CST DESIGN STUDIO

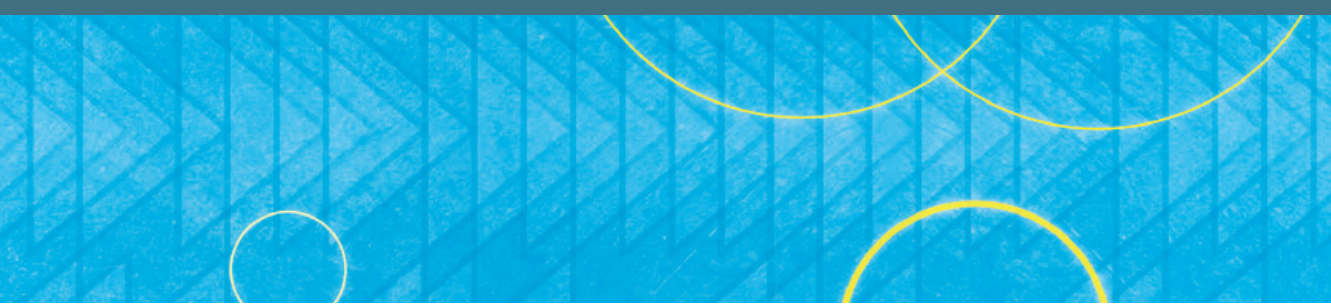**Workflow**

Contents

Chapter 1 — Introduction

Welcome

Welcome to CST DESIGN STUDIO™, the powerful and easy-to-use schematic design tool built for fast synthesis and optimization of complex systems. The tight integration with our electromagnetic field simulators allows considering systems at different levels of detail and takes into account various effects.

CST DESIGN STUDIO is part of the CST STUDIO SUITE®. Please refer to the CST STUDIO SUITE *Getting Started* manual first. The following explanations assume that you already installed the software and familiarized yourself with the basic concepts of the user interface.

Within CST STUDIO SUITE, CST DESIGN STUDIO appears in two different flavors:

- ☐ As a stand-alone tool. It runs independently, without any connections to a specific field simulator project.

- ☐ As an associated view to a CST MICROWAVE STUDIO®, CST CABLE STUDIO®, CST PCB STUDIO®, CST EM STUDIO® or CST PARTICLE STUDIO® project. It represents the schematic view that shows the circuit level description of the current field simulator project.

All steps necessary to set up a simulation in CST DESIGN STUDIO are identical for both flavors described above.

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 should provide you with all the basic information necessary to understand the advanced documentation.

3. Look at the examples folder in the installation directory. The different application types will give you a good impression of what has already been done with the software. Please note that these examples are designed to give you a basic insight into a particular application domain. Real-world applications are typically much more complex and harder to understand if you are not familiar with the basic concepts.

4. Start with your first own example. Choose a reasonably simple example that will allow you to become familiar with the software quickly.

5. After you have worked through your first example, contact technical support for hints on possible improvements to achieve even more efficient usage of CST DESIGN STUDIO.

What is CST DESIGN STUDIO?

CST DESIGN STUDIO is a schematic design tool for system level simulation. Several component libraries are available based on analytical and semi-analytical models. Library models can be expanded by means of field simulators of CST STUDIO SUITE. Measured data are taken into account as TOUCHSTONE or SPICE blocks and the support of the IBIS standard allows an easy I/O device description. A vendor library of linear and non-linear components helps to easily set up a design.

CST DESIGN STUDIO is tightly integrated to various electromagnetic (EM) and multiphysics field simulation tools of CST STUDIO SUITE. A hierarchical task concept and a powerful 3D layout generator make CST DESIGN STUDIO a unique product for system assembly and modeling (SAM). SAM technology is able to extract a complex 3D system by connecting its individual components on a schematic level.

Main Applications for CST DESIGN STUDIO

CST DESIGN STUDIO users will take advantage of its versatility and the seamless workflow between a circuit simulator and electromagnetic (or multiphysics) field simulators. Main applications are the following:

- Antenna module design with system performance optimization including matching/driver networks
- Microwave/RF device and system design, applicable for filters, diplexers, phase shifters, high performance RF amplifiers etc.
- Signal Integrity (SI) simulation of packages, 3D connectors and cables, system channels including high speed and control PBCs
- EMC/EMI analysis of complex systems, considering radiation phenomena from and into connected cable harnesses
- Multiphysics simulations like resonator optimizations to compensate the resonance frequency shift due to temperature depending deformation of the resonator geometry, induced by EM material losses

Learn more about applications of CST DS on www.cst.com.

CST DESIGN STUDIO Key Features

The following list provides an overview of CST DESIGN STUDIO main features. Please note that not all options may be available to you due to license restrictions. Please contact your local sales office for details.

User Interface

☐ Supports drag'n'drop for many operations
☐ Intuitive and easy-to-use schematic view, also supporting drag'n'drop for inserting/manipulating circuit elements

Components / Circuit Models

☐ Several analytical components
☐ Comprehensive analytical and 2D EM based microstrip and stripline component libraries
☐ Active, passive, linear and non-linear circuit elements
☐ Support of hierarchical modeling, i.e. separation of a system into logical parts
☐ Tight integration with 3D EM field simulations like CST MICROWAVE STUDIO®, CST CABLE STUDIO®, CST PCB STUDIO®, CST EM STUDIO®, CST PARTICLE STUDIO® and CST MPHYSICS® STUDIO
☐ Import of net lists and semiconductor device models in Berkeley SPICE, Cadence® Pspice® or Synopsis® HSPICE®* format
☐ Support of the IBIS data file format
☐ Import of measured or simulated data in the TOUCHSTONE file format
☐ Control and use of extensible element library

SAM (System Assembly and Modeling)

☐ 3D representations for individual components
☐ Automatic project creation by assembling the schematic's elements into a full 3D representation
☐ Manage project variations derived from one common 3D geometry setup
☐ Coupled multiphysics simulations by using different combinations of coupled circuit/em/thermal/stress projects
☐ Antenna Array Wizard

Analysis

☐ Global parameterization
☐ Flexible and powerful hierarchical task concept offering nested sequence/parameter sweep/optimizer setups
☐ Parameter sweep task with an arbitrary number of parameters
☐ Optimizer task for an arbitrary number of parameters and a combination of weighted goals
☐ Template-based post-processing for user defined result processing

---

* Only available if the HSPICE simulation kernel is used

- ☐ Tuning parameters by moving sliders and immediately updating the results
- ☐ Powerful circuit simulator, offering DC, AC, S-Parameter, Transient and Harmonic Balance simulations
- ☐ Robust and accurate handling of frequency domain data (e.g. S-Parameters) in time domain
- ☐ Mixer and amplifier simulations
- ☐ Input net list file export for HSPICE
- ☐ Result cache for S-Parameters from CST MICROWAVE STUDIO projects
- ☐ Recombination of fields in CST MICROWAVE STUDIO for stimulations calculated in CST DESIGN STUDIO
- ☐ Fast time domain simulation of coupled problems by transient EM/circuit co-simulation with CST MICROWAVE STUDIO
- ☐ Automatic solver choice that automatically selects either an analytic or numerical evaluation of microstrip and stripline components depending on the validity of the analytic models
- ☐ Consideration of higher order modes
- ☐ SPICE model extraction

Visualization

- ☐ Multiple 1D result view support
- ☐ Displays S-Parameters in xy-plots (linear or logarithmic scale)
- ☐ Displays S-Parameters in Smith charts and polar charts
- ☐ Fast access to parametric data via interactive tuning sliders
- ☐ Automatic parametric 1D result storage
- ☐ Measurement functionality inside the views (axis markers, curve markers)
- ☐ Possibility of keeping and comparing results in user-defined result folders
- ☐ Full 3D layout (assembly) viewer

Result Export

- ☐ Export of S-Parameter data as TOUCHSTONE files
- ☐ Export of result data, e.g. 1D plots, as ASCII files

Documentation

- ☐ Creation and insertion of text boxes and images inside the drawing for documentation purposes
- ☐ Annotations inside the data views

Automation

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

About This Manual

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

The main part of the manual is a Quick Tour (Chapter 2) that will guide you through the most important features of CST DESIGN STUDIO. We strongly recommend that you 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 ⇨ Button name ⇨ 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: *Home: Simulation ⇨ Update (Ctrl+F5)*

☐ 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 ⇨ Sub folder ⇨ Tree item*.
Example: *NT: Tasks ⇨ SPara1 ⇨ S-Parameters ⇨ S1,1*

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 local support center. If you don't know how to contact the support center near you, send an email to info@cst.com.

Chapter 2 — Quick Tour

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

This chapter comprises the following sections:

- ☐ Overview of the User Interface's structure
- ☐ Overview of available elements
- ☐ Creating a system
- ☐ Defining simulation tasks and running a calculation
- ☐ Dealing with parameters
- ☐ Performing a parameter sweep and an optimization
- ☐ Viewing and simulating circuits in 3D

The following explanations are useful for users of the CST DESIGN STUDIO (CST DS) stand-alone version as well as for users of CST MICROWAVE STUDIO (CST MWS), CST EM STUDIO (CST EMS), CST PCB STUDIO (CST PCBS), CST CABLE STUDIO (CST CS), CST MPHYSICS STUDIO or CST PARTICLE STUDIO. All these tools offer a schematic view where a circuit model can be constructed. The simulation setup is also identical for all modules.

The only difference between the schematic main view of the stand-alone version and the associated CST DS schematic view is the presence of a predefined block inside the associated CST DS schematic view. This block represents the corresponding 3D simulation (see topics below for additional information).

Overview of the User Interface's Structure

Before we guide you through your first example, we will explain the interface and its main components. We will do this by means of the CST MWS schematic view because it contains additional elements that need to be explained.

If you are using CST DS, you will see a main window similar to the one shown below immediately after you have started the program. If you are using CST MWS or any other CST product with a tight integration to CST DS, you will need to switch to the CST DS view. Please observe the two tabs within the main view:

3D view tab for   Schematic view tab for
CST MICROWAVE STUDIO view   CST DESIGN STUDIO view
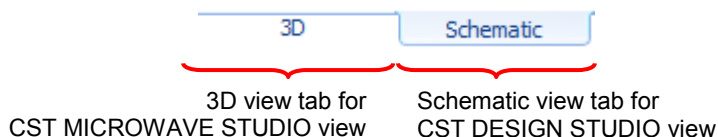
Please click on the *Schematic* view tab now.
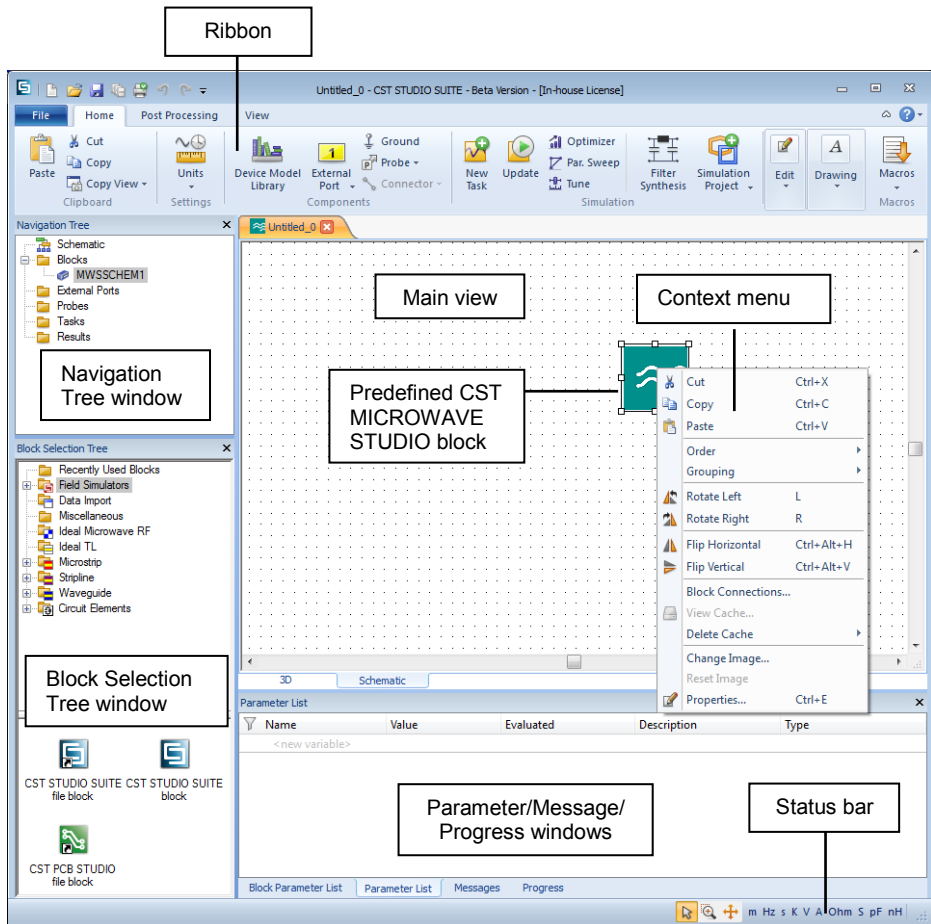
As you can see, the interface consists of several windows:

- ☐ The *main view* consists of a collection of different windows with different views. Each window can visualize a project or any available result. In the above example there are already two windows: The 3D view and the Schematic view. If the views are maximized, they may be selected by the already mentioned view tabs. The contents of a view depend on the selection in the *Navigation Tree* (*NT*).

- ☐ All results and structural details can be accessed through the *Navigation Tree*. It is organized in folders and subfolders with specific contents. When you select an item from the tree, the currently active view visualizes its content in an appropriate manner.

- ☐ The *Block Selection Tree* can be thought of as a library of all elements that are available for creating a design and setting up a simulation. An element may be a circuit element like a resistor or a capacitor, a microwave element, a link to an external simulator, measured S-Parameters, or any other offered element.

- ☐ The *Parameter List* window shows all global parameters that are currently defined. Local parameters of a selected block are accessible through the *Block Parameter List* window.

□   Whenever the program has information for you it will print this text into the *Message* window. It may contain general information, warnings, or errors.

□   The *Progress* window shows the progress of a currently running process, presented by one or more progress bars.

All windows, with the exception of the main view, are freely configurable. You may place them to your favored position. Furthermore, they may be docked into a separate window, as tab into an existing window or removed from the main frame, such that they become a standalone window. The standalone parameter window, for example, looks like this:

▽ Name	Expression	Value	Description	Type
<new variable>				

Parameter List

The next noteworthy element is the *status bar*. The status bar primarily lists the currently selected global units.

The other elements to be mentioned are quite common to all windows programs. The *Ribbon* offers access to the functions of the program (Please have a look into the *CST STUDIO SUITE – Getting Started* document for a more detailed explanation about the Ribbon). Also, quick access to functions is offered in *context menus*. The function types offered in the context menu depend on the current selection (a window, block, navigation tree item or other program elements) and on the current program status.

Overview of Available Components

CST DESIGN STUDIO offers a large variety of elements that can be used to assemble your system. To help you to get started with this collection, this section explains the existing element categories and introduces their most important members.

Components or Circuit Models

A *Component* or a *Circuit Model* implements the physical behavior of a sub-system or represents a lumped circuit element. Throughout the CST DESIGN STUDIO documentation all these elements are referred to as *blocks*. We distinguish between analytical, measured, simulated and some special blocks.

□   Analytical Blocks: Most of the available blocks are analytical or semi-analytical blocks whose physical behaviors are described by parameterized circuit models or mathematical formulas.

□   Measured Blocks: To consider measurement results, CST DESIGN STUDIO offers the TOUCHSTONE block that imports S-Parameters in the well-known TOUCHSTONE format and the IBIS block that interprets the IBIS behavioral descriptions of buffer type components and Capacitance/Inductance/Reactance matrix blocks.

    ☐   Simulated Blocks: These block types reference or store projects of our field simulators e.g. CST MWS, CST CS or external simulators. These blocks keep track of the projects' results and some of them even allow parametric control of the projects from within CST DS. All blocks whose properties can be controlled by free parameters will also be called *parameterized blocks*.

    ☐   Special Blocks: The most important ones are the ground element, the CST DESIGN STUDIO block and the reference block. The ground marks the common ground of a circuit. The CST DESIGN STUDIO block represents a placeholder for a sub-system and therefore supports hierarchical designs. Finally, the reference block defines a common property set that can be assigned to analytical blocks. Reference blocks themselves show no physical behavior.

All these blocks are discussed in more detail in the online documentation.

External Ports

External Ports represent sources or sinks of your system. Depending on the defined task, the type of a source/sink may vary. For instance, a source may deliver a pulse signal for a time domain simulation task and it may deliver a specific complex amplitude for an AC simulation task.

Connectors

A connector is the graphical representation of the electrical connectivity between two elements.

Probes

Probes can be associated with any connector. They record voltages, currents and quantities derived from those for circuit simulation tasks.

Connection Labels

Connection labels are a graphical alternative to connectors. They too define an electrical connection and can be used to make the schematic more readable. They are characterized by a name. All block pins connected to labels with the same name are electrically connected.

Creating a System

Now it is time to create your first circuit in CST DESIGN STUDIO. You will learn how to create a design, how to add components and how to electrically connect them. You will modify the components' properties and use parameters.

The already set up and simulated version of this example, *LumpedFilter.cst* can be found in the *Examples\DS\Workflow* folder of the installation directory.

## Adding and Connecting Components

A simple band pass filter will serve as an example in the following sections. It consists of simple inductors and capacitors that form three resonating elements (LC sections) in so-called pi configuration. The filter's topology is shown below.
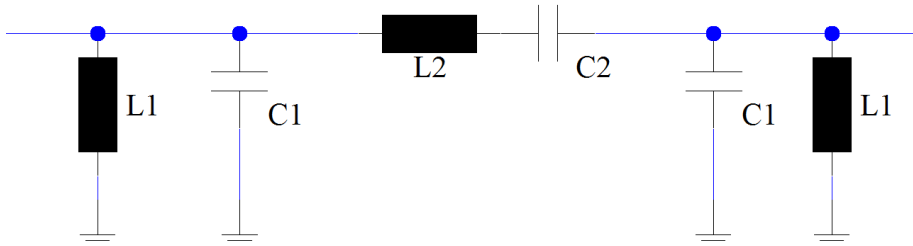
Let us begin the circuit's setup by inserting the first inductor. Select the *Circuit Elements* folder in the block selection tree. You will see all elements collected in this folder in the lower part of the window. Find the symbol for an inductor and press the left mouse button over this type of block. Keep the button pressed, move to the location inside the schematic view where you want to insert the block, and release the button to finish the insertion. During movement inside the schematic view, a template of the component is displayed for better orientation. As shown below, the inserted block is selected and can be moved inside the schematic view.
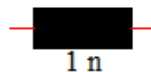
Besides the block symbol in the schematic, a tree item has also been added to the *Blocks* folder of the navigation tree. It has the same name as the block it belongs to. An inductor block's default name is INDn and therefore the added block is called *IND1* unless its name is manually changed afterwards.
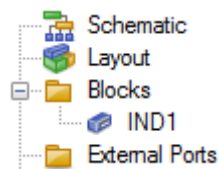

A block tree item may itself contain items. Its sub-items allow the access of block-related results. We will refer to these items later.

A block contains a certain number of internal ports according to the physical behavior attributed to it. These internal ports are the terminals where the block's model can be connected to the outer schematic. The block's symbol represents these ports either by individual pins (normal line) or by bus pins (bold line) that collect several internal ports into one pin symbol.

A pin is represented by a short line adjacent to the block. If it is not connected, this short line is drawn in red, otherwise it is black. Usually, pins have a name to be able to identify them. If a pin represents one internal port, the name is set to the internal port name (except for lumped circuit elements, because these are sufficiently described by their

block images representing them inside the design view). For example, the inserted inductor block has two pins whose short lines are both red because they are not yet connected.

To insert the first capacitor, move the mouse over the left pin of that block inside the *Circuit Elements* folder of the block selection tree. When this pin is highlighted by a red circle, press the left mouse button and drag the capacitor towards the right pin of the previously added inductor. When the two pins contact each other, a red circle is displayed.
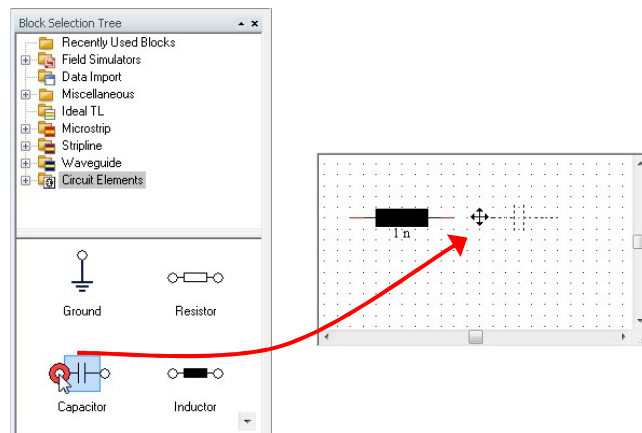
Release the mouse button when the red circle appears. As a result, the capacitor will be inserted and connected with the inductor. A valid connection is indicated by the fact that the colors of the right inductor leg and the left capacitor leg change from red to black.


You may also manually create a connector between two elements. Therefore, place the next inductor by drag'n'drop to the capacitor's right.
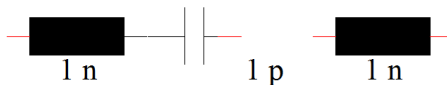
Now move the mouse pointer on the left pin of the right inductor. As soon as the mouse pointer reaches the vicinity of that port it will be highlighted by a red circle and the mouse pointer icon will change to a bordered cross.
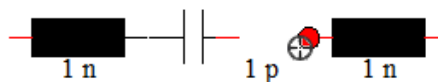
To define the starting point of the connector, double-click on the red circle. The mouse pointer icon changes again and a rubber band line is drawn from the pin to the actual mouse position.
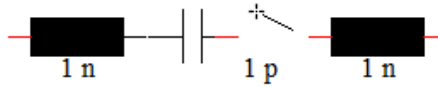
Additionally, some guidance text is displayed in the schematic view, explaining what to do next.

Whenever your mouse pointer meets an element to which the connector can be attached, the element will be highlighted and the mouse pointer icon will change back to the bordered cross. Click on the right pin of the capacitor to finish the connection. As soon as the connector is created it will be drawn as a blue line between the two connected elements.

Please note that a connector has no physical properties, i.e. there is no electrical length associated with a connector. A connector only combines interfaces (i.e. internal ports).

Now, insert another capacitor into your model. Place it to the left of the already inserted components and connect it as shown below:
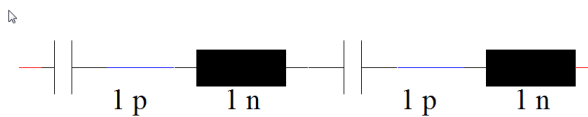
Now rotate the left capacitor and the right inductor by selecting them one after the other and choosing *Home: Drawing ⇨ Arrange ⇨ Rotate Left/Right* , or using the shortcut key *l* or *r* (for left or right). Reposition these two blocks with the arrow keys to obtain the following model:
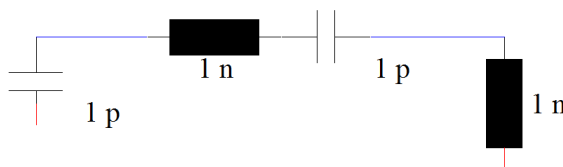
To this point, one pin was connected to exactly one other pin which means that one internal port was connected to exactly one other internal port. We always obtained a one-to-one assignment. However, circuits often have T-junctions or cross-junctions. In CST DS, these junctions are realized by inserting nodes that can be connected to up to four connectors. Such a node is automatically created when you drop a selected pin on a highlighted connection line instead of on another pin, or when you click on a connector while the connector mode is active. Furthermore, if a pin of an element is placed on a connection line, the element will be automatically positioned perpendicular to the connection line in a direction such that the element is moved towards it.

Try this behavior with the next element. Select another capacitor and move it to the design. Initially, it will be horizontally aligned. Now click on a pin of the capacitor in the *Circuit Elements* folder and, in the main window, advance the capacitor from the bottom

to the rightmost horizontal connection line. The element will automatically flip to a vertical orientation, such that the element remains below the horizontal line.
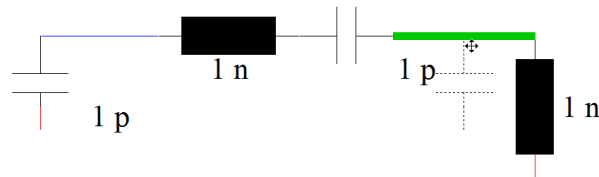

Release the element such that a node will be established. Insert the last inductor and rotate or move the elements until your schematic appears as below:
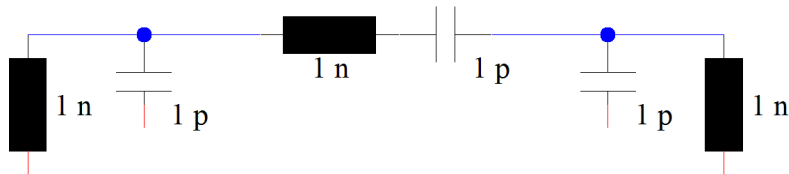
Finally, you need to connect all open pins with ground blocks. You find the ground block in the *Circuit Elements* folder and in the Ribbon under *Home: Components ⇨ Ground* ⏚. However, the fastest method of establishing ground connections is to use the shortcut key *g* when the schematic view is active (you may need to single-click into the schematic view in order to activate it). The shortcut key *g* (for *g*round) creates a ground block with the next mouse click. Try this feature now. After you have pressed *g* you may notice the mouse icon change to a general insertion icon 🖈. Additionally, the same connection highlighting mechanism is activated as for the common drag'n'drop operations.
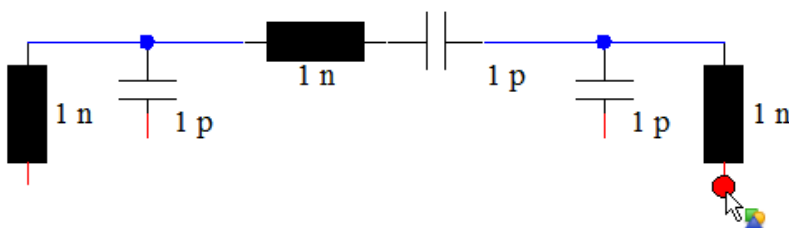

Click on the rightmost open pin and the ground element will be created. Repeat this procedure using the shortcut key *g* with the remaining open pins.
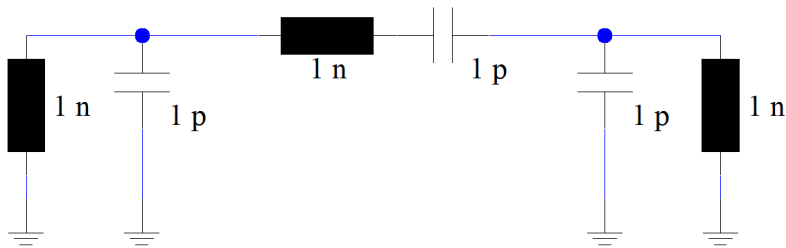
External ports define sources and sinks of a design. Except for some circuit simulation tasks, external ports are required to perform a calculation. This is especially true for S-Parameter calculations.

