

 **CST EM STUDIO**

**Workflow &  
Solver Overview**

## **Copyright**

© CST 2002–2015

CST – Computer Simulation Technology AG

All rights reserved.

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



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

## **Trademarks**

CST, CST STUDIO SUITE, CST MICROWAVE STUDIO, CST EM STUDIO, CST PARTICLE STUDIO, CST CABLE STUDIO, CST PCB STUDIO, CST MPHYSICS STUDIO, CST MICROSTRIPES, CST DESIGN STUDIO, CST BOARDCHECK, PERFECT BOUNDARY APPROXIMATION (PBA), and the CST logo are trademarks or registered trademarks of CST in North America, the European Union, and other countries. Other brands and their products are trademarks or registered trademarks of their respective holders and should be noted as such.

CST – Computer Simulation Technology AG  
[www.cst.com](http://www.cst.com)

# Contents

<b>CHAPTER 1 — INTRODUCTION .....</b>	<b>3</b>
Welcome.....	3
How to Get Started Quickly .....	3
What is CST EM STUDIO?.....	3
Who Uses CST EM STUDIO?.....	4
CST EM STUDIO Key Features .....	5
General.....	5
Structure Modeling .....	5
Electrostatic Solver.....	6
Magnetostatic Solver .....	6
Stationary Current Solver .....	7
LF Frequency Domain Solver .....	7
LF Time Domain Solver .....	8
CST DESIGN STUDIO View .....	8
SAM (System and Assembly Modeling).....	8
Visualization and Secondary Result Calculation.....	9
Result Export .....	9
Automation .....	9
About This Manual.....	10
Document Conventions .....	10
Your Feedback .....	10
<b>CHAPTER 2 — SIMULATION WORKFLOW .....</b>	<b>11</b>
The Structure .....	11
Create a New Project.....	12
Open the QuickStart Guide.....	14
Define the Background Material .....	15
Model the Structure .....	15
Define Coils .....	20
Define Boundary Conditions .....	26
Define Symmetry Conditions .....	28
Generate and Visualize a Tetrahedral Mesh .....	29
Run the Tetrahedral Magnetostatic Solver .....	33
Analyze the Results of the Tetrahedral Solver .....	36
Visualize a Hexahedral Mesh .....	41
Start the Hexahedral Solver .....	43
Analyze the Results of the Hexahedral Solver .....	45
Create a Planar Mesh.....	49
Start the Planar Solver .....	50
Analyze the Results of the Planar Solver .....	52
Accessing the Single-Value Results .....	55
Parameterization and the Automatic Optimization of the Structure .....	57
Summary .....	66
<b>CHAPTER 3 — SOLVER OVERVIEW .....</b>	<b>67</b>
Solvers and Sources .....	67
Magnetostatic Solver .....	70
Nonlinear Materials .....	70
Inductance Calculation .....	70
Current or Voltage Coils .....	70
Coil Segments .....	72
Permanent Magnets .....	74
Current Paths .....	74

Homogeneous Magnetic Field .....	75
Electrostatic Solver .....	76
Open Boundaries.....	76
Potential Sources .....	76
Field Grading .....	77
Charge Sources.....	78
Boundary Potentials.....	79
Stationary Current Solver .....	79
Parameterized Electrical Conductivity .....	79
Current Ports .....	80
Electrical Contact Impedance .....	80
LF Frequency Domain Solver .....	83
Full Wave and Magnetoquasistatic Simulator.....	83
Voltage Paths .....	83
Lumped Network Elements.....	84
Electroquasistatic Simulator .....	85
LF Time Domain Solver .....	86
Magnetoquasistatic Simulator.....	86
Electroquasistatic Simulator .....	86
Workflow.....	87
Signal Definition.....	87
Excitations: Assigning Signals to Sources .....	88
Monitor Definition.....	91
Starting the Simulation.....	94
Coupled Simulations with CST MPHYSICS STUDIO .....	95
Equivalent Circuit EMS/DS Co-Simulation.....	96
<b>CHAPTER 4 — FINDING FURTHER INFORMATION .....</b>	<b>97</b>
The QuickStart Guide .....	97
Online Documentation .....	97
Tutorials.....	98
Examples.....	98
Technical Support.....	99
History of Changes .....	99

# Chapter 1 — Introduction

## Welcome

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

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

## How to Get Started Quickly

We recommend that you proceed as follows:

1. Read the *CST STUDIO SUITE Getting Started* manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. Work through the online help system's tutorials by choosing the example which best suits your needs.
4. Look at the examples folder in the installation directory. The different application types will give you a good impression of what has already been done with the software. Please note that these examples are designed to give you a basic insight into a particular application domain. Real-world applications are typically much more complex and harder to understand if you are not familiar with the basic concepts.
5. Start with your own first example. Choose a reasonably simple example which will allow you to quickly become familiar with the software.
6. After you have worked through your first example, contact technical support for hints on possible improvements to achieve even more efficient usage of CST EM STUDIO.

## What is CST EM STUDIO?

CST EM STUDIO is a fully featured software package for electromagnetic analysis and design of electrostatic, magnetostatic, stationary current and low-frequency devices. It simplifies the process of creating the structure by providing a powerful graphical solid modeling front end which is based on the ACIS modeling kernel. After the model has been constructed, a fully automatic meshing procedure is applied before a simulation engine is started.

A key feature of CST EM STUDIO is the *Method on Demand* approach which allows using the solver or mesh type that is best suited to a particular problem. Most solvers support two different meshing strategies:

- Classic tetrahedral meshes which provide an explicit representation of the geometry and material interface by a surface mesh. Thus material interfaces

are explicitly resolved by the mesh. This geometry resolution is continually improved during an adaptive mesh refinement using CST's True Geometry Adaptation technique.

- Hexahedral grids in combination with the Perfect Boundary Approximation (PBA)® feature. With hexahedral (Cartesian) meshes, interfaces of materials and solids are not represented by surface mesh cells. Therefore the meshing algorithm is very robust, and meshes can be generated even for very complex CAD geometries. The PBA® feature increases the accuracy of the simulation significantly in comparison to conventional Cartesian mesh simulators.

The software contains five different solvers that best fit their particular applications:

- Electrostatic solver
- Magnetostatic solver
- Stationary current solver
- LF Frequency Domain solver
  - magnetoquasistatic
  - electroquasistatic
  - full-wave
- LF Time Domain solver
  - magnetoquasistatic
  - electroquasistatic

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

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

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

## Who Uses CST EM STUDIO?

Anyone who has to deal with static or low-frequency electromagnetic problems can use CST EM STUDIO. The program is especially suited to the fast, efficient analysis and design of components like actuators, insulators, shielding problems, sensors, transformers, etc. Since the underlying method is a general 3D approach, CST EM STUDIO can solve virtually any static and low-frequency field problem.

## CST EM STUDIO Key Features

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

### General

- Native graphical user interface based on Windows 7, Windows 2008 Server R2, Windows 8, Windows 2012 Server, Windows 8.1 or Windows 2012 Server R2
- The structure can be viewed either as a 3D model or as a schematic. The latter allows for easy coupling of EM simulation with circuit simulation.
- Various independent types of solver strategies (based on hexahedral as well as tetrahedral meshes) allow accurate results with a high performance for all kind of low frequency applications
- For specific solvers highly advanced numerical techniques offer features like Perfect Boundary Approximation (PBA)<sup>®</sup> for hexahedral grids and curved and higher order elements for tetrahedral meshes

### Structure Modeling

- Advanced ACIS<sup>1</sup>-based, parametric solid modeling front end with excellent structure visualization
- Feature-based hybrid modeler allows quick structural changes
- Import of 3D CAD data from ACIS SAT (e.g. AutoCAD<sup>®</sup>), ACIS SAB, Autodesk Inventor<sup>®</sup>, IGES, VDA-FS, STEP, Pro/ENGINEER<sup>®</sup>, CATIA 4<sup>®</sup>, CATIA 5<sup>®</sup>, Siemens NX, Parasolid, Solid Edge, SolidWorks, CoventorWare<sup>®</sup>, Mecadtron<sup>®</sup>, NASTRAN, STL or OBJ files
- Import of 2D CAD data from DXF™, GDSII and Gerber RS274X, RS274D files
- Import of EDA data from design flows including Cadence Allegro<sup>®</sup> / APD<sup>®</sup> / SiP<sup>®</sup>, Mentor Graphics Expedition<sup>®</sup>, Mentor Graphics PADS<sup>®</sup>, Mentor Graphics HyperLynx<sup>®</sup>, Zuken CR-5000<sup>®</sup> / CR-8000<sup>®</sup> and ODB++<sup>®</sup> (e.g. Mentor Graphics Boardstation<sup>®</sup>, CADSTAR<sup>®</sup>, Visula<sup>®</sup>)
- Import of PCB designs originating from CST PCB STUDIO<sup>®</sup>
- Import of 2D and 3D sub models
- Import of Agilent ADS<sup>®</sup> layouts
- Import of Sonnet<sup>®</sup> EM models
- Import of a visible human model dataset or other voxel datasets
- Export of CAD data by ACIS SAT, ACIS SAB, IGES, STEP, NASTRAN, STL, DXF™, GDSII, Gerber or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem setup

---

<sup>1</sup> Portions of this software are owned by Spatial Corp. © 1986 – 2014. All Rights Reserved.

## Electrostatic Solver

- Isotropic and (coordinate-dependent) anisotropic material properties
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Sources: potentials, charges on conductors (floating potentials), uniform volume- and surface-charge densities, capacitive field grading
- Force calculation
- Capacitance calculation
- Electric / magnetic / tangential / normal / open / fixed-potential boundary-conditions
- Perfect conducting sheets and wires
- Discrete edge capacitive elements at any location in the structure
- Adaptive mesh refinement in 3D
- Higher order representation of the solution with tetrahedral mesh
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with Mechanical Solver from CST MPHYSICS STUDIO®
- Equivalent Circuit EMS/DS Co-Simulation for constant material properties

## Magnetostatic Solver

- 3D- and 2D<sup>2</sup>- problem support.
- Isotropic and (coordinate-dependent) anisotropic material properties
- Nonlinear material properties
- Laminated material properties
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Sources: coils, permanent magnets, current paths, external fields, stationary current fields, current ports
- Discrete edge inductances at any location in the structure
- Force calculation
- Apparent and incremental inductance calculation
- Flux linkages
- Electric / magnetic / tangential / normal / open boundary-conditions
- Rotational periodicity for 2D problems
- Adaptive mesh refinement for 2D and 3D solver
- Higher order representation of the solution with tetrahedral and triangular meshes
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with Mechanical Solver from CST MPHYSICS STUDIO
- Equivalent Circuit EMS/DS Co-Simulation for constant and nonlinear material properties

---

<sup>2</sup> The 2D solver supports rotational and translational symmetric problems.

## Stationary Current Solver

- Isotropic and (coordinate-dependent) anisotropic material properties
- Nonlinear electrical conductivity properties
- Temperature dependent materials with coupling to CST MPHYSICS STUDIO
- Electric contact resistance
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Sources: current paths, potentials, current ports, coil segments
- Conductance calculation
- Discrete edge resistances at any location in the structure
- Perfect conducting sheets and wires
- Electric / magnetic / normal / tangential boundary-conditions
- Adaptive mesh refinement in 3D
- Higher order representation of the solution with tetrahedral mesh
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with Thermal Solver from CST MPHYSICS STUDIO
- Equivalent Circuit EMS/DS Co-Simulation for constant material properties

## LF Frequency Domain Solver

- Isotropic and (coordinate-dependent) anisotropic material properties
- Temperature dependent materials with coupling to CST MPHYSICS STUDIO
- Support of hexahedral meshes as well as linear and curved tetrahedral meshes
- Electroquasistatic analysis
- Magnetoquasistatic analysis (eddy current approximation)
- Full wave analysis
- Sources for electroquasistatic analysis: potentials
- Sources for full wave and magnetoquasistatic analysis: coils, current paths, voltage paths, external fields ,current ports
- Impedance calculation
- Force calculation
- Perfect conducting sheets and wires
- Lumped R, L, C elements at any location in the structure
- Surface impedance model for good conducting metals
- Electric- / magnetic- and open-boundary-conditions
- Adaptive mesh refinement in 3D
- Higher order representation of the solution with tetrahedral mesh
- Automatic parameter studies using built-in parameter sweep tool
- Automatic structure optimization for arbitrary goals using built-in optimizer
- Network distributed computing for optimizations, parameter sweeps and remote calculations
- Coupled simulations with Thermal Solver from CST MPHYSICS STUDIO

## LF Time Domain Solver

- Isotropic and (coordinate-dependent) anisotropic material properties
- Magnetoquasistatic analysis (eddy current approximation), 3D- and 2D- problem support
- Electroquasistatic analysis
- Nonlinear material properties (within the magnetoquasistatic analysis)
- Iron Loss computation
- Support of linear and curved tetrahedral meshes
- Sources for the magnetoquasistatic analysis: coils, coil segments, current paths, voltage paths, permanent magnets, external magnetic source field
- Sources for electroquasistatic analysis: potentials
- Magnetoquasistatic analysis: perfect conducting sheets and wires
- Electric / magnetic boundary-conditions
- Higher order representation of the solution with tetrahedral mesh
- User defined excitation signals and signal database
- Adaptive time stepping for 3D models
- Rigid body motion for 2D models
- Network distributed computing remote calculations

**Note:** some solvers or features may be available for hexahedral and some may be available for tetrahedral meshes only.

## CST DESIGN STUDIO View

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

## SAM (System and Assembly Modeling)

- 3D representations for individual components
- Automatic project creation by assembling the schematic's elements into a full 3D representation
- Manage project variations derived from one common 3D geometry setup
- Coupled Multiphysics simulations by using different combinations of coupled circuit/EM/Thermal/Stress projects

## Visualization and Secondary Result Calculation

- Multiple 1D result view support
- Online visualization of intermediate results during transient simulations
- Import and visualization of external xy-data
- Copy / paste of xy-datasets
- Fast access to parametric data via interactive tuning sliders
- Automatic parametric 1D result storage
  
- Various field visualization options in 2D and 3D for electric fields, magnetic fields, potentials, current densities, energy densities, etc.
- Animation of field distributions
- Display of source definitions in 3D
- Display of nonlinear material curves in xy-plots
- Display of material distribution for nonlinear materials
  
- Display and integration of 2D and 3D fields along arbitrary curves
- Integration of 3D fields across arbitrary faces
- Hierarchical result templates for automated extraction and visualization of arbitrary results from various simulation runs. These data can also be used for the definition of optimization goals.

## Result Export

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

## Automation

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

## About This Manual

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

The main part of the manual is the *Simulation Workflow* (Chapter 2) which will guide you through the most important features of CST EM STUDIO. We strongly encourage you to study this chapter carefully.

## Document Conventions

- Buttons that should be pressed within dialog boxes are always written in italics, e.g. *OK*.
- Key combinations are always joined with a plus (+) sign. *Ctrl+S* means that you should hold down the “Ctrl” key while pressing the “S” key.
- The program’s features can be accessed through a Ribbon command bar at the top of the main window. The commands are organized in a series of tabs within the Ribbon. In this document a command is printed as follows: *Tab name: Group name ⇔ Button Name ⇔ Command name*. This means that you should activate the proper tab first and then press the button *Command name*, which belongs to the group *Group name*. If a keyboard shortcut exists it is shown in brackets after the command.  
Example: *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: 2D/3D Results ⇔ E-Field [Es] ⇔ Abs*

## Your Feedback

We are constantly striving to improve the quality of our software documentation. If you have any comments regarding the documentation, please send them to your local support center. If you don’t know how to contact the support center near you, send an email to [info@cst.com](mailto:info@cst.com).

## Chapter 2 – Simulation Workflow

The following example shows a fairly simple magnetostatic simulation. Studying this example carefully will allow you to become familiar with many standard operations that are necessary to perform a simulation within CST EM STUDIO.

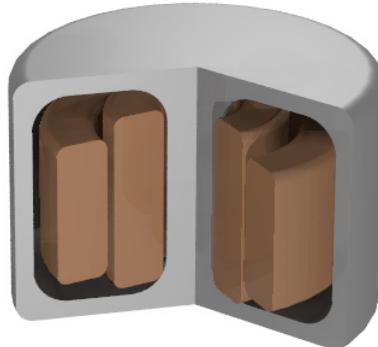
Go through the following explanations carefully even if you are not planning to use the software for magnetostatic computations. Only a small portion of the example is specific to this particular application type since most of the considerations are quite general to all solvers and application domains.

At the end of this example, you will find some remarks concerning the differences between the typical sources and simulation procedures for electrostatic, stationary current, magnetostatic, and low-frequency calculations.

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

### The Structure

In the example, you will model a simple sealed transformer consisting of two coils and an iron core in a cylindrical box. Then you will set up the simulation to compute the magnetic field distribution and inductances. The following picture shows the current structure of interest (it has been sliced open purely to aid visualization). The picture was produced using the POV export option in CST EM STUDIO.



Before you start modeling the structure, let's spend a few moments discussing how to describe this structure efficiently.

CST EM STUDIO allows you to define the properties of the background material. Anything you do not fill with a particular material will automatically be considered as the background material. For this structure, it is sufficient to model only the cylinder box, the iron core and the two coils. The background properties will be set to vacuum.

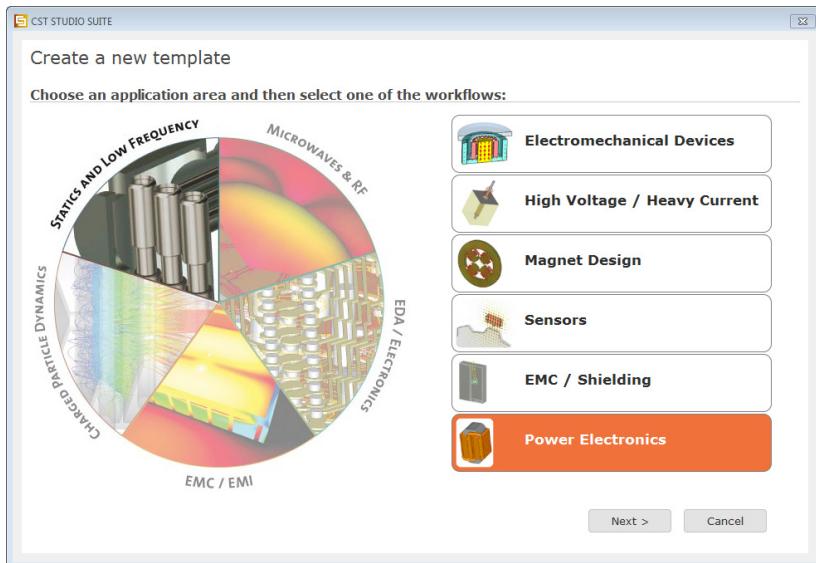
Your method of describing the structure should therefore be as follows:

1. Model the cylindrical box.
2. Model the iron core inside the box.
3. Define the coils.

## Create a New Project

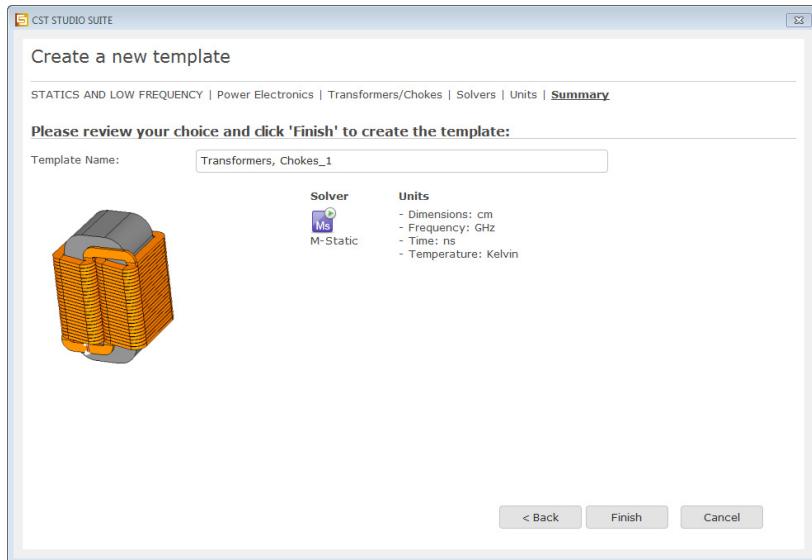
After launching the CST STUDIO SUITE you will enter the start screen showing you a list of recently opened projects and allowing you to specify the application which suits your requirements best. The easiest way to get started is to configure a project template which sets the basic settings that are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

Next you should choose the application area, which is *Statics and Low Frequency* for the example in this tutorial and then select the workflow by clicking on the corresponding entry.



For the sealed transformer, please select *Power Electronics*  $\Rightarrow$  *Transformers/Chokes*  $\Rightarrow$  *M-Static* .

At last you are requested to select the units which fit your application best. For the sealed transformer all dimensions will be given in cm. Therefore select cm from the *Dimensions* drop-down list. For the specific application in this tutorial the other settings can be left unchanged. After clicking the *Next* button, you can give the project template a name and review a summary of your initial settings.



Finally, click the *Finish* button to save the project template and to create a new project with appropriate settings. CST EM STUDIO will be launched automatically due to the choice of the application area Statics and Low Frequency.

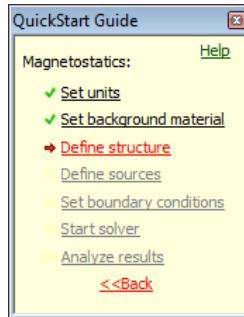
**Please note:** When you click again on *File: New and Recent* you will see that the recently defined template appears below the *Project Templates* section. For further projects in the same application area you can simply click on this template entry to launch CST EM STUDIO with useful basic settings.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

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

## Open the QuickStart Guide

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

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



As the project template has set already the solver type, units and background material, the *Magnetostatic Analysis* is preselected and some entries are marked as done. The red arrow always indicates the next step necessary for your problem definition. You do not have to follow the steps in this order, but we recommend you follow this guide at the beginning to ensure that all necessary steps have been completed.

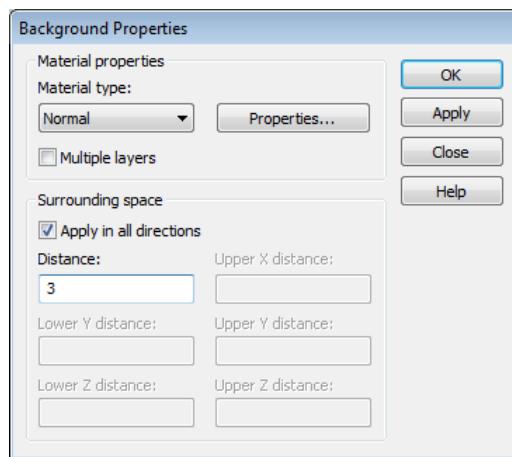
Look at the dialog box as you follow the various steps in this example. You may close the assistant at any time. Even if you re-open the window later, it will always indicate the next required step.

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

## Define the Background Material

As discussed above, the structure will be described within a vacuum world with some surrounding space. The project template has set some typical default values already. Select *Modeling: Materials*  $\Rightarrow$  *Background*  to check or modify the background material settings. For this example enter 3 cm for all directions by checking *Apply in all directions* and enter the *Distance* value.

Confirm by clicking the *Ok* button. (Remember: according to the predefined unit, all geometric settings are in cm.)

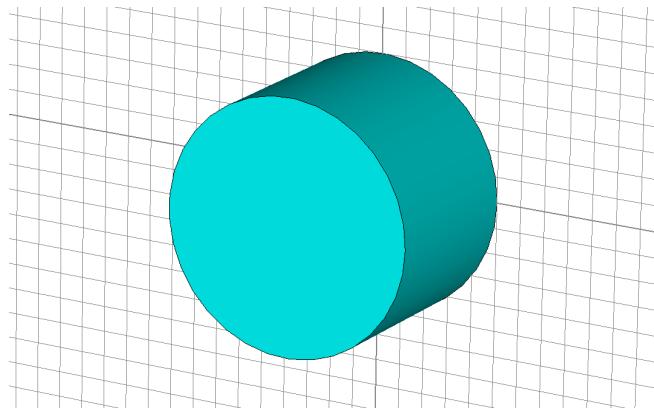


## Model the Structure

First create a cylinder along the z-axis of the coordinate system by the following steps:

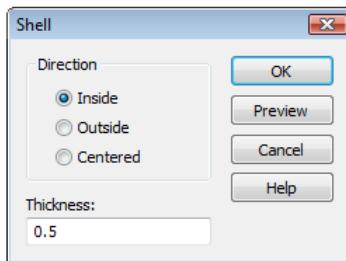
1. Select the cylinder creation tool from *Modeling: Shapes*  $\Rightarrow$  *Cylinder* .
2. Press the *Shift+Tab* key, and enter the center point (0,0) in the xy-plane before pressing the *Return* key to store this setting.
3. Press the *Tab* key again, enter the radius 5 and press the *Return* key.
4. Press the *Tab* key, enter the height as 7 and press the *Return* key.
5. Press *Esc* to create a solid cylinder (skip the definition of the inner radius).
6. In the shape dialog box, enter “cylinder box” in the *Name* field.
7. Select *component1* from the Component dropdown list.
8. Select *[New Material]* from the Material dropdown list. The Material dialog box opens where you should enter the material name “Iron”, select *Normal* properties (*Type*) and set the material properties *Epsilon* = 1.0 and *Mue* = 1000. Now you can select a color and close the dialog box by clicking *OK*.
9. Back in the cylinder creation dialog box, click *OK* to create the cylinder.
10. Finally, save the structure by selecting *File: Save (Ctrl+S)* and entering the name “first example.cst” in a folder of your choice.

