

CST Studio Suite

High Frequency Simulation



3DEXPERIENCE®

Workflow &
Solver Overview

Version 2020.0 - 8/16/2019

Copyright

© 1998–2019 Dassault Systèmes Deutschland GmbH
CST Studio Suite is a Dassault Systèmes product.
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 Dassault Systèmes.

Trademarks

CST, the CST logo, Cable Studio, CST BOARDCHECK, CST EM STUDIO, CST EMC STUDIO, CST MICROWAVE STUDIO, CST PARTICLE STUDIO, CST Studio Suite, EM Studio, EMC Studio, Microstripes, Microwave Studio, MPHYSICS, MWS, Particle Studio, PCB Studio, PERFECT BOUNDARY APPROXIMATION (PBA), Studio Suite, IdEM, Aurorasat, SPARK3D, FEST3D, 3DEXPERIENCE, the Compass logo and the 3DS logo, CATIA, SOLIDWORKS, ENOVIA, DELMIA, SIMULIA, GEOVIA, EXALEAD, 3D VIA, BIOVIA, NETVIBES, 3DSWYM and 3DEXCITE are commercial trademarks or registered trademarks of Dassault Systèmes, a French "société européenne" (Versailles Commercial Register # B 322 306 440), or its subsidiaries in the U. S. and/or other countries. All other trademarks are owned by their respective owners. Use of any Dassault Systèmes or its subsidiaries trademarks is subject to their express written approval.

DS Offerings and services names may be trademarks or service marks of Dassault Systèmes or its subsidiaries.

3DS.com/SIMULIA

Table of contents

Chapter 1 – Introduction.....	5
Welcome	5
How to Get Started Quickly.....	5
CST Studio Suite for High Frequency Simulation	5
Who Uses CST Studio Suite for High Frequency Simulation?	7
Key Features for High Frequency Simulation	7
General.....	7
Structure Modeling	7
Transient Solver	8
TLM Solver	9
Frequency Domain Solver.....	10
Integral Equation Solver.....	12
Multilayer Solver.....	13
Asymptotic Solver	13
Eigenmode Solver	14
Partial RLC Solver.....	14
Schematic View.....	14
SAM (System Assembly and Modeling).....	15
Visualization and Secondary Result Calculation.....	15
Result Export.....	15
Automation	15
About This Manual	16
Document Conventions.....	16
Your Feedback.....	16
Chapter 2 – Simulation Workflow.....	17
Setup the Simulation Model	17
The Structure.....	17
Create a New Project	18
Open the QuickStart Guide	19
Define the Units.....	20
Define the Background Material.....	20
Model the Structure	20
Define the Frequency Range	26
Define Ports.....	27
Define Boundary and Symmetry Conditions	28
Visualize the Mesh	30
Start the Simulation	31
Analyze the Port Modes	34
Analyze the S-Parameters	35
Adaptive Mesh Refinement	37
Analyze the Electromagnetic Field at Various Frequencies	40

Parameterization of the Model	42
Parameter Sweeps and Processing of Parametric Result Data	47
Automatic Optimization of the Structure	52
Comparison of Time and Frequency Domain Solver Results	55
Summary	60
Chapter 3 – Solver Overview	61
Which Solver to Use.....	61
Time Domain Solver.....	64
Transient Solver	64
TLM Solver	64
Frequency Domain Solver.....	67
Integral Equation Solver	77
Multilayer Solver.....	79
Asymptotic Solver	82
Eigenmode Solver.....	86
Partial RLC Solver.....	93
Choosing the Right Port Type	93
Antenna Computations.....	94
Simplifying Antenna Farfield Calculations	98
Sensitivity Analysis.....	99
Digital-Signal Calculations	101
Coupled Simulations	103
Adding Circuit Elements to External Ports	103
Coupled Simulations between High Frequency Solvers	104
Coupled Simulations with Thermal or Mechanical Solvers	105
Coupled Workflow with Cable Simulation	105
Acceleration Features	105
Chapter 4 – Finding Further Information.....	107
The QuickStart Guide.....	107
Online Documentation.....	107
Tutorials and Examples.....	108
Technical Support	108
Macro Language Documentation	108
History of Changes.....	108

Chapter 1 – Introduction

Welcome

Welcome to CST Studio Suite®, the powerful and easy-to-use electromagnetic field simulation software. This program combines a user-friendly interface with unsurpassed simulation performance. CST Studio Suite contains a large variety of solvers for carrying out *High Frequency Simulations*. They are all grouped as a specific *High Frequency Module*, also known as CST Microwave Studio®.

Please refer to the *CST Studio Suite - Getting Started* manual first. The following explanations assume that you have 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 Studio Suite - Getting Started manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. Look at the examples provided in the Component Library (File: Component Library □ Examples). Especially the examples which are tagged as Tutorial provide detailed information of a specific simulation workflow. Press the Help ? button of the individual component to get to the help page of this component. Please note that all 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 basic concepts.
4. Start with your own first example. Choose a reasonably simple example, which will allow you to quickly become familiar with the software.
5. After you have worked through your first example, contact technical support for hints on possible improvements to achieve even more efficient usage of the software.

CST Studio Suite for High Frequency Simulation

CST Studio Suite for High Frequency Simulation is a fully featured software package for electromagnetic analysis and design in the high frequency range. It simplifies the process of creating the structure by providing a powerful graphical solid modeling front end which is based on the ACIS modeling kernel. After the model has been constructed, a fully automatic meshing procedure is applied before a simulation engine is started. An advanced visualization engine and flexible post-processing allow you to analyze and improve your design in a relevant and efficient way.

A key feature of CST Studio Suite is the *Complete Technology* approach which gives the choice of simulator or mesh type that is best suited to a particular problem, seamlessly integrated into one user interface.

Since no one method works equally well for all applications, the software contains several different simulation techniques (time domain solvers, frequency domain solvers, integral equation solver, multilayer solver, asymptotic solver, and eigenmode solver) to best suit various applications.

Each method in turn supports meshing types best suited for its simulation technique. Hexahedral meshes are available in combination either with the Perfect Boundary Approximation (PBA)® feature, and for some solvers additionally with the Thin Sheet Technique (TST)™ extension, or with a powerful octree-based meshing algorithm which efficiently reduces the overall cell count. Applying these advanced techniques usually increases the accuracy of the simulation substantially in comparison to simulation techniques which employ a conventional hexahedral mesh.

In addition to the hexahedral mesh the frequency domain and eigenmode solvers also support linear and curved tetrahedral meshes. Furthermore, linear and curved surface and multilayer meshes are available for the integral equation and multilayer solver, respectively.

The largest simulation flexibility is offered by the **time domain solvers**, which can obtain the entire broadband frequency behavior of the simulated device from a single calculation run. These solvers are remarkably efficient for many high frequency applications such as connectors, transmission lines, waveguide components, and antennas, amongst others.

Two time domain solvers are available, both using a hexahedral mesh, either based on the Finite Integration Technique (FIT) or on the Transmission-Line Matrix (TLM) method. The latter is especially well suited to EMC/EMI/E3 applications.

The time domain solvers are less efficient for structures that are electrically much smaller than the shortest wavelength of interest. In such cases it may be 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 use of an unstructured tetrahedral mesh is advantageous to resolve very small geometric details. Besides the general purpose broadband frequency sweep, the frequency domain solver also contains alternatives using fast reduced order model techniques to efficiently generate broadband results. The frequency domain solver supports hexahedral as well as tetrahedral meshes.

For electrically large structures, volumetric discretization methods generally suffer from dispersion effects and thus require a very fine mesh. CST Studio Suite therefore contains an **integral equation based solver** which is particularly suited to solving this kind of problem. The integral equation solver uses a curved triangular and quadrilateral surface mesh which becomes very efficient for electrically large structures. The Multilevel Fast Multipole Method (MLFMM) solver technology ensures an excellent scaling of solver time and memory requirements with increasing frequency. For lower frequencies where the MLFMM is not as efficient, direct and iterative Method of Moments solvers are available.

The systematic design of antennas of different shapes can be greatly facilitated by means of a characteristic mode analysis (CMA) by providing physical insight into the behavior of a conducting surface. The CMA-tool, which is built into the integral equation solver and the multilayer solver, automates the process of calculating and analyzing such characteristic modes.

Despite its excellent scalability, even the MLFMM solver may become inefficient for electrically extremely large structures. Such very high frequency problems are best solved in CST Studio Suite by using the **asymptotic solver** which is based on the so-called ray-tracing technique.

For structures which are mainly planar, such as microstrip filters or printed circuit boards, this particular property can be exploited in order to gain efficiency. The **multilayer solver**, based on the Method of Moments, does not require discretization of the transversally infinite dielectric and metal stackup. Therefore, the solver can be more efficient than general purpose 3D solvers for this specific type of application. Moreover, the solver is also capable of performing CMA.

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 Studio Suite also features an **eigenmode solver**, available either on hexahedral or tetrahedral meshes, which efficiently calculates a finite number of modes in closed electromagnetic devices.

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

Each solver's simulation results can be visualized with a variety of different options. A strongly interactive interface will help you to quickly achieve the desired insight into your device.

The last – but certainly not least – of the outstanding features is the full parameterization of the structure modeler, which enables the use of variables in the definition of all geometric and material properties of your component. In combination with the built-in optimizer and parameter sweep tools, CST Studio Suite is capable of both the analysis and design of electromagnetic devices.

Who Uses CST Studio Suite for High Frequency Simulation?

Anyone who has to deal with electromagnetic problems in the high frequency range should use CST Studio Suite. The program is especially suited to the fast, efficient analysis and design of components like single and multi-element antennas (including phased arrays), filters, transmission lines, couplers, connectors (single and multiple pin), printed circuit boards, resonators, optical devices, and many more. Due to the various independent solver strategies CST Studio Suite can solve virtually any high frequency field problem.

Key Features for High Frequency Simulation

The following list gives you an overview of the main features of CST Studio Suite for High Frequency Simulations. Note that not all of these features may be available to you because of license restrictions. Please contact a sales office for more information.

General

- Native graphical user interface based on Windows 7 (SP 1 or later), Windows 2008 Server R2 (SP 1 or later), Windows 8.1, Windows 2012 Server R2, Windows 10, Windows Server 2016 and Windows Server 2019
- 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.
- Various independent solver strategies (based on hexahedral as well as tetrahedral meshes) allow accurate results with a high performance for all kinds of high frequency applications
- For specific solvers, highly advanced numerical techniques offer features like Perfect Boundary Approximation (PBA), Thin Sheet Technique (TST) or octree-based meshing for hexahedral grids and curved and higher order elements for tetrahedral meshes

Structure Modeling

- Advanced ACIS-based, parametric solid modeling front end with excellent structure visualization
- Feature-based hybrid modeler allows quick structural changes
- Import of 3D CAD data from ACIS® SAT/SAB, CATIA®, SOLIDWORKS®, Autodesk Inventor, IGES, VDA-FS, STEP, PTC Creo, Siemens NX, Parasolid, Solid Edge, CoventorWare, Mecadtron, NASTRAN, STL or OBJ files
- Import of 2D CAD data from DXF™, GDSII and Gerber RS274X, RS274D files
- Import of EDA data from design flows including Cadence Allegro® / APD® / SiP®, Mentor Graphics Expedition®, Mentor Graphics PADS®, Mentor Graphics HyperLynx®, Zuchen CR-5000® / CR-8000®, IPC-2581 and ODB++® (e.g. Altium Designer, Mentor Graphics Boardstation®, CADSTAR®, Visula®)
- Import of OpenAccess and GDSII-based integrated-circuit layouts via CST Chip Interface
- Import of PCB designs originating from CST PCB STUDIO®
- Import of 2D and 3D sub models
- Import of Sonnet® EM models, National Instruments Microwave Office™ and Keysight Technologies ADS® layouts
- Import of a visible human model dataset or other voxel datasets
- Export of CAD data to ACIS SAT/SAB, IGES, STEP, NASTRAN, STL, DXF, GDSII, Gerber or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem setup

Transient Solver

- Fast and memory efficient Finite Integration Technique (FIT)
- Efficient calculation for loss-free and lossy structures
- Direct time-domain analysis and broadband calculation of S-parameters from one single calculation run by applying DFTs to time signals
- Possibility to suppress the disk storage of time signals
- Calculation of field distributions as a function of time or at multiple selected frequencies from one simulation run
- Solver stop criteria based on S-parameters, radiated power and probe results
- Adaptive mesh refinement in 3D using S-Parameter or 0D results as stop criteria
- Shared memory parallelization of the transient solver run and of the matrix calculator
- MPI Cluster parallelization via domain decomposition
- Support of hardware acceleration (selected NVIDIA and AMD GPUs)
- Combined simulation with MPI and hardware acceleration
- Support of Linux batch mode and batch queuing systems (e.g. Slurm, PBS Pro, LSF, SGE) including native shell support
- Support of more than 2 billion mesh cells (with MPI)

- Isotropic and anisotropic material properties
- Frequency dependent material properties with arbitrary order for permittivity and permeability as well as a material parameter fitting functionality
- Gyrotropic materials (magnetized ferrites) as well as field-dependent microwave plasma
- Non-linear material models (Kerr, Raman)
- Spatially varying material models (general or with specialized radial dependency) with optional dispersive behavior and 3D material monitors
- Surface impedance models (tabulated surface impedance, Ohmic sheet, lossy metal, corrugated wall, material coating, metal surface roughness)
- Frequency dependent thin panel materials defined based on a multilayered stackup or an S-Matrix table (isotropic and symmetric)
- Special perforation materials like wire mesh and air ventilation panels (isotropic)
- Time dependent conductive materials (volumetric or lossy metal type)
- Temperature dependent materials with coupling to the Thermal Solver from CST Studio Suite

- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Selective calculation of higher order port modes
- Automatic waveguide port mesh adaptation with optional result re-usage of identical ports
- Multipin and single-ended ports for (Q)TEM mode ports with multiple conductors
- Broadband treatment of inhomogeneous ports
- Multiport and multimode excitation (sequentially or simultaneously)
- PEC or PMC shielding functionality for waveguide ports
- Plane wave excitation (linear and broadband circular or elliptical polarization)
- Excitation by external nearfield sources imported from CST Studio Suite or Sigrity® tools or NFS *nearfield* scan data.
- Online TDR analysis with Gaussian or rectangular shape excitation function
- User defined excitation signals and signal database
- Simultaneous port excitation with different excitation signals for each port and broadband phase shift
- Single port excitation with user definable amplitude and phase setting
- Transient EM/circuit co-simulation with network elements
- AC radiation or irradiation cable co-simulation
- Transient radiation, irradiation or bi-directional cable co-simulation
- 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
- Inhomogeneous port accuracy enhancement for highly accurate S-parameter results, considering also low loss dielectrics
- Single-ended S-parameter calculation
- Possibility to use waveguide ports as mode monitors only

- S-parameter sensitivity and yield analysis
- Combined linear and logarithmic sampling of 1D spectral 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 or currents in time and frequency domain
- 1D power loss results (time and frequency domain) per material or solid
- Calculation of time averaged power loss volume monitors
- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.) with and without farfield approximation at multiple selected frequencies
- Broadband farfield monitors and farfield probes to determine broadband farfield information over a wide angular range or at certain angles
- Antenna array farfield calculation
- Radar Cross Section (RCS) calculation
- Calculation of Specific Absorption Rate (SAR) distributions
- Export of field source monitors, which then may be used as excitation for other high frequency solvers inside CST Studio Suite
- Discrete edge and face elements (lumped resistors) as ports
- Ideal voltage and current sources for EMC problems
- Discrete edge and face R, L, C, and (nonlinear) diode lumped elements at any location in the structure
- General lumped element circuit import from SPICE or Touchstone files
- Visualization of discretized wire endpoint connectivity
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Coupled simulations with the Thermal Solver from CST Studio Suite
- Network distributed computing for optimizations, parameter sweeps and multiple port/mode excitations

TLM Solver

- Time domain Transmission-Line Matrix (TLM) method with octree-based meshing
- Efficient calculation for loss-free and lossy structures
- Direct time-domain analysis and broadband calculation of S-parameters from one single calculation run by applying DFTs to time signals
- Applicable to EMC/EMI applications such as radiated and conducted emissions and immunity, EMP and lightning, electrostatic discharge (ESD), high speed interference and shielding analysis
- Solver stop criteria based on S-parameters, radiated power and probe results
- Support of GPU acceleration
- Isotropic and anisotropic materials (including materials with axes not aligned to the mesh)
- Frequency dependent material properties with arbitrary order for permittivity and permeability as well as a material parameter fitting functionality
- Gyrotropic materials with homogeneous biasing field
- Frequency dependent thin panel materials defined based on a multilayered stackup or an S-Matrix table
- Special perforation materials like wire mesh and air ventilation panels
- User defined excitation signals and signal database
- Simultaneous port excitation with different excitation signals for each port and broadband phase shift
- Transient EM/circuit co-simulation with network elements
- AC radiation or irradiation cable co-simulation
- Transient radiation, irradiation or bi-directional cable co-simulation

- Excitation by external nearfield sources imported from CST Studio Suite or Sigrity® tools or NFS *nearfield* scan data.
- Compact models which avoid excessively fine meshes for:
 - slots, seams and gaskets
 - multi-conductor wires
 - conductive coatings and absorbers
- Broadband compact antenna radiation sources based on the Equivalence Principle
- 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 or currents in time and frequency domain
- 1D power loss results (time and frequency domain) per material or solid
- Calculation of time averaged power loss monitors
- Antenna farfield calculation (including gain, beam direction, etc.)
- Broadband farfield monitors and farfield probes to determine broadband farfield information over a wide angular range or at certain angles
- Radar Cross Section (RCS) calculation
- Calculation of Specific Absorption Rate (SAR) distributions
- Export of field source monitors, which then may be used as excitation for other high frequency solvers inside CST Studio Suite
- Cylinder scan for emissions analysis yielding peak radiated fields vs. frequency
- Discrete edge or face elements (lumped resistors) as ports
- Ideal voltage and current sources for EMC problems
- Lumped R, L, C elements at any location in the structure
- Visualization of discretized wire endpoint connectivity

Frequency Domain Solver

- Efficient calculation for loss-free and lossy structures
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Adaptive mesh refinement in 3D using various stopping criteria: S-parameters or probe results at multiple frequency points, broadband S-parameters, as well as 0D and 1D result templates
- Special mesh refinement for singular edges
- True Geometry Adaptation
- Option to maintain the tetrahedral mesh during optimization and parameter sweep with small geometric changes
- Fast broadband adaptive frequency sweep for S-parameters and field probes
- Equidistant, logarithmic and user defined frequency sweeps and evaluation for 1D results
- Continuation of the solver run with additional frequency samples
- Low frequency stabilization
- Direct and iterative matrix solvers with convergence acceleration techniques
- Higher order representation of the fields, with either constant or variable order (with tetrahedral mesh)
- Support of Linux batch mode and batch queuing systems (e.g. OGE, LSF)
- Isotropic and anisotropic material properties
- Arbitrary frequency dependent material properties (general purpose sweep with tetrahedral mesh)
- Surface impedance model for good conductors, Ohmic sheets and corrugated walls, as well as frequency-dependent, tabulated surface impedance data and coated materials (with tetrahedral mesh)
- Inhomogeneously biased ferrites with a static biasing field (general purpose sweep with tetrahedral mesh), based on SAM (System and Assembly Modeling)
- Temperature dependent materials with coupling to the Thermal Solver from CST Studio Suite
- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Automatic waveguide port mesh adaptation (with tetrahedral mesh)

- Multipin ports for TEM modes in ports with multiple conductors
- Simultaneous excitation with individual amplitude and phase shift settings for selected excitations (with tetrahedral mesh)
- PEC or PMC shielding functionality for waveguide ports
- Plane wave excitation with linear, circular or elliptical polarization (with tetrahedral mesh), as well as plane waves in layered dielectrics (general purpose sweep)
- Discrete edge and face elements (lumped resistors) as ports (face elements with tetrahedral mesh, numerical face port solver for arbitrary shaped geometries with general purpose sweep)
- Ideal current source for EMC problems (general purpose sweep with tetrahedral mesh)
- Nearfield source imprint on open boundaries, lossy metal, and Ohmic sheets (general purpose sweep with tetrahedral mesh)
- Lumped R, L, C elements at any location in the structure
- Arbitrary shaped lumped elements (general purpose sweep with tetrahedral mesh)
- General lumped element circuit import from SPICE and Touchstone files (general purpose sweep with tetrahedral mesh)

- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters
- Single-ended S-parameter calculation, with native single-ended field monitors for tetrahedral mesh
- S-parameter sensitivity and yield analysis (with tetrahedral mesh)

- High performance radiating/absorbing boundary conditions
- Conducting wall boundary conditions (with tetrahedral mesh)
- Periodic boundary conditions including phase shift or scan angle
- Unit cell feature to simplify the simulation of periodic antenna arrays or of frequency selective surfaces (general purpose sweep)
- Convenient generation of the unit cell calculation domain from arbitrary structures (with tetrahedral mesh)
- Floquet mode ports (periodic waveguide ports)
- Fast farfield calculation based on the Floquet port aperture fields (general purpose sweep with tetrahedral mesh)

- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents, power flows, current densities, surface and volumetric power loss densities, electric energy densities, magnetic energy densities
- Antenna farfield and farfield probe calculation (including gain, beam direction, side lobe suppression, etc.) with and without farfield approximation
- Antenna array farfield calculation
- RCS calculation (with tetrahedral mesh)
- Calculation of SAR distributions (with hexahedral mesh)
- Export of field source monitors (with tetrahedral mesh), which then may be used as excitation for other high frequency solvers inside CST Studio Suite
- Export of fields for corona discharge and multipactor analysis with Spark3D (tetrahedral mesh only)

- 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
- Network distributed computing for frequency samples and remote calculation
- Coupled simulations with the Thermal Solver and the Stress Solver from CST Studio Suite

- Besides the general purpose frequency sweep, a fast reduced order model technique, specifically designed for the efficient calculation of broadband results such as S-parameters, field probes and far-field probes, is available.

Integral Equation Solver

- Broadband calculation of S-parameters also for near- and farfield excitations
- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents
- Antenna farfield calculation (including gain, beam direction, side lobe suppression, etc.)
- RCS calculation
- Fast monostatic RCS sweep
- Characteristic Mode Analysis (including modal weighting coefficient calculation)
- Supports antenna coupling workflow
- Export of field source monitors, which then may be used as excitation for other high frequency solvers inside CST Studio Suite

- Waveguide port excitation
- Plane wave excitation
- Nearfield source excitation
- Farfield source excitation
- Farfield source excitation with multipole coefficient calculation
- Receiving farfield source and nearfield source excitation
- Current distribution
- Discrete edge and face port excitation
- Face lumped R, L, C elements

- Symmetries are supported for discrete ports, waveguide ports, plane wave and farfield excitations.
- MPI parallelization for MLFMM and direct solver
- Support of GPU acceleration for MLFMM and direct solver
- Support of combined MPI & GPU acceleration
- Support of Linux batch mode and batch queuing systems (e.g. OGE, LSF)
- Infinite electric and magnetic ground planes
- Infinite Real Ground option
- Multithread parallelization
- Efficient calculation of loss-free and lossy structures including lossy waveguide ports
- Surface mesh discretization (triangles and quadrilaterals)
- Wire mesh discretization with special junction meshing strategy
- Support of Curved Mesh (quadrilateral and triangular surface mesh elements)
- Handling of layered media which enables simulation of windshield antennas etc.
- Isotropic material properties
- Coated materials
- Arbitrary frequency dependent material properties
- Surface impedance models (tabulated surface impedance, Ohmic sheet, lossy metal)
- Automatic fast broadband adaptive frequency sweep
- User defined frequency sweeps
- Low frequency stabilization
- Direct and iterative matrix solvers with convergence acceleration techniques
- Higher order representation of the fields including mixed order
- Single and double precision floating-point representation

- Port mode calculation by a 2D eigenmode solver in the frequency domain
- Automatic waveguide port mesh adaptation
- Simultaneous excitation with individual amplitude and phase shift settings for selected excitations
- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters
- 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
- Network distributed computing for frequency sweeps

Multilayer Solver

- Broadband calculation of S-parameters
- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents
- Waveguide (multipin) port excitation
- Discrete face port excitation
- Plane wave excitation
- Characteristic Mode Analysis (including modal weighting coefficient calculation)
- Face lumped R, L, C elements
- Multithread parallelization
- MPI parallelization for the direct solver
- Efficient calculation of loss-free and lossy structures
- Surface mesh discretization (curved triangles and quadrilaterals)
- Support of Curved Mesh (quadrilateral and triangular surface mesh elements)
- Automatic edge mesh refinement is available for finite-thickness and infinitely thin conductors
- Isotropic material properties
- Arbitrary frequency dependent material properties
- Automatic fast broadband adaptive frequency sweep
- User defined frequency sweeps
- Re-normalization of S-parameters for specified port impedances
- Phase de-embedding of S-parameters
- Simultaneous excitation with individual amplitude and phase shift settings for selected excitations
- 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
- Network distributed computing for frequency sweeps

Asymptotic Solver

- Specialized tool for fast monostatic and bistatic RCS sweeps and antenna farfield calculations
- Fast ray tracing technique including multiple reflections and edge diffraction (SBR) by using either independent rays or ray-tubes
- Supports antenna coupling workflow
- Field of view analysis
- Multiple plane wave excitations with different polarization types
- Farfield source excitation
- Nearfield source excitation
- Receiving farfield source and nearfield source excitation
- Robust surface mesh discretization including curved meshes
- PEC and vacuum material properties
- Complex surface impedance materials
- Coated materials (incl. frequency dependent and angle dependent properties)
- Thin dielectrics (incl. frequency dependent and angle dependent properties)
- User defined frequency sweeps and angular sweeps
- Visualization of rays and their amplitudes, including multiple reflections
- Visualization of points where the rays initially hit the structure
- Computation of range profiles and sinograms
- Computation of scattering hotspots
- Computation of RCS maps
- Tabulated export of raw solver data
- Calculation of electric and magnetic fields

- Export of field source monitors, which then may be used as excitation for other high frequency solvers inside CST Studio Suite
- Multithread parallelization
- Support of GPU acceleration for field sources and bistatic calculations
- 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
- Network distributed computing for excitation angles
- Network distributed computing for near- and farfield sources

Eigenmode Solver

- Calculation of modal field distributions in closed / opened loss-free or lossy structures
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Isotropic and anisotropic materials
- Multithread parallelization
- Adaptive mesh refinement in 3D, with True Geometry Adaptation
- Open, conducting wall, and periodic boundary conditions including phase shift
- Accurate calculation of losses and internal / external Q-factors for each mode (directly or using a perturbation method)
- Discrete L, C elements at any location in the structure
- Target frequency can be set (calculation within the frequency interval)
- Calculation of all eigenmodes in a given frequency interval
- Sensitivity analysis with respect to materials and geometric deformations defined by face constraints (with tetrahedral mesh)
- 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
- Coupled simulations with the Thermal Solver and the Stress Solver from CST Studio Suite
- Export of fields for corona discharge and multipactor analysis with Spark3D (tetrahedral mesh only)

Partial RLC Solver

- Calculation of equivalent circuit parameters (partial inductances, resistances, and capacitances) and optional SPICE export
- For a detailed description consult the online documentation

Schematic View

- The schematic view shows the circuit level description of the current high frequency simulation project
- Allows additional wiring, including active and passive circuit elements as well as more complex circuit models coming from measured data (e.g. Touchstone or IBIS files), analytical or semi analytical descriptions (e.g. microstrip or stripline models) or from simulated results (CST Studio Suite projects)
- Offers many different circuit simulation methods, including transient EM/circuit co-simulations
- All schematic elements as well as all defined parameters of the connected high frequency simulation project can be parameterized and are ready for optimization runs
- Geometry creation by assembling the components on the schematic in 3D
- Flexible and powerful hierarchical task concept offering nested parameter sweep / optimizer setups
- Recombination of 3D fields for stimulations calculated by the circuit simulator

SAM (System Assembly and Modeling)

- 3D representations for individual components
- Automatic project creation by assembling the schematic's elements into a full 3D representation
- Fast parametric modeling front end for easy component transformation and alignment
- Manage project variations derived from one common 3D geometry setup
- Coupled Multiphysics simulations by using different combinations of coupled circuit/EM/Thermal/Stress projects
- Hybrid Solver Task (uni- or bi-directional coupling of 3D high frequency solvers)
- Antenna Array Task

Visualization and Secondary Result Calculation

- Multiple 1D result view support
- 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
- Automatic parametric 1D result storage
- 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
- Calculation and display of farfields (fields, gain, directivity, RCS) in xy-plots, polar plots, scattering maps, radiation plots (3D)
- Nearfield cylinder scan visualization
- Calculation of Specific Absorption Rate (SAR) including averaging over specified mass
- Calculation of surface losses by perturbation method and of the Q factor
- 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 field plots
- Export of farfield data as excitation for integral equation or asymptotic solver
- Export of frequency domain nearfield data from transient or frequency domain solver, for use as excitation in transient solver

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 you to get a quick start with CST Studio Suite. It is not intended to be a complete reference guide for 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 *Simulation Workflow* (Chapter 2) which will guide you through the most important features of CST Studio Suite for High Frequency Simulation. We strongly encourage you to study this chapter carefully.

Document Conventions

- Buttons that should be pressed within dialog boxes are always written in italics, e.g. *OK*.
- Key combinations are always joined with a plus (+) sign. *Ctrl+S* means that you should hold down the *Ctrl* key while pressing the *S* key.
- The program's features can be accessed through a Ribbon command bar at the top of the main window. The commands are organized in a series of tabs within the Ribbon. In this document a command is printed as follows: *Tab name: Group name* \Rightarrow *Button name* \Rightarrow *Command name*. This means that you should activate the proper tab first and then press the button *Command name*, which belongs to the group *Group name*. If a keyboard shortcut exists, it is shown in brackets after the command.
Example: *View: Change View* \Rightarrow *Reset View (Space)*
- The project data is accessible through the navigation tree on the left side of the application's main window. An item of the navigation tree is referenced in the following way: *NT: Tree folder* \Rightarrow *Sub folder* \Rightarrow *Tree item*.
- Example: *NT: 1D Results* \Rightarrow *Port Signals* \Rightarrow *i1*

Your Feedback

We are constantly striving to improve the quality of our software documentation. If you have any comments regarding the documentation, please send them to your support center: 3DS.com/Support.

Chapter 2 – Simulation Workflow

The following example shows a fairly simple S-parameter calculation of a coaxial connector. Studying this example carefully will help you become familiar with many standard operations that are important when performing a high frequency simulation with CST Studio Suite.

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 applications.

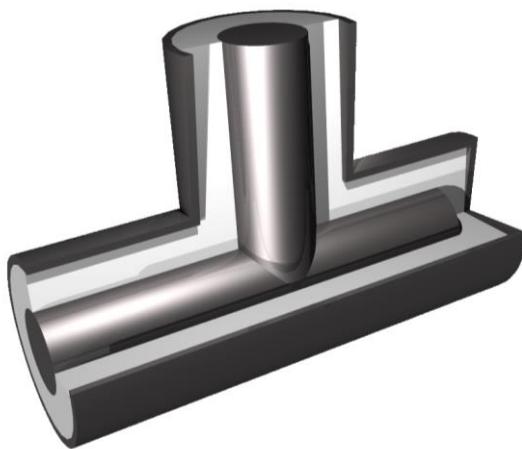
In subsequent sections you will find some remarks concerning how typical procedures may differ for other kinds of simulations.

Setup the Simulation Model

The following explanations describe the “long” way to open a particular dialog box or to launch a particular command. Whenever available, the corresponding Ribbon 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 activating the command from the context menu) is omitted. You should regularly open the context menu by clicking the right mouse button, to check available commands for the currently active mode.

The Structure

In this 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 fields inside this structure at various frequencies. The picture below shows the current structure of interest (it has been sliced open to aid visualization), and was produced using the POV export option.



Before you start modeling the structure, let's spend a few moments discussing how to construct this structure efficiently. Due to the outer conductor of the coaxial cable, the structure's interior 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 just the dielectric and inner conductor in a perfectly conducting block.

In order to simplify this procedure, CST Studio Suite allows you to define the properties of the background material. Any part of the simulation volume that you do not specifically fill with some 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.

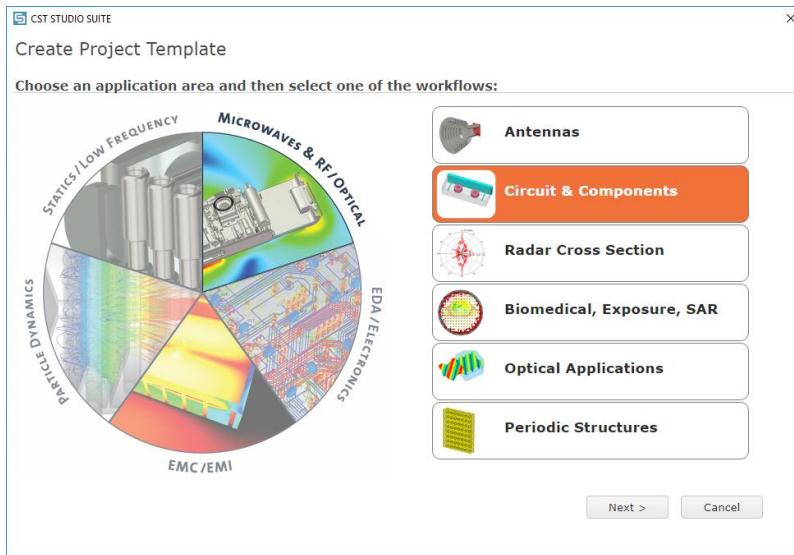
The method of constructing the structure should therefore be as follows:

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

Create a New Project

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

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

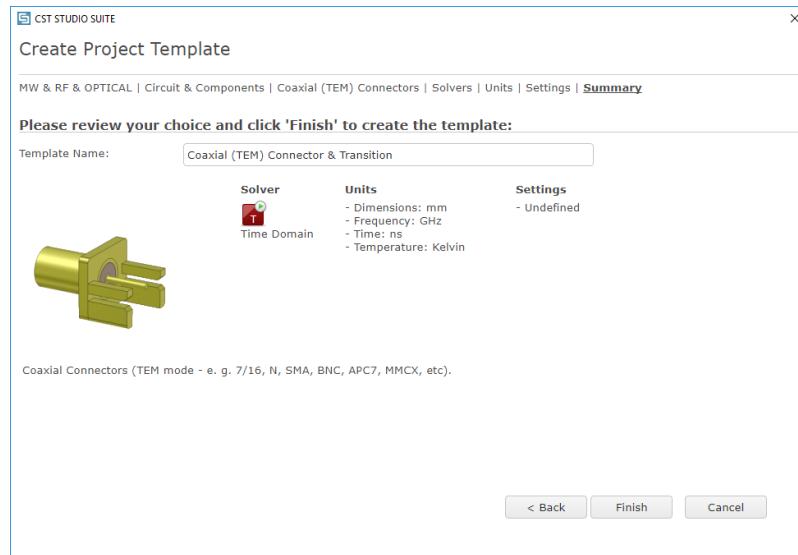


For the coaxial structure, please select *Circuit & Components* \Rightarrow *Coaxial (TEM) Connectors* \Rightarrow *Time Domain* .

Now you are requested to select the units which best fit your application. For this example, we can leave the pre-selected units as follows:

Dimensions:	mm
Frequency:	GHz
Time:	ns

On the next page it is possible to already define a frequency range as well as typical 3D field monitor specifications for your template, in case these settings are reusable for several of your models. However, we will define these settings later on during the model setup, so we can continue by again clicking the *Next* button. Now you can give the project template a name and review a summary of your initial settings:



Finally click the *Finish* button to save the project template and to create a new project with appropriate settings. The high frequency module will be launched automatically due to the choice of the application area *MW & RF & Optical*.

