T-FLEX Analysis 17

USER MANUAL

FINITE ELEMENT ANALYSIS

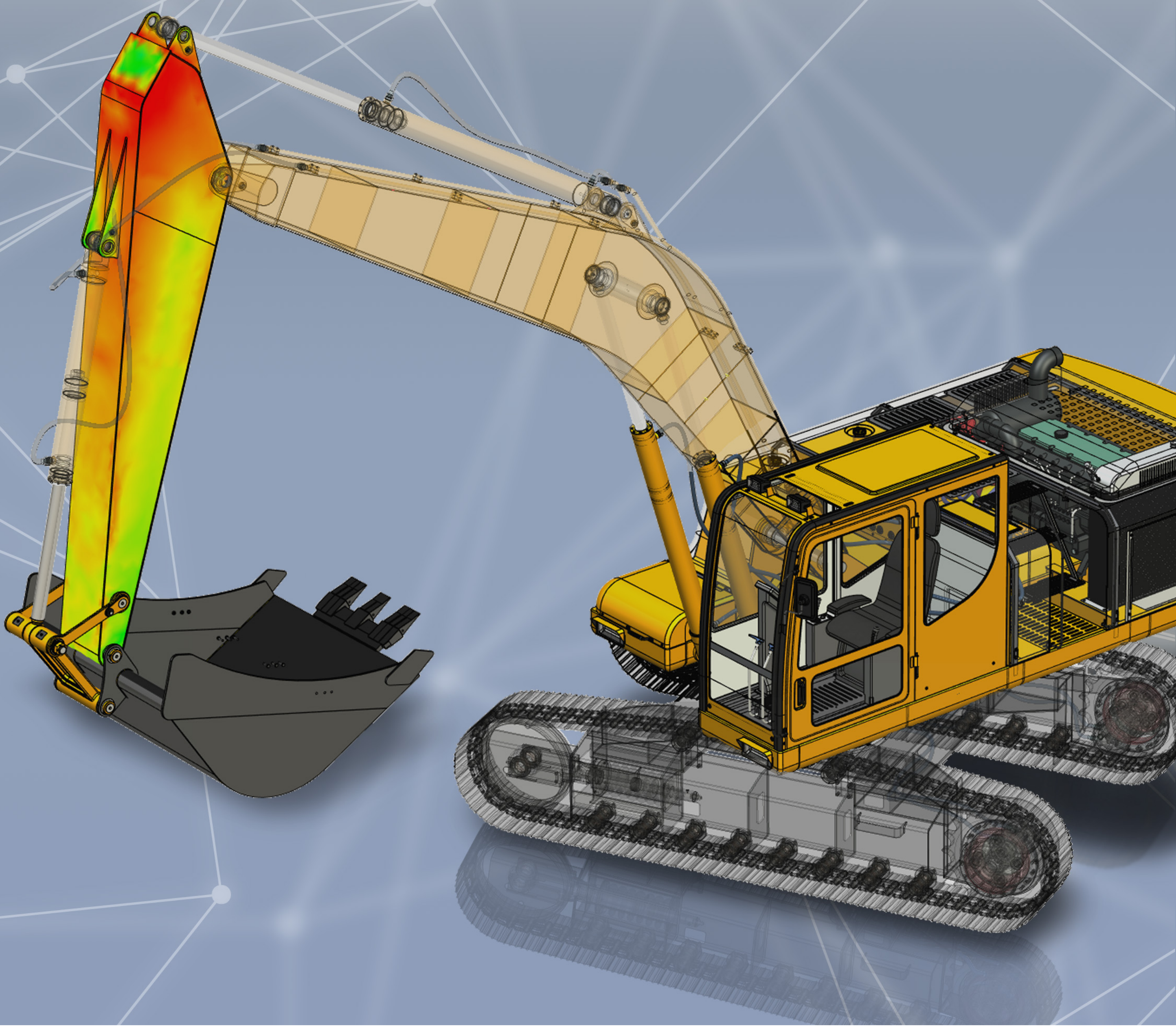


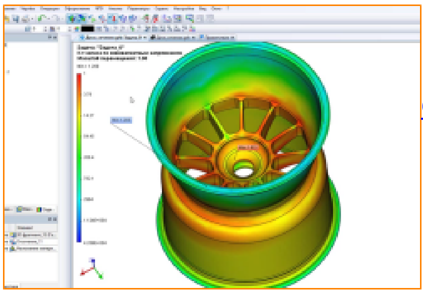
Table of Contents

Welcome

Welcome to T-FLEX Analysis help!

Here you may find the detailed description of all the commands.

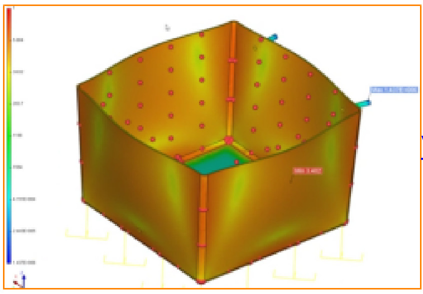
Using Contents and Search windows.



Quick Start

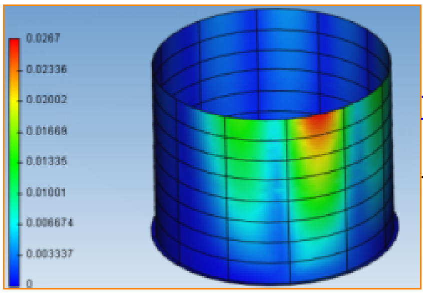
Brief overview of the basics of FEA studies creation; vocabulary of T-FLEX Analysis.

Quick start - videos



Verification Examples

Examples of solving different types of studies.



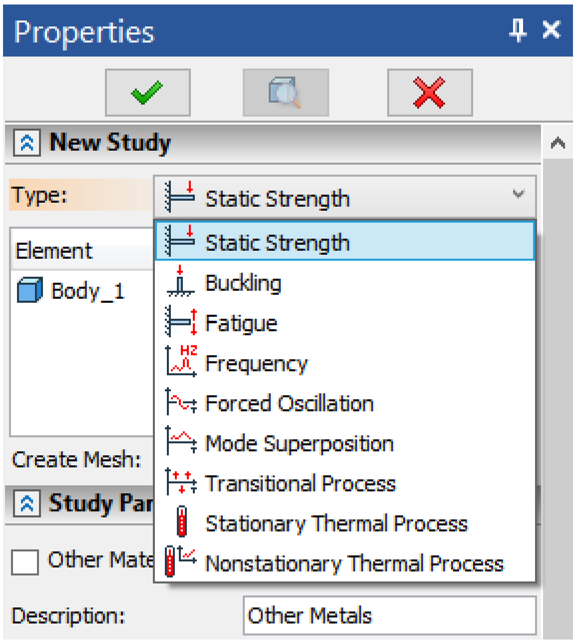
Tutorial course – videos

Tutorial presents fundamentals of T-FLEX Analysis application on the basis of various studies and examples.

T-FLEX Analysis Fundamentals

Introduction

T-FLEX Analysis is an environment for finite element calculations integrated with T-FLEX CAD. With the help of T-FLEX Analysis, a T-FLEX CAD user can perform mathematical modeling of common physical phenomena and solve important practical problems arising in everyday design practice.
All calculations rely on the finite element method (FEM). At the same time, an associative relationship is maintained between the three-dimensional model of a part and the finite element model used in calculations. Parametric notifications of the original solid model are automatically propagated into the meshed finite element model.
The following calculations modules are available.



Finite element analysis studies types

Static Strength allows calculating the state of stresses and strains in a structure under the impact of constant in time forces applied to the model. It is also possible to account for stresses building up due to thermal material expansion/contraction, or for structural deformations introduced by known displacements. By using the Static analysis study, the user can evaluate the strength of a structure developed by him, with respect to admissible stresses, identify the most vulnerable parts of the structure and introduce the necessary changes, optimize the design.
Buckling is important when designing structures, whose operation implies lasting influence of loads ranging in intensity. With the help of this module, the user can evaluate safety margin by the so-called «critical load». Significant nonelastic strains may occur suddenly within compressed parts of a structure, which could likely cause its rupture or serious damage.
Fatigue. Certain parts of machines, mechanisms and also structural elements during the time of their service are subjected to the loads that change with time. Material's resistance to the effects of these loads considerably differs from the material's resistance to static load or impact. For studying material's strength when the time-dependent loads are applied, the Fatigue **Analysis Module** is used in T-FLEX Analysis.
Express Analysis. Evaluation module for static analysis calculations.
Frequency allows calculating natural (resonant) frequencies of a structure and the respective vibration modes. Based on the calculation results, the product is assessed on the presence of resonant

frequencies in the operating frequency range. The developer can enhance reliability and performance of a product by optimizing the design in such a way as to exclude resonance occurrences.

Forced Oscillation is carried out for prediction of response of the structure subjected to harmonically varying external loads. **External loads** include kinetic and/or kinematic excitation. In addition, the damping of the system can be taken into account.

The goal of the forced oscillation analysis is to find the dependence of the system’s response on the frequency of the driving loads. The results of the analysis include the amplitudes of displacements, oscillation accelerations and oscillation overload for the given driving frequency. From the results of the analysis for a range of frequencies we can obtain dependence of the amplitudes and oscillation accelerations on the frequency of the driving loads, which is important when estimating vibrostability of the system in the given frequency range.

Mode Superposition allows you to simulate the behavior of a mechanical system under the loads that are non-constant in time. The solution is provided as decomposition according to the previously calculated natural modes of oscillations of the structure. This type of dynamic study is convenient for the analysis on long time intervals, because it has high performance (if calculated natural modes of oscillations already exist).

Transitional Process allows you to simulate the behavior of a mechanical system in time. Full equations systems are calculated on each time step, which imposes high requirements on computational resources. This method is generic and provides high calculation accuracy, allowing, in particular, to solve problems in nonlinear formulation.

Stationary Thermal Process provides possibility of the steady (stationary) temperature field evaluation that appears in the product under the impact of sources of heat and radiation.

Nonstationary Thermal Process calculates thermal processes that are non-stationary in time.

Both thermal analysis studies can be used independently for calculating temperature and heat fields through the volume of a structure, as well as in combination with static analysis for evaluating thermal deformations building up in the product.

Mathematical Background of T-FLEX Analysis

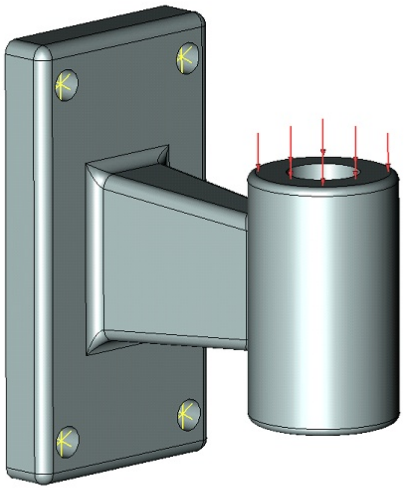
Engineering design often requires investigation of the most important physical and mechanical properties of parts, assemblies, or the entire product. For example, in a design one must evaluate the strength of parts under specified loads or maximum deformations of a product's body. For a long time, the only means for evaluating physical and mechanical properties of products was assessment based on approximate analytical or semiempirical methods, listed in industry guides. The accuracy of such methods is generally not high, with respect to real-life design objects. Consequently, significant «safety factors» (as with respect to the strength) are incorporated, in order to lower the risks of an unviable design.

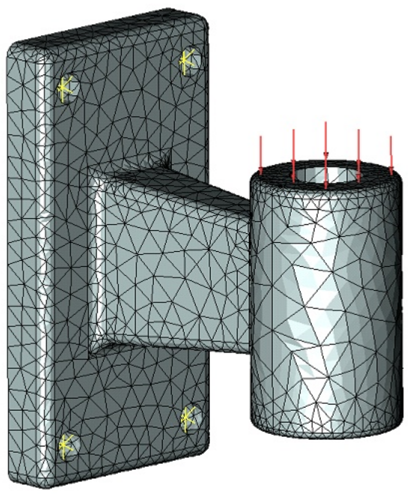
Emergence of computers and development of computer science led to big changes in traditional approaches to engineering calculations. From the mid-60s of the 20th century, the leading method of numerical solving a wide variety of physical problems became finite element method (FEM). The special features of the FEM that put it in the commanding position in the applied computational mathematics are such inherent qualities as:

- versatility – the method is suitable for solving all kinds of different problems of mathematical physics (mechanics of deformable solids, heat transfer, electrodynamics);
- good algorithmization – the suitability for developing software suites that cover a wide scope of applications;
- good numerical stability of FEM algorithms.

Emergence of personal computers and their increasingly wide use for design purposes impacted the accelerated development and availability of finite element analysis application systems that do not

require the user to be deeply proficient in FEM theory, eliminate labor-intensive operations of manual preparation of initial data and offer excellent opportunities of processing results of mathematical modeling.
T-FLEX Analysis belongs to modern finite element analysis systems, oriented at a wide range of users, who, by the nature of their responsibilities, face the requirement of assessing product behavior under conditions of various physical influences. T-FLEX Analysis is oriented at a nonspecialist in the area of finite element analysis and does not require the user to have in-depth knowledge in the area of mathematical modeling for effective use of the system. Nevertheless, correctness of results of a mathematical modeling and their appropriate assessment are determined to a significant degree by the user's proficient approach to formulating physical problems, which are to be solved with the help of T-FLEX Analysis.
The center point of the finite element method is in replacing the original spatial structure of a complex shape by a discretized mathematical model that appropriately represents the physical essence and properties of the original product. The most important element in this model is the product's finite element discretization - which implies building a set of elementary volumes of the specified shape (the so-called finite elements, FE), combined in a united system (the so-called finite element mesh).
T-FLEX Analysis is oriented at solving physical problems in spatial formulation. The product's mathematical approximation uses its equivalent replacement by a mesh of tetrahedral elements. A tetrahedral finite element is convenient for automatic generation of the computational mesh, since the use of tetrahedra permits a high-accuracy approximation of a however complicated product shape.



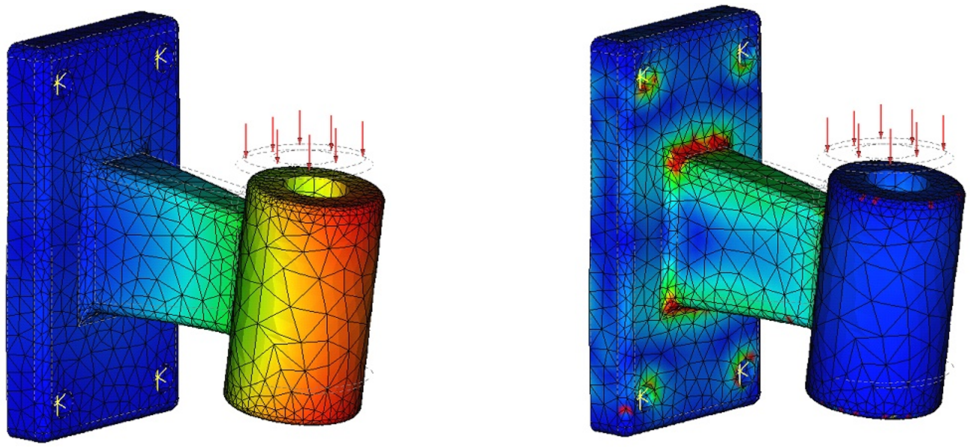


Original structure and its finite element discretization

The structure that itself represents a distributed system of a complex geometrical shape is represented as a union of finite elements. The finite elements that approximate the original structure are considered connected to each other at the corner points - the nodes, in each of which the three translational degrees of freedom are introduced (for mechanical problems). The external loads applied to the structure are converted to equivalent forces applied to the nodes of finite elements. Restraints on the structure's motion (fixings) are also transferred to finite elements that model the original object. Since the shape of each FE is defined in advance, and its geometrical characteristics are known, as well as the material properties, therefore a system of linear algebraic equations (SLAE) can be written out for each FE that is used for modeling the structure, describing displacements of FE nodes under the influence of forces applied at these nodes.
By writing out a system of equations for each finite element that is involved in approximating the original physical system, we study those together and get a system of equations for the entire

structure. The order of this system of equations is equal to the product of the number of movable nodes in the structure and the number of degrees of freedom introduced in one node. In T-FLEX Analysis, this usually amounts to tens or hundreds thousand algebraic equations.

By building the system of equations for the entire structure and solving it, we get the values of the sought physical measure (for example, displacements) in the nodes of a finite element mesh, as well as additional physical measures, for example, stresses. Those values will be approximate (with respect to the theoretically possible «exact» solution of the respective differential equation of mathematical physics), however with the miscalculation error being possibly very small – fractions of a percent on test problems having «exact» analytical solution. The error of the solution obtained as the result of a finite element approximation is usually decreasing smoothly with the increased degree of elaboration on the modeled system discretization. In other words, the greater is the number of FE involved in a discretization (or the smaller are the relative dimensions of a FE), the more accurate is the resulting solution. Naturally, a more dense subdivision of FE demands more computational power.



Results of finite element modeling (displacements and stresses)

The described algorithm of finite element modeling is applicable for solving various problems, which a modern engineer may encounter – heat transfer, electrodynamics, etc. Due to advantages accounted for above, FEM became the leading method of computer modeling of physical problems and, in fact, associates with a whole branch of the modern IT industry, known by the acronym CAE (Computer Aided Engineering).

System Requirements

Mathematical modeling of physical phenomena belongs to the class of the resource-intensive problems that require serious computational resources. That is why, for efficient use of the finite element modeling system it is recommended to use the most powerful computers accessible to the user. Moreover, increase in the dimensionality of the solved problem can be achieved by using 64-bit operating systems.

T-FLEX Analysis is available in two versions depending on the edition of the Windows operating system:

1) T-FLEX Analysis for Windows 32-bit. The distinctive feature of the 32-bit operating systems is the existence of «physical» bound on the maximum volume of addressed information (about 2 GB), which limits capabilities needed for analysis of systems with large number of finite elements.
2) T-FLEX Analysis for Windows 64-bit. This system works on the processors that support 64-bit instructions. Operating systems with digit capacity 64-bit allow the user to address significantly larger volumes of information and solve the problems of higher dimensionality.

Computer parameters for work with T-FLEX Analysis

Minimal	
Processor	Intel or AMD with support SSE3
Hard drive space (for storing calculation results)	3 GB
RAM	2 GB
Operating system	Microsoft® Windows® 7
Graphics card	graphics card with support for OpenGL 3.3 and higher
Recommended	
Processor	Core i7 or similar
RAM	16 GB (and larger)
Operating system	Windows 8.1 x64, 10 x64
Graphics card	NVIDIA or AMD high-performance graphics card with 1gb RAM or larger supporting OpenGL 4.2 or higher

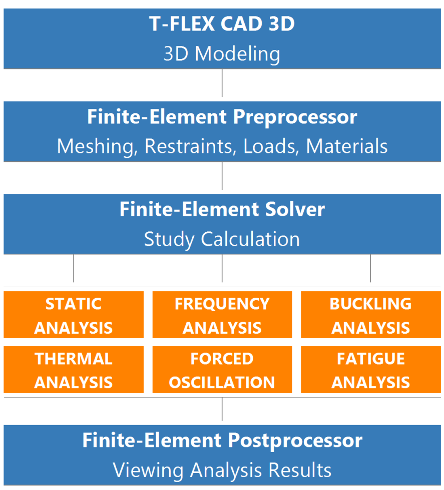
It is not recommended to use integrated graphics cards.

Structural Organization of T-FLEX Analysis

T-FLEX Analysis is organized in a modular structure, which enables the user with a flexible approach to setting up an engineer's work seat. The standard system installation package includes the following modules:

- preprocessor – the module that prepares a finite element model;
- specialized solver - the user can choose one or more out of the available solvers, depending on the posed tasks.
- postprocessor – the module for visualizing and evaluating results.

T-FLEX Analysis Workflow



Steps of Structural Analysis

Any type of analysis is performed in several stages. Listed are the steps required for conducting an analysis. Following are the steps required for running calculations:

1. Build three-dimensional model of the part;
2. Create «Study». A study is created for one or more connected solid bodies («glued» connection);
3. Define more the material;
4. Generate finite element mesh;
5. Apply boundary conditions reflecting the essence of the physical phenomenon being analyzed;
6. Run calculations;
7. Analyze results.

The listed steps of analysis are valid for most types of studies realized in «T-FLEX Analysis» system.
The difference in the respective modeling steps for different types of studies is usually in the types of applied boundary conditions, which depend on the calculation type. For example, in static analysis and in buckling, the role of the boundary conditions is played by the forces and restraints on the product, in frequency analysis – usually restraints only, and in thermal analysis – temperature and heat impact.

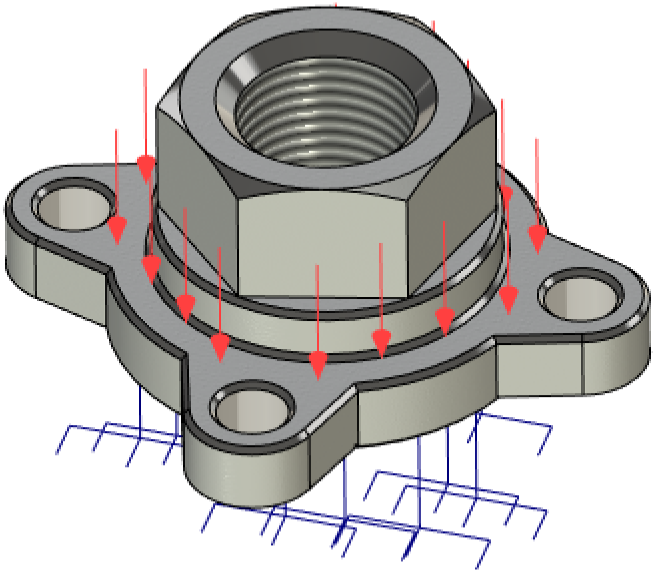
Express Analysis

The express analysis module is a limited trial version of "T-FLEX Analysis", specifically tuned for running simplified strength studies. Express analysis module is included into "T-FLEX CAD" distribution and does not require any additional licences.
A special study type "Express Analysis" is available in express analysis module. The study is a static analysis with certain limitations. It is the only study type available without license for "T-FLEX Analysis".
Data from an express analysis study file can be read using "T-FLEX Analysis". You can also change a study type for it and perform calculations for a more complex analysis model.
An express analysis module has the same menu items as full "T-FLEX Analysis" module but commands that are unavailable for express analysis are disabled.
An express analysis module has the following limitations with respect to the full "T-FLEX Analysis" module:

- Analysis model can include only one element. I.e. you can calculate single parts but you can't calculate an assembly.
- FEA mesh has a simplified parameters dialog. Mesh parameters are specified using "Low/High" controller.
- Only force and pressure loads are available.
- Only full restraint is available.
- User cannot modify calculation options.
- Export of calculation results is not available.
- The following types of calculation results are available: "Displacement, magnitude", "Equivalent Stress", "Factor of safety by equivalent stress"

Quick Start

Let's review a general algorithm of using T-FLEX Analysis system, based on the example of static strength analysis. Suppose, we are required to perform analysis of the strain state of the "Body" structure, whose one face is subjected to a distributed load, and the supporting bottom surface is fully fixed.



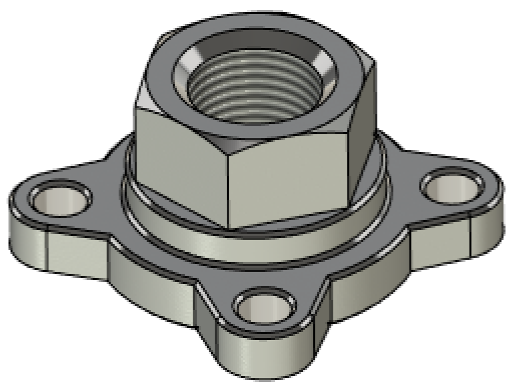
Topics in this section:

Preparing Spatial Solid Model of a Part

For analysis, you need to have a three-dimensional solid model of the part. The user can create the module in the T-FLEX CAD three-dimensional modeling environment. This can be a «working» model, containing projections and complete working drawings, which could be part of an assembly or a subject to calculating numerical sequences for CAM processing.

> For calculation purposes, it is helpful to create in advance a special optimized version of the model (an optimized copy maintaining a parametric relationship with the original). For example, one can delete small features from the original model, which are not significant in the calculation (such as small unimportant holes). In this case, the calculations will run faster, and the finite element mesh can be created easier. To correctly apply loads, it is sometimes necessary to create special "spot" faces at some locations on large faces.

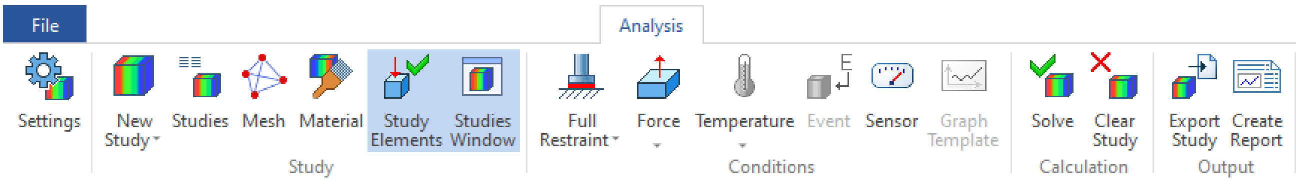
By using the T-FLEX CAD command **File | Import**, one can load into the system a model for analysis, that was created in another spatial modeling system.



Original structure

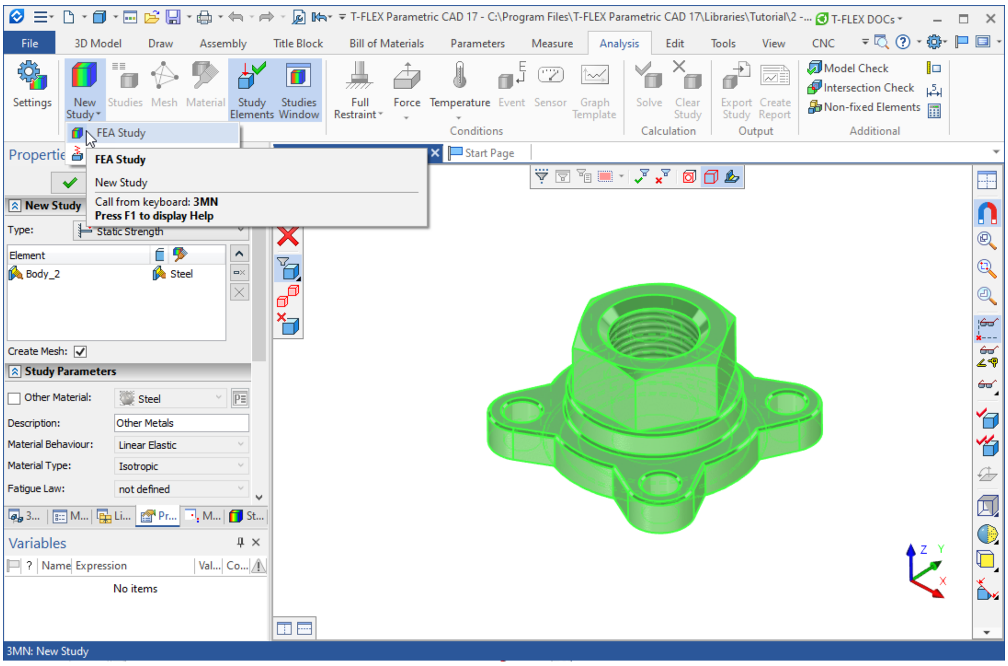
Creating Study

Once a three-dimensional model of the part is built in T-FLEX CAD or imported into the system, you can proceed to preparing the finite element model.
Any type of calculations in T-FLEX Analysis begins with creating a «Study» using the **New Study** command on the **Analysis** tab on the Ribbon. All commands for analysis are located on the tab.



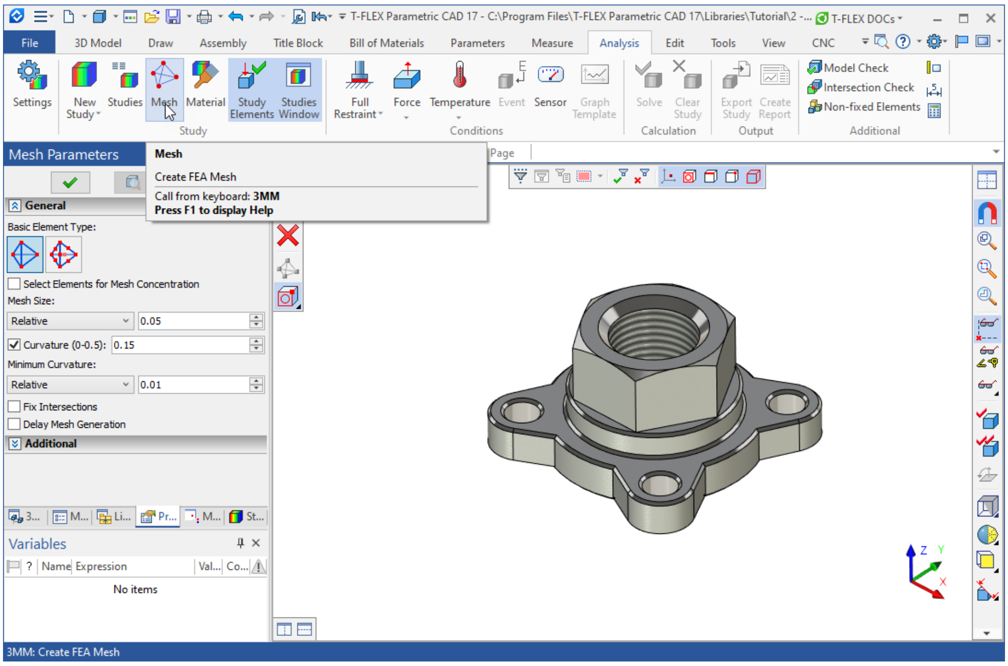
When creating a study, the user defines its type (**Static Strength, Buckling, Fatigue, Express Analysis, Frequency, Forced Oscillation, Mode Superposition, Transitional Process, Stationary Thermal Process, Nonstationary Thermal Process**). Additionally, if more than one solid body is present in the scene, then you need to specify, for which body in the scene you are creating the study.

Let's create a study of the type **Static Strength** for our model part.

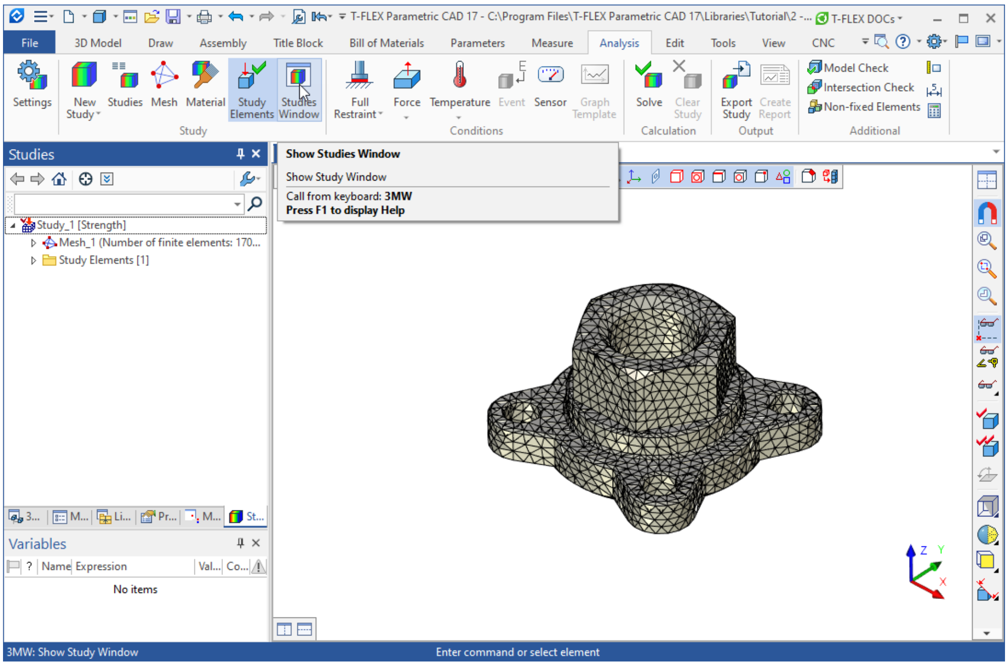


Creating finite element analysis problem

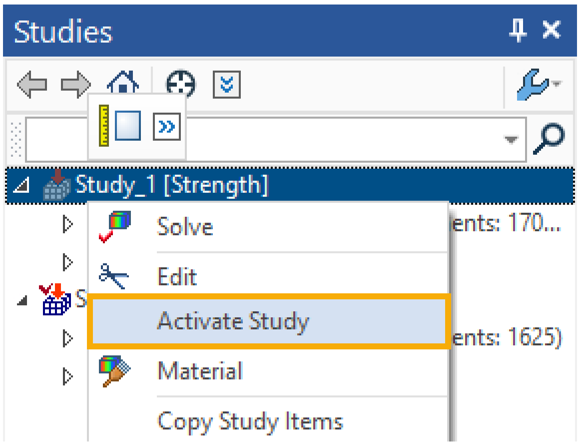
By default, the **Analysis** > **Mesh** command is started automatically when creating a new study. Thus, upon the successful study creation, a dialog appears providing controls for finite element mesh generation; upon the successful completion of the latter, we obtain a meshed model, made of tetrahedra approximating the solid model of the part.



The **Analysis** > **Show Studies Window** command opens the study window, which displays the studies presented in the current document and their elements in a tree view.
The just created study becomes active. The newly created study elements and the issued Analysis commands will pertain to the active study.

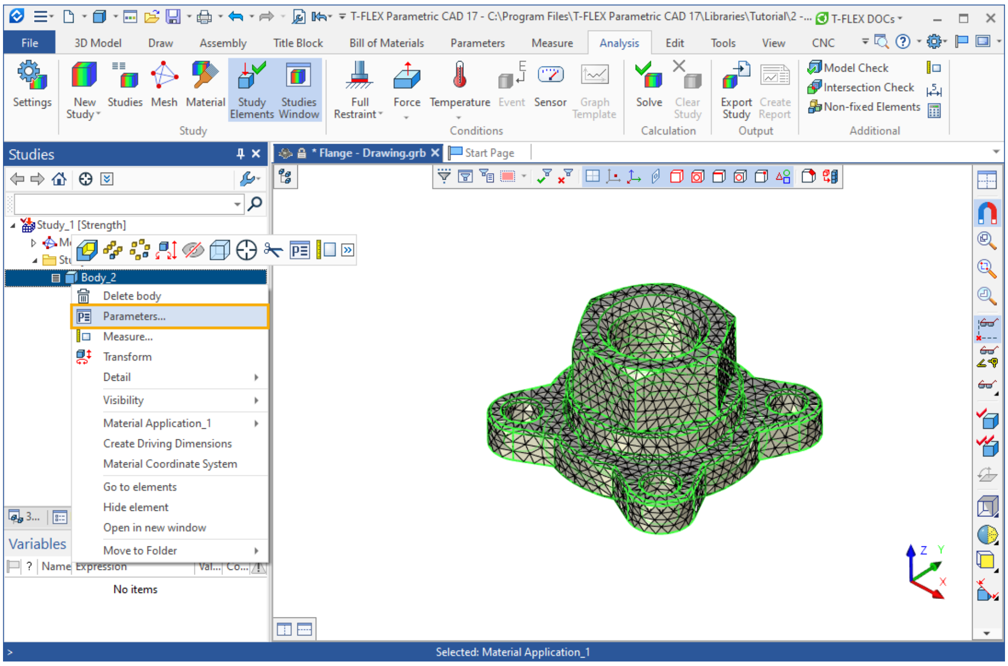


If there are many different studies in the document, then only one of them can be active. Switching an active study is done via the context menu accessed by understudy name. The **Activate Study** command is provided for inactive studies.



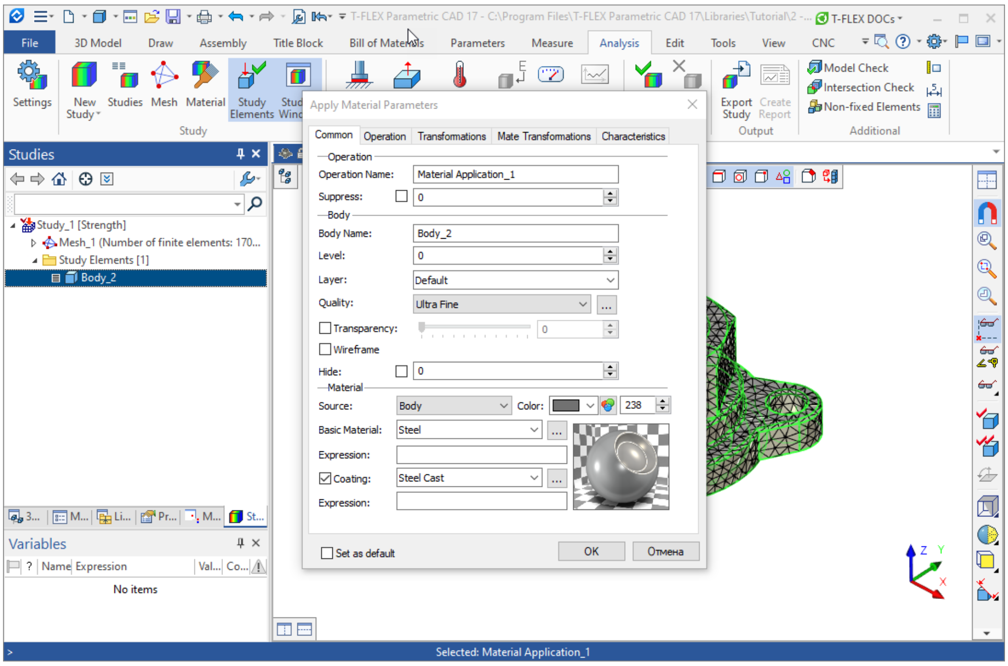
Assigning Material

To perform any calculations with the solid model of a part, you need to define the material, from which it is made. T-FLEX Analysis provides two ways of defining a material for performing analysis. By default, calculations use the material properties «from study Operation/ body». Material assignment for a three-dimensional model is done in the operation's parameters window. To check or modify the material in this case, call the operation's parameters window from the context menu by  on the three-dimensional body, resulting from the operation, or on the operation name in the studies window.

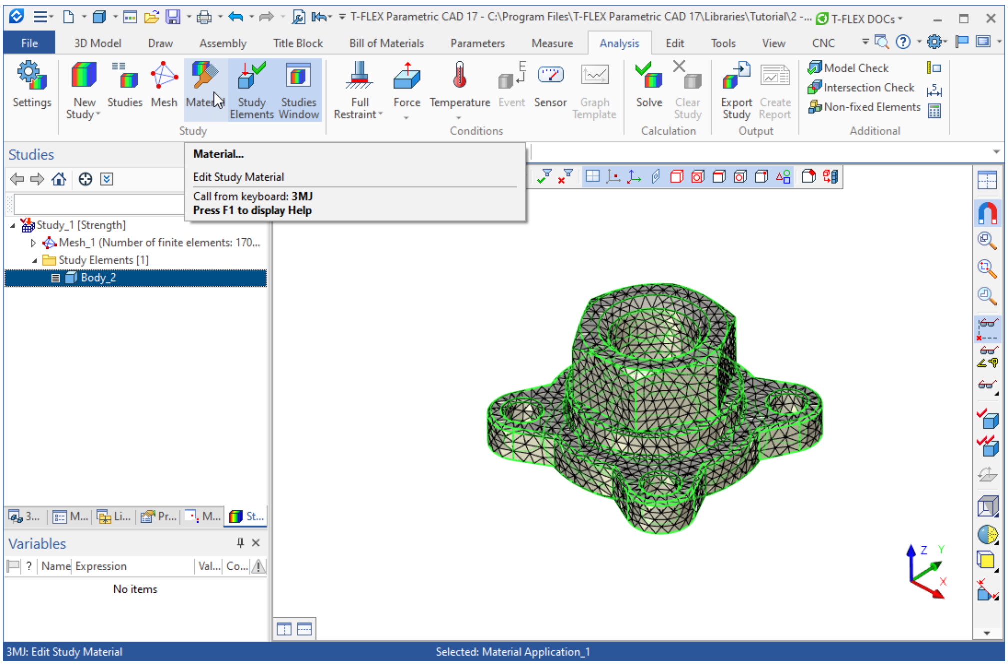


The operation's parameters dialog lets you select a material from the standard T-FLEX CAD material library. If necessary, the user can add to the standard T-FLEX CAD materials database one's own materials and modify properties of any materials in this library.

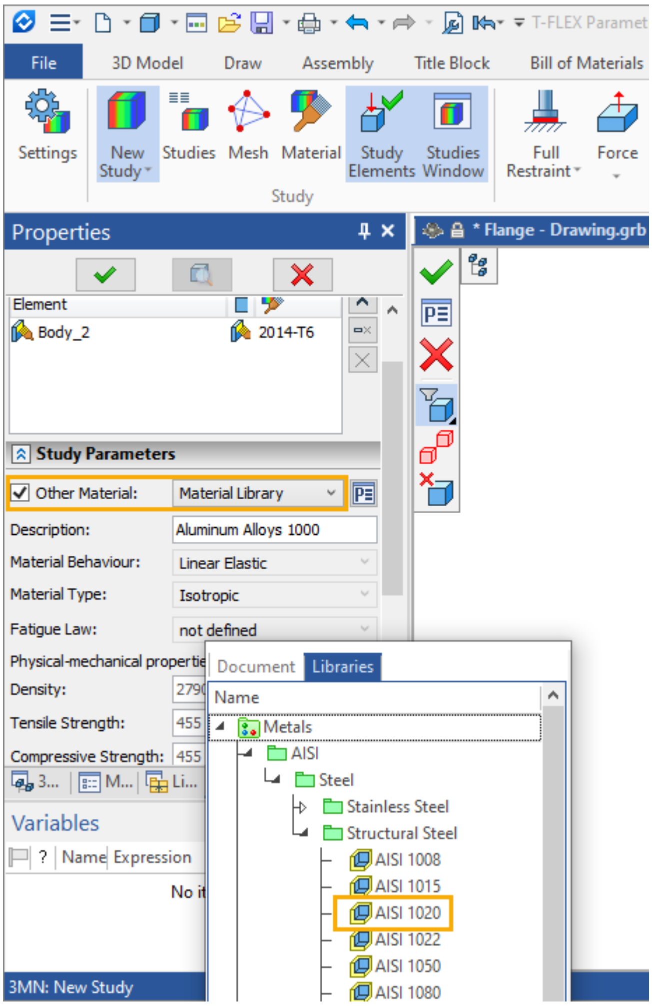
> For detailed information on handling materials in T-FLEX CAD, refer to T-FLEX CAD documentation.



You can select a material for a study from the database of materials in the study parameters window. You can quickly open this window for setting up materials using the **Material** command on the Ribbon panel or from the context menu of the task tree displayed in the task window.



In the parameters window, select the **Other Material** option, select Material Library in the drop-down list and select the required material.
Set for our model the material "Steel/AISI 1020" from the T-FLEX Analysis material base.



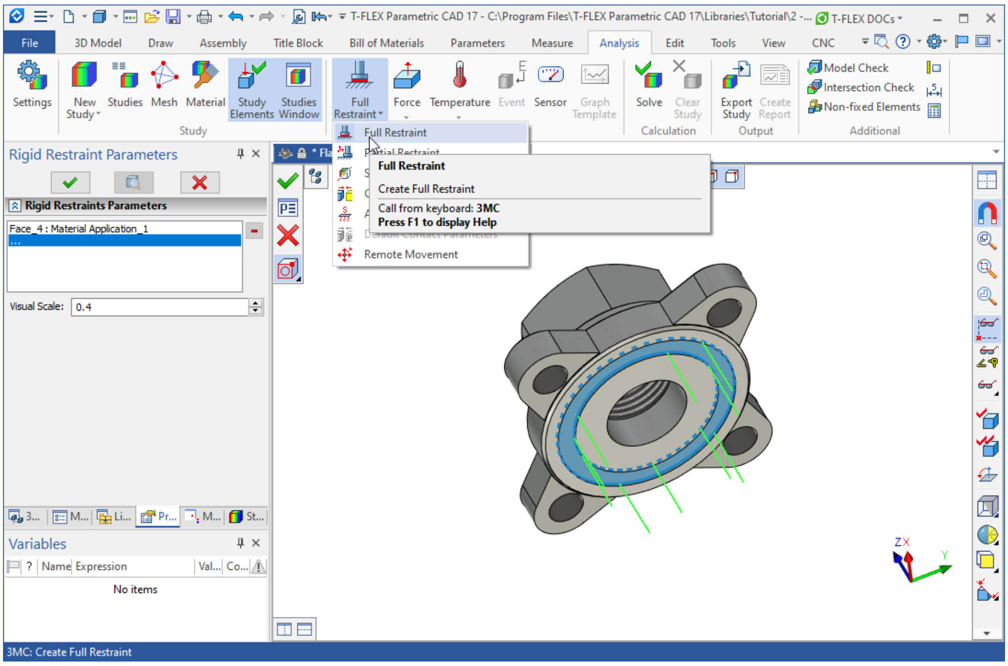
Defining Restraints

In order to successfully solve a physical problem in a finite element formulation, in addition to creating a finite element mesh it is also necessary to correctly define the so-called «boundary conditions». In statics, their role is played by restraints and external loads applied to the system.

The commands are provided in T-FLEX Analysis for defining restraints: Full Restraint, Partial Restraint, Contact and Additional Stiffness.

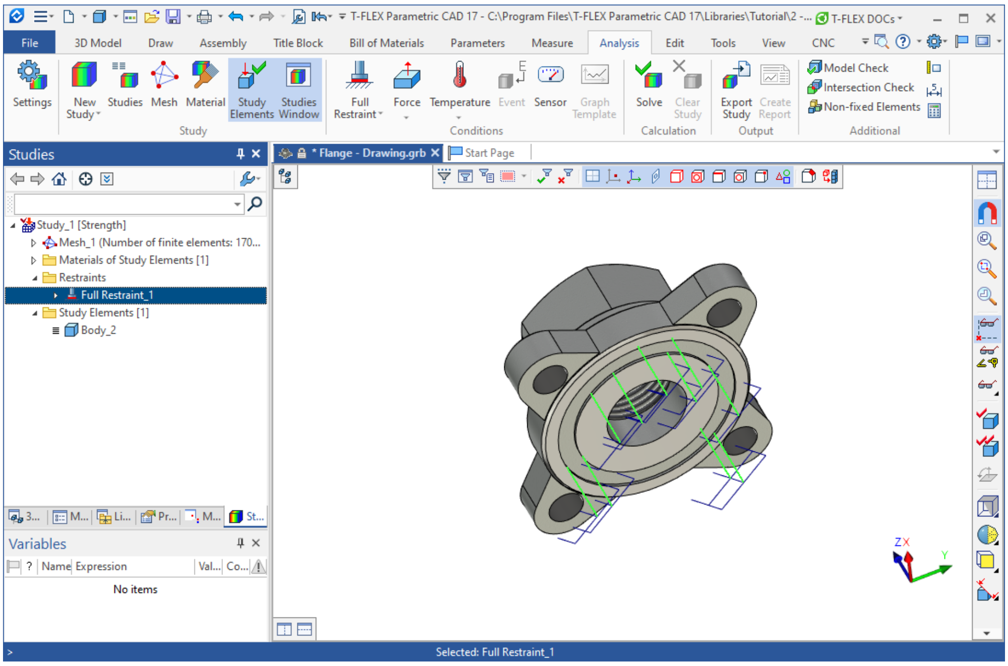
The **Full Restraint** command is used with the model's vertices, faces and edges. It asserts that a given element of the three-dimensional body is fully fixed, that is, it maintains its original position and does not change location under the impact of loads applied to the system.

By using the command **Full Restraint** that located on the Ribbon in the **Conditions** group, specify a fixed face of the model by selecting .



When defining boundary conditions, the finite element mesh gets automatically hidden in order to let you apply boundary conditions to elements of the three-dimensional solid model (faces, edges, vertices).

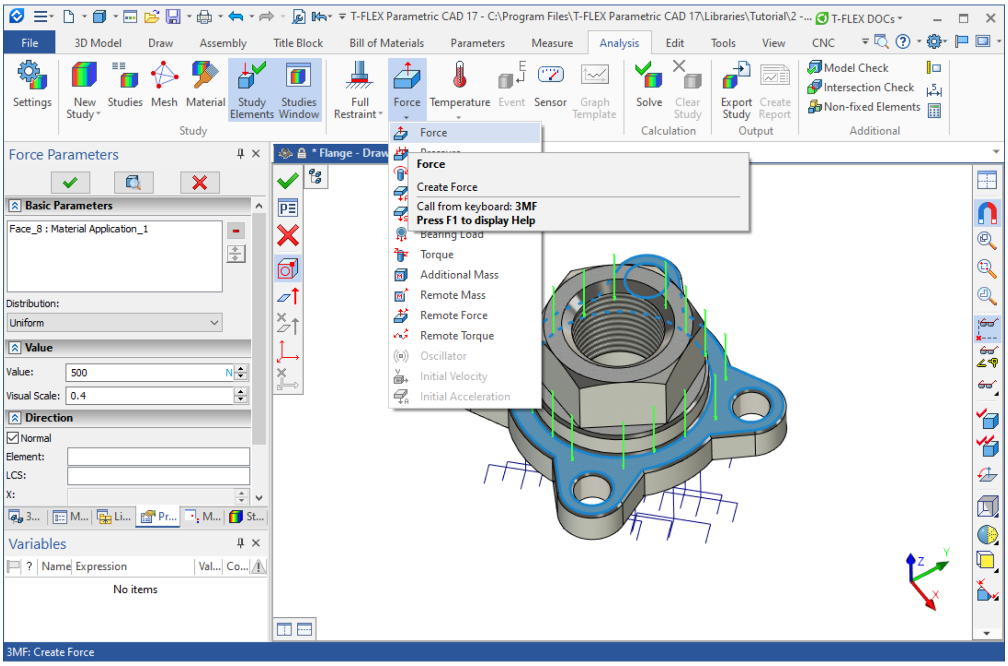
Upon successful completion of the restraints creation command, the corresponding elements are displayed in the studies tree of the studies window, signifying presence of the respective boundary conditions. Restraints on the face are also displayed by special three-dimensional elements (decorations) in the model window of T-FLEX CAD.



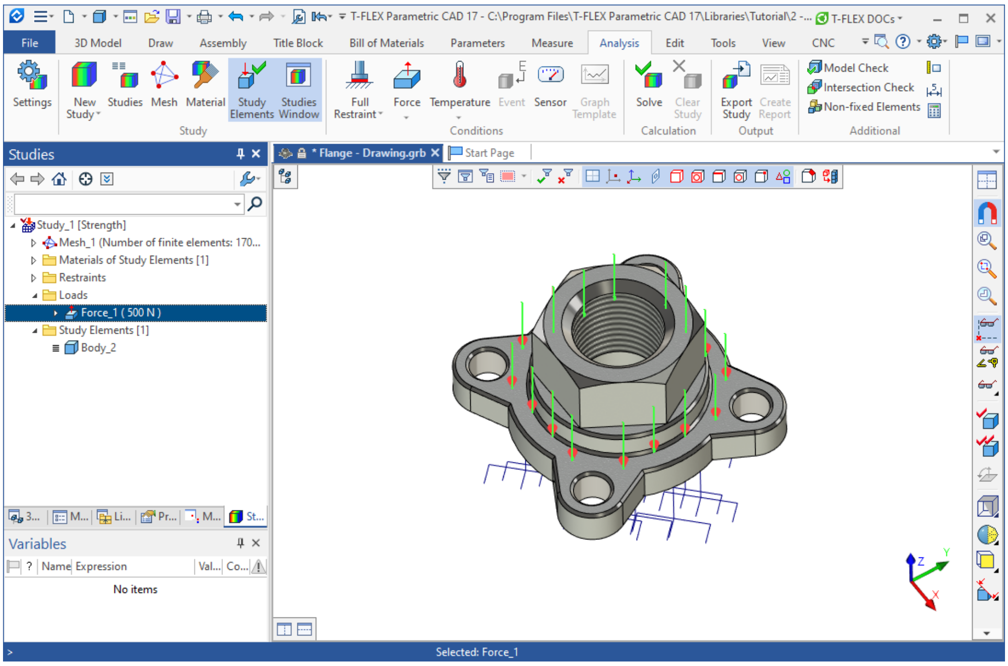
Defining Loads

A set of special commands are provided in T-FLEX Analysis for defining loads, accessible from the **Force** drop-down list of the **Conditions** group on the Ribbon.

Using the Analysis > Force command, select a face of the «Flange», to which the load is applied. In the command's parameters dialog, specify the force value in the **Value** field (500 Newtons). The specified force will be distributed evenly over the selected face. Originally, the force direction is assumed to be normal to the selected flat face. If desired, one can specify the direction vector of the force.



Upon completion of the loads creation command, the introduced loads are shown by special marks on the three-dimensional model of the part, applied to the appropriate model elements.

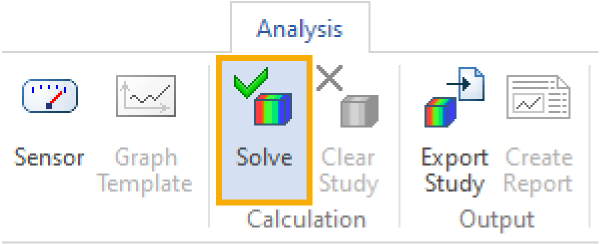


Upon a successful completion of the loads creation command, there are all four elements in the studies tree, required for running the calculations:

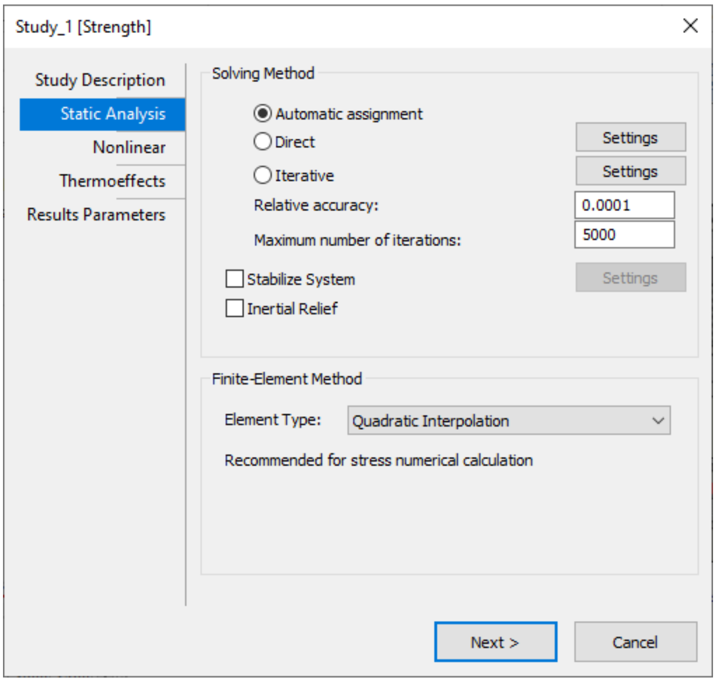
- mesh;
- material;
- restraints;
- loads.

Running Calculations

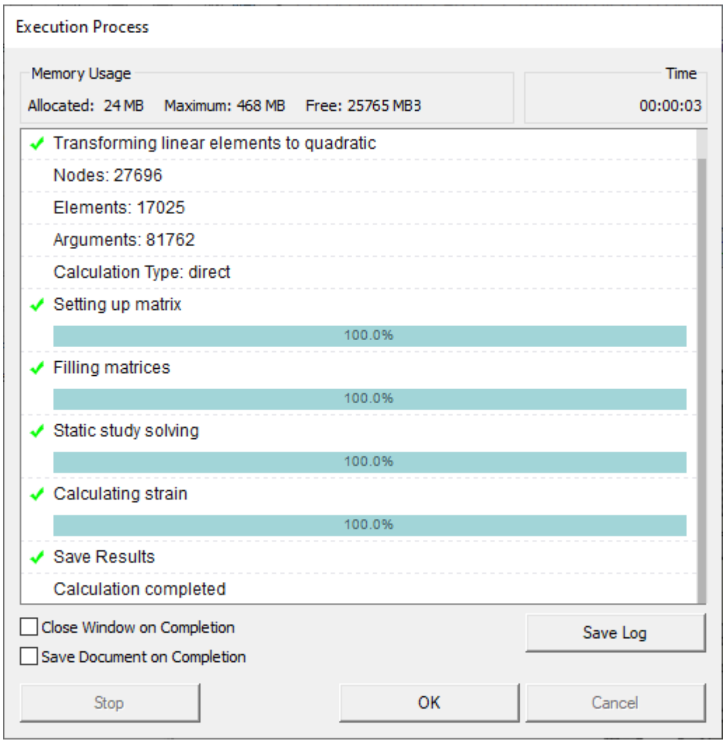
After creating a finite element mesh and applying boundary conditions, you can launch the command **Analysis** > **Solve** and start the process of generating systems of linear algebraic equations (SLAE) and their solving.



The **Solve** command can also be accessed from the context menu of the respective study in the studies tree displayed in the studies window.
The modes of generating the SLAE and methods of their solution are selected automatically by the processor of the T-FLEX Analysis. The user can manually modify calculation options in the study's parameters dialog, which opens automatically before the beginning of calculations.

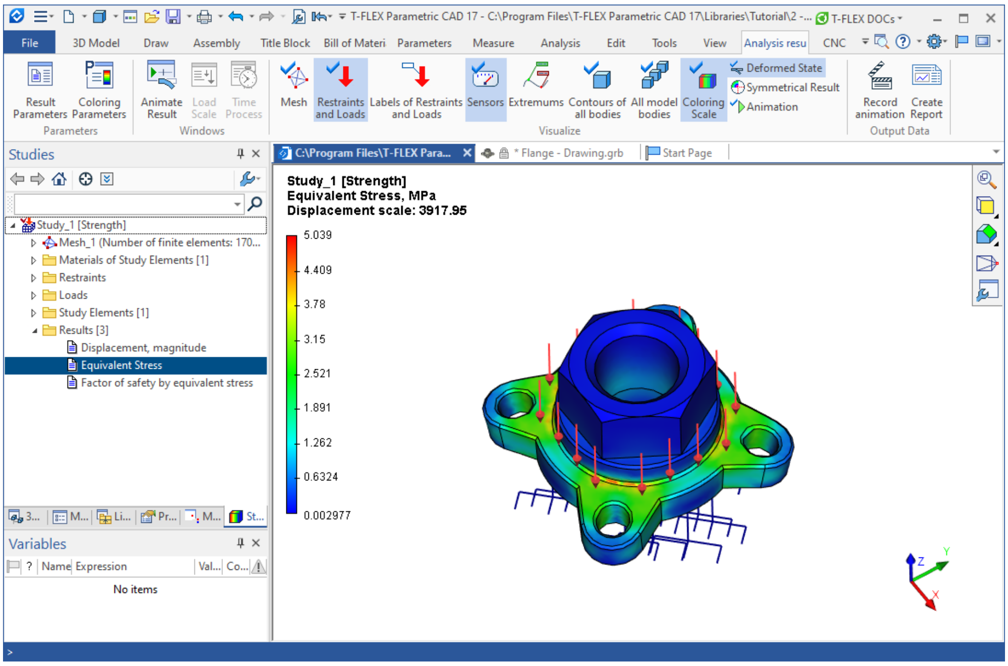


While solving SLAE, a dialogue is provided, that displays solution steps. The process of solving SLAE might take significant time for studies using meshes of a large number of tetrahedra. Once solving completes, the respective informational message is output.

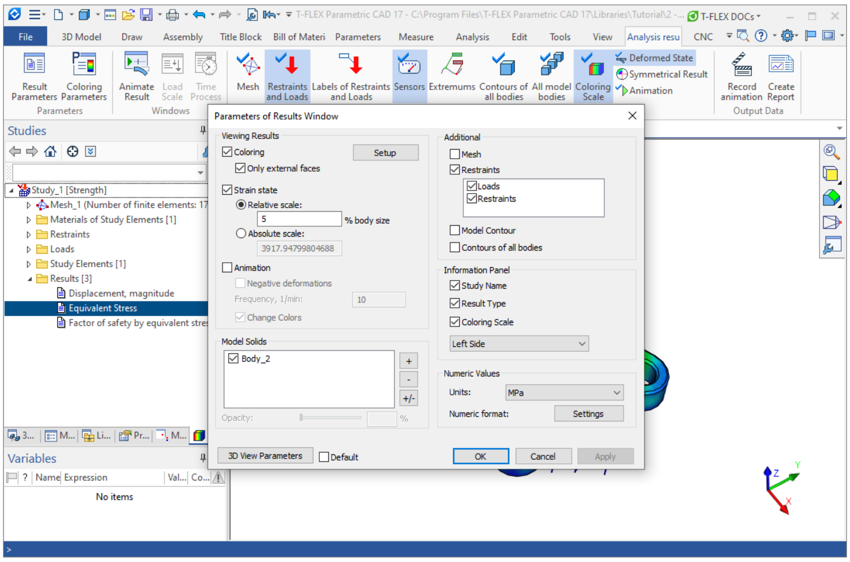


Analyzing Calculation Results

Calculation results are displayed in the studies tree. Access to results is provided from the context menu for the study selected in the studies tree, by the **Open**» or **Open in New Window** command, as well as by . Results are visualized in a separate 3D window of T-FLEX CAD. Several windows with the results from the same or different studies can be opened simultaneously. The user has an access to all zooming and panning commands working on the meshed model with the applied calculation results, just like those used with three-dimensional models in T-FLEX CAD. Additionally, there is a set of specialized commands and options, providing various tools for processing calculation results. Let us briefly mention the most important ones.

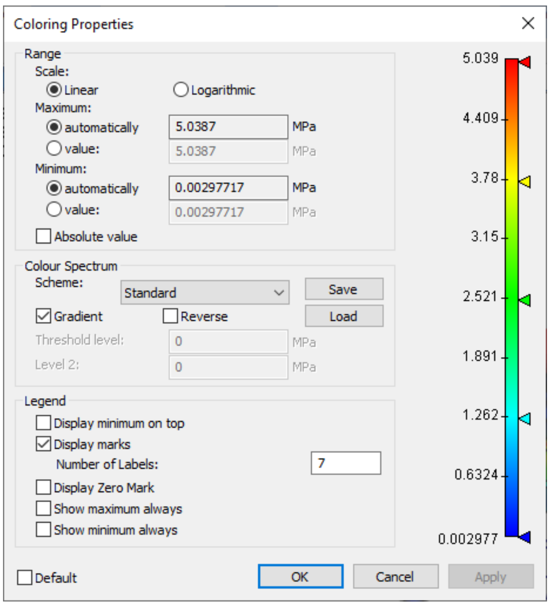


Meshed Model Display Management. The functionality is accessed by double-clicking in the solution viewer window or by the context menu command **Result Properties**. The user can specify various modes of displaying calculation results – over the mesh, without mesh, with or without displaying contour of the original part and other bodies present in the assembly, displaying deformed state, animating the image, etc.
Animation parameter allows recreating the studied model behavior under a smoothly varying loading, with simultaneous display of stress or displacement fields corresponding to the varying load.

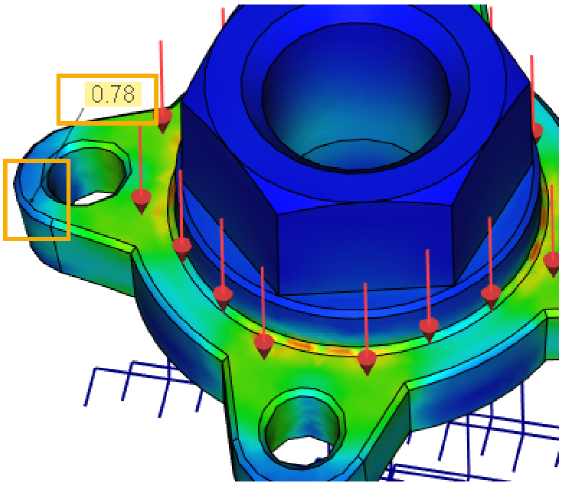


Animation parameter allows you to reproduce the behavior of the model under study under a smoothly changing load, while displaying the stress or displacement fields corresponding to the variable load.

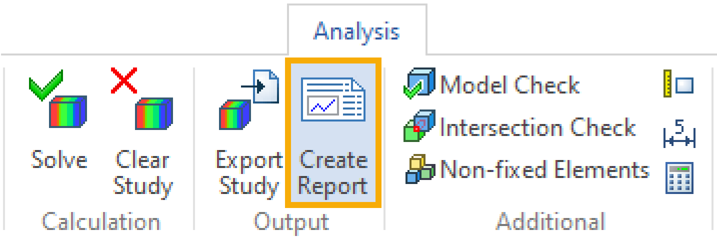
Scale setup. This functionality is accessed by double-clicking on the scale in the results viewer or by the **Coloring Properties...** command in the context menu of the calculation results viewing window. The user has a range of opportunities for setting up display of numerical values. One can use several predefined scale types, and, additionally, a unique capability of setting up a flexible scale with an arbitrary color palette. Also, there is a possibility to specify the minimum and maximum values of the scale, select the display of the logarithmic scale.



Dynamic Result Sampling. T-FLEX Analysis Postprocessor offers a convenient capability for sampling the result directly under the mouse pointer. The user needs to simply point the mouse at the location of interest on the meshed model, and the exact result value will be displayed at that spot. Sampling also works in the mode of displaying the deformed model state. To sample inner portions of the model, you can use a T-FLEX CAD tool called **Clip Plane**.



Creating Report – results of a solved study can be saved as a separate electronic document. The dialog for generating a report of the active study is accessible via the Analysis > Create Report menu or from the **Report...** context menu item of the study selected in the studies tree.



Preparing Finite Element Model

The main purpose of the T-FLEX Analysis Preprocessor is preparing initial data for the physical problem to be analyzed in the form of a finite element model, which would adequately reflect on the geometrical and physical properties of the part being modeled. This finite element model is then processed by the T-FLEX Analysis Processor, which results in a solution to the posed problem. Preparing a finite element model does not require specific knowledge of the finite element analysis from a user. It is conducted on the basis of a geometrical model interactively, using the Preprocessor commands, whose function is described in this chapter. Use of the Preprocessor results in a finite element model of the part, containing:

- finite element mesh;
- materials data;
- initial and boundary conditions, corresponding to the physical problem being modeled.

The order of building a finite element model in T-FLEX Analysis is arbitrary in most cases, meaning that the user can first build the finite element mesh, and then apply boundary conditions, or, on the contrary, first specify loads and restraints, and only afterwards generate a mesh of finite elements. Nevertheless, an unavoidable condition for a proper finite element model is the presence of all its required components – a mesh of finite elements (tetrahedral or rectangular), material properties and external impacts on the system.
The mesh and boundary conditions are visually displayed in the T-FLEX CAD model window directly (as the mesh) or by using special notations (boundary conditions). With this visual representation, the user can assess correctness of the data specified.

Features of working with a parametric model

The T-FLEX CAD model is usually parametric. The advantages of the parametric model can be fully used in the calculation. As a result of parametric changes in the 3D model, all elements of studies (loads, restraints, mesh) associated with the model can be automatically regenerated, and they do not need to be re-defined. Some elements of studies that require noticeable computational resources (for example, grids) are sometimes more convenient to update manually. In order to update any element of the study, it is necessary to select the **Refresh** command in the context menu invoked by pressing on the study element.
When a parent element disappears (for example, a face to which a force is applied), manual editing of the problem element may be required to take into account changes in the model.
After updating the study data, you will need to re-run the calculation to get the actual results.

Topics in this section:

- Types of Finite-Element Models
- Purpose and Role of Meshes
- Types and Role Initial and Boundary Conditions

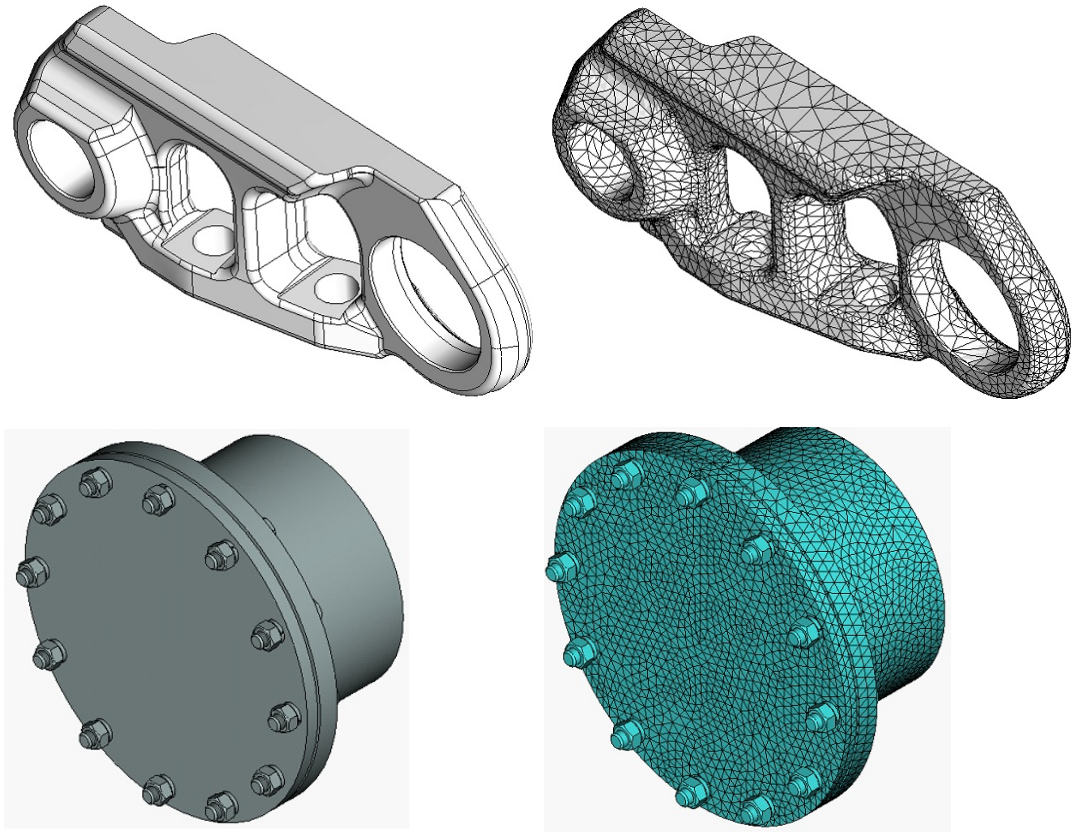
Types of Finite-Element Models

Depending on geometric features of the analyzed structure, in the T-FLEX Analysis it is possible to construct any of three kinds of finite-element models:

- tetrahedral finite element model;
- laminar finite element model;
- hybrid finite element model.

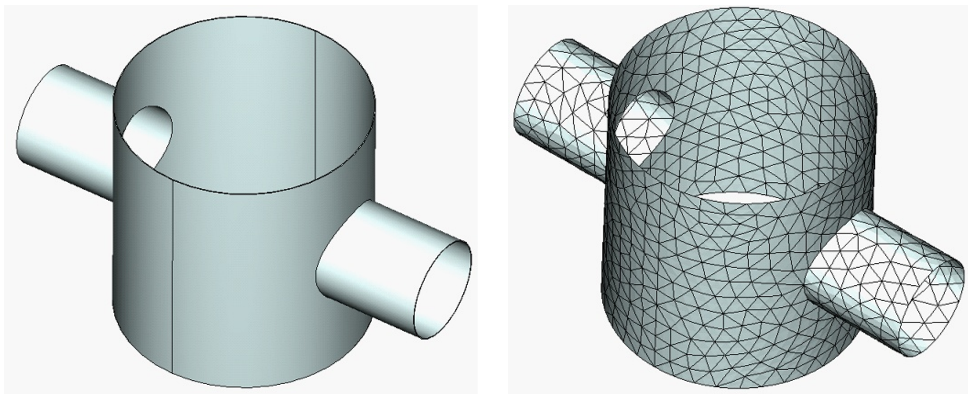
Let us consider the cases of using each type of the finite element meshes in detail.

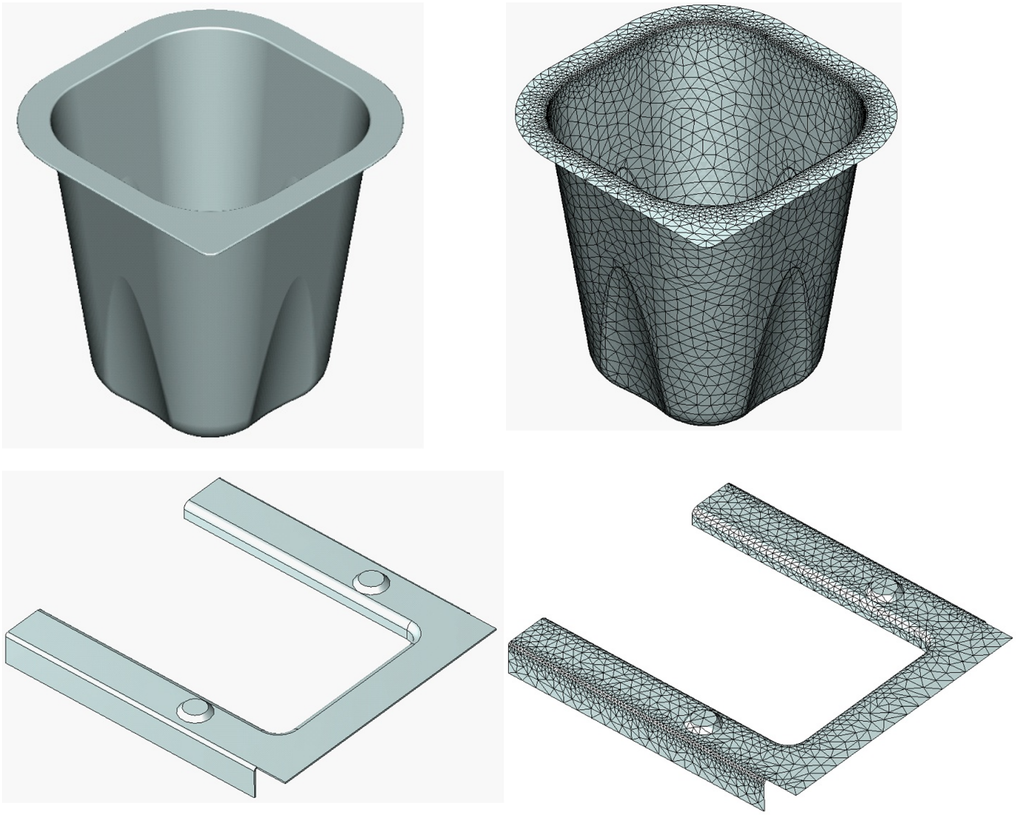
Tetrahedral finite-element model. In this case, to approximate geometry of the modeled part, its representation by finite elements of tetrahedral shape is used. Tetrahedral finite element mesh well approximates the arbitrarily complex shape of parts and provides satisfactory results of modeling physical problems for objects of arbitrary shape, whose characteristic sizes along three space dimensions (length, width, height) are comparable with each other. Most parts and joints of the standard mechanical and instrumental engineering equipment fall into this category.



Typical mechanical engineering objects and their tetrahedral finite element models

Laminar finite element model. Substantial class of structures used in people's life has a special geometric shape when one of the dimensions (thickness) is considerably smaller than two other dimensions – width and length. Such structures are usually called thin-walled. For example, in mechanical engineering these structures can serve as the shells of various machines, spiral of turbines; in instrumental engineering – flexible elastic elements: accordion boots, membranes, including crimp, plate springs; in civil engineering – coatings, floors, ramps, sheds and aprons, in shipbuilding – hulls of ships; in aircraft industry – fuselage and wings of aircrafts; in industry – various tanks: cisterns, reservoirs, etc.

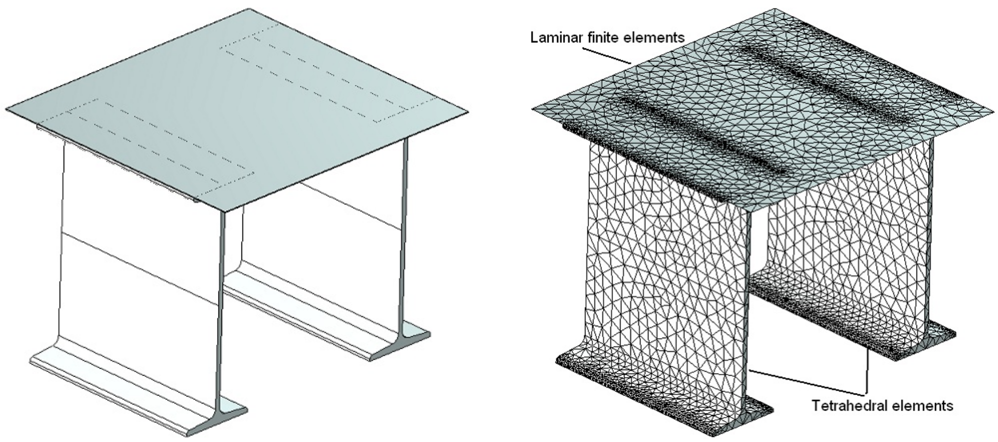




Examples of thin-walled structures and their laminar finite element models

For finite element analysis of thin-walled structures, instead of tetrahedral elements, it is possible to use laminar (shell) finite elements that allow the user to obtain a satisfactory solution with smaller computational effort than when using three-dimensional finite elements.

Hybrid finite-element model includes finite elements of both types simultaneously – parts of the structure corresponding to volumetric bodies, with comparable sizes along three space dimensions, are approximated with tetrahedral elements. Thin-walled parts of the structure are approximated with laminar finite elements.

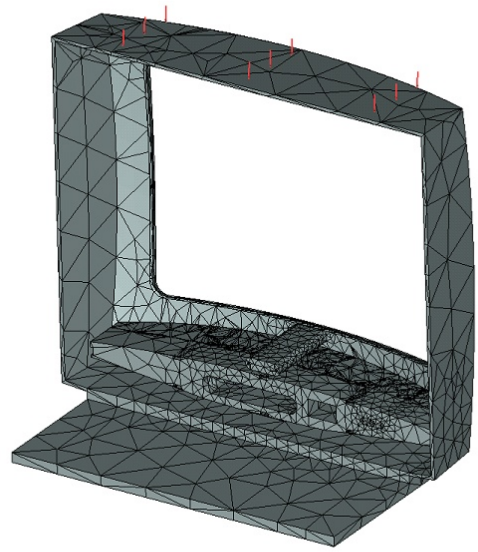


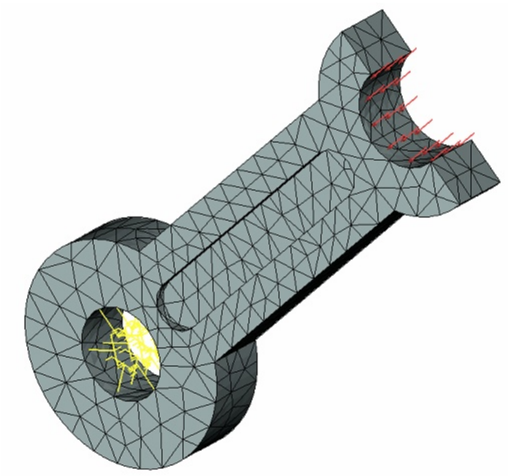
Examples of structures and their hybrid finite element models

Purpose and Role of Meshes

The main purpose of a finite element mesh is to adequately approximate geometry of the body being modeled, accounting for all features of the part geometry significant to the solution. The T-FLEX Analysis Preprocessor uses an effective automatic generator of finite element meshes, which lets the user control various modes of mesh generation in order to obtain meshes of the desired quality on different models. In T-FLEX Analysis, volumetric tetrahedral and triangular surface finite elements are used in finite element meshes, which, in theory, allow approximation with any required accuracy. Nevertheless, there are several preliminary recommendations regarding adequacy of calculation models using finite elements.

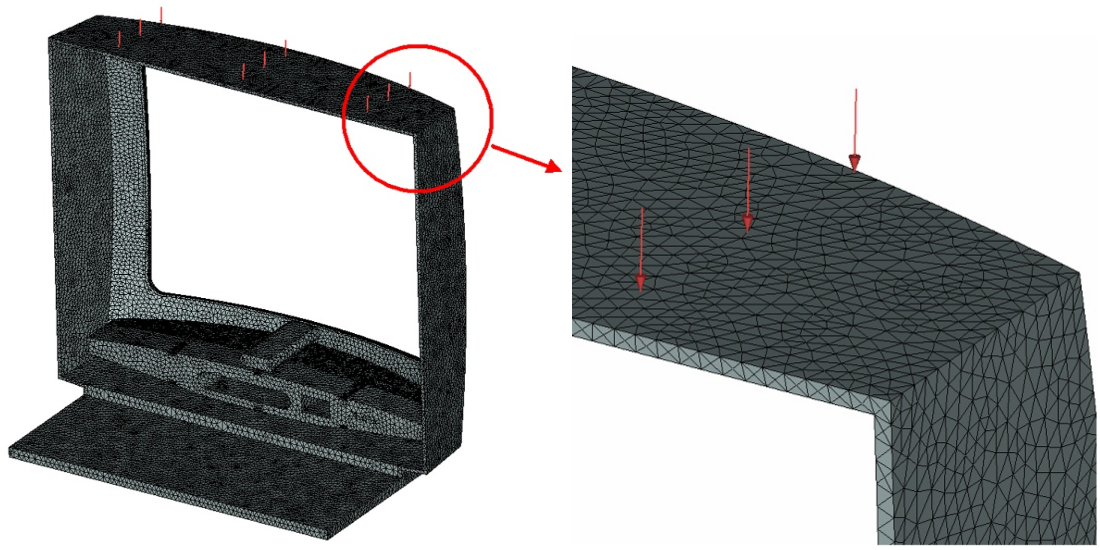
Firstly, quality of a solution may depend on the **shape** of the involved finite elements. Best results of finite element modeling are achieved, if the elements (tetrahedrons and triangles) forming the meshed model are close to equilateral ones. This is especially important for tetrahedral elements. Vise versa, if a meshed model contains elements, whose element-generating edges vary in their size greatly, then the modeling results could be of an insufficient accuracy. In such cases, it is desirable to minimize the number of such improper elements by means of the options provided in the finite element mesh generator.





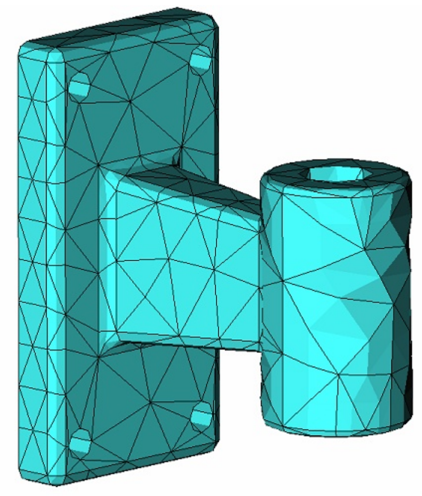
«Poor» mesh of finite element model «Good» mesh of finite element model

Thus, a user needs to control «quality» of the constructed finite element model based on a visual inspection or with the help of **Grid settings**, aiming at possibly more uniform shape distribution of the elements involved in the mesh

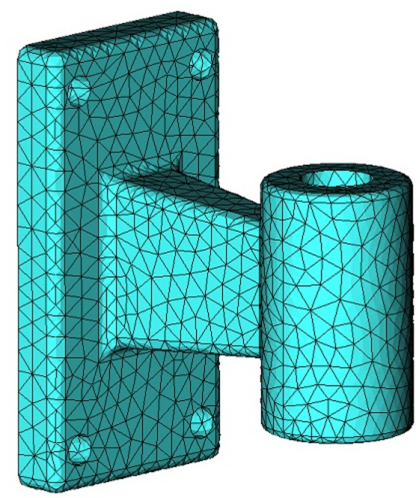


More adequate mesh obtained after using mesh parameters settings

Secondly, besides the shapes of the finite elements, the solution quality is directly affected by the degree of discretization of the original geometrical model, that is «density» of the finite element mesh. The user can control this mesh generator's parameter by specifying a relative or absolute mean size of the finite elements approximating the body geometry, or by varying parameters that affect mesh generation on curvilinear models. Usually, a finer division yields better results in terms of accuracy. Nevertheless, remember that approximating a model by a large number of small finite elements inevitably leads to a high-order system of algebraic equations, which could adversely affect the speed of calculations. Quality of a finite element model can be assessed by subsequently solving several studies with ever-increasing degree of discretization. If the solution (such as maximum displacements and stresses) no longer shows significant difference on a denser mesh, then, to a great certainty, one can regard it as an optimal discretization level, so that a higher rate of discretization is unjustified.



Relative size of 0.2



Relative size of 0.05

> In many cases, consider the estimated minimum level of a body's division as that delivering two to three layers of finite elements in the direction of applying loads and anticipated displacements.

Additionally, the mesh generator provides means for creating user-imposed mesh «refinement» in the areas of the model with sharp variations in the curvature, where one would expect high gradients of the sought values (stresses, for example).
Thus, one should pay much attention to the meshed model being generated for a finite element model, watching that the finite element mesh corresponded to the model geometry and had a satisfactory quality from the viewpoint of insuring a reliable and trustworthy solution to the physical problem being modeled.

Types and Role Initial and Boundary Conditions

Initial conditions are used in studies that use time dependencies and vary according to the type of calculated physical study in the following way:
The following initial conditions are used for the **Mode Superposition** and **Transitional process** study types:
Specified in text menu or ribbon:

- Initial velocity,
- Initial acceleration.

Specified in the study parameters:

- Initial displacements from static analysis,
- initial displacements from any dynamic analysis step.

The following initial conditions are used for **Nonstationary thermal processes studies**:
Set in text menu or ribbon:

- Initial temperature.

Set in study parameters:

- Initial temperature from results of stationary thermal analysis or any step of the nonstational thermal analysis.

Boundary conditions

Boundary conditions differ, depending on the type of the physical problem being modeled, as follows.

In the case of the **Static Strength**, Buckling problem types, the following represent boundary conditions:

- «Full Restraint» restraint;
- «Partial Restraint» restraint;
- «Contact» restraint;
- «Symmetry» restraint;
- «Additional Stiffness» restraint;
- «Remote Movement» restraint;
- «Force» load;
- «Pressure» load;
- «Centrifugal Force» load;
- «Acceleration» load;
- «Bearing Force» load;
- «Torque» load;
- «Additional Mass» load;
- «Remote Mass» load;
- «Remote Force» load;
- «Remote Torque» load;
- «Temperature» thermal load.

In the case of the **Forced Oscillation** problem types, the following represent boundary conditions:

- «Full Restraint» restraint;
- «Partial Restraint» restraint;
- «Contact» restraint;
- «Additional Stiffness» restraint;
- «Force» load;
- «Centrifugal Force» load;
- «Pressure» load;
- «Acceleration» load;
- «Bearing» load;
- «Torque» load;
- «Oscillator» load.

In the case of the Thermal Analysis problem type, the following represent boundary conditions:

- «Temperature» thermal load;
- «Initial Temperature» thermal load;
- «Heat Flux» thermal load;
- «Convection» thermal load;
- «Heat Power» thermal load;
- «Radiation» thermal loa;
- «Thermal contact» constraint.

The essence of a physical problem is determined by the type of initial and boundary conditions applied to the system. To obtain a correct and trustworthy solution, the user needs to imagine well the physical side of the phenomenon being analyzed, in order to specify initial and boundary conditions corresponding to real conditions affecting the product in its life cycle. The result of solving a study will be fully determined by the composition and parameters of conditions, specified by the user. A solution could be obtained that does not reflect on the essence of the physical phenomenon being analyzed, if the user fails to interpret correctly the meaning of a mechanical or thermal load or restraint. Note that the process of designating conditions cannot be totally automated, therefore the user is charged with the responsibility of correctly applying loads and restraints on the system, from the prospective of the physically solvable problem.

Managing Studies

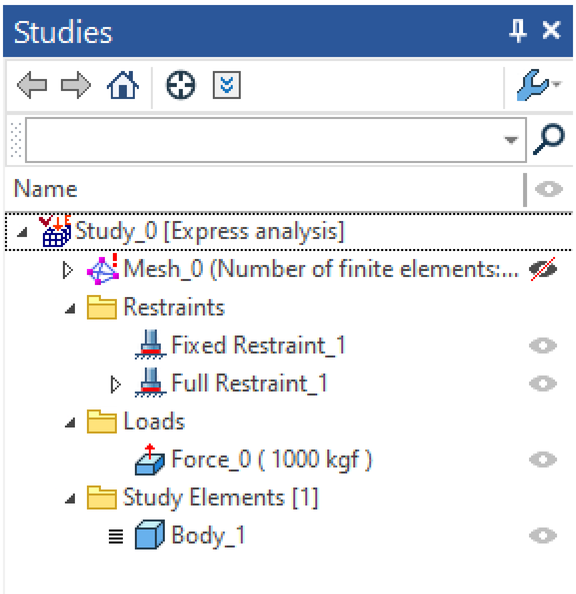
To create a new study, use the command New Study:

Icon	Ribbon
	Analysis > Study > New Study > FEA Study
Keyboard	Textual Menu
<3MN>	Analysis > New Study > FEA Study

Study is a special system object uniting data and elements required for running a specific calculation of a model. A study contains necessary settings of calculation parameters, as well as information on the used objects (solid bodies and/or shells), on the basis of which the finite element model is built, loads, restraints and finite element mesh. After completing calculations, the study also contains solution results. The type of a calculation to run is specified for a study: static, frequency, thermal, buckling analysis.

A special **Studies** tool window is provided for handling studies (the same functions are provided in the **3D Model** window).

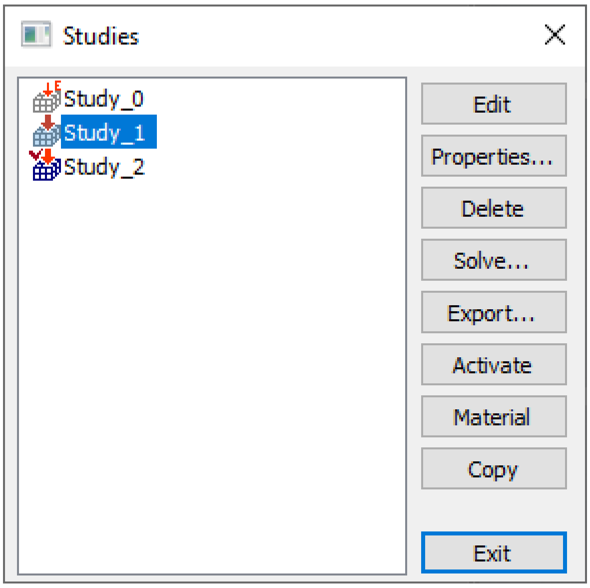
The studies window displays in a tree layout complete information about prepared studies within a given document, and about all elements included in each study. The window provides a quick access to elements of each study. Each type of study, as well as every study element, is marked with a specific icon. Some study elements (loads, restraints, results) are joined into groups.



A special command is provided for managing the list of studies:

Icon	Ribbon
	Analysis > Study > Studies
Keyboard	Textual Menu
<3ML>	Analysis > Studies

The dialog window of this command displays the list of all studies existing in the current document. The buttons for calling main commands are to the right of the list.
To quickly create similar studies of the same type and for the same model (for example, to compare solution results on different meshes or with different material), one can use the study copying functionality.
By default the option **Copy mesh** is turned on, that is, the study is copied with all elements contained in it, except the results. If this option is turned off, the finite element mesh will not be copied, i.e., for the considered studies the mesh will be common.



It is possible to change the type of the copied study. It should be borne in mind that after changing the type of problem, the available calculation results will become irrelevant.
Several studies can be created in one document for running different calculations. The study currently being worked on is called active. The active study's icon has a red check in the studies window. To make another study active, use the context menu command **Activate**. Working in the studies window is done using the context menu that provides all necessary commands. The contents of a context menu depend on what study element was right-clicked .

Topics in this section:

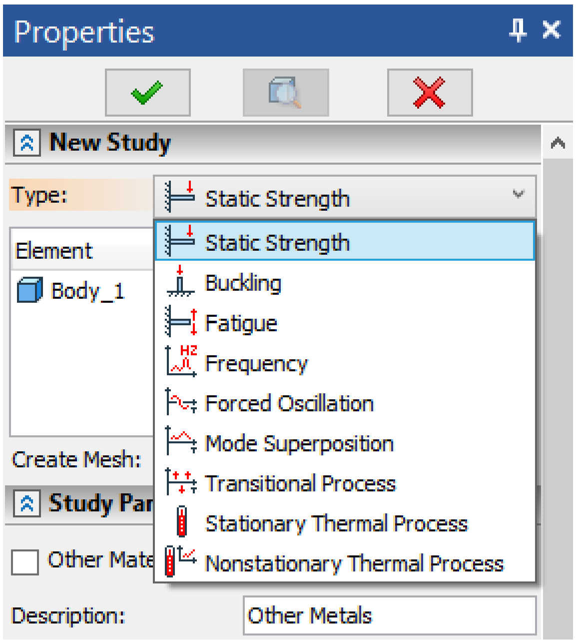
- Creating a Study
- General Parameters of Studies
- Defining Material
- Meshing
- Customization and Utility Commands
- Loads and Restrictions Actuality
- Export
- Visual Scale

Creating a Study

To create a new study, use the command:

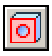
Icon	Ribbon
	Analysis > Study > New Study > FEA Study
Keyboard	Textual Menu
<3MN>	Analysis > New Study > FEA Study

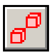
After calling the command, you can select the type for the study being created in the parameters window. The study is created on the basis of one or more solid-creating operations. If the scene contains a single body, it is selected automatically



If there are more than one suitable objects in the scene, then the user shall select desired ones. Selection of objects can be carried out with the help of the option:

 <E> Select Body

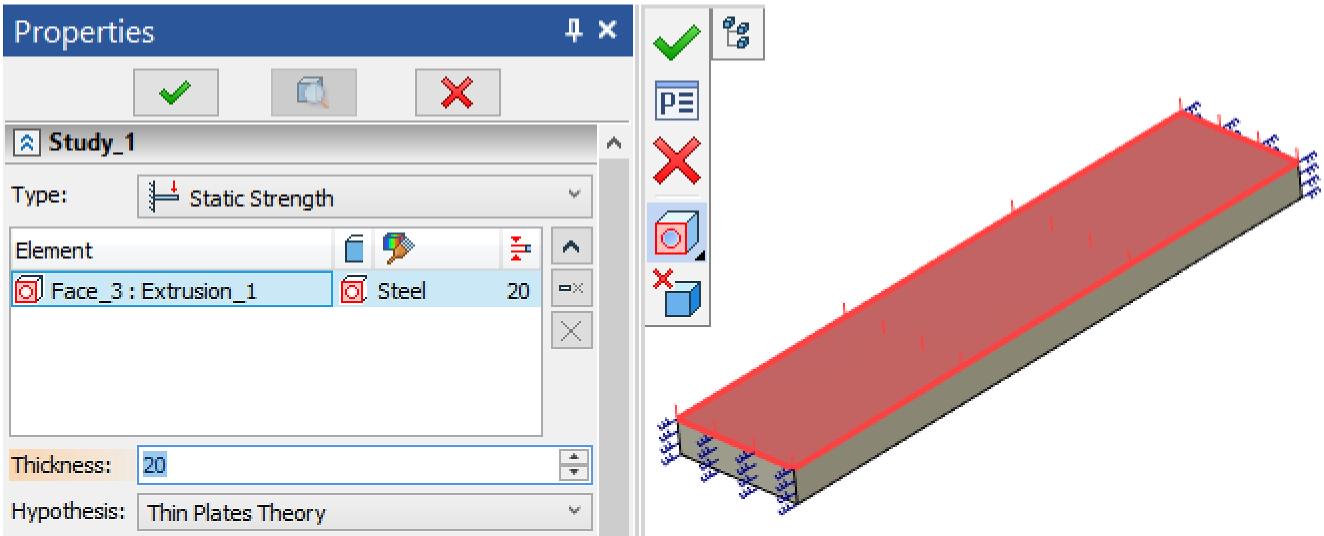
 <E> Select Face

 <S> Select All Elements

The user can cancel selection of all objects with the help of the option:

 <R> Cancel selection

When analyzing thin-walled structures, the user can determine which fragments of the model have to be discretized with laminar (triangular) finite elements and which fragments – with tetrahedral finite elements. That is why, for **Elements of Study**, which will take part in calculation, it is required to select the faces and/or bodies. For each selected face, specify the **Thickness**.



Also, for faces and surfaces, you can select a calculation hypothesis: **Hypothesis** parameter. There are two options available:

- **Thin Plates Theory**;
- **Thick Plates Theory**.

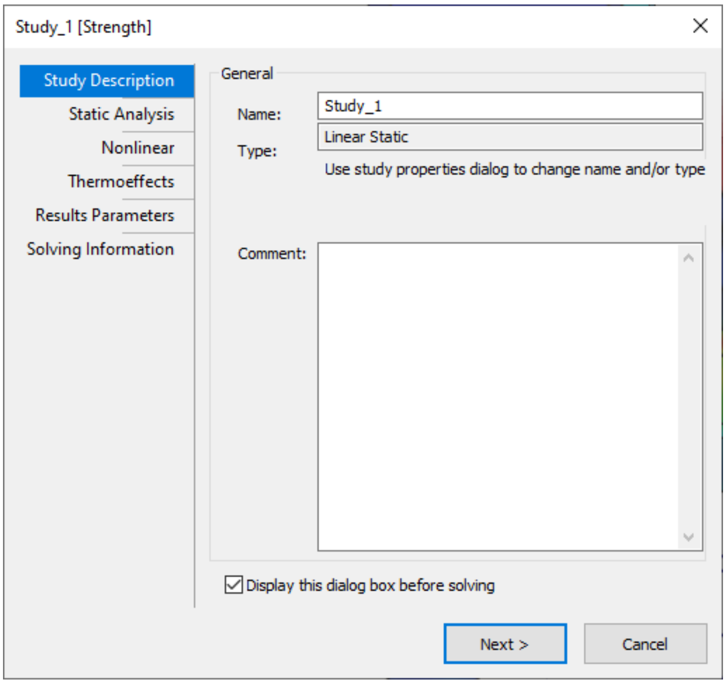
The system allows using several selected elements of the study in analysis, including elements of different types – bodies and faces (in this case the result is the so-called «hybrid model», consisting of shells and solid bodies). Given that, all elements of the study are treated as a single whole (similar to a glue joint), and one mesh is calculated for them. That is why, every element of the study necessarily has to be contiguous with at least one of the remaining elements taking part in analysis, and these elements cannot penetrate into each other.

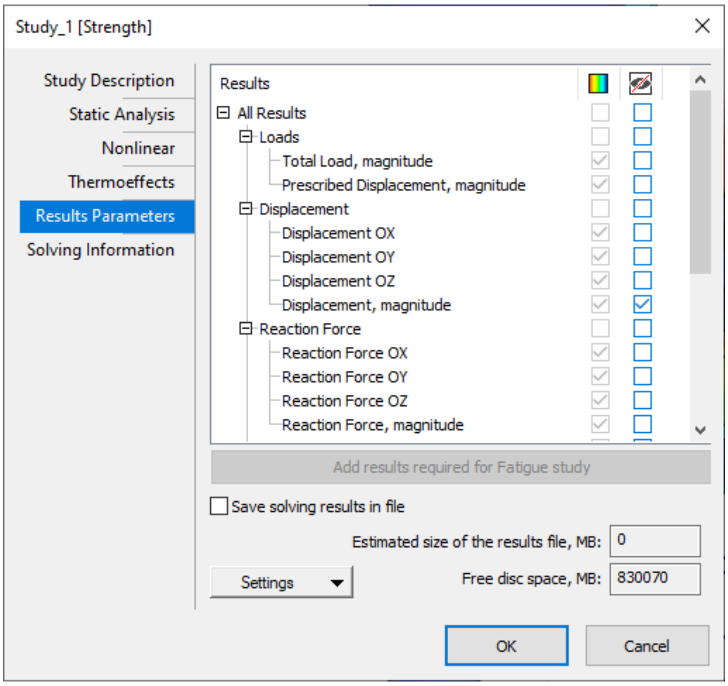
For each element, the material properties can be specified.

A created study gets a certain set of settings, new parameters and solving methods – depending on the chosen type. Other study settings can be edited after its creation in the parameters dialog. A study's parameters dialog box may automatically appear before running calculations or when calling the respective command from the context menu, called by right-clicking the study in the studies window, as well as from the studies list management window.

General Parameters of Studies

A number of similar properties exists in all types of studies, defined on the **Study Description** and **Results Parameters** tabs in the parameters dialog.





On the **Study Description** tab the user can specify the study's name, modify its type (Static Analysis, Frequency Analysis, Stability Analysis, Thermal Analysis) and enter the comments. Comments are used for recording necessary explanations, and are output at the time of generating a report.

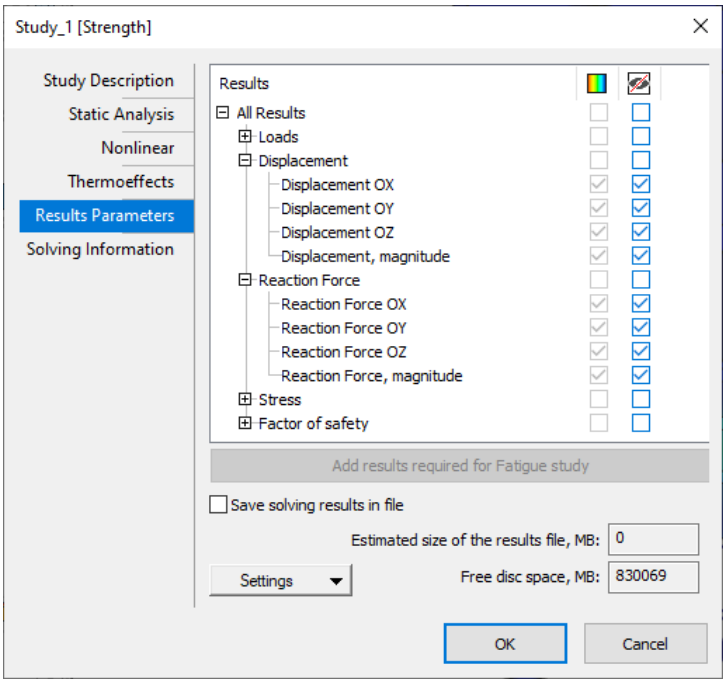
We recommend turning on the **Display this dialog box before solving** flag. This allows specifying study parameters and adjusting calculation algorithms before the execution.

On the **Results Parameters** tab the option of saving the problem's calculation results in a separate external file can be enabled. By default, calculation results are saved in the main T-FLEX document file (with extension .grb). When the external storage of results is enabled, the file with the name «Document name_Study's name» and extension .tfa is created in the folder of original document.

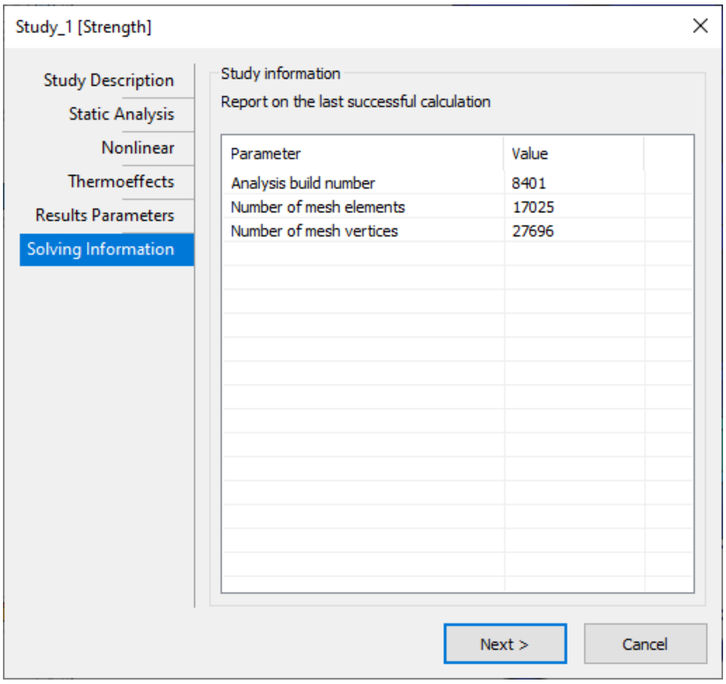
This option can be useful when solving problems with large volumes of results, and also can speed up the saving of a document (this depends on operating system).

By clicking the **Results Parameters** button, you can configure the list of results that will be added to the study tree after the calculation is completed.

The user can set checkmarks against any item if one is planning to investigate the corresponding result in the future. The marked items will be output in the studies window. The desired results could further be loaded into the calculation results view window.
The user can customize the results list either before or after calculations are completed. The total calculation run time does not depend on the number of output results. The system will calculate all results anyway, but will display in the studies window only those selected by the user.



On the **Solving Information** tab we can see different information about the current problem after its solution: the start and end time and date of calculation, the solver version number, number of finite elements, dimensionality of the problem, runtime and other calculation parameters. This data is stored inside the problem and allow the user to easily find out when exactly and with what method the saved problem has been solved.

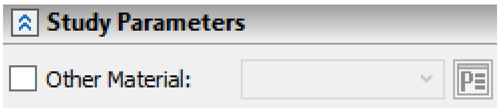


Sets of settings and parameters corresponding to different types of studies are described in more detail in the corresponding sections.

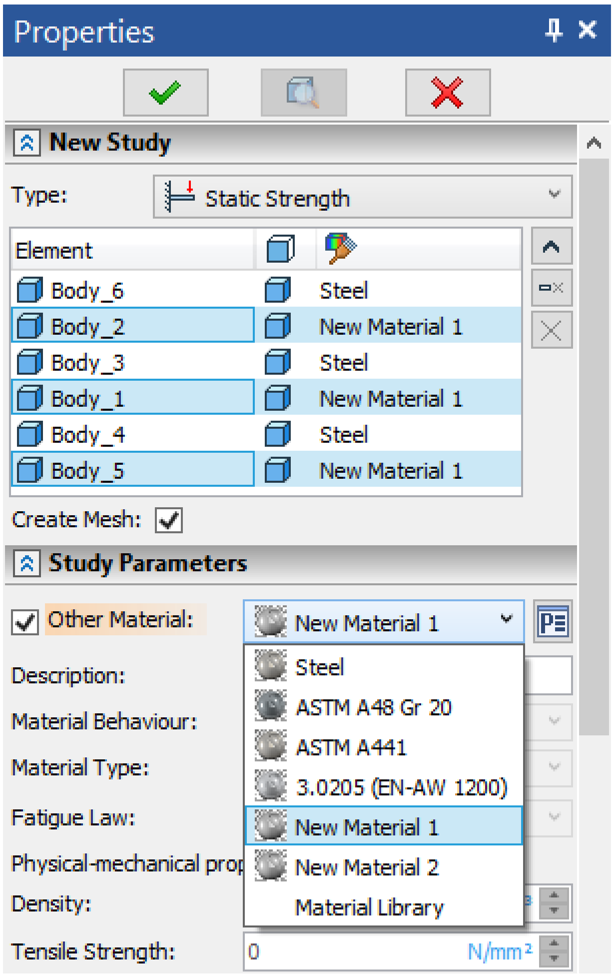
- Static Analysis
- Fatigue Analysis
- Frequency Analysis
- Forced Oscillation
- Dynamic Studies
- Thermal Analysis

Defining Material

The assignment of the material for the study element is carried out when creating or editing the study on the **Study Parameters** tab. By default, the material of the study element is inherited from the CAD model.



For each element of the study, you can assign your own material. Also, you can assign a material to a group of elements at the same time - for this, you need to select them in the list of study elements. The option **Other Material** activates the material selection list of the document from which the material is assigned for the selected study elements.



> The material of the study element is always the material of the document.

The **Material Library** option, available in the list of document materials, opens a special window for quick viewing of document materials and libraries. The window is created to quickly add the library material to the document materials in automatic mode, with the subsequent assignment of an

element (or several elements) to it as a material. Having selected the material of the library in the material viewer on the **Libraries** tab, it will be automatically added to the materials of the document and assigned as the material of the study element.
Document materials are local materials whose properties are defined only within the current document. In order for the document material to be dependent on the external library, it must be linked to the library material using a special command in the **Materials** window.

> Each document material can be associated with the specified library material and its properties can be read automatically.

To edit a study in order to assign materials to study elements, a special command is provided.

Icon	Ribbon
	Analysis > Study > Material
Keyboard	Textual Menu
<3MJ>	Analysis > Material

The command is not obligatory for use when assigning materials to study elements, since is an additional entry point into the study editing command. The command is intended to facilitate navigation.

Topics in this section:

Materials

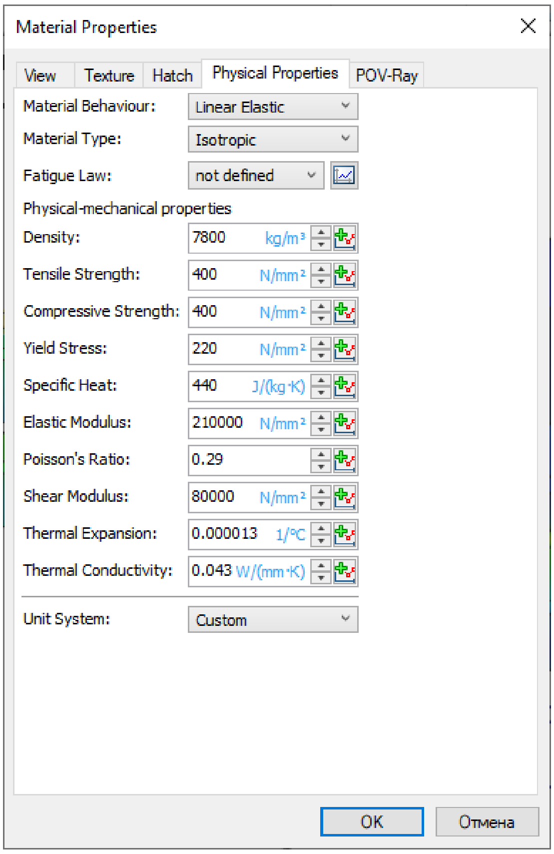
Material – is an element of the T-FLEX CAD. It contains the list of characteristics of a real material which the user deals with in real life.
Characteristics of material can be conditionally divided into two groups. Characteristics of the first type are the ones affecting the display of three-dimensional objects in a 3D window. Characteristics of the second type – are various physical parameters of material such as density, elasticity modulus, strength limit in tension, etc. The characteristics of the second type are necessary for carrying out calculations.
A part's response to loading depends on what material it is made of. The program needs to know elastic properties of the material, from which the part consists. The program supports **isotropic** and **anisotropic** materials, which can be **orthotropic** and **transversely isotropic**. Moreover you can set dependencies of material properties from temperatures.
By default, material properties used for a study's calculations inherit from the subject operation's parameters. Specifying an operation's material is described in the **T-FLEX CAD** help.

Consider Dependence of Material Properties from Temperature

You can specify dependency from temperature for any material property using graphs.

To set dependence of material property from temperature you need to add graph using the  button to the right from the parameter. After that, the graph editor is opened. Here you can set the dependence. More information about that can be found in Using graphs to set dependences from time or temperature section.

You need to activate option **Consider dependence of physical properties from temperature** on the **Thermoeffects** tab in the parameters of the current study to consider temperature dependences.



Additionally, there is an alternative approach to specifying a study's material properties. Isotropic materials are such that physical properties of the material (modulus of elasticity, Poisson's coefficient, coefficients of heat conduction and linear thermal expansion) are considered invariant with respect to orientation of the body in the space, i.e., identical in all directions. Overwhelming majority of structural materials used in mechanical engineering and instrumental engineering are usually considered isotropic.

Additionally, there is an alternative approach to specifying a study's material properties.

Isotropic Materials

Isotropic materials are characterized by the fact that the physical properties of the material (elastic coefficient, Poisson's ratio, thermal conductivity and linear expansion coefficients) are considered invariant to the direction of orientation of the body in space, i.e. the same in all directions. The overwhelming majority of structural materials used in mechanical engineering and instrumentation are usually considered isotropic.

Anisotropic Materials

Anisotropic materials differ from isotropic materials in that their physical properties (elastic, Poisson's ratios, thermal conductivity, etc.) have different values depending on orientation of the physical body in space. Among the entire variety of anisotropic structural materials, the so-called orthotropic and transversely isotropic materials have the most significant practical value. T-FLEX Analysis provides the capability of working with both these types of anisotropic materials.

Orthotropic Materials

Orthotropic material – is a type of anisotropic material for which there are three mutually orthogonal planes of elastic symmetry, with respect to which the material's characteristics remain invariant. Such materials include wood, paper, veneer (if we ignore inhomogeneous spatial distribution of fibers, i.e., the dimensions of the sample are sufficiently large), composite materials of regular structure (for example, laminate fiberglass plastic, fabric fiberglass plastic).
For orthotropic materials, the generalized Hook's law can be written as:

$$\left.\begin{aligned}
\varepsilon_x &= \frac{1}{E_1}\sigma_x - \frac{\nu_{21}}{E_2}\sigma_y - \frac{\nu_{31}}{E_3}\sigma_z, \\
\varepsilon_y &= -\frac{\nu_{12}}{E_1}\sigma_x + \frac{1}{E_2}\sigma_y - \frac{\nu_{32}}{E_3}\sigma_z, \\
\varepsilon_y &= -\frac{\nu_{13}}{E_1}\sigma_x - \frac{\nu_{23}}{E_2}\sigma_y + \frac{1}{E_3}\sigma_z, \\
\gamma_{xy} = \frac{1}{G_{12}}\tau_{xy}, \quad \gamma_{yz} &= \frac{1}{G_{23}}\tau_{yz}, \quad \gamma_{xz} = \frac{1}{G_{13}}\tau_{xz}
\end{aligned}\right\}$$

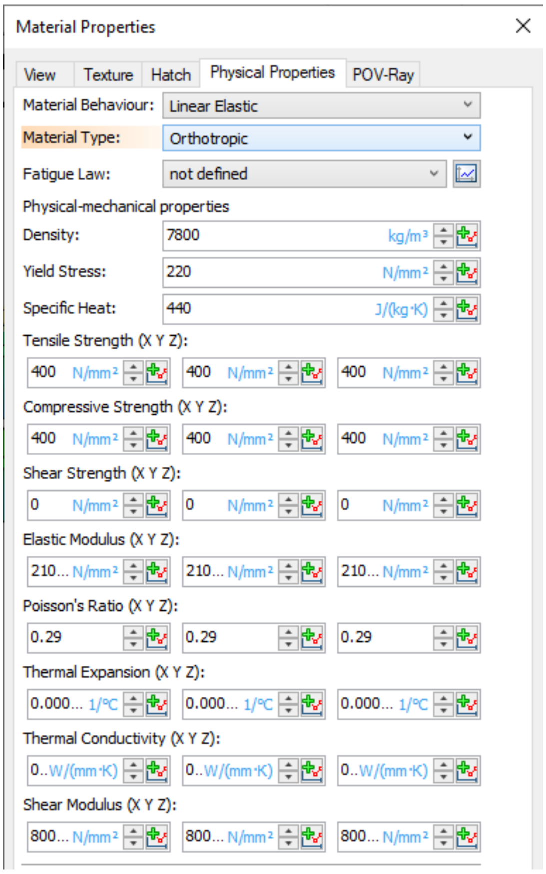
From 12 coefficients of this equation (elastic constants) only 9 are independent, since due to the symmetry of right-hand side of the system of equations of the generalized Hook's law the following relationships hold:

$$E_1\nu_{21} = E_2\nu_{12}, \;\; E_2\nu_{32} = E_3\nu_{23}, \;\; E_3\nu_{13} = E_1\nu_{31}$$

The shear moduli G_{ij} are independent of other elastic constants. However, several materials satisfy additional connections between the shear modulus and elastic modulus:

$$G_{12} = \frac{E_1 E_2}{E_1(1 + 2\nu_{12}) + E_2}$$

To define an orthotropic material, it is required in the **Material Properties** dialog, **Physical Properties** tab, to specify the structure of the material: «Orthotropic». After that, the group of parameters for specifying properties of an orthotropic material will appear.



In this dialog, the following parameters are specified:

Elastic Modulus: E_1, E_2, E_3.

Poisson's Ratio: ν_{21}, ν_{23}, ν_{31}.

Shear Modulus: G_{12}, G_{23}, G_{13}.

Coefficients of linear thermal expansion along the axes of the coordinate system: α_1, α_2, α_3.

Coefficients of heat conduction along the axes of the coordinate system: λ_1, λ_2, λ_3.

The coordinate system specified for each body defines direction of axes of symmetry. The **Orthotropic** structure of the material must be specified in the material's properties. The same orthotropic material can be specified for several bodies, and for each material the directions of axes of symmetry can be specified by a separate coordinate system. By default, the global coordinate system is used.

The principal directions of elasticity (normals to the planes of symmetry) will be directed along the axes of the coordinate system specified for the body. To specify the coordinate system for given body select the **Material Coordinate System** command.

To see which coordinate system is related to the given orthotropic body, call the same command for the body in the study tree.

Transversely-isotropic Materials

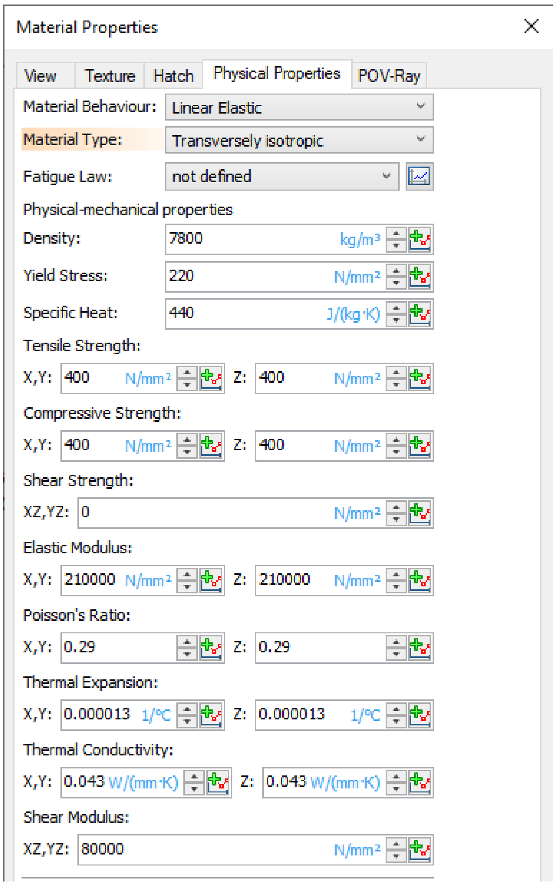
Transversely–isotropic material – is a type of anisotropic material each point of which has parallel planes of elastic symmetry, in which the material's characteristics remain invariant regardless of orientation. The axes of rotational symmetry are located orthogonally to the planes of symmetry. Examples of such materials are multi-layer pipes in which across the layers (along the radius) the properties of the material are different from those along the tangent and generating line.

For transversely-isotropic materials, the generalized Hook's law can be written as:

$$\left.\begin{aligned}
\varepsilon_x &= \frac{1}{E}\left(\sigma_x - \nu\sigma_y\right) - \frac{\nu'}{E'}\sigma_z,\\
\varepsilon_y &= \frac{1}{E}\left(-\nu\sigma_x + \sigma_y\right) - \frac{\nu'}{E'}\sigma_z,\\
\varepsilon_y &= -\frac{\nu'}{E'}\left(\sigma_x + \sigma_y\right) + \frac{1}{E'}\sigma_z,\\
\gamma_{xy} = \frac{1}{G'}\tau_{xy},\quad \gamma_{yz} &= \frac{1}{G'}\tau_{yz},\quad \gamma_{xz} = \frac{1}{G'}\tau_{xz}
\end{aligned}\right\}$$

Transversely-isotropic material is characterized by the following elastic constants: elastic modulus E, Poisson's ratio ν, and shear modulus G, acting on the planes of symmetry, and, moreover, $G = \frac{E}{2(1+\nu)}$. In all planes orthogonal to the planes of symmetry act the elastic modulus E', Poisson's ratio ν', and shear modulus G', and for several materials the following relation can be satisfied:

$$G' = \frac{EE'}{E(1+2\nu') + E'}.$$



To specify a transversely-isotropic material, it is required, in the «Material's properties» dialog opened with the **Material Properties** dialog, **Physical Properties** tab, to specify the structure of the material **Transversely-isotropic**. After that, a group of parameters for specifying the properties of the material will appear. In this dialog the following parameters are specified:

Elastic moduli in the direction of the plane of symmetry and orthogonally to it: E, E';

Poisson's ratios in the direction of the plane of symmetry and orthogonally to it: ν, ν';

Shear moduli in the direction of the plane of symmetry and orthogonally to it: G, G';

Coefficients of thermal expansion λ, λ' and thermal conductivity α, α'.

The system of coordinates in which these parameters act is specified in a similar way to orthotropic body (see above).

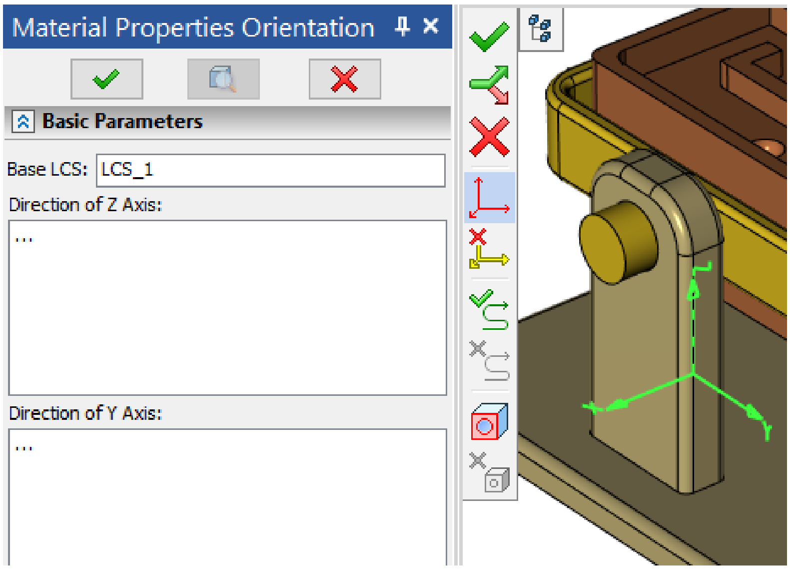
When specifying properties of any anisotropic material, the typical order of actions is the following:

1. Create a new material.
2. In the **Material Properties** dialog, **Physical Properties** tab, specify the structure of the material: **Orthotropic** or **Transversely-isotropic**. Input elastic constants, thermal constants.
3. Assign the anisotropic material to bodies.
4. After creation of the study, for each anisotropic body assign the coordinate system by using the command of the context menu of each body. If necessary, specify the law of the change of the direction of the Z-axis and the law of the change of the direction of the Y-axis along the normal to the surface.
5. Return to specification of the boundary conditions of the study.

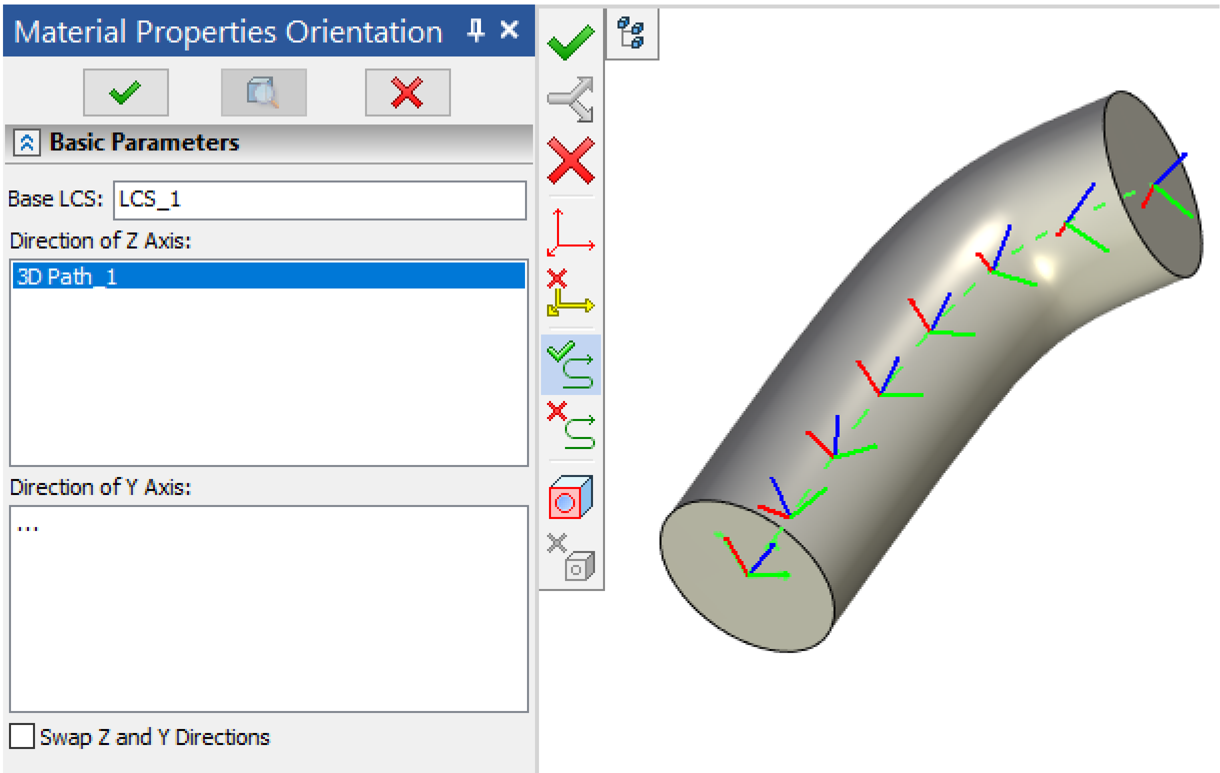
Material Coordinate System

Пиктограмма	Лента
	Analysis > Study > Material Coordinate System
Клавиатура	Текстовое меню
	Analysis > Material Coordinate System

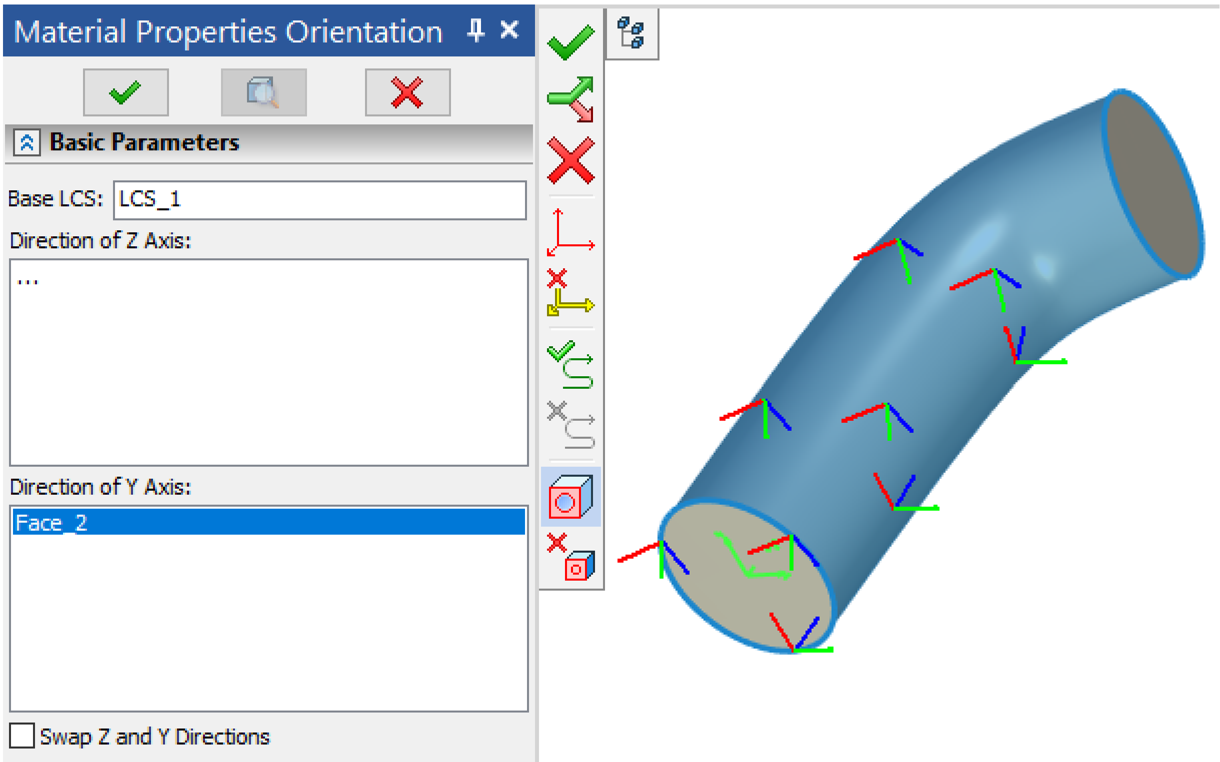
To specify the material coordinate system call this command and after that indicate LCS. The selected LCS will be included into the list as the base coordinate system and along the direction of its axes will be selected the axes of symmetry of orthotropic or transversely isotropic bodies.



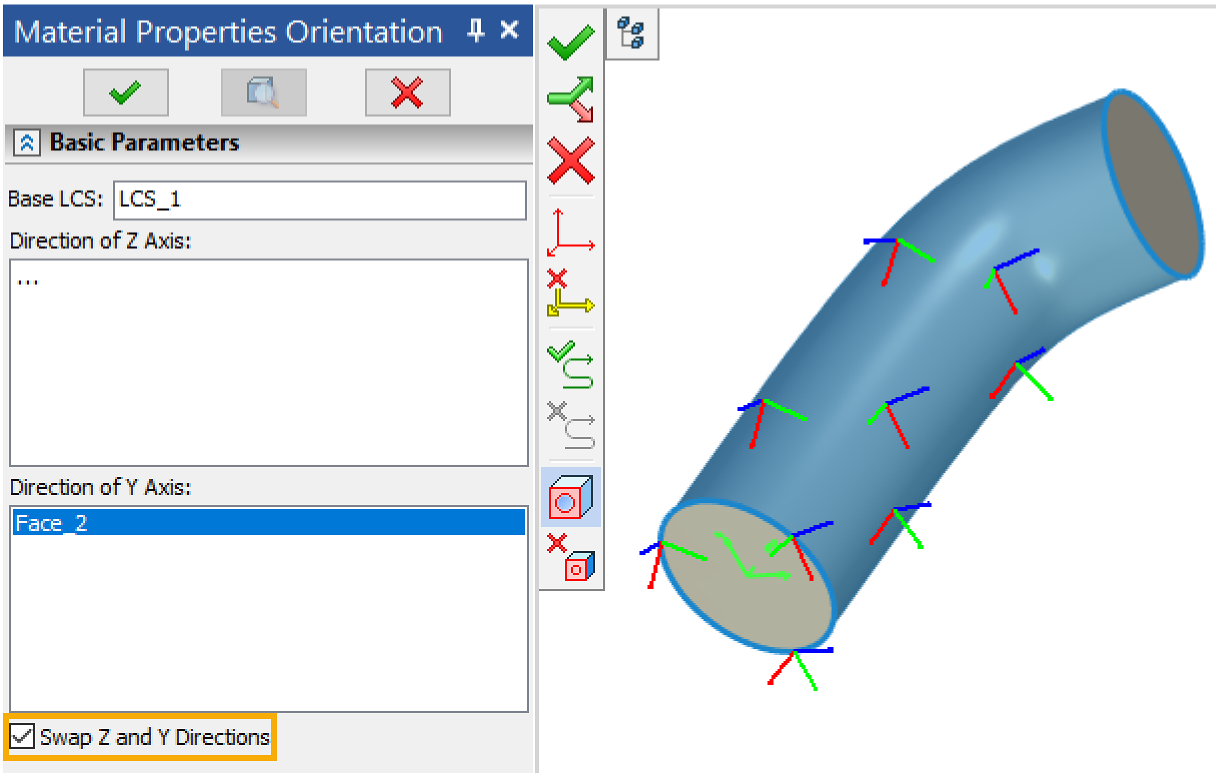
Moreover, if the anisotropic body is deformed or generated by extrusion, it is also possible to specify the law of the change of the direction of the Z-axis (for orthotropic or transversely isotropic bodies) and, in addition, of the Y-axis (for orthotropic body).



