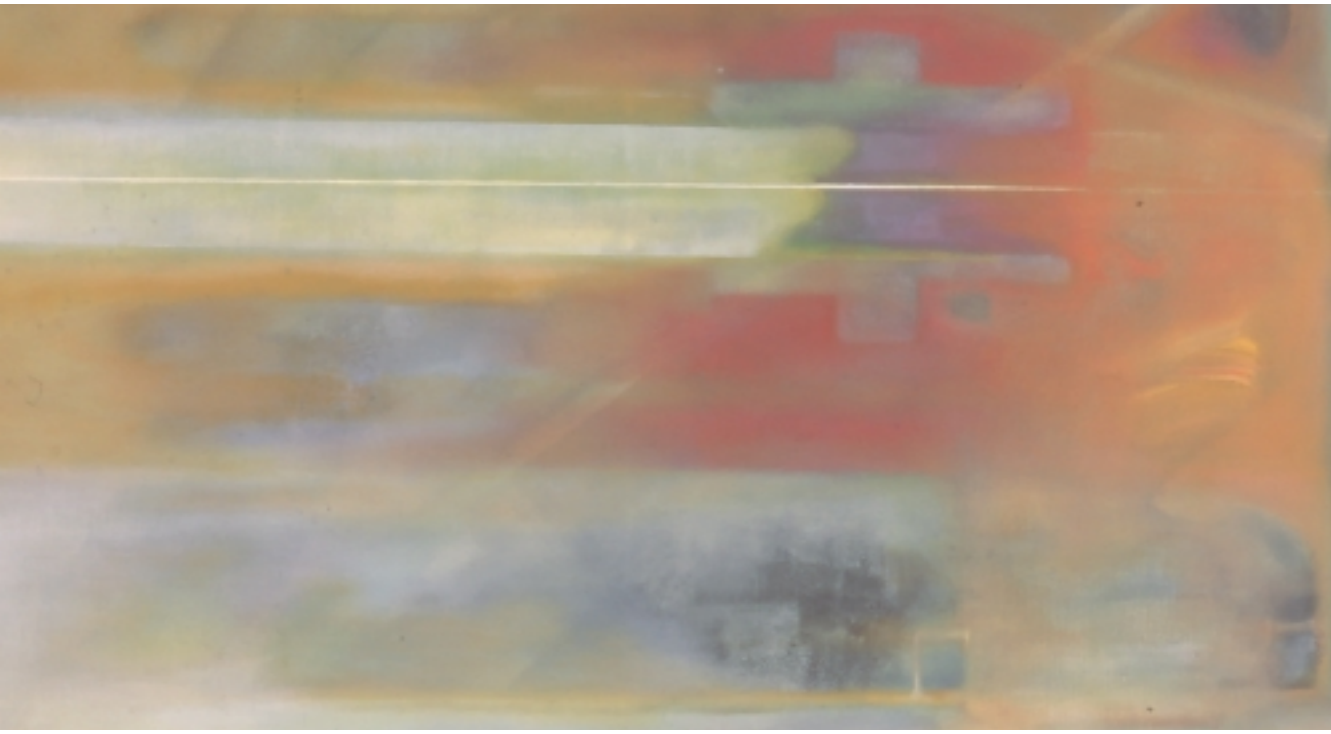
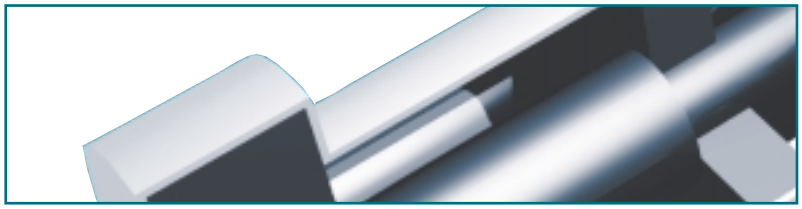
HF DESIGN AND ANALYSIS

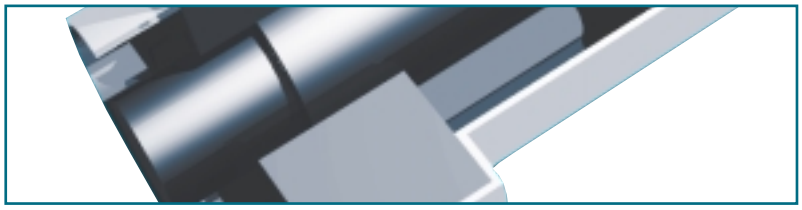
CST MICROWAVE STUDIO®



GETTING STARTED · VERSION 4



CST – Computer Simulation Technology
www.cst.de



July 15, 2002

Contents

Chapter 1 — Introduction

Welcome

Welcome to CST MICROWAVE STUDIO®, the powerful and easy to use electromagnetic field simulation software. This program combines both a user friendly interface and simulation performance in an unsurpassed manner.

Because of the native Windows based user interface, you will feel familiar with the simulation environment straight away. This means that you can immediately start caring about your actual electromagnetic problem rather than dealing with a cryptic proprietary user interface. An excellent visual feedback at all stages of the simulation process allows you to obtain a very steep learning curve.

How to Get Started Quickly?

We recommend that you proceed as follows:

1. Work through this document carefully. It should provide you with all the basic information necessary to understand the advanced documentation.
2. Work through the tutorials by picking the example which best fits your needs.
3. Have a look at the examples folder in the installation directory. The different application types will give you a good impression of what has already been done with the software. Please note that these examples are designed to give you a basic insight into a particular application domain. Real world applications are typically much more complex and harder to understand if you are not familiar with the device.
4. Start with your own first example. Please choose a reasonably small and simple example, which will allow you to quickly become familiar with the software.
5. After you have worked through your first example, contact technical support in order to get some hints for possible improvements to achieve an even more efficient usage of CST MICROWAVE STUDIO®.

What is CST MICROWAVE STUDIO®?

CST MICROWAVE STUDIO® is a fully featured software package for electromagnetic analysis and design in the high frequency range. It simplifies the process of inputting the structure by providing a powerful solid modeling front-end which is based on the famous ACIS modeling kernel. Strong graphic feedback simplifies the definition of your device even further. After the component has been input, a fully automatic meshing procedure (based on an expert system) is applied before the simulation engine is started.

The simulator itself features the new Perfect Boundary Approximation (PBA method) and its Thin Sheet Technique (TST) extension, which increases the accuracy of the simulation by an order of magnitude in comparison to conventional simulators. Since no method works equally well in all application domains, the software contains four different simulation techniques (transient solver, frequency domain solver, eigenmode solver, modal analysis solver) which best fit their particular applications.

The most flexible tool is the **transient solver**, which can obtain the entire broadband frequency behavior of the simulated device from only one calculation run (in contrast to the frequency stepping approach of many other simulators). This solver is very efficient for most kinds of high frequency applications such as connectors, transmission lines, filters, antennas and many more.

However, efficient filter design often requires the direct calculation of the operating modes in the filter rather than an S-parameter simulation. For these cases, CST MICROWAVE STUDIO® also features an **eigenmode solver** which efficiently calculates a finite number of modes in any loss-free electromagnetic device.

When investigating highly resonant structures such as narrow bandwidth filters, a time domain approach may become inefficient, because of the slowly decaying time signals. The usage of advanced signal processing techniques (AR-filters) provided by CST MICROWAVE STUDIO® allows speeding up these simulations by orders of magnitude compared to standard time domain methods. Furthermore, CST MICROWAVE STUDIO® also contains a so called **modal analysis solver** which works in combination with the eigenmode solver. After the modes of a filter have been calculated this very efficient technique can be used to derive the S-parameters for the filter with little additional simulation time.

The transient solver becomes less efficient for low frequency problems where the structure is much smaller than the shortest wavelength. In these cases it can be advantageous to solve the problem by using the **frequency domain** solver. This approach is most efficient when only a few frequency points are of interest.

If you are unsure which solver best fits your needs, please contact your local sales office for further assistance.

Each of these solvers' simulation results can then be visualized with a variety of different options. Again, a strongly interactive interface will help you quickly achieve the desired insight into your device.

The last – but not least – outstanding feature is the full parameterization of the structure modeler, which enables the use of variables in the definition of your component. In combination with the built in optimizer and parameter sweep tools, CST MICROWAVE STUDIO® is capable of both the analysis and design of electromagnetic devices.

Who Uses CST MICROWAVE STUDIO®?

Anybody who has to deal with electromagnetic problems in the high frequency range. The program is especially suited to the fast, efficient analysis and design of components like antennas, filters, transmission lines, couplers, connectors (single and multiple pin), printed circuit boards, resonators and many more. Since the underlying method is a general three dimensional approach, CST MICROWAVE STUDIO® can solve virtually any high frequency field problem.

The software is based on a method, which requires the discretization of the entire calculation volume; the applications are therefore limited by the electrical size of the structures. A very important feature of the transient solver is the excellent linear scaling of the computational resources with structure size. Currently, modern personal computers allow the simulation of structures with a size of up to about 100 wavelengths.

CST MICROWAVE STUDIO® Key Features

The following list gives you an overview of CST MICROWAVE STUDIO®'s main features. Please note that not all of these features may be available to you because of license restrictions. Please contact a sales office for more information.

General

- Native graphical user interface based on Windows 95/98, Windows NT, Windows 2000 and Windows XP.
- Fast and memory efficient FI-method
- Extremely good performance due to Perfect Boundary Approximation (PBA) and the new Thin Sheet Technique (TST).

Structure Modeling

- Advanced ACIS[1] based, parametric solid modeling front-end with excellent structure visualization
- Feature based hybrid modeler allows quick structural changes
- Import of 3D CAD data by SAT (e.g. AutoCAD ®), IGES, STEP, ProE®, CATIA 4® or STL files
- Import of 2D CAD data by DXF, GDSII and Gerber RS274X, RS274D files
- Import of a visible human model dataset
- Export of CAD data by SAT, IGES, STEP, STL, DXF, DRC or POV files
- Parameterization even for imported CAD files

Transient Simulator

- Efficient calculation for loss-free and lossy structures
- Broadband calculation of S-parameters from one single calculation run by applying DFT's to time signals
- Calculation of field distributions as a function of time or at multiple selected frequencies from one simulation run
- Adaptive mesh refinement in 3D
- Parallelisation of the transient solver using up to 32 processors on a PC

- Isotropic and anisotropic material properties
- Frequency dependent material properties
- Gyrotropic materials (magnetized ferrites)
- Surface impedance model for good conductors

- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Multipin ports for TEM mode ports with multiple conductors
- Multiport and multimode excitation
- Plane wave excitation

[1] Portions of this software are owned by Spatial Corp. © 1986 – 2002. All Rights Reserved.

- S-parameter symmetry option to decrease solve time for many structures
- Auto-regressive filtering for efficient treatment of strongly resonating structures
- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters
- Full de-embedding feature for highly accurate S-parameter results

- High performance radiating/absorbing boundary conditions
- Conducting wall boundary conditions
- Periodic boundary conditions without phase shift

- Calculation of various electromagnetic quantities such as: Electric fields, magnetic fields, surface currents, power flows, current densities, power loss densities, electric energy densities, magnetic energy densities in time and frequency domain
- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.)
- Antenna array farfield calculation
- RCS calculation
- Calculation of SAR distributions

- Discrete elements (lumped resistors) as ports
- Ideal voltage and current sources for EMC problems
- Lumped R, L, C, (nonlinear) Diode elements at any location in the structure
- Rectangular shaped excitation function for TDR analysis
- User defined excitation function

- Automatic extraction of cascaded SPICE (R, L, C, G) network models. Verification of the result by running SPICE.

- Automatic parameter studies by using the built in parameter sweep tool.
- Automatic structure optimization for arbitrary goals using the built in optimizer

Frequency Domain Simulator

- Efficient calculation for loss-free and lossy structures
- Isotropic and anisotropic material properties
- Adaptive frequency sampling for automatic frequency sweeps
- User defined frequency sweeps

- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters

- High performance radiating/absorbing boundary conditions
- Periodic boundary conditions including phase shift

- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.)
- Antenna array farfield calculation
- Calculation of electromagnetic quantities such as: Electric fields and magnetic fields.

- Discrete elements (lumped resistors) as ports
- Lumped R, L, C elements at any location in the structure

- Automatic extraction of cascaded SPICE (R, L, C, G) network models. Verification of the result by running SPICE.

Eigenmode Simulator

- Calculation of modal field distributions in closed, loss free structures
- Isotropic and anisotropic materials
- Parallelisation using up to two processors on a PC

- Periodic boundary conditions including phase shift
- Calculation of losses and Q-factors for each mode (perturbation method)

- Automatic parameter studies using the built in parameter sweep tool.
- Automatic structure optimization for arbitrary goals using the built in optimizer

Modal Analysis Simulator

- Broadband calculation of S-parameters from the modal field distributions calculated using the eigenmode solver.
- Isotropic and anisotropic materials
- Parallelisation using up to two processors on a PC

- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters
- Calculation of losses and Q-factors for each mode (perturbation method)

- Automatic extraction of cascaded SPICE (R, L, C, G) network models. Verification of the result by running SPICE.

- Automatic parameter studies by using the built in parameter sweep tool.
- Automatic structure optimization for arbitrary goals by using the build in optimizer

Visualization

- Display of port modes (with propagation constant, impedance, etc.)

- Display of S-parameters in xy-plots (linear or logarithmic scale)
- Display of S-parameters in smith charts and polar charts
- Online visualization of intermediate results during simulation

- Various field visualization options in 2D and 3D for electric fields, magnetic fields, power flows, surface currents, etc.
- Movie display of field distributions

- Display of farfields (fields, gain, directivity, RCS) in xy-plots or polar plots.
- Display of farfields (fields, gain, directivity, RCS) in scattering maps and radiation plots (3D)

- Display and integration of 2D and 3D fields along arbitrary curves.

Result Export

- Export of S-parameter data as TOUCHSTONE files
- Export of result data such as fields, curves, etc. as ASCII files
- Export screen shots of result field plots

Automation

- Powerful VBA (Visual Basic for Applications) compatible macro language including editor and macro debugger
- OLE automation for seamless integration into the Windows environment (e.g. Microsoft Office, MATLAB, AutoCAD, MathCAD, Windows Scripting Host, etc.)

About This Manual

This manual is primarily designed to enable a quick start to CST MICROWAVE STUDIO®. It is not intended to be a complete reference guide to all the available features but it will give you an overview of the key concepts. Understanding these concepts will then allow you to learn how to use the software efficiently with the help of the online documentation.

The main part of the manual is a so-called Quick Tour (Chapter 2) which will guide you through the most important features of CST MICROWAVE STUDIO®. We strongly encourage you to study this chapter carefully.

Document Conventions

- Commands which are accessed through the main window are printed as follows*: menu bar item⇨menu item.* This means that you first should press the “menu bar item” (e.g. “File”) and then select the corresponding “menu item” from the opening menu (e.g. “Open”).

- Buttons which should be pressed within dialog boxes are always written in italics, e.g. *Ok*.

- Key combinations are always joined with a plus (+) sign. Ctrl.+S means that you should hold down the “Ctrl.” key whilst pressing the “S” key.

Your Feedback

We are constantly striving to improve the quality of our software documentation. If you have any comments on the documentation, please send them to: support@cst.de

Contacting CST – Computer Simulation Technology

CST – Computer Simulation Technology would be happy to receive your feedback. If you have any questions concerning sales, please contact your local sales office. Whenever you have problems using our software, please do not hesitate to contact Technical Support as described below.

CST Headquarters

CST – Computer Simulation Technology
Bad Nauheimer Strasse 19
D-64289 Darmstadt
Germany
Phone: +49 (0)6151-7303-0
Fax: +49 (0)6151-7303-10
Email: info@cst.de
WWW: http://www.cst-world.com

Worldwide Distribution Agents

For an up-to-date list of support centers, please refer to our homepage at http://www.cst-world.com.

Technical Support

Before contacting Technical Support you should check your manual and the Help system. If you still can not solve your problem, product support can be obtained in several ways:

Support area on our homepage

In this area you will find a regularly updated list of frequently asked questions. You can also access the latest tutorials online or download these files to your computer. Furthermore you may also download online updates and patches from this area.

In order to access the support area, you will need a password which is obtained by registering online when entering the support area. You will then receive a password within a day by e-mail.

You might find it useful to browse in this area on a regular basis to get all the latest information regarding improvements and corresponding patches and updates.

E-mail support

Please e-mail our support group (support@cst.de) with any problems which do not appear in the Frequently Asked Questions page. You will usually receive an answer within one working day. If necessary, our support group will give you a call, so please note your direct dial phone number in your e-mail.

Support via telephone and fax

In very urgent cases you may also call our telephone support. In this case, please contact your local dealer (see our homepage at http://www.cst-world.com). If the problem cannot be solved immediately you will receive a call from a member of our support group.

Chapter 2 — Installation

Installing CST MICROWAVE STUDIO® is a simple process. This chapter explains everything you need to know regarding installation. It covers the following sections:

- ⇨ Installation requirements
- ⇨ Installation instructions
- ⇨ Providing password information

Installation Requirements

Software Requirements

- The software runs under any one of the following operating systems: Windows NT 4.0, Windows 2000, Windows XP, Windows 95, Windows 98, Windows Me

Hardware Requirements

- Intel Pentium II or greater, PC Compatible (Intel Pentium III or IV processor strongly recommended).
- 100% OpenGL compatible graphics card
- CD-ROM drive
- 128 MB RAM (256 MB recommended)
- 1 GB free disk space.

Licensing Options

The software can be licensed either as a single PC version or as a network version. The single PC license allows the software to be started on a single PC as many times as process licenses are available. In contrast, the network license allows the software to be started on several PC's which are connected to a license server in a local area network.

Since this document is designed to help you with your first steps in CST MICROWAVE STUDIO®, we assume that you currently have a single PC evaluation license. In the following we will therefore focus on the installation of the single PC license. Please refer to the *Advanced Topics* manual on how to install the network license.

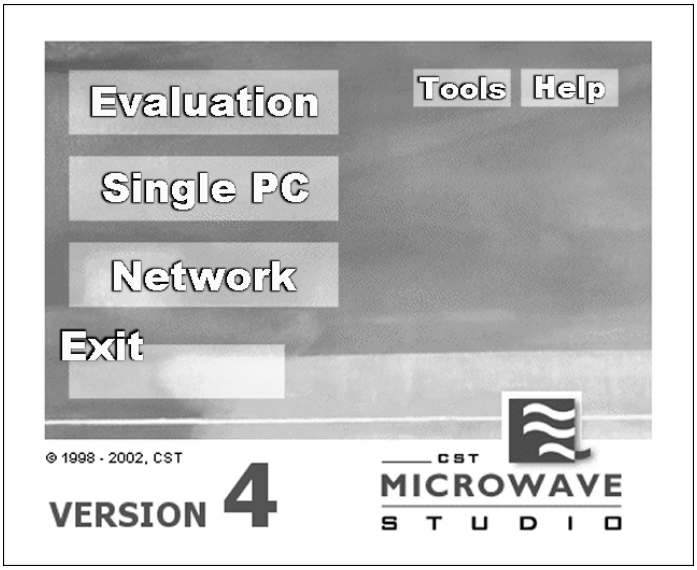
Installation Instructions

Before the software can be installed, the hardlock (dongle) has to be connected to the printer port. To do this, please switch off your computer, remove the printer cable (if any), connect the hardlock to the printer port and make sure that it fits properly. You may then re-attach the printer cable.

Note: If you have other hardlock-protected software products installed, some interference between the hardlocks is possible. In such cases, please remove all other hardlocks and re-try the installation of CST MICROWAVE STUDIO®. Please contact technical support for more information.

When you are installing the software on a Windows NT, Windows 2000 or Windows XP system you will need administrator privileges in order to start the installation. If you don't have these privileges on your local computer, please ask your system administrator for assistance.

Installing CST MICROWAVE STUDIO®'s software is very simple. Insert the appropriate CD-ROM into the drive and wait for the installation program to start. If you have deactivated the auto-run feature for your CD drive, you can start the installation by manually double-clicking at the *Setup* program in the CD's root directory. You should finally see the following screen:



If you are currently evaluating the software, please click on the evaluation license button. A single PC license installation works in the same way as the evaluation license procedure described below. For installing network licenses, please refer to the *Advanced Topics* manual.

After pressing the button, the actual software installation will be started. You should now follow the instructions on the screen and make sure that you read every screen as you advance. We recommend you to use the *Typical* setup to ensure that you can access all examples which might be of interest to you.

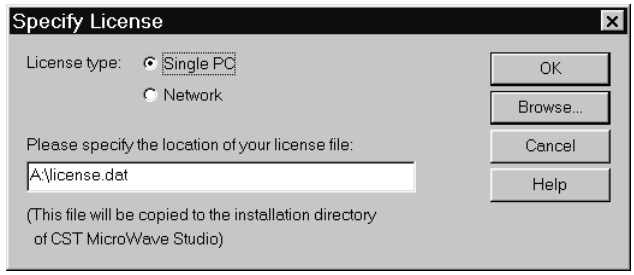
After the successful installation and rebooting of your computer, the software is ready to use. Please note that on Windows NT, Windows 2000 or Windows XP systems, you should again log in with administrator privileges in order to set up the license as described in the next section.

You can now start CST MICROWAVE STUDIO® by selecting the appropriate item in the *Start* menu.

Note: For the single PC version, the program's license is bound to a hardlock (dongle). You may install the program on several PCs, but you may only use it on one PC at a time. Please contact your local dealer for more information.

Providing Password Information

When you start CST MICROWAVE STUDIO® for the first time or whenever the license has expired, a dialog box will appear:



In this window you may specify the location of a valid password file. Usually you will receive your license file by e-mail. Please use your e-mail system to save the attached file to the hard disk and specify the location of this file in the dialog box by using the *Browse* button. The license file will be copied to the proper location and the software will be started afterwards.

Sometimes a license disk comes with the software. If this is the case just enter the disk into drive A: and press *Ok*. The program will now copy the license file to the proper location and start the software.

If your license is about to expire, a message window will appear when you start the software. If this happens, please contact you local dealer to extend the license period.

After the license has been properly set up and the software is running, you can log in without administrator privileges on Windows NT, Windows 2000 or Windows XP.

Chapter 3 — Quick Tour

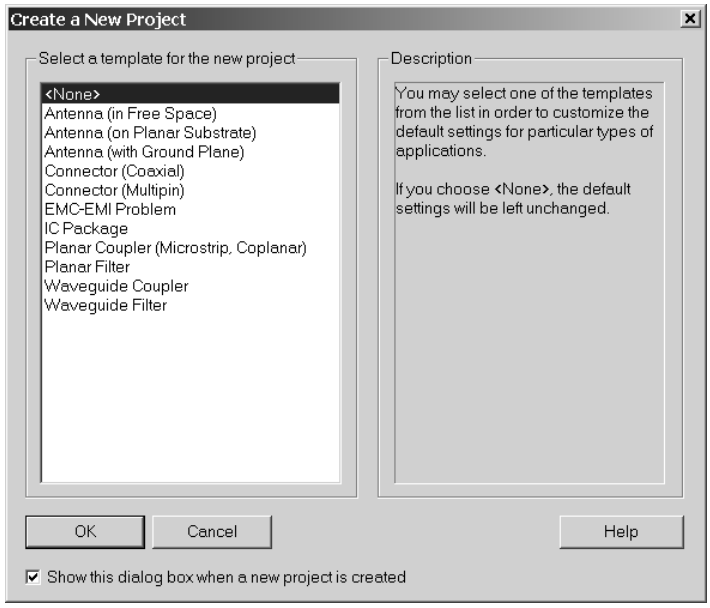
CST MICROWAVE STUDIO® is designed for ease of use. However, to get started quickly you will need to know a couple of things. The main purpose of this chapter is to provide an overview of the software's many capabilities. Please read this chapter carefully, as this may be the fastest way to learn how to use the software efficiently.

This chapter covers the following sections:

- ⇨ Overview of the user interface structure.
- ⇨ Creating and viewing some simple structures.
- ⇨ Modeling and simulating a simple coaxial cable bend with a stub.
- ⇨ Parameter studies of the model and automatic optimization of the structure.

Starting the software

After starting CST MICROWAVE STUDIO® and confirming to create a new project by pressing the *Ok* button in "Welcome" dialog box, you will see the following window:



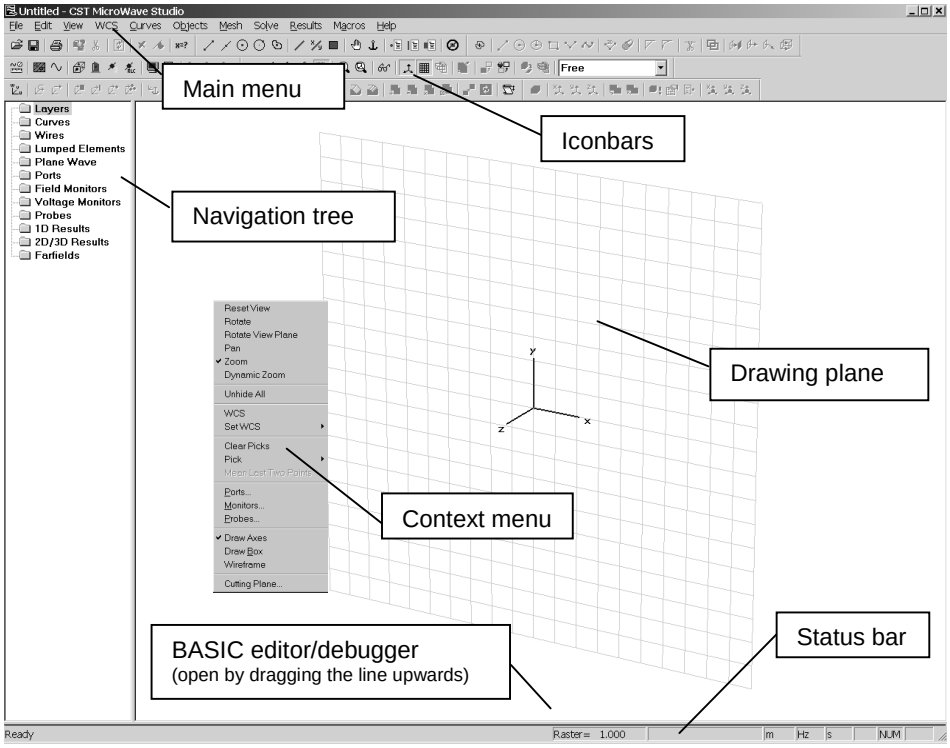
This dialog box will always appear when a new project is created.

Here you can select one of the predefined templates in order to automatically set proper default values for the particular type of device which you are going to analyze. Although all of these settings can be changed manually at any time later, it is more convenient to start with proper defaults, especially for new users. However, as an advanced user you can customize the predefined templates or add new ones.

For the first part of this introduction you may simply choose <*None*> and press the *Ok* button.

Overview of the User Interface's Structure

The following picture shows a screenshot of CST MICROWAVE STUDIO®'s main window .



The navigation tree is an essential part of the user interface. From here you may access structural elements as well as simulation results. The following sections will explain the many different items in this tree window.

The context menus are a very flexible way of accessing the frequently used menu commands for the current context. The contents of this menu (which can be opened by pressing the right mouse button) change dynamically.

The drawing plane is the plane on which you will draw the structure's primitives. As the mouse is only a two dimensional locator, even when defining three dimensional structures, the coordinates must be projected onto the drawing plane in order to specify a three dimensional location. Since you may change the location and orientation of the drawing plane by means of various tools, this feature makes the modeler very powerful.

The most advanced part of the user interface is the built-in BASIC interpreter. This language is almost 100% compatible with the Visual Basic for Application language. You

may use this language either for creating your own structure library or for the automation of common tasks. However, when you are just getting started with CST MICROWAVE STUDIO® you will have relatively little to do with this feature. We will therefore ignore the BASIC interpreter here and discuss it in more detail in the online documentation and the *Advanced Topics* manual.

The other elements of the user interface are standard for a Windows based application and so we assume that you are familiar with these controls.

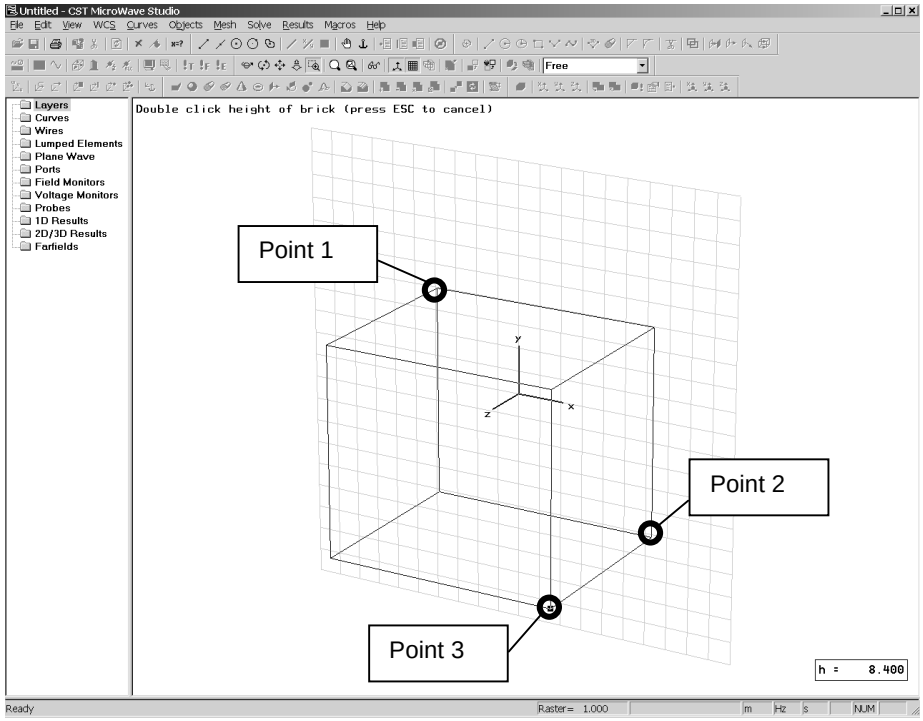
Creating and Viewing Some Simple Structures

After the lengthy explanation of the user interface, let's jump into the procedure of creating a simple structure. Many complex structures are composed of very simple elements or so-called primitives. In the following we will draw such a primitive: a brick.

Create a First "Brick"

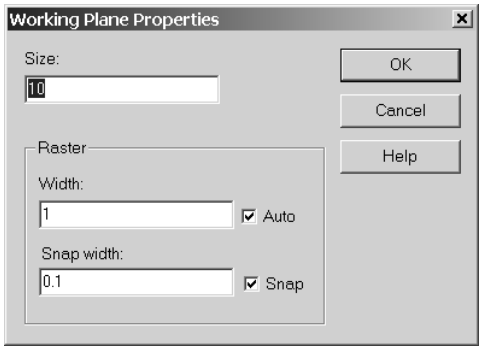
1. As your very first step, activate the "Brick" tool by pressing the corresponding icon in the object toolbar:  (you could also chose *Objects⇨Basic Shapes⇨Brick* from the main menu). Now you will be prompted to select the first point of the brick's base in the drawing plane (see the text line in the main window).
2. You may set a starting point by double-clicking a location on the drawing plane.
3. Now you can select the opposite corner of the brick's base on the drawing plane by double-clicking on it.
4. In the third step you have to define the height of the brick by dragging the mouse. Double-click to fix the height of the brick.
5. Finally a dialog box will open which shows you the numerical values of all coordinate locations you have entered. Pressing *Ok* here will store the settings and create your first primitive. Congratulations!

The following picture gives an overview of the three double-clicks used to define the brick:



Before we continue drawing some other simple shapes, we will spend some time on the different methods of setting a point.

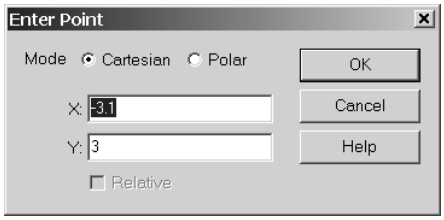
The simplest way to set a point is to double-click its location on the drawing plane, as above. However, in most cases the structure coordinates have to be entered at a high precision. In this case, the snap to grid mode should be activated. You will find the corresponding option dialog box under "*Edit⇨Working Plane Properties*" in the menu bar. The following dialog box will appear:



Here you may specify whether the mouse coordinates should *Snap* to a raster (which is the default) or not. Furthermore you may specify the raster *Snap width* in the corresponding field. The raster *Width* entry only influences the size of the raster which is drawn on the screen. The coordinate mapping is independent from this setting.

Please note that pressing the Help button in a dialog box always opens a particular help page containing more information about the dialog box and its settings.

Another way to specify a coordinate is by pressing the *TAB* key whenever a location is expected. In this case a dialog box will appear in which you may numerically specify the location. The following example shows a dialog box which is shown when the first point of a shape has to be defined:



You may specify the position either in *Cartesian* or in *Polar* coordinates. The latter type is measured from the origin of the coordinate system. The *Angle* is between the x-axis and the location of the point, and the *Radius* is the point's distance from the origin.

When the first point has been set, the *Relative* option will be available. If you check this item, the entered coordinates are no longer absolute (measured from the origin of the coordinate system), but relative to the last point entered.

The coordinate dialog boxes will always show the current mouse location in the entry fields. However, often a point should be set in the center of the coordinate system (0,0). If you press *Shift+TAB*, the coordinate dialog box will open with zero values in the coordinate fields.

The third way to enter accurate coordinates is by clicking estimated values using the mouse and then correcting the values in the final dialog box. You may skip the definition of points by using the mouse, at any time, by pressing the *ESC* key. In this case, the shape dialog box will open immediately.

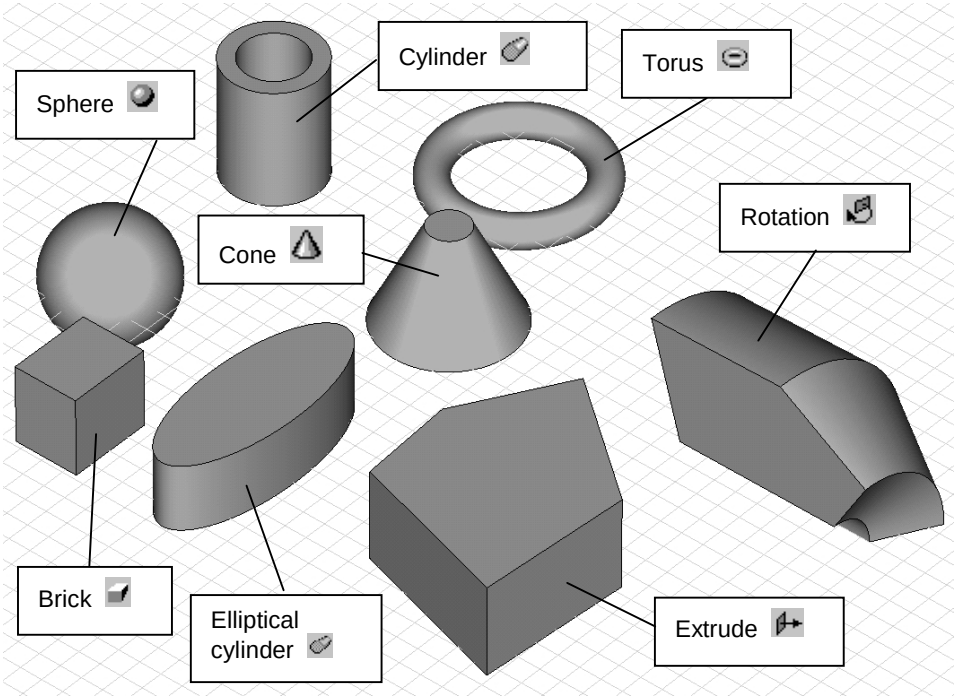
By pressing the *ESC* key twice, the shape generation can be aborted. Pressing the *Backspace* key deletes the previously selected point. If no point has been selected, the shape generation will also be aborted.