An external port may be inserted by the same drag'n'drop procedure as for a block. You find the port component in the *Circuit Elements\Sources/Ports* folder of the block selection tree. Alternatively, you will find the port in the Ribbon *Home: Components* ⇨ *External Port* 🔳 or you may take advantage of the shortcut key *p* (for *p*ort) that works very similar to the previously introduced shortcut key *g*. Press *p* and click inside the main view to create the first port at your current mouse pointer position. Create a second port in the same manner and locate the ports as in the picture below. Note that ports are automatically numbered sequentially, starting at one. However, port numbers can be changed by the user as will be explained later.
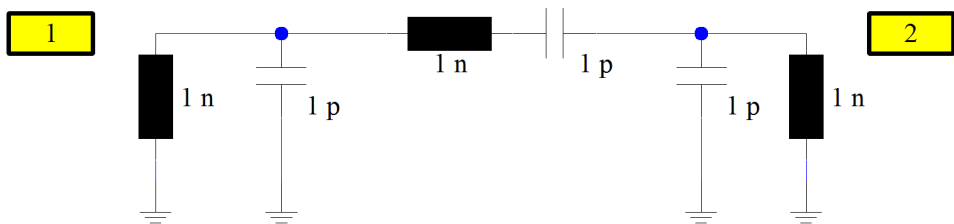
The connection of an external port symbol to a pin or a node is established in the same way as the manual connection of two blocks: Perform a double click on the external port No. 1 and afterwards single-click on the edge of the connection line to its right. Do the same for the external port No. 2.
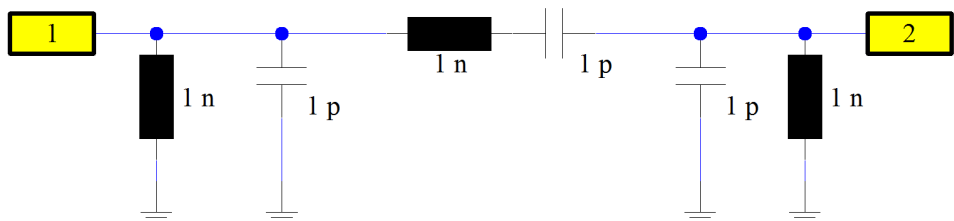
Now, all ports should be connected. Please make sure that there are no red lines in your design view that would indicate unconnected pins. If necessary, repeat one of the actions explained above to establish the missing connection.
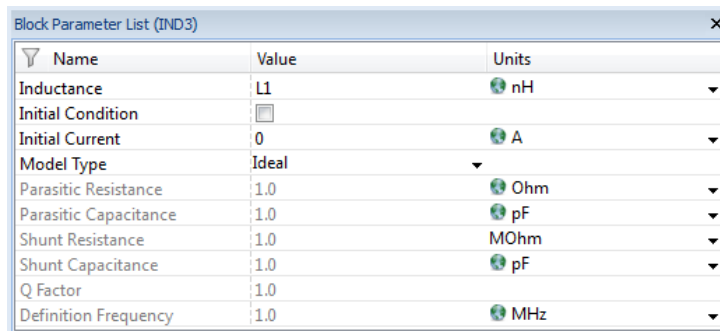
Nodes and edges of connection lines can also be moved manually to different locations. For more complex circuits it may sometimes be useful to modify the automatic layout.

As a summary, all important shortcut keys are listed in the following table:

Shortcut key	Description
g	Place a ground element. Simply click on the end of a connection line after activating this shortcut
p	Place a port element. Click on the end of a connection line to place the port
l	Rotates the selected element counter clockwise
r	Rotates the selected element clockwise
c	Starts the *connector mode*

Changing Properties of a Block

After creating the circuit's topology we want to assign the values for C1, C2, L1 and L2 to the blocks' corresponding properties. This is easily accomplished by selecting a block and editing its properties in the docked parameter window shown below:

The content of the list depends on the block's type. The properties' names should clearly reflect the physical property to which they belong. However, if there are some doubts about the meaning of a property, the online help will provide more information.

You can edit a value associated with a parameter after clicking on it. If you enter an invalid value, an error message will be displayed. By default, the units of all parameters are associated with the global units defined for the project (we will refer to those settings later).

To edit the value for *Inductance*, perform a double-click in the *Value* column of the *Inductance* row. You may also choose a different unit within the selector box in the *Unit* column. However, for our example, keep the default units. Initial conditions may be useful in transient simulation. We do not require them in S-Parameter simulations. Therefore, we leave the *Initial Condition* checkbox and the *Initial Value* row untouched.

Select the blocks one after the other and specify the following values:

Element Name	Value
L1	1.6 nH
L2	44 nH
C1	35 pF
C2	1.2 pF

Now your model should look similar to the picture below. Please make sure that all values have been set correctly.
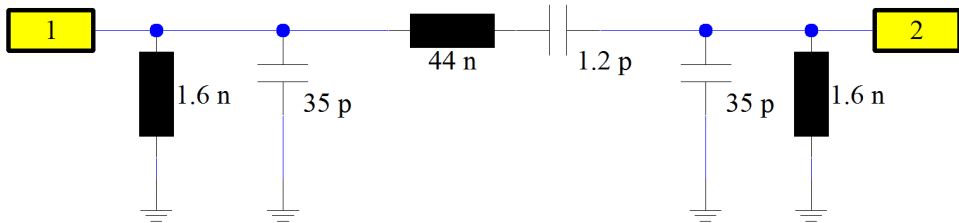
An alternative method of changing the properties of a block is to open the block property dialog box by selecting the block and choosing *Home: Edit* ⇨ *Properties* ⇨ *Properties* or using the block's context menu in the schematic view or in the navigation tree. Furthermore, the block property dialog box can be accessed by double-clicking on a block inside the schematic view.

Now open the block property dialog box of the leftmost inductor. The dialog box generally consists of several sheets whose appearance may differ depending on the type of the selected block. In particular, the blocks that reference a file show some important differences that is referred to in one of the following sections.
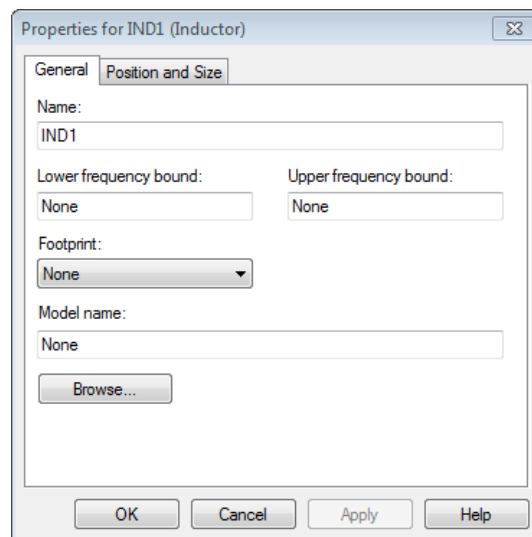

Changing Properties of an External Port

To modify the properties of an external port, open the corresponding property dialog box by selecting an external port and choosing *Home: Edit* ⇨ *Properties* ⇨ *Properties*, or by

double-clicking on the external port inside the schematic view. This dialog box is very similar to the block properties dialog box explained in the previous section.

The port property dialog box contains three tabs labeled *General*, *Font* and *Position and Size*.
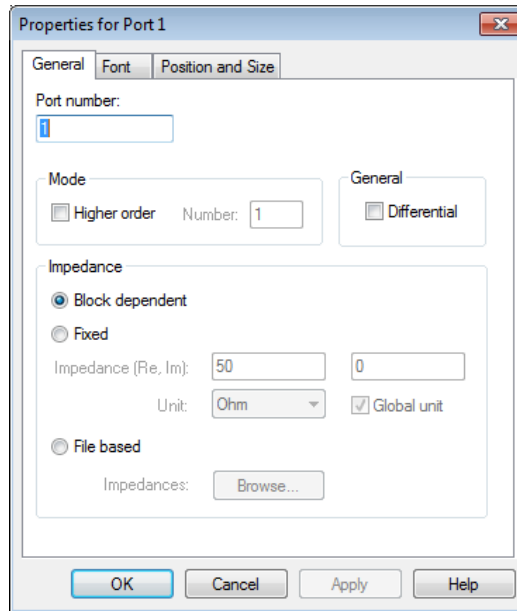

The *General* page displays the port number that can be modified within this dialog box. Note that only positive integer numbers are valid. Additionally, there are three frames:

- The *Mode* frame provides the labeling of the external port with a mode number. If you check the *Higher order mode* box the external port's name will be expanded to p(n), where p is the port number and n the number specified in the *Number* field.

- The characteristic impedance is an important property of an external port. The *Impedance* frame specifies how this impedance will be determined. If the *Block dependent* radio button is enabled (which is the default) the impedance will be set according to the specific impedance of the attached block. However, some blocks do not have a characteristic impedance (e.g. circuit elements like capacitors or inductors). In these cases, the impedance must be set manually. If it has not been set, the external port's impedance will be automatically set to the default value of 50 Ω and the message window will display an appropriate message, indicating this situation. If the *Fixed* radio button is enabled the port impedance will be set according to the manually entered real and imaginary part. If the *File based* radio button is enabled, a frequency-dependent impedance will be read from a TOUCHSTONE file. If the impedance in the two latter cases is different from the impedance of the attached block, the block's S-Parameters will be internally renormalized.

- By default, the common ground (that does not need to be explicitly defined, but can also be a point at infinity) represents the reference node for an external port. For circuit simulations, you might want to define a differential port that refers to a node
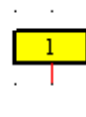
inside your circuit. In this case, switch on the *Differential* property inside the *General* frame. The external port will be expanded by a pin (as shown in the image on the right hand side) to which you can connect the reference node.

Let us return to our example of a band pass filter. This filter has been designed for a 50 Ω environment. Therefore, we will switch on the *Fixed* property for both ports and keep the default value of 50 in the *Impedance* field as shown below.
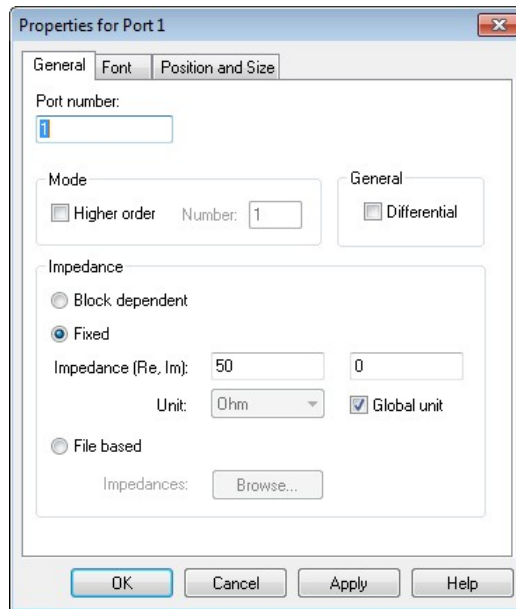

Performing a Simulation

This section will demonstrate how to generate the results that you are interested in. Therefore, global settings are explained and a *simulation task* is defined.
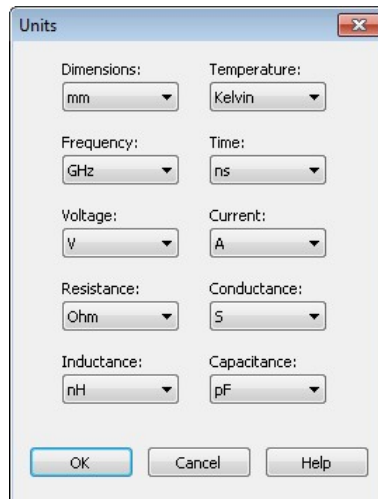
Unit Settings

To this point, we have assigned some values to the element's properties and decided to keep their association with the global project units. Therefore, if you change the global inductance unit e.g. from nH to uH, you scale all inductances referring to the global unit by a factor of 1000, because the values assigned to the properties are retained. To avoid this scaling you may select local units for each block.

You can check whether a property refers to the local or to the global unit by having a look in the block property dialog box:
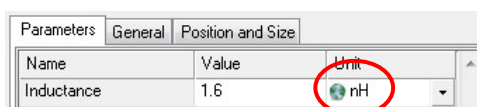
If you see the *world* icon 🌐 in the *Units* drop-down list, global unit will be taken for this value.

The global units currently used in your project are displayed in the status bar. They can be modified from the *Units* dialog box. To open it, choose *Home: Settings ⇨ Units* 🕐.


In our example, all inductances are given in nH and all capacitances are given in pF, which are the default settings for *Inductance* and *Capacitance* properties. However, our circuit should operate in the MHz / µs range. So please change the *Frequency* and *Time* settings accordingly and press the *OK* button. Please note, that in the status bar the frequency and time units have changed correspondingly.

Defining Simulation Tasks

In order to obtain information about the filter's characteristics, we intend to calculate the S-Parameters for our design.

To define a new task, choose *Home: Simulation ⇨ New Task* 📊.
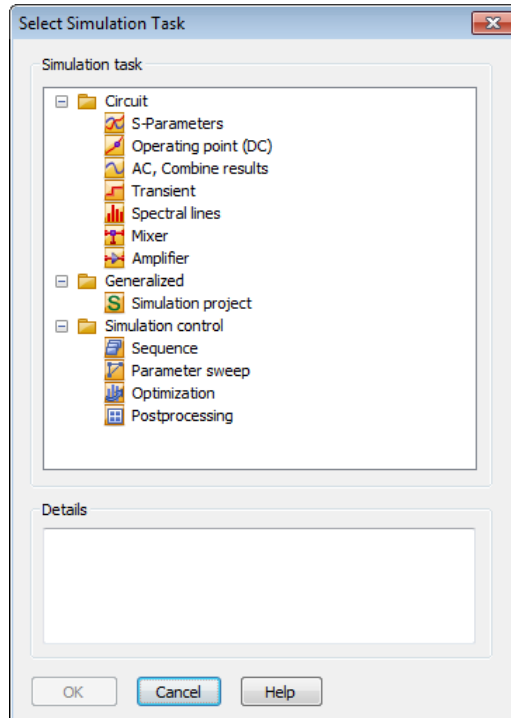
The *Select Simulation Task* dialog box shows a tree view of all available tasks. The *Details* frame displays some information about the selected task.

As you can see, the S-Parameter calculation is only one of several tasks that can be performed by CST DESIGN STUDIO. You can find a detailed explanation of all these simulation methods in the online documentation.

Depending on the selected task, another dialog box will be opened (after pressing the *OK* button) in which you can define the specific settings.

Select *Circuit ⇨ S-Parameters* and press the *OK* button. The *S-Parameter Simulation Task* dialog box will be opened where you can input S-Parameter specific settings.
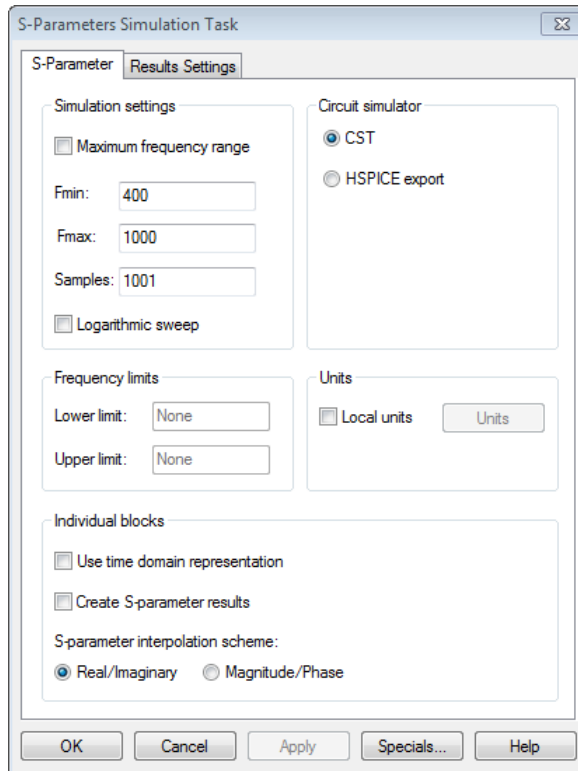
The dialog has two tabs. The *S-Parameter* tab, in which you can specify the simulation settings and the *Results Settings* tab, in which you can specify which results are to be calculated.
We stay with the defaults in the *Results Settings* tab. S-Parameters, port impedances, and balances will be calculated.

The *S-Parameter* tab has five frames:

☐   Inside the *Simulation Settings* frame the frequency range for the S-Parameter calculation and the number of frequency samples are specified. There is a check box labeled *Maximum frequency range*. If this property is switched on, the maximum valid frequency range will be used for the simulation. Note that frequency bounds must be shown in the *Frequency limits* frame if you choose this option. If there is a valid frequency range, this option is switched on by default. In addition to this control, there are three edit fields: The *Fmin* and *Fmax* fields are only editable if the *Maximum frequency range* option is switched off. There, you can enter values that must be within the range given by *Lower limit* and *Upper limit*. Finally, you should specify the number of frequency samples to consider in the *Samples* edit field. You may also choose the *Logarithmic sweep* option to perform a logarithmic sweep instead of a linear sweep inside the specified frequency range. However, *Fmin* must not be zero in this case.

☐   Inside the *Circuit simulator* frame you may choose to perform a CST simulation or export the circuit from the schematic into a HSPICE® netlist file.

☐   Inside the *Frequency limits* frame, the largest frequency range for which the model is valid is displayed. If your model does not contain any frequency-bound

blocks, *None* is displayed for the lower and upper limits. Otherwise, the range represents the intersection of all block ranges.

☐ Inside the *Units* frame you may select the frequency unit that all task properties refer to. By default, the global frequency unit is set there, but you can also choose a local unit that is different from the global unit.

☐ Inside the *Individual blocks* frame you may choose to store the S-Parameter results for the individual blocks that are calculated while the simulation task is running. Depending on the number of blocks that your model contains, this option may slow down the simulation significantly. Furthermore, you can choose the interpolation scheme for the imported blocks' data here. The selection of *Real/Imaginary* may lead to small inaccuracies for the amplitude and phase and vice versa.

As the frequency range of interest we choose $400 \leq f / \text{MHz} \leq 1000$, i.e., we specify 400 for *Fmin* and 1000 for *Fmax*. We keep the default value of 1001 for *Number of frequency samples*. Press *OK* to confirm your settings.

The task item *S-Para1* is added to the *Tasks* folder inside the navigation tree as shown below.
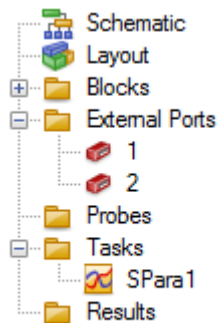
Double-clicking on it reopens the *S-Parameter Simulation Task* dialog box where you can modify the settings for this task. You can also rename or delete it using the item's context menu.

You can add an arbitrary number of tasks to your project. Some of the tasks like *Parameter sweep* or *Optimizer* tasks may also be nested. Each task can be moved or modified from the navigation tree. If you invoke an update of the results for your design, all simulation tasks will be performed one after the other, but you can also update individual tasks via their context menu. If you want to exclude a task from the update loop but do not wish to delete it, you can simply disable it. Therefore, open the tree item's context menu by right-clicking the corresponding task's item and choosing *Disable*. Disabling a task will recursively disable all its child tasks as well. To re-enable it, carry out the same procedure choosing *Enable* instead.

Starting a Simulation

After all required settings have been established, a calculation of the S-Parameters according to the defined task can be performed. Choose *Home: Simulation ⇨ Update*
or use the shortcut key *Ctrl+F5* to update the results of all simulation tasks. As

mentioned above, all simulation tasks are executed one after the other during an update operation. You should examine the message window where information about the simulation progress is displayed in addition to warnings and error messages. For our example, just two lines are displayed indicating the beginning and the end of the execution of the task:

ⓘ Performing task SPara1...

ⓘ Task SPara1 successfully completed.
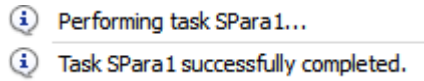
Visualization of the Results

This section will explain how to view results from inside the navigation tree. We distinguish standard results that are automatically generated by CST DESIGN STUDIO from user-defined results that are added by the user. In the user-defined results, single results of the current result set can be inserted.

Standard Results

CST DESIGN STUDIO automatically generates result folders associated with the executed simulation tasks. For instance, for the S-Parameter simulation task, result folders are added that contain S-Parameters, port impedances, and the power balance as a function of frequency for the complete design. The result views of these folders can be activated by clicking on the item inside the navigation tree.
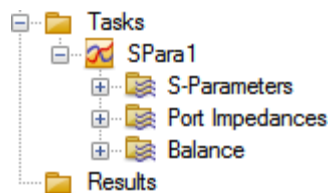
Inside these result folders are tree items that are related to single curves of e.g. the S-Parameters.
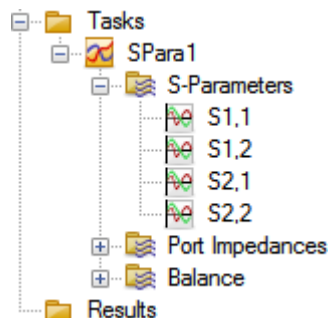

To view the S-Parameters of the complete design, click on the *S-Parameters* folder:
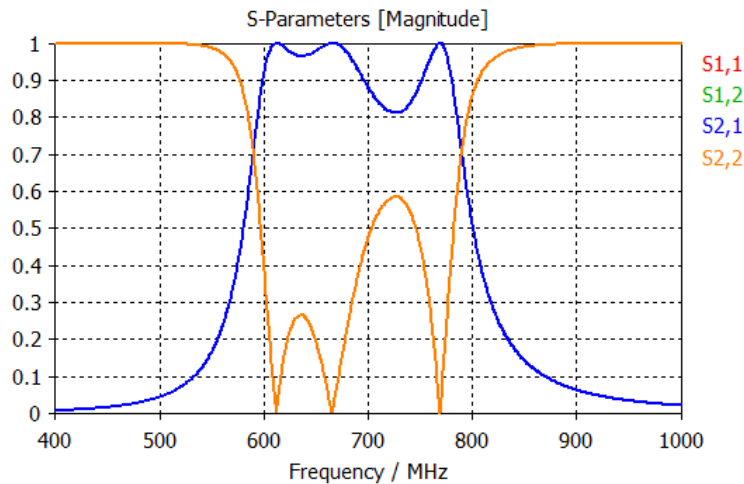
Note that the S11 and S12 curves are not visible here because they are hidden by the S22 and S21 curve, respectively. Double click on a label in the legend on the right hand side to see one particular curve emphasized. Double clicking somewhere in an empty area restores the original view. The customization of plots is introduced in another section.

Initially, a result view shows the magnitude of the tree items in the result folder. In fact, our model represents a band pass filter. A better idea of its performance is given by the *Magnitude in dB* representation. Switch to *Magnitude in dB* by choosing *1D Plot: Plot Type ⇨ dB* 🔲, obtaining the following plot:
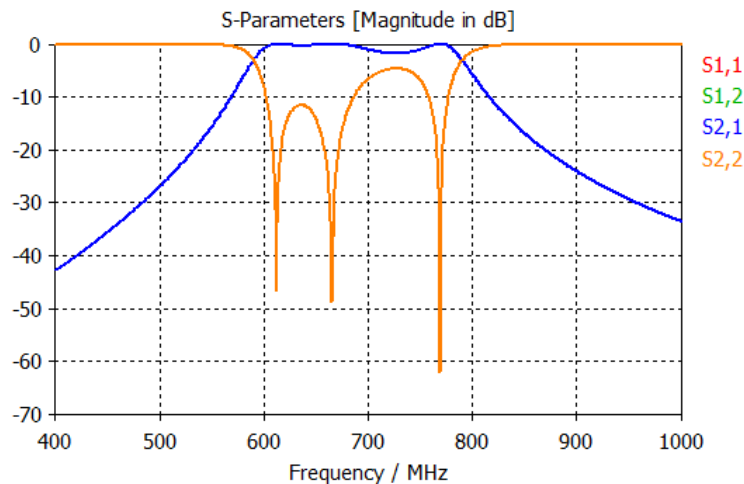
Initially, the *Zoom* mode (*View: Mouse Control ⇨ Zoom* 🔍) is active: You can define a zoom rectangle by clicking inside the view, keeping the mouse button pressed, moving the cursor to a different location and releasing the button. Immediately after releasing the button, a more detailed view will be displayed. There are additional modes that can be activated via *View: Mouse Control*. To navigate inside a zoomed view, activate the *Pan* mode (*View: Mouse Control ⇨ Pan* ✛). It allows moving the view in vertical and horizontal directions.

In addition to the modes, there are some viewing tools available such as axis markers, measure lines, and curve markers. They are activated via *1D Plot: Markers*. These tools are for performing measurements inside a plot view. The following information can be obtained using these tools:

- ☐ The axis marker (*1D Plot: Markers ⇨ Axis marker* ⬓) is a vertical line that is initially located in the middle of the x-axis. Its current x-value and the y-values of the intersections of the axis marker and the curves are displayed. Thus you can retrieve the position of a pole, for instance.

- ☐ The measure lin*es (1D Plot: Markers ⇨ Measure Lines* ⬓) are two pairs of lines, one pair in parallel to the x-axis and one pair parallel to the y-axis. The difference between the values of each pair is displayed as well as the measure lines' current x-values or y-values, respectively. Thus you can retrieve the minimum and the maximum value of a curve and the distance between them, for instance.

To demonstrate how measure lines can be utilized, let us assume that we would like to have a filter characteristic for our band pass filter as follows:

Description	Frequency range	Condition
Stop band	400 < f / MHz < 550	|S11| = maximal
Transition region	550 < f / MHz < 610	-
Pass band	610 < f / MHz < 790	|S11| = minimal
Transition region	790 < f / MHz < 850	-
Stop band	850 < f / MHz < 1000	|S11| = maximal

The frequency range of the pass band can be represented within the plot very easily, using the measure lines . Choose *1D Plot: Markers ⇨ Measure Lines* ⬓ to switch them on. Move one of the vertical measure lines to 610 MHz by clicking on it and dragging it to the desired position while keeping the mouse button pressed. Alternatively, when double-clicking on the vertical measure line, you can directly enter the desired value 610.
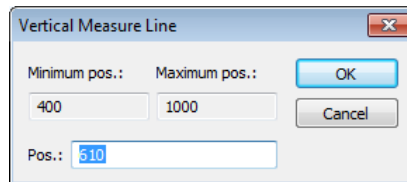
The current position of the axis marker will be plotted below the frequency axis. In the same manner, move the other vertical measure line to 790 MHz to have a visualization of the pass band of our filter.

