder by selecting *Objects* ⇨ *Basic Shapes* ⇨ *Cylinder* ( ). When you are requested to enter the cylinder's center, press *Shift+Tab* and check the coordinate values U=0, V=0 before clicking the *OK* button.

Afterwards, press the *Tab* key again to set the cylinder's outer radius to 200. The height of the cylinder should then be set to "310 / 2" (you are currently modeling only one half of the cylinder) in the same way. After skipping the definition of the inner radius by pressing the *Esc* key, the cylinder creation dialog box should appear:



In this dialog, you should now assign a proper *Name* (e.g. “teflon1”) to the shape before you enter the expression “–310 / 2” in the *Wmin* field to properly set the cylinder’s full length.

You can skip the *Component* setting because the complete connector will be constructed as one component. However, the cylinder’s material is currently set to Vacuum. In order to change this, select “[New material…]” in the *Material* dropdown list, opening the material parameter dialog box:

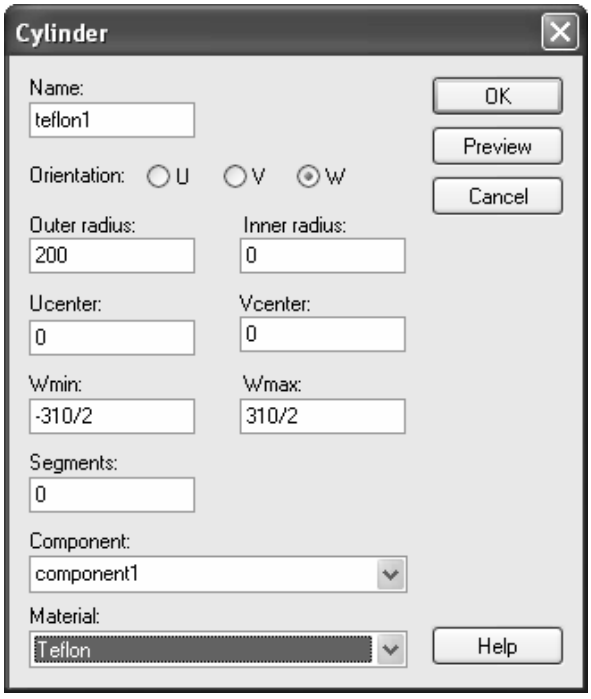


In this dialog box you should set the *Material name* to “Teflon” and the *Type* to be a “Normal” dielectric material. Afterwards, you can specify the dielectric constant of Teflon by entering “2.04” in the *Epsilon* field. Select the *Change* button in the *Color* frame and choose a color.

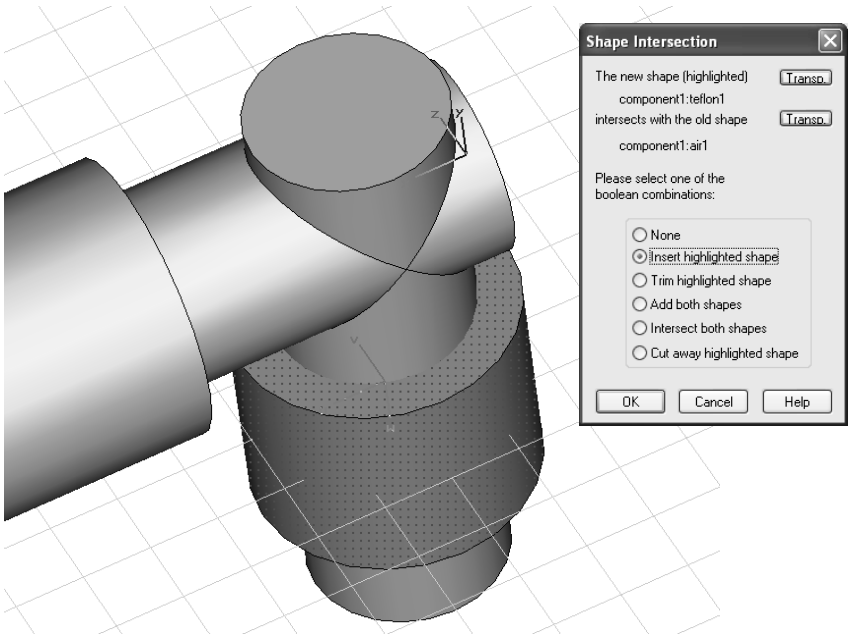
Finally, you should check your settings in the dialog box again before pressing the *OK* button to store the material’s parameters.

> **Please note:** The defined material “Teflon” will now be available inside the current project for the further creation of other solids. However, if you also want to 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 *Load from Material Library* in the *Materials* context menu in the navigation tree.

The dialog for the cylinder creation should now look as follows:



After checking the current settings, create the cylinder by pressing the *OK* button. Since the Teflon cylinder overlaps the previously modeled air parts, the shape intersection dialog box will appear again:

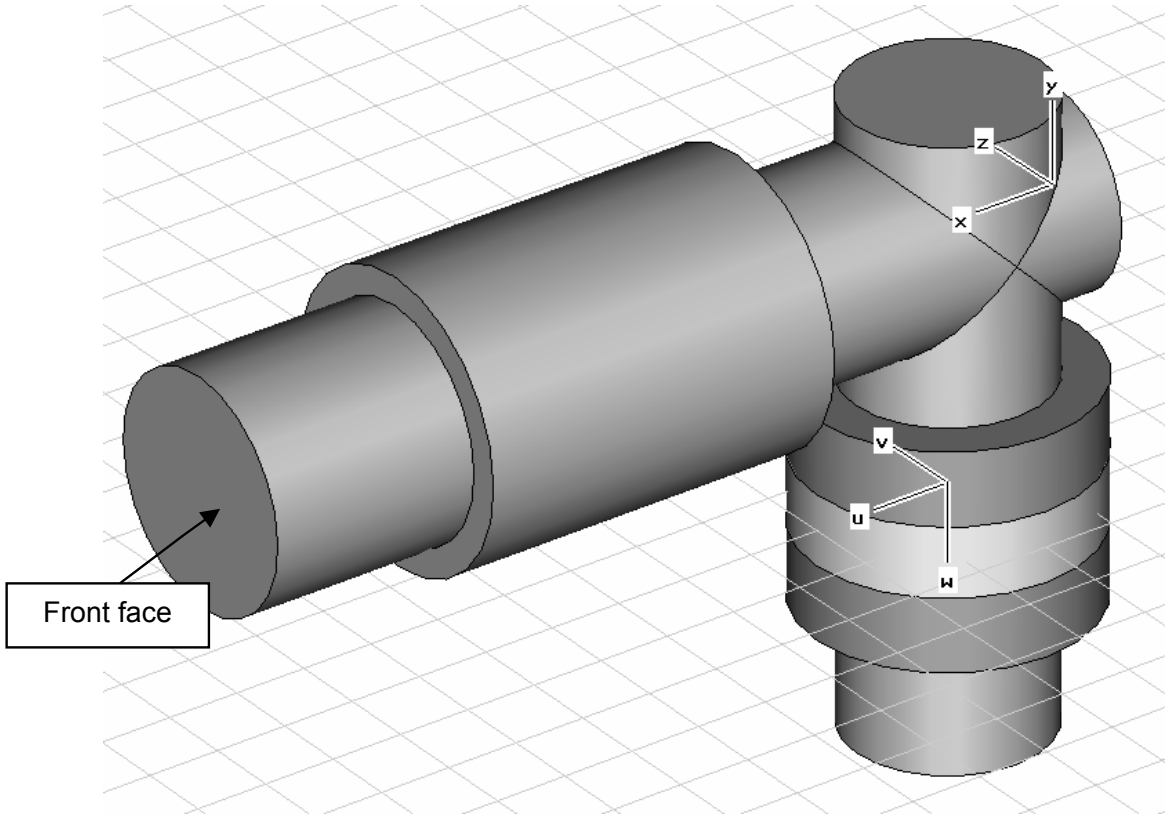


Here, you should choose to insert the new Teflon cylinder into the air part by selecting *Insert highlighted shape* before pressing the *OK* button. Please refer to the *Getting Started* manual for more information on Boolean operations.

Afterwards, the rubber ring inside the first Teflon cylinder can be modeled analogously:

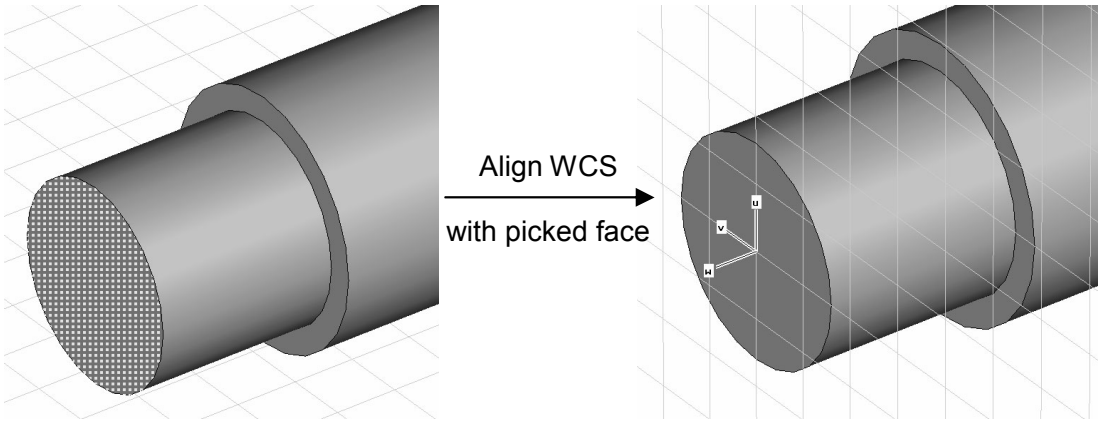
1. Activate the cylinder creation tool (*Objects ⇨ Basic Shapes ⇨ Cylinder,*).
2. Press *Shift+Tab* and set the center point to *U* = 0, *V* = 0.
3. Press *Tab* and set the *Radius* to 200.
4. Press *Tab* and set the *Height* to 100/2.
5. Press *Tab* and set the inner *Radius* to 140.
6. Set the *Name* to "rubber" and enter "–100/2" in the *Wmin* field.
7. Select "[New Material…]" from the *Material* dropdown list to create a new material.
8. In the material properties dialog box set the *Material name* to "Rubber", its *Type* to "Normal" and its dielectric constant *Epsilon* to 2.75.
9. Choose a color by pressing the *Change* button and confirm the material creation by pressing *OK*.
10. Back in the cylinder creation dialog box, verify the material assignment to "Rubber" and press the *OK* button.
11. In the shape intersection dialog box, choose *Insert highlighted shape* and press *OK*.

After successfully performing all above steps, your model should look as follows:



Before you continue with the construction of the two remaining Teflon rings, the working coordinate system should be aligned with the front face shown in the picture above.

Therefore, please activate the pick face tool by either selecting *Objects ⇨ Pick ⇨ Pick Face* from the main menu, pressing the corresponding toolbar button or just using the shortcut *f* (while the main view is active). Afterwards, the front face should be selected by double-clicking on it. The working coordinate system is aligned with the front face by selecting *WCS ⇨ Align WCS With Selected Face* (or shortcut *w*):

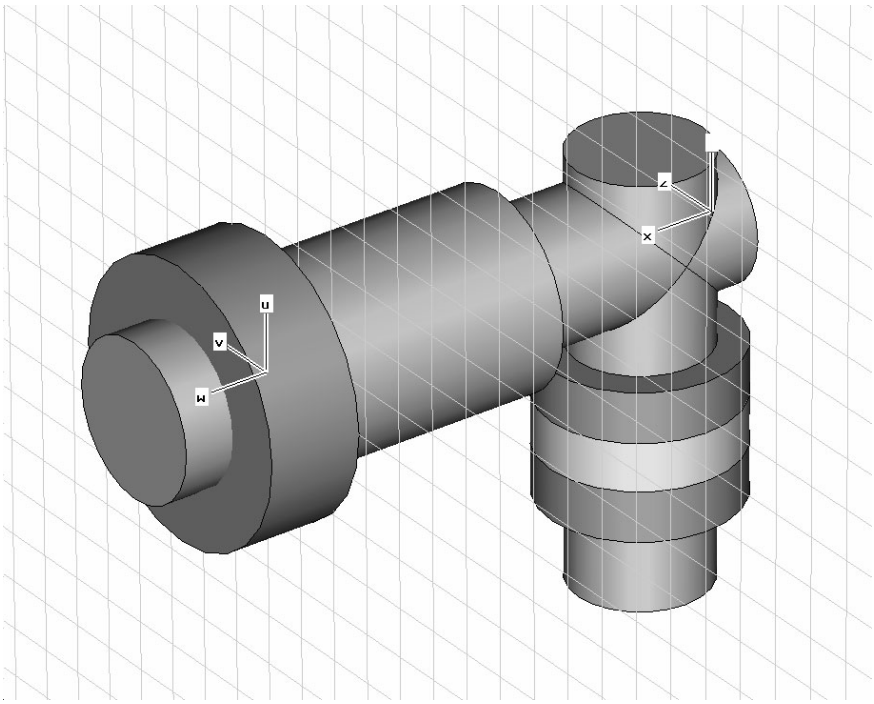


The next step is to move the WCS to the location of the second Teflon cylinder's base. Therefore, select *WCS* ⇨ *Move Local Coordinates* () to open the corresponding dialog box:

Please enter –290 in the DW field before pressing *OK*. Now you can straightforwardly model the second Teflon cylinder as shown in the structure's drawing.

1. Activate the cylinder creation tool (*Objects* ⇨ *Basic Shapes* ⇨ *Cylinder,* ).
2. Press *Shift*+*Tab* and set the center point to *U* = 0, *V* = 0.
3. Press *Tab* and set the *Radius* to 280.
4. Press *Tab* and set the *Height* to 190.
5. Press *Tab* and set the inner *Radius* to 90.
6. In the cylinder creation dialog box set the *Name* to "teflon2" and select the previously defined *Material* "Teflon" before pressing *OK*.
7. In the shape intersection dialog box, choose *Insert highlighted shape* and press *OK*.

The model should then look as follows:



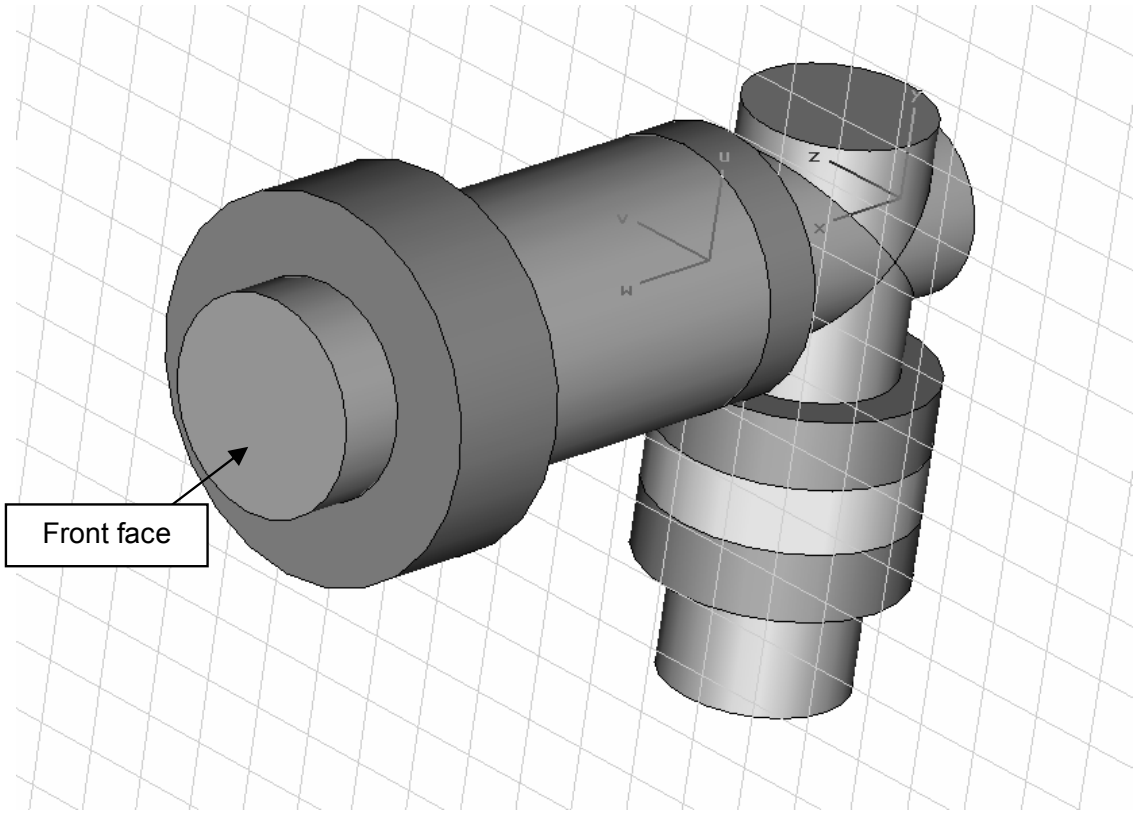
For the creation of the third Teflon ring you should again move the WCS to the proper location:

1. Select *WCS* ⇨ *Move Local Coordinates* (🔄) to open the move WCS dialog box.
2. Enter –600 in the *DW* field and press *OK*.

The construction of the Teflon ring can then be performed by:

1. Activate the cylinder creation tool (*Objects* ⇨ *Basic Shapes* ⇨ *Cylinder,* 🛢).
2. Press *Shift+Tab* and set the center point to *U* = 0, *V* = 0.
3. Press Tab and set the *Radius* to 200.
4. Press Tab and set the *Height* to 90.
5. Press *Esc* to skip the definition of the inner radius.
6. In the cylinder creation dialog box, set the *Name* to “teflon3” and select the previously defined *Material* “Teflon” before pressing *OK*.
7. In the shape intersection dialog box, choose *Insert highlighted shape* and press *OK*.

After successfully completing these steps, the model should look as follows:

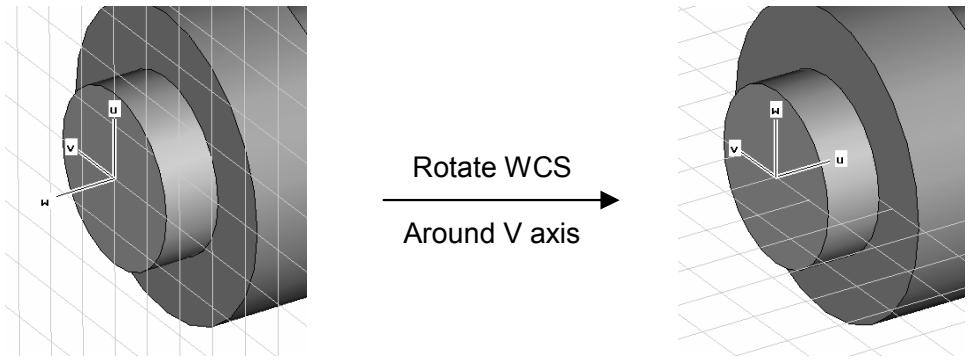


☐ Model the Inner Conductor

Creation of the inner conductor of the coaxial cable can again be simplified by aligning the working coordinate system with the front face shown in the above picture:

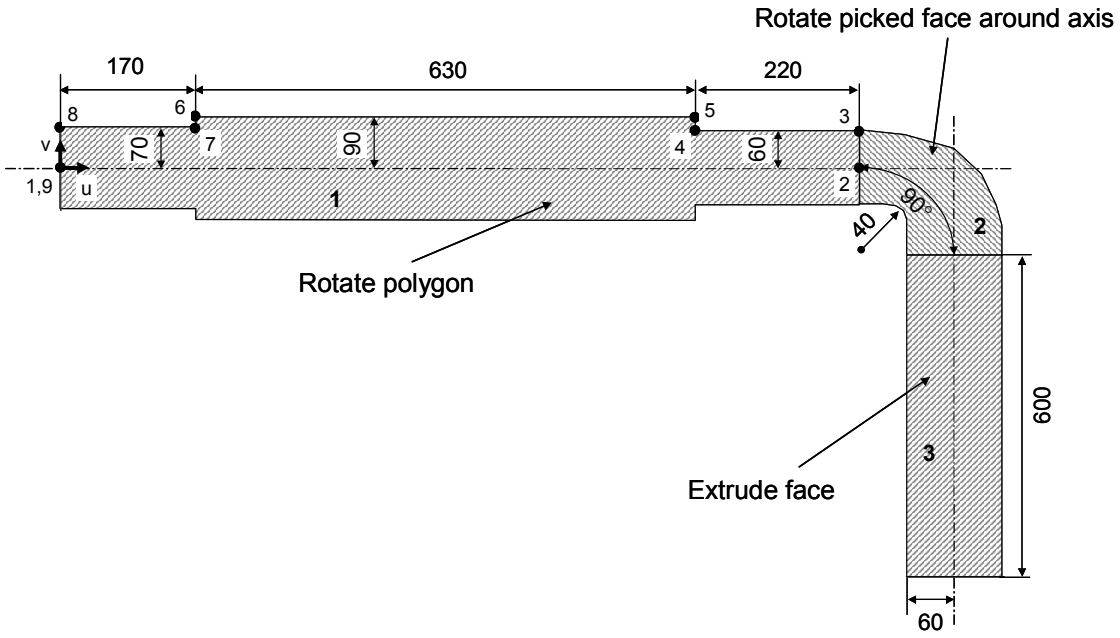
1. Activate the pick face tool (*Objects* ⇨ *Pick* ⇨ *Pick Face*, *f*)
2. Double-click on the front face shown above.
3. Align the WCS with the picked face (*WCS* ⇨ *Align WCS with Selected Face*, *w*)

The new location of the working coordinate system should then be shown as pictured below:



Next, the WCS should be rotated such that its u axis points into the structure by invoking the command *WCS* ⇨ *Rotate +90° around V axis* (*Shift+v*).

Now the first part of the inner conductor can be modeled by a figure of rotation that is shown in the schematic drawing below:



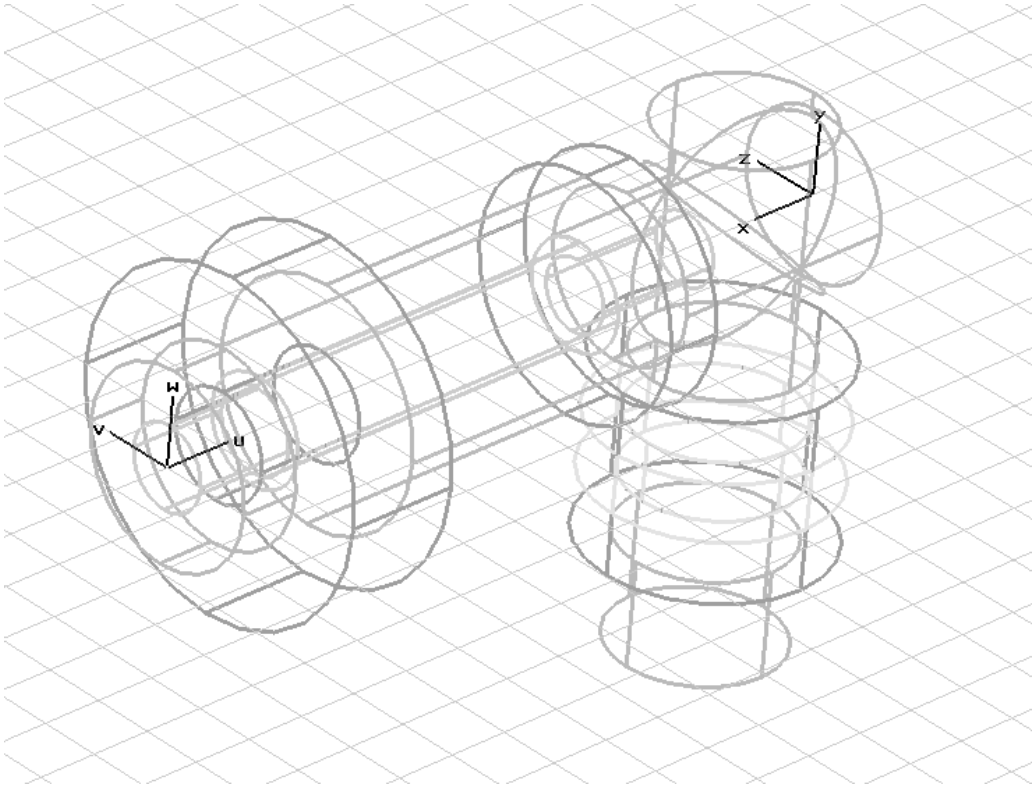
Since the construction of a figure of rotation has already been explained in detail earlier in this tutorial, we will only give a short list of steps:

1. Activate the wireframe visualization mode by pressing the toolbar icon or using the shortcut *Ctrl*+*w*. This will allow you to follow the construction; otherwise, the parts of the inner conductor will be hidden inside the previously created shapes.
2. Activate the figure of rotation tool (*Objects* ⇨ *Rotate*,).
3. Enter the points shown in the table below by pressing the *Tab* key and specifying the coordinate values numerically:

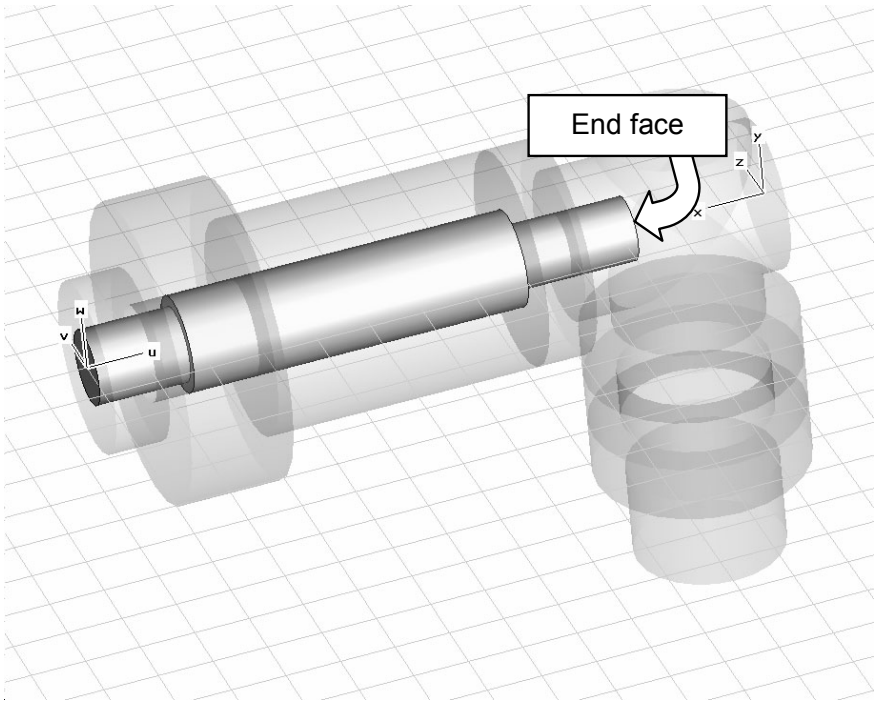
Point	U	V
1	0	0
2	1020	0
3	1020	60
4	800	60
5	800	90
6	170	90
7	170	70
8	0	70
9	0	0

4. Once the last point has been defined the polygon is closed. The rotate profile creation dialog box will open. In this dialog box check the points' coordinates, set the *Name* of the shape to "conductor1" and change the *Material* assignment to "PEC" before pressing *OK*.

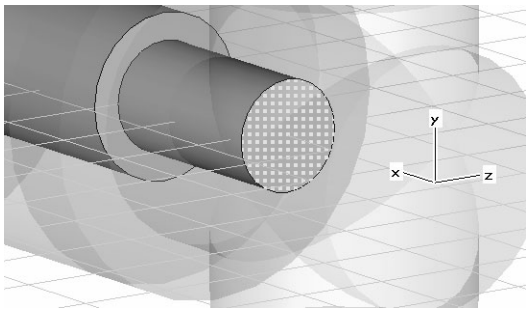
After successfully performing these steps, the structure should look as follows:



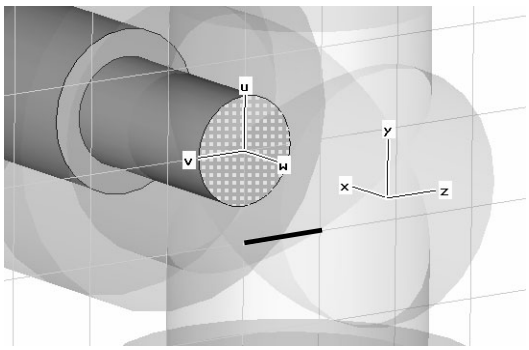
Please select the inner conductor by either double-clicking on it in the view or by selecting *component1* ⇨ *conductor1* from the navigation tree. Once this is done, you may deactivate the wireframe plot mode (or *Ctrl*+*w*):



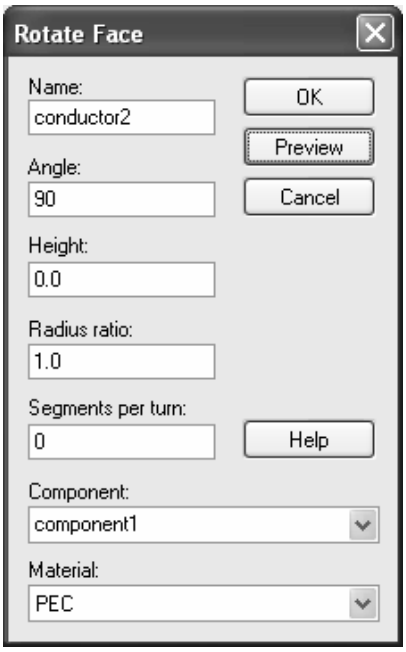
The next step is to align the working coordinate system with the inner conductor's end face shown above. Therefore, rotate the view to ensure that the end face is visible (select *View* ⇨ *Mode* ⇨ *Rotate* and drag the mouse while pressing the left button). Afterwards, activate the face pick tools (*Objects* ⇨ *Pick* ⇨ *Pick Face*, *f*) and double-click on the desired face.



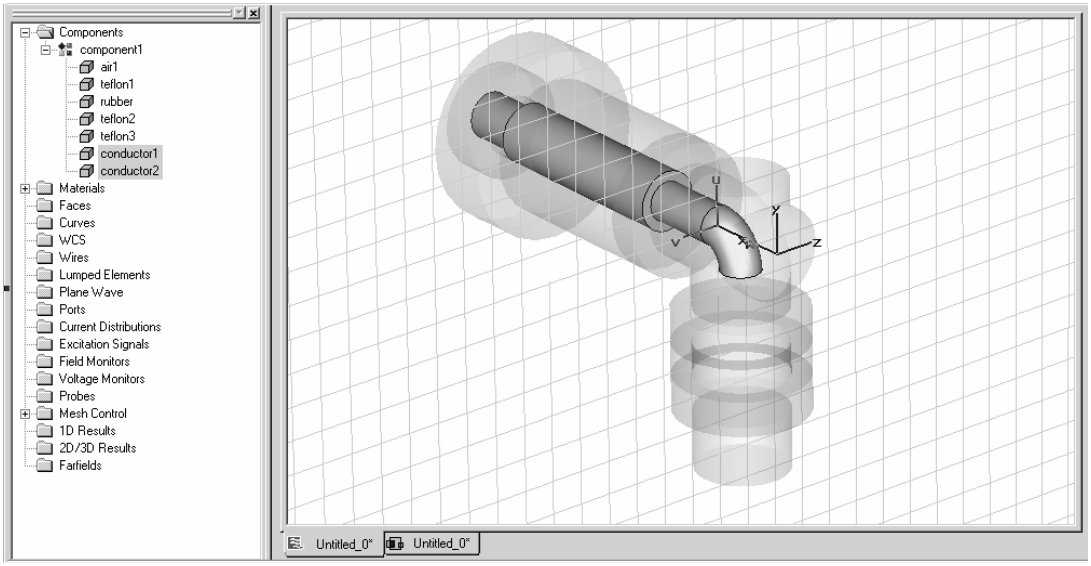
Now, align the WCS with this face by pressing *WCS* ⇨ *Align WCS with Selected Face* (*w*). The next step is to define a rotation axis by selecting *Objects* ⇨ *Pick* ⇨ *Edge from Coordinates* ( ). Enter the first point's coordinates by pressing the *Tab* key (*U* = -100, *V* = 0) and the coordinates of the second point (*U* = -100, *V* = -100) in the same manner. Finally, check and confirm the settings in the dialog box.



Afterwards you need to select the previously picked face once again (*Objects* ⇨ *Pick* ⇨ *Pick Face*, + double-click on the face). Activate the rotate face mode by selecting *Objects* ⇨ *Rotate*, ( ). Because a face has been previously picked, the definition of polygon points is skipped and a dialog box is opened immediately:

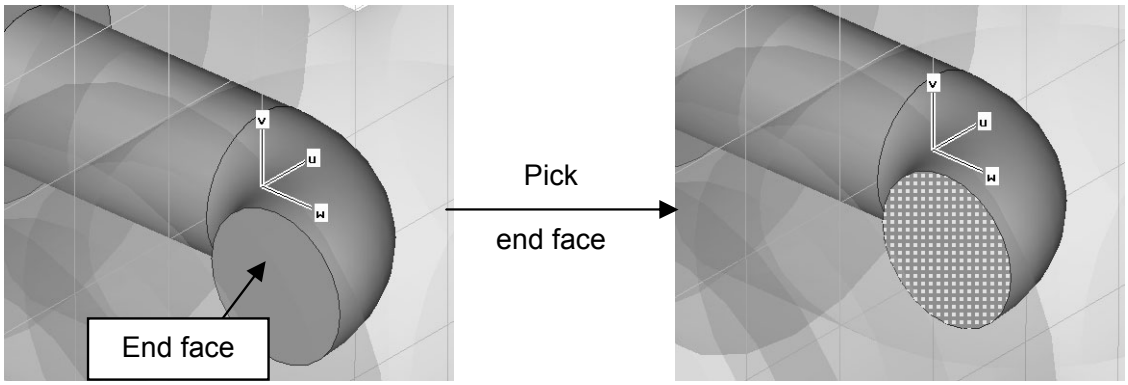


In this dialog box, you should first assign a proper *Name* (e.g. “conductor2”) to the shape. Because the rotation axis is aligned with the negative v-axis direction and the rotation angle is specified in a right-handed system, the *Angle* must be set to 90 degrees (the rotation axis is visualized by a blue arrow while the dialog box is open). If not already set, change the *Material* assignment to “PEC” and press the *OK* button.



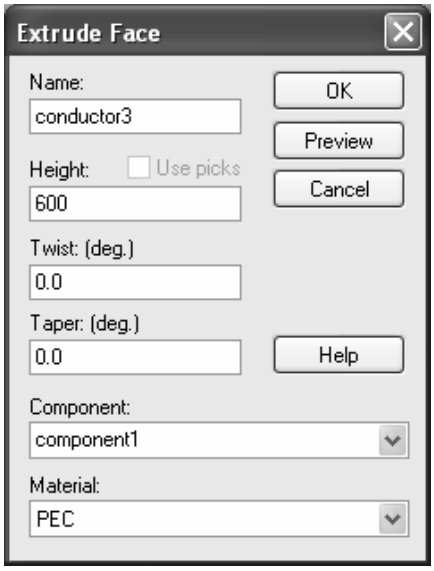
The last step in the model’s geometric construction process is to create the third conductor by extruding the end face of the figure of rotation defined above.

Rotate the view to obtain a picture similar to the following:

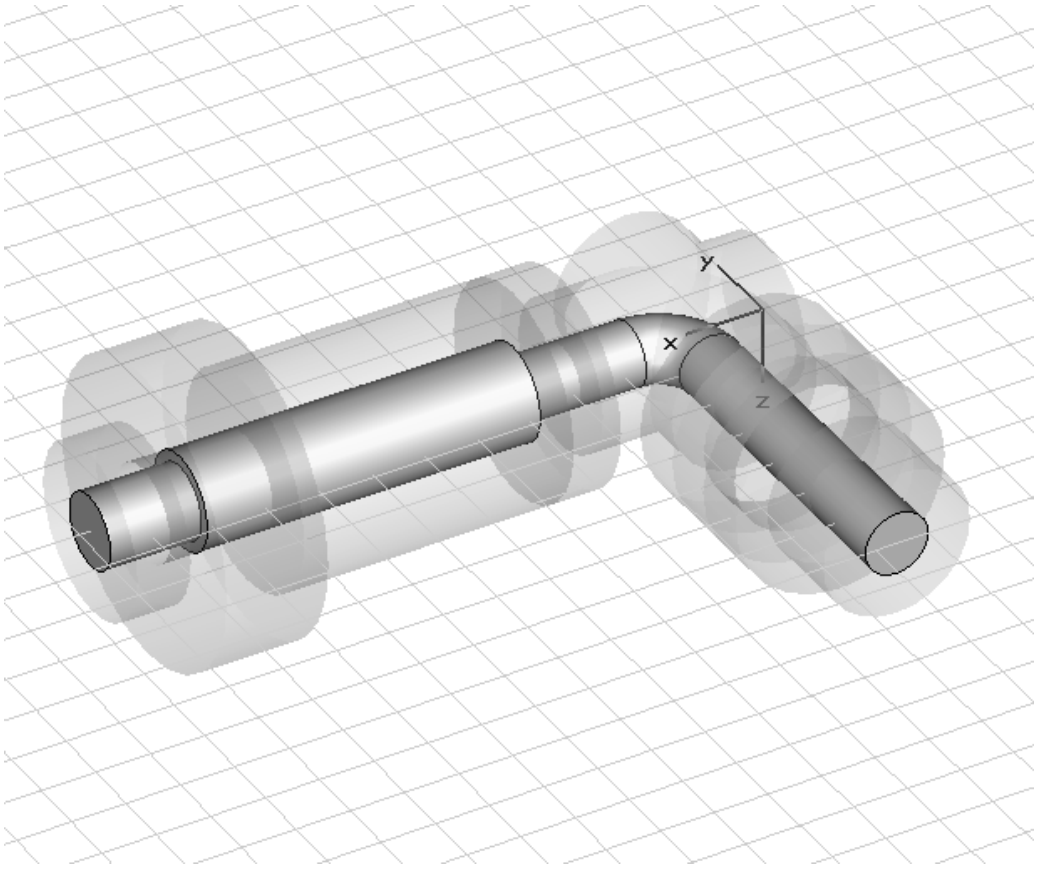


Afterwards, select “conductor2” and pick the end face of the conductor as shown in the above picture (*Objects* ⇨ *Pick* ⇨ *Pick Face*, + double-clicking on the end face).

You can now open the Extrude Face dialog box by selecting *Objects* ⇨ *Extrude* or pressing the toolbar button . As with the figure of rotation, a polygon definition mode will be entered by the extrude tool if no face has been previously selected.



In this dialog box, please assign a proper *Name* (e.g. “conductor3”), set the *Height* to 600 (mil) and the *Material* assignment to “PEC” before pressing the *OK* button. Your structure should then look as follows:



Common Solver Settings

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

- Define Ports

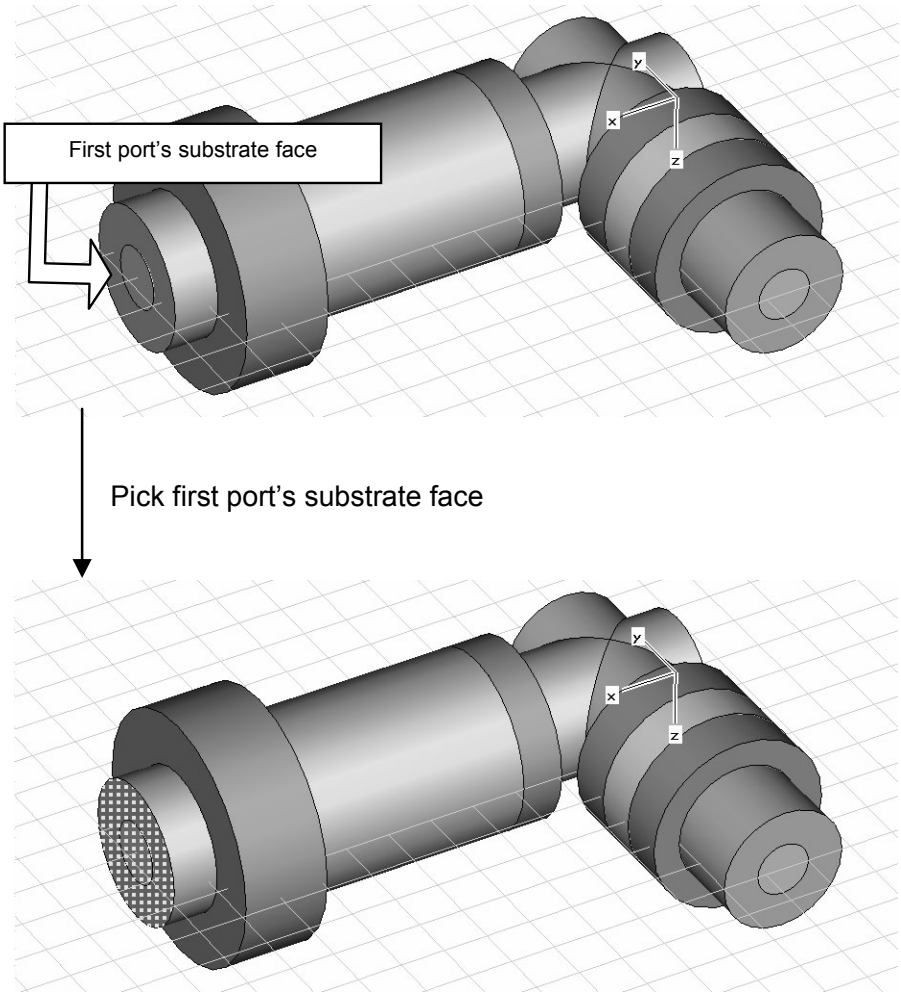
Each port that is defined will simulate an infinitely long waveguide (here a coaxial cable) that is connected to the structure at the port's plane. Waveguide ports are the most accurate way to calculate the S-parameters of filters and should therefore be used in this case.

Since a waveguide port is based on the two dimensional mode patterns in the waveguide's cross-section, the port must be defined large enough to entirely cover these mode fields. In the case of a coaxial cable, the port has to completely cover the coaxial cable's substrate.

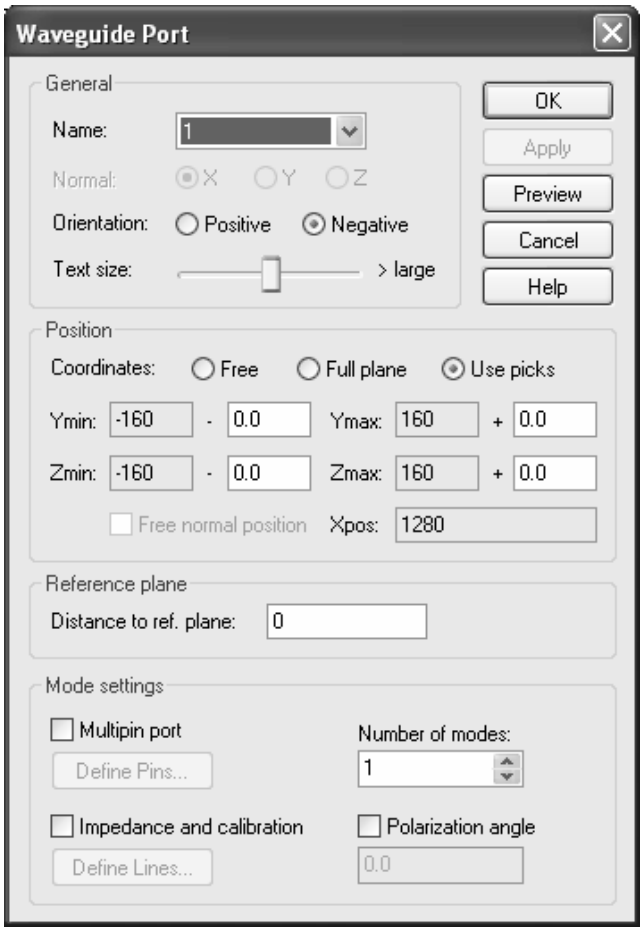
Before you continue with the port definition, please clear the selection by either double-clicking on the view's background or selecting *Components* in the navigation tree.

The port's extent can be defined either numerically or, as is more convenient here, by picking the face to be covered by the port. Therefore, activate the pick face tool (*Objects*

⇨ *Pick* ⇨ *Pick Face*, *f*) and double-click the substrate's port face at the first port as shown in the pictures below:

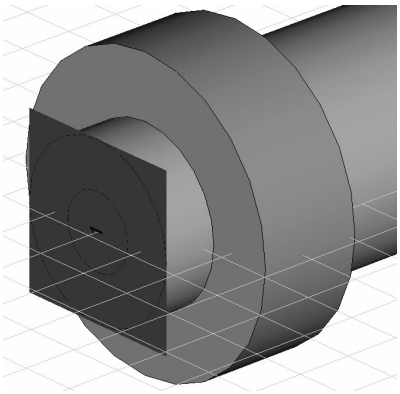


Please open the waveguide dialog box (*Solve* ⇨ *Waveguide Ports*,) to define the first port 1:



Whenever a face is picked before the port dialog is opened, the port's location and size will automatically be defined by the picked face's extent. Thus the port's *Position* (transversal as well as normal) is initially set to *Use picks*. You can accept this setting. The next step is to choose how many modes should be considered by the port. For coaxial devices, we usually have only a single propagating mode. Therefore, you should simply keep the default of one mode.

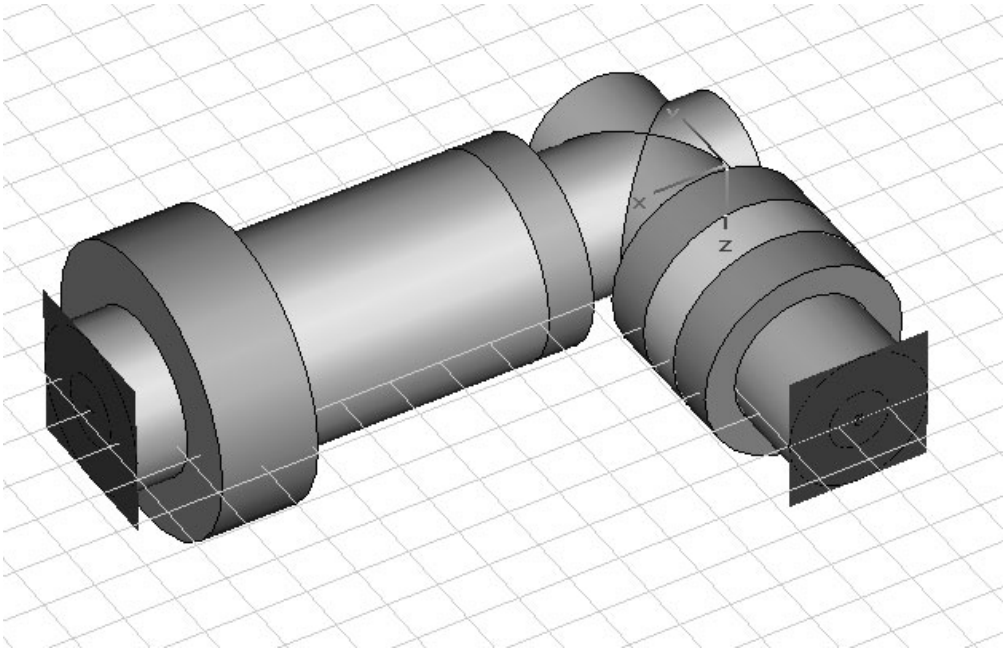
Finally, check the settings in the dialog box and press the *OK* button to create the port:



Now you can repeat the same steps for the definition of the second port:

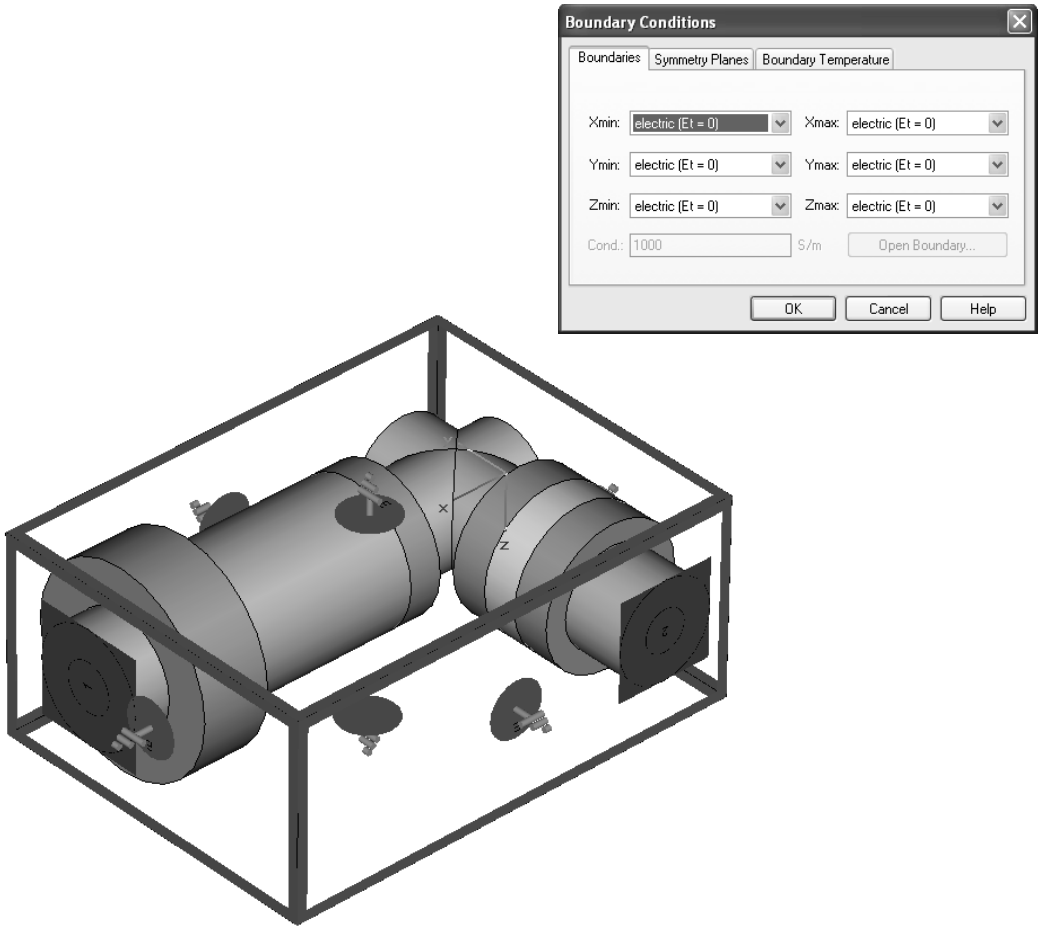
1. Pick the corresponding substrate's port face (*Objects* ⇨ *Pick* ⇨ *Pick Face*,).
2. Open the waveguide dialog box (*Solve* ⇨ *Waveguide Ports*,).
3. Press *OK* to store the port's settings.

Your model including the ports should now look as follows:



☐ Define Boundary Conditions and Symmetries

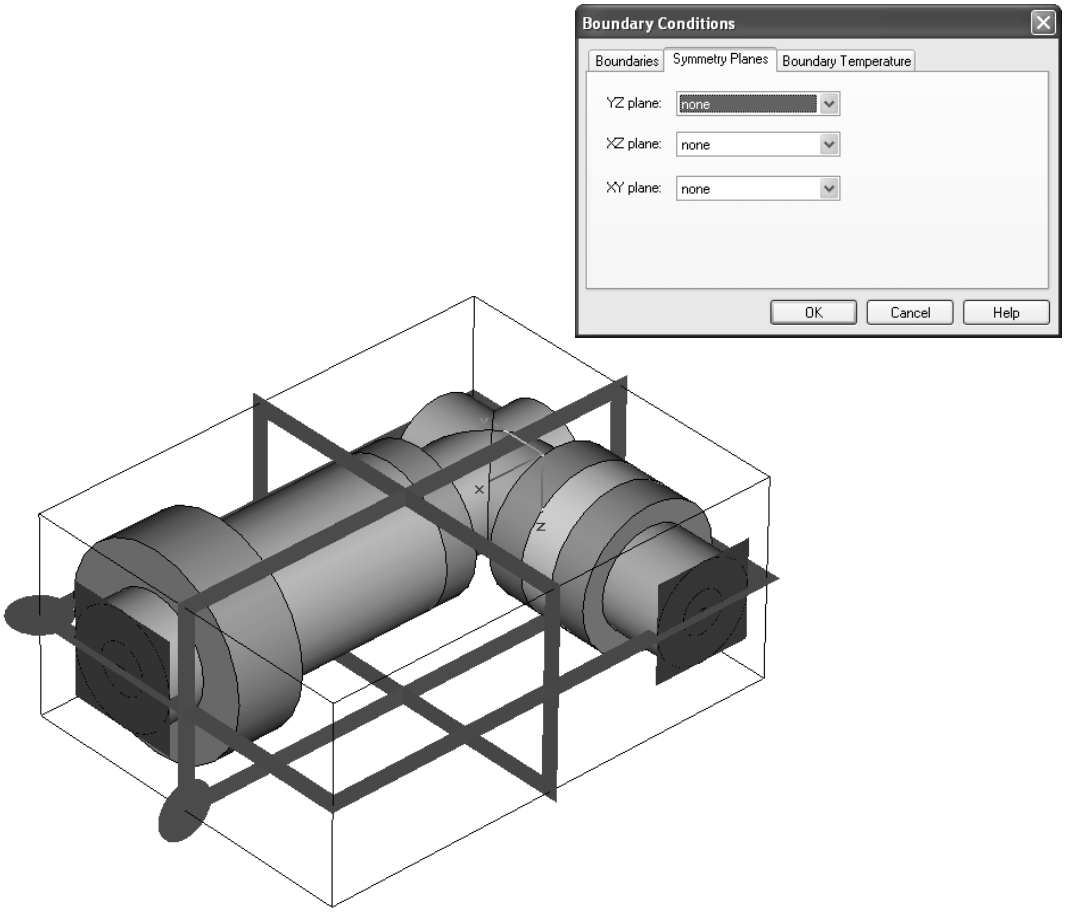
Before you start the solver, you should always check the boundary and symmetry conditions. This is most easily accomplished by entering the boundary definition mode by pressing the tool bar item or selecting *Solve* ⇨ *Boundary Conditions*. The boundary conditions will then become visualized in the main view as follows:



Here, all boundary conditions are set to “electric” which means that the structure is embedded in a perfect electrically conducting housing. These defaults (set by the template) are appropriate for this example.

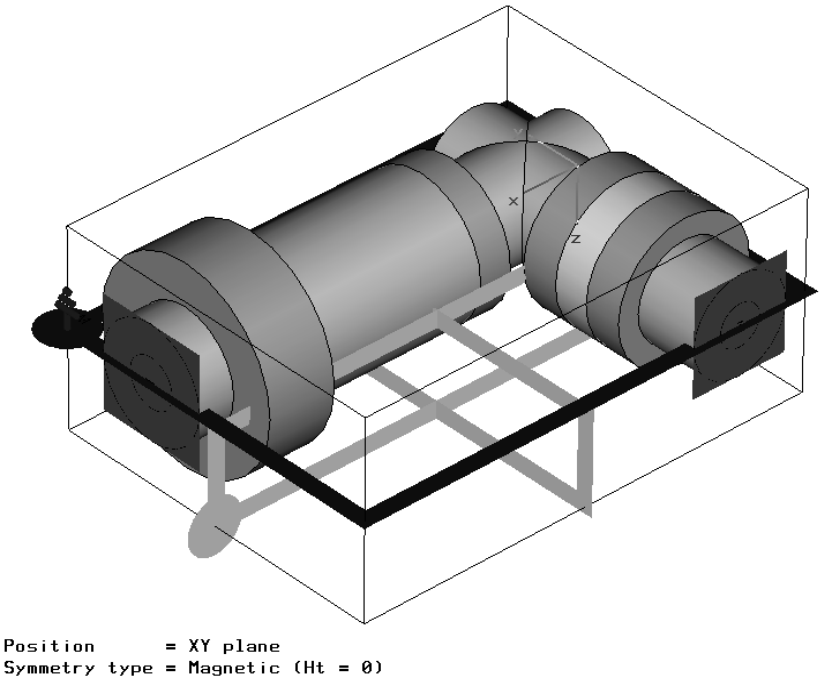
Due to the structure’s symmetry to the XY plane and the fact that the magnetic field in the coaxial cable is perpendicular to this plane, a symmetry condition can be used. This symmetry reduces the time required for the simulation by a factor of two. You should also refer to the example in the *Getting Started* manual for more information on symmetry conditions.

Please enter the symmetry plane definition mode by activating the *Symmetry Planes* tab in the dialog box. The screen should then look as follows:



By setting the symmetry plane XY to magnetic, you force the solver to calculate only the modes that have no tangential magnetic field component on these planes (thereby forcing the electric field to be tangential to these planes).

After these settings have been made, the structure should look as follows:



Please note that you also could double-click on the symmetry plane's handle and choose the proper symmetry condition from the context menu.

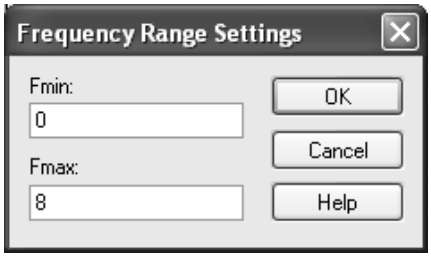
Finally, press the *OK* button to complete this step.

In general, you should always make use of symmetry conditions whenever possible to reduce calculation times by a factor of two to eight.

☐ 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).

For this example, we will choose a frequency range from 0 to 8 GHz. Open the frequency range dialog box (*Solve* ⇨ *Frequency*, ∿) and enter the range of 0 to 8 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz and is displayed in the status bar):

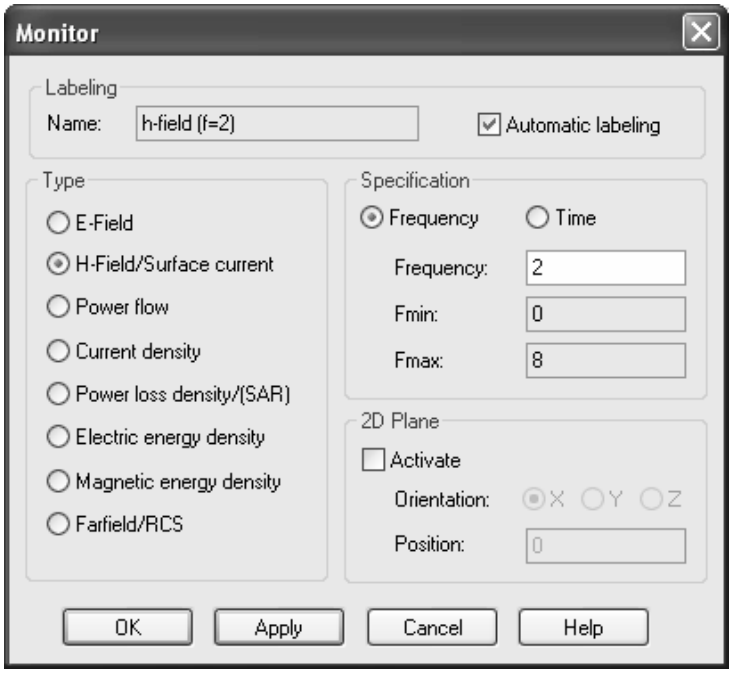


☐ Define Field Monitors

CST MICROWAVE STUDIO® uses the concept of "monitors" to specify which field data to store. In addition to choosing the type, you can also choose whether the field is recorded at a fixed frequency or at a sequence of time samples (the latter case applies only to the transient solver). You may define as many monitors as necessary to obtain the fields at various frequencies. For the transient solver, all monitors are calculated from a single simulation run by the Fourier Transform. Consequently, the computational overhead for a defined monitor is relatively small.

Please note that an excessive number of field monitors may significantly increase the memory space required for the simulation.

Assume that you are interested in the current distribution on the coaxial cable's conductors at several frequencies (2, 4, 6 and 8 GHz). To add field monitors, select *Solve* ⇨ *Field Monitors* from the main menu or press the corresponding icon in the toolbar .



In this dialog box, you should first select the *Type H-Field / Surface current* before specifying the frequency for the monitor in the *Frequency* field. Afterwards, press the *Apply* button to store the monitor's data. Please define monitors for the following frequencies: 2, 4, 6, 8 (with GHz being the currently active frequency unit). Please make sure that you press the *Apply* button for each monitor (the monitor definition is then added in the *Monitors* folder in the navigation tree).

After the monitor definition is completed, you can close this dialog box by pressing the *OK* button.

S-Parameter Calculation

A key feature of CST MICROWAVE STUDIO® is the *Method on Demand* approach that allows a simulator or mesh type that is best suited to a particular problem. Another benefit is the ability to compare the results obtained by completely independent approaches. We demonstrate this strength in the following two sections by calculating the S-parameters with the transient solver as well as the frequency domain solver. The transient simulation uses a hexahedral mesh while the frequency domain calculation is performed with a tetrahedral mesh. Both sections are self-contained parts and it is sufficient to work through only one of them, depending on what solver you are interested in. The chapter ends with a comparison of the two methods.

Please note that not all solvers may be available to you due to license restrictions. Please contact your sales office for more information.

Transient Solver

☐ Frequency Range Considerations for the Transient Solver

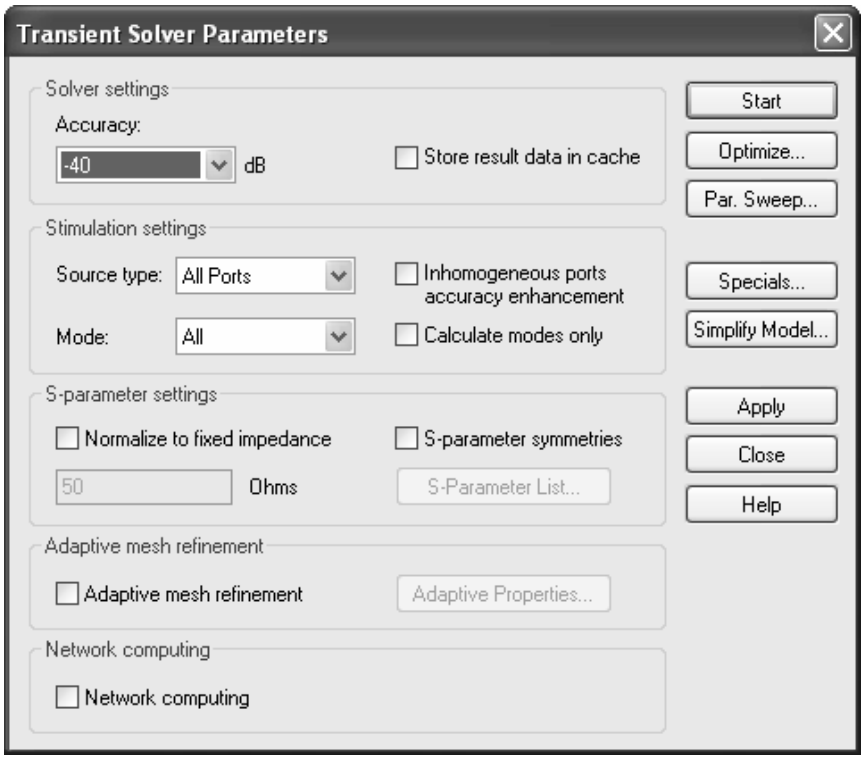
In contrast to frequency domain tools, a transient solver's performance can be degraded if the frequency range is chosen to be too small (the opposite is usually true for frequency domain solvers).

We recommend using reasonably large bandwidths of 20% to 100% for the transient simulation. In this example, the S-parameters are to be calculated for a frequency range between 0 and 8 GHz. With the center frequency being 4 GHz, the bandwidth (8 GHz – 0 GHz = 8 GHz) is 200% of the center frequency, which is acceptable. Thus, you can simply choose the desired frequency range between 0 and 8 GHz.

> **Please note:** Assuming that you are interested primarily in a frequency range of e.g. 11.5 to 12.5 GHz (for a narrow band filter), then the bandwidth would only be about 8.3%. In this case, it is logical to increase the frequency range (without losing accuracy) to a bandwidth of 30%, which corresponds to a frequency range of 10.2 – 13.8 GHz. This extension of the frequency range could speed up your simulation by more than a factor of three!
>
> 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's parameters are specified in the "Transient Solver Parameters" dialog box that can be opened by selecting *Solve* ⇨ *Transient Solver* from the main menu or by pressing the corresponding icon  in the toolbar.



Because this two port structure is lossless, the transient solver will need to calculate only a single port to obtain the full S-matrix, even if you specify *All Ports* for the *Source type*.

In this case, you can keep the default settings and press the *Start* button to start the calculation. A progress bar and an abort button appear in the status bar:.

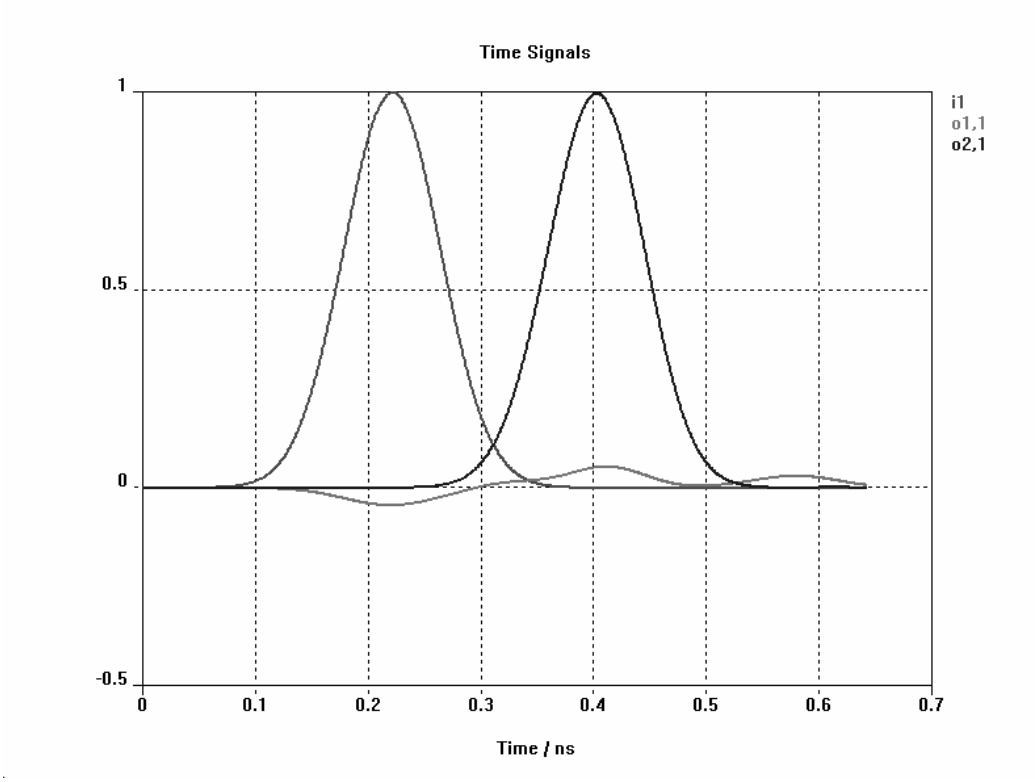
When the simulation has finished or has been aborted, both items disappear. During the simulation, the "Message Window" will show some details about the performed simulation.