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



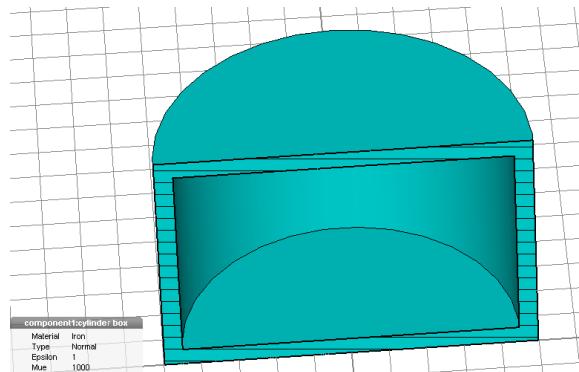
Please note that you can switch on or off the multicolored axes or the axes at the origin in the *View Options* dialog box (*View: Options*  $\Rightarrow$  *View Options* (*Alt+v*) 

The next step is to shell the cylinder. Select the cylinder in the navigation tree (*NT: Components*  $\Rightarrow$  *component1*  $\Rightarrow$  *cylinder box*) and open the shell dialog by selecting *Modeling: Tools*  $\Rightarrow$  *Shape Tools*  $\Rightarrow$  *Shell Solid or Thicken Sheet*. Enter the *Thickness* 0.5 and select *Inside* as the direction.



To observe the result, activate the cutting plane view via *View: Sectional View*  $\Rightarrow$  *Cutting Plane*  $\Rightarrow$  *Cutting Plane* (*Shift+C*) . You can adjust the cutting plane settings either by using the up/down arrow keys or by entering the *Cutting Plane Properties* dialog box (*View: Sectional View*  $\Rightarrow$  *Cutting Plane*  $\Rightarrow$  *Properties* ).

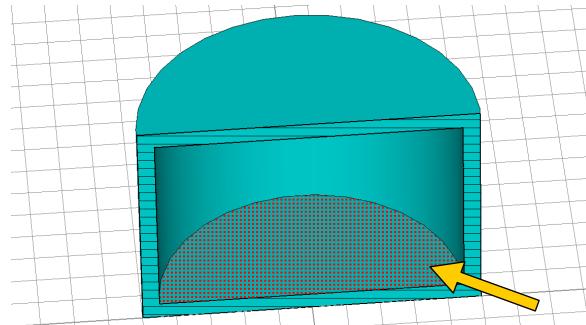
To look into the box, you might have to rotate the view. Activate the rotation mode by selecting *View: Mouse Control*  $\Rightarrow$  *Rotate*  $\Rightarrow$  *Smart Mouse Pointer* . Then press the left mouse button and move the mouse until the view looks like this:



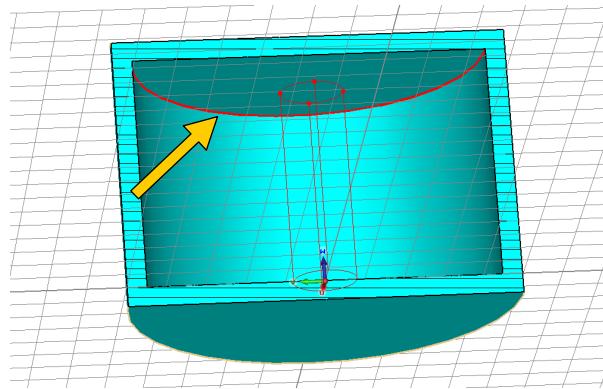
It is also possible to hold down the *Ctrl* button to activate the rotation mode for as long as *Ctrl* is pressed.

The next step is to create a second cylinder inside the box. The center of the new cylinder's base should align with the center of the box's inside face. To this end, first align the local coordinate system (WCS) with the lower inside z face of the box:

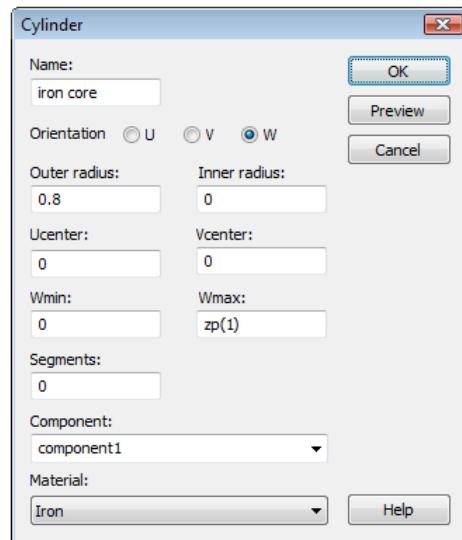
1. Select *Modeling: Picks*  $\Rightarrow$  *Picks*
2. Double-click on the box's lower inside z-plane. The selected face should now be highlighted:



3. Now choose *Modeling: WCS*  $\Rightarrow$  *Align WCS* (Shortcut: *w*).
4. Select the cylinder creation tool *Modeling: Shapes*  $\Rightarrow$  *Cylinder* .
5. Press the *Shift+Tab* key, and enter the center point (0,0) in the uv-plane and press the *Return* key.
6. Press the *Tab* key again, and enter a radius of 0.8 and press the *Return* key.
7. Select *Modeling: Picks*  $\Rightarrow$  *Pick Point*  $\Rightarrow$  *Pick Circle Center* .
8. Set the cylinder's height by picking the highlighted circle of the upper inner face of the box with a double-click. You might have to rotate the structure a little bit to get a better view:

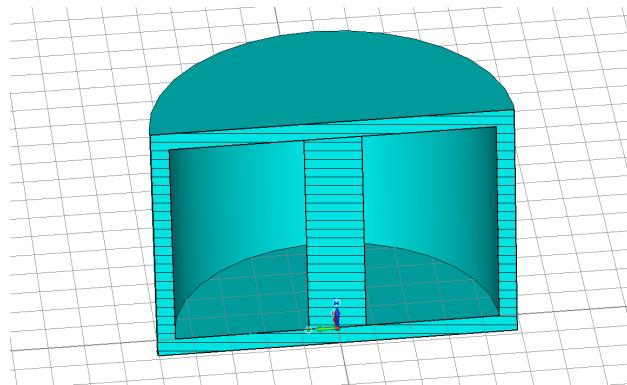


9. Press *Esc* to create a solid cylinder (skip the definition of the inner radius).
10. In the shape dialog box, enter “iron core” in the *Name* field.
11. Select the component “component1” from the component list.
12. Select the material “Iron” from the material list.



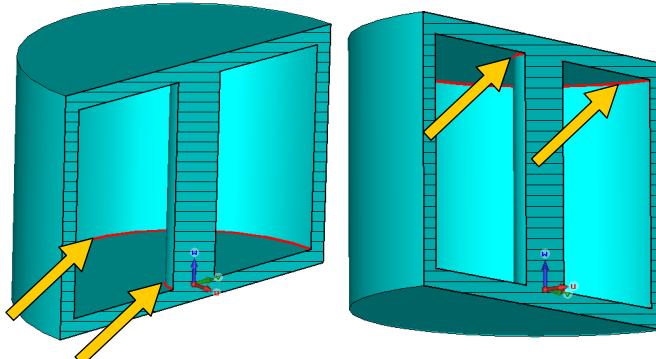
13. Click the *Ok* button.

The result of these operations should look like this:

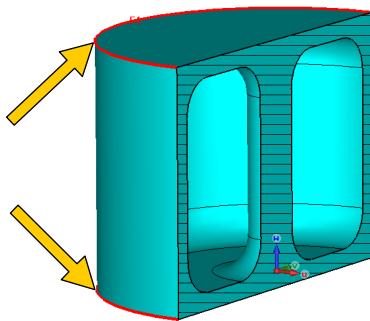


Sharp edges are, in general, responsible for field singularities. Therefore, we will blend the edges of the iron core and the cylinder box. Before we can do this, the two bodies have to be united. Thus, select the cylinder box (either in the navigation tree or by double-clicking on it in the main view). Then choose *Modeling: Tools*  $\Rightarrow$  *Boolean*  $\Rightarrow$  *Add* and select the iron core. Confirm the operation by pressing the *Enter* key. The iron core entry will vanish from the navigation tree and only the cylinder box remains in the *NT: Components*  $\Rightarrow$  *component1* folder.

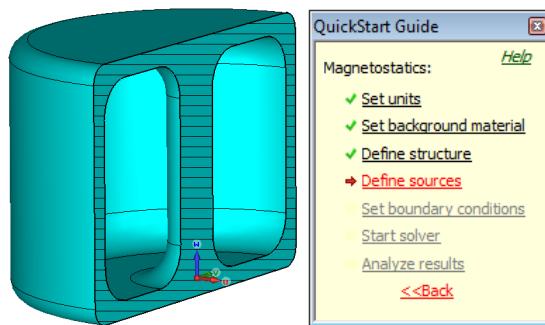
Now you can select the edges to blend. All inner edges shall be blended with radius 1, the outer edges of the cylinder box with radius 0.5. Hence, activate the pick edge tool *Modeling: Picks*  $\Rightarrow$  *Picks* (Shortcut: *e*) and pick first all inner edges:



Finally enter the *Blend Edges* dialog box via *Modeling: Tools*  $\Rightarrow$  *Blend*  $\Rightarrow$  *Blend Edges* and enter the radius 1.0. Confirm this setting by pressing *OK*. Next pick the two outer edges of the cylinder box.



Open the *Blend Edges* dialog again and enter the radius 0.5. Leave the dialog via the OK button. The cylinder box should look now as depicted below:



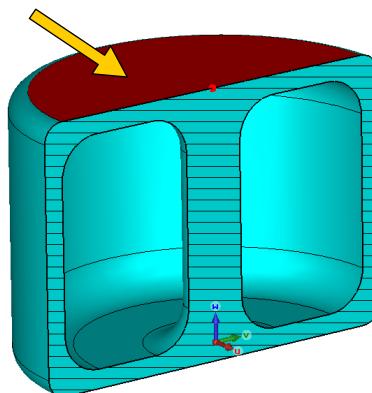
Looking at the *QuickStart Guide*, you will see that now it is time to define the sources for the magnetic field simulation.

## Define Coils

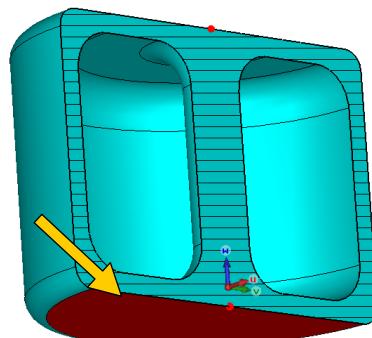
In CST EM STUDIO, a coil is defined as an *a-priori* known current- or voltage-distribution which should be constant over the cross-section of the coil body for this example. Consequently, the coil represents the equivalent distribution of the current of a realistic coil with many turns with small-scale variations averaged out.

The creation of a coil is quite similar to the definition of a solid by curves. First of all, you have to move the working coordinate system to the right position:

1. Select *Modeling: Picks*  $\Rightarrow$  *Pick Point*  $\Rightarrow$  *Pick Face Center* (shortcut A).

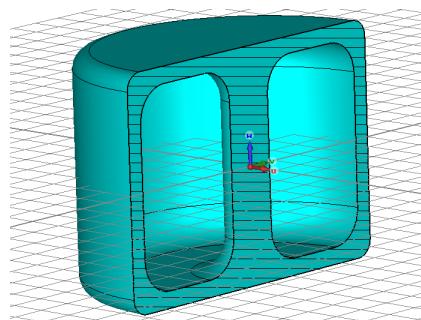


2. Double-click on the upper outside face of the box as highlighted.
3. Select *Modeling: Picks*  $\Rightarrow$  *Pick Point*  $\Rightarrow$  *Pick Face Center*  again.



4. Double-click on the lower outside face of the box as highlighted.
5. Select *Modeling: Picks*  $\Rightarrow$  *Pick Point*  $\Rightarrow$  *Mean Last Two Points*.
6. Select *Modeling: WCS*  $\Rightarrow$  *Align WCS* .

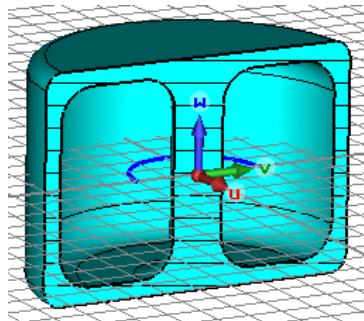
Now the working coordinate system should be placed as depicted in the next figure. At any time the *Working Plane* can be enabled or disabled using *View: Visibility  $\Rightarrow$  Working Plane* (shortcut Alt+W).



To define the path of the first coil, carry out the following:

1. Select *Modeling: Curves*  $\Rightarrow$  *Curves*  $\Rightarrow$  *Circle* .

2. Press the *Shift+Tab* key and enter the center point (0,0) in the uv-plane. Then press the *Return* key to store this setting.
3. Press the *Tab* key again, and enter the radius 2.
4. In the circle dialog box, enter “coil path 1” in the *Name* field.
5. Click *OK* to create the circle.



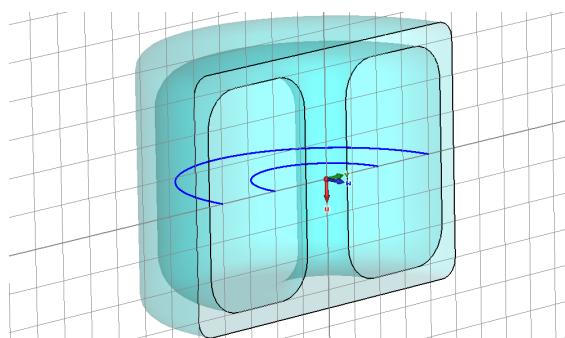
The path for the second coil is created in the same way:

1. Select *Modeling: Curves*  $\Rightarrow$  *Curves*  $\Rightarrow$  *Circle*
2. Press the *Shift+Tab* key, and enter the center point (0,0) in the uv-plane before pressing the *Return* key to store this setting.
3. Press the *Tab* key again, and enter the radius 4.
4. In the circle dialog box, enter “coil path 2” in the *Name* field.
5. Select *[New Curve]* from the *Curve* drop-down list.
6. Click *OK* to create the circle.

**Please note:** We put all path and profile curves into separate *Curve* folders just to simplify blending the coils’ edges afterwards.

To define the profile paths of both coils, you first need to rotate the working coordinate system around the v-axis:

1. Press *Shift+V* or select *Modeling: WCS*  $\Rightarrow$  *Transform WCS* and activate the *Rotate* control in the *Transform Local Coordinate System* dialog box and enter 90 for the V component.
2. Click *OK* to finish the rotation of the working coordinate system.



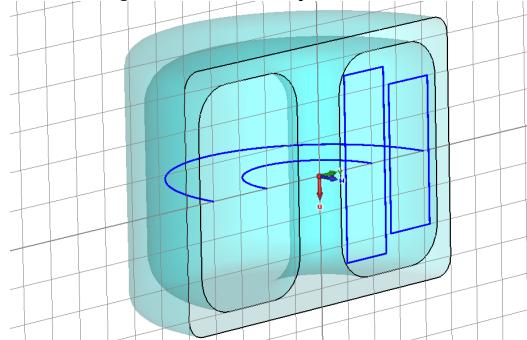
For the definition of the first profile curve, perform the following steps:

1. Select *Modeling: Curves*  $\Rightarrow$  *Curves*  $\Rightarrow$  *Rectangle* 
2. Press the *Tab* key, and enter the first point (-2.5, 1) in the uv-plane before pressing the *Return* key to store this setting.
3. Press the *Tab* key again, and enter the second point (2.5, 2.5) and press the *Return* key.
4. In the rectangle dialog box, enter “profile path 1” in the *Name* field.
5. Select *[New Curve]* from the *Curve* drop-down list.
6. Click *OK* to create the rectangle.

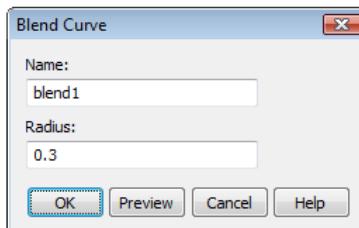
The second profile can be created as follows:

1. Select *Modeling: Curves*  $\Rightarrow$  *Curves*  $\Rightarrow$  *Rectangle* 
2. Press the *Tab* key, and enter the first point (-2, 2.7) in the uv-plane before pressing the *Return* key to store this setting.
3. Press the *Tab* key again, and enter the second point (2, 4.2) and press the *Return* key.
4. In the rectangle dialog box, enter “profile path 2” in the *Name* field.
5. Select *[New Curve]* from the *Curve* drop-down list.
6. Click *OK* to create the rectangle.

Now your model should look like the one depicted below. You may need to click on the components folder in the *Navigation Tree* if only the last created curve is still highlighted.

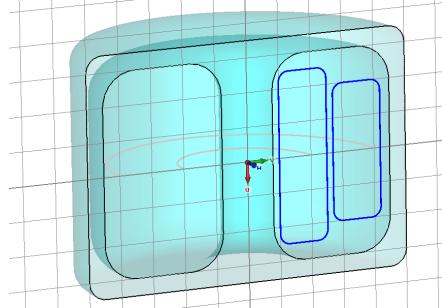


Like for the cylinder box, it is meaningful to blend the coil edges as well. This can be done by blending the corners of the profile paths. Select *NT: Curves*  $\Rightarrow$  *curve3*  $\Rightarrow$  *profile path 1* or double click on the appropriate rectangle in the main view. Now choose *Modeling: Curves*  $\Rightarrow$  *Curve Tools*  $\Rightarrow$  *Blend Curve* . You will be asked to double-click on a point to which the blend is to be applied. Choose one of the four corners of the rectangle. The *Blend Curve* dialog box will pop up. Enter the radius 0.3.



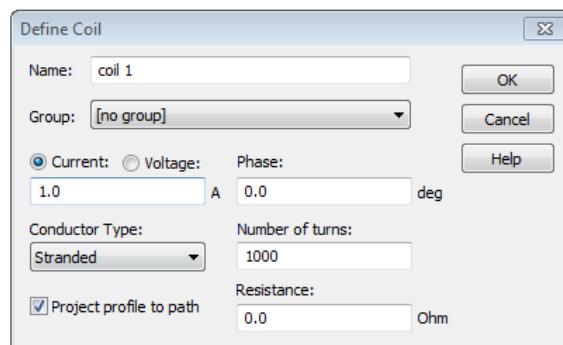
Confirm this setting by pressing OK and repeat the same steps to blend the other three corners of the *profile path 1* rectangle as well.

Next, the corners of the *profile path 2* rectangle need to be blended in completely the same manner. Select *NT: Curves*  $\Rightarrow$  *curve4*  $\Rightarrow$  *profile path 2*, choose *Modeling: Curves*  $\Rightarrow$  *Curve Tools*  $\Rightarrow$  *Blend Curve*  and pick a corner of the highlighted rectangle. Enter the radius 0.3 in the *Blend Curve* dialog box and repeat these steps for the remaining corners. The profile curves should then look as depicted below:



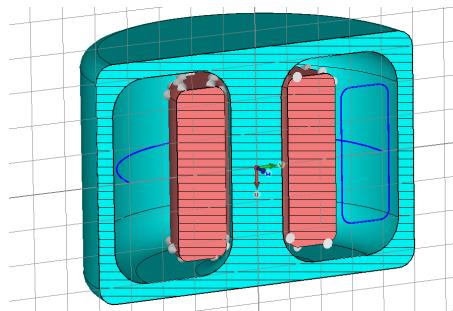
Finally, the coils can be created from the profile and path curves:

1. Select *Simulation: Sources and Loads*  $\Rightarrow$  *Coil*  $\Rightarrow$  *Coil* .
2. Move the mouse cursor to “profile path 1” until it is highlighted. Then double-click to select it (the inner profile curve).
3. Move the mouse cursor to “coil path 1” and select it by double-clicking.
4. In the *Define Coil* dialog box, enter “coil 1” in the *Name* field, select the type *Current*, enter 1 A for the current value and 1000 in the *Number of turns* field. (Do not change the *Conductor Type*, *Phase* or *Resistance* values.) Coils can be gathered into the so-called coil groups. For more information about this, please refer to the online help.



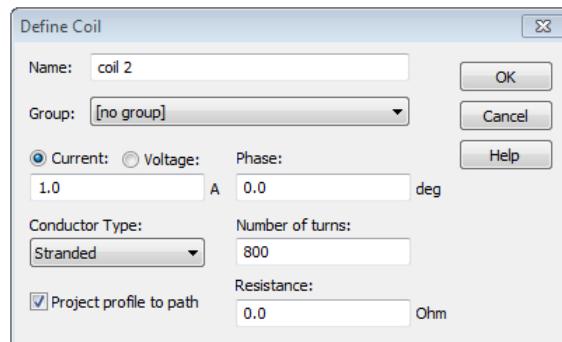
5. Click *OK* to create the coil.

Now your model should look like the one depicted below. You may need to click on the components folder in the *Navigation Tree* if the coil is not highlighted.



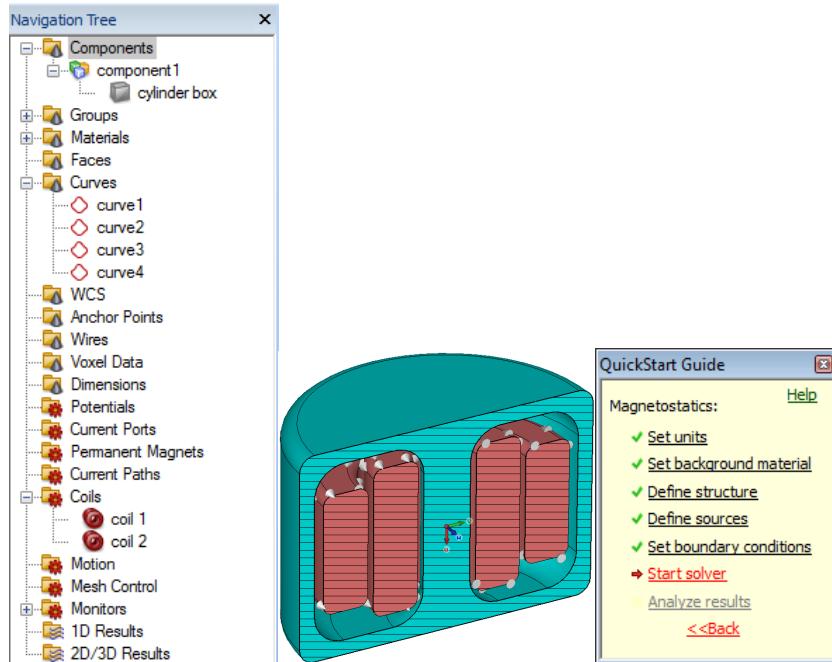
The same procedure can be applied for the second coil:

1. Select *Simulation: Sources and Loads*  $\Rightarrow$  *Coil*  $\Rightarrow$  *Coil*
2. Move the mouse cursor to “profile path 2” until it is highlighted. Then double-click to select it.
3. Move the mouse cursor to “coil path 2,” and select it by double-clicking.
4. In the *Define Coil* dialog box, enter “coil 2” in the *Name* field, 1 A for the value of a current and 800 in the *Number of turns* field.



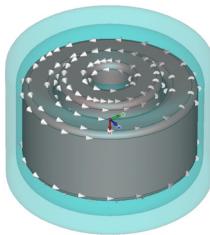
5. Click *OK* to create the coil.

Congratulations! You have just created your first structure within CST EM STUDIO. The view should now look like this after the working plane (*View: Visibility*  $\Leftrightarrow$  *Working Plane* (*Alt+W*) ) has been switched off:

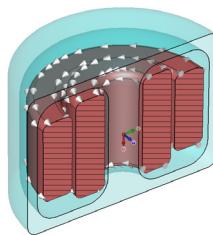


**Please note:** As the project template has set some default boundary conditions which are applicable in most use cases, the corresponding entry in the *QuickStart Guide* is already checked. Nevertheless, you should always check if the model can be simplified, e.g. by symmetry conditions. We will discuss this in the next section.

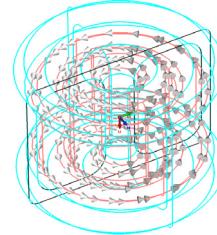
The following gallery shows some views of the structure using different visualization options:



Shaded view  
(deactivated working plane  
iron material properties:  
50% transparency)



Shaded view,  
(cutting plane active)  
Wireframe



Wireframe view,  
(View: Visibility

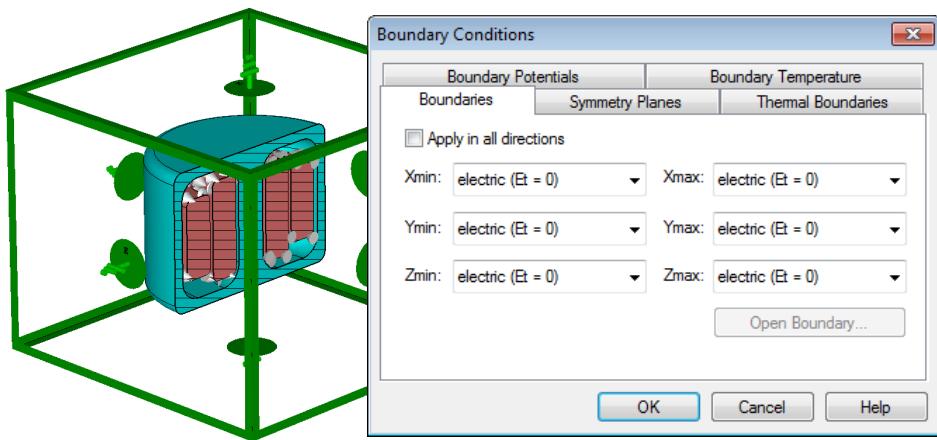
## Define Boundary Conditions

The simulation of this structure is performed only within the bounding box enclosing the structure together with some background material. The space occupied by the structure and background material is called the "computational domain" in the sequel.

Note that the restriction to a bounded computational domain is artificial for our example (keeping in mind the transformer structure in open space). However, in this simple case, the magnetic flux is concentrated in the core material. Therefore, the artificial boundary will not considerably disturb the solution though the added space around the structure is not very large.

