CST Studio Suite

Thermal and Mechanical Simulation

3DEXPERIENCE®

Workflow &
Solver Overview

Version 2020.0 - 8/16/2019

Copyright

Trademarks

3DS.com/SIMULIA

Table of contents

Chapter 1 – Introduction

Welcome

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

Please refer to the *CST Studio Suite - Getting Started* manual first. The following explanations assume that you have already installed the software and familiarized yourself with the basic concepts of the user interface.

How to Get Started Quickly

We recommend that you proceed as follows:

1. Read the *CST Studio Suite - Getting Started* manual.
2. Work through this document carefully. It provides all the basic information necessary to understand the advanced documentation.
3. Look at the examples provided in the Component Library (*File: Component Library ⇨ Examples*). Especially the examples which are tagged as *Tutorial* provide detailed information of a specific simulation workflow. Press the *Help* button of the individual component to get to the help page of this component. Please note that all these examples are designed to give you a basic insight into a particular application domain. Real-world applications are typically much more complex and harder to understand if you are not familiar with the basic concepts.
4. Start with your own first example. Choose a reasonably simple example which will allow you to quickly become familiar with the software.
5. After you have worked through your first example, contact technical support for hints on possible improvements to achieve even more efficient usage of the software.

What is CST MPhysics Studio?

CST MPhysics Studio is a software package from the CST Studio Suite 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 Design 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 (SP 1 or later), Windows 2008 Server R2 (SP 1 or later), Windows 8.1, Windows 2012 Server R2, Windows 10 and Windows Server 2016
- 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-based, parametric solid modeling front end with excellent structure visualization
- Feature-based hybrid modeler allows quick structural changes
- Import of 3D CAD data from ACIS® SAT/SAB, CATIA®, SOLIDWORKS®, Autodesk Inventor, IGES, VDA-FS, STEP, PTC Creo, Siemens NX, Parasolid, Solid Edge, CoventorWare, Mecadtron, NASTRAN, STL or OBJ files
- Import of 2D CAD data 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®, IPC-2581 and ODB++® (e.g. Altium Designer, 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 to ACIS SAT/SAB, IGES, STEP, NASTRAN, STL, DXF, GDSII, Gerber or POV files
- Parameterization for imported CAD files
- Material database
- Structure templates for simplified problem setup

Mechanics Solver

- Temperature dependent Young's modulus
- Displacement boundary condition
- Traction boundary condition
- Thermal expansion
- Neo-Hookean material model for simulation of large deformations
- Various stress plots: von Mises, hydrostatic and tensor components
- Strain plots including visualization of the volumetric strain
- Nonlinear solver computes the Green-Lagrange and the Almansi-strain as well as the 2nd Piola-Kirchhoff and Cauchy stress tensors
- Displacement plot including visualization of deformed mesh
- Import of force densities from EM-solvers
- Export of deformed structure to CST Microwave Studio

Thermal Steady State Solver

- Isotropic and anisotropic material properties
- Bioheat material properties
- Nonlinear material properties (Bioheat properties and thermal conductivity)

- Thermal contact resistance
- Moving media
- 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, CST EM Studio or CST PCB 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 conductance matrix calculation
- Equivalent Circuit EMS/MPS/DS Co-Simulation for linear problems

Thermal Transient Solver

- Isotropic and anisotropic material properties
- Bioheat material properties
- Nonlinear material properties (Bioheat properties, thermal conductivity and heat capacity)
- Thermal contact resistance
- Moving media
- Convection for human voxel models
- Heat transfer by conduction in volumes
- Heat transfer by convection and radiation through surfaces
- Sources: fixed, initial 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, CST EM Studio or CST PCB Studio, crashed particle loss distribution from CST Particle Studio
- Adiabatic / fixed or floating temperature / open boundary conditions
- Low or high order time integration method, constant or adaptive time step width
- Network distributed computing for remote calculations
- Calculation of CEM43°C thermal dose in biological tissues

Conjugate Heat Transfer Solver

- Steady-state solver for incompressible laminar or turbulent flows
- Conjugate heat transfer between solids and fluids
- Boussinesq approximation for buoyancy force in flows
- Surface-to-surface radiation with automatic calculation of view factors
- Opening: velocity- and pressure-based inlets and outlets
- Walls: slip/no slip, isothermal and adiabatic
- Internal heat sources
- External heat sources imported from CST Microwave Studio or CST EM Studio
- Axial fan model support
- Planar and volume flow resistance model support
- Two-resistor component model support
- Thermal contact properties: resistance
- Thermal surface properties: surface emissivity and heat transfer coefficient
- Full GPU acceleration support

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

- ☐ Online visualization of intermediate 1D results during simulation
- ☐ Import and visualization of external xy-data
- ☐ Copy / paste of xy-datasets
- ☐ Fast access to parametric data via interactive tuning sliders
- ☐ Automatic saving of parametric 1D results
- ☐ Multiple 1D result view support

- ☐ Various 2D and 3D field visualization options 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 with 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.

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: Change View ⇨ Reset View (Space)*
- The project data is accessible through the navigation tree on the left side of the application's main window. An item of the navigation tree is referenced in the following way: *NT: Tree folder ⇨ Sub folder ⇨ Tree item*.
- Example: *View: Visibility ⇨ Wire Frame (Ctrl+W)* 

Your Feedback

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

Chapter 2 – Simulation Workflows

This chapter contains two workflow examples demonstrating the basic features of CST MPhysics Studio. In the first example, a very simple structural mechanics model of an accelerometer is created. This workflow describes in detail, how to generate a model geometry, assign material properties and sources, generate a mesh and run the simulation. Besides, the visualization and interpretation of structural mechanics results are discussed.

In the second workflow example, a coupled simulation is configured. First, a high frequency electromagnetic solver calculates the ohmic losses in the walls of an HF-filter. Then, these losses are imported by the Conjugate Heat Transfer solver in order to model the heating processes in the filter and its surrounding environment.

Studying these examples carefully will help you to become familiar with many standard operations that are important when performing simulations 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.

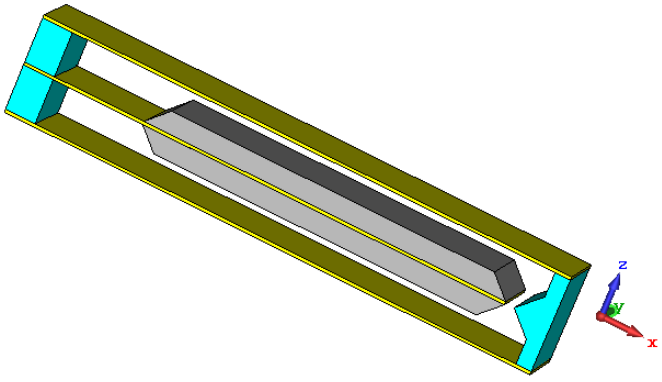
The following explanations describe the “long” way to open a particular dialog box or to launch a particular command. Whenever available, the corresponding Ribbon item will be displayed next to the command description. Because of the limited space in this manual, the shortest way to activate a particular 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.

Simulation Workflow: Structural Mechanics

In this example you will model a simple accelerometer. At first, the geometry of the 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

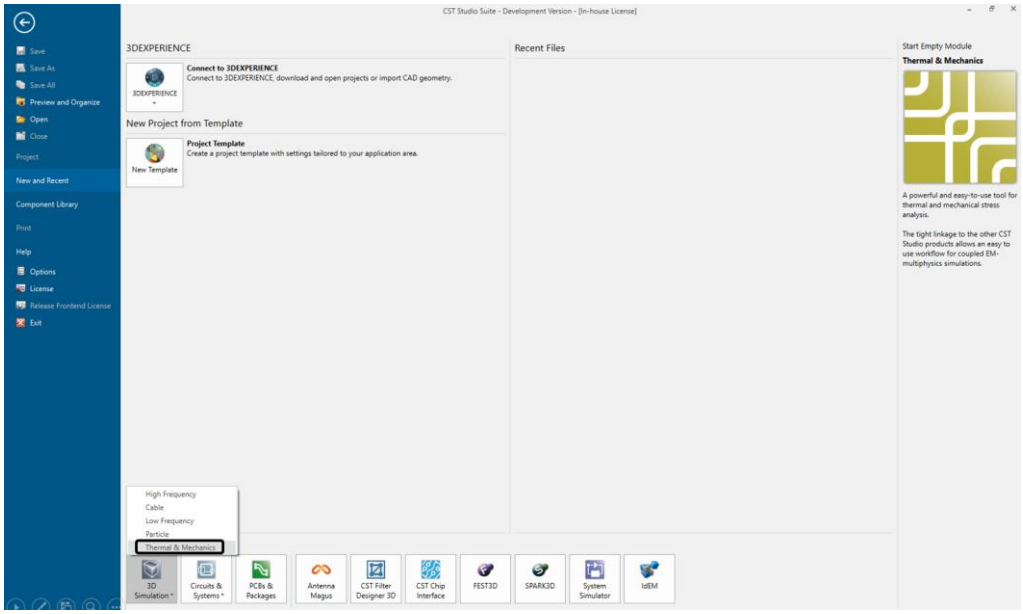
The following picture demonstrates the spatial structure of a simple accelerometer. It consists of two fixed flat conductors with a potential difference applied, and a movable conductor between them.



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 Studio Suite, please select Thermal and Mechanics from the list of installed modules:

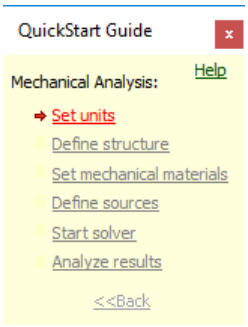


After a new CST MPhysics Studio project is created, you can switch the problem type to *Mechanics* by selecting *Home: Edit ⇨ Problem Type ⇨ Mechanics* .

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:



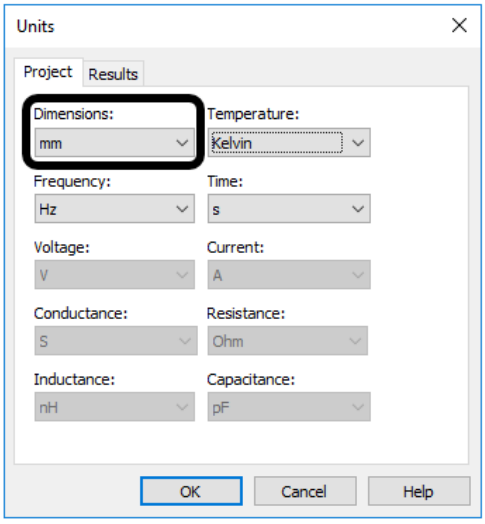
The red arrow always indicates the next step necessary for your problem definition. You do not need to process the steps in this order, but we recommend that you follow this guide at the beginning in order to ensure all necessary steps have been completed.

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:



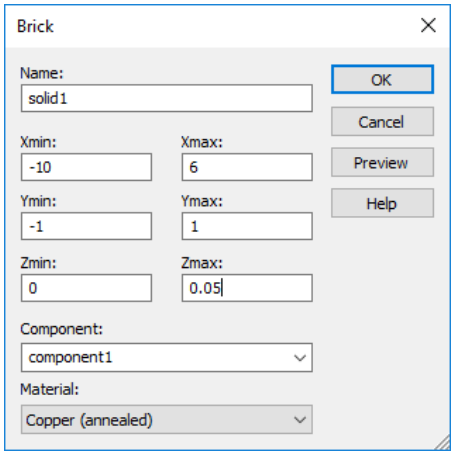
Model the Structure

The first step is to create a brick.

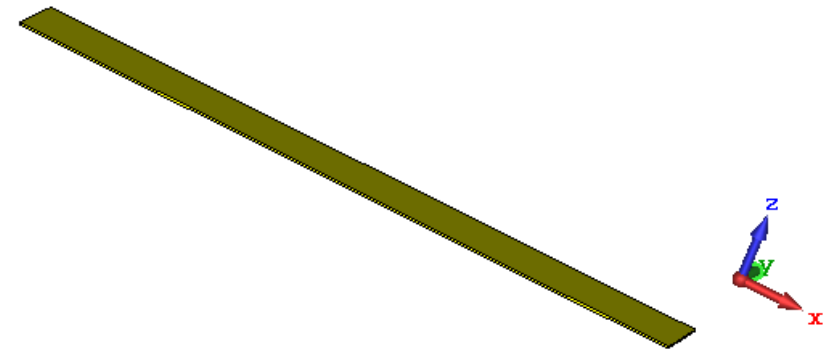
1. Select the brick creation tool from the main menu: *Modeling: Shapes* ⇨ *Brick* .
2. Press the *Escape* key in order to open the dialog box.
3. Fill up the brick size fields as it is shown in the table below.

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

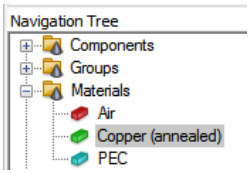
4. In order to select the material, click on the corresponding combo box and select *Copper (annealed)*. This material is predefined for CST MPhysics Studio projects.



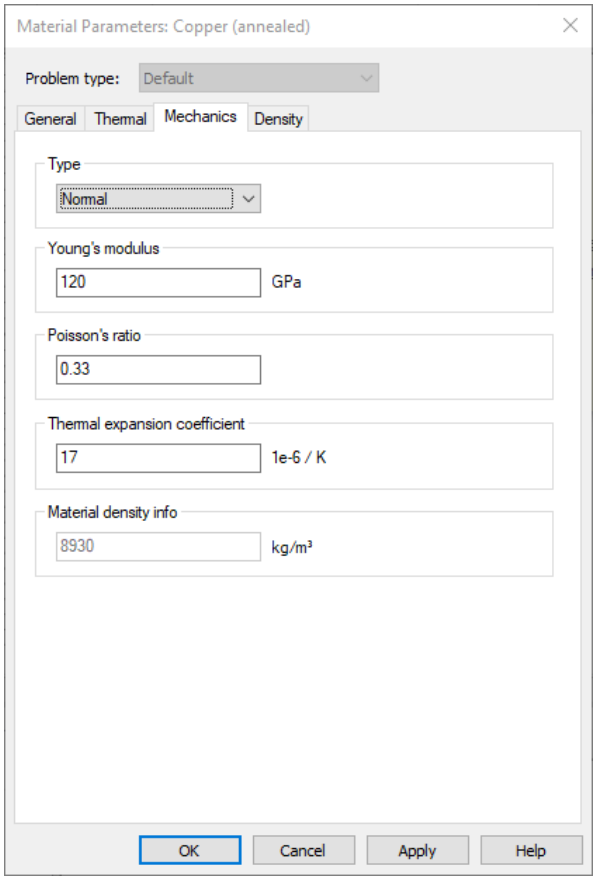
5. Now click the *OK* button. A new brick has been created:



1. 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.



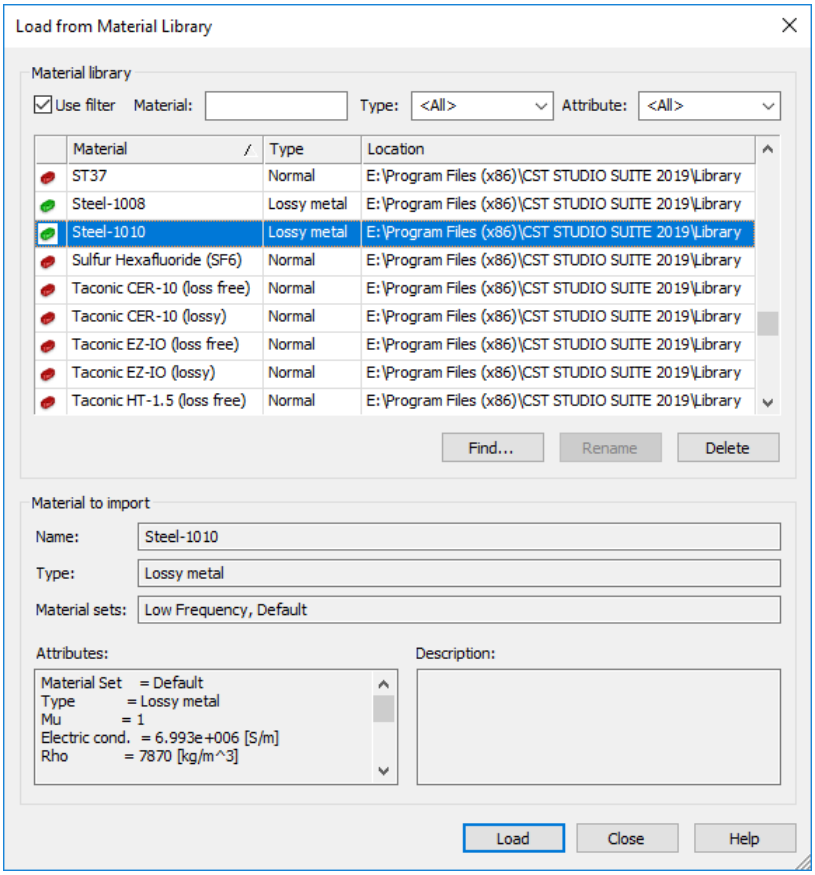
In this tab you can change the mechanical properties of the selected material. These are the three most important mechanical properties:

- *Young's modulus* defines the stiffness of the 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.