Transient Solver Results

Congratulations, you have simulated the coaxial connector using the transient solver! Let's look at the results:

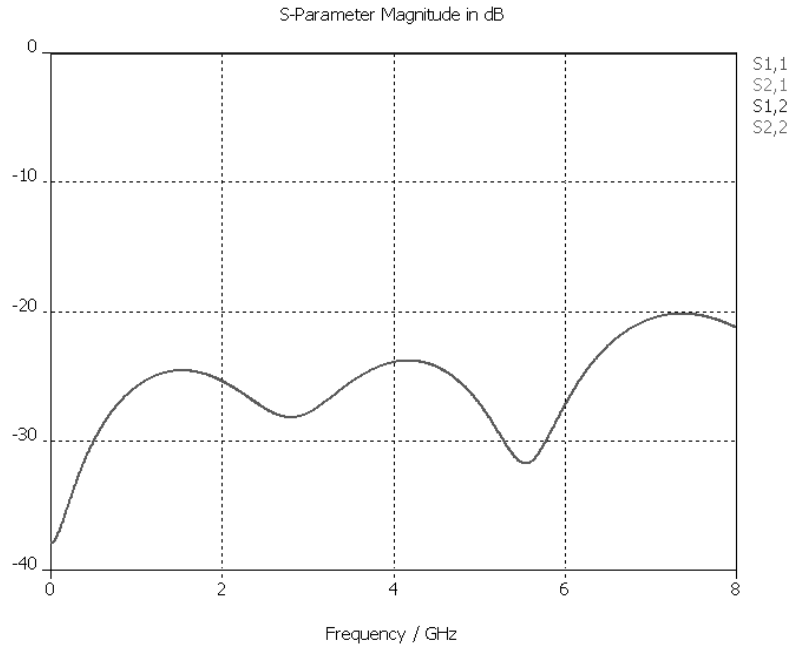
- 1D Results (Port Signals, S-Parameters)

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



This plot shows the incident, reflected and transmitted wave amplitudes at the ports versus time. The incident wave amplitude is called i1 and the reflected or transmitted wave amplitudes of the two ports are o1,1 and o2,1. These curves show the delay in the transition from the input port to the output port and a relatively small reflection at the input port.

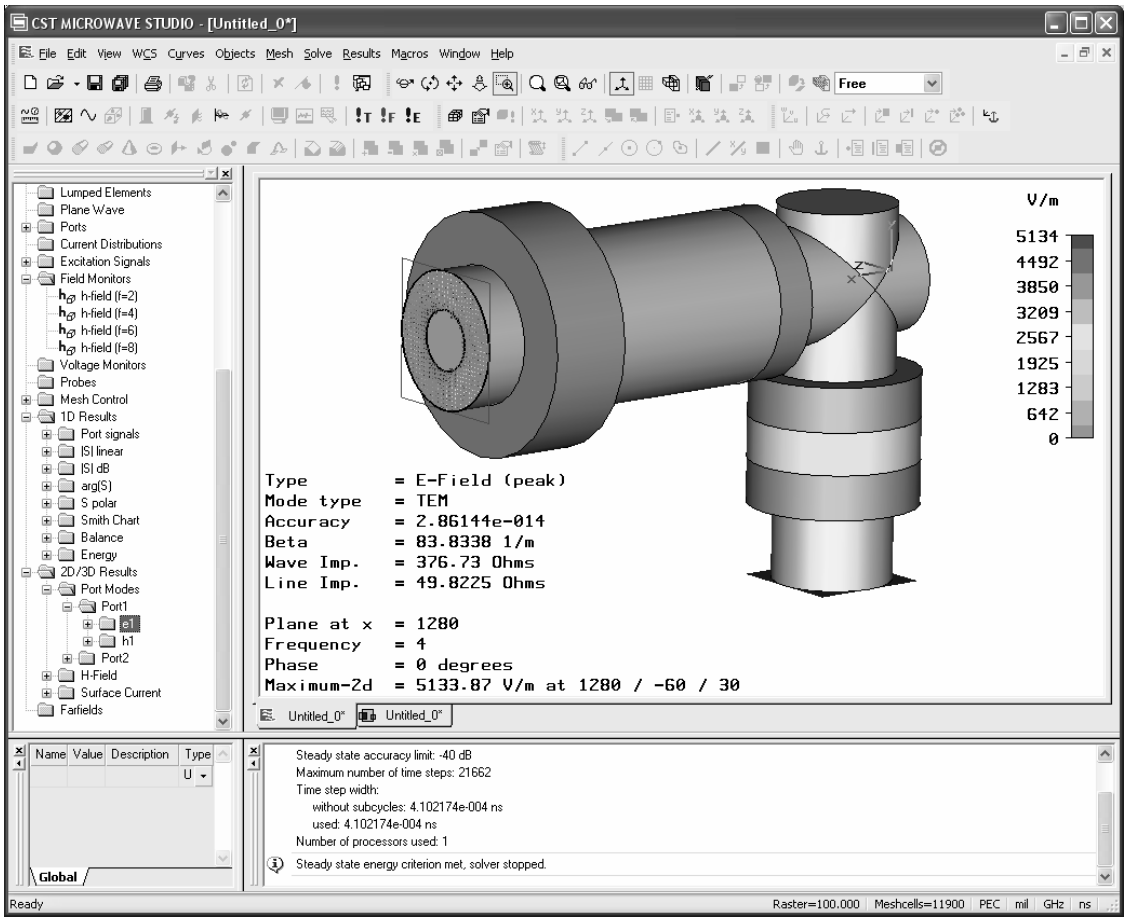
The S-Parameters magnitude in dB scale can be plotted by clicking on the *1D Results* ⇨ *S dB* folder.



As expected, the input reflection S1,1 is quite small across the entire frequency range.

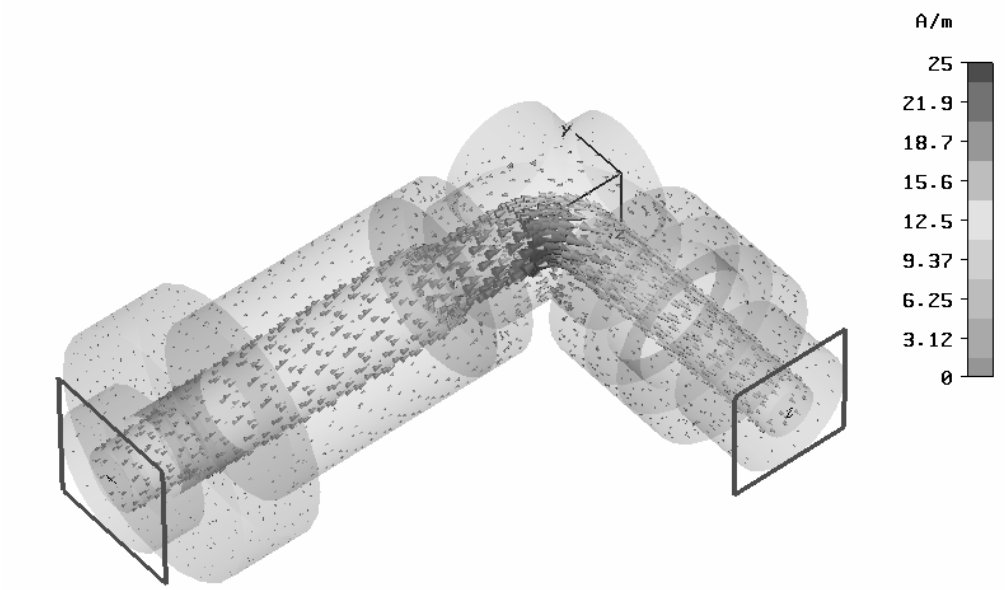
☐ 2D and 3D Results (Port Modes and Field Monitors)

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. 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 (please refer to the *Getting Started* manual for more information on how to change the plot's parameters):



The plot also displays some important properties of the mode, such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

The three-dimensional surface current distribution on the conductors can be shown by selecting one of the entries in the *2D/3D Results* ⇨ *Surface Current* folder from the navigation tree. The surface current at a frequency of 2 GHz can thus be visualized by clicking at the *2D/3D Results* ⇨ *Surface Current* ⇨ *h-field (f=2) [1]* entry (you may need to activate the transparent plotting option by selecting *Results* ⇨ *All Transparent,*)




```
Type        = Surface Current (peak)
Monitor     = h-field (f=2) [1]
Maximum-3d  = 24.9827 A/m at 242.17 / -64.1935 / 7.34764e-015
Frequency   = 2
Phase       = 225 degrees
```

You can toggle an animation of the currents on and off by selecting the *Results* ⇨ *Animate Fields* item. The surface currents for the other frequencies can be visualized in the same manner.

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.

In the following section, we provide hints on how to minimize these errors and achieve highly accurate results.

☐ 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 themselves 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 using the *Accuracy* setting in the transient solver control dialog box. The coaxial connector template has already set the default to –40dB because this is a good compromise between speed and accuracy for this type of device. However, you may set the accuracy to –50 dB or even –60 dB with relatively little increase in computation time.

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

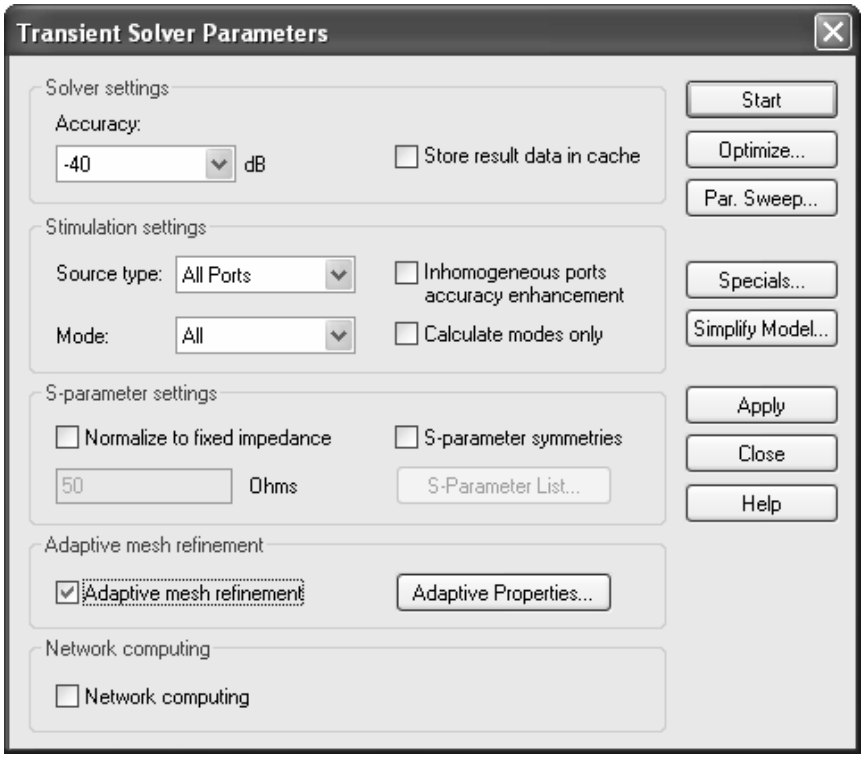
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

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

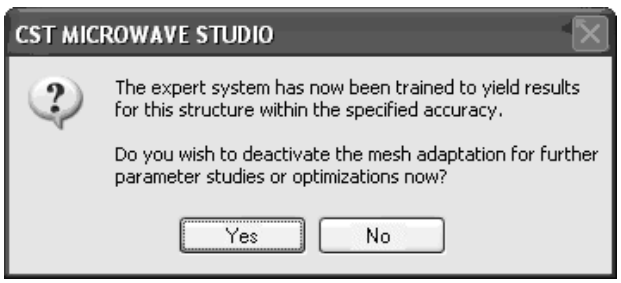
☐ Effect of the Mesh Resolution on the S-parameter's Accuracy

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. If these 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 the expert system. The easiest way to test the accuracy of the results is to use the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the "Transient Solver Parameters" dialog box (*Solve ⇨ Transient Solver):*



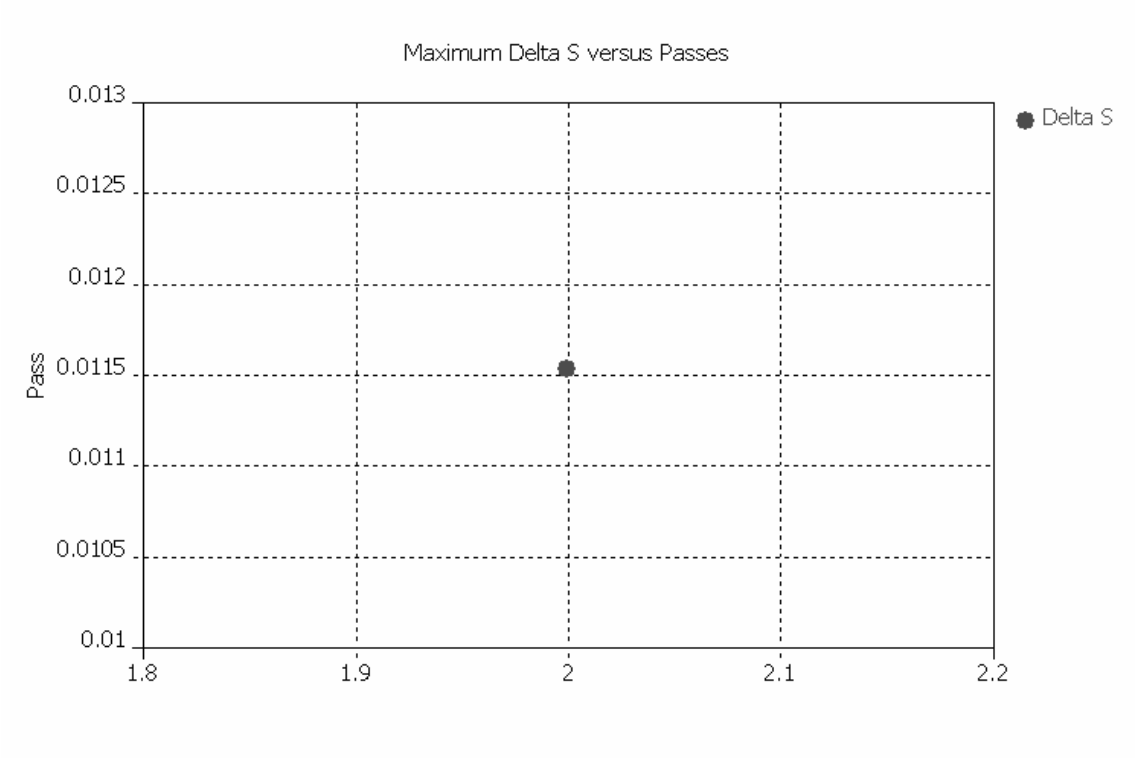
After activating the adaptive mesh refinement tool, you should now start the solver again by pressing the *Start* button. After a couple of minutes (during which the solver is running through mesh adaptation passes), the following dialog box will appear:



This dialog box informs you that the desired accuracy limit (2% by default) could be met by the adaptive mesh refinement. Because the expert system's settings have now been adjusted such that this accuracy is achieved, you may switch off the adaptation procedure for subsequent calculations (e.g. parameter sweeps or optimizations).

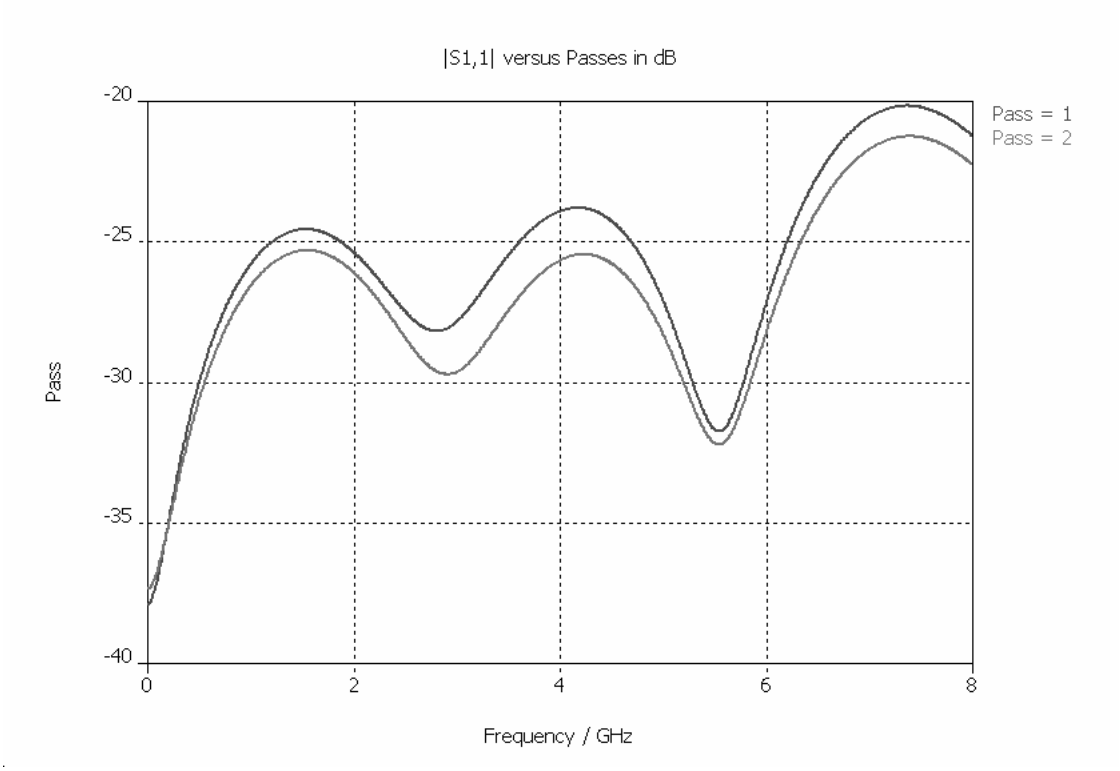
You should now confirm the deactivation of the mesh adaptation by pressing the *Yes* button.

After the mesh adaptation procedure is complete, you can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *Delta S* from the navigation tree:



As evidenced in the above plot, only two passes of the mesh refinement were required to obtain highly accurate results within the given accuracy level that is set to 2% by default.

The convergence process of the input reflection S1,1 during the mesh adaptation can be visualized by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *|S| dB* ⇨ *S1,1* from the navigation tree:



The convergence process of the other S-parameters magnitudes and phases can be visualized in the same manner.

The major advantage of this expert system based mesh refinement procedure over traditional adaptive schemes is that the mesh adaptation needs to be carried out only once for each device to determine the optimum settings for the expert system. There is then no need for time consuming mesh adaptation cycles during parameter sweeps or optimization.

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 grid type they are based on. The general purpose frequency domain solvers are available for hexahedral grids as well as for tetrahedral grids. The availability of a frequency domain solver within the same environment offers a very convenient way to cross-check results produced by the time domain solver with minimal additional effort.

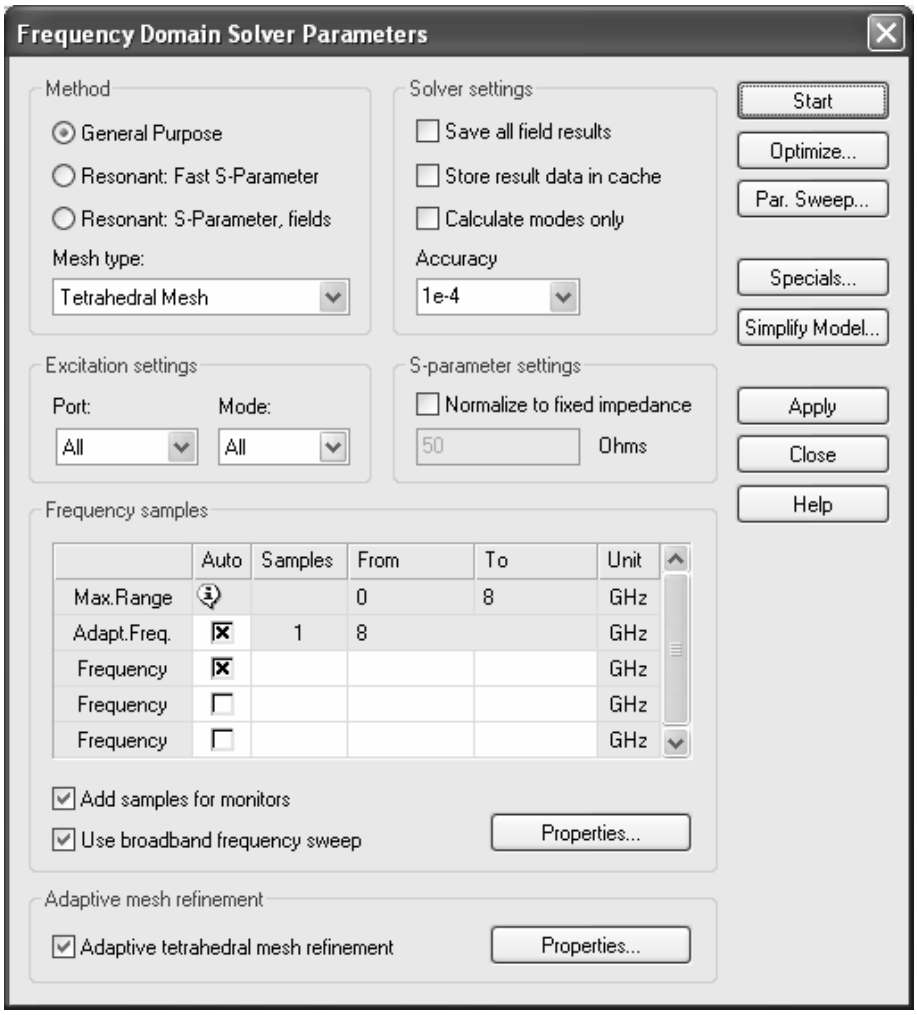
☐ Making a Copy of Transient Solver Results

Before performing a simulation with a frequency domain solver, you may want to keep the results of the transient solver in order to compare the two simulations. he copy of the current results is obtained as follows: Select, for example, the *|S| dB* folder in *1D Results*, then press *Ctrl+c* and *Ctrl+v.* The copies of the results will be created in the selected folder. The names of the copies will be *S1,1_1, S2,1_1* etc. You may rename them to *S1,1_TD*, *S2,1_TD* and so on using the *Rename* command from the context

menu. Use *Add 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.

☐ Frequency Domain Solver Settings

The "Frequency Domain Solver Parameters" dialog box is opened by selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu or by pressing the corresponding icon in the toolbar.



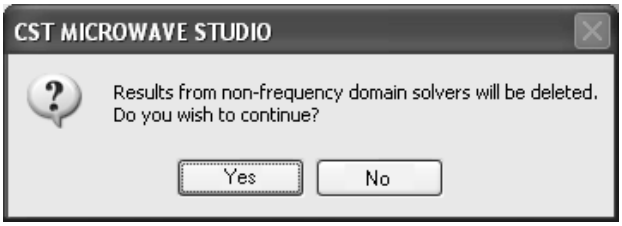
There are three different methods to choose from. For the example here, choose the *General Purpose* frequency domain solver. In the *Mesh Type* combo box you may choose between Hexahedral and Tetrahedral Mesh. Please choose *Tetrahedral Mesh.*

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 the 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 check box is activated by default.

All other settings may also be left unchanged. Press *Start* to begin the calculation.

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



Press *Yes* to acknowledge the deletion. A progress bar and an abort button appear in the status bar showing information about the solver stages.

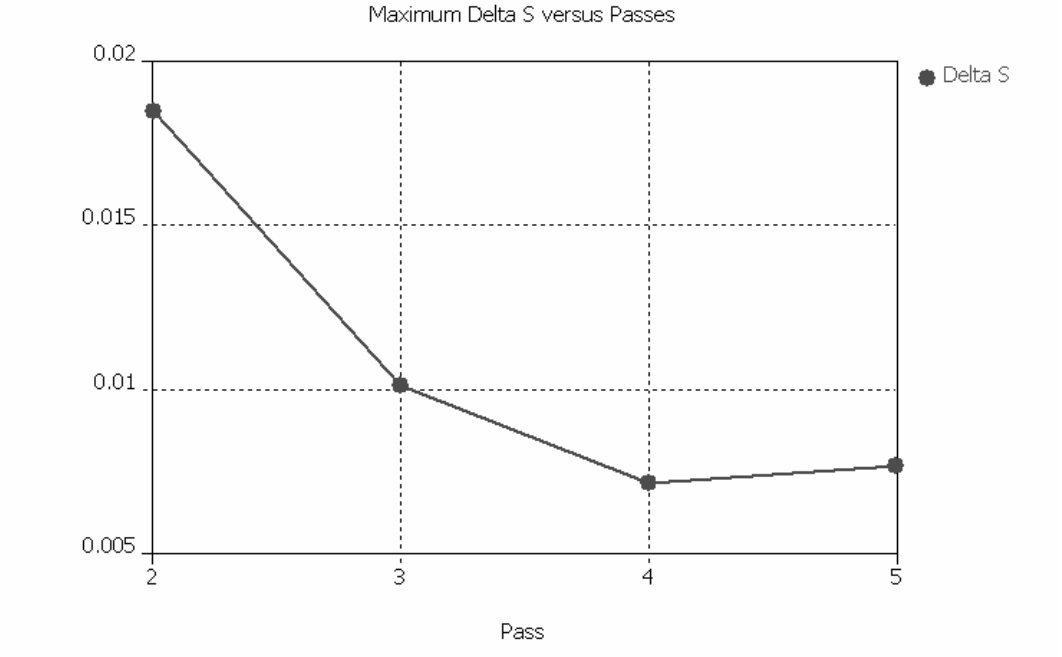
After the desired accuracy for the S-Parameter has been reached, the simulation stops. When the simulation has finished or if it has been aborted, both items disappear again. During the simulation, the "Message Window" will show some details about the performed simulation.

Results of the Frequency Domain Solver

Congratulations, you have simulated the coaxial connector using the frequency domain solver! Let's review the results:

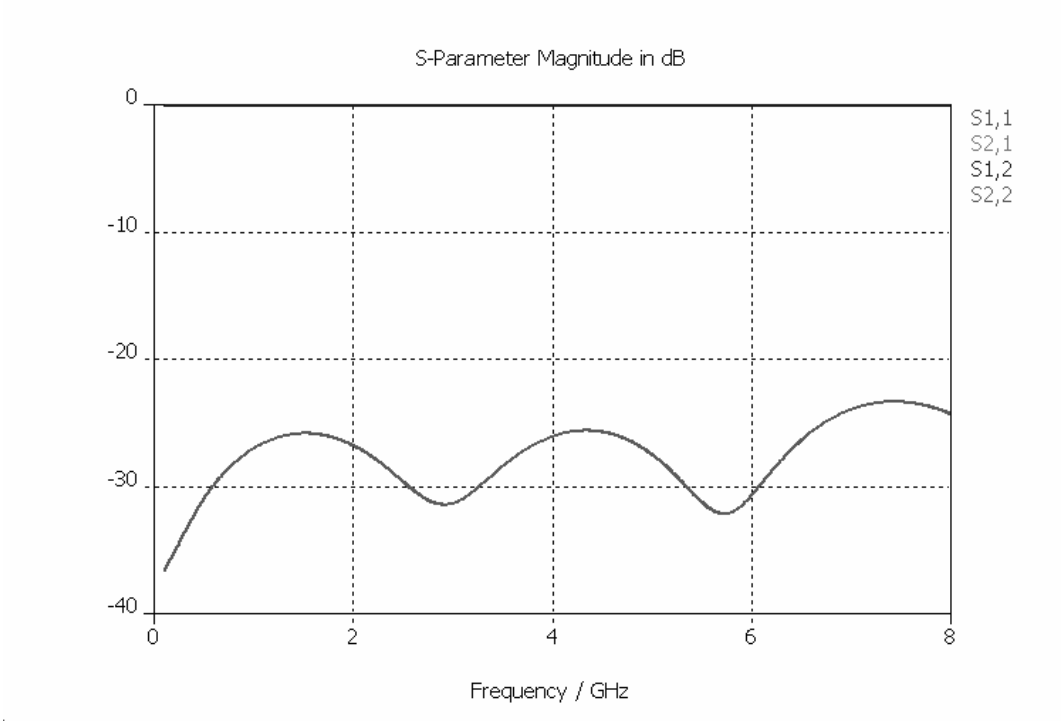
- 1D Results (S-Parameters)

You can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *Delta S* from the navigation tree:



As evident in the above plot, four passes of the mesh refinement were required to obtain highly accurate results within the given accuracy level, which is set to 1% by default.

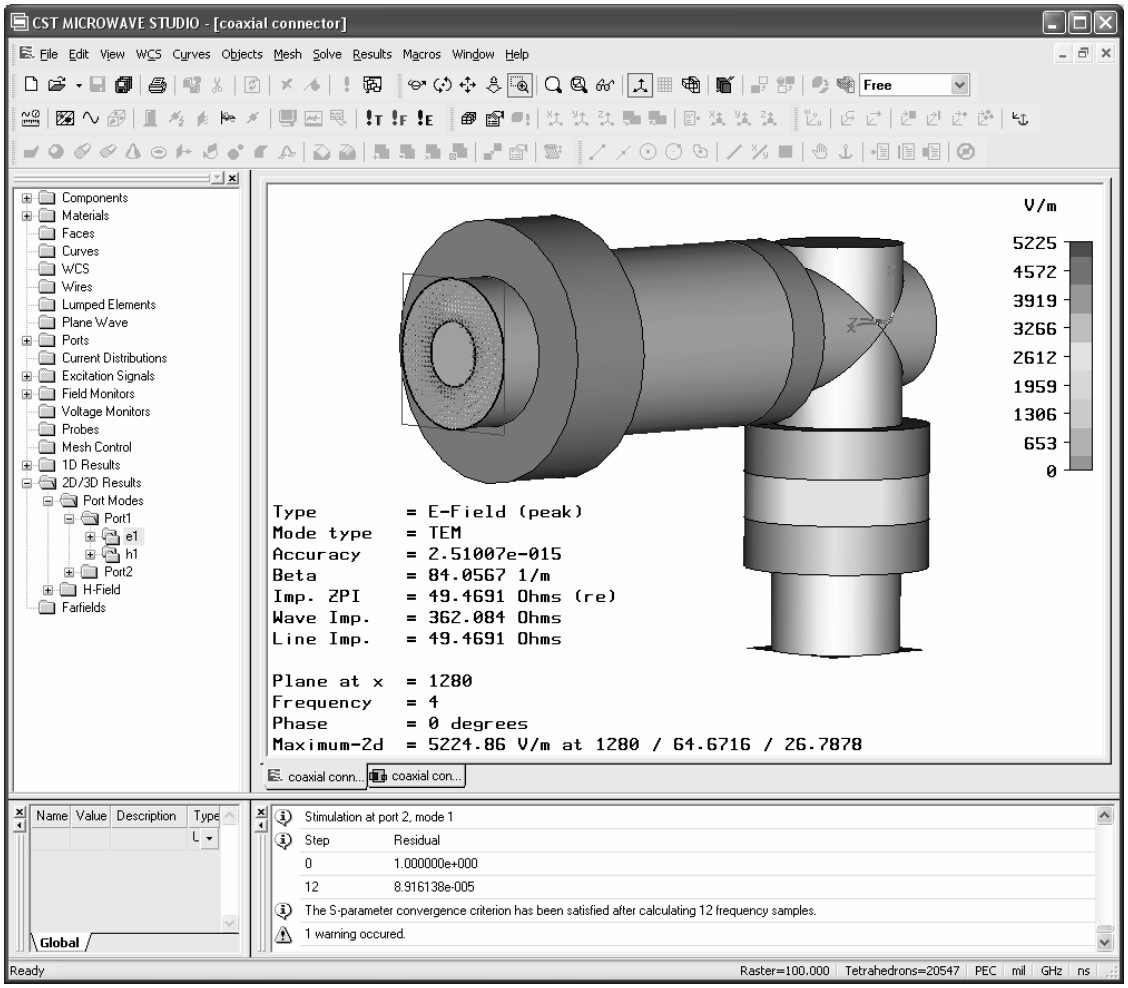
You can view the S-parameters magnitude in dB scale by selecting *1D Results* ⇨ |*S*| *dB* in the navigation tree.



As expected, the input reflection S1,1 is quite small across the entire frequency range.

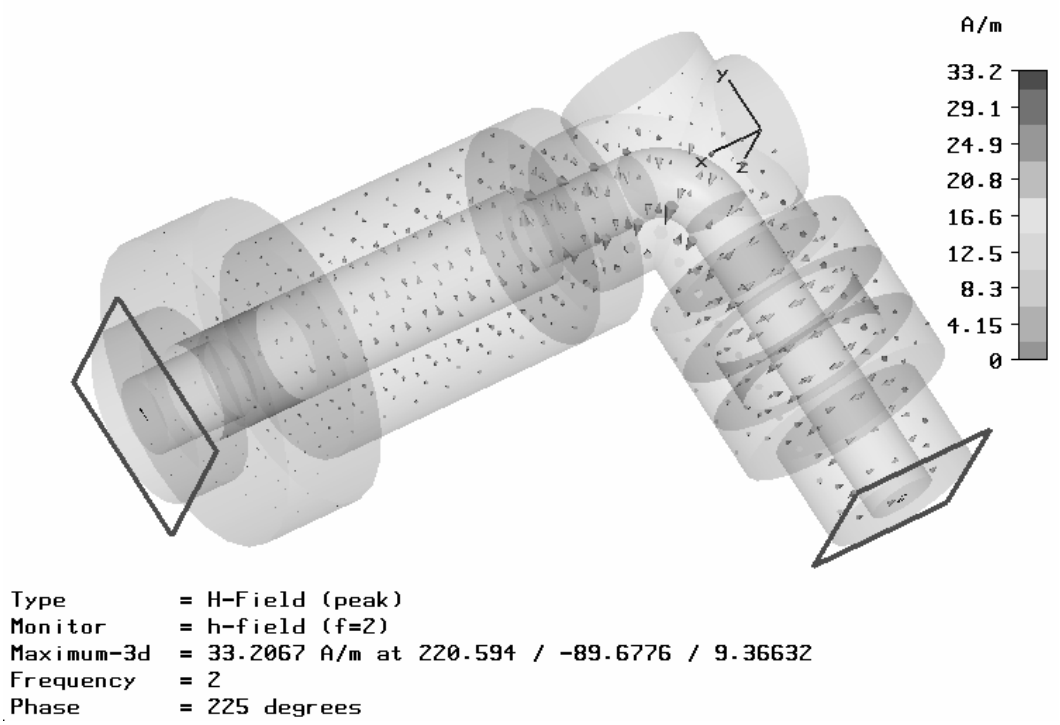
☐ 2D and 3D Results (Port Modes and Field Monitors)

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 selecting *Results ⇨ Select Mode Frequency* from the main menu and change the frequency to 4 GHz. Please confirm your setting by pressing *OK*. After properly rotating the view, you should obtain a plot similar to the following picture (please refer to the *Getting Started* manual for more information on how to change the plot's parameters):



The plot also shows some important properties of the mode, such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

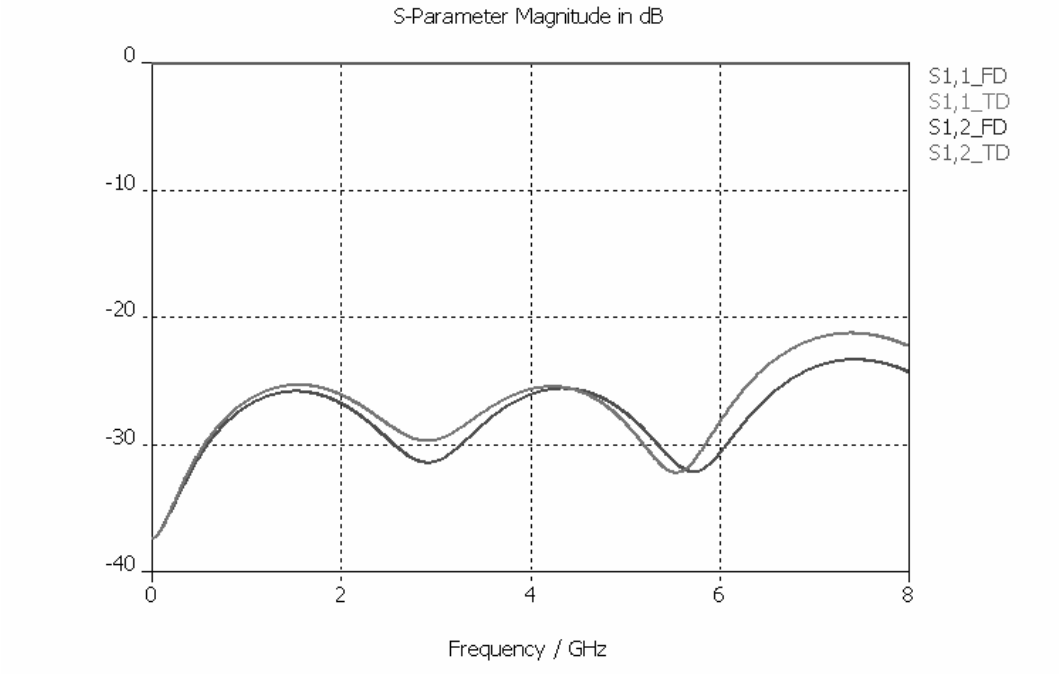
The three-dimensional h-field distribution can be displayed by selecting one of the entries in the *2D/3D Results ⇨ H-Field* folder from the navigation tree. The magnetic field at a frequency of 2 GHz can thus be visualized by clicking at the *2D/3D Results ⇨ H-Field ⇨ h-field (f=2) [1]* entry (you may need to activate the transparent plotting option by selecting *Results ⇨ All Transparent,*)



You can toggle an animation of the currents on and off by selecting *Results* ⇨ *Animate Fields*. The surface currents for the other frequencies can be visualized in the same manner, as shown above.

Comparison of the Solver Results

The next plot shows the S-parameters S1,1 and S1,2 resulting from the time domain and frequency domain simulations. Plotting the S-parameter curves in the same graph allows for a better comparison.



As you can see, the results agree very well. Since the results are not converged to the highest possible accuracy level, there is still a slight difference between the curves. When refining the accuracy limit in the adaptive mesh refinement the difference will become negligible.

The following table shows the number of cells and the simulation time when using the solver with the default settings for the adaptive mesh refinement:

	Transient Solver	Frequency Domain Solver
Mesh	30420 hexahedral cells	20547 tetrahedrons
Simulation time	39 seconds	149 seconds

Getting More Information

Congratulations! You have just completed the Coaxial Structure tutorial that should have provided you with a good working knowledge on how to use CST MICROWAVE STUDIO® to calculate S-parameters. The following topics have been covered:

1. General modeling considerations, using templates, etc.
2. Model a coaxial structure using the rotate, cylinder and extrude tools and define the substrates.
3. Define ports.
4. Define frequency range, boundary conditions and symmetry planes.
5. Define field monitors for surface current distributions.
6. Start the transient or the frequency domain solver.
7. Visualize port signals and S-parameters.
8. Visualize port modes and surface currents.
9. Check the truncation error of the time signals
10. Obtain accurate and converged results using the automatic expert system based mesh adaptation.

You can obtain more information for each particular step from the online help system that can be activated either by pressing the *Help* button in each dialog box or by pressing the *F1* key at any time to obtain context sensitive information.

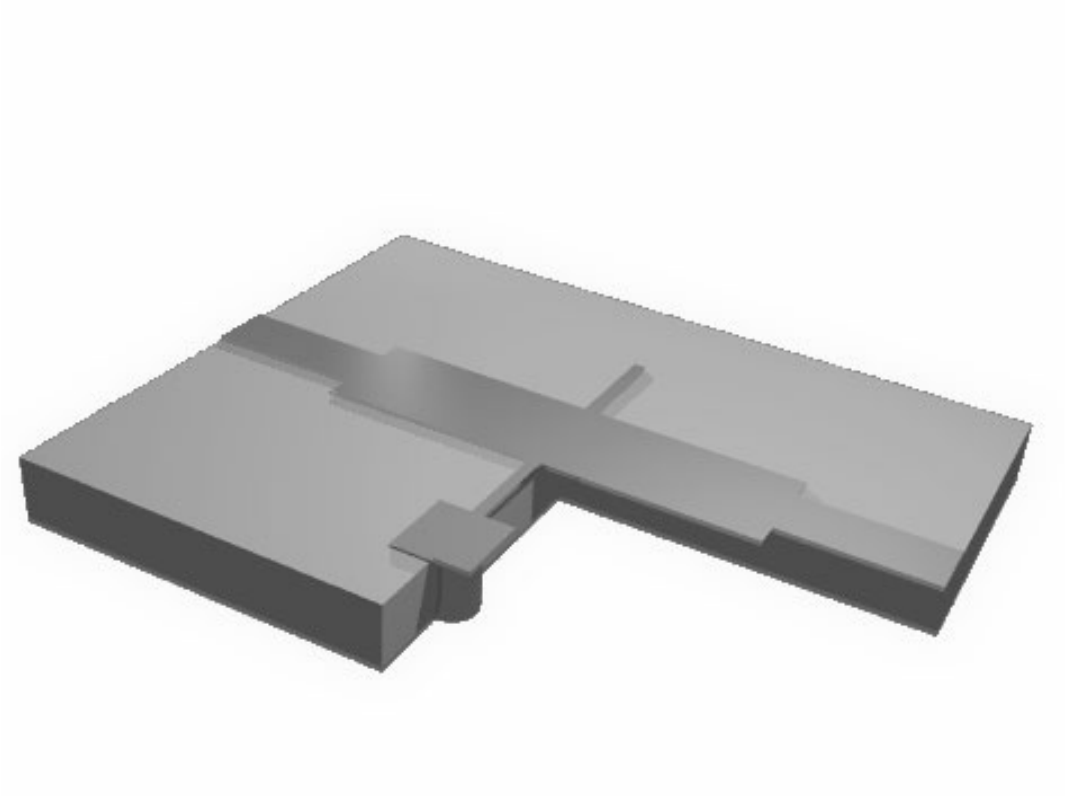
In some cases we have referred to the *Getting Started* manual that is also a good source of information for general topics.

In addition to this tutorial, you can find some more S-parameter calculations in the “examples” folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that will give you some more information about the particular device.

Finally, you should refer to the *Advanced Topics* manual for more in depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please also visit one of the training classes held regularly at a location near you. Thank you for using CST MICROWAVE STUDIO®!

Planar Device Tutorial



Geometric Construction and Solver Settings **78**

Introduction and Model Dimensions 78
Geometric Construction Steps 79

S-Parameter and Field Calculation **98**

Transient Solver 98
Accuracy Considerations 103
Frequency Domain Solver 106
Accuracy Considerations 112
Comparison of the Solver Results 113

Getting More Information **114**

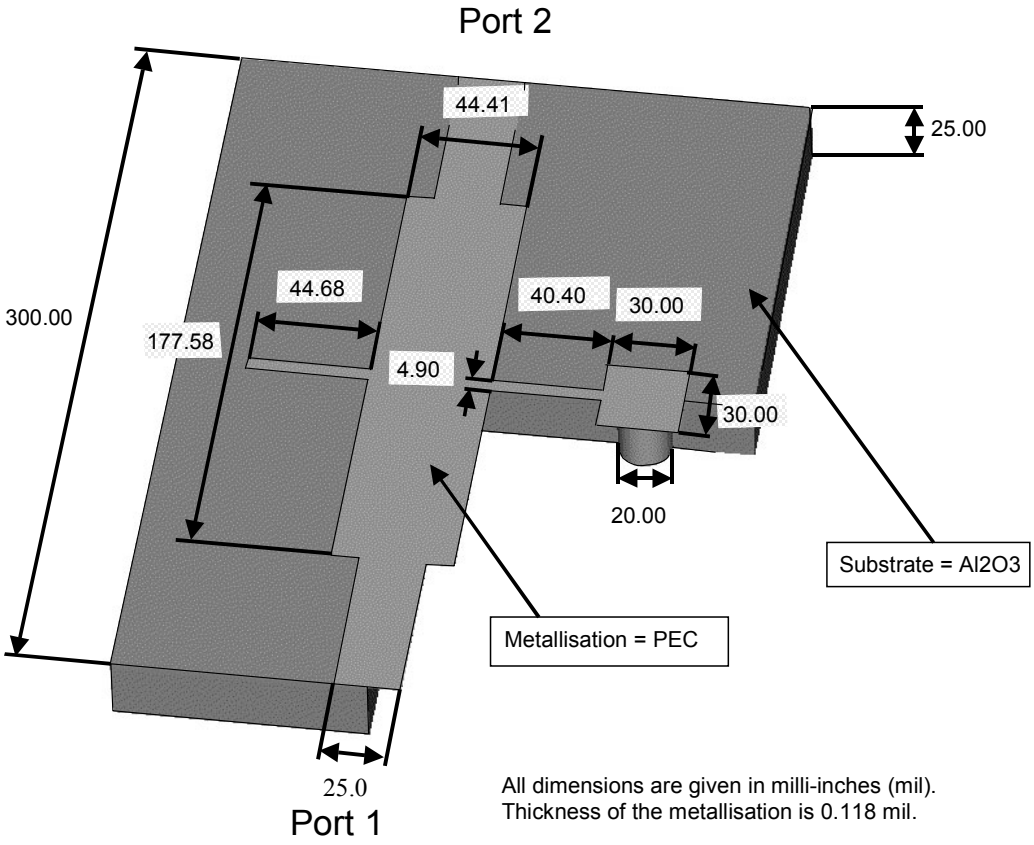
Geometric Construction and Solver Settings

Introduction and Model Dimensions

In this tutorial you will learn how to simulate planar devices. As a typical example for a planar device, you will analyze a Microstrip Phase Bridge. The following explanations on how to model and analyze this device can be applied to other planar devices, as well.

CST MICROWAVE STUDIO® can provide a wide variety of results. This tutorial, however, concentrates solely on the S-parameters and surface currents.

We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.



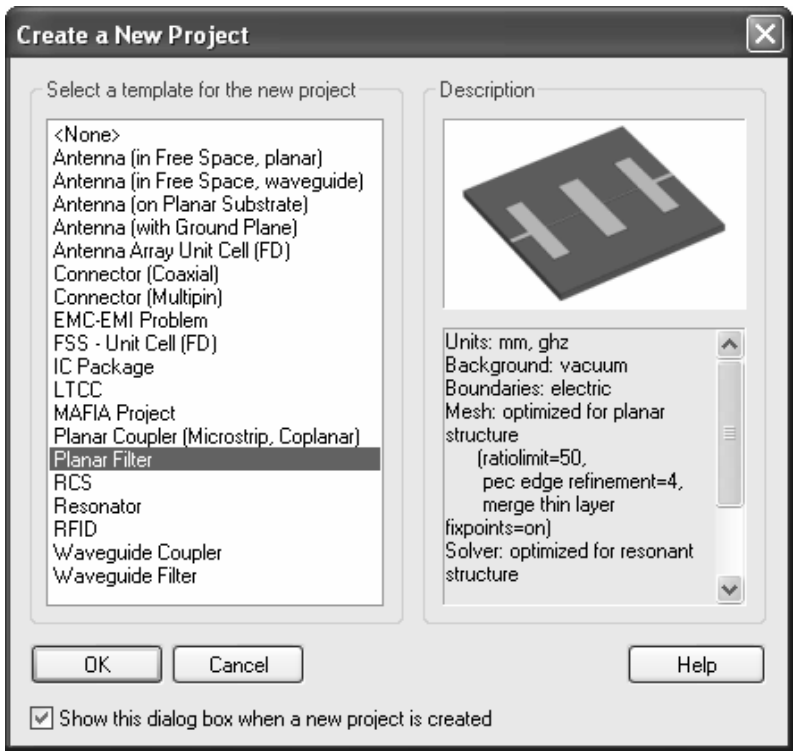
The structure depicted above consists of two different materials: The aluminum oxide substrate (Al_2O_3) and the stripline metallization. There is no need to model the ground plane since it can easily be described using a perfect electric boundary condition.

Geometric Construction Steps

This tutorial 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.

☐ Select a Template

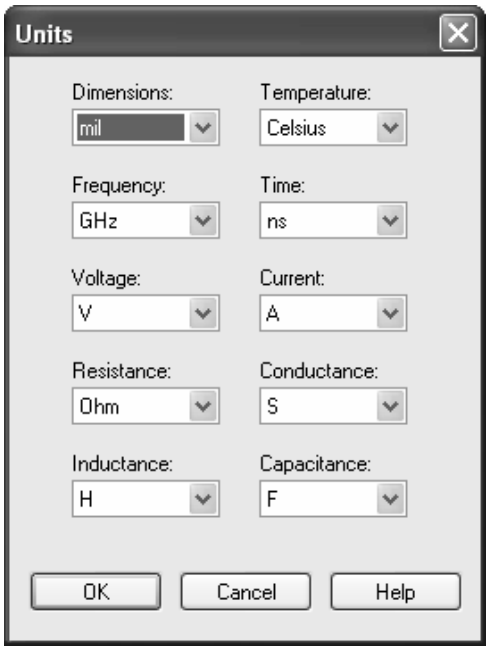
Once you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the "Planar Filter" template should be selected.



This template automatically sets the units to mm and GHz, the background material to vacuum and all boundaries to perfect electrical conductors. Because the background material has been set to vacuum, the structure can be modeled just as it appears. Furthermore, the automatic mesh strategy is optimized for planar structures and the solver settings are adjusted to resonant behavior.

☐ Set the Units

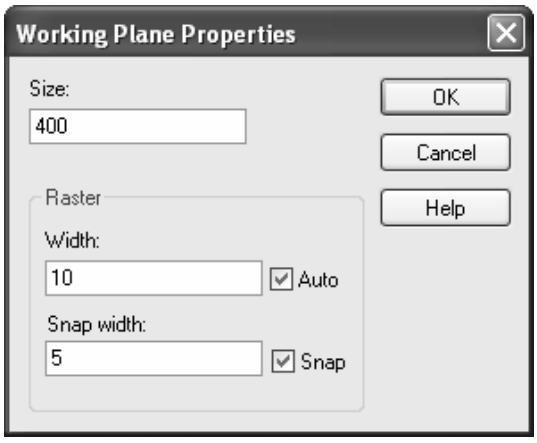
As mentioned, the template has automatically set the geometrical units to mm. However, since all geometrical dimensions are given in mil for this example, you should change this setting manually. Therefore, open the units dialog box by selecting *Solve* ⇨ *Units* from the main menu:



Here, you should set the *Dimensions* to *mil* and press *OK*.

☐ Set the Working Plane's Properties

The next step is usually to set the working plane properties to make the drawing plane large enough for your device. Because the structure has a maximum extension of 300 mil along a coordinate direction, the working plane size should be set to at least 400 mil. These settings can be changed in a dialog box that opens after selecting *Edit* ⇨ *Working Plane Properties* from the main menu. Please note that we will use the same document conventions here as introduced in the *Getting Started* manual.



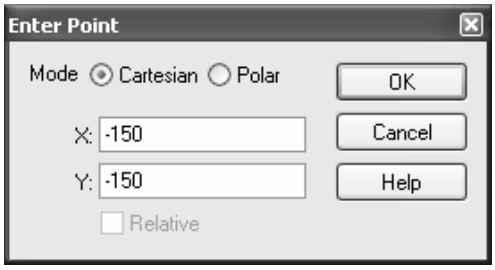
In this dialog box, you should set the *Size* to 400 (the unit that has previously been set to mil is displayed in the status bar), the *Raster width* to 10 and the *Snap width* to 5 to

obtain a reasonably spaced grid. Please confirm these settings by pressing the *OK* button.

☐ Draw the Substrate Brick

The first construction step for modeling 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 (*Objects ⇨ Basic Shapes ⇨ Brick,*).

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

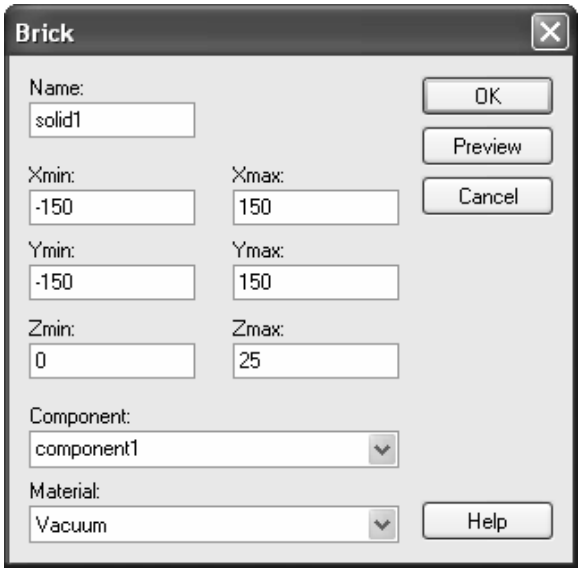


In this example, you should enter a substrate block that has an extension of 300 mil in each of the transversal directions. The transversal coordinates can thus be described by X = -150, Y = -150 for the first corner and X = 150, Y = 150 for the opposite corner, assuming that the brick is modeled symmetrically to the origin. Please enter the first point's coordinates *X* = -150 and *Y* = -150 in the dialog box and press the *OK* button.

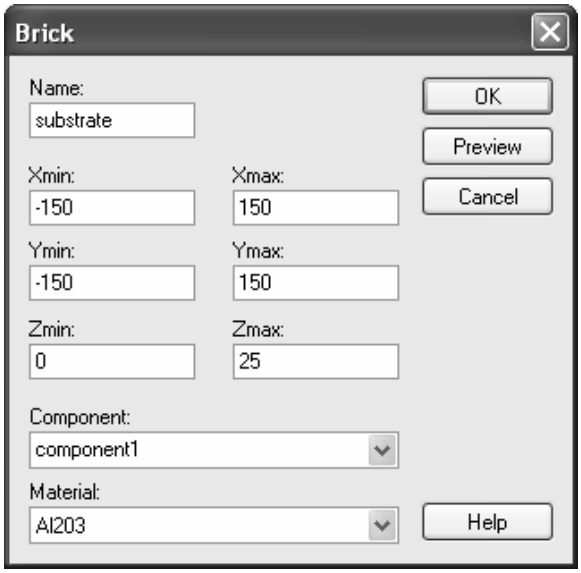
You can repeat these steps for the second point:

1. Press the *Tab* key
2. Enter *X* = 150, *Y* = 150 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 25 and pressing the *OK* button. Now the following dialog box will appear showing you a summary of your previous input:



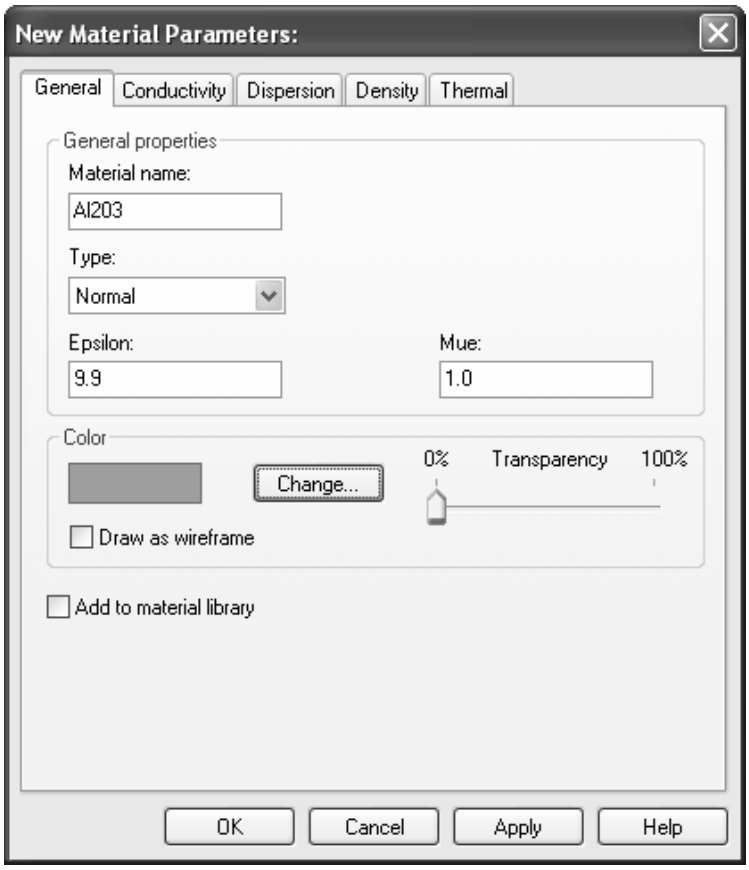
Please check all these settings carefully. If you encounter any mistake, please change the value in the corresponding entry field.



You should now assign a meaningful name to the brick by entering e.g. “substrate” in the *Name* field. Since the brick is the first object you have modeled thus far, you can keep the default settings for the first *Component* (“component1”).

> **Please note:** The use of different components allows you to combine several solids into specific groups, independent of their material behavior. However, in this tutorial it is convenient to construct the complete microstrip device as a representation of one component.

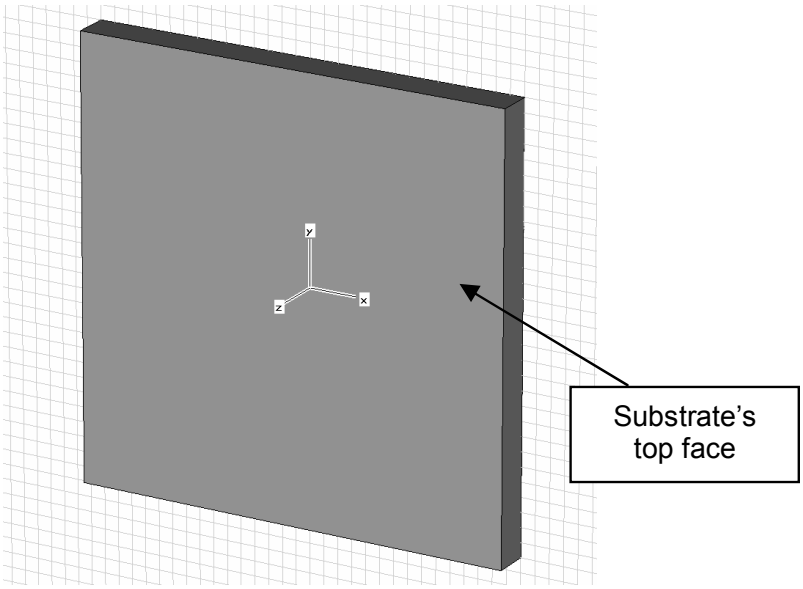
The *Material* setting of the brick must be changed to the desired substrate material. Because no material has yet been defined for the substrate, you should open the layer definition dialog box by selecting “[New Material…]” from the *Material* dropdown list:



In this dialog box you should define a new *Material name* (e.g. Al2O3) and set the *Type* to a *Normal* dielectric material. Afterwards, specify the material properties in the *Epsilon* and *Mue* fields. Here, you only need to change the dielectric constant *Epsilon* to 9.9. Finally, choose a color for the material by pressing the *Change* button. Your dialog box should now look similar to the picture above before you press the *OK* button.

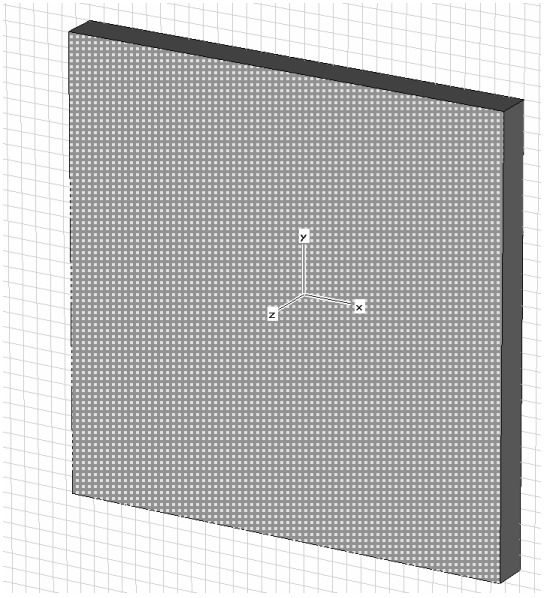
> **Please note:** The defined material "Al2O3" will now be available inside the current project for the creation of other solids. However, if you also want to save this specific material definition for other projects, you may check the button *Add to material library*. You will have access to this material database by clicking on *Load from Material Library* in the *Materials* context menu in the navigation tree.

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* key in order to zoom the structure to the maximum possible extent):

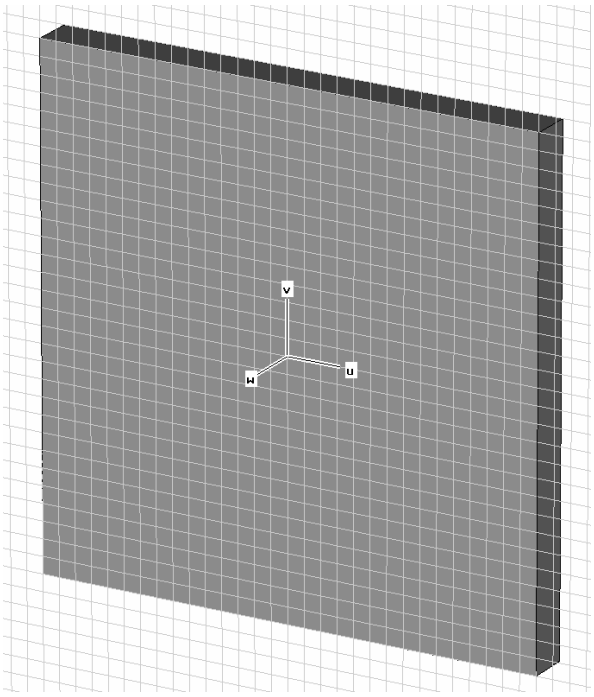


☐ Model the Stripline Metallization

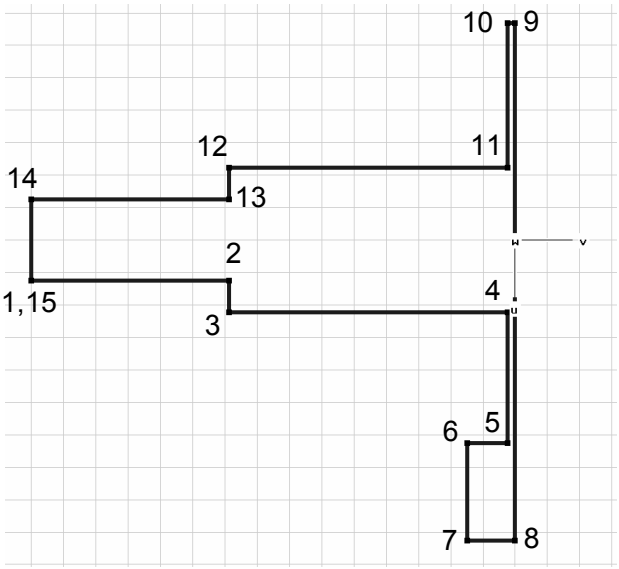
The next step is to model the stripline metallization on top of the substrate. Therefore, you should first move the drawing plane on top of the substrate. This can be easily achieved by activating the face pick tool (*Objects ⇨ Pick ⇨ Pick Face*, *f*) and double-clicking on the substrate's top face as shown above. The face selection should then be visualized as in the following picture:



After the face has been selected, you can align the working coordinate system with its plane. Therefore, please either select *WCS ⇨ Align WCS with Selected Face* from the main menu, press the toolbar button , or simply use the shortcut *w*. Now the drawing plane will be aligned with the top of the substrate (you may switch off the visualization of the global coordinate axes by pressing *Ctrl*+*a*):



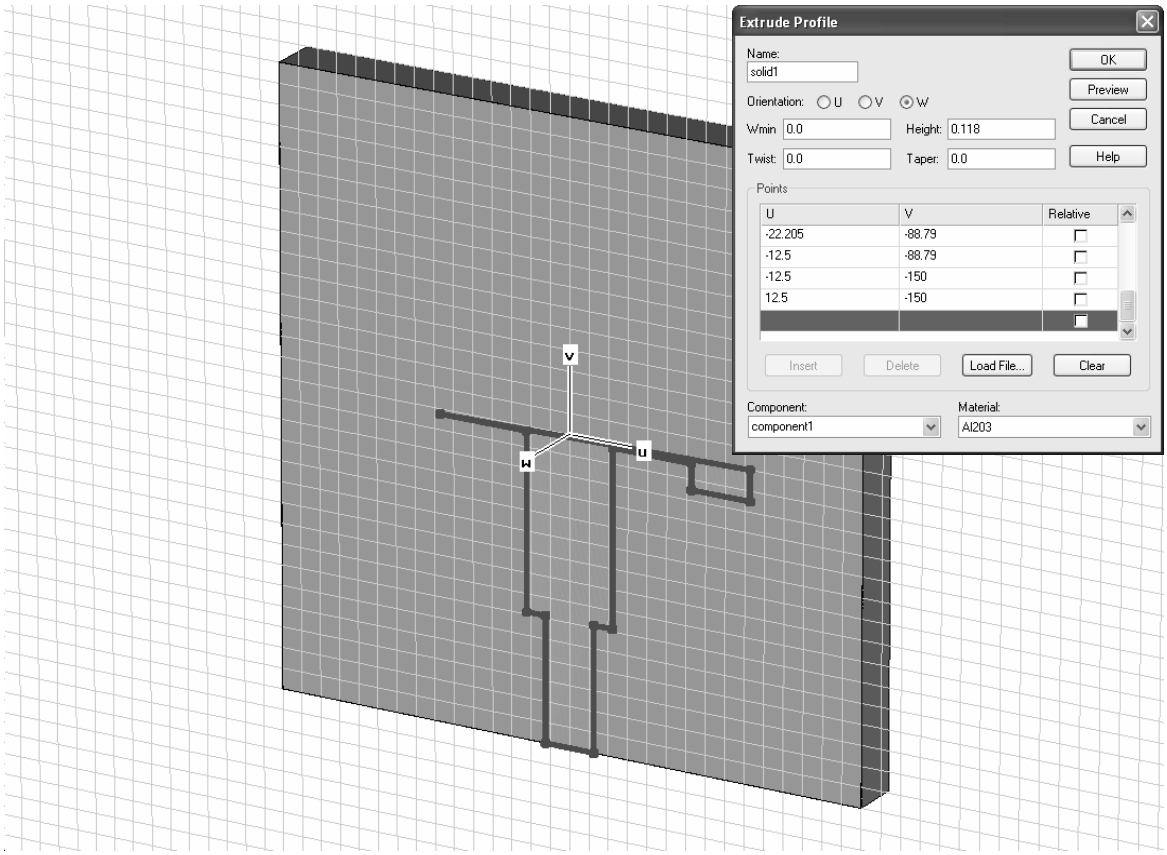
The easiest way to draw the metallization is to use a polygonal extrusion. This tool can be entered by selecting *Objects* ⇨ *Extrude* or by pressing the toolbar button . After the polygonal extrude mode is active, you are requested to enter the polygon's points. For each of these points you should press the *Tab* key and enter the point coordinates manually according to the following table (you may either enter the expressions or the absolute values given in brackets):



Point	U coordinate	V coordinate
1	25 / 2 (=12.5)	-150
2	25 / 2 (=12.5)	-177.58 / 2 (= -88.79)
3	44.41 / 2 (=22.205)	-177.58 / 2 (= -88.79)
4	44.41 / 2 (=22.205)	-4.9 / 2 (= -2.45)
5	44.41 / 2 + 40.4 (=62.605)	-4.9 / 2 (= -2.45)
6	44.41 / 2 + 40.4 (=62.605)	-30 / 2 (= -15)
7	44.41 / 2 + 40.4 + 30 (=92.605)	-30 / 2 (= -15)

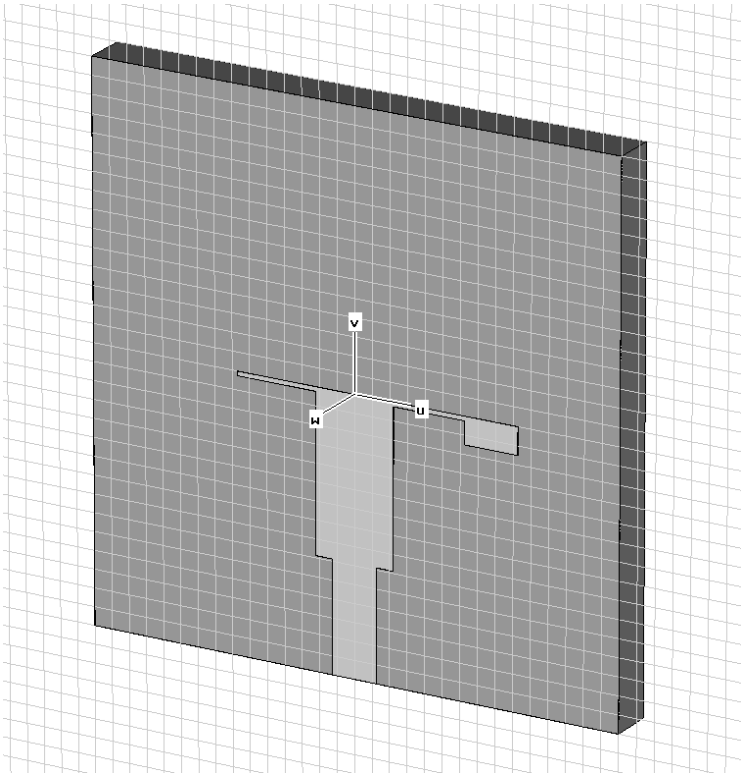
8	44.41 / 2 + 40.4 + 30 (=92.605)	0
9	-44.41 / 2 – 44.68 (= -66.885)	0
10	-44.41 / 2 – 44.68 (= -66.885)	-4.9 / 2 (= -2.45)
11	-44.41 / 2 (= -22.205)	-4.9 / 2 (= -2.45)
12	-44.41 / 2 (= -22.205)	-177.58 / 2 (= -88.79)
13	-25 / 2 (= -12.5)	-177.58 / 2 (= -88.79)
14	-25 / 2 (= -12.5)	-150
15	25 / 2 (=12.5)	-150

Please note that we do not recommend entering the points relative to each other here because this would make the detection of mistakes during the coordinate input more difficult. After you have entered the last point from the table that closes the polygon, the extrusion tool requests you to enter the height. Please press the *Tab* key again and enter the *Height* as 0.118. Afterwards, your screen should look similar to:

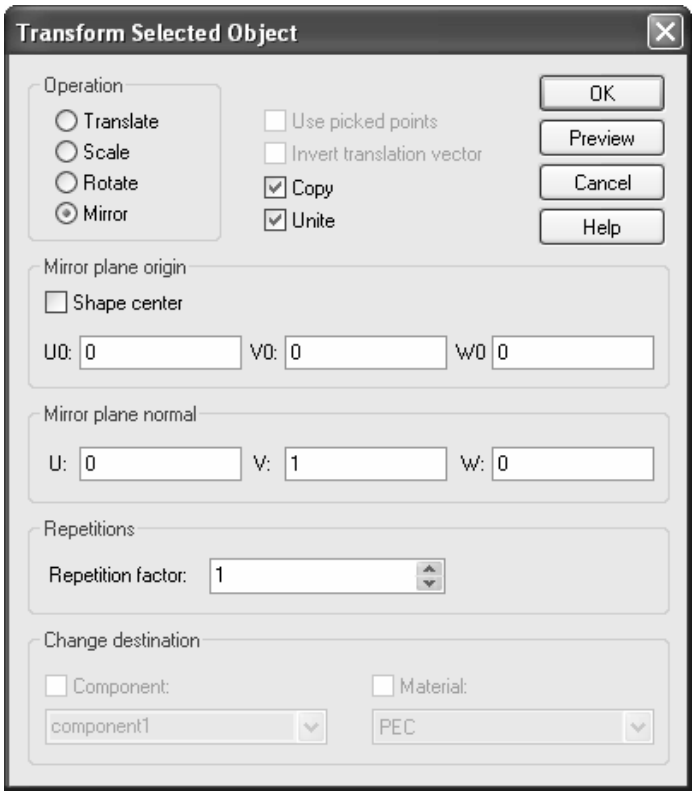


If your polygon does not look like the one in the picture above, please double-check your input in the dialog's point list. Afterwards, please assign a *Name* to the solid (e.g. "stripline") and change the *Material* assignment to be a perfect electrical conductor (PEC).

