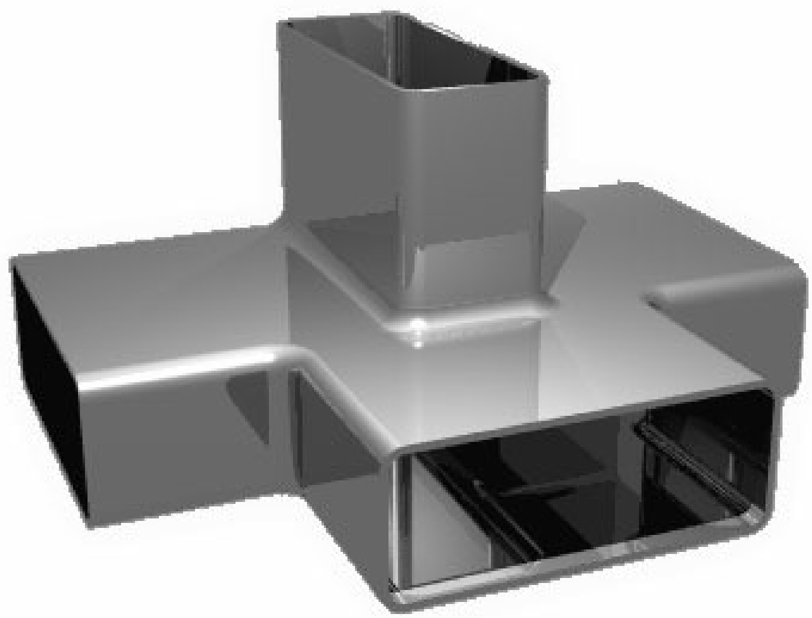
Rectangular Waveguide Tutorial



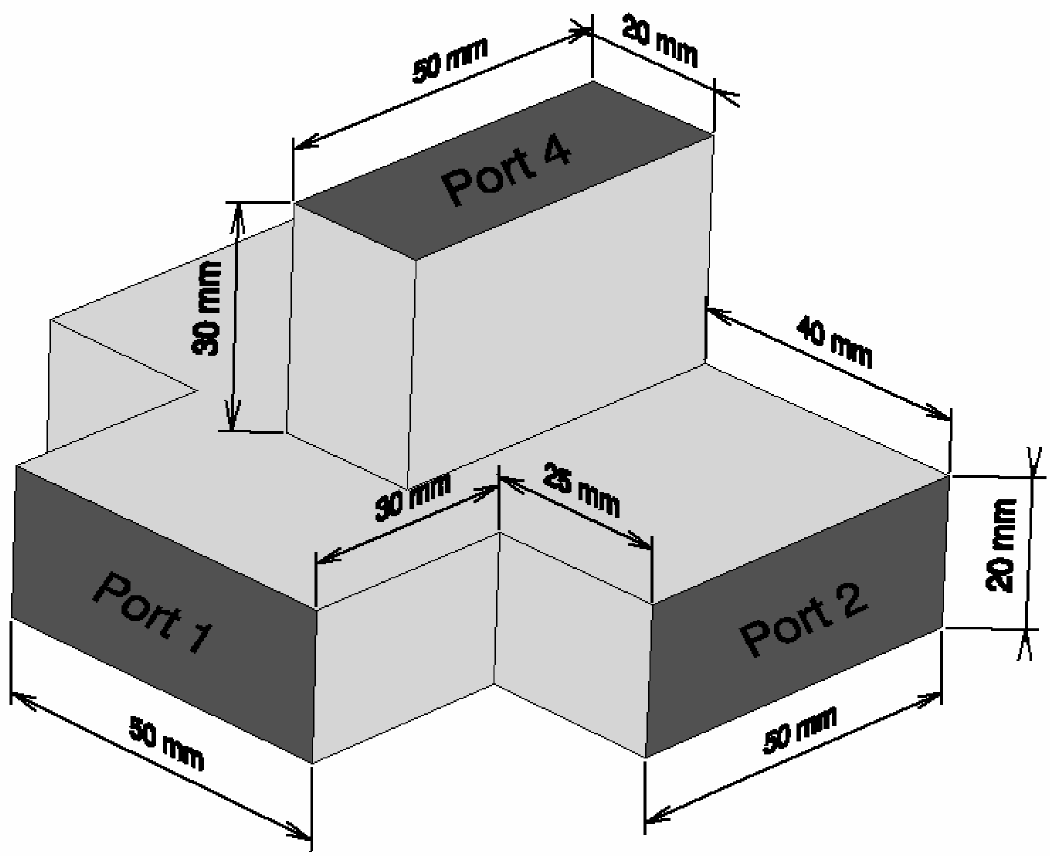
Geometric Construction and Solver Settings

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.

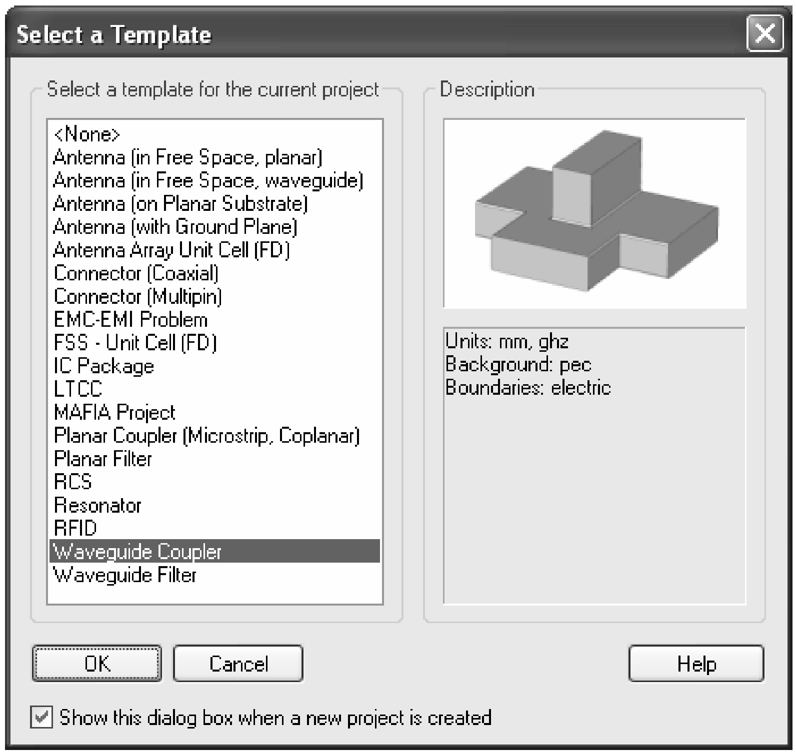
We strongly suggest that you carefully read through the CST MICROWAVE STUDIO® *Getting Started* manual before starting this tutorial.



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.

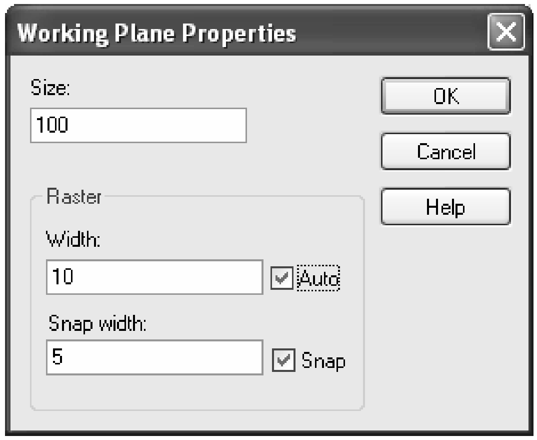


This template automatically sets the units to mm and GHz, the background material to PEC (which is the default) and all boundaries to be perfect electrical conductors.

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.