Please note that another mode exists for the generation of bricks. When you are requested to pick the opposite corner of the brick's base, you may also specify a line rather than a rectangle. In this case, you will be asked to specify the width of the brick as a third step before specifying the height. This feature is quite useful for construction tasks such as building a micro strip line centered on a substrate.

Since picking two points on a line is quite sensitive to exact mouse positioning, the mouse movement will be restricted to 90 degree offsets from a first point when the *Shift* key is being held down while the mouse is moved across the screen.

An Overview of the Basic Shapes Available

The following picture gives a brief overview of all basic shapes which can be generated in a similar way to the brick (as described above).

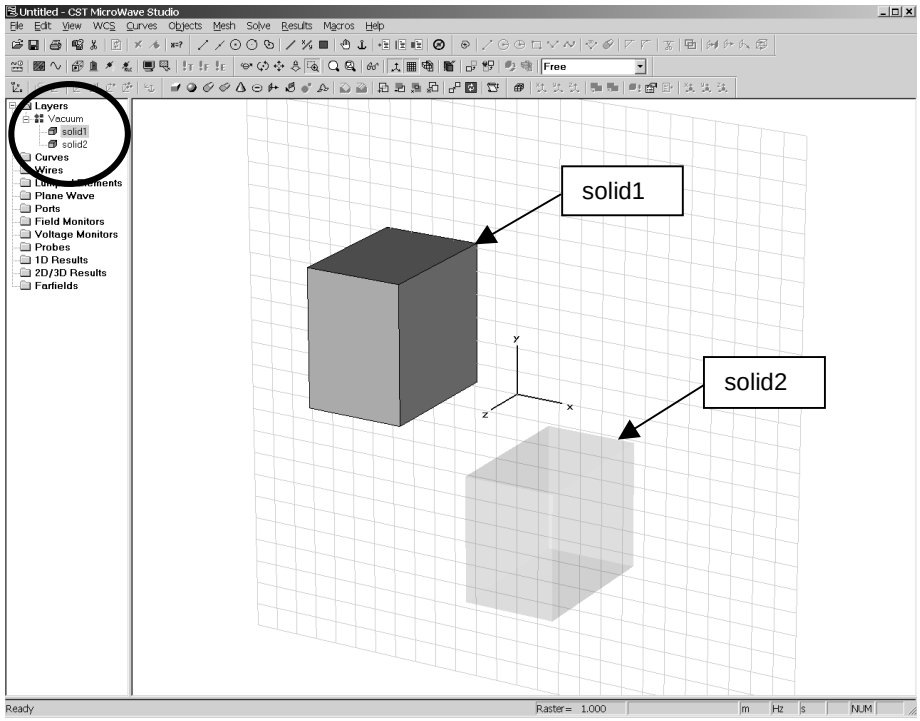


At this stage it is advisable to play around a bit with the shape generator in order familiarize yourself with the user interface.

Selecting Previously Defined Shapes and Grouping Shapes into Layers

After a shape has been defined, it is automatically cataloged in the navigation tree. All shapes can be found in the *Layers* folder. If you open this folder you will find a subfolder called: *Vacuum*. This folder contains all defined shapes. The name for each primitive is assigned to the final shape dialog box when the shape is created. The default names start with “solid” followed by an increasing number: solid1, solid2, ..., etc.

You may select a shape by simply clicking on the corresponding item in the navigation tree. Please note that after selecting a shape it will be displayed opaquely while all other shapes are drawn transparent (see the picture below). This is how CST MICROWAVE STUDIO® visualizes shape selection. A shape can also be selected by double-clicking on it in the main window. In this case the corresponding item in the navigation tree will also be selected. Please take a few seconds to familiarize yourself with the shape selection mechanism.



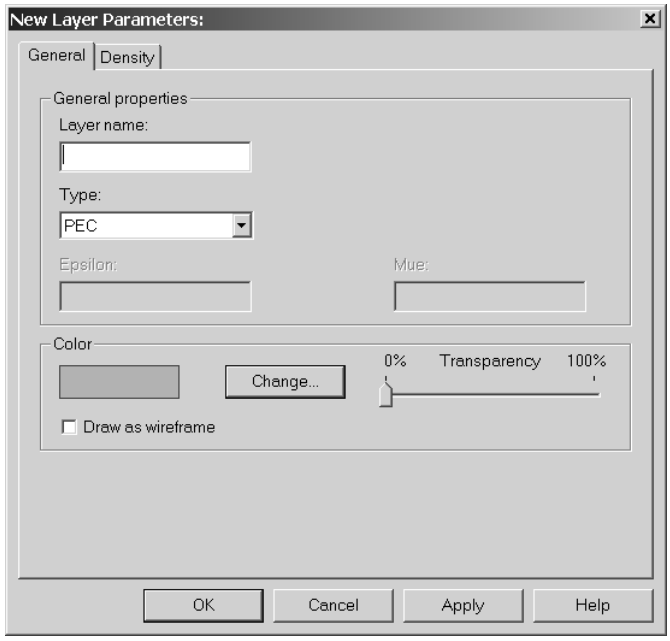
You may change the name of a shape by selecting it and then choosing *Edit⇨Rename Object* from the menu bar or pressing the *F2* key. The name of the shape can now be changed by editing the item text in the navigation tree.

Now that we have discussed how to select an object, we should spend a little time on the grouping of shapes in so called layers. Each layer is a subfolder of the *Layers* folder in the navigation tree. Each individual layer folder contains an arbitrary number of shapes.

The purpose of the layer structure is to group together objects with same material properties. In other words, the material properties (and colors) do not belong to the shapes directly but to the corresponding layer. This means that all shapes in a particular layer have the same material properties and are drawn in the same color.

The only way to change the material properties or the color of an individual shape is to assign it to another layer. This can be done by selecting the shape and choosing *Edit⇨Change Layer* from the menu bar (this option can also be found in the context menu when a shape is selected). The following dialog box will then open:

In this dialog box you may select an existing layer from the list or define a new one by pressing the *New* button. In the latter case another dialog box opens:

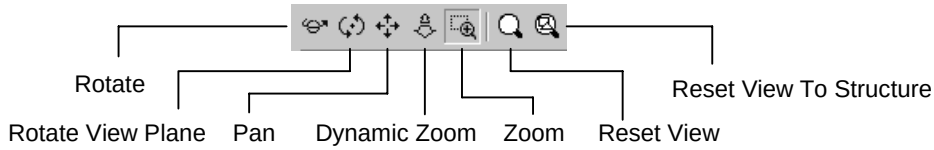


In this dialog box you must specify the *Layer name* and the *Layer type* (perfect conductor, normal dielectric, anisotropic or surface impedance material). You can also change the color of the layer by pressing the *Change* button. After pressing the *Ok* button, the new layer will be stored and appears in the list of layers.

Changing the View

So far we only have created and viewed the shapes by using the default view. You can change the view at any time (even during shape generation) by some simple commands which will be explained in the following.

The view will change whenever you drag the mouse whilst holding down the left button, depending on the mode selected. The mode can either be selected from the main menu by choosing *View⇨Mode⇨Rotate/Rotate View Plane/Pan/Zoom/Dynamic Zoom* or by selecting the appropriate item from the view toolbar:



The mode setting effects the behavior as follows:

- **Rotate**: The structure will be rotated around the two screen axes.
- **Rotate View Plane**: The structure will be rotated in the screen's plane.
- **Pan**: The structure will be translated in the screen plane following the mouse cursor movement.
- **Dynamic Zoom**: Moving the mouse upwards will decrease the zoom factor while moving the mouse downwards increases the zoom factor.
- **Zoom**: In this mode a rubber band rectangle will be defined by dragging the mouse. After releasing the left mouse button, the zoom factor and the view location will be updated so that the rectangle fills up the screen.

The dynamic view-adjusting mode is always left when the left mouse button is released. You can reset the zoom factor by choosing *View⇨Reset View* from the main menu or from the context menus. Alternatively you could press the corresponding item in the view toolbar.

One of the most important view-changing commands is activated by *View⇨Reset View to Structure* or by pressing the *Space*-bar. This command will zoom the defined structure to a point where it fits well into the drawing window.

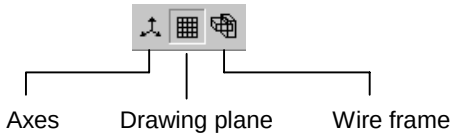
Since changing the view is a frequently used operation which will sometimes be necessary even during the process of interactive shape creation, some useful shortcut keys exist. Please press the appropriate keys and drag the mouse with the left button pressed:

- ***Ctrl*:** Same as "rotate" mode.
- ***Shift*:** Same as "plane rotation" mode.
- ***Shift + Ctrl:*** Same as "pan" mode.

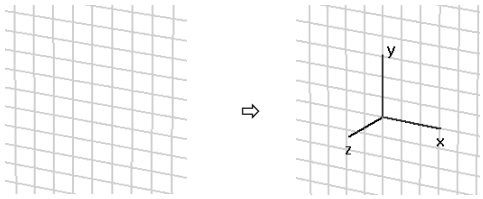
A mouse wheel has the same effect as the "dynamic zoom".

In addition to the options described above, some specific settings are available to change the visualization of the model.

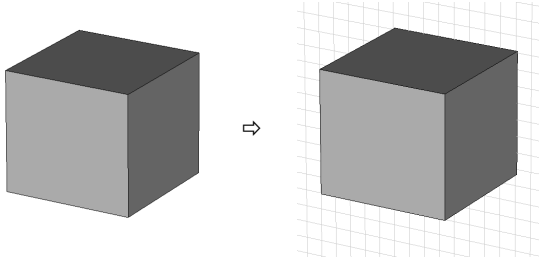
All these settings can be specified by choosing the appropriate item from the *View* menu. Furthermore these settings can be specified with the corresponding item in the view toolbar:



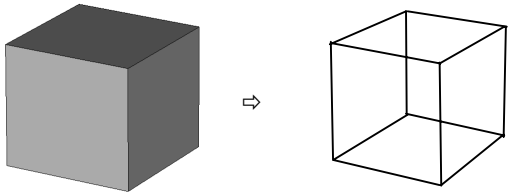
Axes (*View⇨View Options* dialog box, *Ctrl*+*A*): This option specifies whether the coordinate system is displayed or not:



Working plane (*View⇨View Options* dialog box, *Alt*+*W*): With this flag you may specify whether the drawing plane is visible or not.



Wireframe (*View⇨View Options* dialog box, *Ctrl*+*W*): This flag indicates whether all shapes are displayed as simple wire models or as solid shaded objects.



Applying Geometric Transformations

So far you have seen how to model simple shapes and how to change the view of your model. The first of the more advanced operations on your model are geometric transformations.

In the following we assume that you have already selected the shape to which a transformation shall be applied (e.g. by double-clicking on it in the main view).

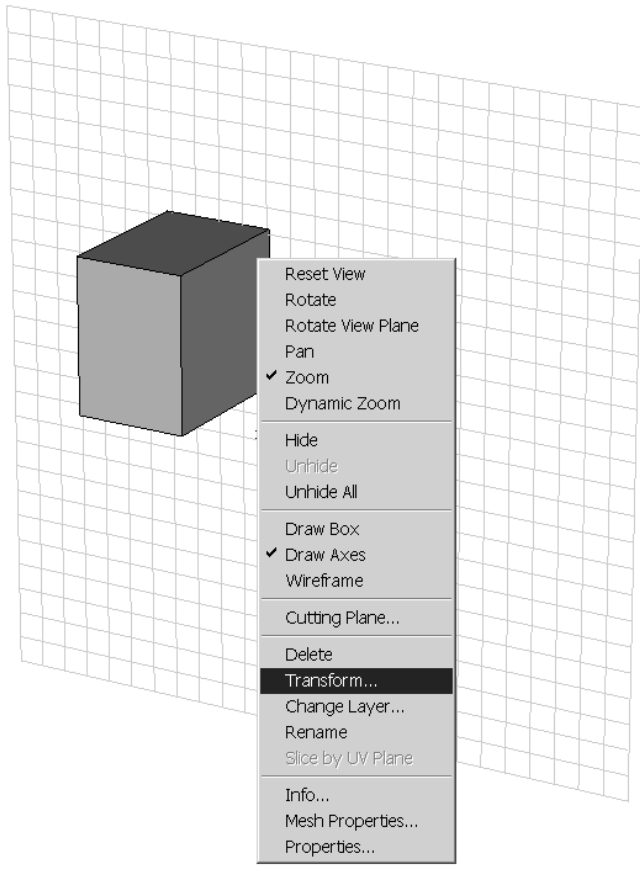
You can then open the transformation dialog box either in the main menu by choosing *Objects⇨Transform Shape*, by choosing the item *Transform* from the context menu, or by pressing  in the objects toolbar.

In the dialog box you are invited to choose one of the following transformations:

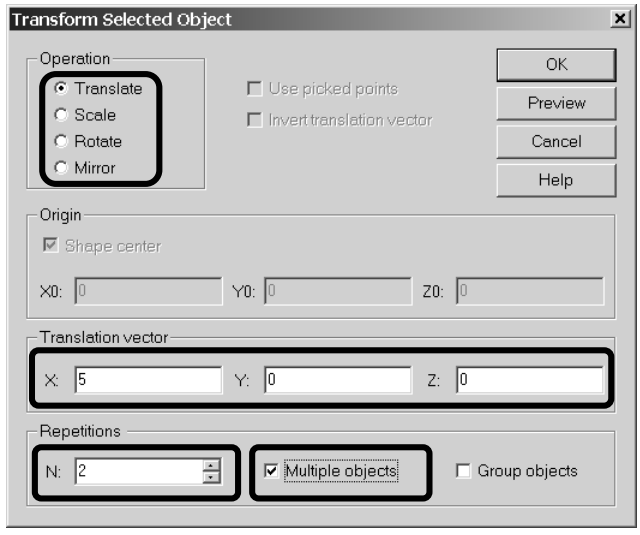
- **Translate**: This transformation applies a translation vector to the selected shape.
- **Scale**: By choosing this transformation you can scale the shape along the coordinate axes. You may specify different scaling factors in the different coordinate directions.
- **Rotate**: This transformation applies a rotation of the shape around a coordinate axis by a fixed angle. You may additionally specify the rotation center in the *Origin* field. The center may either be the center of the shape (calculated automatically) or any specified point. The rotation angle and axis settings are specified by entering the corresponding angle in the entry field for the corresponding axis. (e.g. entering 45 in the y field while leaving all other fields set to zero performs a rotation around the y axis of 45 degrees.)
- **Mirror**: This transformation allows you to mirror the shape at a specified plane. A point on the mirror plane is specified in the *Origin* field and the plane's normal vector is given in the *Mirror plane normal* entries.

For all these transformations you may specify whether the original shape should be kept (*Multiple objects* option) or deleted. Furthermore you can specify in the *Repetitions* field how many times the same transformation is to be applied to the shape (each time producing a new shape when the *Multiple objects* option is active).

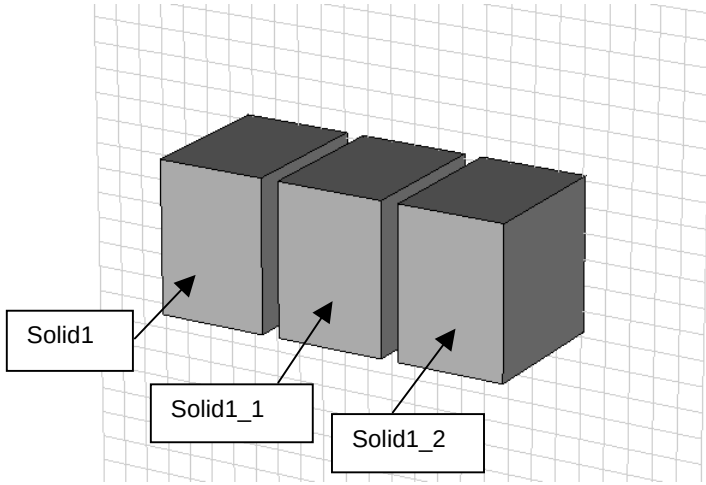
A final simple example will demonstrate the usage of the transformation dialog box. Assume a brick has been defined and selected as depicted below. Open the transform dialog box by choosing the appropriate item from the context menu (or *Objects⇨Transform Shape*).



Now apply a translation to the shape by a translation vector (5, 0, 0) and produce multiple objects as the transformation is applied twice:



You will end up obtaining the following shapes:

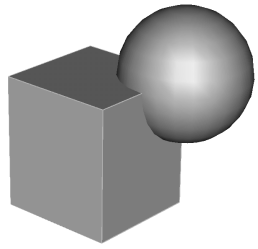


Please note that for each transformation the name of the transformed shape is either kept (no multiple objects) or extended by appendices _1, _2, etc. in order to obtain unique names for the shapes.

Combine Shapes by Using Boolean Operations

Probably the most powerful operation to create complex shapes is the combination of simple shapes by using so called Boolean operations. These operations allow you to add two shapes together, to subtract one shape from another, to insert shapes into each other, and to intersect two shapes.

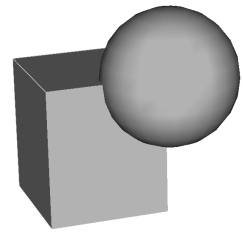
In the following we will consider two shapes – a sphere and a brick – on which we will perform Boolean operations.



In the following list we will name all available Boolean operations and show the resulting body for each combination:

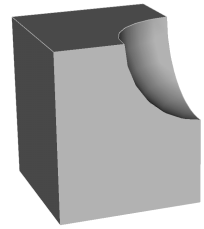
Add brick to sphere:

Add both shapes together to obtain one single shape. The resulting shape will get the name and layer of the first shape



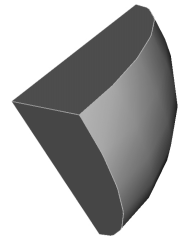
Subtract sphere from brick

Subtract the first shape from the second to obtain one single shape. The resulting shape will get the name and layer of the shape from which the other shape is subtracted.



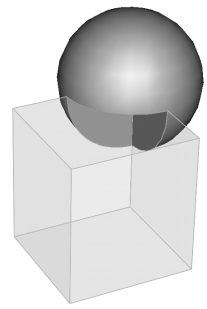
Intersect brick and sphere

Intersect two shapes to form a single shape. The resulting shape will get the name and layer from the first shape of this operation.



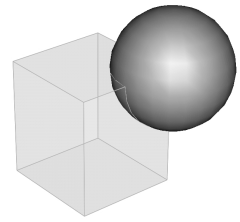
Trim sphere
= Insert brick into sphere

The first shape will be trimmed by the boundary of the second shape. Both shapes will be kept. The resulting shapes will have no intersecting volume.



Insert sphere into brick
= Trim brick

The first shape will be inserted into the second one. Again both shapes will be kept. The resulting shapes will have no intersecting volume.



Please note that not all of these Boolean operations above are directly accessible. As you see, some of the operations are redundant (e.g. a trimming operation can be replaced by an insertion operation when the order of the shapes is reversed.)

The following Boolean operations are available from the main menu by choosing the corresponding items: *Objects⇨Boolean⇨Add/Subtract/Intersect/Insert*. All operations are only accessible when a shape has been selected (in the following referred to as

"first" shape). After the Boolean operation has been activated you will be prompted to select the "second" shape. Pressing the *RETURN* key or selecting  from the objects toolbar will perform the Boolean combination. The result depends on the type of the Boolean operation and is given in the following list:

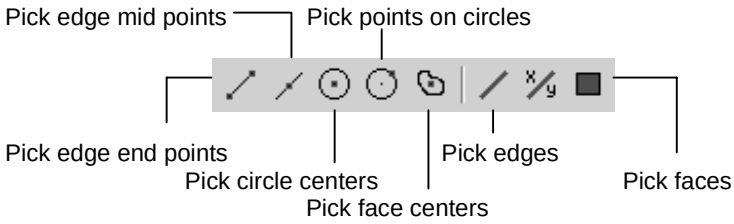
- **Add:** Add the second shape to the first one. Keeps the name and layer of the first shape.
- **Subtract:** Subtract the second shape from the first one. Keeps the name and layer of the first shape.
- **Intersect:** Intersect the first with the second shape. Keeps the name and layer of the first shape.
- **Insert:** Insert the second shape into the first one. Keeps both shapes while only changing the first shape.

The trim operations are only available in a special "Shape intersection" dialog box which appears when a shape is created which intersects or touches areas with existing shapes. This dialog box will be explained later.

Pick Points, Edges or Faces from Within the Model

Many construction steps require the selection of points, edges or faces from the model. The following section explains how to select these elementary entities interactively.

For each of these so called "pick operations" you first have to select the appropriate pick tool. This may be selected either from the menu item *Objects⇨Pick⇨Pick Point/...* or from an item in the pick toolbar:



After activating a pick tool, the mouse cursor will change indicating that a pick operation is in progress. In addition to this, all pickable elements (e.g. points, edges or faces) will be highlighted in the model. Now you may double-click on an appropriate item (point, edge or face). Alternatively you may cancel the pick mode by pressing the *ESC* key, selecting *Leave pick mode* from the context menu or pressing the  item in the main toolbar.

Note: You can not pick edges or faces of a shape when another shape is currently selected. In this case you should either select the proper shape or deselect all shapes.

As soon as you double-click in the main view, the pick mode will be terminated and the selected point, edge, or face will be highlighted (if any). Please note that if the  symbol is pressed in the pick toolbar, the pick operation will not terminate after double-clicking. In this case you have to cancel the pick mode as described above. This mode is useful when multiple points, edges or faces have to be selected and it would be cumbersome to re-enter the pick mode several times.

The following list gives an overview of which entities can be picked in the various pick modes and what effect this picking will have. There are also some interesting shortcuts for efficiently activating the pick modes. These shortcuts are available only when the main structure view is the active window. You can activate this view by clicking in it with the left mouse button. In the following list, all shortcuts are depicted in brackets besides the corresponding pick operation.

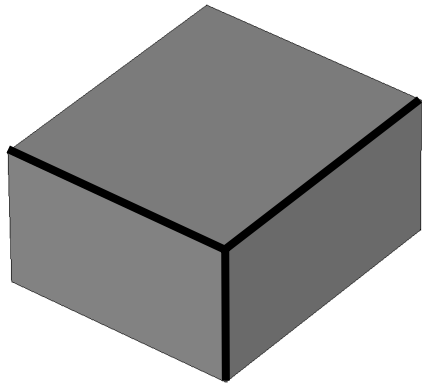
- **Pick edge end points (P):** Double-click close to the end point of an edge. The corresponding point will then be selected.
- **Pick edge mid points (M):** Double-click on an edge. The mid point of this edge will then be selected.
- **Pick circle centers (C):** Double-click on a circular edge. The center point of this edge will be selected. Please note that the edge need not necessarily belong to a complete circle.
- **Pick points on circles (R):** Double-click on a circular edge. Afterwards an arbitrary point on the circle will be selected. This operation is useful when matching radii in the interactive shape creation modes.
- **Pick face centers (A):** Double-click on a planar face of the model. The center point of this face will then be selected.
- **Pick edges (E):** Double-click on an edge of the model, this will then be selected.
- **Pick faces (F):** Double-click on a face of the model, this will then be selected.
- **Pick edge chain (Shift+E):** Double-click on an edge of the model. If the selected edge is a free edge, a connected chain of free edges will be selected. If the selected edge is connected to two faces, a dialog box will appear with which you can specify which one of the two possible edge chains bounding the faces will be selected. In both cases the selection chain stops at previously picked points, if any.
- **Pick face chain (Shift+F):** Double-click on a face of the model. This function will then automatically select all faces connected to the selected face. The selection stops at previously picked edges, if any.

The pick operations for selecting points from the model are also valid in the interactive shape creation modes. Here, whenever a double-click is requested to enter the next point, the pick mode may alternatively be entered. After leaving this mode the picked point will be taken as the next point for the shape creation.

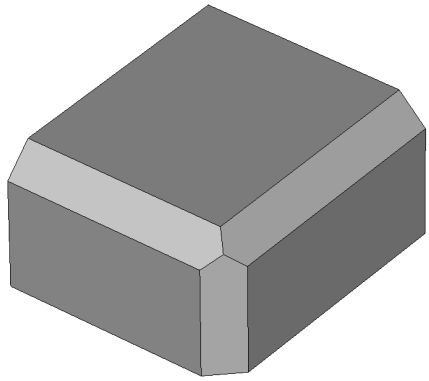
Previously picked points, edges or faces can be cleared by using the *Objects⇨Clear Picks* command (shortcut D in the main view) or by pressing  in the pick toolbar.

Chamfer and Blend Edges

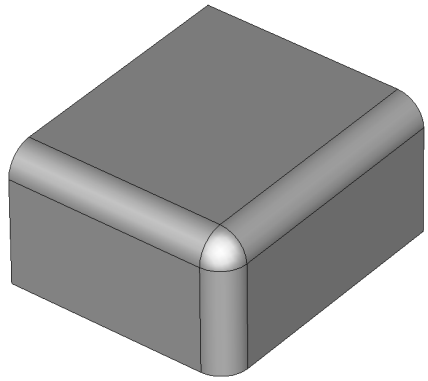
One of the most common applications for picked edges are the chamfer and blend edge operations. In the following we assume that you have created a brick and selected some of its edges, as shown in the following picture:



Now you can perform a chamfer edge operation which can be activated either by choosing *Objects⇨Chamfer Edges* from the main menu or by pressing in the objects toolbar. In the following dialog box you can specify the width of the chamfer. The structure should end up looking similar to the one depicted below:

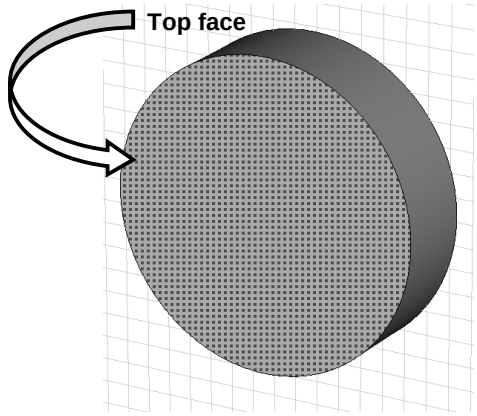


Alternatively you could perform a blend edges operation which is activated either by choosing *Objects⇨Blend Edges* in the main menu or by pressing in the objects toolbar. In the following dialog box you can specify the radius of the blend. The result should look similar to the following picture:

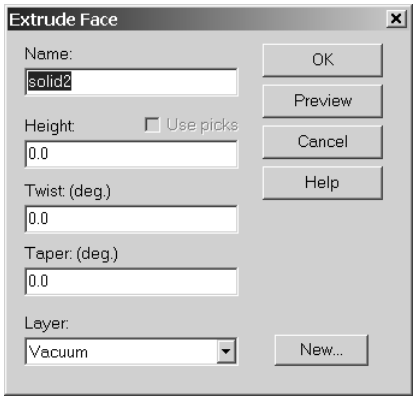


Extrude, Rotate and Loft Faces

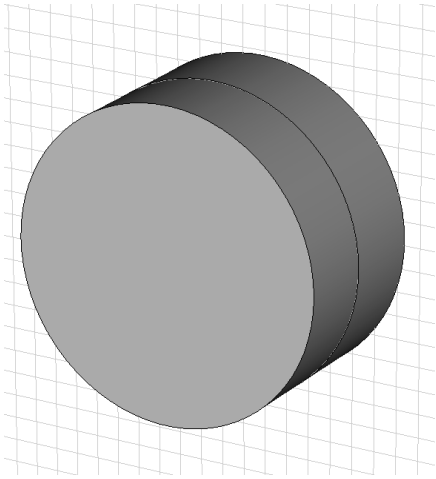
The chamfer and blend tools are very common operations on picked edges, and the extrude, rotate and loft operations are equally typical construction tools for use on picked faces. In the following we assume an existing cylinder with a picked top face:



Now we can extrude this face by simply selecting the *Objects⇨Extrude* ( ) tool. When a planar or cylindrical face is picked before this tool is activated, the extrusion refers to the picked face and the dialog box will open immediately:

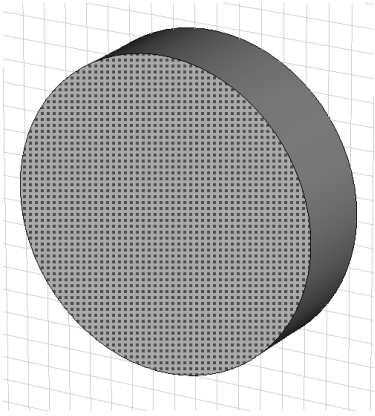


If no face is picked in advance, an interactive mode will be entered in which you can define polygon points for the extrusion profile. However, in this example you should enter a height and press the *Ok* button. Finally your structure should look as follows:

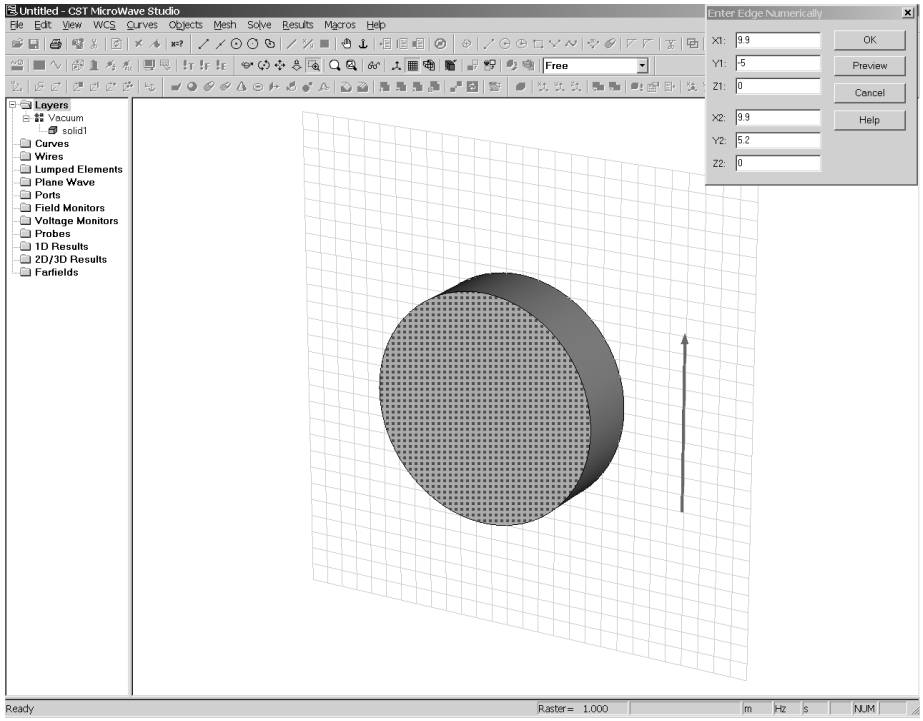


The extrusion tool has created a second shape by the extrusion of the picked face.

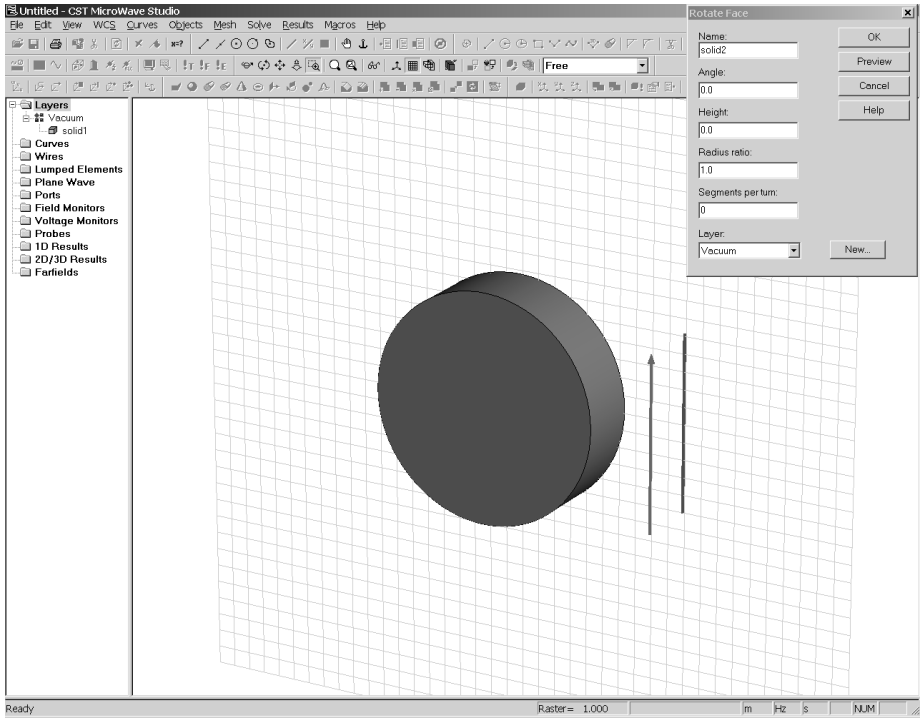
For the rotation, you should start with the same basic geometry as before:



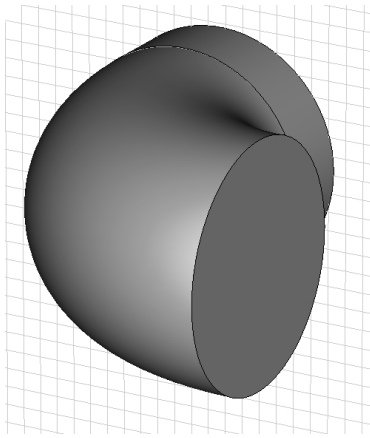
The rotation tool requires the input of both a rotation axis and a picked face. The rotation axis can either be a linear edge picked from the model or a numerically specified edge. In this example, you should specify the edge by selecting the *Objects⇨Pick⇨Edge from Coordinates* ( ) tool from the pick toolbar. Afterwards you will be requested to pick two points on the drawing plane to define the edge. Please select two points similar to those in the following picture:



In the numerical edge dialog box press the *Ok* button to store the edge. Afterwards you can activate the rotate face tool by selecting *Objects⇨Rotate*.

