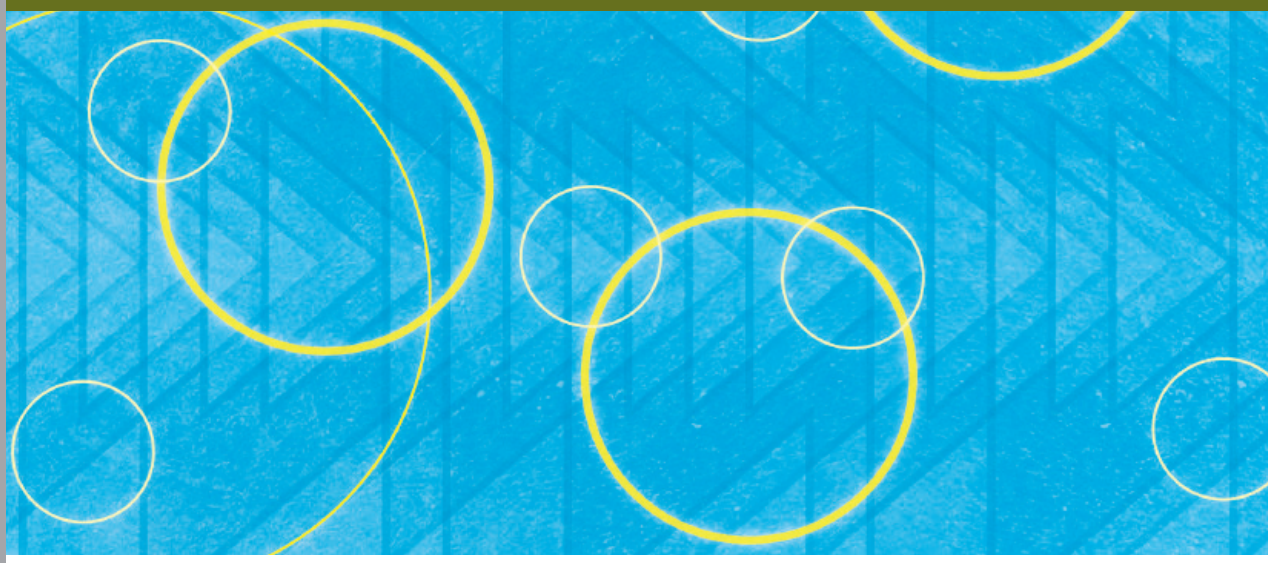
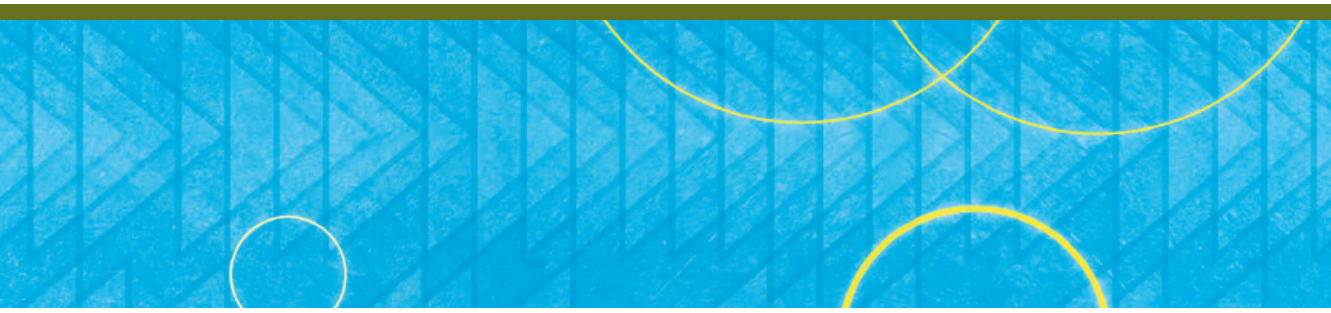
CST MPHYSICS STUDIO



Workflow & Solver Overview

CST STUDIO SUITE® 2015

Copyright



Trademarks

CST – Computer Simulation Technology AG
www.cst.com

Contents

Chapter 1 — Introduction

Welcome

Welcome to CST MPHYSICS® STUDIO, the powerful and easy-to-use tool for solving mechanical and thermal problems. This program combines a user-friendly interface with unsurpassed simulation performance.

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

How to Get Started Quickly

We recommend that you proceed as follows:

1. Read the *CST STUDIO SUITE Getting Started* manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. 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 idea 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 become familiar with the software quickly.
6. After you have worked through your first example, contact technical support for possible hints on how to use CST MPHYSICS STUDIO even more efficiently.

What is CST MPHYSICS STUDIO?

CST MPHYSICS STUDIO is a software package from the CST STUDIO family which allows thermal and mechanical simulations. It simplifies the process of defining the structure by providing a powerful solid modeling front end which is based on the ACIS modeling kernel. Strong graphic feedback simplifies the definition of your device even further. After the component has been modeled, a fully automatic meshing procedure is applied before a simulation engine is started.

A key feature of CST MPHYSICS STUDIO is its tight integration with the other CST STUDIO products. This allows an easy to use workflow for coupled EM-multiphysics simulations

A further 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 MPHYSICS STUDIO is capable of analyzing and designing thermal and mechanical aspects of devices.

Who Uses CST MPHYSICS STUDIO?

Anyone who needs to investigate thermal and mechanical aspects of electromagnetic devices – of course it is also possible to use the product standalone, but the full set of capabilities deploys when coupling the thermal and mechanical simulators with other products from the CST STUDIO SUITE family such as CST MICROWAVE STUDIO®, CST EM STUDIO® or CST PARTICLE STUDIO®.

CST MPHYSICS STUDIO Key Features

The following list gives you an overview of the CST MPHYSICS STUDIO 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 thermal simulation parameters with circuit simulation.
- Various independent types of solver strategies (based on hexahedral as well as tetrahedral meshes) allow accurate simulations with a high level of performance for a wide range of multi-physical applications.
- For specific solvers highly advanced numerical techniques offer features like Perfect Boundary Approximation® (PBA) for hexahedral grids and curved and higher order elements for tetrahedral meshes.

Structure Modeling

- Advanced ACIS[1]-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®), ACIS SAB, Autodesk Inventor®, IGES, VDA-FS, STEP, Pro/ENGINEER®, CATIA 4®, CATIA 5®, Siemens NX, Parasolid, Solid Edge, SolidWorks, CoventorWare®, Mecadtron®, NASTRAN, STL or OBJ files
- Import of 2D CAD data by DXF, GDSII and Gerber RS274X, RS274D files
- Import of EDA data from design flows including Cadence Allegro® / APD® / SiP®, Mentor Graphics Expedition®, Mentor Graphics PADS®, Mentor Graphics HyperLynx®, Zuken CR-5000® / CR-8000® and ODB++® (e.g. Mentor Graphics Boardstation®, CADSTAR®, Visula®)
- Import of PCB designs originating from CST PCB STUDIO®
- Import of 2D and 3D sub models
- Import of Agilent ADS® layouts
- Import of Sonnet® 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™, Gerber, DRC or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem setup

Mechanics Solver

- Temperature dependent Young's modulus
- Displacement boundary condition
- Traction boundary condition

[1] Portions of this software are owned by Spatial Corp. © 1986 – 2014. All Rights Reserved.

- Thermal expansion
- Neo-Hookean material model for simulation of large deformations
- Various Stress plots: Von Mises, Hydrostatic and Tensor components
- Strain plot including the visualization of the volumetric strain
- Nonlinear solver computes the Green-Lagrange and Almansi strain as well as the 2nd Piola-Kirchhoff and Cauchy stress tensors
- Displacement plot including a deformed mesh plot
- Import of force densities from EM-solvers
- Export of deformed structure to CST MICROWAVE STUDIO

Thermal Stationary Solver

- Isotropic and anisotropic material properties
- Bioheat material properties
- Nonlinear material properties (Bioheat properties and thermal conductivity)
- Thermal contact resistance
- Convection for human voxel models
- Heat transfer by conduction in volumes
- Heat transfer by convection and radiation through surfaces
- Sources: fixed and floating temperatures, heat sources, eddy current and stationary current loss fields, volume/surface power loss distributions in dielectric or lossy metal materials imported from CST MICROWAVE STUDIO or CST EM STUDIO, crashed particle loss distribution from CST PARTICLE STUDIO
- Adiabatic / fixed or floating temperature / open boundary conditions
- 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

Thermal Transient Solver

- Isotropic and anisotropic material properties
- Bioheat material properties
- Nonlinear material properties (Bioheat properties, thermal conductivity and heat capacity)
- Thermal contact resistance
- Convection for human voxel models
- Heat transfer by conduction in volumes
- Heat transfer by convection and radiation through surfaces
- Sources: fixed and floating temperatures, heat sources, eddy current and stationary current loss fields, volume/surface power loss distributions in dielectric or lossy metal materials imported from CST MICROWAVE STUDIO or CST EM STUDIO, crashed particle loss distribution from CST PARTICLE STUDIO
- Adiabatic / fixed or floating temperature / open boundary conditions
- Network distributed computing remote calculations

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 simulation
- Import and visualization of external xy-data
- Copy / paste of xy-datasets
- Fast access to parametric data via interactive tuning sliders

- Various field visualization options in 2D and 3D for thermal fields, heat flow densities, displacement fields, stress fields, etc.
- Animation of field distributions

- 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. 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 MPHYSICS 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 MPHYSICS 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 ⇨ Heat Flow Density [THs] ⇨ X*

Your Feedback

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

Chapter 2 — Simulation Workflow

The following example demonstrates how to use the basic features of the structural mechanics solver. Studying this example carefully will help you to become familiar with many standard operations that are important when performing a simulation with CST MPHYSICS STUDIO.

In the subsequent chapters you will find some remarks concerning the extended features of the solvers omitted in the tutorial part of this documentation.

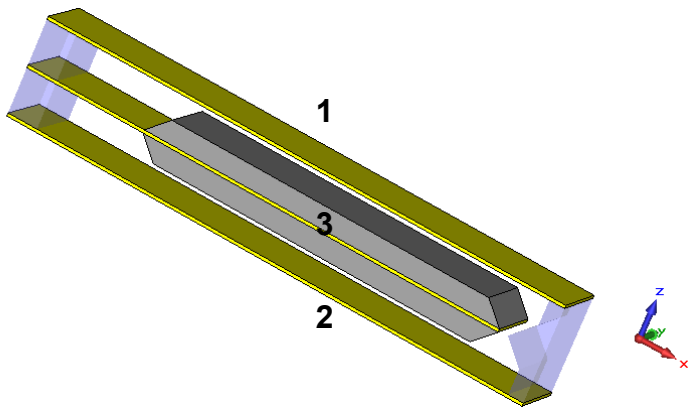
The following explanations describe the “long” way to open a particular dialog box or to launch a particular command. Whenever available, the corresponding Ribbon item will be displayed next to the command description. Because of the limited space in this manual, the shortest way to activate a particular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

Structural Mechanics Solver

In this example you will model a simple accelerometer. At first, the spatial structure will be created, and material properties will be defined. Then, boundary conditions will be specified and the solver will be configured and started. Finally, it will be shown how the solution results should be interpreted.

The Structure

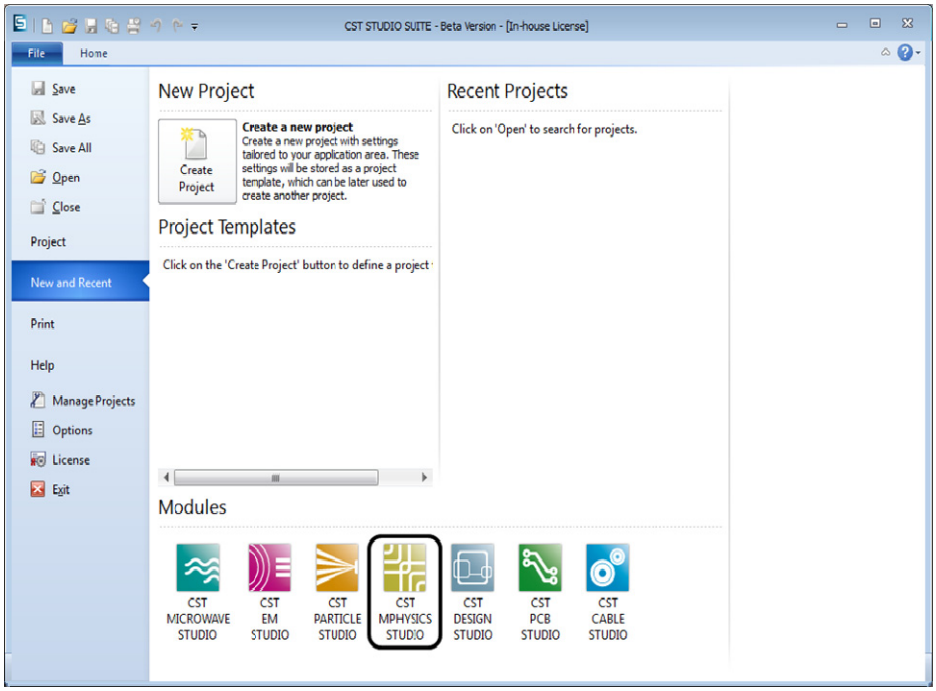
The following picture demonstrates the spatial structure of a simple accelerometer. It consists of two fixed flat conductors with a potential difference applied, and a movable conductor between them.



If the system moves with acceleration, the inertial force pushes the movable conductor towards one of the fixed ones. The potential difference, e.g., between the conductors 2 and 3 changes proportionally.