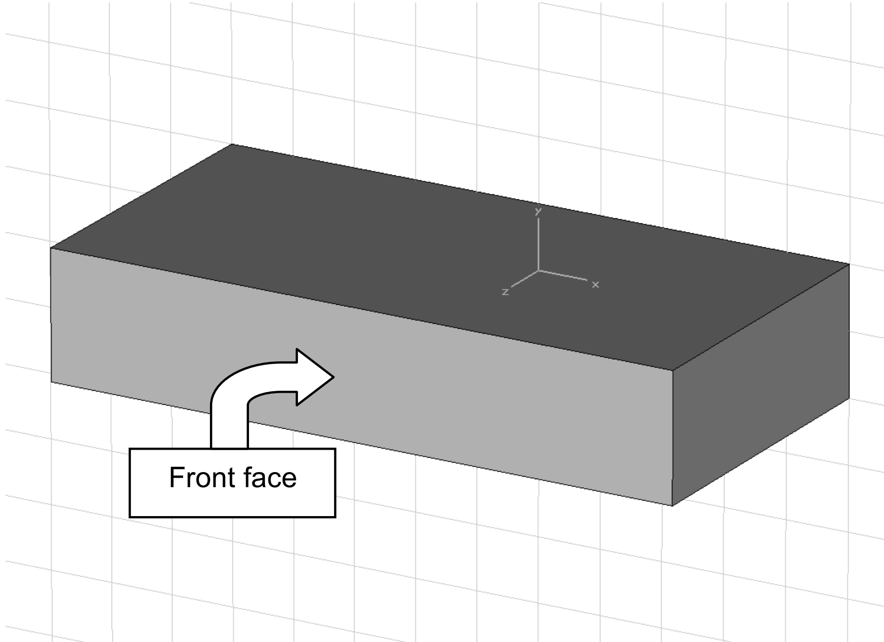
☐ Define the First Brick

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.

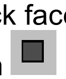
Xmin	-50
Xmax	50
Ymin	-10
Ymax	10
Zmin	0
Zmax	50



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 **<u>W</u>**orking **<u>C</u>**oordinate **<u>S</u>**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

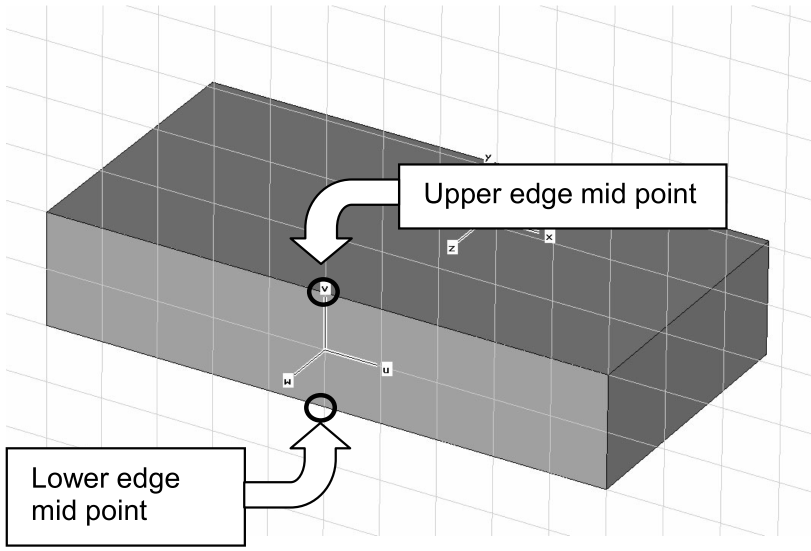
To add the waveguide belonging to port 1 to the front face, as shown in the above picture, activate the "Pick face" tool with one of the following options:

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.

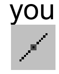
Now simply double-click on the front face of the brick to complete the pick operation.

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

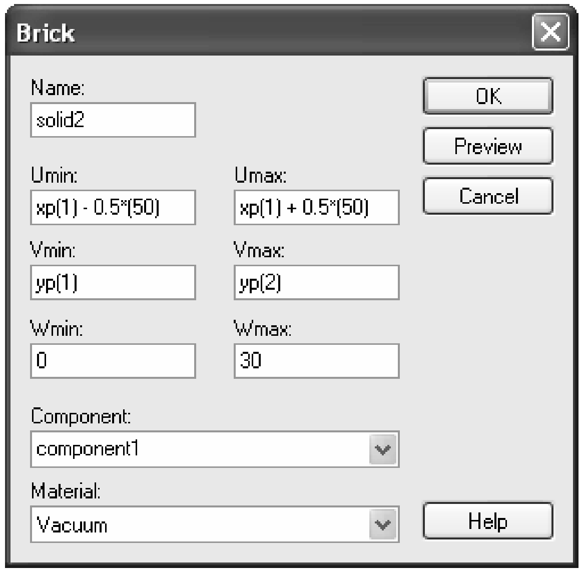
With the WCS in the right location, creating the second brick is quite simple. Start the brick creation mode with either the main menu's *Objects ⇨ Basic Shapes ⇨ Brick* or the corresponding icon . Please remember that all values used for shape construction are relative to the uvw coordinate system as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

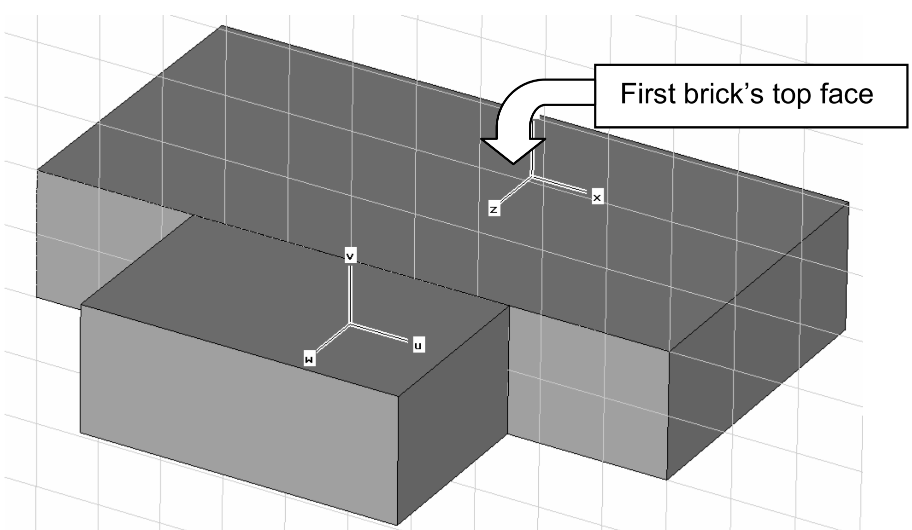


Some of the coordinate fields now contain mathematical expressions because some of the points were entered using the pick tools. Here, the functions xp(1), yp(1) represent the point coordinates of the first picked point (the midpoint of the first brick's lower edge). Analogously, the functions xp(2)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

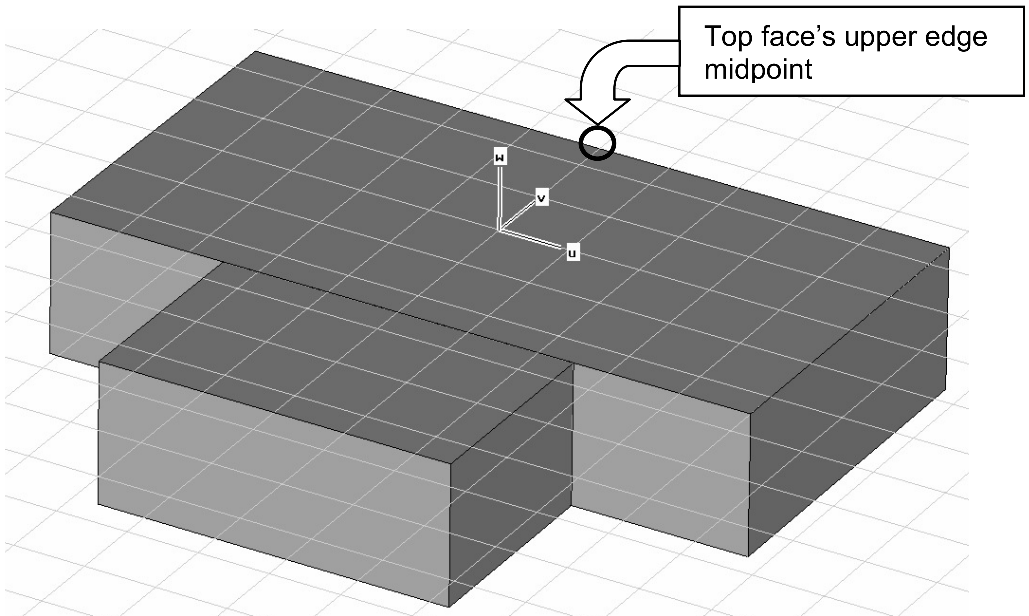
Finally, you should confirm the brick's creation again by pressing the *OK* button. Let's now construct the third brick.



☐ 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*.