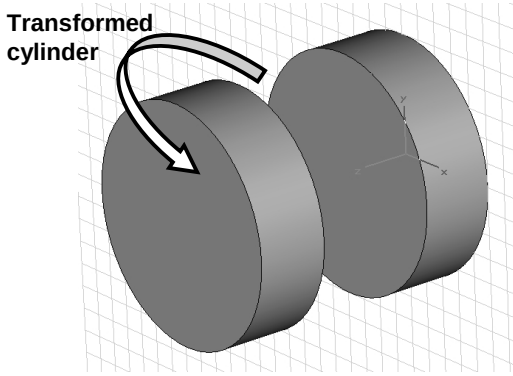


The previously selected rotation axis is automatically projected into the face's plane (blue vector) and the rotation tool dialog box is opened immediately. In this dialog box you can specify an *Angle* (e.g. 90 degrees) and press *OK*. The final shape should now look as follows:

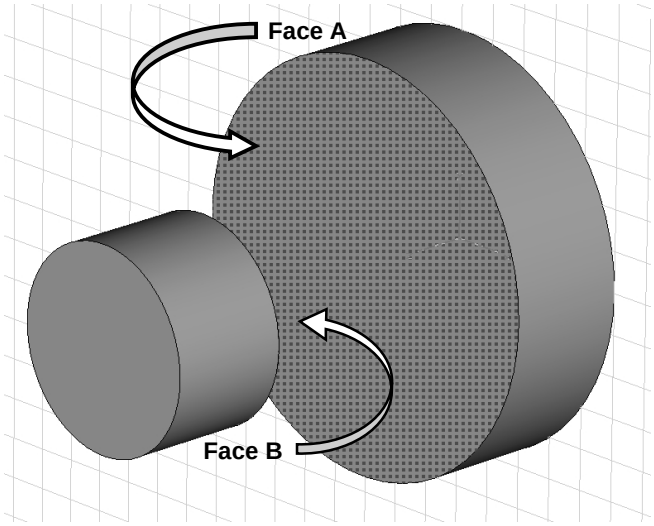


Please note that the rotate tool enters an interactive polygon definition mode similar to the one/that in the extrude tool if no face is picked before the tool is activated.

One of the more advanced operations is the generation of lofts between picked faces. As an example, please construct the following model by defining a cylinder (e.g. radius=5, height=3) and transforming it along its axis by a certain translation (e.g. (0,0,8)) using the *Multiple objects* option:

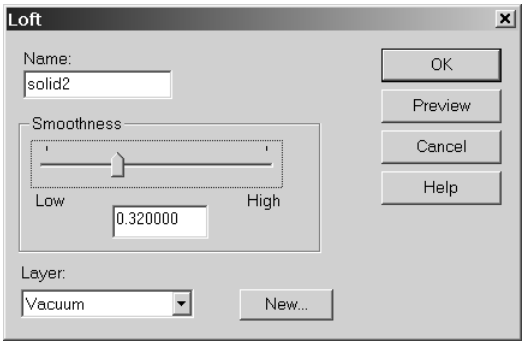


Your next step is to select the transformed cylinder and apply a scaling transformation to it by shrinking its size along the x and y axes by 0.5 while keeping the z-scale at 1.0:

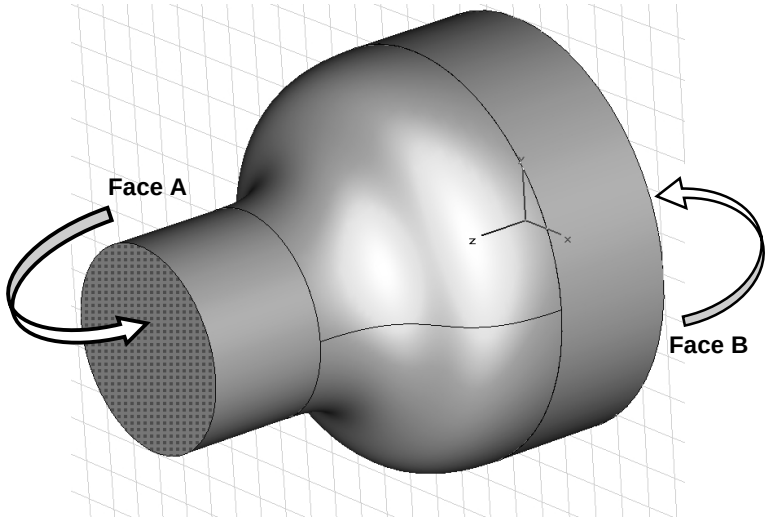


Now you should pick the adjacent top and bottom faces of the two cylinders as shown in the picture above. Afterwards you can activate the loft tool by selecting Objects⇨*Loft* (). In the following dialog box you can set the smoothness to a reasonable value and press the *Preview* button to get an impression of the shape. Please drag the *Smoothness* slider such that the shape has a relatively smooth transition between the two picked faces before pressing *Ok*.

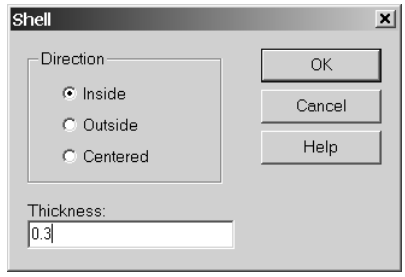
Note: You should select the corresponding shape before picking its face. Since all other shapes become transparent, it is easier to pick the desired face even "through" other shapes.



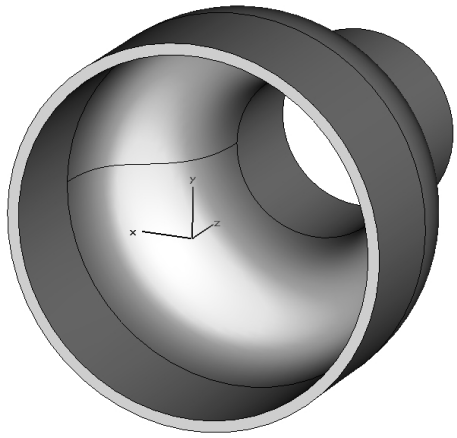
Finally your model should look like the following picture (please note that the actual form of the lofted shape depends on the setting of the smoothness parameter).



Finally you can add all shapes together by using multiple *Objects⇨Boolean⇨Add* operations. Now you can pick the two planar top and bottom faces of the shape, select the shape by double-clicking on it and invoke the *Objects⇨Shell* tool.



In the dialog box you can specify a *Thickness* (e.g. 0.3) and press the *Ok* button. After a couple of seconds, your model should look similar to the following picture:



Picking the two faces before entering the shell operation has the effect, that the selected faces will later be openings in the shelled structure. If no faces are selected, the structure will be shelled to form a hollow solid.

Local Coordinate Systems

Besides the Boolean operations, the ability to create local coordinate systems adds a great deal of flexibility to the modeler. In the above sections we have described how to create simple shapes which are aligned with the axes of a global fixed coordinate system.

The aim of a local coordinate system is to allow the easy definition of shapes even when they are not aligned in the global coordinate system.

The local coordinate system consists of three coordinate axes. In contrast to the global x, y and z axes, these axes are called the u, v and w axes. The local coordinate system is also known as the Working Coordinate System (WCS).

Either the local or the global coordinate system can be active at any time. “Active” here means that all the geometric data is specified in this coordinate system from now on. You may activate or deactivate the local coordinate system either from the *WCS⇨Local Coordinate System* item in the main menu, from the *WCS* context menu item, or by pressing the  item in the WCS toolbar. Each of these user interface items toggles the local coordinate system on or off.

You have now learned what a local coordinate system (WCS) is and how it can be activated, but you still need to know how to define this system in order to align its axes with the desired location.

The most common way to define the orientation of a local coordinate system is to pick points, edges or faces on the model and align the WCS with these entities.

- When a **point** is selected, the origin of the local coordinate system may be translated onto this point (*WCS⇨Align WCS with Selected Point*).
- When **three points** are selected, the u/v plane of the WCS can be aligned with the plane defined by these points (*WCS⇨Align WCS with 3 Selected Points*).. Additionally this function will move the origin of the WCS onto the first selected point.
- When an **edge** is selected, the u axis of the WCS may be oriented such that it becomes parallel to the selected edge (*WCS⇨Align WCS with Selected Edge*).
- Finally, a **planar face** could be selected to which the u/v plane of the WCS can be aligned (*WCS⇨Align WCS with Selected Face*).

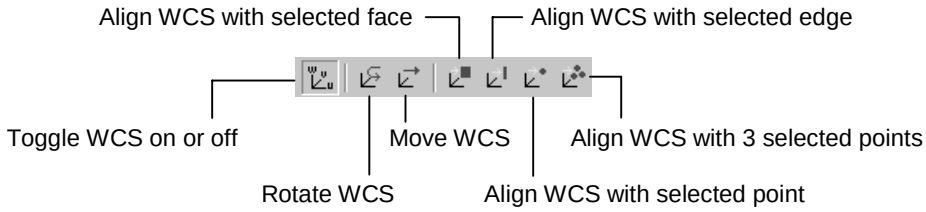
After picking a point, edge or face on the model, you could alternatively press the *W* key in order to align the WCS with the most recently picked item. Together with the available shortcut keys for the pick mode, this is the most efficient way of changing the location and orientation of the WCS.

Besides the ability to align the WCS in various ways with entities selected from the model, there are three further ways of defining the local coordinate system:

- **Define local coordinate system parameters directly**:(*WCS⇨Define Local Coordinates*) In this dialog box you may enter the origin and the orientation of the w-axis and the u-axis directly.
- **Move local coordinate system**: (*WCS⇨Move Local Coordinates*) In this dialog box you may translate the origin of the local coordinate system by a specified translation vector.
- **Rotate local coordinate system**: (*WCS⇨Rotate Local Coordinates*) By using this dialog box you can rotate the local coordinate system around one of its axes by a specified rotation angle.

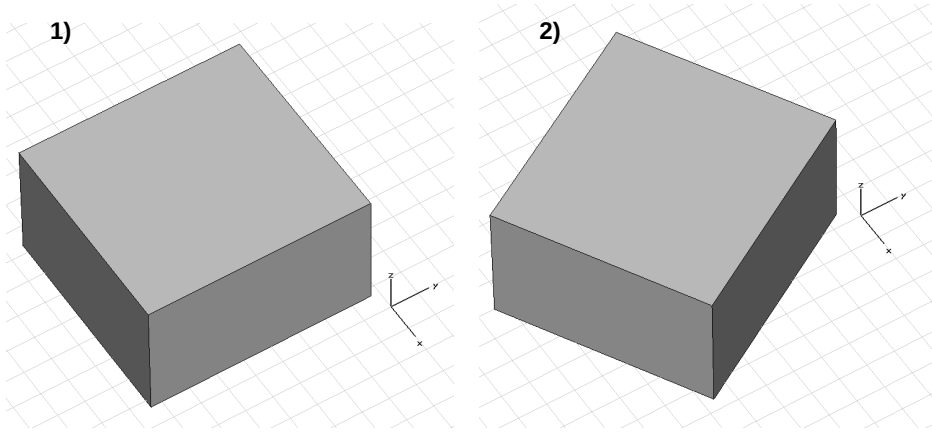
The second and third options become especially powerful when they are combined with the pick alignment options described above.

Most of the operations on the local coordinate system are also accessible from the WCS toolbar which is shown below:

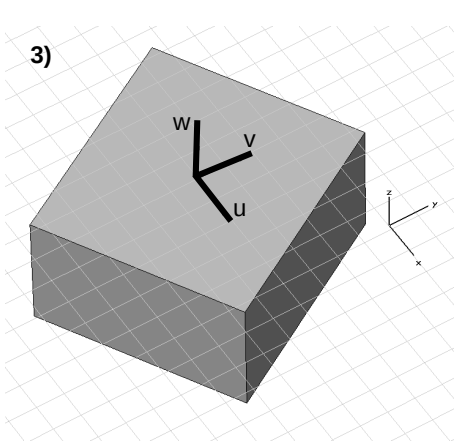


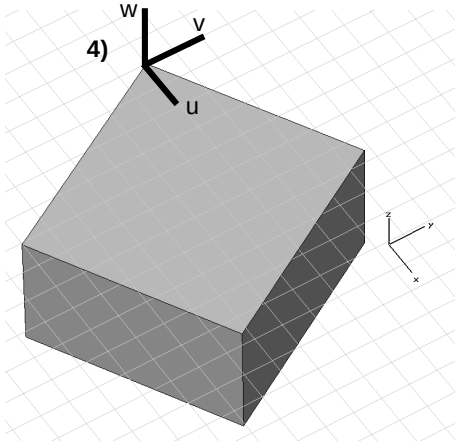
The following example should give you an idea of what can be done by efficiently using local coordinate system specifications:

The first step is to create a brick in global coordinates. Afterwards rotate the brick around the z-axis by 30 degrees using the transform dialog box:

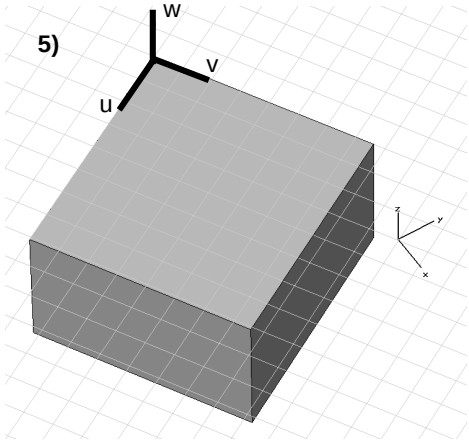


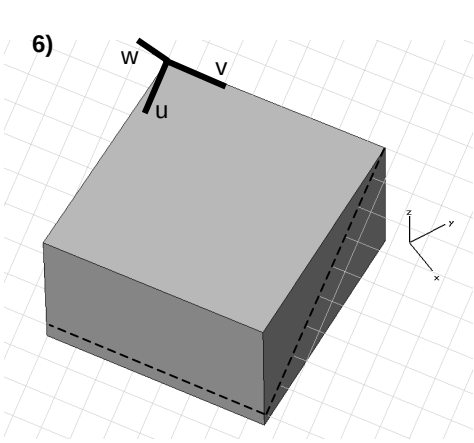
In the next step activate the local coordinate system and align it firstly with the top face of the brick, then with one of the top face's vertices:



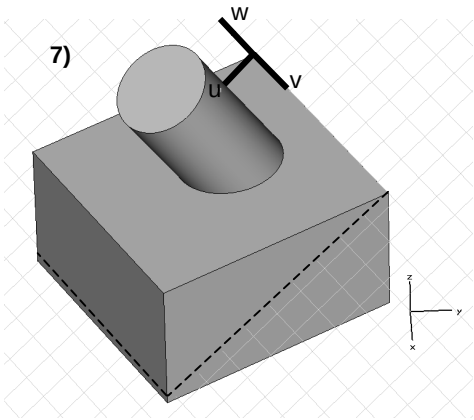


Next, align the coordinate system with one of the edges of the brick's top face and then rotate the coordinate system by thirty (30) degrees around its v-axis:





Finally create a new cylinder in the local coordinate system. As soon as you have defined the cylinder, a dialog box will open asking for the Boolean combination of the two intersecting shapes. In this dialog box choose *Add shapes* and press *Ok*:



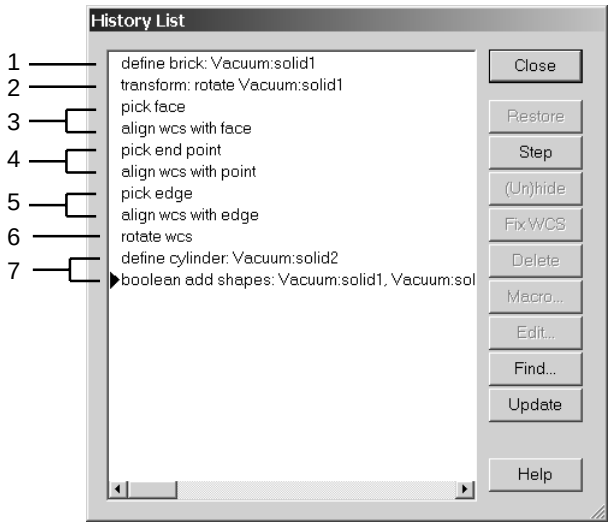
The History List

Up to now, you have created some basic structures and performed some simple geometric transformations. Mistakes during the structure generation can always be corrected by using the *Edit⇨Undo* command which simply removes the most recent construction step.

However, sometimes it may become necessary to return to a previous step in the structure generation in order to change, delete or insert some operations.

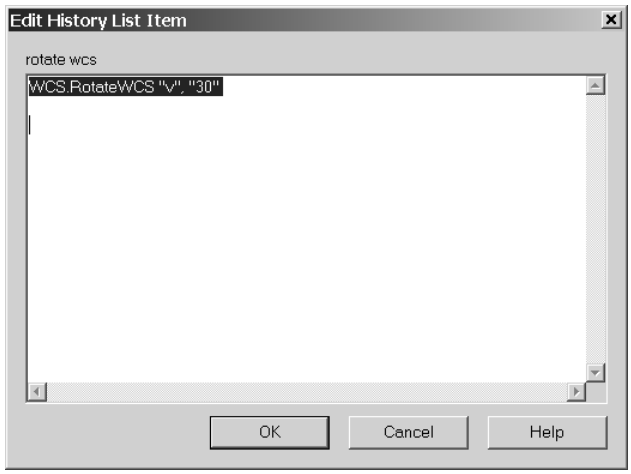
This typical task is supported by CST MICROWAVE STUDIO® via the so called History List. All relevant structural modifications are recorded in a list which can be shown by choosing *Edit⇨History List* or by pressing  in the objects toolbar.

In the following, we assume that you have created the structure consisting of a brick and a cylinder as shown in the last section covering local coordinate systems. In this case, the history list will look like the following picture:

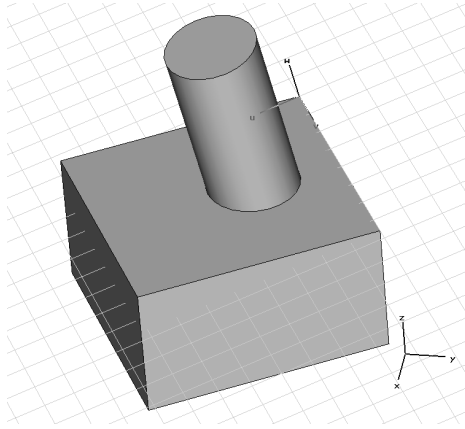


The list will show all previous operations in a chronological order. The marker ▶ indicates the current position of the structure creation in the history list. You may restore the structure creation at any step in the history list by selecting the corresponding line and pressing the *Restore* button. Pressing the *Step* button will go to the next step in the history list. You could now play around a bit with these functions.

Pressing the *Update* button will completely regenerate the structure. The *Edit* button allows you to perform changes to previous operations. In this case you should select the "rotate wcs" line and press the *Edit* button. The following dialog box will appear:



The text is actually the command, in macro language, which will perform the corresponding task. Here the first argument "v" is the rotation axis while the second argument specifies the rotation angle. You should now change the rotation angle to ten (10) degrees and press the *Ok* button. Back in the history list, you should press the *Update* button in order to regenerate the structure. Your structure should end up looking like the following picture:



In general, the history functionality allows you to perform changes to the model quickly and easily without needing to reenter the modified structure. However, some care has to be taken when history items are altered since this may result in strong topological

changes appearing in the model. This often happens when some history items are deleted or new items are inserted. In such cases, pick operations might select incorrect points, edges, or faces (sometimes because the original picked items no longer exist).

As an example of such an incident, assume that you have deleted the creation of the first brick from the history list. In this case the pick of the brick's top face in order to align the WCS with this face will obviously fail.

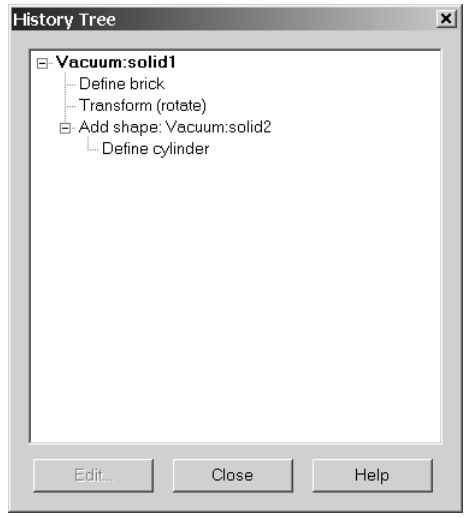
In such cases we recommend that you work through the history list from the beginning in order to properly adjust the picks when they are needed. Even in this extreme case, the work needed to change the model is much less effort than completely reentering the model. Please refer to the online documentation for details.

The History Tree

The History List, as described in the previous section, is definitely the most powerful tool to edit the structure's generation. However, in many cases only some parameters of the basic shapes or transformations need to be changed. In these cases, using the History Tree function is much more convenient.

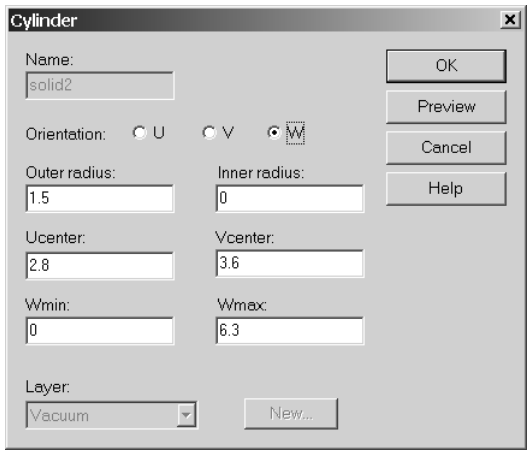
Assume that you want to change the radius of the cylinder in the previous example. You could open the history list and edit the generation of the cylinder. However, you can also select the corresponding shape by double-clicking at it and choose *Edit⇨Object Properties* or *Properties* from the context menu.

Now a dialog box (the so called History Tree) opens showing the construction of the selected shape:

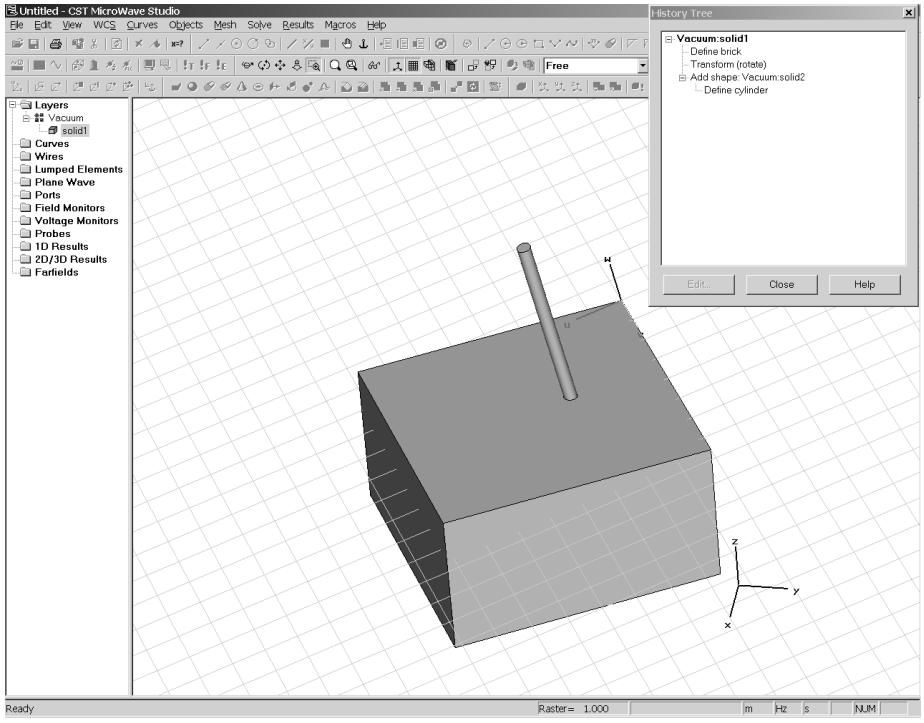


You can now simply click at the "Define cylinder" item. As soon as you have selected an editable operation from the History Tree, the corresponding structure element will be highlighted in the main view. Please note that subsequent transformations will not be considered by this highlighting functionality.

After pressing the *Edit* button in the History Tree dialog box, the well known cylinder creation dialog box opens showing the parameters of the cylinder:



You can now alter the cylinder radius and press the *Preview* button. You will now get an impression of how the structural changes will influence your model. After you are happy with the result, press the *Ok* button in order to update the structure. Finally, your screen should look as follows:



You should now play around a little with the History Tree to get a idea of which changes can be applied to the existing structure using this functionality. Please note that subsequent transformations will not be visualized by the *Preview* option in the shape dialog box, but will be applied when the model is updated.

Curve Creation

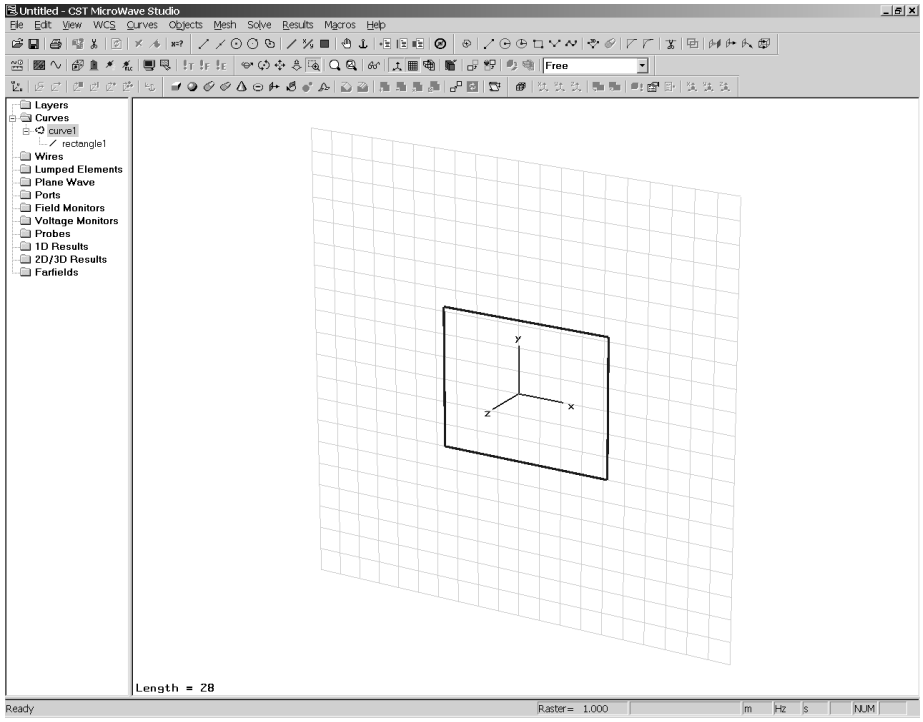
The previous chapters have shown how a model can be generated from three dimensional primitives and their modifications by using powerful operations such as blending, lofting, shelling, etc.

Another complex shape generation option is based on so called *curves*. A curve is a three dimensional line which is drawn on the drawing plane. After a curve has been defined, it can be used for more advanced modeling operations.

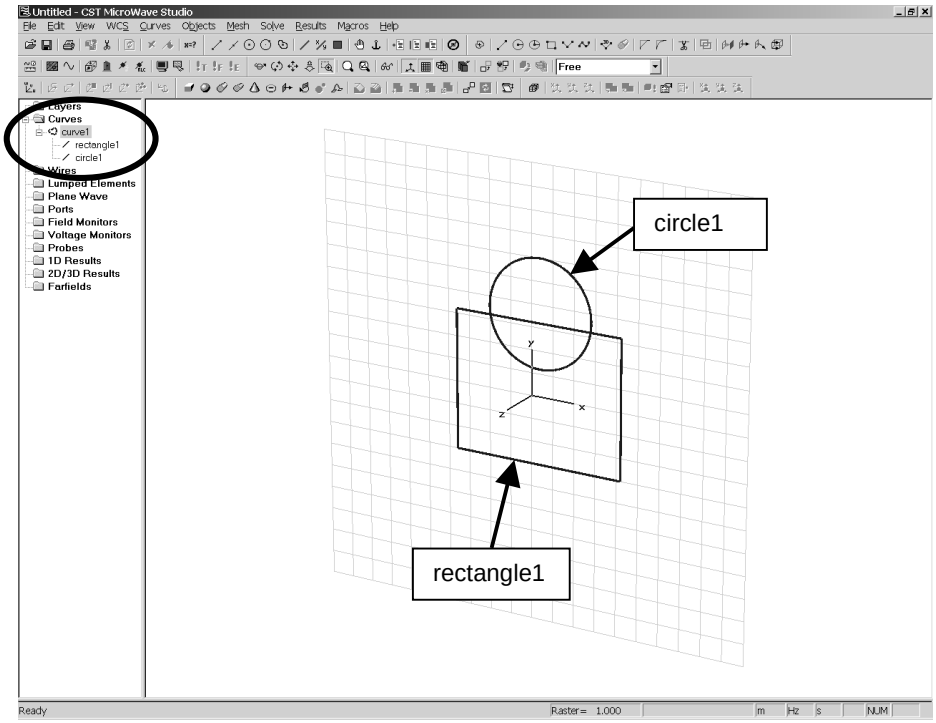
The following explanations can only give you a very basic introduction into the way curve modeling works. A detailed description of all possibilities would easily exceed the scope of this document. Please refer to the online documentation for more information.

As a simple example, you should firstly create a new curve by selecting *Curves⇨New Curve* ( ) from the main menu. This operation will create a new item called “curve1” in the navigation tree’s *Curves* folder.

Now you should activate the rectangle creation by choosing *Curves⇨Rectangle* ( ) before drawing a rectangle on the working plane. Please note that the creation of curve items works in a similar way to the construction of solid primitives. Your result should then look as follows:



In a next step you can draw a circle on the drawing plane which overlaps one of the rectangle‘s edges. The circle creation is activated by choosing *Curves⇨Circle* ( ). Afterwards, your screen should look similar to the following:

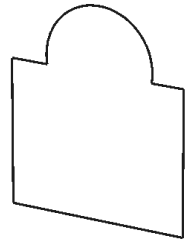


As a result of the previous steps you now have two curve items *rectangle1* and *circle1* in a curve named *curve1*. The navigation tree reflects this relationship.

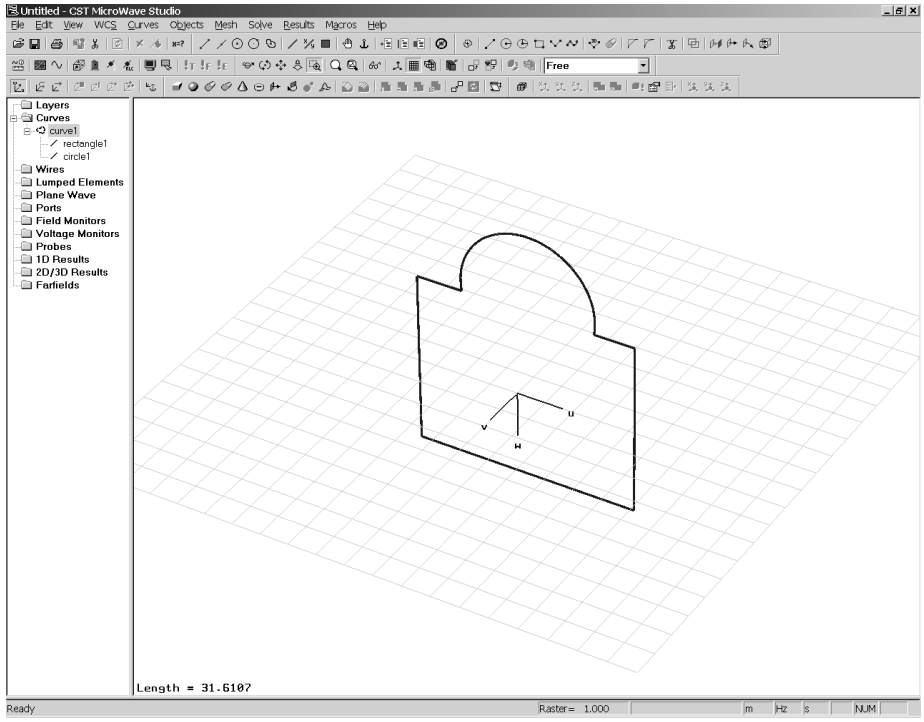
In a next step we will trim both curve items such that the resulting curve only contains the outlines of both curve items. To accomplish this task, you should first select one of the curve items, e.g. *rectangle1* (either in the navigation tree or by double-clicking on it in the main view). Afterwards you can activate the *Trim Curves* operation by choosing *Curves⇨Trim Curves* ( ) from the main menu.

Now you will be prompted to select the item to be trimmed with the rectangle. Therefore select the circle and confirm your selection by pressing the RETURN ( ) key.

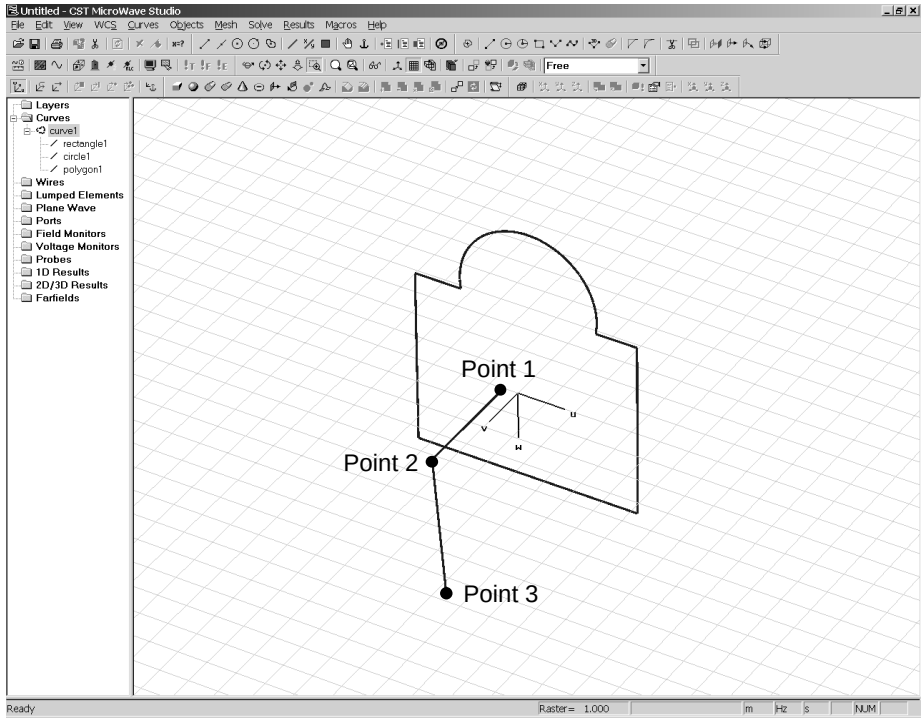
The next step will prompt you to double-click on any curve segments you wish to delete from the model. When you move the mouse across the screen, all selectable curve segments at the mouse location will be highlighted. You should now delete two segments such that the result looks similar to the following picture before you press RETURN ( ) to complete the operation.



Now you can activate the local coordinate system and rotate it around its u-axis. Your screen should now look as follows:



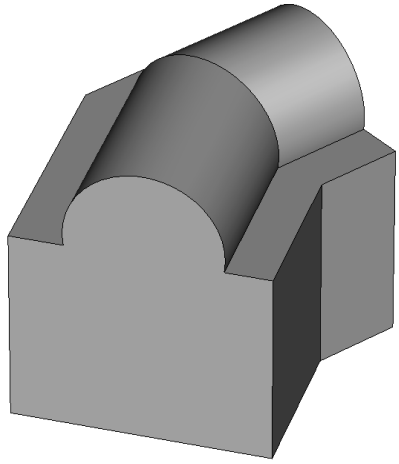
The next action is to draw an open polygon consisting of three points on the drawing plane *(Curves⇨Polygon,* ).



Based on these two disjoint curves you are now going to create a solid by using the *sweep curves* operation which can be invoked by choosing *Curves⇨Sweep Curve* ( ) from the main menu.