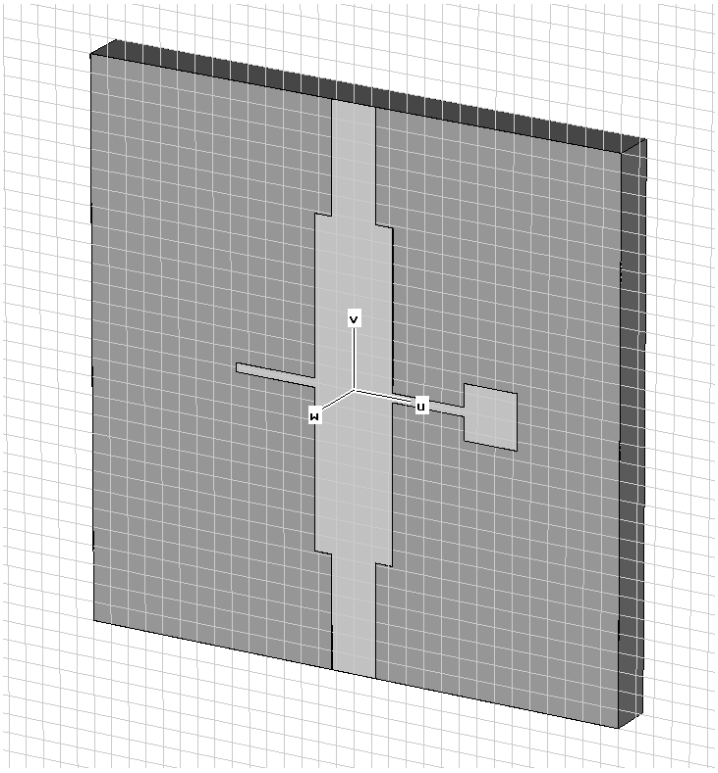
After finally pressing the *OK* button, the structure should look as follows:



To this point, you have modeled half of the stripline structure. The other half can be created by mirroring the structure at the UW plane of the working coordinate system. Please select the stripline by double-clicking on it (the substrate will then become transparent) and then open the transform dialog box (*Objects* ⇨ *Transform,*) :



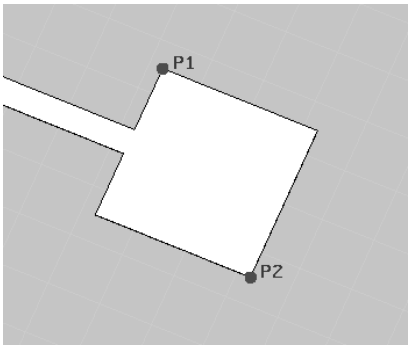
In this dialog box you should change the *Operation* to *Mirror* before you set the *V*-coordinate of the *Mirror plane normal* to 1. Afterwards, please switch on the *Copy* as well as the *Unite* option to copy the existing shape before mirroring it and to unite the original shape with the mirrored copy. You may check your settings using the *Preview* button. Finally, press *OK* to create the full stripline. Your model should then look as follows:



☐ Model the Via

After successfully modeling the stripline structure, the next step is to model the via that should be located in the center of the square pad. The alignment between these two geometric elements can be specified by moving the working coordinate system to the center of the pad.

Please activate the pick point tool (*Objects ⇨ Pick ⇨ Pick Point*,) and double-click on one of the corners of the pad. Repeat the same steps in order to pick the point from the opposite corner, as well. The picture below shows an example of how your structure should now appear:

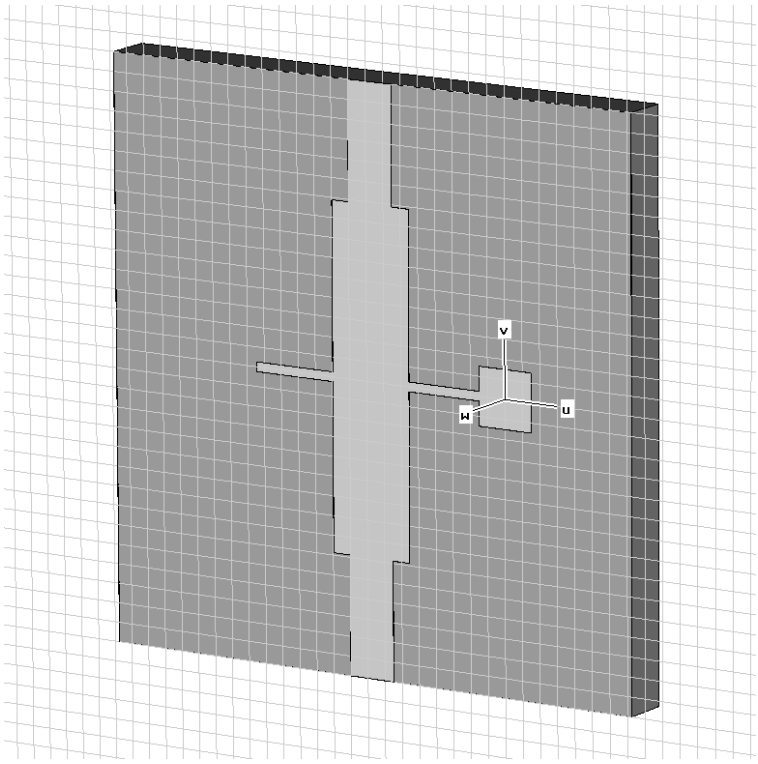


Please note: Due to the small thickness it may be difficult to pick the appropriate points on the drawing plane because they are very close to the points from the metallization's top face. In order to simplify this task, CST MICROWAVE STUDIO® offers a "Snap points to drawing plane" feature (*Objects ⇨ Pick ⇨ Snap Points to Drawing Plane*) that is activated by default. This feature will automatically snap all picked points onto the workplane if they are indistinguishably close to it in the current view. Zooming into the structure would allow you to pick points from the metallization's top face as soon as they are clearly distinguishable from the points belonging to the bottom face.

In this example, we make use of this feature to ensure that the picked points are located on the drawing plane.

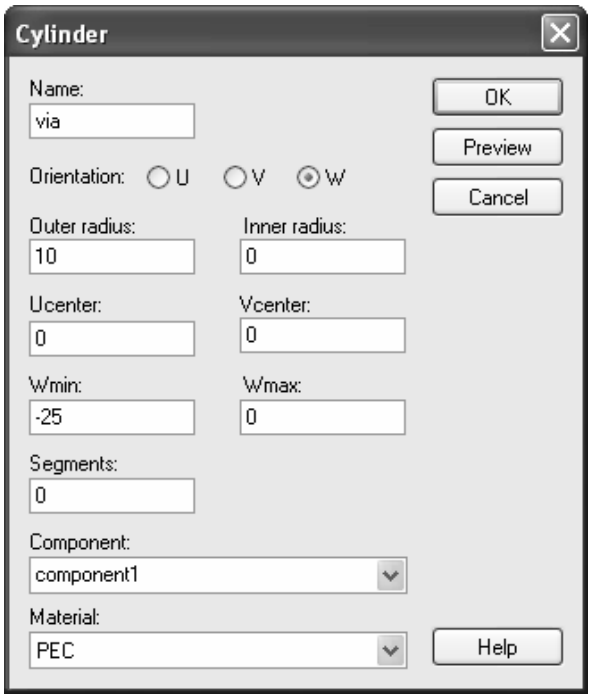
If you have made a mistake, please clear all picked points (*Objects ⇨ Clear Picks*) and try again. The next step is to replace both points by a point in the middle of both. This can be easily achieved by invoking the command *Objects ⇨ Pick ⇨ Mean Last Two Points*. Now a single point should be selected in the middle of the pad.

Now the working coordinate system can be aligned with this point by selecting *WCS ⇨ Align WCS with Selected Point* or by pressing the toolbar button . Your structure should then look like the following picture:

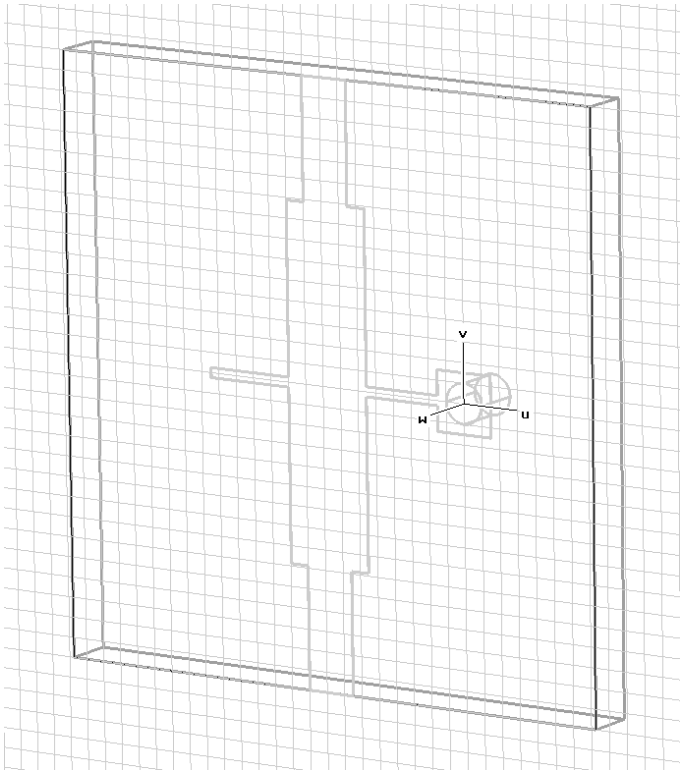


The via can now be created using the cylinder tool: *Objects ⇨ Basic Shapes ⇨ Cylinder,* . Once the cylinder creation mode is active you are requested to pick the center of the cylinder. Because this is now the origin of the working coordinate system, you can simply press *Shift+Tab* to open the dialog box for numerically entering the coordinates and confirm the settings by pressing *OK* (please note that holding down the *Shift* key while pressing the *Tab* key opens the dialog box with the coordinate values initially set to zero rather than to the current mouse pointer's location).

You are now requested to enter the outer radius of the via. Please press the *Tab* key again and set the *Radius* to 10 before pressing the *OK* button. The *Height* of the cylinder can then be set to –25 in the same manner. Please skip the definition of the inner radius by pressing the *Esc* key (the via should be modeled as solid cylinder here) and check your settings in the following dialog box:



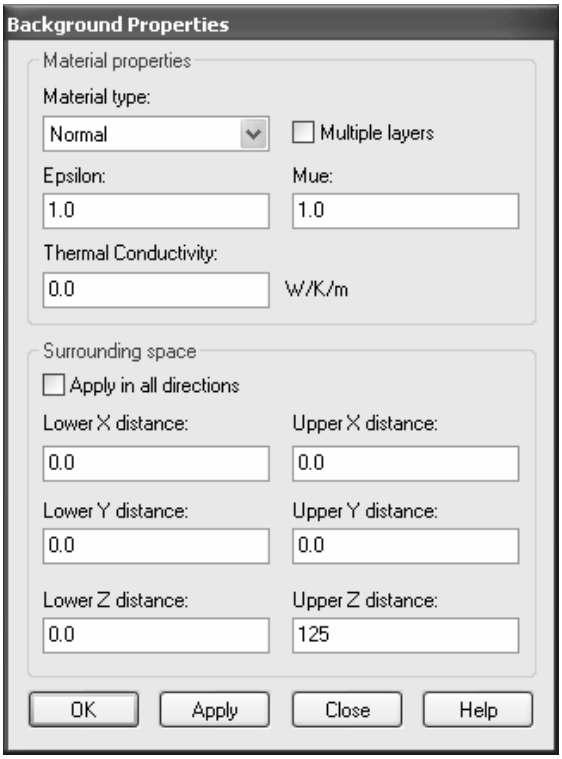
Finally set the *Name* of the cylinder to "via" and the *Material* assignment to "PEC" and press the *OK* button. The model should then finally look as follows (please use *Ctrl+w* to toggle the wireframe visualization mode on and off):



- Add Space on Top of the Stripline

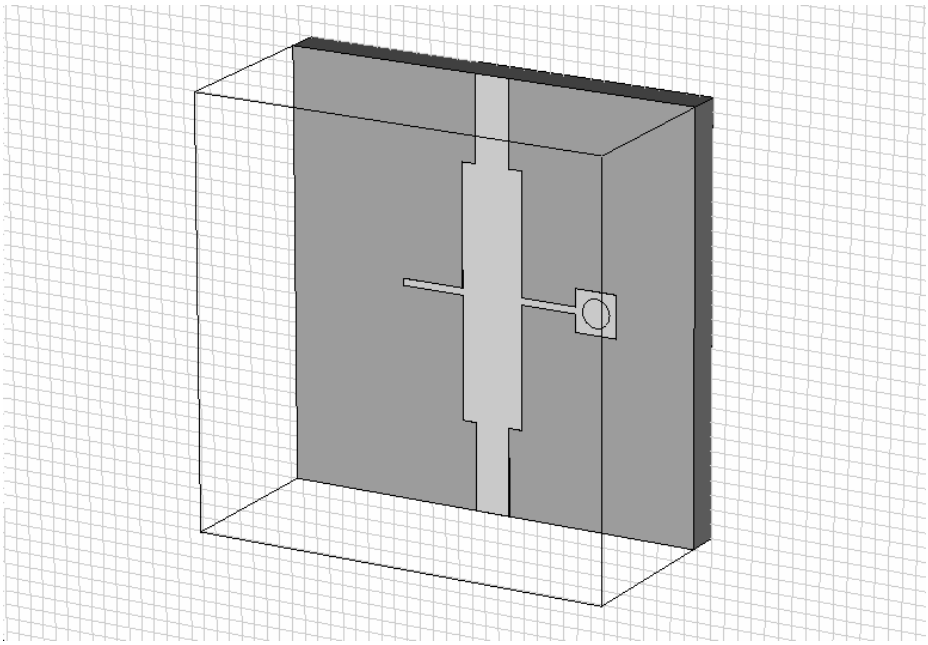
Since the structure will be embedded in a perfect electrically conducting box, some space is needed between the metallization layer and the top wall of the enclosure.

This can be easily achieved in the *Background Properties* dialog box that can be opened by selecting *Solve* ⇨ *Background Material* ( ).



In most cases it is sufficient to add an additional space of about five times the height of the substrate on top of the metallization. Thus, you should now enter 125 (= 5 • 25) in the *Upper Z distance* field and press *OK*.

The structure should then finally appear as shown in the picture below (the working coordinate system has been deactivated using *WCS* ⇨ *Local Coordinate System*,). The additional space is indicated by the box.



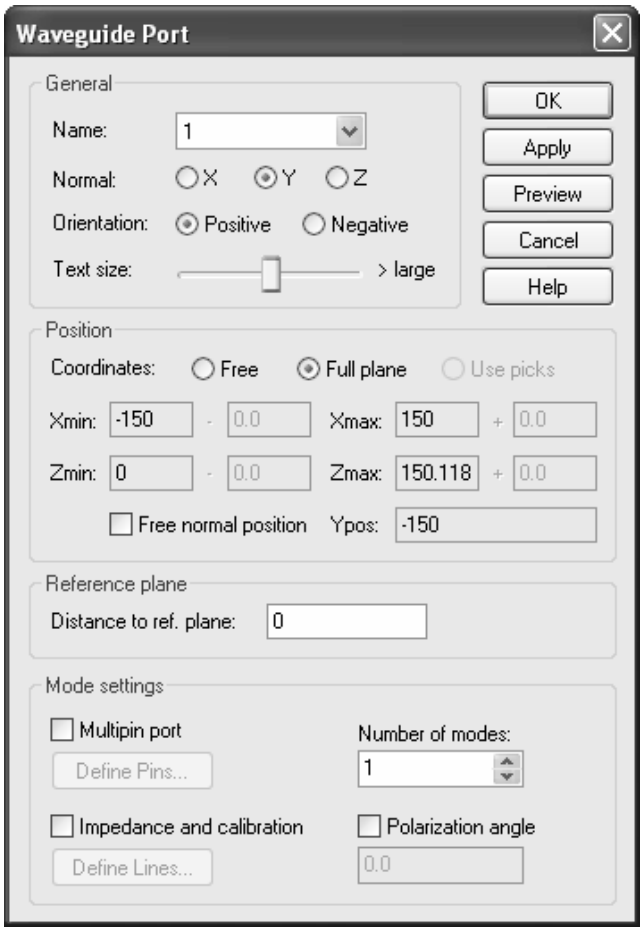
☐ Define Ports

The next step is to add the ports to the microstrip device for which the S-parameters will later be calculated. Each port will simulate an infinitely long waveguide (here stripline) structure that is connected to the structure at the port's plane. Waveguide ports are the most accurate way to calculate the S-parameters of microstrip devices and should thus be used here.

A waveguide port extends the structure to infinity. Its transversal extension must be large enough to sufficiently cover the microstrip mode. On the other hand, it should not be chosen excessively large in order to avoid higher order mode propagation in the port. A good choice for the width of the port is roughly ten times the width of the stripline. A proper height is about five times the height of the substrate.

Applying these guidelines to the example here, you find that the optimum port's width is roughly 250 mil and that its height should be about 125 mil. In this example, the whole model has a width of 300 mil and a height of 150 mil. Because these dimensions are close to the optimal port size you can simply take these dimensions and apply the port to the full extension of the model. Read the *Getting Started* manual to obtain more information on defining waveguide ports.

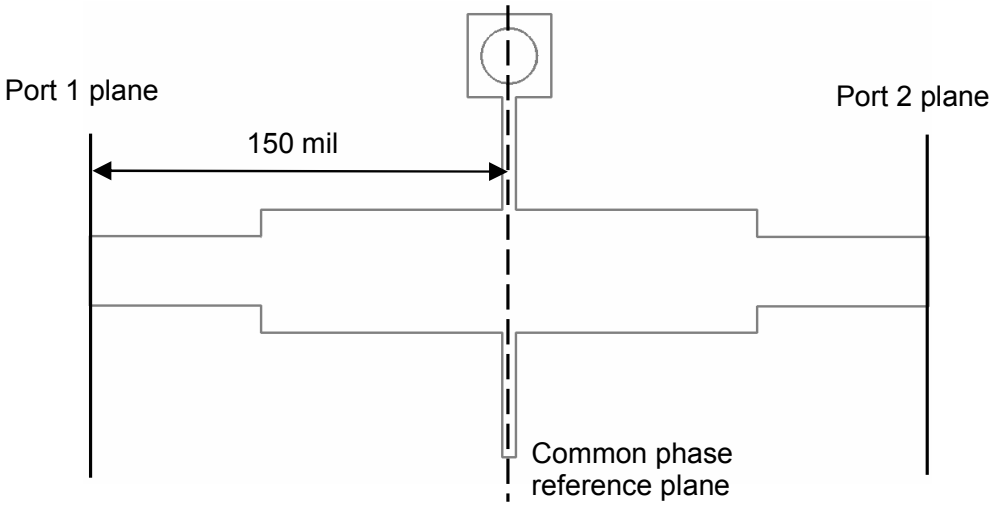
Please open the waveguide port dialog box (*Solve* ⇨ *Waveguide Ports*,) to define the first port:



Here, you should set the *Normal* of the port's plane to the *Y*-direction and its *Orientation* in the positive Y-direction (*Positive*). Because the port should extend across the entire boundary of the model, you can simply keep the *Full plane* setting for the transversal position. Without the *Free normal position* check button activated, the port will be allocated as default on the boundary of the calculation domain.

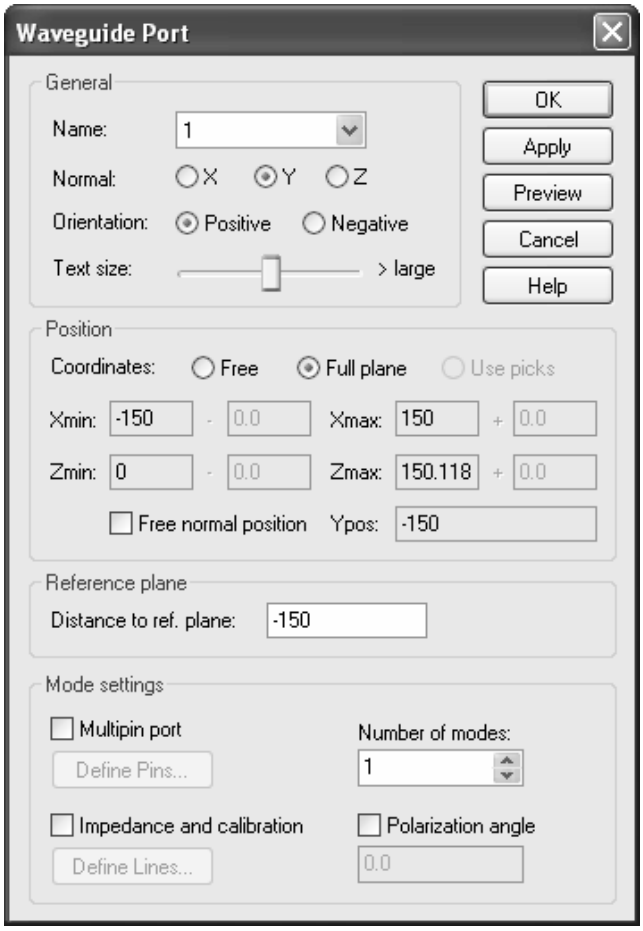
The next step is to choose how many modes should be considered by the port. For microstrip devices, a single mode usually propagates along the line. Therefore, you should keep the default setting of one mode.

Let's assume that you are interested in the additional phase shift of the device compared to a microstrip line of the same length. In this case, you could move the phase reference plane for both ports to the center of the structure as shown below:



Therefore, please enter the distance between the port's plane and the phase reference plane in the *Dist. to ref. plane* field. Please note that you must enter a negative number (-150) to move the reference plane inwards. After entering the distance, you may press the *Tab* key to move the focus to the next dialog element. After a reference plane distance has been set, the location of this plane will be visualized in the main view.

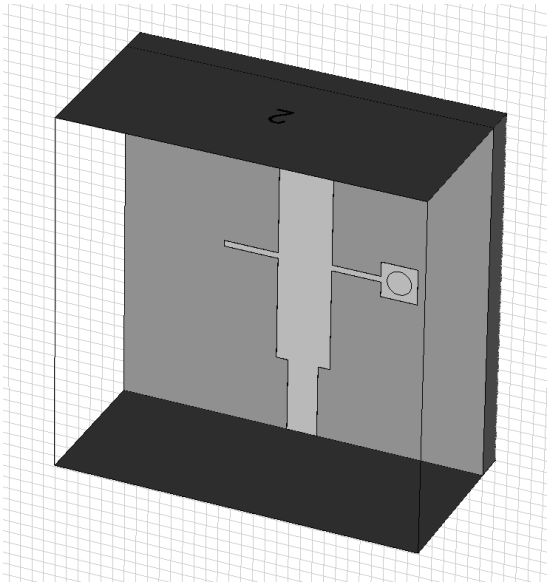
Please finally check the settings in the dialog box and press the *OK* button to create the port:



You can now repeat the same steps for the definition of the opposite port 2:

1. Open the waveguide port dialog box. (*Solve* ⇨ *Waveguide Ports*,)
2. Set the *Normal* to Y.
3. Set the *Orientation* to *Negative*.
4. Enter the reference plane distance of –150.
5. Press *OK* to store the port's settings.

Your model should now look as follows:



☐ Define the Boundary Conditions

In this case, the structure is embedded within a perfect electrically conducting enclosure. Because this is the default set by the template for planar structures, you do not need to change any settings here.

☐ Define the Frequency Range

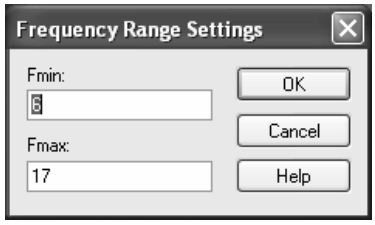
The frequency range for the simulation should be chosen with care. In contrast to frequency domain tools, the performance of a transient solver can be degraded if the chosen frequency range is too small (the opposite is usually true for frequency domain solvers).

We recommend using reasonably large bandwidths of 20% to 100% for the transient simulation. In this example, the S-parameters are to be calculated for a frequency range between 6 and 17 GHz. With the center frequency being 11.5 GHz, the bandwidth (17 GHz – 6 GHz = 11 GHz) is about 96% of the center frequency, which is sufficiently large. Thus, you can simply choose the frequency range as desired between 6 and 17 GHz.

> **Please note:** Assuming that you were interested primarily in a frequency range of e.g. 11.5 to 12.5 GHz (for a narrow band filter), then the bandwidth would only be about 8.3%. In this case, it would make sense to increase the frequency range (without losing accuracy) to a bandwidth of 30% that corresponds to a frequency range of 10.2 – 13.8 GHz. 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. to 0.01 GHz.

After the proper frequency band for this device has been chosen, you can simply open the frequency range dialog box (*Solve* ⇨ *Frequency*,) and enter the range from 6 to

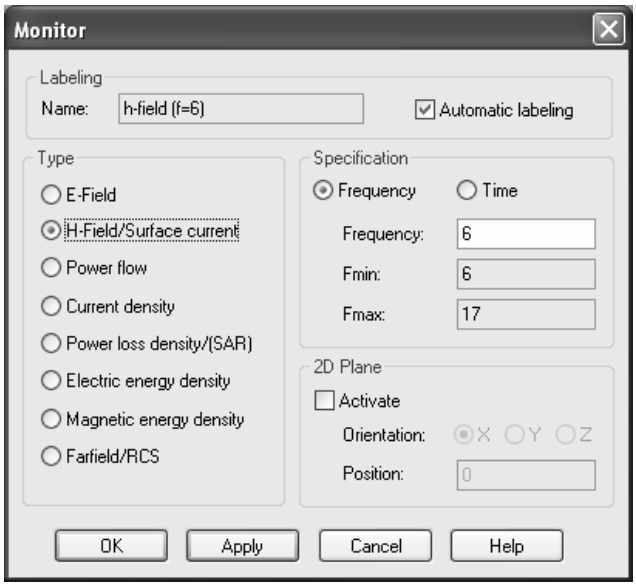
17 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz and is displayed in the status bar):



☐ Define Monitors for the Surface Current

In addition to the S-parameters, an interesting result for microstrip devices is the current distribution as a function of frequency. The transient solver in CST MICROWAVE STUDIO® is able to obtain the surface current distribution for an arbitrary number of frequency samples from a single calculation run. You can define "field monitors" to specify the frequencies at which the field data shall be stored.

Please open the monitor definition dialog box by selecting *Solve* ⇨ *Field Monitors* ( ):



In this dialog box you should select the *Type H-Field/Surface current* before you specify the frequency for this monitor in the *Frequency* field. Afterwards, you should press the *Apply* button to store the monitor's data. Please define monitors for the following frequencies: 6, 9, 12, 15 (with GHz being the currently active frequency unit). Please make sure that you press the *Apply* button for each monitor. The monitor definition is then added in the *Monitors* folder in the navigation tree. The volume in which the fields are recorded is indicated by a box.

After the monitor definition is complete, you can close this dialog box by pressing the *OK* button.

S-Parameter and Field Calculation

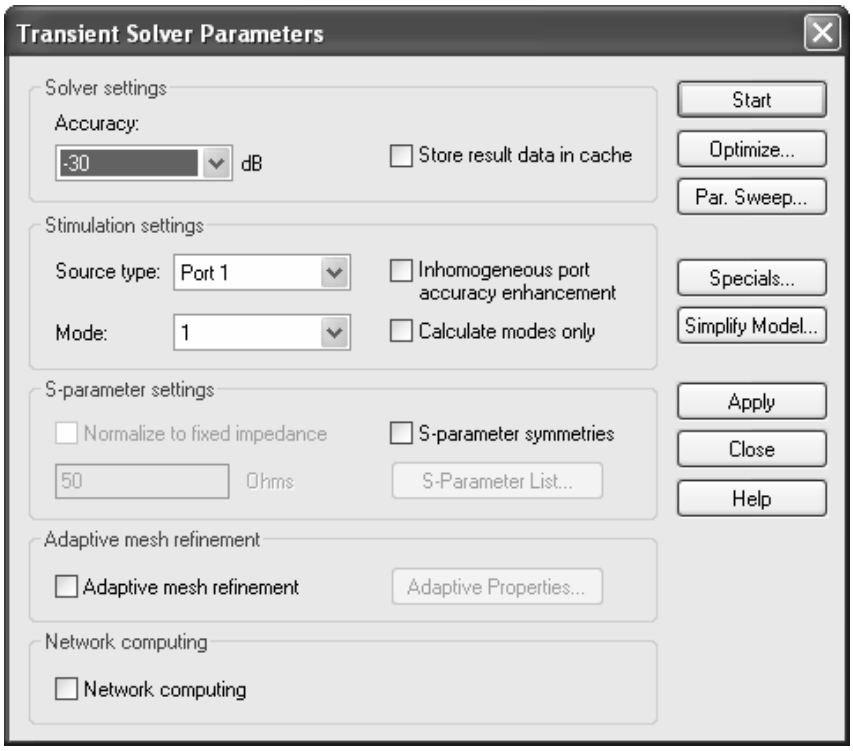
A key feature of CST MICROWAVE STUDIO® is the *Method on Demand* approach that allows a simulator or mesh type that is best suited to a particular problem. Another benefit is the ability to compare results obtained from completely independent approaches. We demonstrate this strength in the following two sections by calculating the S-parameters with the transient solver and the frequency domain solver. The transient simulation uses a hexahedral mesh while the frequency domain calculation is performed with a tetrahedral mesh.

Both sections are self-contained parts and it is sufficient to work through only one of them depending on what solver you are interested in. The chapter ends with a comparison of the two methods.

Please note that not all solvers may be available to you due to license restrictions. Please contact your sales office for more information.

Transient Solver

The transient solver's parameters are specified in the solver control dialog box that can be opened by selecting *Solve* ⇨ *Transient Solver* from the main menu or by pressing the corresponding icon in the toolbar.



Because the structure is fully symmetric, it is sufficient to calculate the S-parameters S1,1 and S2,1 to get all of the information about the device. Both results can be obtained by exciting the structure at port 1 only, so change the *Source type* to Port 1.

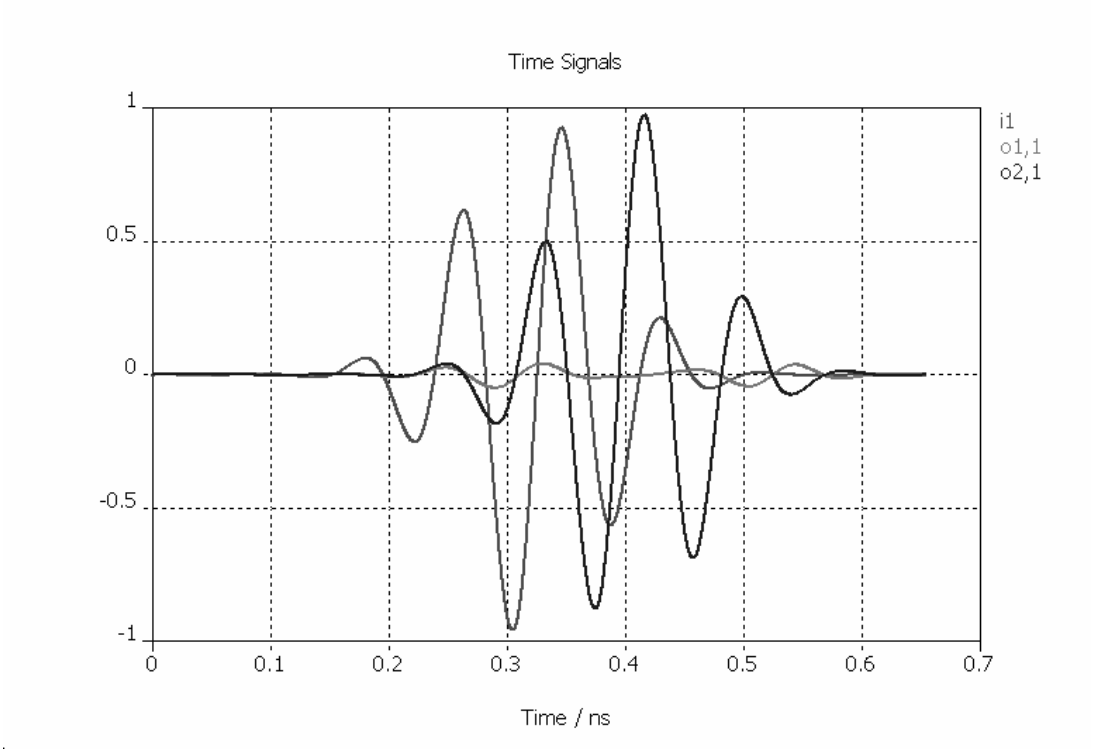
Finally, press the *Start* button to start the calculation. A progress bar and abort button appear in the status bar, displaying some information about the solver stages.

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

Congratulations, you have simulated the microstrip phase bridge using the transient solver! Let's review the results.

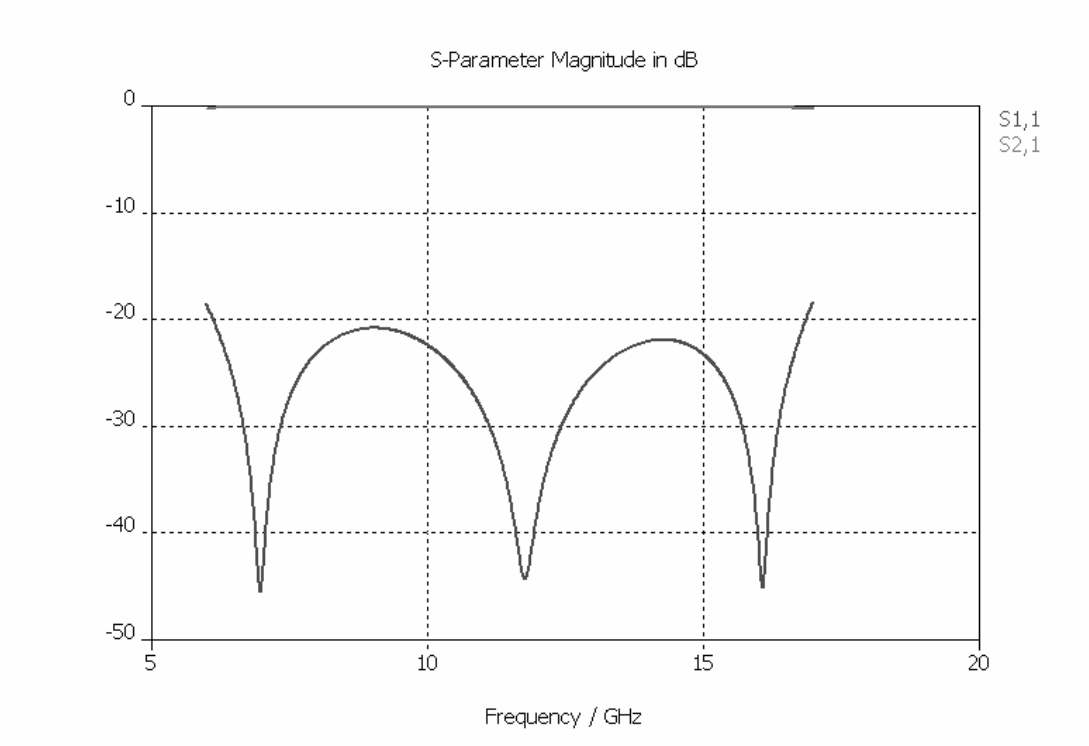
- ☐ 1D Results (Port Signals, S-Parameters)

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



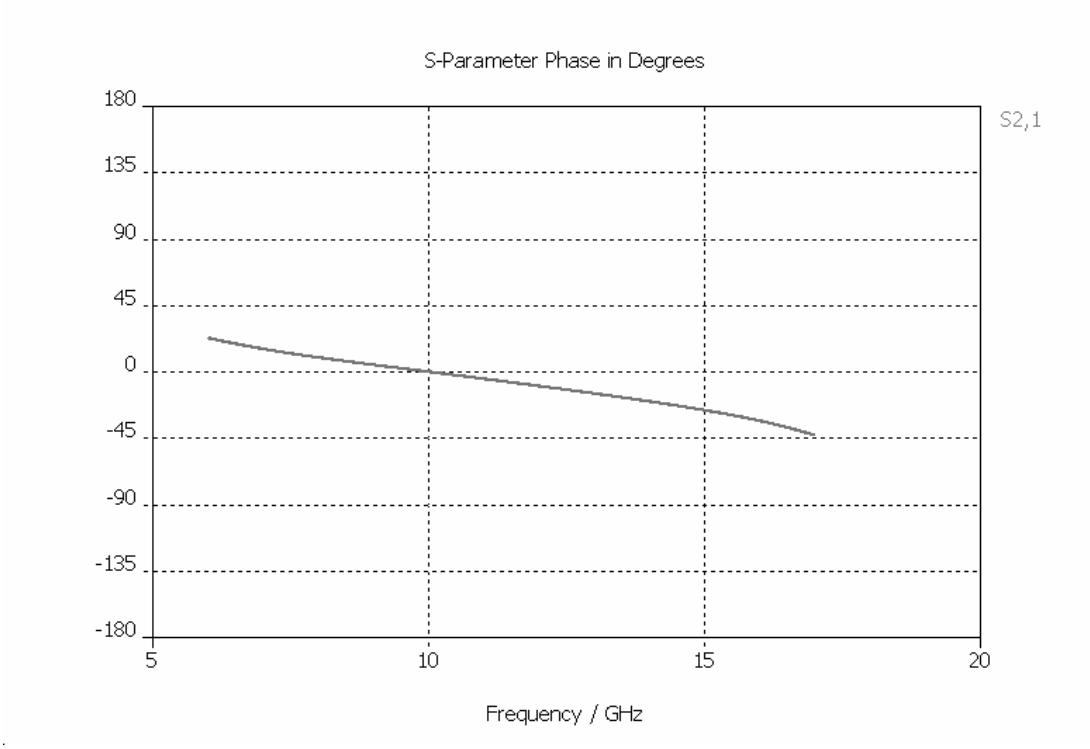
This plot shows the incident, reflected and transmitted wave amplitudes at the ports versus time. The incident wave amplitude is called i1 and the reflected or transmitted wave amplitudes of the two ports are o1,1 and o2,1. These curves already show that the reflection is quite small for this device.

The S-parameters magnitude in dB scale can be plotted by clicking on the *1D Results* ⇨ */S/dB* folder.



As expected, the input reflection S1,1 is quite small (less than –20 dB) for most of the frequency range.

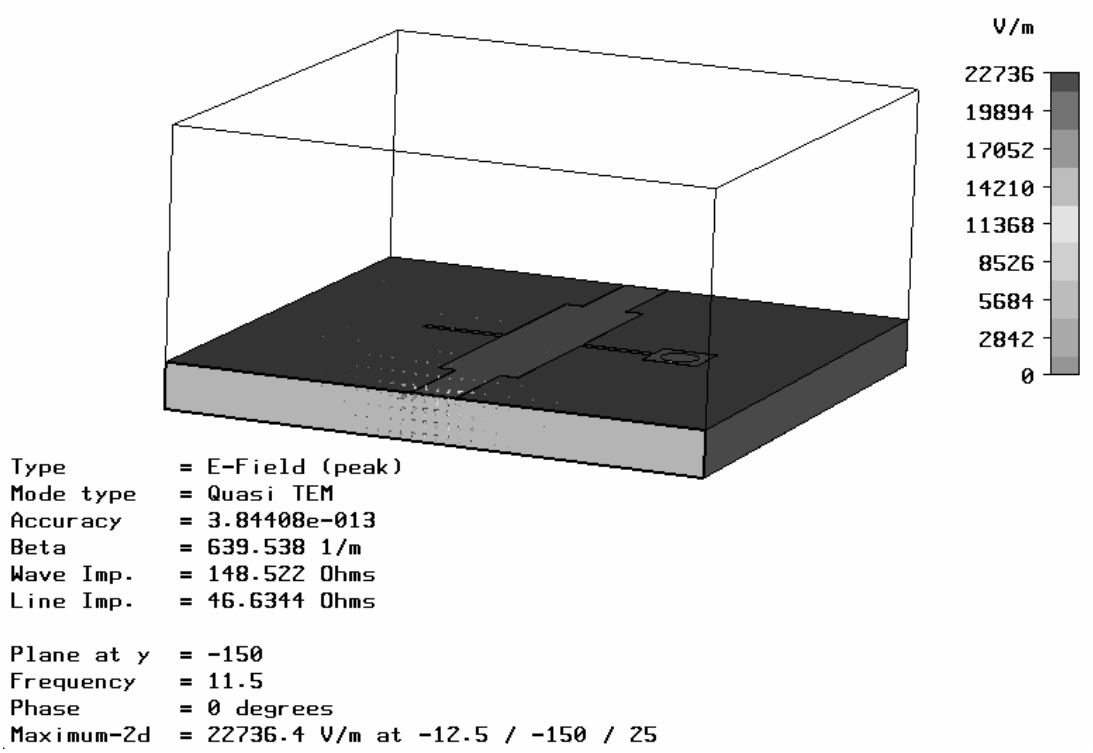
The most important S-parameter information for a phase shifter is the transmission phase that can be visualized by clicking on the *1D Results* ⇨ *arg(S)* folder. If you want to visualize the phase curve for S2,1 only, you can also select the sub-item *1D Results* ⇨ *arg(S)* ⇨ *S2,1*:



Please note: Because the reference plane is set to the center of the structure, this is the additional phase shift compared to the standard stripline.

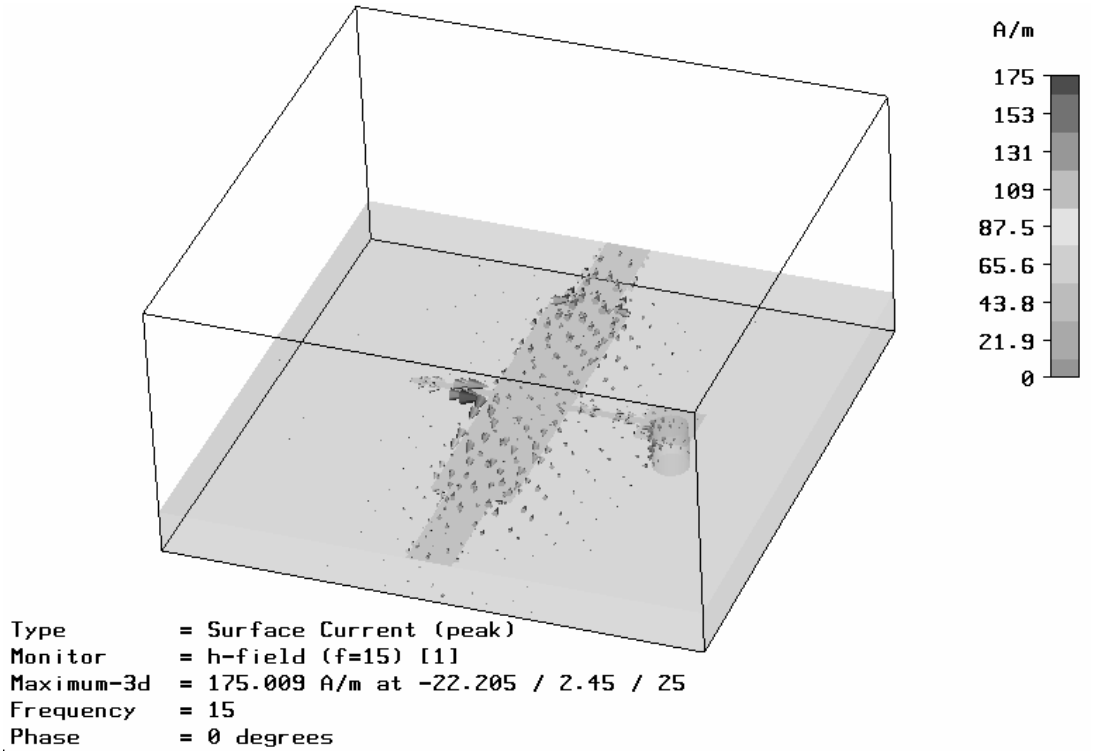
- 2D and 3D Results (Port Modes and Field Monitors)

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 of the navigation tree. To visualize the electric field of the fundamental port mode you should 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. Please refer to the *Getting Started* manual for more information on how to change the plot's parameters.



The plot also shows some important properties of the mode such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

The three-dimensional surface current distribution on the conductors can be shown by selecting one of the entries in the *2D/3D Results ⇨ Surface Current* folder in the navigation tree. The surface current at a frequency of 15 GHz can thus be visualized by clicking at the *2D/3D Results ⇨ Surface Current ⇨ h-field (f=15) [1]* entry:



You can toggle an animation of the currents on and off by selecting *Results ⇨ Animate Fields*. The surface currents for the other frequencies can be visualized in the same manner.

Accuracy Considerations

The transient S-parameter calculation is mainly 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.

In the following section we provide hints on how to minimize these errors and obtain highly accurate results.

- 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 themselves 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 using the *Accuracy* setting in the transient solver control dialog box. The default value of –30 dB will usually give sufficiently accurate results for coupler devices. However, to obtain highly accurate results for filter structures it is sometimes necessary to increase the accuracy to –40 dB or –50 dB.

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

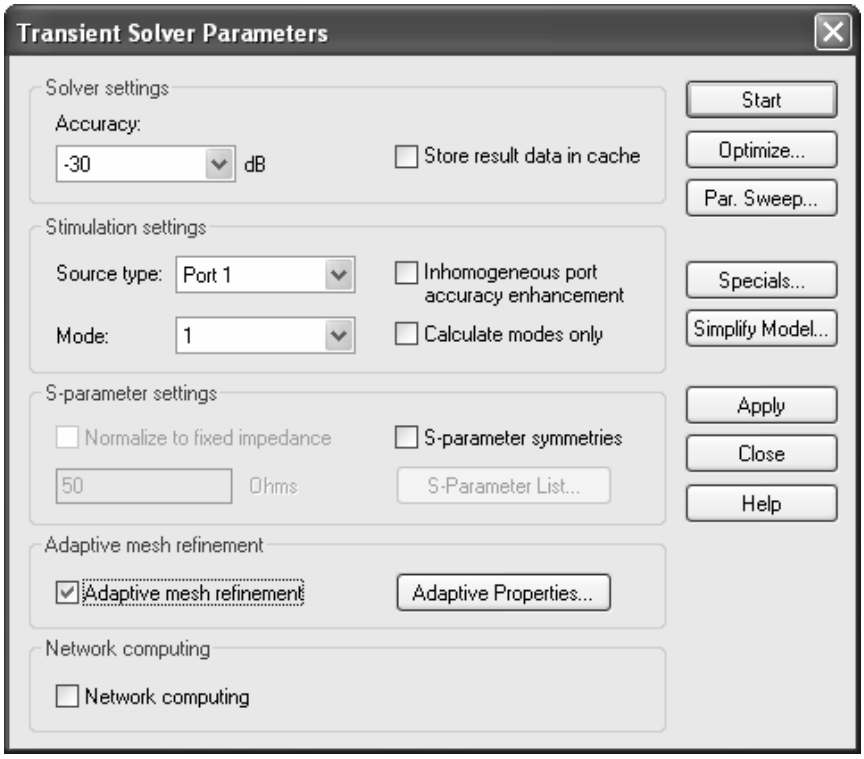
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

The following rule may also be useful: If you find a large ripple in the S-parameters, increase the solver's accuracy setting.

☐ Effect of the Mesh Resolution on the S-parameter's Accuracy

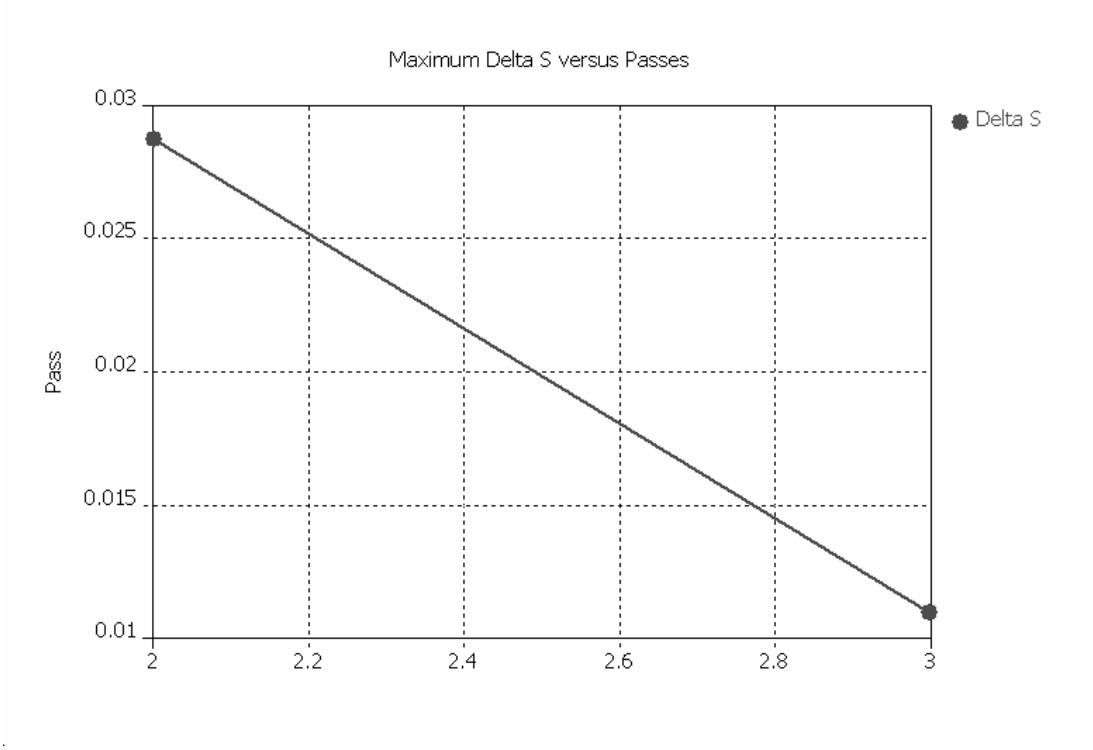
Inaccuracies arising from the finite mesh resolution are usually more difficult to estimate. The only way to ensure the accuracy of the solution is to increase the mesh resolution and recalculate the S-parameters. If these results no longer change significantly when the mesh density is increased, then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

In the example above, you have used the default mesh that has been automatically generated by an expert system. The easiest way to test the accuracy of the results is to use the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (*Solve ⇨ Transient Solver):*

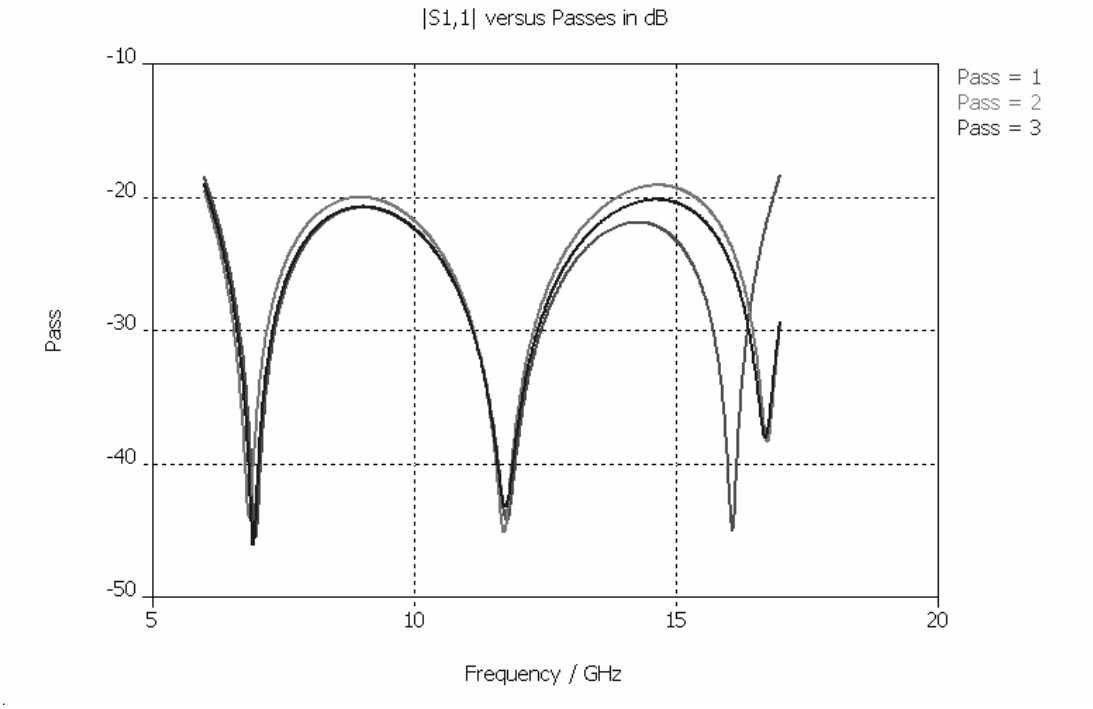


Please note that the previously selected template has changed the default settings to the energy based adaptive strategy that is more convenient for planar structures. Thus, you only have to activate the *Adaptive mesh refinement* tool in the "Transient Solver Parameters" dialog and start the solver again by pressing the *Start* button.

The solver is now running through several mesh adaptation passes until the desired accuracy limit (2% by default) is reached. After the mesh adaptation procedure is complete, you can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results ⇨ Adaptive Meshing ⇨ Delta S* from the navigation tree:

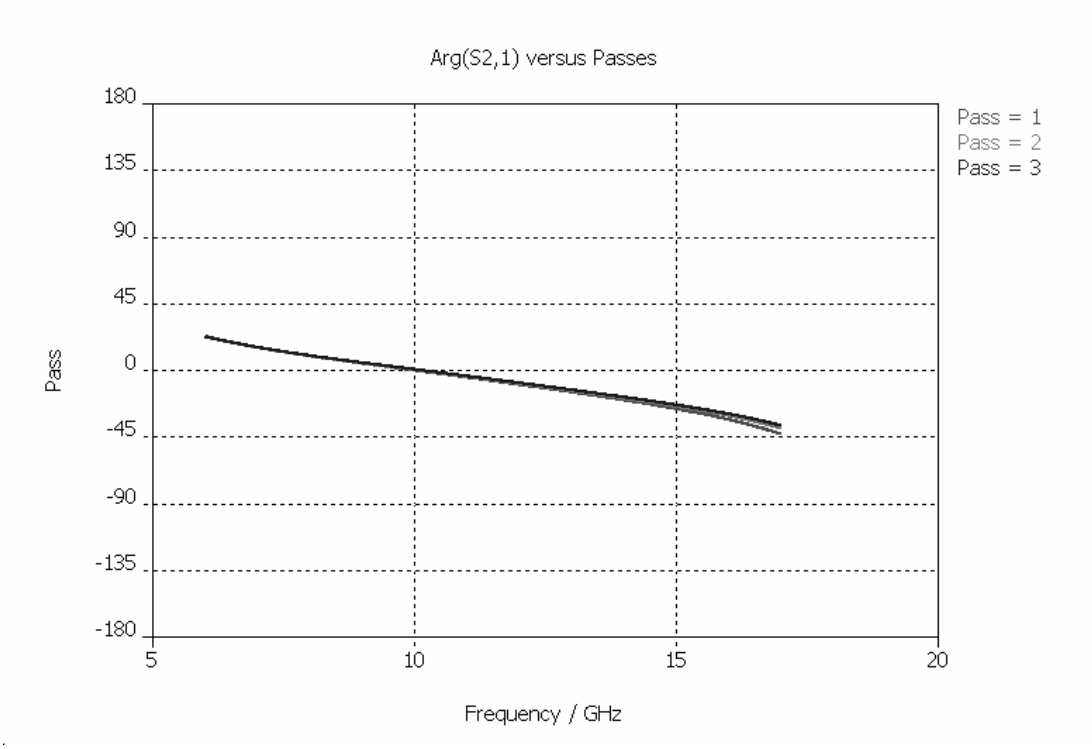


Visualization of the S-parameter curves for different adaptation passes provides deeper insight into the performance of the mesh adaptation. The following plot is obtained by selecting *1D Results ⇨ Adaptive Meshing ⇨ |S| dB ⇨ S1,1* from the navigation tree:



The positions of the extremal values tend to converge as the refinement proceeds. This clearly illustrates the positive effect of the mesh adaptation. The convergence process of the other S-parameters' magnitudes and phases can be visualized in the same

manner. By inspecting the plots, you can confirm that the important results for the transmission phase are quite stable:



Please note: Refer to the *Getting Started* manual for information on *Template Based Postprocessing* for automated extraction and visualization of arbitrary results from various simulation runs.

Frequency Domain Solver

CST MICROWAVE STUDIO® offers a variety of frequency domain solvers that are specialized for different type of problems. They differ not only by their algorithms but also by the grid type they are based on. The general purpose frequency domain solvers are available for hexahedral grids as well as for tetrahedral grids. The availability of a frequency domain solver within the same environment offers a very convenient way to cross-check results produced by the time domain solver with minimal additional effort.

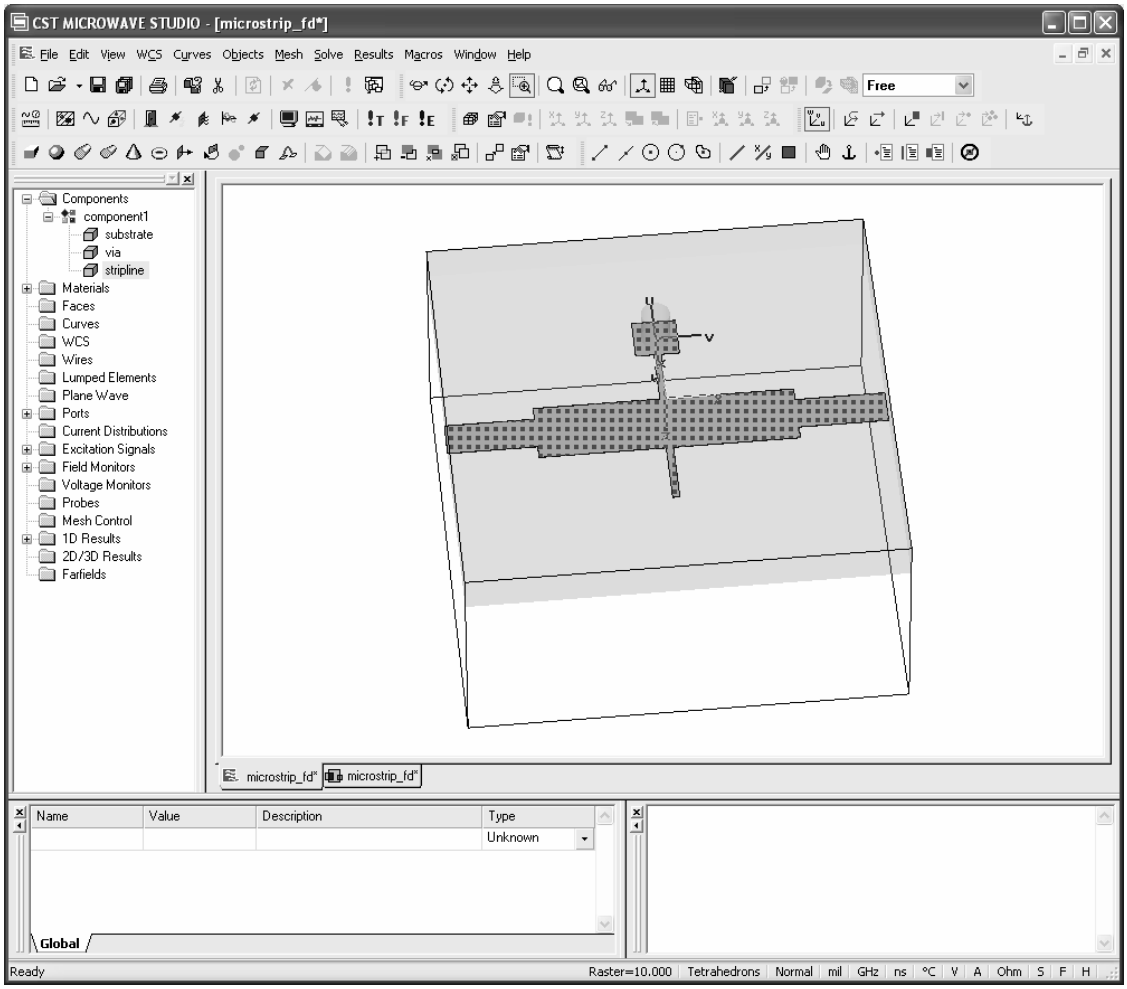
☐ Making a Copy of Transient Solver Results

Before performing a simulation with a frequency domain solver, you may want to keep the results of the transient solver in order to compare the two simulations. The copy of the current results is obtained as follows: Select, for example, the |*S*| *dB* folder in *1D Results*, then press *Ctrl+c* and *Ctrl+v.* The copies of the results will be created in the selected folder. The names of the copies will be *S1,1_1, S2,1_1* etc. You may rename them to *S1,1_TD*, *S2,1_TD* and so on with the *Rename* command from the context menu. Use *Add 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.

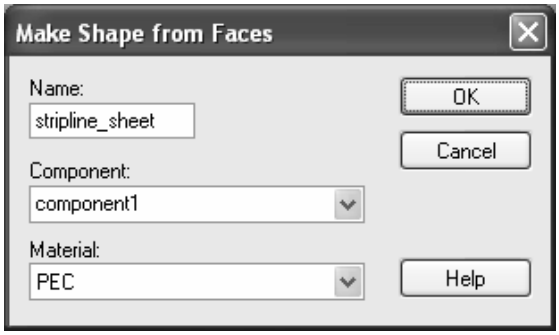
☐ Optimizing Structure for Tetrahedral Mesh

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 stripline in our example. Because this

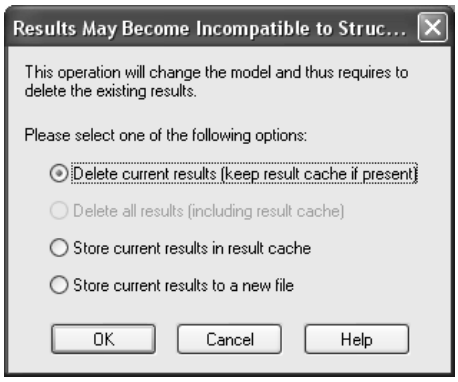
thickness has a rather small influence on the results compared to a zero thickness, we rebuild the stripline as a PEC sheet. First, select the stripline in the navigation tree and then the stripline's bottom face using the face pick tool (*Objects ⇨ Pick ⇨ Pick Face*, ■):



Open the "Make Shape from Faces" dialog box by selecting *Objects ⇨ Face Healing Tools ⇨ Shape from Picked Faces,* . Assign a name to the new shape by entering e.g. "stripline_sheet" in the *Name* field. Press *OK* to create the new solid.



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 apears:

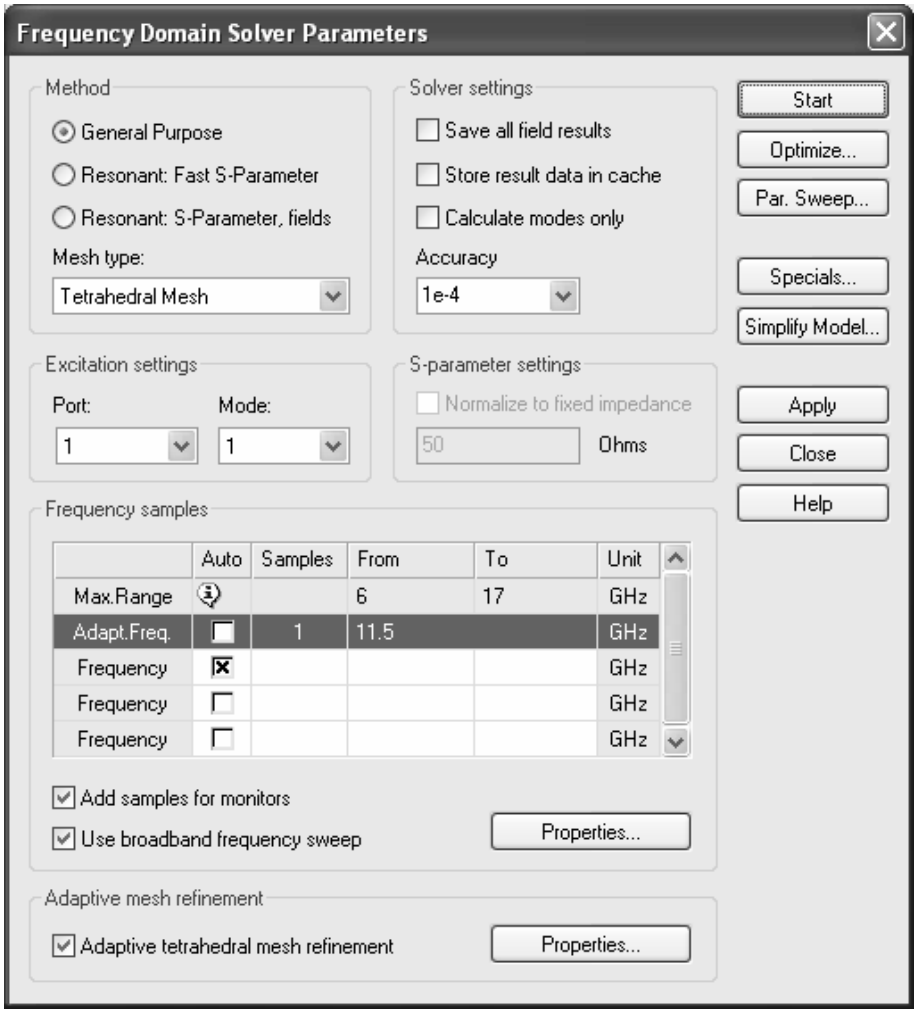


Press *OK* to acknowledge deletion of the previous results.