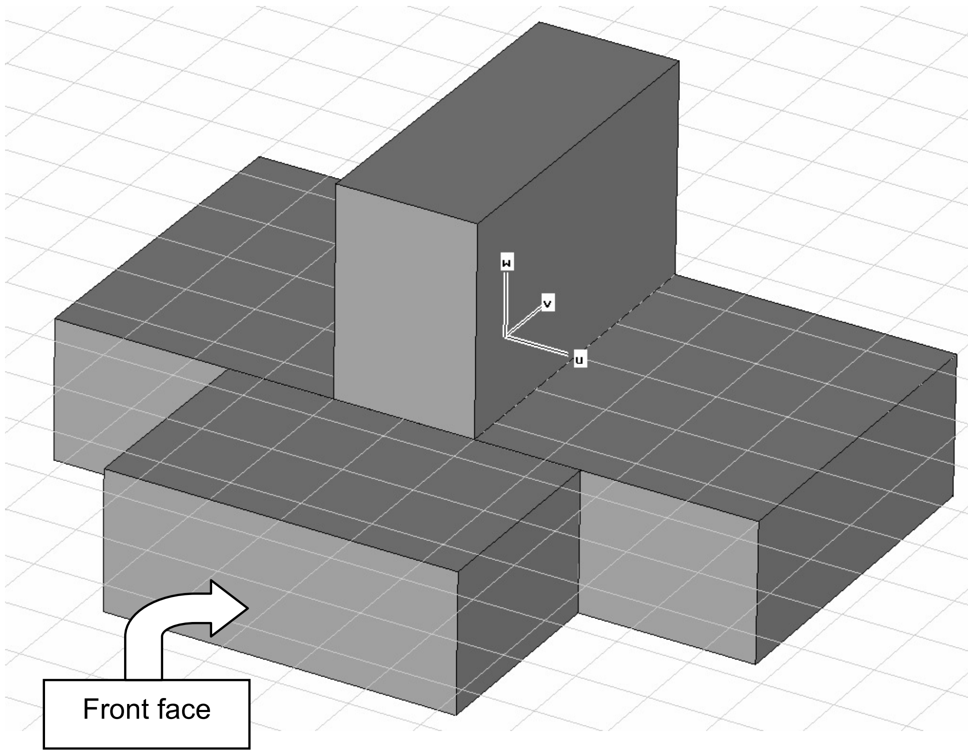
☐ Construct the Third Brick

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 .

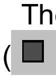
When you are requested to enter the first point, you should activate the midpoint edge pick tool (shortcut *m*), as you did for the previous brick, and double-click on the top face's upper edge midpoint (see picture above).

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.

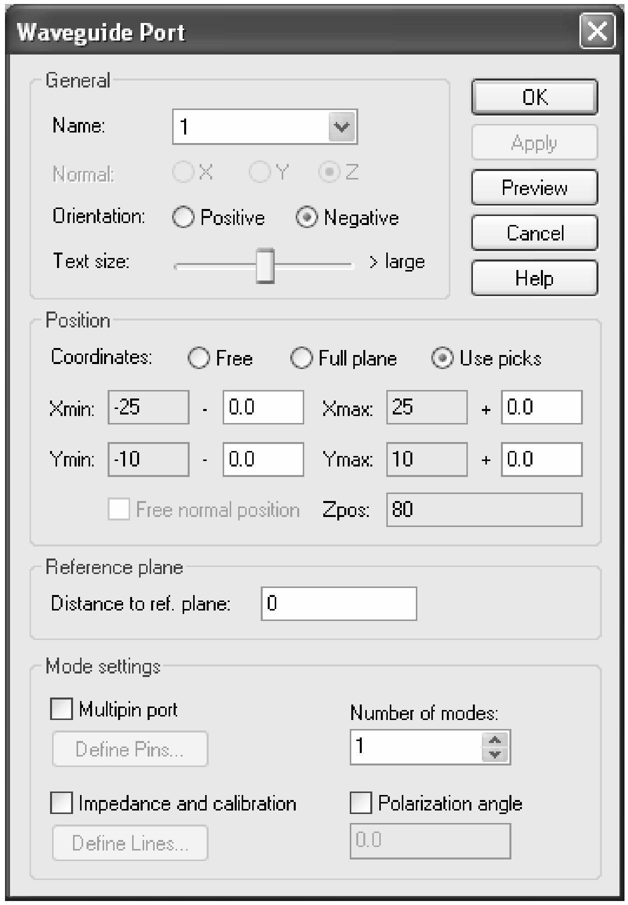
The last brick is also created as a vacuum material and belongs to the component “component1”. Finally, confirm these settings in the brick creation dialog box. Now the structure should look as follows:



☐ 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.

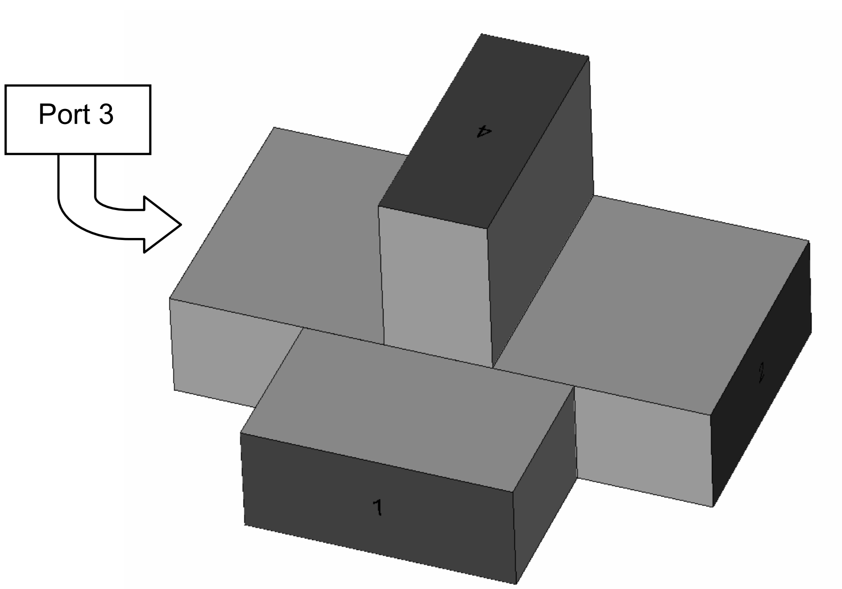
Once the port’s face is selected you can open the waveguide port dialog box either by selecting *Solve* ⇨ *Waveguide Ports* from the main menu or by pressing on the “Define waveguide port” icon . The settings in the waveguide port dialog box will automatically specify the extension and location of the 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.

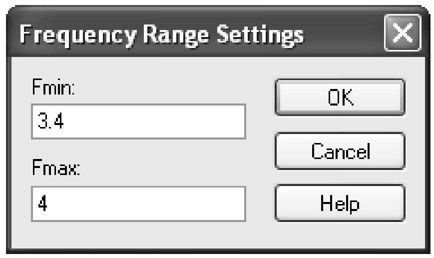
☐ Define Ports 2, 3, 4

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

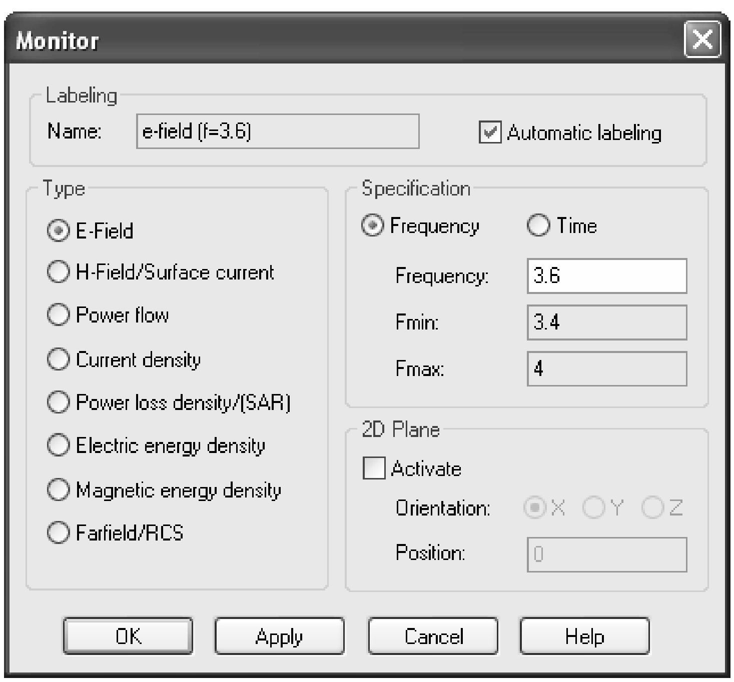
The frequency range for this example extends from 3.4 GHz to 4 GHz. Change *Fmin* and *Fmax* to the desired values in the frequency range settings dialog box (opened by pressing the “Frequency range” icon or choosing *Solve* ⇨ *Frequency*) and store these settings by pressing the *OK* button. Please 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.