Create a New Project

After starting CST DESIGN ENVIRONMENT, please select CST MPHYSICS STUDIO from the list of installed modules:

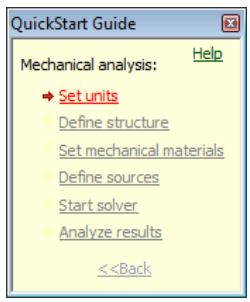


After a new CST MPHYSICS STUDIO project is created, you can switch the problem type to *Mechanics* by selecting *Home: Simulation* ⇨ *Problem Type* ⇨ *Mechanics* .

Open the QuickStart Guide

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

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



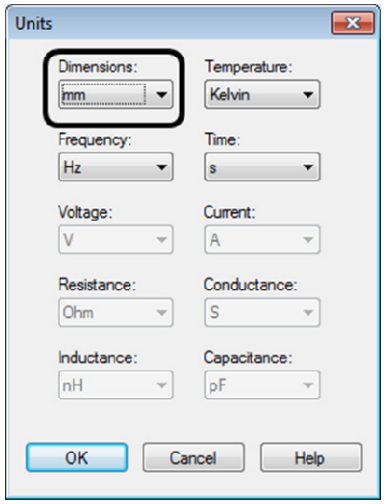
The red arrow always indicates the next step necessary for your problem definition. You do not need to process the steps in this order, but we recommend you follow this guide at the beginning in order to ensure all necessary steps have been completed.

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

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

Define the Units

By default, m is selected as the dimensions unit. Please change this setting by selecting *Home: Settings* ⇨ *Units*. In the *Units* dialog, please select *mm* for dimensions:



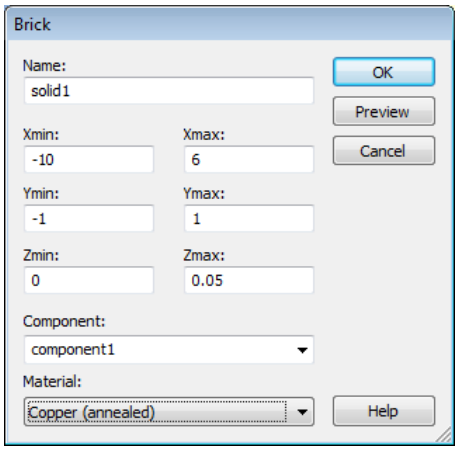
Model the Structure

The first step is to create a brick.

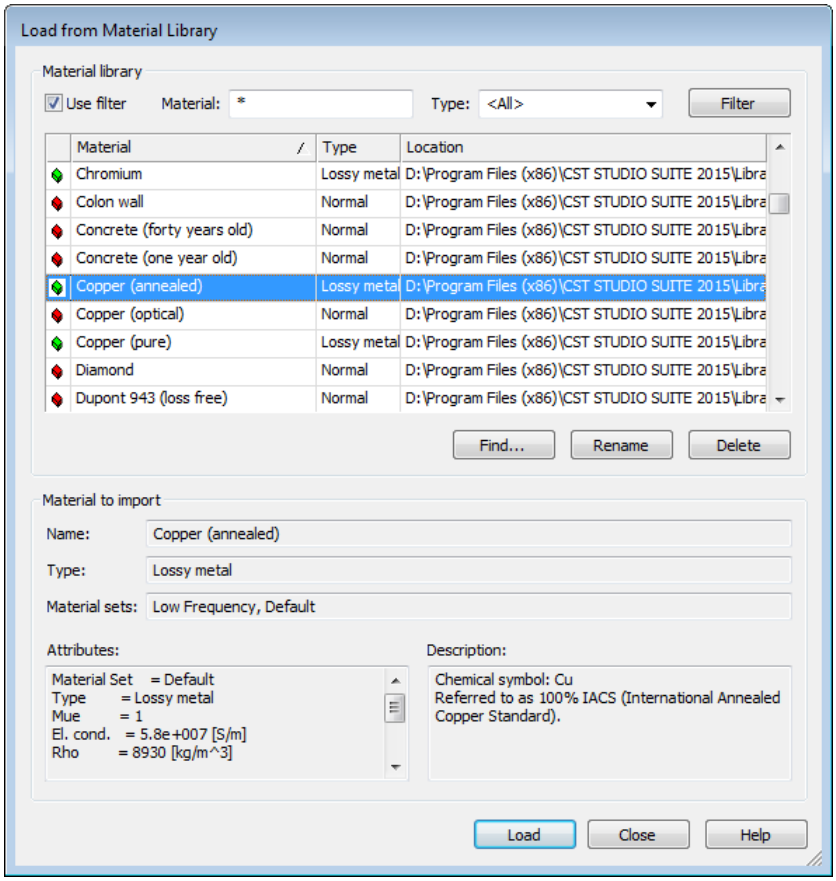
1. Select the brick creation tool from the main menu: *Modeling: Shapes* ⇨ *Brick*.
2. Press the *Escape* key, in order to open the dialog box.
3. Fill up the brick size fields as it is shown in the table below.

Xmin	-10	**Xmax**	6
Ymin	-1	**Ymax**	1
Zmin	0	**Zmax**	0.05

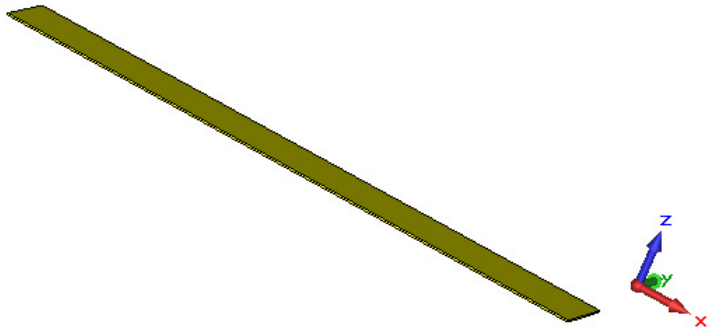
4. In order to select the material, click on the corresponding combo box and select *[Load from Material Library...]*.



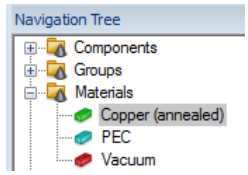
5. In the dialog *Load from Material Library*, select *Copper (annealed)* from the material list and click the *Load* button.



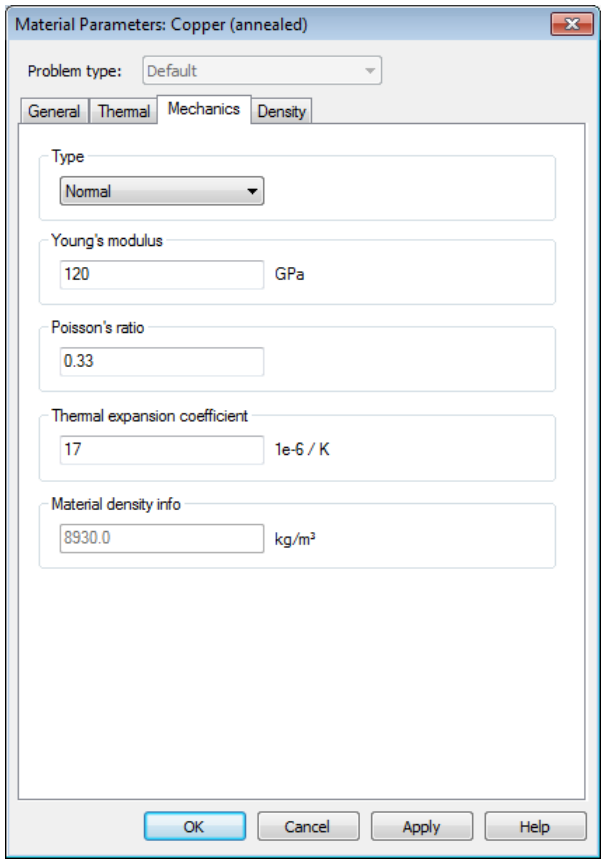
6. Finally, click the *OK* button in the *Brick* dialog. Now a new brick has been created:



7. Let us explore the material properties of the newly created object. Open the Materials folder in the Navigation Tree and double-click the item *Copper (annealed)*.



The dialog box *Material Parameters: Copper (annealed)* appears where various properties of copper can be modified. Select the tab *Mechanics* in this dialog box.



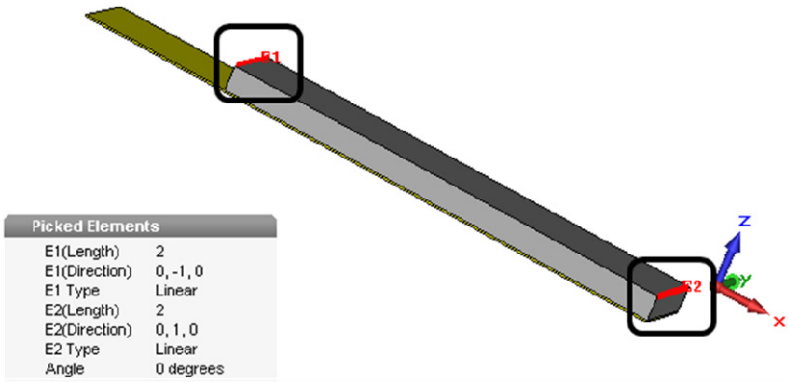
In this tab you can change the mechanical properties of the selected material. There are three most important mechanical properties:

- *Young's modulus* defines the stiffness of an isotropic elastic material. It is normally measured in GPa, or kN/mm^2. The typical values vary between 0.01 GPa (rubber) and over 1000 GPa (diamond). It is important to know the value of this material parameter very well, since it has a large influence on the accuracy of the solution.
- *Poisson's ratio* defines the scale of the transverse contraction of a longitudinally stretched body. This parameter can vary between -1 and 0.5, whereas most of the materials are characterized by a positive Poisson's ratio.
- *Thermal expansion coefficient* is the strain of a body if its temperature changes by 1 K. This value is utilized to compute strain induced by an external temperature field.

8. Now press Cancel and create a new brick (*Modeling: Shapes ⇨ Brick*) with the following size:

Xmin	-6	**Xmax**	6
Ymin	-1	**Ymax**	1
Zmin	0.05	**Zmax**	0.7

9. Select the material Steel-1010 from the material library.
10. By selecting *Modeling: Picks ⇨ Picks (S)* activate the general pick tool to pick two edges of the second brick, as shown in the picture below:

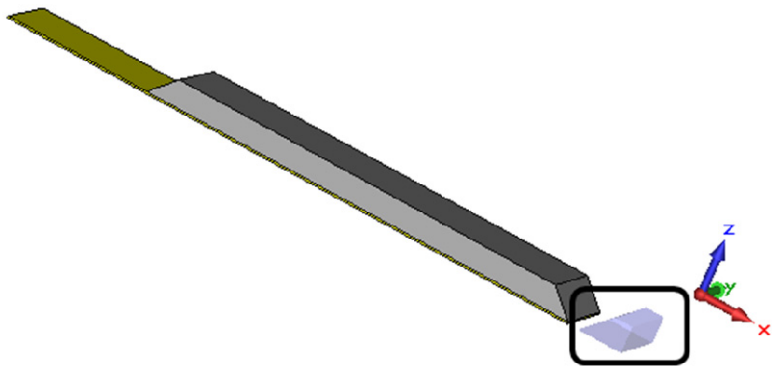


11. Select *Modeling: Tools ⇨ Blend ⇨ Chamfer Edges* in order to chamfer the selected edges. Enter the chamfer width of 0.65, and keep the default angle of 45° in the appearing dialog box and click the *OK* button.
12. Again, open the Brick dialog and enter the following values:

Xmin	6.3	**Xmax**	7.5
Ymin	-1	**Ymax**	1
Zmin	0	**Zmax**	0.7

Select material *Plexiglas* for it from the material library.