As soon as this operation is activated you will be prompted to select the profile curve. Therefore double-click on the curve consisting of the rectangle and the circle.

After the profile is selected you will be requested to double-click on the path curve which is given by the polygon's curve here. After closing the upcoming dialog box by pressing *Ok*, the resulting shape should look as follows:

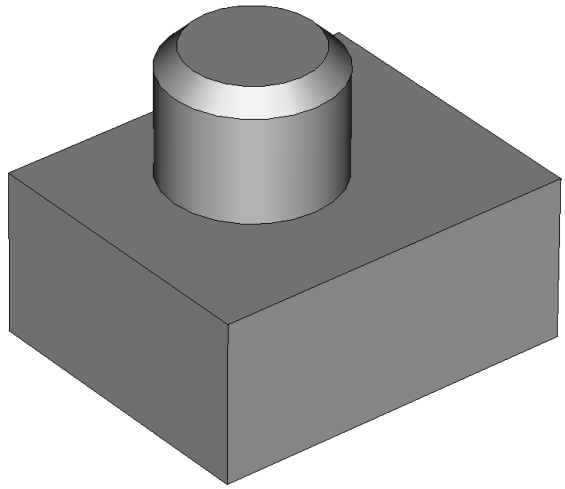


This short introduction into curve modeling can only provide a very basic understanding of these powerful structure drawing tools. You should now experiment a little bit with the curve modeling features to become more familiar with this kind of structure modeling. Please refer to the online documentation for more details.

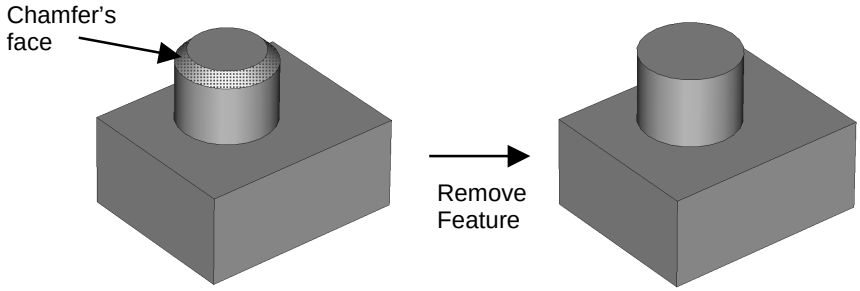
Local Modifications

So far we have focused on how to change a structure which has entirely been constructed within CST MICROWAVE STUDIO®. However, sometimes the model will consist of imported geometry for which no information about the modeling process is available.

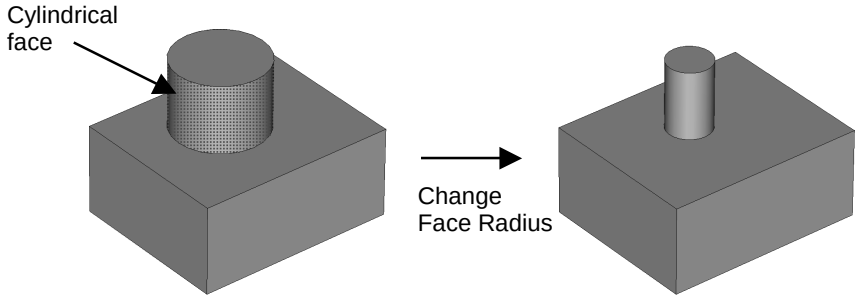
This section will illustrate that even in these cases the structure can be parameterized by using so-called *Local Modifications*. Before we can start using these advanced modeling tools you should first create a structure similar to the following (a brick united with a cylinder and a chamfer operation being applied to the cylinder's top edge):



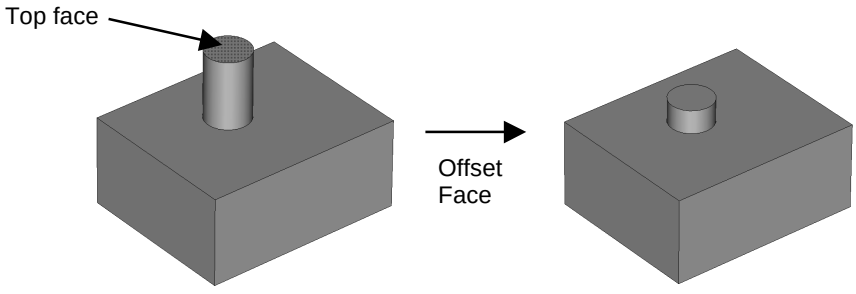
In this structure you should firstly use the pick face tools in order to select the chamfer's face. Afterwards you can invoke the *remove feature* command by selecting *Objects⇨Local Modifications⇨Remove Feature* from the main menu.



As you can see, the gap which would be produced by simply removing the face will automatically be closed by the *remove feature* operation. Afterwards you should pick the cylindrical face and select the *Objects⇨Local Modifications⇨Change Face Radius* command. Now a dialog box will open where you can specify a new radius of the cylinder.



Finally you can pick the cylinder's top face and invoke the *Objects⇨Local Modifications⇨Offset Faces* command. Specifying an offset in the dialog box will move the face while again closing any gaps:



The Local Modifications are very powerful modeling operations. However, the modifications will fail if there is no unique solution for closing the gaps. You should play a little bit with these tools in order to get an impression of what is possible.

The First Real World Application Example

The previous chapters have focused on the elementary concepts of the structure modeler. Now it is time to start on the first real world application example.

The following example shows a fairly simple S-parameter calculation. Studying this example carefully will allow you to become familiar with many standard operations which become important when performing a simulation within CST MICROWAVE STUDIO®.

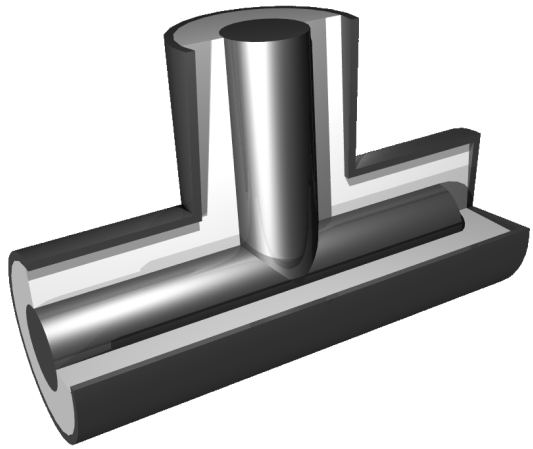
Please go through the following explanations carefully, even if you are not planning to use the software for S-parameter computations. Only a small portion of the example is specific to this particular application type while most of the considerations are quite general for all solvers and application domains.

In subsequent sections you will find some remarks concerning the differences between the typical simulation procedures for antenna calculations, eigenmode calculations and modal analysis simulations.

The following explanations will always describe the “long” way to open a particular dialog box or to launch a particular command. Whenever available, the corresponding toolbar item will be displayed next to the command description. Due to the limited space in this manual, the shortest way to activate a particular command is by either pressing a shortcut key or by activating the command from the context menu will be omitted. You should regularly open the context menu to check the available commands for the currently active mode.

The Structure

In the example you will model a simple coaxial bend with a tuning stub. You will then calculate the broadband S-parameter matrix for this structure before having a look at the electromagnetic field inside this structure at various frequencies. The following picture shows the current structure of interest (it has been sliced open purely to aid visualization). The picture has been produced by using the POV export option in CST MICROWAVE STUDIO®.



Before you start modeling the structure let's spend a few moments discussing how to describe this structure efficiently. Due to the outer conductor of the coaxial cable, the structure is sealed as if it were embedded in a perfect electric conducting block (apart, of course, from the ports). For simplification, you can thus model the problem without the outer conductor and instead embed it in a perfect conducting block.

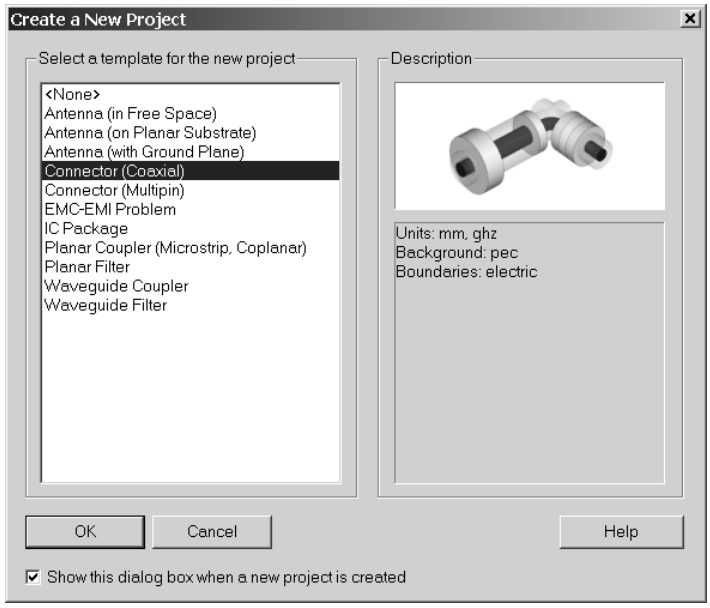
In order to simplify this procedure, CST MICROWAVE STUDIO® allows you to define the properties of the background material. Everything which you do not fill with a particular material will automatically be filled with the background material. For this structure, it would be sufficient to only model the dielectric parts and define the background material as a perfect electric conductor.

Your method of describing the structure should therefore be as follows:

1. Model the dielectric (air) cylinders
2. Model the inner conductor inside the dielectric part.

Start CST MICROWAVE STUDIO®

After starting the software and choosing to create a new project you are requested to select a template for a structure which is closest to your device of interest.

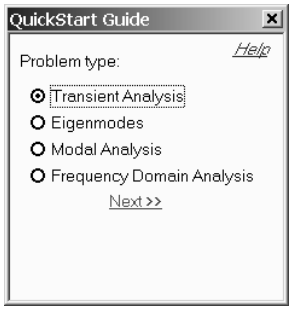


For this example you should select the coaxial connector template and press *Ok*. Now the software's default settings will be adjusted in order to simplify the simulation setup for the coaxial connector.

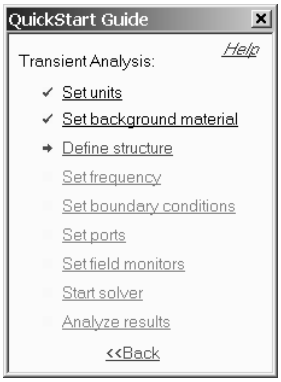
Open the Quick Start Guide

An interesting feature of the online help system is the so called *Quick Start Guide*, an electronic assistant that will guide you through your simulation. You can open this assistant by selecting *Help⇨Quick Start Guide* if it does not show up automatically.

The following dialog box should now be positioned in the upper right corner of the main view:



If your dialog box looks different, press the *Back* button in order to get the dialog above. In this dialog box you should select the *Problem Type* “Transient analysis” and press the *Next* button. The following window should appear:



The red arrow will always indicate the next step necessary for your problem definition. You may not have to process the steps in this order, but we recommend that you follow this guide at the beginning in order to ensure that all necessary steps are completed.

Please have a look at the dialog box as you follow the various steps in this example. You may close the assistant at any time. Even if you re-open the window later, it will always indicate the next required step.

Define the Units

The coaxial connector template has already made some settings for you. The defaults for this structure type are geometrical lengths in mm and frequencies in GHz. You could change these settings by entering the desired settings in the units dialog box (*Solve⇨Units*), but for this example you should just leave the settings as specified by the template.

Define the Background Material

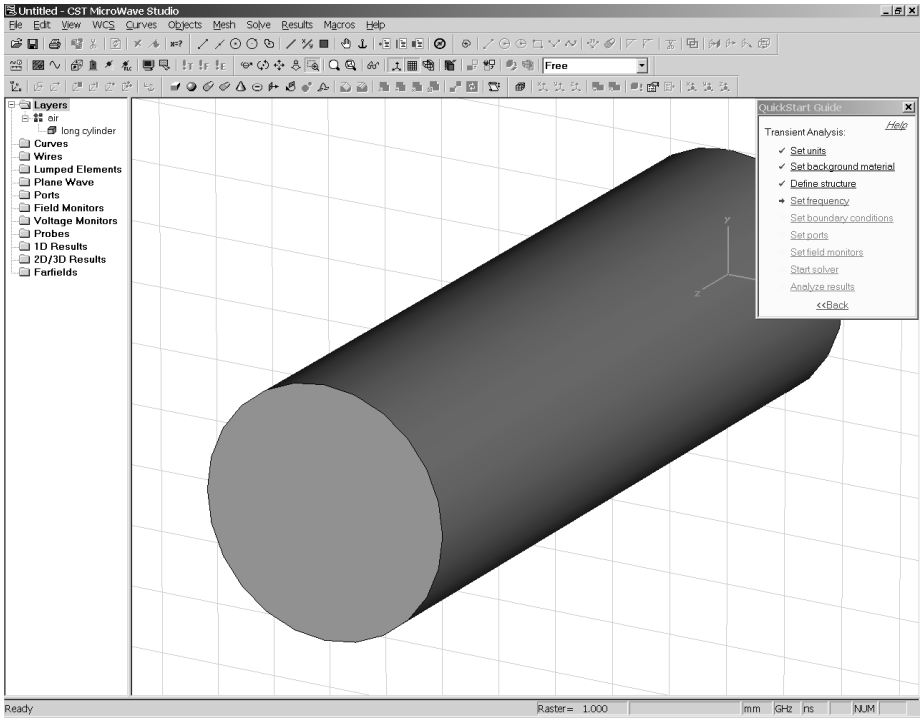
As discussed above, the structure will be described within a perfectly conducting world. The coaxial connector template has set this typical default value for you. In order to change this settings, you could make your changes in the corresponding dialog box (*Solve⇨Background Material*). For this example you don't need to change anything here.

Model the Structure

As the first step you should create a cylinder along the z-axis of the coordinate system with the following steps:

1. Select the cylinder creation tool from the main menu: *Objects⇨Basic Shapes⇨Cylinder* ().
2. Press the *Shift+TAB* key and enter the center point (0,0) in the xy-plane before pressing the *RETURN* key to store this setting.
3. Press the *TAB* key again, enter the radius 2 and press the *RETURN* key.
4. Press the *TAB* key, enter the height 12 and press the *RETURN* key.
5. Press *ESC* to create a solid cylinder (skip the definition of the inner radius).
6. In the shape dialog box enter "long cylinder" in the *Name* field.
7. You could simply select the predefined material *Vacuum* (which is very close to air) from the list in the *Layer* field. Here we are going to create a new layer "air" in order to show how the layer creation procedure works. Therefore press the *New* button beneath the layer settings.
8. In the layer creation dialog box, enter the layer name "air", select *Normal* dielectric properties (*Layer Type*) and check the material properties *Epsilon* = 1.0 and *Mue* = 1.0. Finally you may select a nice color and close the dialog box by pressing the *Ok* button.
9. Back in the cylinder creation dialog box you should press the *Ok* button to create the cylinder.

The result of all these operations should look like the picture below. You can press the *Space* bar to zoom to a full screen view.

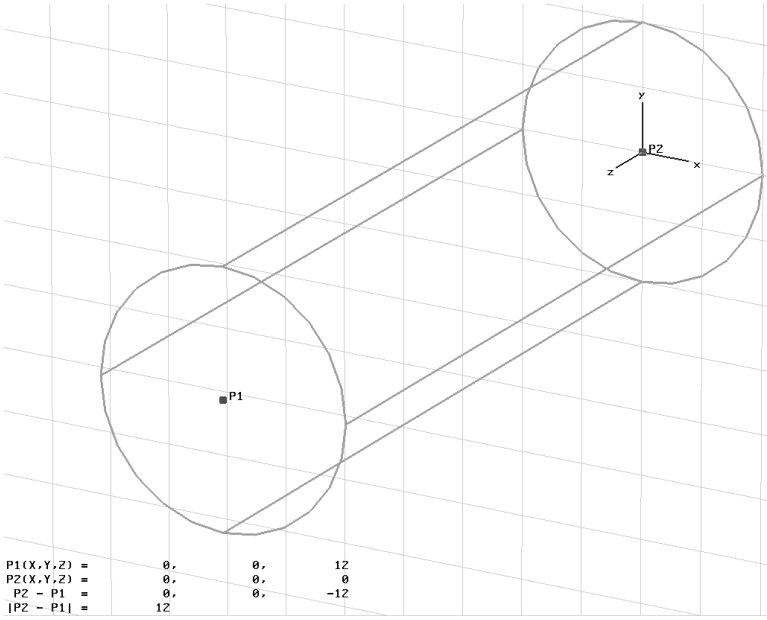


The next step is to create a second cylinder perpendicular to the first. The center of the new cylinder's base should be aligned with the center of the first one.

The first step in defining the second cylinder is to:

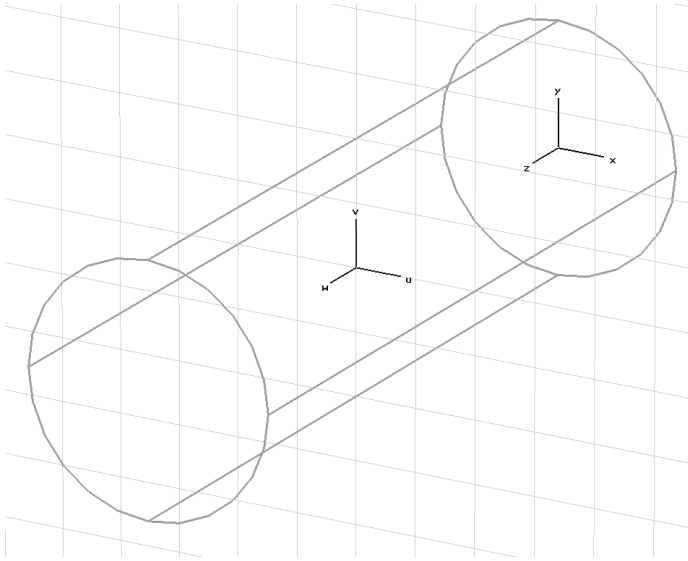
1. Select the wire frame draw mode: *View⇨ View Options* ( ) or use the shortcut Ctrl.+W.
2. Activate the "circle center" pick tool: *Objects⇨Pick⇨Pick Circle Center* ( )
3. Double-click on one of the cylinder's circular edges and a point should have been added in the center of the circle.
4. Perform steps 2 and 3 for the cylinder's other circular edge.

Now the construction should look like the following:



The next step is to replace the two selected points by a point in between the two. Therefore select : *Objects⇨Pick⇨Mean Last Two Points* from the menu.

You can now move the origin of the local coordinate system (WCS) to this point by choosing *WCS⇨Align WCS With Selected Point* ( ) from the main menu. The screen should look like this:

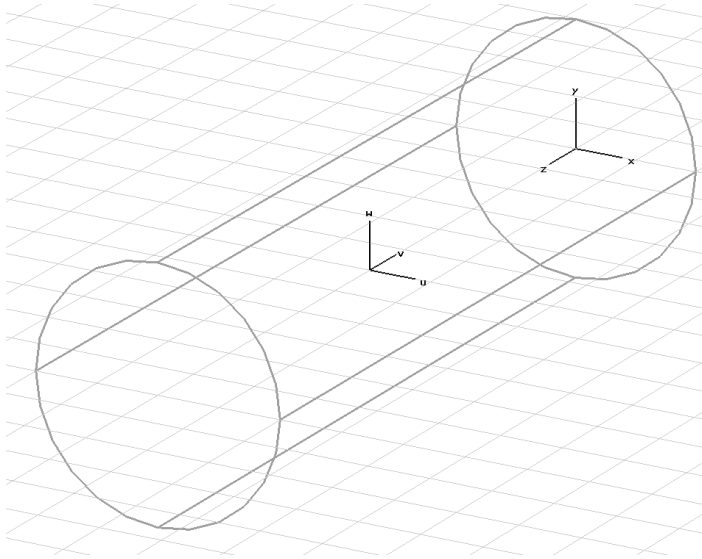


The next step is to align the w axis of the WCS with the proposed axis of the second cylinder.

1. Select *WCS⇨Rotate Local Coordinates* () from the main menu.
2. Select the *U* axis as rotation *Axis* and enter a rotation *Angle* of –90 degrees.
3. Finally press the *Ok* button.

Alternatively you could also press Shift+U to rotate the WCS by 90 degrees around its u axis. Thus pressing Shift+U three times has the same effect as the rotation by using the dialog box described above.

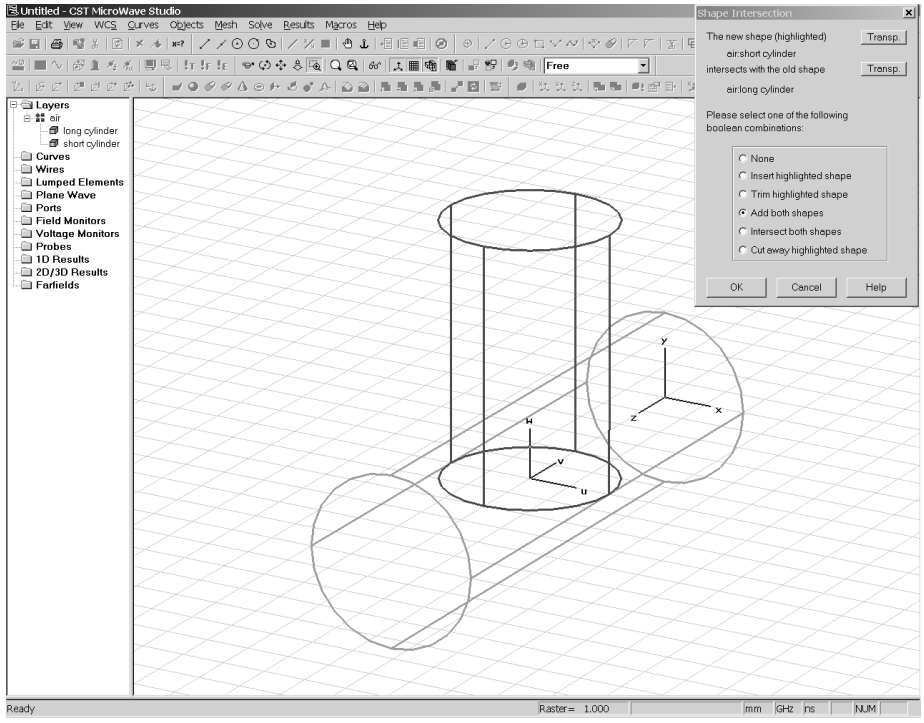
Now the structure should look like this:



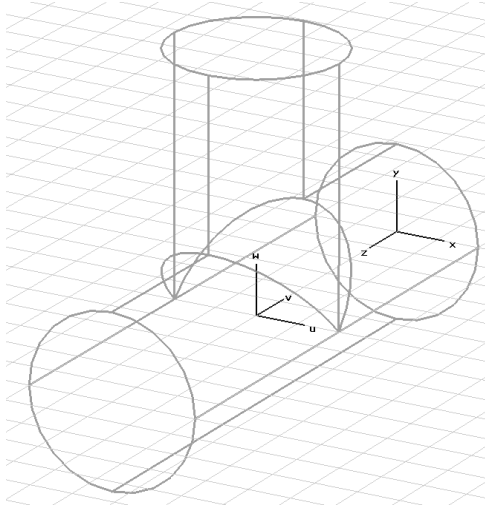
The next step is to create the second cylinder perpendicular to the first one:

1. Select the cylinder creation tool from the main menu: *Objects⇨Basic Shapes⇨Cylinder* ().
2. Press the *Shift+TAB* key and enter the center point (0,0) in the uv-plane.
3. Press the *TAB* key again and enter the radius 2.
4. Press the *TAB* key and enter the height 6.
5. Press *ESC* to create a solid cylinder.
6. In the shape dialog box enter “short cylinder” in the *Name* field.
7. Select the layer “air” from the layer list and press the *Ok* button.

Now the program will automatically detect the intersection between these two cylinders.



In the “Shape intersection” dialog box you should choose the option *Add both shapes* and press *Ok*. Finally the structure should look like this:



The creation of the dielectric air parts is finished. The following operations will now create the inner conductor inside the air.

Since the coordinate system is already aligned with the center of the second cylinder, you can start to create the first part of the conductor straight away:

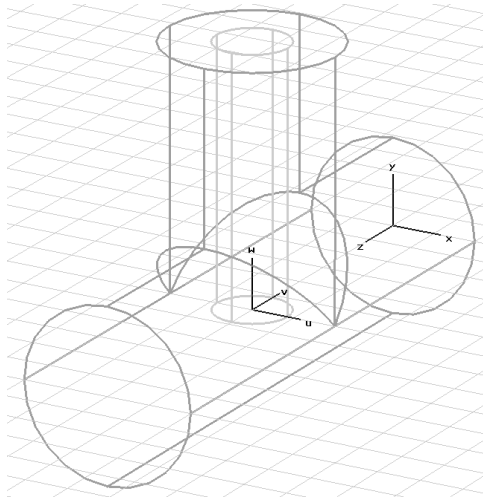
1. Select the cylinder creation tool from the main menu: *Objects⇨Basic Shapes⇨Cylinder* ( ).
2. Press the *Shift+TAB* key and enter the center point (0,0) in the uv-plane.
3. Press the *TAB* key again and enter the radius 0.86.
4. Press the *TAB* key and enter the height 6.
5. Press *ESC* to create a solid cylinder.
6. In the shape dialog box enter “short conductor” in the *Name* field.
7. Select the predefined *Layer* type PEC (perfect electric conductor) from the List of available layers and press the Ok button to create the cylinder.

At this point we should briefly discuss the intersections between shapes. In general, each point in space should be identified with one particular material. However, perfect electric conductors can be seen as a special kind of material. It is explicitly allowed that a perfect conductor is present at the same point as a dielectric material. In such cases, the perfect conductor will always be the dominant material.

On the other hand, two different dielectric shapes must not overlap each other. Accordingly, two different perfect conductors must not either. So the only exception is a dielectric material which may overlap a perfect conductor.

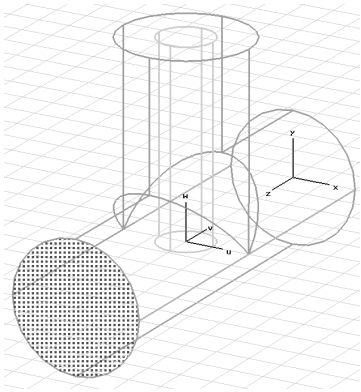
Background information*: Some structures contain extremely complex conducting parts embedded within dielectric materials. In such cases, the overall complexity of the model can be significantly reduced by NOT intersecting these two materials. This is the reason why CST MICROWAVE STUDIO® allows this exception. However, you should always make use of this feature whenever possible, even in such simple structures as this example.*

The following picture shows the structure as it should currently look:



Now you should add the second conductor. Firstly align the local coordinate system with the upper z circle of the first dielectric cylinder:

1. Select *Objects⇨Pick⇨Pick Face* ( ) from the main menu.
2. Double-click on the first cylinder’s upper z-plane. The selected face should now be highlighted:

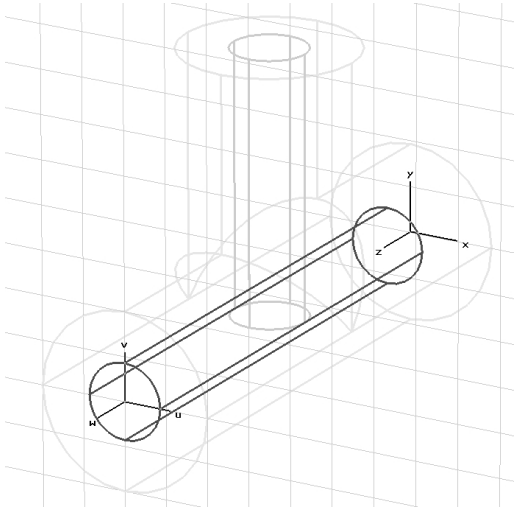


3. Now choose *WCS⇨Align WCS With Selected Face* ( ) from the main menu.

The w-axis of the local coordinate system is now aligned with the first cylinder's axis, so you can now create the second part of the conductor:

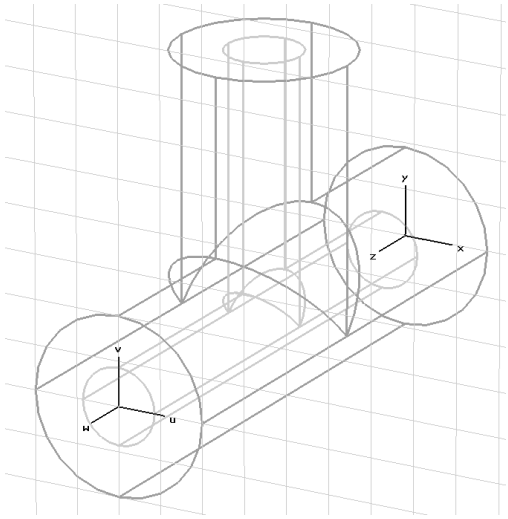
1. Select the cylinder creation tool from the main menu: *Objects⇨Basic Shapes⇨Cylinder* ( ).
2. Press the Shift+TAB key and enter the center point (0,0) in the uv-plane.
3. Press the *TAB* key again and enter the radius 0.86.
4. Press the *TAB* key and enter the height –11.
5. Press *ESC* to create a solid cylinder.
6. In the cylinder creation dialog box enter "long conductor" in the *Name* field.
7. Select the *Layer* name "PEC" from the list and press the *Ok* button.

The newly created cylinder intersects with the dielectric part as well as with the previously created PEC cylinder. The Shape intersection dialog box will request the desired relationship between the two perfect conducting cylinders:

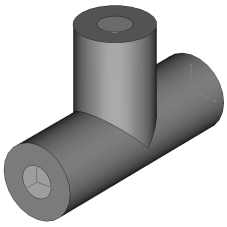


Here you should select *Add shapes* and press *Ok*.

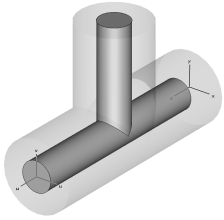
Congratulations! You have just created your first structure within CST MICROWAVE STUDIO®. The view should now look like this:



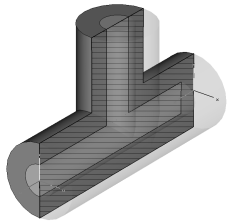
The following gallery shows some views of the structure available using different visualization options:



Shaded view
(deactivated working plane)



Shaded view,
(conductor selected)



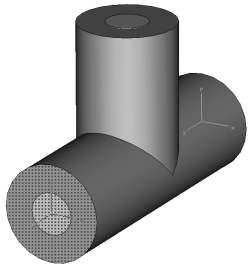
Shaded view,
(cutplane activated
View⇨*Cutting Plane*, 
Appearance of part above cutplane = transparent)

Define Ports

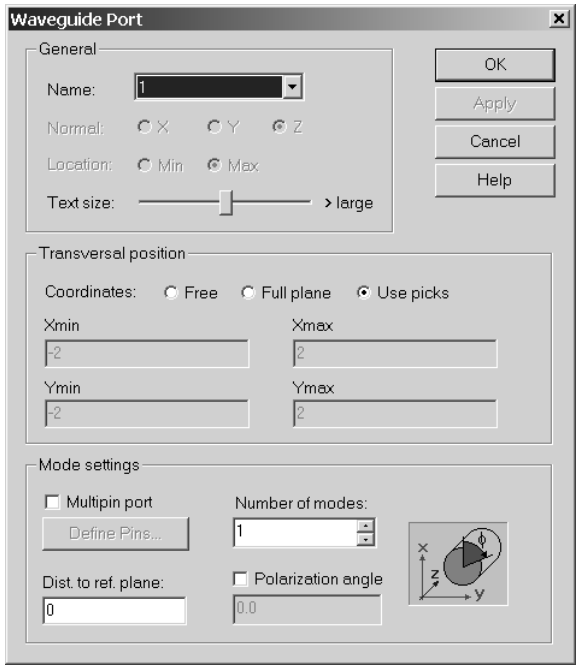
The following calculation of S-parameters requires the definition of so called ports, through which the energy enters and leaves the structure. This can be done by simply selecting the corresponding faces before entering the ports dialog box.

For the definition of the first port please perform the following steps:

1. Select *Objects⇨Pick⇨Pick Face* ( ) from the main menu.
2. Double-click on the upper z-plane of the dielectric part. The selected face will be highlighted:

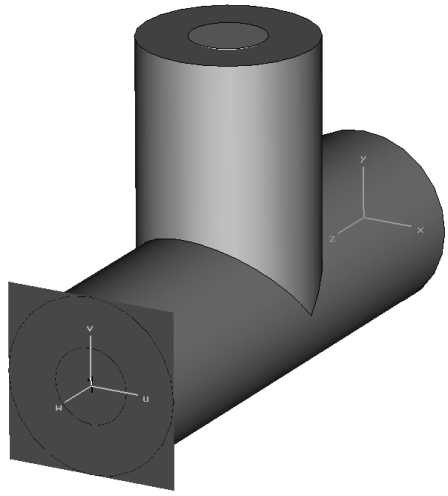


3. Open the ports dialog box by selecting *Solve⇨Waveguide Ports* ( ) from the main menu:

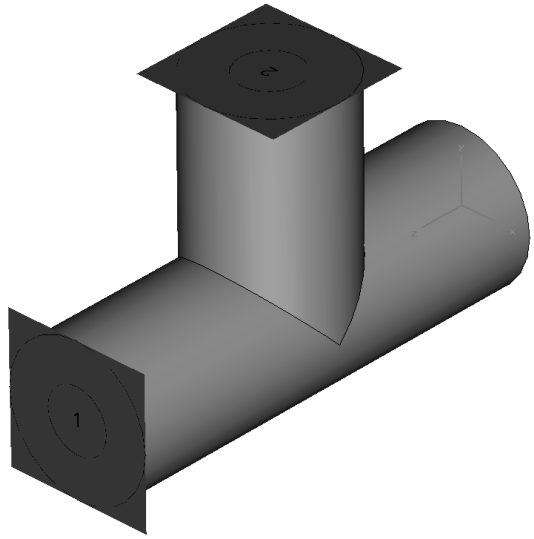


Everything is already set up correctly for the coaxial cable, so you can simply press *Ok* in this dialog box.

