



CST MICROWAVE STUDIO®

3D EM FOR HIGH FREQUENCIES



GETTING STARTED

C S T S T U D I O S U I T E ™ 2 0 0 6

Copyright

© 1998-2005
CST GmbH – Computer Simulation Technology
All rights reserved.

Information in this document is subject to change without notice. The software described in this document is furnished under a license agreement or non-disclosure agreement. The software may be used only in accordance with the terms of those agreements.



No part of this documentation may be reproduced, stored in a retrieval system, or transmitted in any form or any means electronic or mechanical, including photocopying and recording for any purpose other than the purchaser's personal use without the written permission of CST.

Trademarks

CST MICROWAVE STUDIO, CST DESIGN ENVIRONMENT, CST EM STUDIO, CST PARTICLE STUDIO, CST DESIGN STUDIO are trademarks or registered trademarks of CST GmbH.

Other brands and their products are trademarks or registered trademarks of their respective holders and should be noted as such.

Contents

September 14th, 2005

CHAPTER 1 — INTRODUCTION	3
Welcome.....	3
How to Get Started Quickly	3
What is CST MICROWAVE STUDIO®?	3
Who Uses CST MICROWAVE STUDIO®?	5
CST MICROWAVE STUDIO® Key Features	5
General.....	5
Structure Modeling	5
Transient Simulator.....	6
Frequency Domain Simulator	7
Eigenmode Simulator	8
Schematic View	8
Visualization and Secondary Result Calculation.....	8
Result Export	9
Automation	9
About This Manual.....	9
Document Conventions	9
Your Feedback	9
CHAPTER 2 — QUICK TOUR.....	11
Start the Software.....	11
Overview of the User Interface Structure.....	12
Create and View Some Simple Structures.....	14
Create a First “Brick”	14
An Overview of the Basic Shapes Available	16
Select Previously Defined Shapes, Group Shapes into Components and Assign Material Properties	17
Change the View	21
Apply Geometric Transformations	23
Combine Shapes Using Boolean Operations	25
Pick Points, Edges or Faces from within the Model.....	27
Chamfer and Blend Edges.....	29
Local Coordinate Systems.....	36
The History List.....	39
The History Tree	41
Curve Creation	43
Trace Creation	46
Bondwire Creation	48
Local Modifications	50
The First Real World Application Example	52
The Structure	52
Start CST MICROWAVE STUDIO®	53
Open the Quick Start Guide.....	54
Define the Units	55
Define the Background Material	55
Model the Structure	55
Define the Frequency Range	62
Define Ports	63
Define Boundary and Symmetry Conditions	65
Visualize the Mesh	67
Start the Simulation	68
Analyze the Port Modes.....	70
Analyze the S-Parameters.....	72

Adaptive Mesh Refinement.....	74
Analyze the Electromagnetic Field at Various Frequencies.....	77
Parameterization of the Model.....	82
Parameter Sweeps and Processing of Parametric Result Data.....	88
Automatic Optimization of the Structure	95
Comparison of Time and Frequency Domain Solver Results	99
Summary	102
Which Solver to Use	103
General Purpose Frequency Domain Computations	106
Resonant Frequency Domain Computations	111
Resonant: Fast S-Parameter	111
Resonant: S-Parameter, fields.....	112
Eigenmode (Resonator) Computations	115
Choose the Right Port	119
Discrete Ports	120
Waveguide Ports	121
Empty Waveguides.....	121
Coaxial Waveguides.....	124
Microstrip Lines	126
Coplanar Lines	130
Multipin Waveguides.....	136
Handle Multipin Waveguides Using Standard Ports	141
Inhomogeneous Waveguides	142
Antenna Computations	144
Simplifying Antenna Farfield Calculations.....	147
Digital Calculations	148
Add Circuit Elements to External Ports.....	151
SPICE Network Model Extraction	157
Model Order Reduction Based Network Parameter Extraction.....	157
Transmission Line Based Network Parameter Extraction.....	161
Cadence® Allegro / APD Plug in	165
Installation	165
Exporting Models from Cadence® Allegro / APD	167
Export Full Design	169
Export Area.....	170
Export Selected Nets	172
Export Custom Selected Objects	174
Agilent ADS® Co-Simulation	177
Design Kit Installation	177
Static Link between CST MICROWAVE STUDIO® and Agilent ADS®	179
CST MICROWAVE STUDIO® and Agilent ADS® Co-Simulation.....	180
CHAPTER 3 — FINDING FURTHER INFORMATION	185
The Quick Start Guide	185
Tutorials.....	186
Examples.....	187
Online Reference Documentation.....	187
Referring to the <i>Advanced Topics</i> Manual.....	187
Technical Support.....	188
Macro Language Documentation.....	188
History of Changes	188
APPENDIX A — LIST OF SHORTCUT KEYS	189
General Shortcut Keys Available in Main Structure View	189
Shortcut Keys Available in Edit Fields	190
Shortcut Keys Available in VBA Editor	191

Chapter 1 — Introduction

Welcome

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

CST MICROWAVE STUDIO® is embedded into the CST DESIGN ENVIRONMENT™. Please refer to the CST DESIGN ENVIRONMENT™ First Steps manual first. The following explanations assume that you already installed the software and familiarized yourself with the basic concepts of the user interface.

How to Get Started Quickly

We recommend that you proceed as follows:

1. Read the CST DESIGN ENVIRONMENT™ First Steps manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. Work through the tutorials by choosing the example which best suits your needs.
4. 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.
5. Start with your own first example. Choose a reasonably simple example, which will allow you to become familiar with the software quickly.
6. After you have worked through your first example, contact technical support for hints on possible improvements to achieve 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 ACIS modeling kernel. Strong graphic feedback simplifies the definition of your device even further. After the component has been modeled, a fully automatic meshing procedure is applied before a simulation engine is started.

A key feature of CST MICROWAVE STUDIO® is the *Method on Demand™* approach which allows using the simulator or mesh type that is best suited to a particular problem.

All simulators support hexahedral grids in combination with the Perfect Boundary Approximation (PBA® method). Some solvers also feature the Thin Sheet Technique (TST™) extension. Applying these highly advanced techniques normally increases the accuracy of the simulation substantially in comparison to conventional simulators. Since no method works equally well in all application domains, the software contains three different simulation techniques (transient solver, frequency domain solver, eigenmode solver) to best fit their particular applications. The frequency domain solver also contains specialized methods for analyzing highly resonant structures such as filters. Furthermore, the frequency domain solver supports both hexahedral and tetrahedral types of mesh.

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 step approach of many other simulators). This solver is remarkably efficient for most kinds of high frequency applications such as connectors, transmission lines, filters, antennae and more.

The transient solver is less efficient for electrically small structures that are much smaller than the shortest wavelength. In these cases it is advantageous to solve the problem by using the **frequency domain** solver. The frequency domain solver may also be the method of choice for narrow band problems such as filters or when the usage of tetrahedral grids is advantageous. Besides the general purpose solver (supporting hexahedral and tetrahedral grids), the frequency domain solver also contains fast alternatives for the calculation of S-parameters for strongly resonating structures. Please note that the latter solvers are currently available for hexahedral grids only.

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

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

Each solver's simulation results can 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®?

Anyone who has to deal with electromagnetic problems in the high frequency range should use CST MICROWAVE STUDIO®. The program is especially suited to the fast, efficient analysis and design of components like antennae (including arrays), filters, transmission lines, couplers, connectors (single and multiple pin), printed circuit boards, resonators and many more. Since the underlying method is a general 3D approach, CST MICROWAVE STUDIO® can solve virtually any high frequency field problem.

The software is based on a method that requires the discretization of the entire calculation volume; the applications are therefore limited by the electrical size of the structures. An 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 roughly 100 wavelengths. For large antenna arrays, the frequency domain solver offers a very powerful Unit Cell feature where the periodicity of the structure is taken into account.

CST MICROWAVE STUDIO® Key Features

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

General

- Native graphical user interface based on Windows 2000 and Windows XP
- Fast and memory efficient F1-method
- Extremely good performance due to Perfect Boundary Approximation (PBA®) for solvers using hexahedral grids. The transient and eigenmode solvers also support the Thin Sheet Technique (TST™). Hexahedral grids are supported by all solvers.
- The structure can be viewed either as a 3D model or as a schematic. The latter allows for easy coupling of EM simulation with circuit simulation.

Structure Modeling

- Advanced ACIS¹-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®), Autodesk Inventor®, IGES, VDA-FS, STEP, ProE®, CATIA 4®, CATIA 5®, CoventorWare®, Mecadtron® or STL files
- Import of 2D CAD data by DXF, GDSII and Gerber RS274X, RS274D files
- Import of Agilent ADS® layouts
- Import of Sonnet em® models (8.5x)
- Import of a visible human model dataset or other voxel datasets
- Export of CAD data by SAT, IGES, STEP, STL, DXF, DRC or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem description

¹ Portions of this software are owned by Spatial Corp. © 1986 – 2005. All Rights Reserved.

Transient Simulator

- Efficient calculation for loss-free and lossy structures
- Broadband calculation of S-parameters from one single calculation run by applying DFTs 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
- Parallelization 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 (subsequently or simultaneously)
- Plane wave excitation (linear, circular or elliptical polarization)
- Excitation by a current distribution imported from SimLab®

- 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, voltages in time and frequency domain
- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.) with and without farfield approximation. Farfield probes to determine broad band farfield data at certain angles.
- 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 signals and signal database
- Simultaneous port excitation with different excitation signals for each port

- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and multiple port/mode excitations

Frequency Domain Simulator

- Efficient calculation for loss-free and lossy structures including lossy wave guide ports
- General purpose solver supports both hexahedral and tetrahedral meshes
- Isotropic and anisotropic material properties
- Arbitrary frequency dependent material properties
- Automatic fast broadband adaptive frequency sweep
- User defined frequency sweeps
- Adaptive mesh refinement in 3D
- Direct and iterative matrix solvers with convergence acceleration techniques
- 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 or scan angle
- Unit cell feature simplifies the simulation of periodic antenna arrays (tetrahedral mesh only)
- Floquet mode ports (periodic wave guide ports)
- 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
- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.) with and without farfield approximation
- Antenna array farfield calculation
- RCS calculation
- Calculation of SAR distributions
- Discrete elements (lumped resistors) as ports
- Lumped R, L, C elements at any location in the structure
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations and parameter sweeps
- Besides the general purpose solver, the frequency domain solver also contains two solvers specialized on strongly resonant structures (hexahedral meshes only). The first of these solvers calculates S-parameters only whereas the second also calculates fields with some additional calculation time, of course.

Eigenmode Simulator

- Calculation of modal field distributions in closed loss free or lossy structures
- Isotropic and anisotropic materials
- Parallelization using up to two processors on a PC
- Adaptive mesh refinement in 3D
- Periodic boundary conditions including phase shift
- Calculation of losses and internal / external Q-factors for each mode (directly or using perturbation method)
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations and parameter sweeps

Schematic View

- Allows for the connection of arbitrary networks to EM ports. These networks can contain any combination of R/L/C circuit elements, ideal phase shifters, perfect absorbers, variable reflections, directional couplers, 3dB splitters, CST MICROWAVE STUDIO® netlist files and ports.
- All circuit simulation capabilities licensed for CST DESIGN STUDIO™ can also be used within this schematic view.
- The schematic view and the 3D view are synchronized automatically.

Visualization and Secondary Result Calculation

- Displays S-parameters in xy-plots (linear or logarithmic scale)
- Displays S-parameters in smith charts and polar charts
- Online visualization of intermediate results during simulation
- Import and visualization of external xy-data
- Copy / paste of xy-datasets
- Fast access to parametric data via interactive tuning sliders
- Displays port modes (with propagation constant, impedance, etc.)
- Various field visualization options in 2D and 3D for electric fields, magnetic fields, power flows, surface currents, etc.
- Animation of field distributions
- Display of farfields (fields, gain, directivity, RCS) in xy-plots, polar plots, scattering maps and radiation plots (3D)
- Display and integration of 2D and 3D fields along arbitrary curves
- Integration of 3D fields across arbitrary faces
- Automatic extraction of SPICE network models for arbitrary topologies ensuring the passivity of the extracted circuits
- Combination of results from different port excitations
- Hierarchical result templates for automated extraction and visualization of arbitrary results from various simulation runs. These data can also be used for the definition of optimization goals.

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 (Microsoft Office®, MATLAB®, AutoCAD®, MathCAD®, Windows Scripting Host, etc.)

About This Manual

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

The main part of the manual is the 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 accessed through the main window menu are printed as follows: *menu bar item* \Rightarrow *menu item*. This means that you first should click 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 clicked 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 while 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 your local support center. If you don't know how to contact the support center near you, send an email to info@cst.com.

Chapter 2 — Quick Tour

CST MICROWAVE STUDIO® is designed for ease of use. However, to get started quickly, you need to know a couple of things. The main purpose of this chapter is to provide an overview of the software's capabilities. 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:

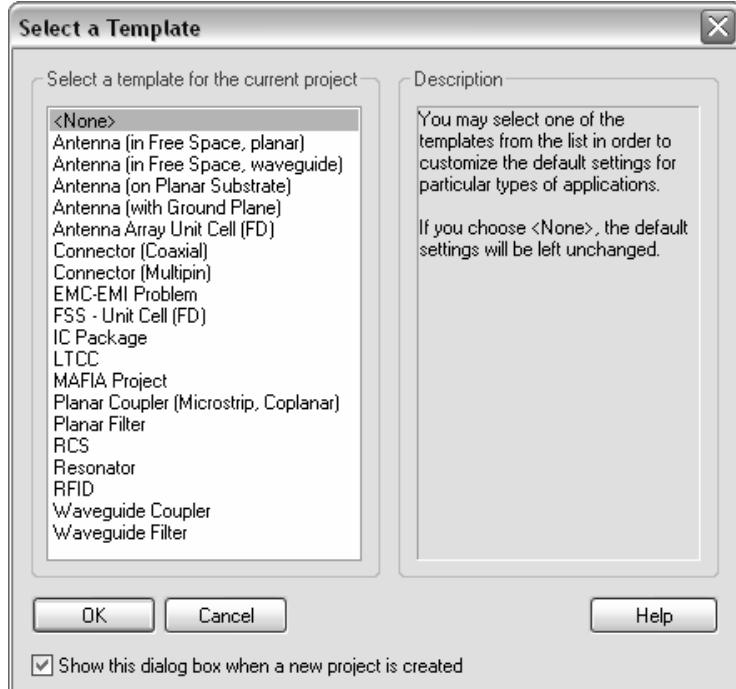
- Start the software
- Overview of the user interface structure
- Create and view some simple structures
- The first real world application example
- Which solver to use
- General purpose and resonant frequency domain computations
- Eigenmode (resonator) computations
- Choose the right port
- Antenna computations and digital calculations
- Add circuit elements to external ports
- SPICE network model extraction
- Cadence® Allegro / APD plug in and Agilent ADS® co-simulation

Start the Software

After starting CST DESIGN ENVIRONMENT™, you will be prompted to open an existing file or create a new project:



In this dialog box, select a CST MICROWAVE STUDIO® project and click the *OK* button. You can make this selection the default whenever the CST DESIGN ENVIRONMENT™ is started by checking the *Always start with the selected module option*. Once CST MICROWAVE STUDIO® is initialized, you will see the following window:



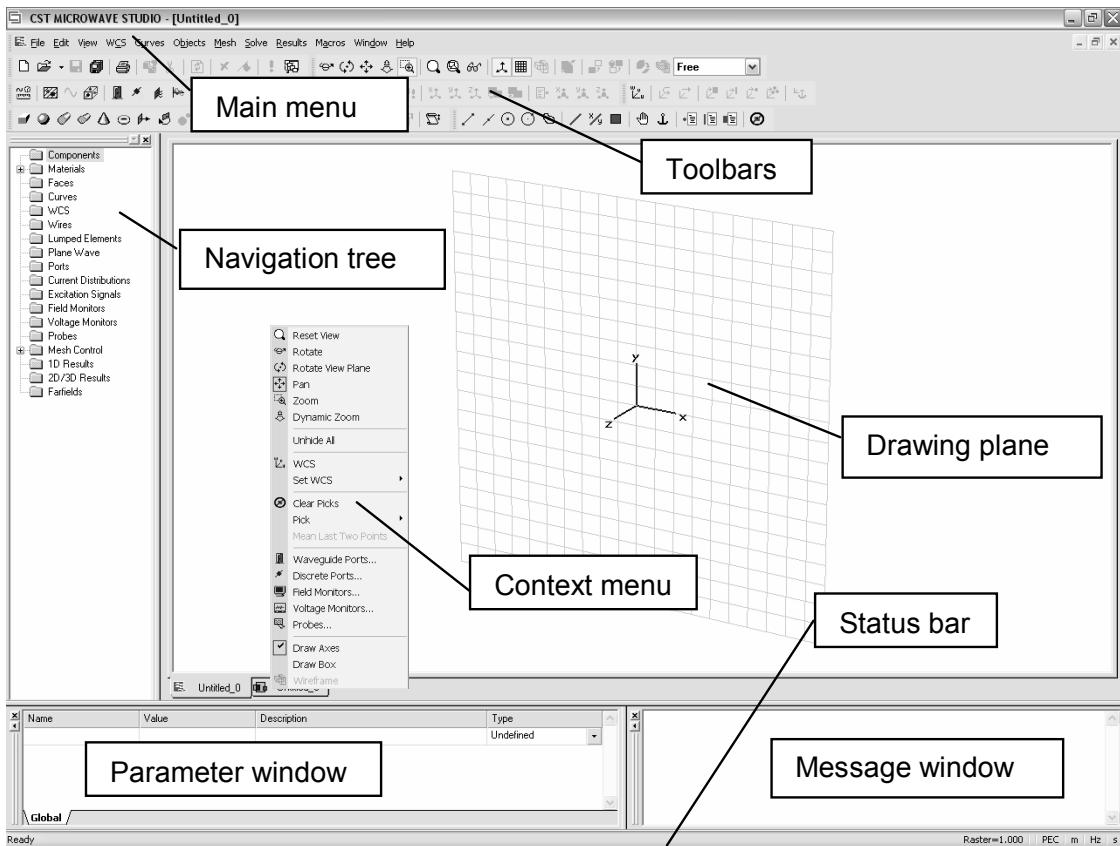
This dialog box will always appear when a new project is created.

Here you can select one of the predefined templates to automatically set proper default values for the particular type of device you want to analyze. Although all of these settings can be changed manually at any time, 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, simply select *<None>* and click the *OK* button.

Overview of the User Interface Structure

The following picture shows a screenshot of CST MICROWAVE STUDIO® as it is embedded in CST DESIGN ENVIRONMENT™. Refer to the CST DESIGN ENVIRONMENT™ documentation for more information on how to customize the user interface according to your needs.



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 explain the different items in this tree window.

The context menus are a flexible way of accessing 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 2D locator, even when defining 3D structures, the coordinates must be projected onto the drawing plane in order to specify a 3D 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 parameter window displays a list of all previously defined parameters together with their current values.

The message window displays information text (e.g. solver output) whenever applicable.

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

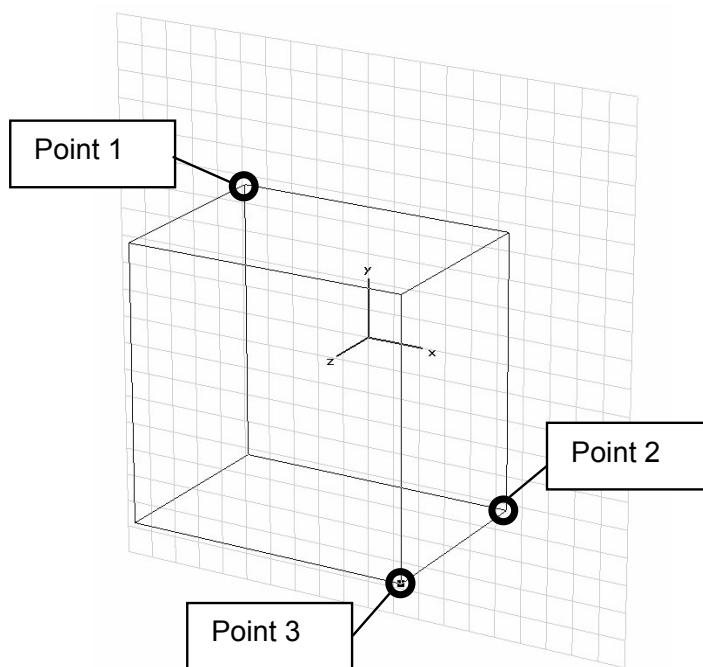
Create and View Some Simple Structures

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 draw such a primitive: a brick.

Create a First “Brick”

1. Activate the “Brick” tool by clicking the corresponding icon in the object toolbar:  (You can also select *Objects*⇒*Basic Shapes*⇒*Brick* from the main menu). 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. Next, 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 showing the numerical values of all coordinate locations you have entered. Click *OK* here to store the settings and create your first primitive!

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



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

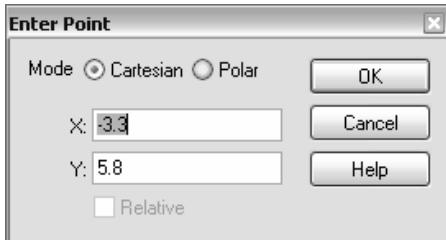
The simplest way to set a point is to double-click its location on the drawing plane as above. However, in most cases the structure coordinates have to be entered at a high precision. In this case, the snap-to-grid mode should be activated. You will find the corresponding option dialog box under *Edit*⇒*Working 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 influences only the size of the raster which is drawn on the screen. The coordinate mapping is independent from this setting.

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

Another way to specify a coordinate is to press 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 that appears when the first point of a shape must 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 always show the current mouse location in the entry fields. However, oftentimes a point should be set to 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 using the mouse at any time by pressing the *Esc* key. In this case, the shape dialog box will open immediately.

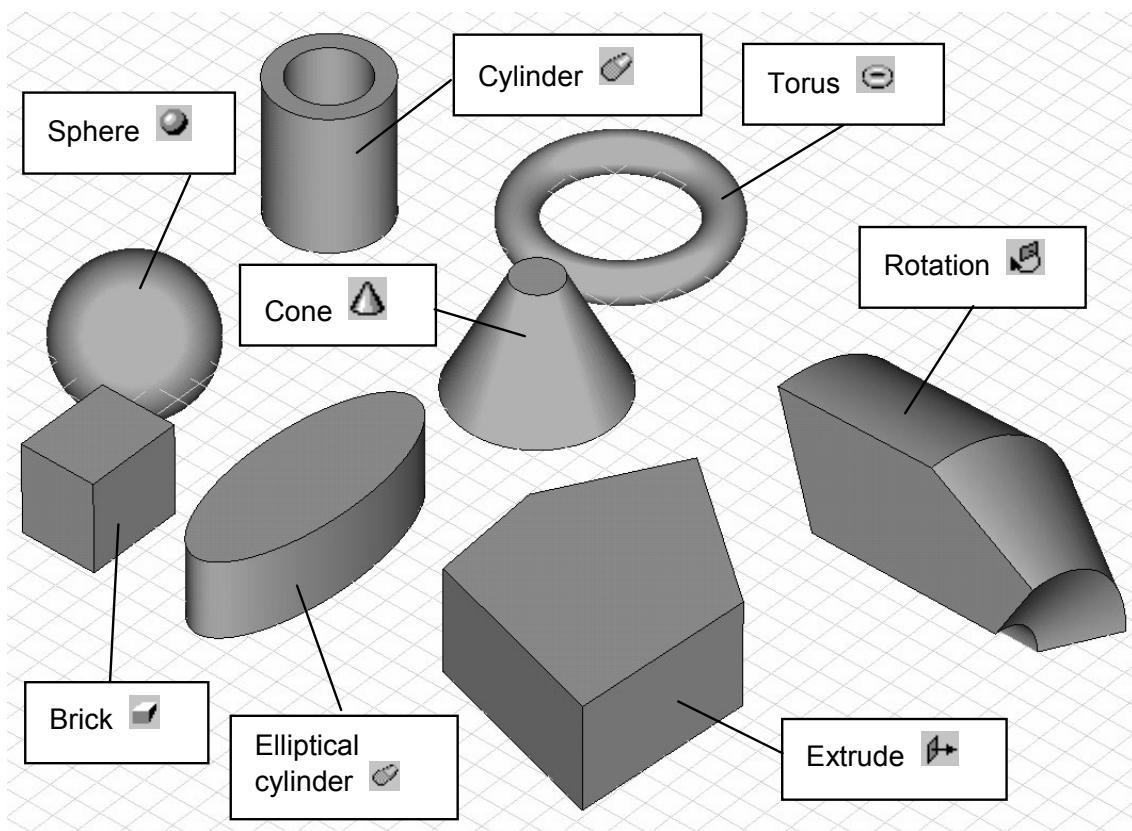
Pressing the *Esc* key twice aborts the shape generation. Pressing the *Backspace* key deletes the previously selected point. If no point has been selected, the shape generation will also be aborted.

Please note that another mode exists for the generation of bricks. When you are asked 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.

To facilitate this, a feature exists which allows the line definition to be restricted to orthogonal movements from the first selected point. Simply hold down the shift key and move the mouse to define the next point.

An Overview of the Basic Shapes Available

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

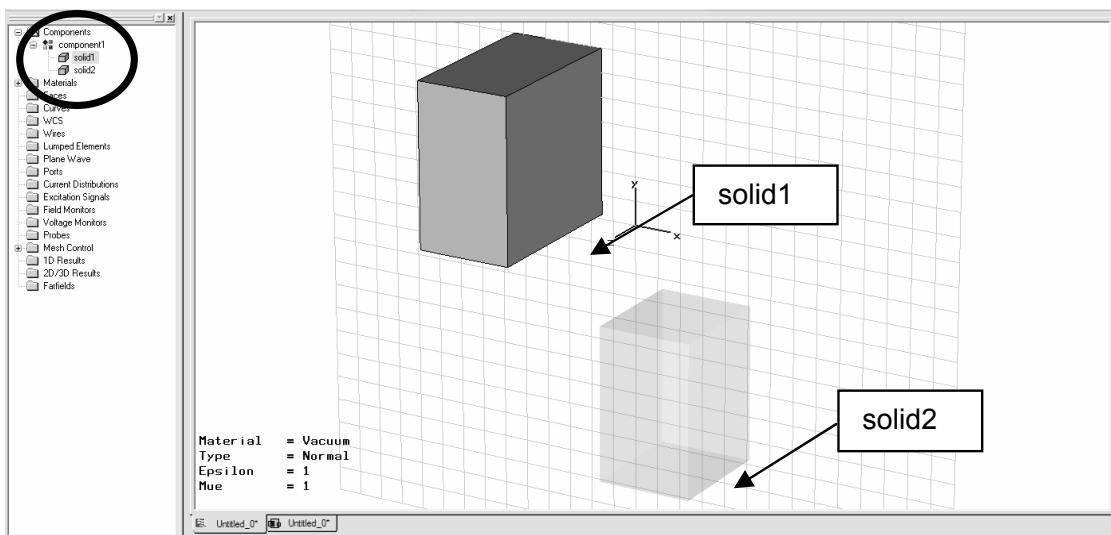


At this stage you should play around a bit with the shape generator to familiarize yourself with the user interface.

Select Previously Defined Shapes, Group Shapes into Components and Assign Material Properties

After a shape has been defined, it is automatically cataloged in the navigation tree. You can find all shapes in the *Components* folder. If you open this folder, you will find a subfolder called *component1*, which 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 clicking on the corresponding item in the navigation tree. Note that after you select a shape, it will be displayed opaquely while all others will be drawn transparently (see the picture below). This is how the modeler 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. Holding down the *Ctrl* key while double-clicking at a shape in the view allows you to select multiple shapes. You may also select ranges of shapes in the navigation tree by holding down the *Shift* key while clicking on the shapes’ names. Take a few seconds to familiarize yourself with the shape selection mechanism.

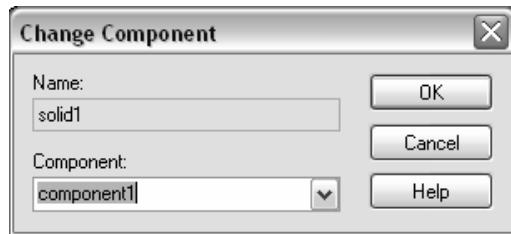


You may change the name of a shape by selecting it and choosing *Edit*⇒*Rename Object* from the menu bar or by pressing the *F2* key. You can then change the name of the shape by editing the item text in the navigation tree.

Now that we have discussed how to select an object, we should spend a little time on the grouping of shapes into components. Each component is a subfolder of the *Components* folder in the navigation tree. Each individual component folder can contain an arbitrary number of shapes.

The purpose of the component structure is to group together objects which belong to the same geometrical component, e.g. connectors, antennae, etc. This hierarchical grouping of shapes allows simplified operations on entire components such as transformations (including copying), deletions, etc.

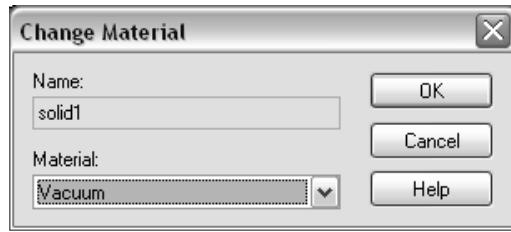
You can change the component assignment of a shape by selecting the shape and choosing *Edit*⇒*Change Component* from the menu bar (this option can also be found in the context menu when a shape is selected). The following dialog box will open:



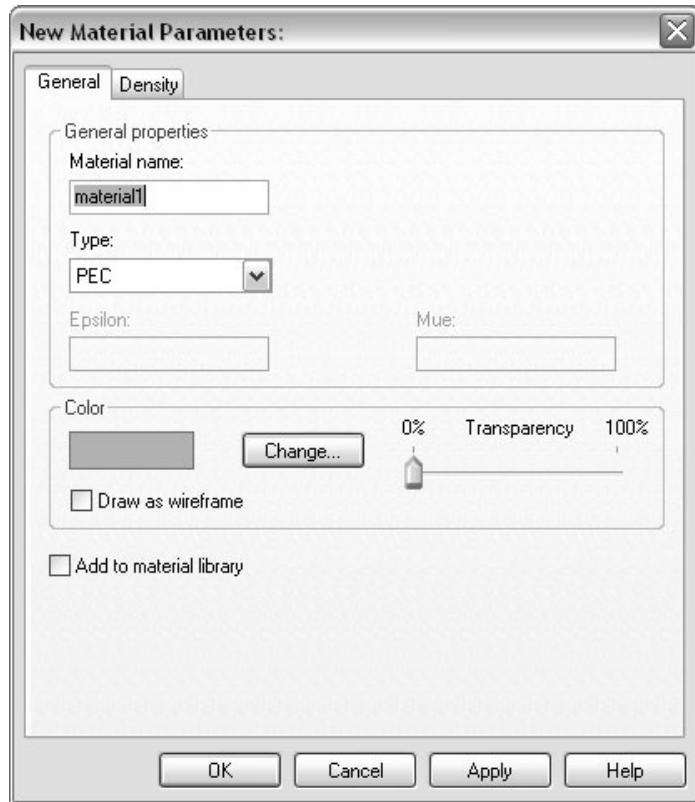
In this dialog box, you can select an existing component from the list or create a new one by simply typing its name in the entry field. You may also select [New Component] from the list. In the latter case, the newly created components will be automatically named as component1, component2, etc.

The component assignment of a shape has nothing to do with its physical material properties. In addition to its association to a particular component, each shape is assigned a material that also defines the color for the shape's visualization. In other words, the material properties (and colors) do not belong to the shapes directly but to the corresponding material. This means that all shapes made of a particular material 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 material by selecting the shape and choosing *Edit*⇒*Change Material* from the menu bar (this option can also be found in the context menu when a shape is selected). The following dialog box will open:

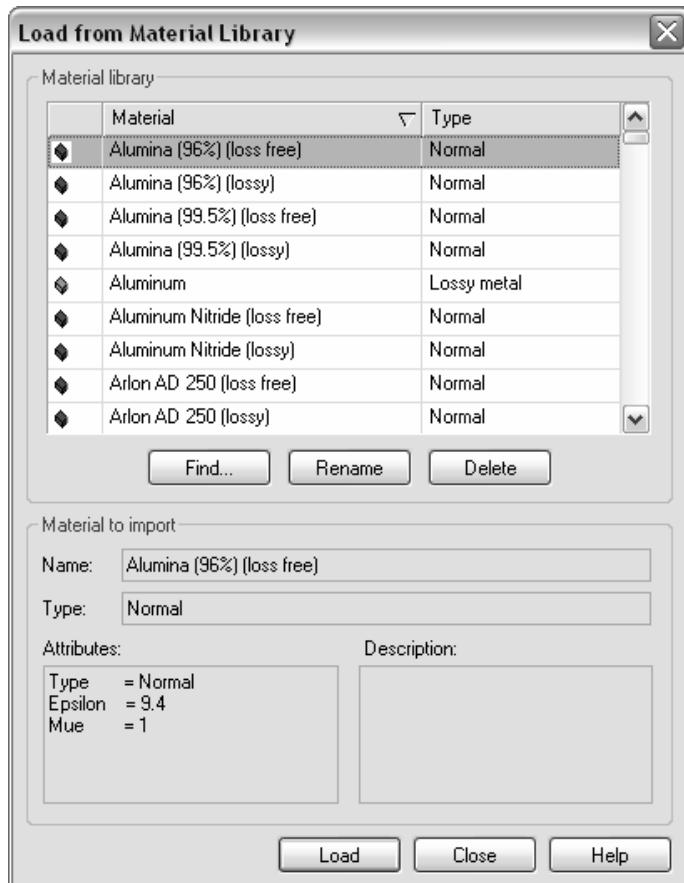


In this dialog box you may select an existing material from the list or define a new one by selecting the item [*New Material...*] from the list. In the latter case, another dialog box will open:



In this dialog box you must specify the *Material name* and the *Material type* (perfect conductor, normal dielectric, anisotropic, lossy metal, corrugated wall and ohmic sheet material). Note that the material types *corrugated wall* and *ohmic sheet* are available for the tetrahedral frequency domain solver only. You can also change the color of the material by clicking the *Change* button. After clicking the *OK* button, the new material will be stored and appears in the *Materials* folder in the navigation tree. Selecting a particular material in the navigation tree also highlights all shapes that belong to this material. All other shapes will then be drawn transparently.

In order to simplify the definition of frequently used materials, a material database has been implemented. Before a material definition from the database can be used, you must add it to the current project by selecting *Solve*⇒*Materials*⇒*Load from Material Library*. This operation will open the following dialog box displaying the contents of the database:

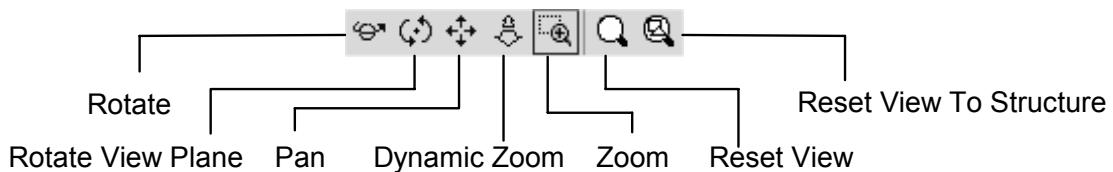


You may select an existing material from the list and click the *Load* button to add the material definition to the Materials folder in the navigation tree. Once the material is available in this folder, it can be used in the current project. You can also add a material that has been defined in the current project to the database by selecting the material in the navigation tree and then choosing *Solve*⇒*Materials*⇒*Add to Material Library*.

Change the View

So far we have created and viewed the shapes by using the default view. You can change the view at any time (even during shape generation) using some simple commands as explained below.

The view will change whenever you drag the mouse while holding down the left button, according to the selected mode. You can select the mode from the main menu by choosing *View*⇒*Mode*⇒*Rotate/Plane/Pan/Zoom/Dynamic Zoom* or by selecting the appropriate item from the view toolbar:



The mode setting affects 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 upward will decrease the zoom factor while moving the mouse downward will increase the zoom factor.
- Zoom**: In this mode a rubber band rectangle will be defined by dragging the mouse. After you release 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 ends when you release the left mouse button. You can reset the zoom factor by choosing *View*⇒*Reset View* from the main menu or from the context menus. Alternatively you can select 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. Furthermore, pressing *Shift+Space* will zoom to the currently selected shape rather than the entire structure.

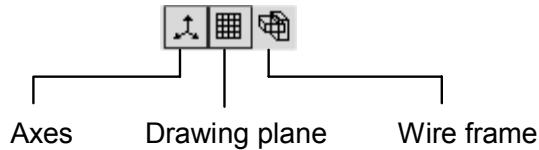
Since changing the view is a frequently used operation that will sometimes be necessary even during the process of interactive shape creation, some useful shortcut keys exist. Press the appropriate keys, and drag the mouse while pressing the left button:

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

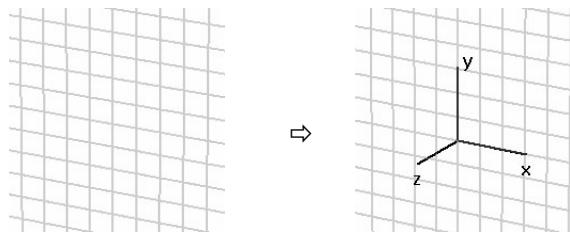
A mouse wheel has the same effect as the "dynamic zoom."

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

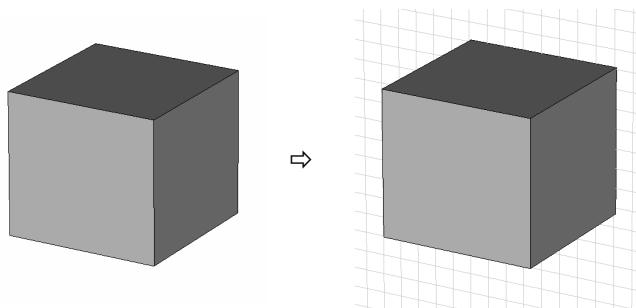
All these settings can be specified by choosing the appropriate item from the *View* menu. Furthermore these settings can be specified with the corresponding item in the view toolbar:



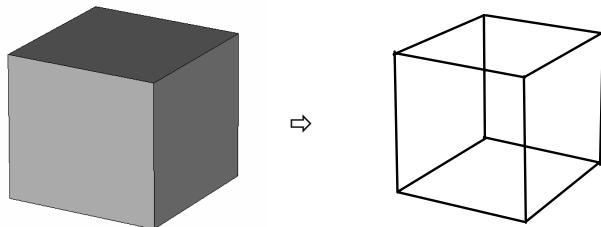
Axes (*View*⇒*View Options* dialog box, *Ctrl+A*): This option specifies whether or not the coordinate system is displayed:



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



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.



Apply 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 is geometric transformations.

We assume that you have already selected the shape (or multiple shapes) to which a transformation will be applied (e.g. by double-clicking on a shape in the main view).

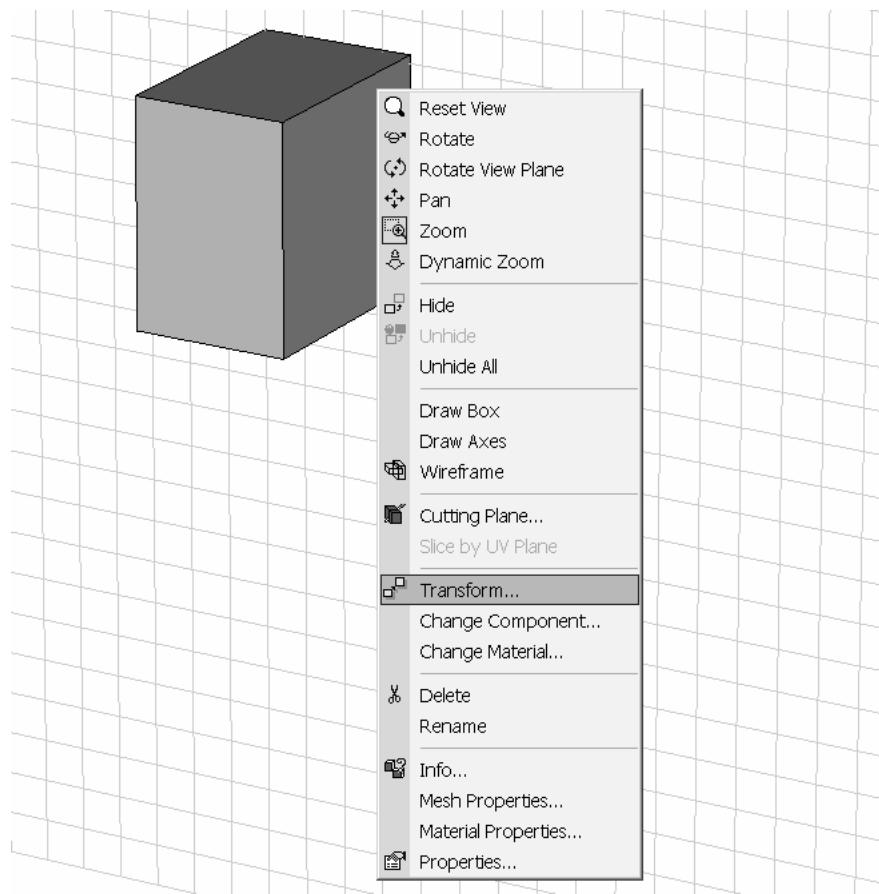
You can then open the transformation dialog box by choosing *Objects*⇒*Transform Shape* in the main menu, by choosing the item *Transform* from the context menu, or by selecting  in the objects toolbar.

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

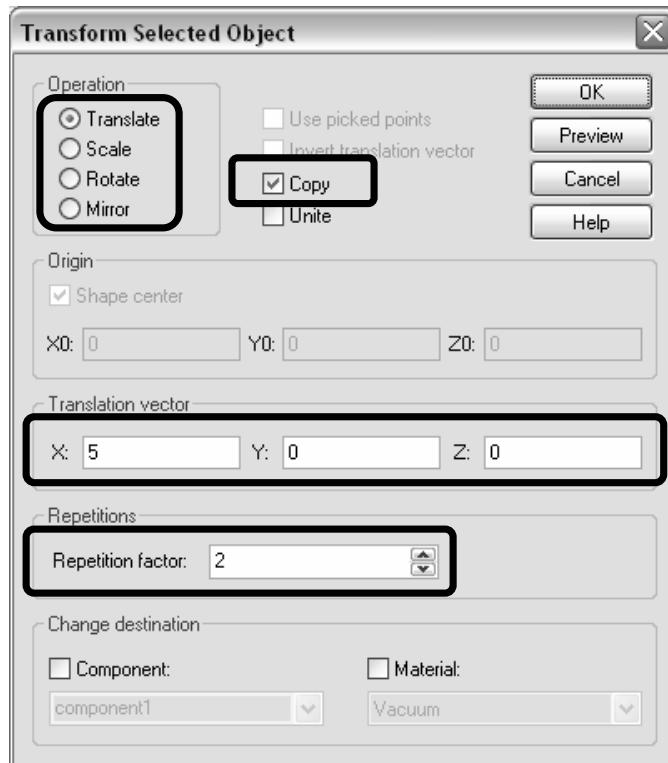
- Translate:** This transformation applies a translation vector to the selected shape.
- Scale:** By choosing this transformation, you can scale the shape along the coordinate axes. You may specify different scaling factors in the different coordinate directions.
- Rotate:** This transformation applies 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 be the center of the shape (calculated automatically) or any specified point. Specify the rotation angle and axis settings 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 (*Copy* option) or deleted. Furthermore you can specify in the *Repetition factor* field how many times the same transformation will be applied to the shape (each time producing a new shape when the *Copy* option is active).

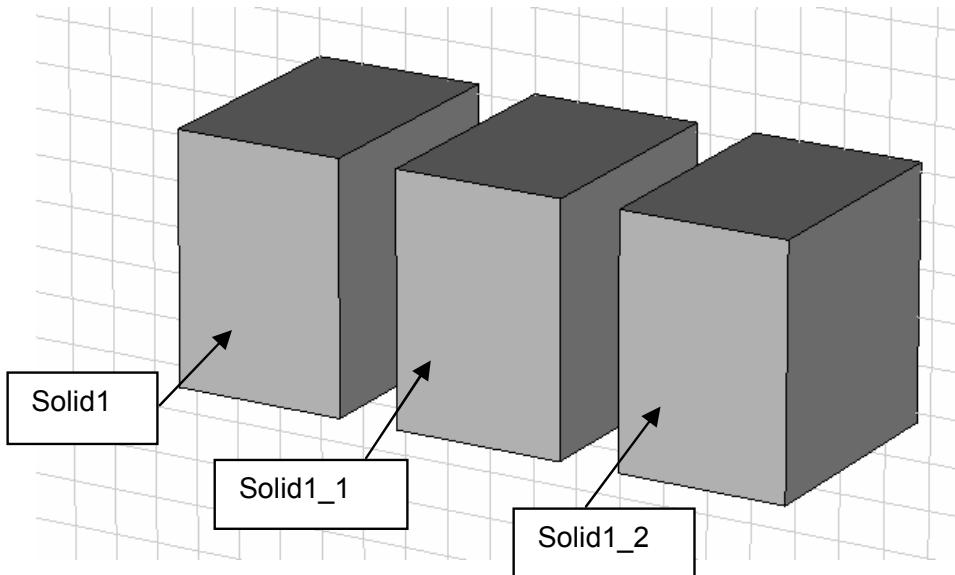
A final example will demonstrate the usage of the transformation dialog box. Assume a brick has been defined and selected as depicted below. Open the transform dialog box by choosing the appropriate item from the context menu (or *Objects*⇒*Transform*).



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



You should obtain the following shapes:

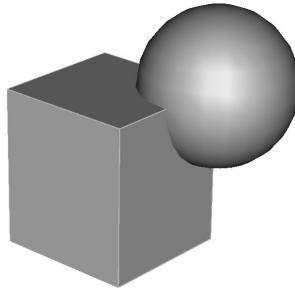


Note that for each transformation the name of the transformed shape is either kept (no *Copy* option) or extended by appendices _1, _2, etc. in order to obtain unique names for the shapes.

Combine Shapes Using Boolean Operations

Probably the most powerful operation to create complex shapes is combining simple shapes using Boolean operations. These operations allow you to add shapes together, to subtract one or more shapes from another, to insert shapes into each other, and to intersect two or more shapes.

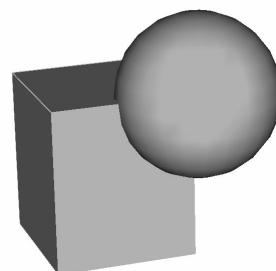
Let's consider two shapes – a sphere and a brick – on which to perform Boolean operations.



This list names all available Boolean operations and shows the resulting body for each combination:

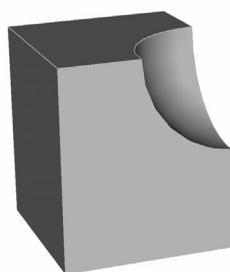
Add brick to sphere

Add both shapes together to obtain a single shape. The resulting shape will assume the component and material settings of the first shape.



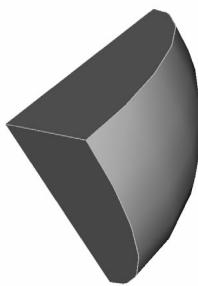
Subtract sphere from brick

Subtract the first shape from the second to obtain a single shape. The resulting shape will assume the component and material settings of the shape from which the other shape is subtracted.



Intersect brick and sphere

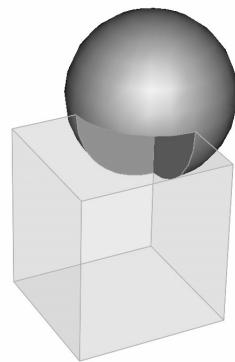
Intersect two shapes to form a single shape. The resulting shape will assume the component and material settings from the first shape of this operation.



Trim sphere

= Insert brick into sphere

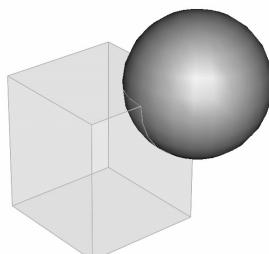
The first shape will be trimmed by the boundary of the second shape. Both shapes will be kept. The resulting shapes will have no intersecting volume.



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.



Note that not all of the Boolean operations above are directly accessible. As you can 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).

You can access the following Boolean operations from the main menu by choosing the corresponding items: *Objects*⇒*Boolean*⇒*Add/Subtract/Intersect/Insert*. Operations are accessible only 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 performs the Boolean combination. The result depends on the type of Boolean operation and is listed below:

- Add:** Add the second shape to the first one – keeps the component and material settings of the first shape.
- Subtract:** Subtract the second shape from the first one – keeps the component and material settings of the first shape.
- Intersect:** Intersect the first with the second shape – keeps the component and material settings of the first shape.
- Insert:** Insert the second shape into the first one – keeps both shapes while changing the first shape only.

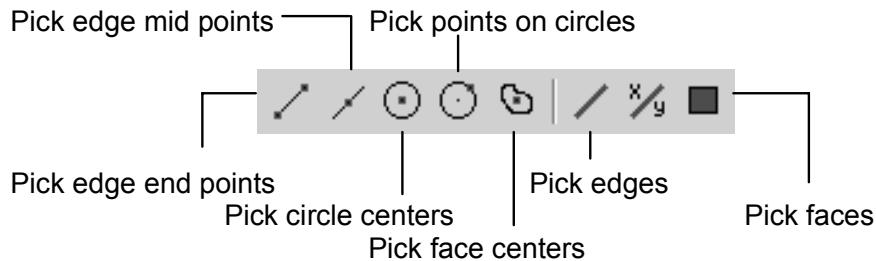
The trim operations are only available in a special “Shape intersection” dialog box which appears when a shape is created that intersects or touches areas with existing shapes. This dialog box will be explained later.

When multiple shapes are selected, you can access the Boolean add operation to unite all selected shapes. You can also select more than one shape when you are prompted to specify the *second* shape for Boolean subtract, intersect or insert operations.

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 the “pick operations,” you must first select the appropriate pick tool from either the menu item *Objects*⇒*Pick*⇒*Pick Point...* or from an item in the pick toolbar:



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

Note: You cannot 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. Note that if the  symbol is selected 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 which 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 parentheses next to the corresponding pick operation.

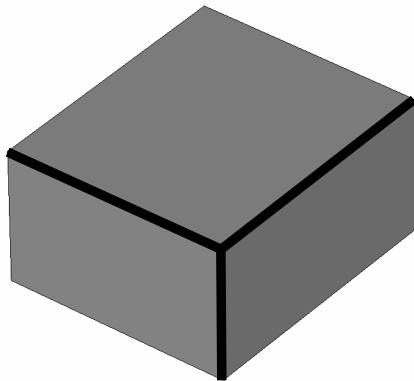
- Pick edge end points (P):** Double-click close to the end point of an edge. The corresponding point will then be selected.
- Pick edge mid points (M):** Double-click on an edge. The mid-point of this edge will then be selected.
- Pick circle centers (C):** Double-click on a circular edge. The center point of this edge will be selected. The edge need not necessarily belong to a complete circle.
- Pick points on circles (R):** Double-click on a circular edge. Afterward 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 to select it.
- Pick faces (F):** Double-click on a face of the model to select it.
- 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 in 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 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 you are requested to double-click in order to enter the next point, you may alternatively enter the pick mode. After leaving this mode, the picked point will be taken as the next point for the shape creation.

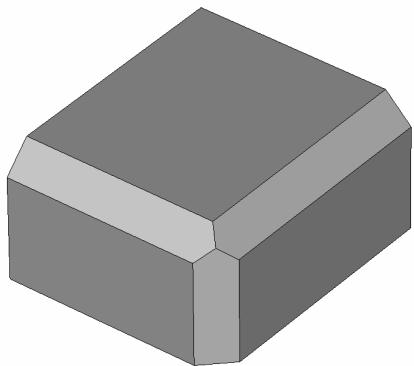
Previously picked points, edges or faces can be cleared by using the *Objects*⇒*Clear Picks* command (shortcut D in the main view) or by selecting  in the pick toolbar.

Chamfer and Blend Edges

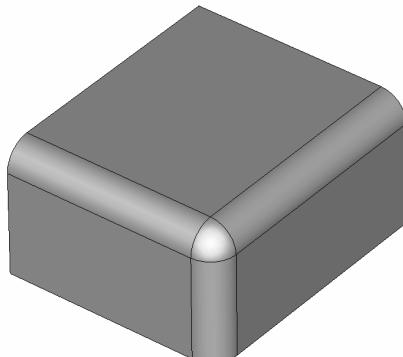
One of the most common applications for picked edges is the chamfer and blend edge operation. We assume you have created a brick and selected some of its edges, as shown in the following picture:



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

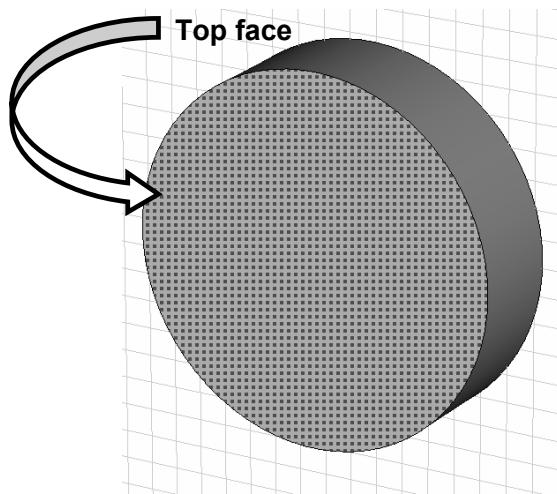


Alternatively you can perform a blend edges operation which is activated either by choosing *Objects*⇒*Blend Edges* in the main menu or by selecting  in the objects toolbar. In the following dialog box, you can specify the radius of the blend. The result should look similar to the following picture:

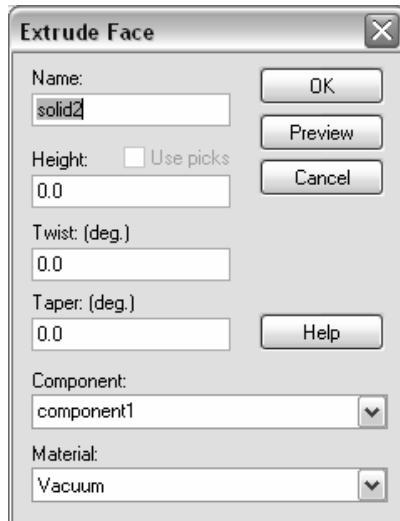


Extrude, Rotate and Loft Faces

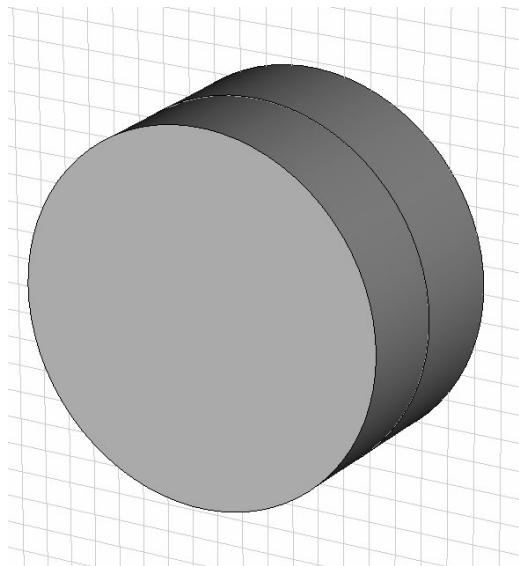
The chamfer and blend tools are common operations on picked edges; and the extrude, rotate and loft operations are equally typical construction tools for use on picked faces. In the following, we assume an existing cylinder with a picked top face:



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 opens immediately:

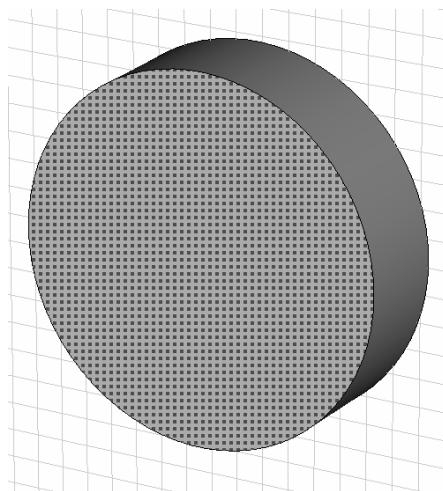


If no face is picked in advance, an interactive mode will be entered in which you can define polygon points for the extrusion profile. However, in this example you should enter a height and click the *OK* button. Finally, your structure should look as follows:

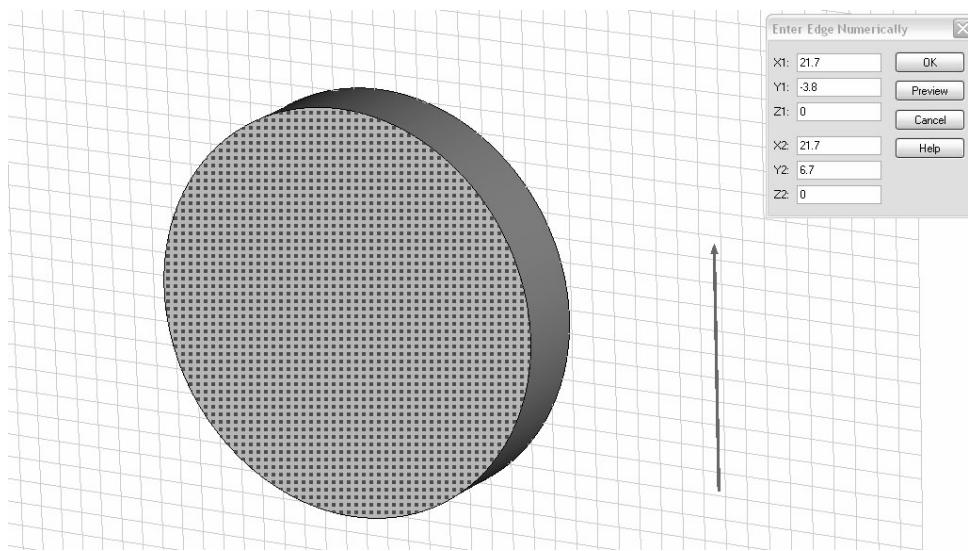


The extrusion tool has created a second shape by the extrusion of the picked face.

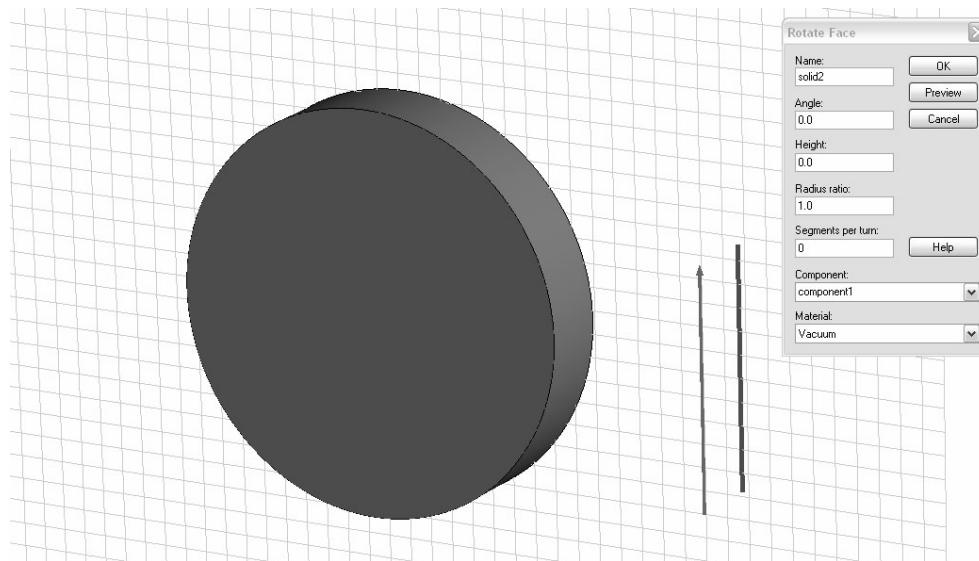
For the rotation, you should start with the same basic geometry as before:



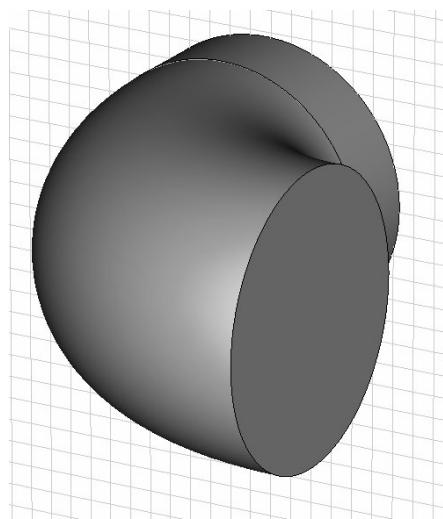
The rotation tool requires the input of both a rotation axis and a picked face. The rotation axis can be a linear edge picked from the model or a numerically specified edge. In this example, you should specify the edge by selecting the *Objects*⇒*Pick*⇒*Edge from Coordinates* () tool from the pick toolbar. Afterward 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, click the *OK* button to store the edge. Afterward you can activate the rotate face tool by selecting *Objects* \Rightarrow *Rotate*.