13. Pick and chamfer one edge with the chamfer width of 0.7, and keep the default angle of 45° in order to obtain the following structure:



14. Create the following bricks:
    - One of material *Plexiglas* with the following size:

Xmin	7	**Xmax**	7.5
Ymin	-1	**Ymax**	1
Zmin	0.7	**Zmax**	1.5

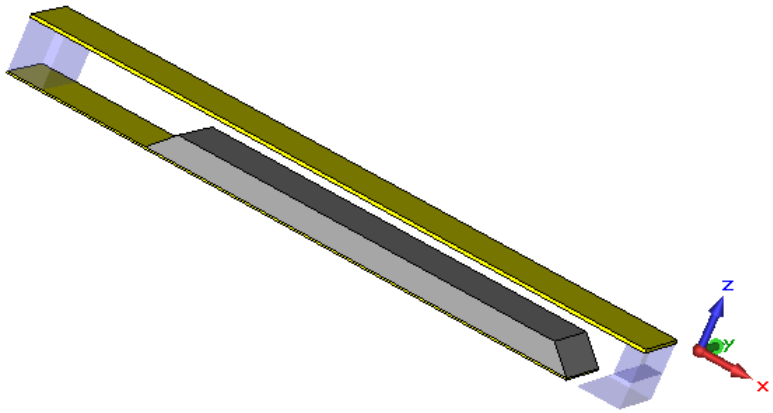
- Another of *Plexiglas* with the following size:

Xmin	-10	**Xmax**	-9
Ymin	-1	**Ymax**	1
Zmin	0.05	**Zmax**	1.5

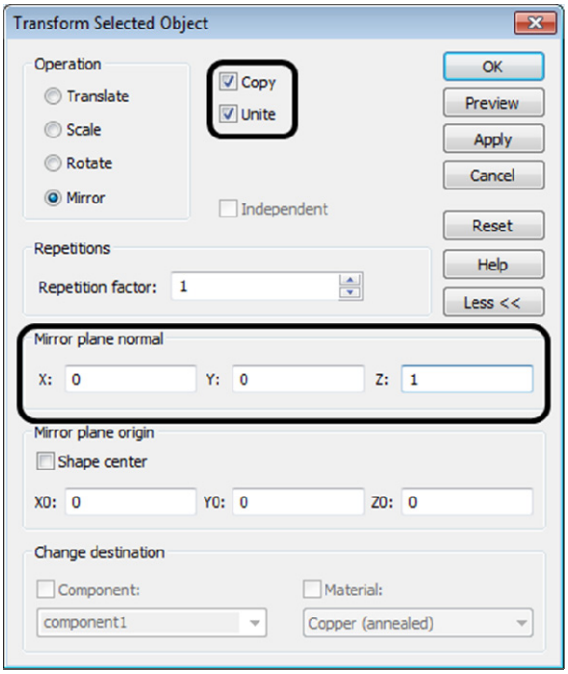
- A last one made of *Copper (annealed)* with the following dimensions:

Xmin	-10	**Xmax**	7.5
Ymin	-1	**Ymax**	1
Zmin	1.5	**Zmax**	1.6

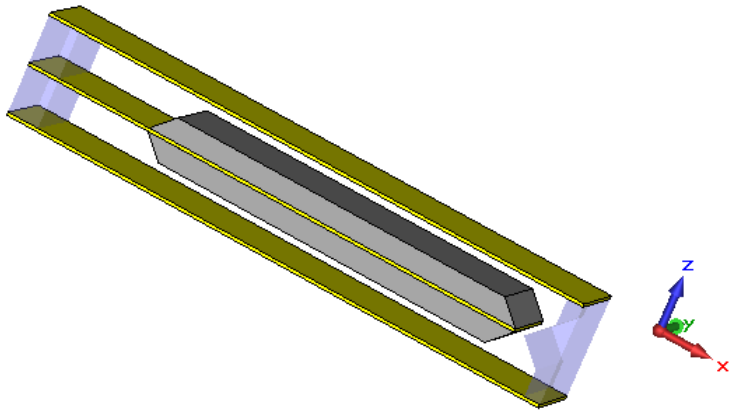
- The result should be as shown in the following picture:



15. In Navigation Tree to the left of the main document window, open the item *Components* and select *component1*. Afterwards activate *Modeling: Tools* ⇨ *Transform* .
16. In the dialog *Transform Selected Object* select the operation *Mirror*, check the boxes *Copy* and *Unite* and set the mirror plane normal to 0, 0, 1, as shown in the following picture:



17. Click *OK* button. Now the geometric structure setup is complete:

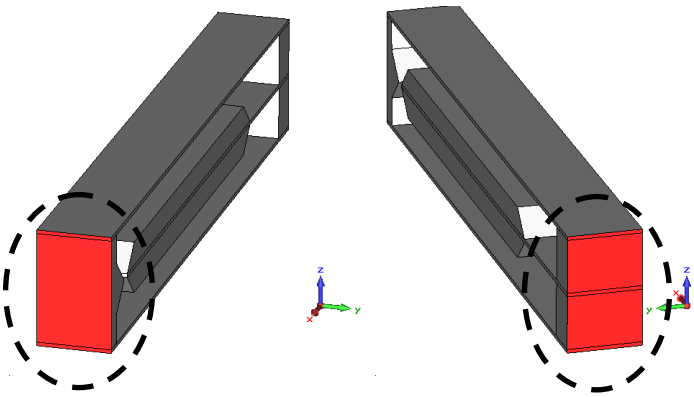


Traction and Displacement Boundaries

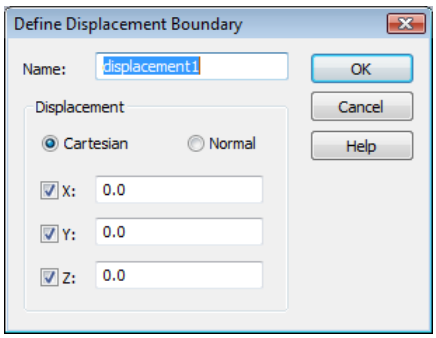
After the spatial structure has been built, the next step is to define the displacement and traction boundaries. Displacement boundaries refer to the surfaces of the model which have been shifted by a certain distance in a certain direction. To fix a surface at its initial position it is also possible to set the displacement values to zero.
Traction boundaries are the surfaces, where a certain pressure in a certain direction is applied. Both displacements and tractions are defined as vectors in the Cartesian coordinate system.

In the present example let us fix the both sides of the model and apply a pressure to the middle electrode, which would mimic the influence of inertial forces during acceleration. The following steps must be performed:

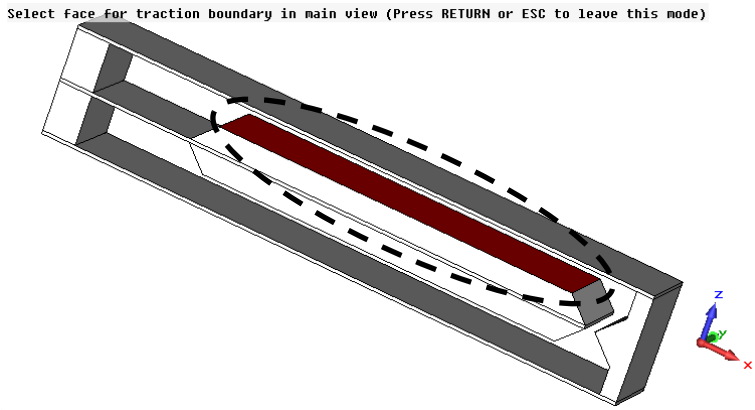
1. Press the toolbar button *Simulation: Boundaries* ⇨ *Displacement Boundary* .
2. Select the side faces of the model, as shown in the picture below (you have to select five faces at x-min and 3 faces at x-max):



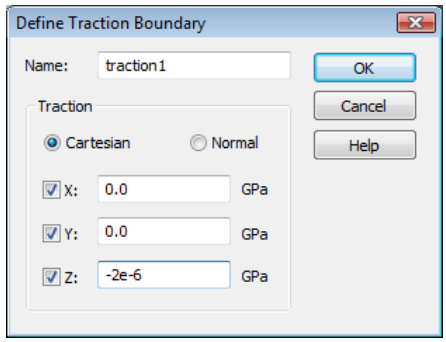
After pressing the *Return* key, the dialog box *Define Displacement Boundary* will appear:



3. Keep the zero values for all the components of the displacement vector and press *OK* button. Now the sides of the model are fixed in space.
4. Press the toolbar button *Simulation: Boundaries* ⇨ *Traction Boundary* .
5. Double-click the upper surface of the third electrode:



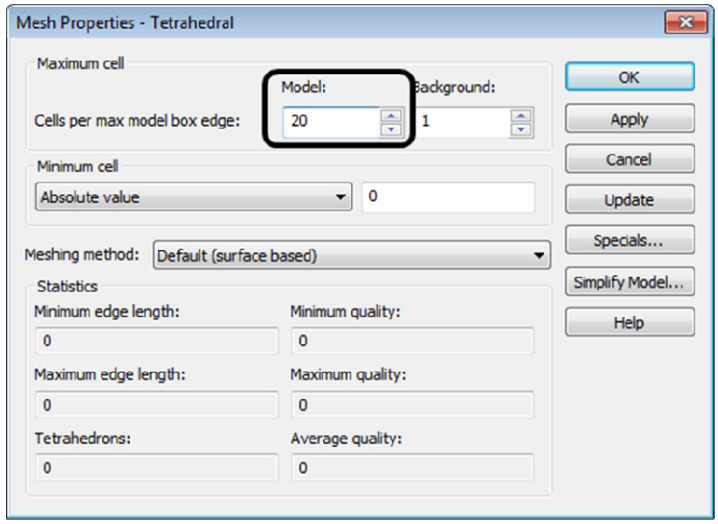
After pressing the *Return* key, the dialog box *Define Traction Boundary* will appear:



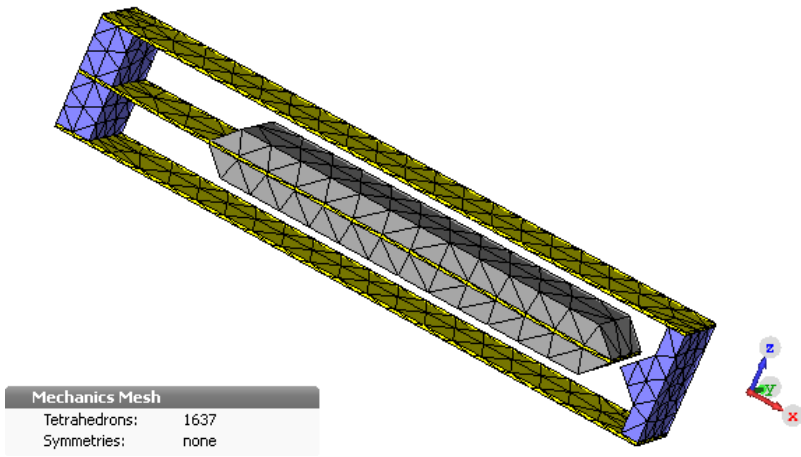
6. Put the value of -2e-6 GPa as the Z-coordinate of the traction vector. This means, that the pressure of 2 kPa is applied towards the negative direction of the Z-axis. This pressure would roughly correspond to the acceleration of ca. 17*g, or 170 m/s^2, into the positive Z-direction.

Mesh Settings

The structural mechanics solver is quite sensitive with respect to the quality of discretization. In order to obtain reliable results, the default mesh density needs to be increased. To do this, press the toolbar button *Simulation: Mesh ⇨ Global Properties* . In the *Mesh Properties – Tetrahedral* dialog, change the *Cells per max model box edge* setting for *Model* to 20:



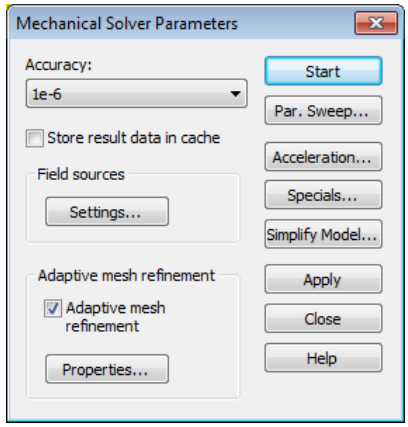
This will increase the density of generated mesh. In order to check the resulting mesh, you may press the Update button:



Now press the OK button to accept the changes and close the window.

Start the Simulation

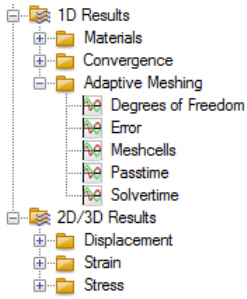
Now, as all the settings have been made, it is time to start the mechanical solver. Press the toolbar button *Simulation: Solver* ⇨ *Setup Solver* . The structural mechanics solver parameter dialog box appears.



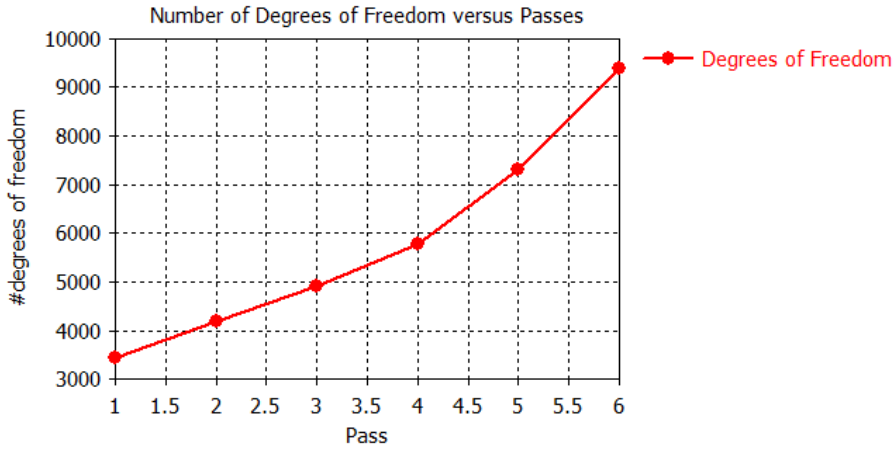
You can click the *Help* button in order to learn more about the controls in this dialog box. For now, the default settings are good enough, so just click the *Start* button. After the computation has been started, you can control the execution of the solver in the *Progress* and *Messages* windows.

Analyze the Solution of the Tetrahedral Solver

After the mechanical solver finishes the computation, several items appear in the Navigation Tree.

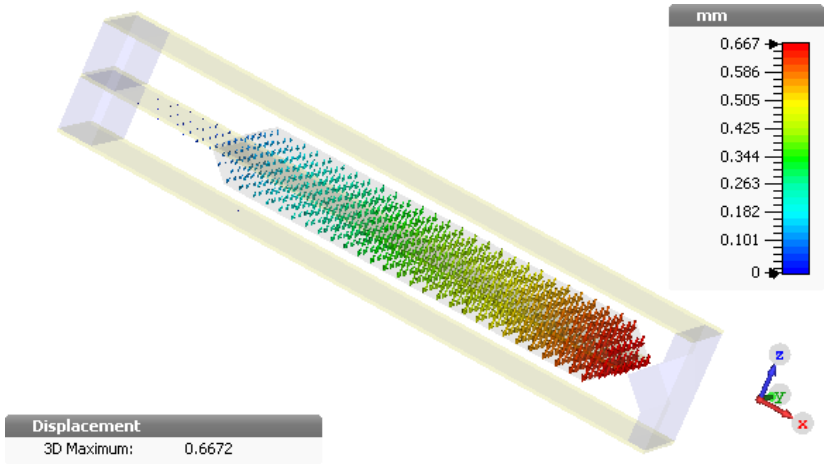


The directory *NT: 1D Results* ⇨ *Adaptive Meshing* contains information on the adaptive mesh refinement performed by the solver. Here you can inspect the number of cells in the mesh for each iteration step, time, used by the solver to generate the solution, as well as the relative error of the solution. For example, in the picture below you can see the number of degrees of freedom in the solution for each step of mesh refinement. Please note that the exact values may be slightly different on different systems.



