CST CABLE STUDIO

Workflow & Solver Overview

CST STUDIO SUITE® 2015

Copyright

Trademarks

CST – Computer Simulation Technology AG
www.cst.com

Contents

Chapter 1 — Introduction

Welcome

Welcome to CST CABLE STUDIO, the powerful and easy-to-use package for analyzing conducted transmission, EMI (Electromagnetic Interference) and EMS (Electromagnetic Susceptibility) on complex cable structures. This program combines transmission line theory, circuit simulation and 3D "full-wave" calculation methods in a convenient and sophisticated way, which makes it highly suitable for simulating cables inside electrically large systems.

CST CABLE STUDIO is embedded into the design environment of CST STUDIO SUITE, which is explained in the *Getting Started* manual. The following explanations assume that you have already installed the software and familiarized yourself with the basic concepts of the user interface.

How to Get Started Quickly

We recommend that you proceed as follows:

- Read the *CST STUDIO SUITE Getting Started* manual.
- Work through this document carefully. It should provide you with all the basic information necessary to understand the advanced documentation found in the online help.
- Please look at the 'Examples' folder in the installation directory. The different application examples will give you a good impression of what can be accomplished with the software. Please note that these examples are designed to give you basic insight into a particular application domain.
- Start with your first own example. Create your first CABLE STUDIO model and simulation. Choose a reasonably small and simple harness that will allow you to quickly become familiar with the software.

What is CST CABLE STUDIO?

CST CABLE STUDIO is an electromagnetic simulation tool specially designed for fast and accurate simulation of real-world cables inside electrically large systems by combining transmission line theory, circuit simulation and 3D full wave calculation methods. It closes the gap between pure 2D cross-section analysis and pure 3D full wave analysis.

CST CABLE STUDIO offers a user interface that makes it easy to define a complex cable harness. The 3D topology can be defined either from scratch or by loading an existing harness via a NASTRAN or STEP AP212-KBL import filter. Separate dialog boxes enable the definition of four basic types of cables: single wires, ribbon cables, twisted pairs and shielded cables (compact or braided shields). Any combination of these basic cable types can be defined and stored in a user defined library. A bundle dialog box allows the convenient definition of exact or random cable cross-sections. The transfer impedance of a shielded cable can be either directly defined or calculated by using the built-in transfer impedance calculator, which extracts the impedance from the geometric characteristics of the shield. In addition, it is possible to load and assign measured transfer impedance curves.

CST CABLE STUDIO generates equivalent circuits from the cable harness based on classical transmission line theory. It automatically meshes the cable harness along its length and calculates the transmission line parameters on these segments. Skin effect and dielectric loss are modeled in both frequency and time domain simulations. The equivalent circuits can be exported in several SPICE formats.

CST CABLE STUDIO uses the powerful 3D solid modeling front end of CST MICROWAVE STUDIO to set-up or import arbitrary metallic 3D shapes, ranging from simple ground planes to complex chassis structures. Moreover, the powerful 3D full wave solvers from CST MICROWAVE STUDIO are used to calculate the electromagnetic fields in the environment surrounding the cable.

CST CABLE STUDIO uses CST DESIGN STUDIO's easy-to-use schematic to define passive and active devices at cable terminations. The powerful built-in network simulator in CST DESIGN STUDIO enables the simulation of a whole system consisting of the equivalent circuit of the cable harness and its terminations.

CST CABLE STUDIO controls the data exchange between the circuit simulation engine and the various 3D EM solvers for currents (on the cable harness) and electromagnetic fields (around the cable harness). This enables simulating both the effects of fields coupling into the cables and fields radiating from the cables.

Applications

- Transmission and crosstalk simulations of extended cable structures in time and frequency domains
- EMI: analysis of radiated electromagnetic fields from complex cables lying inside open metallic structures
- EMS: analysis of coupled electromagnetic fields into complex cables lying inside open metallic structures

CST CABLE STUDIO Key Features

An overview of the main features of CST CABLE STUDIO is provided in the following list. Please note that availability of some of the listed features may depend on the individual license configuration. Please contact your local sales office for details.

For the circuit simulator only some selected key features are listed below. Additional information can be found in the *CST DESIGN STUDIO manual* and *Online Help*.
For the 3D solid modeling front end and the 3D full wave simulation only some selected key features are listed below. A full list can be found in the *CST MICROWAVE STUDIO Workflow and Solver Overview* manual.

General

- Native graphical user interface based on Windows operating systems.
- Tight interface to CST DESIGN STUDIO and CST MICROWAVE STUDIO enabling cable modeling, circuit simulation and 3D full wave analysis in one environment.
- Transmission line modeling method for fast and accurate simulation of TEM propagation modes inside complex cable structures.
- Hybrid method combining transmission line analysis with 3D “full-wave” analysis.

Harness Structure Modeling

- Easy definition of complex harness topology.
- Import of harness via NASTRAN and STEP AP212-KBL.
- Interactive cable editing dialog boxes for all relevant types of cables.
- Parameterization of cable position

Harness Electric Modeling

- Automatic meshing and extraction of 2D transmission line parameters.
- Modelling of all relevant cable types in any combination (single wire, ribbon cables, twisted pair and shielded cables).
- Consideration of skin and proximity effects as well as dielectric loss in time and frequency domains.
- Consideration of transfer impedance for braided shields Export of equivalent SPICE circuits.

Circuit Simulator

- Schematic editor enables the easy definition of passive and active devices on the cable's equivalent circuit.
- Fast circuit simulation in time and frequency domains.
- Import of SPICE sub-circuits (Berkley SPICE syntax).
- Support of IBIS models.
- Import and Export of S-Parameter data via TOUCHSTONE file format.
- Parameterization of termination circuitry and parameter sweep.

3D Full Wave Simulator

- Automatic transfer of impressed common mode currents on cable bundles from circuit simulator to the 3D "full-wave" simulator.
- Automatic transfer of induced voltages on cable bundles to circuit simulator.
- Advanced solid modelling of the definition of scattering or antenna structures.
- Import of 3D CAD data by SAT, Autodesk Inventor®, IGES, VDA-FS, STEP, ProE®, CATIA 4®, CATIA 5®, CoventorWare®, Mecadtron®, NASTRAN or STL files for the definition of scatter and antenna structures.
- Plane wave excitation (linear, circular, elliptical polarization).
- Ideal voltage and current sources for antenna excitation.
- Accurate and efficient time domain solvers, based on the Finite Integration Technique (FIT) or on the Transmission-Line Matrix (TLM) method.
- Fully automatic creation of hexahedral grids in combination with the Perfect Boundary Approximation (PBA), Thin Sheet Technique (TST) or Octree-based meshing.
- Calculation of various electromagnetic quantities such as electric fields, magnetic fields, surface currents and voltages.

About This Manual

This manual is primarily designed to enable a quick start on the modeling capabilities of CST CABLE 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 one to learn the software efficiently with help of the online documentation.

To learn more about the circuit simulator, please refer to the *CST DESIGN STUDIO* manual.

To learn more about the 3D full wave simulator, please refer to the *CST MICROWAVE STUDIO* manual.

The next chapter, *Overview*, is dedicated to explaining the general concepts behind CST CABLE STUDIO and to show the most important objects and related dialog boxes. Chapter *Examples* will guide you through three important examples which provide a good overview of the capabilities. We strongly recommend studying both chapters 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 parentheses after the command. Example: *View: Visibility ⇨ Wire Frame (Ctrl+W)*
- 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: 1D Results ⇨ Port Signals ⇨ i1*

Your Feedback

We are constantly striving to improve the quality of our software documentation. If you have any comments regarding the documentation, please send them to info@cst.com.

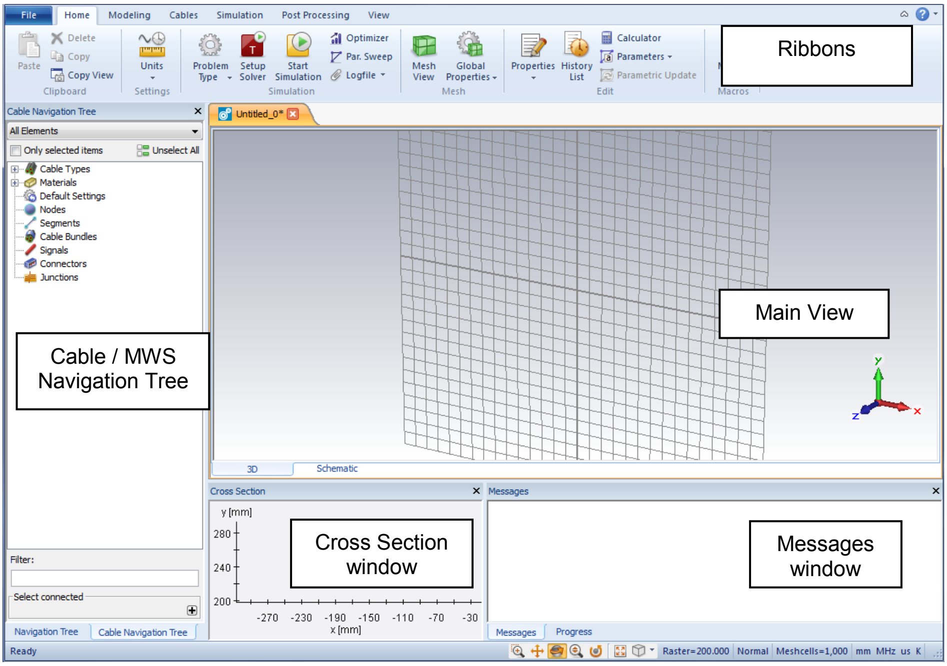
Chapter 2 — Overview

CST CABLE STUDIO is designed for ease of use. However, to get started quickly you will need to know the basic concepts behind it. The main purpose of this chapter is to provide an overview of the most important objects and dialog boxes.

User Interface

Launch CST STUDIO SUITE from the Start menu or by clicking on the desktop icon. In the *Modules* list click on *CST CABLE STUDIO*. A new CST CABLE STUDIO project is opened with an empty Main View.

Main Frame



The user interface is divided into five sub-windows:

- The **Main View** allows the 3D visualization of the harness and its surrounding metallic components.
- The **Cross Section** window shows the 2D visualization of cable cross sections.
- The **Cable/MWS Navigation Tree** frame enables switching between the **Cable Navigation** and the **MWS Navigation Tree.** The *Cable Navigation Tree* allows access to all objects necessary to define a complete cable assembly in 3D. When selecting an item it will be displayed in the Main View, Cross Section View or in both depending on the object's characteristics. The *MWS Navigation Tree* allows access to all MWS related objects, thereby allowing full access to solid modeling

and 3D full wave simulation technology. When selecting an item it will be displayed in the *Main View*.

- The **Messages** window shows general information, solver progress, warnings and errors during project set-up or simulation.

Interface to CST DESIGN STUDIO

Below the Main View there are two different tabs:

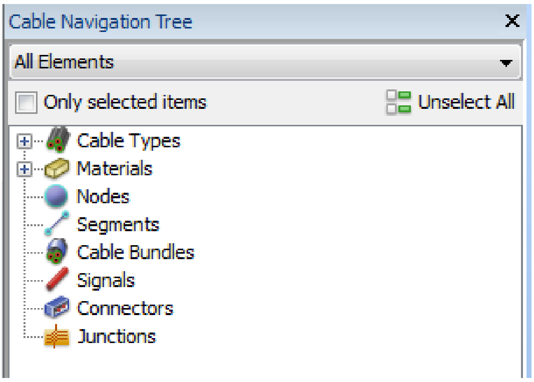
Initially the **3D** view is active. Selecting the **Schematic** tab changes the view to CST DESIGN STUDIO. This provides access to the schematic editor and the circuit simulator. The following list gives an overview on the meaning and usage of the two different tabs:

- The **3D** tab presents all objects and dialog boxes necessary to define and edit cable bundles inside a 3D metallic environment. It includes appropriate solver technology to generate equivalent circuits which are passed to the Schematic tab.
  It further enables the hybrid methods for radiation and irradiation by exchanging the common mode current and voltages of a cable between the circuit simulator and the 3D transient solvers.

- The **Schematic** tab is used to define and edit loads on the equivalent circuit of the cable harness with the help of a schematic editor. It further enables the circuit simulation of the whole system in time and frequency domains, while maintaining a tight interface with the 3D transient solvers to easily exchange impressed currents and voltages.

How to Define a Cable Assembly

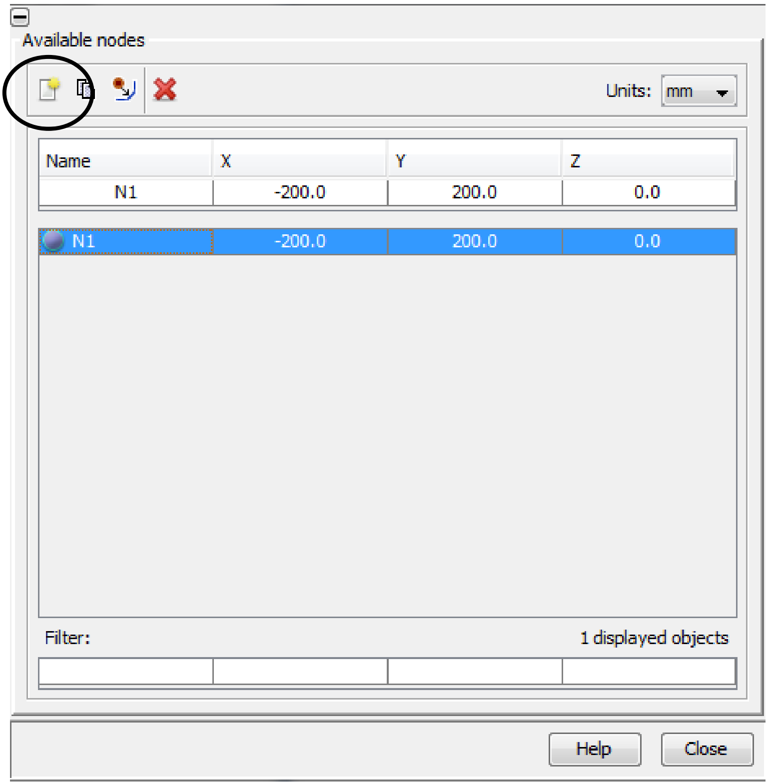
This section will explain how to set up a complete cable assembly. It is a basic procedure that we recommend reading carefully before starting with the examples in the next chapter. In order to set up a cable assembly, cable bundles where different cable types can be placed must be defined first. This can be done by using any existing curve (*Cables: Curves ⇨ Curves*) or by using the following three objects: **Nodes**, **Segments** and **Cable Bundles**, which can be seen as separate sub-folders in the *Cable Navigation Tree*.



The Trace of a Cable Bundle

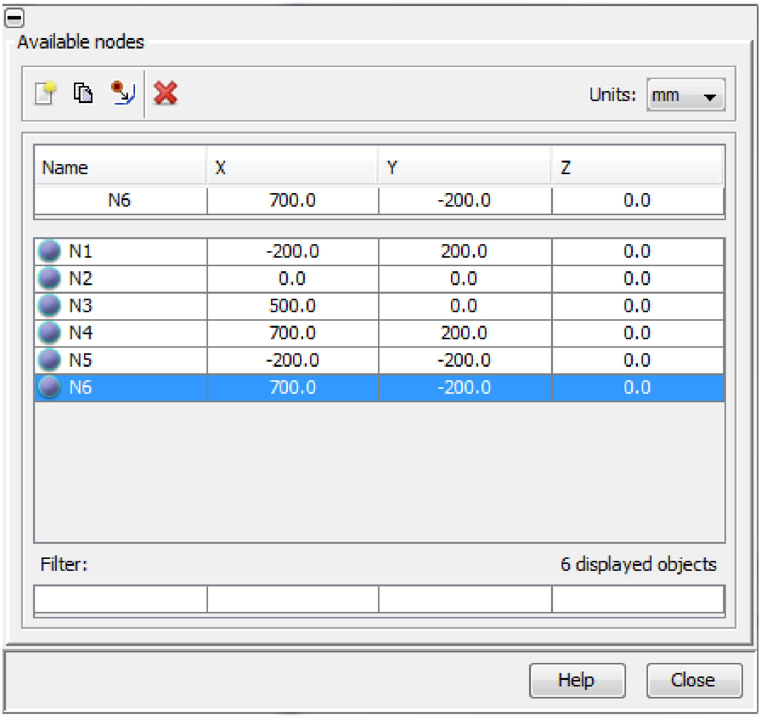
Nodes and Segments build a 3D graph, which are used for defining paths. Such paths are called *Traces* and they can be assigned to any number of different cables. All cables on a *Trace* build a *Cable Bundle,* and all those cables are automatically coupled (by the mutual inductances and capacitances between the cables). There is one restriction: a trace from a cable bundle cannot build a closed loop.

We start with the definition of six nodes which we will use later to define two separate traces. We double-click on *Cable Navigation Tree: Nodes* and get the following dialog box:

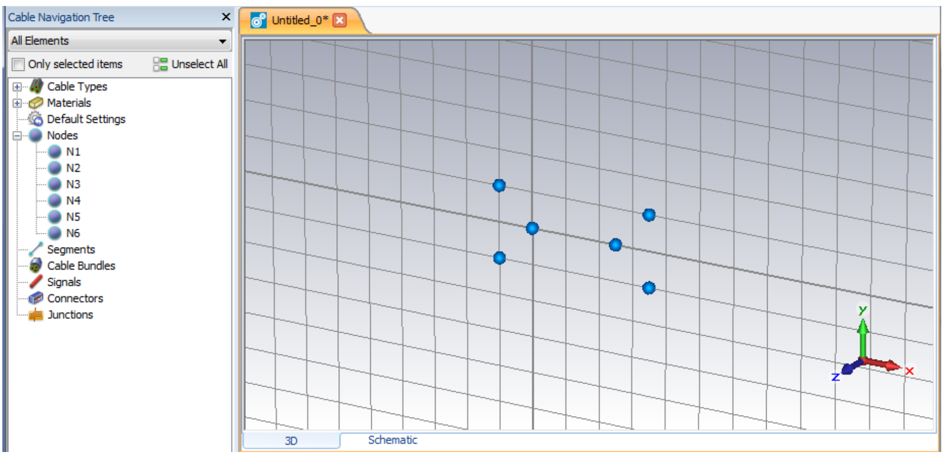


To define a first node we select the marked symbol on the top left and enter *-200.0* for x and *200.0* for y.

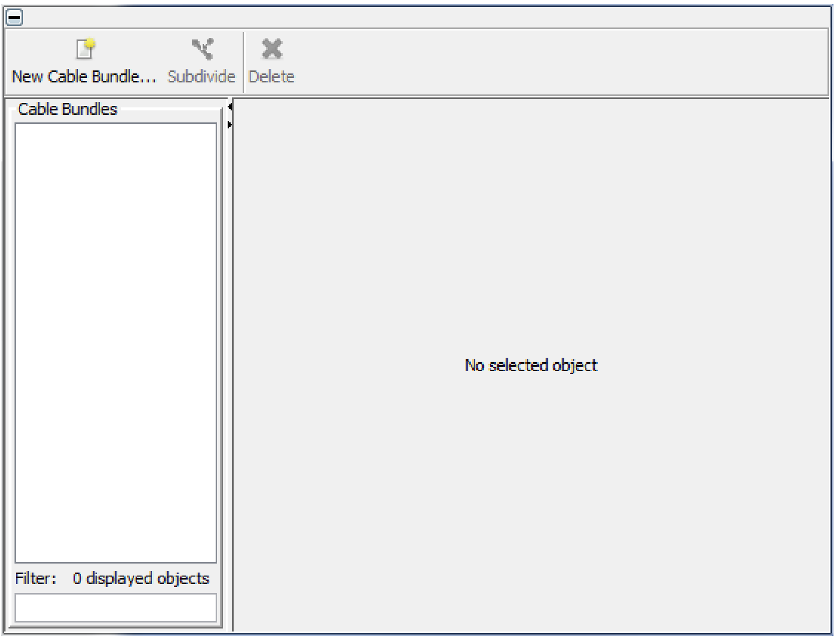
This procedure has to be repeated for five additional nodes according to the list below:



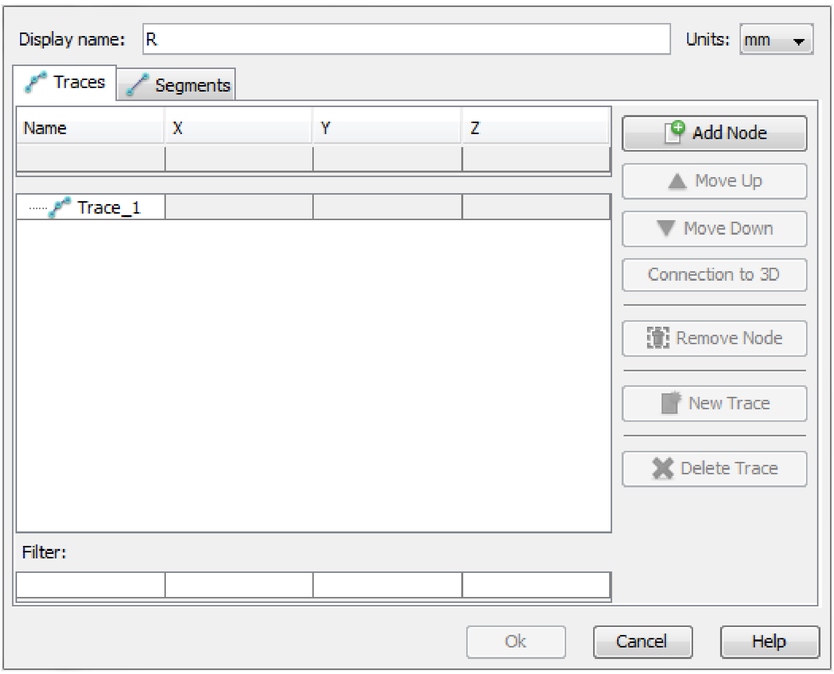
After closing the dialog box, we see six nodes appearing in the *Cable Navigation Tree* and in the *Main View*:



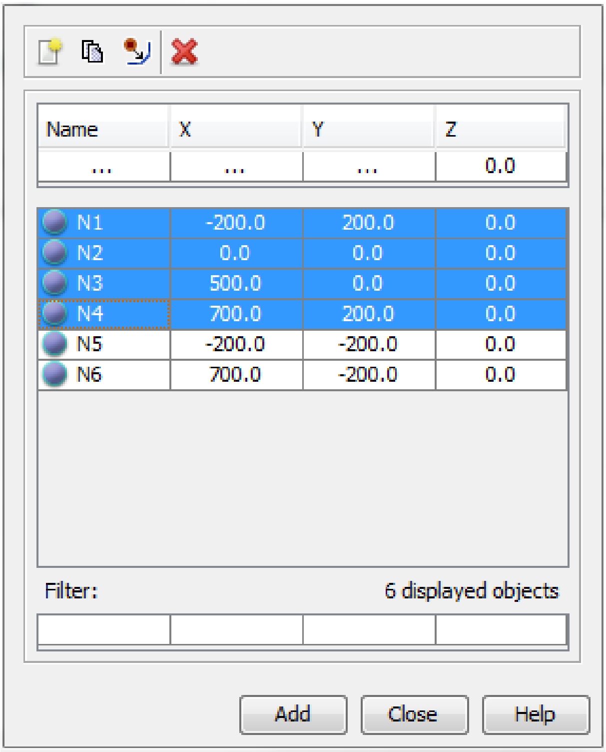
In order to define a trace along the node sequence *N1-N2-N3-N4* we press *Cables: Edit Cabling ⇨ Cable Bundles*. The following dialog box will appear:



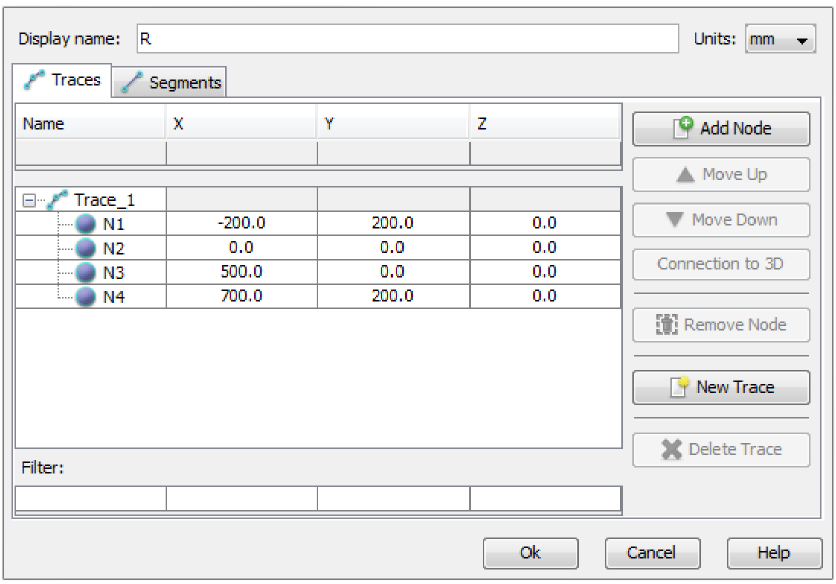
We click on the *New Cable Bundle* button on the top left side of the dialog box and get the following view:



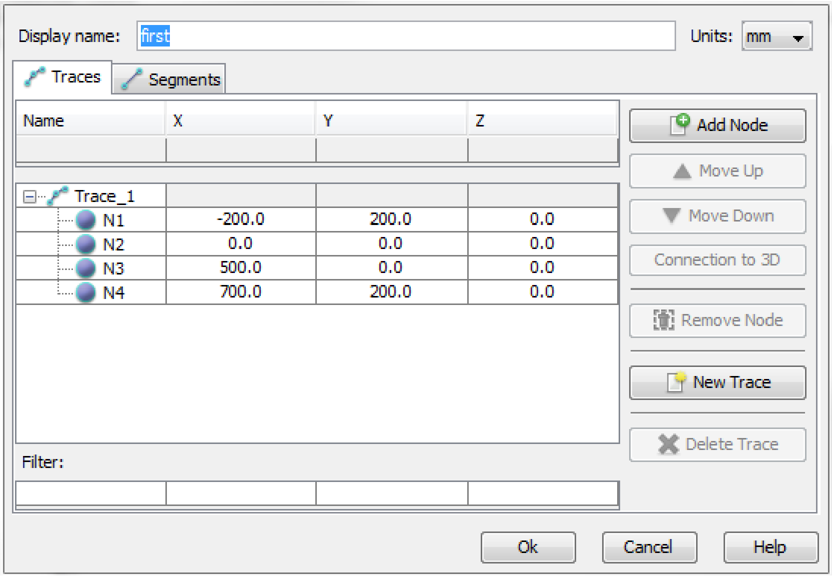
To define the trace *N1-N2-N3-N4*, we press the *Add Node* button and multi-select the first four nodes in the following dialog box:



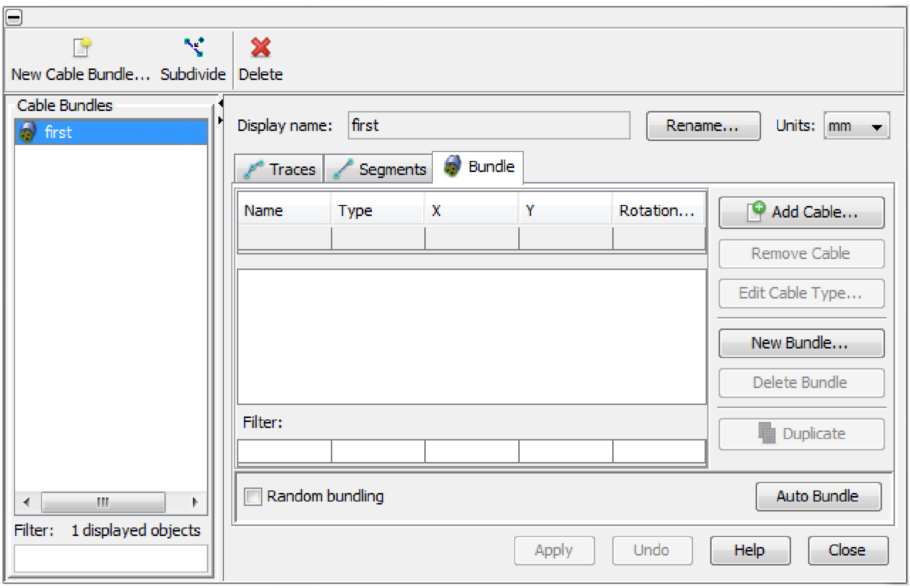
After pressing the *Add* button, the four nodes will disappear from the list of *Available Nodes*. We close this dialog box and see the first trace defined:



We name the Cable Bundle “first” in the *Display name* field as shown in the figure below and press the *Ok* button.