2. Now press *Cancel* and start creating a new brick (*Modeling: Shapes* ⇨ *Brick*) with the following size (please do not press *OK* yet):

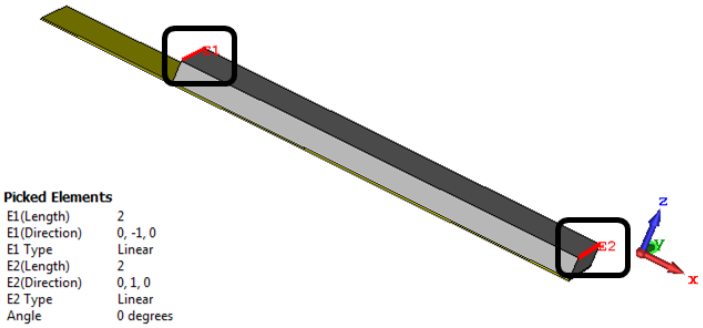
Xmin	-6	**Xmax**	6
Ymin	-1	**Ymax**	1
Zmin	0.05	**Zmax**	0.7

3. In order to change the material for the new solid, select *[Load from Material Library…]* in the *Material* combo box. The dialog box *Load from Material Library* appears. Select the material Steel-1010 and press the button *Load*.



4. Now press the button *OK* in the *Brick* dialog box. A new brick consisting of Steel-1010 is created.

5. 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:

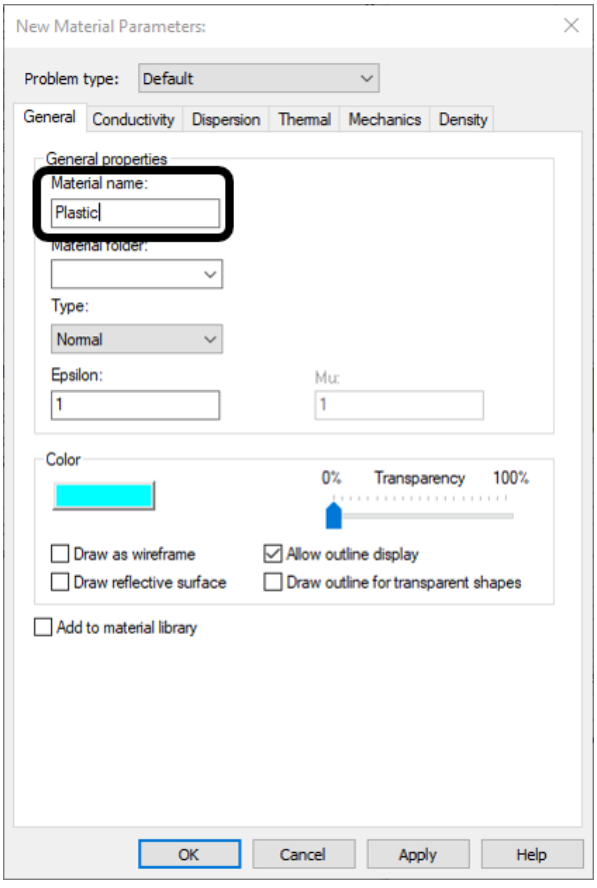


6. Select *Modeling: Tools ⇨ Blend ⇨ Chamfer Edges* in order to chamfer the selected edges. Enter the chamfer width of 0.65, and keep the default angle of 45° in the appearing dialog box and click the *OK* button.

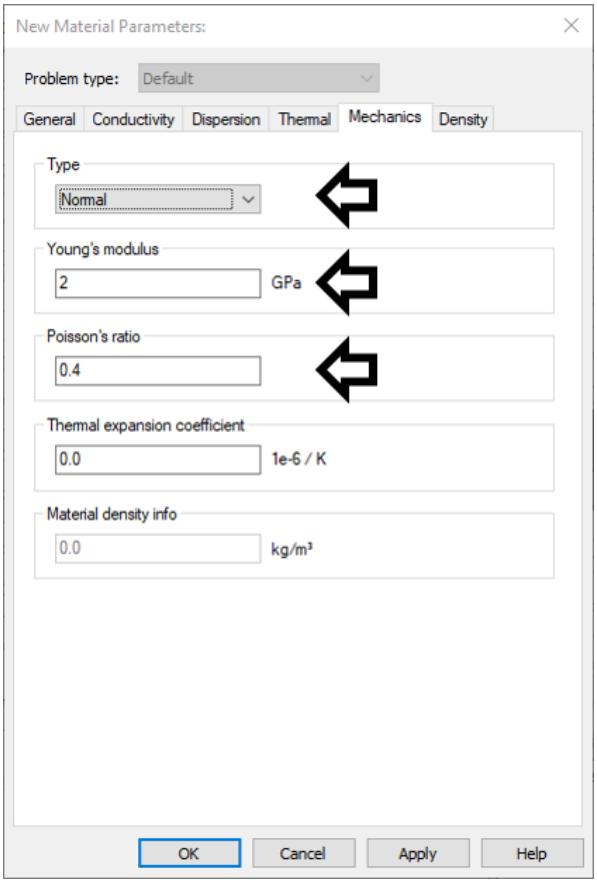
7. Again, open the Brick dialog and enter the following values:

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

For the new brick a new material should be created. Select *[New Material…]* in the *Material* combo box. The New Material Parameters dialog is shown. In the General tab, enter *Plastic* for the *Material name*.

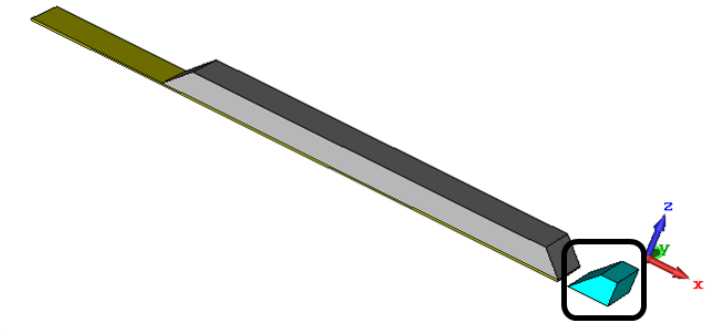


After that, switch to the Mechanics tab in this dialog, select *Normal* for material *Type*, set the *Young's modulus* to 2 GPa and the *Poisson's ratio* to 0.4.



Confirm your settings with *OK*.

8. Pick and chamfer one of the upper edges with the chamfer width of 0.7, and keep the default angle of 45° (*Modeling: Tools ⇨ Blend ⇨ Chamfer Edges*) in order to obtain the following structure:



9. Create the following bricks:
   - One of *Plastic* with the following size:

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

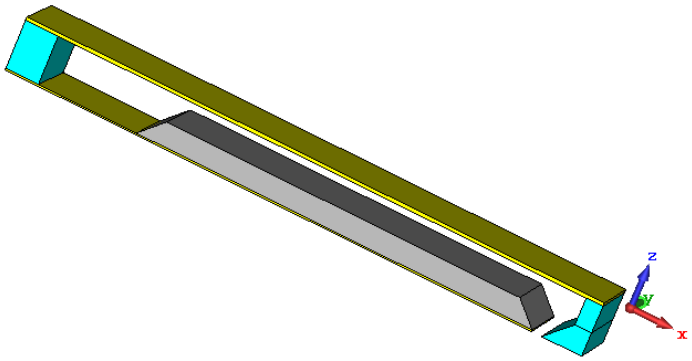
   - Another one of *Plastic* with the following size:

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

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

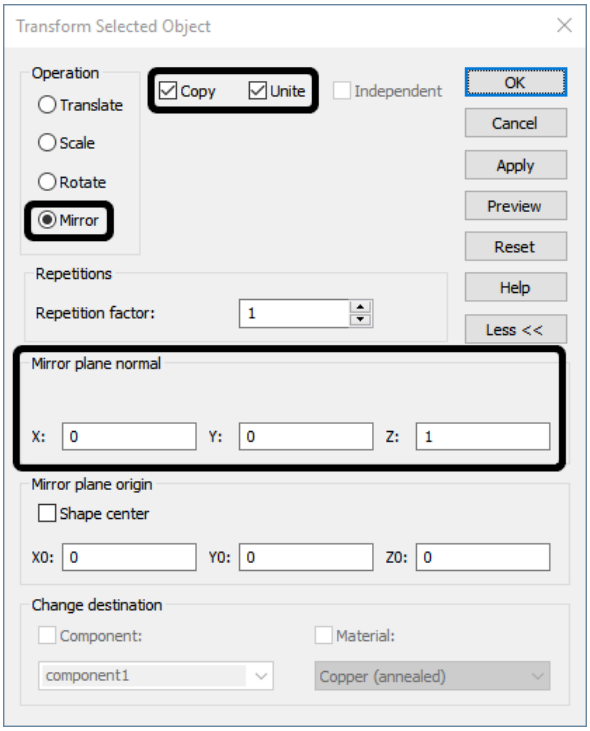
Xmin	-10	**Xmax**	7.5
Ymin	-1	**Ymax**	1
Zmin	1.5	**Zmax**	1.6

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

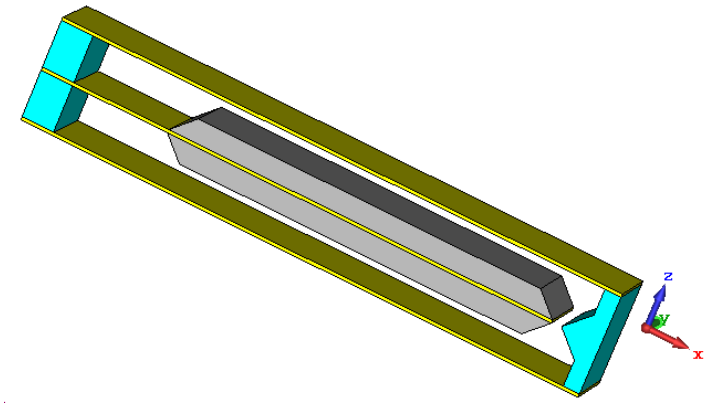


10. In Navigation Tree to the left of the main document window, open the item *Components* and select *component1*. Afterwards activate *Modeling: Tools ⇨ Transform* .

11. 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:



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

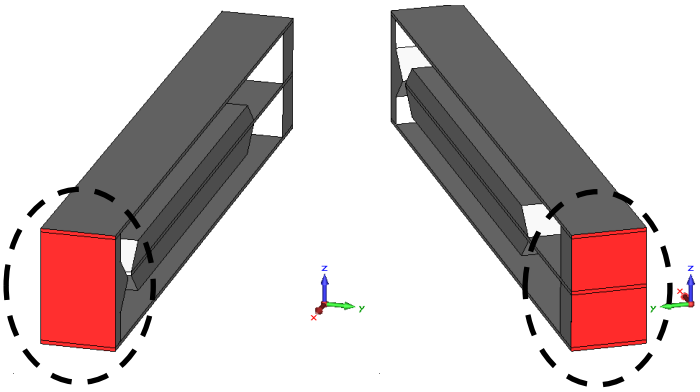


Traction and Displacement Boundaries

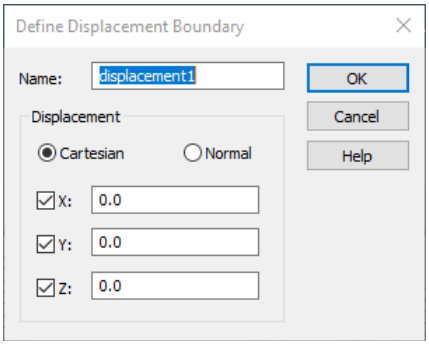
After the spatial structure has been built, the next step is to define the displacement and traction boundaries. Displacement boundaries refer to the surfaces of the model which have been shifted by a certain distance in a certain direction. To fix a surface at its initial position it is also possible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

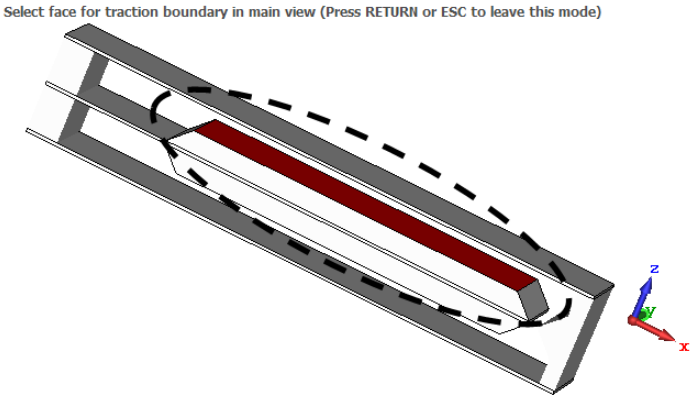
1. Press the toolbar button *Simulation: Boundaries ⇨ Displacement Boundary* .
2. Select the side faces of the model, as shown in the picture below (you have to select five faces at x-min and 3 faces at x-max):



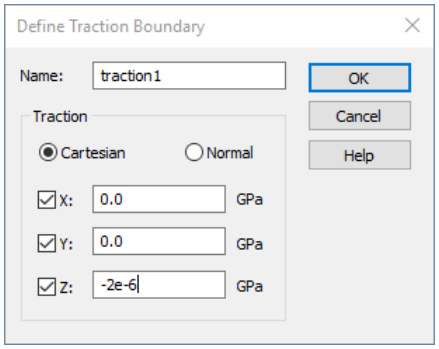
After pressing the *Return* key, the dialog box *Define Displacement Boundary* will appear:



3. Keep the zero values for all the components of the displacement vector and press the *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:



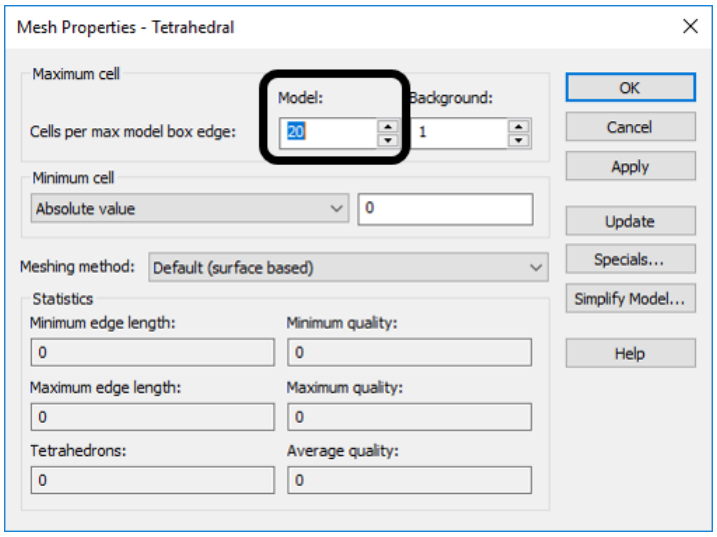
After pressing the *Return* key, the dialog box *Define Traction Boundary* will appear:



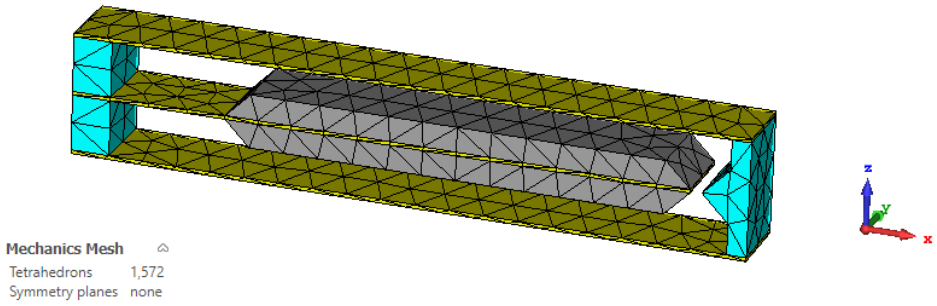
6. Put the value of -2e-6 GPa as the Z-coordinate of the traction vector. This means that the pressure of 2 kPa is applied towards the negative direction of the Z-axis. This pressure would roughly correspond to the acceleration of 17*g, or 170 m/s^2, into the positive Z-direction.