To specify the law of the change of the direction of the Z-axis, it is possible to select the 3D path (without specifying the base LCS) or several paths running together – in the latter case, the direction of the axis of symmetry at each point of the body will be determined according to the direction of the tangent in the nearest point of one of the paths.



To specify the law of the change of the direction of the Y-axis, it is necessary to select one or several curvilinear or plane surfaces (without specifying the base LCS). The direction of the Y-axis at each point of the body will be determined according to the direction of the normal in the nearest point of the surfaces.



The **Swap Z and Y Directions** switch serves for using instantaneous XY-axes, located in the plane tangent to the selected surface, as a plane of symmetry of a transversely isotropic body (see below). In this case, parameters E, ν, G will be related to the XY-plane, and parameters E′, ν′, G′ – to all of the planes containing the Z-axis.

Material Library

As mentioned earlier, each study element material is a document material, i.e. local material defined only within the scope of the document.

> You cannot create a direct link of a study item material to an external library material. You can use the **Link Materials** command to set the relationship between document materials and library materials.

This solves the following important tasks.

- The materials of the study elements will never be lost when transferring a document with calculations to other workplaces.
- Study element materials can be quickly and easily linked to materials from any external library without changing the material names each time and without editing the links in each study element.
- User control of the process of changing the physical parameters of the materials of the elements of the study.

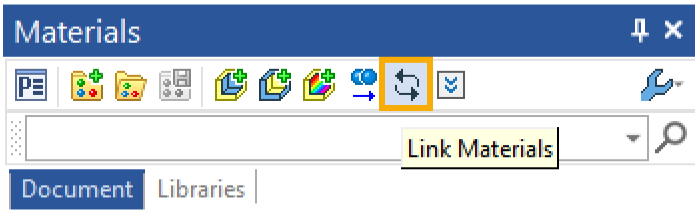
Work with materials is carried out in the **Materials** window.

Icon	Ribbon
	Assembly > Style > Materials
Keyboard	Textual Menu
<Alt+8>	Customize > Tool Windows > Materials

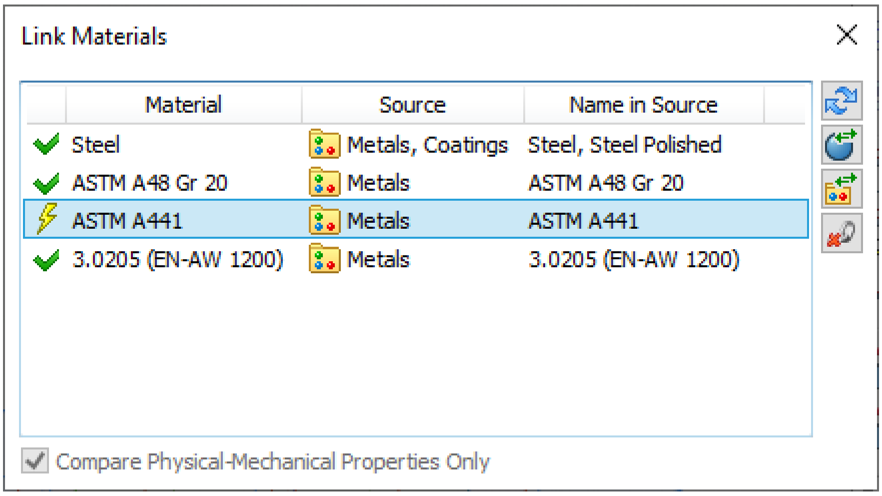
A detailed description of all the possibilities of working with the **Materials** window can be found in the **T-FLEX CAD** help. Working with the **Materials** window provides the user with many scenarios for working with document materials, which are then assigned as the materials of the study elements .
You can create your own custom libraries based on existing materials from other libraries, you can create each material from scratch, you can copy and edit materials from other libraries. You can only use materials from other libraries and not create a custom library. Document materials can be named like in a library or different names can be given. You can copy the library material into the document and specify the linking, or you can first create the document material and then link it to the library material.

> To transfer the properties of the library material to the document material, you need to link them.

The materials of the document will be automatically linked with the materials of the standard library if they were copied from it.
Link Materials command is available in the **Materials** window.



This command calls a special linking window.



If the physical parameters of the document material differ from the linked library material, then the icon will be in the first column. If the physical parameters are identical, then the icon will be in the first column. If it is impossible to automatically link a material because the library contains several identical materials, the first column will contain an icon .
Column **Material** is the name of the document material.
Column **Source** is the name of the library with the linked material.
Column **Name in Source** is the name of the library material being linked.
The following commands are available in the window.

Update . The command is available for document materials in which the physical parameters were found to differ from the physical parameters of the linked library material. The command changes the physical parameters of the document material making them identical to the library material.

Link with Library Material . Allows you to manually set the linking of the document material to the necessary material in a specific library. When this command is called, a window for viewing the materials of the document and libraries will open.

Link with Same Library Materials . Allows you to automatically set the linking of the document material to the material of the specified library based on the identity of the physical parameters.

Unlink . Breaks the linking of the document material to the library material.

Meshing

For mesh manipulations, use the command:

	Ribbon
	Analysis > Study > Mesh
Keyboard	Textual Menu
<3MM>	Analysis > Mesh

The mesh creation command can be automatically called after completing creation of the new study.

The command launches the mesh management procedure for the active study. Depending on the existence of a mesh in the active study, the system will either create a new or edit the existing mesh.

A **mesh** is created based on the operation selected at creation of the current active study. Only one mesh can be created for one study.

When creating a mesh, one can select model elements to obtain local zones of refined mesh. This is done with the purpose of getting more accurate calculation results at the critical spots of the model.

The user can select the elements for improving mesh with the help of the following automenu options:

<L> Select Elements to refine Mesh

It is possible to select 3D nodes, vertices, edges and faces:

<A> Select Faces, Edges, Vertices

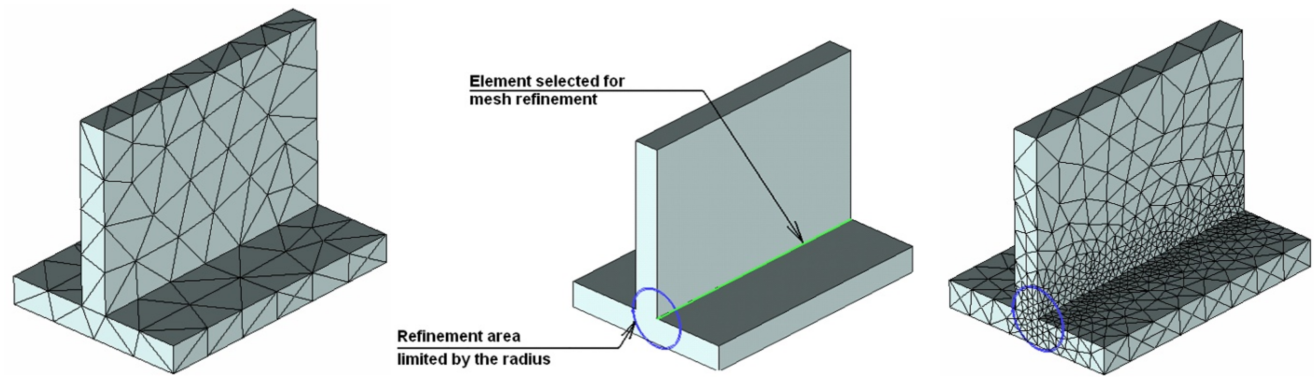
<P> Select 3D Node

<V> Select Vertex

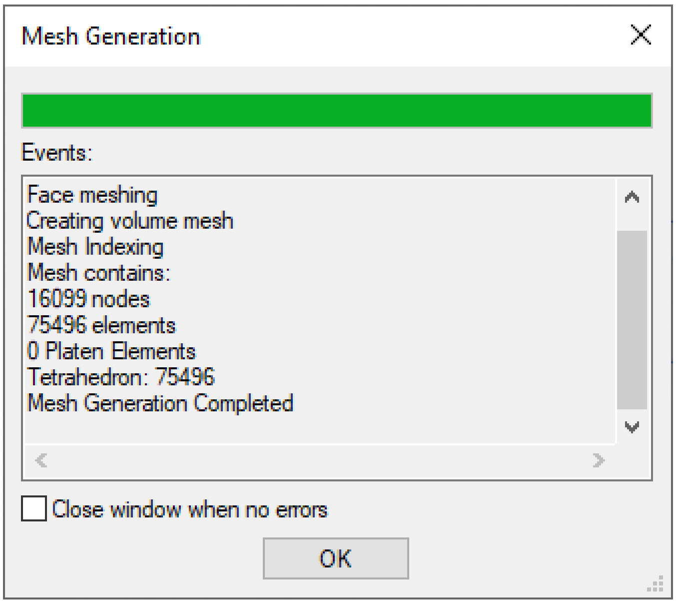
<E> Select Edge

<F> Select Face

Within the reach of the **Refinement radius** (see below) around the selected element, the size of mesh elements will be equal to the size specified in the mesh parameters for the selected refinement element.



At the mesh calculation time, the system displays a tool window that tracks the progress of the generation process. The window has a **Cancel** button that allows terminating the mesh calculation process.



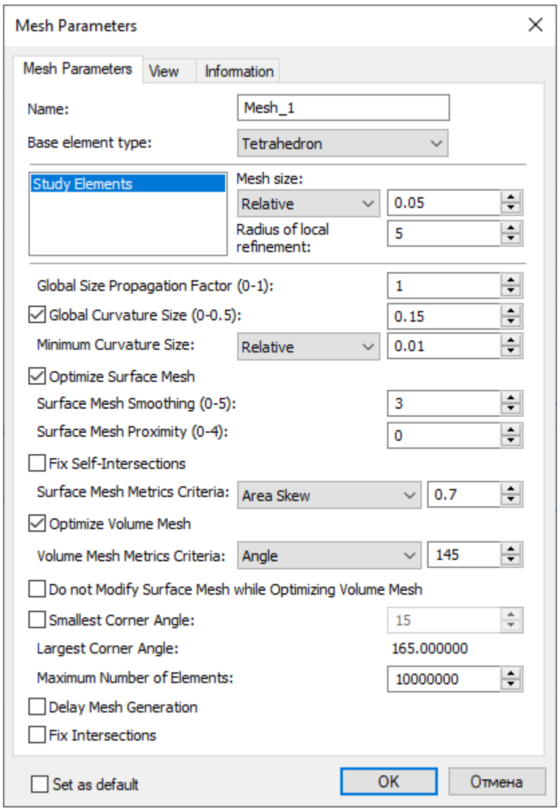
As the parametric model changes, the mesh may require an update. The system can automatically update the mesh, if the respective setting is made in the mesh parameters. Start the mesh update command manually from the context menu by right-clicking  the mesh in the studies window.

Topics in this section:

- Meshing Parameters

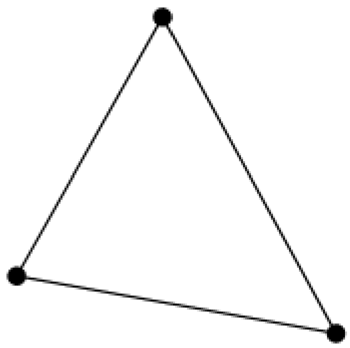
Meshing Parameters

Settings for the mesh being generated can be made either in the parameters window or in the identical parameters dialog.

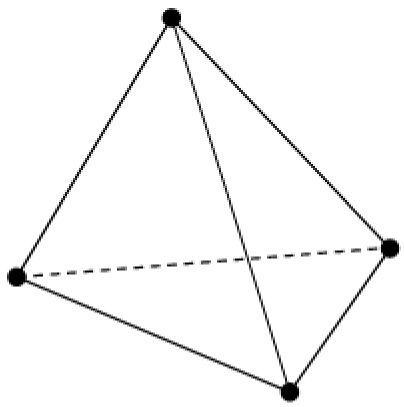


There are three variants for finite elements used in T-FLEX Analysis, namely:

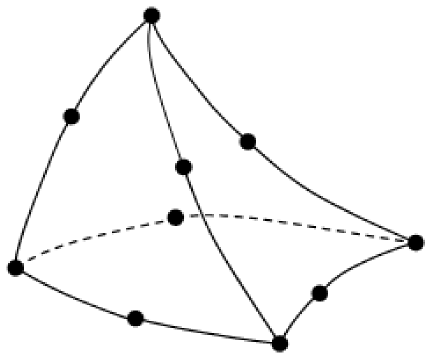
- Three-nodal triangular finite element is used for plate and cover digitization.
- Four-nodal flat-faced tetrahedral finite element (linear tetrahedron) is used for 3D body digitization.
- Ten-nodal curvilinear tetrahedral finite element (quadratic tetrahedron) is used for detailed 3D body digitization. Element mesh in this case is build basing on the four-nodal element mesh. 10-nodal element mesh is used by default for most of the tasks. You can change mesh type from ten-nodal to four-nodal in calculation parameters dialog.



3-nodal triangular finite element



4-nodal flat-faced tetrahedral finite element



10-nodal curvilinear tetrahedral finite element

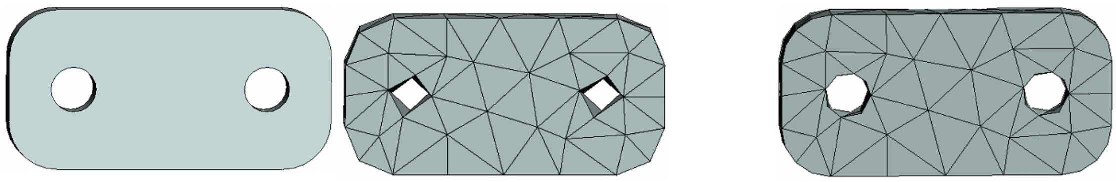
Using a curvilinear finite element enables you to approximate a complex geometry of the body boundaries more precisely and obtain higher solution accuracy with fewer elements. Hence, to describe complex body geometry more precisely you need either more 4-nodal elements with straight-lined sides (faces), i.e. flat-faced finite elements, or curvilinear elements.

It is worth noting that with the same step of discretization, creating the mesh with curvilinear elements requires more time than generation of the mesh with straight-edged elements, especially for the

models with large number of radii and fillets. In certain cases, the mesh with curvilinear elements cannot be generated at all, or its generation may take an unacceptably long time.
At the same time, difference between the results obtained on the meshes with curvilinear and straight-edged finite elements (as, for example, extreme displacements and stresses) vanishes to zero, when using a sufficiently fine discretization.

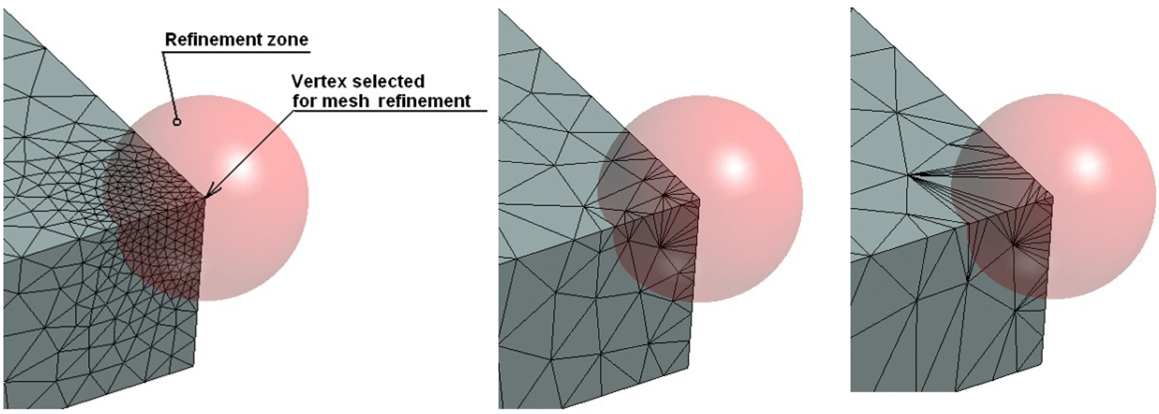
> Consequently, if constructing a mesh with curvilinear finite elements fails on a particular model of a complex geometrical shape or generating such a mesh takes too long time, then we recommend building a mesh of straight-edged finite elements with a sufficiently small discretization step, and use the latter for calculations instead.

The diagram below shows examples of dividing a model into finite elements of each type. The size of mesh elements is somewhat exaggerated for better visual effect, as compared to what is required for calculations.



Original model Mesh of straight-edged elements Mesh of curvilinear elements

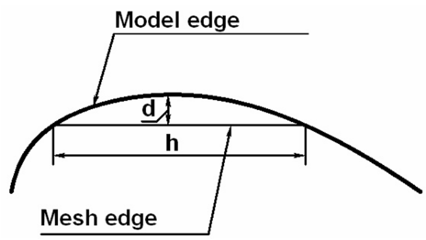
Setting up mesh updating parameters is done by selecting a choice in the dropdown list. Two choices are provided – on request ("ask") and automatically (at model regeneration).
Mesh size. The finite element edge size in the mesh being generated can be specified as relative or absolute. In the first case, the size of a edge is defined as a fraction of the model-outlining box's longest side. For the absolute size, a finite element edge is defined in the model units. The specified size is adjusted by the system to eventually get all mesh elements with edges of approximately the same size nearing the value set in the parameters. The model elements selected for mesh refinement allow setting only the absolute size.
Global Size Propagation Factor. Controls the speed of mesh variation from reduced-size mesh cells to large cells of the general size. If the factor equals 1 (default), then the mesh size nearly doubles with each following element up until its size reaches the large mesh size. With the reduction of the factor value, the transition of sizes occurs in a lesser number of steps (large leaps in the element size). If the factor is equal 0, then a cell's size jumps to coarse without transition. Normally, the values near one are used most in practice.



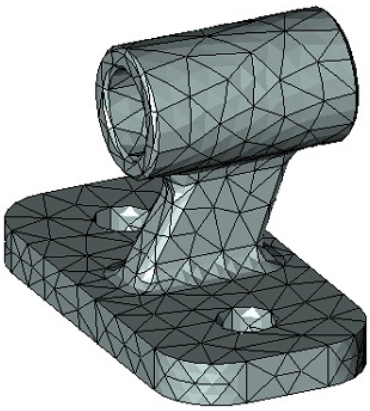
Propagation factor = 1 Propagation factor = 0.5 Propagation factor = 0

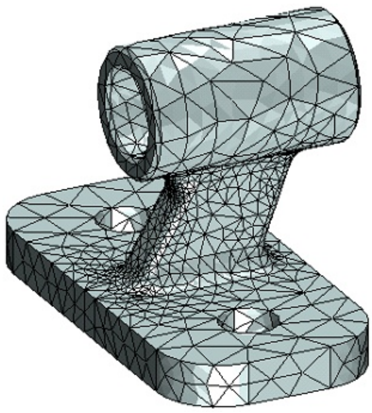
Refinement Radius. This parameter can be set only for model elements selected for local mesh refinement. Refinement radius determines the size of the zone around the element, within which the mesh is constructed with enhanced individually specified properties – usually, finer meshing. The

distance is counted from the element selected for refinement. The absolute mesh size for each auxiliary element is specified separately. In practice, this capability can be used for achieving more accurate calculation results by the processor within approximately same calculation time, using more coarse overall mesh for the entire model, while a more elaborated mesh at critical points.
Curvature Ratio. This parameter enables automatic processing of curved surfaces and sets a limit on the minimum size of a mesh element during such processing. The limitation is defined by the bending factor, that is evaluated as the ratio of the depth d to the chord length h (see the diagram). The limitation can be specified in the range 0 ч 0.5.



Minimum Curvature. Works together with «Curvature ratio». This sets the ultimate minimum size of a curve segment, to which it can be divided. This parameter is introduced for limiting the number of mesh elements – this is because automatic subdivision of surfaces could go on indefinitely on some models, such as a cone, in order to meet the specified bending amount condition. This parameter permits any values greater than zero. Minimum curvature value may be set in relative or absolute units, which are defined in the same way as when specifying **Mesh size** parameter.

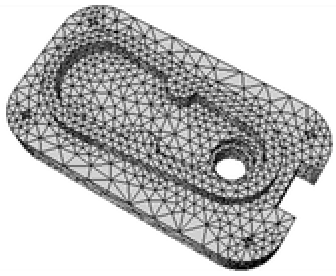




Curve processing disabled Curve processing enabled

Optimization Options provide control over the process of generating an improved-quality mesh. When the generator creates the mesh, it first calculates a preliminary mesh. After that, the program can apply certain manipulations to the obtained preliminary mesh in order to improve its quality. Those manipulations are divided into two stages: optimization, which changes connectivity between mesh vertices, and Smoothing, which replaces mesh vertices. Smoothing is driven by a number in the range 0 ч 5. A greater number yields greater smoothing. A high degree of smoothing will temper transitions in the mesh size. One can distinguish the surface and the volume mesh optimization processes. In most cases, when those are unnecessary, you can disable these options. This will speed up the mesh generation process.
Surface mesh proximity is equal to the value in 0 ч 4 range. The value specifies the desired number of elements that lay within the thin walls. The real number of elements may differ from the specified because the specified value is average. The parameter does not apply to the hybrid meshes in which thin walls are specified using a surface mesh.

Surface mesh proximity is disabled



Surface mesh proximity is enabled

Fix Self-Intersections. Enables checking of the mesh for self-intersections after generation. If they are found the system tries to fix them automatically. Mesh self-intersections appear when coarse sizes are set for the model with highly curved geometry.

Surface/Volume mesh metrics criteria. A mesh is generated in two steps. At first, the surface is divided, then a volume mesh is created on the basis of the surface mesh. Therefore, you can separately set metric criteria for surface (the first mesh generation step) and for the volume (the second mesh generation step). You can select one of the six metric criteria. A metric criteria specifies the optimal value to which the system should aim to when generating the grid.

Angle. The maximum angle of triangle for the surface mesh elements is set or the maximum dihedral angle of tetrahedron for the volume mesh is set. The value lies in range 145°<=0<180°. The value by default is 145°.

Area Skew (for a surface) and Volume Skew (for volume elements).

A criteria for surface elements is determined by the expression:

$$K = \frac{S_{opt} - S}{S_{opt}},$$

where K is the Area Skew criteria, S is an area of a triangular element, Sopt – optimal area of the triangular element. The Sopt value is determined by the expression:

$$S_{opt} = \frac{3\sqrt{3}}{4} R^2,$$

where R is a radius of a circumscribed circle for the triangular element.

A criteria for volume elements is determined by the expression:

$$K = \frac{V_{opt} - V}{V_{opt}},$$

where K is the Volume Skew criteria, V is a volume of a tetrahedral element, Vopt is an optimal area of the tetrahedral element. The Vopt value is determined by the expression:

$$V_{opt} = \frac{8\sqrt{3}}{27} R^3,$$

Where R is a radius of a circumscribed sphere for the tetrahedral element.

For both surface and volume elements, the K criterion can vary from 0 to 1. When K = 0, the element is equilateral (equilateral triangle or tetrahedron), when K=1 the element is degenerate element, so the value 1 cannot be specified. It is known from practice that a qualitative surface mesh is obtained at K<0.8 (the default value of an area screw is K=0.7), and a high-quality volume mesh is obtained at K<0.9 (by default, the value of an area screw is K=0.85).

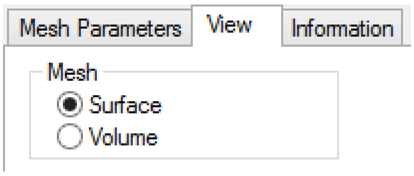
Sides Ratio. The criterion is defined as the aspect ratio of an element sides: the ratio of the largest side to the smallest. The criterion can vary from 1 to infinity: 1 is an equilateral element, infinity is degenerate. By default, for surface triangular elements, the value of criterion is 6, for volume tetrahedral elements the value of criterion is 12.
Angle Skew. The method for calculating the criterion is close to calculating the criteria for Area Skew and Volume Skew. The criterion can vary from 0 to 1. An angle skew value equal to 0 means that the element is equilateral. An angle skew value of the criterion equal to 1 means that the element is degenerate (the value of the criterion equal 1 cannot be set). By default, the surface elements are set to Angle Skew equal to 0.7, for volume elements, the default value is 0.85.
Ratio of Length to Area (for the surface) and **Ratio of Length to Volume** (for volume elements). For surface elements, the criterion is calculated by the expression:

$$K = 3\left(\frac{4S}{L^2}\right)^2,$$

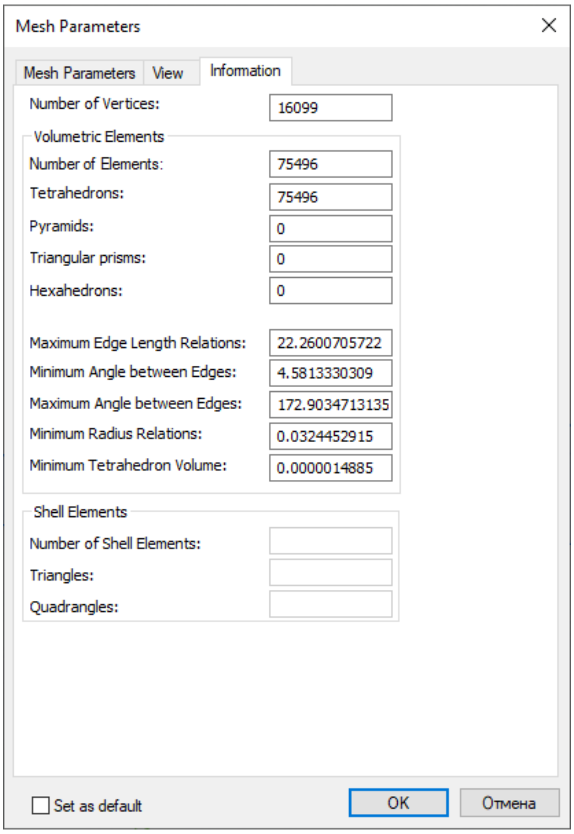
where K is a metric criterion, S is the area of a triangular element, L is the root-mean-square length of the element sides.
For volumetric elements, the criterion is calculated by the expression:

$$K = 3\left(\frac{72V}{L^3}\right)^2,$$

where V is the volume of the tetrahedral element.
For both surface and volume elements, the criterion can take values from 0 (worst value) to 1. By default, the criterion value for both triangles and tetrahedra is 0.15.
Jacobian Ratio. The value of the criterion can vary from 0 (worst value) to 1. By default, for both surface and volume elements, the value of the criterion is 1. This criterion is valid only for square flat elements or prismatic volume elements. If another type of elements is selected, then the default metric criterion will be used: Angle.
Do not modify surface mesh while optimizing volume mesh. Allows optimizing the volume mesh so as not to affect the surface mesh obtained at the first stage of the generation. Otherwise, the mesh on the surface can be changed in the optimization. This capability is useful in the cases when the user wants to maintain the mesh structure of the part's surface, that was obtained as a result of adjusting grid settings, yet still pursues optimization of the volume mesh.
Smallest corner angle defines the admissible range of angle values between a mesh element's (tetrahedron's) edges. The greatest triangle's angle is calculated automatically (180°-a_min). At the same time, note that angles outside this range could still be present due to other factors.
Maximum Number of Elements. This parameter sets a limitation on the total number of mesh elements. This functionality is provided to prevent accidental creation of a too large number of mesh nodes, which might significantly slow down both generation of the mesh itself and the following solving. If the number of elements exceeds the specified limit while generating the mesh, then the system outputs the appropriate message and terminates mesh generation. In such a case, to obtain a mesh with the specified number of elements, use more relaxed settings.
Delay mesh generation. The flag allows saving the user-defined parameters of mesh generation in the study without the creation of the mesh in the current moment.
On the **View** tab, one can specify the type of mesh rendering. The surface mesh helps assess most of the main parameters of the obtained mesh. In this way, the mesh portions in the interior of the model's volume are not shown, helping speedy system operation when rotating the 3D scene.



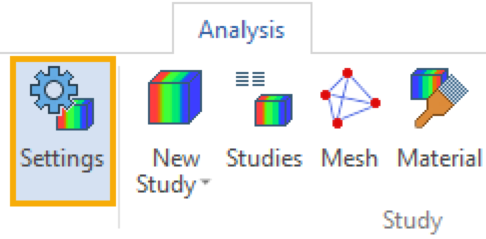
The volume mesh rendering shows the entire mesh, including its portions within the interior of the model's volume. In this mode, the system may experience a slowdown when rotating the 3D scene. In such a case, it would be best to use the wireframe mode for the 3D scene.
On the **Information** tab you can get information about certain parameters of the obtained mesh: the total number of vertices, the number of finite elements, etc. All those parameters help assisting the quality of the resulting mesh generation. Some of the parameters require additional explanation:



Maximum Edge Length Relations is the characteristic referring to the mesh element that has the overall greatest ratio of its longest and shortest edges.
Maximum/minimum Angle Between Edges. Reports the actually resulting maximum and minimum angles between edges of mesh elements.
Maximum Radius Relations. Reports the smallest ratio of the radii of the inscribed and circumscribed spheres of a tetrahedron.

Customization and Utility Commands

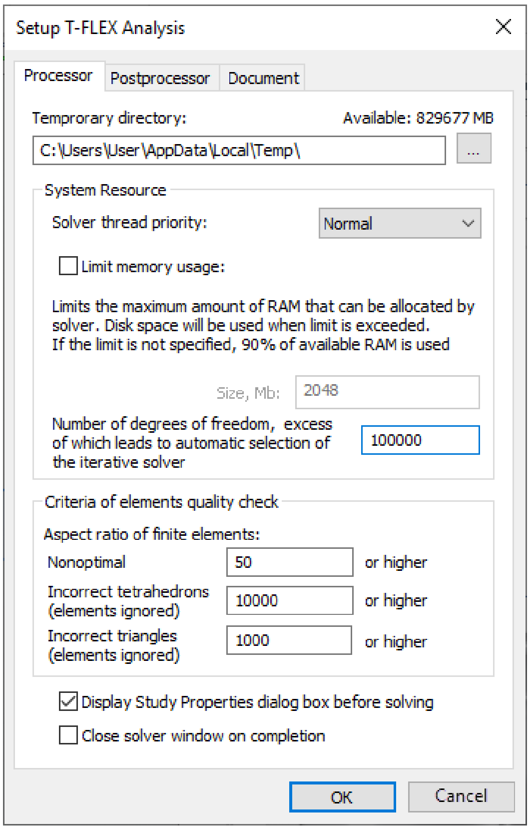
Using the **Analyze> Settings** command, the user can define the global settings of the T-FLEX Analysis system.



The following parameters are defined on the **Processor** tab.

Temporary directory - defines the path to the directory in which intermediate working data are saved when solving systems of equations. By default, work files are saved in the folder specified by the "TEMP" ("TMP") Windows system variable. The user can override this path if necessary.
The following parameters are defined in the **Resources** group:

Solver thread priority - allows the user to define the system priority of the modules responsible for solving systems of equations. Depending on the priority set, the Windows operating system will allocate system resources, giving preference to the higher priority. For example, if you plan to solve a large problem using disk space for a long time, the user can pre-set the priority to **Below Normal** or **Low**, which will allow him to work in parallel in other Windows applications without special restrictions.



Limiting memory usage - allows the user to set the amount of RAM, upon exceeding which the system will switch to the mode of solving equations using disk memory, which usually takes much more time.

Number of degrees of freedom, excess of which leads to automatic selection of the iterative solver. The number of degrees of freedom for choosing an iterative method for solving systems of algebraic equations. The default is 100 000.

In the **Criteria for elements quality check** group, the aspect ratios of finite elements (tetrahedrons and triangles) are set at which the system will consider them as "not optimal" or "erroneous". Erroneous elements are forcefully removed from the grid, because lead to singularity of systems of equations (impossibility to solve them). Usually these are either practically flat or point-degenerate tetrahedrons, or triangular finite elements degenerate into a line. The user can change these parameters if necessary.

Display Study Properties dialog box before solving – the active control includes the mode of automatic invocation of the study parameters dialog when the **Analysis** > **Solve** command is initialized for all studies (default setting).

Close solver window on completion - the control element turns on the mode of automatic closing of the information window with the display of the process of solving systems of equations in all problems. By default, this mode is not activated.

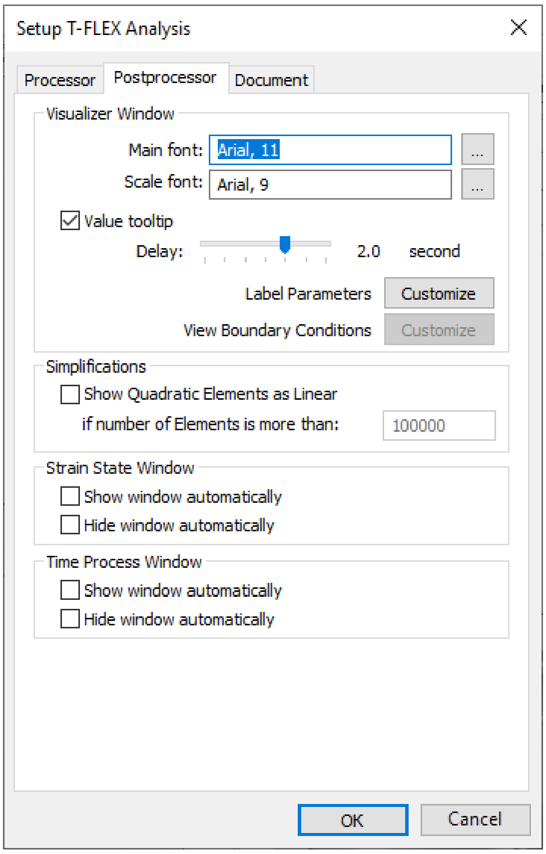
On the **Postprocessor** tab, the user can define global settings for the visualization of results that apply to all studies.

Main font – sets the default font for text information displayed in the Postprocessor visualizer window (task name, result type, etc.).

Scale font – sets the font for displaying the numeric values of the color scale.

The control **Value tooltip** enables the mode, in which, when you hover the cursor over a region of the model, a tooltip appears in the **Postprocessor** window with an interpolated value of the result corresponding to the coordinate of the model under the cursor.

The **Delay** control allows you to set the time interval for the tooltip to appear.

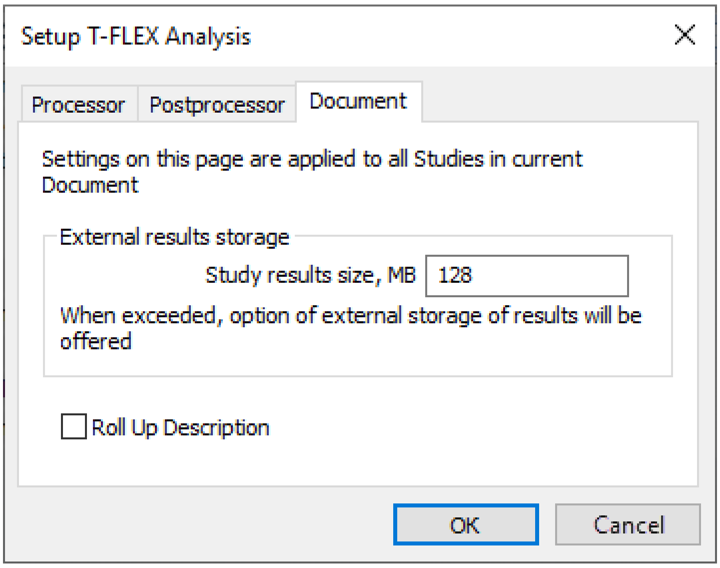


In the **Simplification** group, a threshold for the number of finite elements is set, at which the calculation results in the **Postprocessor** window are displayed only for the main nodes of quadratic finite elements, and the results at the nodes located in the middle of the element edges are omitted (this does not apply to extreme values). This mode allows to significantly speed up the loading of results into the **Postprocessor** window on very large grids (for example, more than 10 000 000 elements).

The options **Show window automatically**, **Hide window automatically** in the groups **Strain State Window** and **Time Process Window** allow you to control the display of floating panels **Deformed State** and **Time Process**, respectively.

On the **Document** tab, you can activate the option to save the results of calculating studies in a separate external file. By default, the calculation results are saved in the main file of the T-FLEX document (extension .grb). When external storage of results is activated, a file with the name "Document name_Problem name" and the extension .tfa is created in the directory of the source document).

This option can be useful when solving problems with large volumes of results, and can also speed up the saving of a document (depending on the operating system).

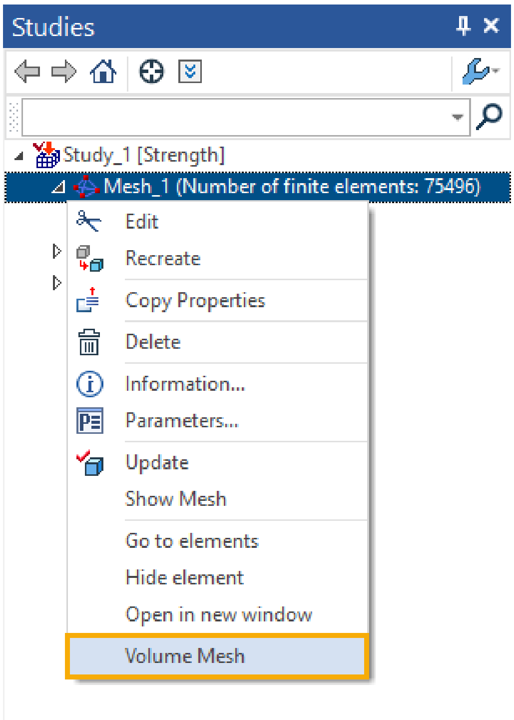


Show Study Elements

As you work with studies, various study elements can be displayed in the 3D scene – loads, restraints, mesh. Special commands are provided for handling those displays.

Icon	Ribbon
	Analysis > Study > Study Elements
Keyboard	Textual Menu
<3MH>	Analysis > Show Study Elements

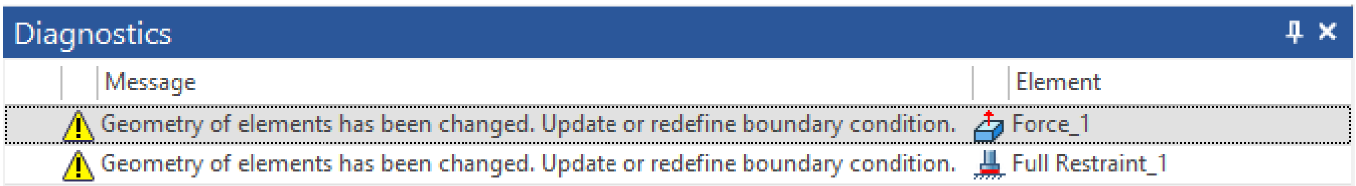
With this command, you can enable or disable the mode of displaying all loads and restraints in the 3D scene. This command is available in the context menu upon selecting a study.



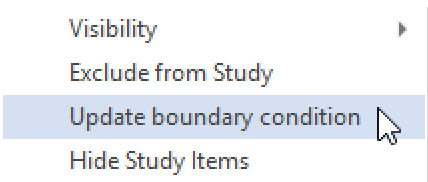
A generated surface or volume mesh can be displayed in the 3D window. A surface mesh is automatically displayed after finishing the generation process. When the mesh is displayed, the model is not shown. If you need to work with the model (for example, to define a restraint on a model's face), the system automatically hides the mesh. To view the mesh again, one can use the **Show Mesh** command. This command is available in the context menu upon selecting the **Mesh** element in the studies window. This menu also contains commands for switching between the surface and volume mesh views. The commands are **Volume Mesh** and **Surface Mesh**.

Loads and Restrictions Actuality

You should update loads and restrictions when the body topology, which is included to the calculated model, was changed. The diagnostics window will show you a warning.



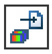
You can update loads and restrictions using the Update boundary condition command. The command can be found in the context menu of the corresponding task element.



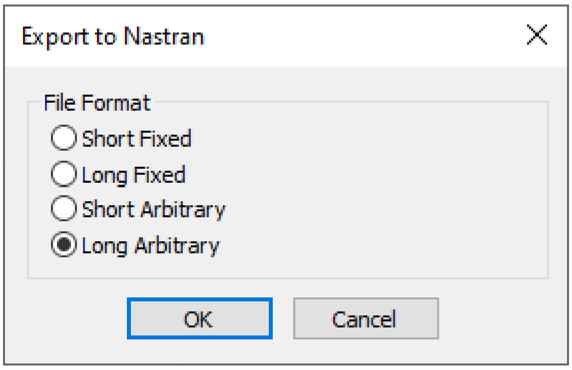
You should delete or create a new load or restriction in the case when it does not comply to the new topology of the body.

Export

Prepared initial data of each study can be exported in the Nastran (*.dat) format. To export an active study, use the command:

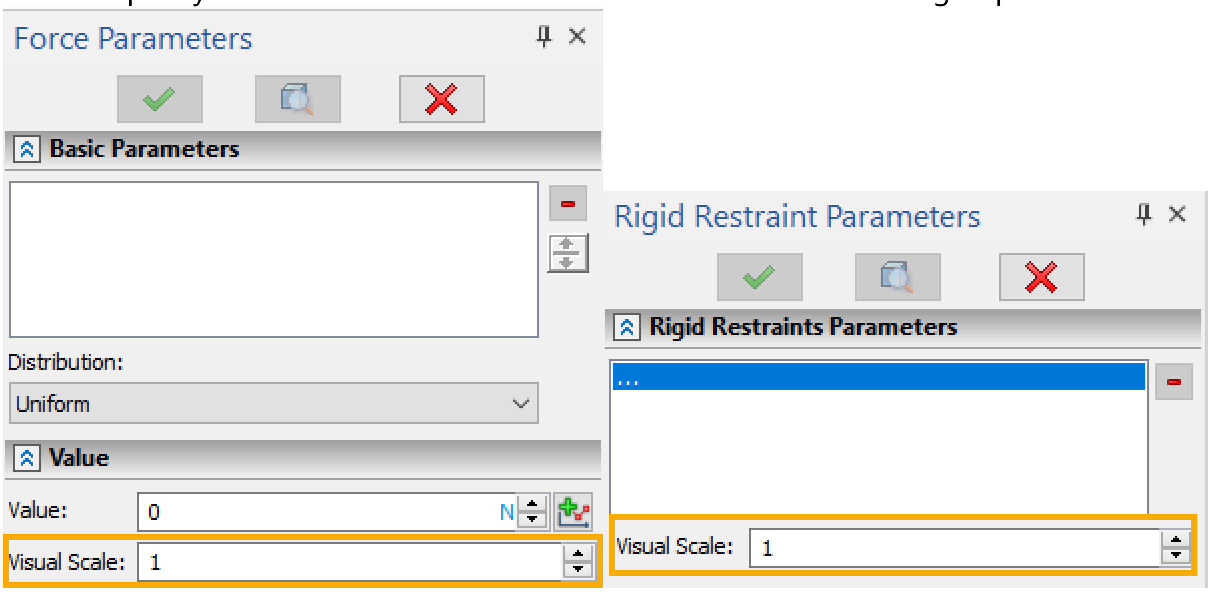
Icon	Ribbon
	Analysis > Output > Export Study
Keyboard	Textual Menu
<3MX>	Analysis > Export

Also, this command is accessible via the context menu upon selecting the study  in the window **Studies** or in the window **3D Model**.
After calling the command, the standard file-saving dialog appears.
Next, you need to specify a combination of two properties of the format:
Short/Long - defines the precision of the output parameters (the maximum number of digits in decimal positions, including the decimal point): short - up to 8 decimal digits, long - up to 16 decimal digits.
Fixing/Arbitrary - defines the type of export data: fixed - data are output by columns of a fixed size, arbitrary - data are output separated by commas.



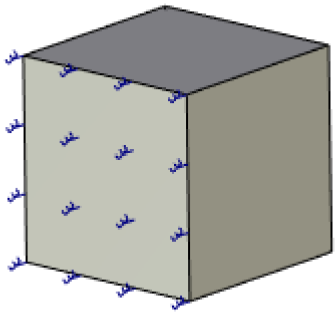
Visual Scale

You can specify **Visual Scale** for most commands from the **Conditions** group of the ribbon.

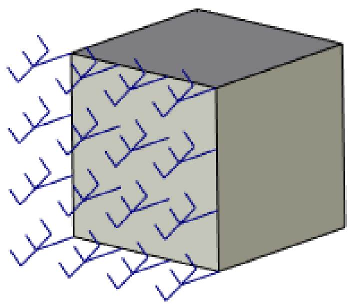


Field for specifying a **Visual Scale** value might be located in various tabs of the **Parameters** window depending on the selected command.
This value defines a **Condition's** visualization size in the 2D scene.

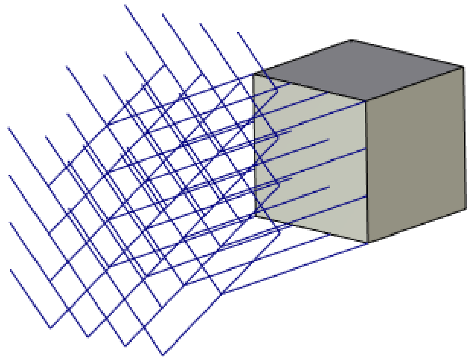
Examples of visualization with different scale values for the same condition are shown below:



Visual Scale = 0.25



Visual Scale = 1



Visual Scale = 5

Defining Initial Conditions

Initial conditions are used for modeling of processes that change in time. You can use the initial conditions to set values of nonstationary processes at time zero.

Topics in this section:

- Mechanical Initial Conditions
- Thermal Initial Conditions

Mechanical Initial Conditions

Mechanical initial conditions are used in dynamic analysis studies (Mode superposition study, dynamic nonstationary processes).

Topics in this section:

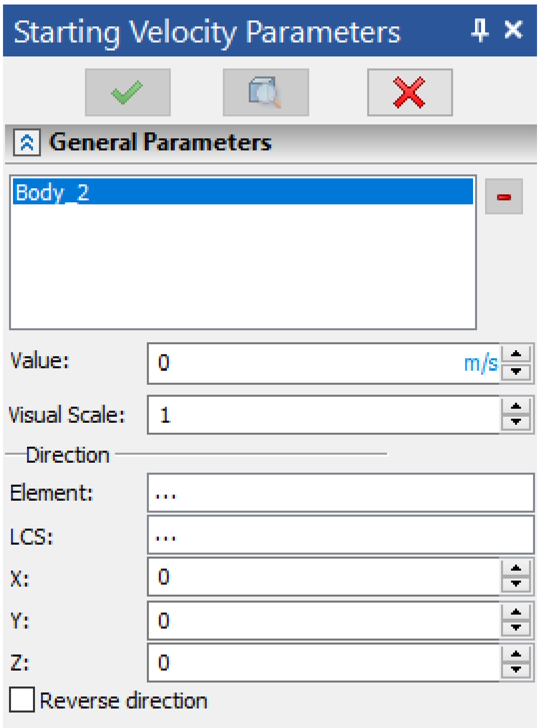
- Initial Velocity
- Inicial Acceleration

Initial Velocity

Use the following command to specify **Initial Velocity** boundary condition:

Icon	Ribbon
	Analysis > Conditions > Force > Initial Velocity
Keyboard	Textual Menu
<3MV>	Analysis > Load > Initial Velocity

You need to select model elements to apply loads after the command call. Selected elements are added to the list in the upper part of the **General Parameters** tab of the **Parameters** window.



You can select all bodies to specify their initial velocity. Use automenu options for this purpose:

	<M>	Select All Solids

Value. Here you can insert the value of velocity.

Units. You can set the following units for initial velocity: m/s, cm/s, in/s.

Direction of Load

Use the following option to set the direction of initial velocity using 3D model object:

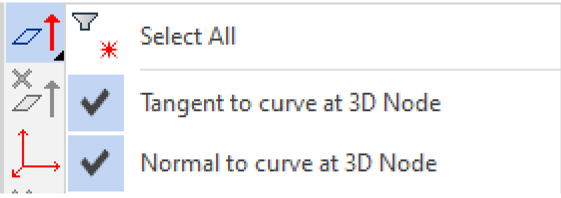
	<D>	Select direction

To cancel direction selection use option:

	<U>	Cancel direction selection

You can set **Reverse direction** flag to change direction on the opposite.

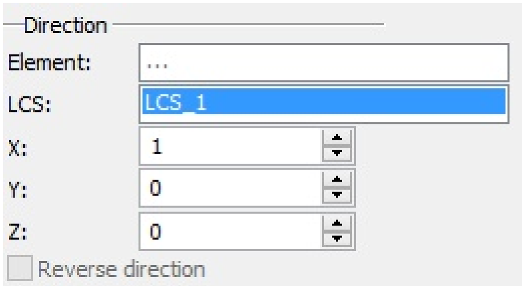
The drop-down list of filters allows to specify elements that can be selected for direction defining.



To set the direction of initial velocity using LCS use option:

	<C>	Select LCS
	<K>	Cancel LCS selection

In this case, the direction is defined as radius vector. The radius vector is specified by values of guide cosines in the X, Y and Z fields. The reverse direction is specified by the sign "-" before the value.



The sequence of actions for initial velocity specifying:

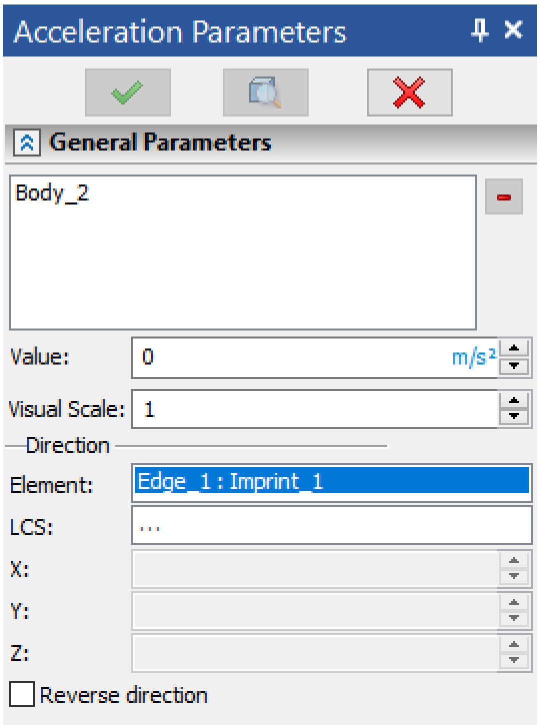
1. Activate Initial velocity command .
2. Select body or several bodies.
3. Specify initial velocity value.
4. Specify units.
5. Specify direction.
6. Apply command.

Initial Acceleration

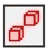
Use the following command to specify initial acceleration:

Icon	Ribbon
	Analysis > Conditions > Initial Acceleration
Keyboard	Textual Menu
<3MZ>	Analysis > Load > Acceleration

You need to select model elements to apply loads after the command call. Selected elements are added to the list in the upper part of the **General Parameters** tab of the **Parameters** window.



You can select all bodies to specify their initial velocity. Use automenu options for this purpose:

	<M>	Select All Solids

Value. Here you can insert the value of velocity.

Units. You can set the following units for initial acceleration: m/s^2, cm/s^2, in/s^2.

Direction of Load

Use the following option to set the direction of initial acceleration using 3D model object:

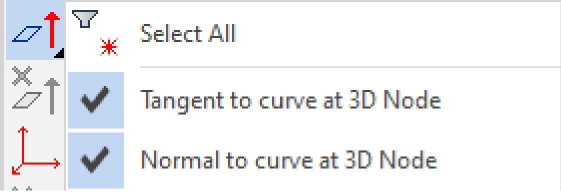
	<D>	Select direction

To cancel direction selection use option:

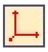
	<U>	Cancel direction selection

You can set **Reverse direction** flag to change direction on the opposite.

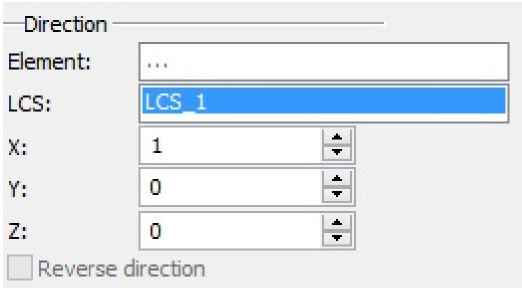
The drop-down list of filters of filters allows to specify elements that can be selected for direction defining.



To set the direction of initial acceleration using LCS use option:

	<C>	Select LCS
	<K>	Cancel LCS selection

In this case, the direction is defined as radius vector. The radius vector is specified by values of guide cosines in the X, Y and Z fields. The reverse direction is specified by the sign "-" before the value.



Sequence of actions for initial acceleration specifying

1. Activate Initial acceleration command.
2. Select body or several bodies.
3. Specify initial acceleration value.
4. Specify units.
5. Specify direction.
6. Apply command.

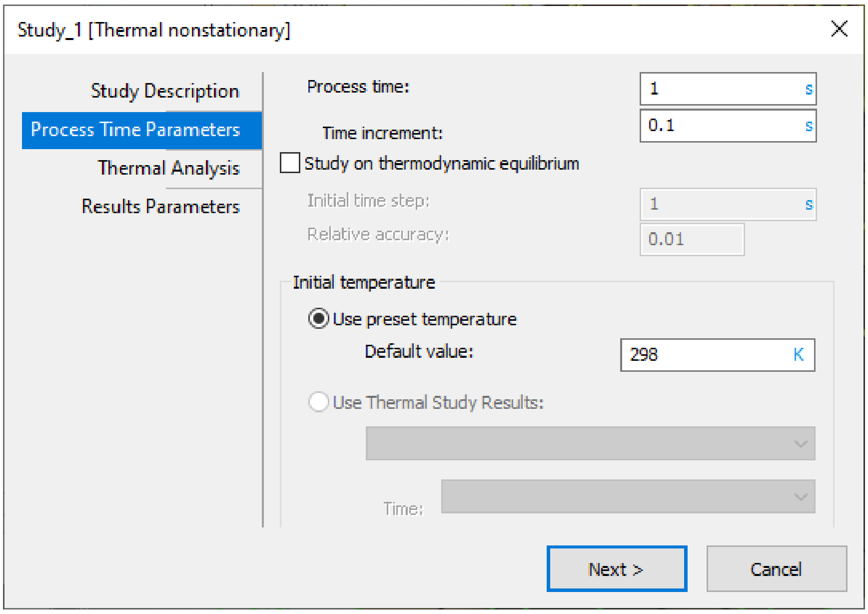
Thermal Initial Conditions

To specify the initial condition use command:

Icon	Ribbon
	Analysis > Conditions > Temperature
Keyboard	Textual Menu
<3TT>	Analysis > Thermal Load > Temperature

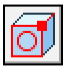
The thermal initial conditions type is used upon modeling thermal analysis study.

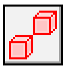
The **initial temperature** condition is used for specifying of the temperature in the nonstationary thermal study. The thermal load defines the temperature of the selected model elements at time zero. In those nodes of the finite element mesh that do not belong to the specified elements of the model, the initial temperature will be set to the "default" value. It is defined in the **Process Time Parameters** dialog box, which can be called from the context menu of the thermal analysis task, on the **Process Time Parameters** tab.

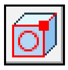


Initial temperature can be applied to the body, face, edge or vertex of the model.

Use the following options of the automenu to select model elements:

 <E> Select element for loading

 <M> Select All Solids

Selection filters are available upon click and hold  on the  **Select elements for loading** option:

<Alt+Num+> Select All

<Ctrl+Alt+V> Select Vertex

<Ctrl+Alt+E> Select Edge

<Ctrl+Alt+Q> Select Face

<Ctrl+Alt+B> Select Body

Same filters are available in the Filter Toolbar:

In the Select Face mode additional filters are available in the automenu:

<C> Select all faces of body including contacting

<N> Select all faces of body excluding contacting

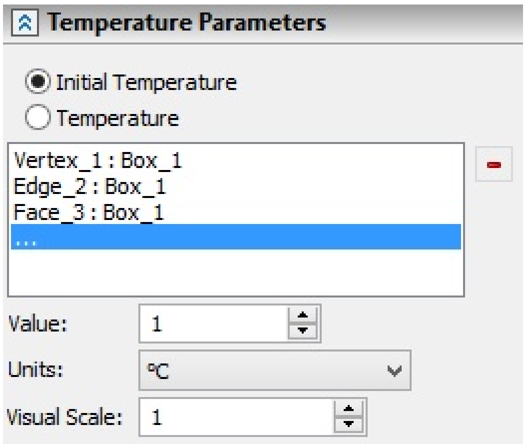
<S> Select all faces of bodies used in study

Selected elements are added to the list in the **Parameters** window.

In the **Temperature parameters** window you need to set the following parameters:

- Value,

- Units C, K, F.



The **initial temperature** is shown in the 3D scene in the same way as the common **temperature** load.

The sequence of actions for initial temperature specifying

1. Activate Temperature command.
2. Set Initial temperature flag.
3. Select body, face, edge, vertex or several elements.
4. Specify initial temperature value and units.
5. Apply the command.

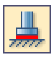
Defining Restraints

The location for specifying a restraint can be a face, edge or vertex of the subject body. The system supports several types of restraints: **full restraint**, **partial restraint**, **symmetry**, **contact**, **additional stiffness**, **remote movement**. A restraint is added to the active study and can be related only to elements of the body that is used in the active study. To avoid a failure when solving, you need to create enough restraints for the model, for example, one full restraint.

Topics in this section:

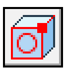
Full Restraint

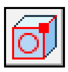
To specify a full restraint, use the command:

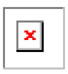
Icon	Ribbon
	Analysis > Conditions > Full Restraint > Full Restraint
Keyboard	Textual Menu
<3MC>	Analysis > Restraint > Full Restraint

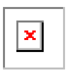
This type of boundary conditions locks all degrees of freedom for the selected object. A full restraint can be applied to a face, edge or vertex of the model.
To specify a restraint, you need to select a model element. Faces, edges and vertices are available for selection.

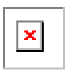
Use the following option of the automenu to select model elements:

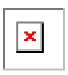
 <E> Select element for fixing

Selection filters are available upon click and hold  on the  **Select elements for fixing** option:

 **<Alt+Num+>** Select All

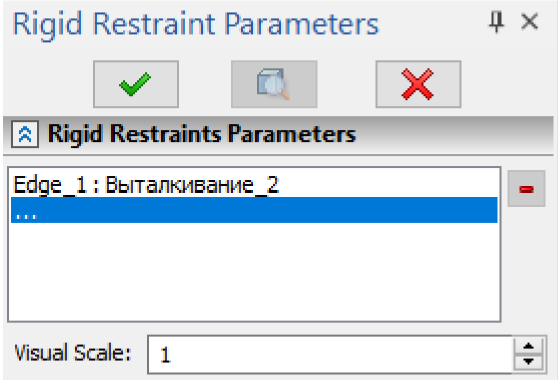
 **<Ctrl+Alt+V>** Select Vertex

 **<Ctrl+Alt+E>** Select Edge

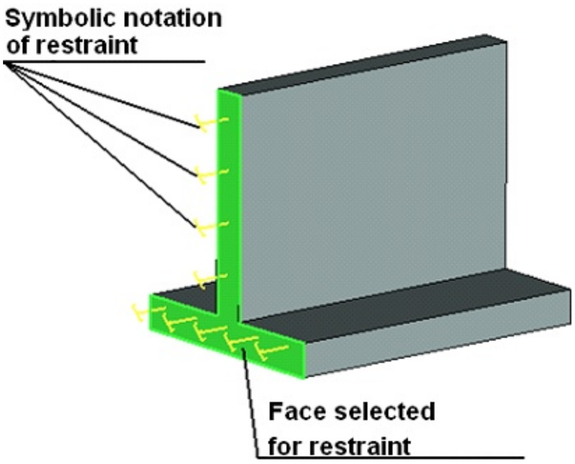
 **<Ctrl+Alt+Q>** Select Face

Same filters are available in the Filter Toolbar:

Selected elements are added to the list in the **Parameters** window.

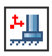


Upon selecting an element, the symbolic notation of the full restraint appears in the 3D window.



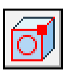
Partial Restraint

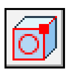
To specify a partial restraint, use the command:

Icon	Ribbon
	Analysis > Conditions > Full Restraint > Partial Restraint
Keyboard	Textual Menu
<3MU>	Analysis > Restraint > Partial Restraint

When defining a partial restraint, the user is offered to manually specify restraints on different degrees of freedom. When using only partial restraints, you need to ensure the sufficient number of restraints for fixing the model.
To specify locations of a partial restraint, select an edge, face or vertex.

Use the following option of the automenu to select model elements:

 <E> Select element for fixing

Selection filters are available upon click and hold  on the  **Select elements for fixing** option:

<Alt+Num+> Select All

<Ctrl+Alt+V> Select Vertex

<Ctrl+Alt+E> Select Edge

<Ctrl+Alt+Q> Select Face

Same filters are available in the Filter Toolbar:

Upon selecting an element, the symbolic notation of the full restraint appears in the 3D window.

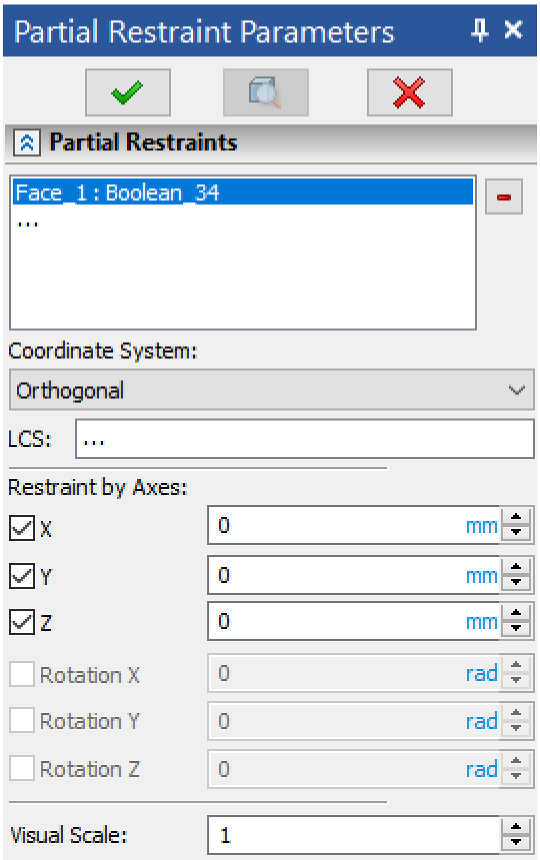
Selected elements are added to the list in the **Parameters** window.

Next, you need to define restraints by degrees of freedom. The user can work in one of the three types of coordinate systems – Cartesian, cylindrical or spherical. A local coordinate system is used for binding the coordinate system in question to the model. It is worth noting that in the case when the user did not define the local coordinate system, the partial constraints will be defined with respect to the global coordinate system.
To select coordinate system use the following automenu option:

<C> Select LCS

<K> Cancel LCS selection

Each coordinate system allows restraining displacements in three degrees of freedom. An activated box item of the respective degree of freedom in the selected coordinate system means that displacements are fully constrained in this direction (if the value is equal 0), or that a known displacement is specified (if the value in the respective text field is not zero). A cleared flag means no restraint is applied in this degree of freedom. By default, all displacements in all three directions are blocked. If necessary, the user can lift up existing restraints or add new ones.

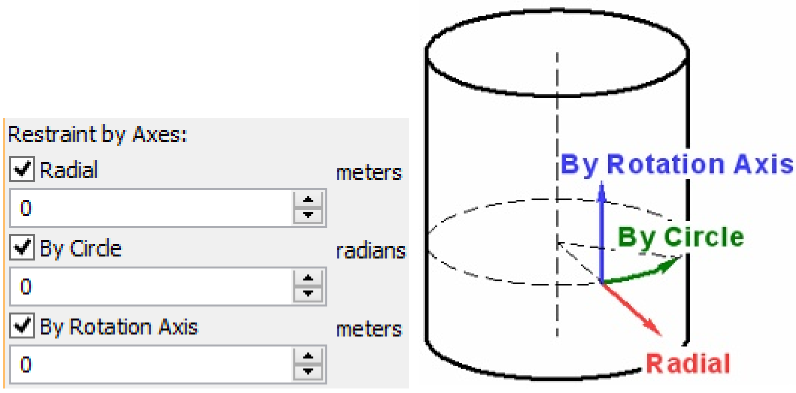


Parameters **Rotation about X**, **Rotation about Y**, **Rotation about Z** are required to specify rotations with respect to the axes of the coordinate system (local or global) to solve problems of plate (or shell) deformation. Given that, triangular elements must be used for discretization of the computational domain.

If the value of the rotation is equal to 0, it means that, along this direction the rotation is fully restrained. If the rotation value is not zero, then the known rotation is specified. The absence of flag (option is turned off) means that restraint of the rotation with respect to the given axis is not defined. By default, restraints of rotations with respect to the axes of the selected coordinate system are absent.

A cylindrical coordinate system allows constraining displacements in:

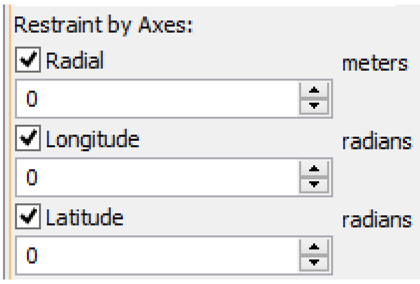
- Radial
- By Cirle
- By Rotation Axis

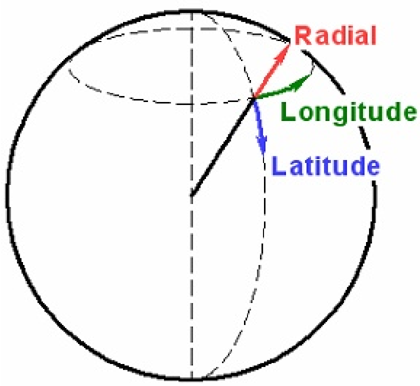


The Rotation Axis is directed along Z axis of global coordinate system, or along Z axis of local coordinate system (if use LCS).

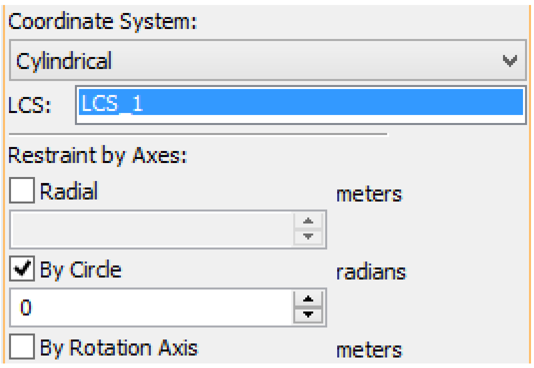
A spherical coordinate system allows constraining dimensions in:

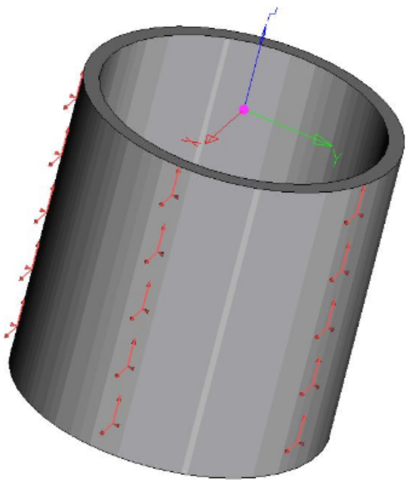
- Radial
- Longitude
- Latitude



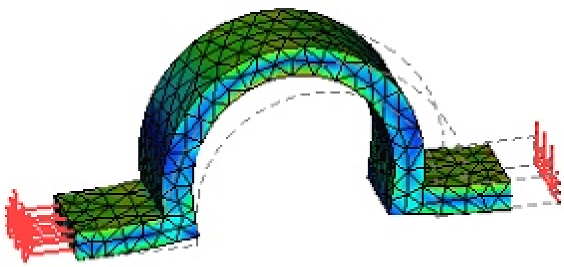


Shown below is an example of a partial restraint on a surface in a cylindrical coordinate system. In this case, partial restraints are defined in the "circumferential" direction, whereas there are no restraints in the radial direction and along the rotation axis, meaning that the revolution about the own axis is excluded for the shown body. The symbolic notation for those restraints is special marks oriented in the respective directions.



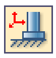


The **Partial Restraint** command also provides another useful functionality. The user can specify known displacements for the structure, such as a known in advance strain in the structure. For this, specify the value of fixed displacement of a model element along some of the coordinate axes in the **Partial Restraint** command's parameters window. Static analysis will be performed with this condition (do not use for dynamic study).



Note that a static solution is possible in this case without applying additional (force) loads. In this way, one can evaluate the stress developing in a strained structure when the quantitative values of the strain (displacements) are known.

Typical sequence for specifying a restraint "Partial Restraint"

1. Initiate the **Partial Restraint** command .

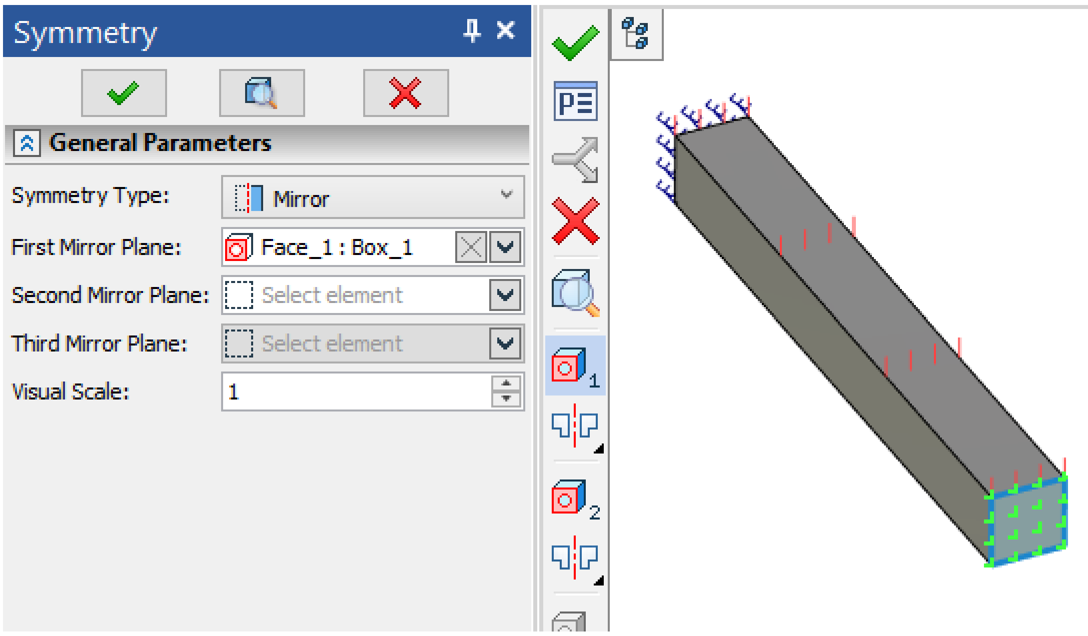
2. Select element to fix.
3. Select LCS.
4. Mark the necessary limits for displacements by the axes and define their values.
5. Complete the command.

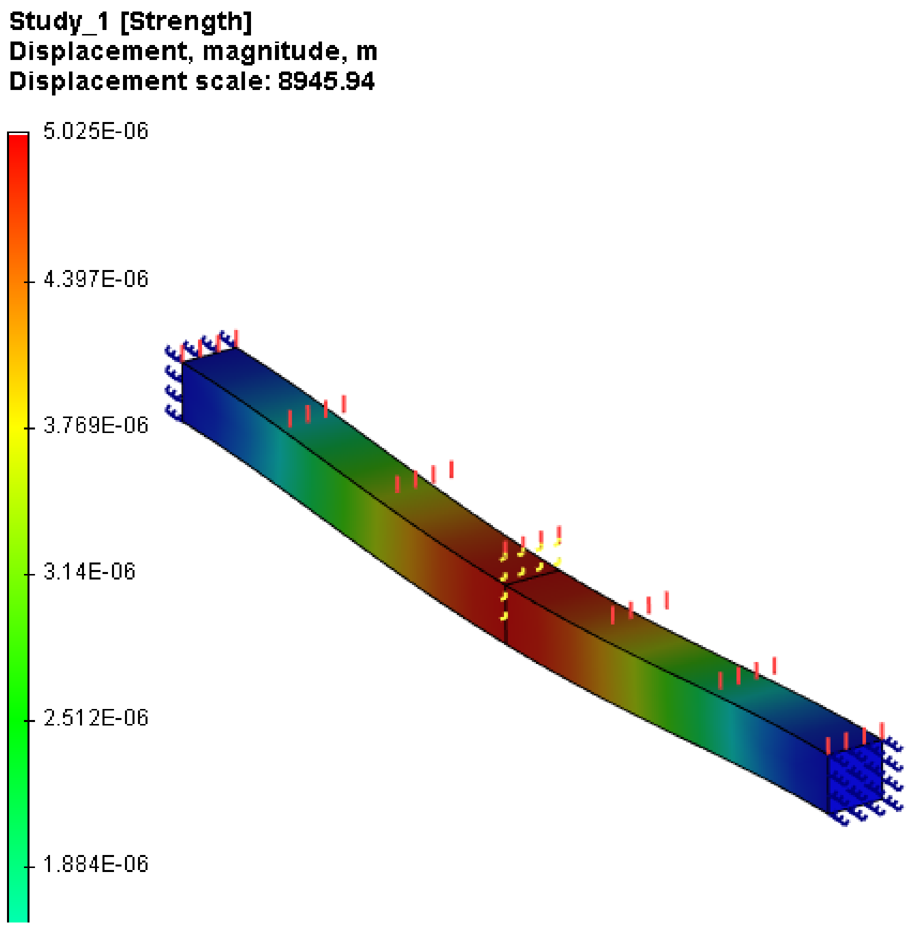
Symmetry

Use the following command for symmetry creation:

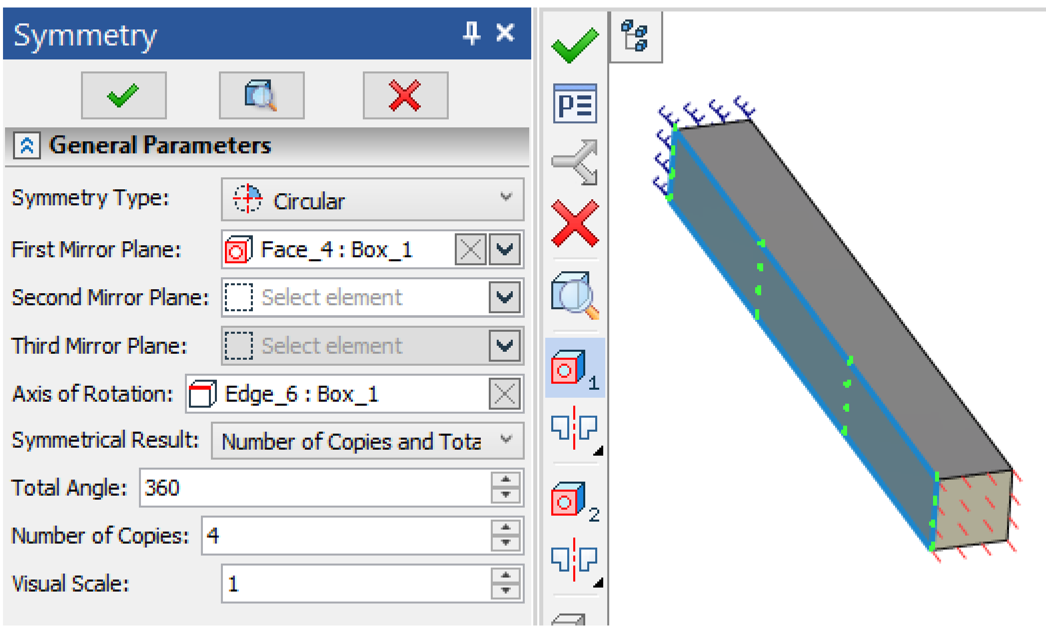
Icon	Ribbon
	Analysis > Conditions > Full Resistance > Symmetry
Keyboard	Textual menu
<3M3D>	Analysis > Resistance > Symmetry

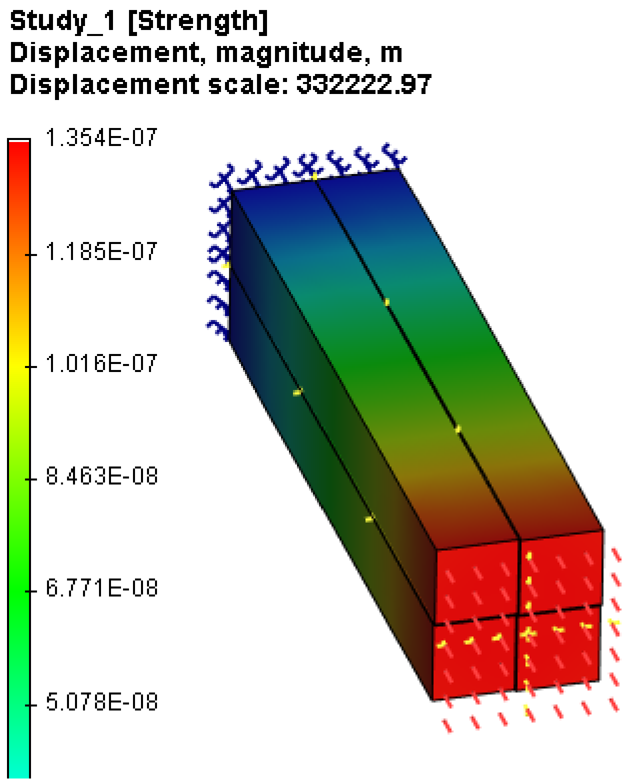
In many practical cases, mechanical structures have a symmetrical geometric shape and can be subjected to symmetrical loading. In these cases, the simulation result will also be symmetric, and it becomes possible to significantly reduce the complexity of the task being solved by calculating only one part of the symmetric structure. To do this, you should correctly set the boundary conditions (fixation). In the general case, in mechanical problems the symmetry condition is specified by the prohibition of displacement in the direction perpendicular to the plane of symmetry in the absence of restrictions in the plane of symmetry. In thermal tasks, to define symmetry, it is enough not to apply any thermal boundary conditions in the plane of symmetry.
Select flat faces of the 3D model to set symmetry planes.





Mirror Symmetry





Circular Symmetry

Typical sequence for specifying a restraint "Symmetry"

- Activate **Symmetry** command;
- Select the body faces that defines symmetry plane;
- Finish the command.

Contact

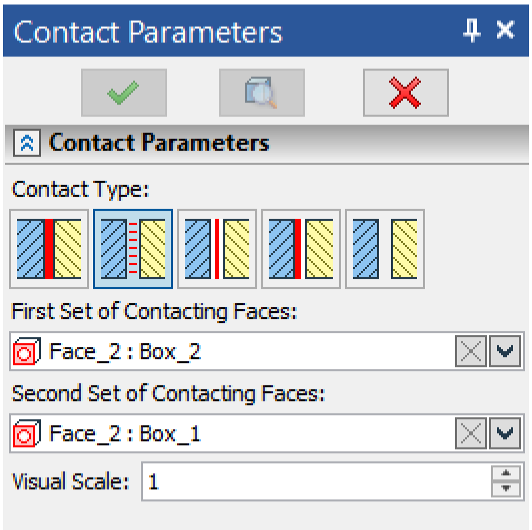
To define a contact, use the command:

Icon	Ribbon
	Analysis > Conditions > Full Restraint > Contact
Keyboard	Textual Menu
<3MI>	Analysis > Restraint > Contact

Contact restraints are needed in studies of contacting bodies.

To define a contact, you need to select the contacting faces of two bodies. Next, select one of four contact types:

- **Rigid Constraint**
- **No Contact**
- **Tangency**
- **Rigid Wall**
- **Gap**



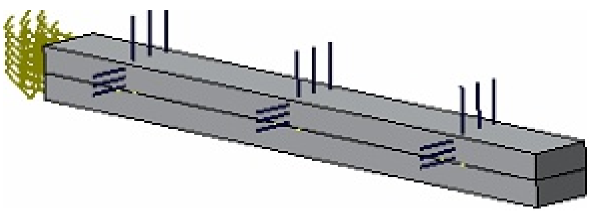
The contact type **Rigid Constraint** is used in the case when it is necessary to bond the contacting surfaces of the bodies. The bodies are considered as bonded in this case, so that relocations of a face in one body result in relocations of the other body faces without any restrictions. If the bodies are made of materials with different physical characteristics, then the finite element model correctly accounts for the different material properties of different faces of contacting bodies.

If the contact area is not subjected to any restraints, then use the **No Contact** type. In this case, the contacting surfaces can freely move with respect to each other. Therefore, when using this contact, one should be on guard against mutual penetration of contacting faces when a load is applied.

The **Tangency** contact differs from **No Contact** in that it bans mutual penetration of the contacting faces. This contact type allows modeling such physical phenomena as sliding of one body along another one, occurrence of gaps at the part connection locations due to deformation, etc. We would like to also note that using the **Tangency** contact implies existence of a physical contact between body faces in the initial state of the structure being analyzed.

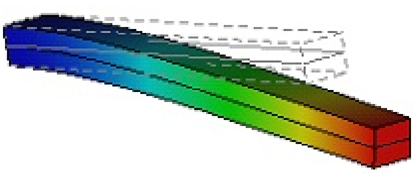
Rigid Wall is used to model a contact of a body with a rigid surface, whose deformation can be neglected for the modeling purposes. In this case, all that is necessary is to define the faces of the first body that contact the **Rigid Wall**.

The dependency of a model behavior on various contact types can be illustrated with the following example. Two beams are kept together using a contact restraint, with one end fixed, and a distributed force acting normal on the top and side surfaces of the first beam.



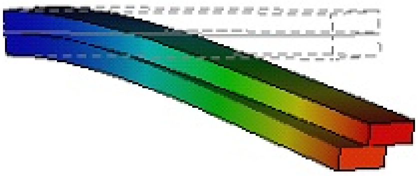
Initial Model

If the **Rigid Constraint** type is used, then the combined beam is deformed as a single solid.



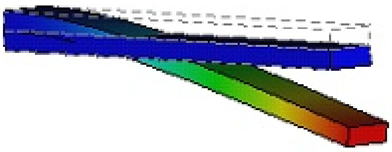
Contact type **Rigid Constraint**

In the case when the **Tangency** contact is used, one can see that the top beam makes the lower one deformed and at the same time slides along it.



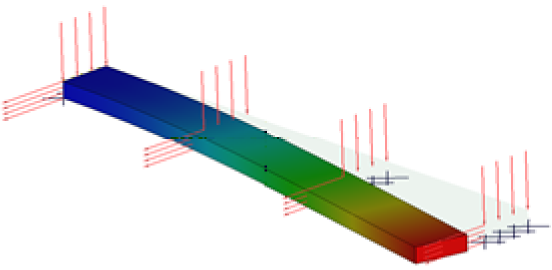
Contact type **Tangency**

When using **No Contact**, one can observe penetration of the top beam into the lower one, which shall be avoided when designing assembly models.



Contact type **No Contact**

The Rigid wall contact type allows to set body faces that specify the surfaces in space which the body can't penetrate (virtual absolutely rigid wall).



Contact type **Rigid wall**

The **Gap** contact type allows you to specify the faces that do not contact in the initial position of the bodies, but for which it is necessary to take into account the contact in the solution process, when the faces will be in contact during the deformation of the bodies.

Default Contact Parameters

It is possible to set contacts by default:

Icon	Ribbon
	Analysis > Conditions > Full Restraint > Default Contact Parameters
Keyboard	Textual Menu
<3MK>	Analysis > Full Restraints > Default Contact Parameters

This command defines the contact type to use by default. This serves to define global contact parameters for all bodies in contact. For example, if a combined structure is being calculated, that consists of several rigidly connected parts, then defining the default contact as **Rigid Constraint** helps avoid manual definition of the contact type for all surfaces in contact. Default contact parameters can be redefined with the help of the **Contact** command.

Typical sequence for specifying a restraint "Contact"

1. Initiate the **Contact** command .
2. Select contacting faces of the first body.
3. Select contacting faces of the second body.
4. Define the contact type.
5. Complete the command.

Additional Stiffness

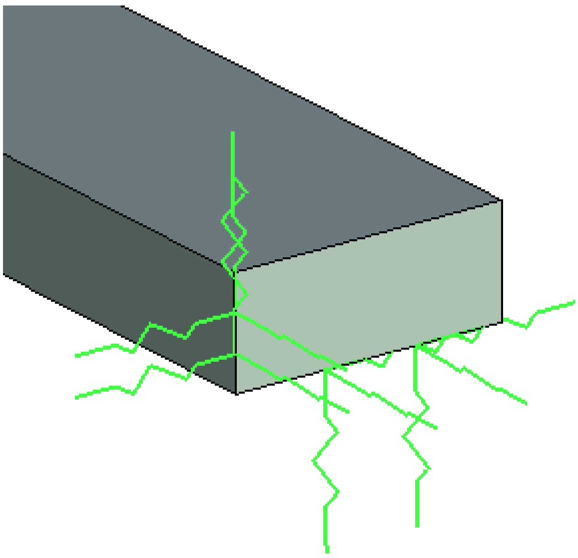
To define an elastic foundation, we use the command:

Icon	Ribbon
	Analysis > Conditions > Full Restraint > Additional Stiffness
Keyboard	Textual Menu
<3M3S>	Analysis > Restraint > Additional Stiffness

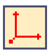
This type of restraints allows us to define elastic interaction on the boundary of a body. Additional stiffness is used when modeling the contact of a body with an external elastic medium which is deformed together with the body. For example, the frame of a machine tool is deformed together with the foundation that is based on elastic damping elements (dampers). Also, a dam supported by the bedrock, a railway etc. can serve as an example of a body connected with the elastic medium.
Mathematically Additional stiffness can be envisioned as a set of weightless springs of identical stiffness attached to the boundary of the body.
After the command is invoked it is required to select elements of the model for application of the load. With the help of the automenu select an element of the model:

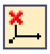
 <E> Select element for fixing

Faces, edges and vertices are available for selection. After the element has been selected, the element's symbolic representation appears in 3D window.



To select the local coordinate system in which the stiffness coefficients will be defined, use the following options:

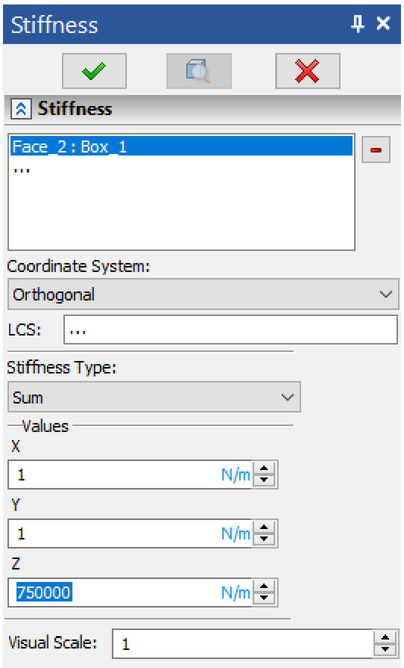
 <C> Select LCS

	<K>	Cancel LCS selection

If LCS is not set, the global coordinate system is used by default.
You can select the following parameters in the parameters window:

- Coordinate system: orthogonal, cylindrical, spherical.
- Stiffness: type sum or distributed.
- Values (1 N/m by default).
- Units: N/m, lbf/in, kgf/cm.

The stiffness type **Sum** is used when the total stiffness of the interacting elastic medium is known and this value is uniformly distributed over the resultant area of faces/resultant length of edges/resultant number of vertices (by one or each direction).
The stiffness type **Distributed** is used when the specific stiffness of the foundation per unit area is known (by one or each direction).



Typical sequence for specifying a restraint "Additional Stiffness"

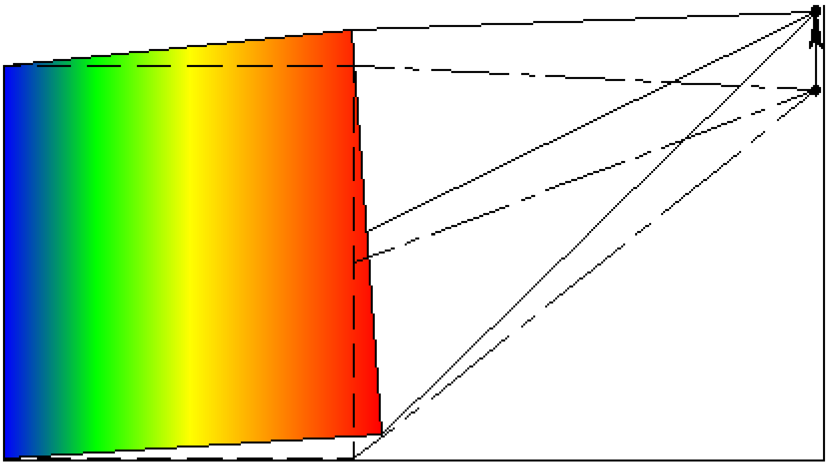
1. Initialize **Additional Stiffness**  command.
2. Select edges, ribs, vertices.
3. Select coordinate system and specify its type.
4. Specify stiffness coefficients values by axes and units.
5. Finish input.

Remote Movement

Use the following command to specify remote movement:

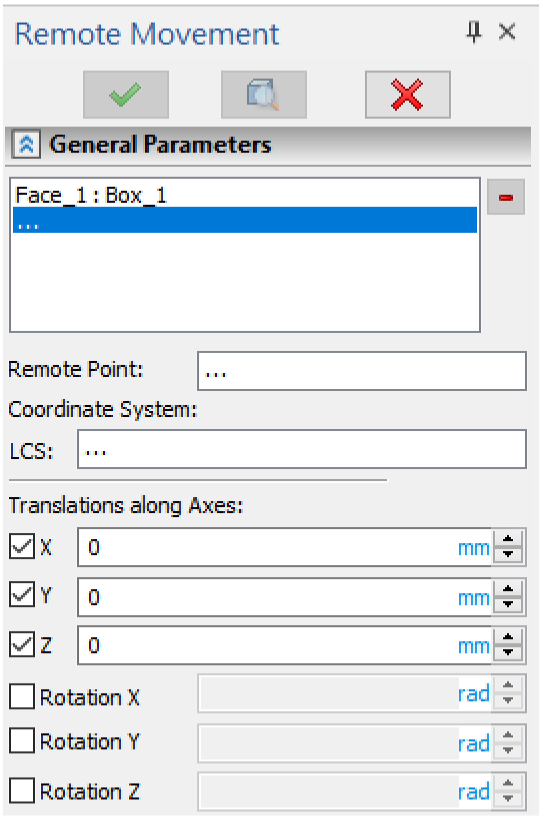
Icon	Ribbon
	Analysis > Conditions > Restraint > Remote Movement
Keyboard	Textual menu
<3M3Z>	Analysis > Restraint > Remote Movement

When specifying a remote movement, the user may set the movement/rotation of a certain point with known coordinates, rigidly connected with the faces/edges of the FE model, in directions/around the axes of the selected LCS.



The remote point is connected by invisible rigid (inelastic) rods with the selected face/edge of the FE model and is moved/rotated in its LCS, while the points are transferred to the face/edge by rigid rods. It should be noted that the selected face/edge of the FE model becomes rigid.

To specify the location of the remote movement application, you should select an edge or a face. It should be noted that if a face is selected, then the edges can no longer be selected and vice versa.



Next you should select a remote point. 3D nodes, vertices, centers of edges, etc. can be selected in accordance with the filter settings.

After that it is necessary to choose the LCS, in which movements/turns will be determined. If the LCS is not selected, the global coordinate system is used.

Next, you should specify the movement of the remote point in three degrees of freedom. The active control element relative to the corresponding degree of freedom of the selected coordinate system means that in this direction a full restriction of movement is set (if the value is 0) or a known movement is specified (if the value in the corresponding text field is non-zero). The absence of a flag means that there is no restriction on this degree of freedom. By default, any movement in all three directions is limited. If necessary, you can remove existing limitations or add new ones.

The parameters **Rotate by X**, **Rotate by Y**, **Rotate by Z** are necessary in order to define rotations about the axes of the coordinate system (local or global). If the value of the rotation is 0, this means that in this direction a complete rotation restriction is set. If the rotation value is not 0, then the current rotation is indicated. The absence of a flag (the option is disabled) means that there is no rotation restriction with respect to this axis. By default, there are no rotation restrictions with respect to the axes of the selected coordinate system.

Typical sequence for specifying a restraint "Remote Movement"

- Initiate the **Remote Movement** command;
- Select faces/edges for load application;
- Select a remote point (3D node or vertex);
- Select a coordinate system;
- Mark the necessary movements/rotations along the axes and set their values;
- Complete the command.

Defining Loads

Loads and fixings are necessary in order to determine the conditions under which the analyzed model is located. The result of the analysis directly depends on the specified loads and fixings. Loads and fixings are applied to geometric objects in the model (bodies, faces, edges, and vertices) and are fully associative, meaning they can be automatically rebuilt when the model topology changes.
They are placed in the **Loading** and **Fixing** folder of the study window tree.
The types of loads and fixings depend on the type of study. You can work with them using the appropriate system commands, as well as the context menu commands that are called when you right-click in the study tree.

Mechanical load

Mechanical initial conditions are used in dynamic analysis studies (Linear dynamic study, calculation of dynamic non-stationary processes).

Thermal load

This type of initial conditions is used for modeling thermal analysis studies
Thermal transfer is the process of transferring heat from an area with a higher temperature to an area with a lower temperature.

Summary table of loads

Load type	Application location	Related objects	Input parameters
Concentrated force	Vertex	Objects selected for defining direction, local coordinate system	Unit system, force amount
Uniformly distributed Force	Face, edge	Objects selected for defining direction, local coordinate system, normal to selected face	Unit system, force amount
Non-uniformly distributed force	Face	Objects selected for defining direction, local coordinate system, normal to selected face	Unit system, force amount, distribution law
Bending moments	Vertex, face, edge	Local coordinate system	Unit system, magnitude of bending moments
Uniform pressure	Face, edge	Objects selected for defining direction, local coordinate system, normal to selected face	Unit system, pressure amount
Non-uniform pressure	Face	Objects selected for defining direction, local coordinate system, normal to selected face	Unit system; pressure distribution law

Hydrostatic pressure	Face	Local coordinate system	Fluid density; unit system

Centrifugal Force	Body	Objects selected for defining axis, local coordinate system	Angular velocity and angular acceleration values, unit system
Acceleration	Body	Objects selected for defining direction, local coordinate system.	Unit system, acceleration amount
Bearing force	Cylindrical face	Objects selected for defining direction, local coordinate system.	Unit system, force amount
Torque	Face	Objects selected for defining axis, local coordinate system	Unit system, moment amount
Additional Mass	Vertex, Edge, Face	Objects selected for defining axis of gravity; local coordinate system	Unit system, mass value
Oscillator	Vertex, Edge, Face	Local coordinate system	Unit system, displacement, velocity or acceleration

Temperature	Body, face, edge, vertex		Magnitude of load, units
Heat Flux	Face		Magnitude of load, units
Heat Power	Body, face, edge, vertex		Magnitude of load, units
Convection	Face		Heat transfer coefficient, temperature of environment, units
Radiation	Face		Radiation type, emissivity, temperature of environment, units, radiation view factor of a face

Topics in this section:

- Mechanical Loads
- Thermal Loads
- Using Graphs to Specify Parameters
- Editing Loads and Restraints

Mechanical Loads

This type of loads is used when modeling the problems of linear and nonlinear static analysis of structures' strength (Static Analysis), when calculating the magnitude of critical buckling loads and, corresponding to them, shapes of structures (Buckling Analysis), and also – for modeling problems of cyclic loading which take into account fatigue phenomena.

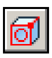
Topics in this section:

Force

Use the following command to set **Force** load:

Icon	Ribbon
	Analysis > Conditions > Force > Force
Keyboard	Textual Menu
<3MF>	Analysis > Load > Force

Force is a type of loading used to specify a concentrated load, and also for specifying a total magnitude of distributed load.
After calling the command, select the model elements for applying the load. Use the following menu option:

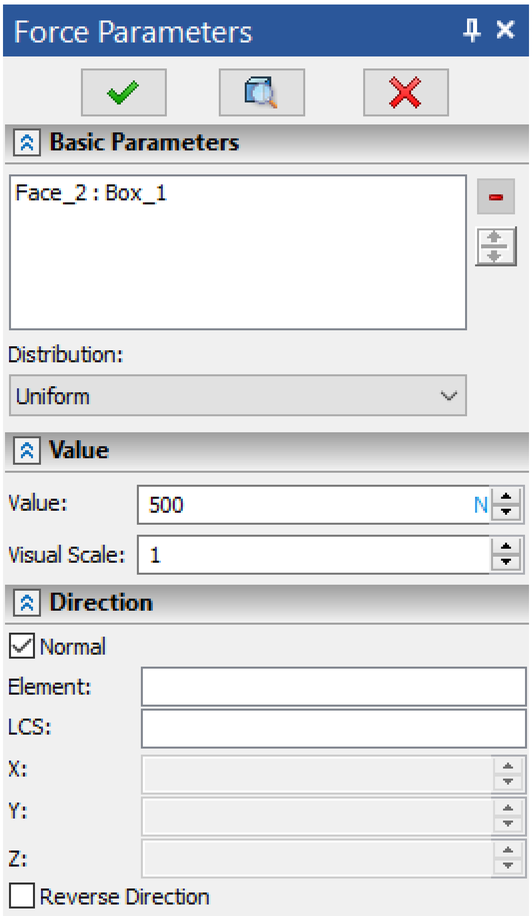
 **<E>** Select Element for loading

Select faces, edges or vertices of the model being analyzed. The selected objects are added to the list.
Also, since with the help of Force, the user can specify the total magnitude of linear (surface) load, it is also necessary to define the way this load is distributed along the edge length (over the face area). In the parameters window, it is required to specify the load type:

- **uniform force**
- **non-uniform force**

Force as a total magnitude of the uniformly distributed load

When defining the **Force** as a total magnitude of uniformly distributed load, it is required to specify the numeric value, units and direction of its action.



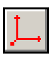
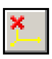
Value. The numeric value is defined as a total equivalent magnitude of uniformly distributed along the edge length (over the face area) load.
When the Force is uniformly distributed, the load per unit length of the edge (per unit area of the face) is equal to the ratio of the specified load magnitude to the edge length (face area).
It is worth noting that if the load magnitude is specified simultaneously for several elements (it is allowed to select the elements of a single type only: vertices, edges or faces), its total value will be distributed between them in the following way:

- The load magnitude equal to the ratio of the specified load to the total area of the faces will be acting on a unit area of each face;
- The magnitude equal to the ratio of the specified load to the total length of edges will be acting on a unit length of each edge;
- Each vertex will be subjected to the part of the force equal to 1/n, where n- is the number of vertices.

You may set dependence of force to time only for nonstationary studies.

> More information can be found in [Using Graphs to Specify Parameters](#).

Units. For the load **Force** the following units can be used: N, lbf, kgf.
Direction. As a direction of the **Force**, the user can select the normal to the loaded face, the element of a 3D model or a certain radius-vector, specified in the selected by the user local coordinate system (if the local coordinate system is not specified, the global coordinate system will be used by default).
To work with the local coordinate system, use the following options:

	<C>	Select LCS
	<K>	Unselect LCS

For specifying the load direction with the help of a 3D model, use the following automenu option:

 <D> Select direction

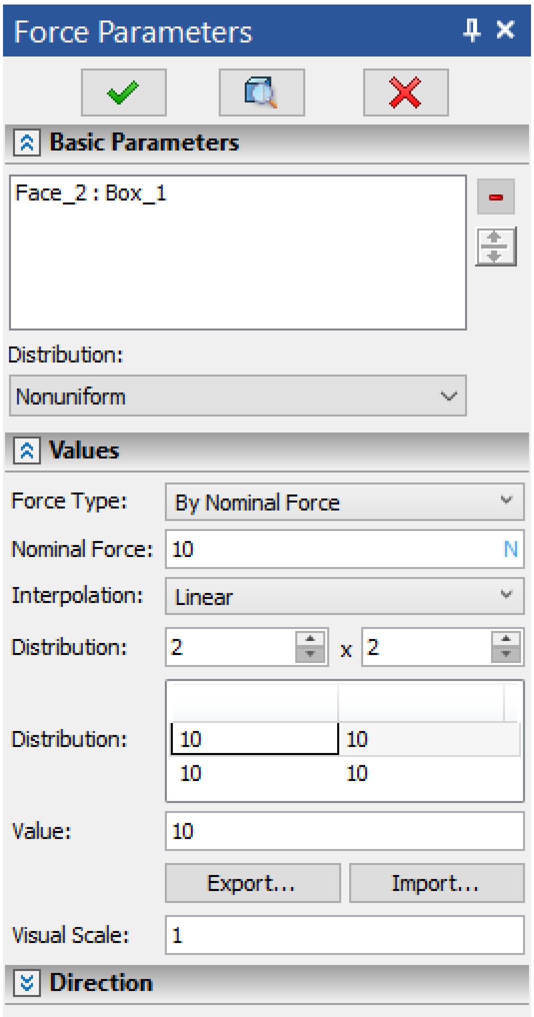
To cancel selection of direction, use the option:

 <U> Cancel direction selection

For a quick change of **Force** direction to reverse, the user can turn on the flag **Reverse Direction**.

Force as a total magnitude of the non-uniformly distributed load

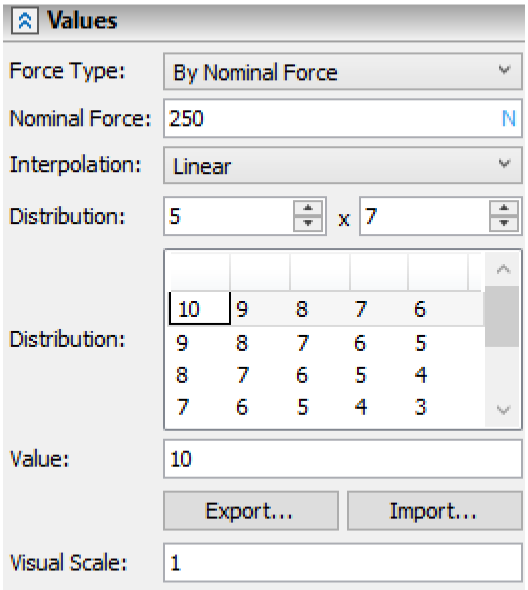
When defining the **Force** as a total magnitude of non-uniformly distributed load, in addition to the numeric value, units and direction of its action, it is required to specify a qualitative distribution law, according to which the load will be distributed over the face area.

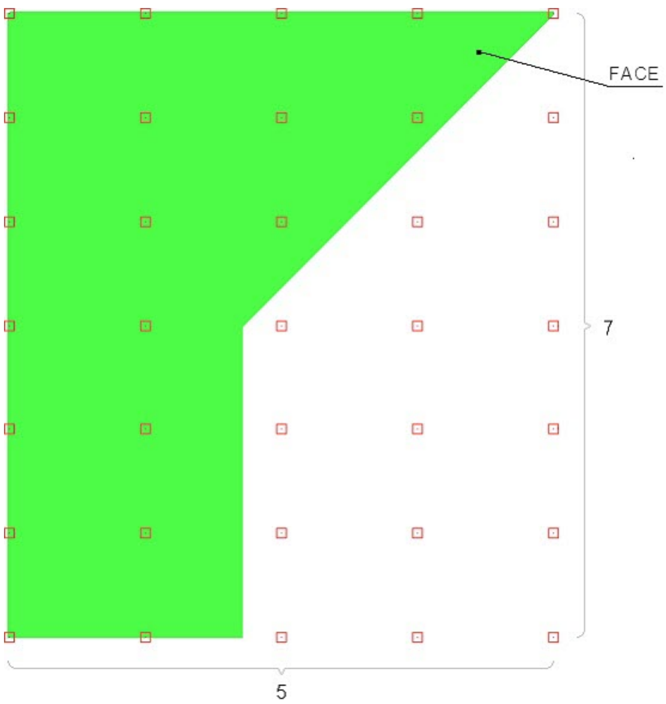


Value. We will define the numeric value of the force as a total magnitude of load non-uniformly distributed over the face area.

Units. For the Force, the following units can be used: N, lbf, kgf.

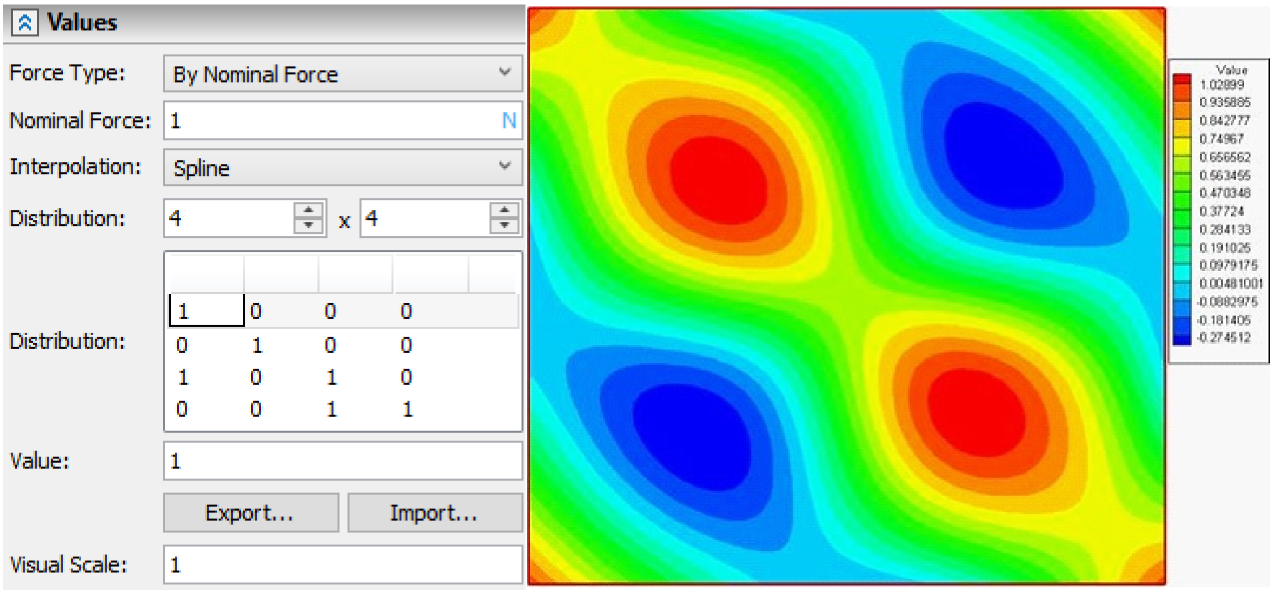
Distribution Law. On the rectangle, circumscribing the face selected as the domain of application of non-uniform load, a uniform grid of nodes is generated. The density of grid is determined by the number of rows and columns in the distribution table. The value of the function at a corresponding node of the grid is specified in the cells of the table.



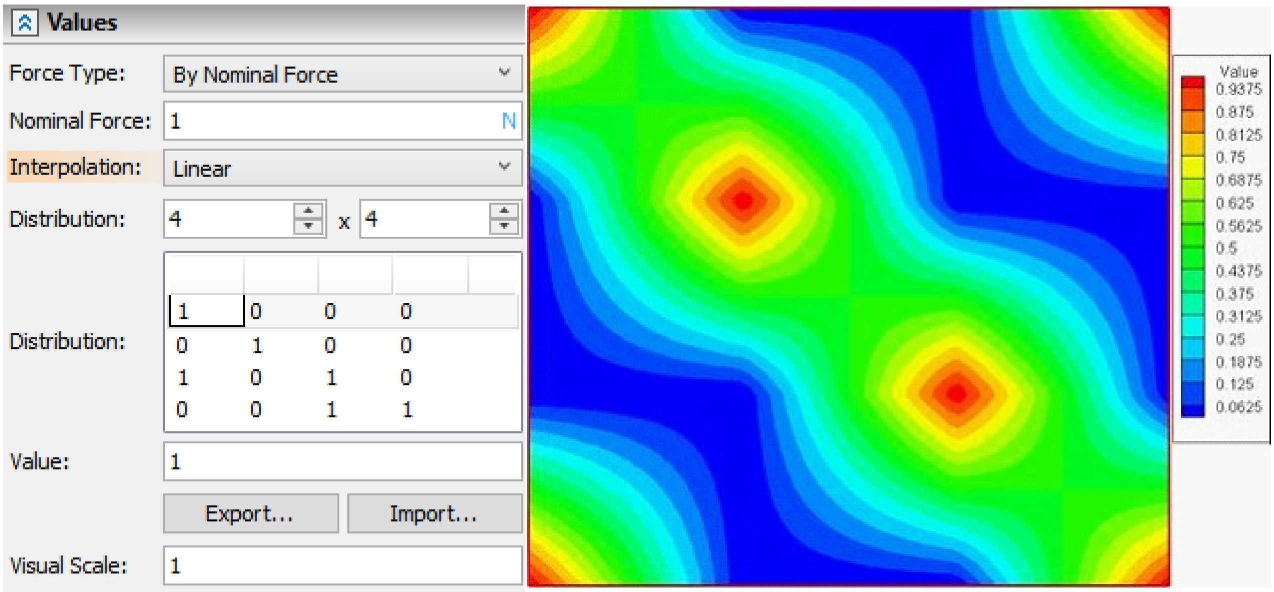


Interpolation. Since the values of the distribution function are known only at the grid points (i.e, specified by the table), it is necessary to extend definition of this function to an arbitrary point on the face. In the T-FLEX Analysis there are two ways of defining the function from the known values: linear interpolation (linear dependence is constructed between the values at the grid points) and spline construction.

Figures show examples of non-uniform load distributions obtained with the help of linear interpolation and spline construction.



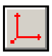
Example of non-uniform load distribution function obtained by spline construction

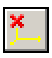


Example of non-uniform load distribution function obtained by using linear interpolation

Direction of Load. For a direction of the Force, the user can select the normal to the loaded face, the element of a 3D model or a radius-vector, specified in the selected by the user local coordinate system (if the local coordinate system is not specified, the global coordinate system will be used by default).

To work with the local coordinate system, use the following options:

 <C> Select LCS

 <K> Unselect LCS

For specifying direction of the force with the help of an object of a 3D model, use the following automenu option:

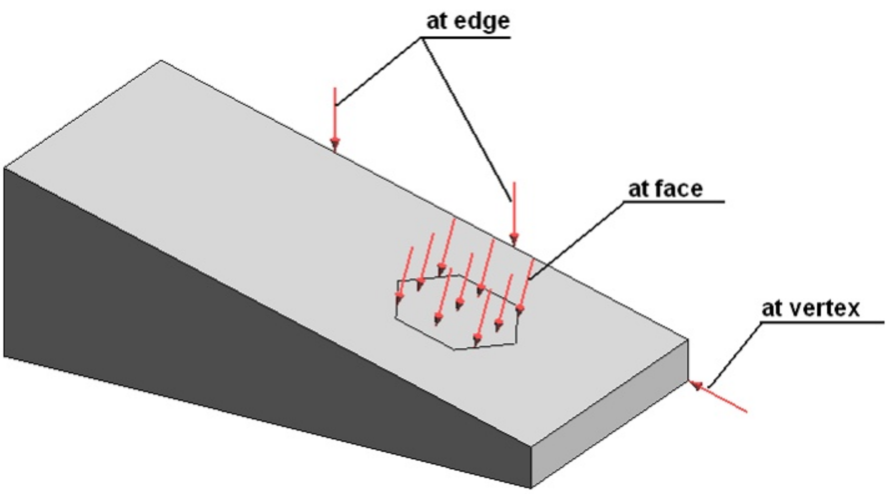
 <D> Select direction

To cancel selection of direction, use the option:

 <U> Cancel direction selection

For a quick change of **Force** direction to reverse, the user can turn on the flag **Reverse Direction**.
In the 3D scene the load **Force** is shown with arrows. The arrows show the direction of the load.
In many cases, the distributed load has to be applied only to a certain part of the edge or face, corresponding to the domain of action of the external load, and not to the entire face or edge of the model.

> To apply the load to the part of the face, first the geometry of the desired shape must be created on the face, and then use the **Imprint Elements** operation on the **Surfaces** tab of the ribbon.



Specifying load **Force**

Typical sequence for specifying a load "Force"

1. Call the command **Force** .
2. Select face, edge, vertex, node or a sequence of elements.
3. Specify the load magnitude.
4. Specify the units.
5. Select the load type: uniform or non-uniform.
6. For a non-uniform load, specify the qualitative distribution law.
7. Specify the direction of load action.
8. Complete the command.

Pressure

Use the following command to set **Pressure** load:

Icon	Ribbon
	Analysis > Conditions > Force > Pressure
Keyboard	Textual Menu
<3MS>	Analysis > Load > Pressure

Pressure represents a loading type used for specifying a distributed load.
After invoking this command, it is required to select the model’s elements for application of load. With the help of automenu option:

 <E> Select Element for loarding

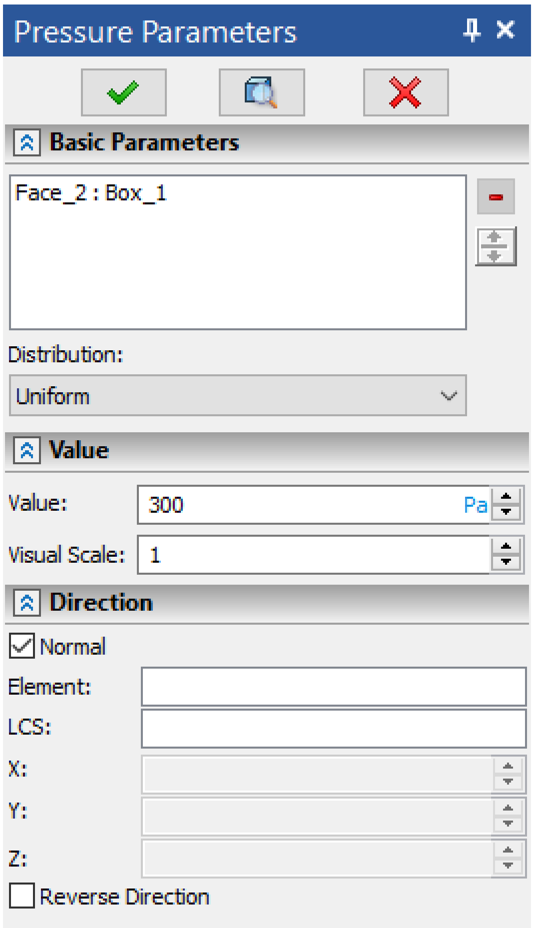
select the face or the edge of the analyzed model. Selected elements will be added to the list.

Since with the help of option Pressure it is possible to define only distributed load, it is required to define the type of this distribution along the edge length or over the face area. In the parameters window, specify the load type:

- **uniform**
- **non-uniform**
- **hydrostatic**

Pressure as a Uniformly Distributed Load

When defining the **Pressure** as a uniformly distributed load, it is necessary to specify the numeric value, units, and direction of its action.



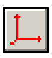
Value. The value is defined as a magnitude of load acting on a unit area of face or unit length of an edge. You can specify dependence of pressure from time only for nonstationary studies.

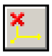
> More information can be found in Using Graphs to Specify Parameters.

Units. For a load Pressure, applied to a face, the following units can be used: N/m2, lbf/in2, kgf/cm2. For a load Pressure, applied to an edge, the following units can be used: N/m, lbf/in, kgf/cm.

Direction. As a direction of Pressure, the user can select the normal to the loaded face, the element of a 3D model or a certain radius-vector, specified in the selected by the user local coordinate system (if the local coordinate system is not specified, the global coordinate system will be used by default). By default, in the local coordinate system, the direction of load is set along the X-axis.

To work with the local coordinate system, use the options:

 <C> Select LCS

 <K> Unselect LCS

To specify the direction of the load **Pressure** with the help of a 3D model, use the automenu option:

 <D> Select direction

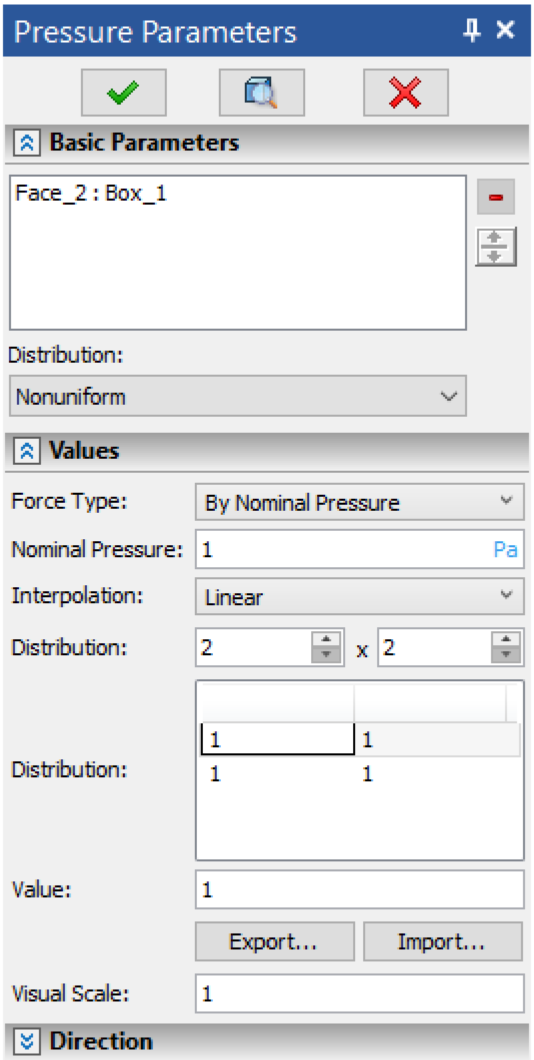
To cancel selection of direction, use the option:

 <U> Cancel direction selection

For a quick change of load direction to an opposite one, the user can activate the flag **Reverse Direction**.

Pressure as a Non-Uniformly Distributed Load

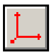
When defining the Pressure as a non-uniform load, it is necessary to specify the units, direction and distribution law according to which a given load will be defined at each point of a face.

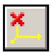


Distribution law. On the rectangle, circumscribing the face selected as the domain of application of a non-uniform load, a uniform grid of nodes is generated. The density of grid is determined by the number of rows and columns in the distribution table. The value of the function at a corresponding node of the grid is specified in the cells of the table.

Interpolation. Since the values of the distribution function are known only at the grid points (i.e, specified by the table), it is necessary to extend definition of this function to any point on the face. In the T-FLEX Analysis there are two ways of defining the function from the known values: bilinear interpolation (linear dependence is constructed between the values at the grid points) and spline construction.

Units. For load **Pressure**, applied to a face, the following units are used: N/m2, lbf/in2, kgf/in2.
Direction. As a direction of **Pressure**, the user can select the normal to the loaded face, the element of a 3D model or a certain radius-vector, specified in the selected by the user local coordinate system (if the local coordinate system is not specified, the global coordinate system will be used by default). By default, in the local coordinate system, the direction of load is set along the X-axis.
To work with the local coordinate system, use the options:

 <**C**> Select LCS

 <**K**> Unselect LCS

To specify the direction of the load Pressure with the help of a 3D model, use the automenu option:

 <**D**> Select direction

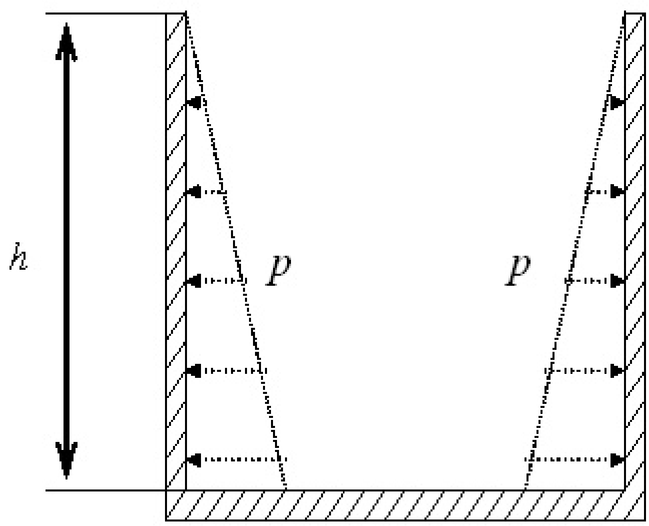
To cancel selection of direction, use the option:

 <**U**> Cancel direction selection

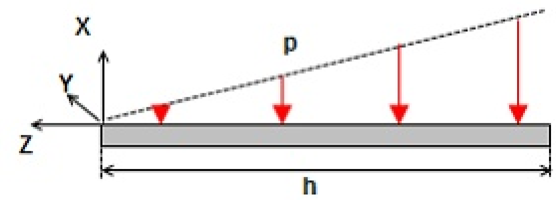
For a quick change of load direction to an opposite one, the user can activate the flag **Reverse Direction**.

Hydrostatic Pressure

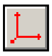
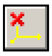
Hydrostatic pressure (or pressure of liquid) is a special case of a non-uniformly distributed load **Pressure**. An example of such load is the liquid pressure exerted on walls of a vessel which changes with height as p= h, where h – is the height of a liquid column of density .



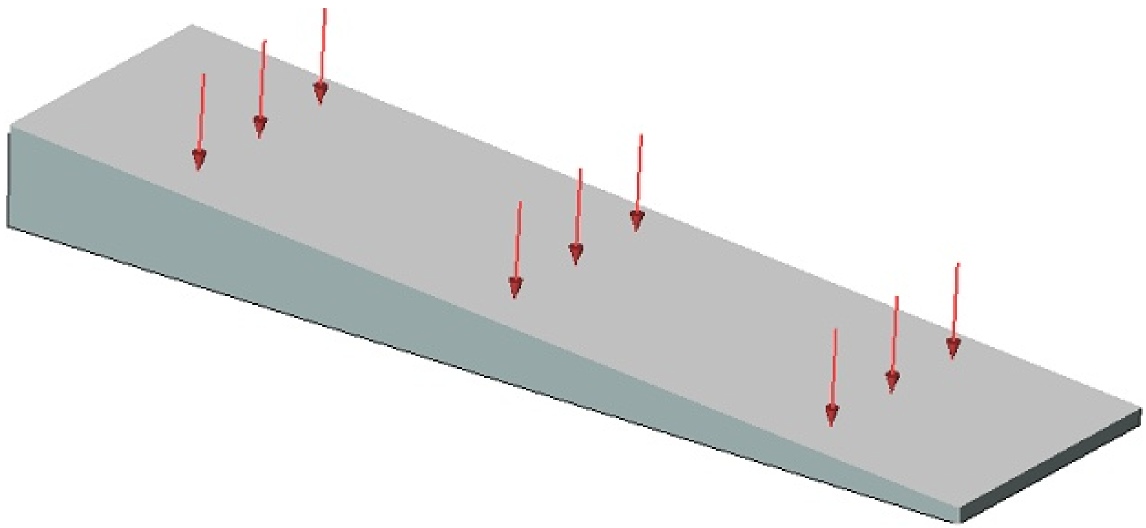
When defining **Pressure** as a hydrostatic load, it is necessary to specify the liquid density, units, and define the direction of load.
Direction of Load Change. The load **Hydrostatic Pressure** acts along the normal to the loaded face. It is required to specify direction of load change, that is, the direction along which the load is increased.
To specify direction of the load change, it is required to select the local coordinate system. The direction of load change (increase) will be determined by the direction opposite to the direction of the Z-axis of the selected coordinate system:



The zero value of the load corresponding to an upper level of liquid coincides in space with the coordinate Z=0 of local coordinate system determining direction of the load change.
To work with the local coordinate system, use the options:

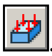
	<C>	Select LCS
	<K>	Unselect LCS

In a 3D scene, the load **Pressure** is shown in the following way:



Specifying load «Pressure»

Typical sequence for specifying a load "Pressure"

1. Call the command **Pressure** .
2. Select face, edge or a sequence of elements.
3. Select load type: uniform, non-uniform or hydrostatic.
4. For uniformly distributed load, specify a numeric value.
5. For non-uniformly distributed load, specify the distribution law.
6. For hydrostatic pressure specify density of liquid and select LCS of upper liquid level.
7. Specify units.
8. Specify direction.
9. Complete the command.

Centrifugal Force (Rotation)

Use the following command to set **Rotation** load:

Icon	Ribbon
	Analysis > Conditions > Force > Rotation
Keyboard	Textual Menu
<3RB>	Analysis > Load > Rotation

Rotation represents a loading type used for simulating a centrifugal force, which arises upon uniform, or uniformly accelerated, rotation of an object.
After invoking this command, select one or several solid bodies for applying the load. They will be added to the list.

It is possible to select all bodies in a study at once using the following automenu option:

<M> Select All Solids

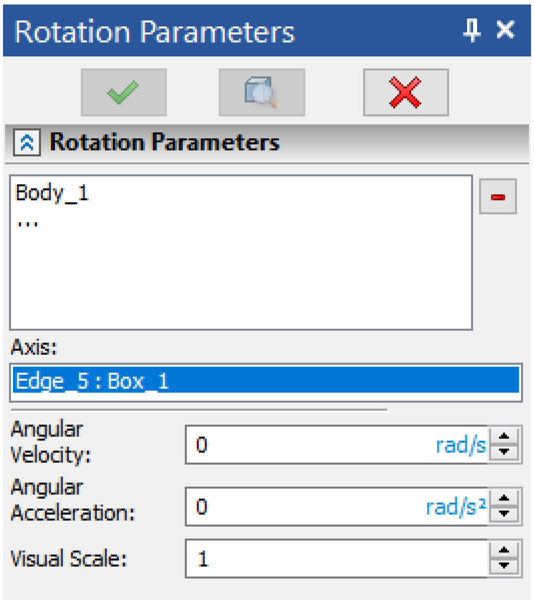
Rotation always takes place about a certain axis, thus, it must be specified. As an axis of rotation, you can use an element of a 3D model (edge, axis of a cylindrical face, etc.), or a specially constructed line (for example, a 3D path constructed by two 3D nodes), or one of the axes of the local coordinate system. Direction of an axis of rotation defines the direction of load (according to a right-hand screw rule).
For selecting a rotation axis, use the automenu option:

<A> Select axis of rotation

As the rotation axis, either an element of a 3D model (an edge, an axis of a cylindrical face, etc.) or a specially constructed line (for example, a 3D path constructed from two 3D nodes) or one of the axes of the local coordinate system.
To cancel selection of axis of rotation, use the option:

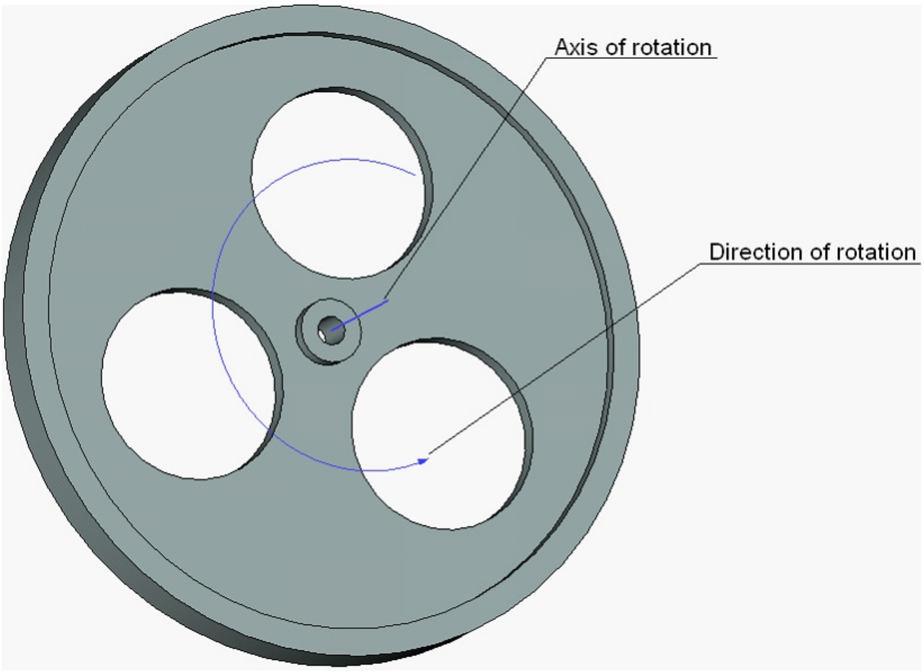
<C> Cancel axis selection

In the load's parameters window, it is necessary to specify the magnitude of angular velocity and angular acceleration.



For angular velocity, the following units can be used: radian per second [rad/sec], degrees per second [deg/sec], the number of revolutions per second [Hz], the number of revolutions per minute [rpm].

For angular acceleration, the following units can be used: radian per second squared [rad/sec^2], degrees per second squared [deg/sec^2], the number of revolutions per second squared [Hz/sec^2], the number of revolutions per minute squared [rpm^2].
In the 3D scene the load **Rotation** is shown in the following way:



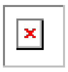
Specifying rotation

Typical sequence for specifying a load "Rotation"

1. Call the command **Rotation** .
2. Specify axis of rotation.
3. Specify the magnitude and units for angular velocity and angular acceleration in the command's parameters window.
4. Complete the command.