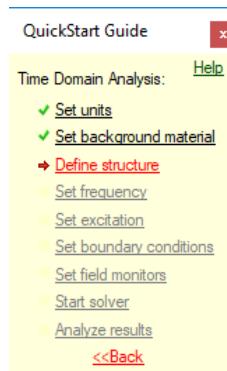
Please note: When you click again on *File: New and Recent* you will see that the recently defined template appears in the *Project Templates* section. For further projects in the same application area you can simply click on this template entry to launch the high frequency module with useful basic settings. It is not necessary to define a new template each time. You are now able to start the software with reasonable initial settings quickly with just one click on the corresponding template.

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

Open the QuickStart Guide

An interesting feature of the online help system is the QuickStart Guide, an electronic assistant that will guide you through your simulation. If it does not show up automatically, you can open this assistant by selecting *QuickStart Guide* from the dropdown menu next to the Help button  in the upper right corner.

The following dialog box should then be visible at the upper right corner of the main view:



As the project template has already set the solver type, units and background material, the *Time Domain Analysis* is preselected and some entries are marked as done. The red arrow always indicates the next step necessary for your problem definition. You do not have to follow the steps

in this order, but we recommend you follow this guide at the beginning to ensure that 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 *QuickStart 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 units in *mm* and frequencies in *GHz*. You can change these settings by entering the desired values in the units dialog box (*Home: Settings \Rightarrow Units* ), but for this example you should just leave the settings as specified by the template.

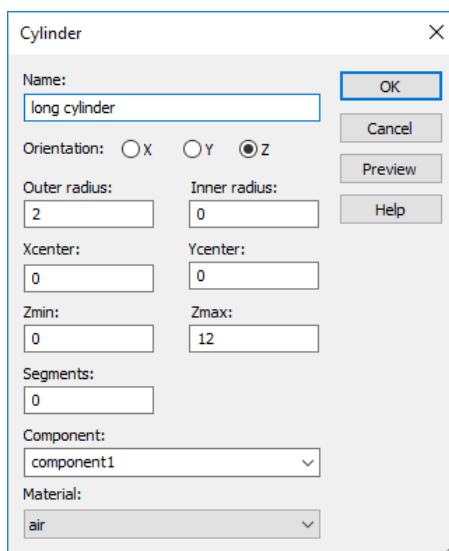
Define the Background Material

As discussed above, the structure will be described within the perfectly conducting background material which the coaxial connector template has set for you. In order to change it you may enter the corresponding dialog box (*Modeling: Materials \Rightarrow Background* ). For this example, however you do not need to change anything.

Model the Structure

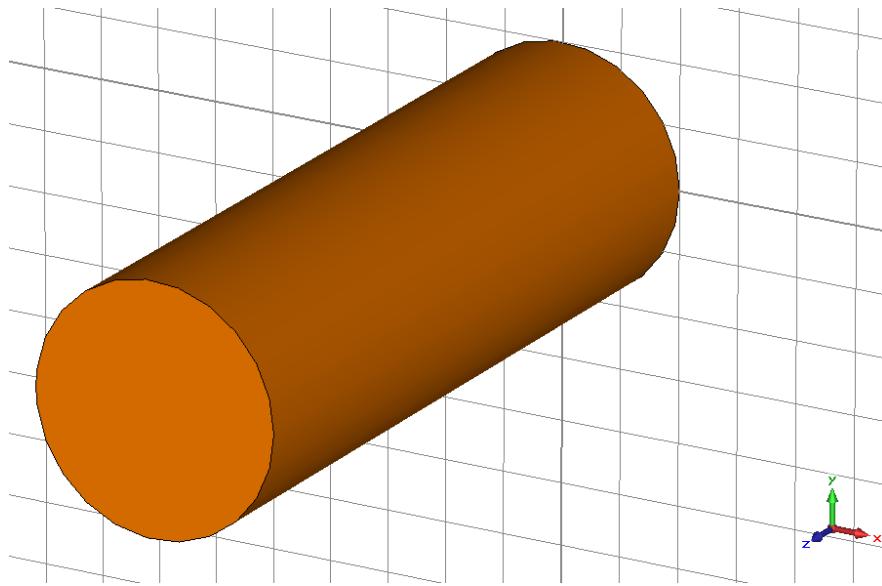
The first step is to create a circular cylinder along the z-axis of the coordinate system:

1. Open the cylinder creation dialog by selecting *Modeling: Shapes \Rightarrow Cylinder* .
2. Press the *Shift+Tab* keys 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 (skipping the definition of the inner radius).
6. In the shape dialog box, enter “long cylinder” in the *Name* field.
7. You could simply select the predefined material *Vacuum* (which is very similar 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 *Material* dropdown list.
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



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.

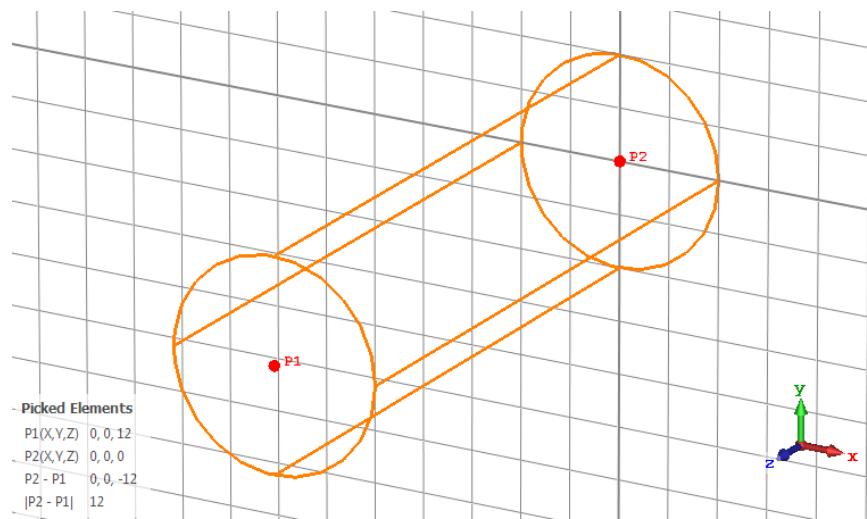


The next step is to create a second cylinder perpendicular to the first. The center of the new cylinder's base should be aligned with the center of the first cylinder.

Follow these steps to define the second cylinder:

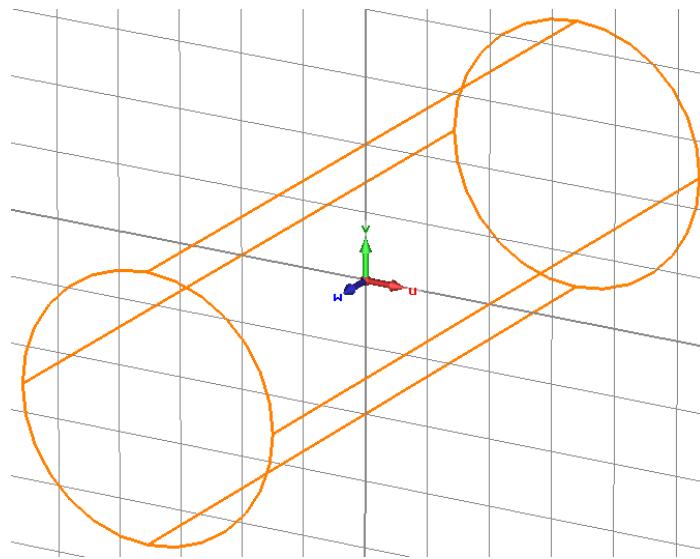
1. Select View: *Visibility* \Rightarrow *Wire Frame* (**Ctrl+W**) to activate the wire frame draw mode.
2. Activate the “circle center” pick tool: *Modeling: Picks* \Rightarrow *Pick Points* \Rightarrow *Pick Circle Center* (**C**) .
3. Double-click on one of the cylinder’s circular edges so that a 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 this:



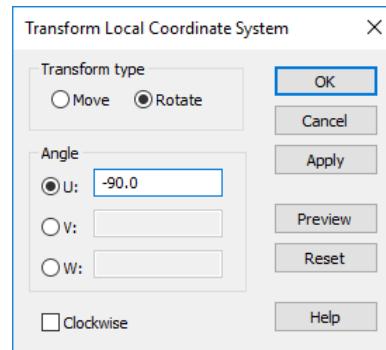
Next replace the two selected points by a point half way between the two by selecting *Modeling: Picks* \Rightarrow *Pick Points* \Rightarrow *Mean Last Two Points* (**Ctrl + Shift + M**).

You can now move the origin of the local working coordinate system (WCS) to this point by choosing *Modeling: WCS* \Rightarrow *Align WCS* \Rightarrow *Align WCS with Selected Point* or *WCS* \Rightarrow *Align WCS (W)* . The screen should look like this:



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

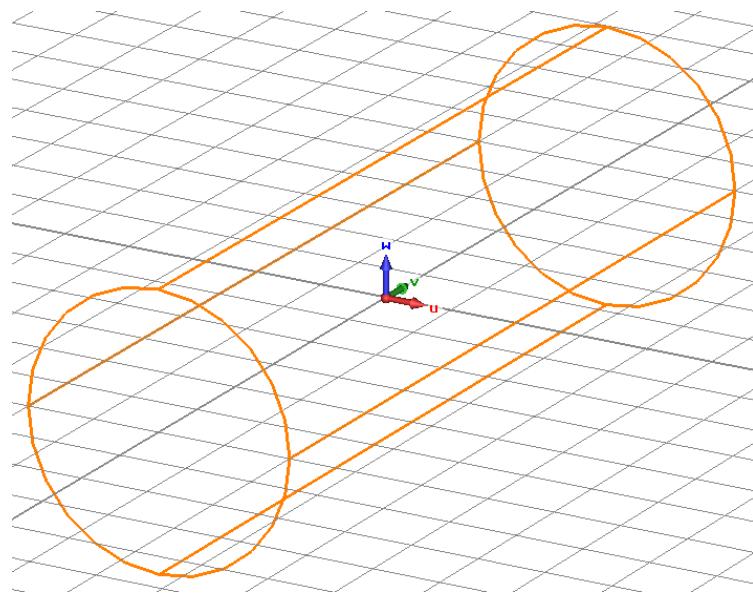
1. Select *Modeling: WCS* \Rightarrow *Transform WCS* .
2. Select *Rotate* as *Transform type*.
3. Select *U* as rotation axis and enter a rotation *Angle* of -90 degrees.
4. Click the *OK* button.



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.

Furthermore, you can also perform the transformation interactively with the mouse after selecting *Modeling: WCS* \Rightarrow *Transform WCS* .

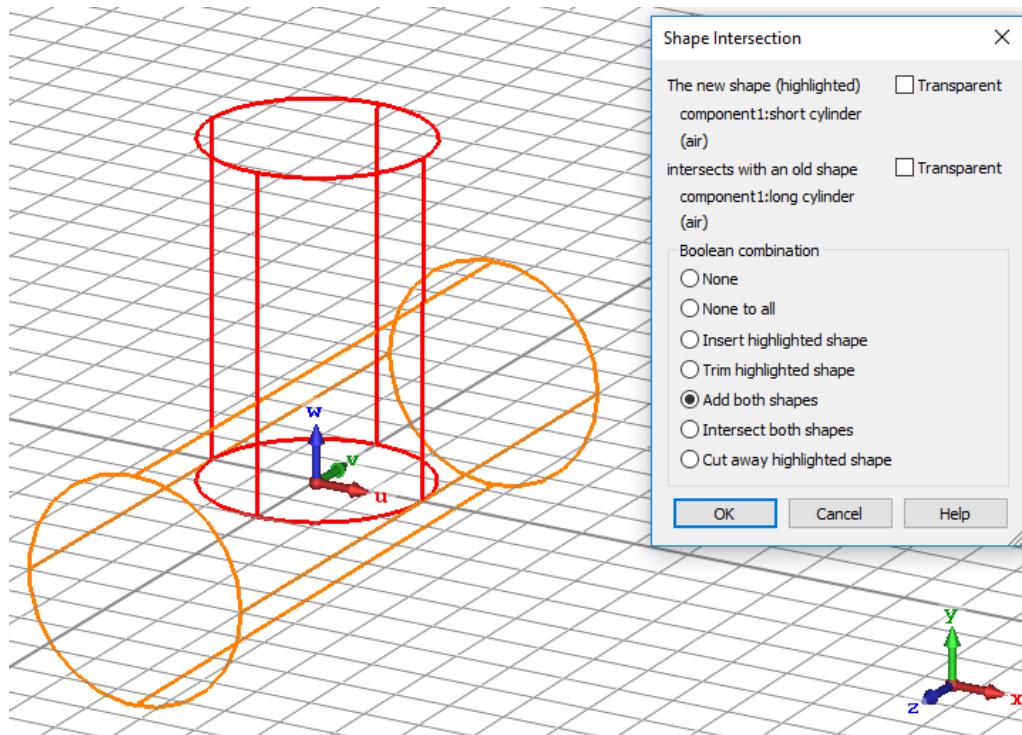
Now the structure should look like this:



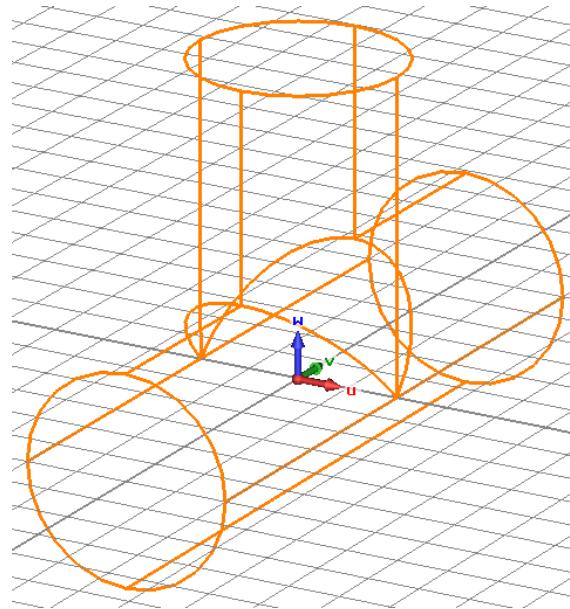
The next step is to create the second cylinder perpendicular to the first one:

1. Open the cylinder creation dialog by selecting *Modeling: 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.

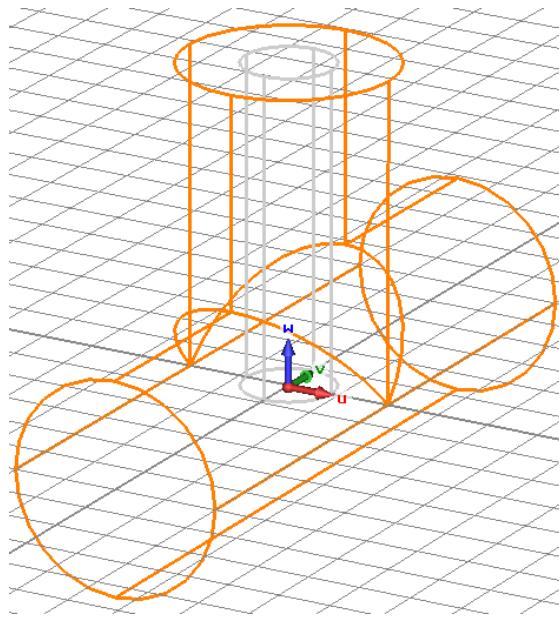
Since the coordinate system is already aligned with the center of the second cylinder, you can go ahead and start to create the first part of the conductor:

1. Open the cylinder creation dialog by selecting *Modeling: 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 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.

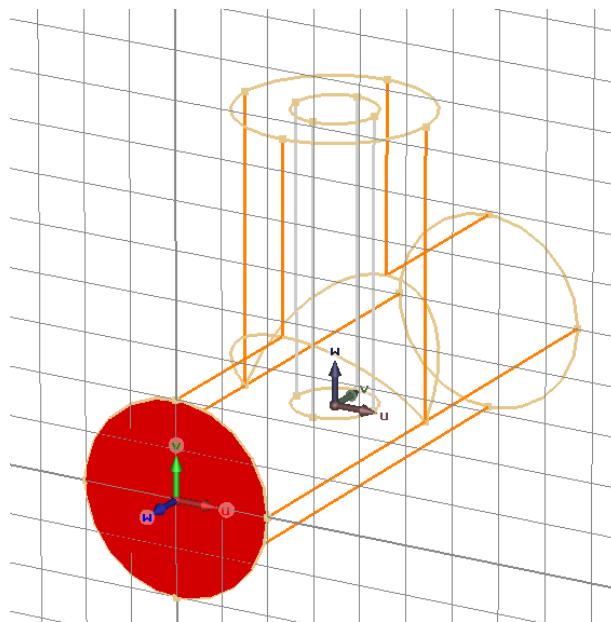
At this point we should briefly discuss the intersections between shapes. In general, each point in space should be identified with one particular material. However, perfect electric conductors can be seen as a special kind of material. It is allowable for a perfect conductor to be present at the same 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. Therefore, the intersection dialog box will not be shown automatically in the case of a perfect conductor overlapping with a dielectric material or with another perfect conductor. On the other hand, two different dielectric shapes may not overlap.

Background information: Some structures contain extremely complex conducting parts embedded within dielectric materials. In such cases, the overall complexity of the model can be significantly reduced by NOT intersecting these two materials. This is the reason why CST Studio Suite allows this exception for the high frequency module. However, you should 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 appear:



Now you should add the second conductor. First align the local working coordinate system with the upper z-circle of the first dielectric cylinder by selecting *Modeling: WCS* \Rightarrow *Align WCS* and double-click on the first cylinder's upper z-plane:



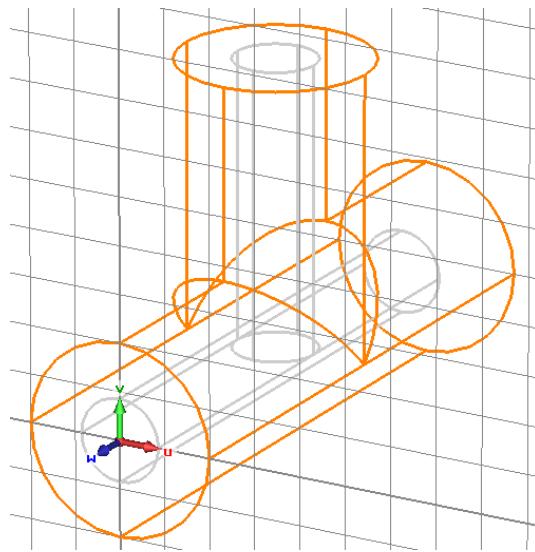
The w-axis of the local coordinate system is aligned with the first cylinder's axis, so you can now create the second part of the conductor:

1. Open the cylinder creation dialog by selecting *Modeling: 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 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*.

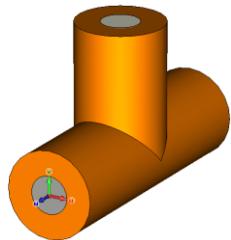
The newly created cylinder intersects with the dielectric part as well as with the previously created PEC cylinder. Even if there are two intersections (dielectric / PEC and PEC / PEC), the *Shape*

intersection dialog box will not be shown here since both types of overlaps are well defined. In both cases the common volume will be filled with PEC.

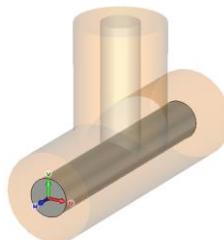
Congratulations! You have just created your first structure within CST Studio Suite. The view should now look like this:



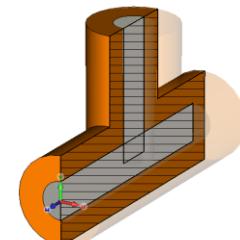
The following gallery shows some views of the structure available using different visualization options:



Shaded view
(deactivated working
plane,
View: Visibility \Rightarrow
Working Plane (Alt+W)

Shaded view
(long conductor
selected)

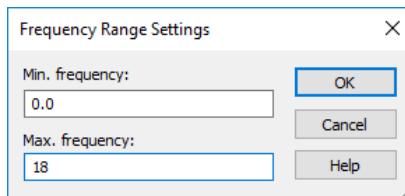


Shaded view with
cutplane
(*View: Sectional View* \Rightarrow
Cutting Plane (Shift+C)

Appearance of part
above
cutplane = transparent)

Define the Frequency Range

The next important setting for the simulation is the frequency range of interest. If not already specified by your template settings, you can specify the frequency by choosing *Simulation: Settings \Rightarrow Frequency* .



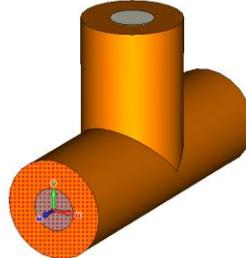
In this example you should specify a frequency range between 0 and 18 GHz. Since you have already set the frequency unit to GHz, you need to define only the absolute numbers 0 and 18 (the status bar always displays the current unit settings).

Define Ports

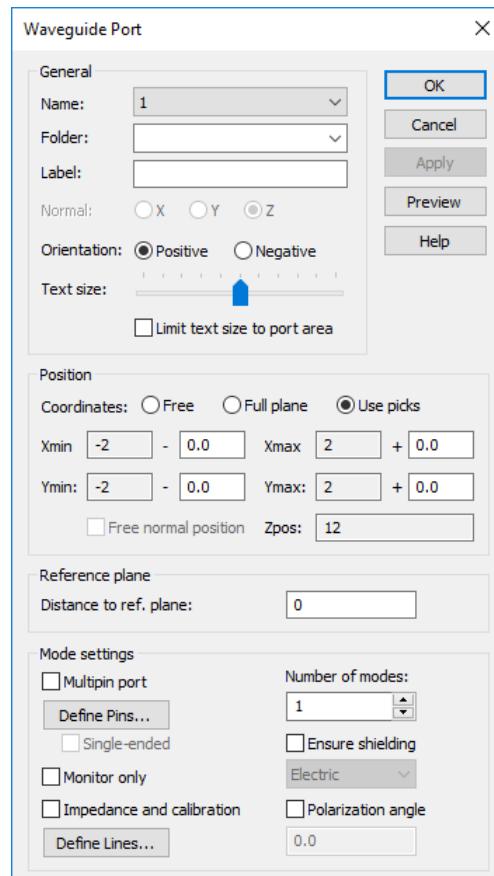
The following calculation of S-parameters requires the definition of ports through which energy enters and leaves the structure. You can do this by simply selecting the corresponding faces before entering the ports dialog box.

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

1. Select *Simulation: Picks* \Rightarrow *Pick Points, Edges or Faces (S)*.
2. Double-click on the upper w-plane aligned face of the dielectric part. The selected face will be highlighted:

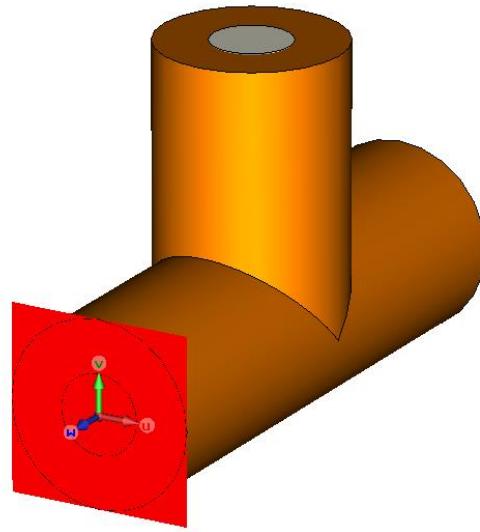


3. Open the ports dialog by selecting *Simulation: Sources and Loads* \Rightarrow *Waveguide Port*:

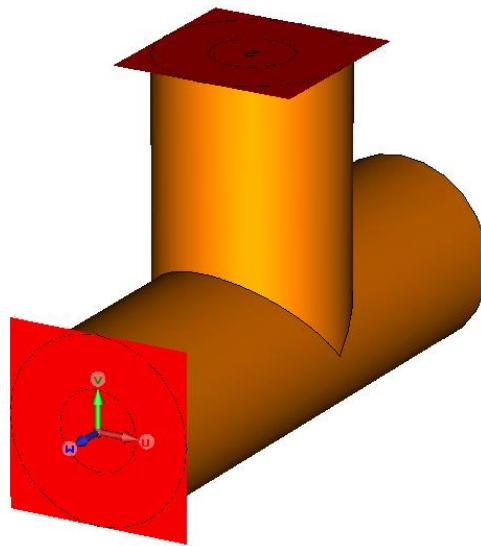


Everything is already set up correctly for the coaxial cable simulation, so you can simply click *OK* in this dialog box.

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:

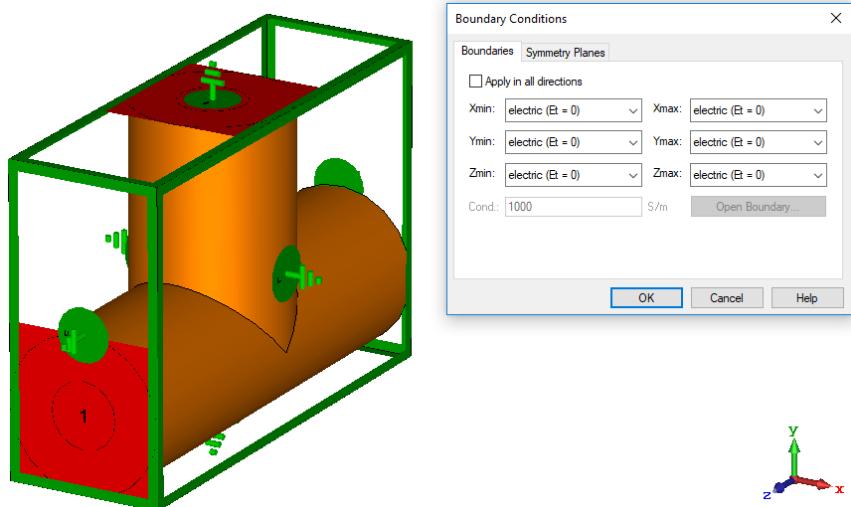


The correct definition of ports is very important for obtaining accurate S-parameters. Please refer to the **Choosing the Right Port** section later in this manual for more information about the correct placement of ports for various types of structures.

Define Boundary and Symmetry Conditions

The simulation of this structure will only be performed within the bounding box of the structure. You must specify a boundary condition for each plane (X_{\min}/X_{\max} / Y_{\min}/Y_{\max} / Z_{\min}/Z_{\max}) of the bounding box.

The boundary conditions are specified in a dialog box you can open by choosing *Simulation: Settings \Rightarrow Boundaries*

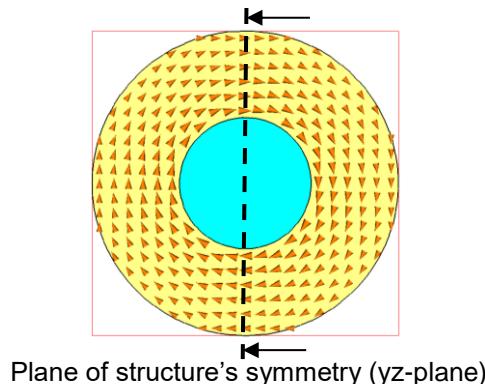


While the boundary dialog box is open, the boundary conditions will be visualized in the structure view as in the picture above.

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.

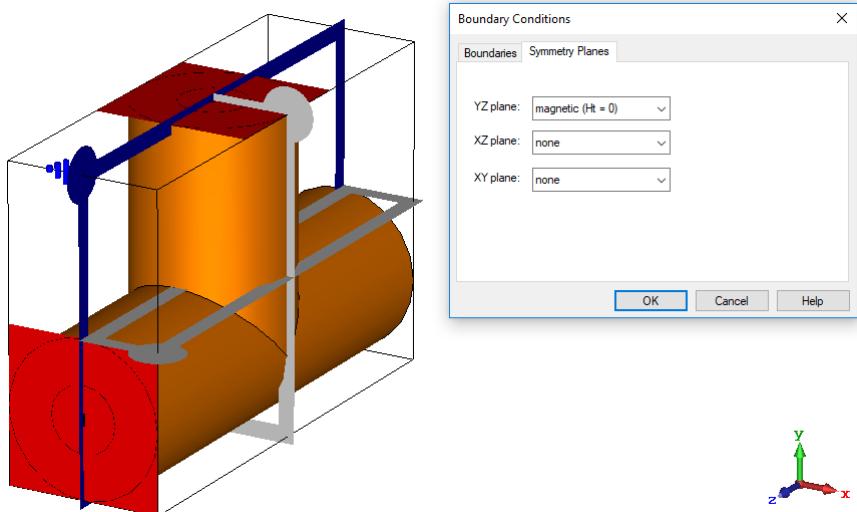
In our example, the structure is symmetric in the yz -plane (perpendicular to the x -axis) in the center of the structure. The excitation of the fields will be performed by the fundamental mode of the coaxial cable for which the magnetic field is shown below:



The magnetic field has no component tangential to the plane of the structure’s symmetry (the field is always oriented perpendicular to this plane). If you specify this plane as a “magnetic” symmetry plane, you can direct CST Studio Suite 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 ($H_t=0$)* in one of two ways. Either select the appropriate option in the dialog box, or double-click on the corresponding symmetry plane visualization in the view and select the proper choice from the context menu. Once you have done so, your screen will appear as follows:



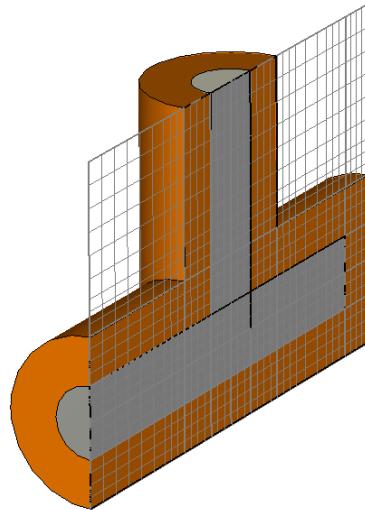
Finally click *OK* in the dialog box to store the settings. The boundary visualization will then disappear.

Visualize the Mesh

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

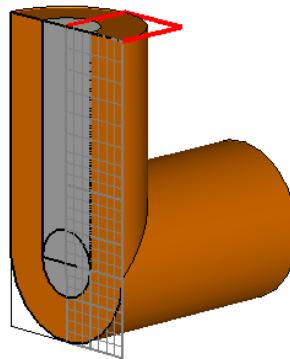
The hexahedral mesh generation for the structure analysis will be performed automatically based on an expert system. However, in some situations it may be helpful to inspect the mesh in order to improve the simulation speed by changing the parameters for the mesh generation.

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



You can modify the orientation of the mesh plane by adjusting the selection in the *Mesh: Sectional View* \Rightarrow *Normal* dropdown list or just by pressing the *X/Y/Z* keys. Move the plane along its normal direction using the *Up/Down* cursor keys. The current position of the plane will be shown in the *Mesh: Sectional View* \Rightarrow *Position* field.

Because of the symmetry setting, the mesh plane extends across only one half of the structure, what can be seen by e.g. changing the plane normal to the *z*-direction:



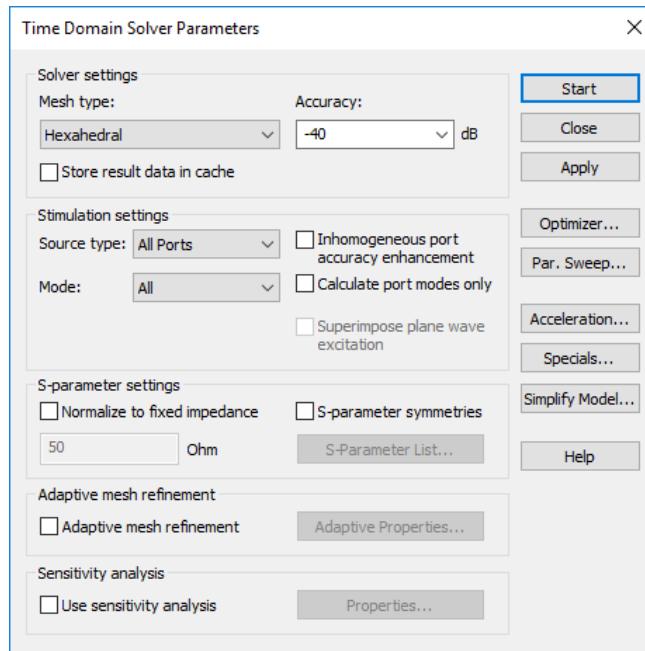
In this view also the PBA representation of the curved structure is seen in the mesh cells that are partly filled with PEC and partly with air.

There are some thick mesh lines shown in the mesh view. These mesh lines represent important planes (so-called snapping planes) at which the expert system finds it necessary to place mesh lines. You can control these snapping planes in the Special Mesh Properties dialog by selecting *Mesh: Mesh Control / Simulation: Mesh \Rightarrow Global Properties \Rightarrow Specials \Rightarrow Snapping*.

In most cases the automatic mesh generation will produce a reasonable initial mesh, but we recommend that you later spend some time reviewing 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 toggling *Mesh: Close \Rightarrow Close Mesh View* or just by pressing the *ESC* key.

Start the Simulation

After defining all necessary parameters, you are ready to start your first simulation from the transient solver control dialog box by selecting *Simulation: Solver \Rightarrow Setup 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 and S21 will be calculated. Setting the *Source Type* to *Port 2* will calculate S22 and S12.

If the full S-matrix is needed, you may also set the *Source Type* to *All Ports*. In this case a 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 example you should compute the full S-matrix and leave *All Ports* as your *Source type* setting.

The calculated S-parameters will always be normalized to the port impedance (which will be calculated automatically) by default. For this model, the port impedance will be approximately

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

for the coaxial lines with the specified dimensions and dielectric constants. However, sometimes you may need the S-parameters for a fixed normalization impedance (e.g. 50 Ohm), so in such a case you should 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 Ohm. Note that the re-normalization of the S-parameters is possible only when all S-parameters have been calculated (*Source Type* = *All Ports*).

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

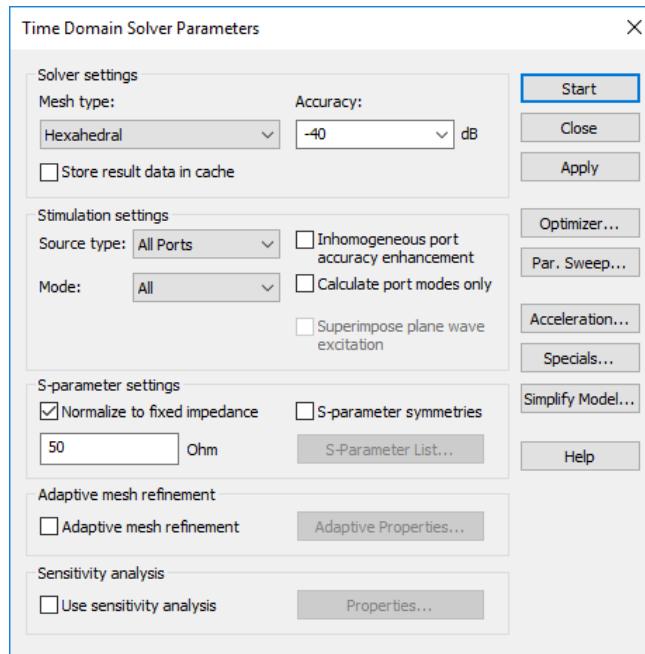
In order to obtain the S-parameters, the transformation of the time signals into the frequency domain requires the signals to have sufficiently decayed to zero. Otherwise a truncation error will occur, causing ripples on the S-parameter curves.