Mesh Settings

The structural mechanics solver is quite sensitive 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:



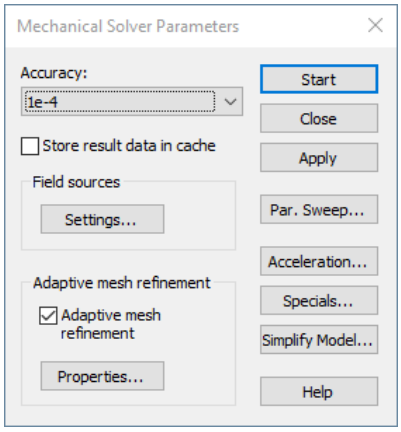
This will increase the density of generated mesh. In order to check the resulting mesh, you may press the Update button:



Press the OK button to accept the changes and close the window.

Start the Simulation

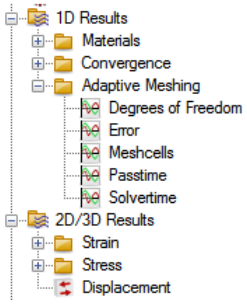
Finally, after all the settings have been made, it is time to start the mechanical solver. Press the toolbar button *Simulation: Solver ⇨ Setup Solver* . The structural mechanics solver parameter dialog box appears.



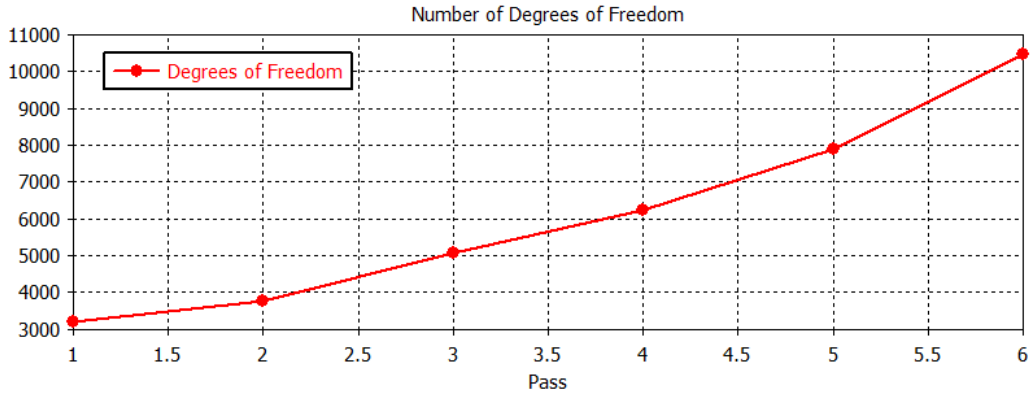
You can click the *Help* button in order to learn more about the controls in this dialog box. For now, the default settings are good enough, so just click the *Start* button. After the calculation has been started, you can control the execution of the solver in the *Progress* and *Messages* windows.

Analyze the Solution of the Tetrahedral Solver

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

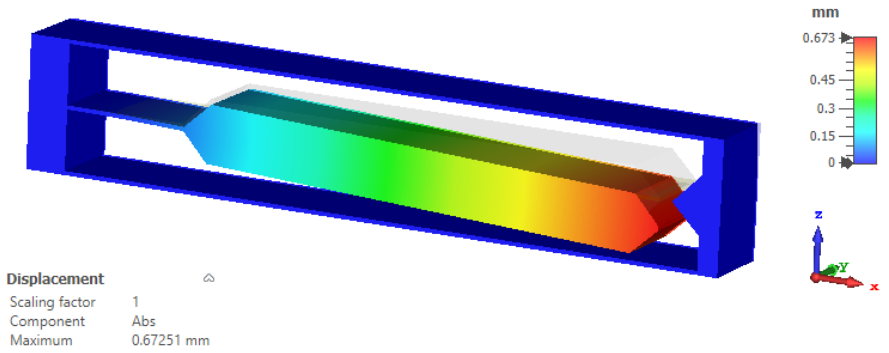


The directory *NT: 1D Results ⇨ Adaptive Meshing* contains information on the adaptive mesh refinement performed by the solver. Here you can inspect the number of cells in the mesh for each iteration step, time used by the solver to generate the solution, as well as the relative error of the solution. 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.



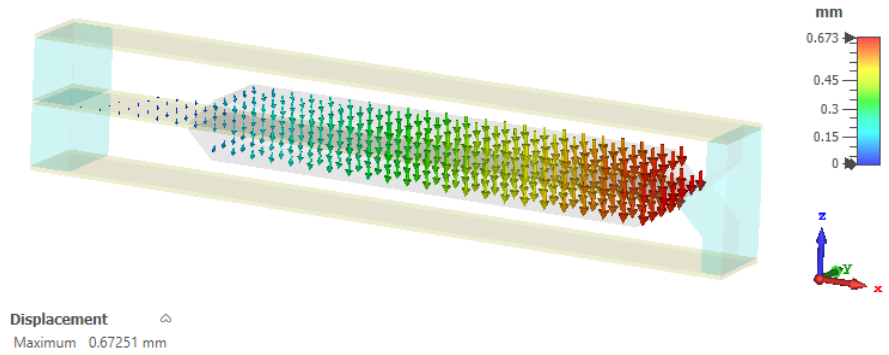
The directory *2D/3D Results* contains the distributions of displacement, strain and stress within the solution domain. If a temperature distribution has been imported from the thermal solver, it will be mapped to the tetrahedral mesh and will be available for display here as well.

A click on the item *NT: 2D/3D Results* ⇨ *Displacement* displays a deformation plot of the body deformation.



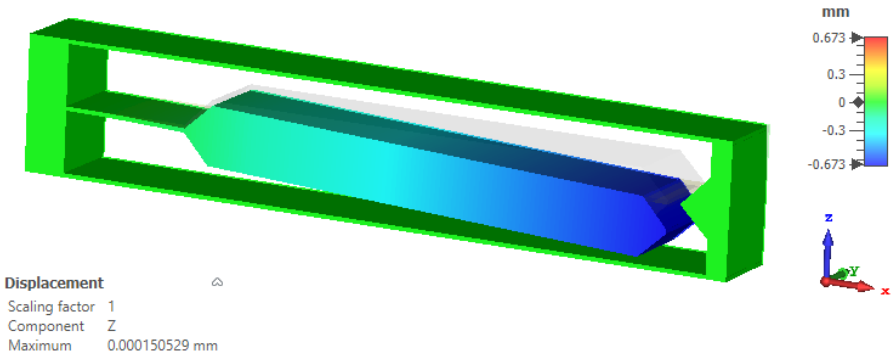
Here the original shape of the model is shown semi-transparently whereas the scalar plot of absolute displacement is shown on the solid deformed shape.

Selecting Arrows from the plot type pull-down menu in the 2D/3D Plot ribbon displays a vector plot of the body deformation, as shown in the following picture.



Selecting Contour from the plot type pull-down menu displays a scalar plot and enables the vector component pull-down menu in the 2D/3D Plot ribbon, which contains the following items:

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

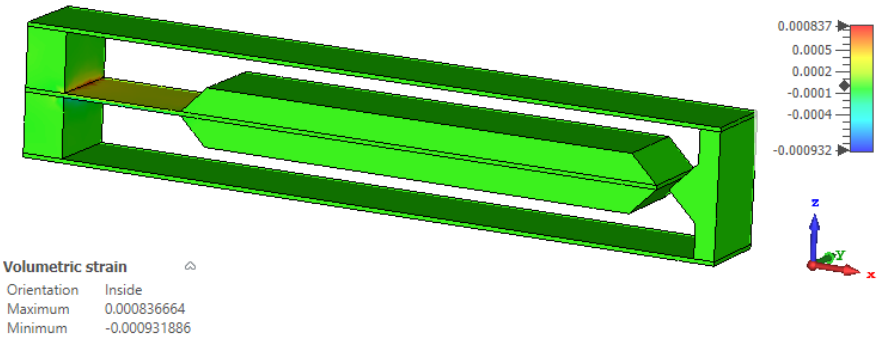


- The item *Abs* demonstrates the distribution of the absolute value of displacement within the solution domain.
- The items *Normal* and *Tangential* demonstrate the length of the corresponding projection of displacement vector onto each body surface.

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

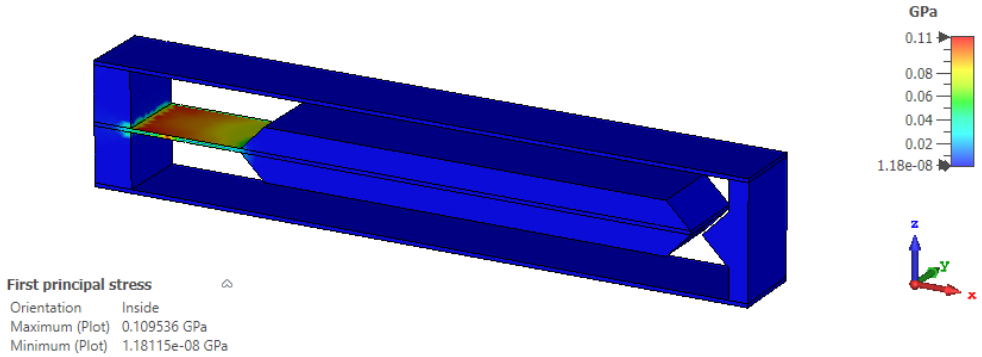
- Directory *Components* contains the components XX, YY, ZZ, XY, XZ and YZ of the strain tensor.

- 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.



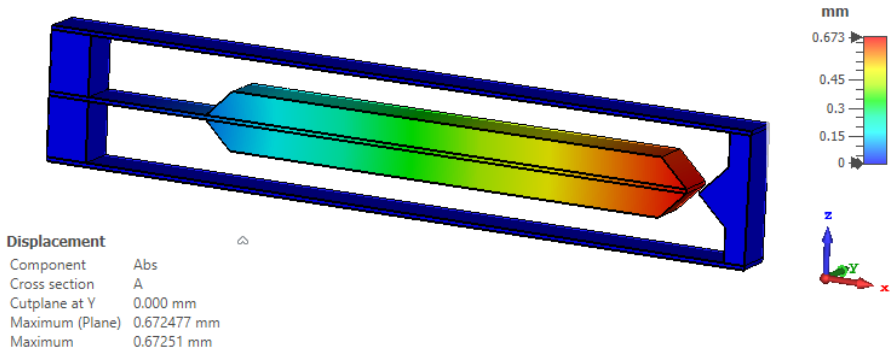
Finally, Navigation Tree item *NT: 2D/3D Results ⇨ Stress* contains the following entries:

- 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 useful feature is the visualization of computation results on a cutting plane. Select *2D/3D Plot: Sectional View ⇨ Fields on 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 select the *Arrow* plot type in the *2D/3D Plot* ribbon (of course, for a plot with vector data like *Displacement*). In this case the Fields on Plane 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.

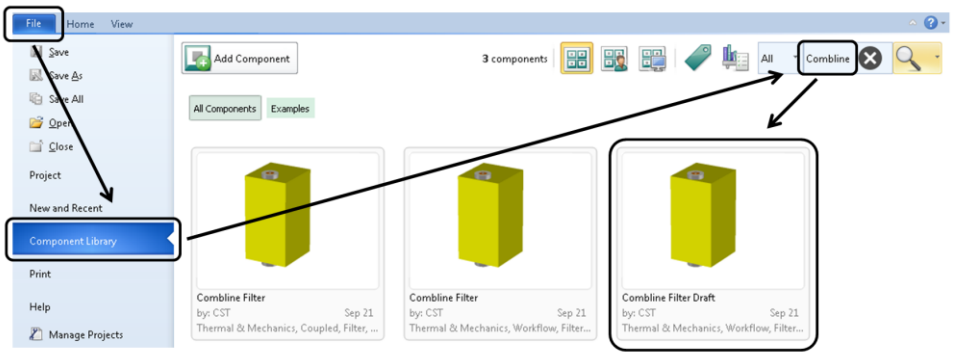
Simulation Workflow: Coupled EM-CHT Simulation

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. In addition, 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.

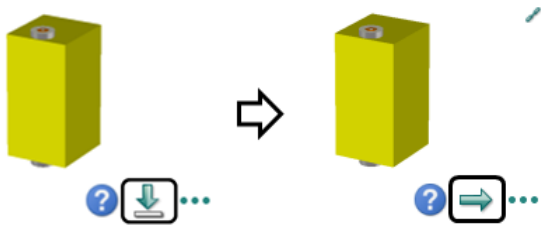
The typical workflow is demonstrated with an EM-Thermal coupled project. The simulated device consists of a filter placed on a horizontal support and surrounded by air. The EM solver is first used to perform a frequency domain analysis of the filter and to calculate the field distribution and the ohmic losses. The energy lost by the filter is transformed into heat which increases the filter temperature. The conjugate heat transfer (CHT) solver is then used to compute the temperature increase. The ohmic losses are imported as heat sources into the thermal simulation and a thermal analysis of the filter is performed by including the cooling effect of the surrounding air.

EM-Thermal Link Set-up

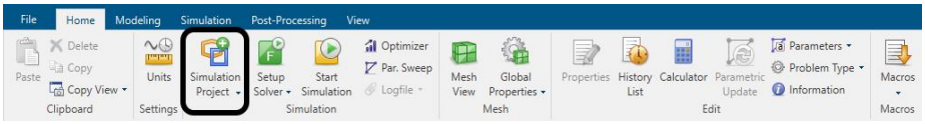
Please open the project “Combline Filter Draft” located in the Component Library. To access the example, please select the File tab, then select Component Library, type “Combline” into the search field on the top right and press Enter:



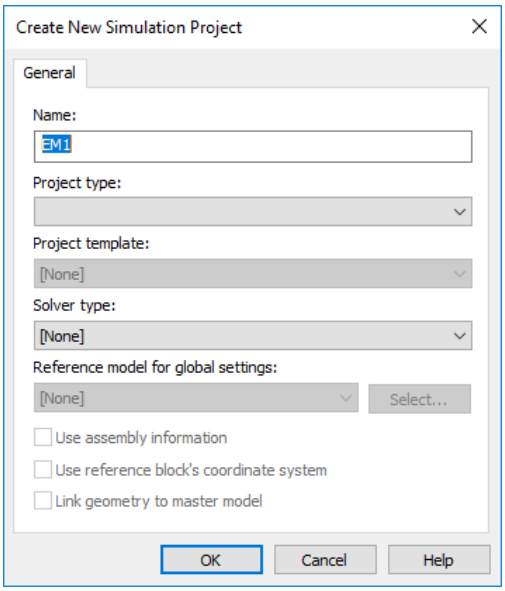
To open the project you have to download a copy first, by clicking on the Download symbol. Once this is done you are ready to open the example by clicking on the Open Project symbol.