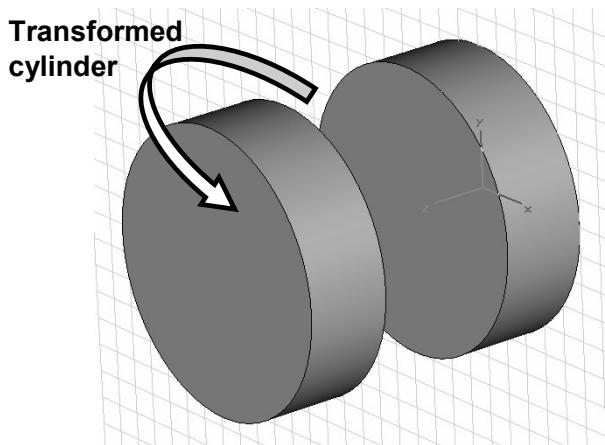


The previously selected rotation axis is automatically projected into the face's plane (blue vector), and the rotation tool dialog box opens immediately. In this dialog box, you can specify an *Angle* (e.g. 90 degrees) and click *OK*. The final shape should look as follows:

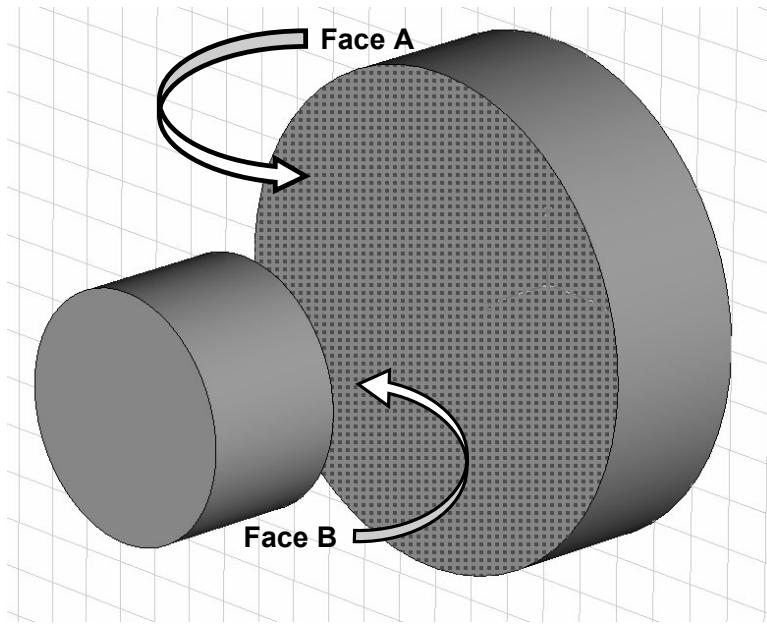


Note that the rotate tool enters an interactive polygon definition mode similar to the one in the extrude tool if no face is picked before the tool is activated.

One of the more advanced operations is generating lofts between picked faces. To practice, 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 *Copy* option:



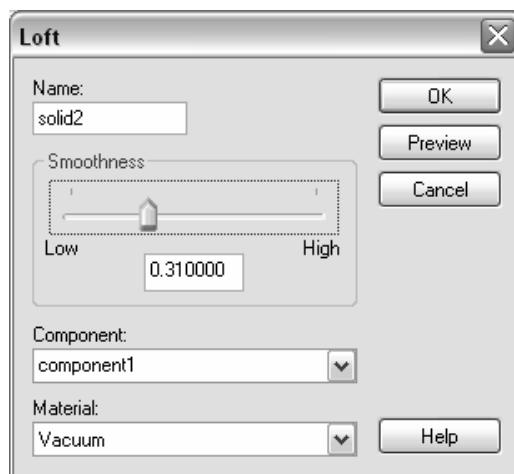
Next select the transformed cylinder and apply a scaling transformation to it by shrinking its size along the x and y axes by 0.5 while keeping the z-scale at 1.0:



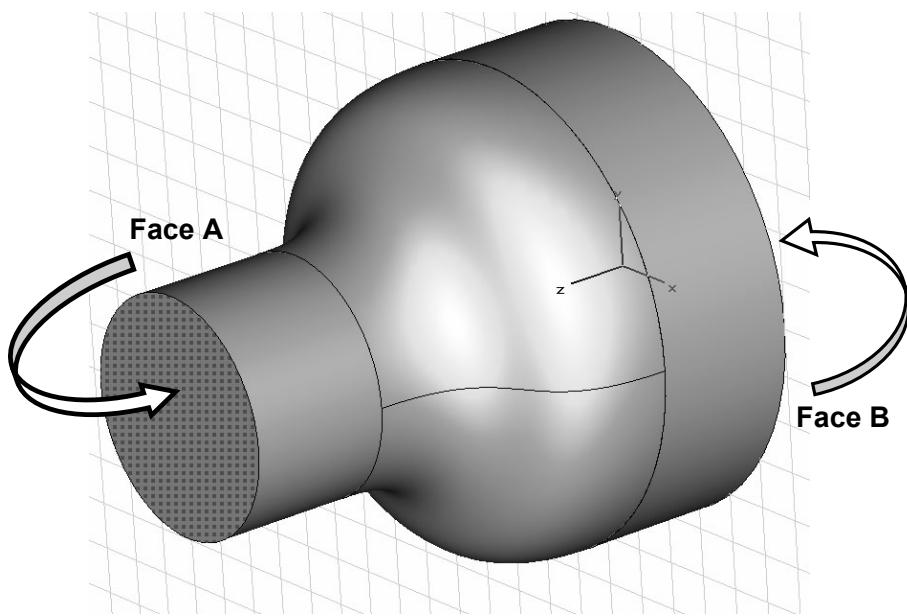
Now pick the adjacent top and bottom faces of the two cylinders as shown above.

Afterward you can activate the loft tool by selecting Objects \Rightarrow Loft (). In the following dialog box you can set the smoothness to a reasonable value and click the *Preview* button to get an impression of the shape. Drag the *Smoothness* slider such that the shape has a relatively smooth transition between the two picked faces before clicking *OK*.

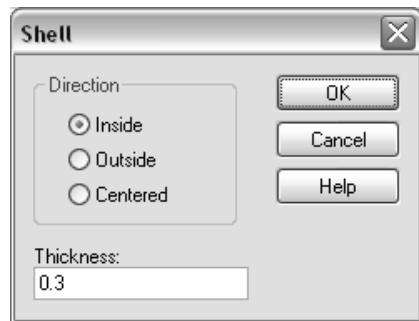
Note: You should select the corresponding shape before picking its face. Since all other shapes become transparent, it is easier to pick the desired face even “through” other shapes.



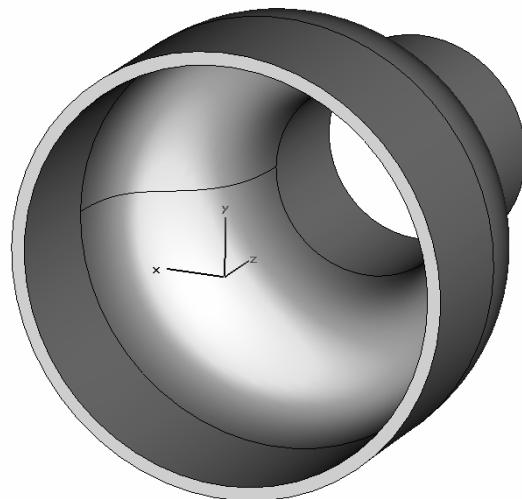
Finally your model should look like the following picture (note that the actual form of the lofted shape depends on the setting of the smoothness parameter).



Finally, add all shapes together by selecting all three (holding down the *Ctrl*-key) and using the *Objects*⇒*Boolean*⇒*Add* operation. Now you can pick the two planar top and bottom faces of the shape, select the shape by double-clicking on it and initiate the *Objects*⇒*Shell Solid or Thicken Sheet* tool. Note that the shell command will be accessible only when a shape has been selected previously.



In the dialog box, you can specify a *Thickness* (e.g. 0.3) and click 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

The ability to create local coordinate systems adds a great deal of flexibility to the modeler. In the above sections we describe how to create simple shapes that 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 are specified in this coordinate system from now on. You may activate or deactivate the local coordinate system from the *WCS \Rightarrow Local Coordinate System* item in the main menu, from the *WCS* context menu item, or by selecting 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 \Rightarrow 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 \Rightarrow 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 \Rightarrow Align WCS with Selected Edge*).
- Finally, a **planar face** can be selected to which the u/v plane of the WCS can be aligned (*WCS \Rightarrow Align WCS with Selected Face*).

After picking a point, edge or face of 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.

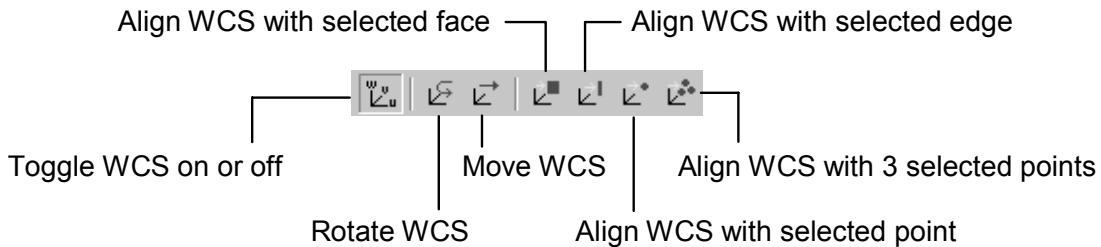
Besides aligning the WCS with entities selected from the model, there are three further ways of defining the local coordinate system:

- Define local coordinate system parameters directly:** (*WCS \Rightarrow 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 can 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.

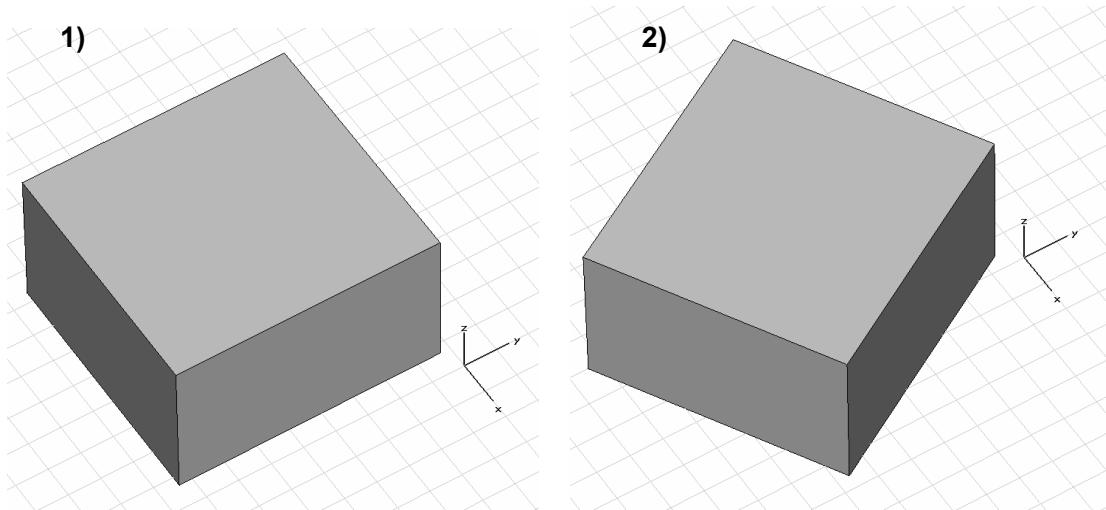
The second and third options are especially powerful when combined with the pick alignment options described above.

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

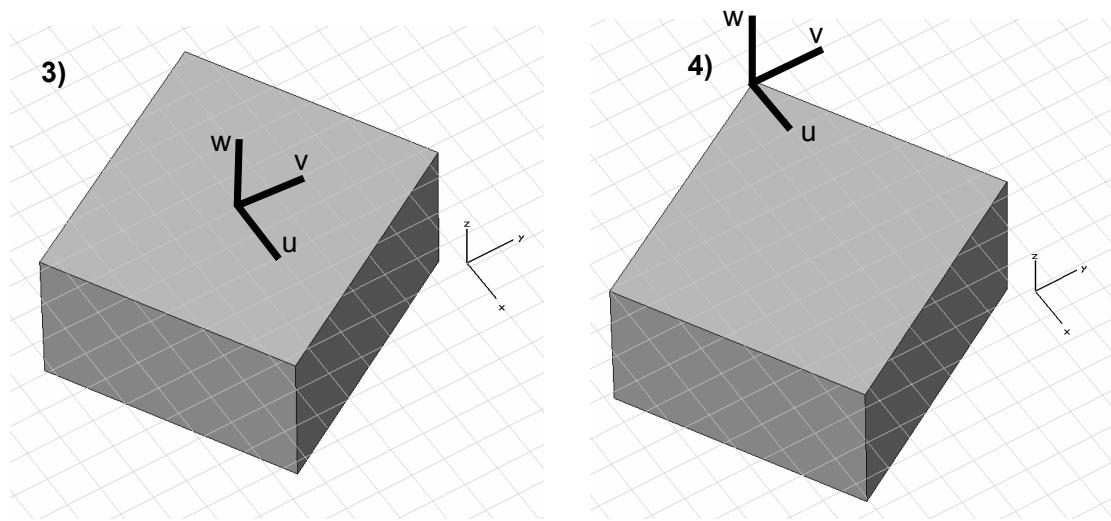


The following example should give you an idea of what can be done by efficiently using local coordinate system specifications:

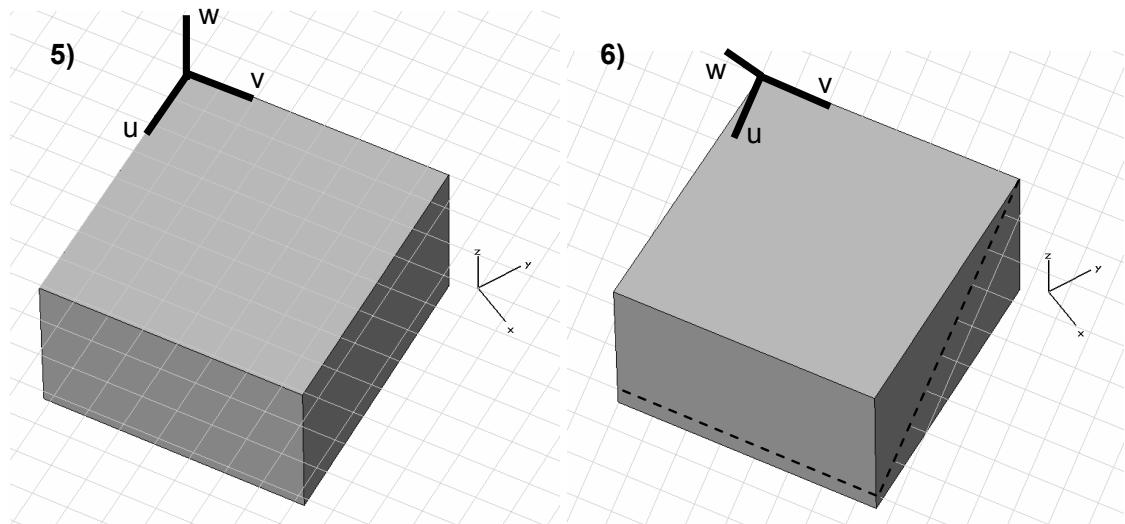
The first step is to create a brick in global coordinates. Then rotate the brick around the z-axis by 30 degrees using the transform dialog box:



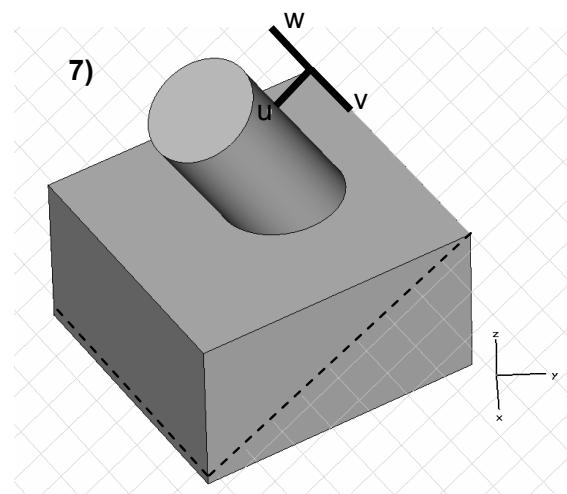
Next activate the local coordinate system, and align it first with the top face of the brick and then with one of the vertices on the top face:



Now align the coordinate system with one of the edges of the brick's top face, and then rotate the coordinate system 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 click *OK*:



The History List

Up to now, you have created some basic structures and performed some simple geometric transformations. You can always correct mistakes made during the structure generation by using the *Edit*⇒*Undo* command to remove 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 “History List.” All relevant structural modifications are recorded in a list you can see by choosing *Edit*⇒*History List* or by clicking  in the objects toolbar.

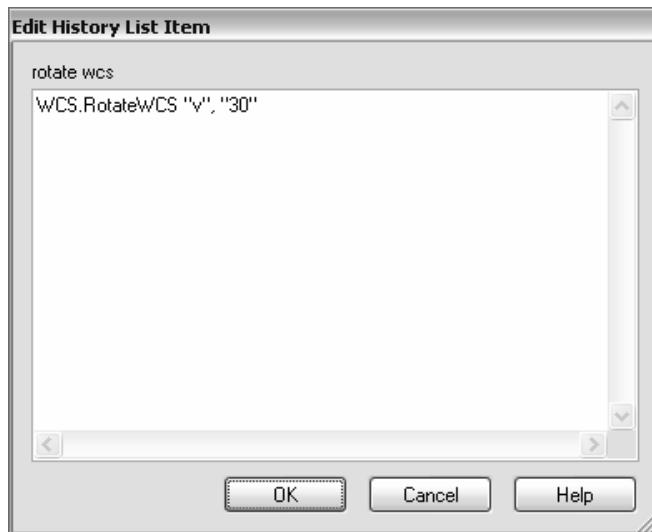
In the following, we assume you have created the structure consisting of a brick and a cylinder as shown in the last section covering local coordinate systems.

In this case, the history list will look like the following picture:

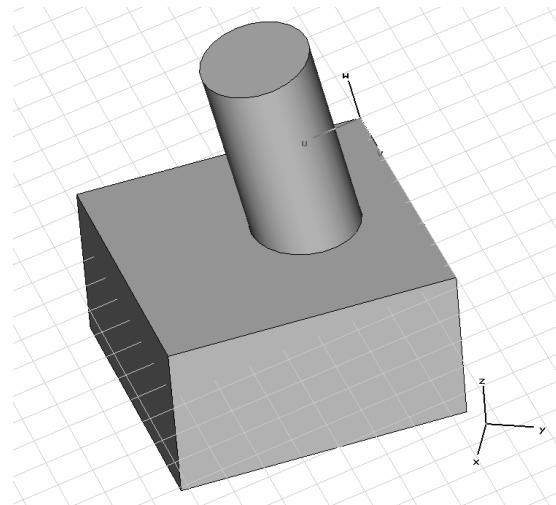


The list shows all previous operations in chronological order. The marker ► indicates the current position of the structure creation in the history list. You may restore the structure creation to any step in the history list by selecting the corresponding line and clicking the *Restore* button. Clicking the *Step* button will take you to the next step in the history list. You can now experiment a bit with this feature.

Clicking the *Update* button completely regenerates the structure. The *Edit* button allows you to perform changes to previous operations. In this case, select the “rotate wcs” line and click 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 10 degrees and click the *OK* button. Back in the history list, you should click the *Update* button to regenerate the structure. Your structure should look like the following picture:



In general, the history functionality allows you to perform changes to the model quickly and easily without having to re-enter 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, assume 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.

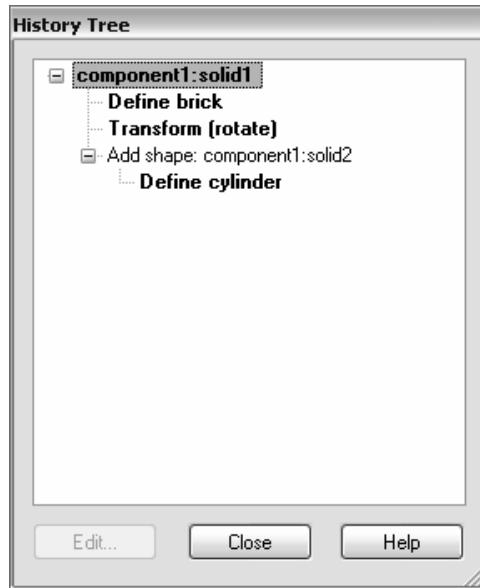
In such cases we recommend you work through the history list from the beginning in order to properly adjust the picks when needed. Even in this extreme case, the work needed to change the model takes much less effort than completely re-entering the model. Refer to the online documentation for details.

The History Tree

The History List is 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.

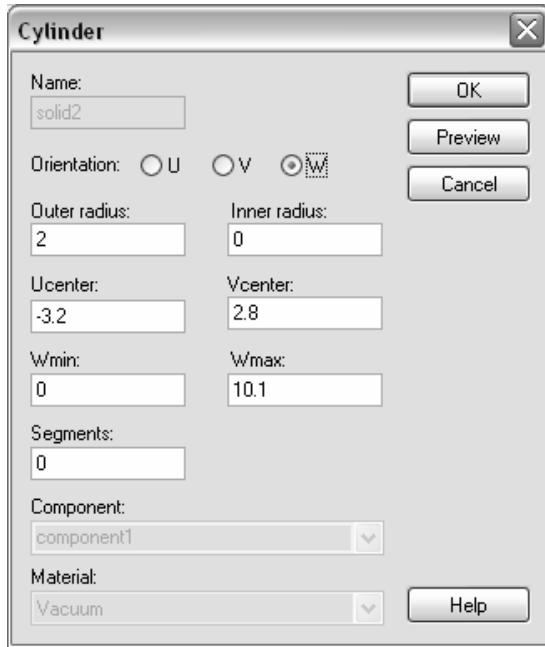
Assume that you want to change the radius of the cylinder in the previous example. You can open the history list and edit the generation of the cylinder. However, you can also select the corresponding shape by double-clicking it and then choosing *Edit⇒Object Properties* or *Properties* from the context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

A dialog box (the History Tree) will open showing the construction of the selected shape:



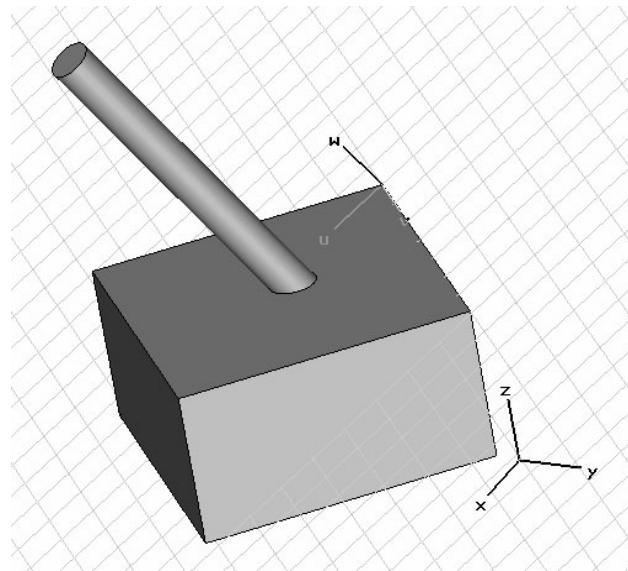
You can now simply click the “Define cylinder” item. As soon as you have selected an editable operation from the History Tree, the corresponding structure element will be highlighted in the main view. Please note that subsequent transformations will not be considered by this highlighting functionality.

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



You can now alter the cylinder radius and click the *Preview* button. You will get an impression of how the structural changes will influence your model. If you are happy with the result, click the *OK* button to update the structure.

Finally, your model should look as follows:



Play around a little with the History Tree to get an idea of what changes can be applied to the existing structure using this functionality. Note that subsequent transformations will not be visualized by the *Preview* option in the shape dialog box but will be applied when you update the model.

Curve Creation

The previous chapters show how a model can be generated from 3D primitives and their modifications by using powerful operations such as blending, lofting, shelling, etc.

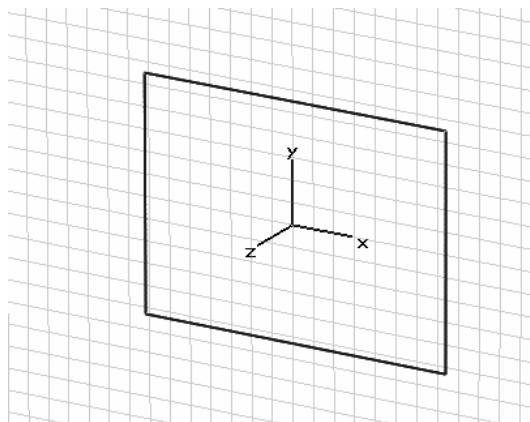
Another complex shape generation option is based on *curves*. A curve is a 3D line drawn on the drawing plane. After a curve has been defined, it can be used for more advanced modeling operations.

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

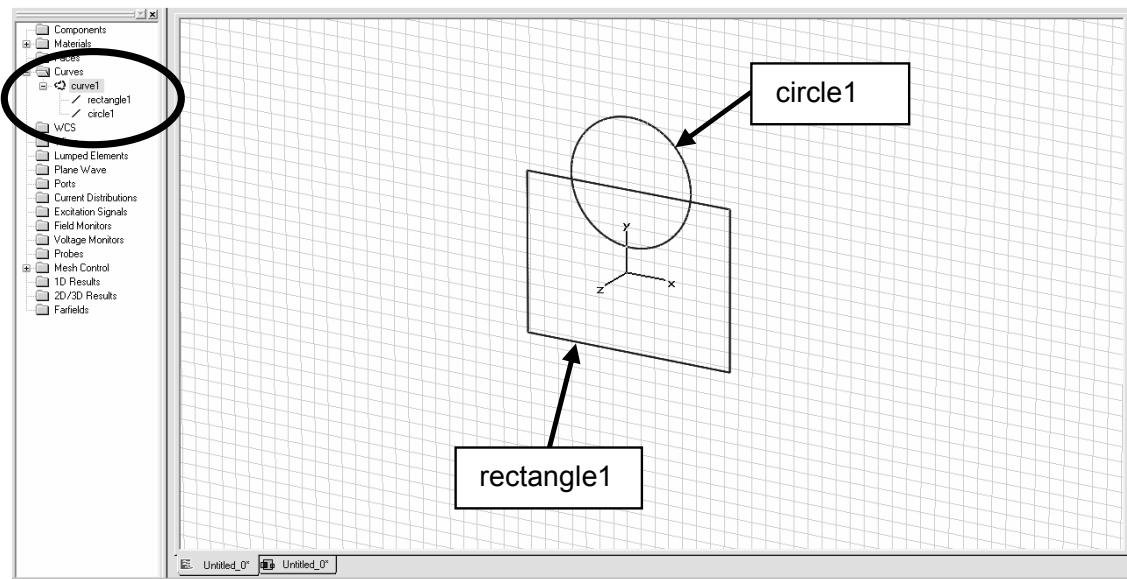
To practice, you should first create a new curve by selecting *Curves*⇒*New Curve* () from the main menu. This operation will create a new item called “curve1” in the navigation tree’s *Curves* folder.

Now activate the rectangle creation by choosing *Curves*⇒*Rectangle* () before drawing a rectangle on the working plane. Creating curve items is similar to constructing solid primitives.

Your result should look as follows:



Next draw a circle on the drawing plane which overlaps one of the rectangle’s edges. Activate the circle creation by choosing *Curves*⇒*Circle* () . Afterward, your screen should look similar to the following:

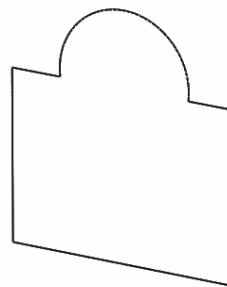


As a result of the previous steps, you now have two curve items – *rectangle1* and *circle1* – in a curve named *curve1*. The navigation tree reflects this relationship.

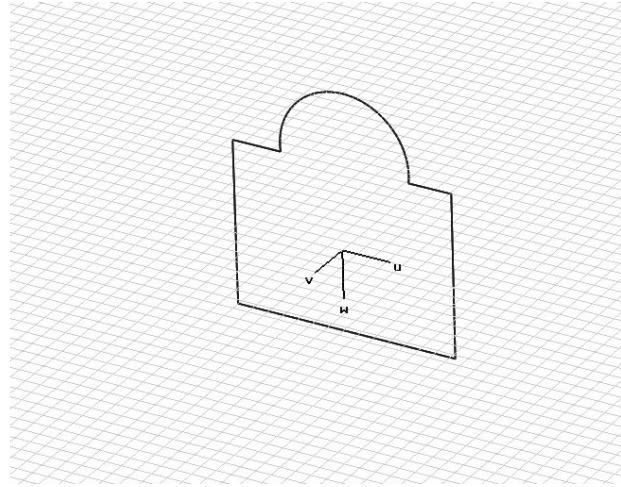
Now let's trim both curve items so that the resulting curve contains only the outlines of both curve items. First select one of the curve items, e.g. *rectangle1* (either in the navigation tree or by double-clicking on it in the main view). Afterward activate the *Trim Curves* operation by choosing *Curves*⇒*Trim Curves* () from the main menu.

You will be prompted to select the item to be trimmed with the rectangle. Select the circle and confirm your selection by pressing the *Return* () key.

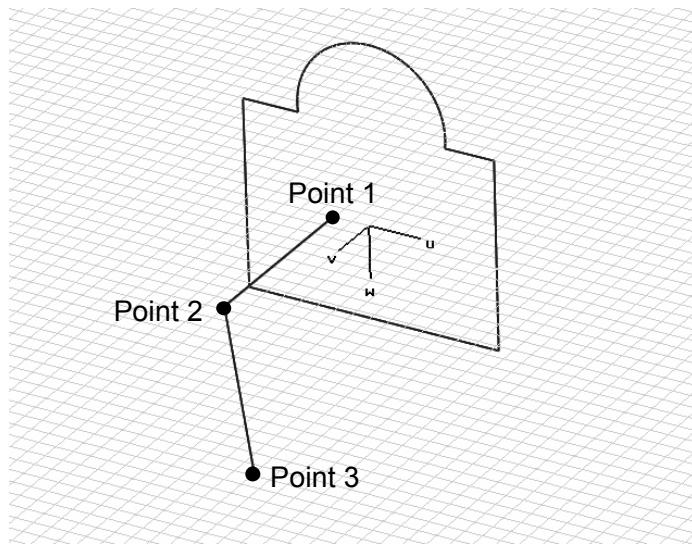
The next step will prompt you to double-click on any curve segments you wish to delete from the model. When you move the mouse across the screen, all selectable curve segments at the mouse location will be highlighted. You should now delete two segments so that the result looks similar to the following 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 model should look as follows:



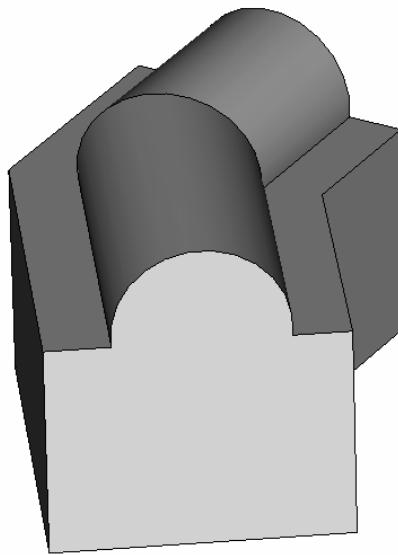
The next action is to draw an open polygon consisting of three points on the drawing plane (*Curves*⇒*Polygon*, ).



Based on these two disjoint curves, you are going to create a solid using the *sweep curves* operation which can be initiated 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. 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 given by the polygon's curve here. After you close the resulting dialog box by clicking *OK*, the final shape should look as follows:



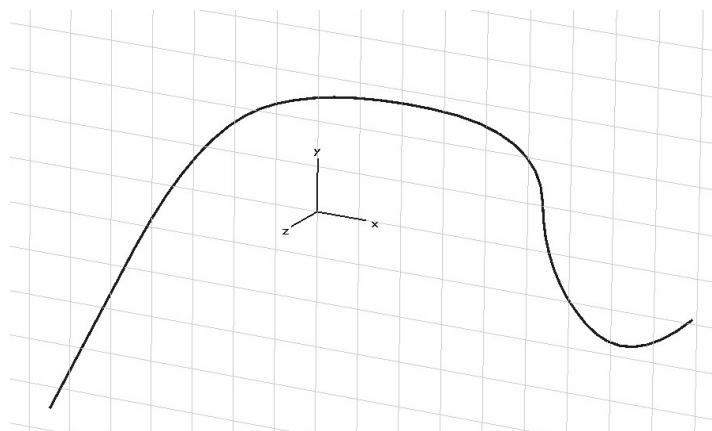
This short introduction into curve modeling provides a very basic understanding of these powerful structure drawing tools. You should experiment a little with the curve modeling features to become more familiar with this kind of structure modeling. Please refer to the online documentation for more details.

Trace Creation

The next section focuses on a rather tedious part of the model creation: the definition of conducting traces. Some structures (e.g. printed circuit boards) require many traces which often entail many time-consuming construction steps.

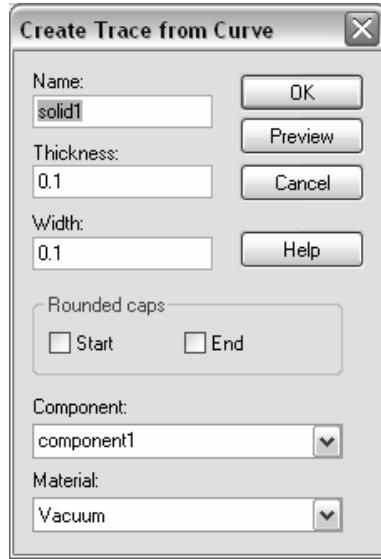
To simplify this task, a trace tool has been added to allow the creation of solid traces with finite width and thickness based on the definition of curves.

To practice using this powerful tool, draw an open but otherwise continuous curve such as the following:



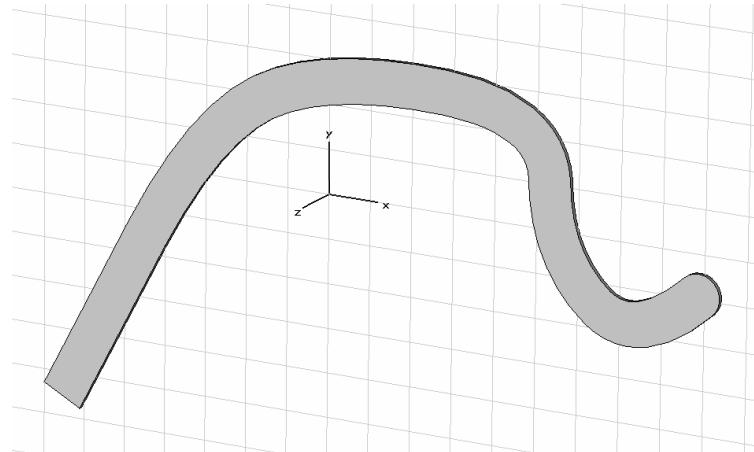
Based on this curve, you can now easily create a trace by choosing *Curves*⇒*Trace From Curve* (✉). As soon as this operation is activated, you will be prompted to select the trace's curve.

After you double-click on the previously defined curve, the following dialog box will open:



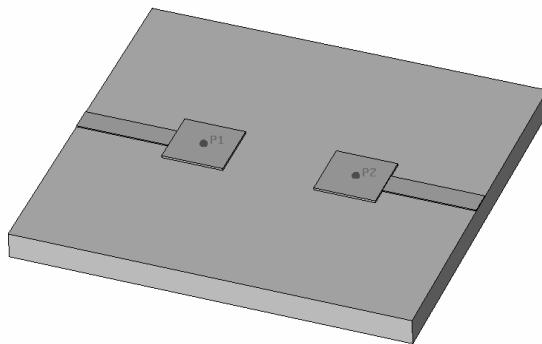
In this dialog box, you can specify the metallization *Thickness* and the *Width* of the trace. You can also specify whether the trace should have rounded caps (instead of rectangular caps) at the start or end of the trace's path.

The resulting trace might look as follows (rounded caps at the end of the trace only):

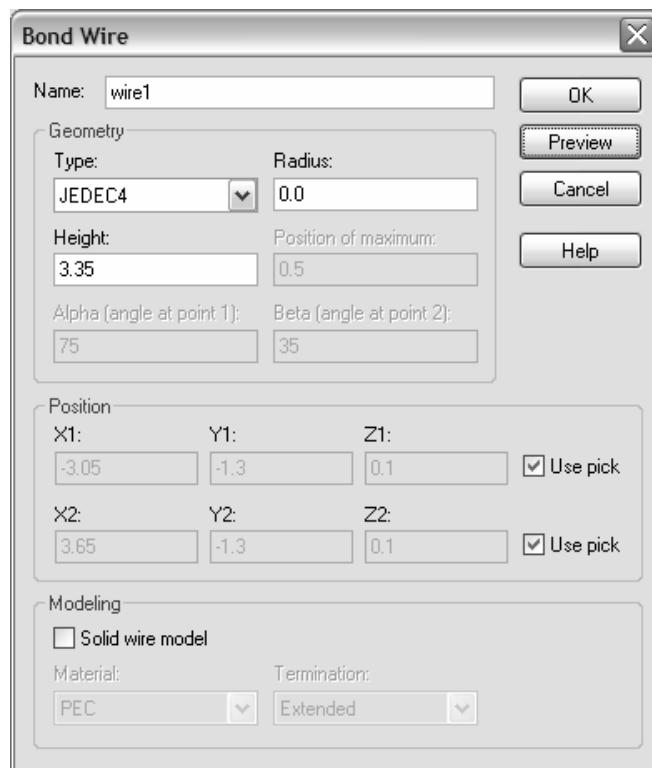


Bondwire Creation

You can create bondwires as frequently used structure elements easily using the bondwire tool. The easiest way to define a bondwire between two points is to pick those points first as shown in the following picture:



Once the points are picked, you can open the bondwire dialog box by choosing *Objects*⇒*Basic Shapes*⇒*Bondwire* ().

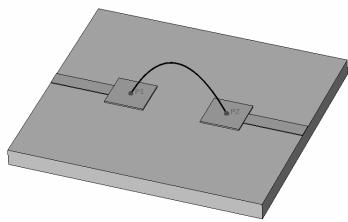


You can also open the dialog box without any points being picked previously. In this case, you may specify the coordinates of the bondwire's start and end points numerically.

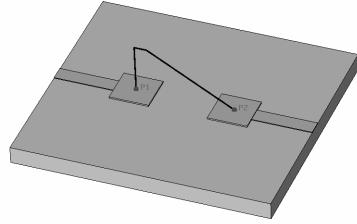
The type of the bondwire can be spline, JEDEC4 or JEDEC5. The location of the spline's maximum can be specified whereas the other two models accept the standardized parameters.

The following picture shows the three different types of bondwires:

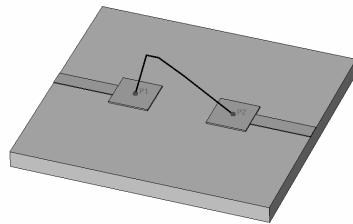
Spline



JEDEC4



JEDEC5



Refer to the online documentation for more information about JEDEC parameters.

You may also assign a finite radius to the wire by specifying a non-zero entry in the *Radius* field. However, the wire will still be modeled as infinitely thin, but the solver module will apply a special model to the wire in order to take the finite radius into account. Please note that this feature is not supported by solvers based on tetrahedral grids.

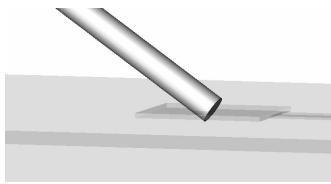
In addition to this option of modeling the bondwire as infinitely thin wire, the dialog box also supports the creation of solid bondwires by offering the *Solid wire model* option. As for every other solid, a solid bondwire needs a material to be assigned to it.

The *Termination* of the bondwire may be set to any one of the following types:

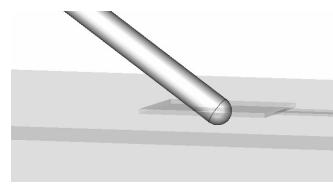
- Natural:** The wire will be a solid tube with perpendicular cuts at the end.
- Rounded:** The wire will be terminated by a part of a sphere.
- Extended:** This is the most powerful option. In this case, the software detects the plane in which the bondwire ends. Then the wire extends toward this plane in order to ensure an optimal connection with this plane.

The following picture illustrates the three types of termination:

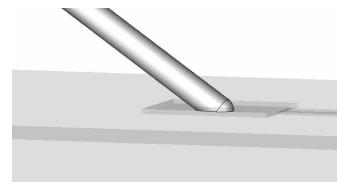
Natural



Rounded



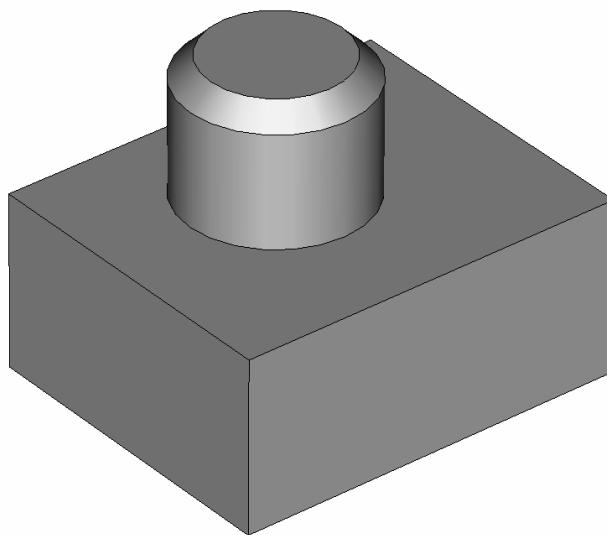
Extended



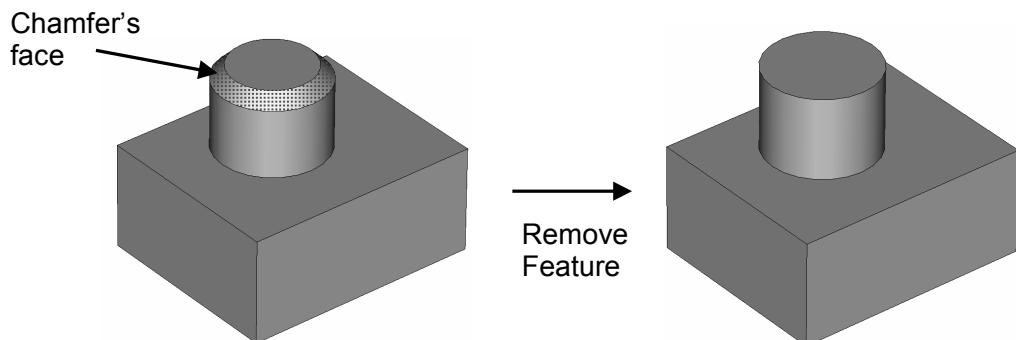
Local Modifications

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

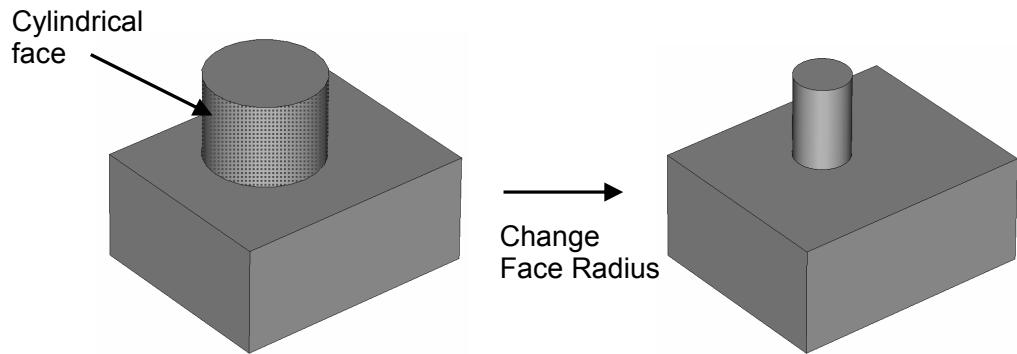
This section illustrates that even in these cases the structure can be parameterized using *Local Modifications*. Before you can start using these advanced modeling tools, you must create a structure similar to the following (a brick unified with a cylinder and a chamfer operation applied to the cylinder's top edge):



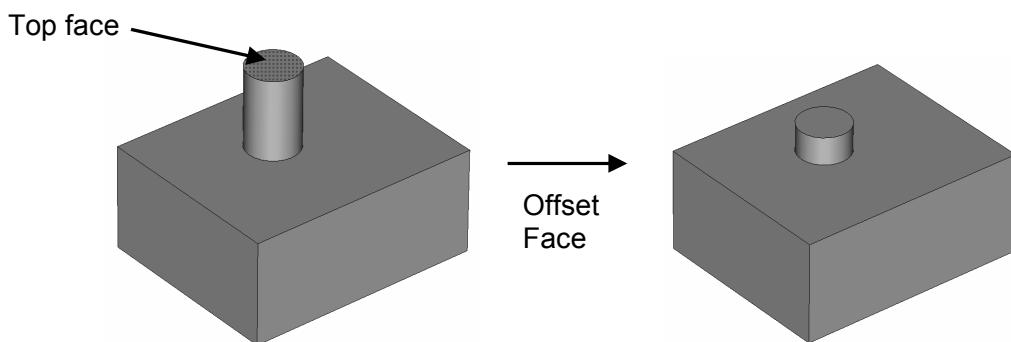
In this structure you should first use the pick face tools in order to select the chamfer's face. Then you can initiate the *remove feature* command by selecting *Objects*⇒*Local Modifications*⇒*Remove Feature* from the main menu.



As you can see, the gap that would be produced by simply removing the face will automatically be closed with the *remove feature* operation. Afterward, you should pick the cylindrical face and select the *Objects*⇒*Local Modifications*⇒*Change Face Radius* command. 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 initiate 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 powerful modeling operations. However, the modifications will fail if there is no unique solution for closing the gaps. You should play around a bit with these tools to get an impression of what is possible.

The First Real World Application Example

The previous chapters focus 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 help you become familiar with many standard operations that are important when performing a simulation with CST MICROWAVE STUDIO®.

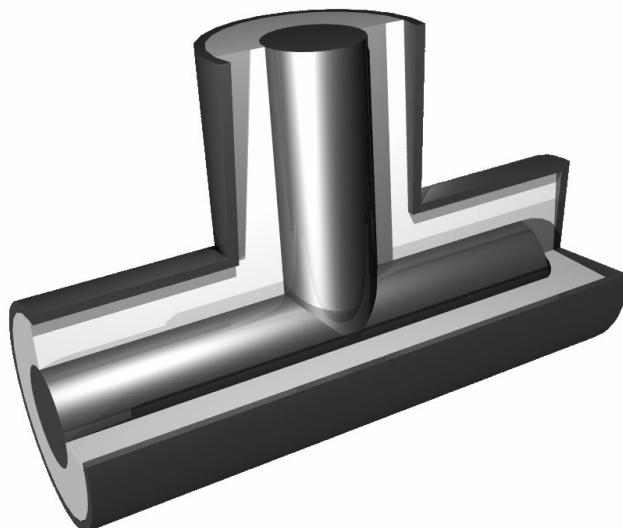
Go through the following explanations carefully, even if you are not planning to use the software for S-parameter computations. Only a small portion of the example is specific to this particular application type while most of the considerations are general to all solvers and application domains.

In subsequent sections you will find some remarks concerning the differences of the typical procedures for other kinds of simulations.

The following explanations 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. Because of the limited space in this manual, the shortest way to activate a particular command (i.e. by either pressing a shortcut key or by activating the command from the context menu) is omitted. You should regularly open the context menu to check 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 looking at the electromagnetic field inside this structure at various frequencies. The following picture shows the current structure of interest (it has been sliced open to aid visualization). The picture is produced using the POV export option.



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.

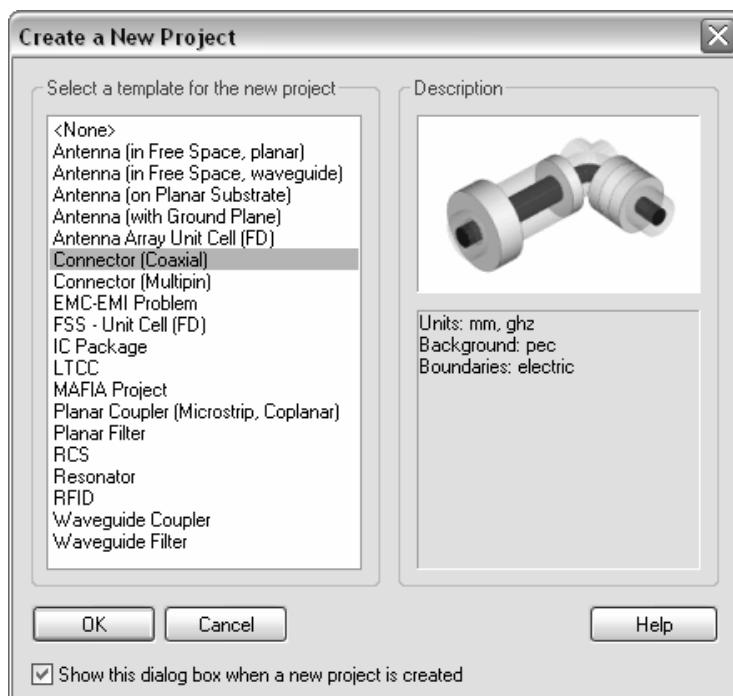
In order to simplify this procedure, CST MICROWAVE STUDIO® allows you to define the properties of the background material. Anything you do not fill with a particular material will automatically be filled with the background material. For this structure, it is sufficient to model the dielectric parts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

Your method of describing the structure should be as follows:

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

Start CST MICROWAVE STUDIO®

After starting CST DESIGN ENVIRONMENT™ and choosing to create a new CST MICROWAVE STUDIO® project, you will be asked to select a template for a structure which is closest to your device of interest.

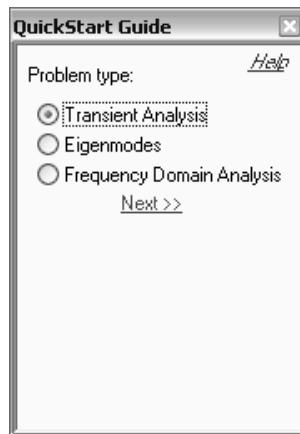


For this example, select the coaxial connector template and click **OK**. The software's default settings will adjust in order to simplify the simulation set up for the coaxial connector.

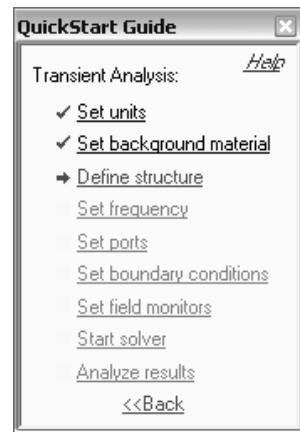
Open the Quick Start Guide

An interesting feature of the online help system is the *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, click the *Back* button to get the dialog above. In this dialog box you should select the *Problem Type* “Transient analysis” and click the *Next* button. The following window should appear:



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

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.

If you are unsure of how to access a certain operation, click on the corresponding line. The Quick Start Guide will then either run an animation showing the location of the related menu entry or open the corresponding help page.

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 can 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.

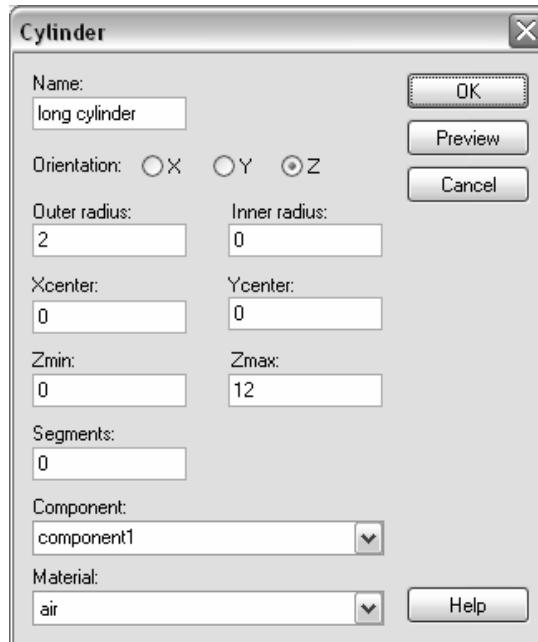
Define the Background Material

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 these settings, you may make changes in the corresponding dialog box (*Solve*⇒*Background Material*). For this example, you don't need to change anything.

Model the Structure

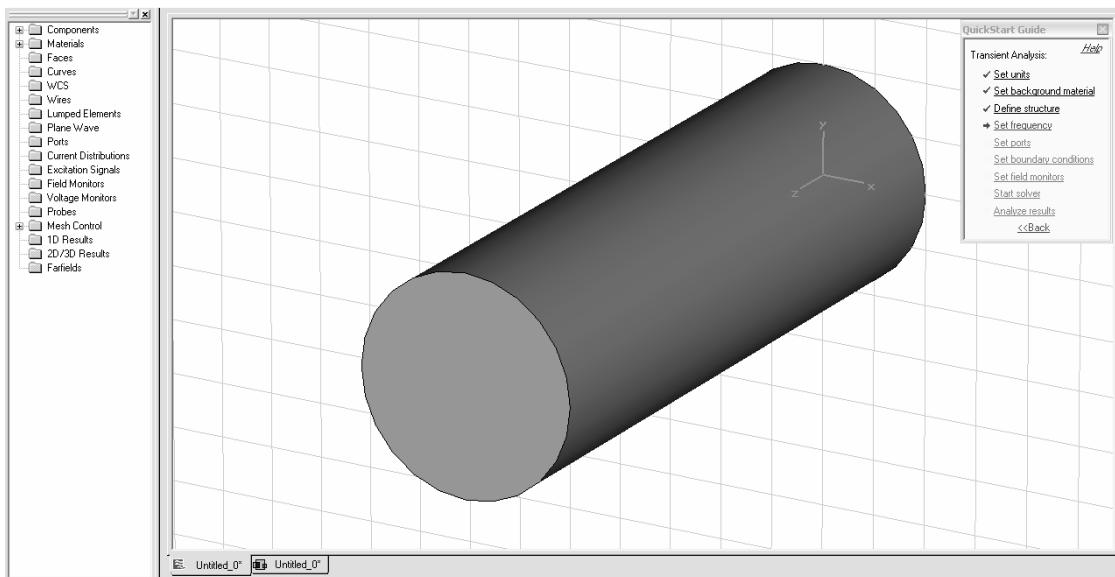
The first step is to create a cylinder along the z-axis of the coordinate system:

1. Select the cylinder creation tool from the main menu: *Objects*⇒*Basic Shapes*⇒*Cylinder* (C).
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 may simply select the predefined material *Vacuum* (which is very close to air) from the list in the *Material* field. Here we are going to create a new material “air” to show how the layer creation procedure works, so select the *New Material* entry in the list of materials.
8. In the material creation dialog box, enter the *Material name* “air,” select *Normal* dielectric properties (*Type*) and check the material properties *Epsilon* = 1.0 and *Mue* = 1.0. Then select a color and close the dialog box by clicking *OK*.
9. In the cylinder creation dialog box, your settings should now look as follows:



Finally, click *OK* to create the cylinder.

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

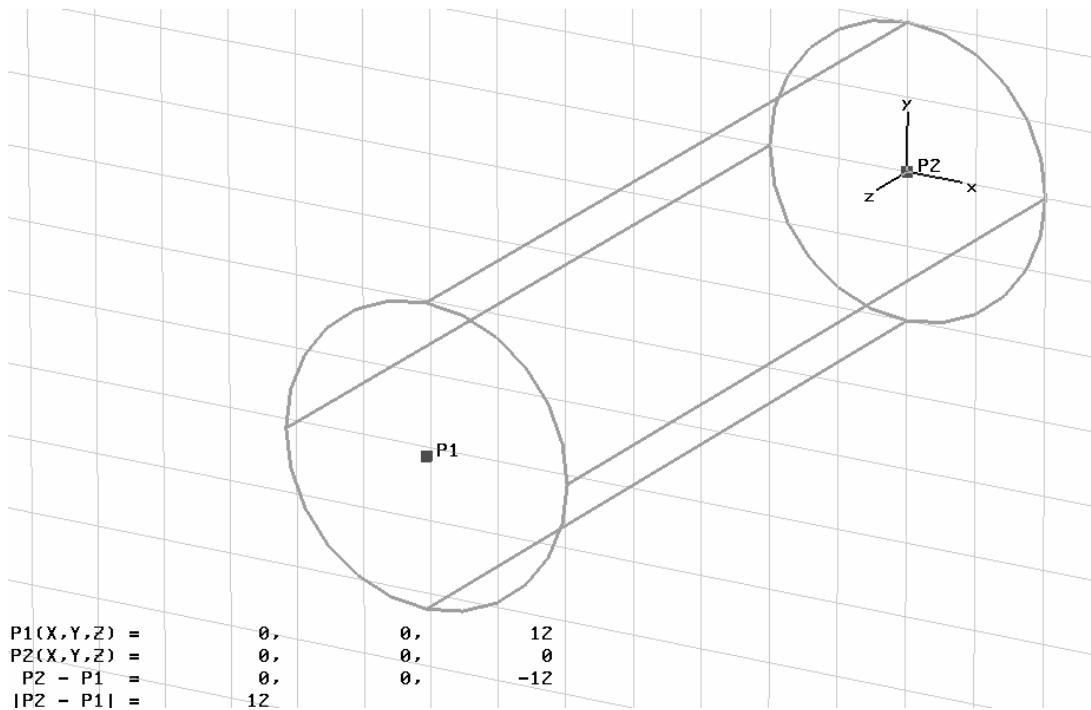


The next step is to create a second cylinder perpendicular to the first. The center of the new cylinder's base should be aligned with the center of the first one.