In order to get a well-defined problem, you must specify the behavior of the field at the boundary of the computational domain by setting a boundary condition for each plane ( $X_{\min}/X_{\max}/Y_{\min}/Y_{\max}/Z_{\min}/Z_{\max}$ ).

The boundary conditions are specified in a dialog box which you can bring up by choosing *Simulation: Settings  $\Rightarrow$  Boundaries* .



While the boundary dialog box is open, the boundary conditions will be visualized in the structure view as in the picture above. You can change boundary conditions within the dialog box or interactively in the view by double-clicking on the corresponding boundary symbol, and then select the appropriate type from the context menu.

The project template has already set "electric ( $Et = 0$ )" boundary conditions for every face. You do not need to change the default setting. Do NOT click OK.

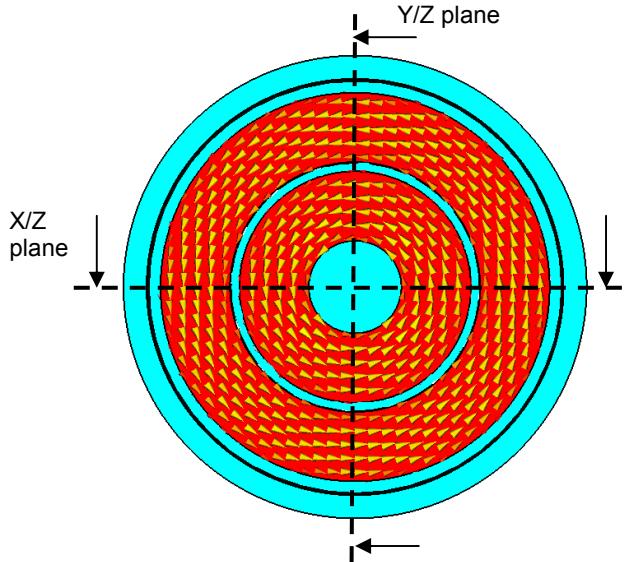
**Background information:** Electric boundary conditions ("electric ( $Et=0$ )") force the tangential electric field to zero. For non-zero frequencies, Faraday's Law implies a zero normal component of the magnetic flux density  $B$ . Viewing magnetostatics as a static limit of Maxwell's equations justifies this implication even for the magnetostatic case. Consequently, an electric boundary condition always forces a zero normal component of the magnetic flux density, i.e. the  $B$ -field is purely tangential, and no flux can leave the computational domain at this face. Note that this also applies to the boundary of perfectly electric conductors (PECs), which play the role of interior boundary conditions.

Another important boundary condition is the "magnetic ( $Ht=0$ )-condition, which forces a zero tangential magnetic field, i.e. the magnetic field is purely normal at a face defined as "magnetic." This consideration is used in the next sub-section.

## Define Symmetry Conditions

In addition to the boundary planes, you can specify "symmetry planes". Each specified symmetry plane reduces the simulation time and the required memory by a factor of two. In our example, the structure is symmetric with respect to the Y/Z plane (perpendicular to the x-axis). A second symmetry plane applies to the X/Z plane.

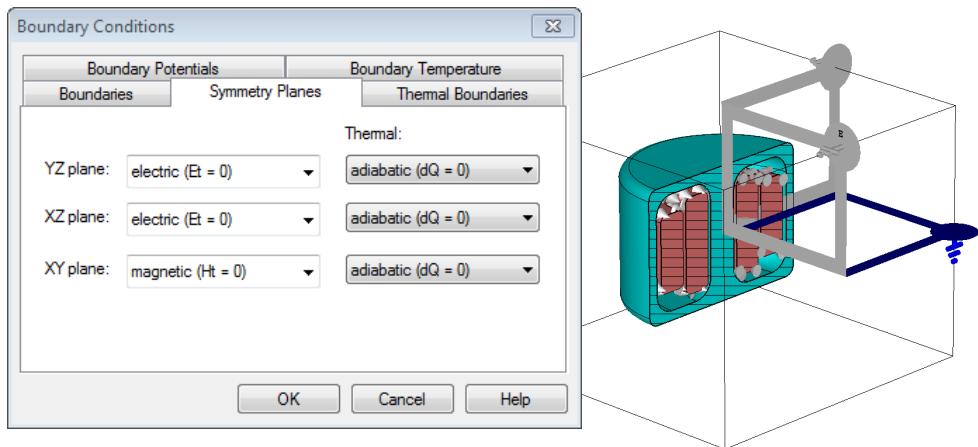
The excitation of the fields is performed by the currents in the coils for which the current pattern is shown below:



The electric symmetry planes for the magnetic field can be applied if the current pattern is parallel on the plane to its normal.

The resulting magnetic field has no component normal to the X/Z and Y/Z planes (the entire field is oriented tangential to these planes). Moreover, the fields have no component tangential to the X/Y plane. If you specify X/Z and Y/Z planes as "electric" and X/Y as "magnetic" symmetry planes, you can advise CST EM STUDIO to limit the simulation to 1/8 of the actual structure while taking these symmetry conditions into account.

To specify the symmetry condition, click on the *Symmetry Planes* tab in the *Boundary Conditions* dialog box. For the YZ- and XZ-plane symmetry, you can choose "electric" by either selecting the appropriate choice in the dialog box, or by double-clicking on the corresponding symmetry plane visualization in the view and selecting the appropriate choice from the context menu. For XY-plane symmetry, choose "magnetic." Once you have done this, your model and the dialog box will appear as follows:



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

As shown by the *QuickStart Guide*, the model is now completely defined, and you are ready to start the magnetostatic solver.

In order to get a discrete version of the defined model that can be solved numerically, a *mesh* must be provided for the computational domain. CST EM STUDIO features two independent solvers based on tetrahedral and hexahedral meshes, respectively. Let's start with the tetrahedral solver.

## Generate and Visualize a Tetrahedral Mesh

The tetrahedral mesh generation for the structure is performed fully automatically when the tetrahedral magnetostatic solver starts.

It is also possible to generate the mesh separately before starting the solver. This may be helpful in order to get an impression of the mesh quality and mesh resolution. Furthermore, it is possible to fine-tune the mesh before running the computation using *a priori* knowledge about the solution. Let's use this second possibility and generate the mesh separately.

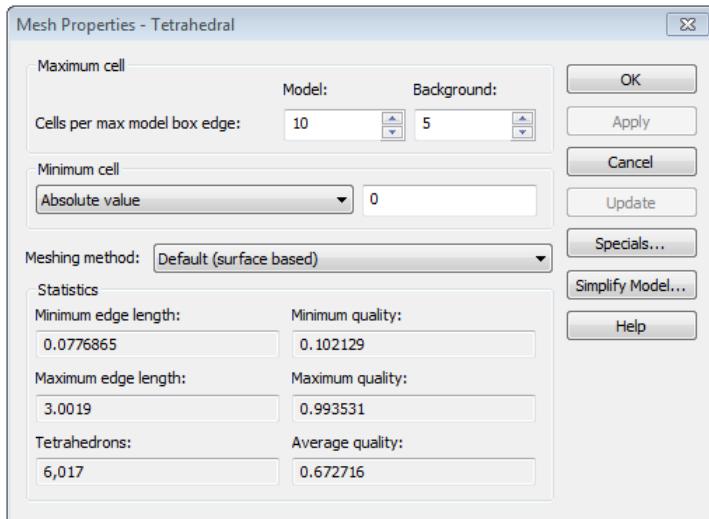
First, open the *Mesh Properties* dialog by selecting *Simulation: Mesh  $\Rightarrow$  Global Properties  $\Rightarrow$  Tetrahedral*. The dialog box "Mesh Properties – Tetrahedral" will open. In order to get a reasonable overall mesh resolution of the problem, you can increase the *Maximum cell*. In general it is sufficient, however, to refine the mesh locally, i.e. only at certain critical parts of the geometry. This can be achieved by running the solver with the fully automatic energy-based adaptive refinement. Thus we start with a rather coarse mesh and leave the *Cells per max model box edge* at the value 10 for the model and at 5 for the background.

**Background information:** The results are strongly influenced by the mesh resolution. The automatic mesh generator analyzes the geometry and tries to refine the mesh locally taking geometric features into account (e.g. curvature-based refinement with tetrahedral meshes or expert system-based approach with hexahedral meshes). However, due to the complexity of electromagnetic problems, this approach may not be able to determine all critical domains in the structure. To circumvent this problem, CST

*EM STUDIO features an adaptive mesh refinement that uses the results of a previous solver run in order to optimize the mesh. The adaptive mesh refinement can be activated by checking the corresponding option in the solver parameter dialog box.*

Now click the *Update* button in the *Mesh Properties* dialog box to start the mesh generation. You will see a progress bar displaying the current status of the mesh generation.

When the mesh generation process has finished, the progress bar disappears. You will see that the entries in the *Mesh summary* frame of the *Mesh Properties Dialog* have been updated.



In the *Statistics* frame, you can get information about

3. the minimum and maximum mesh-edge lengths,
4. the number of tetrahedrons,
5. the maximum / minimum, and average mesh quality.

The number of tetrahedrons and the edge lengths give you information about the size and resolution of the discretized model.

Please mind that the mesh size and the results might differ slightly depending on the operating system and the architecture of the machine with which they are calculated.

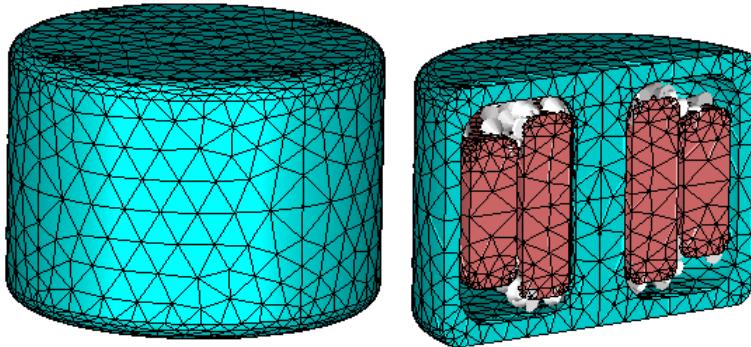
**Background information:** Generally, due to the finiteness of the mesh density, the computed results differ from the exact solution. The introduced error is called the discretization error. Increasing the mesh density will usually lead to more precise results, yet the computation time and the necessary memory size will increase.

The quality of a tetrahedron is positive and less than or equal to one. The value “1” indicates the highest (equilateral tetrahedron), the value “0” the lowest quality (zero volume tetrahedron). Please refer to the online help for an exact definition of quality.

**Background information:** Not only the mesh density but also the mesh quality has a strong influence on the results. A very low mesh quality may lead to a bad approximation of the model. Moreover, a low mesh quality may reduce the speed of an iterative solver.

*This is the reason why it is always meaningful to have a look at the mesh when running a simulation.*

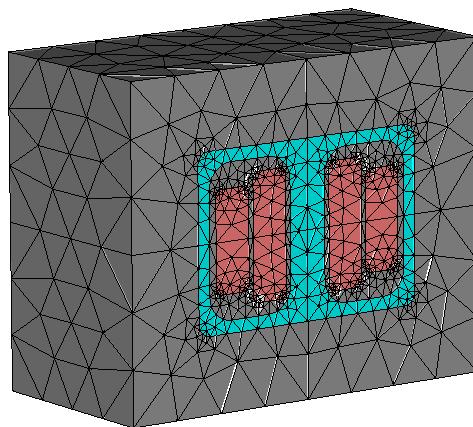
Now close the *Mesh Properties* dialog by clicking the *OK* button. You can visualize the mesh by entering the mesh view (*Simulation: Mesh*  $\Rightarrow$  *Mesh View* ). The mesh should look similar as illustrated below. To inspect the mesh in the interior of the structure activate the cutting plane by selecting *View: Sectional View*  $\Rightarrow$  *Cutting Plane*  $\Rightarrow$  *Cutting Plane (Shift+C)* .



The automatic curvature refinement leads to a local refinement along the blended edges. By default, the mesh transition from the coarser to the finer mesh regions is very rapid. This transition can be smoothed in the *Specials* dialog-box of the Global Mesh Properties dialog (*Mesh: Mesh Control*  $\Rightarrow$  *Global Properties*  $\Rightarrow$  *Tetrahedral* ) which may also improve the mesh quality. Please refer to the Online Help for more details. For this model, the default settings are sufficient.

Remember that you have reduced the computational model by defining symmetry planes. Therefore, only 1/8 of the computational domain is meshed. Nevertheless, the mesh is visualized for the complete structure by mirroring the missing parts. You can easily see the symmetry planes in the mesh-view.

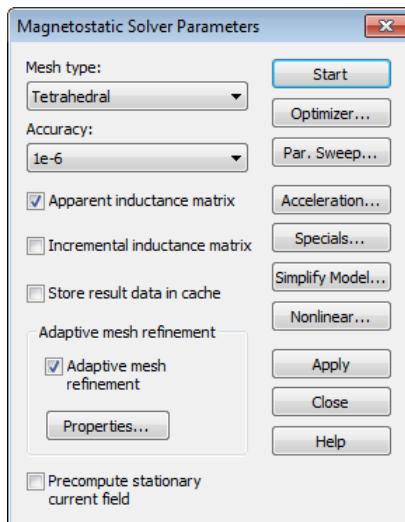
Finally, let's look at the mesh of the surrounding space. Activate the visualization of the background material by selecting *View: Options*  $\Rightarrow$  *View Options (Alt+v)* , and then select the *Background material* checkbox in the *Draw* frame of the *General Tab*. The displayed mesh should look similar to the following picture:



Before you go on, you should deactivate the visualization of the background material by selecting *View: Options*  $\Rightarrow$  *View Options (Alt+v)* and un-checking *Background material*. Leave the mesh view by selecting *Mesh*  $\Rightarrow$  *Close Mesh View* .

## Run the Tetrahedral Magnetostatic Solver

The simulation is started from the *Magnetostatic Solver Parameters* dialog box which can be opened via *Home: Simulation  $\Rightarrow$  Setup Solver* .

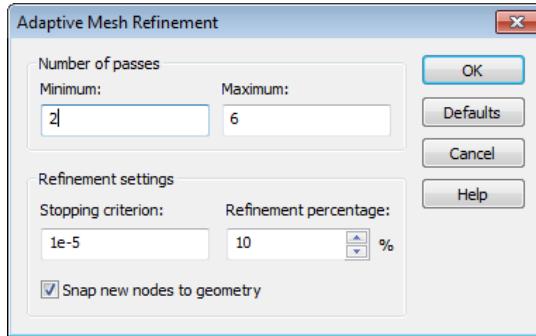


Make sure the *Mesh Type* "Tetrahedral" is selected. In the *Accuracy* drop-down list, a stopping criterion for the iterative linear equation system solver can be selected. For the example model, leave the *Accuracy* value at 1e-6.

**Background Information:** While the solution accuracy mainly depends on the discretization of the structure and can be improved by refining the mesh, the numerical error of the linear equation system solver introduces a second error source in field simulations (iteration error). Choosing a small Accuracy value reduces this error at the expense of a longer calculation time. Usually, an accuracy setting of "1e-6" is sufficient, but in some cases it might be necessary to select a smaller value, particularly if you receive a warning that the results are not accurate. Furthermore, with increasing mesh density (i.e. smaller discretization error) you should also increase the solver accuracy by selecting a smaller Accuracy value.

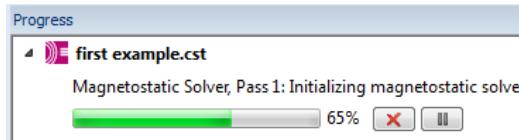
Furthermore, activate the calculation of the *Apparent inductance matrix*. Please note that the *Adaptive mesh refinement* is switched on already. This setting is meaningful as the initial mesh is rather coarse. During the solver run, several mesh refinement passes are performed automatically until the energy value does not change significantly between two subsequent passes. The default termination criterion is an energy deviation of 1% (or less). You can fine-tune these settings in the *Adaptive Mesh Refinement* dialog box.

Click the *Properties...* button to enter the *Adaptive Mesh Refinement* dialog box. Change the *Stopping criterion* to 1e-5 and verify that the checkbox *Snap new nodes to geometry* is checked. This feature will ensure that new nodes that are generated on the surface mesh during the mesh adaption will be projected to the original geometry, so that the approximation of curved surfaces is improved after each adaptation step. The dialog box should now look as follows:



Close the dialog with the **OK** button and finally start the simulation procedure by clicking **Start**.

Several progress bars like the one depicted below will appear in the status bar informing you about the current solver status.



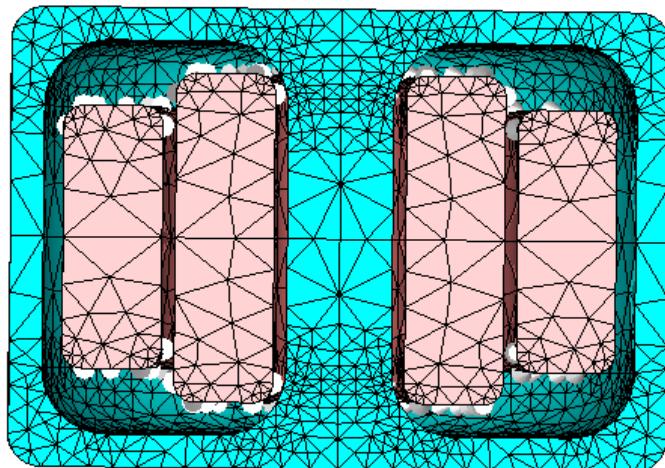
These are the steps of the tetrahedral magnetostatic solver run:

1. **Computing coil(s):** This first calculation step must be performed to calculate the discrete representation of coil current patterns.
2. **Initializing magnetostatic solver:** During this step, your input model is checked for errors such as invalid overlapping materials, not well-defined sources, etc.
3. **Assembling system:** The linear system of equations is generated.
4. **Constructing pre-conditioner:** This includes construction steps for the pre-conditioner of the solver, e.g. an LU-decomposition, a construction of hierarchy for a multigrid solver etc.
5. **Solving linear system:** During this stage, the equation system is solved yielding the unknown field.
6. **Estimating error** (only during mesh adaption pass): The local error for each element is estimated (error distribution).
7. **Marking elements for refinement** (only during mesh adaption pass): Based on the computed error, a certain number of elements will be marked for refinement.
8. **Adapting mesh** (only during mesh adaption pass): The mesh is refined taking the marked elements into account.
9. **Inductance computation** (only if switched on): The apparent and/or incremental inductance matrix is calculated.
10. **Post processing stage:** From the field solution other fields and additional results like the energy within the structure are computed.

If the adaptive mesh refinement is switched on, some of the steps are repeated until a predefined stopping criterion is met.

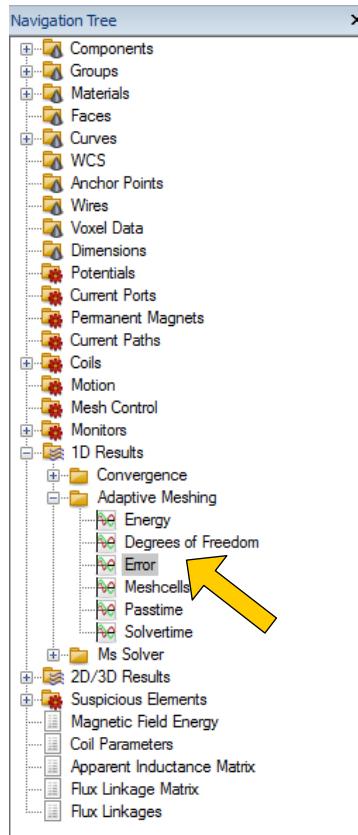
For this simple structure, the entire analysis (including adaptive mesh refinement) usually takes only a few minutes to complete on a today's standard computer.

If you activate the mesh view (*Home: Mesh  $\Rightarrow$  Mesh View* ) while the adaptive solver is running, you can observe how and where the mesh is refined after each pass. After the solver has finished, the mesh should look like depicted in the following picture (deviations are possible since the initial mesh can differ slightly depending on the operating system and the architecture of the machine):



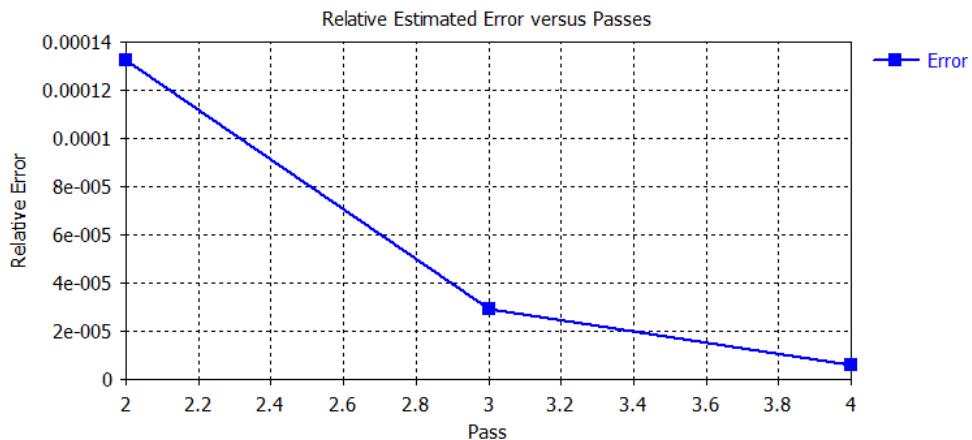
## Analyze the Results of the Tetrahedral Solver

After the solver run you can access the results via the *navigation tree*, see below.

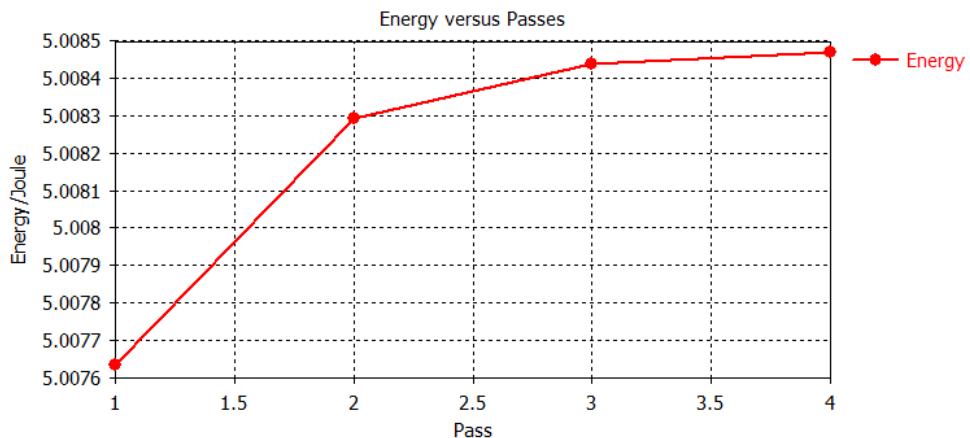


Already while the adaptive solver is running you can watch the progress of the mesh refinement and the convergence behavior in the *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing* folder.

Click, for instance, on *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Error*. This folder contains a curve which displays the change of the relative energy of two subsequent simulations. The curve below shows that the maximum difference of the relative change of the energy is below the desired stopping criterion of 1e-5.



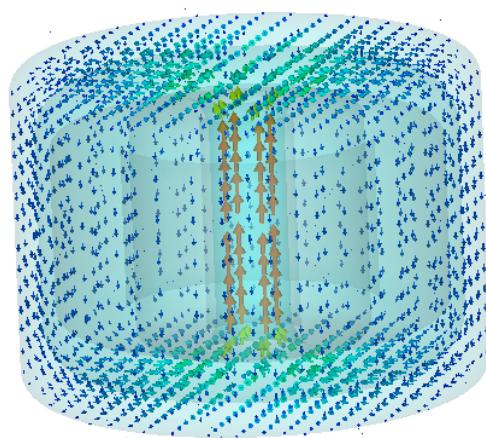
Additionally, the convergence of the energy can be visualized by selecting *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Energy*.



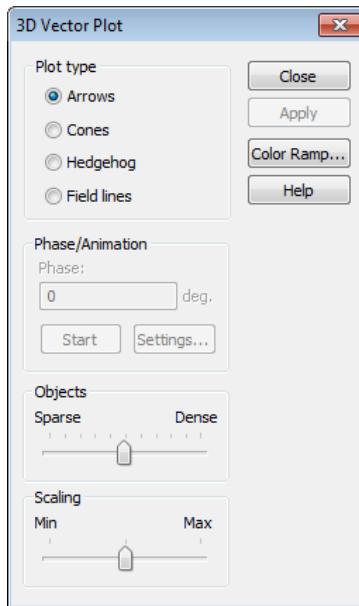
Please remember that the curves can differ slightly when computed on a 32 bit or 64 bit machine. Furthermore, the number of passes needed for convergence can deviate owing to the machine architecture.

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

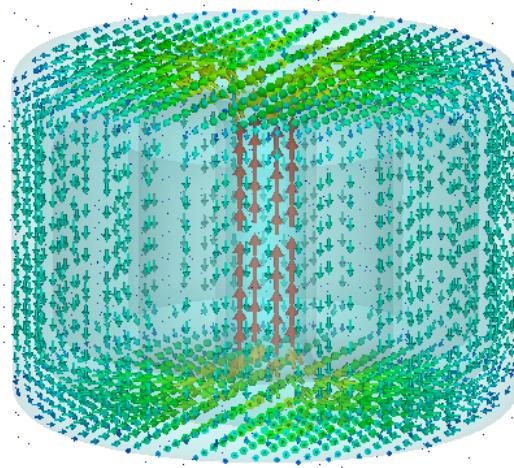
You can visualize the magnetic flux density by choosing *NT: 2D/3D Results*  $\Rightarrow$  *B-Field* to get you an impression of the B-field inside the transformer. After you select this folder, a plot similar to the following should appear:



It might be necessary to adjust the size (scaling) and the density of the arrow objects to obtain a better view. You can modify the plot properties by selecting *2D/3D Plot: Plot Properties* (or by selecting *Plot Properties* from the context menu in the main view). The following dialog box will open:



To enlarge the number or size scaling of the drawn arrow objects, move the sliders in the *Objects* frame to the right. Furthermore, the logarithmic plot option can be activated. If you do this the plot should now look similar to the following picture. Finally, close the *3D Vector Plot* dialog box.