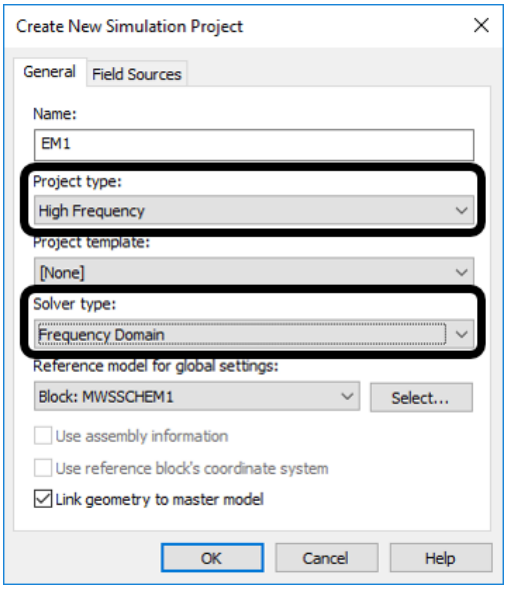
Start the automatic creation of a coupled electromagnetic/thermal computation by selecting *Home: Simulation ⇨ Simulation Project ⇨ EM-Thermal Coupling ⇨ Uni-directional*.



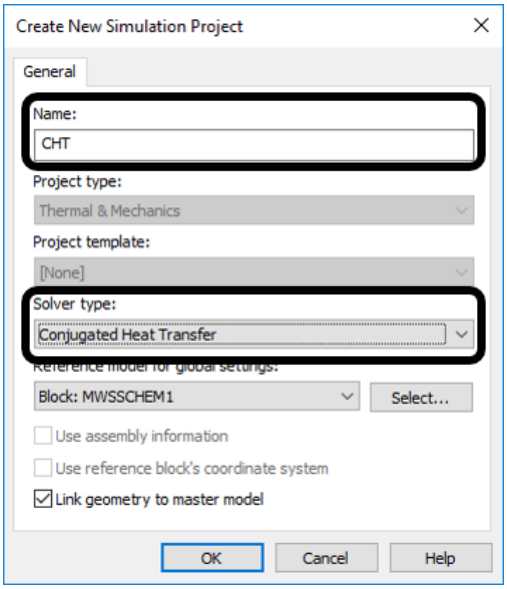
A dialog box appears to create the first part of the EM-Thermal link:



The EM simulation project is named EM1 and will be performed by the frequency domain solver of MWS. Select *High Frequency* as Project type and *Frequency Domain* as Solver type. All the settings from the master model can be inherited by selecting its schematic block as the reference model.



After you click the *OK* button, a new simulation project called EM1 is first created and added to the Tasks folder in the Navigation Tree in the Schematic view; then a dialog box appears to create the second part of the EM-Thermal link:

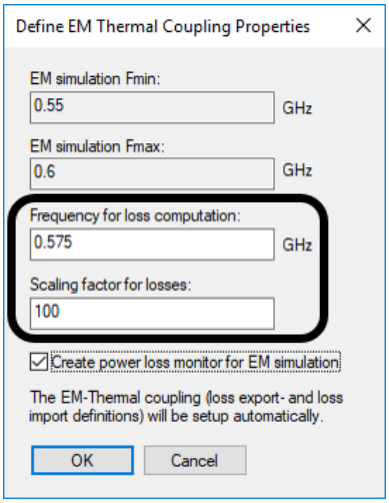


In the dialog box, the project type Thermal & Mechanics is already chosen, so only select the thermal solver type *Conjugate Heat Transfer* and rename the project to CHT. After pressing OK, a thermal simulation task using the CHT solver is created.

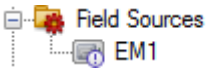
Loss Import

In the next step, you are invited to define the frequency at which the thermal losses should be computed and exported. The losses directly exported by the EM solver are by default calculated for an input peak power of 1W. The simulated device however may be operated at a different

input power therefore the exported losses must be proportionally rescaled. For an operational input power of 100W, assign the value 100 to the Scaling factor for losses entry:

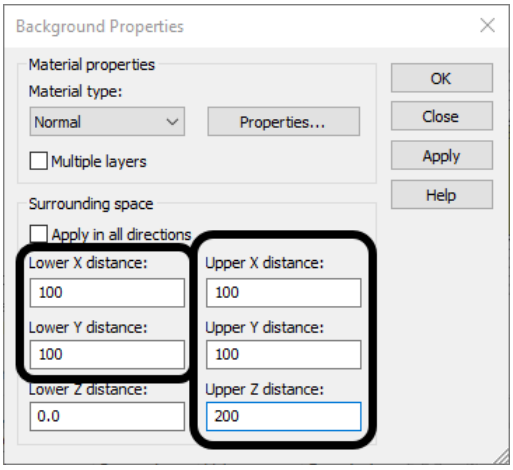


Click *OK*. The corresponding monitors and field imports are configured automatically in both simulation projects. Now you may switch to the thermal project (select the CHT project tab in the main view) and configure additional material properties, necessary thermal sources, boundary conditions and calculation parameters.
In the navigation tree a field source called EM1 (named after the name of the EM project of the EM-Thermal link) has been automatically added and configured. Edit it to reconfigure it if necessary. The exclamation mark indicates that losses are missing because the EM1 simulation has not yet been performed.



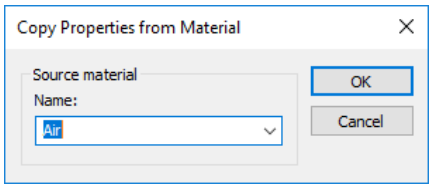
Background and Boundary Conditions

To take into account the effect of natural convection it is necessary to create some space around the device so that the airflow induced by heated device can be simulated:

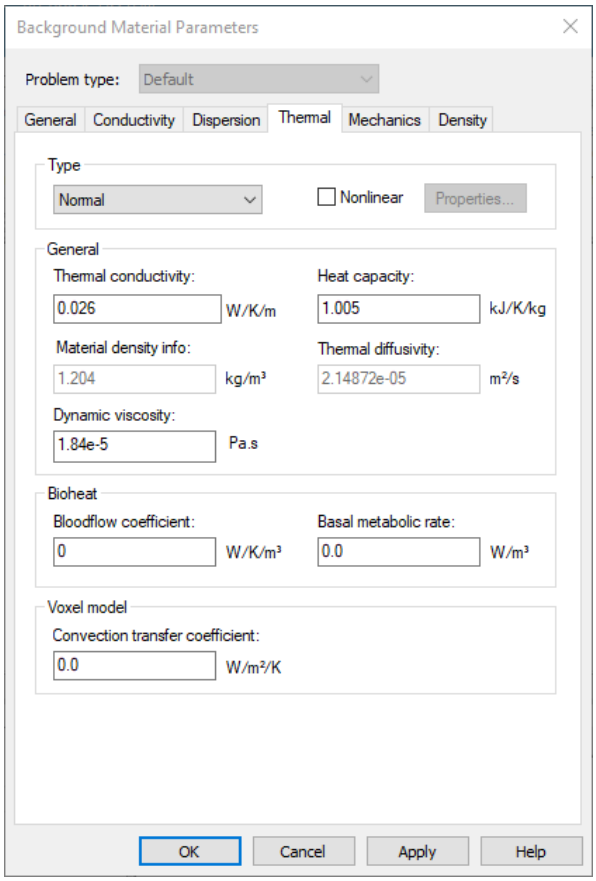


Some of the materials contained in the original model are missing thermal properties. In order for the CHT solver to work properly, the density, heat capacity and thermal conductivity of the background and of all the solids must be defined (>0). In addition, the dynamic viscosity of the background material must be specified. To include the contribution of radiation from a solid, the emissivity has to be specified as well. By default, radiation calculation is not activated but can be enabled in the solver parameter dialog (see section Solver Parameters).

The background material properties (*Simulation: Settings ⇨ Background* ) need to be copied from the material “Air”. To do this click on the *Properties…* button and then on the *Copy Properties from Material…* button in the General tab of the material dialog.

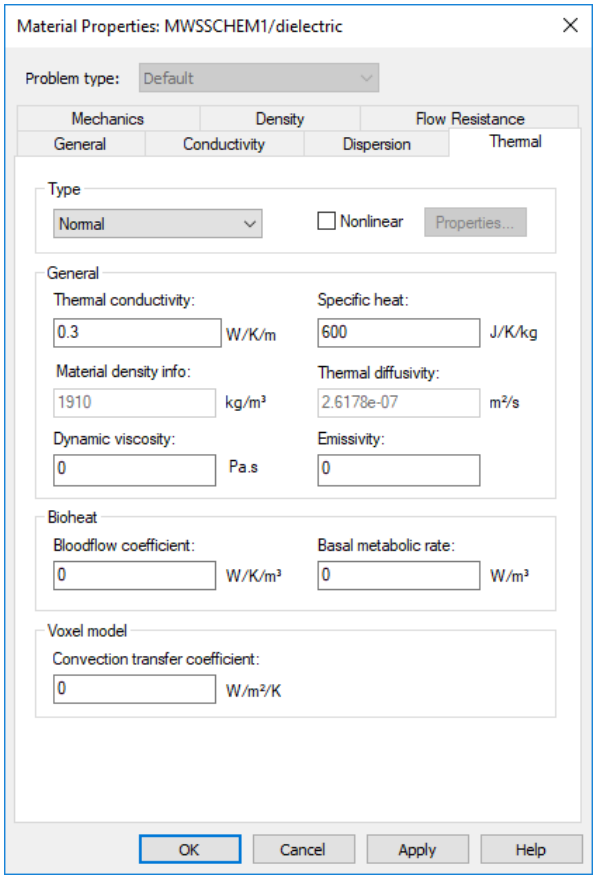


Select the Thermal tab and check the properties before closing the dialog with OK.

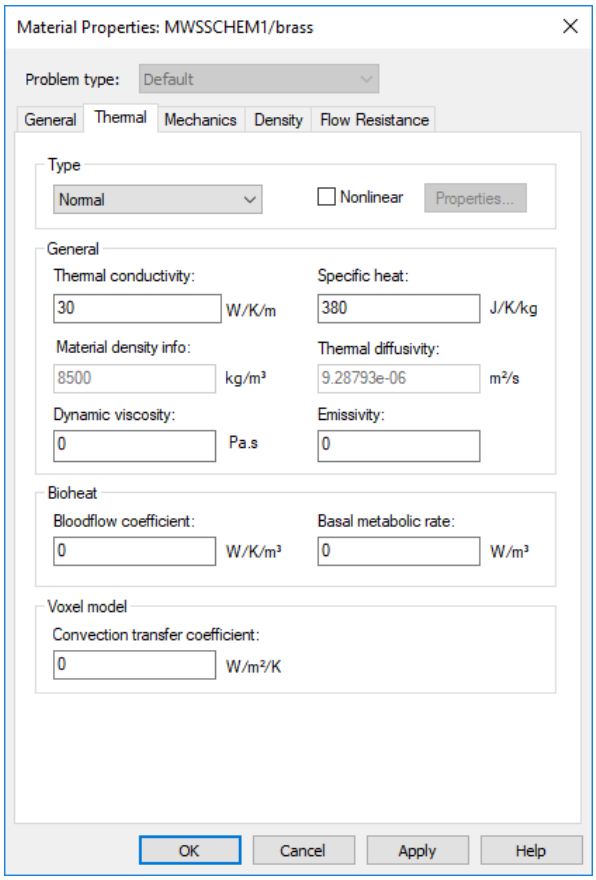


Note: solids made of material PEC will be automatically replaced in the solver by the material Copper (annealed).

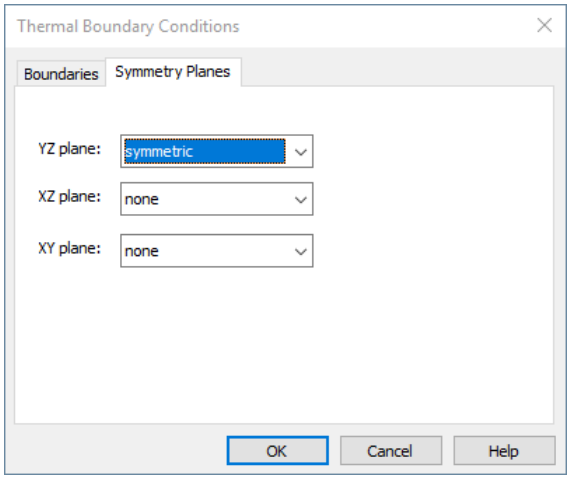
Make sure that the material named “MWSSCHEM1/dielectric” has the following thermal properties (the density value can be entered under the Density tab):



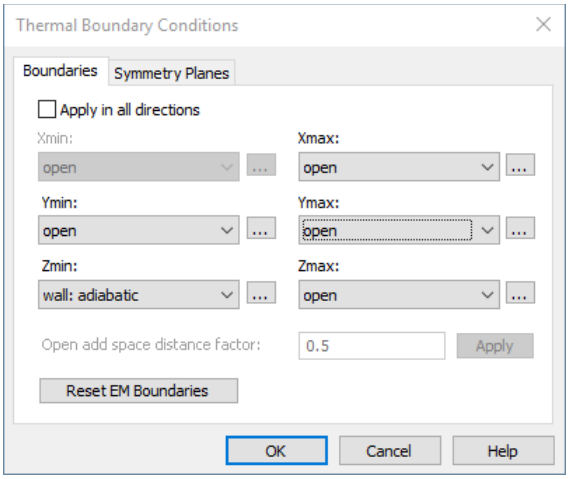
Please check the material named “MWSSCHEM1/brass” in the same manner:



Open *Simulation: Settings ⇨ Boundaries* to exploit the symmetry of the model with respect to the YZ plane and set-up the YZ symmetry boundary condition:

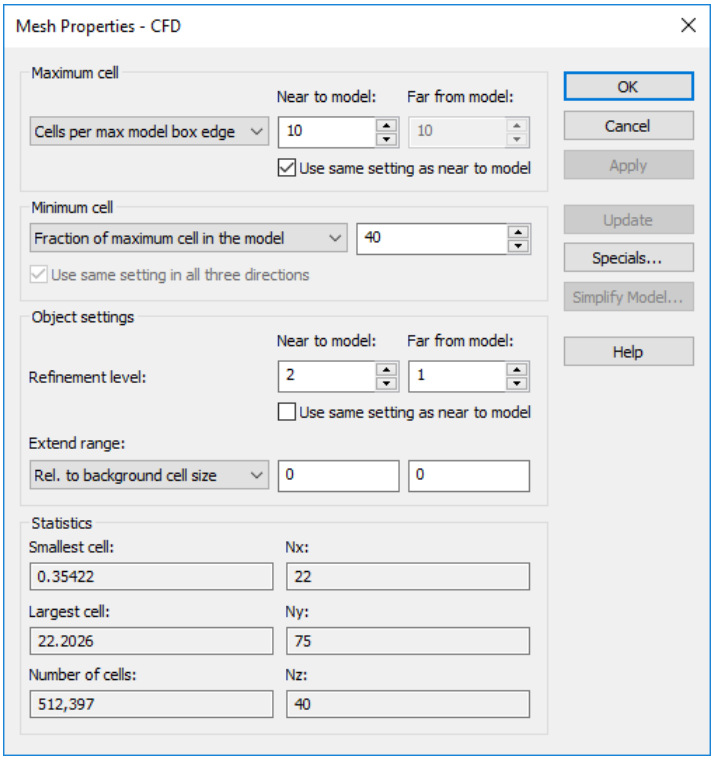


Assuming the device is positioned horizontally, the horizontal support is modeled as an adiabatic wall while the other sides of the computational domain are set to open:



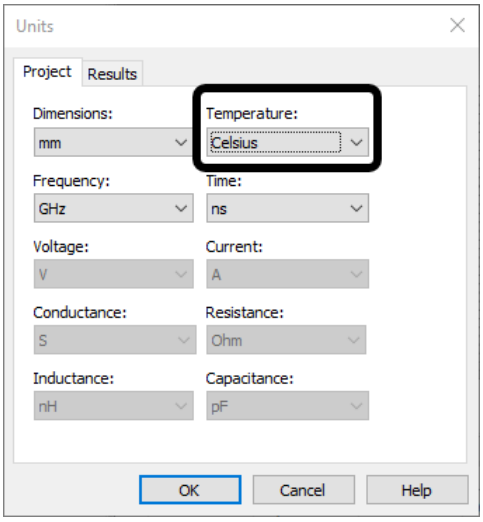
Mesh Settings

Please change the mesh type to CFD (*Simulation: Mesh ⇨ Global Properties ⇨ CFD*) and open the mesh properties dialog. Adjust the Minimum cell setting to a fraction of 40 for the maximum cell in the model.