Once the first port has been defined, the structure should look like this:



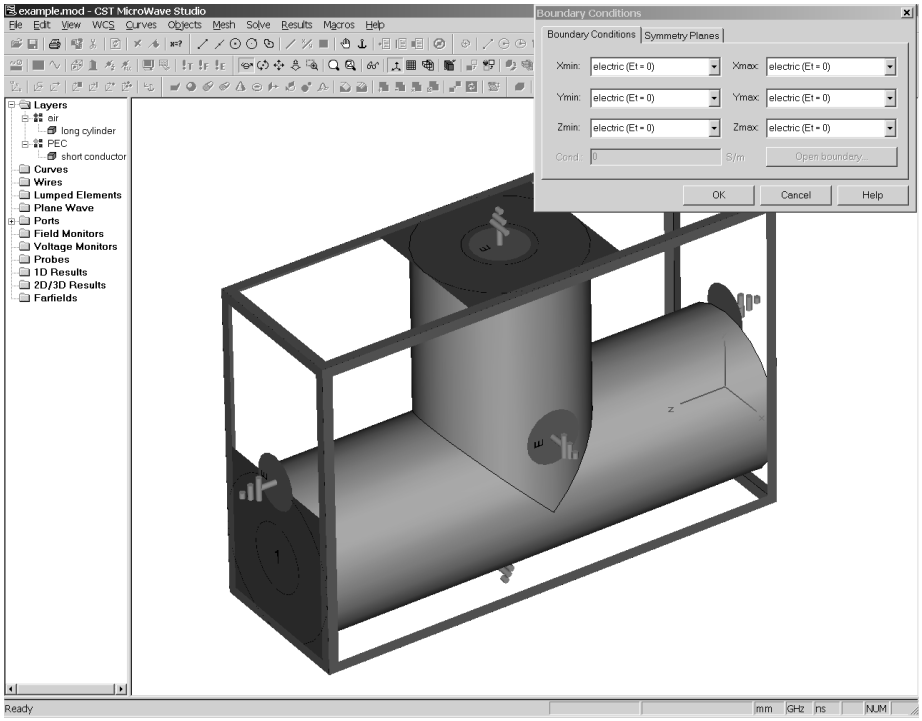
You can now define the second port in exactly the same way. The picture below shows the structure after the definition of both ports:



Define Boundary and Symmetry Conditions

The simulation of this structure will only be performed within the bounding box of the structure. You may, however, specify certain boundary conditions for each plane (Xmin/Xmax/Ymin/Ymax/Zmin/Zmax) of the bounding box.

The boundary conditions are specified in a dialog box which can be opened by choosing *Solve⇨Boundary Conditions* from the main menu.

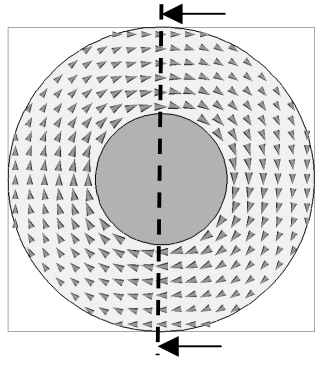


While the boundary dialog box is open, the boundary conditions will be visualized in the structure view like in the picture above.

In this simple case, the structure is completely embedded in perfect conducting material, so all the boundary planes may be specified as “electric” planes (which is the default).

In addition to these boundary planes, you can also specify so called “symmetry planes”. The specification of each symmetry plane will reduce the simulation time by a factor of two!

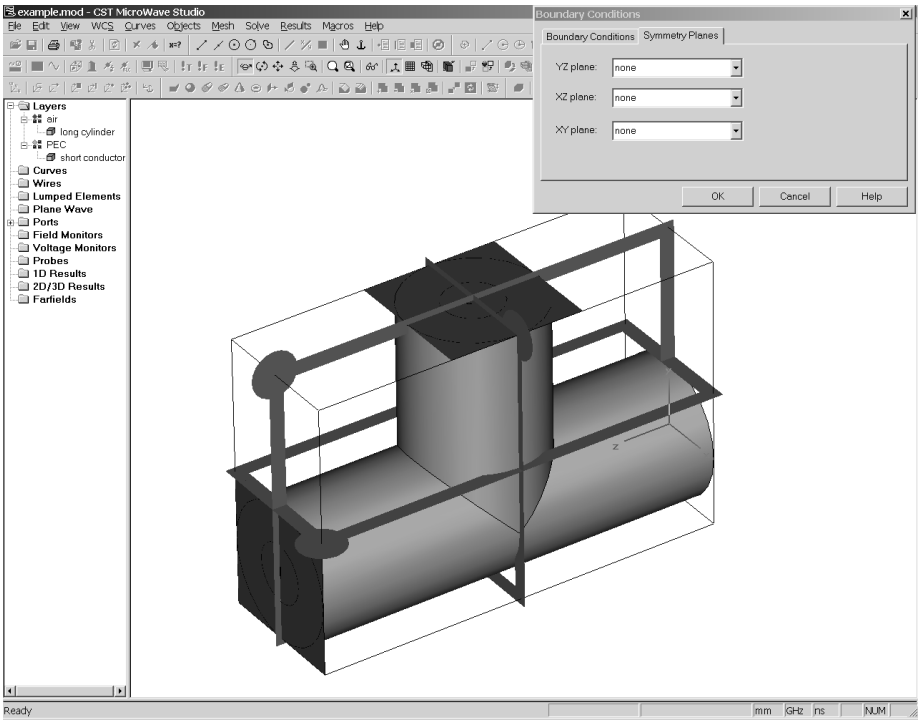
In our example, the structure is symmetric to a Y/Z plane perpendicular to the x-axis in the center of the structure. The excitation of the fields will be performed by the fundamental mode in the coaxial cable for which the magnetic field is shown below:



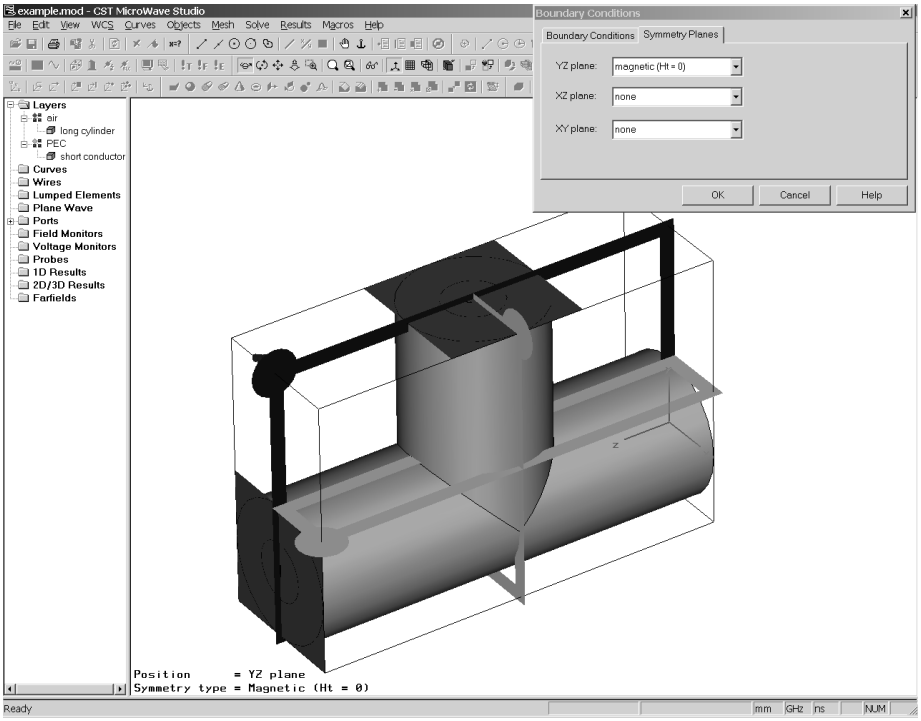
Plane of structure’s symmetry (Y/Z plane)

The magnetic field has no component tangential to the plane of the structure's symmetry (the entire field is oriented perpendicular to this plane). If you specify this plane as being a "magnetic" symmetry plane, you can advise CST MICROWAVE STUDIO® to limit the simulation to one half of the actual structure while taking the symmetry conditions into account.

In order to specify the symmetry condition you first need to click on the *Symmetry Planes* tab in the boundary conditions dialog box. Afterwards your screen should look as follows:



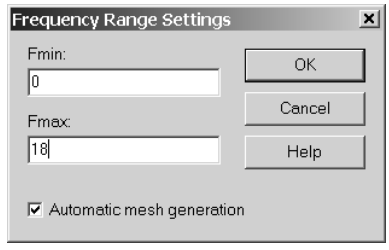
So for the yz-plane symmetry you can choose *magnetic* by either selecting the appropriate choice in the dialog box or by double-clicking at the corresponding symmetry plane visualization in the view and selecting the proper choice from the context menu. Once you have done this your screen will appear as follows:



Finally you can press *Ok* in the dialog box to store the settings. The boundary visualization will then disappear.

Define the Frequency Range

The last important setting for the simulation is the frequency range of interest. The frequency can be specified by choosing *Solve⇨Frequency* ( ) from the main menu:

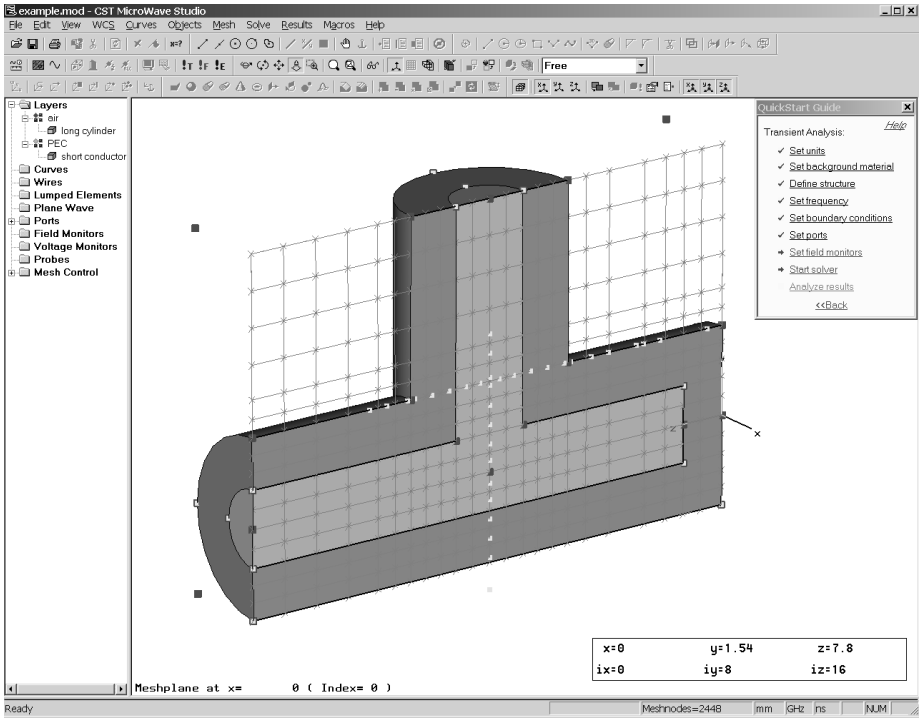


In this example you should specify a frequency range between 0 and 18 GHz. Please note that you have already set the frequency unit to GHz, so here you only need to define the absolute numbers 0 and 18 (the status bar always displays the current unit settings).

Visualize the Mesh

The mesh generation for the structure analysis will be performed fully automatically, based on an expert system. However, in some situations it may be helpful to inspect the mesh in order to improve the simulation speed by changing the parameters for the mesh generation.

The mesh can be visualized by entering the mesh mode (*Mesh⇨Mesh Mode* ()). For this structure, the mesh information will be displayed as follows:



One 2D mesh plane will always be kept in view. Because of the symmetry setting, the mesh plane only extends across one half of the structure. The orientation of the mesh plane can be modified by choosing *Mesh⇨X/Y/Z Plane Normal* (/). The plane can be moved along its normal direction by *Mesh⇨Increment/Decrement Index* (/) or by pressing the *Up* / *Down* cursor keys.

The red points in the model are critical points (so called fixpoints) where the expert system finds it necessary to have mesh lines at this location. In addition to this the yellow dots show points where the automatic mesh generation finds the need for improving the mesh density.

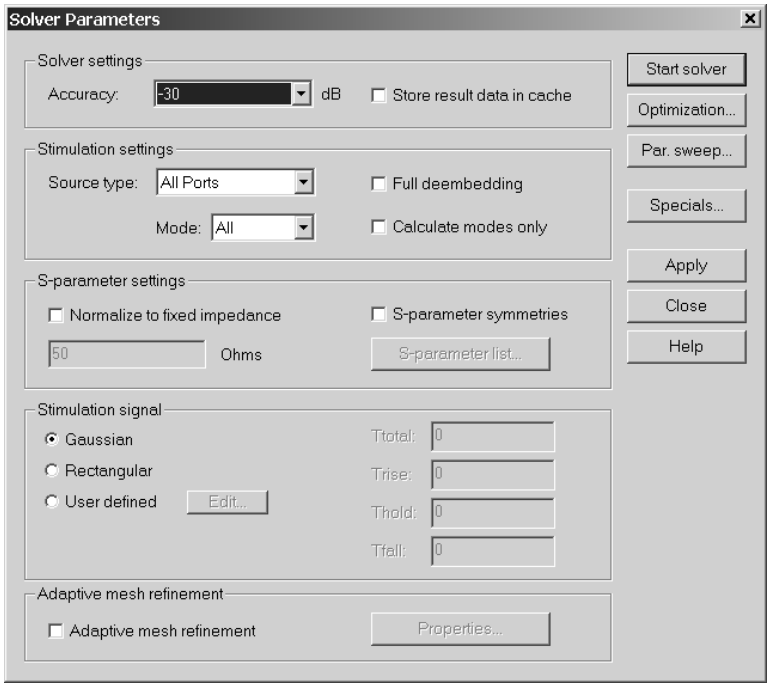
In most cases the automatic mesh generation will produce a sufficient mesh but we do recommend that you later spend some time on the mesh generation procedures in the online documentation when you feel familiar with the standard simulation procedure.

You should now leave the mesh inspection mode by again toggling: *Mesh⇨Mesh Mode* ().

Start the Simulation

After you have defined all necessary parameters you are ready to start your first simulation.

The simulation is started from within the transient solver control dialog box: *Solve⇨Transient Solver* (!T).



In this dialog box you can specify which column of the S-matrix should be calculated. Therefore select the *Source type* port for which the couplings to all other ports will then be calculated during a single simulation run. In our example, setting the *Source Type* to *Port 1*, the S-parameters S11, S21 will be calculated. Setting the *Source Type* to *Port 2* will calculate S22 and S12.

In some cases where the full S-matrix is needed, you may also set the *Source Type* to *All Ports* which implies that one calculation run will be performed for each port. However for loss free, two port structures (like the structure investigated here), the second calculation run will not be performed since all S-parameters can be calculated from one run using analytic properties of the S-matrix.

In this case you should compute the full S-matrix and leave *All Ports* as your *Source Type* setting.

The S-parameters which are calculated will always be automatically normalized to the port impedance (which will be calculated automatically). In this case the port impedance will be approximately

$$138 \cdot \log(\frac{2}{0.86}) = 50.58 \text{ Ohms}$$

for the coaxial lines with the specified dimensions and dielectric constants. However, sometimes you need the S-parameters for a fixed normalization impedance (e.g. 50 Ohms), therefore check the *Normalize to fixed impedance* button and specify the desired normalization impedance in the entry field below. In this example you should calculate the S-parameters for a reference impedance of 50 Ohms. Please note that the re-normalization of the S-parameters is only possible when all S-parameters are calculated (*Source Type = All Ports*).

While the solution accuracy mainly depends on the discretization of the structure and can be improved by refining the mesh, the truncation error introduces a second error source in transient simulations.

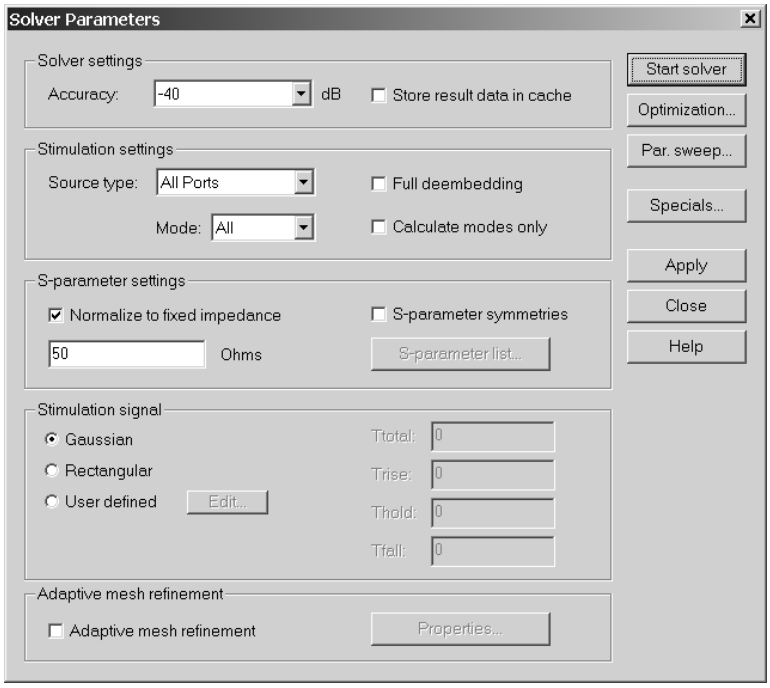
The transformation of the time signals into the frequency domain in order to obtain the S-parameters (as a frequency domain result), requires the signals have sufficiently decayed to zero. Otherwise a truncation error will occur which causes ripples on the S-parameter curves.

CST MICROWAVE STUDIO® features an automatic solver control which stops the transient analysis when the energy inside the device, and thus the time signals at the ports, have sufficiently decayed to zero. The ratio between the maximum energy inside the structure at any time and the limit at which the simulation will be stopped is specified the *Accuracy* field (in dB).

In this example we will limit the maximum truncation error down to one percent for which you should set the solver *Accuracy* to –40 dB.

Please note that the solver will excite the structure with a frequency independent power of 1 W. All field data obtained during the simulation will be normalized to this input power level.

After setting all these parameters, the dialog box should look like this:



You can now start the simulation procedure by pressing the *Start solver* button. A few progress bars will appear through which you will be kept up to date with the solvers progress:

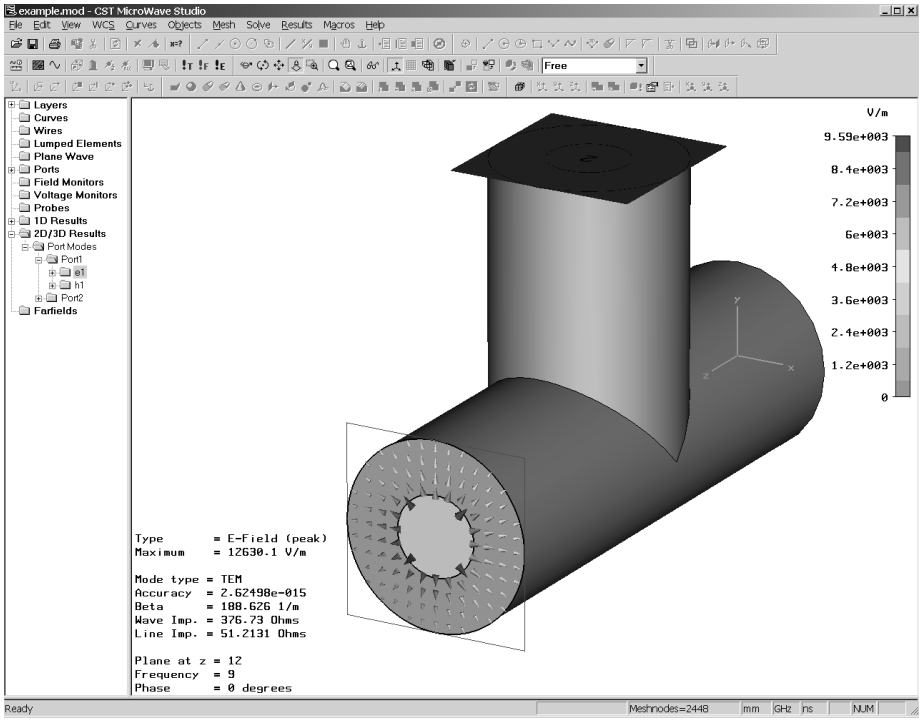
1. **Checking model:** During this step, your input model will be checked for errors such as invalid overlapping materials, etc.
2. **Calculating matrix and dual matrix:** During these steps the system of equations, which will subsequently be solved, will be set up.
3. **Calculating the port modes:** In this step the solver calculates the port mode field distributions and propagation characteristics as well as the port impedances'. This information will be used later in the time domain analysis of the structure.
4. **Processing excitation:** During this stage an input signal will be fed into the stimulation port. The solver then calculates the resulting field distribution inside the structure as well as the mode amplitudes at all other ports. From this information the frequency dependent S-parameters are calculated in a second step using a Fourier Transformation.
5. **Transient analysis:** After the excitation pulse has vanished, there is still electromagnetic field energy inside the structure. The solver then continues to calculate the field distribution and the S-parameters until the energy inside the structure has decayed to under a certain limit (specified by the *Accuracy* setting in the solver dialog box).

For this simple structure, the entire analysis only takes a few seconds to complete.

Analyze the Port Modes

After the solver has completed the port mode calculation you can have a look at the results (even if the transient analysis is still running).

In order to visualize a particular port mode you first have to choose the solution from the navigation tree. You can find the mode in port 1 from: *NT* (stands for the navigation tree*)⇨2D/3D Results⇨Port Modes⇨Port1*. If you open this subfolder you may select either the electric or the magnetic mode field. By selecting the folder for the electric field of the first mode *e1*, the port mode and its relevant parameters will be displayed in the main view:

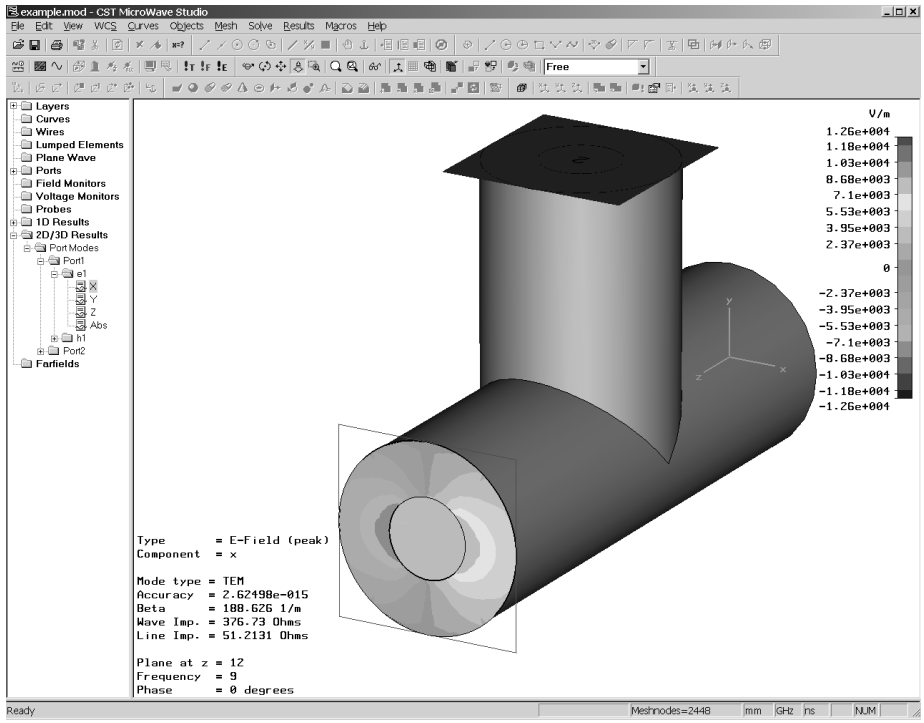


Besides the information about the type of mode (here TEM), you will also find the propagation constant (beta) at the central frequency. Additionally the port impedance is calculated automatically (line impedance).

You will find that the calculated result for the port impedance of 51.21 Ohms agrees quite well with the analytical solution of 50.58 Ohms. The small difference of 1.25% is caused by the discretization of the structure. By increasing the mesh density, the agreement between simulation and theoretical value will get even better. However, the automatic mesh generation always tries to choose a mesh which provides a good tradeoff between accuracy and simulation speed.

The number and size of arrows can be adjusted in the dialog box which can be opened by choosing *Results⇨Vector Plot* (or *Plot Properties* in the context menu).

Furthermore you may perform a scalar field visualization by opening the *e1* folder and selecting one of its field components (e.g. *X*). The selected field component will be visualized as a contour plot by default:

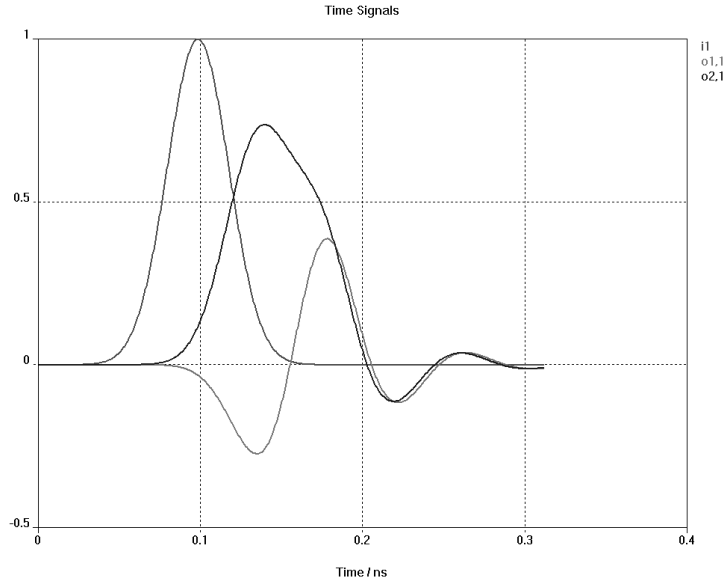


You may change the type of the scalar visualization by selecting a different visualization option in the corresponding dialog box: *Results⇨Scalar Plot* (or *Plot Properties* in the context menu).

You should play around a bit here to become familiar with the different visualization options before you proceed with the next step.

Analyze the S-Parameters

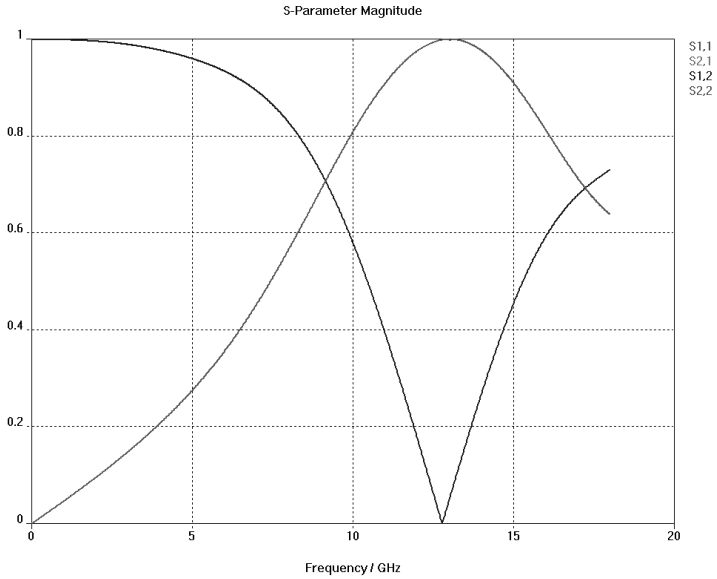
After the simulation has finished you should always have a look at the time signals of the port modes. You can visualize these signals by choosing *NT*(navigation tree*)⇨1D Results⇨Port signals*. After selecting this folder the following plot should appear:



The input signals are named with reference to their corresponding port: i1 (for port 1), i2 and so on. The output signals are similarly named by “o1,1”, “o2,1”, etc. where the number following the comma indicates the corresponding excitation port.

In order to obtain a sufficiently smooth frequency spectrum of the S-parameters, it is important that all time signals have decayed to zero before the simulation stops. The simulation will stop automatically when this criteria has been sufficiently met.

The most interesting results are, of course, the S-parameters themselves: You may obtain a visualization of these parameters in linear scale by choosing *NT⇨1D Results⇨|S| linear*.

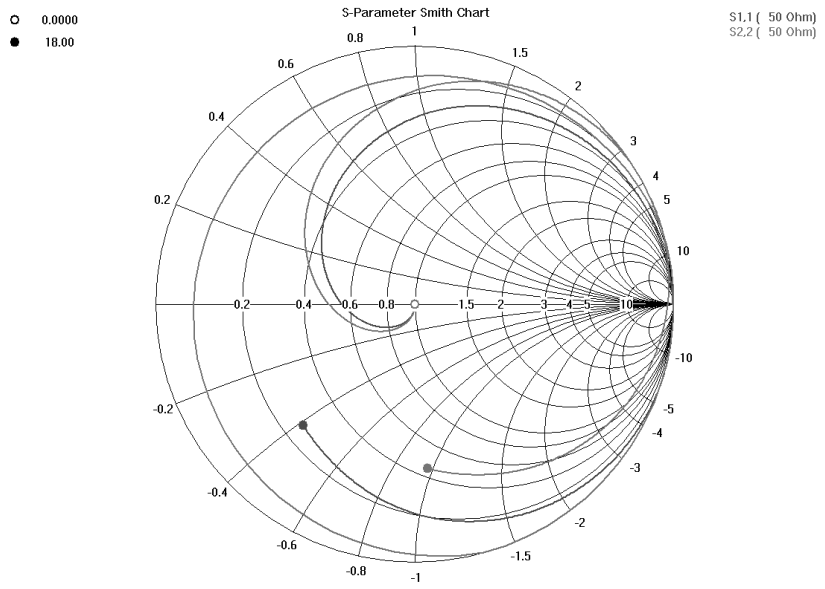


You can change the axis scaling by selecting *Results⇨1D Plot Options⇨Plot Properties* from the main menu (or the context menu). In addition you can display and hide an axis marker by toggling *Results⇨1D Plot Options⇨Show Axis Marker*. The marker can be moved either with the cursor keys (*Left* or *Right*) or by picking and dragging it with the mouse.

The marker helps to determine the minimum of the transmission (S1,2 or S2,1) being at about 12.79 GHz.

In the same way as above, the S-parameters can be visualized in logarithmic scale (dB) by choosing *NT⇨1D Results⇨|S| dB*. The phase can be visualized by choosing *NT⇨1D Results⇨arg(S)*.

Furthermore the S-parameters can be visualized in a Smith Chart (*NT⇨1D Results⇨Smith Chart*).

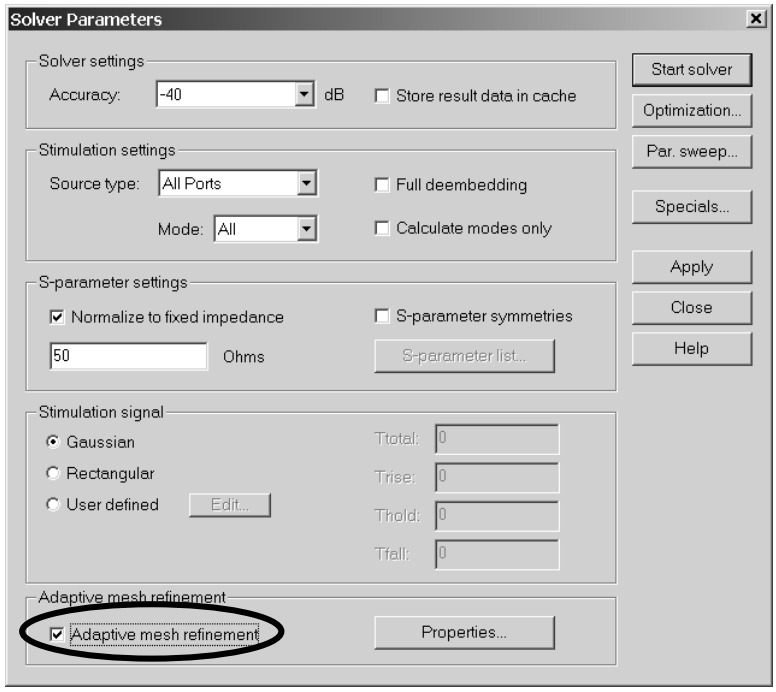


In this plot you can add markers to the curves by simply double-clicking on the corresponding positions on the curves. You may delete these markers in a properties dialog box: *Results⇨1D Plot Options⇨Plot Properties* (or *Plot Properties* from the context menu).

Adaptive Mesh Refinement

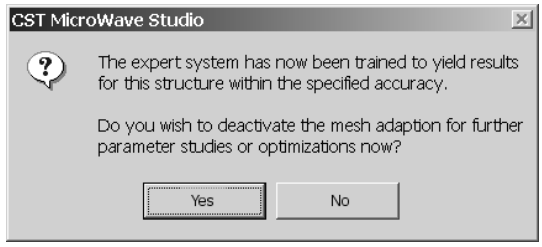
As already mentioned above, the mesh resolution does influence the results. The expert system based approach analyzes the geometry and tries to identify the parts which are critical to the electromagnetic behavior of the device. The mesh will then automatically be refined in these regions. However, due to the complexity of electromagnetic problems this approach may not be able to determine all critical domains in the structure. To circumvent this problem, CST MICROWAVE STUDIO® features an adaptive mesh refinement which uses the results of a previous solver run in order to improve the expert system's setting.

The adaptive mesh refinement can be activated by checking the corresponding option in the solver control dialog box.



When the solver is then started, several mesh refinement passes are performed automatically until the S-parameters do not change significantly between two subsequent passes.

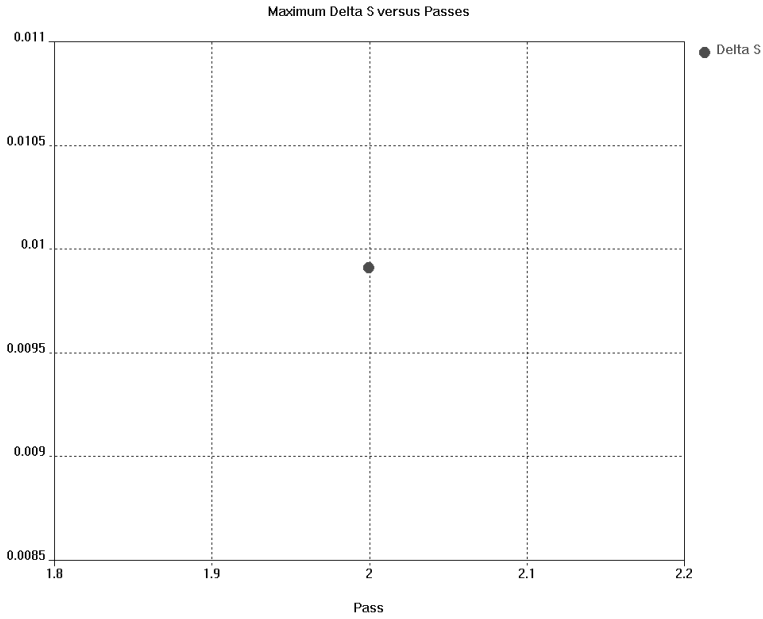
After two passes have been completed, the following dialog box will appear:



Since the automatic mesh adaption procedure has successfully adjusted the expert system's settings in order to meet the given accuracy level (2% by default), you may now switch off the adaptive refinement procedure for subsequent calculations. The expert system will now apply the determined rules to the structure even if it is modified afterwards. This extremely powerful approach allows you to run the mesh adaption procedure just once and then perform parametric studies or optimizations on the structure without the need for further mesh refinement passes.