Now we can check the performance of our current filter design within the pass band. Move one of the horizontal measure lines to the maximum |S11| value in the range of the pass band. The maximum value is plotted left of the measure line.
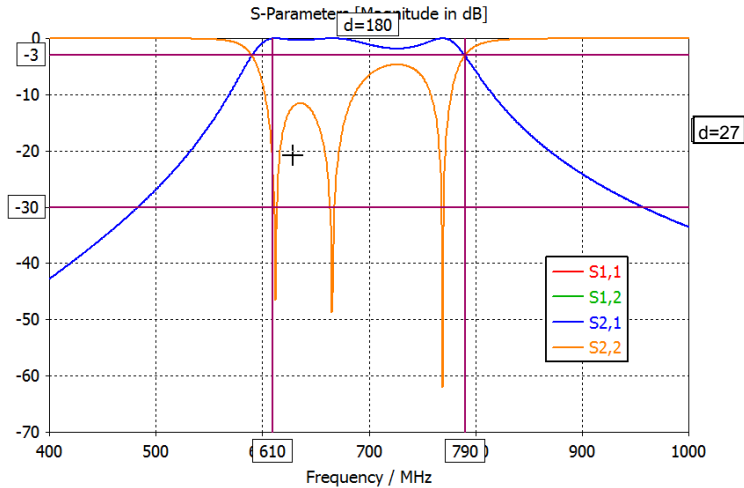
As you can see, the performance already looks reasonable but the curves suggest that an overall minimum of S11 within the pass band has not yet been reached.

In a subsequent section we will demonstrate how to optimize our filter design. We will introduce parameters, study their influence by performing a parameter sweep and finally optimize the filter using the built-in optimizer tool.

But first, let us return to the visualization subject. The following sections will teach you how to modify the plot's properties. Furthermore, we will show you how to create user-defined result views and add data there.

Customizing Result View Properties

In addition to the buttons used to switch between the visualization types and interaction modes, there are more options to manipulate the plot:

☐ Several plot options can be set in the *1D Plot: Plot Properties ⇨ Properties* dialog box.

☐ Individual plots can be shown or hidden by using *1D Plot: Plot Properties ⇨ Select Curves.*

☐ Selecting *1D Plot: Windows ⇨ New Plot Window* 🖼 opens another plot view, initially displaying the same contents as the current one, but in *Magnitude* representation. To switch between the plots you may click on the corresponding tab at the bottom of the main window. Alternatively, you can use the context menu of the corresponding tree item: If it refers to more than one plot, it contains the entry *Next* to activate the next view. Note that the plots are organized in cyclic order: After you have activated the last one, choosing *Next* will activate the first view.

Let us now examine the Plot Properties dialog box. Choose *1D Plot: Properties ⇨ Plot Properties ⇨ Properties* 🖺 or use the plot's context menu item *Plot Properties*:
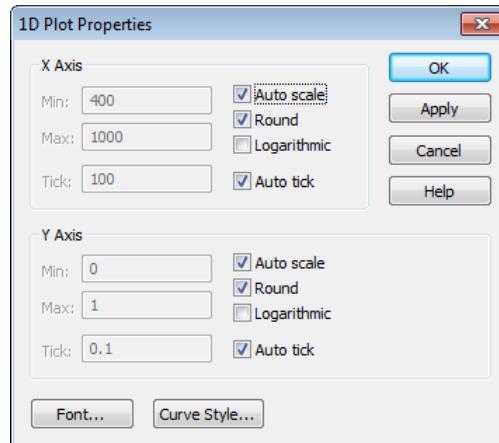
□ Within the *X Axis* frame you can customize the range and appearance of the x-axis and the positions of the horizontal ticks:

▪ If *Auto scale* is chosen, the plot's minimum and maximum abscissa values are automatically calculated. To specify your own values, switch off this option and edit the *Min* and *Max* fields.

▪ The *Round* option expands the plot range to the next rounded minimum and maximum abscissa values. This option is only available if *Auto scale* is set.

▪ Ticks subdivide an axis into intervals of identical size. Switch on *Auto tick* for an automatic calculation of an interval's width. To specify your own tick width, switch off this option and specify *Tick*.

▪ Choose the *Logarithmic* option to establish a logarithmic axis. A logarithmic axis does not allow customized ticks. Furthermore, you have to ensure that all axis values are positive.

□ Within the *Y Axis* frame the settings described for the x-axis can be applied to the y-axis. In phase plots, there is a further option, *Wrap phase*, that limits the display of a phase to -180° < arg < 180°.

□ The *Font…* button leads to a dialog box where you can specify the font for the title, axis labels, etc.

□ The *Curve style…* button leads to a dialog box where you can manipulate the appearance of your curves within the plot.

To exclude some curves from the current plot view, choose *1D Plot: Properties ⇨ Plot Properties ⇨ Select Curves* or choose *Select Curves* from the view's context menu.
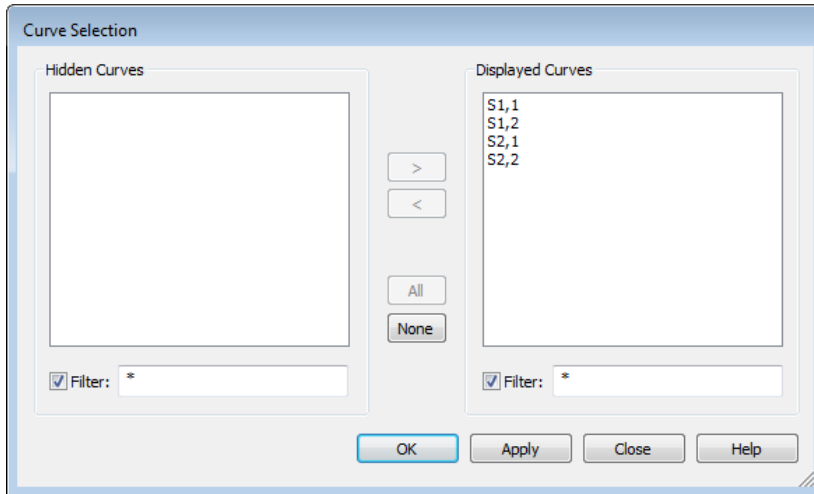
The *Select Curves* dialog box will open that consists of two list boxes: The box labeled *Hidden Curves* shows a list of the curves that are currently not displayed and the box *Displayed Curves* contains the curves that are currently displayed. Use the buttons > and < to move entries from one list box to the other or press *All* or *None* to move all entries to one of the boxes. Pressing *OK* or *Apply* applies the selection to the plot.

If you want to exclude only distinct curves from the plot there is an even faster way to do this. Select the result item in the result tree and select *Hide* from its context menu.

User-Defined Result Views

In addition to the standard result views that are automatically generated, a user-defined result plot can also be created:

Select the *Results* item inside the navigation tree and choose *Add Result Plot* from the item's context menu. A tree item is added to the folder with an editable label. This tree item represents a new result folder. You may enter a name for it or accept the default name *Result*.

Click on this new tree item to activate the folder's view. Since the folder is initially empty, the view only displays the message "Result not available". To add data to the new result folder, choose *Manage results…* from the item's context menu to open the *Manage results* dialog box.

In this dialog box all existing results are listed. To add the reflection factor S1,1 choose the *S1,1* entry in the *Result items* list, as shown below.
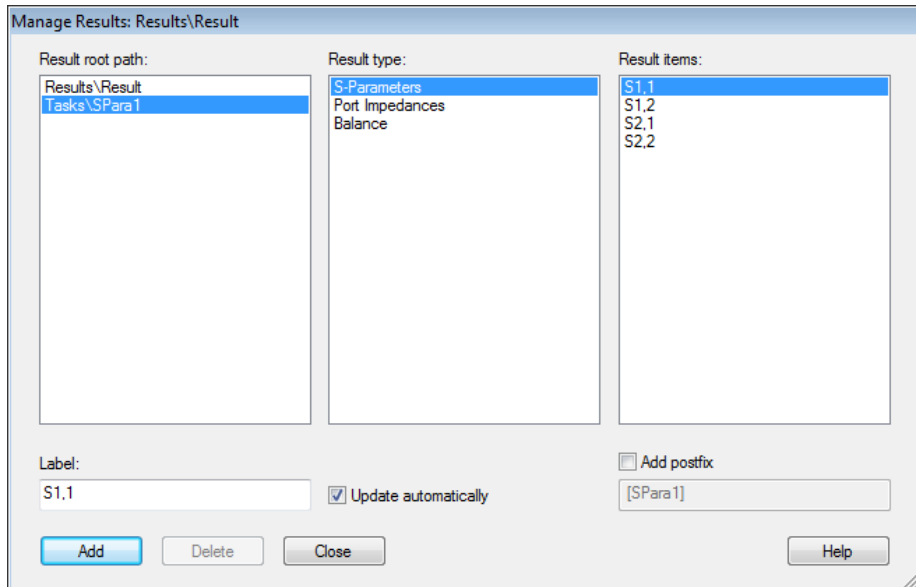
By default the result name will be used as a default value for the plot label. For our example, please change the label to *Initial S1,1*.

In the dialog box you will also find an *Update automatically* option. If this setting is switched on, a result reference is created that is updated whenever a simulation has been started. Please uncheck this setting now, since we would like to preserve the results of this simulation run.

Press the *Add* button and the selected curve will be added to the result folder.
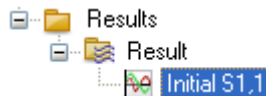
If the *Update automatically* option had been switched on, the result icon would have shown a small link symbol at its lower left corner.

Adding a result item into a user defined result folder can also be done via drag'n'drop. Select S1,2 from the result tree and drag it onto the *Result* folder while pressing the *Ctrl* key:
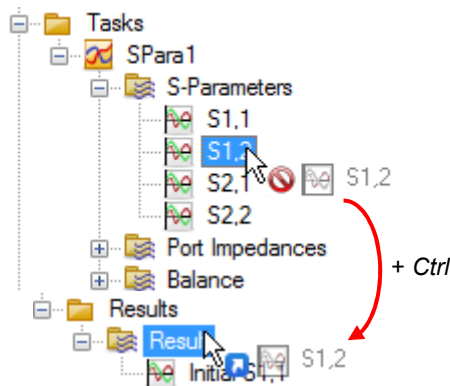
If you now release the mouse button, a copy of S1,2 is created in the *Result* folder. A reference would have been created, if you had not pressed the *Ctrl* button when releasing the mouse button. Please rename the *S1,2* curve to *Initial S1,2* by selecting the *Rename* option via the context menu.

If you now select the *Result* folder, you will get the following plot:
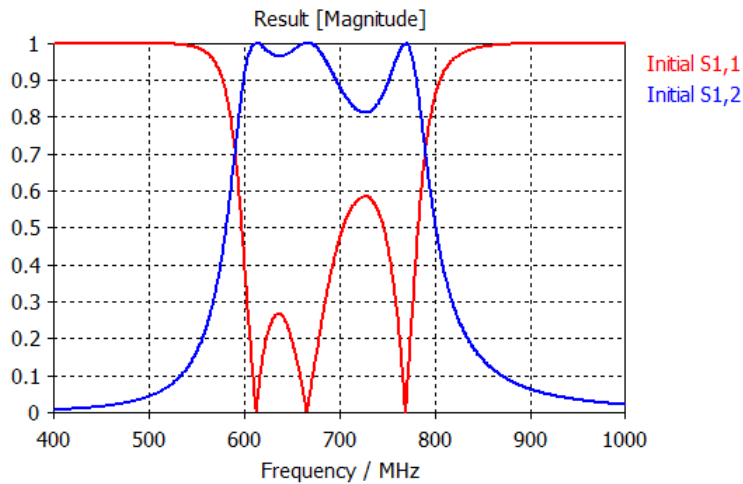
Parameterization and Optimization

The parameterization of a design enables you to easily consider variations. If properties are associated with parameters, the properties can be changed and therefore influence the design's behavior. In the following section, the use of parameters will be shown and a parameter sweep will be performed. However, a common application of parameters is their optimization with respect to goal functions that will also be explained in this section.

Using Parameters

CST DESIGN STUDIO offers the possibility to deal with global variables that may serve as parameters for global settings or block properties. Working with parameters is straightforward: First, you need to define them inside the parameter list control; you may then assign them to a property (including mathematical expressions containing parameters).

The parameter list control displays a table consisting of five columns labeled *Name*, *Value*, *Evaluated*, and *Description*, and *Type*. The table itself initially shows one single empty line (except for the *Name* column displaying *<new variable>*) as shown below. If you define a new parameter, it does not matter which column you edit first. The definition of a parameter is not completed until a name has been specified. However, we recommend that you start with the first column.

Parameter List				✕
▽ Name ▲	Value	Evaluated	Description	Type
<new variable>				
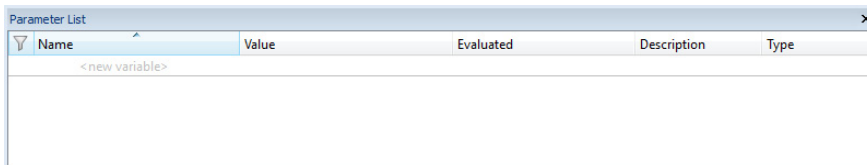
Valid names are all strings that are valid variable names in VBA. In particular, they must not be interpreted as a VBA command and must not contain special characters such as spaces, etc. Valid values are all expressions consisting of mathematical VBA functions, real numbers and previously defined parameters. The specification of a description is optional. Within the *Type* column you should specify the type of unit that a parameter is associated with. This information is only applied if the current design is used as a sub-model or if parameterized contents are copied to another model to properly scale the concerned properties. Moreover, the parameter list control provides the following features:

☐ A subset of parameters can be displayed by clicking on the filter symbol (▽) left of *Name* and typing the substring to be matched.

☐ A parameter can be removed from the list and will be deleted from your project, as well. Therefore, select the parameter row by clicking on any field in this row and choose *Delete* from the context menu. If the selected parameter is used somewhere in your project the following message box will appear:
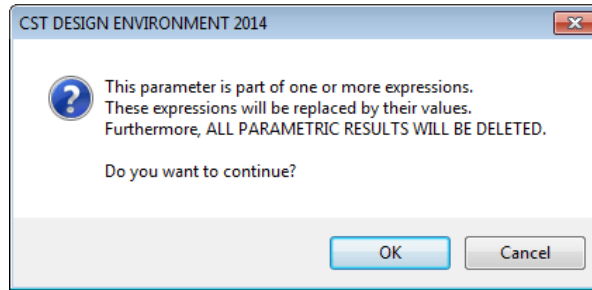
Pressing *OK* would delete all parametric results and it would replace the selected parameter by its value everywhere it is in use.

☐ The name of a parameter inside the list is editable. Note that renaming a parameter means that it will be deleted and a new one will be defined. Therefore, renaming may also lead to the same message described above.

☐ To check whether a parameter is in use (and whether you can rename or delete it without any consequences), select it and choose *Dependencies* from the parameter list control's context menu. If it is used for any properties, a dialog box is opened containing a list of those properties. Otherwise, a message will be displayed, indicating that the parameter is not used.

☐ A parameter can be replaced by its value by choosing *Replace parameter by value* from its context menu. The parameter will not be deleted afterwards.

☐ When performing simulations for different parameter values, results of every parameter set are stored. To identify these result sets, each result of a specific parameter set is associated to a unique integer run ID. This ID can be used to reset all parameter values connected to this specific result set. Choose *Set Parameters to Run ID* from the parameter list control's context menu to use this functionality. You will learn more about parametric result handling further below.

For our example, we are going to introduce four parameters: C1, C2 (representing the capacitances of the capacitors) and L1, L2 (representing the inductances of the inductors). To start to define C1, click the left mouse button on the top left cell inside the parameter window that will then become editable. Enter the name *C1* and press the *Tab* key to change to the *Value* column or click the left mouse button over that cell. We recommend using the *Tab* key as it allows you to add your entries very quickly. The value 0 has been automatically assigned to the new parameter.

The value can now be edited (after selecting it by mouse click or using the *Tab* key). Enter *35* there and press *Tab* again to change to the *Description* column where you can optionally give a short description of the parameter. If you press *Tab* again, a new row will be added to the table and its *Name* cell will be selected and editable. Continue as described for all parameters.
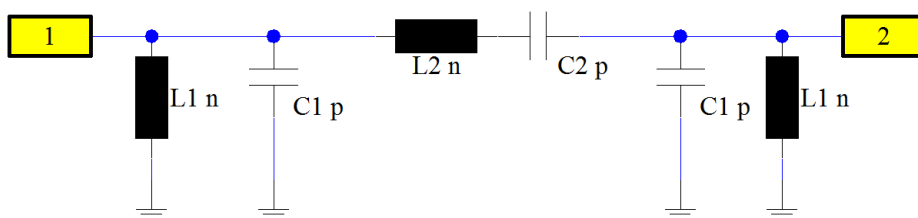
Finally, do not forget to select the correct type for each parameter. In our case, this would be either *Capacitance* or *Inductance*. After everything is set, the parameter list should look as follows:

Now, select the leftmost inductor inside the schematic view. With the selection the parameter window will change and show the local parameters of the selected block. Replace the current value for the inductance by the previously defined parameter *L1*.


Repeat this procedure for the remaining elements until your drawing looks like the figure below:

Note that although the blocks parameters have changed from numbers to parameter names the results are still valid. The values of their associated results are still the same.

To check our model once more and become more familiar with the parameter list control, single click into the schematic view to activate the global parameter list and click the right mouse button over the row belonging to the parameter C1. Then, choose *Dependencies* from the activated context menu. The following *Dependencies* dialog box appears and displays the blocks and properties that depend on the selected parameter.
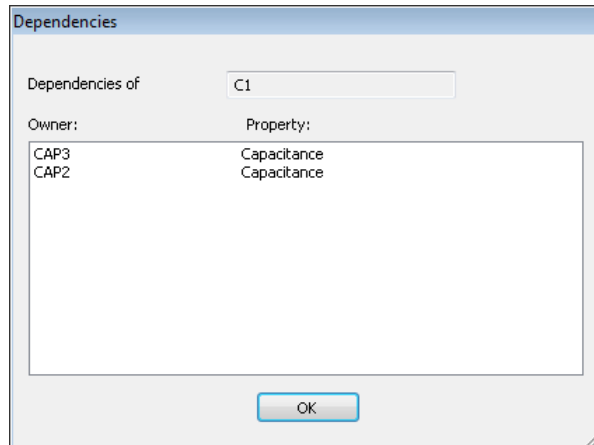
After closing this dialog box by pressing the OK button, press the *Ctrl+F5* key to update the results. All standard result views are automatically generated that show the same contents as before.

Let us now modify the parameter C2. Click the left mouse button on its *Value* cell and enter *1.4* there.

Please note that the plot view in the main view window shows a grey background now to indicate that the actual result is not valid anymore because of the parametric change. However, the S-Parameters in the navigation tree are still valid, because the results for the previous parameter set are still available and still valid for this topology.

On a topological change, such as adding another circuit element to the schematic, the S-Parameters in the result tree would also have been invalidated. In the tree, outdated results would have been indicated by changed S-Parameter tree item icons (⬚):
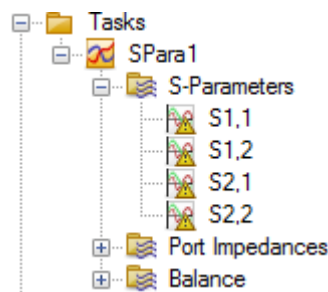
Let us now move back to the actual workflow. Activate the design's S-Parameter view now by clicking on the *S-Parameters* folder and change the plot to *Magnitude in dB*

representation by selecting the corresponding button in the *Plot Tools* toolbar. Then, update the results (*Ctrl+F5*). The results are now calculated for the current parameters. After the calculation is completed, the 'out of date' result icons in the result tree are changed to the normal result view items. Of course, the S-Parameter results have changed according to the parameter modification:
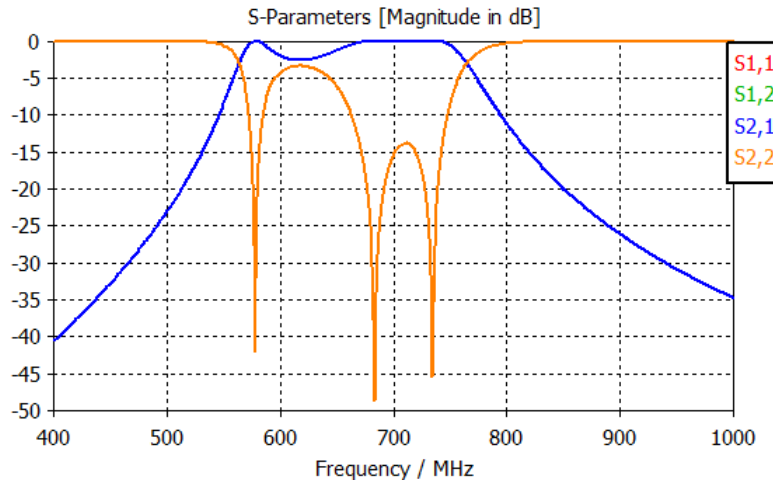

To continue, reset the parameter *C2* back to *1.2*, delete the results via *Postprocessing: Manage Results ⇨ Delete Results* ✖ and selecting *All results created by simulation tasks*. Then update the results again.

Performing a Parameter Sweep

Because you have successfully introduced parameters, it might be interesting to see how the results change when these parameters are modified. The easiest way to obtain these varying results is to perform a parameter sweep task.

To create a new parameter sweep task, choose *Home: Simulation ⇨ New Task ⇨ Parameter Sweep* ☑. As for the S-Parameter task, the new parameter sweep task will be listed in the navigation tree.
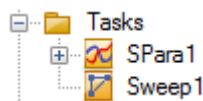

Like all other *Simulation control* tasks, a parameter sweep task cannot be executed on its own. It needs to be associated with another simulation task. In our case, we want the parameter sweep to execute an S-Parameter task for each parameter combination. This relation is represented in the tree by defining an S-Parameter task as a child entry of the parameter sweep task. This can be done by moving task *S-Para1* onto *Sweep 1* via drag'n'drop:

As a second step, we must define the results of interest that are to be recorded during the sweep. For user-defined results this needs to be done via a postprocessing task (*Home: Simulation ⇨ New Task ⇨ Postprocessing* 🖩) as child of the S-Parameter task. For built-in results such as the S-Parameters generated by the S-Parameter task there is no need to use postprocessing tasks, since they are stored in a parametric fashion anyway.

Please view the properties of the S-Parameter task *SPara1* by choosing *Properties* from its context menu. There, select the *Results Settings* tab. In this dialog you can specify which results are to be stored during simulation and in which format they are to be stored.
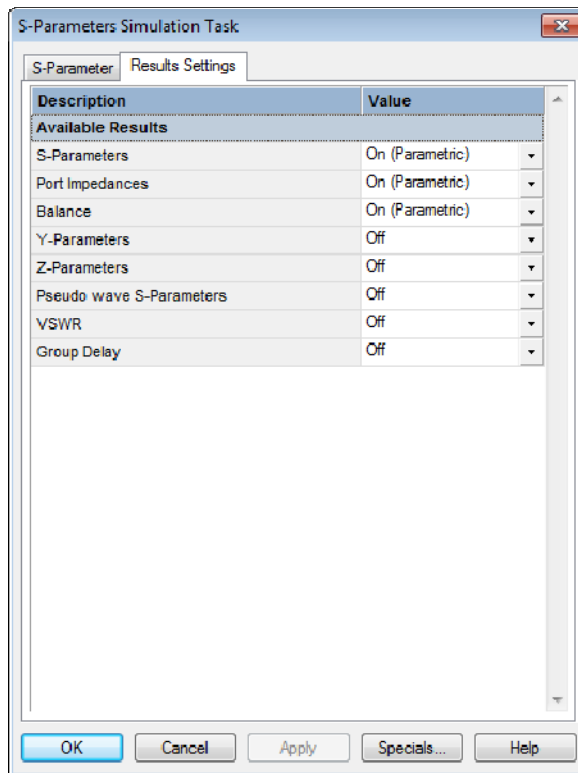
All results that are set to *On w*ill be stored as a function of global parameters, namely, as a function of the parameter(s) that we are going to define as sweep variable(s).
Leave the dialog by pressing the *OK* button.

Now let us return to the definition of the parameter sweep itself. Open the *Parameter Sweep* dialog box by double clicking on the tree entry *Sweep1*.
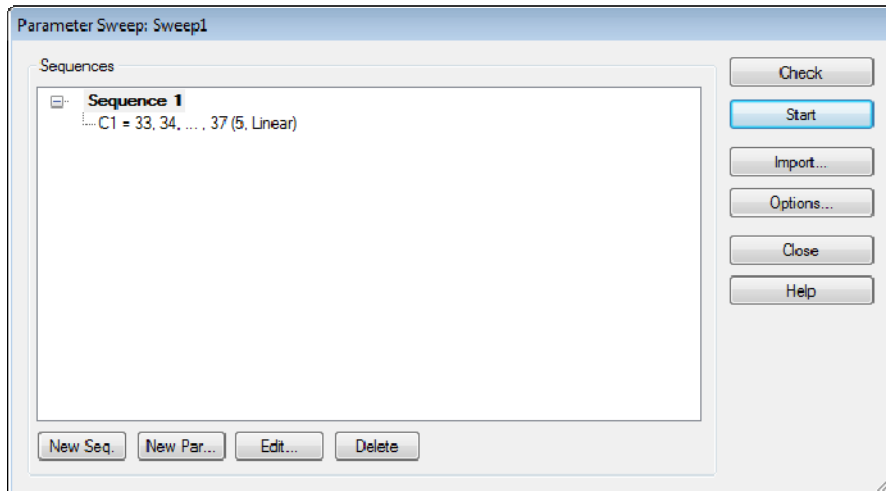
Within the *Sequences* frame you can specify the parameters to sweep, the number of steps to perform, etc. It contains a list that displays the defined sequences and the parameter ranges assigned to the sweep. Furthermore, there are buttons to add and delete a sequence or a parameter, respectively.

Let us now perform a parameter sweep step-by-step. First, we create a new sweep sequence by pressing *New Seq*. A sequence *Sequence1* is added to the list of sequences that is selected right after its creation, which causes the buttons *New Par*, *Edit* and *Delete* to become active.

Pressing the *Edit* button makes the name of the sequence editable. Clicking on the *Delete* button simply deletes the sequence.