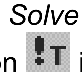
Calculation of Fields and S-Parameters

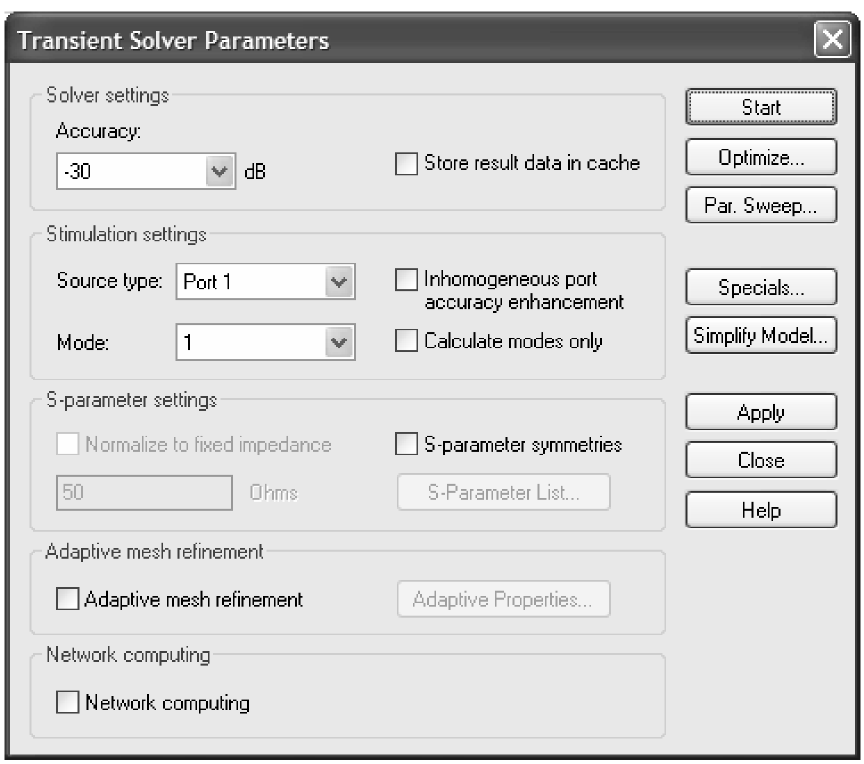
A key feature of CST MICROWAVE STUDIO® is the *Method on Demand* approach that allows a simulator or mesh type that is best suited for a particular problem. Another benefit is the ability to compare the results obtained by completely independent approaches. We demonstrate this strength in the following 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.

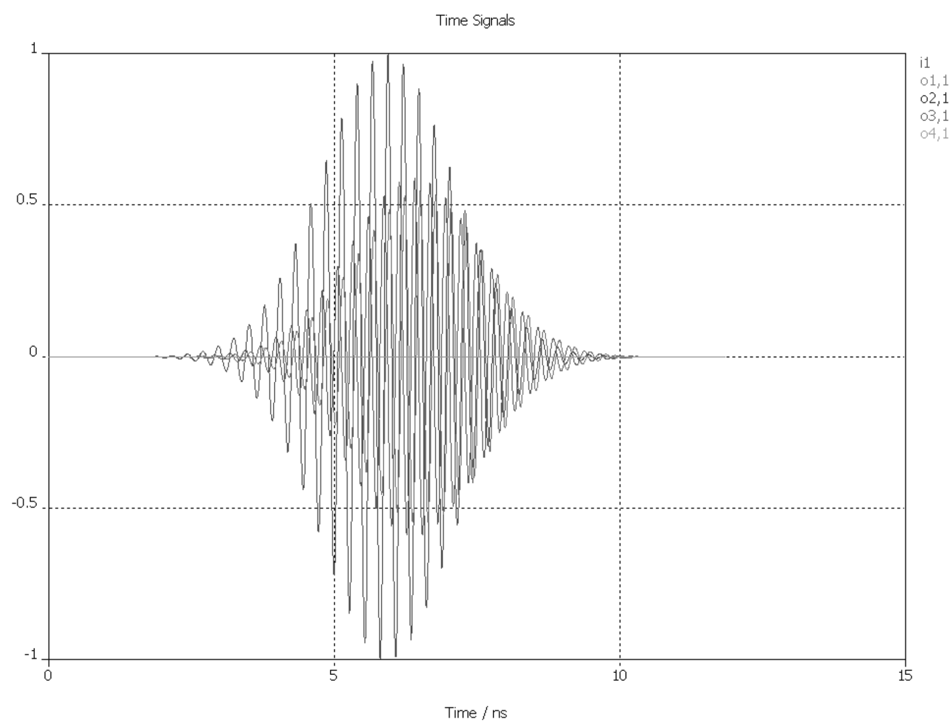
Finally, press the *Start* button to begin the calculation. A progress indicator appears in the status bar displaying some information about the calculation. If any error or warning messages are produced by the solver, they will be displayed in the message window that will be activated automatically,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

Transient Solver Results

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

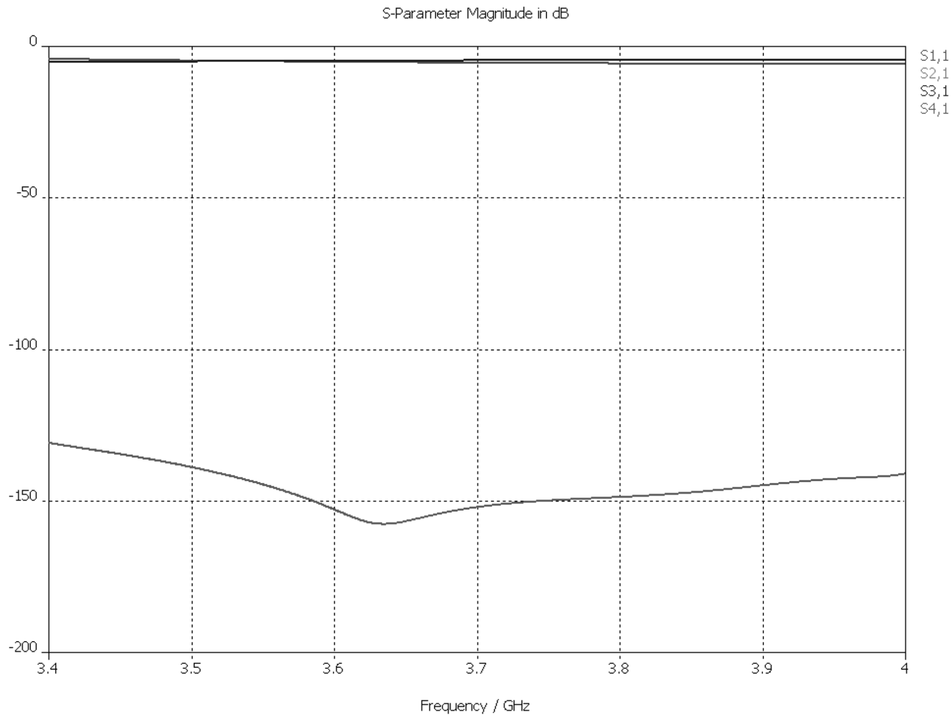
☐ 1D Results (Port Signals, S-Parameters)

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).