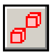
Acceleration

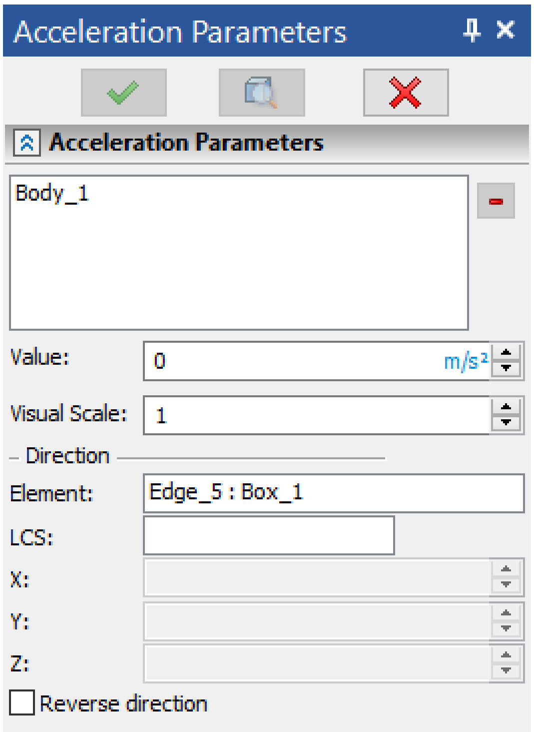
Use the following command to set **Acceleration** load:

Icon	Ribbon
	Analysis > Conditions > Force > Acceleration
Keyboard	Textual Menu
<3MA>	Analysis > Load > Acceleration

Acceleration creates a uniform impact on any body with a mass. This impact is uniformly distributed over the entire volume of the selected body. Use of this type of loading allows, for example, simulating the load of the own weight under the force of gravity.
After invoking this command, it is necessary to select the body (or several bodies) for applying the load. They will be added to the list.

It is possible to select all bodies in a study at once using the following automenu option:

 <M> Select All Solids



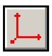
In the parameters window specify:

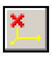
- Value of load (you can specify dependence of acceleration from time for nonstationary studies.

More information in [Using Graphs to Specify Parameters] section.

- Units: m/sec2, cm/sec2, in/sec2;
- Direction of load.

Direction of load. As a direction of acceleration, the user can select the element of a 3D model or a certain radius-vector, specified in the selected by the user local coordinate system (if the local coordinate system is not specified, the global coordinate system will be used by default).
By default, in the local coordinate system the direction of load is set along the X-axis.
To work with the local coordinate system, use the options:

 <C> Select LCS

 <K> Unselect LCS

For specifying direction of load **Acceleration** with the help of an object of a 3D model, use the following automenu option:

 <D> Select direction

To cancel direction selection, use the option:

 <U> Cancel direction selection

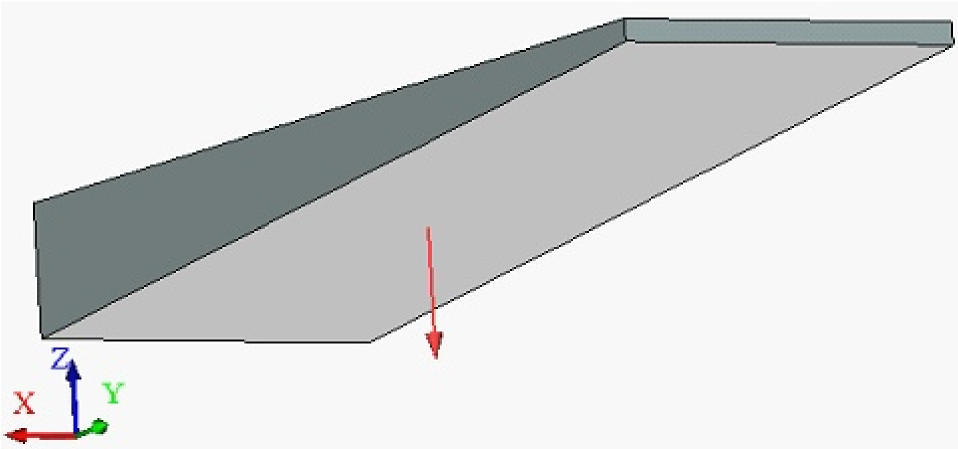
For a change of load direction to an opposite one, the user can activate the flag «Reverse direction».

Gravity

Icon	Ribbon
	Analysis > Conditions > Force > Gravity
Keyboard	Textual Menu
<3MG>	Analysis > Load > Gravity

For a quick specification of the gravity force, there is a **Gravity force** command, which sets the value of acceleration equal to ~9.81 m/s^2 and sets the direction of load along the Z-axis of the global coordinate system equal to -1 for all bodies in a study.

In the 3D scene, the load **Acceleration** is shown in the following way:



Specifying load «Acceleration»

Typical sequence for specifying a load "Acceleration"

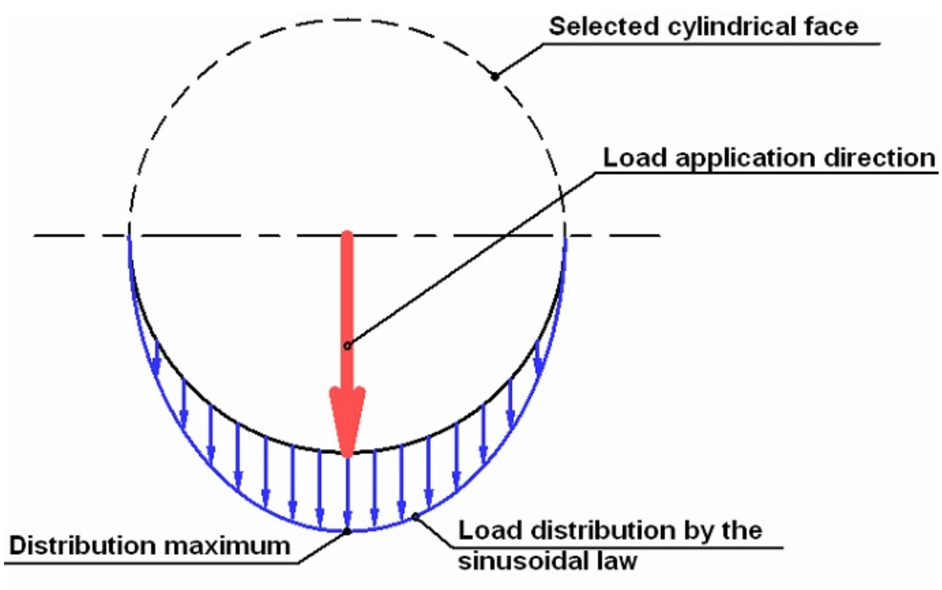
1. Call the command **Acceleration** .
2. Specify the load magnitude.
3. Specify the direction of load
4. Complete the command.

Bearing Force

Use the following command to set **Bearing Force** load:

Icon	Ribbon
	Analysis > Conditions > Force > Bearing Force
Keyboard	Textual Menu
<3MB>	Analysis > Load > Bearing Force

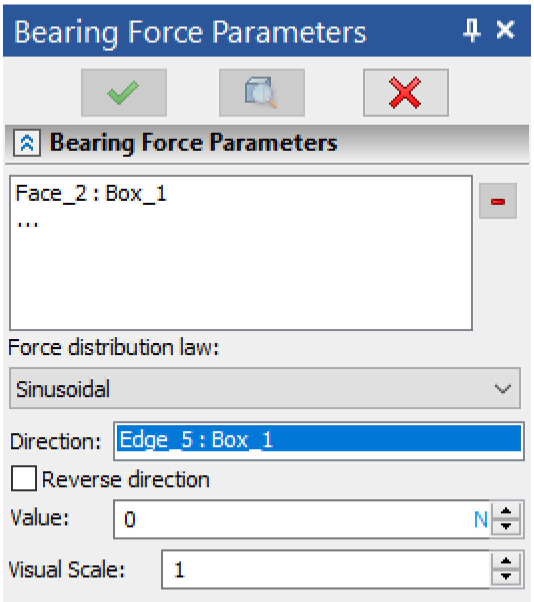
Bearing Force simulates the load occurring under a direct impact of such parts as an axle, a bearing or a shaft. A cylindrical face is used as the location for applying the load. The applied force is distributed according to the sinusoidal law (see the diagram).



After invoking this command, it is required to select elements (cylindrical faces) of the model for applying the load. With the help of automenu option:

 <F> Select cylindrical face

select a cylindrical face of an analyzed model. The selected element will be added to the list.



In the parameters window specify:

- Magnitude of load;
- Units: N; kgf, lbf;
- Direction of load.

Load direction. As a direction of Bearing load, the user can select the element of a 3D model or coordinate axes of the coordinate system.

You can specify dependence of load from time for nonstationary studies.

More information in Using Graphs to Specify Parameters section.

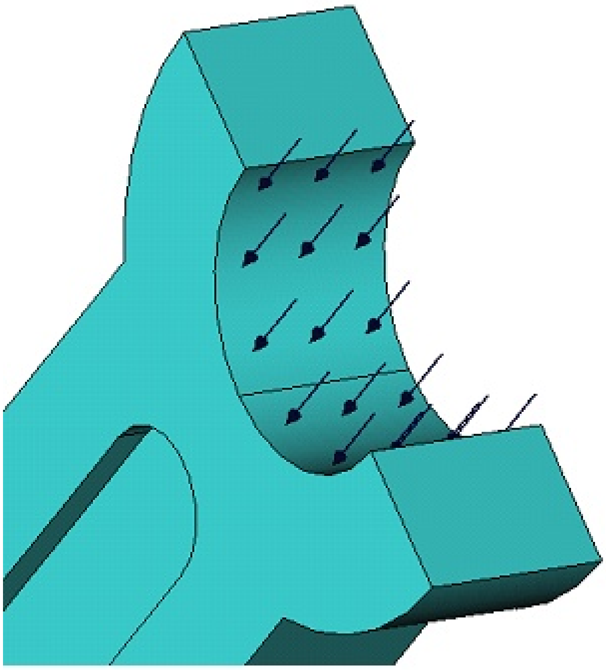
For specifying the direction of Bearing load use the automenu option:

 <D> Select direction

As the direction of the load, either an element of a 3D model (an edge, an axis of a cylindrical face, etc.) or a specially constructed line (for example, a 3D path constructed from two 3D nodes) or one of the axes of the local coordinate system.
To cancel selection of direction, use the option:

<U> Cancel direction selection

To change a direction of load to an opposite one, the user can activate the flag **Reverse direction**.
In the 3D scene **Bearing Force** is shown in the following way:



Specifying «Bearing force»

Typical sequence for specifying a load "Bearing Force"

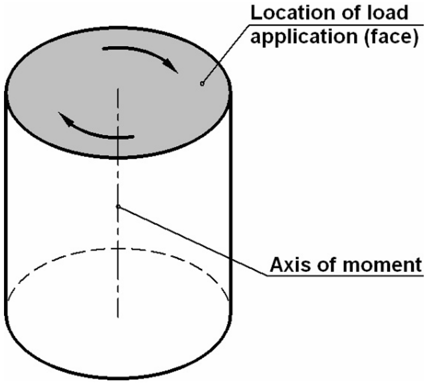
1. Call the command **Bearing Force** .
2. Select a cylindrical face or a set of faces.
3. Specify the magnitude of load.
4. Specify the load direction
5. Complete the command.

Torque

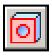
Use the following command to set **Torque** load:

Icon	Ribbon
	Analysis > Conditions > Force > Torque
Keyboard	Textual Menu
<3MQ>	Analysis > Load > Torque

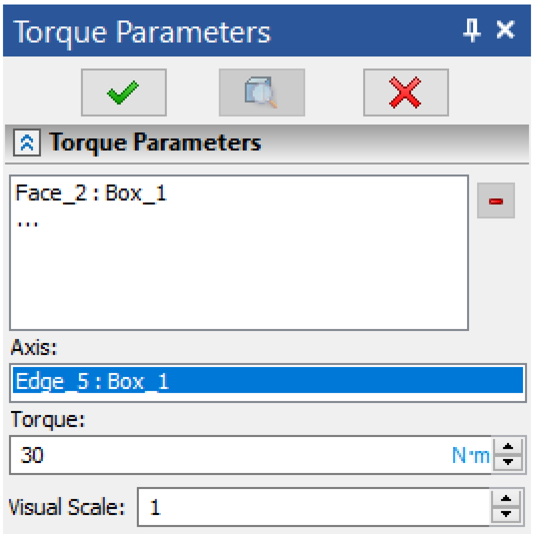
Torque is the impact of a force moment of the specified magnitude distributed over the selected face.



You can use faces as the location of the load application. To select faces, use the automenu option:

 **<F>** Select Face

Selected faces are entered in the list.



In the parameters window specify:

- The magnitude of load;
- Units: N-m, kgf-cm, lbf-in;
- The axis of torque.

The direction of the torque axis (or axis of rotation) defines the direction of load (according to a right hand rule). As a direction of axis of rotation, the user can select an element of a 3D model (edge, axis of a cylindrical face, etc.), or a specially constructed line (for example, a 3D path, constructed by two 3D nodes), or one of the axes of the local coordinate system.

You can specify dependence of load from time for nonstationary studies.

> More information in Using Graphs to Specify Parameters section.

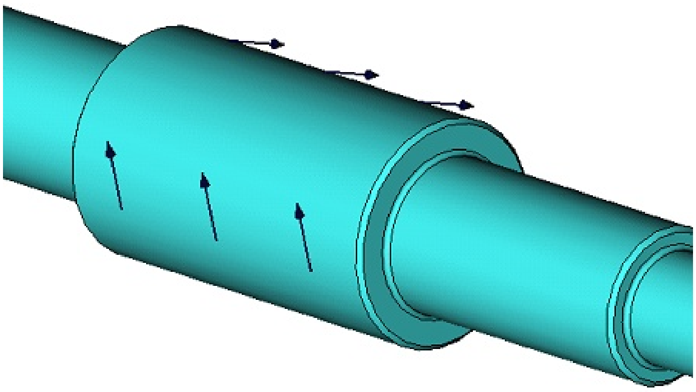
To select the axis of rotation, use the automenu option:

 **<A>** Select axis of rotation

To cancel selection of axis rotation, use the option:

 **<C>** Cancel axis selection

In the 3D scene the load **Torque** is shown in the following way:



Specifying load **Torque**

Typical sequence for specifying a load "Torque"

1. Call the command **Torque** .
2. Select loaded faces of body
3. Specify magnitude of load
4. Specify the axis of moment
5. Complete the command.

Oscillator

Use the following command to set **Oscillator** load:

Icon	Ribbon
	Analysis > Conditions > Force > Oscillator
Keyboard	Textual Menu
<3M3O>	Analysis > Load > Oscillator

Oscillator (kinematic loading) constitutes a type of loading used in dynamic analysis studies: Forced Oscillation, Mode Superposition, Transitional Processes to specify the so-called **kinematic excitation of vibrations**. In the **Forced Oscillation** study, an oscillator is used for specifying harmonic oscillatory motion of structure element experiencing vibration. In the other two studies, it is possible to specify a displacement of the structure or its elements according to the arbitrary time graph.

More information in Using Graphs to Specify Parameters section.

After this command is invoked, it is necessary to select the elements of the model for specifying the loadings. With the help of the option of the automenu:

<E> Select element for kinematic loading

Select elements of the analyzed model. The selected element will be added to the list.
It is possible to select all bodies in a study at once using the following automenu option:

<M> Select all bodies

In the Parameters window specify:

- Type of the values of the loading along the axes;
- Magnitude and direction of the loading;
- Units for different types of loadings: m, m/s, m/s^2;

- Phase shift, units: degrees, radians.

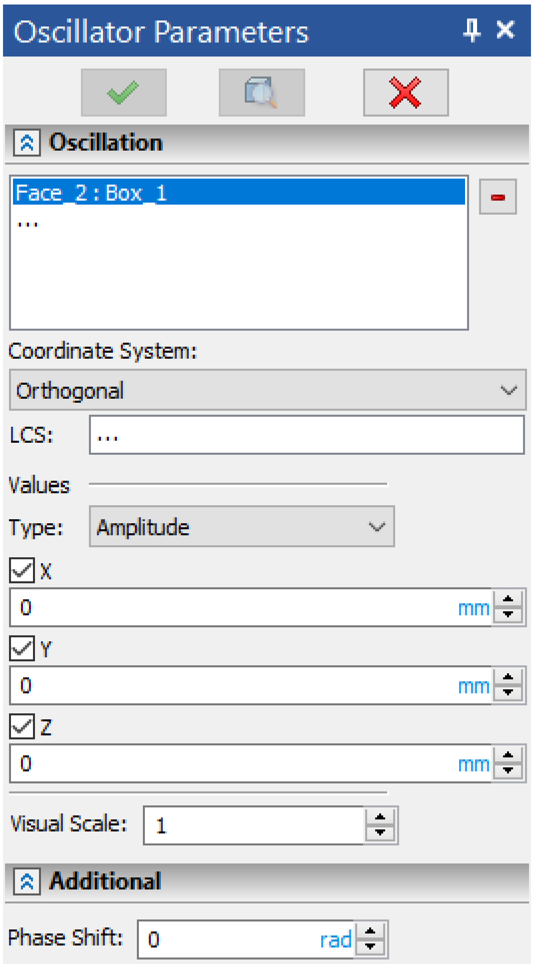
Values along the axes, type. Before specifying the values of the kinematic loading, it is required to select the type of the parameter that determines the amplitude of vibrations. Quantitatively, the amplitude of the motion of an element of the structure can be specified in several equivalent ways via the amplitudes of: displacements U_m, mm, cm, m, in, ft;

velocity $U_m = U_m \cdot \omega_f$

acceleration $U_m = U_m \cdot \omega_f^2$ *mm/s², cm/s², m/s², in/s², ft/s².*

overloading $U_m / g = U_m \cdot \omega_f^2 / g$, times;

where ω_f – frequency of forced vibrations, rad/s; *g* – gravitational acceleration, m/s².

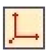
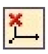


As can be seen, all variables, except the amplitude of displacement, are secondary and are uniquely expressed in terms of the amplitude of vibration and frequency of the forced vibration.

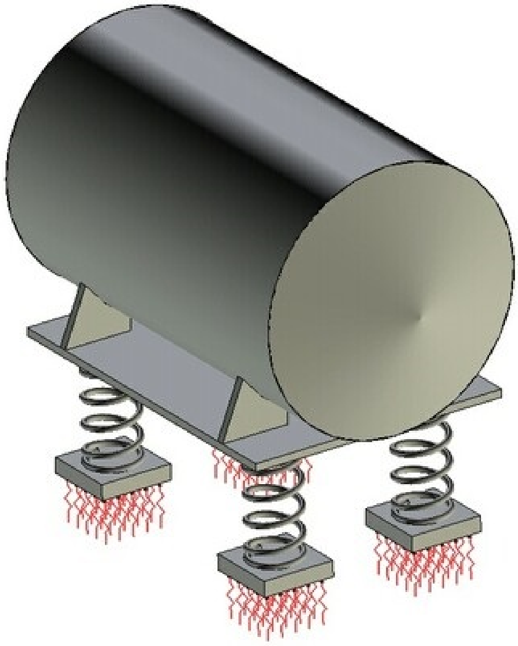
The frequency of vibration is specified in the analysis settings dialog called **Forced harmonic vibrations** right before the execution of analysis.

Magnitude and direction of action of kinematic loading. As a direction of loading, it is possible to select the radius-vector defined in the local coordinate system selected by the user (if the local coordinate system is not specified, the global coordinate system will be used by default). Fill in the check boxes opposite those axes along which the magnitude of the corresponding component of the load will be specified.

To work with the local coordinate system, use the following options:

	<C>	Select coordinate system
	<K>	Cancel selection of coordinate system

The values of the components of the loading, decomposed along the coordinate axes, are entered into the fields corresponding to the axes marked with a check. By default, the values for the amplitude of displacements are specified in mm.
When, in the study, there are several harmonic excitations (kinetic or kinematic) the **Phase Shift** parameter is used to specify lagging or advancing phase shift of one loading with respect to another one.
In 3D scene the **Oscillator** load is displayed in the following way:



Displaying **Oscillator** loading

Typical sequence for specifying a load "Oscillator"

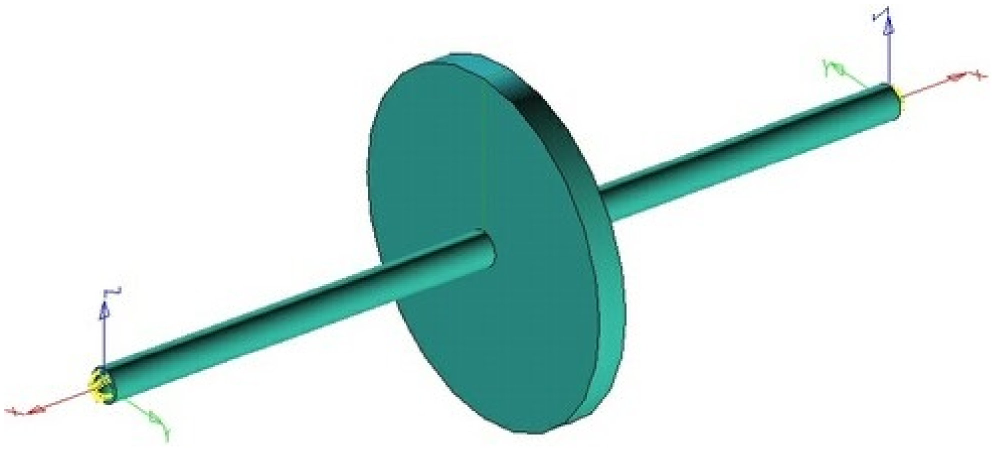
1. Call the **Oscillator** command .
2. Select faces, edges, vertices and/or bodies; select the coordinate system.
3. Specify values and units for the vibrational amplitudes of components of displacements, velocity or acceleration in the command's parameters window.
4. Specify phase shift and its units.
5. Complete the command.

Additional Mass

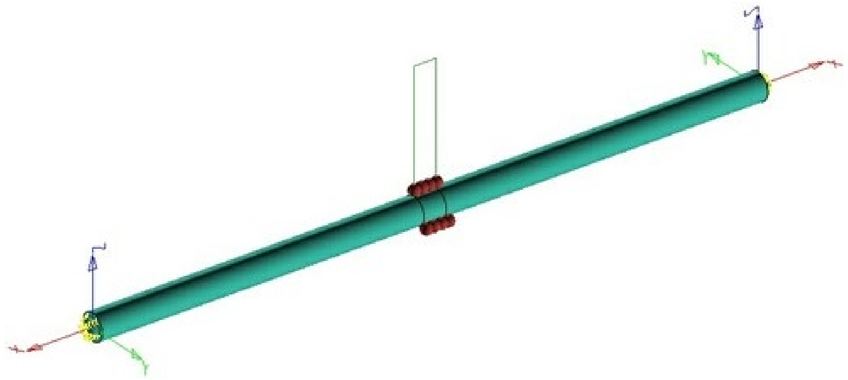
Use the following command to set **Additional Mass** load:

Icon	Ribbon
	Analysis > Conditions > Force > Additional Mass
Keyboard	Textual Menu
<3M3M>	Analysis > Load > Additional Mass

Additional mass constitutes a type of loading used for specifying additional inertia load produced by the part of the structure not included explicitly into the study. Used in all types of studies.



Initial full-size model

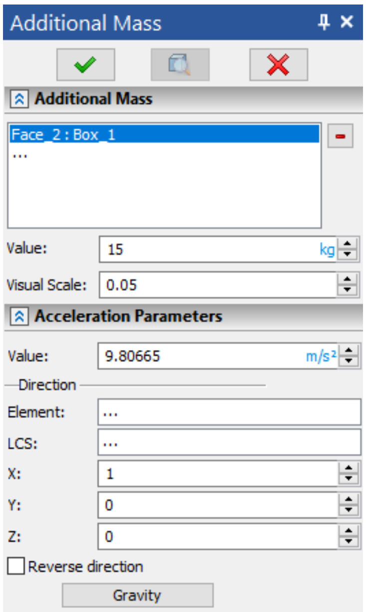


Replacement of the disk with the **Additional Mass**

After calling this command, it is necessary to select the elements of the model for applying the load. With the help of the option of the automenu:

 <E> Select element for loading

select faces, edges or vertices of the analyzed model. Selected elements will be added to the list. Select bodies of the analyzed model. They will be added to the list.



On the **Load Parameters** tab specify:

- Magnitude of loading in kg; it is possible to use variables which contain the result of measurement of the mass of some body or operation. You can specify dependence of mass from time for nonstationary studies.

More information in Using Graphs to Specify Parameters section.

- Units of measurement: kg, pounds;

On the **Acceleration Parameters** tab specify:

- Acceleration value;
- Units: m/s^2, in/s^2, cm/s^2.

It is to be noted that the replacement of the body with an additional mass applied to the same faces which were touched by the replaced body, simplifies the finite element model, but at the same time does not allow us to take into account the spatial distribution of mass. This fact must be considered especially in frequency analysis problems, in which the spatial distribution of mass plays a key role in forming the spectrum of natural frequencies of the structure. Therefore, such a replacement is usually admissible only for calculation of the first (lowest) natural frequency.

Typical sequence for specifying a load "Additional Mass"

1. Call the **Additional Mass** command .
2. Select faces, edges, vertices.
3. Specify values and units of measurement.
4. Specify values and units of acceleration,
5. Complete the command.

Remote Mass

Use the following command to set **Remote Mass** load:

Icon	Ribbon
	Analysis > Conditions > Force > Remote Mass
Keyboard	Textual Menu
<3M3Y>	Analysis > Load > Remote Mass

Remote mass is a type of loading that is used to specify an additional inertial load from a part of the structure that is not explicitly included in the task. In contrast to the additional mass, the load is applied to the distance point, which represents the center of mass of the discarded part of the structure, and is not simply distributed evenly along the face. In static tasks, only mass is taken into account. In dynamic tasks, inertia moments are taken into account as well. It is used in all types of tasks.

You should select the model elements to apply load after calling of the command. Use the following automenu option:

<E> Select element for loading

Select faces, ribs and vertexes of the calculated model. The selected elements will be added to the list.

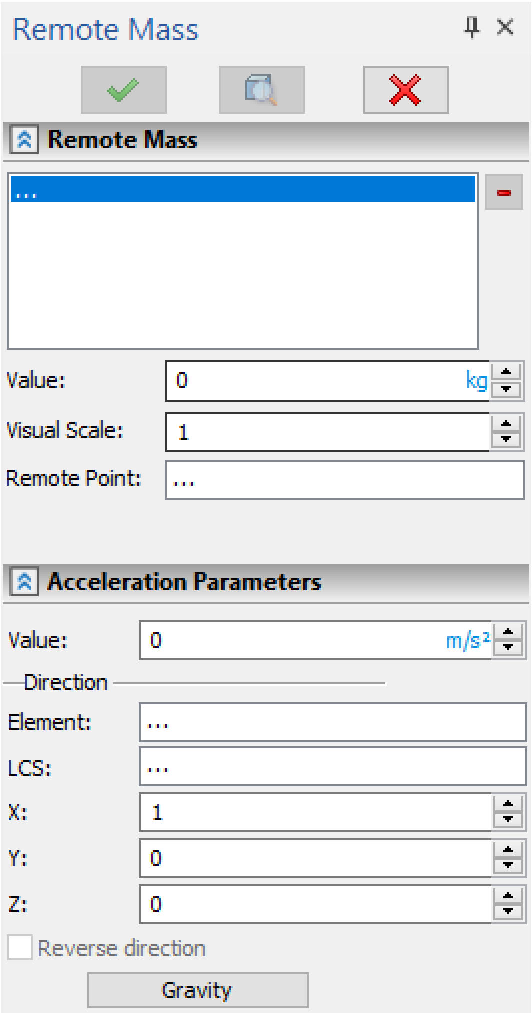
There are features of the load applying in the statics and dynamics tasks.

In the statics studies you should specify:

- Value. You can use variables in which the result of measuring the mass of a body or operation is recorded. In non-stationary tasks it is possible to specify the dependence of the mass value on time.

More information in Using Graphs to Specify Parameters section.

- Mass measurement units: kg, lbs.
- Remote point. This point represents the center of mass of the discarded part of the structure, replaced by the distance mass. As a point, you can choose a 3D node, the beginning of an LCS, the top of a wire or solid geometry, etc. according to filter settings.
- In the **Acceleration Parameters** group, set the acceleration value that determines the effective weight;
- Acceleration units: m/s^2, cm/s^2, $inch/s^2$ lbs/s^2.
- LCS for the direction of acceleration.
- Performing the last 3 steps can be replaced by pressing the **Gravity** button.



Statics

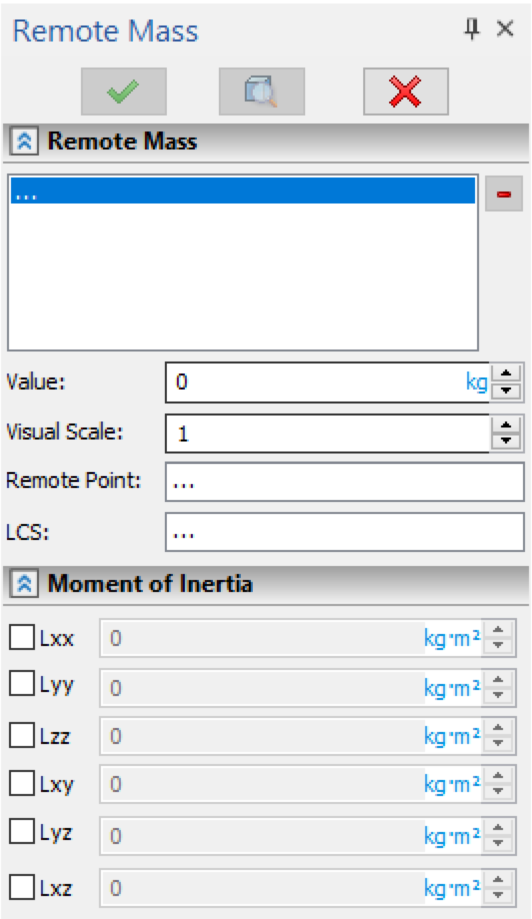
In the dynamics studies, you should specify:

- **Remote point**. This point represents the center of mass of the discarded part of the structure, replaced by the distance mass. As a point, you can choose s 3D node, LCS, the top of a wire or solid geometry, etc. according to filter settings.

- **Value**. You can use variables in which the result of mass measuring of a body or operation is recorded. In non-stationary tasks it is possible to specify the dependence of the mass value on time.

More information in Using Graphs to Specify Parameters section).

- Mass measurement units: kg, lbs;
- Moment of inertia. Moment of inertia is calculated according to a coordinate system placed at the center of gravity of the discarded structure, replaced by a remote mass, the axes of which are aligned with the axes of the global coordinate system. For structures symmetrical with respect to the LSC, the moments of inertia of Lxx, Lyy, Lzz are sufficient. The moments may coincide depending on the number of axes of symmetry.
- Units of moment of inertia: kg/m^2, g/cm^2, lbs/in^2 lbs/ft^2.
- LCS according to which the moments of inertia are calculated.



Dynamics

Typical sequence for specifying a load "Remote Mass"

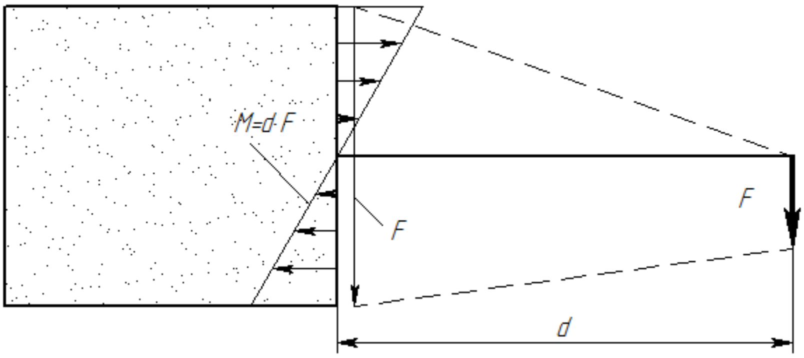
1. Call the **Remote Mass** command;
2. Select faces, edges, vertices;
3. Set values and units;
4. Set the value and direction of acceleration;
5. Complete the command.

Remote Force

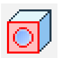
Use the following command to set **Remote Force** load:

Icon	Ribbon
	Analysis > Conditions > Force > Remote Force
Keyboard	Textual menu
<3M3F>	Analysis > Load > Remote Force

Remote Force is used to set the total value of the distributed load. The load does not act directly on the face, but is transmitted from the distance point by means of inelastic rods. The rods connect the distance point to the face, taking into account the moments that arise.



After calling the command, you should select the model elements for the load application. Use the following automenu option:

 **<F>** Select Face

Select faces of the calculated model. The selected elements will be added to the list.

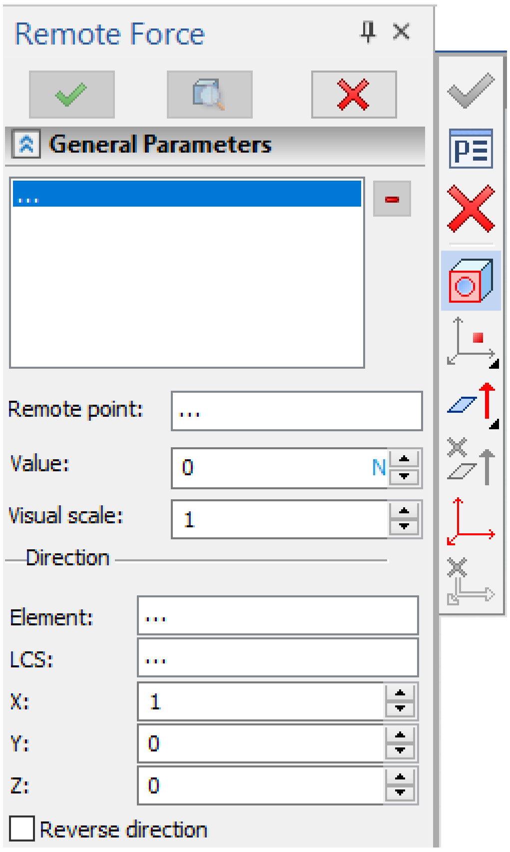
Remote point. This point represents the point of application of a **Remote Force**. As a point, you can choose s 3D node, LCS, the top of a wire or solid geometry, etc. according to filter settings.

Value. It is a total value of the load evenly distributed over the face area. You can specify the dependence of load on time only in non-stationery tasks.

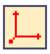
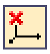
> More information in Using Graphs to Specify Parameters section.

Measurement units. For the **Remote Force** load the following units of measurement can be set: N, kN, MN, psi, dyn, kgf.

Direction. As a direction of the Remote force, you can select an element of the 3D model or some radius vector specified in the local coordinate system selected by the user. If the local coordinate system is not specified, then the global coordinate system will be used by default.



Use the following options to work with a LCS:

	<C>	Select LCS
	<K>	Cancel LCS selection

Use the following automenu option to specify the direction of a **Remote Force** using a 3D model object:

	<D>	Select direction

Use the following option to cancel the direction selection:

	<U>	Cancel direction selection

To change the direction of a **Remote Force** you can set the **Reverse direction** flag.

Typical sequence for specifying a load "Remote Force"

1. Call the **Remote Force**  command.
2. Select faces for load application;
3. Select a remote point (3D node or vertex);
4. Select a coordinate system;
5. Set the load value and units;
6. Complete the command.

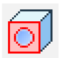
Remote Torque

Use the following command to set **Remote Torque** load:

Icon	Ribbon
	Analysis > Conditions > Force > Remote Torque
Keyboard	Textual Menu
<3M3Q>	Analysis > Load > Remote Torque

Remote Torque is used to set the total torque given directly to the face. Creation of a remote point is not required, it is enough only the magnitude and direction of the torque.

After calling the command, you should select the model elements for the load application. Use the automenu option:

 <F> Select Face

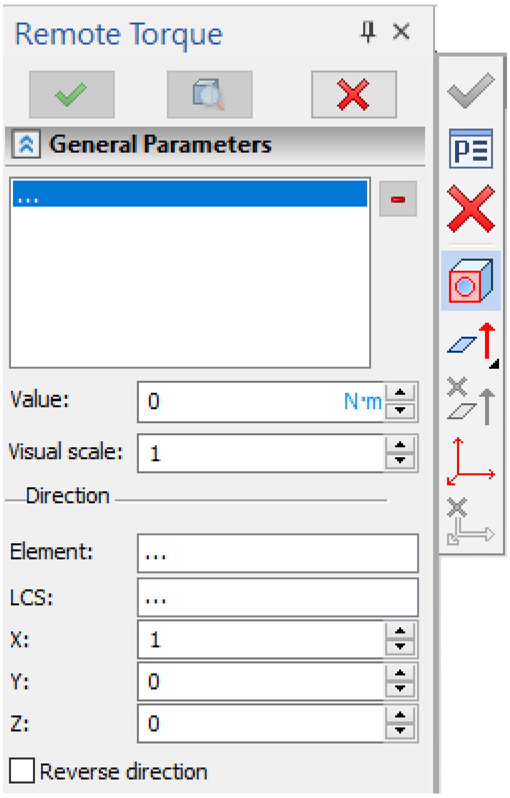
Select faces of the calculated model. The selected elements will be added to the list.

Value. It is a total value of the load evenly distributed over the face area. You can specify the dependence of load on time only in non-stationery studies.

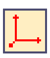
> More information in Using Graphs to Specify Parameters section.

Measurement units. For the **Remote Torque** load the following units of measurement can be set: N*m, kgf*cm, lbf*in.

Direction. The direction is used to specify a rotation axis of a **Remote Torque**. You may select an element of a 3D model or a radius vector to specify the direction. The vector should be specified in the selected local coordinate system. If the local coordinate system is not specified, then the global coordinate system will be used by default.



Use the following options to work with a LCS:

 <C> Select LCS

	<K>	Cancel LCS selection

Use the following automenu option to specify the direction of a **Remote Torque** using a 3D model object:

	<D>	Select direction

Use the following option to cancel the direction selection:

	<U>	Cancel direction selection

To change the direction of **Remote Torque** you can set the **Reverse direction** flag.

Typical sequence for specifying a load "Remote Torque"

1. Call the **Remote Torque** command.
2. Select faces for load application;
3. Select a coordinate system;
4. Set the load value and units;
5. Complete the command.

Thermal Loads

This type of loading is used in the heat transfer problems.
Heat transfer is the process of transferring the heat from one region with higher temperature to the region with lower temperature.

Topics in this section:

- Temperature
- Heat Flow
- Heat Power
- Convection
- Radiation
- Thermal Contact

Temperature

Use the following command to set **Temperature** load:

Icon	Ribbon
	Analysis > Conditions > Temperature > Temperature
Keyboard	Textual Menu
<3TT>	Analysis > Thermal Load > Temperature

Temperature characterizes a thermal state of a body and determines how warm it is.

The load **Temperature** is used for defining invariant in time constant temperature of elements of the model in steady state and transient thermal analysis, and also for defining temperature difference in the static analysis of structure’s strength when solving the thermo-elastic problems.

The **Initial Temperature** option is used for defining initial temperature in transient thermal analysis. This initial condition defines the temperatures of selected elements of the model at time equal to zero.

> In the nodes of finite element mesh which do not belong to the selected elements of the model, the initial temperature will be assigned the «default» value. The default value is defined in the dialog window **Study's Parameters** on the tab **Parameters**.

Temperature can be applied to a model body, a face, edge or vertex.
Use the following options of the automenu to select model elements:

<E>	Select element for loading
<M>	Select All Solids

Selection filters are available upon click and hold on the **Select elements for loading** option:

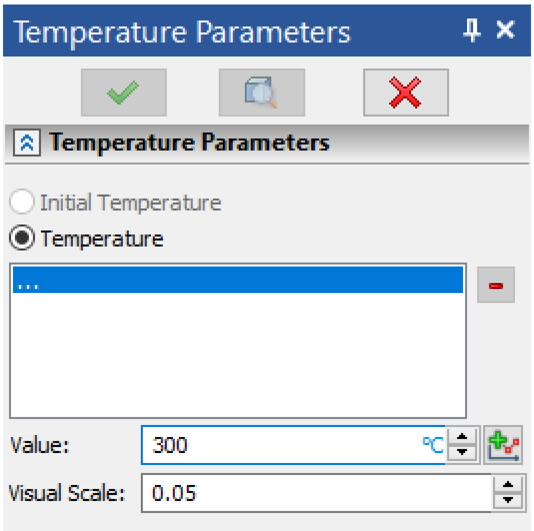
<Alt+Num+>	Select All
<Ctrl+Alt+V>	Select Vertex
<Ctrl+Alt+E>	Select Edge
<Ctrl+Alt+Q>	Select Face
<Ctrl+Alt+B>	Select Body

Same filters are available in the Filter Toolbar:

In the Select Face mode additional filters are available in the automenu:

<C>	Select all faces of body including contacting
<N>	Select all faces of body excluding contacting
<S>	Select all faces of bodies used in study

Selected elements are added to the list in the **Parameters** window.



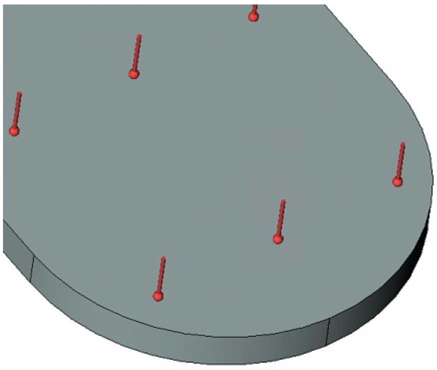
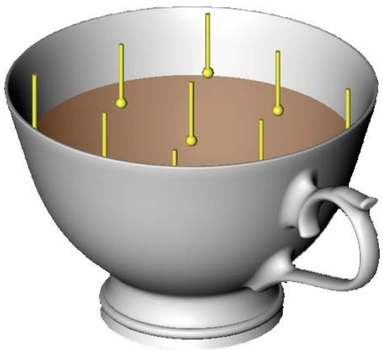
In the Parameters window of the load **Temperature**, it is required to specify the following parameters:

- The magnitude of the load (You can specify dependence of temperature from time for **Nonstationary Thermal Process** study).

More information in Using Graphs to Specify Parameters section.

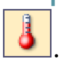
- Units: K, C, F.

In the 3D scene the **Temperature** is shown in the following way:

Specifying thermal load **Temperature** | Specifying initial temperature in transient thermal analysis

Typical sequence for specifying a load "Temperature"

1. Call the command **Temperature** .
2. Activate option **Temperature** or **Initial Temperature**.
3. Select body, face, edge, vertex or a set of elements
4. Specify the value of temperature and units.
5. Complete the command.

Heat Flow

Use the following command to set **Heat Flow** load:

Icon	Ribbon
	Analysis > Conditions > Temperature > Heat Flow
Keyboard	Textual Menu
<3TF>	Analysis > Thermal Load > Heat Flow

Load **Heat Flow** allows the user to specify the amount of heat transferred across the unit surface area per unit of time, that is, define a specific heat flow.
Heat flow can be applied to faces of the model. For selecting faces, use the automenu option:

<F> Select Face

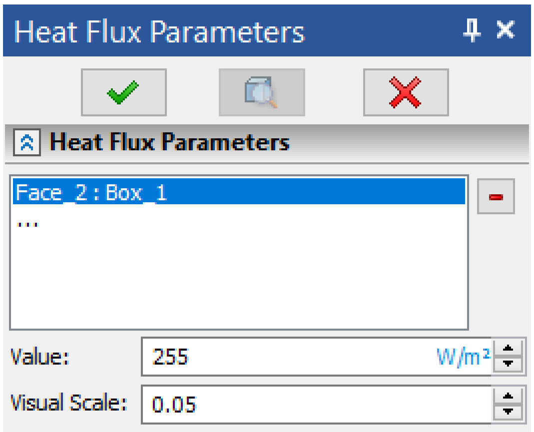
In the Select Face mode additional filters are available in the automenu:

<C> Select all faces of body including contacting

<N> Select all faces of body excluding contacting

<S> Select all faces of bodies used in study

Selected elements are added to the list.



In the parameters window of the load **Heat Flow** it is required to specify the following parameters:

- The heat flow magnitude;
- Units: W/m^2, W/cm^2, BTU/s*in^2.

Negative value of the heat flow signifies that through the specified face the body looses the energy.
You can specify dependence of heat flow from time for **Nonstationary Thermal Process** study.

> More information in Using Graphs to Specify Parameters section.

In the 3D scene Heat flow is shown in the following way:

Specifying thermal load **Heat Flow**

Typical sequence for specifying a load "Heat Flow"

1. Call the command **Heat Flow** ;
2. Select a face;
3. Specify the magnitude of load;
4. Specify units;
5. Complete the command.

Heat Power

Use the following command to set **Heat Power** load:

Icon	Ribbon
	Analysis > Conditions > Temperature > Heat Power
Keyboard	Textual Menu
<3TP>	Analysis > Thermal Load > Heat Power

Load **Heat Power** allows the user to define:

- volume power of thermal energy sources;
- amount of heat transferred through an arbitrary surface per unit of time, called a power of heat flow (heat power).

Heat power can be applied to a body, face, edge or a vertex of a model.
Use the following options of the automenu to select model elements:

<E> Select element for loading

<M> Select All Solids

Selection filters are available upon click and hold on the **Select elements for loading** option:

<Alt+Num+> Select All

<Ctrl+Alt+V> Select Vertex

<Ctrl+Alt+E> Select Edge

<Ctrl+Alt+Q> Select Face

<Ctrl+Alt+B> Select Body

Same filters are available in the Filter Toolbar:

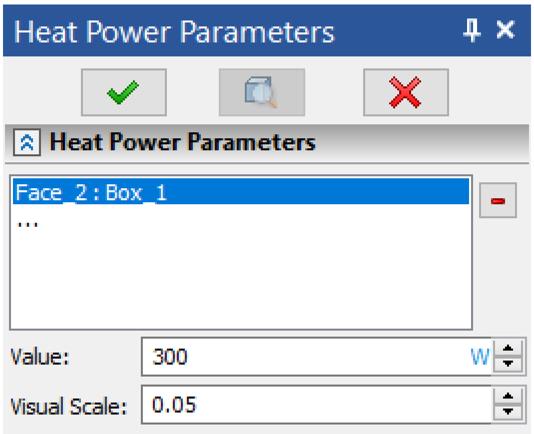
In the Select Face mode additional filters are available in the automenu:

<C> Select all faces of body including contacting

<N> Select all faces of body excluding contacting

<S> Select all faces of bodies used in study

Selected elements are added to the list in the **Parameters** window



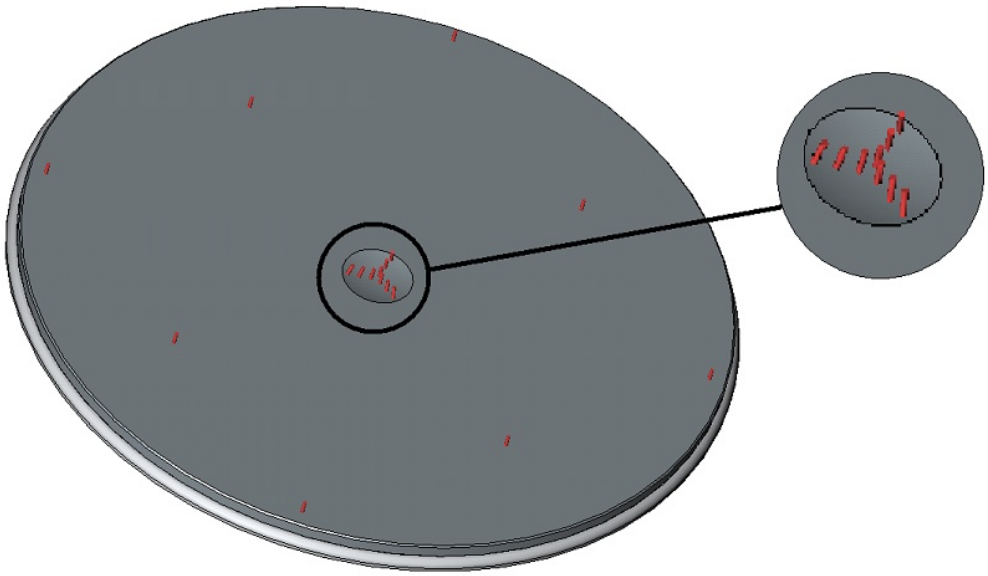
In the parameters window specify:

- Magnitude of load;
- Units: W, BTU/sec.

The negative value of this thermal load signifies that a body looses the energy.
You can specify dependence of heat power from time for **Nonstationary Thermal Process** study.

> More information in Using Graphs to Specify Parameters section.

In the 3D scene **Heat Power** is shown in the following way:



Specifying thermal load **Heat Power**

Typical sequence for specifying a load "Heat Power"

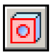
1. Call the command **Heat Power** 
2. Select body, face, edge, or a vertex.
3. Specify the magnitude of load;
4. Specify the units
5. Complete the command.

Convection

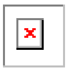
Use the following command to set **Convection** load:

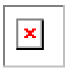
Icon	Ribbon
	Analysis > Conditions > Temperature > Convection
Keyboard	Textual Menu
<3TC>	Analysis > Thermal Load > Convection

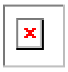
Convective heat transfer is a process of transferring the heat between the surface of a solid body and external environment (gas, liquid).
Load **Convection** allows the user to specify the amount of heat emitted by the unit surface per unit of time when the temperature difference between the surface and external environment is one degree, that is, specify the heat transfer coefficient.
Load **Convection** is defined for faces of the model. For selecting faces use the automenu option:

 <F> Select Face

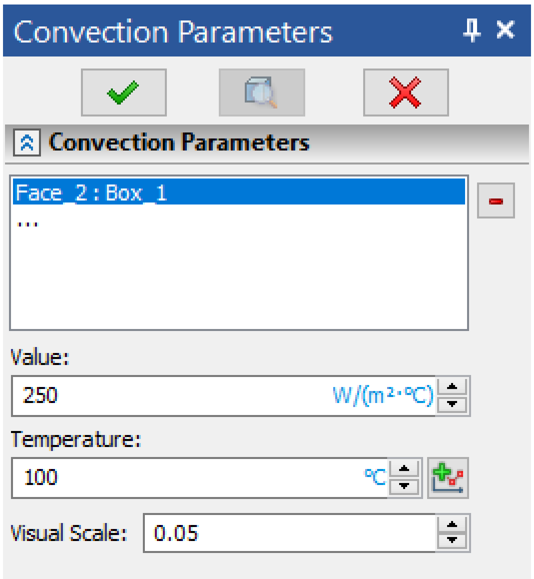
In the Select Face mode additional filters are available in the automenu:

 <C> Select all faces of body including contacting

 <N> Select all faces of body excluding contacting

 <S> Select all faces of bodies used in study

Selected elements are added to the list.



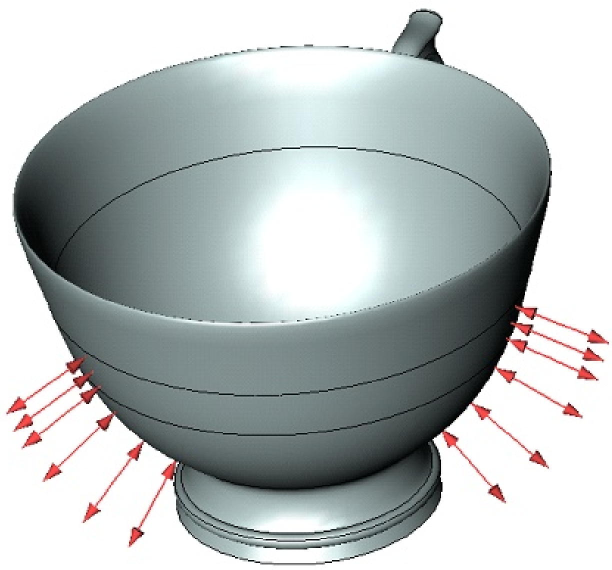
In the parameters window of the load **Convection** it is necessary to specify the following parameters:

- heat transfer coefficient;
- Units: W/m^2*C, W/cm^2*C, BTU/sec*in^2*F;
- Temperature of external environment (liquid or gas);
- Units: Kelvin [K], degrees of Celsius [C], degrees of Fahrenheit [F].

You can specify dependence of convection from time for **Nonstationary Thermal Process** study.

> More information in Using Graphs to Specify Parameters section.

In the 3D scene the load **Convection** is shown in the following way:



Specifying thermal load **Convection**

Typical sequence for specifying a load "Convection"

1. Call the command **Convection** .
2. Select a face.

3. Specify the heat transfer coefficient and temperature of the environment.
4. Complete the command.

Radiation

Use the following command to set **Radiation** load:

Icon	Ribbon
	Analysis > Conditions > Temperature > Radiation
Keyboard	Textual Menu
<3TR>	Analysis > Thermal Load > Radiation

Heated bodies emit the thermal radiant energy into the environment. Thermal radiation hitting a certain body is partially reflected, partially absorbed, and partially passes through the body. Boundary condition **Radiation** allows to determine the properties of bodies in terms of its ability to emit and absorb radiant energy – radiation factor.

The radiation factor equal to “1” corresponds to completely black body, i.e. the body that emits the maximum possible amount of radiant energy according to the Stefan-Boltzmann law and absorbs all incident radiant energy.

The radiation factor equal to “0” corresponds to an ideal mirror, which does not emit when heated to any temperature and reflects all incident radiant energy, i.e. excluded from the calculation.

Intermediate values of the radiation factor are equal to the ratio of the radiating ability of the body to the emissivity of a blackbody, i.e. show how the body emits less energy than a black body would radiate at the same temperature. Known or initial temperatures and sources of heat energy are specified in addition to the emissivity/absorption for the correct formulation of the problem on heat transfer by radiation. They are applied to the bodies participating in the study.

> Only bodies for which **Radiation** boundary condition was applied participate in convection calculation. Bodies for which radiation factors were not applied are ignored

Load **Radiation** is defined for faces of the model.

For selecting faces use the automenu option:

<E> Select Element for loading

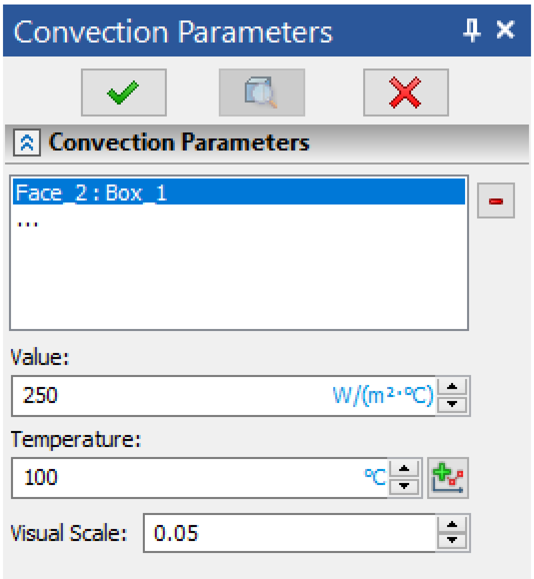
When selecting faces additional filters are available in the automenu:

<C> Select all faces of body including contacting

<N> Select all faces of body excluding contacting

<S> Select all faces of bodies used in study

Selected elements are added to the list.



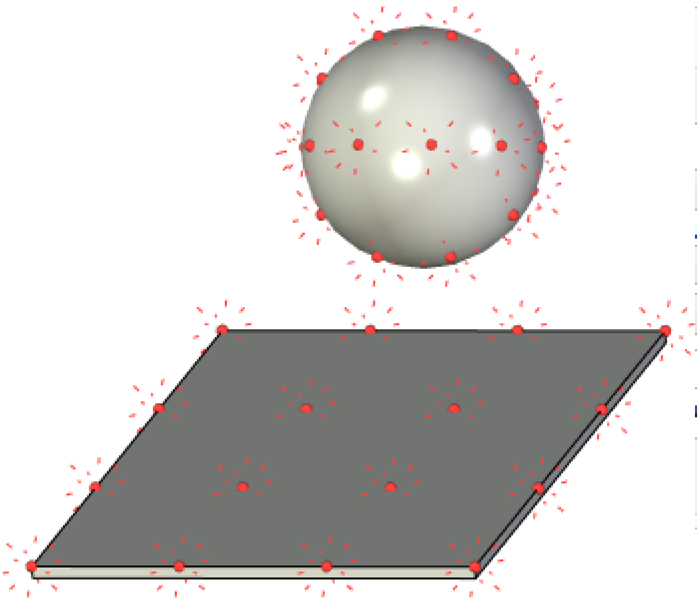
In the parameters window of the load **Radiation** it is required to specify the following parameters:

- Radiation type: radiation into space, radiation between faces;
- Emissivity;
- Temperature of environment;
- Units: Kelvin [K], degrees of Celsius [C], degrees of Fahrenheit [F].

You can specify dependence of radiation factor from time for **Nonstationary Thermal Process** study.

> More information in Using Graphs to Specify Parameters section.

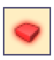
The load **Radiation** is shown in the 3D scene in the following way:



Specifying thermal load **Radiation**

It is recommended to use **Nonstationary thermal study** with active **Study on thermodynamic equilibrium** option for calculation using **Radiation** load in the **Radiation between faces** mode. The option can be found on the **Parameters** tab in the study parameters.

Typical sequence for specifying a load "Radiation"

1. Call the command **Radiation** .
2. Specify radiation type.

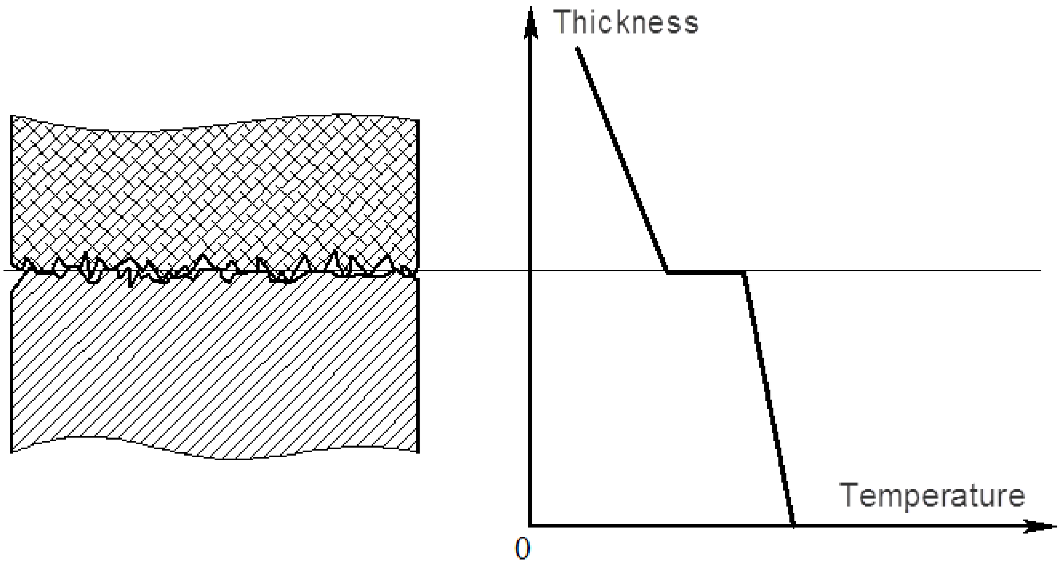
3. Specify emissivity.
4. Specify temperature of environment and units.
5. Specify radiation viewfactor of a face.
6. Complete the command.

Thermal Contact

Use the following command to set **Thermal Contact** load:

Icon	Ribbon
	Analysis > Conditions > Temperature > Thermal contact
Keyboard	Textual Menu
<3T3C>	Analysis > Thermal Load > Thermal contact

Contact between physical bodies practically never can be considered ideal. Because of the roughness of contact surfaces, microscopic gaps are created at the interface between two bodies which can be filled with air or another surrounding environment. This surrounding environment has heat conduction coefficients different from those of the solid bodies in contact. As a result, at the interface between the bodies in contact, the continuous temperature field suffers a jump which is usually caused by worsened conditions for conduction of the thermal energy at the interface between the bodies in contact.



This physical phenomenon is called thermal resistance.
After the command is called, the following options of the automenu become available:

	<C>	Select contact surfaces
	<1>	Selection of faces 1
	<X>	Cancel selection of all faces

With the left mouse button, indicate the faces taking part in the thermal contact. It is possible to indicate several sets of faces if they have similar parameters of the thermal contact.
After that, it is required to select the type of the thermal resistance: total or distributed.

- Total thermal resistance has the units of measurement degrees/W. Its magnitude characterizes the total loss of thermal power across the entire surface of contact (the surface can be compound).
- Distributed resistance characterizes the loss of thermal flow per unit area of the surface of contact. The units of measurement are degrees*m^2/W.

Approximate values of coefficients of heat conduction are shown in the following table:

Contact surfaces	**Thermal resistance (m^2 ·°K/ W)**
Iron/aluminum	$2{,}22 \cdot 10^{-5}$
Copper/Copper	$1 \cdot 10^{-4}$ - $4 \cdot 10^{-5}$
Aluminum/aluminum	$4{,}54 \cdot 10^{-4}$ – $8{,}33 \cdot 10^{-5}$
Stainless steel/ stainless steel	$5 \cdot 10^{-4}$ – $2{,}7 \cdot 10^{-4}$
Stainless steel/ stainless steel (dispersed gaps)	$5 \cdot 10^{-3}$ – $9{,}09 \cdot 10^{-4}$
Ceramics/ceramics	$2 \cdot 10^{-3}$ – $3{,}33 \cdot 10^{-4}$

Typical sequence for specifying a load "Thermal Contact"

1. Initialize the **Thermal Contact** command .
2. Select contact surfaces of the first body.
3. Select contact surfaces of the second body.
4. Specify type of contact.
5. Complete the command.

Using Graphs to Specify Parameters

Boundary conditions or material properties that depend on time or temperature can be used in the dynamic and nonstationary thermal studies.
For example, it can be:

- Gradual increase of force with time;
- Dependence of the convection factor on the surface temperature of bodies from which the convection occurs;
- Dependence of the linear expansion factor of the material from the temperature.

Dependence of material properties from temperature can be also specified in static, buckling, etc. studies.
Graphs in boundary conditions or material properties are used for specifying the parameters.

Specify Dependence of Value from Time

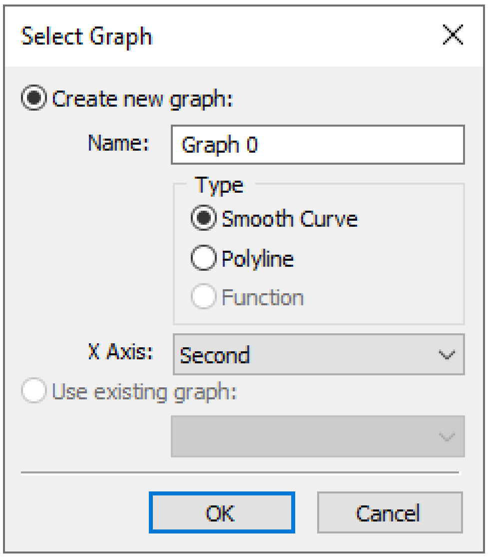
Dependence of value from time is used in dynamic analysis studies (Mode Superposition, Transitional processes) and Nonstationary thermal processes. You can specify the dependence for the following boundary conditions:“

- Force
- Pressure
- Acceleration
- Bearing Force
- Torque

- Additional Mass
- Ocsillator
- Temperature
- Heat Flux
- Heat Power
- Convection
- Radiation

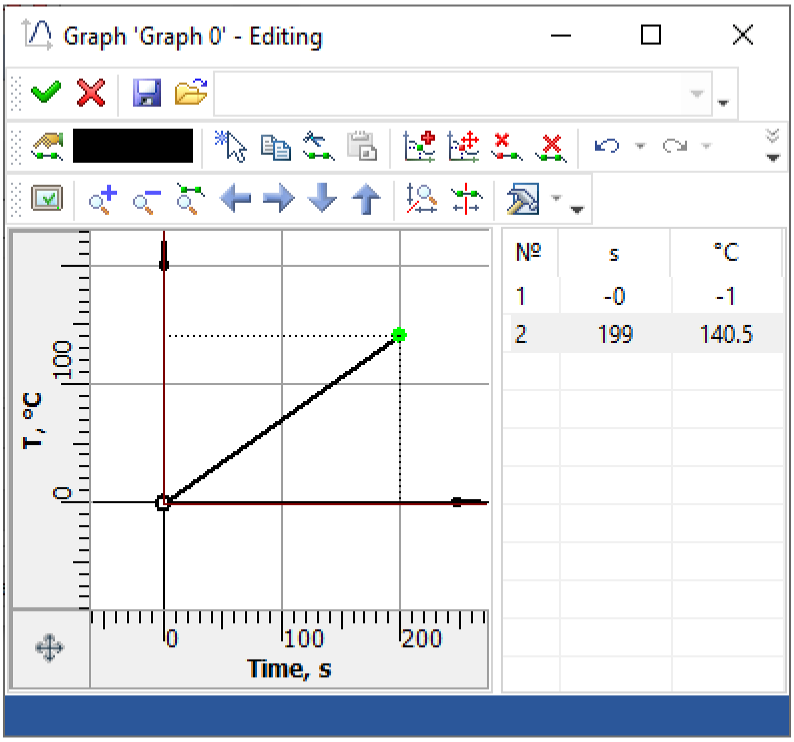
You need to press  button right to the value in the parameters window.

Dialog for the graphic creation is opened after that. You need to specify a name for the graph, select type smooth curve/polyline and press **OK**.

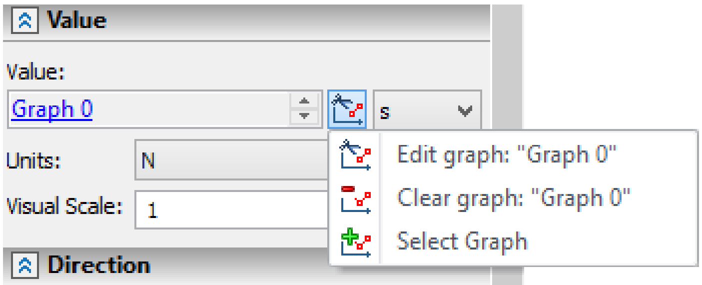


You need to specify point after that. Creation, edition, saving and loading of graphs is no different from standard T-FLEX CAD graphs.

You can paste the points from clipboard (delimiter of argument/function is Tab) or Excel table. The created graph is stored in the current document and is available in **Edit Graph Functions** command. You can also download T-FLEX CAD graph of .tflaw format and use it for the parameter.



After the graph is added, the parameter value changes to the graph name and the list appears. You can select units of the argument (s, min, h) in the list. Units correspond the units that are set for the parameter for which the graph is used.



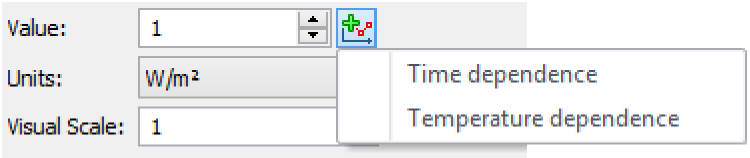
Pressing of  button calls list that allows to edit, clear current graph or select a new graph. When the graph is deleted, it is unbound from the parameter but stored in the document. You need to use **PL: Edit Graph Functions** command to delete it from the document.

Specify Dependence of Value from Temperature

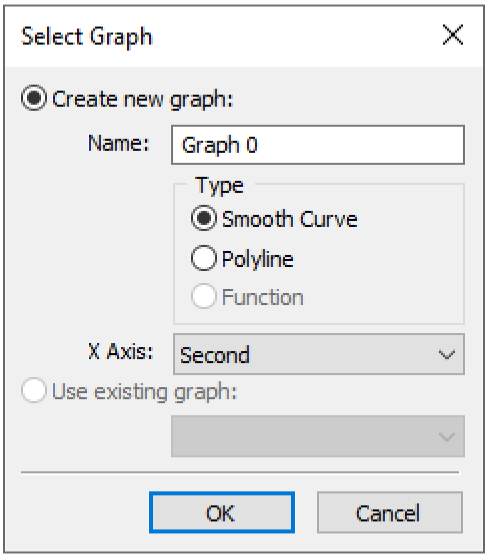
Dependences of values from temperature can be specified in stationary and nonstationary processes. It is allowed for the following thermal boundary conditions:

- Temperature
- Heat Flow
- Heat Power
- Convection
- Radiation

Dependence from temperature can be also applied for any of the material properties.
To specify dependence of parameter from temperature you need to press button right to the value in the parameters window. In the drop-down list, you should select **Temperature dependence**.

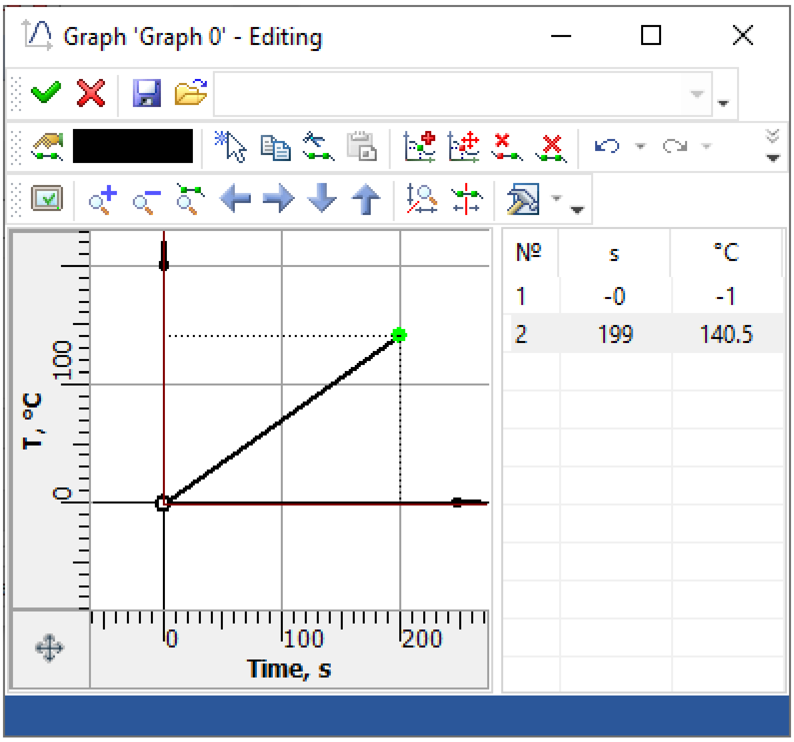


Dialog for the graphic creation is opened after that. You need to specify a name for the graph, select type smooth curve/polyline and press **OK**.

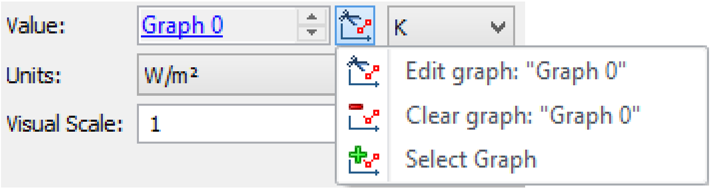


You need to specify point after that. Creation, edition, saving and loading of graphs is no different from standard T-FLEX CAD graphs.

You can paste the points from clipboard (delimiter of argument/function is Tab) or Excel table. The created graph is stored in the current document and is available in **Edit Graph Functions** command. You can also download T-FLEX CAD graph of .tflaw format and use it for the parameter.



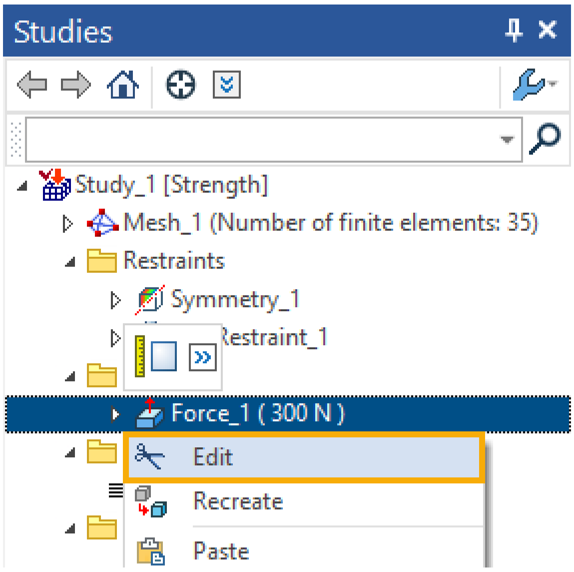
After the graph is added, the parameter value changes to the graph name and the list appears. You can select units of the argument (K, F, C) in the list. Units correspond the units that are set for the parameter for which the graph is used.



Pressing of  button calls list that allows to edit, clear current graph or select a new graph. When the graph is deleted, it is unbound from the parameter but stored in the document. You need to use **Edit Graph Functions** command to delete it from the document.

Editing Loads and Restraints

To modify specified loads and restraints, use the **Edit** command, available in the context menu on right clicking  a study element in the studies window or in the **3D Model** window.

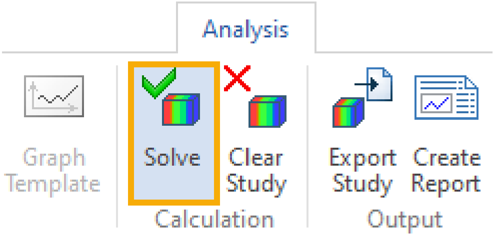


Further user actions of modifying loads or restraints are little different from the process of their creation. In the parameters window, you can modify numerical parameters, while the appropriate automenu options (see above) let you cancel selection and then specify new defining model elements.

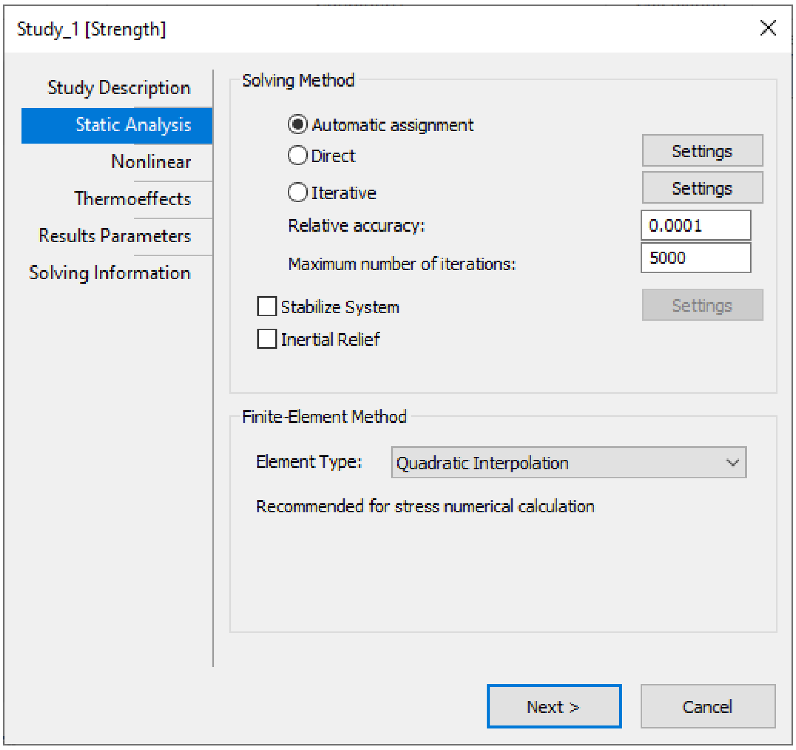
Upon finishing entering changes, confirm your actions by clicking .

Study Solve

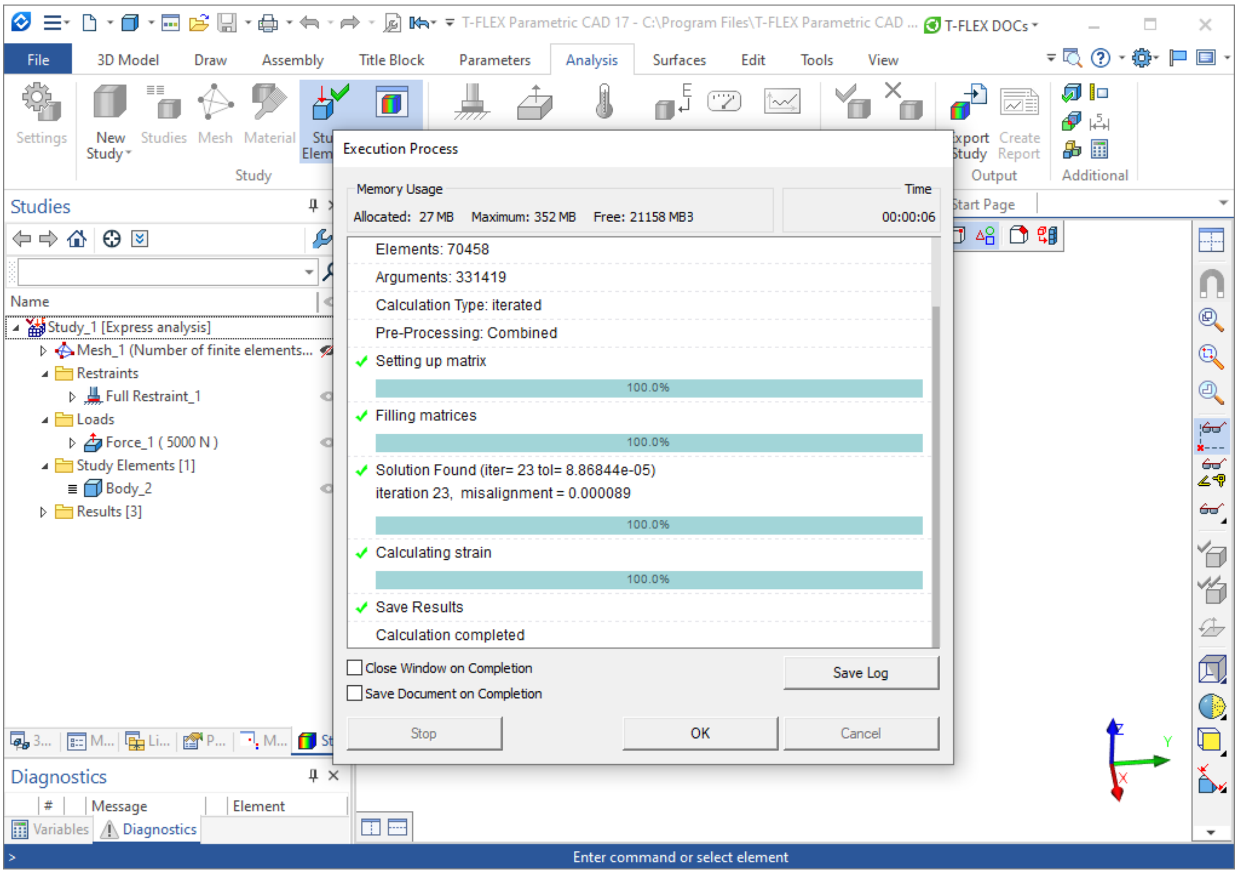
After creating a finite element grid and applying boundary conditions, you can initialize the **Analysis** > **Solve** command and start the process of forming systems of linear algebraic equations (SLAE) and solving them.



You can also access the **Solve** command from the context menu of the corresponding study in the study tree displayed in the study window.
Modes for forming SLAE and methods for solving them are selected automatically by the T-FLEX Analysis processor. The user can independently change the calculation options in the study parameters dialog, which opens by default before starting the calculation.



The process of solving systems of linear equations can take a significant amount of time on studies with meshes of large number of tetrahedra. At the end of the calculation, the corresponding information message is displayed.



This command is used to start solving the active study according to its source data and calculation parameters.

By default, after initializing the command, the study parameters dialog appears with several switchable tabs. The user can change the default study parameters. You can also access the study parameters by clicking on the study name in the study tree or from the context menu by clicking on the name of the selected study in the study tree. The user-defined study parameters are saved when the document is saved, and they are inherited when the study is copied. The main purpose of study parameters is to set the modes required for Processor operation, and lists of results displayed after calculation in the study tree.

Sets of settings and parameters corresponding to different types of studies are described in more detail in the sections:

- Static Strength
- Buckling Analysis
- Fatugue Analysis
- Frequency Analysis
- Forced Oscillation
- Dynamic Studies
- Thermal Analysis

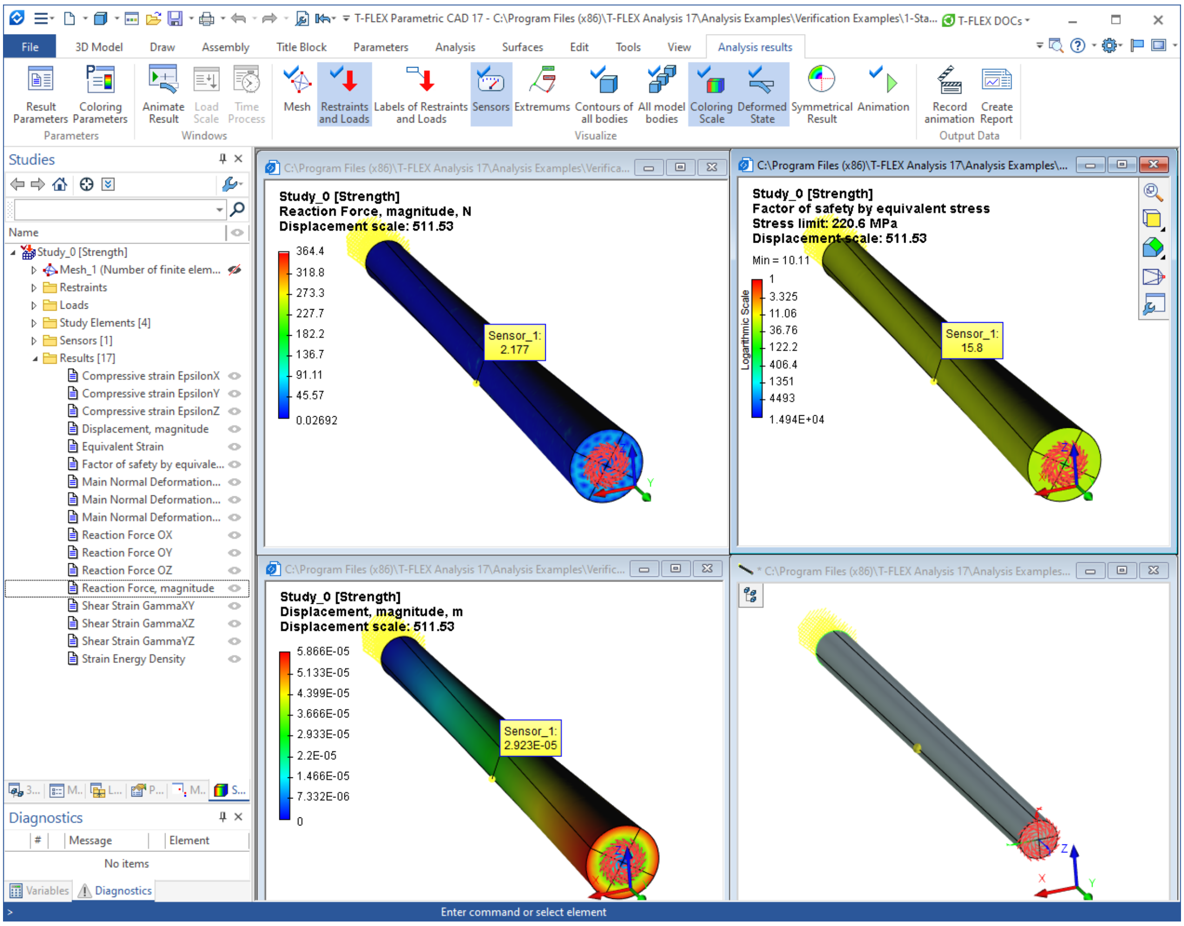
Static Analysis

Static Strength

The main goal of the static strength analysis of structures is evaluation of a stress state of a structure subjected to constant in time (static) forces. This evaluation of the stress state is usually performed with the purpose of probing the adopted design features against the strength criterion. The strength criterion is generally formulated as follows:

> The stress σ developing in a structure under applied external forces must be less than the safe stress $[\sigma]$ for the given structural material after applying the strain margin of safety factor: K_{safe}: $K_{safe} \cdot \sigma \le [\sigma]$

The static analysis module of the T-FLEX Analysis Finite Element modeling system serves for calculating a static stress state of three-dimensional structures in T-FLEX CAD environment. The static analysis module works directly with three-dimensional T-FLEX CAD models and and does not require additional constructions for solving a particular three-dimensional model.



The main results of a static solution are:

- structure's displacements field at the calculation points of the finite element mesh;
- relative strain field;
- stress components field;
- strain energy;
- node response;
- the field of the strain safety factor distribution over the volume of the structure;

This data is normally enough for predicting the structure behavior and making decisions for optimizing geometrical shape of a part with the goal of insuring the main strength criteria of parts.

Topics in this section:

- Details of Static Analysis Steps
- Evaluation of Results
- Study Parameters
- Optimization
- Appendix (References)

Details of Static Analysis Steps

Static Analysis of a model is performed in several stages. Listed are the elements required for conducting an analysis. To run a static analysis, complete the following steps:

Step 1. Creating three-dimensional solid model of a part

Before starting working in T-FLEX Analysis system, the user shall prepare a three-dimensional solid model to be evaluated. A solid model can be built in T-FLEX CAD environment or imported from other systems. Static analysis can be performed over one or multiple operations-bodies.

Step 2. Creating "Study"

A **Study** is created by the command:

Icon	Ribbon
	Analysis > Analysis > New Study > FEA Study
Keyboard	Textual Menu
<3MN>	Analysis > New Study > FEA Study

To perform a static analysis, when creating a study, the user specifies its type - **Static strength** in the command's parameters window. If there are multiple bodies in the scene, then you need to select one or several contacting bodies, for which a new study will be created.

Step 3. Defining material

One of the required elements of any solution is the study's material. Detailed description of material defining methods for calculations is provided in the respective section of the preprocessor description.

Step 4. Creating mesh

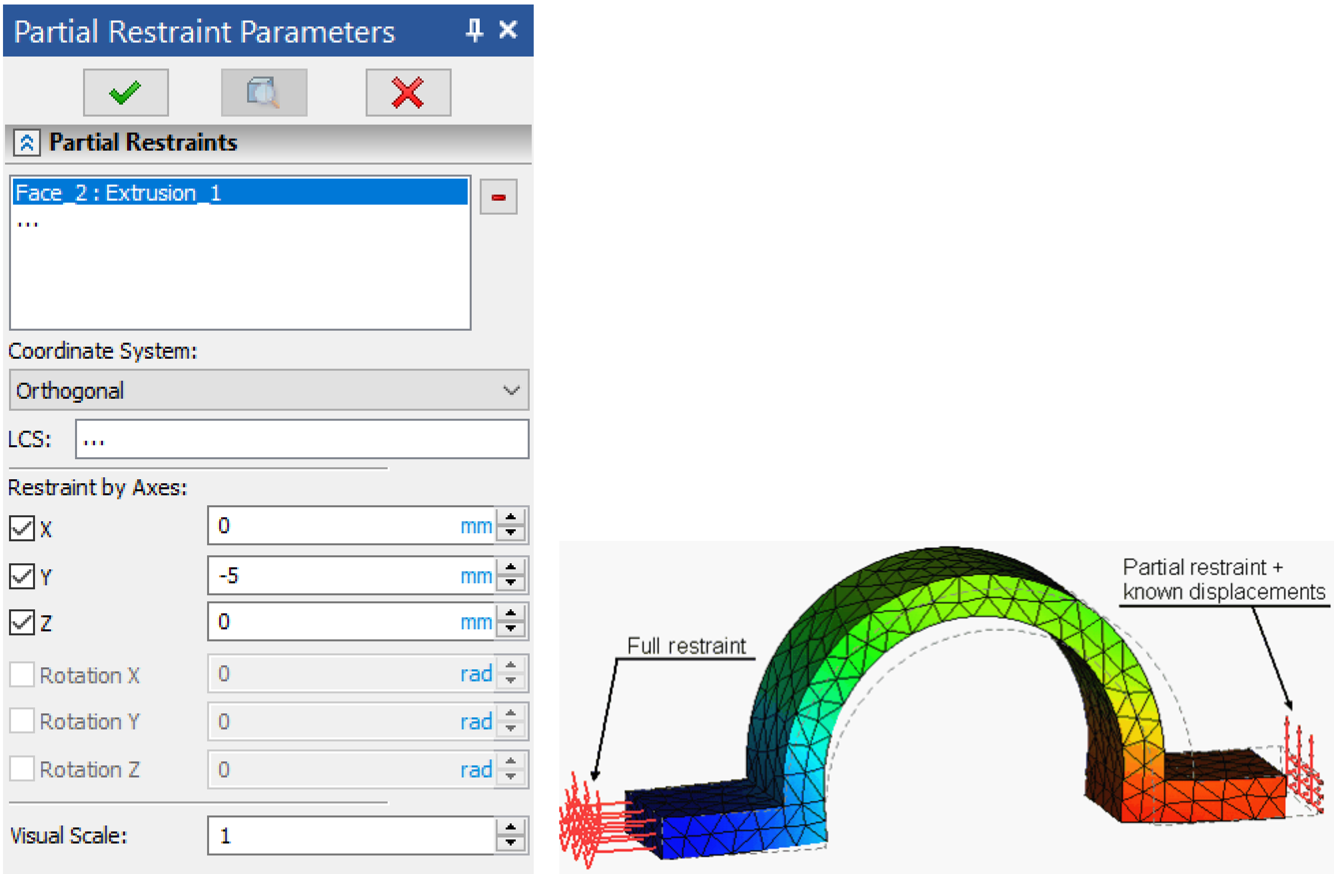
To perform Finite Element modeling, you need to construct a finite element mesh. By default, the mesh construction command initiates automatically when creating a study. The user can also create a mesh using the T-FLEX Analysis command **Analysis > Mesh**. When creating a mesh, the user defines various parameters of discretizing a solid-state model. The finite element mesh can significantly influence the quality of the obtained solution in the cases of complex spatial configuration of parts. The finite element mesh generation parameters are reviewed in detail in the respective section of T-FLEX Analysis preprocessor description.

Step 5. Applying boundary conditions

In static analysis, boundary conditions are represented by restraining methods and external loads applied to the system. The boundary conditions creating stage is very important and requires good understanding by the engineer of the essence of the study being solved. Therefore, think over the physical aspects of the study thoroughly before applying boundary conditions.
Defining restraints is a necessary condition for running a correct static analysis. The combined limitation on the body movement must satisfy the following condition:

> To be suited for a static analysis, a model should have restraints preventing free movement in the space as a solid body. Failing to meet this condition will cause incorrect results of Finite Element analysis or abortion of computations.

Two commands are provided in T-FLEX Analysis for defining restraints: **Full Restraint** and **Partial Restraint**. The **Full Restraint** command sets a fully fixed (immovable) state for the selected model element. The **Partial Restraint** command allows selectively limiting model element motion along the axes of the chosen coordinate system.
The "Partial Restraint" command also provides another useful functionality. The user can specify known displacements for the structure, such as a known in advance strain in the structure. For this, specify the value of fixed displacement of a model element along some of the coordinate axes in the **Partial Restraint** command's parameters window. Static Analysis will be performed with this condition accounted for. Note that a static solution is possible in this case without applying additional (force) loads. In this way, one can evaluate the stress developing in a strained structure when the quantitative values of the strain (displacements) are known.



Example of using known displacements

A number of specialized commands is provided in T-FLEX Analysis to define loads; those allow defining main types of loads (Force, Pressure, Centrifugal Force, Acceleration, Bearing Force, Torque). Detailed description of all types of loads is provided in the preprocessor description.
Note yet another functional capability of a static solution in T-FLEX Analysis. The user can define a structure's stress state analysis not only under various forces, but also under thermal loads - the **Thermoelasticity** Study. As known, structural materials develop linear strain under the thermal impact - expand under heating and shrink under cooling. Changes in a body's dimensions cause strain and a stressed state. T-FLEX Analysis accounts for changing temperatures. To define temperatures when accounting for transient temperature fields, use the command:

Icon	Ribbon
	Analysis > Conditions > Temperature > Temperature
Keyboard	Textual Menu
<3TT>	Analysis > Thermal Load > Temperature

At the same time, you need to enable the **Consider Thermoeffects** option on the **Thermoelasticity** tab of the static study parameters dialog in order to account for thermal loads in the static solution. You will also need to define the temperature of "zero" strain, which corresponds to the no-stress state of the model, and to define the working temperature field (details are in the section **Settings of Linear Statics Processor**).

Step 6. Running calculations

Once a finite element mesh is built for the model and boundary conditions are applied (restraints and loads), you can start the process or creating and solving linear algebraic equations of the static analysis. Use the following command to start solving the active study:

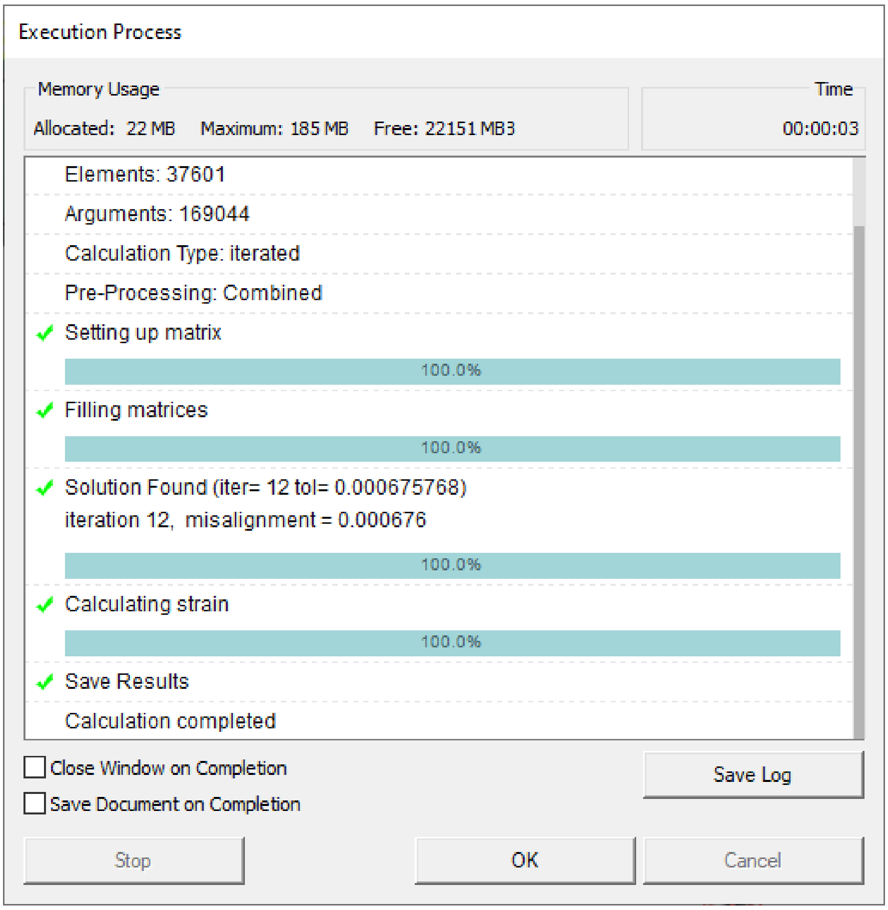
Icon	Ribbon
	Analysis > Solve > Solve
Keyboard	Textual Menu
<3MY>	Analysis > Solve

The selected study's calculation can be started from the context menu by clicking on the name of the selected study in the studies tree.

By default, the "Study parameters" dialog of the static analysis opens automatically before calculations. In this dialog, the user can define the desired options and settings of the solution, as well as specify the types of solution data displayable in the studies tree. Detailed description of study settings purpose is further available in the section **Settings of Linear Statics Processor**. Most of the settings are selected by the processor automatically depending on the number of dimensions in the study being solved and imposed boundary conditions.

Clicking the **OK** button in the study parameters dialog launches the process of building and solving systems of linear algebraic equations. The stages of solving equations and additional reference information are displayed in a special information pane. Clicking the **Close** button in the information pane terminates calculations. The **Close this dialog box on solving completion** flag will force automatic closing of the solution steps monitoring window after finishing solving equations.

The flag **Save Document on solving completion** will force automatic saving of calculation results and all changed data in the active document.



The following reference data is output in the information window:

Nodes - the number of nodes in the computational finite element mesh.

Elements - the number of tetrahedrals in the finite element mesh.

Arguments - the number of equations of linear statics.

Calculation Type - the algorithm used for solving equations. Types of possible algorithms and their use are described in the section **Settings of Linear Statics Processor**.

Solution Found – tells that system of equations was successfully calculated. There is also auxiliary information in the brackets: iter – number of executed iteration (if iterative method was used), tol – miscalculation of result after calculation.

The calculation steps are also visually displayed as a dynamically updating scale. Additionally, the time elapsed from the start of calculations is shown. The **Memory usage** group shows the current state of memory and it shows if the computer being used is suitable for solving large problems. The **Allocated** field shows how much RAM is occupied at the current moment (including swap memory). The **Maximum** field – a pick value reached during the solution time. The **Free** field is a size of free physical random access memory. If this value decreases to zero, the swap memory is used. Moreover, the time elapsed from the start of calculation and the percentage of solution at the current iteration are displayed. After finishing calculations, the user must close the auxiliary window (unless the auto close option is enabled).

Step 7. Analysis of static solution results

After completing calculations, a new **Results** folder appears in the studies tree. By default, this one displays the results defined on the **Results** tab of the **Study parameters** dialog. Overall, the user can access 38 solutions in the result of the static analysis, sorted out into 6 groups.

Displacements group includes the following results.

Δ_x - Component of displacement vector for a node of the finite element mesh along the OX-axis of the global coordinate system;

Δ_y - Component of displacement vector for a node of the finite element mesh along the OY-axis of the global coordinate system;

Δ_z - Component of displacement vector for a node of the finite element mesh along the OY-axis of the global coordinate system.

Displacements, absolute value – the absolute value of the nodal displacements of the model, defined for each node according to the formula: $\Delta = (x_i^2 + y_i^2 + z_i^2)^{1/2}$, where *x, y, z* - displacement vector components for the i-th node of the finite element mesh.

Group "Stresses" includes the following results:

σ_{eq} - relative equivalent stresses evaluated from components of the stress tensor according to the formula:

$$\sigma_{eq} = (((\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6 \cdot (\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2))/2)^{1/2};$$

σ_x - normal stress in the direction of the *OX*-axis of the global coordinate system;

σ_y - - normal stress in the direction of the *OY*-axis of the global coordinate system;

σ_z - normal stress in the direction of the *OZ*-axis of the global coordinate system;

τ_{xy} - shear stress acting in the direction of the *OY*-axis of the global coordinate system on a plane with the normal vector parallel to the *OX*-axis;

τ_{xz} - shear stress acting in the direction of the *OZ*-axis of the global coordinate system on a plane with the normal vector parallel to the OX-axis;

τ_{yz} - shear stress acting in the direction of the *OZ*-axis of the global coordinate system on a plane with the normal vector parallel to the OY-axis;

$\sigma_1, \sigma_2, \sigma_3$ - principal stresses $\sigma_1 \geq \sigma_2 \geq \sigma_3$.

Stress intensity is defined in the following way: $\sigma_1 = max(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|)$

Group «Safety factor by stresses» includes the following results:

Safety factor by equivalent stresses represents the ratio of admissible for a given structural material stresses $[\sigma]$ to the equivalent stresses:

$$K = [\sigma]/\sigma_{eq};$$

Safety factor by shear stresses is evaluated as:

$$K_\tau = [\sigma]/2\tau_{max}, \quad \tau_{max} = (\sigma_1 - \sigma_3) \cdot 0{,}5;$$

Safety factor by normal stresses is evaluated as:

$$K_n = [\sigma]/\sigma_1;$$

A material's safe stress is defined in the material properties in the standard T-FLEX CAD library or in the appropriate field of the study's materials library. The yield limit is accepted as the safe stress for plastic materials.

Group «Deformation» includes the following results:

ε_{eq} - relative equivalent strains expressed in terms of components of the strain tensor by the formula:

$$\varepsilon_{eq} = \frac{2}{3}\sqrt{\frac{3(\varepsilon_x^2 + \varepsilon_y^2 + \varepsilon_z^2)}{2} + \frac{3(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{xz}^2)}{4}};$$

ε_x - relative normal strain in the direction of the *OX*-axis of the global coordinate system;

ε_y - relative normal strain in the direction of the *OY*-axis of the global coordinate system;

ε_z - relative normal strain in the direction of the *OZ*-axis of the global coordinate system;

γ_{xy} - shear strain in the *OXY* plane;

γ_{xz} - shear strain in the *OXZ* plane;

γ_{yz} -shear strain in the *OYZ* plane.

$\varepsilon_1, \varepsilon_2, \varepsilon_3$ - principal stresses $\varepsilon_1 \geq \varepsilon_2 \geq \varepsilon_3$.

Strain Energy Density. The result reflects volume distribution of strain energy over the model.

Group «Reactions». The result reflects forces building up in the supporting (fixed) nodes of the finite element model.

F_x -reaction force in the direction of the *OX*-axis of the global coordinate system;

F_y - reaction force in the direction of the *OY*-axis of the global coordinate system;

F_z - reaction force in the direction of the *OZ*-axis of the global coordinate system;

Reaction force (absolute value) – the magnitude of the absolute value of the nodal reaction forces of the model defined for a node as $F_{mi} = (F_{xi}^2 + F_{yi}^2 + F_{zi}^2)^{1/2}$, where F_x- x -component, F_y- y-component, F_z – z -component of the reaction force for the i-th node of the finite element mesh.

"Total Load" group displays the loads applied to a finite-element model as the effective node responses. This type of data represents reference information.

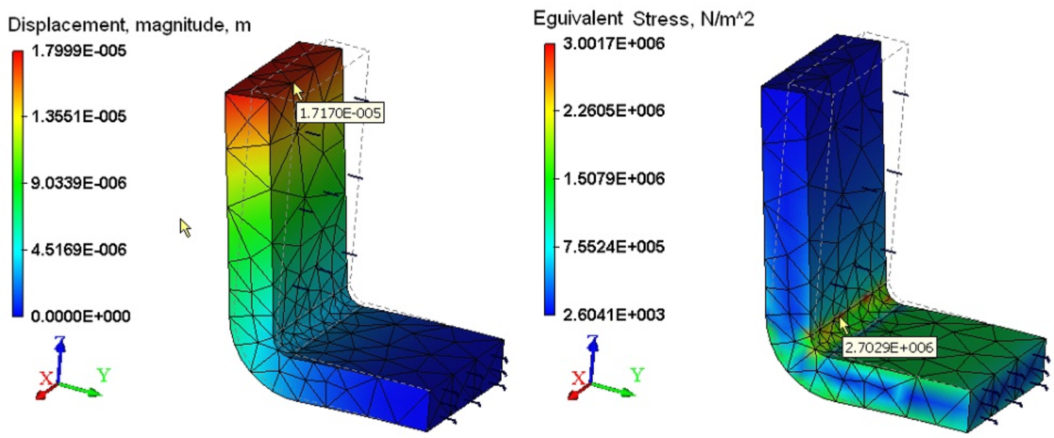
Temperature. This result shows distribution of temperature field over the volume of the model.

Evaluation of Results

Once the study calculation is completed successfully, you should analyze obtained results in order to make conclusions on probabilistic static strength of the structure. In most cases, three types of solution suffice - displacements, stresses and the strain safety factor. A typical sequence of steps for validating the results of Finite Element modeling is as follows:

Displacement Analysis

In the studies tree, use the context menu command **Open** or **Open in new window** to open the **Displacement, magnitude** solution. We can visually estimate the pattern and the ranges of the stress-strain state of a structure. It is necessary to analyze displacements in order to verify correctnes of applied loads and to assert correctnes of the found solution as a result of solving systems of equations. If the results of displacements analysis indicate that a solution to the study is found and the pattern of the structure's strain state matching the expected, then you can proceed to the next step.



The diagram of absolute displacements and equivalent stresses

Stress Analysis

Open the **Equivalent Stress** result. One can visually assess the pattern of the calculated equivalent stress. The stress gradients are illustrated by color transitions. The color code scale displayed in the calculation results view window helps reading the approximate value of the displayed result. If you point the mouse to the area of interest on the model, then a tooltip will pop up, displaying the value of the evaluated measure, interpolated by the nearest nodes around the pointer location. The **Equivalent stress** result lets the user make the following conclusions:

1. Determine, at what locations and in which elements of the structure the largest stress develops;
2. By comparing the maxima of the calculated stresses with the allowable stress for the model material, one can assess the degree of the structural strength.

Safety factor estimate

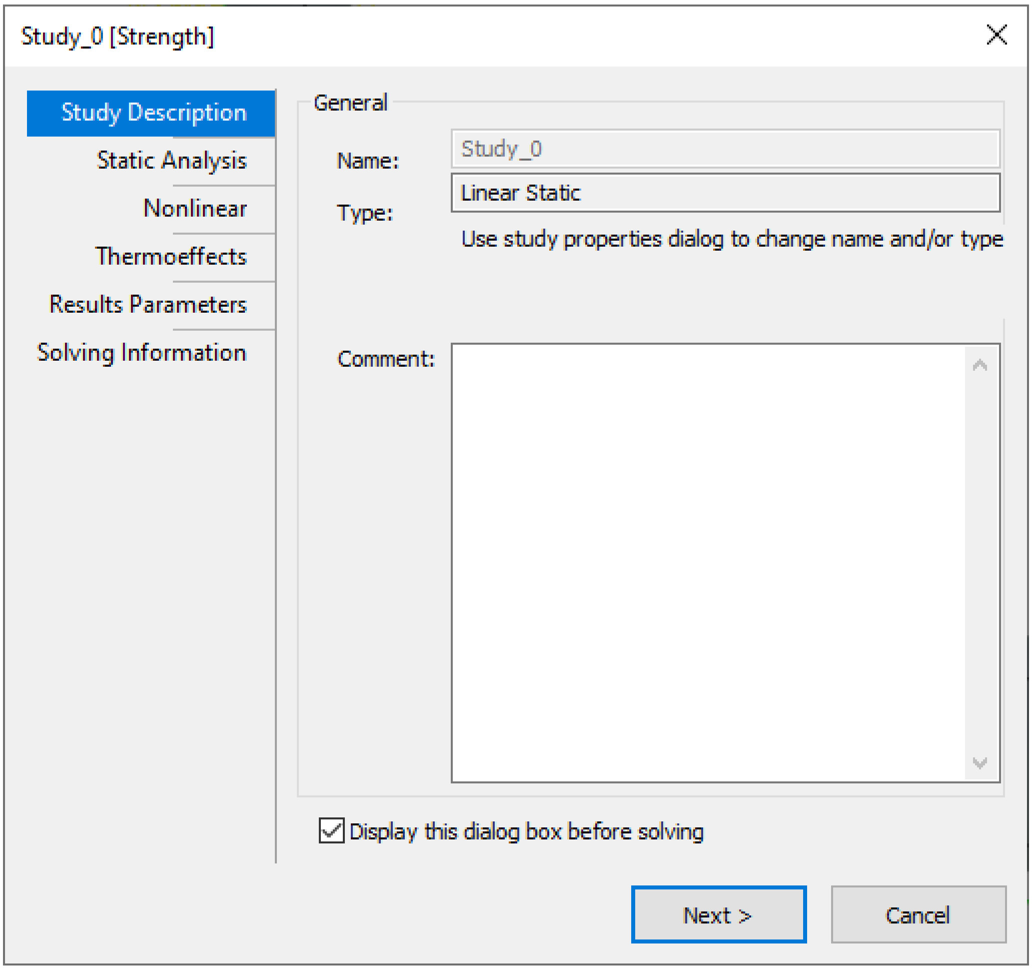
Open the **Factor of safety by equivalent stress** result. This result allows estimating the quantitative ratio of the safe stress to the calculated equivalent stresses specified in the material properties. By default, the result is shown on the logarithmic scale in order to reduce the range of color gradients. If the ratio of the safe and calculated stresses approaches one or becomes less than that, then the strength criterion no longer holds and, therefore, the design must be altered.

Study Parameters

The user-defined study parameters are saved together with the document and are inherited upon copying a study. The main purpose of the study parameters is defining options required for the processor, the result listings to be displayed after completing calculations in the studies tree, as well as keeping the descriptive attributes of the study, as the name or a comment. The static analysis solution parameters dialog has five tabs.

"Study Description" and "Static Analysis" Tabs

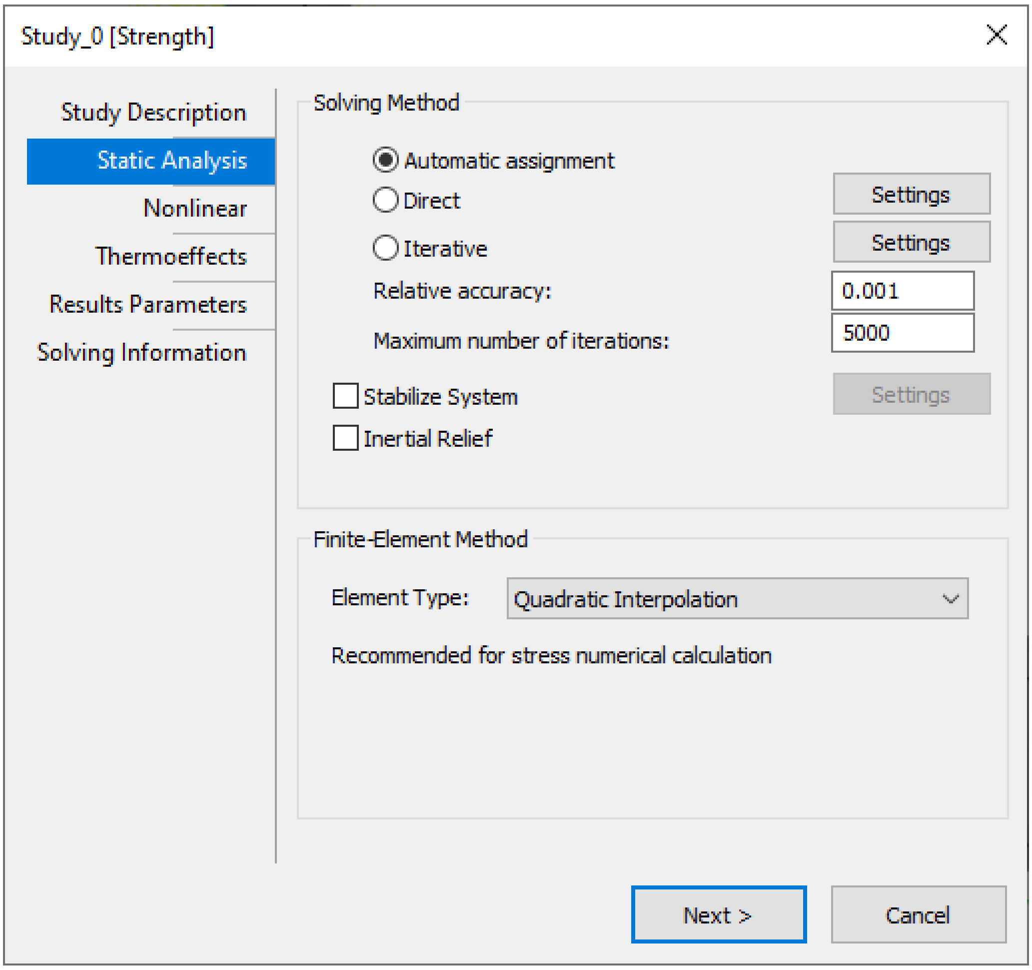
The **Study Description** tab serves for defining descriptive parameters of the current study.



In the **Name** field, the user can edit the study name assigned the system default at creation. This name will be further on displayed in the studies tree, in the results window and in the report. The **Type** control serves for defining the study type. Note that T-FLEX Analysis allows changing an existing study type to another one from the list of study types available to the user. For example, the user can create a study of the **Static Strength** type, and then change the type, for example, to **Stability** or **Frequency Analysis**.

The **Comment** edit box lets the user entering arbitrary text information pertaining to the current study. This information will be used in the future for generating a report based on the study solution results.

The **Static Analysis** tab serves for defining processor properties for solving linear statics equations. The control elements in the "**Solving method**" group let the user define the methods of solving systems of algebraic equations of linear statics.



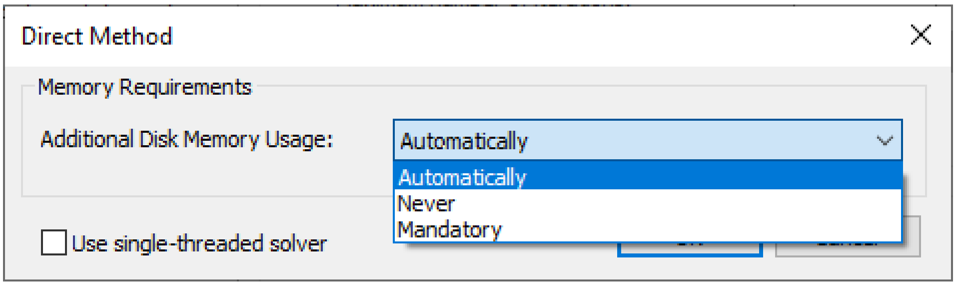
Direct method. The system of equations is solved by Gauss method via LU decomposition of the stiffness matrix. This method is effective for solving the system of equations constructed on the basis of the linear finite elements. In certain cases, the use of direct method can be also justified for analysis of the system with the help of quadratic finite elements. It can be used instead of iterative method, if the iterative algorithm does not converge to the stable solution, or if the convergence speed is very small (the number of iterations is several thousands). This situation can be observed for «thin» problems (the model is flat or stretched), and also, for a large number of finite elements which are considerably different from equilateral elements (when the ratio of the lengths of the finite element edges are on the order of hundreds or thousands).

Iterative. The systems of equations are solved by iterative methods. This method is used by default for solving systems of equations, built based on a quadratic finite element. The following two options can be set for the iterative method: relative accuracy and the maximum number of iterations.

Relative accuracy - the accuracy of the achieved iterative solution. The smaller is the specified miscalculation error, the greater number of steps (iterations) will be required.

Maximum number of iterations - the critical number of iterations, after reaching which the iterative solving of the system of equations terminates, even if the required solution precision was not achieved.

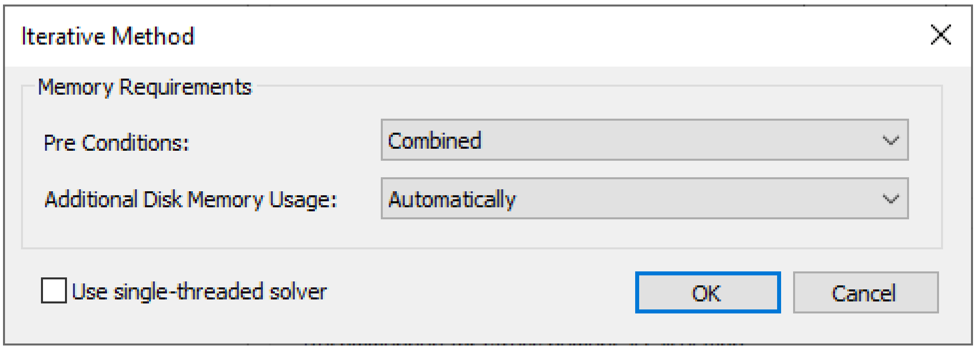
The user can also manage interaction with external (disk) memory of the computer system when solving **SLAE** by a direct or iterative method (**Settings** button).



There are three options for using additional disk memory: automatically, never, mandatory. The use of additional disk memory allows you to save the stiffness matrix decomposition.Using additional disk memory for solving systems of equations is necessary only when the memory requirement for keeping intermediate matrixes exceeds the computer's RAM. Note also that the running time for studies with a large number of dimensions using external storage could be significant due to a large number of operations on sequential data read-write.

High volumes of disk storage may be needed for keeping intermediate matrixes (up to several Gigabytes). Make sure there is enough disk space before solving studies in large dimensions using external storage.

If the user disabled the possibility of using the disk space while solving a high order system of equations, an abnormal termination of the process may abort calculations in the event of the memory consumption for keeping the matrix decomposition approaching 2 Gigabyte (for Windows 32-bit).



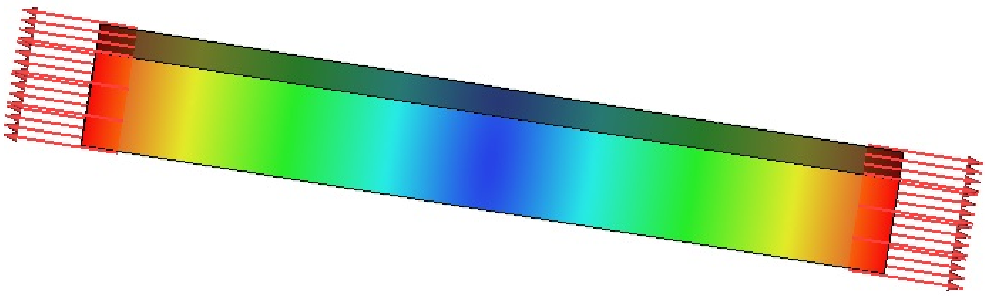
In the iterative method settings the user can select **Pre Conditions** for the system of equations. Pre Conditions is a certain numerical procedure that allows us to speed up the process of solving the system of linear equations by an iterative method. In T-FLEX Analysis 4 types of preconditioners are available:

- **Combined** (by default) – usually provides the fastest solution with minimum number of iterations but at the same time the expenses of random access memory needed for execution of this iterative method are the largest.
- **Incomplete** expansion – memory expenses do not exceed the doubled size of the memory required for storage of the stiffness matrix, the number of iterations is decreased, but this type of preconditioner is not as effective as the combined preconditioner.
- **Diagonal and Singular** – provides minimum requirements to the additional random access memory but requires large number of iterations to obtain the solution, and therefore, is rarely used for solving problems of practical interest.

Finite-Element Method. By default, all calculations use quadratic approximation for displacements, regardless of what kind of finite element mesh was constructed for the model. If the user is only interested in qualitative results, that is, he is only interested in relative distribution of stress fields, using a rather fine mesh, then one can use the linear element solution, which runs much faster than the quadratic counterpart. The hybrid element is used for static strength analysis of the models containing both linear plate-like and 3D volume elements (so called «hybrid» models).

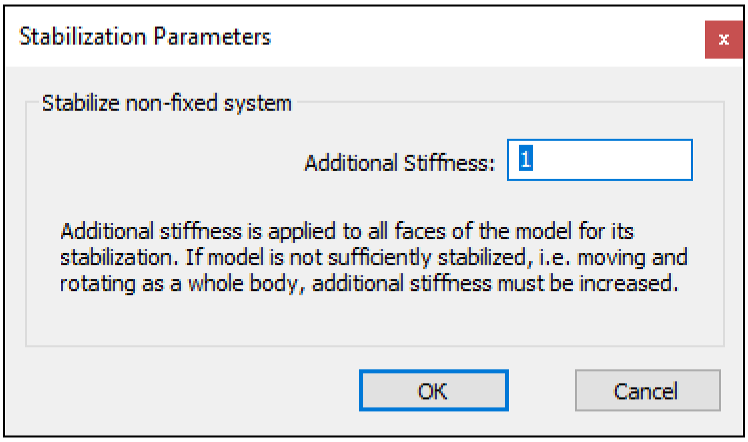
> The tetrahedral linear element analysis provides **insufficient** accuracy of quantitative results. Maximum displacement and stress results are much smaller via the calculation by linear tetrahedral finite elements, rather than those achieved by more accurate methods. It is strongly recommended to use quadratic element calculations for quantitative evaluation (the default mode).

Stabilize System. This mode allows us to carry out analysis of systems which are non-fixed in space but equilibrated by forces. Usually, if in finite-element calculations we use a model which is not constrained in space, i.e., has the capability to move in space as a rigid body, the finite element static analysis becomes impossible (Obviously, the body which is not fixed will "fly" away to infinity under the action of applied loads). The **Stabilize System** option allows us to overcome this limitation if the system is equilibrated in space by forces. The principle of stabilization is the following: on all faces of the model the elastic elements are applied the stiffness of which is sufficiently small compared to the stiffness of the model and does not influence considerably the distribution of deformation (and, hence, the stresses).
Moreover, the elastic elements make impossible displacement of the model in space as whole, and therefore, make the stiffness matrix definite which allows us to solve the finite-element problem. It is to be noted that in order to avoid the large displacements of the finite element model, the applied system of forces must be self-equilibrated. If, however, the large displacement of the model as a whole takes place in space, it is possible to make use of additional partial constraints of the face or edge to avoid rotation of the model.
While doing so, it is important to remember that additional constraints must be selected in such a way that a certain deformation mode of the body is not provoked under conditions of the given system of forces (do not cause additional reaction forces, for example, constrain displacements in those directions along which the force components are equal to zero or along axes of symmetry).



Analysis of a non-fixed model equilibrated by forces

The **Settings** button, located next to the **Stabilize System** flag, allows us to select the magnitude of additional stiffness in N/m, suitable for conditions of the given problem.



Thermoeffects Tab

The **Thermoeffects** tab allows defining the methods for calculating thermal loads.

Consider thermoelasticity. Includes the mode of calculating loads building up in a structure due to the linear expansion forces under the condition of heat applied to the body.

Temperature of zero deformations: - the initial body temperature, at which there is no thermal strain and there is no stress caused by difference in temperatures. The user can specify temperature values in one of the existing scales: **K** - Kelvins; **C** - Celsius; **F**- Fahrenheit. Define the method of specifying thermal loads in the **Temperature fields** group.

Uniform temperature - the value of a uniform temperature field is specified in the chosen units, which affects all studied bodies.

Use preset temperature - thermal loads are included in the static analysis, that were defined by the command **Analysis** > **Thermal Load** > **Temperature**.

Use Thermal Study Results - available solution of the thermal analysis study is used for defining the thermal loading. In the drop-down list select the name of the solved thermal analysis study and (if necessary) the time instant, to which the solution pertains. Please note that certain conditions are to be met for using thermal analysis results as the initial temperature conditions:

Identity condition of finite element meshes in static and thermal analyses. The simplest way of achieving such identity is the use of the "Copy" command available in the context menu. The sequence of steps can be, for example, as follows:

1. Create a study of the type **Stationary Thermal Process**, generate a mesh, define boundary conditions, and run;
2. Create a study copy using the **Copy** command;
3. Go to study editing and change the study type to **Static Analysis**.

As a result, we have two studies of different types, but with identical finite element meshes.

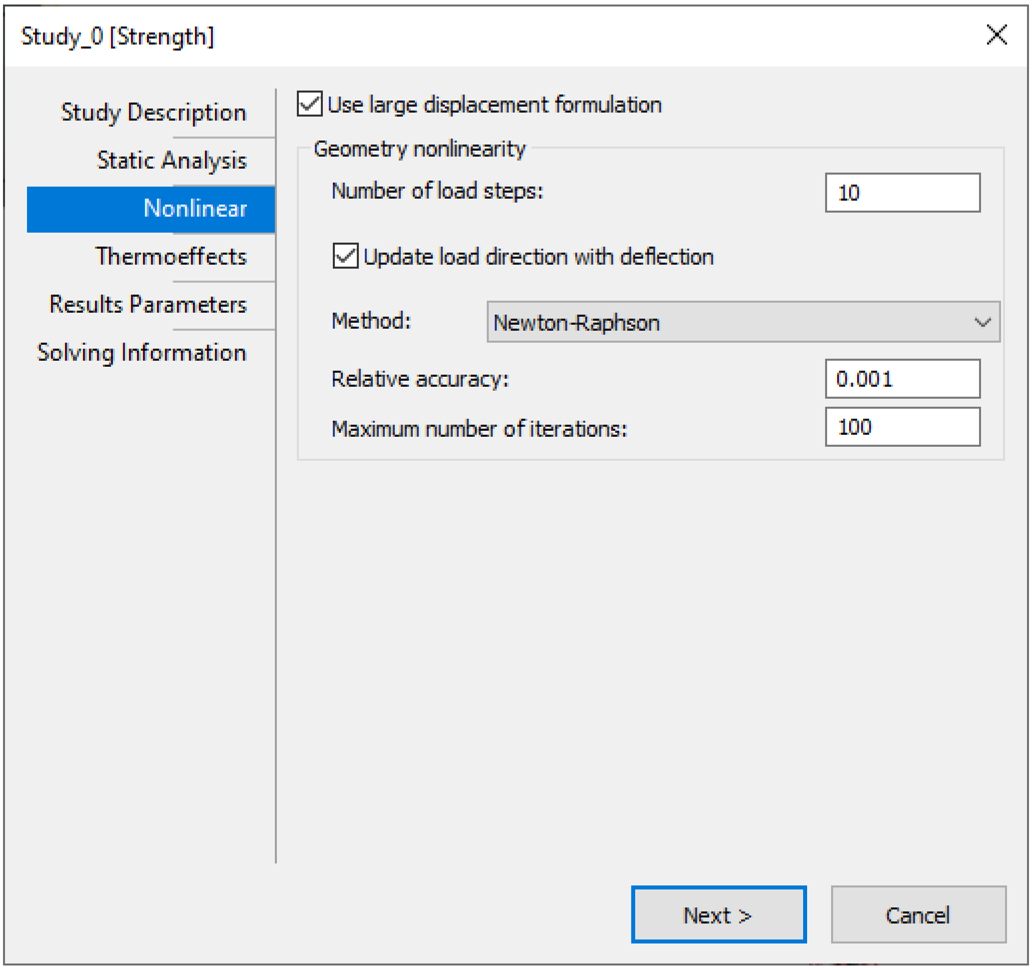
The **Calculate using linear element** parameters on the **Solve** tab of the study parameters dialog should use the same settings in both studies. For example, if the thermal analysis is done by linear elements, then the static analysis based on thermal analysis results can also be run only by linear elements.

Results Parameters Tab

The **Results Parameters** tab allows defining the result types displayable in the studies tree after finishing calculations.

Nonlinear Tab

The tab **Nonlinear** allows the user to carry out static analysis taking into consideration large displacements.

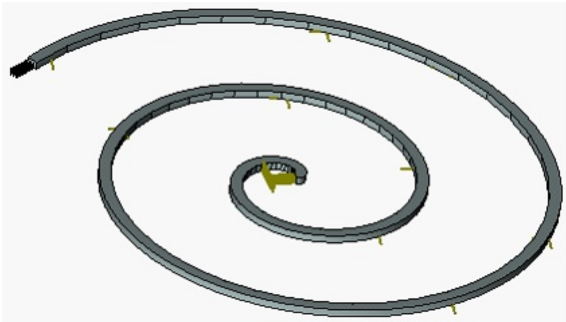


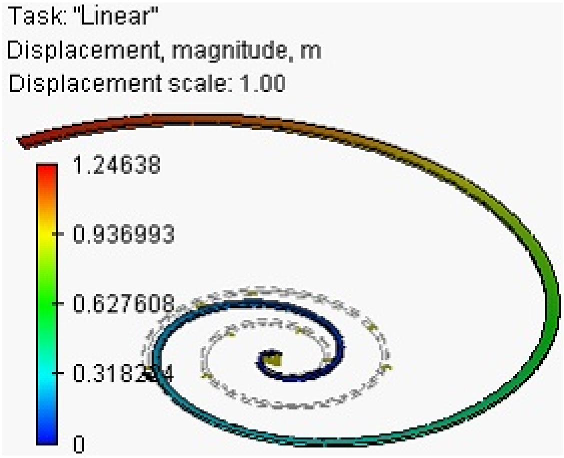
In practice, there are situations in which displacements of certain points of the structure reach significant values under the action of external loads. These problems are especially important in aircraft and space industry, when designing radio-telescopes, cooling towers and other thin-walled structures. In these cases the nonlinear effects should be taken into consideration, since the assumptions on which the linear analysis is built are not valid.

The option **Use large displacement formulation** should be activated in cases when at least one of the following assumptions of the linear analysis is violated:

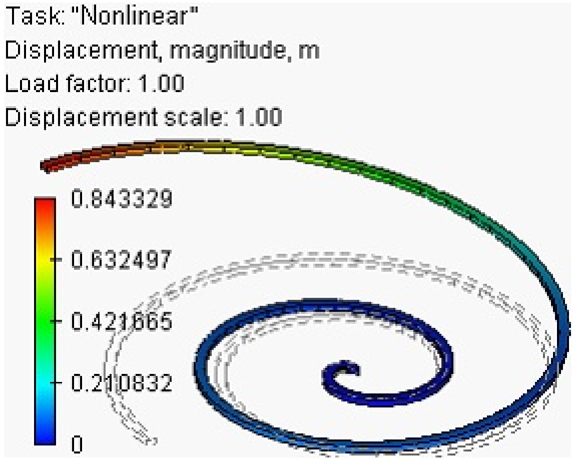
1. Resulting deformations are sufficiently small, so the stiffness changes caused by the load can be ignored;
2. In the process of applying the load, boundary conditions do not change amplitude, direction and distribution.

For example, the linear analysis of the spiral-like part subject to the load applied at the end edge gives an error of approximately 30% compared to the nonlinear analysis. This difference in results arises due to small displacement assumption adopted in linear analysis.





Linear analysis



Nonlinear analysis

Controls in the group **Geometry nonlinearity** allow the user to customize the process of solution of geometrically nonlinear problems.

For solving such problems, a time stepping nonlinear solver organizes the process of incremental loading of structure and gives the solution of the linearized system of equations at each step for the current increment of the load vector, formed for a specific loading.

Number of load steps. This option allows the user to set the number of steps during which the load will be changing from zero to a specified value. Theoretically, all solutions can be found within one step for total value of acting load. However, there arises the possibility of non-uniqueness of solution, and, moreover, the found solution may not have physical meaning. In such cases, it is reasonable to specify the load incrementally and obtain nonlinear solution for each increment. From the computational point of view, it is often efficient because the nonlinear effects will be getting smaller at each step. If the increments of load are sufficiently small in magnitude, each incremental solution can be found within one step with a high degree of accuracy. By default, the number of steps is set to 10.

Update load direction with deflection allows the user to account for change in the load vector, while applying the loading, according to the deformed geometry of the model.

Solution method. By default, the Newton-Raphson method of solving the system of nonlinear equations is used.

At each step of load application, the system of the linear algebraic equations is being solved until the **relative error** between two consecutive solutions becomes smaller than the prescribed tolerance.

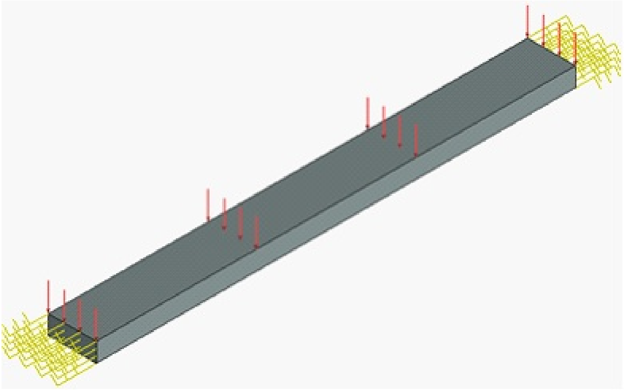
If the **maximum number of iterations** reaches the value larger than the specified one, the calculations are terminated.

Optimization

Optimization problem (or the problem of finding an optimal solution) is designed for automatic search of the values of variables which satisfy the specified constraints in the most accurate way. In the T-FLEX Analysis the sought variables can be, for example, parameters of the part's geometry, properties of the material, from which it is manufactured, (Young's modulus, Poisson's coefficient, etc) and, also, applied loads. The distinct feature of the optimization problem used in the T-FLEX Analysis is that the search of the optimal solution is carried out taking into consideration the calculation results of the finite element analysis problem.