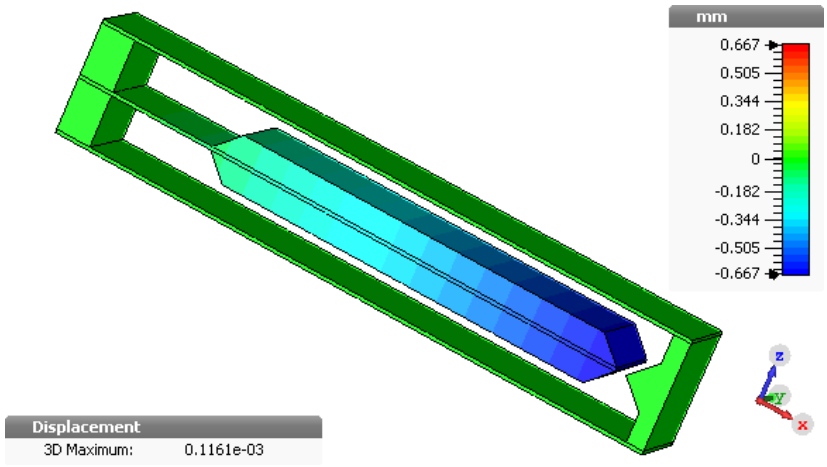
The directory *2D/3D Results* contains the distributions of displacement, strain and stress within the solution domain. If a temperature distribution has been imported from the thermal solver, the results of its interpolation onto the tetrahedral mesh are also displayed here.

A click on the item *NT: 2D/3D Results* ⇨ *Displacement* displays a vector plot of the body deformation, as shown in the following picture.

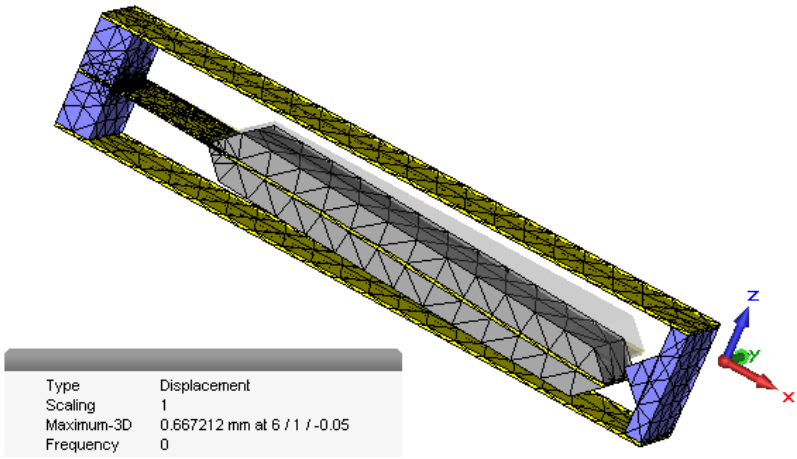


This Navigation Tree item also contains the following sub-items:

- Clicking on *X*, *Y* or *Z* sub-items displays the corresponding component of the displacement vector. The example below demonstrates the displacement of the solution domain in the Z-direction.



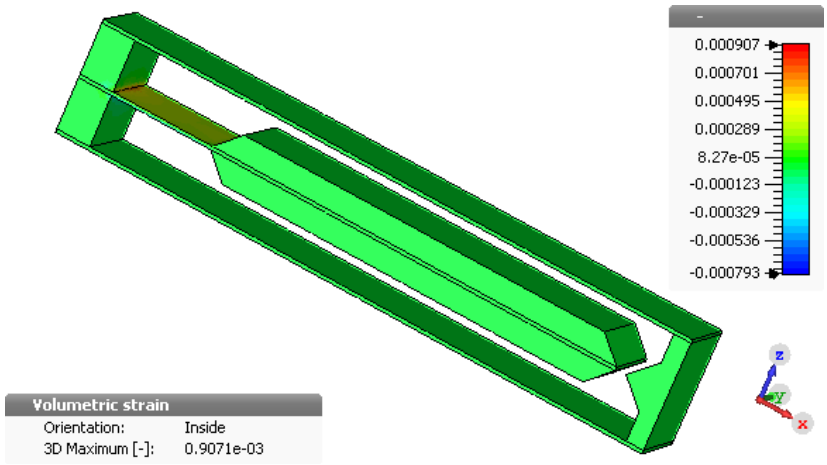
- Sub-item *Abs* demonstrates the distribution of the absolute value of displacement within the solution domain.
- Sub-item *Normal* and *Tangential* demonstrate the length of the corresponding projection of displacement vector onto each body surface.
- Sub-item *Deformed Mesh* demonstrates the actual deformation of the solution domain, as shown in the picture below. Since the amplitude of the mesh deformation is normally quite small compared to the linear dimensions of the model, it can be adjusted by choosing *2D/3D Plot: Plot Properties* ⇨ *Properties* (or *Plot Properties* in the context menu). A dialog box will be shown, where the mesh deformation can be scaled.



Here, the original shape of the model is shown semi-transparently, whereas the deformed tetrahedral mesh is solid.

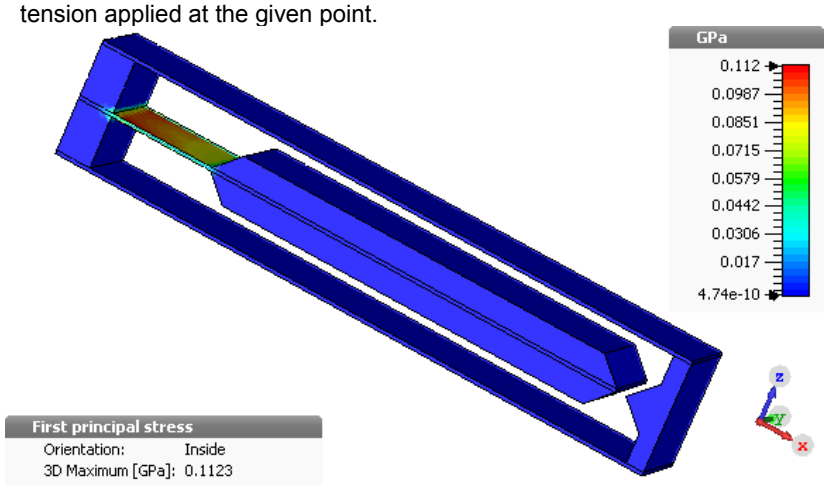
Navigation Tree item *NT: 2D/3D Results* ⇨ *Strain* contains the following sub-items:

- Directory *Components* contains the components XX, YY, ZZ, XY, XZ and YZ of the strain tensor.
- Sub-item *Volumetric* displays the distribution of the volumetric strain in the model, which means the relative volume change in each node of the solution domain. The negative values mean contraction, whereas the positive values mean expansion.



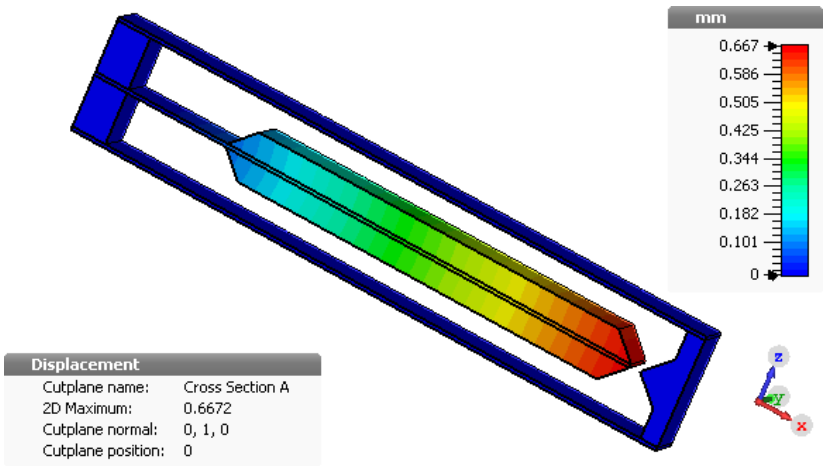
Finally, Navigation Tree item *NT: 2D/3D Results ⇨ Stress* contains the following entries:

- Directory *Components* contains the components XX, YY, ZZ, XY, XZ and YZ of the stress tensor.
- The tree-entry *Von Mises* displays the distribution of von Mises stress within the solution domain. If this stress at some location is higher than the yield strength of the corresponding material, plastic deformation takes place in this location. Von Mises stress is always positive.
- The tree-entry *Hydrostatic* displays the hydrostatic stress distribution, reproducing the change of the volume in the stressed body. The negative values mean contraction forces.
- The tree-entry *First Principal Stress* displays the distribution of the largest eigenvalue of the stress tensor in the solution domain. The first principal stress is the largest tension applied at the given point.



Another interesting feature is the visualization of computation results on a cutting plane. Select *2D/3D Plot: Sectional View ⇨ 3D Fields on 2D Plane* from the toolbar to enter this mode. By default, the cutting plane is perpendicular to the X-axis. Its orientation can be modified from the toolbar by changing the *2D/3D Plot: Sectional View ⇨ Normal* setting. Also the position of the cutting plane can be changed in this way.

In the following picture the distribution of the absolute value of displacement vector is shown on the cutting plane perpendicular to the Y-axis.



Vector fields can be visualized on a cutting plane in the same manner. Just click on the Navigation Tree item: *NT: 2D/3D Results ⇨ Displacement*. In this case the 3D-to-2D mode stays activated.

Summary

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

1. Model the structures by using the solid modeler;
2. Define and modify various material parameters;
3. Assign displacement and traction boundaries;
4. Start the structural mechanics solver;
5. Explore the results of adaptive mesh refinement;
6. Visualize various distributions delivered by the mechanical solver;
7. Visualize the deformation of the mesh and scale it.

If you are familiar with all these topics, you have a very good starting point for further improving your usage of CST MPHYSICS 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 demonstrated how to perform a simple mechanical simulation. The general workflow for thermal problems is quite similar to a mechanical application.

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

Mechanical Solver:

- Displacement boundary: *Simulation: Boundaries ⇨ Displacement Boundary*
- Traction boundary: *Simulation: Boundaries ⇨ Traction Boundary*
- External temperature or force distribution:
  Simulation: Sources ⇨ Field Import

Thermal Solvers:

- Fixed temperature: *Simulation: Sources and Loads ⇨ Temperature Source*
- Heat source: *Simulation: Sources and Loads ⇨ Heat Source*
- Volume heat source: *Simulation: Sources and Loads⇨ Volume Heat Source*
- Thermal losses from an electromagnetic or particle simulation:
  Simulation: Sources and Loads ⇨ Thermal Losses
- Thermal contact resistance:
  Simulation: Sources and Loads ⇨ Contact Impedance
- Convection and radiation at surfaces:
  Simulation: Sources and Loads ⇨ Thermal Surface

Mechanical Solver

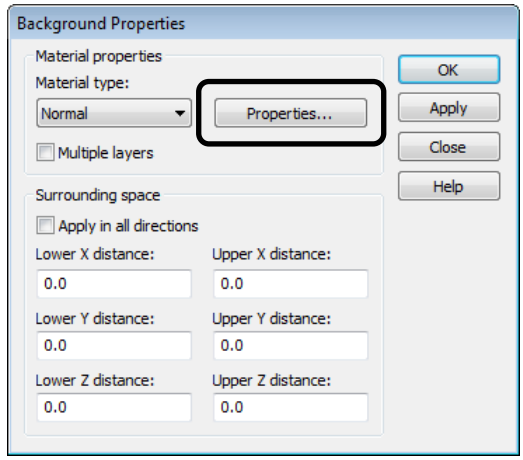
The mechanics solver is a tetrahedral based solver for structural mechanic problems. Its main application is computing deformations driven by thermal expansion and external forces. The deformation results can be used for a subsequent High Frequency Electromagnetic analysis with the tetrahedral based frequency domain solvers from CST MICROWAVE STUDIO.
Refer to the chapter *Simulation Workflow* for a description of the basic features. The import of temperature and force density distributions is described in the section *Workflow for Coupled Simulations*.

Thermal Solvers

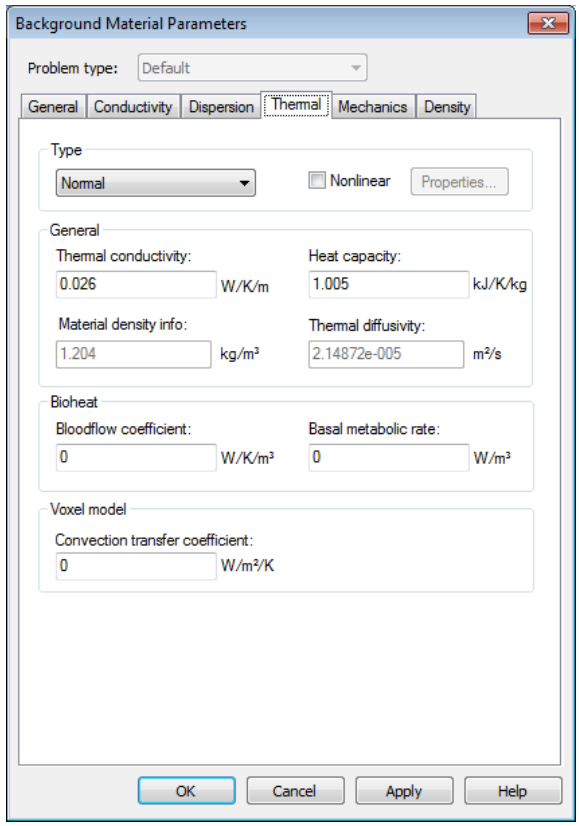
CST MPHYSICS STUDIO includes a stationary and a transient thermal solver, based both on hexahedral and tetrahedral grids. Their main applications are solving stationary or transient temperature problems resulting from various types of losses.
In addition, the thermal solvers are also well suited to compute standalone thermal problems. The following section demonstrates the most important aspects of a thermal simulation with CST MPHYSICS STUDIO.