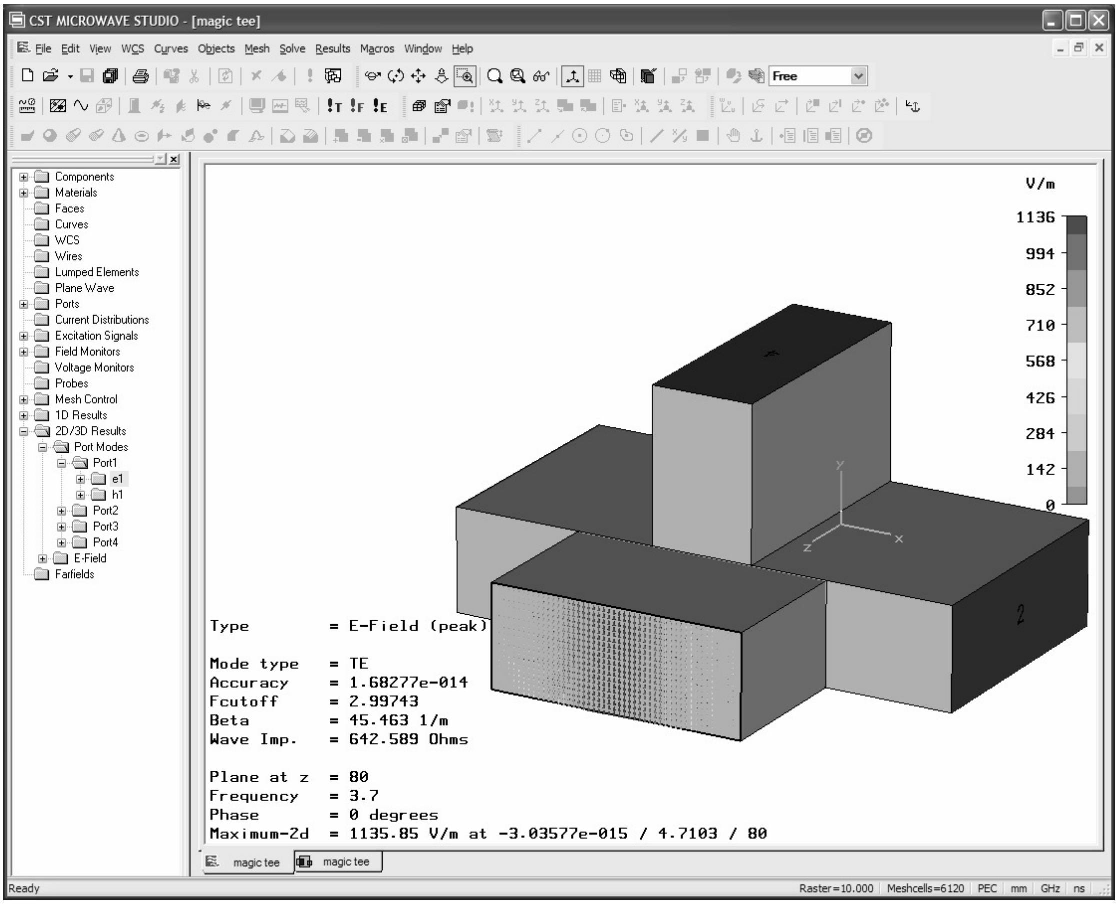
The S-parameters can be plotted in dB by clicking on the *1D Results* ⇨ /*S* /*dB* folder.



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.

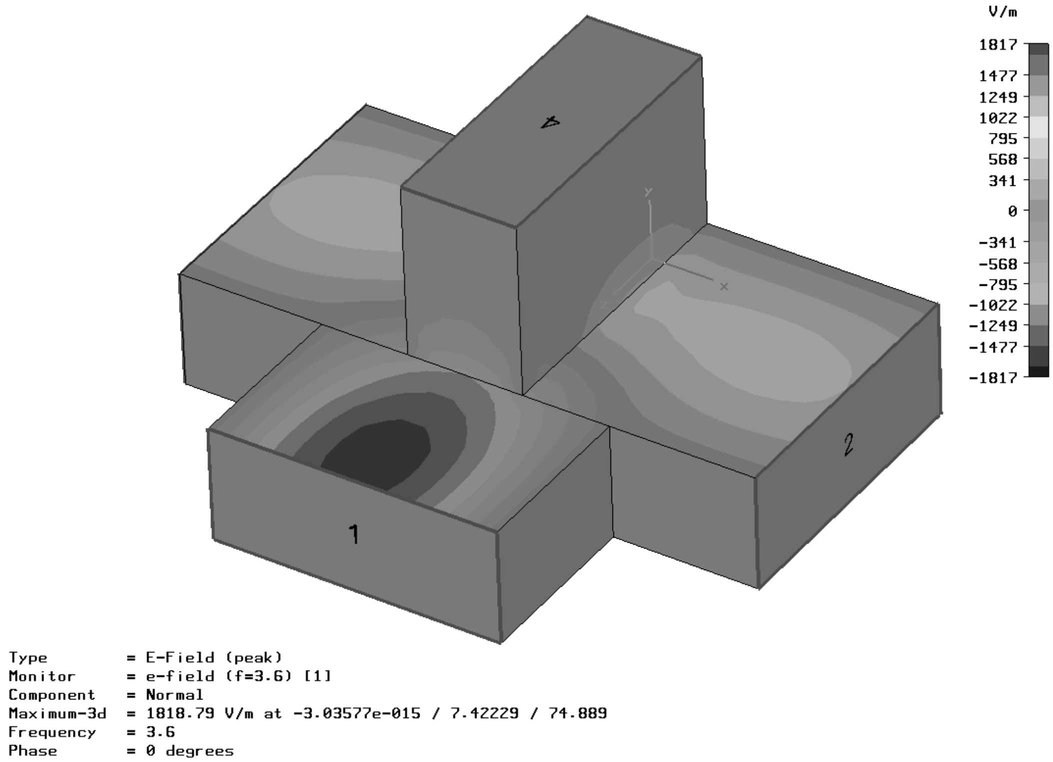
☐ 2D and 3D Results (Port Modes and Field Monitors)

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.



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:

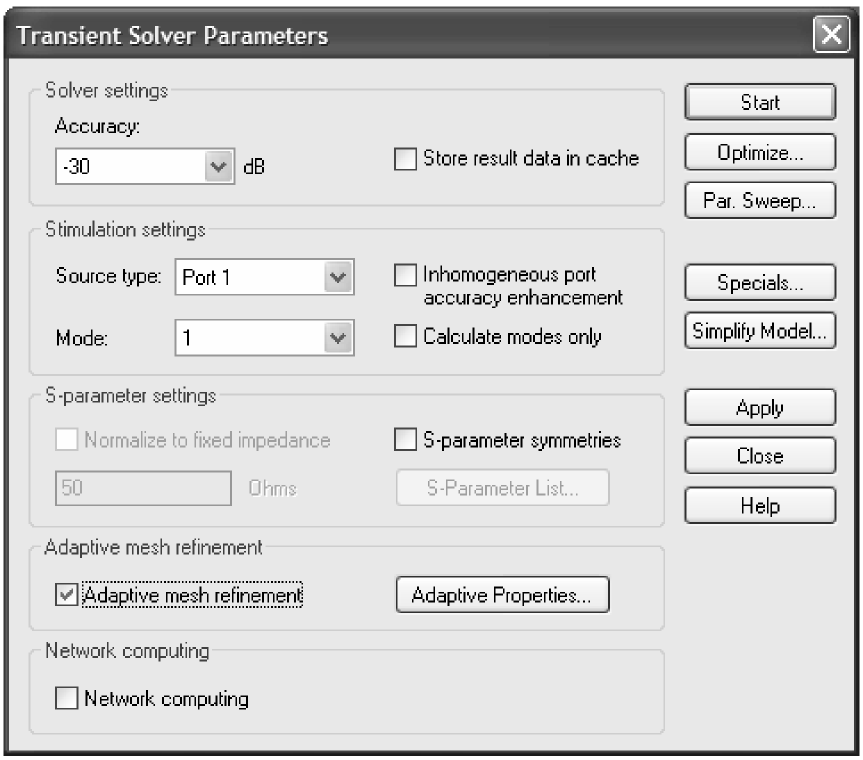
Desired Accuracy Level	Accuracy Setting (Solver control dialog box)
Moderate	-30dB
High	-40dB
Very high	-50dB

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.