Clicking on *New Par* opens the *Parameter Sweep Parameter* dialog box where we can select a parameter and specify the range of values assigned to it during the sweep. Choose the parameter *C1* as the first parameter to alter during the sweep.
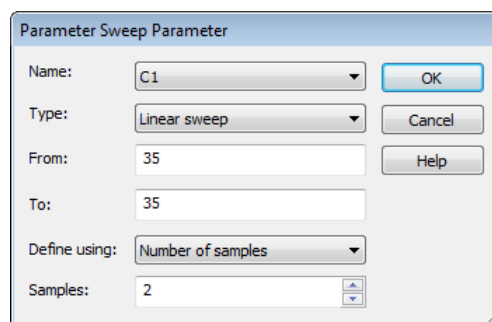
There are various sweep types such as logarithmic sweep or a sweep with arbitrary sweep points. We stay with the preselected linear sweep. Let us specify an appropriate parameter range for the sweep. Enter *33* as the lower limit (*From*), *37* as the upper limit (*To*) and assign *5* to the *Samples* field as shown below.
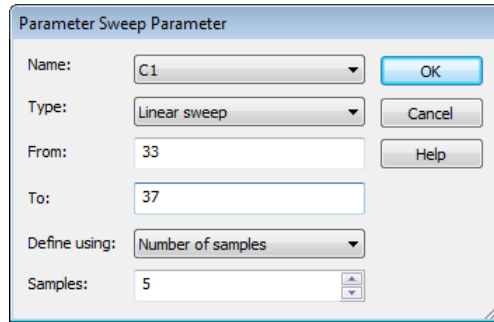
Press *OK* to add the parameter variation to the sequence. The parameter variation is added to the sequence and is displayed in the *Sequences* frame:
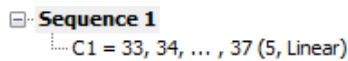

CST DESIGN STUDIO is able to perform parameter sweeps with an arbitrary number of parameters. For demonstration purposes it is sufficient to consider only one parameter. If you define more than one parameter variation and assign it to a sequence, calculations for all combinations of the possible parameter values are performed during the parameter sweep.

After successfully defining parameter variations the *Start* and *Check* buttons of the *Parameter Sweep* dialog box become active.

During a parameter sweep the values that will be assigned to the parameters might exceed the range of valid values for some of the associated properties. For example, the transmission lines' characteristic impedances must be greater than zero; a value less than zero would lead to an error. Clicking on *Check* performs a sequential update of the databases. If an update fails, a message will be displayed such that you can modify your settings.

Please perform a check (which finishes successfully for our settings) and start the sweep afterwards by pressing the *Start* button. After the parameter sweep is complete, an info message will appear.
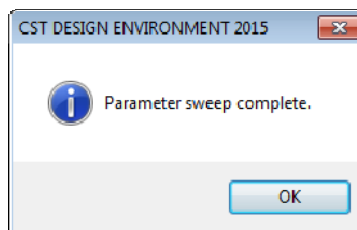

Confirm by pressing *OK* and *Close* the parameter sweep dialog box.

To view the results for *S1,1* Select *NT: Tasks ⇨ Sweep1 ⇨ S-Para1 ⇨ S-Parameters ⇨ S1,1*. The folder's result view is opened, showing the results of the last calculated parameter combination. To see the results as a function of the swept parameter *C1,* choose *1D Plot: Plot Mode ⇨ Parametric Plot* 🖾

Two new views open. One plot view, showing the S-Parameter S1,1 as a function of the swept parameter C1 and a table view, showing the associated *Parameter View*.

Plot view:
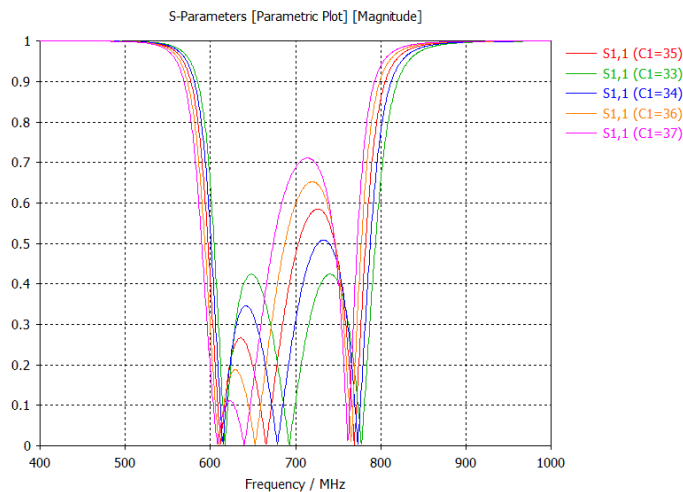
Parameter View:

Schematic Run ID		Parameters	Tasks\Sweep1\SPara1\S-Parameters
	Status	C1	S1,1
1	Calculated	33	1DC
2	Calculated	34	1DC
3	Calculated	35	1DC
4	Calculated	36	1DC
5	Calculated	37	1DC

In the plot view, you can see the parameter value of the swept parameter *C1* as label text. When sweeping two or more parameters, only the unique *Run ID* of each simulation run is shown to keep the labels short.

The mapping between Run ID and its associated parameter values is presented in the *Parameter View* window.

In fact, the results have been stored not only depending to parameter C1 but also depending to the other defined global variables (C2, L1, L2). The default parametric view realizes that these parameters are constant and therefore hides them from the view. To get a complete view of all considered variables activate the parametric plot view and deselect *Visibility and Sorting: Hide Constant Parameters*:

Schematic Run ID		Parameters				Tasks\Sweep1\SPara1\S-Parameters
	Status	C1	C2	L1	L2	S1,1
1	Calculated	33	1.2	1.6	44	1DC
2	Calculated	34	1.2	1.6	44	1DC
3	Calculated	35	1.2	1.6	44	1DC
4	Calculated	36	1.2	1.6	44	1DC
5	Calculated	37	1.2	1.6	44	1DC

Close the Parameter view via *Parameter View: Close ➪ Close Parameter View* ✖.

Performing an Optimization

CST DESIGN STUDIO offers a very powerful built-in optimization feature that is able to consider an arbitrary number of parameters and a combination of differently weighted goal functions.

In the following, we will optimize the characteristics of our band pass filter. The optimization goals for our filter will be as follows:
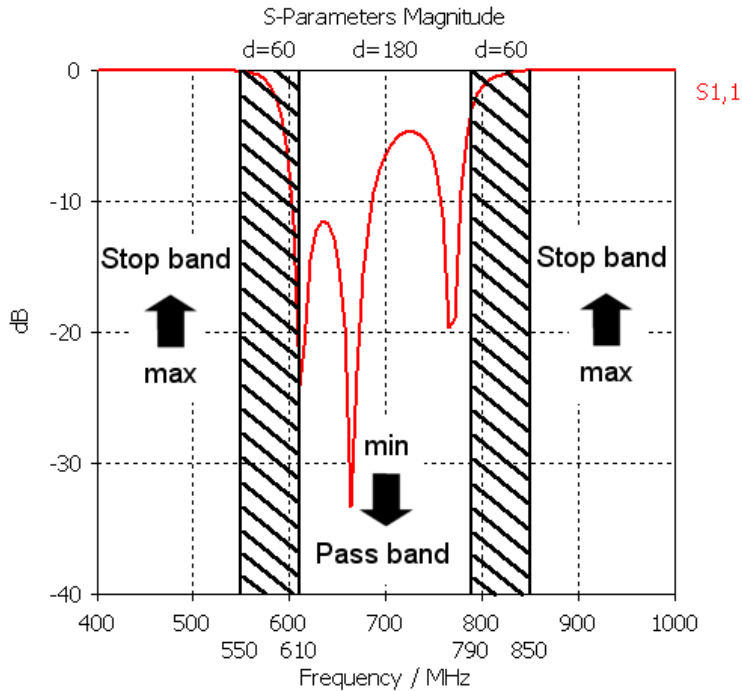
To achieve this goal, we need to define a new optimizer task. Choose *Home: Simulation ⇨ New Task ⇨ Optimization* 📊. By default, it is named *Opt1*.

Very similar to the parameter sweep task, we need to connect an S-Parameter task to the optimizer task. The easiest way to do this is to duplicate the *S-Para1* task and move it into the optimizer task. You will find the *Duplicate* operation in the task's context menu. After the duplication and drag'n'dropping the duplicated task into the optimization task, the task structure in the navigation tree should look as follows:
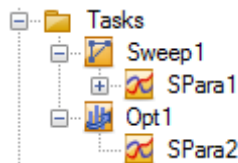
Now open the optimizer dialog box by double clicking on the corresponding tree item *Opt1*. The initially displayed *Settings* page allows specifying the optimization algorithm and shows a list of all previously defined parameters that can be used for the optimization.

For our optimization we will choose the *Trust Region Framework*. This algorithm uses a *Domain accuracy* value that controls its convergence behavior. Please refer to the online help for a more detailed explanation. In our example we stay with the defaults.

The parameter list shows all available parameters and their optimization settings, namely the parameters' minimum and maximum values, the initial value, the current value, and the best value achieved in the previous optimization. To the left hand side of the parameters a column of check boxes is displayed, where you can select the parameters to use for the optimization.

Select C1, C2 and L1 to include them in the optimization process.
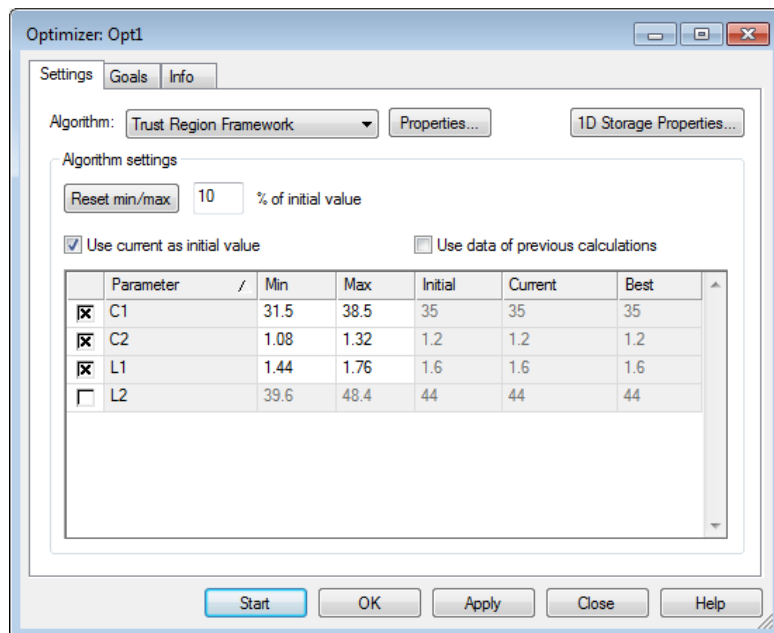
For the selected parameters you can specify the minimum and maximum values and, if the *Use current as initial values* option at the top of the list is switched off, the initial value. By default, this option is switched on to initialize each parameter by its current value. For our example please disable this option.

The optimizer attempts to optimize a goal function by evaluating the goal function for different parameter sets. Since the evaluation of the goal function might be very expensive, the aim of every optimizer is to find the function's optimum with as few evaluations as possible. Therefore, different algorithms are available that are suitable for different types of problems. Each algorithm uses its own strategies to reduce the number of simulation runs. Some use an interpolation technique, others like the *Trust Region Framework* use cached data from already simulated results when needed.

The 10% parameter range in the edit field associated with the *Reset min/max* button on top of the parameter list restricts the variability of the parameters. Clicking on the *Reset min/max* button automatically adjusts the range of the selected parameters: The entries in the *Min* column are set to the initial values minus the specified percentage; the entries in the *Max* column are set to the initial values plus the specified percentage.
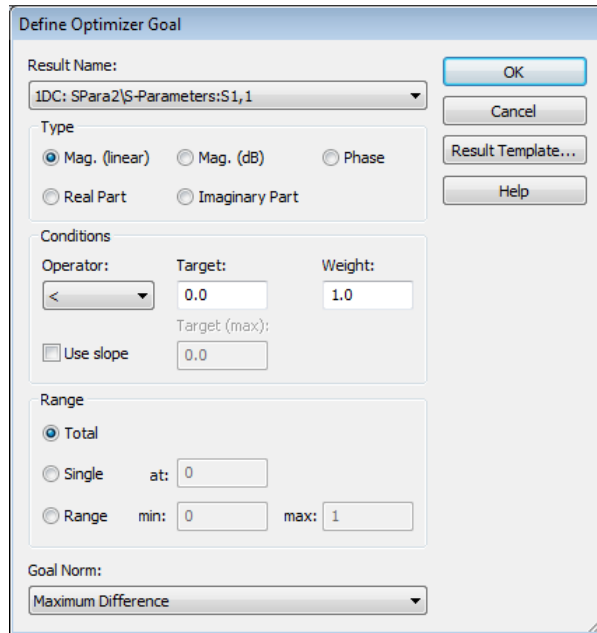
Enter a value of 30% and press the *Reset min/max* button to update the Min/Max values in the table.

Now, we need to specify the goal function for our optimization. Change to the *Goals* page by clicking on the corresponding label at the top of the dialog box.
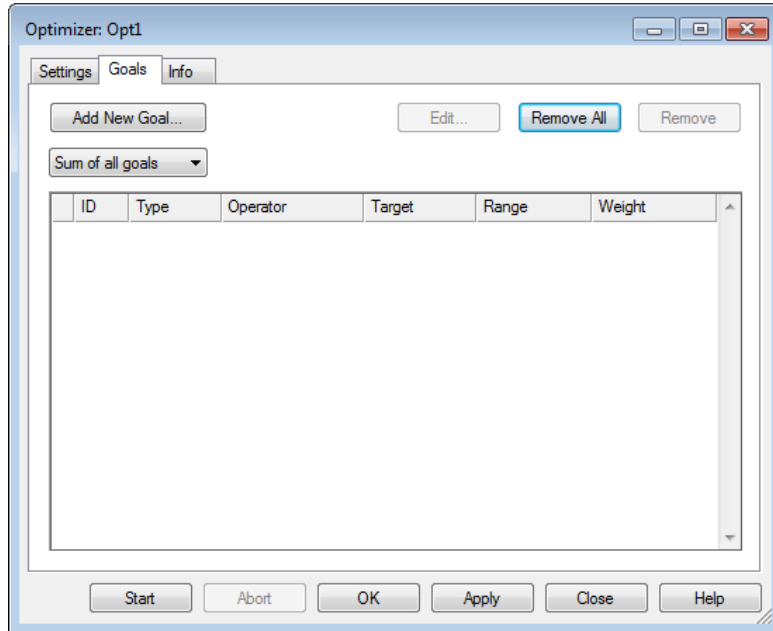
The *Goals* page allows manipulating goals used for the optimization process. Every goal is based on built-in results of the underlying tasks and/or on the results of postprocessing templates and some general rules how to calculate a goal function value from the template's result. In our application, we optimize S-Parameters for which we can use the build-in results without needing to define any postprocessing tasks.

To add a new goal, click on the *Add New Goal…* dialog button. From a drop-down list you can select a specific result that is to be used for the goal definition. We keep the default result name since we want to define optimization goals on S1,1.

The prefix *1DC* in the result name indicates that the task's S-Parameters are of the complex 1D Result type. The appearing dialog box allows us to complete the goal definition.

- ☐ In the *Type* frame the type of result which the goal definition should act on can be specified. We want to define goals on the magnitude of S1,1 and leave this frame untouched.

- ☐ In the *Conditions* frame you may assign an objective for the selected data. We will describe the proper definition of such a target in detail later on. Furthermore, you can specify a weight for the goal.

- ☐ In the *Range* frame you can specify the abscissa interval (range) within which the condition should be satisfied. There are three options: You may optimize either at a single value (select *Single*), at a given abscissa range (select *Range*) or for the total abscissa range (select *Total*). The default option is *Total*. In many cases the abscissa values will be frequency values.

Let us take a closer look at the definition of a condition by considering some examples.

- ☐ Specifying constraints:
  - ▪ If you want the selected data not to exceed a certain value, perform the following settings: Choose the operator '<' from the *Operator* list and specify the maximum value of the chosen result inside the *Target* edit field.
  - ▪ For the specification of a lower data limit just select the operator '>' and perform the same steps.
  - ▪ Use the operator '=' to optimize the parameters such that the template result equals the target value.

- ☐ Moving the minimum or maximum value to a specific abscissa value: To move the minimum value of the goal function to a specific abscissa value,

select the operator *move min* and specify the abscissa value where you want the minimum to be moved to in the *Target* field. Because a value will be moved along the abscissa, the *Single* setting in the Range frame will be disabled for this type of goal. You can analogously move the maximum value to a certain location using the operator *move max*.

☐ If there is more than one goal, you may adjust their influence on the optimization by specifying a weight. It can be any positive number.

For our example, we need to define three goals specifying constraints for certain ranges.

Description	Frequency range	Condition
Stop band	400 < f / MHz < 550	|S11| maximal
Transition region	550 < f / MHz < 610	-
Pass band	610 < f / MHz < 790	|S11| minimal
Transition region	790 < f / MHz < 850	-
Stop band	850 < f / MHz < 1000	|S11| maximal

The first goal definition should look like this:

After all three goals have been defined the *Goals* page should look like this:
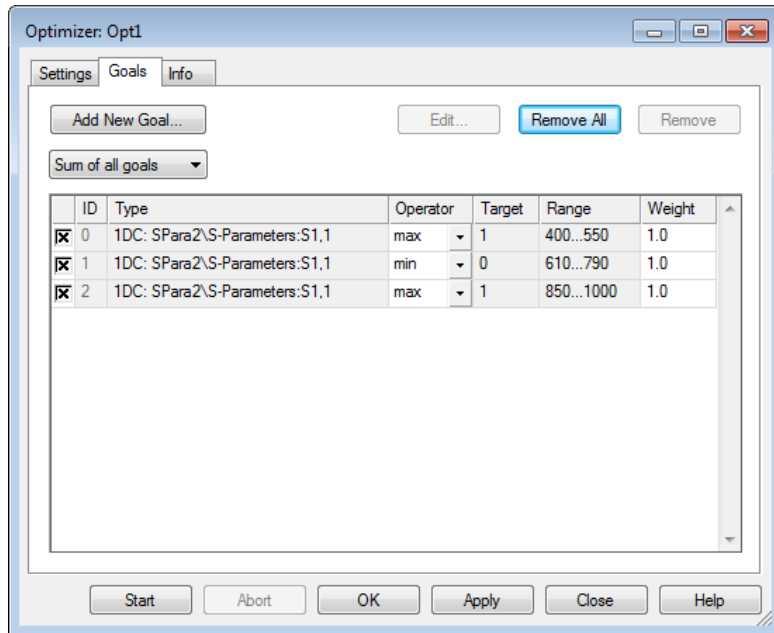
The check boxes in the first column of the table can be used to enable/disable the defined goals. In our case all goals need to be enabled.

Now press the *Start* button. As soon as the optimizer is started, the solver run *Info* page will be available, displaying some information about the optimization process. To visualize the optimization process, click on the *Opt1\S-Para2\S-Parameters* tree item.

When the calculations are performed, the *Info* page reports the progress of the optimization. Among the values displayed are the number of evaluations (some of them are based on a calculation, while others are reloaded), the first, the last and the best goal function values and the best parameter values thus far.

After successful completion of the optimization the *Info* page displays the results as shown below:
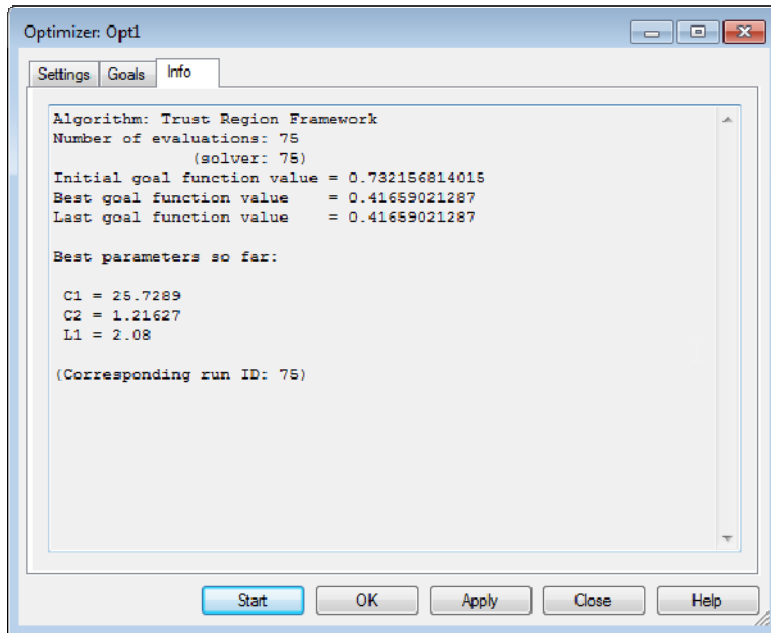
During the optimization, the goal function value decreased by a considerable amount, indicating that the optimizer was able to improve the filter's characteristic.

Now that we have an optimized result, it would be interesting to see a comparison between the optimized and the initial S11. Select the *Tasks\Opt1\Result Curves_SPara2_S-Parameters_S1,1* tree item to visualize these curves.
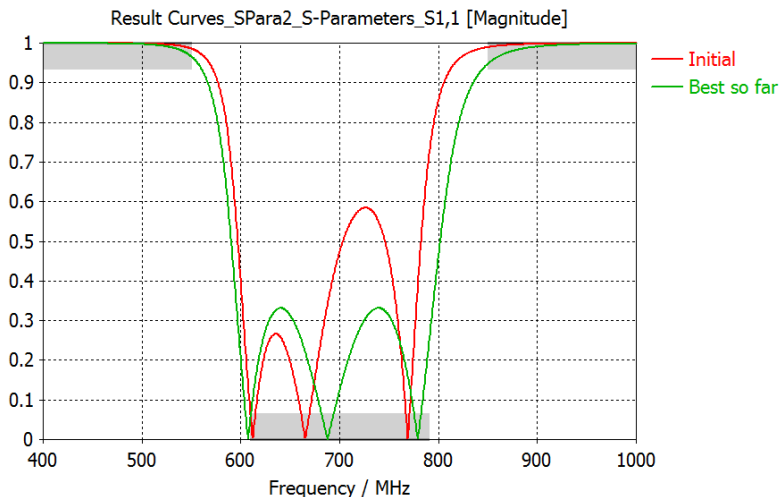
Inside this plot the goal definitions are also shown.

To get a plot in dB please change the plot scaling to *Magnitude in dB* and slightly change the y-Axis range: Open the *1D Plot Properties* dialog box via the context menu of the schematic view, disable the *Auto scale* for the y-Axis and set its min value to -14.

Result Curves_SPara2_S-P...rs_S1,1 [Magnitude in dB]
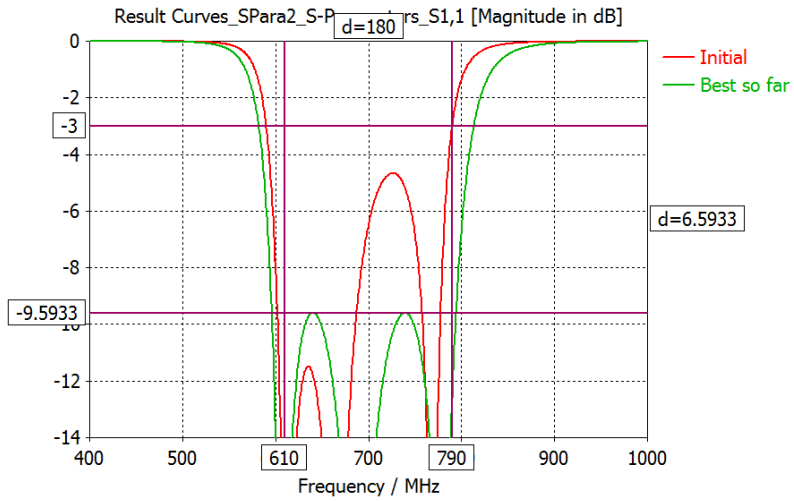
You can verify that the insertion loss throughout the pass band could be improved, indeed, by the optimization.

Chapter 3 — System Assembly and Modeling

This chapter introduces the system simulation capabilities of CST DESIGN STUDIO.

The first section describes how planar circuits can be simulated with different accuracy levels. It starts with a schematic representation where each block is described by an analytical model. This is followed by a full 3D field simulation that is performed on a 3D model derived from the original schematic.

The second section describes how a complete 3D model can be derived from schematic blocks. The layout view is used to align the components to each other. Simulation projects are used to create partial results and combine them into the final results for the whole structure.

The third section describes how variations of a master project may be simulated and compared. For this purpose, simulation projects are derived from the master project. Results of the variants are compared by using a postprocessing task in the master project.

Planar Circuits

When working with distributed elements such as microstrip and stripline waveguides, CST DESIGN STUDIO combines the advantages of having a pure schematic and a full 3D representation of such elements. To show this ability we will use the already prepared *S-Parameter Lowpass Geometry Only.cst* project in the *Examples\DS\S-Parameter\Circuit-EM\Lowpass Filter\S-Parameter Lowpass_data* folder of the installation directory. You can also find an entirely set up and simulated version of this example in the *Examples\DS\S-Parameter\Circuit-EM\Lowpass Filter* directory.
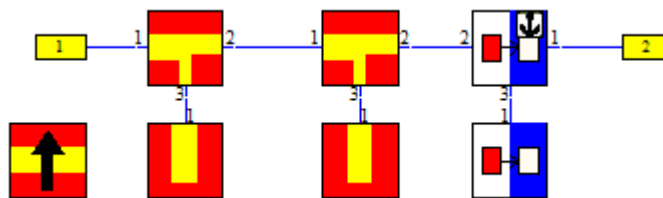
The circuit consists of three microstrip T-junctions and three microstrip open-ended stubs. Since two of the T-junctions and open-ended subs are identical, one of each is represented as a *clone block*. Clone blocks refer to any already defined block and behave identically to the referenced ones.