The time domain solver features an automatic control that stops the transient analysis when the energy inside the device, and thus the time signals at the ports, have decayed sufficiently close 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).

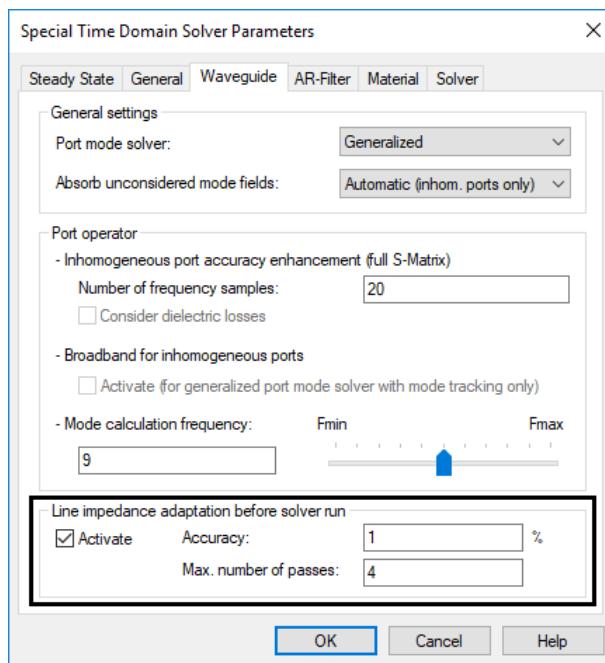
The chosen coaxial connector template already set the solver *Accuracy* to -40 dB to limit the maximum truncation error to 1% for this example.

The solver will excite the structure with a Gaussian pulse in the 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 peak.

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



In order to also achieve accurate results for the line impedance values of (Q)TEM modes, an adaptive mesh refinement in the port regions is performed as a pre-processing step before the transient simulation itself is started. This procedure refines the port mesh until a defined accuracy value or a maximum number of passes has been reached. These settings can be adjusted in the following dialog box *Simulation* \Rightarrow *Solver* \Rightarrow *Setup Solver* \Rightarrow *Specials* \Rightarrow *Waveguide*:



Since we want to simulate a coaxial structure with static port modes, we keep the adaptation enabled with its default settings. You can now close the *Specials* dialog box without any changes and then start the simulation by clicking the *Start* button.

A progress bar will appear in the progress window which will update you on the solver's progress. You can activate this window by selecting *View*: *Window* \Rightarrow *Windows* \Rightarrow *Progress Window*. Information text regarding the simulation will appear above the progress bar. The most important stages are listed below:

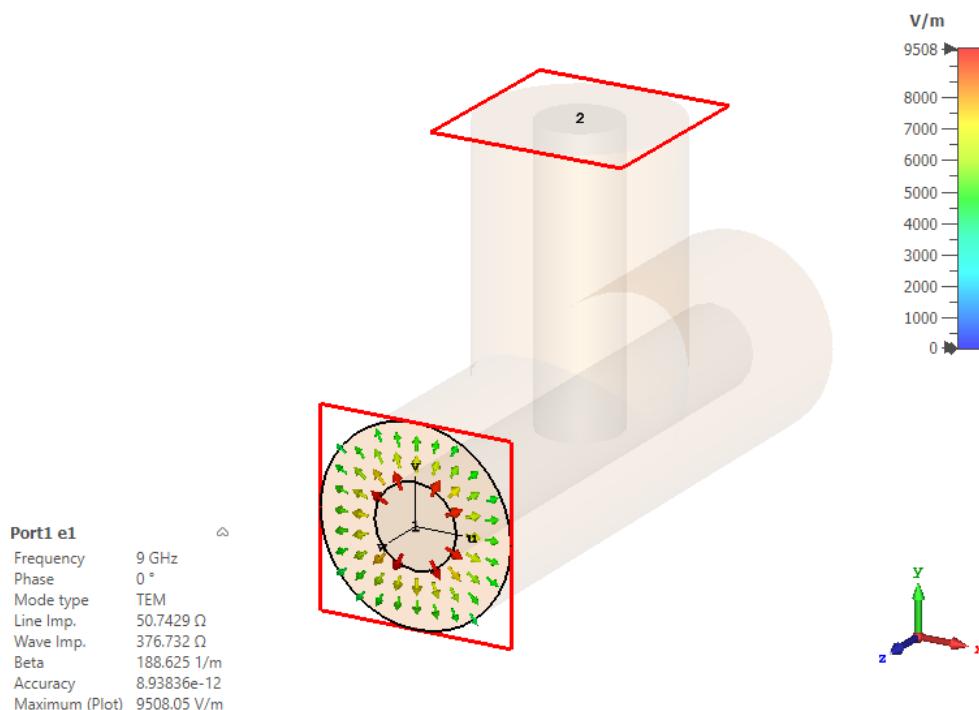
- Analyzing port domains:** During this first step, the port regions are analyzed for the port mesh adaptation to follow.
- Port mode calculation:** Here, the port modes are calculated during the port mesh adaptation. This step is performed several times for each port until a defined accuracy value or a maximum number of passes has been reached.
- Calculating matrices: Processing CAD model:** During this step, your input model is checked and processed.
- Calculating matrices: Computing coefficients:** During this step, the system of equations which will subsequently be solved is set up.
- Data rearrangement: Merging results:** For larger models the matrices are calculated in parallel and the results are merged at the end.
- Transient analysis: Calculating port modes:** In this step, the solver calculates the port mode field distributions and propagation characteristics as well as the port impedances if they have not been previously calculated. This information will be used later in the time domain analysis of the structure.
- 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.
- Transient analysis: Transient field analysis:** After the excitation pulse has vanished, there is still electromagnetic field energy inside the structure. The solver continues to calculate the field distribution and the S-parameters until the energy inside the structure and the port signals 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 while 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 at port 1 from *NT* (navigation tree): *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 item for the electric field of the first mode *e1* will display the port mode and its relevant parameters in the main view:

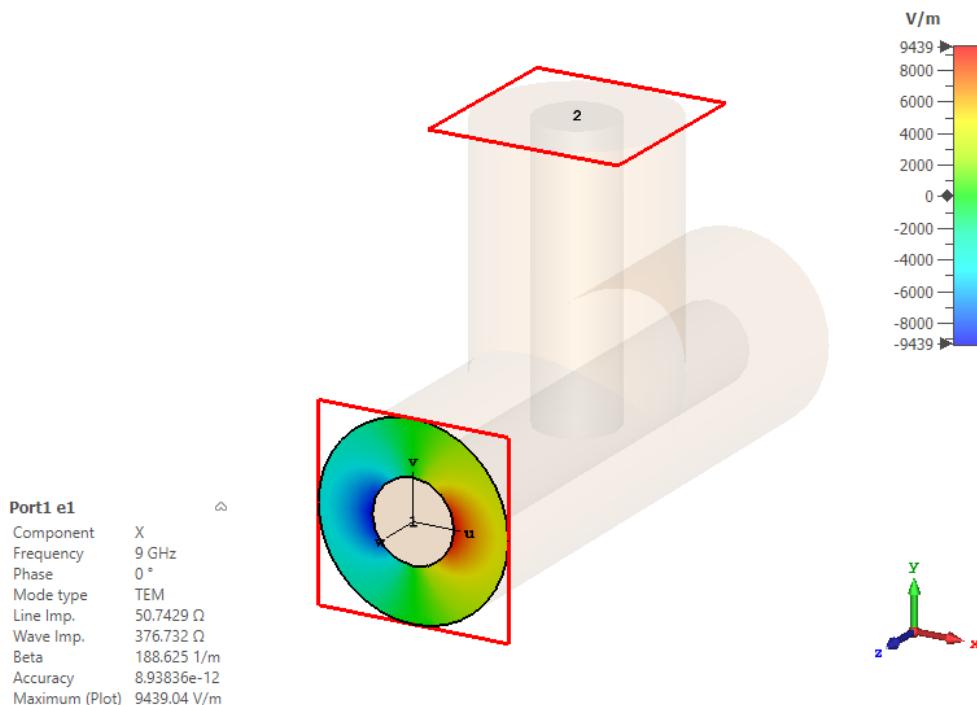


Besides information on the type of mode (in this case TEM), you will also find the propagation constant (beta) at the center frequency. Additionally, the port impedance is calculated automatically (line impedance).

You will find that the calculated result for the port impedance of 50.75 Ohm agrees well with the analytical solution of 50.58 Ohm after the port mesh adaptation has run. The small difference is caused by the discretization of the structure. The agreement between simulation and theoretical value can be improved by defining a smaller value for the *Accuracy limit* of the port mesh adaption or by increasing the overall initial mesh density. 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 *2D/3D Plot: Plot Properties* \Rightarrow *Properties* (or *Plot Properties* in the context menu).

You may visualize the scalar fields by opening the *e1* item and selecting *Contour* from the plot type pull-down menu in the 2D/3D Plot ribbon *2D/3D Plot: Plot Type* \Rightarrow *Contour*. The field component *Abs* will be visualized as a contour plot by default. To visualize the field component *X* you can select it from the field component pull-down menu or from the context menu:

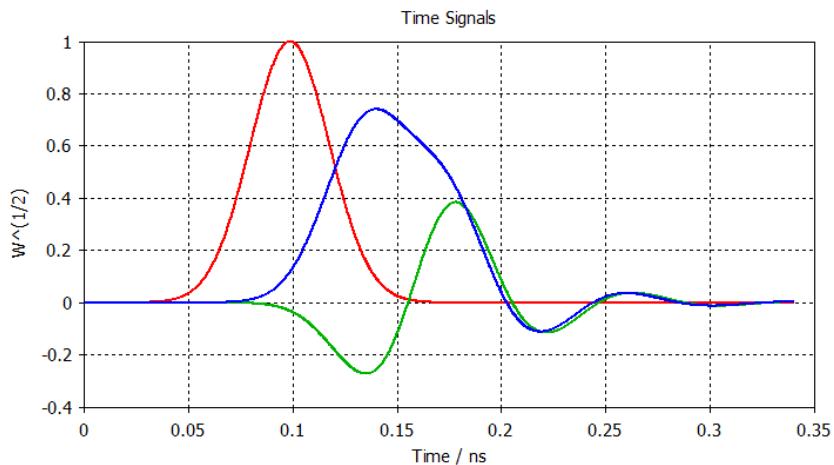


You may again change the type of the scalar visualization by selecting a different visualization option in the corresponding dialog box: *2D/3D Plot: Plot Properties* \Rightarrow *Properties* (or *Plot Properties* in the context menu).

Please experiment with the various settings in this dialog to become familiar with the different visualization options before you proceed with the next step.

Analyze the S-Parameters

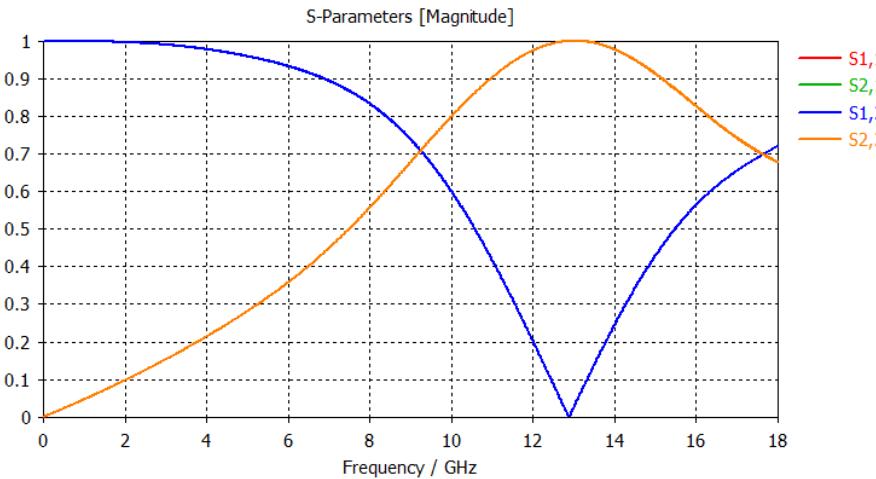
After a simulation has finished, you should always look at the time signals of the port modes. You can visualize these signals by choosing *NT: 1D Results* \Rightarrow *Port signals*. After selecting this folder, the following plot should appear:



The input signals are named with reference to their corresponding ports: i_1 (for port 1), i_2 and so on. The output signals are similarly named “ $o_{1,1}$ ”, “ $o_{2,1}$ ”, etc., where the number following the comma indicates the corresponding excitation port.

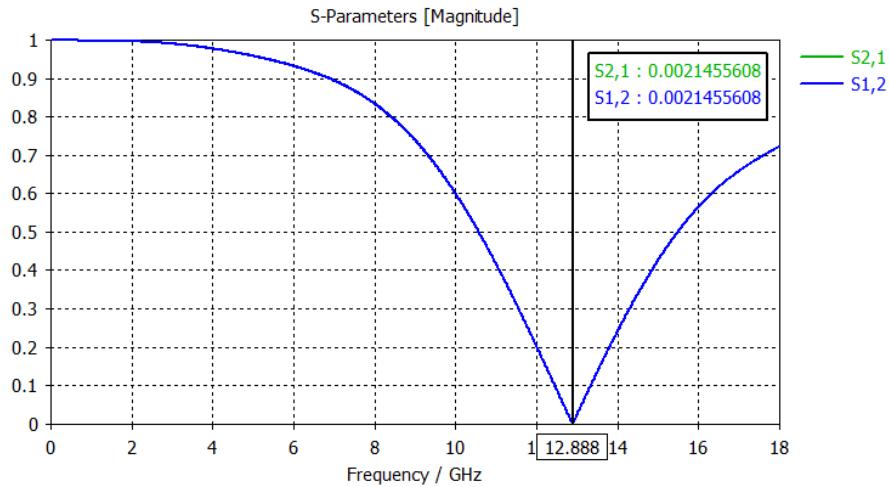
To obtain a sufficiently smooth frequency spectrum of the S-parameters, it is important that all time signals decay to zero before the simulation stops. The simulation will stop automatically when the solver *Accuracy* criterion is met.

The results in which we are most interested in this case are the S-parameters themselves. You may obtain a visualization of these parameters in linear scale by choosing *NT: 1D Results* \Rightarrow *S-Parameters* and selecting *1D Plot: Plot Type* \Rightarrow *Linear*.



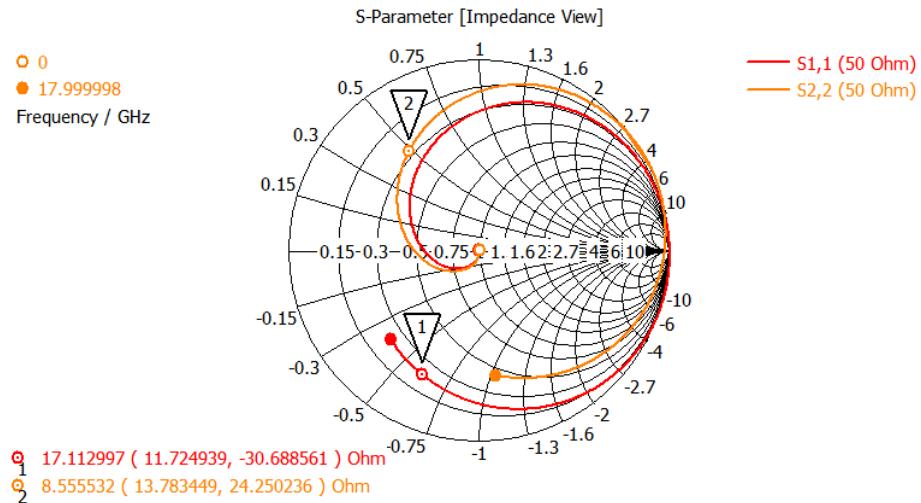
You can change the axis scaling by selecting *1D Plot: Plot Properties* \Rightarrow *Properties* (or *Plot Properties* in the context menu). In addition, you can display and hide an axis marker by toggling *1D Plot: Markers* \Rightarrow *Axis Marker*. The marker can be moved either with the cursor keys (*Left* or *Right*) or by dragging it with the mouse.

The marker can also be adjusted automatically to determine the minimum of the transmission ($S_{1,2}$ or $S_{2,1}$) at about 12.9 GHz by selecting *1D Plot: Markers* \Rightarrow *Axis Marker* \Rightarrow *Move Marker to Minimum*. You can restrict the view to specific curves only by multi-selection in the navigation tree or by choosing *Select curves* via the context menu to show an unambiguous minimum value.



In the same way as above, the S-parameters can be visualized in logarithmic scale (dB) by choosing *NT: 1D Results* \Rightarrow *S-Parameters* and *1D Plot: Plot Type* \Rightarrow *dB* in the context menu. The phase, the real or imaginary part of the selected result can also be visualized.

Furthermore, the S-parameters can be presented in a Z or Y Smith Chart plot (*1D Plot: Plot Type* \Rightarrow *Z Smith Chart* or *1D Plot: Plot Type* \Rightarrow *Y Smith Chart* , respectively).

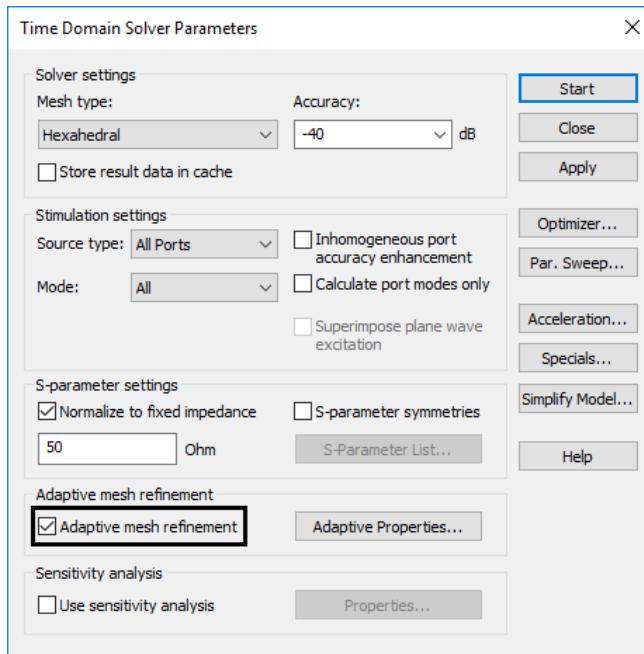


In all 1D plots multiple curve markers can be added by selecting *1D Plot: Markers* \Rightarrow *Curve Markers* \Rightarrow *Add Curve Marker (M)* as shown for example in the Smith Chart view above. The individual markers can be moved along the curve by picking and dragging them with the mouse. You may activate or deactivate the visualization of all markers by choosing *1D Plot: Markers* \Rightarrow *Curve Markers* \Rightarrow *Show Curve Markers* or delete them all with the option *1D Plot: Markers* \Rightarrow *Curve Markers* \Rightarrow *Remove All Curve Markers*.

Adaptive Mesh Refinement

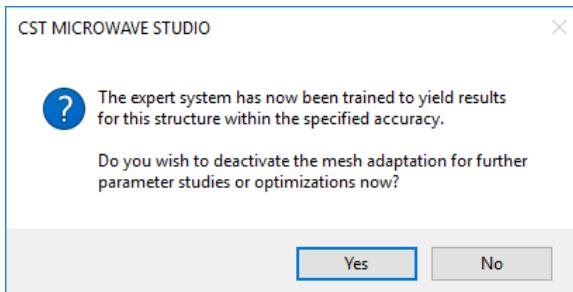
As mentioned above, the mesh resolution influences the results. The expert system-based mesh generator analyzes the geometry and tries to identify the parts that are critical to the electromagnetic behavior of the device. The mesh will then automatically be refined in these regions. However, due to the complexity of electromagnetic problems, this approach may not be able to determine all critical domains in the structure. To circumvent this problem, the transient solver 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. The S-Parameter based stop criterion is activated by default, but it is also possible to use any kind of 0D result template instead, or the two approaches in combination. Please refer to the online help for more detailed information.

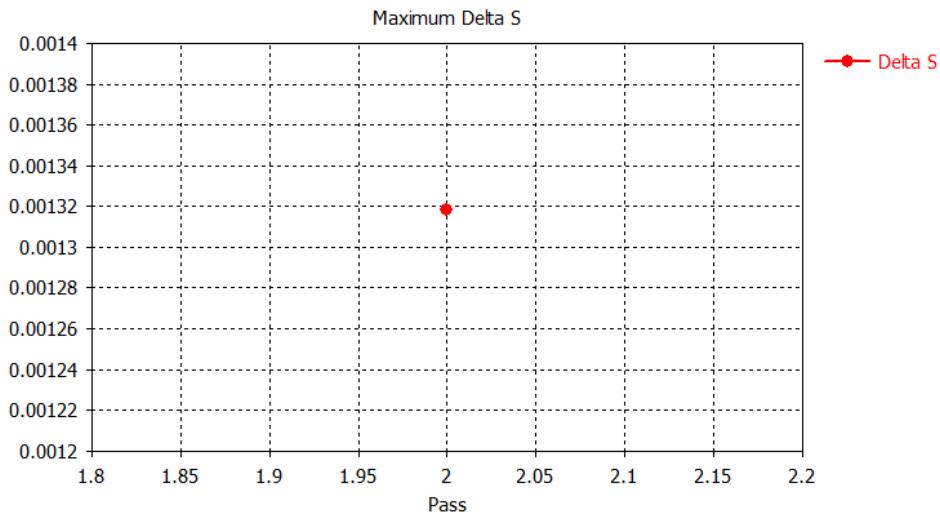
After two passes have been completed, the following dialog box will appear:



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 afterwards. This powerful approach allows you to run the mesh adaptation procedure just once and then perform parametric studies or optimizations on the structure without the need for further mesh refinement passes.

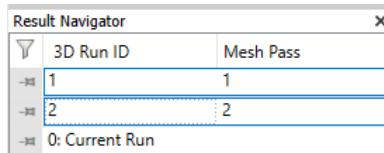
You should now confirm deactivation of the mesh adaptation by clicking the **Yes** button.

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* \Rightarrow *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* \Rightarrow *Adaptive Meshing* \Rightarrow *Delta S*.

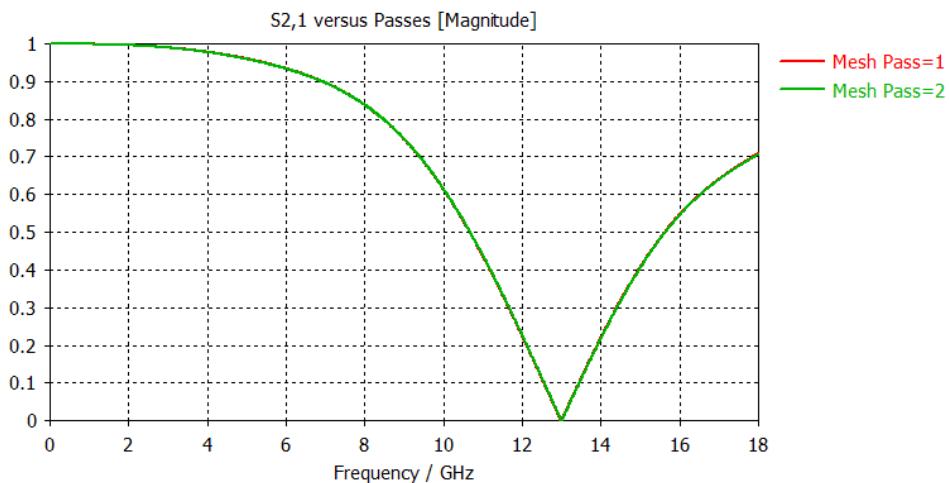


Since the mesh adaptation required only two passes for this example, the *Delta S* curve consists of a single data point only. The result shows that the maximum difference between the S-parameters from both runs is below 1% over the whole frequency range. The mesh adaptation stops automatically when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

The S-parameter results will be automatically stored for the different mesh adaptation runs and can be selected with help of the Result Navigator. If the window is not visible it can be activated by selecting *View: Window \Rightarrow Windows \Rightarrow Result Navigator*.



The convergence of the S-parameter results can be visualized by selecting for example *NT: 1D Results \Rightarrow S-Parameters \Rightarrow S2,1* and activating *1D Plot: Plot Type \Rightarrow Linear*:



You can see that the expert system based mesher provided a good initial mesh for this structure. The convergence of the S-parameters shows only small variations from the results obtained using the expert system generated initial mesh to the converged solution.

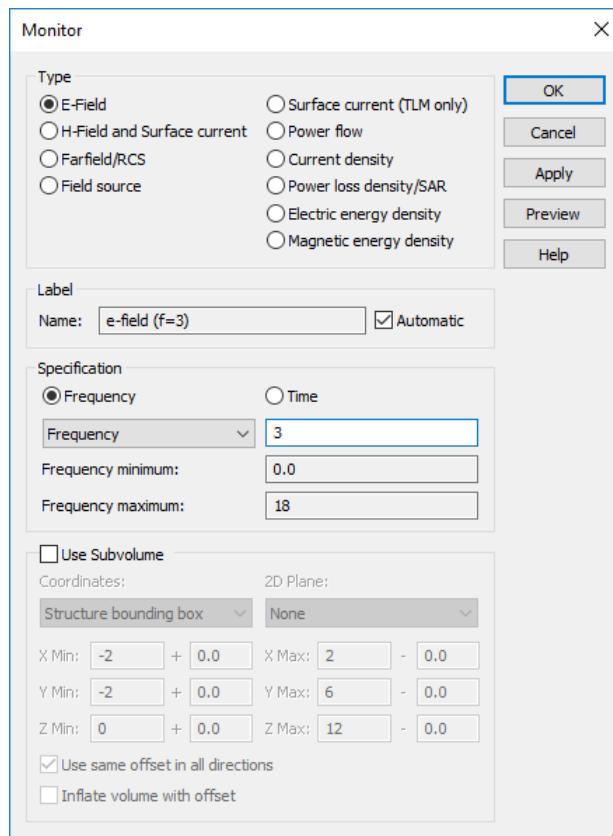
In practice it often proves wise to activate the adaptive mesh refinement to ensure convergence of the results. (This might not be necessary for structures with which you are already familiar when you can use your experience to refine the automatic mesh.)

Analyze the Electromagnetic Field at Various Frequencies

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

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

Monitors can be defined in the dialog box that opens upon choosing *Simulation: Monitors \Rightarrow Field Monitor*. 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:

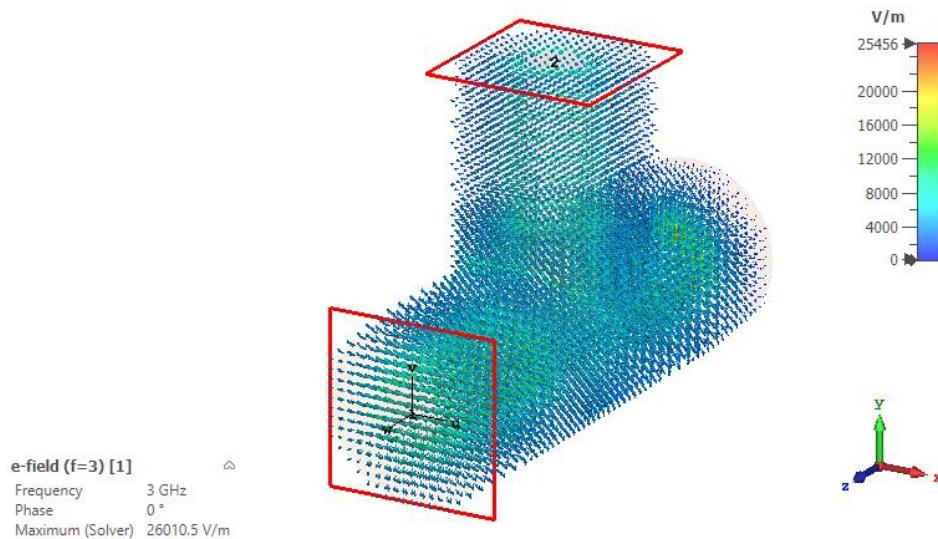
Field type	Frequency / GHz
E-Field	3
E-Field	12.9
H-Field and Surface current	3
H-Field and Surface current	12.9

All defined monitors are listed in the *NT: Field Monitors* folder. Within this folder you may select a particular monitor to reveal its parameters in the main view.

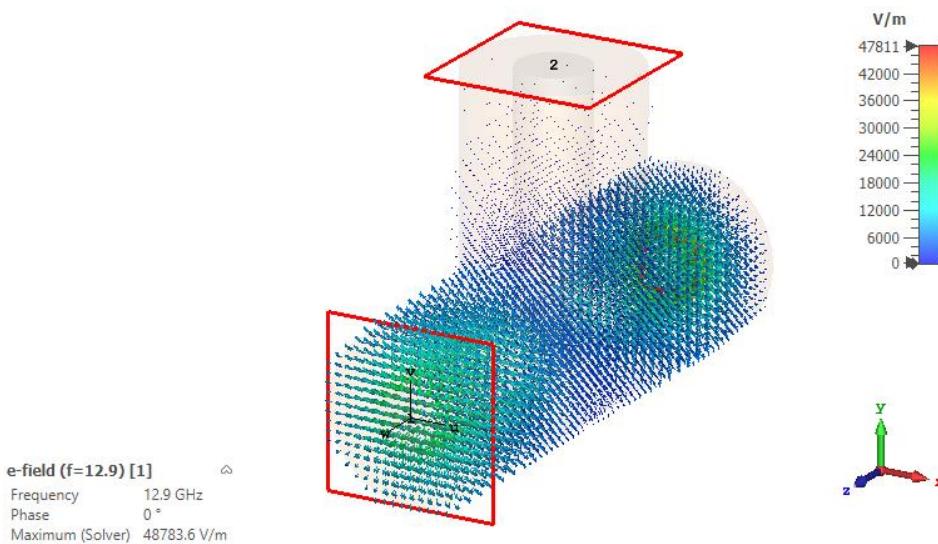
You should now run the simulation again. Without the need to change further solver settings you can press *Home: Simulation* \Rightarrow *Start Simulation*  to directly start the solver run without opening the dialog box. When the simulation finishes, you can visualize the recorded fields by choosing 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 type (*E-Field/H-Field/Surface Current*).

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 square brackets. However, in this loss-free example the second run is not necessary, so you will find that the monitor data for the second run is not available. You can instruct 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 *General* tab in the solver's *Specials* dialog box.

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

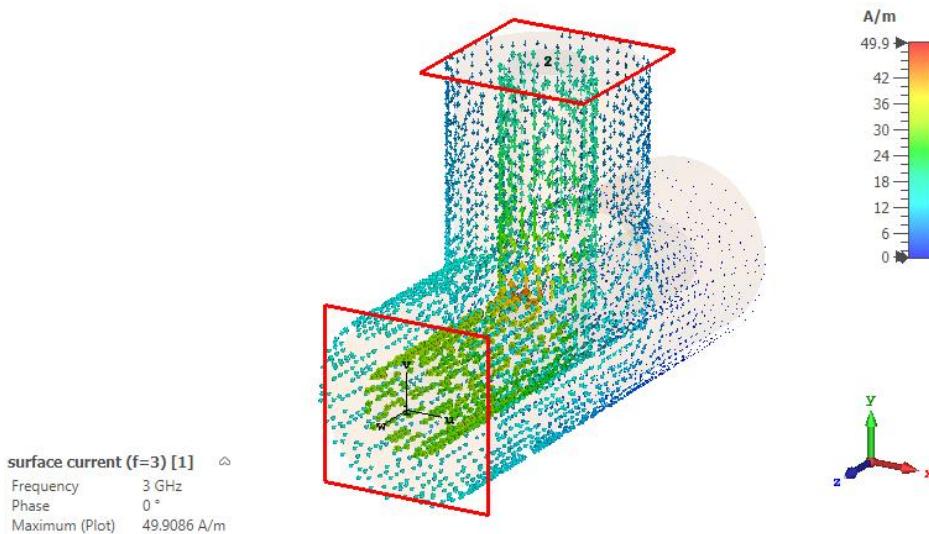


If you select the electric field at 12.9 GHz (*NT: 2D/3D Results* \Rightarrow *E-Field* \Rightarrow *e-field(f=12.9)[1]*) you obtain the following plot:



Please experiment with the various field visualization options for the 3D vector plot (*2D/3D Plot: Plot Properties* or *Plot Properties* from the context menu).

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



You may change the plot options in the plot dialog box by selecting *2D/3D Plot: Plot Properties* . You can obtain a field animation by clicking *2D/3D Plot: Plot Type* *Animate Fields* . Here the phase of the field will be automatically varied between 0 and 360 degrees. You can stop the animation by clicking the button again or just 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 12.9 GHz, you will see that almost no current passes the 90-degree bend of the coaxial cable.

After obtaining a rough overview of the 3D electromagnetic field distribution, you can inspect the fields in more detail by analyzing some cross sectional cuts through the structure. In order to do this, choose an electric or magnetic field (no surface currents) for display and select *2D/3D Plot: Sectional View* *Fields on Plane* . The same plot options are available in the 2D plot mode that you have already used for the port mode visualization. Since the data is 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 by defining *2D/3D Plot: Sectional View* *Cutting Plane Normal* and *Position* or just by toggling the arrow controls shown in the main view.

Due to the limited space, not all plotting options can be explained here. Please refer to the online help for more detailed information and examples.

Parameterization of the Model

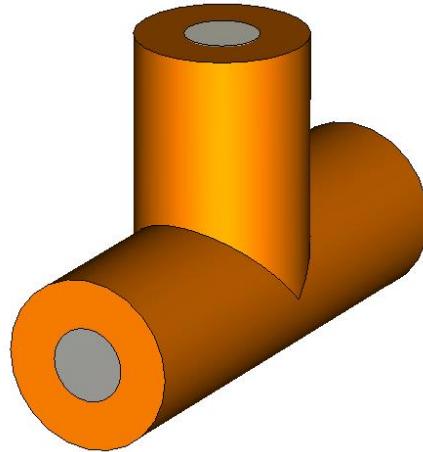
The steps above demonstrated how to enter and analyze a simple structure. However, structures will usually be modified in order to improve their performance. This procedure may be called “design” in contrast to the “analysis” done before.

CST Studio Suite offers a lot of options to parametrically describe the structure in order to easily change its parameters. In general all relevant structural modifications are recorded in the so-called *history list*, which can be opened by choosing *Modeling: Edit* *History List* .

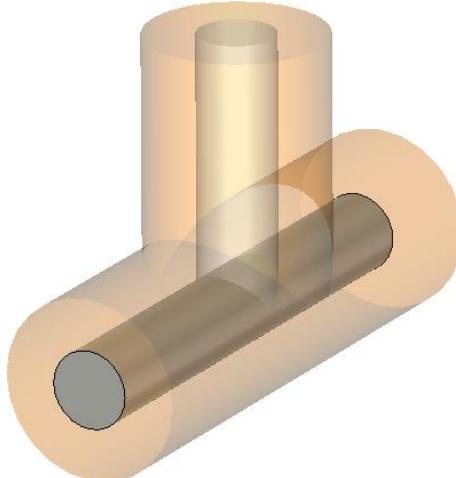
Please refer to the *CST Studio Suite - Getting Started* document for further information on this general option.

However, for simple parameter changes an easier solution is available. 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: Components* folder.