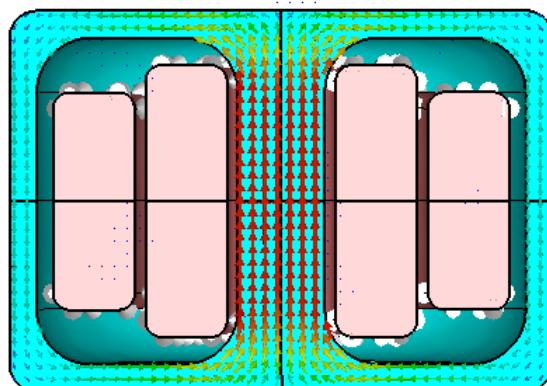
To get an even better view, you can plot the field on a 2D plane. Select **2D/3D Plot: Sectional View**  $\Rightarrow$  **3D Fields on 2D Plane** . Again, to adjust the plot quality, you can select **2D/3D Plot: Plot Properties**  $\Rightarrow$  **Properties**, and move the **Arrows** and **Scaling** sliders.

Before you continue, ensure that the local coordinate system is not active. In order to deactivate the local coordinate system, select **Modeling: WCS**  $\Rightarrow$  **Local WCS**  $\Rightarrow$  **Local WCS** . Note that it may be necessary to click on the **NT: Components** folder first.

After reselecting **NT: 2D/3D Results**  $\Rightarrow$  **B-Field** switch off the “All Transparent” mode by clicking on **2D/3D Plot: Plot Properties**  $\Rightarrow$  **All Transparent** . Furthermore, use the **View** tab to adjust the view properly:

1. Select “Right” from the dropdown list in **View: Change View**.
2. Activate the Plane Rotation Mode (**View: Mouse Control**  $\Rightarrow$  **Rotate in Plane** .
3. Turn the plot 90 degrees by holding the left mouse button and moving the mouse.
4. Select **View: Change View**  $\Rightarrow$  **Reset View**

A plot similar to the following should appear:



Afterwards, switch on the “All Transparent” mode again via *2D/3D Plot: View Options* *All Transparent* and deactivate the 2D plot mode by selecting *2D/3D Plot: Sectional View* *3D Fields on 2D Plane* .

**Please note:** At the right top corner in the main view, you can usually see a color ramp which you can adjust by dragging its small markers or in the plot properties. By default, it is scaled to the overall maximum of the 3D Field you are viewing. From time to time it may happen that, for example, the maximum of an active 2D cut plane is much smaller than the 3D maximum. In order to get a meaningful impression of the field then, it might be necessary to scale the color ramp a bit. This can be done, for example, in the context menu (right-click in the main view) by selecting *Smart Scaling* . To reset the view to the default, select *Reset Scaling* from the context menu.

The inductance matrix was computed after the last adaptive run. The result can be found in the text file *NT: Apparent Inductance Matrix*. A table containing the self- and mutual inductances is shown. The self-inductance of every coil is printed on the main diagonal. The secondary diagonal elements show the mutual inductances.

Apparent Inductance Matrix:

	coil 1	coil 2
coil 1	3.085418e+000 H	2.465616e+000 H
coil 2	2.465616e+000 H	2.000288e+000 H

Finally, let's look at the total magnetic energy in the computational domain. Double-click on *NT: Magnetic Field Energy* to reveal the following:

Magnetic energy in background	:	1.324785e-002 J
Magnetic energy in component1:cylinder box	:	4.978225e+000 J
Magnetic energy in coil 1	:	1.349755e-002 J
Magnetic energy in coil 2	:	3.498575e-003 J
Total magnetic energy	:	5.008469e+000 J

Magnetic co-energy in background	:	1.324785e-002 J
Magnetic co-energy in component1:cylinder box	:	4.978225e+000 J
Magnetic co-energy in coil 1	:	1.349755e-002 J
Magnetic co-energy in coil 2	:	3.498575e-003 J
Total magnetic co-energy	:	5.008469e+000 J

The energy and co-energy is shown for each solid separately. Note that energy and co-energy are exactly the same since only linear materials have been used in the model.

Now leave the text info window by clicking **OK**.

Remember that the major advantage of the tetrahedral mesh is the explicit representation of the geometry, even in the course of adaptive refinement. A proper resolution of non-planar surfaces is very important, in particular, to model jumps in the

field components at material interfaces. For very complex geometries, however, the generation of the tetrahedral mesh is sometimes rather time-consuming and requires a sufficient quality of the CAD data. With the simplicity of hexahedral meshes combined with the Perfect Boundary Approximation an optional method is available.

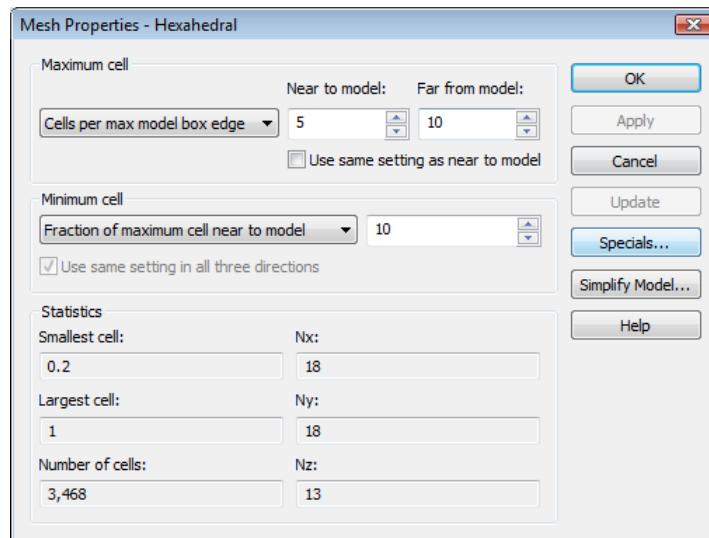
In the following subsections, let's compute the same model applying the hexahedral magnetostatic solver. Again, we will look at the mesh parameters and visualization and then turn to the solver itself.

## Visualize a Hexahedral Mesh

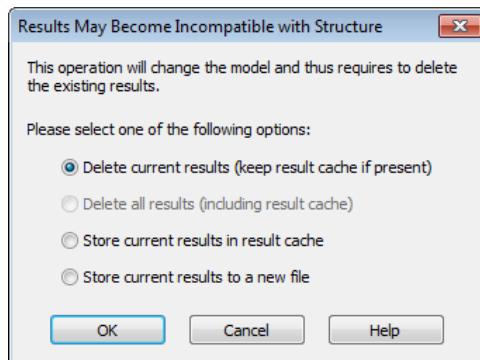
The hexahedral mesh generation for the structure analysis is performed fully automatically based on an expert system. As for tetrahedral meshes, it may be helpful in some situations to inspect the mesh before starting the solver in order to improve the simulation speed by changing the parameters for the mesh generation.

Note that in CST EM STUDIO generating hexahedral meshes is very fast compared to generating tetrahedral meshes. The reason is that by applying the Perfect Boundary Approximation feature, hexahedral meshes do not need to resolve the geometry: i.e. interfaces of materials and solids are not represented by a surface mesh as they are for tetrahedral meshes.

First, you must switch from tetrahedral to hexahedral meshing. Select *Home: Mesh*  $\Rightarrow$  *Global Properties*  $\Rightarrow$  *Hexahedral*. Then *Global Mesh Properties - Hexahedral* dialog box will open automatically. For the purpose of this tutorial, the *Maximum cell - Near to model* value can be changed to 5.

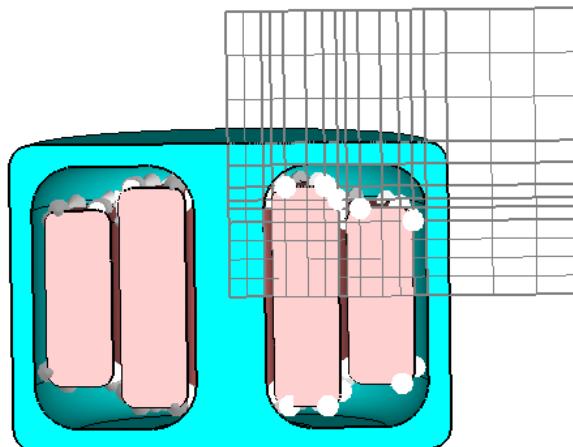


When you click the *OK* button you will be informed that the results have to be deleted.



Confirm the deletion of the results by clicking **OK**.

A hexahedral mesh will be generated automatically without further action. You can visualize the mesh by entering the mesh view (*Home*: *Mesh*  $\Rightarrow$  *Mesh View* ). For this structure, the mesh information will be displayed as follows:



One 2D mesh plane will always be kept in view. Because of the symmetry settings, the mesh only extends across 1/8 of the structure (the mesh plane extends to 1/4). You can modify the orientation of the mesh plane by choosing *Mesh: Sectional View*  $\Rightarrow$  *X/Y/Z Normal* (*X/Y/Z-keys*). You can move the plane along its normal direction with *Mesh: Sectional View*  $\Rightarrow$  *Position* or by pressing the *Up / Down* cursor keys.

In most cases the automatic mesh generation produces a sufficient mesh, but we recommend that you spend some time later on studying the mesh generation procedures in the online documentation once you feel familiar with the standard simulation procedure.

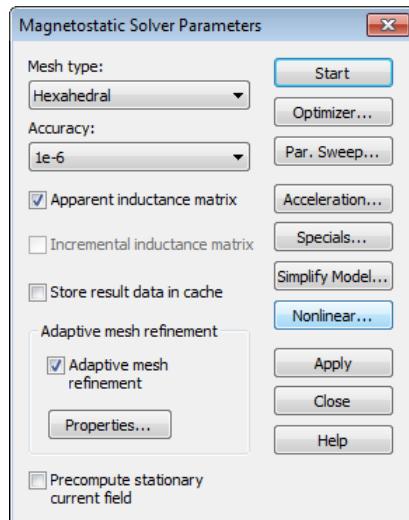
Leave the mesh inspection view by again via *Mesh: Close*  $\Rightarrow$  *Close Mesh View* .

## Start the Hexahedral Solver

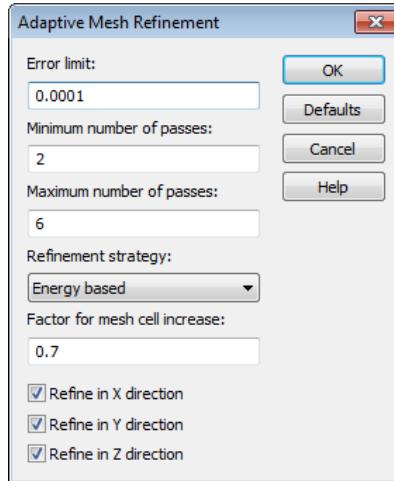
After you have defined all necessary parameters, you are ready to start your first simulation using the hexahedral solver. Again, start the simulation from the magnetostatic solver dialog box: *Home: Simulation  $\Rightarrow$  Setup Solver* . Within the solver dialog box, the "Hexahedral" mesh should be selected in the *Mesh Type* drop-down list. In order to compute inductances from the magnetic field, the box *Apparent inductance matrix* has to be checked. Ensure that the *Adaptive mesh refinement* is switched on (this is not the default for hexahedral meshes). Please recall the remarks on adaptive mesh refinement made in the section *Generate and Visualize a Tetrahedral Mesh*. They apply to hexahedral meshes as well.

The *Accuracy* value can be left unchanged. Please note that what is mentioned concerning the accuracy value in the tetrahedral solver subsection (e.g. its dependence on the discretization) also applies to the hexahedral solver.

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



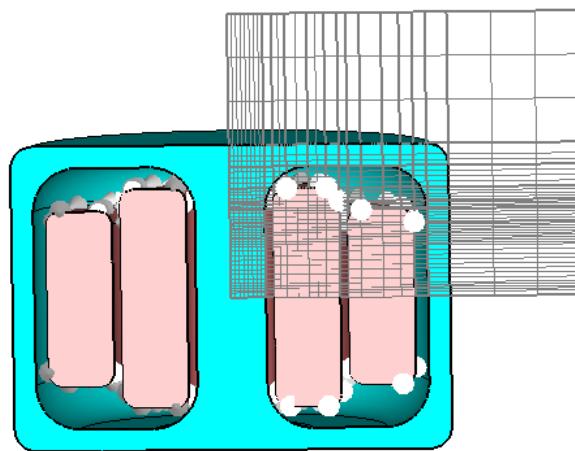
Next enter the *Properties* dialog of the adaptive mesh refinement. The *Error limit* should be changed to 0.0001. The other settings can be kept at their default values.



Confirm your setting by pressing **OK**. Now start the simulation procedure by clicking **Start**. A few progress bars will appear in the status bar to keep you up-to-date with the solver's progress:

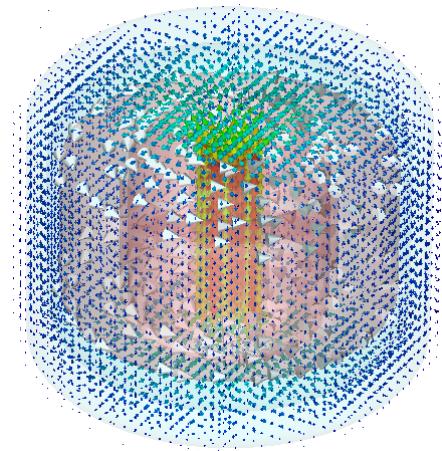
1. **Calculating coil excitations:** This first calculation step must be performed to calculate the discrete representation of coil current patterns.
2. **Checking model:** During this step, your input model is checked for errors such as invalid overlapping materials, etc.
3. **Calculating matrix and dual matrix:** During these steps, the system of equations is set up, which will be solved subsequently.
4. **Solving linear system:** During this stage, a linear equation solver calculates the field distribution inside the structure.
5. **Post-processing:** From the field distribution, additional results like the inductance matrix or the energy within the calculation domain are calculated.

As for the tetrahedral solver, some error estimation and mesh refinement steps are performed in the case of adaptive mesh refinement. Note that several linear systems will be solved during the computation in order to compute all entries of the inductance matrix. For this simple structure, the entire analysis takes only a few seconds per adaption pass. After the simulation the mesh should look similar to this:

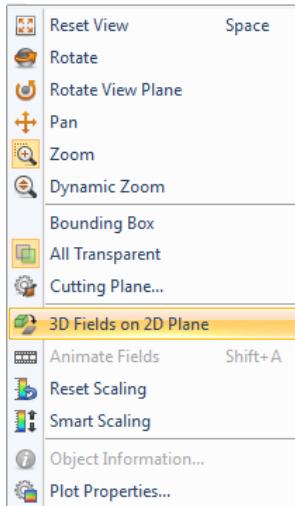


## Analyze the Results of the Hexahedral Solver

Now you can generate similar result plots as you did for the tetrahedral solver-run: Visualize the magnetic flux density by choosing *NT*  $\Rightarrow$  *2D/3D Results*  $\Rightarrow$  *B-Field*. After you select this folder and fine-tune the plot properties in *2D/3D Plot: Plot Properties*  $\Rightarrow$  *Properties*, a plot similar to the following should appear.



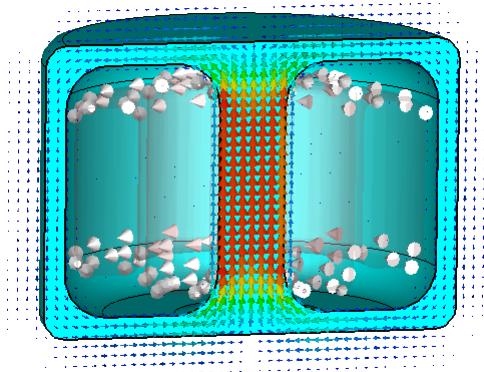
Again, for a 2D-view, you can select *2D/3D Results: Sectional View*  $\Rightarrow$  *3D Fields on 2D Plane* . This is also possible using the context menu by pressing the right mouse button in the main view and selecting *3D Fields on 2D Plane* (see below).



To improve the plot quality, select *2D/3D Results: Plot Properties*  $\Rightarrow$  *Properties* (or select *Plot Properties* from the context menu) and increase the number of arrows and their size by moving the *Arrows* and *Size* slider slightly to the right, then leave the dialog box by clicking *Close*.

Now switch off the “All Transparent” mode within the previously shown context menu (or via *2D/3D Plot: View Options*  $\Rightarrow$  *All Transparent*). Again, use the *View* tab to adjust the view properly: select “Right” from the drop-down list in *View: Change View* and activate the Plane Rotation Mode via *View: Mouse Control*  $\Rightarrow$  *Rotate in Plane*, turn the plot 90 degrees by holding the left mouse button and moving the mouse, and select *View: Change View*  $\Rightarrow$  *Reset View* to adjust the plot size.

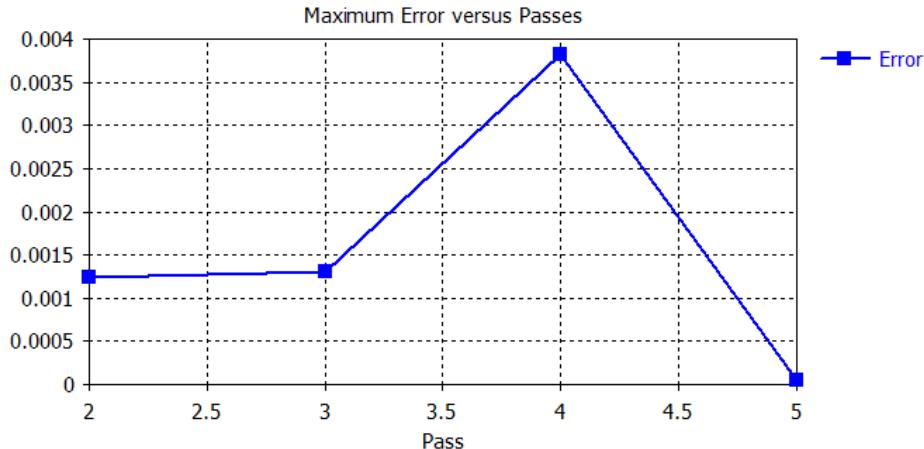
A plot similar to the following should appear:



To observe field values at certain positions, activate *2D/3D Plot: Evaluate Fields*  $\Rightarrow$  *Field at Cursor*. The field values will be displayed in the lower left corner of the main view. Note that for the scalar fields and for the vector fields projected on the plane you can add points to a *List of Field Values* with a double click in the main view.

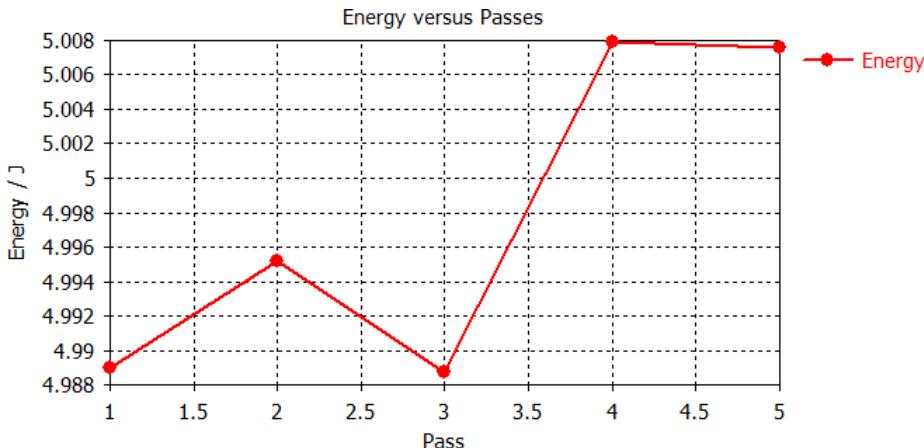
Several mesh refinement passes were performed automatically until the energy value did not change significantly between two subsequent passes. The default termination criterion is an energy deviation of 1% (or less).

The progress of the mesh refinement can be checked in the *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing* folder. This folder contains a curve that displays the energy error of two subsequent simulations. This plot can be viewed by selecting *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Error*:



This result shows that the maximum difference of the energy error is below 0.05 %, i.e. below the error limit prescribed in the adaptive mesh refinement *Properties*.

Additionally, the convergence of the energy can be visualized by selecting *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Energy*:



It can be seen that the hexahedral mesh generator already provides a good mesh for a first calculation. The small energy error shows that the adaptive mesh refinement is able to confirm that variations are reduced to a minimum.

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

Now let's compare the magnetic energy computed by the hexahedral solver to the one computed by the tetrahedral solver. Double-click on *NT: Magnetic Field Energy*. This opens a text-box showing

Magnetic field energy: 5.007607e+000 J

This is very similar to the value computed by the tetrahedral solver. The difference comes from the non-zero discretization errors. Moreover, fewer meshcells have been used for the hexahedral discretization. Leave the text box by clicking *OK*.

In the solver dialog box, you have chosen to calculate the inductance matrix. To view the inductance matrix, select *NT: Apparent Inductance Matrix*:

Apparent Inductance matrix

---

---

	coil 1	coil 2
coil 1	3.084813e+000 H	2.465185e+000 H
coil 2	2.465184e+000 H	2.000035e+000 H

---

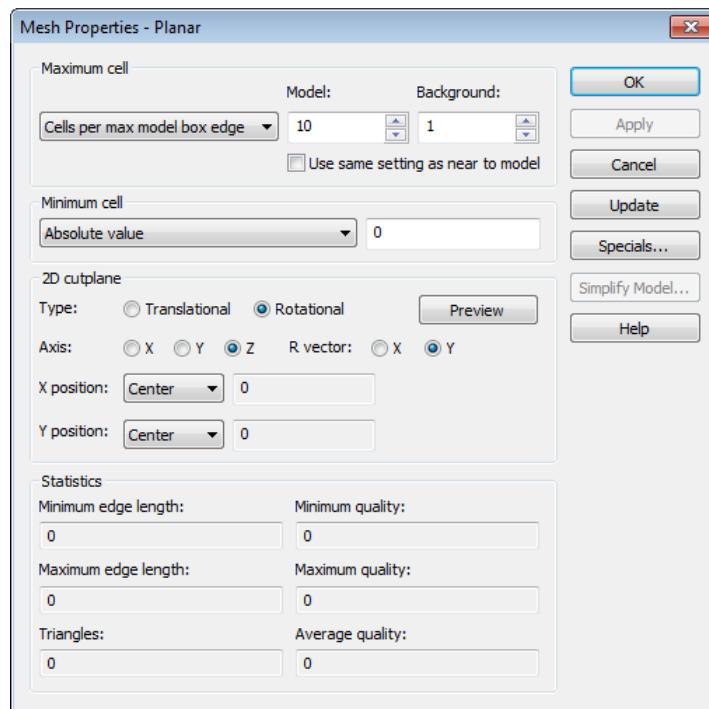
---

The self-inductance of every coil is printed on the main diagonal. The off-diagonal entries provide the mutual inductances. The results are in good agreement with those obtained with the tetrahedral mesh. Leave the text box by clicking *OK*.

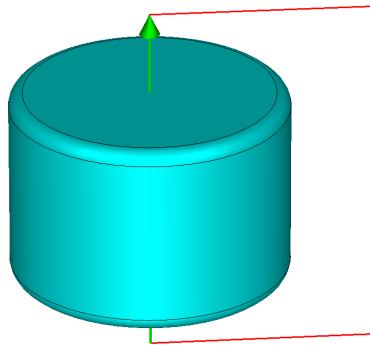
## Create a Planar Mesh

For axis symmetric structures or structures for which boundary effects for one spatial dimension can be neglected, the 2D solver can be applied. The structure is designed as 3D model and cut by a user defined plane. Compared to the 3D solvers, choosing this option might save a lot of computation time. Even if your device is not perfectly symmetric, this solver can give good estimates when starting with a new design.

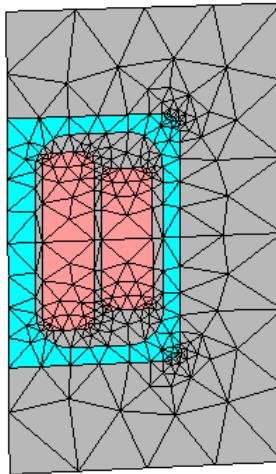
First, you must switch from hexahedral to planar meshing. Select *Home: Mesh  $\Rightarrow$  Global Properties  $\Rightarrow$  Planar*. The cutting plane alignment description as well as the 2D mesh setting are available then in the Mesh Properties dialog box, which will open automatically.



Select *Rotational* for the symmetry type and *Z* for the axis. The axis should be centered in the 3D domain, therefore select *Center* for the X- and Y-position. Finally select *Y* for the *R vector*. The *Preview* button allows checking the settings in the main view:



A first mesh can be created directly with the *Update* button. It will be shown a few seconds later:

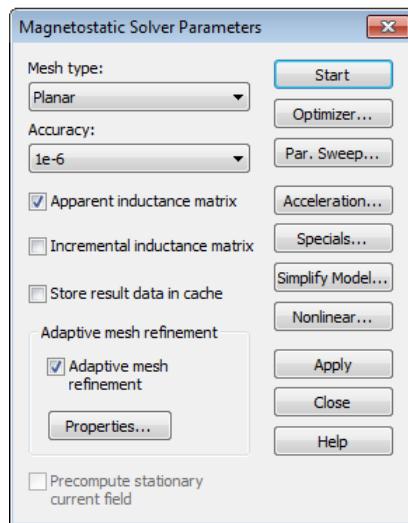


Finally, leave the Mesh Properties dialog box by pressing *OK*.