Now delete the old solid “stripline”. Select the solid in the navigation tree and choose *delete* from the context menu. It is now time to start the solver.

☐ Frequency Domain Solver Settings

Open the “Frequency Domain Solver Parameters” dialog box by selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu or by pressing the corresponding icon in the toolbar.



In the *Mesh Type* combo box you may choose between Hexahedral or Tetrahedral Mesh. Please choose *Tetrahedral Mesh.* Switch off the *Auto* button for the adaptation frequency in the *Frequency Samples* frame. Now you can change the adaptation frequency to 11.5 GHz to optimize the mesh at a frequency within the passband of the filter device.

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 obtain the continuous S-parameter values. The frequency samples are chosen automatically 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 check box is activated by default. All other settings should be left unchanged. Everything is now ready; you may press *Start* to begin the calculation.

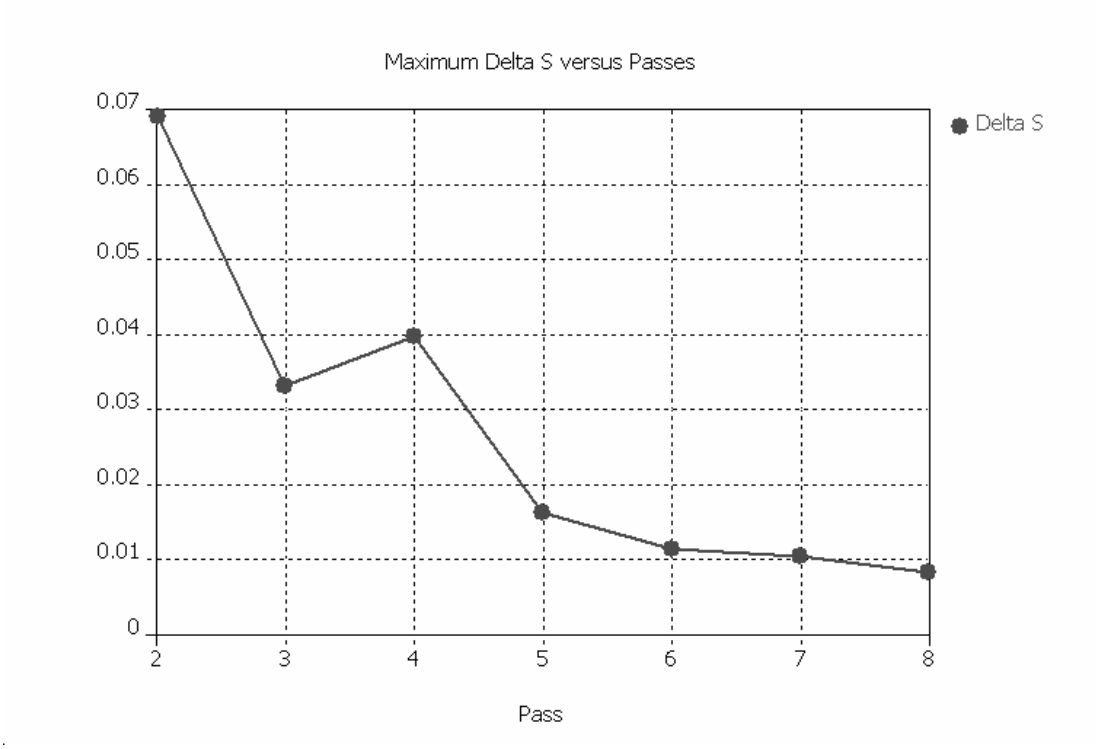
A progress bar and abort button appear in the status bar displaying some information about the solver stages:

During the simulation, the "Message Window" will show details about the performed simulation. After the maximum number of adaptation passes has been reached, the simulation stops. The warning indicates that the desired accuracy limit (1% by default) has not been reached twice. Nevertheless, the maximum deviation of the S-parameters is below 1% after the eighth pass.

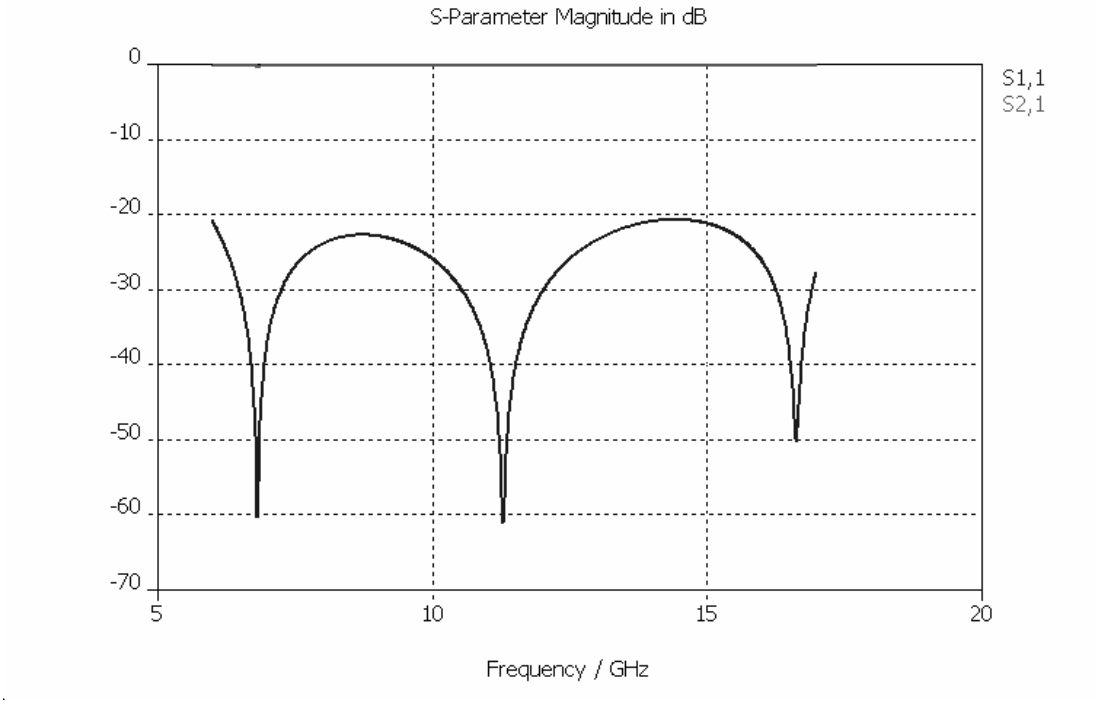
Congratulations, you have simulated the microstrip phase bridge using the frequency domain solver! Let's review the results.

- 1D Results (S-Parameters)

You can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *Delta S* from the navigation tree:



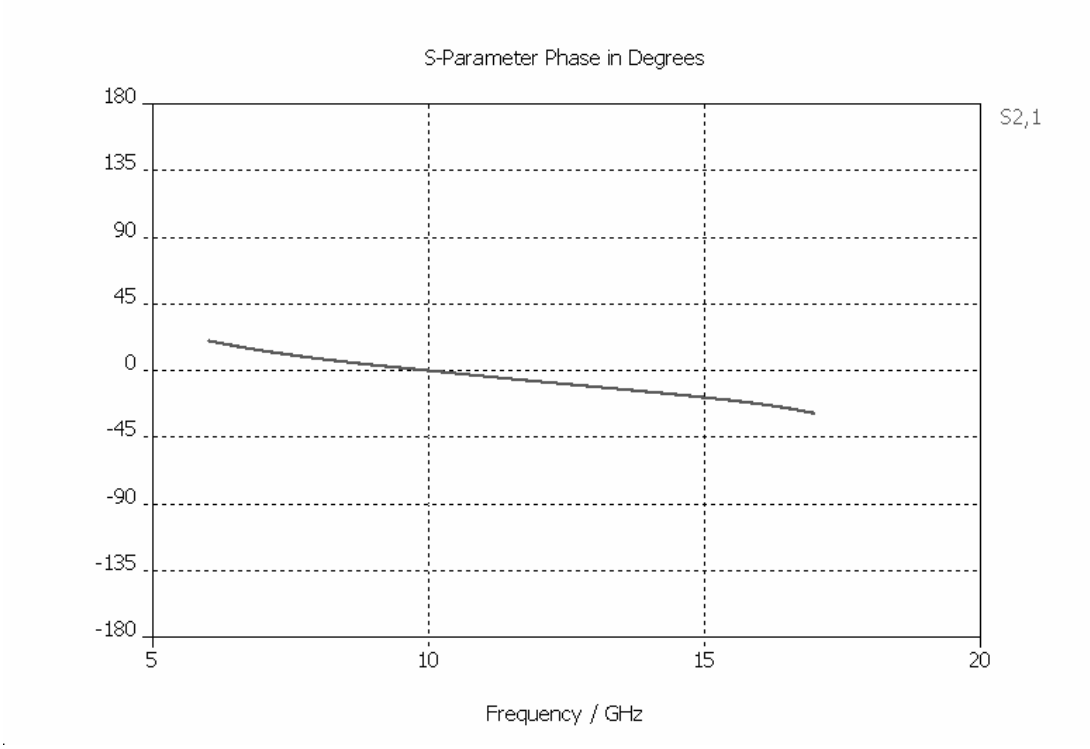
As evident in the above plot, the maximum deviation of the S-parameters is already beneath 2% after the fifth pass. The S-parameters magnitude in dB scale can be plotted by clicking on the *1D Results* ⇨ *S dB* folder:



The input reflection S1,1 is quite small (less than –20dB) for the entire frequency range.

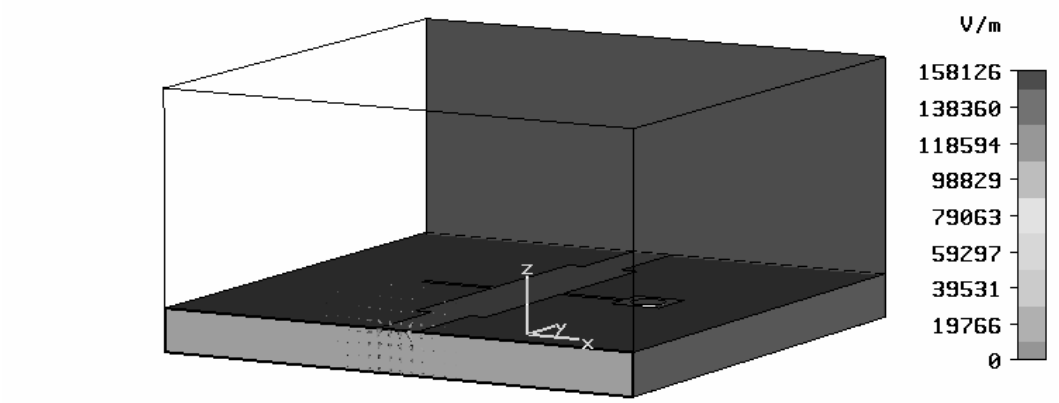
The most important S-parameter information for a phase shifter is probably the transmission phase that can be visualized by clicking on the *1D Results* ⇨ *arg(S)* folder.

If you want to visualize the phase curve for S2,1 only, you can also select the sub-item *1D Results ⇨ arg(S) ⇨ S2,1*:



☐ 2D and 3D Results (Port Modes and Field 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 fundamental port mode you should click on the *e1* folder. Open the "Select Port Mode" dialog box by selecting *Results ⇨ Select Mode Frequency* from the main menu and change the frequency to 11.5 GHz. Please confirm your setting by pressing *OK*. After properly rotating the view and tuning the settings in the plot properties dialog box, you should obtain a plot similar to that shown below (please refer to the *Getting Started* manual for more information on how to change the plot's parameters):



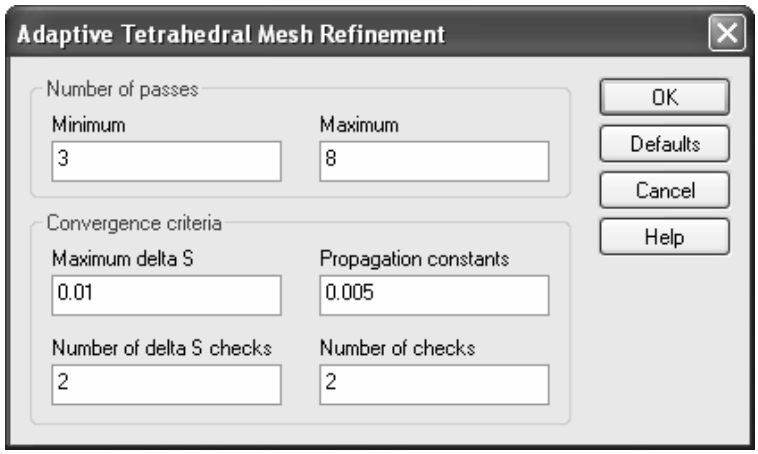

```
Type         = E-Field (peak)
Mode type    = Quasi TEM
Accuracy     = 9.71336e-014
Beta         = 641.064 1/m
Imp. ZPI     = 47.453 Ohms (re)
Wave Imp.    = 163.411 Ohms
Line Imp.    = 47.453 Ohms

Plane at y   = -150
Frequency    = 11.5
Phase        = 0 degrees
Maximum-2d   = 158126 V/m at 12.5709 / -150 / 25
```

The plot also shows some important properties of the mode such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

Accuracy Considerations

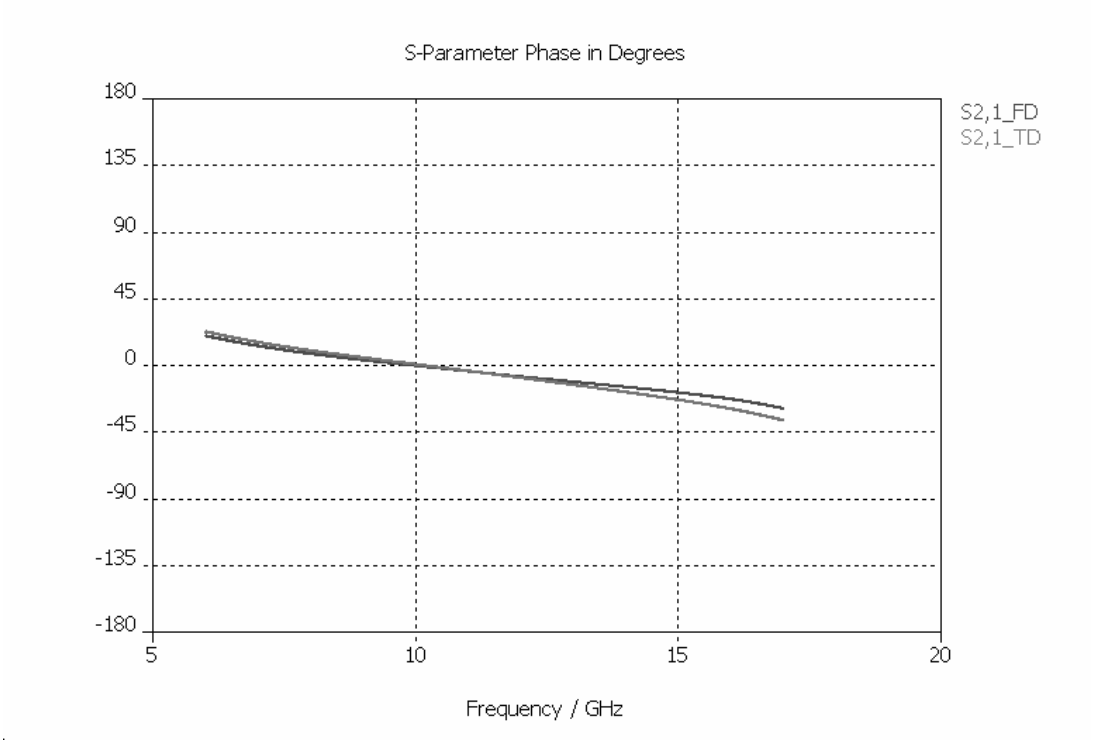
The results of the frequency domain solver using the tetrahedral mesh are mainly affected by the inaccuracies arising from the finite mesh resolution. In the case of a tetrahedral mesh the adaptive mesh refinement is switched on by default. The mesh adaptation is performed by checking the convergence of the S-parameter values at the highest simulation frequency. The adaptation is oriented towards achieving highly accurate S-parameter calculations.

If the quality of the results seems unsatisfactory, additional mesh refinement can be performed. For example, three additional mesh adaptation passes can be forced by re-starting the frequency domain solver without changing any parameters. Three mesh adaptation passes will be performed according to the “Minimum number of passes“ setting. This setting can be accessed by pressing “Properties” in the “Adaptive Mesh Refinement” frame of the “Frequency Domain Solver Parameters” dialog:



Comparison of the Solver Results

The next plot shows the phase of the S-Parameter S2,1 resulting from the time domain and frequency domain simulations. Plotting the S-Parameter curves in the same graph allows for a better comparison of the results.



As you can see, the results agree very well. Because the results are not converged to the highest possible accuracy level, there is still a slight deviation. This deviation will be reduced when refining the accuracy limit in the adaptive mesh refinement. The difference of the stripline's thickness also influences the variations in the S-parameters.

The following table shows the number of cells and the simulation time when running the solvers with the default settings for the adaptive mesh refinement:

	Transient Solver	Frequency Domain Solver
Mesh	56595 cells	13719 tetrahedrons
Simulation time	215 seconds	371 seconds

Getting More Information

Congratulations! You have just completed the planar device tutorial that should have provided you with a good working knowledge on how to use the transient solver to calculate S-parameters. The following topics have been covered:

1. General modeling considerations, using templates, etc.
2. Model a planar structure by using the extrude tool, define the substrate and create a via.
3. Define ports.
4. Define frequency range and boundary conditions.
5. Define field monitors for surface current distributions.
6. Start the transient or the frequency domain solver.
7. Visualize port signals and S-parameters.
8. Visualize port modes and surface currents.
9. Obtain accurate and converged results using the adaptive mesh refinement.

You can obtain more information for each particular step from the online help system that can be activated either by pressing the *Help* button in each dialog box or by pressing the *F1* key at any time to obtain context sensitive information.

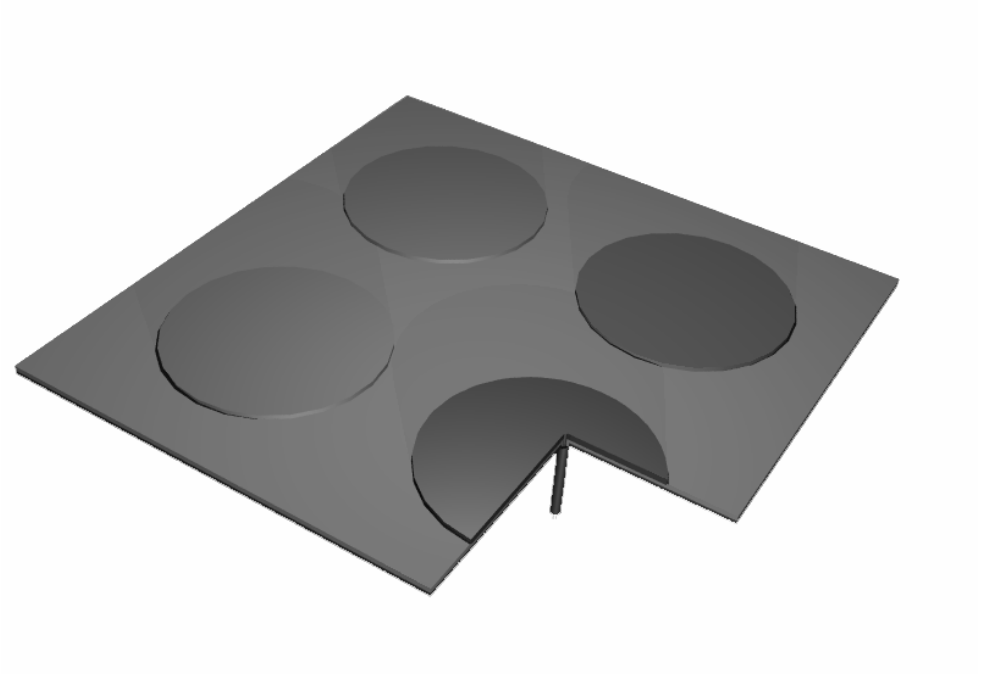
In some cases we have referred to the *Getting Started* manual that is also a good source of information for general topics.

In addition to this tutorial you can find some more S-parameter calculation examples for planar structures in the examples folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that will give you some more information about the particular device.

Finally, you should refer to the *Advanced Topics* manual for more in-depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please also visit one of the training classes held regularly at a location near you. Thank you for using CST MICROWAVE STUDIO®!

Antenna Tutorial



Geometric Constructions **116**

Introduction and Model Dimensions 116
Geometric Construction Steps 117

Common Solver Settings **128**

S-Parameter and Farfield Calculation **132**

Transient Solver 132
Transient Solver Results 134
Frequency Domain Solver 140
Frequency Domain Solver Results 145

Patch Antenna Array **149**

Antenna Array Calculation 150
Geometric Construction Steps 152
Combine Results 155
Simultaneous Excitation 159

Getting More Information **163**

Geometric Constructions

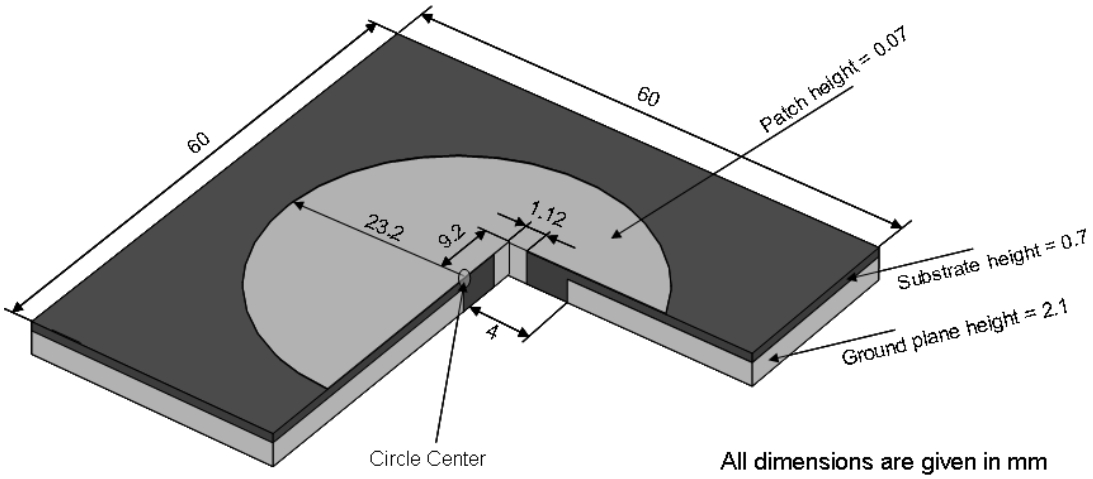
Introduction and Model Dimensions

In this tutorial you will learn how to simulate antenna devices. As a typical example you will analyze 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 tutorial 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.

We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.



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.

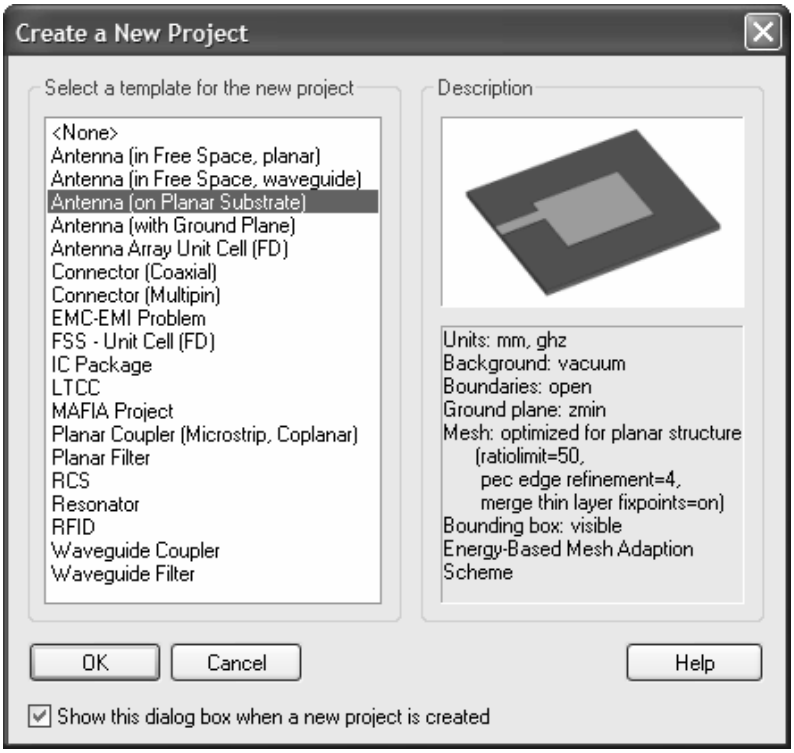
Geometric Construction Steps

This tutorial 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 *Edit* ⇨ *Undo* facility in the event that you want to cancel the last construction step.

☐ Select a Template

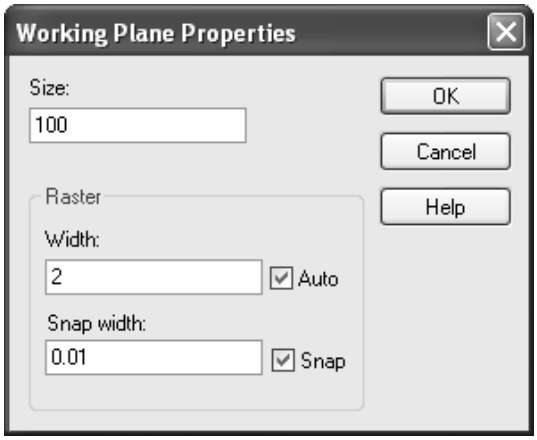
After you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that best fits your current device. Here, the "Antenna (on Planar Substrate)" template should be chosen.



This template automatically sets the units to mm and GHz, defines the background material to vacuum and selects appropriate boundary conditions (see chapter "Boundary Conditions"). Because the background material has been set to vacuum, the structure can be modeled just as it appears on your desk.

☐ Set the Working Plane's 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 *Edit* ⇨ *Working Plane Properties* from the main menu. Please note that we will use the same document conventions here as introduced in the *Getting Started* manual.

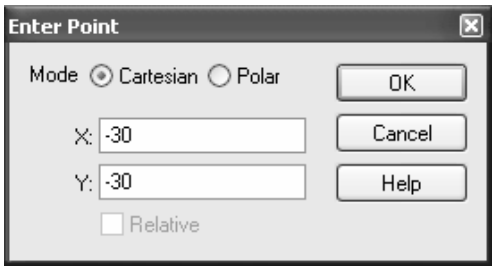


In this dialog box, you should set the *Size* to 100 (the unit which has previously been set to mm by the chosen template is displayed in the status bar), the *Raster width* to 2 and the *Snap width* to 0.01 to obtain a reasonably spaced grid. Please confirm these settings by pressing the *OK* button.

☐ Draw the Substrate Brick

The first construction step for modeling 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 now (*Objects ⇨ Basic 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:

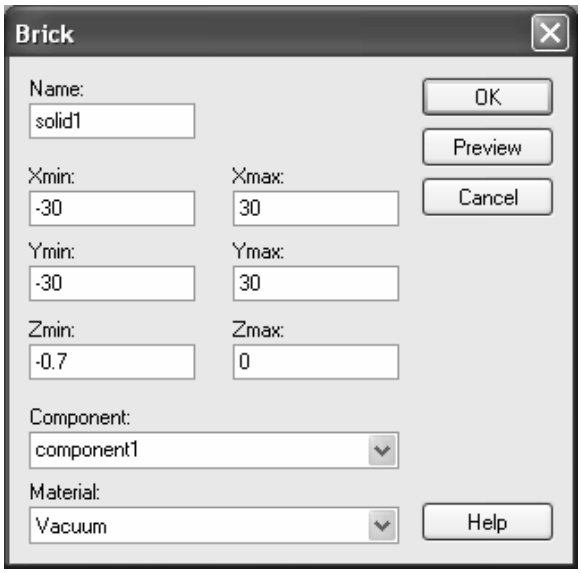


In this example, you should enter a substrate block that has an extension of 60 mm in each of the transversal directions. The transversal coordinates can thus be described by X = -30, Y = -30 for the first corner and X = 30, Y = 30 for the opposite corner, assuming that the brick is modeled symmetrically to the origin. Thus, please enter the first point's 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:

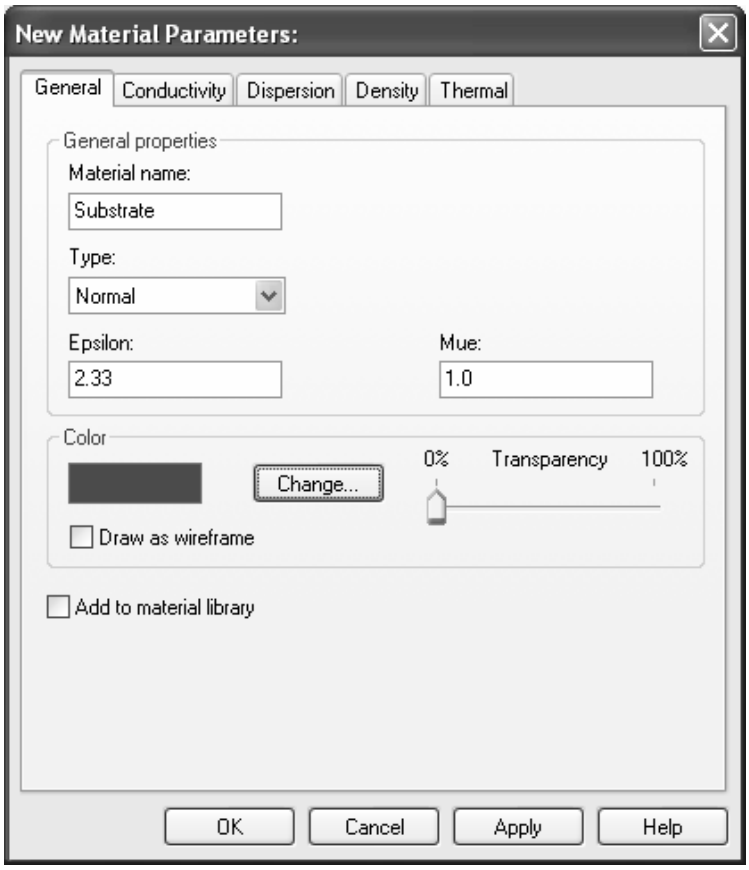


Please check all these settings carefully. If you encounter any mistakes, please change the value in the corresponding entry field.

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”).

> **Please note:** The use of different components allows you to combine several solids into specific groups, independent of their material behavior. However, in this tutorial, 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.

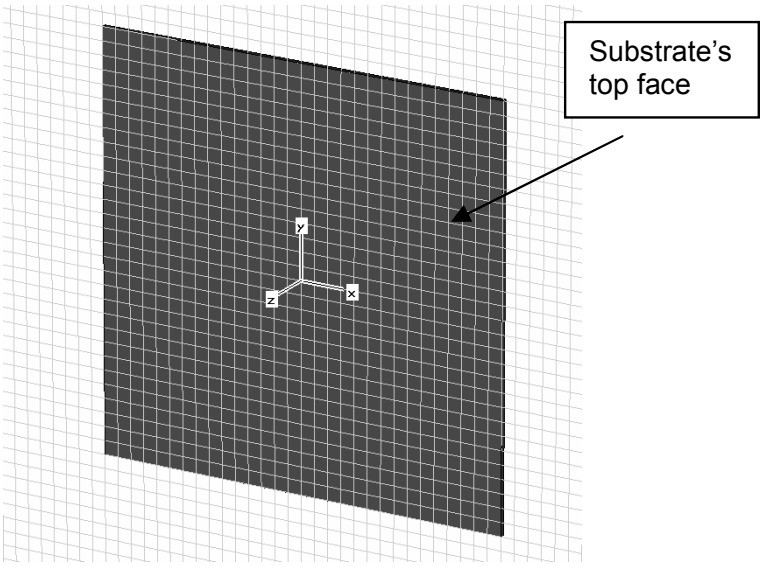
Finally, you need to define the substrate material. Because no material has yet been defined for the substrate, you should open the “New Material Parameters” dialog box by selecting “[New Material…]” from the *Material* dropdown list:



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 *Mue* fields. Here, you only need to change the dielectric constant *Epsilon* to 2.33. Finally, choose a color for the layer by pressing the *Change* button. Your dialog box should now look similar to the above picture before you press the *OK* button.

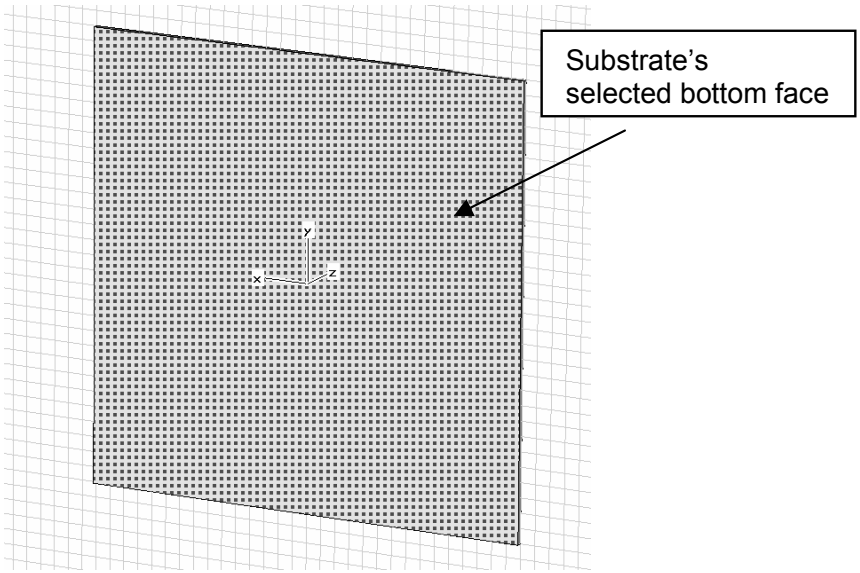
> **Please note:** The defined material “Substrate” will now be available inside the current project for the further creation of other solids. However, if you want to also save this specific material definition for other projects, you may check the button *Add to material library*. You will have access to this material database by clicking on *Load from Material Library* in the *Materials* context menu in the navigation tree.

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):

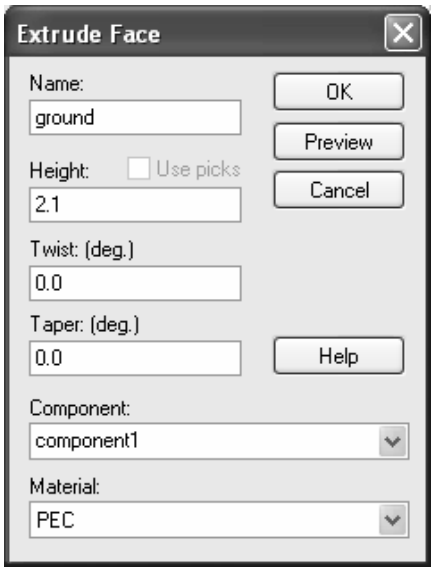


☐ Model the Ground Plane

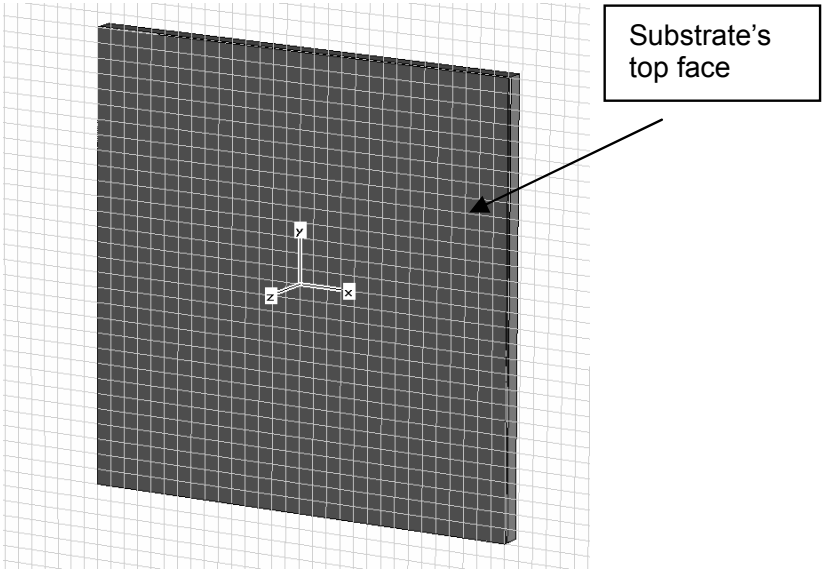
The next step is to model the ground plane of the patch antenna. Because the antenna will be excited by a coaxial feed at the bottom face, the electric boundary at *Zmin* defined by the previously chosen template is not suitable as a ground plane. Consequently, a metallic brick must be additionally defined. This can be easily achieved by activating the face pick tool (*Objects* ⇨ *Pick* ⇨ *Pick Face*,) and double-clicking on the substrate's bottom face. The model is rotated by activating the rotation mode *View* ⇨ *Mode* ⇨ *Rotate* ( ). The face selection should then be visualized as follows:



You can now extrude the selected face with the "Extrude" tool ( ). 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 modeled 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:

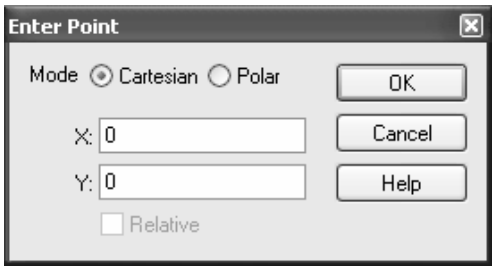


After entering a suitable name (e.g. “ground”) in the *Name* field and confirming your settings with the *OK* button, the current structure will look like this (rotated again to the see the top face):

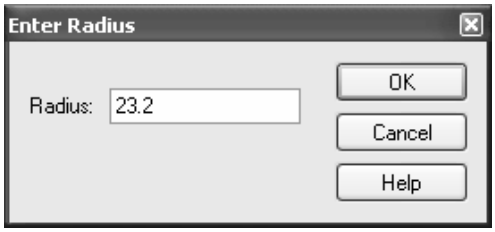


☐ Model the Patch Antenna

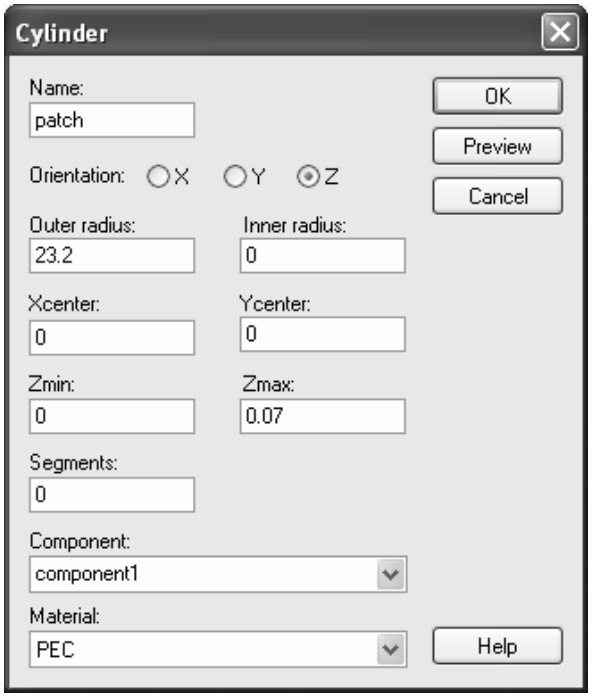
After the ground plane has been defined, the patch antenna must be modeled as a cylindrical shape on the substrate’s top face. Please activate the cylinder creation mode (*Objects ⇨ Basic Shapes ⇨ Cylinder,*). Similar to the construction of the substrate’s brick, enter the coordinates numerically by pressing the *Tab* key to open the following dialog box:



Here, enter the center 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:

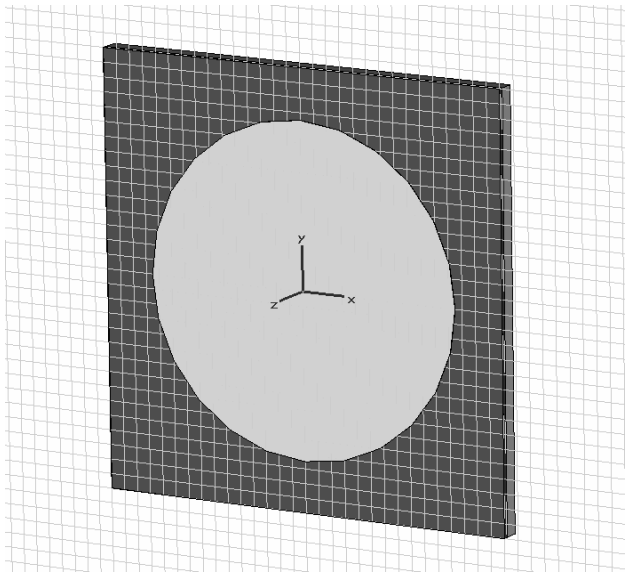


Skipping the entry dialog for the inner radius by pressing the *Esc* button will lead to the following dialog box that provides a summary of your entered parameters:



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 field. After you apply the settings with the *OK* button, your screen should show the following structure:



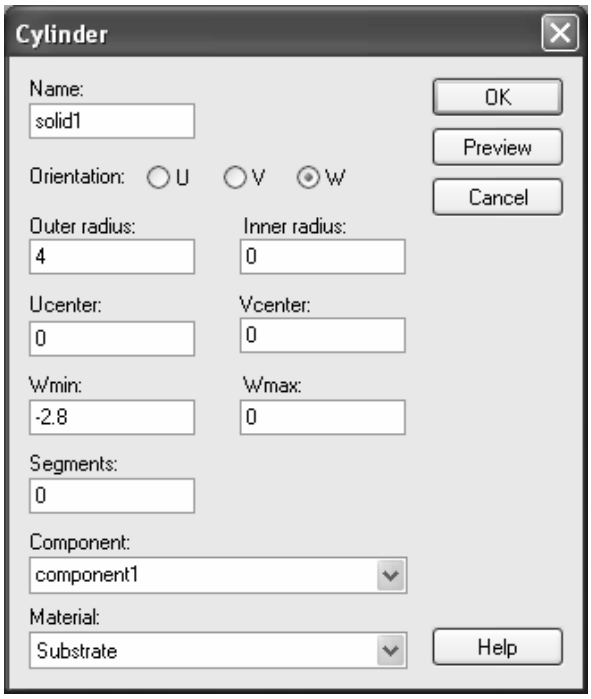
☐ Model the Coaxial Feed

The last modeling 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 (*WCS ⇨ Local Coordinate System*,).

To define the new center point for the coaxial feed the local coordinate system is moved along the positive v-direction (*WCS ⇨ Move Local Coordinates*,). Therefore, please enter a value of 9.2 mm in the following dialog box:

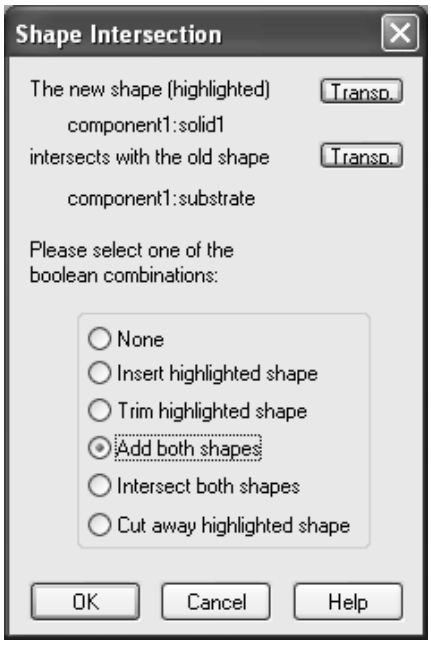
Now it is possible to design the coaxial feed by constructing two cylindrical shapes, similar to the previously defined circular patch.

Please activate the cylinder creation mode again (*Objects ⇨ Basic 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. Please check your settings against the following dialog box:

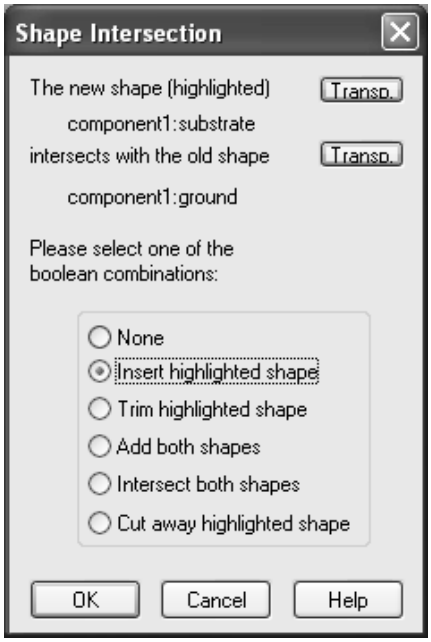


Select the previously defined "Substrate" material from the *Material* dropdown list and create the cylinder with the *OK* button.

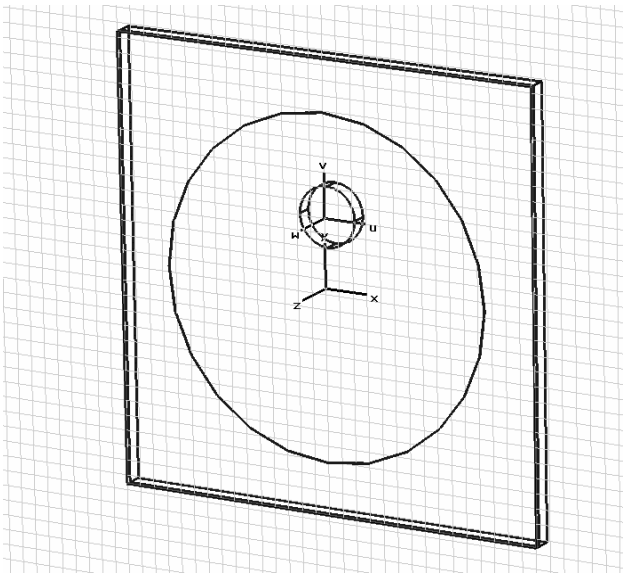
As a result, the cylinder shape component1:solid1 intersects with two already existing shapes, the solid "component1:substrate" and the ground plane "component1:ground". Here it is necessary to determine the type of intersection for the shapes. It is more convenient to combine the two substrate materials 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*:



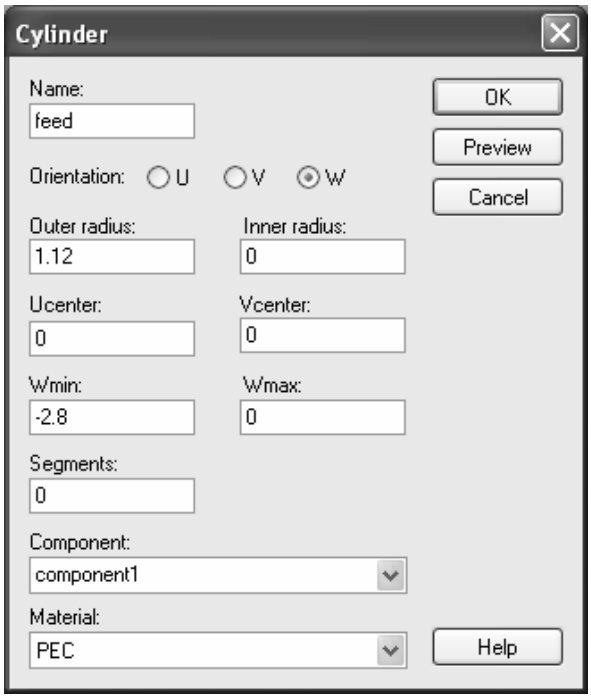
In the second case, the substrate cylinder must be inserted into the PEC material of the ground plane. Please mark the radio button *Insert highlighted shape* from the "Shape Intersection" dialog window as presented below and confirm again with *OK*:



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



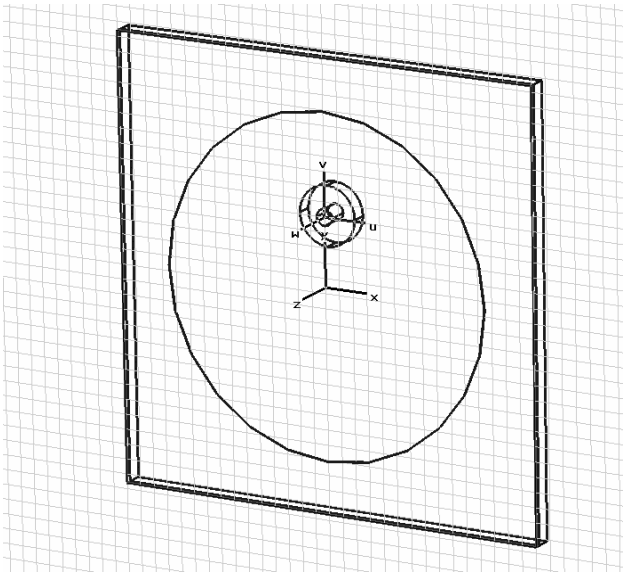
The inner conductor is constructed by defining another cylinder made of PEC material. Please define the cylinder with an outer radius of 1.12 mm and again an extension of 2.8 mm in the negative w-direction. The cylinder creation dialog should appear as follows:



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.

> **Please note:** 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. Refer to the *Getting Started* manual for more details.

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



Common Solver Settings

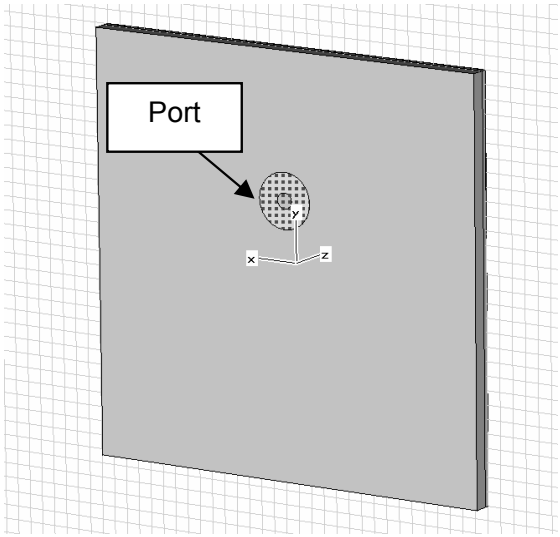
To this point, only the structure itself has been modeled. Now it is necessary to define some solver-specific elements. For an S-parameter calculation you must define input and output ports. Furthermore, the simulation needs to know how the calculation domain should be terminated at its bounds.

☐ Define the Waveguide Port

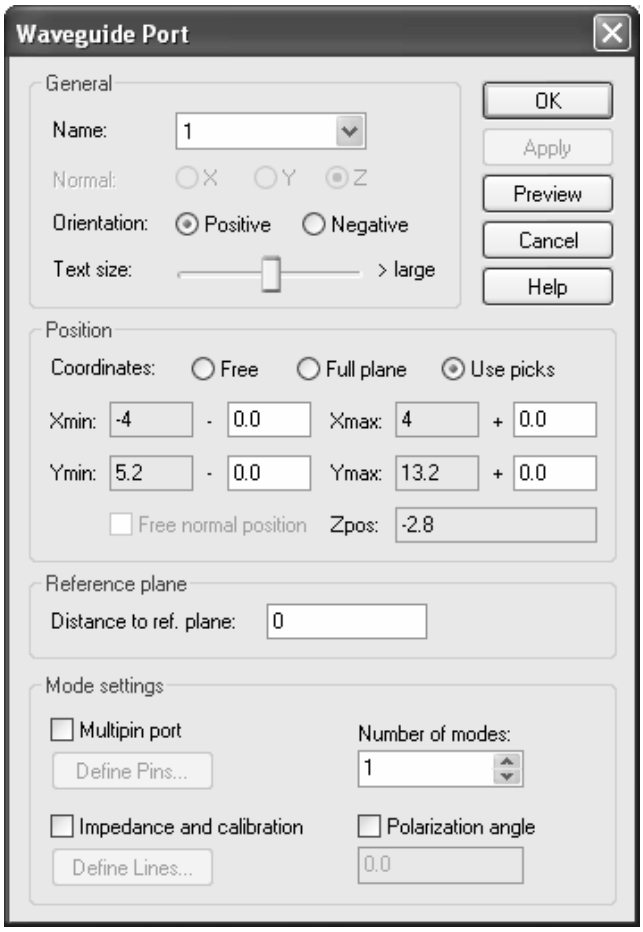
The next step is to add the excitation port to the patch antenna device, for which the reflection parameter will later be calculated. The port simulates an infinitely long coaxial waveguide structure that is connected to the structure at the port's plane.

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 (*Objects ⇨ Pick ⇨ Pick Face*,) of the coaxial feed ("Substrate" material) as demonstrated below (the model is rotated again to the bottom side first):

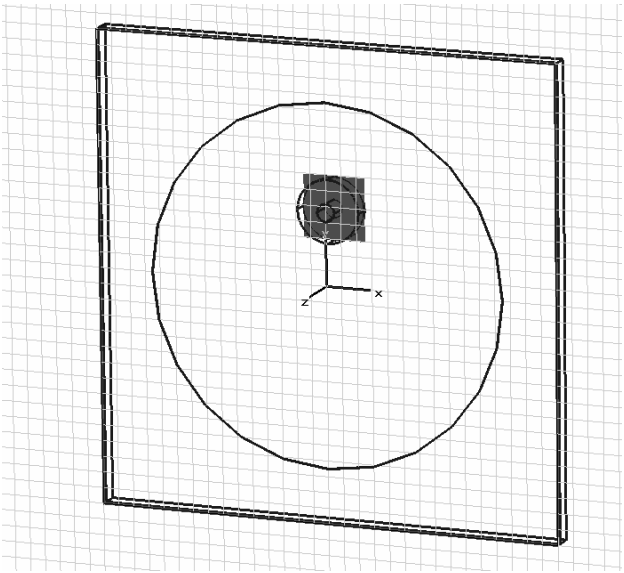


Please now open the waveguide dialog box (*Solve ⇨ Waveguide Ports*,) to define the port:



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.

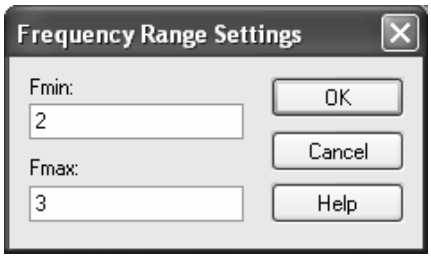
Please confirm your port settings with the *OK* button to finally create the port. After rotating the model again to the top face, your model should now look as follows (please use again *Ctrl+w* or  to toggle the wireframe visualization mode on and off):



☐ Define the Frequency Range

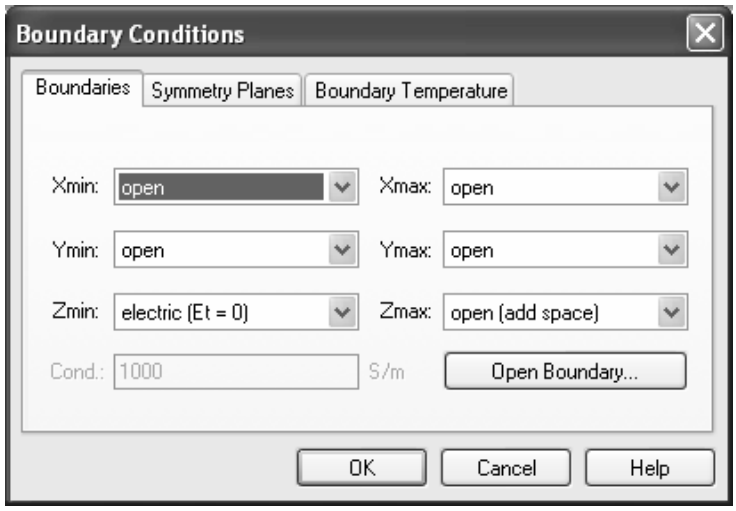
The frequency range for the simulation should be chosen with care. Different considerations must be made when using a transient solver or a frequency domain solver (see next chapter for details).

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 (*Solve* ⇨ *Frequency*,) and enter the range from 2 to 3 (GHz) before pressing the *OK* button (the frequency unit has previously been set to GHz by the selected template and is displayed in the status bar):

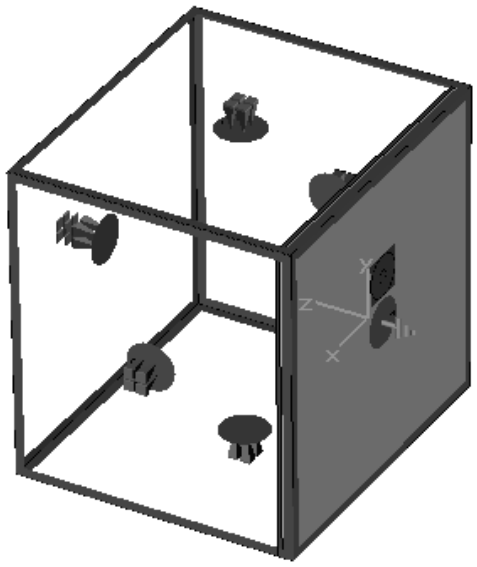


☐ Boundary Conditions

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 *Solve* ⇨ *Boundary Conditions* ( ):



When the dialog box opens, all currently selected boundary conditions are simultaneously displayed in the main view:



When you selected the “Antenna (on planar substrate)” template at the beginning of this tutorial the boundary conditions were already properly set for this structure. At the ground plane, an *electric* boundary condition has been 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.

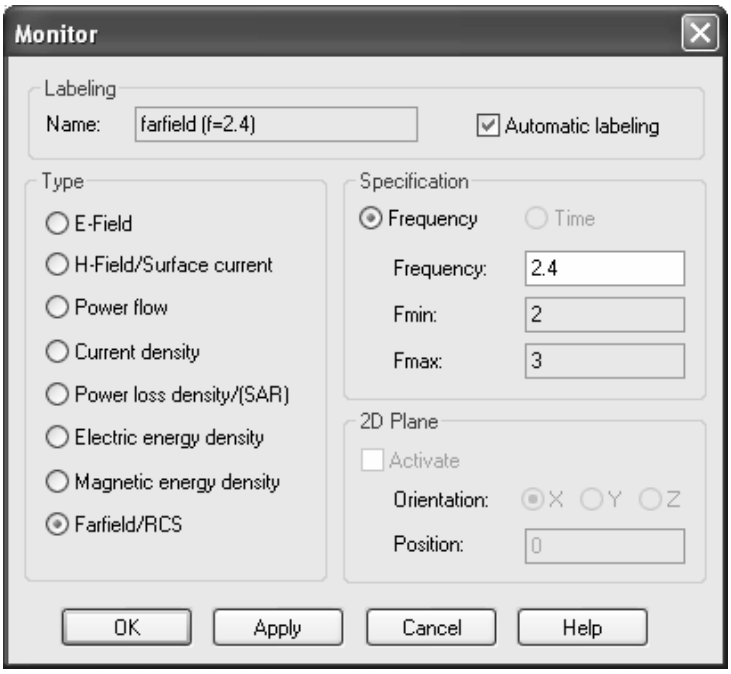
> **Please note:** As a general rule, the open boundary conditions work best if they are at least 1/8 wavelength apart from the field source. “Open (add space)” already incorporates this rule and automatically adds the correct amount of background space to the structure.
>
> Because the “open (add space)” 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 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 *Solve* ⇨ *Monitors* ( ):



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 monitor's 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 *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. We demonstrate this strength 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.

Please note that not all solvers may be available to you due to license restrictions. Please contact your sales office for more information.

Transient Solver

- Frequency Range Considerations for the Transient Solver

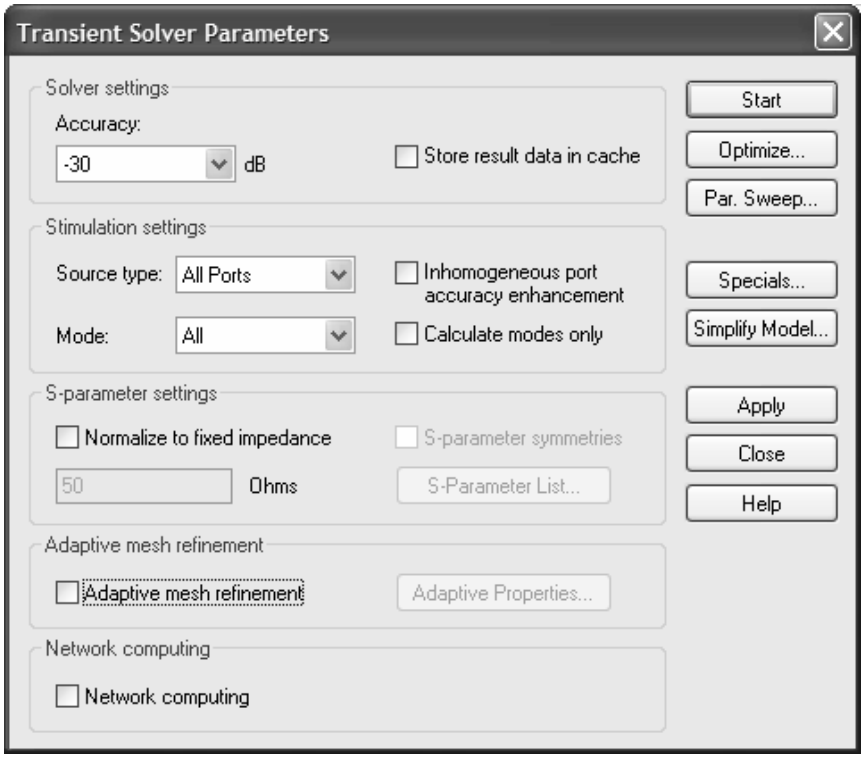
We recommend using reasonably large bandwidths of 20% to 100% for the transient simulation. In this example, the S-parameters are to be calculated for a frequency range between 2 and 3 GHz. With a center frequency of 2.5 GHz, the bandwidth (3 GHz – 2 GHz = 1 GHz) is 40% of the center frequency, which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

Please note: 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.

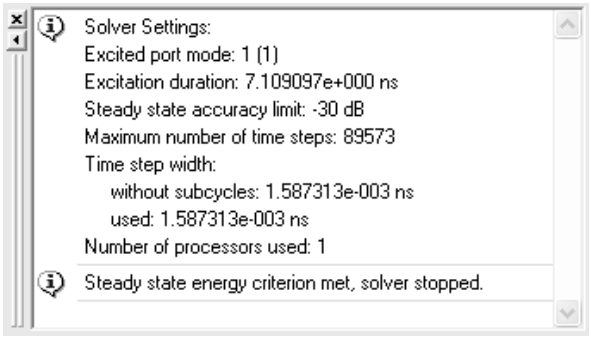
☐ Transient Solver Settings

The solver's parameters are specified in the "Transient Solver Parameters" dialog box that can be opened by selecting *Solve* ⇨ *Transient Solver* from the main menu or pressing the corresponding icon ( ) in the toolbar:



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 display additional information:



Please note: If there are any warning or error messages during the simulation they will be written into the message window, as well.

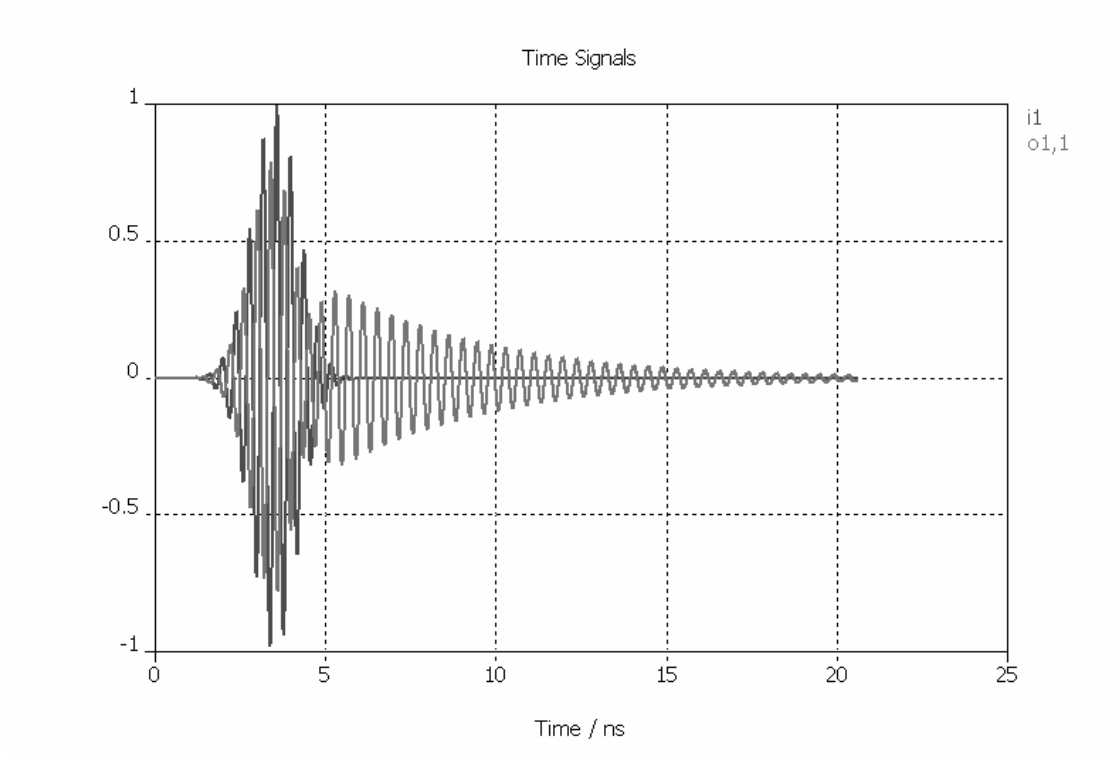
Transient Solver Results

Congratulations, you have simulated the circular patch antenna using the transient solver! Let's review the 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.

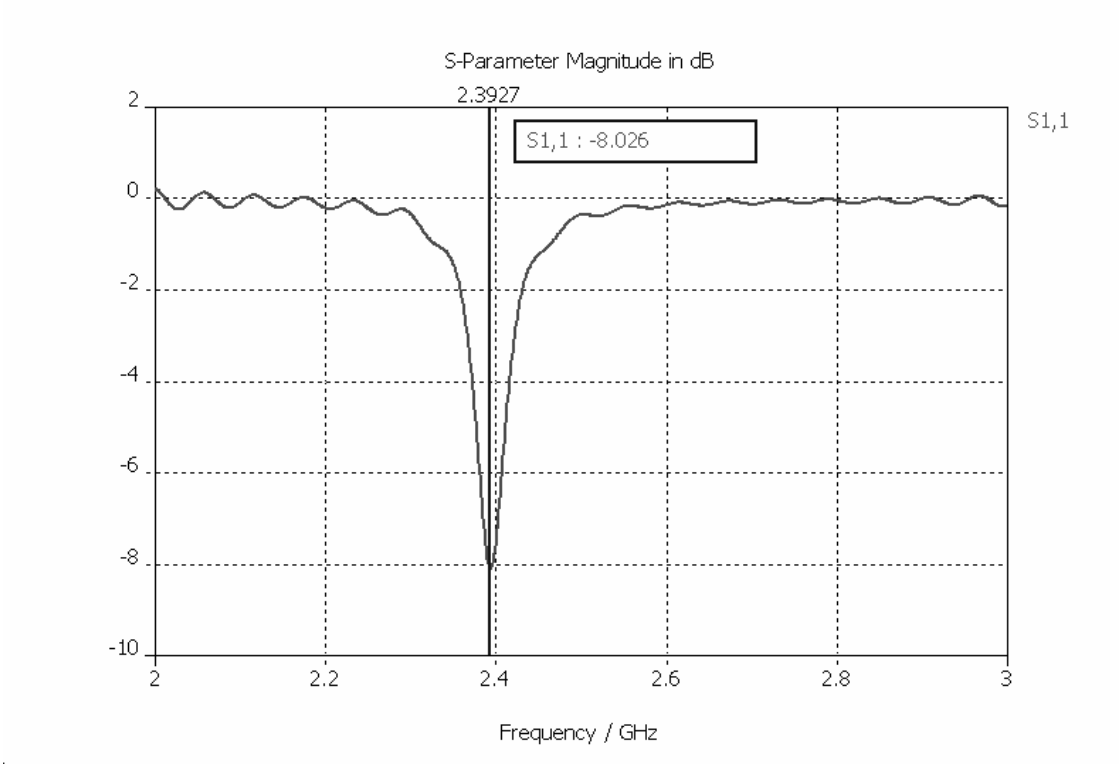
Please note: It is possible to observe the progress of the results during the computation. In order to get the complete information, however, wait until the solver has finished.