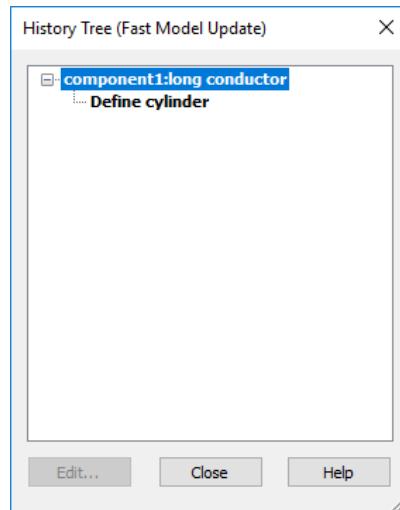
Select all ports by clicking on the *NT: Ports* folder. Then press the right mouse button to choose *Hide All Ports* from the context menu. The structure plot should look like this (the local working coordinate system can be deactivated by selecting *Modeling: WCS* \Rightarrow *Local WCS* ):



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

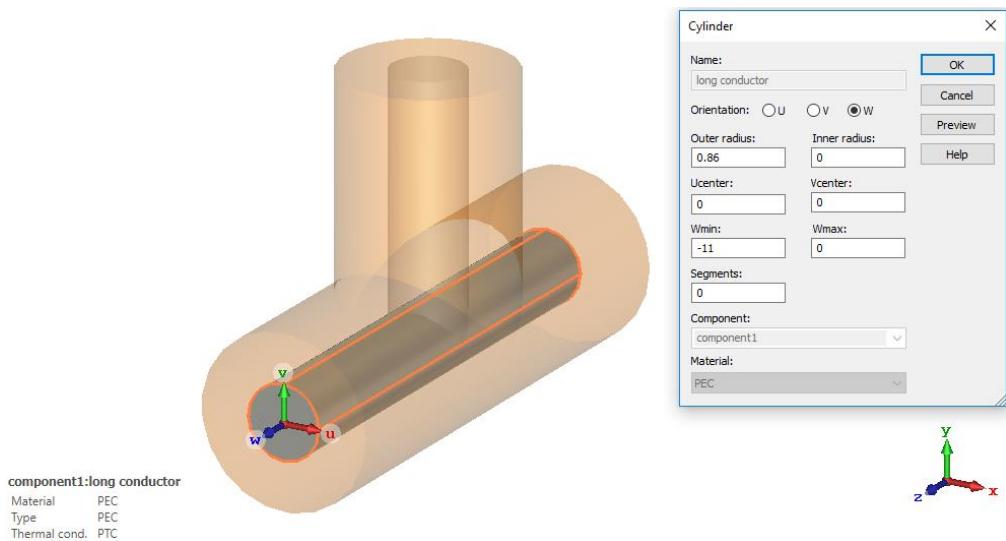


You can now choose *Modeling: Edit* \Rightarrow *Properties*  (or *Properties* from the context menu) which will open a list showing the history of the shape's creation:

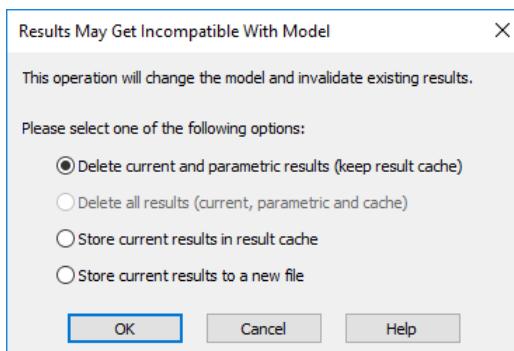


Select the “Define cylinder” operation in the tree folder “component1:long conductor” from the History Tree (see above). The corresponding shape will be highlighted in the main window.

After clicking the *Edit* button in the History Tree dialog, the cylinder creation dialog box will appear showing the parameters of this shape:

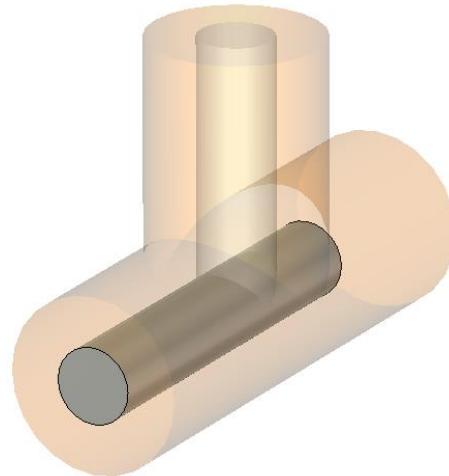


In this dialog box you will find the length of the cylinder ($W_{min} = -11$) as it was previously specified during the shape creation. Change this parameter to a value of -9 and click *OK*. Since you are going to change the structure, the previously calculated results will no longer match the modified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



Here you may specify whether to store the old model with its results in a cache or as a new file, or just to 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 may now dismiss the History Tree dialog box by clicking the *Close* button.

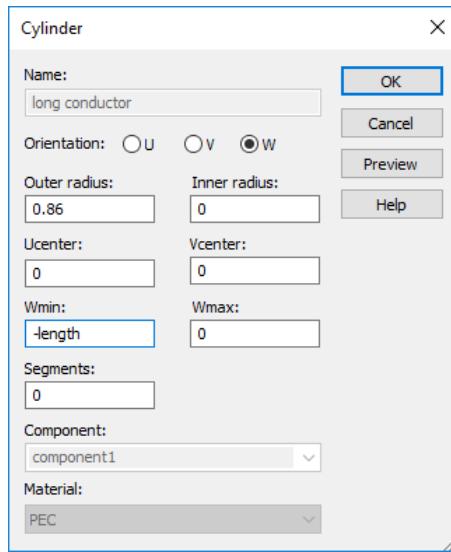
Generally, you can change all geometric parameters of any shape by selecting the shape and editing its properties. This fully parametric structural modeling is one of the most outstanding features of CST Studio Suite.

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

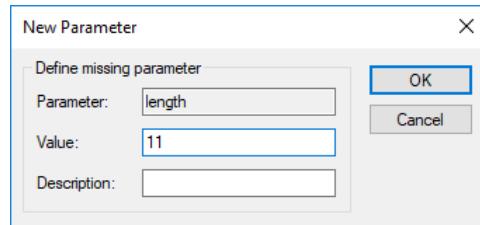
Changes in parameters occasionally alter the topology of the structure so severely that the structure update may fail. In this case, the History List function offers powerful options to circumvent these problems. Please refer to the online documentation or contact technical support for more information.

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. Open the cylinder dialog box again as described above, and then 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:



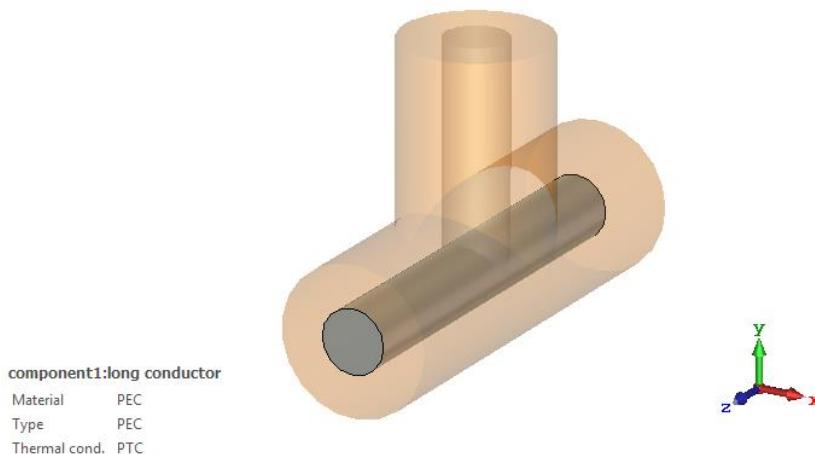
You can now assign a value to the new parameter by entering 11 in the *Value* field. You may also enter some text in the *Description* field so that you can later remember the meaning of the parameter. Click *OK* to create the parameter and update the model. Finally, dismiss the History Tree dialog box by clicking the *Close* button.

All defined parameters will be listed in the Parameter List window, which can be activated by selecting *View: Window \Rightarrow Windows \Rightarrow Parameter List*.

	Name	Expression	Value	Description
->	length	= 11	11	
<new param...>				

You can change the value of this parameter in the *Value* field. Afterwards, a message in the main view informs you to press *Home: Edit \Rightarrow Parametric Update (F7)* :

Some variables have been modified. Press 'Home: Edit->Parametric Update (F7)'

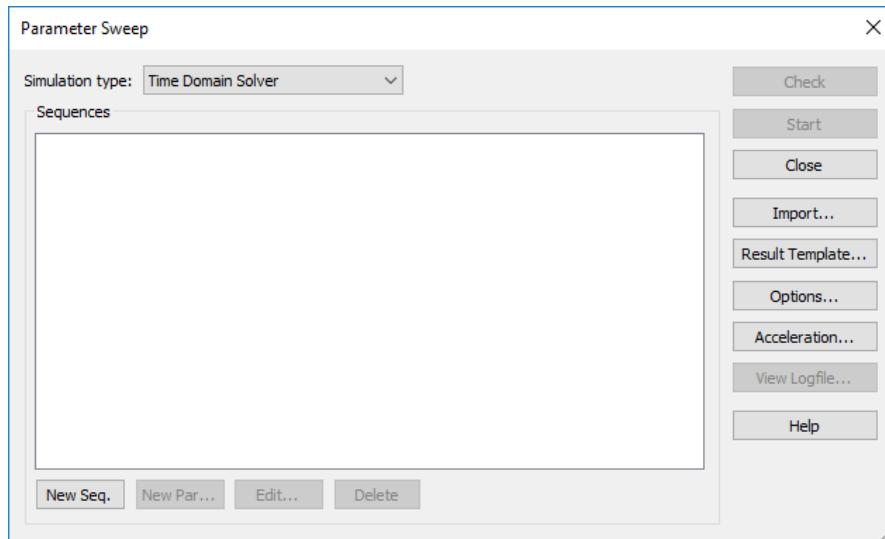


You can also select *Update Parametric Changes* from the context menu which appears when you press the right mouse button in the Parameter List window.

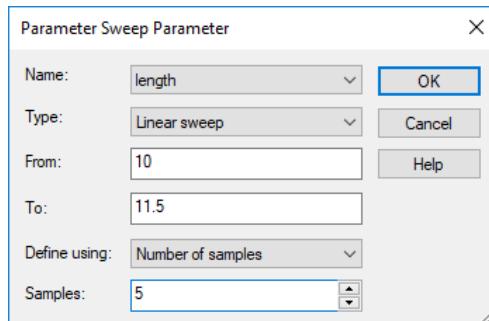
When performing this update operation, the structure will be regenerated according to the current parameter value. You can verify that parameter values between 7 and 11.5 yield a sensible geometry. The function *Home: Edit* \Rightarrow *Parameters* \Rightarrow *Animate Parameter* is also useful in this 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 this result variation is to use the Parameter Sweep tool by selecting *Simulation: Solver* \Rightarrow *Par. Sweep* (or from within the *time domain solver* control dialog box by clicking on the *Par. Sweep* button):

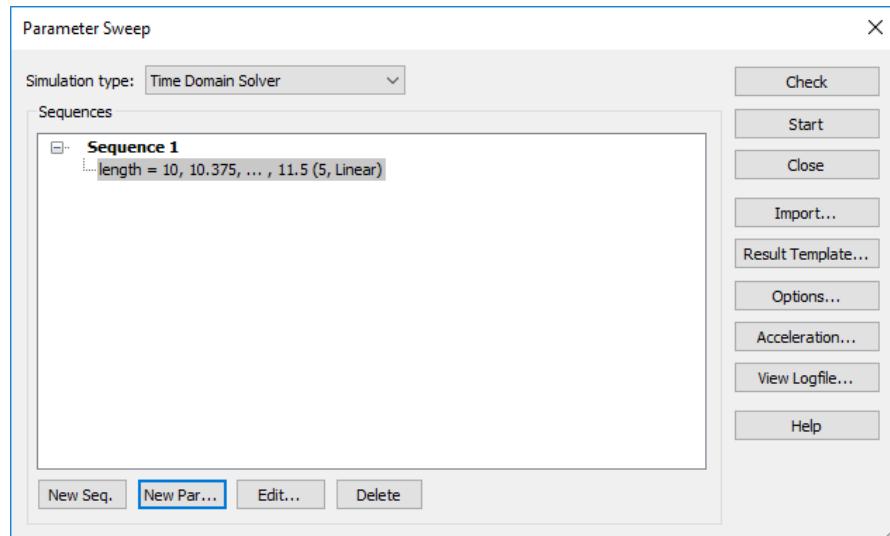


In this dialog box you can specify calculation "Sequences" which will consist of various parameter combinations. To add such a sequence, click the *New Seq.* button now. Then click the *New Par.* button to add a parameter variation to the sequence:



In the resulting dialog box you can select the name of the parameter to vary in the *Name* field. Then you can specify different sweep types to define the sampling of the parameter space (*Linear sweep*, *Logarithmic sweep*, *Arbitrary points*). Depending on this selection the sampling can be defined further, e.g. the linear sweep option allows us to define the lower (*From*) and upper (*To*) bounds for the parameter variation as well as the definition of either the number of samples or the step width.

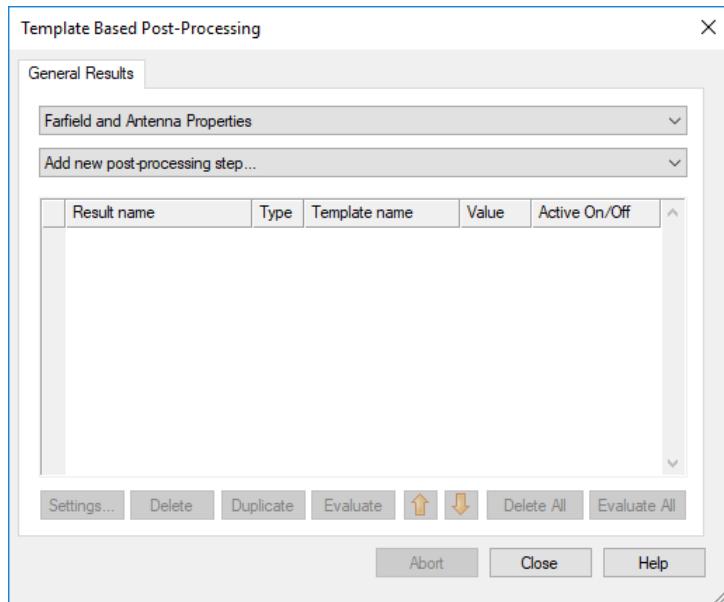
In this example, you should perform a linear sweep from 10 to 11.5 with 5 samples. After you click the *OK* button, the parameter sweep dialog box should look as follows:



Note that you can define an arbitrary number of sequences which each can contain an unlimited number of different parameter combinations.

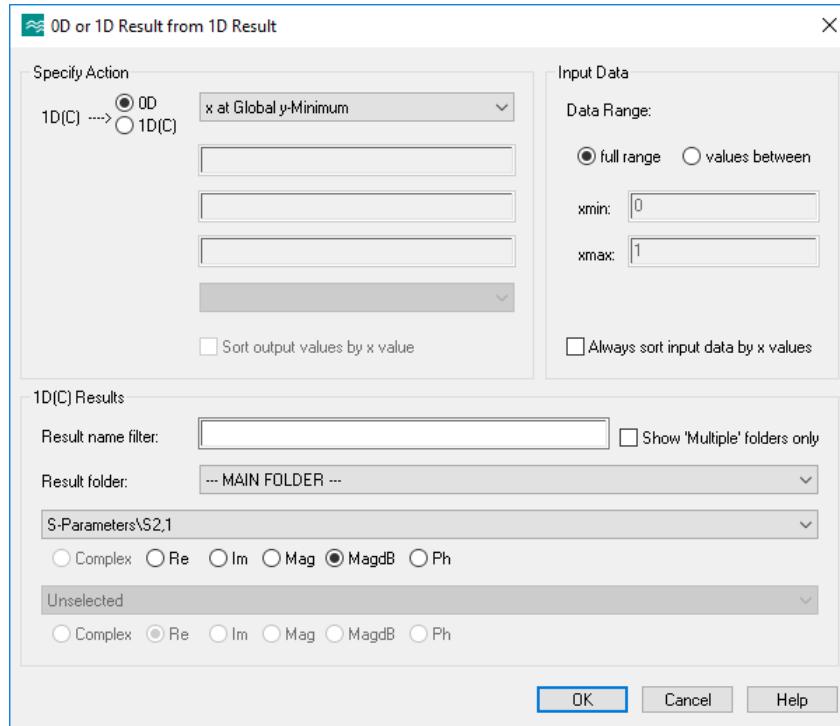
In the next step you have to specify which results you are interested in. With the help of the automated *Parametric Result Storage* it is possible to store any one dimensional result curve parametrically during parameter sweep sequences. A special parametric plot option allows the convenient display of this data. Please refer to the online documentation and the *CST Studio Suite – Getting Started* manual for more information about this convenient functionality.

Besides this general option it is also possible to setup specific *Result Templates*, which allow in addition the definition of various secondary results. Pressing the corresponding button, the global *Template Based Post-Processing* dialog box opens, in which you can define various post-processing steps which will be automatically computed after each simulation run. Please note that this dialog can also be accessed directly by choosing *Post-Processing: Tools* \Rightarrow *Result Templates*



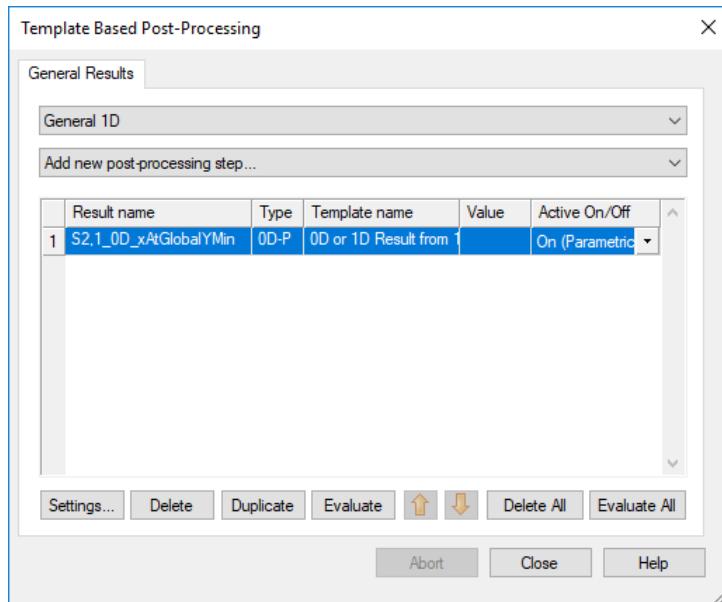
Now we want to investigate how the location of the transmission minimum changes as a function of length. This information can be defined as a single data point result (or so-called 0D result).

Select the *General 1D template list* and choose *0D or 1D Result from 1D Result (Rescale, Derivation, etc.)* to open a dialog box in which you can specify details about the post-processing step:



Since you want to know the location of the curve's (y-) global minimum, after selecting *0D* in the *Specify Action* frame you should choose *x at Global y-Minimum* as the desired result. You can now choose the desired result by selecting the *MagdB* part of the S-parameter result *S-Parameters\S2,1*. Clicking *OK* will complete the definition of the specific post-processing step in this example.

The new result template was added to the list:



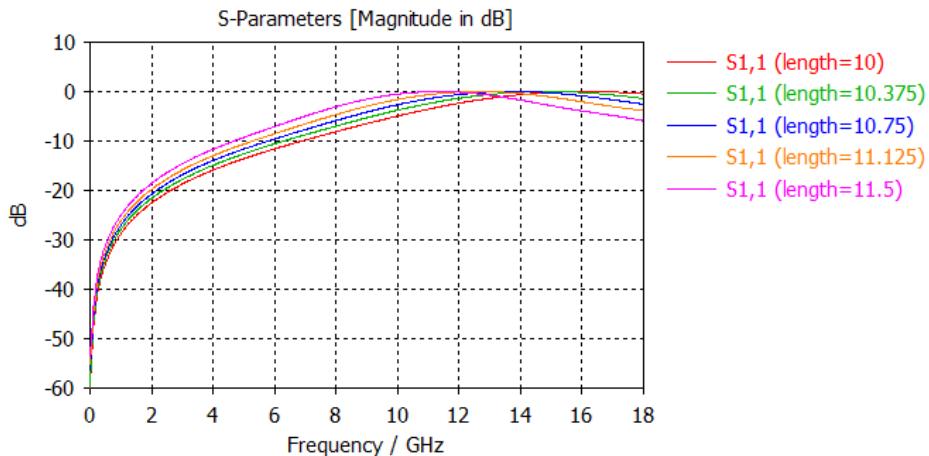
All defined post-processing operations are automatically carried out after every solver run, and the result of each of these steps is stored as a parametric result.

Please now accept the settings by pressing the *Close* button and start the parameter sweep by clicking *Start*.

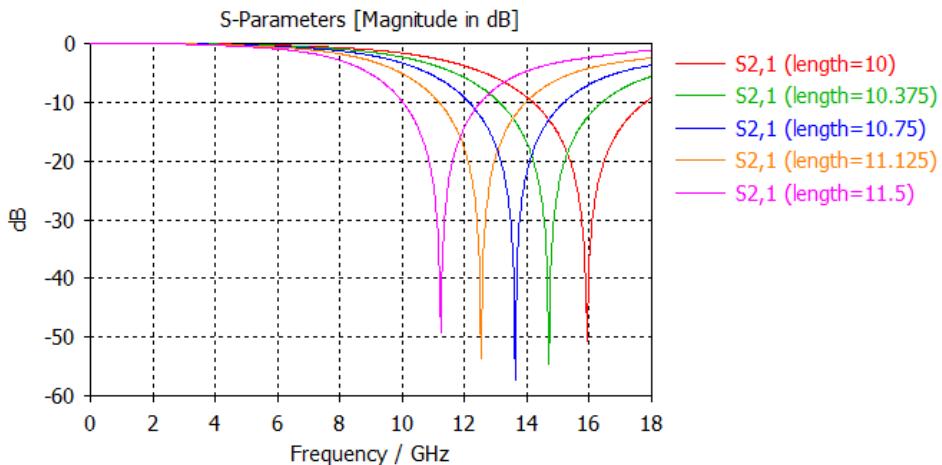
Note that the parameter sweep uses the previously specified solver settings. If you want to change the solver settings (e.g. to activate the adaptive mesh refinement), make sure that the modified settings are stored by clicking *Apply* in the solver control dialog box.

After the solver has finished, close the dialog box by clicking the *Close* button. The navigation tree will contain a new folder called “Tables” where you will find the results of the defined post-processing steps.

But first we can have a look at the basic parametric results of the parameter sweep. Please select the S-parameter result *NT: 1D Results* \Rightarrow *S-Parameters* \Rightarrow *S1,1* and *1D Plot: Plot Type* \Rightarrow *dB* to obtain the following view:



Similarly, you can also plot the magnitude of the transmission coefficient by selecting *NT: 1D Results* \Rightarrow *S-Parameters* \Rightarrow *S2,1* and *1D Plot: Plot Type* \Rightarrow *dB*

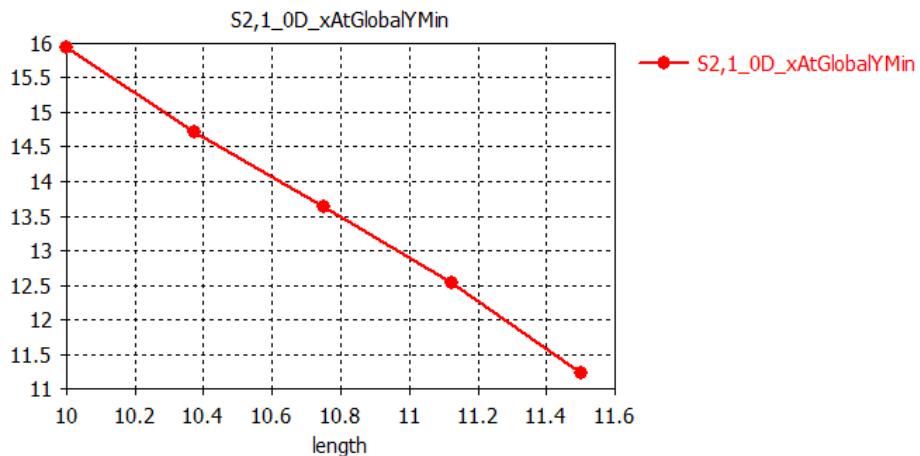


As you see, all available results as well as the last or current result are shown together in one plot. Again, with help of the Result Navigator window it is now possible to easily select any result combination of the previously calculated parameter values. If the window is not visible it can be activated by selecting *View: Window \Rightarrow Windows \Rightarrow Result Navigator*.

Result Navigator		
	3D Run ID	length
-#	1	10
-#	2	10.375
-#	3	10.75
-#	4	11.125
-#	5	11.5
-#	0: Current Run	

Please refer to the online documentation and the *CST Studio Suite – Getting Started* manual for more information about the possibilities to plot parametric result data.

Finally, the result of the previously defined 0D result template can be accessed from the *NT: Tables \Rightarrow 0D Results \Rightarrow S2,1_0D_xAtGlobalYMin* folder:



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

Because of the limited scope of this manual, we have only given a very brief introduction to the many options of storing and displaying parametric data, for example by filtering for parameter range, so please refer to the online documentation and the *CST Studio Suite – Getting Started* manual for more information.

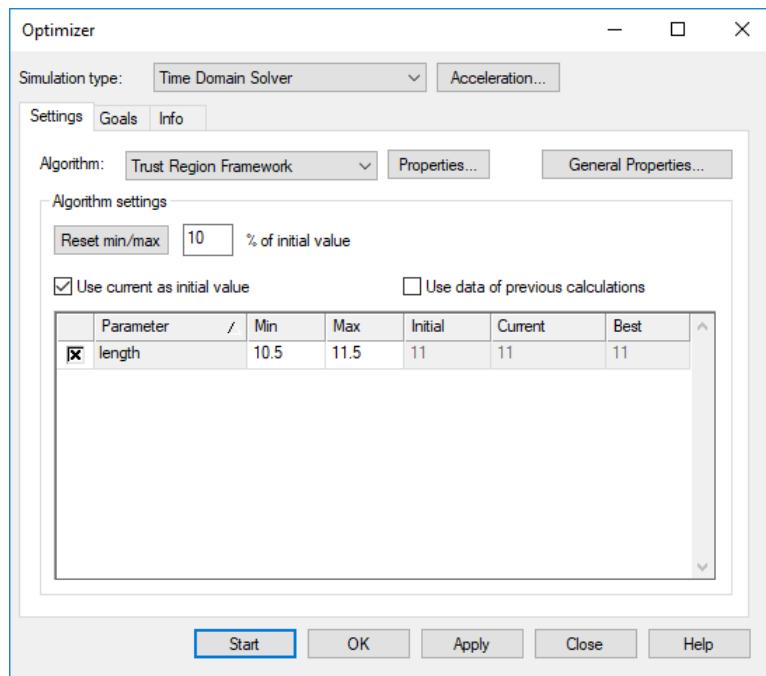
Automatic Optimization of the Structure

Let's now assume that you wish to modify the structure so that the minimum of the transmission S21 is at 13 GHz (which can be achieved somewhere within the parameter range of 10.5 to 11.5 according to the curve above). By measuring the curve (activate the axis marker tool by choosing *1D Plot: Markers \Rightarrow Axis Marker* ), you can check that the desired parameter value is between 10.9 and 11. However, determining the exact parameter value may be a lengthy task which can be performed equally well automatically.

CST Studio Suite offers a powerful built-in optimizer feature for this kind of parametric optimization.

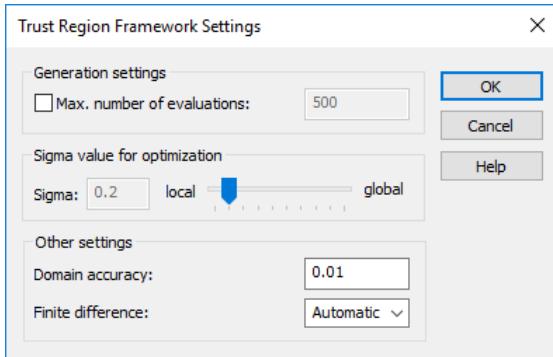
Before you start optimizing this structure, set the length parameter to a value within the valid parameter range (e.g. 11) and update the structure. You must enter the modeler mode (e.g. by clicking on the *Components* folder in the navigation tree) before you can modify the parameters.

To use the optimizer, please select *Simulation: Solver \Rightarrow Optimizer*  to open the optimizer control dialog box (or from within the *time domain solver* control dialog box by clicking on the *Optimizer* button):



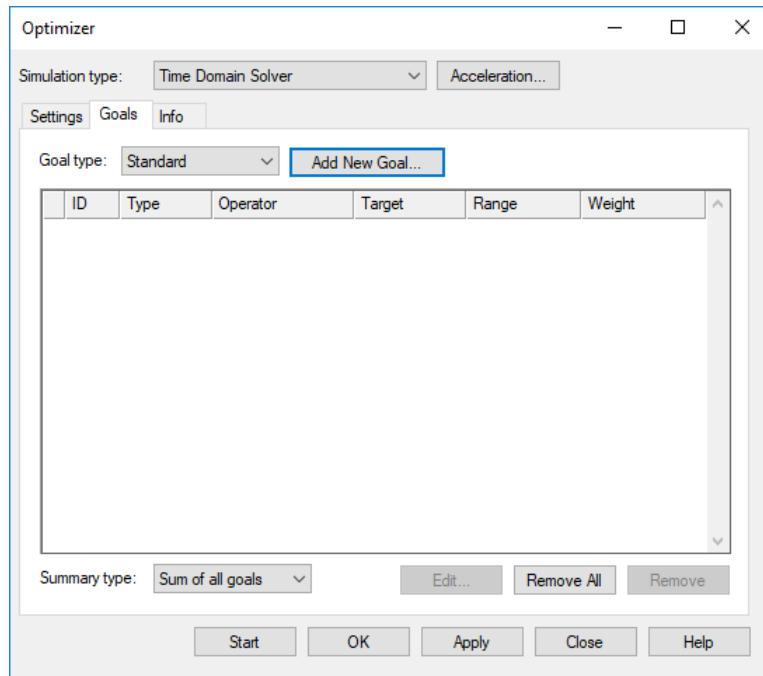
First check the desired parameter(s) for the optimization in the *Settings* tab of the optimization dialog box (the "length" parameter should be checked). Now specify the minimum and maximum values to be allowed for this parameter during the optimization. Enter a parameter range between 10.5 and 11.5.

The default *Trust Region Framework* method will be used for the optimization run. The optimizer settings can be accessed by pressing the *Properties* button:

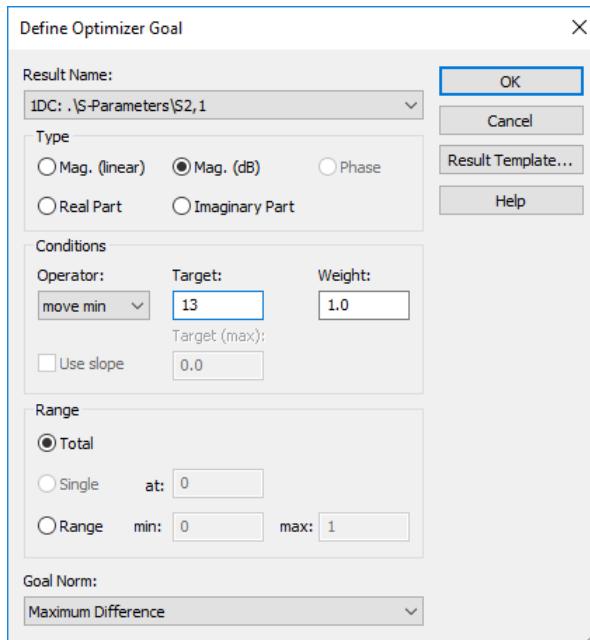


In our example it is sufficient to keep all default settings, so we can directly close the dialog by pressing *OK*. Please refer to the online documentation for more information on these settings and about the different available optimization techniques.

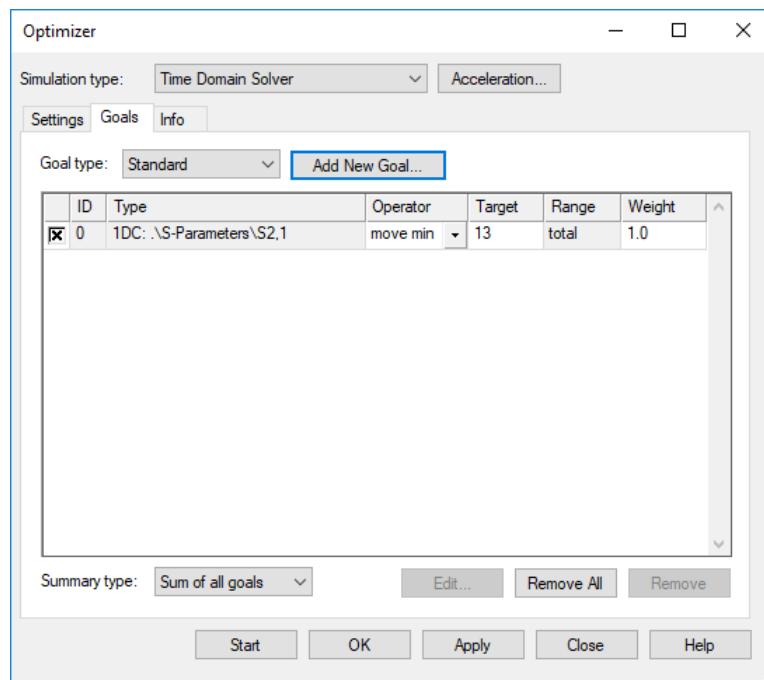
The next step is to specify the optimization goal(s) by clicking on the *Goals* tab.



Goals are based either on previously calculated results or on defined result templates. In this example the target is to move the minimum of the S-parameter S21 to a given frequency. Two goal types are available: the default type *Standard* as well as the type based on *Filter Designer 3D*. Please refer to the online help to find more information about the latter one. We keep the goal type *Standard*. By clicking on the *Add New Goal* button, the following dialog box should appear, where you can select the desired complex S-parameter *S2,1* as *Mag.(dB)*:

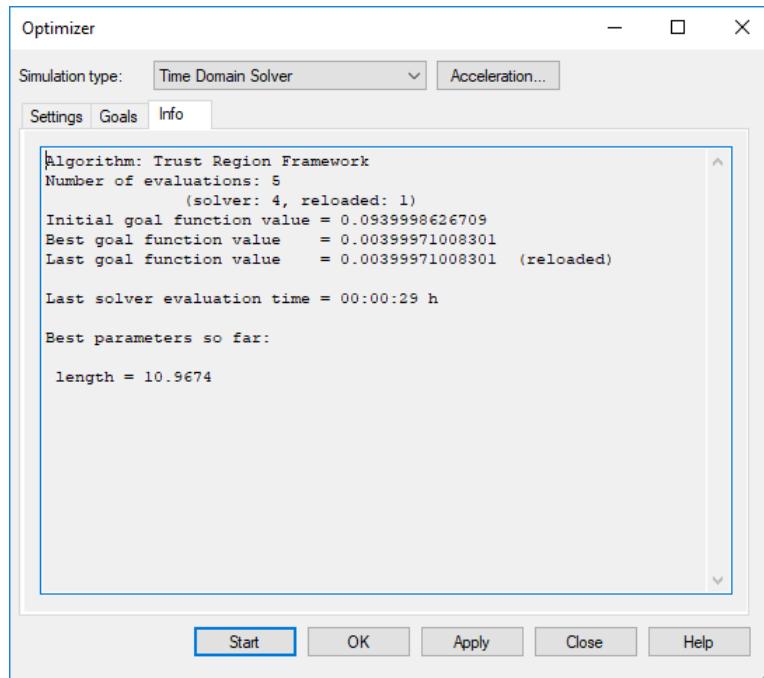


Now specify the goal for the previously specified S-parameter data. Since you want to move the minimum of S21 dB in this example, you should select the *move min* operator in the *Conditions* frame. Afterwards, set the *Target* frequency to which the minimum should be moved to 13 GHz. If more than one minimum exists in the S-parameter data, you can limit the frequency range in which the minimum will be searched for in the *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:



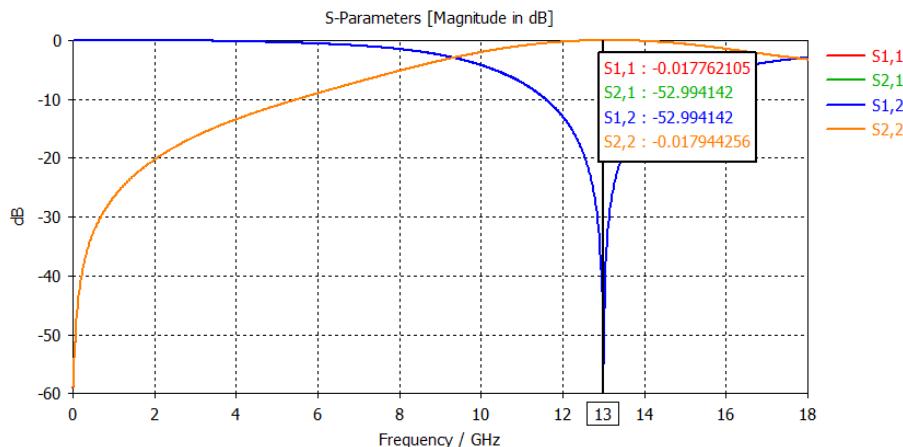
Since you have now specified optimization parameters and goals, the next step is to start the optimization by clicking the *Start* button. The optimizer will show the progress of the optimization in an output window in the *Info* tab which will be activated automatically.