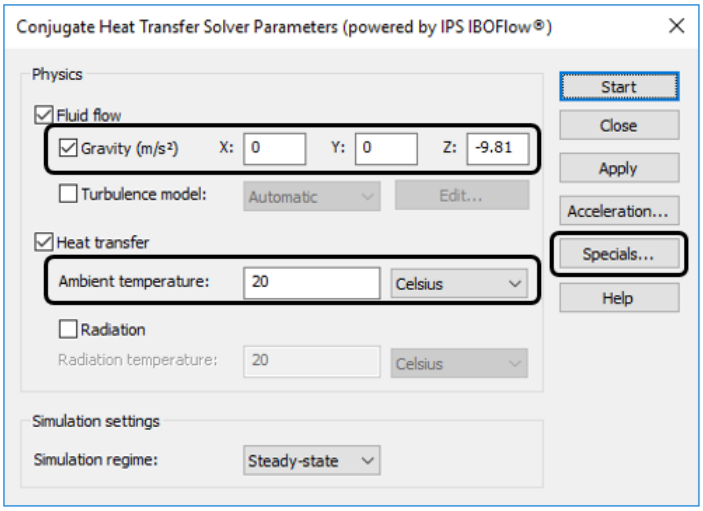
Solver Parameters

Please change the default temperature unit to *Celsius* in the project Units dialog (*Home: Settings ⇨ Units*) to have the results presented in Celsius later.

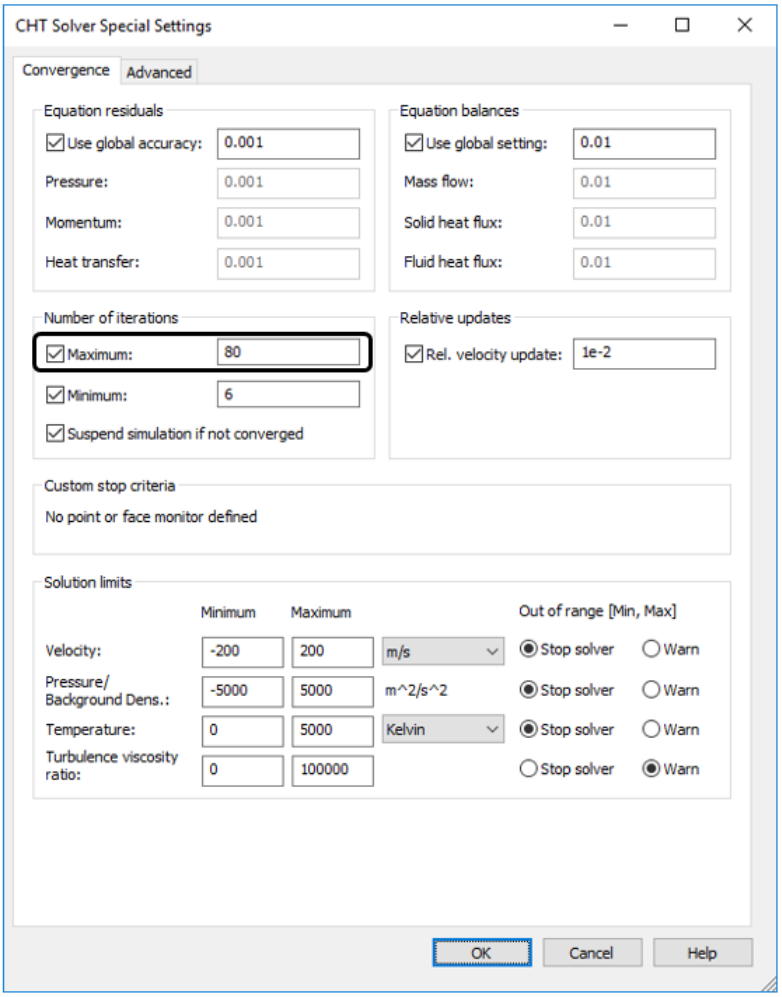


Then open the solver parameter dialog (*Simulation: Solver ⇨ Setup Solver*). Activate the *Gravity* check box to simulate the effect of natural convection. Adjust the *Ambient temperature* unit to *Celcius* and specify the ambient conditions of 20°C.

If Radiation is turned on, the radiation temperature can be used as reference temperature when the contribution of open boundary conditions to radiation is taken into account.. Please note that we perform this simulation without radiation in order to reduce the computation time for this tutorial.



Open the *CHT Solver Special Settings* dialog by pressing the button *Specials…* and limit the *Number of iterations* to a maximum of 80 in order to shorten the otherwise lengthy simulation time. Please note that the results obtained after 80 iterations are not fully converged. If more accurate results are desired change the number of iterations to automatic calculation (i.e. switch off the *Maximum* checkbox). With current settings, the overall simulation time should be less than 20 minutes.

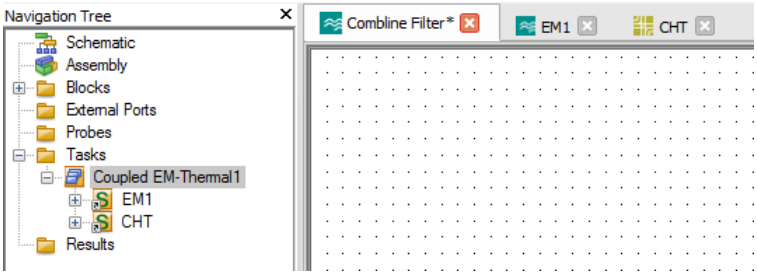


Apply all settings with the *OK* or *Apply* buttons before closing the CHT Solver Special Settings and Conjugate Heat Transfer Solver Parameters dialog box.

Coupled Run

Switch back to schematic of the master project (first tab) and therein to the *Schematic* view (select the appropriate tab at the bottom of the main view). Press the button *Home: Simulation ⇨ Update*  . At first, the EM 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.

Alternatively, right click on *NT: Tasks ⇨ Coupled EM-Thermal1* and *Update* the task.



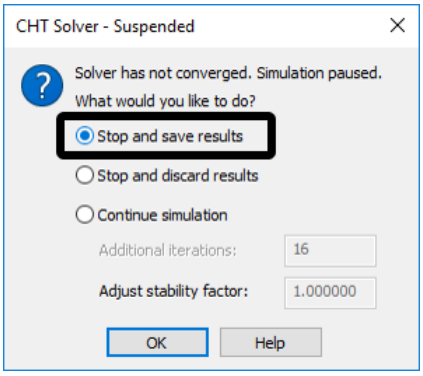
A progress bar will appear in the progress window which will update you on the task progress. You can activate this window by selecting *View: Window ⇨ Windows ⇨ Progress Window*. Information text regarding the simulation will appear above the progress bar. The most important stages are listed below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

1. **Updating tasks: 1of 1:** the selected task includes the previously created EM and CHT simulation.
2. …. the EM simulation is performed….
3. **CHT solver: Surface mesh generation:** the solid surfaces are triangulated.
4. **CHT solver: Octree grid generation:** the CFD mesh is constructed by using the solid surface triangulations.
5. **CHT solver: Importing surface/volume losses:** the losses from the EM simulation are imported and mapped into the CFD mesh.
6. **CHT solver: Upgrade grid:** inactive cells are removed from the CFD mesh.
7. **CHT solver: Iterations:** the simulation is performed.

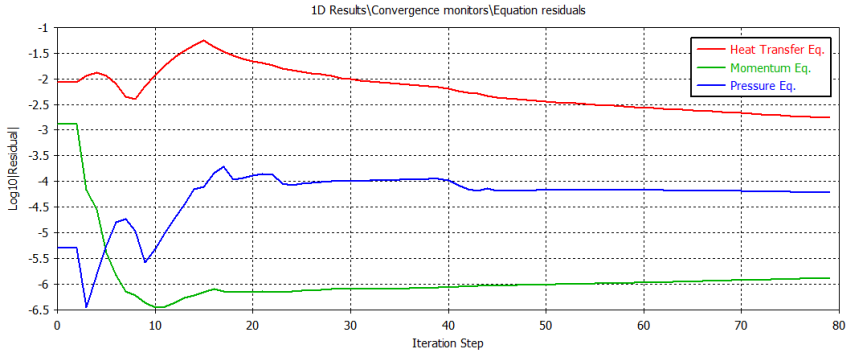
Simulation Results

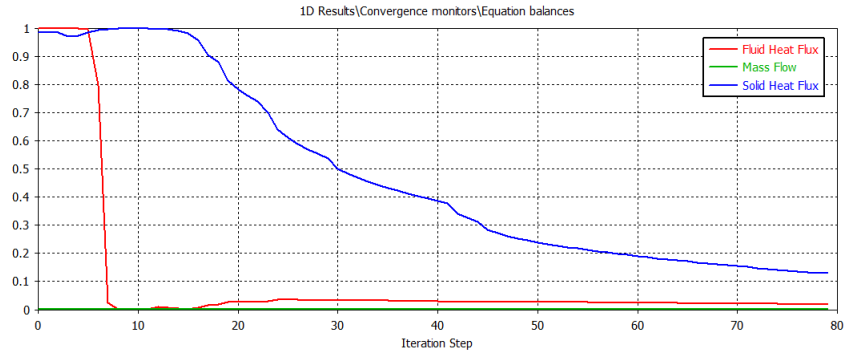
Once the EM simulation has been completed please leave the schematic and return to the CHT simulation. Follow the progresses of the CHT simulation by looking at the convergence monitors in the *NT: 1D Results ⇨ Convergence monitors ⇨ Equation residuals* and *NT: 1D Results ⇨ Convergence monitors ⇨ Equation balances*.

The simulation completes 80 iterations before stopping. The following dialog box pops up because the simulation has not fully converged (i.e. several convergence criteria have not been met due to the low maximum number of iterations):

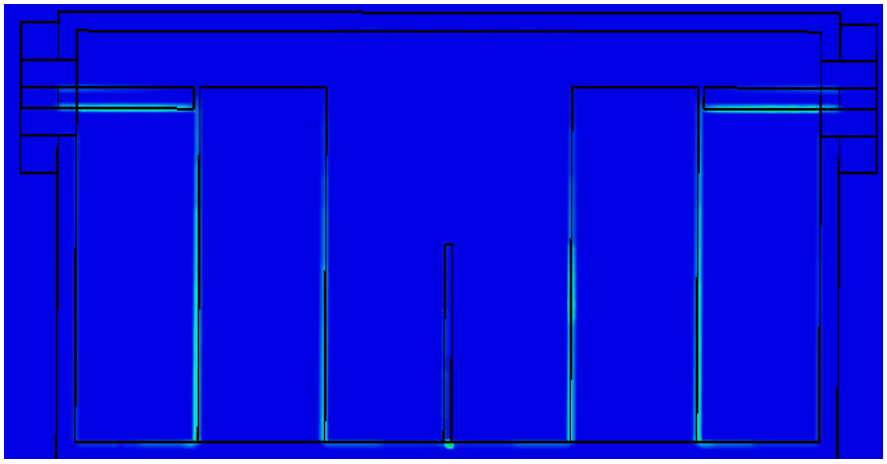


Choose *Stop and save the results* and press *OK*.

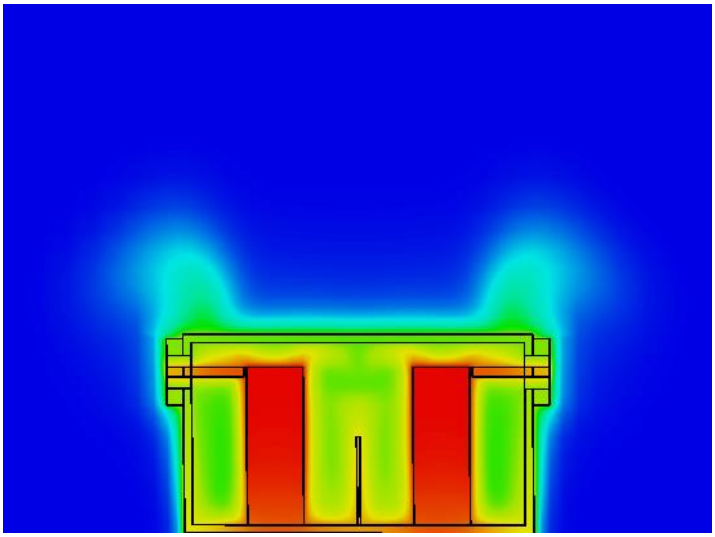




To visualize the loss imported from the EM simulation, select *NT: 2D/3D Results ⇨ Heat source densities* and a cut plane, for instance X=0. Please note that the losses can only be visualized on cut planes (check ribbon *2D/3D Plot: Sectional View ⇨ Fields on Plane*). Observe that the losses are the highest on the walls of the coaxial feeds and of the cylinders.



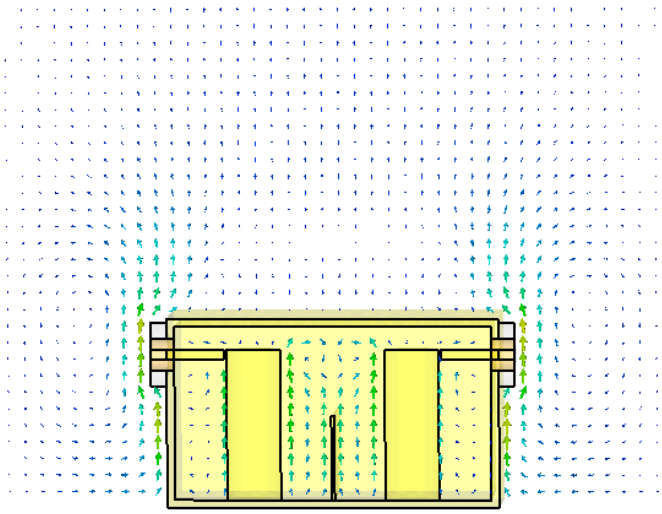
Once the simulation has stopped, visualize in the same cut plane the temperature by selecting *NT: 2D/3D Results ⇨ Temperature*.



The temperature increases are the highest where the losses are also the highest. The air in contact with the walls of the filter heats up and carries the heat away which cools down the filter. One can observe that the simulation has not converged to a steady-state solution in the whole domain because the heat carried by the air flow has not yet reached the top boundary of the computational domain. Still, the simulation has enough progressed to show a correct temperature distribution inside the filter.

The CHT solver takes into account the air cooling effect by simultaneously calculating the heat transfer in the fluid and solid domains and the air flow caused by the temperature gradients and gravity. This key feature differentiates the CHT solver from the thermal solvers which do not solve for the air flow and thus can simulate neither natural nor forced convection.

The air flow can be visualized by selecting *NT: 2D/3D Results ⇨ Velocity*



The velocity vector plot shows the air circulating inside the filter as well as the heated air ascending and being replaced by air at ambient temperature.

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 the online help tutorials.

Chapter 3 – Solver Overview

Solvers and Sources

Various simulation types differ in the definition of materials, boundary conditions and sources. The way 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 for each solver are explained below.