Problem of optimizing beam thickness

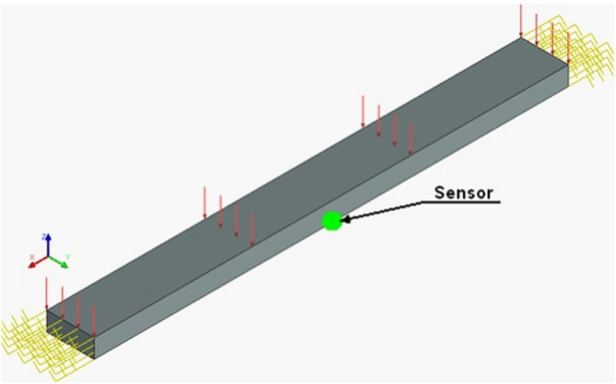
This example illustrates the use of optimization for a 3D model taking into account finite element analysis calculation results. The example represents a solution to the problem of beam deflection. The beam clamped at both sides is subjected to the force 25 N.



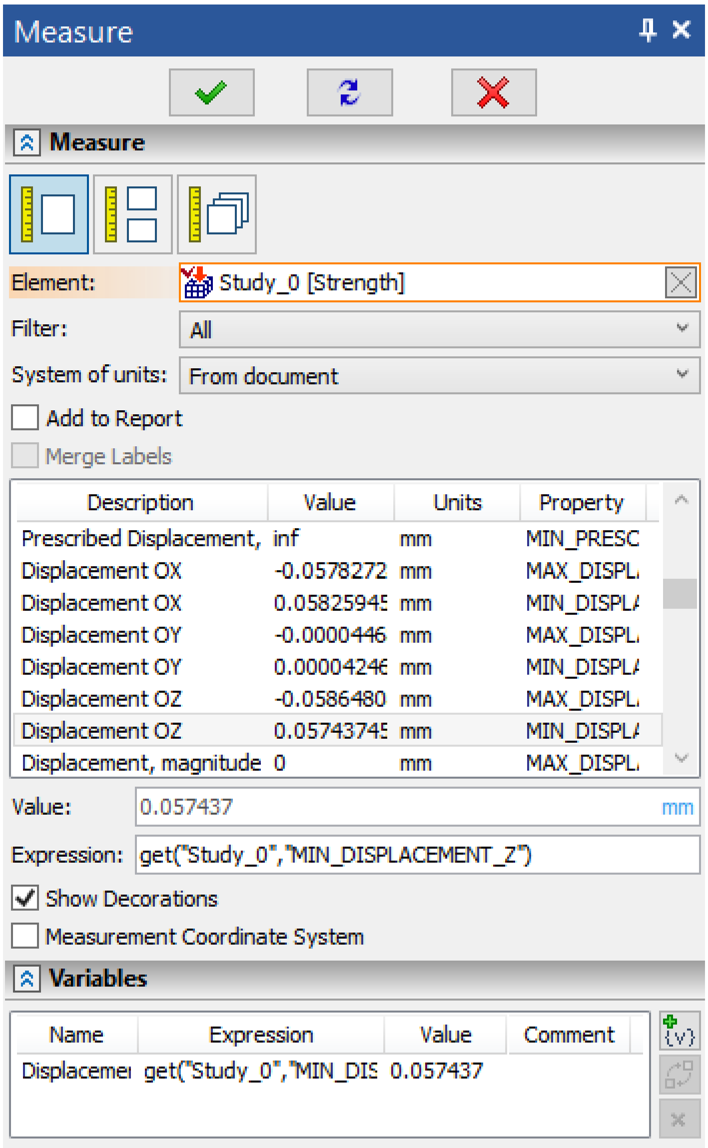
It is required to determine an admissible thickness of the beam at which the maximum value of the deflection in the direction coinciding with the direction of load does not exceed 1e-6 m in absolute value. Length of the beam is equal to 500 mm, width of the beam is 50 mm, thickness of the beam is 20 mm. The beam is manufactured from steal AISI 1020.

Let us first create the problem of static analysis, build finite element mesh, specify material, and apply the load and restraints.

In order to use the finite element analysis calculation results in the optimization problem, which we are going to solve next, let us create a sensor at the middle of the beam edge since the largest displacements are expected here.



Next, after carrying out the static analysis of the problem, let us create a variable with the help of a sensor. As a variable, we choose «Displacement OZ» and call it «Displacement_Z».



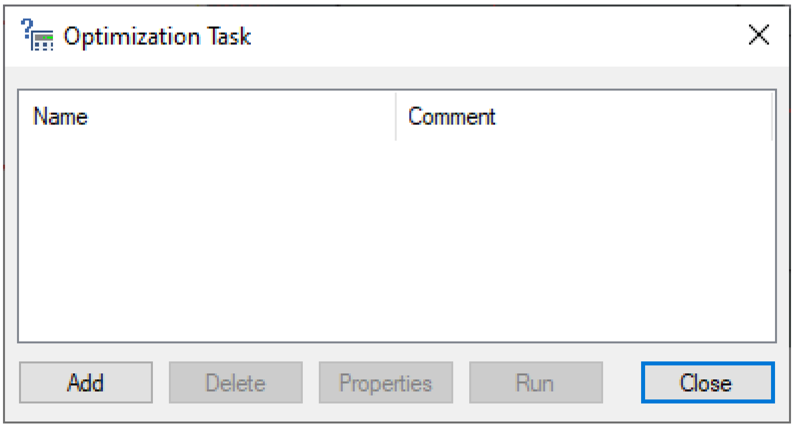
Before the next step, you need to create a variable **Thickness** equal to 1.

Let us create optimization problem. The optimization problem can be created with the help of command:

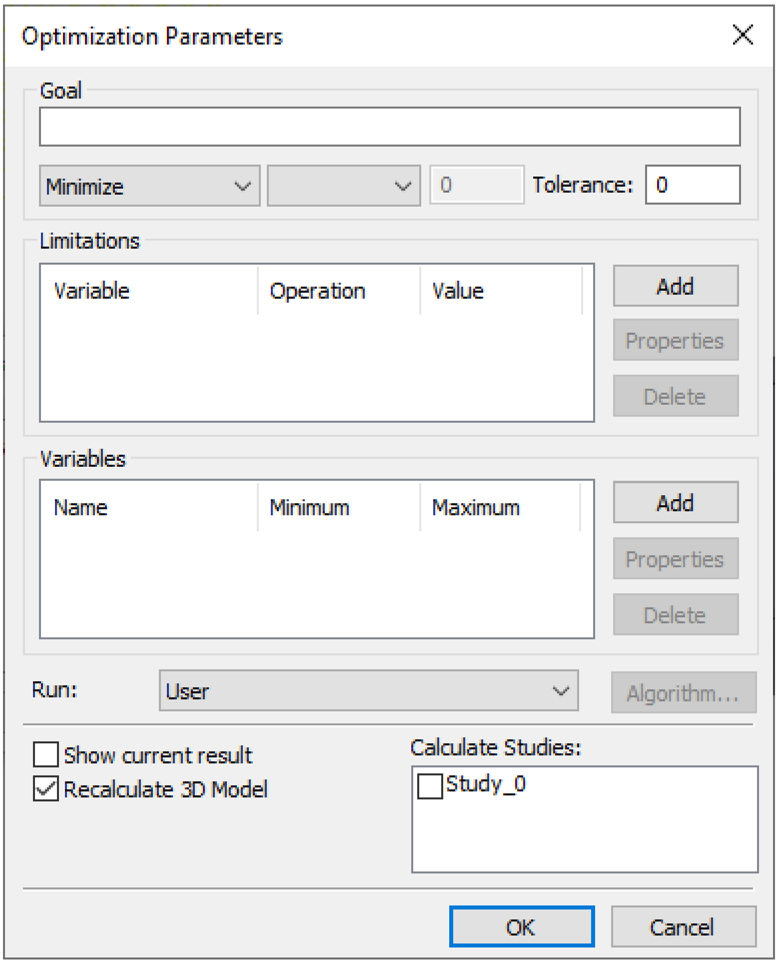
Icon	Ribbon
	Parameters > Tools > Optimize
Keyboard	Textual Menu
<PO>	Parameters > Optimize

This command can be invoked only upon the presence of numeric variables in the document.

After invoking this command, the window **Optimization Task** appears – it contains the list of created optimization studies.

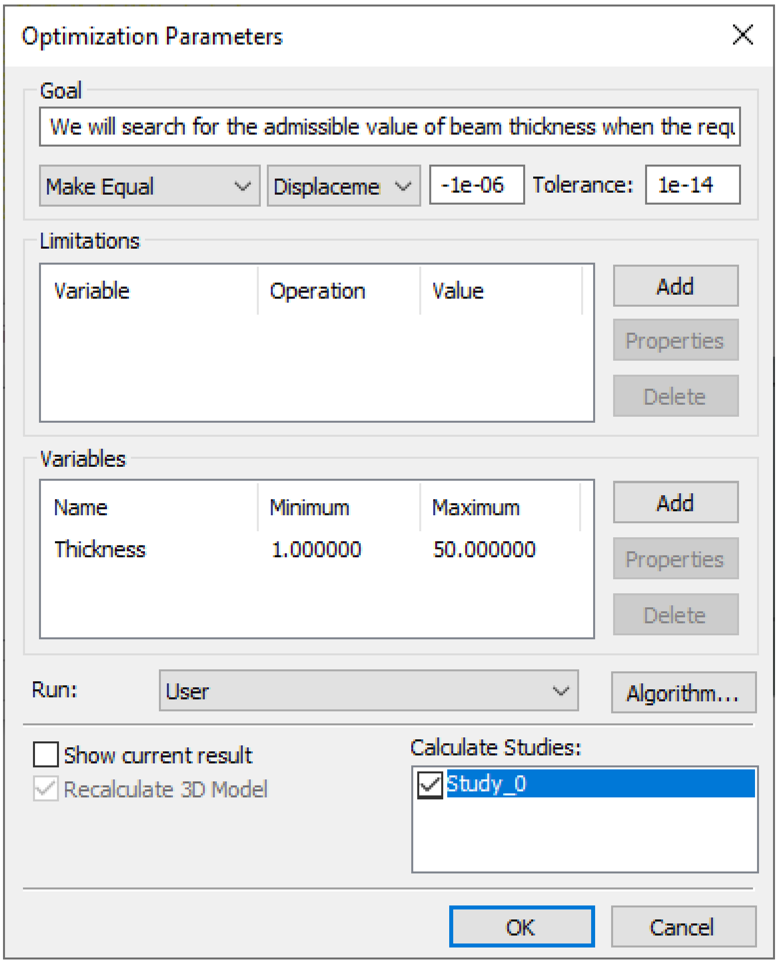


Let us add a new study to this list. After pressing the button **Add** the window **Optimization Parameters** appears on the screen.

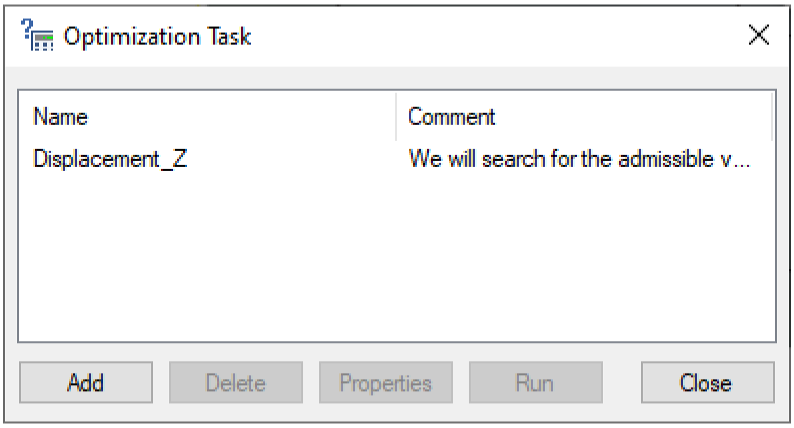


In the text line it is necessary to write down commentary for the current optimization problem.
We will formulate the goal of the optimization problem in the following way: *We will search for the admissible value of beam thickness when the required deflection in the direction of the Z-axis is equal (in absolute value) 1e-6m*. Therefore, we select the value **Make Equal**, indicate the variable «Displacement_Z» and enter its objective value equal to -1E-06. We will set the error in the found value (**Tolerance**) equal to 1E-014.
In the group **Variables** let us specify the definition domain of the variable **Thickness**. We set the minimum value equal to **1** mm, the maximum value equal to **50** mm.
Since upon changing the beam thickness its deflection along the Z-axis also changes, while solving optimization problem it will be necessary to recalculate 3D model and the static analysis problem for

each found value of the thickness. Thus, in the field **Calculate Studies** it is necessary to select the study whose results are used for searching the optimal value of the variable **Thickness.**



Press the button **OK**. The window **Optimization Task** will appear again. To perform calculation, it is necessary to select the required study from the list of studies and press the button **Run**.



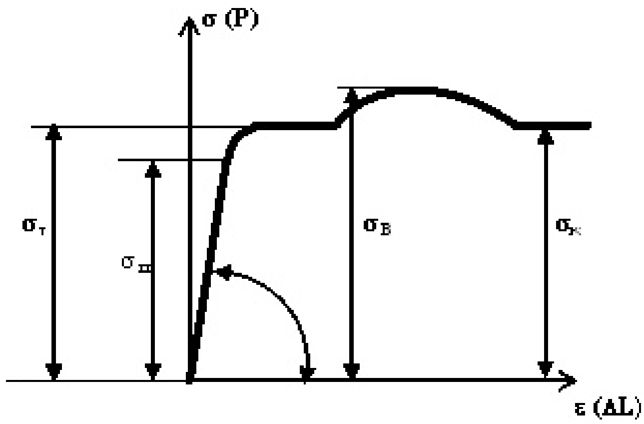
As a result of solving optimization study, we obtained the value of the beam thickness equal to 21.09mm, whereas the deflection along the Z-axis is equal to -9.07355E-007 m.

Appendix (References)

Properties of Structural Materials

A proper assignment of material properties used in the structure is an important prerequisite for correctness of finite element analysis. The main properties of structural materials used for strength analysis in T-FLEX Analysis are:

A material's Elastic Modulus E, [N/m^2] - is the ratio of stress with respect to relative strain $\sigma = E\varepsilon$ developing in a prism-shape speciment subjected to an axial force in a tensile test. In this case, a uniform stress state exists in the mid-part of the specimen in the longitudal direction. The value of the Elastic Modulus E on the strain graph $\sigma = f(\varepsilon)$ is numerically equal to the tangent of the tilt angle of the linear segment: $E = tg\beta$ on the stress graph when testing a specimen. The physical sense of the *E* modulus is described as the stress required for doubling the specimen length. However, the value of elastic elongation seldom reaches even 1% for most of solid bodies.
The stress graph of the tested speciment has several special points corresponding to changes in physical properties of the material and is used for evaluating the degree of material reliability under load.
Elasticity limit sn - the stress that is the upper bound in effect of purely elastic strain.



The stress (dilation) diagram for plastic materials (for example, low-carbon steel)

Yield limit ***sT***. Further elongation of the specimen (for example, for low-carbon steels) occurs practically with no increase in the load.This phenomenon is called "**plastic flow**", and the horizontal part of the diagram immediately to the right from the bend point is called the **plastic flow range**. In many structural materials the plastic flow range is not so prominently visible, as in low-carbon steels. The notion of the **conditional yield limit** σ_s is introduced for such materials; this is the stress corresponding to the residual (plastic) strain equal to s %. Usually, s = 0.2%. The Yield Limit for plastic materials is selected as the strength criterion - the maximum **safe stress [y]**. Reaching stresses corresponding to the yield limit causes irreversible plastic strains in the structure, thus breaking its viability and is thus an inadmissible behavior from the safety viewpoint.
The ultimate strength *sr* (rheological resistance) is the stress, upon exceeding which the material rupture occurs. Upon an increase in the loads, there is a moment, after which more strain builds up in the specimen without an increase, or even under a reduction in, the load, up to the rupture.
The Poisson's ratio μ characterizes transverse strain developing in a stretching specimen. In the elastic zone, the strain in the transverse direction is $\varepsilon' = -\mu\varepsilon$, where ε - the strain in the longitudial direction, μ — the Poisson's ratio. For isotropical materials, the Poisson's ratio lies in the range $0 < \mu \leq 0,5$.
For various steel grades, *E = 195-206 GPa*, *μ = 0.23-0.31*; for aluminum alloys, *E = 69-71 GPa, μ = 0.30-0.33.*
Elastic properties of some materials are given in the table (the denominator indicates the respective compression property).

Material	**Property**				
	E, GPa	**sT,MPa**	**sE,,MPa**	**d, %**	**y, %**
Steel ST.3	200	240/240	450/-	26	50

Steel 15	200	210/210	350/-	28	55
Steel 45	200	340/340	610/-	24	45
Steel 30HGSA	200	950/950	1200/-	13	-
Cast iron S-Ch-15-32	150	-	150/640	0,6	45
Copper wire	110	250/250	320/-	15	-
Duralumin D16	75	240/240	420/-	18	-
Delta wood (plywood)	20	-	250/160	-	-
Textolite	30	75/115	127/168	1,5	-

A material's plasticity properties are the relative elongation and relative contraction at rupture:

$$\delta = \frac{l_k - l_0}{l_0} 100\% \quad , \quad \psi = \frac{F_0 - F_k}{F_0} 100\%$$

where l0, F0 -the length of the working part of the specimen and the area of the cross-section before strain; lk - the length of the working part of the specimen after the rupture; Fk - the final area of the cross-section at the specimen's neck after the rupture.

The plastic and brittle material states are distinguished by the amount of relative elongation at rupture. Materials developing sufficiently high values δ *(δ > 10%)* at the point of rupture are referred to as plastic materials; those referred as brittle are the materials with relative elongation of δ *< 3%*. For plastic materials, upon compressing to nearly the yield condition, the σ *= f(ε)* graph pattern is the same as in the case of tension. Under a compression strain, the specimen shortens; meanwhile, its cross-section dimensions grow. For certain plastic materials, it is impossible to find the stress analogous to the tensile rheological resistance due to the specimen's flattening.

Brittle materials exhibit much better ability to resist compression strain, rather then dilational strain; for those, the compression rupture strain exceeds the ultimate tensile strength multifold. Rupture of brittle materials under compression occurs due to cracking.

Volume Stress-Strain State at a Point

The deformed state at a point of a deformable body is described by the symmetrical strain tensor:

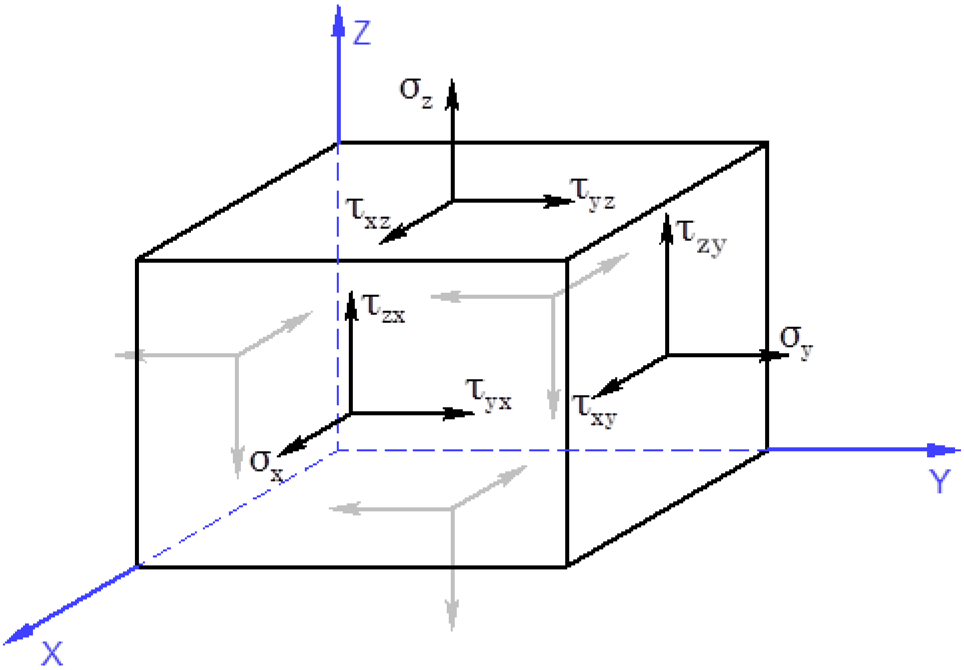
$$\varepsilon = \begin{bmatrix} \varepsilon_x & \frac{1}{2}\gamma_{xy} & \frac{1}{2}\gamma_{zx} \\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & \frac{1}{2}\gamma_{zy} \\ \frac{1}{2}\gamma_{xz} & \frac{1}{2}\gamma_{yz} & \varepsilon_z \end{bmatrix},$$

where $\varepsilon_1, \varepsilon_2, \varepsilon_3$. - the longitudinal relative strain, $\gamma_{xz} = \gamma_{zx}, \gamma_{yx} = \gamma_{xy}, \gamma_{zy} = \gamma_{yz}$ - the angular strain. You can always specify the three orthogonal directions, so that the sheer angles are all zeros, while elongations are $\varepsilon_1 \geq \varepsilon_2 \geq \varepsilon_3$. The strains $\varepsilon_1, \varepsilon_2, \varepsilon_3$ in the directions, for which sheer angles are absent, are called **principal strains** at a point.

Together, the nine stress components (by three per each of the mutually perpendicular facets) make up a physical entity called, stress tensor at a point. The tensor is represented by a symmetrical matrix:

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix},$$

where $\sigma_1, \sigma_2, \sigma_3$ - the compression-tension stress, $\tau_{xz} = \tau_{zx}, \tau_{xy} = \tau_{yx}, \tau_{zy} = \tau_{yz}$, - the shear stress.



The stress state tensor components in an infinitesimal block

The following rule of signs is commonly used for stress tensor components: a component is positive, if it points in the positive direction of the respective coordinate axis of a facet with the positive external normal aligned with one of the coordinates.

Both the stress tensor and the strain tensor possess the symmetry property. $\tau_{xz} = \tau_{zx}$, $\tau_{xy} = \tau_{yx}$, $\tau_{zy} = \tau_{yz}$.

The symmetry conditions of the stress tensor are also referred to as **paired sheer stresses** condition: the sheer stresses acting on two mutually perpendicular facets in the directions orthogonal to the edge in the intersection of those facets are equal in magnitude. Due to these properties, out of nine components of the stress tensor there are six independent ones.

Just like in the case of the strain, the concept of **principal stresses** is introduced in a stress state, $\sigma_1 \geq \sigma_2 \geq \sigma_3$, corresponding to principal strains, related with the stress tensor components by the equation:

$\sigma^3 - J_1\sigma^2 + J_2\sigma - J_3 = 0,$

where

$J_1 = \sigma_x + \sigma_y + \sigma_z,$

$$J_2 = \begin{vmatrix} \sigma_x & \tau_{xy} \\ \tau_{yx} & \sigma_y \end{vmatrix} + \begin{vmatrix} \sigma_x & \tau_{xz} \\ \tau_{zx} & \sigma_z \end{vmatrix} + \begin{vmatrix} \sigma_y & \tau_{yz} \\ \tau_{zy} & \sigma_z \end{vmatrix},$$

$$J_3 = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix}$$

A cubic equation solution has three real roots $\sigma_1, \sigma_2, \sigma_3$, which are commonly ordered as follows: $\sigma_1 \geq \sigma_2 \geq \sigma_3$. The principal stresses possess an important property: the normal stresses on the principal direction-oriented facets are highest among those on any other facets. Also introduced is the concept of mean stress by the formula: $\sigma_0 = (\sigma_x + \sigma_y + \sigma_z)/3 = (\sigma_1 + \sigma_2 + \sigma_3)/3$

Structure's Static Strength Assessment. Strength Theories

The ultimate safe stress state is when material properties undergo a qualitative change - a transition from one mechanical state to another one. For plastic materials, the safe strain state is commonly considered as the condition of developing noticeable residual strain, while for brittle ones - a condition when a material begins to crack. The ultimate state is not admissible for materials. Therefore, when performing strength analysis, pursue the so-called admissible state. It corresponds to the load obtained by dividing the load of the ultimate-strength state by a safety factor. If the safety factors are equal in two stressed states, than those are called equally fail-safe. To compare various strained states, the simple tension (compression) is accepted as the universal measure, with the principal stress σ_{equiv}.

Equivalent stress σ_{equiv} - the stress to be developed in a stretched specimen in order to make its state equally unsafe as a specified stress state. The strength criterion is written out as $\sigma_{equiv} \le [\sigma]$.

Strength theories are hypotheses about criteria describing the conditions of a material reaching the ultimate strength state.

First strength theory

In the first strength theory, a material's ultimate strength refers to the maximum normal stress. According to this theory, the unsafe state occurs when one of the principal stresses reaches a safety threshhold. Accordingly, the magnitude of the maximum principal stresses is limited so as not to exceed the **maximum principal stress** $[\sigma]$. The strength criterion appears as:

$$\sigma^{I}_{equiv} \le [\sigma],$$

where $\sigma^{I}_{equiv} \le |\sigma_1|$, if $\sigma_1 \ge |\sigma_3|$ and $\sigma^{I}_{equiv} = |\sigma_3|$, if $\sigma_1 \le |\sigma_3|$.

Second strength theory

The second strength theory uses the **maximum strain** as the ultimate strength criterion. According to this theory, the unsafe state of a material occurs when linear strain reaches a certain safety threshhold. For a plastic material, the strength criterion appears

as $max|\varepsilon| \le [\varepsilon]$, where $[\varepsilon] = [\sigma]/E$.

If, for example,

$$max|\varepsilon| = \varepsilon_1 = [\sigma_1 - \nu(\sigma_2 + \sigma_3)]/E,$$

then

$$\sigma^{II}_{equiv} = \sigma_1 - \nu(\sigma_2 + \sigma_3) \le [\sigma].$$

For brittle materials, the strength criterion appears as:

$$\varepsilon_{max} \le [\varepsilon_p] = [\sigma_p]/E, \; |\varepsilon_{min}| \le [\varepsilon_c] = [\sigma_c]/E$$

The first theory yields good agreement with experimental data only for brittle materials. The second one is practically abandoned nowadays.

Third strength theory

In the third strength theory, the ultimate strength refers to the **maximum sheer stress**. According to this theory, the unsafe state occurs when the maximum sheer stress reaches a safety threshhold.

The strength criterion appears as: $\tau_{max} = 0{,}5 \cdot (\sigma_1 - \sigma_3)$, where $[\tau] = 0{,}5 \cdot [\sigma]$. Consequently:

$$\sigma^{III}_{equiv} = \sigma_1 - \sigma_3 \le [\sigma].$$

Fourth (energy) strength theory

The fourth strength theory is based on the energy approach, based on the hypothesis that the cause of an unsafe state is the magnitude of the potential energy density of deformation $u_{ф}$, therefore the criterion refers to the density of the potential energy of deformation.

We will derive the formula for the potential energy density due to distortion from the formula for the full potential energy density due to strain, by using the specific Poisson's ratio of $\mu = 0.5$.
That yields:

$$[u^{\phi}] = (1+\mu)\cdot(\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_1\sigma_3)/3E$$

The strength criterion appears as $u_{\phi} \le [u_{\phi}]$, where $[u^{\phi}] = (1+\mu)\cdot[\sigma]^2/3E$.
Consequently:

$$\sigma^{IV}_{equiv} = (\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_1\sigma_3)^{1/2} \le [\sigma]$$

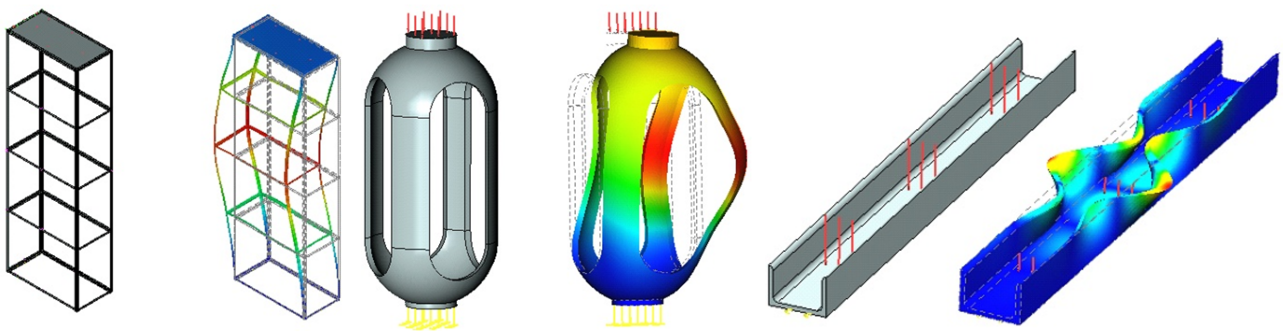
or

$$\sigma^{IV}_{equiv} = (((\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\cdot(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2))/2)^{1/2} \le [\sigma]$$

The third and fourth strength theories produce satisfactory agreement of theoretical calculation results with laboratory test data for **plastic materials** and are widely used in strength analysis. These theories are not applicable for brittle materials.

Buckling Analysis

Equilibrium of a statically loaded structure is called stable, if small disturbances cause small deformations. In certain cases of loading structures, situations are possible that are called **buckling** – when small disturbances from the forces applied to the system cause large structural deformations, which exceed those defined within the framework of the linear theory of elasticity. Loads that cause buckling are called **critical**, and the respective states – **critical states**. Under compressing forces, which even insignificantly exceed the critical value, additional bending stresses reach quite large values and directly threaten structural integrity. Therefore, the critical state which immediately precedes rupture, is considered inadmissible in real-life conditions. The threat of buckling is especially great in compressed zones for light thin-wall structures, such as slender rods, plates and shells. The buckling phenomenon exhibits various forms: completely new forms of equilibrium appear; known stable configurations deteriorate, etc.



The buckling analysis module serves for conducting the so-called **initial buckling** structural study. The result of the study is a coefficient of the critical load, under which the structure may spring into a new equilibrium state, and the shape of the new equilibrium state corresponding to that load. In such a case, a situation is possible, when the critical load, under which buckling occurs, could be much less than the load, under which the maximum strength of the structure will be exceeded based on the linear static stressed state of the structure. In other words, the stresses in the structural material may not reach the ultimate values, but deformations due to buckling may cause structural rupture. Therefore, the buckling condition can be formulated based on the critical load criterion as follows:

> Actual loads applied to a structure, must be less than the estimated critical load, subject to an asserted safety factor: $F_{factual} \cdot K_{safety} < F_{critical}$

Having estimated the value of the critical load, under which the structure may buckle, you can optimize the part in order to achieve the safe condition. For example, for a slender object, you can increase resistance to buckling by reducing the length or increasing the thickness of your object, or create additional ribs for rigidity.

Topics in this section:

Details of Bucking Analysis Steps

Buckling Analysis is performed in several stages. The order of steps for the user to put together a study and run calculations of the structural buckling analysis is mostly similar to the algorithm described for the **Static Strength**. Therefore, in this chapter we will just mention what is special for the buckling analysis:

Step 1. Creating Study

When creating a study, specify its type – **Buckling.**

Step 2. Applying boundary conditions

Just like in the static analysis, the buckling analysis uses restraints and loads as boundary conditions. All types of restraints and all types of forces can be used in the buckling analysis. The thermal impact is defined in the same way as in the static analysis. Defining restraints and forces is a necessary condition for running a correct analysis. The combined limitation on the model must satisfy the following condition:

> To be suited for a buckling analysis, a model should have restraints preventing free movement in the space as a solid body. Failing to meet this condition will cause incorrect results of Finite Element analysis or abortion of computations.

Note also that defining loads properly is important for the initial buckling study to be correctly formulated. In particular, in certain loading cases, the solution to the study may not have physical sense (for example, in the case of a rod being stretched with a tensile longitudinal force). Correctly defined boundary conditions are signified by a positive value of the critical load coefficient, resulting from the study.

Step 3. Running calculations.

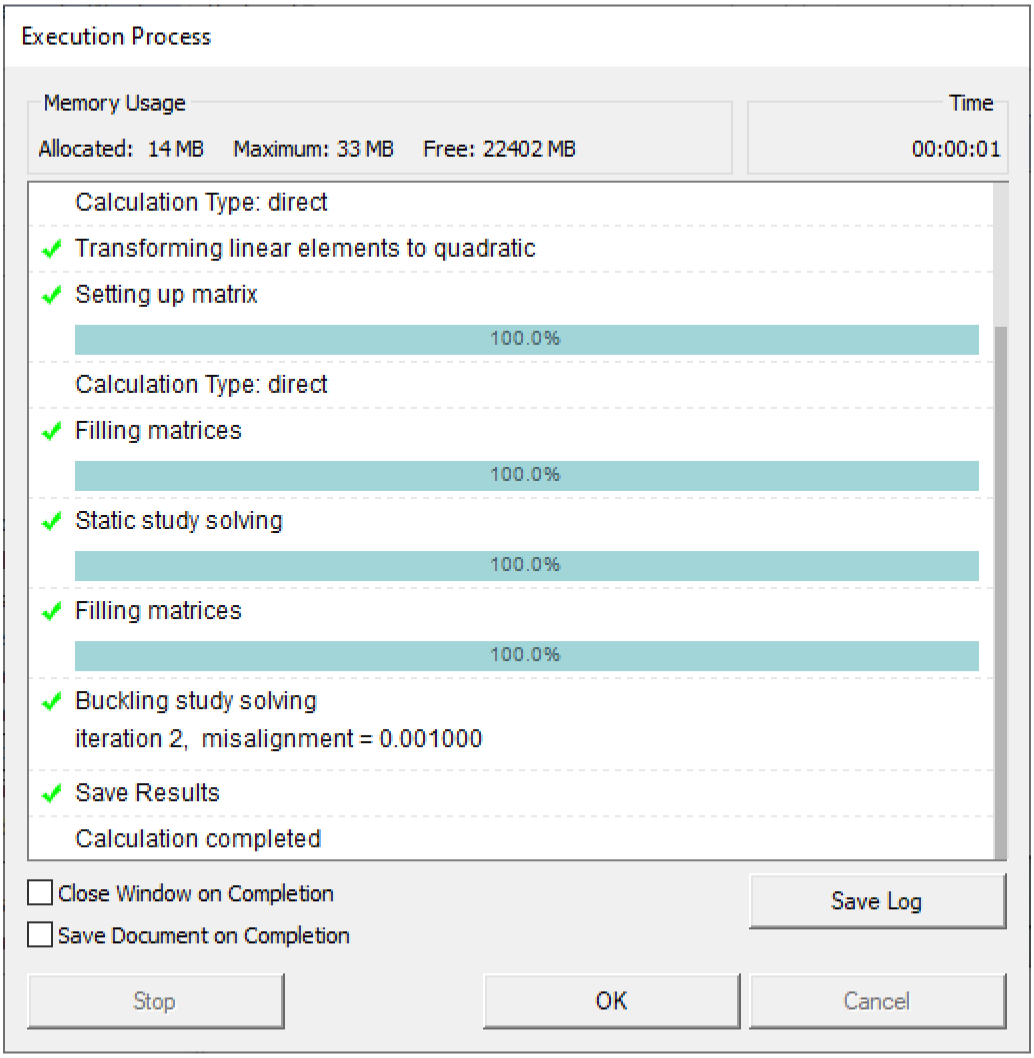
Before running calculations, the user specifies computational algorithms and the number of buckling modes to be analyzed, in the study parameters. The following data is output into the information window when running calculations:

Nodes - the number of nodes in the computational finite element mesh.

Elements - the number of tetrahedra in the finite element mesh.

Arguments – the number of equations used in the calculation.

Calculation complete – this message signifies that the solution process is completed successfully.



Step 4. Results

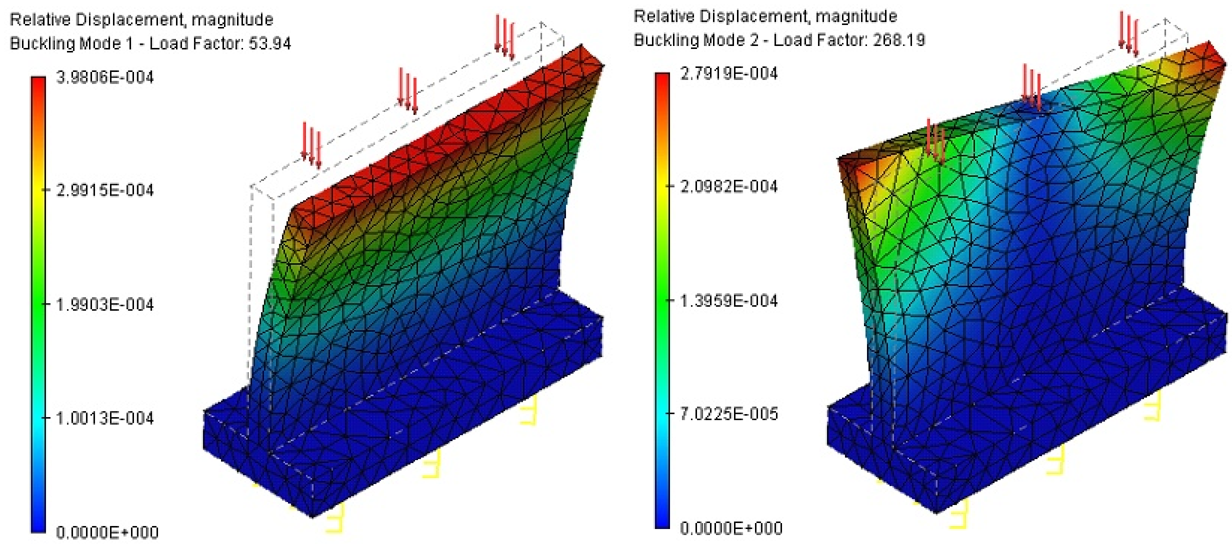
The following are analysis results:

Load Factor – the calculated value of the coefficient, the product of which and the loads applied to the system makes the factual value of the critical load, which brings the system into a new equilibrium state. For example, a distributed force of 1000 H is applied over the model. The Load Factor, as calculated, is equal to 109.18 . That means, that the first mode of an equilibrium state for the given model has the critical load equal to 109180 H.

> Load Factor must be positive. If calculations have resulted in a negative Load Factor, that means, no buckling can be produced by the loads applied to the structure.

Relative displacements, corresponding to a given critical load. This type of result reflects on a buckling mode of the structure corresponding to a certain critical load. The buckling modes displayed in the postprocessor window after completing calculations are relative displacements. By analyzing those modes, you can make a conclusion about the pattern of displacements in a buckling condition. By knowing the expected buckling mode under a certain critical load, one could, for example, introduce an additional restraint or a support in the part of the structure corresponding to the maximum of buckling in this mode, which would effectively alter mechanical properties of the product.

As an additional (reference) result, you can also output displacements of the structure under the applied static loads, whose calculations preceded the evaluation of the critical load factors.



Buckling modes, corresponding to the first and second critical loads on the part

Evaluation of Results

Once the study calculation completes successfully, you should analyze solution results in order to make conclusions on probabilistic buckling of the structure based on results of Finite Element modeling. A typical sequence of steps for validating the results of Finite Element modeling of initial buckling is as follows:

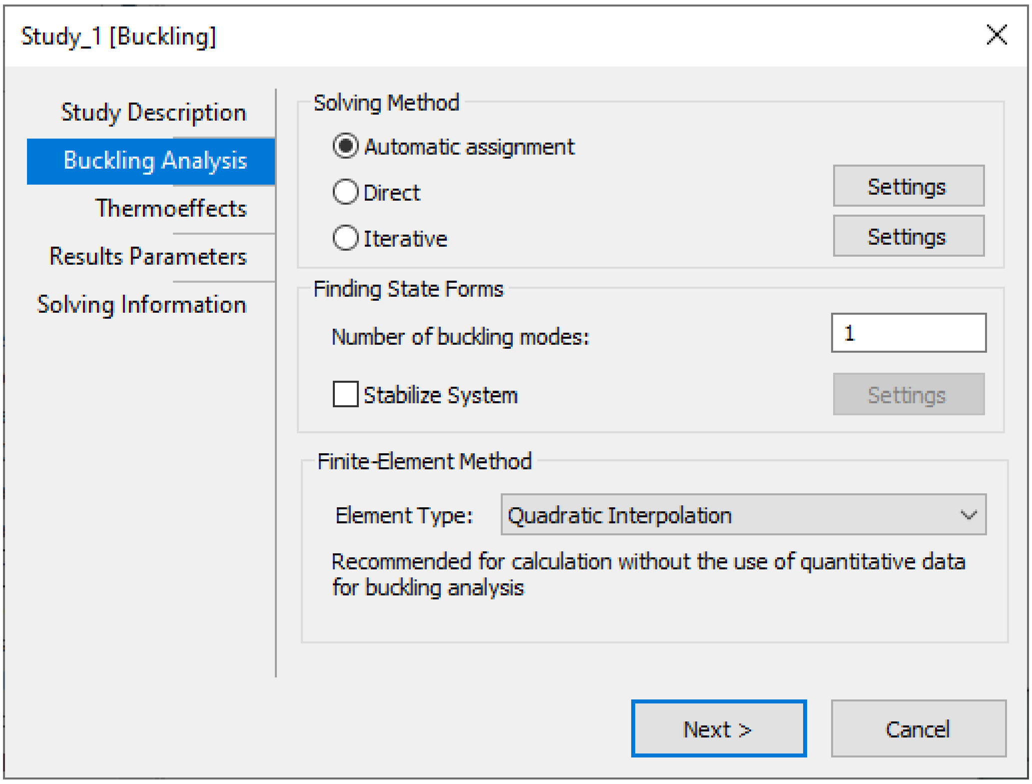
1. **Solution evaluation**. As was mentioned earlier, the Load Factor must be positive. If the factor came out negative, that means the loads applied to the structure do not produce system buckling.
2. **Load factor evaluation**. If the Load Factor is positive and is less than 1, that means system buckling will occur under the specified loads, and that the design of the structure needs enhancement. If the Load Factor is positive and is greater than 1, that means there is no buckling threat for the structure under the specified loading conditions.
3. **Buckling modes analysis**. In the studies tree, use the context menu command **Open** or **Open in new window** to open the "Buckling mode 01" solution, corresponding to the smallest critical loading. We can visually estimate the pattern of the strained state of the structure. The buckling analysis allows making a conclusion about directions and locations of maximum displacements, corresponding to a critical load. This information can be used for optimizing the product's design with the purpose of increasing its resistance to buckling.

Study Parameters

Upon initializing the **Analysis** > **Solve** command, the dialog of defining study parameters appears by default, with several tabs to switch between. The user can change the default study parameters. To access a study's parameters, one can also double-click a study's name in the studies tree, or in the context menu right-click on the name of the selected study in the studies tree. The user-defined study parameters are saved together with the document and are inherited upon copying a study. The main purpose of this study parameters is defining the modes of the Processor's operation, as well as the lists of results and the number of buckling modes displayable in the studies three after calculations.

On the **Study Description** tab, you can define or modify the descriptive parameters of the current study: the name, the study type, the comment.

On the **Buckling Analysis** tab, you can define processor properties for solving the equations.

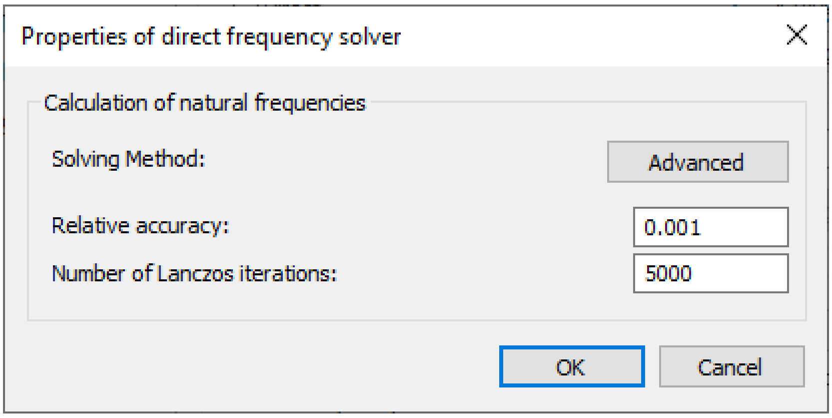


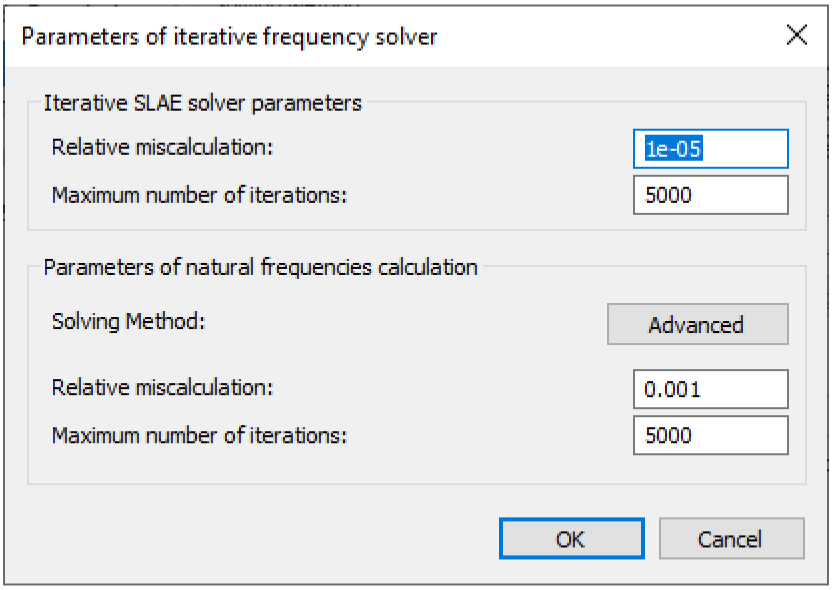
"Solving Method" group:

Automatic assignment – equations solution method is automatically selected by the system based on the total number of equations. By default, the threshold number is equal to 100 000 equations (degrees of freedom) and is set on the **Settings > Processor** page. If the total number of equations exceeds this value, an iterative method is used for solving equations; otherwise the system of equations is solved by direct methods.

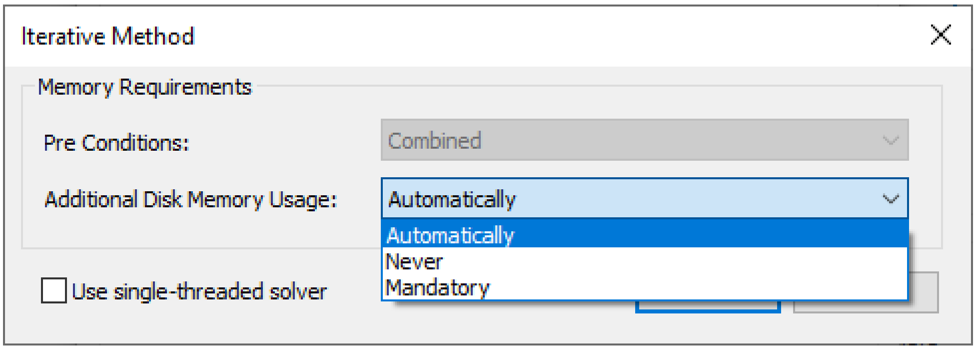
Direct – the system of equations is solved by Gauss method, which takes a lot of RAM as compared to the iterative method. If sufficiently large amount of RAM is available, then the direct method renders the solution quicker than the iterative method. The direct method is preferred for relatively small problems with the number of degrees of freedom smaller than 100 000, however this number can increase depending on the amount of available RAM. This method is also preferred for linear finite elements. After pressing the **Settings** button, the window of properties of the direct frequency solver appears in which the user can select the relative tolerance for finding critical loads and the number of Lanczos iterations (see below).

Iterative – the system of equations is solved by iterative methods, which do not require complete inversion of the matrix, which takes lesser RAM. Calculation time is approximately proportional to the number of natural vibration modes sought. If the finite element mesh contains a lot of elements that do not have the optimal shape, e.g., stretched, then the convergence rate is significantly reduced. After pressing the **Settings** button, the window of properties of iterative stability/frequency solver appears, in which the user can select the relative tolerance for solving the system of linear equations (residue) and the maximum number of iterations in the group of settings for linear equations iterative solver; the relative tolerance for finding the natural frequencies and the maximum number of iterations for determining the eigenvalues can be specified in the group of settings for eigenvalues (natural frequencies) determination (see below).





When you press **Advanced** button, it is available to indicate possibility of using additional disk memory: **Automatically, Never, Mandatory**.



"Finding State Forms" group

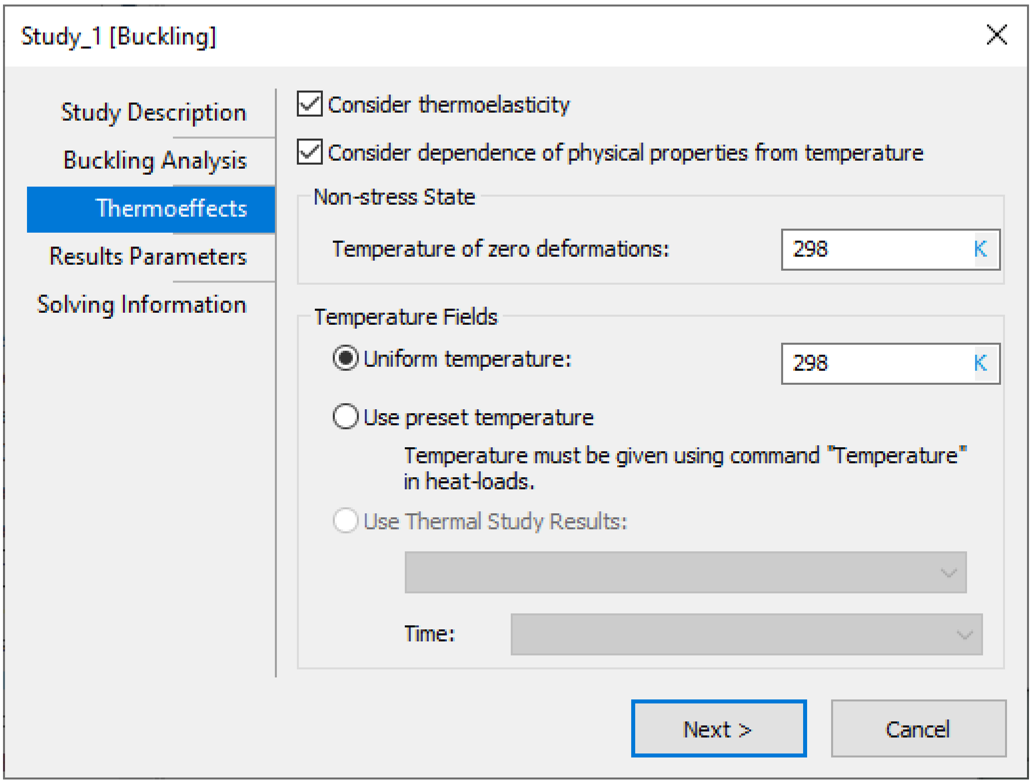
Number of buckling modes. The user can specify the number of critical loads and respective buckling modes to be identified. For practical purposes, the most important is the first mode, corresponding to the minimum critical load. Nevertheless, the user may also find the critical loads of other buckling modes.

In the group **Finite-Element Method** the user can set the mode **Linear Interpolation**. This facilitates much faster calculation for an approximate estimate of the buckling mode amplitudes relative distribution on a sufficiently fine mesh.

The linear element analysis provides **insufficient** accuracy of calculating critical loads. Critical load results are much greater (by a factor of tens or hundreds of times) via the calculation by linear finite elements, rather than those achieved by more accurate methods. It is strongly recommended to use only quadratic element calculations for quantitative evaluation of the critical loads (the default mode).

"Trermoeffects" tab

Parameters of temperature changes influence on the materials properties are specified on the **Thermoeffects** tab.



Consider Thermoelasticity. If the flag is enabled the influence of heat stress to the rigidity of construction and changes of critical load value will be considered. Otherwise, the condition of heat stress will not be considered.

If the option **Consider dependence of physical properties from temperature** is enabled, the material properties will be taken from the graphs showing dependences of materials properties to the temperature. The properties are set in the parameters for each material.

Other parameters on the tab work in the same way as in another static analysis studies.

The **Results Parameters** tab sets the displayable result types in the studies tree after finishing calculations.

Fatigue Analysis

Certain parts of machines, mechanisms and also structural elements during the time of their service are subjected to the loads that change with time. Material's resistance to the effects of these loads considerably differs from the material's resistance to static load or impact.

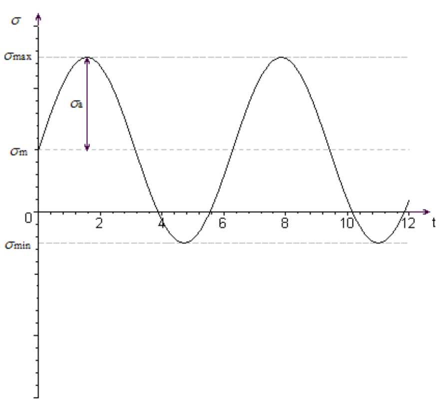
For studying material's strength when the time-dependent loads are applied, the Fatigue Analysis Module is used in T-FLEX Analysis.

By the fatigue of the material we imply the process of gradual accumulation of defects in the material, subjected to time-dependent stresses, which results in creation of crack, its propagation and ultimate failure of the workpiece.

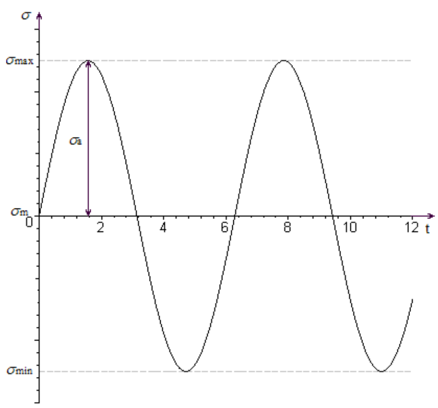
Stress cycle. Main characteristics

After a certain number of repeatedly applied loadings (or stress cycles) the ultimate failure of the workpiece can occur, while for the same loading independent of time the failure does not occur.
The number of stress cycles until the moment of failure depends on the magnitude of σ_a (stress amplitude) and changes in broad limits. When the stress is large, it is sufficient to have 5 10 cycles for the failure. When the stress becomes smaller, a workpiece can sustain millions and milliards of cycles, and for very small loads – the workpiece can work indefinitely long.
We distinguish between maximum σ_{max} and minimum σ_{min} stresses of the cycle which signify the largest and the smallest in algebraic value stresses of the cycle. As the average stress σ_m and the stress amplitude σ_a of the cycle we take:
$\sigma_m = (\sigma_{min} + \sigma_{max}) \cdot 0{,}5$, $\sigma_a = (\sigma_{max} - \sigma_{min}) \cdot 0{,}5$.
The difference between the maximum and minimum stresses of the cycle, i.e., $2\sigma_a = \sigma_{max} - \sigma_{min}$ is called the stress range.
The cycle in which the maximum and minimum stresses in absolute value are different is called asymmetric.
A special case of the asymmetric cycle is a pulsating cycle, in which the minimum stress of the cycle is equal to zero: $\sigma_{min} = 0$.
The cycle in which the maximum and minimum stresses are equal in absolute value but opposite in sign is called symmetric.
To characterize the degree of the asymmetry of the stress cycle, we introduce the cycle asymmetry coefficient by which we imply the ratio of the minimum stress of the cycle to the maximum stress: $R = \sigma_{min} / \sigma_{max}$.

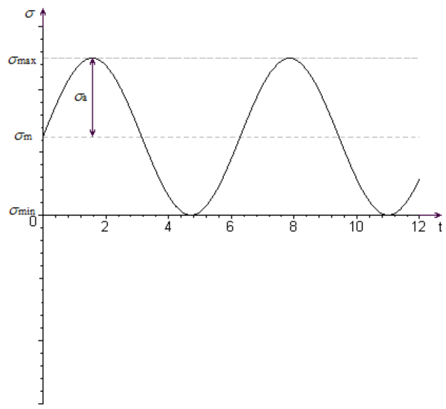
Stress cycle type	$R = \sigma_{min} / \sigma_{max}$	σ_{min}	σ_{max}	$\sigma_m = (\sigma_{min} + \sigma_{max}) \cdot 0{,}5$	$\sigma_a = (\sigma_{max} - \sigma_{min}) \cdot 0{,}5$	$2\sigma_a$
Asymmetric	R_1	$R_1 \sigma_{max}$	σ_{max}	$0{,}5(1 + R_1)\sigma_{max}$	$0{,}5(1 - R_1)\sigma_{max}$	$(1 - R_1)\sigma_{max}$
Pulsating	0	0	σ_{max}	$0{,}5\sigma_{max}$	$0{,}5\sigma_{max}$	σ_{max}
Symmetric	-1	$-\sigma_{max}$	σ_{max}	0	σ_{max}	$2\sigma_{max}$



Asymmetric cycle (R= − 0.2)



Symmetric cycle (R= − 1)



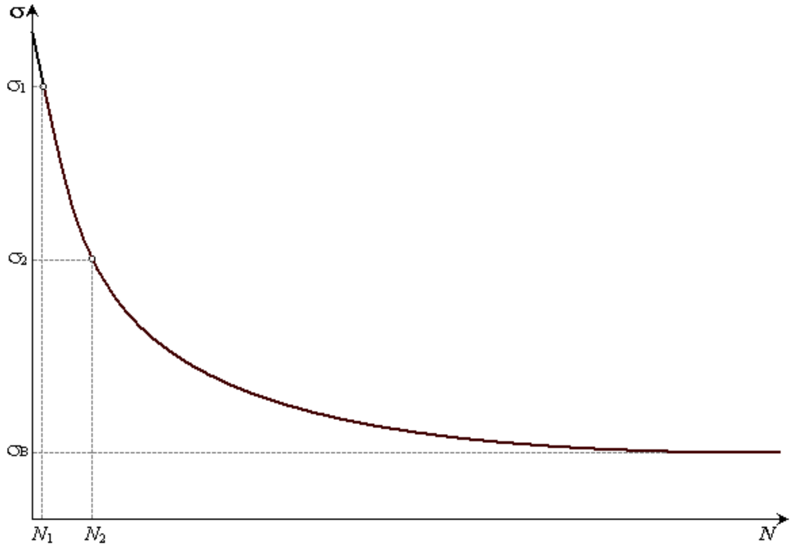
Pulsating cycle (R= 0)

S-N curve

S-N curve constitutes an experimental curve constructed by an assembly of points such that the abscissa of each of the points is equal to the number of cycles N before the failure, and the ordinate – the fatigue-limit for the given number of stress cycles.

> Fatigue-limit is the maximum stress by an absolute value which does not cause failure of the material. The fatigue-limit depends on the cycle asymmetry coefficient and is denoted by σ_R. The minimum value of the fatigue-limit is achieved for the symmetric cycle.

S-N curve shows the rule (pattern), according to which with the increase in the number of cycles the maximum stress for which the failure of the material occurs decreases.



After a certain number of cycles the ordinates σ of the S-N curve practically remain constant. Because of this fact the number of cycles (during fatigue testing of the material) is bounded by a certain limit which is called the base number of cycles. If the material sustains the base number of cycles, then it is assumed that the stress in the material does not exceed the endurance limit σ_B.

Stress Adjustment Methods

Let σ^* be the adjusted sign-alternating stress, σ_T – yield stress, $\sigma_П$ – ultimate strength. Then:

1. **Soderberg method:** $$\sigma^* = \frac{\sigma_a}{1-\left(\frac{\sigma_m}{\sigma_T}\right)}$$
2. **Goodman method used for brittle materials:** $$\sigma^* = \frac{\sigma_a}{1-\left(\frac{\sigma_m}{\sigma_П}\right)}$$
3. **Gerber method used for ductile materials:** $$\sigma^* = \frac{\sigma_a}{1-\left(\frac{\sigma_m}{\sigma_П}\right)^2}$$

Estimating fatigue resistance characteristics in complex stress state

The strength criteria for time-dependent stresses in general are similar to the strength criteria for static analysis but as a maximum admissible stress the fatigue-limit σ_R is used. Thus, in the fatigue analysis the hypotheses for checking the fatigue strength by the criterion of inadmissibility of plastic deformations take the form $\sigma^* \le \sigma_R$. Let us recall the general expressions for standard hypotheses of strength checking (by plasticity criterion) used for estimating the strength of structures:

1. **Hypothesis of Tresca – St Venant** (hypothesis of maximum shear stresses):

$$\sigma_T < (\sigma_1 - \sigma_3), \ |\sigma_1| \geq |\sigma_2| \geq |\sigma_3|;$$

2. **Hypothesis of Huber – Mises – Hencky** (hypothesis of distortion energy)

$$\sigma_T = (((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2)/2)^{1/2};$$

3. **Hypothesis of Gesta – Mohr** (hypothesis of maximum principal stresses)

$$\sigma_T < \sigma_1, \ |\sigma_1| \geq |\sigma_2| \geq |\sigma_3|;$$

where σ_T – yield stress, $\sigma_1, \sigma_2, \sigma_3$ – principal stresses.

Therefore, in the fatigue analysis we also obtain three kinds of safety coefficients that correspond to each of the generally accepted strength checking theories.

Topics in this section:

- Details of Fatigue Analysis Steps
- Evaluation of Results
- Single-event Fatigue Analysis
- Multi-event Fatigue Analysis

Details of Fatigue Analysis Steps

Before carrying out the fatigue strength analysis, it is first required to examine the effect of the static loading on the given workpiece or structure (i.e., perform static analysis). This is required in order to find out whether the workpiece experiences damage under the action of the given load. If the workpiece experiences damage for the given static load (safety coefficient less than 1), then it is meaningless to carry out the fatigue analysis.

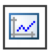
Preliminary static analysis is also necessary since the stresses obtained as a result of the static analysis (equivalent or principal) will be used for analysis of fatigue strength as the amplitudes of cyclic stresses.

In addition, when performing the fatigue analysis it is required that the **S-N curve** be determined for the material from which the product in question is made.

Step 1. Fatigue Curve

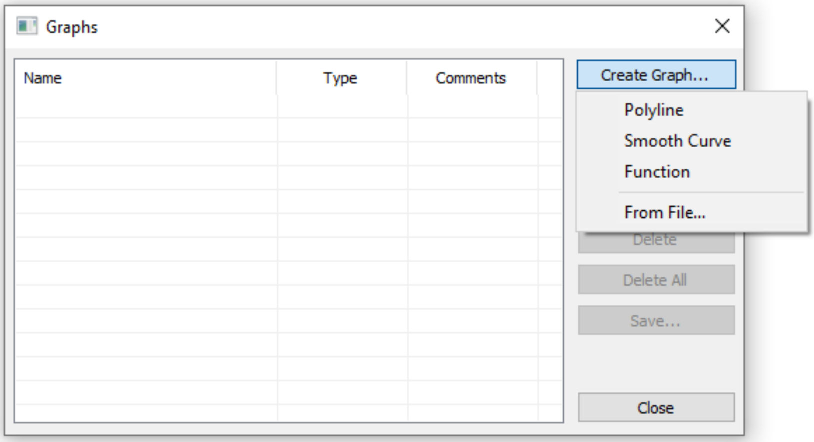
The S-N curve can be selected from already existing curves or specified independently.

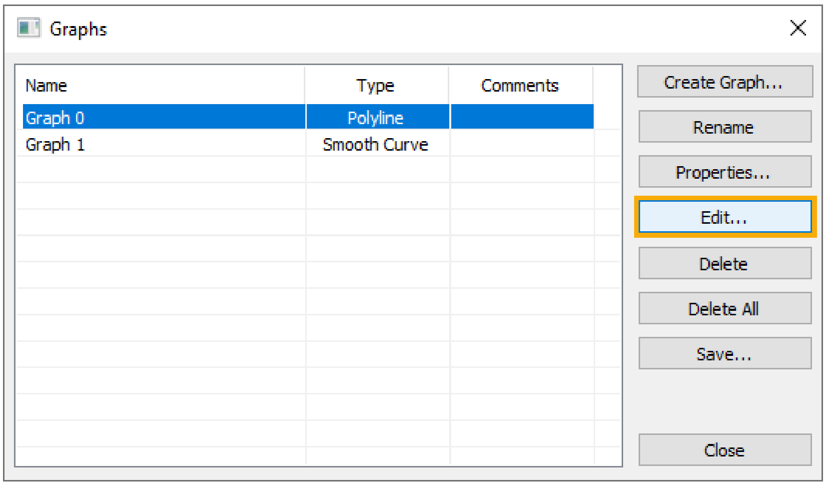
With the help of the command:

Icon	Ribbon
	Analysis > Conditions > Graph Template
Keyboard	Textual Menu
<3GT>	Analysis > Graph Template

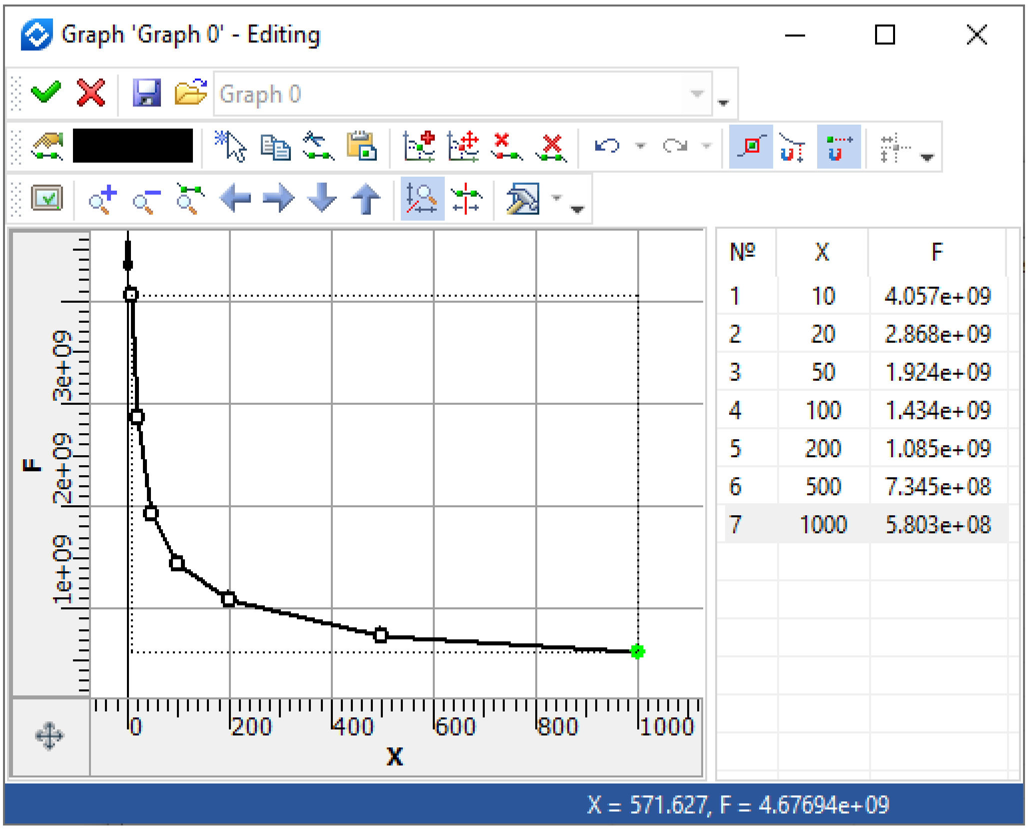
It is possible to call the dialog to work with the S-N curves. In this way the user can create new curves, edit or remove already existing curves.

The type of the curve (polyline, smooth curve, function) is determined when creating the graph.

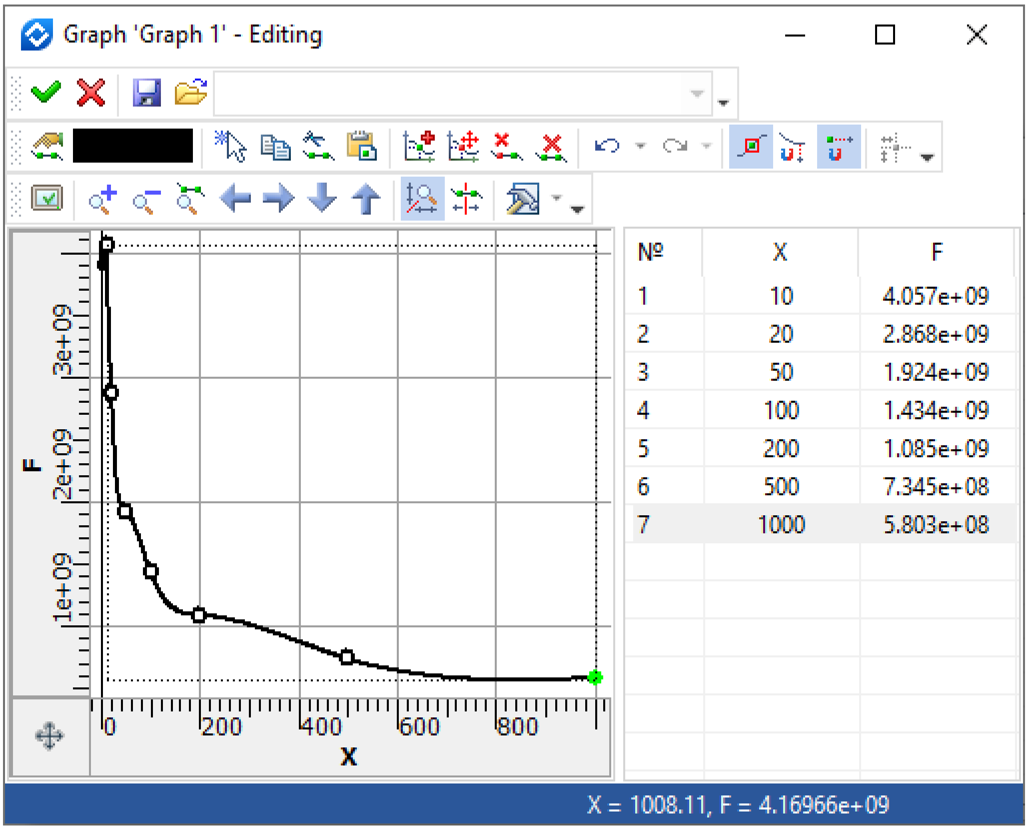




After the curve has been created (its type and name are defined), it is required to specify the nodes of the curve and the values of the function at these nodes.

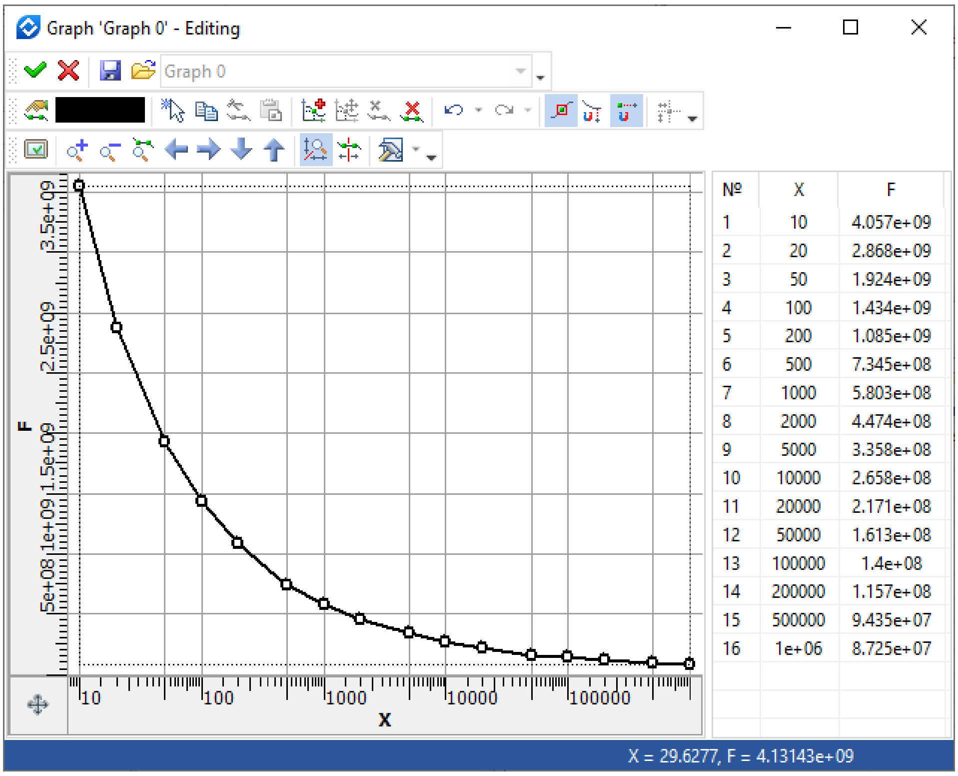


Polyline



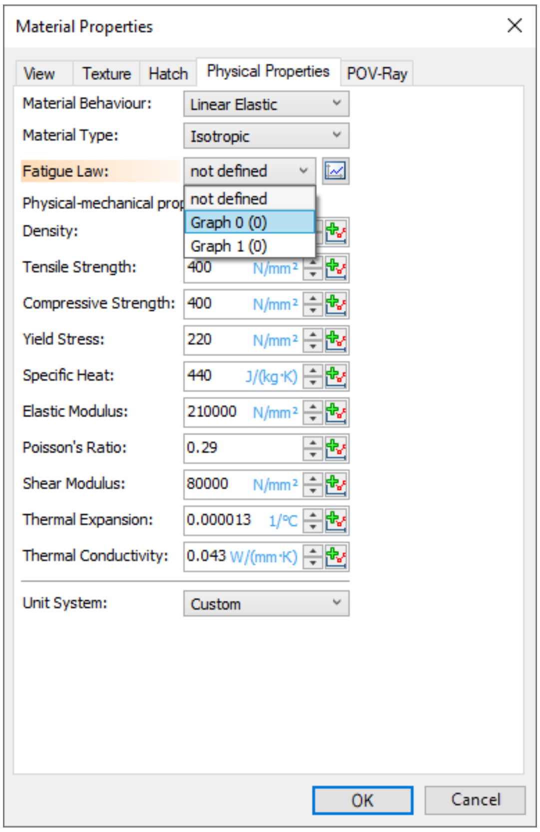
Smooth curve

The final appearance of the given S-N curve can be the following:



The graph of this curve can be saved into the external file for it to be used again later on in other fatigue calculations.

To do this, open the materials window and in the **Document** tab open the properties window for the material of the current study. Go to the **Physical Properties** tab and select the fatigue (S-N) curve.



If 3D model is an assembly, i.e., it consists of 3D fragments, then in order to attach the S-N curve to the material of the fragment, we select 3D fragment in the studies tree, then, with the help of the

context menu, call the **3D Fragment Parameters** dialog. In the dialog, set the source - **Body** and select the material, then click the button (with the help of the ... button).
In the **Material's Properties** dialog that appears we select **Physical Properties** tab and select the S-N curve in the **Fatigue Law** parameter.

Step 1. Fatigue Curve

This study is created with the help of the command:

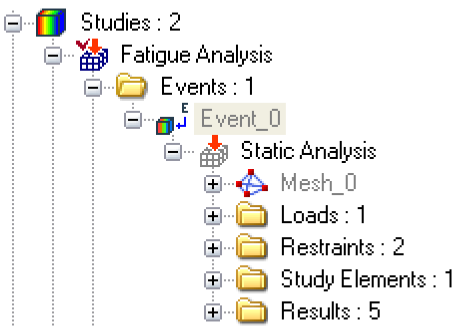
Icon	Ribbon
	Analysis > Analysis > New Study > FEA Study
Keyboard	Textual Menu
<3MN>	Analysis > New Study > FEA Study

To carry out the fatigue analysis, when creating the study the user indicates its type – **Fatigue Analysis** in the command's parameters window.
After creating the study of the **Fatigue Analysis** type, it is required to create one or more events in the study with the help of the command:

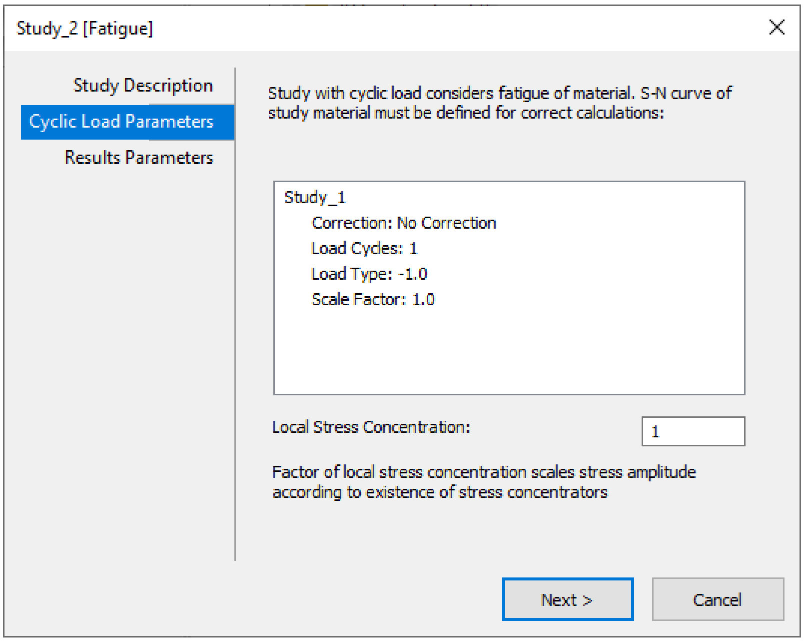
Event

Icon	Ribbon
	Analysis > Conditions > Event
Keyboard	Textual Menu
<3CE>	Analysis > Event

In a single-event analysis it is assumed that all loads applied to the system are changing in cyclic manner by the same rule (parameters such as the number of cycles and the type of a cyclic change are the same for all loads). Multi-event analysis allows us to estimate the action of several forces with different parameters of the cyclic load (different number of cycles or non-coinciding types of the cyclic loading change). Each event is added to the **Events** folder in the studies tree.



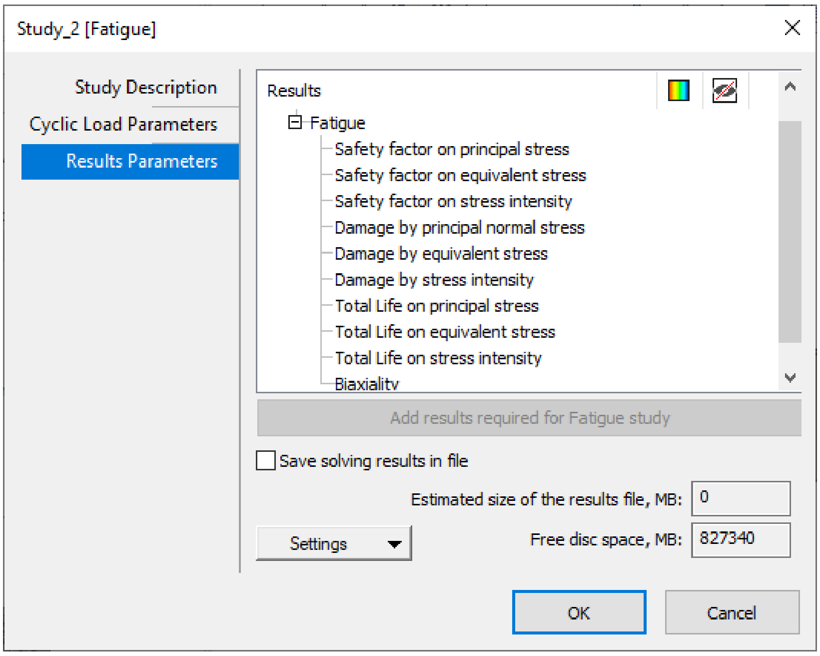
It is important to note that to carry out a multi-event fatigue analysis, it is required that the finite element meshes of all statics studies included in the fatigue analysis coincide and the same body of each statics study be made of the same material.



Then it is required to add the completed static analysis study and specify **parameters of the cyclic loading**: number of cycles, loading type, stress adjustment method (this is specified only in case if the cycle asymmetry coefficient R does not coincide with the coefficient of the given S-N curve), scale effect coefficient (stress scaling coefficient). Also in the **Study Parameters (fatigue analysis)** dialog on the **Parameters** tab it is possible to specify the local stress intensity coefficient (by default this coefficient is equal to 1).

Evaluation of Results

After completing the analysis, the new folder **Results** will appear in the studies tree. The list of displayed results can be specified using the **Results Parameters** tab of the study parameters dialog.



From the results of the fatigue analysis 10 results in total are accessible to the user.

- **Safety factor on principal stresses** (hypothesis 2);
- **Safety factor on equivalent stresses** (hypothesis 1);
- **Safety factor on stress intensity** (hypothesis 3).

The fatigue strength safety coefficient is the ratio of the fatigue-limit σ_R, determined based on the given S-N curve for a given number of loading cycles, to the adjusted amplitude of the cycle σ*, which is calculated from the stresses obtained in the static analysis study: $K = \sigma_R / |\sigma^*|$.

This type of the results is available only for a single-event analysis.

The stresses σ are evaluated according to the corresponding plasticity conditions:

- hypothesis of distortion energy: $\sigma = (((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2)/2)^{1/2}$;
- hypothesis of maximum shear stresses: $\sigma = \sigma_1 - \sigma_3$, $|\sigma_1| \geq |\sigma_2| \geq |\sigma_3|$;
- hypothesis of maximum principal stresses $\sigma = \sigma_1$.

- **Damage by principal normal stress** (hypothesis 3);
- **Damage by equivalent stress** (hypothesis 2);
- **Damage by stress intensity** (hypothesis 1);

This result is displayed in percentage and characterizes the extent of damage of the structure subjected to the cyclic stresses for the given number and type of loading cycles indicated in the analysis.

If the structure is subjected to n1 cycles of sign-alternating stress S1, n2 cycles of sign-alternating stress S2 , n3 cycles of sign-alternating stress S3 ,... , nk cycles of sign-alternating stress Sk, then the total extent of damage D is calculated as: $D = \sum_{i=1}^{k} \frac{n_i}{N_i}$, where Ni is equal to the number of cycles required to cause damage for Si.

- **Total life on principal stresses**.
- **Total life on equivalent stresses**.
- **Total life on stress intensity**.

This type of the results is available only for a single-event analysis.

The result shows the minimum number of cycles Nmin required to induce fatigue damage.

- **Biaxiality** – the ratio of the smallest principal sign-alternating stress (different from 0) to the largest principal sign-alternating stress: b = σ_3 / σ_1, $|\sigma_1| \geq |\sigma_2| \geq |\sigma_3|$.

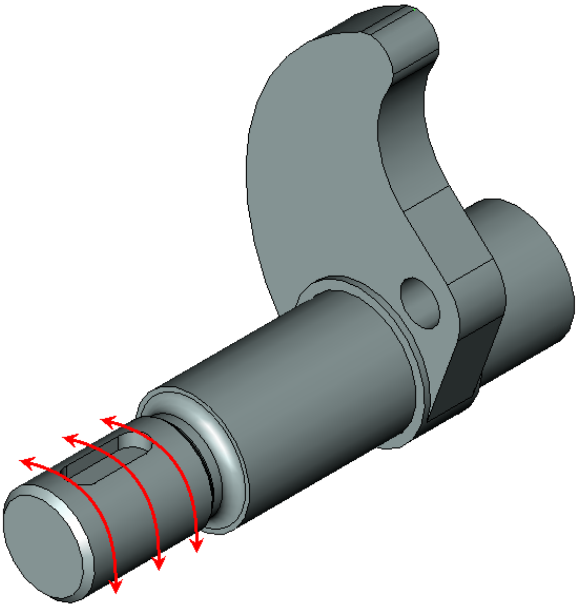
Biaxiality characterizes inequality of the amplitudes of the principal stresses at a point and characterizes spatially heterogeneous nature of the principal stress distribution over the volume of the body at each point. Biaxiality value equal to 1 corresponds to the isotropic stress state $\sigma_1 = \sigma_2 = \sigma_3$ at a point.

This type of the results is available only for a single-event analysis.

Single-event Fatigue Analysis

Let the shaft shown on the picture be subjected to the **Torque** load. Let us assume that upon the startup of the shaft and upon its braking, a bidirectional (reversible) load arises. Let us determine how many startups and stops the shaft will endure for the given torque.

The workpiece is subjected to the load periodically changing with time the amplitude values of which are symmetric with respect to 0.

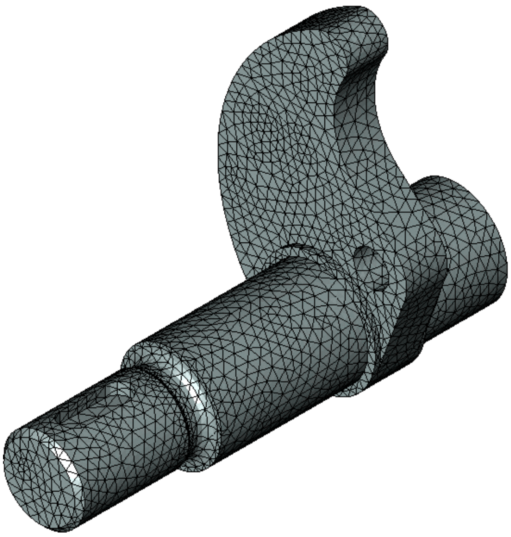


We first assume that the load is static (i.e., does not change with time) and perform the static strength analysis for the workpiece subjected to such a load.

Static Analysis

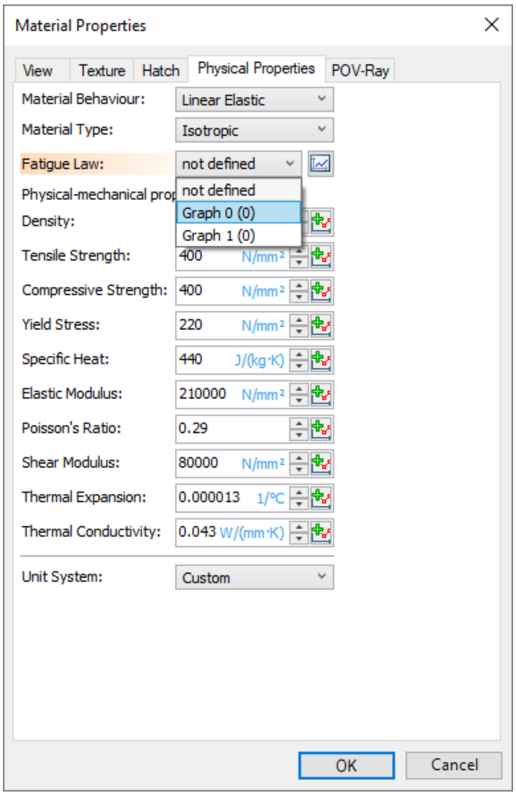
To solve Static Analysis problem it is required to:

- **Create finite-element mesh;**

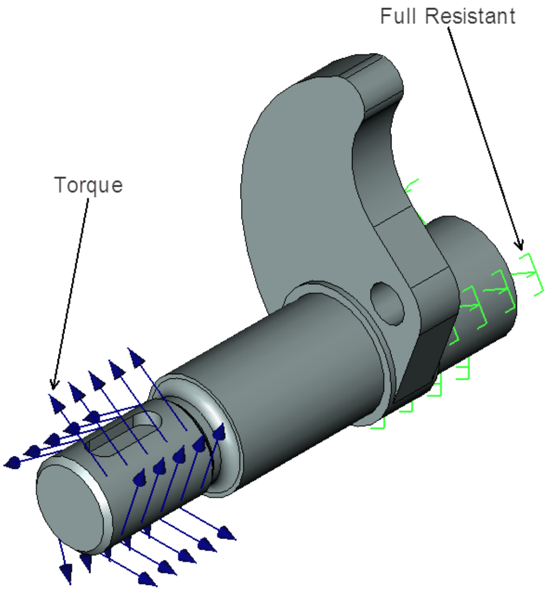


- **Specify material of the study;**

> Later on when executing the fatigue strength analysis, it will be required to define the S-N curve for the material of the problem. That is why the S-N curve for the material must be specified.

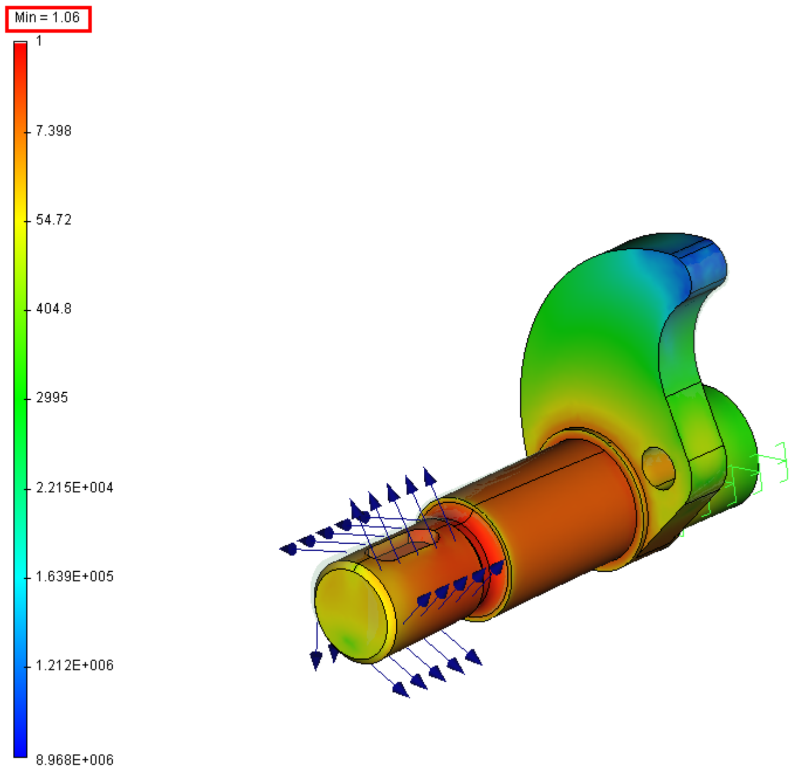


- **Apply loads and specify constraints;**



Magnitude of «Torque» is 50 N-m

After performing the analysis, the obtained results must be analyzed. In particular we are interested in the safety coefficient. Let us consider, for example, **Safety coefficient by equivalent stresses**.



The minimum value of the safety coefficient is greater than 1, hence the workpiece will not be damaged for the static load. In the case it makes sense to consider the action of the cyclically varying loads applied to this workpiece.

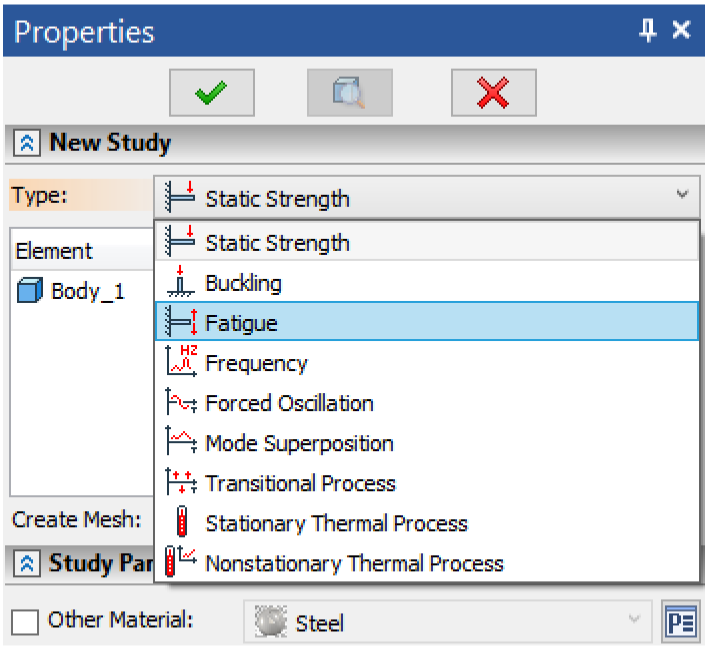
Fatigue analysis

Now after we have completed the workpiece static strength analysis and confirmed that the workpiece will not experience damage for a one-time loading, we can proceed to the fatigue strength analysis.

Let the workpiece be now subjected to the cyclically changing load **Torque**. In what follows the loading cycle will be specified as symmetric *(R= 1)*. The cyclic stress amplitude will be found from the stresses obtained in the static analysis.

> It should be noted that the loading amplitude can be scaled (decreased or increased by several times) with the help of **Scale Factor Value** (see below).

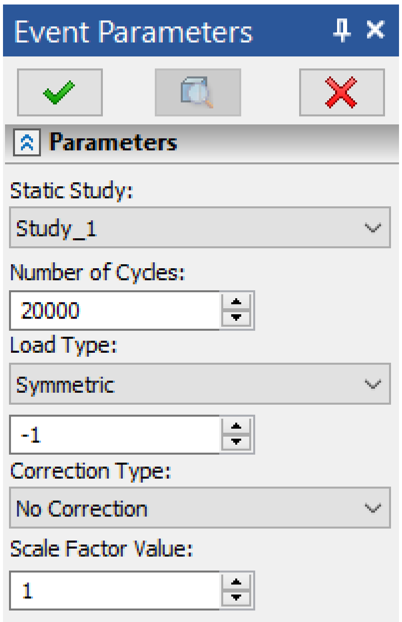
Let us create the Fatigue analysis study (**Analysis** > **New Study** > **FEA Study** > **Fatigue**).



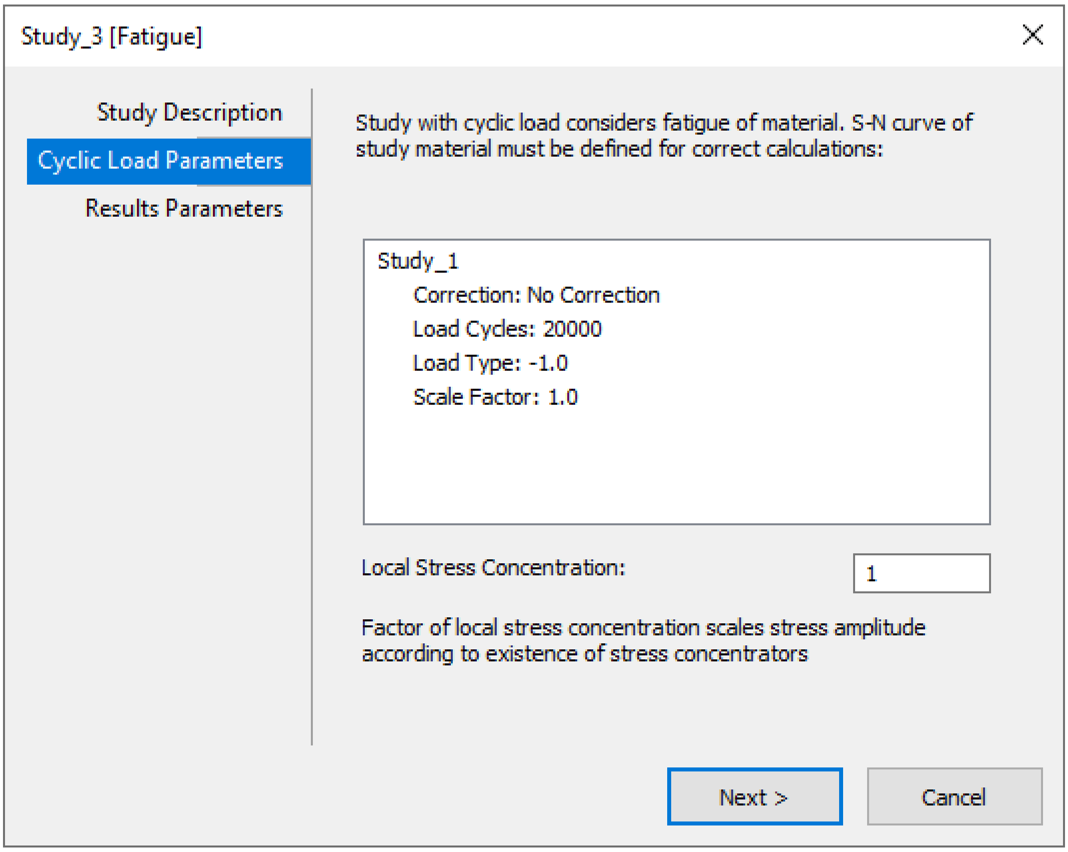
Let us create the event **Analysis** > **Event**.

On the **Event Parameters** tab we select the completed static problem and specify parameters of the cyclic loading: number of cycles, load type, correction type (specified in case if the cycle asymmetry coefficient R does not coincide with the coefficient of the given S-N curve), scale factor value (stress scaling coefficient).

Also on the **Event Parameters** tab the local stress intensity factor can be specified (by default it is equal to 1).



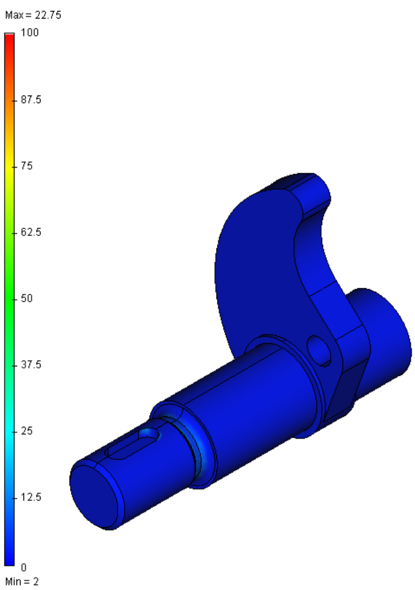
Before starting the analysis of the study, you can view the information about specified parameters of the fatigue study:



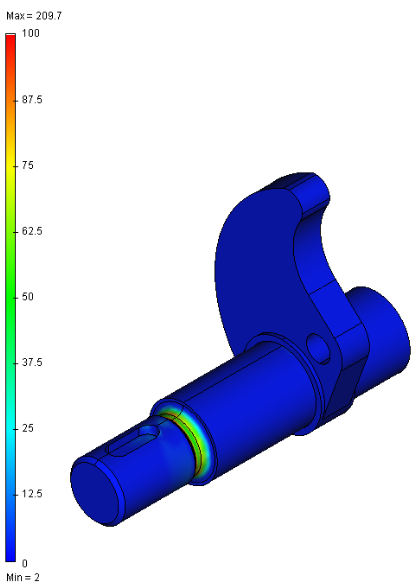
Results of Analysis

1. Damage

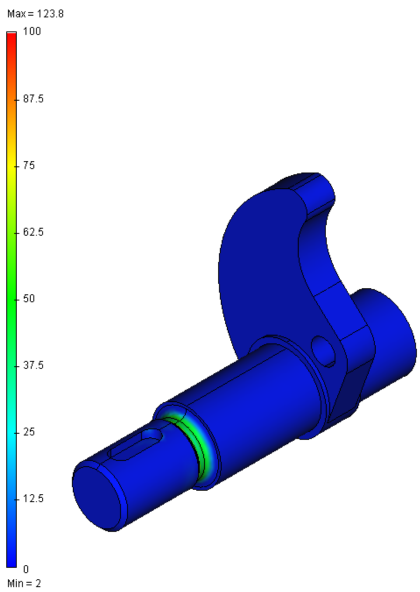
Red zones on the model correspond to the segments with the damage of 100% and signify insufficient stability with respect to the cyclic loads in these segments of the structure.



Damage by principal normal stress



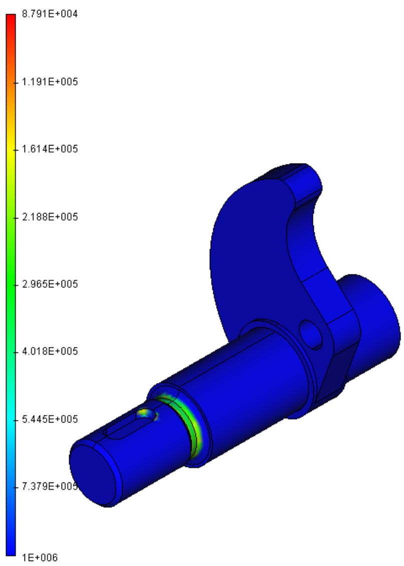
Damage by stress intensity



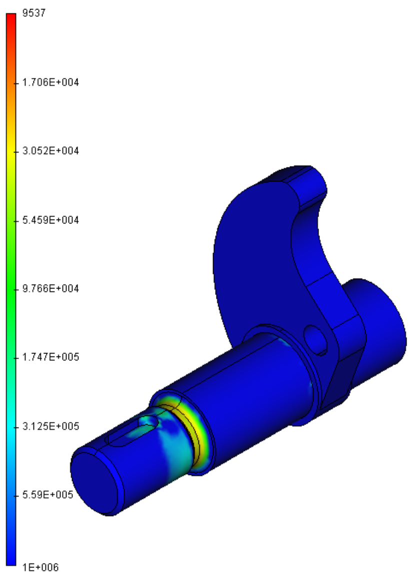
Damage by equivalent stress

2. Total life

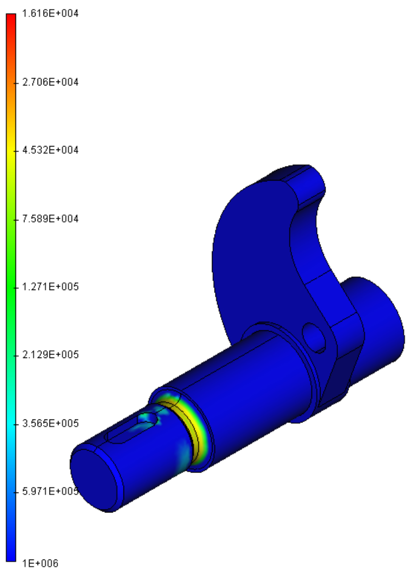
Zones shown with red color have the smallest total life and blue ones – the largest.



Total life on principal stresses



Total life on stress intensity



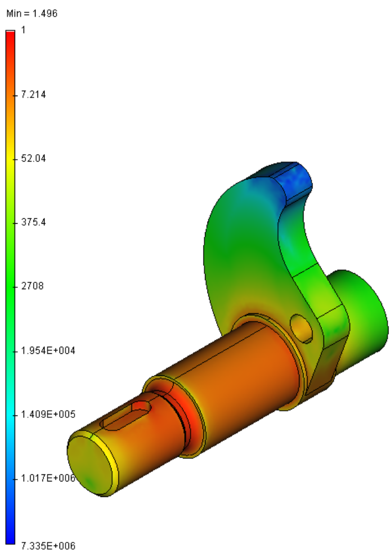
Total life on equivalent stresses

This group of results shows that before the failure the workpiece will sustain the following number of loading cycles (i.e., the number of startups and stops):

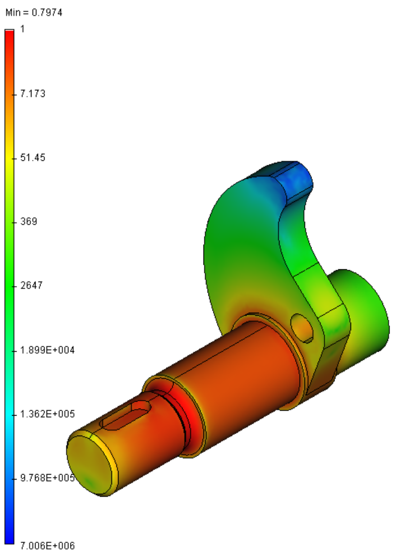
Type of the «Total life» result	Number of cycles
by principal stresses	87910
by stress intensity	9537
by equivalent stresses	16160

Therefore, if we adopt the most frequently used criterion (by equivalent stresses), we can make a conclusion about insufficient reliability of the workpiece, based on the specified number of the loading cycles (20000 cycles).

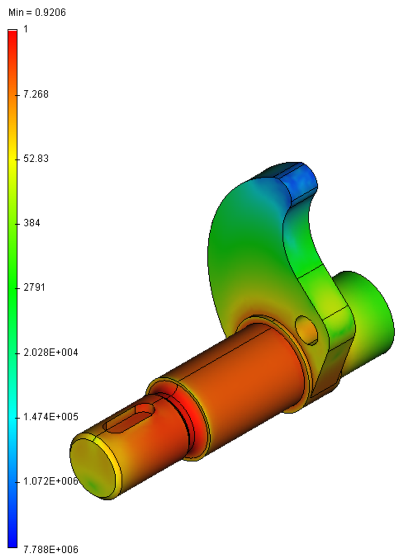
3. The safety factor shows us the fatigue strength safety coefficient for the specified cyclic loading (20000 cycles) and also it tells us about possible issues with the fatigue strength of the given workpiece, minimum by two criteria.



Safety factor on principal stresses

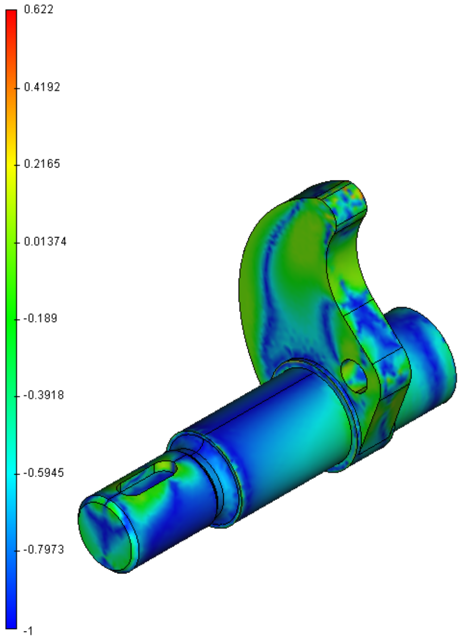


Safety factor on stress intensity



Safety factor on equivalent stresses

4. Biaxiality characterizes inequality of amplitudes of the principal stresses.

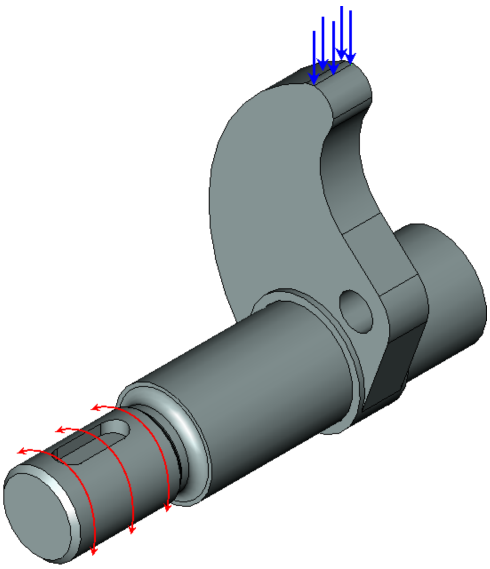


Multi-event Fatigue Analysis

To carry out the fatigue strength analysis, it is first required to examine the action of the static load on the workpiece (i.e., carry out static analysis). The stresses (equivalent or principal) obtained in the static analysis will be used in the fatigue analysis as the amplitudes of the cyclic stresses.

Now let us consider the fatigue strength analysis of the workpiece subjected to several cyclically varying loads. All loading cycles considered have different characteristics, i.e., the types of loading cycles do not coincide and the number of loading cycles can also be different.

Let the workpiece shown on the figure be subjected to the bidirectional load **Torque**, and in addition a pulsating load **Force** is applied.



By the pulsating load **Force** in the given problem we understand the maximum value of the load applied to the cam for one turn of the shaft.

Both loads (**Torque** and **Force**) are independent of each other. That is why the fatigue analysis will be multi-event.

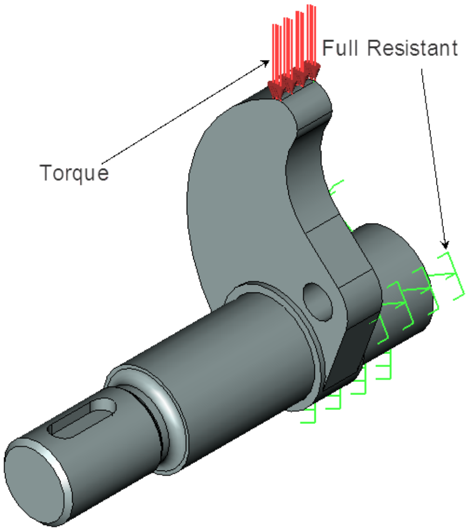
Static Analysis

Firstly we treat the specified loads as static (i.e., not changing with time) and perform the workpiece's static strength analysis individually for each of the loads. That is, we will solve the static analysis study with the **Torque** load and the static analysis study with the **Force** load.

We note that in order to carry out a multi-event fatigue analysis it is required that the finite element meshes of both static analysis studies included into the fatigue analysis be coincident and the same body of each static analysis study be made of the same material.

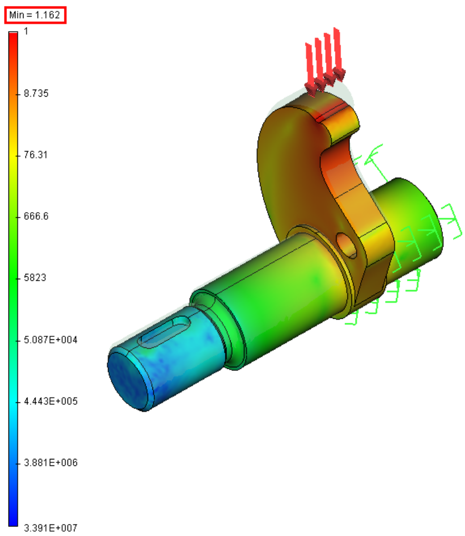
For the static analysis problem with the **Torque** load all input data remain unchanged (see above).

For static analysis study with the **Force** load, the material characteristics and restraints imposed on the workpiece did not change, but the load changed. Load's parameters are given below:



Magnitude of the **Force** load is equal to 3000 N

After the analysis has been completed, it is required to analyze the obtained results. Let us consider **Safety coefficient by equivalent stresses**.



The minimum value of the safety coefficient is greater than 1, and hence, upon static loading the workpiece will not break down. Then it is meaningful to consider the action of the cyclically varying loads applied to this workpiece.

Fatigue Analysis

Now, after we have completed the static strength analysis for the workpiece, we can proceed to the fatigue strength analysis.

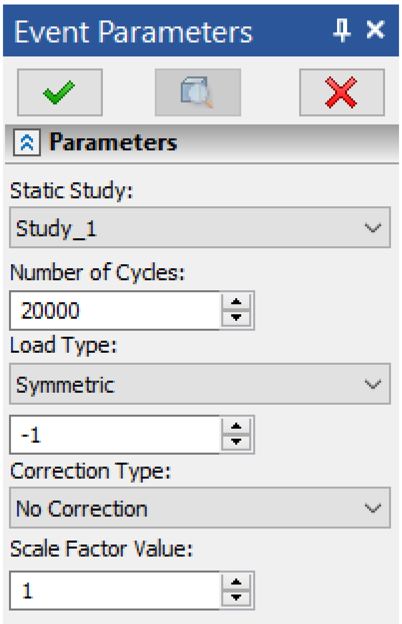
Let the workpiece be subjected to the cyclically varying loads: **Torque** and **Force**. In what follows the loading cycles will be specified as symmetric (R= 1) and pulsating (R = 0), respectively. The loading cycles' amplitudes will be determined from the stresses computed in the static analysis.

Let us create the Fatigue Analysis Problem.

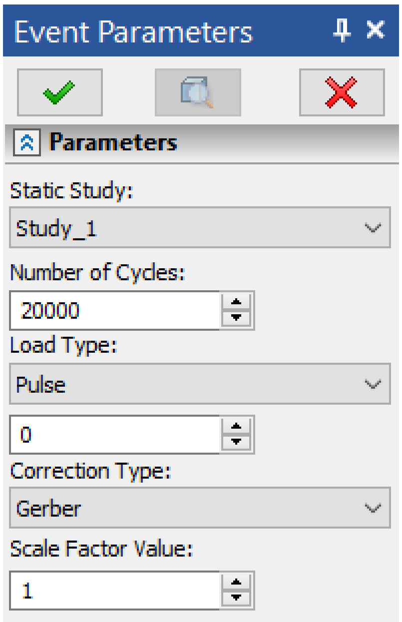
We create two events: for the **Torque** load and for the **Force** load.

For the **Torque** loading, we specify the same parameters as for the single-event analysis.

For the **Force** loading, on the **Event Parameters** tab we select the completed static analysis problem with the corresponding load and specify parameters of the cyclic loading: number of cycles, loading type, stress adjustment method, scale effect coefficient (stress scaling coefficient).



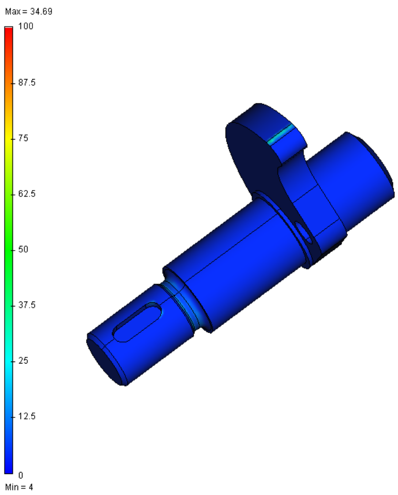
Torque



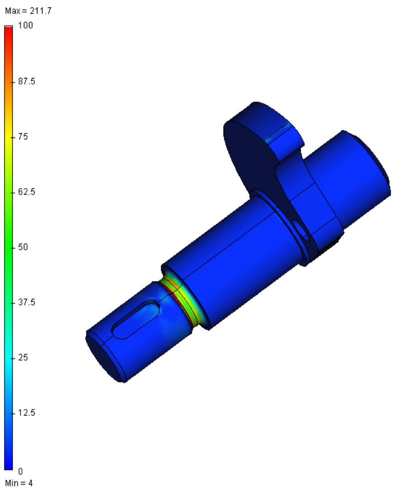
Force

Analysis results

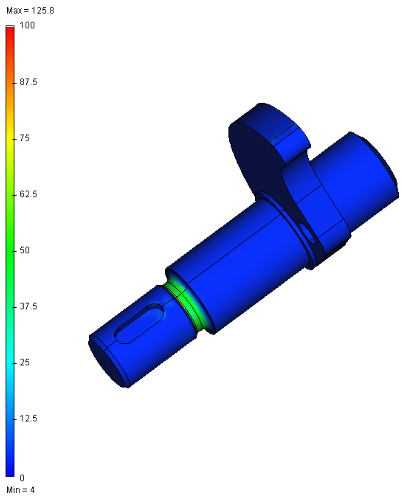
For multi-event fatigue analysis we obtain only one group of the analysis results: **Damage Resource**. Red-colored zones on the model correspond to the segments with damage of 100% and signify insufficient stability to the action of cyclic loadings in these parts of the structure.



Damage by principal normal stress



Damage by stress intensity



Damage by equivalent stress

Dynamic Analysis

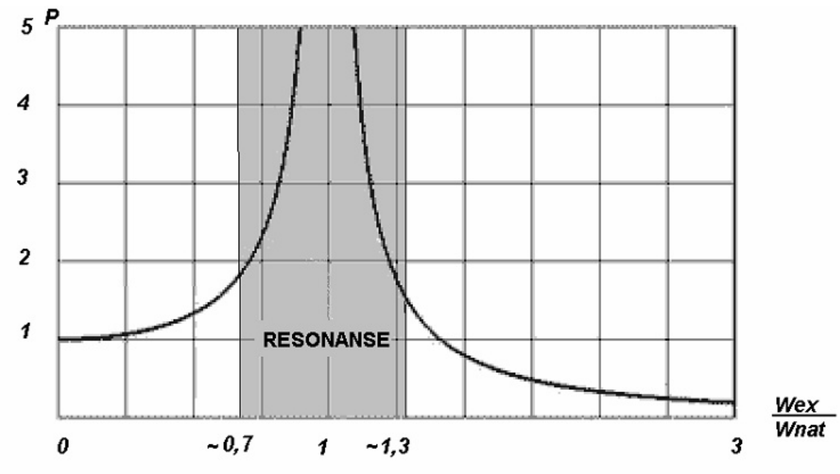
Frequency Analysis

The frequency analysis module serves for calculating natural frequencies (resonant frequencies) of a structure's vibrations and the respective vibration patterns. The study of calculating natural frequencies and the respective vibration patterns arises in many practical cases of analyzing a structure's dynamical response under varying loads. A most widespread situation is when it is necessary to assure at the design stage a low possibility of the mechanical phenomenon of the resonance under operating conditions. As is known, the essence of the resonance is a significant increase in the magnitude of induced vibrations (by dozens of times and even more) at certain frequencies of an external disturbance – the so-called resonant frequencies. In most cases, the occurrence of resonance is an unwanted phenomenon from a product's safety viewpoint. Probing a structure's natural properties against the possibility of a resonance in the operating range of external exciting frequencies at the design stage helps introducing changes in the structure that can alter the natural frequencies spectrum. This could help avoid or significantly lower the possibility of resonance during operation. Thus, the vibro-stability condition with respect to the natural frequencies criterion can be formulated as follows:

> A structure's natural frequencies must fall outside the external exciting frequency range: $f_i \notin [0.7f_{min}^{ex}; 1.3f_{max}^{ex}]$.

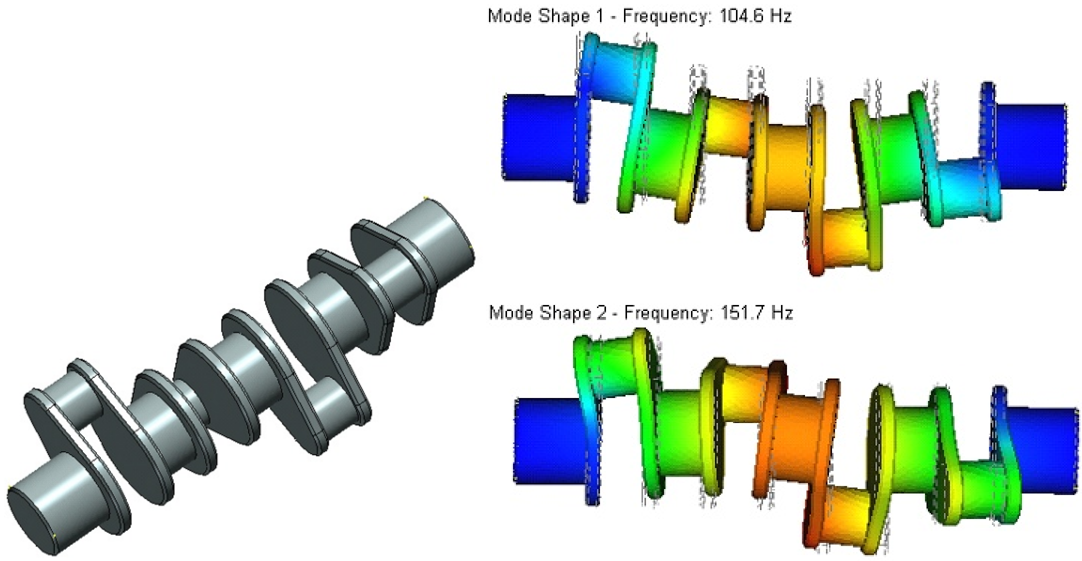
where f_i- the i-th natural frequency of the structure. Usually, the greatest danger is presented by resonance at lower natural frequencies *($i \leq \sim 5$)*, since that is where the most mechanical energy is concentrated;

f_{min}^{ex}, f_{max}^{ex} – the lowest and the highest frequencies of the known range of external exciting vibration.



Variation in the amplitude magnification factor with respect to the natural frequency to the external exciting frequency ratio in a system with an insufficient damping

By having evaluated natural frequencies of a structure's vibrations at the design stage, you can optimize the structure with the goal of meeting the frequency vibro-stability condition. To increase natural frequencies, you would need to add rigidity to the structure and (or) reduce its weight. For example, in the case of a slender object, the rigidity can be increased by reducing the length and increasing the thickness of the object. To reduce a part's natural frequency, you should, on the contrary, increase the weight or reduce the object's rigidity.



Thus, by calculating resonant frequencies at the design stage using the frequency analysis module and optimizing the part's mass-rigidity properties, the user can raise reliability of the structure being developed from the viewpoint of its vibro-stability and vibrational strength.

Topics in this section:

Details of Frequency Analysis Steps

Frequency analysis is performed in several stages. The sequence of the user's steps for putting together a study and running a structure's frequency calculation is in many parts similar to the algorithm described for the Static Analysis. Therefore, we will point out in this chapter only certain details specific to stability calculations:

Step 1. Creating "Study"

When creating a study, specify the study type - **Frequency Analysis** in the command's parameters window.

Step 2. Applying boundary conditions

In a frequency analysis study, the boundary conditions are solely defined by restraints. Defining restraints is a necessary prerequisite for performing a correct frequency calculation. The combined restraints on a body's motion must satisfy the following condition:

> To be subjected to frequency analysis, a model must be restrained so as to exclude its free motion in the space as a solid body. Failing to meet this condition will cause incorrect results of Finite Element modeling or abortion of computations.

Step 3. Solving

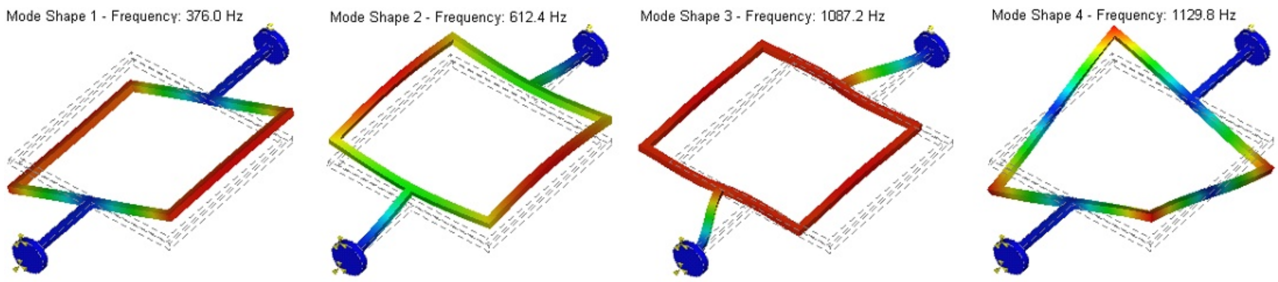
Before running calculations, the user should specify the number of natural frequencies and, if necessary, elaborate on the solution algorithm.

Step 4. Analysis of frequency calculation results

The results of a frequency analysis are:

Natural vibration frequency (Hz) – corresponds to the expected resonant frequency of the structure. In theory, the number of natural frequencies is unlimited for any body. The results reflect only the frequencies for selected modes of natural vibration.

Natural vibration mode with respect to a given frequency. Let us illustrate the physical meaning of the "vibration mode" term. A vibration mode shows what will be the relative deformations (displacements) in a structure in the case of resonance at the respective natural frequency. Please take a special notice on that the vibration modes displayed in the Postprocessor window after completing calculations are relative amplitudes of vibration. By analysing those modes, one can make conclusions about the pattern of resonant displacements, but not about their factual amplitude. By knowing the expected vibration mode at a certain natural frequency, you can introduce an additional restraint or support at the part of the structure corresponding to the maximum vibration in this mode, which would effectively manage the part's natural properties.
By default, the vibration modes are displayed in the Postprocessor window without color-coding; the latter can be enabled in the visualization parameters.



Consider also the convenience of animation for analyzing the pattern of the structure's motion at a certain frequency. Remember to enable the **Animation** option in the parameters of the calculation results window (accessible by  in the **Postprocessor window**) in order to have an animation, and specify the desired animation parameters.

Frequency Analysis Processor Settings

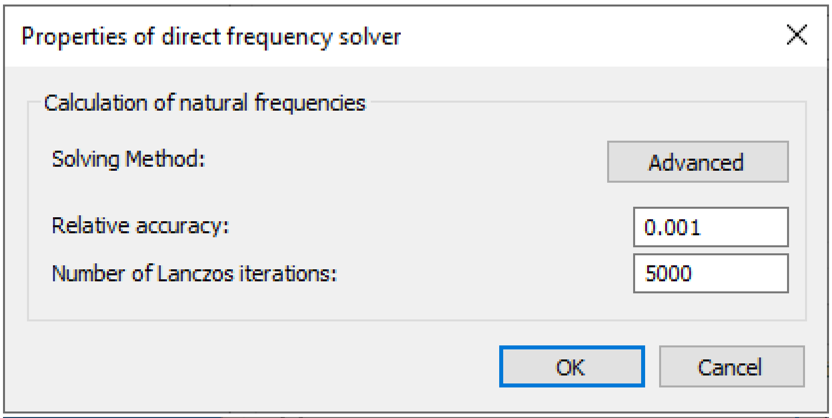
"Study Description" and "Natural Frequencies Analysis" Tabs

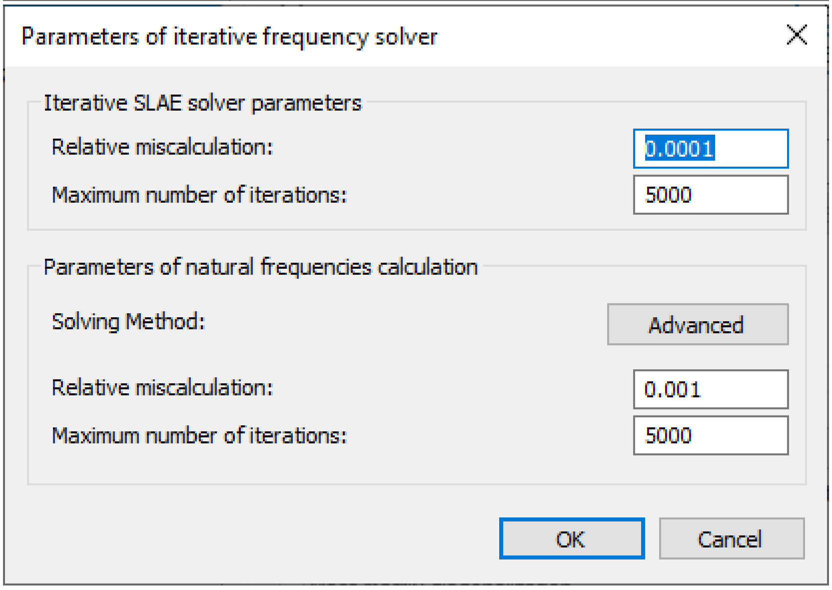
On the **Study Description** tab, you can define or edit the descriptive attributes of the current study, as the name or a comment.
The **Natural Frequencies Analysis** tab serves for defining processor properties for solving equations.

"Solving Method" group

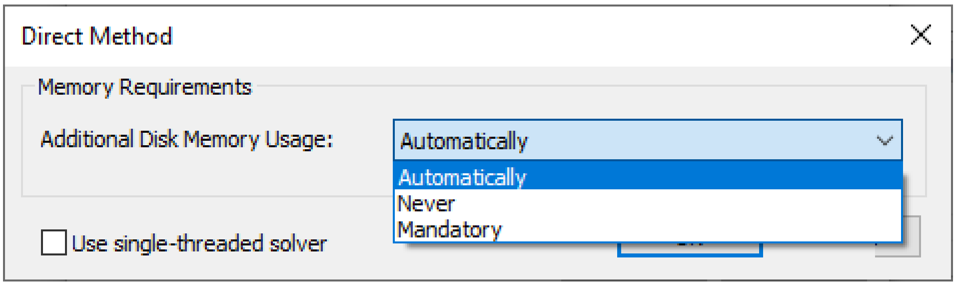
Automatic assignment – equations solution method is automatically selected by the system based on the total number of equations. By default, the threshold number is equal to 100 000 equations (degrees of freedom) and is set on the **Settings** > **Processor** page. If the total number of equations exceeds this value, an iterative method is used for solving equations; otherwise the system of equations is solved by direct methods.
Direct – the system of equations is solved by Gauss method, which takes a lot of RAM as compared to the iterative method. If sufficiently large amount of RAM is available, then the direct method renders the solution quicker than the iterative method. The direct method is preferred for relatively small problems with the number of degrees of freedom smaller than 100 000, however this number can increase depending on the amount of available RAM. This method is also preferred for linear finite elements. After pressing the **Settings** button, the window of properties of the direct frequency solver appears in which the user can select the relative tolerance for finding critical loads and the number of Lanczos iterations (see below).
Iterative – the system of equations is solved by iterative methods, which do not require complete inversion of the matrix, which takes lesser RAM. Calculation time is approximately proportional to the number of natural vibration modes sought. If the finite element mesh contains a lot of elements that do not have the optimal shape, e.g., stretched, then the convergence rate is significantly reduced. After

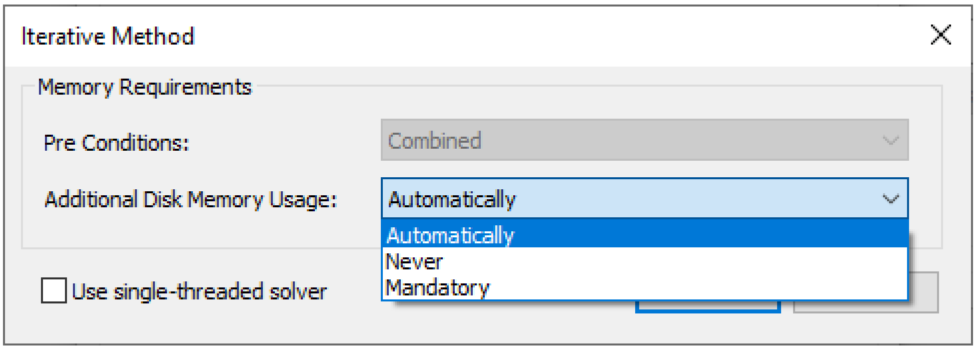
pressing the (**Settings**) button, the window of properties of iterative stability/frequency solver appears, in which the user can select the relative tolerance for solving the system of linear equations (residue) and the maximum number of iterations in the group of settings for linear equations iterative solver; the relative tolerance for finding the natural frequencies and the maximum number of iterations for determining the eigenvalues can be specified in the group of settings for eigenvalues (natural frequencies) determination (see below).





When you press **Advanced** button, it is available to indicate possibility of using additional disk memory: **Aulomatically, Never, Mandatory**.





"Spectrum of natural frequencies" group

Specify parameter **Number of frequencies**, that is, the user can define the number of lower natural frequencies of the structure to be evaluated;

Specify the **natural frequencies range** if the frequencies are sought within the specific range of the spectrum; it is required to specify the upper and lower limits of the range (the larger number of natural frequencies belong to this range the more accurate the evaluations will be)

If the **Ignore Mechanical Loads** option is enabled, the forces will not be taken into account; otherwise the natural frequencies will be sought for the stressed state of the structure

Enable the option **Non Stationary System** in case if the restraints applied to the model are not sufficient to eliminate the model's rigid body motion in space.

The **Don't calculate natural modes** option increases calculation and saves memory.

"Finite-Element Method" Group

In the group **Finite-Element Method** the user can set the **Linear Interpolation** option, if interested in qualitative results only, that is, when the one is only interested in relative assessment of vibration amplitude patterns.

> Please note that a linear element solution provides insufficient accuracy of determining the numerical values of natural frequencies. The frequency values achieved in a linear finite element calculation could be much greater than the values achieved by using more accurate methods. You are recommended to use quadratic element calculations (the default mode) for quantitative evaluation of natural frequencies.

Mass matrix diagonalization mode enables the user to decrease the amount of memory required to solve the system of linear algebraic equations. At the same time, the accuracy of obtained results becomes slightly worse.