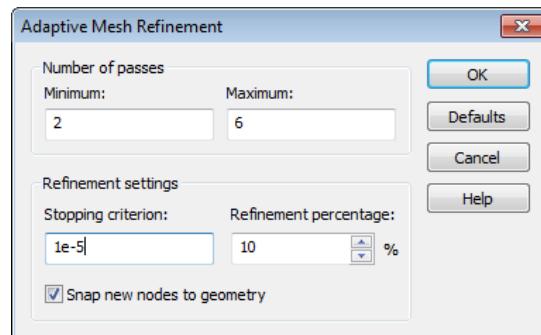
## Start the Planar Solver

After you have defined all the necessary parameters, you are ready to start your first simulation using the planar solver. Again, start the simulation from the magnetostatic solver dialog box: *Home: Simulation*  $\Rightarrow$  *Setup Solver* . Within the solver setup menu, the "Planar" mesh should be selected in the *Mesh type* drop-down list. In order to compute the apparent inductances, the box *Apparent inductance matrix* has to be checked. Ensure that the *Adaptive mesh refinement* is switched on. The *Accuracy* value can be left unchanged.

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



Next, enter the *Properties* dialog of the adaptive mesh refinement. Change the error limit to 1e-5. The other settings can be kept at their default values.



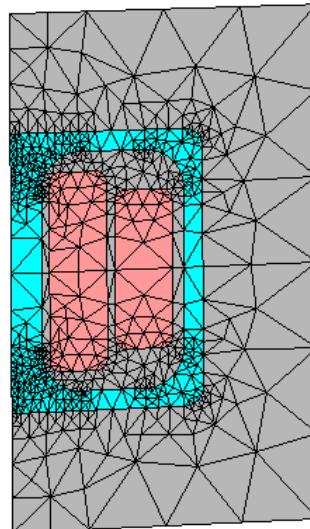
Finally, close the dialog with the *OK* button and start the simulation procedure by clicking on *Start*. Like in the case of the previous simulations, several progress bars will appear in the status bar informing you about the current solver status.

These are the steps of the planar magnetostatic solver run:

1. **Computing coil(s):** This first calculation step must be performed to calculate the discrete representation of coil current patterns.
2. **Initializing magnetostatic solver:** During this step, your input model is checked for errors such as invalid overlapping materials, not well-defined sources, etc.
3. **Assembling system:** The linear system of equations is generated.

4. **Constructing pre-conditioner:** This includes construction steps for the pre-conditioner of the solver, e.g. an LU-decomposition, a construction of hierarchy for a multigrid solver etc.
5. **Solving linear system:** During this stage, the equation system is solved yielding the unknown field.
6. **Estimating error** (only during mesh adaption pass): The local error for each element is estimated (error distribution).
7. **Marking elements for refinement** (only during mesh adaption pass): Based on the computed error, a certain number of elements will be marked for refinement.
8. **Adapting mesh** (only during mesh adaption pass): The mesh is refined taking the marked elements into account.
9. **Inductance computation** (only if switched on): The apparent and/or incremental inductance matrix is calculated.
10. **Post processing stage:** From the field solution other fields and additional results like the energy within the structure are computed.

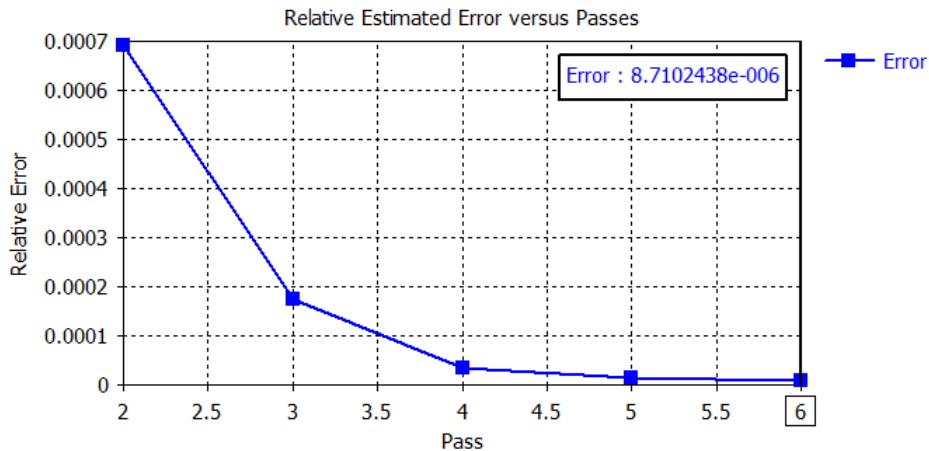
After the solver has finished, the mesh should look similar to the one depicted in the following picture (deviations are possible since the initial mesh can differ slightly depending on the operating system and the architecture of the machine):



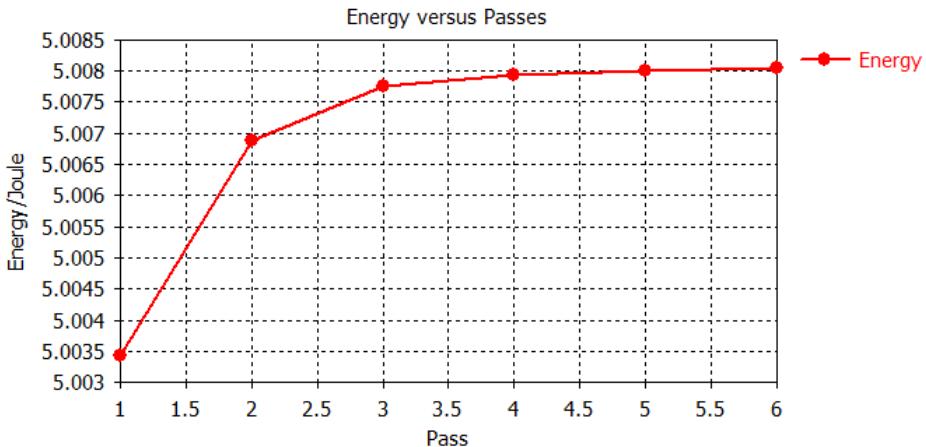
## Analyze the Results of the Planar Solver

Already during the planar solver run, you can watch the progress of the mesh refinement and the convergence behavior in the *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing* folder.

Click, for instance, on *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Error*. This folder contains a curve which displays the change of the relative energy of two subsequent simulations. From this result, we can observe that the maximum difference of the relative change of the energy is below the desired stopping criterion 1e-5:

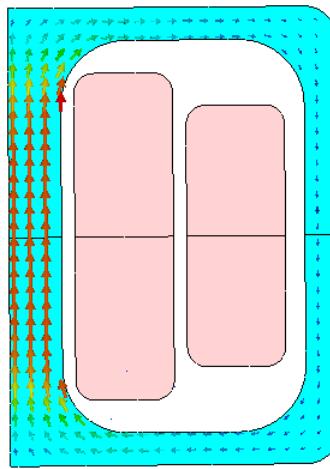


Additionally, the energy convergence can be visualized by selecting in the *NT: 1D Results*  $\Rightarrow$  *Adaptive Meshing*  $\Rightarrow$  *Energy*:



The curves can slightly differ when computed on a 32 bit or a 64 bit machine. The number of adaptation passes needed for convergence can also deviate depending on the machine architecture.

Now, you can visualize the magnetic flux density by choosing *NT: 2D/3D Results*  $\Rightarrow$  *B-Field*. After you select this folder and fine-tune the plot properties in *2D/3D Plot: Plot Properties*, a plot similar to the following one should appear:



Completing the analysis of the planar solver results, let's compare the magnetic energy and the apparent inductance values computed by this solver to the ones computed by the 3D solvers.

To view the magnetic energy result, double click on *NT: Magnetic Field Energy*. This opens a text-box showing

Magnetic energy in background	:	1.270819e-002 J
Magnetic energy in component1:cylinder box	:	4.978334e+000 J
Magnetic energy in coil 1	:	1.350715e-002 J
Magnetic energy in coil 2	:	3.497639e-003 J
<hr/>		
Total magnetic energy	:	5.008047e+000 J
<hr/>		
Magnetic co-energy in background	:	1.270818e-002 J
Magnetic co-energy in component1:cylinder box:	:	4.978334e+000 J
Magnetic co-energy in coil 1	:	1.350715e-002 J
Magnetic co-energy in coil 2	:	3.497639e-003 J
<hr/>		
Total magnetic co-energy	:	5.008047e+000 J

These results are very similar to the ones computed by the tetrahedral and hexahedral solvers. Leave the text box by clicking *OK*.

The results for the apparent inductance computation can be found in the text file *NT: Apparent Inductance Matrix*. The self-inductance of every coil is printed on the main diagonal while mutual-inductances are the off-diagonal entries.

Apparent Inductance Matrix:

	coil 1	coil 2
coil 1	3.085161e+000 H	2.465610e+000 H
coil 2	2.465410e+000 H	2.000113e+000 H

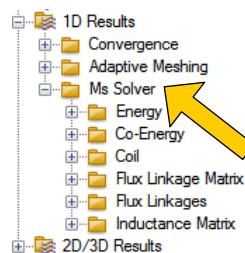
These results are also in good agreement with those computed with the 3D solvers. Leave the text box by clicking *OK*.

## Accessing the Single-Value Results

CST EM STUDIO offers various options to access the single-value results which are briefly discussed in what follows.

### Access from the Navigation Tree

A big portion of single-value results relevant for the specific solver can be found in the *NT*. For the magnetostatic solver, result values for energy, co-energy, coil characteristics, flux linkages and inductances are quickly accessible from *NT:1D Results*  $\Rightarrow$  *Ms Solver* folder:

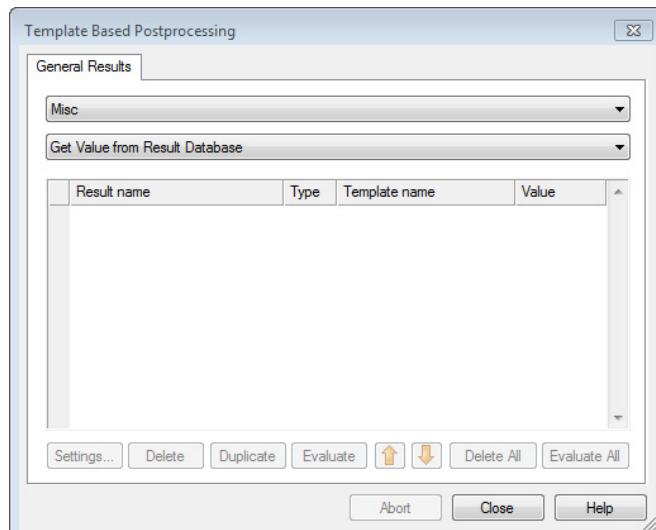


The same data and more complex post processing results are also available via the *Template Based Post Processing* tool as described below.

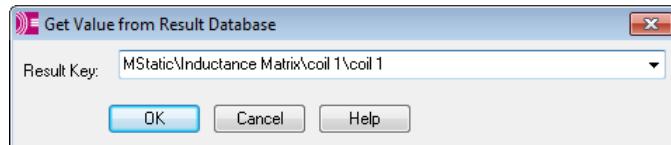
### Access via the Template Based Post Processing Tool

All single-value results are stored in result database which can be accessed via the *Template Based Post Processing* tool. In the subsequent workflow, we would like to trace the inductances for multiple solver runs. Therefore, open the *Template Based Postprocessing* dialog box via *Post Processing: Result Templates*  $\Rightarrow$  *Template Based Post Processing* (*Shift+P*) .

Select the template group *Misc* and the postprocessing step *Get Value from Result Database* to access the values from the result database.

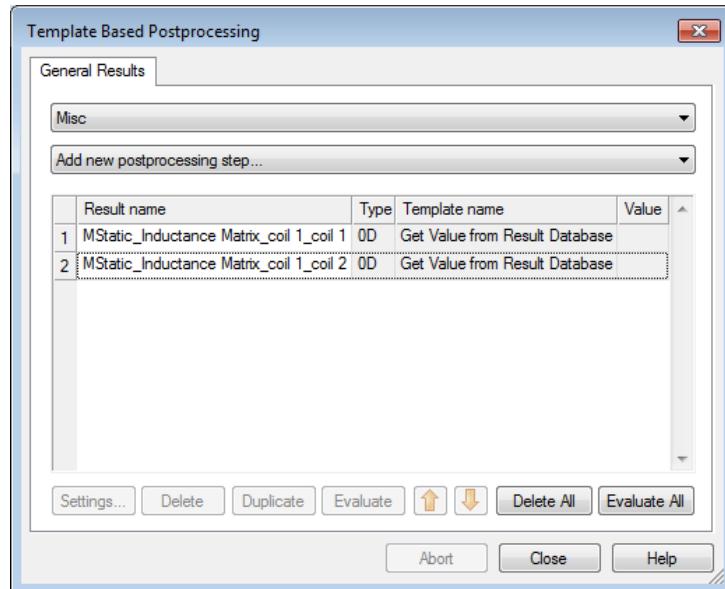


Another dialog box will pop up, where you can select the result in which you are interested. Please note that database entries can be found in the corresponding dropdown list only when the results from a previous calculation are still available. In this case select *MStatic\Inductance Matrix\coil 1\coil 1*.



If you have deleted the results beforehand, you have to enter the corresponding string *MStatic\Inductance Matrix\coil 1\coil 1* into the edit field.

In this example, we would like to watch the self-inductance of coil 1 and the mutual inductance between coil 1 and coil 2. Therefore, define two templates in the described way, each confirming with the OK button. The *Template Based Postprocessing* dialog box should finally look as follows:



When you press the *Evaluate All* button (while solver results are available), the results previously seen in the *Apparent Inductance Matrix* text file will appear in the *Value* column. Finally, leave the dialog box with the *Close* button.

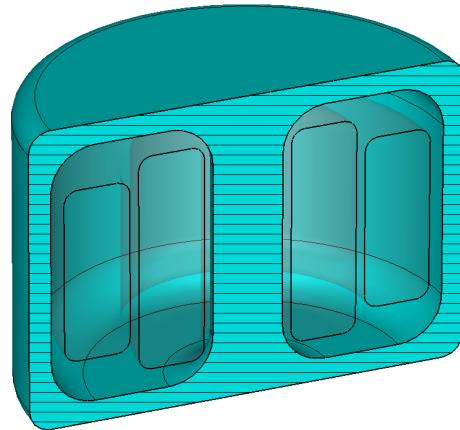
## Parameterization and the Automatic Optimization of the Structure

The steps above demonstrate how to enter and analyze a simple structure. However, structures are usually analyzed to improve their performance. This procedure is called “design” in contrast to “analysis”.

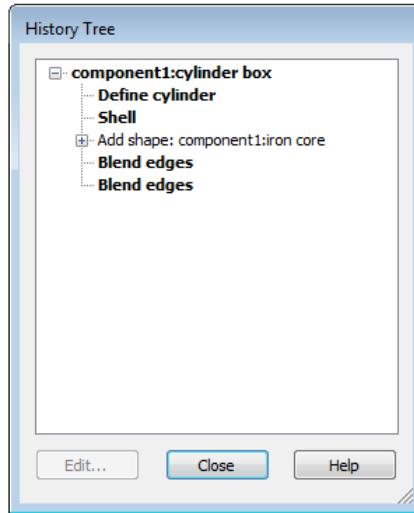
After you receive some information on how to improve the structure, you will need to change the structure’s parameters. This could be done by simply re-entering the structure but this is not the most efficient solution.

CST EM STUDIO offers various options to describe the structure parametrically in order to change the parameters easily. The *History List* function, described in the *CST STUDIO SUITE Getting Started* manual, is a general option, but for simple parameter changes there is an easier solution, which is described below.

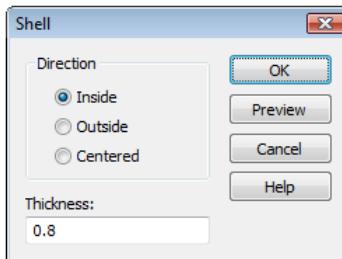
Let’s assume you want to change the thickness of the transformer’s box. The easiest way to do this is to select the box by clicking on *NT: Components*  $\Rightarrow$  *component1*  $\Rightarrow$  *cylinder box*. You may also need to rotate the structure in order to see a plot similar to the following (the cutting plane is still switched on):



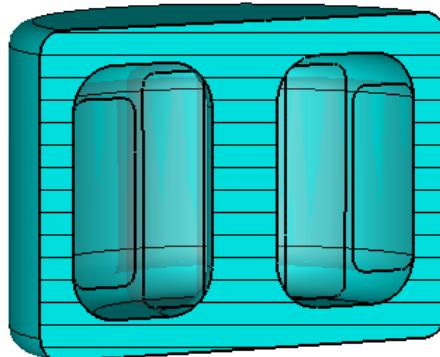
You can now choose *Modeling: Edit*  $\Rightarrow$  *Properties* (*Ctrl+E*) (or *Properties* from the context menu) to open a list showing the history of the shape’s creation:



Select the “Shell” operation from the history tree (see above). After you click *Edit*, the shell dialog box, you will find the thickness of the box (*Thickness = 0.5*) as specified during the shape creation. Change this parameter to a value of 0.8 and click *OK*.



Confirm the deletion of the results by clicking *OK*.  
The structure plot will change showing the new structure with the different box thickness:

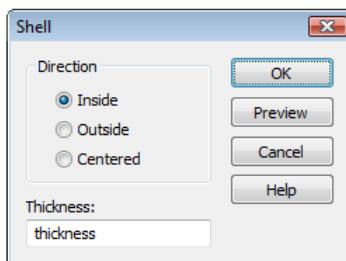


You can generally change all parameters of any shape by selecting the shape and editing its properties. This fully parametric structural modeling is one of CST EM STUDIO's most outstanding features.

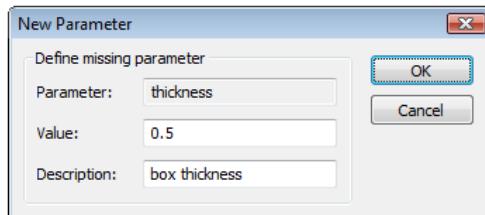
The parametric structure definition also works if some objects have been constructed relative to each other using local coordinate systems. In this case, the program will try to identify all the picked faces according to their topological order rather than their absolute position in space.

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

You may also assign variables to the structure parameters: Select the “Shell” operation from the history tree again (the dialog box should be still open) and click *Edit*. Now enter the string “thickness” as depicted below.



Then click *OK*. A new dialog box will open asking you to define the new parameter “thickness.” Here enter 0.5 in the *Value* field. You may also provide a text in the *Description* field so that you can later remember the meaning of the parameter.



Closing this dialog box by clicking *OK* defines the parameter and updates the model. Now also close the History Tree window. Note that all defined parameters are listed in the parameter docking window:

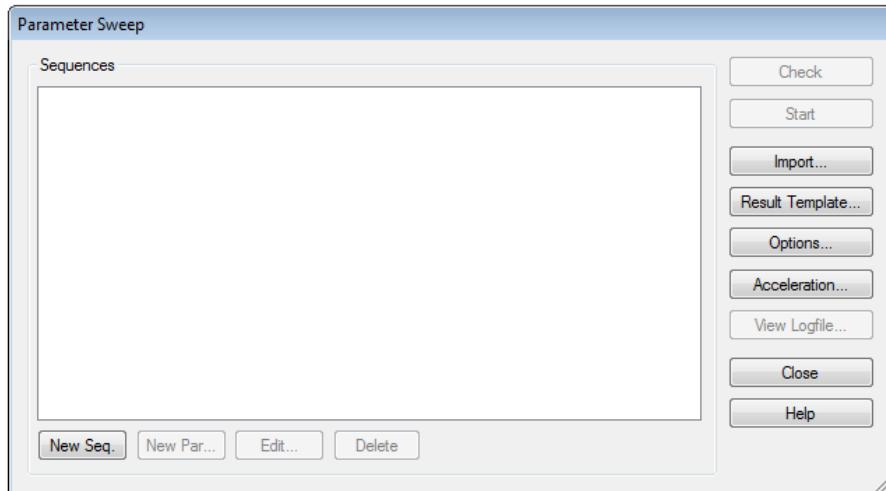
Parameter List				
	Name	Expression	Value	Description
	thickness	= 0.5	0.5	None
<new variable>				

You can change the value of parameters by clicking on the corresponding entry in the *Value* column of the parameter window and entering a new value. If you do this the message “*Some variables have been modified. Press ‘Home: Edit ⇨ Parametric Update*  (F7)” will appear in the main view. Note that the entry in the *Type* column can be ignored.

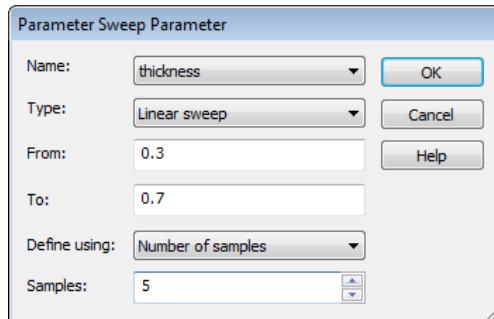
When you perform this update operation, the structure will be regenerated according to the current parameter value. You can verify that parameter values between 0.3 and 0.7 give useful results. The function *Modeling: Edit*  $\Rightarrow$  *Parameters*  $\Rightarrow$  *Animate Parameter* is also useful in this regard.

It is also possible to define a new parameter by entering it in the parameter window.

Since you now parameterized your structure successfully, it might be interesting to see how the apparent inductance values change when the thickness of the box is varied. The easiest way to obtain these variation results is to use the Parameter Sweep tool accessible from within the magnetostatic solver dialog box (*Simulation: Solver*  $\Rightarrow$  *Setup Solver* ). Note that the *Planar Mesh* type with *Adaptive mesh refinement* is still selected. Click the *Par. Sweep...* button to open the following dialog box:



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



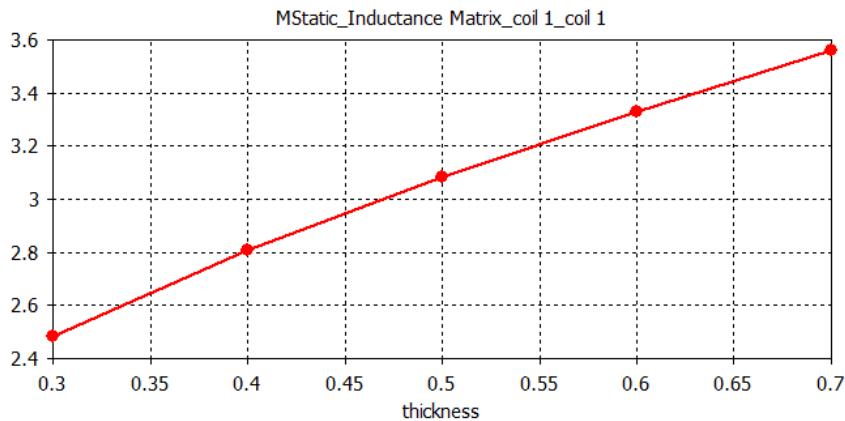
In the dialog box that appears, you can select the name of the parameter to vary in the *Name* drop-down list. After selecting the item to sweep, you can specify the lower (*From*) and upper (*To*) bounds for the parameter variation. Finally enter the number of steps in which the parameter should be varied in the *Samples* field.

In this example the *thickness* of the box should be swept *From 0.3 To 0.7 in 5 Samples*. After you click *OK*, the parameter sweep setting will appear in the *Sequences* frame. Note that you can define an arbitrary number of sequences each containing an unlimited number of different parameter combinations.

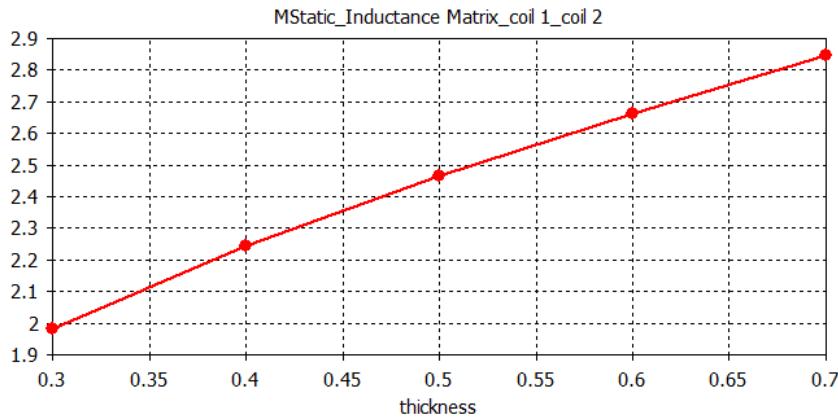
In the next step you must specify which results you are interested in. These are traced by the parameter sweep. Therefore you should click the *Result template* button to open the *Template Based Postprocessing* dialog box. As we have defined the result templates of interest already, you can simply verify that the *Inductance Matrix* templates for *coil 1/coil 1* and *coil1/coil 2* are listed, and then close the dialog box again with *Close*.

Back in the *Parameter Sweep* dialog box, run the parameter sweep by clicking *Start*. A progress bar in the *Progress* window shows the current status of the parameter sweep.