The blocks have the following properties:

Reference Block

Height	0.71
Thickness	0.035
Epsilon	3.5
Tan. Delta	0.006

MSOPEN 1, 2

Length	L1, L3
Width	W1, W3

MSTEE 1,2

Width1	1.56, W2
Width2	W2, W2
Width3	W1, W3
Length1	6, 0.0
Length2	L2, 0.0
Length3	0.0, 0.0

The parameters are defined as follows:

L1	6.5
L2	8.1
L3	8.6
W1	1.7
W2	2.0
W3	1.5

In many cases it is important to check if the resulting 3D structure, e.g. the layout of the circuit, is correct. It may happen that the resulting structure overlaps or that distances between elements are too small, such that unwanted coupling may occur. Choose *NT: Layout* or *Home: Edit ⇨ Layout* 🔲 or *NT: Layout*. A new view will open, showing the 3D-structure:
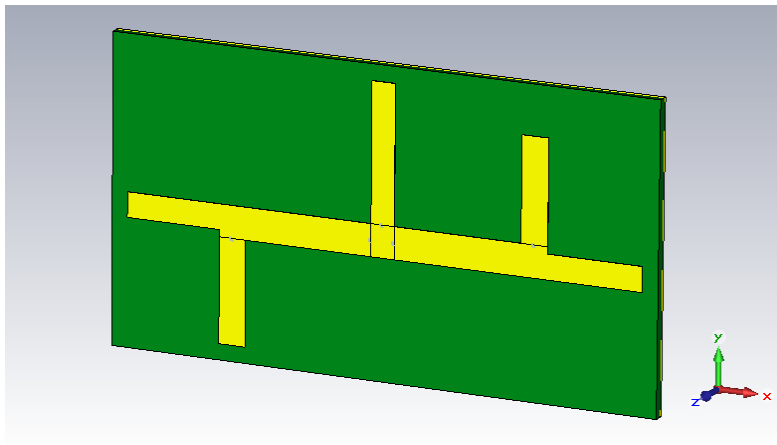
The layout view tries to automatically snap the components at their port location to get proper positioning of each block. However, as you can see, some orientations that are not relevant for the circuit simulation are not yet defined. They may need some manual adjustments. In our case we want to have all three fingers at one side. Select block *MSTEE1_CLONE1* which is the block that has the wrong orientation. Now you may select *Layout Flip* in the *Parameter List* of the block:
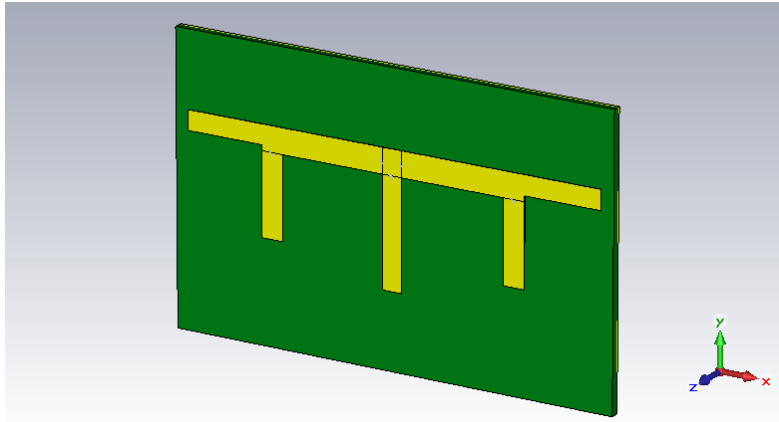
The layout view recognizes the change and asks for updating the view. After updating the desired result is obtained:
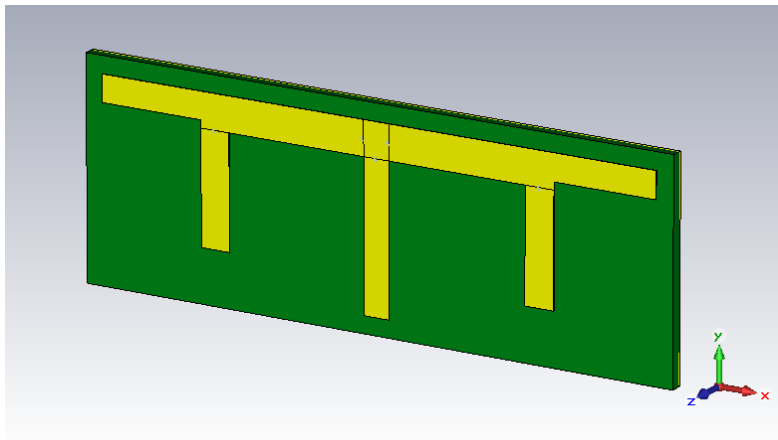

For the later 3D simulation it is useful to already enlarge the substrate in the transverse direction. Most field simulators need some space between the structure and the boundary of the simulation domain. Select the substrate in the layout view by double-clicking on it and change *Substrate Ymin* and *Substrate Ymax* to *6* in the *Parameter List*.

Again an update is requested. Pressing *F7* changes the view:

To simulate the S-Parameters, an S-Parameter task needs to be set up as described in previous sections. The frequency range should be Fmin = 0 and Fmax = 7 GHz. The results can be seen in the following graph:
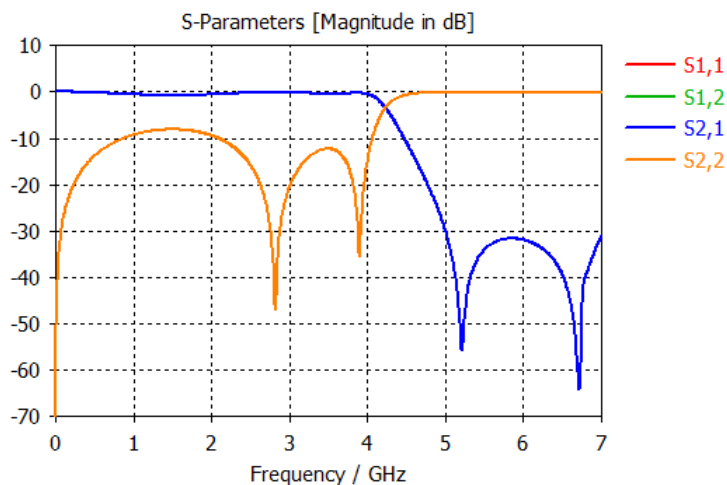
Of course, these are the results of the circuit simulation based on the analytical models of the used blocks. To obtain the results of the entire structure in 3D, some additional steps need to be done.

To set up a 3D model, a so called *Simulation project* must be created.

A simulation project is a complete .cst project that is automatically created by using the assembly information of the schematic. Additionally, all necessary solver settings and possibly some structure modification can be done there.

Note: Generally, the created 3D structure of a simulation project is linked to the blocks of the schematic such that parametric changes in the blocks are propagated to and reflected by the simulation project as well.

Go to the schematic view and choose *Home: Simulation ⇨ New Task* 🗹 and double click on *Simulation project*. Alternatively, just press the corresponding button in the

Ribbon (*Home: Simulation ⇨ Simulation Project ⇨ Select Block Representation* 🔳). The schematic view will be colored in a light yellow indicating that the *simulation project mode* is active and a *Simulation Project* tab will be shown in the Ribbon.

When in simulation project mode, the following buttons of the *Simulation Project* Ribbon tab are important:

3D Model	Schematic Model	Ignore in Simulation	Create Simulation Project	Close Simulation Project Mode

The buttons are grouped in different tabs as follows:

- ☐ *Simulation Project: Model Representation ⇨ 3D Model* 🔳: Considers the selected blocks as 3D
- ☐ *Simulation Project: Model Representation ⇨ Schematic Model* 🔳: Considers the selected blocks as schematic elements
- ☐ *Simulation Project: Model Representation ⇨ Ignore in Simulation* 🔳: Ignores the selected blocks
- ☐ *Simulation Project: Create Project ⇨ Create Simulation Project* 🔳: Ends the simulation project mode
- ☐ *Simulation Project: Close ⇨ Close Simulation Project Mode* 🔳: Exits the simulation project mode

Note: Only blocks that will be represented as 3D elements are linked to the master project. Blocks that are created as schematic elements are copied.

Since we want to add all blocks into a full 3D simulation press *Ctrl+A* to select all elements on the schematic and afterwards press *Simulation Project: Model Representation ⇨ 3D Model* 🔳. End the simulation project mode with *Simulation Project: Create Project ⇨ Create Simulation Project* 🔳.

We could also have used the short cut *Home: Simulation ⇨ Simulation Project ⇨ All Blocks as 3D models* 🔳 instead of individually selecting all blocks.

The appearing dialog box allows to specify some properties of the simulation project. Please choose *Full_3D* as *Name* and select *CST MICROWAVE STUDIO* as *Project type*. Since we want to simulate a planar microstrip filter structure we should define a template that sets several solver settings specific to this kind of structure. Select *Select template…* in the *Project template* drop down box. A new dialog box opens that allows either to select an already existing template or to create a new one. Let us quickly create an appropriate new template:

- ☐ Press *New Template…*
- ☐ Click on the *MV & RF & Optical* piece in the pie chart
- ☐ Double click on *Circuit & Components*
- ☐ Double click on *Planar Filters*
- ☐ Double click on *Frequency Domain (Fast Reduced Order)* (If the FD-Solver is not included in your license, you may choose any other listed solver)
- ☐ Leave all unit settings unchanged and click *Next*
- ☐ Ignore all additional settings in the page and directly press *Next*

☐ Specify *My Planar Filter* as template name and press *Finish* to leave this dialog box
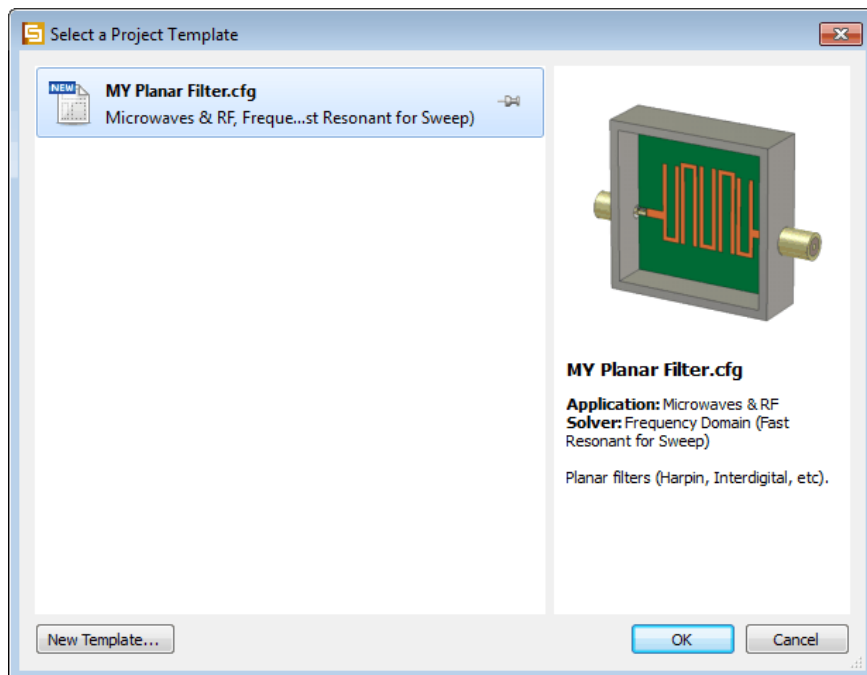
You will find more information on project templates in the *CST STUDIO SUITE Getting Started* document.

Press OK to return to the *Create New Simulation Project* dialog box.

Finalize the dialog settings by choosing the *Frequency Domain* solver and by selecting *Task: SPara1* as *Reference model for global settings*.
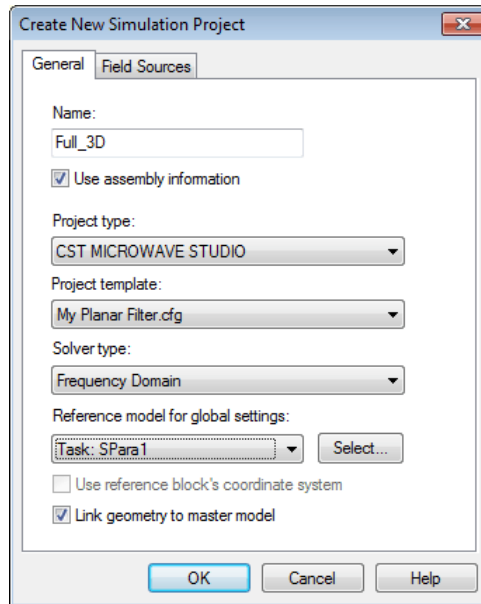
While the project template makes sure that the 3D simulation project gets the appropriate settings for background material and boundary conditions, the *Reference model* setting makes sure that the same frequency range as in the S-Parameter task is set. The frequency range of the reference task will overwrite any frequency range definition in the selected project template

Press *OK* to create the new simulation project.

Finally everything is in place to start the simulation. In the schematic view of the main project, select *Home: Simulation ⇨ Update* . Now the regular S-Parameter task as well as the simulation project is simulated. After the simulations have finished, all results can be found in the result tree. To view the simulation projects' results, select *NT: Tasks ⇨ Full_3D ⇨ 3D Model Results ⇨ 1D Results ⇨ S-Parameters*:
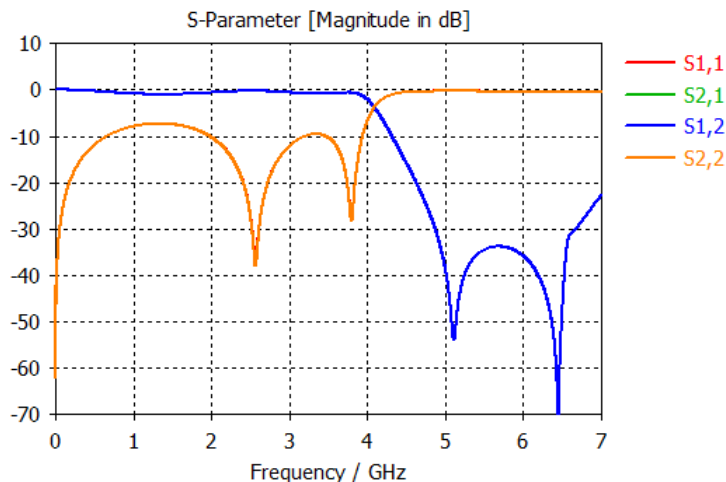
As you can see, the results from schematic and from 3D compare very well. The approach to compose the structure of different elements was reasonable in this case.

Assemblies

In the previous section we have seen how a full 3D model can be created from microstrip and stripline blocks. These blocks were aligned almost automatically in the layout view. The layout view can also be used to create a 3D representation of arbitrary 3D structure elements. To demonstrate this functionality we would like to create a horn-reflector antenna. If you do not want to create the structures on your own, you can find the already set up and simulated example *Assembly.cst* in the *Examples\DS\Workflow\Assemblies* directory of your installation.

As the name already suggests, a horn-reflector antenna consists of a horn antenna that illuminates a reflector dish. We can create these two structures by the help of two online macros.
Please create a new CST MWS project without any project templates and select *Home: Macros ⇨ Macros 📝 ⇨ Construct ⇨ Parts ⇨ Reflector dish*. In the *Construct Reflector dish* macro definition dialog box set the following values

| Dialog box setting	Value
Diameter of the reflector (D)	60
Focal length of the reflector	0.5*D
Specify whether the reflector has finite (1) or no thickness (0)	0

and save the project as *Reflector.cst*.

Now open another new CST MWS project (again without any project template) and create the horn antenna by selecting *Home: Macros ⇨ Macros 📝 ⇨ Construct ⇨ Demo Examples ⇨ Horn antenna*. The horn antenna is already parameterized as listed in the *Parameter List* window. Change the parameter according to the table below:

| Horn parameter	Value
Fctr	5
Fmax	5
Fmin	4
Horn_length	100
Taper_angle	20
Wall_thickness	10
Waveguide_height	20
Waveguide_width	40

Save the project as *Horn_Antenna.cst*
With these two structures, we already have the basic elements of our antenna at hand.
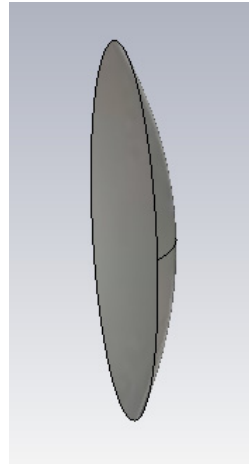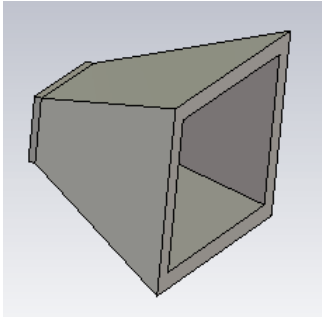
In the next step we need to assemble these two structures into one. Please close all open projects and create a new CST DESIGN STUDIO project. Save it as *Assembly.cst*. In the *Block Selection Tree* select *Field Simulators* and place a CST STUDIO SUITE block onto the schematic (Please have a look in the *Quick tour* section for an explanation how to do it). Select *Reflector.cst* in the file selector dialog box and press *OK*. Create another CST MWS block with the Horn antenna.

Note: You can place a CST MWS block into the schematic also by directly dragging the desired .cst file from the windows file explorer onto the schematic.

Please save the project and choose *NT: Layout* or *Home: Edit ⇨ Layout* 🟩. A new view will open containing all blocks having a 3D representation:
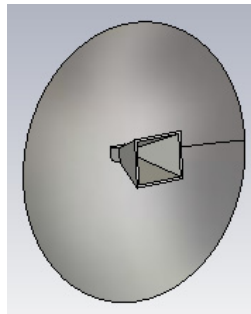

(The grid has been hidden pressing *View: Visibility ⇨ Working Plane*.)

Note: In the assembly view, all included 3D projects are automatically scaled to the units chosen in the assembly project.

Since there is no information about how to automatically align the two structures, they are positioned relative to the origin in the same way as they were in the original projects. To place them correctly, we need to transform the horn antenna manually.

Transformations can be done by selecting the desired block and choosing *Home: Tools ⇨ Transform* 🔧.
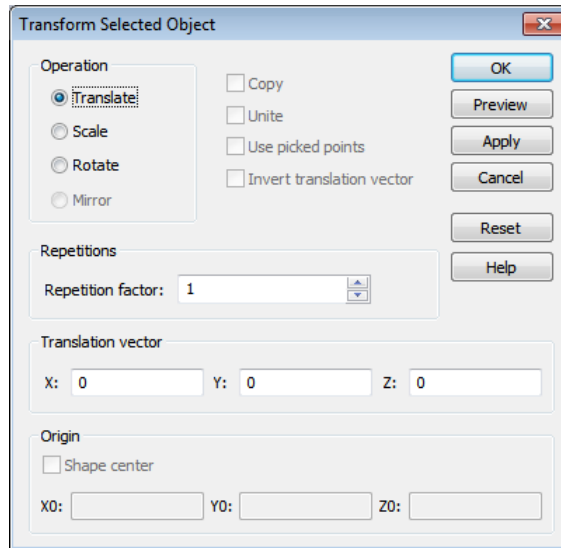
Here all basic transformations can be performed. To let the horn antenna illuminate the dish from the focal point and at an angle of 45 degrees, the horn must be transformed step by step as described in the following table:

Purpose of transformation	Operation	Settings
Centre to origin	Translate	X = -20, Y = -10
Correct radiation direction	Rotate	X = 180, Origin Z0 = 53
Move to focal point	Translate	Z = 300
Rotate to final location	Rotate	X = 45

Now the assembly is ready:
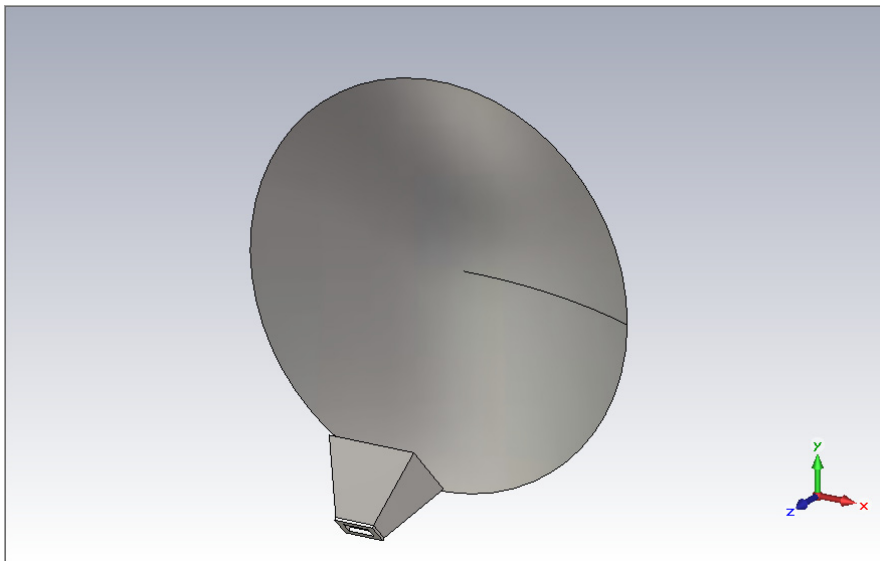
The shown structure is a 3D representation of all elements shown in the schematic. It is not yet a project that can be simulated in 3D. To do so, a *Simulation project* needs to be created. This time we use the shortcut: *Home: Simulation ⇨ Simulation Project ⇨ All Blocks as 3D models* from the schematic view.

Adjust all settings in the *Create New Simulation Project* dialog box as shown below:
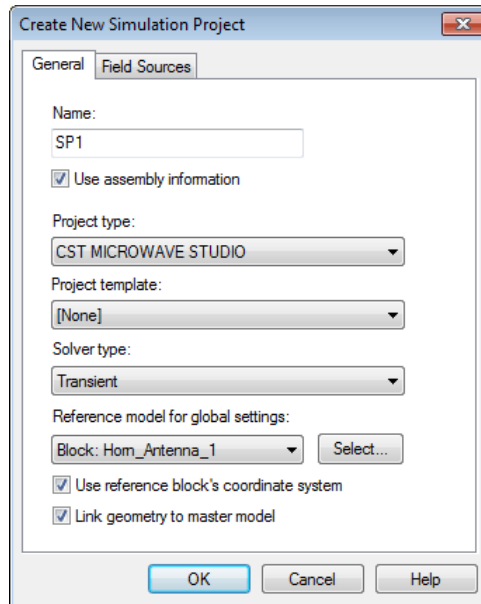
If, as in our case, a block has been chosen in the *Reference model for global settings* drop down box, the simulation project will be generated in a way that most of its solver settings are the same as defined in the reference model. You can click on *Select* to see which solver settings will be considered. This setting is very useful since in many cases it will save you a lot of configuration time.

Another quite important setting is the *Use reference block's coordinate system* setting. It ensures that the coordinate system of the new project is the same as the one used in the reference model. All other structure elements are imported relative to this coordinate system. This is necessary if the used solver has some constraints on the orientation of some solver features. For instance, the time domain solver we have chosen requires all wave guide ports to be aligned perpendicular to the global coordinate system. Since this is the case for the reference model, its coordinate system should be preserved, while all other structure elements need to be transformed accordingly.

After pressing *OK* in the dialog box, a new CST MWS project with the correctly assembled structure is created, nearly ready for simulation. Use the mouse to rotate the objects (move cursor with pressed left mouse button) if necessary to get a better view.
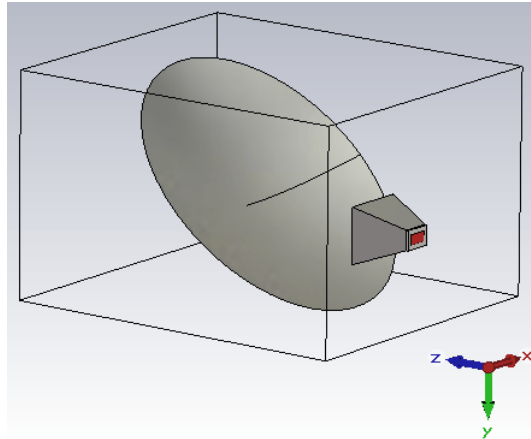
The only settings that need adjustment are the symmetry planes. Currently there are two symmetry planes: One in the YZ plane and one in the XZ plane. For the horn antenna these settings were correct, but for the assembled project the XZ plane is not valid any more.

In the opened CST MWS please select *Simulation: Settings ⇨ Boundaries* 📦, go to the *Symmetry Planes* tab and remove the *electric* symmetry in the *XZ plane*. Press *OK* to close the dialog box.

Now switch back to CST Design Studio Schematic and update all tasks. The field solver will start, and after some time, list its results in the result tree.
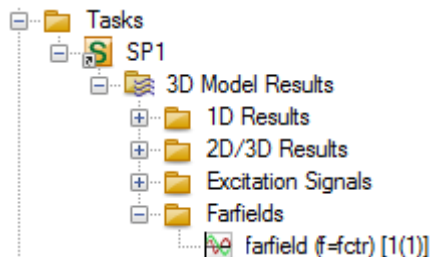
To visualize the resulting farfield just click on the corresponding tree item.
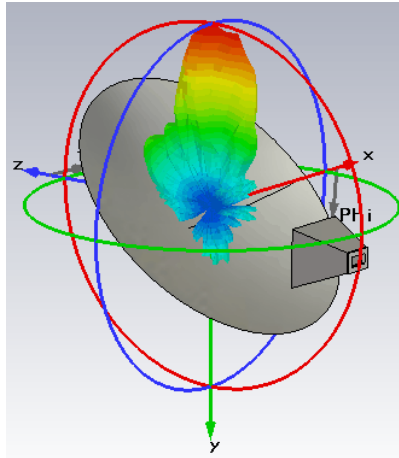
For this plot the *Farfield Plot Visibility ⇨ Show structure* option was switched on.

Managing Variations

A common task is to simulate variations of an already assembled structure. Additionally, all these variations should be automatically maintained or managed by a master setup in order to keep track of the entire project.

Let us have a further look at the example above. There are several possible scenarios for it. For instance, one might want to:

☐ Simulate the dish with the I-Solver, using the farfield result of the horn antenna as illuminating source.

☐ Include some fixture that connects the horn with the dish.

☐ Do parametric studies.

☐ And many more…

Unfortunately, within this document we can give you only a very brief introduction to this large variety of possibilities. Therefore we will concentrate on the first mentioned workflow. To do so we simply continue working on the example of the previous chapter. For not getting confused we recommend to disable the already existing simulation project by right clicking on the tree item *NT: Tasks ⇨ SP1* and selecting *Disable*.
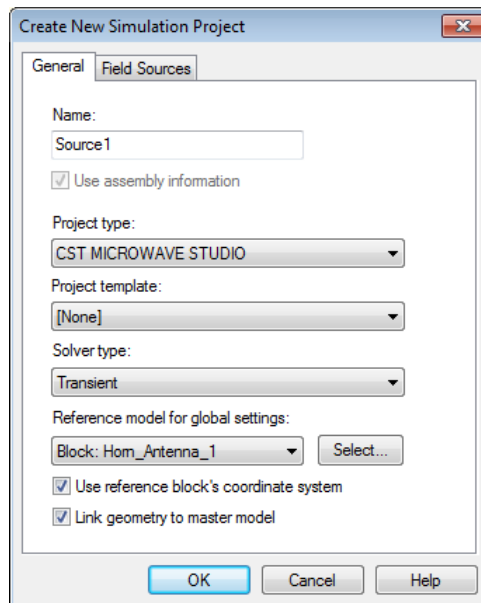
Field Source Coupling

There are two aspects when simulating the dish illuminated with a farfield source.

☐ Different parts of the assembly need to be extracted and simulated in separate projects.