"Thermoeffects" Tab

Parameters of influence of the temperature changes on the materials properties are specified on the **Thermoeffects** tab.

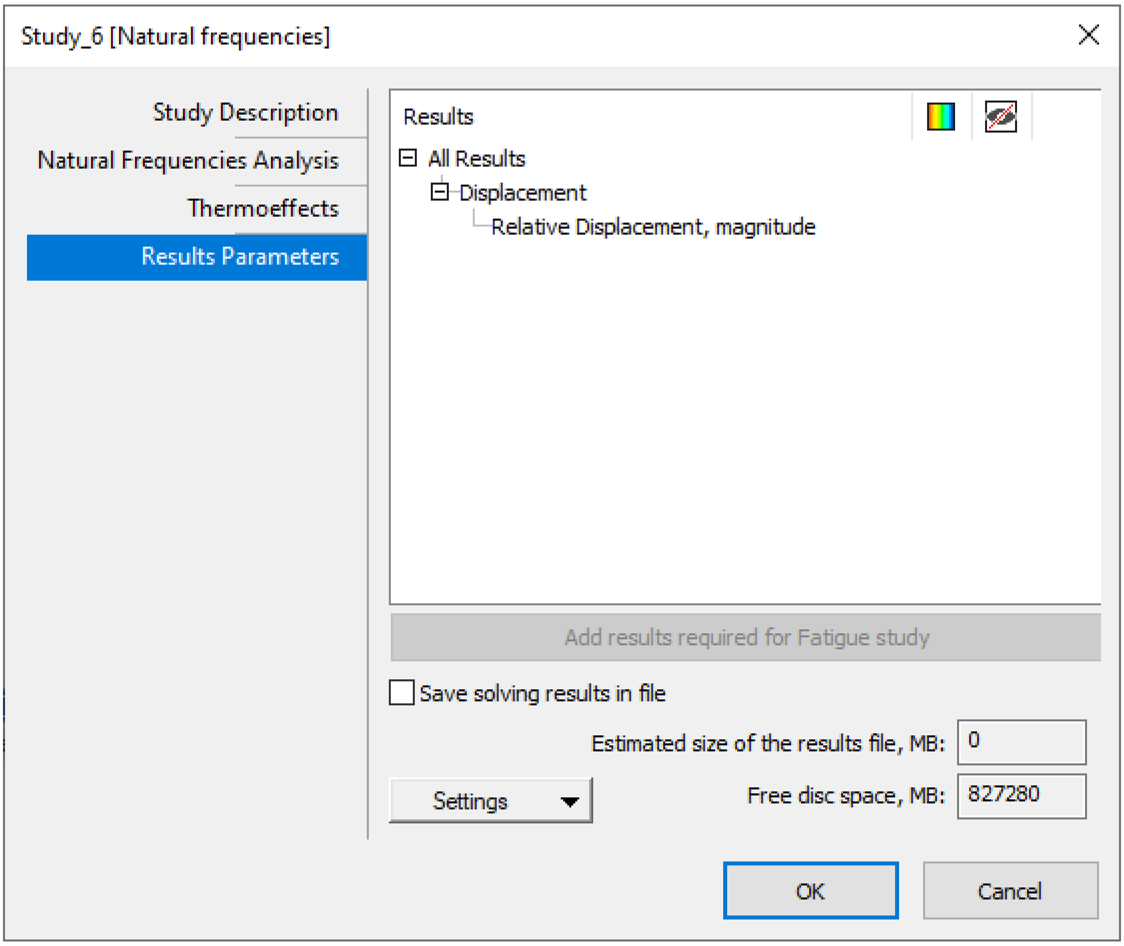
Consider thermoelasticity. If the flag is enabled the influence of heat stress to the rigidity of the construction and changes of natural frequencies value will be considered. Otherwise, the condition of heat stress will not be considered.

If the option **Consider dependence of physical properties from temperature** is enabled, the material properties will be taken from the graphs showing dependences of materials properties to the temperature. The properties are set in the parameters for each material.

Both options can be considered together or separately. Other parameters on the tab work in the same way as in another static analysis studies.

"Results Parameters" Tab

The **Results Parameters** tab defines the result types displayable in the studies tree after finishing calculations. In the frequency analysis, the user can assess only relative displacements, either their absolute value or a value in the direction of the respective axes of the global coordinate system.



Forced Oscillation

Forced oscillation analysis is carried out for prediction of response of the structure subjected to harmonically varying external loads. External loads include kinetic and/or kinematic excitation. In addition, the damping of the system can be taken into account.

The goal of the forced oscillation analysis is to find the dependence of the system's response on the frequency of the driving loads. The results of the analysis include the amplitudes of displacements, oscillation accelerations and oscillation overload for the given driving frequency. From the results of the analysis for a range of frequencies we can obtain dependence of the amplitudes and oscillation accelerations on the frequency of the driving loads, which is important when estimating vibrostability of the system in the given frequency range.

Introductory Information

The «Forced oscillations » module of the system of finite element modeling T-FLEX Analysis can be used for analysis of steady-state forced oscillation of the following types:

- Forced oscillation of the system without damping subjected to harmonic driving load. These oscillations for the system with many degrees of freedom are described by the following system of linear differential equations:

$$[M]\{U\} + [K][U] = [\{F_0 \cos(\omega t + \varphi)\}],$$

where:

M – symmetric square mass matrix,

K – symmetric square stiffness matrix of the system,

F_0 – vector of amplitudes of the driving load,

ω – frequency of the driving load,

U, $\ddot{U}$ – vectors of coordinates of points of the system which change their location with time t and their accelerations,

φ – initial phase of the actuator.

- Forced oscillation of the system with damping subjected to harmonic driving load. These oscillations are described by the following system of linear differential equations:

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K][U] = [\{F_0 \cos(\omega t + \varphi)\}],$$

where:

C – symmetric square damping matrix,

$\dot{U}$ – velocity vector for the points of the system. Rayleigh damping is adopted in the system, i.e., it is proportional and defined by the expression $[C] = a[M] + b[K]$, where a – mass proportionality coefficient; b – stiffness proportionality coefficient (both coefficients are scalars).

- Forced oscillation of the system of two indicated types that are caused by harmonic motion of foundations, i.e., initiation of oscillational motion of one or several supports of the system. Differential equations that describe this type of oscillations are similar to those given above and are different in that the harmonic driving load in the right-hand side of the equations is evaluated by the formula:

$$(k/m)x_{och}\cos(\omega t + \varphi).$$

> Several driving loads and/or supports' displacements can be applied to the system but their frequencies must be the same.

Rotation of the shaft or spindle, having imbalance, located on elastic supports can serve as an example of the harmonic driving force. Kinematic excitation is used in those cases when the magnitudes of the driving forces are not known but the amplitudes of oscillations of certain elements of the structure are known.

When analyzing forced oscillation it is important to take into account the influence of **damping forces**. **Damping** is called a process of dissipation of the energy of mechanical oscillations resulting in a gradual decay of system's oscillations once initiated. Damping forces can have different origin: friction between dry surfaces with sliding, friction between lubricated surfaces, internal friction, air or water resistance, etc. Usually it is assumed that the damping force is proportional to the velocity (viscous damping). Resistance forces that change in an arbitrary way are replaced by equivalent damping forces based on the condition that during one cycle they dissipate the same amount of energy as the real forces. Equation of forced oscillation with damping for i-th mass, which is the solution of the differential equations described above, can be written as:

$$u_i = e^{-nt}(C_1 \cos \omega_\delta t + C_1 \sin \omega_\delta t) + C_3 \cos \omega_\beta t + C_4 \sin \omega_\beta t;$$

where

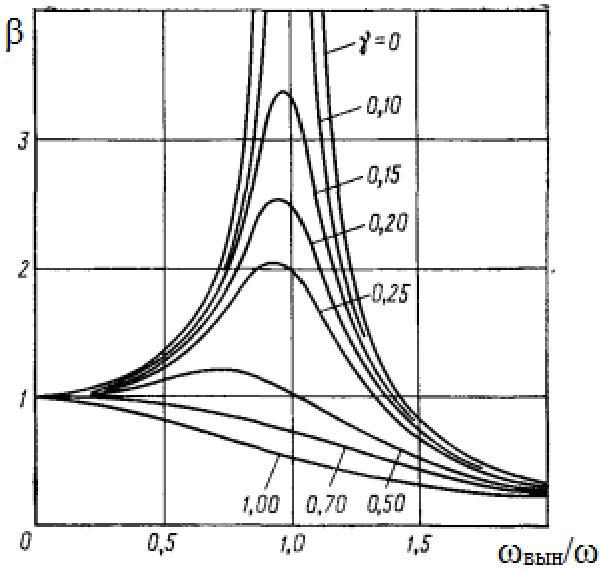
ω_δ – circular damped frequency;

ω_β – circular frequency of the driving force;

$2n=c_i/m_i$,

where c_i – damping coefficient for i-th mode, m_i – mass.

The expressions contains two terms: the term in the brackets describes the damped free oscillations occurring with the damped frequency, which is only slightly different from the undamped natural frequency; the remaining part represents undamped forced oscillation with the frequency of the driving load ω^{β}.



To explain the influence of damping, let us consider a plot on which is shown the dependence of the amplitude amplification factor $\beta = (1 - \omega^{\delta 2}/\omega^{2} + (2\gamma\omega^{\delta}/\omega)^{2})^{-1/2}$ on the ratio of frequencies of the forced and free oscillations $\omega_{вын}/\omega$ for different values of the damping coefficient $\gamma = n/\omega^{\delta} = c/c_k^{\delta}$, where c_k^{δ} – critical coefficient of viscous damping, for which oscillations do not occur and the motion of the system monotonically decays. From the figure we can see that when the frequency of the forced oscillation is small compared to the natural frequency of free oscillations, translation of points of the system is approximately equal to the translation that occurs for the statically applied driving force. When the driving harmonic force has high frequency, then regardless of the damping coefficient the applied force does not cause forced oscillation of the system that has low natural frequency. In both cases $\omega^{\delta} << \omega$ and $\omega^{\delta} >> \omega$ damping does not have any effect on the forced oscillations, but when the ratio of the mentioned frequencies is close to one then damping has a significant effect on the amplification factor. If the damping coefficient is small, the largest influence of damping is observed closer to resonance frequencies, which is very important to take into account when analyzing the structure. For analyzing forced oscillation of the structure in proximity to natural frequencies the import of the natural frequencies values from the results of the frequency analysis is provided (see below).

The damping coefficient γ for j-th mode is related to the proportionality coefficients a, b by the expression $\gamma_j = (a + b\omega_j^2)/2\omega_j$. The magnitude of the damping coefficient γ takes the values from 0,01 for weakly damped systems (all-steel parts); 0,02 – 0,04 (steel structures with non-detachable joints deformed below the yield strength); 0,03 – 0,07 (steel structures with dismountable joints); 0,05 for rubber; up to 0,15 for strongly damped systems.

If damping coefficients for i-th and j-th modes are known, the proportionality coefficients are calculated from the formula:

$$a = 2\omega_i\omega_j(\gamma_j\omega_i - \gamma_i\omega_j)/(\omega_i^2 - \omega_j^2) \text{ and } b = 2(\gamma_i\omega_i - \gamma_j\omega_j)/(\omega_i^2 - \omega_j^2).$$

If the coefficient a is equal to zero then this damping is called relative, and the damping coefficient for j-th mode is proportional to the circular frequency of this mode without damping. Therefore, oscillations that correspond to the highest modes will decay faster. If the coefficient *b* is equal to zero then this damping is called absolute and the damping coefficient by *j*-th mode is inversely proportional to the circular frequency of this mode without damping. Therefore, oscillations that correspond to the lowest modes will decay faster.

The results of the analysis in the forced oscillation module include the following quantities:

- Displacements amplitudes in nodes of the finite-element mesh U_m.
- Oscillation acceleration in nodes of the finite element mesh which are expressed in terms of the amplitudes U_m as $U_m = U_m \omega_{\text{вын}}^2$.
- Oscillation overloads defined as the ratio of the oscillation acceleration to the gravitational acceleration U_m/g.

Topics in this section:

Special Features of Forced Oscillation Analysis Stages

The forced oscillation analysis is carried out in several stages. The sequence of user's actions for study preparation and execution of this type of analysis is in many ways similar to the algorithm described for Static Analysis. That is why in this Section we describe only certain aspects which are characteristic to the forced oscillation analysis:

Step 1. Creation of «Study»

When creating a study we need to specify its type – **Forced Oscillations** in the command's parameters window.

Step 2. Imposing boundary conditions

In the forced oscillation analysis, as in the static analysis, constraints and loads play the role of the boundary conditions. In the given type of analysis all types of constraints and all types of loads can be used. Specifying constraints and loads is a mandatory condition to properly execute the analysis. Cumulatively imposed constraints on the displacements of the body must satisfy the following condition:

> For execution of the static analysis the model must have a constraint that excludes its free displacement in space as a rigid body. If this condition is not satisfied this will lead to incorrect results of the finite element modeling or interruption in the calculation process. In addition, the kinematic load «oscillator» can replace partial or full constraints.

Step 3. Analysis execution

Before execution of the analysis the user specifies, in the parameters of the study, the values for frequencies of the driving loads, for which the forced oscillation analysis will be performed, and also the magnitude of damping.

Step 4. Getting and analyzing results

The forced oscillation analysis results include: amplitudes of displacements, oscillation accelerations or oscillation overloads, phases of oscillations. There is a capability of viewing the deformed state of the structure in various phases.

Forced Oscillation Analysis Preprocessor Settings

Formulation of study

Specifying constraints. The model can be restricted both by partial and full constraints. If partial constraints are specified it is not permitted to specify displacements different from zero.

Specifying forces. To specify the amplitudes of the forces, the following types of loads can be applied to the faces, edges or vertices of the model (acceleration can also be applied to the entire body):

- Force,
- Pressure,
- Acceleration,
- Cylindrical loading,
- Moment.

Several loadings can be applied to the system of bodies simultaneously, but all of them must have equal frequency. The initial phase of oscillations can be specified for all admissible load types.

Specifying kinematic loads (oscillator). To specify the oscillation amplitude of the foundation, it is required to apply, to the elements of the model, the loading “oscillator” which can be used instead of full or partial constraint. This loading is applied to the faces, edges, vertices of the bodies, and can also be applied to individual bodies of the assembly model. For specifying direction of oscillations, the LCS is selected in the command and the direction itself is indicated by the checkmark that corresponds to the axis of LCS.

The type of kinematic loading is selected from the drop-down list and can be the following:

- Amplitudes of displacements of points,
- Velocity,
- Acceleration,
- Overload (g).

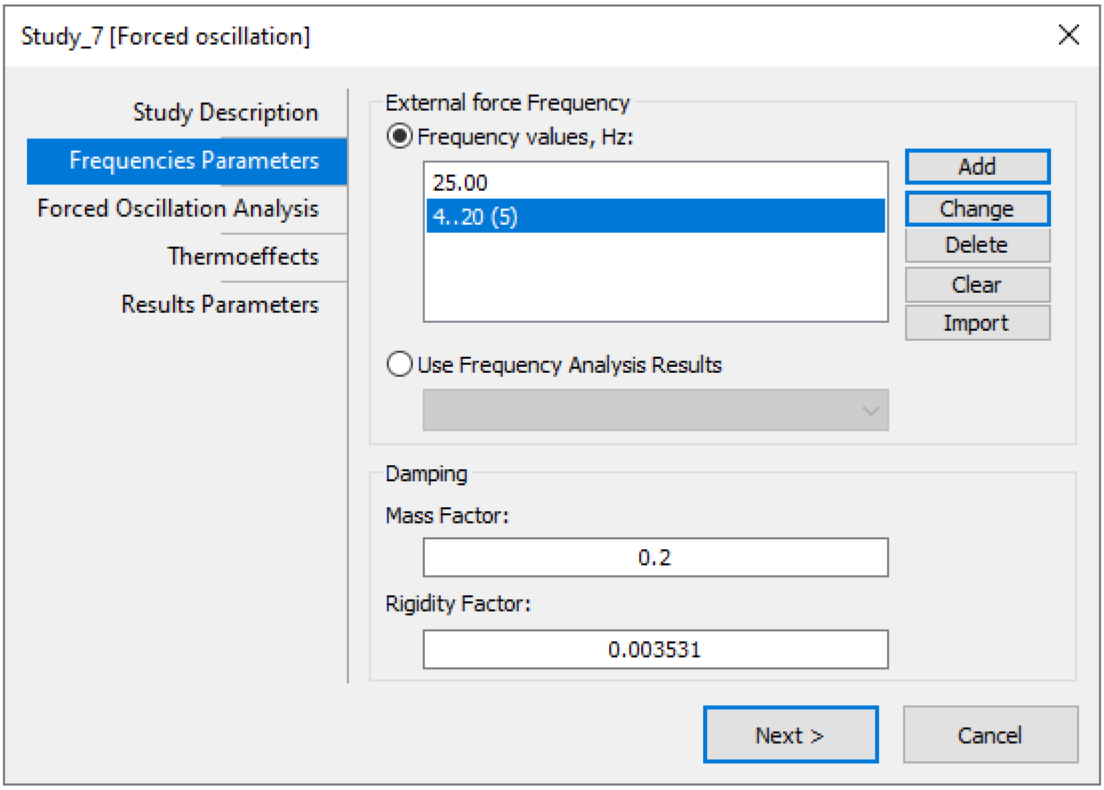
The phase shift measured in degrees or radians can be specified in a separate field.

When the loading “oscillator” is combined with partial constraints for the same element of the model, directions of the oscillations and constraints must be different.

Forced oscillation analysis processor settings

On the **Study Description** tab we can define or change the descriptive parameters of the current problem: name, problem type, comments.

On the **Frequencies Parameters** tab we specify the main settings for forced oscillation analysis.

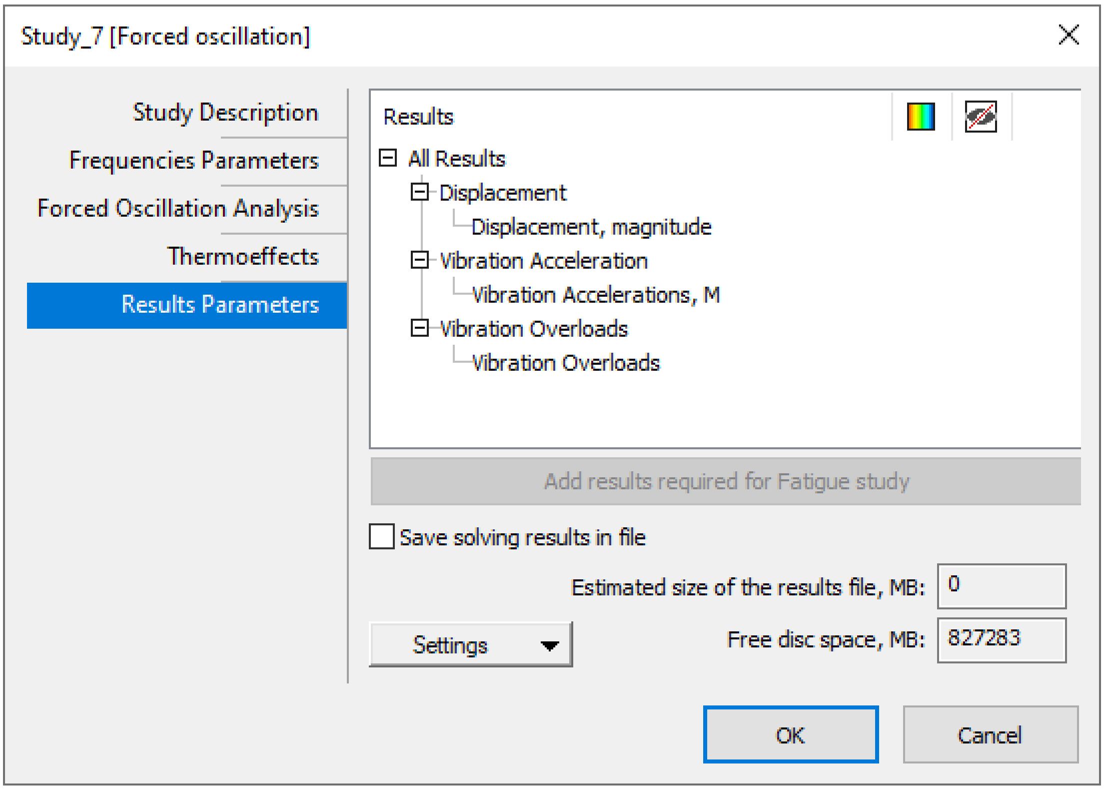


In the group of parameters **External force Frequency** we specify the values for the frequencies of the external loads. New values of frequencies can be added in several ways.
The **Add** button allows us to add both a single value of frequency and a range of frequencies, in which the initial value, final value and the increment are specified.
The **Import** button invokes the dialog window in which the values of the resonance frequencies can be imported from the results of the frequency analysis completed earlier.
In case when it is required to execute the forced oscillation analysis only on all resonance frequencies, which were found in the earlier performed frequency analysis, enable the control element «**Use frequency analysis results**». The associated connection with the results of the selected frequency analysis is supported here, i.e., when the results of the frequency analysis are changed, the values of the updated natural frequencies will automatically be used in the forced oscillation analysis.
The following editing operations are available for already existing list of frequency values:

- the **Change** button allows us to rewrite the value of one specific frequency,
- the **Delete** button removes the selected value of the frequency from the list,
- the **Clear** button clears the entire list of values of frequencies.

In the **Damping** group of parameters the structure's damping coefficients value is specified.

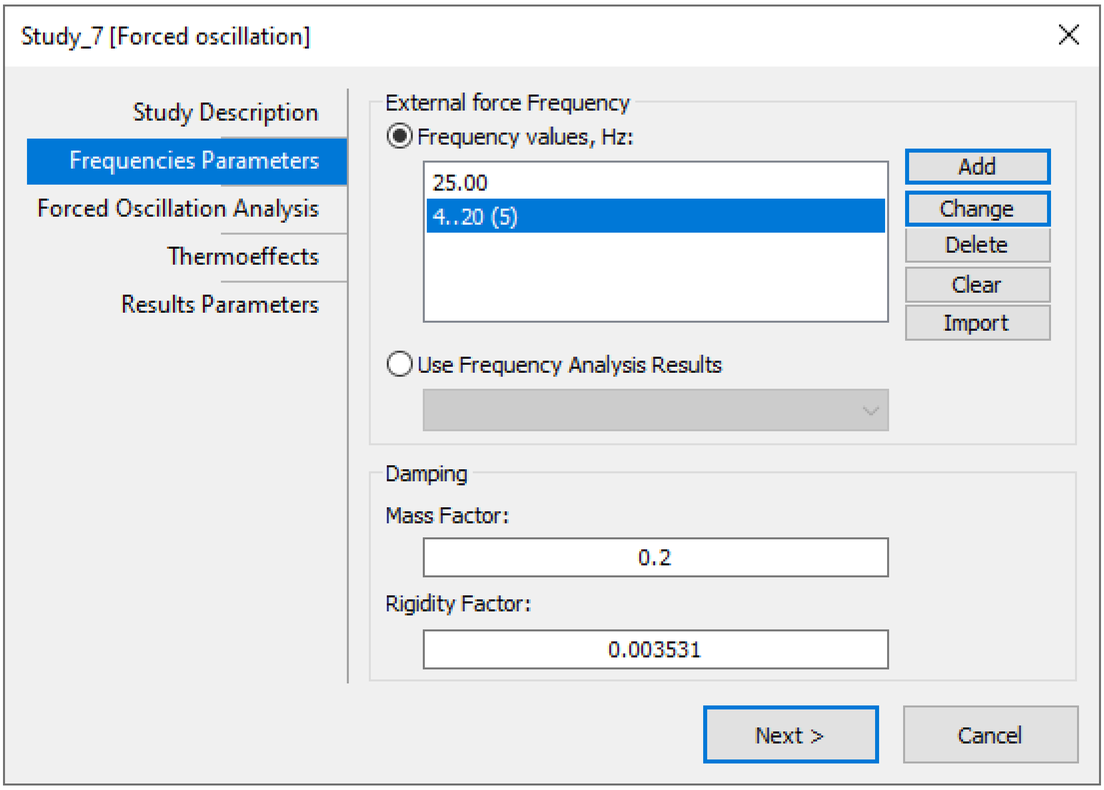
The **Forced Oscillation Analysis** tab allows us to specify properties of the processor for solution of the system of equations. Parameters that control the processor's settings are analogous to the parameters of the Static Analysis.
Parameters of temperature changes influence on the materials properties are specified on the **Thermoeffects** tab.



If the option **Consider dependence of physical properties from temperature** is enabled, the material properties will be taken from the graphs showing dependences of materials properties to the temperature. The properties are set in the parameters for each material.
Other parameters on the tab work in the same way as in another static analysis studies.
The **Results Parameters** tab allows us to define the type of results, which will be displayed in the tree of studies after completion of the analysis.

Postprocessor Settings and Forced Oscillation Results Analysis

On the **Results** tab, the following type of results grouped into 4 groups can be defined:



The «**Loads**» group includes the following results: the components and the magnitude of loads applied to the nodes of the finite element model. This type of results is considered as reference information.

The «**Displacements**» group includes:

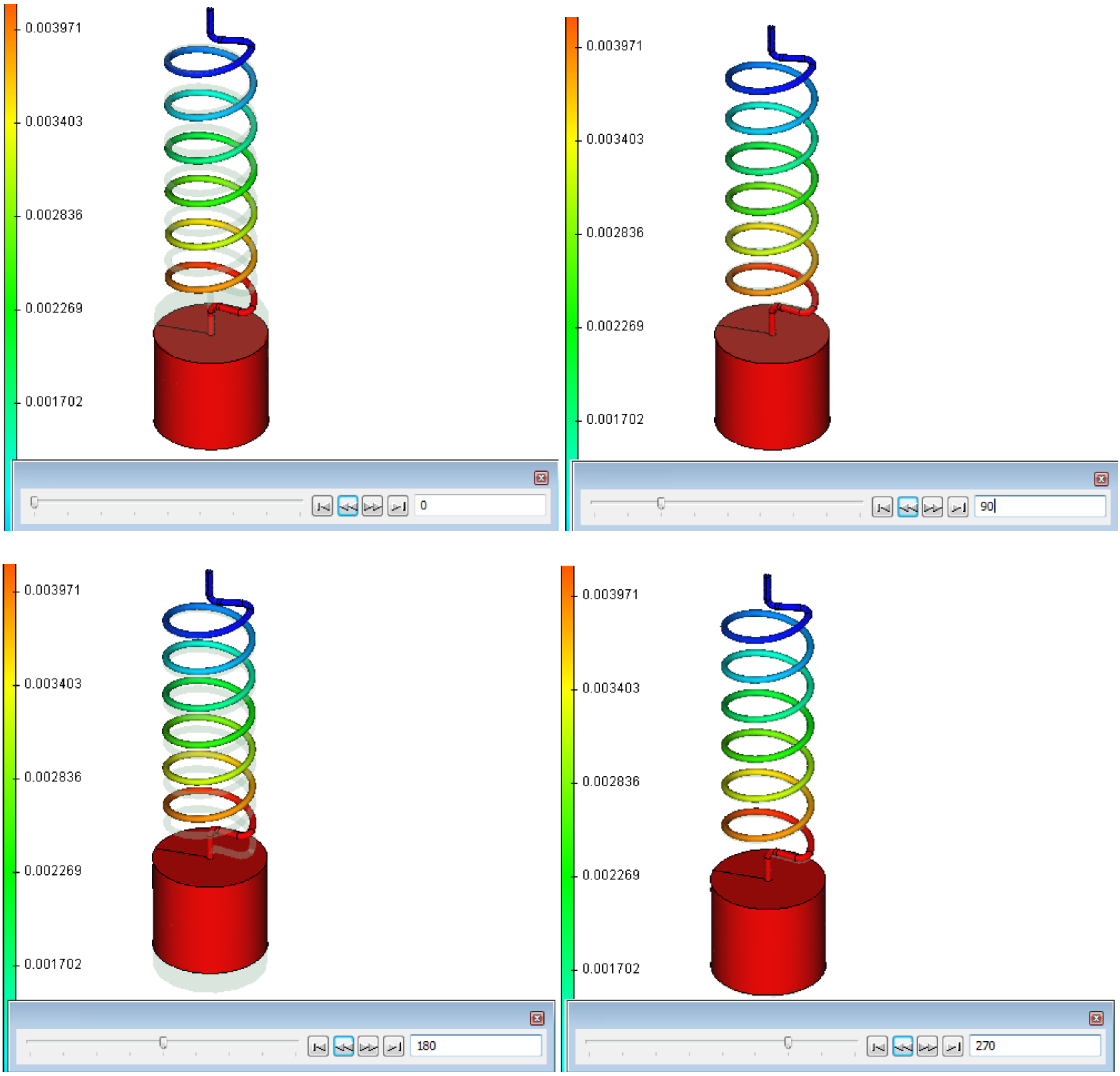
- displacements of points of the finite element model taking into account the phase shift with the respect to the actuator's phase in the direction of the axes of the global coordinate system: $U_x = (ReU_x^2 + ImU_x^2)^{1/2}$, $U_y = (ReU_y^2 + ImU_y^2)^{1/2}$, $U_z = (ReU_z^2 + ImU_z^2)^{1/2}$, and also the magnitude of the displacement $U = (U_x^2 + U_y^2 + U_z^2)^{1/2}$.
- the real part of the displacement in the direction of the axes of the global coordinate system: $\mathrm{Re}(U_x)$, $\mathrm{Re}(U_y)$, $\mathrm{Re}(U_z)$, and also the magnitude of the real part of the displacement $\mathrm{Re}U = (\mathrm{Re}U_x^2 + \mathrm{Re}U_y^2 + \mathrm{Re}U_z^2)^{1/2}$.
- the imaginary part of the displacement in the direction of the axes of the global coordinate system: $\mathrm{Im}(U_x)$, $\mathrm{Im}(U_y)$, $\mathrm{Im}(U_z)$, and also the magnitude of the imaginary part of the displacement $\mathrm{Im}U = (\mathrm{Im}U_x^2 + \mathrm{Im}U_y^2 + \mathrm{Im}U_z^2)^{1/2}$.
- amplitudes of the displacements of points of the finite element model (not accounting for the phase shift with the respect to the actuator's phase) in the direction of the axes of the global coordinate system: U_{Xm}, U_{Ym}, U_{Zm}, and also the magnitude of the amplitude $U_m = (U_{Xm}^2 + U_{Ym}^2 + U_{Zm}^2)^{1/2}$.
- Phase angles for displacements components of points of the finite element model in the direction of the axes of the global coordinate system with respect to the phase of the actuator $\varphi_{U_x} = arctg\left(\frac{\mathrm{Im}\,U_x}{\mathrm{Re}\,U_x}\right)$, $\varphi_{U_y} = arctg\left(\frac{\mathrm{Im}\,U_y}{\mathrm{Re}\,U_y}\right)$, $\varphi_{U_z} = arctg\left(\frac{\mathrm{Im}\,U_z}{\mathrm{Re}\,U_z}\right)$, and also the magnitude of the phase angle $\varphi_U = arctg\left(\frac{\mathrm{Im}\,U}{\mathrm{Re}\,U}\right)$.

In case if the phase shift is specified, individual diagrams for the real and imaginary parts of the displacements and also the absolute value of the phase at points of the finite element model can be outputted.

In the **Vibration Acceleration** group, the output of diagrams of amplitudes of oscillation accelerations for points of the finite element model $U_m = U_{m\ вын}\,\omega^2$ can be requested. The phase of vibration accelerations is different by 180° (π radians) from the phase of displacements.

In the **Vibration Overloads** group, the output of diagrams of vibration overloads U_m/g measured with respect to gravitational acceleration can be requested.

After the results window has been opened, the phase control element becomes available in the postprocessor, which allows us to trace the change in the structure's shape for different vibration process evolution phases:



The phase value can be specified by translation of the slider or by inputting the value directly into the numeric field.

To obtain ARC (amplitude resonance curve), it is required to create at least one sensor (see the Use of Sensors for Analyzing Results section) and the template of the graph containing one sensor. After that it is possible to create ARC for a point, at which the sensor is defined, by selecting (with a right mouse button) the **Display Plot** option in the context menu of the forced frequencies analysis results. The graph for the sensor selected in the **Sensors' Configurations** field will be displayed. In this way it is possible to trace the change of ARC when transitioning from one sensor to another. On the other hand, if several sensors have been created, then it is possible to display simultaneously on the same graph all curves that show the change in the result's value (for example, displacement or acceleration) on the given frequency when transitioning from one sensor to another sensor – for all frequencies. In order to accomplish this, it is required to prepare the template of the graph that contains all sensors.

Dynamic Studies

The dynamic analysis module of T-FLEX Analysis finite-element modeling system is used for calculation of changing in time stress state three-dimensional structures in T--FLEX CAD. The module works with three-dimensional T--FLEX CAD models and do not usually requires separate additional constructions for calculation of the three-dimensional model.

In the dynamic analysis the mechanical system is considered under the action of a time-varying external influences of the forces, pressures, accelerations, etc. applied to the system. The examples of Dynamic analysis studies are shaft or spindle spin up, the vehicle passage on the bridge, seismic vibrations, filling of hopper with sand, etc. A stress state of the mechanical system will also change in time. Analysis of such situations is important in many practical cases as dynamic stresses and inertial forces can have a significant influence on the overall performance of the mechanical system. As a result of the dynamic study solution we receive the same calculation results as for the static analysis (displacements, stresses, factor of safety) but each result presented for a point in time corresponding to the set time interval and the specific time step interval. The system user has a possibility to see the system strains and stresses changes in time and to predict the system behavior under the complex system of external changing in time influences.

A dynamic strength condition is formulated in the following way in the general case:

Stresses σ_{dyn} that appear in the structure under the action of external forces applied to the structure must be less than the allowable stresses $[\sigma]$ for the structural material with the use of a correction factor of safety SF by dynamic strength $\sigma_{dyn} \cdot SF \le [\sigma]$.

As for the static analysis the main calculation results are displacements, stresses, factor of safety in time-varying function.

Mathematical statement of dynamic study by finite element method looks in the following way:

$$[M][U] + [C][U] + [K][U] = \{F(t)\}, \qquad (1)$$

where:

M - symmetrical square matrix *K* of the system stiffness,

F(t) - vector of generalized forces, applied in the system nodes,

C - symmetrical square matrix of the system damping,

U - vector of generalized displacements, applied in the system nodes,

U, U - vectors of generalized velocities and accelerations, applied in the system nodes.

From the point of view of mathematical implementation there are two main approaches to the solution of systems of equations arising in the description of dynamic studies: mode superposition and transitional process.

Input data for dynamic calculation differ according to the study type. Consider parameters of each type of dynamic studies separately.

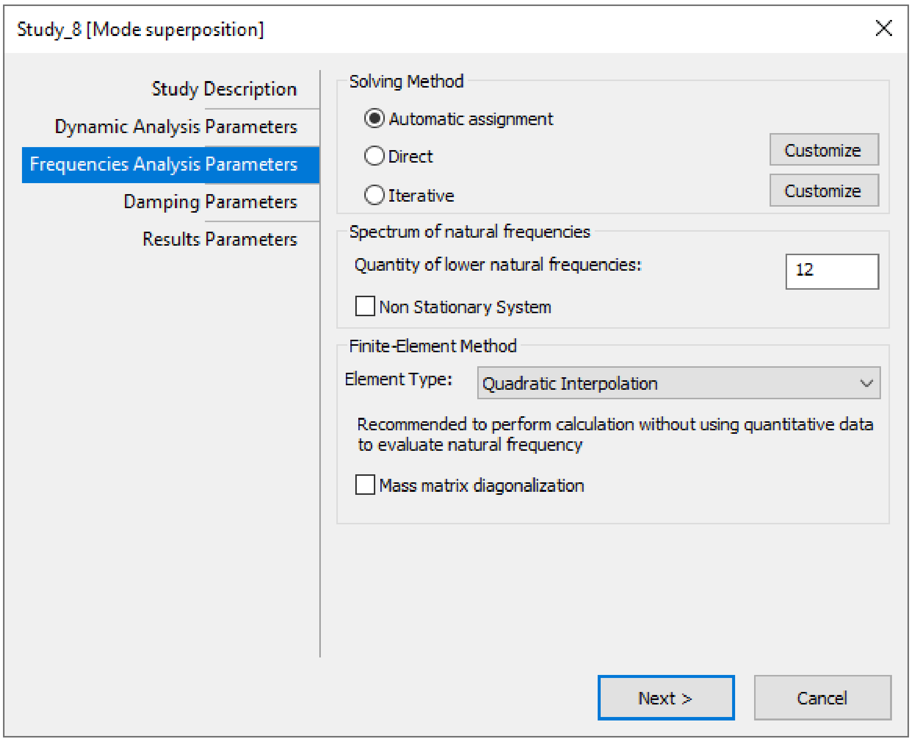
Mode Superposition

In the **mode superposition** the system (1) that consists of N related differential equations of motion is transformed in the equivalent system that consists of n independent ordinary differential equations. For this purpose the natural frequencies and oscillation forms of the system are calculated. Then, using

fundamental properties of oscillation forms, the system (1) that consists of many thousands and millions of equations is transformed in the system of independent differential equations of dimension equal to the number of oscillation forms selected for equalization. The resulting solution of the system (1) is presented as the sum of oscillation forms that were used for equalization, with calculated weighting factor. The advantage of the method is its computational efficiency at the dynamic study solution stage, i.e. solving of the reduced equations system is much more effective then solving of full equations system (1). You should take into account that preliminary calculation of natural frequencies and oscillation forms is a resource-demanding task and it should be taken into account when selecting method of dynamic task solution. One more important restriction of the mode superposition is impossibility of its usage for nonlinear studies.
Accuracy of the method depends on the oscillation forms, selected for equalization, and their quantity. Quantity of natural modes necessary for the exact calculation is in the range of 10-15 depending on the complexity of the model and the nature of the modeled impact. The greater the quantity, the more accurate the result, but the calculation time is increased correspondingly.
In whole the studies type may be recommended for modeling of long time linear dynamic processes. When the types and quantity of oscillation forms are selected correctly the method provides satisfactory accuracy and efficiency of the solution.

Before calculation of the Mode superposition user should define natural oscillation forms for which the equalization will be performed. It can be done by one of the two ways:

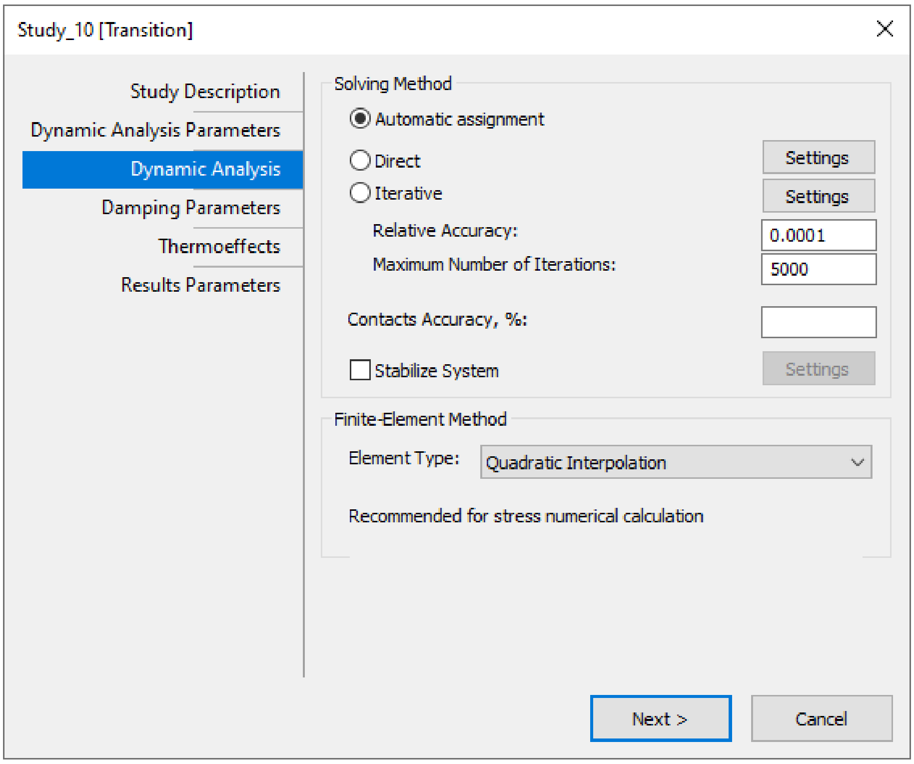
- On the **Frequencies Analysis Parameters** tab in the "Use frequency analysis results" group select already calculated study of frequency analysis (study type: "Frequency"). All natural modes from the selected study will be used for finding solution of mode superposition.
- Activate natural modes solving mode directly in the dynamic study. ш You need also select quantity of natural modes. Other control elements mainly coincide with the corresponding parameters of frequency analysis.



Dynamic Transitional Process

In the **Transitional Process** mode the solution of the equation (1) is found using numerical integration in time. Full equation system (1) is solved on each time step so the method is resource-demanding.

The study type may be recommended for modeling of short time dynamic processes. The solution is exact, but it requires increased computational resources.
Parameters of equations systems solving on the **Dynamic Analysis** tab repeat the same parameters for static analysis.



Topics in this section:

- Features of Dynamic Analysis Stages
- Parameters of Finite Time and Step of Modeling
- Algorithm of Evaluation of Dynamic Analysis Results

Features of Dynamic Analysis Stages

The analysis is performed in several stages. Sequence of user actions during the study preparation and dynamic calculation of the structure is similar to the algorithm described for the strength analysis. Thats why in the chapter we will describe only some features typical for the dynamic calculation:

Step 1.Study creation

When you create study you should select its type - Mode superposition or Transitional processes.

Step 2. Apply boundary conditions

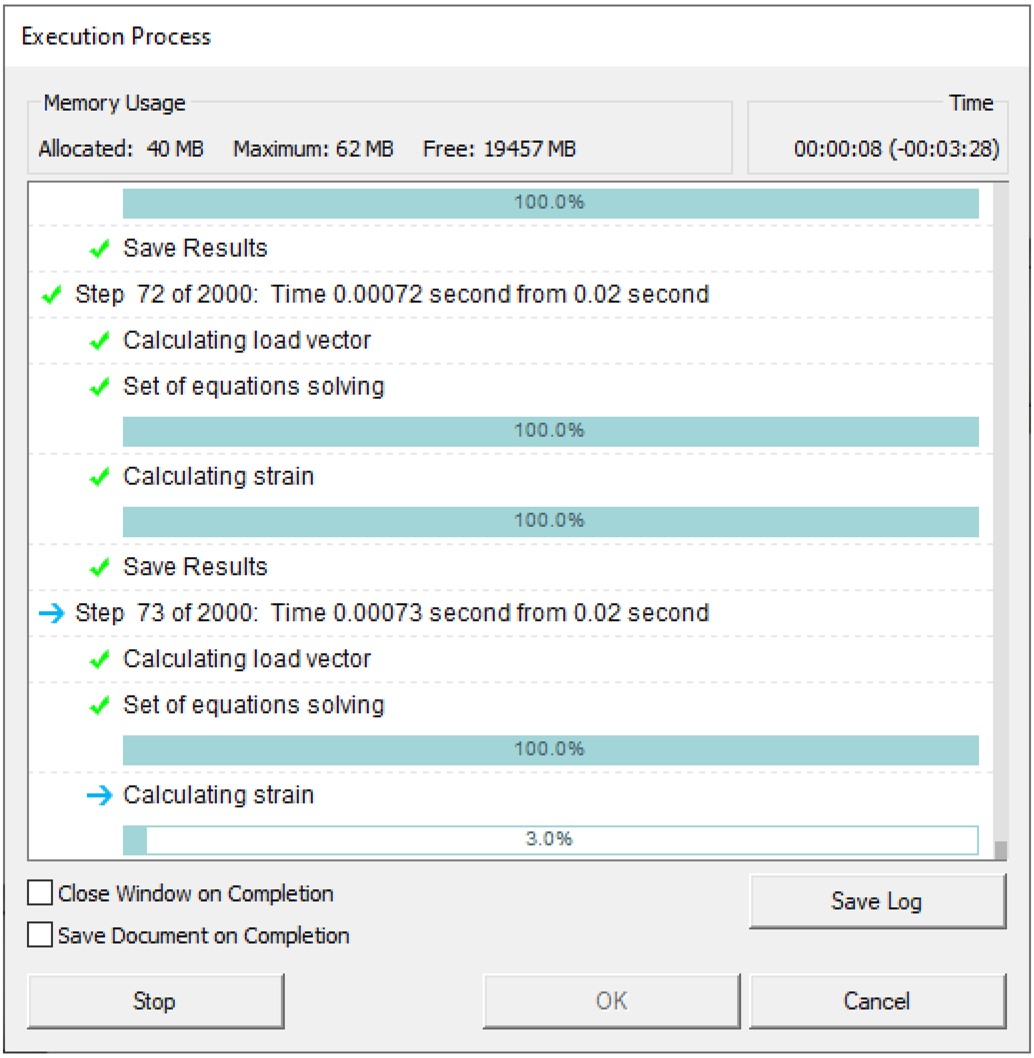
In the dynamic analysis as in the static analysis boundary conditions are loads and restraints. You can use all types of restraints and all types of force loads used in the static strength analysis. In addition, you can use initial conditions such as Initial velocity, Initial acceleration, and oscillator load. All loads may be specified using variables in time. You can find more information in the "Specify dependence of Value from Time" section.

- **Thermoeffects** are defined as for the strength analysis but thermal stresses are not taken into account. Only changes of physical properties of materials according to the temperature are considered.
- Defining of **restraints**, **oscillator** or **force loads** is the mandatory clause of the correct calculation. Thus, in contrast to static analysis there is no mandatory clause of model displacement as a whole body prohibition, i.e. for the calculation implementation it is enough

to apply single force for not long time period of calculation during which the model will be displaced for the finite distance.

Step 3. Solve Study

The user specifies time step of integration, selected according to the clause described above, method of time integration and, if necessary, one of the initial conditions receiving method in the study parameters before study solving: displacements from the static study, displacements, velocities and accelerations from another dynamic study with time specifying.
Information and diagnostics are displayed in the message window.
Nodes - number of nodes in the calculated finite-element mesh.
Elements - number of tetrahedrons/triangles in the finite-element mesh.
Arguments - number of equations participating in the solving.
Calculation type - direct or iterative.
Step ... of ... - current step and overall number of steps.
Calculation complete - indicates that the solving is finished successfully.



Step 4. Solving results analysis

After the solving is finished a new **Results** folder appears in the **Studies** window. Results specified on the **Results Parameters** tab in the **Parameters** dialog are displayed in the folder by default. Only 49 results grouped in 8 groups are available to the user after the dynamic analysis.

Parameters of Finite Time and Step of Modeling

User should specify the time step and finite time of modeling before running the dynamic calculation. The parameters are set on the **Dynamic Analysis Parameters** tab.

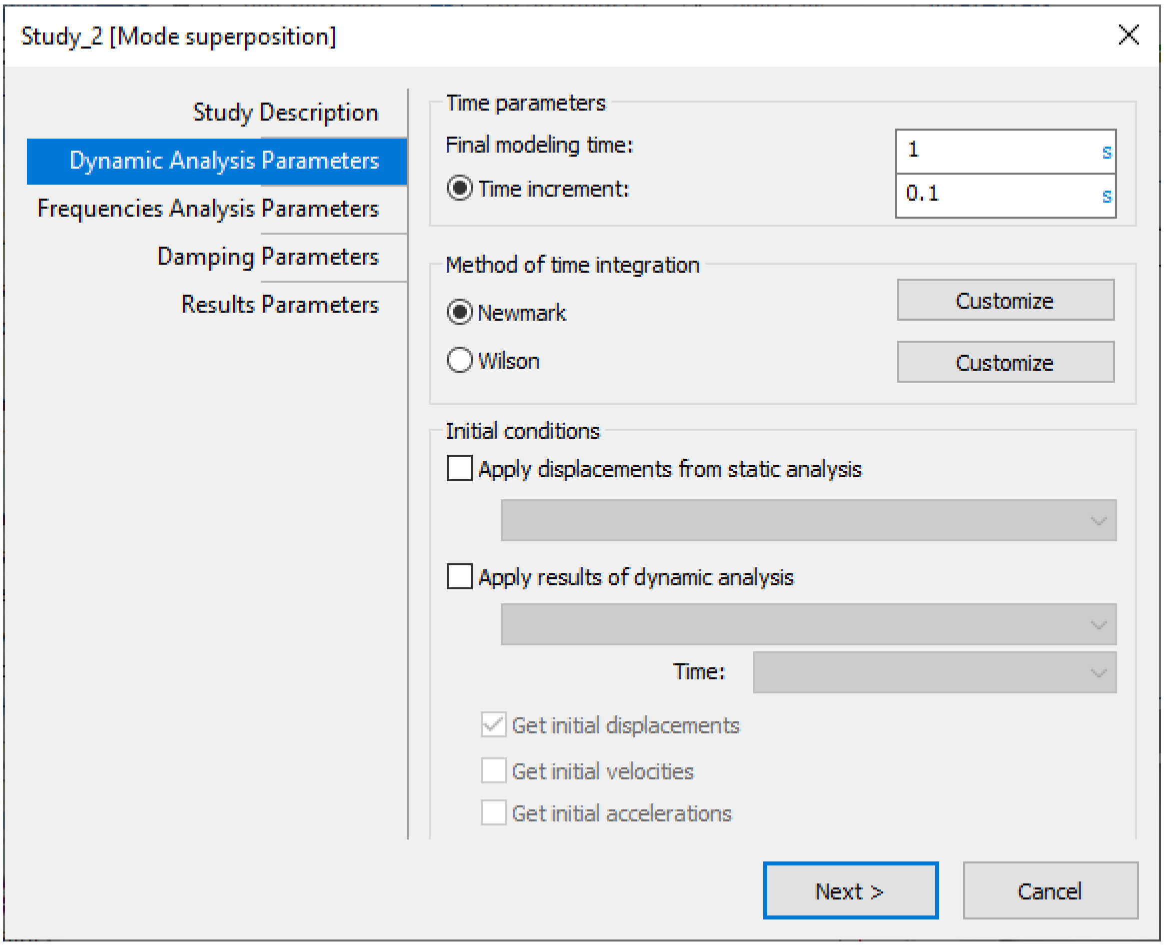
Time step

The time step should be selected carefully as it critically influence on the accuracy of the numerical integration and the received solution. For many practical calculations the maximum value of time step Δt can be estimated by expression:

$$\Delta t \leq T_{min} / 20,$$

where T_{min} is the smallest period of natural system oscillations and (or) interval of external variable load change. You should take into account that it is the maximal estimation and sometimes you have to choose a lot smaller step than this estimation. The maximum number of time steps in the current version is 65535.

Finite time of modeling is specified according to the conditions of calculated study. Too many steps can lead to long calculation time and fill disk space that required to store results.



Parameters of Time Integration Method

Difference scheme are used for the equations integration in both cases: Newmark or Wilson method.
Ntt/

The Newmark method is based on the following approximation of velocities and displacements:

$$\{\dot{U}(t+\tau)\} = \{\dot{U}(t)\} + \tau[(1-\delta)\{\ddot{U}(t)\} + \delta\{\ddot{U}(t+\tau)\}]$$

$$\{U(t+\tau)\} = \{U(t)\} + \{\dot{U}(t)\}\tau + [(0.5-\alpha)\{\ddot{U}(t)\} + \alpha\{\ddot{U}(t+\tau)\}]\tau^2,$$

where α, δ - parameters that determine the accuracy and stability of integration.

For $\alpha = 1/6$, $\delta = 1/2$ accelerations vary linearly within the interval of integration; preferred for linear studies;

For $\alpha = 1/4$, $\delta = 1/2$ accelerations remain constant within the interval of integration step. In this case the method unconditionally converge; used for nonlinear study. The condition of the method convergence on the assumption of proper choice of the time step: *a > 1/8*.
The Wilson method is based on the fact that the vector of accelerations and loads vary linearly in the interval $(t+\theta\tau)$, where $\theta \geq 1$, based on the following approximation of the accelerations:

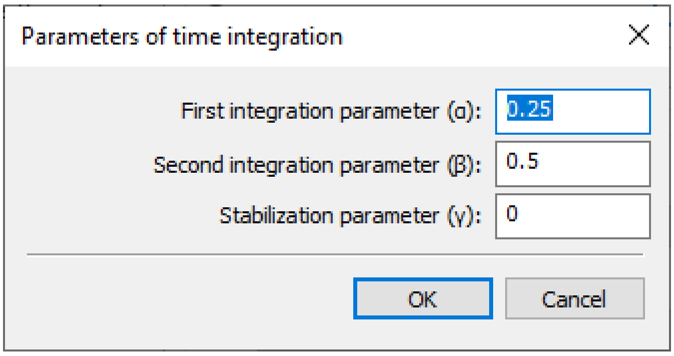
$$\{U(t+\tau)\} = \{U(t)\} + \Delta t(\{U(t+\theta\tau)\} - U(t)\})/\theta\tau,$$

The expressions for velocities and displacements are received by integration of the equation:

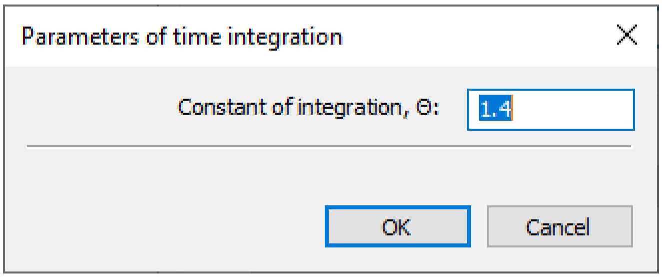
$$\{U(t+\theta\tau)\} = 6(\{U(t+\theta\tau)\} - U(t)\})/\theta^2\tau^2 - 6(\{U(t)\}\theta\tau - 2\{U(t)\})/\theta\tau,$$

$$\{U(t+\theta\tau)\} = 3(\{U(t+\theta\tau)\} - \{U(t)\} - 2\{U(t)\} - 0.5\theta\tau\{U(t)\})/\theta\tau,$$

Minimal value of θ=1,37, using which the method steadily converges, is θ=1,4.
The parameters are set on the **Parameters tab**. Specifying of time integration parameters for the Newmark method is performed by pressing on the **Customize** button near the method switcher. The parameters dialog is opened in this case. By default, the parameters are configured to unconditionally convergence of the method.



Specifying of parameters for the Wilson method is performed by pressing on the **Customize** button near the method switcher. By default, the integration parameters are configured to unconditionally convergence of the method.

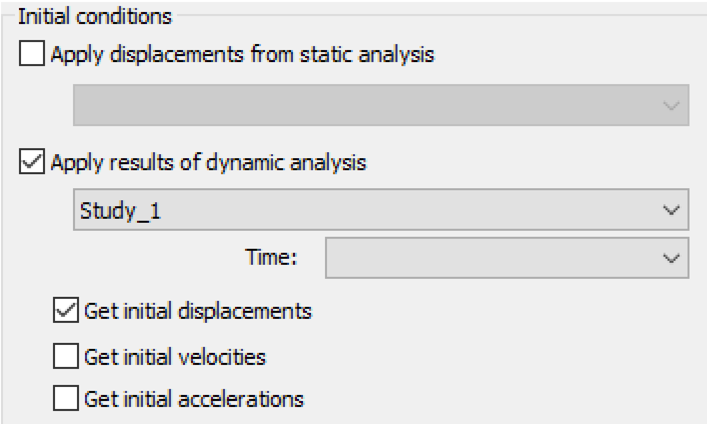


Assign initial Conditions

Besides specifying of the initial velocities and accelerations using special commands, user may also define initial conditions using the results of previously solved statics or dynamics studies. Important condition for such usage is the requirement of identity of the finite element meshes in both tasks. The parameters are set on the **Dynamic Analysis Parameters** tab in the **Initial conditions** section. You can use one of the two ways:

- Apply displacements from the static analysis.
- Apply results of dynamic analysis.

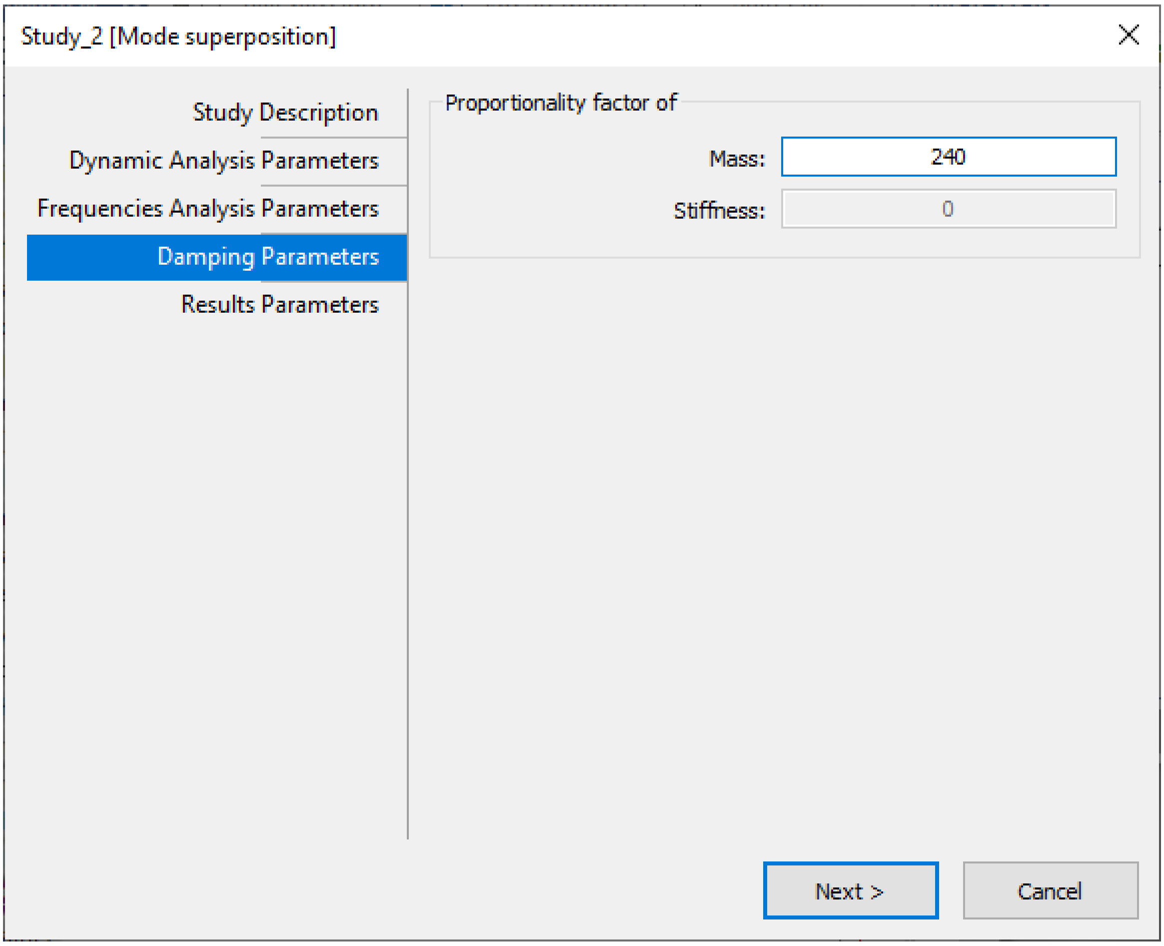
In the last case you can apply not only displacements but also velocities and accelerations from another study to the initial step.



Damping Accounting

The Rayleigh damping is accepted in the system, i.e. damping matrix is proportional to the stiffness and mass matrices and is defined by the expression *[C] = a[M] + b[K]* where a is the proportionality factor of the masses; b – proportionality factor of stiffness (both coefficients are scalars).

The parameters are set on the **Damping Parameters** tab . The parameters on the tab are similar to the parameters of forced oscillation analysis. The difference is that the proportionality factor of stiffness is not set for the Mode superposition study type.

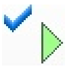


Thermoeffects Accounting

The **Thermoeffects tab** allows to activate considering dependence of material properties from the temperature mode.

Consider Dependence of Physical Properties from Temperature When the mode is active material properties will be read from the graphs of temperature dependences that are set in the parameters of study materials.
You can specify way of thermal load assigning for physical properties of materials in the **Temperature fields** section. The loads are used for calculation of materials physical parameters using graphs of temperature dependences, specified in their parameters. The control elements on the tab are similar to the parameters of static analysis.

Algorithm of Evaluation of Dynamic Analysis Results

You should analyze the received results after successful calculation of the study to conclude about the probable dynamic strength of the structure. The three types of results are enough in many cases - displacements, stresses and factor of safety by stresses. By default, each result is opened on the last time step, which may not be enough to obtain a dynamic picture of changes of strains and stresses. Therefore, when you analyze the results you should use timescale and graphs, created according to the sensors.
Enable/disable time process pan. For the dynamic studies it does meter in what period of time is to consider the state of strain since the peak of the stresses or displacements may appear at any time step. To display all results according to the time steps you should enable time process using  icon or enable animation using  icon on the ribbon.
Using graphs for the results analysis is described in the corresponding section of "**Preprocessor**" chapter.
General results of dynamic calculations (with time step) are:

- fields of the structure displacements in nodes of the finite-element mesh;
- relative strains fields;
- stress components fields;
- strain energy;
- node response;
- fields of the strain safety factor distribution over the volume of the structure;
- velocity field in the computation points of the finite-element mesh;
- acceleration field in the computation points of the finite-element mesh.

Usually this data is sufficient to predict the behavior of the structure and make a decision to optimize the geometric shape of the product to ensure the basic conditions of the products dynamic strength.

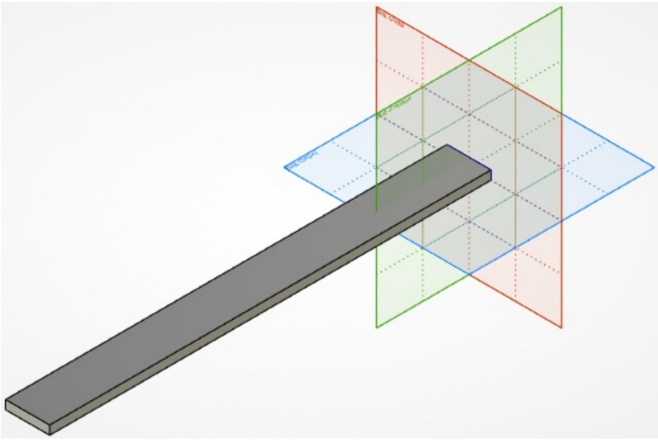
Calculation Example. Dynamic analysis of cantilever beam with varying load

To illustrate functioning of dynamic analysis module we will use the example of cantilever fixed rectangular beam with dimensions 1000x100x20. The force increasing from 0 to 500 is applied to the free end. The force is acting during 2 sec after which it remains. It is necessary to define the maximal displacements of the beam end and dynamic stresses in the closing.

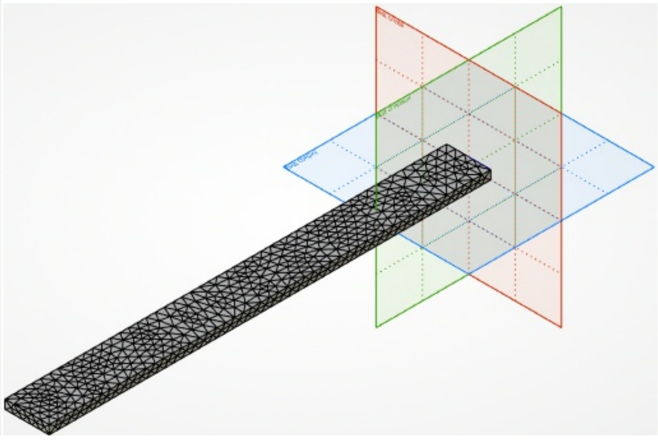
Dynamic Transitional Process

Step 1. Creation of study, creation of mesh, applying material

Use the **New Study** command to create the **Transitional Processes** study on the basis of the body - beam with dimensions 1000x100x20. Create finite-element mesh. You need to define material parameters of the model. By default the properties **From study Operation/Body** are used in the calculation, i.e. the material properties are automatically taken from the solid model of the product. The mode is convenient if bodies with different materials participate in the calculation of the assembly model. In this case the "Steel" material was applied during 3D model creation. The physico-chemical properties exist in the T-FLEX CAD base.



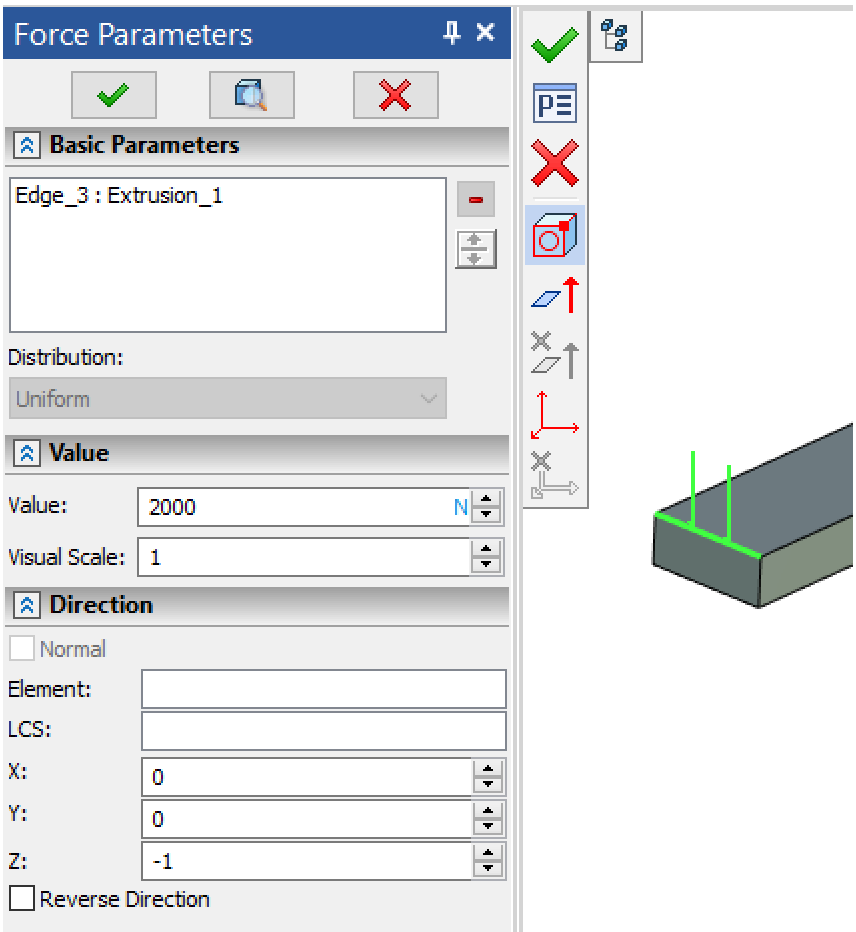
3D model



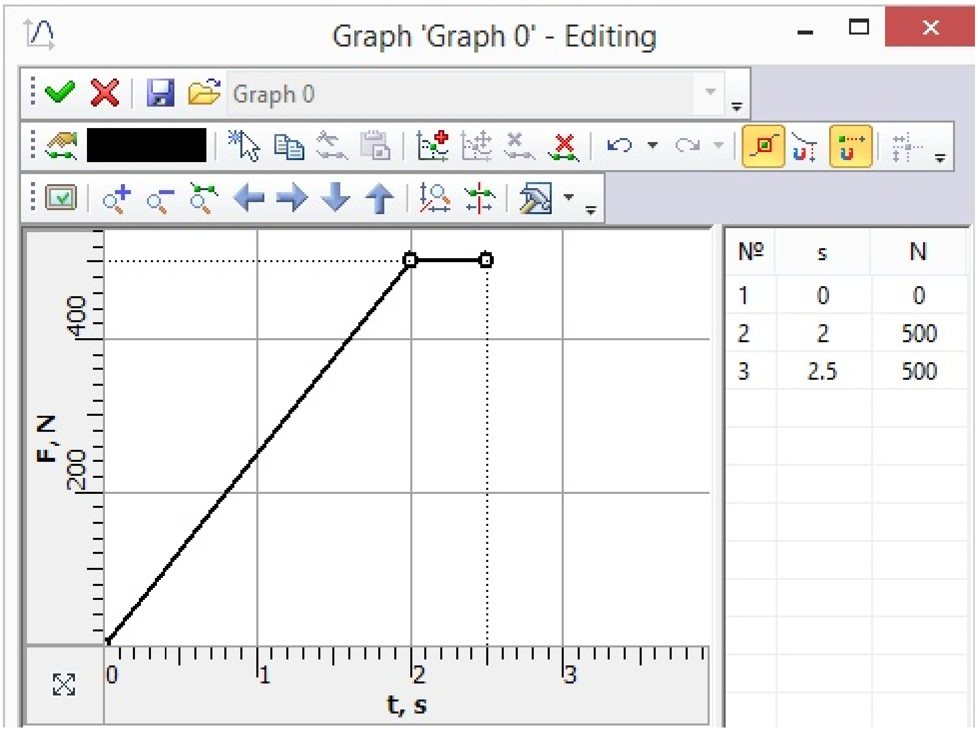
Created finite-element mesh

Step 2. Apply initial and boundary conditions

Initial conditions and loads should be pecified. Initial conditions will be zero velocities and accelerations that need not be set separately. Apply full restraint to one end and vertical force that varies according to the graph to the other end.



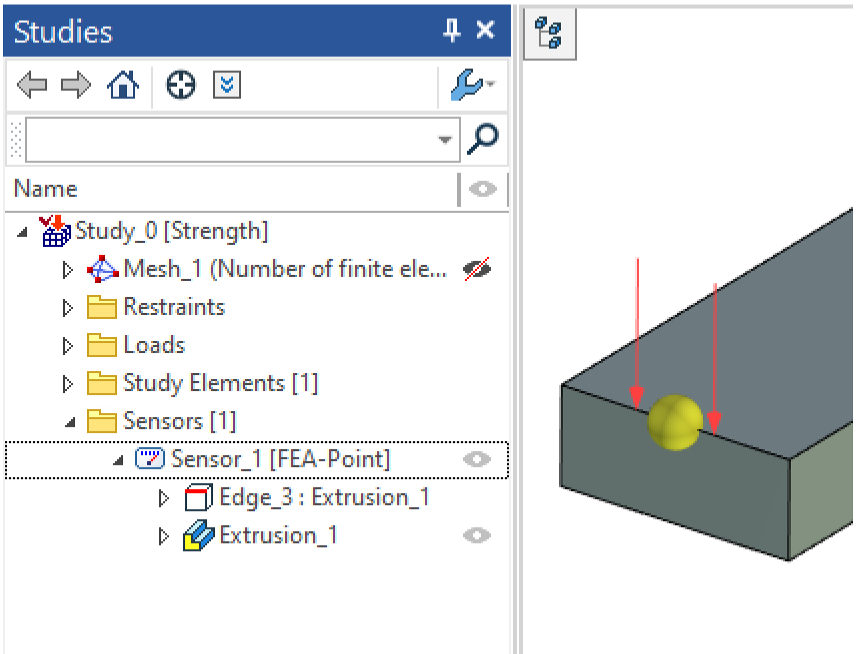
Varying load to the face



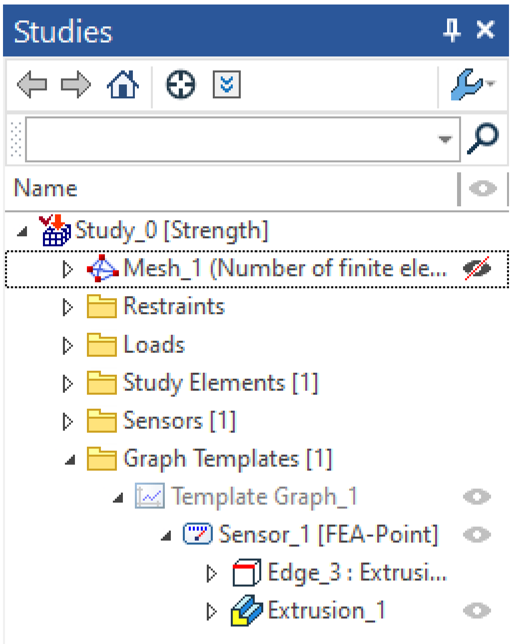
Graph of varying load

Step 3. Creation of sensors and graphs templates

To define the maximum deflection and velocity of the beam end it is necessary to create sensor using which the graph template is created:



Sensor in the middle of the edge

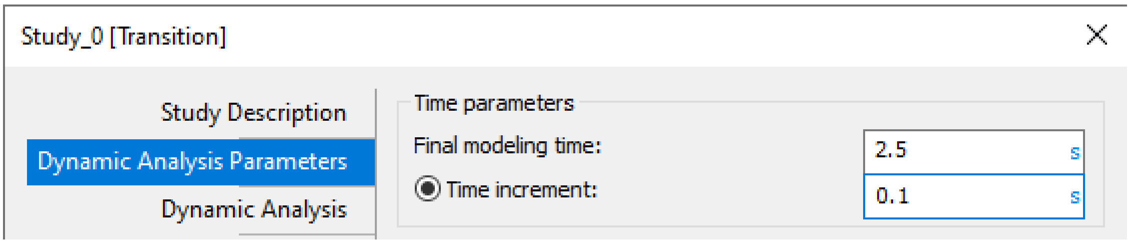


Graph template created using sensor in the study window

Step 4. Solve Study

The study calculation is started after the restraints and loads are defined.

You should specify time step and finite time of the calculation in the study properties. According to the slow load change the 0.025 sec time step is defined.

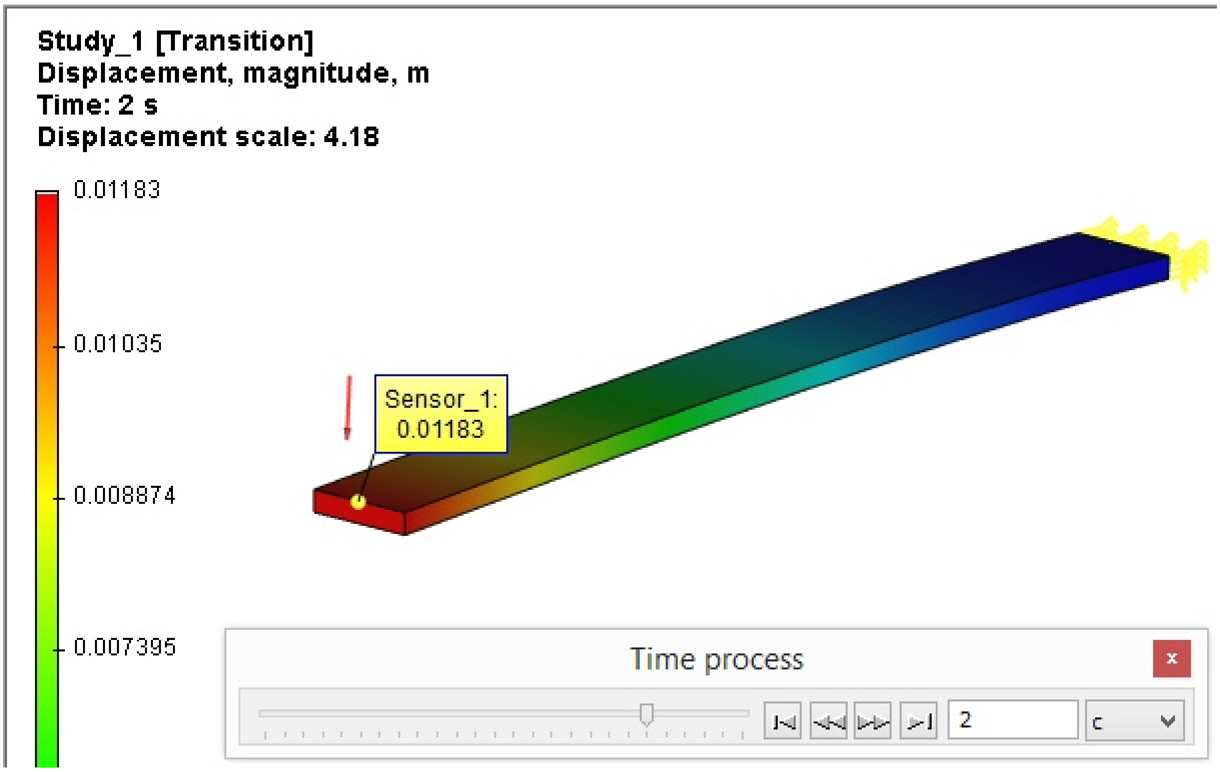


Specify step and time of the modeling

Step 5. Solving results analysis

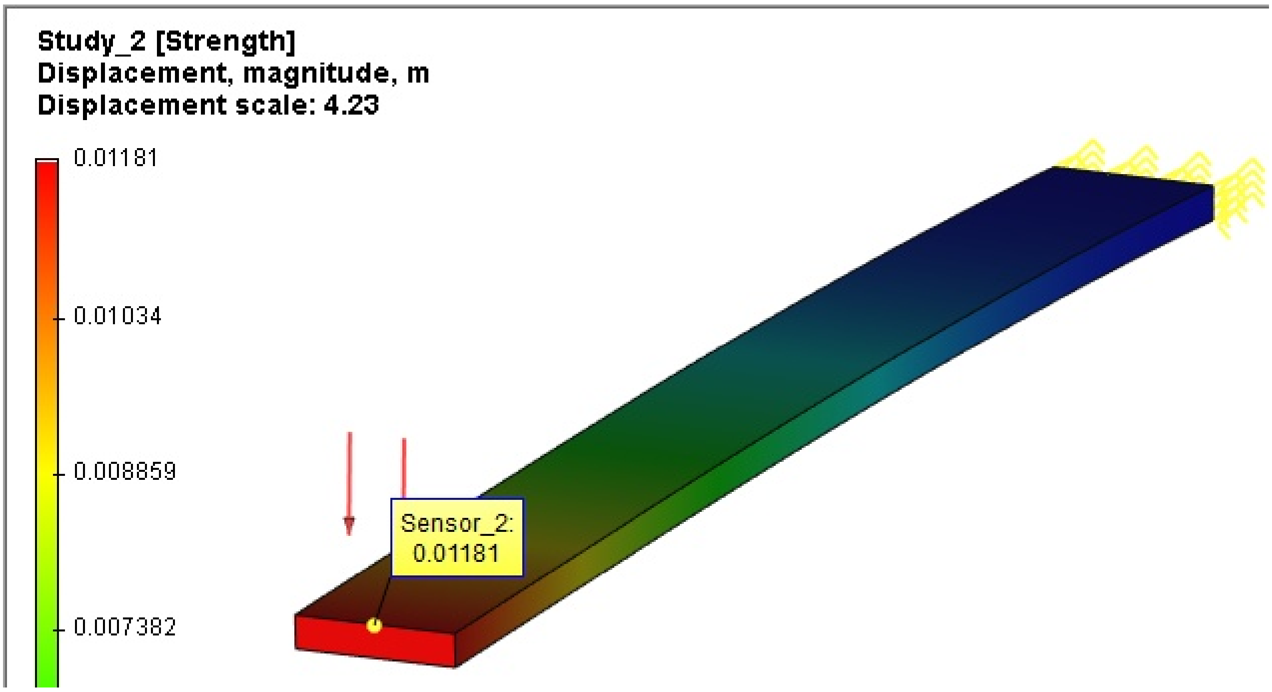
After the calculation you need to open displacements result.

After "**Time process**" control pan activation we can watch the deformation process of the beam step by step. The beam will deform to the maximum value under the load and then the beam oscillates by inertia because it is still loaded with applied force. Oscillations are continuous so that the damping is not used in the study.



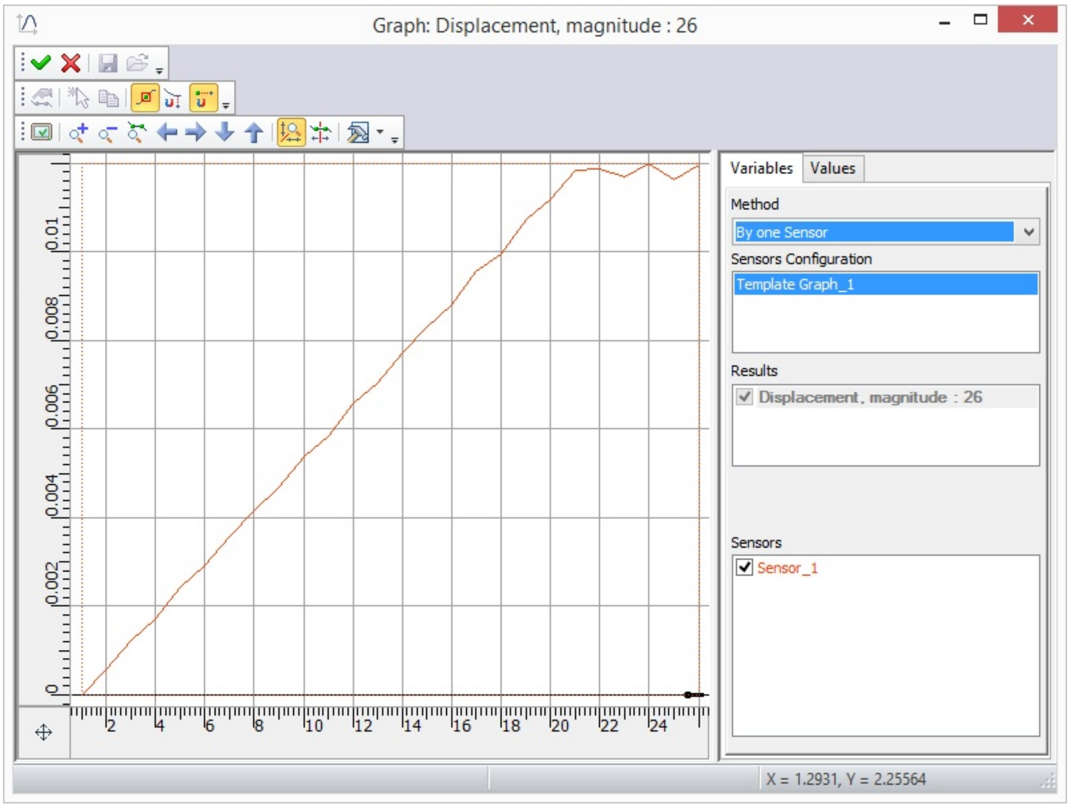
Displacements - time process

Deflection of the same beam under the constant force, received after static strength study calculation. The results practically coincide.



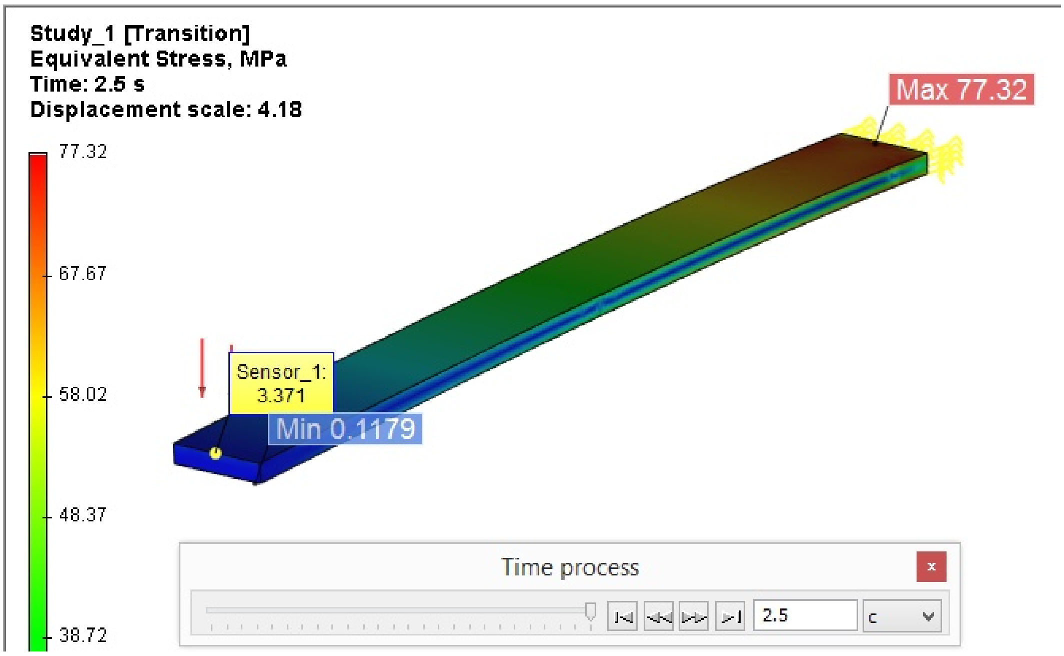
Comparison with the static displacement from the same load

On the displacements graph created using Sensor_1 you may see that the beam end continues to oscillate with small amplitude with regard to the maximum deflection loaded with the applied force of 500 N.



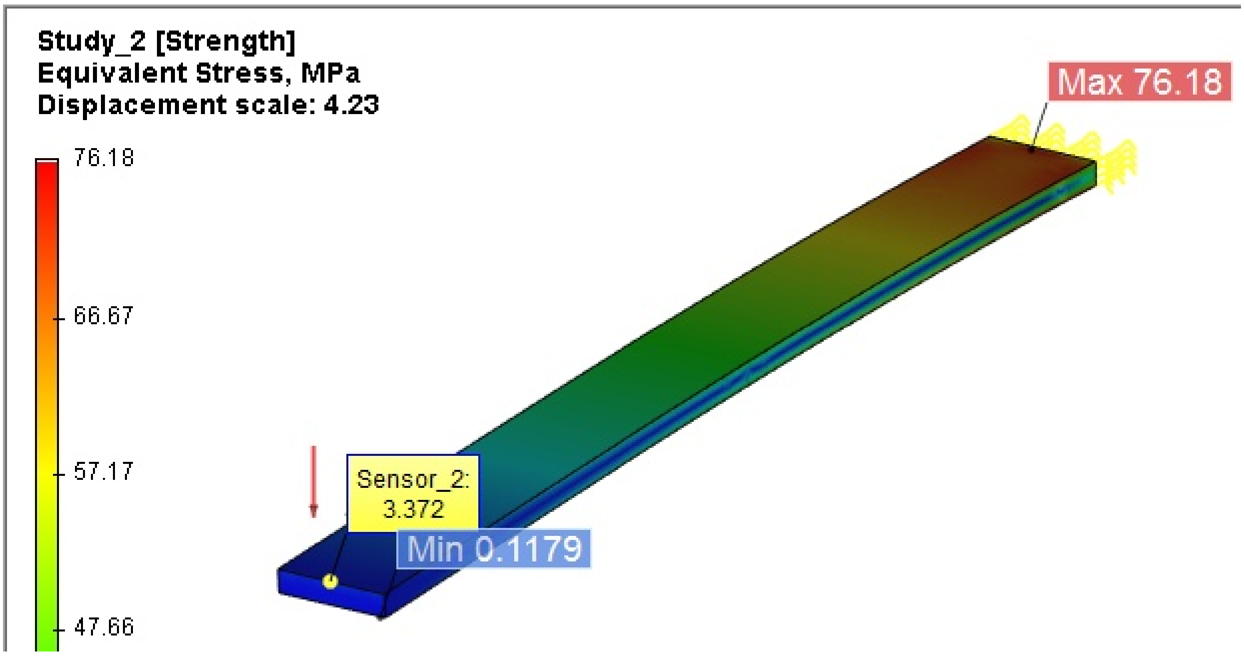
Displacements - time process

The maximal stresses can be shown on the stresses diagram. Display extremums.



Dynamic

Make sure that received dynamic stresses are close to the stresses received in the static calculation.



Comparison with static stresses

Mode Superposition

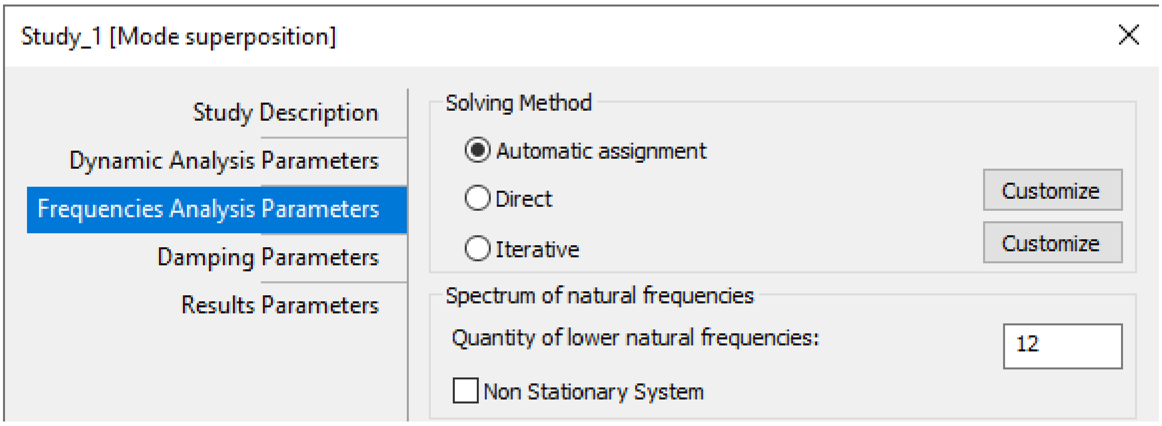
Calculate previous study using mode superposition.

Step 1. Creation or copying of the study

Use the **New Study** command to create the Mode superposition study on the basis of the same beam or copy previous study and change its type to Mode superposition in the parameters in order not to apply twice the same loads and restraints.

Step 2. Calculation of natural modes

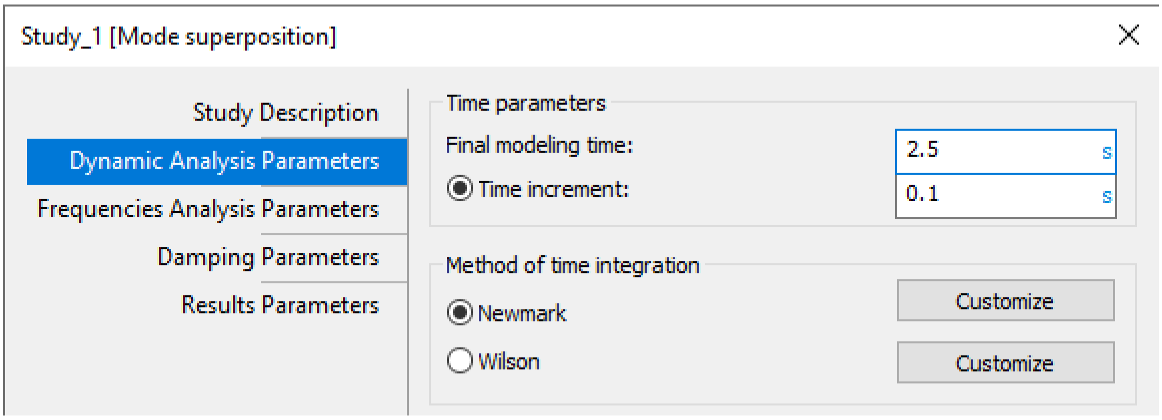
On the **Frequencies Analysis Parameters tab** the system offers to use 12 natural modes for the calculation.



Specify number of calculated natural modes

Step 3. Solve Study

The study calculation is started after the restraints and loads are defined.
You should specify the same time step and finite time of the calculation as for the previous example in the study parameters on the **Dynamic Analysis Parameters** tab.

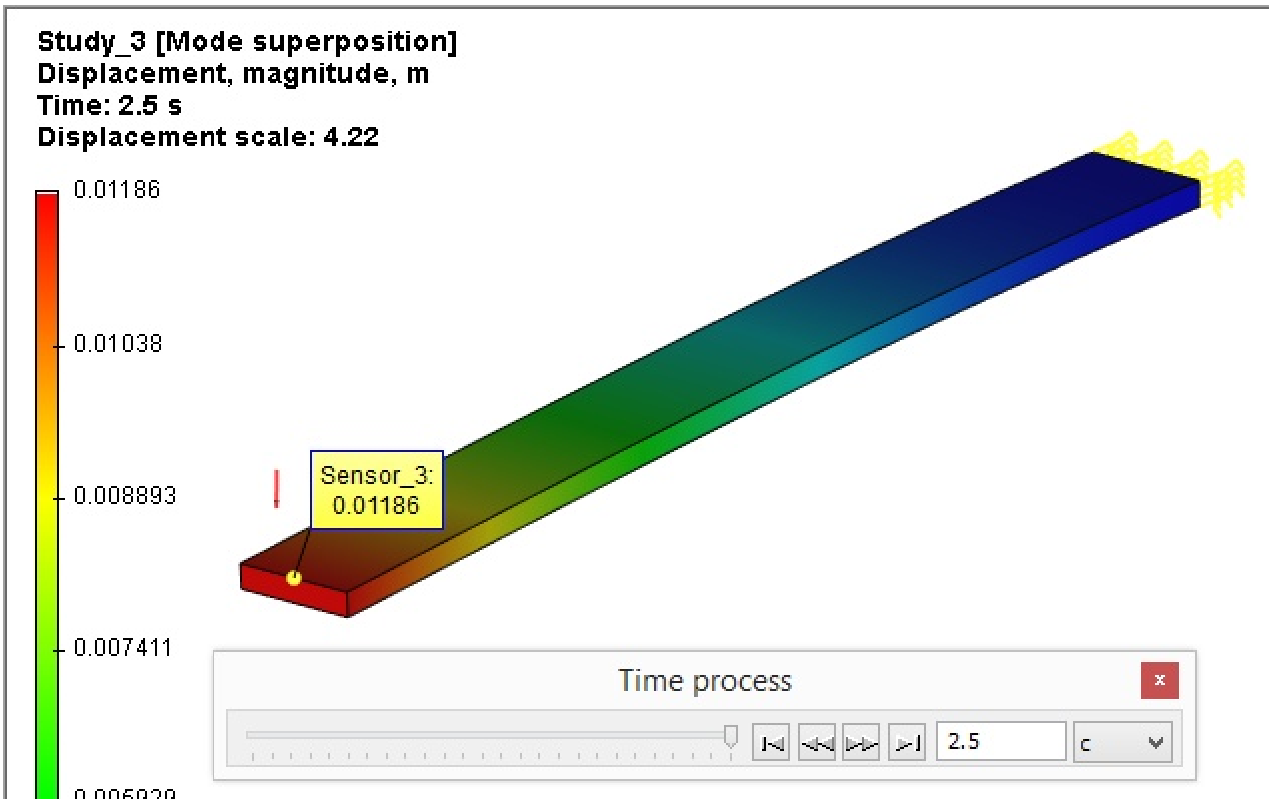


Specify step and time of the modeling

Step 4. Solving results analysis

After the calculation you need to open displacements result.

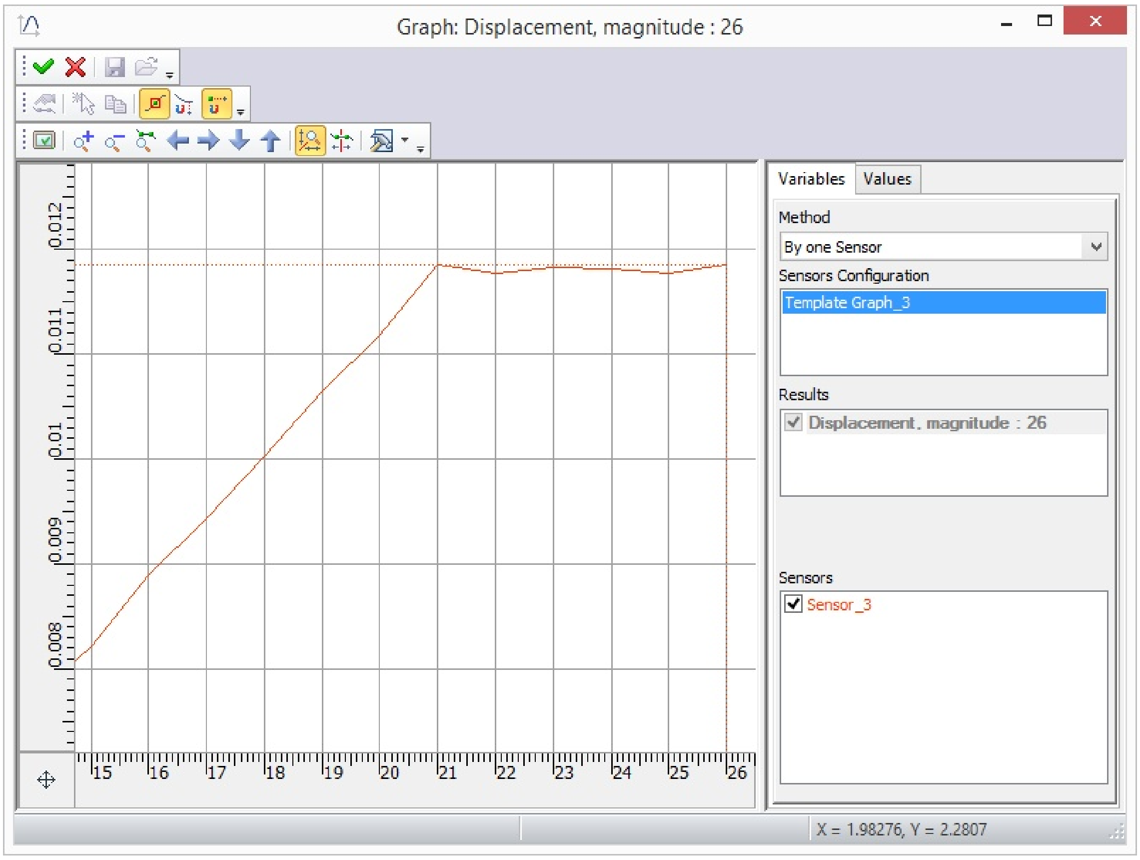
After the calculation is complete you should activate "Time process" control pan to watch the deformation process of the beam step by step. You can see that as for the Transitional process study the beam will deform under the load to the maximum value and then the beam oscillates by inertia with small amplitude near the maximum value of the deflection. The amplitude of the deflection practically coincide with the value found in the Transitional processes and Static strength studies.



Displacements - time process

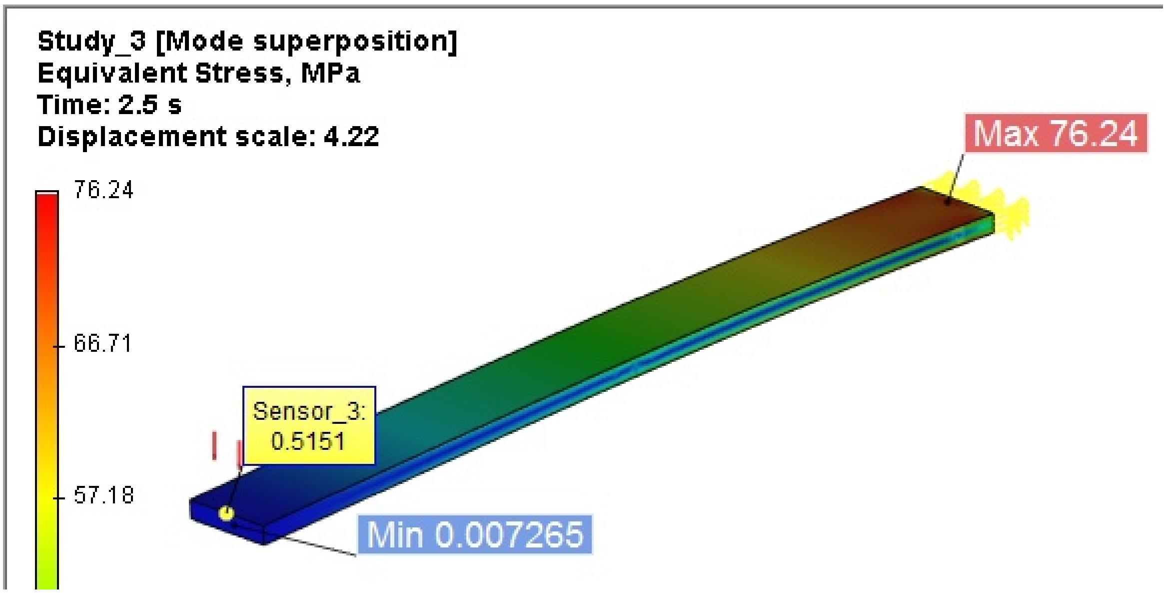
On the displacements graph created using Sensor_3 you may see that the beam end continues to oscillate with small amplitude with regard to the maximum deflection.

Oscillations are continuous so that the damping is not used in the study.



Displacements - time process

When extrema are displayed on the stresses diagram you can see that the computation stresses are close by their values to the previously calculated values.



Dynamic stresses in the Mode superposition

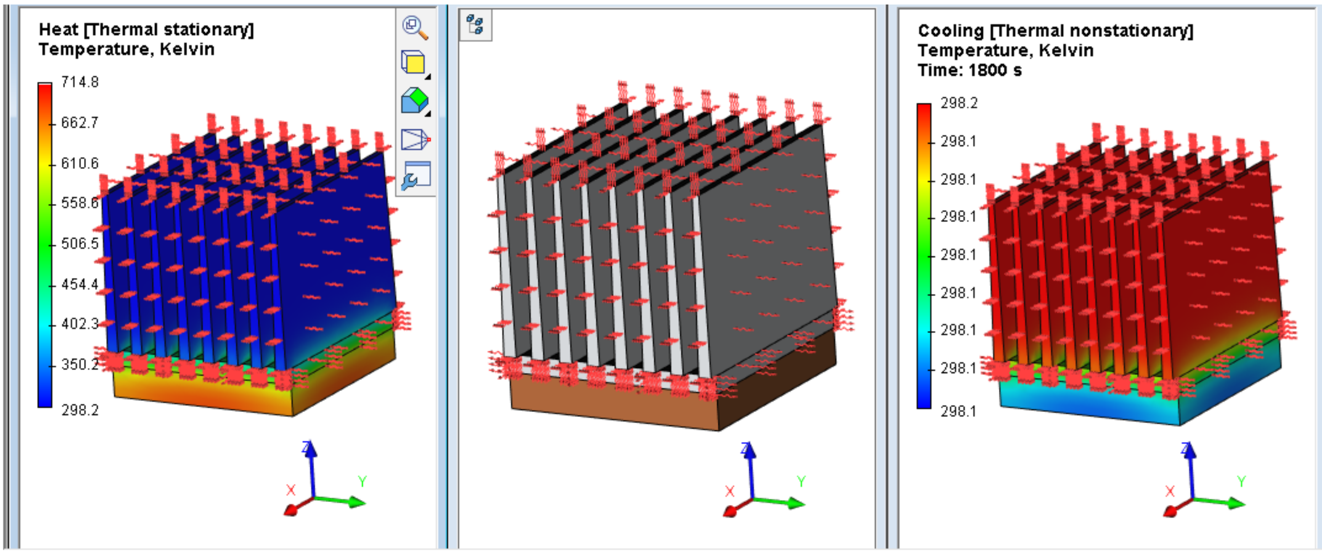
Conclusion

According to the solved study you may see that both of the methods of dynamic study calculation lead to the same result and can be used for the dynamic studies calculation.

Thermal Analysis

The thermal analysis module serves for solving heat transfer and thermal conduction problems. A typical goal of performing thermal analysis is finding temperature fields and heat (thermal) flux within a product's volume. T-FLEX Analysis supports two ways of formulating a thermal analysis problem:

- **Stationary Thermal Process** – calculating temperature fields and heat flux distribution under the assumption of an infinitely long time passing after applying thermal loads. A body's temperature does not change with time in the steady state, so that an elementary body volume loses as much energy to the environment per the time unit as it gains from outside or from internal heat sources.
- **Nonstationary Thermal Process** – temperature fields calculation occurs as a function of time. Thermal energy sources, temperatures and thermophysical properties of the system may change in time. The temperature field distribution pattern changes with time in the analyzed physical system, so that the study results in obtaining temperature fields at each time instant of a certain time period set forth by the user.



Examples of a stationary (left) and nonstationary (right) thermal processes

Topics in this section:

- Details of Thermal Analysis Steps
- Thermal Analysis Processor Settings
- Steady State
- Trancient Mode

Details of Thermal Analysis Steps

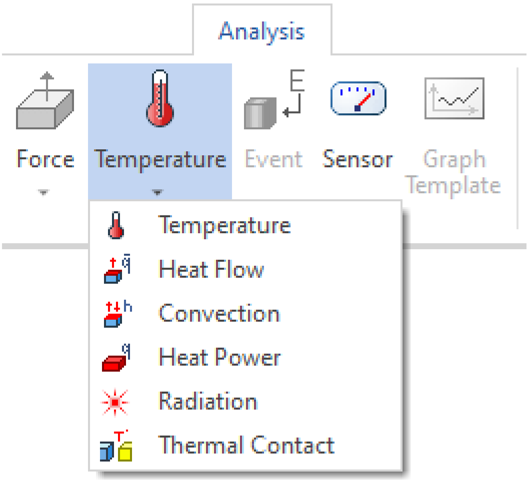
Thermal analysis is performed in several stages. The sequence of the user's steps for putting together a study and running a thermal study of a structure is in many parts similar to algorithms of working with other study modules of T-FLEX Analysis. Therefore, we will point out in this chapter only certain details specific to thermal studies.

Step 1. Creating Study

When creating a study, specify its type – **Stationary Thermal Analysis** or **Nonstationary Thermal Analysis**. As in other study types, building a finite element mesh is required, for approximating the structure's geometry.

Step 2. Applying boundary conditions

In the thermal analysis, the boundary conditions are represented by the boundary and initial temperatures, heat power sources, heat flow, and conditions of heat exchange between the model and environment – convection and radiation applied to the model.



Commands for defining boundary conditions of thermal analysis.

When defining thermal loads, you need to distinguish and appropriately use the two options of defining the «Temperature» load (see Preprocessor) – Initial Temperature and Temperature. The initial temperature is used for defining thermal loads at the initial (zero) moment of time for the transient thermal analysis only. All thermal loads defined without the «initial» flag are considered constant (invariable) in both the stationary and nonstationary thermal analysis.

Step 3. Solving

Before running calculations, the user can specify adjust algorithms for solving systems of equations on the **Thermal Analysis** tab.

Step 4. Analysis of thermal solution results

The results of a thermal analysis are:

Temperature fields – temperature distribution over the model's volume.

Thermal gradients by the X, Y, Z axes, and the magnitude of the thermal gradient – reflect on the degree of temperature changes by the respective axes of the coordinate system.

Resulting thermal flux by the X, Y, Z axes, and the magnitude of the resulting thermal flux –show the rate of thermal energy transfer, determined from the solution to the thermal analysis study.

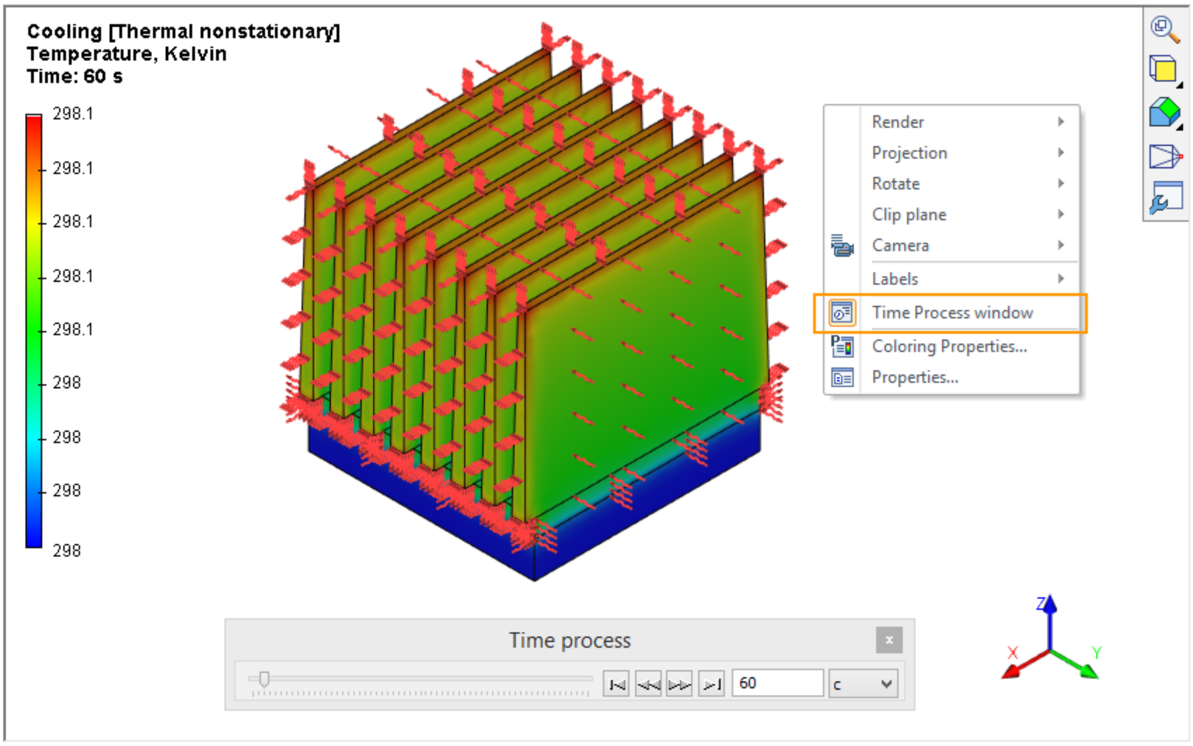
Magnitudes of the thermal (temperature) gradient and the resulting heat flux are determined as the square root of the sum of the squares of the respective coordinate-projected components.

Besides the mentioned results, the following reference data can be displayed in the postprocessor window:

- **Prescribed thermal flux** corresponds to the specified initial parameters of thermal loads.
- **Prescribed temperature** – constant thermal loads applied to the model.
- **Initial temperature** – the initial temperature field applied to the model (for the transient thermal analysis).

The methods for analyzing results of thermal analysis accepted in the T-FLEX Analysis Postprocessor, are in general similar to the methods of examining results in other analysis modules. Let us mention some specific Postprocessor tools, which can be used for analyzing results of **Nonstationaty thermal process**.

Solving a **Nonstationaty thermal process** study results in a large set of data, whose total number is equal to the number of time steps specified by the user. T-FLEX Analysis provides the user with a convenient visual interface for managing the entire array of data resulting from calculations. For this purpose, a **Time process** dialog panel can be called from the results viewing window's context menu, that can be used by the user to quickly switch to the desired result on the time scale.



Use of the **Time process** window for managing access to the results of a Nonstationary Thermal Process study

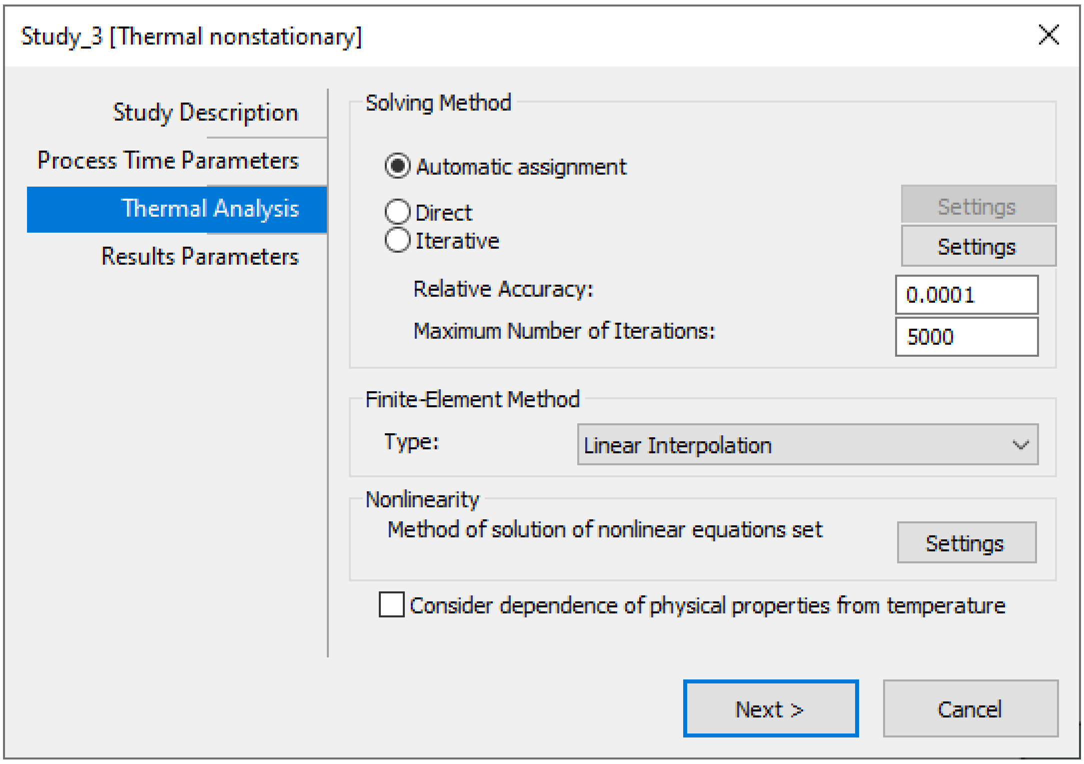
Thermal Analysis Processor Settings

Stationary Thermal Processes

The **Study Description, Thermal Analysis** and **Results Parameters** tabs are the same for the both of thermal studies types.

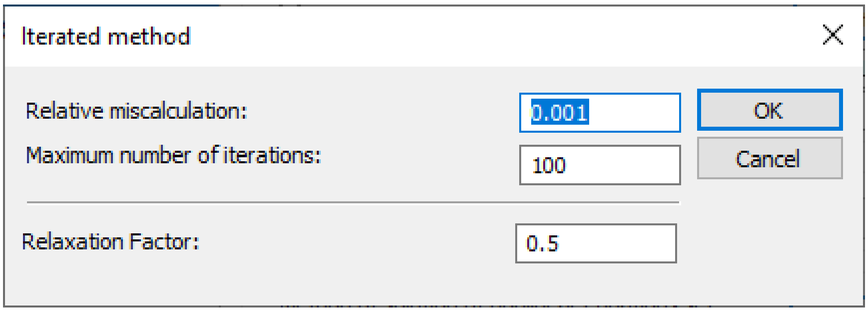
On the **Study Description** tab, you can define or edit the descriptive attributes of the current study, as the name or a comment. The **Thermal Analysis** tab contains settings for solving systems of algebraic equations, with their meanings similar to the settings of the **Static Analysis** study (see the respective section). Note that the **Linear Interpolation** mode can be used in most cases of thermal analysis, facilitating much faster calculations. The finite element is used by default as, unlike studies in **Statics**, **Frequency analysis** and **Buckling**, results of temperature distribution over the model's volume can be calculated quite accurately. Calculation of the study using quadratic finite element may be required if it is necessary to use results of the thermal study (temperatures distribution) in the mechanical study for thermoeffects calculation.

When you solve study, using the quadratic element, the **Iterated** and **Automatic assignment** calculation modes become available. When **Automatic assignment** mode is chosen, the system assigns method of solution based on total number of equations.



Thermal studies parameters dialogs

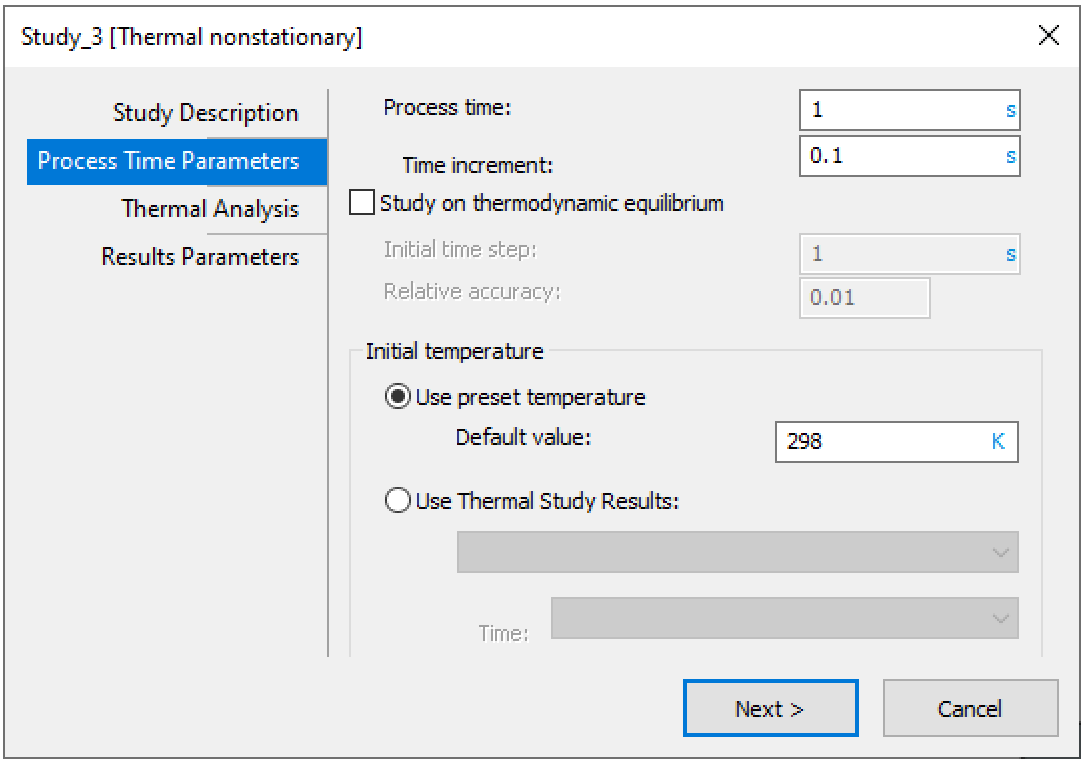
You can open the dialog of the iterated method of nonlinear equations set solution using the **Settings** button in the **Nonlinearity** group.



Nonstationary Thermal Processes

The user should set process time, time step and initial temperature using the **Process Time Parameters** tab before the calculation of the nonstationary thermal process study.

The **Study on thermodynamic equilibrium** study type is also available on the **Process Time Parameters** tab. For the study type, you should set **Initial time step** and **Relative accuracy**. Time step of the following iterations will be chosen automatically during calculation. The result of calculation using the option contains the temperature that established in equilibrium and time, in which the equilibrium temperature established. The study type is recommended for calculation of the thermal studies that take into account radiation between faces.



The **Use preset temperature** option allow the user to define as an initial temperature:

- initial temperature prescribed with the help of the command **Analysis** > **Thermal Load** > **Temperature**;
- the default value of temperature at those finite element nodes where the initial temperature was not defined by the user.

The **Use Thermal Study Results** control allows defining the initial temperature by the results of an earlier conducted thermal analysis. This dialog item becomes accessible to the user, if there are earlier conducted thermal studies present in the model. In the drop-down list select the name of a solved thermal analysis study and, if necessary, the time instant, to which the solution pertains. Please note that certain conditions are to be met for using thermal analysis results as the initial temperature conditions:

Identity condition of finite element meshes in both thermal analyses. The simplest way of achieving such identity is the use of the **Copy Study Items** command available in the context menu of the study. The sequence of steps can be, for example, as follows:

1. create a study of the **Thermal Analysis** type, generate a mesh, define boundary conditions, and run. We assume that the solved temperatures will be used for defining initial temperatures in another study of the transient thermal analysis;
2. create a study's copy using the **Copy Study Items** command;
3. if the Stationary **Thermal Process study** was copied, then we change the study type to the **Nonstationary Thermal Study** using **Edit** option from the context menu;
4. define boundary conditions of a transient study in thermal analysis. On the **Parameters** tab of the study's parameters, select the first study and, if that's a transient analysis, the desired time step.

As a result, we have two studies with identical finite element meshes.

The **Calculate using linear element** parameter on the **Thermal Analysis** tab of the study parameters dialog should use the same settings in **both** studies. For example, if the first thermal analysis is done by linear elements, then the second thermal analysis based on the former thermal analysis results can also be run only by linear elements.

Note also that solving a nonstationary thermal study requires more CPU time as compared to the stationary thermal study, since in the former case the systems of algebraic equations are solved at each time step defined by the user.

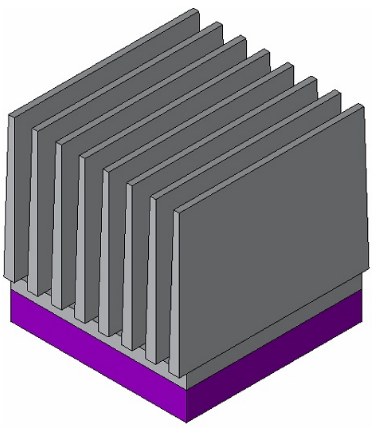
Steady State

Thermal Analysis of a Cooling Radiator. Steady State

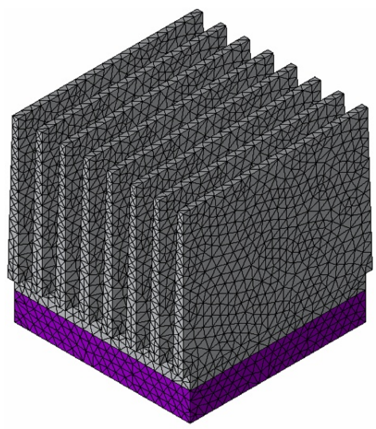
Required is an evaluation of a passive cooling radiator efficiency for the semiconductor electronic device with the maximum dissipating power of 15 Watt. The permissible temperature of the microchip's body is 75°C in the operating range of ambient temperatures from 25°C to 55°C. An aluminum alloy radiator is used for cooling the device and is mounted at the top of the microchip's body. To improve heat dissipation, the body of the microchip is made of copper.

Step 1. Creating «Study», meshing, and assigning material

Create a study of the **Stationary Thermal Process** type using the command **Analysis** > **New Study** based on two bodies – the microchip and the radiator. Generate a finite element mesh. You also need to define parameters of the part's material. By default, calculations use material properties **From Study Operation/Body**, that is, the material properties are automatically obtained from the product part's solid model. This is especially convenient when a study includes bodies from different materials representing parts of assembly models. In our case, the **Aluminum** material was defined at creation of the 3D model of the radiator, with its physical and chemical properties contained in the T-FLEX CAD 3D database. Microchip is made of **Copper**.



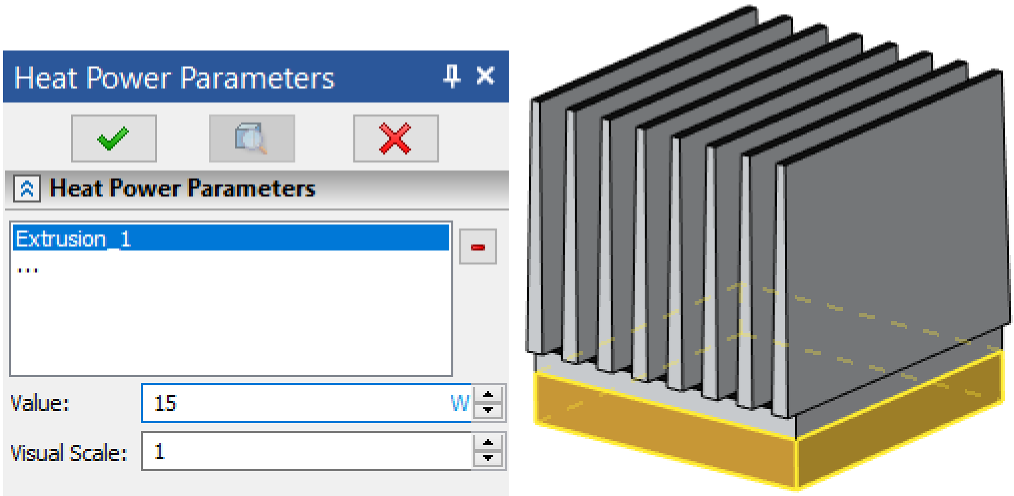
Three-dimensional model of the microchip with a passive cooling radiator



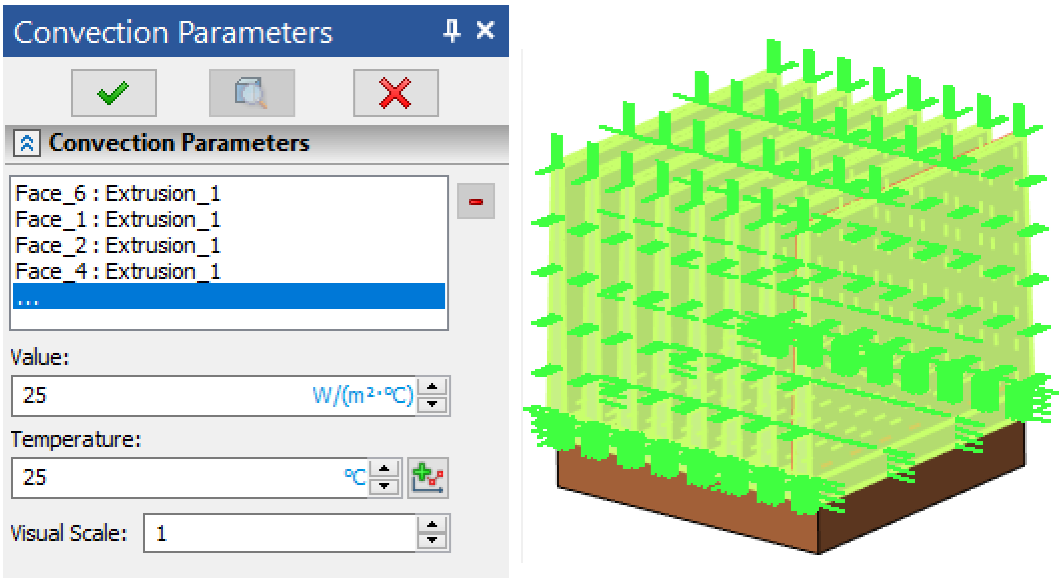
Resulting finite element mesh

Step 2. Applying boundary conditions

Let us specify thermal loads for the model. We will apply the «Heat power» load of 15 Watt to the volume of the microchip, and define the «Convection» boundary condition on the external heat-sinking radiator surfaces with the convection parameter of 25 Watt/(m2 h°C) and ambient temperature of (25°C). We can disregard in this study the heat exchange factor of mutual and ambient radiation, since their radiation contribution is vanishingly small at the expected temperatures (tens of degrees Celsius). Upon completing the commands of building the finite element mesh and defining thermal loads, we get a calculations-ready finite element model.



Defining **Heat Power** load

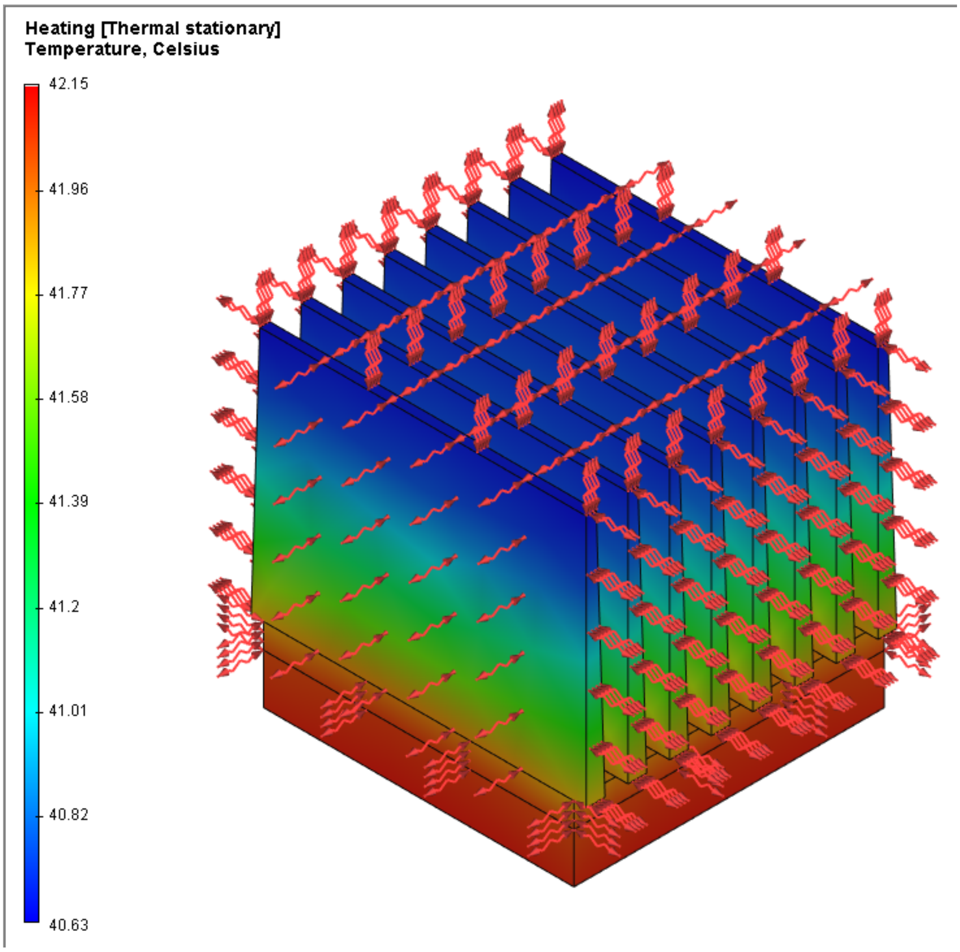


Defining **Convection** load

Step 3. Running calculations and analyzing results

We will start the thermal analysis by running the command **Analysis** > **Solve**. Use the **Linear** Interpolation mode on the **Thermal Analysis** tab to speed up the calculations.

The list of calculation results is displayed in the **Studies** window and can be accessed by the context menu in the calculation results window. The maximum temperature according to the heat and analysis results is 42.15 °C at the convection temperature equal to 25 °C. We will then edit the convection temperature using the **Edit** command of the studies tree context menu, setting the operational ambient temperature to its upper limit (55°C), and then rerun calculations. We will obtain the maximum temperature of the microchip equal to 72.15 °C. The conclusion is the radiator does fulfill the required temperature condition for the device in the entire specified range of the device's operational temperatures. The study is complete.



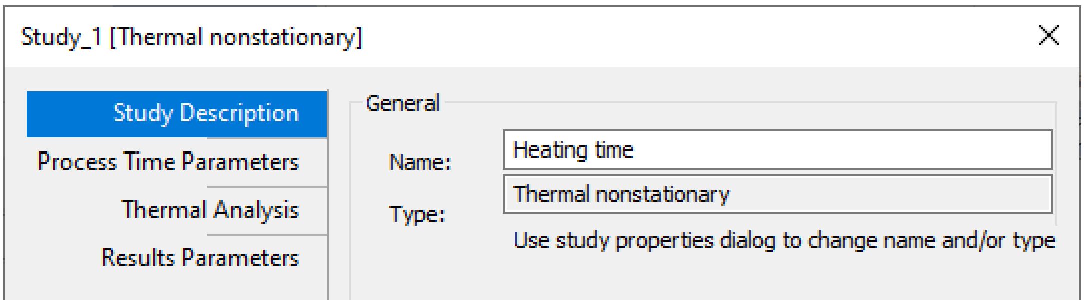
Transient Mode

Calculating the Time of Heating up the Cooling Radiator. Transient Mode

Let us estimate the time required for the device to reach a steady thermal state. To do this, let's run a transient thermal analysis of the «microchip+radiator» system.