Background Material

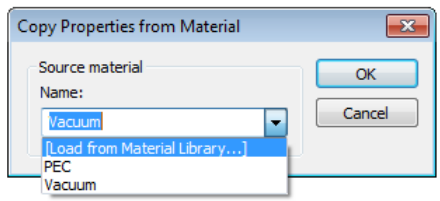
The first step for setting up a thermal simulation is to define an appropriate background material. Open the material background properties dialog box by selecting *Modeling: Materials* ⇨ *Background* :



To edit the thermal properties, select *Normal* for the *Material type*, afterwards open the material dialog box by pressing *Properties...* and select the *Thermal* property page:



In most cases it is advisable to define a "Normal" Material Type with a thermal conductivity of air (air at 300 K: 0.026 $WK^{-1}m^{-1}$). For a transient simulation a heat capacity and a material density also need to be defined. The easiest way to assign these values is to copy the properties from *Air* in the material library. Press the *Copy Properties from Material…* button in the *General* tab, select *[Load from Material Library…]* in the Copy Properties from Material dialog box:



Now select *Air* from the material list.

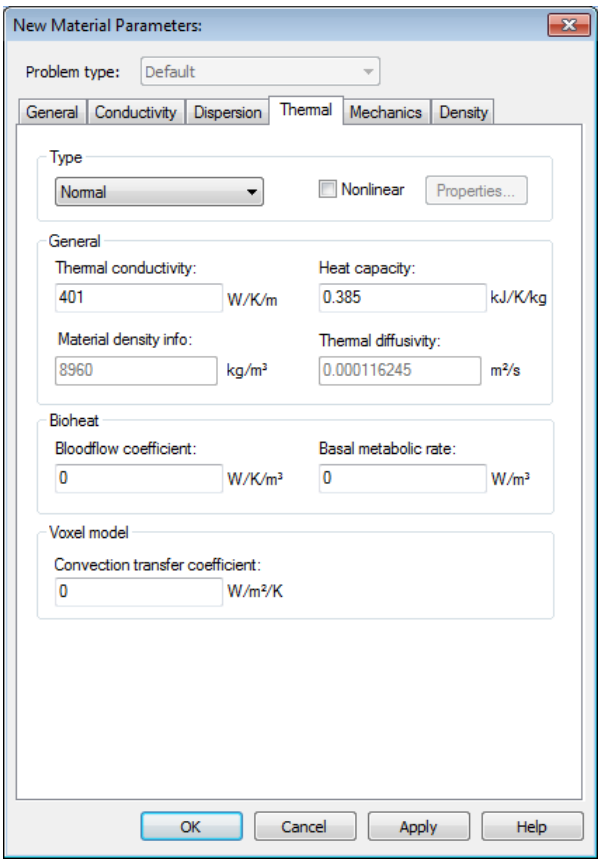
Material Properties

The material parameters for a thermal problem can be defined inside the material parameters dialog box: *Modeling: Materials* ⇨ *New/Edit* ⇨ *New Material* . Select the *Thermal* tab.

Here you can specify a thermal conductivity for your material in W K^{-1} m^{-1} in case a *Normal* or *Anisotropic* thermal material *Type* has been selected. If a temperature dependent thermal conductivity, heat capacity and/or blood flow coefficient should be taken into account, activate the checkbox *Nonlinear* and define the material curve by entering the corresponding dialog box via *Properties*…

If you select a PTC (Perfect Thermal Conductor) type, an infinite thermal conductivity is assumed. A body with PTC material assigned always has a uniform temperature.

For transient thermal problems (see also the next chapter) the heat capacity and the material density play an important role in the simulation process. These parameters determine how much energy per degree Kelvin is stored in a certain amount of mass or volume.



Because the thermal diffusivity plays an important role for the transient simulation process, it is shown here as well. The diffusivity can be calculated from the conductivity, the heat capacity and the density as follows:

$$\alpha = \frac{k}{\rho \cdot c_P \cdot 1000}$$

With:

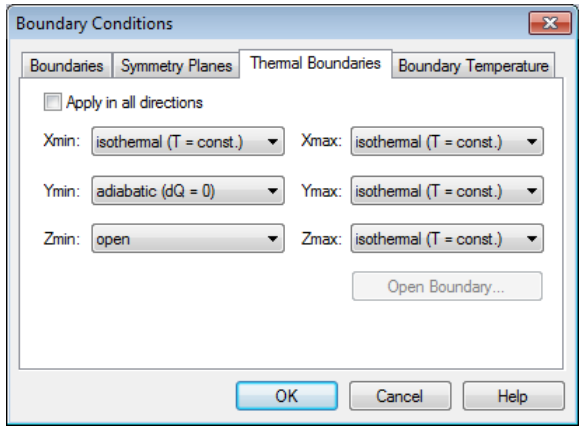
α: Diffusivity [m² / s]
k: Thermal conductivity [W / K /m]
ρ: Density [kg / m³]
c_P: Specific heat capacity [kJ / K / kg]

Nonlinear heat capacity can be used for simulation of material phase change in transient computations. This can be achieved by a local increase of heat capacity for a small interval of temperatures. For more information on simulation of phase changes, please refer to the online help.

For simulations which involve biological materials heating mechanisms of living tissue can be taken into account (see also: Bioheat Source below). In addition it is possible to define a convection coefficient for surface materials of human voxel models (typically: skin).

Boundary Conditions

The boundary conditions for the thermal solver can be defined in the *Thermal Boundaries* tab of the *Boundary Conditions* dialog box (*Simulation: Settings ⇨ Boundaries*)



For the *"open"* boundary condition, it is assumed that the temperature value approaches the ambient temperature with increasing distance from the structure (the ambient temperature can be defined inside the thermal solver dialog box). Apply this type of boundary condition if thermal conduction through the surrounding background material plays an important role for your problem.

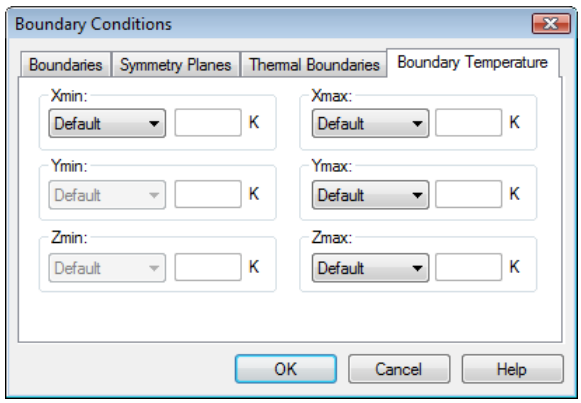
When no heat flow leaves the computational domain through a boundary, use the "*adiabatic"* boundary condition. In case the conductive heat flow of an open structure can be neglected, you can use these boundary conditions instead of "*open*" boundary conditions (if radiation or convection effects dominate).

The "*isothermal"* boundary condition forces the temperature to be constant at this boundary. As a consequence, the tangential component of the heat flow density is forced to be zero here.

The following table shows an overview, where **T** is the temperature and **Q** is the heat flux density:

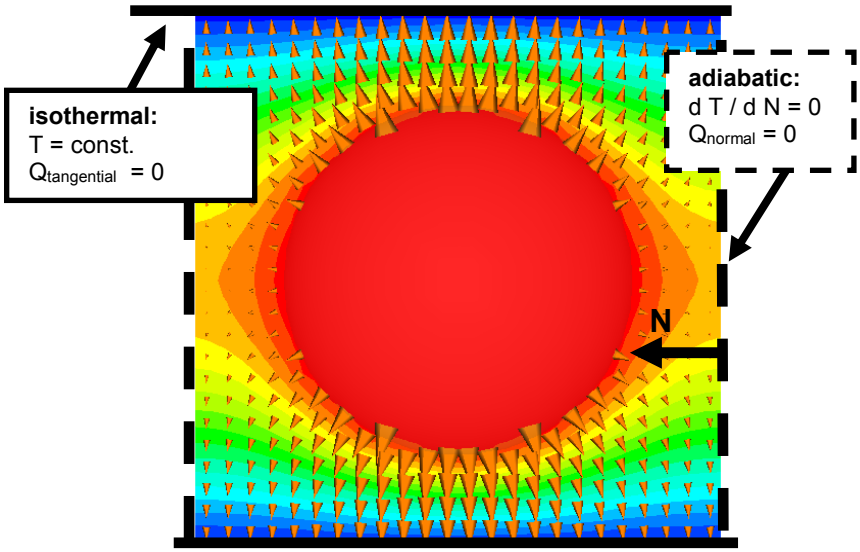
	Temperature (**T**)	Heat Flow (**Q**)
Isothermal	**T** = const.(fixed or floating)	$\mathbf{Q}_{tangential} = 0$
Adiabatic	d **T** / dN = 0	$\mathbf{Q}_{normal} = 0$
Open	$\lim_{R\to\infty}(\mathbf{T}) = T_{Ambient}$	

The temperature value at "*isothermal"* boundaries can be specified inside the *Boundary Temperature* tab:



Note that if no temperature value is defined at an "isothermal" boundary the ambient temperature is used (default value).

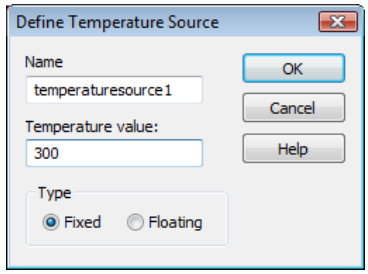
The picture below illustrates an example of how thermal fields are influenced by the different boundary types. It shows a metal sphere at a constant temperature, which is surrounded by a material with constant thermal conductivity.



The thermal solvers can handle five different types of sources or loss mechanisms, which are listed below:

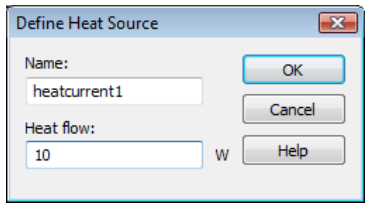
Temperature Source

This source is available via *Simulation: Sources and Loads* ⇨ *Temperature Source* . This source type can only be assigned to a surface of an object with PTC material properties. You can choose between a fixed temperature value and a floating temperature. A floating temperature is a constant temperature distribution with zero heat flow from or into the associated surface.



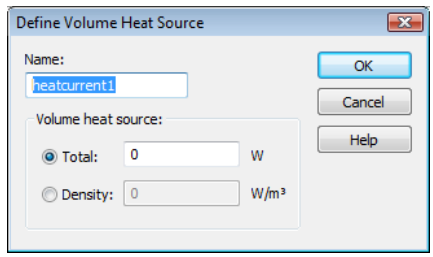
Heat Source

This source is available via *Simulation: Sources and Loads* ⇨ *Heat Source* , it can only be assigned to a PTC solid. A heat source defines the total heat flow from a PTC surface. Therefore a heat source with zero heat flow and a floating temperature are identical.



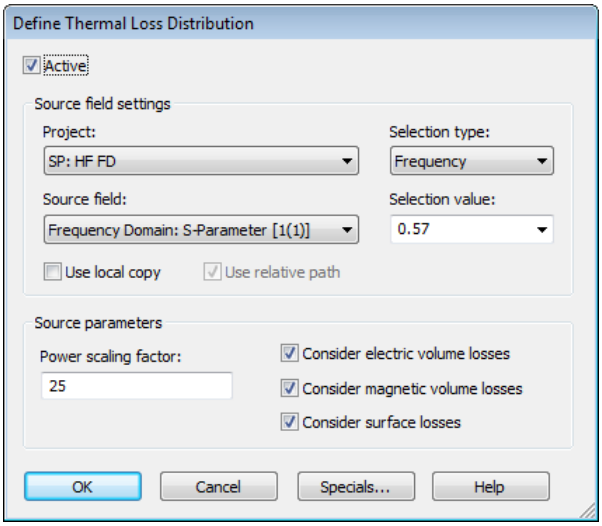
Volume Heat Source

This source is available via *Simulation: Sources and Loads* ⇨ *Volume Heat Source* , it can only be assigned to a non-PTC solid with a non-zero thermal conductivity. A volume heat source defines the thermal power evenly released within the solid. The user may define the total power released within the solid (*Total*) or the volume heat density (*Density*).



Thermal Loss Distribution

This source is available via *Simulation: Sources and Loads* ⇨ *Thermal Losses*. Thermal losses can occur inside electric materials with finite conductivity, on surfaces of good conductors, inside dispersive materials or at materials where particles hit the surface. These loss distributions can be imported and used as thermal sources inside thermal conductive materials. If previously calculated loss distributions are present, you can edit setting by reopening the dialog box (*Simulation: Sources and Loads* ⇨ *Thermal Losses*).



It is possible to choose source fields from the same project or from an external project. The following table shows a list of loss types and which solver from the CST STUDIO SUITE can create these losses.