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| dB* folder from the navigation tree. The following screenshot shows the reflection parameter:

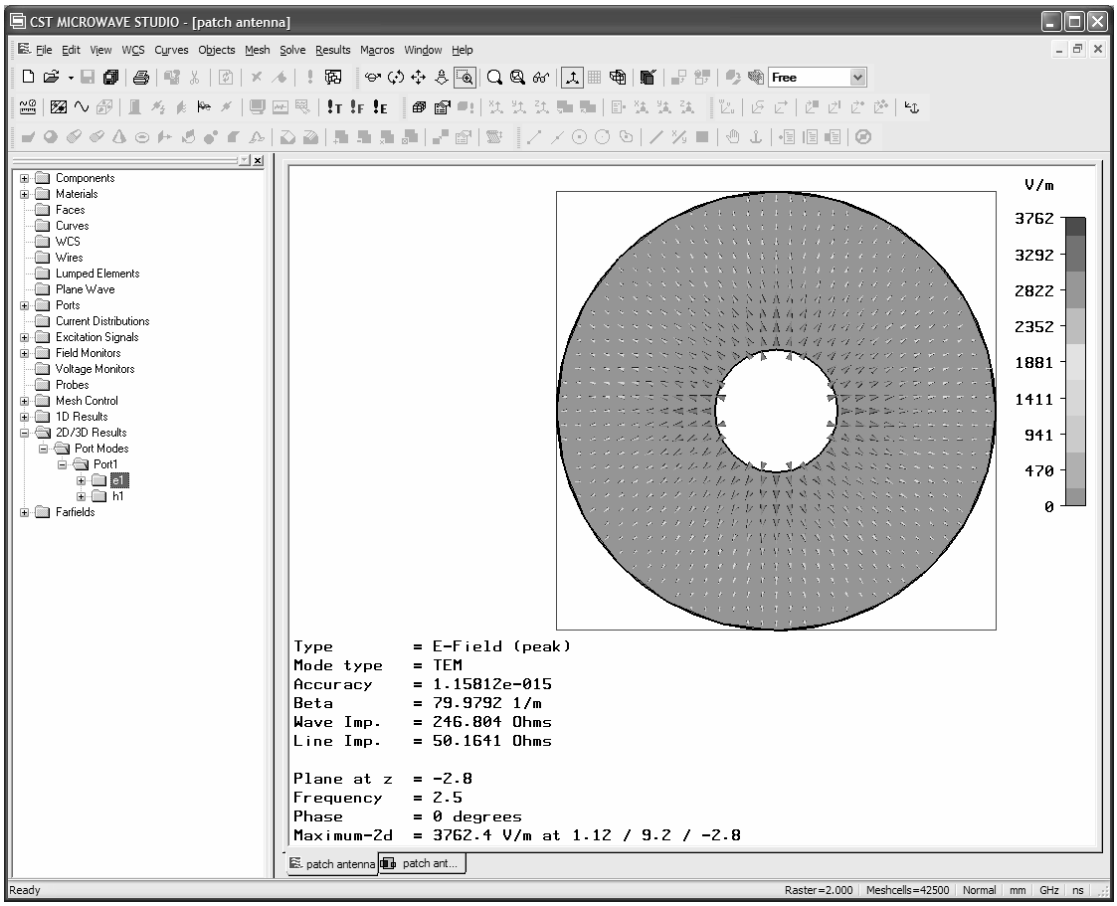


It is possible to precisely determine the operational frequency for the patch antenna. Activate the axis marker by pressing the right mouse button and selecting the *Show Axis Marker* option from the context menu. Now you can move the marker to the S11 minimum and pinpoint a resonance frequency for the 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. More information about this type of numerical error is available in the “Accuracy Considerations” chapter.

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

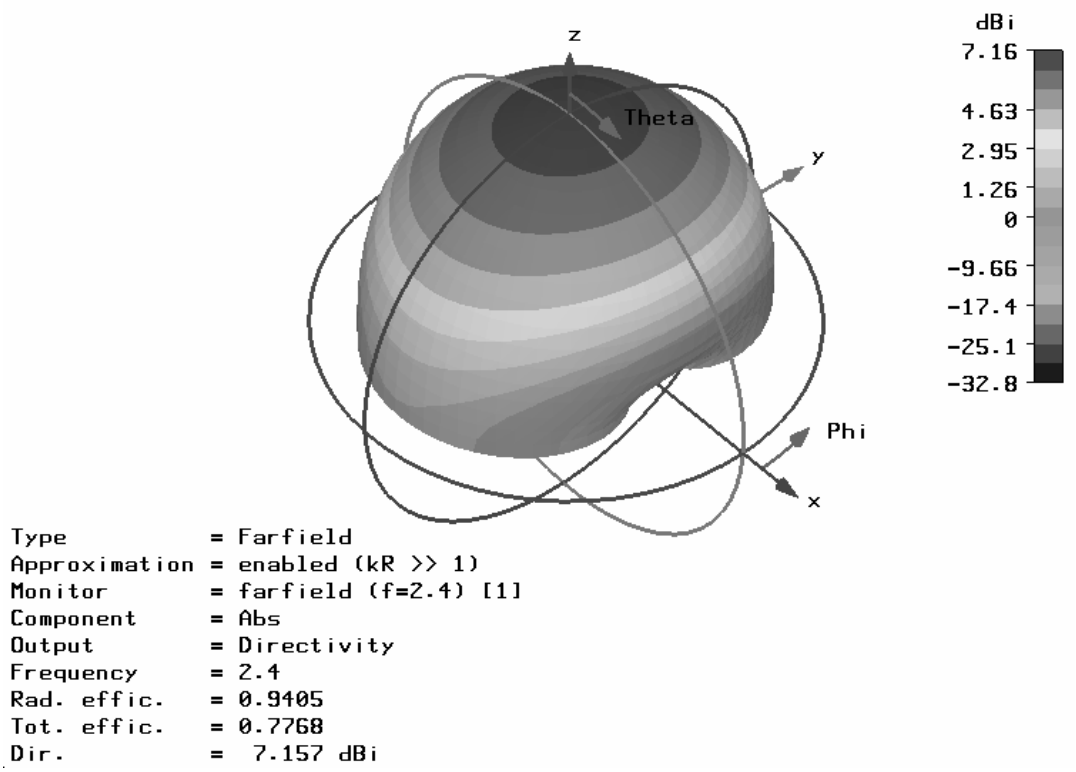
You should first inspect the port modes that can be easily displayed by opening the *2D/3D Results ⇨ Port Modes ⇨ Port1* folder from the navigation tree. To visualize the electric field of the fundamental port mode, click on the *e1* folder. After properly rotating the view and tuning some settings in the plot properties dialog box, you should obtain a plot similar to the following picture (please refer to the *Getting Started* manual for more information on how to change the plot’s parameters):



The plot also shows some important properties of the coaxial mode such as TEM 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:



Please note: You have the option to change the *Results* ⇨ *Plot Properties* ⇨ *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.

In the following section, we provide hints how to minimize these errors and achieve highly accurate results.

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 –30 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 –40 dB or –50 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:

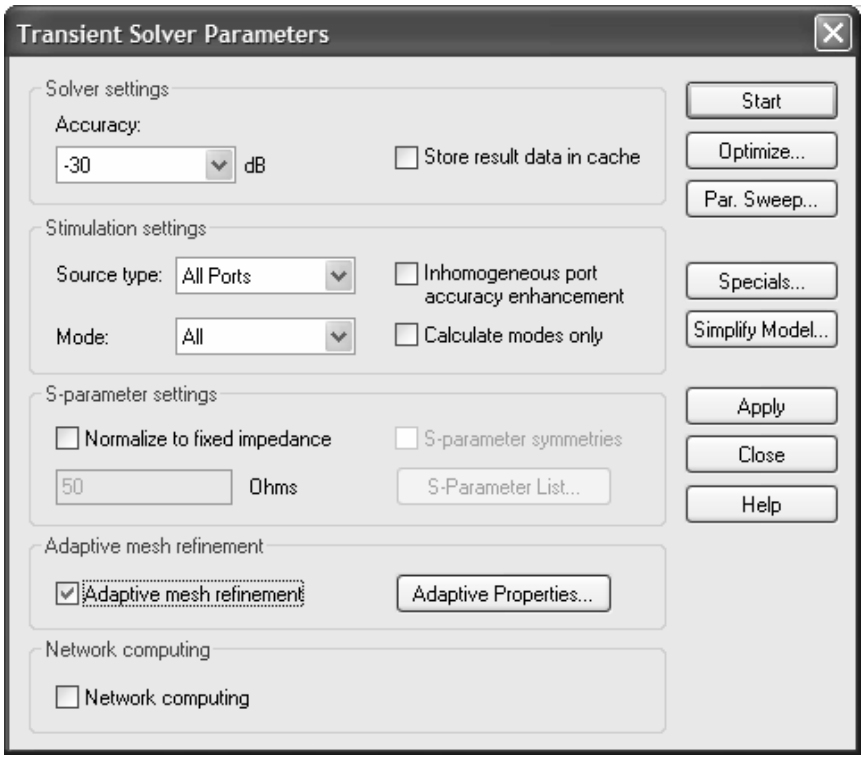
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

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

2. Effect of the Mesh Resolution on the S-parameter's 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.

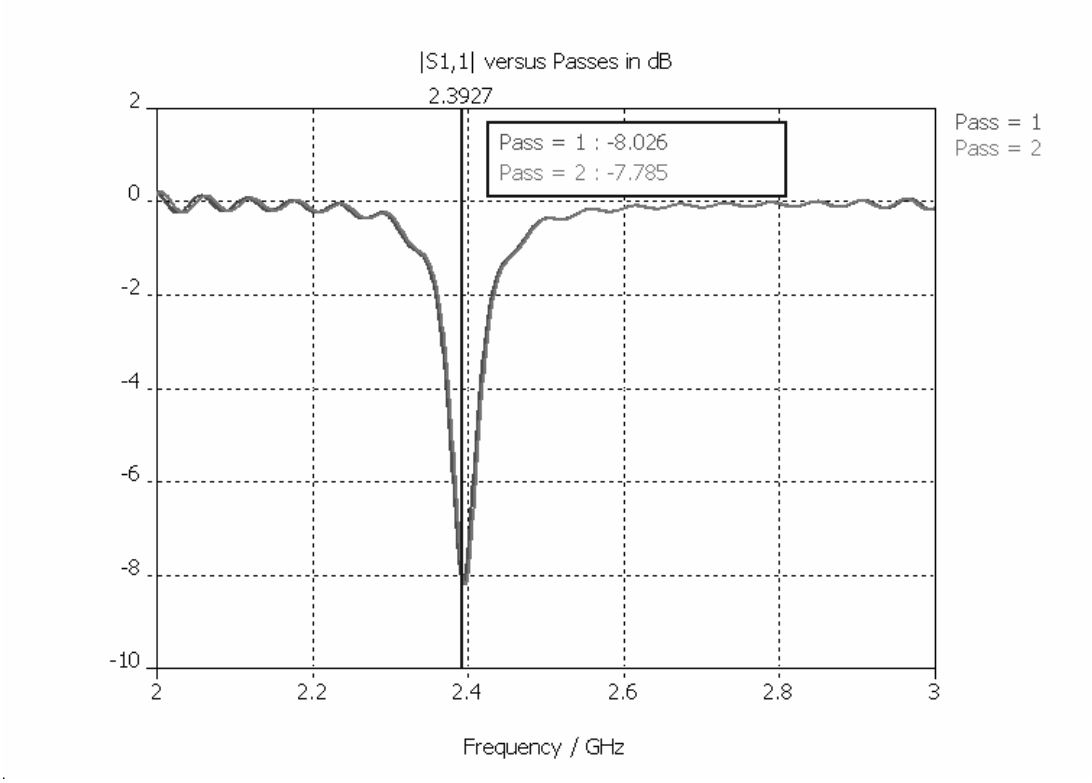
In the example above, you have used the default mesh that has been automatically generated by an expert system. The accuracy of the results is most easily tested with the full automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (*Solve ⇨ Transient 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. Thus, you only have to activate the *Adaptive mesh refinement* tool in the "Transient Solver Parameters" dialog and start the solver again by pressing the *Start* button.

In this example, only one adaptation pass is necessary to obtain a suitable result. This means that the maximum deviation of the S-parameters between the first and the second runs is less than 2%. Obviously, in this example the mesh created by the expert system was already quite acceptable for an accurate solution.

The convergence process of the input reflection S1,1 during the mesh adaptation can be visualized by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *|S| dB* ⇒ *S1,1* from the navigation tree:



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.

> **Please note:** Refer to the *Getting Started* manual for more information on using *Template Based Postprocessing* for automated extraction and visualization of arbitrary results from various simulation runs.

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

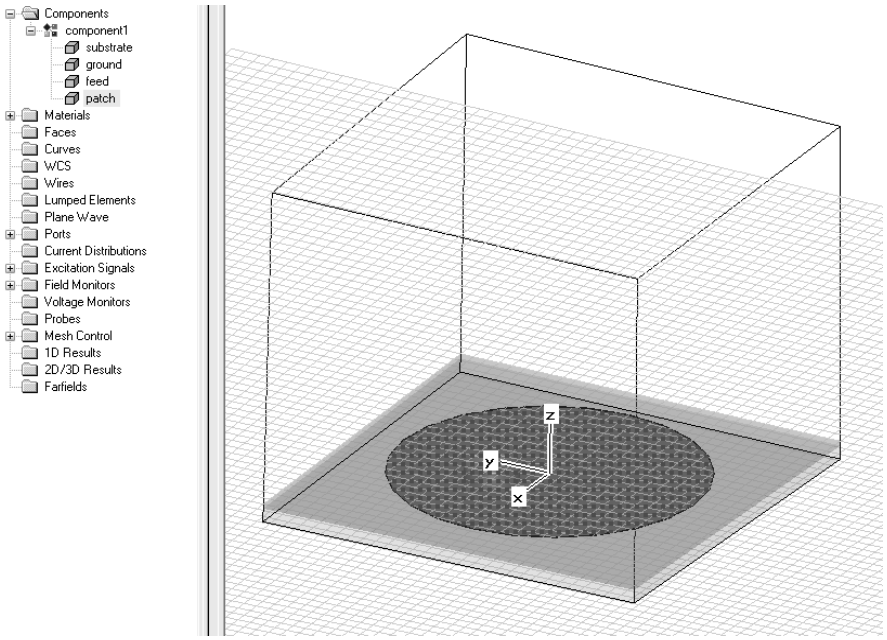
Before performing a simulation with a frequency domain solver, you may want to keep the results of the transient solver to allow for an easy comparison of the two simulations. To obtain the copy of the current results: Select, for example, the |*S*| *dB* folder in *1D Results*, then press *Ctrl+c* and *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 *Add 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.

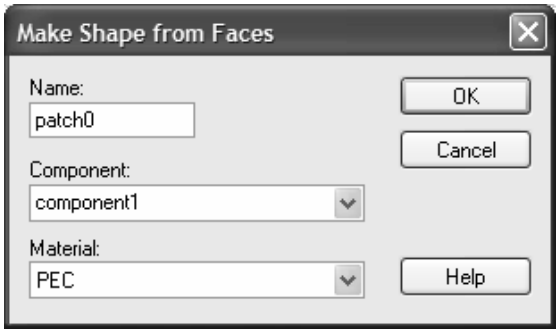
☐ Optimize Structure for Tetrahedral Mesh

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, we rebuild the patch 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 (*Objects ⇨ Pick ⇨ Pick Face*, ■):

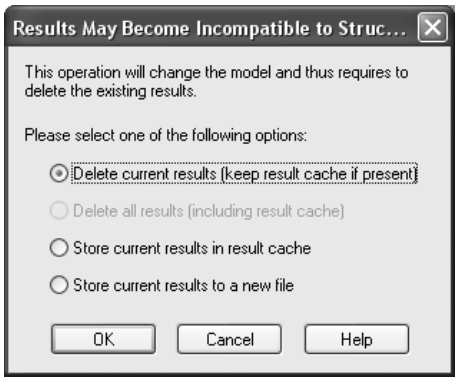


A PEC sheet is easily created from the selected face by applying *Objects ⇨ Face Healing Tools ⇨ Shape from Picked Faces,*):

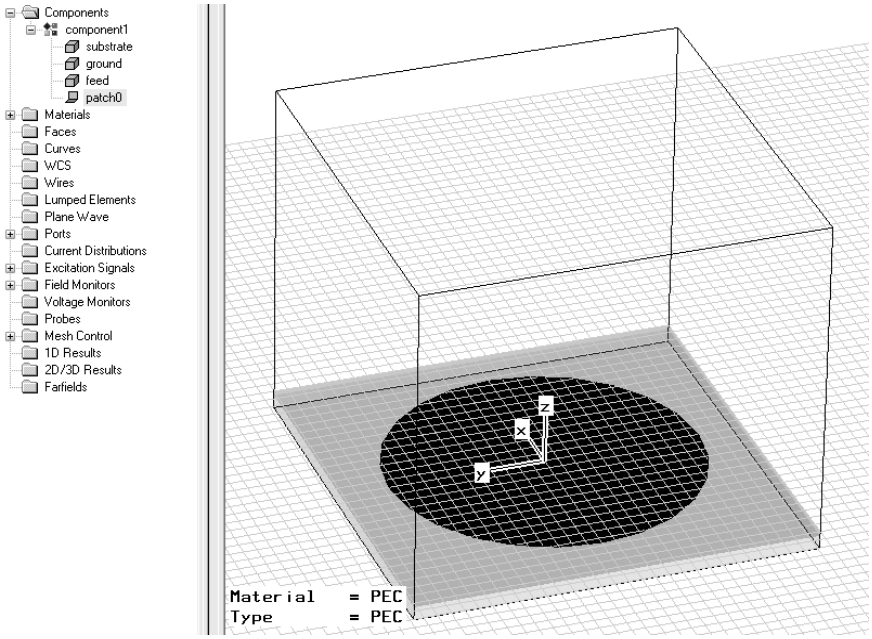


Enter a suitable name for the new shape (*patch0*) and confirm the creation by pressing the *OK* button. Finally, delete the old patch (*component1 ⇨ patch*) so that only the newly created patch with zero height (*component1 ⇨ patch0*) remains.

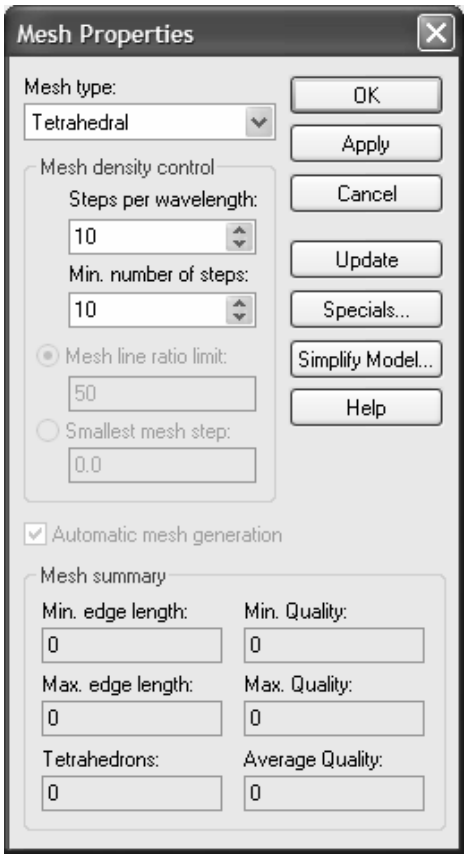
There may be old results present from the previous transient solver run that will be overwritten when changing the model. In this case, the following warning message appears:



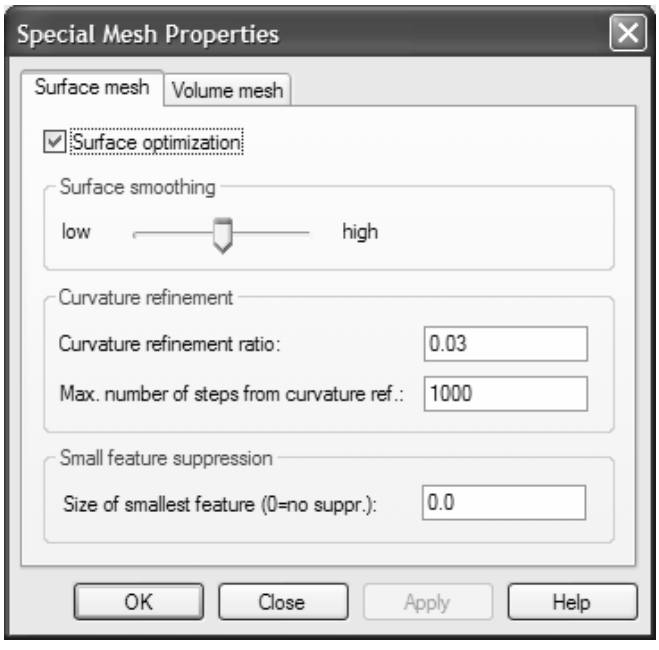
Press *OK* to acknowledge deletion of the previous results.



In order to allow a tetrahedral-based calculation, we change the *Mesh type* from hexahedral to tetrahedral in the "Mesh Properties" dialog box (*Mesh* ⇨ *Global Mesh Properties*,). This selection can also be performed directly in the "Frequency Domain Solver Parameters" dialog box when choosing the desired solver type, as demonstrated in the following chapter. Furthermore, regarding the circular geometry of the patch antenna and its coaxial feed, it is advisable to refine the default mesh settings. This ensures a homogenous size of the tetrahedra before starting the mesh adaptation. Therefore, please increase the *Steps per wavelength* to a value of 10 in the "Mesh Properties" dialog box:



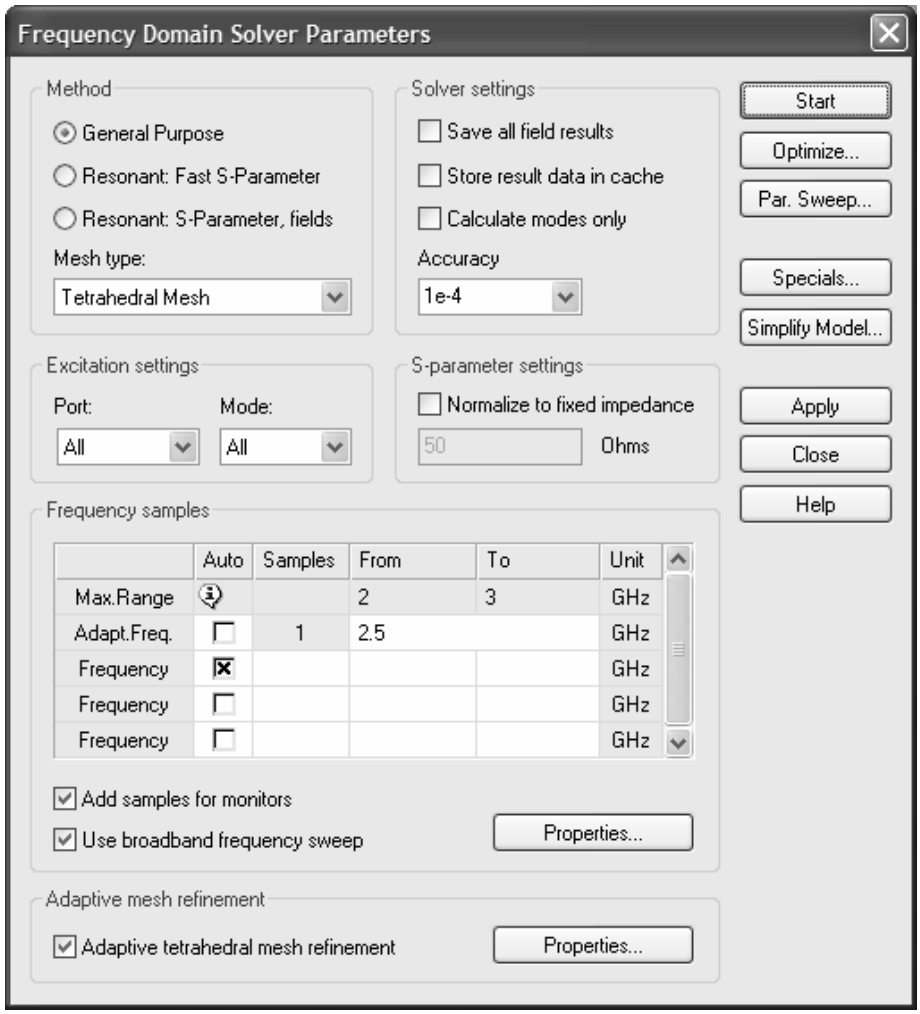
Afterwards, open the "Special Mesh Properties" dialog box by pressing the *Specials* button. Please reduce the *Curvature refinement ratio* to 0.03 and increase the *Maximal number of steps from curvature refinement* to 1000 to allow a better starting approximation of the circular elements. Press the *Help* button to obtain more information on the settings.



Please check the settings and confirm them by clicking the *OK* buttons in both dialog boxes.

☐ Frequency Domain Solver Settings

The “Frequency Domain Solver Parameters” dialog box is opened by selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu or by pressing the corresponding icon in the toolbar.

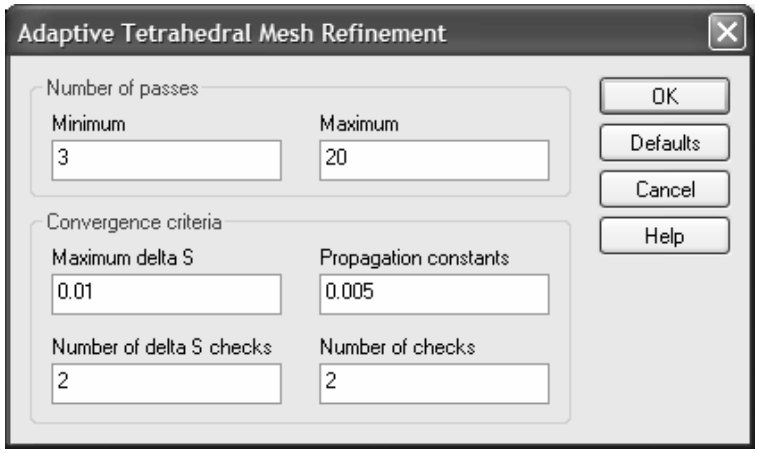


There are three different methods to choose from. For the example here, please choose the *General Purpose* frequency domain solver. In the *Mesh Type* combo box you may choose between *Hexahedral* and *Tetrahedral Mesh*. Due to the previously made settings in the “Mesh Properties” dialog box, the *Tetrahedral Mesh* is already selected.

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 check box is activated by default.

Please note that it is necessary to choose a suitable *Adaptation Frequency*, correspondent to the assumed radiation behavior of the antenna. A good choice for an antenna structure is usually the center frequency of the calculation range, so in this example we enter the value of 2.5 GHz in the dialog box (see picture above). Furthermore, to ensure that the adaptation will satisfy the desired accuracy limit, we increase the *Maximum number of passes* to 20 in the "Adaptive Tetrahedral Mesh Refinement" dialog box 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 status bar, displaying some information about the solver stages:

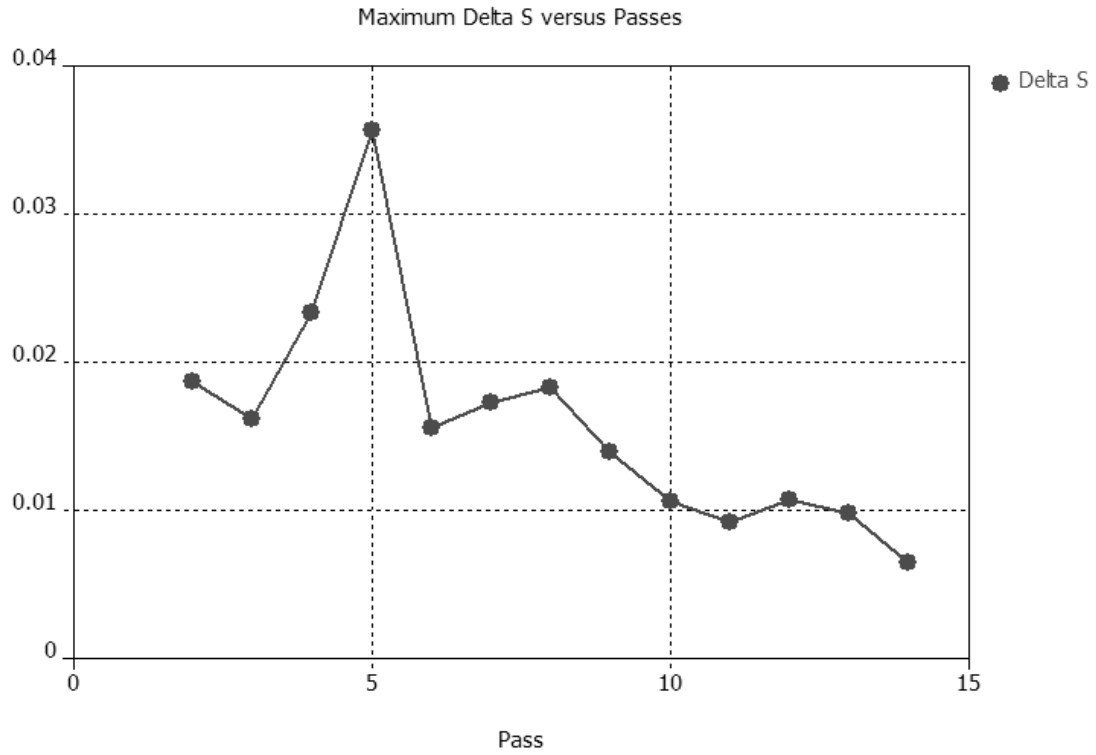
After the desired accuracy for the S-parameter has been reached, the simulation stops. When the simulation has finished or has been aborted, both items disappear again. During the simulation, the "Message Window" will display details about the performed simulation.

Frequency Domain Solver Results

Congratulations, you have simulated the patch antenna using the general purpose tetrahedral frequency domain solver! Let's review the results.

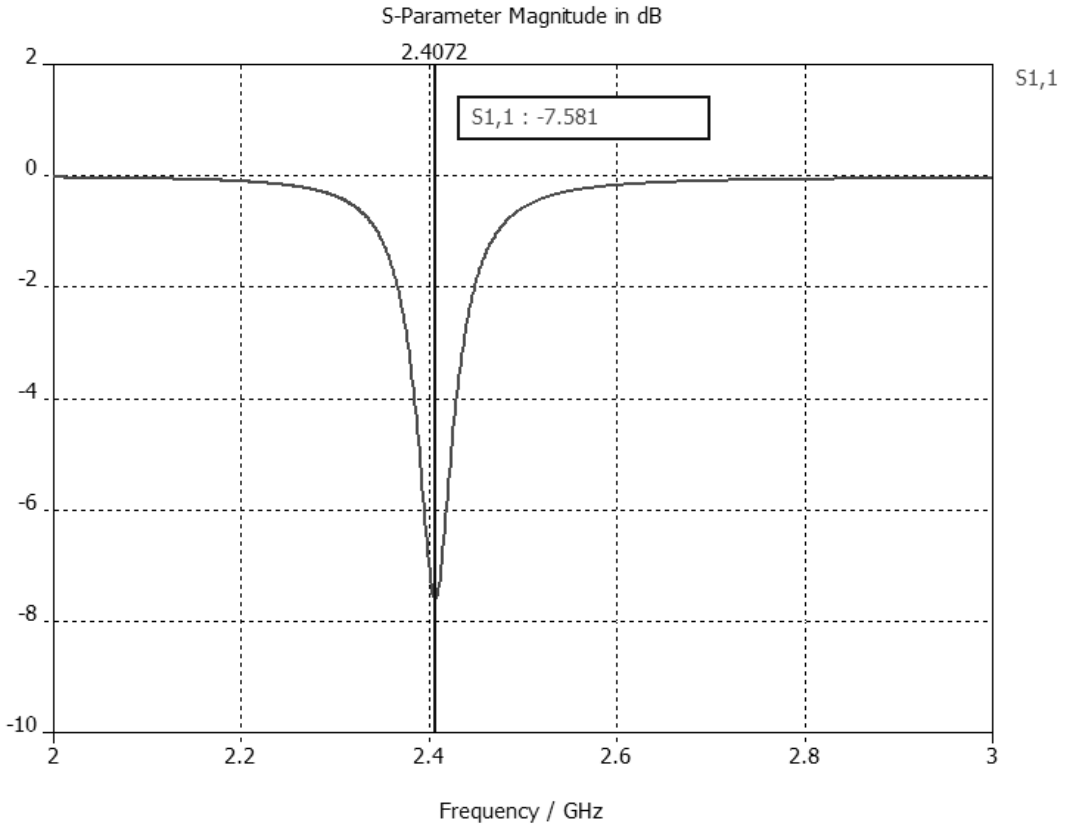
- 1D Results (S-Parameters)

You can visualize the maximum difference of the S-parameters for two subsequent passes by selecting *1D Results* ⇨ *Adaptive Meshing* ⇨ *Delta S* from the navigation tree:



As evident from the above diagram, several passes of the mesh refinement were required to obtain highly accurate results within the given accuracy level that is set to 1% by default.

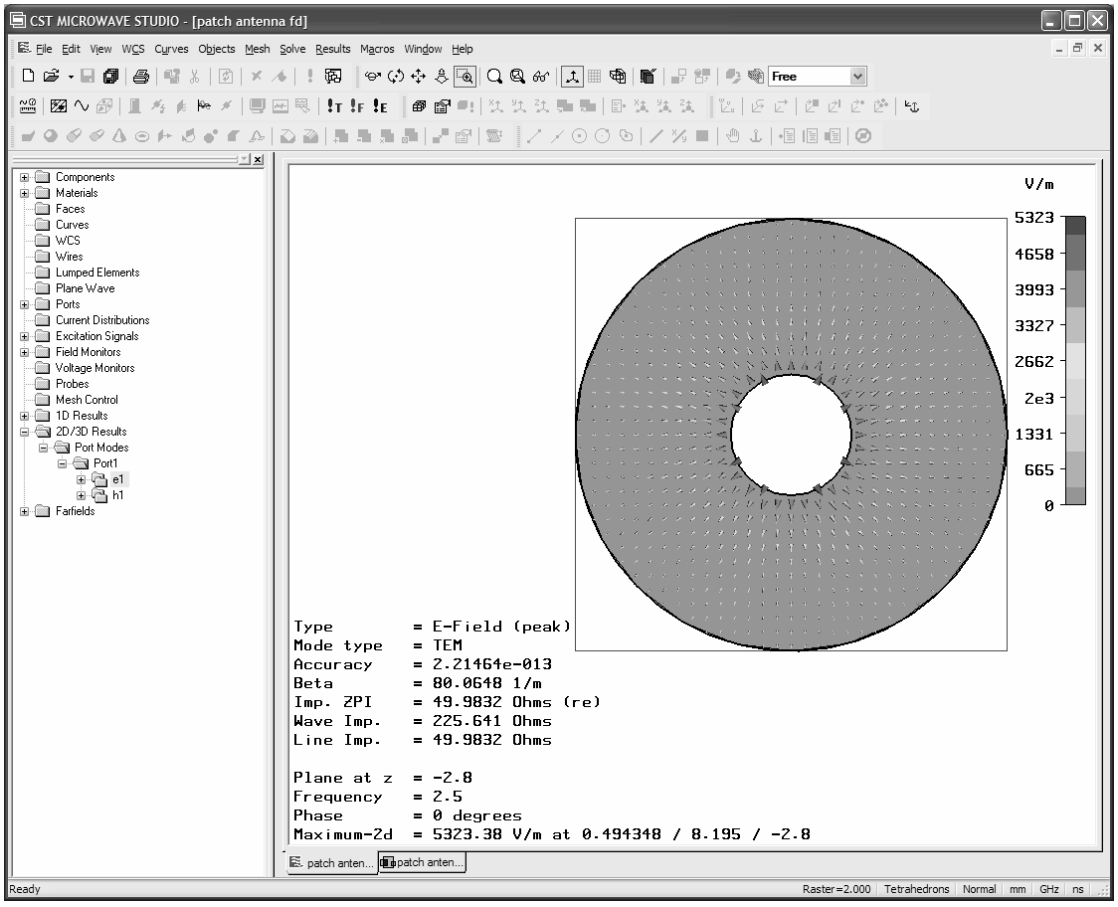
You can view the S-parameters magnitude in dB by selecting *1D Results* ⇨ |*S*| *db* in the navigation tree:



It is possible to precisely determine the operational frequency for the patch antenna. Activate the axis marker by pressing the right mouse button within the main window 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.

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

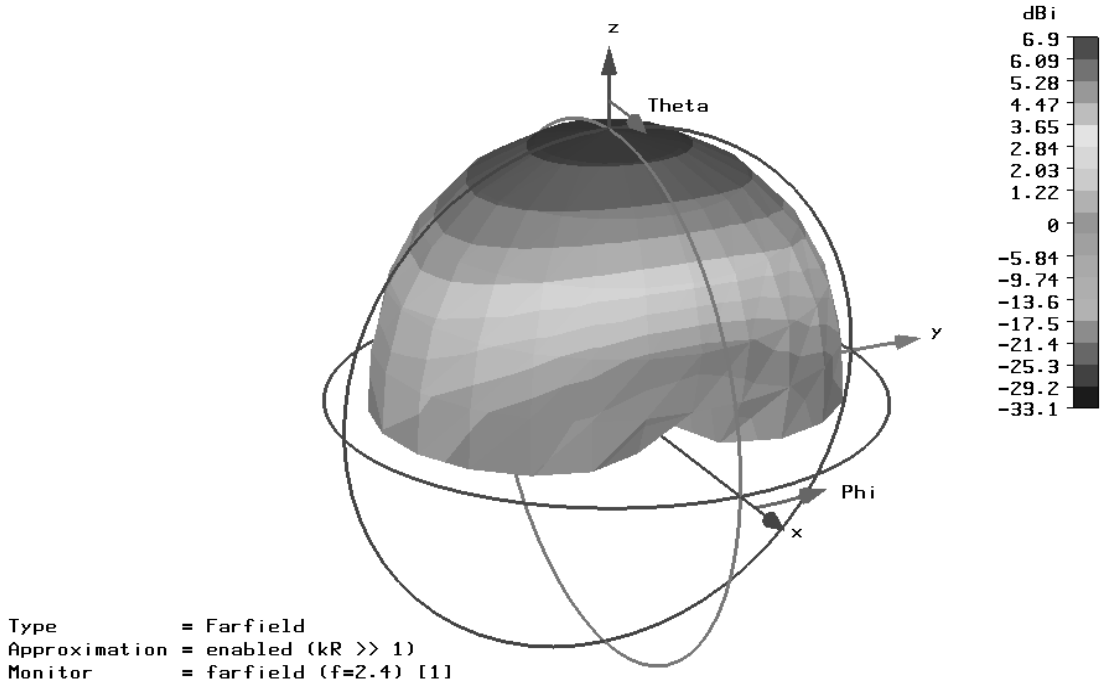
Finally, you can observe the 2D and 3D field results. You should first inspect the port modes that can be easily displayed by opening the *2D/3D Results ⇨ Port Modes ⇨ Port1* folder from the navigation tree. To visualize the electric field of the port mode, please click on the *e1* folder. Open the "Select Port Mode" dialog box by selecting *Results ⇨ Select Mode Frequency* from the main menu and changing the frequency to 2.5 GHz. Please confirm your setting by pressing *OK*. 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 (please refer to the *Getting Started* manual for more information on how to change the plot's parameters):



The plot also shows some important properties of the mode such as mode type, propagation constant and line impedance. The port mode at the second port can be visualized in the same manner.

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 position of the color ramp can be modified by clicking on the *Results ⇨ Plot Properties ⇨ Color ramp* button):




```
Type          = Farfield
Approximation = enabled (kR >> 1)
Monitor       = farfield (f=2.4) [1]
Component     = Abs
Output        = Directivity
Frequency     = 2.4
Rad. effic.   =  1.002
Tot. effic.   = 0.7940
Dir.          =  6.901 dBi
```

Please note: You have the option of changing the *Results ⇨ Plot Properties ⇨ Step* to 5 degrees for a better angle accuracy of the plot.

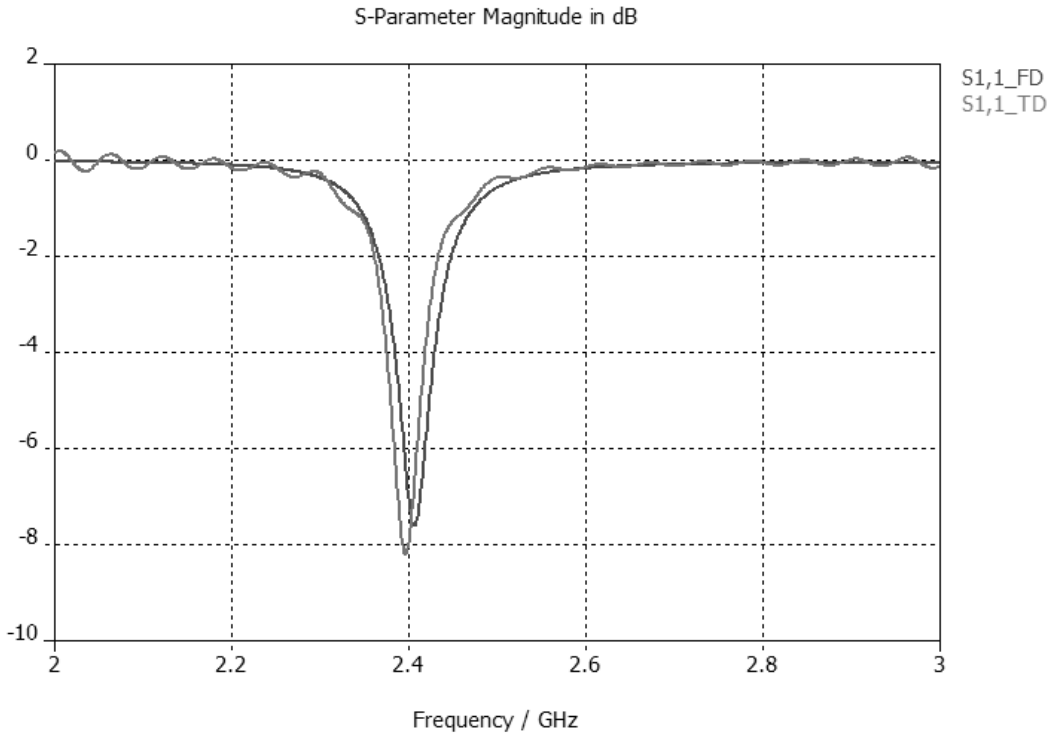
You can see that the maximum power is radiated in the positive z-direction. Note that there are several other options available to plot a farfield: the Polar plot, the Cartesian plot and the 2D plot.

☐ Comparison of the Solver Results

The following table shows the number of hexahedral mesh cells or tetrahedra and the simulation time for the transient and frequency domain simulations, respectively, presented in the previous chapters (the mesh resolution refers to the last adaptation run; the simulation time covers the complete simulation process including all performed adaptation passes):

	Transient Solver	Frequency Domain Solver
Mesh	73372 hexahedral cells	42953 tetrahedrons
Simulation time	13 min. 48 sec.	19 min. 07 sec.

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



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 behavior 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, we will concentrate purely on the farfield calculations. All other results can be analyzed in a manner similar to the first part of this tutorial.

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.

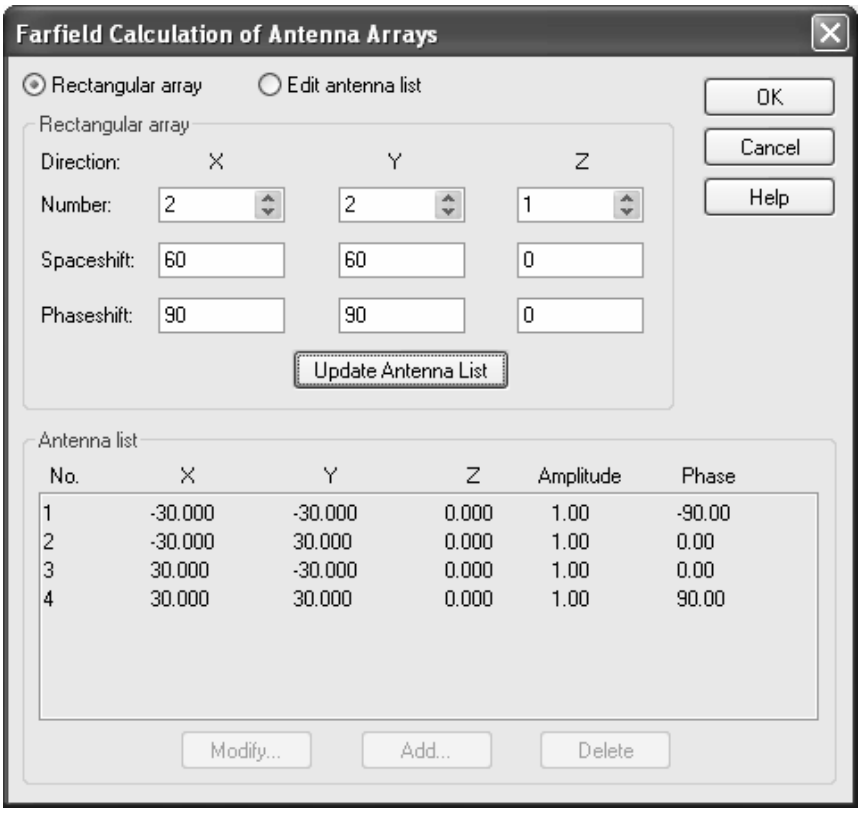
The array calculation will be done in three steps: First, the antenna array feature is applied to the farfield results of the single patch antenna. Afterwards, the structure is physically expanded to the 2x2 antenna array and we use a result combination of a sequential excitation as well as a simultaneous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

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, we will expand the antenna example by constructing a 2x2 antenna array pattern, 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 and open the corresponding dialog box by pressing *Results ⇨ Plot Properties [Array]* and selecting the radio button *Antenna array.*

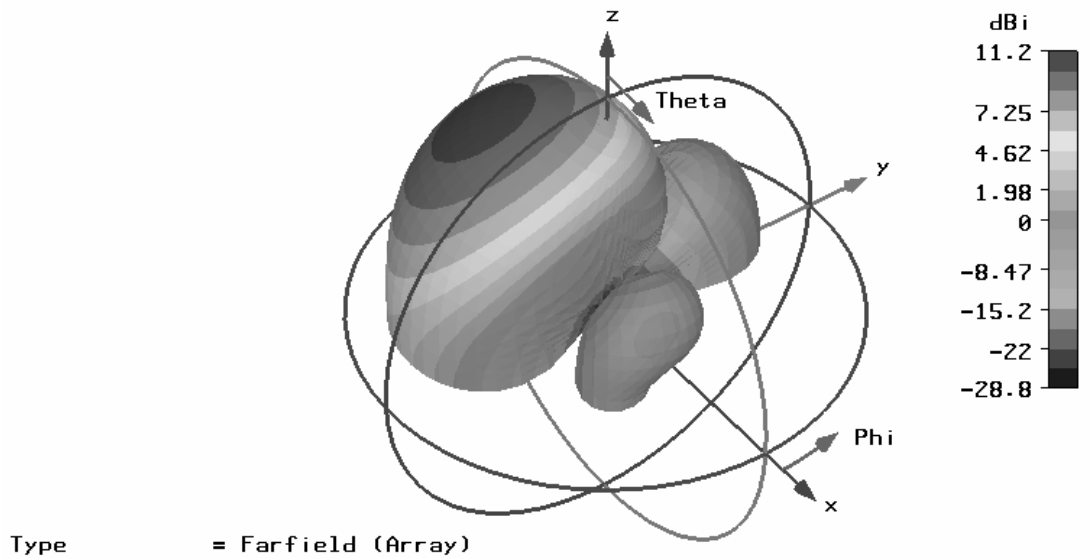
Afterwards, select *Properties* and insert the values to create the rectangular 2x2 array pattern in the XY-plane with a spatial shift of 60 mm (due to the dimension of the antenna substrate) and a phase shift of +90 degrees:



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.

> **Please note:** 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:




```
Type          = Farfield (Array)
Approximation = enabled (kR >> 1)
Monitor       = farfield (f=2.4) [1]
Component     = Abs
Output        = Directivity
Frequency     = 2.4
Rad. effic.   = 0.9405
Tot. effic.   = 0.7768
Dir.          =  11.21 dBi
```

Please feel free to define some more array patterns to analyze 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 (*Results ⇨ Plot Properties*) back to *Single antenna*.

Geometric Construction Steps

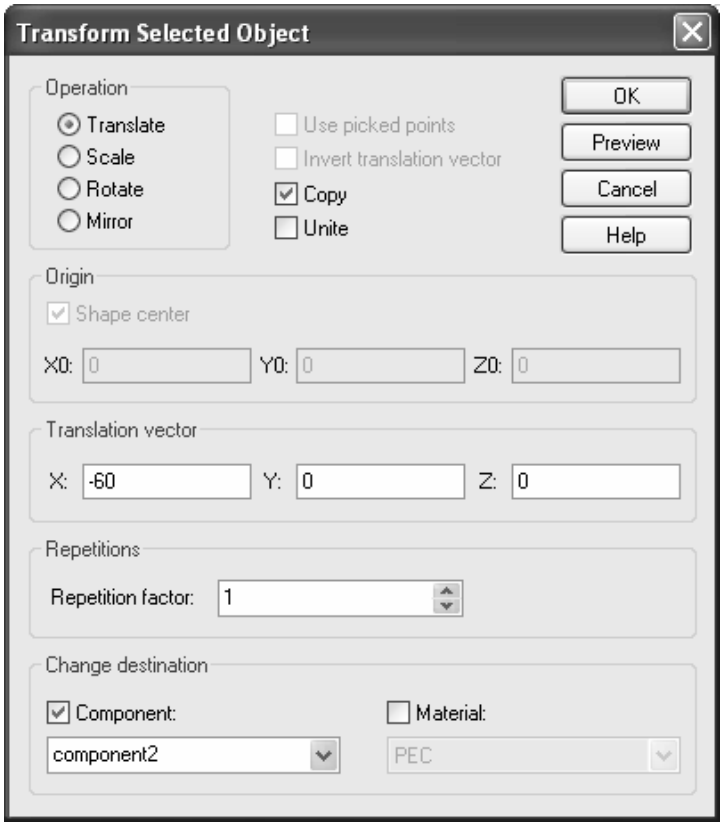
The construction of the array is based on the translation feature for selected objects that will be applied to the complete component of the single patch antenna. This procedure is also applicable to single objects or an arbitrary multiple selection of objects, even when selected from different components.

Before you start, please switch off the local coordinate system, if necessary, by clicking *WCS ⇨ Local Coordinate System* or the icon .

Now, please select “component1” from the “Components” folder in the navigation tree and perform a transformation of the complete component (press the toolbar icon or enter the *Objects ⇨ Transform* menu). 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 to create the destination group “component2”.

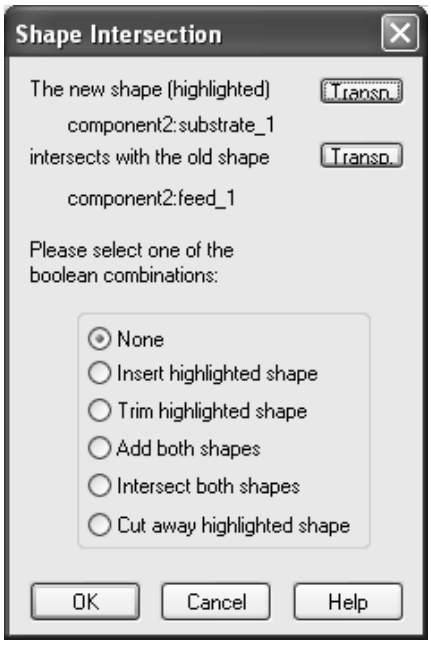
Finally, perform the translation operation with the *OK* button:



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*.

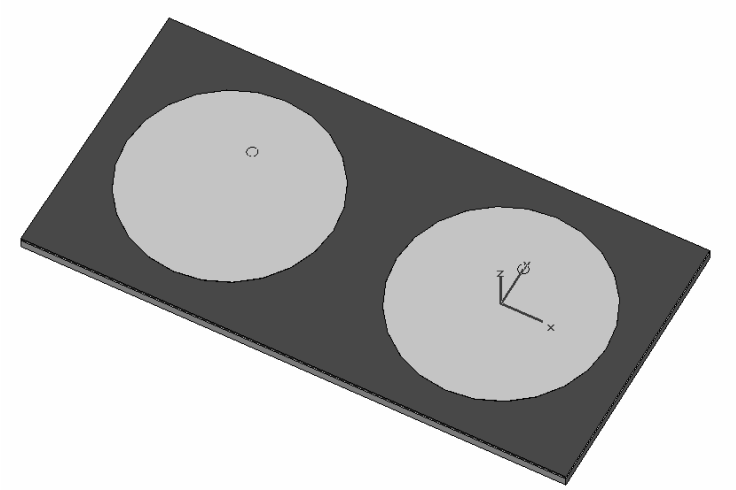
Now, CST MICROWAVE STUDIO® copies the selected component to the coordinates of its translated position and creates a new component, containing all translated single shapes of the patch antenna.

The “Shape Intersection” dialog box that appears to inform you about the intersection of the solids “component2:substrate_1” and “component2:feed_1” can be skipped with the default setting (activated *None* button):

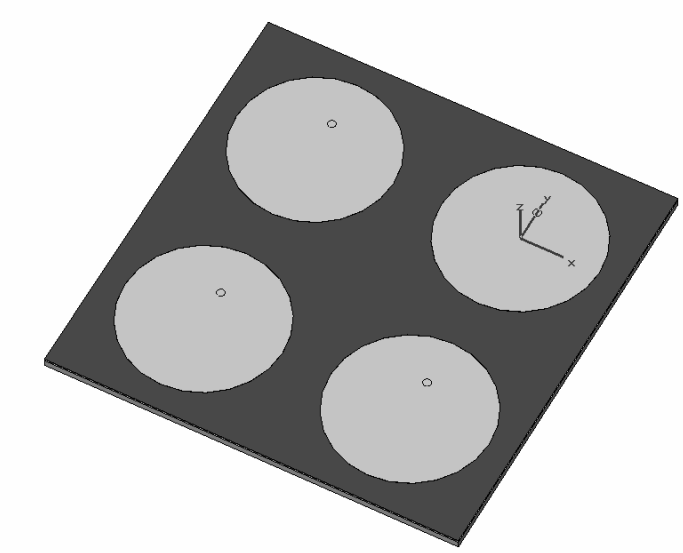


Please note: In this case, only one "Shape intersection" dialog window will appear because the PEC shapes are created ***after*** the shape with normal material (here: "Substrate"). This implies that the PEC shapes are *automatically inserted* into the intersected shape. Refer to the *Getting Started* manual for more details.

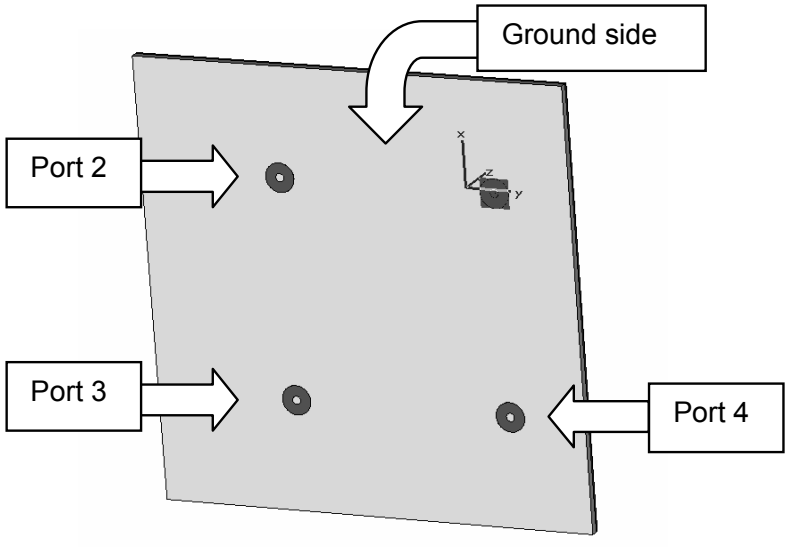
The resulting structure, consisting of two identical patch antennas, will then look as follows:



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.

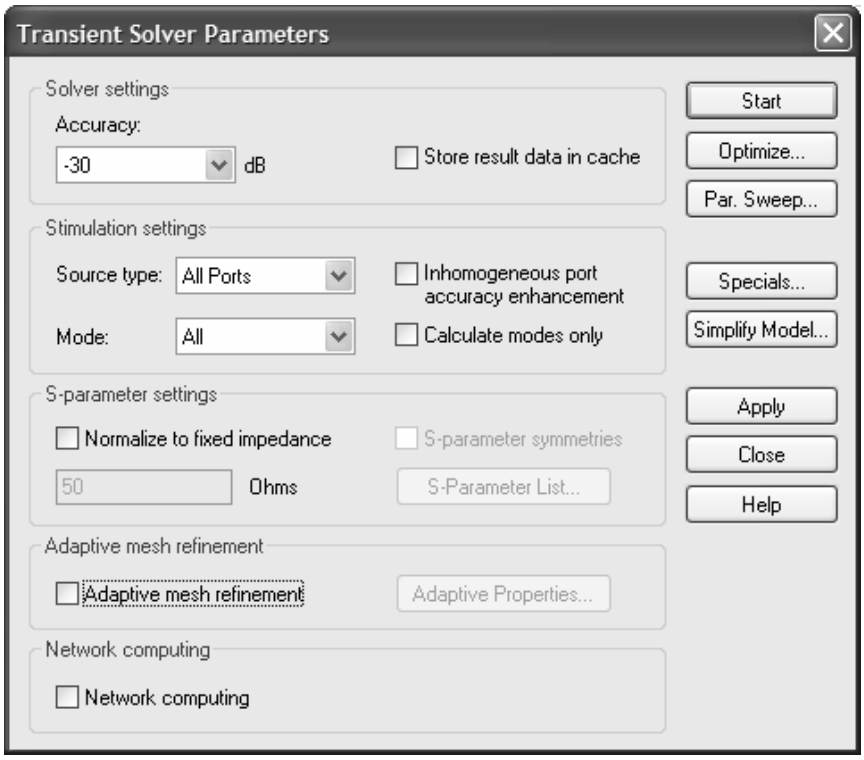


Combine Results

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.

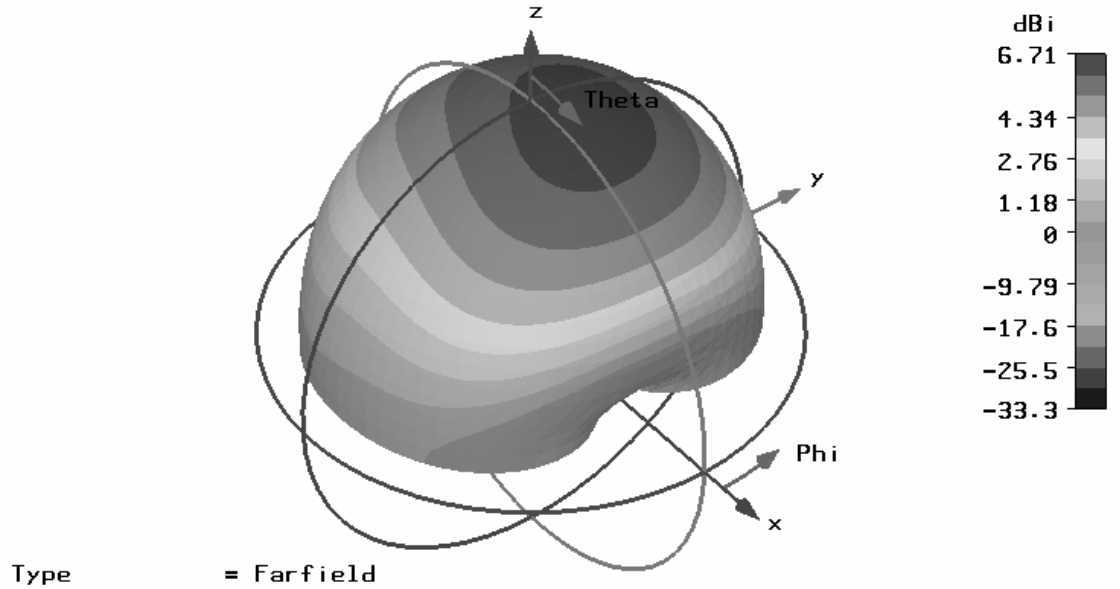
☐ Define the Solver's Parameters and Start the Calculation

As described earlier, the solver's parameters are specified in the "Transient Solver Parameters" dialog box that can be opened by selecting *Solve* ⇨ *Transient Solver* from the main menu or by pressing the corresponding icon in the toolbar:



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* check box 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 to bring your plot into view:

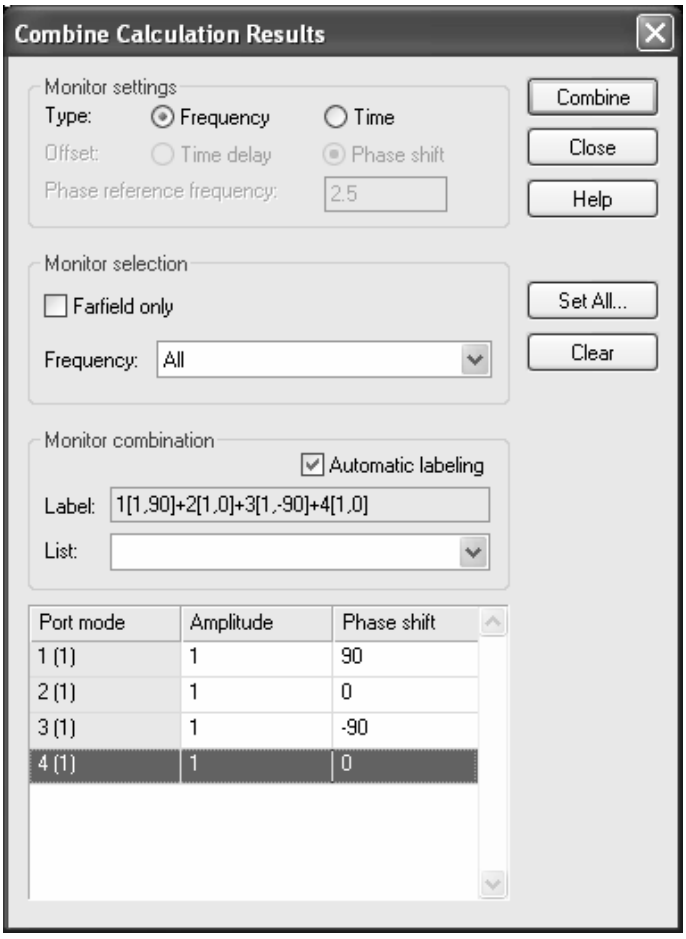



```
Type          = Farfield
Approximation = enabled (kR >> 1)
Monitor       = farfield (f=2.4) [1]
Component     = Abs
Output        = Directivity
Frequency     = 2.4
Rad. effic.   = 0.9525
Tot. effic.   = 0.7070
Dir.          =  6.706 dBi
```

☐ Combine Results

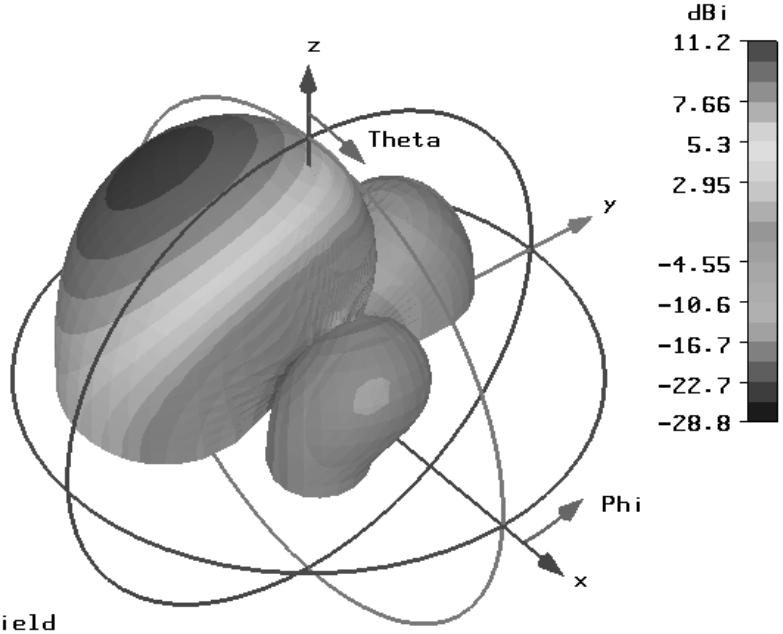
The most interesting results, however, are those where all patches are driven at one time with a pre-determined amplitude and phase variation being taken into account. This can be produced by CST MICROWAVE STUDIO® using the “Combine results” option. Press *Results* ⇨ *Combine Results* to bring the “Combine 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, we define identical antenna settings, i.e. the first antenna receives a phase shift of +90 degrees, the third antenna –90 degrees and the two remaining antennas 0 degrees:



Please note: The corresponding monitor label will be generated automatically due to the settings of the port mode combination; however, by disabling the *Automatic labeling* 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:




```
Type           = Farfield
Approximation  = enabled (kR >> 1)
Monitor        = farfield (f=2.4) [1[1,90]+2[1,0]+3[1,-90]+4[1,0]]
Component      = Abs
Output         = Directivity
Frequency      = 2.4
Rad. effic.    = 0.9517
Tot. effic.    = 0.7059
Dir.           =  11.19 dBi
```

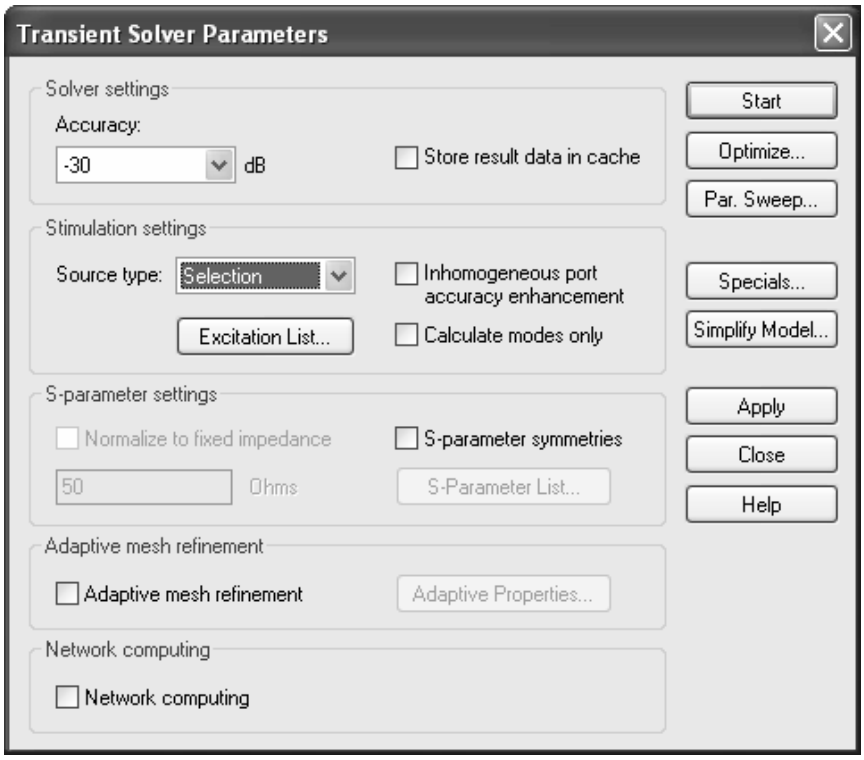
As you can see, the combination of results is time efficient because you do not have to restart the solver; just enter the amplitude and phase and view the combined generated farfield plot.