Follow these steps to define the second cylinder:

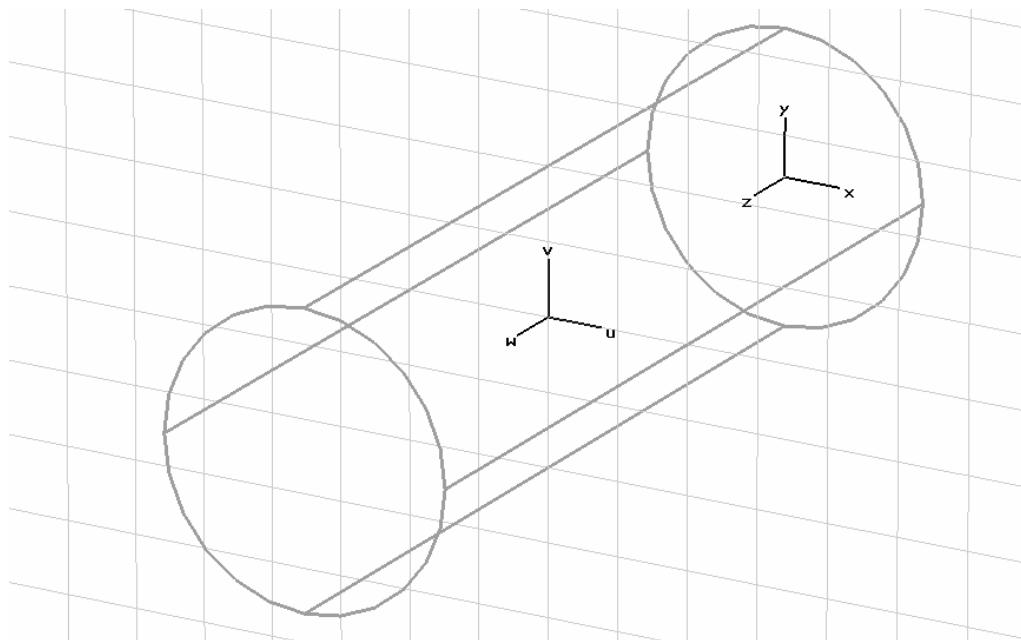
1. Select the wire frame draw mode: *View* \Rightarrow *View Options* () or use the shortcut **Ctrl+W**.
2. Activate the “circle center” pick tool: *Objects* \Rightarrow *Pick* \Rightarrow *Pick Circle Center* ().
3. Double-click on one of the cylinder's circular edges so that a point is added in the 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:



Next replace the two selected points by a point in between the two by selecting *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:

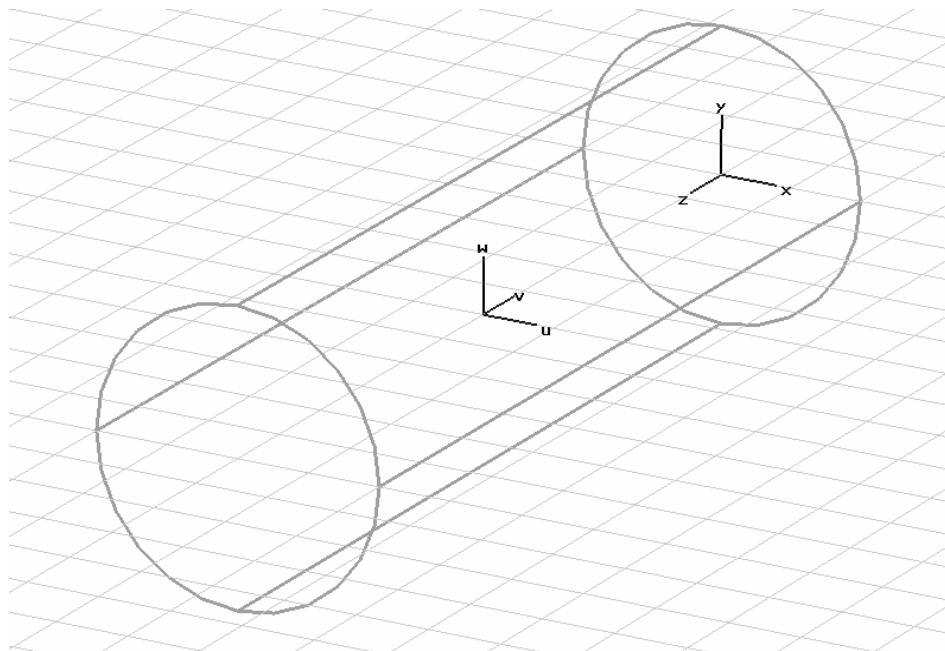


Now align the w axis of the WCS with the proposed axis of the second cylinder.

1. Select *WCS* \Rightarrow *Rotate Local Coordinates* () from the main menu.
2. Select the *U* axis as rotation *Axis* and enter a rotation *Angle* of –90 degrees.
3. Click the *OK* button.

Alternatively you could 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.

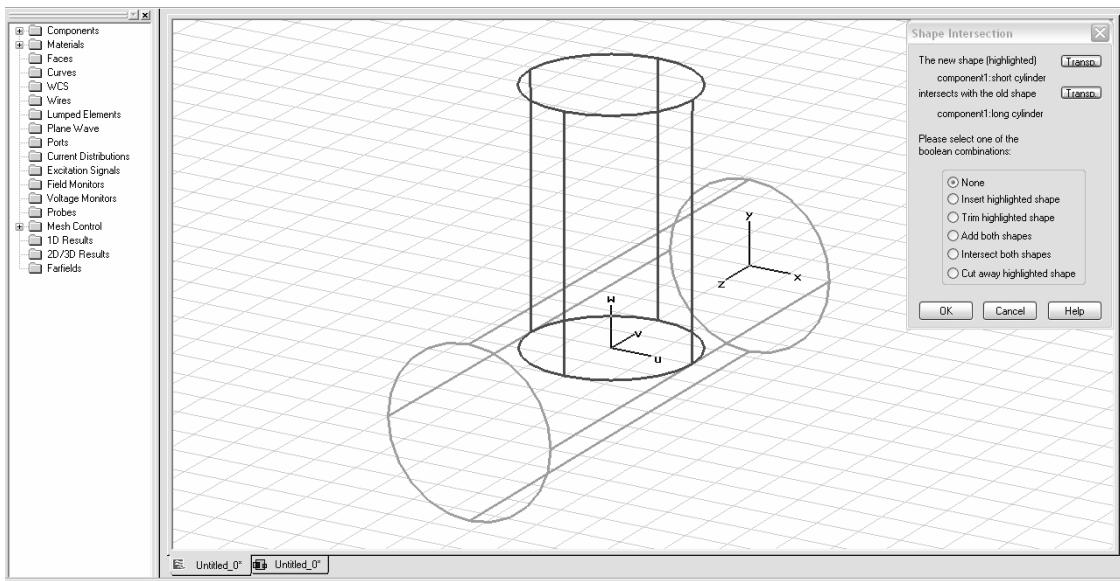
Now the structure should look like this:



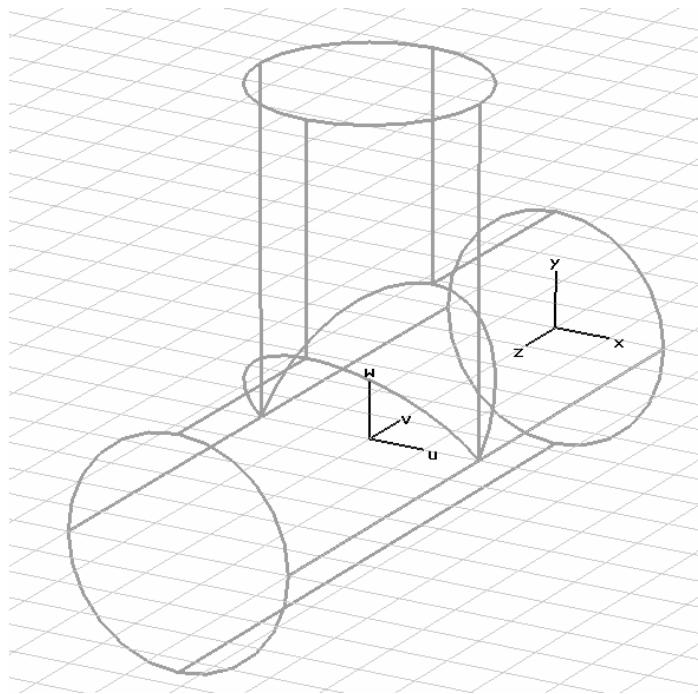
The next step is to create the second cylinder perpendicular to the first one:

1. Select the cylinder creation tool from the main menu: *Objects* \Rightarrow *Basic Shapes* \Rightarrow *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 material “air” from the material list and click *OK*.

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



In the “Shape intersection” dialog box, choose the option *Add both shapes* and click **OK**. Finally the structure should look like this:



The creation of the dielectric air parts is complete. 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 go ahead and start to create the first part of the conductor:

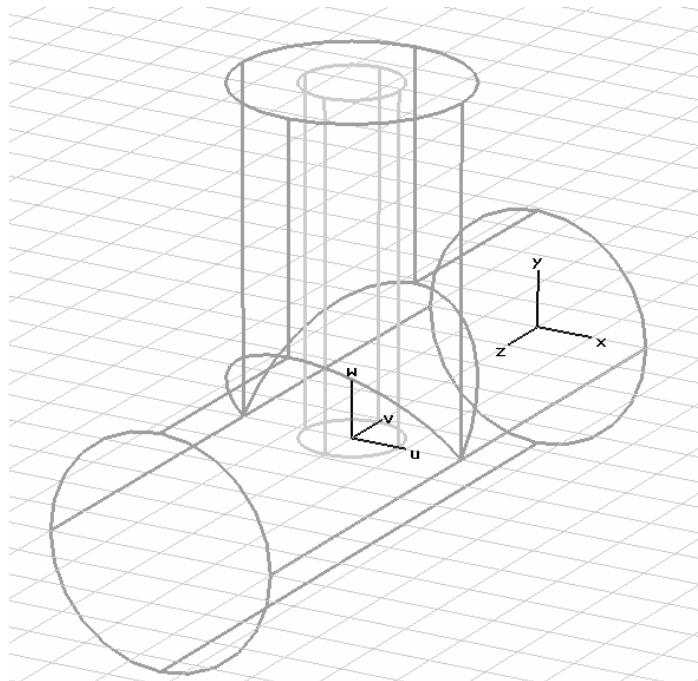
1. Select the cylinder creation tool from the main menu: *Objects*⇒*Basic Shapes*⇒*Cylinder* (C).
2. Press the *Shift+Tab* key and enter the center point (0,0) in the uv-plane.
3. Press the *Tab* key again and enter the radius 0.86.
4. Press the *Tab* key and enter the height 6.
5. Press *Esc* to create a solid cylinder.
6. In the shape dialog box, enter “short conductor” in the *Name* field.
7. Select the predefined *Material PEC* (perfect electric conductor) from the list of available materials and click *OK* 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 allowable for a perfect conductor to be present at the same point as a dielectric material. In such cases, the perfect conductor is always the dominant material. The situation is also clear for two overlapping perfectly conducting materials, since in this case the overlapping regions will also be perfect conductors.

On the other hand, two different dielectric shapes must not overlap each other. Therefore the intersection dialog box will not be shown automatically in case of a perfect conductor overlapping with a dielectric material or with another perfect conductor.

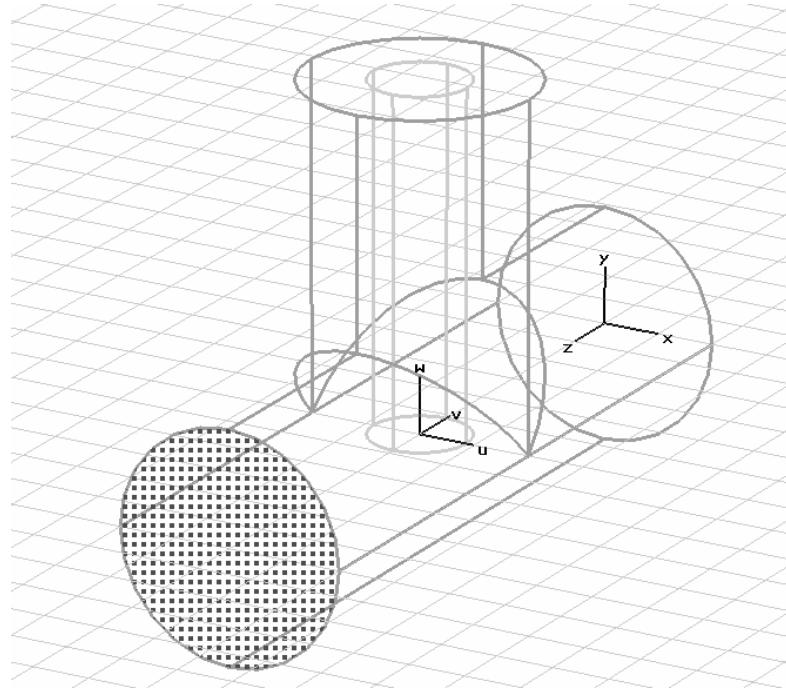
Background information: Some structures contain extremely complex conducting parts embedded within dielectric materials. In such cases, the overall complexity of the model can be significantly reduced by NOT intersecting these two materials. This is the reason CST MICROWAVE STUDIO® allows this exception. 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. First 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:



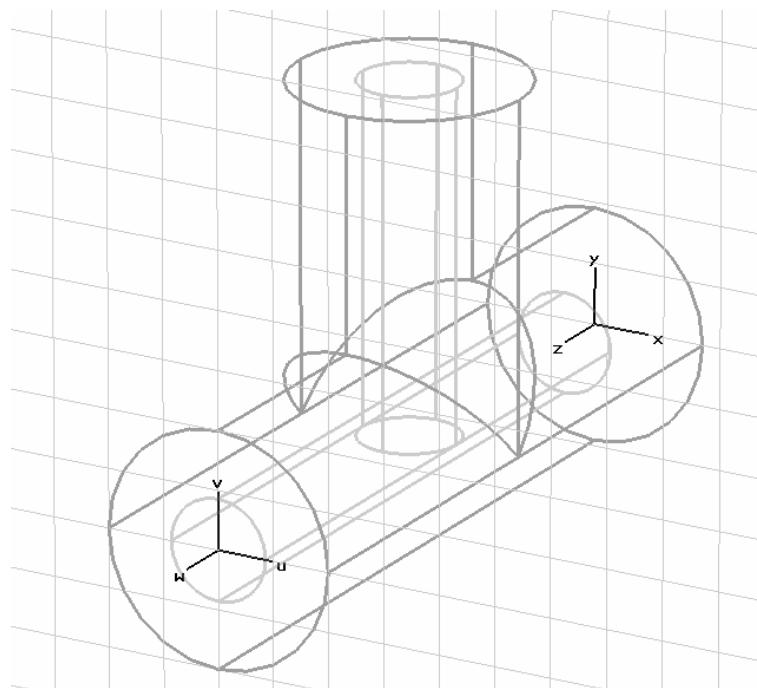
3. Now choose *WCS*⇒*Align WCS With Selected Face* (↖) from the main menu.

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

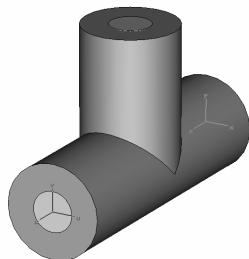
1. Select the cylinder creation tool from the main menu: *Objects*⇒*Basic Shapes*⇒*Cylinder* (cil).
2. Press the *Shift+Tab* key and enter the center point (0,0) in the uv-plane.
3. Press the *Tab* key again and enter the radius 0.86.
4. Press the *Tab* key and enter the height -11.
5. Press *Esc* to create a solid cylinder.
6. In the cylinder creation dialog box enter "long conductor" in the *Name* field.
7. Select the *Material* "PEC" from the list and click *OK*.

The newly created cylinder intersects with the dielectric part as well as with the previously created PEC cylinder. Even if there are two intersections (dielectric / PEC and PEC / PEC), the *Shape intersection* dialog box will not be shown here since both types of overlaps are well defined. In both cases the common volume will be of type PEC.

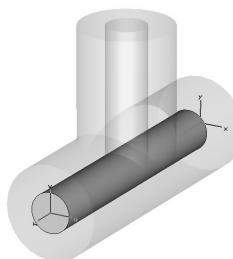
Congratulations! You have just created your first structure within CST MICROWAVE STUDIO®. The view should now look like this:



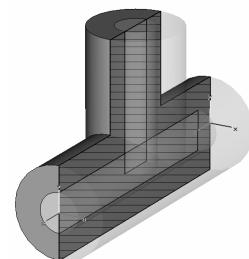
The following gallery shows some views of the structure available using different visualization options:



Shaded view
(deactivated working
plane, *Ctrl+W*)



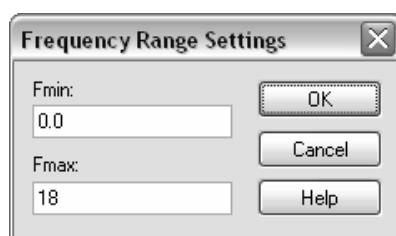
Shaded view
(long conductor
selected)



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

Define the Frequency Range

The next important setting for the simulation is the frequency range of interest. You can specify the frequency by choosing *Solve⇒Frequency* (*Ctrl+F*) from the main menu:



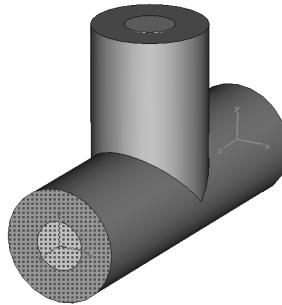
In this example you should specify a frequency range between 0 and 18 GHz. Since you have already set the frequency unit to GHz, you need to define only the absolute numbers 0 and 18 (the status bar always displays the current unit settings).

Define Ports

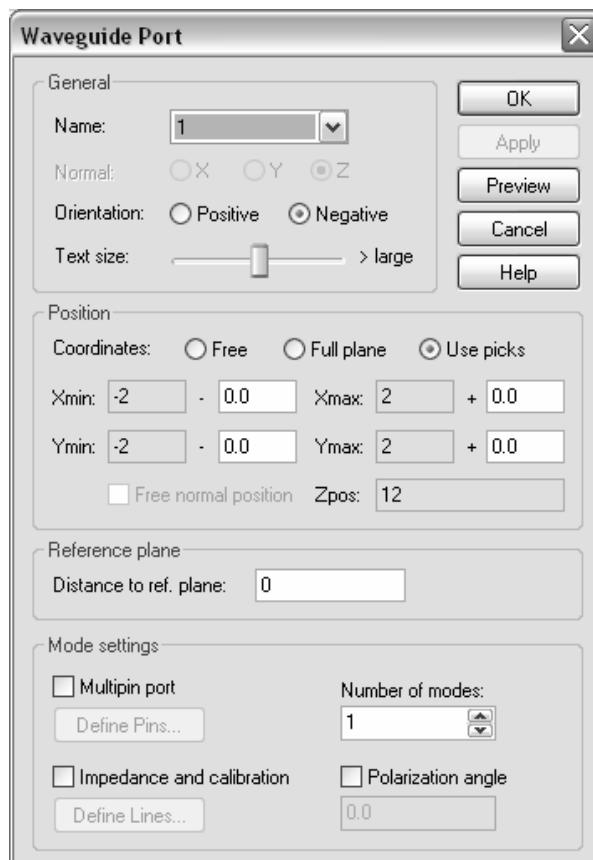
The following calculation of S-parameters requires the definition of ports through which energy enters and leaves the structure. You can do this by simply selecting the corresponding faces before entering the ports dialog box.

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

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

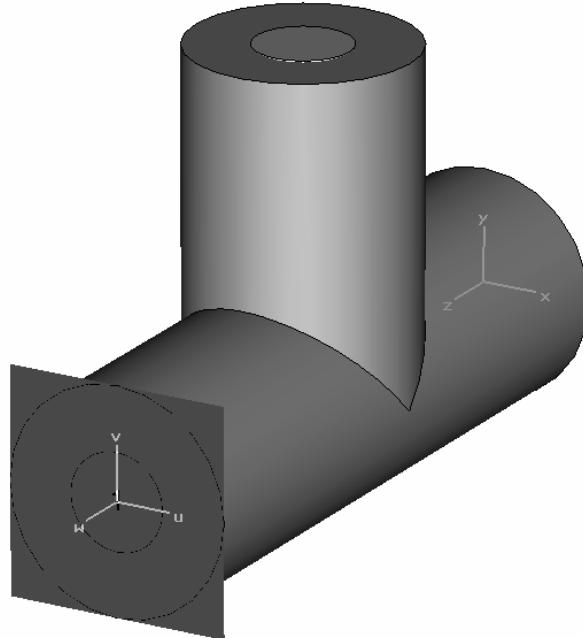


3. Open the ports dialog box by selecting *Solve* \Rightarrow *Waveguide Ports* () from the main menu:

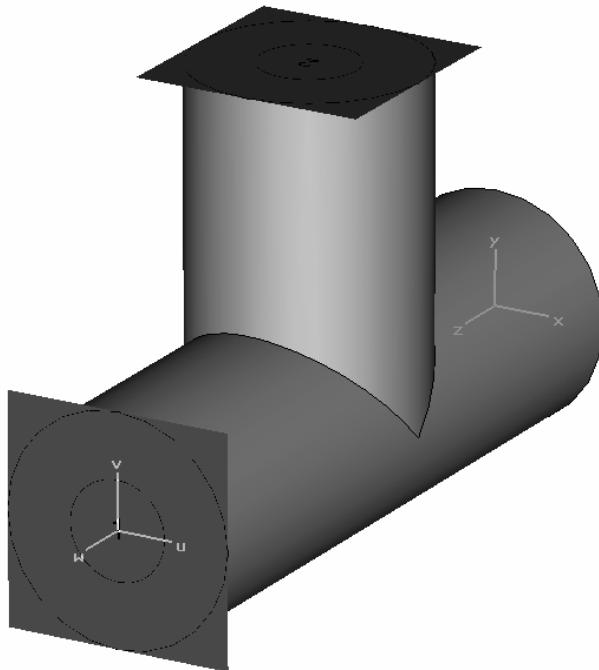


Everything is already set up correctly for the coaxial cable, so you can simply click *OK* in this dialog box.

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:

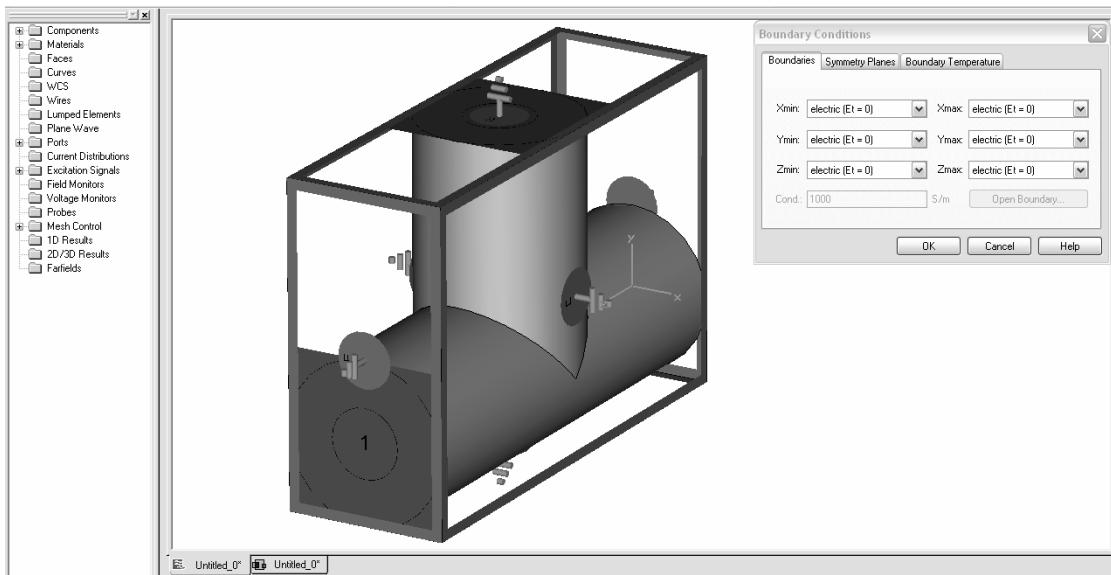


The correct definition of ports is very important for obtaining accurate S-parameters. Please refer to the *Choose the Right Port* section later in this manual to obtain more information about the correct placement of ports for various types of structures.

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 you can open by choosing *Solve* → *Boundary Conditions* from the main menu.

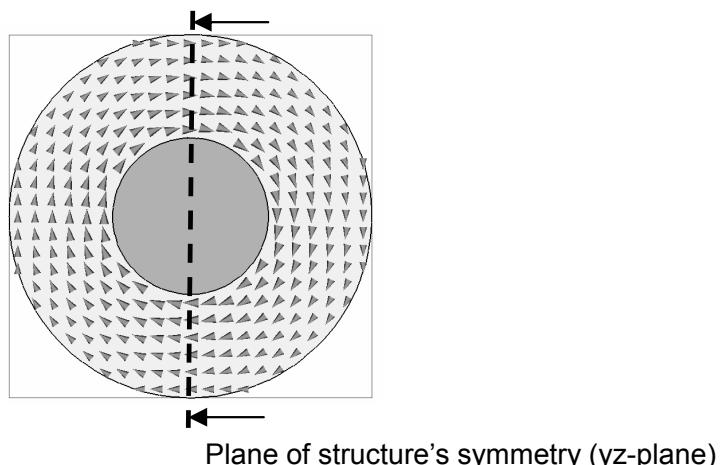


While the boundary dialog box is open, the boundary conditions will be visualized in the structure view as in the picture above.

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 "symmetry planes." The specification of each symmetry plane will reduce the simulation time by a factor of two.

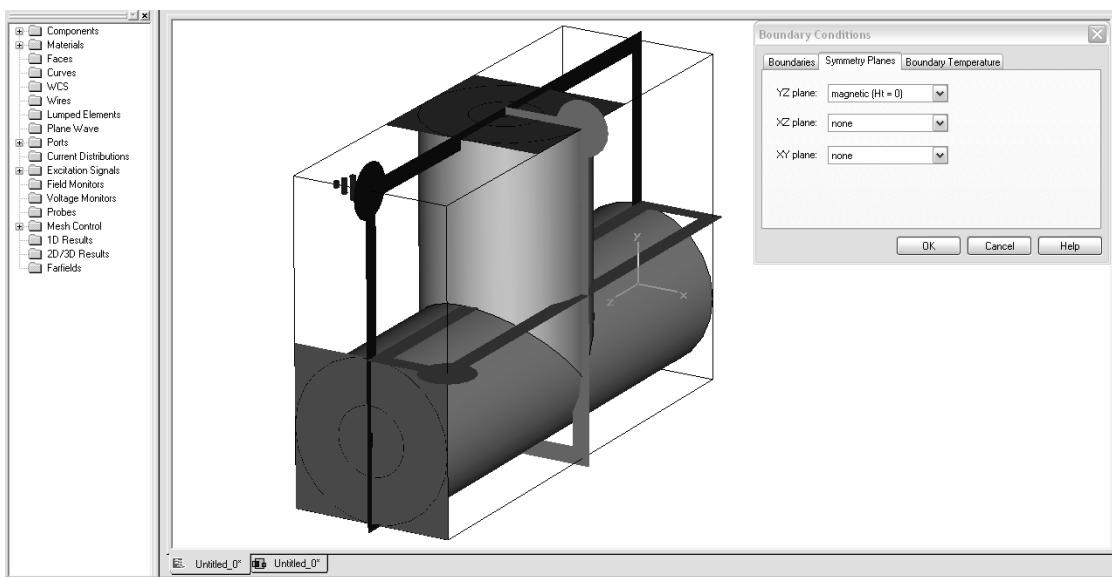
In our example, the structure is symmetric to a *yz*-plane perpendicular to the *x*-axis in the center of the structure. The excitation of the fields will be performed by the fundamental mode of the coaxial cable for which the magnetic field is shown below:



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 a "magnetic" symmetry plane, you can direct 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.

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



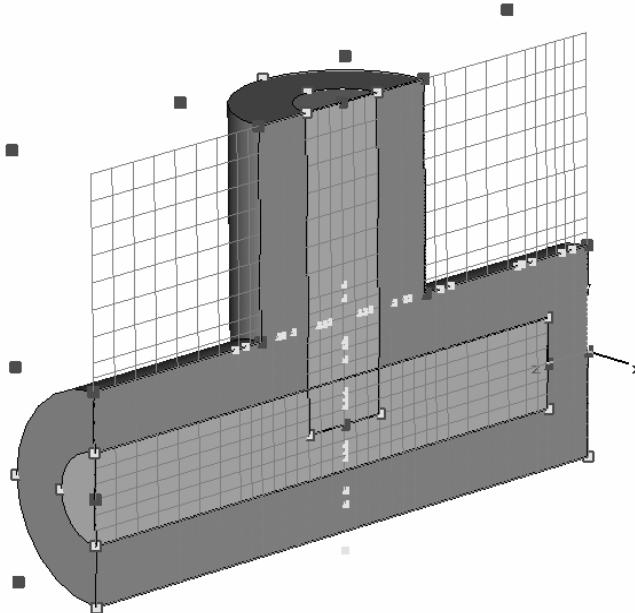
Finally click **OK** in the dialog box to store the settings. The boundary visualization will then disappear.

Visualize the Mesh

In a first simulation we will run the transient simulator based on hexahedral grids. Since this is the default mesh type, we don't need to change anything here. In a later step we show how to apply a tetrahedral mesh to this structure, run the frequency domain solver and compare the results. However, let's focus on the hexahedral mesh generation options first.

The hexahedral mesh generation for the structure analysis will be performed automatically based on an expert system. However, in some situations it may be helpful to inspect the mesh 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 View* (grid icon)). For this structure, the mesh information will be displayed as follows:



One 2D mesh plane is always kept in view. Because of the symmetry setting, the mesh plane extends across only one half of the structure. You can modify the orientation of the mesh plane by choosing *Mesh*⇒*X/Y/Z Plane Normal* (grid icon). Move the plane along its normal direction using *Mesh*⇒*Increment/Decrement Index* (grid icon) or using the Up / Down cursor keys.

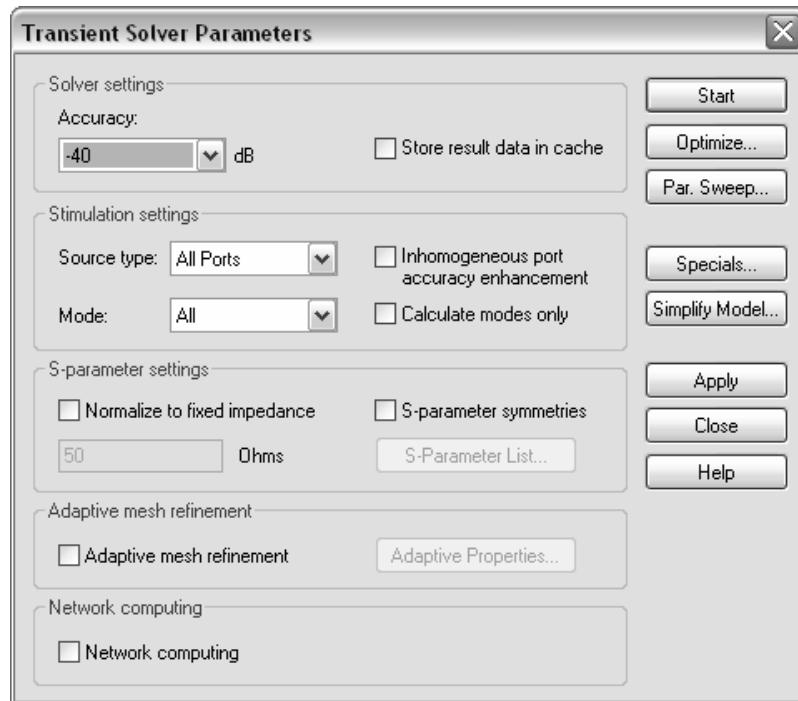
The red points in the model are critical points (so-called fixpoints) where the expert system finds it necessary to have mesh lines at these locations. In addition, the yellow dots show points where the automatic mesh generation finds the need for improving the mesh density.

In most cases the automatic mesh generation will produce a reasonable initial mesh, but we 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 View* (grid icon).

Start the Simulation

After defining all necessary parameters, you are ready to start your first simulation.

Start the simulation from the transient solver control dialog box: *Solve* \Rightarrow *Transient Solver* ().



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, by 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 normalized to the port impedance (which will be calculated automatically) by default. In this case the port impedance will be approximately

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

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), so check the *Normalize to fixed impedance* button and specify the desired normalization impedance in the entry field below. In this example we assume that you want to calculate the S-parameters for a reference impedance of 50 Ohms. Note that the re-normalization of the S-parameters is possible only when all S-parameters are calculated (*Source Type = All Ports*).

While 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.

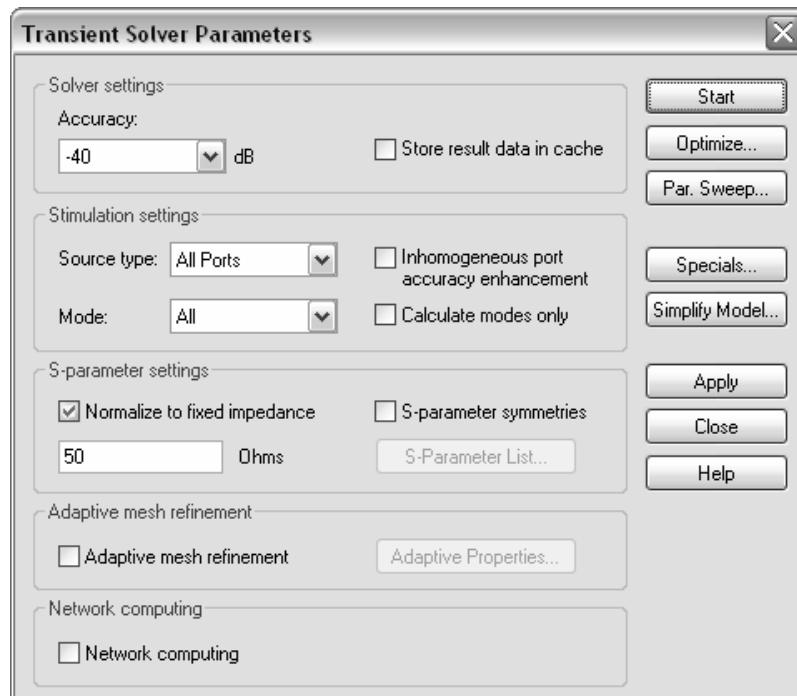
In order to obtain the S-parameters, the transformation of the time signals into the frequency domain requires the signals to have sufficiently decayed to zero. Otherwise a truncation error will occur causing ripples on the S-parameter curves.

CST MICROWAVE STUDIO® features an automatic solver control that stops transient analysis when the energy inside the device, and thus the time signals at the ports, has 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 in the *Accuracy* field (in dB).

In this example we will limit the maximum truncation error down to 1% for which you should keep the default solver *Accuracy* at -40 dB.

The solver will excite the structure with a Gaussian pulse in time domain. However, all frequency domain and field data obtained during the simulation will be normalized to a frequency independent input power of 1 W.

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



You can now start the simulation procedure by clicking the *Start* solver button. A progress bar will appear in the status bar which will update you on the solver's progress.

Information text regarding the operation will appear next to the progress bar. The most important stages are listed below:

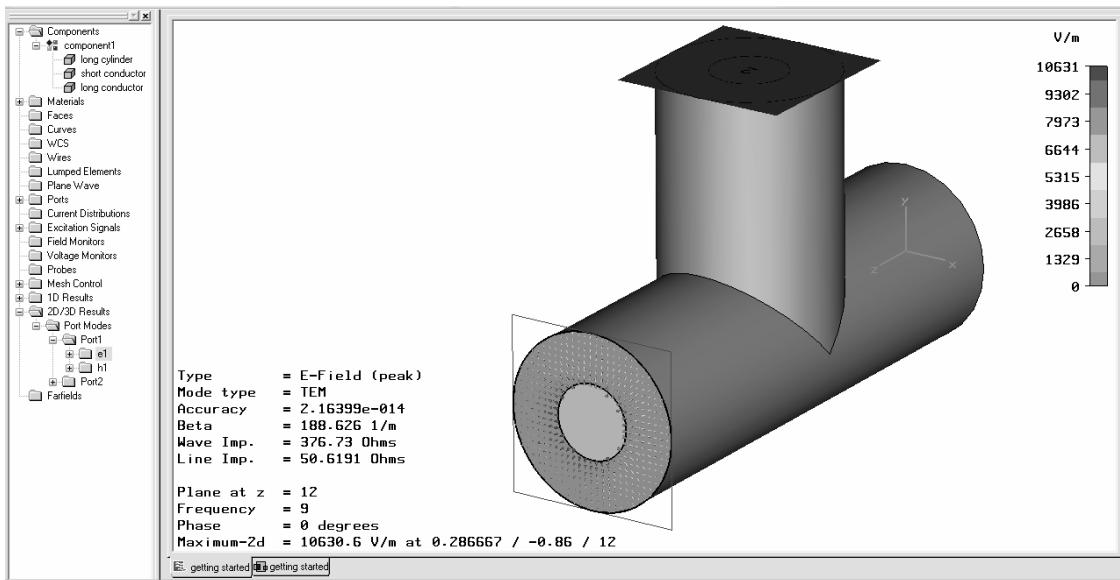
1. **Calculating matrices, preparing and checking model:** During this step, your input model is checked for errors such as invalid overlapping materials.
2. **Calculating matrices, normal matrix and dual matrix:** During these steps, the system of equations, which will subsequently be solved, are set up.
3. **Transient analysis, 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. **Transient analysis, processing excitation:** During this stage, an input signal is 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, transient field 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 below a certain limit (specified by the *Accuracy* setting in the solver dialog box).

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

Analyze the Port Modes

After the solver has completed the port mode calculation, you can view the results (even if the transient analysis is still running).

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

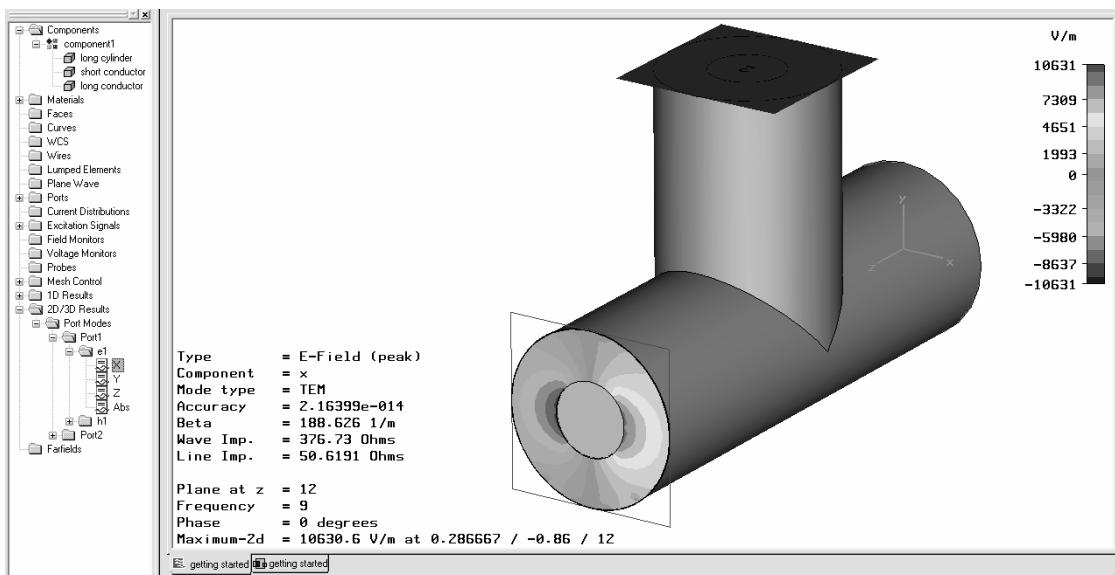


Besides information on the type of mode (here TEM), you will also find the propagation constant (beta) at the central frequency. Additionally port impedance is calculated automatically (line impedance).

You will find that the calculated result for the port impedance of 50.62 Ohms agrees well with the analytical solution of 50.58 Ohms. The small difference is caused by the discretization of the structure. Increasing the mesh density will improve the agreement between simulation and theoretical value. However, the automatic mesh generation always tries to choose a mesh that provides a good trade off between accuracy and simulation speed.

You can adjust the number and size of arrows in the dialog box, which can be opened by choosing *Results*⇒*Plot Properties* (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:

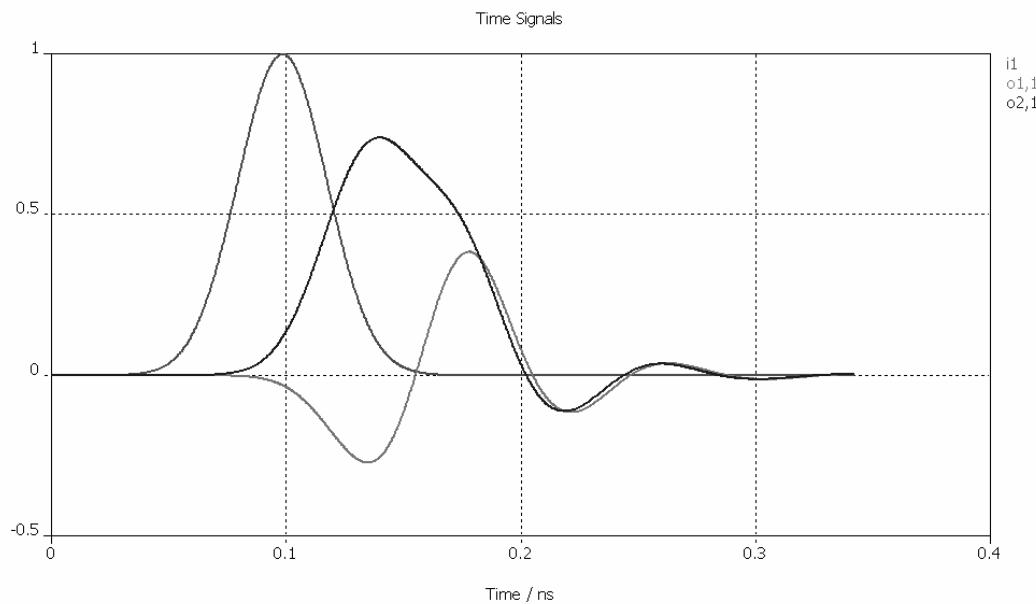


You may change the type of the scalar visualization by selecting a different visualization option in the corresponding dialog box: *Results*⇒*Plot Properties* (or *Plot Properties* in the context menu).

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

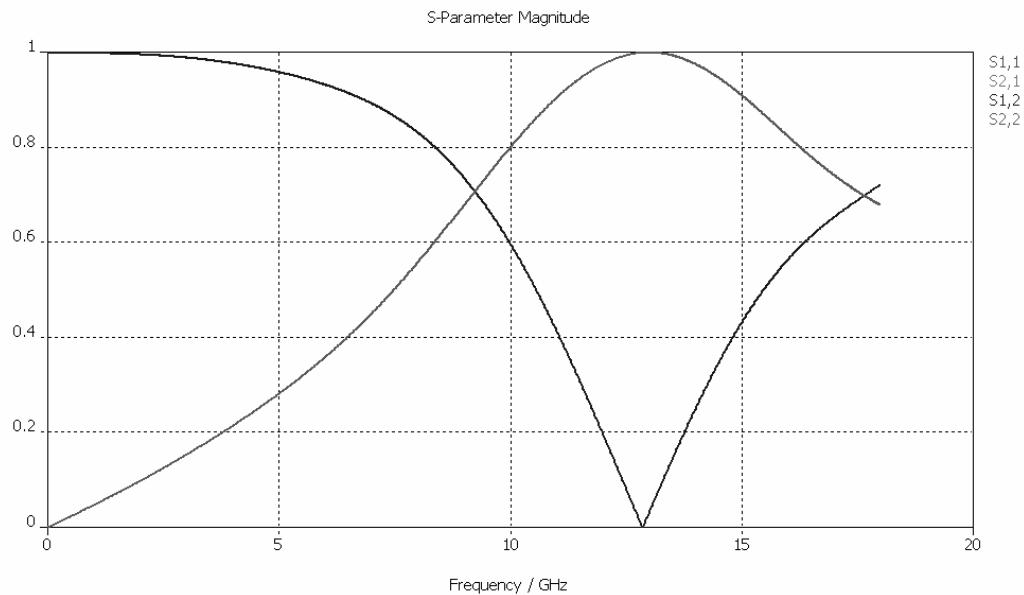
After a simulation has finished, you should always look at the time signals of the port modes. You can visualize these signals by choosing *NT*(navigation tree) \Rightarrow *1D Results* \Rightarrow *Port signals*. After selecting this folder, the following plot should appear:



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

To obtain a sufficiently smooth frequency spectrum of the S-parameters, it is important that all time signals decay to zero before the simulation stops. The simulation will stop automatically when this criterion is 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* \Rightarrow *1D Results* \Rightarrow *|S| linear*.

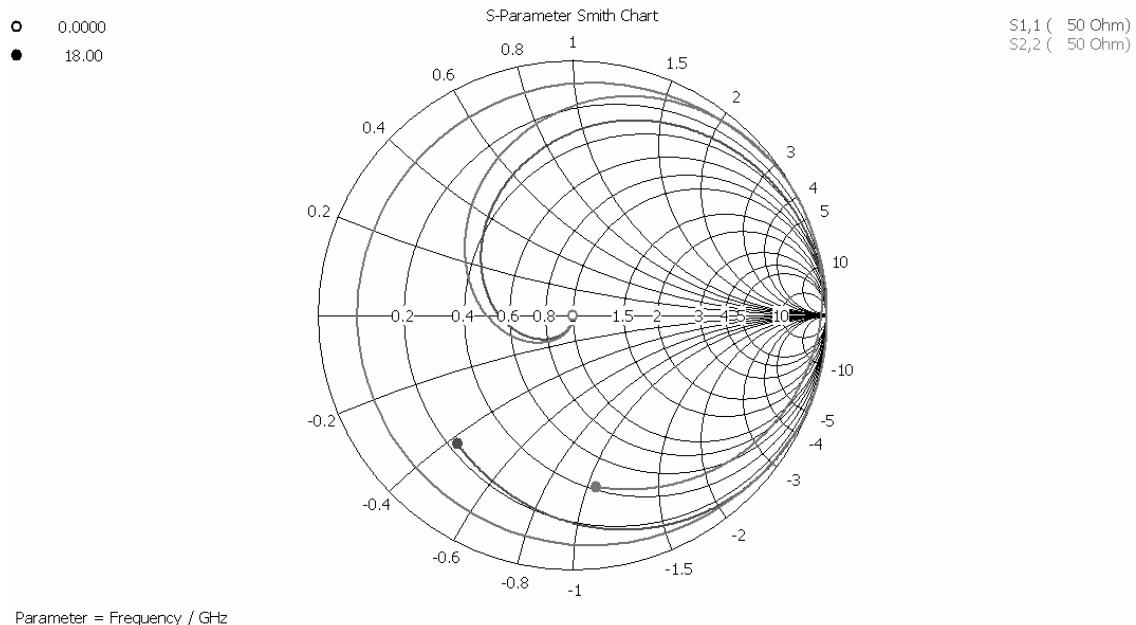


You can change the axis scaling by selecting *Results*⇒*1D Plot Options*⇒*Plot Properties* from the main menu (or the context menu). In addition, you can display and hide an axis marker by toggling *Results*⇒*1D Plot Options*⇒*Show Axis Marker*. The marker can be moved either with the cursor keys (*Left* 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) at about 12.87 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*).

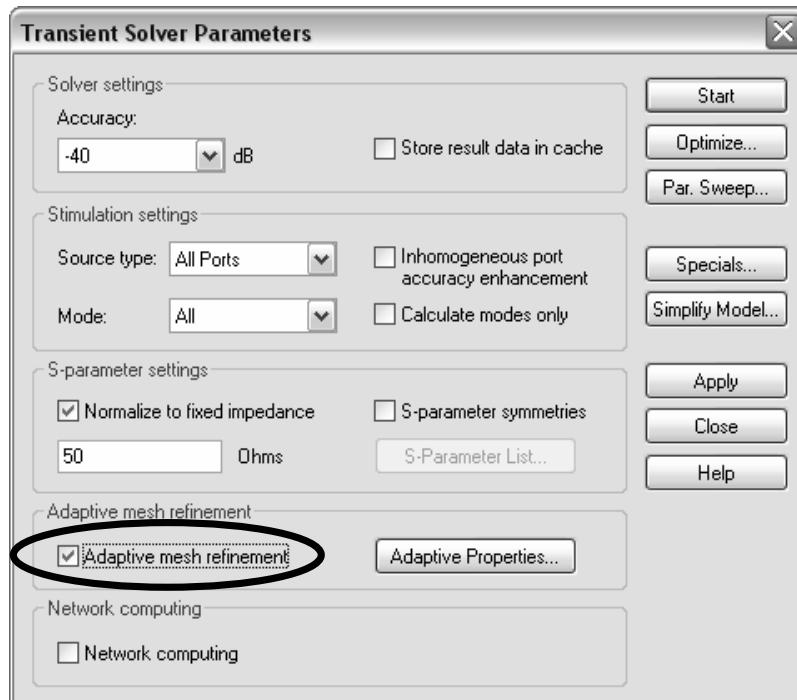


In this plot you can add markers to the curves by simply double-clicking on the corresponding positions on the curves. You may delete these markers in a properties dialog box: *Results*⇒*1D Plot Options*⇒*Plot Properties* (or *Plot Properties* from the context menu).

Adaptive Mesh Refinement

As already mentioned above, the mesh resolution influences the results. The expert system-based approach analyzes the geometry and tries to identify the parts that 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 settings.

Activate the adaptive mesh refinement by checking the corresponding option in the solver control dialog box.



Click the *Start* button. The solver will now perform several mesh refinement passes until the S-parameters no longer change significantly between two subsequent passes.

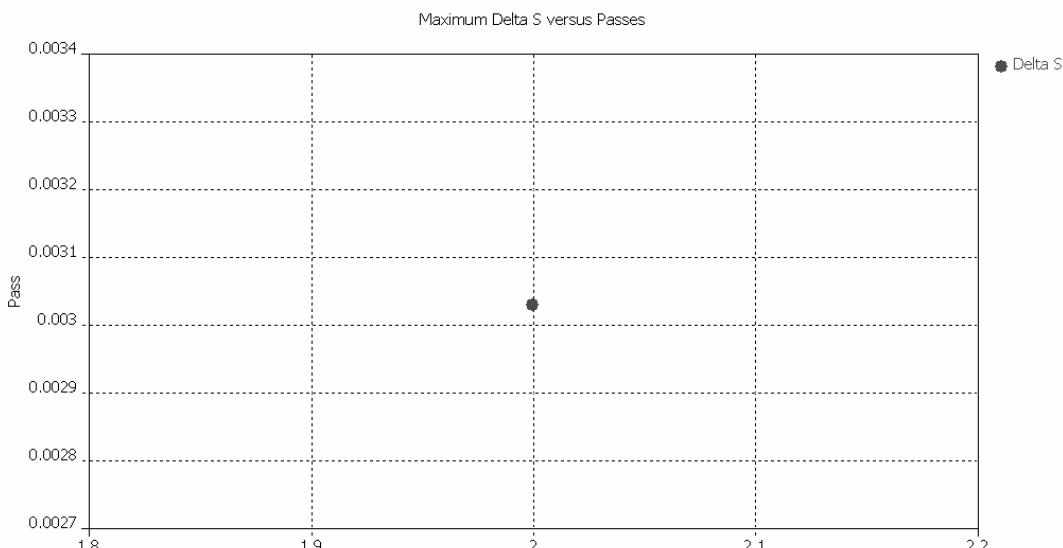
After two passes have been completed, the following dialog box will appear:



Since the automatic mesh adaptation 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 apply the determined rules to the structure even if it is modified afterward. This 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.

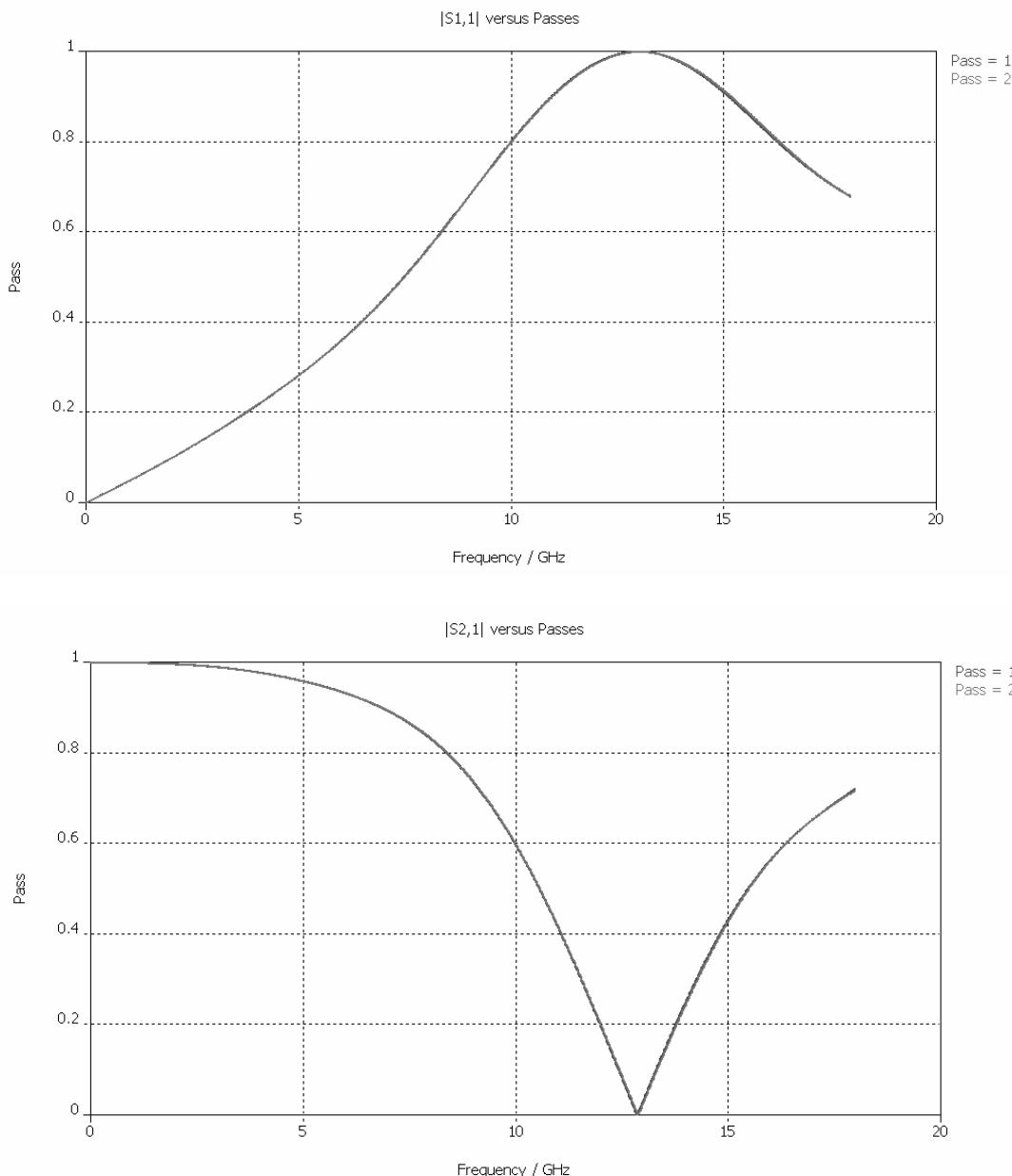
You should now confirm deactivation of the mesh adaptation by clicking the Yes button.

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*.



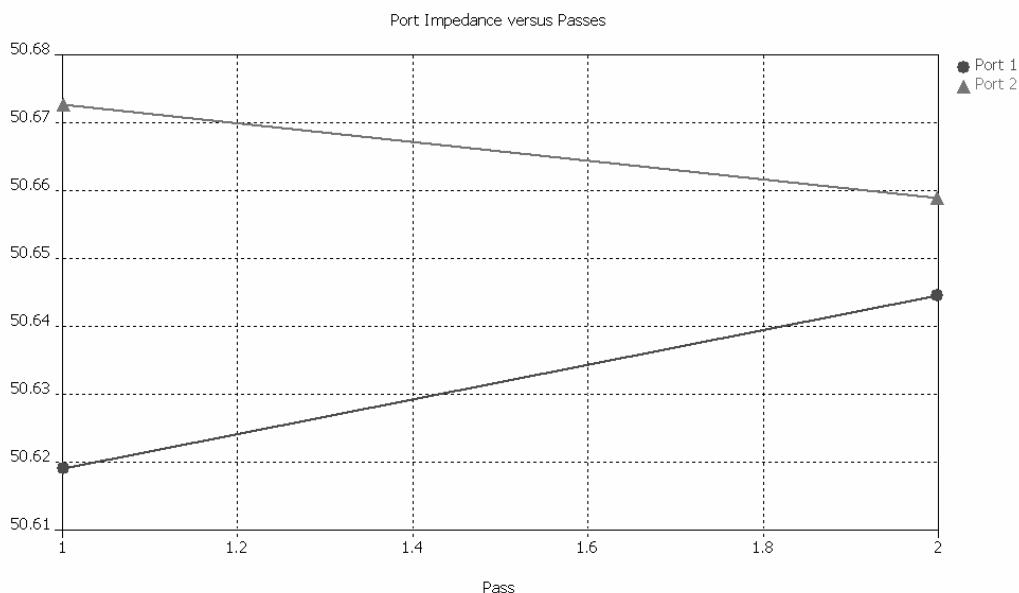
Since the mesh adaptation requires only two passes for this example, the Delta S curve consists of a single data point only. The result shows that the maximum difference of the S parameters from both runs is around 0.3% for the whole frequency range. The mesh adaptation stops automatically when the difference 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| linear⇒S1,1 versus Passes*, and *NT⇒1D Results⇒Adaptive Meshing⇒|S| linear⇒S2,1 versus Passes*, respectively.



You can see that 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 using the expert system generated mesh to the converged solution.

Another result from the adaptive meshing procedure is the ability for you to visualize how the port impedance changes versus the number of adaptive passes by selecting *NT⇒1D Results⇒Adaptive Meshing⇒Port Impedance*.



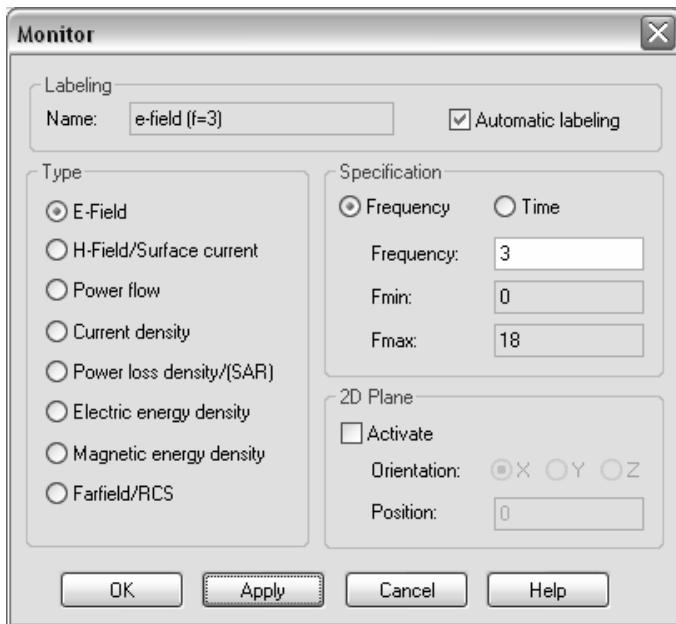
In practice it often proves wise to activate the adaptive mesh refinement to ensure convergence of the results. (This might not be necessary for structures with which you are already familiar when you can use your experience to refine the automatic mesh.)

Analyze the Electromagnetic Field at Various Frequencies

To understand the behavior of an electromagnetic device, it is often useful to get 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 that opens after choosing *Solve* \Rightarrow *Field Monitors* () from the main menu. You may need to switch back to the modeler mode by selecting the *Components* folder in the navigation tree before the monitor definition is activated.



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

For this analysis you should add the following monitors:

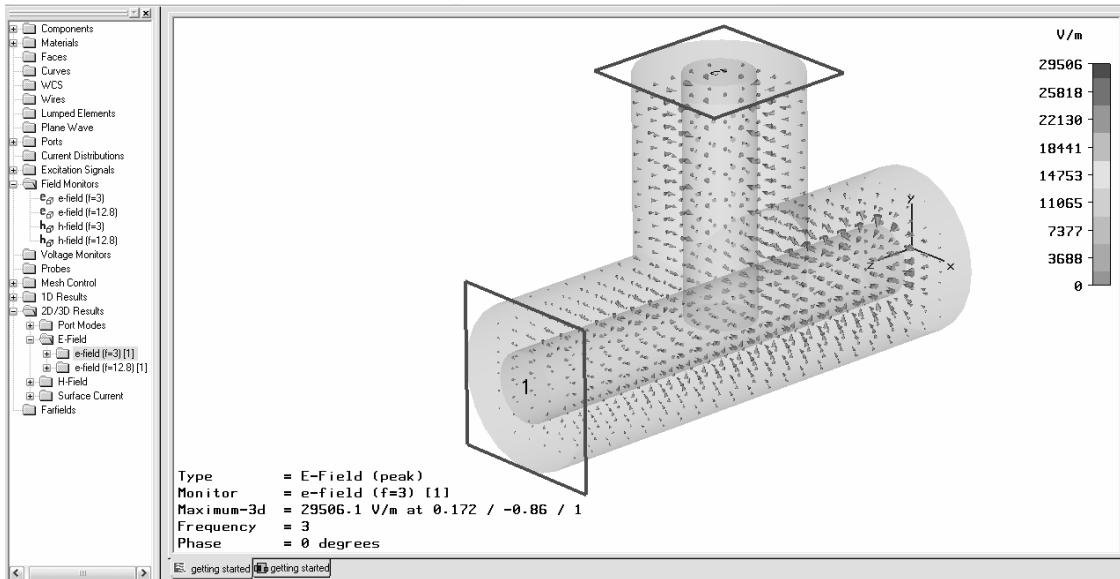
Field type	Frequency / GHz
E-Field	3
E-Field	12.8
H-Field	3
H-Field	12.8

All defined monitors are listed in the *NT(navigation tree)⇒Monitors* folder. Within this folder you may select a particular monitor to reveal its parameters in the main view.