When the optimization has finished, you should confirm that 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 four 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 = 10.9674) and should obtain the following picture (you can activate the axis marker tool by choosing *1D Plot: Markers* \Rightarrow *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 0D result template *S2,1_0D_xAtGlobalYMin* to be equal to the desired resonance frequency of 13 GHz.

Comparison of Time and Frequency Domain Solver Results

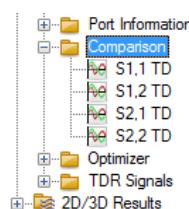
Thus far all explanations have focused on the transient solver. In the next steps you will compare the results of this time domain solver based on a hexahedral mesh with the frequency domain solver using a curved tetrahedral mesh. Since these two simulation methods are based on different techniques, this comparison allows you to judge the accuracy of the results. Although the time domain solver is much faster for this and many other examples, the frequency domain solver may be the better choice for lower frequency problems or resonant devices such as filters. In such cases the primary simulation and optimization should be performed using the frequency domain solver whereas the final verification can then be done using the time domain solver. The

seamless combination of these different techniques in a homogeneous environment is another outstanding feature of CST Studio Suite.

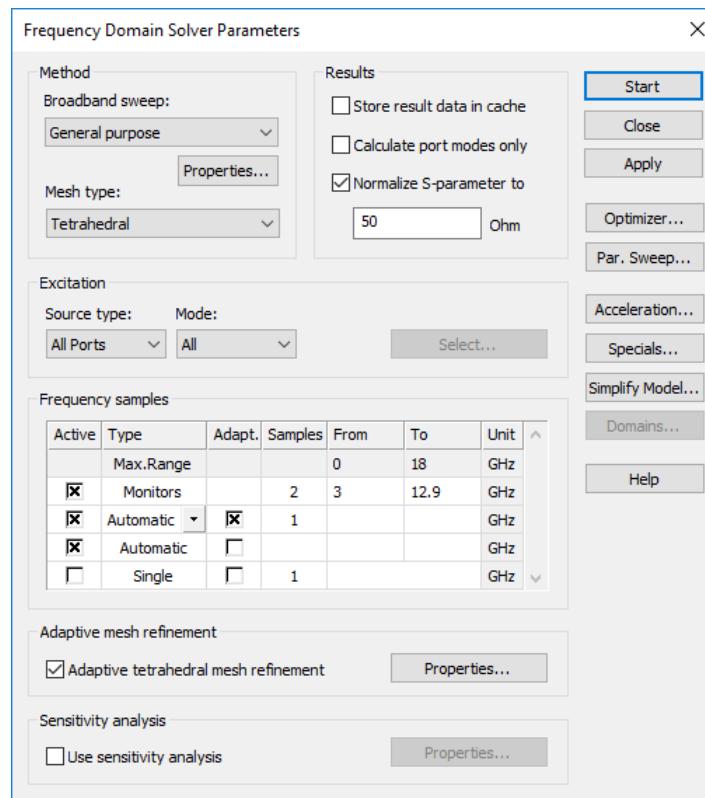
Before you recalculate the S-parameters using the frequency domain solver, you should first copy the results from the time domain solver into a new folder for easier comparison afterwards.

Select the *1D Results* folder in the navigation tree, and choose *New Tree Folder* from the context 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 *NT: 1D Results* \Rightarrow *S-Parameters* folder and choose *Home: Clipboard* \Rightarrow *Copy* . Finally, select the newly created *NT: 1D Results* \Rightarrow *Comparison* folder and choose *Home: Clipboard* \Rightarrow *Paste* .

Note that the copied result curves will neither be deleted nor changed when parameters are changed or S-parameters are recalculated. For organizational purposes you should now click on each of the new curve entries in the *NT: 1D Results* \Rightarrow *Comparison* folder, choose *Rename* from the context menu (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 time domain solver. The navigation tree should finally look as follows:



Once you have saved the time domain solver results for later comparison, you can switch the currently active solver by selecting *Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *Frequency Domain Solver* . Now you can simply open the frequency domain solver dialog box by clicking on the solver icon: *Home: Simulation* \Rightarrow *Setup Solver* :

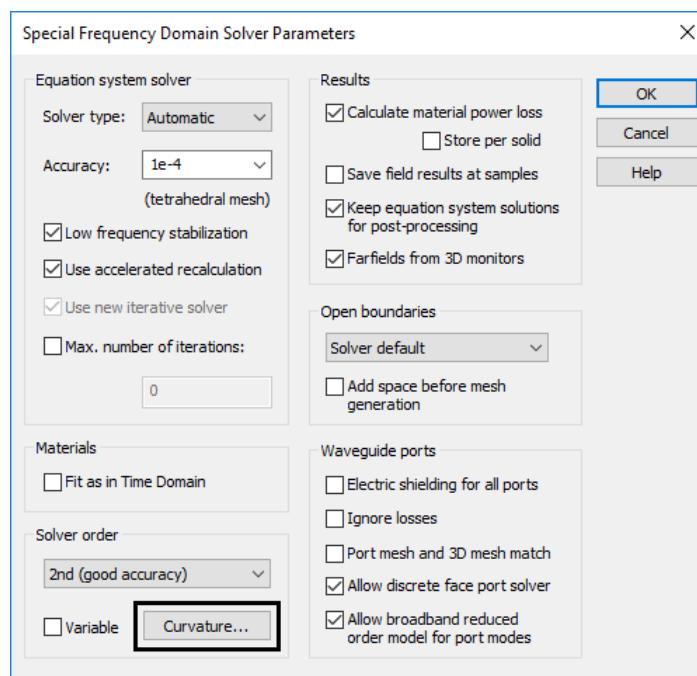


By default, the frequency domain solver uses a tetrahedral mesh, automatic mesh adaptation, and full S-parameter matrix calculation, so you do not need to change anything here. Please make sure that the *Normalize to fixed impedance* check button is activated and that the corresponding value is set to 50 Ohm.

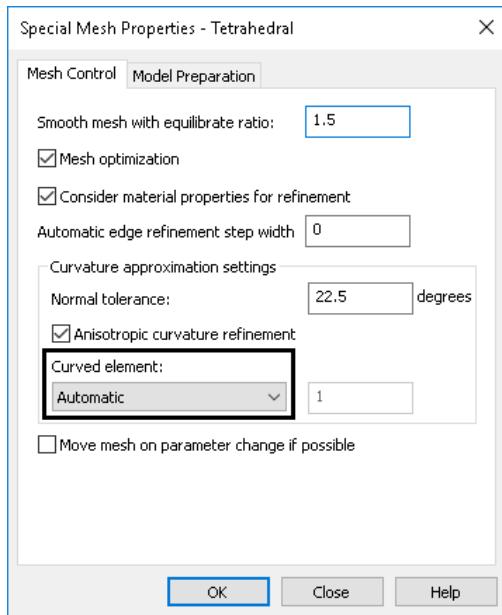
Most of the structure's surfaces are curved. It is therefore recommended to use the curved tetrahedral mesh to obtain more accurate results, and this is the default for newly created projects. Curved elements provide a better approximation of the geometry than linear elements. (The latter are actually a special case of the former: linear elements are "curved" with a curved element order of one.)

For the default "Second" order solver elements, a curved element order equal to two or three is recommended. For higher solver order it is advisable to further increase the curvature order. The curvature order of the elements is by default chosen automatically so that it fits the solver order of the solver selected in *Home: Simulation \Rightarrow Setup Solver*, so in this case no change needs to be applied.

To verify that the curved element order is set to *Automatic*, the special tetrahedral mesh properties dialog needs to be opened. This can generally be accessed by closing the solver dialog and choosing *Home: Mesh \Rightarrow Global Properties* \Rightarrow *Tetrahedral* and the *Specials* button therein. However, the solver specials dialog, accessed by the *Specials* button in the frequency domain solver parameters dialog, provides a direct link to this setting:



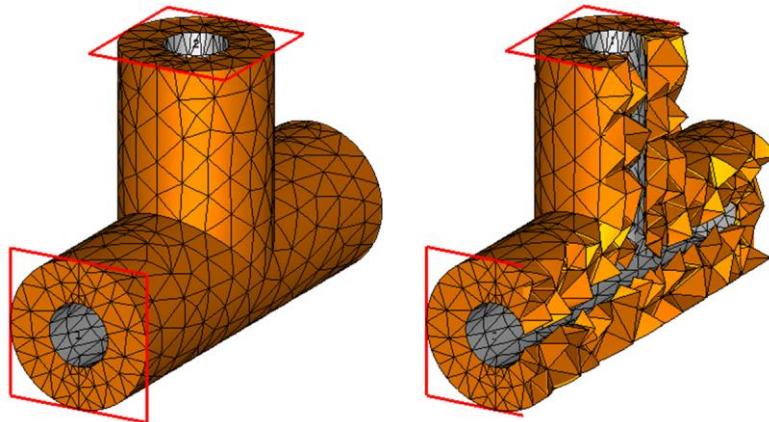
The settings for the solver order (first to third order, possibly variable) and a button *Curvature* are available in the *Solver order* frame. Please follow the *Curvature* link to the special mesh properties to verify that the choice for the *Curved elements* is *Automatic*:



Confirm the settings and close the special mesh properties and the solver special dialog boxes by pressing **OK** to return to the frequency domain solver parameters dialog.

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 (*Home: Mesh \Rightarrow Mesh View*). By selecting *View: Sectional View \Rightarrow Cutting Plane* (*Shift+C*) a cut of the meshed structure is shown:



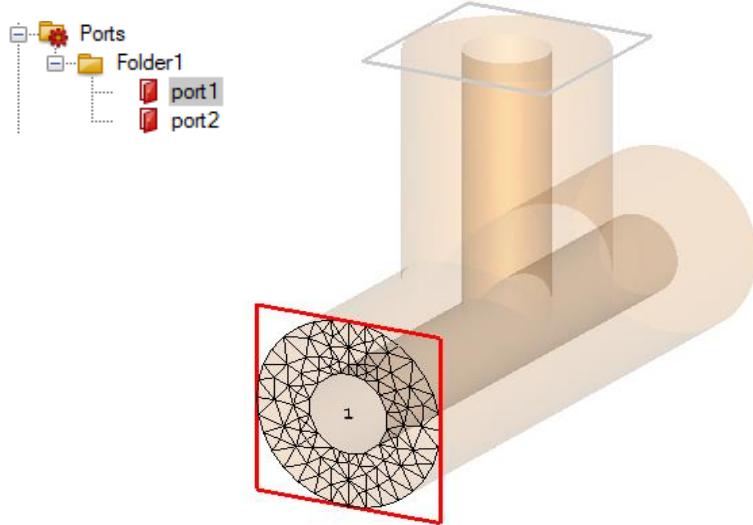
The ports can be made visible again by clicking on the *NT: Ports* folder and then selecting *Show All Ports* from the context menu. The solver first performs a mesh adaptation at the ports before the mesh inside the structure is adapted at the highest frequency of interest in the second step.

The mesh adaptation frequency can be set to other values if necessary and more than one mesh adaptation frequency sample can be defined. Please note that for the sake of accuracy it is important to have a mesh adaptation sample at some frequency where power is delivered into the structure, for instance in the pass band of a filter. If the mesh adaptation frequency is defined at a frequency where most of the input power is reflected, the error indicator will not "see" the possibly more important interior parts of the structure, and the mesh refinement will focus on the terminals of the structure rather than on the inner regions.

The solver may therefore stop the adaptive mesh refinement if the minimum input reflection of all S-parameters at the present adaptation frequency seems to be too high. It attempts to insert new

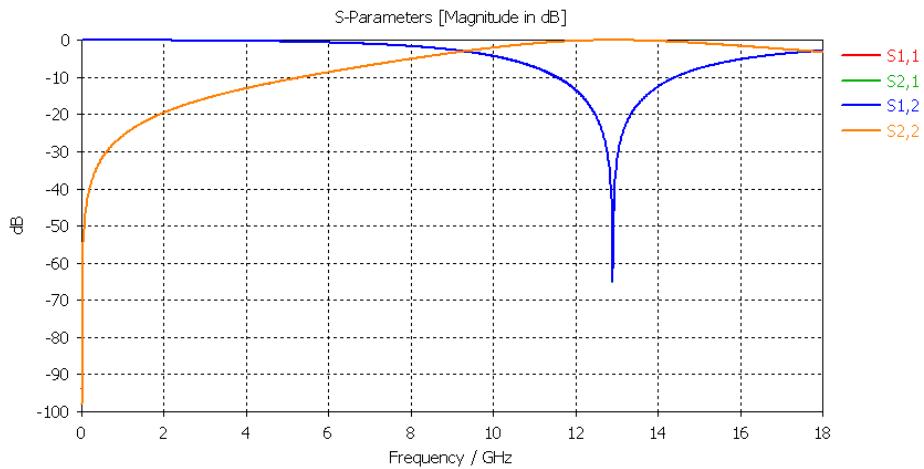
adaptation frequencies with a trial-and-error approach that covers the whole frequency range, starting with monitor frequency samples, if any. The number of attempts to "move" the automatic adaptation frequency samples is limited. If no suitable frequency is found, the adaptive mesh refinement will continue at the first adaptation frequency again. In this case, please choose and define a suitable constant adaptation frequency in the *Frequency samples* frame of the *Frequency Domain Solver Parameters* dialog.

Now click on *NT: Ports \Rightarrow Port 1* in the navigation tree to view the port mesh:

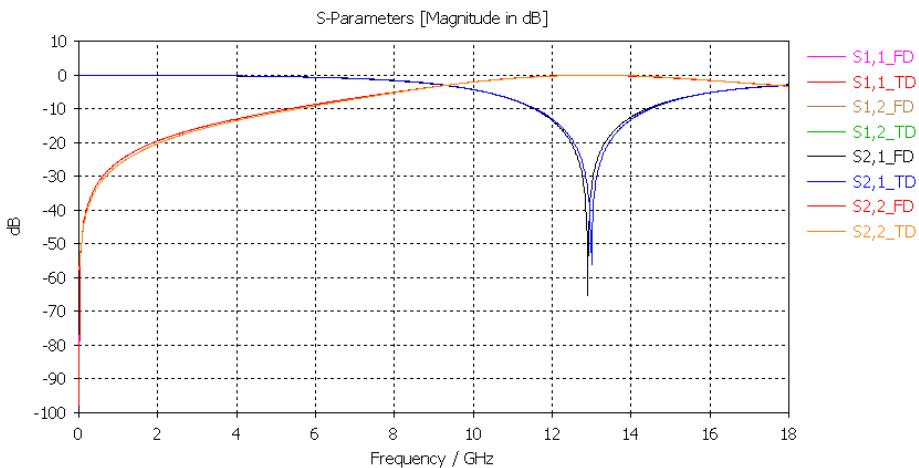


Once the mesh adaptation has converged, the solver calculates the S-parameters as a function of frequency by using its fast sweep capability.

When the solver has finished, you can view the results in logarithmic scale (dB) by choosing *NT: 1D Results \Rightarrow S-Parameters* and *1D Plot: Plot Type \Rightarrow dB*:



As you can see, the results are quite similar to the results previously obtained from the time domain solver. A direct comparison can be made by copying and pasting the frequency domain solver results to the *NT: 1D Results \Rightarrow Comparison* folder as described above. You can add an appendix "FD" to the curve names of the new frequency domain solver results:



As you can see, the results from the time domain solver using a hexahedral mesh and the frequency domain solver using a tetrahedral mesh are in good agreement.

Summary

This example gave you an overview of the key concepts of a high frequency simulation in CST Studio Suite. You should now have a basic idea of how to do the following:

1. Model structures by using the solid modeler
2. Specify the solver parameters, check the mesh and start a time domain 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 time domain solver and the frequency domain solver

If you are familiar with all these topics, you have a very good starting point for improving your usage of CST Studio Suite.

For more information on a particular topic, we recommend that you browse through the online help system which can be opened by selecting *File: Help \Rightarrow Help Contents – Get help using CST Studio Suite*. If you have any further questions or remarks, please do not hesitate to contact your technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Please ask your support center for details.

Chapter 3 – Solver Overview

Which Solver to Use

Since in the previous example we have mainly focused on the transient solver, and to a lesser extent on the general purpose frequency domain solver, it is time to clarify which solver best fits which application. 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 below lists a few typical applications along with the solvers that are most frequently used for solving that particular type of 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, this table should be used as a guideline rather than a rule for which solver to use.

For further guidance CST Studio Suite offers a configuration wizard, which suggests the best suited solver types as well as automatically predefines simulation settings for your specific application. As described in the **Create a New Project** chapter, these so-called *Project Templates* can be defined by selecting *File* \Rightarrow *New and Recent* \Rightarrow *New Project from Template* \Rightarrow *New Template*. Please find more detailed information in the *CST Studio Suite – Getting Started* manual.

Application	Solver Type(s)
Connectors (coaxial, multi-pin)	Transient
Strip lines (microstrip, coplanar lines)	Transient, Frequency Domain, Multilayer
Stripline circuits	Transient, Frequency Domain, Multilayer
Cross-talk calculations	Transient
Printed circuit boards	Transient, Multilayer, Partial RLC
Digital circuit simulation	Transient
Packaging problems	Transient, Frequency Domain, Multilayer, Partial RLC
Network parameter (SPICE) extraction	Transient, Frequency Domain, Partial RLC
Nonlinear diode applications	Transient
EMI problems	Transient, TLM
Radiation problems	Transient, Integral equation, TLM
Shielding (irradiation) problems	Transient, TLM
Monopole, dipole and multipole antennas	Transient
Patch antennas	Transient, Frequency Domain
Conformal antennas	Transient, Frequency Domain
Helical and spiral antennas	Transient, Integral equation
Antenna arrays	Transient

Application	Solver Type(s)
Waveguides (hollow, dielectric, coaxial)	Transient
Transmission line networks	Transient
Optical wave guides	Transient
Optical couplers	Transient
Optical diplexers and filters	Transient, Frequency Domain
Filters and diplexers	Frequency Domain, Transient
Cavities, resonator design	Eigenmode
Traveling wave structures	Eigenmode
Periodic problems (frequency selective surfaces, periodic band gap structures)	Frequency Domain, Eigenmode, Transient
Periodic problem with nonzero phase shift	Frequency Domain, Eigenmode
Periodic problems with non-rectangular lattice (unit cell)	Frequency Domain with Tetrahedral mesh
Antenna placement	Integral equation, Transient
Antenna placement (electrically large)	Integral equation, Asymptotic
RCS (electrically large)	Integral equation, Asymptotic, Transient
Electrically large antennas	Integral equation, Transient

Please note that the application range of the transient analysis can be extended significantly for more resonant devices by applying some advanced digital signal processing techniques rather than simply using a Discrete Fourier Transform. CST Studio Suite 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 degrades for strongly resonant structures or if the device operates at very low frequencies. In such cases the 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, in particular with the reduced order model sweep. On the other hand, the simulation time of the frequency domain solver increases more rapidly with an increase in the number of mesh cells than the simulation time of the transient solver.

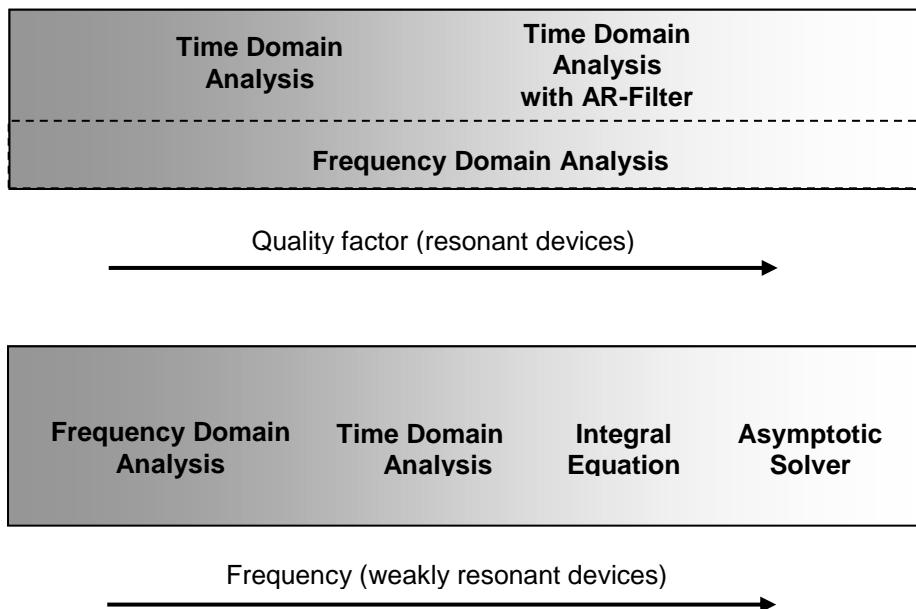
Besides these general considerations, there are also some applications that require the selection of a particular solver since the corresponding electromagnetic problem can be solved only by using the corresponding method:

- Structures containing nonlinear materials or diodes:** The frequency domain solver cannot handle nonlinearities. Therefore, the transient 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 tetrahedrons for the frequency domain solver is in the order of several million, the time domain solvers or the integral equation solver should be used. For electrically very large problems, using the integral equation solver or even the asymptotic solver may be the best option.
- Periodic structures with non-zero 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 the electrical phase angle between the boundary planes and the geometrical angle of incidence are not identical. The frequency domain solver in combination with a tetrahedral mesh also offers a

special Unit Cell feature which allows the simulation of periodic structures with a non-rectangular lattice.

4. **Planar structures:** Predominantly planar structures such as microstrip filters and printed circuit boards can be solved by general purpose 3D solvers (time or frequency domain). However, in order to ideally exploit the planar property of the structure the multilayer solver can be applied to these examples.

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



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

Furthermore, it should be mentioned, that the solvers can be combined with one another to give hybrid solution capability for structures or systems which do not fit neatly into one category, e.g. a small resonant antenna on a large structure such as a motor vehicle.

Time Domain Solver

In CST Studio Suite two high frequency time domain solvers are available, which both work on hexahedral meshes. One is based on the Finite Integration Technique (FIT), just called *Transient solver*, the second one is based on the Transmission-Line Method (TLM) and is referred to as *TLM solver*. Both time domain solvers are launched via the time domain solver dialog box *Simulation: Solver* \Rightarrow *Setup Solver* \Rightarrow *Time Domain Solver*  and can be distinguished in the *Mesh type* dropdown list by either specifying *Hexahedral* to choose the transient FIT solver or *Hexahedral TLM* for the TLM solver.

Transient Solver

The Transient solver applies advanced numerical techniques like the Perfect Boundary Approximation (PBA) in combination with the Thin Sheet Technique (TST) to allow accurate modeling of small and curved structures without the need for an extreme refinement of the mesh at these locations. This allows a very memory efficient computation together with a robust hexahedral meshing to successfully simulate extremely complex structures.

Features like AR-Filtering or S-Parameter symmetries and reciprocity help to increase the performance of this solver. Furthermore, the simulation becomes even more efficient when applying hardware acceleration like GPU or MPI computing as described later on in the general chapter *Acceleration Features*.

The usage of the Transient solver is explained in detail in **Chapter 2 - Simulation Workflow**, showing the basic construction steps of a coaxial connector model as well as the solver setup and some post-processing steps. Therefore, in the next section the TLM solver will be discussed in more detail:

TLM Solver

The TLM solver has many of the features of the Transient solver and shares a similar application range. This section describes the differences in **model definition** between the TLM and the Transient solver:

Materials

Most of the materials which are supported by the Transient solver are also available for the TLM solver.

The unavailable materials are:

- Corrugated wall and frequency dependent Ohmic sheet
- Temperature dependent and spatially varying material

In return the TLM solver is able to model special material types and compact models which will be discussed on the following pages.

Thin panel and coated metal

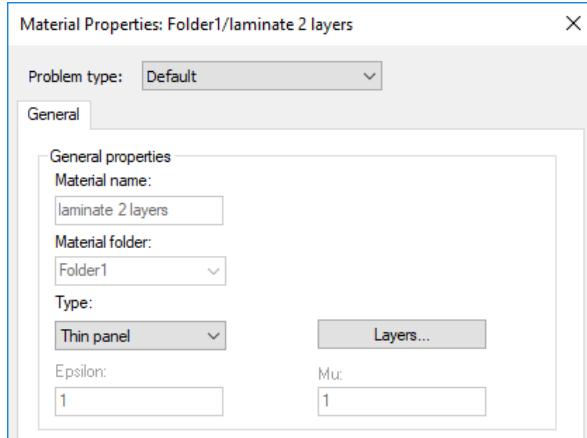
The TLM solver is able to model thin sheets of *Anisotropic* or *Normal* material without needing to add any mesh cells in the thickness of the sheet. This can lead to a significant improvement in simulation time for devices containing such thin sheets, for example radomes or carbon fiber composite surfaces on aircraft.

To take advantage of this feature in modeling penetrable thin objects, define a new material of type *Thin panel* and attach to it any number of layers of *Normal*, *Anisotropic* or *Perforations* material. *Perforations* material is used to define wire meshes that can be embedded in carbon fiber composites for lightning protection, or air ventilation panels.

Alternatively, scattering parameters defining the reflection and transmission for a sheet material can be imported directly into the *Thin Panel* material dialog.

This *Thin Panel* material can be attached to any sheet object. If the layers of the material are asymmetrically defined, it is necessary to attach *Local Solid Coordinates* to each object made of the *Thin panel* material (*Local Solid Coordinates* \Rightarrow *Attach Active WCS* from the context menu).

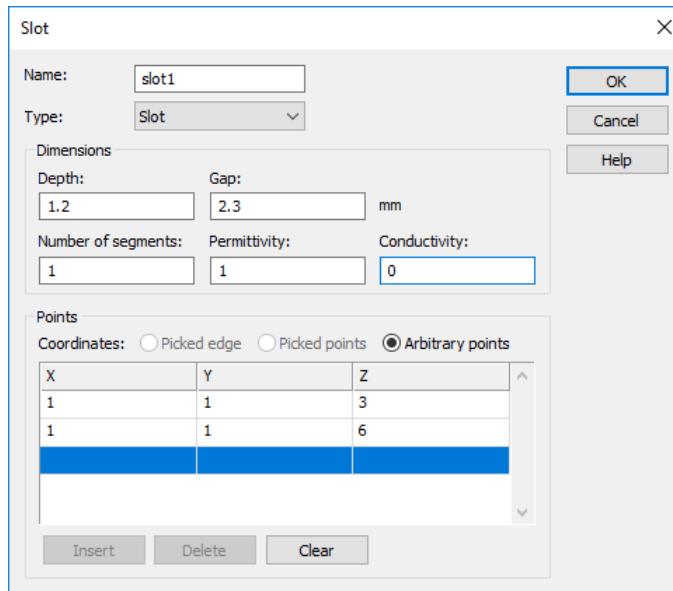
The *W* direction of the *Local Solid Coordinates* is then used to indicate the direction of layer stackup, and the *U* direction is used to indicate the *x* direction of any anisotropic material in the stackup.



Slots and seams

Slots and seams in thin metal sheets can be a significant source of electromagnetic interference. The TLM solver can model these narrow apertures without having to add mesh cells across the gap. This can lead to significantly faster simulations.

To add a slot to a thin metal sheet, first select the sheet object, and then choose *Modeling: Shapes* \Rightarrow *Faces and Apertures* \Rightarrow *Slot* .



The *Type* can be set to *Slot*, *Seam* or *Transfer impedance*.

A *Slot* type should be used when there is a thin gap cut into the metal. The slot dimensions are defined by the *Depth* and *Gap* values. There are limitations to the type of slot that can be modeled. The slot gap should be less than approximately 40% of the corresponding cell size that the slot passes through, and the slot depth should be less than 5 times the cell size normal to the slot plane. *Conductivity* and *Relative permittivity* can be defined to represent a gasket material in the gap.

A *Seam* type should be used when two sheets of metal overlap. It is defined using an *Overlap* and a *Gap*. The number of *Segments* along the slot/seam can be specified to represent electrical connections such as rivets along the length of the slot.

The *Transfer impedance* type is used to represent the frequency dependent penetration of signals through more complex materials in the slot gap.

If a number of points or a curve are picked before the slot dialog is opened, then these will be used to define the path of the slot. Otherwise you must define the path as a series of xyz coordinates along the length. Note that these coordinates must lie on the selected object.

Waveguide ports

Most waveguide ports are supported by the TLM solver but only the fundamental mode can be excited at each port. All waveguide ports in a model must be excited before the TLM solver can generate scattering parameters.

Excitation signal

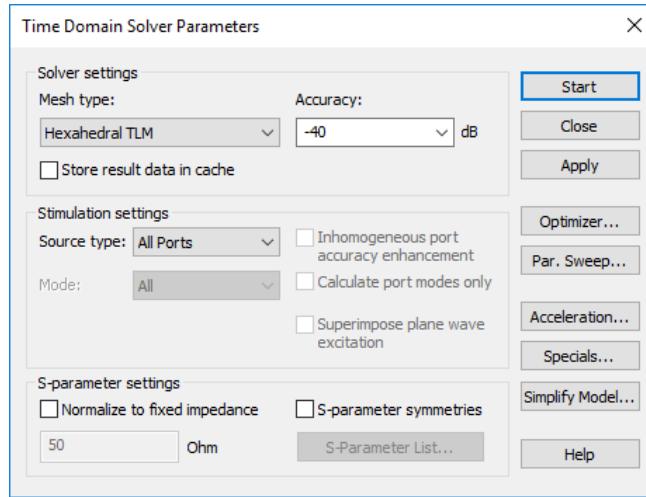
The TLM and Transient solvers use different default excitation signals. If the reference excitation signal is set to *default*, then the TLM solver will apply an impulse excitation which is filtered to the maximum model frequency. The excitation will then have a uniform magnitude across the frequency range of interest.

Mesh definition

The TLM solver uses the same hexahedral mesh as the Transient solver, but has different default values since the TLM solver sometimes needs a finer mesh to capture the geometry accurately. To compensate for the increased mesh fineness, the TLM solver employs an octree-based meshing algorithm to reduce the overall cell count. Small cells are lumped together into larger cells to create a mesh that gradually becomes coarser with increasing distance from the geometry.

Launch the TLM solver and view results

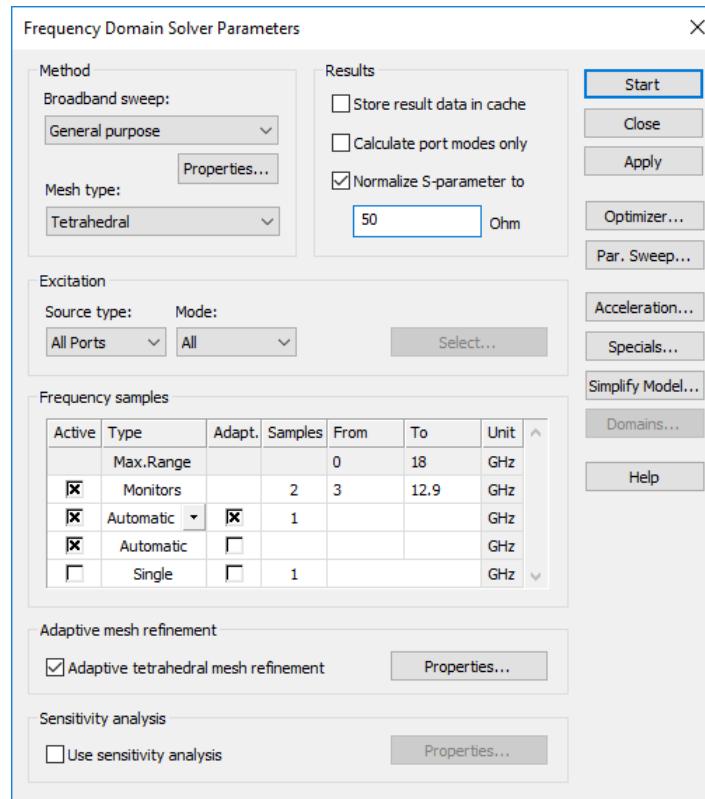
The TLM solver can be launched by choosing *Home: Simulation \Rightarrow Setup Solver \Rightarrow Time Domain Solver* :



The *Mesh type* should be set to *Hexahedral TLM* to launch the TLM solver.

Frequency Domain Solver

The basic procedure of running the frequency domain solver is demonstrated in the previous section **Comparison of Time and Frequency Domain Solver Results**. The following explanations provide some more detailed information about the settings in the frequency domain solver dialog box which you can open by choosing *Home: Simulation \Rightarrow Setup Solver \Rightarrow Frequency Domain Solver* 

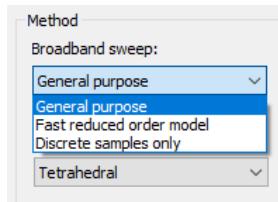


A special feature of the frequency domain solver is the support of both hexahedral and tetrahedral meshes. In most cases, you will compare the results from the tetrahedral frequency domain solver and the hexahedral transient solver, since this allows you to compare results from two completely independent simulation techniques.

An important difference between the transient solver and the frequency domain solver is the number of frequency samples that are 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 does however use special broadband frequency sweep techniques in order to derive the full broadband spectrum from a relatively small number of frequency samples.

The *Method* field in the solver dialog allows choosing the mesh type and the technique to generate results for the whole frequency range:



The frequency domain solver with *General purpose* broadband frequency sweep can be seen as the counterpart of the transient solver.

As an alternative to the *General purpose* sweep, a *Fast reduced order model* sweep is available, which efficiently generates broadband results from very few equation system solver runs.

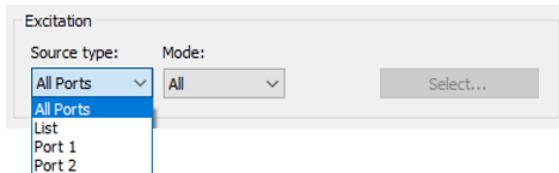
If you are only interested in results at a few specific frequencies, the *Discrete samples only* option may be used.