Simultaneous Excitation

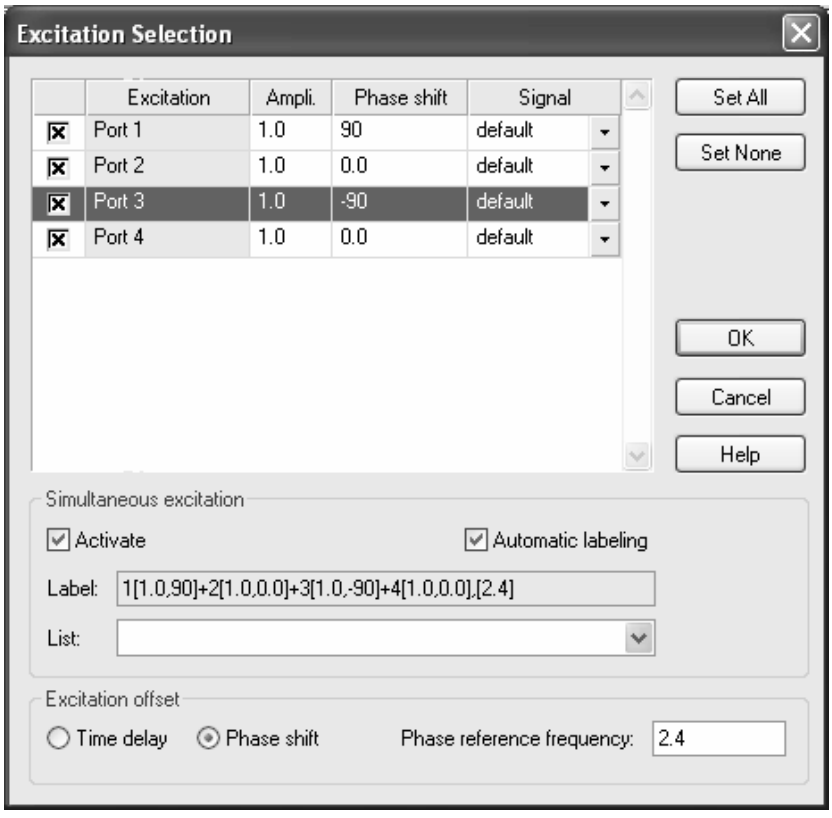
Another possibility for obtaining the farfield result of the array pattern is the excitation of all four ports simultaneously. In contrast to the combine result facility, here only one transient solver run is required. However, the phase shift relationship between the antenna elements has to be known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

> **Please note:** 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 (*Solve ⇨ Transient Solver,*):



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

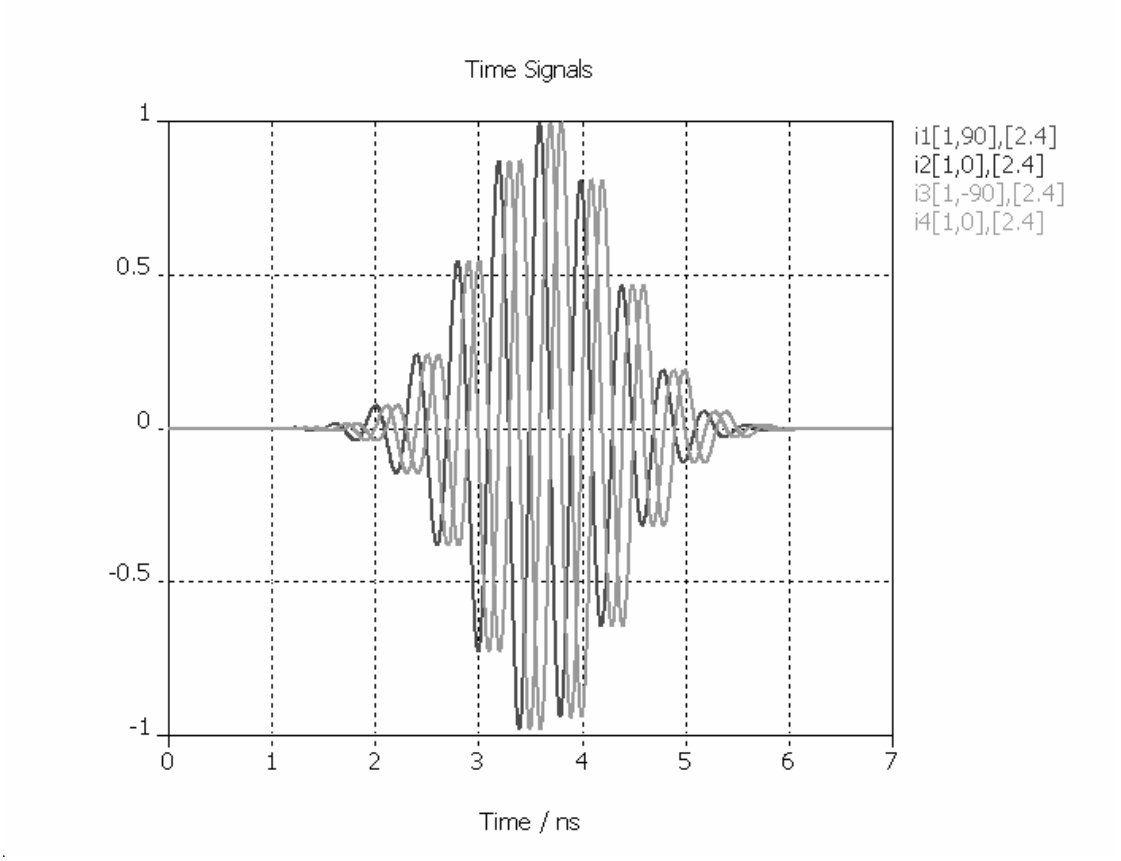


Since all four existing ports will be excited, please activate all available check buttons in the port mode list. In order to enter the amplitude and phase values, you must first activate the *Simultaneous excitation* by checking the corresponding *Activate* button.

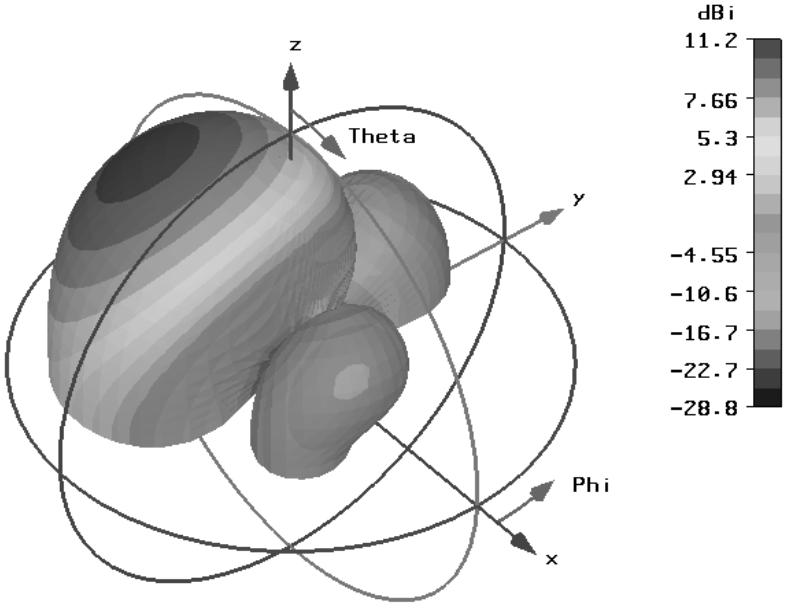
Please activate the *Phase shift* check box and enter a reference frequency of 2.4 GHz so that the result will be compatible to the combined farfield monitor in the previous chapter. Again, all antennas are driven with an amplitude value of one and phase values of 90, 0, -90, 0 degrees. Please double-check 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 (*Solve ⇨ Transient Solver,* !T) 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:




```
Type          = Farfield
Approximation = enabled (kR >> 1)
Monitor       = farfield (f=2.4) [1[1.0,90]+2[1.0,0.0]+3[1.0,-90]+4[1.0,0.0],[2.4]
Component     = Abs
Output        = Directivity
Frequency     = 2.4
Rad. effic.   = 0.9549
Tot. effic.   = 0.7087
Dir.          =  11.19 dBi
```

Please note: If some ports are stimulated simultaneously the magnitude and phase of the normalized signal spectrum at the ports are recorded (F-parameters).

Getting More Information

Congratulations! You have just completed the antenna tutorial that should have provided you with a good working knowledge on how to use CST MICROWAVE STUDIO® to calculate S-parameters and farfield results. The following topics have been covered:

1. General modeling 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.

You can obtain more information for each particular step from the online help system that can be activated either by pressing the *Help* button in each dialog box or by pressing the *F1* key at any time to obtain context sensitive information.

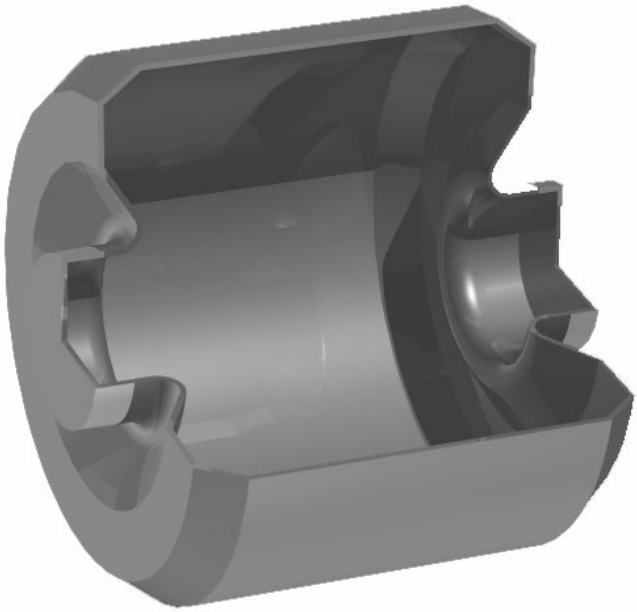
In some cases we have referred to the *Getting Started* manual that is also a good source of information for general topics.

In addition to this tutorial, you can find more S-parameter calculation examples for planar structures or various antenna models in the “Examples” folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that provides more information about the particular device.

Finally, you should refer to the *Advanced Topics* manual for more in-depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please visit one of the training classes held regularly at a location near you. Thank you for using CST MICROWAVE STUDIO®!

Resonator Tutorial



Geometric Construction and Solver Settings **166**

Introduction and Model's Dimensions 166
Geometric Construction Steps 167
Solver Settings 173

Eigenmode Calculation with AKS **176**

Eigenmode Visualization and Q-Factor Calculation **180**

Accuracy Considerations **183**

Eigenmode Calculation with JDM **188**

Getting More Information **191**

Geometric Construction and Solver Settings

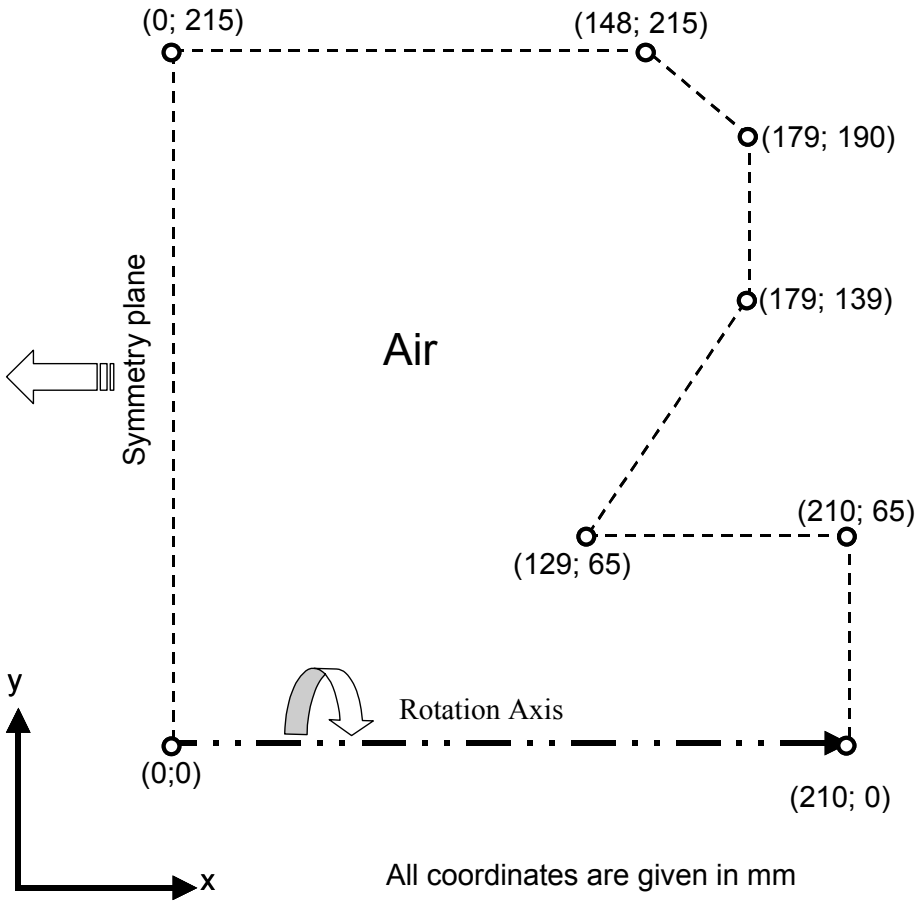
Introduction and Model's Dimensions

This tutorial demonstrates how to simulate resonator devices. As a typical example you will learn how to calculate the eigenmodes of an air filled cavity. The following explanations can be applied to other resonators as well.

Due to its rotational symmetry, the cavity can be easily modeled by rotating a cross section profile around an axis. Furthermore, due to a second symmetry, it is sufficient to draw half of the profile and mirror the structure afterwards to build the full model. After the model is constructed, the analysis process is quite simple. This tutorial will show you how to calculate and visualize a number of eigenmodes for this resonator. In addition, the Q-factors will also be calculated.

> **Please note:** The CST MICROWAVE STUDIO® "Eigenmode solver" license is required for this tutorial. If it is not available, please contact your distribution partner.

The cross section polygon is shown below. Please note than only half of the profile needs to be drawn because of the depicted symmetry. The coordinates are given in mm for each of the polygon's points.

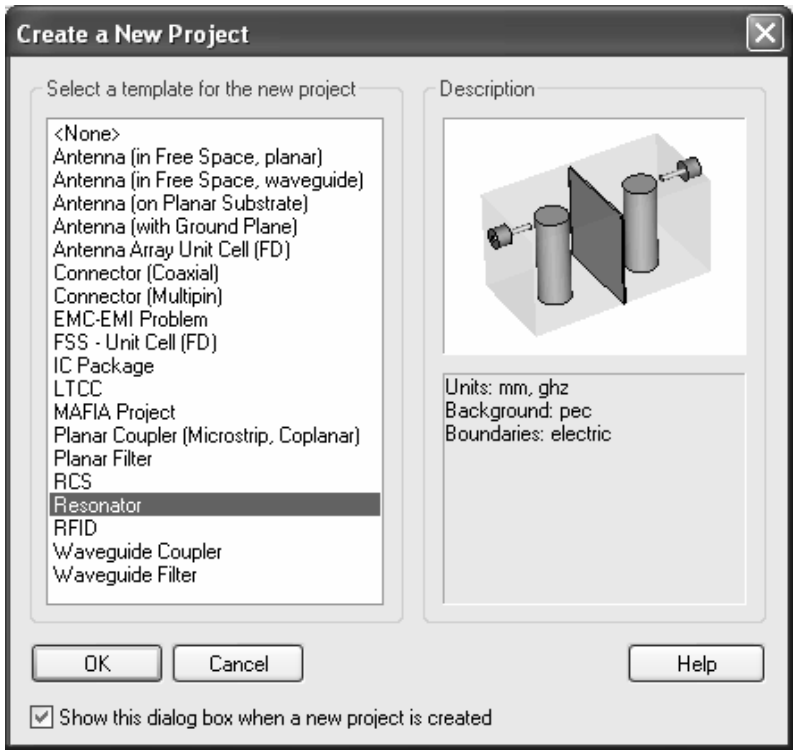


The following sections will guide you through the construction of the cavity's model. Please make sure that you carefully complete each step before you proceed to the next one.

Geometric Construction Steps

☐ Select a Template

Once you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, the program requests that you select a template that best fits your current device. The "Resonator" template should be chosen.

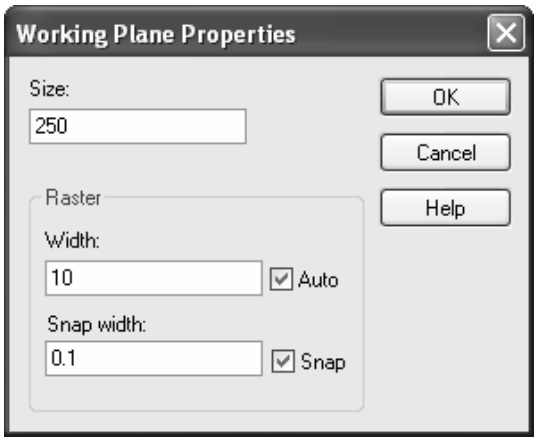


This template automatically sets the units to mm and GHz, the background material to PEC (which is the default) and all boundaries to be perfect electrical conductors. Please select this template and press the *OK* button.

☐ Set Working Plane Properties

Before you start entering the cavity's shape, you should set the working plane to be large enough to contain the model's dimensions. The dialog box can be accessed by selecting *Edit ⇨ Working Plane Properties*.

The largest coordinate value is 215 mm, so a working plane size of 250 mm will suffice. Enter this value in the *Size* field and set the *Raster Width* to 10 mm in order to obtain a reasonably fine grid on the drawing plane. Please note that all geometric settings are in mm because the current geometry unit is set to mm (displayed in the status bar).



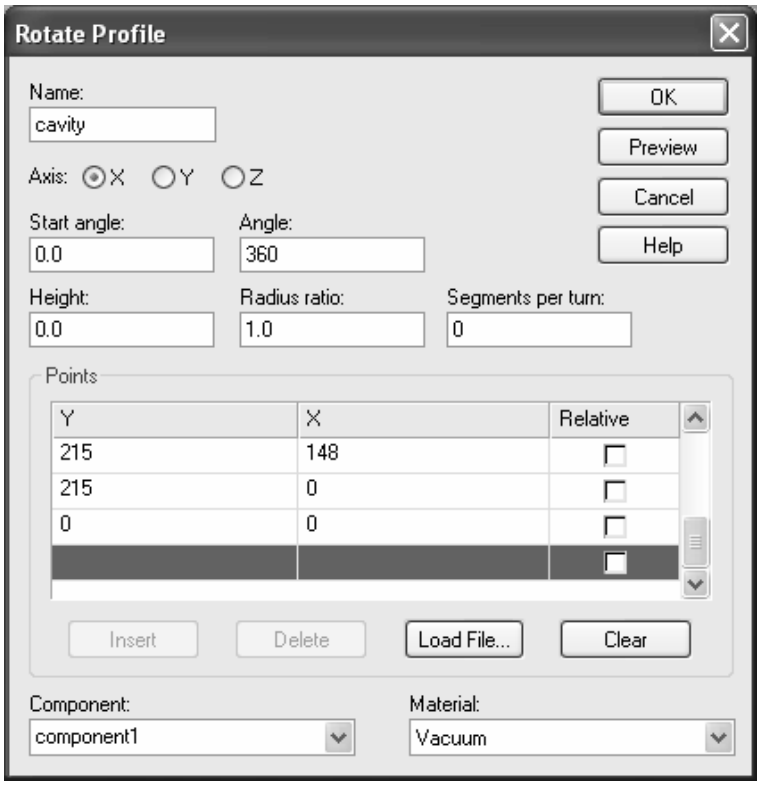
☐ Create Figure of Rotation

After these preparatory steps have been completed, you can now start drawing the figure of rotation. Because the cross section profile is a simple polygon, you do not need to use the curve modeling tools here (please refer to the *Getting Started* manual for information on this advanced functionality). For polygonal cross sections, it is more convenient to use the figure of rotation tool, activated by selecting *Objects* ⇨ *Rotate* from the main menu.

Since no face has been previously picked, the tool will automatically enter a polygon definition mode and request that you enter the polygon's points. You can do this either by double-clicking on each point's coordinates on the drawing plane, or by entering the values numerically. Because the latter approach may be more convenient, we suggest pressing the *Tab* key and entering the coordinates in the dialog box. All polygon points can thus be entered according to the following table (whenever you make a mistake, you can delete the most recently entered point by pressing the *Backspace* key):

Point	X	Y
1	0	0
2	210	0
3	210	65
4	129	65
5	179	139
6	179	190
7	148	215
8	0	215
9	0	0

After the last point has been entered, the polygon will then be closed. The "Rotate Profile" dialog box will then automatically appear.



This dialog box allows you to review the coordinate settings in the table. If you encounter any mistakes you can easily change the values by double-clicking on the incorrect coordinate's entry field.

The next step is to assign a specific *Component* and a *Material* to the shape. In this case, the default settings with "component1" and "Vacuum" are practically appropriate.

> **Please note:** The use of different components allows you to combine several solids into specific groups, independent of their material behavior. However, here it is convenient to construct the complete cavity as a representation of one component.

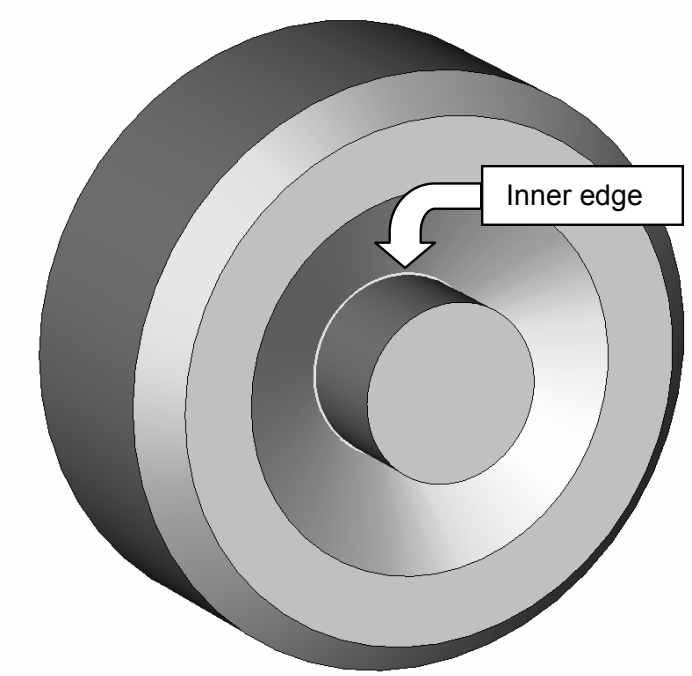
Finally, you should assign a proper *Name* (e.g. "cavity") to the shape and press the *OK* button in order to create the solid.

☐ Pick Edge and Blend Edge

Sharp edges inside cavities are best avoided because strong field singularities will occur at these edges. Therefore, the sharp inner edge (shown on the picture above) should be blended.

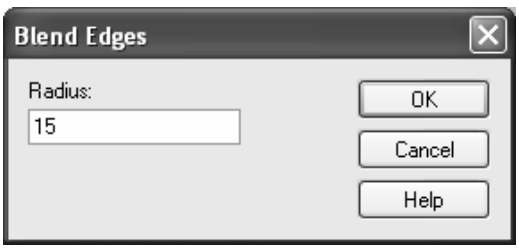
The first step towards blending the edge is to pick the edge in the model by entering the pick edge mode with the toolbar icon, the *Objects ⇨ Pick ⇨ Pick Edge* menu entry, or the shortcut *e* in the main view window. Once this mode is active, which is indicated by all edges in the model being highlighted, you can easily double-click at the inner edge. If the edge is hidden, you can rotate the view using the view changing tools as explained in the *Getting Started* manual. Sometimes it is also advantageous to switch the drawing mode to Wireframe (*View ⇨ View options*).

Once the proper edge has been selected, the model should look as follows:

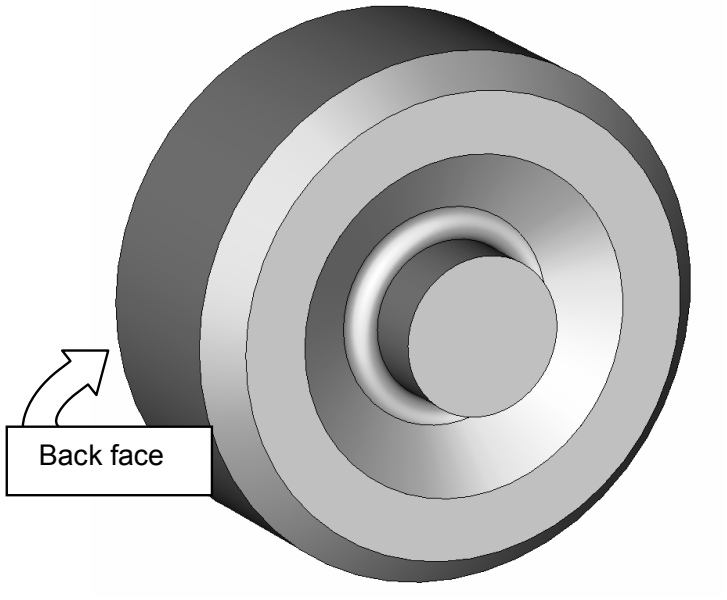


If you accidentally picked the wrong edge, you can delete all picks using *Objects ⇨ Clear Picks* and try again.

The next step is to blend the selected edge. Therefore, press the toolbar icon or select *Objects ⇨ Blend Edges* from the menu. Afterwards, a dialog box will open where you can set the *Radius* of the blend to 15 mm.



Finally press the *OK* button to apply the blend. The structure should then look as follows:

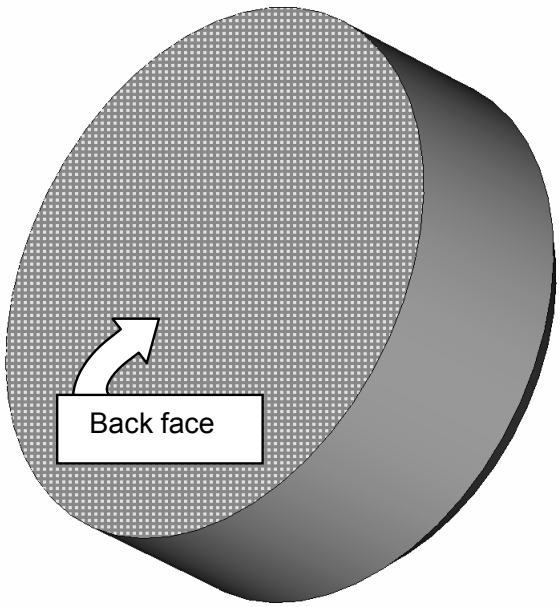


☐ Mirror the Structure to Model the Entire Cavity

Thus far you have successfully modeled half of the cavity. The easiest way to obtain the full model is to mirror this structure at the planar back face.

The first step is to define the mirror plane by picking the corresponding face in the model. Therefore, enter the face pick mode by pressing the toolbar item , selecting *Objects* ⇨ *Pick* ⇨ *Pick Face*, or using the shortcut *f* in the main view window.

Once the face pick mode is active, you should double-click on the planar back face as shown in the picture above. If the face is hidden by the structure, you can use the view changing tools as explained in the *Getting Started* manual. After the face has been selected, the model should look as follows:

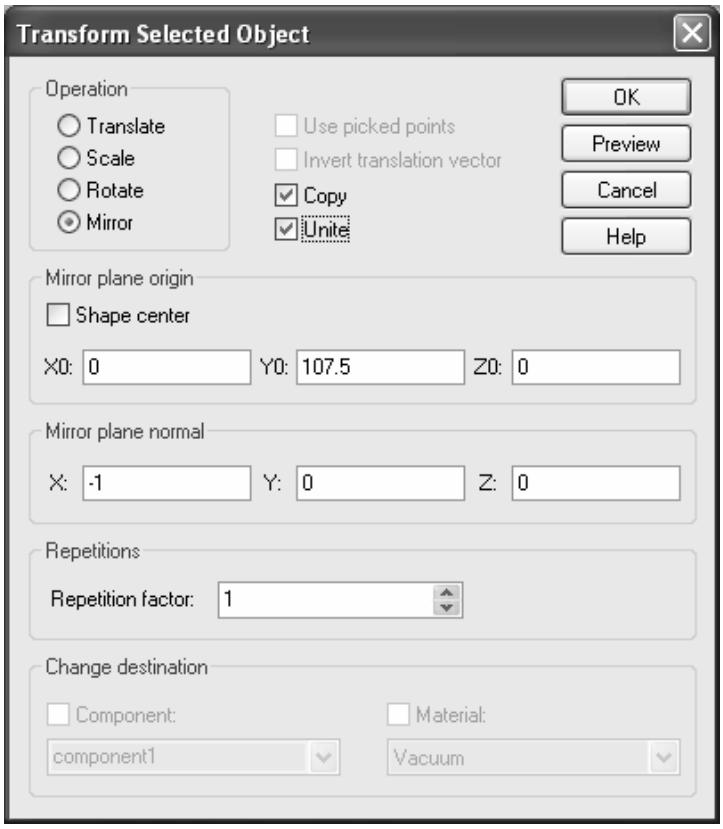


If this is not the case, then please delete all picks with *Objects* ⇨ *Clear Picks* and try again.

You should now select the cavity's solid by double-clicking it. Please confirm that its name becomes highlighted in the navigation tree.

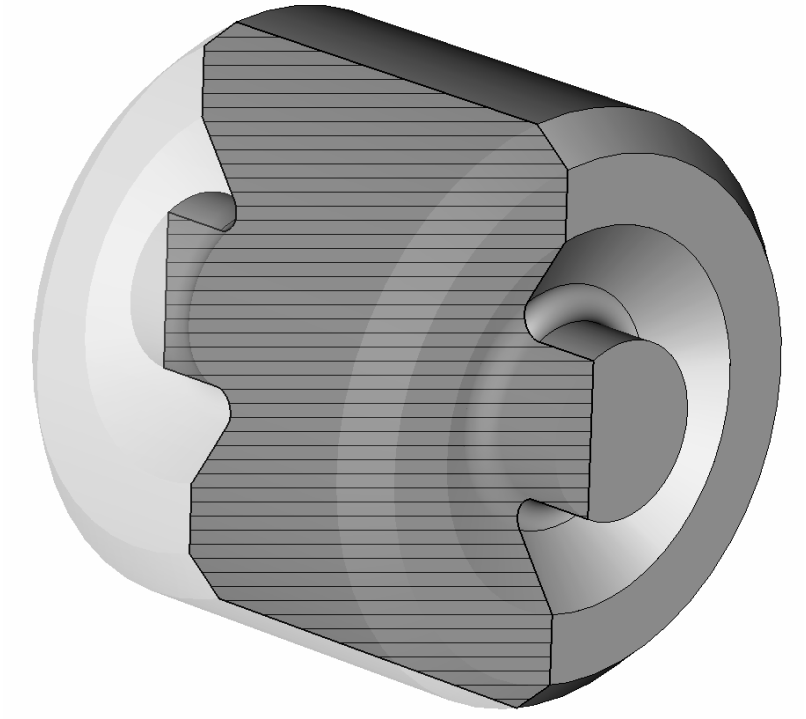
The next step is to use the transform tool to create the mirrored shape by pressing the toolbar item or selecting *Objects* ⇨ *Transform*.

In the dialog box that appears you should select the *Operation Mirror*. The mirror plane's coordinates will then be automatically set according to the previously-picked back face of the cavity, so you do not need to change any of the coordinate settings.



Because the transformation will create a new shape by mirroring the existing one, you have to select the *Copy* option. Furthermore, the original shape should also be combined with the mirrored one to form a single shape. Thus, the *Unite* option must also be switched on.

Finally, you should press the *OK* button to create the entire cavity shown below:



In this picture, the cavity has been sliced for visualization purposes using the *View* ⇨ *Cutting plane* tool.

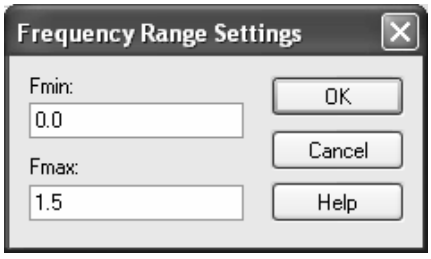
> **Please note:** Select *No cutplane* before closing the dialog in order to leave this mode.

Solver Settings

After you have successfully modeled the cavity's geometry, you need to specify some solver settings, such as frequency range and boundary conditions, before you can finally start the solver to calculate the eigenmodes.

☐ Define the Frequency Range

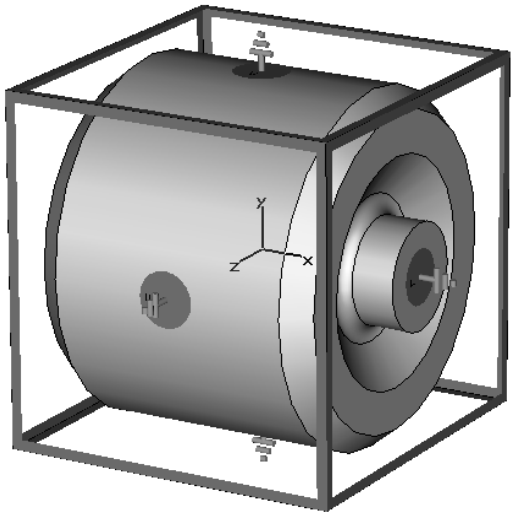
For this device, the first five resonance frequencies are estimated to be below 1.5 GHz. Open the frequency range dialog box either by pressing the icon from the toolbar or by selecting *Solve* ⇨ *Frequency*. In this dialog box you should set the upper frequency limit to 1.5 (please recall that the frequency unit has been set to GHz as shown in the status bar).

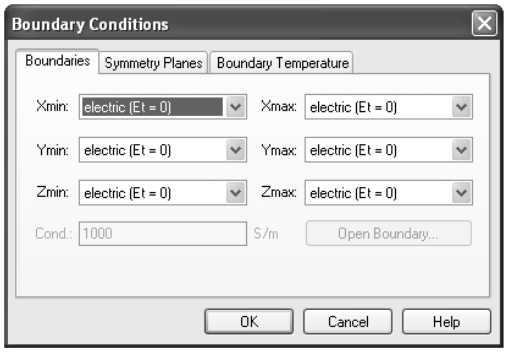


Finally, press the *OK* button to store these settings.

☐ Define Boundary Conditions and Symmetries

You should always check the boundary and symmetry conditions before starting the solver. This is most easily accomplished by entering the boundary definition mode by pressing the tool bar item or selecting *Solve* ⇨ *Boundary Conditions*. The boundary conditions will then become visualized in the main view.

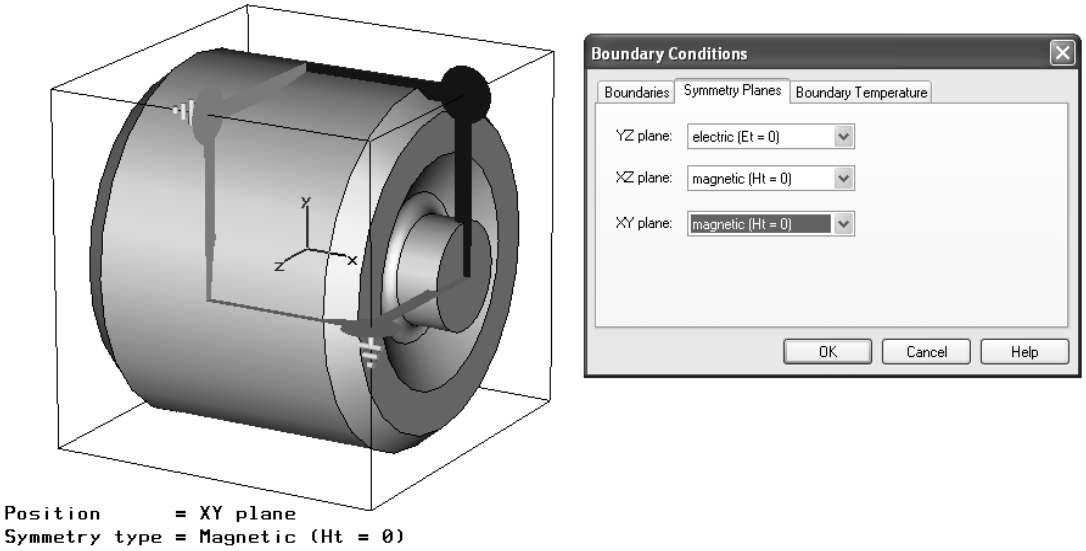




Here, all boundary conditions are set to electric, meaning that the structure is embedded in a perfect electrically conducting housing. These defaults (which have been set by the template) are appropriate for this example.

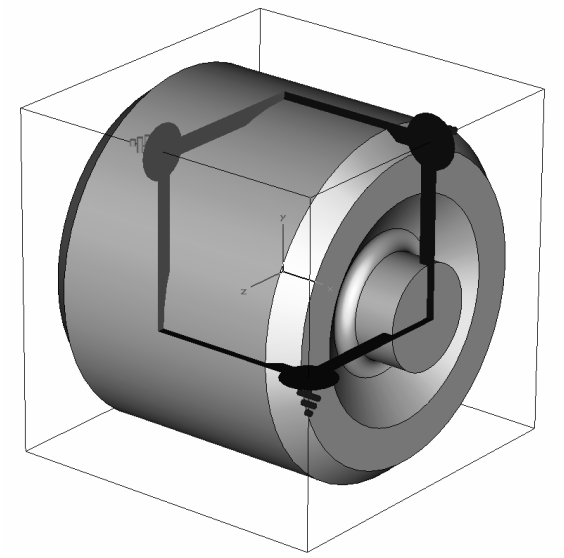
Assume that you are only interested in those modes that have longitudinal electric field components along the x-axis of the device. This *a priori* knowledge about the fields could be used to speed up the calculation by a factor of eight by informing the solver about these symmetry conditions.

Please enter the symmetry plane definition mode by activating the *Symmetry Planes* tab in the dialog box.



By setting the symmetry planes XY and XZ to *magnetic*, you force the solver to only calculate fields that have no magnetic field tangential to these planes (thereby forcing the electric field to be tangential to these planes). Additionally, you can set the YZ symmetry plane to be an electric wall, which implies that the electric field is forced to be normal to this plane.

After these settings have been made, the structure should look as follows:



Please note that you could also double-click on the symmetry planes' handles and choose the proper symmetry condition from the context menu.

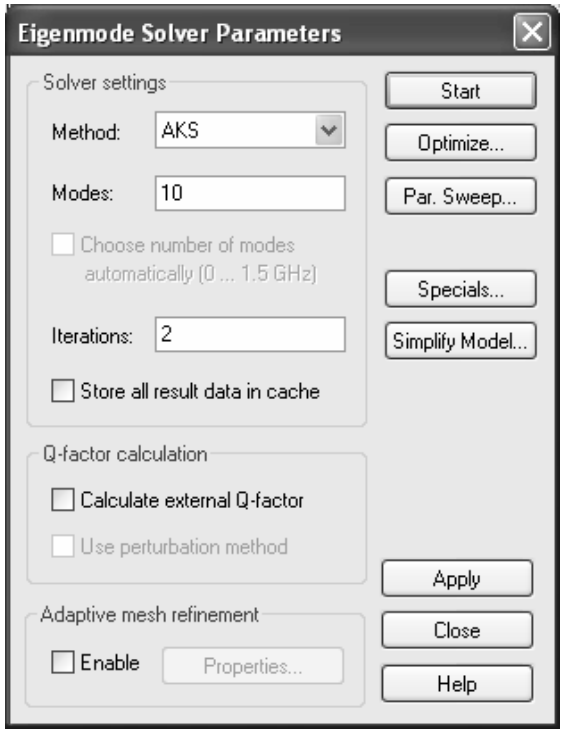
Finally press the *OK* button to complete this step.

In general, you should always make use of symmetry conditions whenever possible in order to reduce calculation times by a factor of two to eight.

Eigenmode Calculation with AKS

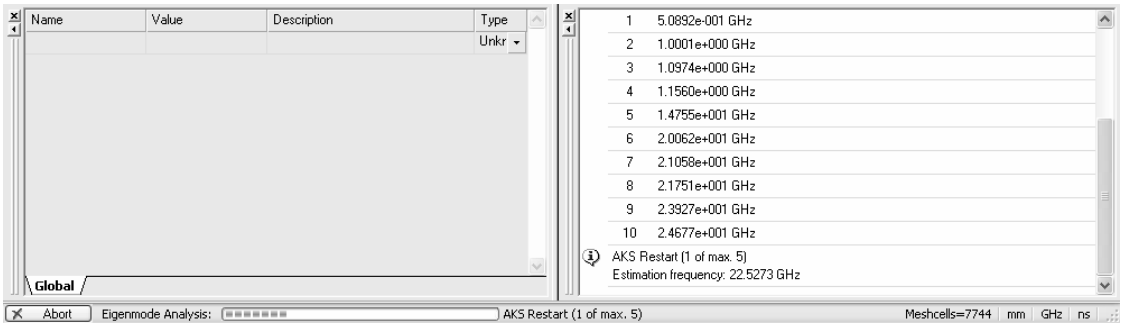
☐ Calculating the Eigenmodes Using Fully Automatic Solver Settings

After completing all of the above steps, you are ready to start the eigenmode calculation. Therefore, open the eigenmode solver control dialog box by pressing the corresponding toolbar icon or selecting *Solve* ⇨ *Eigenmode Solver* from the main menu.



The only setting that commonly needs to be specified here is the number of *Modes* to be calculated. The solver will then calculate this number of modes starting from the lowest resonance frequency. It is usually advantageous to specify more modes than you are actually searching for. Thus, assuming that you want to calculate the first five modes in this example, you should advise the eigenmode solver to calculate 10 modes, which is the default. You, therefore, do not need to change anything here and can press the *Start* button right away.

Progress bars will appear in the status bar, keeping you informed about the current status of the calculation (e.g. matrix calculation, eigenmode analysis).



Due to the Perfect Boundary Approximation®, the number of mesh cells required for discretizing this example is quite small (roughly 7700). This, in fact, corresponds to a system of equations consisting of about 23,100 unknowns. Calculating eigenmodes for such a system takes only a few minutes to complete on a modern PC.

After the solver has finished its work, the resonance frequencies of the first ten modes are displayed in the result window:

Final Results:

Mode	Frequency	Accuracy
1	0.5089 GHz	8.684e-008
2	1.000 GHz	2.147e-008
3	1.097 GHz	1.367e-008
4	1.156 GHz	9.779e-009
5	1.256 GHz	2.480e-008
6	1.463 GHz	1.029e-008
7	1.467 GHz	9.434e-009
8	1.603 GHz	2.697e-008
9	1.697 GHz	4.093e-007
10	1.732 GHz	0.0001066

Optimum guess for the highest eigenfrequency would be: 1.73153.

The accuracy for the mode solution is excellent for all these modes. A mode with an accuracy of less than 1e-3 can usually be considered sufficiently accurate.

In order to review the solver time required to achieve these results, you can display the solver log-file by selecting *Results* ⇨ *View Logfiles* ⇨ *Solver Logfile* from the main menu. Please scroll down the text to obtain the following timing information (the actual values may vary depending on the speed of your computer):

```
Mesh generation time     :       3 s
Solver time              :       26 s

Total time               :       29 s

-------------------------------------------------------------
```

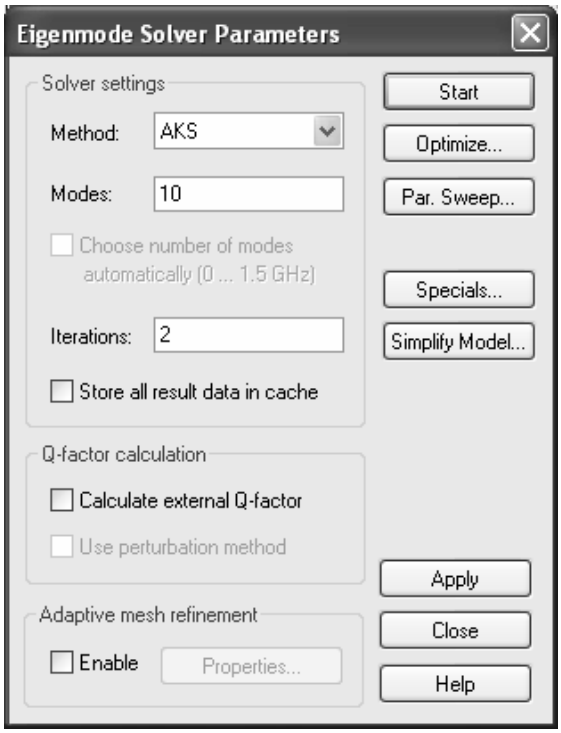
☐ Optimizing the Performance for Subsequent Calculations

To this point, you have successfully calculated the eigenmodes for this device in a reasonable amount of time. However, if you intend to make parametric studies it may be advantageous to speed up the solver for subsequent runs.

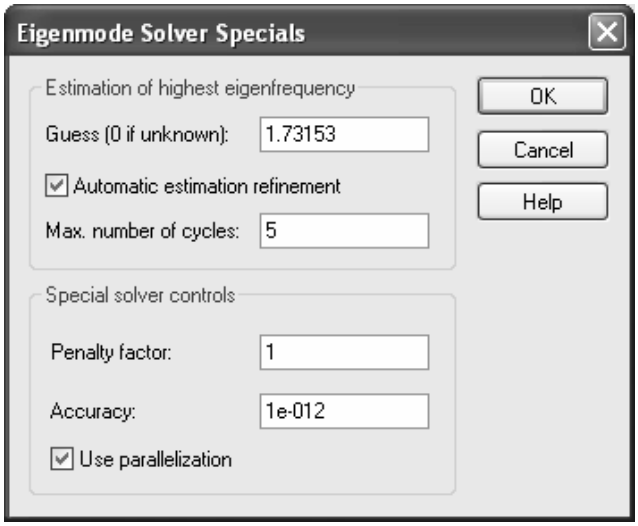
This performance tuning step is quite simple: The eigenmode solver can make use of a guess for the highest eigenmode frequency you are looking for. The eigenmode solver automatically determines this guess from a previous calculation and prints the result in the log-file. This information is shown right below the timing information:

```
------------------------------------------------------------

Optimum guess for the highest eigenfrequency would be: 1.73153

------------------------------------------------------------
```

In order to demonstrate how this information can be used for improving the solver's speed, you should now recalculate the eigenmodes to compare the time required for the simulation. Please enter the eigenmode solver control dialog box once more by pressing the corresponding toolbar icon or selecting *Solve* ⇨ *Eigenmode Solver* from the main menu.



In this dialog box, press the *Specials* button in order to open a dialog box for more advanced settings:



Once you have obtained a guess for the highest eigenfrequency of interest (here 1.73153 GHz), you can enter this value in the *Guess* field. If you don't know this value, just enter zero to let the solver estimate this value automatically. After pressing the *OK* button in this dialog box you can restart the eigenmode solver by pressing the *Start* button.

Again, a progress bar will appear informing you of the status of your calculation. Please note that there is no need for recalculating the matrix because the structure has not been changed.

The solver will finish its work after a short time, giving the same results as before for the eigenfrequencies. The timing information in the log-file should look similar to the data below:

```
-------------------------------------------------------------
Solver Statistics:

  Mesh generation time     :     0 s
  Solver time              :     15 s

  Total time               :     15 s

-------------------------------------------------------------
```

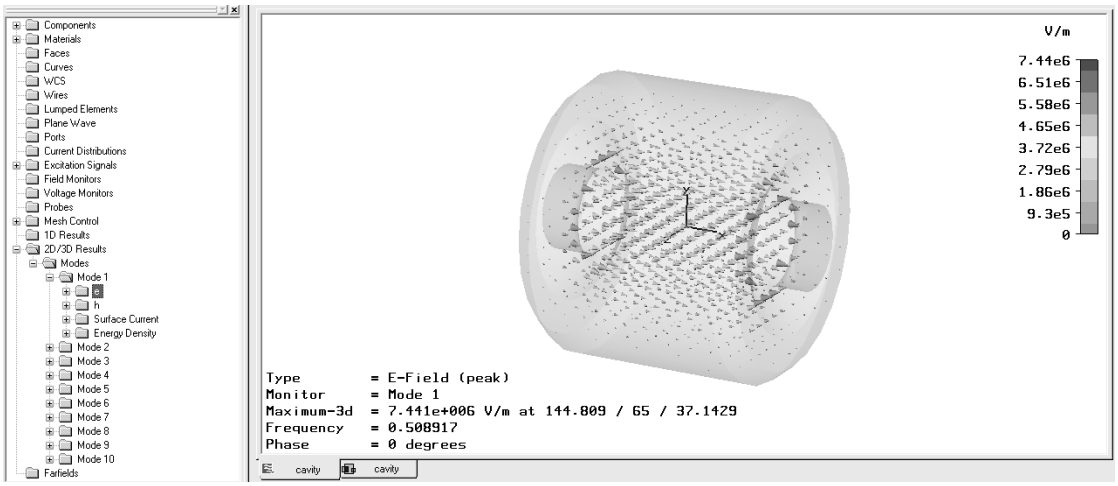
If you now compare the solver times, you can see that specifying the guess for the highest eigenfrequency could speed up the solution process by a factor of 1.5.

Please remember that these steps are only made in order to illustrate how to speed up the solver for parametric sweeps or optimizations. The accuracy of the solution will also be excellent using the fully automatic procedure without this additional setting. For a single analysis of a particular device, this performance tuning step does not make sense, of course.

Eigenmode Visualization and Q-Factor Calculation

☐ Eigenmode Visualization

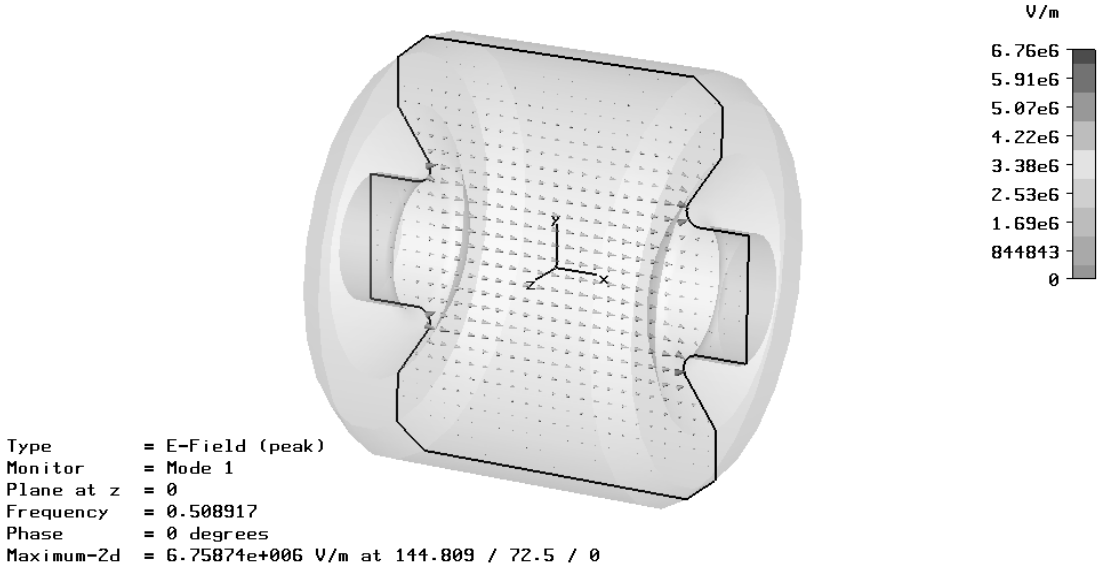
The eigenmode results can be accessed by selecting the corresponding item in the navigation tree from the *2D/3D Results* ⇨ *Modes* folder. The field patterns for each mode will be stored in subfolders named *Mode N* where *N* represents the mode number.



To visualize the electric field of the first mode, select the corresponding item from the navigation bar: *2D/3D Results* ⇨ *Modes* ⇨ *Mode 1*⇨ *e*. The result data will then be visualized in a three dimensional vector plot as shown in the above picture.

> **Please note:** The field amplitudes of the modes will always be normalized such that each mode contains a total energy of 1 Joule.

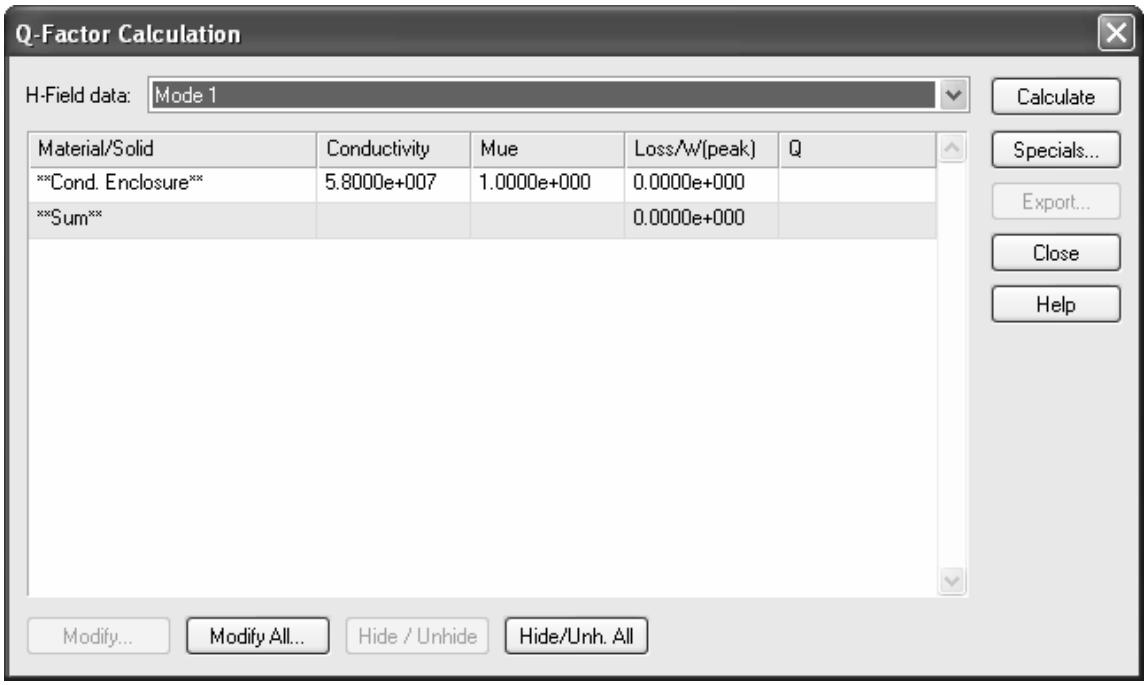
In many cases, it is more important to visualize the fields in a cross section plane. Therefore, please switch to the 2D field visualization mode by pressing the icon or by selecting *Results* ⇨ *3D Fields on 2D Plane*. The field data should then be visualized and appear similar to the picture below. Please refer to the *Getting Started* manual or press the *F1* key for online help to obtain more information on field visualization options.



In addition to the graphical field visualization, some information text containing maximum field strength values and resonance frequencies will also be shown in the main window.

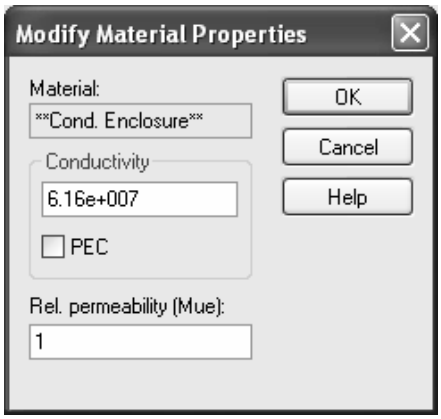
☐ Q Factor Calculation

The eigenmode's internal Q-factors, which are an important quantity for cavity design, can be easily derived from the field patterns. Open the loss and Q-factor calculation dialog box by selecting *Results ⇨ Loss and Q Calculation*.



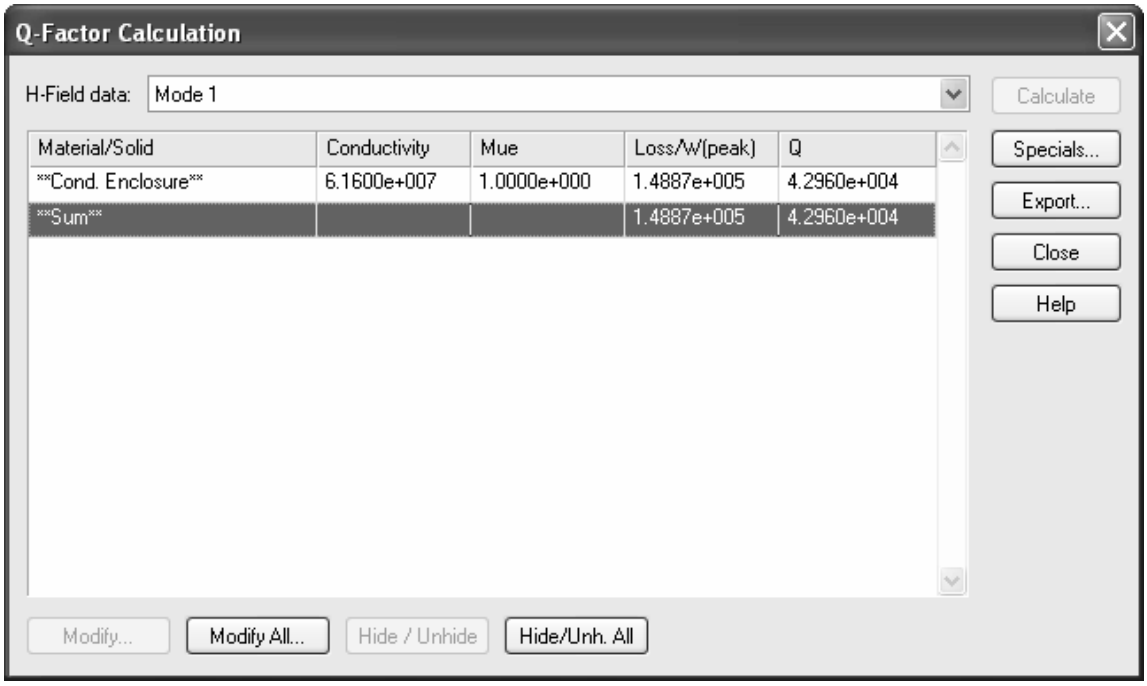
The only setting that requires specification here is the conductivity of the enclosing metal. By default, the conductivity of the *Cond. Enclosure* is set to copper (5.8e7 S/m).

You may change this setting by clicking on the first row and selecting the *Modify* button that opens the following dialog box:



In this example, you can set the *Conductivity* value to silver (6.16e7 S/m) and press the *OK* button.

Back in the loss and Q factor calculation dialog box you should now select the desired Mode from the *H-Field data* list, e.g. the fundamental mode *Mode 1*. Finally, press the *Calculate* button to obtain the Q factor value.



The Q-factor is calculated to be $4.2960 \cdot 10^4$ for the fundamental mode. The Q-factors for the other modes can be calculated similarly.

Accuracy Considerations

The eigenmode calculation is mainly affected by two sources of numerical inaccuracies:

1. Numerical errors introduced by the iterative eigenmode solver.
2. Inaccuracies arising from the finite mesh resolution.

In the following sections we provide hints on how to minimize these errors and achieve highly accurate results.

☐ Accuracy of the Numerical Eigenmode Solver

The first type of error is always quantified as *Accuracy* for each mode after the calculation has finished. The modes can usually be considered sufficiently accurate for most practical applications if the mode accuracy is below 1e-3.

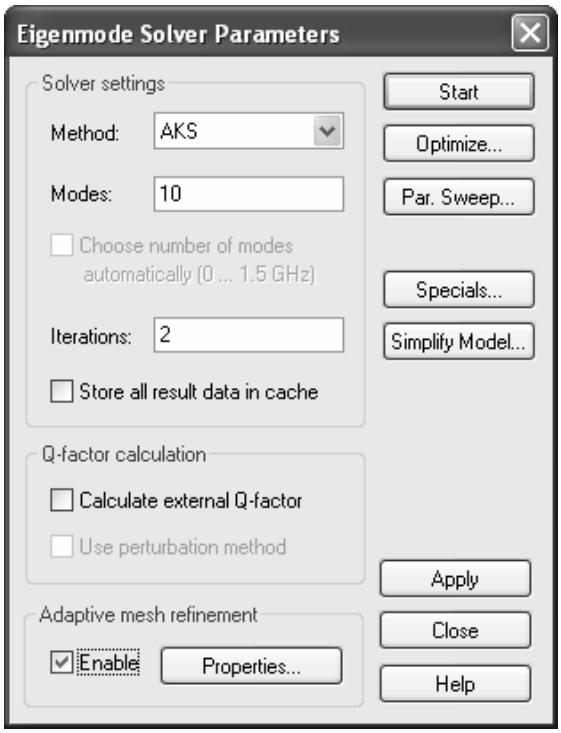
The results can be improved by specifying a proper guess for the highest eigenmode frequency of interest and by increasing the number of iterations for the eigenmode solver. Specifying more than five iterations does not typically improve the results. Sometimes higher order modes are calculated with a lower accuracy than lower order modes. In many cases it may be advantageous to calculate more modes than you are actually looking for, in order to improve the accuracy of the desired (lower) ones.

> **Please note:** In the next section the JDM solver is used. In contrast to the AKS eigenmode solver, the JDM solver requires no guess of the highest eigenfrequency and the eigenmodes are calculated to a prescribed accuracy.

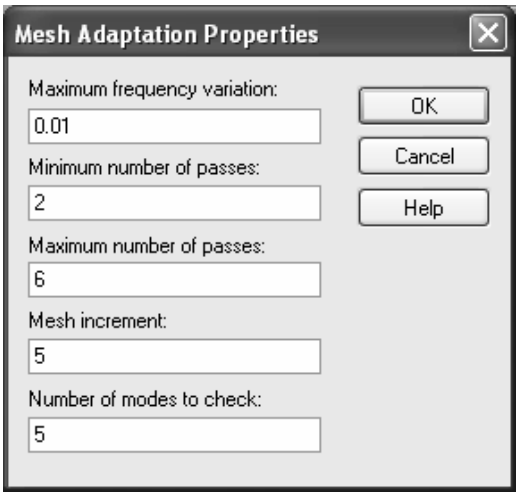
☐ Effect of the Mesh Resolution on the Eigenmode Accuracy

The 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 eigenmodes. If the desired results (e.g. eigenmode frequencies, Q-factors) do not significantly change after increasing the mesh density, then convergence has been achieved.

In the above example, you have used the default mesh that has been automatically generated by an expert system. The easiest method of testing the accuracy of the results is to use the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (*Solve ⇨ Eigenmode Solver):*

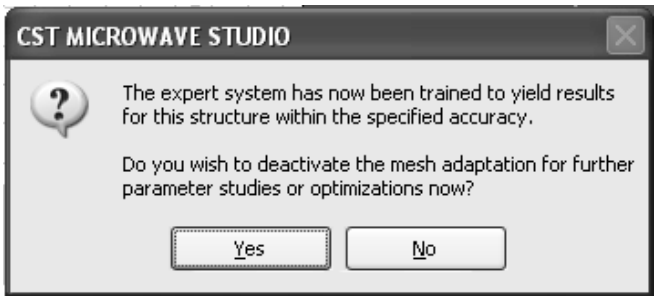


After activating the adaptive mesh refinement tool, the *Properties* button becomes active. Press this button to open the mesh refinement properties dialog box:



Because you are only interested in the first five modes of this structure, you should change the *Number of modes to check* to 5 in order to focus the mesh refinement procedure on these modes. Afterwards, you can close this dialog box by pressing the *OK* button.

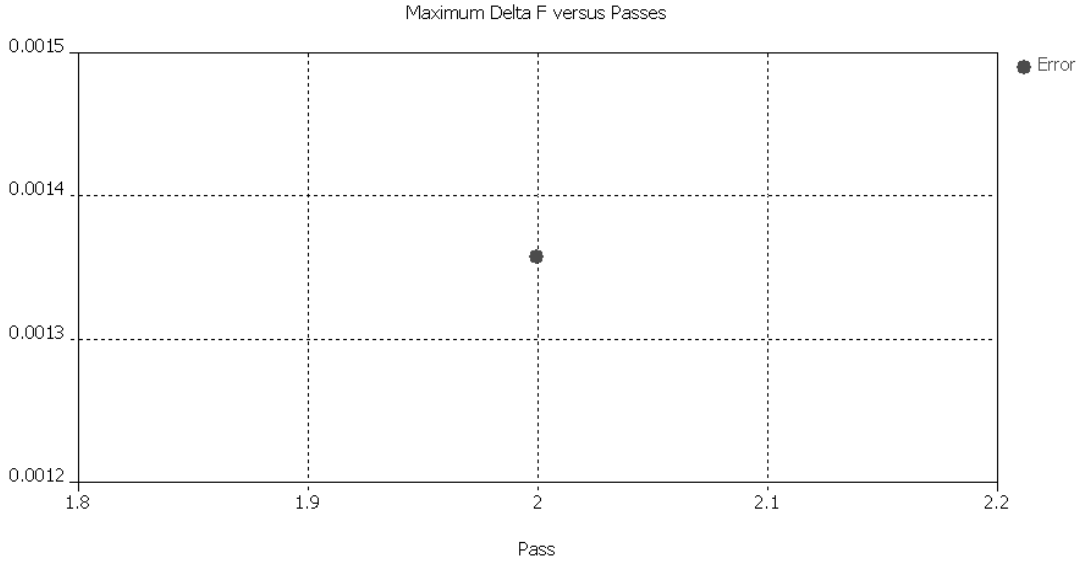
Back in the solver dialog box, you can now start the eigenmode solver again by pressing the *Start* button. After a couple of minutes, during which the solver is running through mesh adaptation passes, the following dialog box will appear:



This dialog box notifies you that the desired accuracy limit (1% by default) could be met by the adaptive mesh refinement. Since the expert system's settings have been adjusted to achieve 1% accuracy, you may switch off the adaptation procedure for subsequent calculations (e.g. parameter sweeps or optimizations).

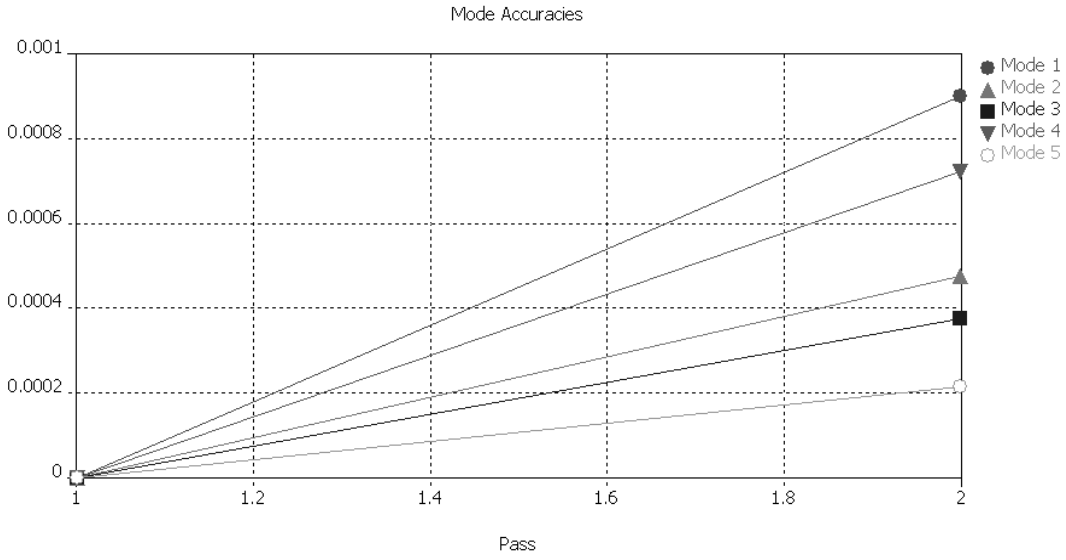
You should now confirm the deactivation of the mesh adaptation by pressing the *Yes* button. The converged results for the eigenmode frequencies will then be shown in the result window.