You should now run the simulation again. When the simulation finishes, 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*).

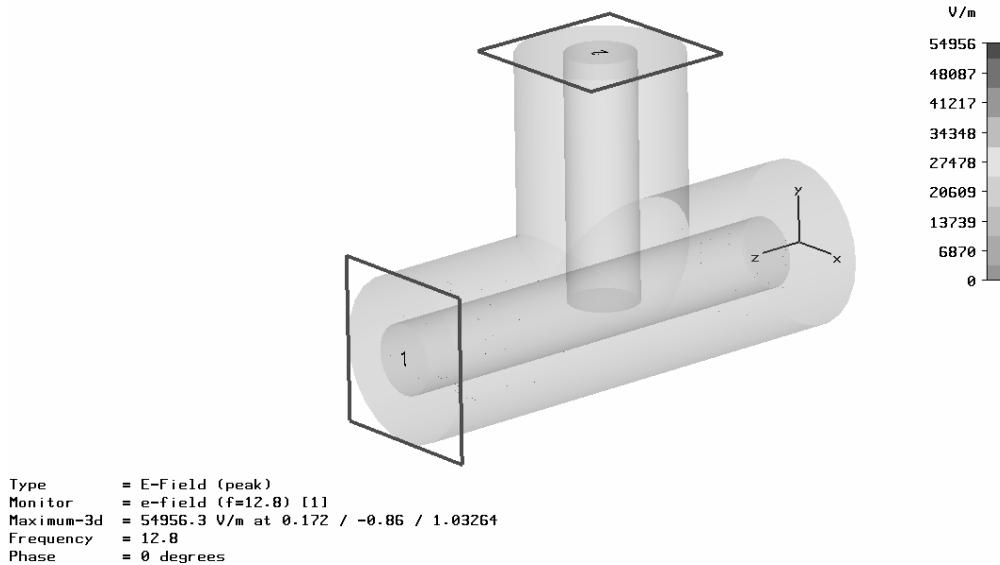
Note: Since you have specified a full S-matrix calculation, two simulation runs would generally be required. For each of these runs, the field would be recorded as specified in the monitors, and the results would be presented in the navigation tree, giving the corresponding stimulation port in brackets. However, in this loss free example, the second run is not necessary, so you will find that the monitor data are 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 3D electric field distribution by selecting *NT* \Rightarrow *2D/3D Results* \Rightarrow *E-Field* \Rightarrow *e-field(f=3)*[1]. The plot should look similar to the picture below:



You should now play around with the various field visualization options for the 3D vector plot (*Results* \Rightarrow *Plot Properties*).

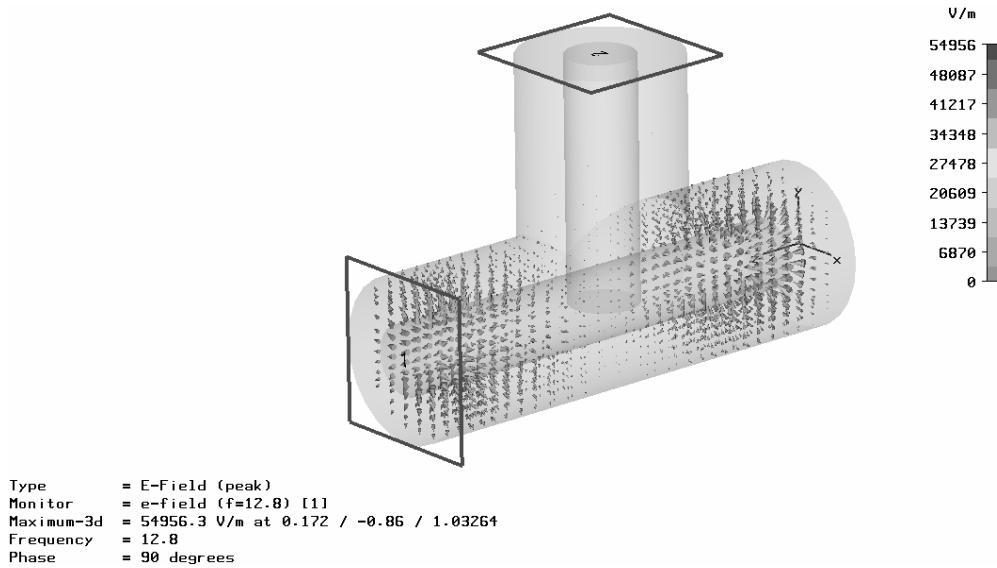
If you select the electric field at 12.8 GHz (*NT* \Rightarrow *2D/3D Results* \Rightarrow *E-Field* \Rightarrow *e-field(f=12.8)*[1]), you may obtain a plot showing very small arrows only:



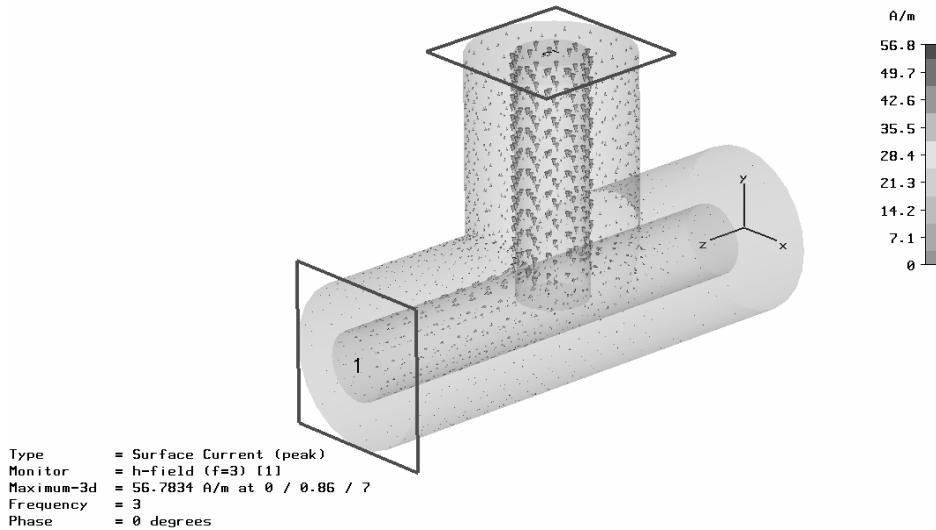
This problem is partly due to the selected phase of the fields which you can change by using the left / right cursor keys or by modifying the *Phase* value to 90 degrees in the *Results* \Rightarrow *Plot Properties* dialog box.

Another reason for the small arrow size is the very large fields at the edges of the conductor. Because of the finite number of arrows drawn in the structure, there might be no arrows at all visualized at the singularities. However, since the field maximum is taken as a reference for the arrow scaling, the smaller fields inside the volume will be visualized with very small arrows only. The visualization can be improved by adjusting the object scaling or density in the *Results* \Rightarrow *Plot Properties* dialog box.

After adjusting these settings, you should obtain a useful plot similar to the following:



The surface currents can be visualized by selecting *NT⇒2D/3D Results⇒Surface Current⇒h-field(f=3)[1]*. You should obtain a plot similar to the following picture:



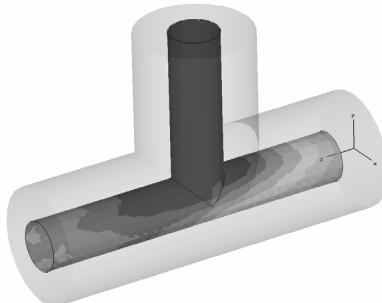
You may now change the plot options in the plot dialog box: *Result⇒Plot Properties* (or *Plot Properties* from the context menu). You can obtain a field animation by clicking the *Start* button located in the *Phase/Animation* frame in this dialog box. Here the phase of the field will be automatically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

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

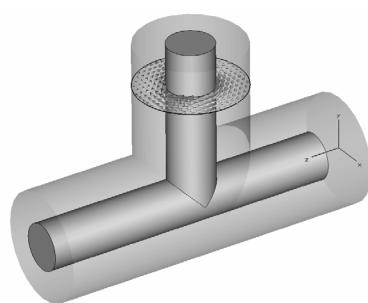
After obtaining a rough overview of the electromagnetic field distribution in 3D, you can inspect the fields in more detail by analyzing some cross sectional cuts through the structure. To do this, select an electric or magnetic field (no surface currents) for display

and toggle the *Results \Rightarrow 3D Fields on 2D Plane* () option on. The same plot options are available in the 2D plot mode you have already used for the port mode visualization. Since the data are derived 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 \Rightarrow Plot Properties* dialog box by changing the *Cutplane Control* and *Location* settings at the bottom of the dialog boxes.

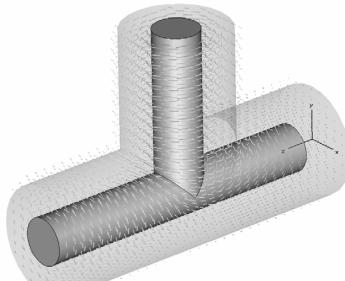
Due to the limited space, not all plotting options can be explained here. However, the following gallery shows some possible plot options. Can you reproduce them?



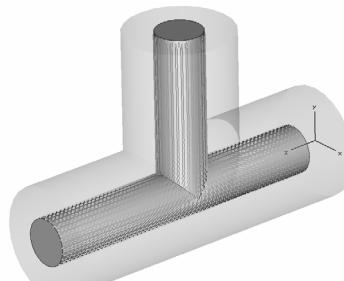
Tangential component of surface current at 3 GHz



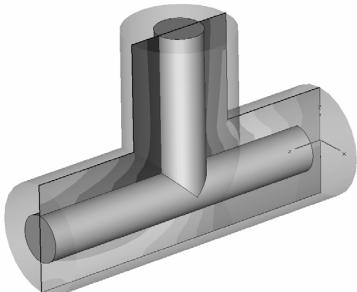
Vector plot of h-field at 3 GHz using *3D Fields on 2D Plane* option



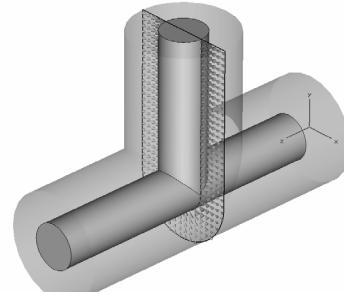
3D vector plot of h-field at 3 GHz using *hedgehog* option



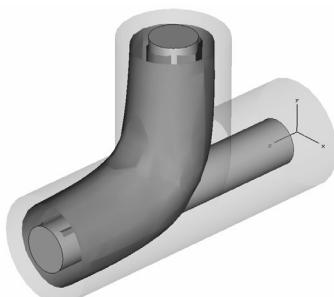
3D vector plot of surface current at 3 GHz using *hedgehog* option



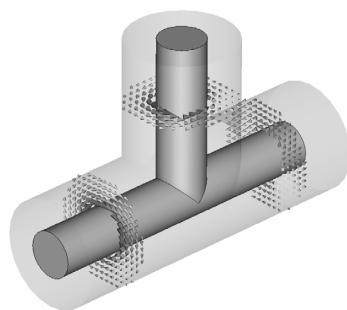
X component of h-field at 3 GHz using *3D Fields on 2D Plane* option



Vector plot of e-field at 3 GHz using *3D Fields on 2D Plane* option



Abs plot of h-field at 3 GHz



Several 3D Field on 2D Plane plots of h-field using *Overlay Multiple Plots* option

Parameterization of the Model

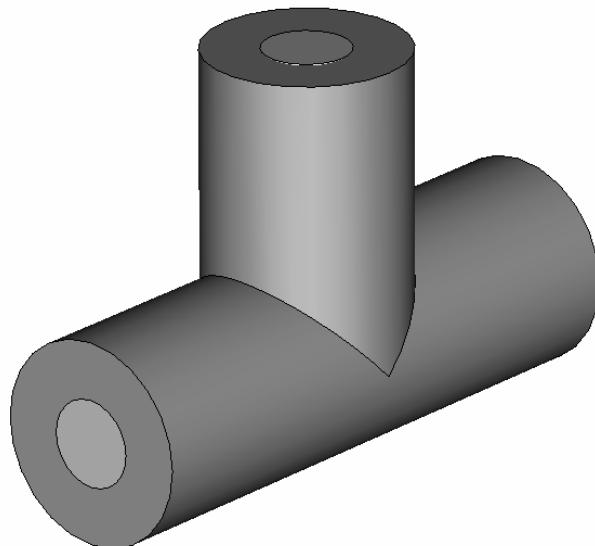
The steps above demonstrate how to enter and analyze a simple structure. However, structures will usually be analyzed in order to improve their performance. This procedure may be called “design” in contrast to the “analysis” done before.

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

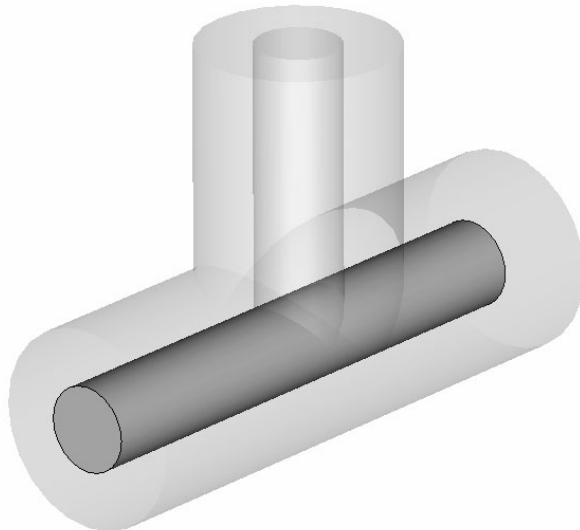
CST MICROWAVE STUDIO® offers a lot of options to parametrically describe the structure in order to easily change its parameters. The *History List* function, as described previously, is a general option, but for simple parameter changes there is an easier solution described below.

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 to enter the modeler mode by selecting the *NT*(navigation tree) \Rightarrow *Components* folder.

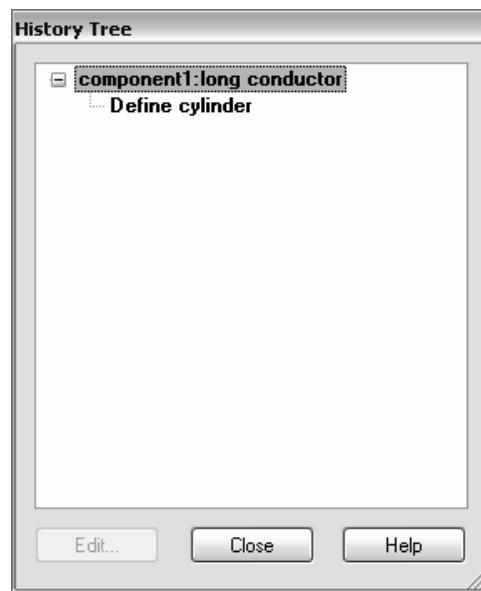
Select all ports by clicking on the *NT* \Rightarrow *Ports* folder. Then press the right mouse button to choose *Hide All Ports* from the context menu. The structure plot should look like this:



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

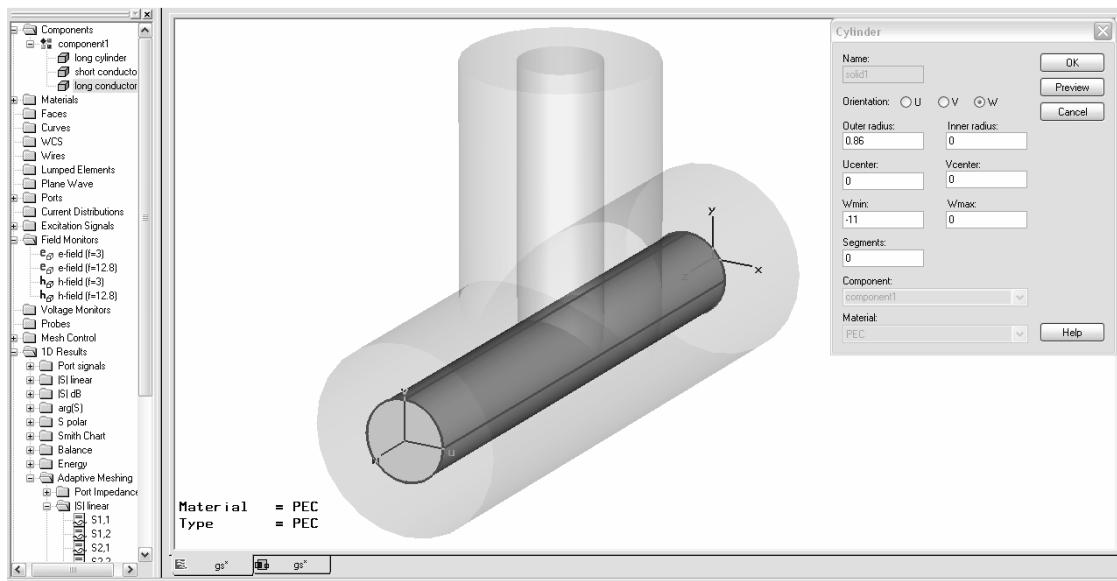


You can now choose *Edit*⇒*Object Properties* (or *Properties* from the context menu) which will open a list showing the history of the shape's creation:

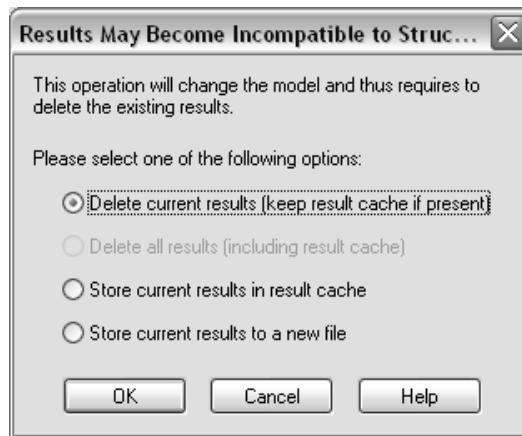


Select the “Define cylinder” operation in the tree folder “component1:long conductor” from the history tree (see above). The corresponding shape will be highlighted in the main window.

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

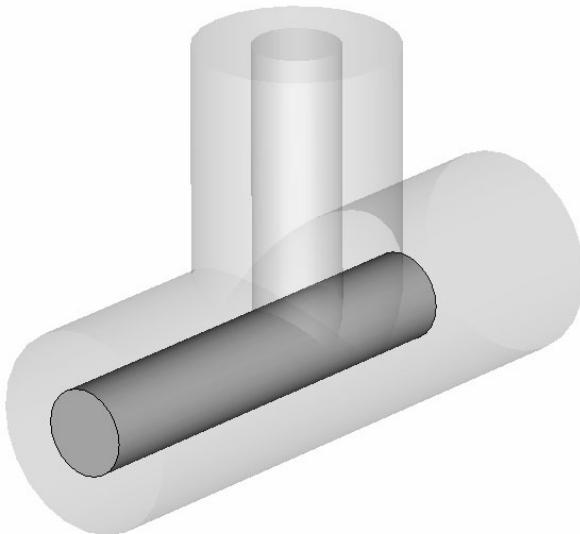


In this dialog box you will find the length of the cylinder ($W_{min}=-11$) as specified during the shape creation. Change this parameter to a value of -9 and click **OK**. Since you are going to change the structure, the previously calculated results will no longer match the current structure; therefore, 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 click **OK**.

After a few seconds, the structure plot will change showing the new structure with the different stub length.



You can now dismiss the History Tree dialog box by clicking the *Close* button.

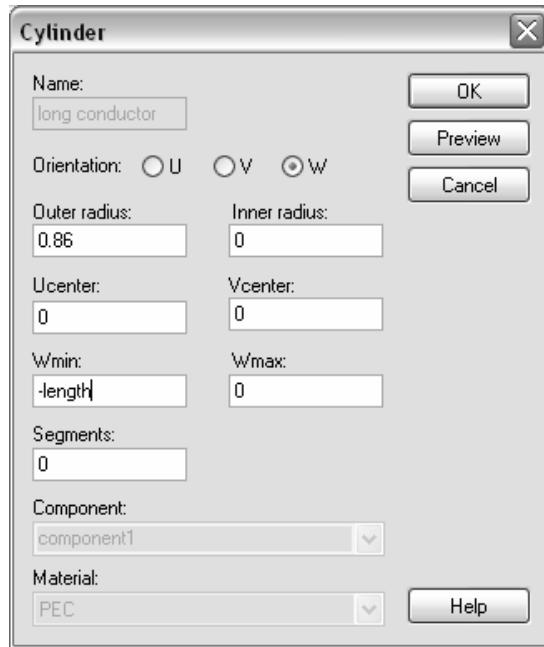
Generally, you can change all parameters of any shape by selecting the shape and editing its properties. This fully parametric structural modeling is one of CST MICROWAVE STUDIO®'s most outstanding features.

The parametric structure definition also works if some objects have been constructed relative to each other 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.

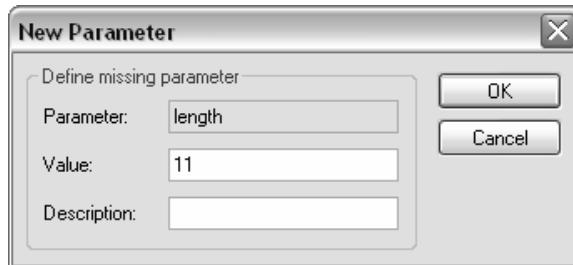
Changes in parameters occasionally alter the topology of the structure too severely so that the structure update may fail. In this case, the History List function offers powerful options to circumvent these problems. Please refer to the online documentation, or contact technical support.

In addition to directly changing the parameters, you may also assign variables to the structure's parameters. The easiest way to do this is to enter a variable name in an expression field rather than a numerical value. Therefore, you should now open the cylinder dialog box again as shown above. Afterward, you can just enter the string “-length” in the *Wmin* field.

The dialog box should look as follows:

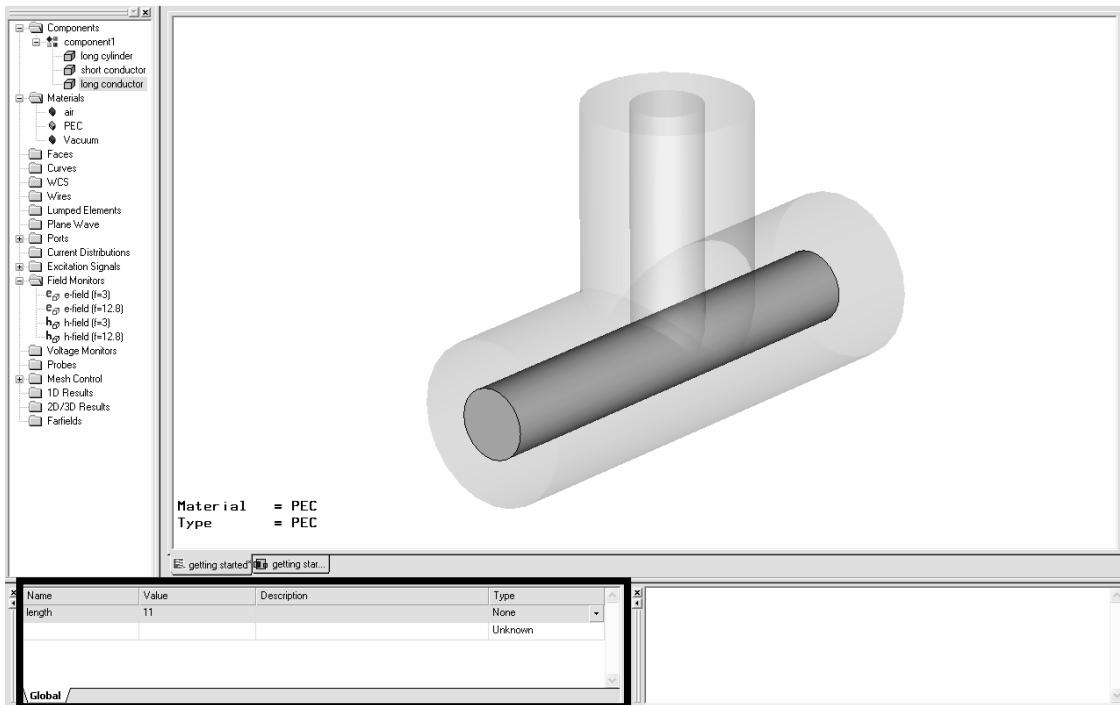


Since the parameter “length” is still undefined, a new dialog box will open after you click OK in the cylinder dialog box:

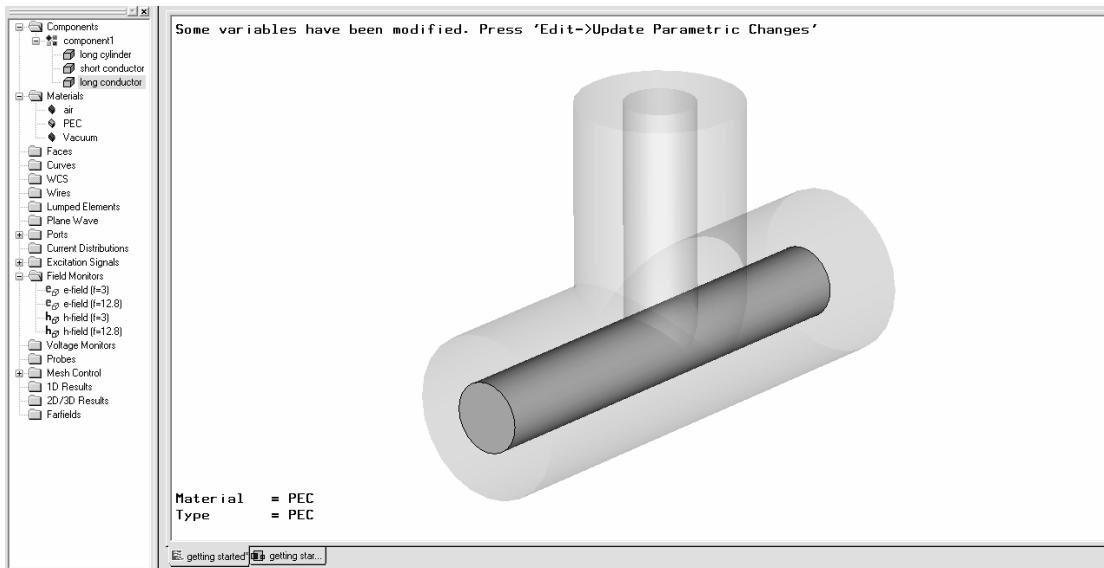


You can now assign a value to the new parameter by entering 11 in the *Value* field. You may also provide a text in the *Description* field so that you can later remember the meaning of the parameter. Click *OK* to create the parameter and update the model.

All defined parameters will be listed in the parameter docking window as shown below:



You can change the value of this parameter in the *Value* field. Afterward, the message “*Some variables have been modified. Press Edit->Update Parametric Changes*” will appear in the main view.

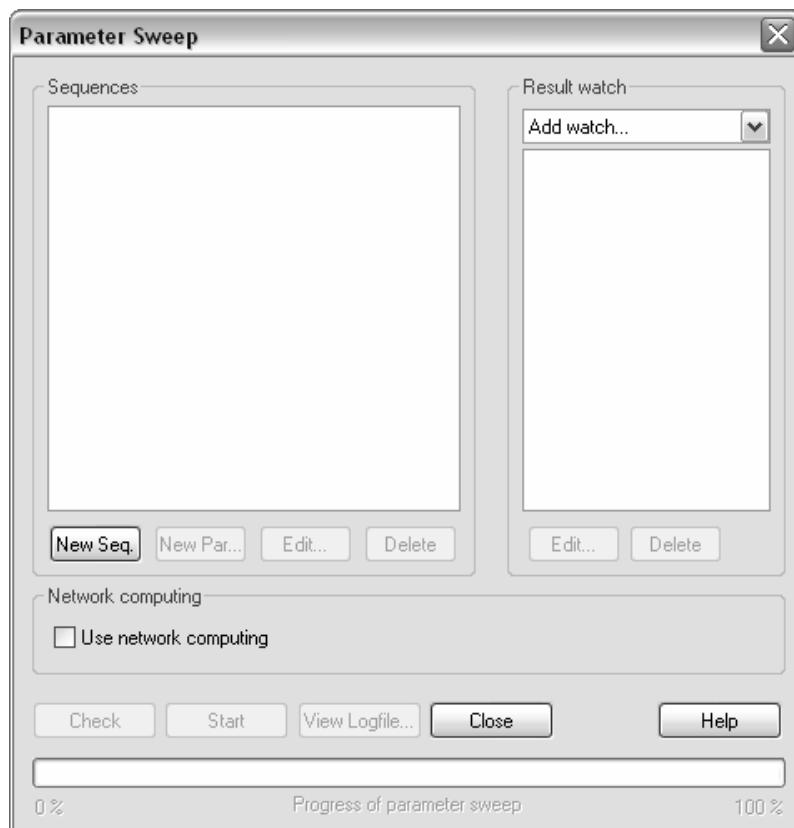


In addition to using the menu command in order to update the structure, you can select the corresponding toolbar button (). You can also select *Update* from the context menu which appears when you press the right mouse button in the parameter list. You may need to click on an empty field in the list first in order to obtain this context menu.

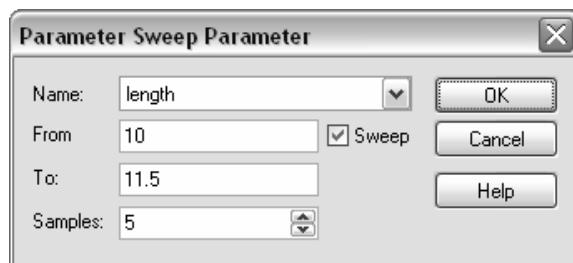
When performing this update operation, the structure will be regenerated according to the current parameter value. You can verify that parameter values between 7 and 11.5 give some useful results. The function *Edit⇒Animate Parameter* is also useful in this regard.

Parameter Sweeps and Processing of Parametric Result Data

Since you have now 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 you can access from within the transient solver dialog box by clicking the *Par. Sweep* button to reveal the following dialog box:

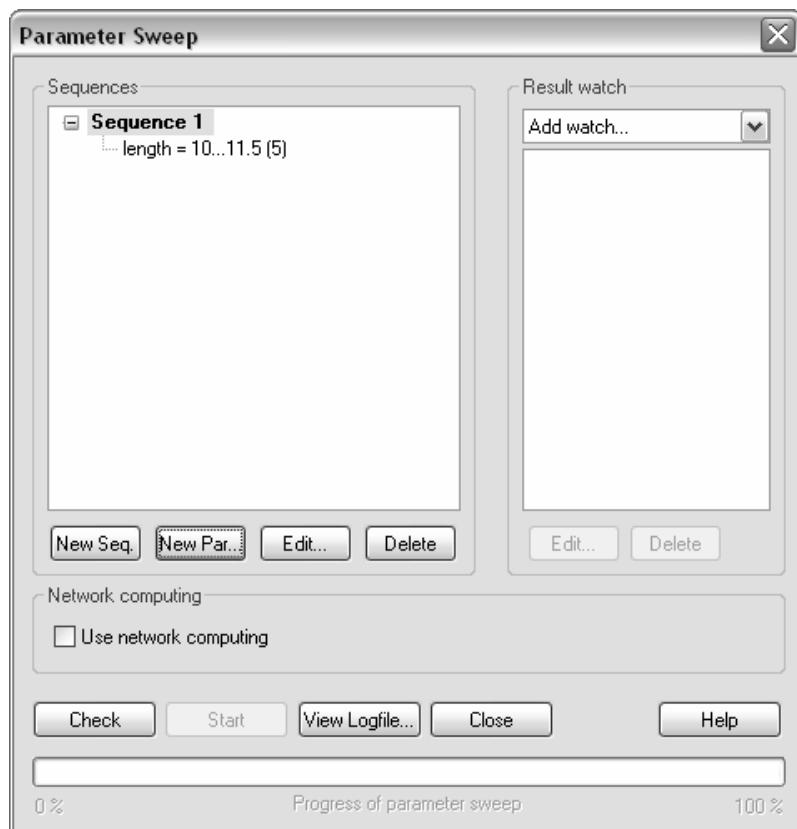


In this dialog box you can specify calculation “sequences” which will consist of various parameter combinations. To add such a sequence, click the *New Seq.* button now. Then click the *New Par...* button to add a parameter variation to the sequence:



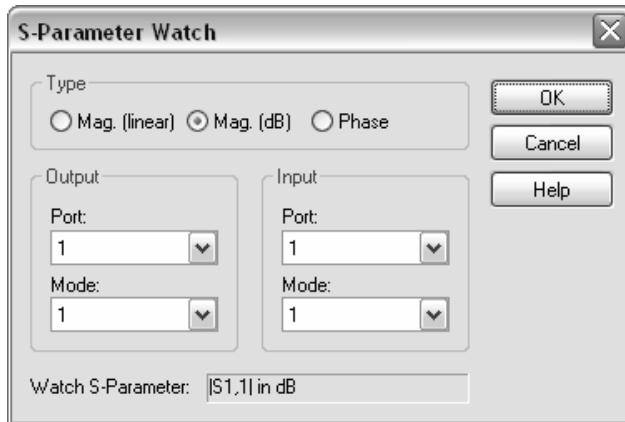
In the resulting dialog box you can select the name of the parameter to vary in the *Name* field. Then you can specify the lower (*From*) and upper (*To*) bounds for the parameter variation after checking the *Sweep* item. Finally enter the number of steps in which the parameter should be varied in the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

In this example, you should perform a sweep *From 10.0 To 11.5 with 5 Samples*. After you click the *OK* button, the parameter sweep dialog box should look as follows:



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. A dialog box opens in which you can specify an S-parameter to store:

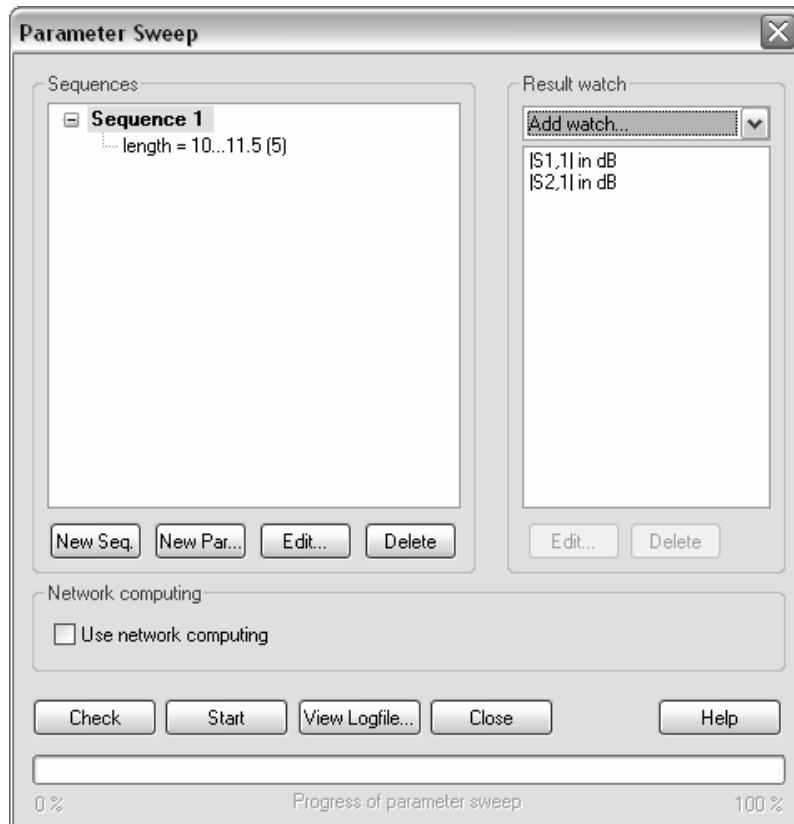


First select the option of recording the magnitude of S1,1 in dB by checking *Mag. (dB)* in the *Type* field and clicking the *OK* button.

Next 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. Click *OK*.

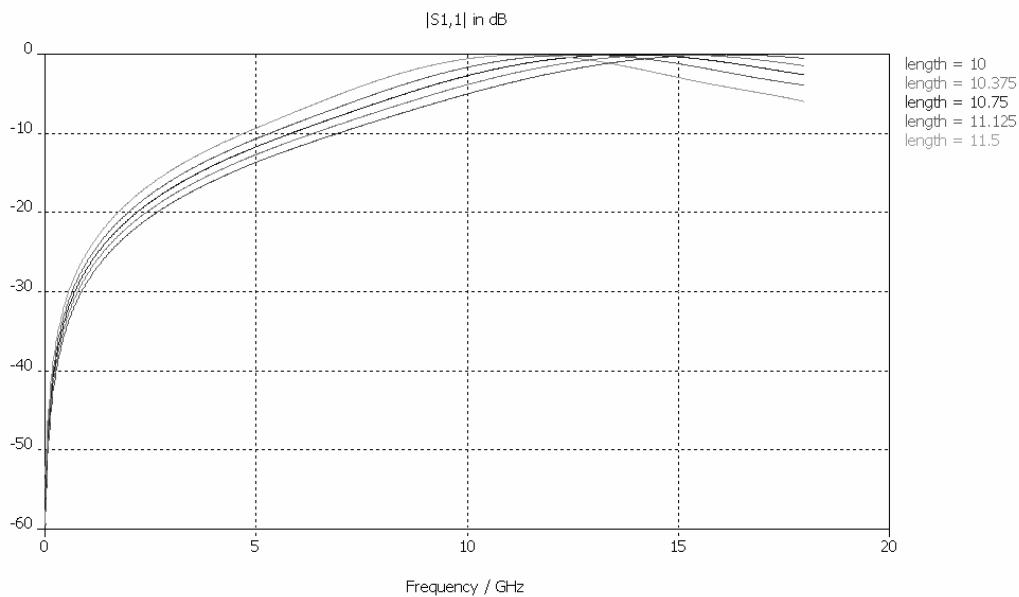
The parameter sweep dialog box should look as follows:



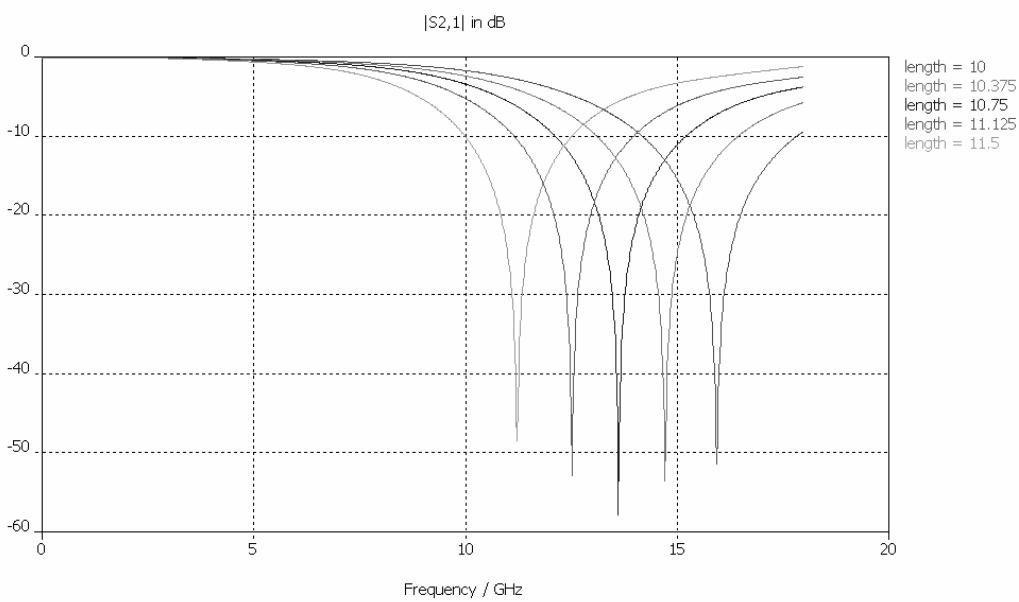
Now start the parameter sweep by clicking the *Start* button.

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), make sure that the modified settings are stored by clicking *Apply* in the solver control dialog box.

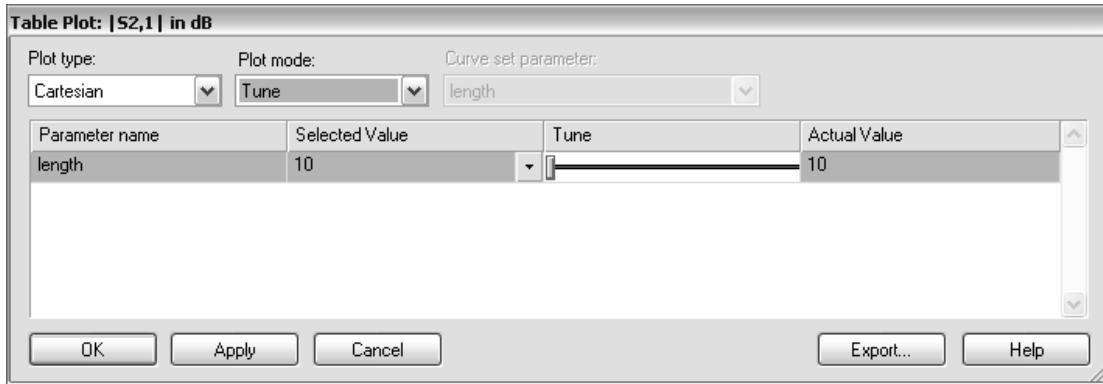
After the solver has finished its work, close the dialog box by clicking the *Close* button. The navigation tree will contain a new item called “Tables” from which you should select the item *Tables* \Rightarrow $|S1,1| \text{ in dB}$ first:



Similarly, you can also plot the magnitude of the transmission coefficient by selecting *Tables* \Rightarrow |S_{2,1}| in dB.



More properties for parametric data visualization can be accessed from the table properties dialog box (*Results*⇒*Table Properties*):

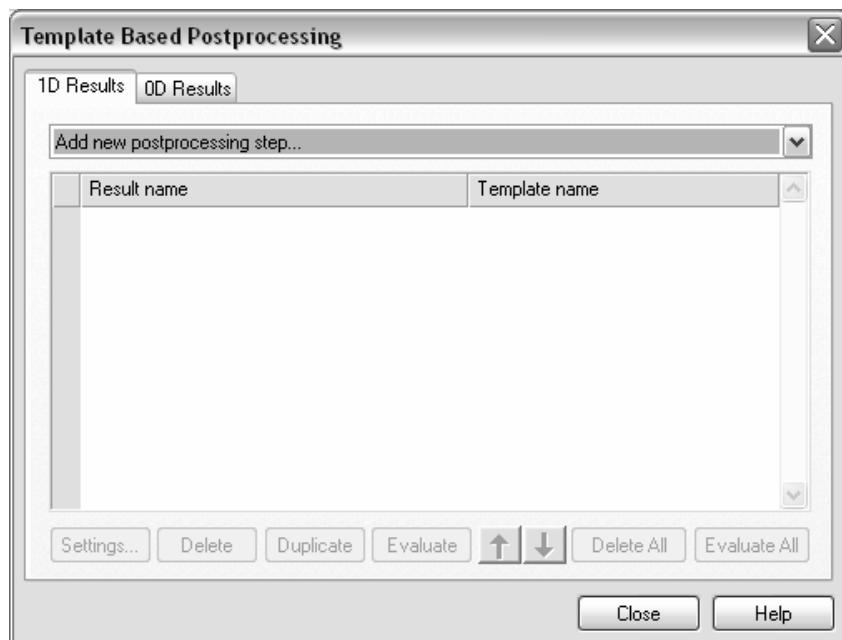


After you change the *Plot mode* to *Tune*, a slider appears which provides quick access to the parametric result data. Dragging the slider allows you to quickly select the corresponding result for the parameter value as shown in the *Selected Value* or *Actual Value* fields.

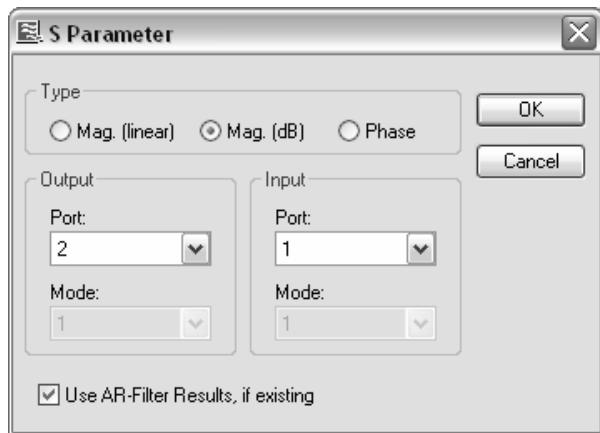
Note that no interpolation of result data is performed here. For more than one parameter it may, therefore, be that no parametric result is available for the currently *Selected Values*. In this case, that data point whose parametric settings are closest to the selected ones will be displayed. However, the actual parametric settings of the currently displayed data point will always be shown in the *Actual Value* field.

Once you close the table properties dialog box by clicking *OK* or *Cancel*, the parametric view will be restored. Refer to the online documentation for more information about the many options for displaying parametric data.

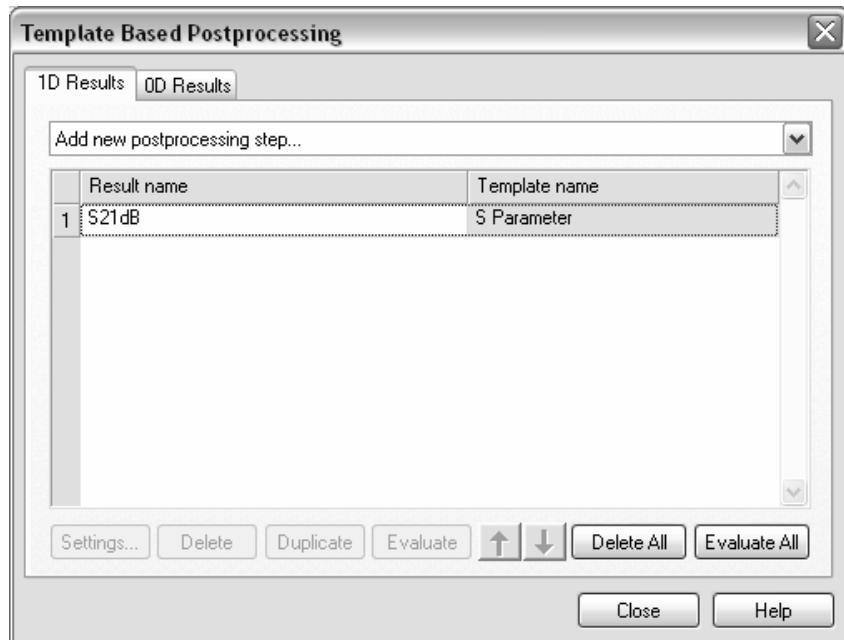
It may be interesting to see how the location of the transition minimum changes as a function of the parameter. This and other special result data can be automatically computed using *Result Templates*. Open the corresponding dialog box by choosing *Results*⇒*Template Based Postprocessing* ():



First, load the transmission S-parameter data into the post processing chain in order to later derive its minimum location. Select the post-processing step *S Parameter* from the list of available 1D result templates to open the following dialog box:

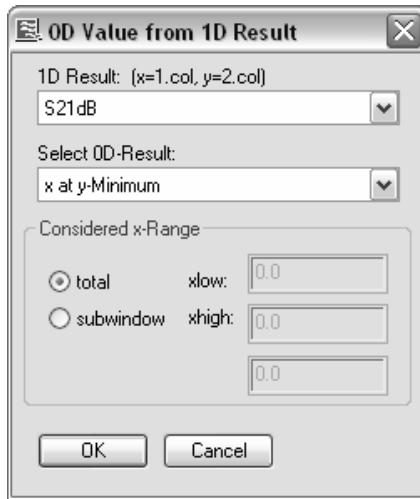


In this dialog box, you should specify $|S_{21}|$ in dB scaling (similar to the settings before in the parameter sweep). The new post-processing step will be listed in the dialog box:



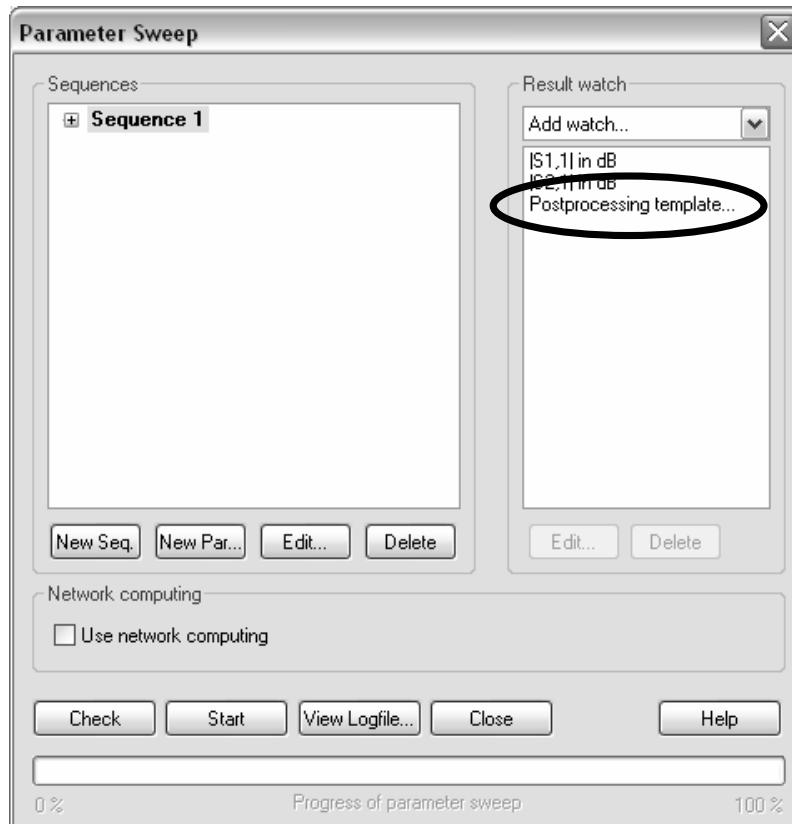
Based on the broadband S-parameter data, you can now extract the location of the minimum which is a single data point (or *0D Result*). Switch to the *0D Results* page and select *0D Value From 1D Result* from the list available 0D result templates.

A dialog box will open where you can specify details about the post-processing step:



The only available 1D data is $|S21|$ in dB, so you have no choice in the *1D Result* list. Since you want to know the location of the curve (*y*-) minimum, you should choose *x at y-Minimum* as the desired result from the *Select 0D Result* list. Clicking *OK* will complete the definition of the specific post processing chain in this example.

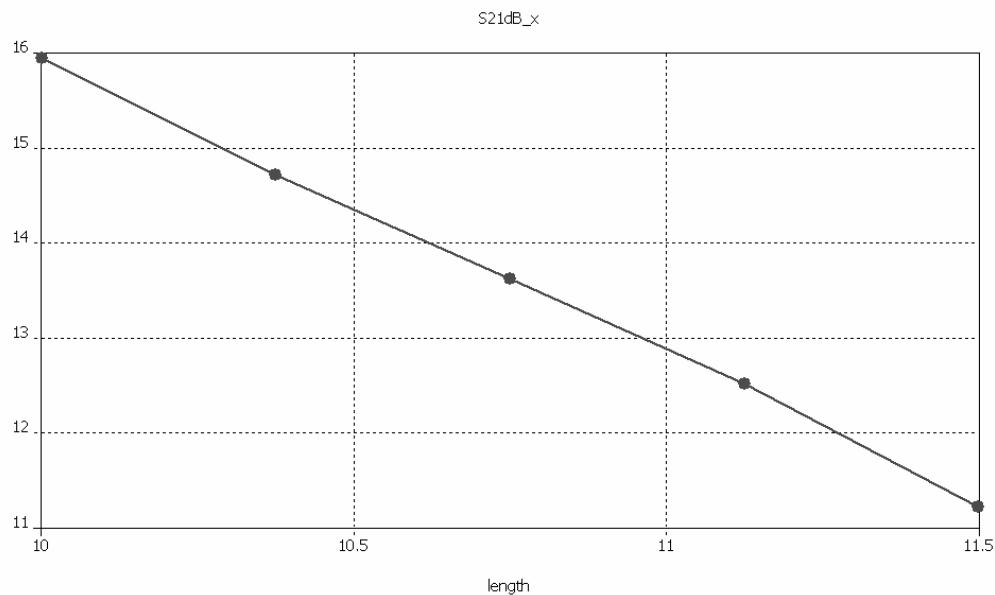
After any solver run, these post-processing operations are automatically carried out, and the result of each of these steps is stored in a table. Once a *Result Template* is defined, a corresponding line is automatically added to the *Result watch* list:



Restart the entire parameter sweep now. You will be asked what to do with the currently available result data. In this dialog box, select *Delete current results* and click *OK*.

Of course, the additional post processing steps could also be defined before the first parameter sweep is started, but for didactical reasons we have separated this task into two steps here.

Once the solver finishes the parameter sweep, you can access the result template data from the *Tables* \Rightarrow *0D Results* \Rightarrow *S21dB_x* folder in the navigation tree:



This curve clearly illustrates how the location (=frequency) of the transition minimum changes as a function of the geometrical parameter.

Because of the limited scope of this manual, we give a very brief introduction to this powerful feature, so please refer to the online documentation for more information.

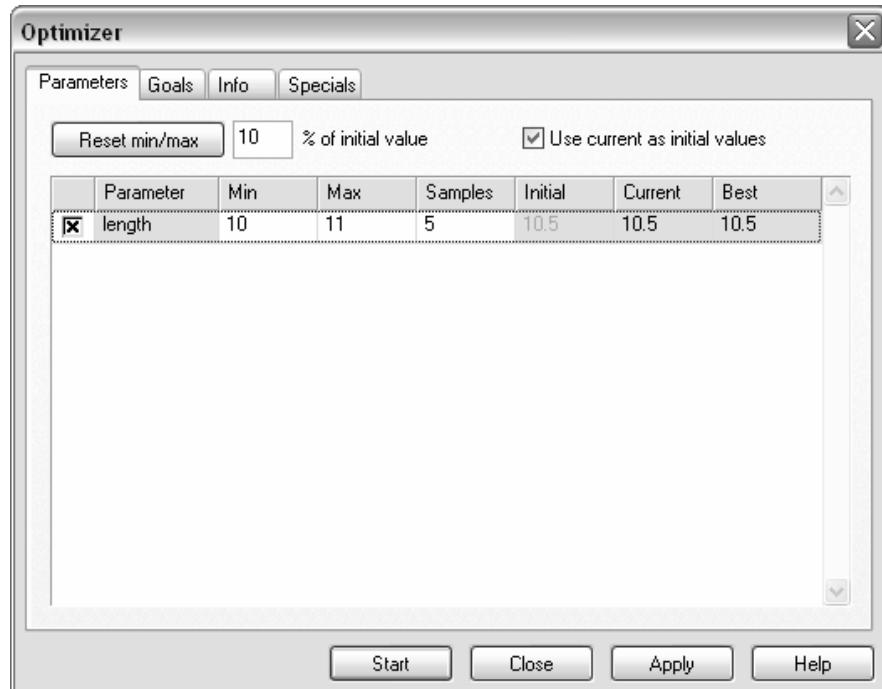
Automatic Optimization of the Structure

Let's now assume you wish to obtain the minimum of the transmission S2,1 at 13 GHz (which can be achieved within a parameter range of 10 to 11 according to the curve above). By measuring the curve (activate the axis marker tool by choosing *Results* \Rightarrow *1D Plot Options* \Rightarrow *Axis Marker*), you can check that the desired parameter value is around 10.96. However, figuring out the exact parameter value may be a lengthy task which can be performed equally well automatically.

Before you continue to optimize this structure, set the length parameter to a value within the valid parameter range (e.g. 10.5) and update the structure. You must enter the modeler mode (e.g. by clicking at the “Components” item in the navigation tree) before you can modify the parameters.

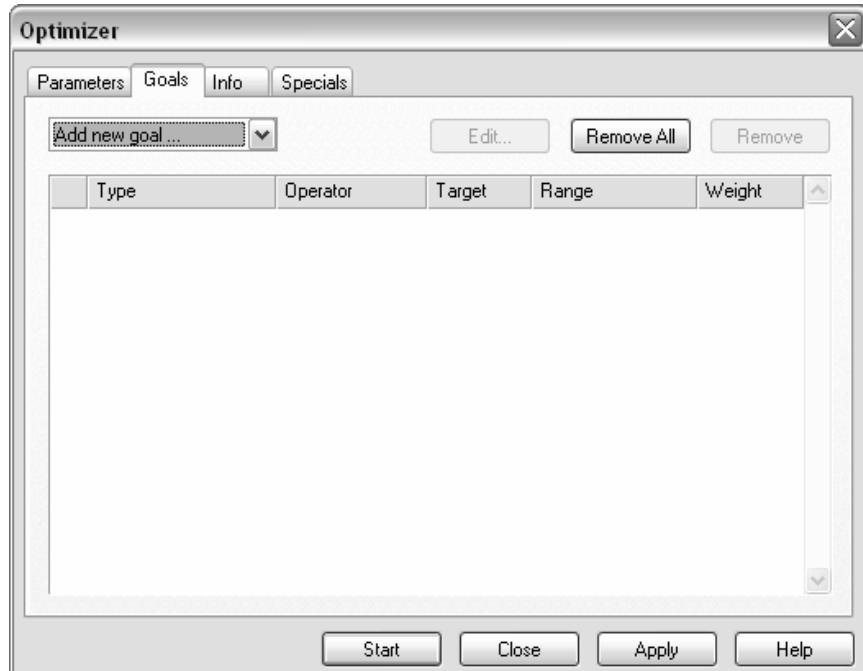
CST MICROWAVE STUDIO® offers a powerful built-in optimizer feature for these kinds of parametrical optimizations.

To use the optimizer, simply open the *transient solver* control dialog box as before. In this dialog box, click the *Optimize* button to open the optimizer control dialog box:



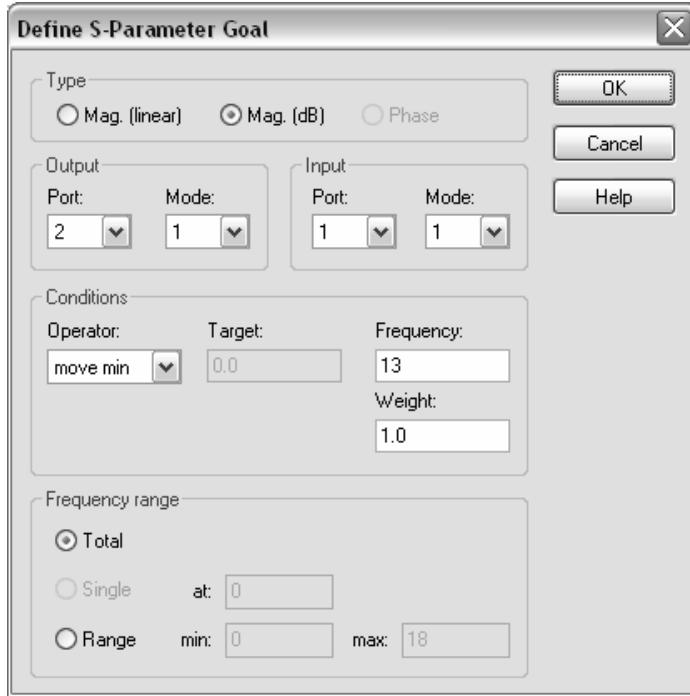
First check the desired parameter(s) for the optimization in the *Parameters* tab of the optimization dialog box (the “length” parameter should be checked). Next specify the minimum and maximum values for this parameter during the optimization. Enter a parameter range between 10 and 11. For this example, the other settings can be kept as default. Refer to the online documentation for more information on these settings.

The next step is to specify the optimization goal. Click on the *Goals* tab.