☐ A fixed sequence of tasks needs to be defined ensuring that the horn antenna is simulated prior to the dish.

To ease the setup, there is a wizard that guides you through the different simulation project creation steps. Select *Home: Simulation ⇨ Simulation Project ⇨ Field Source Coupling.* The schematic will change its color to a light yellow to indicate that we are in the simulation project creation mode and some guiding text is displayed.
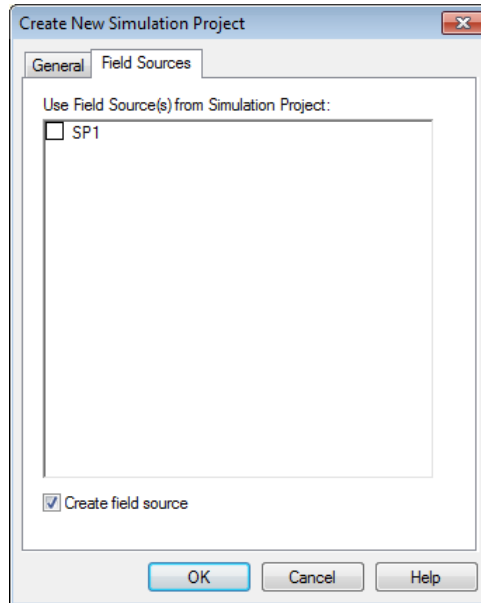
Source Project

As the guiding text says, we first need to specify the project that defines the field source first. Select block *Horn_Antenna_1* and press *Simulation Project: Model Representation ⇨ 3D Model* 🔲 and create the project (*Simulation Project: Create Project ⇨ Create Simulation Project* 🗐)


Make sure that all settings are the same as shown in the picture.

If you switch to the *Field Sources* tab you will see that the field source coupling wizard has already set the *Create field source* check box. This setting makes sure that the farfield results of the source project can be used as farfield sources for the destination project.

Press *OK* to start the project creation. During the process a *Create Field Source* dialog box will appear. Please select *Farfield source* as field source type and choose *Fmin = Fmax* = 5 with *Freq. Samples* = 1 since we are interested in the resulting farfield at 5 GHz only.
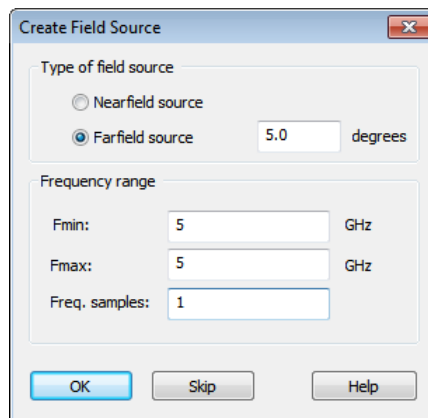
Press *OK* to finish the project creation.

Close the simulation project mode (*Simulation Project: Close* ⇨ *Close Simulation Project Mode*) since we do have only one antenna as farfield source.

Dish Sub Project

Now the guiding text asks us to specify the target project. Include block *Reflector_1* as 3D model and create the simulation project: Select block *Reflector_1*, press *Simulation*

Project: Model Representation ⇨ *3D Model* 🔲 and *Simulation Project: Create Project* ⇨ *Create Simulation Project* 🔲.

Select the *Integral Equation* solver and this time choose *SP:SP1* in the *Reference model for global settings* drop down box. By choosing the simulation project instead of the block *Horn_Antenna_1*, we automatically get the correct symmetry settings for the 3D simulation (those needed to be adjusted when creating SP1).

The *Use reference block's coordinate system* is not available but since the I-Solver does not require any special orientation of the structure it is not necessary anyway.
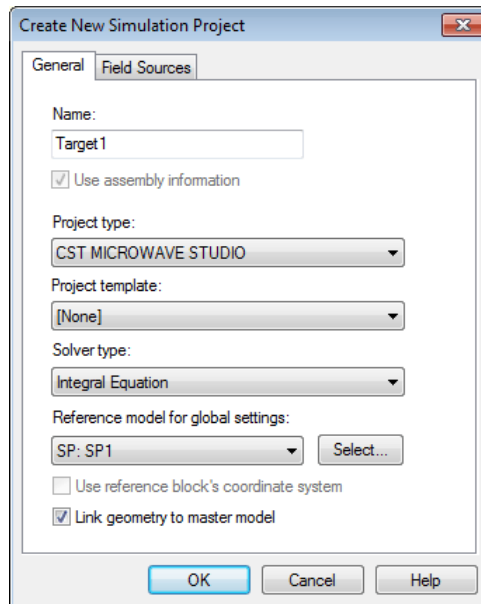

If you now switch to the *Field Sources* tab, you can see that the field source coupling wizard has already selected the *Source1* project as farfield source in the *Target1* project.
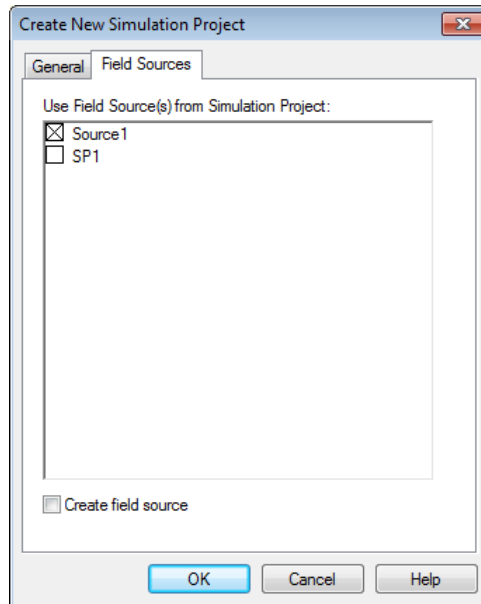
Press *OK* to create the simulation project.

Simulation Sequence

The setup for the two projects is finished now. If you have a look at the navigation tree of the *Assembly* project, you should see the final simulation setup:
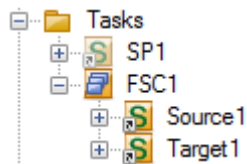
The two simulation projects are already added into a sequence task with the correct order.

To start the entire simulation just press *Home: Simulation ⇨ Update* . First the *Source1* simulation project will be solved, followed by the *Target1* simulation project. Please switch to the *Target1* project after the simulation has been finished and select *NT: Farfield Source ⇨ ffs1*.
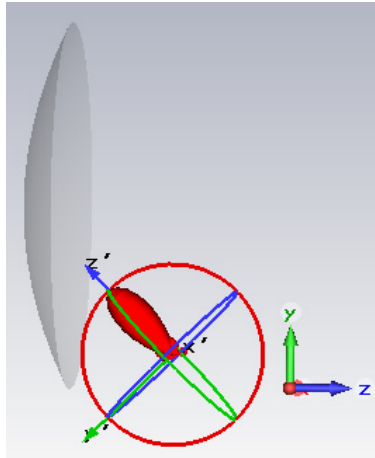
You can see that the farfield source has indeed been imported automatically, retaining the original position of the horn antenna. No manual positioning of the farfield source was necessary.

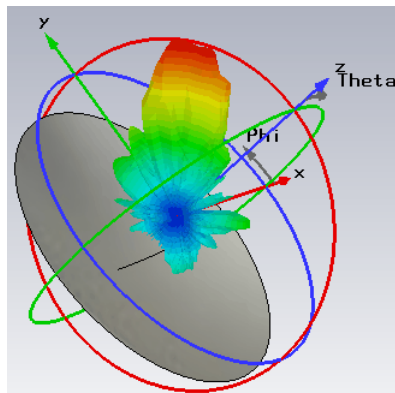In the end, the simulation results of the I-Solver are indeed nearly the same as for the full 3D simulation (*NT: Farfield ⇨ farfield (f=fctr) [ffs 1]*):

Parametric Changes

After having set up the entire project it is very easy to do investigations concerning parametric changes. Please switch to the schematic view of the *Assembly* project and select the horn antenna block. The taper angle of the horn is already parameterized. Please change its value from 20 to 5. If you now switch to the *Source1* and *Target1* projects, you will see the message: *The 3D models are outdated. Press F7 for a complete update (Press ESC to cancel)*. After having changed the parameter, the changes are propagated to each derived project that reacts by the *outdated* message. If you now start the simulation again, all sub-projects will update and simulate in the correct order automatically.
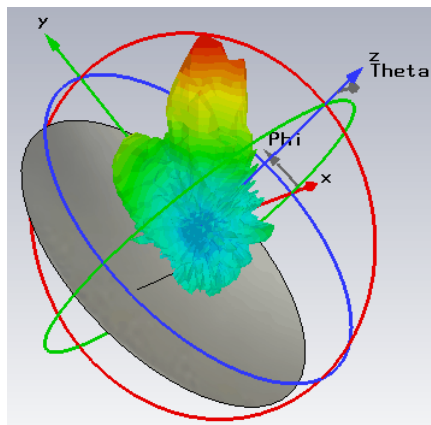
Remarks

Since the shown examples were chosen to explain the principle workflows, the geometries do not represent a 'real world design'. Also some solver settings of the 3D solvers have been chosen to run fast but possibly less accurate. Please have a look at the corresponding CST MWS documentation for proper solver settings.

Chapter 4 — Schematic View

For all CST STUDIO SUITE modules that offer 3D simulations[*], two fundamentally different views on the structure exist. A 3D view that is visible by default and a schematic view, showing an abstract, terminal oriented representation of the structure. The schematic view allows to embed the structure in a more complex environment. Several components may be added, such as resistors, capacitors, transistors, transmission line models or even other 3D CST projects.

This chapter addresses CST STUDIO SUITE users who are mainly dealing with field simulation but who also need the capabilities of a powerful circuit simulator that is tightly coupled with the 3D world.

To demonstrate the main concepts of the schematic view, we will use the RJ45 multipin connector from the CST MICROWAVE STUDIO examples. You can find it in the *Examples\MWS\Transient\S-Parameters\Misc\Rj45* folder of the installation directory. Although this is a CST MWS project, the described workflow similarly applies for other CST STUDIO SUITE projects as well.

Main Concepts

After having opened the project, the schematic view can be activated by selecting the corresponding tab on the bottom of the main view:
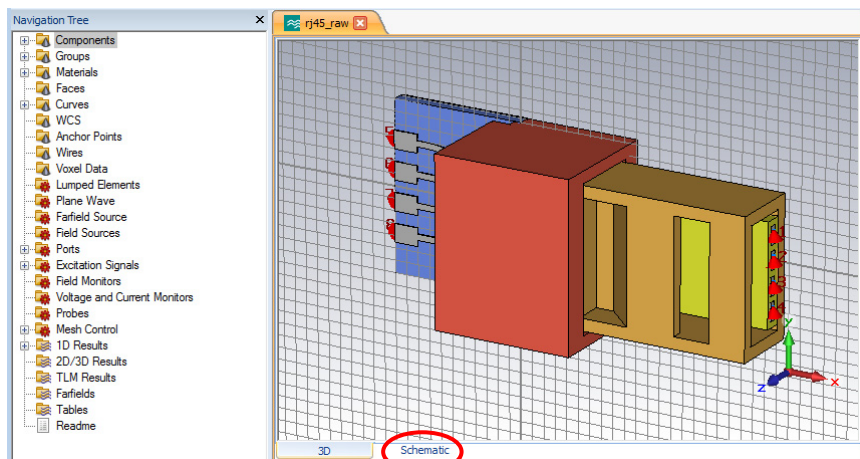
In the beginning, the schematic view shows only one single CST MICROWAVE STUDIO schematic block (MWS block). It represents the 3D structure with its terminals.

---

[*] CST MICROWAVE STUDIO® , CST EM STUDIO® , CST MPHYSICS® STUDIO, CST PARTICLE STUDIO® , CST PCB STUDIO®, CST CABLE STUDIO®
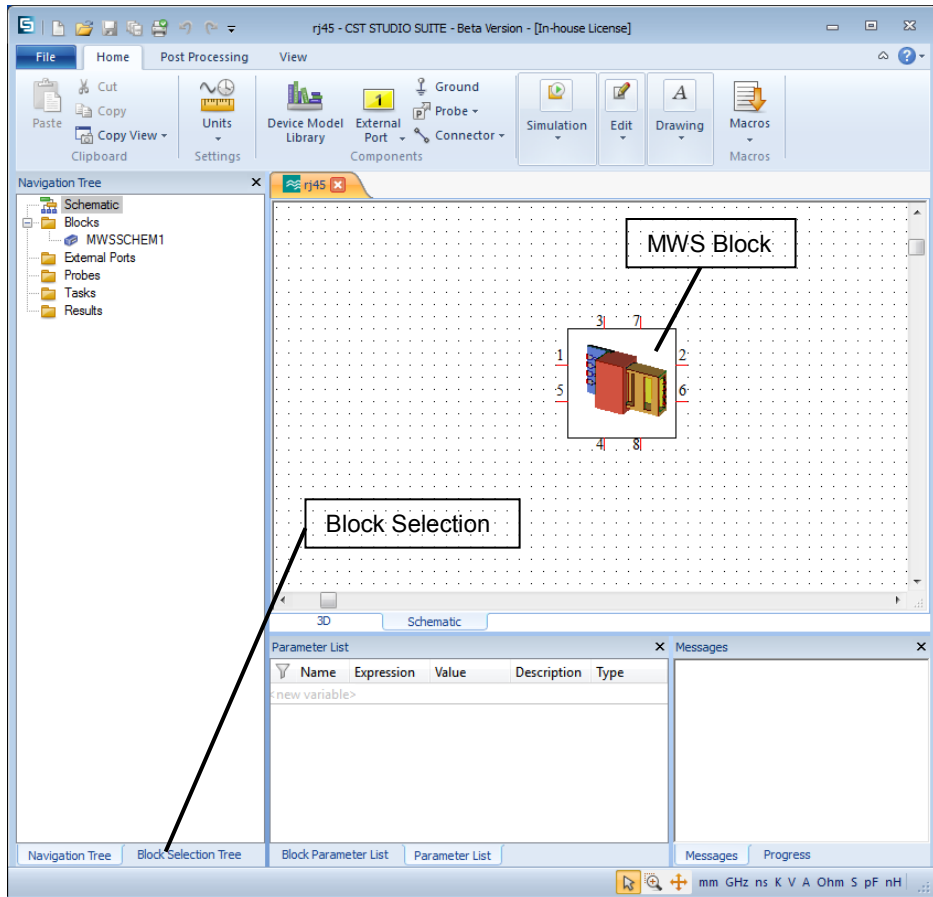
The terminals (pins) are a one-to-one correspondence to the 3D structure's waveguide or discrete ports. The schematic view now allows easy addition of external blocks or circuit elements to the terminals of the 3D structure.

Adding circuit elements to the schematic view is straightforward, simply navigate to the block selection tree, select one of the available blocks within the selected folder, and drag it onto the schematic view. Please refer to the corresponding chapters in the beginning of this document to get more information how to place circuit elements into the schematic.

The following picture shows an exemplary external circuit where the original 8-port structure has been reduced to a 2-port device. This is done by connecting two pins to external ports (the yellow symbols) that define the terminals of the circuit and by terminating all other pins with 50 Ohm resistors:
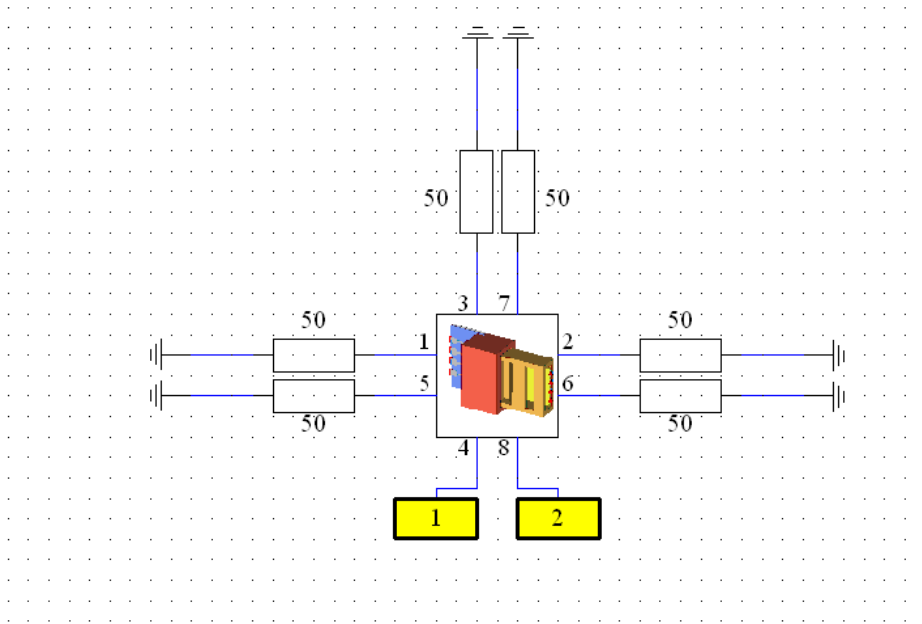
Now, an S-Parameter simulation can be performed. In the example above, the resulting S-matrix will be a 2x2 matrix since only two external ports have been defined.
In order to run an S-Parameter circuit simulation, choose *Home: Simulation ⇨ Update* . The circuit simulator supports a variety of different analysis options, so you will be prompted to select one of them:
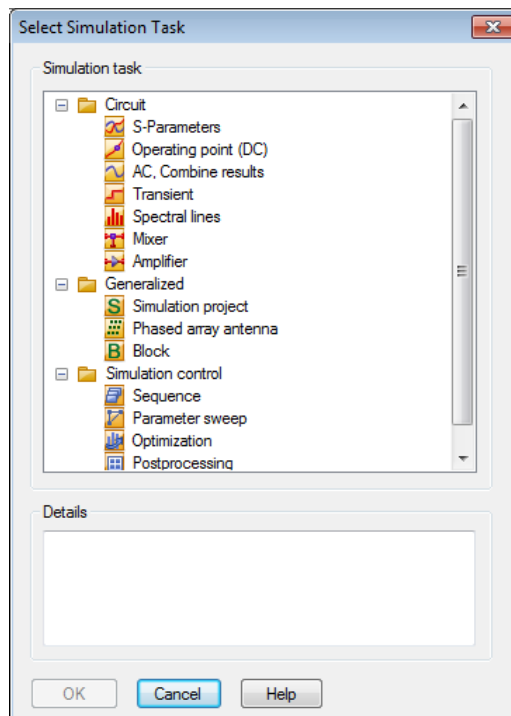
Here you should select *S-Parameters* and click OK. A dialog box will appear in which you can specify the settings of the S-Parameter calculation:
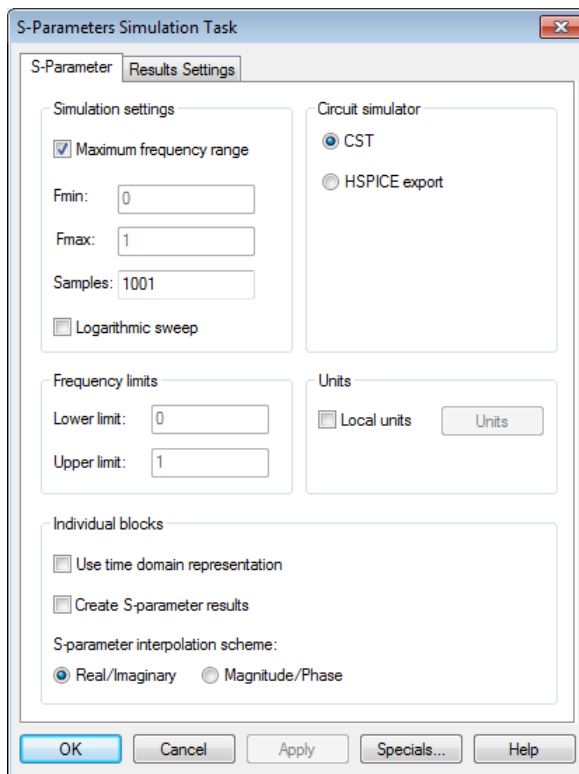
Clicking OK will start the S-Parameter simulation of the coupled EM and circuit problem. If all simulation results of the 3D EM simulation are already present, the circuit simulation will only take a few seconds to complete. Otherwise, the missing EM simulation data will be automatically computed first.

Once the simulation is complete, the results will be added to the navigation tree.
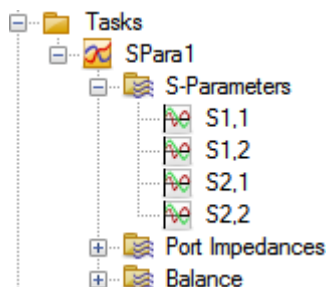
The schematic view also supports the parameterization of the 3D model in a straightforward way. Both, the schematic view and the 3D view use the same project parameters.

The tight integration of the 3D and the schematic modules also allows another type of coupled simulation: The transient EM/circuit co-simulation.

Transient EM/Circuit Co-Simulation
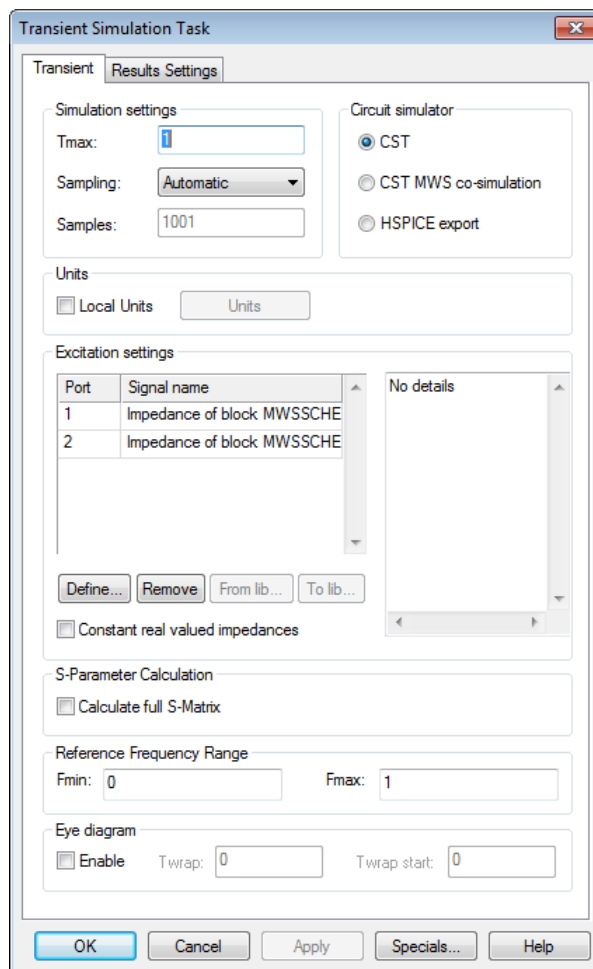
The transient EM/circuit co-simulation is an alternative to the standard EM/circuit co-simulation that is available for coupled circuit-CST MICROWAVE STUDIO problems. The standard EM/circuit co-simulation of CST DESIGN STUDIO and CST MWS uses S-Parameters to describe CST MWS blocks. The computation of S-Parameters by CST MWS requires either a frequency sweep of the frequency domain solver, or one simulation per port of the time domain solver. The resulting S-Parameters describe the block for any combination of port excitations.

The transient EM/circuit co-simulation uses a different approach. Only one simulation of the coupled circuit-EM problem is performed, with exactly the excitation that is defined in the circuit. Both, the circuit and the EM problem are solved simultaneously. This approach may be faster than the standard EM/circuit co-simulation (especially for large numbers of ports), since no general S-Parameters need to be calculated.
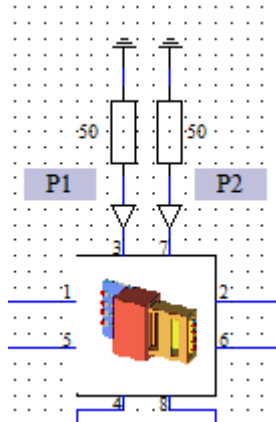

The setup of a transient EM/circuit co-simulation is very similar to the setup of the S-Parameter simulation described above. Open the *Home: Simulation ⇨ New Task* dialog box. Instead of *S-Parameters* choose *Transient* as the task type.

In order to perform a transient EM/circuit co-simulation, select the *CST MWS co-simulation* radio button. Set the total simulation time (*Tmax*) to 5 ns. Leave the *Sampling* setting to *Automatic,* which is the default. The number of samples (*Samples*) is inactive, since it defines the plot resolution of the result curves only if the *Manual* sampling mode is chosen. Now the excitations need to be defined. Mark the first port in the *Excitation settings* frame and press *Define…*. This will bring up the *Excitation signal* dialog box:
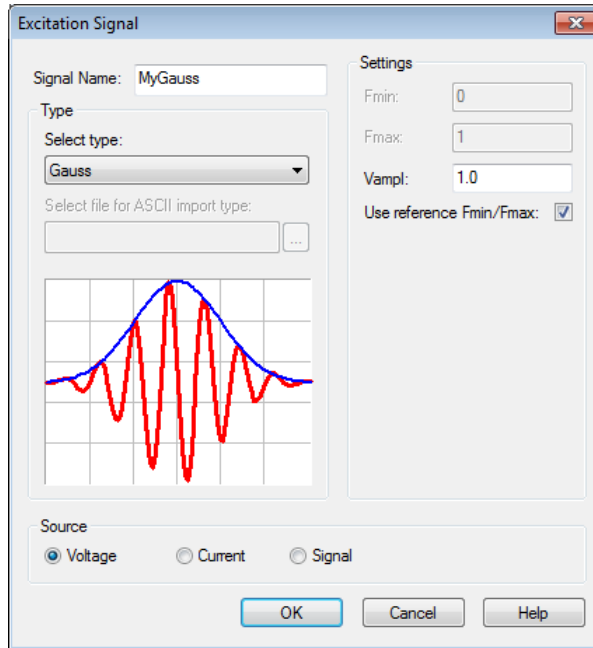
Here the *Use reference Fmin/Fmax* is selected which means that the frequency range of the Gaussian signal will be taken from the task's *Reference Frequency Range* setting. Press *OK* to apply the signal definition. Back in the task dialog box make sure that *Fmax* is set to *1*.

In general, the setup of the transient simulation task is completed now. The only missing bit is to define which signal results of the circuit you are interested in. This is done by probes: A probe can be attached to any connector in your design which will record the associated voltage and current for visualization or further processing. To insert a probe, simply click on *Home: Components ⇨ Probe* 🅿 and afterwards click onto the desired connector. In our simple example we are interested in the voltages and currents at the ports and at pin 7 and 3 of the MWS block. The signals at the ports are monitored automatically, so we need to define only two probes:

Now the setup of the transient EM/circuit co-simulation is completed and you may start the simulation with *Home: Simulation ⇨ Update* . After the simulation has finished, the results appear below the task item *NT: Tasks ⇨ Tran1*.
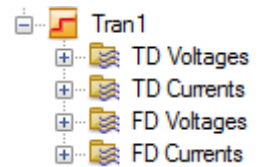
In this simple example you will notice the Gaussian pulse travelling from port 1 to port 2 causing some cross talk in probes *P1* and *P2*. More realistic examples may of course use any of the excitation signals available from the excitation signal library (including user defined signals). Excitation of more than one port is of course also possible.