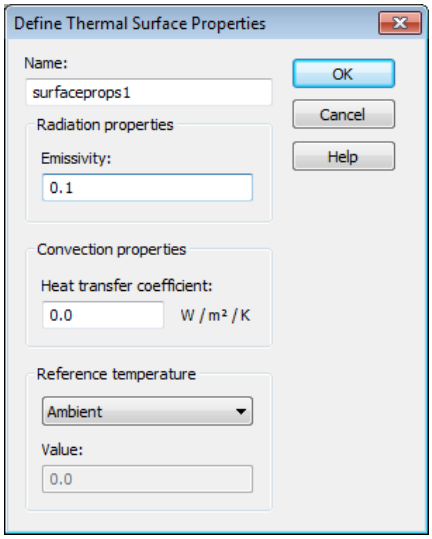
Type of loss	**Created by**
Ohmic (electric vol. losses)	Transient Solver (T), Frequency Domain Solver (F), Eigenmode Solver (E), J-Static Solver (Js), LF-Solver (LF), PIC Solver (Pic), Wakefield Solver (WAK)
Lossy metal (surface losses)	Transient Solver (T), Frequency Domain Solver (F), Eigenmode Solver (E), LF-Solver (LF), PIC Solver (Pic), Wakefield Solver (WAK)
Dispersive (electric and magnetic vol. losses)	Transient Solver (T), Frequency Domain Solver (F), PIC Solver (Pic), Wakefield Solver (WAK)
Crashed particles	Tracking Solver (Trk), PIC Solver (Pic)

For further details refer to the online-help.

Thermal Surface Properties

Thermal surface properties are available via *Simulation: Sources and Loads* ⇨ *Thermal Surface* .

Thermal surface properties can be assigned to surfaces of thermally conductive materials. A thermal surface property definition describes the radiation and convection losses from a surface:



The *Emissivity* ε is a dimensionless constant between 0 and 1 which describes the radiation capability of the selected surface

$$Q_{Radiation} = A_{Surface} \cdot \varepsilon \cdot \sigma \cdot (T^4 - T_{\text{Reference}}^4),$$

where $Q_{Radiation}$ stands for the radiated power, T for the surface temperature, $T_{\text{Reference}}$ for the reference temperature, which can be equal to ambient or user-defined, σ for the Stefan-Boltzmann constant and $A_{Surface}$ for the area for the selected surfaces. An emissivity value $\varepsilon = 0$ means that the surface does not lose thermal power by radiation. A value of 1 means that the thermal power emitted by the surface equals to that of a black body at the same temperature.

The *Heat transfer coefficient* h describes convection processes at the surfaces of conductive materials

$$Q_{Convection} = A_{Surface} \cdot h \cdot (T - T_{\text{Reference}}),$$

where $Q_{Convection}$ denotes the power, T the surfaces temperature, $T_{\text{Reference}}$ the reference temperature and $A_{Surface}$ the area for the selected surfaces.

Bioheat Source

As described above it is possible to assign biological properties to a material. Two different heating mechanisms are available:

The *Bloodflow coefficient* determines the influence of blood at a certain temperature T_{Blood} inside the tissue volume *V*.

$$Q_{Bloodflow} = V \cdot C_{Bloodflow} \cdot (T_{Blood} - T)$$

Depending if the current temperature value *T* is higher or lower than the blood temperature this mechanism cools or heats the surrounding material. The blood temperature value can be edited inside the Specials dialog box of the thermal solvers (*Simulation: Solver* ⇨ *Setup Solver* ⇨ *Specials* or *Simulation: Solver* ⇨ *Setup Solver* ⇨ *Specials*).

An important mechanism of local thermoregulation in the living tissues is the increasing of bloodflow coefficient with rising temperature due to widening of blood vessels (vasodilation). In order to match clinical studies, the bloodflow coefficient is typically assumed to change exponentially with increasing temperature. The parameters of this dependency can be set in the *Nonlinear Thermal Material Properties* dialog, accessible through the *Nonlinear Properties* button in the *Thermal* tab of the *Material Properties* dialog. For more information about these parameters please refer to the online help.

The *Basal metabolic rate* describes the amount of heat $Q_{Metabolic}$ which is produced by tissue per volume *V*.

$$Q_{Metabolic} = V \cdot C_{Metabolic}$$

This kind of sources is only available for hexahedral thermal solvers.

Thermal Contact Impedance

A thermal contact impedance can be defined via *Simulation: Sources and Loads* ⇨ *Contact Impedance*. A contact impedance is equivalent to a thin layer of thermal conductive material at the interface between two (or several) solids. It can be characterized either by lumped parameters (thermal resistance [K/W] and thermal capacitance [J/K]), or by its thickness and the thermal properties of material assigned. Both definitions are equivalent and can be easily converted into each other:

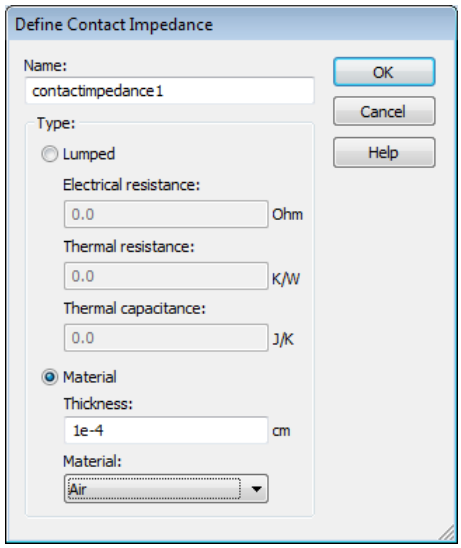
Thermal resistance (K/W): $$R_\theta = \frac{1}{\lambda}\frac{l}{A}$$

Thermal capacitance (J/K): $$C = \int_\Omega c \cdot \rho \, dV$$

Here R_θ and C are the lumped parameters representing the integral thermal resistance and thermal capacitance of the contact layer, respectively. In the material-based representation, thermal conductivity λ, specific heat capacity c, material density ρ and layer thickness l are used. The contact area A is calculated by the solver.

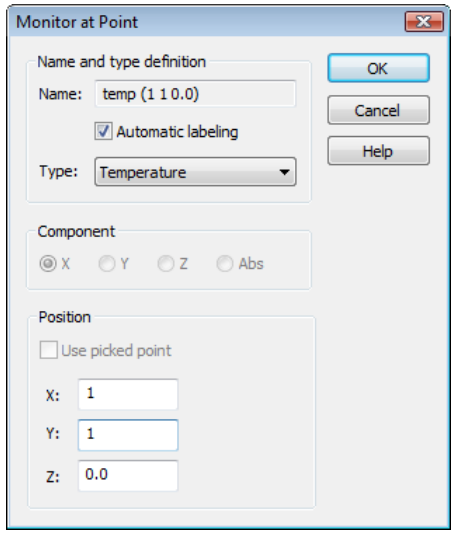
The advantage of contact impedance definition through lumped parameters is the ease and transparency of the parameter values. Besides, it is independent on the contact area A which may vary in case of solid intersections or depending on the mesh settings. On the other hand, the material-based definition offers much more flexibility, for example, it supports nonlinear material properties.

Thermal contact impedances are only supported by tetrahedral-based thermal solvers.



Monitors at Points

The monitors of this kind record scalar values that are defined at a point (e.g. the x-component of the heat current density at a fixed position). You can create such a monitor via *Simulation: Monitors* ⇨ *Monitor at Point* .



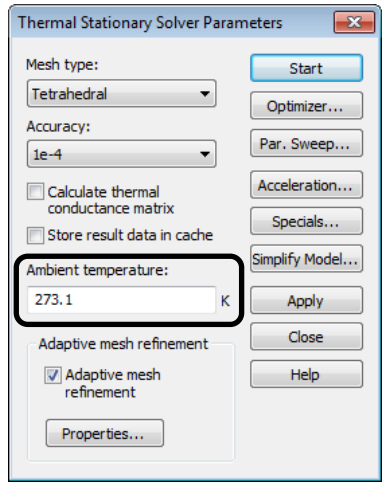
Stationary thermal solver evaluates the temperature values at the monitor points and saves them as 0D data into the Navigation Tree under *NT: Thermal Solver* ⇨ *Temperature 0D* ⇨ *<monitor name>*. Besides, if adaptive mesh refinement is turned on, the tetrahedral-based stationary solver records the temperature value after each refinement step and saves its under *NT: Adaptive Meshing* ⇨ *Temperature 0D* ⇨ *<monitor name>*.

Transient thermal solver records the temperature values at the monitor points during the whole solution time interval.

This monitor type is similar, although not identical, to *Probes* available within CST MICROWAVE STUDIO.

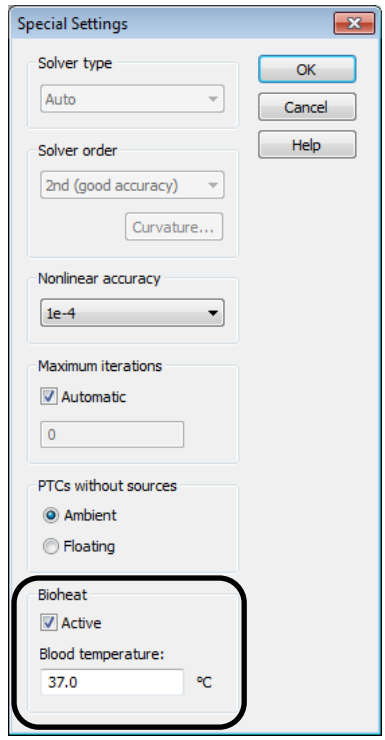
Stationary Thermal Solver Parameters

After the thermal problem has been defined, the stationary solver dialog box can be opened (*Simulation: Solver* ⇨ *Setup Solver*):



Before starting the solver, it is advisable to look at the *Ambient temperature*, which can be the reference temperature for the radiation and convection models as well as for the open boundary condition. Moreover, this temperature may be assigned to PTC regions without user-defined temperature or heat sources.

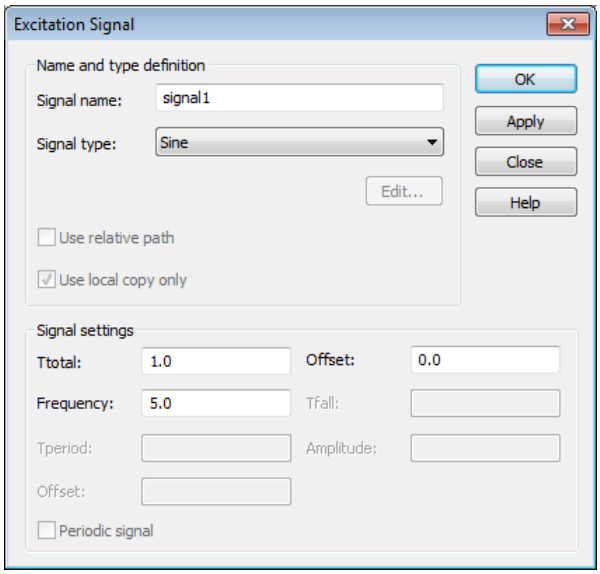
If Bioheat properties must be adjusted, one can open the *Specials* dialog box (the solver *Mesh type* must be set to *Hexahedral*):



This also applies to the transient thermal solver. For further details please refer to the online help.

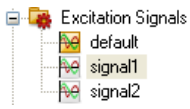
Excitation Signal Settings

For some transient thermal simulations, it is necessary to define time domain excitation signals. A new signal can be defined via *Simulation: Sources and Loads* ⇨ *Signal* ⇨ *New Excitation Signal.* A dialog box opens where a signal type, its parameters and a name can be set.

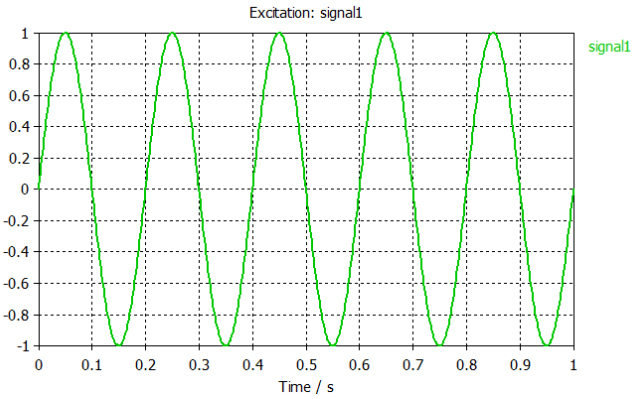


The parameters of the signal depend on the individual signal type and are described in the online help. The parameter *Ttotal* must be set for almost all signal types and defines the size of the definition interval. For time values larger than *Ttotal* the signal is, in general, continued by a constant 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.

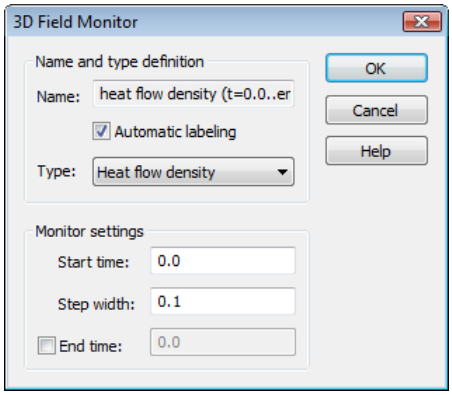


A signal can be displayed by selecting it in the Navigation Tree.