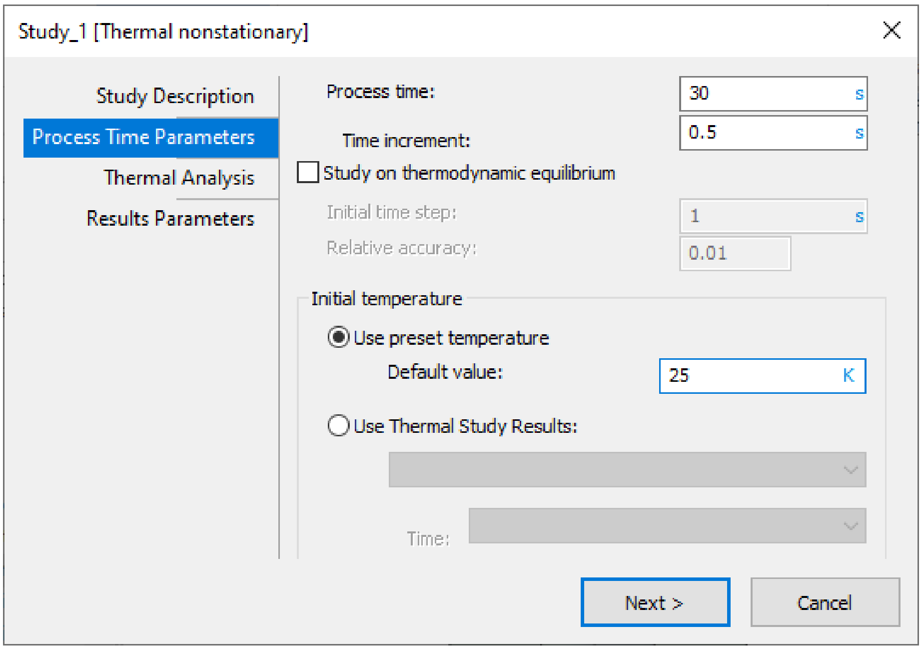
Step 1. Creating study's copy

We will create a copy of the original study in a steady-state thermal analysis using the **Copy** command of the studies tree context menu. On the **Study Description** tab of the study's parameters, change the study name to **Heating time**.



Step 2. Defining parameters of transient analysis

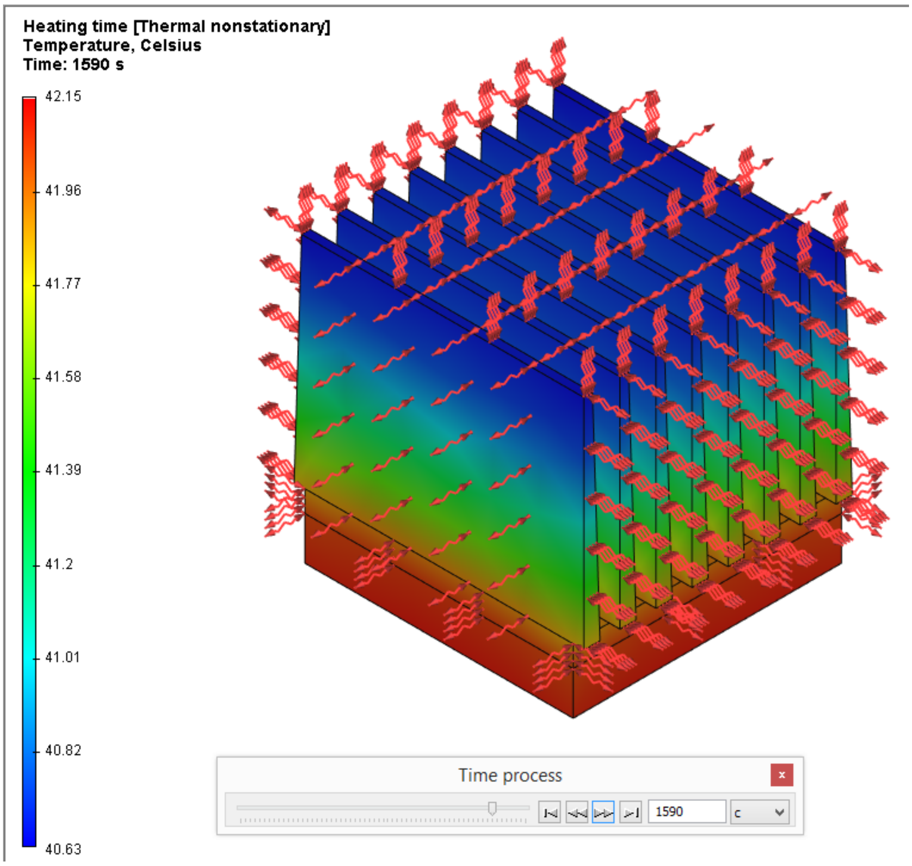
We change the study type to the "Nonstationary Thermal Process" using the **Edit** button from the context menu of the 3D model window. On the **Process Time Parameters** tab of the thermal analysis parameters, we set the finite time – the modeling time of 30 minutes and the modeling step of 0.5 minutes. We activate the **Use preset temperature** flag and enter temperature 25 C.



Defining calculation parameters of transient heat analysis – time and initial temperature

Step 3. Running calculations and analyzing results

After the completion of calculations, you can examine results at each time step. To view such results, we use a floating **Time process** bar that allows the user to quickly switch to the time instant of interest using a slider. With the help of these tools, we determine that a heating of the microchip to the temperature of 42.15°C will occur after approximately 27 minutes.



Result of thermal analysis at time 1590 sec (27 min.)

Then we change the ambient temperature in the study parameters and **Convection** load to the 55°C. We found that the microchip heating to the temperature of 72.15°C after the same 27 minutes.

Calculating Time of Cooling down the Cooling Radiator. Transient Mode

Now, let's evaluate the time required for the device-cooling radiator to cool down after an extended work.

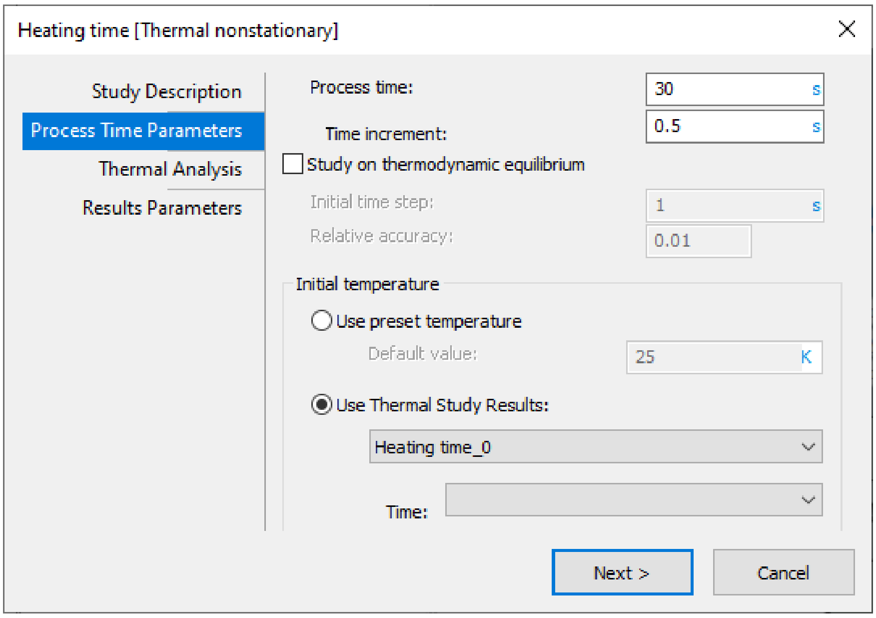
Step 1. Creating study's copy

Let's create a copy of the study of the **Nonstationary Thermal Process**. A **Study Copy** dialog appears when creating a copy of the study. By clearing the flag «**Create Copy of Mesh**», the different studies will be made to use *the same* mesh. This copying mode provides *identity* of finite element meshes two or more studies. Let's call the new study **Cooling time**.

Adjust the boundary conditions of the «Cooling time» study. For this, delete the **Heat power** load from the **Cooling time** study by the studies tree context menu command **Delete**. Now, the **Cooling time** study is ready for defining calculation parameters.

Step 2. Defining parameters of transient analysis

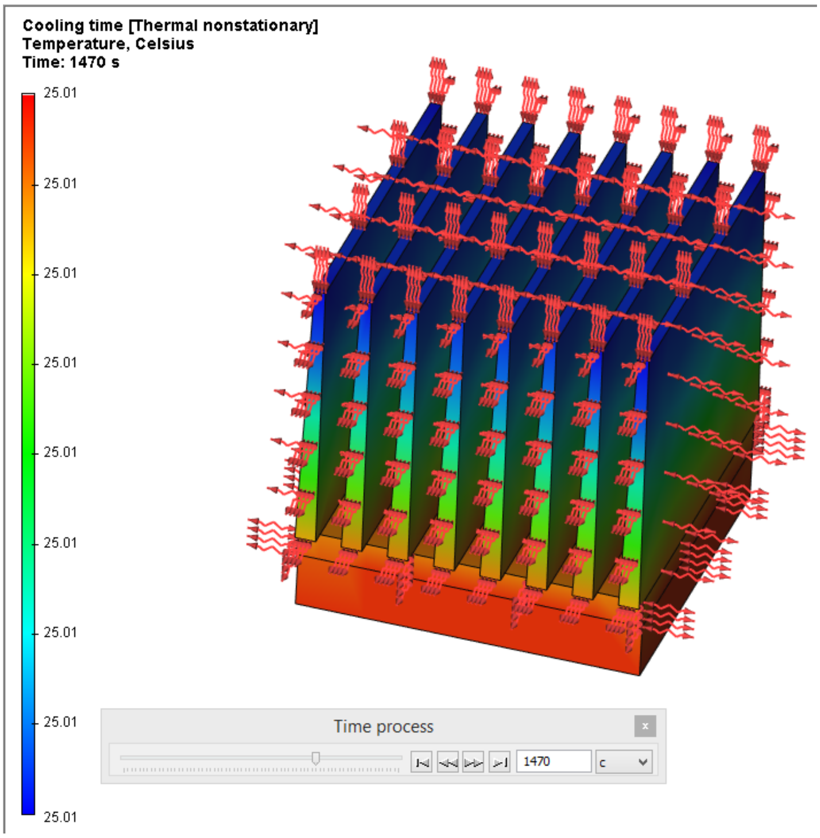
On the Properties tab we left the time analysis parameters without changes – the modeling time of 30 minutes, the modeling step of 0.5 minutes. Let's use already calculated thermal study results as initial model temperature. In our case, we select the **Heating time_0** study from the drop-down list.



Setting up study parameters for calculating cooling process

Step 3. Analyzing calculation results

Let's run calculations and analyze the results. Using the **Time process** bar, we can determine that nearly complete cooling of the radiator will occur in approximately 26 minutes after turning the device off. The device has been working in a stationary mode. The chip temperature will differ from the ambient temperature on 0.1°C.



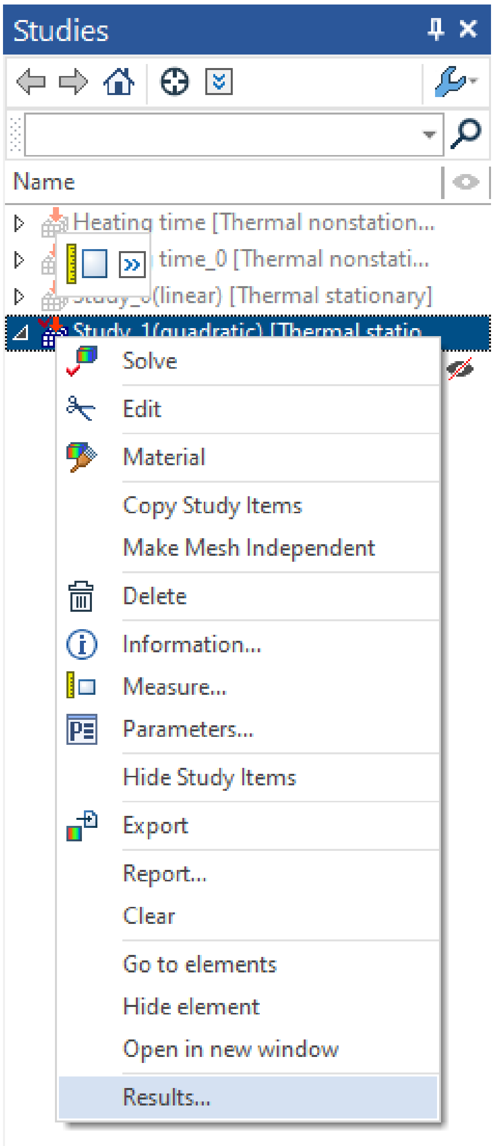
Device cooling calculation. Temperatures distribution at 1470 seconds of the calculation time.

Processing Results (Postprocessing)

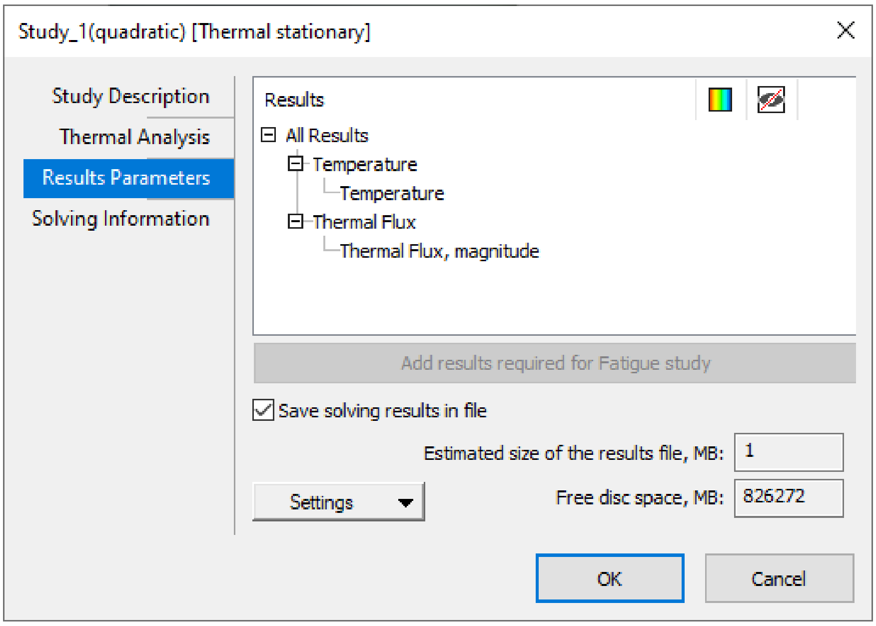
T-FLEX Analysis Postprocessor serves for comprehensive examination of finite element modeling results. A special feature of the T-FLEX Analysis postprocessor is its deep integration with T-FLEX CAD. Calculation results are displayed in a separate window, which is in many aspects of view management is similar to a T-FLEX CAD modeling window. Color-coding of the calculated model can show all results. There is also a tool for sampling exact values at any specific location on the model. When displaying results, the model can be shown in a scalable deformed state. When a special «animation» feature is enabled, one can dynamically view the pattern of changes in deformations from zero to the specified values.

Results settings

The list of finite element calculation results available for viewing is displayed in the studies tree, in the «**Results**» folder. The list of results to be displayed in the studies tree is set up using the «**Results...**» command of the context menu by on the name of the selected study. This command calls the dialog for setting up the results to be displayed in the tree.



Invoking **Results** command



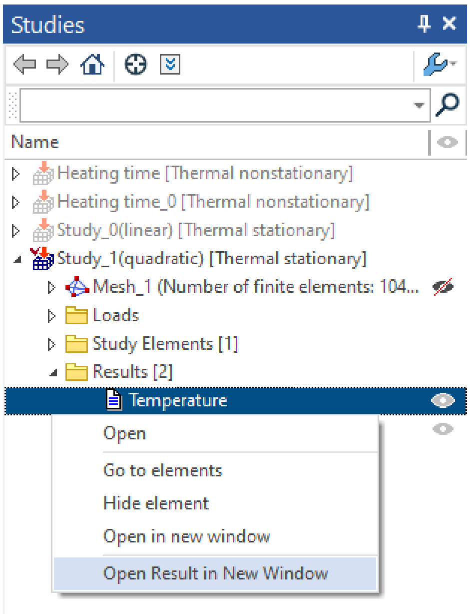
Dialog for customizing the results list

Calling up results

There are several ways to access results for viewing:

1. Double-clicking on the result's name in the studies tree opens the Postprocessor window with the selected result.

2. Accessing the context menu by the right clicking on the result selected in the studies tree and using the command **Open** or **Open Result in New Window**. T-FLEX Analysis Postprocessor supports use of multiple windows. Several windows with different results can be opened simultaneously, as well as several windows with the same result.

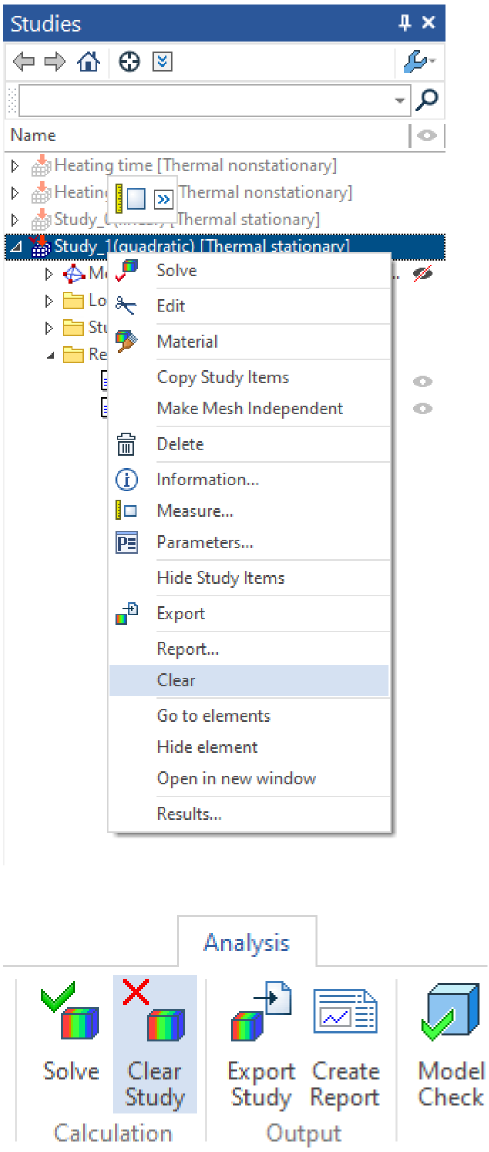


Context menu to open result

To delete a result from the studies tree, use the **Delete** command of the context menu accessible by right clicking on the selected result in the studies tree. The result will no longer be displayed in the studies tree; you can, however, still add it back to the tree, using the above-mentioned command **Result Parameters...**.

> You do not need to rerun study calculations when adding/deleting results in the list.

To actually delete all results (thus making the study «unsolved»), use the command **Clear Study**, accessible for the active study from the main menu or command **Clear**, accessible from the context menu by right clicking on the name of the selected study in the studies tree.



Invoking command for deleting all calculation results

Results of a study's calculation and meshes can be stored together with the model in a *.grb file. The user shall be aware, however, that storing those data increases the file size considerably. If you need to achieve a minimum file size, we recommend clearing calculation results in all studies before saving the mesh. In this way, the boundary conditions are left unaffected. Upon the next opening, you would have to create the mesh again and run calculations.

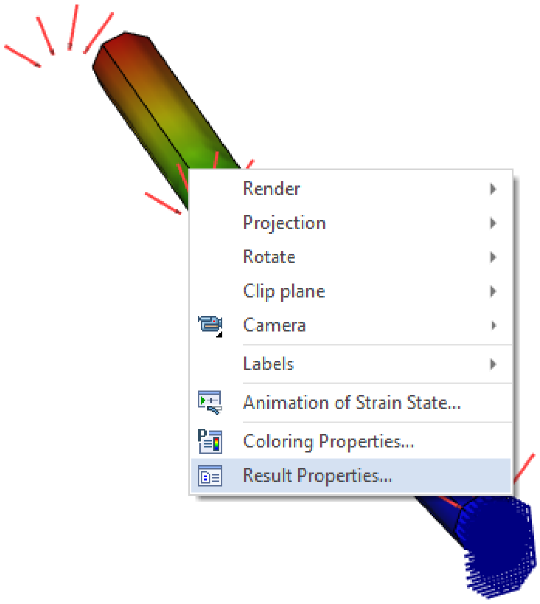
Topics in this section:

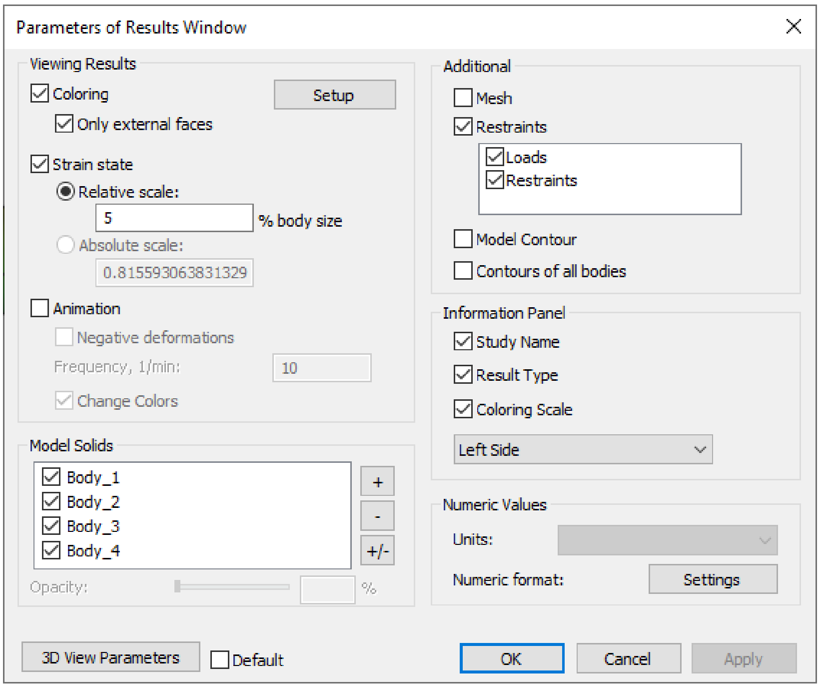
- Customizing Calculation Results Window
- Color Scale Setup
- Analysis Results Tab Commands

- Use of Sensors for Analysis of Results
- Using Graphs for Analyzing Results
- Resultant Value
- Record Animation
- Construction Section Views
- Generating Reports

Customizing Calculation Results Window

The viewer's settings are accessed by double-clicking  in the results viewer window or from the context menu.





Dialog for customizing parameters of calculation results window

Parameters dialog of the calculation results window has five groups of settings.

"Viewing Results" group

The **Viewing Results** group provides the following controls:

Coloring. This flag toggles the mode of colored rendering of the calculated model. Coloring is done according to the properties of calculation results and settings of the color scale. A color filling some area in the model corresponds to a certain numerical value. Color scale setup is done in a separate dialog (see below).

Only external faces. Enables the mode of showing the mesh on the surface only. When the flag is off, the entire volume mesh is displayed.

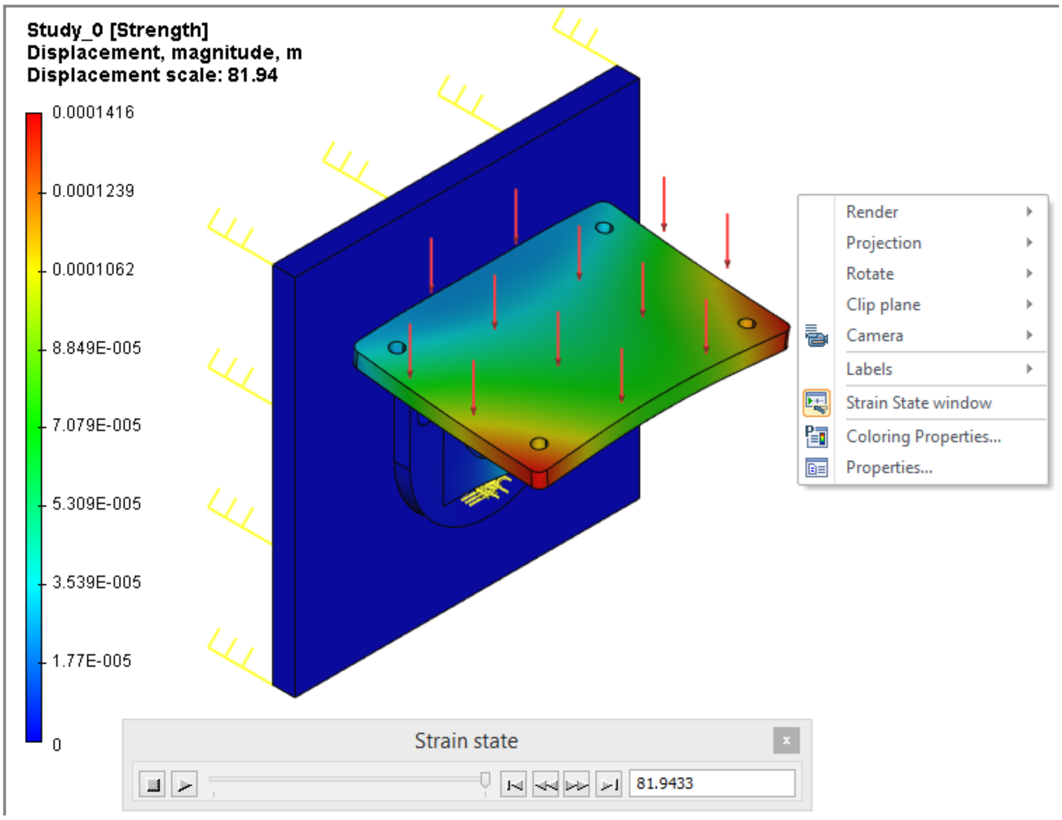
Strain state. This flag controls the way of rendering the resulting finite element model: it can be shown either in the deformed state, or as the original.

Scale. Sets the scale of deformations for the calculated model. It can be defined in terms of a relative or absolute value.

> Initially, the scale is picked up by the system automatically, but the user can change it as desired.

The **displacement scale** is a visual parameter in the results window, which characterizes the deformed state taking into account the scaling.

Animation. This flag turns on the animation mode of the postprocessor window, in which the deformation values smoothly vary from zero to the calculated value. To control animation and the deformed state rendering, one can also use a floating bar, that can be accessed from the context menu in the postprocessor window (the command **Strain state window** or **Time process window**, depending on the type of the result).



Use of the floating bar for controlling deformed state display and animation

Frequency. This parameter sets the rate, at which the full animation cycle completes. The number stands for the fraction of a minute, in which the full animation cycle completes.

Variable Coloring. Controls color variance during animation. Changes in colors can be synchronized with changes of the deformed state – from zero to final values. When showing negative deformations, the colors are not inverted.

Negative Values. This option enables the mode, in which the results displayed during animation first reach zero, and then go to the values equal to the calculated ones, but with the opposite sign. This creates the «vibration» effect of the calculated model, as if the load were periodically changing its sign to the opposite.

Solids lets you manage the bodies in the Postprocessor window, which are part of the assembly model, when evaluating assemblies. The user can tune of the flags corresponding to one or many parts of the assembly, after which those will no longer be displayed in the postprocessor window. By using the **Opacity** control, you can also manage transparency of the assembly parts displayed in the calculation results window. These tools help visualize the result fields inside the assembly model, by temporarily hiding obstructing objects.

Display group of parameters provides control elements, which define the visibility of auxiliary images around the calculated model for better results interpretation.

Mesh. Controls visibility of the mesh facets in the calculation results window.

Loads/Restraints. Controls visibility of all boundary conditions employed in the current study. The list of boundary condition types is displayed in a separate window. Visibility of each element in this list can be controlled individually. At the right of the window, there are buttons for managing the list elements. Using those buttons, you can enable (+), disable (-) or invert (+/-) visibility of all boundary conditions.

Model Contour. By enabling this flag, you make the contours of the original body subjected to the calculation appear as dotted lines in the calculation results window. This capability can be helpful for comparing the deformed state of the model with the original.

Contours of all Bodies. Upon enabling this flag, the rest of the bodies in the 3D scene, that were not subject to the calculations, are also displayed in dotted lines in the calculation results window.

«Information Bar» group

Information Bar group of parameters contains controls for adjusting the amount of displayed textual and graphic information:

Coloring Scale. This flag enables visibility of the color scale for more intuitive interpretation of the calculation results. The scale range and colors can be customized (see «Color scale setup» below). The scale is displayed in the left or right of the calculation results window. It depends on parameter chosen in pull-down list.

Study Name. This flag enables the display of the current study name in the calculation results window.

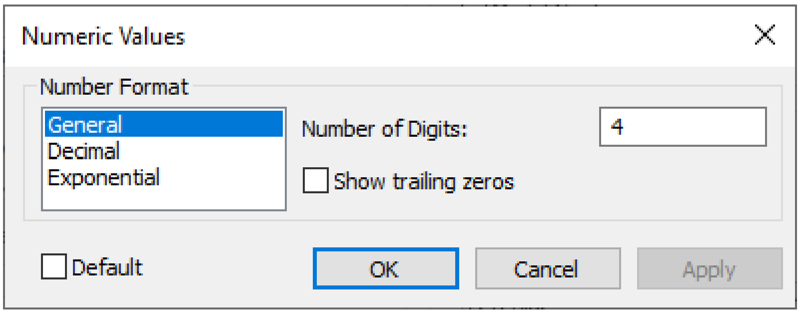
Result Type. This flag enables the display of calculation type name.

"Number Format" group

The **Number Format** group provides the following controls:

Units. Serves to define the measurement units (meter, inch, millimeter) to be used for displaying the result.

Format of Values parameter sets the format of the scale numbers for the viewing convenience - it can be decimal, exponential or general (mixed). The general format represents values up to the 1000 in the decimal format, those greater than 1000 - in the exponential format. The number of significant digits for the exponential format and the number of decimal digits for the decimal format are set in the provided field at the right. "Extra" zeros can be automatically discarded by disabling the flag "show trailing zeros".

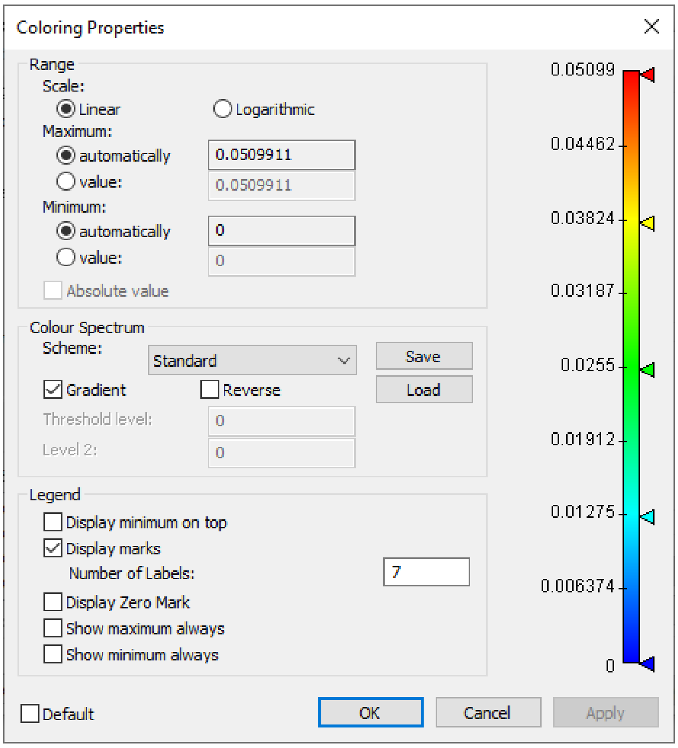


Color Scale Setup

The color scale settings dialog can be accessed from the parameters window of the calculation results (the button **Settings**), or from the context menu, accessible in the calculation results window by clicking . Perform color scale settings in order to associate the desired colors with the obtained values. The system does the initial setup automatically. It evenly distributes the standard palette (spectrum) of five colors between the maximum and the minimum values obtained in the current calculation.

«Range» group

Range group of parameters sets parameters of distributing numerical values over the color scale.
Linear/Logarithmic. The logarithmic scale is used by default for displaying the "Factor of safety" result. This is done in order to get a more detailed color picture in the most important subrange – where the factor's values are near 1. The user can also switch the display mode to the linear scale for this result. For other results, the linear scale is always used.
Maximum. This interface item serves to set the correspondence between the maximum value and the topmost color on the scale. By default, the maximum value is set, but the user can enter one's own value of interest in the respective field.

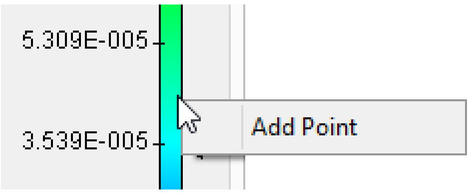


Minimum. This allows defining the correspondence between the minimum value and the bottommost color on the scale. Otherwise, this interface item is analogous to the previous one.

Absolute value. Enables the viewing mode of rendering absolute values in the nodes (disregarding the sign). This is an auxiliary mode. It can be used for analyzing results in the case when you are interested in the magnitudes of values of displacement components or other measures.
Input of the maximum and minimum values can be used for a custom setup of results rendering. For example, to display the «Factor of safety» result, you can limit the maximum value of the safety factor, in order to achieve a more intuitive result picture in the postprocessor window.

«Color spectrum» group

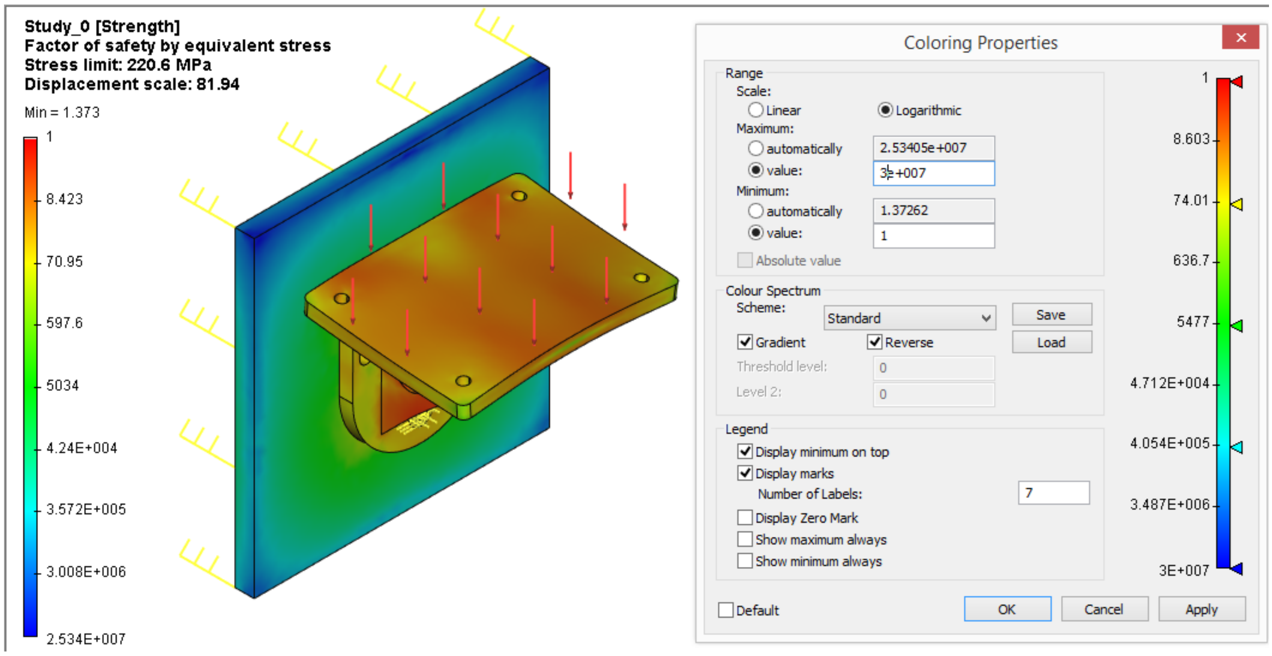
«Color spectrum» group of parameters allows adjusting the number of colors in the color scale.
Scheme. There are seven three predefined settings and one custom.
Standard. This option displays the scale of five main colors.
Shades of gray. This option enables a grayscale display.
Full Spectrum. When using this option, the scale appears as a rainbow of seven main colors.
Besides that, there are the following color schemes: **Maximum**, **Minimum**, **Threshold**, **Range**.



Custom. You can manually set up the color scale, and then save your choice into a special file for future quick loading (a file with the extension *.col). The setup is performed in the right hand side part of the dialog, where the color image of the scale is displayed. On the right of the scale, there are several triangular tags marking places for fixed colors on the scale. Using those tags, you can set a new place for a color on the scale. To move a tag, depress the on it and while holding down the button, drag. To create a new tag with a new color, perform at the right of the scale. A standard Windows dialog for defining color will appear. To delete a tag, drag it beyond the color scale. To change the color assigned to the respective tag, double-click . To access this scale settings, you can also use buttons **Save** or **Load** in the “Colour Spectrum” group.
Gradient. This flag (enabled by default) serves to set smooth transition from one color to another in the color scale.
Reverse. Serves to invert the color scale.

"Legend" group

The **Legend** group provides the following controls:
Display minimum on top. This flag reverses the values scale. This mode is used by default when displaying the "Factor of safety" result in the static analysis. This is convenient for displaying critical values that are close to 1, in Red.
Display Marks. When this flag is enabled, the marks with numerical values will be displayed on the color scale in the results window.
Number of Labels. Sets the number of displayed marks.
Display Zero Mark. Serves to enable the display of the zero value mark on the color scale.
Show maximum always. When this flag is enabled, the maximum value string is displayed in the calculation results window.
Show minimum always. When this flag is enabled, the minimum value string is displayed in the calculation results window.



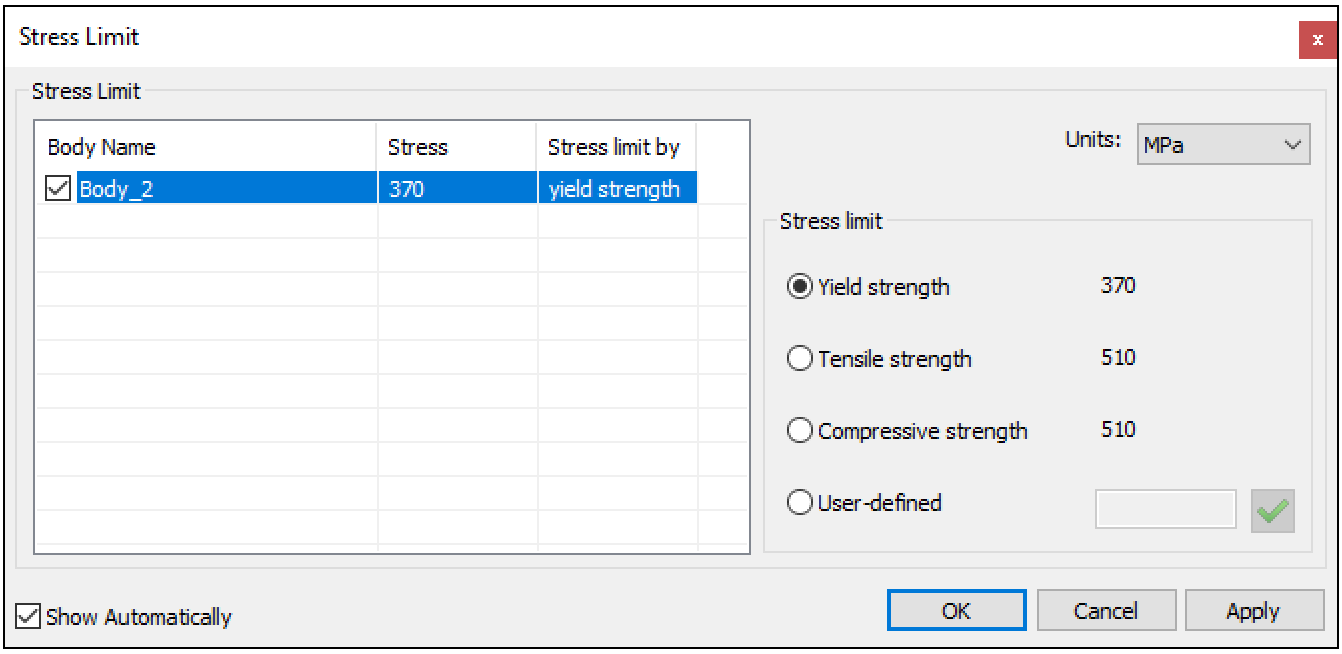
Setting the range of scale values for displaying results

Analysis Results Tab Commands

The Analysis results tab becomes active in Ribbon when you switch to viewing results. The tab includes the following groups and commands.

"Parameters" group

Data Parameters. The command calls the dialog where you can enter limit stress values to determine the safety factor. For ductile materials, the yield strength can be the limit stress. For brittle materials (as well as for ductile materials), the ultimate tensile stress or ultimate compression strength can be the limit stress. The choice depends on the nature of the stressed state in the investigated area or you may enter user-defined limit stress value.



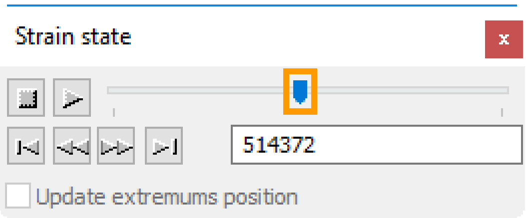
The safety factor shows the ratio of equivalent stresses to limit stresses.

Coloring Parameters. Calls the color scale setup dialog window. For more details switch to [Color Scale Setup](#).

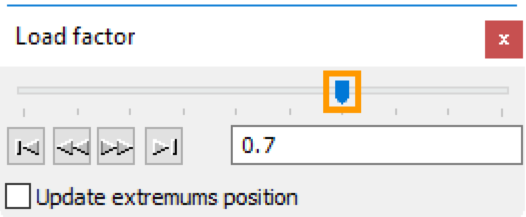
Result Parameters. Calls the result parameters setup dialog window. For more details switch to Customizing Calculation Results Window.

"Windows" group

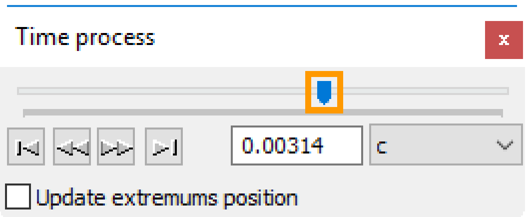
Animate Result. The command is active for stationary processes. Calls a dialog designed as a player which enables you to change the movement scale from zero to the value given in the color scale setup dialog. The dialog of the deformed state represents a background animation player.



Load Scale. The command is available for nonlinear tasks solved with gradual means. The command calls a dialog designed as a player. The player switches calculation steps creating animation effect (calculation steps are determined in the study calculation dialog in the tab Nonlinear).



Time Process. The command is active for dynamic processes proceeding with time. Calls a dialog designed as a player which enables you to change the time lapse of the process from zero to the value given in the study parameters (the process calculation finish time). The time process dialog represents a background animation player.



"Visualize" group

Mesh. Show/hide the finite elements mesh of the model.
Restraints and Loads. Show/hide all the boundary conditions (stresses, fixes).
Labels of Restraints and Loads. Show/hide labels of boundary conditions (stresses, fixes).
Sensors. Show/hide sensors.
Extermums. Show/hide extremums.
Contours of all bodies. Show/hide the contours of initial bodies. It is convenient for understanding the distortions caused by deformation.
All model bodies. Show/hide the contours of the bodies not involved in the task.
Coloring Scale. Show/hide the color representation (according to scale) of the calculated fields.
Deformed State. Show/hide the deformed state (taking into account the representation settings in the **Windows** group).

Symmetrical Result. In case Symmetry limit was applied, you can show/hide symmetrical representation of the calculated body.

Animation. Enables/disables post-processor window animation mode, which shows gradual changes of the deformation value from zero to the calculated value. For more details switch to Customizing Calculation Results Window.

"Output Data" group

Record Animation. Calls the animation creation dialog. For more details switch to “Record Animation”.

Create Report. Calls the report creation dialog. For more details switch to “Generating Reports ”.

Use of Sensors for Analysis of Results

Sensors can be created only for active problem of the finite element or dynamic analysis with the help of the command:

Icon	Ribbon
	Analysis > Conditions > Sensor
Keyboard	Textual Menu
<3MD>	Analysis > Sensor

In the T-FLEX Analysis there is a possibility to probe the results of finite element analysis in a certain point specified by the user with the help of sensors and visualize the results with the help of plots.

Sensor is designed for extracting results of calculation of a finite element analysis problem in a specified by the user point.

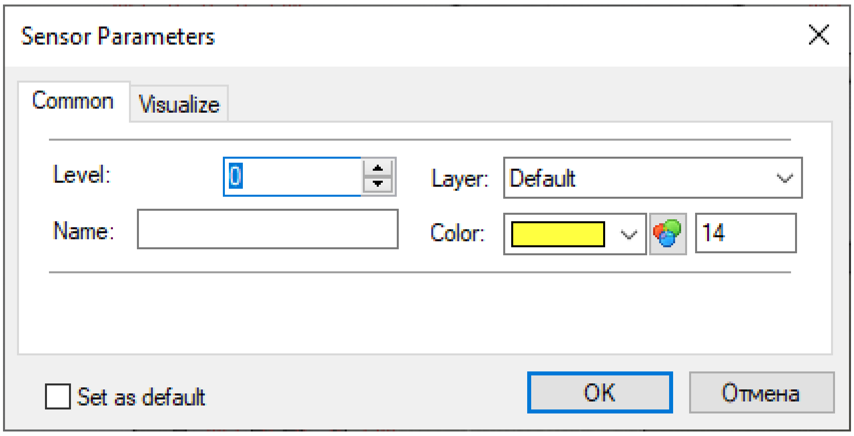
Sensors are also used for extracting control data when solving optimization problems using the results of the finite element calculations. Sensors can be installed on 3D nodes or vertices of the body. After the sensor is set up, the various data from it can be read by the «**Measure**» command and written to a variable (for example, to store control data in optimization problems based on data of finite element analysis). Also, the sensor can serve as a permanent label for displaying the value at a given point in the results window. This can be useful if it is required to measure the result (for example, the stress) at a point or at number of points with the given coordinates.

In the parameters window the sensor type is defined as a FEA point since the sensor is used for the problem of the finite element analysis.

For specifying a point, where the sensor will be created, use the automenu option:

<V> Select point

This point can be a vertex on a profile/path, vertex on a body, center of a curve/edge, middle point of an edge, center of a sphere/torus or a 3D node.



In the parameters window other parameters of the sensor can be specified. As all T-FLEX objects, the sensor has general parameters (name, color, layer, etc.). In addition, visualization parameters and the name can be defined for the sensor. By default, the sensors are given the name Sensor_n, where n is a sequence number.
The size of sensor visualization sphere in the 3D scene window is set on the **3D > View** tab in the dialog, which can be invoked with the **Set Document Parameters** command.

In the «Size» parameters group it is required to select the size of Coordinate systems. To enforce these changes it may be required to regenerate the 3D model with the help of the **Full Regeneration** command, **<Shift-F7>**.
After creation of the sensor it will be displayed in the study's tree in the **Sensors** folder:

To do that, first, the user has to solve the finite element analysis problem (even if the problem has already been solved), and then from the context menu, invoked on selecting **Sensor** with a right mouse button in the window **Studies**, call the command **Measure**.

Measure dialogue

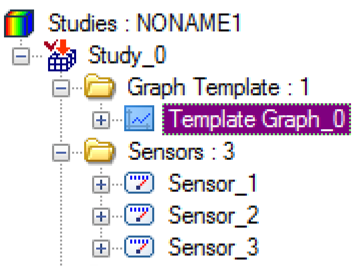
- From the appeared parameters dialog, in the Table **Property**, the user can select the desired type of result and view its value.
- Clarification for the keywords located in the first column of the table is given in the field **Definition** below the table.
- Selected value can be saved with the help of the option **Variable**. To do that, select the option **Create** in the group **Variable**, specify the name of the variable and press the button **Apply**.

Using Graphs for Analyzing Results

Graph template is created with the command:

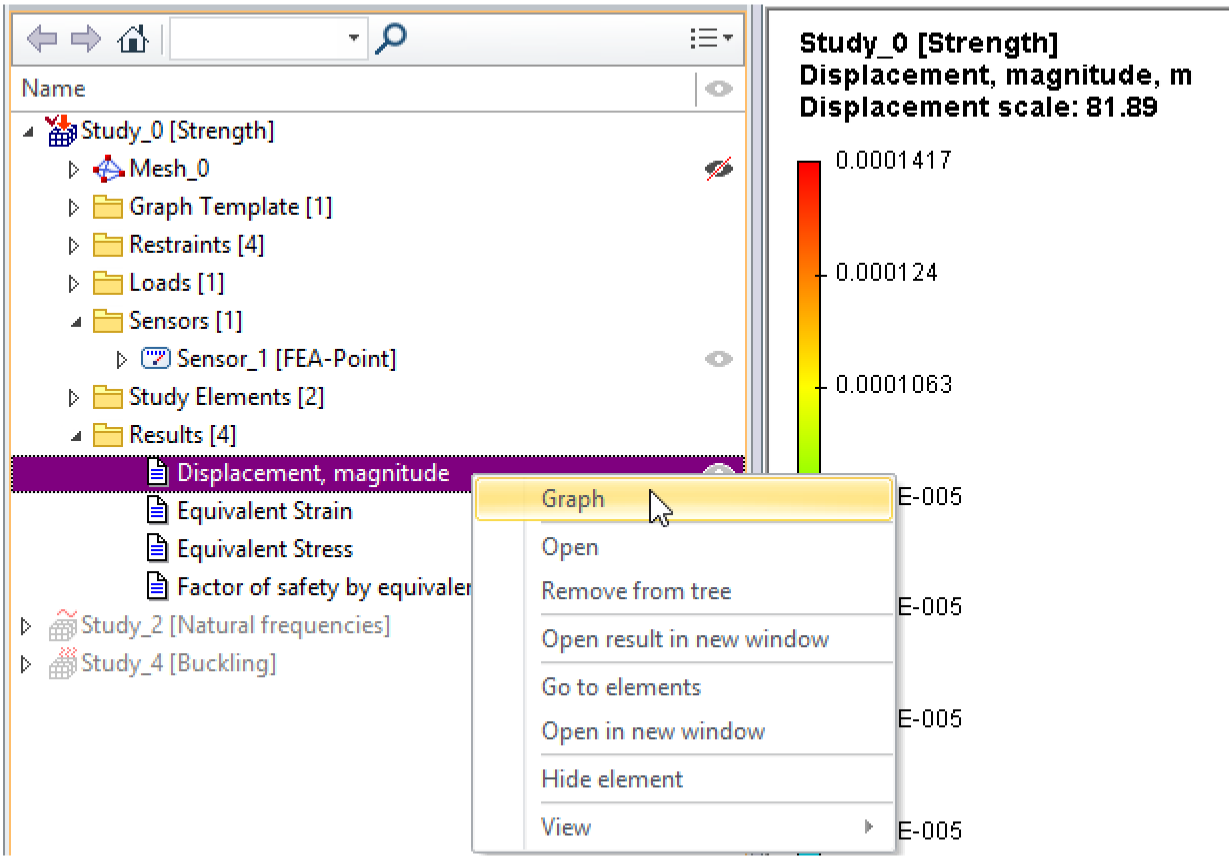
Icon	Ribbon
	Analysis > Conditions > Graph Template
Keyboard	Textual Menu
<3GT>	Analysis > Graph Template

Graphs are used for visualization of the data that is measured from several sensors in the form of curves and for creation of relationships. For example, it is possible to measure the temperature at various distances from the cylinder's center by creating several sensors located on the same radius-vector and plot the temperature as a function of radius or construct the plot showing how the stress changes in the given cross-section.
To create a graph, it is required to have several sensors and, based on these sensors, create a graph template, i.e., register the sensors in the given template. At the same time it is important to note the order in which the sensors are located and, if necessary, position the sensors in the order of increasing distance to provide for monotonicity of the curve (exclude self-intersections).



After the graph template has been created it will be displayed in the studies tree in the «Graph Template» folder, and the number appearing right after the colon is equal to the number of graph templates created for the given study.
Inside the study there can be several graph templates, for example, one template, created for a series of sensors located along the vertical axis of a cross-section, another template – along the horizontal axis.

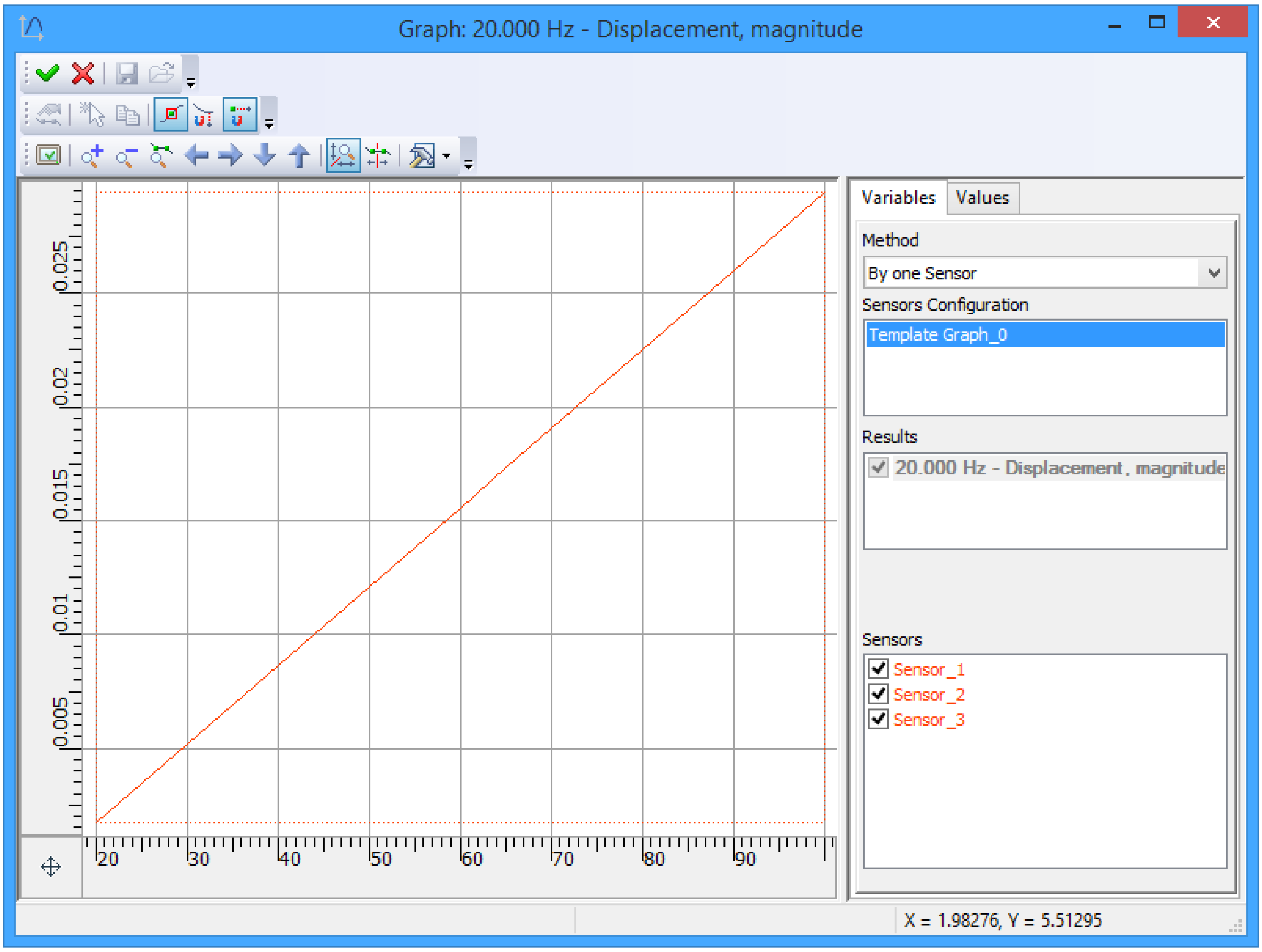
To create a graph based on a graph template, it is required to open one of the results with , invoke the context menu (with the right mouse button) for a specific result in the study's tree and select the **Graph** option.



After that the graph for the given type of the results will be created. In the «Method» field the user can select one of the graph construction methods: by several sensors or by one sensor:

- The graph construction method by several sensors is used when a single result of certain type was specified (displacement, stress) and for this result there is a series of measurements from the sensors located at different points – a spatial dependence of the quantity is constructed.
- The graph construction method by one sensor is used when a multiple result was specified (for example, a series of temperature values at a point at different moments of time in the heat transfer analysis of non-stationary processes or a series of vibrational acceleration values at a point for different forcing frequencies) – dependence of the quantity on time or frequency is constructed.

The graph template, with a corresponding set of sensors related to the template, can be selected from the graph view window in the «Sensors configuration» field, in which the graph templates created in the given study are listed.



In the **Sensors** field the list of sensors related to the given graph template is shown.
In the **Results** field, the multiple results of non-stationary heat transfer analysis or forced vibrations analysis are listed. For a single result, the name of the result is shown.
In the **Values** field the coordinates of points of the graph are shown – this data is linked to the values from the sensors and cannot be edited but can be copied to the clipboard with the subsequent insertion into another graph or electronic table. It should be noted that graphs in the problems of finite element analysis, including dynamic analysis, are the T-FLEX CAD objects of the type «**Graph**», which are created with the help of the command:

Icon	Ribbon
	Parameters > Tools > Graphs
Keyboard	Textual Menu
<PL>	Parameters > Edit graph function

Therefore, all operations can be applied to them as to ordinary graphs, for example, the graph() function can be used for reading intermediate values (interpolation). For more accurate interpolation it is convenient to select the graph type «smooth curve» or copy the points together with the coordinates to the new graph of the type «smooth curve». More detailed information on this can be found in T-FLEX CAD Help.

Standard sequence of actions when creating a sensor:

- Initialize the command  **Create sensor**.

- Select 3D nodes, vertices for specifying sensor location.
- If necessary, specify the sensor name and other general parameters.
- Complete the command.

Standard sequence of actions when creating a graph template:

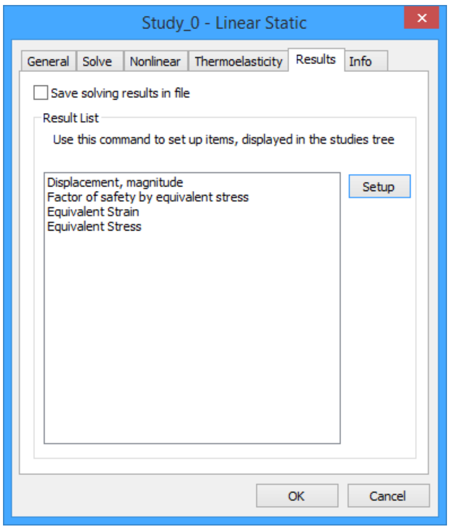
- Initialize the command **Create graph template**.
- Select all or only the required sensors, if necessary correct their order while observing their location in the 3D scene window.
- If necessary, specify the graph template name and other general parameters.
- Complete the command.

Standard sequence of actions when creating a graph:

- In the study's tree select a single or multiple result in the **Results** folder and invoke the context menu for it (right mouse button).
- In the context menu select the **Graph** option.
- In the graph edit window that appears select the graph construction method in the **Method** field and graph template in the **Sensors configurations** field.
- Complete the command with the button.

Resultant Value

This command allows us to sum up the values of reaction forces at the nodes of finite element mesh on the selected faces, edges or vertices of the finite element model. This command is available for the results: «Reaction Force, magnitude», «Reaction Force OX», «Reaction Force OY», «Reaction Force OZ» and can be invoked from the context menu for the result in the study's tree. After this command is invoked, the 3D scene window appears in which we can select the model's faces, edges, and vertices for which the resultant reaction force needs to be found. The command is used in the static analysis studies.
Display of the specified results in the study's tree can be enabled from the study's parameters window that is invoked with the automenu option .
The «Setup» button in the study's parameters window can be used to invoke the results settings dialog in which the desired results must be checked.



After the «Measurer» command from the result's context menu is invoked (click with a right mouse button on the result) it is required to select elements of the model for application of the load.

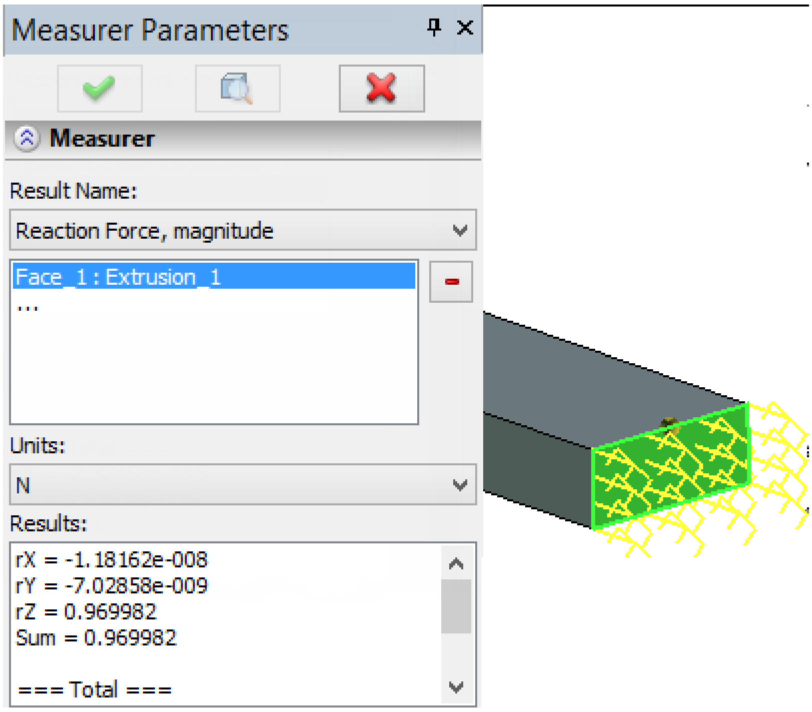
With the help of automenu, select faces, edges or vertices of the model being analyzed:

 <E> Select Element to Measure

The selected elements will be added to the list.

The user can select the units of measurement for reaction force – N, pounds, or kg.

It should be noted that in most cases the given command is used for measuring reaction forces at the rigid support.



Standard sequence of actions when measuring a resultant reaction force value:

- Enable display of the «Reaction Force» results in the study's tree.
- Select faces, edges, vertices.
- Specify units of measurement.
- Complete the command.

Record Animation

The command calls the dialog of recording background animation as a file.

Icon	Ribbon
	Analysis results > Output > Record Animation

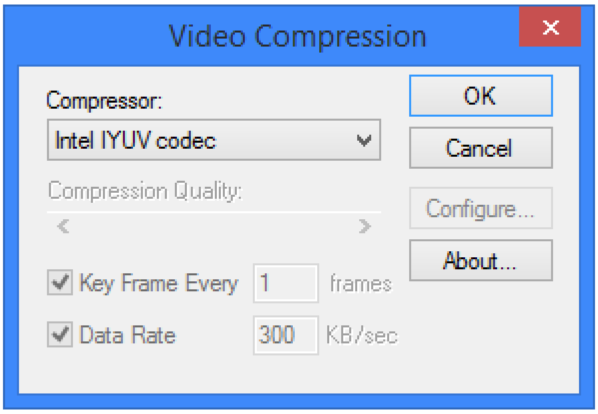
"Video Parameters" group

File Name. Set name and path ... to the animation file being created.

Frames per Second. Determines the number of frames per second in the animation being created.

Compression. A graphical button **Compression** calls the *.avi file compression parameters setup window.

Compressor chooses compression software. Compression Quality defines the file compression quality. Key Frame Every defines the number of frames between key frames. Data rate defines the transmission rate (Kb/s). The graphical button **Configure** outputs the dialog window for inputting corresponding settings of the chosen compression software.



Animation Parameters group. The parameters **Frequency**, **Change Color**, **Negative Values** are similar to the ones for background animation setup.

Frequency. This parameter sets the rate, at which the full animation cycle completes. The number stands for the fraction of a minute, in which the full animation cycle completes.

Change Color. Controls color variance during animation. Changes in colors can be synchronized with changes of the deformed state – from zero to final values. When showing negative deformations, the colors are not inverted.

Negative values. This option enables the mode, in which the results displayed during animation first reach zero, and then go to the values equal to the calculated ones, but with the opposite sign. This creates the «vibration» effect of the calculated model, as if the load were periodically changing its sign to the opposite.

View Point group

Use Current View Point. The animation will be recorded according to the current point of view at the stage.

Use Camera. This option is available in case there are cameras in the model. For more details switch to the «Cameras» section in the CAD help.

Dimensions group

Use Current View Dimensions. In case the flag is on, the content of the current 3D-view window is recorded. If the flag is off, the user can set his/her own width and height values of the image being saved.

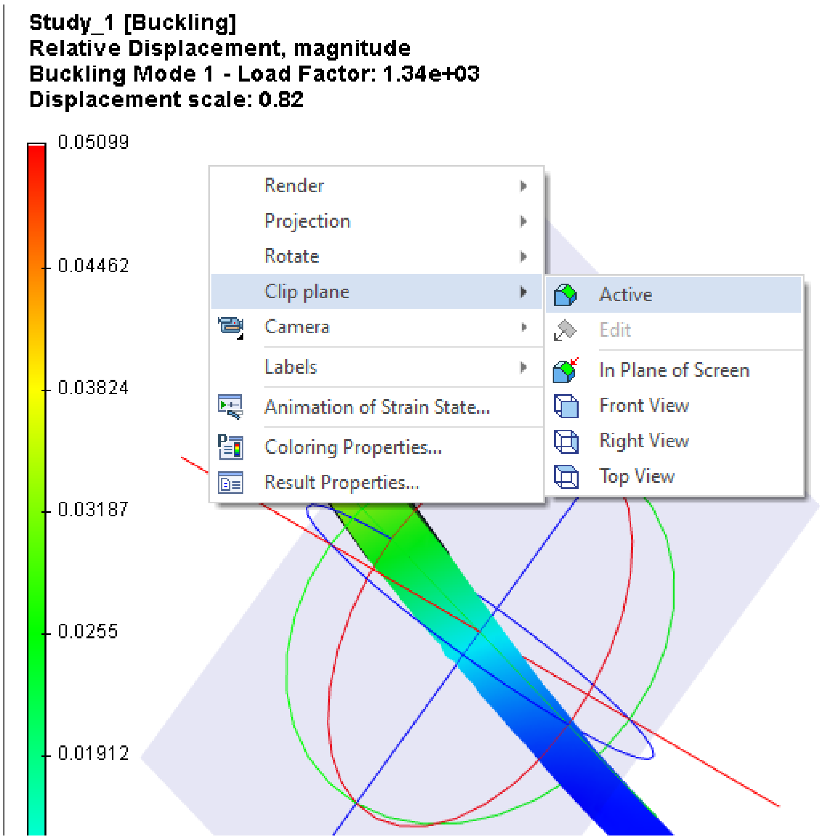
Show Video flag. When the flag is on, the created file is automatically opened for review after the animation is recorded.

Construction of Section Views

The T-FLEX Analysis allows the user to construct sections of finite element mesh by a given user-defined plane.

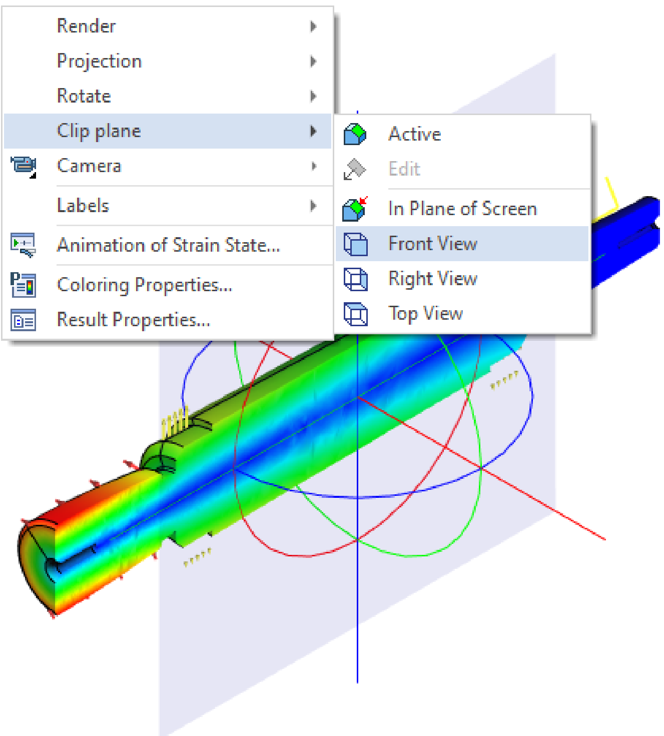
The section of the finite element mesh can be constructed only when the finite element analysis problem is solved successfully.

To construct a section, the user has to select the command **Clip Plane** > **Active** from the context menu that can be invoked by pressing the right mouse button in the **calculation results window**.



Then, the section plane has to be placed at a required position.

To rotate the plane with respect to axes of the local coordinate system (LCS), control objects: , , are used. The control objects: , , are used for translation along the axes of the LCS. Color of each control object matches the color of the axes of LCS plane. Initially, the axes of the local coordinate system are aligned with the axes of the global coordinate system.

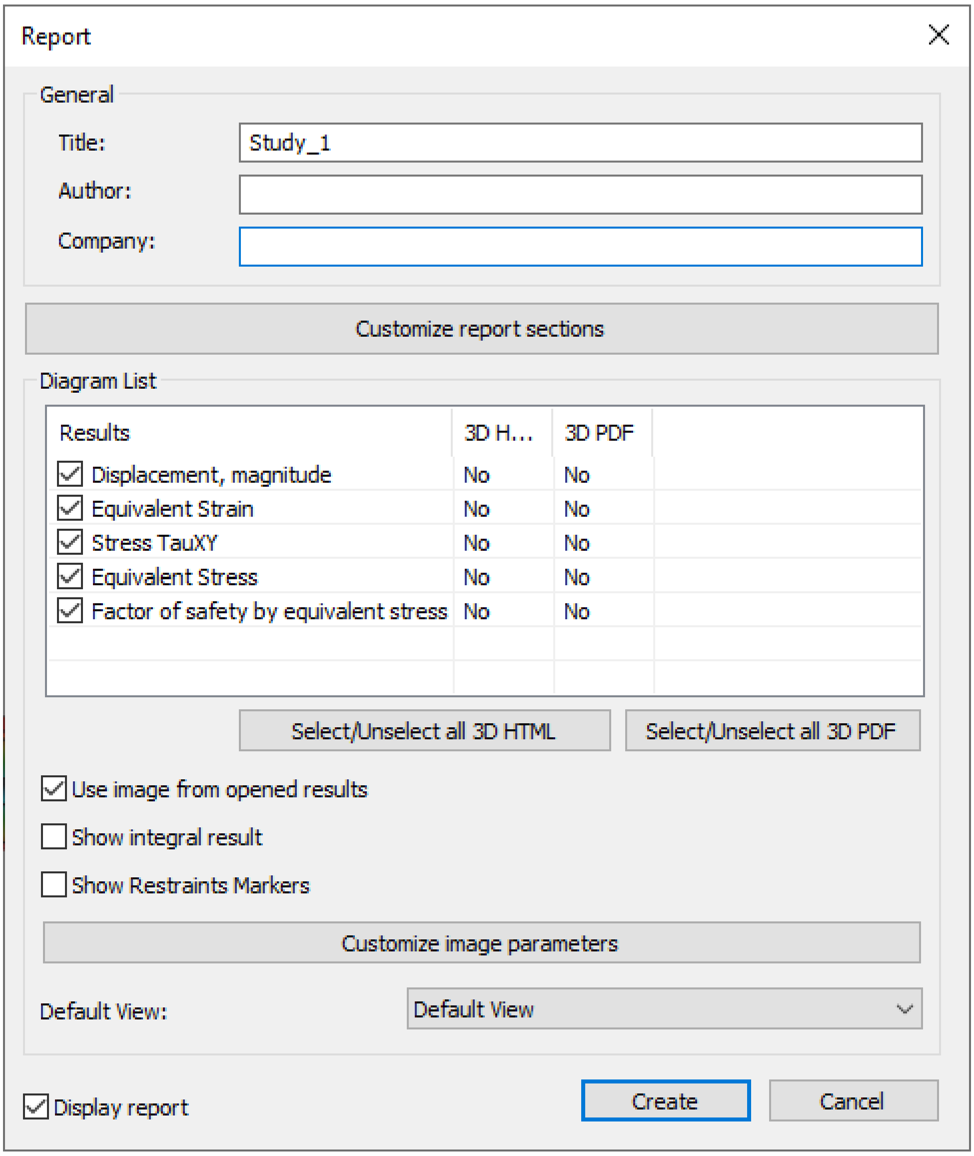


After fixing the position of the section plane, the user has to invoke the context menu, by pressing the right mouse button in the calculation results window, and select the command **Clip Plane** > **Active**. The section plane will be created.
The capability of constructing a section of the finite element mesh is very important in case when the user needs to know the solution inside the structure.

Generating Reports

The user can create electronic documents containing basic information about a calculated study, which are independent of T-FLEX Analysis. Reports are generated in the html format, so that viewing them is possible in any browser, for example, MS Internet Explorer or MS Word. To create a report of the active study, use the command **Analysis** > **Create Report**.

The report settings dialog can also be accessed from the context menu by right clicking  on the name of the selected study, via the command **Create Report**.
A report contains basic information about the model, materials, and the computational finite element mesh, as well as colored result diagrams, which are displayed in the studies tree or opened in the calculation results view windows at the moment.
Let's review the main controls of the report generation dialog.

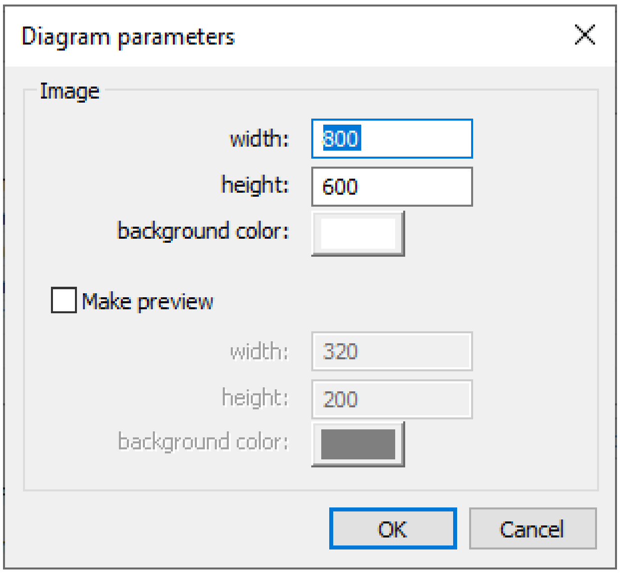


General group contains information on the name of the study, for which the report is being generated (**Title**), information about the report creator (**Author** - by default, the information is accessed from the document parameters), and company information, which is also accessed from the document parameters by default.

Diagram list control lets the user check-mark the result types, whose graphical images will be added to the generated report.

Button **Customize report sections** above the **Diagram list** group allows to choose sections for adding into report. Button **Customize image parameters** is provided under the list of diagrams. You can use it for calling the dialog for defining diagram display parameters. Here you can specify the picture size in pixels, and the background color. You can also enable creation of preview and set up its image. In this way, the main report document would include reduced-size images of result diagrams, with the full-size images being accessible by clicking  on the small image, when viewing the report file, for example, in the Internet Explorer.

You can use check boxes in the result table to enable/disable the creation of 2D image and 3D model results in VRML format with coloring and boundary conditions.



In the report, next to the picture, there will be available the link to the vrml model of the corresponding result. Note that for viewing vrml-model via the Internet Explorer, it is necessary to use independent plugin, displaying vrml-model (for example, Cortona Vrml Client, http://www.cortona3d.com or another analog).

Default View allows adding to the report the currently opened diagrams in their current orientation, the way they appear in the calculation results window. If there are no open result windows, or the control is not active, then the diagrams are added to the report in the default orientation («axonometric front»).

Report Templates

As we mentioned, study reports are generated using templates. A template is a common html document. The template that is installed with the system has the name TFA_common.html and is located by default in the program folder of the T-FLEX CAD installation. The contents of a template can be edited in a textual or html editor. A user familiar with html programming can make changes to the standard template, or create one's own template based on it, and then use the new template for generating reports on studies. The idea of generating a report by template is simple. The reports generator looks through the template text and analyzes its contents. The template contains special fields using reserved notations (so-called «tags»); such tags are replaced by the respective values of parameters of a study, for which the report is being generated. Table 1 lists the tags and their values used in the report template.

Table 1
List of Tags for Generating Reports

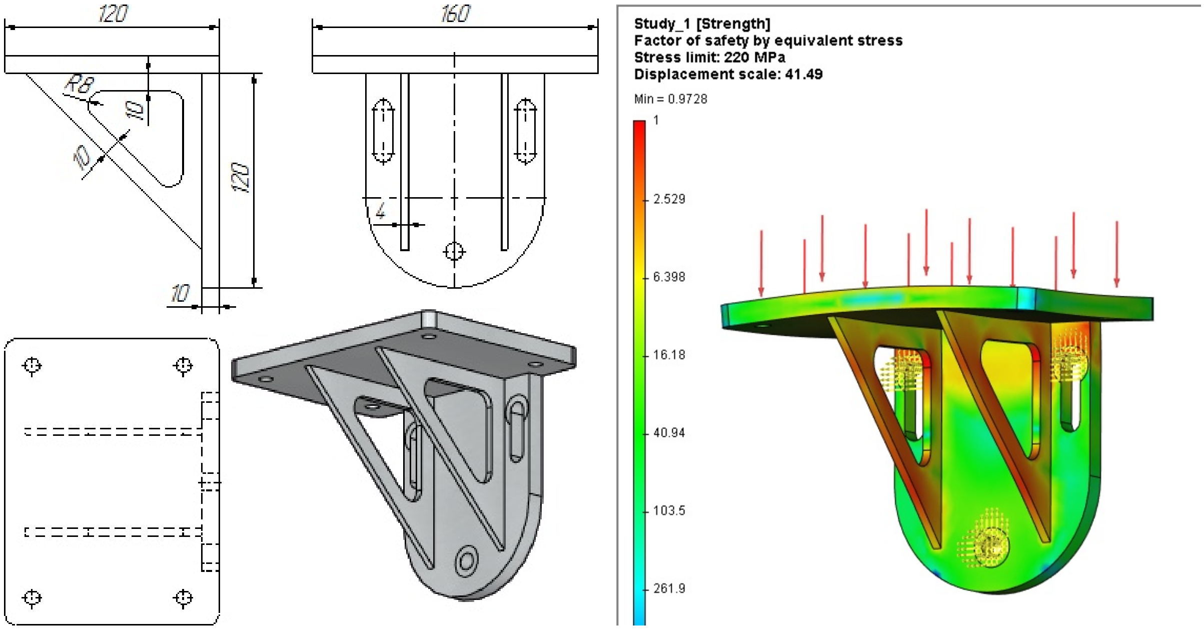
Tag	Tag value
$(TaskName)	Study title
$(TaskComment)	Comments to the study defined in the study parameters dialog
$(TaskType)	Study type
$(TaskAuthor)	Author
$(TaskCompany)	Organization
$(Date)	Date and time of generating the report
$(File)	The name and path to the model file
$(SolidName)	The name of the operation, for which the study was created
$(MaterialNumber)	Material ID number
$(MaterialElasticity)	Elasticity (Young's) Modulus
$(MaterialPoisson)	Poisson's Ratio
$(MaterialDensity)	Density
$(MaterialThermalCond)	Thermal Conductivity
$(MaterialExpansion)	Thermal Expansion Modulus
$(MaterialStress)	Allowable Stress
$(MaterialSpecificHeat)	Specific Heat
$(MeshName)	Mesh ID
$(MeshType)	Element type
$(MeshNodesNum)	Number of nodes
$(MeshElementsNum)	Number of elements
$(ConditionName)	Name (ID) of the boundary condition
$(ConditionParent)	The model element, for which the boundary condition was created
$(ConditionParameters)	The boundary condition value (load, temperature, etc.)
$(HTML_TR.Result)	Results
$(ResultName)	Name of the result
../../../$(ResultBitmap)	Link to the result image

Example of Interpreting Result

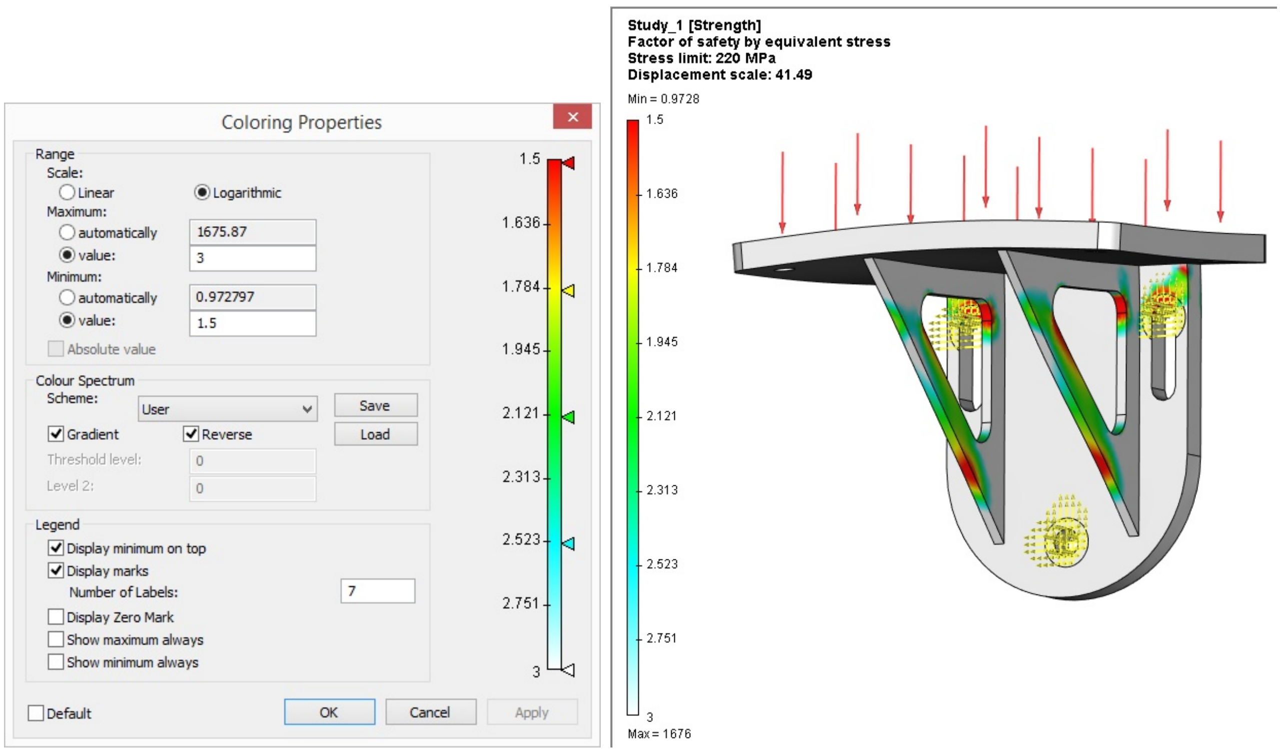
In this paragraph, we will provide an example of detailed result interpretation for a specific structural study, and then take all necessary measures to fix the model's flaws.

The original model is a bracket part, whose drawing is shown below. The bracket is loaded with the force of 3500 kilograms, evenly distributed over the horizontal plate. The model material is steel. There are three restraints: a full restraint for the vertical plate and two partial restraints for each of the three holes of the vertical plate. One partial restraint is in the bolt head zone and it restraints

displacements along Z and X axis. The second partial restraint is in the hole and restraints displacements along the Y axis.
The main criterion for assessing the structural strength is the Factor of safety (FS), as we mentioned earlier. The minimum FS value for this part should be no less than 1.5. After getting first results, one can see the general picture of the factor distribution, shown on the color diagram at the right.

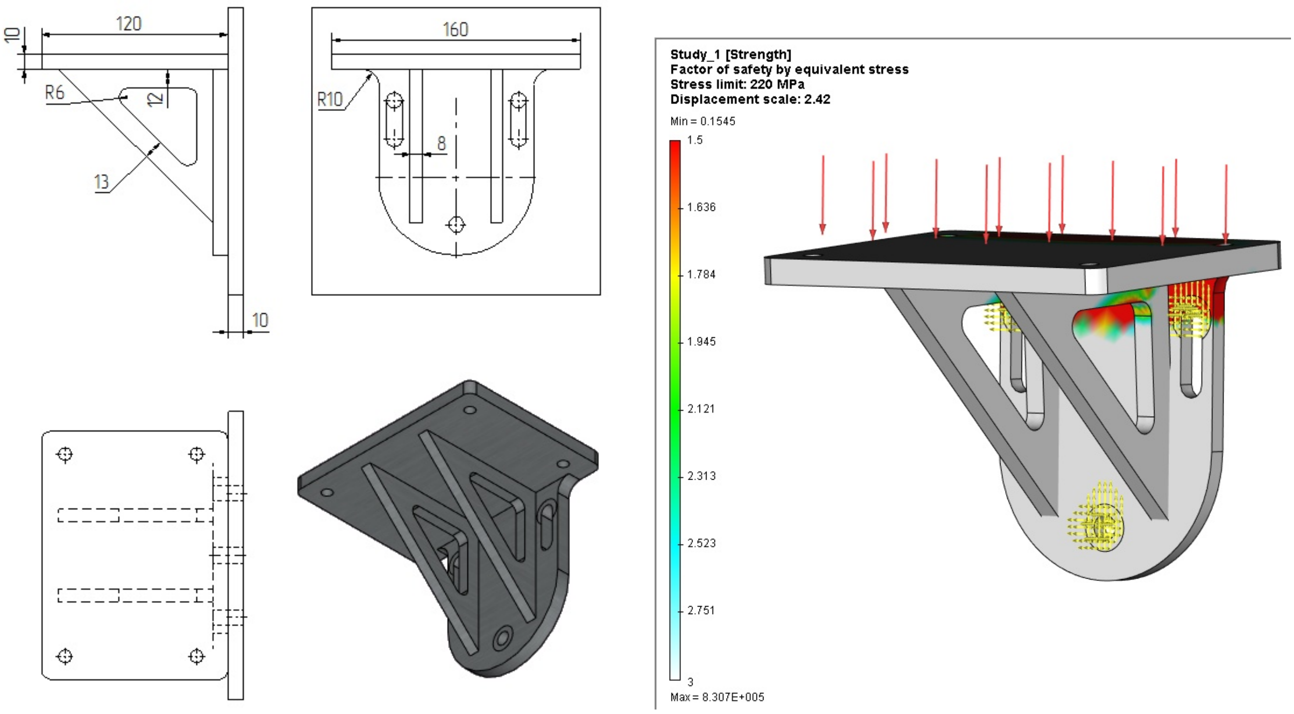


In this case, we are interested in the zones, in which the FS approaches critical values. We can scan the model using the mouse and pointing it at the places of interest, getting the result in the pop-up tooltip. However, for better visual representation of such zones, let's set up the color scheme as shown in the next figure, including changing the color for the maximum value of the safety factor (see section "**Color Scale Setup**").

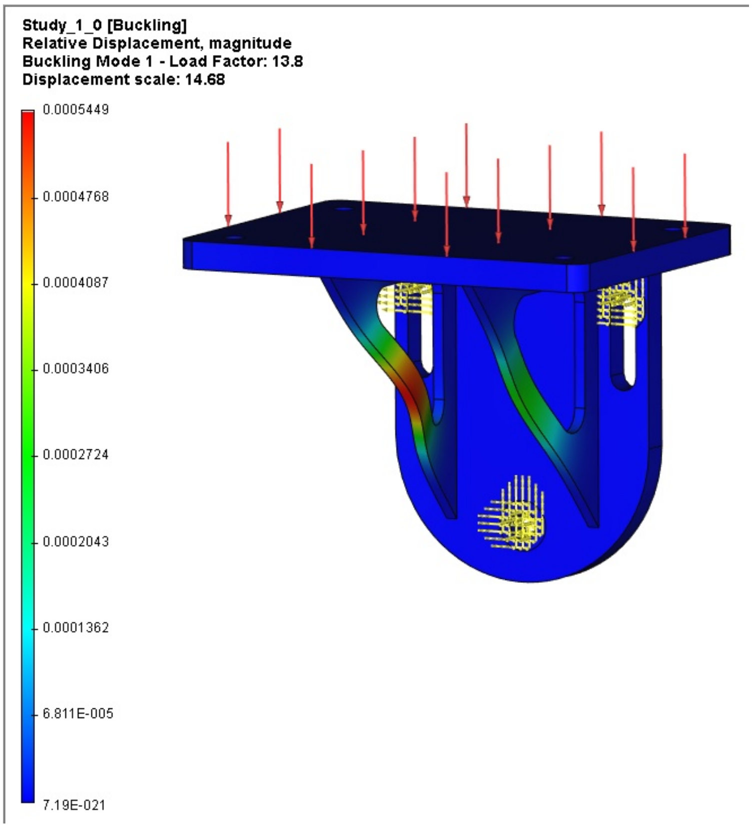


We specify the range of values of interest from 1.5 to 3. For convenience, we will correct color assignments. Anything greater than 3 will be displayed White – we are not interested in those zones

for the time being. The critical zones with the value below 1.5 will be Red. The rest of the values in the range will be assigned a color according to the color scale settings.
In this way of displaying the result, one can instantly notice the places of the model that require fortification. We also see from the scale, that the minimum FS value is 0.97, which is not admissible.
We then proceed with fixing model flaws. First of all, let's strengthen the pillars by increasing the thickness and other dimensions, and fillets to the corners of the vertical and horizontal plates connection. The new bracket drawing is shown below.



After updating the mesh and running calculations the second time, we can see that the factor of safety no longer reaches critical values in the «problematic» zones.
Also, when analyzing the general picture of a factor of safety distribution, one can discover zones of excessive strength. This zones reduction allows saving material. Thus, it is possible to reduce the thickness of the elements with excessive strength and carry out test calculations to achieve an optimal result.
To draw the final conclusion about the strength of the mounting, we must run the buckling analysis of the part. Such an analysis indicates that the critical safety factor for the mode 1 and this type of loading is 13.8, which means a sufficient margin of structural strength against buckling.


Study_1_0 [Buckling]
Relative Displacement, magnitude
Buckling Mode 1 - Load Factor: 13.8
Displacement scale: 14.68
0.0005449
0.0004768
0.0004087
0.0003406
0.0002724
0.0002043
0.0001362
6.811E-005
7.19E-021

Frequently Asked Questions

General

Computer system requirements

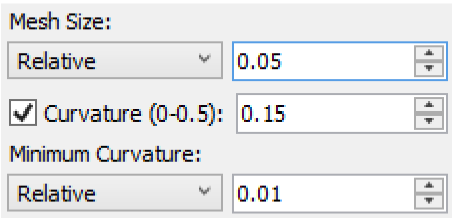
Mathematical modeling is resource consuming and requires a lot from a computer system. First, RAM (internal memory) is important for calculating processes. A modern multicore unit based on Intel (i5, i7, Xeon etc) processor with RAM of 32 GB or more is recommended.

Recommendations on finite element mesh parameters

Mesh designing is an important preparatory stage in finite element modeling. There is a big set of options for building the mesh in the system.

The higher is the quality of the mesh, the longer is calculating. The user should choose an optimum quality to speed ratio.

Three-dimensional finite elements should not be flat or elongated. They should approximate the model geometry uniformly and in detail.

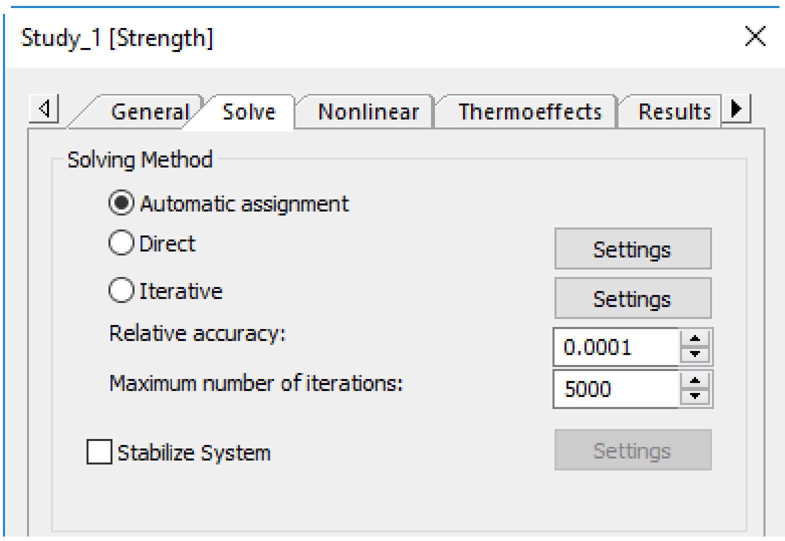


Key parameters for mesh generation

The parameter “Size” should be properly set, so that tetrahedrons tend to be equilateral. If the model is curved, you can control the approximation smoothness with “Curvature” parameter. The smaller it is, the finer and smoother is the mesh on curvilinear models.

Choosing calculation method — direct or iterative?

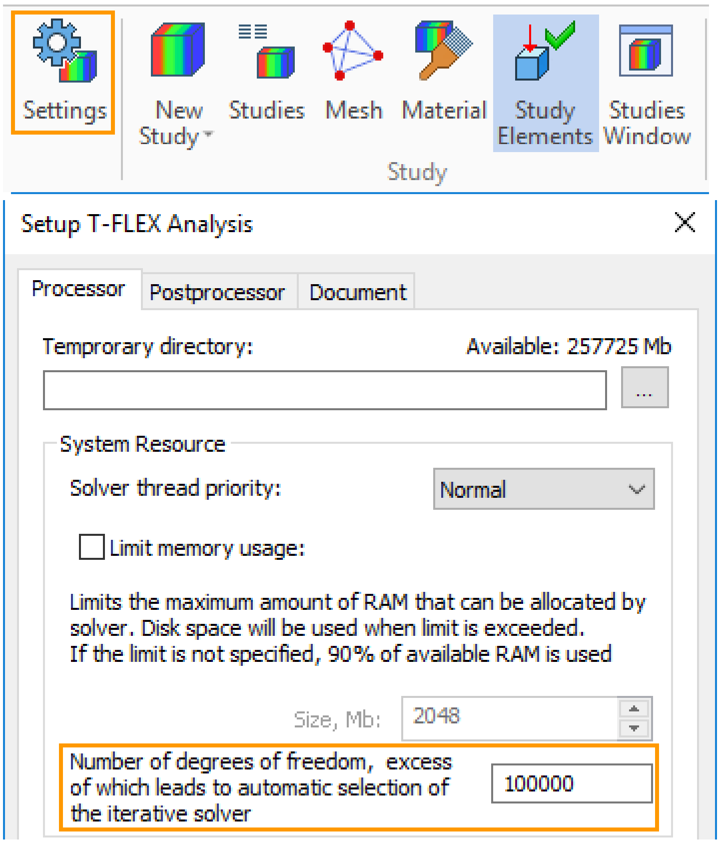
In finite element modeling systems of linear equations of high dimensionality are being solved. By default the system automatically chooses a direct or iterative solution method according to the number of equations. If the number of equations is totally less than the given limit (by default it is 100 000 equations), the direct method is chosen. Otherwise it is the iterative one.



Choosing the system of linear equations solution method

In general the direct method provides a more qualitative solution from the point of view of calculus mathematics. As a rule it requires more internal memory than the iterative method, therefore it is reasonable to use it on a computer with a big RAM.

Iterative method requires less internal memory and enables users to solve the problems with more degrees of freedom than the direct one on the same computer. However, the time required to solve a system of linear equations depends on the particular model and its finite element partition and varies in different cases.



Criterion setting for automatic selection of system of linear equations solution method

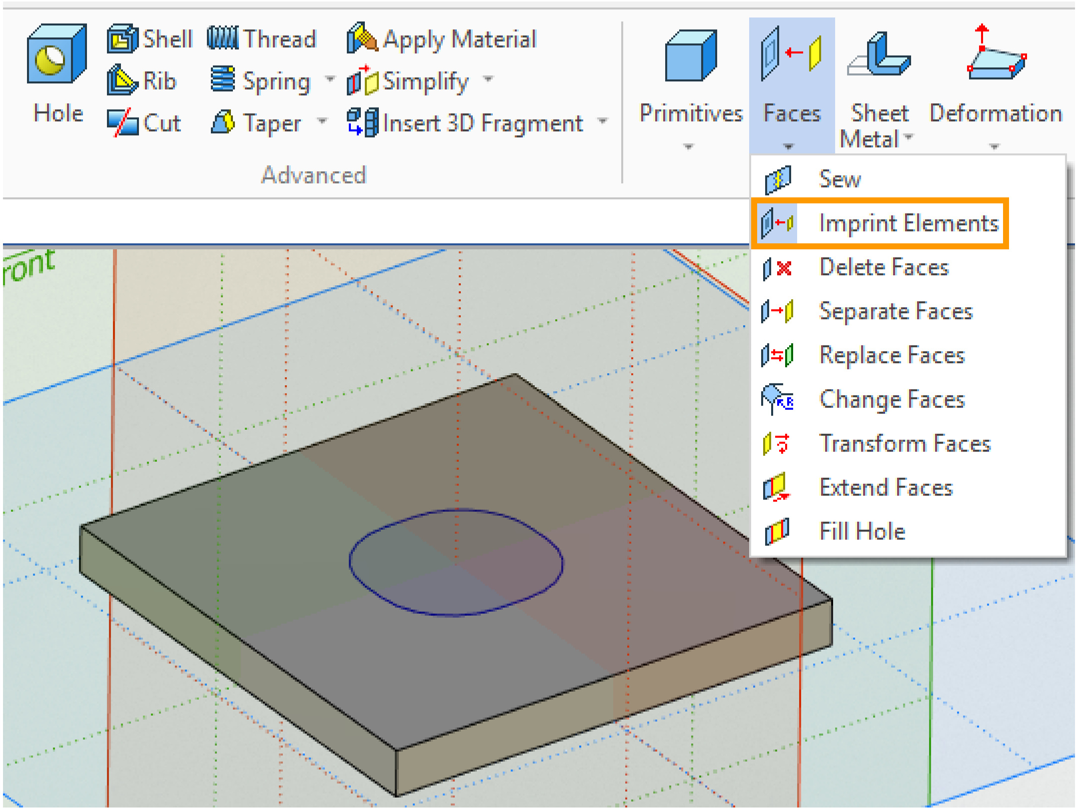
The user can set a limit for automatic selection manually, according to his/her computer system properties.

Direct method is also recommended for solving the problems of “contact” type.

How to apply a workload onto a part of the face?

In practical calculations it is often necessary to apply a load onto a part of a construction element face. For that purpose some preliminary actions should be taken and the user should create the same geometry on the required construction element as is the load application area. To do that, a profile on the chosen face should be created and the command “Imprint Elements” should be executed (see the picture).

It should be noted that for three-dimensional tasks (a finite element is a tetrahedron) boundary conditions (loading and fixing) at the apex (point) of the 3D model should be avoided. The reason for that is the fact that a singularity appears in this place (pressure and deformation tending to infinity) which distorts the result. In real physical conditions the area of applying load/fixing is always different from zero.



"Imprint Elements" command to create the load application area

In plate-type tasks the use of apices/nodes for boundary conditions application is taken into account in the mathematical model so it is justified.

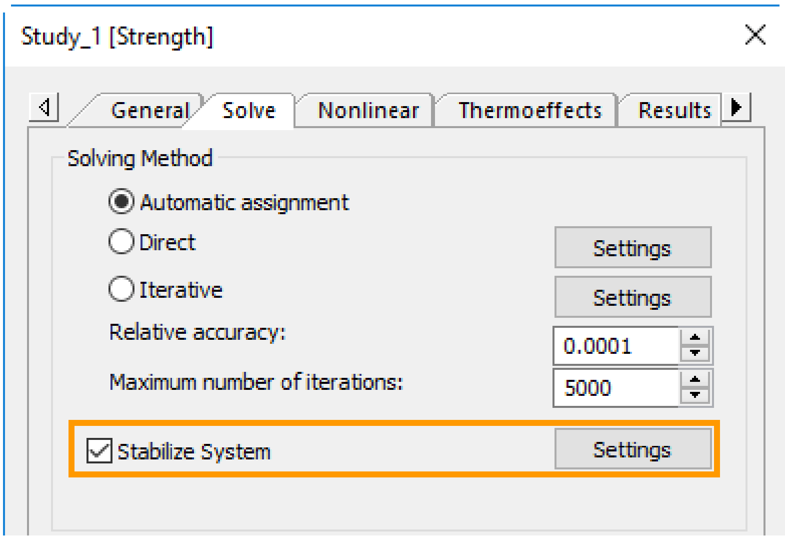
Static Capacity Tasks

What does the message "Study is underdetermined. Not enough boundary conditions" mean?

In case you see the message: "Study is underdetermined. Not enough boundary conditions" when calculating, there are the following variants possible:

1) Check that there are enough fixings. For strength calculation tasks it is usually necessary to ensure that the construction is immobile as a single unit, otherwise it will go to infinity and the calculation will fall apart.

2) Switch on "Stabilize System" option in the task settings. This option enables users to calculate unfixed systems, balanced in space by forces.



"Stabilize system" setting

3) Check material properties. It is possible that too wide spacing is set in the properties or the values are out of tolerable limits, for example Poisson coefficient is equal to 0,5 (it must be less than that).

Incorrect material data

Check if all physical properties necessary for the calculation are set. To calculate properly, the following data should be set for all the task materials:

1) Elastic Modulus E. For the metals usually used in constructing the value is in the interval 43000 ч 220000 N/mm^2.

2) Poisson's Ratio ν**.** The value is in the interval 0< ν <0.5.

To take into account the gravity force or "Acceleration" load, set the density value ρ. The density of the materials usually used in constructing is in the interval 0.00174 ч 0.00785 g/mm^3.

To take into account **Thermoeffects,** set **Thermal Expansion** λ. The value (for metals) is in the interval 8•10^{-6} ÷ 2,5•10^{-5} deg^{-1}.

Shear Modulus G is calculated automatically from **Elastic Modulus** and **Poisson's Ratio** by the formula: G=E/2(1+ν) and doesn't require special setting.

Tension diagram has value leaps and red points

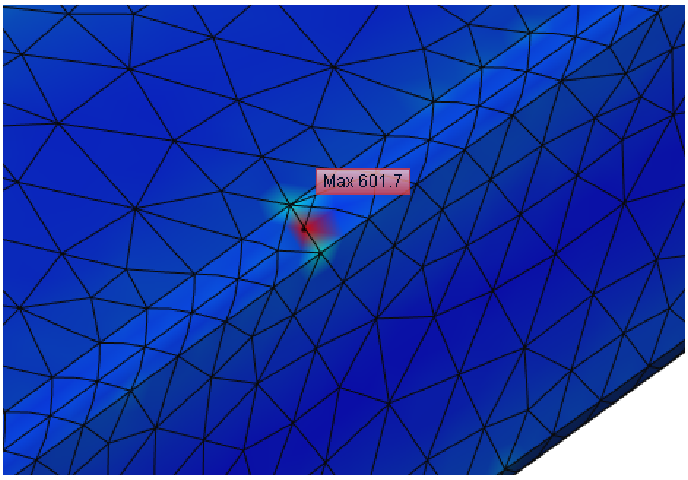
Sometimes tension diagram represents the model in blue with one or two red points – maximum values of tension, which are as a rule much higher than the tension values in other parts of the model. This situation usually takes place because there is an element of "poor" shape in the calculation mesh (it can be elongated, close to flat or degenerate into a segment element). Another reason for that is small size of the element (fraction of a millimeter).

Reasons and ways to fix the problem:

1) The mesh is too coarse. You should redesign the finite element mesh with other parameters. It is usually recommended to reduce the finite element size.

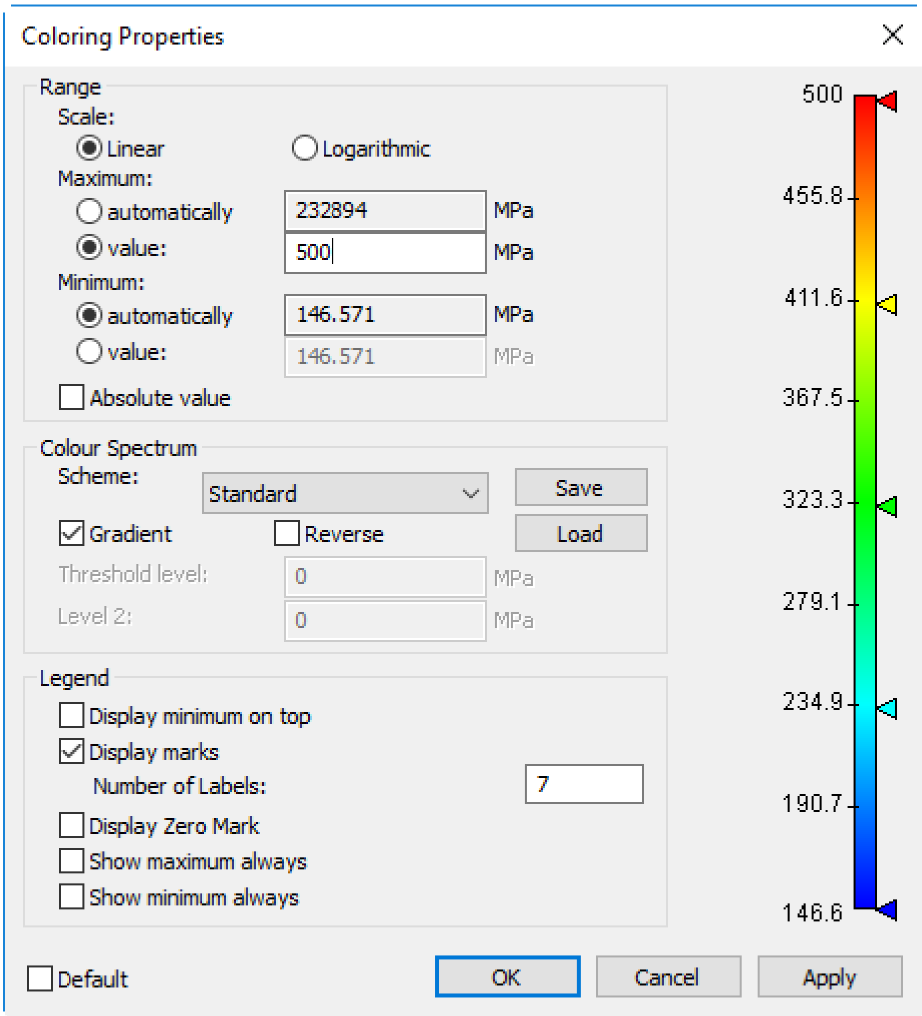
2) The mesh contains incorrect or degenerate elements which were deleted. If deleted elements were located in places of high tension, breaks and leaps in tension can appear. The mesh needs redesigning with other parameters, you should set mesh thickening in places of tension concentration found in the previous calculation.

3) Iterative method poor accuracy. You should reduce the relative error of the iterative method (increase accuracy). It will lead to an increase in calculation time but can eliminate the stains if they appear due to insufficient calculation accuracy of the iterative method. You can also try to solve the problem with the direct method if your computer parameters enable you to do that. In this case the calculation error is minimal.



Example. A diagram with a tension leap.

When analyzing a diagram with a tension leap, you can use color scale settings to eliminate the tension peak from the diagram. To do that, you need to set tension maximum manually.

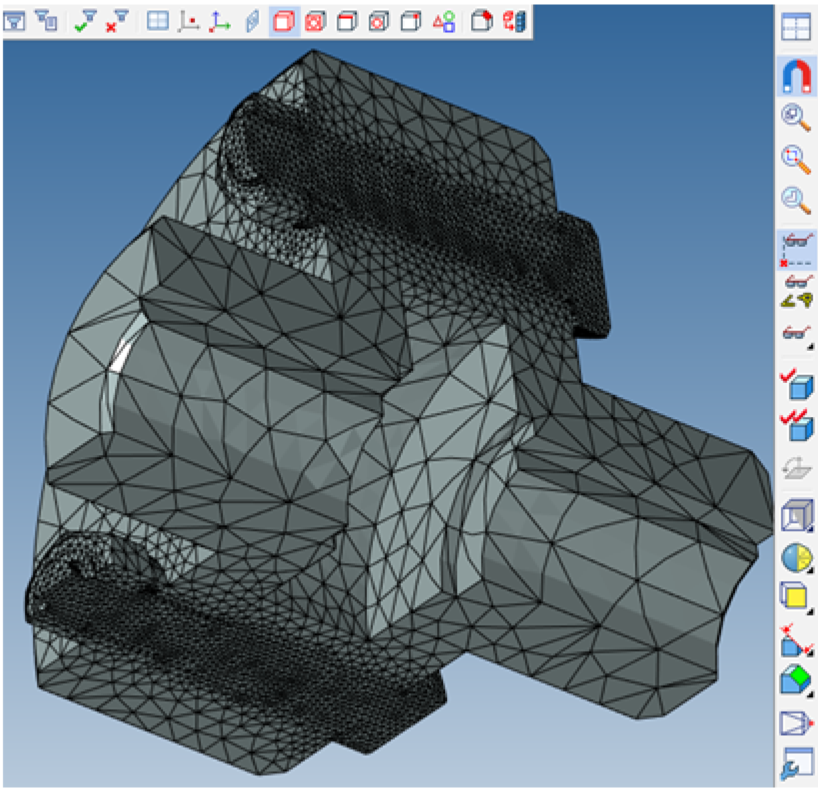


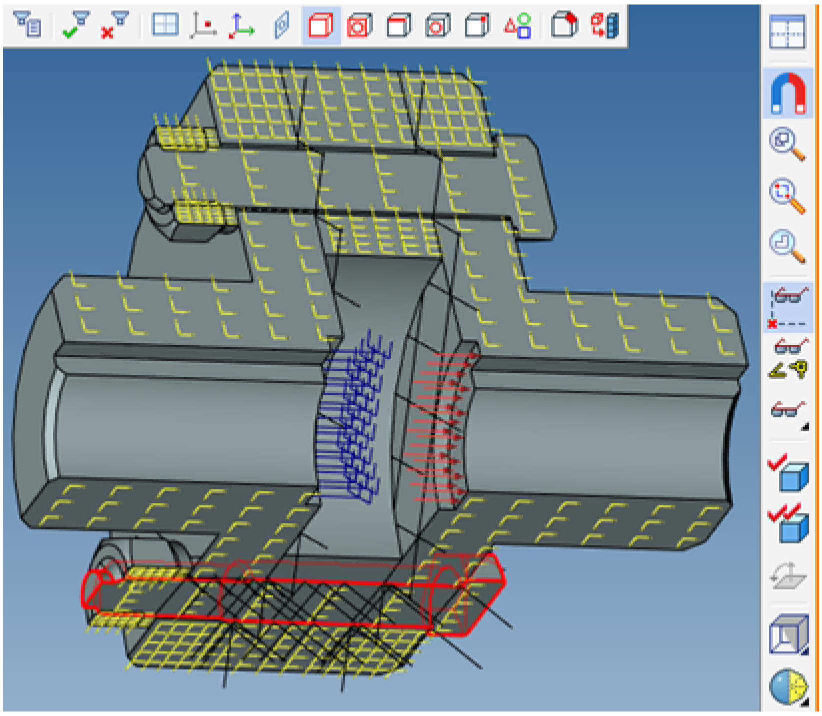
The displayed value maximum limitation

Bolt joint calculation

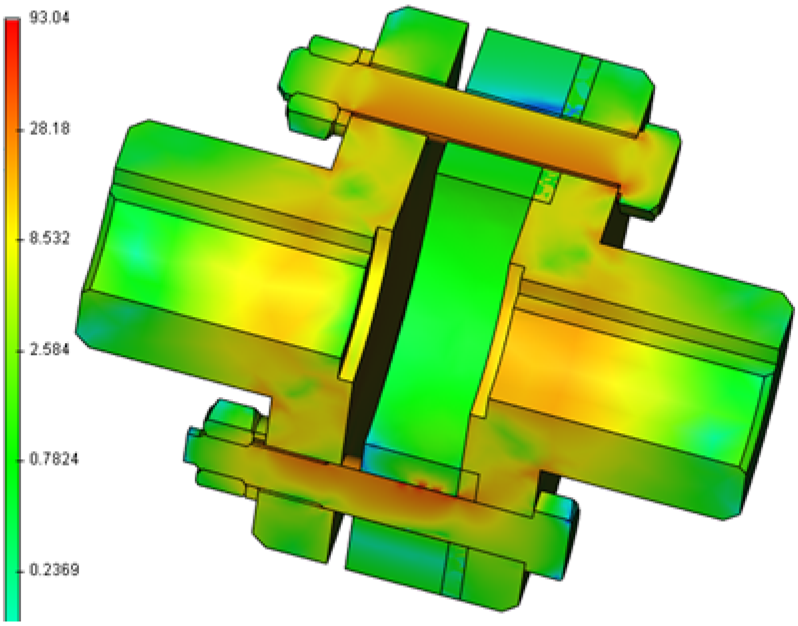
Bolt joints modeling is currently realized in T-FLEX Analysis similarly to modeling other 3D constructions, i. e. by means of designing a full 3D model of the joint. The threaded joint is considered to provide an immovable connection, i.e. the bodies are in rigid connection in place of the thread. When calculating bolt joints the most relevant things are tensile and sheer stresses in the bolt. If there are bolt joints in the assembly model, you can use a bolt model, set the contacts between the parts being joined (bolt head and washer/nut are left in rigid connection with them both), thicken the mesh in the place of joint, cut off the symmetrical parts in the whole assembly (if there are any) and calculate as usual. As preliminary tightening is not taken into account, multiply the calculated stresses in the bolt or nut by 1,3...1,5 depending on tightening being controlled or not and according to that estimate the bolt joint safety factor. In terms of thread shearing, set off a band on the bolt cylinder for the first 3 turns with the help of face separation, set a rigid contact on it and a touching on the rest of the face and calculate as usual.

The picture shows an example of designing a mesh with thickening in fixings.





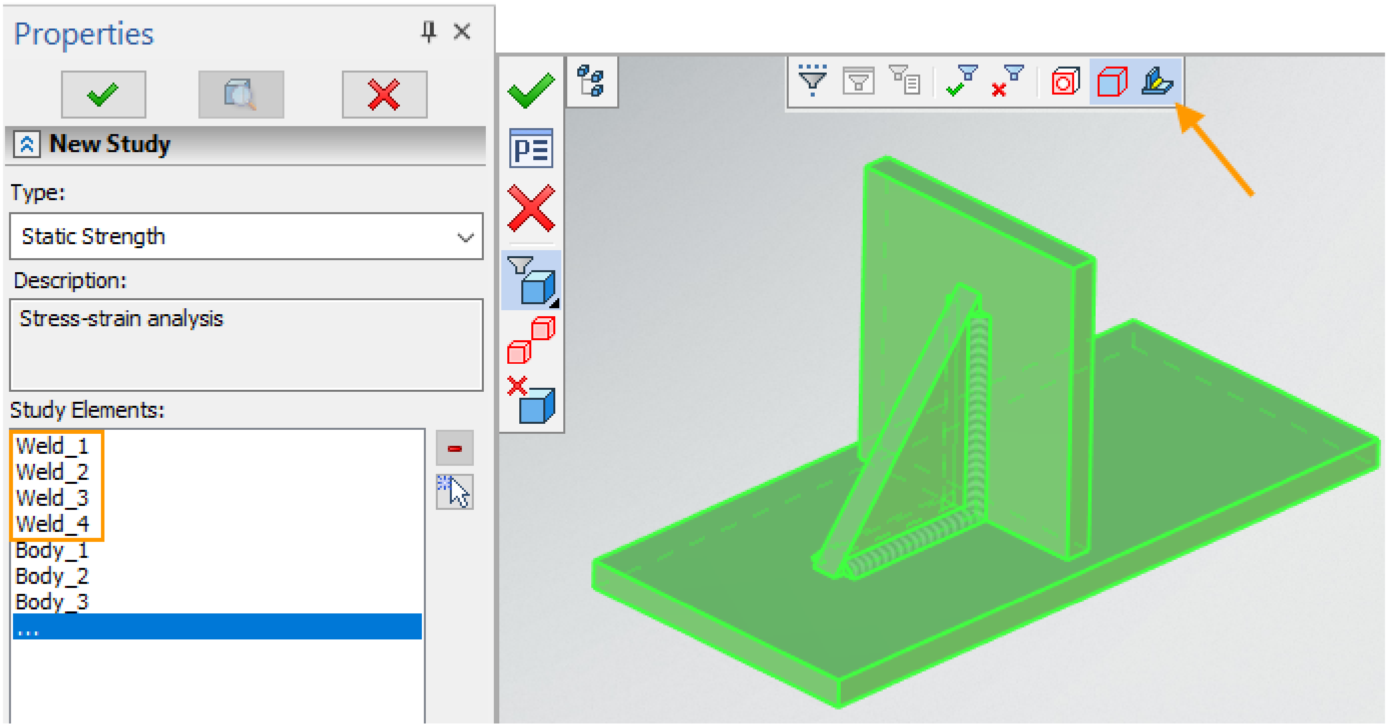
A model with symmetry and contact conditions applied



Result of calculation

Welded seam calculation

There is a special command for creating welded seams in T-FLEX – **Weld**. Welded seams can participate in finite element studies. When creating a study, welds are taken into account, because a corresponding setting of the filter is on by default:

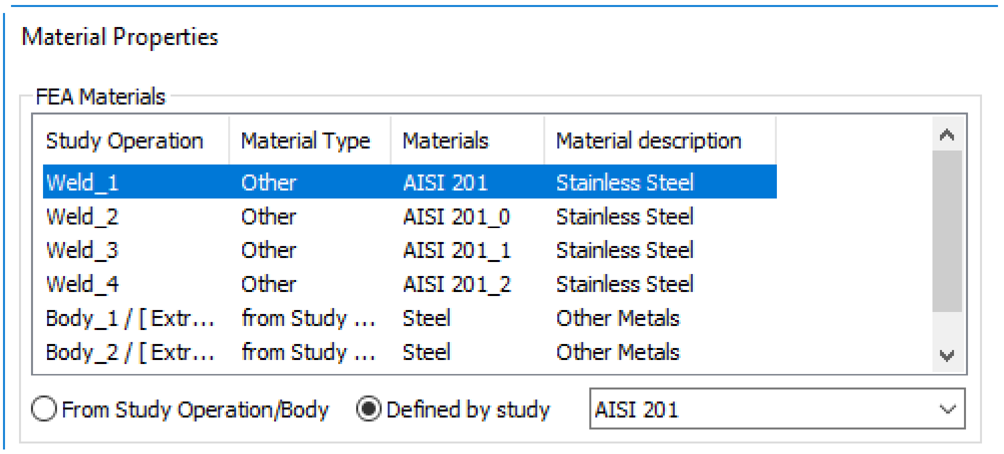


Adding welded seams to the study

By means of calculating welded seams you can estimate the flow of forces and moments between the structural parts as well as their tension and deformation caused by the forces and moments acting through welded seams. You can estimate tensions appearing in the weld. As internal thermal stresses in the welds are not taken into account, their impact can be considered through a coefficient corresponding to the type of welding.

When creating the mesh, it is recommended to use **Fix intersections** option because the welds can intersect as it is shown in the picture.

After creating the study, you should assign a material to every welding seam, then calculation can be performed.

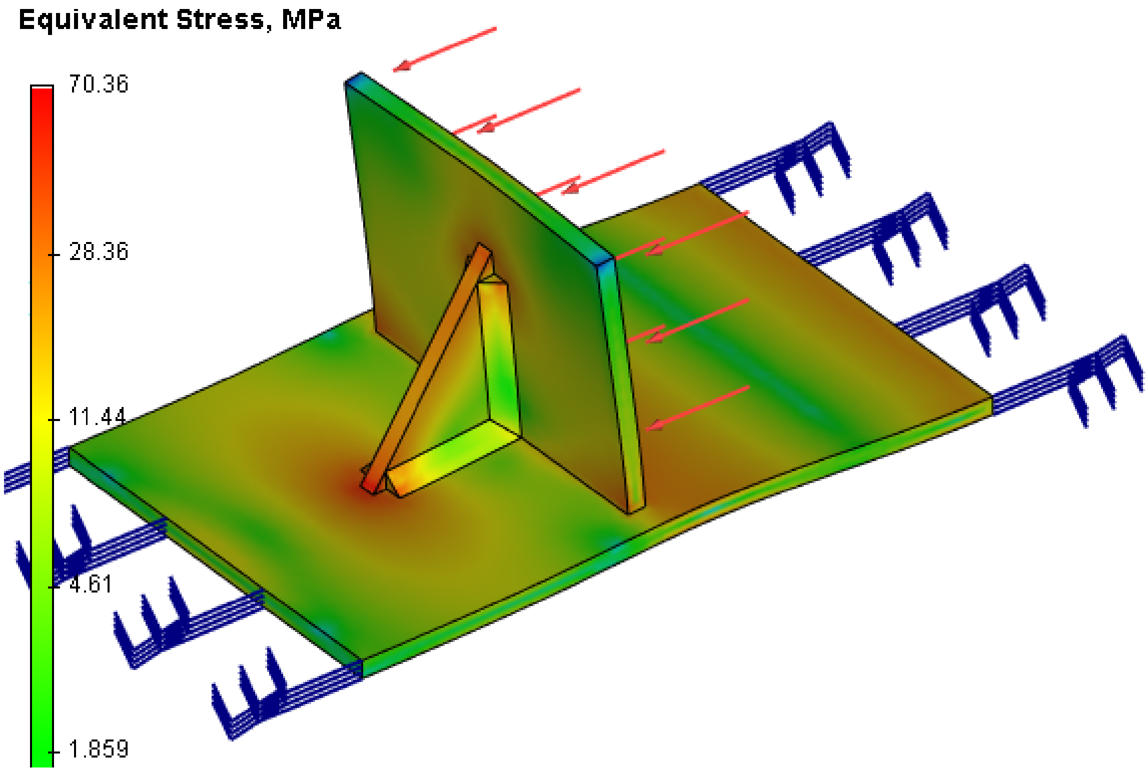

Material Properties

FEA Materials

Study Operation	Material Type	Materials	Material description
Weld_1	Other	AISI 201	Stainless Steel
Weld_2	Other	AISI 201_0	Stainless Steel
Weld_3	Other	AISI 201_1	Stainless Steel
Weld_4	Other	AISI 201_2	Stainless Steel
Body_1 / [Extr...	from Study ...	Steel	Other Metals
Body_2 / [Extr...	from Study ...	Steel	Other Metals

From Study Operation/Body · Defined by study · AISI 201

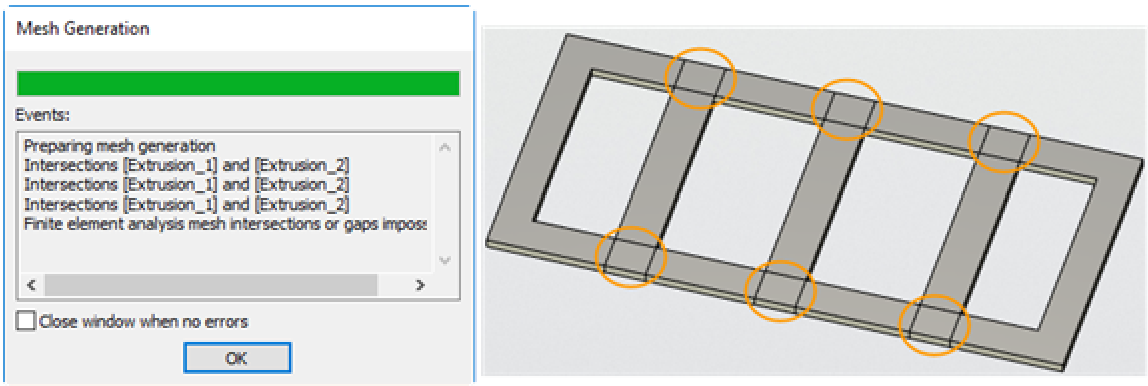
Assigning material to the welds



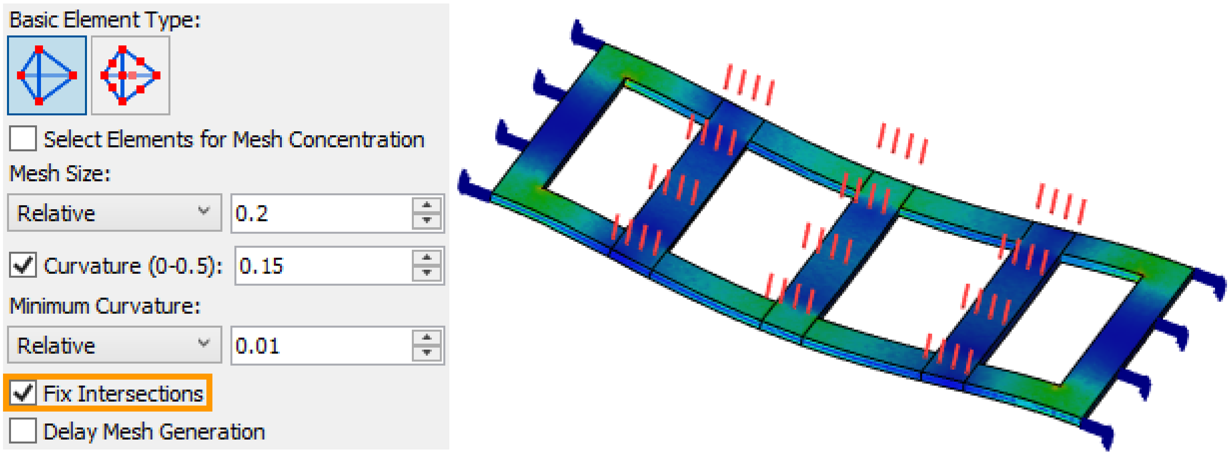
You can control mesh concentration at the weld by means of activationg **Select Elements for Mesh Concentration** option, choosing the rib under the weld (or several welds) and varying the **Refinement Radius** and **Propagation** parameters to get appropriate mesh accuracy.

Eliminating the intersections

For the mesh to be designed properly it is essential that there are no intersections in the model. You can use <QI> **Intersection Check** command to detect intersections and then use common modeling tools to eliminate them. In case of threaded joints intersections cannot be eliminated by modeling, so they are eliminated automatically by the program and require no actions from the user. If assembly model has many minor intersections which take time and effort to eliminate or it was imported and is difficult to edit, you can use **Fix Intersections** option in the mesh parameters menu. This option automatically corrects the intersecting bodies' geometry by means of Boolean operations "intersection" and "subtraction" and makes it possible to design the mesh for the corrected model.



3D model with intersections between bodies



Using automatic correction of intersections option to correct geometric errors of the model

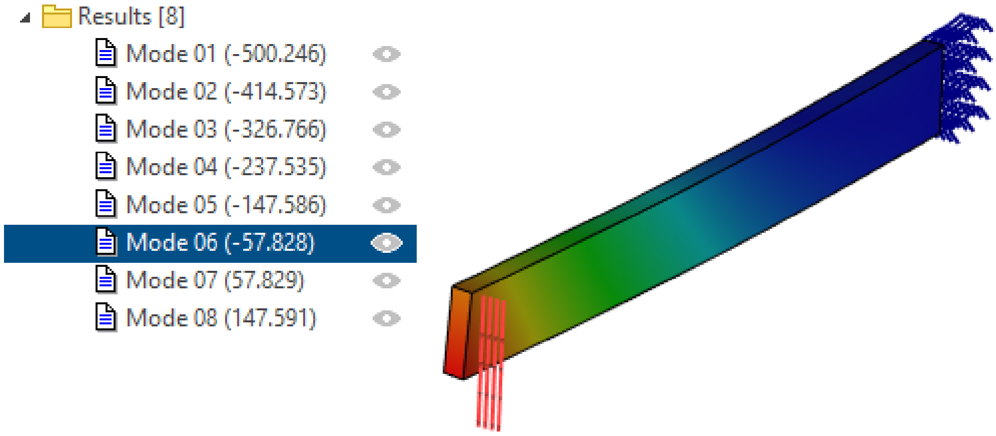
The user should understand that automatic correction changes the initial parts geometry for the calculation, which is not always acceptable. It is perfect if the initial model has no intersections.

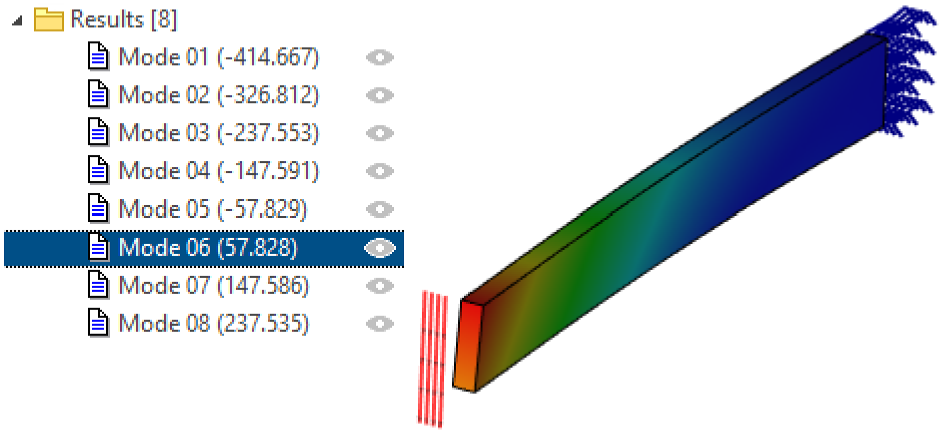
Buckling Problems

Negative values of critical load

Negative value of critical load coefficient means that at given direction of load action there is no loss in stability at the particular form. If the load direction is inverted, it is likely that calculated stability loss form corresponding to this negative value appears. In the construction stability calculation only those forms are usually taken into account, for which the critical load value is positive, starting with the first positive one.