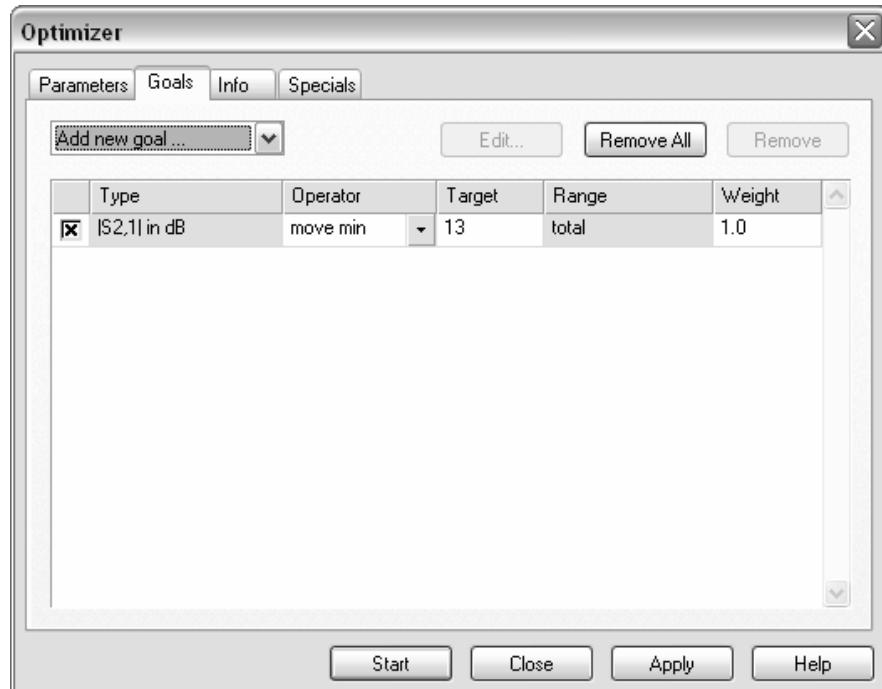
Now you can specify a list of goals to achieve during the optimization. In this example, the target is to move the minimum of the S-parameter S2,1 to a given frequency, so define an “S Parameter” goal by selecting it from the list that appears when you select

the *Add new goal* entry. Once you have selected the proper goal type, the following dialog box should appear:



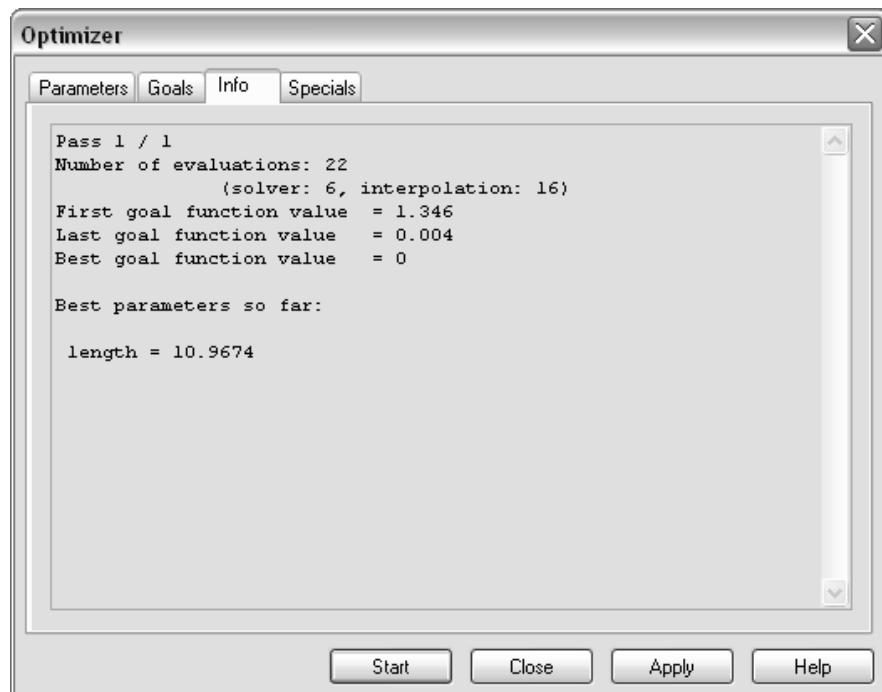
In this dialog box, you should first select the magnitude of the S-parameter in dB for optimization by clicking on the *Mag.(dB)* entry in the *Type* frame. Next specify which S-parameter should be optimized. Select S2,1 by setting *Port* = 2, *Mode* = 1 in the *Output* frame and *Port* = 1, *Mode* = 1 in the *Input* frame.

Now specify the actual goal for the previously specified S-parameter data. Since you want to move the minimum of S2,1 in this example, you should select the *move min* operator in the conditions frame. Afterwards, 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 defaults. After you click *OK*, the optimizer dialog box should look as follows:



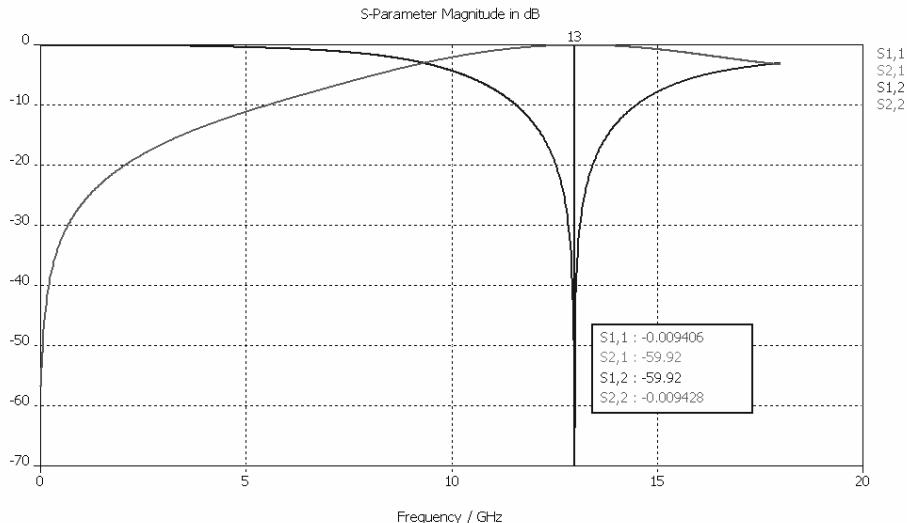
Since you now have specified optimization parameters and goals, the next step is to start the optimization procedure by clicking the *Start* button. The optimizer will show the progress of the optimization in an output window in the *Info* tab which will be activated automatically.

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



Note that due to the sophisticated optimization technology, only six transient solver runs were required to find the optimal solution with high accuracy.

You can now visualize the S-parameters for the optimal parameter setting (length =10.967) and should obtain the following picture (you can activate the axis marker tool by choosing *Results*⇒*1D Plot Options*⇒*Axis Marker* to verify that the location of the peak is at 13 GHz).



Instead of defining a “move min” goal for the optimization, you could also have chosen to optimize the value of the previously defined result template *S21dB_x* to be equal to the desired resonance frequency of 13 GHz.

Comparison of Time and Frequency Domain Solver Results

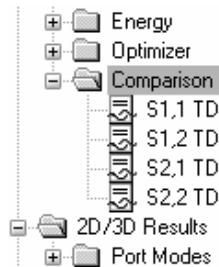
So far, all explanations have focused on the transient solver. In the next steps you will compare the results of the transient solver based on hexahedral grids with the frequency domain solver using tetrahedral grids. Since these two simulation methods are based on different techniques, this comparison allows you to verify the accuracy of the results. Although the transient solver is much faster for this and many other examples, the frequency domain solver may be the better choice for lower frequency problems or filters. In the latter case, the primary simulation and optimization should be performed using the frequency domain solver whereas the final verification can then be done using the transient solver. The seamless combination of these different techniques in a homogeneous environment is another outstanding feature of CST MICROWAVE STUDIO®.

The frequency domain solver may not be available to you due to license restrictions. Please contact your sales office for more information.

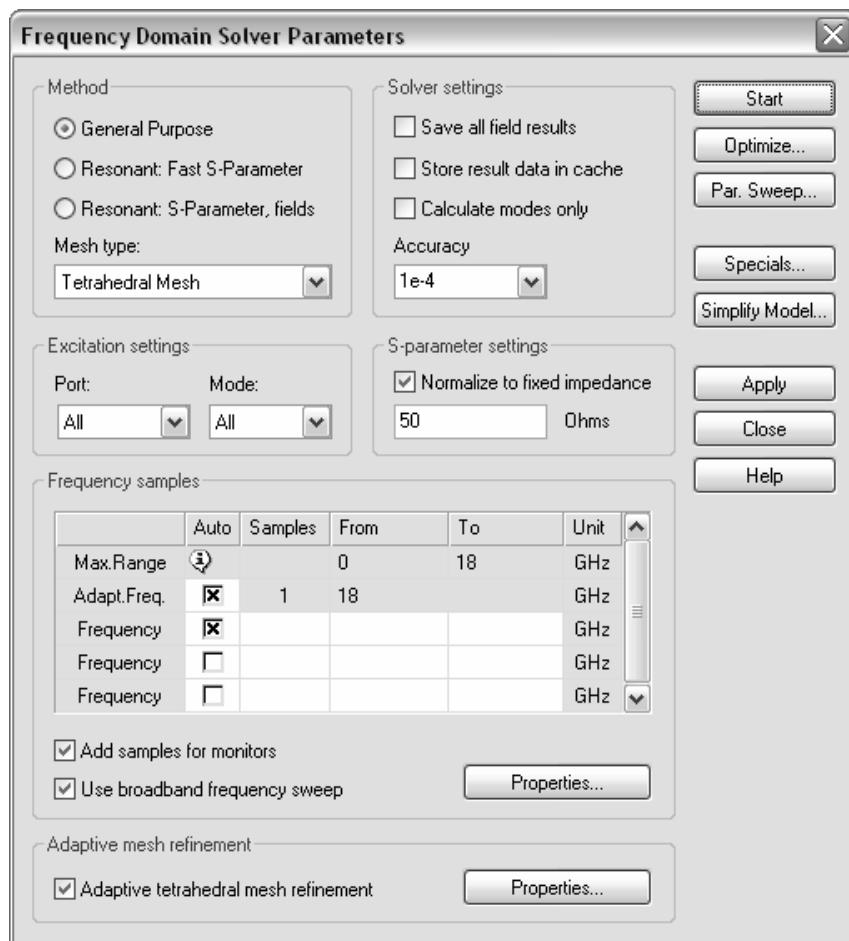
Before you recalculate the S-parameters using the frequency domain solver, you should first copy the results from the transient solver into a new folder for easier comparison afterward.

Select the *1D Results* folder in the navigation tree, and choose *Edit*⇒*Add New Tree Folder* from the main menu. You can then assign a name (e.g. “Comparison”) to the newly created navigation tree item. After creating the new folder, you can select the *1D Results*⇒*|S| dB* folder and choose *Edit*⇒*Copy Result Curve(s)*. Finally, select the newly created *1D Results*⇒*Comparison* folder and choose *Edit*⇒*Paste Result Curve(s)*. Note that the copied result curves are fixed and will be neither deleted nor changed when parameters are changed or S-parameters are recalculated. For organizational purposes,

you should now click at each of the new curve entries in the *1D Results*⇒*Comparison* folder, choose *Edit*⇒*Rename* (or just press the *F2* key) and add an appendix TD to the curve name in order to indicate that this is a result from the transient solver. The navigation tree should finally look as follows:

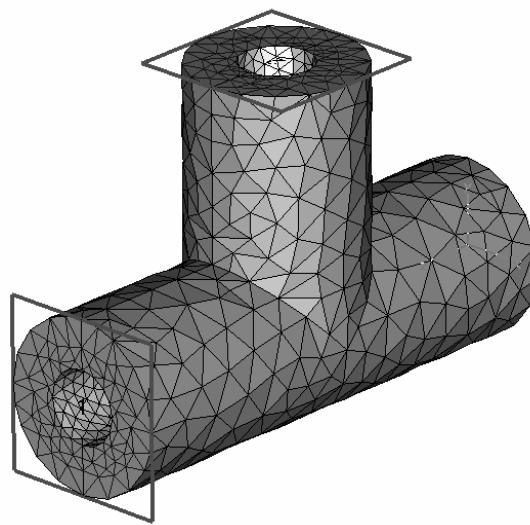


Once you save the transient solver results for later comparison, you can simply open the frequency domain solver dialog box by choosing *Solve*⇒*Frequency Domain Solver* which will open the following dialog box:



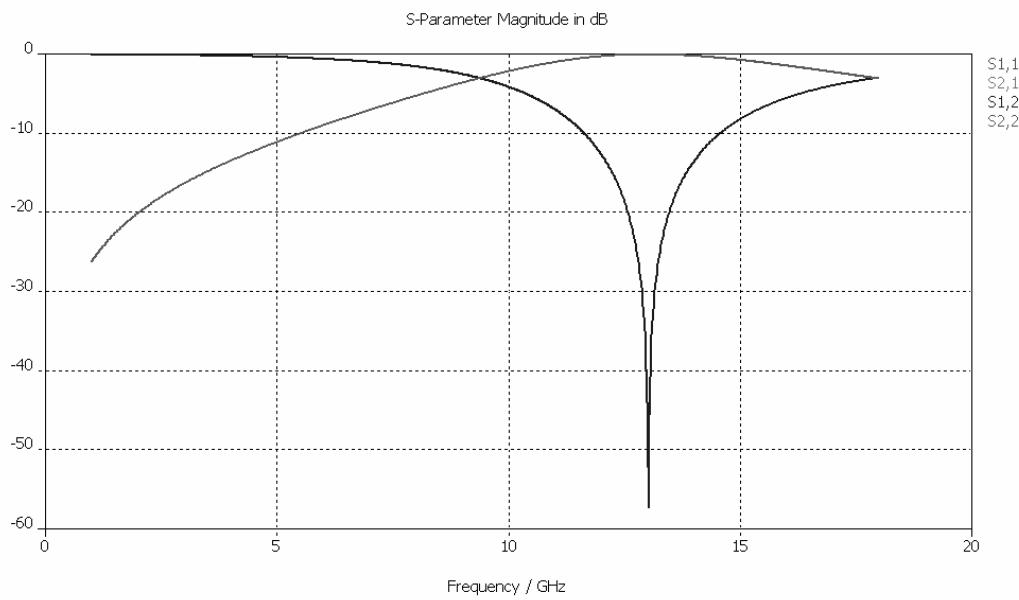
The default settings of the frequency domain solver use a tetrahedral mesh, automatic mesh adaptation and full S-parameter matrix calculation so that you do not need to change anything here. The S-parameter normalization to 50 ohms also activates automatically since you have already specified this for the transient simulation before. You can now perform the frequency domain simulation by clicking the *Start* button and confirming the deletion of the non-frequency domain solver results.

In order to see the tetrahedral mesh used for this simulation while the solver is running, activate the mesh mode (*Mesh*⇒*Mesh View*):

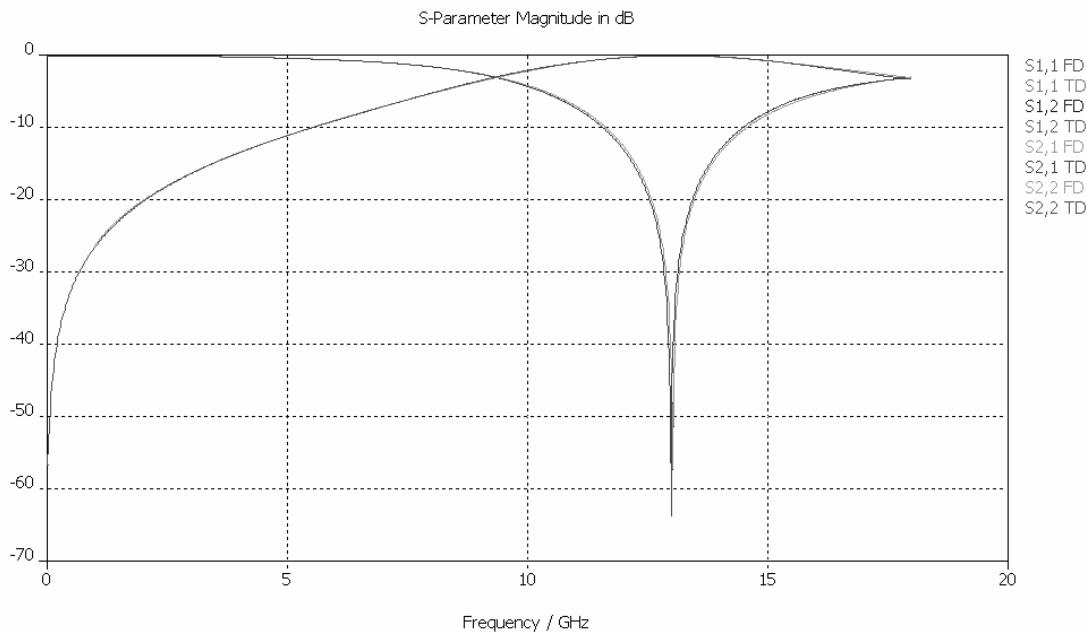


The solver first performs a mesh adaptation at the ports first before the mesh inside the structure is adapted at the highest frequency of interest in the second step. Once the mesh adaptation has converged, the solver calculates the S-parameters as a function of frequency by using its fast sweep capability.

When the solver has finished its work, you can again view the results by selecting *1D Results*⇒*|S| dB* from the navigation tree:



As you can see, the results are quite similar to the results previously obtained from the transient solver. A better comparison can now be achieved by copying and pasting the frequency domain solver results to the *1D Results*⇒*Comparison* folder again. You should again add an appendix “FD” to the new frequency domain solver results:



As you can see, the results from the transient solver using hexahedral grids and the frequency domain solver using tetrahedral grids are in excellent agreement here.

Summary

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

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 using structure parameters
9. Use the parameter sweep tool and visualize parametric results
10. Use result templates for customized post processing
11. Perform automatic optimizations
12. Compare the results from the transient solver and the frequency domain solver

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 look at the contents page of the online help manual which can be opened with *Help* \Rightarrow *Contents*. If you have any further questions or remarks, 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. Ask your support center for details.

Which Solver to Use

Since we have mainly focused on the transient solver and somewhat on the general purpose frequency domain solver in the previous example, it's time to clarify which solver fits which application best. The transient solver is 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 table here lists a few typical applications together with the solvers that 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. Furthermore, depending on the particular structure, it may be that other solvers are more efficient for a particular application than those shown in the table. Therefore use this table as a guideline rather than a rule for which solver to use.

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

Application Name	Solver Type(s)
Optical diplexers and filters	Transient, General Purpose Frequency Domain, Resonant Fast S-parameter Frequency Domain
Filters and diplexers	Transient, General Purpose Frequency Domain, Resonant Fast S-parameter Frequency Domain
Cavities, resonator design	Eigenmode
Traveling wave structures	Eigenmode
Periodic problems (frequency selective surfaces, periodic band gap structures)	General Purpose Frequency Domain, Eigenmode, Transient
Periodic problem with nonzero phase shift	General Purpose Frequency Domain, Eigenmode

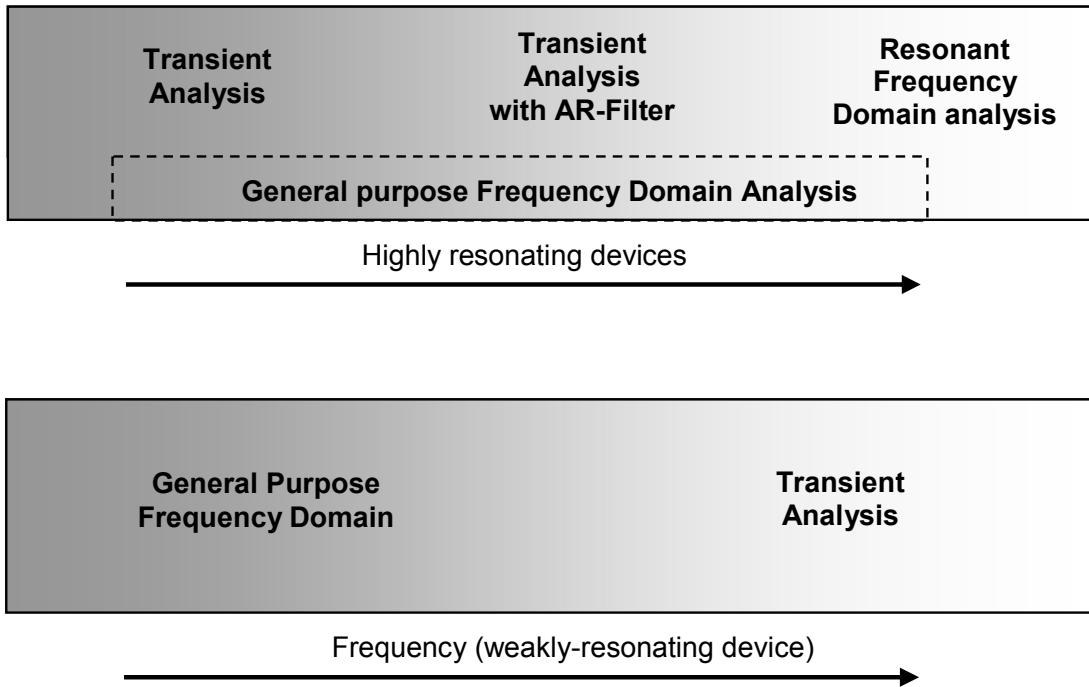
Please note that the application range of the transient analysis can be extended quite a way toward more resonant devices by applying some advanced digital signal processing techniques rather than simply using a Discrete Fourier Transform. CST MICROWAVE STUDIO® features an Auto Regressive (AR) Filter 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 general purpose frequency domain solver may be faster, especially since in most cases a few frequency samples are sufficient to characterize the structure's behavior by using the fast broadband frequency sweep tool. 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.

Besides these general considerations, there are also some applications that require the selection of a particular solver since the corresponding electromagnetic problem can be solved only by using the corresponding method:

- Structures containing nonlinear diodes:** The frequency domain solver can not handle nonlinearities. Therefore the time domain solver must be used for these applications.
- Very large structures / high frequencies:** The frequency domain solver requires the solution of a matrix equation. This becomes very slow and memory intensive when the number of mesh cells is large. Whenever the number of cells is in the order of several million, the time domain solver should be used.
- Periodic structures with nonzero phase shift:** The transient solver can handle only periodic structures with zero phase shifts, so the frequency domain solver must be used instead. The phase shift between adjacent boundary planes or the geometrical angle of incidence has to be specified in the boundary condition dialog box. Note that both the electrical phase angle between the boundary planes and the geometrical angle of incidence are not identical. The frequency domain solver in combination with tetrahedral grids also offers a special Unit Cell feature allowing the simulation of periodic structures with a non-rectangular lattice.
- Calculation of S-parameters for structures with large losses at the waveguide ports:** The frequency domain solver can handle lossy ports and can calculate waveguide port modes for lossy wave guides.

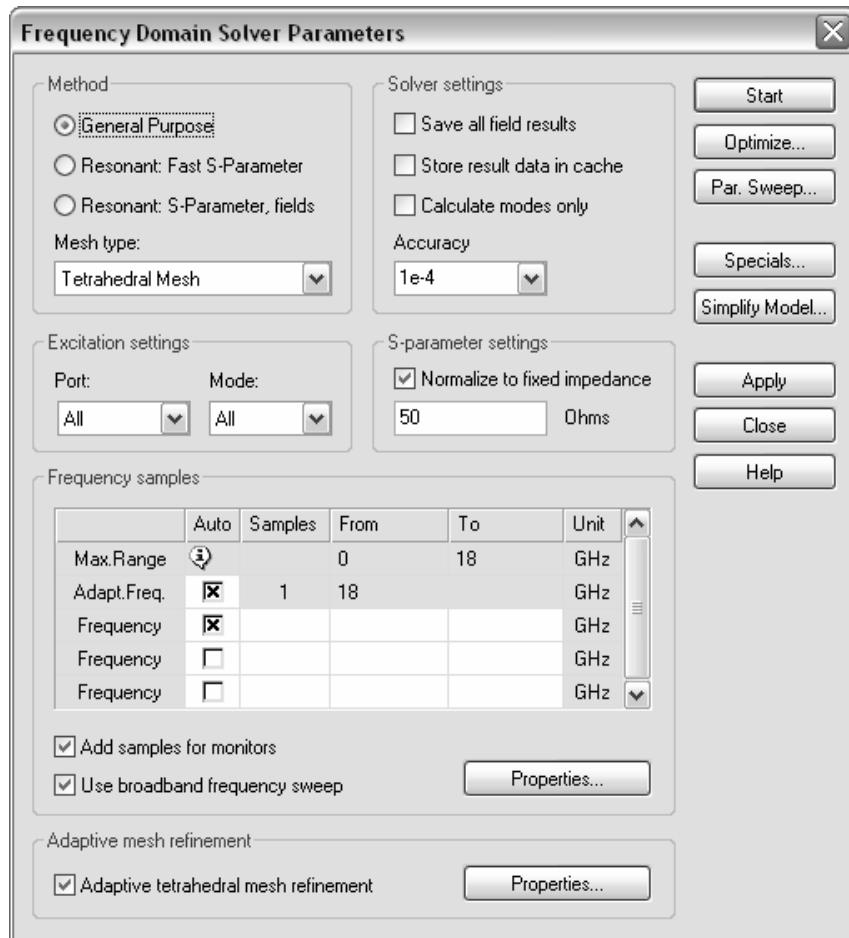
Summarizing these statements, the following diagrams show a rough guideline for the application ranges of the methods:



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.

General Purpose Frequency Domain Computations

The basic procedure of running the frequency domain solver is demonstrated in the example above. The following explanations provide some more information about the settings in the frequency domain solver dialog box which you can open by choosing *Solve*⇒*Frequency Domain Solver*:



The *Method* field allows for the selection of the principal solution method. Refer to the section *Which Solver to Use* earlier in this chapter for more information concerning the difference of these three techniques.

The *General Purpose* solver can be seen as the counterpart of the transient solver. The other two solvers have some very special applications and are described in the next section. The following explanations focus on the *General Purpose* solver.

A special feature of the *General Purpose* solver is the support of both hexahedral and tetrahedral meshes. In most cases, you will compare the results from the tetrahedral frequency domain solver and the hexahedral transient solver. This selection gives you the ability to compare the results from two completely independent simulation techniques.

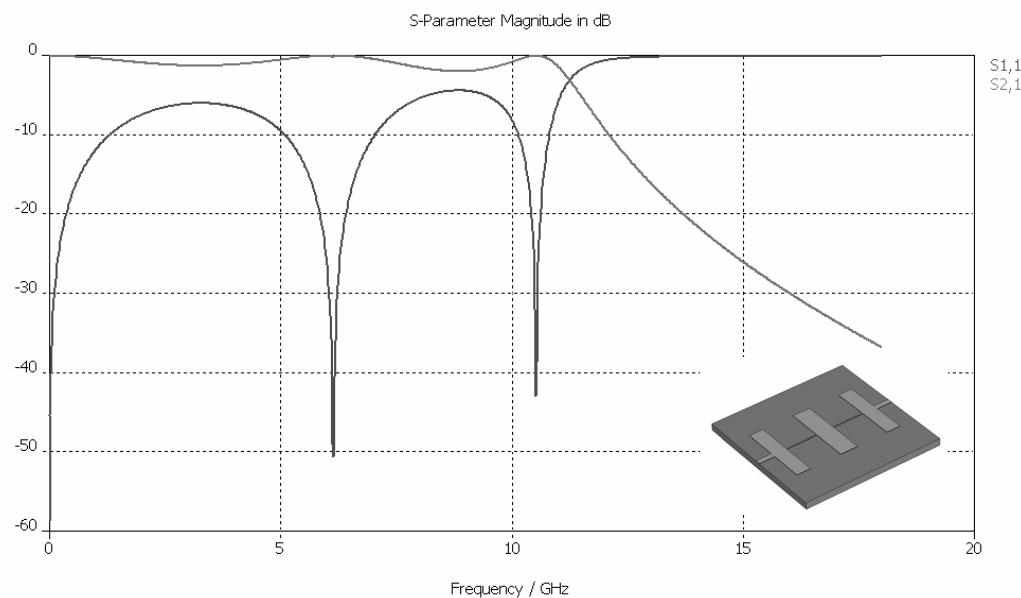
The tetrahedral mesh generation normally generates relatively coarse initial meshes. Therefore we strongly recommend using the *Adaptive tetrahedral mesh refinement* option in order to ensure accurate results.

The mesh adaptation strategies of the transient solver and the frequency domain solver are fundamentally different. The transient solver runs the entire broadband simulation for every mesh adaptation pass and evaluates the worst-case deviation of two subsequent S-parameter results (broadband). The mesh refinement is then done based on information from the broadband result data. In contrast, the frequency domain solver runs the mesh adaptation at a single frequency point only. Once the adaptation is complete, the broadband results are computed by keeping the adapted mesh fixed. Even though this procedure is a little less safe in general, it normally provides quite good results.

Since the frequency domain solver mesh adaptation runs only at a single frequency point, the location of this point within the frequency spectrum is very important.

For weakly resonating devices, it is usually a good choice to select the highest frequency of interest for the mesh adaptation (which is the default setting). This will ensure that fields with the shortest wavelength in the frequency sweep are still sampled properly.

The situation is different for strongly resonating devices as shown in the following picture (e.g. *Low Pass Filter (3 Stubs)* example):



This low pass type of structure has very low transmission at the highest frequency of interest. Running the mesh adaptation at this frequency will not get sufficient information about the actual filter characteristics. Therefore the adaptation will keep refining the mesh around the input port since all the energy is stored there and too little information is available about the behavior of the fields inside the structure.

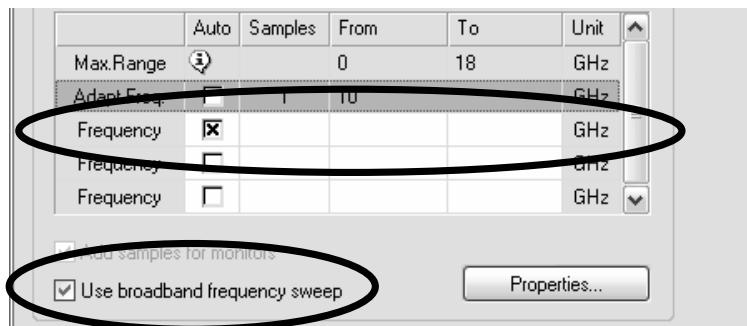
In cases like the one shown above (typically filters), it is very important to specify the adaptation frequency such that it is located in the pass band of the filter. This can be done by un-checking the *Auto* button in the *Adapt.Freq.* line in the frequency list. Afterwards you can specify the adaptation frequency in the *From* column of the list.

	Auto	Samples	From	To	Unit
Max.Range	<input type="button" value="..."/>	0	10	GHz	
Adapt.Freq.	<input type="checkbox"/>	1	10	GHz	
Frequency	<input checked="" type="checkbox"/>			GHz	
Frequency	<input type="checkbox"/>			GHz	
Frequency	<input type="checkbox"/>			GHz	

Another important difference between the transient solver and the frequency domain solver is the number of frequency samples to be calculated. Whereas in the time domain the number of frequency samples has almost no influence on the solver time, a classical frequency domain calculation has to carry out the simulation frequency point by frequency point. Every frequency point requires a complete solver run.

The frequency domain solver used by CST MICROWAVE STUDIO®, however, uses a special broadband frequency sweep technique in order to derive the full broadband spectrum from a relatively small number of frequency samples.

In order to make use of this technique, you should allow an automatic sampling of frequency points by checking the *Auto* button in a *Frequency* line and then activating the *Use broadband frequency sweep* option. The solver will then automatically adapt the selection of frequency points so that the broadband curve can be obtained by calculating a minimum number of samples.



To store the fields at particular frequencies, monitors need to be defined in advance as described previously for the transient solver. These monitor frequencies are then added to the list of calculated frequency points if the *Add samples for monitors* option is activated.

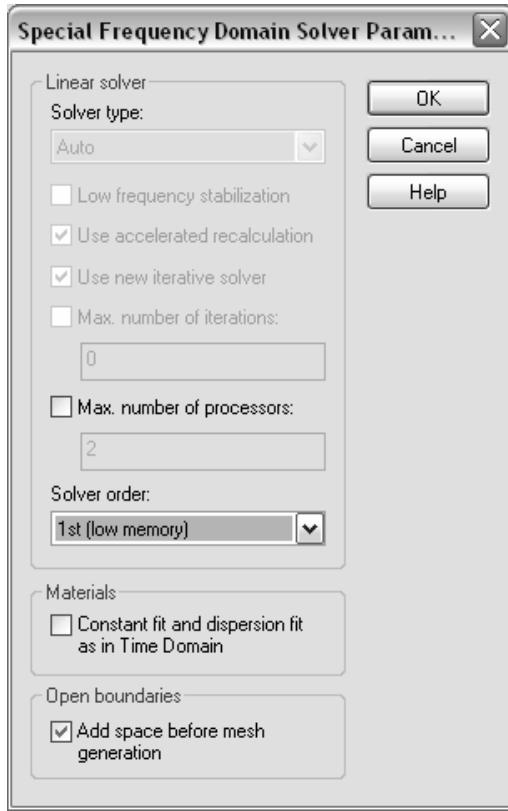
The frequency domain solver cannot calculate the fields at a frequency of zero. Therefore a zero frequency will automatically be shifted to a reasonably small value.

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

By default, the tetrahedral frequency domain solver uses second order elements to get an excellent sampling of the fields at high frequencies. This also allows the use of relatively few elements per wavelength as compared with the first order elements used by the solvers based on hexahedral grids.

However, some structures require a relatively fine mesh due to their geometry which is much finer than required to properly sample the wave phenomena. Typical application examples for this are often PCB types of structures or IC packages. In such cases, using first order elements rather than the standard second order elements can significantly reduce simulation time and memory requirement significantly.

To use first order elements, select *1st (low memory)* in the *Solver order* field in the *Specials* dialog box:



Another important difference between the frequency domain solver and the transient solver is the way both simulators handle dispersive materials.

For a given list of material parameters at various frequencies, the transient solver always needs to fit a certain dispersion model to the data (first or second order). During the simulation, the broadband material behavior will then be taken from the model rather than using the originally specified data.

However, since the frequency domain solver computes the broadband sweep by a sequence of individual frequency point calculations, the solver can simply interpolate the given list of frequency points directly. As a result, the frequency domain solver can use user-specified material property tables more directly than the transient solver can.

For the sake of comparing these two solvers results, it may be advantageous to advise the frequency domain solver to use the same material model fitted data as the transient solver does by checking the *Constant fit and dispersion fit as in Time Domain* box in the solver *Specials* dialog box.

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

1. Select an appropriate project template (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define the structure.
5. Set the frequency range.
6. Set the boundary conditions (optional).
7. Define the excitation ports.
8. Set the monitors (optional).
9. Start the frequency domain solver.
10. Analyze the results (S-parameters, field patterns, result templates, etc.).

Resonant Frequency Domain Computations

Besides the general purpose solver mentioned above, the frequency domain solver also features two additional solver methods specialized for the simulation of strongly resonant structures such as filters:

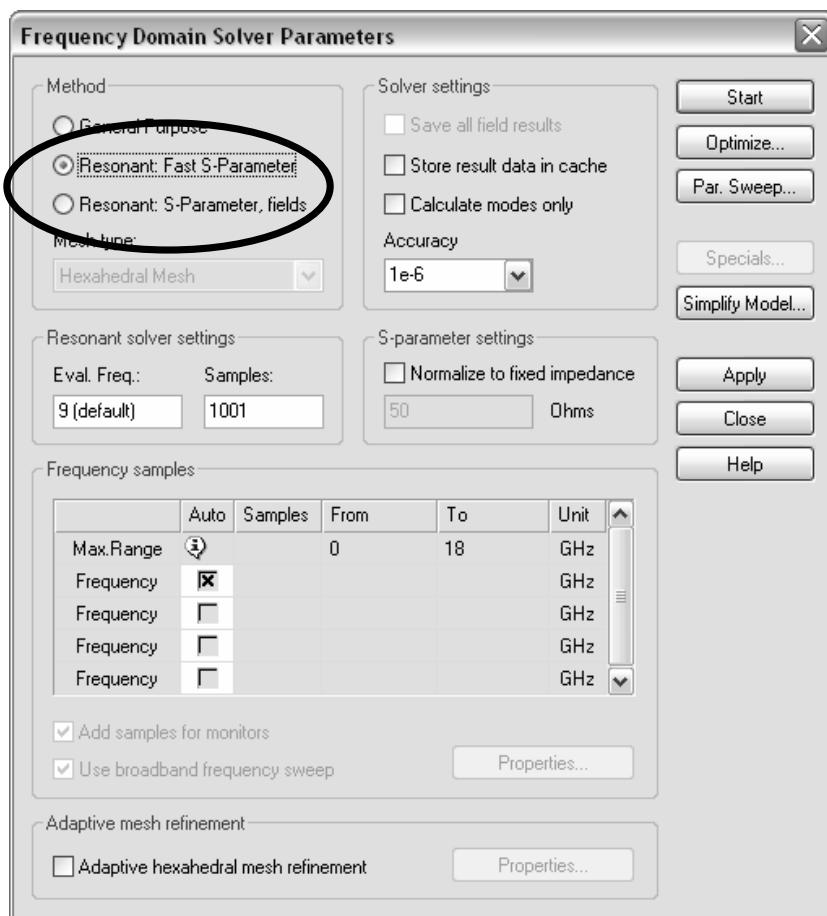
1. Resonant: Fast S-Parameter
2. Resonant: S-Parameter, fields

Please note that both solvers are currently based on hexahedral grids only. For more information, refer to the **Filter** tutorial.

Resonant: Fast S-Parameter

The *Resonant: Fast S-Parameter* method calculates only the S-parameters but does not produce any field results. The solver is able to handle weak losses such as lossy dielectrics, etc. The strong advantage of this method is its performance which can, in some cases, be significantly better than any other simulation method. Despite its limitations, the performance advantage makes this solver the ideal choice for a certain type of applications.

Using this solver is relatively straightforward. The first step is to select the corresponding method in the solver control dialog box (*Solve* \Rightarrow *Frequency Domain Solver*):



The *Accuracy* setting is not critical so that it can normally be kept at the default of 1e-6. The number of uniformly spaced frequency samples can be specified in the *Samples* field in basically the same way as it is done for the transient solver. The number of frequency samples only slightly influences the performance of the simulation, but excessive values should be avoided since this may require a lot of disk space and may slow down further post processing operations.

The only setting which requires a bit more explanation here is the *Eval.Freq.* field. The default setting equals the center frequency of the simulation frequency range. Material properties (including losses) will be calculated at this frequency and remain fixed over the entire frequency band. The same happens for waveguide ports for which the mode patterns are also calculated at the center frequency. While the port mode pattern does not change as a function of frequency for hollow waveguides or coaxial lines, it does change for micro strips and coplanar lines. Therefore, using a fixed mode pattern may cause some mismatch at the ports and result in slightly inaccurate S-parameters for frequencies farther away from the *Eval.Freq.* However, the resulting error normally is quite small. Especially for narrowband structures (which are the primary application area for this solver), the frequency bands of interest are relatively small, and the variation of the material properties or port modes within these frequency bands is negligible.

After running the solver by clicking the *Start* button, you can access its S-parameter results as usual from within the navigation tree.

The following summarizes the simulation flow for this solver module:

1. Select an appropriate project template for filter structures (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define structure.
5. Set frequency range.
6. Set closed boundary conditions (optional).
7. Define excitation ports.
8. Start *Resonant: Fast S-parameter* frequency domain solver.
9. Analyze results (S-parameters, result templates).

Resonant: S-Parameter, fields

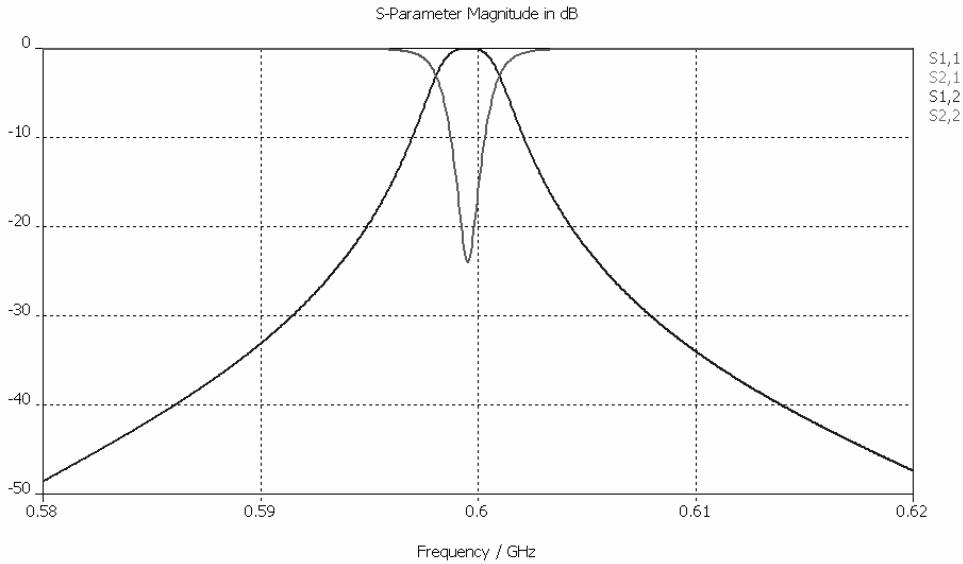
The main difference between the *Resonant: S-Parameter, fields* solver and the *Resonant: Fast S-Parameter* solver is the calculation of fields in addition to the S-parameters.

However, performance is significantly affected by the field calculation such that this solver should only be used when the calculation of fields is essential. Note that the *Resonant: S-Parameter, fields* solver is limited to handle loss free structures only.

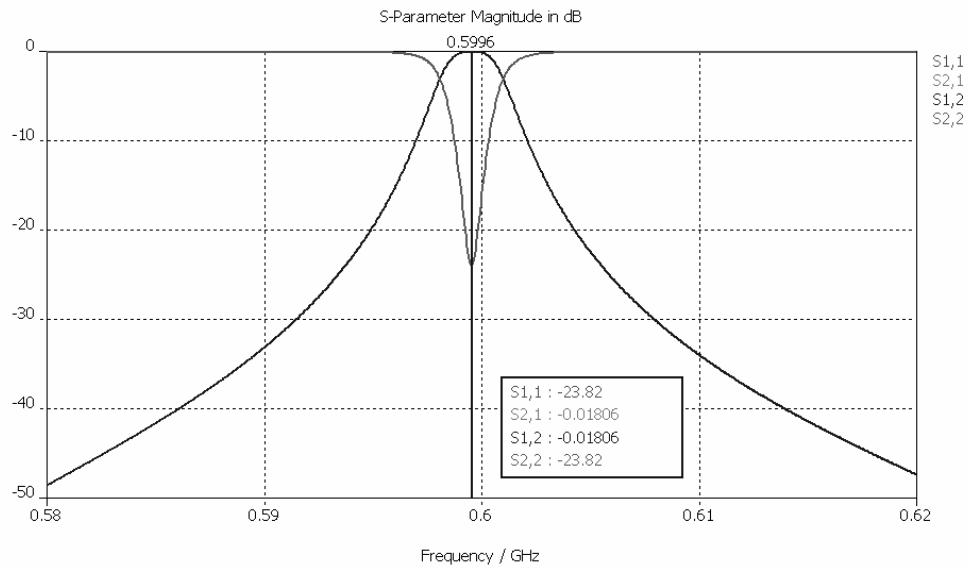
The specification of which fields to calculate is done by the standard monitor concept as explained earlier in this manual.

A very interesting feature of this solver is that some intermediate information concerning the fields is stored even if no field monitors are specified. Once a simulation is finished and the S-parameters are visualized, it is relatively fast and straightforward to obtain the fields at certain frequencies.

To demonstrate this feature, let us assume that you ran a simulation for a filter structure using the *Resonant: S-Parameter, fields* solver and are now inspecting the S-parameters:



You may now be particularly interested in the fields at the resonance peak. The easiest way to obtain this information is to place the axis marker at the location of the resonance (*Results*⇒*1D Plot Options*⇒*Show Axis Marker*.):



Finally choose *Results*⇒*Calculate fields at axis marker* to obtain the fields at this particular frequency. The field computation itself will be relatively quick since a lot of intermediate data have already been stored during the initial S-parameter calculation.

All other settings are identical for the *Resonant: Fast S-Parameter* described above. Therefore the typical simulation flow is as follows:

1. Select an appropriate project template for filter structures (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define structure.
5. Set frequency range.
6. Set closed boundary conditions.
7. Define excitation ports.
8. Define monitors (optional).
9. Start *Resonant: S-parameter, fields* frequency domain solver.
10. Analyze results (S-parameters, result templates, fields, etc.).

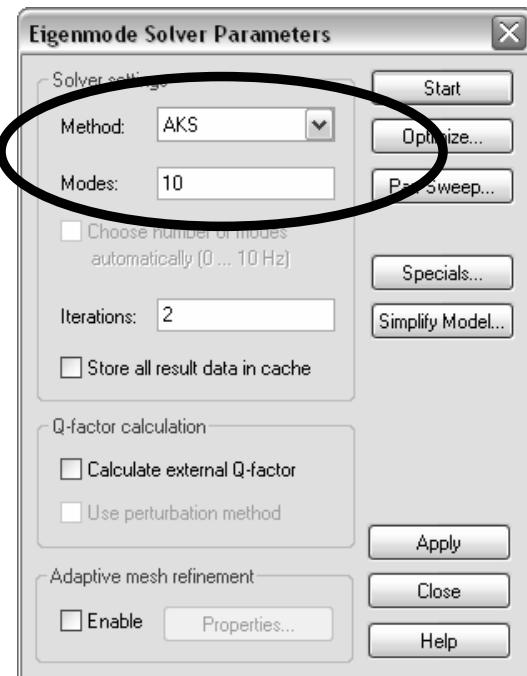
Eigenmode (Resonator) Computations

The eigenmode solver calculates a finite number of modal field distributions in a closed device. CST MICROWAVE STUDIO® provides two different eigenmode solvers: AKS (*Advanced Krylow Subspace*) and JDM (*Jacoby Davidson Method*).

These methods work on a completely different mathematical foundation. The *JDM* solver can be considered as a more robust solver technology, but the *AKS* solver may be faster if many modes are to be calculated. Therefore we recommend the *JDM* solver only if a small number of modes (e.g. 1-5) has to be calculated. Otherwise the *AKS* solver should be used.

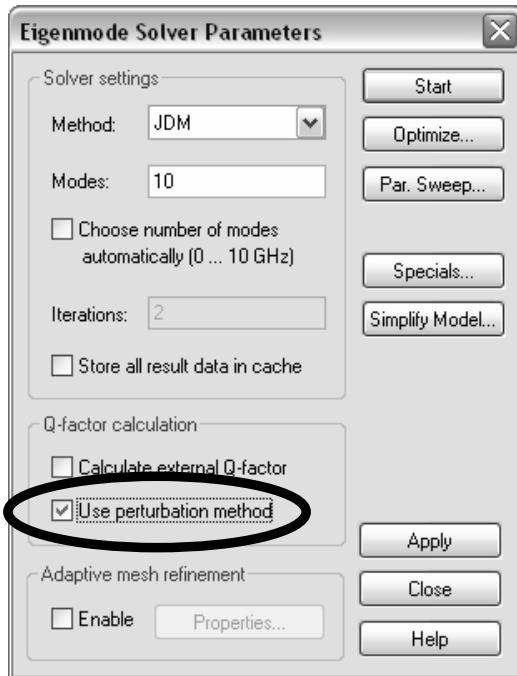
Since the eigenmode analysis does not require the definition of excitation ports, this step can be omitted. The definition of monitors is also not necessary because the modes themselves contain all available information about the device.

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



The solution of lossy eigenmode problems is a challenging task which implies that the proper consideration of losses will significantly slow down the simulation. Even if the *JDM* solver is able to directly solve the lossy eigenmode problem, it may sometimes be advisable (especially for very small losses) to first calculate the loss free eigenmode problem and then obtain losses and Q-factors of the device using a perturbation method in the post processing.

The perturbation method requires material losses to be defined before the eigenmode simulation is started. Running the *AKS* solver will always calculate the loss free problem by simply ignoring the loss definition. The *JDM* solver can be advised to do so by activating the *Use perturbation method* box as shown below:



In the eigenmode solver control dialog box, the most important controls are the *Method* (as discussed above) and the number of *Modes*.

The typical simulation procedure is as follows:

1. Depending on the number of modes, choose the proper solver method:
 - For the direct solution of lossy problems choose *JDM*.
 - For loss free problems with a small number of modes (e.g. 1-5) choose *JDM*.
 - For loss free problems with many modes (e.g. >5) choose *AKS*.
2. Enter the desired number of *Modes* (N). The solver will then compute the first N modes of the device. For the *AKS* solver, it is often advantageous to specify more modes to be calculated than you actually need, e.g. enter 20 modes to be calculated if you actually need 15. In most cases it is a good choice to calculate at least the first 10 modes of the device.
3. Click the *Start* button.

After the solver has finished, a summary of the calculated modes will appear in the message window:

Final Results:		
Mode	Frequency	Accuracy
1	32.88 GHz	1.029e-011
2	34.56 GHz	1.231e-011
3	39.23 GHz	1.156e-011
4	43.68 GHz	4.529e-010
5	43.68 GHz	9.247e-011
6	46.64 GHz	1.062e-012
7	47.84 GHz	9.006e-013
8	54.79 GHz	1.885e-006
9*	54.79 GHz	0.003998
10	56.38 GHz	3.369e-007

* = This mode has a bad accuracy.

⚠ The lowest Eigenmode frequency is greater than the upper frequency range limit. Please check the frequency range.

Optimum guess for the highest eigenfrequency would be: 56.3821.

⚠ 1 warning occurred.

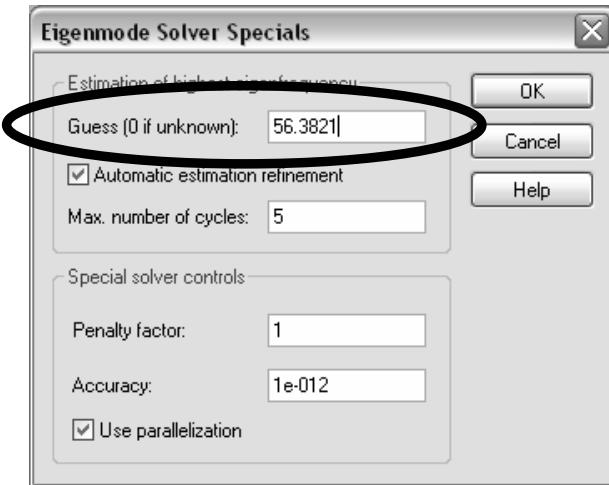
When using the AKS solver, sometimes a few of the higher modes will not be calculated with sufficient accuracy and thus be marked with “*”. However, this does not affect the accuracy of the lower modes and is the reason you should specify more modes to calculate than you actually need.

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

Performing estimation refinement passes reduces the performance of the AKS eigenmode calculation. To speed up the AKS eigenmode calculation in these cases, you can manually enter a guess for the frequency of the highest mode you are looking for. The AKS eigenmode solver automatically derives such a guess from previously calculated results and prints this value in the message window.

⚠ The lowest Eigenmode frequency is greater than the upper frequency range limit. Please check the frequency range.
Optimum guess for the highest eigenfrequency would be: 56.3821.
⚠ 1 warning occurred.

You can set this guess by opening the special settings dialog box by clicking the **Specials** button in the solver control dialog box. In the **Guess** field you should enter the proposed guess as 56.3821 (GHz) in this example. If you are unsure about this setting you should specify zero for automatic estimation. Note that this setting is used only by the AKS method.



This guess will now affect all subsequent calculations and should speed up the AKS solver significantly.

You can access the eigenmode solver results for the N-th mode from the navigation tree:

Navigation tree	Type of result
2D/3D Results \Rightarrow Modes \Rightarrow Mode N \Rightarrow e	Electric field
2D/3D Results \Rightarrow Modes \Rightarrow Mode N \Rightarrow h	Magnetic field
2D/3D Results \Rightarrow Modes \Rightarrow Mode N \Rightarrow Surface Current	Surface current field
2D/3D Results \Rightarrow Modes \Rightarrow Mode N \Rightarrow Energy Density	Energy density

Refer to the **Resonator Tutorial** for more information on post-processing the results.

The following summarizes the input necessary for eigenmode calculations:

1. Select an appropriate project template (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define structure.
5. Set frequency range.
6. Set closed boundary conditions (optional).
7. Start eigenmode solver.
8. Analyze results (field patterns, frequencies, losses/Q-factors, result templates, etc.).

Choose the Right Port

The proper definition of ports is essential for accurate S-parameter computations. In measurement setups, the device under test needs to be connected to the network analyzer by using low reflection probes or applying proper de-embedding techniques. Care must be taken with the probe connection because the measured S-parameters will otherwise become inaccurate.

For EM field simulations, generally the same problems exist. The port connection needs to be loss free and have very low levels of reflection. The basic problem here is to launch and extract the fields as seamlessly as possible at the ports. Fringing effects should be kept to a minimum.

In general, two types of ports need to be distinguished:

1. Discrete ports
2. Waveguide ports

Discrete ports can be seen as lumped circuit elements with an internal resistor and a current source in parallel. These ports consist of a single lumped element in the middle and two perfectly electric conducting wires connecting the port to the structure. A certain voltage / current relation is then introduced along the lumped element, and the S-parameters are calculated based on the element's currents and voltages.

It is important to note that there may be a fringing effect at the transition of the structure to the discrete port. This will always be the case when the geometry of the structure's transmission lines is different from the geometry of the discrete ports. Unfortunately, this will be the situation in most cases.

On the other hand, discrete ports provide a convenient and flexible way to attach ports to a given structure. The accuracy of the simulation is normally sufficient when the size of the discrete port is a tenth of a wavelength or less.

The most accurate results can be obtained by using **waveguide ports**. These ports normally provide very low levels of reflection and distortion and, therefore, are the best choice whenever very high accuracy is required.

Note that CST DESIGN STUDIO™ is capable of de-embedding the port influence from the S-matrix by removing the effect of the port to structure transmission matrix from each of the ports. Refer to the CST DESIGN STUDIO™ *Getting Started* manual for more information.

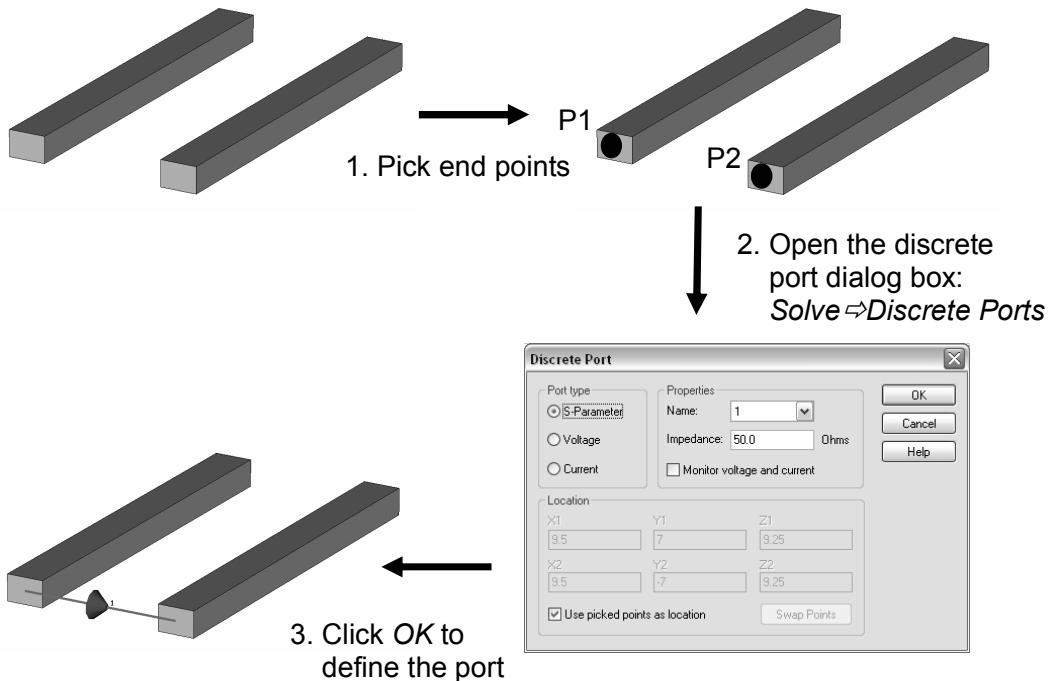
The following sections focus on the definition of discrete ports and waveguide ports. For waveguide ports the most common types of transmission lines are treated separately. The sections are designed so that each is self-contained. Therefore you may use them as a reference once a particular type of port needs to be defined rather than reading through all of them at once.

Discrete Ports

Although the waveguide ports are the most accurate way to terminate a waveguide, discrete ports are sometimes 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 antennae 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 waveguide 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 end points from the structure using the common pick tools and then to enter the discrete port dialog box:



Note: The *S-Parameter* ports are the most frequently used *Port Types* and contain internal resistors for calculating S-Parameters with fixed reference impedances. In contrast, the *Voltage* and *Current* ports are ideal sources. The latter types of ports are normally used for EMC types of application.

Waveguide Ports

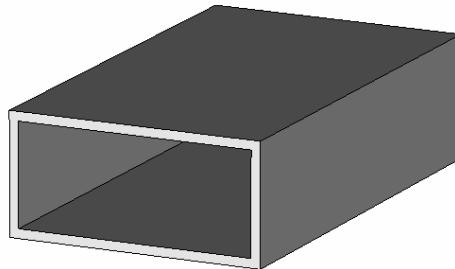
This kind of port simulates an infinitely long waveguide connected to the structure. The waveguide modes travel out of the structure toward the boundary planes thus leaving the computation domain with very low levels of reflections.

Very low reflections can be achieved when the waveguide mode patterns in the port match perfectly with the mode patterns from the waveguides inside the structure. CST MICROWAVE STUDIO® uses a 2D eigenmode solver to calculate the waveguide port modes. This procedure can provide very low levels of reflection below -100dB in some cases.

In general, the definition of a waveguide port requires enclosing the entire field filled domain in the cross section of the transmission line with the port area. The eigenmode solver can then calculate the exact port modes within these boundaries. The strategies to properly define waveguide ports depend a bit on the type of the transmission line. In the following we focus on the most commonly used applications.

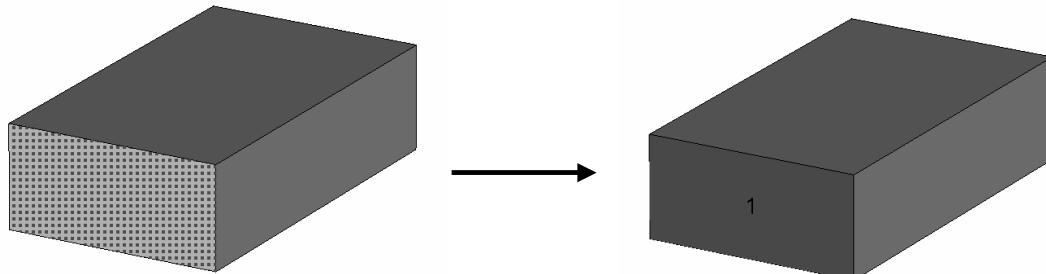
Empty Waveguides

The most frequently used waveguide type of this category is the rectangular waveguide shown below:



The port assignment for this type of waveguide is quite simple. However, it is important to make sure that the port covers the entire waveguide cross-section. The easiest way to define a port's face is to pick structure elements such as points / edges / faces before opening the port definition dialog box. The port dimensions will then automatically adjust to the bounding box of the picked elements.

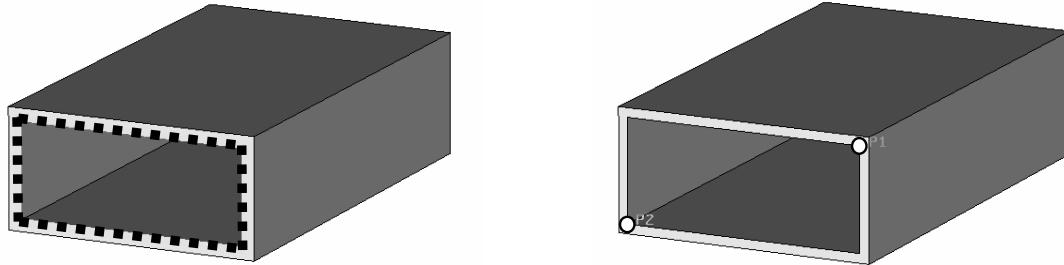
If the interior of the waveguide is modeled by a dielectric material block, you can simply pick the end face of the waveguide as shown in the picture below:



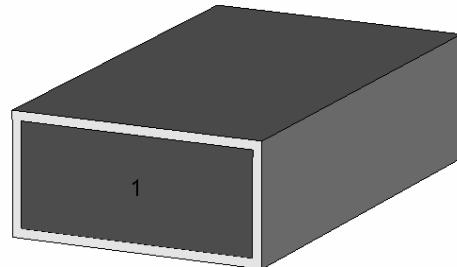
The bounding box of the picked end face will then exactly define the dimension of the port.

When the background material is not made of PEC material, it is often required to model the wall of the waveguide, too. For a construction like the one shown in the first picture above, the end face of the rectangular waveguide has not been modeled directly.