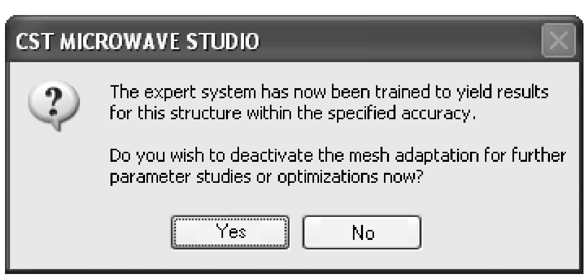
☐ Effect of the Mesh Resolution on the S-parameter's Accuracy

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*):



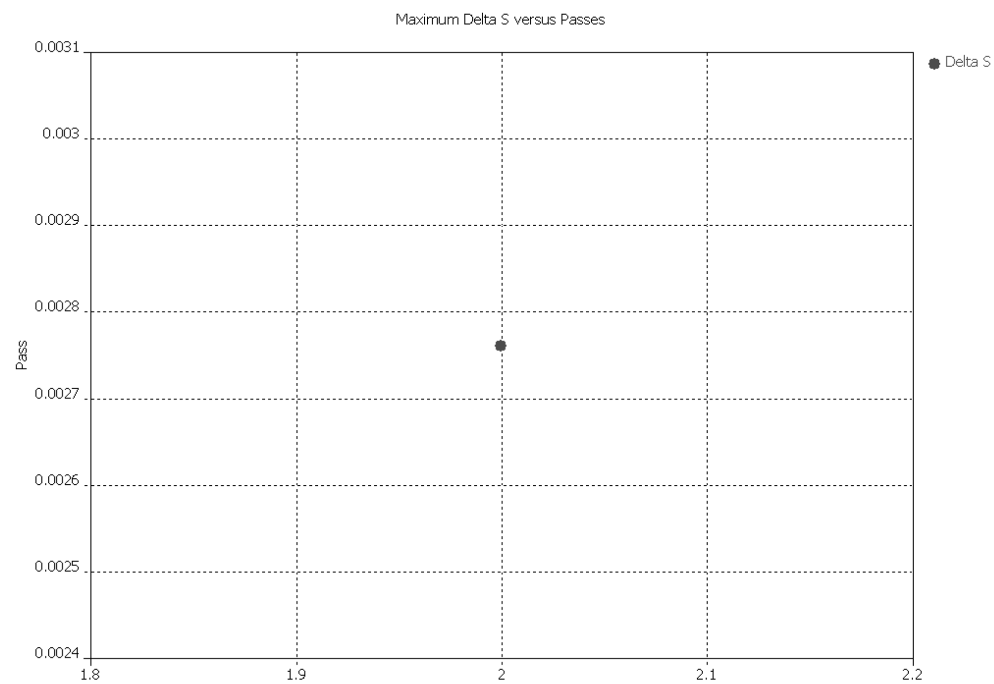
After activating the adaptive mesh refinement tool, you should now start the solver again by pressing the *Start* button. After a couple of minutes (during which the solver is running through mesh adaptation passes), the following dialog box will appear:



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).

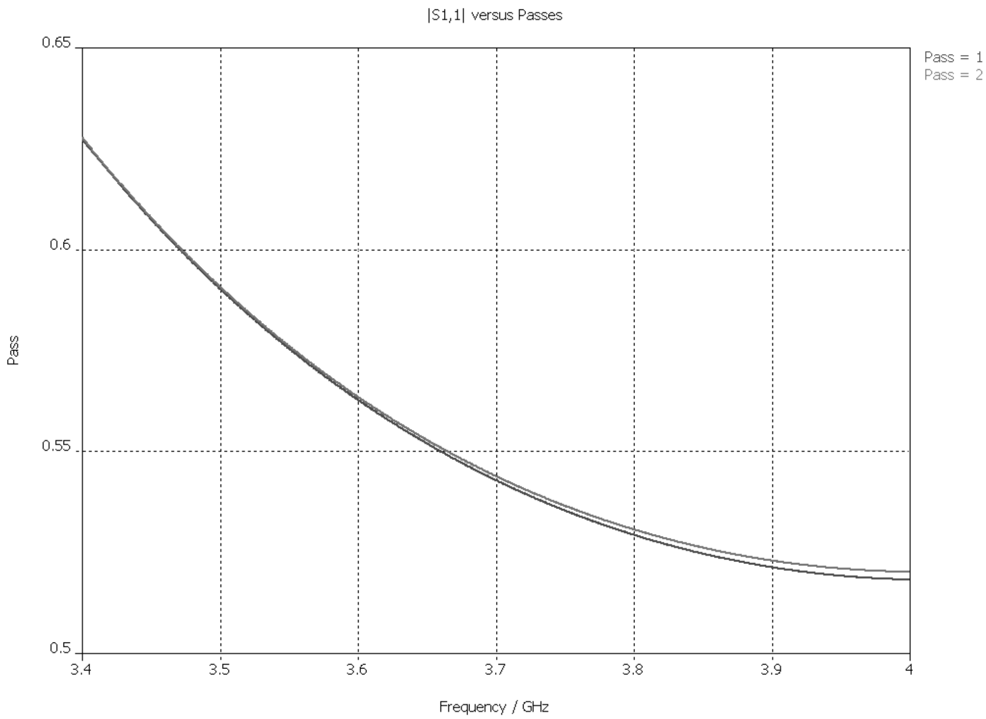
You should now confirm the deactivation of the mesh adaptation by pressing the *Yes* button.

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

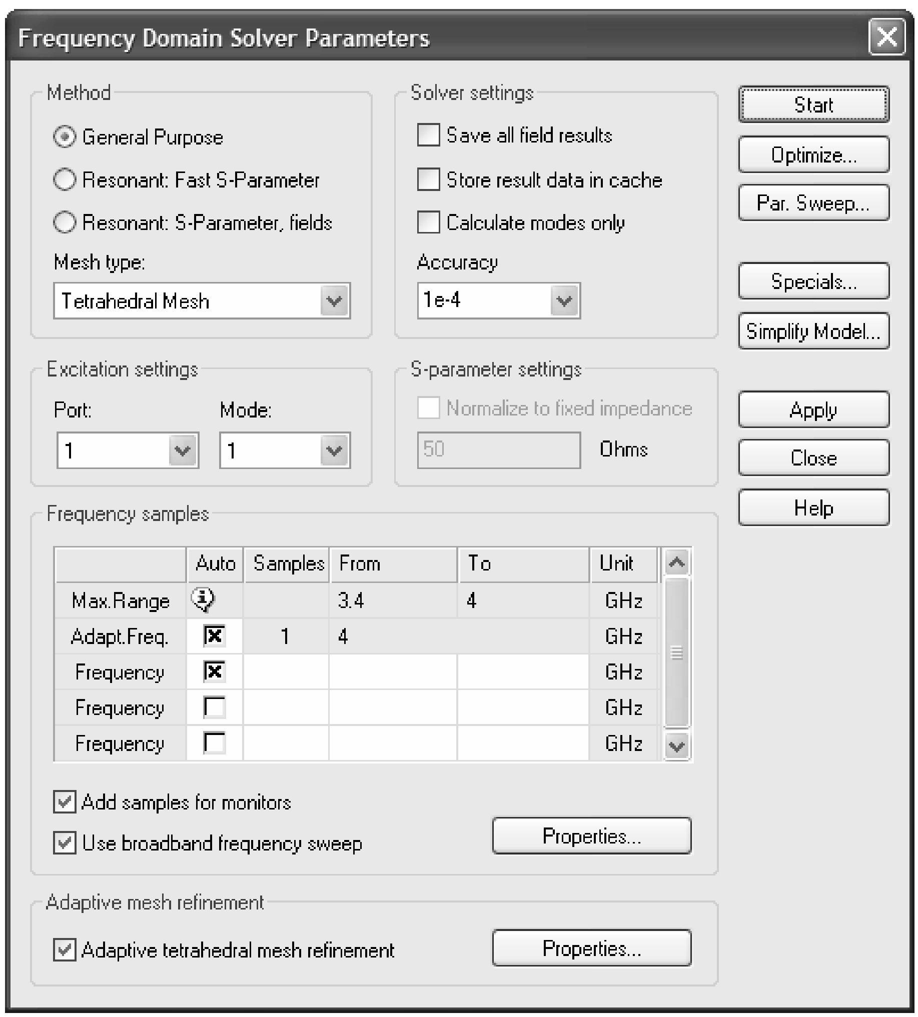
CST MICROWAVE STUDIO® offers a variety of frequency domain solvers specialized for different types of problems. They differ not only by their algorithms but also by the grid type they are based on. The general purpose frequency domain solver is available for hexahedral grids, as well as for tetrahedral 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).