3D Field Monitors

In contrast to stationary solvers, field distributions delivered by transient solvers need to be requested by the user in advance by defining Field Monitors via *Simulation: Monitors* ⇨ *Field Monitor* . A dialog box opens where the type of the field, the start time and the sample step width can be defined:



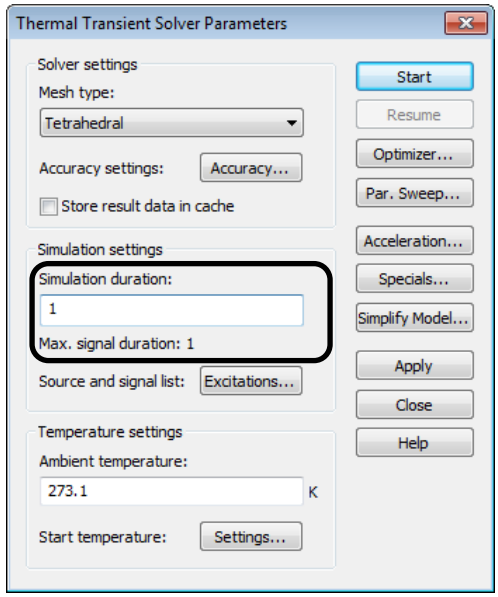
Two field types are available: Temperature and Heat Flow Density. After the solver run has been completed, the recorded result can be accessed via the *2D/3D Results* folder in the Navigation Tree. The scalar or vector field can be animated over the defined time period.

Transient Thermal Solver Settings

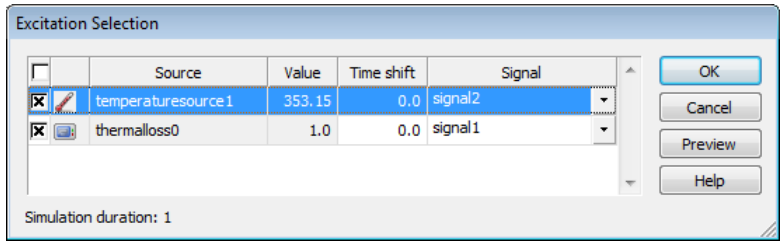
You can switch between the stationary and transient thermal solvers by selecting either

Home: Simulation ⇨ *Setup Solver* ⇨ *Thermal Stationary Solver* or
Home: Simulation ⇨ *Setup Solver* ⇨ *Thermal Transient Solver*.

After selecting the transient solver, the solver parameters dialog box can be opened by clicking on the icon in the *Home* or the *Simulation* ribbon (*Simulation: Solver* ⇨ *Setup Solver*). Before starting the transient thermal solver a valid *Simulation duration* time must be entered:



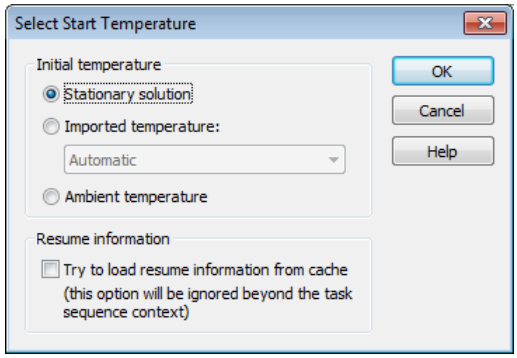
Most source types can be weighted with a previously defined excitation function, when pressing the *Excitations* button:



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 Δt can be defined for each source. The resulting time dependent excitation f is the product of the source value v (e.g. the temperature) and the (possibly shifted) assigned signal s:

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

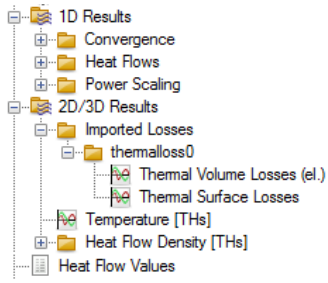
The initial temperature distribution can be defined in the Select Start Temperature dialog, which can be called by pressing the *Start temperature: Settings* button. The default setting is to solve the stationary thermal problem with the initial values of sources defined. Besides, it is possible to import a temperature distribution from an external thermal solution as well as assign the ambient temperature to the whole solution domain.



The solver parameters dialog box also allows changing the ambient temperature in the currently active unit. Moreover the accuracy settings are accessible via the *Accuracy* button and can be edited in case simulation speed or accuracy is not sufficient. For further details please refer to the online help.

Result Types

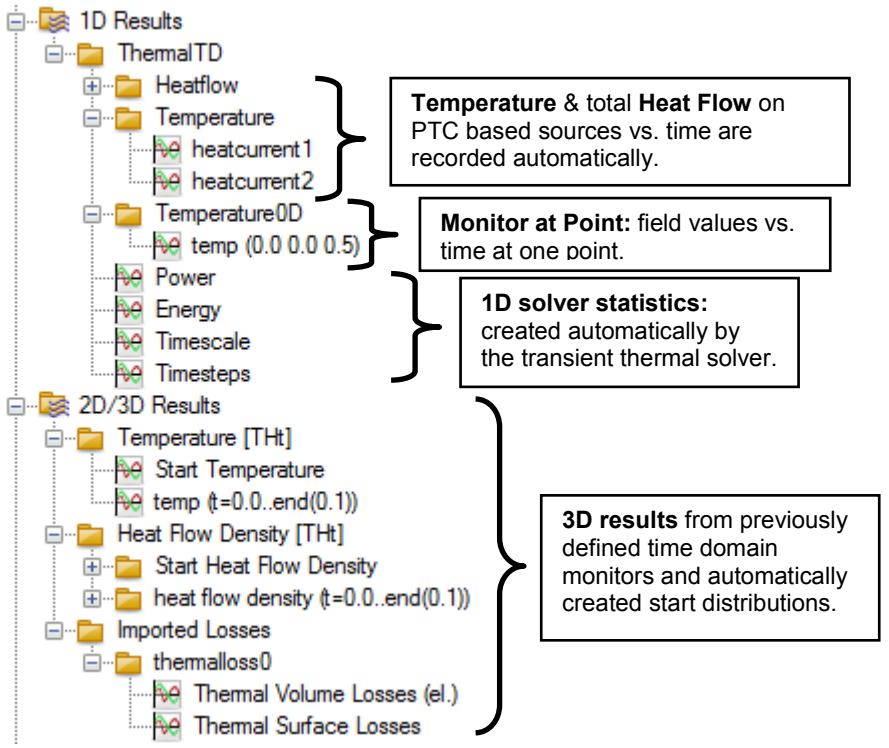
After a **stationary thermal simulation** run has been completed successfully, new result entries appear in the navigation tree:



The directory *1D Results* contains the convergence curve, heat flow values for the heat sources as well as power scaling values for imported fields.

In the directory *2D/3D Results*, beside the scalar temperature field the heat flow density can be seen, which is a vector field showing the heat flow inside thermal conductive materials. Moreover, a text file is written where the total heat flow for every source is listed. In case field losses were imported, further information like interpolated loss distributions as well as the scaling factor is presented.

The **transient thermal solver** creates a different output in the navigation tree:



If time domain temperature monitors have been defined for the transient thermal solver, the associated results will be listed under *2D/3D Results* as well. In addition, a couple of time signals are added to the 1D Results section:

- **ThermalTD / Energy** describes the total amount of energy in the computation domain vs. time.
- **ThermalTD / Timesteps** carries information about the time-step-width vs. computation step of the adaptive time-stepping scheme.
- **ThermalTD / Timescale** shows how the simulated time evolves vs. computation steps.
- **ThermalTD / Power** shows the total amount of power entering/leaving the thermal conductive regions.

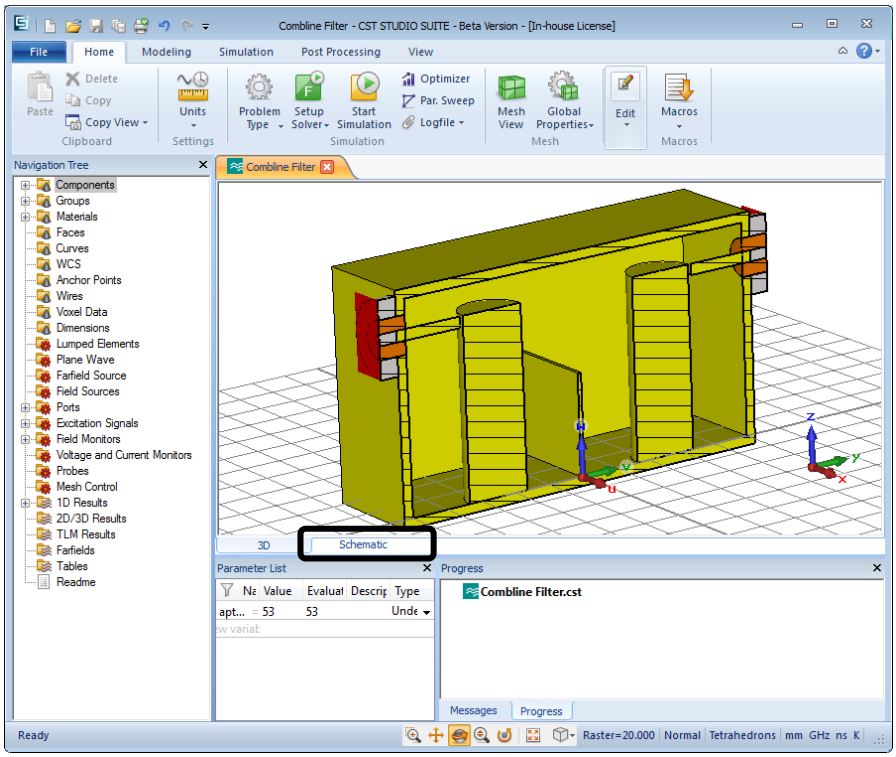
These 1D signals can be updated during the simulation process by selecting the tree item and pressing *1D Plot: Plot Properties ⇨ Update Results* or the F5 key.

Chapter 4 — Workflows for Coupled Simulations

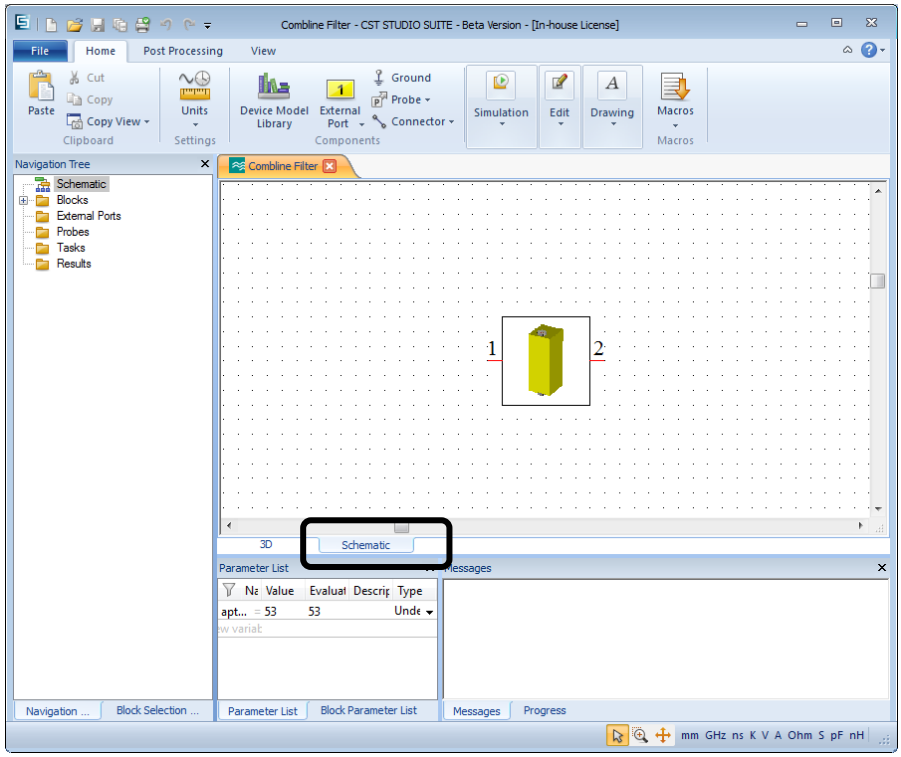
Coupled simulations are the main application field for CST MPHYSICS STUDIO. The new parametric multi-physics workflow simplifies the management of coupled simulation projects, which share the same model geometry (called the Master Model). Changes in the Master Model are directly transferred to the subprojects. Besides, this workflow supports the definition of global parameters, which are shared between the subprojects, as well as the usage of parameter sweeps or optimization sequences.

The typical workflow is demonstrated with an EM-Thermal coupling:

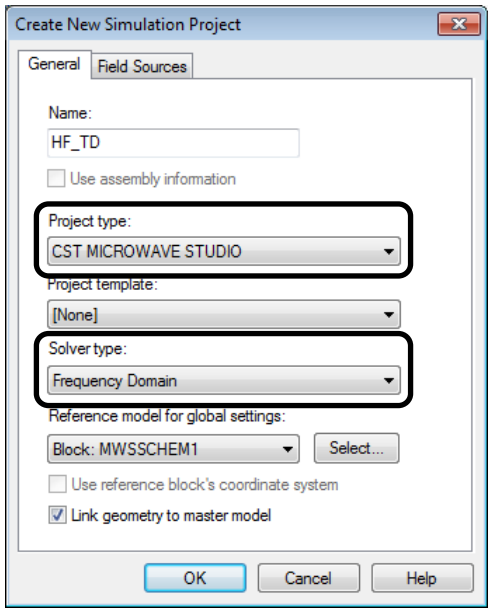
1. Create a new (or open an existing) EM- or MW-Studio project, where the model geometry, material properties and global parameters are defined.



2. Switch to the schematic view by selecting the corresponding schematic tab under the main window. All MWS-, EMS- and PCB- projects contain this view.