For further information about transient simulation in general and transient EM/circuit co-simulation in particular, please refer to the online help.

# Chapter 5 — Schematic View in CST CABLE STUDIO / CST PCB STUDIO

This chapter covers the specific aspects of the schematic view of CST CABLE STUDIO and CST PCB STUDIO

Schematic view in CST CABLE STUDIO

CST CABLE STUDIO (CST CS) is a tool especially designed for simulating cable harnesses. The basis of CST CS is a CST MWS module, tailored to the needs of cable modeling.

CST CABLE STUDIO not only simulates the behavior of cable harnesses themselves. It may also simulate the effects of a 3D environment on the harness. Such an environment may be chassis or, even more complex, an antenna imposing a 3D field to the harness. Generally, a CST CS project may contain a cable harness with terminals, a 3D structure with field excitations like 3D ports.

As for the other modules, the schematic view shows a schematic block that represents the electrical model of the project. It shows pins for all defined 3D ports and cable terminals. In the example below you can see a simple single wire on a ground plane that is illuminated by a dipole antenna. The dipole is excited by a discrete 3D port. Correspondingly, the schematic view shows a CS block with two pins for the two cable terminals and one for the 3D port.
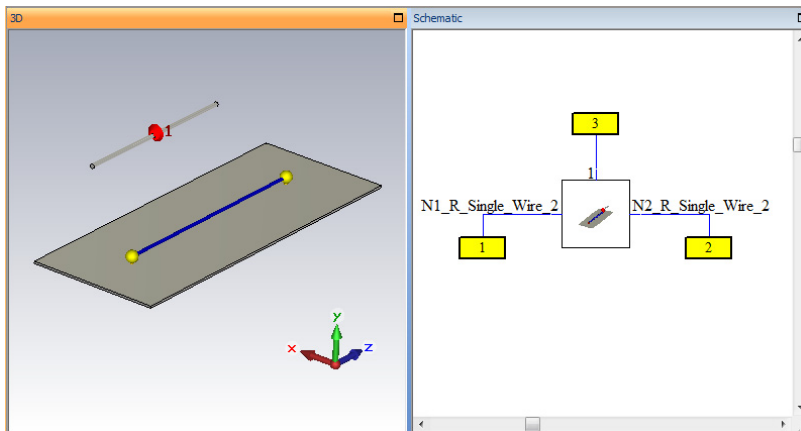
If a cable without any 3D environment is simulated, CST CS creates an equivalent transmission line circuit model from the cable harness. In a second step this model is used by any circuit solver of the schematic to perform the desired simulation. Such a simulation may be for instance a transient task or an AC task. This procedure already describes the fundamental difference between the simulation of cable models and the simulation of general 3D structures. While a 3D field solver is able to directly calculate results in terms of fields, currents, voltages or S-Parameters without using a solver from the schematic view, the cable modeling of CST CS is not. To simulate a cable model a circuit simulator is always needed.

When simulating a cable in a 3D environment, the solving process incorporates the field solver as well as the circuit simulator. Both simulators solve their domain and exchange all necessary information to finally obtain a coupled solution.

For more information about CST CABLE STUDIO please have a look into the *CST CABLE STUDIO – Workflow and Solver Overview* document.

Schematic View in CST PCB STUDIO

CST PCB STUDIO (CST PCBS) is a tool especially designed for simulating printed circuit boards. Since the analysis of a PCB has many different aspects, CST PCB STUDIO also offers a variety of different modeling/solver modules that cover the different simulation needs. For instance, you will find modules for solving SI-TD problems as well as IR-Drop or PI analyses.

Many of the solver modules heavily use the schematic view. They automatically set up a circuitry that not only contains an equivalent electrical model of the PCB but also elements and excitations defined by the imported layout. This circuit is then simulated by an appropriate simulation task to gain the desired results. CST PCBS heavily uses SAM to create several simulation projects that cover and manage different simulation aspects of the entire board.

For more information about CST PCBS please have a look into the *CST PCB STUDIO – Workflow and Solver Overview* document

Chapter 6 — CST STUDIO SUITE Projects in CST DESIGN STUDIO

This section explains how 3D field simulating CST STUDIO SUITE projects[*] (CST 3D project) can be added to a CST DESIGN STUDIO design.

Whenever you want to incorporate a CST 3D project into your CST DS design, you have the choice to either use a parameterized block or a file reference block.

Parameterized blocks:

- ☐   Maintains a copy of the original project.

- ☐   Allows parametric control from within CST DESIGN STUDIO.

- ☐   In case of a CST MWS project, takes advantage of the global result cache, a repository of S-Parameter results of already calculated CST MWS projects.

File Reference blocks:

- ☐   Maintains a reference to the original project.

- ☐   Recognizes project changes to provide most up-to-date results.

To give you an idea of the capabilities offered by these blocks, we will construct an example and demonstrate the key features in the following chapters. Since the main concepts are the same for all CST 3D projects, we will explain them in case of a CST MICROWAVE STUDIO project.

Example Introduction

The model used for the example is shown in the image below. A rectangular patch antenna with two feeds having impedances of approx. 100 Ω is connected to two impedance transformers. If you do not want to set up the model manually, you can find the already set up and simulated *Antenna.cst* version in the *Examples\DS\S-Parameter\Circuit-EM\Matched Antenna* folder of the installation directory. Also, the individual parts of the antenna as the *transformer.cst* and the *patch.cst* can be found in the *Examples\DS\S-Parameter\Circuit-EM\Matched Antenna\Antenna_data* directory.

---

[*] CST MICROWAVE STUDIO® , CST EM STUDIO® , CST MPHYSICS® STUDIO, CST PARTICLE STUDIO® , CST PCB STUDIO® (only file blocks), CST CABLE STUDIO®
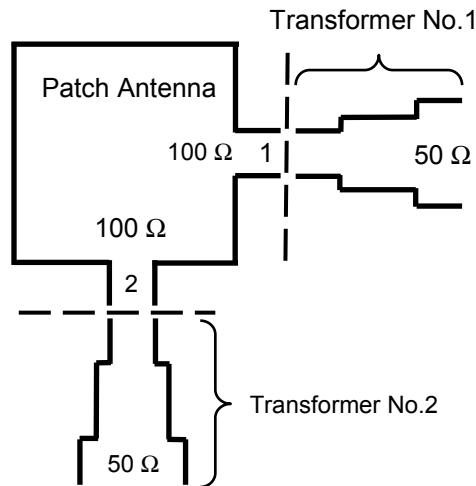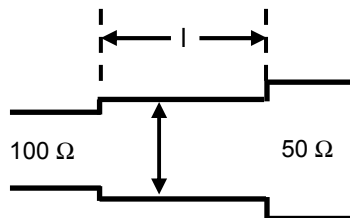
We assume a fixed antenna design with no parameterization. The antenna radiation frequencies are as follows:

$$f_1 = 7 \text{ GHz for excitation at port No.1}$$
$$f_2 = 7.5 \text{ GHz for excitation at port No.2.}$$

The transformers consist of two microstrip step discontinuities with a microstrip line of length l and width w in between:

To distinguish between the widths of transformer 1 and 2, we will call their widths w_1 and w_2, respectively.

The other line widths are defined by the patch antenna's port impedances of approx. 100 Ω at port two and 50 Ω at port one.

Our design goal represents a typical matching problem and can be formulated as follows: Determine values for l_1, l_2, w_1 and w_2 to obtain minimal reflection at the transformers' input ports (50 Ω) for the original antenna frequencies.

To simplify this task, we reduce the number of parameters by choosing $l_1 = l_2 = 7$ mm. Thus, w_1 and w_2 remain as parameters to be optimized for demonstration purposes.

CST MICROWAVE STUDIO Models

First we must set up the CST MICROWAVE STUDIO models that we will use within our CST DESIGN STUDIO project. If you are not familiar with CST MWS, you may read

through the *CST STUDIO SUITE Getting Started* and *CST MICROWAVE STUDIO Overview* manuals first.
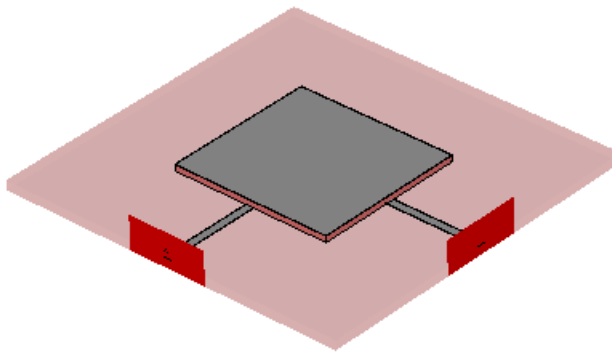
For both the antenna and transformer models, the substrate and the metallization will have the following values:

Name	Value
Dielectric constant ε_r	2.2
Height	0.794 mm
Metallizations' thickness	0.05 mm
Metallizations' Material	PEC

The hexahedral transient field solver will be used.

Antenna

The image below shows the rectangular patch antenna.

The dimensions are the following:

| Name	Value
Size	12.6mm x 13.6 mm
h_{add}	0.5 mm
Microstrip line width	0.7 mm
l_{add}	4 mm
Distance from patch to port	8 mm

The patch is elevated from the substrate by the additional h_{add}. The space between the patch and the substrate is also filled with substrate material. Therefore, the resulting substrate height below the patch is h_{patch} = 1.294 mm.

The feeding microstrip lines are located on the original substrate and end below the patch. The length l_{add} defines the distance between the patch's edge and line's end (as shown in the image below).
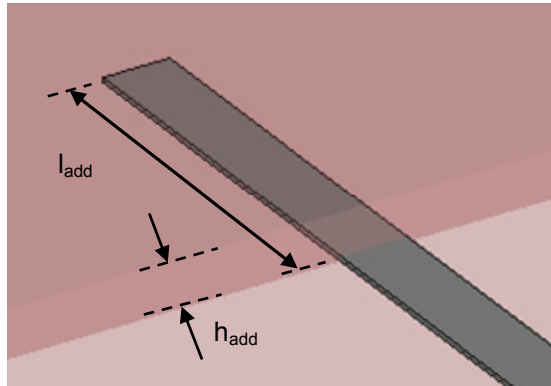
The frequency range is set to $5 \leq f / GHz \leq 10$. To get decent mesh and solver settings for this project, a project template for planar antennas has been used.

The S-Parameters S1,1 and S2,2 show minima at the antenna's radiation frequencies f_1 and f_2:
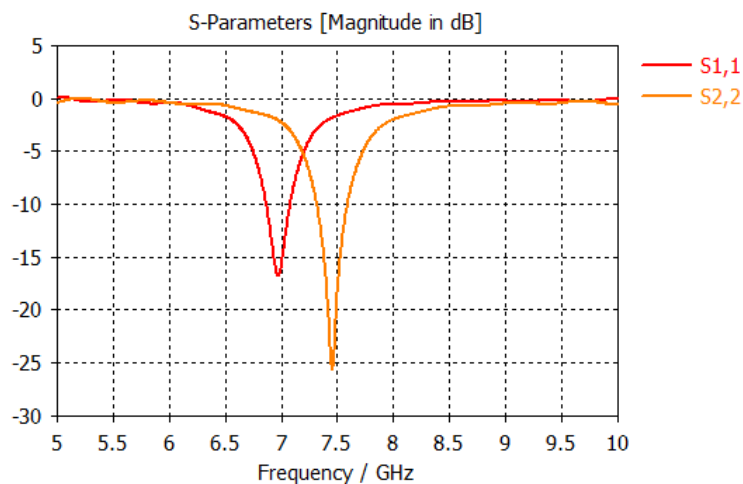
Finally, two farfield monitors should be defined at 7 GHz and 7.5 GHz to perform a final antenna calculation from within CST DESIGN STUDIO.

Transformer

Although CST DESIGN STUDIO provides an analytical model for the microstrip step discontinuity, a CST MICROWAVE STUDIO model of the transformer is used to demonstrate the parametric control of CST MWS projects from within CST DS.
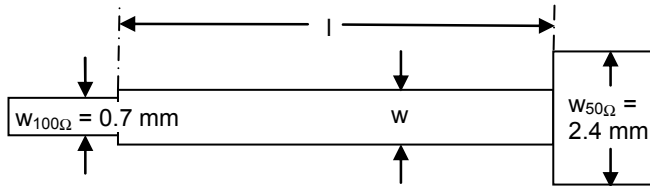
Two parameters are defined for the model of the transformer: w for the transformer's width and I for the transformer's length. The widths for the input and output microstrip lines are defined by their impedances of 50 Ω and 100 Ω. Considering the substrate defined above, we obtain the following dimensions:

Name	Value
$w_{100\Omega}$	0.7 mm
$w_{50\Omega}$	2.4 mm
I	7 mm
$w_{initial}$	1 mm

To get a reasonable accuracy to speed relation, a project template for planar filters has been used but with a disabled mesh adaption.

CST DESIGN STUDIO Modeling

As mentioned in the introduction to this chapter, CST DS provides two types of CST 3D project blocks. In this section, we will give a more detailed overview over the properties and usage of these blocks.

CST STUDIO SUITE File Block

A block of this type holds a reference to a CST 3D project. You find it in the *Field Simulators* folder of the block selection tree. If the underlying project is a CST MICROWAVE STUDIO project, a CST MWS file block will be created. This block basically represents the S-Parameters of the referenced project[*]. Additionally, the block keeps track of modifications of the project such that CST DESIGN STUDIO makes sure that it uses the most up-to-date results. If some required results are missing in the CST MWS project – e.g. the project has not been simulated yet or only a subset of all existing ports have been excited – the required CST MWS simulation is automatically started from CST DS before the circuit simulation is performed.

The usage of a CST SUTIO SUITE file block is quite simple: After dropping this type of block from the block selection tree into the schematic, the *Import CST STUDIO SUITE File* dialog box is opened where you may browse for a CST 3D project file. Alternatively, a CST STUDIO SUITE file block may be created by dragging a CST 3D project from a file browser onto the schematic while pressing the CTRL key.

There are two additional properties that can be set while inserting the block, but that can also be modified later by customizing the block's property dialog box:

---

[*] Other modules may represent different results: A CST EMS project for instance will provide an impedance matrix.

☐ *Store Relative Path*: You can either store the relative path or the absolute path from the current CST DS project to the selected CST 3D project. This option is disabled if the CST DS project has not yet been saved (as there is no path for this project at all). Then, the absolute path will be automatically stored. Both options make sense, depending on how you wish to deal with the project in the future. If there is a project repository on a server it is useful to provide absolute paths because you just need to send the CST DS project file to a colleague who may also access the server. On the other hand, if you want to send a project file to someone who cannot access the server, perhaps by e-mail, it is more useful to copy the CST project to a local folder and provide the relative path.
**NOTE:** The relative path option is available only if both files are located on the same drive.

☐ *Use AR filter whenever possible* (only for CST MWS projects): Use the AR filter function of CST MWS to extract the S-Parameters from the time domain calculation.

A CST STUDIO SUITE file block is represented by a small image of the 3D model and a small arrow on the lower left indicating that it is the file block. The number of (block) ports corresponds to the number of (waveguide or discrete) ports[*] defined in the CST MWS model.
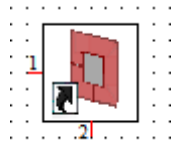
Let us examine the contents of a CST MWS file block's property dialog box. The initial page shown is the *General* page.
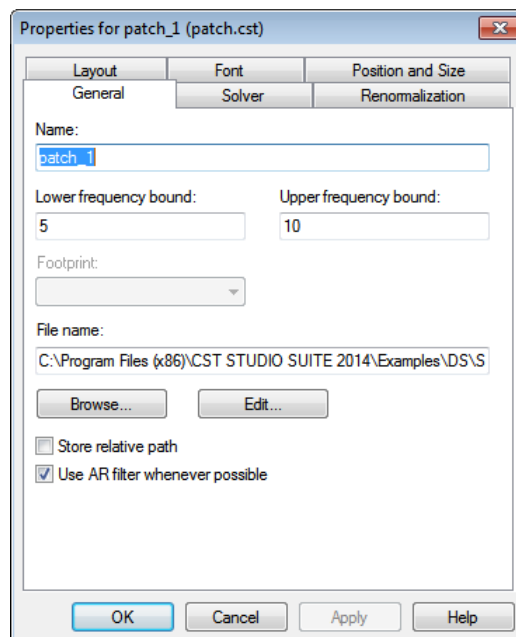
A CST MWS file block is always frequency bound. As usual, the limits are displayed in the *General* page as shown above. Furthermore, the file that the block refers to is

---

[*] Other modules may define different terminal types than waveguide or discrete ports.

displayed there. You can again choose between an absolute path and a relative path. If the CST DS project has not yet been saved, this option will be disabled and the absolute path will be considered.

Moreover, there are two buttons: *Browse* and *Edit*.
Pressing *Browse* opens the *Import CST MICROWAVE STUDIO File* dialog box where you can browse for a different project. Frequency bounds and the number of ports will be changed according to the new file's contents. Connections will be kept if there are some ports with identical names contained in the new project, otherwise the links will be deleted.
Pressing *Edit* opens the CST MWS project in a new tab or activates the project's tab if it is already open.

CST STUDIO SUITE Block

A CST STUDIO SUITE block allows you to parametrically deal with a CST STUDIO SUITE project. You find it in the *Field Simulators* folder of the block selection tree. After dropping this block into the schematic, the *Import CST STUDIO SUITE File* dialog box will be opened where you can browse for the project. Alternatively, a CST STUDIO SUITE block may be created by dragging a CST 3D project from a file browser onto the schematic.

In contrast to the CST STUDIO SUITE file block, this type of block does not refer to the selected project. Instead, the essential project files will be stored by the block. To recalculate the results or to open this project in a corresponding CST STUDIO SUITE module, these project files will be copied into a sub-folder of the current project's folder and opened from there. However, you cannot open this project in a standalone CST STUDIO SUITE module, since it is controlled by the CST DS project.

The most relevant feature supported by this type of block is the control of the CST project's parameters from within CST DS. Whenever you select a CST STUDIO SUITE block, the *Block Parameter List* window shows all parameters defined in the related project.

Result Cache (CST MICROWAVE STUDIO only)

A very useful addition to the parameter control is the usage of a cache, in which already calculated results for S-Parameters and port impedances are stored. Such a cache entry is created after every CST MWS simulation. This cache will be reused if you associate another block with this project, even if it was copied to a different location or stored under a different file name. A new cache entry is added only if the project was modified in the meantime. However, a cache entry is not identified by the project's path or file name, but rather by the project's contents.

A cache may be used by a group of users and each user may use several caches simultaneously. To define the caches that you want to use, choose *Edit: Result Cache ⇨ Set Cache Path*. The following dialog box is opened where you can specify an arbitrary number of cache paths. The paths can be ordered according to their priority (imagine that two caches may contain entries for the same project).
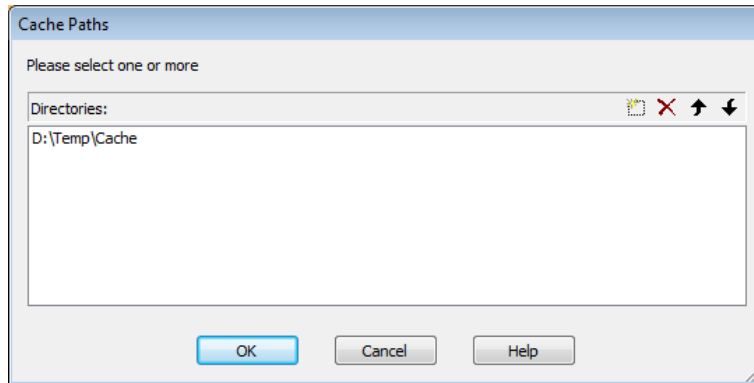
The results for each project are stored separately for different solver and mesh settings. They can be accessed (for read only) by selecting the CST MWS block and choosing *Edit: Result Cache ⇨ View Cache*

or by choosing *View Cache* from the block's context menu. The *Cache Overview* dialog box is then opened. Within this dialog box you can manage the cache entry that is associated with the block:
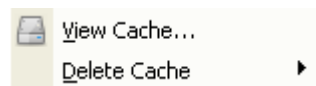
☐ The two selector boxes at the top of the dialog box allow you to switch between the stored parameter sets. Several sets can be stored for each solver that can be selected with the control to the right.

☐ You may delete the currently selected set from the cache by pressing the *Delete* button.

☐ To display the results associated with the currently selected parameter set, just press the *Show Set* button.

☐ Furthermore, the cache path that this entry is based on is displayed.
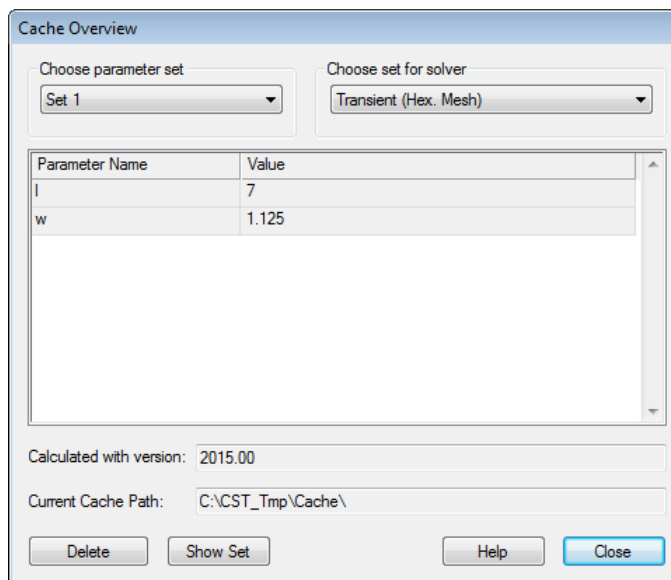
To quickly remove all cache entries (or all cache entries related to a single solver), you may more efficiently use *Edit: Result Cache* ⇨ *Delete Cache* or *Delete Cache* from the block's context menu.

It is worth mentioning that the availability of a result cache is very powerful, especially if interpolation (that is referred to below) is used. Once the cache is filled, e.g. by a parameter sweep, you can quickly obtain interpolated results from 3D simulations and even perform an optimization without waiting for time-consuming 3D simulations.

CST DESIGN STUDIO Simulation

The CST DESIGN STUDIO model of our example consists of three blocks:

- ☐ A CST MICROWAVE STUDIO file block is used for the antenna because it is a fixed model that does not have any parameters.

- ☐ Two CST MICROWAVE STUDIO blocks are used to create two independent copies of the previously created transformer model. Both models offer the parameter w that will be used to optimize each transformer from within CST DESIGN STUDIO.
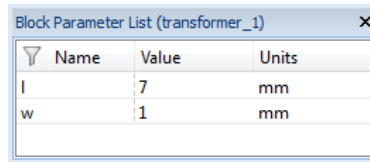
Add these blocks to your (initially empty) project, and connect them as shown below. The 100 Ω ports of the transformers are connected to the antenna. Also, insert two external ports there and connect them to the 50 Ω ports of the transformers.
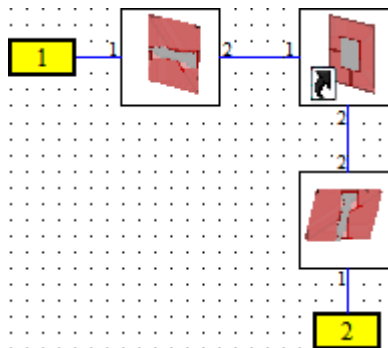
Now some parameters need to be defined. These parameters will be used to optimize the match between the antenna and the external ports. Use the docked parameter list control as explained during the *Quick Tour*, and create the parameters width1 = 1.0 and width2 = 1.0.

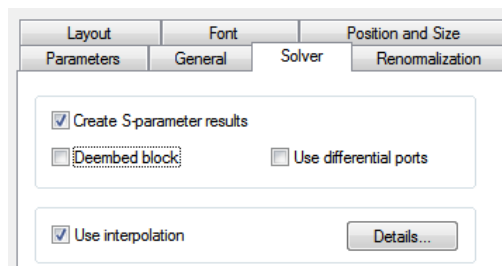Whenever a CST MICROWAVE STUDIO block is selected, its parameters are displayed in the docked parameter list control:

The transformer blocks show two parameters: w = 1 mm and l = 7 mm. They are initially set by the corresponding CST MICROWAVE STUDIO project. Modify the parameter w as follows:

Transformer No. 1: (The one connected to the antenna's port No.1) w = width1
Transformer No. 2: w = width2

CST DESIGN STUDIO is able to evaluate results for a CST MICROWAVE STUDIO block by interpolation. The interpolation uses the results of a number of specific supporting points in the parameter space to interpolate the results for all other parameter values that are 'inside' the values of the supporting points. The supporting points are created by sampling each parameter by a given number of samples within a given parameter range. Doing so, the supporting points form an equidistant grid throughout the parameter space.

Note: To actually get an interpolated value, the results of 2N-1 supporting points (with N = Number of parameters) are needed. This means, that for a project with two parameters three simulations are needed to get the actual result. Therefore using the interpolation scheme is mainly useful when performing a parameter sweep or an optimization run.

To utilize this feature, check *Post Processing: Manage Results ⇨ Result Cache ⇨ Use Interpolation* to make sure that the interpolation is turned on. Moreover, proper interpolation settings need to be chosen for each block. Open the *Block Properties* dialog box of one of the transformer blocks and go to the *Solver* property page:

Make sure that the *Use interpolation* option is switched on (this is the default setting). Click *Details* to assign proper interpolation ranges to the block's parameters. The *Interpolation Details* dialog box opens to display a list of the block's parameters, their ranges of validity (if they are limited), ranges for the interpolation, and numbers of samples for the interpolation.
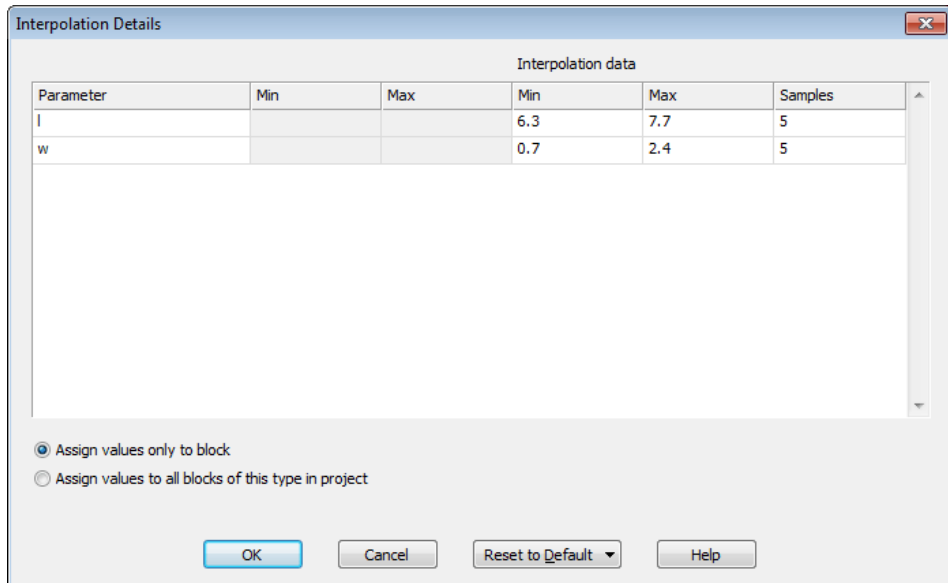
Because only the parameter w will be modified during our optimization process, keep the values for l. For w, choose an interpolation range proposed by the width of the adjoining strip lines: 0.7 ≤ w ≤ 2.4. Keep 5 for the number of samples for this parameter.