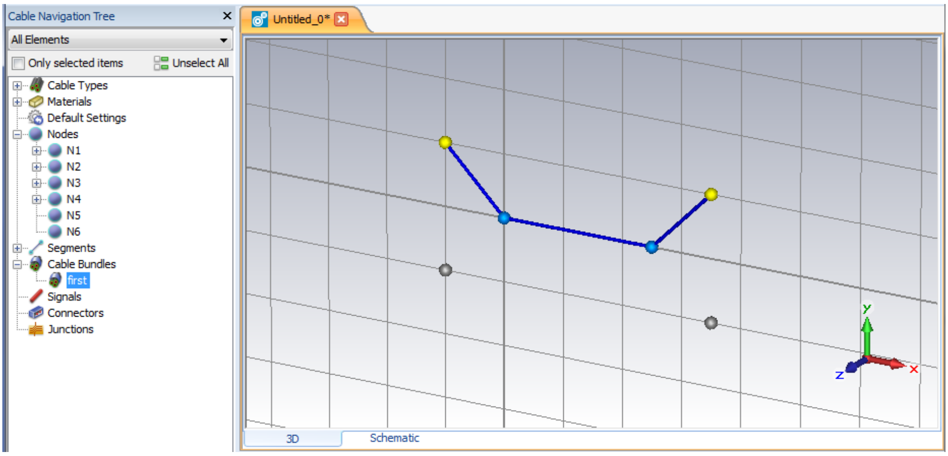


The next dialog box will be launched automatically and will allow you to start inserting cables:



This dialog box allows you to start defining the layout of the cables.

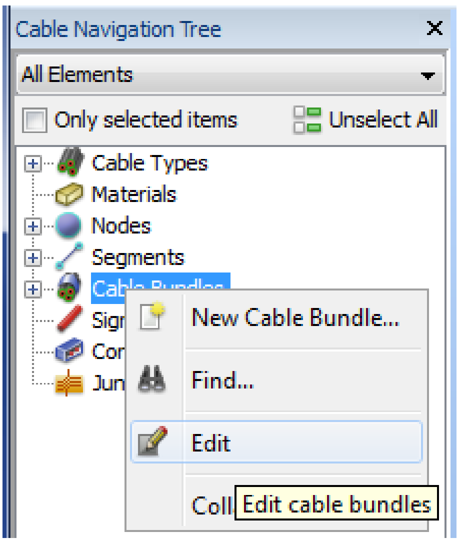
But before continuing we close the dialog box to see what we have done so far. We will see the trace of the cable bundle marked in blue color inside the *Main View*.



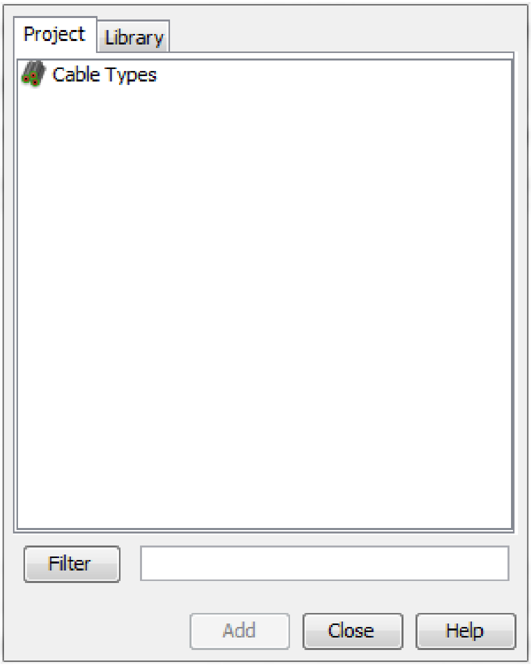
The cable bundle is also added to the corresponding folder in the *Cable Navigation Tree*.

Inserting Cables

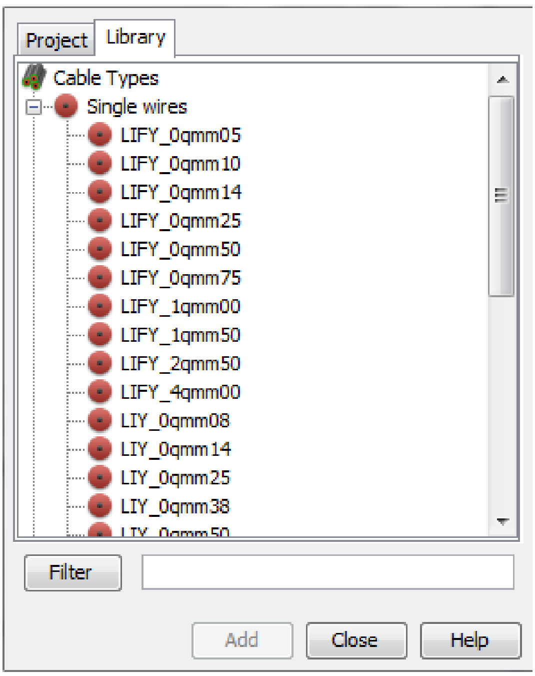
After the position of the cable bundle is defined we have to insert the cables. This can be done easily with the help of the previous dialog box. You can return to the dialog box by selecting *Cable Navigation Tree: Cable Bundles*, right mouse clicking and selecting *Edit* from the pull-down menu:



The dialog box appears again. The left column lists the only cable bundle we have defined so far. In order to place two single wires inside the bundle, we press the *Add Cable* button. The following sub-dialog box will be displayed:

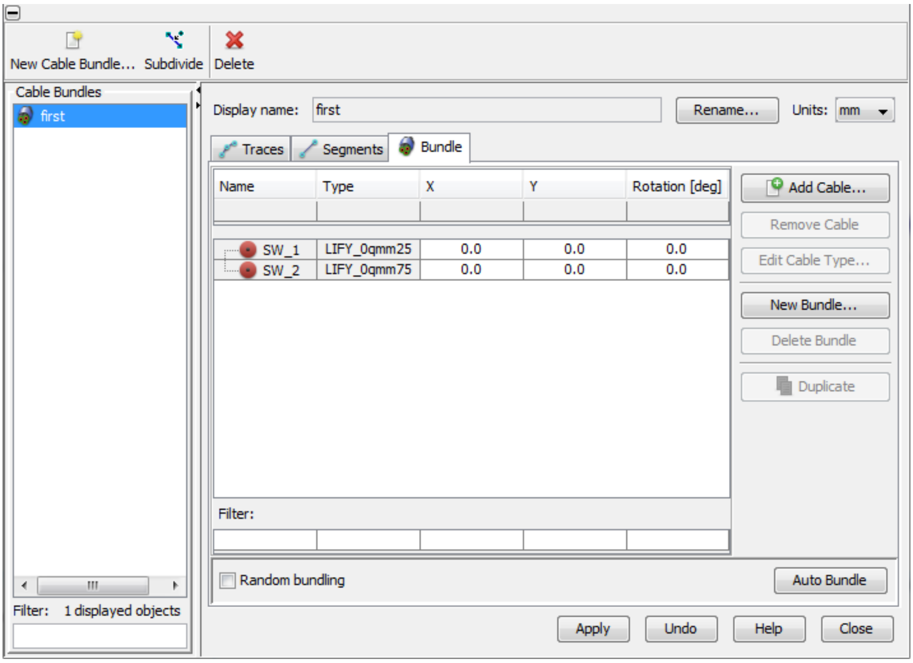


Since there is no cable type in the current project yet, we select the *Library* tab:

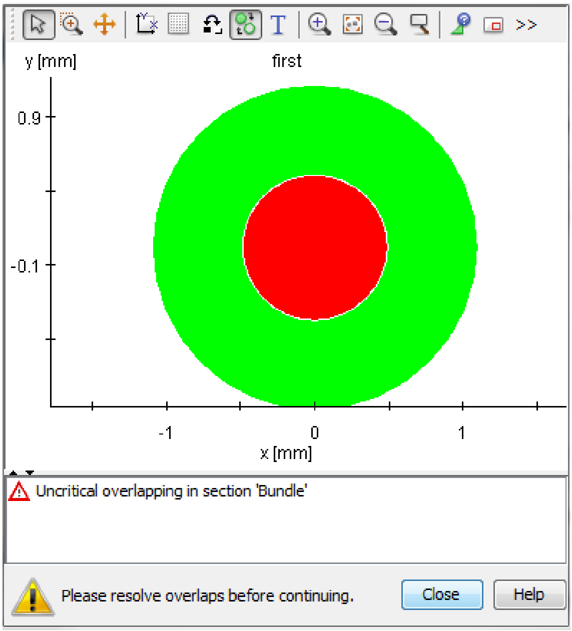


From the main class *Single wires* we select the wire type *LIFY_0qmm25* and press the *Add* button. Then, we repeat the procedure with wire type *LIFY_0qmm75* and press *Close*.

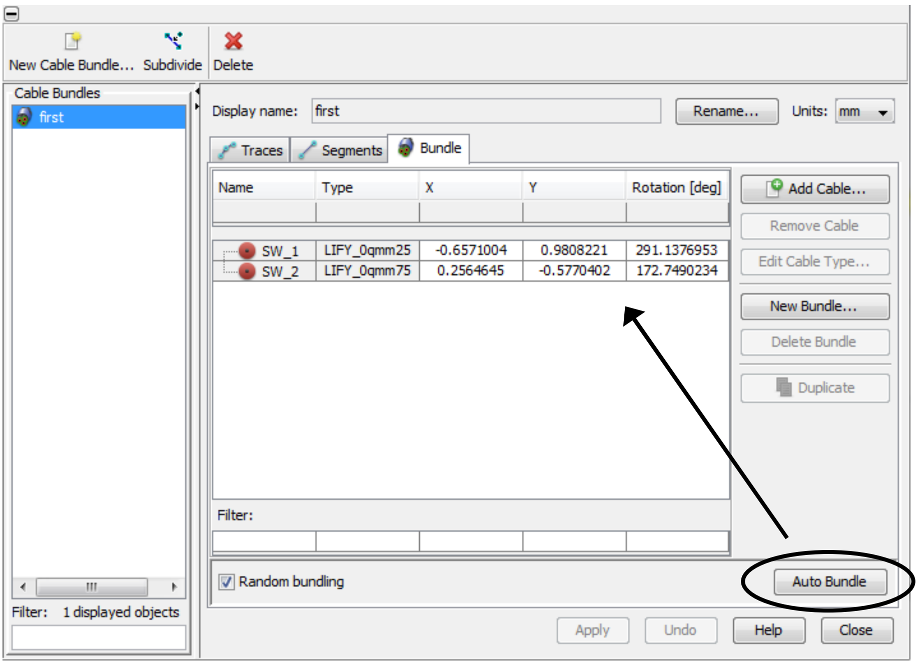
The cable bundle dialog box should look like the figure below:



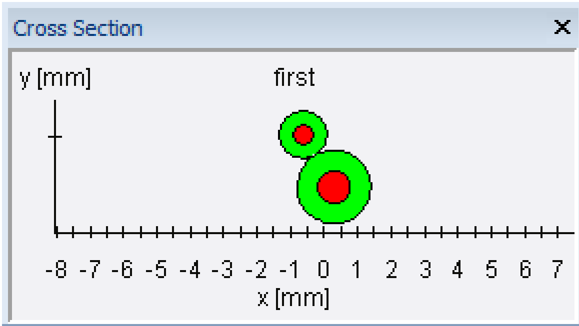
Both wires are at position *x=0, y=0, z=0*. After pressing *Apply* a warning message will appear, indicating that the cross sections of the two wires are overlapping:



In order to choose physically correct positions, there are two possibilities. One way is to enter the precise coordinates for each wire. The other way is to press the button *Auto Bundle*.

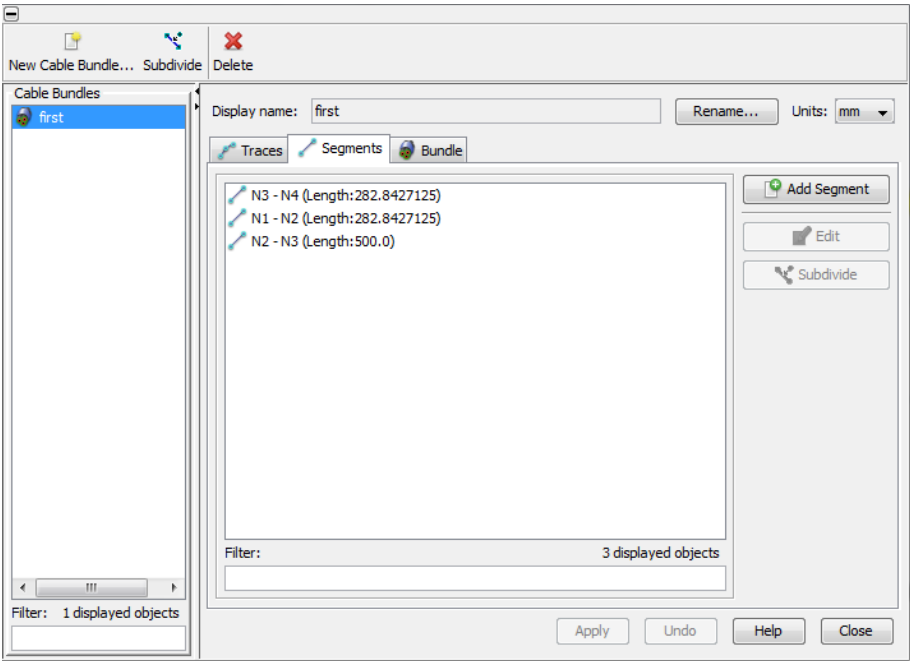


Auto Bundle generates an arbitrary configuration for a physically correct bundle which reduces the effort by the user. In many cases any proposal of a cross section is sufficient because the exact position of wires inside a bundle is not defined in many real-world configurations. After pressing *Auto Bundle* the *Cross Section View* will show a similar configuration:

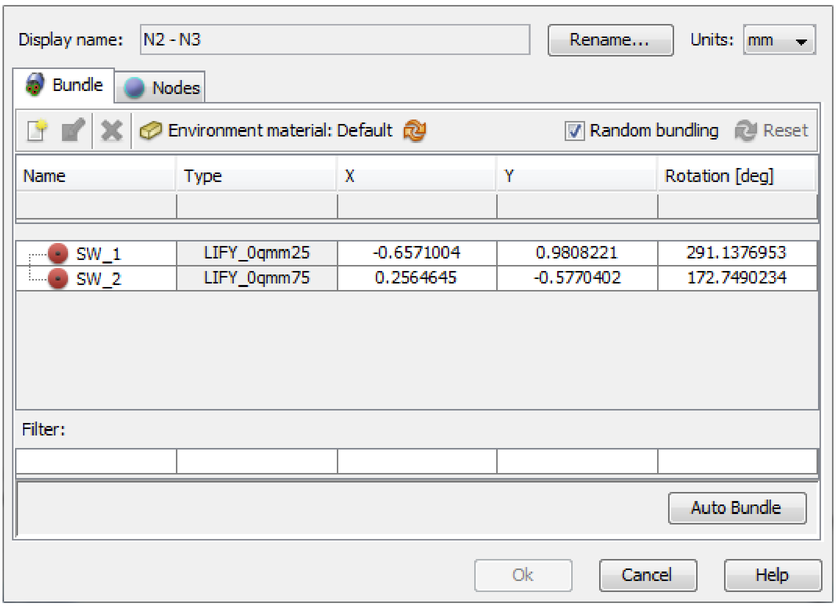


As long as any segment of the topology is only used by a single cable bundle, the representative cross section is adapted to all the segments used by that cable bundle.

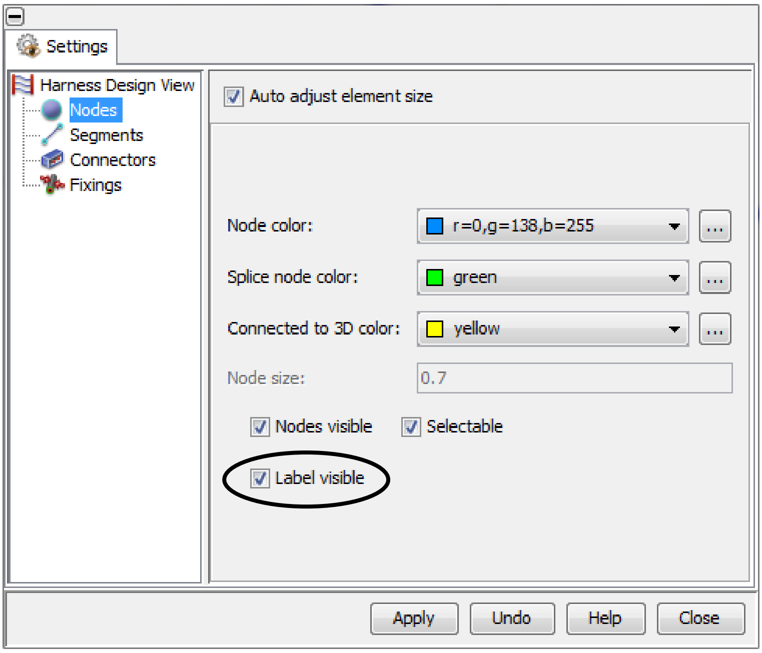
This can be seen if we select the *Segments* tab in the right column of the dialog box:



If we click on any of the three segments in the list we will see the same bundle cross section in the *Cross Section* window. When double-clicking on a segment a dialog box will appear showing the same positions:

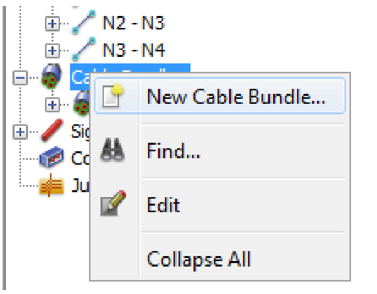


This is no longer true if there are two or more cable bundles using a common segment. For this reason we will define a second cable bundle on path *N5-N2-N3-N6*. In order to show the arrangement of the existing nodes, we select *Cables: Edit Cabling ⇨ Properties ⇨ Harness View Attributes*. The following dialog box will appear:

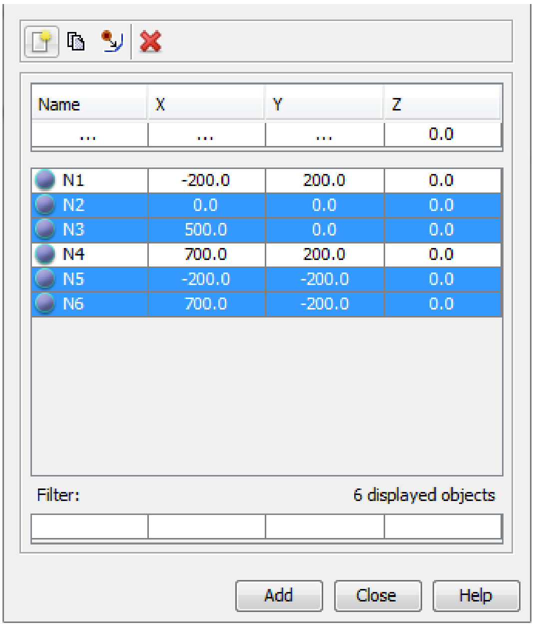


We check *Label visible*, press the *Apply* button and close the dialog box. After doing so the names of the nodes are displayed in the *Main View*.

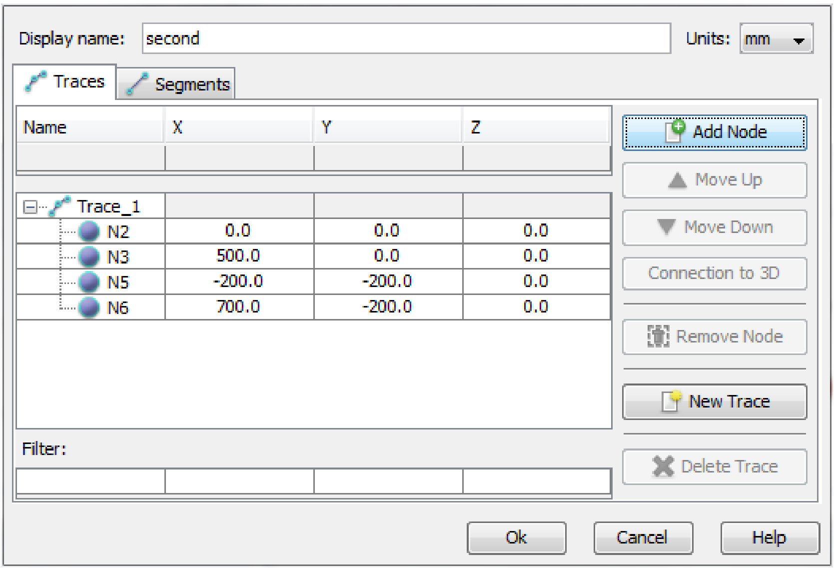
To generate the second cable bundle, we select *Cable Navigation Tree: Cable Bundles ⇨ New Cable Bundle* by using the right mouse button:



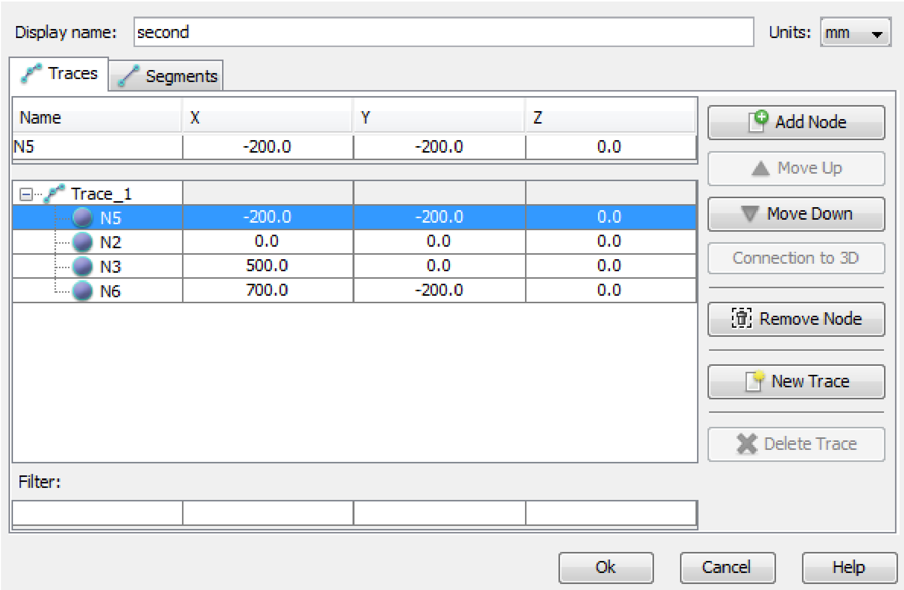
A new dialog box will appear. First we enter “second” as *Display Name* for the new cable bundle. Then, we press the *Add Node* button and select the nodes *N2*, *N3*, *N5* and *N6* as shown in the figure below:



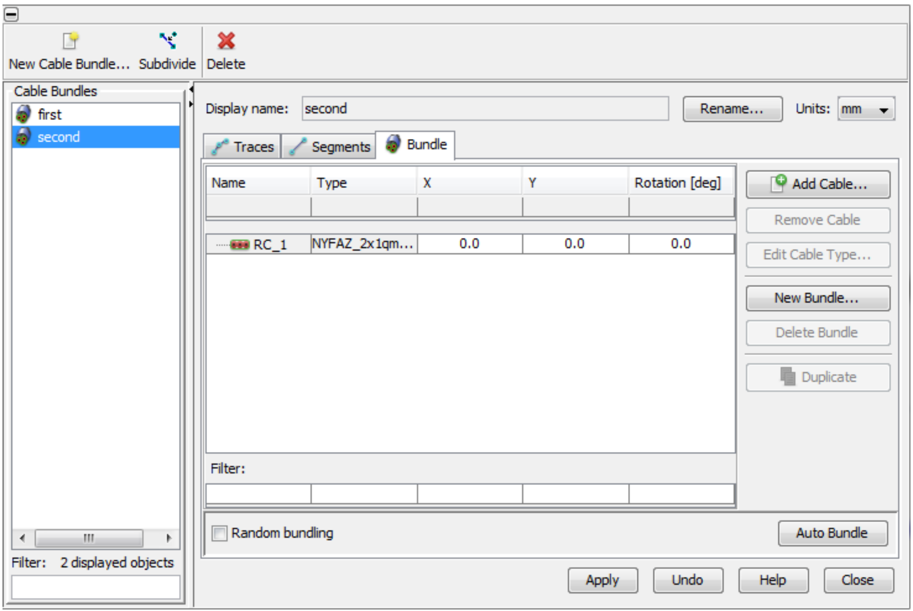
After pressing *Add*, the trace consisting of the correct nodes but incorrect sequence *N2-N3-N5-N6 will appear*:



In order to get the correct node sequence *N5-N2-N3-N6*, we have to move the node *N5* to the first position. This can be done by selecting the node and moving it up by using the *Move Up* button. In the end the trace definition should look like in the figure below:

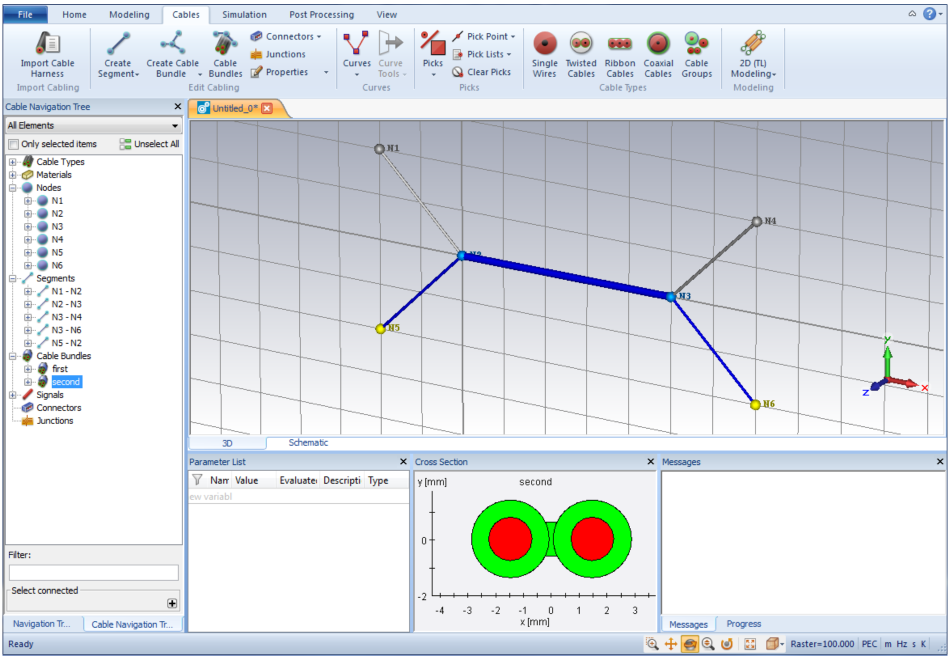


After pressing *Ok* the cable bundle dialog box allows us to insert cable. We press the *Add Cable* button, switch to the Library tab and add *NYFAZ_2x1qmm50* from the cable type group *Ribbon cables*:

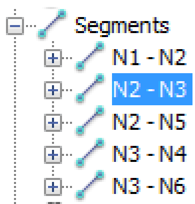


Now, we press *Apply* and close the dialog box.