We therefore recommend to pick the edges on the circumference of the waveguide or to pick two opposite points on the corners:

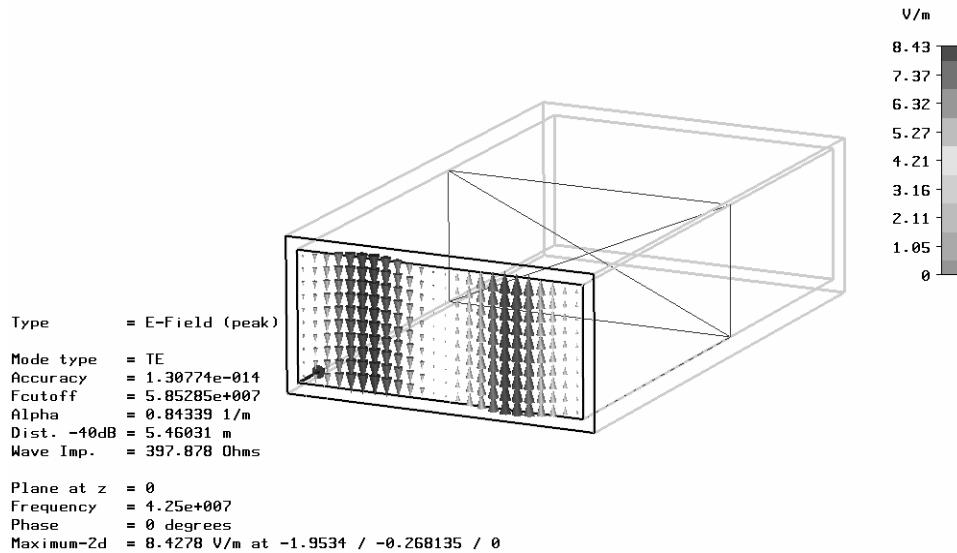


In both cases, the bounding box of the picked elements will be large enough to cover the entire field filled cross-section of the waveguide. The resulting port definition will then look as follows:



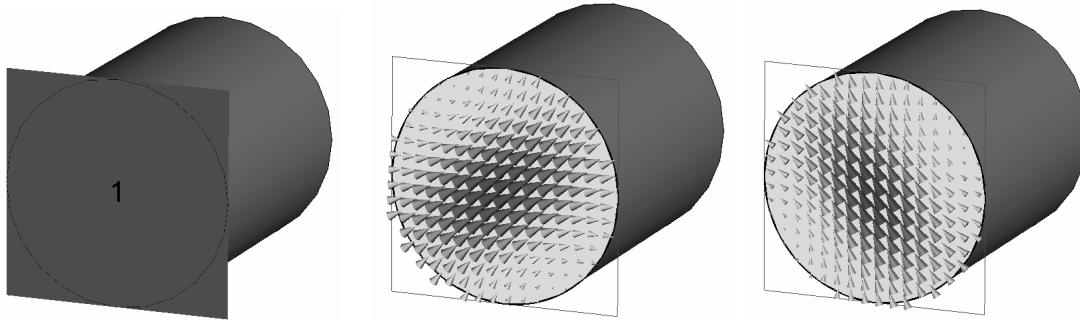
CST MICROWAVE STUDIO® solvers generally allow using more than just the fundamental mode in the waveguide port. This becomes especially important when higher order modes need to be taken into account. As a rule, the number of modes to be considered at a waveguide port should at least be the number of propagating modes.

If an evanescent port mode result is selected, a box will appear visualizing the distance from the port where the port mode has decayed by -40dB:



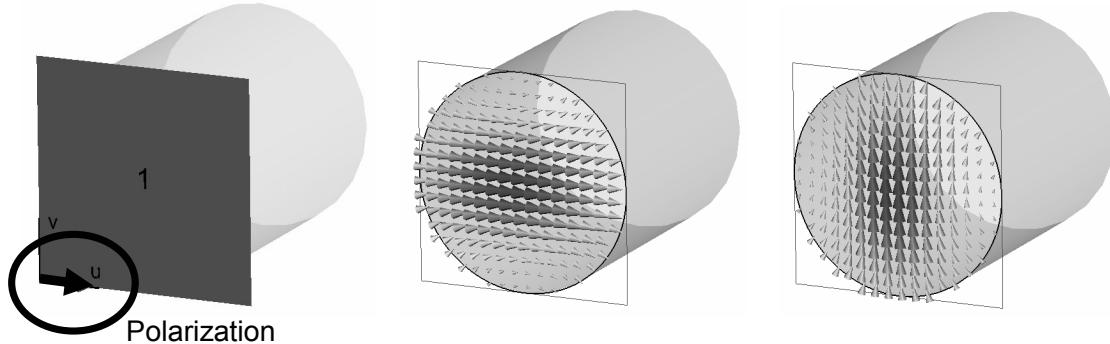
Multiple waveguide modes can convert energy into each other at the structure's discontinuities. Due to these phenomena, the S-parameters of the propagating modes can also be affected by the evanescent modes. Therefore a certain number of evanescent modes needs to be taken into account. We recommend that the number of modes used for the simulation be chosen such that the -40dB distance of the last considered mode is shorter than the distance to the next discontinuity. This setting will ensure that all modes not considered for the simulation will make only very small contributions to the mode conversion.

In the case of quadratic or circular waveguides, the mode polarization is an issue. For the example below, two propagating modes are calculated. By default, the orientation of these modes is arbitrary:



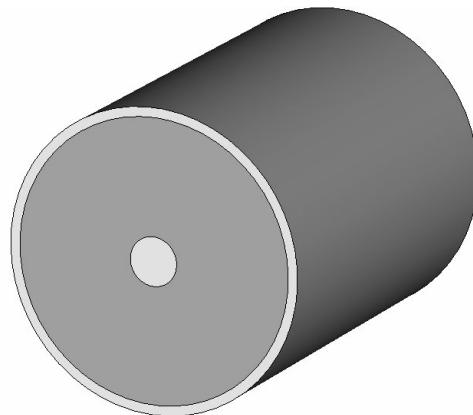
Note that the waveguide port is still shown as a rectangular face. However, the computation of the waveguide mode is automatically restricted to the interior of the waveguide. The most important point here again is that the port's face covers the entire cross-section of the waveguide.

You can specify the polarization of the modes by setting the *Polarization angle* in the waveguide port dialog box. The orientation of the fundamental mode's electric field is then visualized by an arrow in the port face which is shown when the corresponding port is selected. The second mode will then be calculated perpendicularly to the first one:



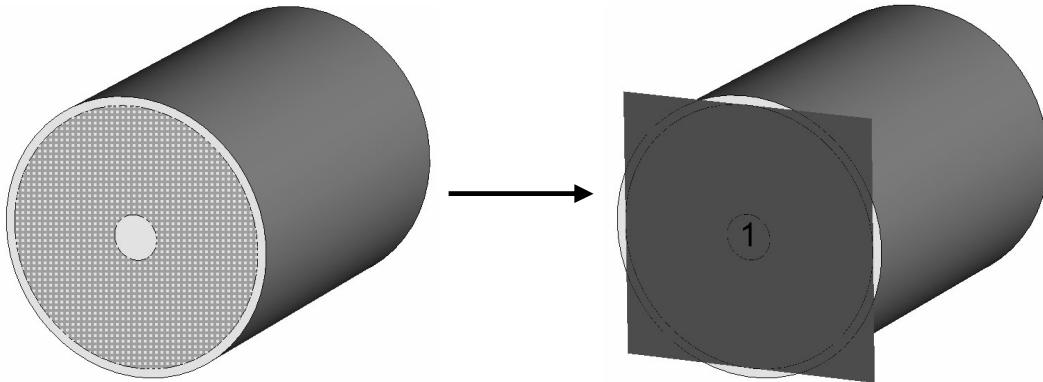
Coaxial Waveguides

The most frequently used waveguide type of this category is the coaxial cable shown in the picture below:



The port assignment for this type of waveguide is very simple. However, it is important to make sure that the port covers the entire cross-section of the coaxial cable. The easiest way to define the port face is to pick structure elements such as points / edges / faces before opening the port definition dialog box. The port dimensions will then automatically adjust to the bounding box of the picked elements.

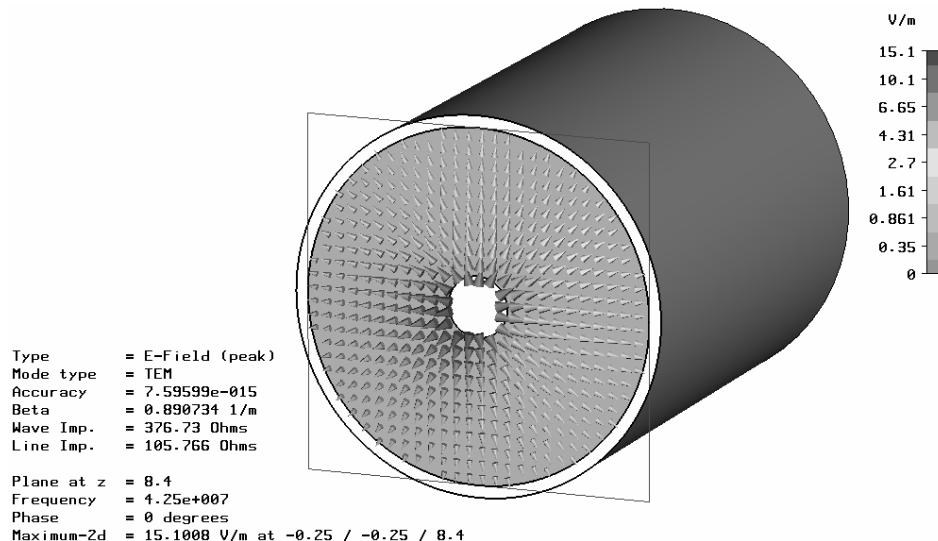
In most cases, the dielectric part of the coaxial cable will be modeled with a solid cylinder. If this is the case, simply pick the end face of the dielectric as shown below:



The bounding box of the picked end face will then define the dimension of the port. Note that the waveguide port is still shown as a rectangular face. However, the computation of the waveguide mode will be automatically restricted to the interior of the coaxial cable.

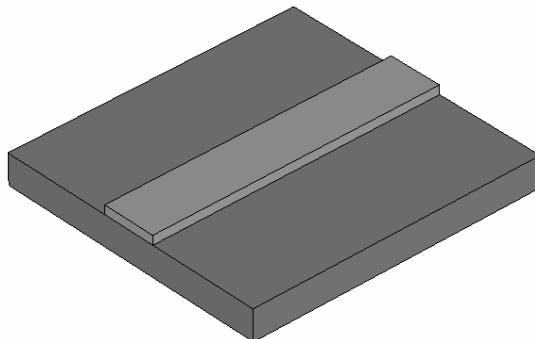
If the end face of the dielectric can not be picked for some reason, you can also pick edges or points on the circumference of the coaxial cable as described for the empty waveguides.

In most cases, you will need to consider the fundamental mode of the coaxial cable so that you don't need to worry about the number of modes for the port. The mode is automatically polarized so that the electric fields point from the inner conductor to the outer conductor as shown in the following picture:



Microstrip Lines

The microstrip line is one of the most commonly used transmission lines for high frequency devices.



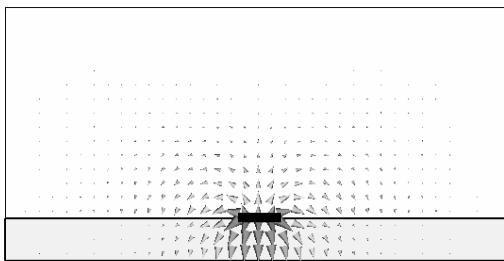
Unfortunately, this line type is relatively complex from an electromagnetic point of view. Therefore a few things need to be considered when defining ports for this type of structure.

This type of structure requires modeling the air above the microstrip line, too. The easiest way to achieve this is to specify an extension in the *Background Material* dialog box as shown in the *Planar Structure* tutorial.

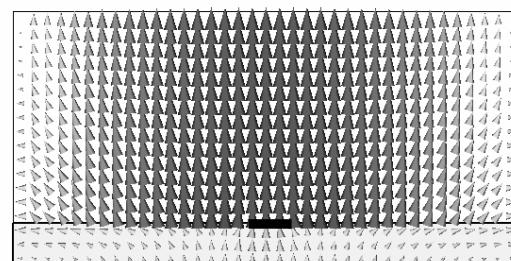
In general, the size of the port is a very important consideration. On one hand, the port needs to be large enough to enclose the significant part of the microstrip line's fundamental quasi-TEM mode. On the other hand, the port size should not be chosen unnecessarily large because this may cause higher order waveguide modes to propagate in the port.

The following pictures show the fundamental microstrip line mode and higher order mode.

Fundamental mode



Higher order mode



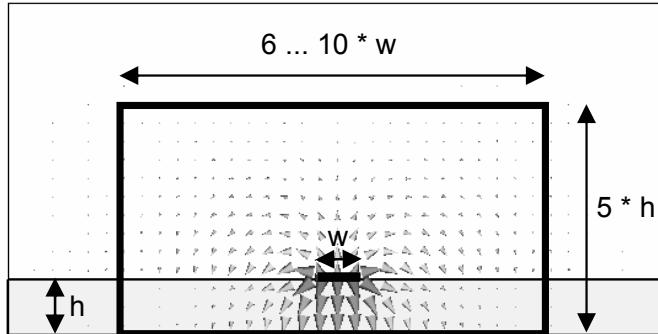
The higher order modes of the microstrip line are very similar to modes in rectangular waveguides. This behavior can be explained by an enclosure that is automatically added along the port's circumference for the port mode calculation. The boundary conditions at the port's edges will adopt the settings from the 3D model. In case of an "open" boundary in the 3D model, a "magnetic" port boundary will be used. This enclosure around the port causes the port to behave like a rectangular waveguide and thus is responsible for the particular pattern of the higher order modes.

The larger the port, the lower the cut-off frequency of these modes. Since the higher order modes are somewhat artificial, they should not be considered in the simulation.

Therefore, the port size should be chosen small enough that the higher order modes can not propagate, and only one (fundamental) mode should be chosen at the port.

If higher order microstrip line modes become propagating, this normally results in very slow energy decays in the transient simulations and sharp spikes in the frequency domain simulation results, respectively. On the other hand, choosing a port size too small will cause degradation of the S-parameter's accuracy or even instabilities of the transient solver. If you experience an unexpected behavior like this, check the size of the ports.

As a rule, the size of the port should be chosen according to the following picture:

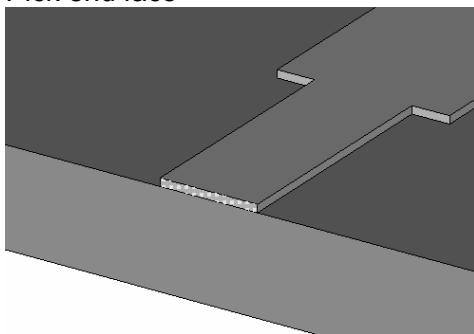


Ideally you would make the port 10 times as wide as the width of the microstrip line, but this may be reduced to around 6 times the microstrip line width in case of geometrical constraints.

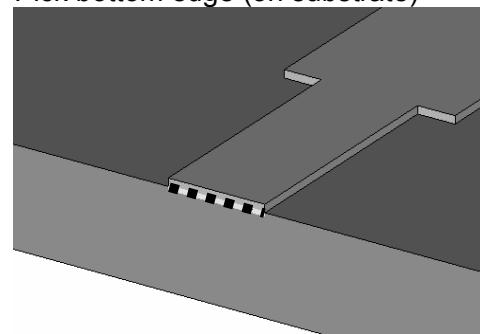
One way to define the port for this type of structure is to enter all its coordinates numerically. However, since this is a cumbersome solution, we will illustrate how ports for microstrip lines can be defined based on picked geometry.

You can pick the entire end face of the microstrip line or pick the edge of the end face which is located on the surface of the dielectric substrate. The following pictures illustrate these options:

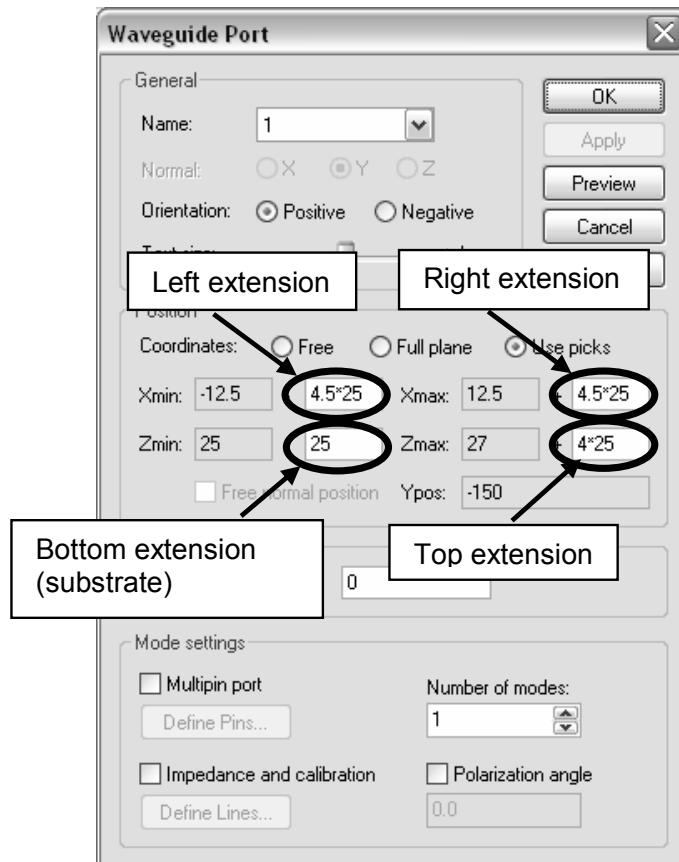
Pick end face



Pick bottom edge (on substrate)

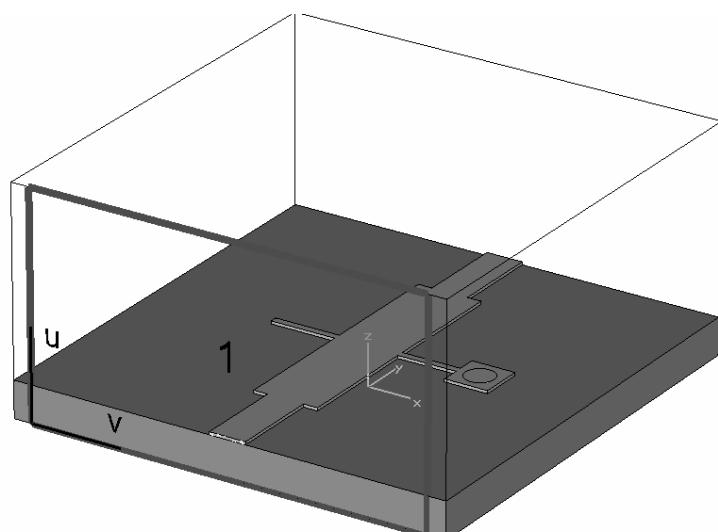


After picking the geometry, you can open the port definition dialog box (*Solve*⇒*Waveguide Ports*). In this dialog box, simply specify an extension of the port around the picked geometry by entering the distances in the corresponding entry fields. In case of a single picked line, you also need to set the port's normal direction and orientation since it cannot be automatically derived from the picked geometry.



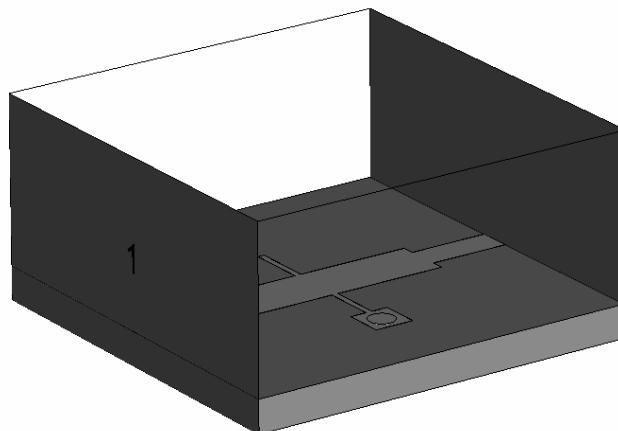
You need an extension space of ideally 4.5 times the width of the microstrip line (here 25) at each side of the microstrip line. Furthermore, you should specify a distance of 4 times the substrate height (here 25) above the strip line.

Note: Make sure to enter exactly the height of the substrate for the port extension below the microstrip line or you will introduce some unwanted additional space between the substrate and the ground metallization, or the port may not be connected to ground at all. We therefore recommend defining a parameter for the substrate's height (e.g. `substrate_height`). This parameter can then be used for specifying the bottom extension as shown above.



The explanations given above mainly refer to the unshielded microstrip lines. When a shielded microstrip problem needs to be simulated, the port can usually be chosen in such a way that it covers the entire dimension of the structure. The physical problem of unwanted box resonances appears much earlier in these cases than the artificial problem of unwanted propagating modes so that you do not need to worry about the latter here.

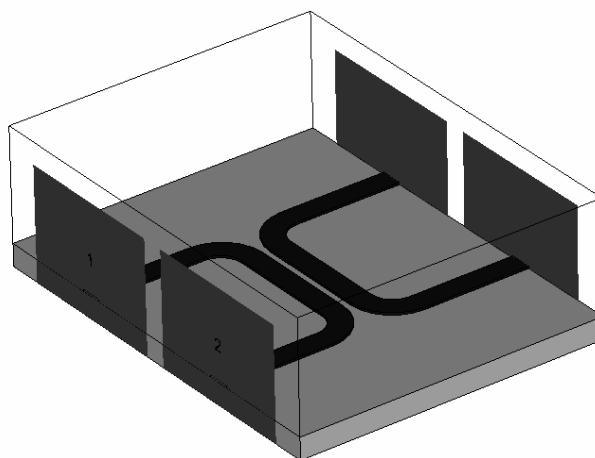
The picture below shows the assignment of ports for a shielded structure with perfect electric conducting walls at each side:



The easiest way to define a waveguide port covering the entire boundary face of the computation domain is to use the *Full plane* option in the port definition dialog box rather than using previously picked elements.

Some structures require more than one microstrip line being located at the same boundary of the computation domain. In general, you should try to avoid this situation if possible because it adds complexity to the port assignment.

However, if the microstrip lines are far enough away from each other that the interaction of fields is negligible, then ports can be assigned in the same way as described above using the same port size rule. An example for a structure like this is shown in the picture below:



If the lines get even closer, then they no longer work as two independent microstrip lines since the mode fields interact with each other and the waveguide becomes a *multipin waveguide*. Refer to the explanations later in this section for this type of waveguide.

Another important aspect in the simulation of microstrip lines is that the mode pattern is frequency dependent (unlike the mode patterns in empty guides or coaxial lines).

The frequency domain solvers automatically recalculate the mode patterns for every frequency point so that this frequency dependent behavior does not cause a difficulty for the analysis.

In contrast, the time domain solver uses the same mode pattern for the entire frequency band which may cause port mode mismatches at frequencies other than the mode calculation frequency. The error increases with increasing distance to the mode calculation frequency.

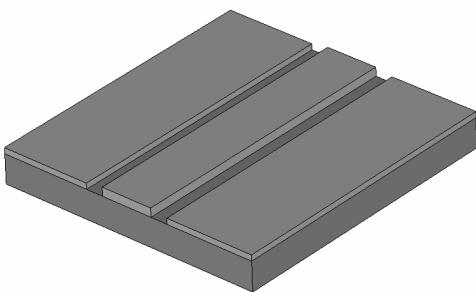
By default, the transient solver computes the mode pattern at the center frequency of the frequency band, but this behavior can be changed by specifying the *Mode calculation frequency* in the solver specials dialog box on the *Waveguide page*.

Despite this small mismatch at the ports, broadband simulation results will still be sufficiently accurate in most cases. However, very high accuracy requirements or very large bandwidths may require you to activate the *Inhomogeneous Ports* option in the transient solver control dialog box. This feature will improve the accuracy of the S-parameters but on the other hand will slow down the simulation. Refer to the online documentation for more information concerning this feature.

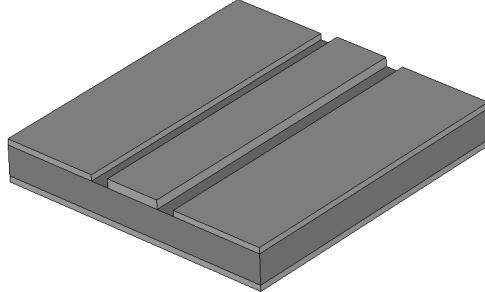
Coplanar Lines

The coplanar line is a frequently used transmission line for high frequency devices. Depending on whether the substrate is backed by a metallic shielding or not, the waveguide is either called *ungrounded coplanar line* or *grounded coplanar line*.

Ungrounded coplanar line



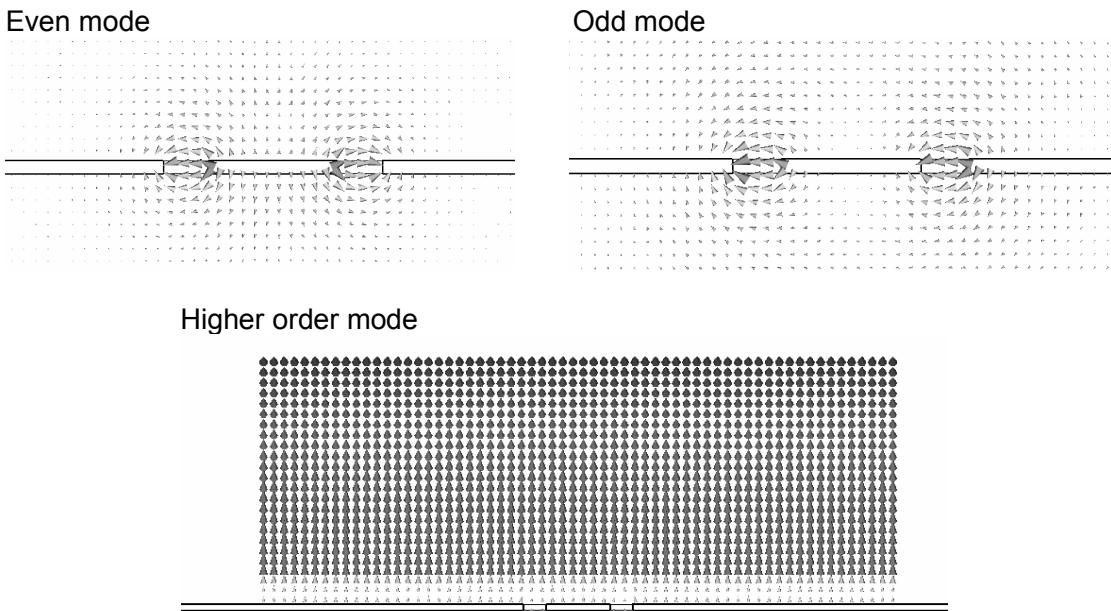
Grounded coplanar line



Unfortunately, this waveguide type is relatively complex from an electromagnetic point of view. Therefore some things need to be considered when defining ports for this type of structure. Many of the explanations in this subsection are similar to those for the *Microstrip line* case. However, we will repeat everything here once again for your convenience.

First of all, the size of the port is a very important consideration. On one hand, the port needs to be large enough to enclose the significant part of the coplanar line field. On the other hand, the port size should not be unnecessarily large because this may cause higher order waveguide modes to propagate in the port.

The following pictures show the fundamental even and odd modes as well as a higher order mode for an ungrounded coplanar waveguide:



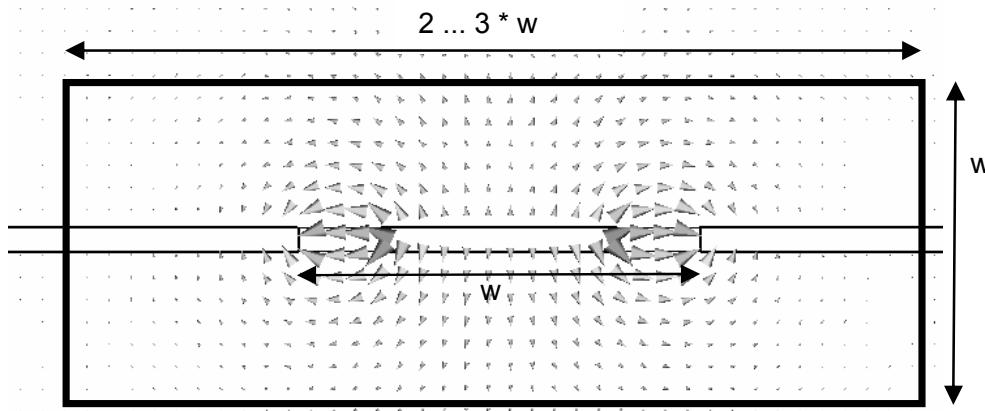
The higher order modes of the coplanar line are very similar to modes in rectangular waveguides. This behavior can be explained by an enclosure that is automatically added along the port's circumference for the port mode calculation. The boundary conditions at the port's edges adopt the settings from the 3D model. In case of an "open" boundary in the 3D model, a "magnetic" port boundary is used. This enclosure around the port causes the port to behave like a rectangular waveguide and thus is responsible for the particular pattern of the higher order modes.

The larger the port, the lower the cut-off frequency of these modes. Since the higher order modes are somewhat artificial, they should not be considered in the simulation. Therefore, the port size should be chosen small enough that the higher order modes can not propagate, and only the fundamental modes should be considered at the port.

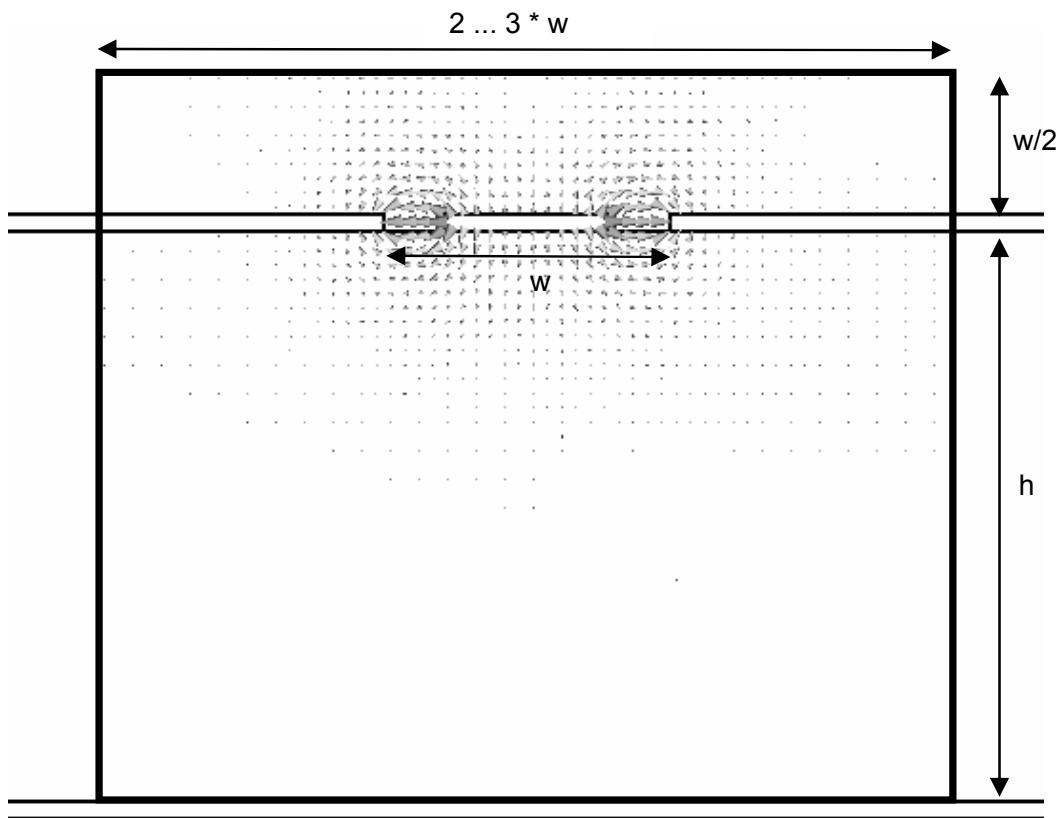
If higher order coplanar line modes become propagating, this normally results in very slow energy decays in the transient simulations and sharp spikes in the frequency domain simulation results, respectively. On the other hand, choosing the port size too small will cause degradation of the S-parameter's accuracy or even instabilities of the transient solver. If you experience an unexpected behavior like this, check the size of the ports.

In the following we will draw a distinction between grounded and non-grounded coplanar lines. The following pictures show the recommended sizes of the port for both cases:

Ungrounded coplanar line



Grounded coplanar line

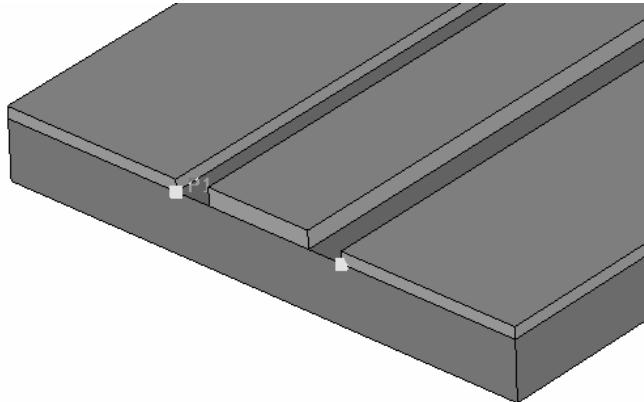


Ideally you would make the port three times as wide as the width of the inner conductor plus the width of the two gaps, but this may be reduced to around two times the inner conductor width plus the gaps in case of geometrical constraints.

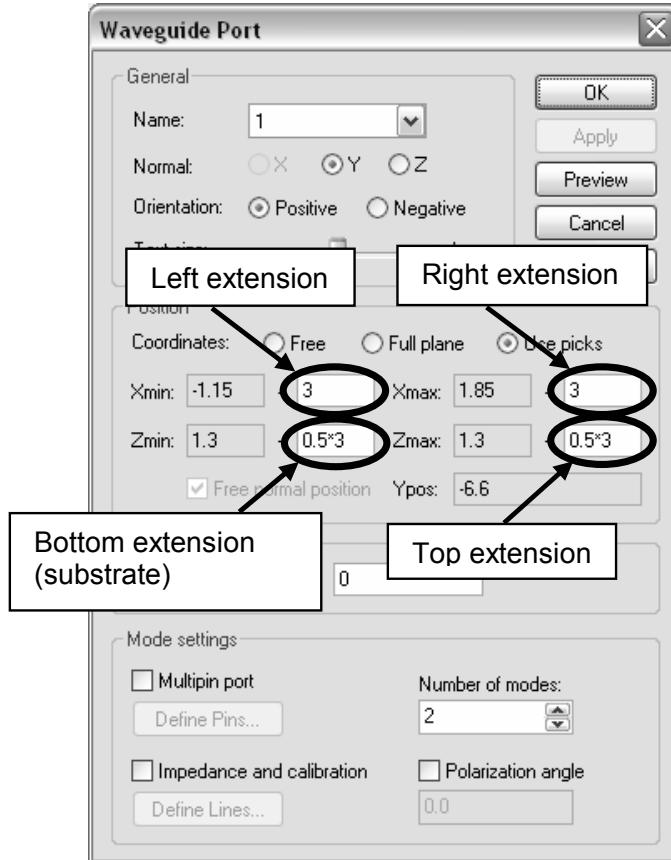
For ungrounded coplanar lines, you should enlarge the port by at least half of the inner conductor's width plus the width of the gaps above and below the metallization. For grounded coplanar lines the same is true above the inner conductor, but you should make the port large enough to touch the ground metallization below.

One way to define the port for this type of structure is to enter all its coordinates numerically. However, since this is of course a very cumbersome solution, we will illustrate how ports for coplanar lines can be defined based on picked geometry.

The easiest way to define a coplanar waveguide port is to pick the two innermost substrate corners of the outer conductors as shown in the picture below:



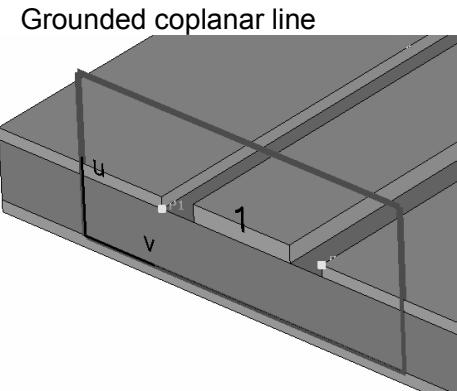
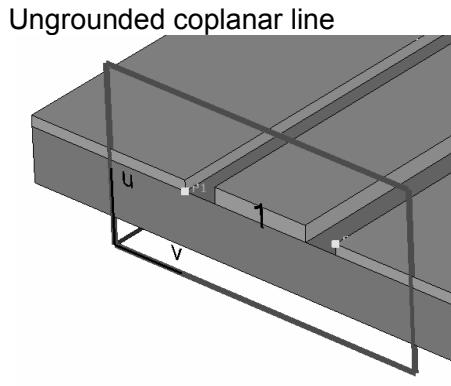
After picking the geometry, you can now open the port definition dialog box (*Solve*⇒*Waveguide Ports*). In this dialog box, simply select the port's normal direction and orientation before you specify an extension of the port around the picked geometry by entering the distances in the corresponding entry fields:



You need an extension space of ideally the width of the inner conductor plus the gaps line (here 3) at each side of the inner conductor line. Furthermore, you should specify a

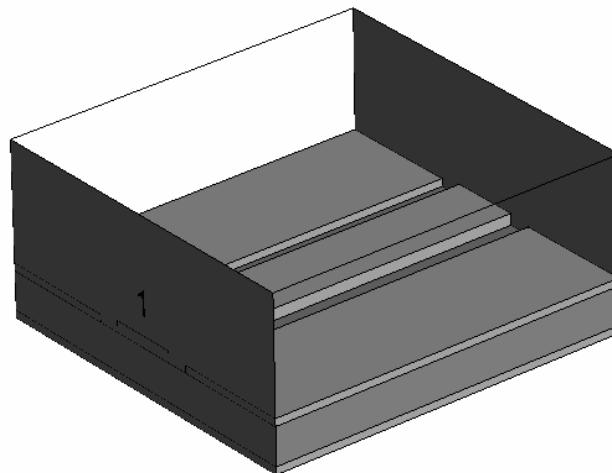
distance of half of the width of the inner conductor plus the gaps above and below the inner conductor.

In the case of grounded coplanar lines, make sure to enter exactly the height of the substrate for the port extension below the inner conductor or you might introduce some unwanted additional space between the substrate and the ground metallization, or the port may not be connected to ground at all.



The explanations above mainly refer to unshielded coplanar lines. When a shielded coplanar line problem needs to be simulated, the port can be normally chosen so that it covers the entire dimension of the structure. The physical problem of unwanted box resonances appears much earlier in these cases than the artificial problem of unwanted propagating modes so that you do not need to worry about the latter here.

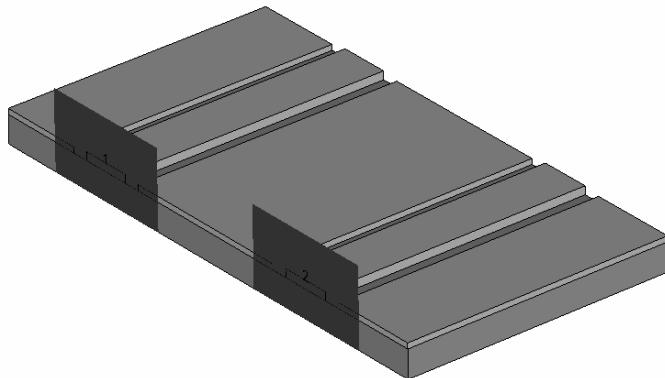
The picture below shows the assignment of ports for a shielded structure with perfectly electric conducting walls at each side:



The easiest way to define waveguide ports covering the entire boundary face of the computation domain is to use the *Full plane* option in the port definition dialog box rather than using previously picked elements.

Some structures require more than one coplanar line located at the same boundary of the computation domain. In general, you should try to avoid this situation if possible because it adds complexity to the port assignment.

However, if the coplanar lines are far enough away from each other that the interaction of fields is negligible, ports can be assigned in the same way as described above using the same port size rule. An example for a structure like this is shown here:



If the lines get even closer, then they no longer work as two independent coplanar lines since the mode fields interact with each other and the waveguide becomes a *multipin waveguide*. Refer to the explanations later in this section for this type of waveguide.

If no symmetry condition is defined in the middle of the coplanar line, both even and odd modes can exist and therefore need to be taken into account. If an electric symmetry condition is specified, only the odd mode can exist. On the other hand, setting a magnetic symmetry condition will consider the even mode only.

If the microstrip line is grounded, another parasitic microstrip mode will exist too as long as no electric symmetry is defined. All this leads us to the following table for the number of modes normally used for the simulation at the coplanar port:

Symmetry	Ungrounded Coplanar	Grounded Coplanar
None	2 modes	3 modes
Electric	1 modes	1 modes
Magnetic	1 modes	2 modes

The corresponding number can be specified in the *Number of modes* field in the port properties dialog box.

Note that the order in which the port modes are calculated may vary depending on the dimensions of the structure as well as the frequency. Therefore we highly recommend always inspecting the port mode results in order to avoid misinterpreting the S-parameter data.

Another important aspect in the simulation of coplanar lines is the fact that the mode pattern is frequency dependent (unlike the mode patterns in empty guides or coaxial lines).

The frequency domain solvers automatically recalculate the mode patterns for every frequency point so that this frequency dependency does not constitute a difficulty for analysis.

In contrast, the time domain solver uses the same mode pattern for the entire frequency band which may cause port mode mismatches at frequencies other than the mode calculation frequency. The error increases with increasing distance to the mode calculation frequency.

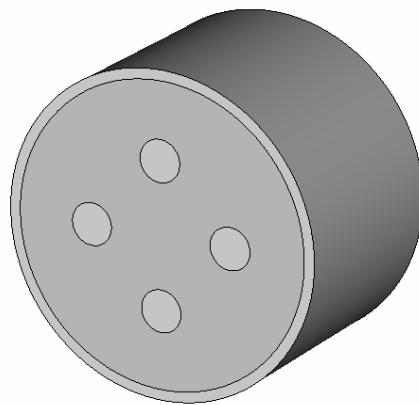
By default, the transient solver computes the mode pattern at the center frequency of the frequency band, but this behavior can be changed by specifying the *Mode calculation frequency* in the solver specials dialog box on the *Waveguide page*.

Despite this small mismatch at the ports, the broadband simulation results are still sufficiently accurate in most cases. However, very high accuracy requirements or very large bandwidths may require you to activate the *Inhomogeneous Ports* option in the transient solver control dialog box. This feature will improve the accuracy of the S-parameters but on the other hand will also slow down the simulation. Refer to the online documentation for more information concerning this feature.

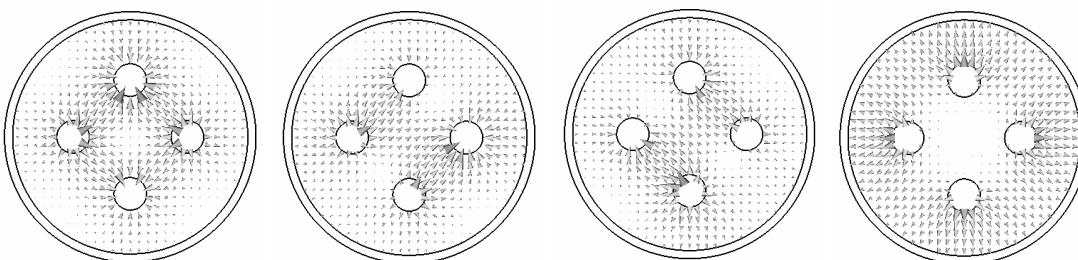
Multipin Waveguides

A *multipin waveguide* is characterized by having more than two conductors (e.g. two inner conductors plus shielding) in the port. According to this definition, a coplanar line is also a multipin waveguide. However, due to the importance of this particular transmission line type, it is explained in detail separately in the preceding subsection.

Let's first consider a simple case of a waveguide being homogeneously filled with a dielectric substrate and having more than two conductors:

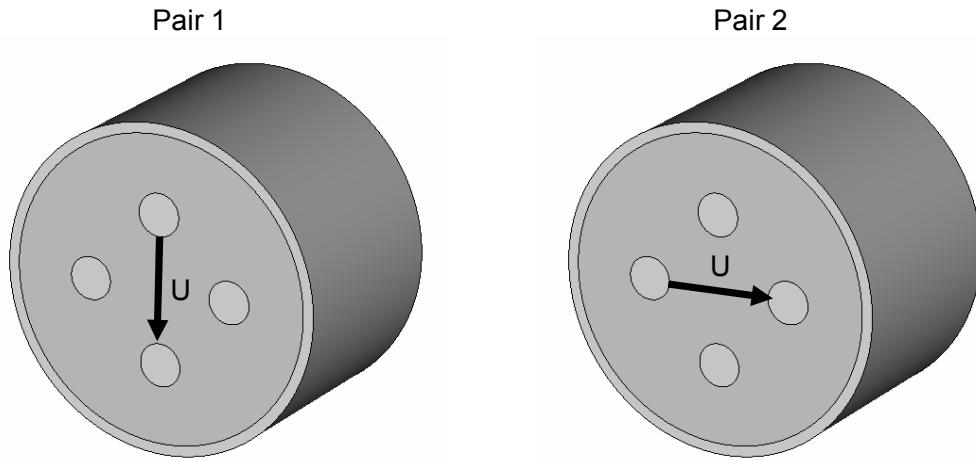


The mode patterns for this type of waveguide are frequency independent. For waveguides having N isolated conductors, $(N-1)$ propagating TEM type modes exist, all sharing exactly the same propagation constant. These modes are also called *degenerated* modes. The electric port mode fields for the example above are shown in the following picture (5 conductors \Rightarrow 4 modes):



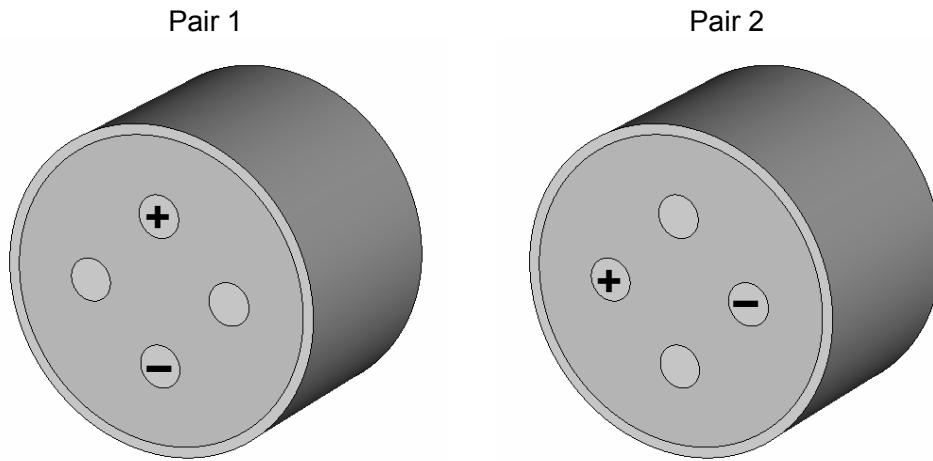
The port mode solver calculates an arbitrary superposition of the fundamental TEM modes, which unfortunately is not often the desired solution. More constraints are required to specify how to superimpose these modes to obtain the favored mode patterns.

Let's now assume that you want to obtain the S-parameters for two differential pairs. Each of these differential pairs is built by two opposite conductors as shown in the following picture:



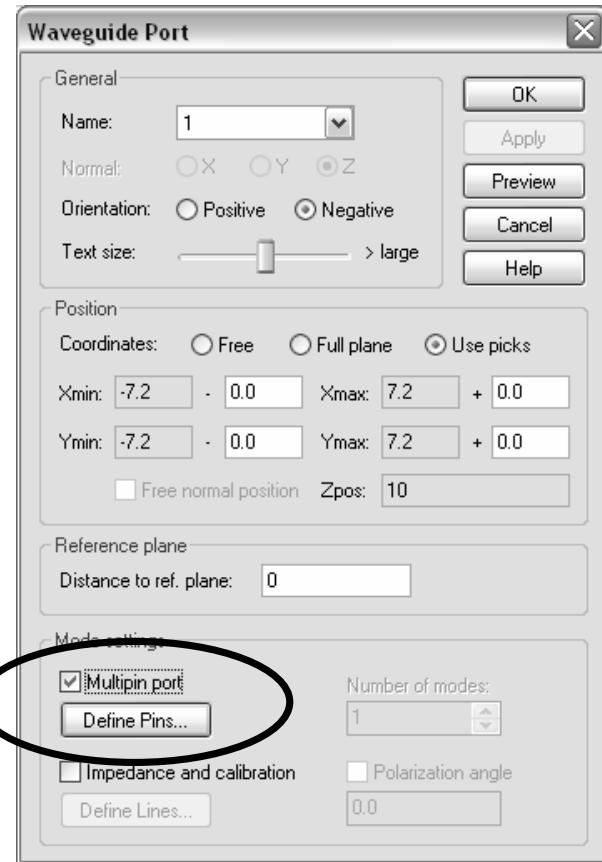
The transient solver features a special *multipin port* option that allows you to specify additional constraints. However, since this option is currently not available for the tetrahedral frequency domain solver and also has some limitations, an alternative approach using standard waveguide ports is explained later in the next subsection (*Handling Multipin Waveguides by Using Standard Ports*).

Each of the desired excitations in the example above corresponds to a particular mode which is a superposition of the structure's TEM modes. The necessary constraint to get the proper modes for the two pairs can be defined by assigning potential values to the conductors. The following picture shows the potential definition which yields the desired mode patterns:

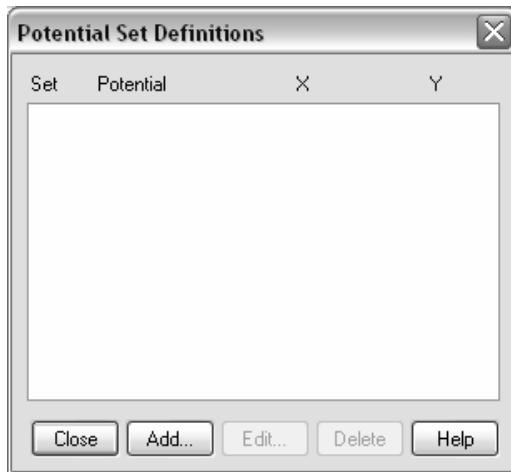


The definition of a *multipin port* starts by defining a port area of correct size (not too large and not too small) as shown in the sections above. Refer to the explanations of the transmission line type closest to your structure for more information.

Once the port size is determined, check the *Multipin port* box in the port dialog box:

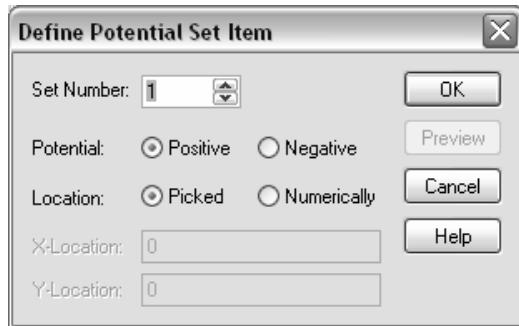


Clicking the *Define Pins* button opens the following dialog box where you can assign the desired potential settings to the conductors:



A *Set* corresponds to a particular mode defined by assigning a certain potential distribution to the conductors. Each *Set* can therefore contain multiple potential definitions and represents a particular superposition of the port's eigenmodes. In the example above, you need to define two *Sets*, each of which represents the mode for one of the differential pairs.

To define the first conductor potential, click the *Add* button to open another dialog box:

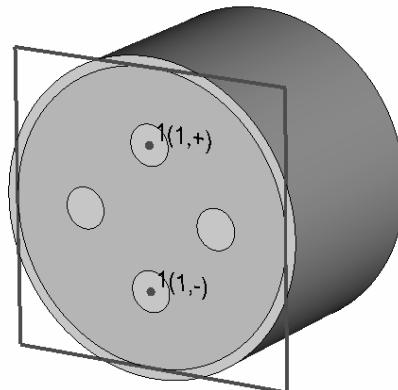


The *Set Number* is the first setting to make here. All *Sets* are simply enumerated, starting with 1. In our example, we define two *sets*, named as 1 and 2.

After selecting the correct set number, you can specify whether you are going to define a positive or negative potential. In this example, you need to define a positive and a negative potential for the two conductors within each pair.

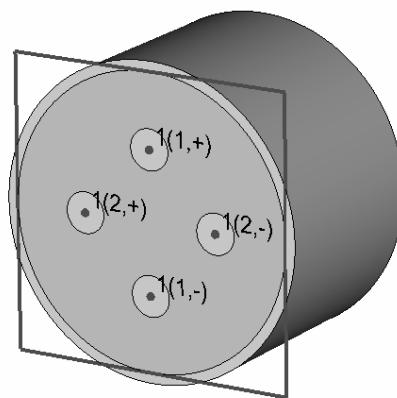
In most cases, it is recommended to place the potential assignment by picking the corresponding conductor in the structure view, so you can simply keep the *Location* being set to *Picked*.

Clicking *OK* enters an interactive mode so that you can simply double-click on an end face of a conductor in order to assign the potential definition there. Once you define the two potentials for the conductors belonging to pair 1, the model should look as follows:

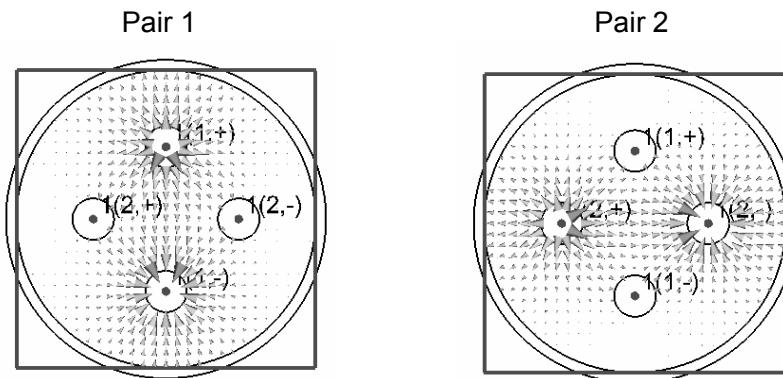


Each of the potential definitions is labeled using the following convention: The port number is followed by a label enclosed in brackets. The first entry within the bracket gives the set number whereas the second entry depicts the type of the potential (positive or negative).

You can now assign the corresponding potentials to the two conductors in the second mode set (corresponding to the second differential pair):



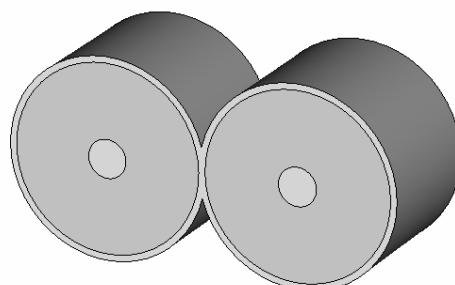
Taking these potential constraints into account, the port modes will be calculated as follows:



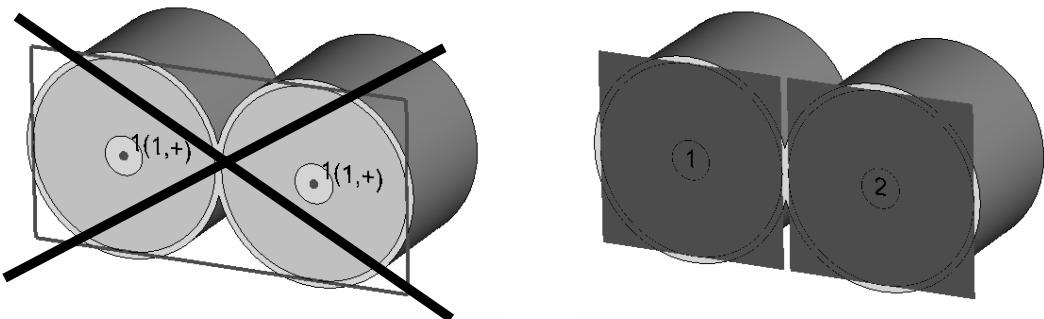
The transient solver and the hexahedral frequency domain solvers then calculate the S-parameters belonging to these mode definitions.

Please note that in order to calculate the interaction of even and odd modes in a single simulation run, it is also possible to use a conductor in several mode sets at the same time.

The *multipin ports* should **not** be applied to conductors that are completely shielded against each other:



For structures like this one, standard ports are preferred as shown below:

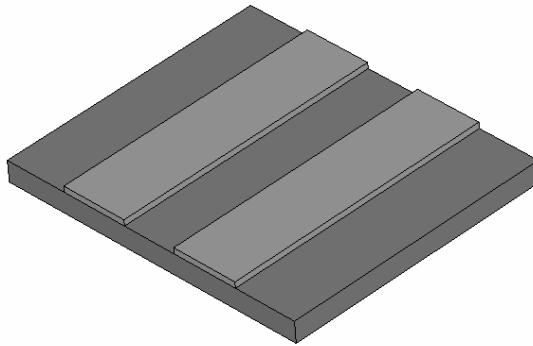


For inhomogeneous waveguides such as microstrip lines, coplanar lines, etc., the modes are generally not degenerated. In those cases, a superposition of the modes is not allowed and may lead to incorrect results. However, for relatively small frequencies or weak inhomogeneities, the modes may still be sufficiently degenerated. The solvers automatically check the level of mode degeneration and show a warning if the *multipin ports* should not be applied.

If the usage of multipin ports is not possible (for whatever reason), the next subsection gives some suggestions of how to handle this situation.

Handle Multipin Waveguides Using Standard Ports

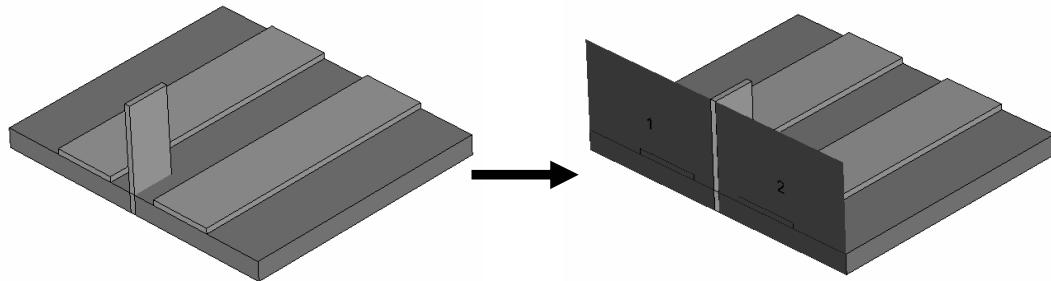
In some cases, multiple transmission lines are located at the same side wall of the bounding box and are too close to each other to be handled as isolated transmission lines.



The preceding subsection explains the *multipin port* feature which provides a good solution in many cases. However, for tetrahedral frequency domain simulations or inhomogeneous waveguides at higher frequencies, this convenient feature cannot be applied.

Please note that closely neighboured transmission lines are truly physically coupled. Therefore all measurement set up will introduce some mismatch which needs to be de-embedded properly. Keep this in mind when comparing S-parameters from simulation and measurement. Disagreement is observed quite regularly when the simulation set up does not replicate the measurement set up.

The easiest way to handle strongly coupled transmission line cases is to introduce a shielding at the ports in order to properly separate them. Afterward, waveguide ports can be individually applied to each of the transmission lines:



Even if this approach introduces some mismatch at the ports, of course, it may still be a viable solution. If you are unsure about the level of reflection added to the S-parameter matrix by this artificial shielding, you can simulate a short line segment alone. Ideally the reflection for the homogeneous line is zero. Therefore the actual reflection level provides some indication about the port mismatch for the particular structure.

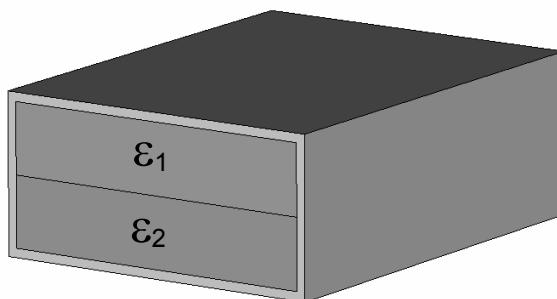
If the former approach does not produce sufficiently accurate results, the only way to handle this problem is to calculate the S-parameters for the multimode matrix by taking all propagating modes into account at each port. In transient simulations, you may also consider using the *Inhomogeneous port accuracy enhancement* option in the solver control dialog box to further increase the accuracy of the broadband S-parameter solution.