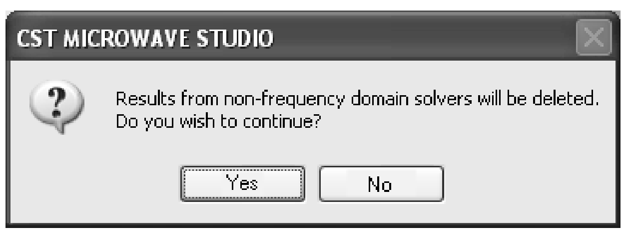
Consequently, we only need to calculate the S-parameters S1,1, S2,1, S3,1 and S4,1. All of the S-parameters can be derived by an excitation at port 1. Therefore, you should change the Source type field in the *Excitation settings* frame to *Port 1* unless already done. If this is set to *All Ports*,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



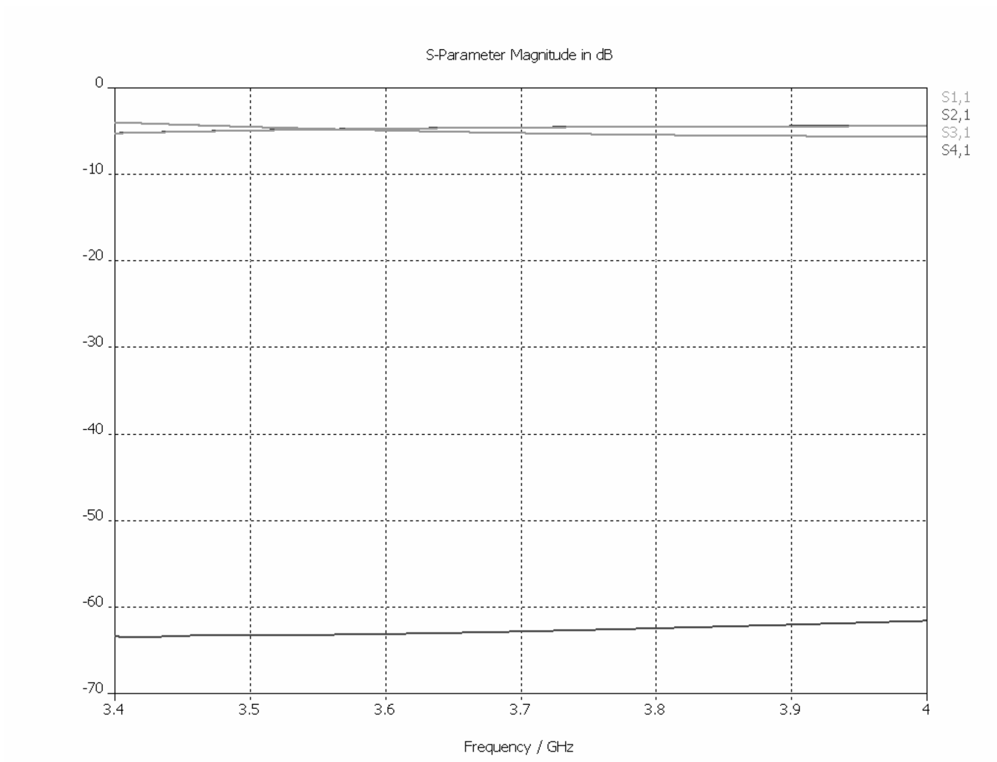
Press *Yes* to acknowledge the deletion. A progress bar will appear at the bottom of the main frame as soon as the solver starts. Additional information about the simulation progress will be shown in the message window that will be activated automatically, if necessary.

Frequency Domain Solver Results

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

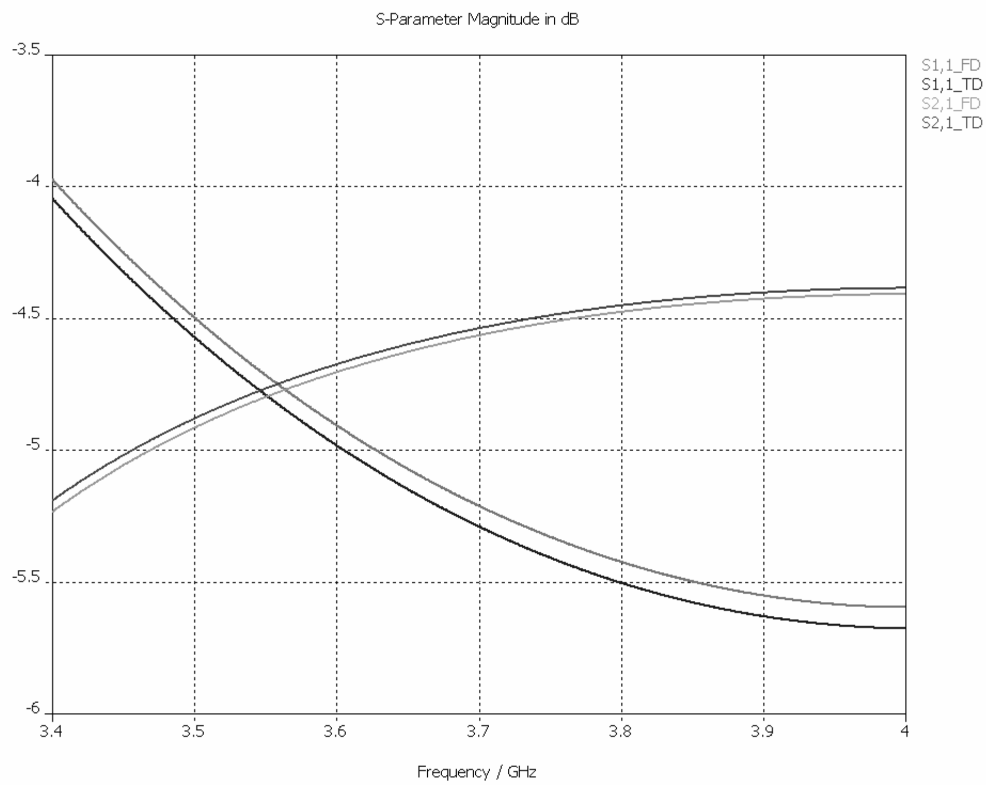
☐ 1D Results (S-Parameters)

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.

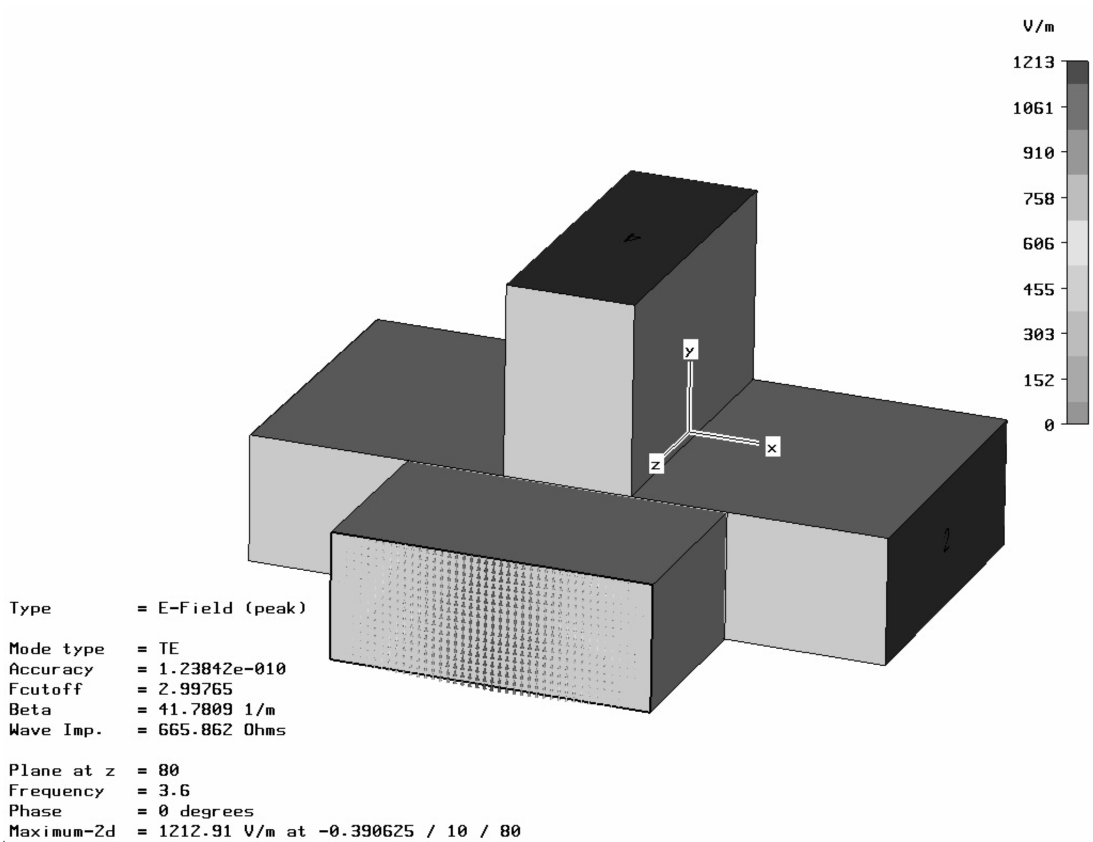
The next figure shows the S-parameters S1,1 and S1,2 for both transient and frequency domain solvers plotted in the same graph. This can be done by copying all these results to an extra folder.