For CPU acceleration and distributed computing options choose *Home: Simulation* \Rightarrow *Solver* \Rightarrow *Acceleration*. Please refer to the chapter **Acceleration Features** or to the online help for more detailed information about the different acceleration features.

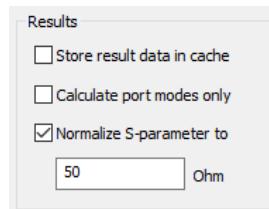
Solver Result Settings

To record the fields at particular frequencies, monitors can be defined in advance as described previously for the transient solver. S-parameters and fields can be accessed as usual from the entries in the navigation tree.

In order to obtain the complete S-matrix and fields, *All Ports* are by default selected as an excitation, which includes both waveguide port modes and discrete ports. If you consider some ports as output terminals only, for instance in a device with higher order waveguide port modes, the amount of result data as well as the simulation time can be reduced by limiting the excitation to some sources only.



Some post-processing steps however may require the full S-matrix and thus *All Ports* and *All Modes*. An example thereof is the normalization of S-parameters which has been enabled for the coaxial connector example:



With *Calculate port modes only* enabled the solver run stops after the waveguide port modes have been calculated without generating any further results. This allows you to quickly check the port modes as described in the chapter **Analyze the Port Modes**.

Store result data in cache creates full backups of the project after parametric changes, for instance in the course of a parameter sweep.

Information about how many monitor samples are left to calculate is displayed in the *Frequency samples* frame in a row labelled *Monitors*, provided in case that some monitors have been defined. If no results have been calculated, the sample count corresponds to the overall number of monitor samples. You can exclude the monitors from being calculated by removing the *Active* flag in this row. In the same way any other sampling row can be ignored by removing the *Active* flag.

Frequency samples						
Active	Type	Adapt.	Samples	From	To	Unit
	Max.Range			0	18	GHz
<input checked="" type="checkbox"/>	Monitors		2 3	12.9		GHz
<input checked="" type="checkbox"/>	Automatic	<input checked="" type="checkbox"/>	1			GHz
<input type="checkbox"/>	Automatic	<input type="checkbox"/>				GHz
<input type="checkbox"/>	Single	<input type="checkbox"/>	1			GHz

By default, frequency samples are added automatically until the S-parameters in the given frequency range are known accurately enough also between the calculated frequency samples. For the *General purpose* sweep, this is indicated by the third sampling row shown above: it has no limit for the number of samples, and blank entries to let *From* and *To* match the global frequency range, in this example from zero up to 18 GHz.

Please note that the frequency domain solver cannot calculate the fields at a frequency of zero. Therefore, a frequency of zero will automatically be shifted to a reasonably small value, and S-parameters will be extrapolated to 0 Hz if the global frequency range starts at zero.

Only sample definitions for the adaptive mesh refinement are considered when the *Fast reduced order model* sweep is selected, since this technique always generates broadband results during the frequency sweep:

Broadband sweep:
Fast reduced order model

Frequency samples						
Active	Type	Adapt.	Samples	From	To	Unit
	Max.Range			0	18	GHz
<input checked="" type="checkbox"/>	Monitors		2 3	12.9		GHz
<input checked="" type="checkbox"/>	Automatic	<input checked="" type="checkbox"/>	1			GHz
	Automatic	<input type="checkbox"/>				GHz
	Single	<input type="checkbox"/>	1			GHz

For the *General purpose* sweep, it is possible to let the solver record electric and magnetic fields and fluxes for all of the frequency samples without an explicitly defined field monitor (option *Save field results at samples* in the *Specials* dialog.)

A convenient feature of the *General purpose* sweep with tetrahedral mesh is the ability to continue a solver run to calculate or even just quickly evaluate additional monitors.

With either sweep method, you can even invoke a single calculation of the fields at a frequency marked in the S-parameter plot. This is described later.

Usually there is no need to change the default settings in the list of *Frequency samples*. However, sometimes it might be helpful to specify additional samples (see also “Frequency Domain Solver Overview” in the online help). One example for such a case is given below.

Adaptive Tetrahedral Mesh Refinement

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

By default, curved elements are used for newly generated projects. In order to enable curved elements for projects created with earlier versions, first select *Home*: *Mesh* \Rightarrow *Global Properties* \Rightarrow *Tetrahedral*. Then push the *Specials* button and choose *Automatic* if this is not yet set.

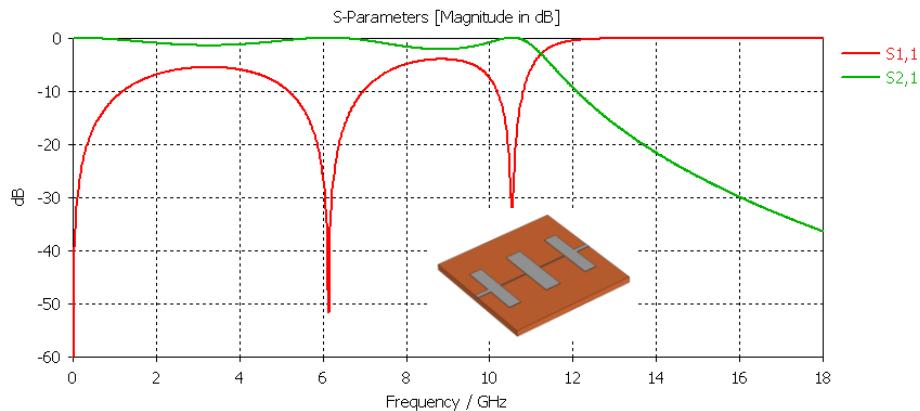
The mesh adaptation strategies of the transient and frequency domain solvers are fundamentally different. The transient solver runs the entire broadband simulation for every mesh adaptation pass and evaluates the worst-case deviation of two subsequent S-parameter results

(broadband.) The mesh refinement is then done based on information from the broadband result data.

In contrast, the frequency domain solver usually runs the mesh adaptation only for a single frequency point at a time. Once the adaptation is complete, the broadband results are computed by keeping the adapted mesh fixed (however, mesh adaptation on broadband results like in the transient solver is available as an option as well, as mentioned below.)

Since by default the frequency domain solver mesh adaptation runs only for a single frequency point at a time, the location of this point within the frequency spectrum is very important. For weakly resonant devices, it is usually a good policy to select the highest frequency of interest for the mesh adaptation. This corresponds with the default setting and will ensure that even the fields with the shortest wavelength in the frequency sweep are sampled properly.

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



This low pass type filter has very low transmission at the highest frequency of interest. Running the mesh adaptation at this frequency will not provide sufficient information about the actual filter characteristics. 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 this, it is very important to specify the adaptation frequency such that it is located in the pass band of the filter. Please note that the solver tries to detect those situations by looking at the minimum input reflection of all S-parameter ports (information or a warning will be displayed in the message window.) If necessary, the adaptation at this frequency sample is stopped and continued at a different frequency:

The input reflection of the S-parameters seems to be large at 18 GHz.
The adaptive mesh refinement at this sample will be stopped. A new mesh adaptation frequency has been added at 9 GHz. [More...](#)

However, you can save some time by manually setting the adaptation frequency to a constant value. This can be done by first selecting *Single* from the *Type* dropdown box of the adaptive mesh refinement line (which has *Adapt.* checked) in the frequency list. Then specify for instance 10 GHz as an adaptation frequency in the *From* column of the list:

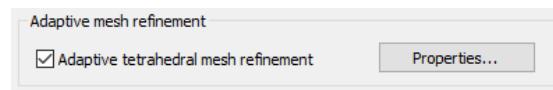
Frequency samples						
Active	Type	Adapt.	Samples	From	To	Unit
	Max.Range			0	18	GHz
<input checked="" type="checkbox"/>	Monitors		2	3	12.9	GHz
<input checked="" type="checkbox"/>	Single	<input checked="" type="checkbox"/>	1			GHz
	Automatic	<input type="checkbox"/>				GHz
	Equidistant	<input type="checkbox"/>				GHz
	Logarithmic	<input type="checkbox"/>	1			GHz

It is possible to define multiple adaptation frequency points, for instance equidistantly distributed or even with logarithmic spacing, by using the corresponding drop down list as shown for the highlighted line in the following figure:

Frequency samples						
Active	Type	Adapt.	Samples	From	To	Unit
	Max.Range			0	18	GHz
<input checked="" type="checkbox"/>	Monitors		2	3	12.9	GHz
<input checked="" type="checkbox"/>	Ex: Single Automatic Equidistant Logarithmic	<input checked="" type="checkbox"/>	1			GHz
						GHz
			1			GHz

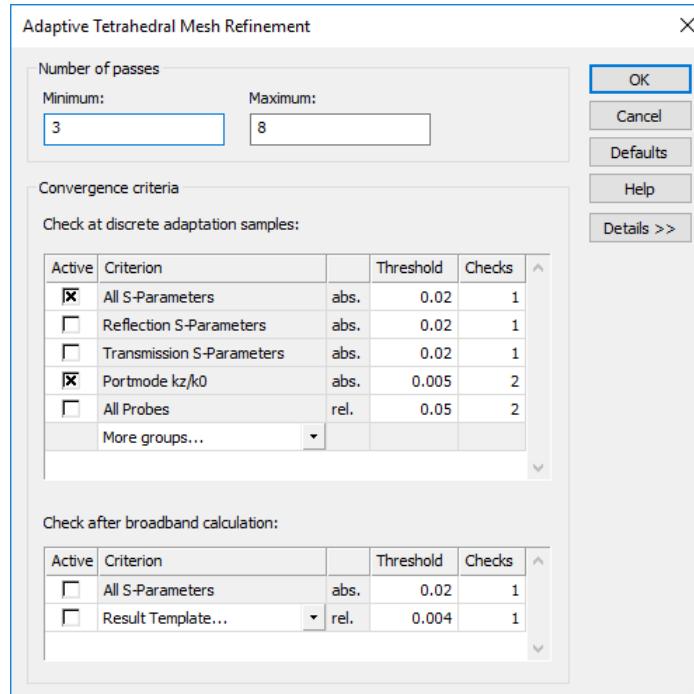
The adaptive mesh refinement will then be sequentially performed at those discrete mesh adaptation frequency samples (two in the example above) before the broadband sweep is started with the adaptively refined mesh.

All settings for the adaptive mesh refinement are displayed if you press on the corresponding *Properties* button:



The adaptive tetrahedral mesh refinement dialog by default allows three to eight mesh refinement passes. If multiple adaptation frequencies are defined as shown above, these limits hold for each mesh adaptation sample individually.

Stopping or convergence criteria are very important for the accuracy of the results. They are defined in the *Convergence criteria* frame:



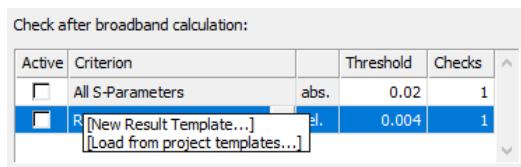
Convergence criteria can be checked after each discrete adaptive mesh refinement sample or after the broadband results are available. Each criterion has a threshold associated with it, and a number of checks. This number defines how often the criterion must fall below the threshold in consecutive mesh adaptation passes until the convergence criterion is considered as being met.

In the case of S-parameters, the criterion (Delta S) is determined as the maximum deviation of the absolute value of the complex difference of the S-parameters between two subsequent passes. By default, all S-parameters are taken into account. In addition, predefined groups for reflection and transmission S-parameters exist, and fully customized groups can be defined as well. The criterion may be calculated in two ways:

- Checking the convergence criterion at discrete adaptation samples means that some mesh adaptation frequencies are calculated in a sequential fashion, and Delta S is used as a criterion for the adaptive mesh refinement loop at those mesh adaptation samples. It refers to the S-parameters at those particular frequencies. For each adaptation frequency, the mesh is refined several times. The broadband frequency sweep is calculated afterwards.
- In order to take the S-parameters in a specified frequency range into account, the broadband frequency sweep must be applied before calculating Delta S as the maximum difference of all S-parameters in the frequency range.

For field probe results, the stopping criterion is relative to the maximum of the probe values of a specific probe type calculated so far.

In addition to the S-parameters and probe results, any *Result Template* may be employed as convergence criterion. In particular, for models without S-parameter ports this is a convenient way to ensure the convergence of the mesh adaption process. An arbitrary number of 0D and 1D Result Templates can be defined and selected in the *Check after broadband calculation* list in the *Result Template...* drop down menu:

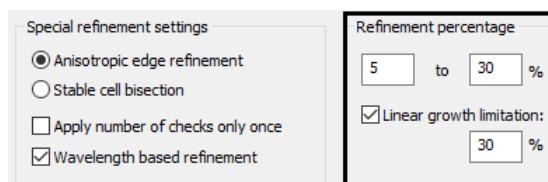


New Result Templates may be defined choosing *[New Result Template...]* in the drop down menu or by *Post-Processing: Tools* \Rightarrow *Result Templates* , enabling user defined specifications of application-tailored convergence criteria. Please refer to the online documentation and the *CST Studio Suite – Getting Started* manual for more information about this versatile functionality.

The convergence criterion *Portmode k_z/k_0* applies to the adaptive mesh refinement during the port mode calculation for waveguide ports. It is the maximum magnitude of the difference of the port modes' complex propagation constant k_z divided by the free space propagation constant k_0 between two port mesh refinement passes.

During the adaptive mesh refinement, newly created nodes in the tetrahedral mesh will be projected onto the original geometry in order to improve the approximation of the geometry (True Geometry Adaptation.)

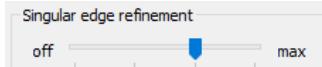
If you expand the *Details* in the *Adaptive Tetrahedral Mesh Refinement* properties dialog by pressing the corresponding button, you can access some special refinement settings and the refinement percentage:



The settings in the *Refinement percentage* frame determine how much the mesh may grow between two consecutive adaptive mesh refinement passes. The default values are a compromise between accuracy and computational resources. A larger mesh growth per pass

might lead to more accurate results in less passes at the cost of higher memory requirements and possibly a longer simulation time. However, a very high mesh growth percentage might lead to mesh refinement also far away from regions of interest. In this case it may be more efficient to perform more mesh adaptation passes with moderate mesh growth for each single pass.

The performance of the adaptive mesh refinement can be further accelerated by an appropriate refinement of mesh edges on conductors. Because of the potentially highly varying field strength and distribution, a special treatment for these areas during the adaptive mesh refinement passes often leads to faster overall convergence and is recommended especially for planar structures as the low pass filter example shown above. Different levels of this strategy are selectable within the *Singular edge refinement* frame.



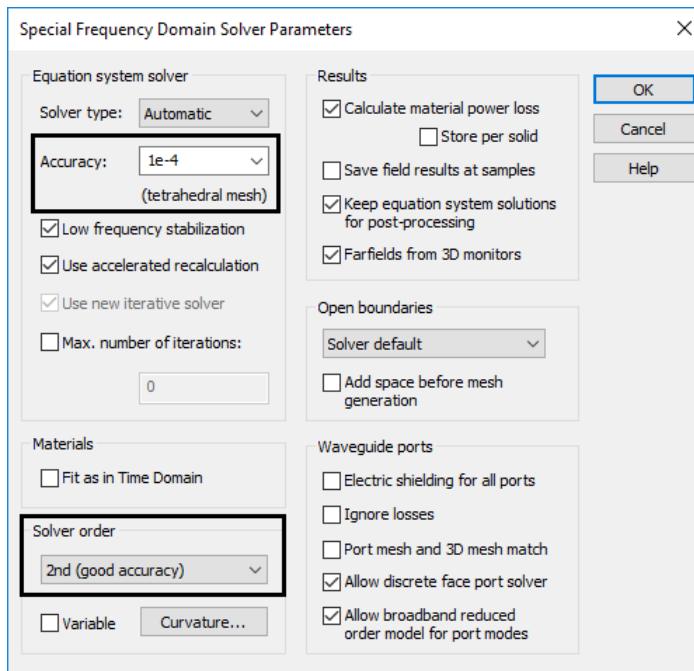
Please close the adaptive tetrahedral mesh refinement dialog to return to the frequency domain solver parameters dialog.

Note that with the *Hexahedral* mesh chosen as the *Mesh type*, the adaptive refinement is performed in a broadband fashion as described for the time domain solver in “Adaptive Mesh Refinement” on page 37. Adaptive mesh refinement frequency samples are therefore ignored for the hexahedral mesh based methods.

Adaptive mesh refinement is one ingredient for reaching a certain level of accuracy. The default settings satisfy the accuracy needs for many applications, with reasonable computational effort. However, if the thresholds of the mesh refinement stop criteria are tightened, it is recommended to change other settings correspondingly.

Solver Order and Curved Elements

Special settings which influence the accuracy of the results as well as the performance of the simulation comprise the *Accuracy* in the *Equation system solver* frame (smaller values represent higher accuracy) and the solver order. In the frequency domain solver dialog, choose the *Specials...* button to open the corresponding dialog:

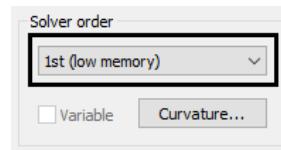


By default, the tetrahedral frequency domain solver uses second order elements to get an excellent sampling of the fields at high frequencies. This also allows the use of relatively few

elements per wavelength by comparison with the first order elements used by the solvers based on hexahedral grids.

A higher solver order allows you to achieve accurate results with less mesh cells and potentially less memory consumption than a lower order if the structure contains electrically large regions free of geometric details. For a given mesh resolution, a higher order will provide more accurate results. However, some structures may need a relatively fine mesh if their geometry is much finer than required to properly sample the wave phenomena. Typical application examples for this are printed circuit boards or integrated circuit packages. In such cases, using first order elements rather than the standard second order elements can reduce simulation time and memory requirement significantly.

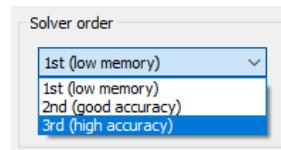
To use first order elements, select *1st (low memory)* in the *Solver order* field in the *Specials* dialog box:



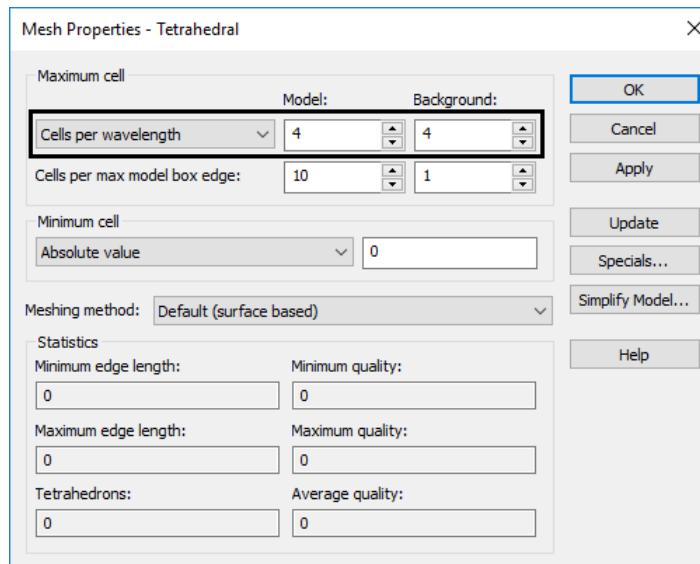
Whenever the solver order is changed, for instance from second to first order, the resolution of the initial mesh and some parameters in the adaptive mesh refinement dialog should be adjusted accordingly.

For new projects, these settings are applied automatically in a way that ensures a suitable resolution of the wavelength in media (for projects generated with earlier versions of CST Studio Suite, please choose *Home: Mesh* \Rightarrow *Global Properties* \Rightarrow *Tetrahedral* and select *Automatic* from the drop down menu in the *Maximum cell* frame). A higher solver order may result in a smaller number of mesh cells.

A third order field approximation scheme is available, and can be selected in the drop down box:

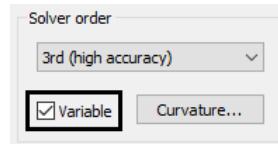


Another reason for choosing higher order is to increase the accuracy of the solver results. As an example for "third order", select *Home: Mesh* \Rightarrow *Global Properties* \Rightarrow *Tetrahedral*, and specify for instance four *Cells per wavelength* as the *Maximum cell* size:



The initial tetrahedral mesh then will be sufficiently dense for second order, but as third order has been chosen, the results are even more accurate for the given mesh.

If the option for *Variable* order is activated, the frequency domain solver with tetrahedral mesh is allowed to use a different solver order for each tetrahedron, rather than constant order throughout the calculation domain.



The solver order's upper limit is then given by the order selected in the drop down combo box left to the *Variable* check box (for instance first to third order, for the selection shown above.)

Enable the *Variable* option if the structure contains electrically small details as well as large voids. The solver will then assign an initial distribution of the solver order to the tetrahedrons, and this distribution may potentially be changed automatically in the course of the adaptive mesh refinement.

Dispersive Materials

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 of general order to the data. During the simulation, the broadband material behavior will then be taken from the model rather than using the originally specified data.

Since the frequency domain solver computes the broadband sweep by a sequence of individual frequency point calculations, the solver can simply linearly 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.

When comparing the results of these two solvers it may be advantageous to configure the frequency domain solver to use the same material model with fitted data as the transient solver. This can be done by checking the *Fit as in Time Domain* box in the *Materials* frame of the solver *Specials* dialog box.



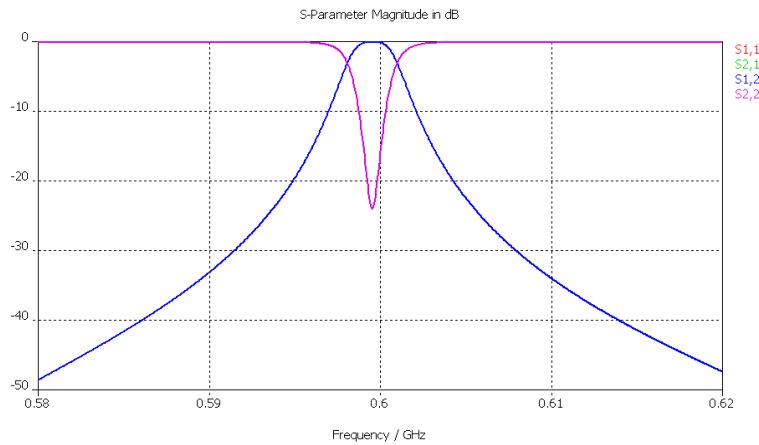
Continued solver runs

You may continue the solver's frequency sweep with additional fixed or automatically chosen samples, newly added field monitors, and additional adaptive mesh refinement after one solver run has finished.

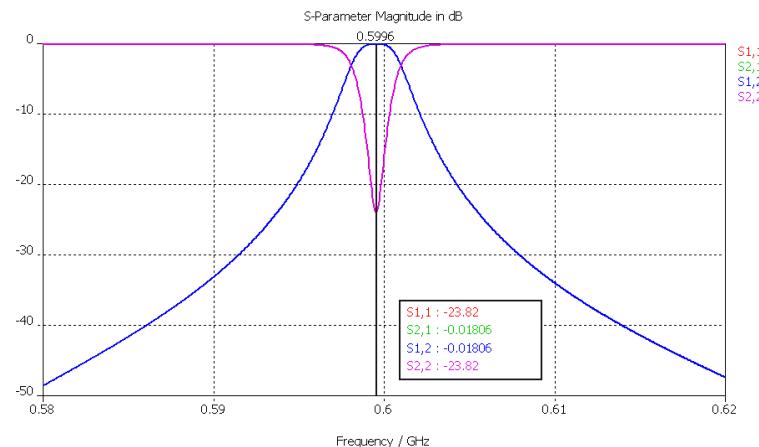
If additional results for already calculated frequencies are requested, for instance by defining new monitors or by using *Calculate Fields at Axis Marker*, the solvers with tetrahedral mesh will attempt to reload the solution to quickly perform additional post-processing steps without the need to solve the equation system again.

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 completed 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 have run a simulation for a filter structure using either the general purpose or the fast reduced order model sweep method 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 (*1D Plot: Markers* \Rightarrow *Axis Marker* \Rightarrow *Move Marker to Minimum*):



Then click on the plot and choose *Calculate Fields at Axis Marker* from the context menu 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.

Workflow Summary

The following summarizes the input necessary for frequency domain analysis:

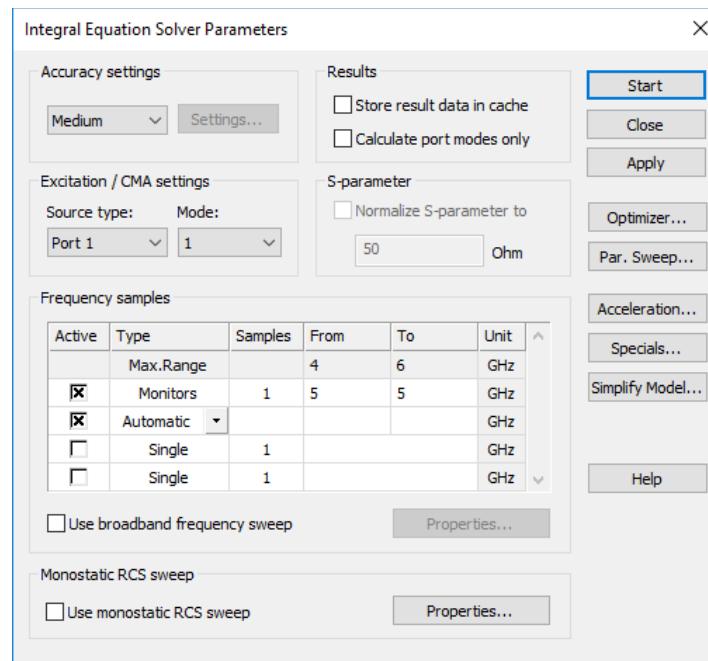
1. Select an appropriate project template (optional).
2. Set units (optional).
3. Set background material (optional).
4. Define the structure.
5. Set the frequency range.
6. Set the boundary conditions (optional).
7. Define the excitation ports.
8. Set the monitors (optional).
9. Select sweep method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

Integral Equation Solver

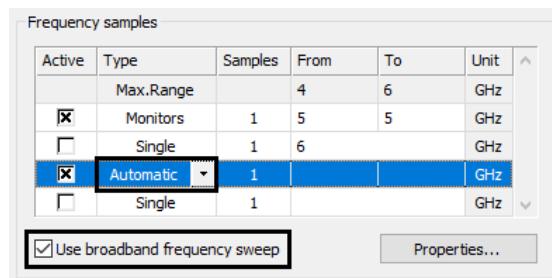
An integral equation solver computation is an analysis in the frequency domain based on a surface and wire mesh. The model setup is very similar to a general purpose frequency domain computation. The following explanations provide some more information about the specific settings in the integral equation solver dialog box.

Integral Equation Solver Parameters

You can open the dialog by choosing *Home: Simulation \Rightarrow Setup Solver \Rightarrow Integral Equation Solver*. As with the general purpose frequency domain computation, an integral equation calculation has to carry out the simulation frequency by frequency. Every frequency point requires a complete solver run.



A special broadband frequency sweep technique can be used 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 selecting the type *Automatic* in the table 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 frequency list.

The integral equation 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 navigation tree.

Acceleration

For CPU and GPU acceleration, distributed computing options and MPI computing settings choose *Simulation: Solver* \Rightarrow *Setup Solver*  \Rightarrow *Acceleration*. Please refer to the chapter **Acceleration Features** or to the online help for more detailed information about the different acceleration features.

Accuracy Settings

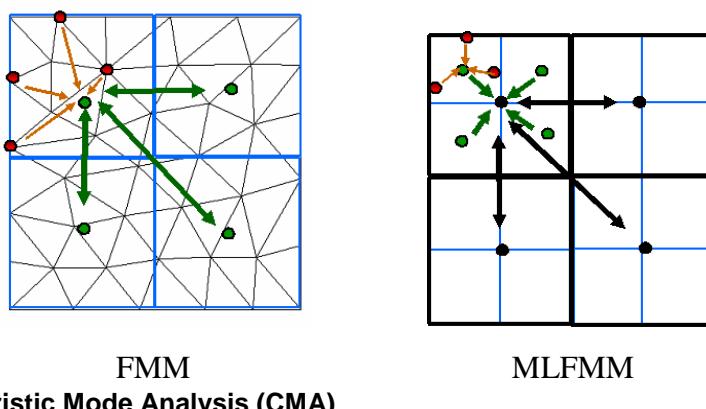
The solver accuracy can be controlled by selecting one of the predefined values (*Low*, *Medium* or *High*) in the *Accuracy* field. Alternatively, selecting the option *Custom* will activate a *Settings* button to open a dialog box for more detailed solver control. Please refer to the online documentation for more information about the available settings within this dialog box.

Special settings

The special settings dialog can be opened by choosing *Simulation: Solver* \Rightarrow *Setup Solver*  \Rightarrow *Specials*. It is possible to enable real ground or infinite PEC or PMC ground and to choose a preconditioner for the linear equation system solver in this dialog. The integral equation solver can make use of 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 integral equation 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.

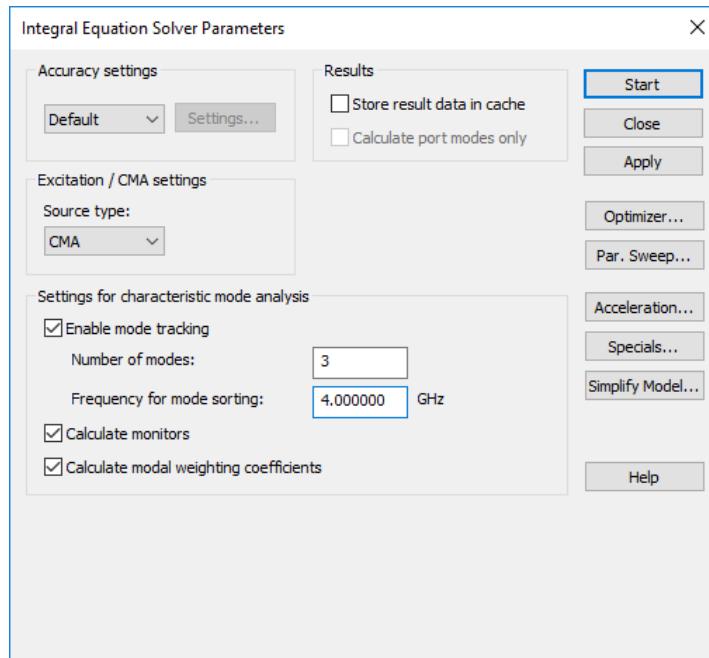
MLFMM

In its standard implementation, the integral method generates a full matrix containing information about the coupling between each pair of surface mesh elements. The Multilevel Fast Multipole Method (MLFMM) is a fast method to reduce the simulation complexity. It uses boxes (clusters of surface mesh elements) to combine the couplings, together with a recursive scheme to increase the efficiency (please see schematic below). The MLFMM speeds up the matrix vector multiplication for an iterative solver and also enhances the memory efficiency. It scales very well for large problems (geometry \gg wavelength) with a complexity of $O(N \log N)$.



Characteristic Mode Analysis (CMA)

A dedicated tool for the efficient analysis of characteristic modes on PEC-structures including the influence of possibly present dielectrics is included with the integral equation solver. It is activated by selecting *CMA* in the *Excitation / CMA settings* frame of the solver dialog.



The analysis can be performed either at discrete, user-defined frequency samples or in a frequency range with automatic mode tracking. If the option *Enable mode tracking* is inactive, the mode analysis is performed at the frequency points as defined in the list of samples. At every discrete frequency, the user-defined *Number of modes* with the largest modal significance is calculated. No mode matching between different samples is done with this setting. In contrast, mode matching and automatic mode tracking are performed if the option *Enable mode tracking* is selected. With this setting, the specified *Number of modes* with the highest modal significance is calculated at the user-defined *Frequency for mode sorting*. These selected modes are tracked over the whole simulation frequency range. Add field monitors and select the option *Calculate monitors* to visualize the eigencurrents and related quantities.

The following summarizes the input necessary for a frequency domain analysis using the integral equation solver:

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.
8. Set the monitors (optional).
9. Start the integral equation solver.
10. Analyze the results (S-parameters, field patterns, result templates, etc.).

Multilayer Solver

For structures which are mainly planar, such as microstrip filters, patch antennas, etc., the multilayer solver might be the best choice. The multilayer solver, based on the method of moments, does not require discretization of the transversally infinite dielectric and metal stackup. Therefore, this solver can be more efficient than general purpose 3D solvers for this specific type of application.

To create an appropriate mesh for the multilayer solver the mesh type *Multilayer* has to be selected (*Simulation: Mesh* \Rightarrow *Global Properties*  \Rightarrow *Multilayer*).

A simulation model consists of two parts:

- The metallic structure modelling the conductors and ports

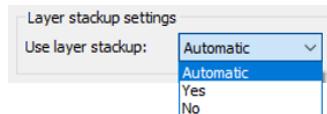
- The layer stackup

The layer stackup will be created automatically if the layers are defined by normal material bricks. The layer stackup can also be defined by using the background dialog (*Modeling: Materials* \Rightarrow *Background* .

Generating the layer stackup from the geometric model

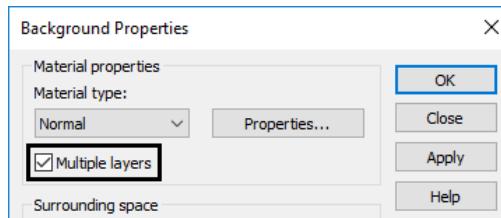
When the stackup is defined in the geometric model by means of several metal/dielectric layers, the mesh generation will automatically exclude the bricks used for layer definition. Metal sheets which define a decoupling plane will be added to the layer stack automatically. Holes in the metal sheets will be considered as apertures in the simulation.

Whether a solid or sheet will be considered for the layer stackup or not can be modified by the local mesh properties dialog.

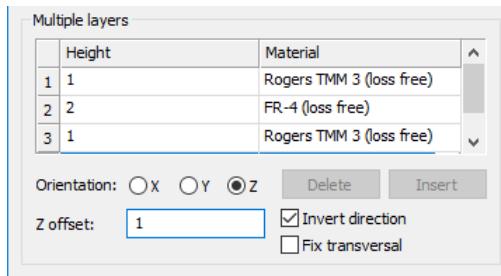


Generating the layer stackup by using the background dialog

The second way of defining the layer stackup is by means of the background properties. In this case, the background dialog has to be expanded by enabling the check box *Multiple layers* first.



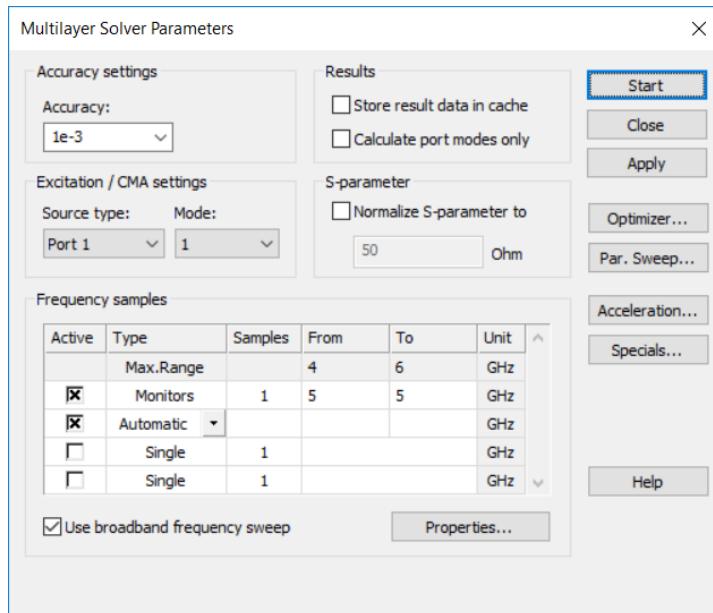
An arbitrary number of dielectric and metal layers can then be defined in the *Multiple layers* frame.