Once the multimode matrix is calculated, single-ended S-parameters can be obtained as a secondary result using the macro *Macros⇒Results⇒S-Parameter⇒Multiline Common+differential mode⇒single ended*. Refer to the macro's documentation for more information.

Inhomogeneous Waveguides

A waveguide is called *inhomogeneous* whenever more than one different dielectric material is present in the cross-section of the waveguide. This is the case for microstrip lines and coplanar lines – these two very common transmission lines are discussed earlier in this section.

The following explanations therefore exclude these types of waveguides and focus on dielectrically loaded waveguides. A typical example is shown in the following picture:



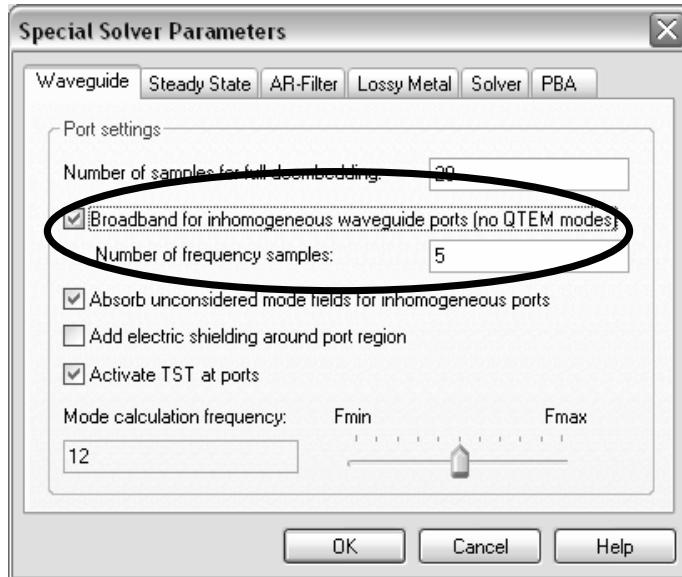
In terms of assigning ports to these waveguides, the procedure is very similar to the *Empty waveguide* case explained earlier in this section. The main difference here is that the port mode pattern is no longer frequency independent.

The frequency domain solvers automatically recalculate the mode patterns for every frequency point so that this frequency dependency does not constitute a difficulty for the analysis.

In contrast, the time domain solver uses the same mode pattern for the entire frequency band which may cause port mode mismatches at frequencies other than the mode calculation frequency. The error increases with increasing distance to the mode calculation frequency.

If you are interested in only very small frequency ranges, a viable solution may be to set the mode calculation frequency equal to the center frequency of this frequency band. This setting can be made by specifying the *Mode calculation frequency* in the solver specials dialog box on the *Waveguide page*.

However, if you are interested in broadband results within larger frequency ranges, the transient solver needs to be advised to make a special (and computationally expensive) treatment for the inhomogeneous ports. The *Broadband for inhomogeneous waveguide ports (no QTEM modes)* option can also be activated on the *Waveguide page* of the solver's specials dialog box:



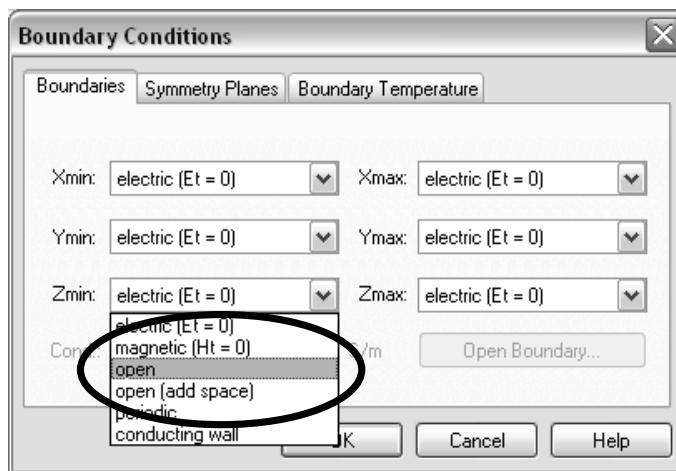
This option will provide much more accurate broadband results for this type of port. Unfortunately, it is limited to the fundamental (propagating) mode only. Refer to the online documentation for more information about this feature.

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.

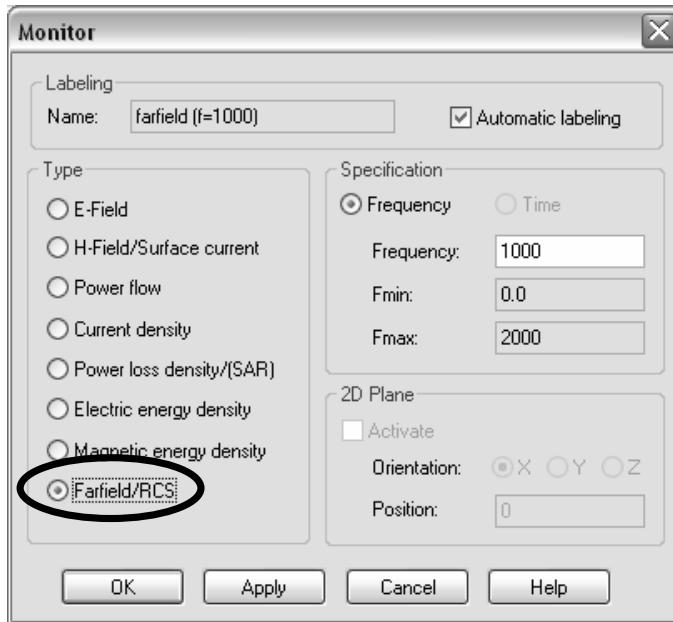
In some rare cases it may be advantageous to use the frequency domain solver. The only difference between these calculations and the procedure described below is that the frequency domain solver needs to be initiated rather than the transient solver. Refer to the frequency domain solver online help for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

With respect to the 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 *Solve* \Rightarrow *Boundary Conditions* () dialog box:



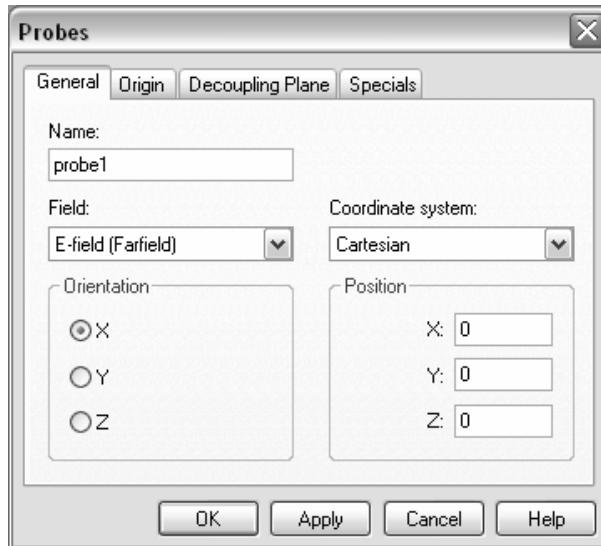
Open boundary conditions require some space between the device and the boundary planes for optimum performance. Since the open boundary conditions are very accurate, only a small distance is necessary. However, if you are not sure about the amount of space needed, simply choose “open (add space)” from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

For the calculation of the antenna farfield gain or directivity patterns (farfield distribution in spherical or Ludwig coordinate systems, left and right hand polarization, axial ratio), “farfield monitors” need to be defined before the simulation starts. Similar to the definition of the other field monitors, an arbitrary number of these monitors can be defined for various frequencies. This means that you can compute the antenna farfield for multiple frequency points from a single transient analysis. The farfield monitors are specified in the *Solve* \Rightarrow *Field Monitors* () dialog box:



After the transient analysis is complete, you can access your farfield results from the *Navigation Tree* \Rightarrow *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.

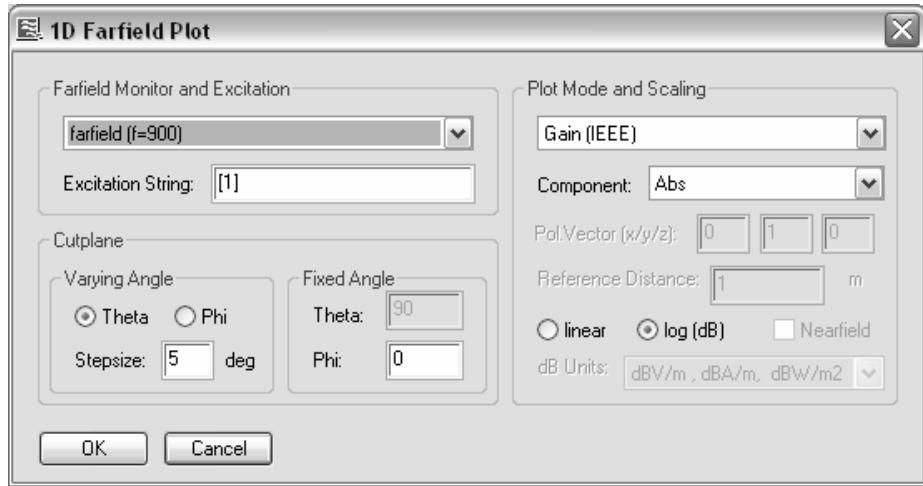
The farfield monitors introduced above are especially useful when the farfield needs to be calculated for a wide angular range (θ , ϕ) but only at a few frequency points. On the other hand, some applications require the calculation of the farfield at a few locations (θ , ϕ) only but as a function of frequency. In these cases, it is advantageous to use the farfield probes: *Solve* \Rightarrow *Probes*, *Field = E-field (Farfield)* or *H-field (Farfield)*:



In this dialog box, you can specify the type of the farfield, the location and the orientation of the desired probe in Cartesian, spherical or Ludwig coordinate systems. Please refer to the online documentation for more information about this feature.

Another very interesting functionality in combination with farfield calculations is the result templates whose basic functionality has already been demonstrated in the previous example. There are also some 1D Result templates available for automated farfield

processing, e.g. the *Farfield (single frq.)* template. Choosing this template from the 1D Result template list (*Results*⇒*Template Based Postprocessing*) will open the following dialog box:



Here you can specify which one of the previously defined farfield monitors should be processed and which farfield component and excitation ([1] corresponds to excitation at port 1) should be considered. Furthermore you need to specify the cutplane in the spherical coordinate system in order to derive a 1D curve from the farfield monitor. The result of this farfield processing template is a single result curve which can then be further processed by other result templates.

As an example, you could extract the location of the maximum by using a 0D result template (*0D Value From 1D Result*) and then use this value for optimizations of the main lobe direction to a certain angular location or magnitude. Please refer to the online documentation and the *Advanced Topics* manual for more information.

The following summarizes the input necessary for antenna calculations:

1. Select an antenna project template (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define structure.
5. Set frequency range.
6. Set (open) boundary conditions (optional).
7. Define excitation ports.
8. Set (farfield) monitors and/or probes.
9. Specify farfield result processing templates (optional).
10. Start transient or general purpose frequency domain solver.
11. Analyze results (input impedance, farfields, etc.).

Simplifying Antenna Farfield Calculations

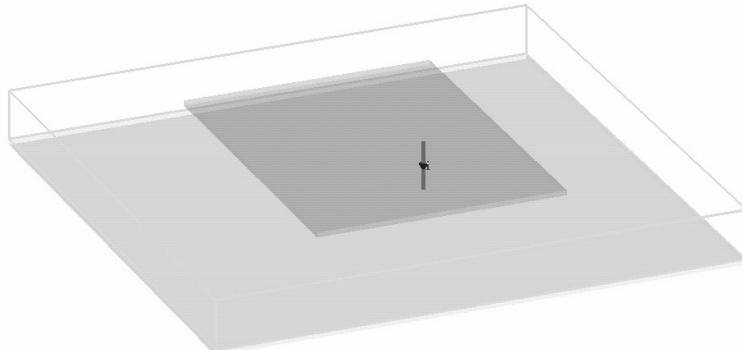
In many cases where only the antenna farfield pattern is of interest rather than the feeding point impedance, it is not necessary to model the actual geometry of the feeding point. However, when you want very accurate results of the antenna's input reflection, it is essential to model the feeding point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

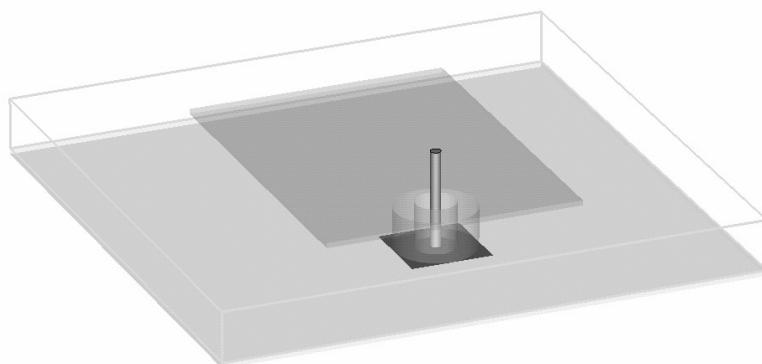
If you start the analysis of a new antenna it is usually a good approach to begin with a discrete port. Since the model is easier to build, you will obtain initial S-parameter and farfield pattern results quickly. This will allow you to assess the principal operation of the antenna before increasing 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 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.

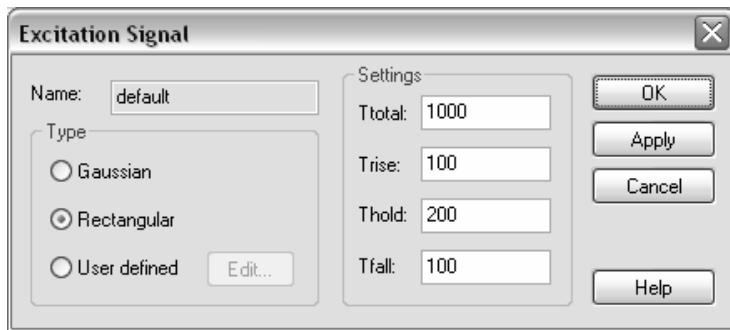
Digital Calculations

A digital calculation is typically performed using the transient solver. Thus the overall simulation procedure is 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.

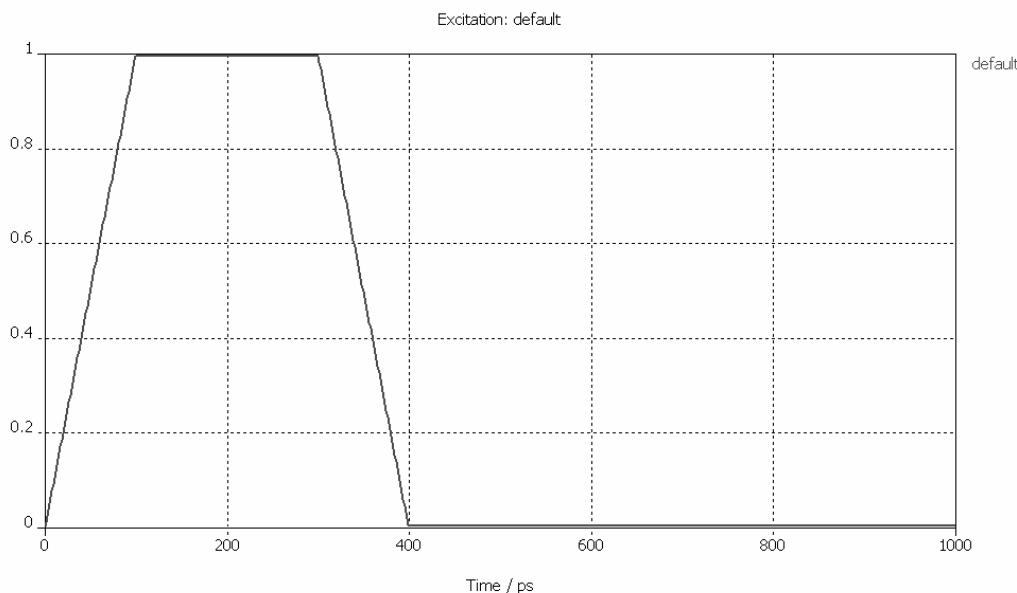
For S-parameter calculations the excitation signal for the transient analysis is typically defined by a Gaussian shape for which the Fourier spectrum is also given by a Gaussian shape covering the entire frequency band of interest. Therefore the time signal is determined mainly by the demands on the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

In contrast, the excitation signal for a digital simulation is described in the time domain by specifying rise-, hold- and fall-times of a rectangular pulse. You can change the excitation signal's shape by first selecting it in the navigation tree *NT*⇒*Excitation Signal*⇒*default* and then changing its properties (*Edit*⇒*Object Properties*) to open the following dialog box:



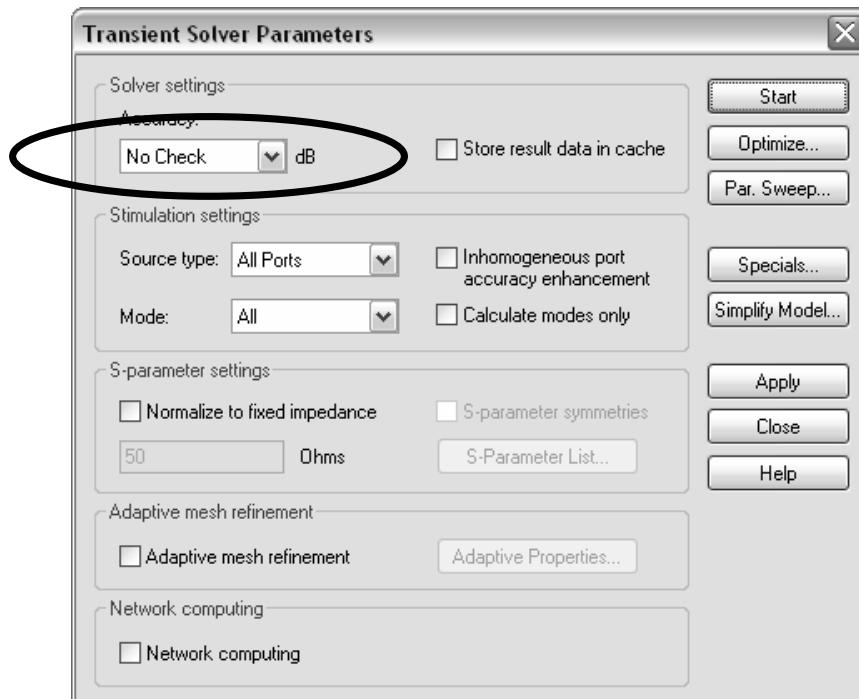
In the example studied above (with the time unit set to ps) the settings define a rectangular shape with a rise-time of 100 ps, a hold-time of 200 ps and a fall-time of 100 ps. The rise- and fall-times of 100 ps correspond to a bandwidth of approximately 10 GHz. The maximum simulation time is given in the *Ttotal* field and is set to 1000 ps in this example. The solver automatically stops after simulating the given total time range. The parameters of the rectangular excitation function are specified in the currently selected time units.

Once the rectangular excitation signal has been defined, it can be viewed by selecting it from the navigation tree *NT*⇒*Excitation Signals*:

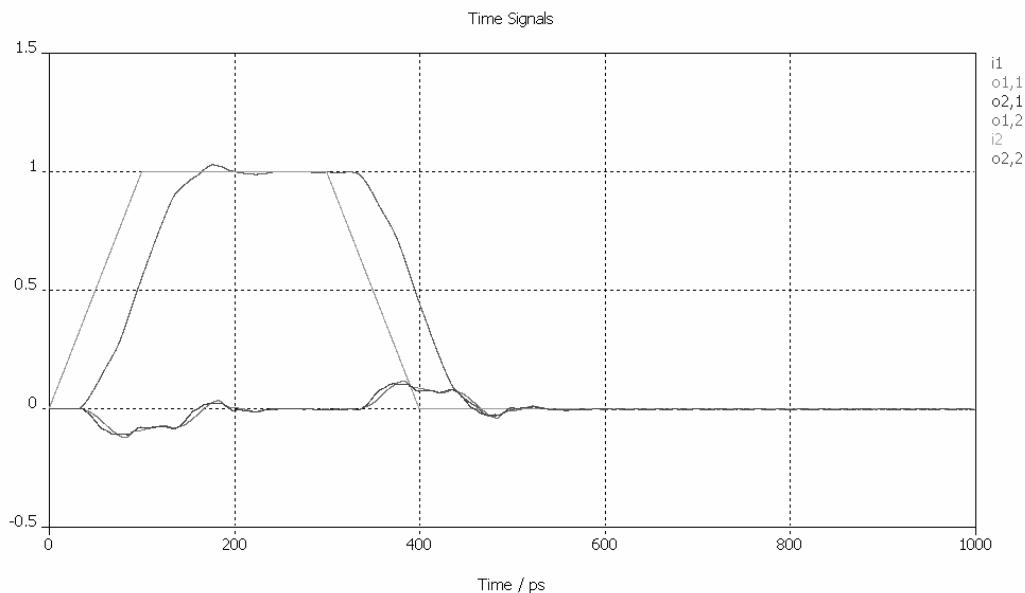


After the default excitation signal has been changed, the modified signal is used for all subsequent transient simulations. You can also specify additional excitation signals in order to excite different ports with individual excitation signals. Refer to the online documentation for more information about this feature.

The transient solver also stops automatically when the energy in the calculation domain is below the level specified in the *Accuracy* field. This may be the case even before the time specified by the *Ttotal* setting (see above) is simulated. To turn off this automatic termination, select *No Check* in the *Accuracy* field as shown in the transient solver control dialog box below:



In our example, the coaxial bend shows the following response to the digital excitation:



The excitation signal “i1” shows the given rise-, hold- and fall-times. The output signal “o2,1” has a distinctly distorted pulse shape (due to the dispersion of the coaxial bend) and a time delay because of the finite length of the transmission line.

In addition to this simplified description of the excitation signal, it is also possible to set a user defined pulse shape. Refer to the online documentation for details.

The following summarizes the input necessary for digital calculations:

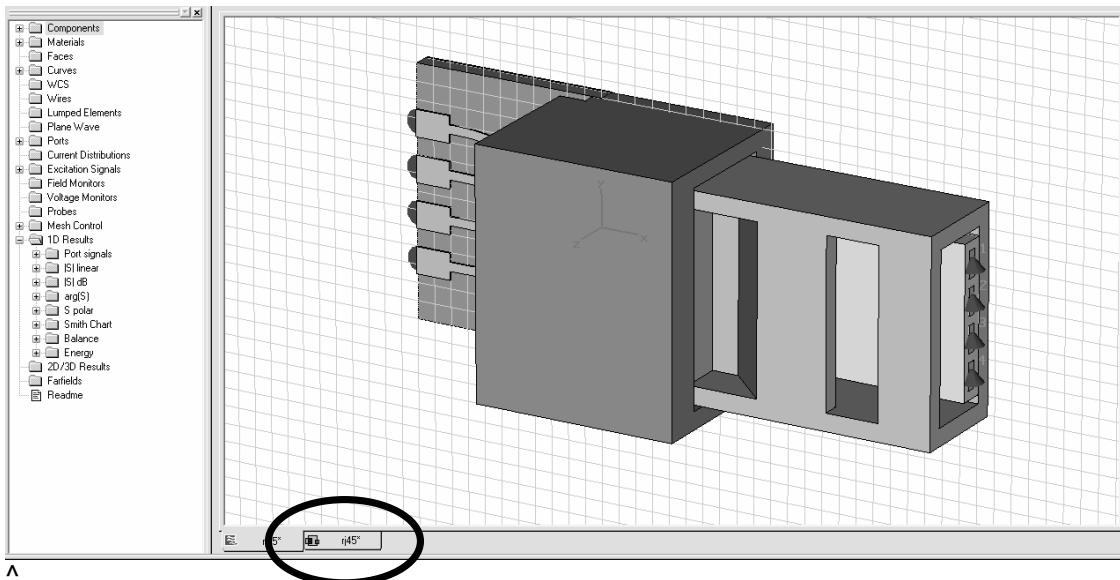
1. Select an appropriate project template (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define the structure.
5. Set the frequency range (covering all desired harmonics).
6. Set the boundary conditions (optional).
7. Define the excitation ports.
8. Set the monitors and/or probes (optional).
9. Define the excitation signal parameters.
10. Start the transient solver.
11. Analyze the results (usually the time signals).

There are some post-processing macros available especially dedicated to digital simulations such as eye diagram computation (*Time Signal Processing* \Rightarrow *Eye Diagram*) or exchange of excitation signals after the simulation (*Time Signal Processing* \Rightarrow *Exchange Excitation*).

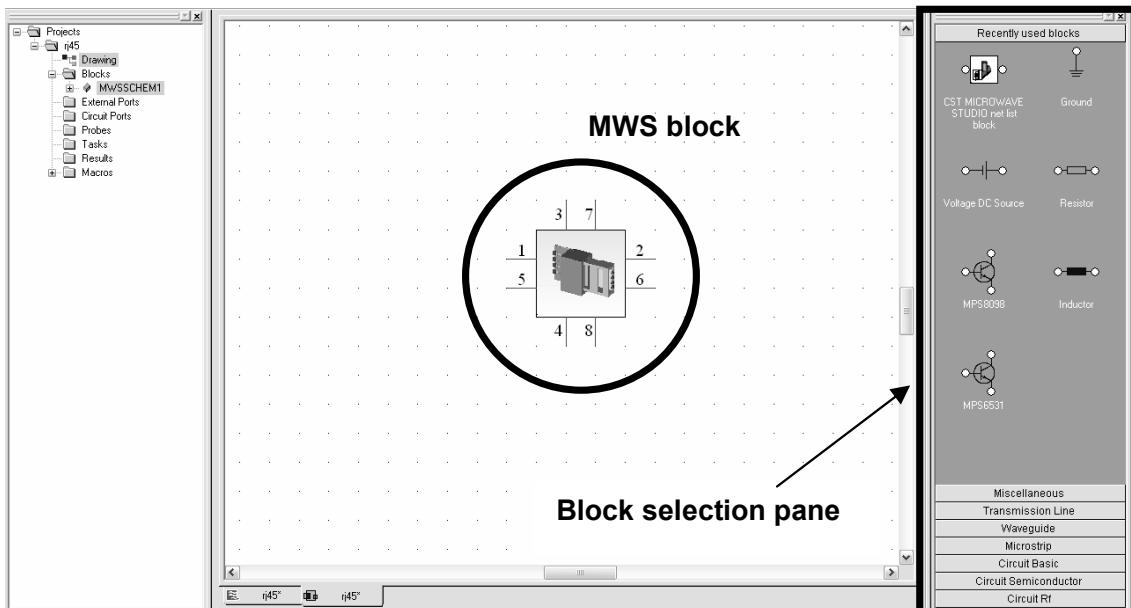
Add Circuit Elements to External Ports

The integration of CST MICROWAVE STUDIO® with CST DESIGN ENVIRONMENT™ allows for a straightforward coupling of EM simulation and circuit simulation.

For each CST MICROWAVE STUDIO® structure, two fundamentally different views on the model exist. The standard view is the 3D model representation which is visible by default. However, in addition, a schematic view can be activated by selecting the corresponding tab under the main view:



Once this view is activated, a schematic canvas is shown where the 3D structure is represented by a single block (MWS block) with terminals:



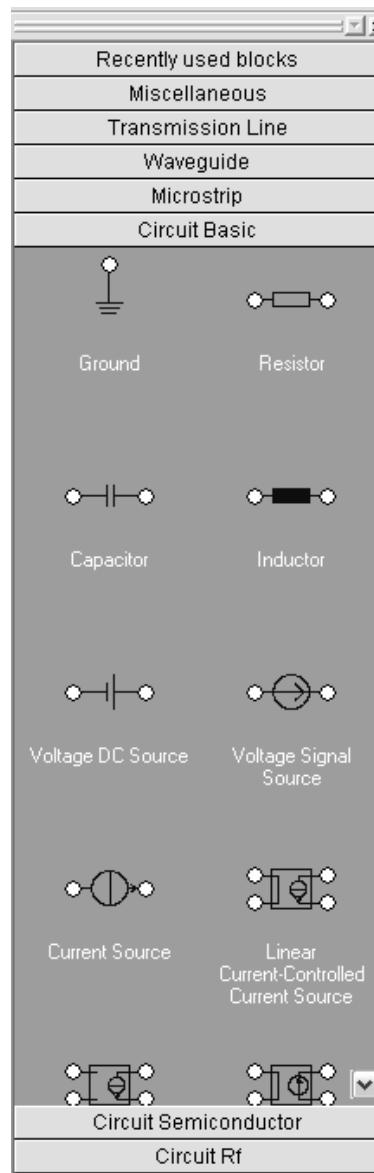
The terminals are a one-to-one correspondence to the 3D structure's waveguide or discrete ports. The schematic view now allows for easy addition of external circuit elements to the terminals of the 3D structure.

Adding circuit elements to the schematic view is straightforward. The general procedure is to select a circuit element from the block selection pane and drag it onto the schematic canvas.

The block selection pane is organized into several categories containing elements that may or may not be accessible to you according to license restrictions. Contact your sales office for more information.

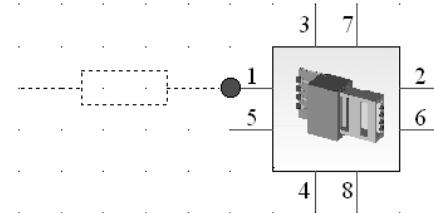
Basic circuit elements such as R, L, C, GND and ports are normally included in the basic CST MICROWAVE STUDIO® license.

You can access the circuit elements from the block selection pane after clicking on the *Circuit Basic* tab:



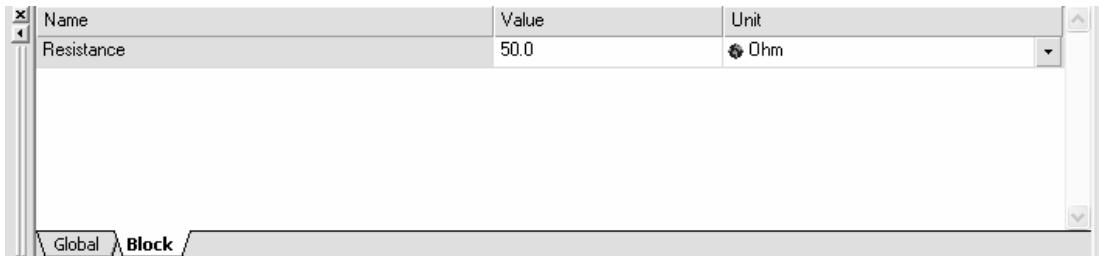
These elements can now either be dragged by pressing the left mouse button on the element itself or even more easily by pressing the left mouse button while the cursor is on one of the element's terminals. Note that an element's terminal is highlighted by a red circle whenever the mouse pointer is located over it.

Once an element is selected at its terminal and then dragged onto the schematic canvas, a red dot will appear whenever the terminal is located close to the end of an already existing connection line (here shown for the RJ45 connector example):



Releasing the mouse button there will then place and connect the circuit element on the canvas. Red terminal lines illustrate that this line is still open-ended whereas blue lines indicate proper connection.

Selecting a circuit element by single-clicking on it will reveal its properties in the parameter window as shown below:



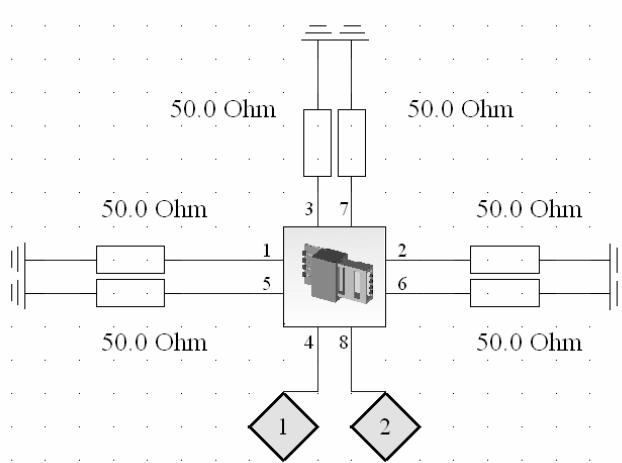
The element's properties can be modified in the usual way by clicking into the corresponding entry field and changing the value.

Ground (GND) elements and ports can also be accessed by dragging and dropping them from the block selection pane (for ports select the *Miscellaneous* section), but convenient shortcuts exist for these frequently used elements:

Shortcut	Description
g	Place a ground element. Simply click on the end of a connection line after activating this shortcut.
p	Place a port element. Click on the end of a connection line to place the port

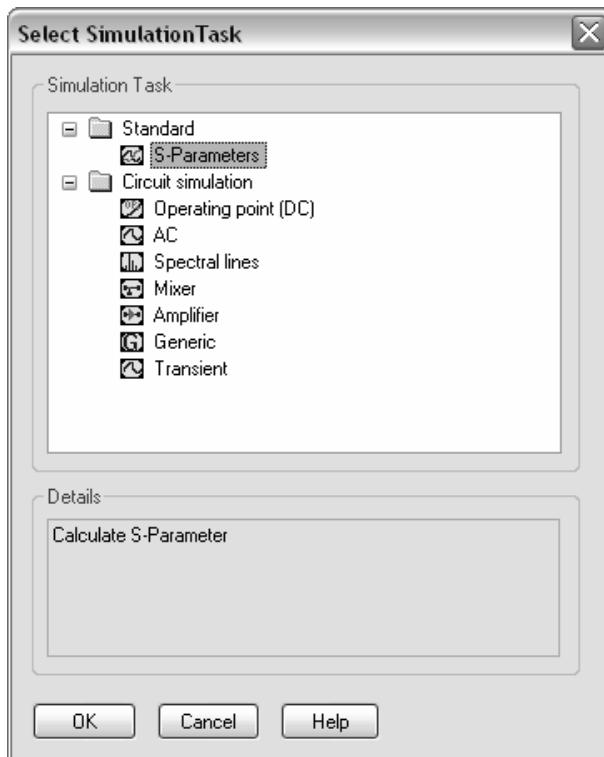
All elements can be dragged with the mouse to a new location while the elements stay connected via the connection lines. Furthermore, after you select an element, you can rotate it by choosing *View*⇒*Tools*⇒*Rotate/Flip Tools*⇒*Rotate Left* (shortcut l) and *View*⇒*Tools*⇒*Rotate/Flip Tools*⇒*Rotate Right* (shortcut r), respectively.

The following picture shows an exemplary external circuit where the original 8-port structure has been reduced to a 2-port device by applying loads to all the other terminals:

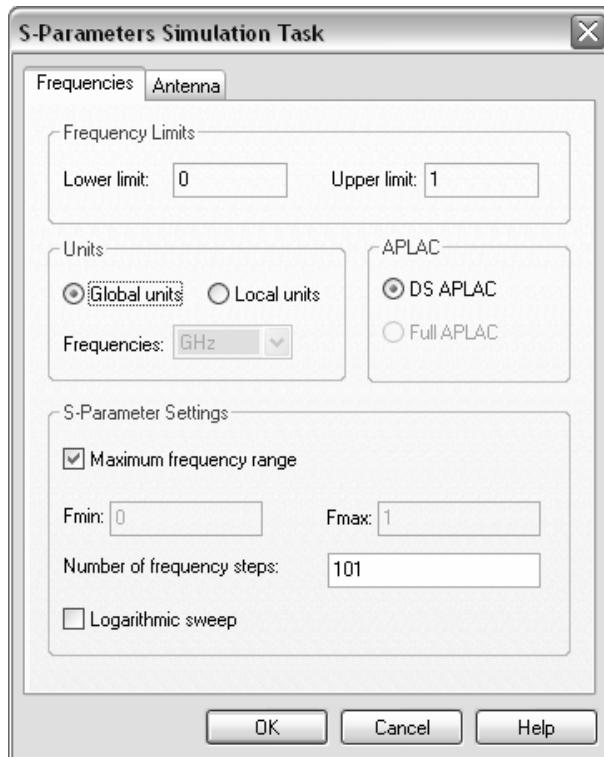


Once all terminals of the MWS block are connected to circuit elements, an S-parameter simulation of the structure including the external circuitry can be performed. The resulting S-matrix will have two ports in the example above, since only two external ports have been placed in the schematic (represented by diamond-like symbols).

In order to run an S-parameter circuit simulation, choose *Simulation* \Rightarrow *Update*. The circuit simulator supports a variety of different analysis options, so you will be prompted to select one of them:



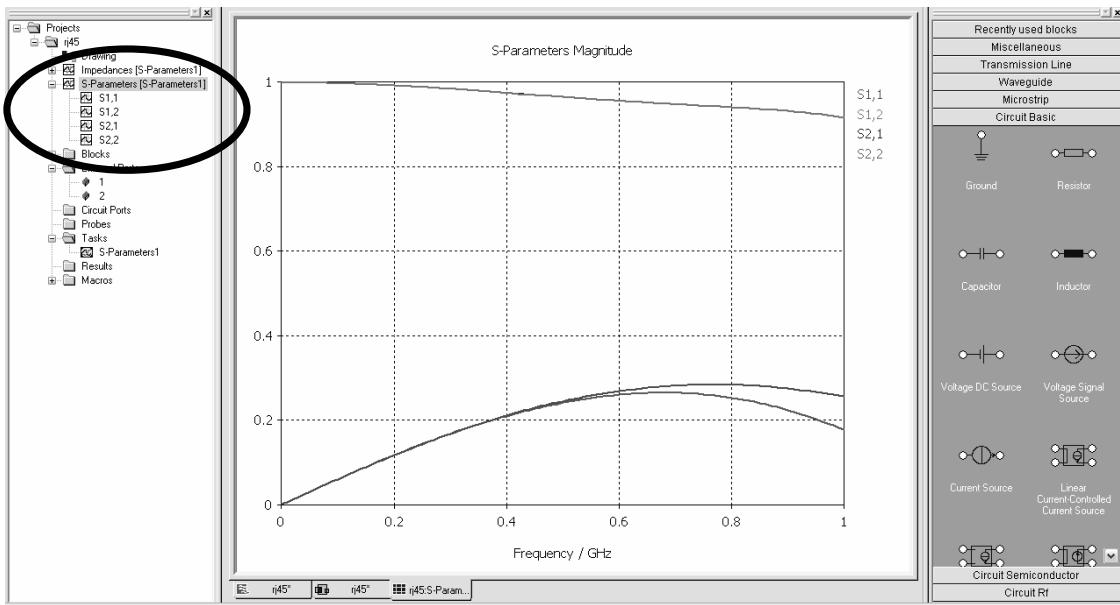
Not all of these options may be available to you due to license restrictions. Here you should select **S-Parameters** and click **OK**. A dialog box will appear in which you can specify the settings of the S-parameter calculation:



The most important setting is the *Number of frequency steps* here. Clicking **OK** will start the S-parameter simulation of the coupled EM and circuit problem. If all simulation results of the 3D EM simulation are already present, the circuit simulation will only take a few seconds to complete. Otherwise, the missing EM simulation data will be automatically computed first.

Once the simulation is complete, the results will be added to the navigation tree. The *Projects* folder will contain a list of sub-folders, each of which represent a currently opened project. All data will be stored under the folder for the corresponding project.

The S-parameters will be shown as top-level results of the project folder. Simply clicking on the *S-Parameters* entry will show the external S-parameters:



The schematic view also supports the parameterization of the 3D model in a straightforward way by making the structure parameters available as block parameters for the MWS block. This tight coupling of the different simulation methods allows for true EM / circuit co-simulation.

For more information on how to use the schematic canvas or the many simulation options, refer to the CST DESIGN STUDIO™ *Getting Started* manual.

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 *Network parameter extraction* features that allow the automatic generation of a SPICE compatible netlists for a given device.

Two different extraction methods are available:

1. The general purpose extraction scheme based on a model order reduction
2. The specialized extraction scheme based on a fixed topology (network of coupled transmission lines)

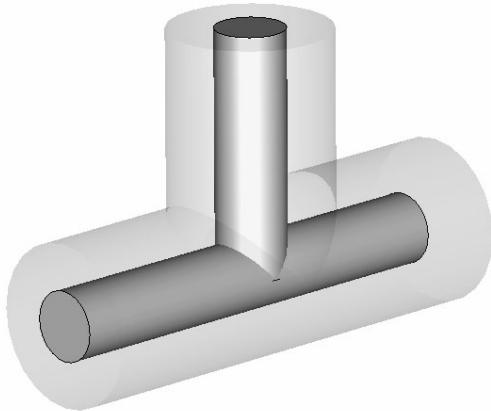
The following sections provide a brief introduction to the particular advantages and the typical applications for each of these schemes. Please refer to the online documentation for more detailed information.

Model Order Reduction Based Network Parameter Extraction

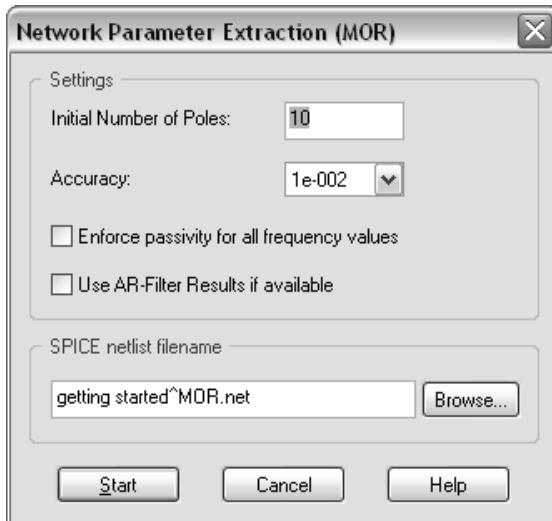
This model is based on a SPICE network with controlled sources and does not assume any particular topology. The advantage of this approach is that the SPICE model can be extracted for arbitrary structures, but on the other hand the network parameters do not have any physical meaning. Therefore the SPICE model should be considered as a black box that has the same electrical behavior at its ports as the full 3D structure.

For the practical usage of this extraction technique, it is usually required to calculate the full generalized S-matrix of the structure first.

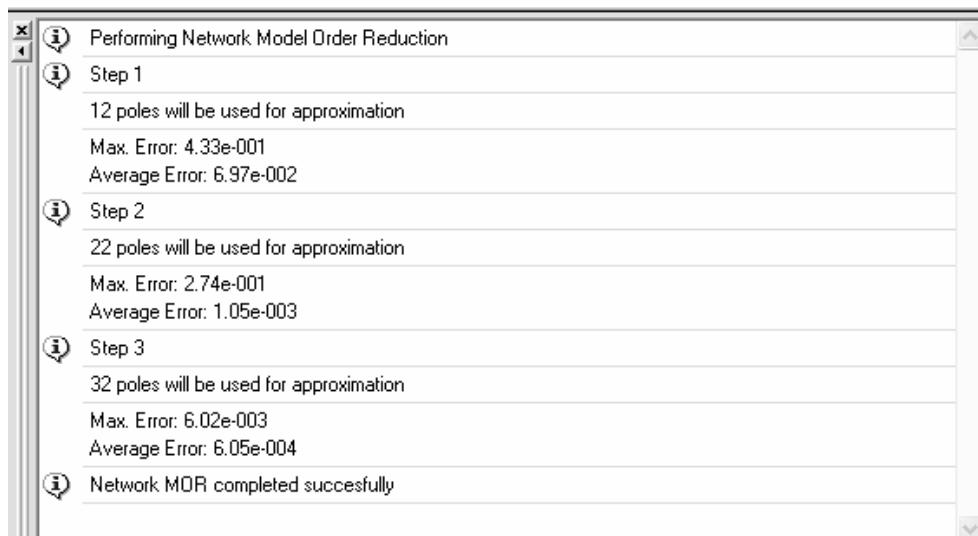
Let us now consider the previous example and derive an equivalent SPICE circuit for the coaxial bend.



Once all generalized S-parameters have been calculated (*All Ports* option, no re-normalization to a fixed port impedance), open the dialog box by choosing *Results* \Rightarrow *Network Parameter Extraction* \Rightarrow *Model Order Reduction*:



In most cases, you can accept the default settings and click the *Start* button. After the network parameter extraction calculation is complete, you should carefully check its output in the message window:



First, check whether the network extraction has completed successfully or whether any warning or error messages have been issued. The generated *SPICE netlist file* containing a subcircuit representing the actual S-parameter data will then look as follows:

```
*****
*** Netlist for Macromodel generated by CST DESIGN ENVIRONMENT ***
*** This file was generated automatically ***
*****
*** Generated 14-Sep-2005, 19:00 ***
*****
*** Connect the external circuit to ports 1, 2 ..... ***
.subckt getting_started 1 2
*** Nodal Ckts Associated With Each State Variable ***
Cn1    node1   0     1e-9
Ra11   node1   0     3.580667898402919e-005
Gb11   node1   0     1     -1.000000000000000e+000
```

```
** End of Nodal Ckt for State Variable #1 **

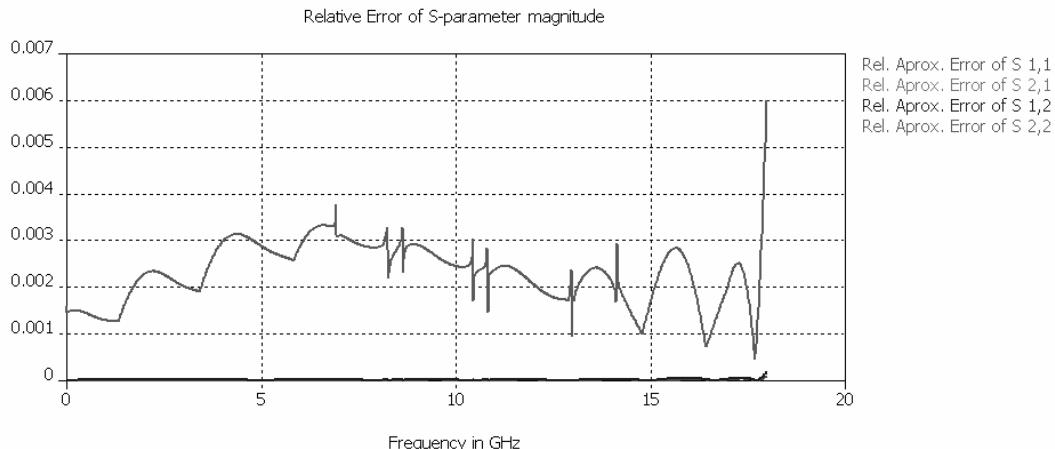
Cn2    node2  0      1e-9
Ra22   node2  0      3.580667898402919e-005
Gb22   node2  0  2  0  -1.000000000000000e+000
** End of Nodal Ckt for State Variable #2 **
```

.

Having verified that the calculation was successful, you should then look at the approximation error which is significantly below 1% for the entire frequency band in this example.

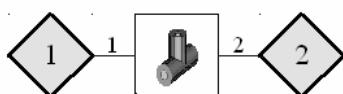
Furthermore, the passivity requirements are met which means that the equivalent SPICE circuit is stable and passive and therefore ideally suited for embedding in larger system or circuit simulations.

Some more information about the results of the network extraction can be found in the navigation tree's *NT*⇒*Network Extraction* folder. Check the agreement of the original data and the data provided by the model order reduction by selecting the *NT*⇒*Network Extraction*⇒*Error*⇒*Relative*⇒|S| linear folder:



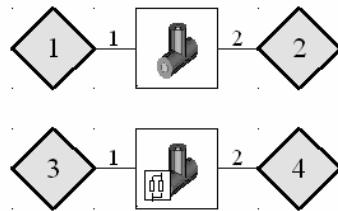
A final verification of the generated netlist can be obtained by loading the netlist into the schematic and comparing its S-parameter results with the original block data.

Now switch to the canvas by clicking the corresponding tab and add two ports to the CST MICROWAVE STUDIO® block representing the current 3D structure. For more information about how to use the schematic canvas, refer to the section *Adding Circuit Elements to the External Ports* earlier in this chapter. The canvas should then look as follows:

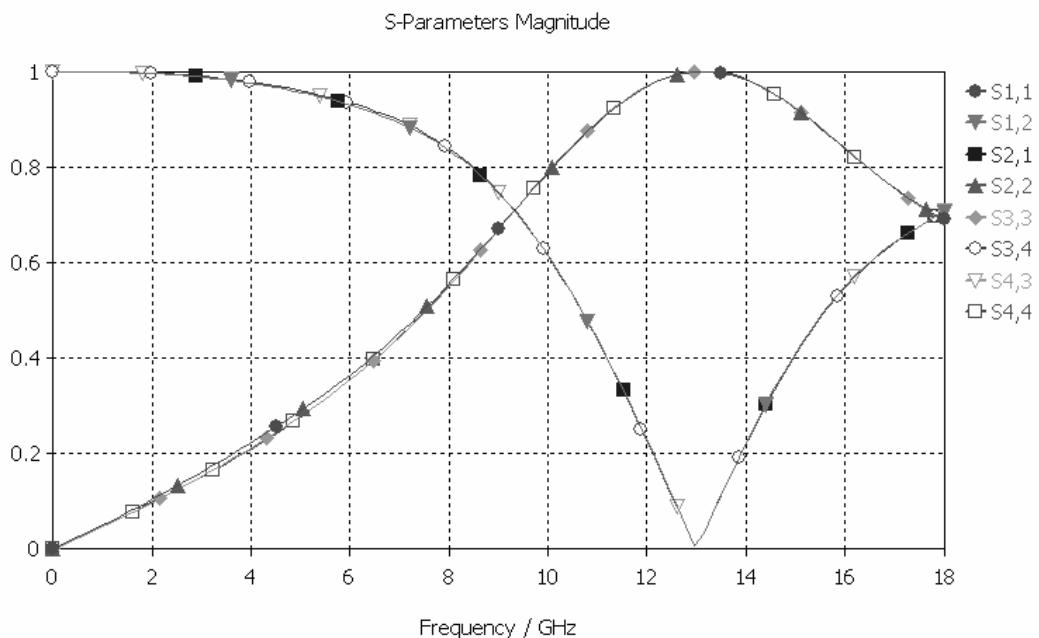


You should now place a CST MICROWAVE STUDIO® netlist file block in the canvas by dragging it from the block selection pane, *Miscellaneous* section. When you are asked to

select a netlist file, simply browse for the file which has previously been generated by the SPICE netlist extraction tool. After adding ports to the netlist block again, you should obtain the following schematic:



Running an S-parameter circuit simulation and looking at the results as described in the section *Circuit Elements to the External Ports* will produce a plot as follows:



As the picture shows, there is excellent agreement between the results obtained by the circuit simulator using the extracted netlist and the original S-parameter data.

Transmission Line Based Network Parameter Extraction

Since this model is based on a network of coupled transmission lines, the approximation is best for transmission line type devices such as connectors or IC packages. It should not be used when the topology of the simulated device is not fundamentally equivalent to a simple network of coupled transmission lines.

The practical usage of this 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.

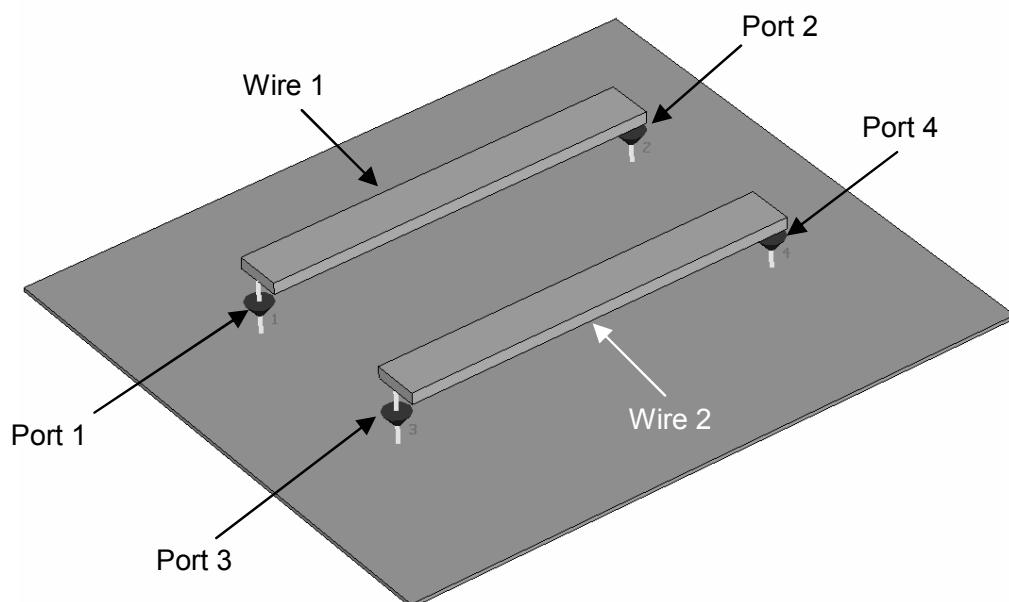
Please note: In contrast to the Model Order Reduction based network parameter extraction, this method requires only the excitation of one port for each wire rather than an excitation at each port of the structure. Therefore it is advantageous not to calculate the full S-matrix in advance by using the *All Ports* setting in the transient solver but to let the network parameter extraction feature do the selection of necessary excitation ports.

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

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

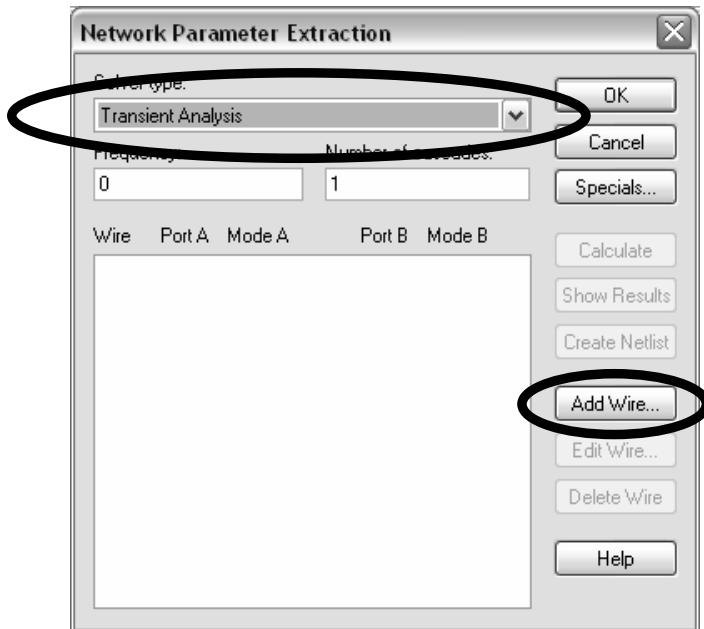
Since the network model assumes that the structure can be described by a lumped element network, the validity of this approach is limited to frequency ranges within which the structure behaves like a transmission line network. Therefore it is usually sufficient to use discrete ports for this kind of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

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



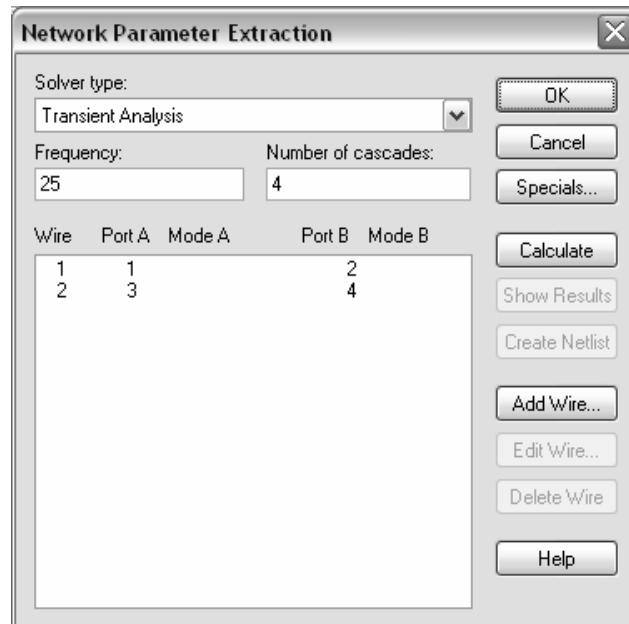
All conductors (including the ground plane) are modeled as perfect conductors here. It is not necessary to model the ground plane because the corresponding boundary plane can simply be set to *electric*.

The next step is to open the network parameter extraction dialog box by selecting: *Results* \Rightarrow *Network Parameter Extraction* \Rightarrow *Transmission Line*:



First specify which solver should be used for the calculation of the network model. This is usually set to *Transient Analysis*.

Then 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 so that the corresponding wave length is approximately 100 times larger than the length (l) of the transmission lines, thus: $f \approx 0.01c/l$.

Now specify the number of cascades so that 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. Refer to the online documentation for more details.

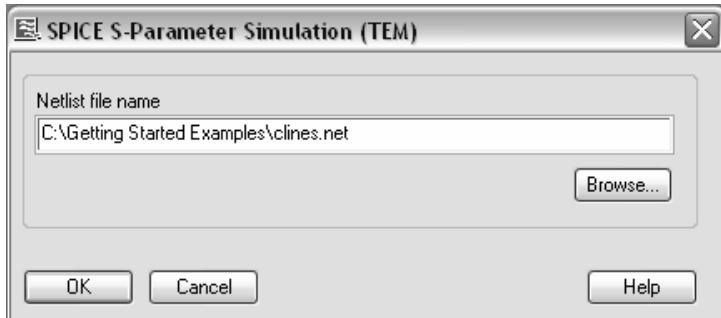
Finally start the network parameter extraction by clicking the *Calculate* button. If the S-parameter results are not yet available, the transient solver will automatically be initiated before the actual SPICE model generation is performed. The transient solver will need to be started only once per wire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

After the solver has finished, click 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 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.

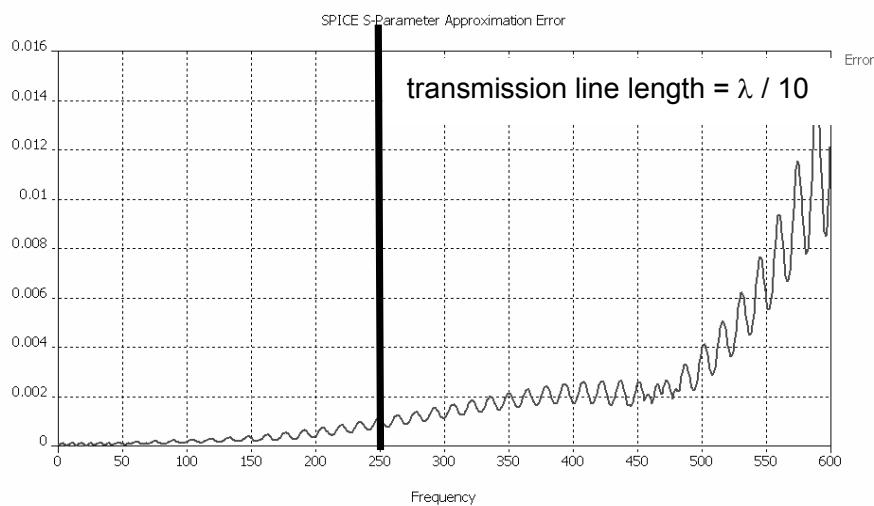
To use this macro you will first need to generate a SPICE netlist by clicking on the *Create netlist* button. Save the netlist with the same name as your structure, and then close the parameter extraction dialog box by clicking *OK*.

The macro can then be called by selecting *Macros* \Rightarrow *Touchstone + SPICE* \Rightarrow *Calculate SPICE S-Parameters (TEM)* which opens the following dialog box:



Check the *Netlist file name* before clicking *OK*. Afterward 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* \Rightarrow *SPICE* will be 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 0.1% up to a transmission line length of a tenth of a wavelength. 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 a third of a wavelength, the approximation error remains roughly below 3%.

Cadence® Allegro / APD Plug in

The Cadence® Allegro / APD Plug in is a component installed on top of an existing Cadence® Allegro or APD installation. The plug in's user interface is then embedded into the corresponding tool's environment and allows exporting models from within the Cadence® flow. The exported models can directly be opened in CST MICROWAVE STUDIO®. The plug in is compatible with both the UNIX and the Microsoft Windows versions of the Cadence® tools.

The following section explains the installation of the Cadence® Allegro / APD plug in first before focusing on how to export models from within the Cadence® flow.

The plug in is written in Cadence's Skill® programming language, so the Cadence® Allegro / APD installation needs to include the Skill language components. As an example, Allegro PCB Studio without the Allegro Performance options does **not** provide a Skill interface.