After the solver has finished its work, leave the *Parameter Sweep* dialog box by clicking *Close*. The navigation tree will contain a new item called “Tables” from which you should select *NT: Tables  $\Rightarrow$  0D Results  $\Rightarrow$  MStatic\_Inductance Matrix\_coil 1\_coil 1* first. You should get a plot similar to the following:



By selecting *NT: Tables  $\Rightarrow$  0D Results  $\Rightarrow$  MStatic\_Inductance Matrix\_coil 1\_coil 2*, you can plot the mutual inductance in the same way:



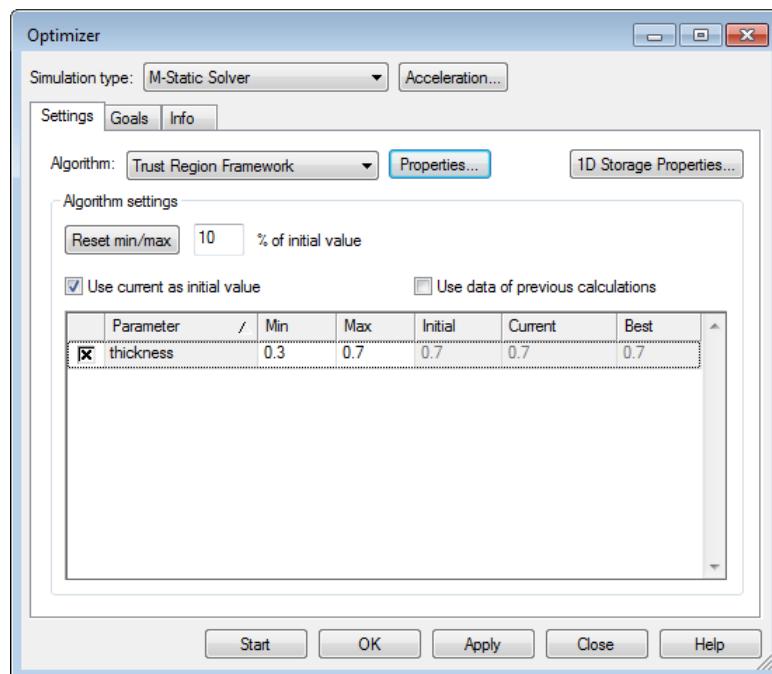
Besides the definition of result templates, it is also possible to store any 1D results automatically during parametric solver runs. Please refer to the online documentation

and the *CST STUDIO SUITE – Getting Started* manual for more information about the many options for storing and displaying parametric data.

Assume that you now want to adjust the self-inductance of coil 1 to a value of 3.2 H (which can be achieved within a parameter range of 0.3 to 0.7 according to the parameter sweep). However, figuring out the proper parameter may be a lengthy task that can be performed equally well automatically.

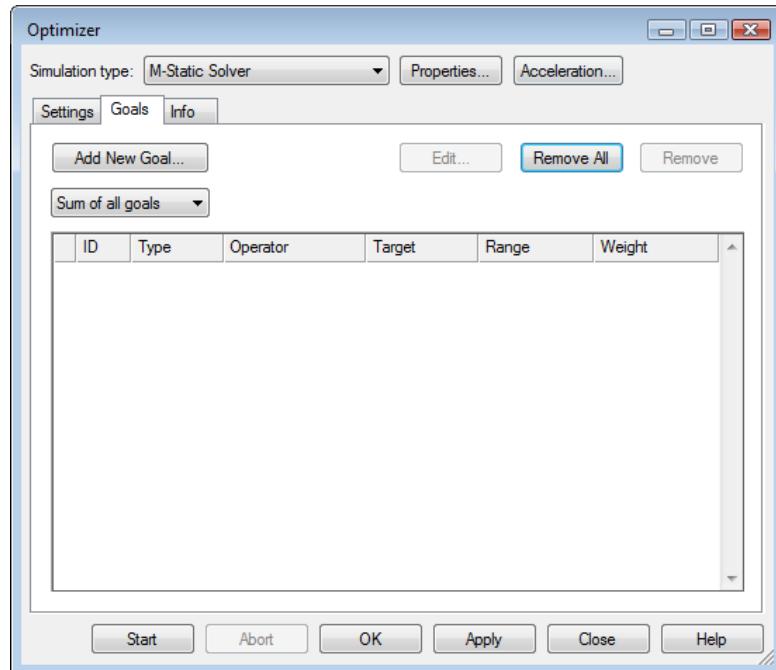
Before you continue to optimize this structure, ensure the thickness parameter is within the valid parameter range (e.g. 0.5). If you have to modify the value, don't forget to update the structure afterwards. Note that you must enter the modeler mode, e.g. by clicking on the “Components” item in the navigation tree, before you modify the parameters.

CST EM STUDIO offers a very powerful built-in optimizer feature for parametrical optimizations. To open the optimizer control dialog box, select *Simulation: Solver*  $\Rightarrow$  *Optimizer*:



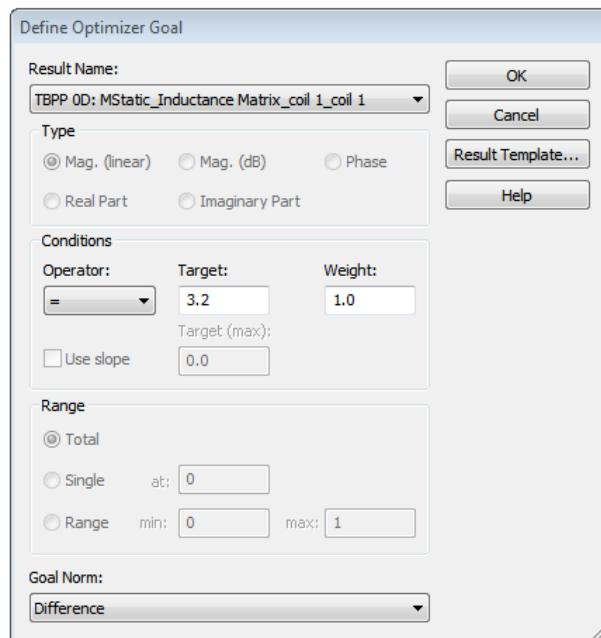
First, check the desired parameter(s) for the optimization in the *Parameter* tab of the optimization dialog box (here the “thickness” parameter should be checked). Next specify the minimum and maximum values for this parameter during the optimization. Here you should enter a parameter range between 0.3 and 0.7. Refer to the online documentation for more information on these settings.

Next specify the optimization goal. Therefore you should click on the *Goals* tab.

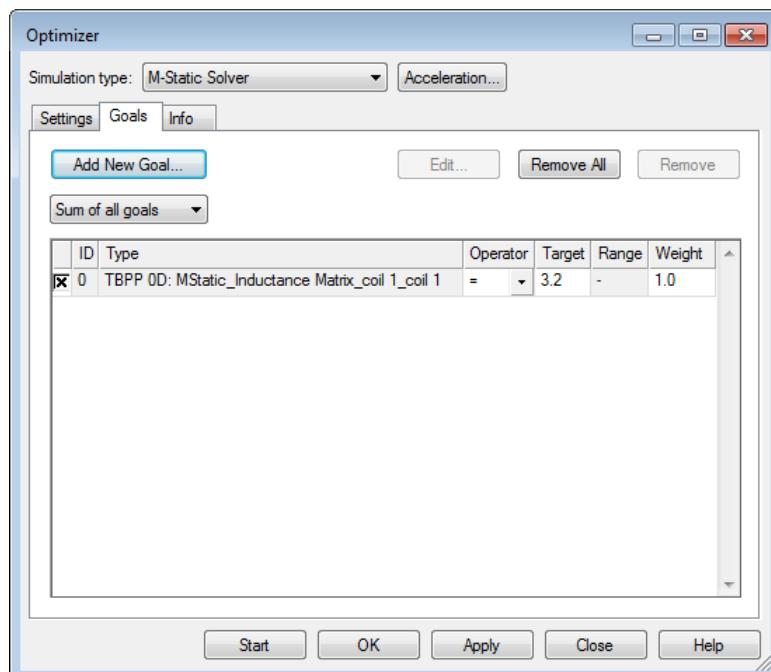


Here you can specify a list of goals to be achieved during the optimization. In this example, the target is to find a parameter value for which the self -inductance of coil 1 is 3.2 H. Therefore you should use a 0D postprocessing template to define the goal.

To this end, click on the *Add New Goal* entry. A new dialog box will open: *Define Optimizer Goal*. For the parameter sweep, we have just defined two postprocessing templates which we can use for the optimizer as well. Since you want to find the thickness value for self-inductance of 3.2 H, select the corresponding result name (*TBPP 0D: MStatic Inductance Matrix\_coil 1\_coil 1*) and the equal operator in the *Conditions* frame and set the *Target* to 3.2:

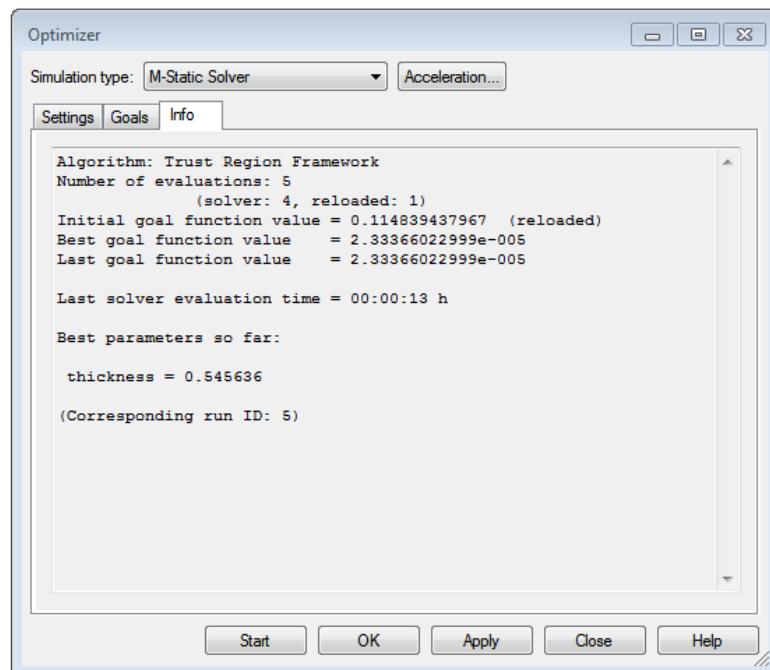


After you click **OK**, the optimizer dialog box should look as follows:



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

When the optimization is done, the optimizer output window shows the best parameter settings to achieve the optimization goal.



Note that due to the sophisticated optimization technology, only a few solver runs were necessary to find the optimal solution with high accuracy.

Now look at the inductance matrix for the optimal parameter setting (thickness = 0.545636) by clicking *NT: Apparent Inductance Matrix*. The computed inductance is very close to the target value:

Apparent Inductance Matrix:

	coil 1	coil 2
coil 1	3.200023e+000 H	2.557707e+000 H
coil 2	2.557707e+000 H	2.074518e+000 H

This ends the first application example.

## Summary

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

1. Model the structures by using the solid modeler
2. Specify the solver parameters, check the mesh and start the simulation using the tetrahedral solver with the adaptive mesh refinement feature
3. Specify the solver parameters, check the mesh and start the simulation using the hexahedral solver with the adaptive mesh refinement feature
4. Visualize the magnetic field distributions
5. Specify the solver parameters, check the mesh and start the simulation using the planar solver with the adaptive mesh refinement feature
6. Define the structure using structure parameters
7. Use the parameter sweep tool and visualize parametric results
8. Perform automatic optimizations

If you are familiar with all these topics, you have a very good starting point for further improvements of your usage of CST EM STUDIO.

For more information on a particular topic, we recommend you browse through the online help system which can be opened via the Help  button in the upper right corner. If you have any further questions or remarks, do not hesitate to contact your technical support team. We also strongly recommend that you participate in one of our special training classes held regularly at a location near you. Ask your support center for details.

# Chapter 3 — Solver Overview

## Solvers and Sources

The previous example demonstrates how to define a coil source for a magnetostatic simulation. The general workflow of electrostatic, stationary current or low-frequency problems is quite similar to a magnetostatic application.

The different simulation types differ in the definition of materials, boundary conditions and excitation sources. The way to define materials and boundary conditions in CST EM STUDIO is quite similar for all solvers, whereas there are larger differences in the definition of sources. For this reason an overview of the sources that are interpreted by each solver is given below.

### Magnetostatic Solver:

- Permanent magnet:  
*Simulation: Sources and Loads*  $\Rightarrow$  Permanent Magnet 
- Current or voltage coil:  
*Simulation: Sources and Loads*  $\Rightarrow$  Coil  $\Rightarrow$  Coil 
- Coil segment:  
*Simulation: Sources and Loads*  $\Rightarrow$  Coil  $\Rightarrow$  Coil segment 
- Current path:  
*Simulation: Sources and Loads*  $\Rightarrow$  Current Path 
- External magnetic field:  
*Simulation: Sources and Loads*  $\Rightarrow$  Magnetic Source Field 
- Stationary current field (via Solver checkbox) – the most important stationary current sources (*Electric Potential*  and *Current Port* 

Typical applications are: magnets, magnetic valves, actuators, motors, generators and sensors

### Electrostatic Solver:

- Potential definition on a PEC (perfect electric conductor) solid:  
*Simulation: Sources and Loads*  $\Rightarrow$  Electric Potential 
- Capacitive field grading on a PEC:  
*Simulation: Sources and Loads*  $\Rightarrow$  Electric Potential   $\Rightarrow$  Field Grading 
- Potential definition on a normal/electric boundary:  
*Simulation: Settings*  $\Rightarrow$  Boundaries   
(select the *Boundary Potentials* tab from within the Boundary dialog)
- Charge definition on a PEC:  
*Simulation: Sources and Loads*  $\Rightarrow$  Electric Charge on PEC 
- Uniform volume- or surface-charge distribution:  
*Simulation: Sources and Loads*  $\Rightarrow$  Electric Charge Distribution 

Typical applications are: high voltage devices, capacitors, MEMS and sensors.

**Stationary Current Solver:**

- Potential definition on a PEC solid:  
*Simulation: Sources and Loads  $\Rightarrow$  Electric Potential* 
- Current port:  
*Simulation: Sources and Loads  $\Rightarrow$  Current Port* 
- Field import:  
*Simulation: Sources and Loads  $\Rightarrow$  Field Import* 
- Current path:  
*Simulation: Sources and Loads  $\Rightarrow$  Current Path* 
- Coil segment:  
*Simulation: Sources and Loads  $\Rightarrow$  Coil segment* 

Typical applications are: sensors, coils, circuit breakers, IR drop simulations and grounding problems.

**LF Frequency Domain Solver (Full Wave and Magnetoquasistatics):**

- Current or voltage coil:  
*Simulation: Sources and Loads  $\Rightarrow$  Coil  $\Rightarrow$  Coil* 
- Coil segment:  
*Simulation: Sources and Loads  $\Rightarrow$  Coil  $\Rightarrow$  Coil segment* 
- Current port:  
*Simulation: Sources and Loads  $\Rightarrow$  Current Port* 
- Current path:  
*Simulation: Sources and Loads  $\Rightarrow$  Path Sources  $\Rightarrow$  Current Path From Curve* 
- Voltage path:  
*Simulation: Sources and Loads  $\Rightarrow$  Path Sources  $\Rightarrow$  Voltage Path from Curve* 
- External magnetic field:  
*Simulation: Sources and Loads  $\Rightarrow$  Magnetic Source Field* 
- Field import:  
*Simulation: Sources and Loads  $\Rightarrow$  Field Import* 

**LF Frequency Domain Solver (Electroquasistatics):**

- Potential definition on a PEC solid:  
*Simulation: Sources and Loads  $\Rightarrow$  Electric Potential* 

Typical applications are: NDT, proximity sensors, inductively coupled power transfer, induction heating, magnetic and electric design of transformers.

**LF Time Domain Solver (Magnetoquasistatics):**

- Permanent magnet:  
*Simulation: Sources and Loads  $\Rightarrow$  Permanent Magnet* 
- Current or voltage coil:  
*Simulation: Sources and Loads  $\Rightarrow$  Coil  $\Rightarrow$  Coil* 
- Coil segment:  
*Simulation: Sources and Loads  $\Rightarrow$  Coil  $\Rightarrow$  Coil segment* 
- Current port:  
*Simulation: Sources and Loads  $\Rightarrow$  Current Port* 
- Current path:  
*Simulation: Sources and Loads  $\Rightarrow$  Path Sources  $\Rightarrow$  Current Path From Curve* 
- Voltage path:  
*Simulation: Sources and Loads  $\Rightarrow$  Path Sources  $\Rightarrow$  Voltage Path from Curve* 
- External magnetic field:  
*Simulation: Sources and Loads  $\Rightarrow$  Magnetic Source Field* 

- Motion:  
*Simulation: Motion* 

**LF Time Domain Solver (Electroquasistatic):**

- Potential definition on a PEC solid:  
*Simulation: Sources and Loads* 

Typical applications are: transient device switching, nonlinear time-dependent problems such as electrical machines, sensors and high-voltage transformers.

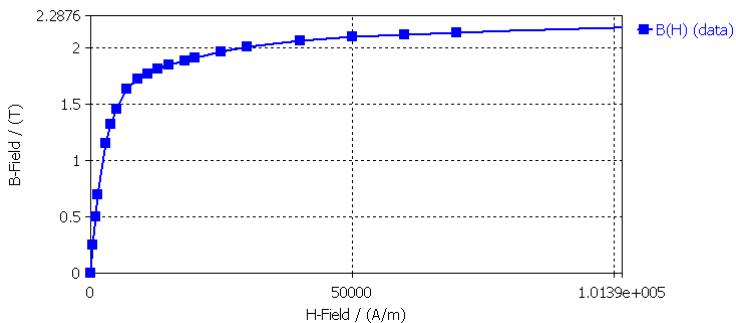
## Magnetostatic Solver

The magnetostatic solver can be used for static magnetic problems. Available sources are current paths, current or voltage coils, coil segments, permanent magnets and homogeneous magnetic source fields as well as the current density field previously calculated by the *stationary current solver*. To use the J-static current density field as magnetostatic source, activate the checkbox *Precompute stationary current field* in the Magnetostatic Solver dialog box. The stationary current field will then be precomputed automatically.

The main task of the solver is to calculate the magnetic field strength and the flux density. These results appear automatically in the navigation tree after the solver run.

## Nonlinear Materials

The magnetostatic solver also features nonlinear materials. These can be defined by creating a BH-curve describing the material. A nonlinear solver will use a smoothed version of this curve in order to improve the convergence. The resulting permeability distribution is also stored and can be accessed in the navigation tree.



## Inductance Calculation

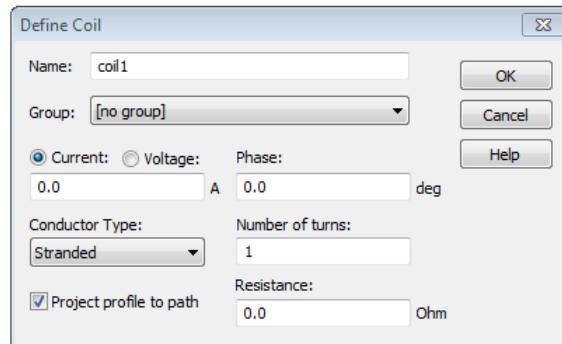
The magnetostatic solver can extract the inductances matrices of coils and coil segments. For nonlinear material properties, the saturation of the material is taken into account. The user may choose between the extraction of the apparent inductance matrix and/or the incremental inductance matrix. For  $n$  coils and coil segments, the computation of each inductance matrix requires the solution of  $n$  equation systems. For constant material properties (i.e. type *Normal*), the apparent and the incremental inductances are identical.

## Current or Voltage Coils

In the section *Define Coils* of the previous chapter, the main ideas of the simulation of coils in CST EM STUDIO are already outlined. Moreover, you can find there a step-by-step description of a coil creation.

Remember that a current coil is defined as an *a-priori* known current distribution which is constant over the cross-section of the coil body. The supporting material has no influence on the source current distribution.

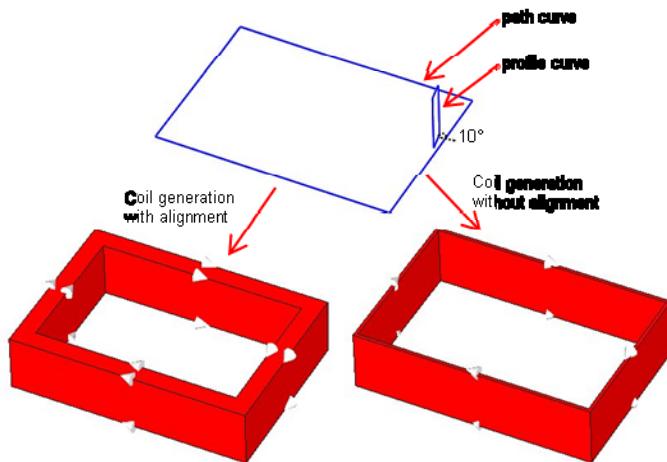
A coil in CST EM STUDIO can be constructed from two curves – the *profile curve* and the *path curve*. To create a current coil, you must define these two curves and then select *Simulation: Sources and Loads*  $\Rightarrow$  *Coil* . You will be prompted to select the coil profile curve and then the coil path curve. When the profile curve can be swept along the path curve successfully, the *Define Coil* dialog box will open automatically.



In this dialog box, you can specify the *Name*, the *Group* and the *Conductor Type* (*Stranded* or *Solid*) as well as the *current* or *voltage* value, the *Number of turns* and the *Ohmic Resistance* of a coil. The *Phase* value is relevant only for LF Frequency Domain simulations.

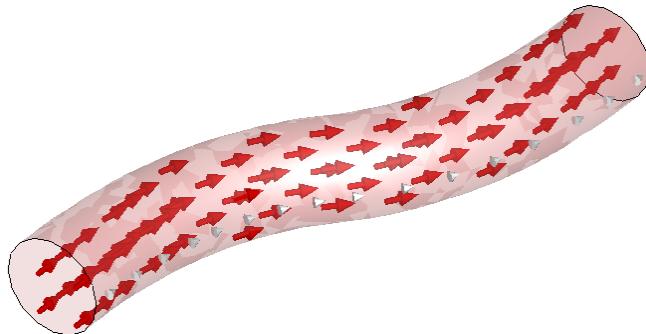
Depending on the physical connections, coil sources can be gathered into so-called coil groups. A current coil group is represented by a series connection of coils characterized by a common current flowing through them. In the absence of mutual inductances or magnetical coupling between the coils, the total inductance of a current coil group can be calculated by just simply summing up the individual coil inductances. In a voltage coil group, the voltage drop across all the coils connected in parallel is the same. For this type of coil connection and in the absence of an additional coupling between the coils, the sum of the reciprocal values of the individual inductances results in the reciprocal value of the group inductance. For more detailed information concerning the coil groups, please refer to the online help.

When the *Project profile to path* checkbox is activated, the profile curve is aligned with the plane which is normal to the path curve. In the following example you can see the profile curve which includes an angle of 10 degrees with the path curve. The coil on the left hand side will be obtained if the alignment is activated. To generate the coil displayed on the right hand side, the alignment is switched off so that the profile is swept unchanged along the path curve.



## Coil Segments

A coil segment is a stranded conductor with a homogeneous current distribution in its cross section. This source type feeds the calculation domain with a given current.

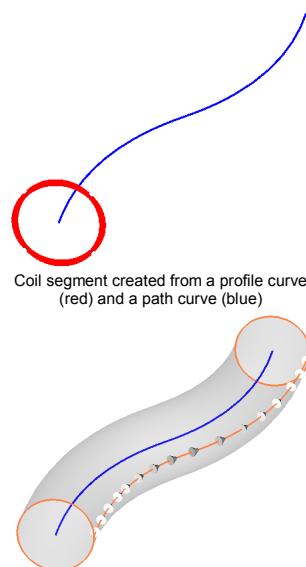
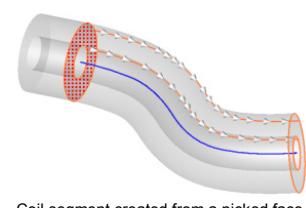
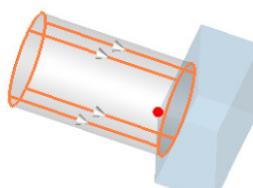
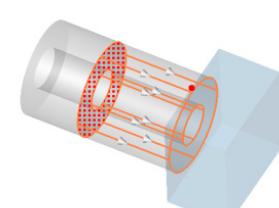
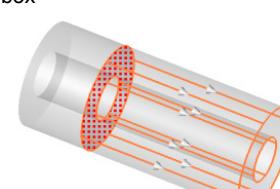


**Please note** that coil segment sources are available for the tetrahedral-based solvers only – if a hexahedral solver is used, a thin path is created at the position of the path curve instead.

There are various ways to define a coil segment. To define the profile of the source one can either pick a planar face before activating this mode or select a planar profile curve in the main plot window. If the tool is activated with a picked planar face, the interactive mode will start with the definition of the path or extrusion. The second step of the construction is to select a path curve. Alternatively, a numerical value could be used for the extrusion of the profile. To skip this step one can press ESC. If the profile is to be extruded to a picked point, it is necessary to pick this point before activating the construction mode.

After the path selection was completed (either by selection or pressing ESC) a dialog box opens where all other settings can be defined.

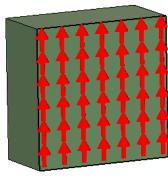
In total there are six different ways to define a coil segment which are summarized in the table below.