To apply this modification to both transformer blocks, choose *Assign values to all blocks of this type in project* at the bottom of the dialog box. It is useful to provide identical interpolation settings for them as they share the same cache since they are derived from the same CST MICROWAVE STUDIO project. Therefore, all results for different parameter sets calculated for the first block will be available for the other one, and vice versa. Because the parameter l will not be modified here, only five 3D calculations (the number of samples specified for w) will be performed at maximum if the values for w do not exceed the interpolation range.

Finally, confirm the settings by clicking *OK* and leave the *Block Properties* dialog box.

To obtain an initial S-Parameter result, an S-Parameter simulation task needs to be defined. Choose *Home: Simulation ⇨ New Task* 🗹 to open the *Select Simulation Task* dialog box, and select *S-Parameters* there.
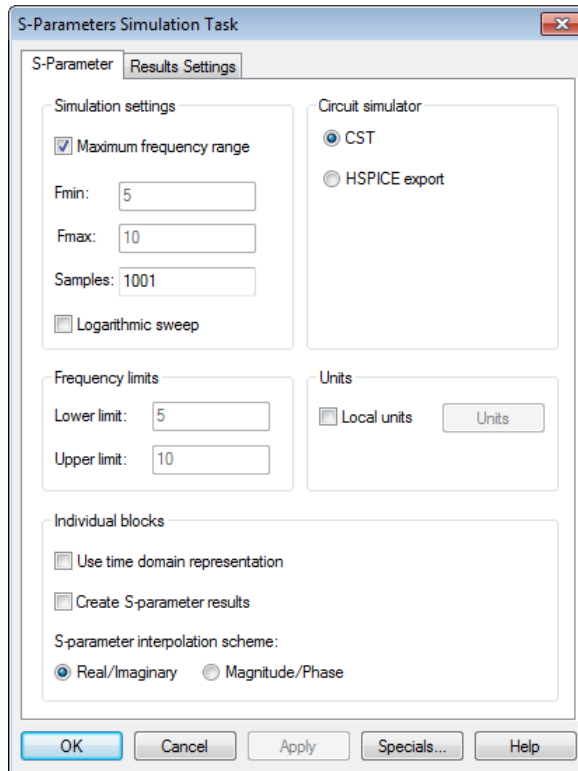
Since frequency ranges are specified for CST MICROWAVE STUDIO projects, the blocks' valid frequency ranges are limited as well. Therefore, the *Maximum frequency range* option can be chosen inside the *Simulation Settings* frame. Keep all default settings, and click *OK* to add the simulation task to the project.

Update the results now by choosing *Home: Simulation ⇨ Update*. For each transformer block three results for different supporting points in the parameter space are needed. These results will then be used to interpolate the result for the actual parameter sets. However, the transformer blocks' parameter values are identical. Consequently, only three 3D calculations are performed for both of them. The remaining ones are read from the result cache.

Look at the S-Parameter results of the circuit now by selecting the corresponding item in the navigation tree and changing the *Plot Type* to *dB* (*1DPlot: Plot Type ⇨ dB*):
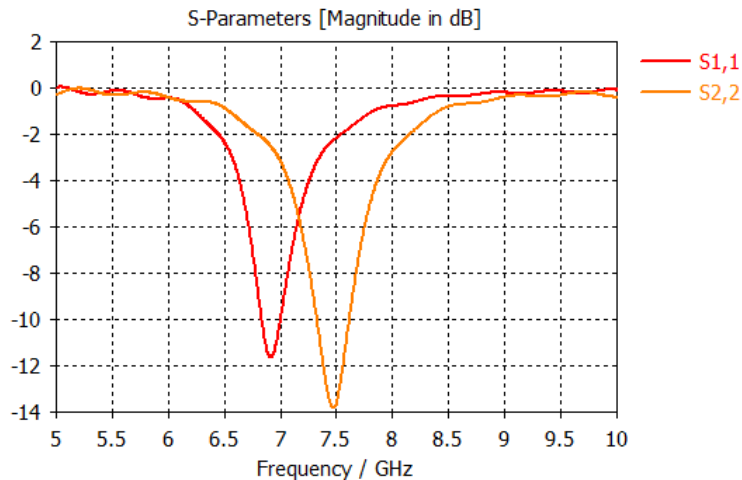
Obviously, this initial result is quite good. The relatively low reflections indicate a reasonable match. However, let us try to obtain even better results by optimization.
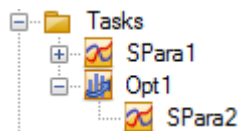
Optimization

The goal of the optimization will be to improve the matching of the two antennas at their resonance frequencies:

<div align="center">
Minimize S1,1 (in dB) for f = 7 GHz and
Minimize S2,2 (in dB) for f = 7.5 GHz.
</div>

How to set up an optimizer task has already been presented in detail during the 'Quick Tour'. Therefore, the following list just briefly sums up how to set up the optimizer task:

□   Create the optimizer task using *Home: Simulation ⇨ New Task*

□   Duplicate the already existing S-Parameter task and move it into the optimizer task

After the task creation, the task structure should look like this:

To properly set up the optimizer, open the optimizer properties dialog box by double clicking on *NT: Tasks ⇨ Opt1.* On this page all important settings for the optimization can be made. The most important one is the choice for an appropriate optimization algorithm. In our case we will choose the *Trust Region Framework*. For more information about this algorithm please have a look at the online help. The page also lists all parameters that will be taken into account during the optimization process. For each parameter the range in which it is allowed to vary and, depending on the optimizer algorithm, a number of samples can be set.
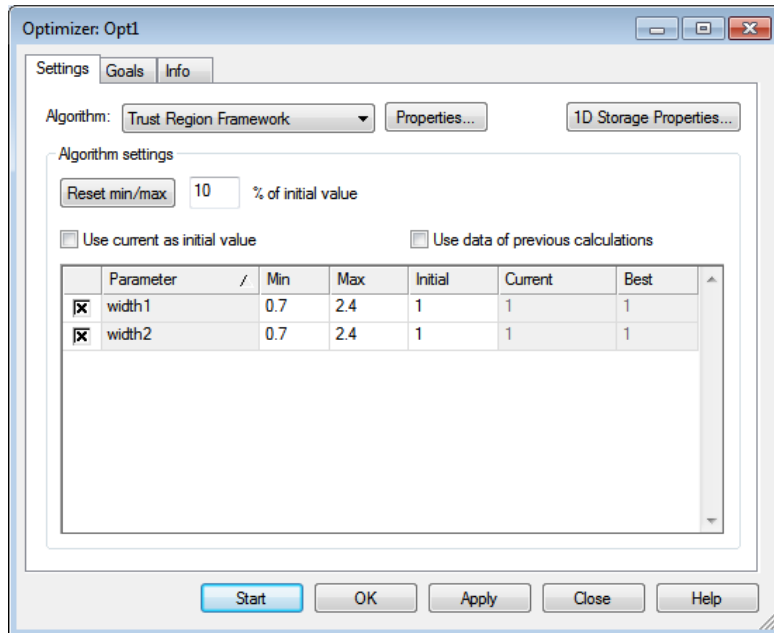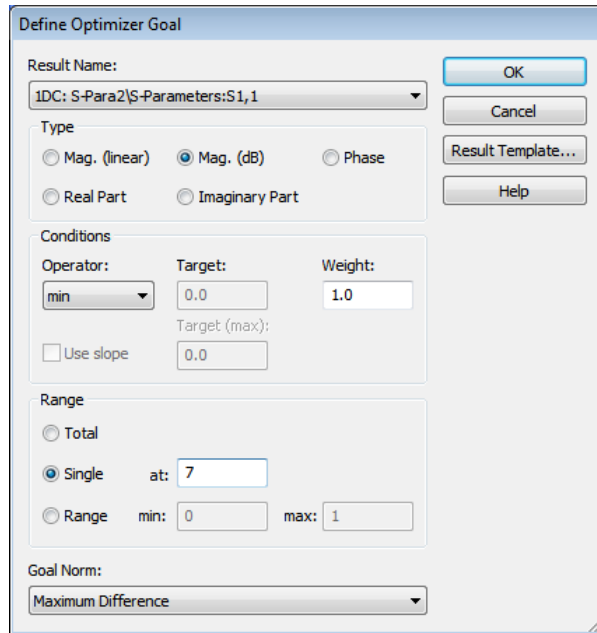
The global parameters "width1" and "width2" are assigned to the properties w of the transformer blocks. The parameter range 0.7 ≤ w ≤ 2.4 should be set for these properties, which are the same as the parameter ranges for the interpolation settings of the transformer blocks. Please disable the *Use current as initial value* to be able to repeat the simulation with the same initial conditions.

Setting up the goals is straightforward. Switch to the *Goals* tab and press the *Add new goal* button to define the first goal.

We want to minimize S1,1 at 7 Ghz. This is reflected by the following dialog settings:

- ☐ Select *1DC: SPara2\S-Parameters:S1,1* as *Result Name*
- ☐ Choose *Mag. (dB)*
- ☐ Specify *min* as operator
- ☐ Change the range to *Single* at 7
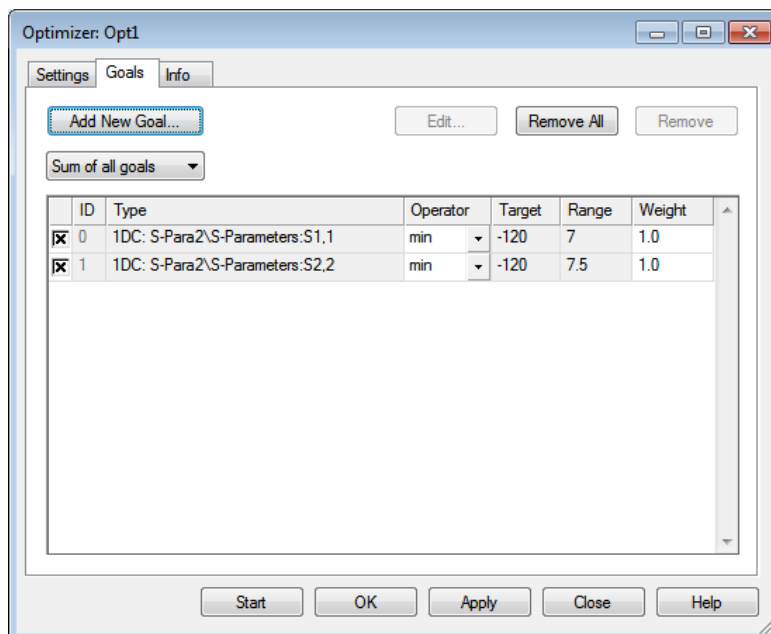
Having done that please create a second goal for S2,2 (in dB) at 7.5 Ghz. The *Goals* page now displays a list of both defined goals:
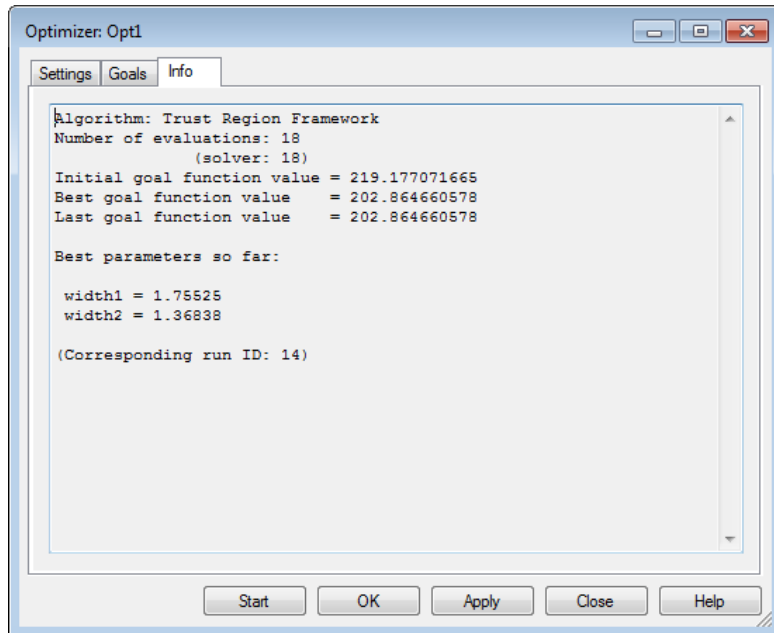

The optimization setup is complete now. Start the optimization by clicking the *Start* button. The *Info* page displays information about the goal values and the optimized parameters. After the optimization is complete, a message appears in the *Info* tab as follows:

As the difference of the first and the best goal values indicates, the optimization was successful. The following picture shows that both, S1,1 and S2,2 could be improved at the desired frequencies:
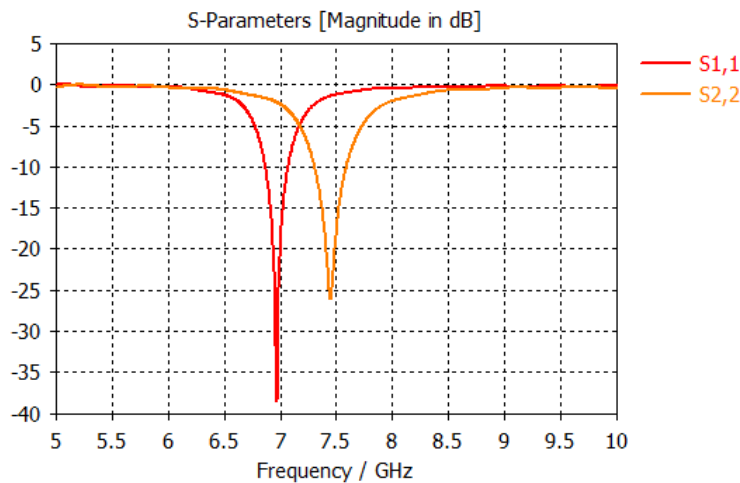
If you want to check whether or not the interpolation worked sufficiently accurate, you may generally disable interpolation (*Post Processing: Manage Resutls ⇨ Result Cache ⇨ Use Interpolation*) and update the S-Parameter task again by selecting *Update* from the S-Parameter task's context menu. In this case the two transformer blocks need to start a 3D simulation since the optimized parameter values are not identical to a supporting value parameter set and therefore the cache has no results for the optimized parameter set right now.
In this example, the interpolated and the simulated results agree very well.

Antenna Calculation

If your model contains a CST MICROWAVE STUDIO schematic block, a CST MICROWAVE STUDIO block or a CST MICROWAVE STUDIO file block, CST DESIGN STUDIO allows you to calculate field values as a result of the excitation coming from the network on the schematic.

Let us now perform a final antenna calculation with the optimized parameters of our CST DS model. Therefore, an AC-task needs to be created. Once again, go to *Home: Simulation ⇨ New Task* 🔳 and create a new *AC, Combine results* simulation task. In the upcoming dialog box, change to the *Combine Results* page where all settings for farfield calculations from within CST DS can be accessed.

By default, the *Combine results* calculation is disabled. Select the *Combine results* option to switch it on. Moreover, the block describing the antenna needs to be selected in the *Block* selector box. Choose the block's name, and make sure that the correct preview image is displayed next to it.
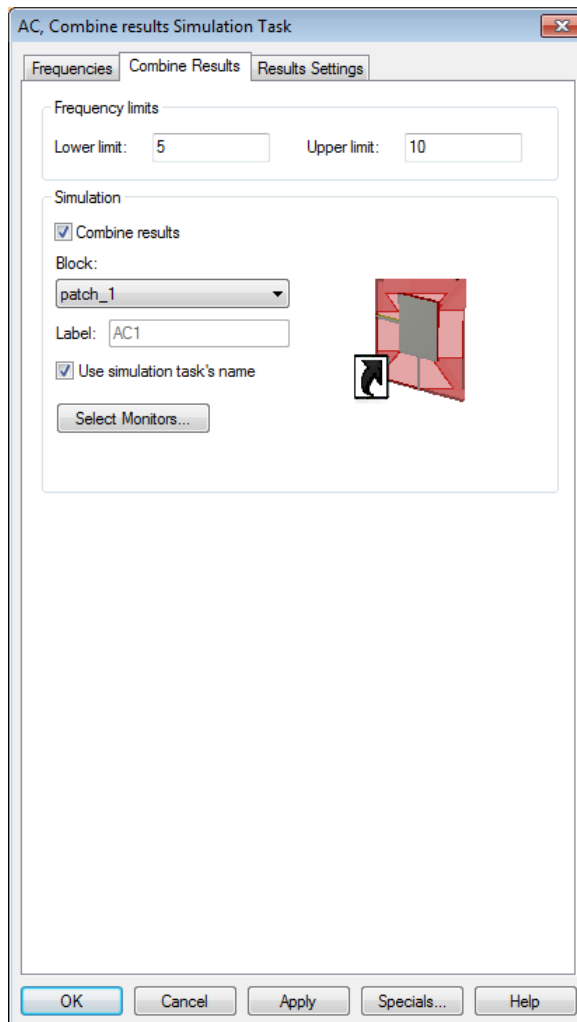
Right now, no excitations have yet been defined for the *AC, Combine results* simulation task. In the *Frequencies* tab, specify a *Signal* source type of amplitude *1* for port No. 1 and a for port No. 2 a S*ignal* source type of amplitude *0*.
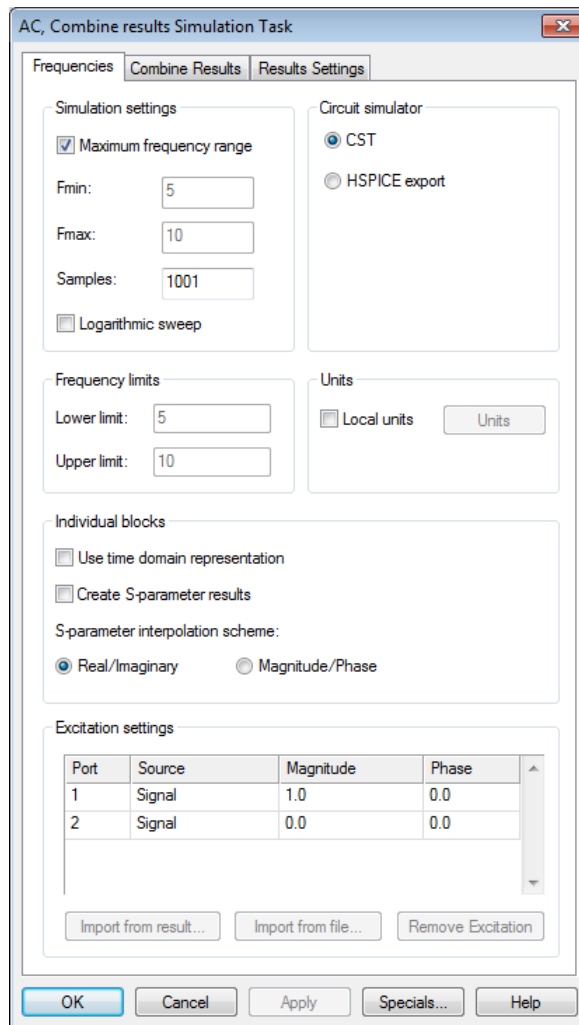
The two monitors at 7 GHz and 7.5 GHz have been specified for the antenna's CST MWS project. Because the task's frequency range is wide enough, the driven farfield results will be computed for both frequencies. Press *OK* to accept all settings and to close the dialog box.

Now, select the new AC-task tree item (*NT: Tasks ⇨ AC1*) and select *Update* in its context menu to start the simulation. If the interpolation is disabled because you have checked the S-Parameters as described in the last section, you will see the following output in the message window:

If you did not re-simulate the S-Parameters and the interpolation is still enabled, the output will show this text:


Note: In both cases, no 3D simulation is performed because the transformer's results and the antenna's results are already available. Before the task is successfully completed, the farfields are calculated for the defined excitation.

The results of the farfield calculations are now available in the antenna block's CST MWS project. You can access them by selecting the block and choosing *Edit* from its context menu that will open the project.

The navigation tree's *Farfields* folder contains additional entries labeled with [AC1]. These items contain the farfields for the excitation as defined in the simulation task.
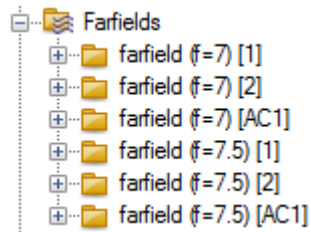
For instance, selecting the item *NT: Farfields ⇨ farfield (f=7) [AC1]* leads to the directivity's magnitude plot in dBi:
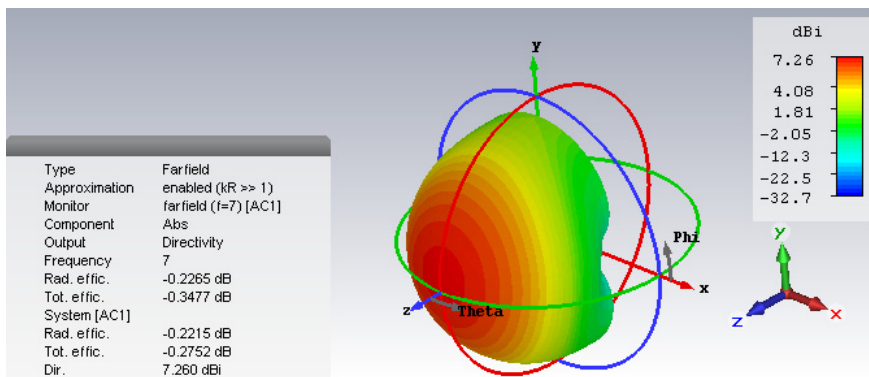
Chapter 7 – Finding Further Information

After carefully reading this manual, you will already have a basic understanding of how to use CST DESIGN STUDIO efficiently for your own problems. However, when you are creating your own designs, many questions will arise. In this chapter we will give you a quick overview of the available documentation.

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 ⇨ CST STUDIO SUITE – 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 MICROWAVE STUDIO Online Documentation.

Examples

The installation directory of CST STUDIO SUITE contains an examples subdirectory containing a couple of typical application examples. A quick overview of the existing examples can be obtained by following the *Examples Overview* link on the online help system's start page. These examples may contain helpful hints that can be transferred to your particular application.

Technical Support

After you have taken your first steps solving your own applications within CST DESIGN STUDIO, please use the *File: Project ⇨ Archive As* function to create an archive containing all relevant files. This archive should then be sent to the technical support team. Even if you have successfully obtained a solution, the problem specification might still be improved in order to get even better results within shorter calculation times.

The preferred option to contact technical support is to submit a support ticket. You can create a new ticket or manage existing tickets from within the support area on our homepage or by selecting *File: Help ⇨ Support Tickets*.

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

command. Please note that the online help system's search function also allows searching in the online content as well.
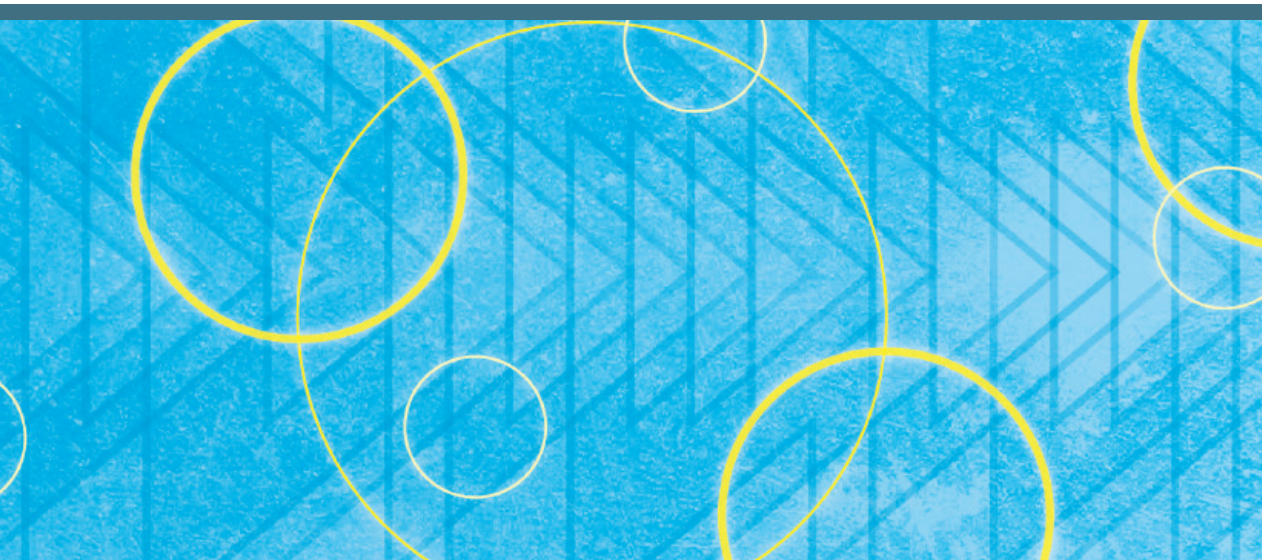
Macro Language Documentation

More information concerning the built-in macro language can be accessed through the VBA overview page of the online help system. The macro language's documentation consists of four parts:

- [ ] An overview and a general description of the macro language.
- [ ] A description of all CST DESIGN STUDIO specific macro language extensions.
- [ ] A syntax reference of the Visual Basic for Applications compatible macro language.
- [ ] Some documented macro examples.

History of Changes

An overview of all new main features of the release can be obtained by selecting the *Spotlight CST STUDIO SUITE 2015* page from the online help system (*File: Help ⇨ CST STUDIO SUITE – Help* 🔵). Also the detailed *History of Changes* can be accessed through the *Spotlight* page in the Online Help. The *Changes in the Service Packs* Page at the same location provides in addition smaller changes released during intermediate service packs. 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.