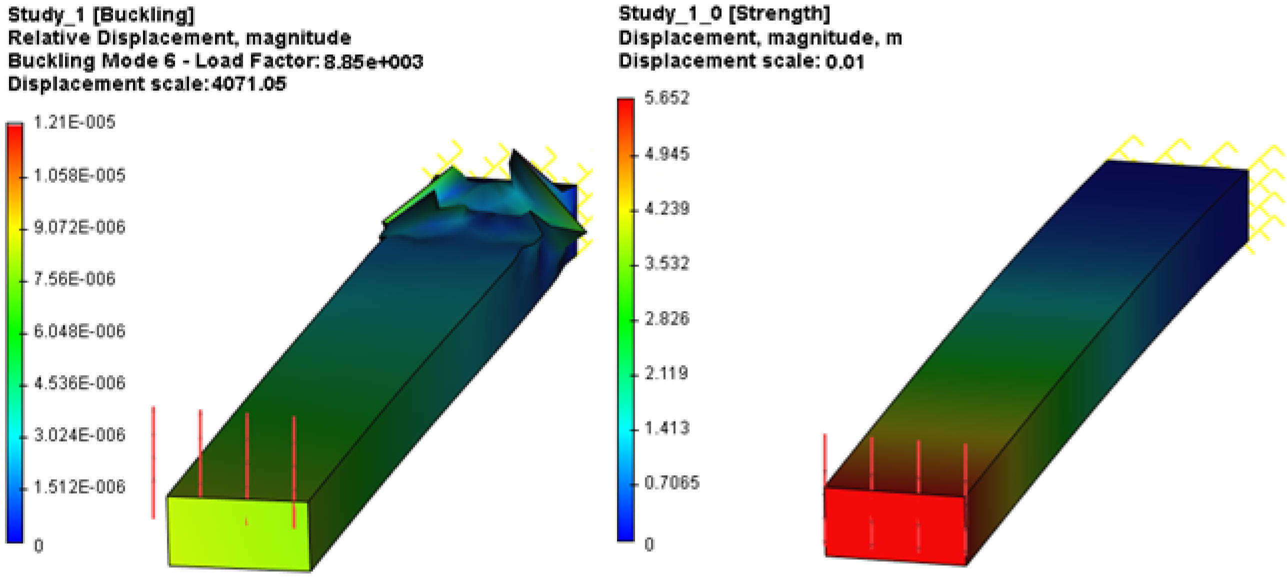
Example. The pictures given below show that Mode 6 (-57,828) is symmetrical relative to the horizontal plane and corresponds to Mode 7 (57,829). When measuring load direction Mode 6 becomes positive and will correspond to Mode 5.





Stability loss form looks strange, needle-shaped or has convexities

When calculating a stability problem first you calculate the static problem. If you get too big movement and stress values, the considered stability loss forms appear. To make sure it is sufficient to calculate a static problem at a critical load corresponding to the given needle-shaped form. To do that, you need to multiply the applied load by the critical load coefficient. In practice in the given load system such stability loss doesn't take place and the construction experiences great deformation staying stable. Therefore, it is necessary to calculate static capacity for it.

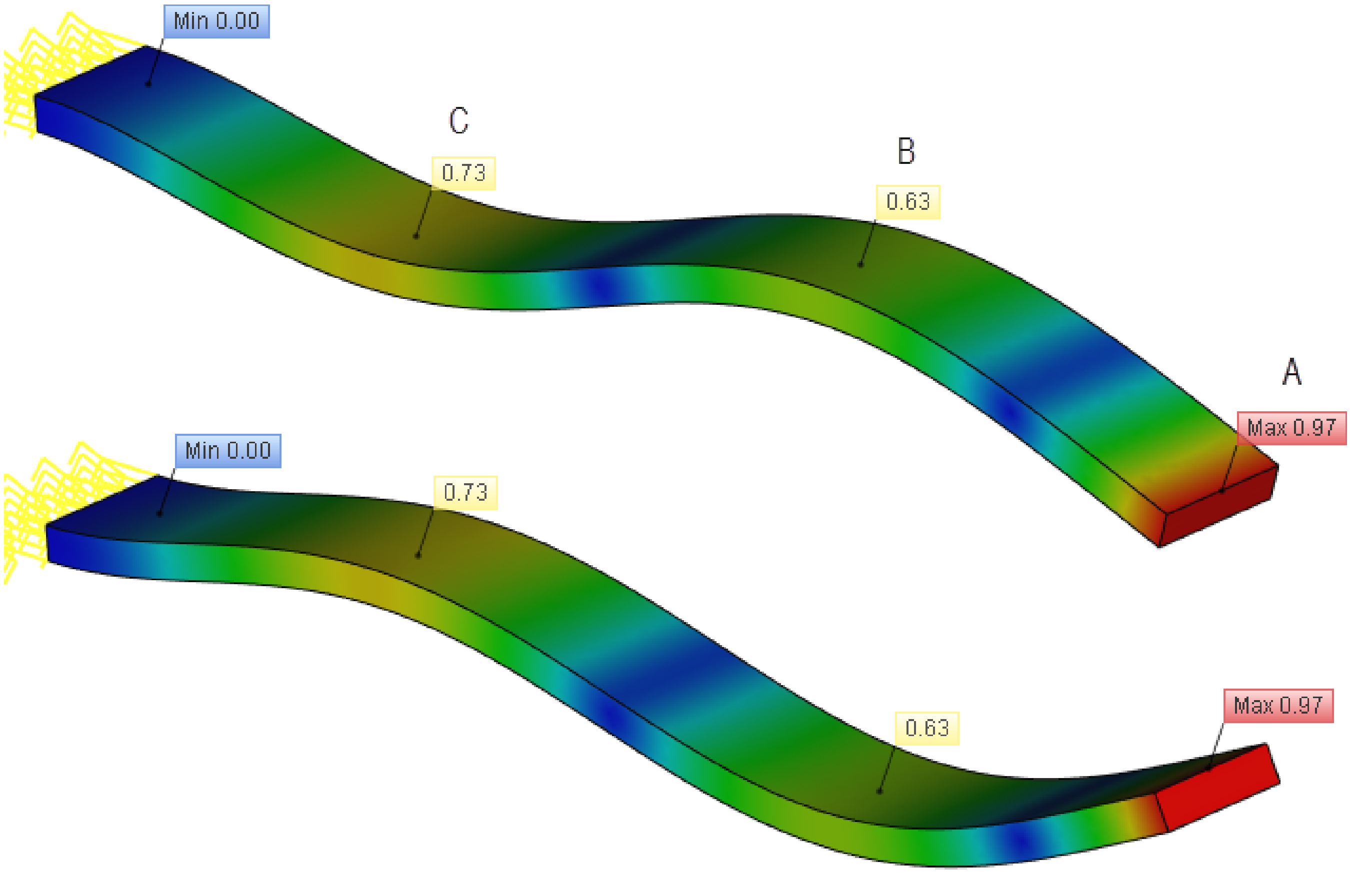


Example. "Needle-shaped" stability loss form and a corresponding abnormally big calculated movement

Frequency Problems

What is the unit of measurement for "Magnitude of Relative Displacement" given in the intrinsic frequencies analysis results?

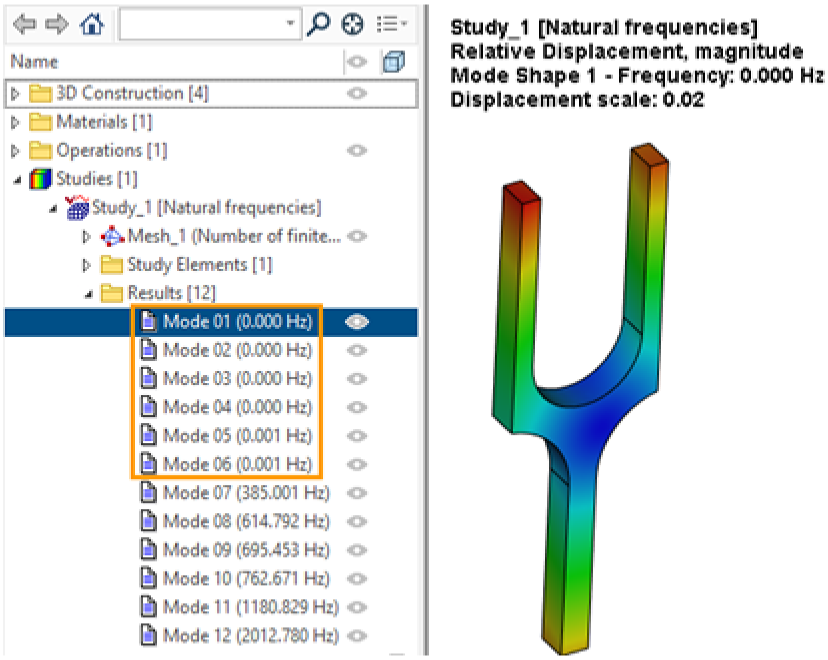
The result of **Relative Displacement, magnitude** shows relative amplitudes of different construction points. The picture below illustrates point B with an amplitude of 0,63 vibrating 0,97/0,63=1,54 times weaker than point A with a maximum amplitude of 0,97. I.e. if point A amplitude is equal to 1 mm, then point B amplitude is equal to 0,65mm, point C amplitude is equal to 0,75 mm. If the vibrations are two times stronger, the amplitudes are 2; 1,3; 1,5 mm correspondingly.

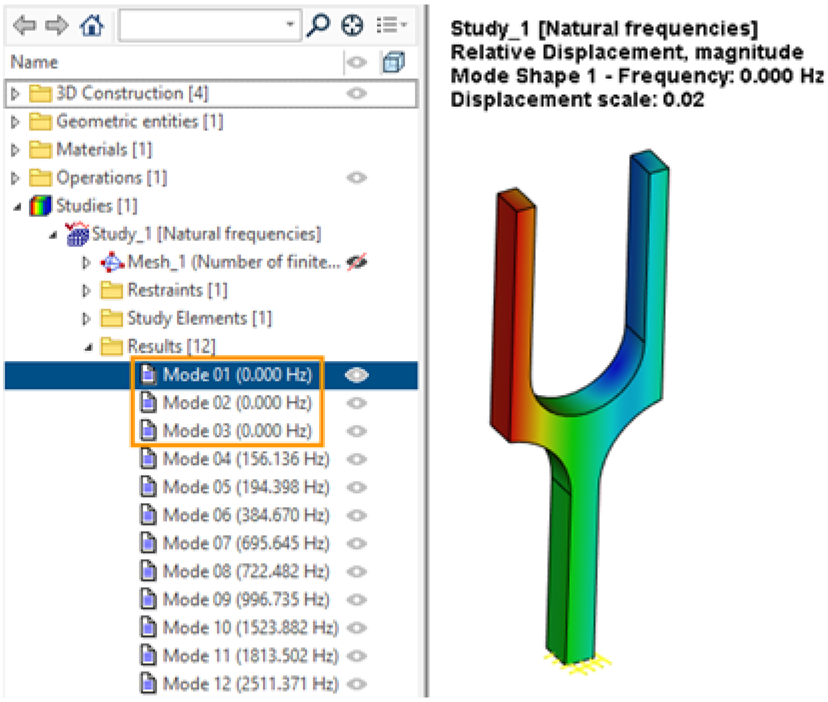


Why do null frequencies appear?

When calculating intrinsic frequencies, it is allowed that the construction is partially fixed or not fixed at all. The construction is considered to be partially fixed when movement of only several of its apices, ribs or planes along one or two axes is restricted. In such a case the calculation proceeds with **Non Stationary System** option on. Null frequencies correspond to an infinite vibration period which takes place if the construction moves as a whole to an infinite distance. The number of null frequencies can reach 6 for a fully unfixed construction (floating in weightlessness) or less if the construction is partially fixed in space.

Example:





Thermoanalysis Problems

Radiation

There are two variants of thermal radiation calculation implemented:

- Radiation to Space.
- Radiation between Faces.

You can choose a variant from the command parameters list.

When creating the Radiation loading, the body always gives and absorbs energy according to the set Radiation parameter through the given surface.

Radiation parameter is a coefficient varying from 0 to 1, which determines radiating ability of the body relative to maximum possible radiation ability of a blackbody at the same temperature.

Therefore, **Radiation** parameter also determines the degree of blackness, i.e. the absorbing ability of the body. Zero corresponds to a body which absorbs no radiation and gives off none as well. One corresponds to a body which absorbs all the external energy and gives off maximum of its own.

It is necessary to define the emitting surface (or several ones if their properties are the same), ambient temperature and **Radiation** parameter for any of the two radiation variants.

Radiation to Space

- *Emission*. The body gives off energy from the defined surface according to the **Radiation** parameter. The body itself cools down and the energy released by the body does not influence other objects of the study.
- *Absorption*. The body absorbs energy through the defined surface according to the **Radiation** parameter and ambient temperature. The body heats up.

Radiation between Faces

- *Emission*. The body gives off energy from the defined surface according to the **Radiation** parameter. If the elements of the surface giving off energy are visible for another surface, for which **Radiation** is defined with the variant **Radiation between Faces**, then part of this energy is absorbed by the latter surface according to its **Radiation** parameter. If the surface elements are not visible for the elements of other surfaces with a similar loading, the energy given off by the body does not influence other objects of the study.
- *Absorption*. The body absorbs energy of the visible elements of other surfaces for which **Radiation** is defined with the variant **Radiation between Faces** through the defined surface according to the **Radiation** parameter. If the surface elements are not visible for the elements of other surfaces with a similar loading, the body absorbs all the ambient energy through the defined surface according to the **Radiation** parameter and ambient temperature. If there are both emitting elements of other surfaces and “void” elements visible for the emitting surface, then the absorbed energy is summed up according to the ratio between voids and visible emitting elements as well as emitting surfaces parameters and ambient temperature.

Whether the elements of one emitting surface are visible for the elements of another emitting surface is defined automatically by the visibility angle parameter and is not controlled by the user.

Important rules for radiation calculation problem formulation

Visibility of emitting surfaces

It should be remembered that calculating the visibility of emitting surfaces requires significant RAM volume. Every new emitting surface added to the calculation increases both required computer resources and calculation time.

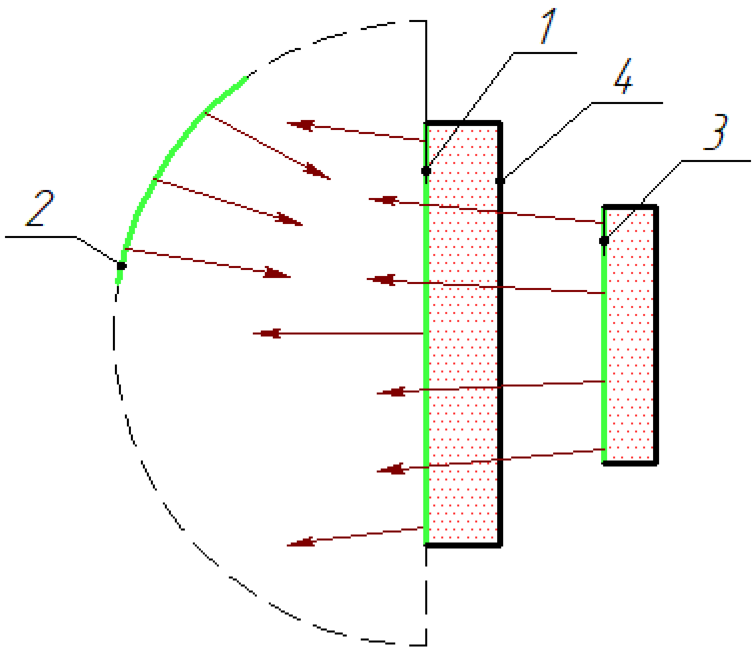
> Don’t add faces with Radiation between Faces loading to the calculation if they are irrelevant for the process under consideration. It will increase efficiency at a minimal loss in accuracy.

All the surfaces without **Radiation between Faces** loading are considered transparent for calculating energy transfer by radiation.

The surface is considered opaque only for the rays whose direction is opposite to its radiation direction. In other words, the surface absorbs only the rays coming in the direction opposite to its radiation direction. It can be explained by the fact that visibility area is built by a hemisphere whose direction coincides with the radiation direction of the surface.

> Body face radiation is directed sideways of the body.

Let’s consider an example. Surface “1” has a **Radiation between Faces** loading and its radiation is directed from the body. As for any surface with a **Radiation between Faces** loading, absorption from other surfaces subject to their visibility is calculated for it. The radiation from a surface (or several ones) at a conditional visibility sphere of surface "1" is represented with area "2". The energy coming from the elements in the conditional visibility area "2" is absorbed by surface "1". Surface "3" radiates in the direction from the body. Surface "4" has no **Radiation between Faces** loading and is transparent for radiation. Surface "1" is also transparent for radiation of surface "3": the elements of surface "3" are out of visibility for the elements of surface "1", so surface "1" doesn't absorb radiation energy of surface "3". Furthermore, surface "3" doesn't absorb radiation energy of surface "1": radiation of surface "1" is directed sideways from surface "3".



Temperature of emitting bodies

In terms of physics it is incorrect to set a **Temperature** boundary condition and **Radiation** thermal loading for one and the same surface as the emitting surface is to lose heat due to giving off radiation and generate it due to absorbing radiation. In other words, emitting surface cannot have a fixed temperature; its temperature is calculated during solving the problem.

> To set the initial temperature of the body which the emitting surface belongs to and the initial temperature of the emitting surface itself, the Initial Temperature condition should be given. In most cases Initial Temperature should be set for the whole body, not its faces separately. The resulting temperature of the emitting surface and its intermediate values during solving the problem are calculated.

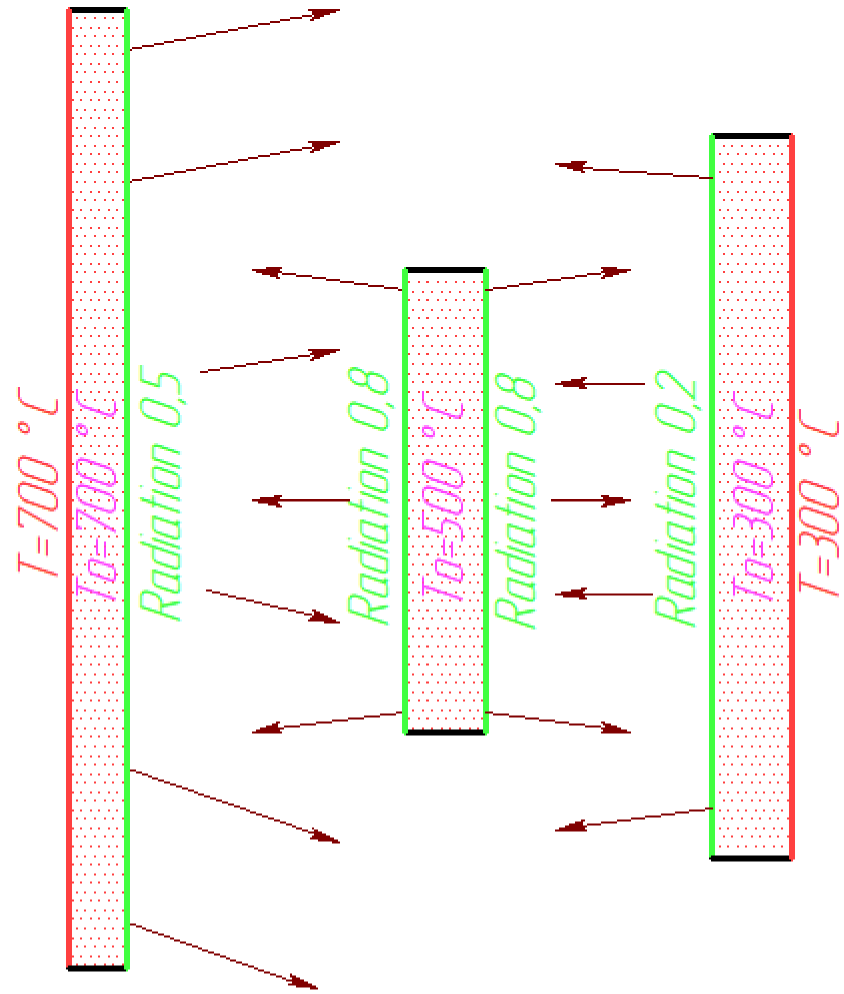
If we know from the statement of the problem that the body with an emitting surface (or several ones) heats up due to some source of heat, this heat gain should be set by means of thermal loading. If we know from the statement of the problem that some part of the body has a constant temperature, a **Temperature** boundary condition should be set for the corresponding body surface (or several ones), at the same time the surfaces with the given temperature must be non-emitting ones.

> If the user sets a Temperature boundary condition for a body with emitting surfaces or for the emitting surface itself – it is a purposeful assumption distorting the temperature field. Such an assumption must be reasoned in terms of the processes under research.

Example of calculation

Let three vertical plates be given.

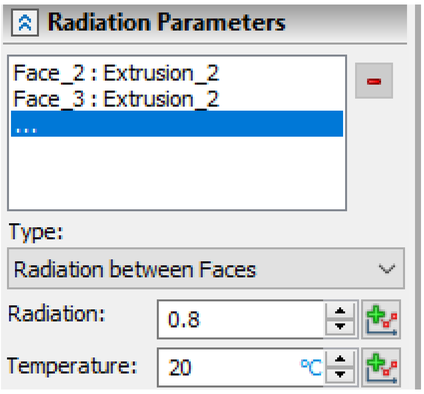
The left plate is the biggest one by area, its temperature is 700 °C and the degree of blackness is 0,5. The central plate is the smallest one by area, its temperature is 500 °C and the degree of blackness is 0,8. The right plate is the bigger than the central one, but smaller than the left one, its temperature is 300 °C and the degree of blackness is 0,2. Let it be known that due to several sources of heat the left plate has a constant temperature of 700 °C at its left face, and the right plate has a constant temperature of 300 °C at its right face. The central plate has no additional sources of heat. The ambient temperature is 20 °C. Let's consider how radiation heat transfer influences the temperature fields of the plates.

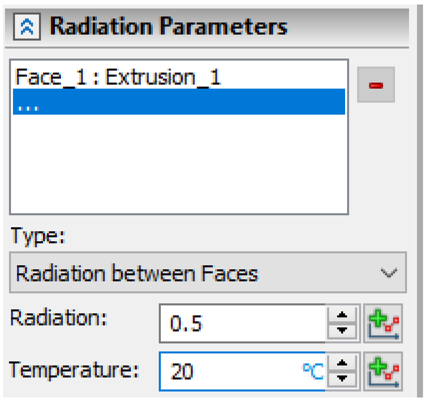


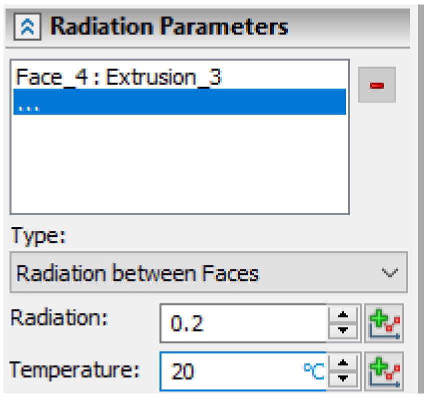
The problem is non-stationary. We create a **Nonstationary Thermal Process** study. We create a mesh with a density corresponding to the required accuracy degree of thermal field calculation.

Let's create a **Radiation** loading with **Radiation between Faces for the right and the left faces of the central plate.** According to the problem statement, **Radiation** parameter equals 0,8, ambient temperature is 20 °C. Ambient temperature is necessarily set for "voids" in the visibility area of the emitting surface elements, in this case the surface absorbs ambient radiation at the given temperature.

Let's also create a **Radiation** loading with **Radiation between Faces for the right surface of the left plate and for the left surface of the right plate according to the** problem statement**.**

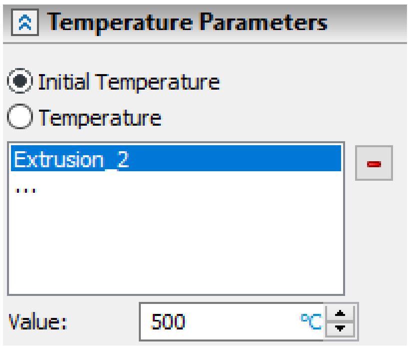


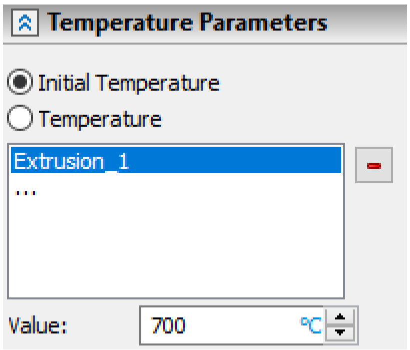


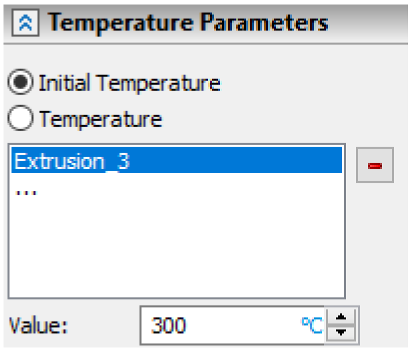


Central plate Left plate Right plate

We set the initial temperature values for the plate bodies **according to the** problem statement.

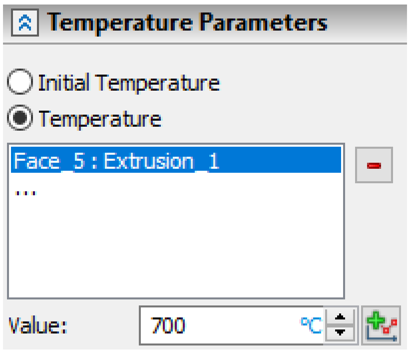


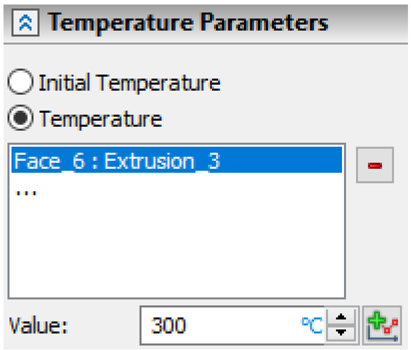




Central plate Left plate Right plate

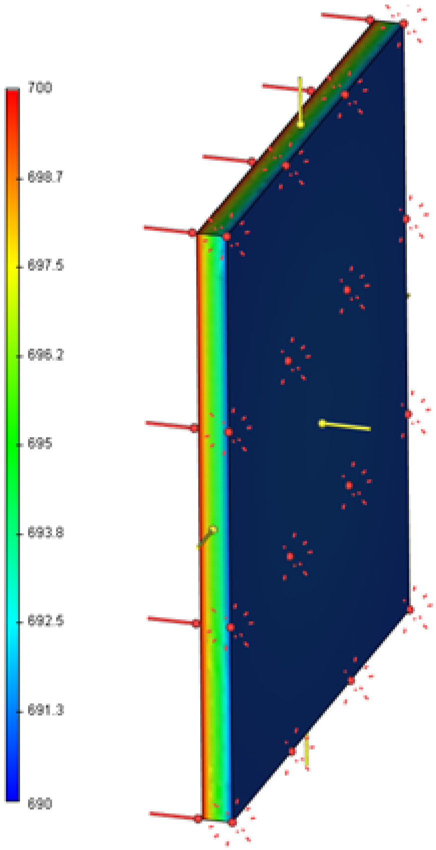
We set fixed temperature values **for the right surface of the right plate and for the left surface of the left plate**



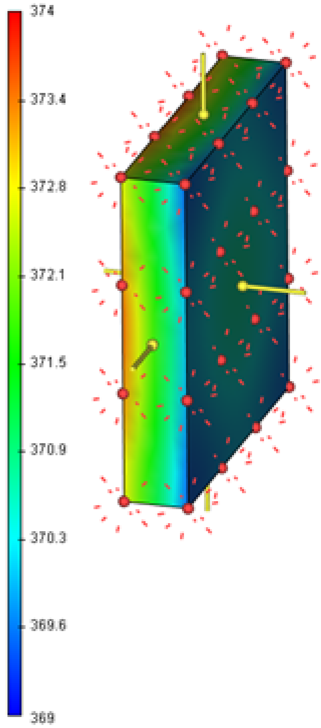


Left plate Right plate

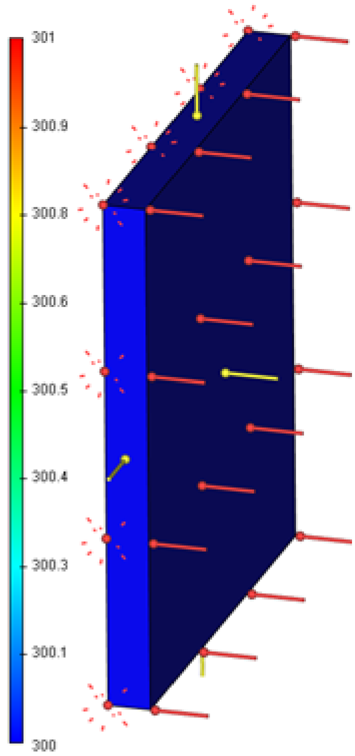
We set calculation time of 100 minutes with a pitch of 1 minute. Let's take a look at the temperature field at the plates at 100 minutes.



With scale setting in the temperature range of the left plate



With scale setting in the temperature range of the central plate



With scale setting in the temperature range of the right plate

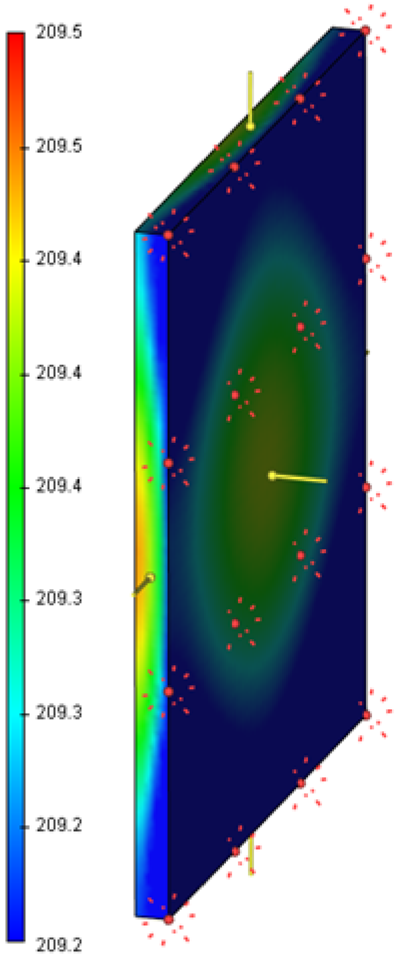
As we can see from the picture, the greatest temperature difference is at the left plate: it has given more energy because of the biggest surface area and relatively high radiation parameter value. At the same time the temperature at the edges of the emitting face is lower than in the centre, which is due to less energy received from emitting bodies.

The temperature of the central plate decreased by more than 100 degrees: the body has no heat sources of its own so it has given off more energy than it has received. At the same time the right side has a lower temperature than the left one: there is a body with a higher radiation parameter to the left than the one to the right.

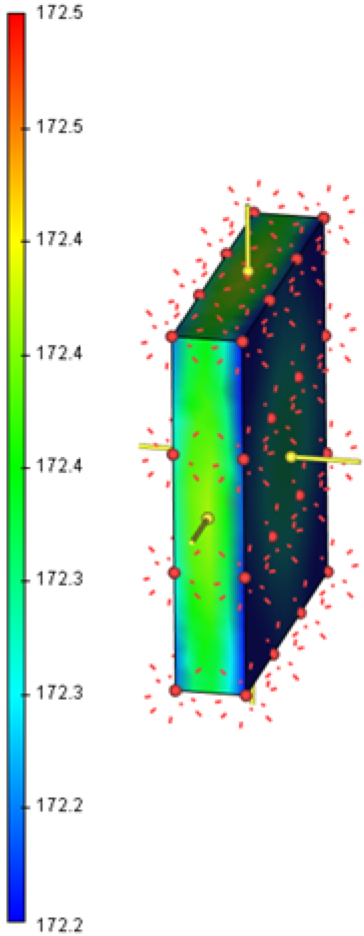
The temperature of the right plate is virtually uniform along the whole volume: the body has a low radiation parameter and gives off little energy at the same time having a constant temperature at the right face.

Let's change the problem statement and remove heat sources of the left and the right plates: we delete **Temperature** restriction at these bodies.

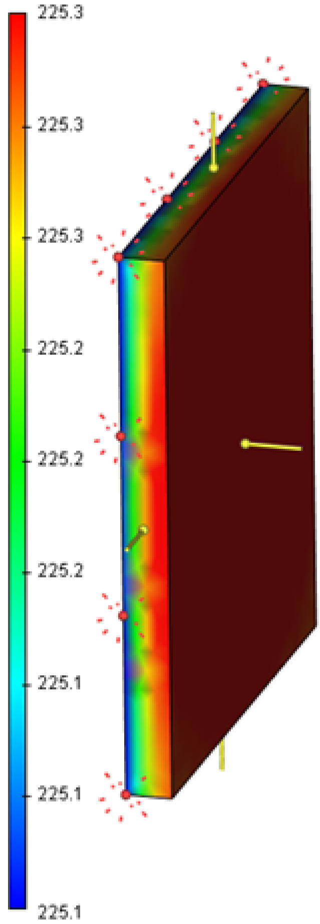
Similarly to the previous step, let's take a look at the temperature field in 100 minutes.



With scale setting in the temperature range of the left plate



With scale setting in the temperature range of the central plate

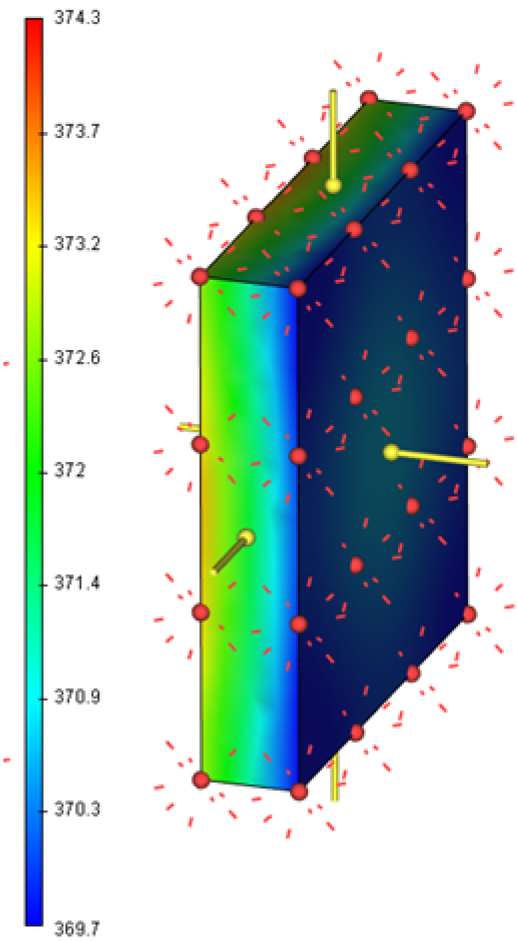


With scale setting in the temperature range of the right plate

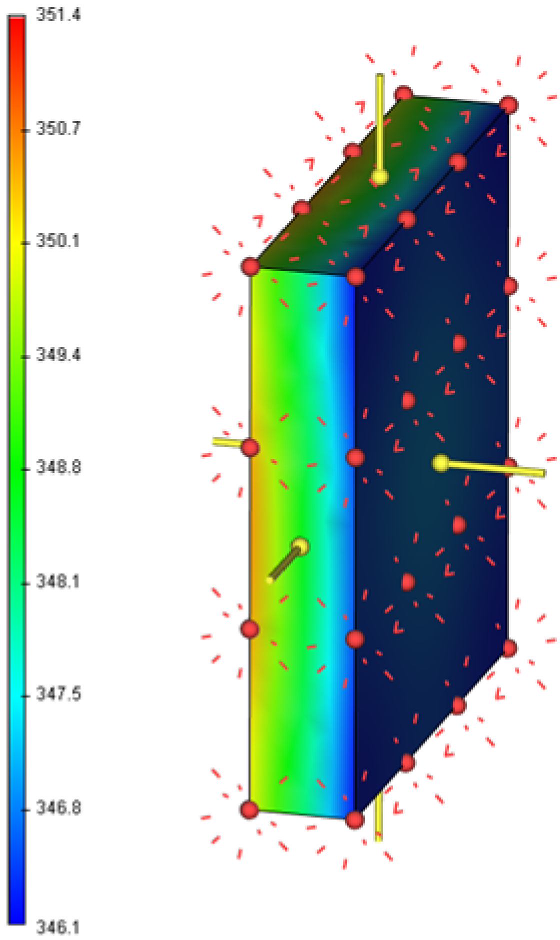
There is a considerable decrease in temperature of all the bodies. At the same time the right plate with the lowest radiation parameter cooled down the least. You can see temperature gradient from the emitting plate to the non-emitting one at the side plates. The centre of the left plate shows a clearly defined circle of increased temperature: the radiation from the central plate to the marginal areas of the left plate brings less energy.

Let's change the problem statement once again: remove the right plate and set a constant temperature for the left face of the left plate. Now there are no visible elements for the right face of the central plate and it gets radiant energy from the environment only. We can change the loading at the right face of the central plate to **Radiation to Space. The change doesn't affect the calculation result in this case (in case the ambient temperature is** 20 °C for the new loading as well). But it speeds up the calculation as when radiating to the environment there is no need to solve the problem of defining radiating elements visibility.

Let's compare the temperature field at the central plate in the first calculation variant (three plates) and in the third one (without the right plate).



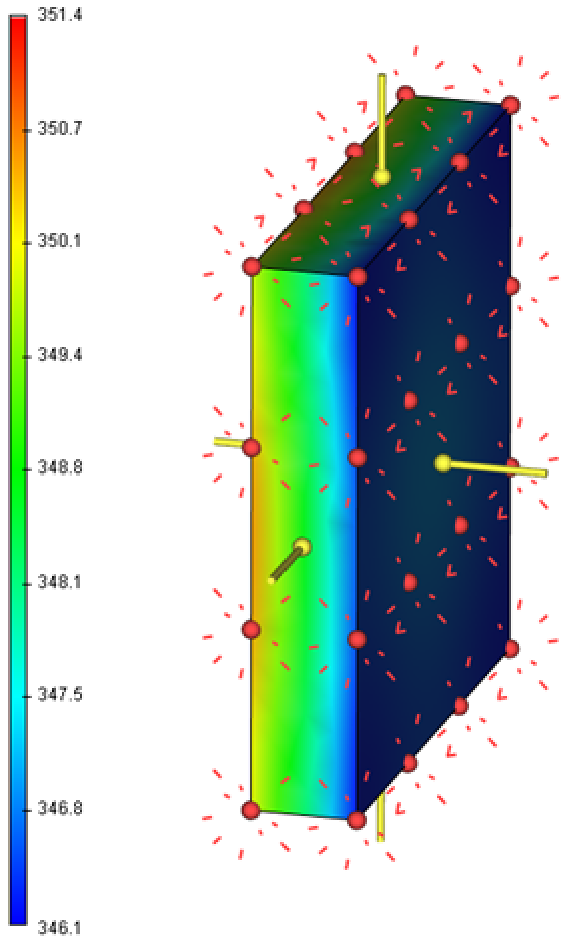
Three plates



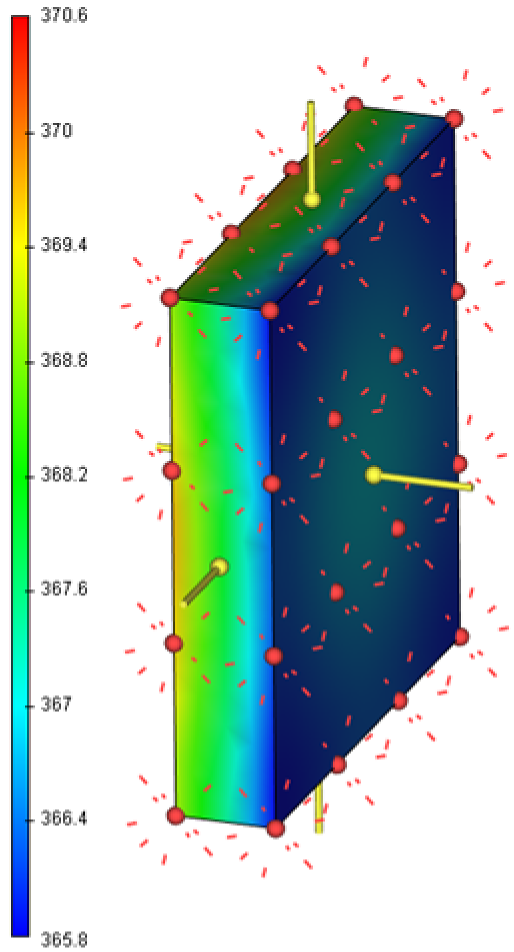
Without the right plate

As you can see from the results, the central plate temperature without energy inflow from the right plate is lower.

Let's change the initial problem statement and set starting temperature of the central plate as 20 °C, it is equal to the ambient temperature. As was done before, let's take a look at the temperature field in 100 minutes. Compare the calculated temperature field at the central plate with the initial variant of the problem statement. As you can see from the pictures, the temperature fields differ by approximately 4 °C at a similar gradient. It proves that by 100 minutes the process approaches a stationary one.



Three plates, initial temperature 500 °C



Three plates, initial temperature 20 °C

Processing of Results (Postprocessing)

How to make a PDF report?

The initial report is generated as html which can be opened in any editor, e.g. in MS Office Word. If needed, you can change the report format by means of this software: save the file as *.pdf (Save as -> other formats -> PDF (*.pdf). In case you use Windows 10 OS, you can print the report as *.pdf from the browser by choosing «Microsoft PDF printer» in the printer settings.

Verification Examples

In this chapter we review the results of solving several model studies that have an analytical solution, in order to assess the accuracy of the finite element analysis system output. All examples brought up here can be found in the "Verification examples" library of files.

Each verification example contains description of the formulation of the physical problem, description of its finite element formulation, and also results of the finite element modelling with the estimate of the relative error of the obtained results as compared to the analytical solution taken as "exact". In several problems we show the plots of the relative errors depending on the degree of refinement of the finite element discretization.

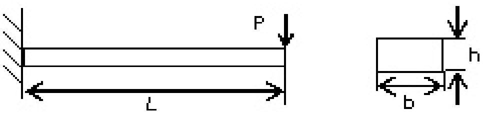
The files of verification examples contain the solved problems that correspond to the given description. For several problems, having a large quantity of results (for example, forced vibrations), the results cannot be saved in the library of verification examples (to decrease the size of the file). In this case, in order to view the results of the finite element modelling, it is necessary to recalculate such problems.

> Remark. It should be noticed that the results of the finite-element calculations depend on the finite element mesh, and therefore, the numerical results of calculations shown in the tables and also their relative errors can be somewhat different from the results saved in the files. This is a normal event for approximate numerical calculations.

Examples of Solving Studies in Statics

Bending of a Cantilevered Beam under a Concentrated Load

Consider a cantilevered beam of length L, loaded with the force P at the right-hand end. The beam cross-section is a rectangle of width b and height h.

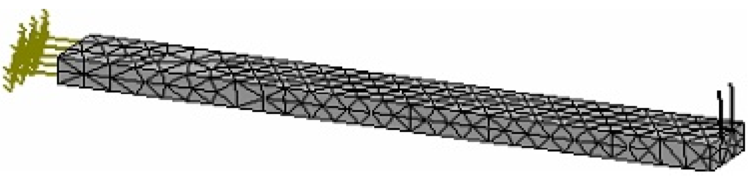


Sought is the maximum beam deflection.

Assume P=2000 N, L=0.5 m, b=0.05 m, h=0.02 m.

Material characteristics assume default values: the Young's modulus E = 2.1E+011 Pa, Poisson's ratio v = 0.28.

The left-hand end of the beam is fixed, and the right-hand end subjected to the load of amount P, directed vertically downward.



Finite-element model of the beam with loads and restraints

The analytical solution appears as:

$$w = \frac{Pl^3}{3EJ} = 1.1905 \times 10^{-2} \text{ m,}$$

where P – is the force, l – the beam length, E – the material Young's modulus, $J = \frac{b \cdot h^3}{12}$ – the section's moment of inertia.

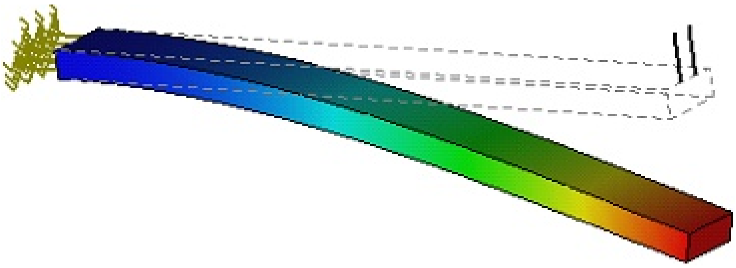
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	**Number of Nodes**	**Number of Finite Elements**
quadratic tetrahedron (10 nodes)	315	764

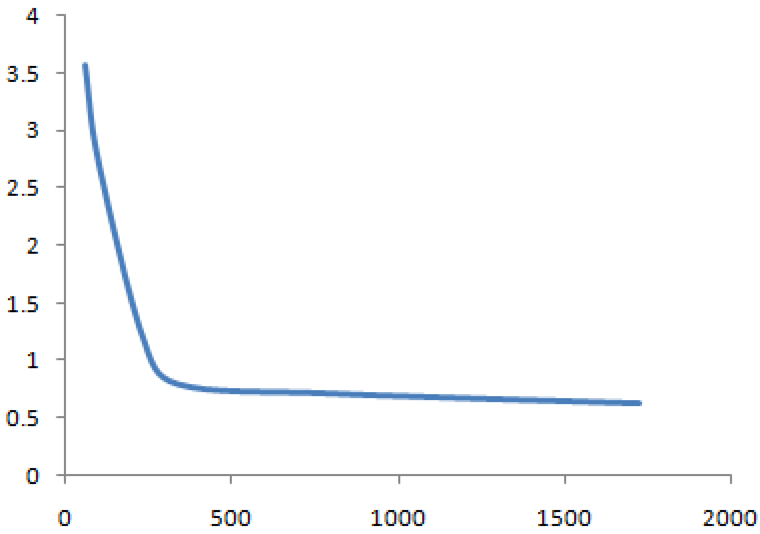
Таблица 2.
Result «Displacement»

Numerical Solution Displacement w^* **,** ***m***	**Analytical Solution Displacement** w **,** ***m***	**Error** $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1.1826E-002	1.1905E-002	0.66



Displacements of beam points

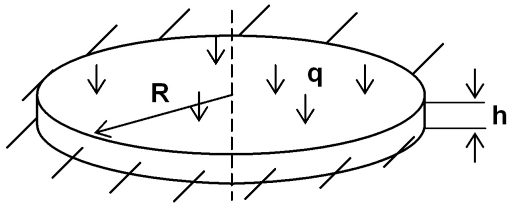
Conclusions:
The relative error of the numerical solution compared to the analytical solution is equal to 0,7% for quadratic finite elements.



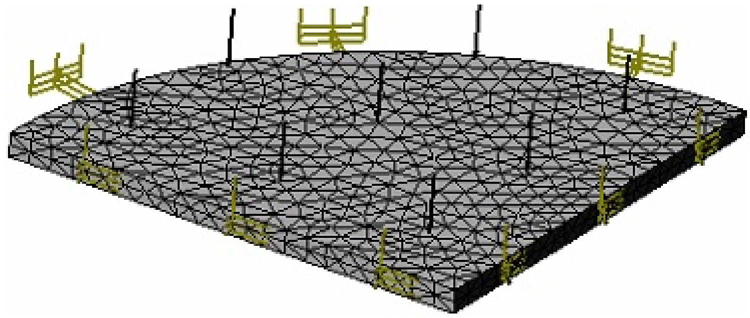
Dependence of the relative error on the number of finite elements

Static Analysis of a Round Plate Clamped along the Contour

We need to find the maximum deflection of a round plate of radius R and thickness h, which is clamped (fixed) along the contour and is loaded with a uniform pressure q distributed on the top face of the plate.



Because of the symmetry in this study, we will work with one quarter of the plate.
Assume the plate radius R=0.2 m, thickness h = 0.003 m, and the pressure q = 10 кN/m2. Material characteristics assume default values: the Young's modulus E = 2.1E+011 Pa, the Poisson's ratio v = 0.28.
Next, we need to apply boundary conditions. The side surface of the plate will be fully restrained, whereas the free faces introduced after discarding s of the plate are subjected to partial restraints in the normal to the faces direction, because the points in the sections cannot have extra displacements in the normal direction due to the symmetry. Pressure in the amount of 10 KN/m2 is applied to the top face of the plate.



Finite element model of the plate with loads and restraints

There is an analytical solution to this study. The deflection at the plate center is calculated by the formula:

$$w = \frac{qR^4}{64D} = 4.8762\times10^{-4}$$ m,

where $D = \frac{Eh^3}{12(1-\nu^2)}$ – flexural rigidity, q – is the pressure amount, R – the plate radius.
The stress on the plate contour is calculated by the formula:

$$\sigma = 0.75q\left(\frac{R}{h}\right)^2 = 3.3333\times10^7$$ Pa.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Finite element mesh parameters

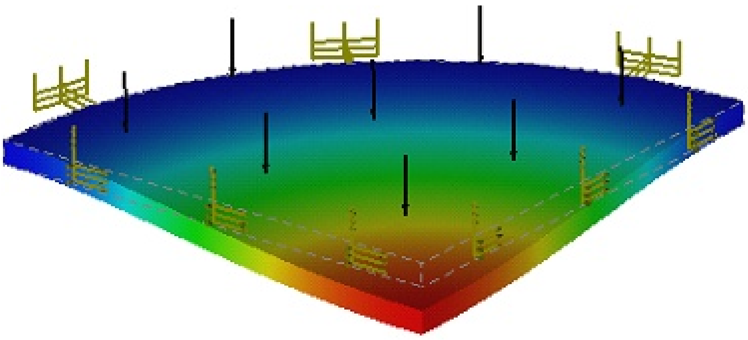
Finite Element Type	**Number of Nodes**	**Number of Finite Elements**
quadratic tetrahedron (10 nodes)	3981	11794

Table 2.
Result «Displacement»

Numerical Solution Displacement w^***, *m***	**Analytical Solution Displacement** w**, *m***	**Error** $\delta = \frac{\lvert w - w^*\rvert}{\lvert w\rvert}\times100\%$
4.8681E-004	4.8762E-004	0.16

Table 3.
Result «Stress»

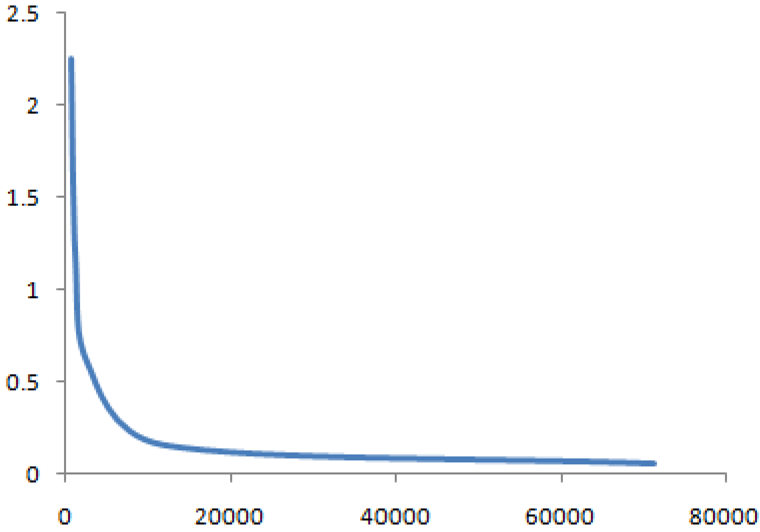
Numerical Solution Stress σ^*, $\frac{N}{m^2}$	Analytical Solution Stress σ, $\frac{N}{m^2}$	Error $\delta = \frac{\lvert\sigma - \sigma^*\rvert}{\lvert\sigma\rvert} \times 100\%$
3.0057E+007	3.3333E+007	9.83



Displacements of plate points

Conclusions:

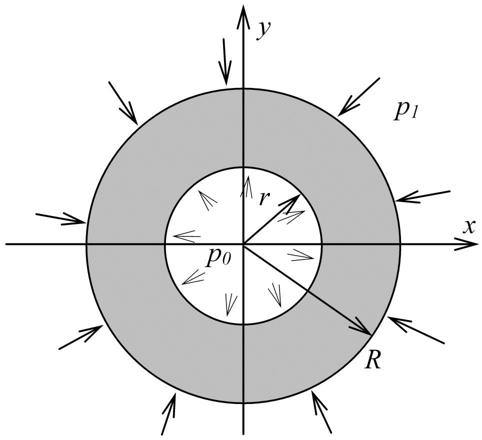
The relative error of the numerical solution compared to the analytical solution was 0,16% for displacements and 9,8% for stresses when using quadratic finite elements. The change in the relative error versus the number of quadratic finite elements is shown on the Figure below.



Dependence of the relative error on the number of finite elements

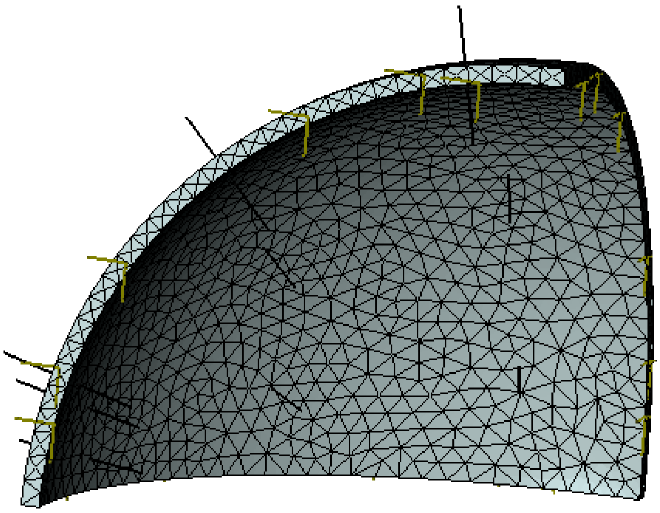
Analysis of a Spherical Pressure Vessel

Given is a spherical vessel with the inner radius r and outer radius R. The vessel is subjected to the internal pressure p0 and external pressure p1. Sought are the displacements of the inner vessel wall. Due to the symmetry of this study, we will consider the 1/8 of the sphere. Assume the following source data: Inner radius r=0.4 m, Outer radius R=0.415 m, Inner pressure p0=200 MPa, Outer pressure p1=120 MPa.



Assume the material properties as $E = 2.1 \cdot 10^{11}$ Pa, $\nu = 0.28$.

As in the previous study, we need to apply the boundary conditions to account for the discarded portion of the sphere. In this case, we need to restrain normal displacements of all flat face points. The pressure in the amount of 200 MPa and 120 MPa is applied at the inner and outer faces, respectively.



Finite element model of 1/8 of a sphere with loads and restraints

Displacement of internal surface of the sphere can be evaluated using the formula:

The analytical solution appears as: $U = A \cdot r + \dfrac{B}{r^2} = 1.4063\,mm,$ where

$$A = \frac{\dfrac{P_1}{r^3} - \dfrac{P_0}{R^3}}{(2\mu + 3\lambda)\cdot\left(\dfrac{1}{R^3} - \dfrac{1}{r^3}\right)},\quad B = \frac{P_1 - P_0}{4\mu\cdot\left(\dfrac{1}{R^3} - \dfrac{1}{r^3}\right)},\quad \mu = \frac{E}{2\cdot(1+\nu)},\quad \lambda = \frac{E\cdot\nu}{(1+\nu)\cdot(1-2\nu)}.$$

In the spherical coordinate system the stresses can be expressed as follows:

$$\sigma_\rho(\rho) = \frac{r^{-3}P_1 - R^{-3}P_0}{(R^{-3} - r^{-3})} - \frac{(P_1 - P_0)}{(R^{-3} - r^{-3})}\rho^{-3},\quad \sigma_t(\rho) = \frac{r^{-3}P_1 - R^{-3}P_0}{(R^{-3} - r^{-3})} + \frac{(P_1 - P_0)}{2\cdot(R^{-3} - r^{-3})}\rho^{-3}.$$

The equivalent stresses are found by the formula:

$$\sigma_{\text{экв}}(\rho) = \sqrt{(\sigma_\rho(\rho))^2 + (\sigma_t(\rho))^2 - 2\cdot\sigma_\rho(\rho)\cdot\sigma_t(\rho)}.$$

The equivalent stresses on the inner surface of the sphere are $\sigma_{equiv}(r) = 1148\,MPa$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	Number of Nodes	Number of Finite Elements
quadratic tetrahedron (10 nodes)	1846	5338

Table 2.
Result «Displacement»

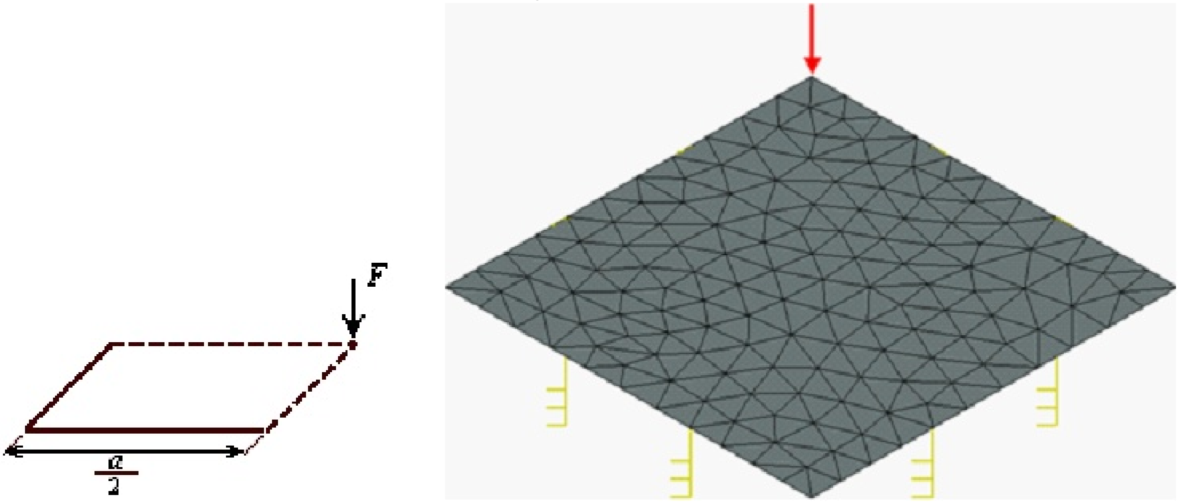
Numerical Solution Displacement w^*, *m*	Analytical Solution Displacement w, *m*	Error $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1.4067E-003	1.4063E-003	0.03

Table 3.
Result «Equivalent stresses»

Numerical Solution Stress σ^*, $\frac{N}{m^2}$	Analytical Solution Stress σ, $\frac{N}{m^2}$	Error $\delta = \frac{\lvert \sigma - \sigma^* \rvert}{\lvert \sigma \rvert} \times 100\%$
1.1499E+009	1.1480E+009	0.17

Square Plate Subjected to Force at Center

Consider a rigidly supported plate subjected to a force applied at the center.
The problem of static analysis is solved by using plate finite elements. Only a quarter of the plate is analyzed due to symmetry conditions on the corresponding edges (restrained displacements in the direction of axis of the local coordinate system normal to the face plane; restrained rotations).



Finite element model of plate with loads and restraints

Analytical solution for the deflection at the point under the force is given by formula: $w = 0.0224\frac{Pa^2}{D}$, $D = \frac{Eh^3}{12(1-\nu^2)}$.

Let us use the following data: the length and width of the plate $a = 500mm$, plate thickness is $h = 3mm$, applied concentrated force is $P = 50\ kgs$ (or $P = 490.3325N$).

Elastic properties of the material are taken as: $\mathrm{E} = 2.1 \times 10^{11}\ Pa$, $\nu = 0.28$.

Analytical solution can be expressed as: $w = 5.3557 \times 10^{-3}\ m$.

After carrying out calculations with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

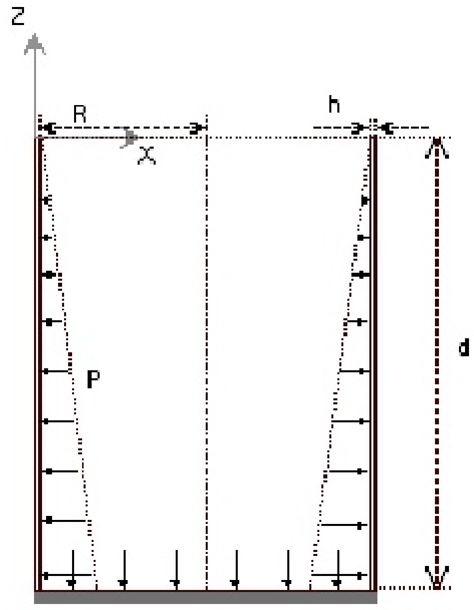
Mesh Number	**Finite Element Type**	**Number of Nodes**	**Number of Finite Elements**
1	quadratic tetrahedron (10 nodes)	8083	24222
2	linear triangle (6 nodes)	3818	7426
3	quadratic triangle (6 nodes)	3818	7426

Table 2.
Result «Displacement»

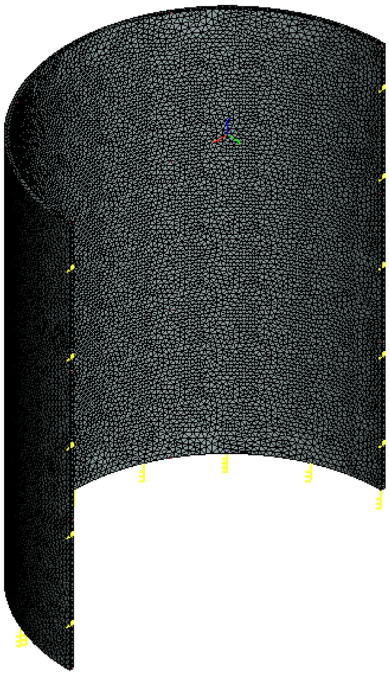
Mesh Number	**Numerical Solution Displacement** w^* **, *m***	**Analytical Solution Displacement** w **, *m***	**Error** $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1	5.3700E-003	5.3557E-003	0.27
2	5.4078E-003	5.3557E-003	0.97
3	5.3763E-003	5.3557E-003	0.38

Cylindrical Reservoir with Walls of Constant Thickness

The reservoir undergoes the pressure of liquid, as shown on the picture. The bottom of reservoir is embedded into an absolutely rigid foundation.



In most cases in practice the thickness of reservoir wall h is small compared to both the radius R, and the depth of the reservoir d. Taking into consideration this and the fact that the bottom of the reservoir does not experience any deformations, it is feasible to model the reservoir as the cylindrical shell whose bottom edge is clamped.



Finite element model of structure with loads and restraints

Analytical solution of the problem takes the form:

$$w = e^{-\beta \hat{z}}\left(C_1 \cos(\beta \hat{z}) + C_2 \sin(\beta \hat{z})\right) - \frac{\rho g (d - \hat{z}) R^2}{Eh},$$

where:

$\hat{z} = z + d$,

$C_1 = \frac{\rho g R^2 d}{Eh}$, $C_2 = \frac{\rho g R^2}{Eh}\left(d - \frac{1}{\beta}\right)$,

$\beta^4 = \frac{3(1-\nu^2)}{R^2 h^2}$,

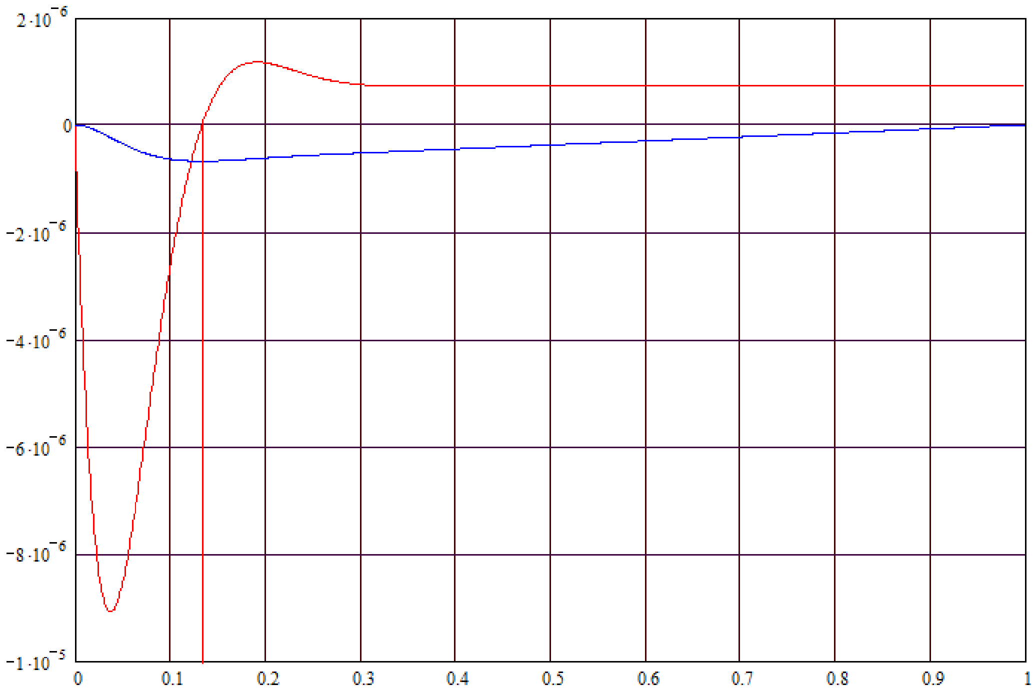
ρ – density of liquid,

g – gravitational acceleration ($\approx 9.8\ m/s^2$)

Let us use the following data: depth of reservoir $d = 1000mm$, radius $R = 200mm$, thickness of reservoir $h = 3mm$, density of liquid $\rho = 1000\ kg/m^3$.

Elastic properties are taken as: $E = 2.1 \times 10^{11}\ Pa$, $\nu = 0.28$.

Thus, $w = 6.1366 \times 10^{-7}\ m$ (maximum is at $\hat{z} = 0.134\ м$ from the foundation – see the graph).



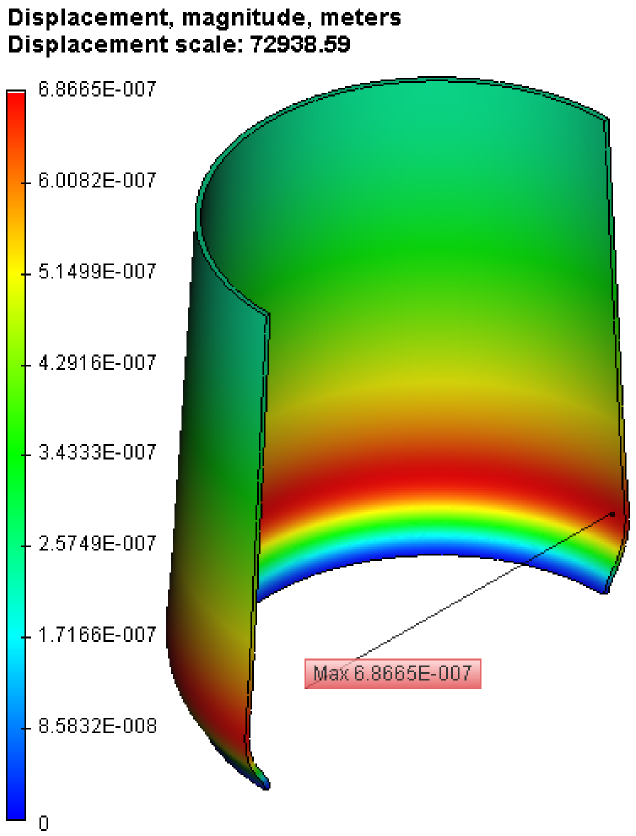
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh Number	Finite Element Type	Number of Nodes	Number of Finite Elements
1	quadratic tetrahedron (10 nodes)	24325	74887
2	linear triangle (6 nodes)	4549	8864
3	quadratic triangle (6 nodes)	4549	8864

Table 2.
Result «Displacement»

Mesh Number	Numerical Solution Displacement w^*, *m*	Analytical Solution Displacement w, *m*	Error $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1	6. 8665E-007	6. 7327E-007	1.98
2	6. 9118E-007	6. 7327E-007	2.66
3	6. 9023E-007	6. 7327E-007	2.52

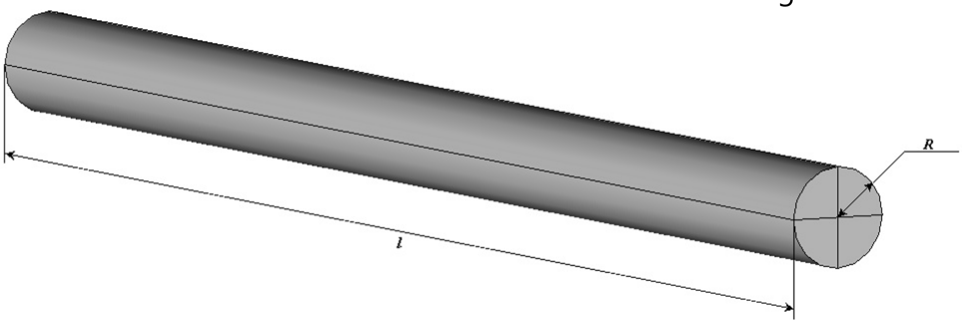


Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements was 2% for quadratic tetrahedral finite elements and 2,6-2,5% for triangular linear/quadratic finite elements – but the required number of finite elements was 8,4 times smaller.

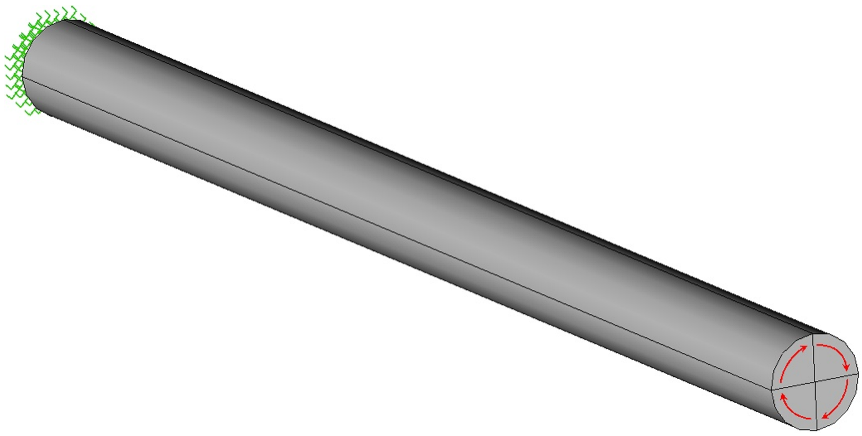
Torsion of Shaft with Circular Cross-section

Consider a shaft with circular cross-section of radius R. Length of shaft is l (see figure).



Select the coordinate system with the z-axis directed along the axis of the shaft, and the coordinate $z = 0$ located at the left edge of the shaft.

The shaft is subjected to the externally applied torque τ. The torque is applied at the right end of the shaft, the left end of the shaft is rigidly clamped.



Let us use the following initial data: length l of the shaft is $0.6m$, radius of cross-section R of the shaft is $0.02m$, the magnitude of the applied torque is $\tau = 100\ N \cdot m$.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $\nu = 0.28$.

To find the angle of twist, let us use the following relation:

$$\varphi = \int_0^z \frac{\tau}{GJ_\rho} dz + \varphi_0,$$

where φ_0 – angle of twist of the cross-section $z = 0$, $G = \dfrac{E}{2(1+\nu)}$ – shear modulus, $J_\rho = \dfrac{\pi R^4}{2}$ – polar moment of inertia of the circular cross-section.

Since, by formulation, the left end of the shaft is clamped, $\varphi_0 = 0$. Then, at a distance $z = 0.5l$ from the clamped edge of the shaft, the angle of twist φ is given by the formula:

$$\varphi_{0.5l} = \frac{0.5\tau l}{GJ_\rho}$$

Thus, $\varphi_{0.5l} = 1.4551 \times 10^{-3} rad$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of Finite Elements**
quadratic tetrahedron (10 nodes)	4526	89457	18264

Absolute value of displacement (at $z = 0.5l$) Δu = 2.9139E-005 m.

Table 2.
Result «Angle of twist»

Numerical Solution **Angle of twist** $\psi = \arcsin\left(\dfrac{\Delta u}{R}\right)$, **rad**	**Analytical Solution** **Angle of twist** φ, **rad**	**Error** $\delta = \dfrac{\lvert\varphi - \psi\rvert}{\lvert\varphi\rvert} \times 100\%$
1.4570E-003	1.4551E-003	0.027

Conclusions:

The relative error of the numerical solution compared to the analytical solution was 0.027%.

Bar Subjected to Self-Weight

Consider a bar of radius R and length l, suspended at the upper edge and stretched under the action of self-weight (see figure).

Let us use the following data: length of bar l is equal to 1m, radius of cross-section of the bar R is equal to 0.02m.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $\nu = 0.28$, $\rho = 7800 \frac{kg}{m^3}$.

Total elongation of the bar under the action of the self-weight can be determined from the formula:

$$\Delta l = \frac{\gamma l^2}{2E},$$

where γ – specific weight of the bar's material, that is $\gamma = \rho \cdot g$, $g = 9.80665 \frac{m}{c^2}$.

The stress in the cross-section of the bar located at a distance x from lower (unconstrained) edge can be evaluated from formula:

$$\sigma = \gamma \cdot x$$

Thus, $\Delta l = 1.8212 \times 10^{-7} m$; $\sigma = 3.8246 \times 10^4 \frac{N}{m^2}$ at $x = \frac{l}{2}$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
quadratic tetrahedron (10 nodes)	2482	16089	9518

Table 2.
Result «Displacement»

Numerical Solution Displacement w^***, m**	**Analytical Solution Displacement** w**, m**	**Error** $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1.8177E-007	1.8212E-007	1.9410E-001

Table 3.
Result «Stress»

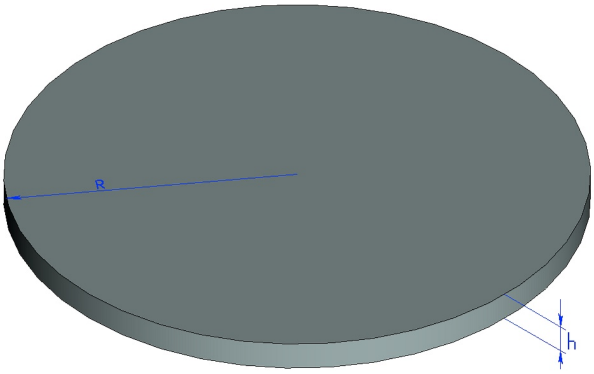
Numerical Solution Stress σ^*, $\frac{N}{m^2}$	**Analytical Solution Stress** σ, $\frac{N}{m^2}$	**Error** $\delta = \frac{\lvert \sigma - \sigma^* \rvert}{\lvert \sigma \rvert} \times 100\%$

3.8249E+004	3.8246E+004	8.0139E-003

Conclusions:
The relative error of the numerical solution compared to the analytical solution was 0.19% for displacements and 0.008% for stresses when using quadratic finite elements.

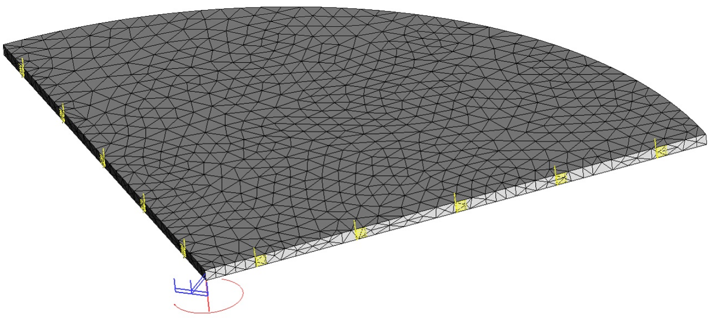
Analysis of Rotating Solid Disc of Constant Thickness

Consider a disc of radius R and thickness h (see figure).



In analysis we consider only $\frac{1}{4}$th part of the disc with application of symmetry conditions at the corresponding edges (restraint of displacements in the direction of the axis of the local coordinate system perpendicular to the plane of the edge).

The disc is subjected to the centrifugal force $F = \rho \cdot \omega^2 R$, where ρ - is the mass of the unit volume of the disc's material, ω - angular velocity of rotation.



Let us use the following data: radius of disc R is equal to $0.457m$, thickness of disc h is equal to $0.01m$, magnitude of angular velocity of rotation ω is equal to $300\frac{rad}{c}$.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $\nu = 0.28$, $\rho = 7800\frac{kg}{m^3}$

For this problem, displacements u can be determined from the formula:

$$u = \frac{1}{E}\left((1-\nu)C_1 r - (1-\nu)C_2\frac{1}{r} - \frac{(1-\nu^2)}{8}\rho\omega^2 r^3\right),$$

where constants $C_1 = \frac{3+\nu}{8}\rho\omega^2 R^2$, $C_2 = 0$ are determined from boundary conditions.

The maximum displacement u_{max} is expected to be at $r = R$, that is $u_{max} = \frac{(1-\nu)\rho\omega^2 R^3}{4E}$.

Stress components σ_r, σ_θ are found as:

$$\sigma_r = \frac{3+\nu}{8}\rho\omega^2\left(R^2 - r^2\right),$$

$$\sigma_\theta = \frac{3+\nu}{8}\rho\omega^2 R^2 - \frac{1+3\nu}{8}\rho\omega^2 r^2.$$

These stresses take the maximum value at the center of the disc where:

$$\sigma_r = \sigma_\theta = \frac{3+\nu}{8}\rho\omega^2 R^2$$

Thus, $u_{max} = 5.7430\times10^{-5}\,m$, $\sigma_r = \sigma_\theta = \sigma = 6.0111\times10^{7}\,\frac{N}{m^2}$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
quadratic tetrahedron (10 nodes)	1545	26874	4340

Table 2.
Result «Displacement»

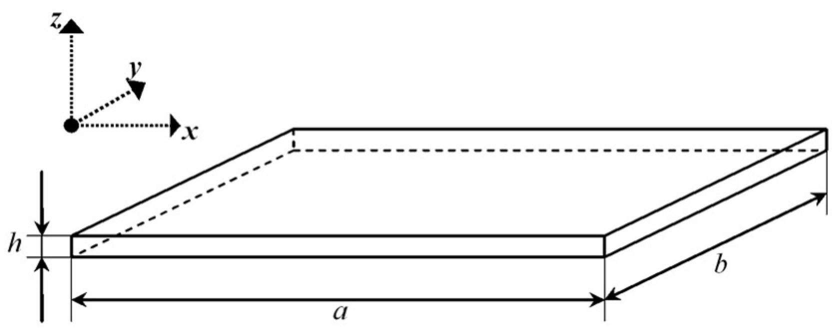
Numerical Solution Displacement w^*, **м**	**Analytical Solution Displacement** w, **м**	**Error** $\delta = \frac{\lvert w - w^*\rvert}{\lvert w\rvert}\times 100\%$
5.8033E-005	5.7430E-005	1.0390E+000

Table 3.
Result «Stress»

Numerical Solution Stress σ^*, $\frac{H}{M^2}$	**Analytical Solution Stress** σ, $\frac{H}{M^2}$	**Error** $\delta = \frac{\lvert \sigma - \sigma^*\rvert}{\lvert \sigma\rvert}\times 100\%$
6.4858+007	6.0111E+007	7

Simply-Supported Rectangular Plate Subjected to Sinusoidal Load

Consider a rectangular plate with the sides a, b and thickness h (see figure).



Thickness of plate h is considerably smaller than length of its sides a, b.

Plate is subjected to the load distributed across the surface of the plate according to the law:

$q = q_0 \sin\left(\frac{\pi x}{a}\right)\sin\left(\frac{\pi y}{b}\right)$, where q_0 is the intensity of the load at the center of the plate.

Consider the case when edges of the plate are simply-supported.

Let us use the following data: length of side a of the plate is equal to $0.5m$, length of b side of the plate is equal to $0.4m$, thickness of the plate $h = 0.003m$, intensity of the load at the center of the plate $q_0 = 100\frac{N}{m^2}$.

Material characteristics: $E = 2.1 \times 10^{11} \Pi a$, $\nu = 0.28$.

Analytical solution of the problem has the form:

$$w = \frac{q_0}{\pi^4 D\left(\frac{1}{a^2}+\frac{1}{b^2}\right)^2}\cdot \sin\left(\frac{\pi x}{a}\right)\cdot \sin\left(\frac{\pi y}{b}\right),$$

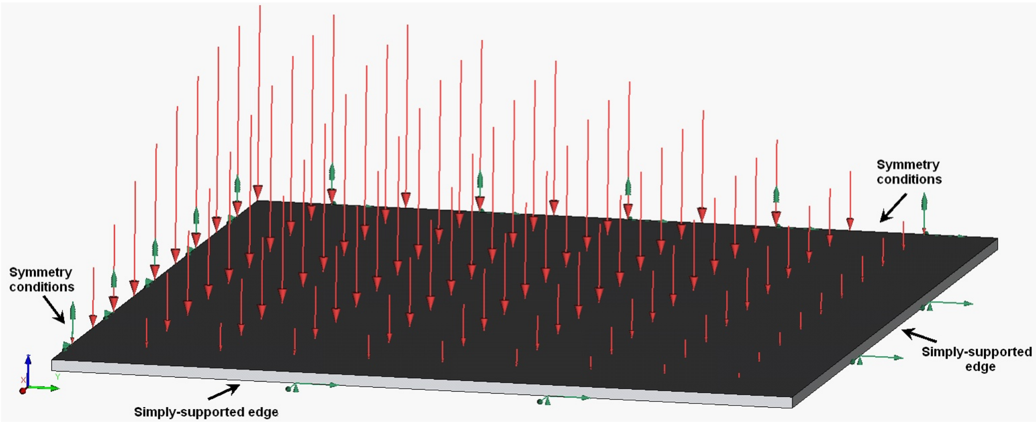
where $D = \frac{Eh^3}{12(1-\nu^2)}$ – cylindrical stiffness of the plate.

The maximum deflection takes place at the center of the plate. Substituting $x = \frac{a}{2}$ and $y = \frac{b}{2}$ into the equation above we obtain:

$$w_{max} = \frac{q_0}{\pi^4 D\left(\frac{1}{a^2}+\frac{1}{b^2}\right)^2}$$

Thus, $w_{max} = 1.9059 \times 10^{-5} m$.

In analysis we consider only $\frac{1}{4}$ th part of the plate with application of symmetry conditions at the corresponding edges (restraint of displacements in the direction of the axis of the local coordinate system perpendicular to the plane of the edge; restraint of rotations).



Model of plate with loads and restraints

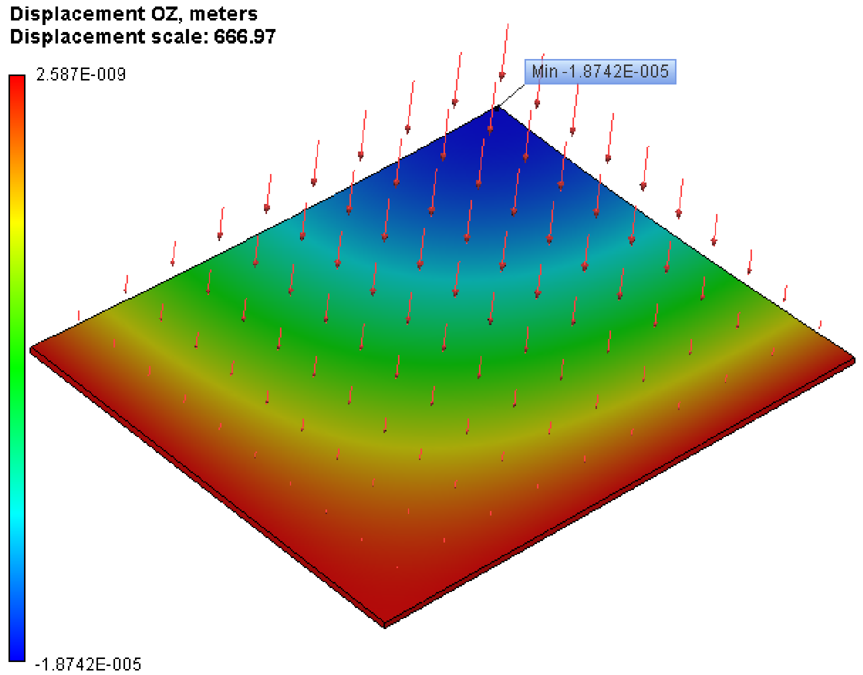
After carrying out calculation with help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh Number	**Finite Element Type**	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
1	quadratic tetrahedron (10 nodes)	5836	104094	17318
2	linear triangle (6 nodes)	2747	16472	5304
3	quadratic triangle (6 nodes)	10797	64782	5304

Table 2.
Result «Displacement»

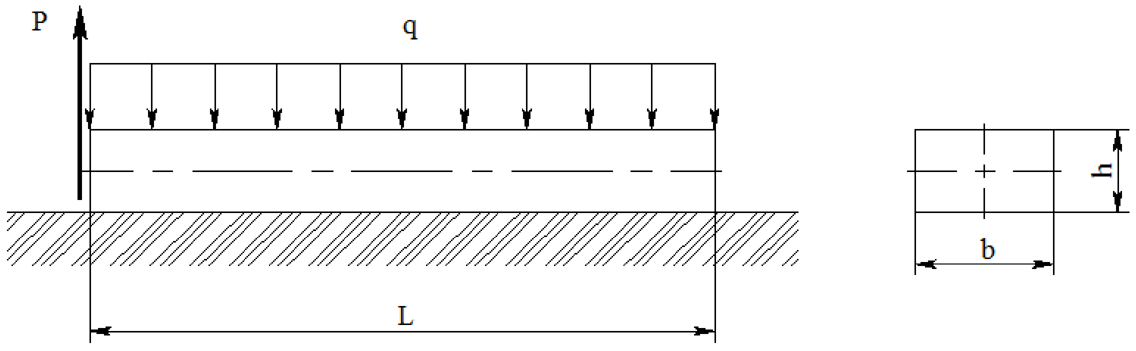
Mesh Number	Numerical solution Displacement w^*, m	Analytical solution Displacement w, m	Error $\delta = \frac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1	1.8741E-005	1.9059E-005	1.671
2	1.8798E-005	1.9059E-005	1.37
3	1.8752E-005	1.9059E-005	1.61



Conclusions:
The relative error of the numerical solution compared with the analytical solution for displacements was 1.67% and 1.60% for quadratic tetrahedron and triangular finite elements, respectively.

Semi-infinite Beam on an Elastic Foundation

Let us consider a beam on an elastic foundation. The length of the beam is L. The cross-section of the beam is a rectangle of width b and height h.

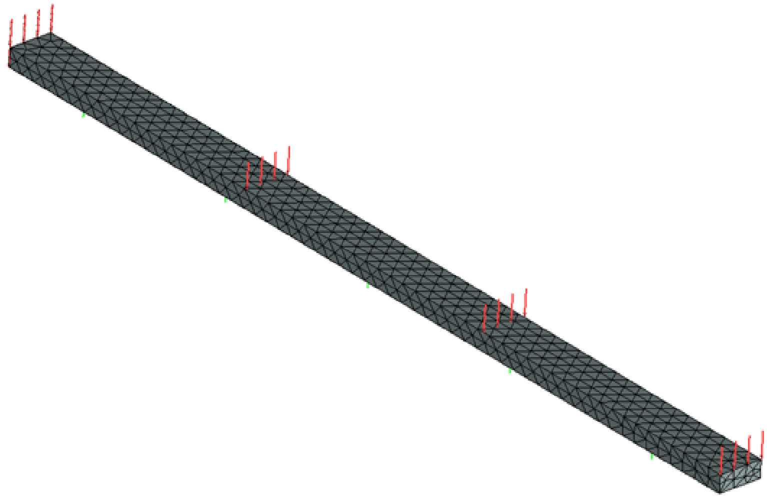


The beam is subjected to the uniformly distributed load q. The force P is applied to the left edge of the beam.

The maximum value of displacement is the sought value.

The following input data are used: q = 1000 Pa, P = 5 N, L = 1 m, b = 0.05 m, h= 0.02 m.

Characteristics of the material (steel): E = 2.1E+011 Pa, ν = 0.28.



Finite element model for specified loading and constraints.

Analytical solution can be found by the formula:

w0max =(2β·P - qL) / k·b,

where J = bh3 / 12 – moment of inertia, k – stiffness coefficient of the supporting layer (k= 3e+06 N/m3),

β = (k·b/4E·J)1/4 = 1.52136. The edge effect can be observed up to a distance of Lкэ ≈ π / β = 1.381 m measured from the left edge of the beam.

Therefore, w0max = (2*1.52136 *1000-200)/(3*106*0.05) = 18.95147 mm.

Deflection of the beam: - qL / k·b = -1,333 mm.

Before carrying out numerical calculations, let us determine the following quantities: area of the face over which the load qL is distributed: S=bL = 0.25 m2; therefore, pressure acting on this plane face is: q= qL*L/S=4000 Pa. The input value of the total stiffness applied to the lower face is: k1=k*S=3*106*0.25= 750000 N/m.

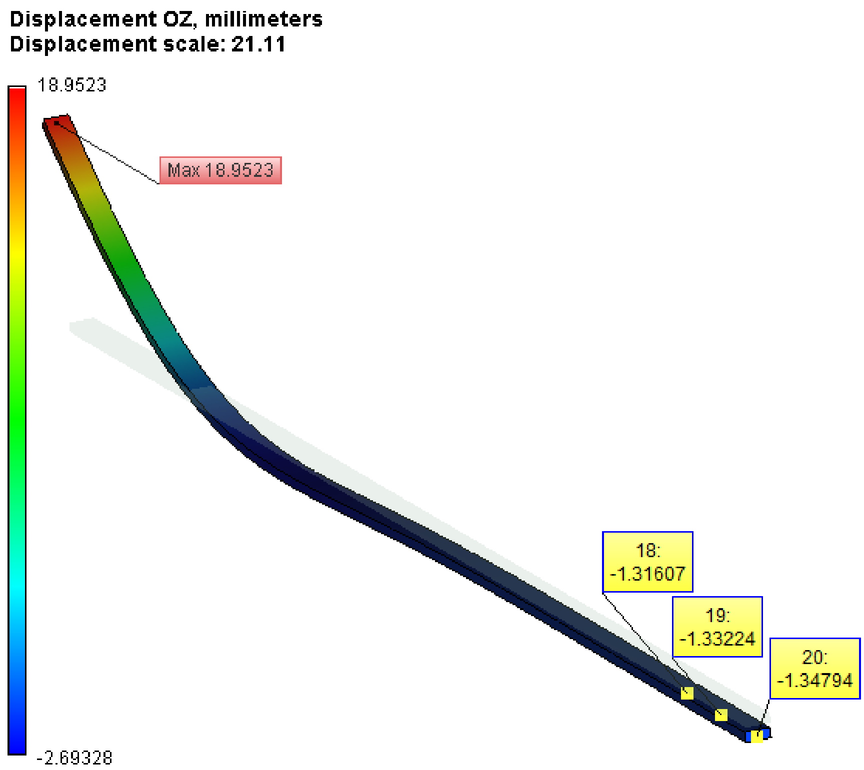
After carrying out calculations with the help of T-FLEX we obtain the following results:

Table 1. Parameters of the finite-element mesh

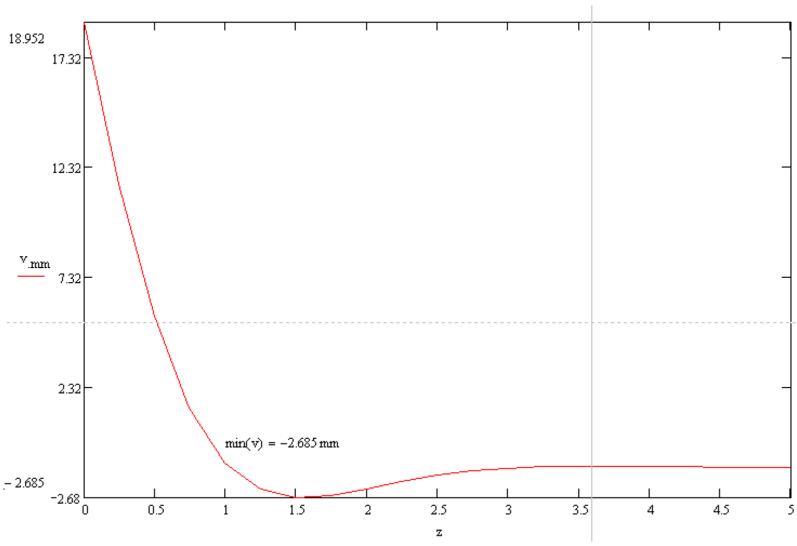
Finite element type	Number of vertices	Number of arguments	Number of finite elements
quadratic tetrahedron (4 nodes)	10100	30300	4821

Table 2. Result "Displacement"

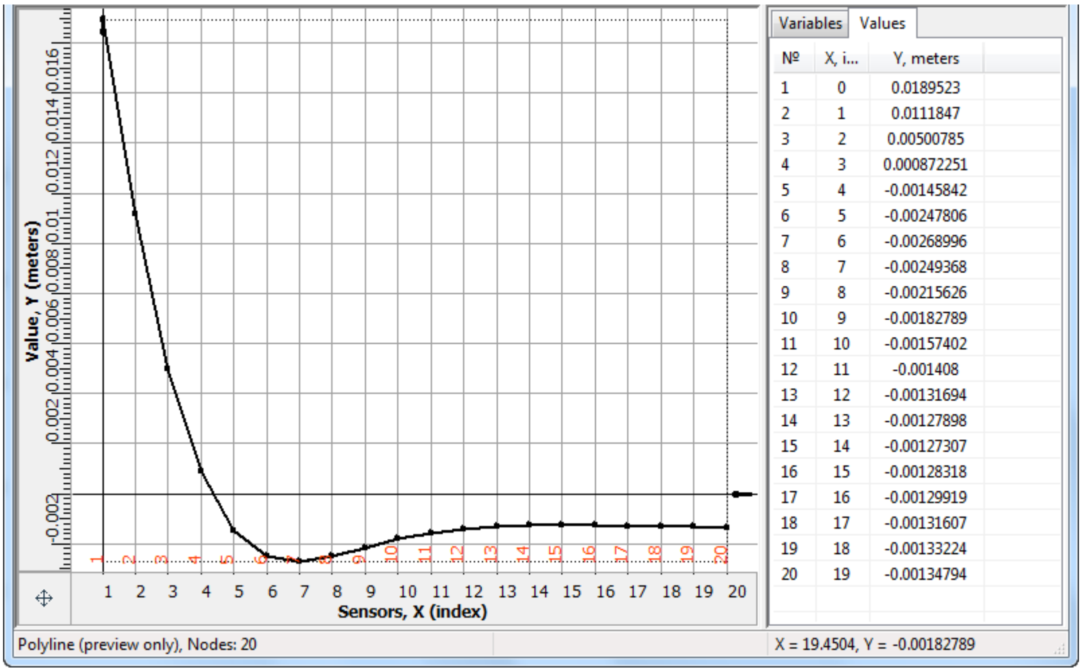
Numerical solution *w**, *mm*	Analytical solution *w*, *mm*	**Error** δ =100%* |*w** - *w* | / |*w*|
18.9523	18.9515	0.004



Results of calculations in T-FLEX Analysis (displacements along the Z-axis)



Plot of deflection of the semi-infinite beam (analytical solution)



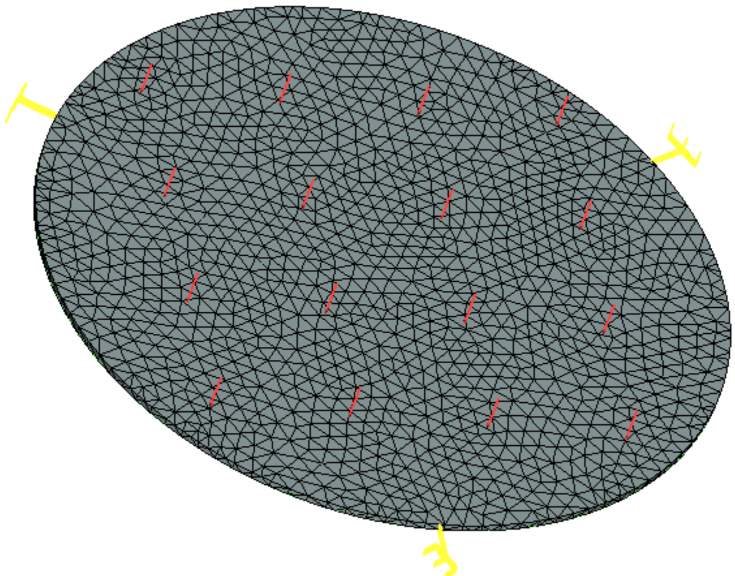
Plot of deflection of the semi-infinite beam (numerical solution)

Conclusion:

The relative error of the numerical solution compared to the analytical solution is equal to 1,3% for quadratic finite elements.

Large Deformation of a Circular Plate

Let us consider a circular plate of radius a and thickness h. The plate is fixed and subjected to the action of distributed load q.



Finite element model for specified loads and constraints.

Let us use the following input data: radius of plate a is equal to 0.25 m, thickness of plate h is equal to 0.005 m, intensity of load q is equal to 1E+011 Pa.

Material properties (steel): E=2.1E+05 MPa and n= 0.28.

Let us use the following approximate formula for calculation of the displacement at the center of the plate:

$$w_0 = \frac{qa^4}{64D} \cdot \frac{1}{1 + 0.488\frac{w_0^2}{h^2}},$$

where

$$D = \frac{Eh^3}{12(1-\nu^2)}$$

flexural stiffness of the plate.

By solving this equation for w0, we obtain the value of the maximum deflection which must occur at the center of the plate: w0= 2.3258E-003 m.

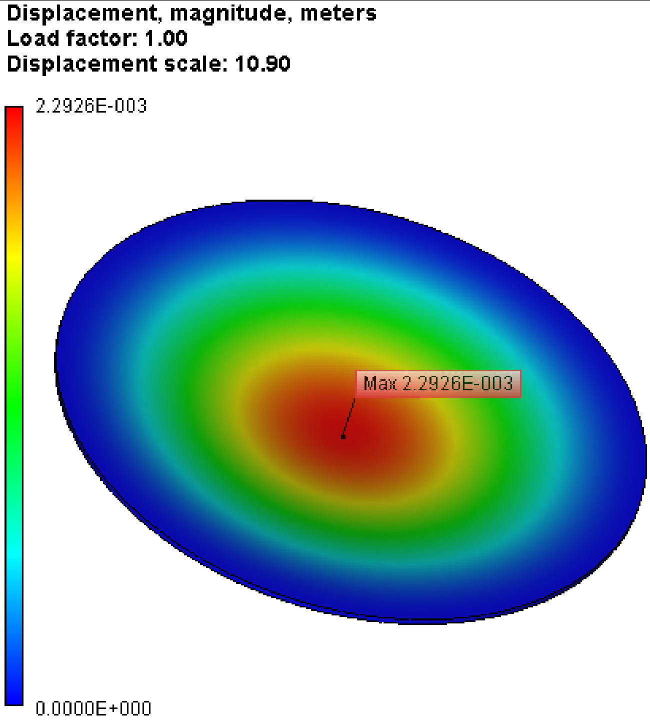
After calculations are performed with the help of T-FLEX, we obtain the following results:

Table 1.Parameters of finite element mesh

Finite element type	Number of vertices	Число конечных элементов
quadratic tetrahedrons	42624	7421

Table 2. Result "Displacement"

Numerical solution w_0*, m	Analytical solution w_0, m	Error δ =100%*|w_0*- w_0|/| w_0 |
2.2926E-003	2.3258E-003	1.3



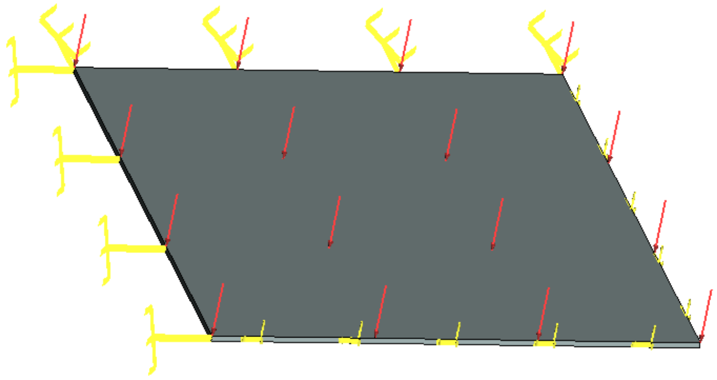
Conclusions:

The relative error of the numerical solution compared with the analytical solution was 1.3% for quadratic finite elements.

Fixed Square Plate under the Action of Distributed Load

Let us consider fixed square plate subjected to pressure.

Due to the symmetry we consider only a quarter of the plate (we constrain displacement along the axes perpendicular to lateral faces).



Finite element model with applied pressure and constraints

Analytical solution for the displacement at the center of the plate can be found from formula:

$$w_{max} = 0.00126\frac{qa^4}{D}, D = \frac{Eh^3}{12(1-\nu^2)}.$$

Let us consider the following input data: length and width of the plate a = 500 mm (for this example, we take the length of the side equal to 250 mm since only a quarter of the original plate is modeled), thickness of plate h = 3 mm, applied pressure q = 800 Pa.
Material's characteristics (steel): E = 2.1E+011 Pa, n = 0.28.
Analytical solution can be expressed as: w = 1.2288E-004 m.

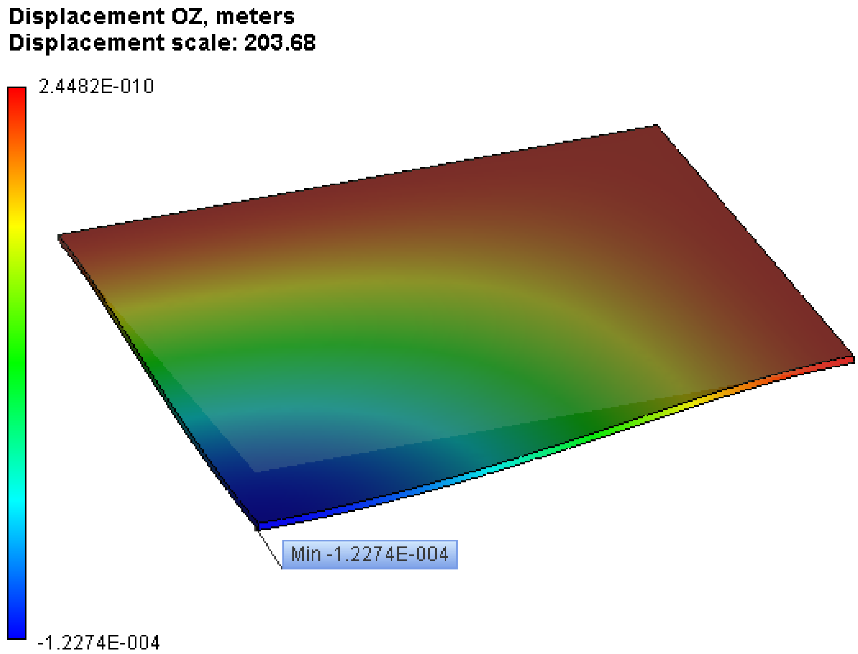
After calculations are performed with the help of T-FLEX, we obtain the following results:

Table 1.Parameters of finite element mesh

Finite element type	Number of arguments	Number of finite elements
quadratic tetrahedrons	32115	5425

Table 2. Result "Displacement"

Numerical solution w^*, m	Analytical solution w, m	Error δ =100%*|w_0*- w_0|/| w_0 |
1.2274E-004	1.2288E-004	0.15

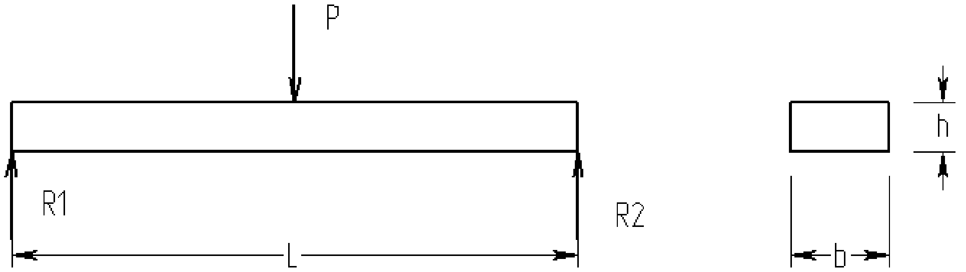


Conclusions:

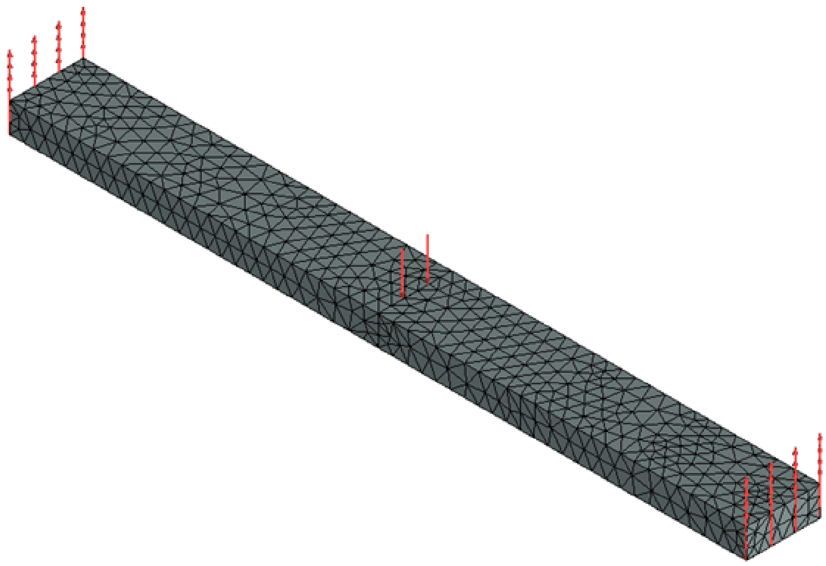
The relative error of the numerical solution for displacements compared to the analytical solution is equal to 0.15% for quadratic finite elements.

Bending of a Beam under the Action of 3 Forces

Let us consider a beam of length L, loaded with a force P in the middle and reaction forces R1, R2 at the ends. The cross-section of the beam is a rectangle of width b and height h.



The value sought – maximum displacement of the beam.
Assume: P = 1000 N, R1=500 N, R2=500 N, L = 0.5 m, b = 0.05 m, h = 0.02 m.
Material's characteristics (steel): E = 2.1E+011 Pa, n = 0.28.
Both ends of the beam are not constrained and they are subjected to loads R1, R2 directed vertically. The force P is applied in the middle of the beam.
Calculations are carried out with the enabled option «Stabilize the system » with additional stiffness equal to 1.



Finite element model for indicated loads and constraints.

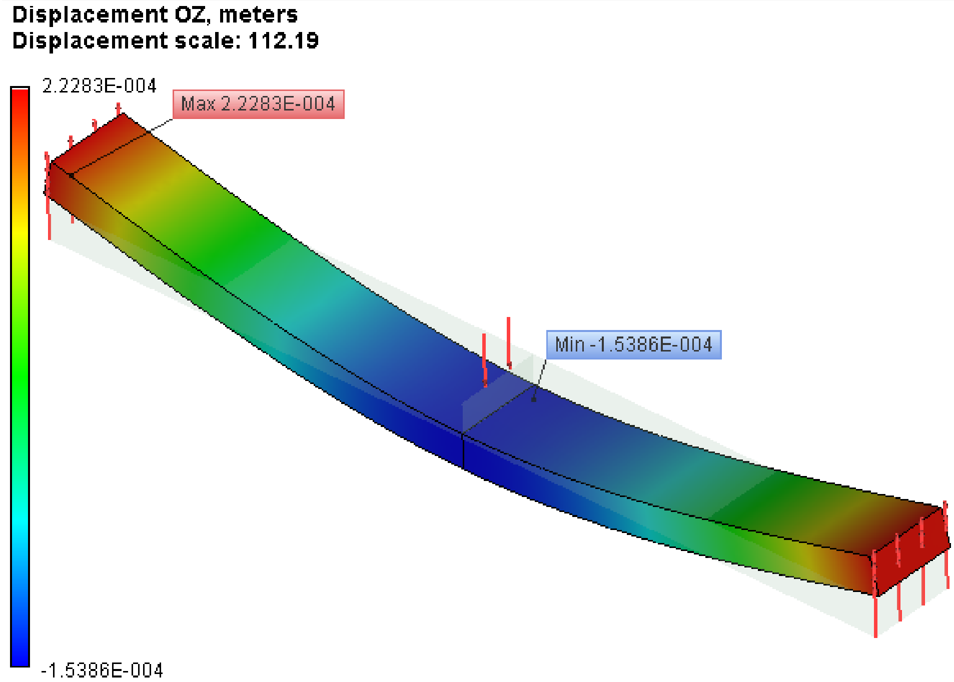
Analytical solution is given by:
w = (P . L3) / (48 . E . J) = 3.720E-004 m
where P – force, L – length of a beam, E – Young's modulus for the material, J = b . h3 / 12 – moment of inertia.
After calculations are carried out with the help of T-FLEX, we obtain the following results:
(displacement is equal to (2.2283E-004)-(-1.5386E-004)=3.7669E-004 m)

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	40626	8622

Table 2. Result "Displacement"

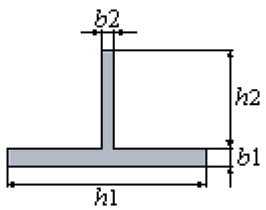
Numerical solution, m	**Analytical solution,** m	**Error** δ =100%* $\|0Z^* - 0Z\| / \|0Z\|$
3.767E-004	3.720E-004	1.26



Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 1.3% for quadratic finite elements.

Consider of a T-shape Beam



The length of the beam is equal to L. The beam is rigidly constrained at the left edge, and the force P is applied at the right edge.

Let us analyze the beam for the following input data: length L of the beam is 1 m, the length of the sides b1,h1,b2,h2 is 0.01 m, 0.1 m, 0.006 m, 0.05 m, respectively, the force P is equal to 100 N.

Material's characteristics (steel): E = 2.1E+011 Pa, n = 0.28.

Finite element model for indicated loads and constraints.

Analytical solution of the problem is sought with the help of the following equation:

$$w = \frac{PL^3}{3EJ}$$

where

$J_Y = \left(J_{Y1} + A_1 z_{01}^2\right) + \left(J_{Y2} + A_2 z_{02}^2\right)$ - moment of inertia with respect to the central axis of inertia;

$J_{Y1} = \frac{h_1 b_1^3}{12}$, $J_{Y2} = \frac{b_2 h_2^3}{12}$;

$A_1 = b_1 \cdot h_1$, $A_2 = b_2 \cdot h_2$;

z01,z02 –distance between the axes Y1 and Y, Y2 and Y, respectively.

Therefore, w= 5.6989E-004 m.

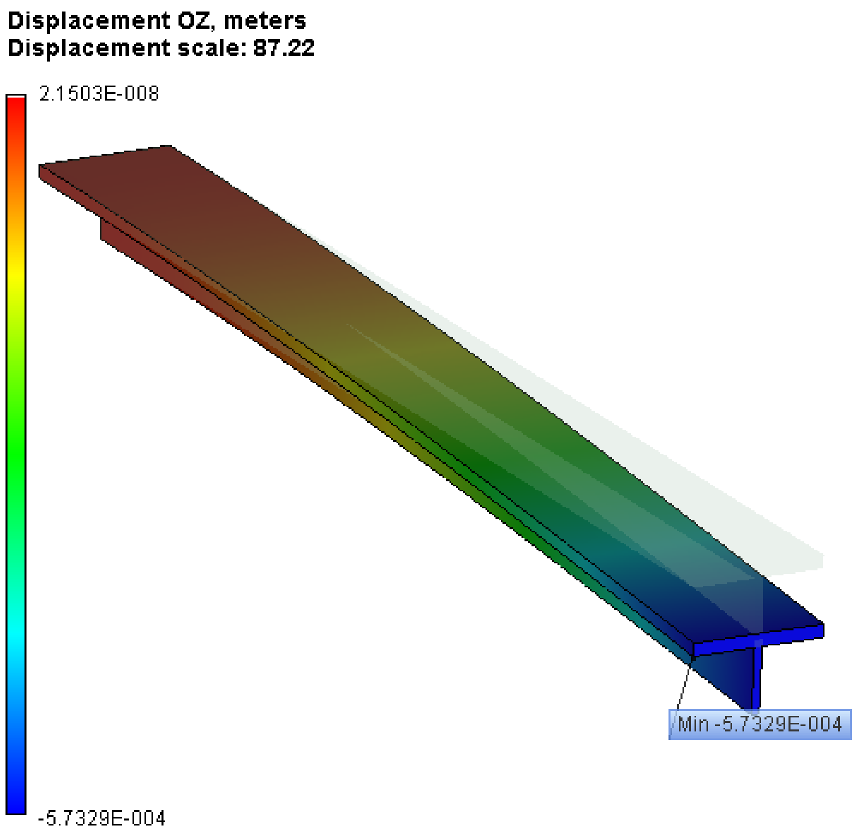
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	49053	7905

Table 2. Result "Displacement"

Numerical solution w^*, m	Analytical solution w, m	Error δ=100%* $\|w^*$ - $w\|$ / $\|w\|$
5.7335E-004	5.6989E-004	0.62

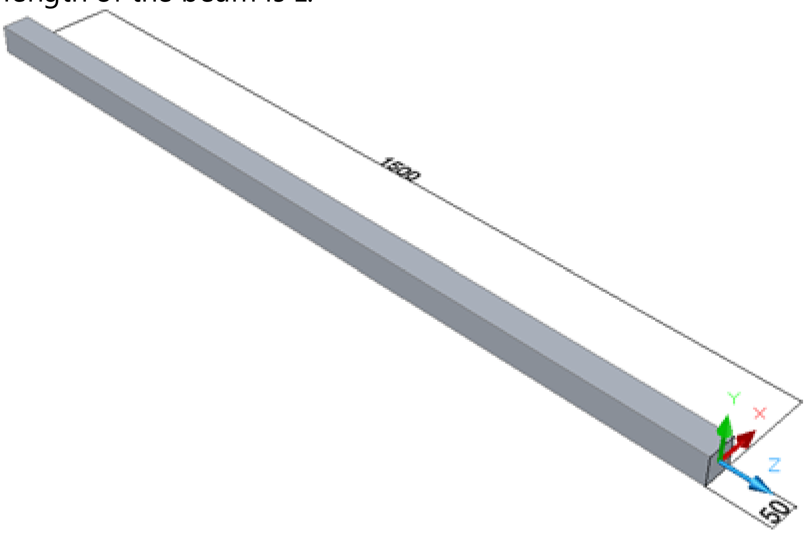


Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.62% for quadratic finite elements.

Torsion of a Beam with a Square Cross-section

Let us consider a beam with a square cross-section. The length of the side of the square is a. The length of the beam is L.



The beam is subjected to the torque Mt which is applied at the right edge perpendicularly. The left edge of the beam is rigidly fixed.

Finite element model for indicated loads and constraints.

Let us use the following input data: length L of the beam is 1.5 m, length of the side of the square a is 0.050 m, the magnitude of the applied torque is Mt is 1000 N-m.
Material's characteristics (steel AISI 1020): E = 2.0E+011 Pa, n = 0.29.
In order to find the angle of twist, we use the following relationship:

$$\varphi = \frac{M_t L}{G J_p},$$

where G=E/2(1+ ν) – shear modulus, Jp= β a4 –polar moment of inertia of a square section, β = 0.1406.
Therefore, f= 2.2168E-002 rad.
The maximum deflection is calculated from the formula:

$$\Delta u = \sin(\varphi) \cdot \frac{\sqrt{2}}{2} a$$

Therefore, Δ u = 7.8371E-004 m.
The maximum shear stress t can be obtained from the following formula:

$$\tau_{max} = \frac{M_t}{\alpha \cdot a^3}$$

where α = 0.208.
Therefore, t = 3.8462E+007 Pa.

After calculations are performed with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

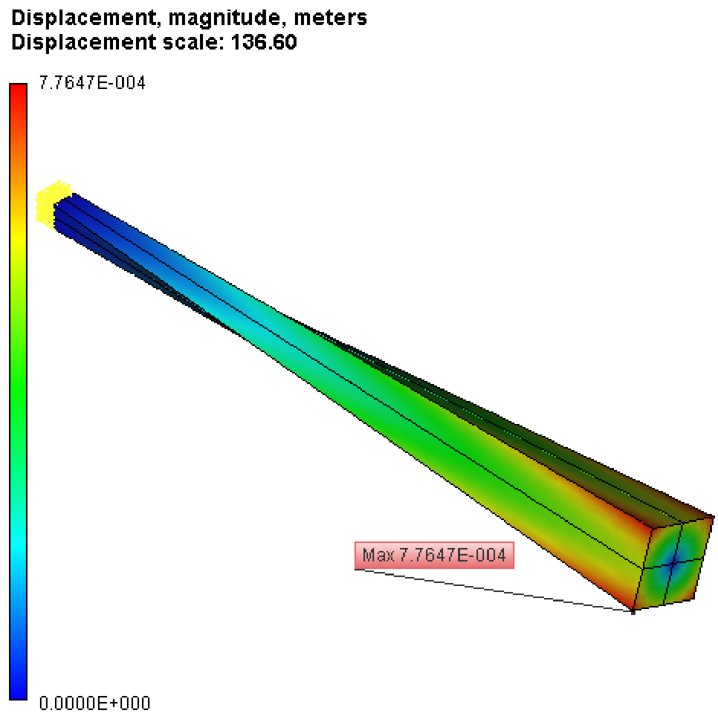
Finite element type	**Number of vertices**	**Number of finite elements**
quadratic tetrahedrons	91878	19876

Table 2. Result "Displacement"

Numerical solution Δu^*, *m*	**Analytical solution** Δu, *m*	**Error** δ = 100%* | Δu^* - Δu | / | Δu |
7.7647E-004	7.8371E-004	0.92

Table 3. Result "Shear stress"

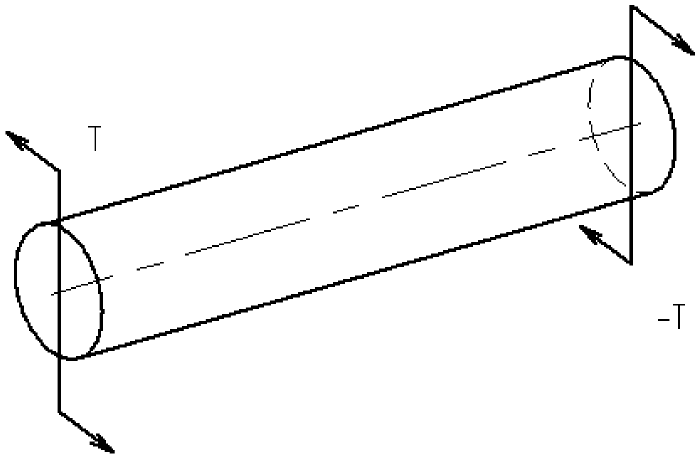
Numerical solution τ^*, *Pa*	**Analytical solution** τ, *Pa*	**Error** δ = 100%*| τ^* - τ | / | τ |
4.0591E+007	3.8462E+007	5.5



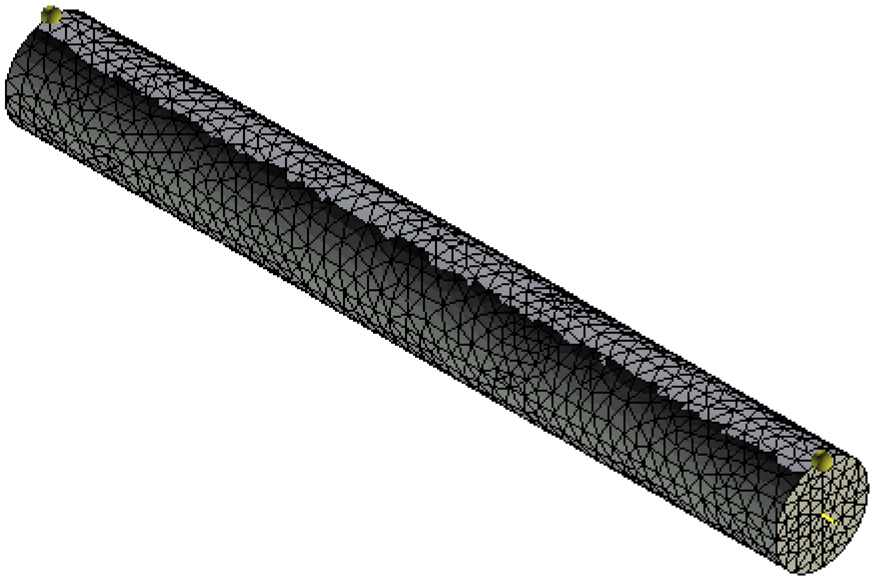
Conclusions:
The relative error of the numerical solution compared to the analytical solution is equal to 0.92% for displacements and 5.5% for stresses when using quadratic finite elements.

Torsion of a Shaft by Two Torques

Let us consider a shaft of length L, diameter d, loaded with two torques T, oriented perpendicularly with respect to each other and applied at the opposite edges of the shaft.



The sought value – maximum value of angle of twist.
Let us take T = 100 N*m, L = 0.5 m, d = 0.06 m.
Material's characteristics (steel): G = 8.203E+010 Pa, n = 0.28.
Both ends of the shaft are not constrained and are subjected to the action of the moments T, the axis of rotation of which coincides with the axis of the cylinder, but the directions are opposite.
Calculations are carried out with the enabled «Stabilize the system» option with additional stiffness equal to 1.



Finite element model for indicated loads and constraints.

Analytical solution is given by:

φ = (T*L)/(G*Jp) = 4.791E-004 rad

w = d*sin(φ/2) = 1.4372E-005 m

where φ – angle of twist, w – displacement of a point, T – torque, L – length of the shaft, G – shear modulus for the given material,

Jp= pd4 / 32 –polar moment of inertia for the circular cross-section.

After calculations are carried out with the help of T-FLEX, we obtain the following results:

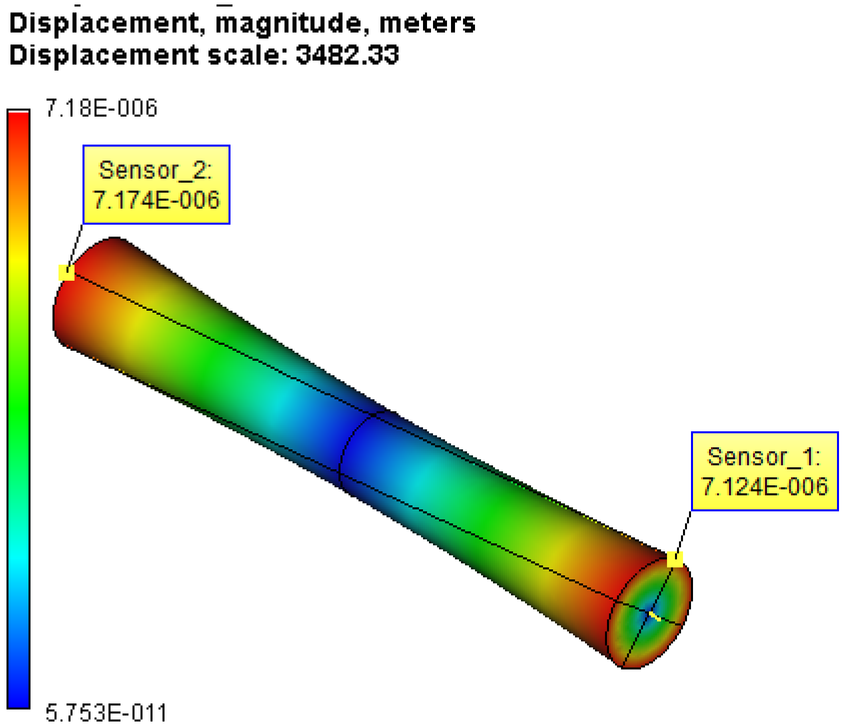
displacement is equal to 7.124E-006+7.174E-006 = 1.429E-005 m

Table 1. Parameters of finite element mesh

Finite element type	**Number of vertices**	**Number of finite elements**
quadratic tetrahedrons	69363	15348

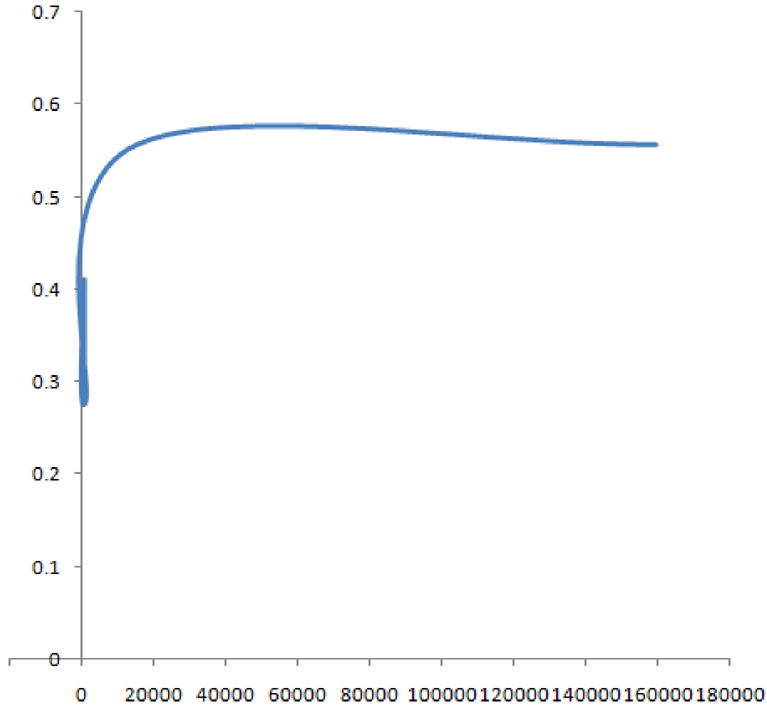
Table 2. Result "Displacement"

Numerical solution w^*, *m*	**Analytical solution** w, *m*	**Error** $\delta = 100\% \cdot \lvert w^* - w \rvert / \lvert w \rvert$
1.4298E-005	1.4372E-005	0.513



Conclusions:

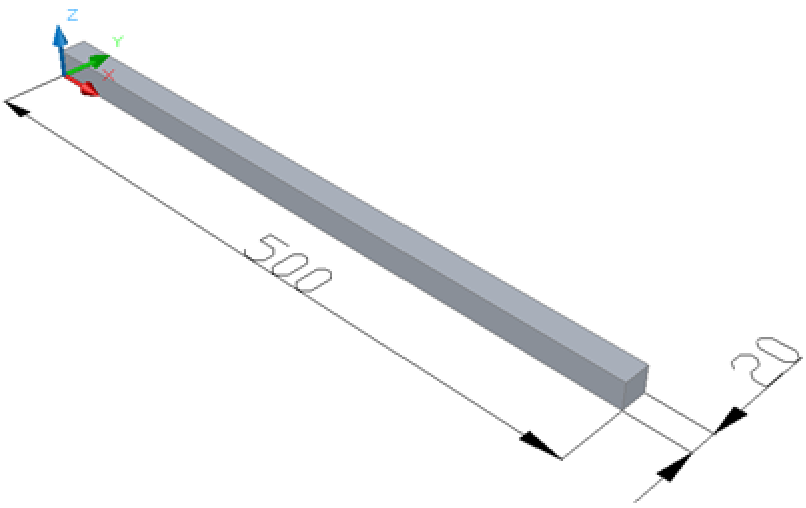
The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.5% for quadratic finite elements. The plot of dependence of the relative error on the number of finite elements shows that influence of stabilization changes the shape of the curve.



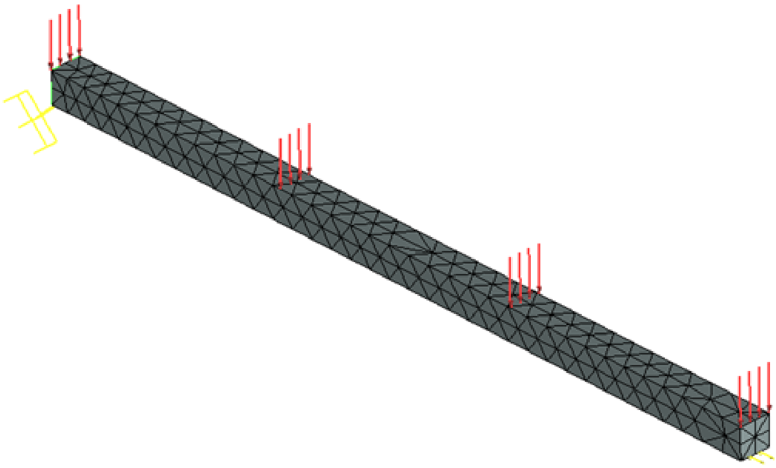
Dependence of relative error on the number of finite elements

Deflection of a Simply supported Beam under uniformly distributed Load

Let us consider a beam under uniformly distributed load q. Length of the beam is equal to L. The cross-section of the beam is a square. The length of the side of the square is a.



The sought value is the maximum displacement along the Z-coordinate.
Let us take the following input data: q = 3000 Pa, L = 0.5 m, a = 0.02 m.
Material's characteristics (steel): Young's modulus E = 2.1E+011 Pa, Poisson's ratio n = 0.28.