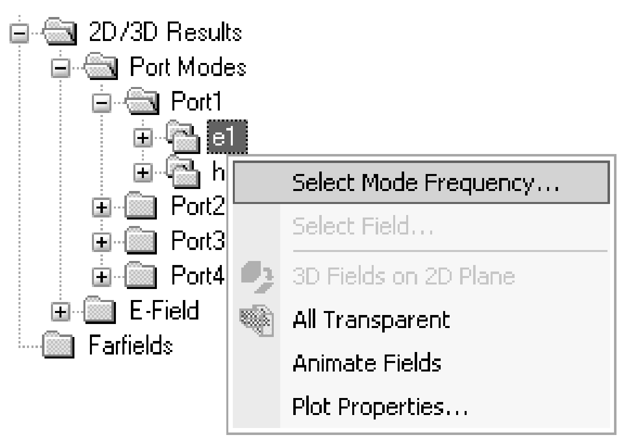
As you can see, the results agree very well. For this specific structure, the transient solver does provide more accurate results by default. The accuracy of the frequency domain simulation can be increased by lowering the accuracy limit for the adaptive mesh refinement.

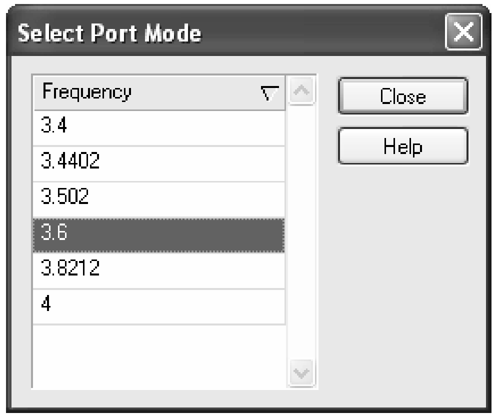
☐ 2D and 3D Results (Port Modes and Field Monitors)

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.

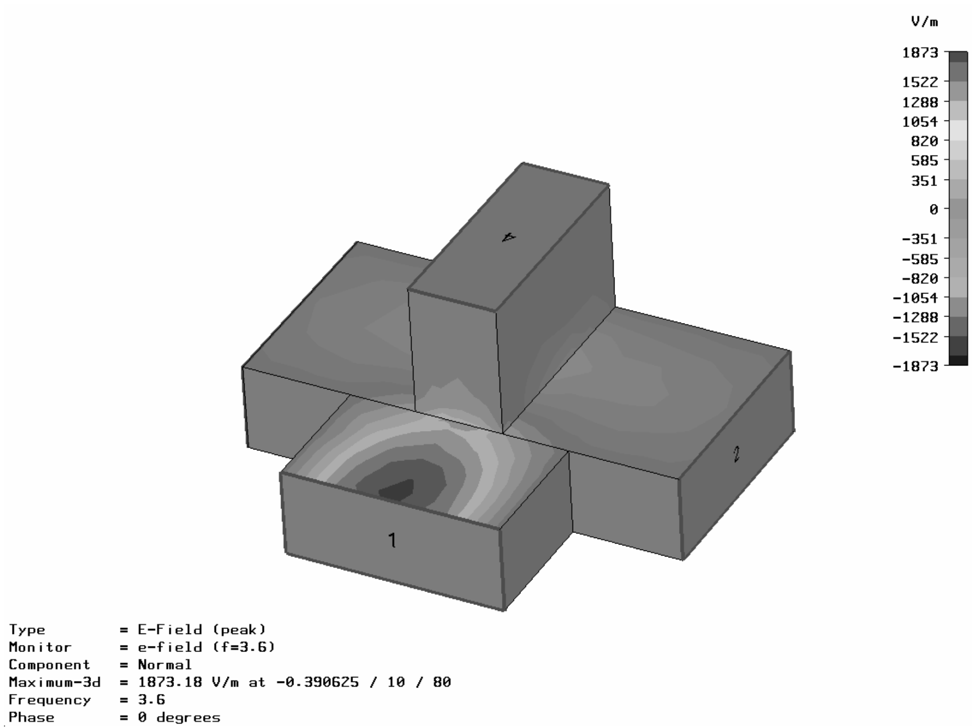


The mode properties shown in the lower left corner of the field plot are close to those computed with the transient solver. Note that the frequency of the mode is not exactly equal to the frequency used by the transient solver. The frequency domain solver calculates the modes for every frequency sample. 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.

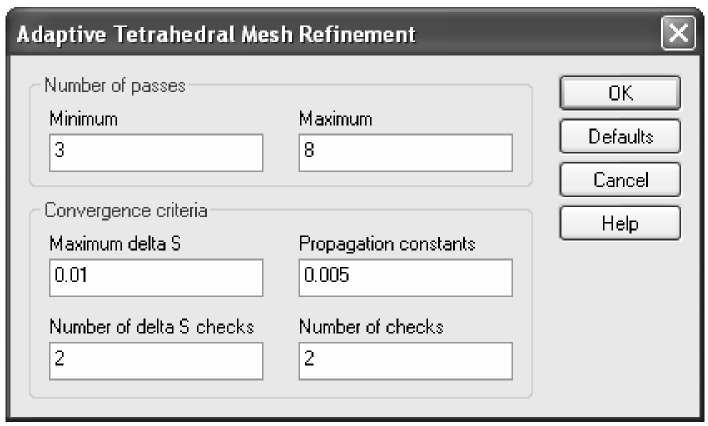


Here, you can observe the effect of the coarse tetrahedral mesh on the computed field: the contours of the computed field distribution are slightly irregular. The next section describes how to influence the mesh refinement and improve the quality of the computed field distribution.

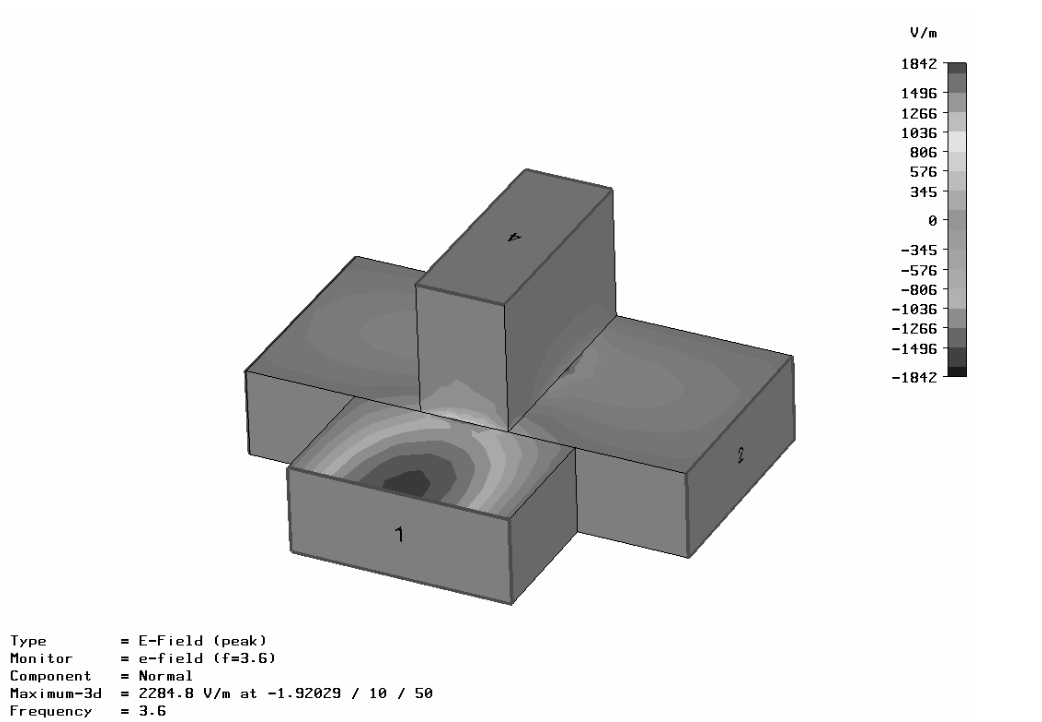
You can display an animation of the fields by checking the *Animate Fields* option in the context menu (right mouse click in the plot window). The appearance of the plot can be changed in the plot properties dialog box that can be opened by selecting *Results ⇨ Plot Properties* from the main menu or *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®!