Mechanical Solver:

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

Thermal and Conjugate Heat Transfer 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 Properties
- Convection and radiation at surfaces:
  Simulation: Sources and Loads ⇨ Thermal Surface
- Initial temperature distribution for a transient calculation:
  Simulation: Sources ⇨ Field Import

Conjugate Heat Transfer Solver:

- Fan: *Simulation: Sources and Loads ⇨ Fan*
- Two-resistor component model:
  Simulation: Sources and Loads ⇨ Two-resistor model

Mechanical Solver

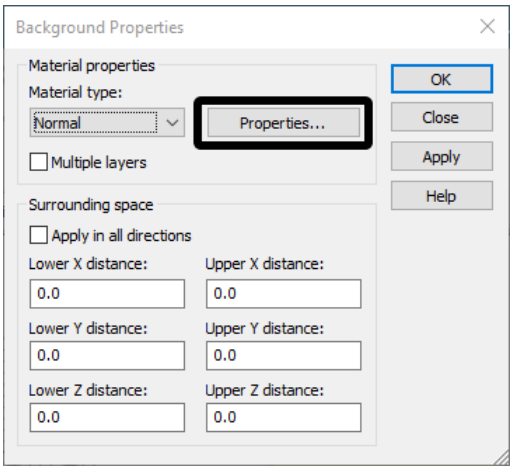
The mechanical solver is a tetrahedral based solver for structural mechanic problems. Its main application is computing deformations driven by thermal expansion and external forces. The deformation results can be used for a subsequent High Frequency Electromagnetic analysis with the tetrahedral based 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 and Conjugate Heat Transfer Solvers

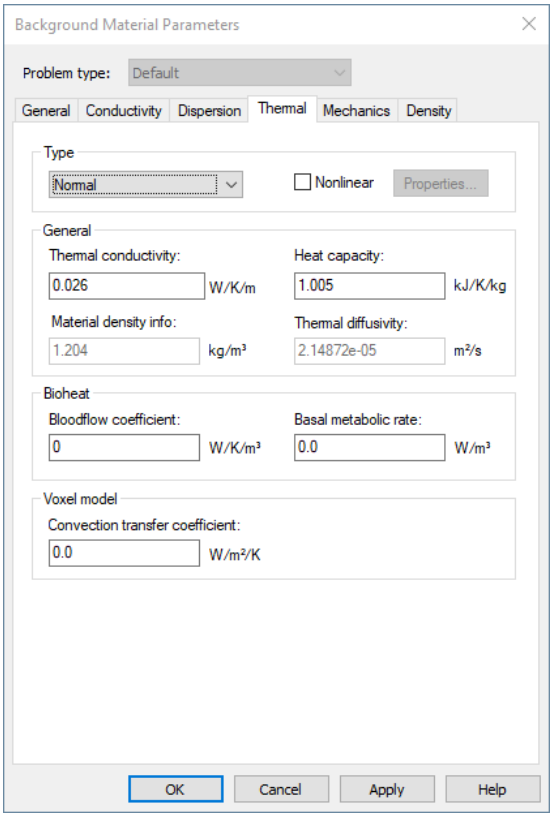
CST MPhysics Studio includes a thermal and a conjugate heat transfer (CHT) solver. The thermal solver is optimized to simulate thermal conduction in the steady state and transient regime and supports hexahedral and tetrahedral grids. The CHT solver is a CFD based heat transfer solver capable of solving thermal conduction, convection and radiation simultaneously in the steady state regime. The main applications of these solvers include solving steady state or transient temperature problems resulting from various types of losses.
In addition, the thermal and CHT 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

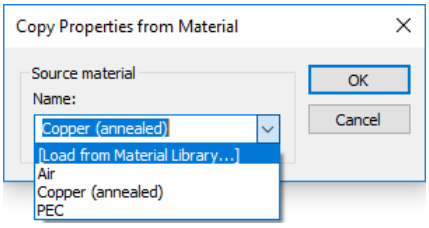
The first step for setting up a thermal simulation is to define the units for temperature and dimension, like it has been described in the chapter *Simulation Workflow*. Afterwards an appropriate background material should be selected. Open the material background properties dialog box by selecting *Modeling: Materials ⇨ Background*:



For thermal problems, the background material is set to Air (thermal conductivity: 0.026 $WK^{-1}m^{-1}$, heat capacity: 1.005$kJK^{-1}kg^{-1}$, density: 1.204 kg/m^3 and dynamic viscosity: 1.84e-5 Pa.s at normal conditions). These settings may be changed by selecting the *Material type* (*Normal* is advisable in most cases), afterwards opening the material dialog box by pressing *Properties...* and select the *Thermal* property page:



The easiest way to assign the necessary values is to copy the properties from an existing material 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 choose the desired material from the material list.

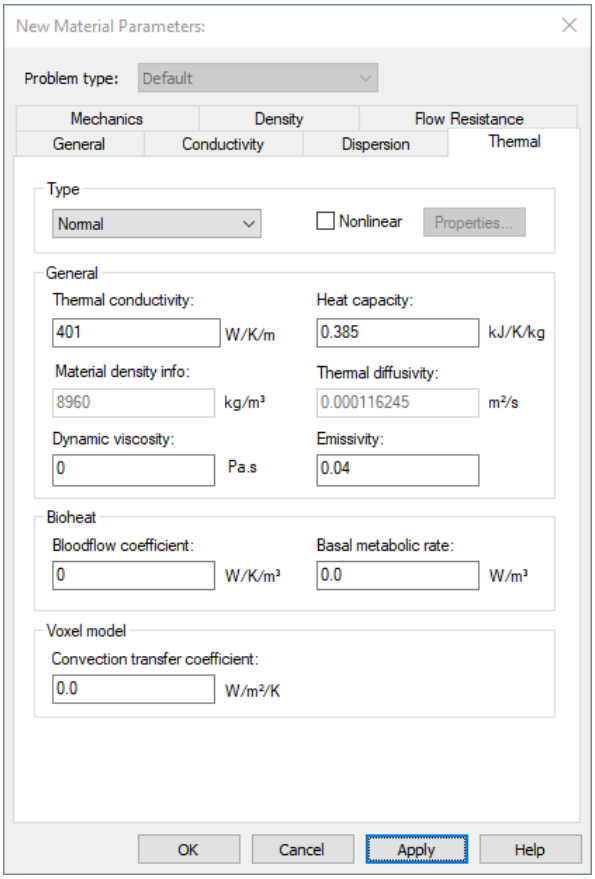
Material Properties

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.

It is necessary to specify a thermal conductivity to perform a thermal or conjugate heat transfer simulation. In the *Thermal* tab please 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*…
Please note that the conjugate heat transfer solver only supports isotropic and constant material parameters.

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.
Please note that the conjugate heat transfer solver replaces PTC with copper.

For transient thermal problems (see also the next chapter) and the conjugate heat transfer solver the heat capacity and the material density must be specified. These parameters determine how much energy per Kelvin is stored in a certain amount of mass or volume:



Specify the material emissivity when radiation is enabled in a conjugate heat transfer simulation.

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 thermal conductivity, the heat capacity and the material density as follows:

$$\alpha = \frac{k}{\rho \cdot c_p \cdot 1000},$$

where

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).

The Flow Resistance material parameter is only supported by the conjugate heat transfer solver. It is used to model the fluid flow behavior across a screen without having to mesh the screen geometry.
A flow going through a sheet with a planar flow resistance experiences a pressure drop which can be expressed as follows:

$$\Delta P = -0.5 \cdot f \cdot \rho \cdot \vec{u} \cdot \vec{n} \cdot \left|\vec{u}\right|,$$

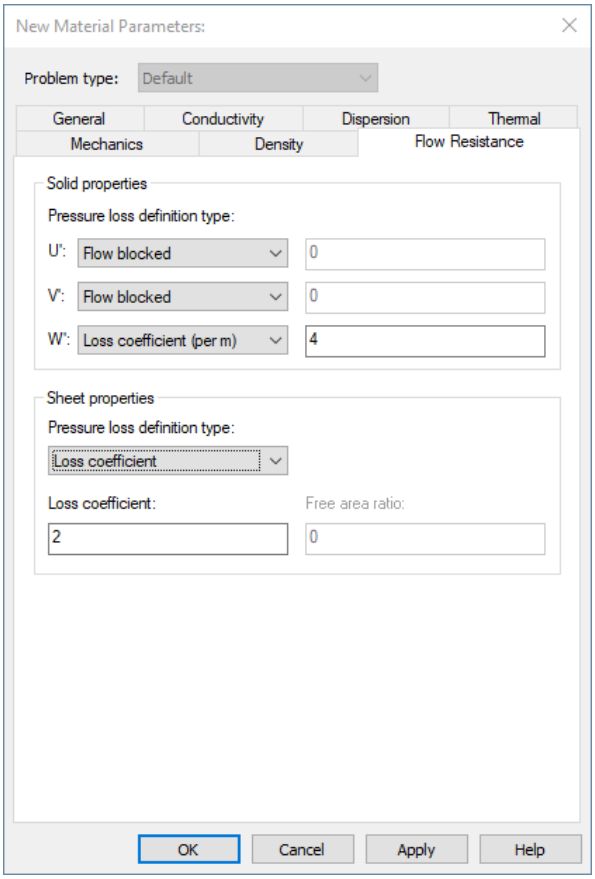
where f is the dimensionless loss coefficient, $\vec{n}$ is the sheet local normal and $\vec{u}$ the flow velocity.

A flow going through a volume resistance experiences a pressure gradient which can be written as follows:

$$\nabla P_j = -0.5 \cdot \rho \cdot \sum_j f_{ij} \cdot u_j \left|u_j\right|,$$

where f_{ij} is the loss coefficient tensor per unit length and u_j is a velocity component in the global coordinate system (X,Y,Z).
The loss coefficient tensor is defined with respect to a local coordinate system (U',V',W') and transformed into a 3x3 tensor in the global coordinate system.

A Flow Resistance assigned to a surface uses the specifications of the sheet properties group whereas a Flow Resistance assigned to a solid uses the specifications of the solid properties.

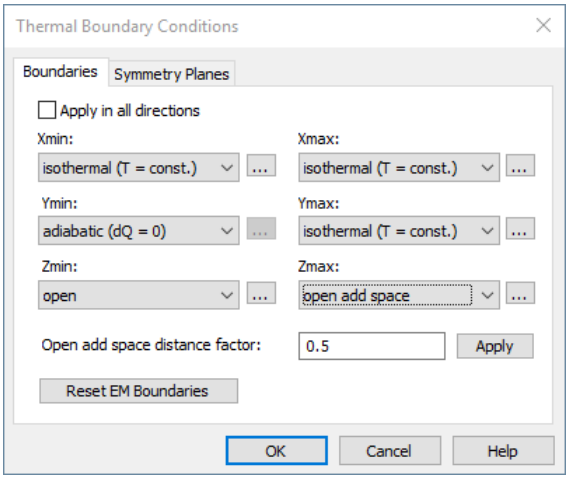


Boundary Conditions

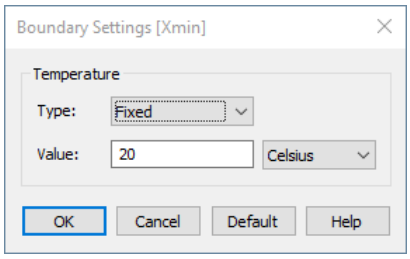
The boundary conditions for the thermal and conjugate heat transfer solver can be defined in the *Thermal Boundaries* tab of the *Boundary Conditions* dialog box (*Simulation: Settings ⇨ Boundaries*)

For Steady State and Transient Thermal Solvers:

The thermal boundary offers the following choices of boundary conditions:



For "*isothermal*" and "*open*" boundaries the temperature settings may be assigned by pressing the corresponding button *[…]*. This button opens the dialog Boundary Settings, in which the temperature value can further be configured, for example, by assigning of a fixed or floating temperature. By default, the option *Unset* is selected, which means the boundary is considered as a PTC surface without sources assigned.



For the *"open"* boundary condition, it is assumed that the temperature approaches the predefined value with increasing distance from the structure. Apply this type of boundary condition if thermal **conduction** through the surrounding background material plays an important role for your problem. In order to consider thermal **convection** effects on the structure, *Thermal surface properties* (see p. 44) should be used.

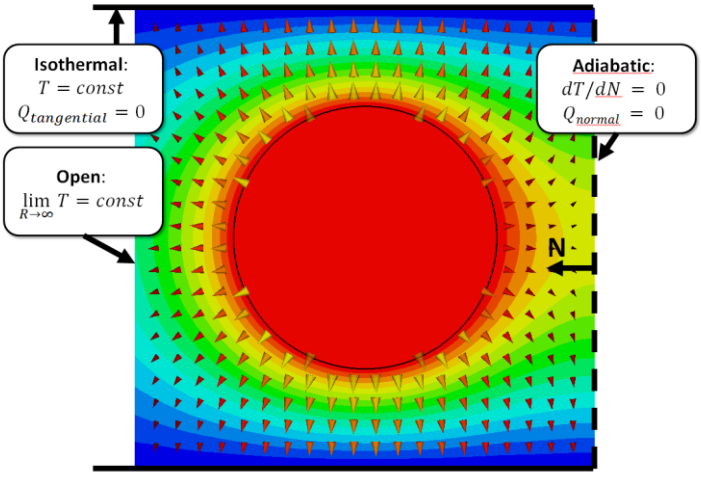
When no heat flow leaves the computational domain through a boundary, use the "*adiabatic"* boundary condition. In case the conductive heat flow of an open structure can be neglected, you can use these boundary conditions instead of "*open*" boundary conditions (if radiation or convection effects dominate).

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:

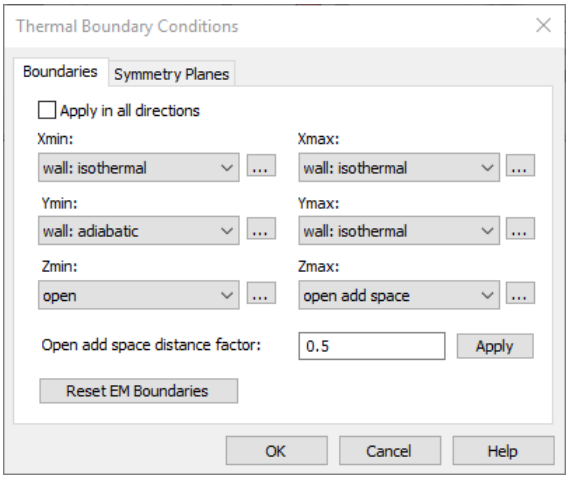
	Temperature (**T**)	Heat Flow (**Q**)
Isothermal	**T** = const (fixed or floating)	$\mathbf{Q}_{\text{tangential}} = 0$
Adiabatic	d **T** / dN = 0	$\mathbf{Q}_{\text{normal}} = 0$
Open	$\lim_{R\to\infty}$ (**T**) = const (fixed or floating)	

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.



For Conjugate Heat Transfer Solver: 

The conjugate heat transfer solver supports similar types of boundary conditions. It is however important to note that it interprets these boundary conditions differently, in particular for the case of open boundaries.



User input for isothermal walls: temperature, emissivity and friction (no-slip/slip) at the boundary wall. The reference temperature used for radiation is the wall temperature.

User input for adiabatic walls: friction at the boundary wall (no heat exchange, zero emissivity).

User input for symmetrical boundaries: none (no friction, no heat exchange, zero emissivity).

User input for open boundaries: flow temperature, flow velocity or flow gauge pressure. An open boundary allows flow to enter and leave the domain, which could be used to model the flow and thermal behavior of an inlet or of an outlet specified by a pressure gauge or a velocity. If the flow temperature is unknown, which is the case for outlets or if the flow direction is unknown set the temperature to unset.

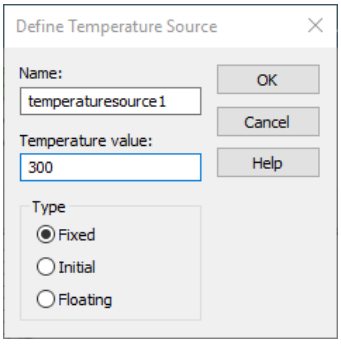
The emissivity is set to 1. The reference temperature used for radiation is the radiation temperature defined in solver parameter dialog.

Sources and Loads

The thermal and conjugate heat transfer solvers can handle several types of sources or loss mechanisms, which are listed below:

Temperature Source

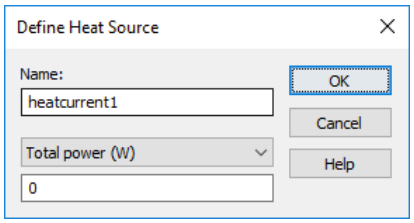
This source is available via *Simulation: Sources and Loads ⇨ Temperature Source* . This source type can be assigned to a surface of an object with PTC material properties or any other material with non-zero thermal conductivity. You can choose between a fixed temperature value and a floating temperature. A floating temperature is a uniform temperature distribution with zero heat flow from or into the associated surface. Besides, for the transient solver an initial temperature source can be defined, which is taken into account only for generation of the initial temperature distribution and ignored during the transient solution.