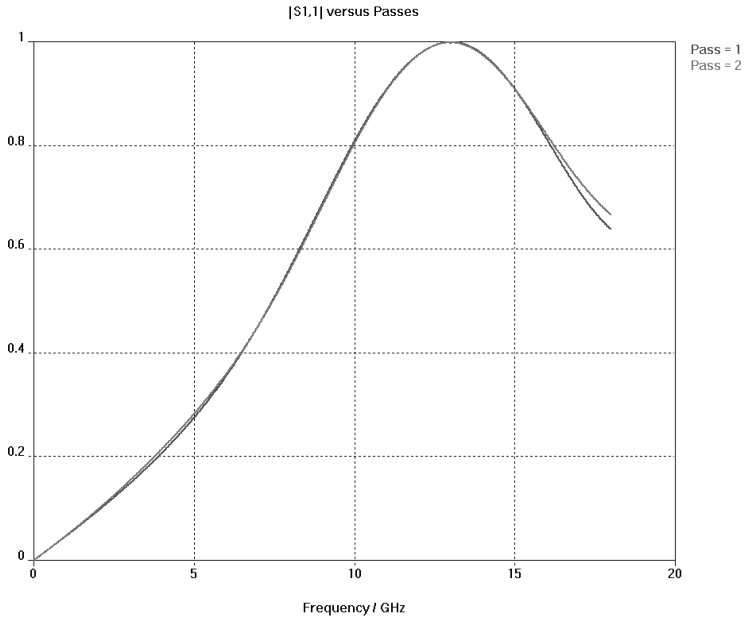
You should now confirm deactivation of the mesh adaption by pressing the *Yes* button.

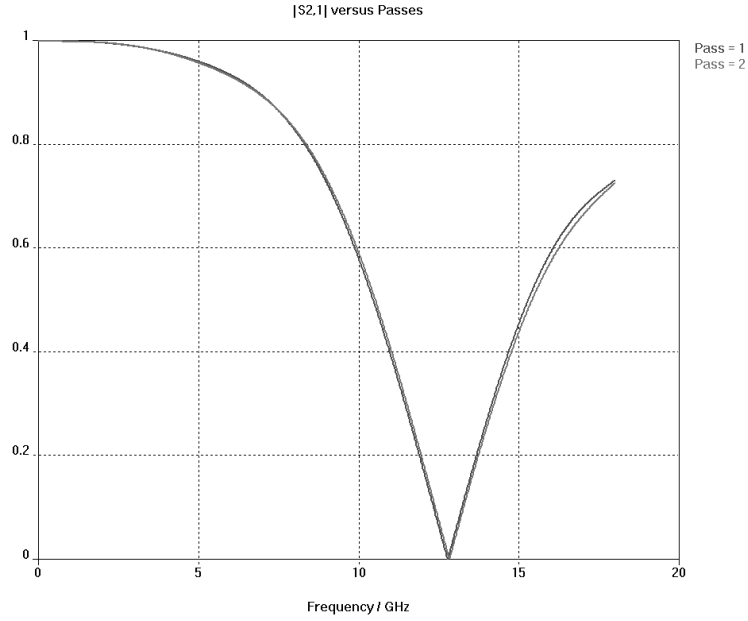
When the analysis has finished, the S-parameters and fields show the converged result. The progress of the mesh refinement can be checked by looking at the *NT⇨1D Results⇨Adaptive Meshing* folder. This folder contains a curve which displays the maximum difference between two S-parameter results belonging to subsequent passes. This curve can be shown by selecting *NT⇨1D Results⇨Adaptive Meshing⇨Delta S*.



Since the mesh adaption only required two passes for this example, the delta S curve consists of a single data point only. The result shows that the maximum difference of the S parameters from both runs is below 1% for the whole frequency range. The mesh adaption stops automatically when the difference is below 2%. This limit can be changed in the adaptive mesh refinement *Properties* (accessible from within the solver dialog box).

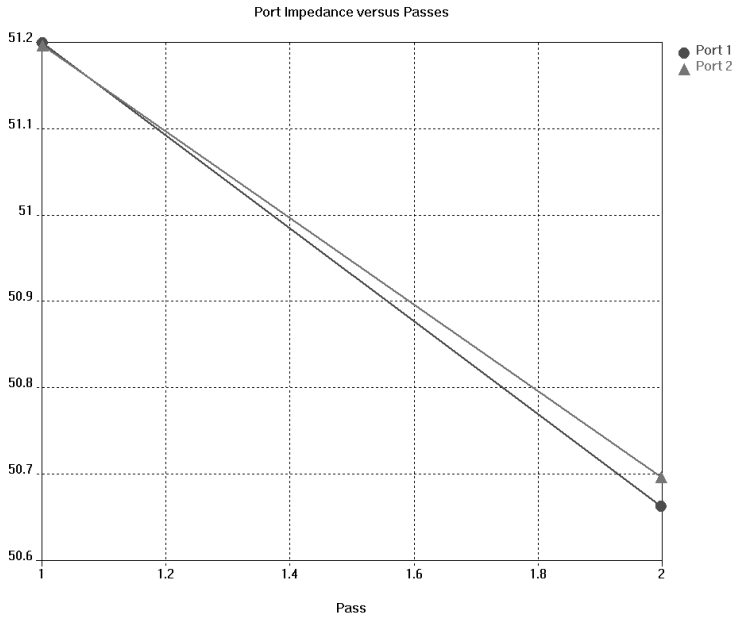
Additionally, the convergence of the S-parameter results can be visualized by selecting *NT⇨1D Results⇨Adaptive Meshing⇨S Magnitude⇨|S1,1| versus Passes,* and *NT⇨1D Results⇨Adaptive Meshing⇨S Magnitude⇨|S2,1| versus Passes*, respectively.





It can be seen that the expert system based meshing provides a good mesh for this structure. The convergence of the S-parameters shows only small variations from the results obtained by using the expert system generated mesh to the converged solution.

As another result from the adaptive meshing procedure you can visualize how the port impedance changes versus the number of adaptive passes by selecting *NT⇨1D Results⇨Adaptive Meshing⇨Port Impedance.*



This result shows that the port impedances improve to a difference of less than 0.25% to the analytical result (50.58 Ohms).

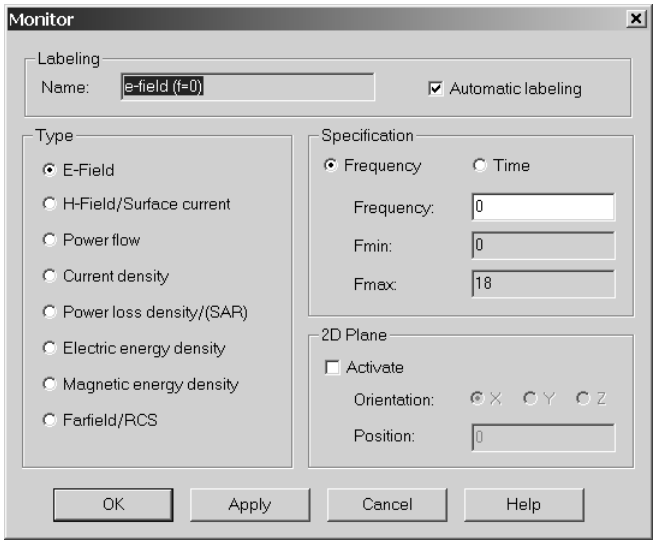
In practice it often proves judicious to activate the adaptive mesh refinement in order to ensure convergence of the results. (This might not be necessary for structures with which you are already familiar where you should use your experience to refine the automatic mesh.)

Analyze the Electromagnetic Field at Various Frequencies

In order to understand the behavior of an electromagnetic device, it is often useful to get an insight into the electromagnetic field distribution. In this example it may be interesting to see the difference between the fields at frequencies where the transmission is large or small.

The fields can be recorded at arbitrary frequencies during a simulation. However, it is not possible to store the field patterns at all available frequencies as this would require a tremendous amount of memory space. You should, therefore, define some frequency points at which the solver will record the fields during a subsequent analysis. These field samples are called monitors.

Monitors can be defined in a dialog box which opens after choosing *Solve⇨Monitors* () from the main menu.



After selecting the proper *Type* for the monitor you may specify its frequency in the *Frequency* field. Pressing *Apply* stores the monitor whilst leaving the dialog box open. All frequencies are specified in the frequency unit which has previously been set to GHz.

For this analysis you should add the following monitors:

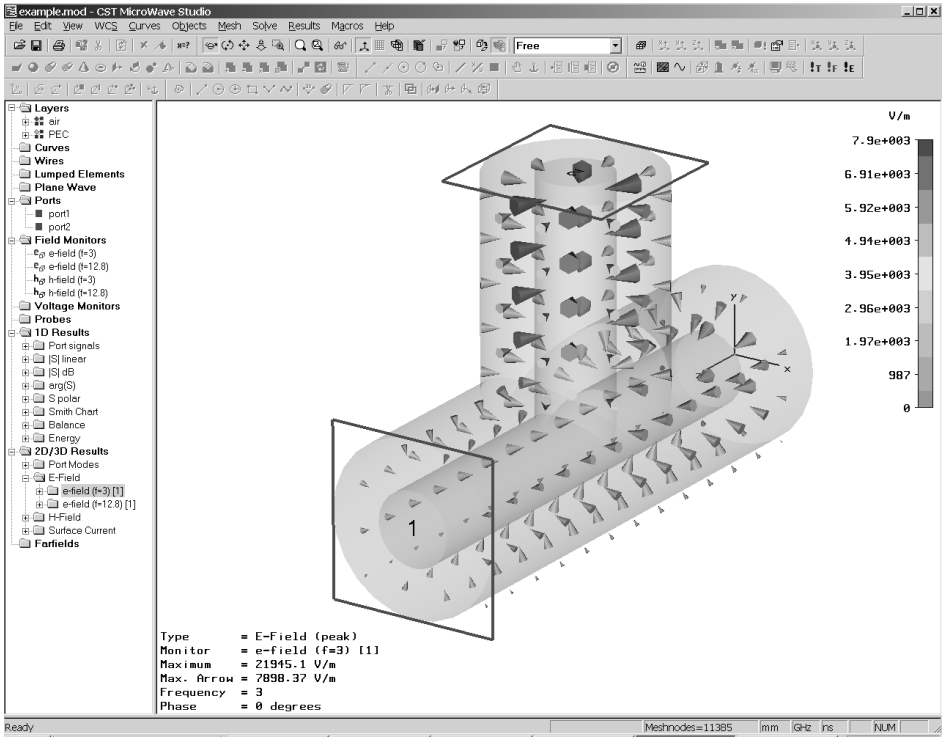
Field type	Frequency / GHz
E-Field	3
E-Field	12.8
H-Field	3
H-Field	12.8

All defined monitors will be listed in the *NT*(navigation tree)⇨*Monitors* folder. Within this folder you may select a particular monitor in order to show its parameters in the main view.

When the simulation has finished you can visualize the recorded field by choosing the corresponding item from the navigation tree. The monitor results can be found in the *NT⇨2D/3D Results* folder. The results are ordered according to their physical quantity (*E-Field/H-Field/Currents/Power flow*).

Note: Since you have specified a full S-matrix calculation, two simulation runs would generally be required. For each of these runs the field will be recorded as specified in the monitors and the results will be presented in the navigation tree, giving the corresponding stimulation port in brackets. However, in this loss free example the second run is not necessary and so you will find that the monitor data is not available. You can advise the solver to perform both simulation runs even if they are not necessary for the S-parameter calculation by deselecting the option *Consider two port reciprocity* under the *Solver* tab in the solver's *Specials* dialog box.

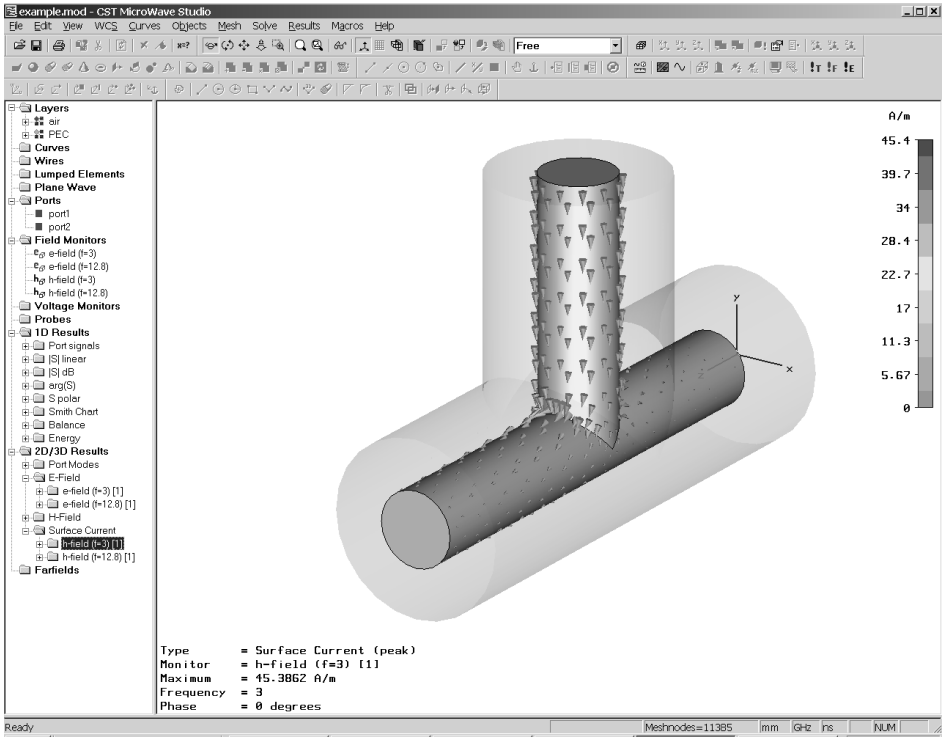
You can investigate the three dimensional electric field distribution by selecting *NT⇨2D/3D Results⇨E-Field⇨e-field(f=3)[1].* The plot should look similar to the picture below:



Note: The quality of the 3D field plots is still being improved. Sometimes arrows may be displayed up to one mesh cell inside the perfect conducting materials, this is only a visualization problem.

You should now play around a little bit with the various field visualization options for the 3D vector plot.

The surface current on the inner conductor can be visualized by selecting the inner conductor by firstly choosing *NT⇨Layers⇨PEC*. The inner conductor should be illustrated solid while the outer dielectric part is shown as transparent. Now you can select the surface current display by selecting *NT⇨2D/3D Results⇨Surface Current⇨h-field(f=3)[1]*. You should finally obtain a plot similar to the following picture:

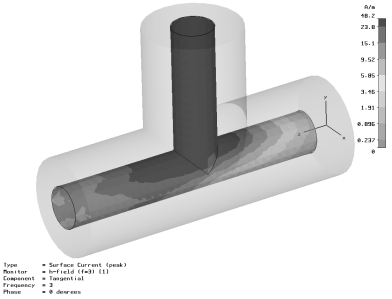


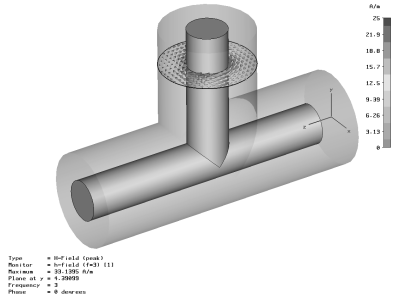
You may now change the plot options in the plot dialog box: *Result⇨Vector Plot* (or *Plot Properties* from the context menu). You can obtain a field animation by pressing the *Start* button located in the *Phase/Animation* frame in this dialog box. Here the phase of the field will be automatically varied between 0 and 360 degrees. The animation can be stopped by pressing the *Stop* button. After clicking in the main view with the left mouse button, you can also change the phase gradually by pressing the *Left* and *Right* cursor keys.

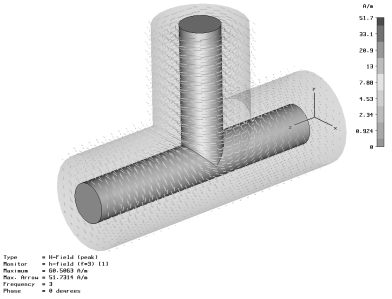
At the frequency of 3 GHz you can see how the current flows through the structure. If you then perform the same steps with the other magnetic field monitor at 12.8 GHz, you can see that almost no current is moving along the 90 degree bend of the coaxial cable.

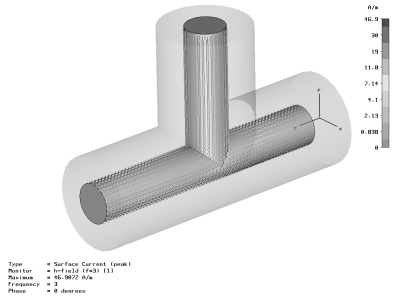
After getting a rough overview of the electromagnetic field distribution in 3D, you can inspect the fields in much more detail by analyzing some cross section cuts through the structure. To do this select an electric or magnetic field (no surface currents) for display and toggle the *Results⇨3D Fields on 2D Plane* ( ) option on. The same plot options are available in the 2D plot mode as you have already used for the port mode visualization. Since the data comes from a 3D result, you may additionally specify the location of the plane at which the fields will be visualized. This can be done in the corresponding *Results⇨Vector Plot* or *Results⇨Scalar Plot* dialog boxes by changing the *Cut plane* settings at the bottom of the dialog boxes.

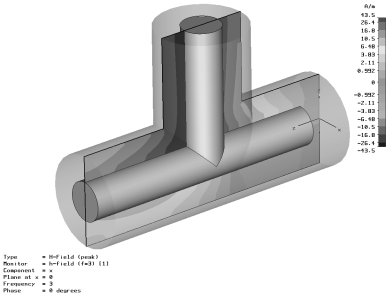
Due to the limited space in this manual, not all the plotting options can be explained here. However, the following gallery shows some possible plot options. Can you reproduce all of them?

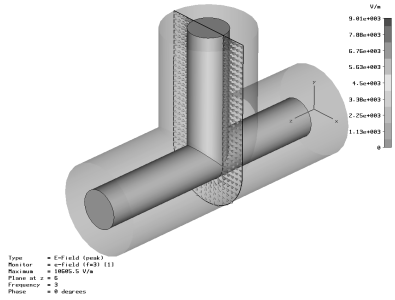


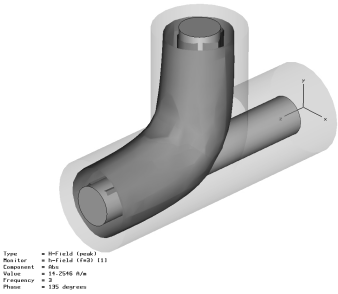


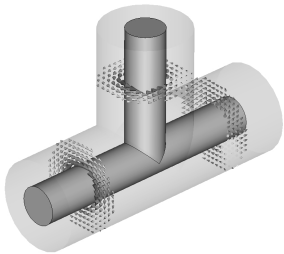












Parameterization of the Model and the Automatic Optimization of the Structure

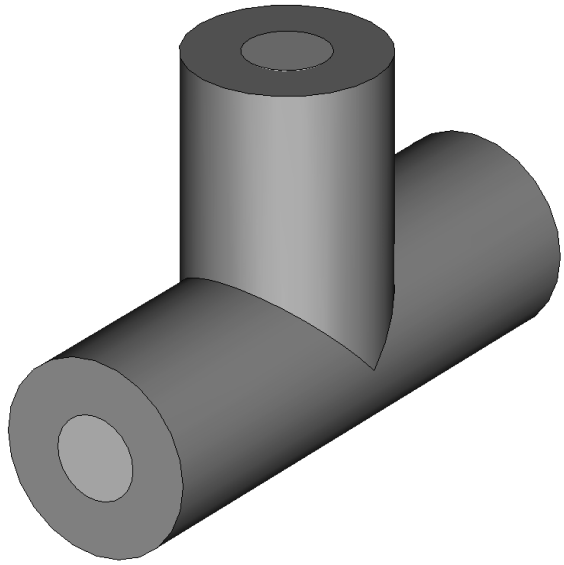
The steps above have demonstrated how to enter and analyze a simple structure. However, structures will usually be analyzed in order to improve their performance. This procedure may be called “design” in contrast to the “analysis” done before.

After you receive some information on how to improve the structure, you will need to change the structure’s parameters. This could be done by simply re-entering the structure which, of course, is not the best solution.

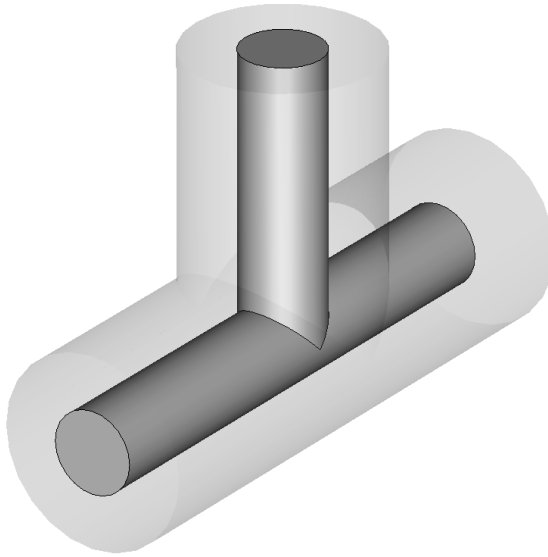
CST MICROWAVE STUDIO® offers quite a lot of options to parametrically describe the structure in order to easily change the parameters. The History List function, as described in a previous chapter, is a very general option, but for simple parameter changes there is an easier solution which will be described below.

Let’s assume that you want to change the stub length of the coaxial cable’s inner conductor. The easiest way to do this is by entering the modeler mode by selecting the “*NT*(navigation tree)⇨*Layers* folder.

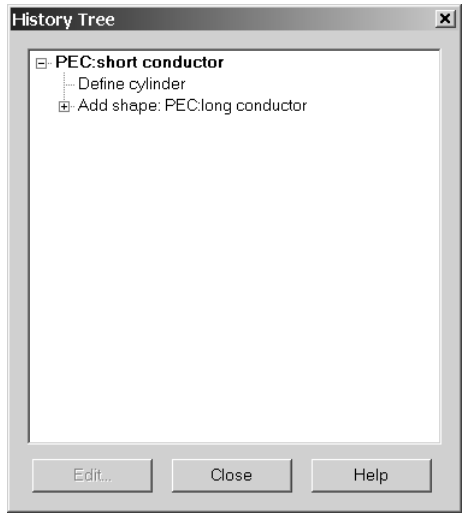
You can now select all ports by clicking on the *NT*⇨*Ports* folder. If you then press the right mouse button, you can choose *Hide all ports* from the context menu. The structure plot should now look like this:



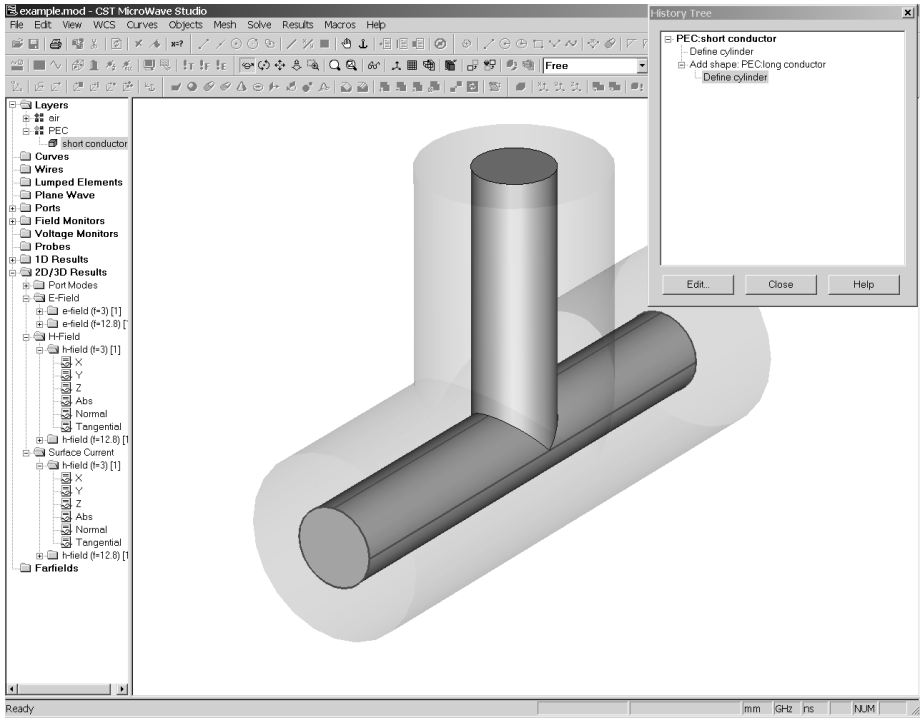
Now select the inner conductor by double-clicking on it with the left mouse button:



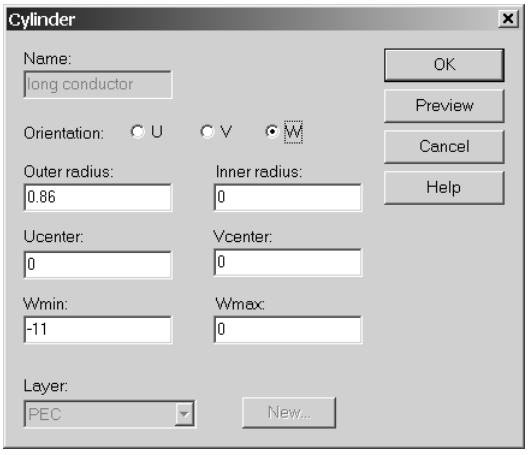
You can now choose *Edit⇨Object Properties* (or *Properties* from the context menu) which will open a list showing the history of the shape's creation:



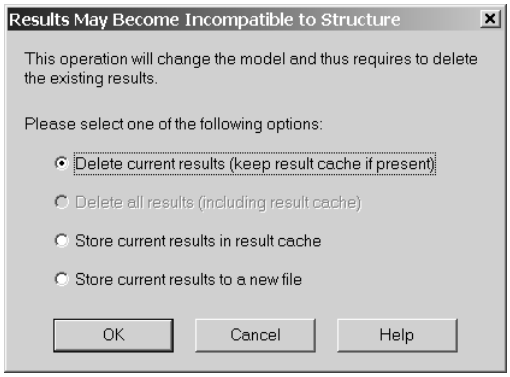
Select the "Define cylinder" operation in the tree folder "PEC:long conductor" from the history tree (see above). Now the corresponding shape will be highlighted in the main window.



After pressing the *Edit* button in the History Tree, a dialog box will appear showing the parameters of this shape.

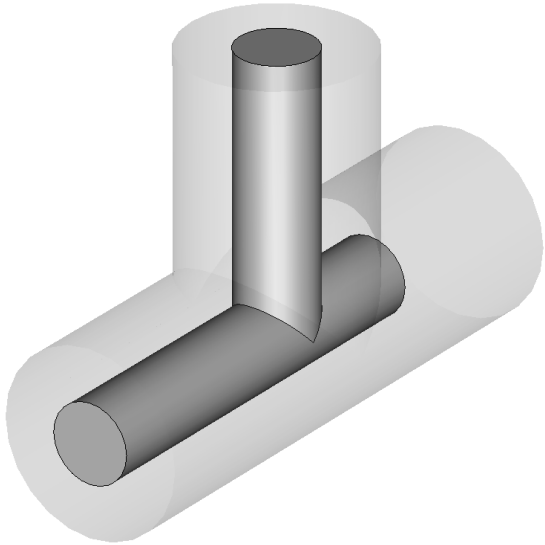


In this dialog box you will find the length of the cylinder (*Wmin*=-11) as specified during the shape creation. Change this parameter to a value of –9 and press *Ok.* Since you are now going to change the structure, the previously calculated results will no longer match the current structure. In such cases the following dialog box will appear:



Here you may specify whether to store the old model together with its results in a cache or as a new file, or just go ahead and delete the current results. In this case you should simply accept the default choice and press *Ok*.

After a few seconds the structure plot changes showing the new structure with the different stub length.

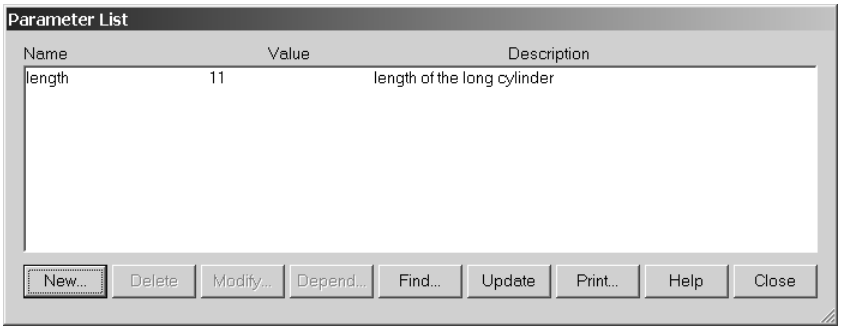


You can generally change all parameters of any shape by simply selecting the shape and editing its properties. This fully parametric structural modeling is one of CST MICROWAVE STUDIO®'s most outstanding features.

The parametric structure definition also works if some objects have been constructed relative to each other, by using local coordinate systems. In this case, the program will try to identify all the picked faces according to their topological order rather than their absolute position in space.

The changes in parameters very occasionally alter the topology of the structure too severely and so the structure update may fail. In this case the History List function offers very powerful options to circumvent these problems. Please refer to the online documentation or contact technical support.

Besides the option to directly change the parameters you may also assign variables to the structure parameters. This can be done by adding a new variable to the parameter list (*Edit⇨Parameters*):



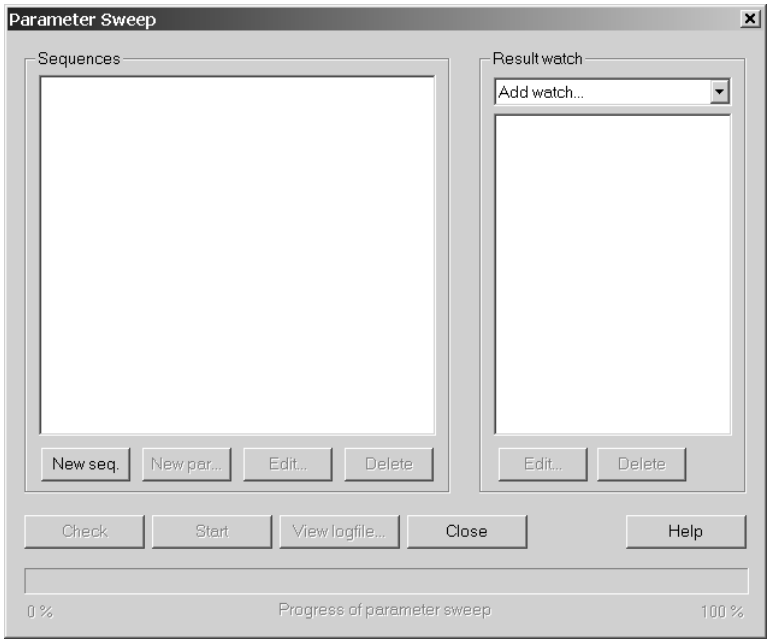
Note: Some dialog box items may be inactive if another structure modification dialog box (e.g. History List) is open. Please close all other dialog boxes before starting to modify the parameter settings.

To add a new structural parameter, press the *New* button, define the name of the parameter (here "length") and assign a current value to it (here 11). You may also enter a text in the *Description* field, so that you can later remember the meaning of the parameter.

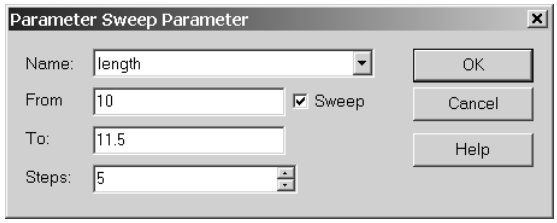
After defining this parameter you should open the parameter dialog box for the long coaxial inner conductor again and assign the expression "-length" to the *Wmin* setting. After pressing *Ok* the structure will be regenerated and should look the same as originally entered.

You can now change the value of this parameter by selecting it and pressing the *Modify* button in the parameter list. After pressing the *Update* button, the structure will be regenerated according to the current parameter value. You can now verify that parameter values between 7 and 11.5 give some useful results.

Since you now have successfully parameterized your structure, it might be interesting to see how the S-parameters change when the length of the conductor is modified. The easiest way to obtain these variation results is to use the Parameter Sweep tool which can be accessed from within the transient solver dialog box by pressing the *Par. sweep* button. This will lead you to the following dialog box:

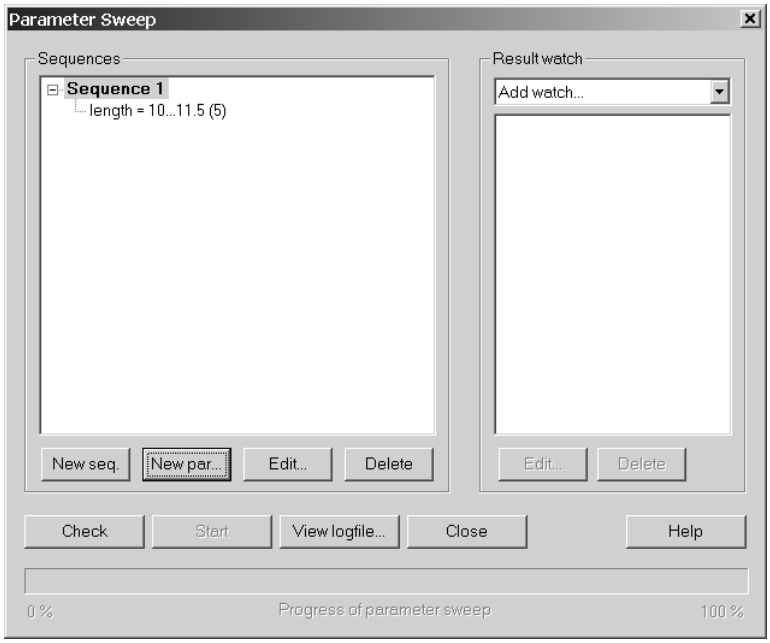


In this dialog box you can specify so called calculation “sequences” which will consist of various parameter combinations. To add such a sequence press the *New seq.* button now. Afterwards you can press the *New par...* button in order to add a parameter variation to the sequence:



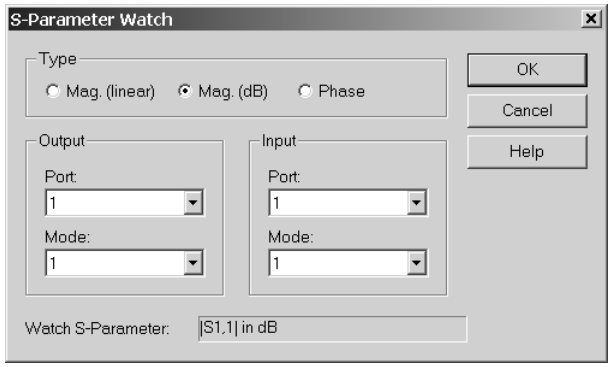
In the ensuing dialog box you can select the name of the parameter to vary in the *Name* field. Afterwards you can specify the lower (*From*) and upper (*To*) bounds for the parameter variation after checking the *Sweep* item. Finally enter the number of steps in which the parameter should be varied in the Steps field.

In this example you should perform a sweep *From* 10.0 *To* 11.5 in 5 *Steps*. After pressing the *Ok* button the parameter sweep dialog box should look as follows:



Please note that you can define an arbitrary number of sequences with each of them containing an unlimited number of different parameter combinations.