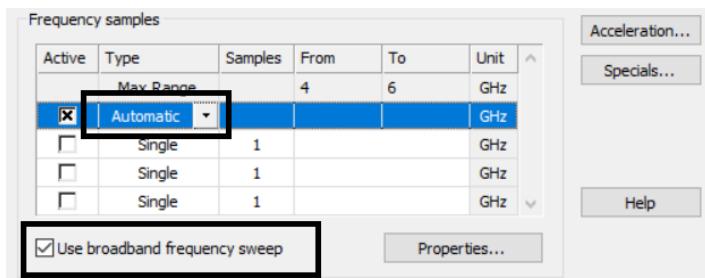
Multilayer Solver Parameters

You can open the multilayer solver dialog by choosing *Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *Multilayer Solver* .

A multilayer calculation has to carry out the simulation frequency by frequency, and every frequency point requires a complete solver run.



A special broadband frequency sweep technique can be used 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 selecting the type *Automatic* in the table 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.



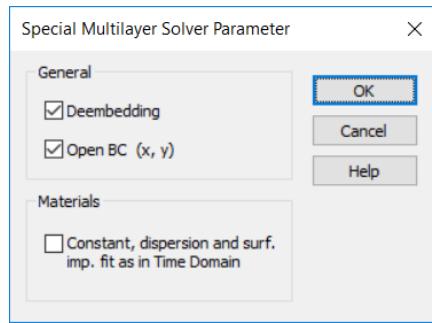
To store the fields at particular frequencies, monitors need to be defined in advance as described previously for the transient solver. These monitor frequencies are then added to the list of calculated frequencies.

For CPU acceleration, distributed computing options and MPI computing settings, choose *Simulation: Solver* \Rightarrow *Setup Solver* \Rightarrow *Acceleration*. Please refer to the chapter **Acceleration Features** or to the online help for more detailed information about the different acceleration features.

The multilayer solver cannot calculate the fields at a frequency of zero. Therefore, a zero frequency will automatically be shifted to a reasonably small value. For very low frequencies the multilayer solver supports low frequency stabilization.

The S-parameter and field results can be accessed as usual from the entries in the navigation tree.

Advanced settings are available in the special multilayer solver settings. This can be opened by choosing *Simulation: Solver* \Rightarrow *Setup Solver* \Rightarrow *Specials*:



General

The *Deembedding* option activates the automatic internal deembedding of waveguide and multipin ports to ensure most accurate S-Parameter results. In addition, the S-Parameters are then normalized to the calculated port impedances.

The multilayer solver uses an open boundary formulation in x- and y-direction and will ignore electric boundary conditions in x- and y- direction by default. This can be changed by deactivating the option *Open BC (x, y)*.

Materials

The multilayer solver can make use of user-specified material property tables more directly than the transient solver can. For the sake of comparing the results of these two solvers it may be advantageous to advise the multilayer 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*.

Characteristic Mode Analysis (CMA)

A dedicated tool for the efficient analysis of characteristic modes is integrated into the multilayer solver. It is activated by selecting *CMA* in the *Excitation / CMA settings* frame of the solver dialog. Please refer to the relevant paragraph in section *Integral Equation Computations* for an explanation of specific settings for CMA.

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

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

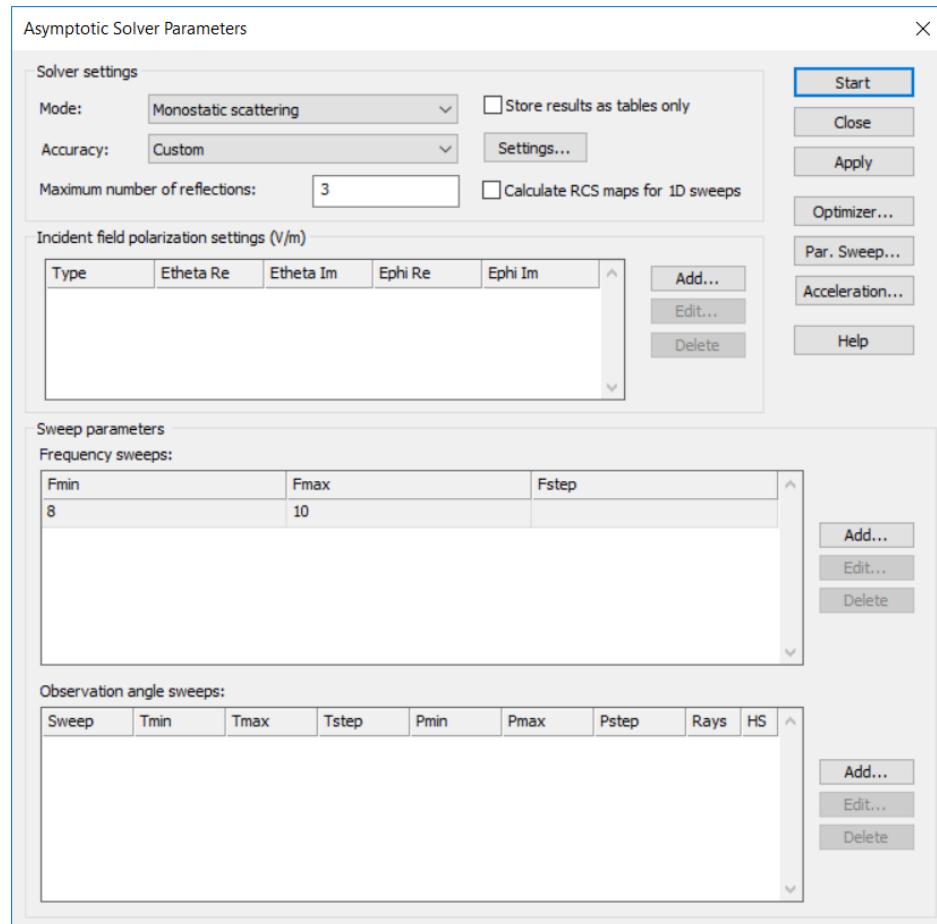
Asymptotic Solver

An asymptotic computation is an analysis in the frequency domain based on a so-called ray-tracing (shooting and bouncing rays) technique. In this approach, the scattered fields are determined by performing a local surface integration of the ray-fields at the ray-object intersections. The solver is typically used for scattering or antenna placement computations of electrically very large objects which are difficult to handle by other EM solution methods.

Due to its limited range of applications, the asymptotic solver's setup is a little different from that of the other more general solvers. The following explanations provide some basic information about the asymptotic analysis workflow. Please refer to the online documentation for more detailed information.

Asymptotic Solver Parameters Dialog:

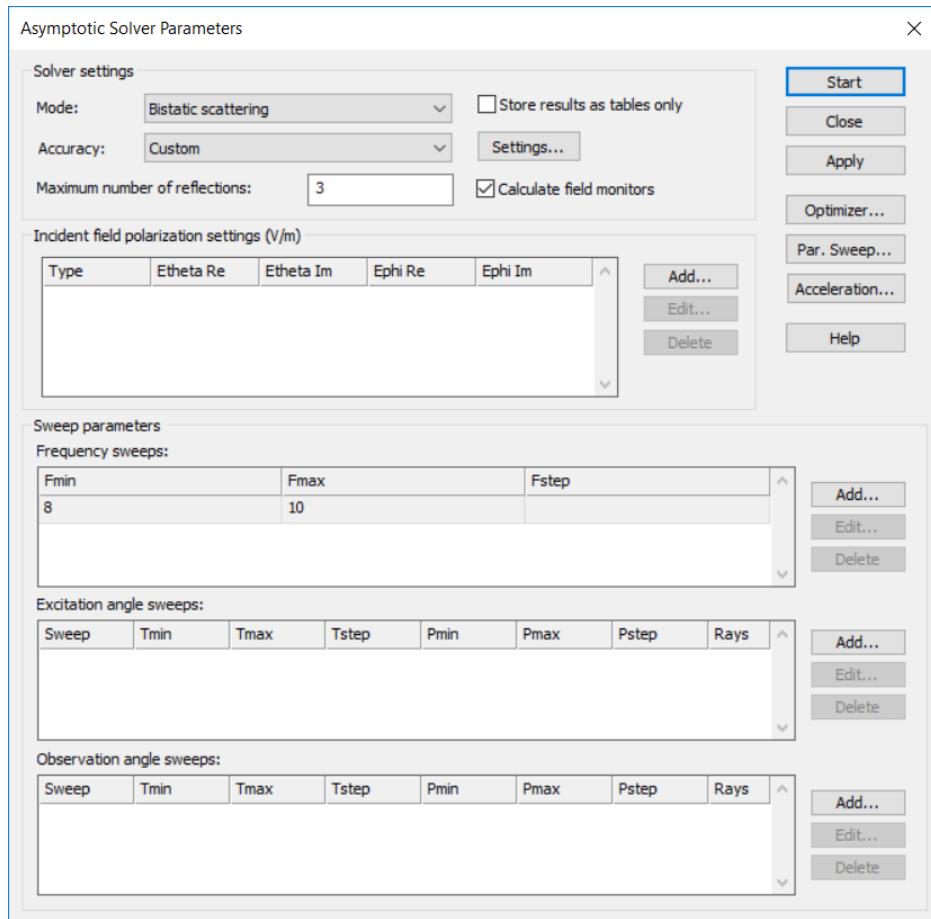
The dialog box can be opened by choosing *Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *Asymptotic Solver* :



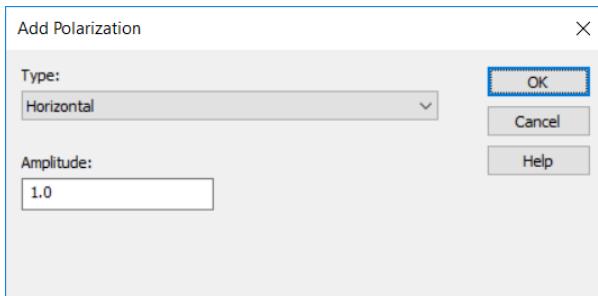
The actual layout of this dialog box will change depending on the selection in the *Mode* field.

For *Monostatic scattering* calculations, the *Sweep parameters* frame will contain two lists. One list specifies the *Frequency sweeps* and the other one describes the *Observation angle sweeps*.

For *Bistatic scattering* calculations excitation directions and observation directions are not identical as in the case of monostatic calculations. Therefore, the *Sweep parameters* frame will contain an additional *Excitation angle sweeps* list:



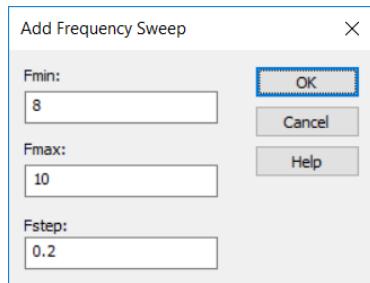
The electric field strength and the polarization of the incident plane wave can be set in the *Incident field polarization settings* frame by adding plane wave definitions to the list. After pressing the *Add* button, the following dialog box will appear:



This dialog box allows you to select a particular type of polarization such as *Horizontal*, *Vertical*, *Left hand circular polarized* or *Right hand circular polarized*. In addition, a *Custom* option can be selected where the complex amplitudes for the incident plane wave's theta and phi components can be specified.

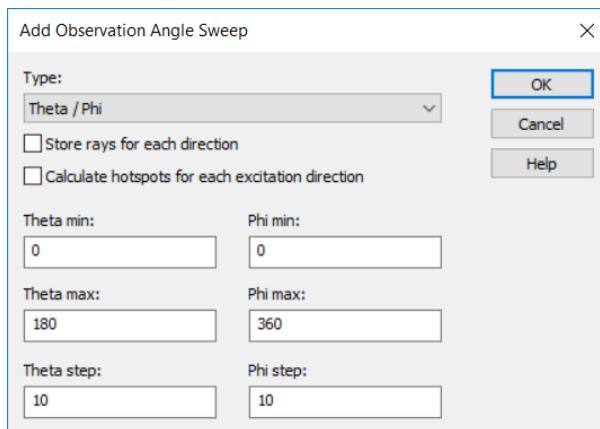
Sweep Definitions

Each of the sweep definition lists can contain a number of individual sweep descriptions. A particular sweep can be added by pressing the *Add* button. For frequency sweeps the following dialog box allows the specification of lower and upper frequency bounds as well as a step width:



A single frequency point can be specified by setting the lower and upper bounds to the same value.

For angular sweeps, the following dialog box will appear:



Here, you can select a particular type of sweep:

- Single Point:* Single theta / phi direction rather than a sweep
- Theta / Phi:* Sweep for both theta and phi angles
- Theta:* Sweep for theta while keeping phi to a fixed value
- Phi:* Sweep for phi while keeping theta to a fixed value

For varying angles theta or phi, upper and lower bounds as well as the corresponding step width are specified in degrees.

In addition, the *Store rays for each excitation direction* option can be checked in which case the solver will store information for a certain number of representative rays. These rays can be visualized by selecting the corresponding result entry in the navigation tree. Please note that for *Bistatic scattering* mode, the *Store rays* option needs to be checked for both the excitation angle sweep as well as the observation angle sweep in order to store the rays for the respective incident / observation angle pairs.

The *Calculate hotspots for each excitation direction* option is only displayed in *Monostatic scattering* mode. Turning this option on for a particular observation angle sweep will calculate hotspot images for each of its excitation / observation directions. A hotspot result can then be visualized by selecting its corresponding result entry in the navigation tree.

Accuracy Settings

The solver accuracy can be controlled by selecting one of the predefined values (*Low*, *Medium* or *High*) in the *Accuracy* field. Alternatively, selecting the option *Custom* will activate a *Settings* button to open a dialog box for more detailed solver control. Please refer to the online documentation for more information about the available settings within this dialog box.

Another important parameter is specified in the *Maximum number of reflections* field. This setting limits the maximum number of reflections for each particular ray as it is bouncing back and forth inside the simulation domain. Typical settings for this parameter are in the range of two to five. The solver will display some statistics about the actual number of multiple reflections, and also will provide some feedback as to whether this parameter may need to be increased further.

For CPU and GPU acceleration as well as distributed computing options choose *Simulation: Solver* \Rightarrow *Setup Solver* \Rightarrow *Acceleration*. Please refer to the chapter **Acceleration Features** or to the online help for more detailed information about the different acceleration features.

In addition to the monostatic and bistatic scattering modes described above, the solver also features a *Field sources* mode which allows scattering computations with farfield (point) or nearfield (box) sources rather than plane waves. Finally, a *Range profiles* mode is offered to efficiently calculate range profiles and sinograms of radar targets. Please refer to the online documentation for more information about these particular modes of operation.

Workflow Summary

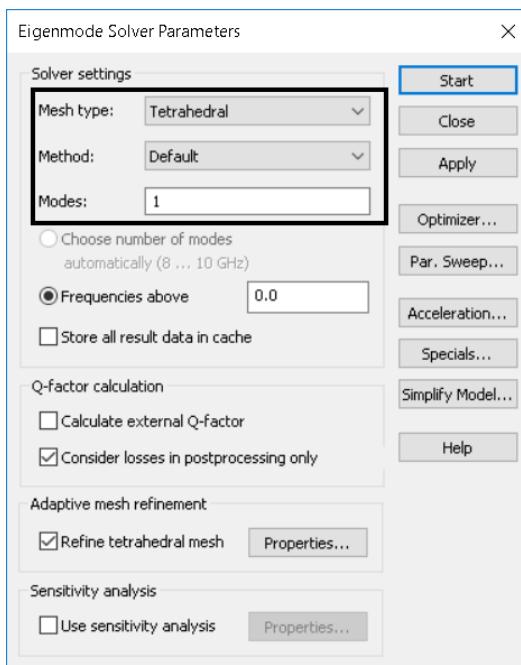
The following list summarizes the input necessary for asymptotic analysis:

1. Select an appropriate project template (optional).
2. Set units (optional).
3. Set background material to vacuum.
4. Define the structure.
5. Set the frequency range.
6. Set all boundary conditions to open.
7. Start the asymptotic solver.
8. Analyze the farfield or RCS results.

Eigenmode Solver

The eigenmode solver calculates a finite number of modal field distributions in a closed device. Linear and curved tetrahedral meshes as well as hexahedral meshes are supported.

Since the eigenmode analysis does not always require the definition of excitation ports, this step can often be omitted. The definition of field 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 (*Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *Eigenmode Solver* \Rightarrow *Eigenmode Solver*), which looks as follows:



The eigenmode solver by default uses the tetrahedral mesh which is therefore described first.

Tetrahedral Mesh

Two different eigenmode solvers are available for the tetrahedral mesh: *Default* and *Preview*.

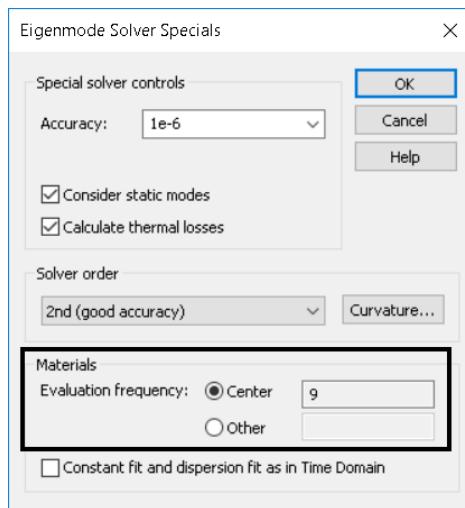
These methods work on completely different mathematical foundations and implement a different set of features. The *Default* solver should be sufficient for most eigenmode analyses and therefore, it should be mostly used. This solver can be considered as a more robust, faster, and memory efficient solver. However, we recommend the *Preview* solver if the analyzed structure contains lossy materials with frequency dependent complex permittivity or permeability. Due to the fact that the *Preview* solver considers the waveguide ports as open, it also computes an accurate external Q-factor for each mode. Also for structures with open or conducting wall boundaries, this solver should be the first choice.

The simulation time increases with the number of modes. Thus, only as many modes as required should be specified in the corresponding field. A strict lower limit to the modes' frequencies can be defined in *Frequencies above*.

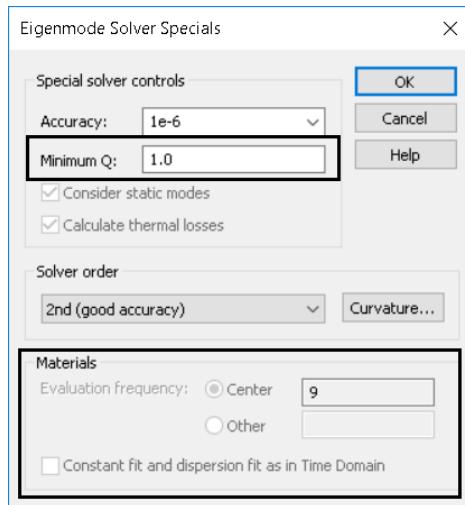
The external Q-factor can be calculated for structures with waveguide ports attached to the device.

By default, losses are ignored for the eigenmode calculation with the *Default* solver itself. This is justified for many applications and results in a better performance of the eigenmode solver. With some level of approximation, losses can be considered by post-processing after the eigenmode solver run.

If the option *Consider losses in postprocessing only* is disabled, lossy and dispersive materials are evaluated at a fixed frequency and the materials' complex permeability and permittivity are then applied to the whole frequency range. This *Evaluation frequency* for the material parameters is defined in the *Specials* dialog and defaults to the center frequency. It can be modified if *Consider losses in postprocessing only* was disabled before:



The option *Consider losses in postprocessing only* is not relevant for the *Preview* solver, as it always considers the defined losses and directly computes a total Q-factor for each calculated mode (e.g. due to volume, external Q losses). Consequently, the *Materials* frame is completely disabled for the *Preview* solver:

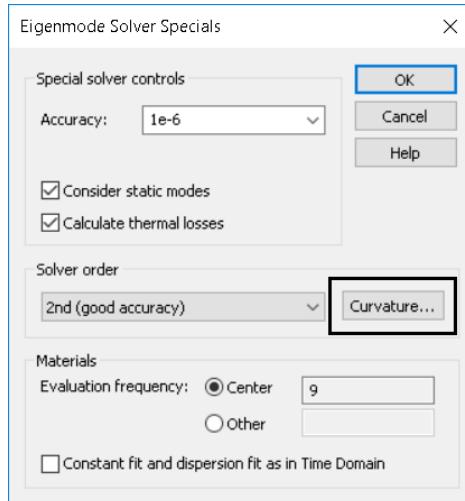


It is important to note that the *Preview* solver calculates complex eigenvalues, where the imaginary part belongs into the defined frequency range and the Q-factor, which is also dependent from the real part of the complex eigenvalue, is greater or equal to the value specified in *Minimum Q*. As already mentioned above, a strict lower limit to the modes' frequencies can be defined in *Frequencies above*.

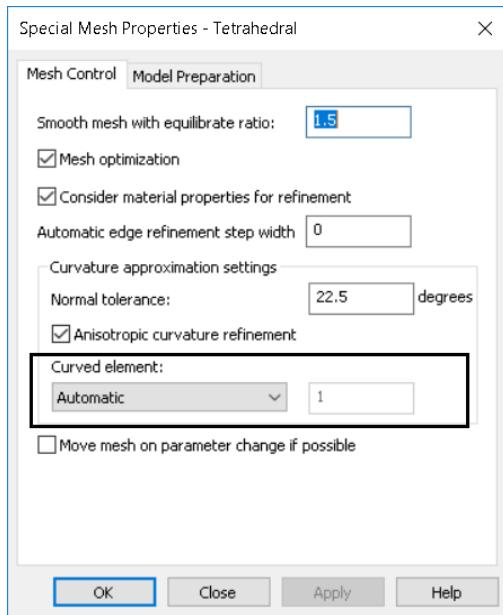
Due to the fact that many applications which require an eigenmode solver have curved surfaces, it is advisable to activate the curved elements for the tetrahedral mesh, since they provide a better approximation of the geometry than linear elements. The latter are a special case of the former: linear elements are "curved" with a curved element order of one. Curved elements are activated automatically for newly created projects.

The curvature order of the elements is usually chosen automatically so that it fits with the solver order of the solver selected in *Home: Simulation* \Rightarrow *Setup Solver* \Rightarrow *Eigenmode Solver*

For projects created with earlier versions, the curved element order can be changed in the special tetrahedral mesh properties. This would require closing the solver dialogs and choosing *Home: Mesh* \Rightarrow *Global Properties* \Rightarrow *Tetrahedral* and the *Specials* therein. However, a link in the solver specials provides direct access to this setting:

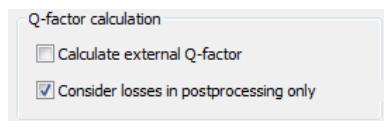


The settings for the solver order (first to third order) and a button *Curvature* are available in the *Solver order* frame. Please follow the *Curvature* link to the special mesh properties. Verify that the choice for the *Curved element* is "Automatic" or change the selection accordingly:



You may confirm the settings and close the special mesh properties dialog and the solver specials dialog by pressing *OK* to return to the eigenmode solver dialog.

Please enable the option *Consider losses in postprocessing only* again if necessary:

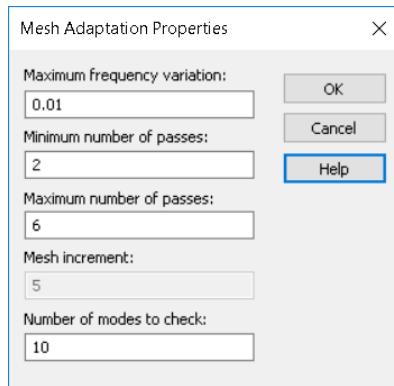


The adaptive tetrahedral mesh refinement is activated by default for new projects to ensure that the results are converged to a certain level of accuracy:



For projects generated with earlier versions, please consider enabling the adaptive tetrahedral mesh refinement. Please note that the *Preview* solver does not support adaptive mesh refinement and therefore, the setting is always disabled for this solver.

Click on *Properties* to open the *Mesh Adaptation Properties* dialog. The stopping criterion for the adaptive mesh refinement of the eigenmode solver is the *Maximum frequency variation*. For each eigenmode, the magnitude of the difference of the eigenmode's frequency between two subsequent passes is calculated. This value is then divided by the corresponding eigenmode frequency at the first of the two subsequent passes. The maximum of these values for all modes up to the *Number of modes to check* finally yields the *Maximum frequency variation*.



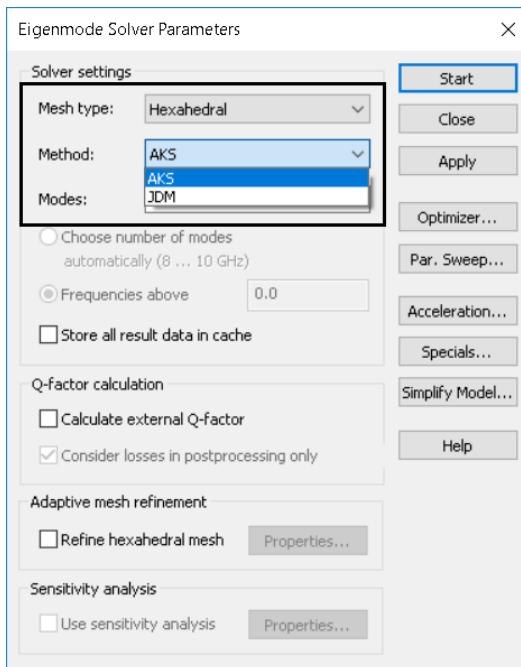
You can now perform the eigenmode simulation by clicking the *Start* button.

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

Results are stored in a common location in the navigation tree for both, the tetrahedral and hexahedral mesh.

Hexahedral Mesh

First change the *Mesh type* to *Hexahedral* in the eigenmode solver dialog box (*Home: Simulation \Rightarrow Setup Solver \Rightarrow Eigenmode Solver*



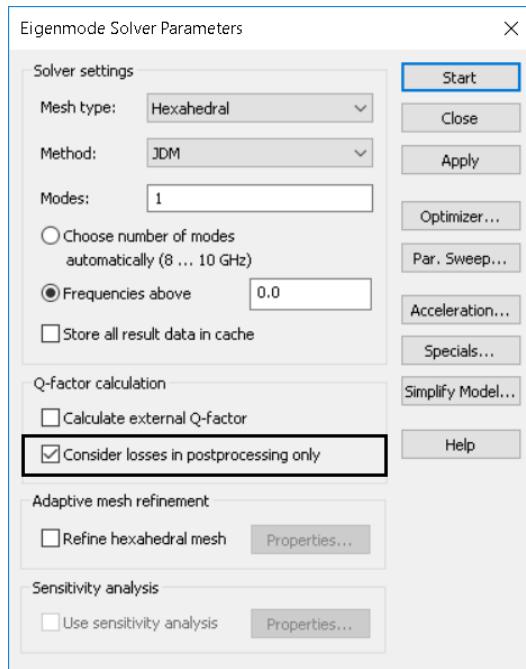
Two different eigenmode solvers are available for the hexahedral mesh: *AKS (Advanced Krylov Subspace)* and *JDM (Jacobi Davidson Method)*.

These methods work on completely different mathematical foundations. 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 especially if a small number of modes (for instance one to five modes) has to be calculated. Otherwise the *AKS* solver should be used.

The solution of lossy eigenmode problems is a challenging task and 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 by default also ignores the losses when *Consider losses in postprocessing only* is enabled as shown below:



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

The typical simulation procedure with a hexahedral mesh is as follows:

1. Depending on the number of modes, choose the proper solver method:
 - For loss free problems with a small number of modes (for instance one to five modes) choose *JDM*.
 - For loss free problems with many modes (for instance more than five modes) choose *AKS*.
 - For the direct solution of lossy problems choose *JDM* and disable *Consider losses in postprocessing only*.
 - If only higher order modes are required with eigenfrequencies above a certain threshold, choose the *JDM* solver and enter a value for *Frequencies above*, which is slightly lower than the threshold.
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 ten modes of the device.
3. Click the *Start* button.

The following description applies to the *AKS* method with ten modes. After the solver has finished, a summary of the calculated modes will appear in the message window:

Eigenmode solver results:		
Mode	Frequency	Accuracy
1	8.091 GHz	3.550e-011
2	10.26 GHz	2.920e-011
3	10.39 GHz	2.623e-012
4	12.60 GHz	3.250e-012
5	15.01 GHz	7.292e-013
6	15.64 GHz	8.377e-013
7	16.07 GHz	1.374e-013
8	16.60 GHz	3.541e-013
9	16.78 GHz	3.891e-013
10	18.34 GHz	9.846e-014

Optimum guess for the highest eigenfrequency would be: 18.3438.

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 a higher number of modes than you actually need.

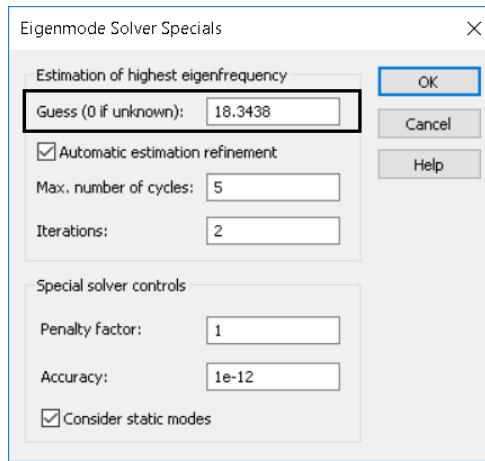
The AKS eigenmode solver internally needs an estimate for the frequency of the highest mode of interest. Usually this frequency is estimated automatically and improved by refinement passes if necessary.

Performing estimation refinement passes reduces the performance of the AKS eigenmode calculation. To speed up the AKS eigenmode calculation in such a case, 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 displays this value in the message window:

8	16.60 GHz	3.541e-013
9	16.78 GHz	3.891e-013
10	18.34 GHz	9.846e-014

Optimum guess for the highest eigenfrequency would be: 18.3438.

You can set this guess in the special settings dialog box, which can be opened by clicking the *Specials* button in the solver control dialog box. In the *Guess* field you should enter the proposed guess as 18.3438 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.

Results

You can access the eigenmode solver results for the Nth 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

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

For CPU acceleration and distributed computing options choose *Home: Simulation \Rightarrow Setup Solver \Rightarrow Eigenmode Solver*  \Rightarrow Acceleration. Please refer to the chapter **Acceleration Features** or to the online help for more detailed information about the different acceleration features.

Workflow Summary

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.).

Partial RLC Solver

The Partial RLC Solver can be used for calculation of equivalent circuit parameters (partial inductances, resistances, and capacitances) and features optional SPICE export. It is of use in the following fields of application, e.g., printed circuit boards, packaging problems, and network parameter extraction.

The excitation for this solver is different from the other solvers and uses RLC nodes instead of ports. Please refer to the online documentation for information on how to set up models for this solver.

Choosing the Right Port Type

The proper definition of ports is essential for accurate S-parameter computations. In measurement set-ups, 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.

In general, the same problems exist for EM field simulations. 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, three types of ports need to be distinguished:

1. Discrete edge ports
2. Discrete face ports
3. Waveguide ports

Discrete edge ports can be seen as lumped circuit elements with an internal resistor and a current source in parallel. Depending on the solver type, these ports consist of a single lumped element in the middle and two perfectly electrically conducting wires connecting the port to the structure or a distributed lumped element along the complete edge. A certain voltage / current relation is then introduced across the lumped element, and the S-parameters are calculated

based on the element's currents and voltages. Any discrete port can also be defined as a current or voltage source.

Discrete face ports are very similar to the discrete edge ports described above. The major difference is that this lumped element is represented by a face rather than an edge. Again, depending on the solver type, these ports consist of a single lumped element in the middle and two perfectly electrically conducting faces connecting the port to the structure or a distributed lumped element over the complete face area. The advantage of the latter type of connection is that the port has a lower self-inductance.

It is important to note that there may be fringing effects at the transition between the structure and the discrete port (of either type). This will always be the case when the geometry of the structure's transmission lines is different from the geometry of the discrete ports, that is, in most cases. Please note that discrete face ports typically introduce smaller discontinuities than discrete edge ports when connected to stripline or microstrip type structures.

Despite these shortcomings, 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.

CST Studio Suite uses a 2D eigenmode solver to calculate the relevant mode patterns in the port plane. Consequently, the definition of waveguide ports requires enclosing the entire field filled domain in the cross section of the port area. This general approach allows the accurate modeling of arbitrary port types, like empty or coaxial waveguides, microstrip or coplanar lines and even more complex setups like multi-conductor, single-ended or periodic waveguide structures. The calculated modes are automatically classified and characteristic properties like wave or line impedance are presented. Please refer to the online documentation for more detailed information.

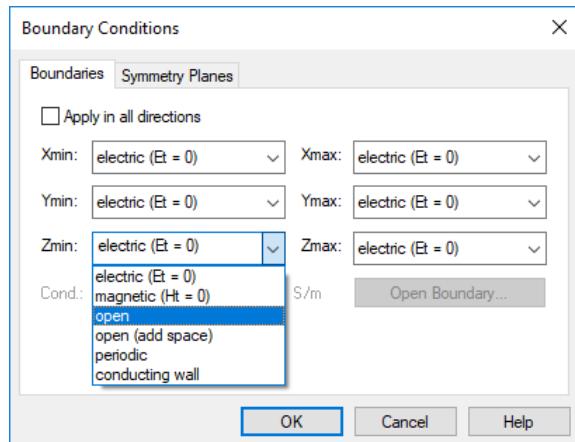
Please note that with the help of the Schematic of CST Studio Suite it is possible to de-embed the port influence from the S-matrix by removing the effect of the port to structure transmission matrix from each of the ports. Please refer to the *CST Studio Suite - Circuit Simulation and SAM (System Assembly and Modelling)* manual for more information.

Please refer to the port overview page in the online help system for more information about all port types.

Antenna Computations

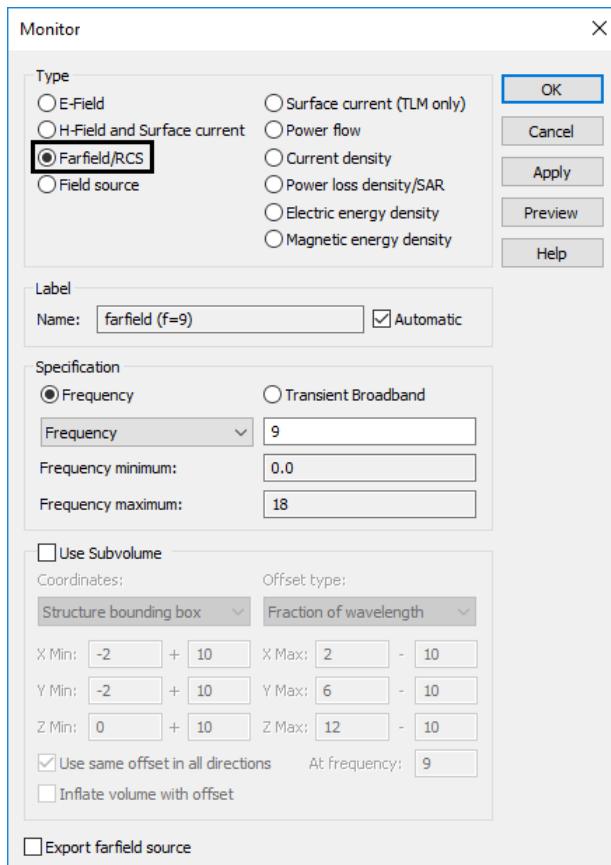
As presented before in the **Which Solver to Use** section, different antenna applications can be optimally solved with appropriate solvers recommended by the configuration wizard. However, some general principles of antenna computations are common, regardless of which solver type is used and will be discussed in the following.