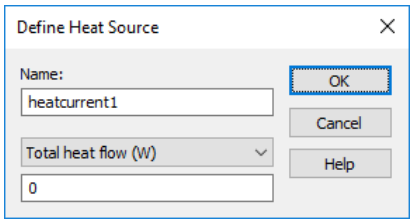
Heat Source

This source is available via *Simulation: Sources and Loads ⇨ Heat Source*

When assigned to a solid with a non-zero thermal conductivity source and that is neither PTC nor PEC it 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*).

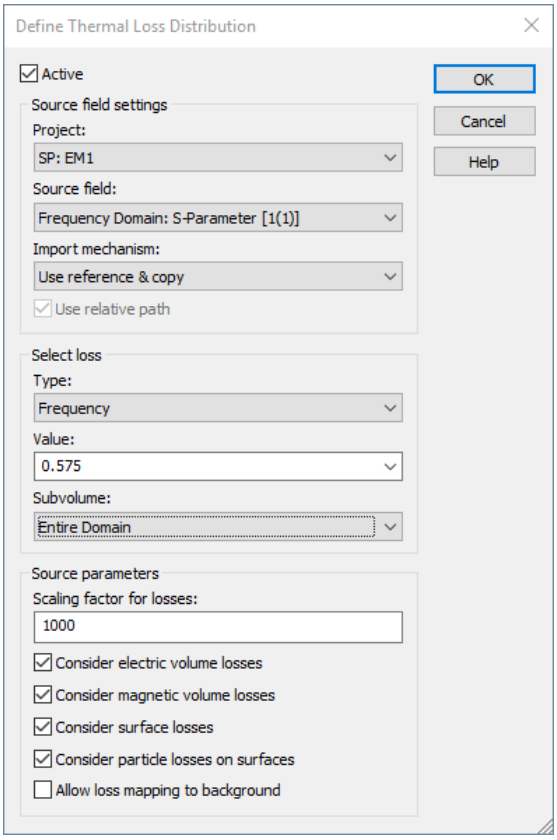


When assigned to a solid that is either PTC or PEC it defines the total heat flow coming from the solid surface. Therefore, a heat source with zero heat flow and a floating temperature are identical.



Thermal Loss Distribution

This source is available via *Simulation: Sources and Loads* ⇨ *Thermal Losses* . Thermal losses can occur inside 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 thermally conductive materials. If previously calculated loss distributions are present, you can edit setting by reopening the dialog box (*Simulation: Sources and Load* ⇨ *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.

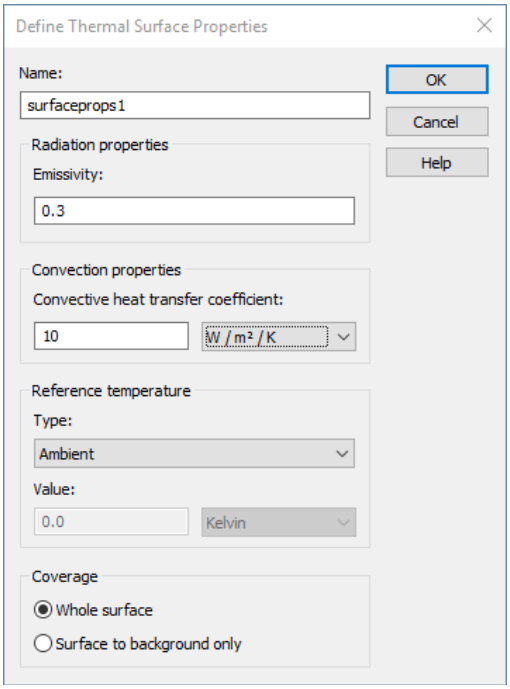
Type of loss	Created by
Ohmic (electric vol. losses)	Transient Solver (), Frequency Domain Solver (), Eigenmode Solver (), J-Static Solver (), LF-Solver (), PIC Solver (), Wakefield Solver (), IR-Drop Solver ()
Lossy metal (surface losses)	Transient Solver (), Frequency Domain Solver (), Eigenmode Solver (), LF-Solver (), PIC Solver (), Wakefield Solver ()
Dispersive (electric and magnetic vol. losses)	Transient Solver (), Frequency Domain Solver (), PIC Solver (), Wakefield Solver ()
Crashed particles	Tracking Solver (), PIC Solver ()

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

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

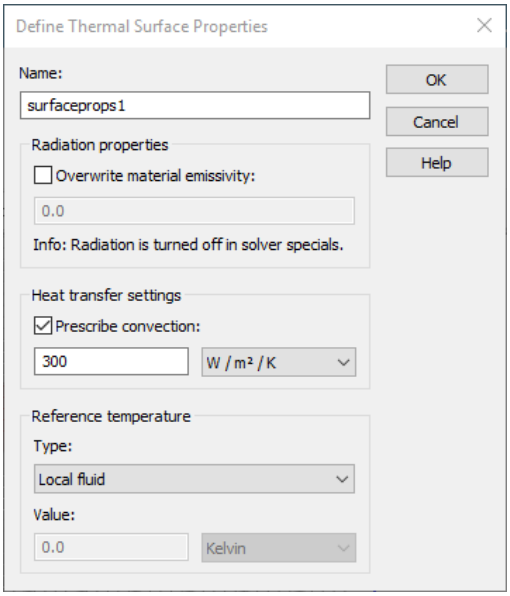
whereas $Q_{Radiation}$ stands for the radiated power, T for the surface temperature, $T_{\text{Reference}}$ for the reference temperature, which can be equal to ambient or user-defined, σ for the Stefan-Boltzmann constant and $A_{Surface}$ for the area for the selected surfaces. An emissivity value $\varepsilon = 0$ 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 *Convective heat transfer coefficient* h describes convection processes between a fluid and the surface of conductive materials:

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

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

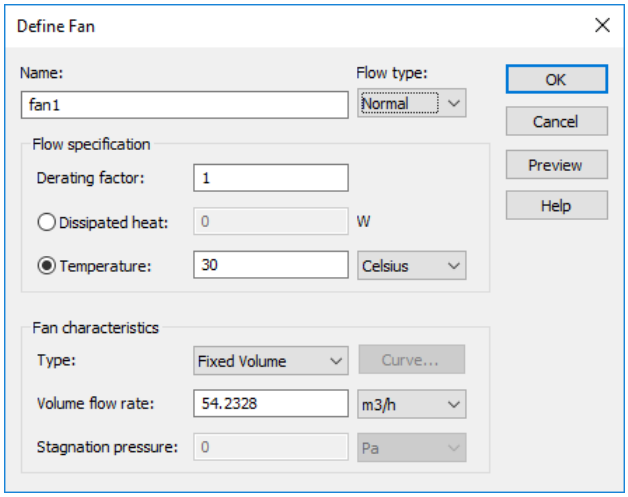
The thermal surface properties dialog includes additional options for the conjugate heat transfer solver. The emissivity of the solid defined by the emissivity of its material can be overwritten by a surface emissivity for the assigned surface. In addition, the local fluid temperature can be used as the reference temperature when convection is prescribed by a heat transfer coefficient.



Fan *(not supported by Thermal solver)*

Axial fans are available via *Simulation: Sources and Loads* ⇨ *Fan* . They are defined by their entry and exit faces. The entry and exit faces must belong to the same lump of the same solid. They can be either assigned both to the same surface if the fan is planar or translated from each other. Note that a planar (infinitely thin) fan can only be created on an outer boundary and can't be created in the interior domain. A non-planar (thick) fan can be created either on the outer boundaries or in the interior domain.

The axial fan behavior can be specified as follows:



The fan characteristics (i.e. fan curve, volume flow rate or stagnation pressure) are given for a quoted speed. The fan however can be operated at a different speed. The derating factor is the ratio of operating speed and quoted speed and is a dimensionless value between 0 and 1. If the derating factor is 0.8, the operating speed will be 80% of the quoted speed and the fan characteristics and the dissipated heat will be adjusted accordingly. The flow temperature can be controlled either by specifying a fixed temperature or the amount of heat dissipated from the flow going through the fan.

The fan characteristics are given by a fan curve defined either by one or two or more points. If a fan curve has only one point its type is *Fixed Volume* and is specified by entering its volume flow rate. If the fan curve has two points its type is *Linear* and is specified by entering its volume flow rate for zero pressure and its stagnation pressure. If the fan curve has more than two points its type is *Nonlinear* and each point can be entered individually by clicking on the *Curve* button.

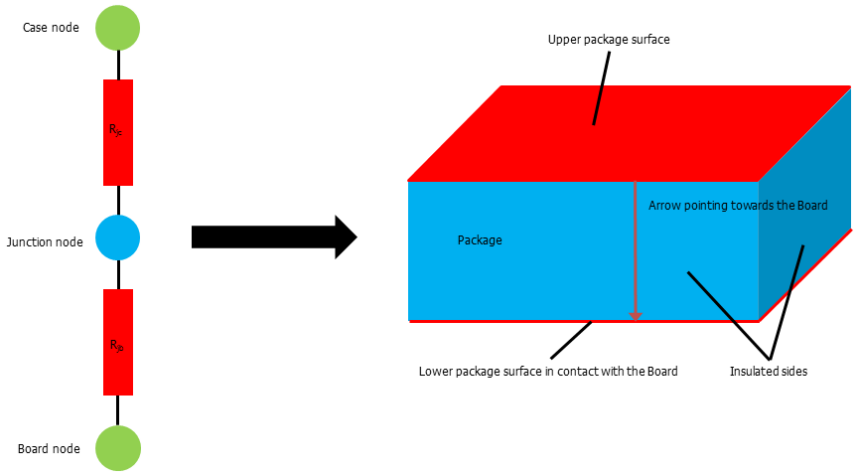
Two-resistor component model *(not supported by Thermal solver)*

Two-resistor component models are *available via Simulation: Sources and Loads ⇨ Two-resistor model* .

The two-resistor component model can be used to approximate the thermal behavior of single-die packages that can be effectively represented by a single junction temperature.

The model is based on the block-and-plate method described in the Two-Resistor Compact Thermal Model Guideline specified in the JEDEC standard JESD15-3.

The 3D representation of the two-resistor model is shown below:



The input parameters of the model are the case node and board node temperatures, which are provided by the heat transfer solvers, and the junction node dissipated power together with the junction-to-case R_{jc} and the junction-to-board R_{jb} thermal resistances that must be provided by the user.

The output parameter of the model is the junction node temperature. In the 3D representation, the package represents the junction node thermal resistance whereas the upper package and the lower package surfaces represent the junction-to-case thermal resistance and the junction-to-board thermal resistance, respectively. The package lateral sides are assumed to be insulated (no heat transfer).
The two-resistor component model has been extended by making it possible to define contact properties on the upper package surface. This is useful when a heatsink covers the upper surface package.

Bioheat Source *(not supported by CHT solver)*

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 the local thermoregulation in living tissues is an increased bloodflow coefficient with rising temperature due to the 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*.

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

Thermal Contact Properties

Thermal contact properties can be defined via *Simulation: Sources and Loads ⇨ Contact Properties* . A contact item is equivalent to a thin layer of thermally conductive material at the interface between two (or several) solids. It can be characterized either by lumped parameters (absolute thermal resistance [K/W] or thermal resistance per unit area [K·m²/W] as well as 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:

Absolute thermal resistance (K/W): $$R_\Theta = \frac{l}{kA}$$

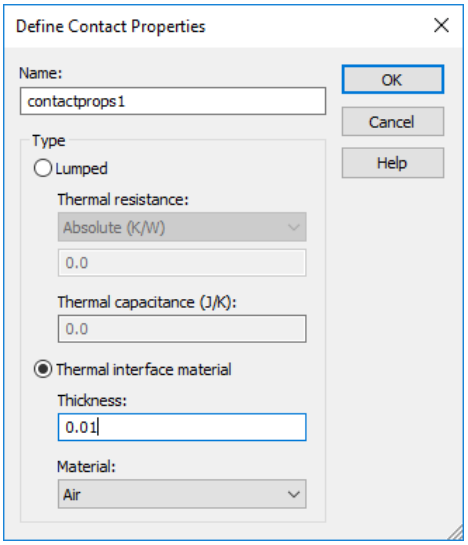
Thermal resistance per unit area (K·m²/W): $$r_\Theta = \frac{l}{k}$$

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

Here R_θ represents the absolute thermal resistance, r_θ the thermal resistance per unit area, C the thermal capacitance of the contact layer. In the material-based representation, thermal conductivity k, specific heat capacity c_P, material density ρ and layer thickness l are used. The contact area A is calculated by the solver.

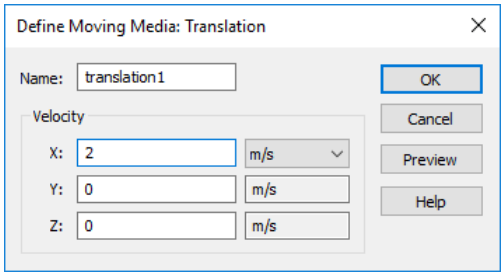
The advantage of contact properties definition through lumped parameters is the ease and transparency of the parameter values. Besides, the absolute thermal resistance is independent from the contact area A which may vary in case of solid intersections or depending on the mesh settings. On the other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

Thermal contact properties are only supported by tetrahedral-based thermal solvers and the conjugate heat transfer solver.



Moving Media *(not supported by CHT solver)*

For each solid containing a non-PTC thermal conducting material, a moving media velocity vector may be assigned via *Simulation: Motion ⇨ Moving Media*.



This vector defines the velocity with which the material comprising the solid is moving relatively to the sources and solid geometry. A typical example would be a very long tube moving through a coil for the purpose of induction heating.

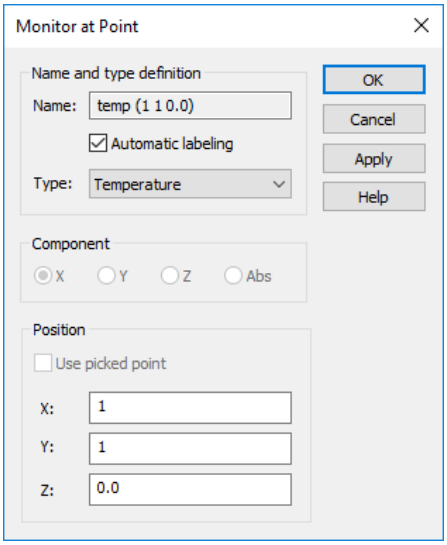
If a velocity vector has been assigned to any solid, the solver saves important information about the distribution and maximum of Peclet number in order to control the solution quality.

Only tetrahedral-based thermal solvers support this feature. In the transient solution, the moving media velocity vector may be made time-dependent by assigning *Excitation Signals* to its components.

You can find more detail about moving media in the online documentation.

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 these monitors via *Simulation: Monitors ⇨ Monitor at Point*.



Steady state 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 steady state solver records the temperature value after each refinement step and saves it 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.

Two additional types of monitor at point are available for the conjugate heat transfer solver. The *Pressure* and *Velocity* types evaluate, respectively, the pressure and velocity at the monitor point at each iteration.
The conjugate heat transfer solver saves the values of the monitor points as 1D data into the Navigation Tree under *NT: 1D Results ⇨ Monitors at Points ⇨ <monitor name>*.

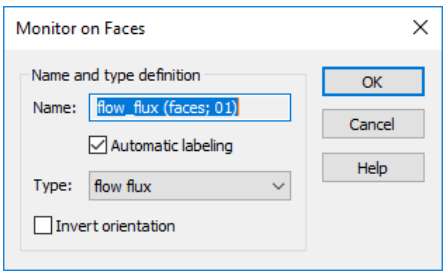
The conjugate heat transfer solver can use the point monitors activated in *Simulation: Setup solver: Accuracy: Custom stop criteria* to detect the convergence of the solver.

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

Monitors on Faces

The monitors of this kind record scalar values defined on a surface. You can create these monitors via *Simulation: Monitors ⇨ Monitor on Faces* .