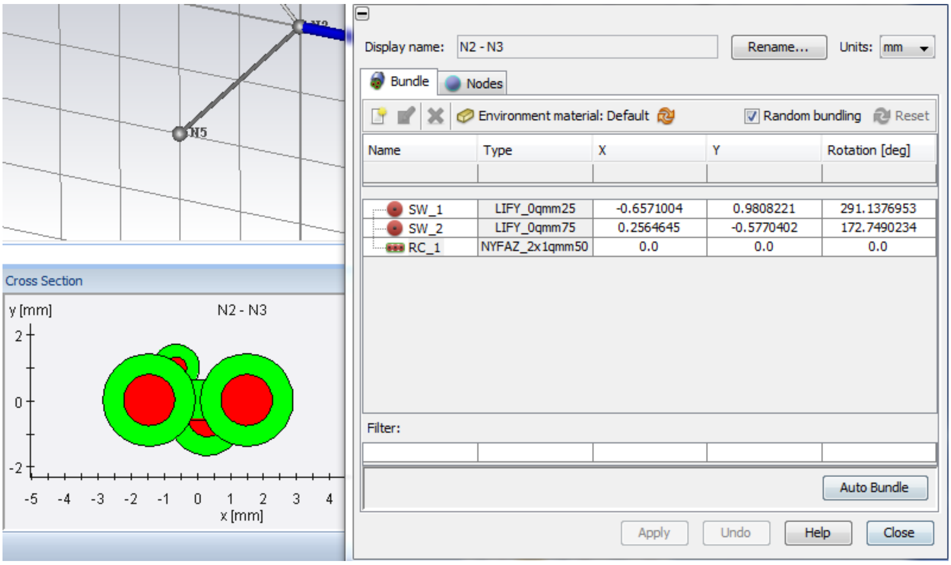
The new cable will be displayed in blue color along *N5-N2-N3-N6* inside the *Main View*. In addition, the ribbon cable can be seen in the *Cross Section* window:



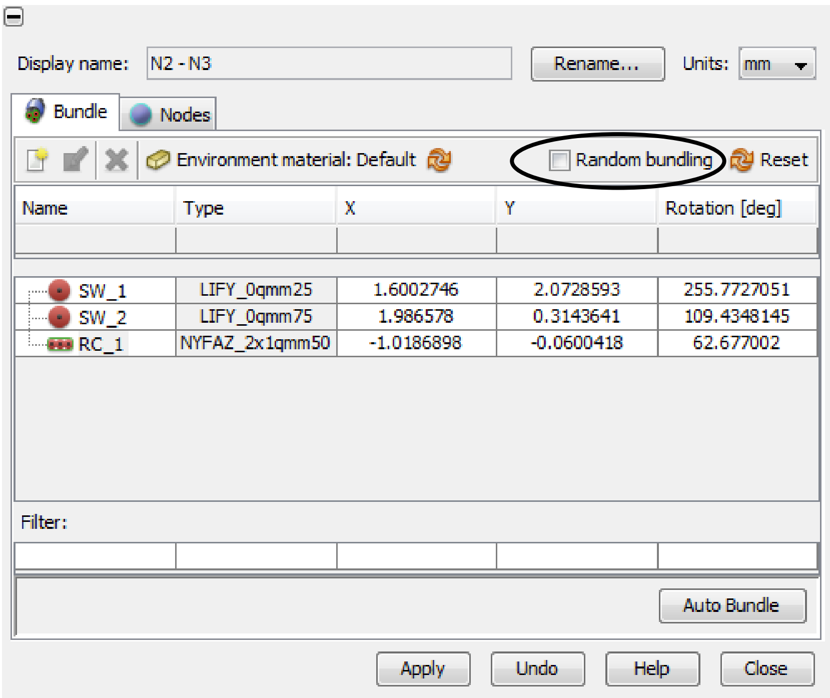
Now we want to investigate the cross section in segment *N2-N3* which is used by both cable bundles. For this purpose we select the corresponding segment (by double-click with the left mouse button) in the *Cable Navigation Tree*:



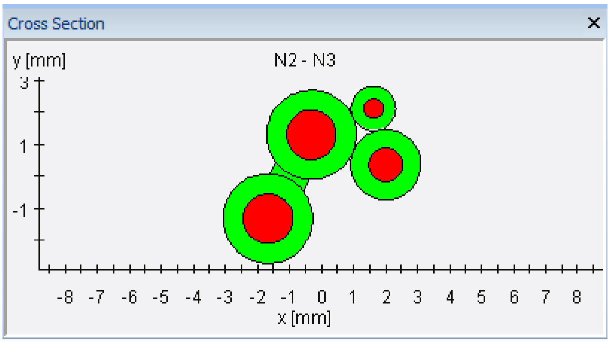
A new dialog box appears and the overlapping wires inside the segment are immediately shown in the *Cross Section* window:



The overlap can be resolved by user interaction or automatic bundling. To resolve it through user interaction, we uncheck the *Random bundling* button in the upper right corner of the dialog box and enter an appropriate coordinate for the new ribbon cable or simply press *Auto Bundle*.



After pressing *Auto Bundle* the *Cross Section* window may look similar to the figure below:



If *Random bundling* is deactivated, the program changes the position values (which were either entered manually or automatically chosen with the help of the *Auto Bundle button*) further in the meshing and modeling process.

The second option would be to ignore the overlapping wires and to let the program resolve the overlap automatically before starting the *2D-TL* solver. In this case, the *Random bundling* button has to be checked. The advantage of this function may not be obvious for such a simple configuration, but it is a powerful function if one has to deal with a complex cable harness consisting of many segments and overlaps.

We finish this section by explaining all terms that have not been introduced so far but will be used in later chapters:

Cable Types

A specific *cable* may be realized with one of the following cable types:

- Single wire
- Twisted cables
- Coaxial cables
- Ribbon cables
- Cable groups

Signals

Every wire inside a cable carries an individual electrical *signal*. For each of these signals a *signal path* is generated. In case of shielded cables, for instance, a *signal path* for every shield will be created. Every signal path starts and ends at a *terminal* where electrical loads or sources may be defined.

Terminals

A *terminal* is the electrical input or output of a signal path.

Connectors

Connectors describe actual physical connectors from a real cable assembly. Each connector can contain a list of *plug-ins* with a specific number of pins, where each pin can be linked to a *terminal*.

Plug-in

A *plug-in* is part of a *connector* and is a collection of pins

Pins (Connector Pins)

A *connector pin* can be linked to a *terminal*. It is part of a *plug-in* and as such also part of a *connector*. It is possible to link more than one *terminal* to a *connector pin*.

Junctions

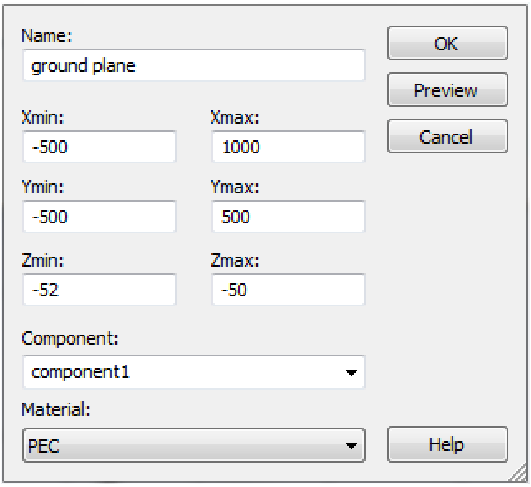
In a similar way that *terminals* are linked to *connector pins*, connector pins can also be connected to other connector pins. This is possible by means of *junctions*.

Components

So far we have generated cables in the 3D space and CST CABLE STUDIO has interpreted each single conductor inside a cable as a potential carrier for a **signal**. Every signal path starts and ends at a terminal where electrical loads or sources may be defined. The user has to provide a current return path for each signal. This can be done either by defining a separate conductor inside the cable or by defining a reference conductor with the help of **components**.

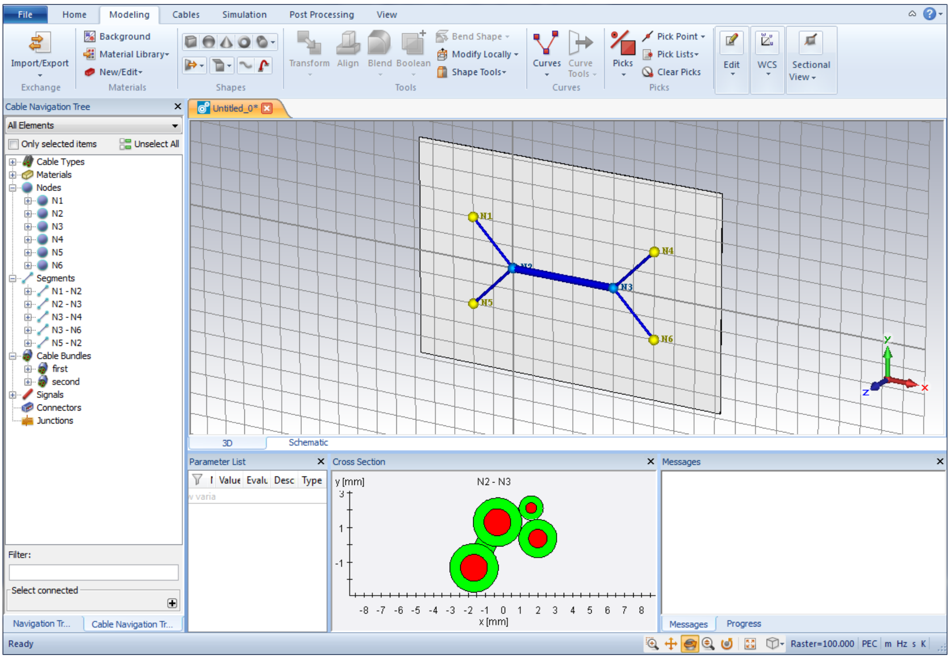
In many cable configurations one can find an additional conducting body acting as reference conductor for the return current or for shielding purposes. In order to define such metallic 3D bodies, the whole range of CST MICROWAVE STUDIO solid modeling possibilities are available. For a detailed explanation on solid modeling the user is referred to the CST MICROWAVE STUDIO manual. For the purpose of this manual the definition of two important metallic bodies will be explained: a simple ground plane and the import of a complex car chassis.

To define a simple ground plane select *Modeling: Shapes⇨ Brick*. Press *ESC* in order to show the dialog box for inserting the coordinates by hand. Inside the dialog box enter the data as shown in the next figure:

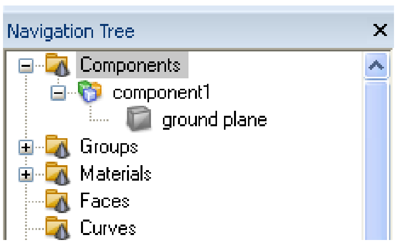


Note that the data field *Material* is set to **PEC**, because only perfect conductors or other metallic materials are considered in the *2D(TL)* modeling process.

After pressing *OK* the new object is displayed in the *Main View*:

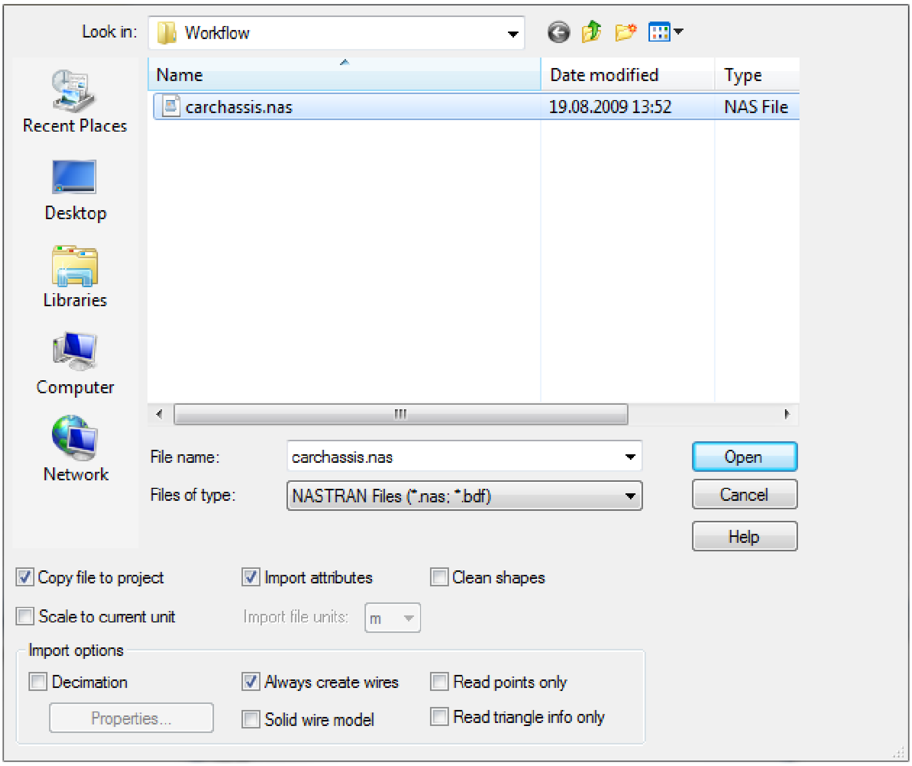


In the (MWS) *Navigation Tree* the new component is listed as an additional object:

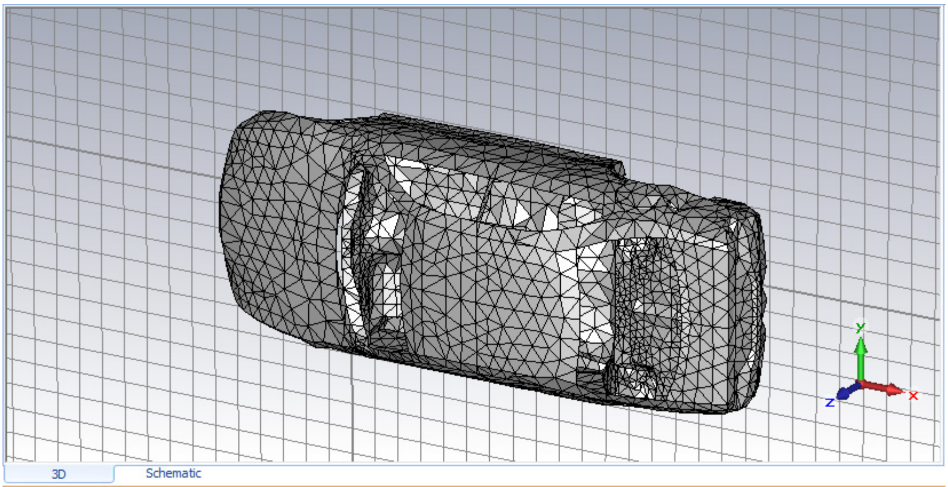


To conclude this section we demonstrate the import of a complex car chassis. For this purpose we close the current project (saving is not necessary), open a new empty project and select *Modeling: Exchange ⇨ Import ⇨ 3D Files ⇨ 3D CAE ⇨ NASTRAN*.

A new dialog box appears allowing the import of an existing NASTRAN file. Use the file browser to navigate to the folder *Examples* of your CST STUDIO SUITE installation directory and select the file *carchassis.nas* in the subfolder *CS/Workflow* as shown in the next figure:



After pressing *Open* the import of the 3D chassis will be performed. The result should look like the figure below:



Chapter 3 — Examples

Having given a short introduction into the theoretical background, the user interface and how to set up a cable harness, this chapter will present three insightful examples on the capabilities of CST CABLE STUDIO. The first two examples deal with the simulation of typical transmission line effects in cables. The third example explains how to proceed when radiation from cables or susceptibility into cables has to be investigated.

Transmission on a Coax Cable

The intention of this example is to acquaint you with the

- Cable library and the definition of materials and cable shields
- 2D modeling dialog box and the generation of an equivalent circuit
- AC- and Transient circuit simulation with preparation of result curves

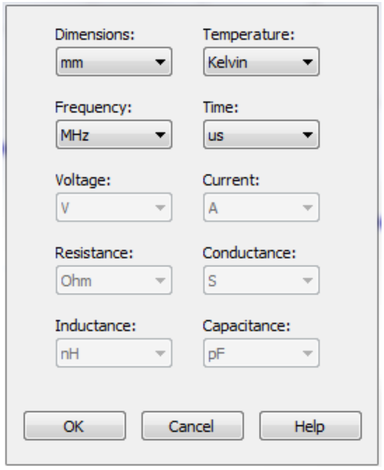
The Structure

In this example a single coaxial cable without any additional reference conductor will be modeled.

The structure supports only the **differential mode**, where a current flows along the inner wire and returns on the inner side of the shield. Due to the absence of an additional conductor, which would allow a return current to flow, the **common mode** is not possible. This fact has to be kept in mind when loading the equivalent circuit in CST DESIGN STUDIO.

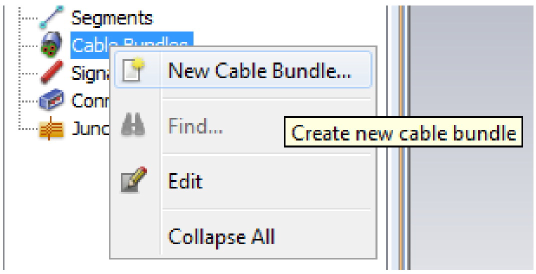
Cable Definition

Create an empty project and save it as **coax cable**. The geometric units of the project can be set with *Home* ⇨ *Settings* ⇨ *Units*:

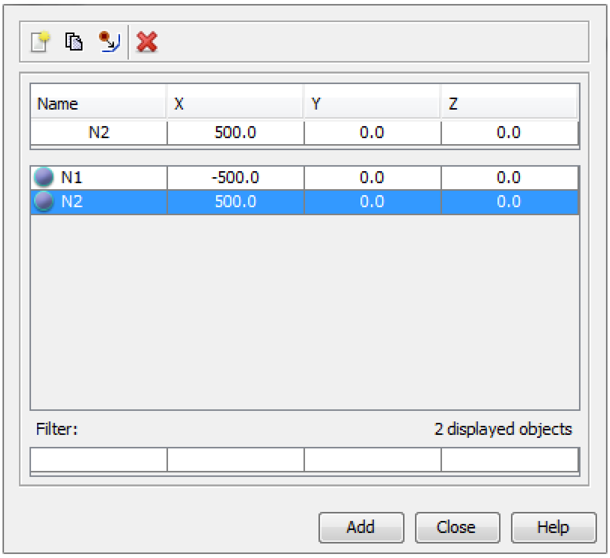


The default units are correct for our example and should be kept.

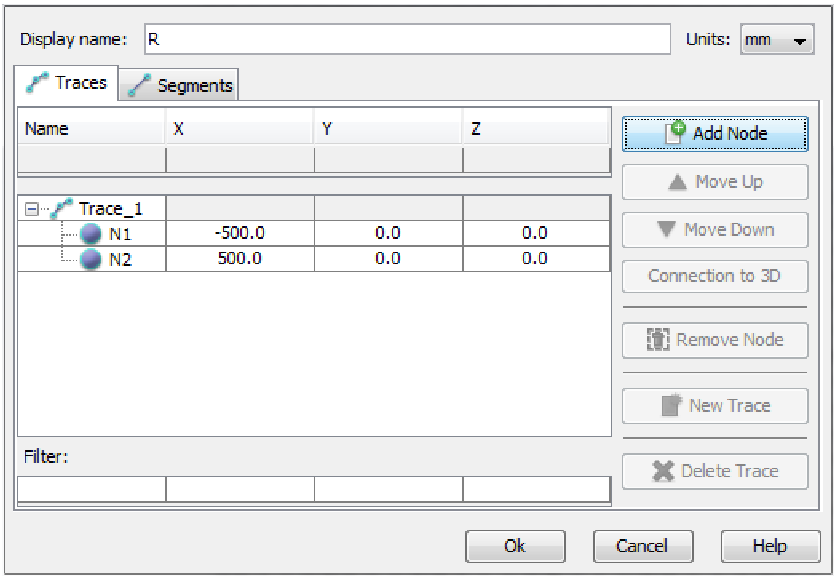
A straight coax cable with a length of *1 m* should be generated. To quickly define the cable we select *Cable Navigation Tree: Cable Bundles* ⇨ *New Cable Bundle* by using the right mouse button:



Then we press the *Add Node* button and enter the following nodes:

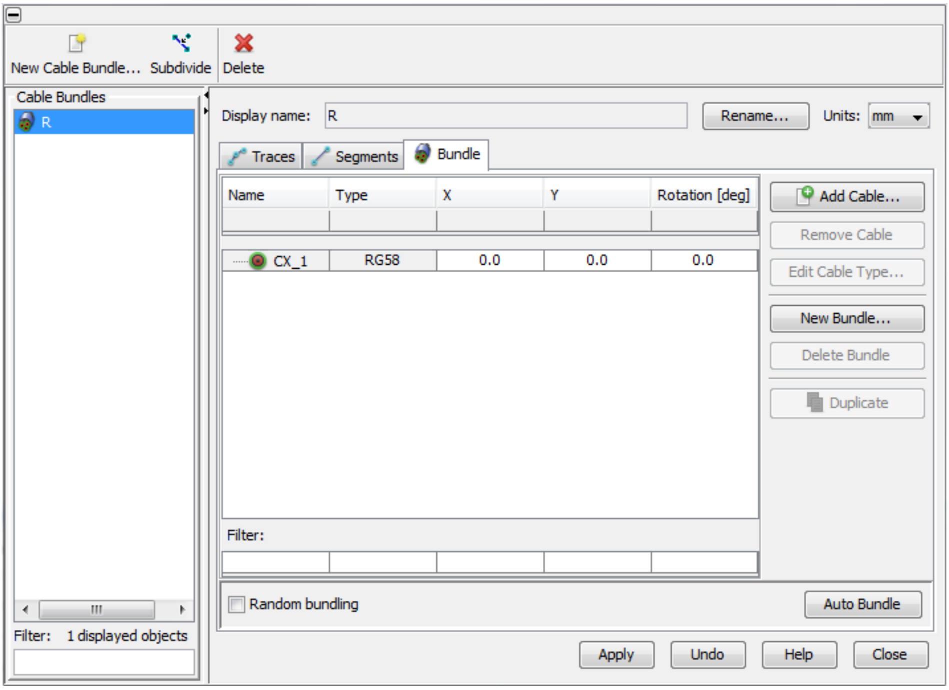


Next we select both nodes (by *Ctrl+left mouse button*), press the *Add* button and close the dialog box. The cable bundle dialog box should look like the figure below:

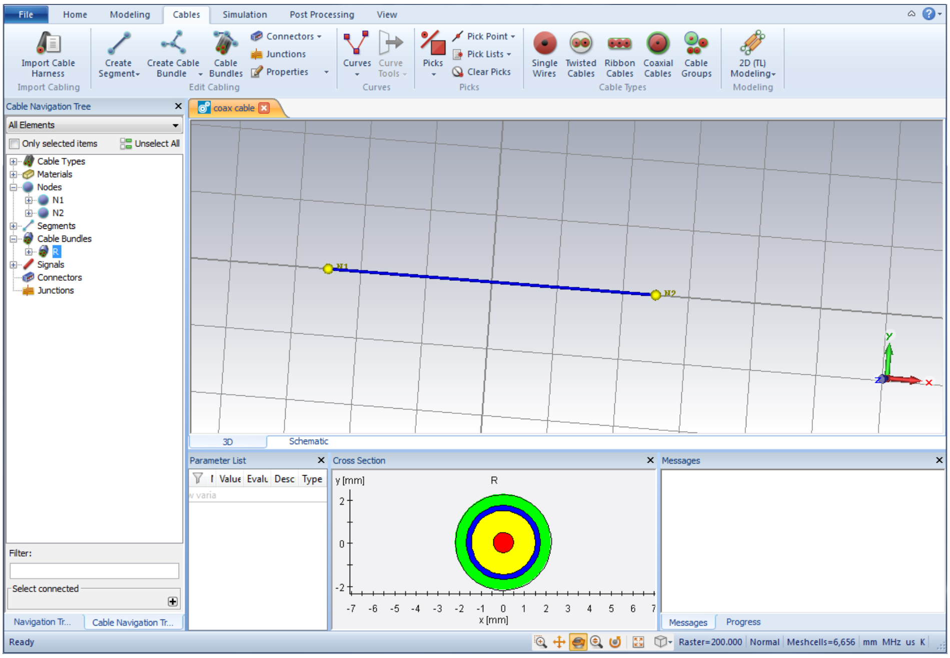


After pressing *Ok* a new dialog box will appear. Here, we press *the Add Cable* button, select *RG58* from the Library, press *Add* and close the dialog box.

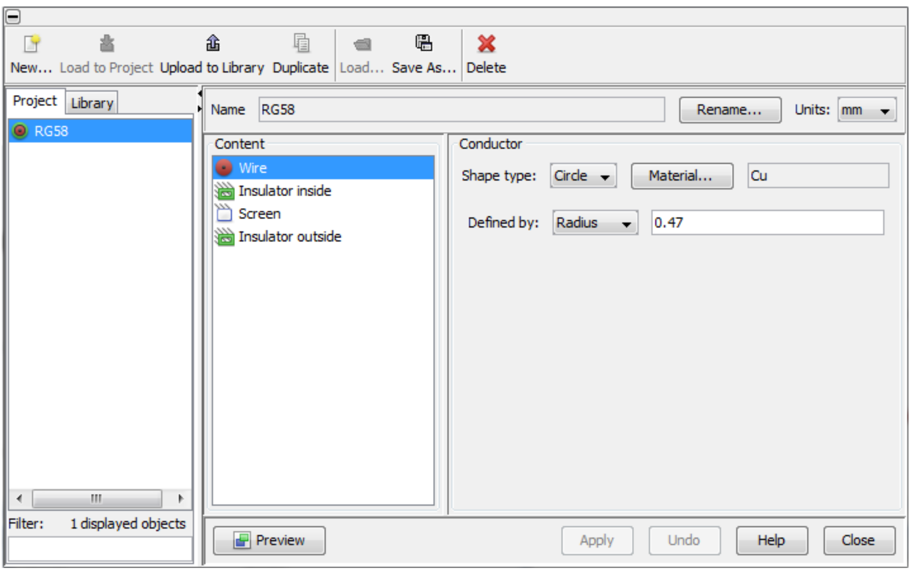
The cable bundle dialog box should look like the figure below:



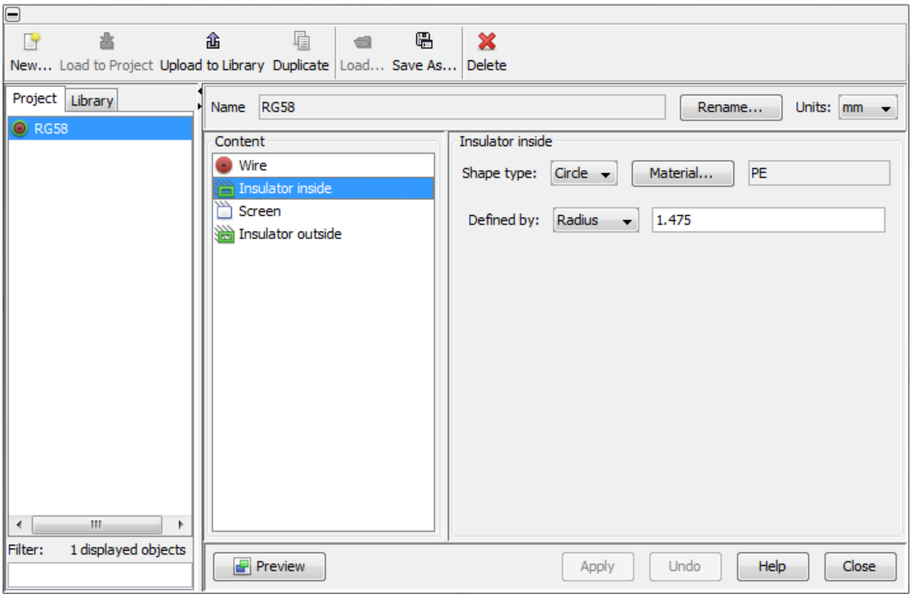
After pressing *Apply* and closing the dialog box the generated cable bundle will be displayed in the *Main View*:



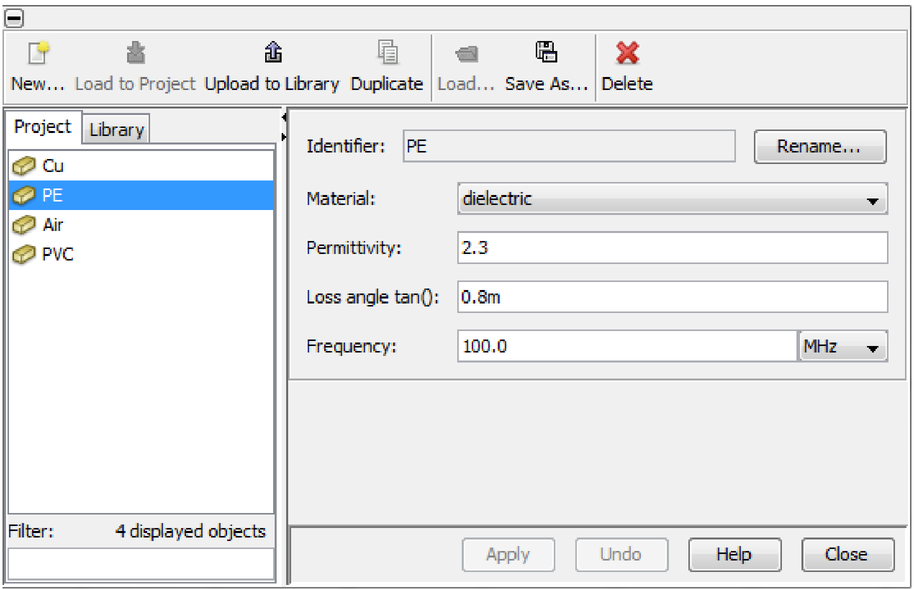
In order to see the characteristics of the cable, we pay a short visit to the *Cable Types* folder. We go into *Cable Navigation Tree: Cables Types* ⇨ *Coaxial cables* and double-click on the *RG58* cable which has been loaded from the Library. The following dialog box will appear, listing the structure of a coax cable with the inner *Wire*, the *Insulator Inside* (inside the screen), the *Screen* and the *Insulator Outside* (outside the screen). On the right side the dialog box displays the corresponding values:



When selecting the content *Insulator Inside* the dialog box shows *PE* as material used for the insulator:

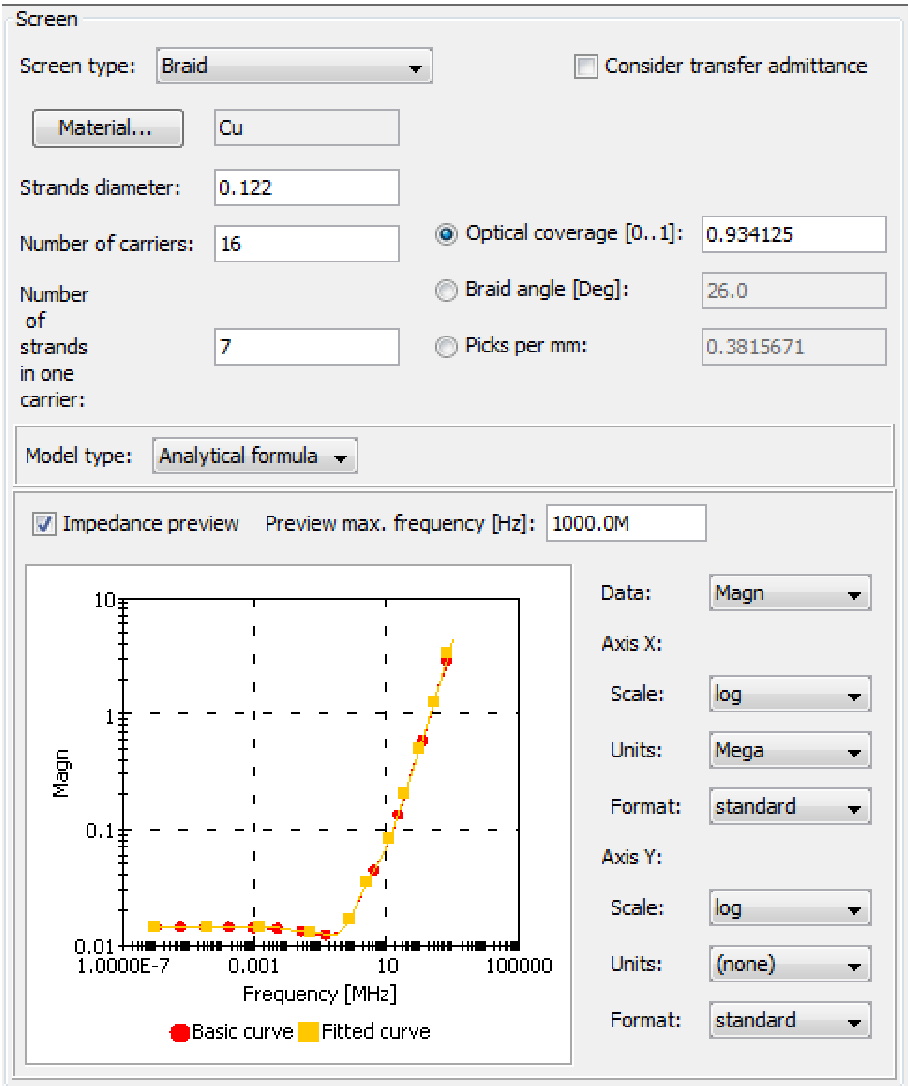


If we want to check the characteristics for this material we have to go to *Cable Navigation Tree*: *Materials* and double-click on *PE*. A dialog box from the Material Library appears, showing the values of the three important characteristics: *Permittivity*, *Loss angle tan()* and *Frequency* (the frequency value belongs to the permittivity value):



CST CABLE STUDIO performs a broadband approximation of the loss angle. This approximation guarantees a constant value of the defined loss angle within the frequency range where the corresponding equivalent circuit of a cable (including the material) is valid.