In the next step you have to specify which results you are interested in as a result of the parameter sweep. Therefore select "*S-Parameter*" from the *Result Watch* combo box. Now a dialog box opens in which you can specify an S-parameter to store:

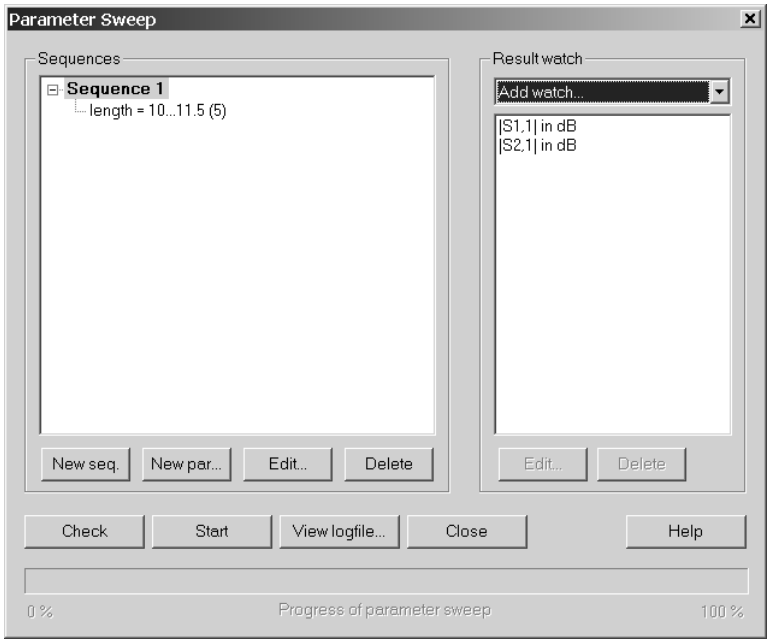


First you should select the option of recording the magnitude of S1,1 in dB by checking *Mag. (dB)* in the *Type* field and pressing the *Ok* button.

Next you should add another "watch" for the magnitude of S2,1 in dB as follows:

1. Select "*S-Parameter*" from *the Result Watch* combo box.
2. Specify *Mag. (dB)* in the *Type* field.
3. Select 2 in the *Output Port* field.
4. Press *OK*.

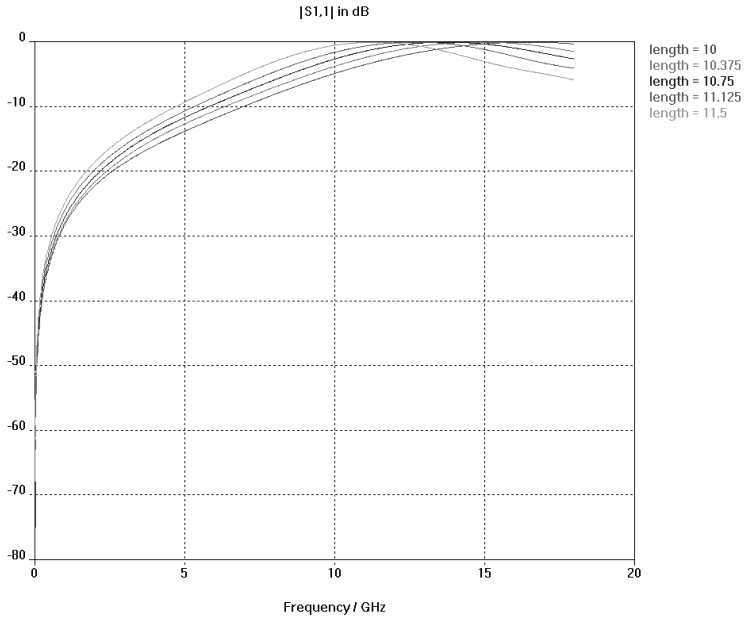
Finally the parameter sweep dialog box should look as follows:



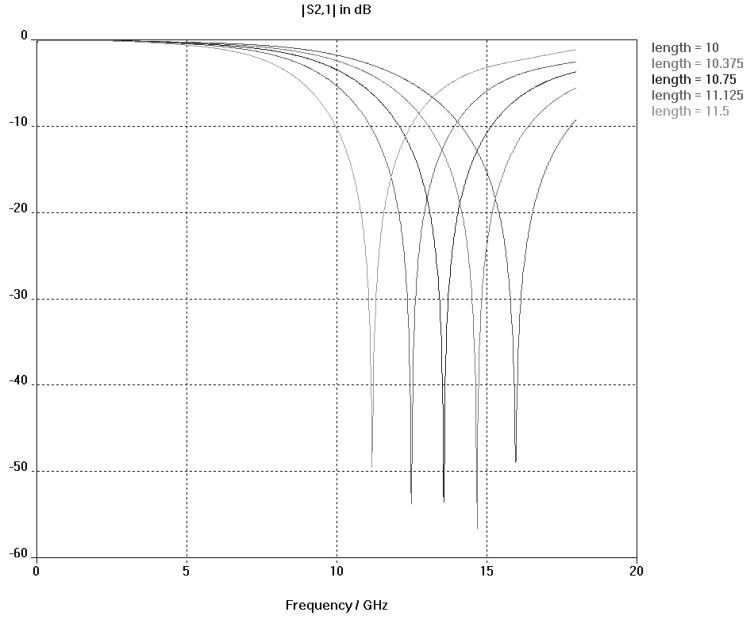
Now you should start the parameter sweep by pressing the *Start* button.

Please note that the parameter sweep uses the previously specified solver settings. If you change the solver settings (e.g. for activating the adaptive mesh refinement), please make sure that the modified settings are stored by pressing the *Apply* button in the solver control dialog box.

After the solver has finished its work you can close the dialog box by pressing the *Close* button. The navigation tree will now contain a new item called “Tables” from which you should now select the item *Tables⇨|S1,1| in dB* first. After setting a proper axis scaling (*Results⇨1D Plot Options⇨Plot Properties...*) you should obtain a plot similar to the following:



Similarly, you can also plot the magnitude of the transmission coefficient by selecting *Tables⇨|S2,1| in dB.*

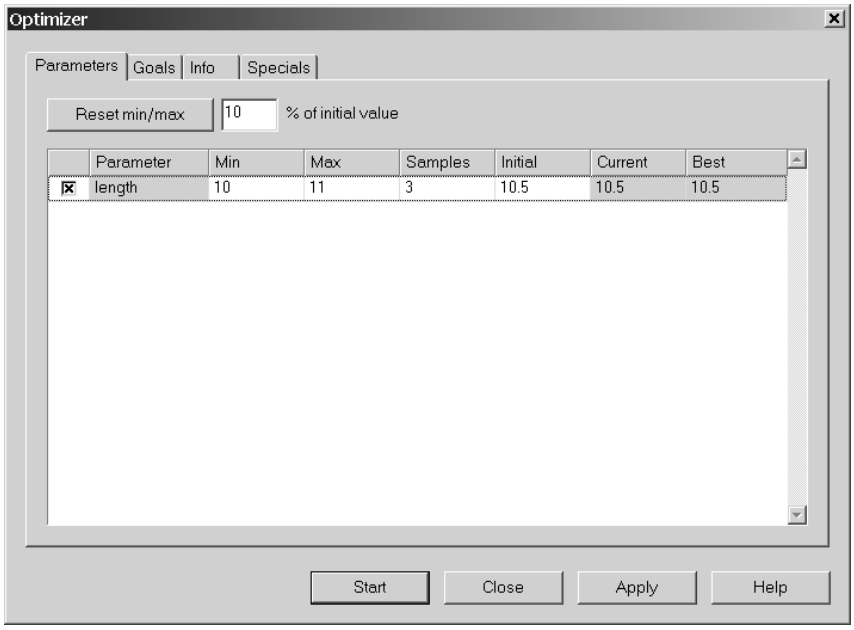


Assume now that you wish the minimum of the transmission S2,1 to be at 13 GHz (which can be achieved within a parameter range of 10 to 11 according to the parameter sweep). However, figuring out the proper parameter might be a lengthy task which can be performed equally well automatically.

Before you continue to optimize this structure you should set the length parameter to a value within the valid parameter range (e.g. 10.5) then update the structure. (please note that you have to enter the modeler mode (e.g. by clicking at the "*Layers*" item in the navigation tree) before you can modify the parameters.

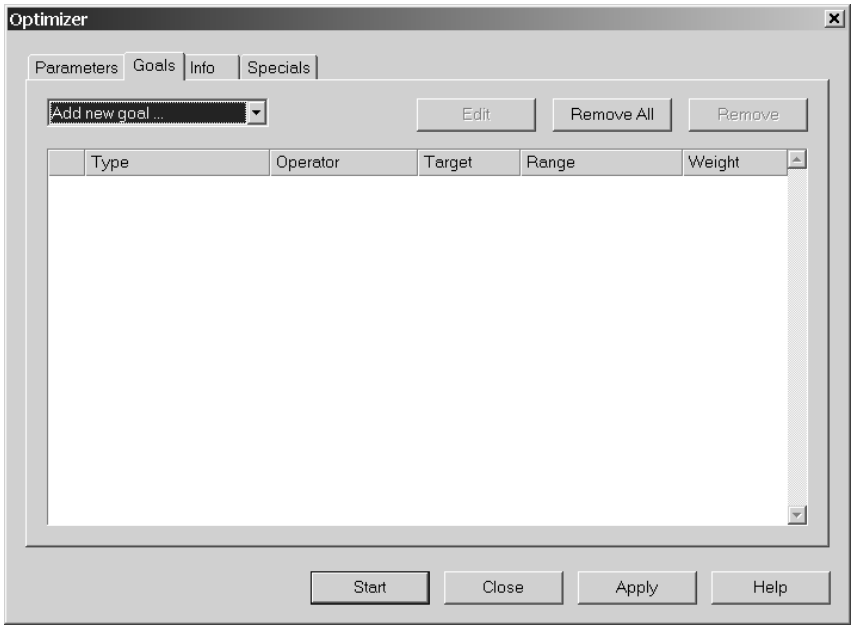
CST MICROWAVE STUDIO® offers a very powerful built-in optimizer feature for these kinds of parametrical optimizations.

To use the optimizer, you only need to open the solver control dialog box as before. In this dialog box you should press the *Optimization* button which then opens the optimizer control dialog box:

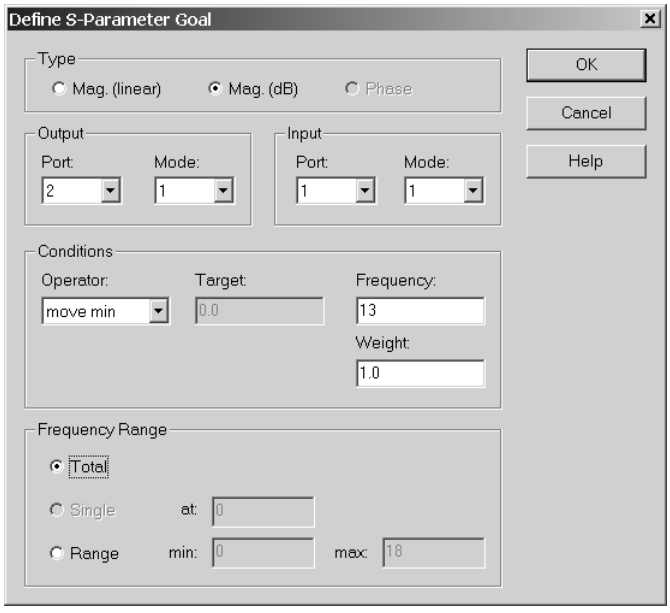


Firstly check the desired parameter(s) for the optimization in the *Parameters* tab of the optimization dialog box (here the "length" parameter should be checked). The next step is to specify the minimum and maximum values for this parameter during the optimization. Here you should enter a parameter range between 10 and 11. For this simple example the other settings can be kept as default. Please refer to the online documentation for more information on these settings.

The next step is to specify the optimization goal. Therefore you should click on the *Goals* tab.



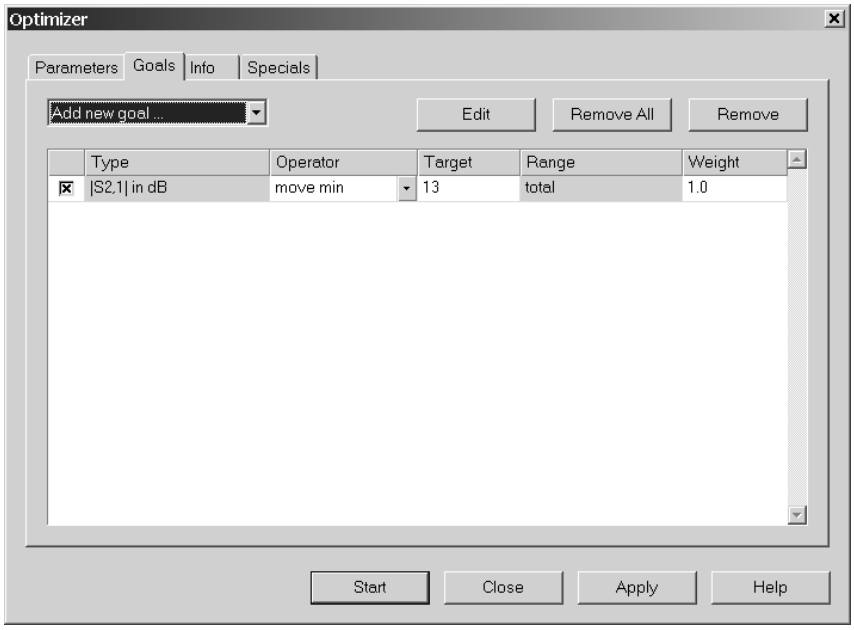
The next step is to specify a list of goals to achieve during the optimization. In this example the target is to move the minimum of the S-parameter S2,1 to a given frequency. Therefore you should define a S-parameter goal by selecting it from the list which appears if you click on the *Add new goal* entry. Once you have selected the proper goal type, the following dialog box should appear:



In this dialog box you should firstly select the magnitude of the S-parameter in dB for optimization by clicking on the *Mag.(dB)* entry in the *Type* frame. The next step is to

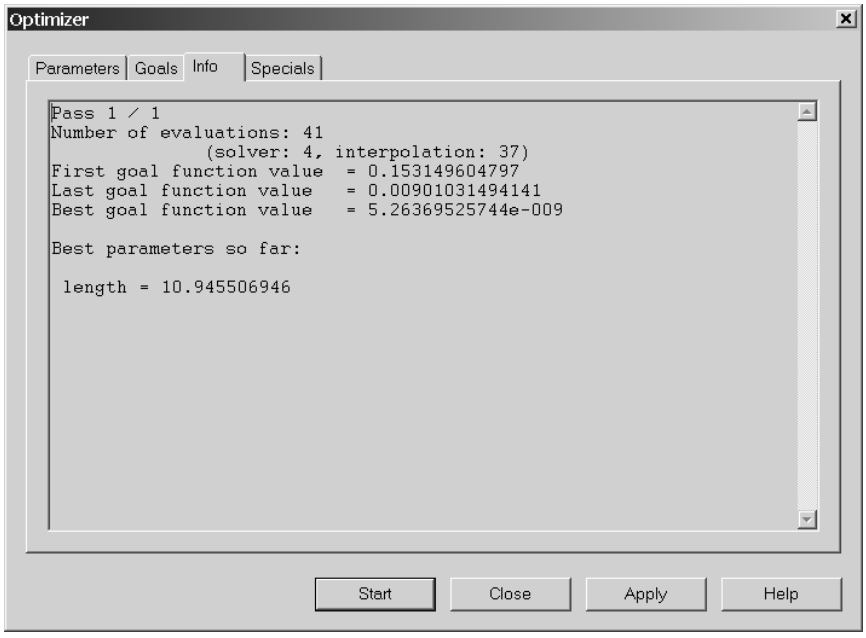
specify which S-parameter should be optimized. You should now select S2,1 by setting *Port* = 2, *Mode* = 1 in the *Output* frame and *Port* = 1, *Mode* = 1 in the *Input* frame.

The next step is to specify the actual goal for the previously specified S-parameter data. Since we want to move the minimum of S2,1 in this example you should select the *move min* operator in the conditions frame. Afterwards, please set the Frequency to which the minimum should be moved to (13 GHz). If more than one minimum exists in the S-parameter data you can limit the frequency range for searching the minimum in the Frequency Range frame. In this example you can just skip these settings and accept the default. After finally pressing Ok, the optimizer dialog box should look as follows:



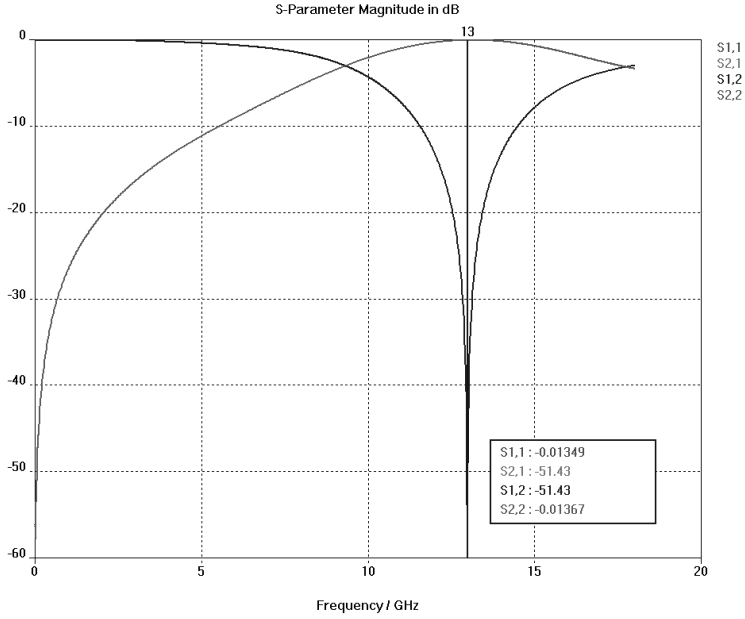
Since we now have specified optimization parameters and goals, the next step is to start the optimization procedure by pressing the Start button. The optimizer will show the progress of the optimization in an output window in the Info tab which will be activated automatically.

When the optimization has finished, you should confirm saving the new parameter settings. The optimizer output window will show the best parameter settings with respect to the given goal.



Please note that due to the sophisticated optimization technology only four transient solver runs were required to find the optimal solution to a very high accuracy.

You can now visualize the S-parameters for the optimal parameter setting (length =10.9455) and should obtain the following picture (you can activate the axis marker tool *Results⇨1D Plot Options⇨Axis Marker* to verify that the location of the peak really is at 13 GHz).



Summary

This example should have given you a very basic overview of the key concepts of CST MICROWAVE STUDIO®. You should now have a basic idea of how to:

1. Model the structures by using the solid modeler
2. Specify the solver parameters, check the mesh and start the simulation
3. Use the adaptive mesh refinement feature
4. Visualize the port modes
5. Visualize the time signals and S-parameters
6. Define field monitors at various frequencies
7. Visualize the electromagnetic field distributions
8. Define the structure by using structure parameters
9. Use the parameter sweep tool for parameter studies
10. Perform automatic optimizations

If you are familiar with all these topics you have a very good starting point for further improving your usage of CST MICROWAVE STUDIO®.

For more information on a particular topic we recommend you to take a look at the contents page of the online help manual which can be opened with *Help⇨Contents*. If you have any further questions or remarks, please do not hesitate to contact your technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Please ask your support center for details.

Which Solver to Use?

Since we have focused on the transient solver in the previous example, it's time to clarify which solver fits best to which application. The transient solver is very general and can solve the widest range of electromagnetic field problems. However, for some applications, specialized solvers will show much better performance while maintaining the same high level of accuracy.

The following table lists a few typical applications together with the solvers which are most frequently used for solving the particular problem. Please note that because of the very wide application spectrum not all possible examples can be listed in the table.

Application Name	**Solver Type(s)**
Connectors (coaxial, multi-pin)	Transient
Strip lines (micro strip, coplanar lines)	Transient, Frequency Domain
Strip line circuits	Transient, Frequency Domain
Cross-talk calculations	Transient
Printed circuit boards	Transient
Digital circuit simulation	Transient
Packaging problems	Transient
Network parameter (SPICE) extraction	Transient, Frequency Domain
Nonlinear diode applications	Transient
EMI problems	Transient
Radiation problems	Transient
Shielding (irradiation) problems	Transient
Monopole, dipole and multipole antennas	Transient
Patch antennas	Transient, Frequency Domain
Conformal antennas	Transient
Helical and spiral antennas	Transient
Antenna arrays	Transient
Waveguides (hollow, dielectric, coaxial)	Transient
Transmission line networks	Transient
Optical wave guides	Transient
Optical couplers	Transient
Optical diplexers and filters	Transient, Frequency Domain, Eigenmode / Modal Analysis
Filters and diplexers	Transient, Frequency Domain, Eigenmode / Modal Analysis
Cavities, Resonator design	Eigenmode

The difference between a port driven analysis (transient, frequency domain and modal analysis) and a typical eigenmode computation is quite obvious, the difference between

transient, frequency domain and modal analysis is unfortunately a bit harder to understand.

A transient analysis is usually quite fast, because by applying a wide band excitation signal, the whole broad band response of a structure can be analyzed within one single calculation run. Therefore the time signals are transformed into the frequency domain by applying a Discrete Fourier Transformation. Such Time Domain to Frequency Domain transformations require that the time signals have sufficiently decayed to zero at the end.

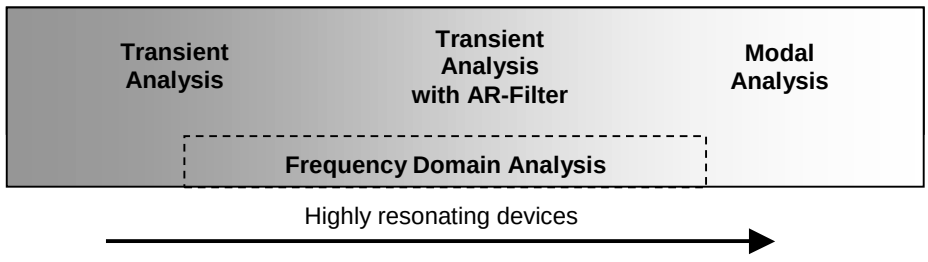
However, if the structure has some very sharp resonances, the field energy may decay very slowly in the device after the excitation has vanished. Such behavior will lead to very long computation times in the transient analysis until the time signals have sufficiently decayed to zero.

For the latter structures the behavior of the device is dominated by the resonant modes from which it becomes clear that the modal analysis will be the appropriate solution method. The modal analysis computes some of the first modes in the device and from the modal distribution, together with the port mode solutions, it finally extracts the S-parameters of the device. The more the behavior of the device is controlled by a few strongly resonant modes, the more efficient this type of analysis becomes.

Please note that the application range of the transient analysis can be extended quite a way towards more resonant devices by applying some advanced digital signal processing techniques rather than simply using a Discrete Fourier Transform. CST MICROWAVE STUDIO® features a so-called Auto Regressive (AR) Filter which is capable of predicting the long term response of a device from a short term response.

The performance of the transient solver also degrades if the device operates at very low frequencies. In these cases the frequency domain solver can be faster, especially if a few frequency samples are sufficient to characterize the structure's behavior. A good example is the extraction of a SPICE model for a structure which is much smaller than the wavelength, which requires the field calculation at a single frequency point only. On the other hand, the performance of the frequency domain solver decreases more rapidly with the number of mesh cells than the performance of the transient solver. Therefore the transient solver should be preferred if the structure is modeled with a large number of mesh cells (e.g. more than 100.000).

Summarizing these statements, we end up with the following diagram showing the application ranges of the methods:



You should now have an impression of the pros and cons of the methods. If you are not sure which solver may best suit your application, please contact your local sales office for assistance.

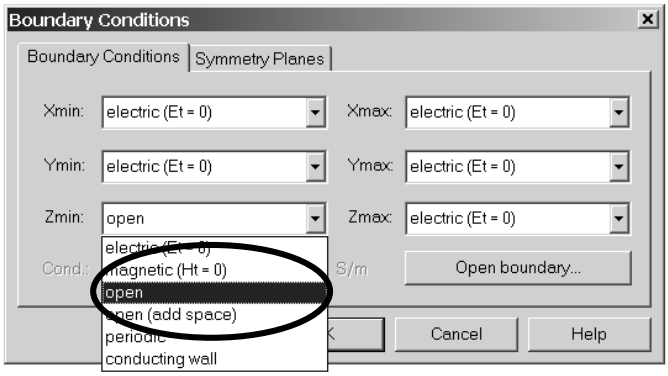
In the following we will point out the differences in the simulation workflow for typical antenna, frequency domain, eigenmode and modal analysis computations in contrast to the S-parameter example discussed earlier in this chapter.

Antenna Computations

An antenna computation is usually performed using the transient solver and is thus quite similar to the procedure explained in the previous example.

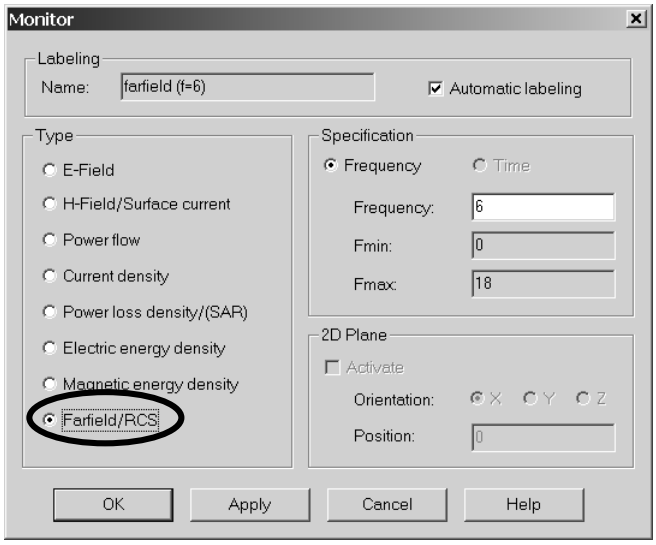
In some rare cases it may be advantageous to use the frequency domain solver. The only difference between these calculations and the procedure described below is that the frequency domain solver needs to be called rather than the transient solver. Please refer to the frequency domain solver online help for more information.

Considering transient analysis, the main difference between an antenna calculation and the S-parameter calculations described earlier in this document lies in the definition of the boundary conditions. Since the antenna radiates into free space, open (or absorbing) boundary conditions must be applied. Therefore simply select "*open*" boundaries in the Solver⇨*Boundary Conditions* dialog box:



Please note that the open boundary conditions require some space between the device and the boundary planes for optimum performance. Since the open boundary conditions are very accurate only a small distance is necessary. However, if you are not sure about the amount of space needed, you may simply choose "*open (add space)*" from the boundary options. In this case, the necessary space is estimated automatically.

For the calculation of the antenna farfield gain or directivity pattern (electric farfield distribution in a spherical coordinate system, left and right hand polarization, axial ratio), so called "farfield monitors" need to be defined before the simulation is started. Similar to the definition of the other field monitors, an arbitrary number of these monitors for various frequencies can be defined. This means that you can compute the antenna farfield for multiple frequency points from one single transient analysis. The farfield monitors are specified in the *Solve⇨Monitors* dialog box:



After the transient analysis is completed, you can access your farfield results from the *Navigation Tree⇨Farfields* folder. Typical antenna characteristics such as main beam direction, gain, efficiency, side lobe suppression, etc. are automatically calculated and displayed. Please refer to the tutorial **Patch Antenna** for more information.

The following table summarizes the input necessary for antenna calculations:

1. Set units
2. Set background material
3. Define structure
4. Set frequency range
5. Set (open) boundary conditions
6. Define excitation ports
7. Set (farfield) monitors
8. Start transient solver
9. Analyze results (input impedance, farfields, etc.)

Please note that some templates are available for antenna calculation which take care of some of the settings above. You should use a proper template whenever possible.

Simplifying Antenna Farfield Calculations

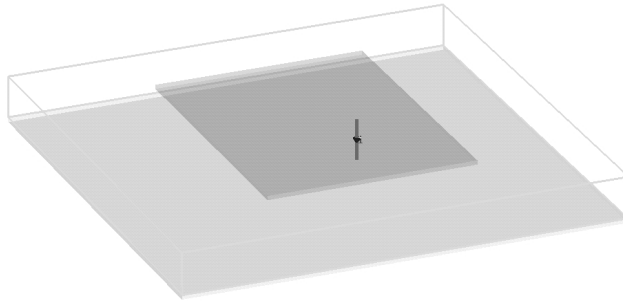
In many cases where only the antenna farfield pattern is of interest rather than the feeding point impedance, it is not necessary to model the actual geometry of the feeding point. However, when you are interested in very accurate results of the antenna's input reflection, it is essential to model the feeding point exactly as it is.

In cases where you are able to use a simplified model, you can use discrete ports rather than waveguide ports (please refer to the **Discrete Ports** section later in this chapter).

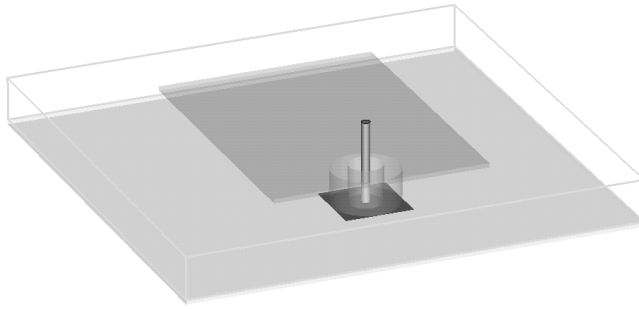
If you start the analysis of a new antenna it is usually a good approach to begin with a discrete port. Since the model is easier to build, you will obtain initial S-parameter and farfield pattern results pretty quickly. This will allow you to assess the principal operation of the antenna before increasing the accuracy (optional) by generating a detailed model of the feeding point geometry.

The following pictures show feeding point models of a simple patch antenna as an example.

a) Simplified model of the feeding point with a discrete port



b) Detailed model of the feeding point using a waveguide port



In the first picture a) the antenna is fed by a discrete port which represents a current source with an internal resistance. This approach delivers accurate farfield results but may yield S-parameters which are not directly comparable to the measurements.

In picture b), the antenna is fed by a coaxial line (as in the real world structure) which gives accurate farfield patterns and S-parameters.

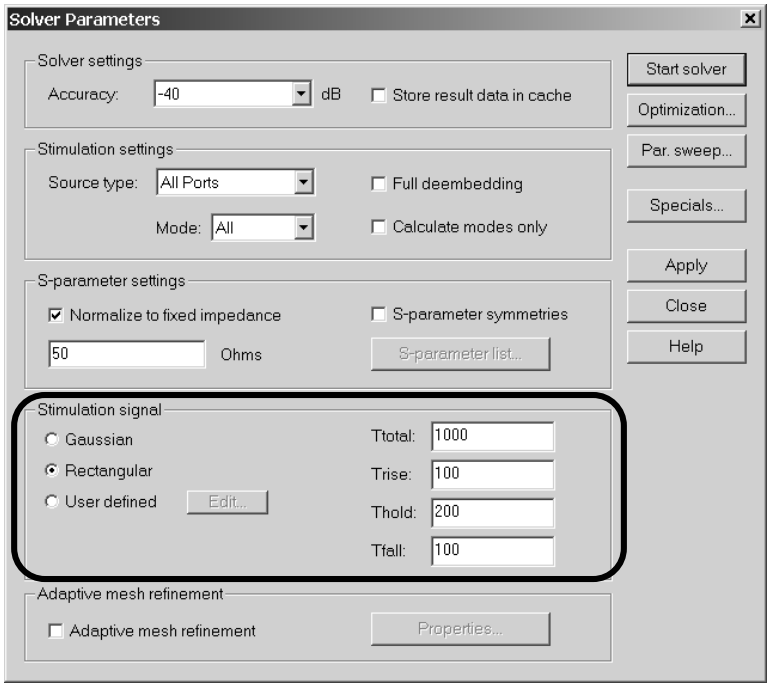
Digital Calculations

A digital calculation is typically performed by using the transient solver. Thus the overall simulation procedure is quite similar to the procedure described earlier in this document.

The main difference between a digital calculation and a typical S-parameter calculation is the definition of the excitation signal.

For S-parameter calculations the excitation signal for the transient analysis is typically defined by a gaussian shape for which the Fourier spectrum is also given by a gaussian shape covering the entire frequency band of interest. Therefore the time signal is determined mainly by the demands on the frequency band.

In contrast to this, the excitation signal for a digital simulation is described in the time domain by specifying rise-, hold- and fall-time of a rectangular pulse in the solver control dialog box:

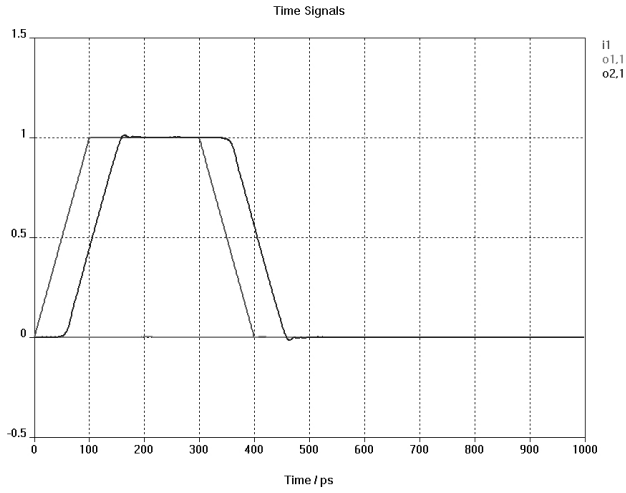


Please note that the parameters of the rectangular excitation function are specified in the currently selected time units.

In the example shown above (with the time unit set to ps) the settings define a rectangular shape with a risetime of 100 ps, a holdtime of 200 ps and a falltime of 100 ps. The rise- and falltimes of 100 ps correspond to a bandwidth of approximately 10 GHz. The maximum simulation time is given in the *Ttotal* field and is set to 1000 ps in this example. The solver will automatically stop after simulating the given total time range.

The solver will also stop automatically when the energy in the calculation domain is below the level specified in the *Accuracy* field. To switch this automatic termination off, please select *No Check* in the *Accuracy* field.

In our example, the structure is a simple microstrip line which shows the following response to the digital excitation:



The excitation signal “i1” shows the given rise-, hold- and falltimes. The output signal “o2.1” has a slightly changed pulse shape (due to the dispersion of the line) and a time delay because of the finite length of the transmission line.

Please note that in addition to this simplified description of the excitation signal it is also possible to set a user defined pulse shape. Please refer to the online documentation for details.

The following table summarizes the input necessary for digital calculations:

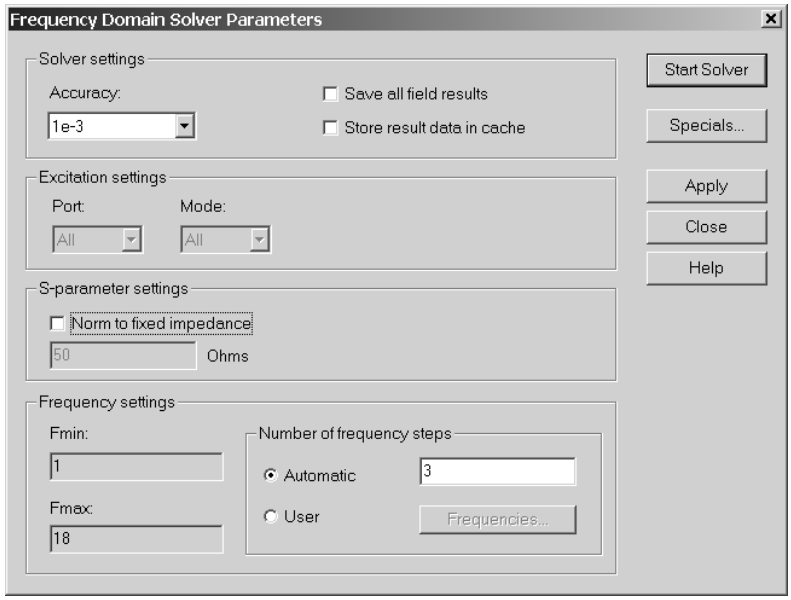
1. Set units
2. Set background material
3. Define the structure
4. Set the frequency range (covering all desired harmonics)
5. Set the boundary conditions
6. Define the excitation ports
7. Set the monitors
8. Define the excitation signal parameters
9. Start the transient solver
10. Analyze the results (usually the time signals)

Please note that some templates are available for various types of structures in order to assist you to specify these settings. You should make use of these templates whenever possible.