Two types of monitors on face are available. The type flow flux is used to monitor the fluid flow, consequently the monitor surfaces must not change the flow and must be borrowed from a dummy solid. A dummy solid is either a solid whose material is exactly the same as the background material or a solid not considered for simulation but considered for the bounding box. If necessary, adjust the local mesh properties of the dummy solid to match those of the background to avoid unwanted mesh refinements around the dummy solid.



The flow flux monitor calculates the mass flow rate, the energy flux and the bulk temperature through the monitor surfaces, respectively defined as:

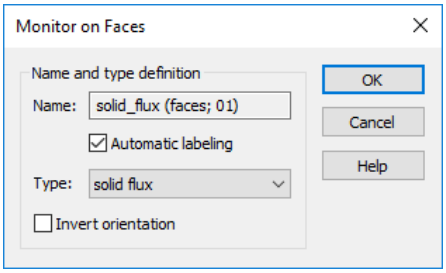
$$\dot{m} = \oiint \rho \mathbf{u} \cdot \mathbf{dA}$$

$$\dot{Q} = \oiint \rho C_p \mathbf{u}(T - T_{amb}) \cdot \mathbf{dA}$$

$$T_b = \frac{1}{\dot{m} C_p} \oiint \rho C_p \mathrm{T}\mathbf{u} \cdot \mathbf{dA}$$

The type solid flux is used to monitor the heat flux and the heat transfer coefficient at solid/fluid interfaces, consequently the monitor surfaces must be borrowed from a solid considered for simulation and considered for the bounding box and whose material is different from the background material:

$$P = \oiint -k \cdot \nabla T \cdot \mathbf{dA}$$

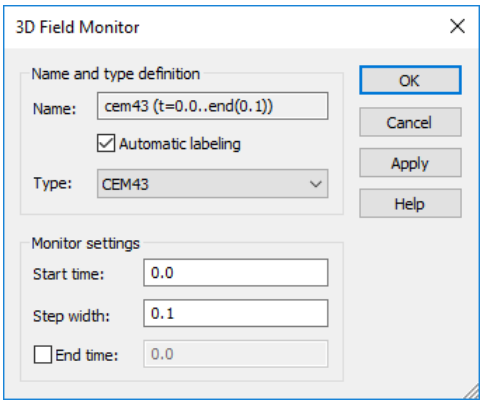


The monitors on faces are evaluated at each iteration and the surface quantities are saved as 1D data into the Navigation Tree under *NT: 1D Results ⇨ Monitors on Faces ⇨ <monitor name>*. The conjugate heat transfer solver can use the face monitors activated in *Simulation: Setup solver: Accuracy: Custom stop criteria* to detect the convergence of the solver.

3D Field Monitors

Note: the conjugate heat transfer solver stores the simulation results obtained for the whole computational domain and ignore the 3D field monitors.

In contrast to steady state 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:

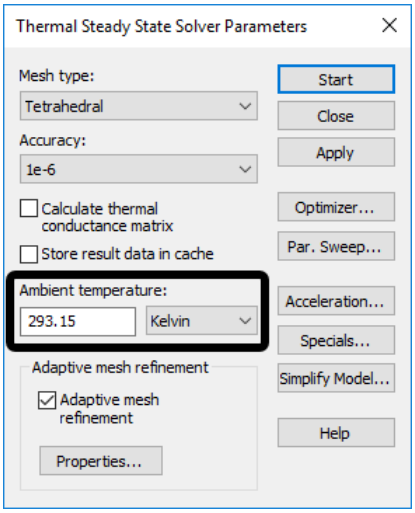


Three field types are available: *Temperature*, *Heat Flow Density* and *CEM43*. The latter monitor represents the distribution of Cumulative Equivalent Minutes at 43°C, which is commonly used to detect the damage of biological tissues exposed to strong electromagnetic fields. After the

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.

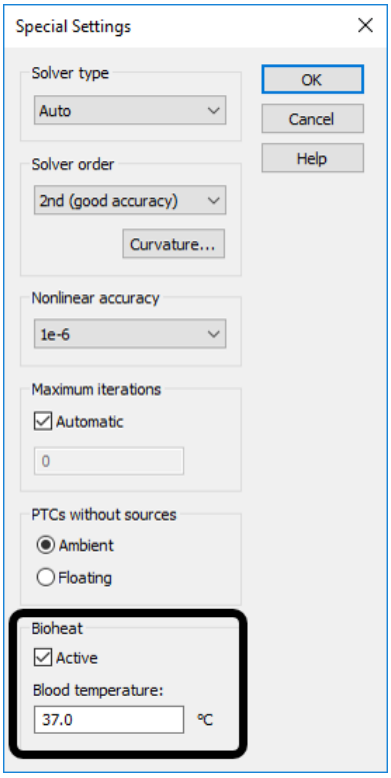
Steady State Thermal Solver Parameters

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



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

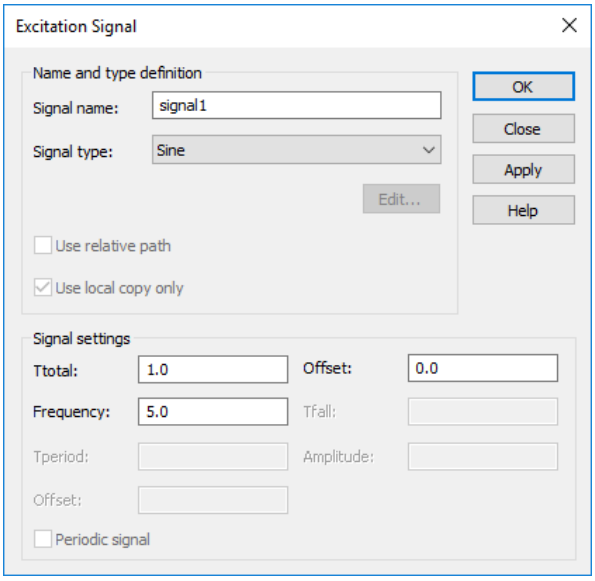
If Bioheat properties must be adjusted, one can open the *Specials* dialog:



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

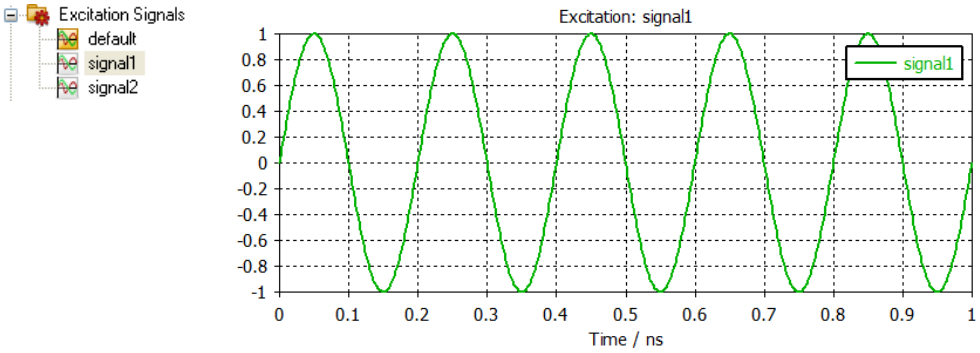
Excitation Signal Settings

For some transient thermal simulations, it is necessary to define time domain excitation signals to model, for example, time varying heat sources. 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 and can be displayed by selection in the Navigation Tree:

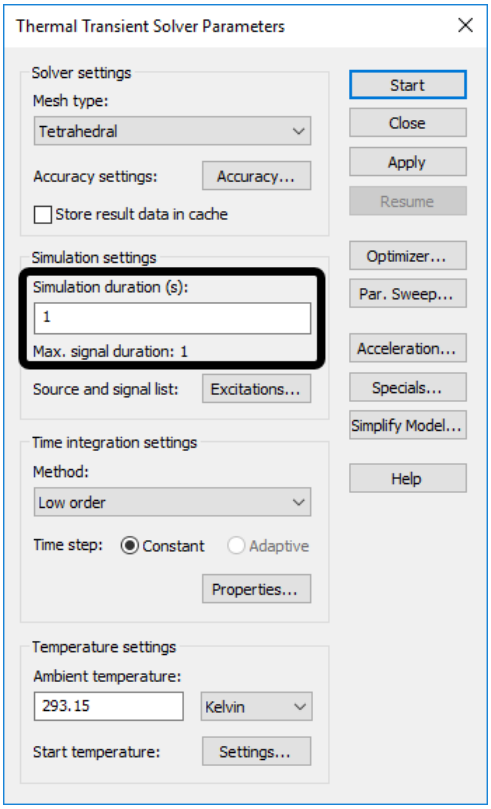


Transient Thermal Solver Settings

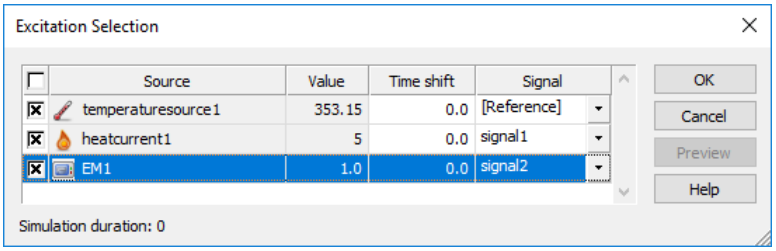
You can switch between the steady state and transient thermal solvers by selecting either

Home: Simulation ⇨ *Setup Solver* ⇨ *Thermal Steady State 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:



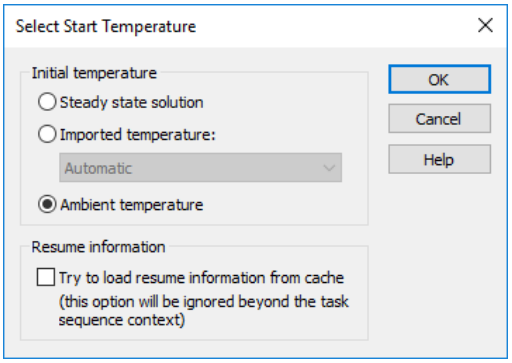
Most source types can be weighted with a previously defined excitation function, when pressing the *Excitations* button:



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 :

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

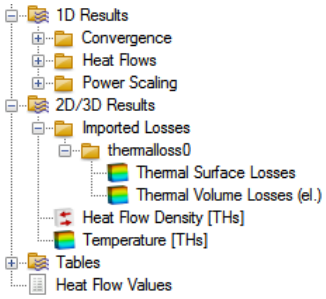
The initial temperature distribution can be defined in the Select Start Temperature dialog, which can be called by pressing the *Start temperature: Settings* button. The default setting is to assign the ambient temperature everywhere except regions with temperature sources. Alternatively, it is possible to assign the solution of the steady-state problem with initial source values as well as import a temperature distribution from an external thermal solution.



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

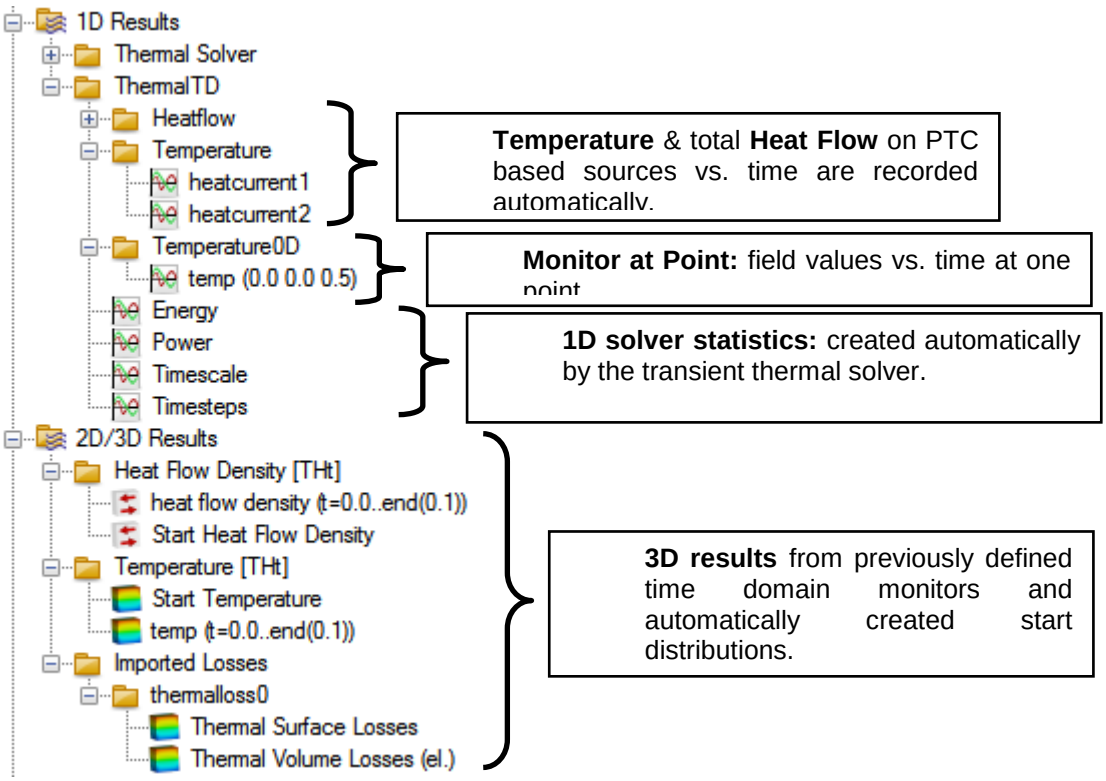
After a **steady state thermal simulation** run has been completed successfully, new result entries appear in the navigation tree:



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 thermally 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:

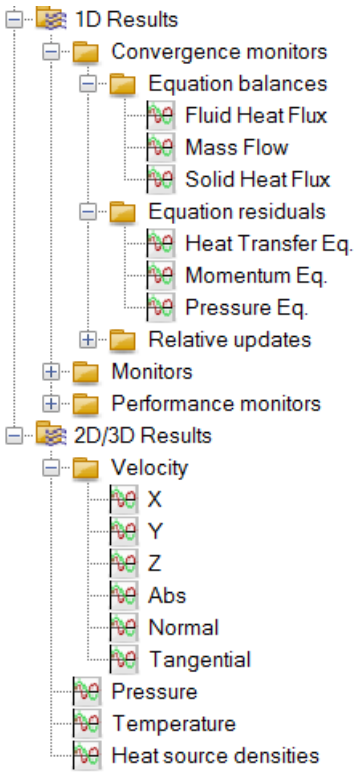


If time domain temperature monitors have been defined for the transient thermal solver, the associated results will be listed under *2D/3D Results* as well. In addition, a couple of time signals are added to the 1D Results section:

- **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.

The **conjugate heat transfer solver** produces the following results



The 1D Results contain solution convergence, point and face monitors as well as performance data, plotted against iterations to give user insights into convergence and solutions.

The 2D/3D results contain velocity, temperature, pressure and heat source densities data, which can be updated during the iteration process using *Plot Properties* ⇨ *Update Results* or the F5 key.

Chapter 4 – Finding Further Information

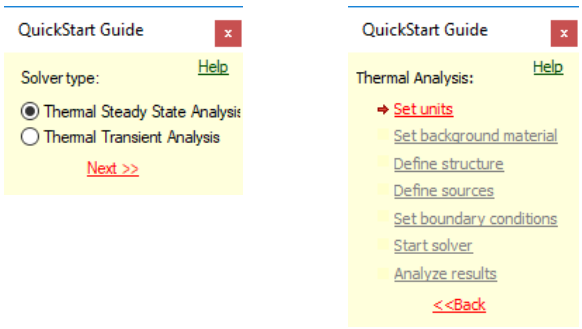
After carefully reading this manual, you will already have some idea of how to use CST MPhysics Studio efficiently for your own problems. However, when you are creating your own first models, some questions may arise. In this chapter, we give you a short overview of the available additional documentation.

The Quick Start Guide

The main task of the Quick Start Guide (not available for Conjugate Heat Transfer solver) 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 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 ⇨ Help Contents*. The online help system includes a powerful full text search engine.

In each of the dialog boxes, there is a specific *Help* button, which 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 and Examples

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

Technical Support

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

Macro Language Documentation

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

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

History of Changes

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