The main difference between an antenna calculation and the S-parameter calculation 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 used. Therefore simply select "open" boundaries in the *Simulation: Settings*  dialog box:



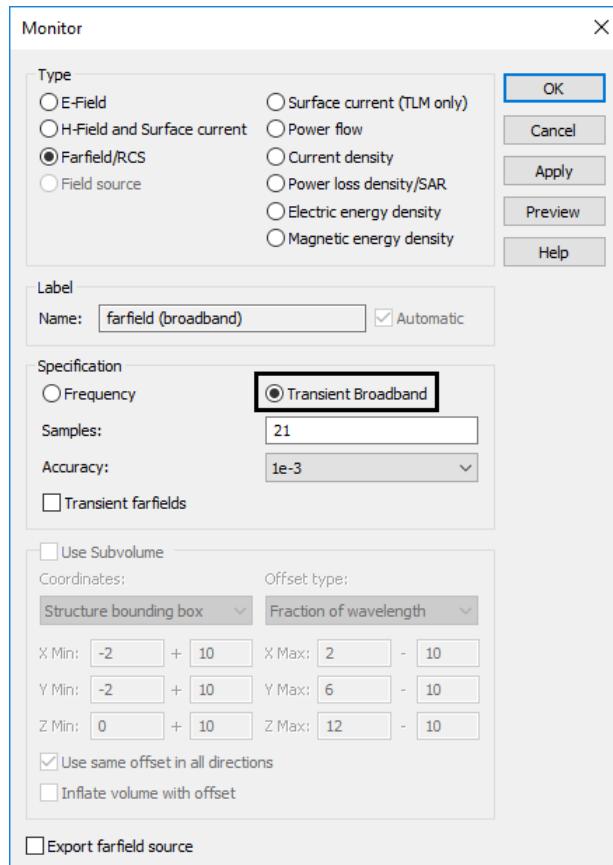
When simulating antennas, open boundary conditions require some space between the device and the boundary planes for optimum performance and accurate farfield calculations. Since the open boundary conditions are very accurate, only a small distance is necessary. However, if you are not sure about the amount of space needed, simply choose “*open (add space)*” from the boundary options. In this case, the necessary space is estimated automatically. The settings for the amount of space to be added can be adjusted in the dialog accessed by the *Open Boundary...* button.

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. Each farfield monitor records the farfield over the sphere in all directions. They can be specified in the *Simulation: Monitors* \Rightarrow *Field Monitor*  dialog box:

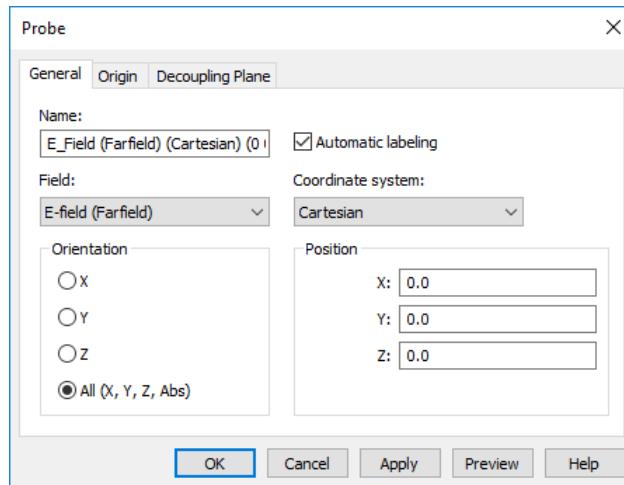


After the simulation is complete you can access your farfield results from the *NT: Farfields* folder. Typical antenna characteristics such as main beam direction, gain, efficiency, side lobe suppression, etc. are automatically calculated and displayed. Please refer to the online help tutorial **Patch Antenna** for more information.

As mentioned above it is possible to define farfield monitors at selected frequencies. However, if you are using the transient solver and are interested in the farfield behavior over a wide frequency range you have the options of either defining a broadband farfield monitor or to use farfield probes. Similar to the frequency farfield monitors, the broadband monitor calculates the farfield data for the full angular range (θ , ϕ) and is activated by checking the corresponding option in the monitor dialog box:

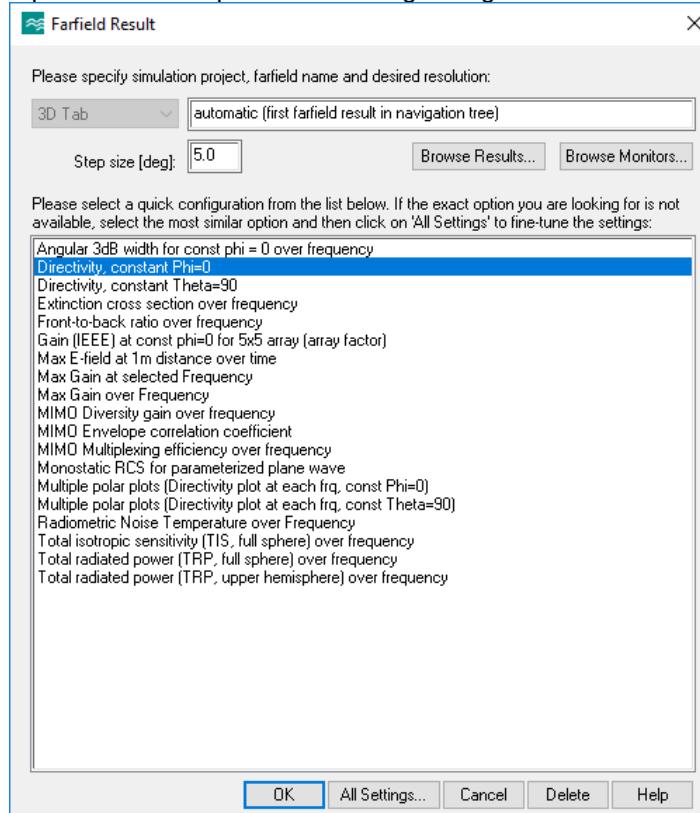


Some applications require the farfield information only at a few (θ , ϕ) locations. In such cases it may be advantageous to use farfield probes: *Simulation: Monitors* \Rightarrow *Field Probe* , *Field = E-field (Farfield) or H-field (Farfield)*:

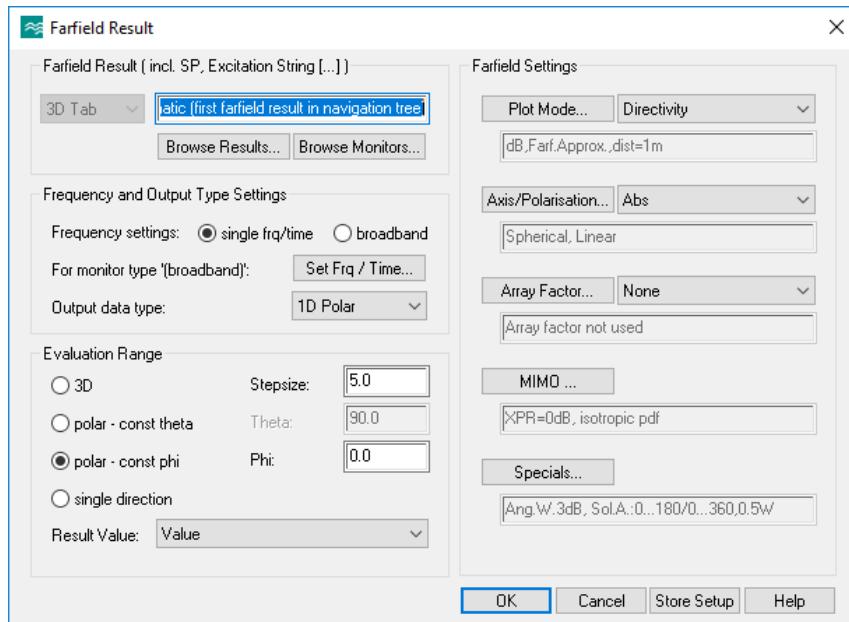


In this dialog box, you can specify the type of farfield, the location and the orientation of the desired probe in Cartesian, spherical or Ludwig coordinate systems. Please refer to the online documentation for more information about this feature.

Another very interesting functionality is the use of result templates in combination with farfield calculations. The basic functionality of result templates has already been demonstrated in the previous example. There are also some automated farfield templates available when selecting *Farfield and Antenna Properties* from the Select Template Group dropdown list (*Post-Processing: Tools* \Rightarrow *Result Templates*). Choosing the *Farfield Result* template from the *Add new post-processing step* dropdown list will open the following dialog box:



Here you can select one of the pre-configured farfield results. However, if needed the corresponding settings can be adjusted in detail by pressing the *All Settings* button:



You can now select which one of the previously defined farfield monitors should be processed with an already performed excitation (e.g. [1] corresponds to excitation at port 1, and [pw] corresponds to a plane wave excitation). You can change several farfield settings such as the farfield component, the polarization, the coordinate system or even an antenna array setup. Finally, the modified settings can be stored as a new configuration by selecting *Store Setup* button.

The result of this farfield processing template is either a single result curve or a 0D value which can then be further processed by other result templates or simulation steps. As an example, you could extract the location of a certain farfield maximum by using a 0D result template and then use this value for an optimization of the main lobe direction to a certain angular location or magnitude. Please refer to the online help system 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 appropriate 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.

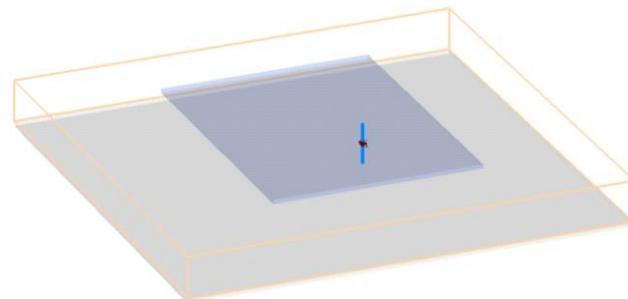
In cases where you are able to use a simplified model, you can use discrete ports rather than waveguide ports (please refer to the **Choosing the Right Port Type** section earlier in this chapter).

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

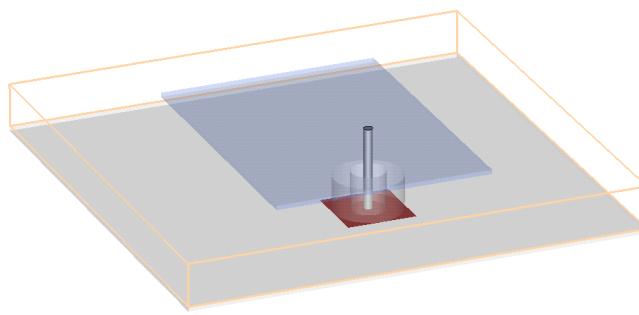
results quickly. This will allow you to assess the principles of operation of the antenna before optionally increasing the accuracy by constructing a detailed model of the feeding point geometry.

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



b) Detailed model of the feeding point using a waveguide



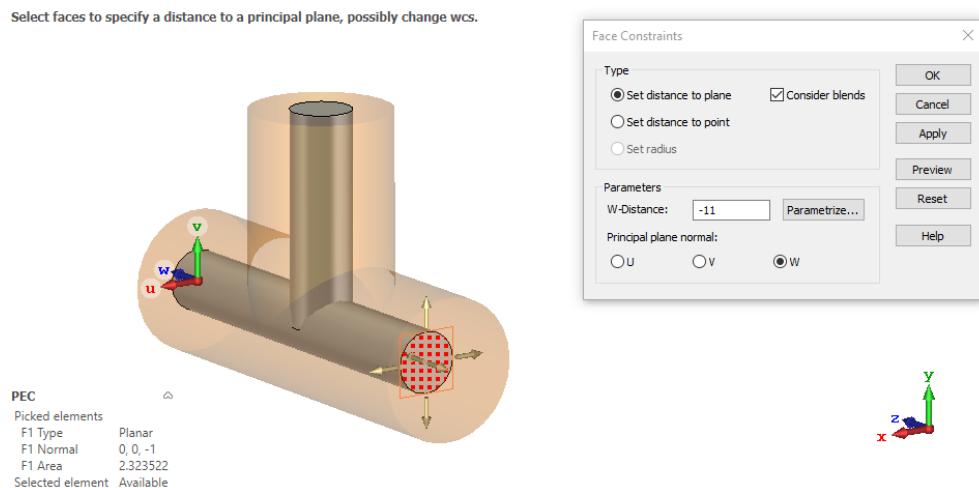
In picture a) the antenna is fed by a discrete edge port which represents a current source with an internal resistance. This approach delivers accurate farfield results but may yield S-parameters which are not directly comparable to the measurements.

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.

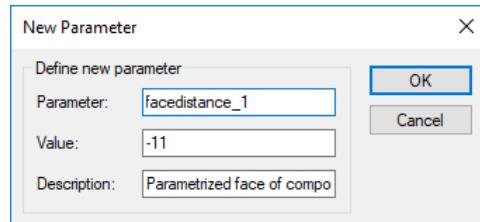
Sensitivity Analysis

Derivatives of S-parameters and of other network characteristics such as Y- and Z-parameters with respect to geometric and/or (simple) material parameters can be calculated via the so-called "sensitivity analysis". This functionality is available with different feature sets for the tetrahedral frequency domain solver as well as for the hexahedral transient solver. The eigenmode solver with tetrahedral mesh can calculate derivatives of the modes' frequencies in the course of the sensitivity analysis.

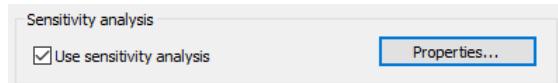
Referring to the coaxial connector example of chapter 2, you can define a face constraint by first selecting the corresponding end face of the inner conductor stub, then defining a geometric face constraint (*Modeling: Tools \Rightarrow Modify Locally \Rightarrow Define Face Constraints*):



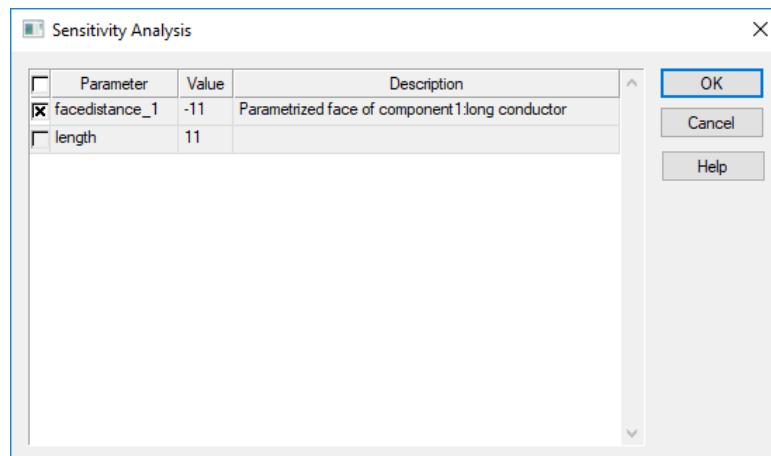
Keep the default selection of *Set distance to plane* to define the new face constraint as the distance of the face to the local coordinate system in *w*-direction. Before closing the dialog, please click on the *Parameterize* button to define a new correspondent parameter with the initial values as shown below:



In the following the sensitivity analysis is performed with the tetrahedral frequency domain solver. In order to consider sensitivity results during the simulation, the *Use sensitivity analysis* box at the bottom of the solver dialog has to be activated:

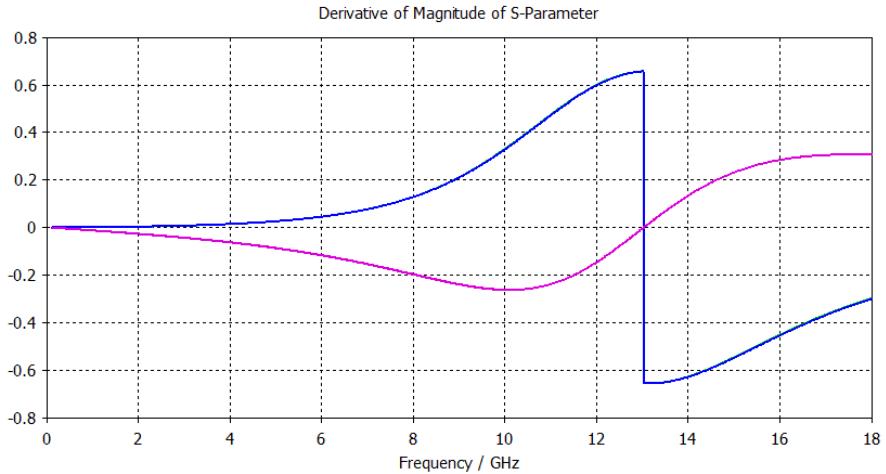


Press the *Properties* button to see the list of parameters that are currently available for the sensitivity analysis. In this case geometric parameter "length" is not available for the sensitivity analysis since it was not defined as a face constraint.

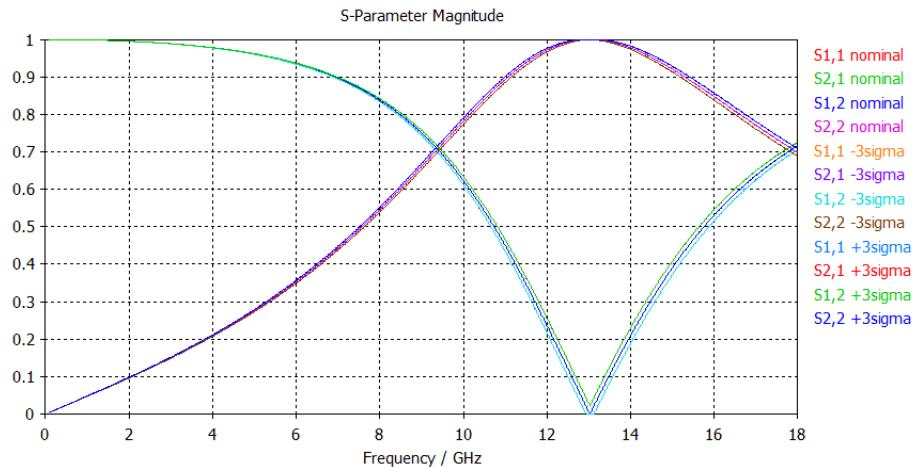


With knowledge of the nominal value and of the first derivative, the sensitivity (i.e. the variation of a network parameter with respect to a design parameter) can be calculated in a small

neighborhood of the nominal value. The results will be displayed in the navigation tree *NT: 1D Results* \Rightarrow *S-Parameter Sensitivity* in separate folders for each design parameter.



As a post-processing step, a yield analysis can be performed using the sensitivity data calculated in the solver run. Select *Post-Processing: Signal Post-Processing* \Rightarrow *Yield Analysis* and find the results again in the navigation tree *NT: 1D Results* \Rightarrow *S-Parameter Yield*:



Please consult the online help for further details about the sensitivity and yield analysis.

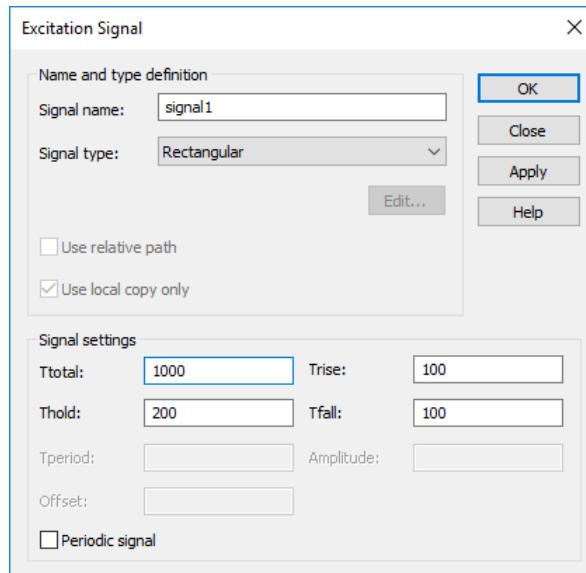
Digital-Signal Calculations

A digital-signal 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.

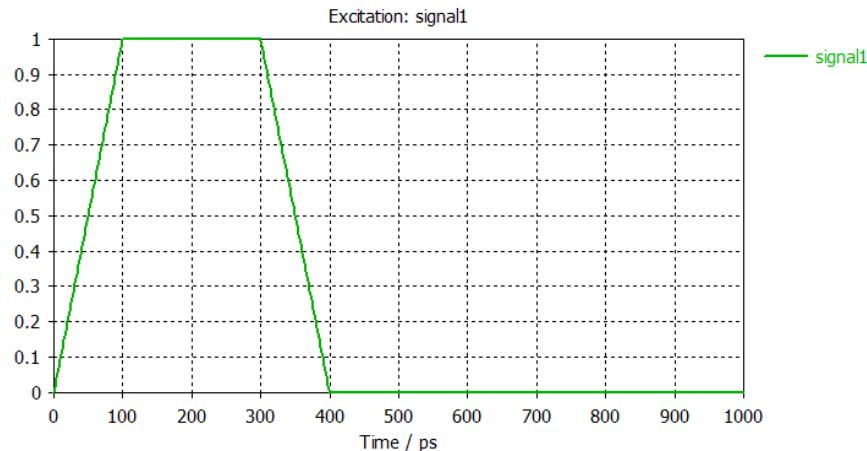
For S-parameter calculations the excitation signal for the transient analysis is typically defined by a Gaussian pulse for which the Fourier spectrum is also given by a Gaussian pulse covering the entire frequency band of interest. Therefore, the time signal is determined mainly by the frequency band.

By 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 define a new excitation signal by clicking on *NT: Excitation Signals* and selecting *New Excitation Signal* from the context menu to open the following dialog box:

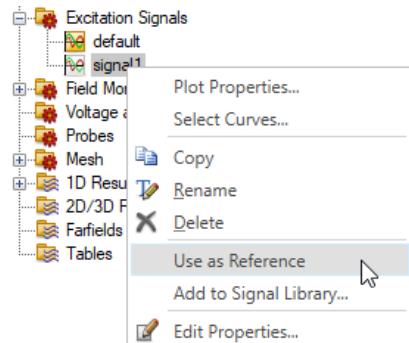


In the example studied above (with the time unit set to *ps*) the settings define a rectangular shape with a rise-time of 100 ps, a hold-time of 200 ps and a fall-time of 100 ps. The rise- and fall-times of 100 ps correspond to a bandwidth of approximately 10 GHz. The maximum simulation time is given in the *Ttotal* field and is set to 1000 ps in this example. For manually defined excitation signals, 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*:

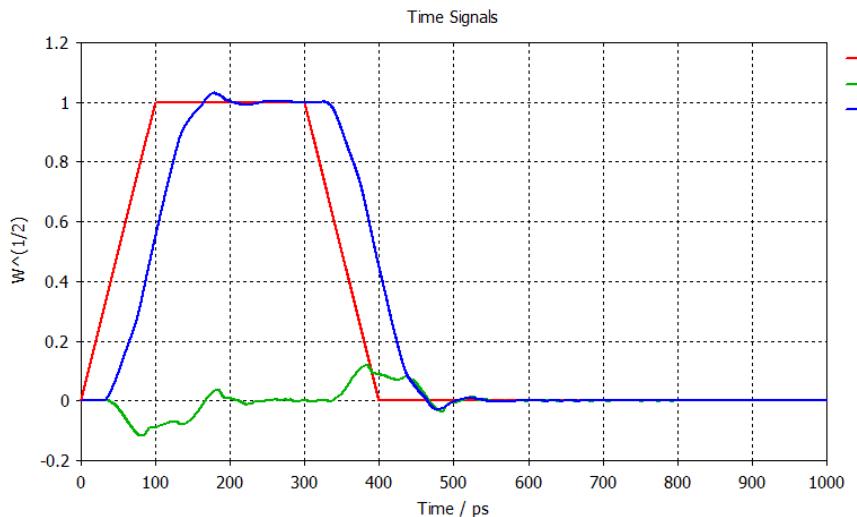


You can now define the rectangular signal *signal1* as the reference signal by selecting *Use as Reference* from the context menu:



The new reference signal is now used for all subsequent transient simulations. However, you can also specify additional excitation signals in order to excite different ports with individual excitation signals. Please refer to the online documentation for more information about this feature.

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 "o_{2,1}" 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. Please 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 which are especially dedicated to digital simulations such as eye diagram computations (*Home: Macros* \Rightarrow *Results* \Rightarrow *Eye Diagram, TDR, etc.* \Rightarrow *Eye Diagram*) or the exchange of excitation signals after the simulation (*Home: Macros* \Rightarrow *Results* \Rightarrow *Eye Diagram, TDR, etc.* \Rightarrow *Exchange Excitation*).

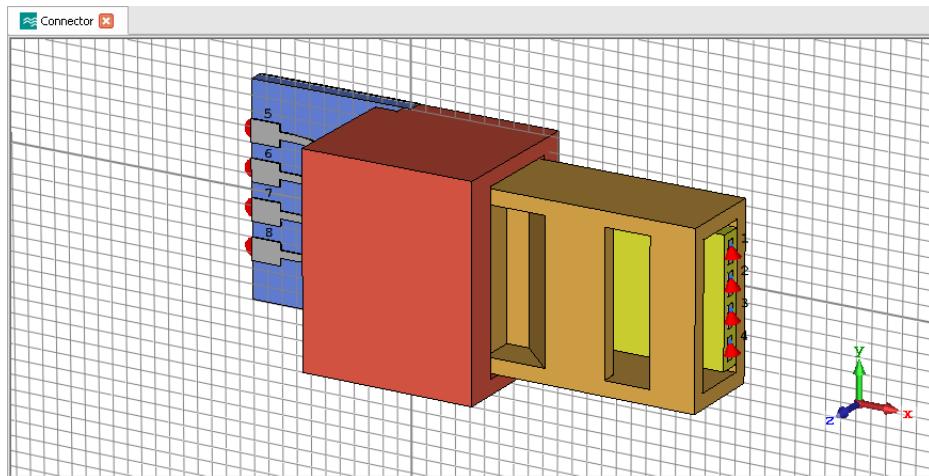
Coupled Simulations

The smooth interaction between different modules or solvers of CST Studio Suite allows for a straightforward coupling of 3D EM simulation with other simulation methods.

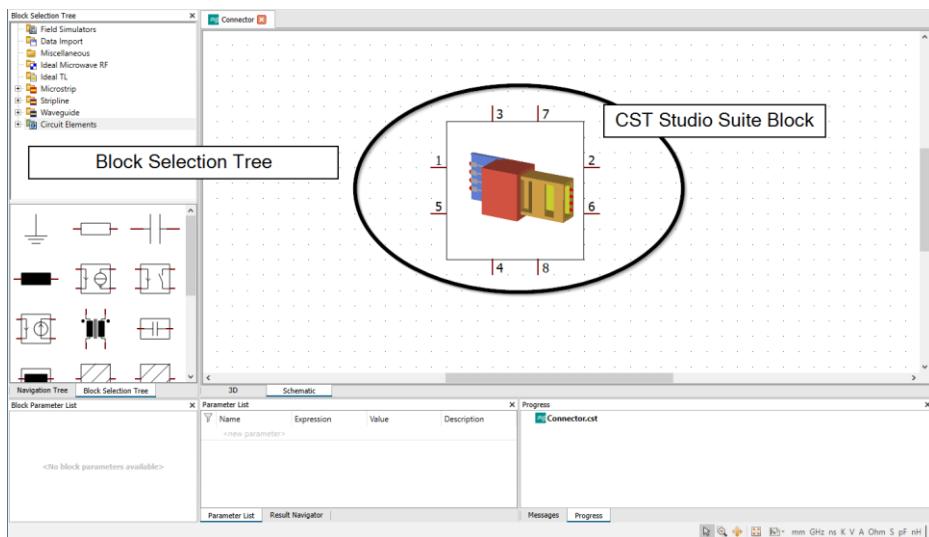
Adding Circuit Elements to External Ports

For each 3D EM simulation setup inside CST Studio Suite two fundamentally different views of 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 (CST Studio Suite block) with terminals:



The terminals have a one-to-one correspondence with the 3D structure's waveguide modes or discrete ports. The schematic view now allows for easy addition of external circuit elements to the terminals of the 3D structure. The connection of these arbitrary networks to the 3D model can either be realized as a standard or a transient EM/circuit co-simulation.

Please refer to the online help system and the *CST Studio Suite - Circuit Simulation and SAM (System Assembly and Modelling)* manual for more information about this topic.

Coupled Simulations between High Frequency Solvers

In order to establish a coupling between 3D high frequency solvers inside CST Studio Suite, near- or far-field data from one solver can be reused as field sources in another solver. This can be useful for antenna placement or EMC radiated emission simulations or to exchange component information without exchanging the model itself. A special *Hybrid Solver Task* is available to simplify the setup of such workflows.

Please refer to the *Field Source Overview* page in the online help description and to the *CST Studio Suite – Circuit Simulation and SAM (System Assembly and Modelling)* document for more detailed information about this topic.

Coupled Simulations with Thermal or Mechanical Solvers

Field monitor results from a high frequency transient, eigenmode or frequency domain solver can be used as a source for thermal simulations. Furthermore, based on these results, a subsequent stress simulation can be performed. The feedback effect of the stress simulation on the EM simulation can then be considered when performing a sensitivity analysis with the frequency domain or eigenmode solver with a tetrahedral mesh.

Please refer to the *CST Studio Suite - Thermal and Mechanical Simulation* document for more detailed information about this multi-physics workflow.

Furthermore, thermal results from the thermal solvers can be used to simulate the effect of temperature dependent material parameters in some of the high frequency time and frequency domain solvers.

Please refer to the *Material Overview* page in the online help description for more detailed information about this topic.

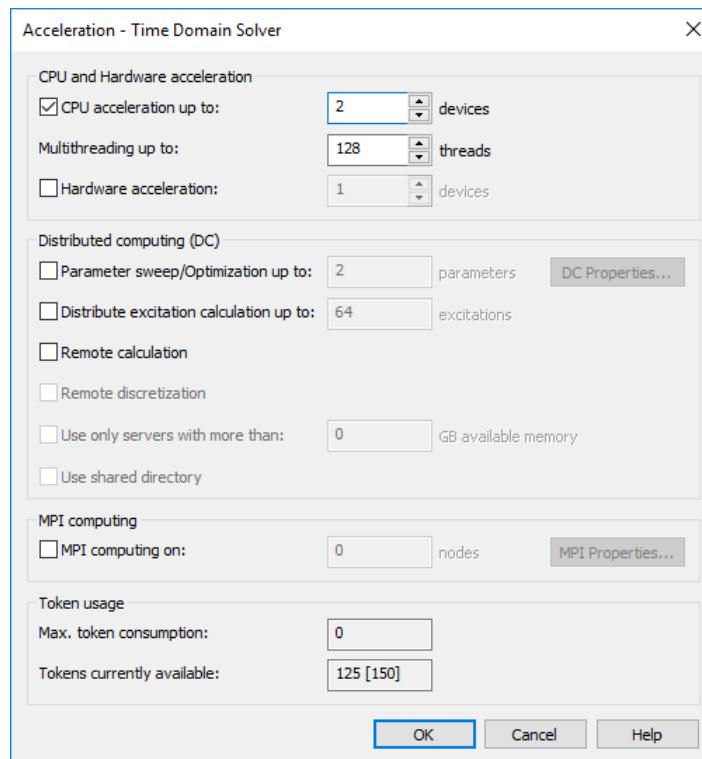
Coupled Workflow with Cable Simulation

Hybrid simulations considering radiation from and irradiation into a cable can be performed using the high frequency time domain 3D field solvers together with the cable modeling tools inside CST Studio Suite. Unidirectional coupling is either done in the frequency or in the time domain, while bi-directional coupling is available when doing a transient simulation.

Please refer to the *CST Studio Suite - Cable Simulation* document for more detailed information about this simulation type.

Acceleration Features

In addition to optimization and parameter sweep techniques, CST Studio Suite offers other more hardware related possibilities to accelerate the simulation. In the case of the transient solver choose *Simulation: Solver* \Rightarrow *Setup Solver*  \Rightarrow *Acceleration* in order to specify the control for CPU and hardware (GPU) acceleration, distributed computing options, as well as MPI computing settings.



Similar options are available for the other solvers in CST Studio Suite. Please refer to the online help (section **Simulation Acceleration**) for more detailed information about the supported solvers and the different acceleration features.

Chapter 4 – Finding Further Information

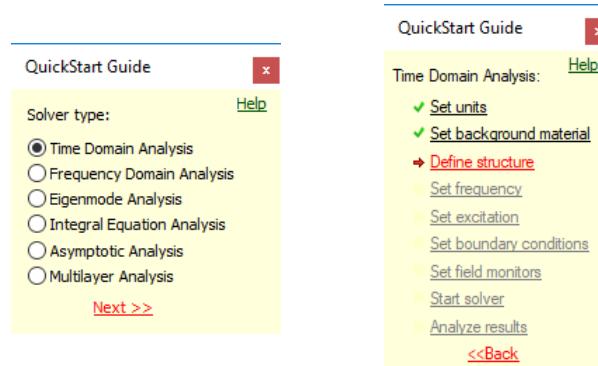
After carefully reading this manual, you will already have some idea of how to use CST Studio Suite efficiently for your own high frequency simulations. However, when you are creating your own first models, some questions will arise. In this chapter, we give you a short overview of the available documentation.

The QuickStart Guide

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

The QuickStart Guide is opened automatically on each project start if the checkbox *File: Options* \Rightarrow *Preferences* \Rightarrow *Open QuickStart Guide* is checked. Alternatively, you may start this assistant at any time by selecting *QuickStart Guide* from the Help button  in the upper right corner.

When the QuickStart Guide is launched, a dialog box opens showing a list of tasks, where each item represents a step in the model definition and simulation process. Usually, a project template will already set the problem type and initialize some basic settings like units and background properties. Otherwise, the QuickStart Guide will first open a dialog box in which you can specify the type of calculation you wish to analyze and proceed with the *Next* button:



As soon as you have successfully completed a step, the corresponding item will be checked and the next necessary step will be highlighted. You may, however, change any of your previous settings throughout the procedure.

In order to access information about the QuickStart Guide itself, click the *Help* button. To obtain more information about a particular operation, click on the appropriate item in the QuickStart Guide.

Online Documentation

The online help system is your primary source of information. You can access the help system's overview page at any time by choosing *File: Help* \Rightarrow *Help* . The online help system includes a powerful full text search engine.

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 block is selected, you will obtain some information about the block's properties.

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

Please refer to the *CST Studio Suite - Getting Started* manual to find some more detailed explanations about the usage of the CST Studio Suite Online Documentation.

Tutorials and Examples

The component library provides tutorials and examples, which are generally your first source of information when trying to solve a particular problem. See also the explanation given when following the *Tutorials and Examples Overview* link  on the online help system's start page. We recommend that you browse through the list of all available tutorials and examples and choose the one closest to your application.

Technical Support

Before contacting Technical Support, you should check the online help system. If this does not help to solve your problem, you find additional information in the [Knowledge Base](#) and obtain general product support at [3DS.com/Support](#).

Macro Language Documentation

More information concerning the built-in macro language for a particular module can be accessed from within the online help system's VBA book: Visual Basic (VBA) Language. The macro language's documentation consists of four parts:

- An overview and a general description of the macro language.
- A description of all specific macro language extensions.
- A syntax reference of the Visual Basic for Applications (VBA) compatible macro language.
- Some documented macro examples.

History of Changes

An overview of important changes in the latest version of the software can be obtained by following the *What's New in this Version* link  on the help system's main page or from the *File: Help* backstage page. Since there are many new features in each new version, you should browse through these lists even if you are already familiar with one of the previous releases.