We end the visit of the characteristics of the RG58 cable by looking at the characteristics of the screen. The dialog box shows that the type of the screen is a *braided shield*. This means the screen does not consist of one compact conductor but is composed of many single conducting filaments. This impacts the shielding quality which is described by the so called **transfer impedance** of the screen.

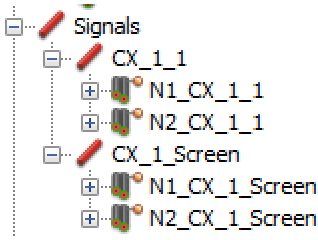


In order to calculate the transfer impedance with the help of an analytical formula (Kley model), five parameters of the braided screen have to be set. Any change of the parameters below will force an update in the impedance display:

- The **inner diameter** of the screen: this is automatically defined by the *Insulator inside* definition of the inner dielectric (see page before).
- **Filament diameter**
- **Number of strands in one carrier**: a carrier is a package composed of a certain number of filaments
- **Number of carriers**: the number of carriers that is used for the braid
- **Braid angle**: the angle with which the carriers are woven. As an alternative the **Optical coverage** or the **Picks per unit length** can be defined instead of the Braid angle.

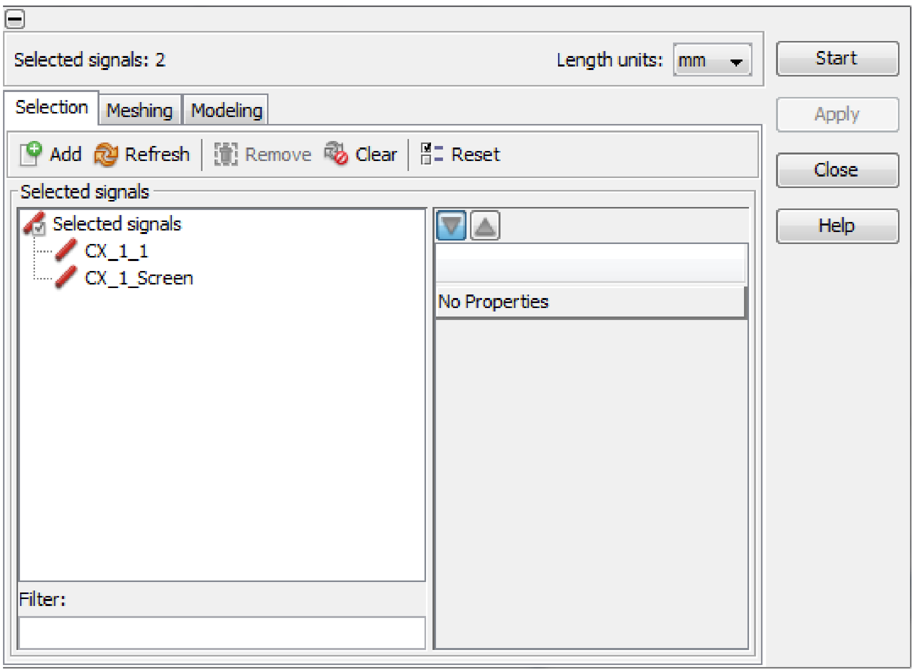
2D Modeling

Before we generate an equivalent circuit let's have a short look at the folder *Signals* in the *Cable Navigation Tree*.

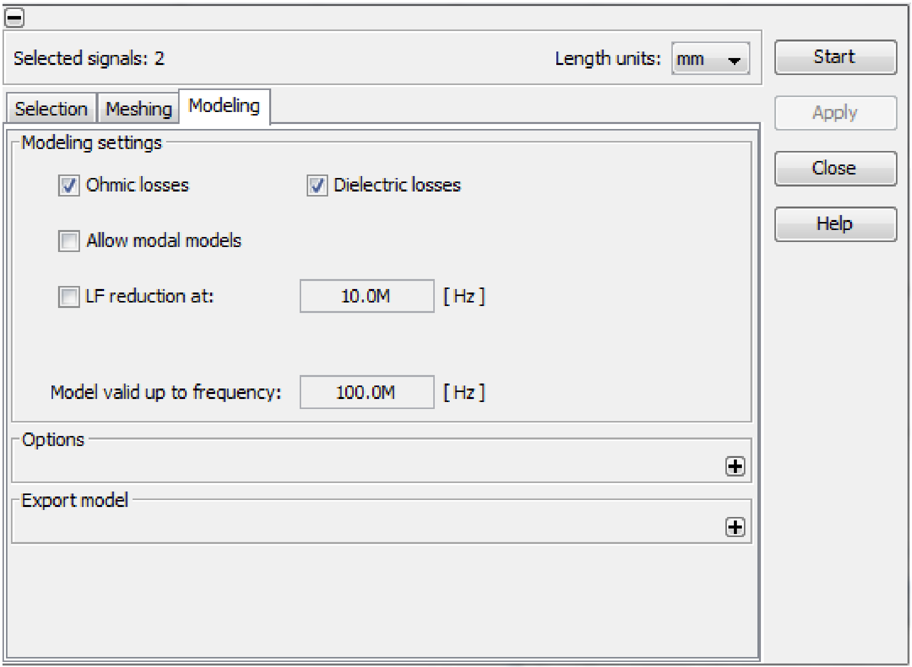


The inner wire and the outer screen of the coax cable are listed as two different signals. The names of the signals are automatically generated by the program. In addition, for each signal two terminals are allocated and can be distinguished by the prefix *N1_* and *N2_*. These terminal names will be used to name the pins of the equivalent circuit as we will see in the next step.

To generate the equivalent circuit, we choose *Cables: 2D (TL) Modeling*. A new dialog box appears where we click on the *Selection* tab and see the two signals that were automatically selected by the program:

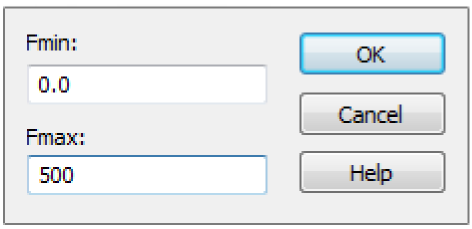


We change to the *Modeling* tab and set the dialog box as follows:



We check *Ohmic losses* and *Dielectric losses* in order to have the program model the skin-effect and the losses resulting from the dielectric loss angle specified for the inner insulator. We then press *Apply*.

An important setting is the maximum frequency up to which the model should be valid. In this example we want the maximum frequency at *500 MHz*. Therefore, we close the dialog, go to *Simulation: Settings* ⇨ *Frequency* and enter the corresponding numbers as shown in the figure below:



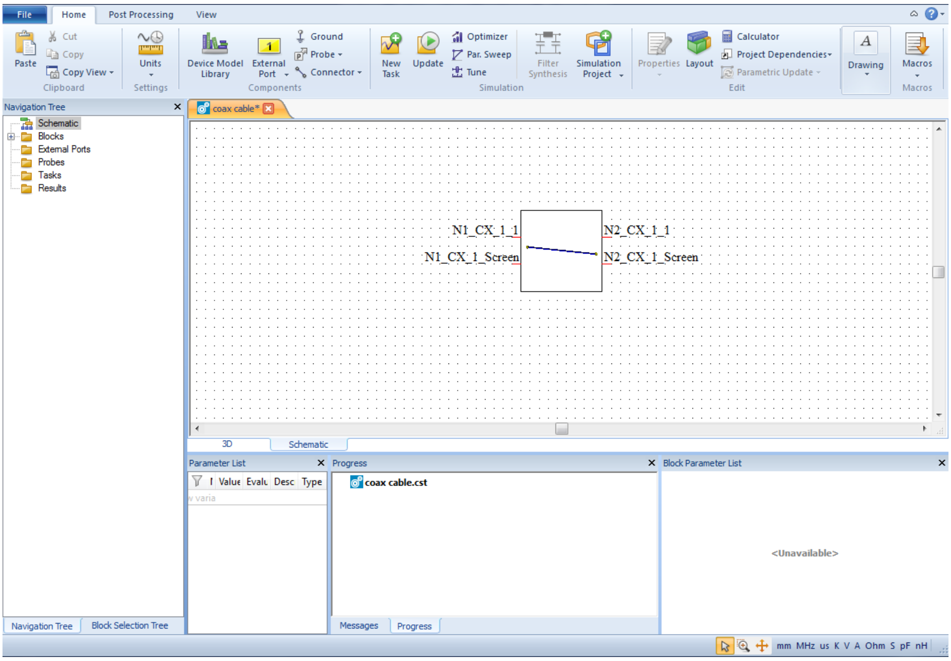
Generally, we recommend that you set the valid frequency range just as high as needed for your application. This is due to two reasons. First, the maximum frequency range affects the complexity of the equivalent circuit - the higher the frequency the more complex the circuit is.

Secondly, every configuration has its own natural frequency limit and above this limit a model can't be produced because of the modeling method used by the program (see explanation for TEM-modes in Chapter 4, *General Methodology*). The limiting factor is the cross section size of the cable bundle – the larger the size of the cross section the lower the maximum possible frequency range. At the end of the modeling process the program reports the maximum achievable frequency.

If we open the *2D (TL) Modeling* again, we see that the frequency parameter has been changed. Now, we press the *Start* button at the upper right corner. Almost immediately, the modeling process finishes and a corresponding schematic symbol is produced. To see the circuit and to set up a simulation we have to change to the *Schematic* tab.

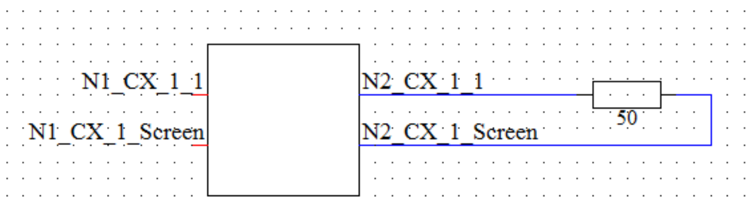
Circuit Simulation and Results

After changing to the *Schematic* tab the schematic symbol with its automatically generated terminal pins is displayed in the *Main View*.

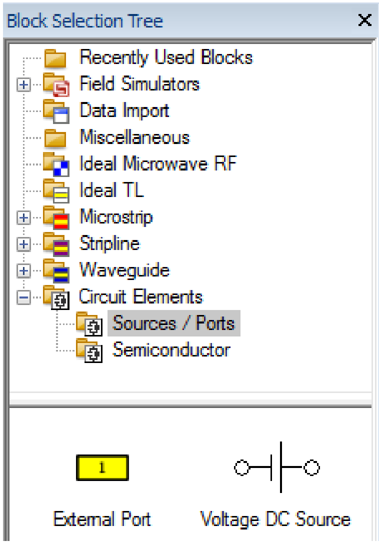


We can now prepare the circuit for an AC- task. As a first step we connect a resistor of *50 Ohm* between both pins on the right hand side of the schematic block. In order to do this, we select *Block Selection Tree*: *Circuit Elements* where we will see different device type symbols. These symbols can be placed in the Main View by *drag-and-drop*.

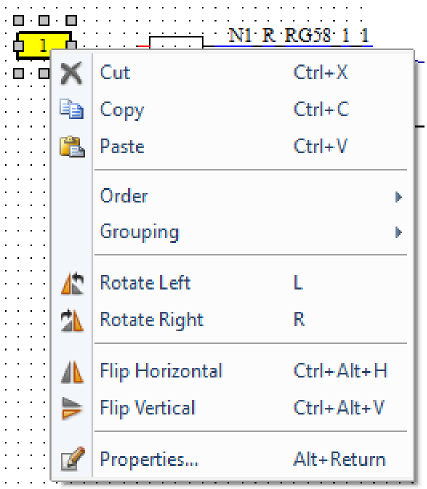
The devices can be electrically connected by selecting *Home: Components* ⇨ *Connector*. For a deeper explanation on how to use the schematic editor we refer to the CST DESIGN STUDIO manual. The schematic should now look like the figure below:



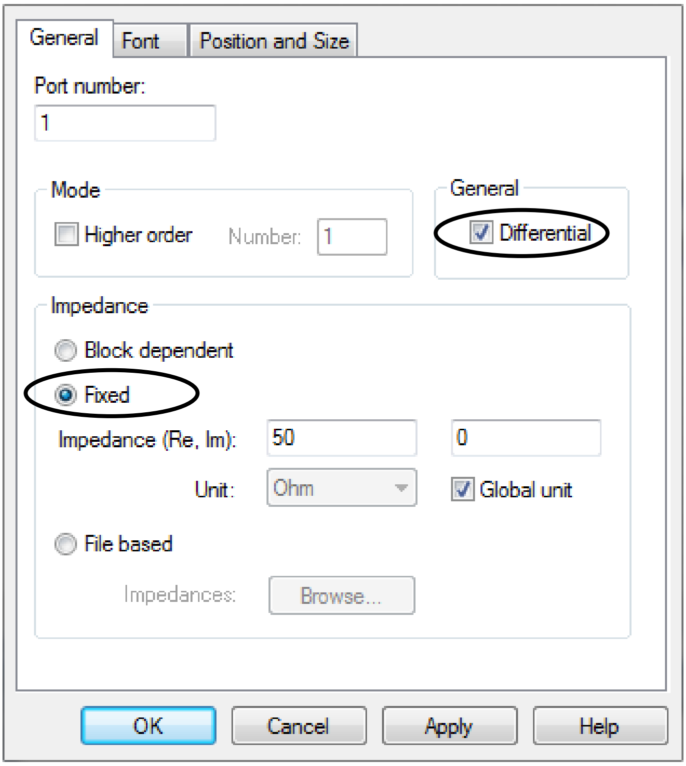
Next we connect another *50 Ohm* resistor on the upper pin on the left side of the block. In order to complete the schematic with an excitation, we need an external port. The corresponding symbol can be found selecting *Block Selection Tree*: *Circuit Elements* ⇨ *Sources/Ports* as shown in the figure below:



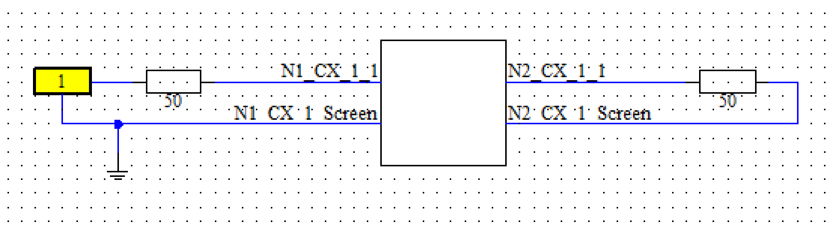
We drag the yellow *External Port* into the *Main View* and enable it for differential loading. To do this we select the symbol by using the right mouse button and choose *Properties* as shown in the figure below:



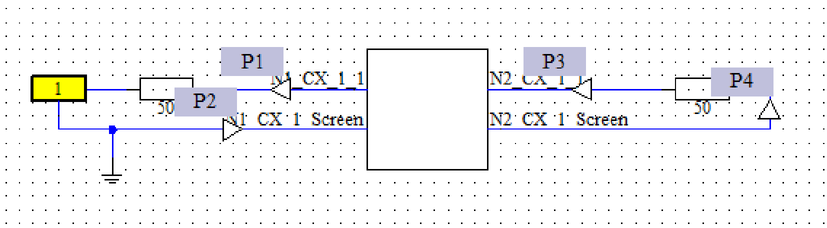
In the following dialog box we first check *Differential* and *Fixed* and then press *OK:*



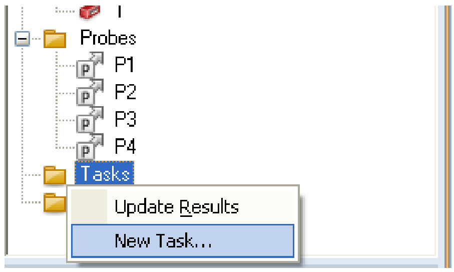
Next we connect the port with the resistor and connect a ground symbol on the lower pin of the left side of the block. The circuit should look like the figure below:



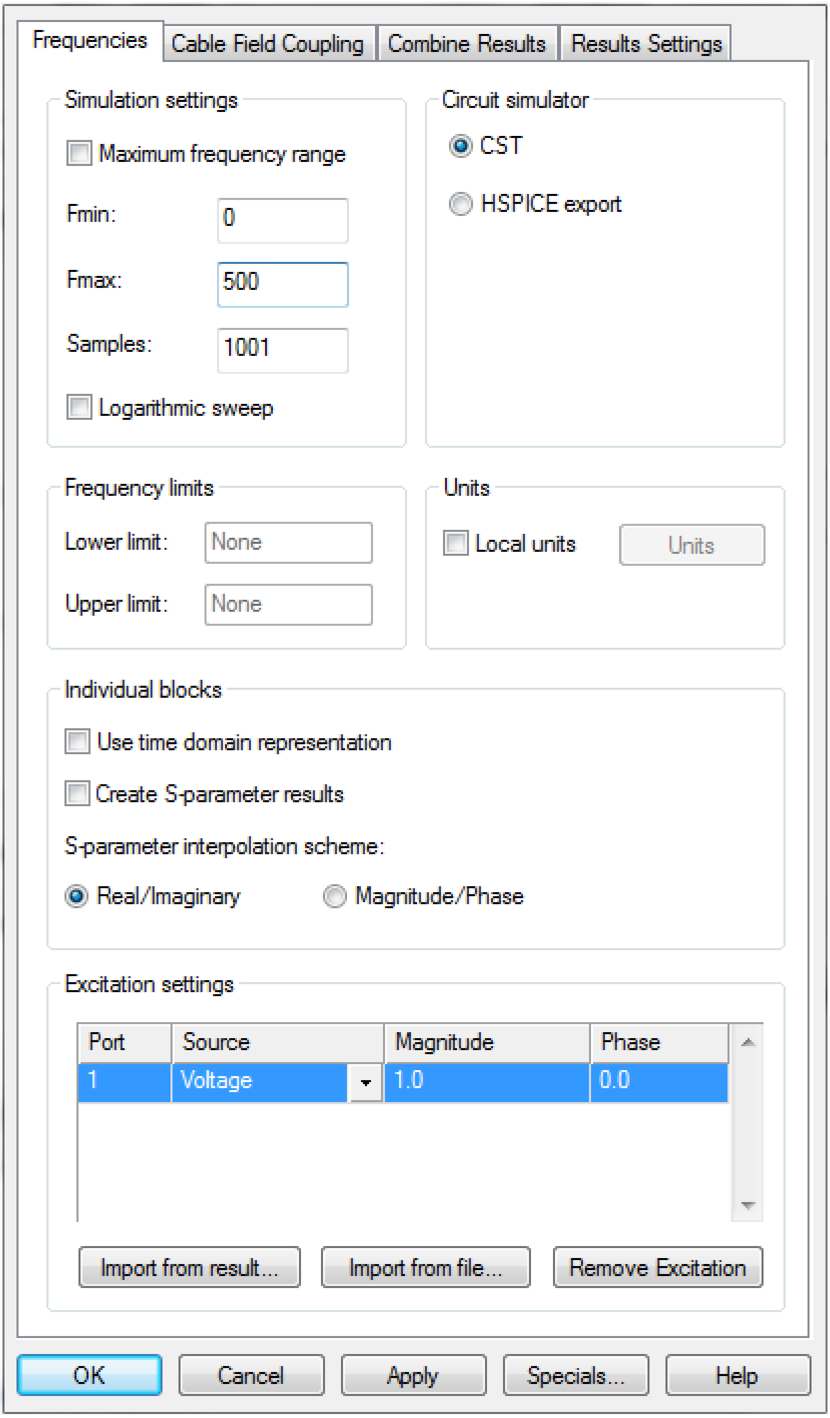
Finally we have to put probes on each terminal. To insert probes, we simply select *Home: Components* ⇨ *Probe* ⇨ *Add Probe*. The complete circuit should look like the circuit in the figure below:



To set up an AC-task we select *Navigation Tree*: *Tasks* ⇨ *New Task* (using the right mouse button):

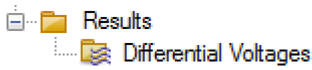


A dialog box will appear where you should select *AC, Combine results* as simulation task. After pressing *OK* the dialog box for the *AC*-task appears where we are able to enter the following settings:

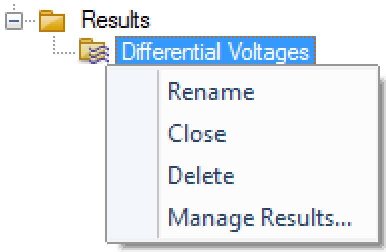


We press *OK* to close the dialog box and start the simulation by pressing *Home: Simulation⇨Update. A*fter a few seconds the *Messages* window will inform us that the simulation task has completed.

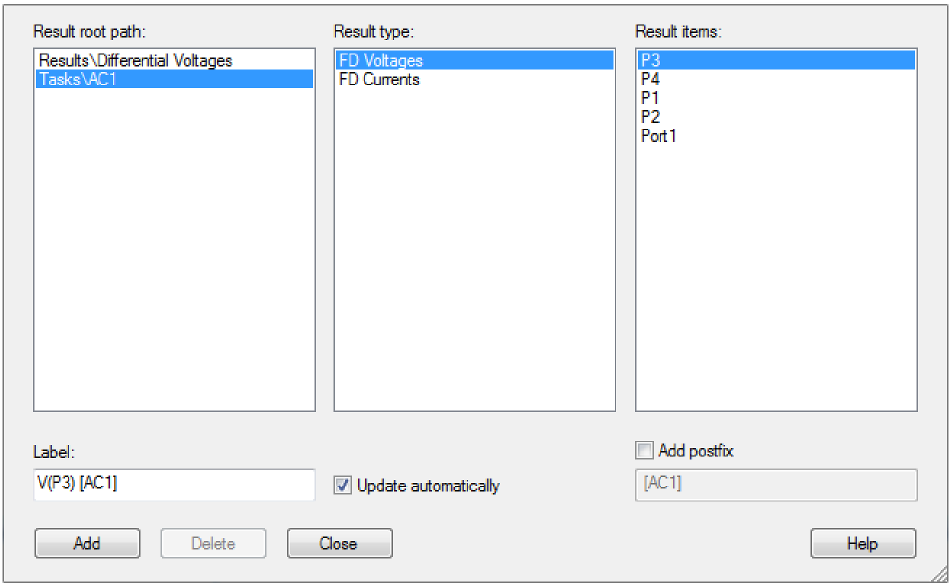
In order to display the differential voltages on both ends of the cable, we have to go into the *Navigation Tree*, select the *Results* folder, choose *Add Result Plot* (by using the right mouse button) and name the new result folder *Differential Voltages* as depicted in the next figure:



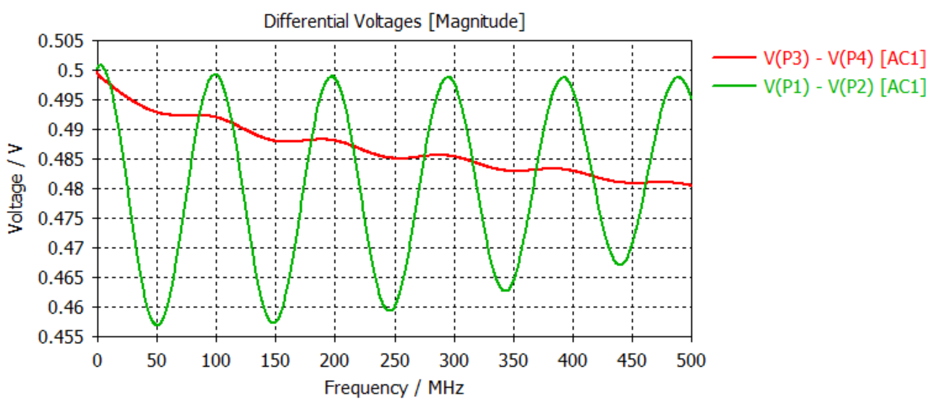
Next, we select *Differential Voltages* and choose *Manage Results* (by using the right mouse button) as can be seen in the figure below:



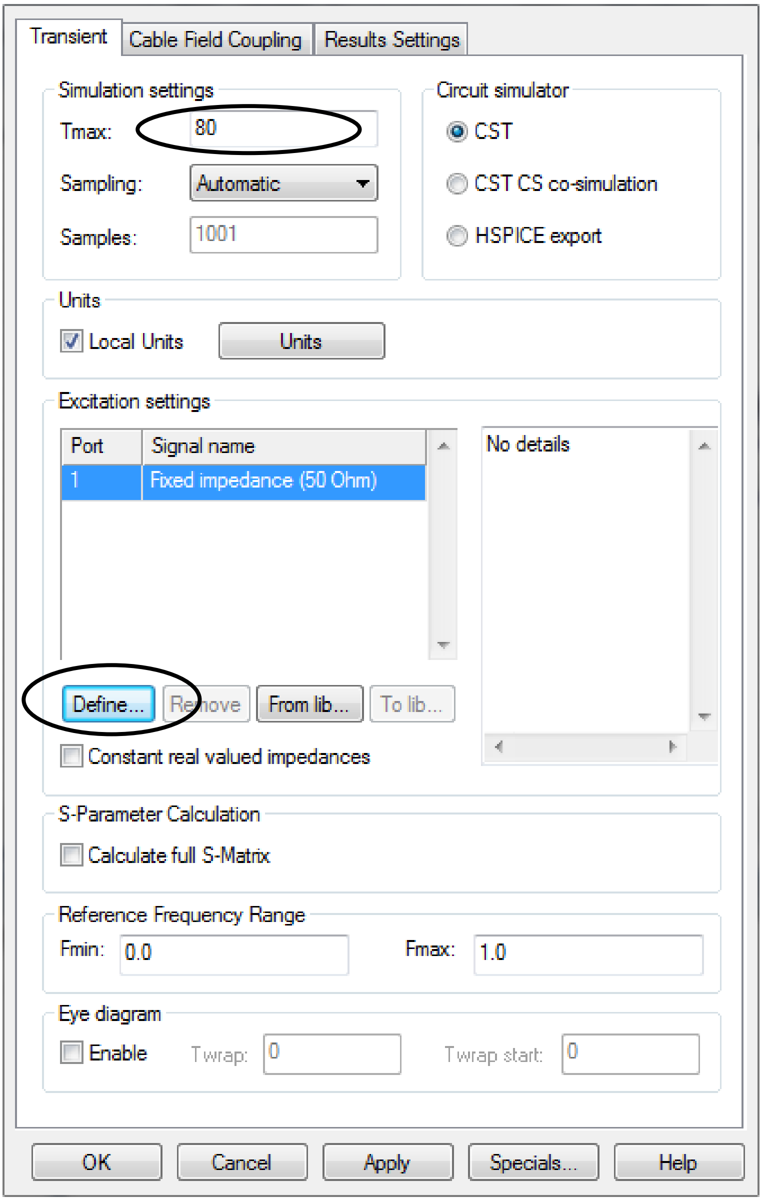
The following dialog box will appear:



We select *FD Voltages* in the mid column. Then we go to the right column and select the first two result items, *P3* and *P4* (by using *Ctrl*+left mouse button), and press *Add*. We repeat this once again for the pair *P1* and *P2*. After closing the dialog box we will see the results in a separate tab:

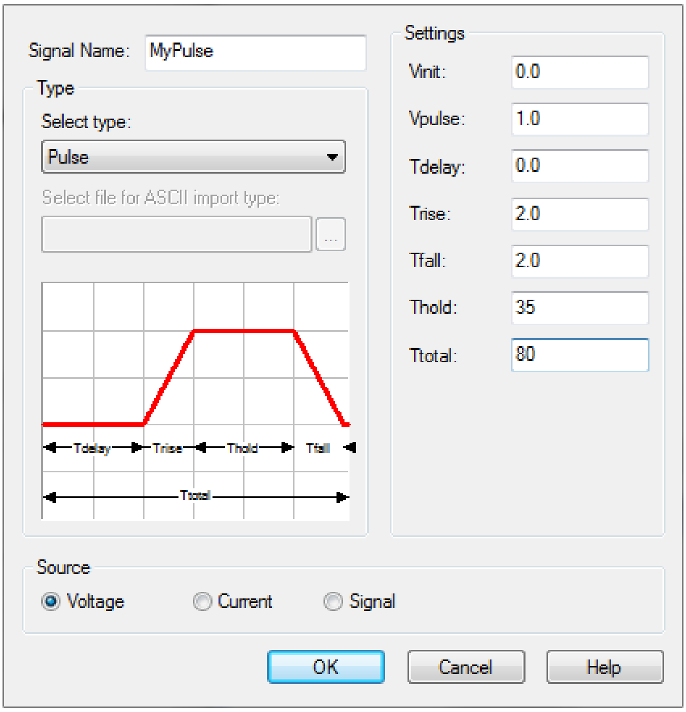


In order to perform a Transient-task, a similar procedure can be applied as it has been explained for the AC-task. We select *Navigation Tree*: *Tasks* ⇨ *New Task* (using the right mouse button). In the following dialog box we select *Transient* as simulation task and press *OK*. A dialog box will appear as shown in the figure below:

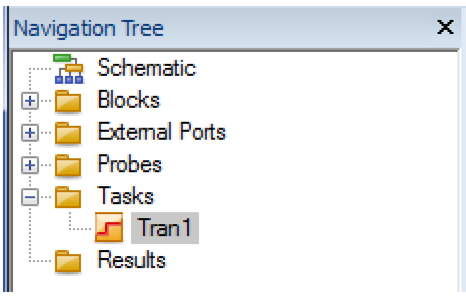


Before defining an excitation signal we change the local time units to *ns* by selecting *Local Units* and clicking the *Units* button. After setting the units we set *Tmax* to *80 ns*. Then we select *Port 1* and press the *Define* button.

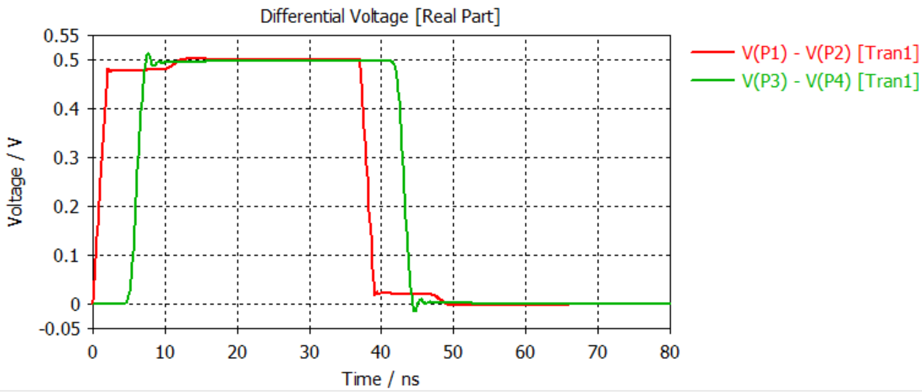
A new dialog box will arise where we select *Pulse* as excitation type. In addition we enter the settings as shown, press *OK* and do another *OK* to finish the Transient-task settings.



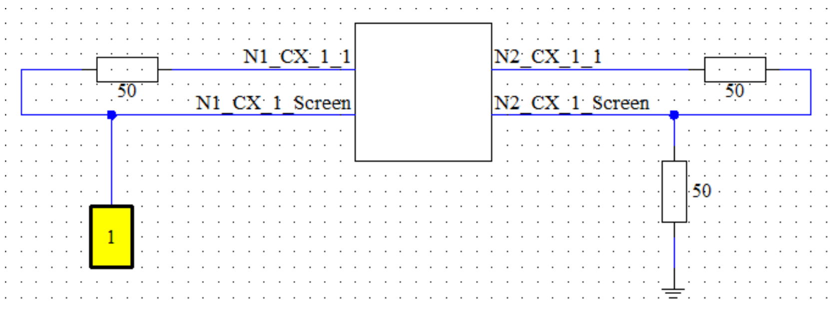
Before starting the new Transient-task we delete the old AC-task and we also delete the corresponding result folder *Differential Voltages*. The *Navigation Tree* should look like the figure below:



Now we start the Transient-task by pressing *Home: Simulation⇨Update.* After a few seconds the task will be completed and we create a new result folder named *Differential Voltage* and add two voltages (between *P1-P2* and between *P3-P4*). We should see the following result:



We want to finish the example with a special note on a general characteristic of this cable model. As mentioned at the beginning of the example (see sub chapter *The Structure*) the model was generated without the presence of an additional reference conductor. Therefore it makes sense to put a pure differential termination on the model's pins as in our example. Any termination that forces currents to an imaginary ground (see figure below) is indeed allowed by the schematic editor but wouldn't lead to any reasonable results, neither in the simulation nor in the real world.



Note: the picture shows the way **not** to do the loading in case of a non-ground referenced structure!

Crosstalk between two Wire Bundles

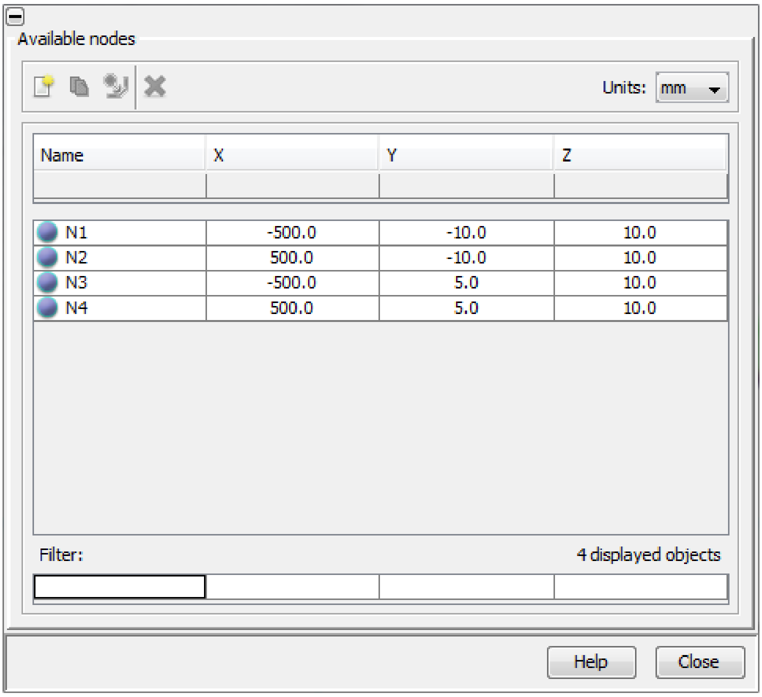
The aim of this example is to acquaint you with the

- concept of connectors
- search distance to couple cable bundles lying in different segments
- S-Parameter analysis
- difference between lumped models and modal models

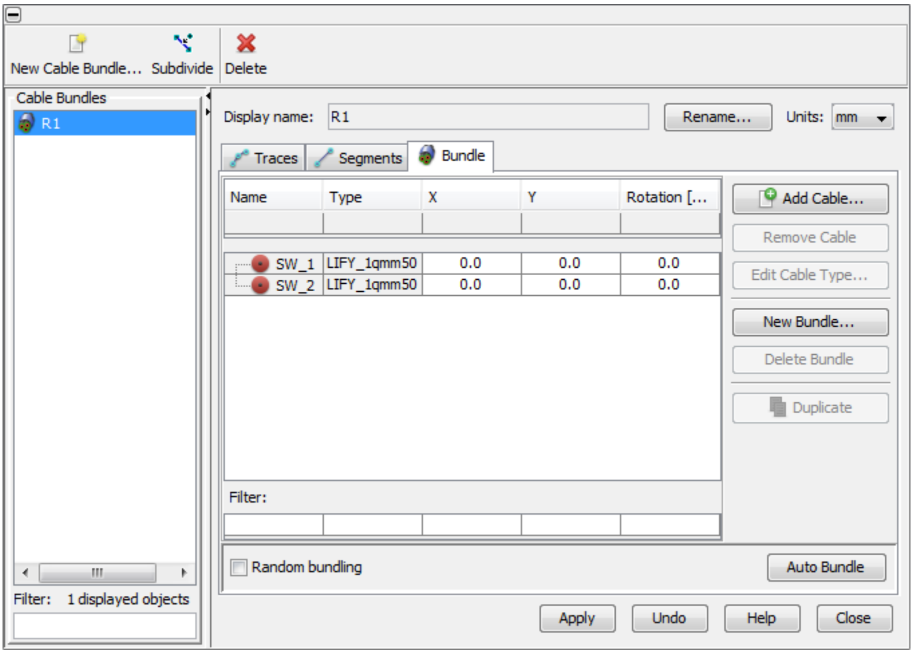
Cable Definition

In this chapter we want to set up a simple cable harness consisting of two separate cable bundles. All cables in a cable bundle are automatically coupled by their mutual capacitances and mutual inductances. Electromagnetic coupling between different cable bundles depends on the given *search distance* during *meshing*.

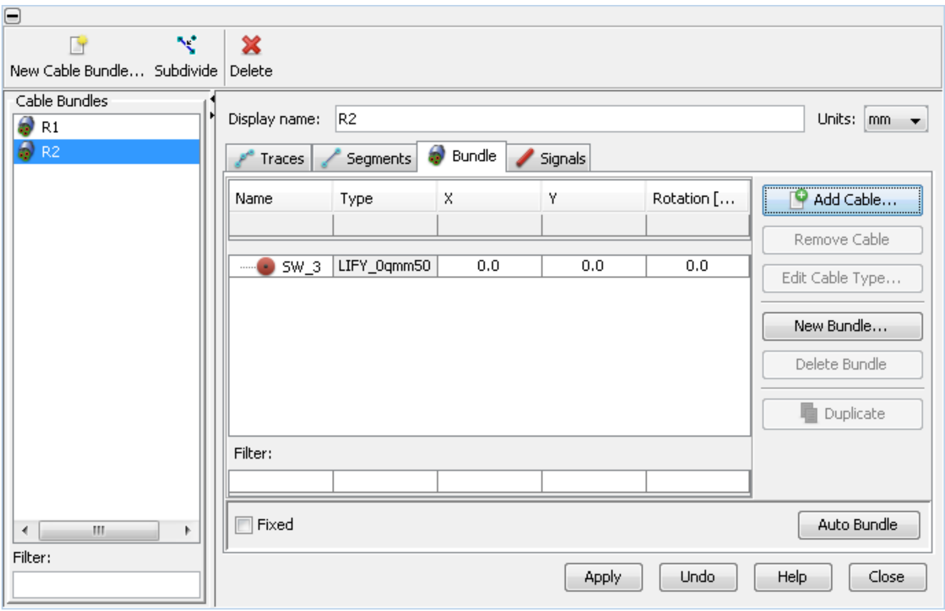
We create an empty project and save it as **two cables**. The geometric units are left to default. We start with the definition of four nodes by double-clicking on *Cable Navigation Tree: Nodes*. After the dialog box appears, we create the nodes by assigning the co-ordinates as shown in the figure below:



Next we allocate a cable bundle with name *R1* between *N1* and *N2* and put two single wires into it as shown in the next dialog box:

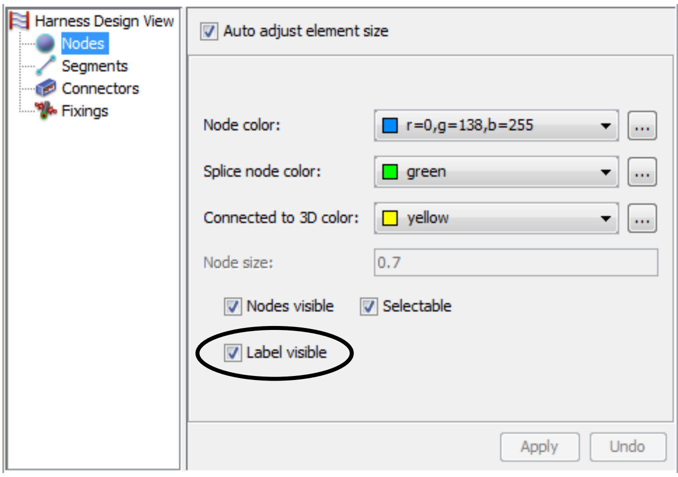


We press *Auto Bundle* and then allocate another cable bundle *R2* between *N3* and *N4* by pressing the *New Cable Bundle* button. We put a single wire into it as shown in the dialog box below:

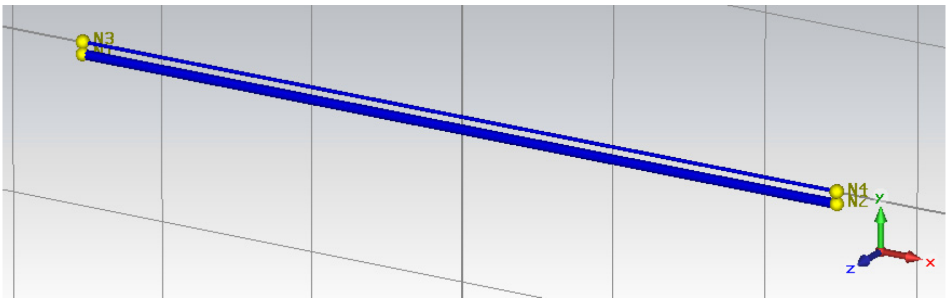


We press *Apply* and close the dialog box.

Next we select *Cables: Edit Cabling ⇨ Properties ⇨ Harness View Attributes* and change the settings for nodes as shown in the following dialog box:



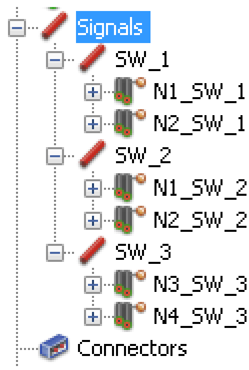
The structure defined so far is displayed in the *Main View*:



Connector Definition

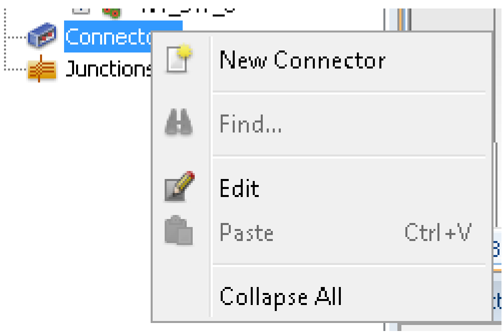
As you may have noticed, the automatically generated pin names on the schematic symbol of the last examples were a bit long and complicated. To get shorter names and gain a better overview we will now introduce **connectors**.

We begin with a short look at the current generated signal and terminal names. To do this, we go into the *Cable Navigation Tree* and expand the *Signals* folder as shown in the figure below.

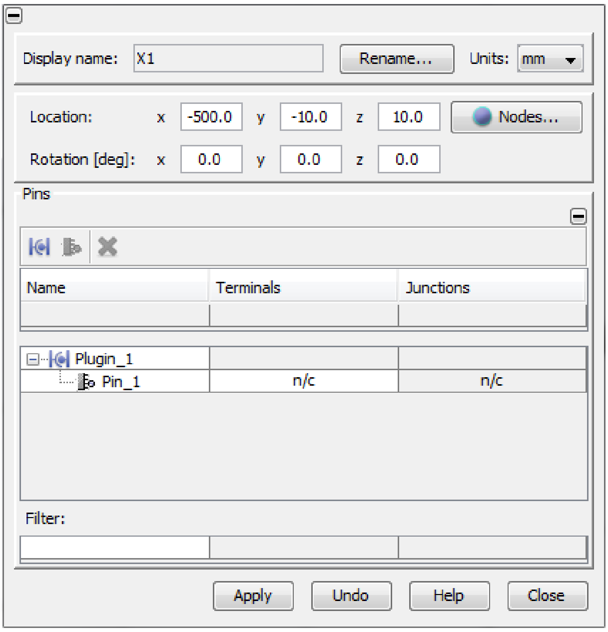


These terminal names (e.g. *N1_SW_1*) will be assigned to the pins of the schematic symbol by default. To avoid this default behavior we start with the definition of the first connector at node *N1*.

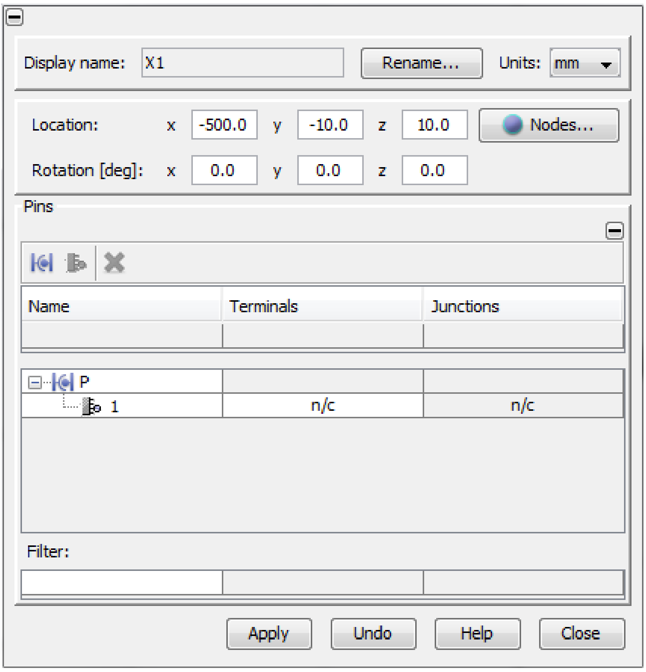
We go again into the *Cable Navigation Tree*, select *Connectors* and choose *New Connector* by using the right mouse button:



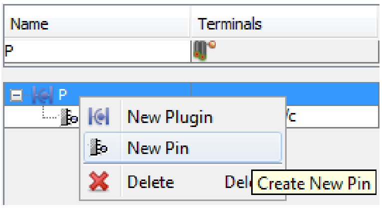
The following dialog box will appear. We press the *Nodes* button, select node *N1* and assign *X1* as name of the connector:



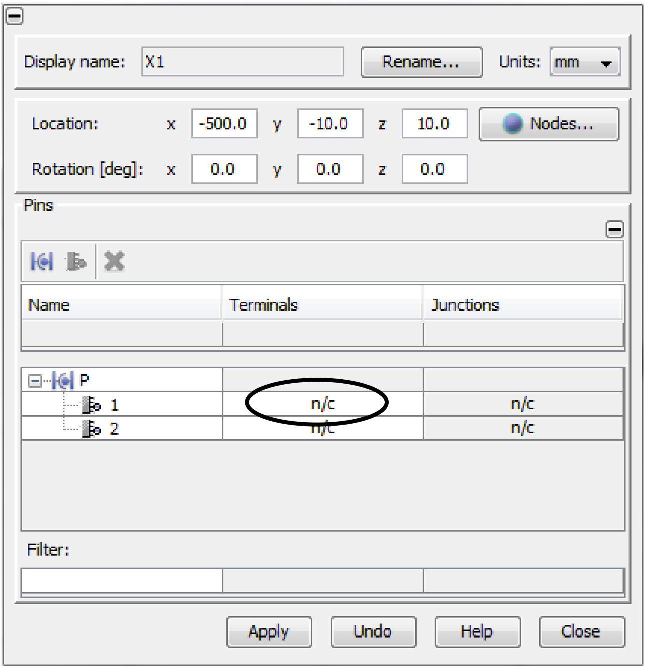
Next we have to define connector pins and assign them to the existing terminals. A plugin is a group of pins inside the connector and a connector can consist of one or more plugins. For our case a single plugin is sufficient and we name it *P*. Below plugin *P* we see the first allocated pin and assign it the name *1*s:



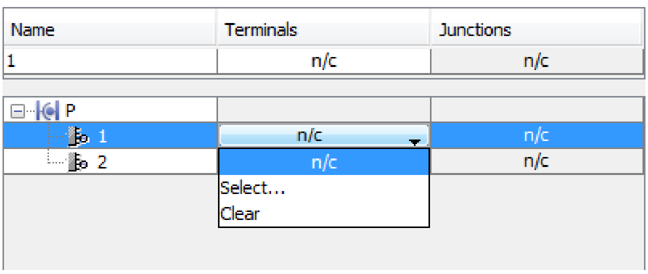
In order to allocate a second pin, we select *P* and choose *New Pin* by using the right mouse button as shown in the figure below:



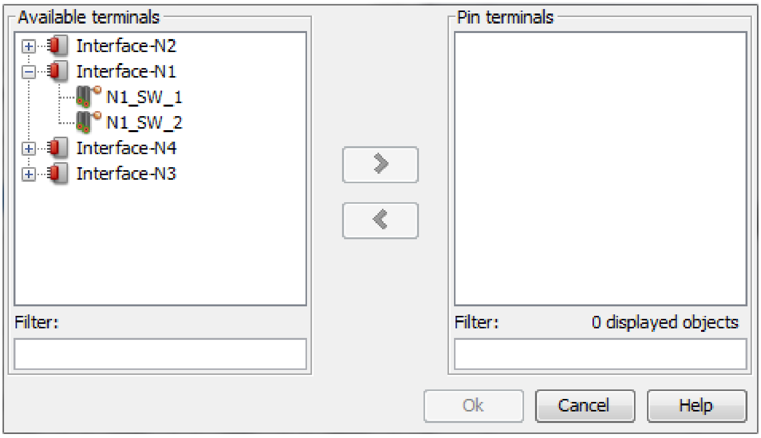
After assigning the name *2* to the new pin the dialog box should look like this:



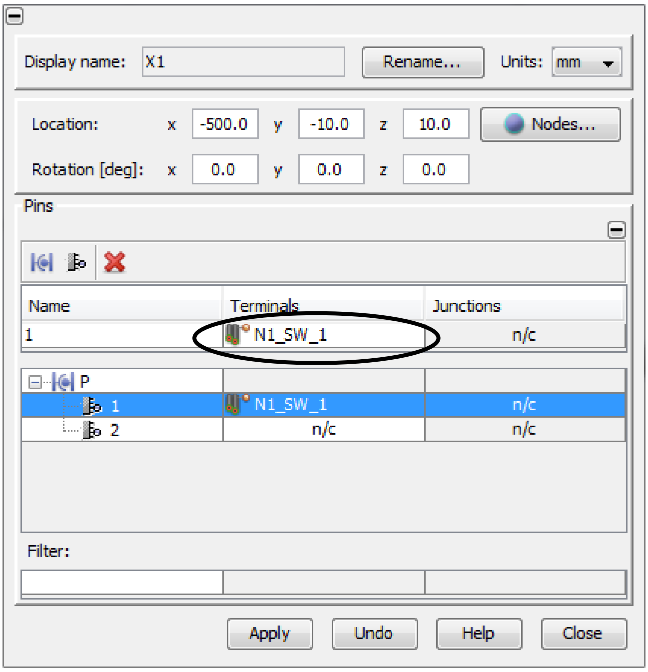
To assign an existing terminal to pin *1* we select the cell in the circle mark (by using the left mouse button). A pop-up menu will appear as shown in the following figure:



After pressing *Select* a new dialog box will appear:

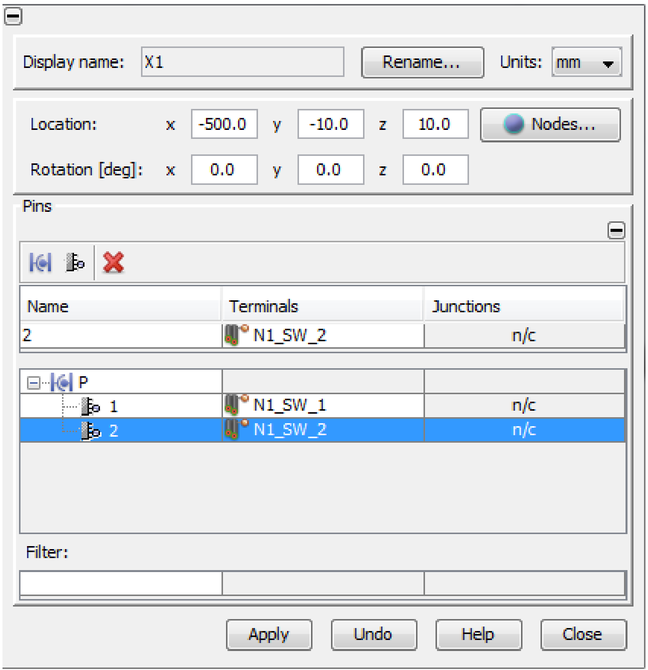


We expand *Interface-N1* inside the left column and see the automatically generated terminal names on node *N1*. We select the terminal *N1_SW_1* and shift it to the right side (by using the arrow in the middle of the dialog box). After pressing *Ok* the assignment for pin 1 is done as seen in the dialog box:

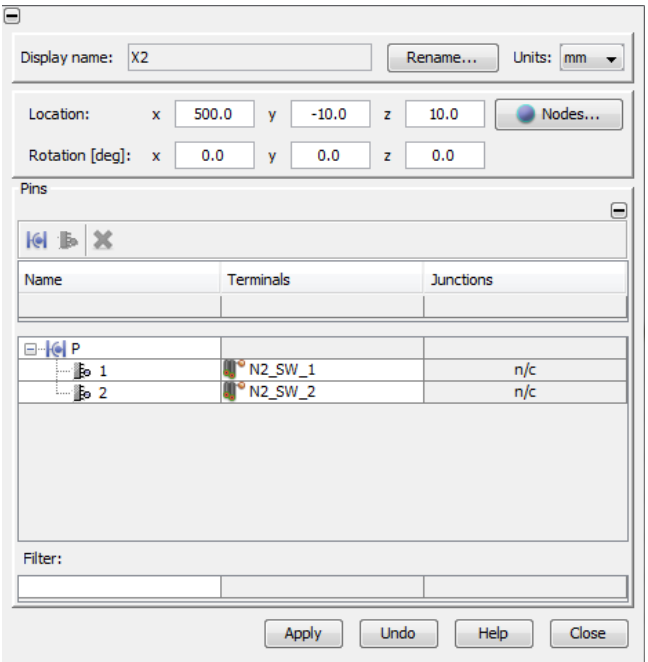


We repeat this procedure for pin *2.*

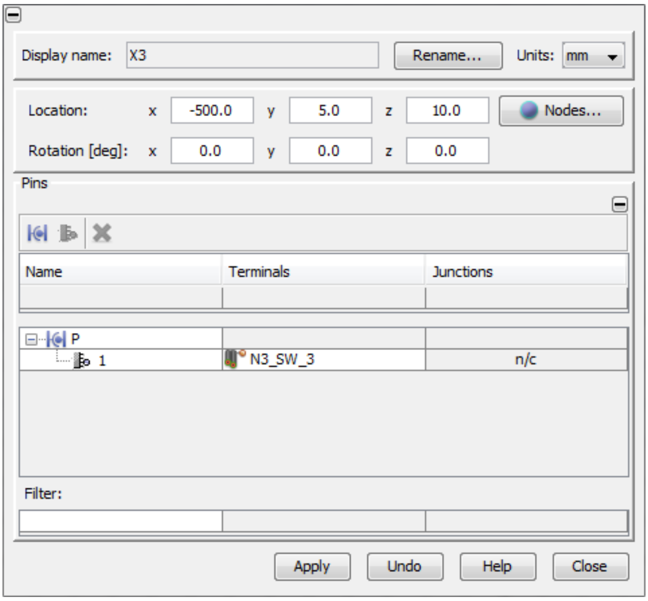
Finally the connector *X1* should be defined as shown in the next dialog box:



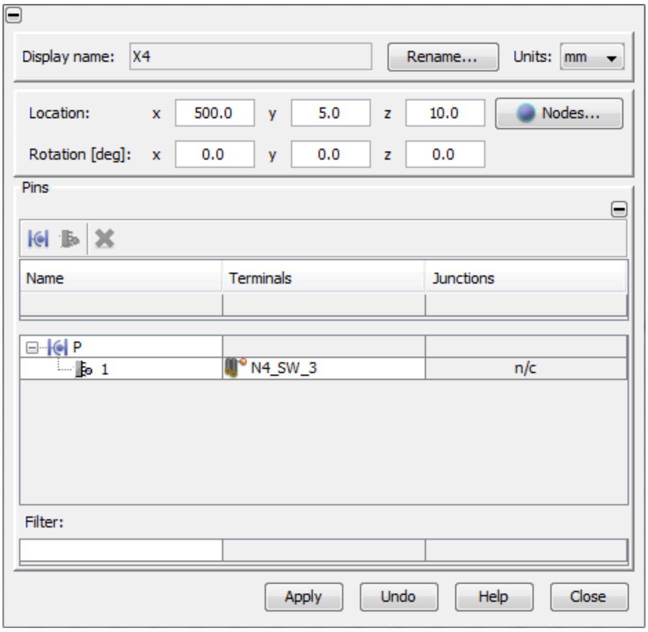
In the following three figures the definition of the other three connectors will be shown:



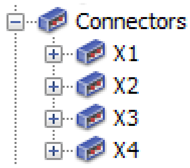
Definition of connector *X3*:



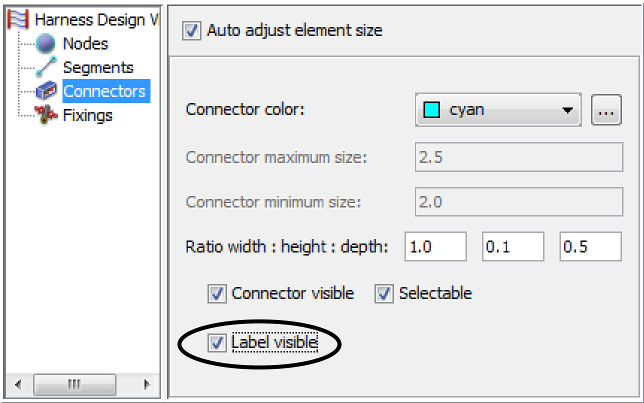
Definition of connector *X4*:



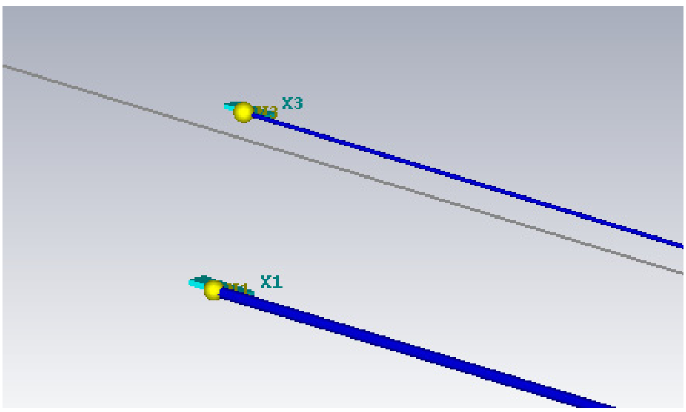
The defined connectors should be listed in the *Cable Navigation Tree* as shown in the figure below:



Next we select *Cables: Edit Cabling* ⇨ *Properties* ⇨ *Harness View Attributes* and change the settings for *Connectors* as shown in the next figure in order to display the connector labels:

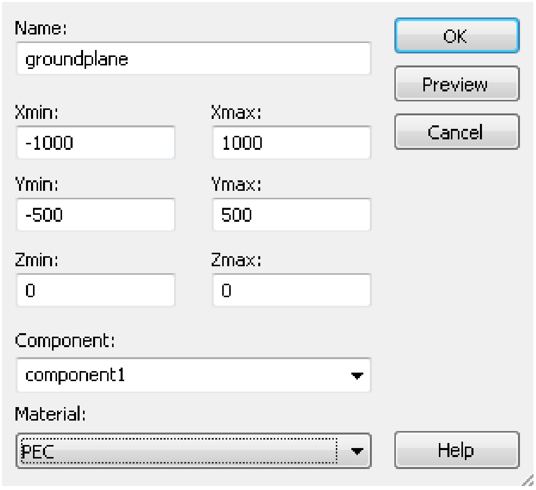


If we zoom into the left side of the harness we get a view as shown in the figure below:

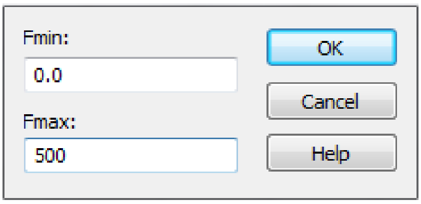


By default, the connectors are displayed with the color *cyan*. We notice that the cable bundle *R1* appears thicker than the cable bundle *R2*. This is due to the fact that there are two wires in the bundle *R1* but only one wire in the bundle *R2*.

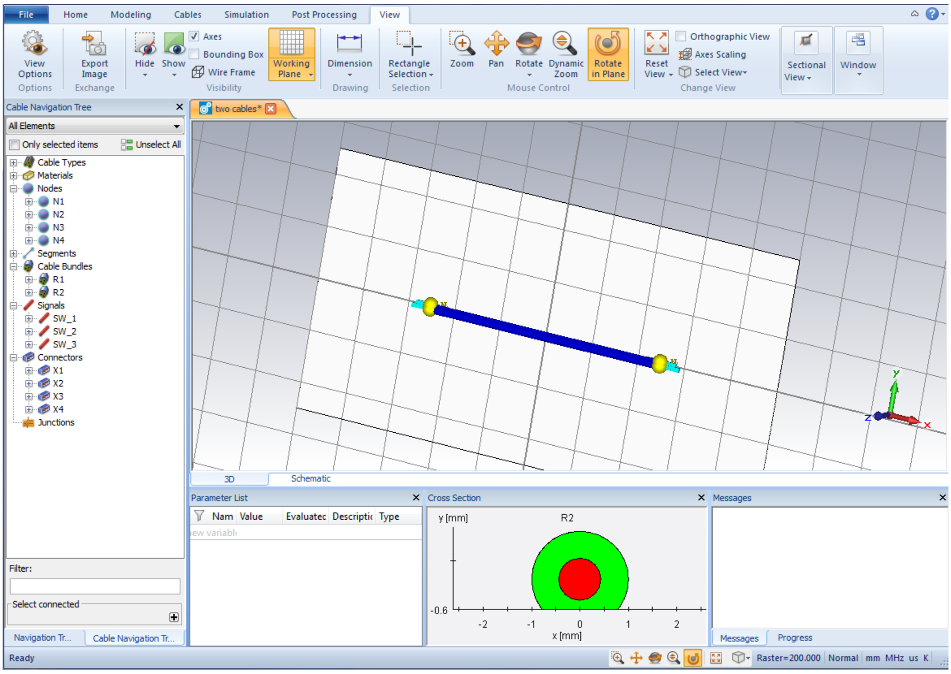
To finish the harness definition we have to add a ground plane. We select *Modeling: Shapes*⇨ *Brick*. After pressing *ESC* the following dialog box will appear where we complete the settings as shown below:



As a last step we change to *Simulation: Settings*⇨ *Frequency* and set *Fmax* to *500 MHz* as shown in the figure below:

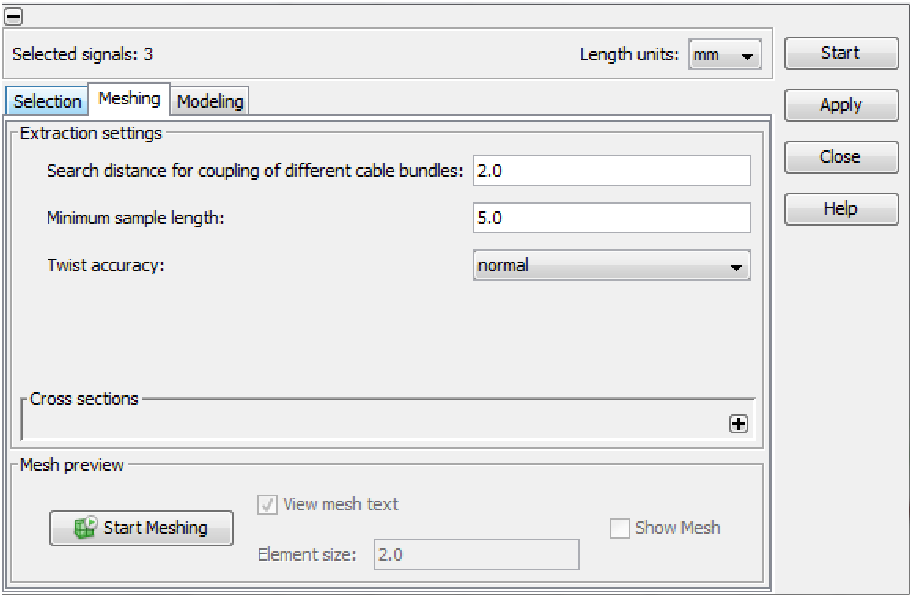


The structure should now look like the figure below:



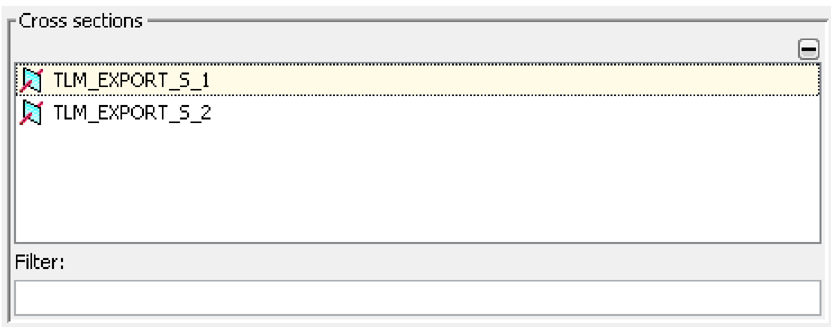
Meshing and Simulation with two Different Search Distances

To generate a first model we select *Cables: 2D (TL) Modeling*. The following dialog box appears. In the Meshing tab we set *Search distance for coupling of different bundles* to *2 mm*.



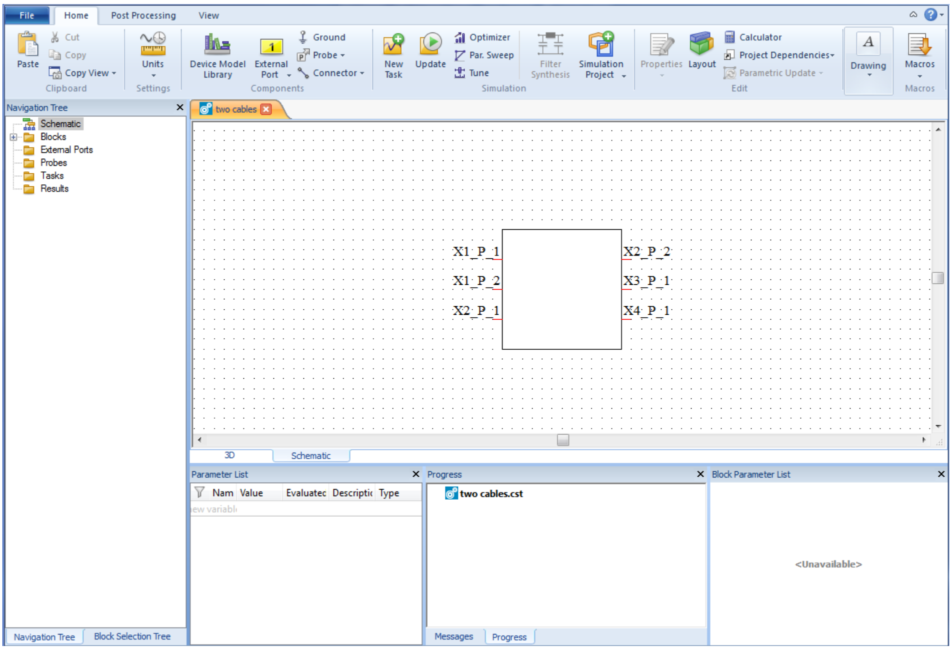
We change to the Modeling tab and see that the value for "*Model valid up to frequency*" is set to *500 MHz*, according to the *Fmax* setting in *Simulation: Settings*⇨ *Frequency*. Now we press the *Start* button on the upper right side

Afterwards we go back to tab Meshing and expand the *Cross sections* frame:

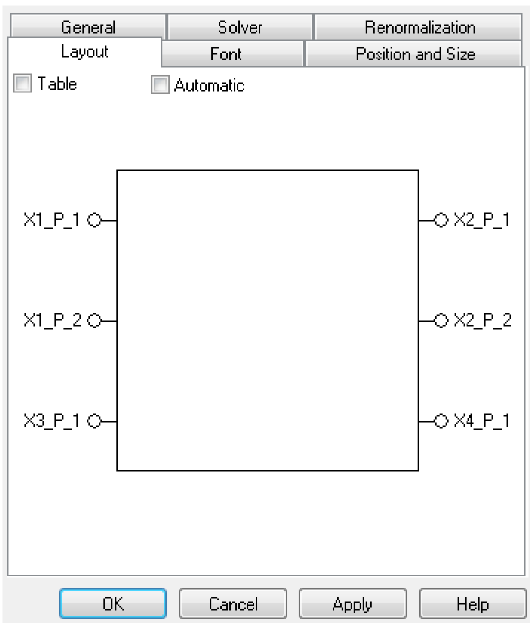


There are two separate cross section items in the list. If we *select* an item the corresponding cross section appears in the *Cross Section* window. Each cable bundle was modeled separately, and this means there won't be any coupling between the cable bundles.

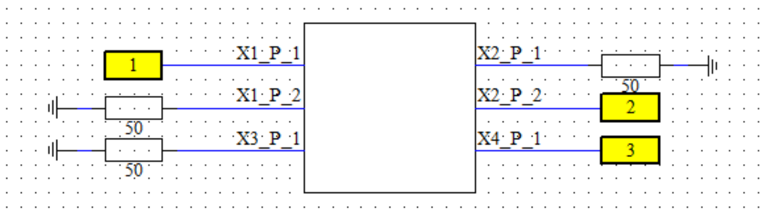
Now let’s change to the *Schematic* tab and set up an S-Parameter-task.



By looking at the pin names of the schematic block you will notice the effect of the connector definition. First, we re-arrange the pin positions. In order to do this, we select the schematic block and choose *Properties* by using the right mouse button. A new dialog box will appear where we select the *Layout* tab. We re-arrange the pins by drag-and-drop as shown in the figure below:

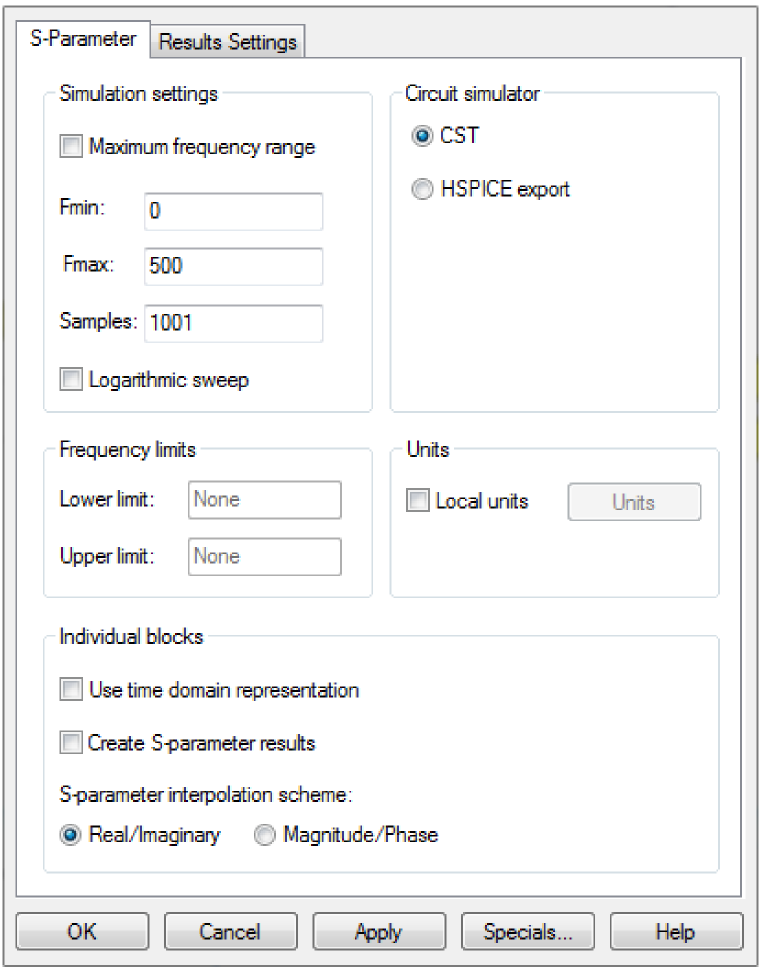


We press OK and add three external ports and three *50 Ohm* resistors to the newly arranged schematic block as shown in the figure below:

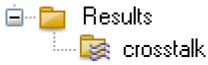


Note: connecting a resistor between a pin and the ground symbol does make sense in this case, because the equivalent circuit behind the schematic symbol includes the information of an existing reference conductor (the ground plane).

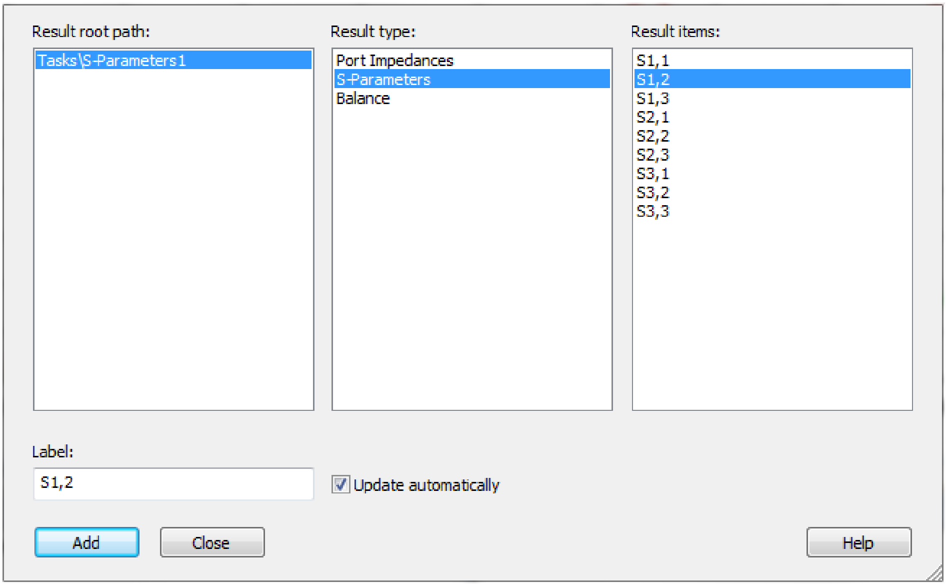
We are now able to set up the S-Parameter task. We select *Navigation Tree*: *Tasks*⇨ *New Task* by using the right mouse button. In the dialog box we select *S-Parameters* and press *OK*. In the appearing dialog box we set the parameters according to the figure below:



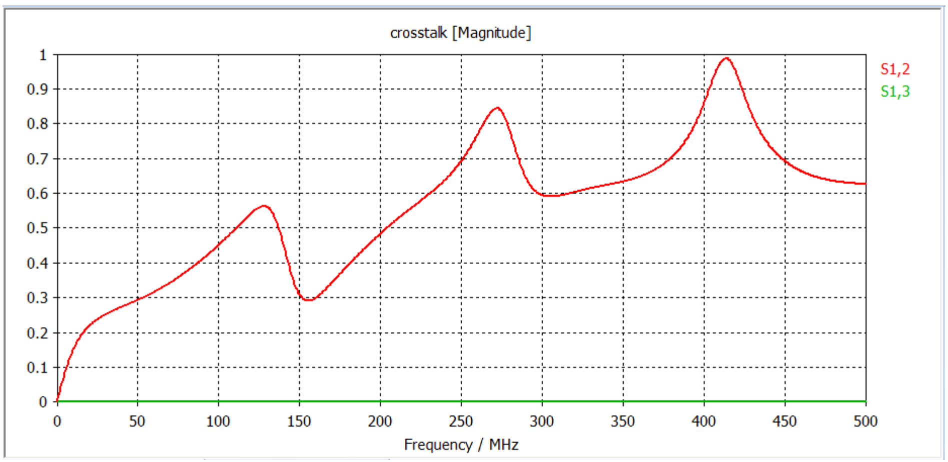
Now we start the simulation by selecting *Home: Simulation ⇨ Update*. After a few seconds the task will be completed. To show the results on port *2* and port *3*, we create a new result folder with name *crosstalk* as shown below:



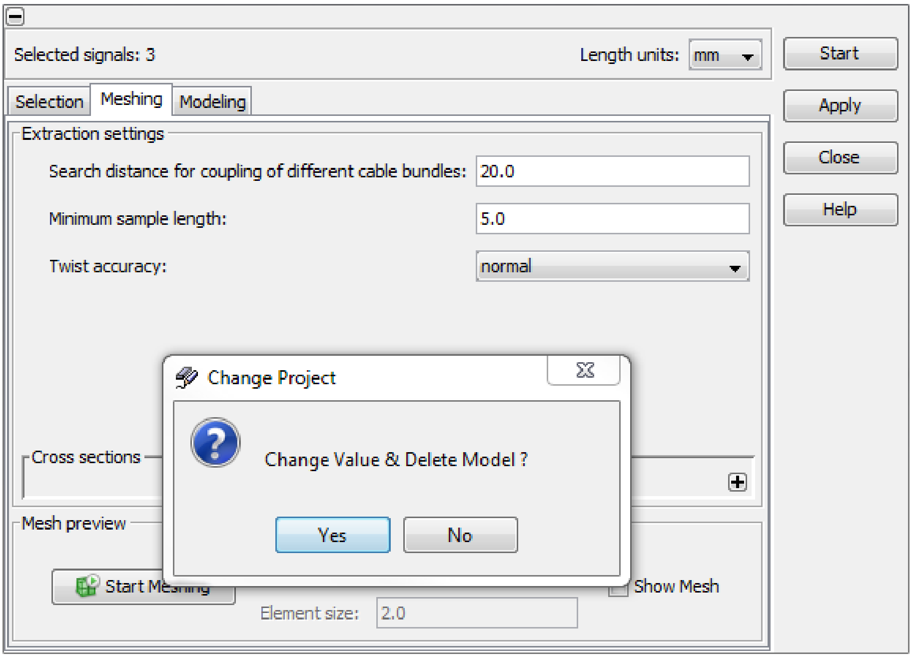
By selecting *crosstalk* and choosing *Manage Results* (using the right mouse button) you will get the following dialog box where you add *S1,2* and *S1,3* to the result folder:



After closing the dialog box you will see crosstalk on pin *X2_P_2* but no cross talk on pin *X4_P_1* as expected.

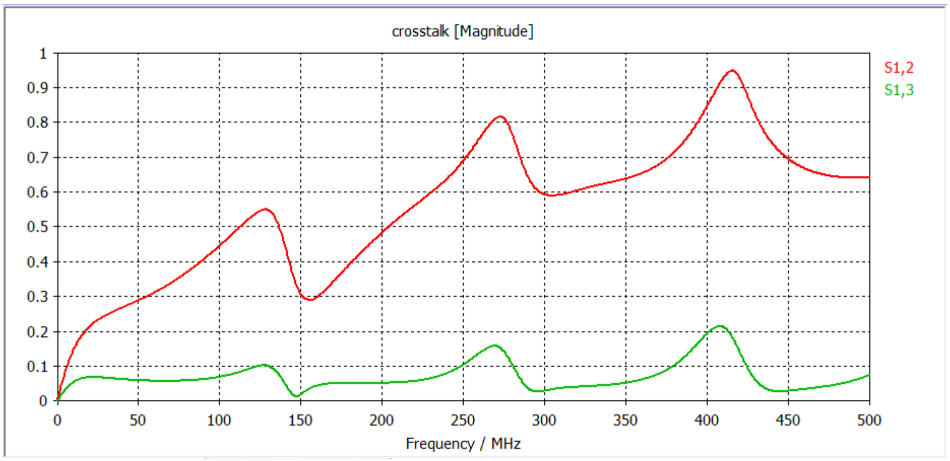


In a second step we go back to the *3D* tab and set the *Search distance for coupling of different cable bundles* to *20 mm* and press *Apply*.

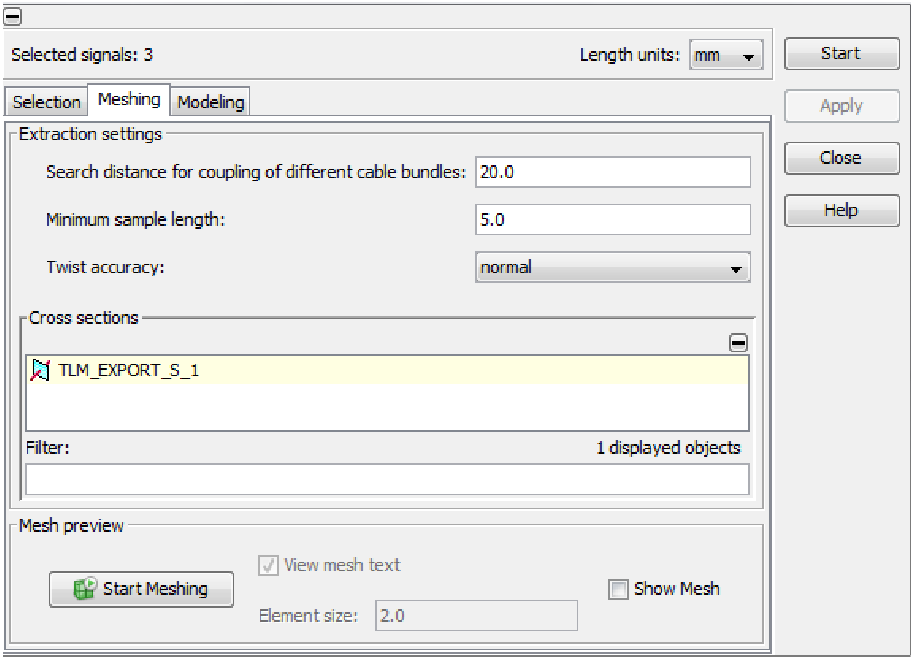


We will be prompted to confirm the change. We press *Yes* and then immediately press the *Start* button again.