	Profile: <b>selected curve</b>	Profile: <b>picked face</b> (needs to be picked beforehand)
Path: <b>selected curve</b>	<p>1. <b>Activate the creation tool</b> 2. Select the closed profile curve 3. Select the open path curve</p>  <p>Coil segment created from a profile curve (red) and a path curve (blue)</p>	<p>1. Pick a planar profile face 2. <b>Activate the creation tool</b> 3. Select the closed profile curve</p>  <p>Coil segment created from a picked face (red dots) and a path curve (blue)</p>
Path: <b>extruded to picked point</b> (needs to be picked beforehand)	<p>1. Pick a point 2. <b>Activate the creation tool</b> 3. Select the closed profile curve 4. Press <i>ESC</i> to open the dialog box</p>  <p>Coil segment created from a profile curve and a picked point</p>	<p>1. or 2. Pick a planar profile face 1. or 2. Pick a point 3. <b>Activate the creation tool</b> 4. Press <i>ESC</i> to open the dialog box</p>  <p>Coil segment created from a picked face (red dots) and a picked point (red)</p>
Path: <b>extrude with given numerical height value</b>	<p>1. <b>Activate the creation tool</b> 2. Select the closed profile curve 3. Press <i>ESC</i> to open the dialog box</p>  <p>Coil segment created from a selected profile curve and a numerical value for the extrusion height</p>	<p>1. Pick a planar profile face 2. <b>Activate the creation tool</b> 3. Press <i>ESC</i> to open the dialog box</p>  <p>Coil segment created from a picked face (red dots) and a numerical value for the extrusion height</p>

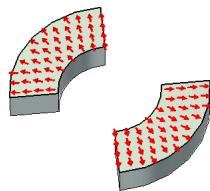
## Permanent Magnets

To define a permanent magnet, you must activate the permanent magnet tool by selecting *Simulation: Sources and Loads*  $\Rightarrow$  *Permanent Magnet* . You will be prompted to select a face of a solid in order to select the magnet's geometry. Pick any solid with "Normal" material properties.

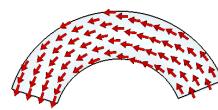
You can define constant, radial or azimuthal magnetizations. For details refer to the online help.



Constant magnetization



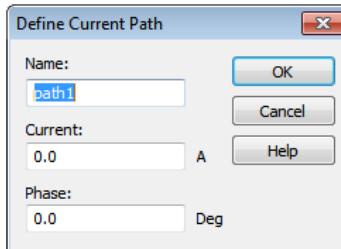
Radial magnetization



Azimuthal magnetization

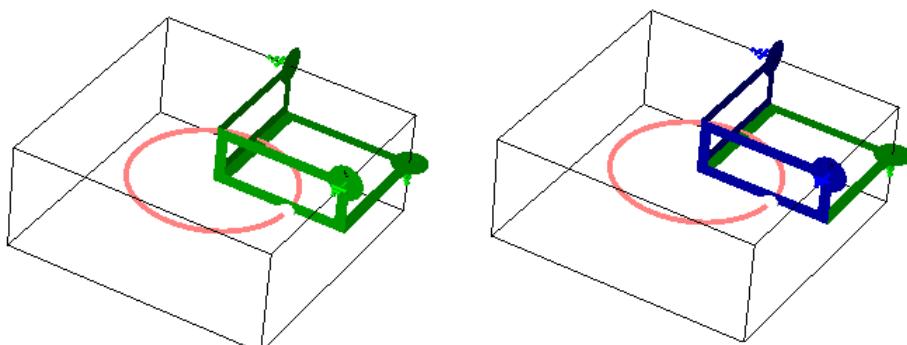
## Current Paths

The definition of a current path is very similar to a coil definition. A single curve must be defined before the current path tool can be activated by selecting *Simulation: Sources and Loads*  $\Rightarrow$  *Current Path* . You will be prompted to select a curve. Then a dialog box appears in order to define the total current through the loop.



The phase value is only relevant for the LF Frequency Domain solver.

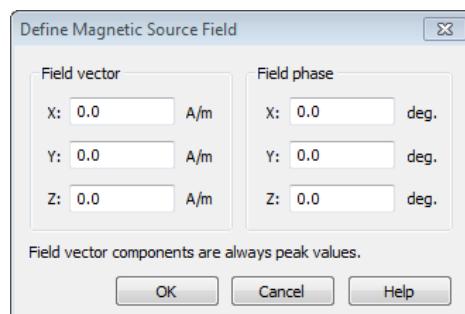
It is important that the current path is closed or that it terminates on a union of perfect electric conductors (PEC) and electric boundary conditions or conductive domains (generating a stationary current field) such that this union forms a closed loop with the current path. Otherwise the problem is not solvable by the magnetostatic solver since such a source violates Ampere's Law.



Left: A circular current path leaves the calculation domain through two electric boundaries – a solvable situation. Due to symmetries, only 1/4 of the structure has to be calculated.  
 Right: A circular current path leaves the calculation domain through two magnetic boundaries – not a solvable situation in magnetostatics.

## Homogeneous Magnetic Field

To simulate structures in a homogeneous magnetic field, it is possible to define such a source by selecting *Simulation: Sources and Loads  $\Rightarrow$  Magnetic Source Field*. The following dialog box allows you to define the magnetic field vector:



Boundaries along the direction of the source field (i.e. boundary faces for which the source field has non-zero flux) have to be set to type “magnetic”. Moreover, to set a valid problem using the tetrahedral solver, one of the remaining faces may also be set to type “magnetic”.

The *Field phase* value is relevant only for LF Frequency Domain simulations.

## Electrostatic Solver

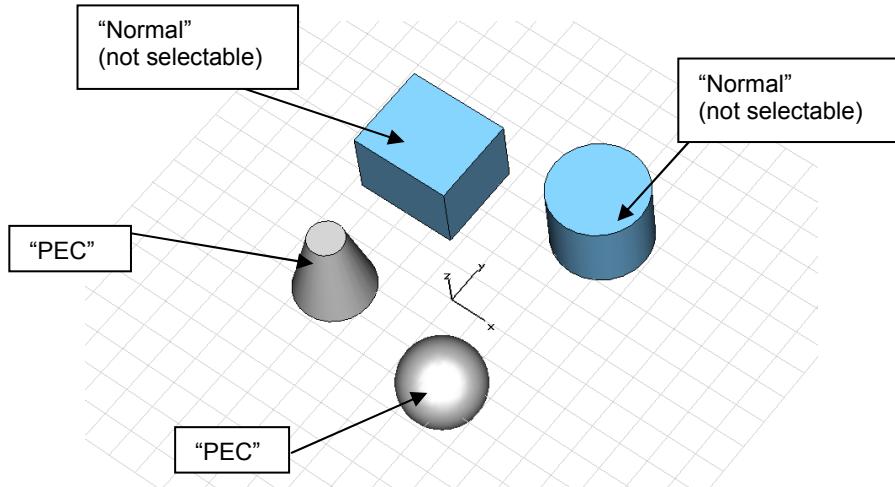
The electrostatic solver can be used for the simulation of static electric problems. Available sources comprise fixed and floating potentials, boundary potentials, charges on PEC solids and homogeneous volume and surface charges. The main task for the solver is to calculate the potential, the electric field strength and the electric flux density. These results appear automatically in the navigation tree after the solver run.

### Open Boundaries

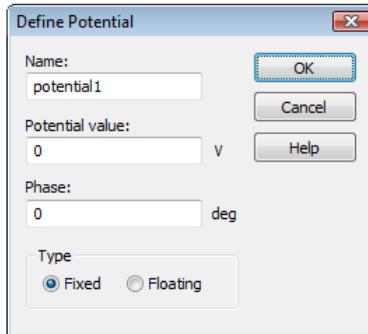
The electrostatic solver features open boundary conditions. These help to reduce the number of mesh nodes when problems in free space are simulated.

### Potential Sources

The most important electrostatic source type is a potential definition. To define a potential on a perfect electric conductor (the solid has to be assigned to PEC material) you must activate the potential tool first via *Simulation: Sources and Loads*  $\Rightarrow$  *Electric Potential* . The first step is to select the surface of a perfect electric conductor carrying the new potential:



After a PEC surface has been selected, the potential dialog appears to assign a *Name*, a *Potential value* and a *Type* for the new source:

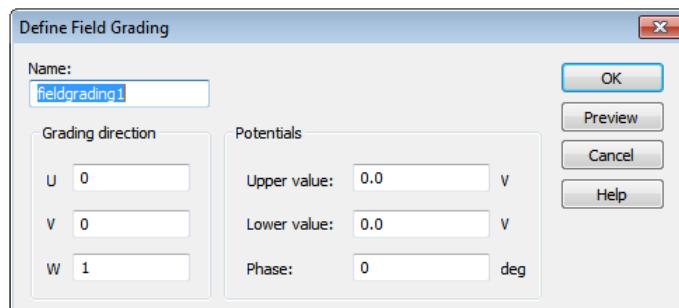


The *Phase* value is relevant only for LF Frequency Domain simulations and thus will be ignored by the described solver.

Note that for a potential of *Type "Floating"*, the value itself is not prescribed, but the resulting constant potential at the solid will obtain a value such that the resulting total charge of the conductor is zero. Consequently, defining a floating potential is equivalent to assigning a zero charge. The charge definition will be discussed later.

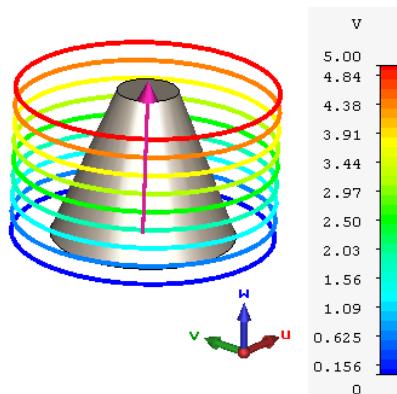
## Field Grading

Capacitive field grading is an electrostatic source characterized by a linear distribution of potential on the PEC solid surface. This source can be created by selecting *Simulation: Sources and Loads*  $\Rightarrow$  *Electric Potential*  $\Rightarrow$  *Field Grading* . Afterwards a surface of a PEC solid can be picked, on which the field grading source is to be created. Then the field grading definition dialog box appears, where all the settings for the source can be defined.



The *Grading direction* is the vector along which the potential value must change linearly. In any plane perpendicular to this vector the potential value on the surface of the PEC object is constant. Upper and lower potential values define the range within which the electrical potential is changing on the surface of the PEC solid.

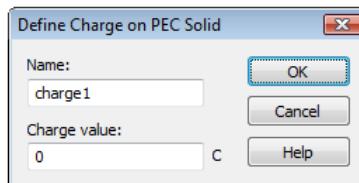
After the necessary values are set, press the *OK* button. A new field grading source is created.



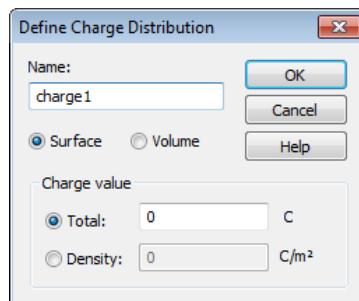
## Charge Sources

Two different charge types exist in CST EM STUDIO: total charges on perfect conductors (resulting generally in a non-uniform surface-charge distribution along the PEC surfaces) and uniform charge distributions on normal material solids.

For the charge definition based on PEC, the first step is very similar to the one carried out with the potential definition. After activating the charge tool via *Simulation: Sources and Loads*  $\Rightarrow$  *Electric Charge on PEC*

, you can pick a surface to which the charge will be applied. Then the charge dialog appears to determine the name and the charge value:
 


For the definition of a uniform charge-distribution definition, the first step is similar again - the only difference is that the source must be assigned to a normal material solid. You cannot define a uniform charge distribution on a PEC material. Use *Simulation: Sources and Loads*  $\Rightarrow$  *Electric Charge Distribution*

 and select a normal material solid. Then the following dialog will appear:
 


Here you can specify a name, a type and a value for the charge distribution. You can define a volume as well as a surface charge distribution. Remember that the latter will generate a jump in the normal component of the electric flux density. Furthermore, you can define the total charge or the charge density value.

## Boundary Potentials

Finally, you can also assign an electrostatic potential to an electric boundary condition from within the boundary dialog. Open the boundary dialog box via *Simulation: Settings*  $\Rightarrow$  *Boundaries*  and select the *Boundary Potentials* tab: In order to specify a boundary potential, select the "Floating" type from the drop down list or select the "Fixed" type and enter a value in the edit field.

A boundary potential can be defined on normal or electric boundary conditions only. Boundaries with different potential values must not be adjacent. Again, you can define a fixed or floating potential.

## Stationary Current Solver

The stationary current solver can be used to simulate DC current distributions. Available sources are potentials, boundary potentials, current paths, current ports and coil segments. Additionally to the modeled structure with defined material properties, lumped network elements, i.e. resistors, may be added into the computational domain. The main task for the solver is to calculate the electric field strength, current density and Ohmic losses. These results appear automatically in the navigation tree after the solver run.

Since the process of defining potential and current path sources is discussed in the two previous sections, we will focus on the current port definition. For a more detailed description of the lumped network element, we refer to the subsection *Lumped Network Elements* in the section *LF Frequency Domain Solver*.

## Parameterized Electrical Conductivity

The stationary current solver supports not only fixed electrical conductivity values (isotropic or anisotropic), but also temperature-dependent and nonlinear characteristics.

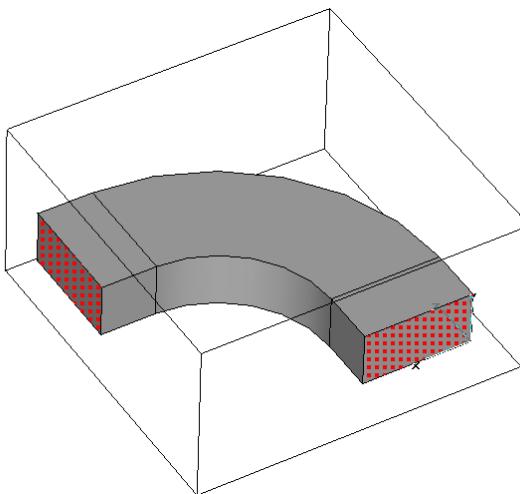
- Temperature-dependent electrical conductivity can be defined by setting the material *Type* in the *General* tab of the *Material Parameters* dialog to *Temp. dependent* and defining the temperature dependency slope of electrical conductivity in the *Temperature-Dependent Materials* dialog, which can be accessed by pressing the *Properties* button in this tab. A temperature field must be imported from a thermal project via *Simulation: Sources and Loads*  $\Rightarrow$  *Field Import* .
- Nonlinear electrical conductivity is defined by creating a E(J) curve in the *Electrical Conductivity Properties* dialog. This dialog is accessible via the *Conductivity* tab of the *Material Properties* dialog. Here, in the group for *Electrical conductivity* check *Advanced* and press the button *Parameters*.

Before setting either parameterization, a default non-zero value of electrical conductivity must be set.

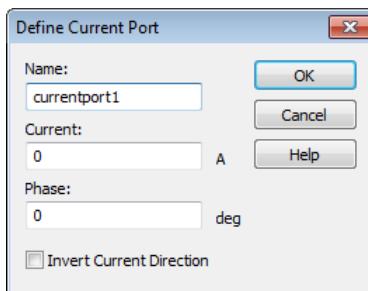
## Current Ports

A current port is a face with a constant potential on a conductive material surface. When using hexahedral meshes, this surface must be located on the computational domain's boundary. The potential value or the total current through the current port can be prescribed. Through the current port, currents can leave or enter the calculation domain. Note that if no potential is prescribed, the sum of the prescribed currents entering and leaving the computational domain must be zero. Otherwise you will define a non-solvable problem.

The following picture shows a simple conductive bend inside the computational domain. The two conducting faces are highlighted.



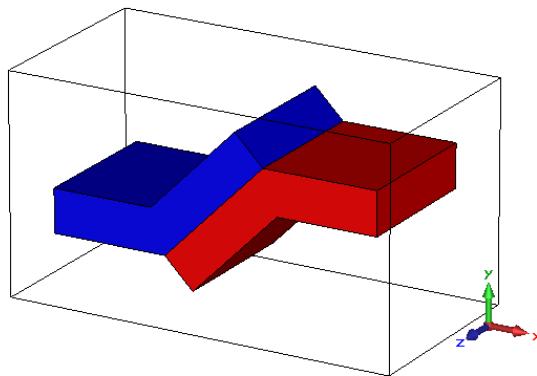
In order to define a current port on one of these faces, select the current port tool via *Simulation: Sources and Loads*  $\Rightarrow$  *Current Port* . Next pick an appropriate face on a conductive material. A dialog box opens where you can define the port's name and the magnitude of the current:



## Electrical Contact Impedance

A contact impedance can be defined via *Simulation: Sources and Loads*  $\Rightarrow$  *Contact Impedance* . It is equivalent to a thin layer of conductive material at the interface

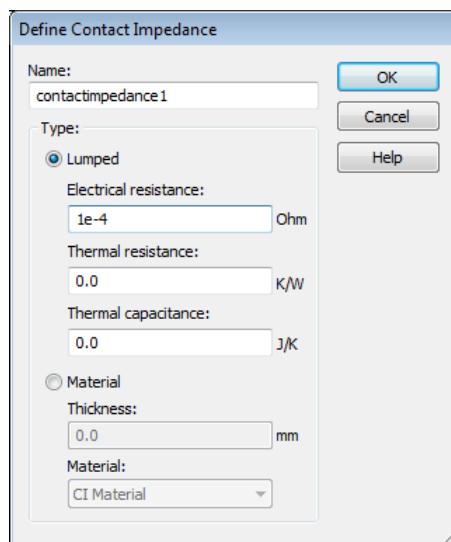
between two (or several) solids. The definition is performed by selecting the solids associated with the “first” and then with the “second” side of the contact surface.



A contact impedance can be characterized either by a lumped parameter (integral electrical resistance in Ohm) or by its thickness and conductivity of the material assigned. Both definitions are equivalent and can be easily converted into each other:

$$R = \frac{1}{\sigma} \frac{l}{A}$$

Here  $R$  is the lumped parameter representing integral resistance. In the material-based representation, electrical conductivity  $\sigma$  and layer thickness  $l$  are used. Contact area  $A$  is calculated by the solver.



The advantage of contact impedance definition through integral resistance is that it is independent on the contact area  $A$  which may vary in case of intersections of associated solids or depending on the mesher settings. On the other hand, the material-based definition offers much more flexibility, for example, it supports nonlinear or temperature-dependent electrical conductivity.

Electrical losses taken place within the contact region are calculated and saved by the stationary current solver as surface losses, so they can be utilized afterwards for a thermal analysis.

Contact impedances are only supported by the tetrahedral-based stationary current solver.

## LF Frequency Domain Solver

The LF Frequency Domain solver can be used to solve electromagnetic field problems with time-harmonic sources and linear materials. In this case, all quantities are time-harmonic and it is possible to solve a complex valued problem in the frequency domain.

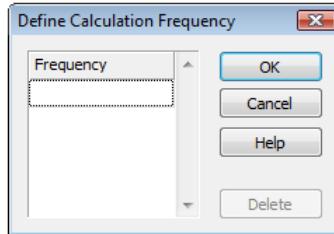
The main task for the solver is to calculate electromagnetic fields and the resulting currents, losses and energies. These results appear automatically in the navigation tree after the solver run has been finished.

The LF Frequency Domain solver includes the following simulators:

- Full Wave simulator
- Magnetoquasistatic simulator
- Electroquasistatic simulator

The full wave simulator solves the full Maxwell's equations. The magnetoquasistatic and electroquasistatic simulators can solve low frequency problems with dominating magnetic (e.g. eddy current problems) or electric energy, respectively. A typical application is the computation of AC current and loss distributions.

In contrast to the static solvers, one or more calculation frequencies must be defined before the LF Frequency Domain solver can start. Open the frequency dialog box *Simulation: Settings  $\Rightarrow$  Frequency*  for this task.



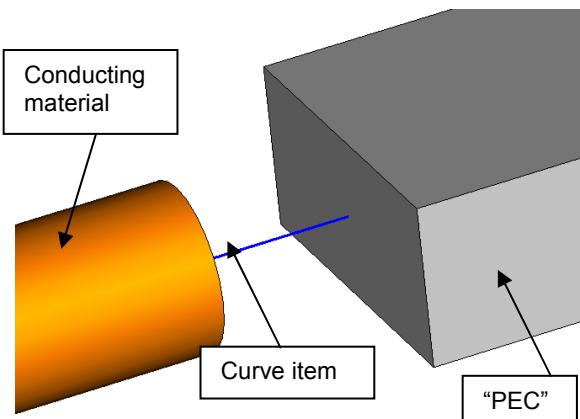
To add a new frequency to the list, select the empty edit field, enter the value and confirm with the *Enter* key. The list becomes operative when you leave the dialog box by clicking *OK*.

## Full Wave and Magnetoquasistatic Simulator

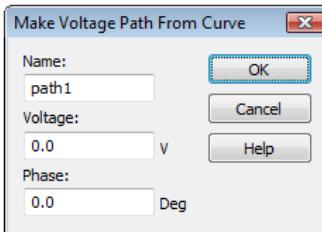
Available sources are current paths, voltage paths and coils. Coil and current path definitions are discussed in the Magnetostatic solver section. One minor difference exists: in addition to the current (or voltage) value, it is possible to assign a phase value to a current path or a coil (for magnetostatic calculations this setting is ignored).

## Voltage Paths

The third source type, the voltage path, is similar to the current path. It is created from a curve path. A typical application is a voltage path connecting two conducting regions, defining a voltage between the conductors.



To define a voltage source, activate the appropriate tool via *Simulation: Sources and Loads*  $\Rightarrow$  *Path Sources*  $\Rightarrow$  *Voltage Path from Curve* . The curve selection modus enables the selection of the curve that is to be transformed into a voltage path. After the appropriate curve has been selected, the voltage path dialog box appears. Here you can determine the element's name, its voltage and phase values.

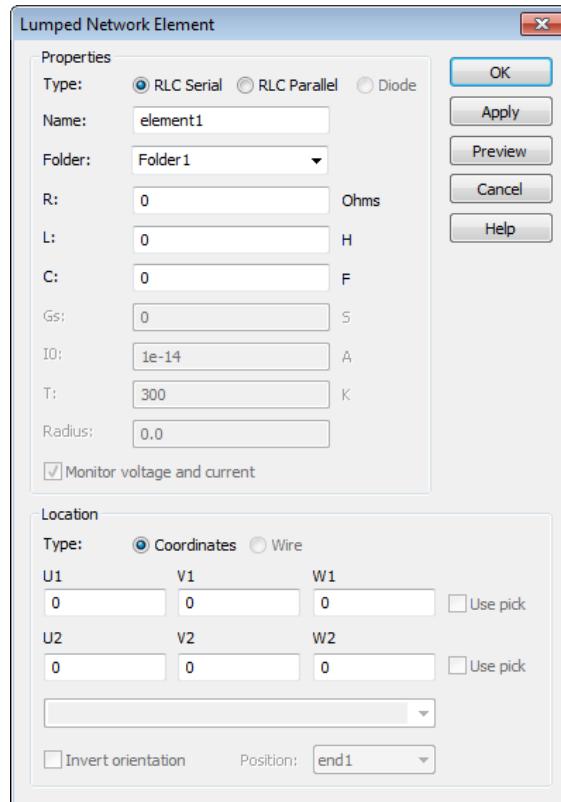


After the definition is complete, the voltage source is listed in the navigation tree folder *Voltage Paths*.

## Lumped Network Elements

The Full wave and the Magnetoquasistatic formulations of the LF Frequency Domain solver account for the inclusion of the lumped network elements in the simulation domain. In this context, one can make use of any parallel or serial circuits consisting of one resistor, one capacitor and one inductor. To add a new network, open the lumped element dialog box, *Simulation: Sources and Loads*  $\Rightarrow$  *Lumped Element* .

In the lumped network element dialog box, the element values as well as the connection type for a lumped element – serial or parallel- are defined. Furthermore, the geometrical location of the lumped element is set in the dialog box, i.e. the starting point and ending point of the network in the computational domain.



## Electroquasistatic Simulator

In the electroquasistatic approximation of the full Maxwell's equations, the time derivative of the magnetic field is ignored in the Faraday-law. Hence, the computed electric field is curl-free in the whole space. As a consequence, electroquasistatic problems can be described by a complex scalar potential which reduces the number of unknowns in the equation system to be solved.

Thus, running the electroquasistatic simulator is usually much faster and more robust than running the full wave simulator on the same mesh. Whenever the time derivative of the magnetic field is negligible in the Faraday-law you should use the electroquasistatic solver to solve a low frequency problem. Typical applications are insulator problems, where the conductivities and magnetic field energies are very low.

Potentials are available as excitation sources. These are already discussed in the electrostatic solver section. Again, a minor difference exists: In addition to the potential value, it is possible to assign a phase value (for electrostatic calculations this setting is ignored). Please refer to the online-help for further details.

## LF Time Domain Solver

The LF Frequency Domain solver can be used to solve electromagnetic field problems with the time-dependent sources driven at low frequencies. This solver includes the following simulators:

- Magnetoquasistatic simulator
- Electroquasistatic simulator

The solver is based on an adaptive *implicit* time-stepping algorithm of high accuracy which needs to solve four linear or nonlinear systems of equations in each time step.

### Magnetoquasistatic Simulator

In the magnetoquasistatic approximation of the Maxwell's equations, the time derivative of the displacement current can be omitted with respect to the conduction currents. Typical use cases are the nonlinear eddy current problems or transient simulations (e.g. switching devices, actuators, sensors).

Within the simulator, supported excitation sources are permanent magnets, current- and voltage-driven coils and wires, coil segments, current ports and transient external magnetic source fields.

The main task for the simulator is to calculate the time evolution of the magnetic and current fields as well as the resulting losses, energies and other derived quantities like forces.

### Electroquasistatic Simulator

The electroquasistatic approximation of the Maxwell's equations is employed when the influence of the magnetic induction can be neglected. Thus, a description of an electroquasistatic field is completed by a scalar potential function which reduces the number of unknowns in the equation system to be solved. Typical use case includes, e.g., a high-voltage bushing.

Electrical potentials are available as excitation sources. These are already discussed in the electrostatic solver section. Please note here again: the *Phase* value, which can be defined together with the potential value, is relevant only for LF Frequency Domain simulations and thus will be ignored by the described simulator.