The plug in is compatible with the following tools:

- AP Designer / AP Engineer
- Allegro PCB Design Expert
- Allegro PCB Designer
- Allegro PCB Studio (with Allegro Performance option)

Installation

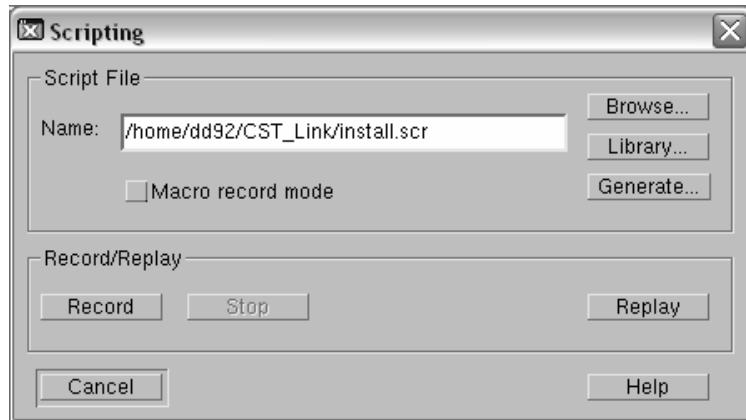
The installation of the plug in has been greatly simplified by an installation program which is provided in form of a Cadence® script file.

First, you need to copy the contents of the *Cadence Allegro APD Plug in* folder in the installation directory of CST STUDIO SUITE™ into your home directory. If you are using FTP to transfer the files, make sure to copy the entire directory structure using the *binary* file transfer option. Afterward, a folder named "CST_Link" should exist in your home directory.

Next, you need to set an environment variable named *CST_Link_HOME* pointing to the full file path name of the "CST_Link" directory. Both the name of the environment variable and the path are generally case sensitive. Make sure that this environment variable is available whenever you run Cadence® Allegro or APD.

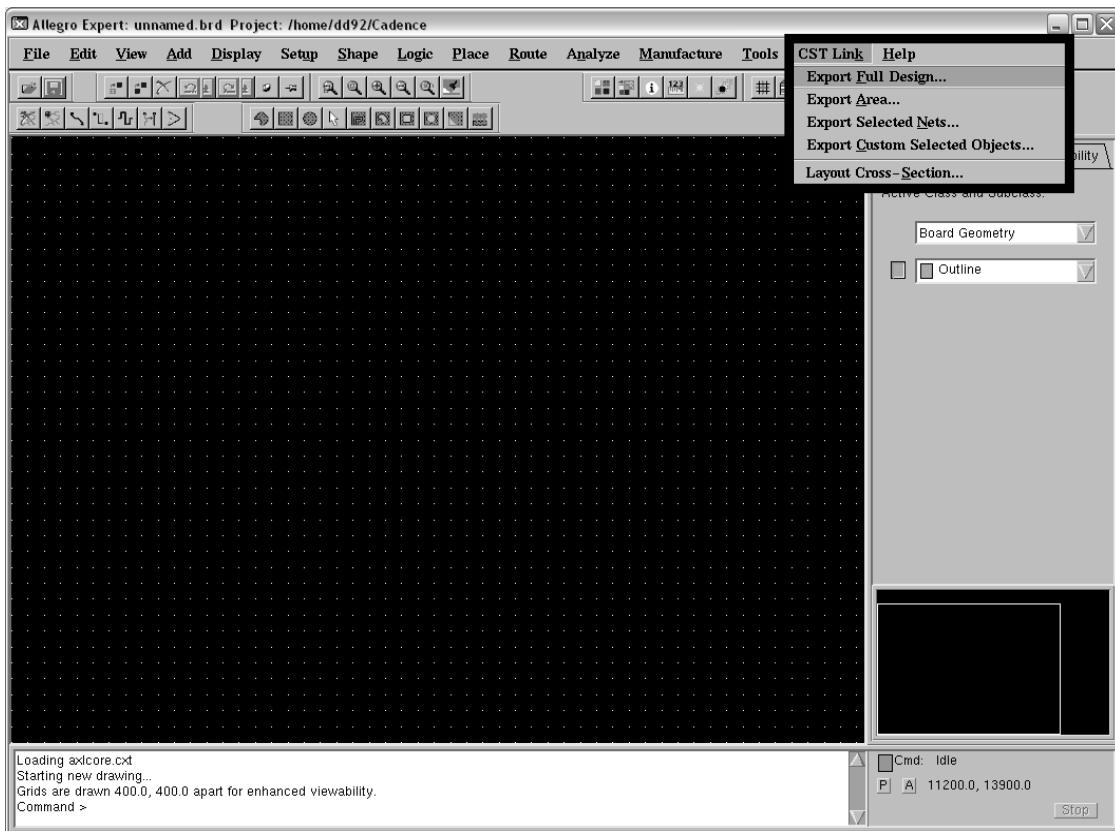
Please note: Use forward slashes for the path specification only, even on Windows operating systems, e.g. use C:/Allegro/CST_Link rather than C:Allegro\CST_Link.

After starting Cadence® Allegro or APD, select the item *File*⇒*Script* from the main menu to open a dialog box similar to the following (depending on the actual version of the Cadence® tool):



Now browse for the file “install.scr” which is contained within the plug in’s “CST_Link” folder. Clicking the *Replay* button installs all required components and creates subdirectories as necessary. Once the installation is complete, the Cadence® program closes automatically.

The plug in can be used after restarting Cadence® Allegro or APD. The following screen shot shows the main window of Cadence® Allegro after successfully installing the plug in:



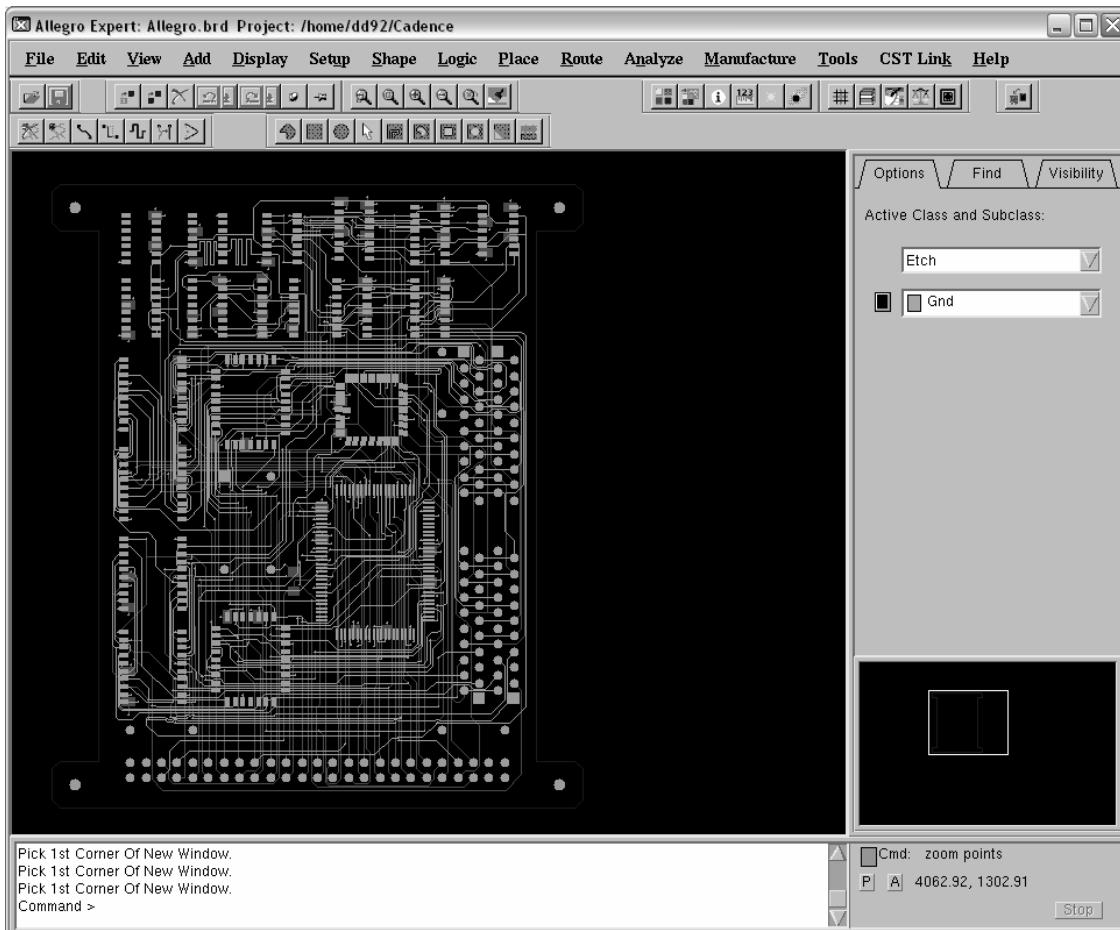
Background information: The installation program has copied the currently used menu configuration files into the “CST_Link” folder and has added the plug in specific menu settings to these files. Furthermore, customized initialization files have been copied to the “pcbenv” folder in your home directory.

The following list shows all files created during this procedure in the pcbevn directory: “allegro.ilinit”, “allegro.ini”, “apd.ilinit” and “cst.ctx”. The plug in can be uninstalled by simply deleting those files together with the entire “CST_Link” folder.

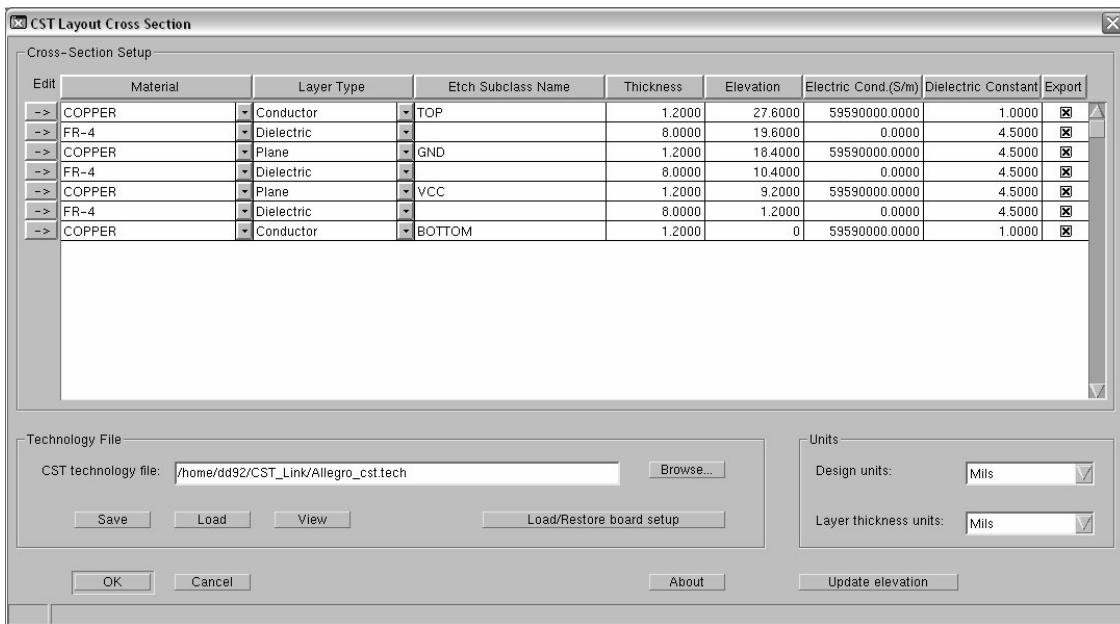
Exporting Models from Cadence® Allegro / APD

We will demonstrate the usage of the plug in for the case of exporting a Cadence® Allegro model. The procedure for exporting Cadence® APD models is similar, and the explanations in this section can be applied in a straightforward manner.

In the following we assume that a Cadence® Allegro model is loaded and the plug in is installed properly:



First, make sure that the layout cross-section information is correctly stored in the technology information by choosing *CST Link*⇒*Layout Cross-Section*:



This dialog box allows you to modify the current stackup (= cross section) information as well as save and load the modified data to an external *CST Technology File*. Clicking the *Load/Restore board setup* button resets the stackup information to the data stored in the board's technology file.

A context menu can be opened by pressing the right mouse button on a layer's “->” button. This menu allows deleting the corresponding layer or adding a new layer above or below the current one. The layer's properties can be changed by directly editing the corresponding entry fields similar to the procedure in Cadence® Allegro.

Unchecking the *Export* option hides the corresponding layer from the export procedure. Please note that this also causes the layer's thickness not to be considered when calculating the stackup's elevations.

After changing the thickness of a layer, click the *Update elevations* button to recalculate the elevation values for all layers. Otherwise, the stackup may contain gaps between adjacent layers and overlapping layers, respectively.

The settings can be confirmed by clicking *OK* which also closes the dialog box.

Once the stackup information is verified, you can choose one the following options to export the model:

- Export Full Design
- Export Area
- Export Selected Nets
- Export Custom Selected Objects

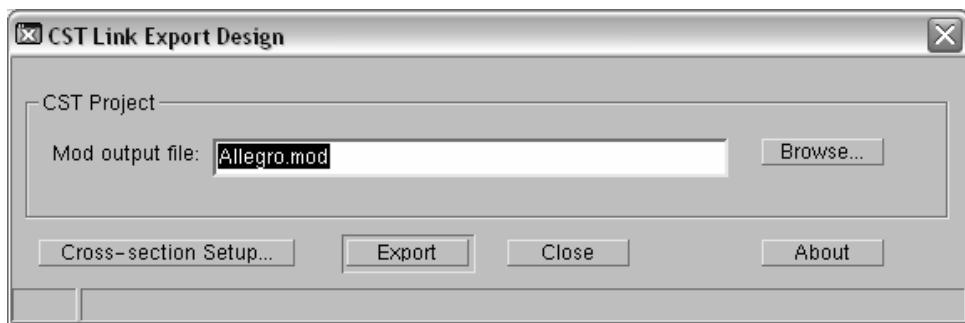
The following sections discuss the usage of each of these options in more detail. The *Export Area* option is probably the most frequently used choice since it combines ease of use with good performance even when working with large models.

For each of these options, two files with the extensions “.mod” and “.par” will be created by the plug in. These files need to be transferred to a location that is directly accessible through CST MICROWAVE STUDIO®. Make sure to activate the *text* mode when using FTP to transfer those files.

The model files contain an intermediate file format after exporting them using the plug in. However, the files can be opened as usual by CST MICROWAVE STUDIO®. The files will then be automatically converted and saved as standard model files. This conversion process requires a Cadence® link license which may not be available to you due to license restrictions. Contact your sales office for more information.

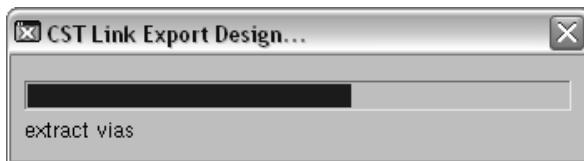
Export Full Design

The easiest way to export the current model is to select the *Export Full Design* option which will open the following dialog box:

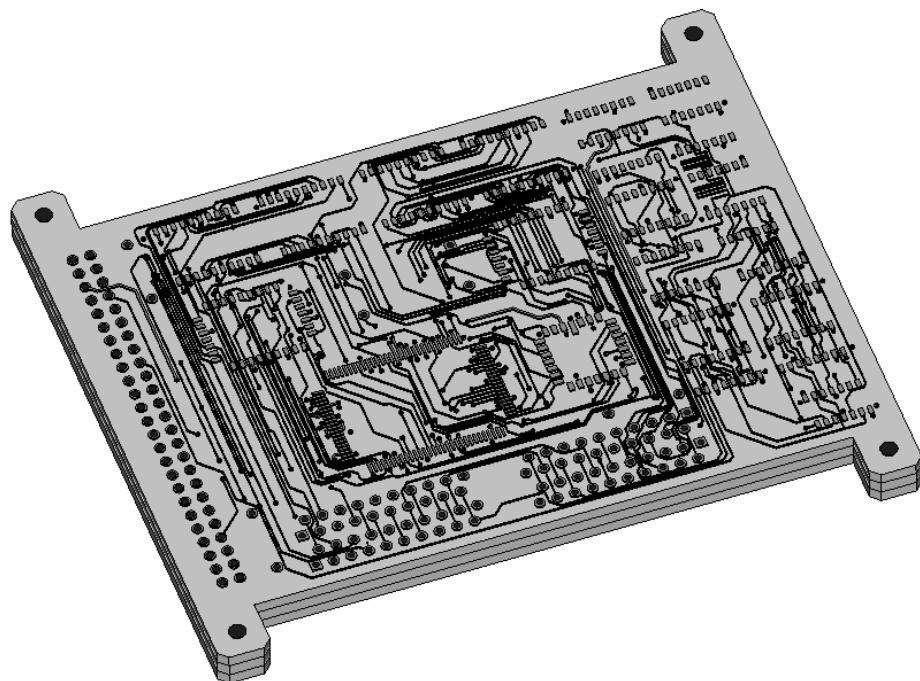


Select a name for the model file (.mod file). As an optional step, you can also open the stackup editor as explained above by clicking the *Cross-section Setup* button.

The actual export procedure can be started by clicking the *Export* button. A progress window will appear keeping you informed of the status of the export process:



The following screen shot shows the entire Cadence® Allegro model after exporting it and opening the corresponding files in CST MICROWAVE STUDIO®:



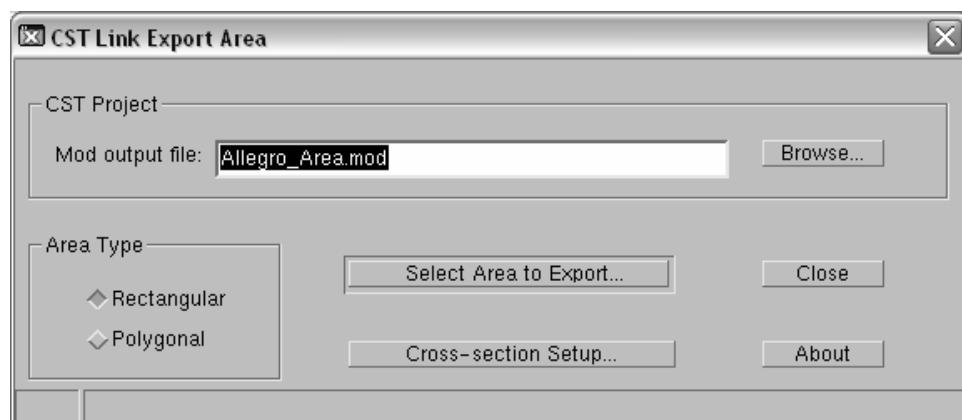
While the simplicity of using this export option clearly is an advantage, its disadvantage is the huge complexity of the exported models. Keep in mind that you are building a 3D CAD model which is usually much more complex than the 2D data in Cadence® Allegro.

Boards having hundreds or thousands of components and traces can easily end up in huge models which can no longer be handled and simulated efficiently. The following export options help to reduce the complexity of the exported models while still transferring all relevant data into the simulation model.

Export Area

The *Export Area* option is relatively simple to use but reduces the complexity of the exported model significantly. In many cases, it is sufficient to simulate only a small part of the entire board using CST MICROWAVE STUDIO®. The *Export Area* selection allows selecting a rectangular or polygonal area whose contents will then be solely exported.

After you select the corresponding option from the menu, the following dialog box will appear:



First specify a name for the exported “.mod” file. You may also open the stackup editor from within this dialog box by clicking the *Cross-section Setup* button.

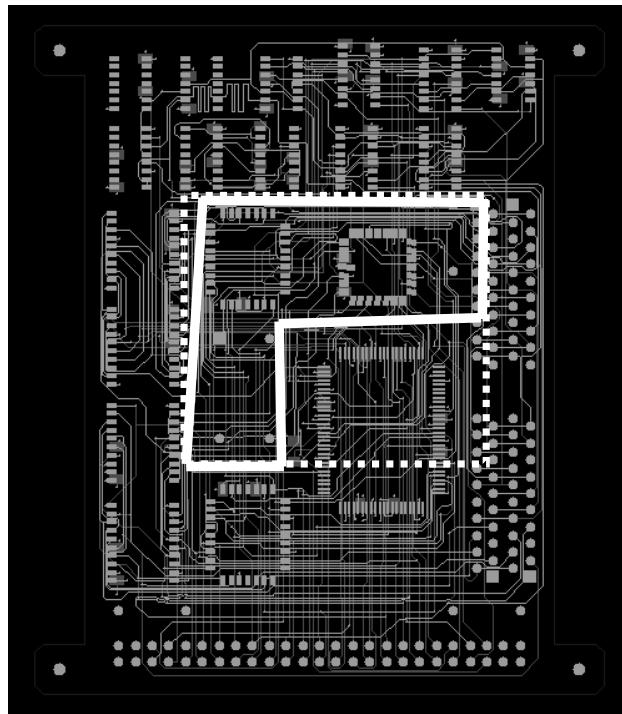
Next select whether you are going to use a *Rectangular* area or the more general case of a *Polygonal* area.

Selecting a rectangular area requires two mouse clicks for the opposite corners only whereas the definition of a polygonal selection requires the definition of all corner points.

Start the actual interactive definition of the area by clicking the *Select Area to Export* button.

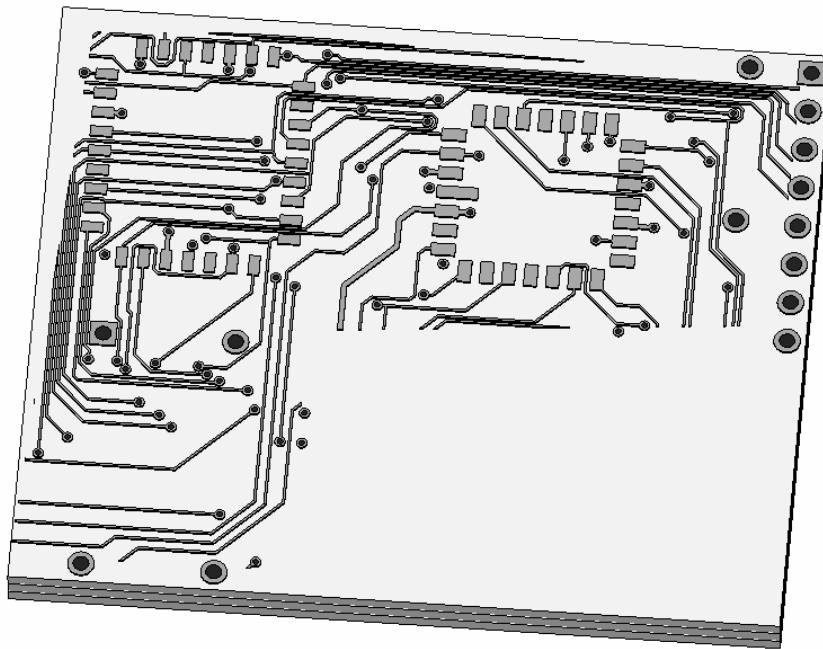
Once this button is clicked, the dialog box disappears and you can start selecting points by single-clicking on the corresponding location on the screen. Pressing the right mouse button opens a context menu which allows you to abort the operation by selecting the *Cancel* item. Choosing the *Oops* entry deletes the previously selected point whereas the *Done* option automatically closes the rectangle or polygon and starts the export process.

In the example above, we select the following polygon (solid white line):



Due to the brick-like shape of the computation domain, the dashed area of the board will be exported rather than the polygonal selection. However, all trace and component geometry will still be clipped at the boundaries of the original polygon.

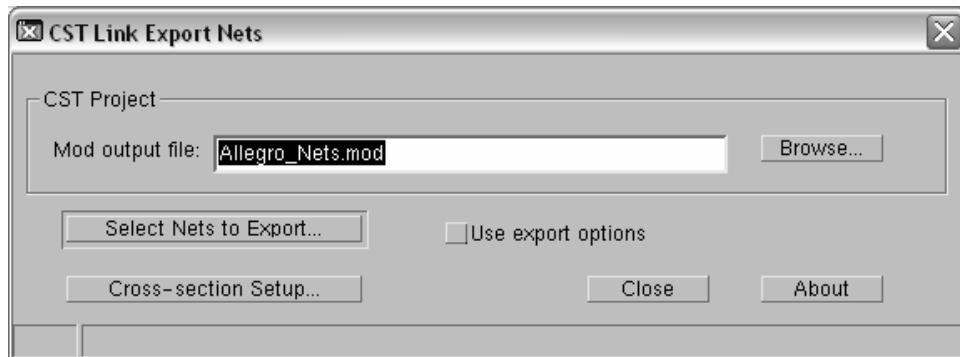
A progress window will be shown during the export process. Opening the exported model in CST MICROWAVE STUDIO® will then reveal the following:



Export Selected Nets

A typical application of EM simulation in the context of board design is analyzing the S-parameters for a set of coupled nets. This mainly requires exporting the geometry of the selected nets together with some neighboring geometry often including GND and Vcc planes. Even if this can be easily achieved using the *Export Area* option described above, the complexity of the exported model may still be too large for some models.

In such cases, the *Export Selected Nets* option provides an alternative solution. The corresponding dialog box looks as follows:



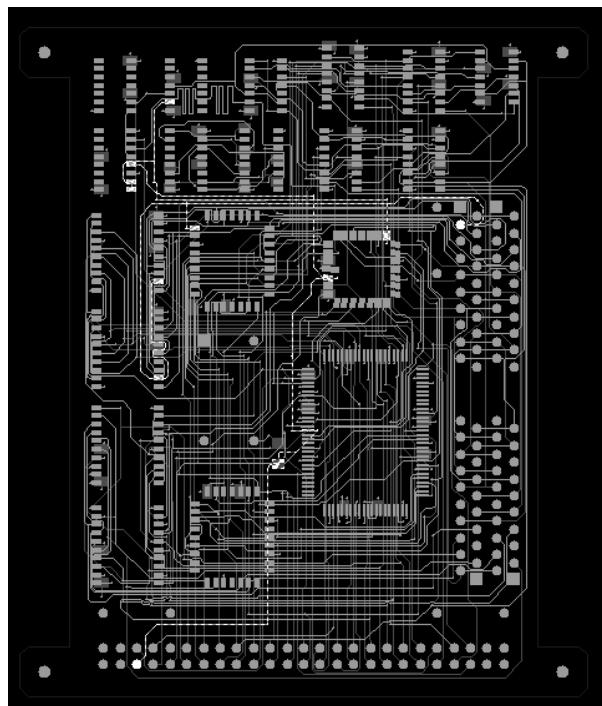
First specify a name for the exported “.mod” file. You can optionally enter the stackup editor from within this dialog box by clicking the *Cross-section Setup* button.

If the *Use export options* button is unchecked, only the selected nets will be exported without any neighboring geometry. If the option is checked, another dialog box will open later on containing parameters to include neighboring geometry.

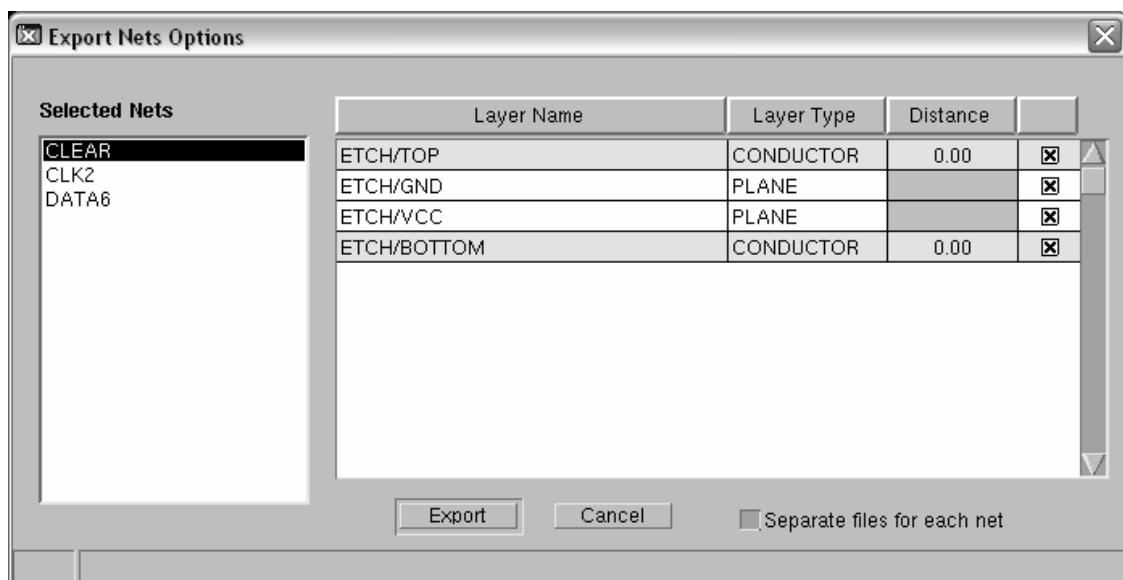
You can start selecting the nets after clicking the *Select Nets to Export* button. The dialog box will be hidden during the interactive selection procedure. You can open a context menu by pressing the right mouse button.

A net can be selected by clicking on it in the main view or by typing its name after choosing *Select By Name* from the context menu. The selection can be aborted by choosing *Cancel* from the context menu. The previously selected net can be deselected again by choosing the *Oops* options from the context menu. Choosing the *Done* item will end the selection and continue with the next step.

The following picture shows the selection of three nets for the example above:



If the *Use export options* button is checked, another dialog box opens after the net selection has finished:



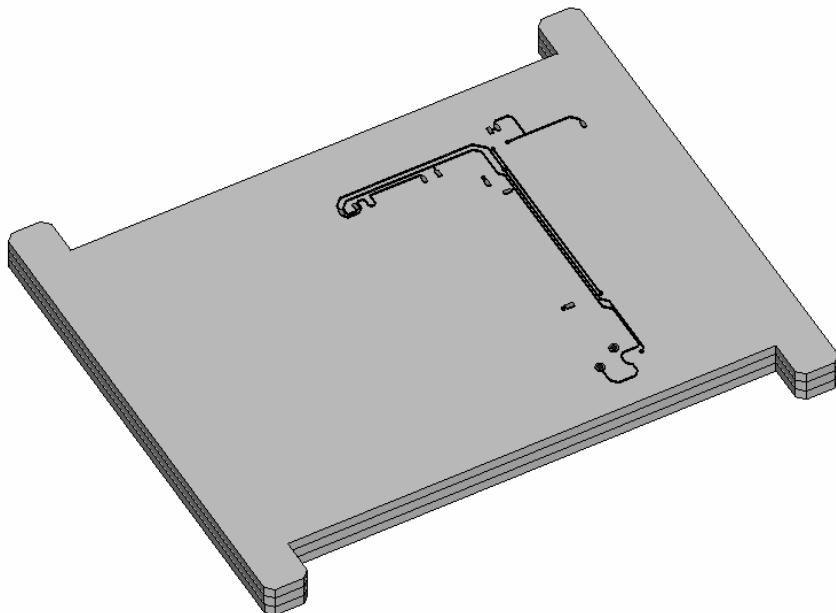
The dialog box shows a list of all previously selected nets. Clicking on a net's name shows a list of all layers that contain parts of the net. This also includes planes that are crossed by the net through vias.

For each of the conducting layers, a *Distance* can be specified. Neighboring geometry will then be exported as well if it is located within the specified distance to the particular net.

The option *Separate files for each net* allows you to specify whether all nets are exported into a single model or whether separate models need to be created for each net. Cross-talk simulations require all nets to be located in the same model, which means that the option needs to be unchecked in this case.

Click the *Export* button to start the model generation.

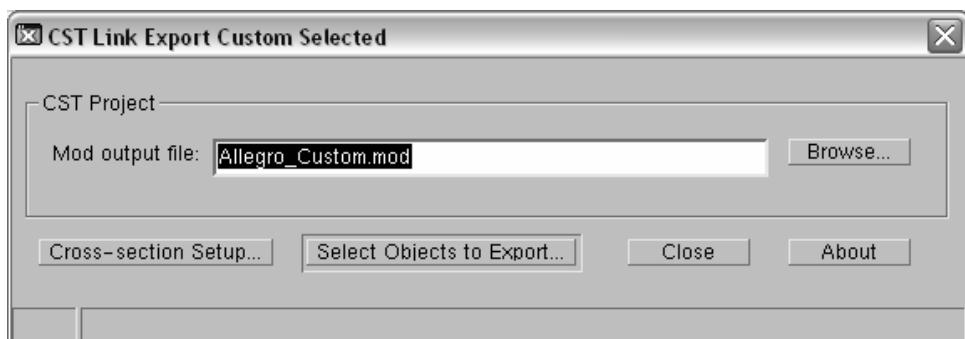
For the example above, we have chosen a small distance around each of the nets and put everything into a single model file:



Export Custom Selected Objects

The *Export Custom Selected Objects* function is one of the most flexible ways to select objects for exporting but also requires the most laborious entity selection.

After choosing this option from the main menu, the following dialog box appears:



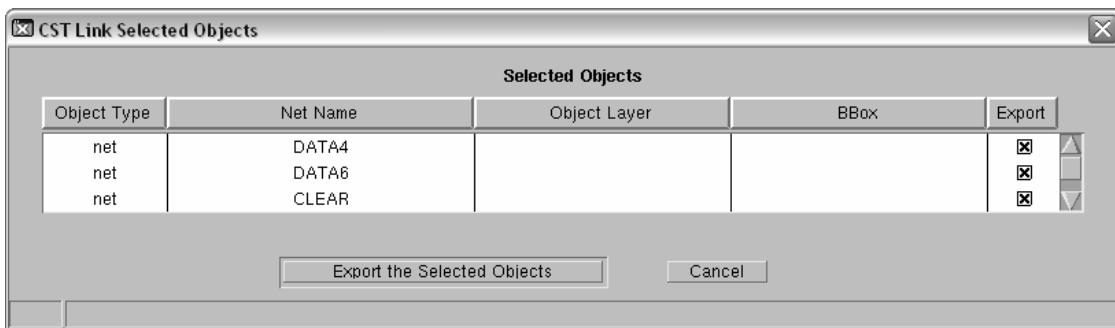
Specify a name for the exported “.mod” file. You can optionally enter the stackup editor from within this dialog box by clicking the *Cross-section Setup* button.

Start the actual entity selection by clicking *Select Objects to Export*. The dialog box will be hidden during the selection process.

You can open a context menu by pressing the right mouse button. From this context menu, you can choose the type of the entity that will be selected by the next mouse click. Available types are the following:

- Net
- Shape
- Cline
- Via
- Pin

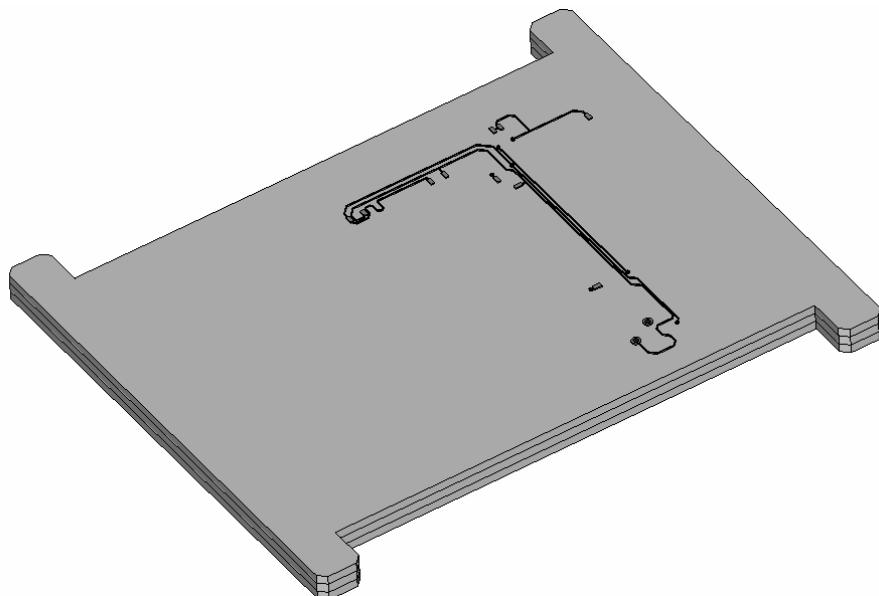
In addition, you can also select a net by specifying its name. End the selection process by choosing either *Cancel* or *Done*. The latter choice will show a summary of all selected entities:



You may now exclude an entity from the export process by unchecking its corresponding *Export* button.

Start the actual export process by clicking the *Export the Selected Objects* button. A progress window will be displayed until the export is complete.

For the example above, the exported model looks as follows:



Note that the selection applies to the metal parts only and the dielectric layers will always be exported. The shape of the dielectric layers is determined by the board's outlines.

Agilent ADS® Co-Simulation

In addition to its tight integration into the CST DESIGN ENVIRONMENT™, CST MICROWAVE STUDIO® features strong interfaces with Agilent ADS®. The latter ranges from using pre-computed S-parameter data in ADS® circuit simulations to using true EM / circuit co-simulations.

CST MICROWAVE STUDIO® manages an ADS® Design Kit which contains a dynamically extended list of EM simulation components. The difference between the levels of integration is how simulation results are handled for a particular component. The integration supports changing parameters of the 3D EM models directly from within the ADS® environment.

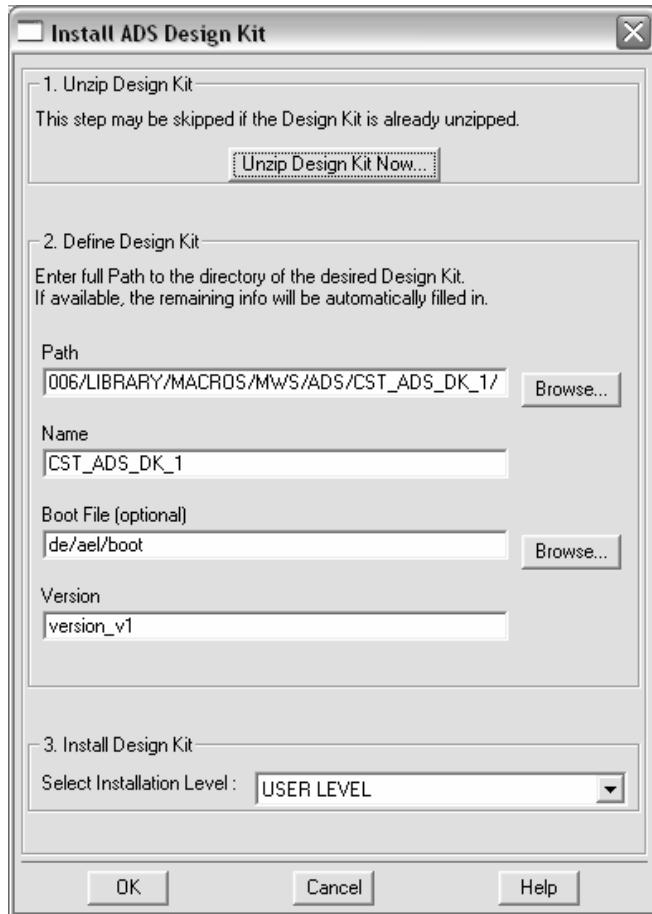
The “static” link (ADS® 2003A and above) requires parametric CST MICROWAVE STUDIO® results to be computed before the component is used within ADS®. The circuit simulator automatically interpolates the S-parameter between the pre-computed samples.

Moreover, the “co-simulation” alternative enables ADS® to launch CST MICROWAVE STUDIO® in order to automatically calculate required data. All information is stored together with the CST MICROWAVE STUDIO® model to avoid unnecessary repetitions of lengthy EM simulations. The co-simulation interface requires ADS® 2005A or above.

In the following sections, we focus on the common Design Kit installation procedure before explaining the usage of both interfaces in more detail.

Design Kit Installation

The Design Kit is required for the static link as well as the co-simulation interface. For the installation of the Design Kit, you can select *Design Kit*⇒*Install Design Kits* from the ADS® main menu to open the following dialog box:



Clicking the *Browse* button next to the *Path* entry allows you to specify the path to the Design Kit which is located in a subfolder of the CST DESIGN ENVIRONMENT™ library folder. Refer to the *CST DESIGN ENVIRONMENT™ First Steps* manual for more information about setting the library folder. By default, the library is located in a subfolder of the CST STUDIO SUITE™ installation directory.

If [library_path] is the actual library folder (e.g. "C:\Program Files\CST STUDIO SUITE 2006\Library"), the Design Kit is stored at the following location:

[library_path]\Macros\MWS\ADS\CST_ADS_DK_1

Make sure to browse for this folder as the Design Kit's path. Once the path is set properly, the remaining fields in the dialog box will be filled in automatically as shown in the picture above.

The Design Kit can now be installed by clicking the *OK* button. Refer to the ADS® documentation for more information on managing Design Kits.

Static Link between CST MICROWAVE STUDIO® and Agilent ADS®

The static link can be used right after installing the Design Kit. In the following, we explain the necessary steps to make a particular project available as a component in ADS®.

After opening the corresponding CST MICROWAVE STUDIO® project, you need to add a 1D result template named *ADS Component Export*. Now all parametric S-parameters which are computed after adding the template will be stored in a local database. Please note that performing model changes other than simple parametric changes will automatically delete this database.

Once the result template is defined, you can run simulations for various parameter settings. All entries in the S-parameter database will then become available for use within ADS®.

Note: If you already have S-parameter data available before adding the result template, you can click the *Evaluate* button in the template based post-processing dialog box in order to add the existing S-parameters to the data set right away. However, only the latest S-parameter results will be added.

After computing all necessary S-parameters, you can create the ADS® Design Kit component by choosing *File*⇒*Export*⇒*ADS Component Export* from the main menu:



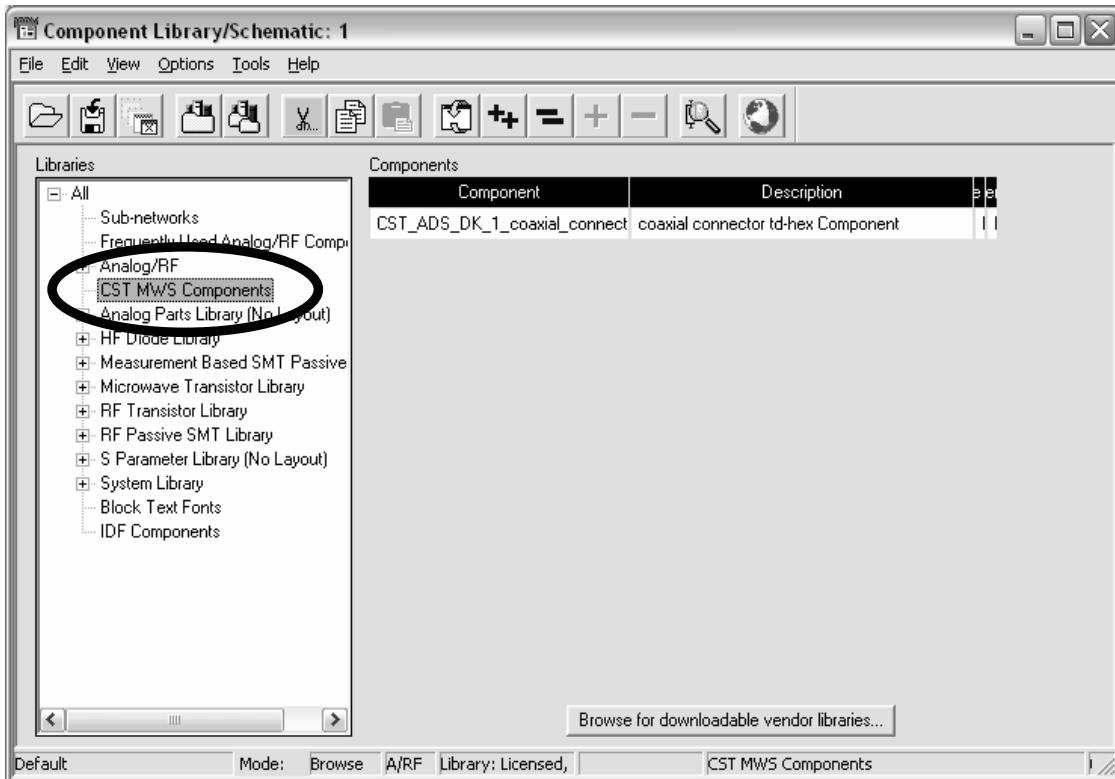
The parameter list shows all parameters that have been varied in the currently available database. Please note that parameters depending on other parameters in form of expressions are not listed here.

Each parameter needs to have a *Type* assigned to it in order to properly handle potentially different unit settings in CST MICROWAVE STUDIO® and ADS®. Make sure to specify the correct type according to the physical meaning of the parameter. In the example shown above, the “radius” parameter is of type *Length*.

Now double-check the types for every parameter, and enter a proper *Component Description*.

Clicking *OK* will create the component and copy the currently available S-parameter data to the Design Kit. Subsequently performed S-parameter simulations will not automatically add new data to the Design Kit.

You need to restart ADS® in order to use the newly created component. The component can then be accessed as usual from the ADS® library browser as shown below:

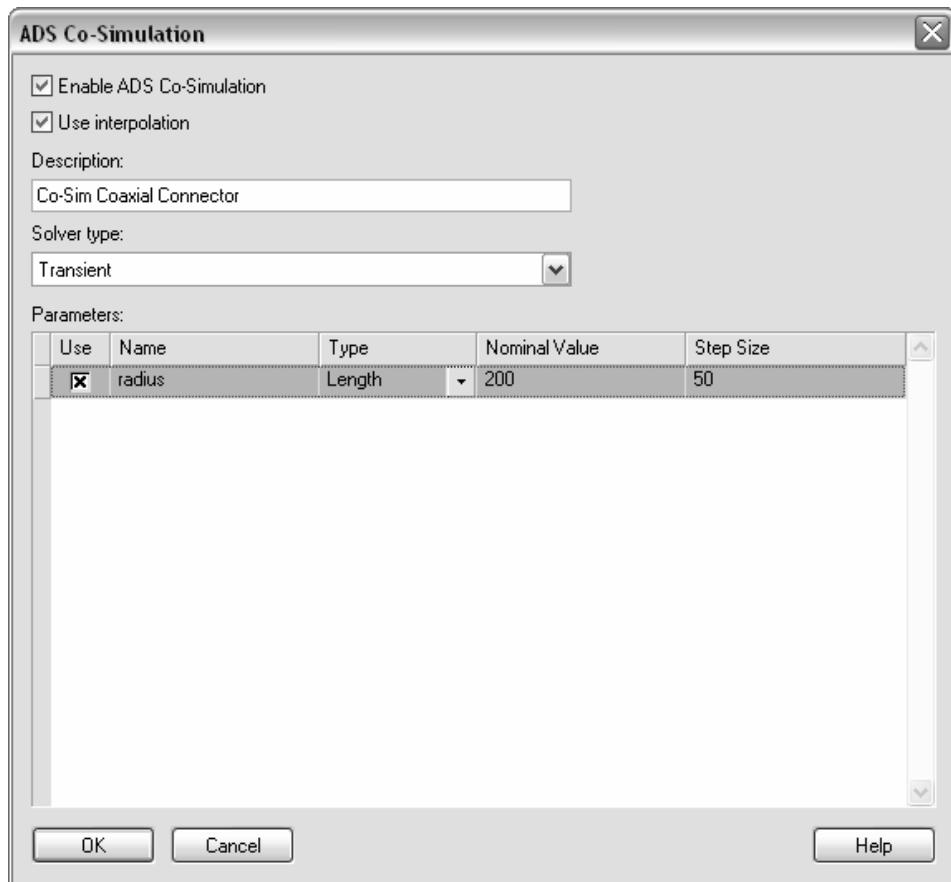


CST MICROWAVE STUDIO® and Agilent ADS® Co-Simulation

A unique feature of the co-simulation link is the ability to automatically launch CST MICROWAVE STUDIO® from within ADS® if new S-parameter data are requested. This advanced interface requires ADS® 2005A or above and needs CST MICROWAVE STUDIO® and ADS® to be installed on the same computer.

Before the co-simulation can actually be used, you need to copy a DLL named "gem_CstCmptDLL29.dll" from the *Agilent ADS Plug in* folder in the installation directory of CST STUDIO SUITE™ to the ADS® *bin* directory. The latter directory is located in the main installation folder of ADS® and already contains a placeholder DLL with the same name. Make sure to override the existing DLL with the new one.

Once the DLL has been installed properly, the actual usage of the co-simulation is simple. It can be activated for a particular model by choosing *Solve*⇒*ADS Co-Simulation* from CST MICROWAVE STUDIO®'s main menu:



First, activate *Enable ADS Co-Simulation* by checking the corresponding option. You can also specify a *Description* text which will later be shown in the ADS® library browser.

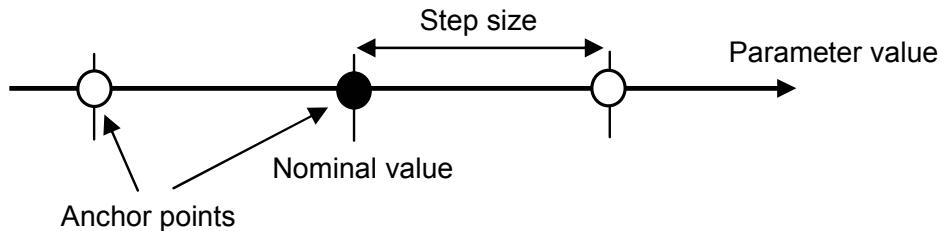
Whenever the calculation of new S-parameter data is required, the co-simulation interface will automatically launch CST MICROWAVE STUDIO®. The *Solver type* setting specifies which solver (transient or frequency domain) will then be used for the calculation.

The parameter list shows currently defined parameters that can be used for the co-simulation. If a parameter depends on another parameter (e.g. in form of an expression), its value cannot be set directly. Therefore, dependent parameters are not shown in this list.

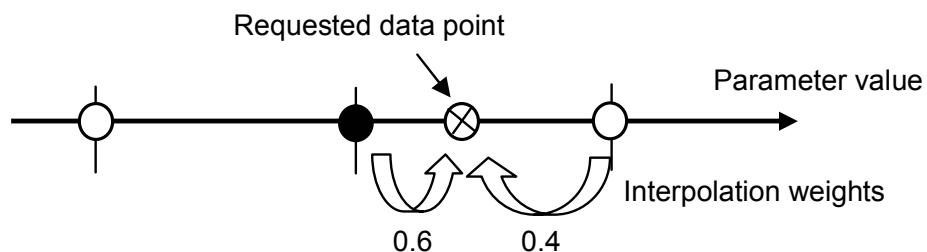
A parameter can be controlled from within ADS® if its *Use* option is checked. This selection allows you to hide parameters from being visible in the ADS® component options.

The *Use interpolation* option specifies whether S-parameter data may be interpolated or not. If this option is switched off, the S-parameters will be directly calculated for every requested parameter combination. If the interpolation feature is activated, the interpolation is done on a regular grid of anchor points. The grid is determined by the *Nominal Value* and *Step Size* settings which can be independently specified for each parameter.

The following picture illustrates how the interpolation scheme works for the simplest case of one parameter:



CST MICROWAVE STUDIO® simulations are only carried out at the uniformly spaced anchor points of the interpolation grid. The grid's origin is specified by the *Nominal value* whereas the *Step Size* determines the distance between two consecutive anchor points.

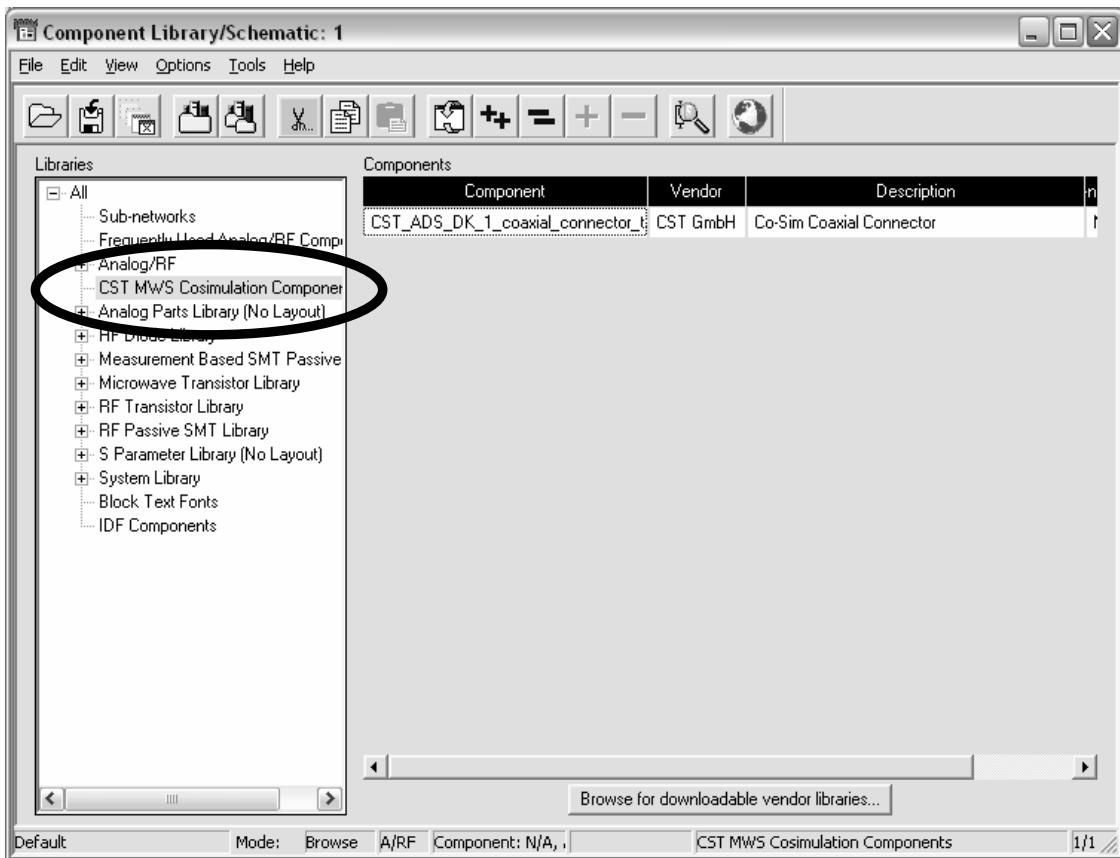


Whenever a new data point is requested, the two closest neighboring anchor points are determined and their contributions to the requested sample are calculated. Based on the corresponding anchor point's S-parameter data and these interpolation weights, the requested S-parameters can then be obtained by linear interpolation. This scheme can be straightforwardly extended to handle multiple parameters as well.

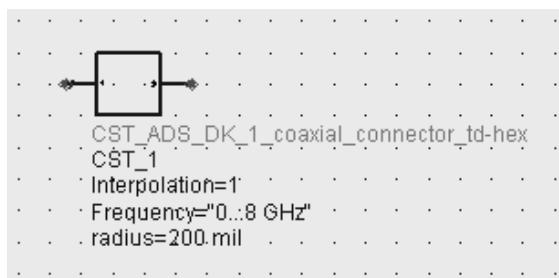
In the dialog box, you can specify each parameter's interpolation options by directly editing the values in the parameter list. Please note that these settings will only be used when the interpolation option is switched on.

Clicking the *OK* button finally creates or updates the Design Kit component. Make sure to restart ADS® in order to update these changes.

The components can now be accessed as usual from the ADS® library browser:



The following picture shows a co-simulation component after it has been placed on the ADS® schematic:



The component's options allow changing the interpolation type (1=yes, 0=no) and CST MICROWAVE STUDIO® model parameters directly from within the ADS® environment. Furthermore the component displays information about the frequency range of the EM simulation.

The Design Kit component is now dynamically linked to the CST MICROWAVE STUDIO® model. Whenever new S-parameter data for a particular parameter combination are computed, these data will become available to ADS® right away.

Note: Whenever a parametric model is used for ADS® co-simulation, it is advisable to test whether the structure update succeeds for various parameter combinations before creating the ADS® Design Kit component.

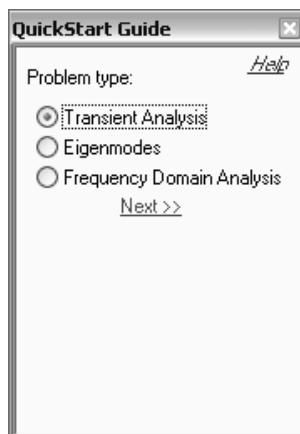
Chapter 3 — Finding Further Information

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 give you a short overview of the available documentation.

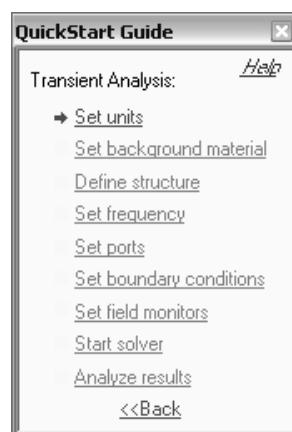
The Quick Start Guide

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, click *Next* 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 Guide* from the menu bar.

In order to access information about the Quick Start Guide itself, click the *Help* button. To obtain more information about a particular operation, 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 you browse through the list of all available tutorials and choose the one closest to your application:

Rectangular Waveguide

This is for typical waveguide structures with rectangular waveguide ports. The S-parameter simulation is performed using both the transient simulator and the general purpose tetrahedral frequency domain simulator.

Coaxial Structure

This is an S-parameter example for a coaxial structure using the transient simulator and the general purpose tetrahedral frequency domain simulator.

Planar Device

This is a common S-parameter example for microstrip and coplanar structures using the transient simulator and the general purpose tetrahedral frequency domain simulator.

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 using the transient simulator and the tetrahedral frequency domain simulator.

Resonator

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

Filter

This example shows an S-parameter simulation for a narrow band filter using the resonant fast S-parameter frequency domain solver. This highly specialized solver is the best choice for this type of application. However, the example also shows the application of the general purpose tetrahedral frequency domain solver as an alternative.

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. .

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 can quickly go through the pages of the tutorial and pick out new information.

Examples

The installation directory of CST STUDIO SUITE™ contains an examples subdirectory consisting of a couple of typical application examples. A quick overview of the existing examples can be obtained by choosing *Help⇒Examples Overview* from the menu bar.

Each of these examples also contains a “Readme” item in the navigation tree. By double-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⇒CST MICROWAVE STUDIO Help* from the menu bar.

In each of the dialog boxes, there is a specific *Help* button which directly opens 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 can get information about the definition of shapes and possible actions.

When 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 more background information about the methods used in CST MICROWAVE STUDIO® together with some more detailed information about advanced topics including the following:

- An introduction into the simulation methods
- Advanced mesh generation
- How to improve the calculation speed
- Introduction to the powerful macro language to automate common tasks
- List of available post processing templates and macros

Technical Support

After you have taken your first steps to solving your own applications within CST MICROWAVE STUDIO®, please use the macro *Macros*⇒*File*⇒*Make Archive* to create a ZIP archive containing all relevant files. This ZIP file should then be sent 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.

The support area on our homepage (www.cst.com) also contains a lot of very useful and frequently updated information. Simple access to this area is provided by choosing *Help*⇒*Online Support*. You only need to enter your user name and password once. Afterwards, the support area will open automatically whenever you choose this menu command.

Macro Language Documentation

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

- An overview and general description of the macro language
- A description of all CST MICROWAVE STUDIO® specific macro language extensions
- A syntax reference of the Visual Basic for Applications compatible macro language
- Some documented macro examples

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 that may be very useful, especially for advanced users.

General Shortcut Keys Available in Main Structure View

You can activate this view 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	Open context sensitive help
F2	Rename the currently selected shape in the navigation tree
F5	Update the visualized 1D result (only while a 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	Copy the currently displayed result curves into clipboard
Ctrl+V	Paste result curves from clipboard into the active result curve plot
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
Shift+C	Activate/deactivate cutting plane view
x	If the cutting plane view is active, the cut is made on the x-plane
y	If the cutting plane view is active, the cut is made on the y-plane
z	If the cutting plane view is active, the cut is made on the z-plane

TAB	Open the numerical coordinate input box (also available in 1D plots for axis marker positioning)
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, increase frequency for visualization of frequency dependent port modes.
Cursor-Down	Move cutplane or meshplane in opposite normal direction, decrease frequency for visualization of frequency dependent port modes.

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
Ctrl+Shift+D	Delete the selected face
Ctrl+Shift+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)
Shift+P	Open result template post processing dialog box
Mouse Wheel	Dynamic zoom view

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

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

Shortcut Keys Available in VBA Editor

Ctrl+N	File new
Ctrl+O	File open
Ctrl+S	File save
Ctrl+P	Print
Ctrl+F	Find
F3	Find again
Ctrl+R	Replace
Ctrl+Z	Undo previous operation
Ctrl+Y	Redo previously undone operation
Ctrl+X	Cut
Ctrl+C	Copy
Ctrl+V	Paste
F1	Context help for the word next to the caret position
F5	Run macro
ESC	Pause macro
F7	Debug step to
F9	Debug break
Ctrl+F9	Add watch
Ctrl+Shift+F9	Clear all breaks
Shift+F9	Quick watch
Ctrl+F8	Debug step out
Shift+F8	Debug step over
F8	Debug step into



© CST GMBH, 2005. ALL RIGHTS RESERVED.