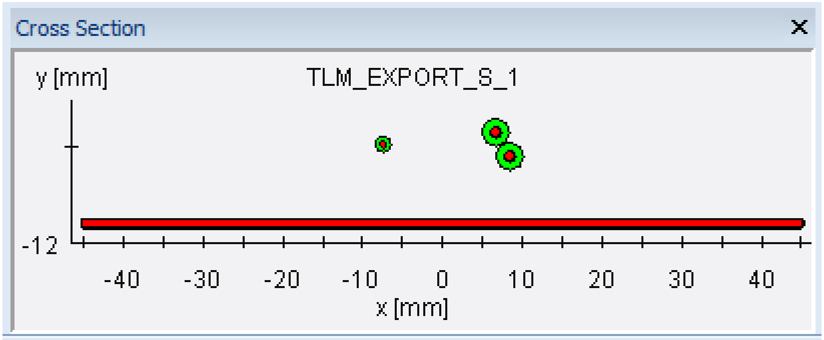
After the modeling has completed, we change to the *Schematic* tab and start the simulation task again. After the simulation has completed the result can be seen by simply selecting the result folder *crosstalk*. As expected, there is now also coupling into pin *X4_P_1*.



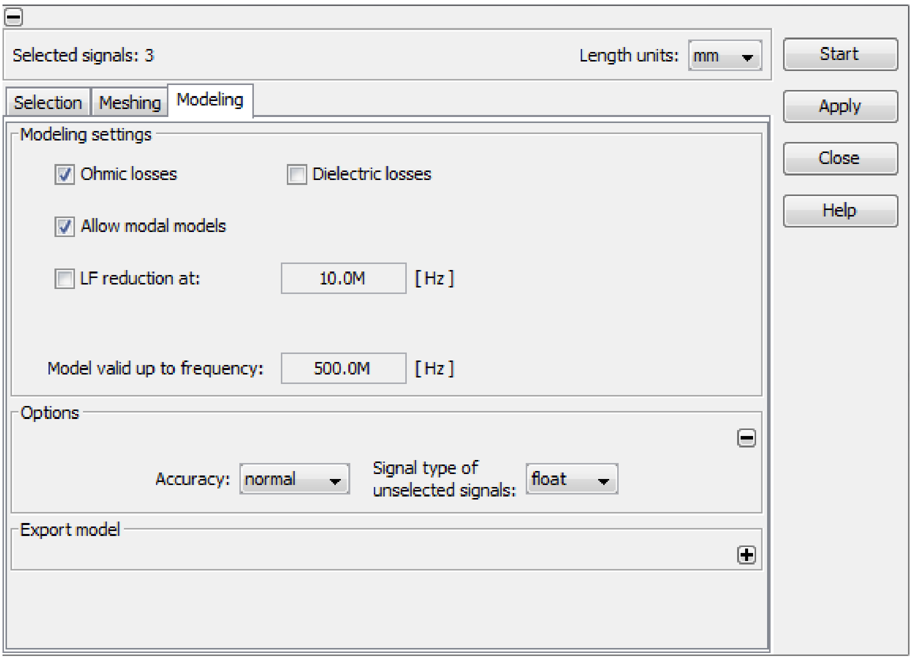
In order to see the reason for this, we change back to the 3D tab and reopen the 2D Meshing tab (inside *Cables: 2D (TL) Modeling)*. In the *Cross Sections* we see a single item as shown in the figure below:



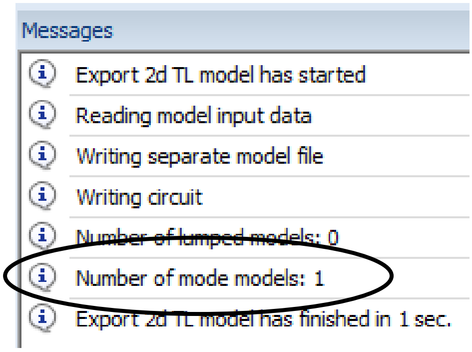
By selecting the item and looking at the *Cross Section* window we see that there are three wires inside the cross section, and this means both cable bundles have been coupled during the modeling:



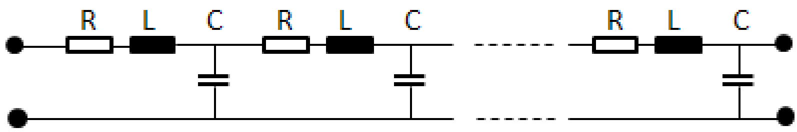
We want to finish the example by going back to the *3D* tab and selecting *Cables: 2D (TL) Modeling* once again. We check *Allow Modal Models* in the *Modeling* tab and after pressing *Apply* we again are prompted to confirm the change. We press *Yes* and then we press *Start*.



After the modeling has completed we have a look in the *Message* Window and see a **modal model** has been generated instead of a **lumped model**:



The difference between *lumped* and *modal* can be best explained using a simple, single transmission line. Using the *lumped modeling approach* a transmission line is approximated by a series of discrete (or lumped) *R, L, C* devices as shown in the next figure:



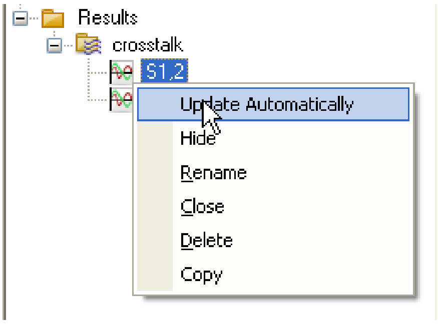
Each *R*-*L*-*C*-combination models a short section of the transmission line. The valid frequency range for the whole model is therefore limited by the length of this unit because the length of the section must be considerably smaller than the shortest wavelength of the propagating signal.

The advantage of the *lumped approach* is its flexible usage inside a circuit simulation and its suitability for modeling non-uniform transmission lines like twisted pairs. Disadvantages arise when dealing with overall lengths of transmission lines much larger than the wavelength of its transmitted signal. In this case the number of necessary section is large and this causes a large number of lumped elements inside the equivalent circuits.

Using the *modal modeling approach* a transmission line is described by its secondary transmission line characteristics like *wave impedance Z* and *propagation delay* τ. The size of the model does not depend on the length of transmission line or on the maximum frequency and this is a big advantage when dealing with long uniform transmission lines. Using this approach for non-uniform transmission lines like twisted pairs requires some simplifications by the program and this may slightly influence the accuracy. The *modal approach* can't be used if the models use hybrid field-to-cable coupling.

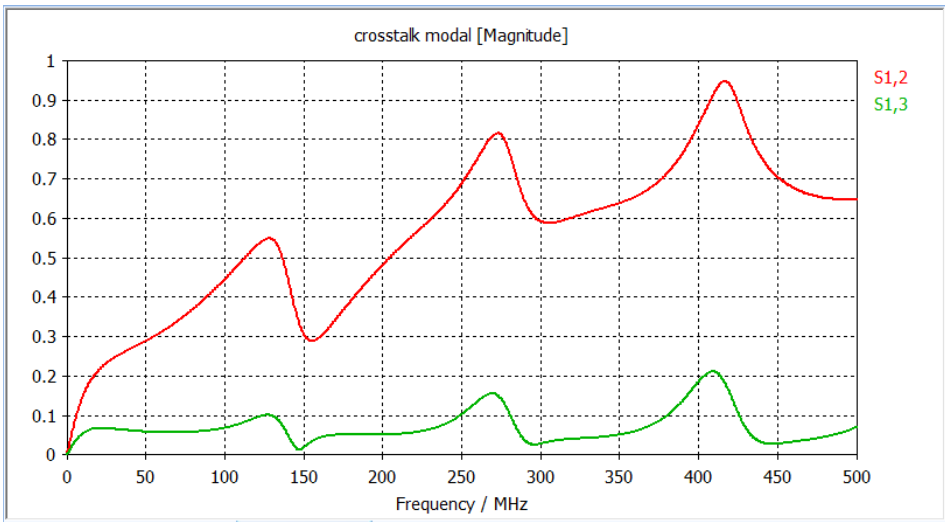
When checking *Allow Modal Models* the program automatically looks for electrically long sections along the cable assembly and models these sections by *modal models* instead of *lumped models*. Note: electrically long means a length that is considerably longer than the wavelength for the maximum frequency set in the same dialog box.

To see how these *modal models* work we change to the Schematic tab. Before starting a new simulation we first save the results of the previous run. In order to do this, we select the existing result curves in folder *crosstalk* and uncheck *Update Automatically* by using the right mouse button (do this for both curves).



Now we run a new simulation and notice that the simulation takes less time than before. After completion of the simulation we create a new result folder and name it *crosstalk modal* as in the figure below:

We add curve *S1,2* and S*1,3*, compare it with the results in folder *crosstalk* and notice no considerable difference:



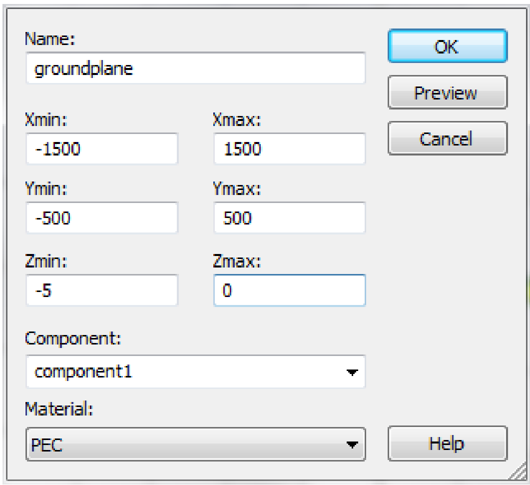
Field Coupling from and into a Twisted Pair

The aim of this example is to acquaint you with the

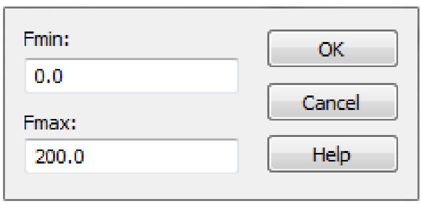
- hybrid method for radiation (*current substitution method*)
- hybrid method for irradiation (*field substitution method*)
- difference between a balanced and un-balanced termination on a twisted pair

Cable Definition

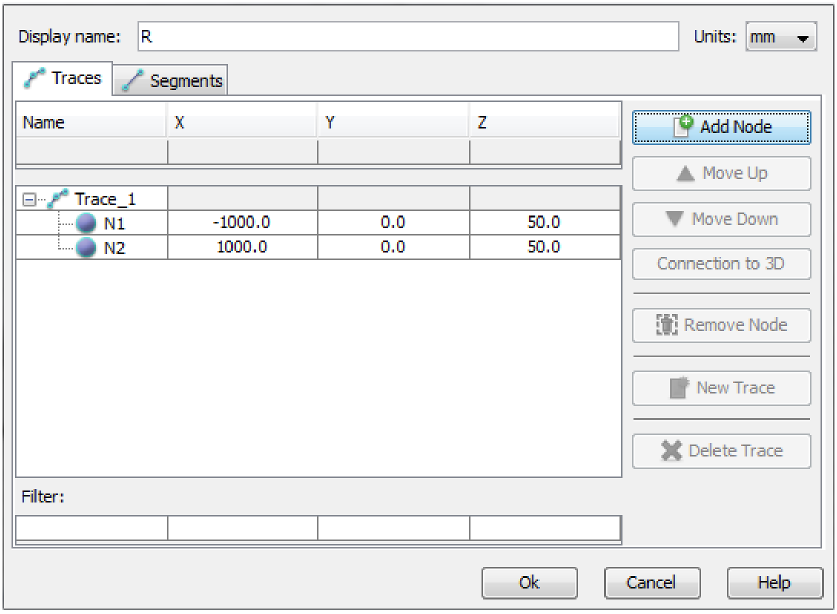
In this chapter we want to set up a simple configuration of a straight twisted pair cable of length *2 m* placed over a ground plane at a distance of *50 mm*. We create an empty project and save it as **twisted pair**. The geometric units are left to default. We start with the definition of the ground plane as shown in the dialog box below:



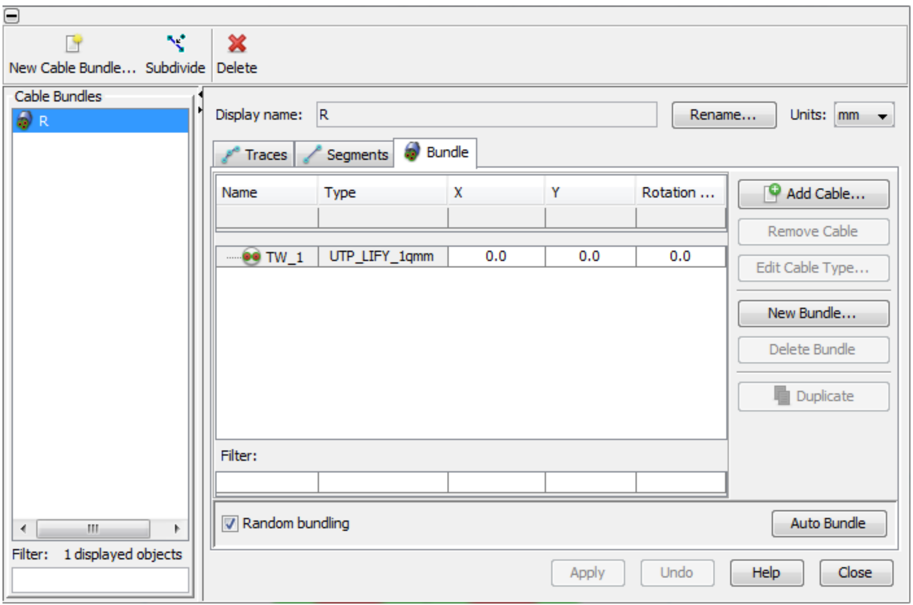
Next, we change to *Simulation: Settings*⇨ *Frequency* and set *Fmax* to *200 MHz* as shown in the figure below:



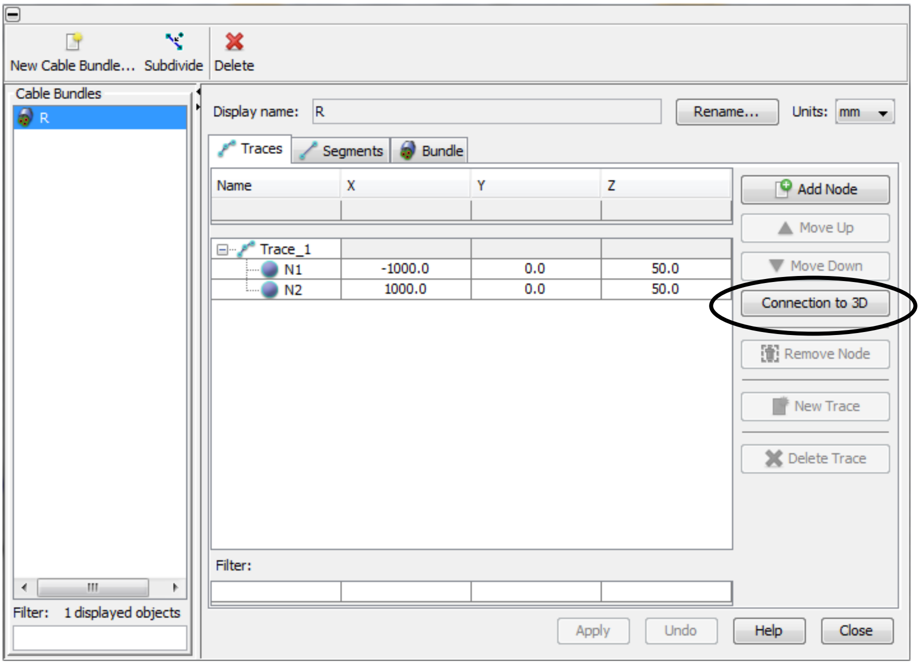
Next we define a cable between nodes *N1* and *N2* as shown in the dialog box below:



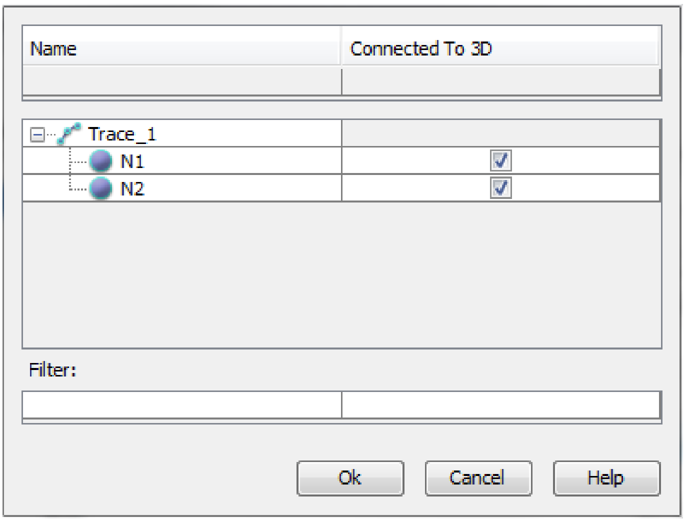
We press *Ok* and see the *Cable Bundle* dialog box where we select the *Bundle* tab and press the *Add Cable* button. We load the standard twisted pair cable from the library and afterwards press *Apply*.



In the following examples we want to connect both cable ends with the ground plane by using capacitors. In order to inform the 3D field solver of such a connection of the cable nodes to the 3D objects, both cable ends have to be marked in a special way. We change to the *Traces* tab of the dialog box and press the *Connection to 3D* button:

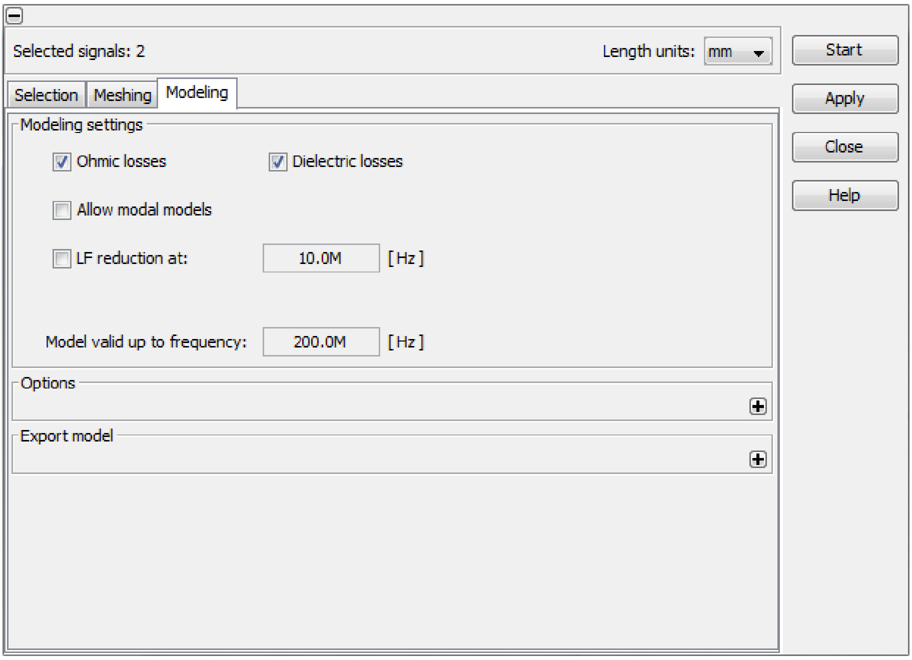


A further dialog box appears which lists both end nodes of the cable bundle.



We see that both nodes are prepared for a connection with the ground plane by default. Note: Nodes prepared for a 3D connection appear in yellow color inside *Main View*, all other nodes will be displayed in blue color.

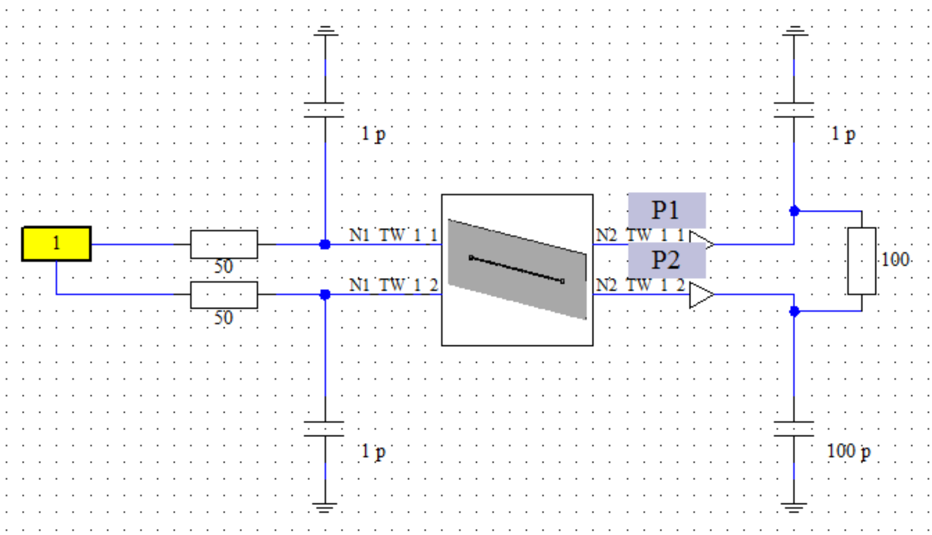
We close the *Enable Connection to 3D* dialog box, open the *2D (TL) Modeling* dialog box and activate *Ohmic losses* and *Dielectric losses* as shown in the figure below:



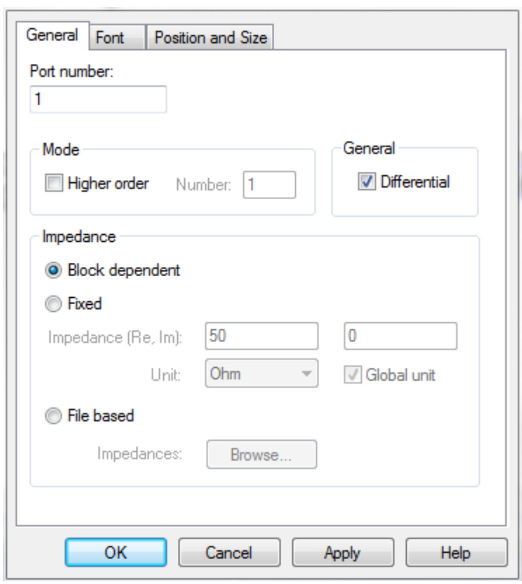
After pressing *Start* the modeling process will be completed within a few seconds.

Hybrid Method for Radiation from a Cable

To simulate the radiation of the cable we now change to the *Schematic* tab and see the generated schematic symbol. We set up a schematic as shown in the figure below. On the left side of the twisted pair we add two capacitors (both with a value of *1 pF*), two resistors (both with value of *50 Ohm*) and a differential port.

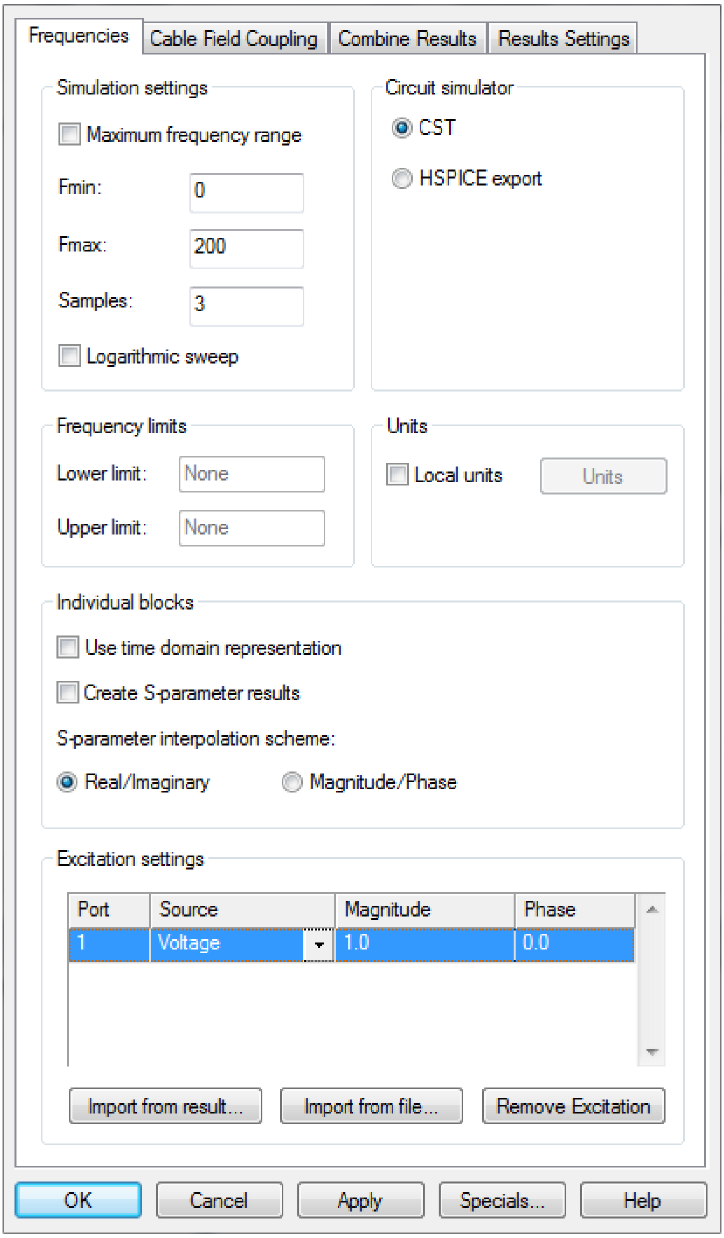


In order to turn a port into a differential port, we right mouse click on the port, select *Properties* from the pull-down menu and check the *Differential* flag as shown in the figure below:

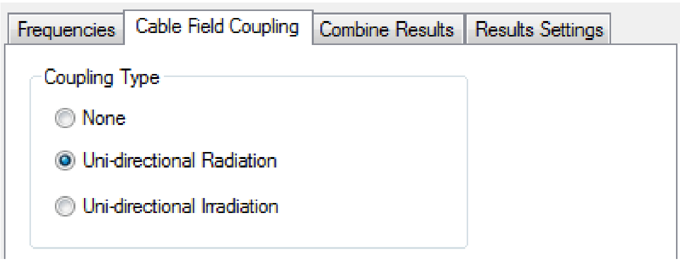


On the right side we add a resistor of *100 Ohm* and two capacitors with different values. The upper has a value of *1 pF*, but the lower has a value of *100 pF*. This difference causes an unbalance that will affect the radiation result. To finish the schematic we add two probes, *P1* and *P2*.

Next we create an AC-task and choose three frequency steps as shown in the next dialog box. The 3D field solvers are indeed able to perform a calculation in a broad frequency range, but since we are only interested in a field distribution plot at a single frequency point, we confine the number of frequencies. In addition, we define a voltage source inside the *Excitation settings* frame as shown in the figure below:

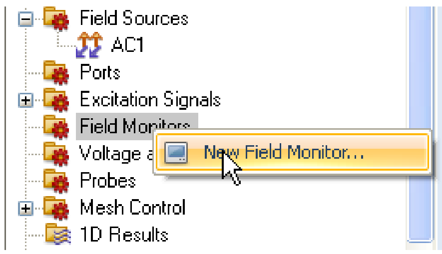


Before we start the simulation we have to prepare the circuit simulation to write out the common mode current along the cable path for the 3D field calculation. Therefore we select the *Cable Field Coupling* tab of the dialog box and select *Uni-directional Radiation* as shown in the figure below:

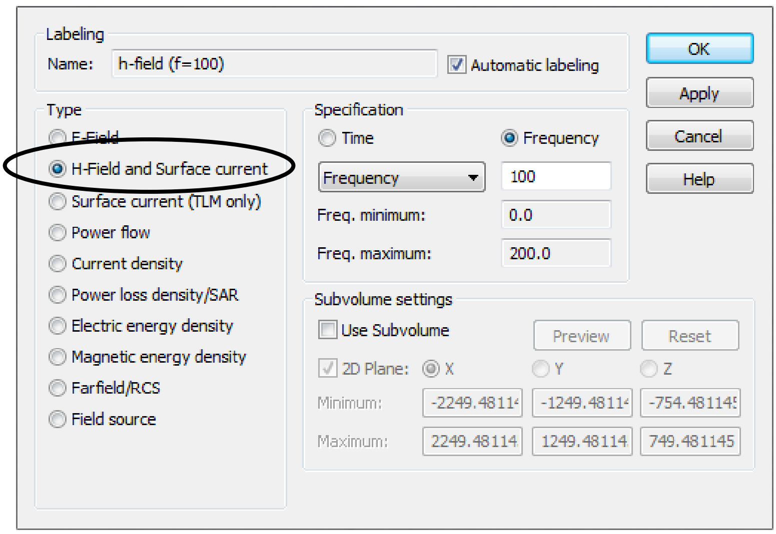


We press *OK* and change to the 3D-tab and see that *Units*, *Background Material*, *Boundary Conditions* and *Frequency* are already set.