Frequency Domain Computations

The frequency domain analysis procedure is very similar to the transient analysis. It can mainly be separated into S-parameter calculations and antenna calculations.

The main difference between frequency domain simulations and the transient analysis runs can be seen in the preparation of the solver control dialog box: *Solve⇨Frequency Domain Solver* when the solver is going to be started.



The most important setting for the frequency domain solver is the number of frequency samples to be calculated. Since the solver is calculating the electromagnetic field solution step by step for each frequency point, the total solver time linearly increases with the number of frequency steps. This behavior is completely different to that of the transient analysis where the number of frequency samples has almost no influence on the solver time.

In order to keep the number of calculated frequency points as small as possible, automatic frequency sampling can be used rather than uniform sampling. To use this option, simply check the *Automatic* button in the *Number of frequency steps* frame. You should then enter the maximum number of frequency steps in the entry field next to this button. If you enter 10 in this field, the solver will calculate the fields (and S-parameters) at the minimum and the maximum frequency plus eight frequency points in between. The solver will automatically adjust the location of these eight points in order to achieve as the best possible approximation of the actual S-parameter curves.

To store the fields at particular frequencies, monitors need to be defined in advance, as described previously for the transient solver. These monitor frequencies will then be added to the list of calculated frequency points.

Please note that the frequency domain solver can not calculate the fields at a frequency of zero. Therefore an error message will appear if the frequency domain solver is started

while the lower frequency limit is set to zero. In this case please increase the lower limit of the frequency domain to a value greater than zero.

The S-parameters and fields can be accessed as usual from the items in the navigation tree.

The following table summarizes the input necessary for frequency domain analysis calculations:

1. Set units
2. Set background material
3. Define the structure
4. Set the frequency range (fmin>0)
5. Set the boundary conditions
6. Define the excitation ports
7. Set the monitors
8. Start the frequency domain solver
9. Analyze the results (S-parameters, field patterns, etc.)

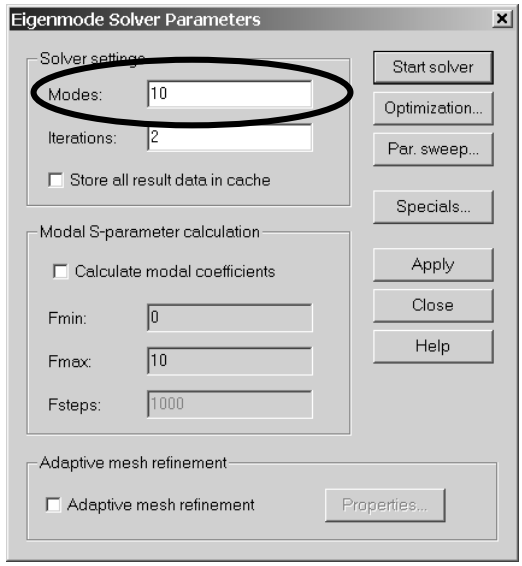
Please note that some templates are available for various types of structures in order to assist you in specifying these settings. You should make use of these templates whenever possible.

Eigenmode (Resonator) Computations

The eigenmode solver calculates a finite number of modal field distributions in a closed device. The solver is currently not capable of solving lossy problems. However, due to a perturbation method, losses and Q-factors of the device can be obtained in the post processor. This well-proven technique works sufficiently accurate in most cases.

Since the eigenmode analysis does not require the definition of excitation ports, this step can simply be omitted. The definition of monitors is also not necessary, because the modes themselves contain all available information about the device.

Thus, after setting up the model, you can immediately proceed to the eigenmode solver dialog box (*Solve⇨Eigenmode Solver*) which looks as follows:

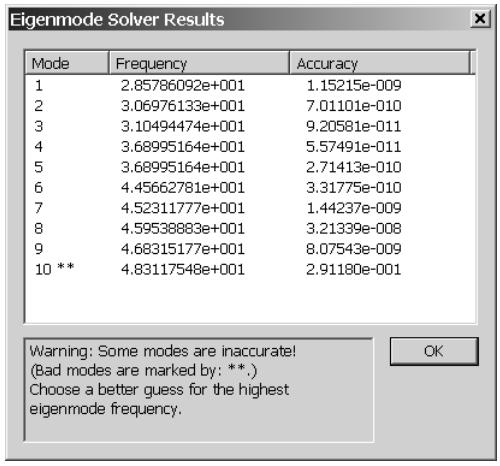


The most important control in this dialog box is the number of *Modes*.

The typical simulation procedure is as follows:

1. Enter the desired number of *Modes* (n). The solver will then compute the first n modes of the device. It is often advantageous to specify more modes to be calculated than you actually need, e.g. enter 10 modes to be calculated if you actually need 5. In most cases it is a good choice to calculate at least the first 10 modes of the device.
2. Press the *Start solver* button.

After the solver has finished, a dialog box will appear showing the results for the mode frequencies and the corresponding accuracy of the modal field solution:



Sometimes a few of the higher modes will not be calculated with sufficient accuracy and thus be marked with **. However, this does not affect the accuracy of the lower modes and is the reason why you should specify more modes to calculate than you actually need.

The eigenmode solver internally needs an estimation for the frequency of the highest mode of interest. Usually this frequency is estimated automatically and refined by refinement passes if necessary.

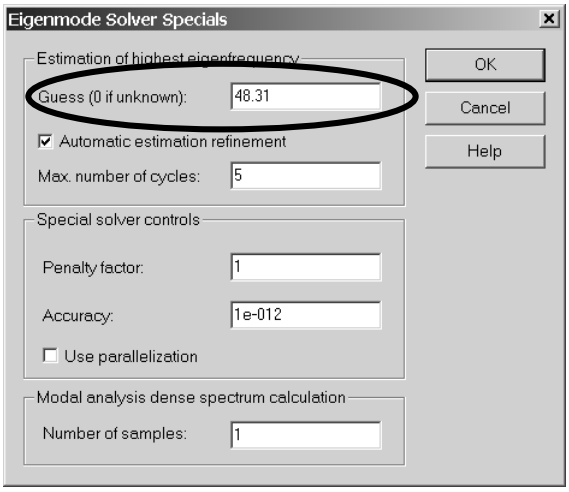
Performing estimation refinement passes reduces the performance of the eigenmode calculation. In order to speed up the eigenmode calculation in these cases, you can manually enter a guess for the frequency of the highest mode you are looking for. The eigenmode solver automatically derives such a guess from previously calculated results and prints this value in the log-file. The log-file can be shown by selecting Results⇨*View Logfiles*⇨*Solver Logfile* and will contain the following information (you may need to scroll the text down in order to see these lines):

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

Optimum guess for the highest eigenfrequency would be: 48.31

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

You can set this guess by opening the special settings dialog box by pressing the *Specials* button in the solver control dialog box. In the *Guess* field you should now enter the proposed guess as 48.31 (GHz) in this example. If you are unsure about this setting you should simply specify zero for automatic estimation.



Back in the eigenmode solver control dialog box, restart the solver by pressing the *Start solver* button again.

The eigenmode solver results for the N-th mode can be accessed from the navigation tree in the *2D/3D Results⇨Modes⇨Mode N⇨e* (for the electric field) and *2D/3D Results⇨Modes⇨Mode N⇨h* for the magnetic field, respectively.

Please refer to the tutorial: **Cavity** for more information on post-processing the results.

The following table summarizes the input necessary for eigenmode calculations:

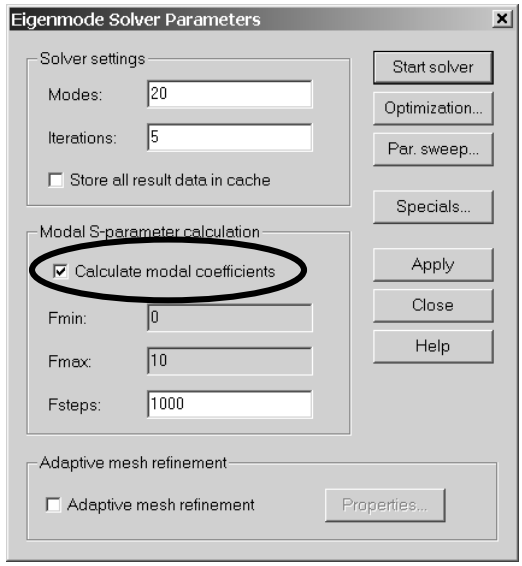
1. Set units
2. Set background material
3. Define structure
4. Set frequency range
5. Set boundary conditions
6. Start eigenmode solver
7. Analyze results (field patterns, frequencies, losses/Q-factors)

Please note that some templates are available for various types of structures in order to assist you to specify these settings. You should make use of these templates whenever possible.

Modal Analysis Computations

The modal analysis works in combination with the eigenmode solver and calculates S-parameters. However, in contrast to the transient solver, the method is limited to wave guide ports and can not currently handle discrete ports.

Compared to the calculation of S-parameters using the transient solver you will only skip the definition of monitors. The simulation itself can be started from within the eigenmode solver dialog box: *Solve⇨Eigenmode Solver*



The settings for the eigenmode solver are the same as explained in the previous section. The only new settings are the entries in the *Modal S-parameter calculation* frame.

In a first step you will need to switch the modal analysis feature on by selecting the *Calculate modal coefficients* check button. Afterwards you only need to define the desired number of frequency samples within the specified frequency. The latter setting has little effect on the simulation time, so you may choose reasonably large values, e.g. 1000.

Finally, just press the *Start* button. After the simulation has finished, you should check the accuracy of the eigenmode solution as described in the previous section.

The resulting modal field patterns can be accessed from the navigation tree in the *2D/3D Results⇨Modes* folder.

The S-parameters can be accessed as usual from the *1D Results⇨|S| linear*, *|S| dB*, *arg(S), S polar*, *Smith Chart* and *Balance* folders.

The eigenmode calculation for filter structures is usually a challenging task for the eigenmode solver. Thus it is a good choice to increase the number of *Iterations* to 5 for

modal analysis calculations. The most critical setting in the modal analysis is the definition of the number of *Modes* to consider for the S-parameter calculation.

Please note: For highly resonant devices, only a few modes will typically be required. However, for weakly resonant structures many modes may be necessary which renders this method inefficient for these kinds of applications (although it can be applied).

The most reliable way to determine the necessary number of modes is to start with a small amount of Modes (e.g. 10-20), perform the simulation, increase the number and check whether the S-parameters change significantly. You should continue this procedure until the S-parameters are sufficiently stable. Finally you can note the required number of modes and apply this value to all your structures of the same type (e.g. for optimizations, etc.).

Especially for highly resonating structures (the typical application domain of this method) only a small number of modes (10-20) will be required to yield accurate solutions. For typical filter structures, a setting of 20 modes will be just fine.

For more information, please refer to the tutorial: **Narrow Band Filter**.

The following table summarizes the input necessary for modal analysis calculations:

1. Set units
2. Set background material
3. Define structure
4. Set frequency range
5. Set boundary conditions
6. Define excitation (waveguide) ports
7. Start modal analysis solver
8. Analyze results (field patterns, frequencies, loss/Q-factors, S-parameters)

Please note that some templates are available for various types of filter structures in order to assist you to specify these settings. You should make use of these templates whenever possible.

Discrete Ports

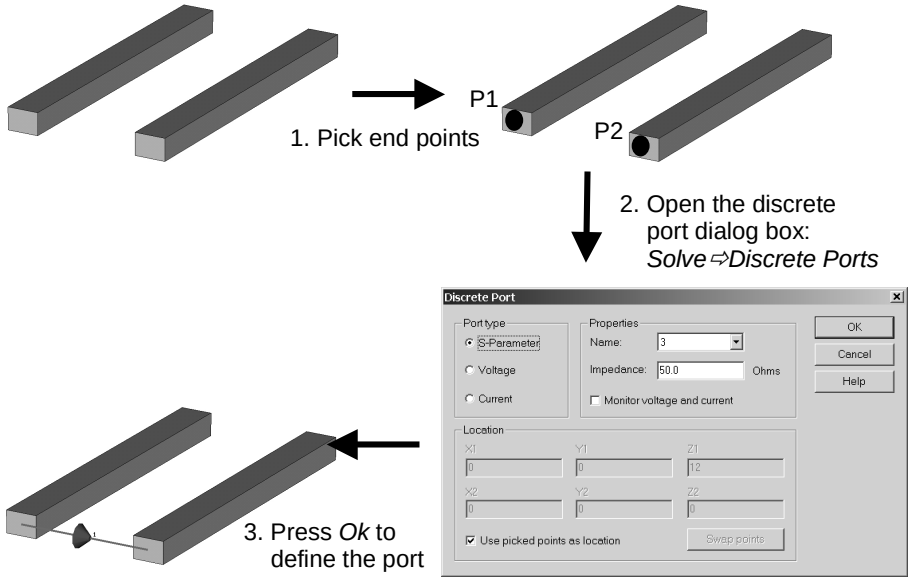
In general, two different types of ports exist for transient and frequency domain analyses. The transient analysis example described earlier in this document only uses so-called waveguide ports.

This kind of port simulates an infinitely long wave guide connected to the structure. The wave guide modes travel out of the structure towards the boundary planes and thus leave the computation domain with very low levels of reflections.

Although the waveguide ports are definitely the most accurate way to terminate a wave guide, sometimes discrete ports are more convenient to use. Discrete ports consist of a current source with an internal resistor and have two pins with which they can be connected to the structure.

This kind of port is often used as feeding point source for antennas or as the termination of transmission lines at very low frequencies. At higher frequencies (e.g. the length of the discrete port is longer than a tenth of a wavelength) the S-parameters may differ from those when using wave guide ports because of the improper match between the port and the structure.

The typical way to define a discrete port is to pick its two endpoints from the structure by using the common pick tools and then enter the discrete port dialog box:



For transient for frequency domain analyses, discrete ports can be used in the same way as waveguide ports.

SPICE Network Model Extraction

Some applications require the derivation of SPICE compatible network models which have the same port behavior as the 3D structure.

Therefore CST MICROWAVE STUDIO® offers a so called *Network parameter extraction* feature which allows the automatic generation of a SPICE compatible netlist for the given device. Since the model is based on a network of coupled transmission lines, the approximation is best for transmission line like devices such as connectors or IC packages.

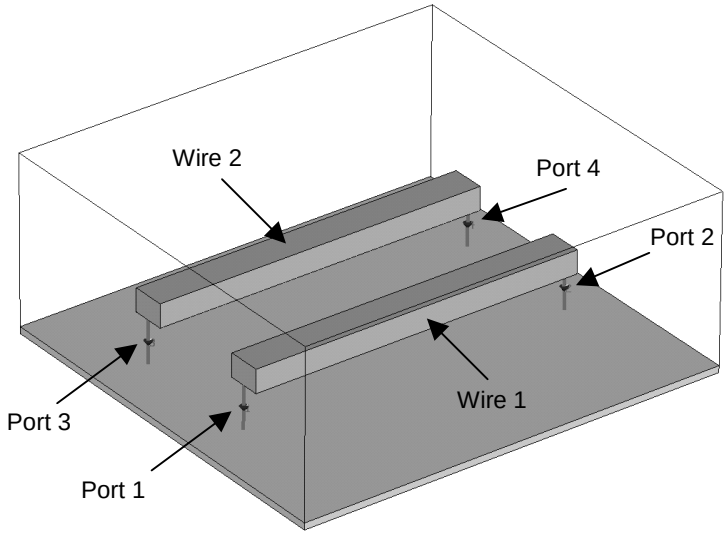
The network parameter extraction is based on the definition of so-called *Wires* for which a T-shaped network representation will be generated. The frequency range for sufficiently accurate representations can be extended by cascading the model.

The extracted network model will describe both the behavior of each of the transmission lines as well as the coupling (cross-talk) between these lines.

The procedure for calculating the network model is quite similar to the simulation procedure of a transient analysis. After modeling the structure, defining its material parameters, its frequency range, etc., the ports need to be defined.

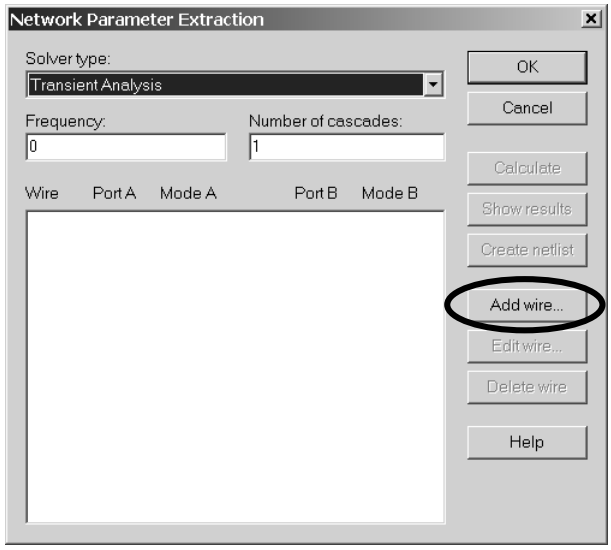
Since the network model assumes that the structure can be described by a lumped element network, the validity of this approach is limited to frequency ranges within which the structure behaves like a transmission line network. Therefore it is usually sufficient to use discrete ports for this kind of calculation.

The following example calculates the network model of two coupled transmission lines. To reproduce this calculation you should enter a structure which looks similar to the picture below:



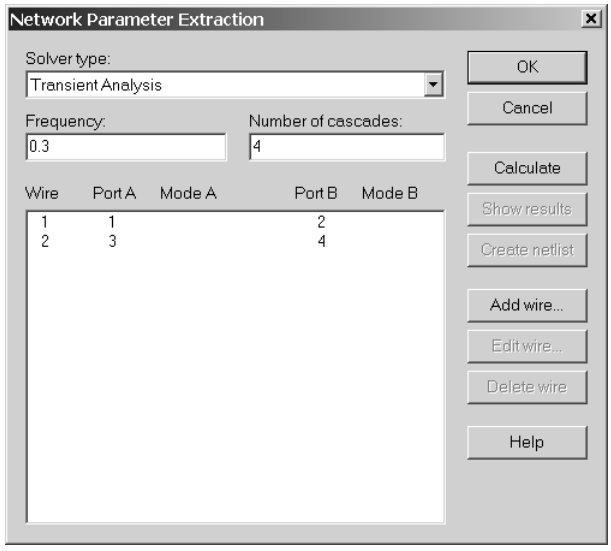
All conductors (including the ground plane) are modeled as perfect conductors here. Please note that it is not necessary to model the ground plane, because the corresponding boundary plane can simply be set to *electric*.

The next step is to open the network parameter extraction dialog box by selecting: *Solve⇨Extract Network Parameters*:



The first step is to specify which solver should be used for the calculation of the network model, this is usually set to *Transient Analysis*.

Afterwards you need to define the wires for which the network model will be generated. In this example, the two conductors above the ground plane will be defined as wires. The first wire starts at port 1 (= Port A) and ends at port 2 (= Port B). Analogously, the second wire is defined between port 3 and port 4. Since only discrete ports are used here, the specification of modes is not required.



Afterwards you should define the *Frequency* for the SPICE model extraction. In most cases this frequency should be set such that the corresponding wave length is approximately 100 times larger than the length (*l*) of the transmission lines, thus: $f \approx 0.01c/l$.

In the next step, the number of cascades should be specified, where both the accuracy and the complexity of the generated SPICE circuit increases with this setting. A value of 4 is a good compromise for many practical applications. Please refer to the online documentation for more details.

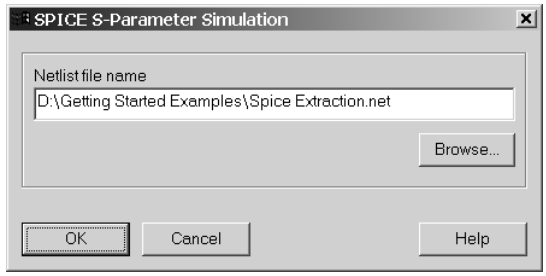
Finally the network parameter extraction can be started by pressing the *Calculate* button. If the S-parameter results are not yet available, the transient solver will automatically be invoked before the actual SPICE model generation is performed. Please note that the transient solver will only to be started once per wire (not once per port as it will be the case for *Source Type* = *All Ports*).

After the solver has finished, you can press the *Show results* button to view the lumped element parameters of the equivalent network model.

To check the quality of the approximation, an S-parameter calculation using SPICE can be performed and the results of this calculation can be compared with the results of the 3D analysis. The difference between these two results then is a direct measure of the error of the approximation. This common task is simplified by a macro which is installed as global macro by default.

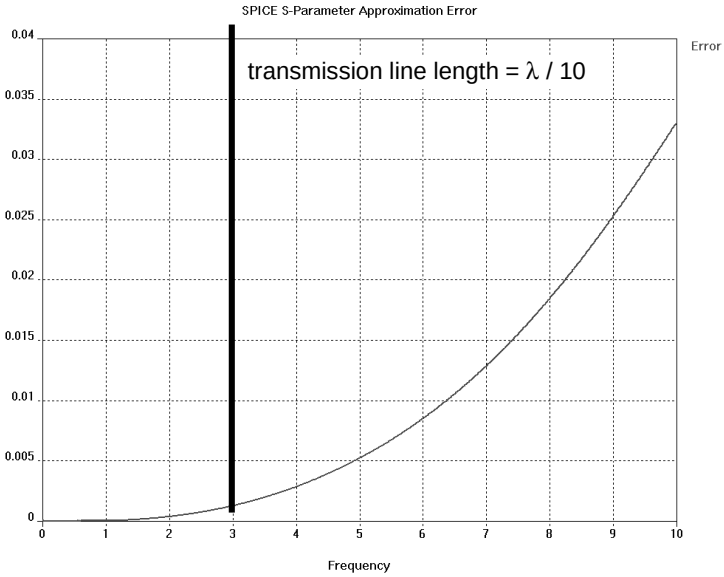
To use this macro you will first need to generate a SPICE netlist by clicking on the *Create netlist* button. Please save the netlist with the same name as your structure and afterwards close the parameter extraction dialog box by pressing *Ok*.

The macro can then be called by selecting *Macros⇨Results / Calculate SPICE S-Parameters* which opens the following dialog box:



Please check the *Netlist file name* before pressing the *Ok* button. Afterwards SPICE will be called automatically before a dialog box appears reporting the successful calculation of the S-parameters.

As a result, a new folder named *1D Results⇨SPICE* has been created containing the SPICE calculated S-parameters and the error curve which shows the maximum error between the network approximation and the real 3D device.



This result shows that the approximation has an error of less than 0.15 percent up to a transmission line length of a tenth of a wave length. The error increases with frequency since the approximation of the 3D structure by a transmission line model becomes less accurate for higher frequencies. Nevertheless even for a transmission line length of more than a third of a wave length, the approximation error remains roughly below 2.5 percent.

The example outlined above should have given you a short introduction into the network parameter extraction feature. Please refer to the online documentation for details and find more examples in the installation directory of CST MICROWAVE STUDIO®.

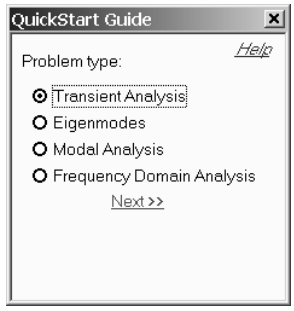
Chapter 4 — Finding Further Information

After carefully reading this manual, you will already have some idea of how to efficiently use CST MICROWAVE STUDIO® for your own problems. However, when you are creating your own first models, a lot of questions will arise. In this chapter we will give you a short overview of the available documentation.

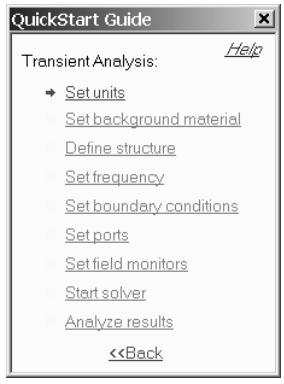
The Quick Start Guide

The main task of the Quick Start Guide is to remind you to complete all necessary steps in order to successfully perform a simulation. Especially for new users – or for those rarely using the software – it may be helpful to have some assistance.

After starting the Quick Start Guide, a dialog box opens in which you can specify the type of problem you wish to analyze:



After the problem type has been selected, you should press the *Next* item in order to proceed to a list of tasks which are either necessary or optional (as indicated) in order to perform a simulation. The following picture shows an example for transient analysis:



You will find that only the very first item on the list is active at the beginning. If you successfully perform the operation indicated by this entry, the next item will become active, and so on. You may, however, change any of your previous settings throughout the procedure.

The Quick Start Guide may be opened as soon as CST MICROWAVE STUDIO® is started. However the Quick Start Guide, will open automatically only when it has been used during the last session. You may start the Quick Start Guide at any time by choosing *Help⇨Quick Start* from the menu bar.

In order to access some information about the Quick Start Guide itself, you may press the *Help* button. To obtain more information about a particular operation, please click on the appropriate item in the Quick Start Guide.

Tutorials

The tutorials will generally be your best source of information when trying to solve a particular problem. We recommend that you browse through the list of all available tutorials and choose the one which is closest to your application:

Magic Tee
A typical wave guide structure. The S-parameter simulation is performed using the transient simulator.

Coaxial Connector
An S-parameter example for a coaxial structure using the transient simulator.

Microstrip Phase Bridge
A common S-parameter example for micro strip and coplanar structures using the transient simulator.

Patch Antenna
This tutorial consists of two parts: A single patch antenna and a four-patch antenna array for which the farfield characteristics are computed for various excitation patterns. Both simulations are carried out by using the transient simulator.

Cavity
This is a typical eigenmode example. The first few modal distributions are calculated by using the eigenmode solver.

Narrow Band Filter
This example shows an S-parameter simulation for a narrow band filter using the modal analysis solver. For highly resonating structures, it often is advantageous to use the modal analysis rather than the transient solver.

The fastest way to solve your particular problem is to study the most appropriate tutorial carefully, understanding the basic concepts before you start modeling your own problem. All tutorials are designed to keep the time required to complete the tutorial to a minimum.

If you are already familiar with CST MICROWAVE STUDIO® (it usually takes a couple of days), it may be no longer necessary to study the tutorials in detail. In this case you could quickly step through the pages of the tutorial and pick out the new information.

Examples

The installation directory of CST MICROWAVE STUDIO® contains an examples subdirectory which contains a couple of typical application examples.

Each of these examples also contains a “Readme” item in the navigation tree. By clicking on these items, you will obtain some information about the particular example regarding structure modeling and simulation procedure.

Although these examples are not explained in as much detail as the tutorials, they may nevertheless contain helpful hints which can be transferred to your particular application.

Online Reference Documentation

You can access the help system’s overview page at any time by choosing *Help⇨Contents* from the menu bar.

In each of the dialog boxes there is a specific *Help* button which will directly open the corresponding manual page. Additionally the *F1* key gives some context sensitive help when a particular mode is active. For instance, by pressing the *F1* key while a basic shape generation mode is active, you will obtain some information about the definition of shapes and possible actions.

Whenever no specific information is available, pressing the *F1* key will open an overview page from which you may navigate through the help system.

Referring to the *Advanced Topics* Manual

This printed manual contains some more background information about the methods used in CST MICROWAVE STUDIO® together with some more detailed information about advanced topics such as:

- Advanced mesh generation
- How to improve the calculation speed?
- Introduction to the powerful macro language to automate common tasks
- Installation of network licenses
- and many more…

Access Technical Support

After you have taken your first steps solving your own applications within CST MICROWAVE STUDIO®, please save your project and send the “.mod” and “.par” files to the technical support team. Even if you have successfully obtained a solution, the problem specification might still be improved in order to get even better results within shorter calculation times.

Macro Language Documentation

More information concerning the built-in macro language can be accessed through the *Help⇨VBA Macro Language* overview page. The macro language's documentation consists of four parts:

1. An overview and a general description of the macro language.

2. A description of all CST MICROWAVE STUDIO® specific macro language extensions.

3. A syntax reference of the Visual Basic for Applications compatible macro language.

4. An introduction to the integrated development environment (IDE) which allows you to build up your own complex macros.

History of Changes

The history of changes between several releases of the program can be opened by selecting *Help⇨History of Changes*. Since there are many new features in each new version, you should browse through the list even if you are already familiar with one of the previous releases.

Appendix A — List of Shortcut Keys

The following list gives an overview of available shortcut keys which may be very useful especially for advanced users.

General Shortcut Keys Available in Main Structure View

This view may be activated by clicking on it with the left mouse button.

ESC	Cancel currently active mode
Ctrl+O	Open new model file
Ctrl+S	Save current model file
DELETE	Delete the currently selected object
F1	Opens context sensitive help
F2	Rename the currently selected shape in the navigation tree
F5	Update the visualized 1D result (only while solver is running)

Ctrl+F	Reset view
Space	Reset view to structure
Shift+Space	Reset view to selection
Alt+V	Open view options dialog box
Ctrl+C	Activate/Deactivate cutting plane
Alt+O	Toggle outline mode
Alt+W	Toggle working plane visualization on or off
Ctrl+A	Toggle axis view on or off
Ctrl+W	Toggle wireframe mode on or off
Shift+A	Toggle field plot animation on or off

TAB	Open the numerical coordinate input box (also available in 1D plots)
Shift+TAB	Open the numerical coordinate input box with zero defaults

Numpad-(5)	Front view
Numpad-(4)	Left view
Numpad-(6)	Right view
Numpad-(8)	Top view
Numpad-(2)	Bottom view
Numpad-(0)	Perspective view

Cursor-Left	Decrement phase (2D/3D plots), move axis marker left (1D plots)
Cursor-Right	Increment phase (2D/3D plots), move axis marker right (1D plots)
Cursor-Up	Move cutplane or meshplane in positive normal direction
Cursor-Down	Move cutplane or meshplane in opposite normal direction

Ctrl+H	Hide selected shape, face or object
Ctrl+U	Unhide all

W	Align the WCS with most recently picked point(s), edge or face
Shift+U	Rotate the WCS around its u axis
Shift+V	Rotate the WCS around its v axis
Shift+W	Rotate the WCS around its w axis
P	Pick point
M	Pick edge mid point
A	Pick face center
R	Pick point on circle
C	Pick circle center

E	Pick edge
F	Pick face
Shift+E	Pick edge chain
Shift+F	Pick face chain
D	Clear picks

Ctrl+R	Remove the selected feature
Alt+U	Delete the selected face
Alt+C	Cover the selected edges
Alt+L	Add linear curve between two picked points
Alt+T	Add tangent curves between two picked points and edges
Alt+P	Split the selected edge at the picked point

Backspace	Delete previous point in generation of basic shapes.
+	Start Boolean add operation for selected shape
-	Start Boolean subtract operation for selected shape
*	Start Boolean intersect operation for selected shape, start trim curves operation for selected curve
÷	Start Boolean insert operation for selected shape
%	Start Boolean imprint operation for selected shape
RETURN	Perform Boolean operation (if active)

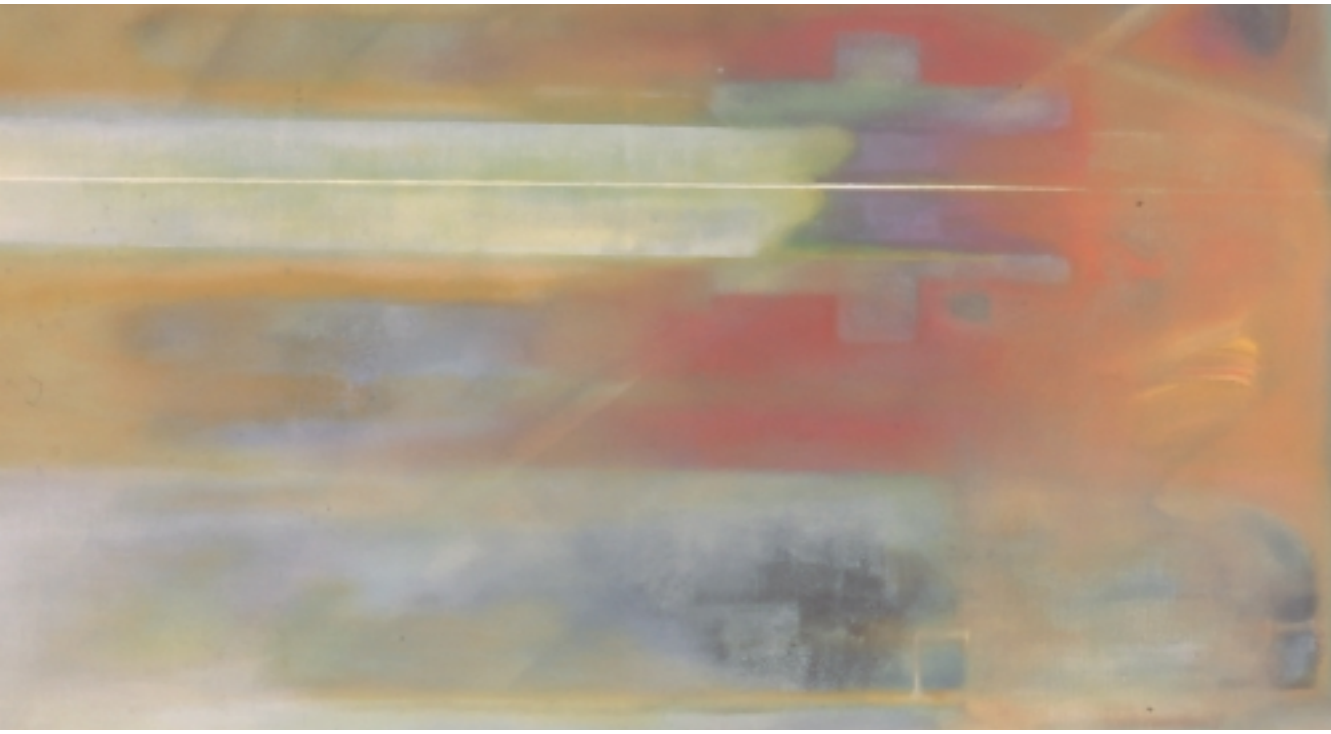
Mouse Wheel	Dynamic zoom view

The following shortcuts are active when the mouse is dragged while the left mouse button is pressed:

Shift	Restrict mouse movement to 90 degree angles (in shape creation) or Planar rotate view (otherwise)
Ctrl	Rotate view
Shift+Ctrl	Pan view

Shortcut Keys Available in Edit Fields

Ctrl+C	Copy selected text into clipboard
Ctrl+V	Paste clipboard to current marker's position
Ctrl+X	Cut selected text
Ctrl+Z	Undo last editing operation
Ctrl+Y	Redo previously undone operation (BASIC editor window only)
F1	Context help for the word next to the caret position (BASIC editor window only).



WWW.CST.DE

CST
COMPUTER SIMULATION TECHNOLOGY