After the mesh adaptation procedure is complete, you can visualize the maximum relative difference in the eigenmode frequencies for two subsequent passes by selecting *1D Results ⇨ Adaptive Meshing ⇨ Error* from the navigation tree:

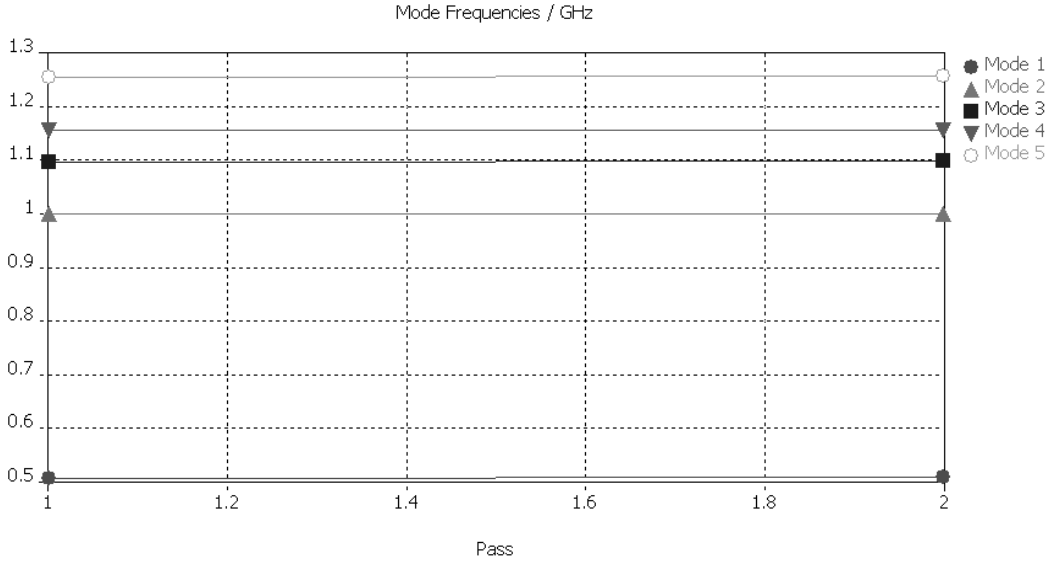


As evident from the above plot, the maximum deviation of the eigenmode's frequencies is below 0.14%, indicating that the expert system based meshing would have been fine for this example, even without running the mesh adaptation procedure.

The eigenmode solver accuracies achieved for the modes during the mesh adaptation can be visualized by selecting *1D Results ⇨ Adaptive Meshing ⇨ Mode Accuracies* from the navigation tree:



The above plot shows that all mode accuracies for both passes are fine. Finally, you can visualize the convergence process of the mode frequencies by clicking on *1D Results ⇨ Adaptive Meshing ⇨ Mode Frequencies*:

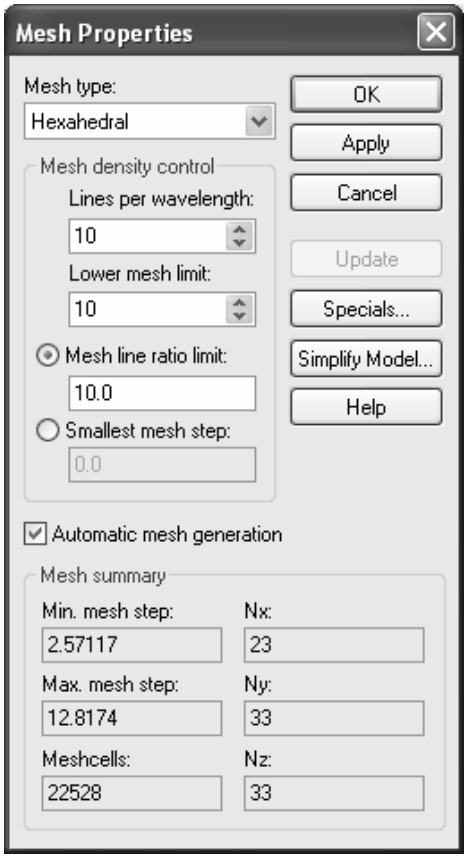


Inspection of the above plot confirms that the results are quite stable.

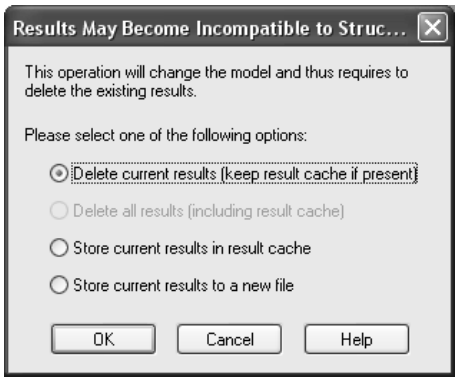
The major advantage of the expert system based mesh refinement procedure over traditional adaptive schemes is that the mesh adaptation needs to be carried out only once for each device in order to determine the optimum settings for the expert system. There is then no need for time consuming mesh adaptation cycles during parameter sweeps or optimization.

Eigenmode Calculation with JDM

CST MICROWAVE STUDIO® offers the possibility to choose the JDM eigenmode solver. This solver is recommended only if a small number of eigenmodes are required (5 modes or less). Since this is a tutorial, it is convenient to reset the mesh settings to the initial values of the mesh adaptation to speed up the calculation. Open the "Mesh Properties" dialog by selecting *Mesh ⇨ Mesh Properties* and set both *Lines per wavelength* and *Lower mesh limit* equal to 10.

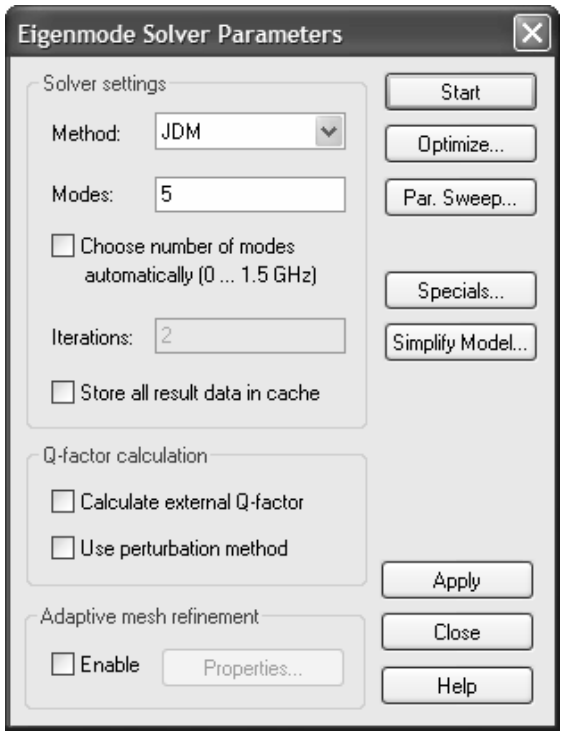


There may be old results present from the previous solver run that will be overwritten when changing the mesh. In this case, the following warning message appears:

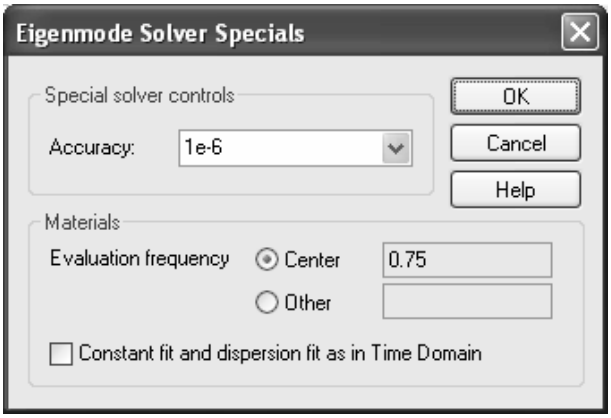


Press *OK* to acknowledge deletion of the previous results.

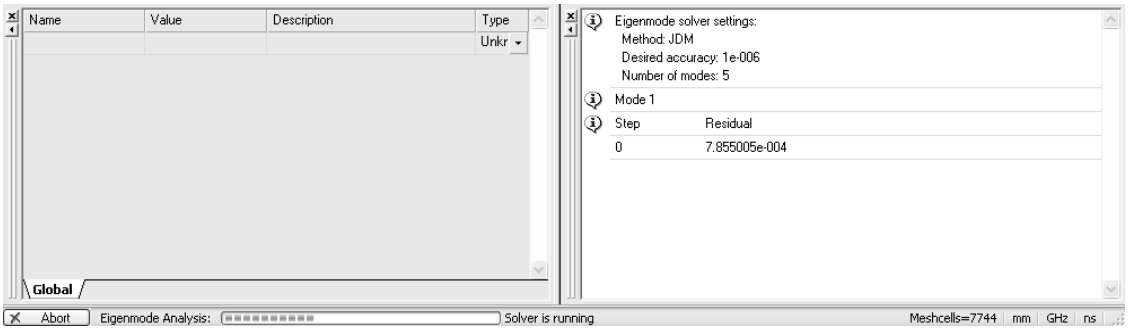
The solver's parameters are specified in the solver control dialog box that can be opened again by selecting *Solve* ⇨ *Eigenmode Solver* from the main menu or by pressing the corresponding toolbar icon . Select from the *Method* drop down menu the eigenmode solver "JDM" and reduce the number of *Modes* to 5. With these settings the solver will calculate 5 modes starting from the lowest resonance frequency.



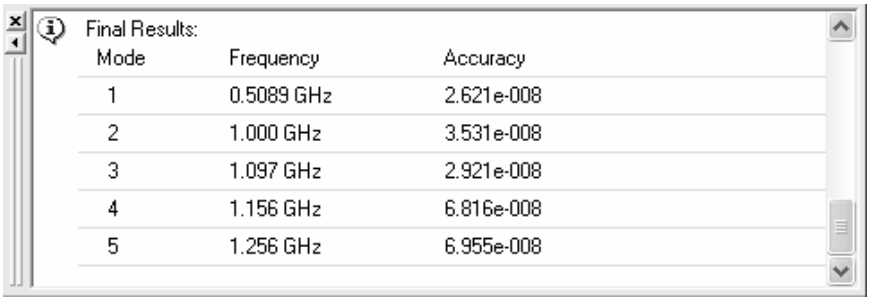
Before starting the eigenmode solver, you may select the *Specials* button in order to change the desired accuracy for the eigenmodes. In this case, the value of 1e-6 is sufficient and you can leave the box unchanged by pressing the *OK* button.



Finally, press the *Start* button. A few progress bars will appear in the status bar again, keeping you informed of the current status of your calculation (e.g. matrix calculation, eigenmode analysis).



After the solver has finished, the resonance frequencies of the first five modes are displayed in the main view.



In order to review the solver time required to achieve these results, you can display the solver log-file by selecting *Results* ⇨ *View Logfiles* ⇨ *Solver Logfile* from the main menu. Please scroll down the text to obtain the following timing information (the actual values may vary depending on the speed of your computer):

```
Mesh generation time     :       3 s
Solver time              :       12 s

Total time               :       15 s

-----------------------------------------------------------------
```

The solver time required for the calculation is comparable to the AKS eigenmode solver.

> **Please note:** In contrast to the AKS eigenmode solver, the JDM requires no guess of the highest eigenfrequency and the eigenmodes are calculated to a prescribed accuracy. The visualization of modes, Q-factor calculation and mesh adaptation are carried out as for the AKS eigenmode solver.
>
> In contrast to AKS, the JDM does not currently support TST mesh cells. Furthermore, the AKS solver should be used if many modes have to be calculated. The JDM eigenmode solver is able to calculate eigenmodes of lossy structures (const. complex permittivity). However, if you require the Q-factors and the losses are small, it is advisable to calculate the eigenmodes of the loss-free structure first. This can be done by activating the *Use perturbation method* check box. Afterwards, you can compute the Q-factors by the post-processing step described previously.

Getting More Information

Congratulations! You have just completed the resonator tutorial that should have provided you with a good working knowledge on how to use the eigenmode solver. The following topics have been covered:

1. General modeling considerations, using templates, etc.
2. Use a figure of rotation and a mirror transformation to model the cavity.
3. Define the frequency range, boundary conditions and symmetries.
4. Run the eigenmode solver and display eigenmode frequencies and field patterns.
5. Optimize the performance of the eigenmode solver for subsequent runs.
6. Calculate the Q-factors of the eigenmodes.
7. Check and, if necessary, improve the accuracy of the eigenmode solver.
8. Obtain accurate and converged results using the automatic expert system based mesh adaptation.
9. Use the JDM eigenmode solver.

You can obtain more information for each particular step from the online help system that can be activated either by pressing the *Help* button in each dialog box or by pressing the *F1* key at any time to obtain context sensitive information.

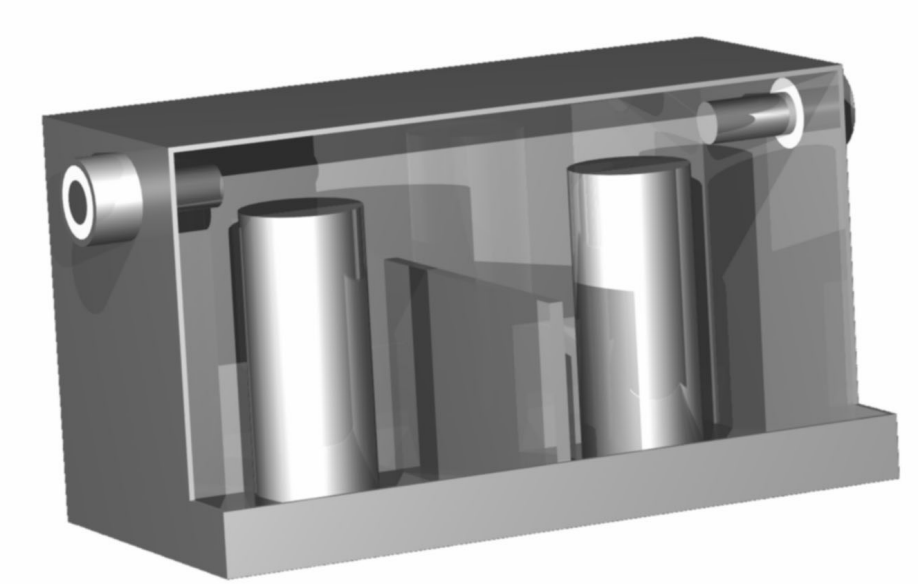
In some cases we have referred to the *Getting Started* manual, which is also a good source of information for general topics.

In addition to this tutorial, you can find some more eigenmode solver examples in the “examples” folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that will give you more information about the particular device.

Finally, you should refer to the *Advanced Topics* manual for more in-depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please also visit one of the training classes that are regularly held at a location near you. Thank you for using CST MICROWAVE STUDIO®!

Filter Tutorial



Geometric Construction and Solver Settings **194**

Introduction and Model Dimensions 194
Geometric Construction Steps 195
Geometric Solver Settings 211

S-Parameter Calculation **216**

Frequency Domain Solver – Resonant: Fast S-Parameters 217
Frequency Domain Solver – General Purpose 222

Getting More Information **233**

Geometric Construction and Solver Settings

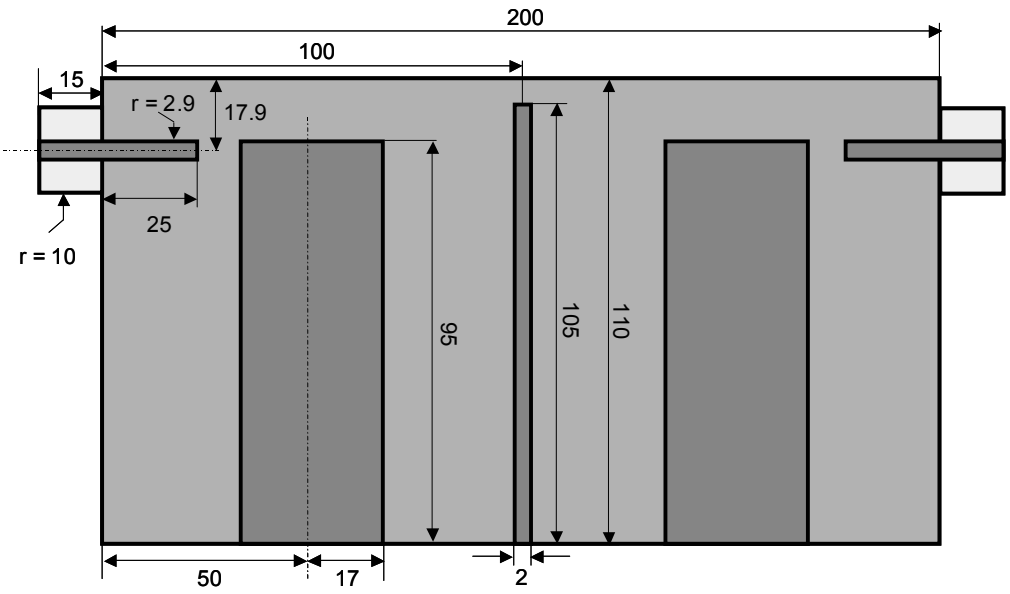
Introduction and Model Dimensions

In this tutorial you will learn how to simulate filter devices. As a typical example for a filter, you will analyze a Narrow Band Filter. The following explanations on how to model and analyze this device can be applied to other filter structures as well.

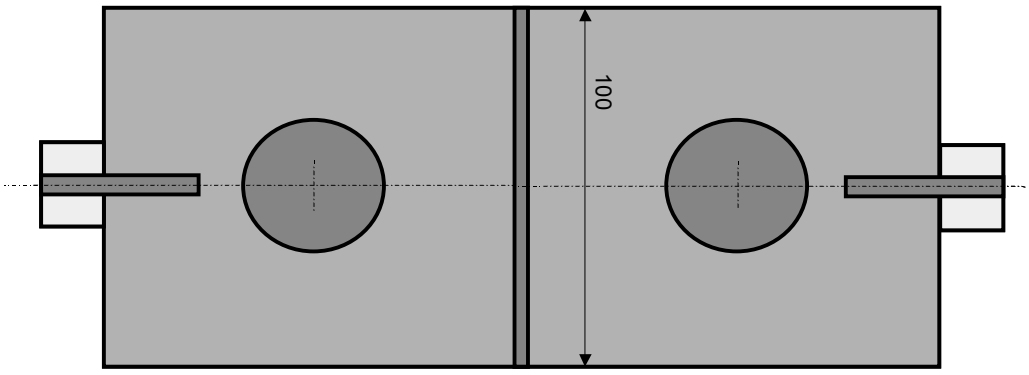
CST MICROWAVE STUDIO® can provide a wide variety of results. This tutorial however, concentrates solely on the S-parameters of the filter.

We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.

The following pictures show the structure and its dimensions in two different cross-sectional planes:



All dimensions are given in mm



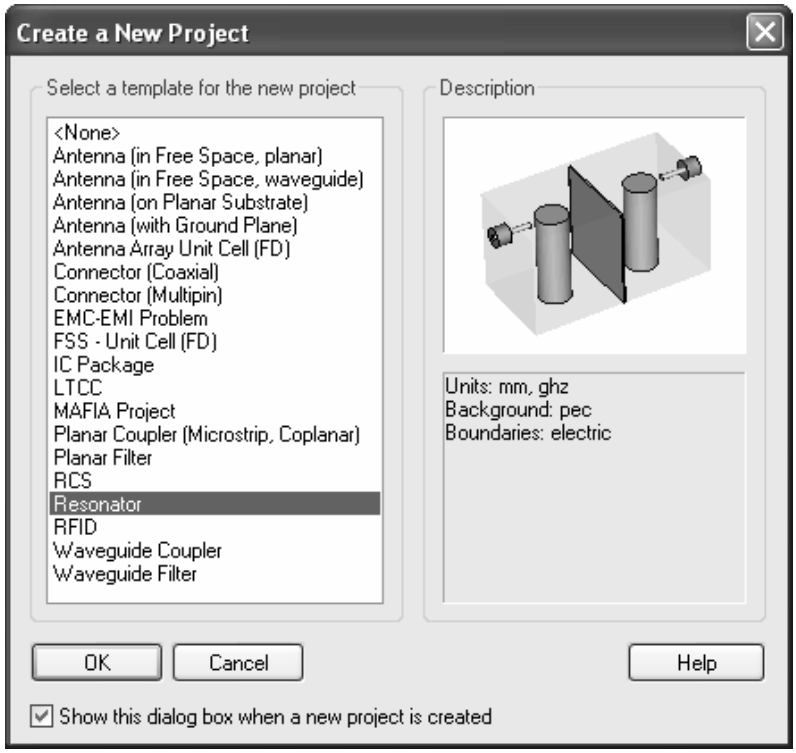
The structure consists of two resonators, each formed by a perfect electrically conducting cylinder in a rectangular cavity. Both resonators are coupled via a rectangular iris. The two coaxial ports are capacitively coupled to the device by extending the coaxial cable's inner conductor into the resonators.

Geometric Construction Steps

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

☐ Select a Template

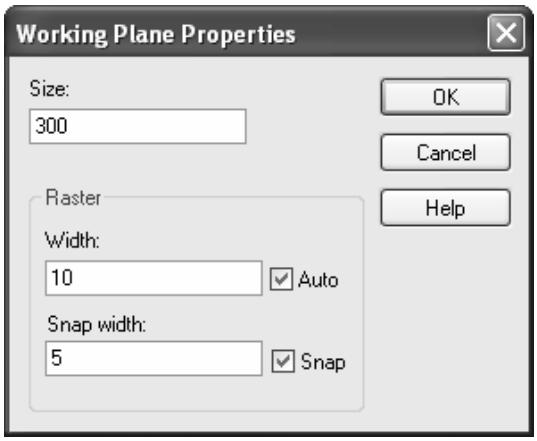
Once you have started CST DESIGN ENVIRONMENT™ and have chosen to create a new CST MICROWAVE STUDIO® project, you are requested to select a template that fits best to your current device. Here, the "Resonator" template should be chosen.



The template automatically sets the units to mm and GHz and the background material to be perfect electrically conducting. Please select this template now and press the *OK* button.

☐ Set the Working Plane's Properties

After the units have been correctly set (which has been done by the template here), the modeling process usually starts with setting the working plane's size large enough for the device. Because the structure has an extension of 200 mm along one coordinate direction, the working plane's size should be set to 300 mm (or more). These settings can be changed in a dialog box that opens after selecting *Edit ⇨ Working Plane Properties* from the main menu. Please note that we will use the same document conventions here as introduced in the *Getting Started* manual.



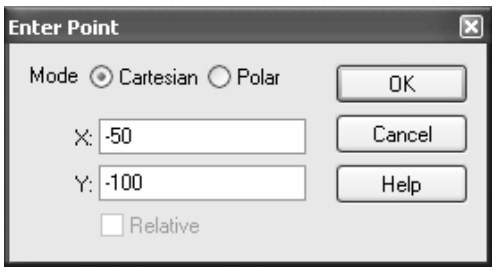
In this dialog box, you should set the *Size* to 300 (the unit that has been previously set to mm is displayed in the status bar), the *Raster Width* to 10 and the *Snap width* to 5 to obtain a reasonably spaced grid. Please confirm these settings by pressing the *OK* button.

☐ Draw the Filter's Housing

Because the background material has been set to electric, you need to model only the interior of the filter. The structure will then be automatically embedded within a perfect electrically conducting enclosure.

Therefore, you should start the structure modeling by entering the filter's housing that can easily be defined by creating an air-brick. Please activate the brick creation tool now by either selecting *Objects* ⇨ *Basic Shapes* ⇨ *Brick* or pressing the corresponding button in the toolbar ( ).

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

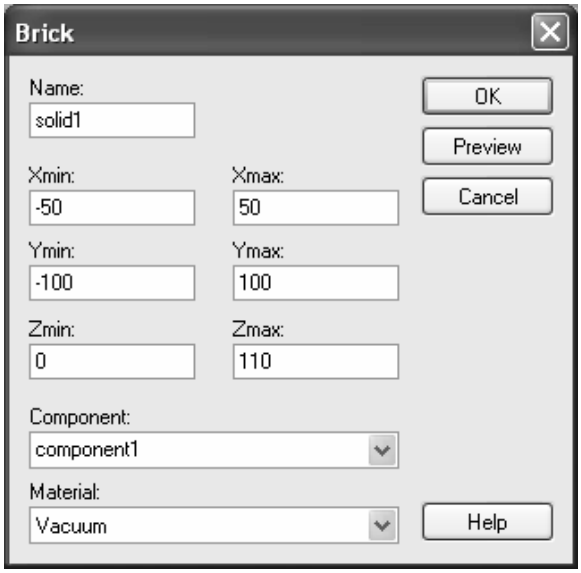


In this example you should create the housing with the transversal extension of 100 x 200 mm. In order to model the structure symmetrically to the origin, you should now enter the coordinates *X* = -50 and *Y* = -100 in the dialog box and press the *OK* button (please remember that the geometric unit is currently set to mm).

The next step is to enter the opposite corner of the brick's base: Press the TAB key again and enter *X* = 50, *Y* = 100 in the coordinate fields before pressing *OK*.

You will now be requested to enter the height of the brick. This can also be achieved by pressing the *Tab* key, entering a *Height* of 110 and pressing the *OK* button again.

After the steps above have been completed, the following dialog box will appear, displaying a summary of your input:

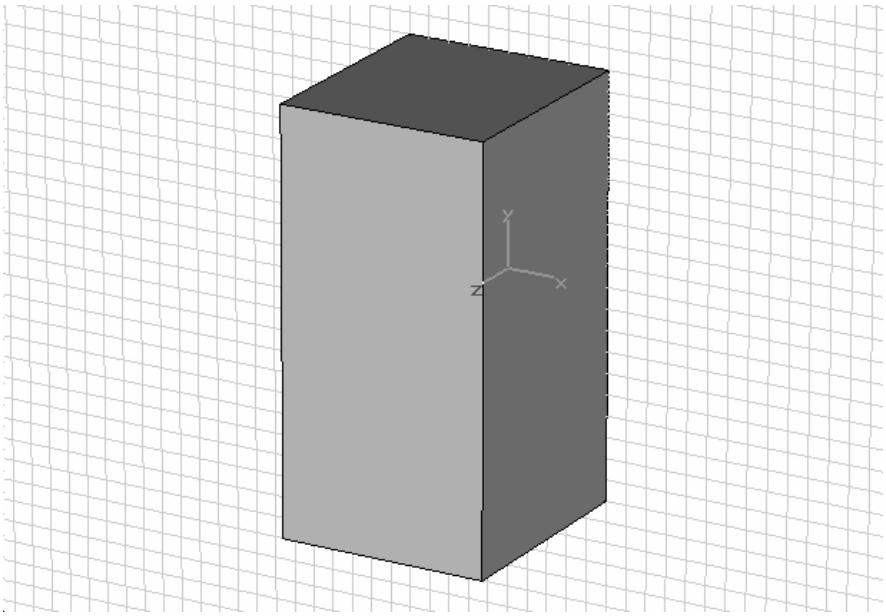


Please check all entries carefully. If you encounter any mistakes, please change the value in the corresponding entry field.

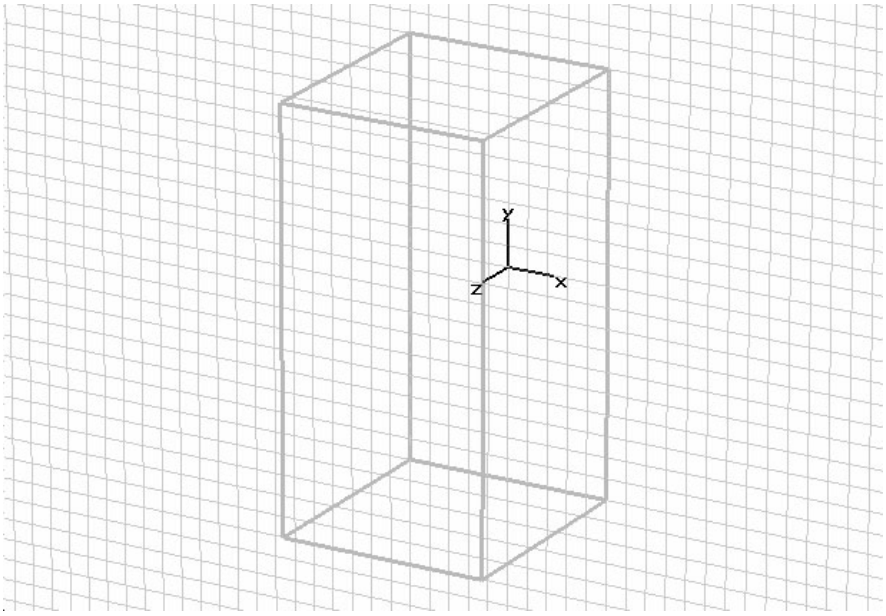
You should then give the shape a meaningful *Name* (e.g. “housing”). Because the housing consists of vacuum, you can keep the *Material* default setting (“Vacuum”) as well as the assignment to the default *Component* “component1”.

> **Please note:** The use of different components allows you to combine several solids into specific groups, independently of their material behavior. However, in this tutorial it is convenient to construct the complete filter device as a representation of one component.

Finally, confirm the creation by pressing *OK*. Your screen should now look as follows (you can press the *Space* key in order to zoom the structure to the maximum possible extent):



Because structures will be inserted into this air brick in the following steps, it is advantageous to switch the display to wireframe mode because otherwise the newly created shapes may be hidden inside the brick. The easiest method of activating the wireframe visualization mode is to press the toolbar icon or use the corresponding shortcut: *Ctrl+w*. The structure should now look as follows:



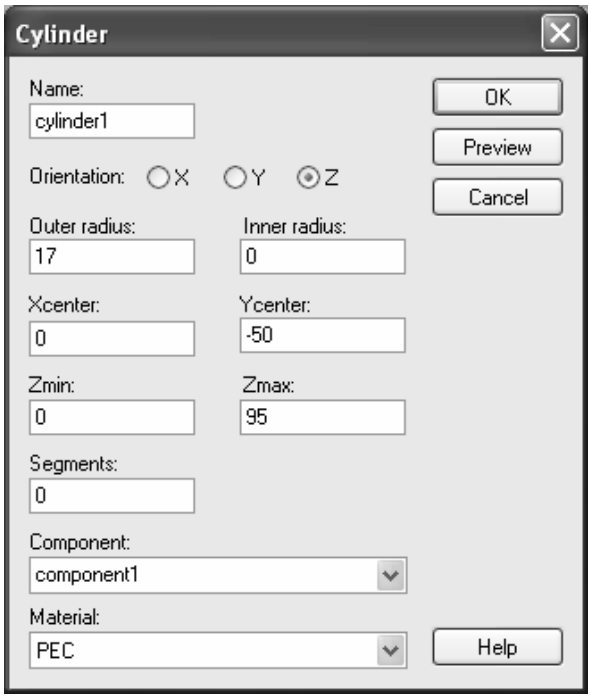
☐ Create the Cylindrical Resonators

The next step is to create the cylindrical resonators inside the air brick. Please activate the cylinder creation tool by either selecting *Objects* ⇨ *Basic Shapes* ⇨ *Cylinder* from the main menu or pressing the corresponding toolbar icon ( ).

The first step in the cylinder creation process is to enter the center point coordinates. This can be achieved numerically by pressing the *Tab* key and entering the dimensions *X* = 0, *Y* = -50 in the dialog box before pressing the *OK* button. In the following sections, we will assume that you always confirm the settings in a dialog box by pressing the *OK* button unless mentioned otherwise.

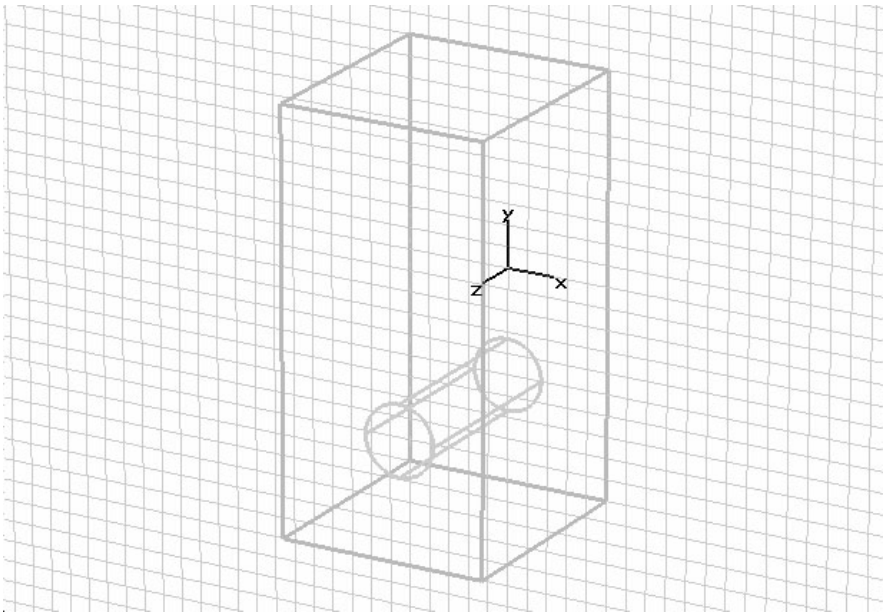
The second step in the cylinder's creation is to specify the outer radius. Similar to the procedure above, you should now set the *Radius* to 17 after pressing the *Tab* key.

After pressing the *Tab* key once more and setting the *Height* to 95, you may skip the definition of the cylinder's inner radius by pressing the *Esc* key. Finally, the following dialog box will appear:



Please check and correct all settings as necessary before specifying the cylinder's *Name* as "cylinder1". To this point, the cylinder consists of vacuum material. However, to specify the cylinder to be a perfect electrical conductor (PEC), you must change the *Material* assignment to "PEC". Because the filter is constructed as one component, you can skip the *Component* setting and confirm the creation of the cylinder by pressing the *OK* button.

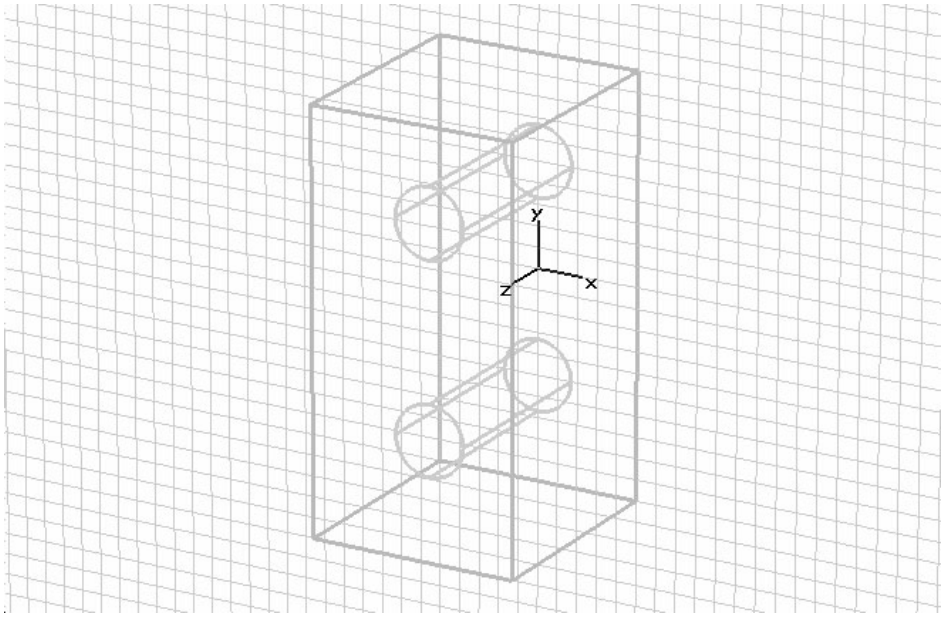
Your screen should then look as follows:



After successfully creating the first cylinder you can now model the second cylinder in the same manner:

1. Activate the cylinder tool: *Objects* ⇨ *Basic Shapes* ⇨ *Cylinder*, .
2. Press the *Tab* key and set the center's coordinates to *X* = 0, *Y* = 50.
3. Press the *Tab* key and set the *Radius* to 17.
4. Press the *Tab* key and set the *Height* to 95.
5. Press the *Esc* key to skip the definition of the inner radius.
6. Set the *Name* of the cylinder to "cylinder2"
7. Change the *Material* assignment to "PEC" and press the *OK* button.

After the successful creation of the second cylinder, the screen should then look as follows:

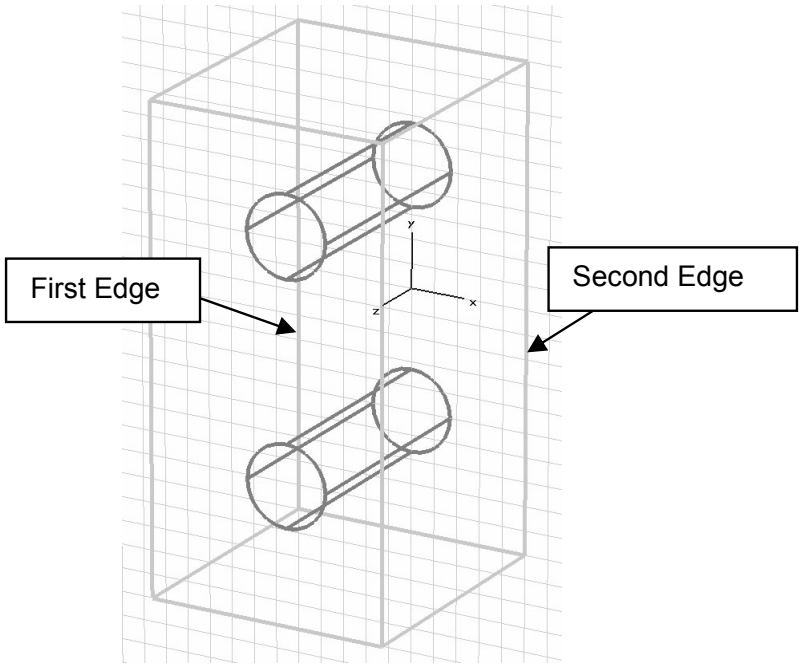


Please note: The creation of the second cylinder could also be achieved by applying a transformation to the first one. For the sake of simplicity, we recommended that you draw the cylinder twice. The application of transformations to copy shapes will be explained later in this tutorial.

☐ Create the Iris between the two Cavities

The next step is to create the rectangular iris between the two cavities. This could easily be achieved by entering its dimensions numerically in the same manner as the creation of the air brick above. However, because the iris should always extend across the entire width of the filter, we will now explain how this can be forced using picked points.

After activating the brick creation tool by selecting *Objects* ⇨ *Basic Shapes* ⇨ *Brick* or pressing the corresponding button in the toolbar ( ), you are requested to enter the first point. Instead of entering the point by double-clicking with the mouse or entering the point numerically by pressing the *Tab* key, you should now activate the pick midpoint tool (*Objects* ⇨ *Pick* ⇨ *Pick Edge Midpoint*,). Afterwards, all straight edges will be highlighted in the model:

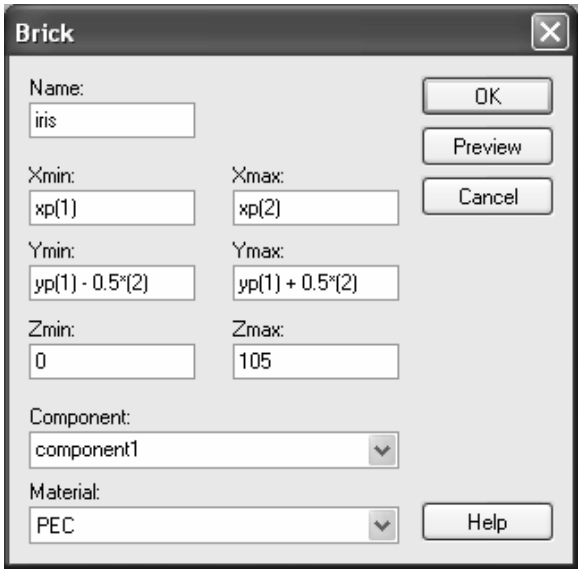


Double-click on the first edge shown in the picture above. By moving the mouse pointer you can now confirm that the first point of the brick is aligned with the mid-point of this edge. Even if the location of the midpoint changes (e.g. by parametrically editing the structure), the first point of the newly created brick will always be linked with the edge midpoint's current position.

You should now repeat the same steps (activate midpoint pick tool, double-click on the edge) with the second edge to specify the brick's second point.

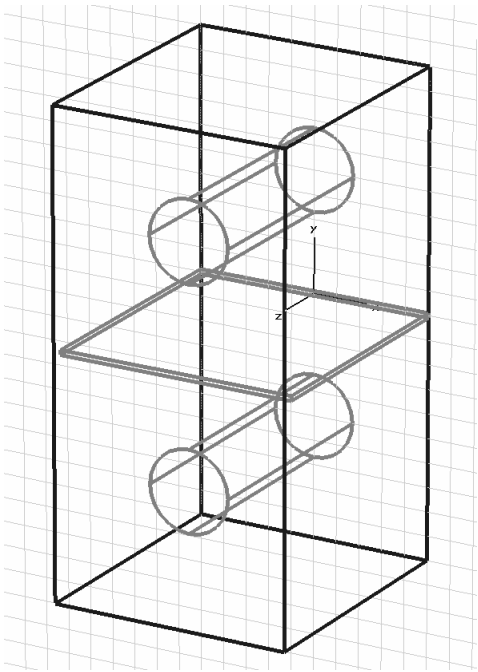
Because both points are now located on a line, the brick creation tool prompts for the width of the brick. You should now press the *Tab* key and set the *Width* of the brick to 2.

In the last step of the interactive brick creation you are requested to enter the brick's height. This can also be accomplished by pressing *Tab* and setting the *Height* to 105. Completing this step will open the following dialog box:



Some of the entry fields now contain expressions that reflect the relative construction of the brick. The expression xp(1), for instance, represents the x-coordinate of the initially picked edge's midpoint.

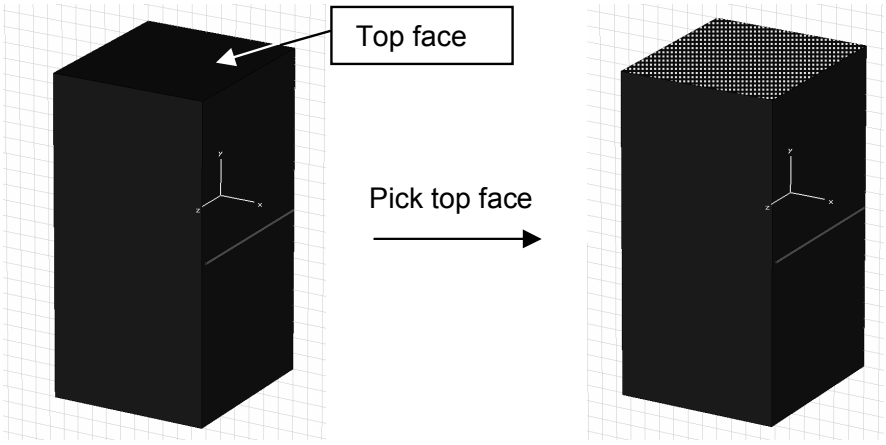
Set the *Name* of the brick to "iris" and the *Material* assignment to "PEC", then press the *OK* button. Your model should now look as follows:



☐ Create the Coaxial Couplings

To this point, you have modeled the filter's internal structure. However, the next step is to model the coaxial couplings on both side walls of the filter.

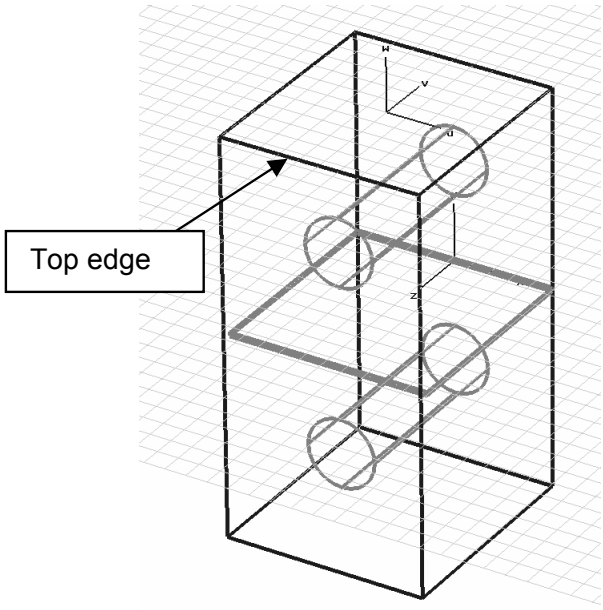
Before you begin modeling the cylinders, you should first align the working coordinate system with one of the side walls of the filter. This will allow you to model the coupling structure in a more convenient way. Please deactivate the wireframe plot mode by pressing the toolbar icon or using the shortcut *Ctrl*+*w*:



You should then activate the face pick tool (*Objects ⇨ Pick ⇨ Pick Face*, shortcut *f*) and double-click on the top face as shown above. The selected face should then be highlighted in the model (see picture above).

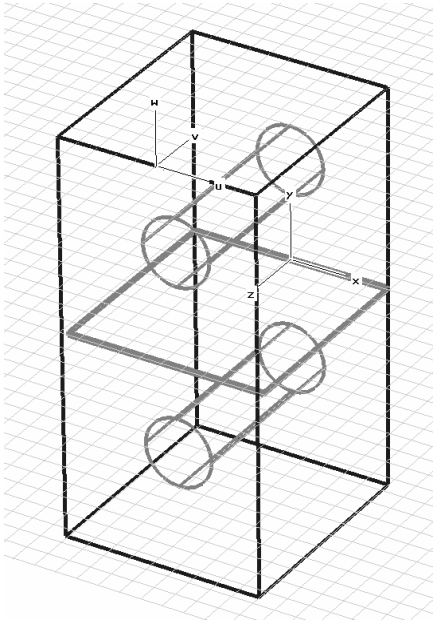
The next step is to align the working coordinate system with the picked face by selecting either *WCS ⇨ Align WCS with Selected Face*, pressing the toolbar button () or by using the shortcut *w* (while the main view is active).

After activating the wireframe drawing mode again (*Ctrl*+*w*), the model should look as follows:



The location of the coaxial coupler's center is located 17.9 mm below the top wall of the filter. Therefore, the next step is to align the working coordinate system with the top wall of the filter, which will make the definition of the coupler's location more convenient.

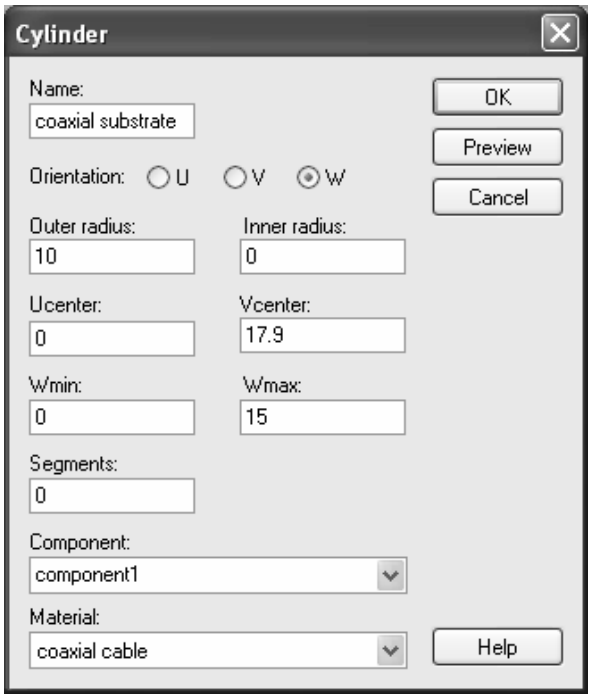
You should now again activate the midpoint pick tool (*Objects ⇨ Pick ⇨ Pick Edge Midpoint*, shortcut *m*) and double-click on the top edge shown in the above picture. Now the midpoint of this edge should become highlighted. You can then align the origin of the working coordinate system with this point by selecting *WCS ⇨ Align WCS with Selected Point*, pressing the toolbar button or just using the shortcut *w*. The following picture shows the new location of the working coordinate system:



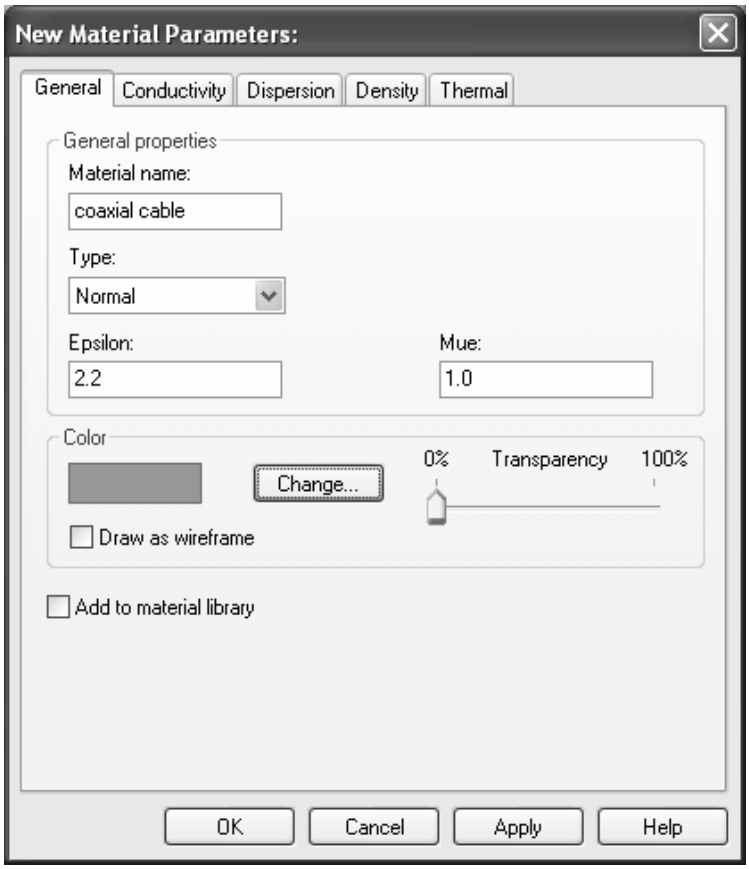
With the working coordinate system aligned this way, the construction of the coaxial coupler is straightforward:

1. Activate the cylinder tool: *Objects ⇨ Basic Shapes ⇨ Cylinder*, .
2. Press the *Tab* key and set the center's coordinates to *U* = 0, *V* = 17.9.
3. Press the *Tab* key and set the *Radius* to 10.
4. Press the *Tab* key and set the *Height* to 15.
5. Press the *Esc* key to skip the definition of the inner radius.
6. Set the *Name* of the cylinder to "coaxial substrate".

The cylinder creation dialog box should then look as follows:



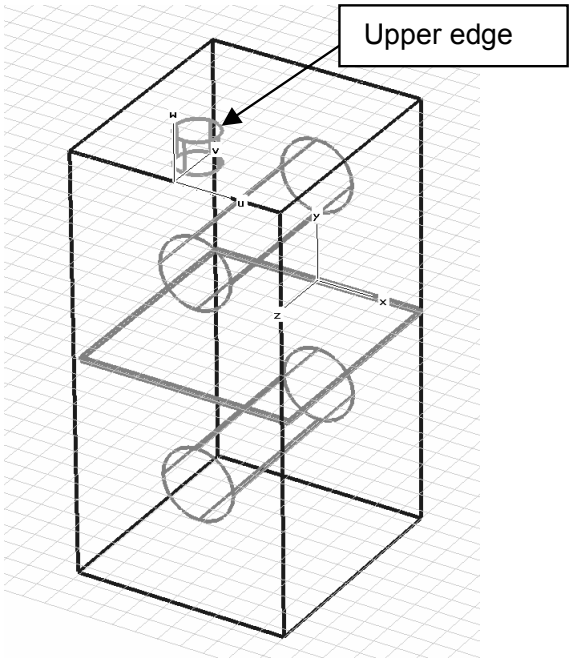
You still need to define the substrate material. Because no material has yet been defined for the substrate, open the material definition dialog box by selecting "[New Material…]" in the *Material* dropdown list:



In this dialog box, you should first define a new *Material name* (e.g. “coaxial cable”) and set the *Type* to a “Normal” dielectric material. Then specify the material properties in the *Epsilon* and *Mue* fields. Here, you only need to change the dielectric constant *Epsilon* to 2.2. Finally, select a color for the layer by pressing the *Change* button. Your dialog box should now look similar to the above picture before you press the *OK* button.

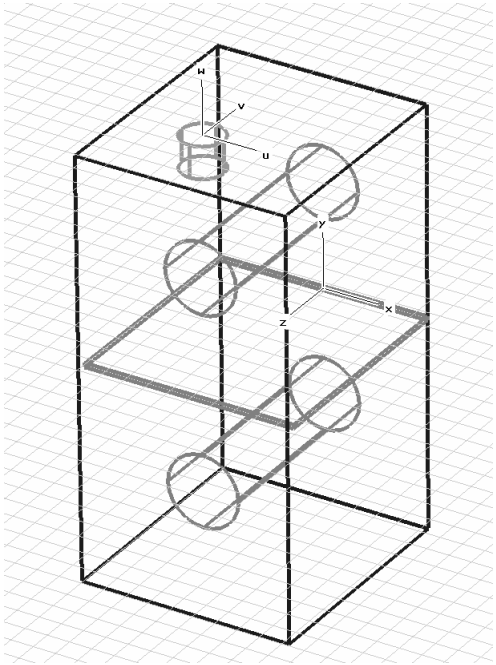
> **Please note:** The defined material “coaxial cable” will now be available inside the current project for the creation of other solids. However, if you also want to save this specific material definition for other projects, you may check the button *Add to material library*. You will have access to this material database by clicking on *Load from Material Library* in the *Materials* context menu in the navigation tree.

Back in the cylinder creation dialog box you can also press the *OK* button to finally create the coaxial coupler’s substrate. Your model should now look as follows:



The next step is to model the inner conductor of the coaxial coupler as a perfect electrically conducting cylinder. Because both cylinders should always be coaxial, it is convenient to move the local coordinate system to the center of the substrate cylinder.

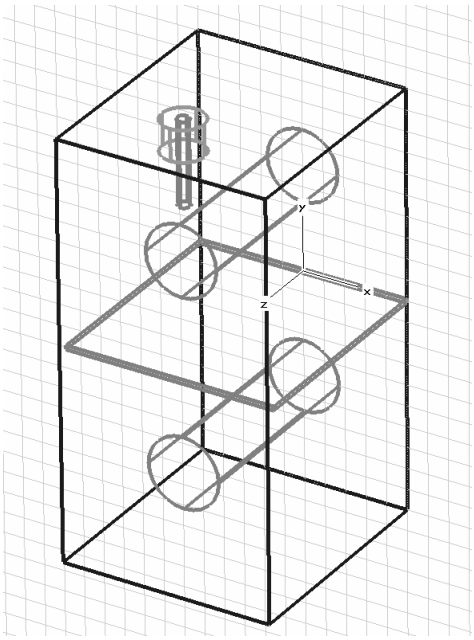
Therefore, activate the circle center pick tool by either selecting *Objects* ⇨ *Pick* ⇨ *Pick Circle Center* from the main menu or pressing the corresponding button in the toolbar, shortcut *c* . Now double-click on the substrate cylinder’s upper edge as shown in the above picture which will highlight the circle center point. Finally, align the working coordinate system with this point by selecting *WCS* ⇨ *Align WCS with Selected Point* or pressing the toolbar button, shortcut *w* . The following picture shows how your model should now look:



The inner conductor of the coaxial connector can now be easily modeled by performing the following operations to create a cylinder:

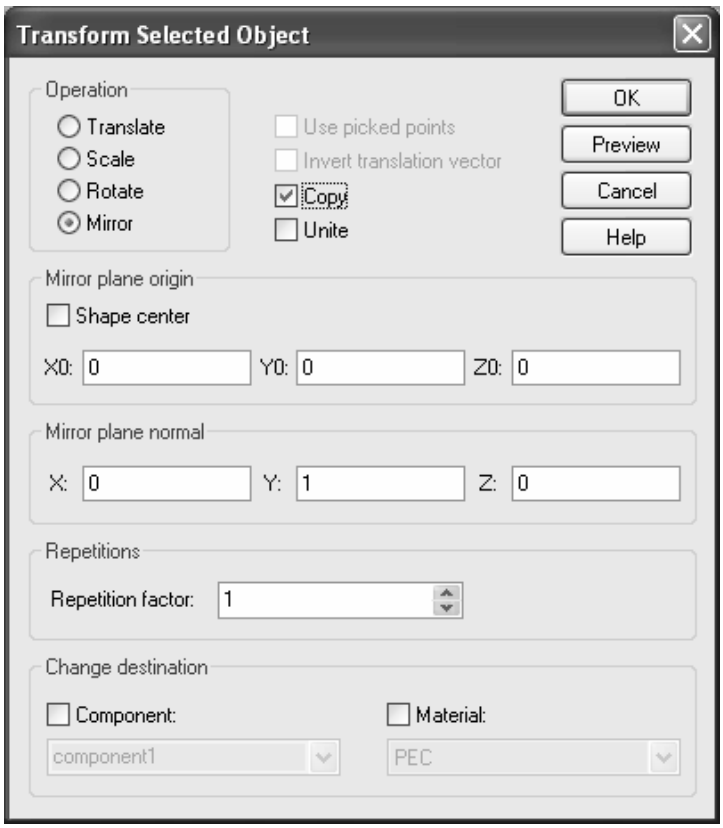
1. Activate the cylinder tool: *Objects* ⇨ *Basic Shapes* ⇨ *Cylinder*, .
2. Press the *Tab* key and set the center's coordinates to *U* = 0, *V* = 0.
3. Press the *Tab* key and set the *Radius* to 2.9.
4. Press the *Tab* key and set the *Height* to -40.
5. Press the *Esc* key to skip the definition of the inner radius.
6. Set the *Name* of the cylinder to "conductor".
7. Change the *Material* to perfect electric conducting ("PEC").
8. Press the *OK* button to finally create the cylinder.

The next step is to deactivate the working coordinate system by selecting *WCS* ⇨ *Local Coordinate System* or pressing the corresponding toolbar button, . After completion of these steps, the model should now look as follows:



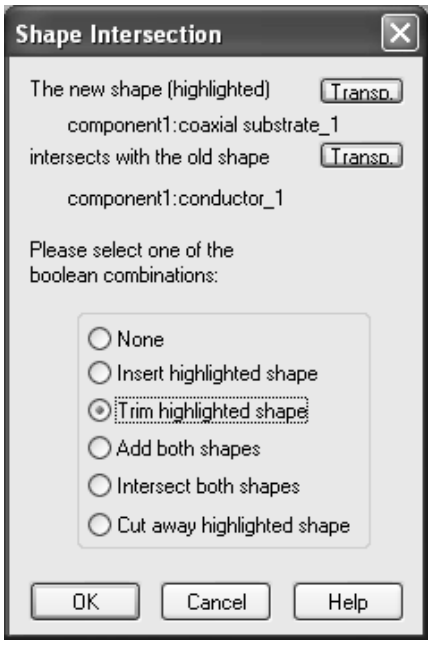
To this point, you have modeled one coaxial coupler but still need to create the second coupler. This is most conveniently achieved by creating a mirrored copy using an appropriate shape transformation. With help of multiple selections, this must be done only once for the complete coaxial coupler. Please select both parts of the connector (*Components ⇨ component1 ⇨ coaxial substrate* and *Components ⇨ component1 ⇨ conductor)* while holding the *Ctrl* key.

Afterwards, open the shape transformation dialog box (*Objects ⇨ Transform* or press the corresponding icon ( )):

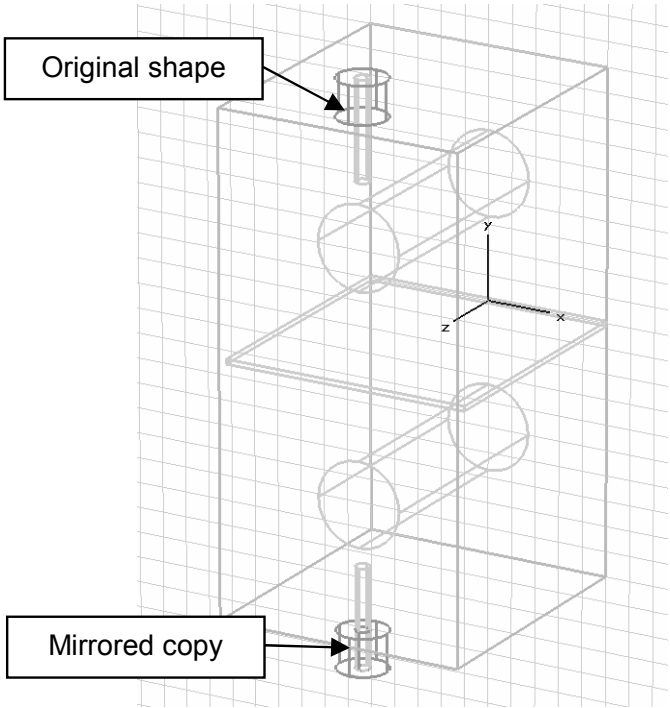


The first action in this dialog is to set the *Operation* to *Mirror*. Then the parameters of the mirror plane are specified. Because this plane should be the XZ plane of the global coordinate system, you only need to set the *Y* coordinate of the *Mirror plane normal* to 1. To create a mirrored copy of the existing multiple selected shape, please enable the option *Copy*. The newly created solids will then also be grouped to the existing component “component1”. Confirm the settings by pressing *OK*.

In the “Shape Intersection” dialog you must mark the radio button *Trim highlighted shape* to get the same coaxial connector as above. Confirm the setting by pressing *OK*.



You end up with the following picture:



Geometric Solver Settings

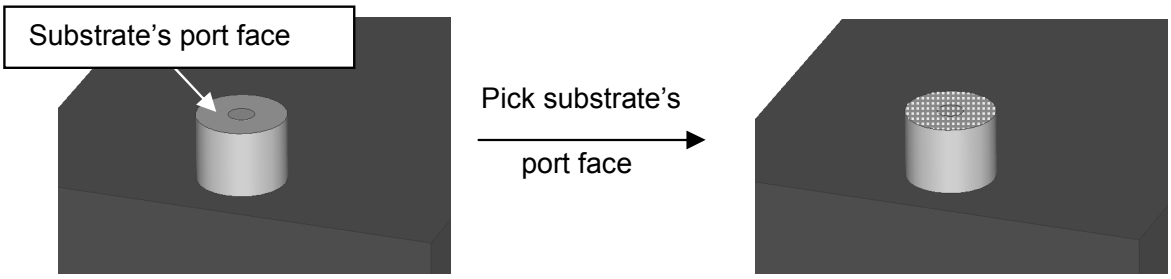
☐ Define Ports

The next step is to add the ports to the filter for which the S-parameters will be calculated. Each port will simulate an infinitely long waveguide (here a coaxial cable) that is connected to the structure at the port's plane. Waveguide ports are the most accurate way to calculate the S-parameters of filters and should thus be used here.

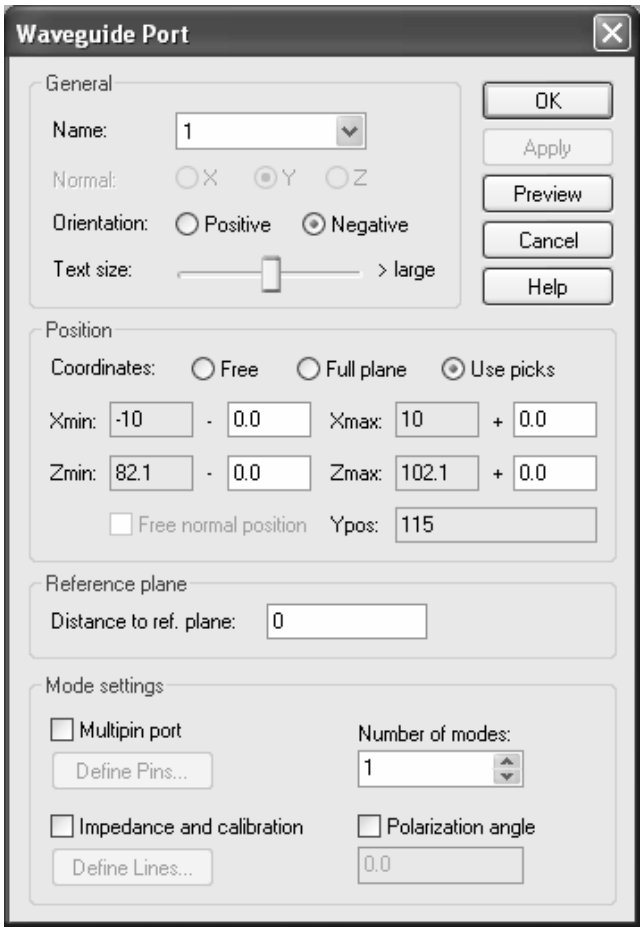
Because a waveguide port is based on the two dimensional mode patterns in the waveguide's cross-section, it must be defined large enough to entirely cover these mode fields. In the case of a coaxial cable, the port therefore must cover the coaxial cable's substrate completely.

Before you continue with the port definition, please deactivate the wireframe visualization by pressing the toolbar icon or using the shortcut *Ctrl+w*.

The port's extent can either be defined numerically or, more conveniently, by picking the face to be covered by the port. Therefore, please activate the pick face tool (*Objects* ⇨ *Pick* ⇨ *Pick Face*, shortcut *f*) and double-click on the substrate's port face on one of the coaxial couplers as shown below:



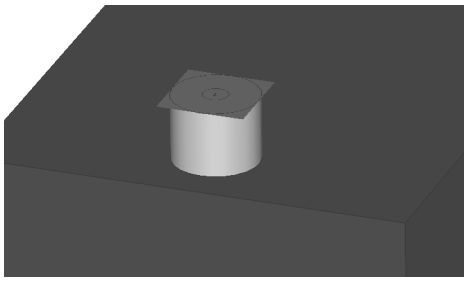
Please now open the waveguide dialog box (*Solve* ⇨ *Waveguide Ports*,) to define port 1:



Whenever a face is picked before the port dialog is opened, the picked face's extent will automatically define the port's location and size. Thus, the port's *Position* is initially set to *Use picks* for the coordinates. You can simply accept this setting.

The next step is to choose the number of modes to be considered by the port. For coaxial devices, we usually have only a single propagating mode. You should therefore keep the default of one mode.

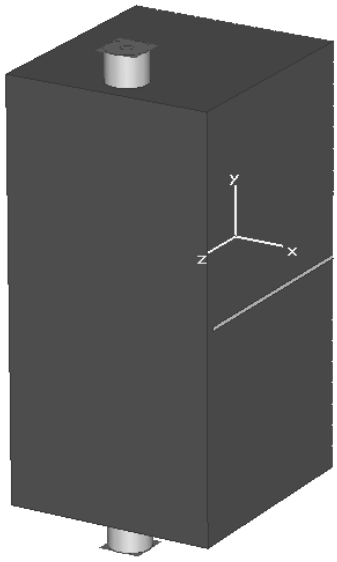
Finally, check the settings in the dialog box and press the *OK* button to create the port:



Now you can repeat the same steps for the definition of port 2.

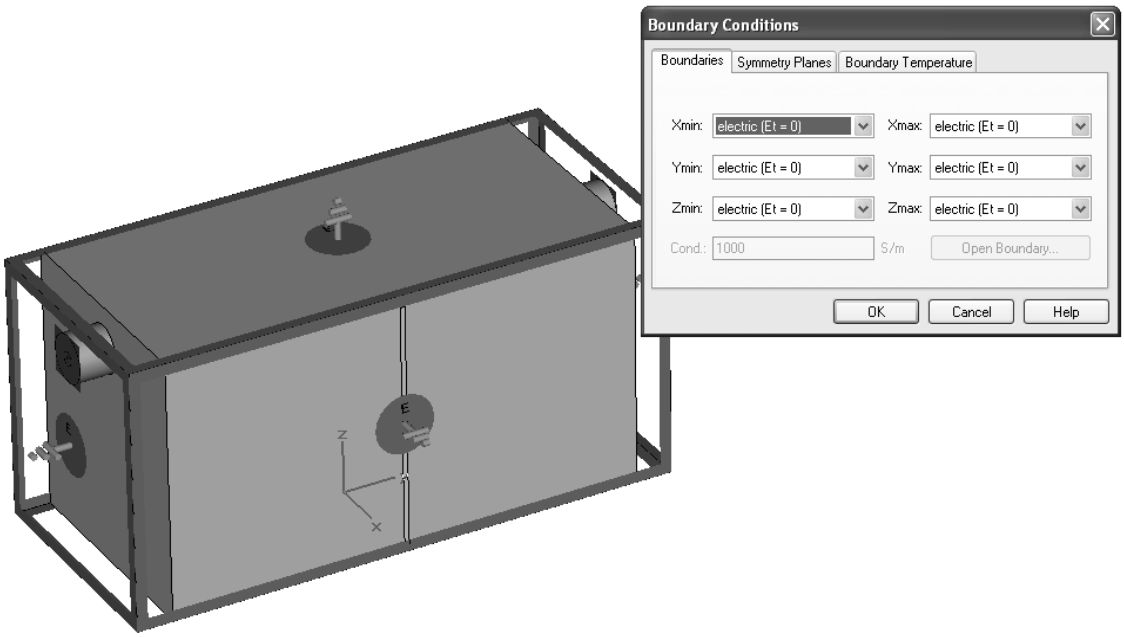
1. Pick the corresponding substrate's port face (*Objects ⇨ Pick ⇨ Pick Face*,).
2. Open the waveguide dialog box (*Solve ⇨Waveguide Ports*,).
3. Press *OK* to store the port's settings.