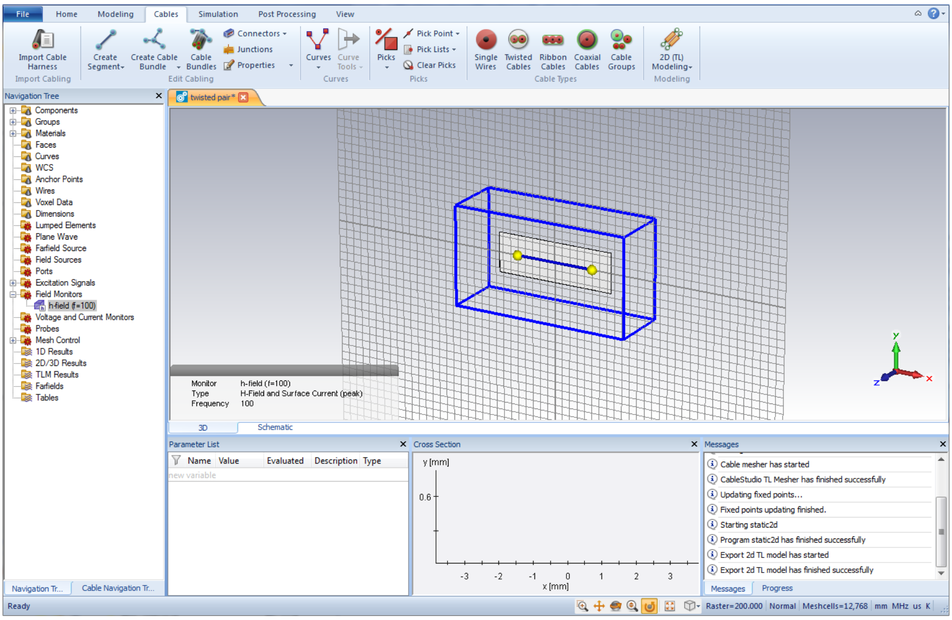
In order to see a distribution of the radiated field, we have to set a field monitor. We select *Navigation Tree*: *Field Monitors* and choose *New Field Monitor* by using the right mouse button:



A dialog box will appear where we select monitor type *H-Field/Surface current*:

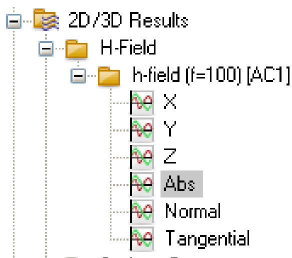


We leave the *Frequency* at the proposed value of *100* MHz, and press *OK*. We now see a field monitor item in the *Navigation Tree* and also a corresponding frame in the *Main View*:



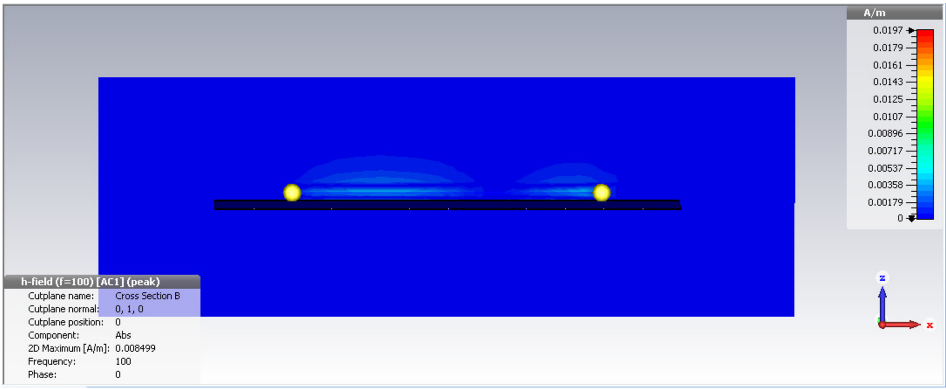
Next we again change to the Schematic tab and start the simulation by pressing the *Update* button.

To display the radiated magnetic field we change back to the 3D-tab, select *Navigation Tree*: *2D/3D Results* ➪ *H-Field* ➪ *h-field (f=100)[AC1]* and finally select item *Abs* as shown in the figure below:

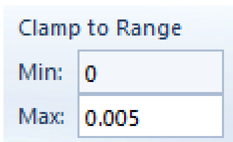


To see the magnetic field distribution on a 2D plane we go to the ribbon *2D/3D Plot* and inside *Sectional View* we set the parameter *Normal* to *Y*. In addition, we select *3D Fields on 2D Plane* and select *Cross Section B* in the pull-down menu.

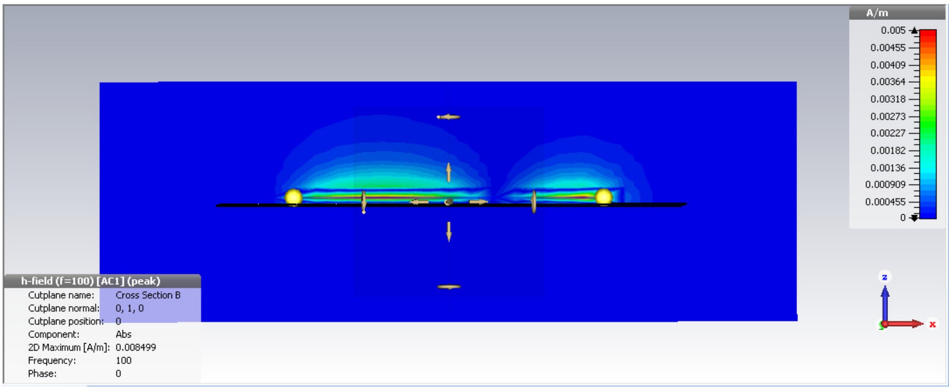
After rotating the whole structure according to the figure below, we see the following magnetic field distribution:



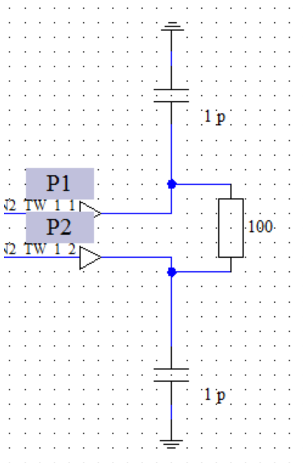
In order to adjust the scale of the displayed field distribution, we go to the ribbon *2D/3D Plot: Solver ⇨ Color Ramp and Scaling* and clamp the minimum and maximum value of the magnetic field to the values depicted below:



The magnetic field distribution will now look like in the figure below:

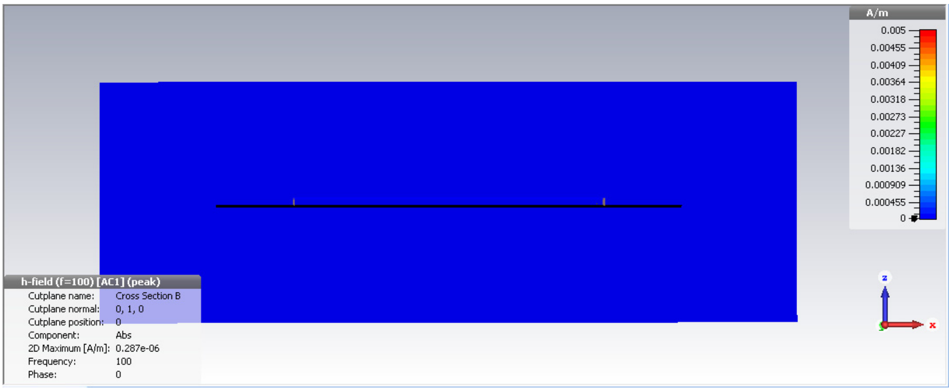


To show the influence of a better balanced termination on the twisted pair, we change to the Schematic tab and modify the *100 pF* capacitor to *1 pF*:



Now the termination is fully balanced and we repeat the whole simulation flow by updating the AC-task once more.

After a few seconds the simulations will be complete. Again we plot the magnetic field distribution and clamp the amplitude to *0.005 A/m* as was done before. We see the magnetic field distribution has vanished:

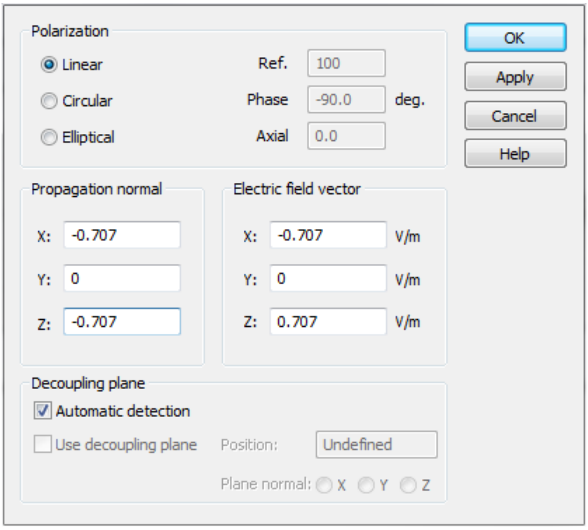


We have now seen how the signals inside a cable and the termination on the cable's ends can influence the 3D field around it. In the next section we will investigate the influence of the 3D field on the cable signals.

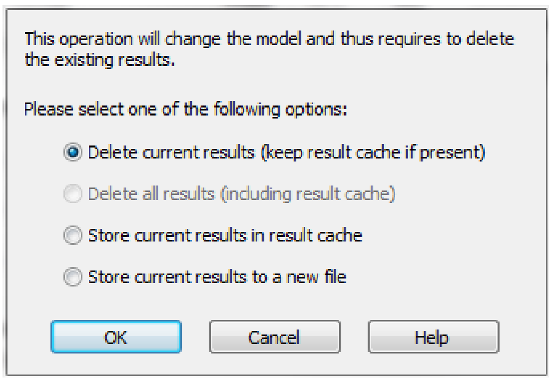
Hybrid Method for Irradiation into a Cable

To see how external fields can be coupled into the existing twisted pair cable, we first create a plane wave excitation in CST CABLE STUDIO. In order to do this, we go into the *Navigation Tree*, close the *2D/3D Results* folder, select the *Plane Wave* folder and choose *New Plane Wave* by using the right mouse button.

In the following dialog box we change the settings as shown in the figure below:

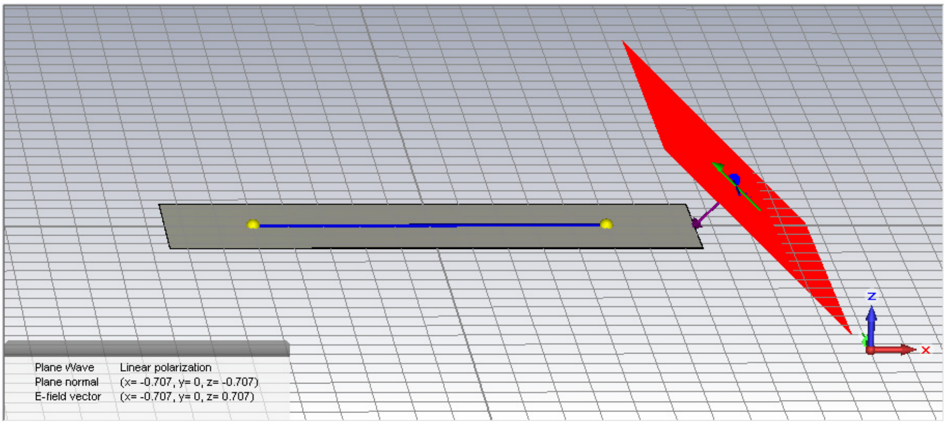


After pressing *OK* we are prompted to confirm the deletion of the old results:

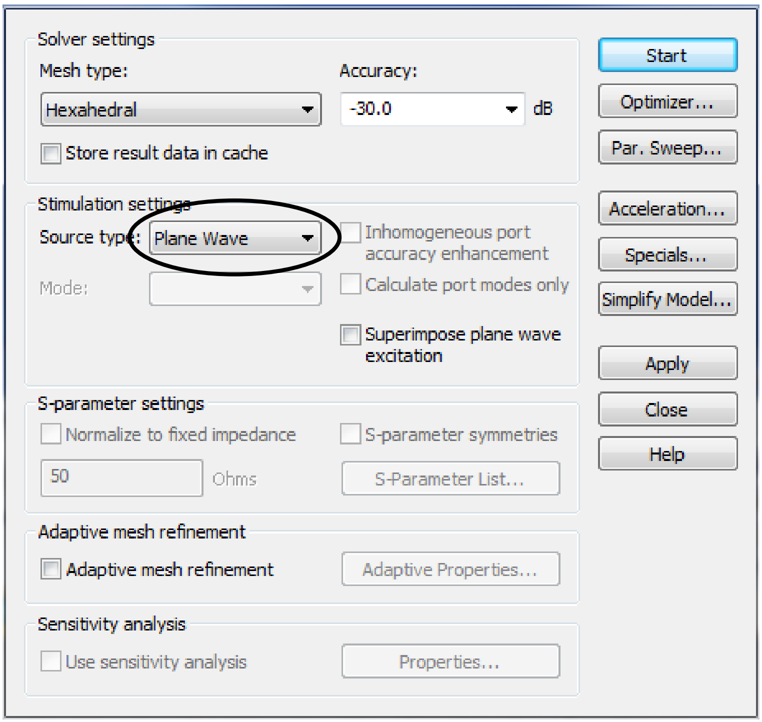


We press *OK* and select the new plane wave item in the *Navigation Tree*.

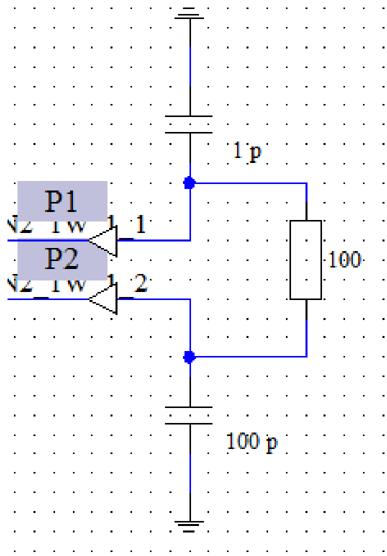
After doing so you will see the plane wave in the *Main View*:



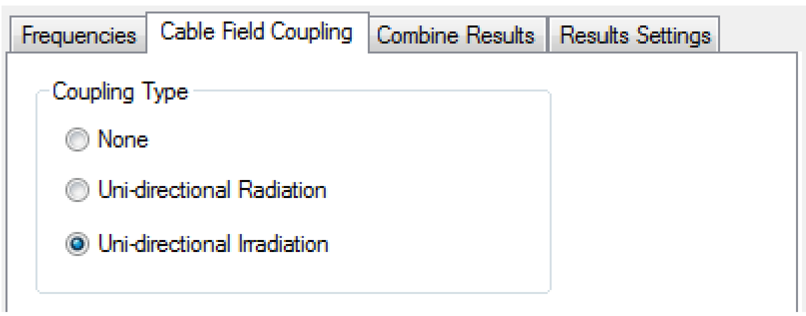
Next, we call the *T-Solver* dialog box (by selecting *Home: Simulation* ⇨ *Setup Solver*) and select *Plane Wave* as *Source type* as shown in the figure below:



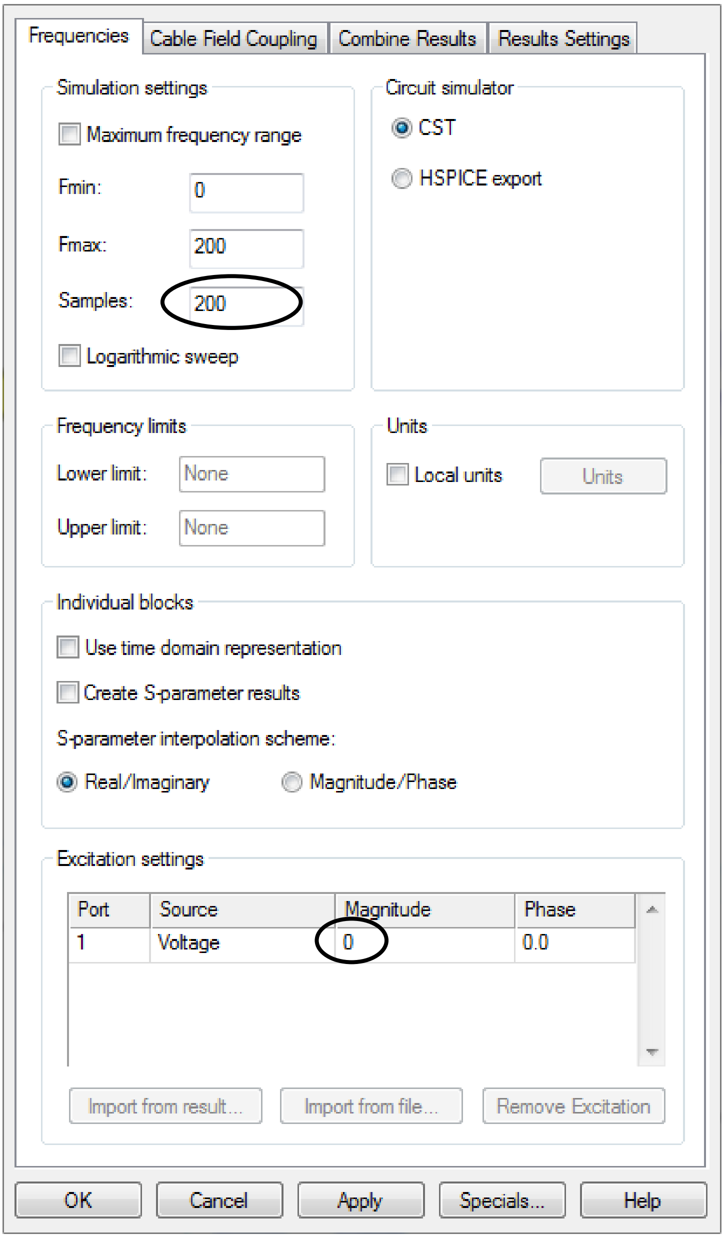
Next, we press *Apply*, close the dialog box and change to the *Schematic* tab where we make three modifications. First, we change the value of the lower capacitor to *100 pF* again:



Next, in order to force the 3D solver to calculate the induced voltages along the cable path, we open the dialog box of the AC-simulation task, select the *Cable Field Coupling* tab and choose *Uni-directional Irradiation* as shown in the figure below:



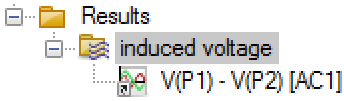
Finally, we change to the *Frequencies* tab and make two more changes: we set the number of frequency steps to *200* and the value of the voltage port to *0* (zero) as shown in the figure below:



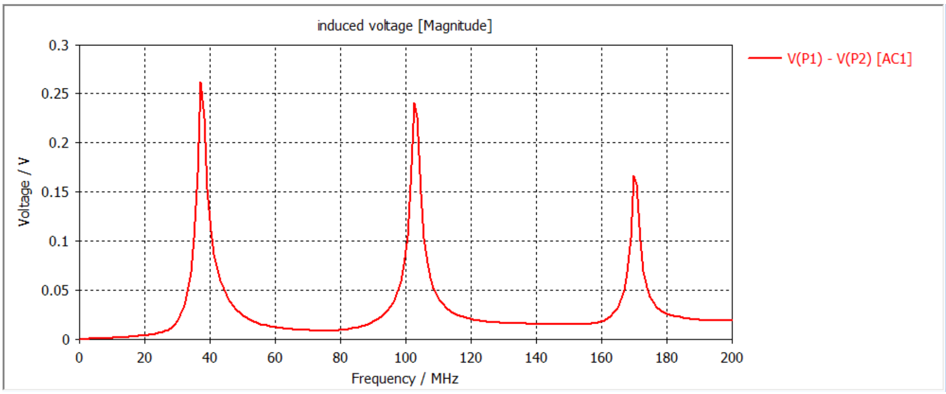
Now we press *OK* and start the simulation. We notice that the 3D field simulation is started first, followed by the circuit simulation with the AC task. The whole sequence takes only a few seconds.

In order to display the induced voltage on the right side of the twisted pair cable, we create a new result folder with name *induced voltage* (by selecting *Navigation Tree*: *Result* and choosing *Add Result Plot* with the right mouse button). Then we add the voltage difference between probes *P1* and *P2* (by choosing *Manage Results* with the help of the right mouse button, selecting both curves in the appearing dialog box and finally pressing *Add*).

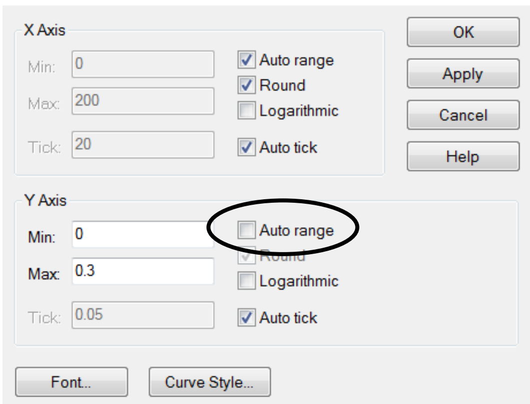
After doing this the result folder should look like in the figure below:



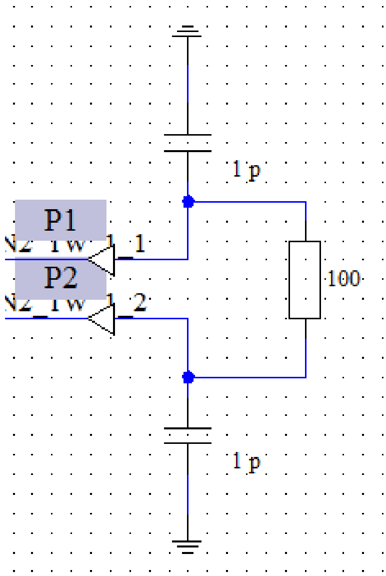
We will get the following result:



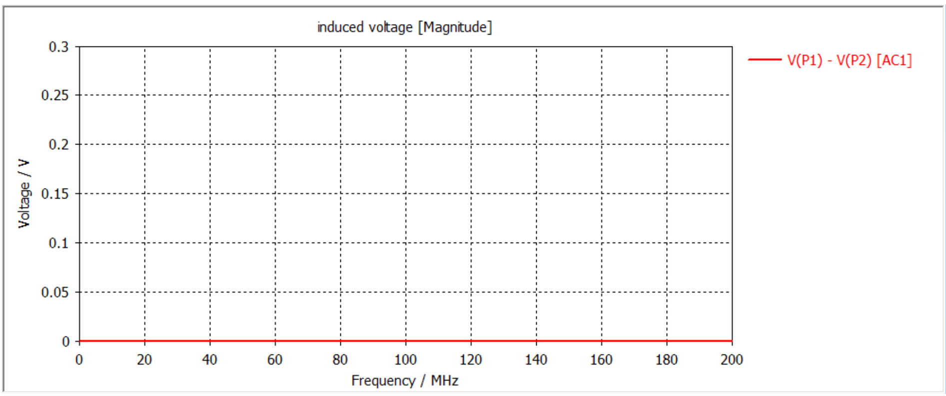
Next, we force the scaling to fixed values. In order to do this, we go with the mouse pointer into the *Main View* and choose *Plot Properties by pressing the right mouse button*. In the dialog box we uncheck *Auto range* of the *Y Axis* field as depicted in the figure below and press *OK*.



In order to see the changes when we balance the termination network on the right side of the cable, we will now change the *100 pF* capacitor to *1 pF*:



After a new simulation run the result will change to a zero line:



Chapter 4 – General Methodology

CST CABLE STUDIO is designed for ease of use. However, to work with the tool in the most efficient way the user should know the principal method behind it. The main purpose of this chapter is to explain the theoretical concepts and the constraints on its use.

The central method of CST CABLE STUDIO is based on classical *transmission line theory*. In this method the geometric and material characteristics of a cable are transformed into an equivalent circuit that can be simulated in time and frequency domains.

Standard Workflow

In the first step a complex cable harness is divided into a finite number of straight segments. For each segment the program checks for any metallic shapes surrounding the cables. All cables in a segment, in combination with additional metallic shapes, define its cross-section. This whole process is called *Meshing*.

In the second step the primary transmission line parameter per unit length (*R’, L’, C’, G’*) is calculated from each segment by a static 2D field solver. Afterwards each segment will be transformed into an equivalent circuit. Finally all circuits will be connected together into one single electrical model representing the whole cable. This process is called *Modeling*.

The second step implies that only **TEM propagation modes** can be considered and this fact causes the following constraints which are described below:

- *TEM* propagation mode means that there are at least two separate conductors necessary to enable one single propagation mode (to enable forward and return current). In general *N* conductors are necessary to enable *N-1* propagation modes. One single wire inside open space without any reference (typical antenna structure) won’t be modeled correctly for a frequency higher than *DC*.

- The generated equivalent circuits are only valid within a frequency range from *DC* to *fmax*. This is due to the fact that the primary transmission line parameters are static parameters and only valid if the geometric dimensions behind the calculation are significantly smaller than the shortest wavelength of the propagating wave.

- Discontinuities like bends, deviations or cable ends will not be considered when using the standard workflow.

In the third step the electrical model of the cable can be further processed in the *Circuit Simulation*. To make this possible the model will be automatically transferred to a circuit simulator where the user is able to define several loadings (passive/active, linear/non-linear) and to calculate the transmission behavior of the cable in time and frequency domains.

Additional Workflow for Uni-and Bi-Directional Cable-to-Field Coupling

Many industrial applications deal with cables inside an additional metallic environment (e.g. ground planes in laboratory set-ups, car chassis). In the presence of such reference conductors one propagation mode is of special interest. This mode is called *common mode* and consists of the sum of the currents of all wires inside the cable bundle and the corresponding return current back through the reference conductor. Significant common mode currents are often the reason for considerable EMI/EMS problems.

If a reference conductor is part of the configuration, the method used by CST CABLE STUDIO is able to calculate the common mode by summing up all currents in the cable bundle during an AC task. The “common mode current” along the cable path can automatically be passed to a 3D full wave solver where it can be used as an impressed field source. This method is called a uni-directional cable-to-field coupling.

If the cable was modeled for *uni-directional coupling* the cable itself is not physically present during the 3D full wave simulation. Because of this, the reaction of the generated field (generated by the impressed current) back to the cable will be neglected. This approach limits the range of applications to configurations where most of the radiated energy will not be scattered back to the cable. This is true for many configurations with cables along open metallic chassis. The assumption is not true in case of a resonant cable inside a nearly closed metallic enclosure. Therefore, when using this *current substitution method* the user has to check if the application fulfills the necessary assumption.

Note: If there is no reference conductor, CST CABLE STUDIO will always predict a common mode current of zero. Any oscillating antenna modes which may exist in the higher frequency range (when dimension of lambda equals length of cable) won’t be considered because the basic method is only able to simulate TEM-modes.

The procedure described above can also be used for considering the *common mode impact* of an external electromagnetic field onto a cable. Here, the 3D full wave solver will calculate the tangential electric field along the cable path (while the cable itself is not physically present). In a next step the solver will automatically converts these values to voltages and finally pass the voltages to a circuit simulator. During an AC-task the voltages can be used to calculate the induced currents on the cable. The limitation of this *field substitution method* is identical to the *current substitution method*.

If the described uni-directional coupling methods are not sufficient, CST CABLE STUDIO also offers the most general method, namely the bi-directional coupling between cable- and field-solver in time-domain. In this case, currents and voltages are exchanged in every time step between the 3D full wave field simulator and the circuit simulator cables and loadings. This method can (and has to) be applied in case of resonating structures, where radiation and irradiation effects act simultaneously.

Chapter 5 – Finding Further Information

After carefully reading this manual, you will have a basic understanding of how to use CST CABLE STUDIO efficiently. 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.

Other Printed Documents

A more detailed introduction into solid modeling and 3D full wave simulation can be found in the *CST MICROWAVE STUDIO – Workflow & Solver Overview* document. You will find it in the same directory as this document.

A more detailed introduction into the circuit simulator can be found in the *CST DESIGN STUDIO Workflow* document. You will find it in the same directory as this document.

Online Reference Documentation

You can access the CST STUDIO SUITE online help system's overview page at any time by choosing *File: Help ⇨ CST STUDIO SUITE-Help*.

Please refer to the CST STUDIO SUITE *Getting Started* manual for more information about how to use the online help system.

Examples

The installation directory of CST STUDIO SUITE contains an examples subdirectory containing some typical application examples. A quick overview of the existing examples can be obtained by browsing through the descriptions of the examples inside the online help system. These examples may contain helpful hints that can be transferred to your particular application.

Access Technical Support

After you have taken your first steps towards solving your own applications in CST CABLE STUDIO, you may send your recent project file (with the extension 'cst') to the technical support team. Even after your problem has been solved further investigation of the project may lead to better and faster results.

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

History of Changes

The history of changes between several releases of the program is also available via the online help system. Since there are many new features in each new version, you should browse through the list even if you are already familiar with one of the previous releases.