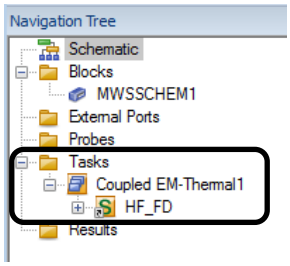


3. Start the automatic creation of a coupled electromagnetic/thermal computation by selecting *Home: Simulation* ⇨ *Simulation Project* ⇨ *Multiphysics (EM-Thermal)*. An invitation will be displayed to create an EM simulation project.
4. Press *Simulation Project: Create Project* ⇨ *Create Simulation Project*. A dialog box appears, where you can assign a unique name for the new simulation project, select the project type, project template, solver type as well as the reference model for global settings:

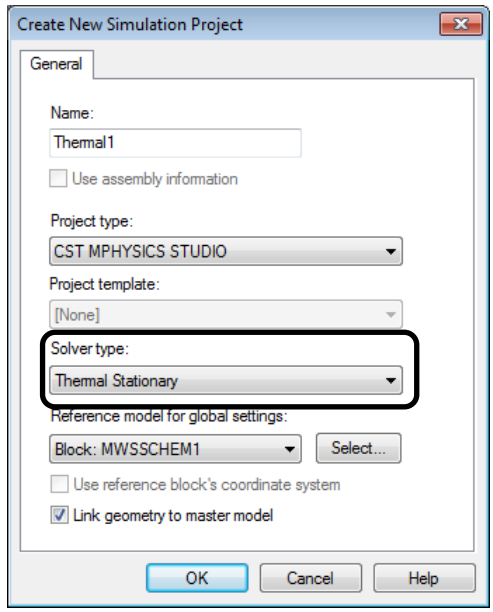


The settings from the master model can be inherited by selecting its schematic block as the reference model.

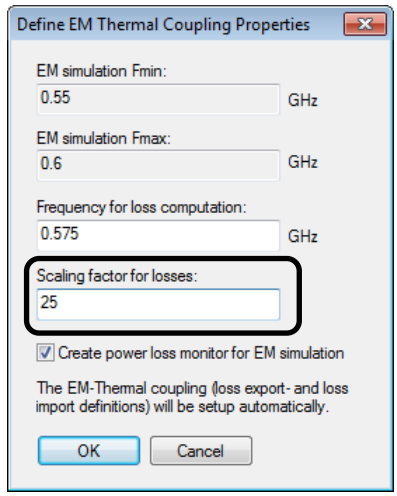
5. After you click the *OK* button, a new simulation project is created and added to the Tasks folder in the Navigation Tree.



6. Going back to the Schematic page of the master project, press *Simulation Project: Create Project ⇨ Create Simulation Project* once again. Now a thermal project must be created. In the appearing dialog box, the project type (CST MPHYSICS STUDIO) is already chosen, so only select the thermal solver type (transient or stationary). After pressing OK, a thermal simulation task is created.

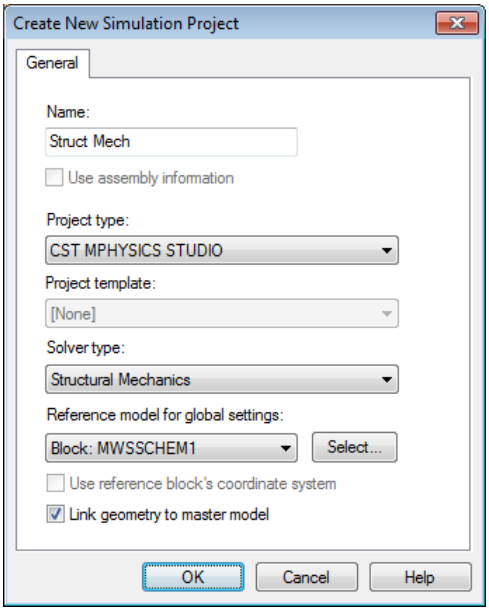


7. In the next step, you are invited to define the frequency at which the thermal losses should be computed and exported, as well as the scaling factor the thermal solver should apply to them. Please assign the value 25 to the latter parameter:

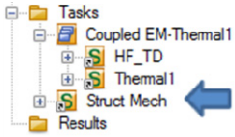


8. After the OK button has been pressed, the corresponding monitors and field imports are configured automatically in both simulation projects. Now you may switch to the thermal project and configure necessary thermal sources, boundary conditions and calculation parameters.
9. Switch back to the master project and press the button *Home: Simulation ⇨ Update* . At first, the electromagnetic calculation will be started. Next, the losses will be computed. Finally, these losses will be imported into the thermal project, and the thermal calculation will be performed.
10. In the next step, we shall use the temperature distribution calculated in the thermal project to simulate the thermal expansion of the system. To do this, a new

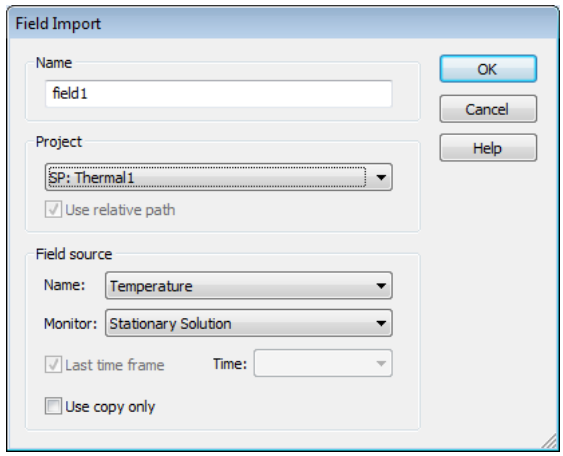
simulation project must be created for structural mechanics simulation. Switch to the *Schematic* view of the master project and press *Home: Simulation* ⇨ *Simulation Project* ⇨ *All Blocks as 3D Model* in the toolbar. Select CST MPHYSICS STUDIO for the *Project type* and Structural Mechanics for the *Solver type*.



11. Press *OK*. A new simulation task has been created:

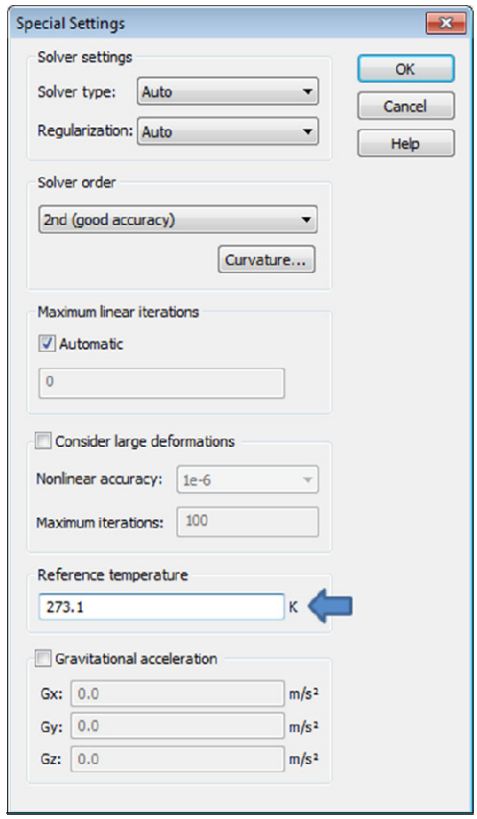


12. Switch to the newly created simulation task and define necessary displacement boundaries to fix the location of the model (like it has been explained in Chapter 2).
13. Now you'll have to import the temperature distribution from the thermal simulation project. To do this, select *Simulation: Sources* ⇨ *Field Import* in the structural mechanics project. In the opening Field Import dialog, select *SP: Thermal1* in the dropdown list *Project*. The field source *Temperature* and monitor *Stationary Solution* will be selected automatically:

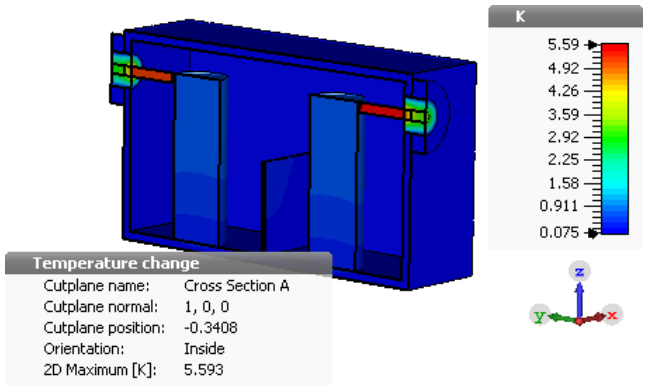


If this did not happen, it means that the temperature solution is missing. Please re-run the thermal stationary solver in the thermal project.

14. Before starting the mechanical calculation, please make sure that the reference temperature in the mechanical solver settings (*Simulation: Solver ⇨ Setup Solver*  *⇨ [Specials…]*) is equal to the ambient temperature in the thermal solver parameters of the thermal project (*Simulation: Solver⇨ Setup Solver* ). Basically it is the temperature change which causes thermal expansion, thus a wrong initial (reference) temperature would cause wrong results.



15. Now run the structural mechanics solver as it has been described in Chapter 2. After the calculation is finished, the deformation of the model, stresses and strains may be investigated. Besides, the distribution of temperature change can be found in *NT: 2D/3D Results⇨ Temperature change*, which may help to control the quality of the temperature import and correctness of settings.



Of course, this short introduction does not cover all details about the possibilities of coupling between various CST STUDIO SUITE projects. For more information, please refer to online help tutorials.

Chapter 5 — Finding Further Information

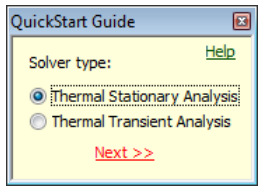
After carefully reading this manual, you will already have some idea of how to use CST MPHYSICS STUDIO efficiently for your own problems. However, when you are creating your own first models, a lot of questions will arise. In this chapter we give you a short overview of the available documentation.

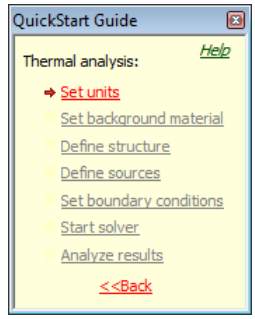
The Quick Start Guide

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

The QuickStart Guide is opened automatically on each project start if the checkbox *File: Options ⇨ Preferences ⇨ Open QuickStart Guide* is checked. Alternatively, you may start this assistant at any time by selecting *QuickStart Guide* from the dropdown list of 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 *Help* ⇨ *Help Contents* from the menu bar. 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 more detailed explanations about the usage of the CST MPHYSICS 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 MPHYSICS 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

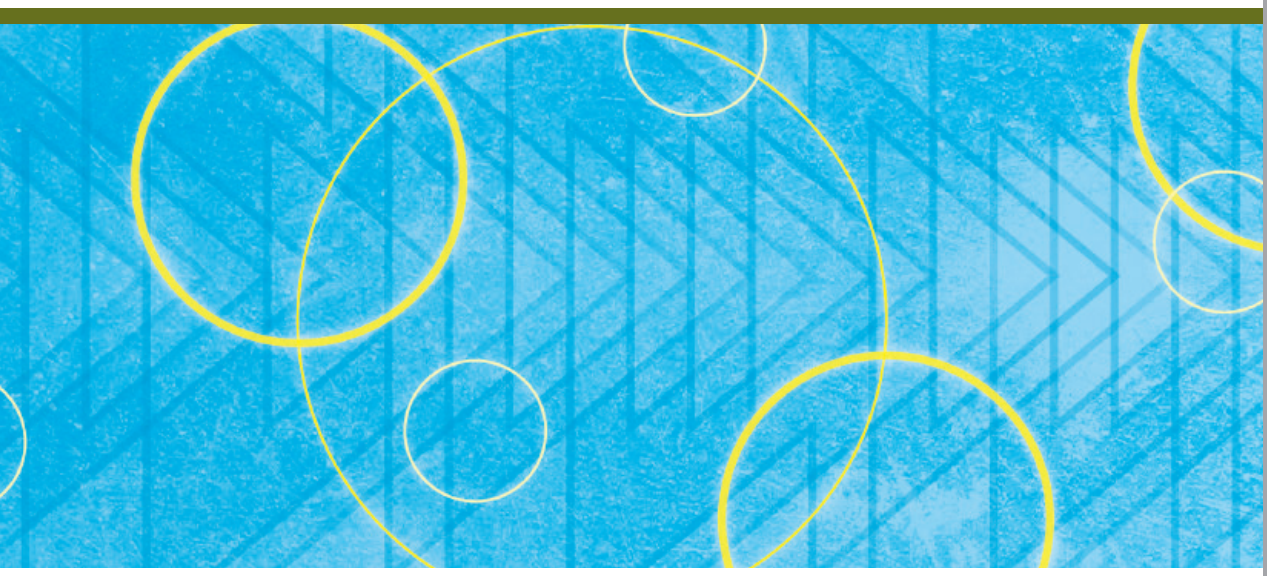
After you have taken your first steps to solving your own applications within CST MPHYSICS STUDIO, please use the *File⇨Archive As* function to create an archive containing all relevant files. This archive should then be sent to the technical support team. Even if you have successfully obtained a solution,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

The preferred option to contact technical support is to submit a 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 ⇨ Support Tickets*.

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

History of Changes

An overview of all new main features of the release can be obtained by selecting the *Spotlight CST STUDIO SUITE 2015* page from the online help system (*File: Help ⇨ CST STUDIO SUITE – Help*). A detailed *History of Changes* can also be accessed through the *Spotlight* page in the Online Help. The *Changes in the Service Packs* page in the same location in addition describes 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