## Workflow

The workflow for a time domain simulation is very similar to the workflow of static and time harmonic simulations. However, some additional steps must be performed before the solver is started:

1. One or more excitation signals must be defined
2. Excitation signals must be assigned to sources
3. Monitors must be defined
4. A simulation duration must be set

These differences result from the fact that additional information is necessary about the time evolution of the excitations and the size of the time interval of interest. Furthermore, storing the whole time history of all results which are computationally available usually needs a lot of disk space. For this reason, the concept of time monitors is introduced, which allows a more specific definition of the results of interest.

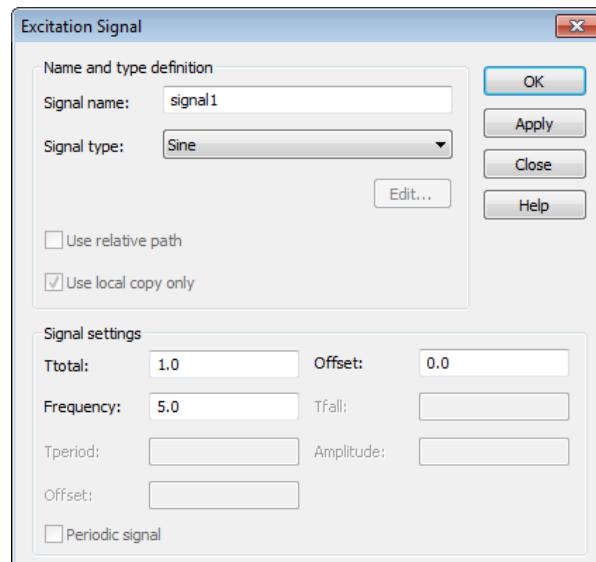
**Note:** The excitation definition as well as the usage of monitors in CST EM STUDIO is very similar to those available in CST MICROWAVE STUDIO.

The following subsections will describe these additional steps in short. For more detailed information, please refer to the online help.

## Signal Definition

In a new project, only a constant "default" signal is defined. For a meaningful simulation with the LF Time Domain Solver, at least one non-constant signal should be defined.

A new signal can be defined via *Simulation: Sources and Loads*  $\Rightarrow$  *Signals*  $\Rightarrow$  *New Excitation Signal* . A dialog box opens where a signal type, its parameters and a name can be set.

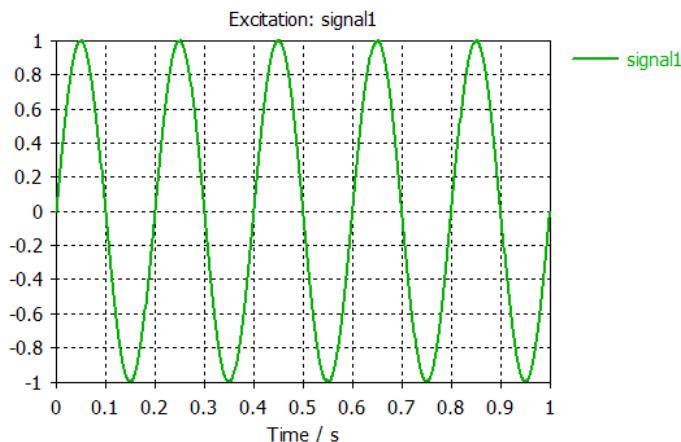


The parameters of the signal depend on the individual signal type and are described in the online help. The parameter *Ttotal* must be set for almost all signal types and defines the size of the definition interval. For time values larger than *Ttotal*, the signal is, in general, continued by a constant value. It is also possible to import a signal or to create a user defined signal or to select a pre-defined signal from the signal database.

All defined signals are visible in the *Signal* folder in the *Navigation Tree*.



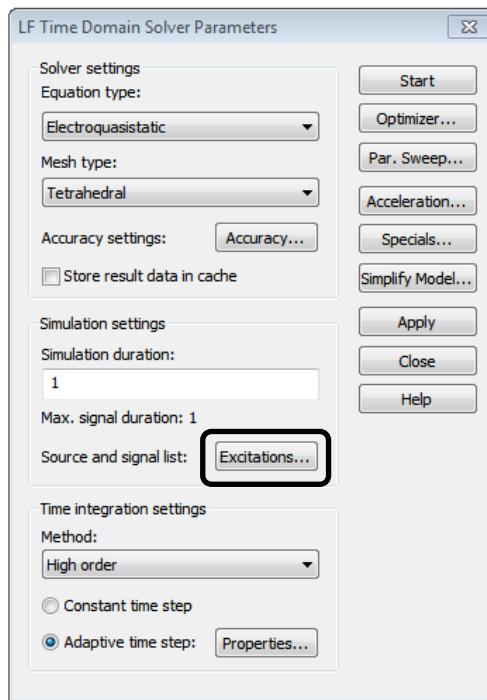
A signal can be displayed by selecting it in the *Navigation Tree*:



## Excitations: Assigning Signals to Sources

As for the static solvers, the source value defines the strength of a source field. The time evolution of a source is defined by assigning a signal to it.

This can be done by opening the solver dialog box via *Home: Simulation*  $\Rightarrow$  *Setup Solver* and pressing the *Excitations...* button.



A sub-dialog opens showing each defined source that can be interpreted by the solver. Also the source values are displayed. Each source can be switched on or off for the simulation. By default, all sources are switched on.

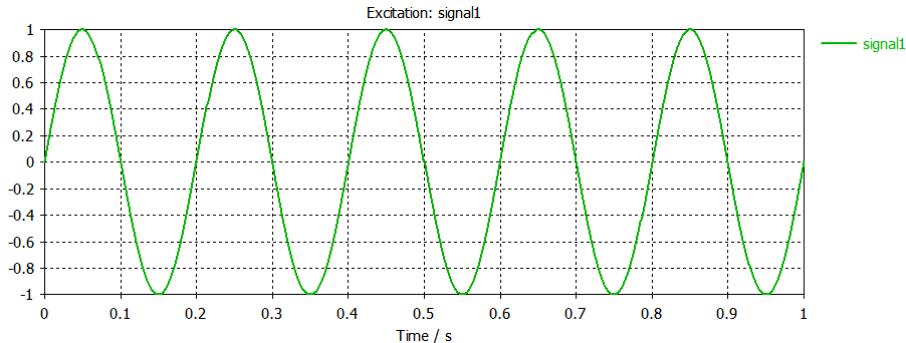
For each source, a signal can be assigned via a drop-down list. The same signal can be assigned to several sources. Optionally, an individual time delay  $\Delta t$  can be defined for each source.

The resulting time dependent excitation  $f$  is the product of the source value  $v$  (e.g. the coil current) and the (possibly shifted) assigned signal  $s$  :

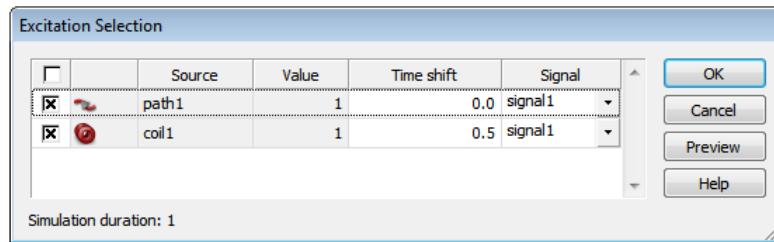
$$f(t) = s(t - \Delta t) \cdot v .$$

### Example

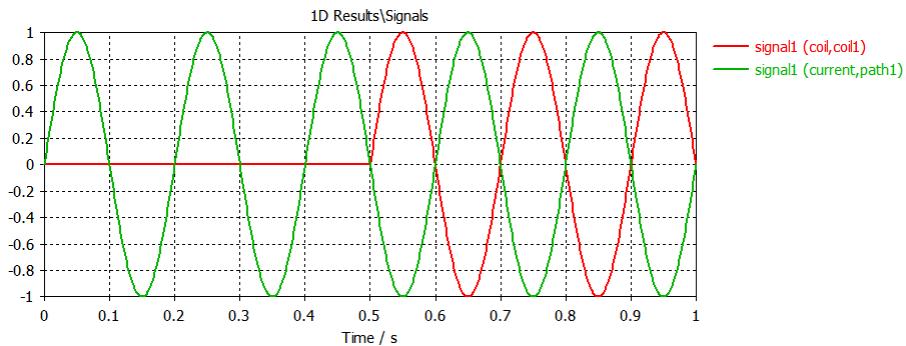
Two sources are defined, one current path with source current 1 A and one coil, also carrying 1 A in each turn. A previously defined signal "signal1" (see image below) is assigned to both sources.



The signal of the coil is shifted by 0.5 s. With these settings, the *Excitation Selection* dialog will look like this:



For this example, the resulting excitations used by the solver look like this:



### Reference Signal

There is always one signal tagged as the 'reference signal'. This signal is highlighted in the *Navigation Tree* by a yellow background. The reference signal can be changed by marking another signal in the tree and selecting *Simulation: Sources and Loads*  $\Rightarrow$  *Signal*  $\Rightarrow$  *Use as Reference*.

By default all sources are set to use the currently defined reference signal. Hence, it is not necessary to visit the *Excitations* sub-dialog of the solver dialog if only one source or only one signal shall be used for the simulation. Then, it is enough to select the desired signal being the reference signal and by default all sources are automatically assigned to this signal.

## Monitor Definition

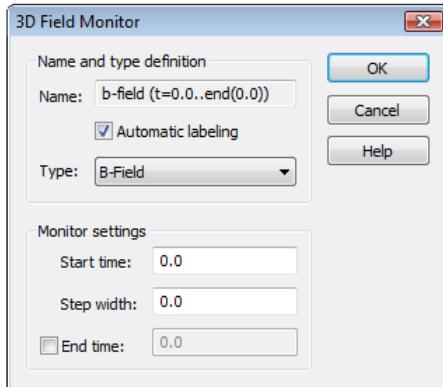
In contrast to the static and time-harmonic solvers, no results will appear automatically in the navigation tree. It is not possible to store the fields and secondary results at every computed time step as this would require a tremendous amount of disk and memory space. You should, therefore, define certain results and time intervals at which the solver will record the desired data. These definitions are called *Monitors*.

Several different kinds of monitors are available in CST EM STUDIO: *3D Field Monitors*, *Monitors at Points*, *Monitors on Edges or Curves*, *Monitors on Faces* and *Monitors on Solids or Volumes*. The 3D Field Monitors yield field plots which can be animated over the simulated time. The other monitors are classified by the objects on which appropriate integral functionals are defined. They yield 1D curves of scalar values versus the simulated time.

All defined monitors are listed in appropriate subfolders of the *Monitors* folder in the *Navigation Tree*. Within this folder, you may select a particular monitor to reveal its parameters in the main view.

### 3D Field Monitors

Several kinds of monitors record 3D vector or scalar fields (e.g. B-field, H-field, E-field, J-field, current density). A 3D Field Monitor can be defined via *Simulation: Monitors*  $\Rightarrow$  *Field Monitor* . A dialog box opens where the type of the field, the start time and the sample step width can be defined:



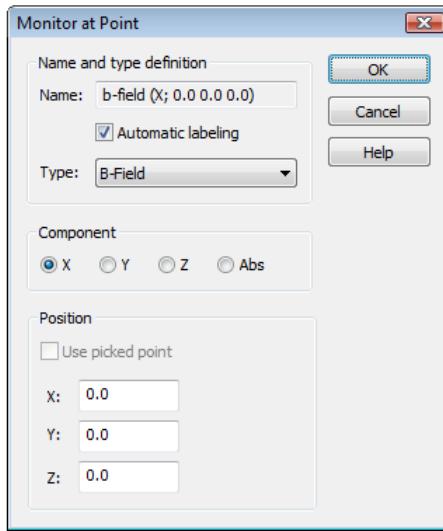
Available field types for the magnetoquasistatic simulator are B-Field, H-Field, E-Field, Cond. Current Densities, and Material (the latter showing the relative permeability). Within the electroquasistatic simulator, the time evolution of the E-field, D-field, Cond. and Displ. Current Densities as well as Potential can be monitored.

After the solver run, the recorded result can be accessed via the *2D/3D Results* folder in the Navigation Tree. The scalar or vector field can be animated over the defined time period.

### Monitors at Points

These kinds of monitors record scalar values that are defined at a point (previously picked or entered numerically), e.g. the x-component of the magnetic flux density at a

fixed position. You can create such a monitor via *Simulation: Monitors  $\Rightarrow$  Monitor on Entity  $\Rightarrow$  Monitor at Point* 



The magnetoquasistatic simulator supports following monitor types: B-Field, H-Field, E-Field, Cond. Current Density and Material.

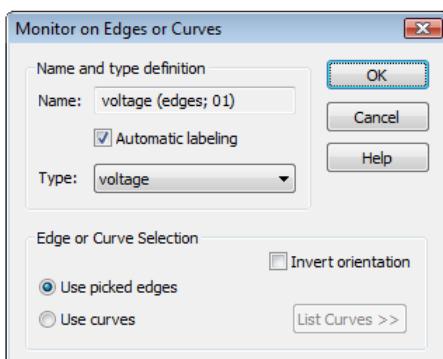
For the electroquasistatic, available monitor types are E-field, D-field, Cond. and Displ. Current Densities and Potential.

The monitor generates a 1D-plot over time during the solver run. The result plot can be accessed in the *Navigation Tree* in the *1D Results* folder.

Please note that this kind of monitor is similar, although not identical, to *Probes* available within CST MICROWAVE STUDIO.

## Monitors on Edges or Curves

These kinds of monitors record scalar values that are defined for (previously picked) model edges or on curve items, currently the voltage and the source current along a path. You can create it via *Simulation: Monitors  $\Rightarrow$  Monitor on Entity  $\Rightarrow$  Monitor on Edge* 

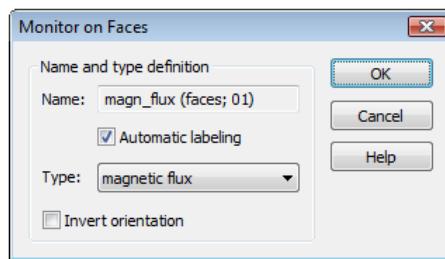


Again, the monitor generates a 1D-plot over time during the solver run and the result plot can be accessed in the *Navigation Tree* in the *1D Results* folder.

For the electroquasistatic simulator, this type of monitors is not supported.

### Monitors on Faces

These kinds of monitors record scalar values (presently only the magnetic flux through a surface within the magnetoquasistatic simulator) that are defined for a (connected set of) model faces which has to be picked (*Simulation: Picks*  $\Rightarrow$  *Picks* Simulation: Monitors  $\Rightarrow$  *Monitor on Entity*  $\Rightarrow$  *Monitor on Face* .

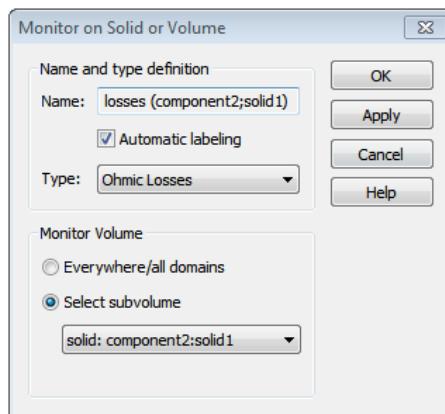


Again, the monitor generates a 1D-plot over time during the solver run and the result plot can be accessed in the *Navigation Tree* in the *1D Results* folder.

For the electroquasistatic simulator, this type of monitors is not supported.

### Monitors on Solids or Volumes

Within the magnetoquasistatic simulator, these kinds of monitors record values that are defined for a solid or volume (the force on a solid etc.). You can create it via *Simulation: Monitors*  $\Rightarrow$  *Monitor on Entity*  $\Rightarrow$  *Monitors on Solid or Volume* .



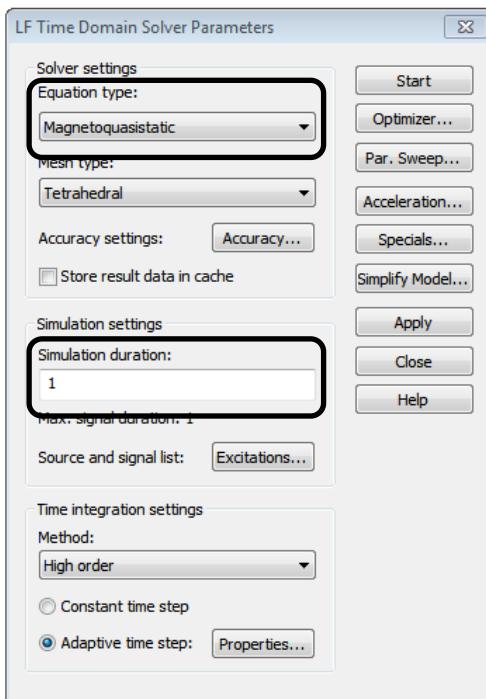
Available monitor types for the magnetoquasistatic simulator are: Ohmic Losses, Force and Torque, Iron Losses as well as App. and Inc. Inductance Matrices.

Again, the monitor generates a 1D-plot over the time during the solver run (or in case of Force monitors one 1D-plot per component) and the result plot can be accessed in the *1D Results* folder.

For the electroquasistatic simulator, this type of monitors is not supported.

## Starting the Simulation

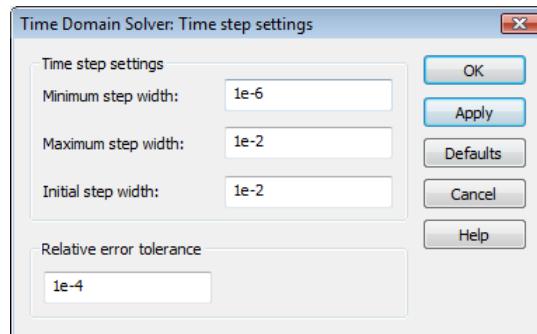
As already mentioned, the solver dialog box can be opened via *Home: Simulation  $\Rightarrow$  Setup Solver*. First of all, define the *Equation type* you are going to employ. Secondly, before starting the simulation, the *Simulation duration* must be entered. This value defines the length of the simulated time interval in the currently active time unit. Note that every simulation starts at time zero.



If at least one non-constant signal is in use, the maximum over all assigned time signal is displayed below the duration entry field (taking possible time shifts into account). This information gives some hint for a reasonable simulation duration and can be used for cross-checking, e.g. to ensure that signals and simulation duration are defined for a similar time period and scale.

Two different time-stepping strategies are available for the solver: Constant and adaptive time-stepping. By default the adaptive time-stepping is enabled. The constant time-stepping may be used for validation purposes or if the adaptive control of the time step should not work as expected. Usually, the adaptive scheme should be preferred since it is more efficient.

It is a good idea to have a look at the parameters of the adaptive time-stepping scheme before the simulation is started. The parameters can be displayed and modified in the *Time step settings* sub-dialog which can be activated by pressing the *Properties* button.



The most important value is the *Relative error tolerance*. The smaller this value the more rigorous is the behaviour of the adaptive scheme, leading to smaller time steps and smaller time-discretization errors. On the other hand, small values will increase the simulation time. Furthermore, you can define upper and lower bounds for the size of a time step and set the size of the initial time step. If you have some knowledge about typical time scales of your model, it might be meaningful to modify the default settings.

Note that for some problems it may be also necessary to increase the accuracy for the solution of the linear (or respectively nonlinear) systems of equations that are solved for each time step. This can be done by pressing the *Accuracy...* button which opens a sub-dialog. However, in most cases, the default-settings can be left unchanged.

Finally, the LF Time Domain solver can be started by pressing the *Solve* button and the results can be analyzed.

## Coupled Simulations with CST MPHYSICS STUDIO

Ohmic losses from the solvers of CST EM STUDIO can be used for thermal simulations in CST MPHYSICS STUDIO. Based on these results, temperature dependent material properties can be updated in the stationary current solver or the low frequency solver. The mechanical solver allows performing a continuative stress simulation on the temperature distribution. Moreover, force density distributions from magnetostatic or electrostatic simulations can be fed into the mechanical solver of CST MPHYSICS STUDIO.

Please refer to the *CST MPHYSICS STUDIO Workflow* document for more detailed information about these multi-physics workflows.

## Equivalent Circuit EMS/DS Co-Simulation

Equivalent circuit parameters describing the physical behavior of the field part of CST EM STUDIO model can be used for the co-simulations within CST DESIGN STUDIO. The extraction of the lumped parameters from the field model is supported by the following tetrahedral mesh based solvers:

- Electrostatic Solver
- Magnetostatic Solver (linear and nonlinear problems)
- Stationary Current Solver

Please note: For nonlinear problems, the equivalent circuit parameters are calculated in the working point determined by the excitation sources defined within CST EM STUDIO. To cover the full parameter space of a nonlinear model, a parameter sweep can be used to retrieve the required data in a convenient way.

For further information, please refer to the examples within the *Equivalent Circuit EMS/DS Co-Simulation* section contained in the *CST EMS Examples* of the online help system.

# Chapter 4 — Finding Further Information

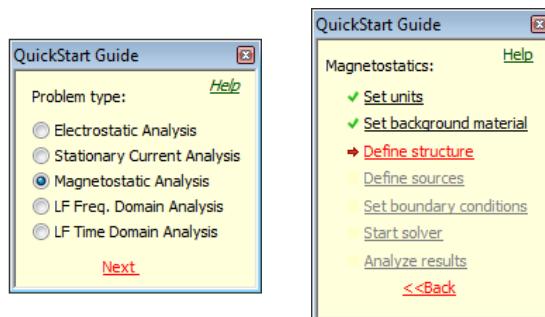
After having read this manual carefully, you should already have some idea of how to use CST EM STUDIO efficiently for your own problems. However, when you are creating your own first models, a lot of questions will arise. In this chapter, we give you a short overview of the available documentation.

## The QuickStart Guide

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

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

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



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

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

## Online Documentation

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

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

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

Please refer to the *CST STUDIO SUITE Getting Started* manual to find some more detailed explanations about the usage of the CST EM STUDIO Online Documentation.

## Tutorials

The online help tutorials will generally be your best source of information when trying to solve a particular problem. You can select an overview page of all available tutorials by following the *Tutorials Overview* link on the online help system's start page.

We recommend you browse through the list of all available tutorials and choose the one closest to your application. The fastest way to solve your particular problem is to study the most appropriate tutorial carefully, understanding the basic concepts before you start modeling your own problem.

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

## Examples

The installation directory of CST STUDIO SUITE contains an *Examples* subdirectory consisting of a couple of typical application examples. A quick overview of the existing examples can be obtained by following the *Examples Overview* link on the online help system's start page.

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

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

## Technical Support

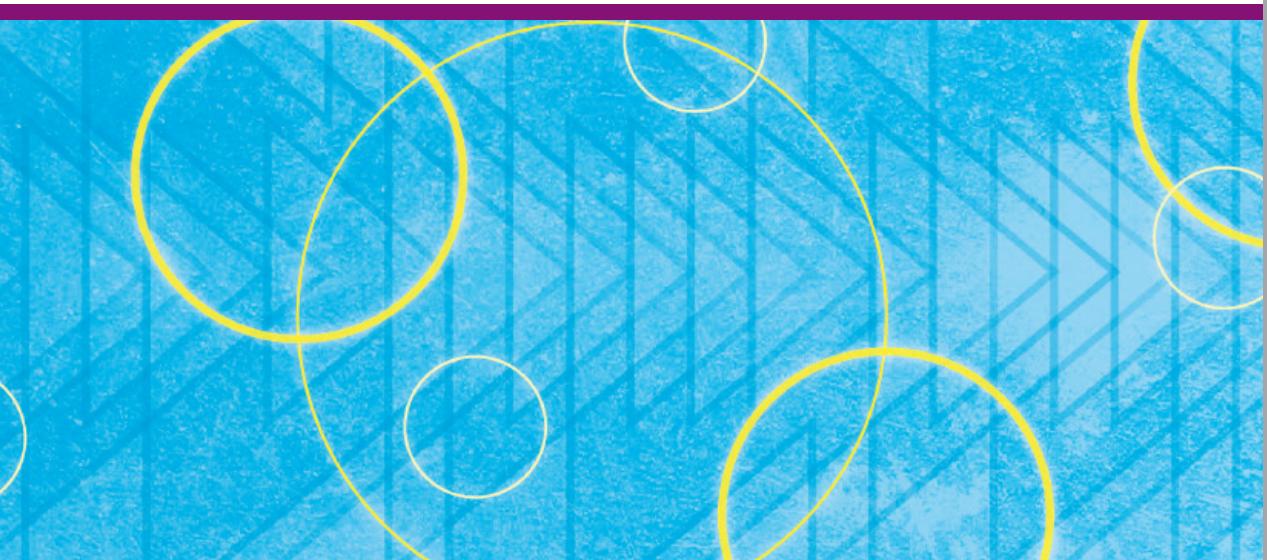
After you have taken your first steps to solving your own applications within CST EM STUDIO, please use the *File: Project*  $\Rightarrow$  *Archive As*  functionality to create an archive containing all relevant files. This archive should then be sent to the technical support team. Even if you have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

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

The support area on our homepage ([www.cst.com](http://www.cst.com)) also contains a lot of very useful and frequently updated information. Simple access to this area is provided by choosing *File: Help*  $\Rightarrow$  *Online Support Area*. You only need to enter your user name and password once. Afterwards, the support area will open automatically whenever you choose this menu command. Please note that the online help system's search function also allows searching in the Online Support content.

## History of Changes

An overview of all new main features of the release can be obtained by selecting the *Spotlight CST STUDIO SUITE 2015* page from the online help system (*File: Help*  $\Rightarrow$  *CST STUDIO SUITE – Help* ). Also the detailed *History of Changes* can be accessed through the *Spotlight* page in the Online Help. The *Changes in the Service Packs* Page at the same location provides additional smaller changes released during intermediate service packs. Since there are many new features in each new version, you should browse through these lists even if you are already familiar with one of the previous releases.



© CST 2015 | CST – Computer Simulation Technology AG | [www.cst.com](http://www.cst.com)