Your model should now look as follows:



☐ Define Boundary Conditions and Symmetries

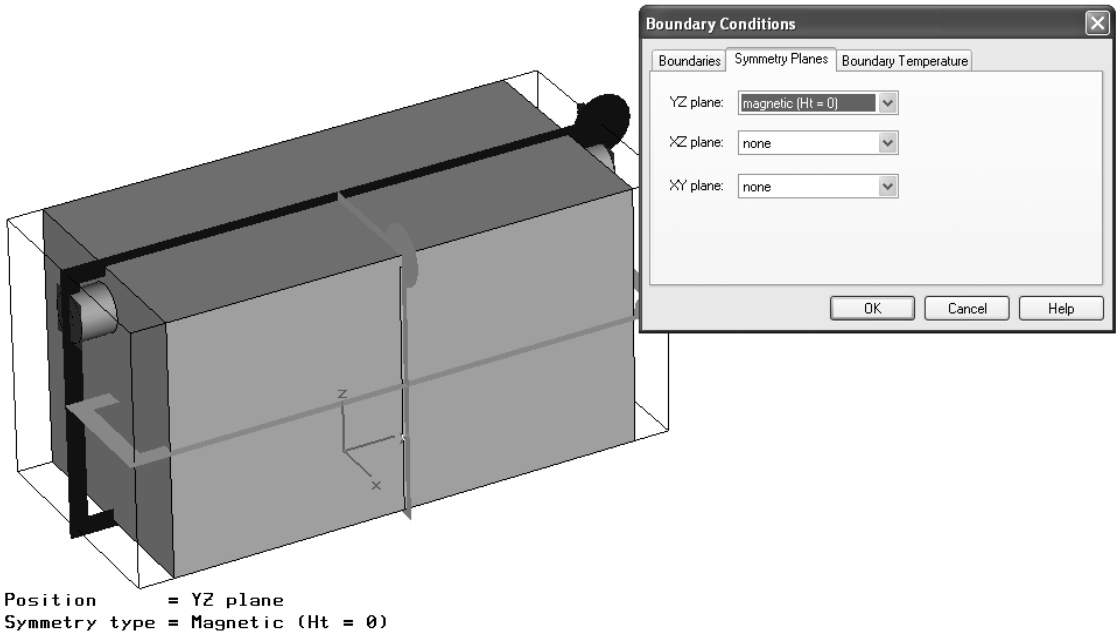
Always check the boundary and symmetry conditions before starting the solver. This is most easily accomplished by entering the boundary definition mode by pressing the tool bar item or selecting *Solve* ⇨ *Boundary Conditions*. The boundary conditions will then become visualized in the main view as follows:



Here, all boundary conditions are set to “electric,” which means that the structure is embedded in a perfect electrically conducting housing. These defaults (that have been set by the template) are appropriate for this example.

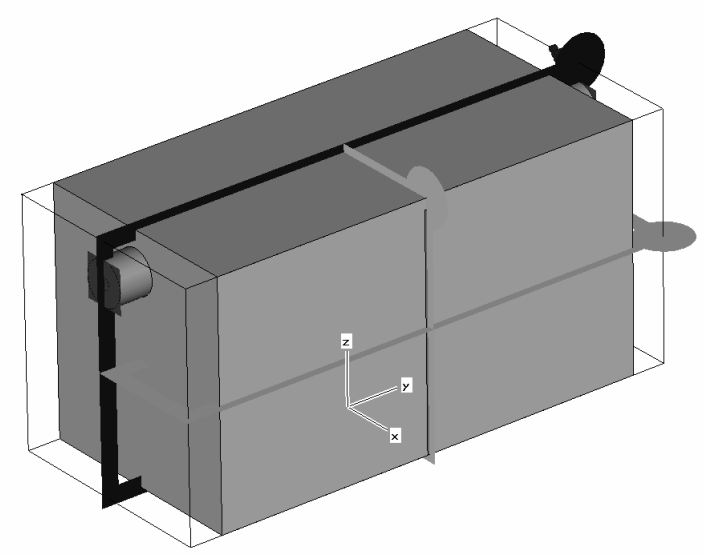
Due to the structure’s symmetry about the YZ plane and the fact that the magnetic field in the coaxial cable is perpendicular to this plane, a symmetry condition can be used. This symmetry will reduce the time required for the simulation by a factor of two. You should also refer to the example in the *Getting Started* manual for more information on symmetry conditions.

Please enter the symmetry plane definition mode by activating the *Symmetry planes* tab in the dialog box.



By setting the symmetry plane YZ to magnetic, you force the solver to calculate only the modes that have no tangential magnetic field component to these planes (thus forcing the electric field to be tangential to these planes).

After these settings have been made, the structure should look as follows:



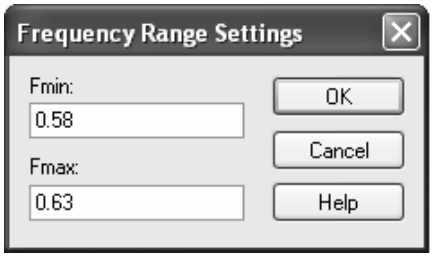
Please note that you could also double-click on the symmetry plane's handle and choose the proper symmetry condition from the context menu.

Finally press the *OK* button to complete this step.

In general, you should always make use of symmetry conditions whenever possible to reduce calculation times by a factor of two to eight.

☐ Define the Frequency Range

The frequency range for this example extends from 0.58 GHz to 0.63 GHz. Change *Fmin* and *Fmax* to the desired values in the "Frequency Range Settings" dialog box (that is opened either by pressing the toolbar icon ∿ or choosing *Solve* ⇨ *Frequency*) and store these settings by pressing the *OK* button (the frequency unit which has previously been set to GHz is shown in the status bar).



S-Parameter Calculation

After completing the above steps, you are ready to start the S-parameter computation. The filter device analyzed here is a narrow band filter for which four different solution methods can be used:

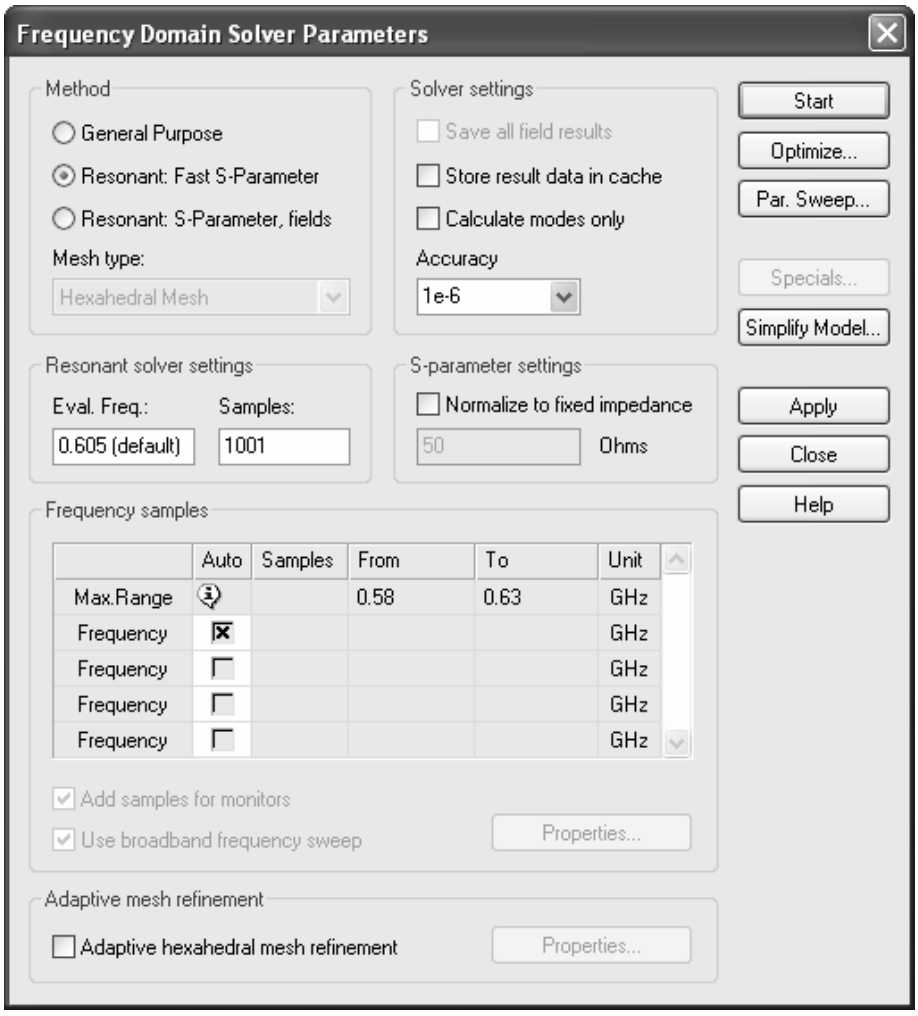
1. Transient solver
2. Frequency domain solver: "General Purpose"
3. Frequency domain solver: "Resonant: Fast S-Parameter"
4. Frequency domain solver: "Resonant: S-Parameter, fields"

The transient solver is the most versatile tool to solve any kind of S-parameter problem. However, for strongly resonant structures (such as the filter demonstrated here), the frequency domain solvers provide an interesting alternative that can be more computationally efficient than the transient solver. Consequently, in this chapter you will learn to use two different types of frequency domain solvers to analyze the device. Please refer to the *Getting Started* manual and the other tutorials on how to utilize the transient solver for S-parameter problems. In the first section, the S-parameters are calculated using the "Resonant: Fast S-Parameter" solver with a hexahedral mesh, while the second simulation is performed with the "General Purpose" solver and the tetrahedral mesh. Two completely different approaches to solving the problem provide proof of the simulation's reliability. Both sections are self-contained parts and it is sufficient to work through only one of them, depending on what solver you are interested in. The chapter ends with a comparison of the two methods.

> **Please note:** Some solvers may not be available to you due to license restrictions. Please contact your sales office for more information.

Frequency Domain Solver – Resonant: Fast S-Parameters

For S-parameter calculations with the frequency domain solver tool you should open the frequency domain solver control dialog box by pressing the corresponding toolbar icon  or selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu.



Since the narrow band filter is a strongly resonant device, we will demonstrate how to use a highly specialized solver for this application.

Click on the “Resonant: Fast S-Parameter” button that enables the corresponding solver. Please check your settings in the dialog box before pressing the *Start* button. A few progress bars will appear, keeping you informed about the current status of your calculation (e.g. matrix calculation …):

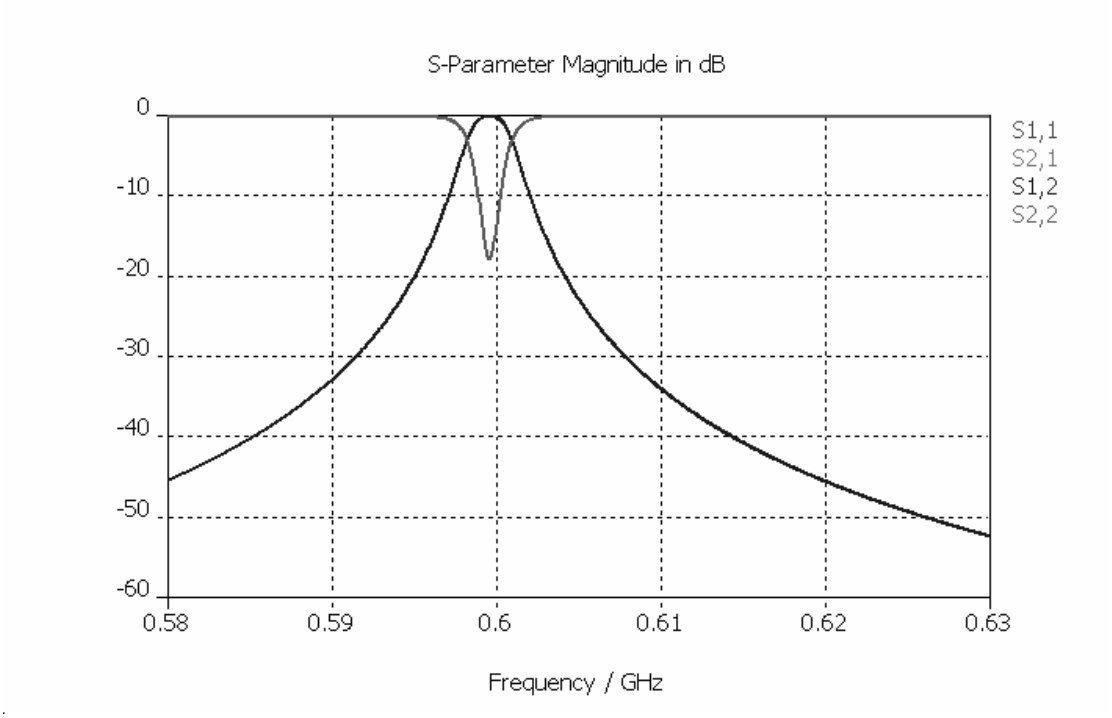
To review the solver time required to achieve these results, you can display the solver log-file by selecting *Results* ⇨ *View Logfiles* ⇨ *Solver Logfile* from the main menu. Please scroll down the text to obtain the following timing information (the actual values may vary depending on the speed of your computer):

```
Mesh generation time     :        4 s
Solver time              :        4 s

Total time               :        8 s

-----------------------------------------------------------------
```

Congratulations, you have simulated the Narrow Band Filter! Let's review the results.

☐ 1D Results (S-Parameters)

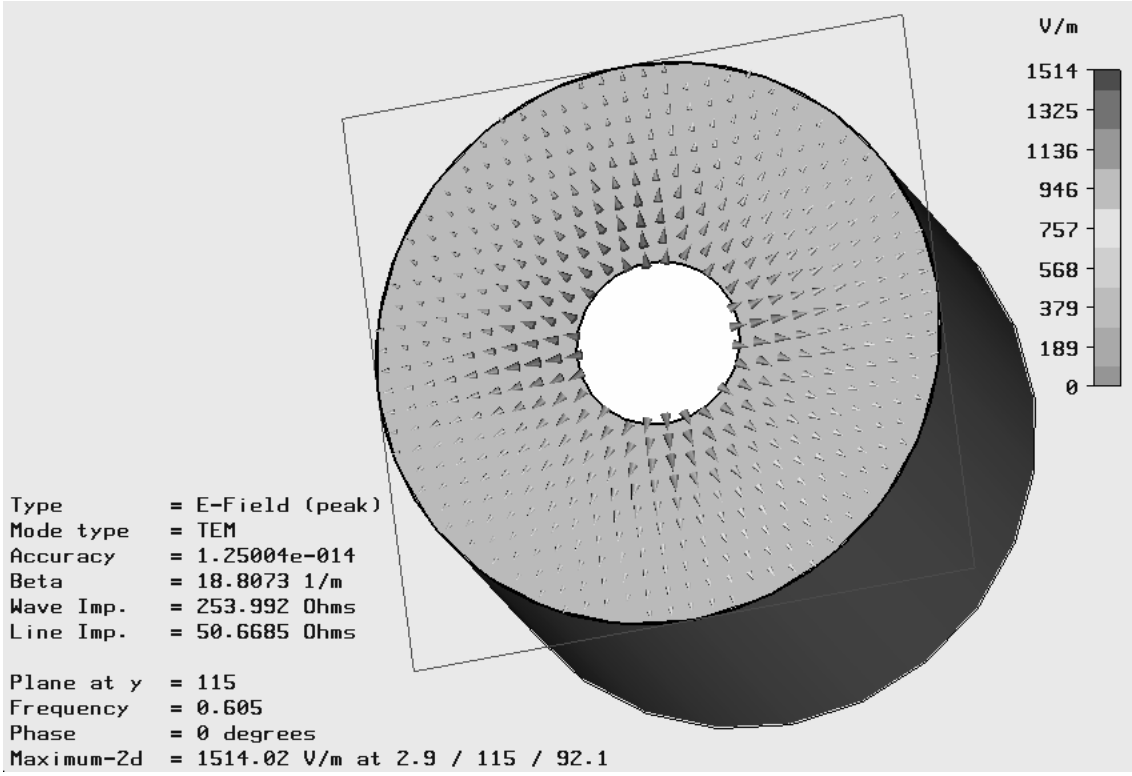
The S-Parameter magnitude in dB scaling can be plotted by clicking on the *1D Results* ⇨ */S/dB* folder.



As expected, the bandwidth of the transmission S2,1 is quite small. It is also obvious that this filter is somehow detuned.

☐ 2D Results (Port Modes)

After inspecting the S-parameters, you can display the 2D 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 fundamental port mode, click on the *e1* folder.



The plot also shows some important properties of the mode, such as mode type, propagation constant and line impedance. The port modes at the second port can be visualized in the same manner.

☐ Accuracy Considerations

The frequency domain solver modules are mainly affected by two sources of numerical inaccuracies:

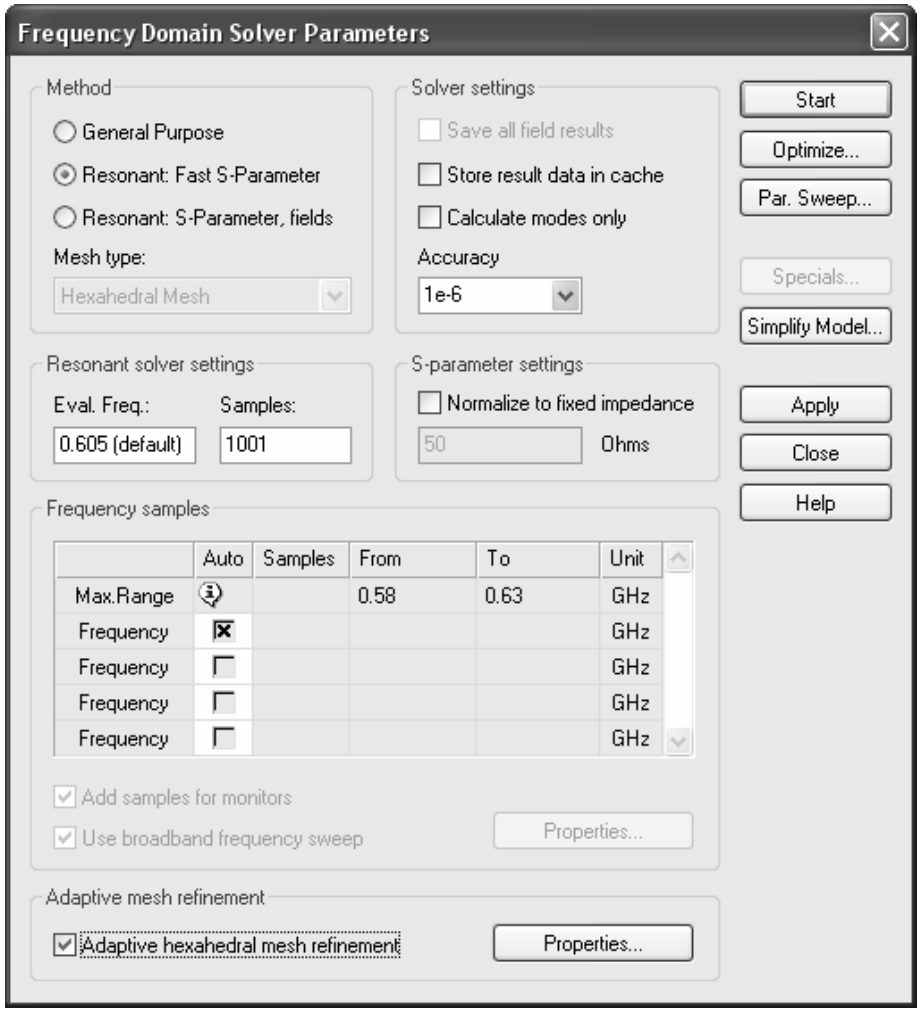
1. Numerical errors introduced by the linear equation system solvers.
2. Inaccuracies arising from the finite mesh resolution.

In the following sections we provide hints on how to minimize these errors and achieve highly accurate results.

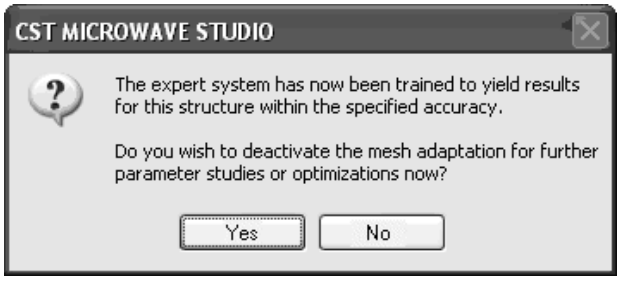
The first type of error is always quantified as *Accuracy* for the solution of a linear equation system. Decreasing this number improves the solutions. The default value of 1e-6 for the “Resonant: Fast S-Parameter” solver is usually sufficient.

The 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 desired results no longer significantly change as 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 easiest way to prove the accuracy of the results is to use the fully automatic mesh adaptation that can be switched on by checking the *Adaptive mesh refinement* option in the solver control dialog box (*Solve ⇨ Frequency Domain Solver):*

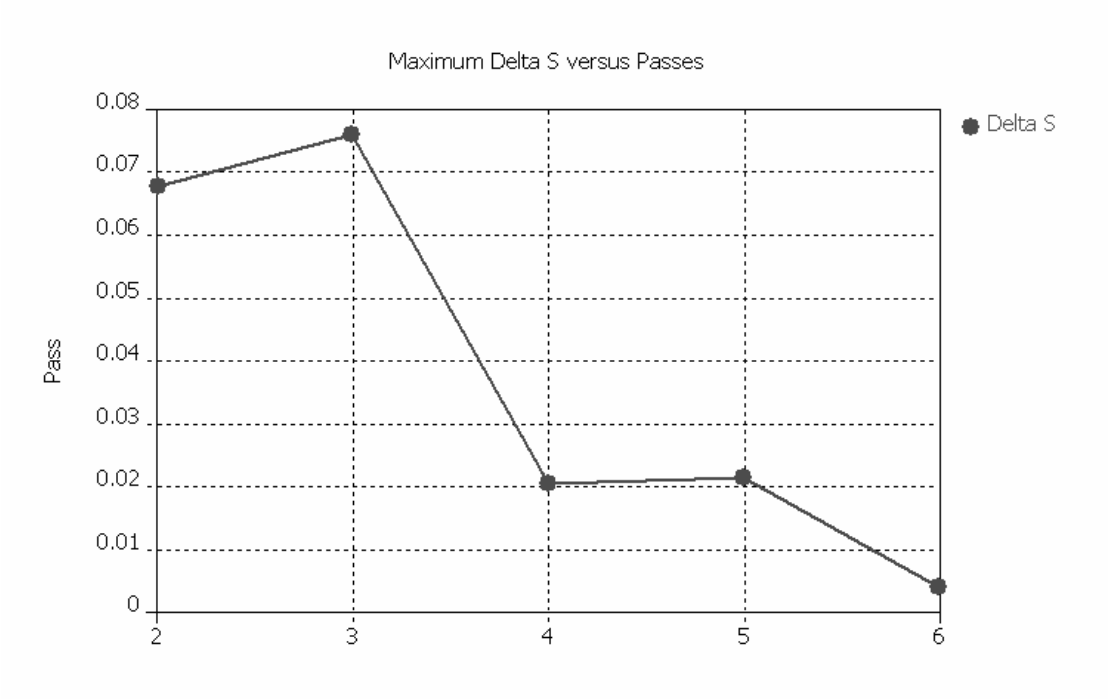


After activating the *Adaptive hexahedral mesh refinement* tool, you should now start the solver again by pressing the *Start* button. After a couple of minutes (during which the solver is running through mesh adaptation passes), the following dialog box will appear:

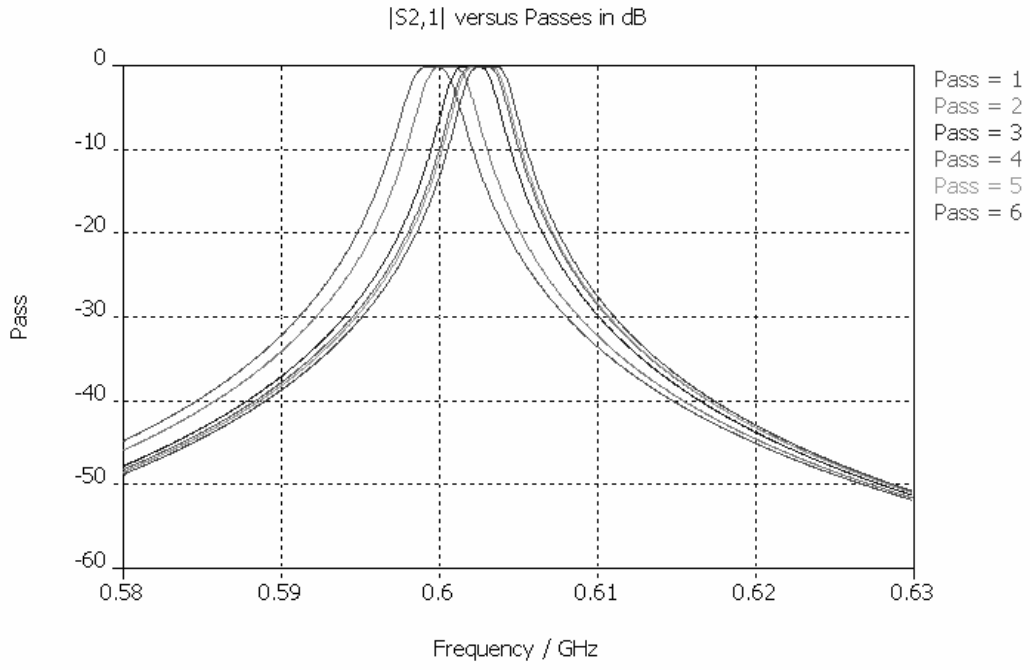


This dialog box notifies you that the desired accuracy limit of the S-parameters (2% by default) could be met by the adaptive mesh refinement. Because the expert system's settings have now been adjusted to achieve this level of accuracy, you may switch off the adaptation procedure for subsequent calculations (e.g. parameter sweeps or optimizations).

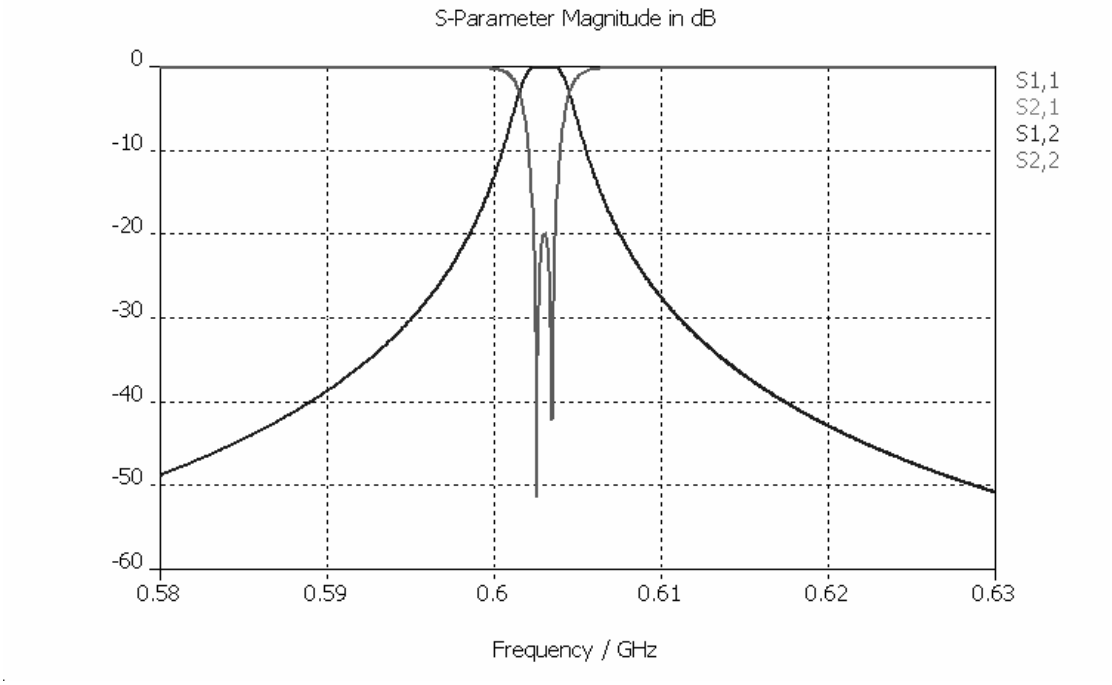
After the mesh adaptation procedure is complete, you can visualize the maximum relative difference of the S-parameters for two subsequent passes by selecting *1D Results ⇨ Adaptive Meshing ⇨ Delta S* from the navigation tree:



As evident from the above figure, the automatic mesh adaptation procedure requires six passes to achieve convergence for this challenging filter structure. This indicates that it is necessary to run the mesh adaptation or perform a manual mesh refinement to achieve highly accurate results. When examining the progression of the transmission S1,2 during the adaptive mesh refinement (*1D Results ⇨ Adaptive Meshing ⇨ |S| dB⇨ S1,2*), you can observe that the pass band has moved towards higher frequencies in the course of the adaptive mesh refinement:



Finally, the most interesting results are the S-parameters as shown below:



The final S-parameters differ significantly from the first results obtained with the initial mesh created by the expert system. In general, accurate S-parameter results for filter structures can only be obtained by mesh convergence studies. These studies can be carried out either by manually changing the settings of the expert system or, as demonstrated here, by running the automatic mesh adaptation tool.

The huge advantage of this expert system based mesh refinement procedure over traditional adaptive schemes is that the mesh adaptation must be carried out only once for each device to determine the optimum settings for the expert system. Afterwards, there is no need for time-consuming mesh adaptation cycles during parameter sweeps or optimizations.

Frequency Domain Solver – General Purpose

☐ Making a Copy of the Previous Solver Results

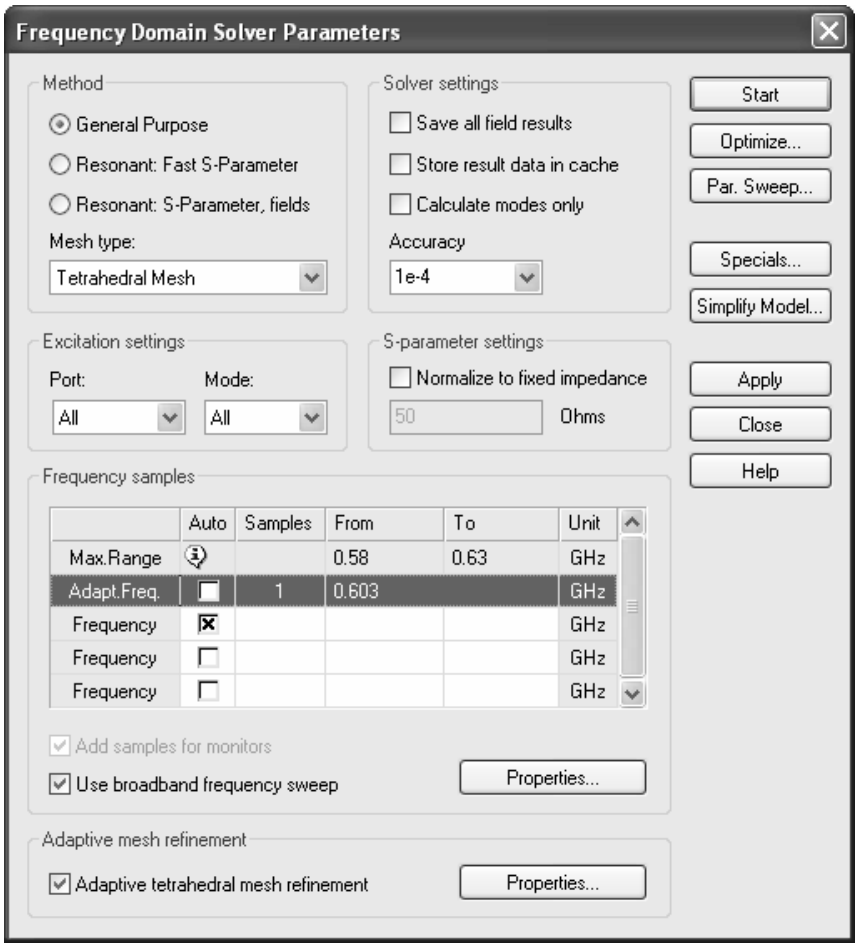
Before performing the simulation with another frequency domain solver, you may want to keep the previous results to compare the two simulations. To obtain the copy of the current results: Select, for example, the |*S*| *dB* folder in *1D Results*, then press *Ctrl+c* and *Ctrl+v.* The copies of the results will be created in the selected folder. The names of the copies will be *S1,1_1, S2,1_1* etc. You may rename them to *S1,1_RF*, *S2,1_RF* and so on using the *Rename* command from the context menu.

Now the “General Purpose” frequency domain solver will be used for the S-parameter calculations. Open the frequency domain solver control dialog box by pressing the corresponding toolbar icon  or selecting *Solve* ⇨ *Frequency Domain Solver* from the main menu. Select “General Purpose” in the method frame. A main advantage of the

General Purpose frequency domain solver is the possibility to choose between two mesh types: Tetrahedral and Hexahedral mesh. For this example, we change the mesh type to "Tetrahedral Mesh". With the default settings the frequency points are determined automatically in the desired frequency range and the S-parameters are interpolated if the button *Use broadband frequency sweep* is activated.

Because the General Purpose frequency domain solver with tetrahedral mesh runs a single frequency adaptive mesh refinement, the adaptation frequency should be moved to the pass band of the filter. From the previous solver run using the "Resonant: Fast S-Parameter" solver, we know that the center of the pass band is close to 0.603 GHz. By default, the adaptation frequency is automatically chosen as the uppermost frequency of the global frequency range. In order to change this behavior, uncheck the "Auto" button in the *Adapt. Freq*. row of the list of frequency samples, and enter the desired adaptation frequency.

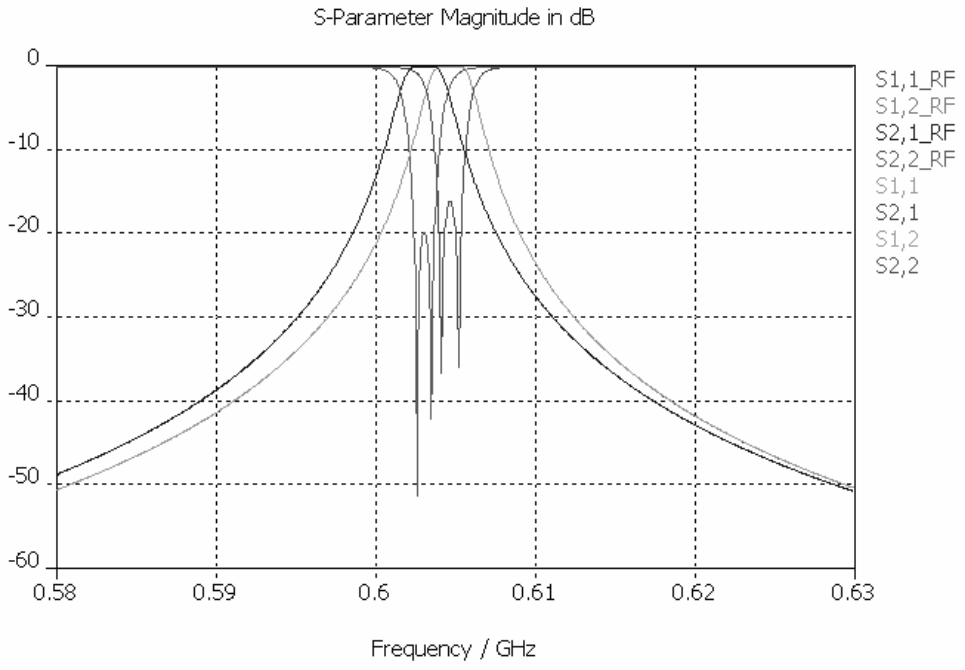
Note that the single frequency adaptive mesh refinement is already activated for the general purpose frequency domain solver with tetrahedral mesh. The broadband frequency sweep is enabled, and the default sampling strategy is given by the row after *Adapt. Freq*.: The *Auto* button is checked, and therefore an adaptive frequency sampling strategy is employed (rather than equidistant sampling). No number of samples is specified, meaning that the solver will calculate as many samples as necessary in order to satisfy the S-parameter sweep's convergence criterion. The *From* and *To* fields are left blank as well, indicating that the global frequency range will be used, as shown in the *Max. Range* line.



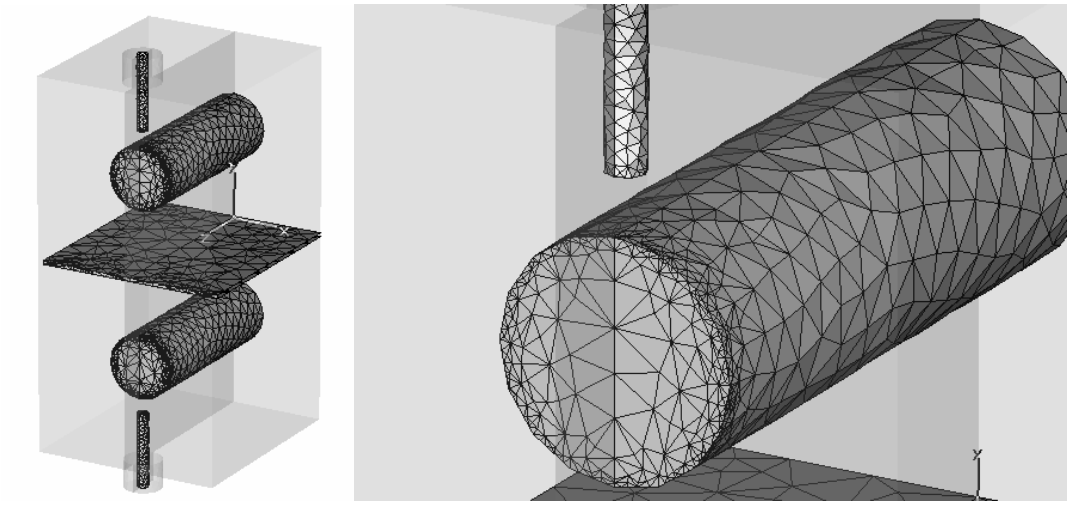
Please start the frequency domain solver by pressing the *Start* button.

The solver first performs the adaptive mesh refinement at 0.603 GHz. Afterwards, the broadband S-parameter sweep adaptively chooses additional frequency samples to obtain the full frequency-dependent S-parameter matrix. A few progress bars are running, keeping you informed about the current status of your calculation (e.g. tetrahedral mesh generation, port mode calculation …).

The solver will finish after a short time and deliver results that show a deviation of the pass band's center frequency compared to the "Resonant: Fast S-Parameter" solver, whose S-parameters are marked by "_RF":



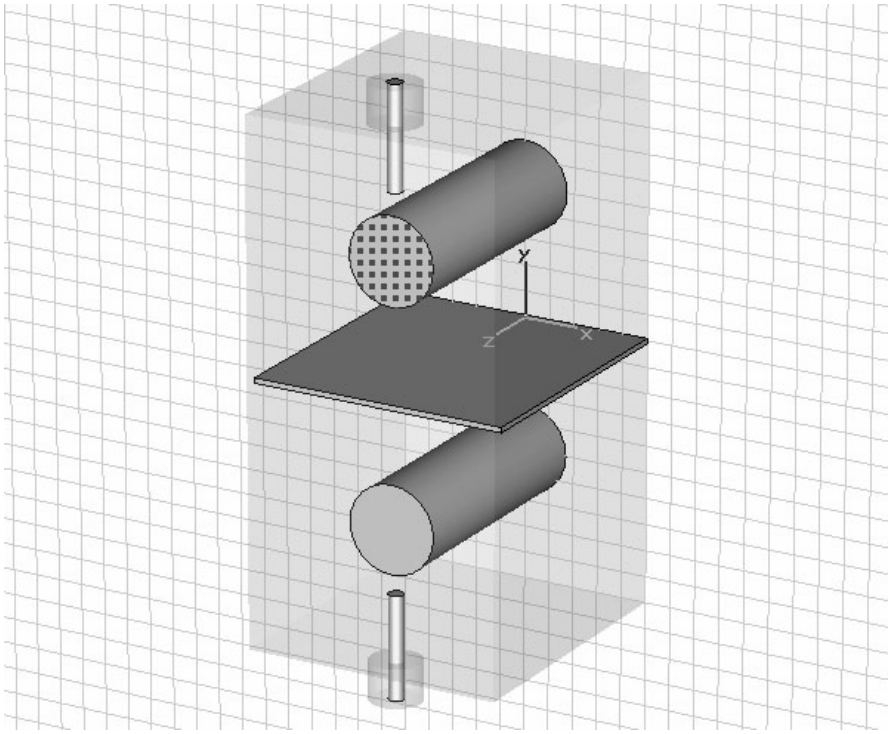
The output window, as well as the mesh adaptation log file (*Results ⇨ View Logfiles ⇨ Mesh Adaptation Logfile*), show a warning that indicates the adaptive mesh refinement has not yet reached the desired accuracy because the maximum number of adaptive mesh refinement passes has been reached. Therefore, examine the mesh by choosing *Mesh ⇨ Mesh View* from the menu or clicking on the icon. Select *Materials ⇨ PEC* from the navigation tree to transparently plot all solids other than PEC solids:



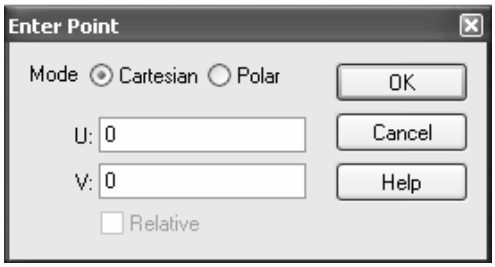
Clearly, the adaptive mesh refinement has strongly refined the mesh near the circular edges of both the cylinder and the coaxial feed's inner conductor, where a singular field can be expected. From this observation it can be concluded that a fine mesh and a good approximation of the circular edge's curvature are essential.

☐ Create Bodies for Mesh Refinement

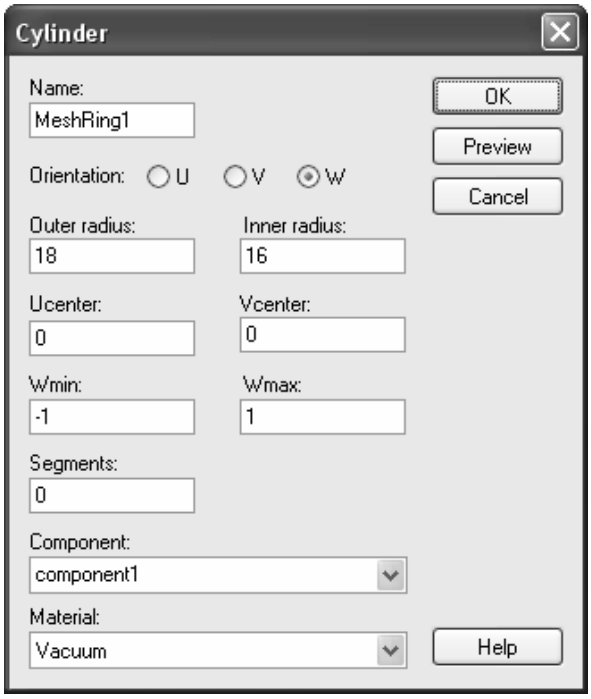
In order to initially get a fine mesh near the circular edges, you can define solids that do not change the physical properties of the model, but allow for a better control of the mesh. Turn off the mesh view by again selecting *Mesh ⇨ Mesh View* from the menu or by clicking on the icon. *Materials ⇨ PEC* should still be selected in the navigation tree so that all solids other than PEC solids are drawn transparently. Use the pick face tool (*Objects ⇨ Pick ⇨ Pick Face*,) to select the top face of the cylinder.



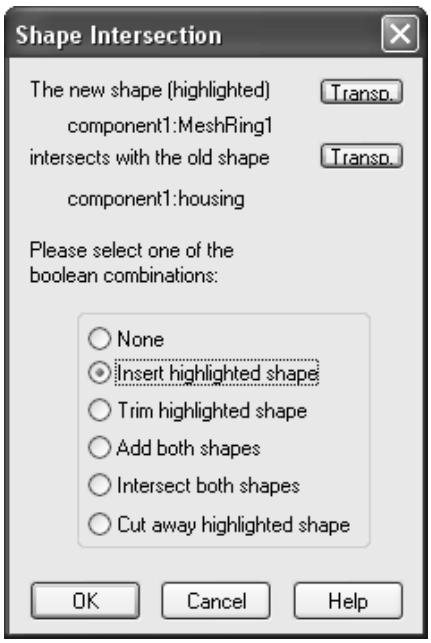
Now align the working coordinate system with the selected face (*WCS ⇨ Align WCS with selected face,*). Choose *Objects ⇨ Basic Shapes ⇨ Cylinder* from the main menu or the corresponding toolbar icon ( ). Its origin can be defined numerically by pressing *Shift+Tab*:



After confirming the settings with *OK*, press the *Esc* key to enter the remaining values as shown below. Note that this will create a vacuum cylinder with inner and outer radii one millimeter smaller or bigger than the radius of the resonator's PEC cylinder.

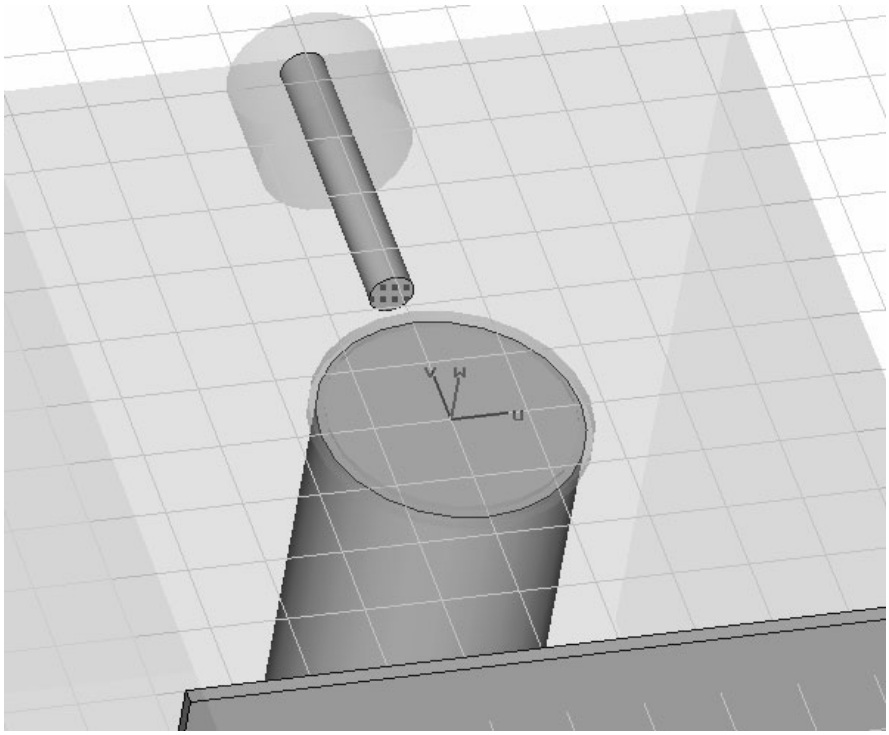


Because results from the previous simulation run are present, you will be prompted to delete those results after confirming the dialog with *OK*. Afterwards, insert the new solid into the housing:

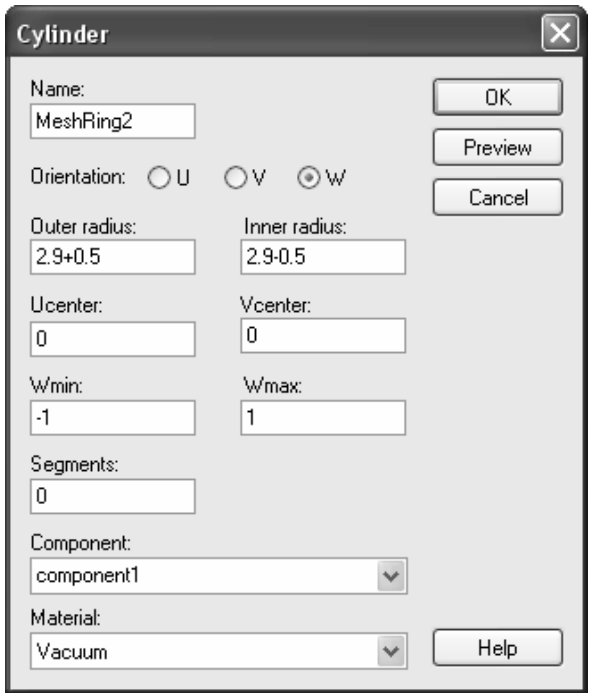


You may cancel the second shape intersection dialog as PEC will always overwrite non-PEC materials.

Using steps similar to those required to create this ring, another one will be defined for the coaxial cable's inner conductor. Select *Materials* ⇨ *PEC*, and then pick the inner conductor's face (*Objects* ⇨ *Pick* ⇨ *Pick Face*, ■) as shown below.

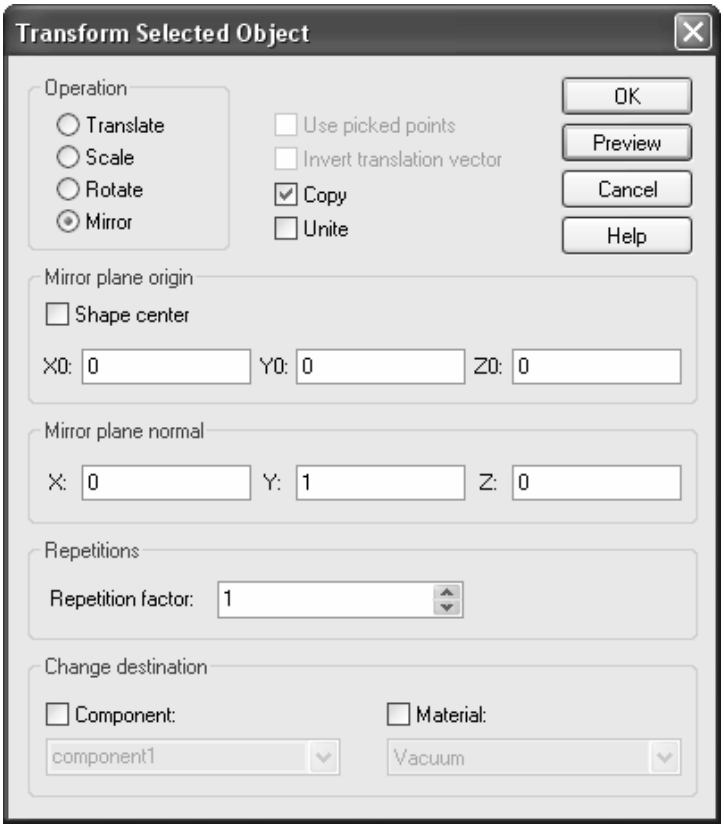


Align the working coordinate system with the selected face (*WCS ⇨ Align WCS with Selected Face,*). Again choose *Objects ⇨ Basic Shapes ⇨ Cylinder* from the main menu or the corresponding toolbar icon ( ), press *Esc* and enter the dialog values as follows:



After confirming these settings, choose “insert” in the first intersection dialog to insert MeshRing2 into the housing, and cancel the second intersection dialog.

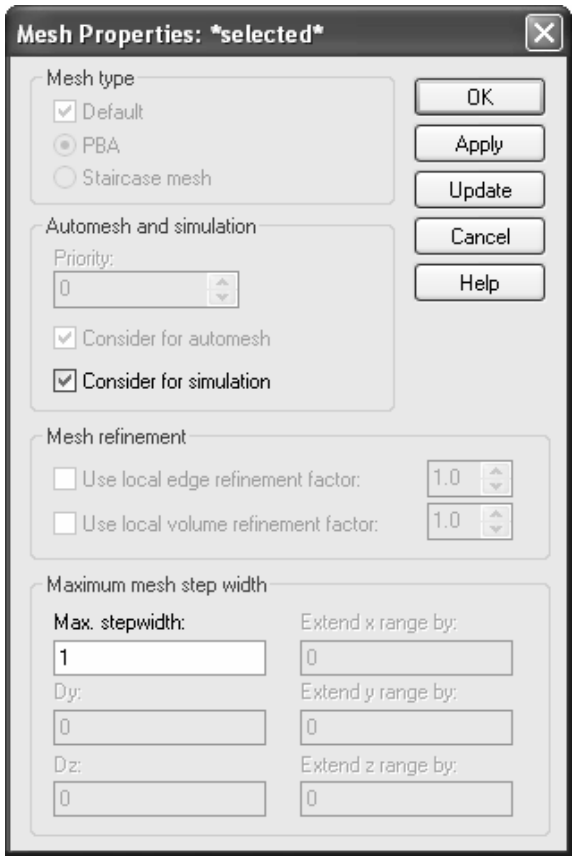
Then switch to the global coordinate system (*WCS ⇨ Local Coordinate System,*). Select both MeshRing1 and MeshRing2 in the navigation tree while pressing the *Ctrl* key, and choose *Objects ⇨ Transform* to mirror the two rings:



In the intersection dialogs that will appear after confirming the mirror operation, *Insert* the two copies of the rings into the housing, and choose *None* for the intersection with the PEC cylinders.

Now you are ready to define the mesh settings for all rings: Select them in the navigation tree while pressing the *Ctrl* key, choose *Edit ⇨ Mesh Properties* from the menu, and set the maximum mesh step width to one millimeter. This maximum mesh step will be applied only for the tetrahedral mesh inside the cylinder rings.

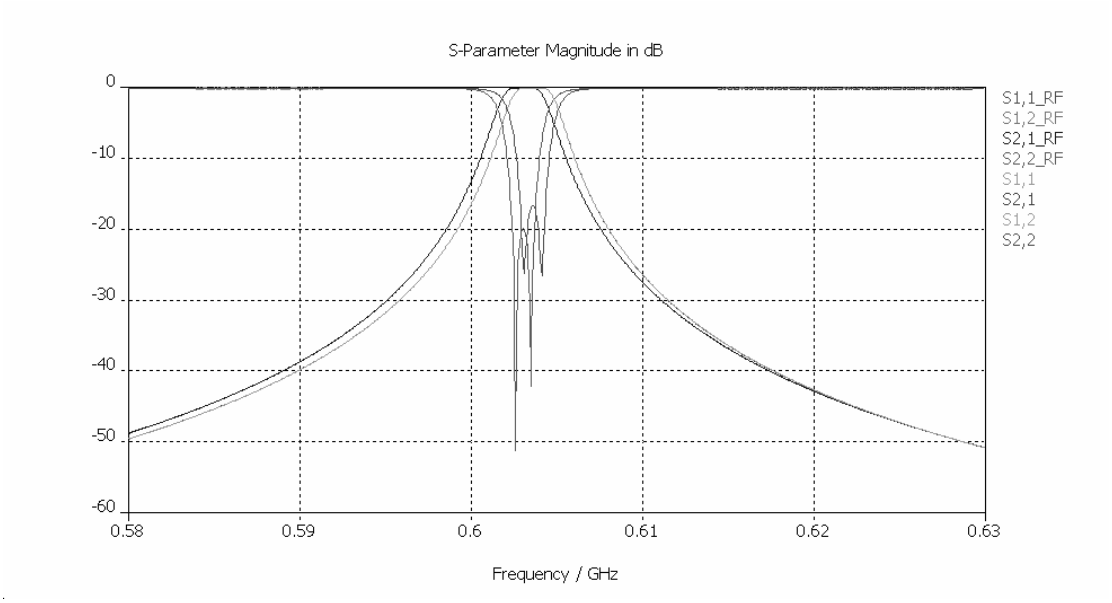
Now start the frequency domain solver again with the same settings as before. After the mesh generation, you can view the effect of the rings on the mesh near the PEC cylinders by selecting *Mesh ⇨ Mesh View* from the menu or by clicking on the icon, and choosing *Materials ⇨ PEC* from the navigation tree.



For a better comparison between the results from the “Resonant: Fast S-Parameter” and the “General Purpose” solvers, the S-parameters from both simulations are plotted again in one graph on the next page.

Note that the “Resonant: Fast S-Parameter” S-parameters seem to approach the final solution from lower frequencies, while those of the “General Purpose” solver with tetrahedral mesh converge from the opposite direction (down to lower frequencies) as the mesh is refined. The difference between the results obtained with the two methods can be used to estimate the accuracy of either result.

As evidenced by the below plot, the results agree well. Because they are not converged to the highest possible accuracy level, there is still a slight difference. After refining the accuracy limit in the adaptive mesh refinement the difference will become negligible.

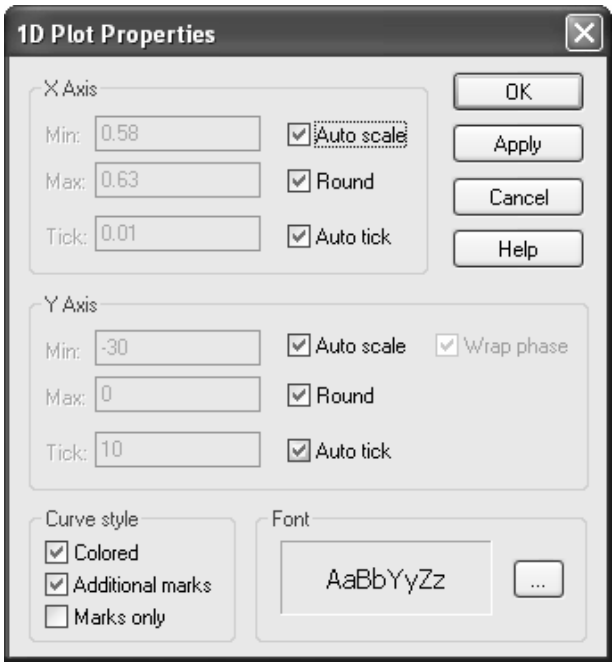


☐ Invoke a Field Calculation from the S-Parameter Curve

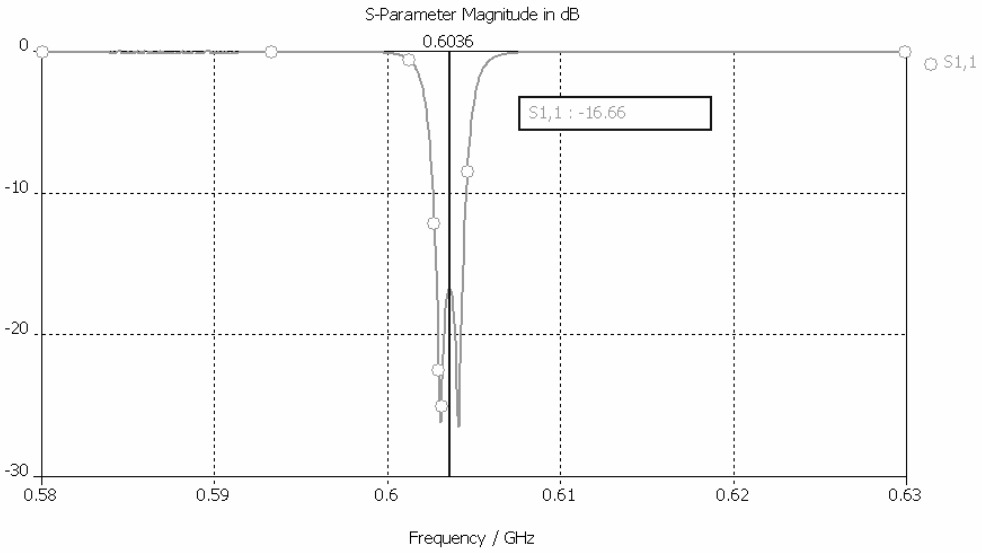
In contrast to the "Resonant: Fast S-Parameter" solver, the general purpose solver is able to calculate fields at certain frequencies. This can either be done via *Monitors* as a pre-processing step (for more information see the *Getting Started* manual), or by calculating the (E, H) fields at the frequency defined by the axis marker. To do so, click for instance on the *1D Results* ⇨ *|S|dB* ⇨ *S1,1* item in the navigation tree. Then select the axis marker by *Results* ⇨ *1D Plot Options* ⇨ *Axis Marker.* In the following dialog, please enter the frequency value where you would like to calculate the fields, e.g. 0.6036, which is approximately the center of the pass band.

Confirm the setting by pressing *OK*.

If you would like to observe where the frequency domain solver has actually placed the frequency samples, you can select *Results* ⇨ *1D Plot Options* ⇨ *Plot Properties* and check the *Additional marks* button. Again, confirm the setting by pressing *OK*.



The S-parameter plot of S1,1 should then look as follows:

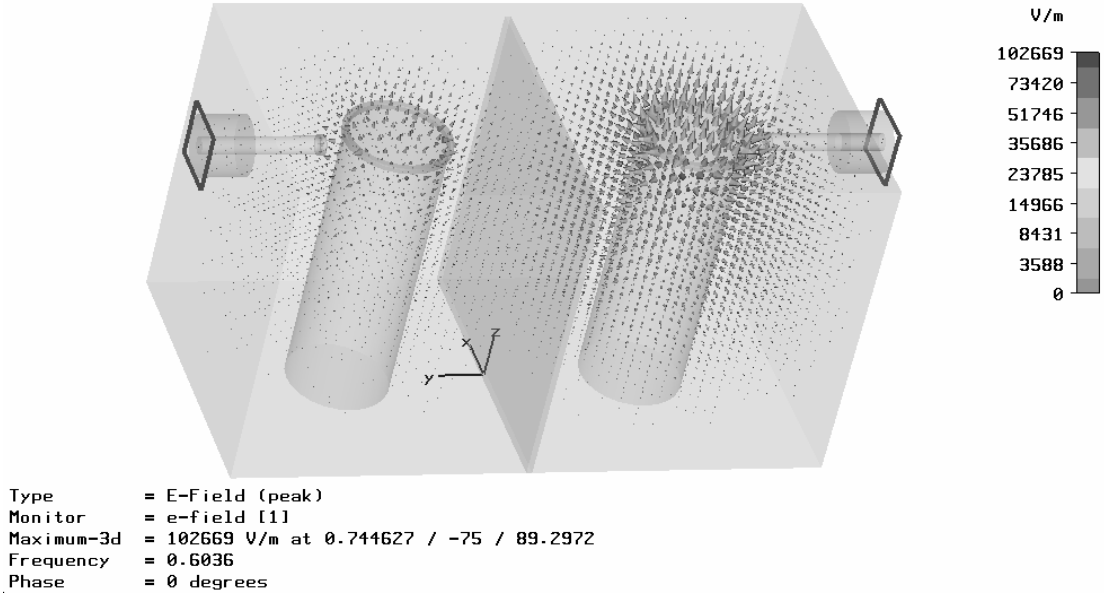


In the S-parameter plot, the axis marker at the desired frequency is plotted and the corresponding S-parameter values are given in a box. Next, select *Results ⇨ Calculate field at axis marker*, or simply click (in the main view) the right mouse button and select *Calculate field at axis marker*.

After the calculation an additional frequency sample mark will appear at 0.6036 GHz, with S1,1 = -16.66, as before.

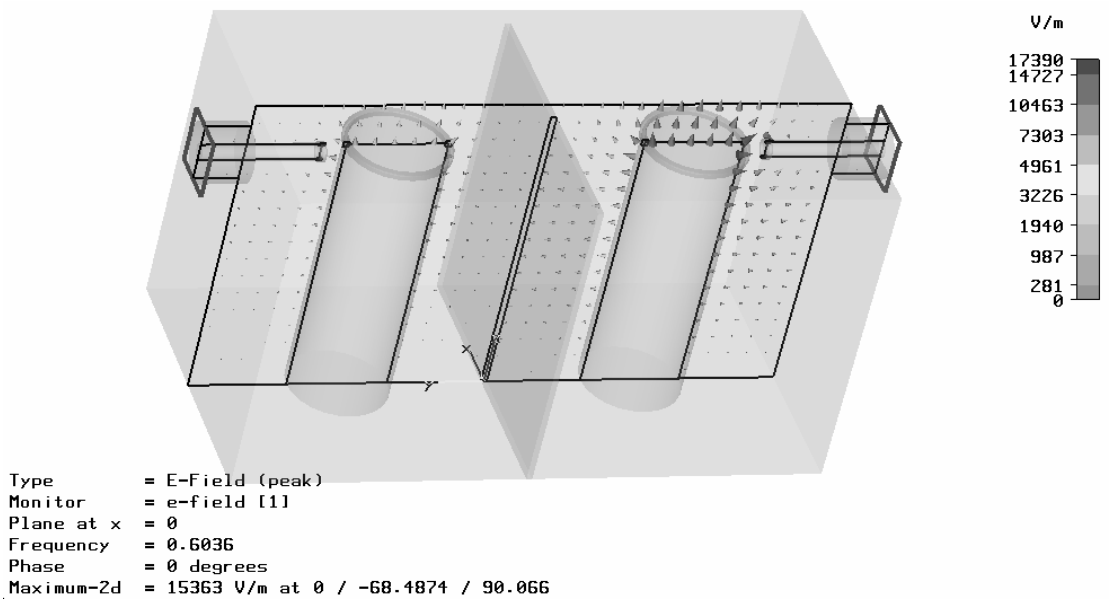
Again a few progress bars will appear, informing you about the current status. When the calculation is finished, a dialog appears that asks if you would like to see the calculated fields.

Please click *Yes*. The main view will automatically show the E-field.



The result data will then be visualized in a three dimensional vector plot as shown in the above picture. You may need to increase the number of arrows in the plot properties to obtain the image above. Please refer to the *Getting Started* manual for more information on how to adjust the plot options.

In many cases, it is more important to visualize the fields in a cross-section of the device. Therefore, please switch to the 2D field visualization mode by pressing the icon or by selecting *Results ⇨ 3D Fields on 2D Plane*. The field data should then be displayed as in the picture below. Again, you will need to increase the number of arrows to obtain this plot. Please refer to the *Getting Started* manual or press the *F1* key for online help to obtain more information on field visualization options.



In addition to the graphical field visualization, some information text containing maximum field strength values, etc. will be also shown in the main window.

Getting More Information

Congratulations! You have just completed the filter tutorial that should have provided you with a good working knowledge on how to use the Frequency Domain solvers to calculate S-parameters of filter structures. The following topics have been covered:

1. General modeling considerations, using templates, etc.
2. Define basic structure elements such as bricks and cylinders.
3. Use the working coordinate system to simplify the shape creation.
4. Create copies of existing shapes by using transformations.
5. Define the frequency range, boundary conditions and symmetries.
6. Run the Frequency Domain solvers and display S-parameters, and field patterns.
7. Obtain accurate and converged results using the automatic expert system based mesh adaptation.

You can obtain more information for each particular step from the online help system that can be activated by pressing either the *Help* button in each dialog box or the *F1* key at any time to obtain context sensitive information.

In some cases we have referred to the *Getting Started* manual, which is also a good source of information for general topics.

In addition to this tutorial, you can find additional Frequency Domain Solver examples in the “examples” folder in your installation directory. Each of these examples contains a *Readme* item in the navigation tree that will give you some more information about the particular device.

You should also consider using the transient solver to calculate the S-parameters for filter structures. Please check out the other tutorials and the transient solver examples in the “examples” folder for more information.

Finally, you should refer to the *Advanced Topics* manual for more in-depth information on issues such as the fundamental principles of the simulation method, mesh generation, usage of macros to automate common tasks, etc.

And last but not least: Please visit one of the training classes that are regularly held at a location near you. Thank you for using CST MICROWAVE STUDIO®!