Finite element model for indicated loads and constraints

Analytical solution is calculated from the formula:

$$w = \frac{q \cdot a}{24EJ}\left(-x^4 + 2Lx^3 - L^3x\right),\ x \in [0, L].$$

Maximum deflection of the beam takes place at x = L / 2:

$$w_{\frac{L}{2}} = -\frac{5}{384} \cdot \frac{q \cdot a \cdot L^4}{E \cdot J},$$

where J = a4 / 12 –moment of inertia.
Therefore, | w | = 1.7439E-005 m.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

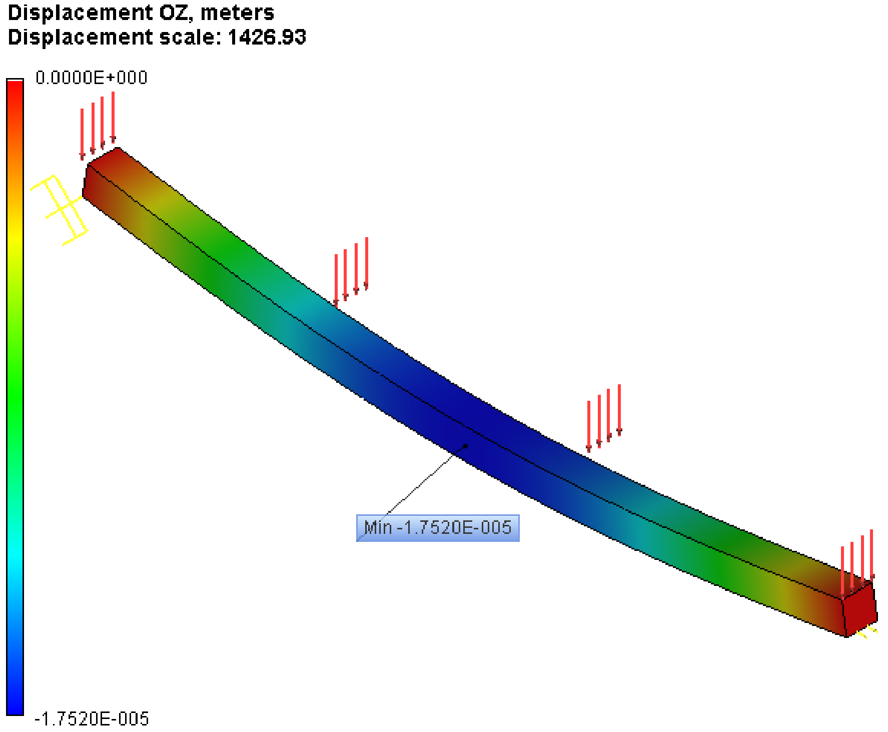
Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of arguments	Number of finite elements
quadratic tetrahedrons	695	12003	2169

Table 2. Result "Displacement"

Numerical solution | *w** |, *m*	Analytical solution | *w* |, *m*	Error δ =100%* |*w** - *w* | / | *w* |

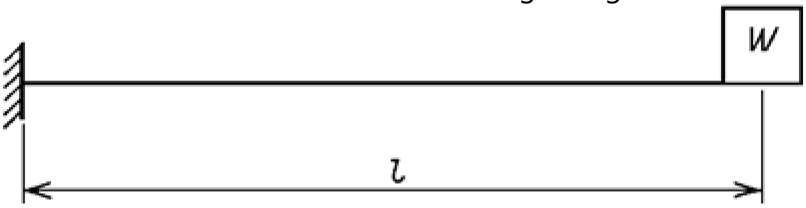
1.752E-005	1.7439E-005	0.4



** Results of numerical solution depend on configuration of the mesh and can be somewhat different from those given in the table.*

Deflection of a Beam with the Load

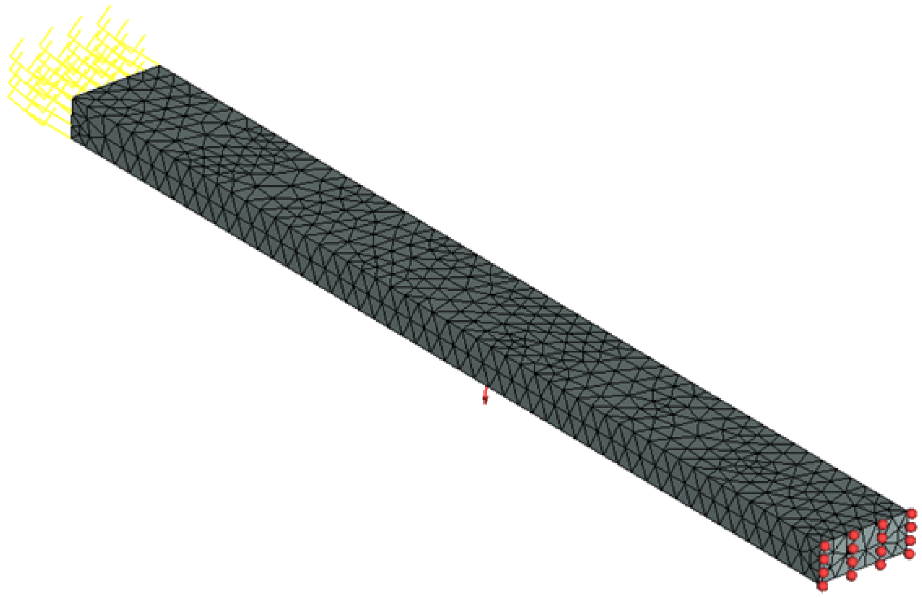
Let us consider a cantilever beam the right edge of which is loaded with the weight.



The length of the beam is L. The cross-section of the beam is a rectangle of width b and height h. Mass of the weight is M. Mass of the beam is m.

m = r F,

where F = b h, - dimensions of the cross-section, r - density of the material of the beam.



Finite element model for indicated loads and constraints.

Let L be 0.5 m, b is equal to 0.02 m, h is equal to 0.05 m.
Material's properties: E = 2.1E+011 Pa, n=0.28, r = 7800 kg / m3.
Mass of the weight M is equal to 20mL kg (i.e., 78 kg).
Analytical solution can be obtained from formula:

$$|z|_{max} = \frac{gL^3}{3EJ}\left(M + \frac{33}{140}mL\right), \quad J = \frac{hb^3}{12}$$

.

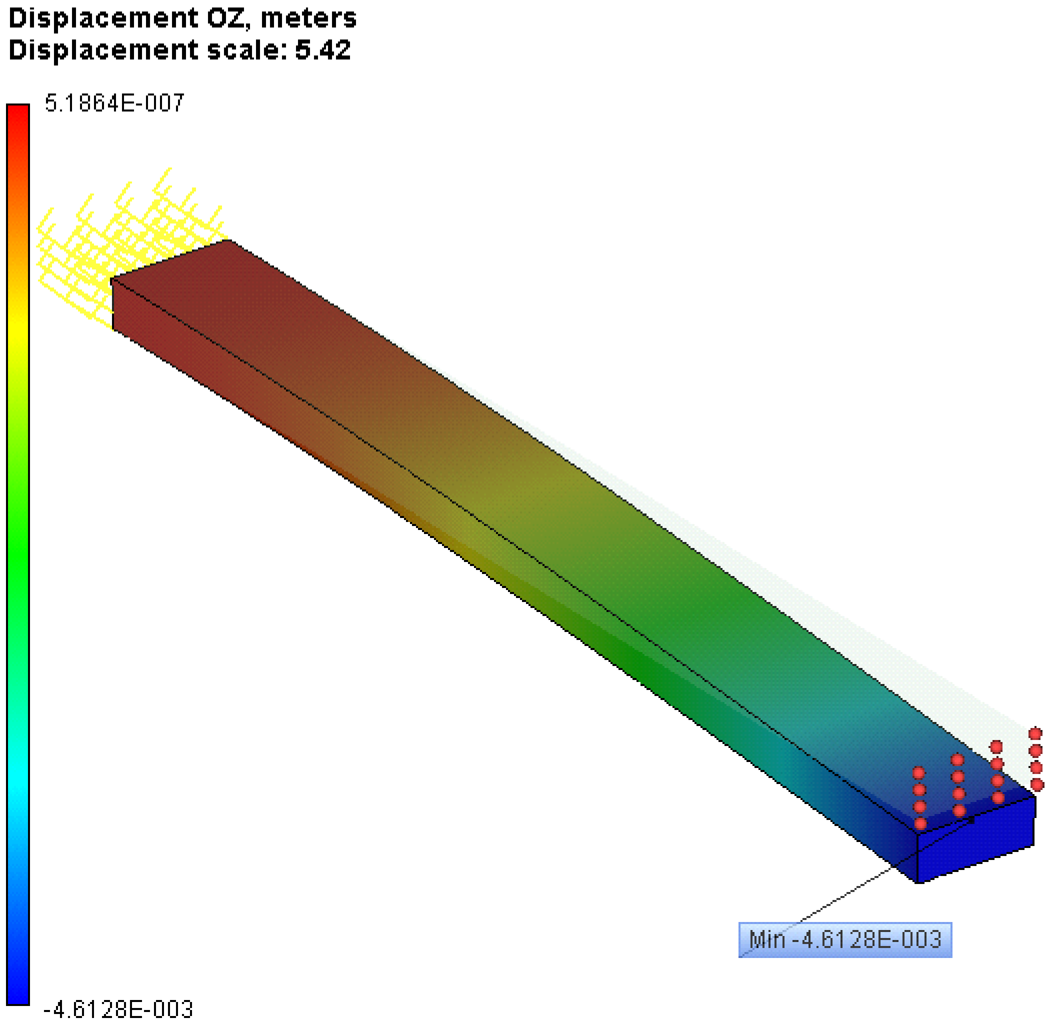
Therefore, |z|max= 4.6067E-003 m.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of arguments	Number of finite elements
quadratic tetrahedrons	2060	40992	8766

Table 2. Result "Displacement"

Numerical solution $\|z\|^*$, m	Analytical solution $\|z\|$, *m*	Error $\delta = 100\%*\| \|z\|^*-\|z\|\| / \|z\|$
4.6128E-003	4.6067E-003	0.13

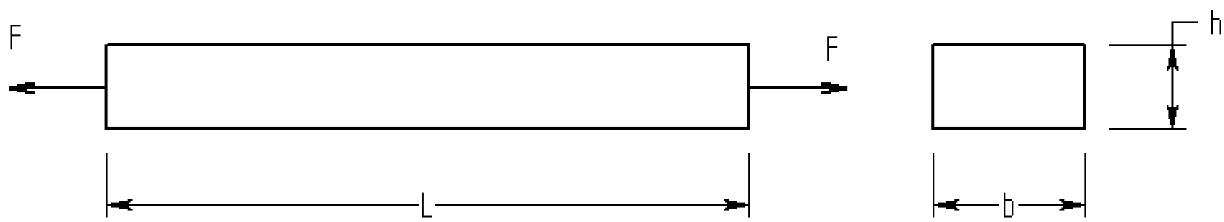


Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.13% for quadratic finite elements.

Stretching of a Beam under the Action of Two Forces

Let us consider a beam of length L, loaded with two forces F, directed perpendicular to both ends of the beam. The cross-section of the beam is a rectangle of width b and height h.



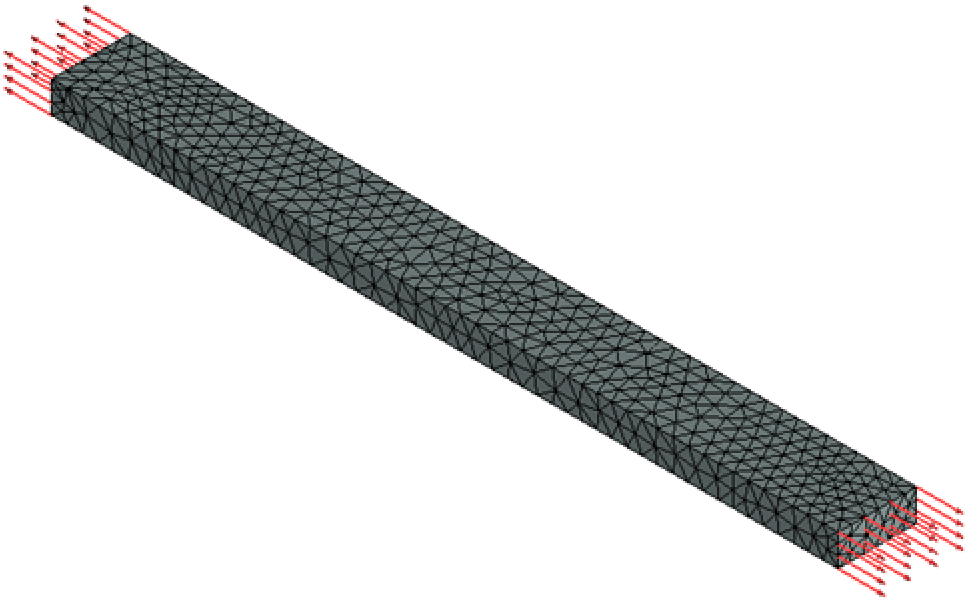
The sought value – maximum stretch.

Let us take F = 1000 N, L = 0.5 m, b = 0.05 m, h = 0.02 m.

Material's characteristics (steel): E = 2.1E+011 Pa, n = 0.28.

Both ends of the beam are not fixed and subjected to the action of the forces F, directed perpendicular to the faces.

Calculations are carried out with the enabled «Stabilize the system» option with additional stiffness equal to 1.



Finite element model for indicated loads and constraints.

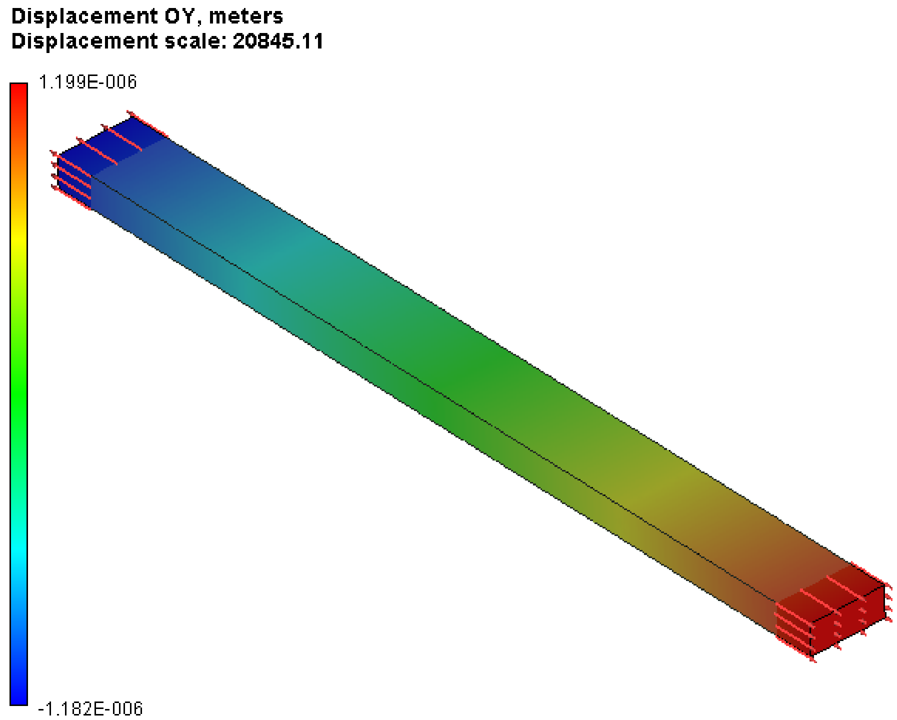
Analytical solution is given by:
w = (F . L) / (A . E) = 2.381E-006 m
where P – force, L – length of a beam, E –Young's modulus for the material, A = b . h - area.
After calculations are carried out with the help of T-FLEX, we obtain the following results:
(displacement is equal to (1.199E-006)+(1.182E-006)=2.381E-006 m)

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of arguments	Number of finite elements
quadratic tetrahedrons	2030	40821	8712

Table 2. Result "Displacement"

Numerical solution $0Y^*$, м	Analytical solution 0Y, m	Error δ =100%* $\lvert 0Y^* - 0Y\rvert$ / $\lvert 0Y\rvert$
2.381E-004	2.381E-004	0.1

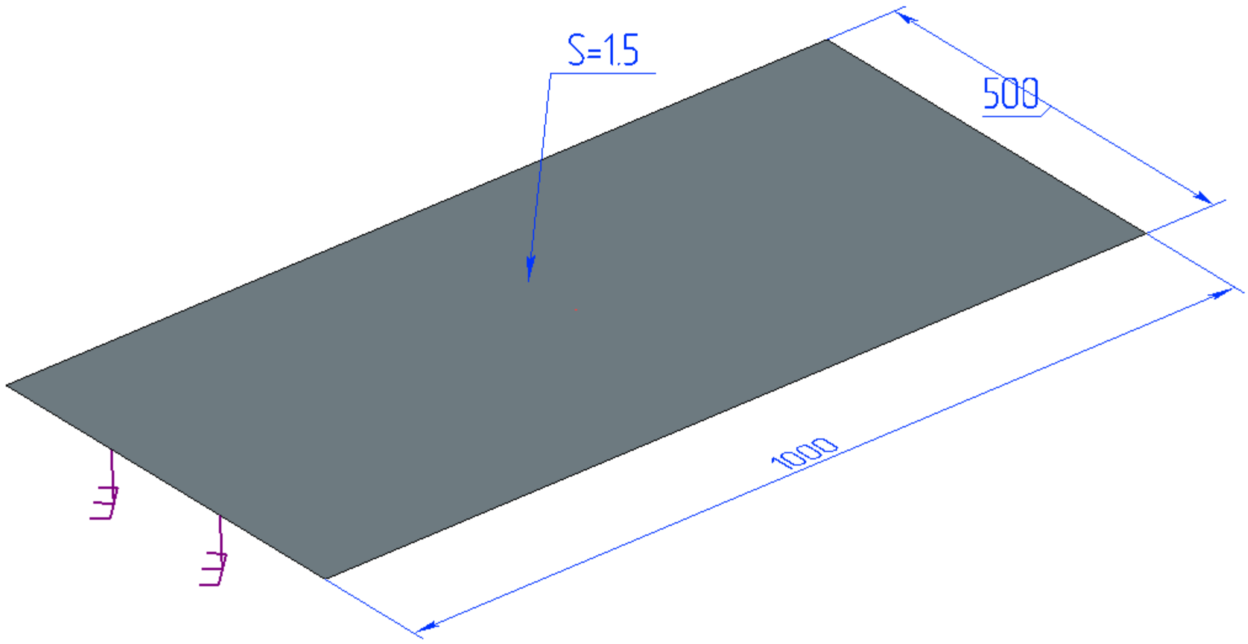


Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.1% for quadratic finite elements.

Deflection of Thin Plate Subjected to Self-Weight (Plate FE)

One edge of the thin plate with dimensions b x h x L = 500 x 1.5 x 1000 mm is fixed and the plate is deflected under self-weight. The material of the plate is a carbon steel with the modulus of elasticity, Poisson ratio and density equal correspondingly: E=2.1E+11 Pa, n=0.28, r = 7800 kg / m3.



Calculated model with boundary conditions

Let us calculate deflection of the plate under self-weight.

$$z = \frac{3\rho \cdot g \cdot L^4}{2E \cdot h^2}$$

Where g - acceleration of gravity, m/s 2.

Calculation using formula gives us the value z = 242.8313 mm.

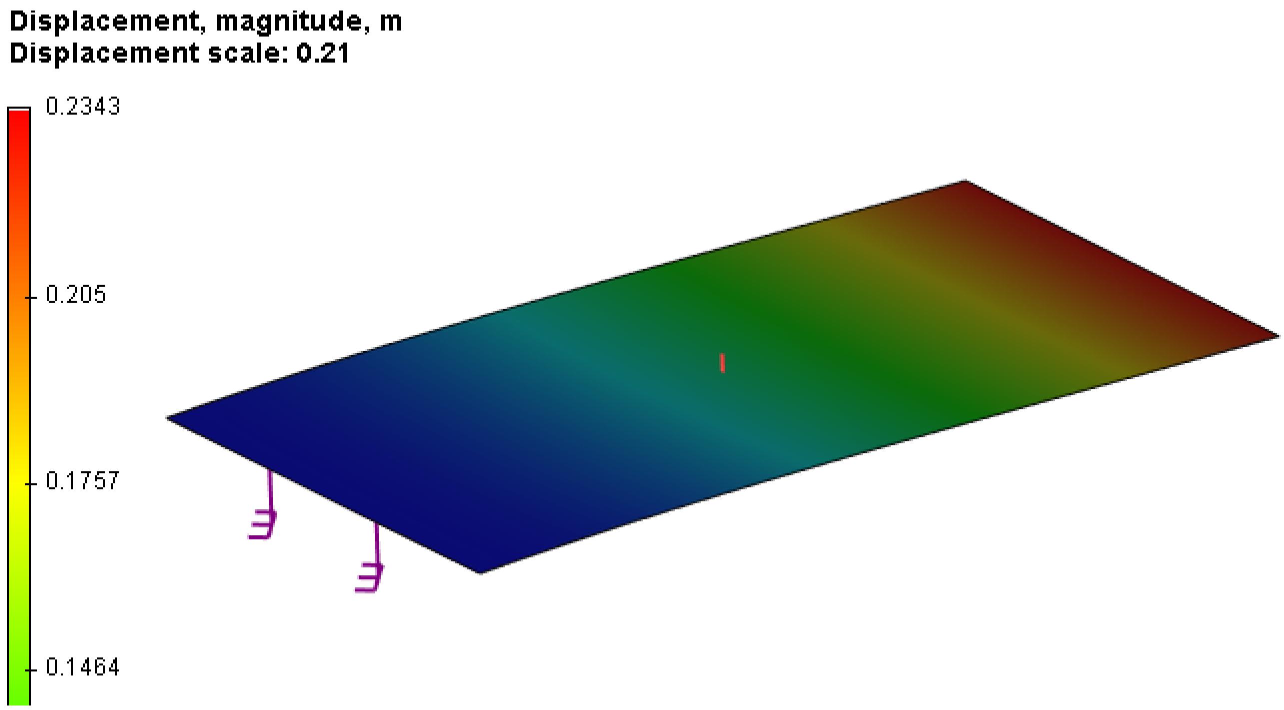
The following results were received after calculation using direct method in T-FLEX analysis:

Table 1

Type of finite element	Number of nodes	Number of finite elements	Number of arguments
Linear triangle (6 nodes)	7485	3338	44592

Table 2

Displacement along Z	Numerical solution,	Analytical solution	Error
z, mm	242.8313	234.3129	-3.5%



Results of finite-element calculation

Deflection of Thin Plate with Single Fixed Edge and One, Two or Three Simply-Supported Edges

Dimensions of the plate are a Ч b Ч s = 1000 Ч 500 Ч 1.5 mm. Pressure q=2000 Pa is applied to the upper face. Let us consider different cases of the plate edges restraints and calculate their displacements. Material: steel with E=2.1E+11 Pa, n=0.28, r = 7800 kg/ m3.

1) One edge is fixed, opposite edge is simply-supported, side edges are free. Displacement is defined using the formula:

Where

2) One edge is fixed, opposite edge is free, side edges are simply-supported. Displacement is defined using the formula:

where D - cylindrical stiffness:

3) One edge is fixed, three side edges are simply-supported. Displacement is defined using the formula:

Calculations gives the following values: D=64,087 Nm; =184.550 mm; = 29.257 mm; = 18.139 mm.

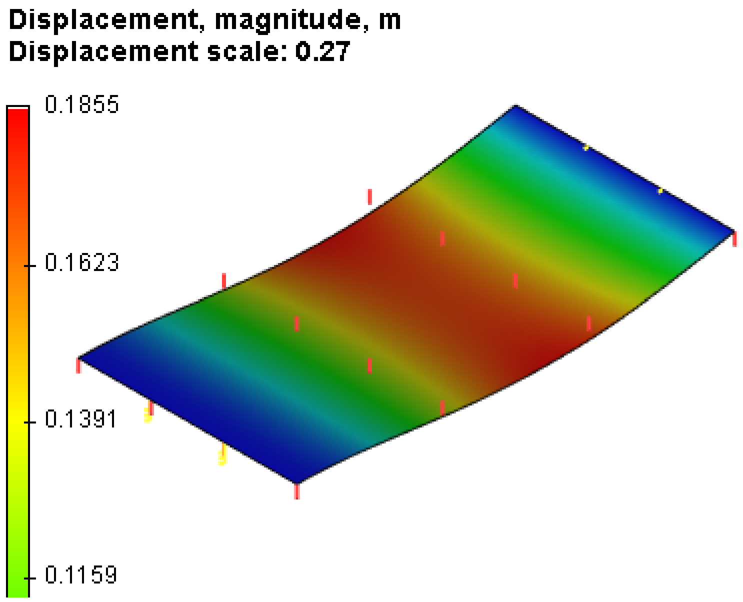
The following results were received after calculation using a direct method in T-FLEX analysis:

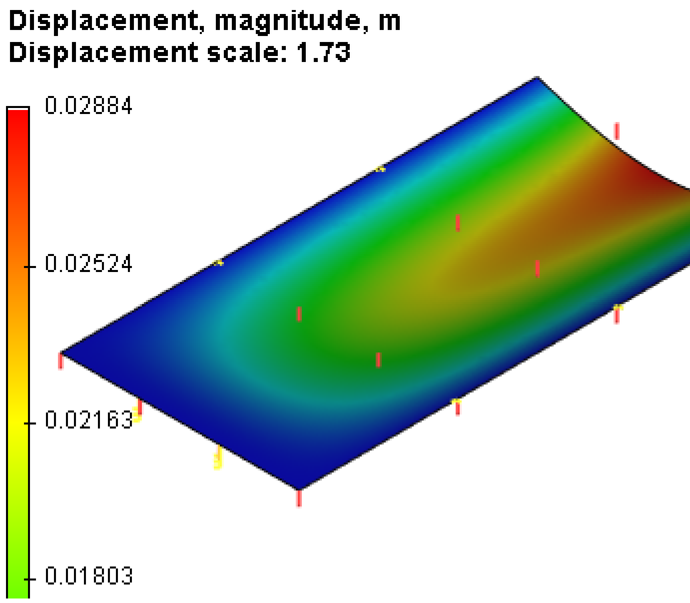
Table 1

Type of finite element	Number of nodes	Number of finite elements	Number of arguments
Linear triangle (6 nodes)	11931	5858	71148

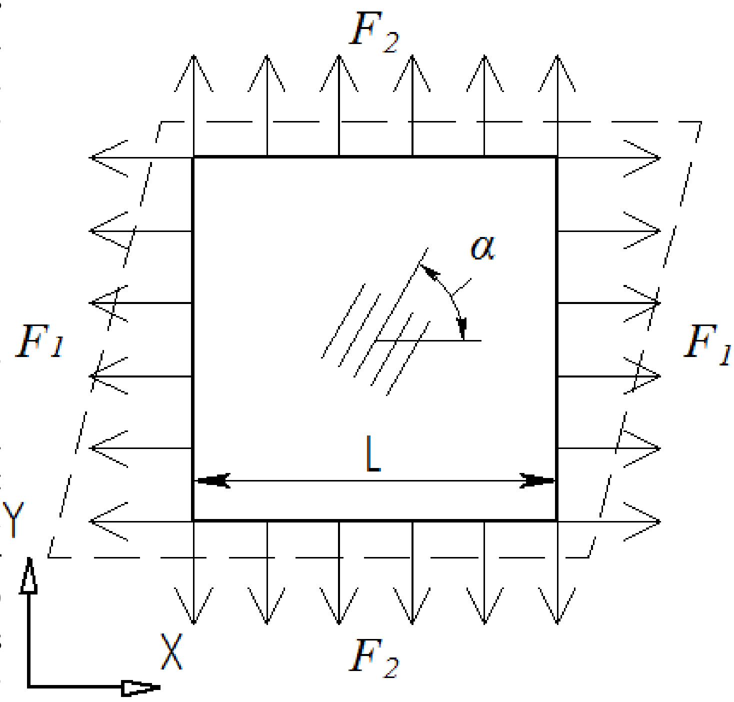
Table 2

Displacement along Z	Numerical solution,	Analytical solution	Error
w_1, mm	184.462	184.550	-0.05%
w_2, mm	28.723	29.257	-1.83%
w_3, mm	18.251	18.139	0.62%





Stresses and Deformations of an Orthotropic Plate in Biaxial Tension



Consider a square plate, whose side length is equal to L, made of an orthotropic material. The plate is loaded with the forces F1, F2 applied at the edges. The thickness of the plate is h.

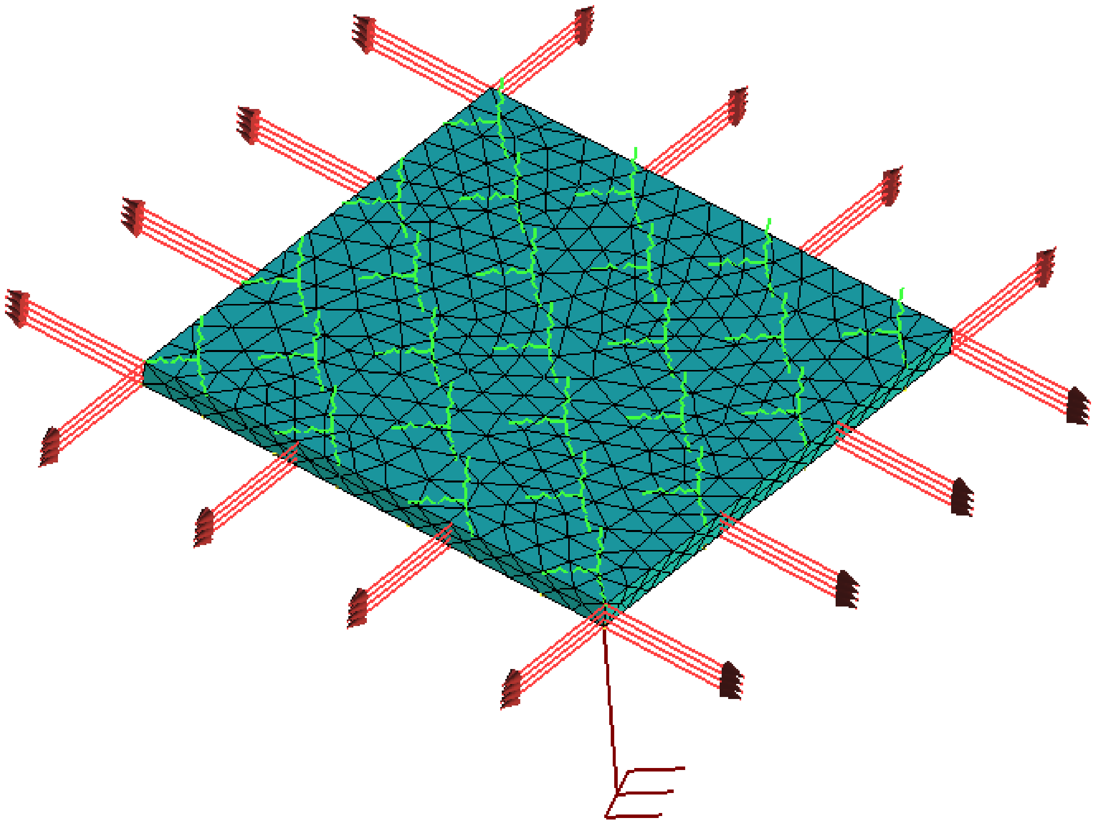
It is required to determine stresses and deformations of the plate.

Let us take F1=20000 N, F2=10000 N, L=0.1 m, h=0.005 m.

Having built up the model, we create the study of the «Static analysis» type and discretize the model into finite elements. Material's parameters are the following: Young's modulus E1=5.59·1010 Pa, E2=1.373·1010 Pa, E3=1.373·1010 Pa, shear modulus G12=5.59·109 Pa, G23=4.904·109 Pa, G31=5.59·109 Pa, Poisson's ratios ν12=0.277, ν23=0.4, ν31=0.068. The principle axis of symmetry makes an angle α=45° with the horizontal axis.

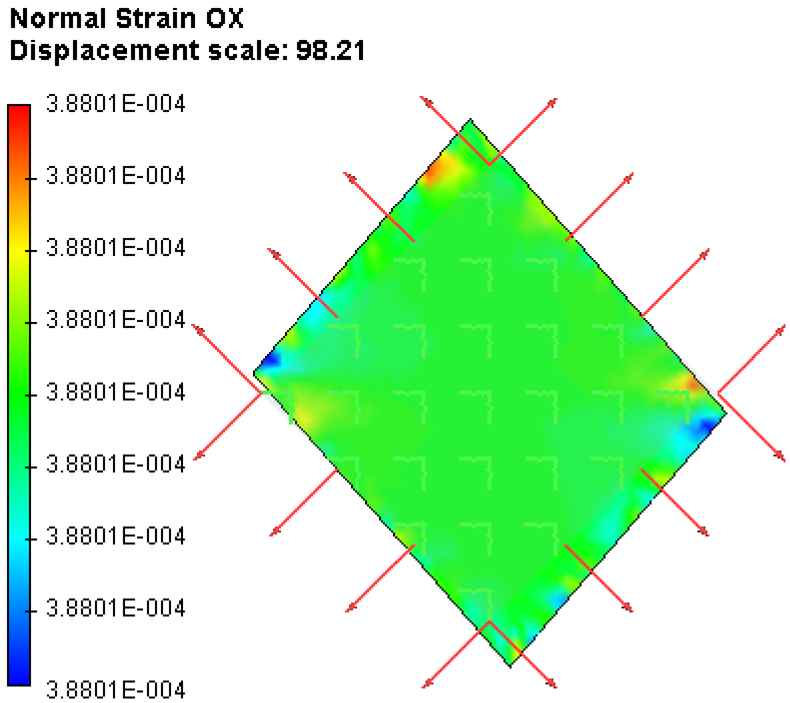
Let us place the plate in such a way that the principle axes of orthotropic symmetry coincide with the axes of the global coordinate system.

Apply full constraint at one of the vertices of the plate, and on the lower face of the plate – partial constraint with the zero displacement along the Z-axis. To stabilize the model, let us apply the elastic foundation on the upper face of the plate with additional stiffness equal to 1 H/m. We apply the normal loading of magnitude F1 on one pair of parallel lateral edges of the plate, and the normal loading of magnitude F2 on another pair.

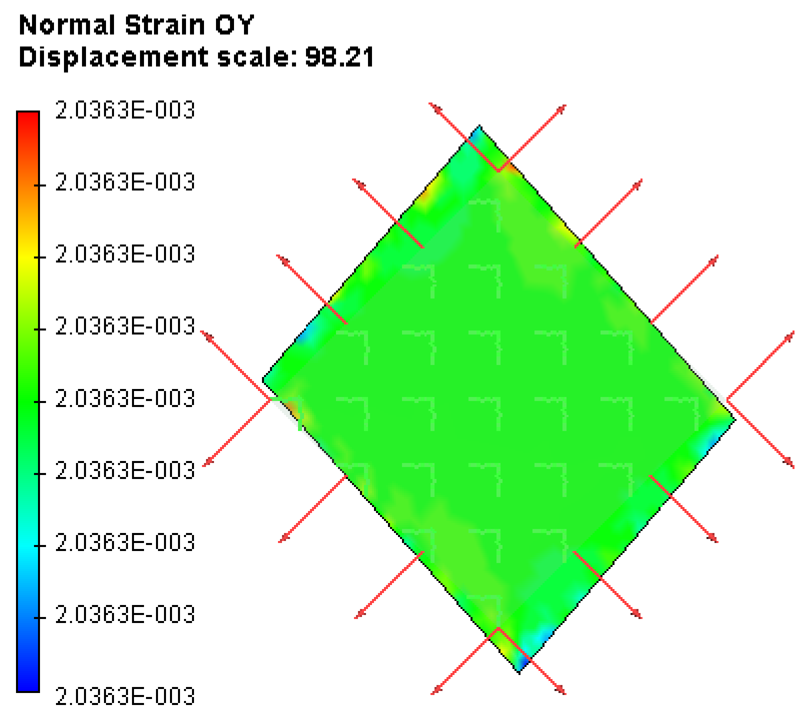


Finite element model of an orthotropic plate with the loads and constraints

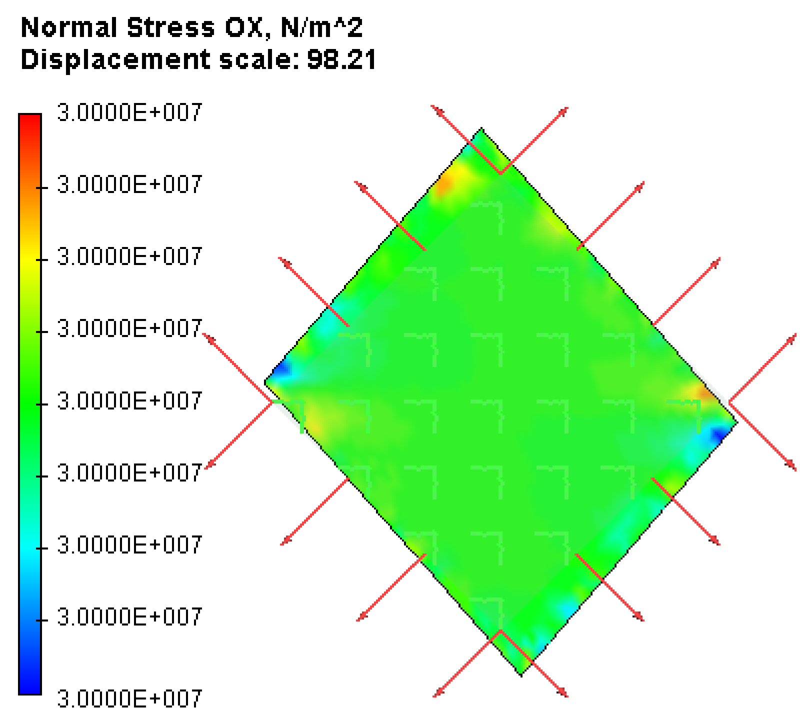
With the help of the «**Analysis**» command, let us carry out the static analysis of the plate. We obtain the results in the form of deformations and stresses.



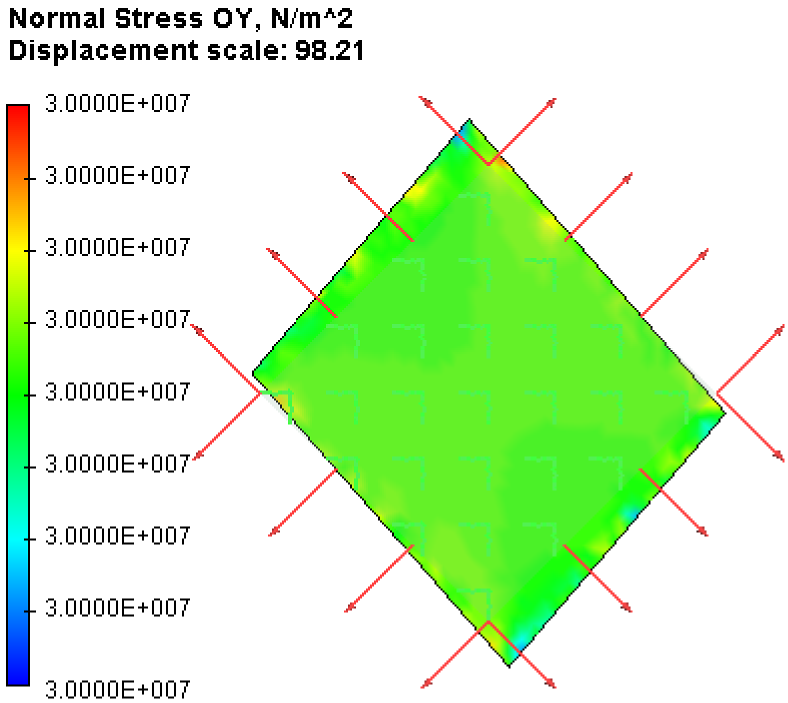
Deformations OX of an orthotropic plate



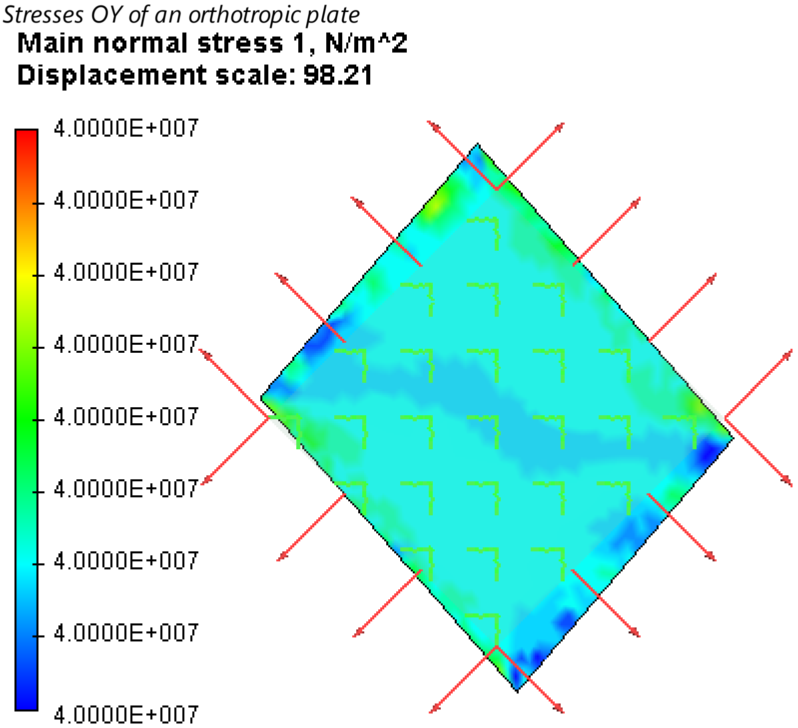
Deformations OY of an orthotropic plate



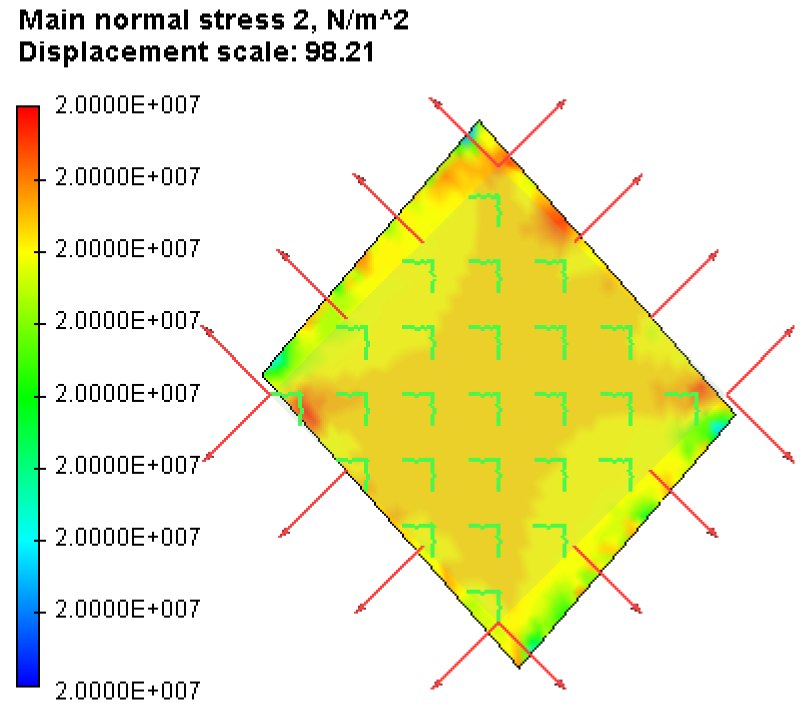
Stresses OX of an orthotropic plate



Stresses OY of an orthotropic plate



Principle stresses 1 of an orthotropic plate



Principle stresses 2 of an orthotropic plate

The average value of the deformation OX εx*=3.8801·10-4, deformation OY εy*=2.0363·10-3,
The average value of the stress OX σx*=3.000·107 Pa, stress OY σy*=3.000·107 Pa.
The principle stresses are equal to: σ1*=4.000·107 Pa, σ2*=2.000·107 Pa.

Analytical solution for principle stresses reads:

$$\sigma_1 = \frac{F_1}{L \cdot h}, \quad \sigma_2 = \frac{F_2}{L \cdot h}$$

where F1,2 – force N, L – length of the plate, h – thickness of the plate, m.
σ1=20000/0.005/0.01=4·107 Pa, σ2=10000/0.005/0.01=2·107 Pa.
Stresses along the axes OX, OY are calculated from the formulas:

$$\sigma_x = \sigma_y = \sigma_\alpha = \sigma_1 \cos^2\alpha + \sigma_2 \sin^2\alpha, \quad \alpha = 45^\circ$$

σx= σy=4·107 · 0.5+2·107 · 0.5=3.0·107 Pa
Deformations along the axes OX, OY are calculated from the formulas:

$$\varepsilon_x = \frac{\sigma_x}{E_1} - \frac{\nu_{21}\sigma_y}{E_2}, \quad \varepsilon_y = \frac{\sigma_y}{E_2} - \frac{\nu_{12}\sigma_x}{E_1}, \quad \nu_{21} = \nu_{12}\frac{E_2}{E_1}$$

ν21=0.277·1.373·1010/5.59·1010=0.06804
εx=3.0·107/5.59·1010 – 0.06804·3.0·107/1.373·1010=3.88·10-4;
εy=3.0·107/1.373·1010 – 0.277·3.0·107/5.59·1010=2.036·10-3;
After calculations are carried out with the help of T-FLEX by a direct method, we obtain the following results:

Table 1

Finite element type	Number of vertices	Number of finite elements
Quadratic tetrahedron	1104	3338

Table 2

Principle stress	Numerical solution, σ^*	Analytical solution σ	Error $\delta_\sigma = \frac{\lvert\sigma - \sigma^*\rvert}{\sigma} \times 100\%$

σ_1 , Pa	4.0×10^7	4.0×10^7	0.00%
σ_2 , Pa	2.0×10^7	2.0×10^7	0.00%

Table 3

Stress OX, OY	**Numerical solution, σ***	**Analytical solution σ**	**Error** $\delta_\sigma = \frac{\lvert\sigma - \sigma^*\rvert}{\sigma}\times100\%$
σ OX, Pa	3.0×10^7	3.0×10^7	0.00%
σ OY, Pa	3.0×10^7	3.0×10^7	0.00%

Table 4

Deformations	**Numerical solution,** ε^*	**Analytical solution** ε	**Error** $\delta_\sigma = \frac{\lvert\sigma - \sigma^*\rvert}{\sigma}\times100\%$
ε OX	3.8801×10^{-4}	3.88×10^{-4}	0.003%
ε OY	2.0363×10^{-3}	2.036×10^{-3}	0.015%

Conclusions:

We obtained a realistic picture of stress distribution. The relative error of the numerical solution compared to the analytical solution for strains is equal to 0.015% for quadratic finite elements.

Thermal Stresses in Bimetallic Element

Bimetallic elements are used in different thermal sensors. Bimetallic element consists of two rigidly connected plates with different coefficients of thermal expansion α_1, α_2. Upon heating, the bimetallic plate is bended due to different elongation of its components (see figure). If one edge of the plate is clamped rigidly, the second edge (free) will have a translation of some magnitude.

Let us compute how the curvature of the surface of seam of the bimetallic element changes depending on the temperature of heating.

Let the bimetallic element consist of plates having thicknesses $h_1 = 1.5mm$, $h_2 = 0.9mm$ and be made from different metals (aluminum $\alpha_1 = 2.4\times10^{-5}\ K^{-1}$ and steel $\alpha_2 = 1.3\times10^{-5}\ K^{-1}$, $E_1 = 6.9\times10^4\ \frac{N}{mm^2}$, $E_2 = 2.1\times10^5\ \frac{N}{mm^2}$, $\nu_1 = 0.33$, $\nu_2 = 0.28$). The plates have equal length $a = 50mm$ and width $b = 10mm$. The element is heated from $T_0 = 298.15\ K$ (or $T_0 = 25\ ^0C$) to $T = 323.15\ K$ (or $T = 50\ ^0C$).

Let $\frac{1}{\rho_0}$ – be the initial curvature of the seam's surface equal to 0; $\frac{1}{\rho}$ – the curvature of the seam's surface after heating. The change of the curvature can be found with the help of the following formula:

$$\frac{1}{\rho}-\frac{1}{\rho_0}=\frac{6(T-T_0)(\alpha_1-\alpha_2)}{\left(\frac{(E_1h_1^2-E_2h_2^2)^2}{E_1E_2h_1h_2(h_1+h_2)}+4(h_1+h_2)\right)}.$$

The curvature of the seam's surface after heating of the given bimetallic element $\frac{1}{\rho}=1.7179\times10^{-4}\ mm^{-1}$.

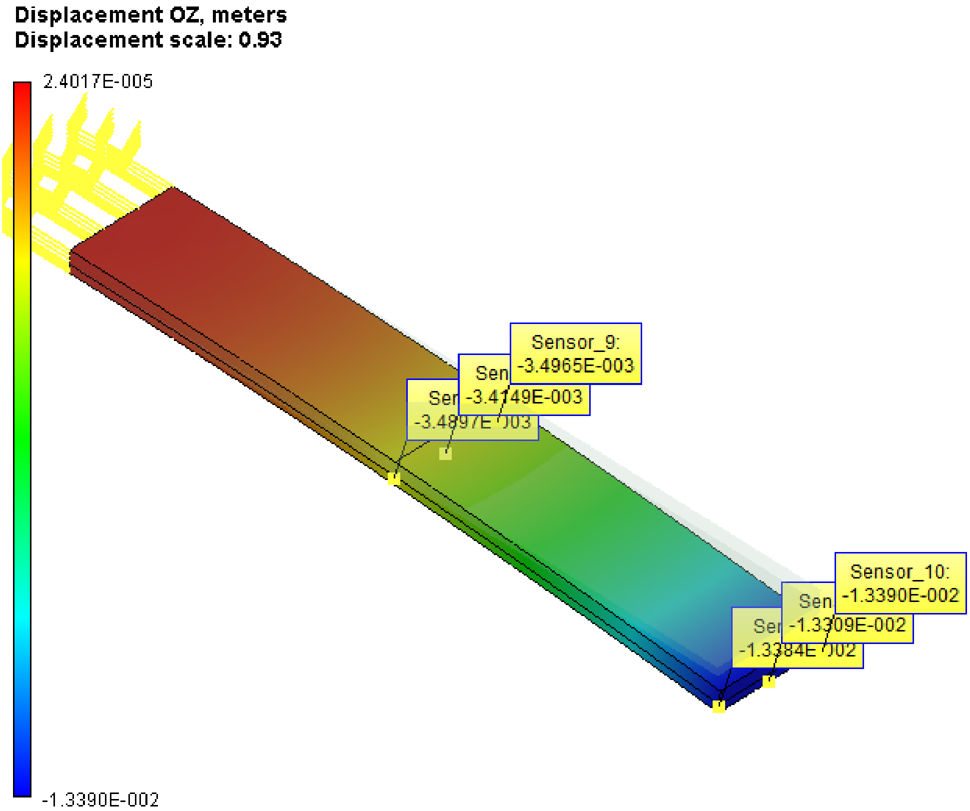
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

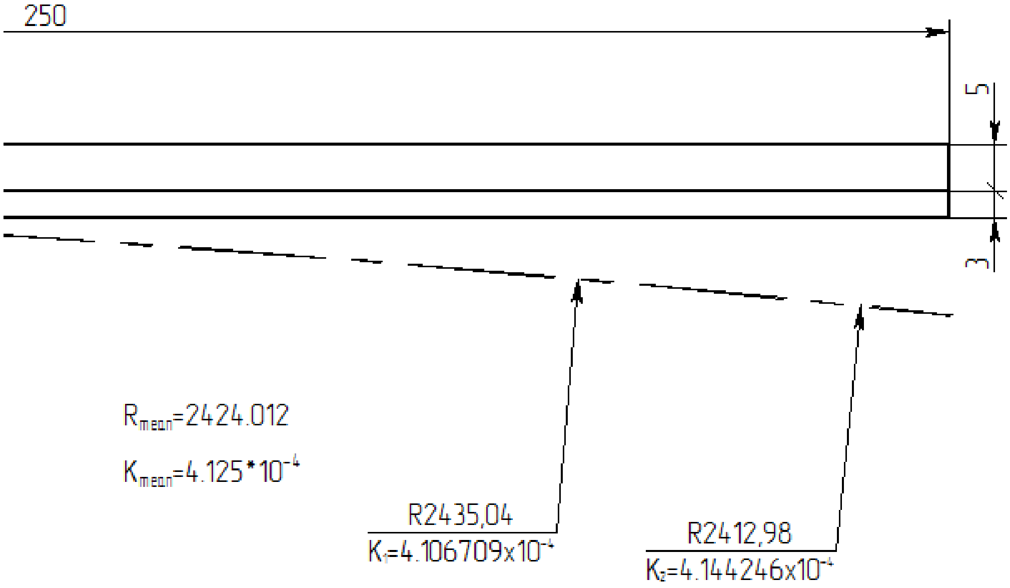
Table 1.
Parameters of finite element mesh

Finite Element Type	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
quadratic tetrahedron (10 nodes)	7932	165231	35667

Table 2.
Result «Curvature of seam's surface»

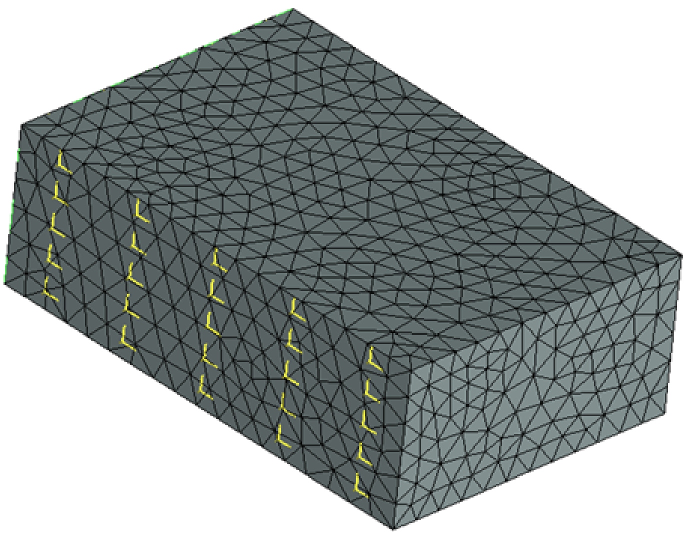
Surface S_{ij} of separation of plates i and j	**Numerical solution** **Curvature** $\frac{1}{\rho_n}$, mm^{-1}	**Analytical solution** **Curvature** $\frac{1}{\rho_a}$, mm^{-1}	**Error** $\delta=\frac{\left\lvert\frac{1}{\rho}-\frac{1}{\rho_n}\right\rvert}{\left(\frac{1}{\rho}\right)}\times100\%$
S_{12}	*125E-004*	*123E-004*	0,049





Thermal Deformation of a 3-D Brick

Let us consider a brick. The length of the brick is L, width is b, height is h.
The sought values – absolute values of deformations along the X, Y, and Z axes caused by the change in temperature.
Let us consider the following input data: L = 0.3 m, b = 0.2 m, h = 0.1 m.
Material's characteristics (steel): E = 2.1E+011 Pa, n = 0.28, coefficient of thermal linear expansion α = 1.3E-005 K -1.
The change of temperature DT is equal to 100o.



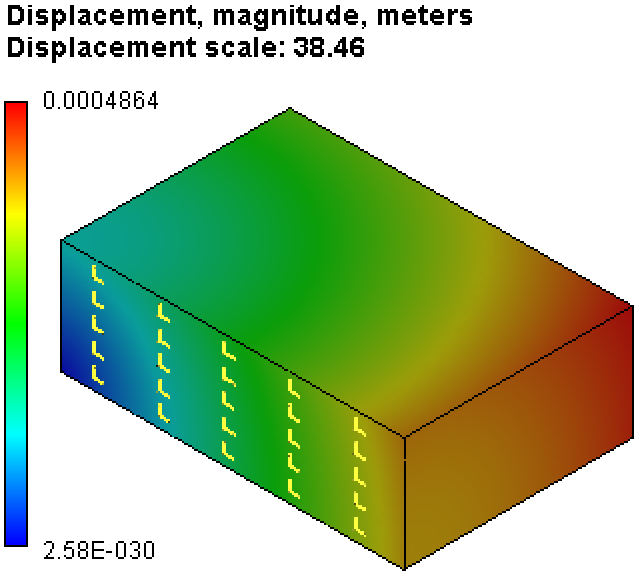
Analytical solution is calculated from the formulas:
Dx = α L DT
Dy = α b DT
Dz = α h DT
Therefore,
Dx = 2.60000000E-004 m
Dy = 3.90000000E-004 m
Dz = 1.30000000E-004 m .
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1.
Parameters of finite element mesh

Finite element type	Number of main nodes	Number of arguments	Number of finite elements
quadratic tetrahedron (10 nodes)	1730	35775	7565

Table 2.
Result "Displacement"

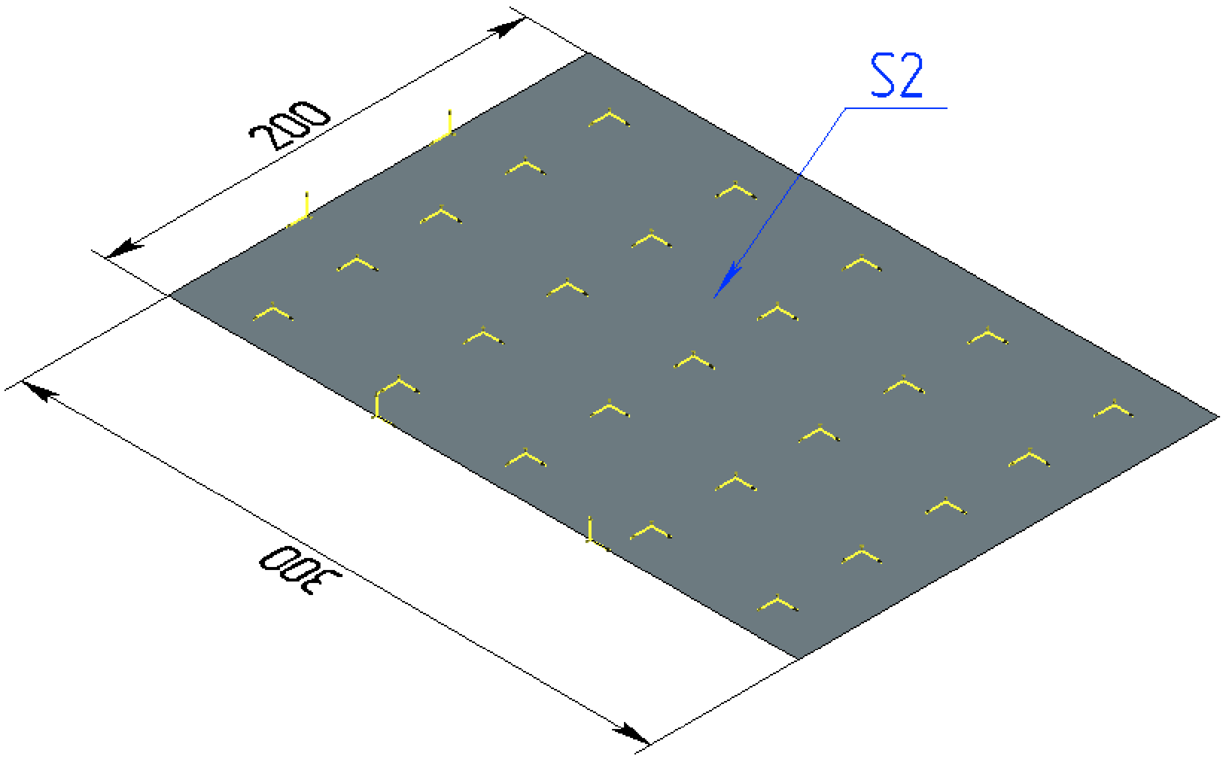
Numerical solution Δ^*, *m*	Analytical solution Δ, *m*	Error $\delta = 100\%*\|\Delta^* - \Delta\|/\|\Delta\|$
3.90000001E-004	3.90000000E-004	0.26E-007
2.60000007E-004	2.60000000E-004	0.27E-007
1.300000004E-004	1.30000000E-004	0.30E-007



Conclusions:
The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.001% for quadratic finite elements.

Thermal Deformation of Plate (plate FE)

Thin plate with dimensions l x b x s = 300 x 200 x 2 is subjected to uniform heating up to the temperature of 100 °C. Material properties (steel): E = 2.1E+011 Pa, n = 0.28, linear expansion coefficient α = 1.3E-005 K -1. Determine thermal deformations dx, dy of the plate with regard to the initial temperature 0 °C.



Calculated model with boundary conditions

Analytical solution is calculated using formulas:

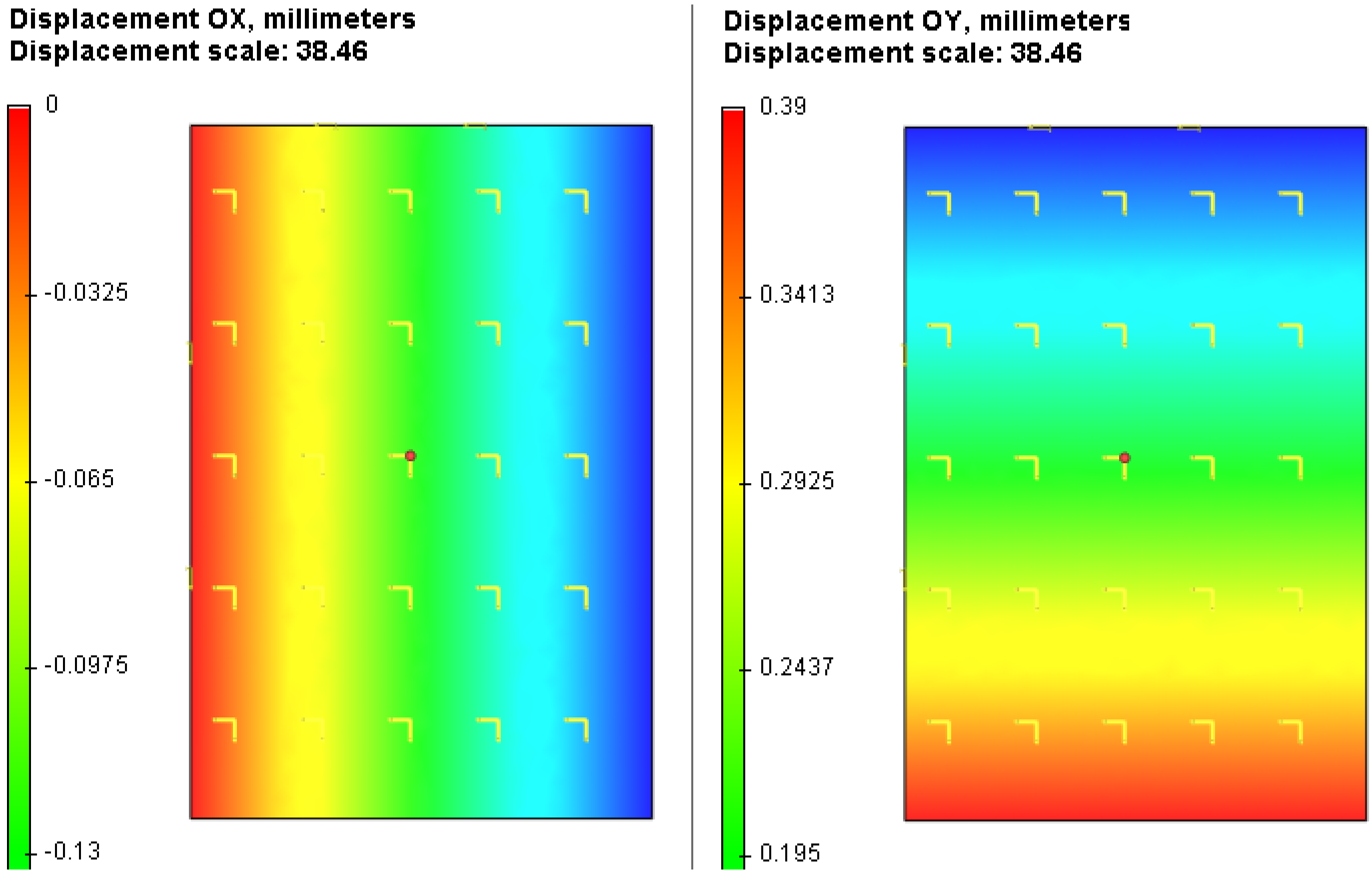
$Dx = \alpha L DT$

$Dy = \alpha b DT$

After the calculations using T-FLEX we receive the following results:

Table 1. Parameters of finite-element mesh			
Type of finite elements	Number of main nodes	Number of arguments	Number of finite elements
Linear triangle (6 nodes)	829	4974	1552

Table 2. "Displacement X, Y" result		
Numerical solution Δ*, *m*	Analytical solution Δ, *m*	Error δ =100%*|Δ* - Δ |/| Δ |
2.60000007E-004	2.60000000E-004	0.27E-007
3.90000001E-004	3.90000000E-004	0.26E-007



Results of calculations in T-FLEX Analysis

Examples of solving Studies with Remote Loads

Framing at a Remote Displacement

Consider a framing consisting of a rod resting upon a hinged bearing and supported by a floating hinge (fig. 1). The cross section of the rod is rectangular (cross section height h is parallel to the figure plane).

E=2E+011Pa, l1=1000 mm, b=50 mm, h=100 mm, d=20 mm, r=300 mm, δ =1000 mm, u_0=5 mm.

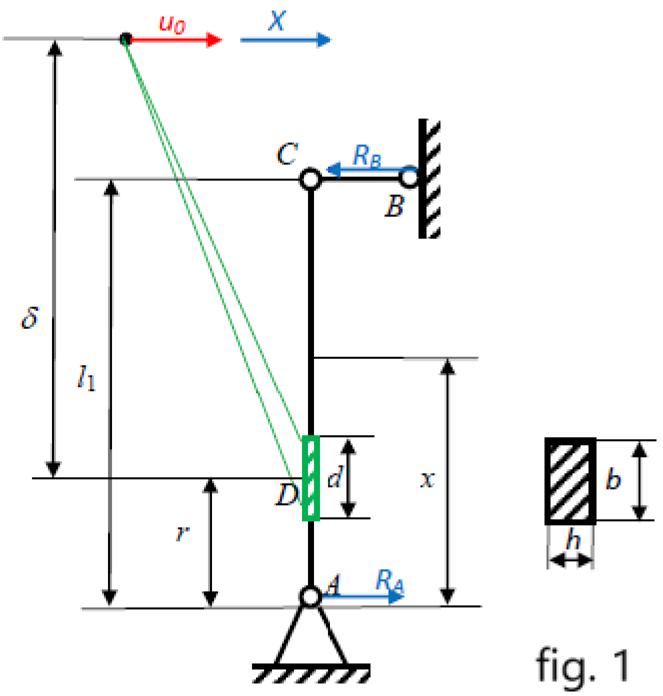


fig. 1

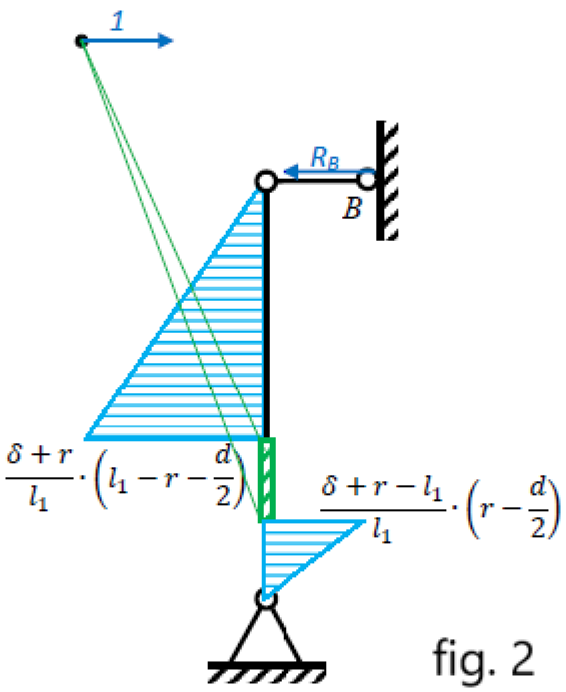


fig. 2

A body with a rigidity much higher than the bending stiffness of rod AC is attached to it (the joint surface is shown as a green rectangle in the picture). Horizontal displacement u_0 is set for a rigid body point. As a result the body moves and turns taking rod AC with it and deforming the framing. Let's find the horizontal displacement of point D and maximum stress in the rod.

u_0 displacement is realized due to force X acting horizontally applied in the same point. The force value is unknown. Force X causes reaction in supports A and B. The framing system is statically determinate and the reactions can be expressed via X:

$$R_A = \frac{\delta + r - l_1}{l_1} X, \quad R_B = \frac{\delta + r}{l_1} X$$

Let's express u_0 displacement via X by means of Mohr integral:

$$u_0 = \int\limits_l \frac{M_X M_1}{EJ} dx$$

where

E is Young modulus; $J = \frac{bh^3}{12}$ is the section inertia moment; M_X is the bending moment due to the force X action; M_1 is the bending moment due to the action of a unit force applied in the point with the given displacement u_0 in the direction of this displacement (fig. 1 and 2); the integral is taken along the lengths of rods AB and BC.

As the application and direction point of force X and the unit force is the same,

$M_X = X M_1$ therefore:

$$u_0 = \frac{X}{EJ} \int\limits_l {M_1}^2 dx$$

Bending moment M_1 at the rigid body and the rod joint surface can be left undetermined as this part of the rod has an infinite rigidity and Mohr integral along this interval is equal to 0 (just as it is along the rigid body volume).

Hence:

$$M_1(x) = \begin{cases} -\dfrac{\delta + r - l_1}{l_1} x, & if \ 0 \le x \le r - \dfrac{d}{2} \\ \dfrac{(\delta + r)(l_1 - x)}{l_1}, & if \ r + \dfrac{d}{2} \le x \le l_1 \end{cases}$$

$$u_0 = \frac{X}{3EJ{l_1}^2} \left[(\delta + r)^2 \left(l_1 - r - \frac{d}{2} \right)^3 + (\delta + r - l_1)^2 \left(r - \frac{d}{2} \right)^3 \right]$$

Now we express force X via the given displacement u_0:

$$X = \frac{3EJ{l_1}^2}{(\delta + r)^2 \left(l_1 - r - \frac{d}{2} \right)^3 + (\delta + r - l_1)^2 \left(r - \frac{d}{2} \right)^3} u_0$$

M_X diagram is X times bigger than M_1 one and maximum normal stresses in the cross sections of rod AC are determined with the following formula:

$$\sigma^{max}(x) = \frac{M_X}{W} = X\frac{M_1(x)}{W}$$

where $W = bh^2/6$ is the section modulus at bending.

Maximum normal stress in the whole framing acts with the maximum M_1 (fig. 2) in the section and is:

$$\sigma_{AC}^{max} = \frac{\delta + r}{l_1} \cdot \left(l_1 - r - \frac{d}{2}\right)\frac{X}{W}$$

Now we determine the horizontal displacement of point D (fig.1) by means of Mohr integral:

$$u_D = \int_l \frac{M_X M_1^D}{EJ} dx = \frac{X}{EJ}\int_l M_1 M_1^D\, dx$$

Here M_1^D is the bending moment of the rod sections appearing due to the unit force action applied in point D in the horizontal direction (in the point and in the direction of displacement) (fig. 3). As a result:

$$u_D = \frac{X}{3EJl_1{}^2}\left[r(\delta + r)\left(l_1 - r - \frac{d}{2}\right)^3 - (l_1 - r)(\delta + r - l_1)\left(r - \frac{d}{2}\right)^3\right]$$

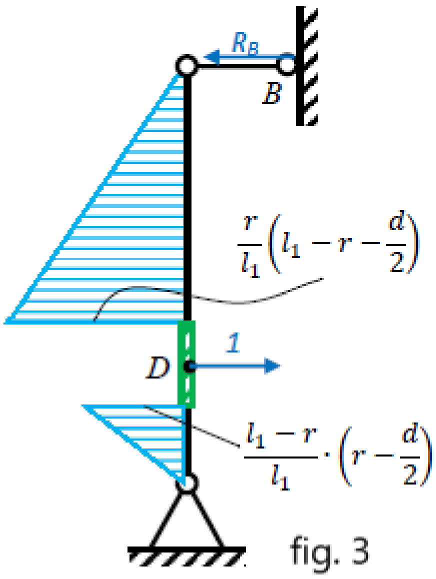


fig. 3

If we substitute the expression for X into this formula, bending rigidity EJ gets cancelled and point D horizontal displacement is independent of the shape and size of the cross section as well as of the rod material.

Jz=3.333E-08 m^4, Wz=3.333E-06 m^3, X=214.812 N, σ_{max}=54.455 MPa,

u_D=1.115 mm.

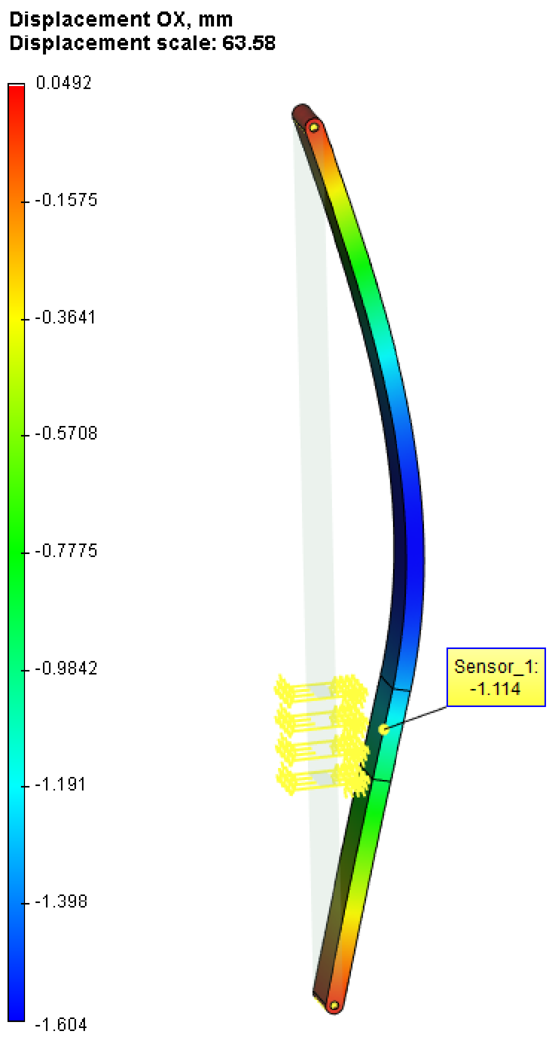


Fig. 4. Calculation result

The results are given in the table.

Table 1.				
Finite elements mesh parameters				
	Finite element type	Number of vertices	Number of finite elements	
	Quadratic tetrahedrons	3157	11438	
Table 2.				
Calculation results, Displacement in the direction of X, mm				
	Numerical solution w^*	Analytical solution w, m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$	
	1,114	1,115	0,09 %	

A Curvilinear Bar under the Action of a Remote Moment

Let’s consider a bar whose axis is a quarter of a circumference and whose cross section is a rectangle (Fig.1). The long side h is parallel to the drawing plane and the short side b is perpendicular to the drawing plane. The left end of the bar is fixed and the right one holds a body whose rigidity is much higher than that of the bar. Moment M is applied to the body in the plane of the picture. b=10 mm, h=20 mm, R=300 mm, E=2E+011 Pa, M=100 Nm.

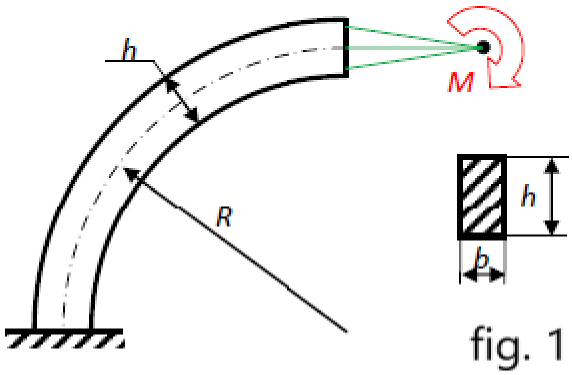


fig. 1

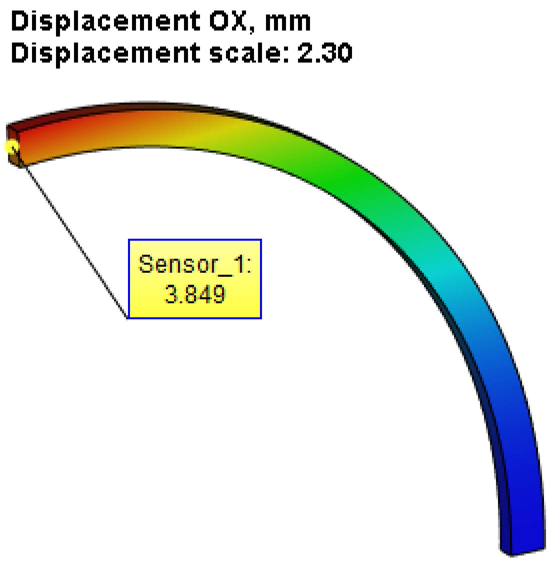
The action of the body onto the bar can be taken into account via moment M applied to the right end of the bar. The formulate determining maximum normal stresses in the cross section, horizontal and vertical displacements of the free end of the bar (to be precise, the displacement of the cross section center of gravity at the free end) are determined according to the strength of materials methods [1].

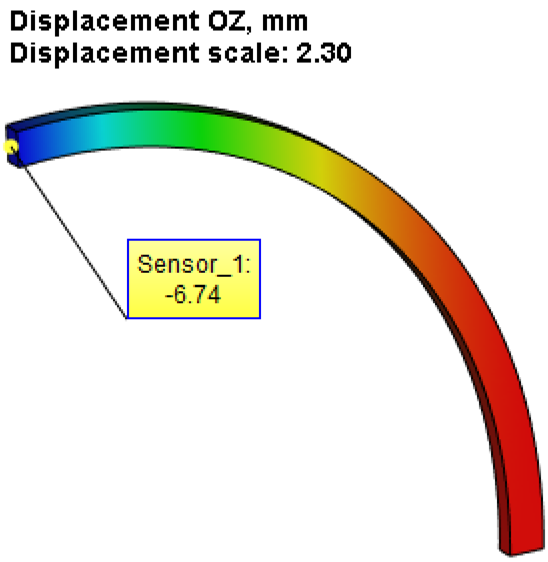
$$\sigma^{max} = \frac{M}{W}, \quad u^{Hor} = \left(\frac{\pi}{2} - 1\right)\frac{MR^2}{EJ}, \qquad u^{Vert} = \frac{MR^2}{EJ}$$

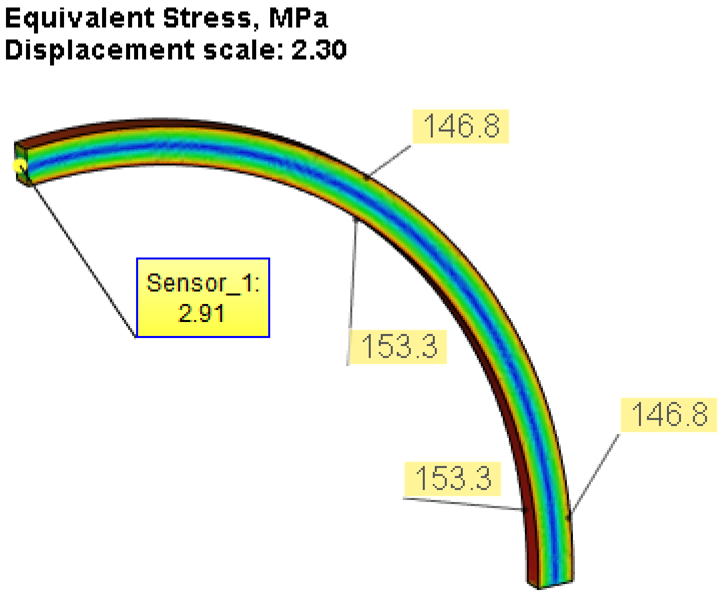
Here E is Young modulus, W is the section modulus at bending, J is the section axial moment of inertia:

$$J = \frac{bh^3}{12}, \qquad W = \frac{bh^2}{6}$$

J=6.667E-09 m^4, W=6.667E-09 m^3. σ_{max}=150 MPa, u^{hor}=3,853 mm, u^{vert}=6,75 mm.







The results are given in the table:

Table 1.			
Finite elements mesh parameters			
	Finite element type	Number of vertices	Number of finite elements
	Quadratic tetrahedrons	2092	7582
Table 2.			
Calculation results, Displacement in the directions of X,Z mm			
	Numerical solution w^*	Analytical solution w, m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$
	3.849	3.853	0.10
	6.74	6.75	0.14
Table 3.			
Calculation results, Equivalent stress, MPa			
	Numerical solution w^*	Analytical solution w, m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$
	150	153,3	2,2

Broken Bar under the Action of a Remote Force

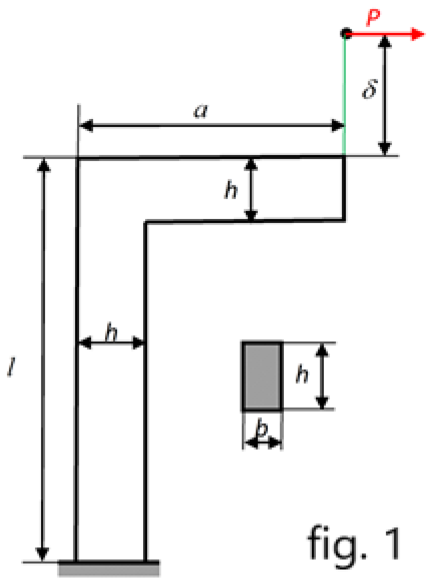


fig. 1

Let's consider a bar consisting of two rectangular horizontal and vertical parts (Fig.1). The action of the body onto the bar can be taken into account via moment M applied to the right end of the bar. The formulate determining maximum normal stresses in the cross section, horizontal and vertical displacements of the free end of the bar (to be precise, the displacement of the cross section center of gravity at the free end) are determined according to the strength of materials methods [1]. The bar cross section is a rectangle. The long side h is parallel to the drawing plane and the short side b is perpendicular to the drawing plane. The butt end of the bar is fixed and the top one holds a body whose rigidity is much higher than that of the bar. Force F is applied to the body. b=10 mm, h=20 mm, l=300 mm, a=100 mm, δ=100 mm, E=2E+011 Pa, P=500 N.

$$\sigma^{max} = \frac{M^{max}}{W}, \quad u^{hor} = \frac{Pl^2}{6EJ}(3\delta + 2l), \qquad u^{vert} = \frac{Pa}{2EJ}(2\delta l + l^2 + a\delta)$$

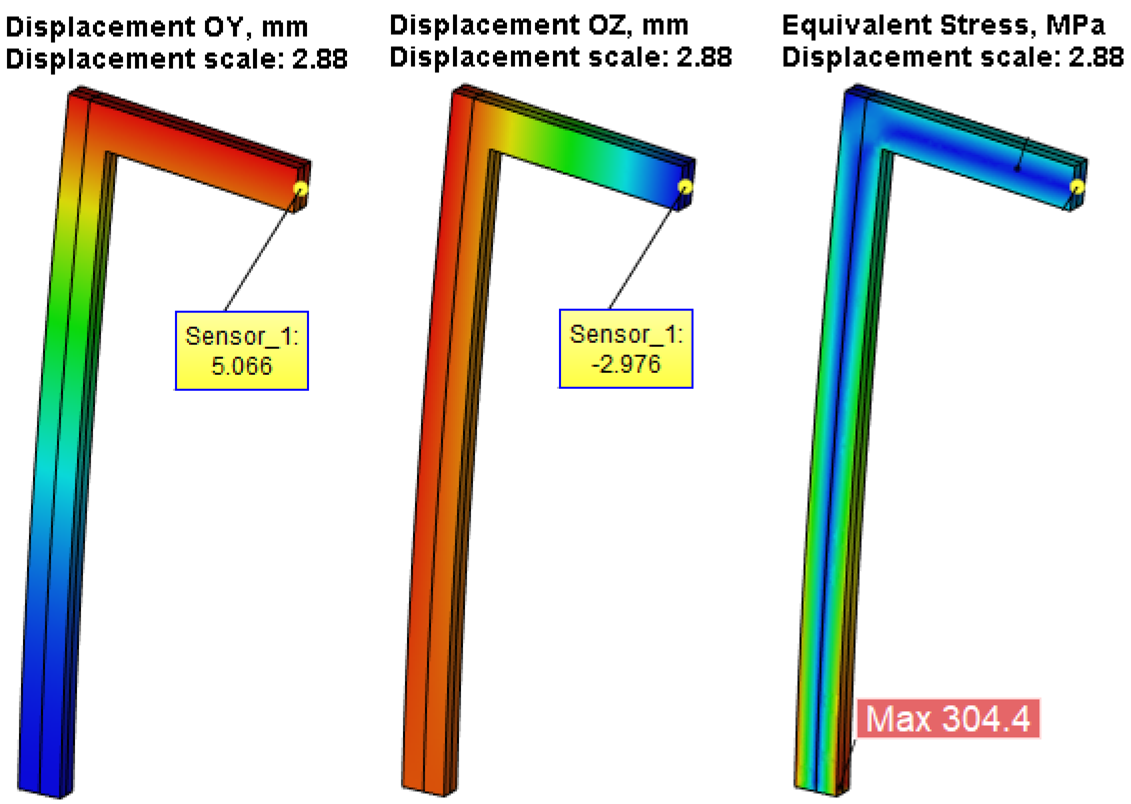
Here E is Young modulus, W is the section modulus at bending, J is the section axial moment of inertia:

$$J = \frac{bh^3}{12}, \qquad W = \frac{bh^2}{6}$$

Maximum bending moment appears at the butt end of the bar and is equal to:

$$M^{max} = P \cdot (\delta + l)$$

J=6.667E-09 m^4, W=6.667E-09 m^3. σ_{max}=300 MPa, u^{hor}=5,063 mm, u^{vert}=3,0 mm.



The results are given in the table:

Table 1.			
Finite elements mesh parameters			
	Finite element type	Number of vertices	Number of finite elements
	Quadratic tetrahedrons	2060	7576

Table 2.			
Calculation results, Displacement in the directions of X,Z mm			
	Numerical solution w^*	Analytical solution w , m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$
	5,066	5,063	0.06
	2,976	3,0	0.8

Table 3.			
Calculation results, Equivalent stress, MPa			
	Numerical solution w^*	Analytical solution w , m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$
	300	304,4	1,44

A Beam with a Distant Mass Loading

Let's consider a beam of length l and rectangular cross section b × h. The left end of the beam is rigidly fixed and the right one holds a cube whose rib's length is H (Fig. 1). The beam and the cube are made of the same material. Gravity force acts upon the points of the beam and the cube. b=20 mm, h=50 mm, l=1000 mm, H=100 mm, δ =100 mm, E=2E+011 Pa, ρ=7900 kg/m^3.

Now we find the beam axis displacement and normal stresses in its cross sections.

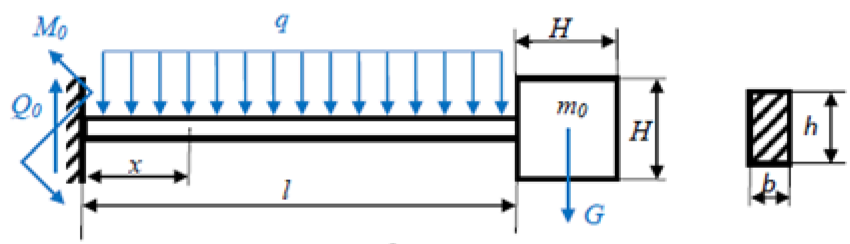


fig. 1

The rigidity of the cube is considerably higher than that of the beam so we can treat the cube as an absolutely rigid body.

Gravity force $G = \rho H^3 g$ acts in the center of mass of the cube, where ρ is the material density, g=9.807 m/sec^2 is the gravitational acceleration. The weight load applied at the points of the beam is brought to uniformly distributed along the axis linear load with the intensity: $q = \rho b h g$.

As the beam is statically determinate, it is sufficient to merely define the axial distribution of the bending moments and lateral forces:

$$M(x) = -\left[\left(l - x + \frac{H}{2}\right)G + q\frac{(l-x)^2}{2}\right], \quad Q(x) = \frac{dM(x)}{dx} = G + q(l - x)$$

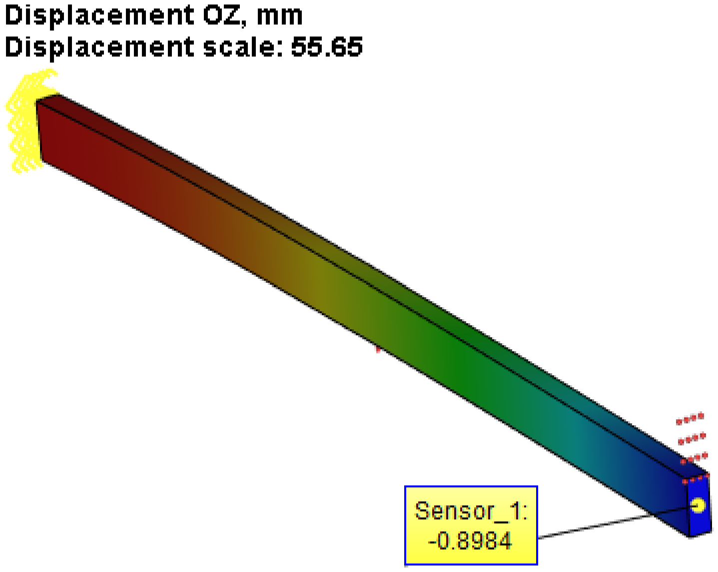
Maximum (as an absolute value) normal stress in the section with the coordinate x is calculated by the formula: $\sigma^{max}(x) = M(x)/W$, where $W = bh^2/6$ is the section moment at bending. The maximum stress in the whole beam is at its fixing point (x=0).

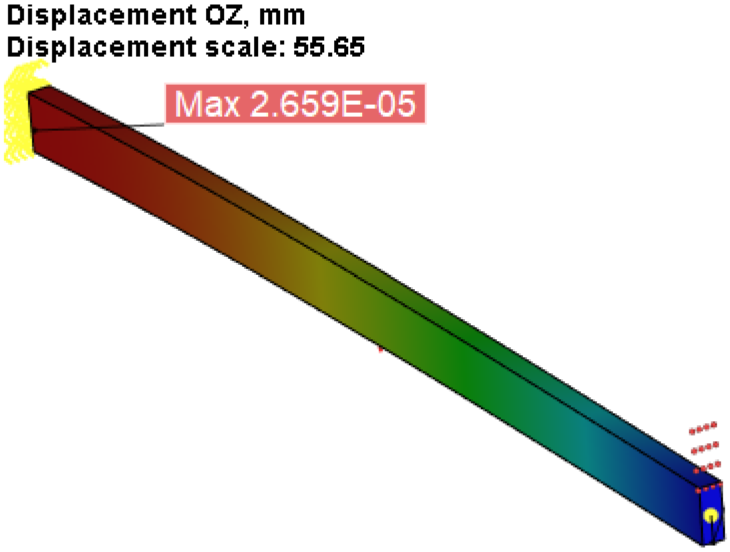
The beam axle deflections are calculated by initial parameters method:

$$v(x) = \frac{1}{EJ}\left[-M_0\frac{x^2}{2} + Q_0\frac{x^3}{6} - q\frac{x^4}{24}\right]$$

Here: $M_0 = |M(0)|$ is the reactive fixed-end moment; $Q_0 = |Q(0)|$ is the reactive fixed-end force; E is Young modulus; $J = bh^3/12$ is the axial inertia moment of the section.

J=2.083E-07 m^4, W=8.333E-06 m^3, M_0=120.082 Nm, Q_0=154.945 N, q=77,473 N/m, G=77,473 N, σ_{max}=14,41 MPa, *v(l)*=-0.899 mm.





The results are given in the table:

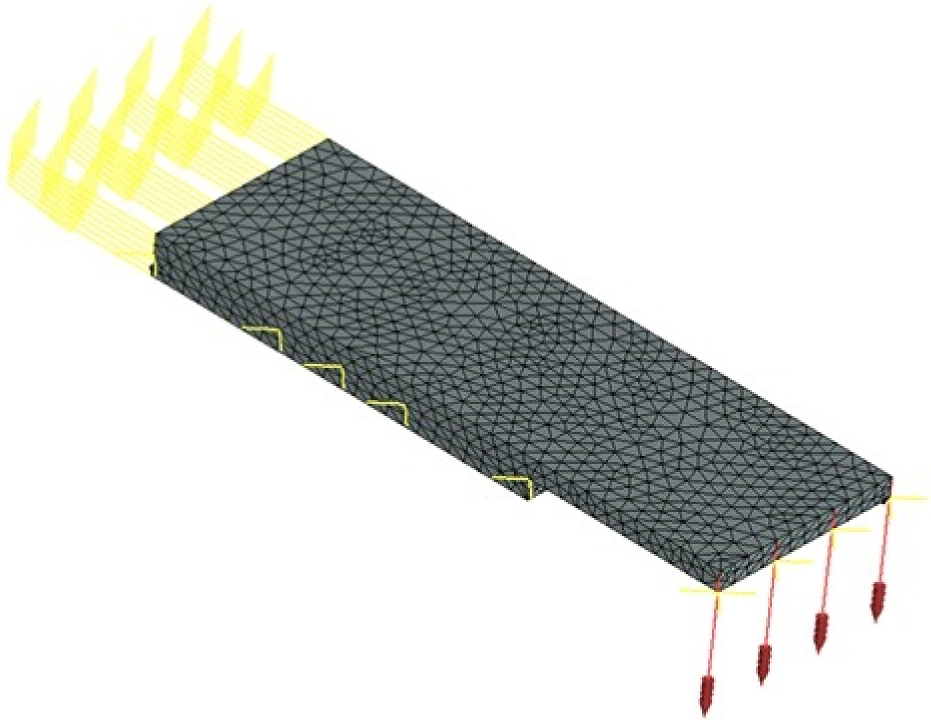
Table 1.

Finite elements mesh parameters				
	Finite element type	Number of vertices	Number of finite elements	
	Quadratic tetrahedrons	2429	9320	
Table 2.				
Calculation results, Displacement in the directions of X,Z mm				
	Numerical solution w^*	Analytical solution w, m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$	
	0.898	0.899	0.11	
Table 3.				
Calculation results, Equivalent stress, MPa				
	Numerical solution w^*	Analytical solution w, m	Error $\delta_u = \frac{\lvert w - w^* \rvert}{w} \times 100\%$	
	14.38	14.41	0.21	

Example of solving Problems with Contacts

Contact of a Flat Spring

Let us consider a flat spring composed of two parts. The length of the first plate is 3L, the length of the second plate is 2L. The plates have equal width b and equal height h. The plates are fixed at the left and loaded at the right by a load P. It is assumed that the plates are not bonded. Each plate can move freely with respect to another one (without friction).



Finite element model for indicated loads and constraints.

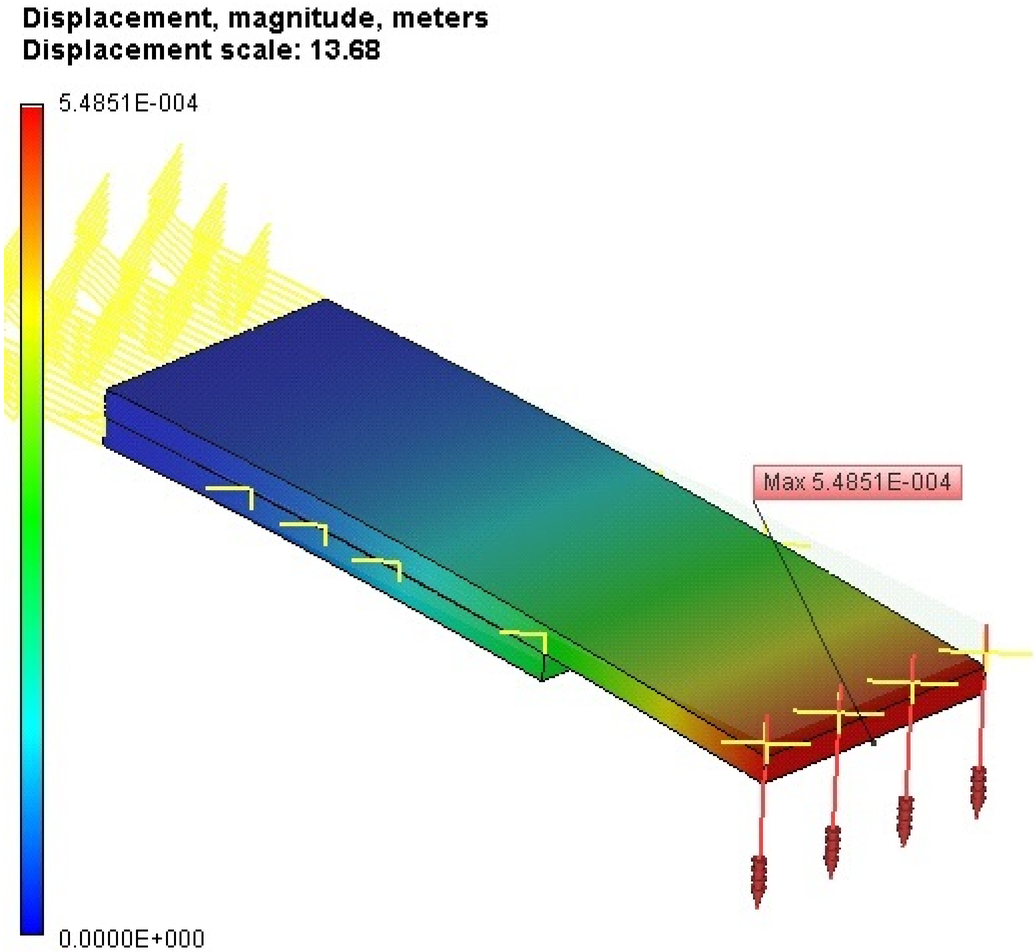
Let us take the following input data: length L is equal to 0.05 m, width b is 0.05 m, height h of each plate is 0.005 m and the magnitude of the applied force P is 100 N.
Material's characteristics (steel): E = 2.1E+011 Pa, Δ = 0.28.
The maximum vertical displacement Δz can be calculated as follows: Δz= 118*P*L3 / 24*E*J , where P- applied force, L - length, J – axial moment of inertia.
J= b*h3 / 12, where b - width and h – height of each plate.
Calculations obtained from the formulas given above yield the following results: Δz = 5.6190E-004 m.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	2143	7632

Table 2. Result "Displacement"

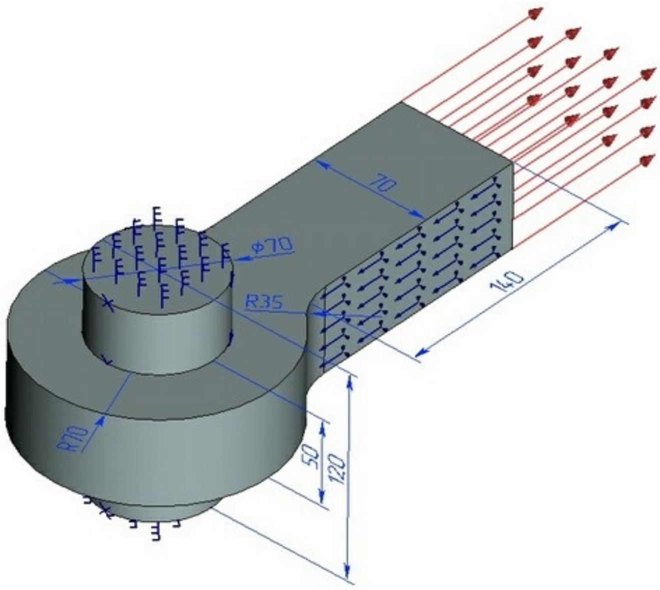
Numerical solution Δz^*, m	Analytical solution Δz, m	Error $\delta=100\%*\lvert\Delta z - \Delta z^*\rvert/\lvert\Delta z\rvert$
5.4829E-004	5.6190E-004	2.42



Conclusions:
The relative error of the numerical solution compared to the analytical solution is equal to 2.4% for quadratic finite elements.

Contact between Axis and Sleeve

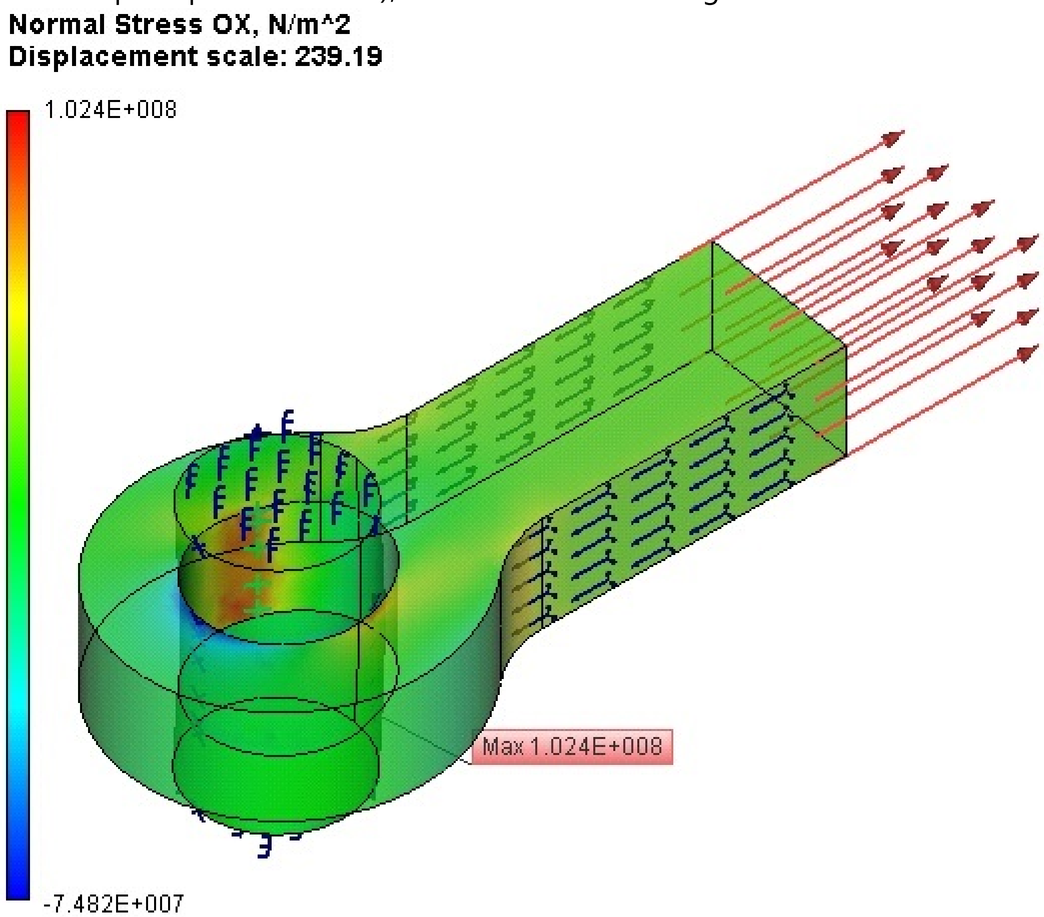
Let us consider the contact between the axis and sleeve along the cylindrical surface; friction is not taken into account (see the figure). At the free edge of the sleeve the normal force is applied P = 100 KN.



Let us take the following input data: diameter of the axis d = 30 mm, length of the axis H = 40 mm, external radius of the sleeve R = 25 mm, thickness of the sleeve h = 10 mm, width of the sleeve's bar b = 20 mm, length of the sleeve's bar b1 = 40 мм, radius of rounding of the sleeve is r1= 10 mm.
Material's characteristics: $E = 2.1\times10^{11}$ Pa, $\nu = 0.28$.
The normal stress in the cross-sections of the bar can be determined from the semi-empirical formula [1, p. 190]:

$$\sigma = k\frac{P}{h(2R-d)},$$

where P – normal force, N; k – stress intensity factor, k = 3,6.
Calculation performed from the formulas given above yields the value $\sigma = 1{,}029\cdot10^{8}$ Pa.
After calculations are carried out with the help of T-FLEX (it is possible to estimate the magnitude of σ from the principle stresses σ1), we obtain the following results: the value of σ1 = 1,027·108 N/m2.



Results are shown in the table.

Table 1. Parameters of finite element mesh

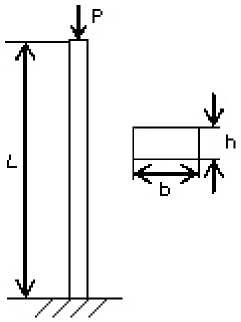
Mesh parameters			
Finite element type	quadratic tetrahedron (10 nodes)		
Number of nodes	2241		
Number of finite elements	10368		
Results of calculations			
	Numerical solution, w^*	Analytical solution, w	Error $\delta_u = \frac{\lvert w - w^*\rvert}{w}\times100\%$
Stress σ, N/m^2	$1{,}024\cdot10^8$	$1{,}223\cdot10^8$	19 %

Examples of solving Buckling Studies

In this chapter we review the results of solving several model studies that have an analytical solution, in order to assess the accuracy of the finite element analysis system output. All examples brought up here it can be found in the "**Verification examples**" library of files.

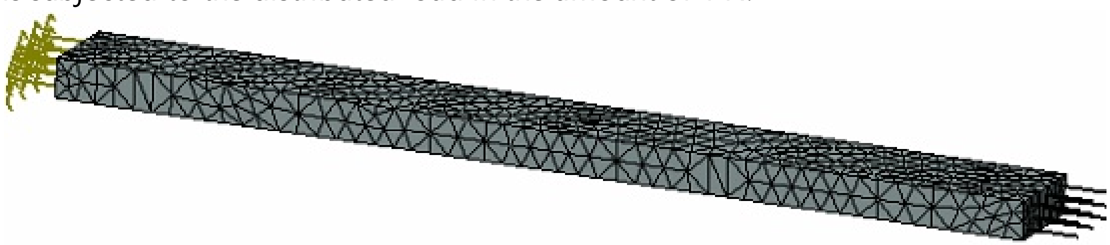
Buckling Analysis of a Compressed Straight Beam

Let's review the buckling analysis of a straight beam compressed with an axial symmetrical load (the Euler's problem). A straight beam of the length l, width and height of the cross-section – b and h respectively, is cantilevered at one end, and a compressing load P acting on the other end. Sought is the load factor corresponding to the start of the beam buckling. Assume the beam length equal to 0.5 m, and the cross-section dimensions b=0.05 m, h=0.02 m.



Material characteristics assume default values: Young's modulus E = 2.1E+011 Pa, Poisson's ratio v = 0.28.

Let's define the boundary conditions as follows. The bottom face is fully restrained, and the upper one is subjected to the distributed load in the amount of 1 N.



Finite element model of the beam for buckling analysis

The analytical solution to determine the critical load appears as:

$$P_{krit} = \frac{\pi^2 EJ}{(\mu l)^2} = 6.9087\times10^4\ N,$$

where E – the Young's modulus, J – the moment of inertia, l – the beam length, μ – the length factor that depends on the support arrangements and the beam loading method. In this case, μ =2.

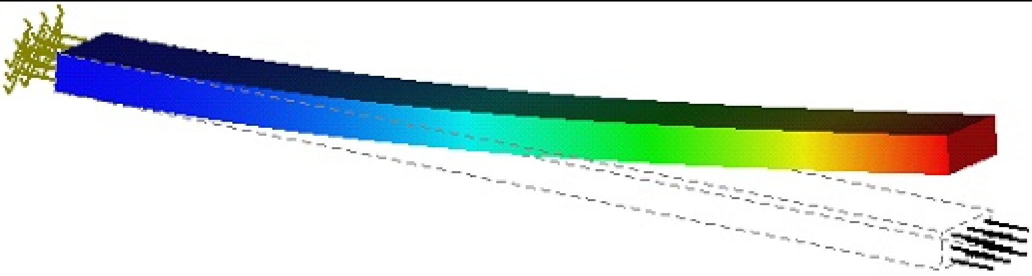
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite Element Type	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	839	2183

Table 2.
Result «Critical load»

Numerical solution Critical load σ_{krit}, $\frac{N}{m^2}$	Analytical solution Critical load p_{krit}, $\frac{N}{m^2}$	Error $\delta = \frac{\left\| p_{krit} - \sigma_{krit} \right\|}{\left\| p_{krit} \right\|} \times 100\%$
6.9341E+004	6.9087E+004	0.37

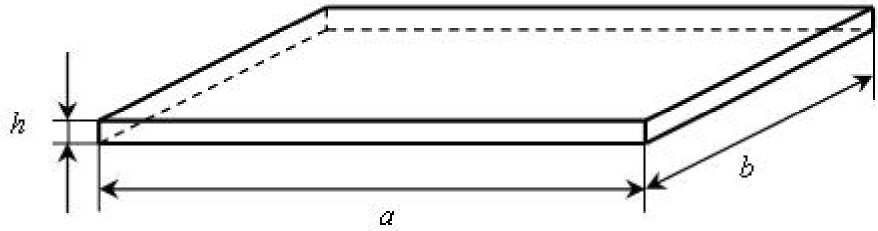


First buckling mode of the beam

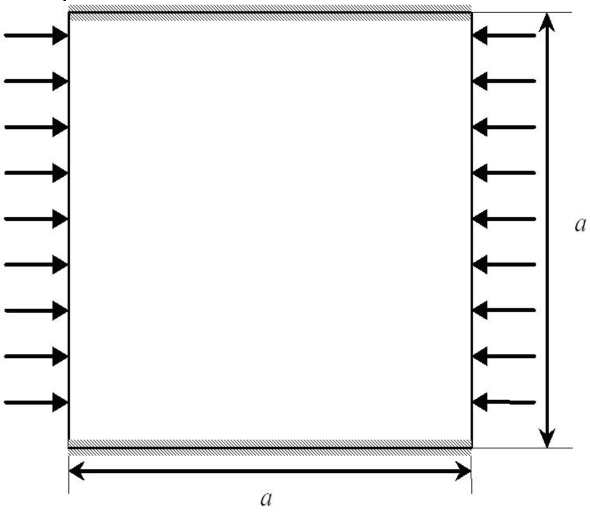
Conclusions:
The relative error of the numerical solution compared to the analytical solution is equal to 0.4% for quadratic finite elements.

Buckling of Square Plate (+ Plate FE)

Consider a square plate with a side a and thickness h (see figure).



The thickness of plate h is much smaller than the length of its side a.
The plate is uniformly compressed in a transversal direction.
Consider the case when the loaded edges of plate are simply-supported; non-loaded edges are clamped.



Let us use the following data: plate side length $a = 500mm$, thickness of plate $h = 3mm$, applied distributed force $P = 1\frac{N}{m^2}$.

Elastic properties are taken as: $E = 2.1\times10^{11}\ Pa$, $\nu = 0.28$.

Analytical solution for this problem is given by: $\sigma_{crit} = K\frac{\pi^2 D}{a^2 h}$,

where E – Young's modulus,

$D = \frac{Eh^3}{12(1-\nu^2)}$ – cylindrical stiffness of plate,

K – coefficient whose value depends on the type of the supports of the plate edges (in this case $K = 7.69$).

Thus, $\sigma_{crit} = K\frac{\pi^2 D}{a^2 h} = 0.5188\times10^8\left[\frac{N}{m^2}\right]$

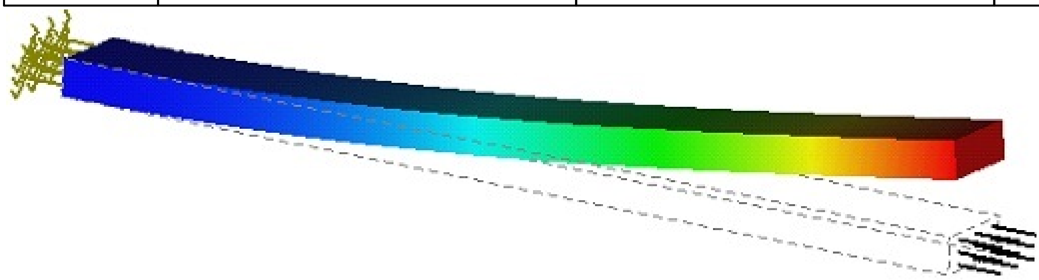
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh Number	Finite Element Type	Number of nodes	Number of finite elements
1	linear triangle (6 nodes)	2238	4306
2	quadratic triangle (6 nodes)	2238	4306
3	quadratic tetrahedron (10 nodes)	4594	13345

Table 2.
Result «Critical load»

Mesh Number	Numerical solution Critical load q, $\frac{N}{m^2}$	Analytical solution Critical load σ_{krit}, $\frac{N}{m^2}$	Error $\delta = \frac{\lvert\sigma_{krit} - q\rvert}{\lvert\sigma_{krit}\rvert}\times100\%$
1	0.5260E+008	0.5188E+008	1.39
2	0.5281E+008	0.5188E+008	1.80
3	0.5277E+008	0.5188E+008	1.72



Conclusions:

The relative error of the numerical solution compared to the analytical solution did not exceed 2% for quadratic finite elements.

Buckling of Rectangular Plate (+ Plate FE)

Consider a rectangular plate with sides a, b and thickness h (see the figure).

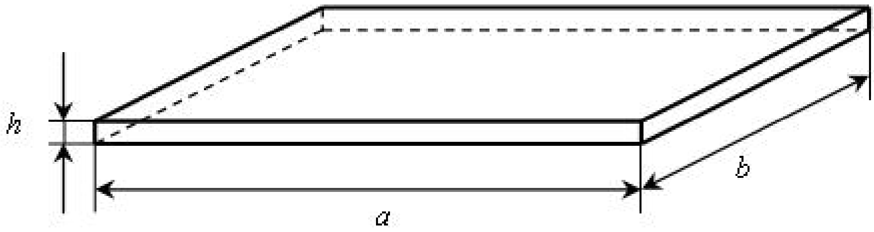
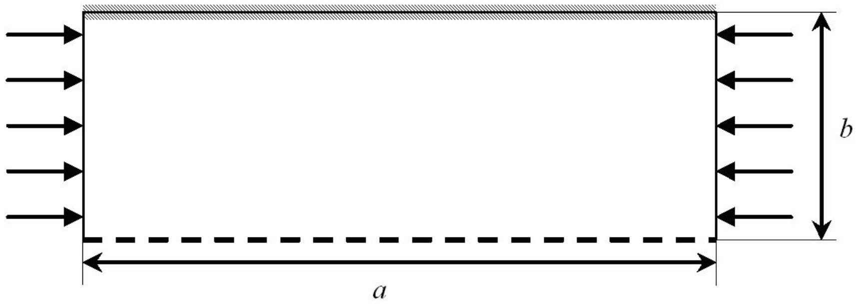


Plate thickness h is considerably smaller than two other dimensions a and b.
The plate is uniformly compressed in a transversal direction.
Consider the case when the loaded edges of plate are simply-supported; one of the non-loaded edges is clamped, another non-loaded edge is free.



Let us use the following data: length and width of the plate $a = 800mm$, $b = 500mm$, plate thickness $h = 3mm$, applied distributed load $P = 1\frac{N}{m^2}$.
Elastic characteristics: $E = 2 \times 10^{11} Pa$, $\nu = 0.25$.

Analytical solution for this problem is given by: $\sigma_{crit} = K\frac{\pi^2 D}{b^2 h}$,
where E – Young's modulus,

$D = \frac{Eh^3}{12(1-\nu^2)}$ – cylindrical stiffness of plate,

K – coefficient whose value depends on the type of the supports of the plate edges and the ratio $\frac{a}{b}$ (in this case $K = 1.33$).

Thus, $\sigma_{crit} = K\frac{\pi^2 D}{b^2 h} = 0.8401 \times 10^7 \left[\frac{N}{m^2}\right]$.

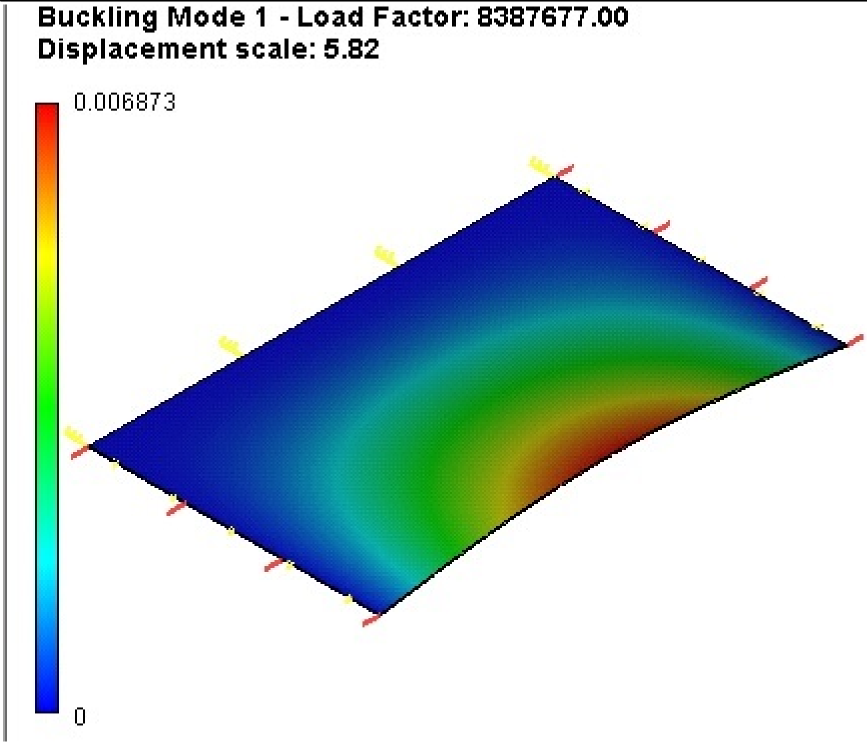
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh Number	Finite Element Type	Number of nodes	Number of finite elements
1	linear triangle (6 nodes)	2105	4040
2	quadratic triangle (6 nodes)	2105	4040
3	quadratic tetrahedron (10 nodes)	4450	12833

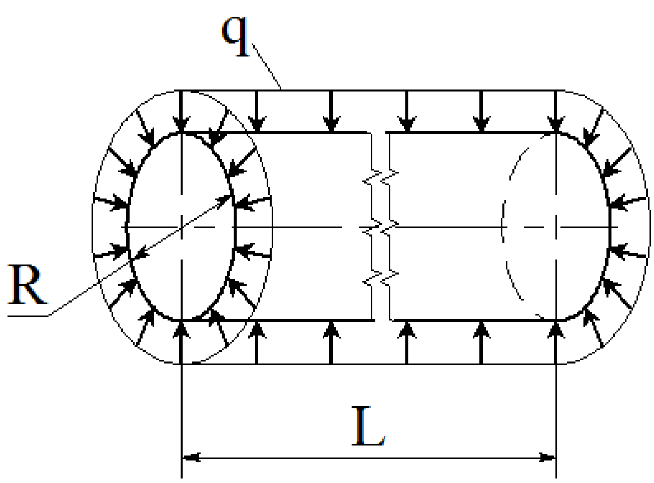
Table 2.
Result «Critical load»

Mesh Number	Numerical solution Critical load q, $\frac{N}{m^2}$	*Analytical solution* Critical load σ_{krit}, $\frac{N}{m^2}$	Error $\delta = \frac{\lvert\sigma_{krit} - q\rvert}{\lvert\sigma_{krit}\rvert} \times 100\%$
1	0.8370E+007	0.8401E+007	0.37
2	0.8391E+007	0.8401E+007	0.12
3	0.8388E+007	0.8401E+007	0.15



Stability of Thin-walled Pipe Compressed under Pressure

Let us consider study of defining critical pressure q cr of the pipe with length L = 1000 mm, radius R = 56 mm, thickness h = 6 mm. Material - steel with .



Scheme of pipe stressing under external pressure

Displacements of the pipe edges are restricted only in radial and circumferential directions.

The magnitude of the critical pressure is determined by the formula:

$$q_{кр} = \frac{D \cdot R \cdot [(m\pi/L)^2 + (n/R)^2]^2}{n^2} + \frac{E \cdot h \cdot (m\pi/L)^4}{R \cdot n^2 \cdot [(m\pi/L)^2 + (n/R)^2]^2}$$

where m, n - number of half-waves axial and along circumference correspondingly;

D - cylindrical stiffness:

$$D = \frac{E \cdot h^3}{12(1 - \nu^2)}$$

h, L, R – thickness, length and radius of the pipe correspondingly;

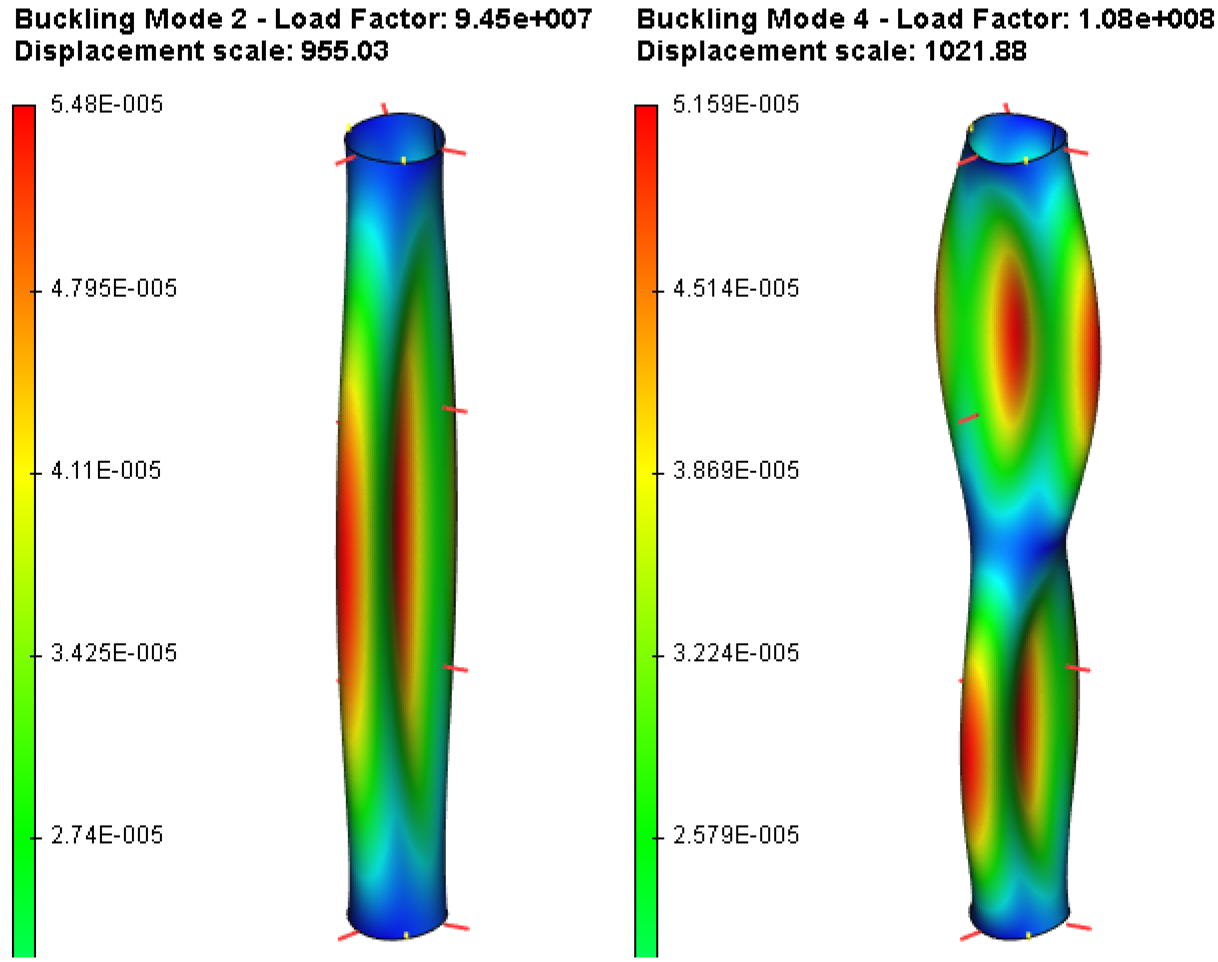
E, ν – Young's modulus and Poisson ratio for the material.

Let us calculate the critical pressure for the following half-wave numbers: (m, n) = (1, 2); (2, 2):

The following results were received after calculation using T-FLEX analysis:

Table 1. Parameters of finite-element meshes				
Mesh number	Type of finite elements	Number of nodes	Number of finite elements	Number of arguments
1	linear triangle (6 nodes)	32895	65606	197370

Table 2."Critical load" Result				
Mesh number	Numerical solution Critical Pressure		Analytical solution Critical Pressure	
1	m=1,n=2	9.4498E+007	9.5204E+007	-0.74
1	m=2,n=2	1.0778E+008	1.0436E+008	3.27



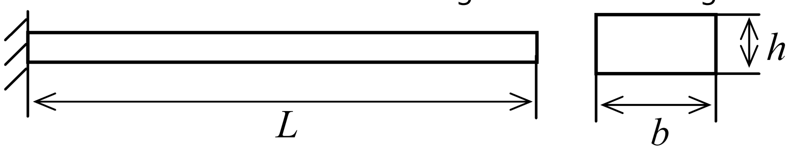
Result of calculation using finite elements

Examples of Frequency Analysis Study

In this chapter we review the results of solving several model studies that have an analytical solution, in order to assess the accuracy of the finite element analysis system output. All examples brought up here it can be found in the "Verification examples" library of files.

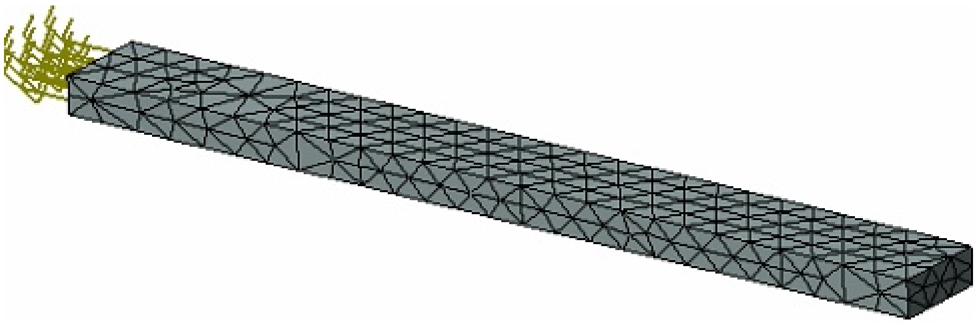
Determining Natural Frequencies of Beam Vibration

Given is a cantilevered beam of length L with a rectangular cross-section of width b and height h.



Sought are the three natural frequencies of the beam.
Assume L=0.5 m, b=0.05 m, h=0.02 m.

The material properties are: the Young's modulus $E = 2.1 \cdot 10^{11}$ Pa, the Poisson's ratio $\nu = 0.28$, the density $\rho = 7800 \, {}^{kg}/_{m^3}$.



Finite element model of the beam with restraints

The analytical solution appears as:

$$f_i = \sqrt{\frac{E \cdot J}{\rho \cdot F}} \frac{1}{2\pi} \left(\frac{k_i}{l}\right)^2,$$

where f_i - natural frequencies, E – the material Young's modulus, J – the moment of inertia, c – the material density, F – the area of the cross section, l – the beam length, k_i - the factor that depends on the vibration mode ($k_1 = 1.875,\ k_2 = 4.694,\ k_3 = 7.855$).

The results are as follows:

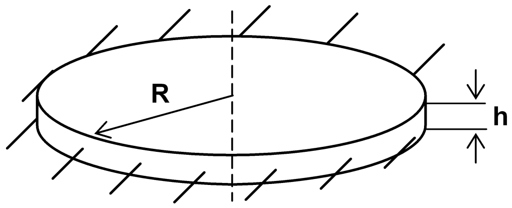
Vibration mode	T-FLEX solution, Hz	Analytical solution, Hz	Error, %
	67.3	67.0	0.4
	419.1	420.2	0.3
	1162.4	1176.7	1.3

Conclusion:

The relative error of the numerical solution compared to the analytical solution is equal to 0,4-1.3% for the first and third forms when using quadratic finite elements and grows with the increase in the form number.

Determining the First Natural Frequency of a Round Plate

Sought is the natural frequency of the first vibration mode of a round plate of radius R and thickness h, clamped along the contour.

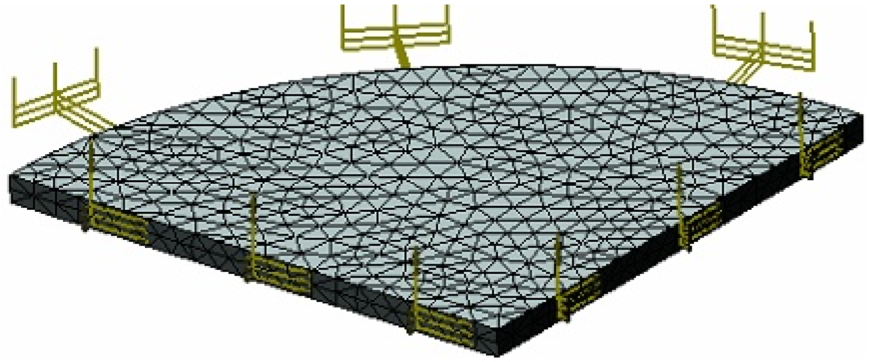


Assume the plate radius equal to R=0.2 m, the plate thickness h=0.01 m. The material properties are: the Young's modulus $E = 2.1 \cdot 10^{11}$ Pa, the Poisson's ratio $\nu = 0.225$, the density $\rho = 7800\ \text{кг}/\text{м}^3$. Due to the symmetry, we will consider the quarter of the plate and apply the appropriate boundary conditions.

Let us calculate the first natural frequency using, first, tetrahedral finite elements and then – triangular elements. Obtained results are compared with the analytical solution which is given by:

$$f = \frac{10.21}{R^2 \cdot 2\pi}\sqrt{\frac{D}{\rho \cdot h}} = 624.5Hz$$
,

where R – plate radius, n – density of material, h – thickness of material, $D = \frac{E \cdot h^3}{12 \cdot (1-\nu^2)}$ – flexural stiffness.



Finite element model of the plate with restraints

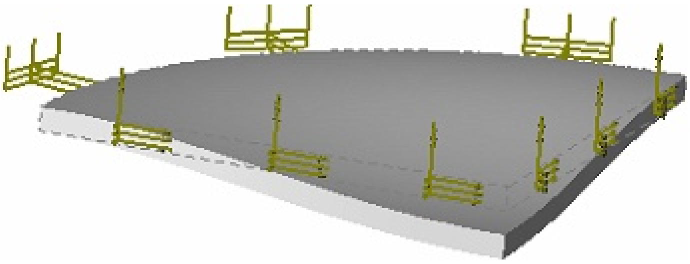
After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh number	**Finite element type**	**Number of nodes**	**Number of finite elements**
1	linear triangle (6 nodes)	1865	3580
2	quadratic triangle (6 nodes)	1865	3580
3	quadratic tetrahedron (10 nodes)	3938	11549

Table 2.
Result «Natural frequencies»

Mesh number	**Numerical solution** **Natural frequency** λ_1, ***Hz***	**Analytical solution** **Natural frequency** f_1, ***Hz***	**Error** $\delta = \frac{\lvert f_1 - \lambda_1 \rvert}{\lvert f_1 \rvert} \times 100\%$
1	623.0	624.5	0.24
2	621.5	624.5	0.48
3	622.1	624.5	0.39



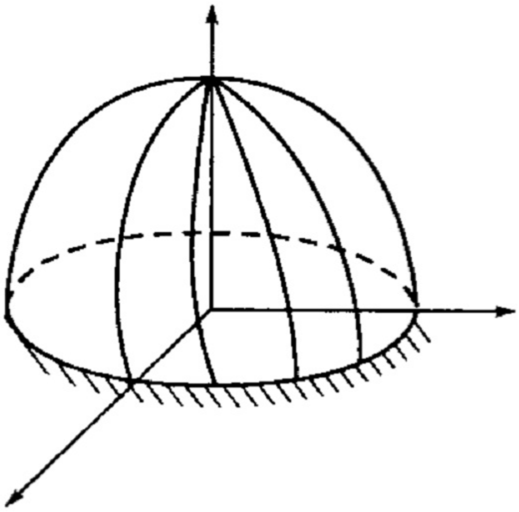
First vibration mode

Conclusions:

The relative error of the numerical solution compared to the analytical solution for the first form is equal to 0,5-0.3% for quadratic triangular and tetrahedral finite element, respectively, however, calculation with the tetrahedral elements required 3 times larger number of finite elements.

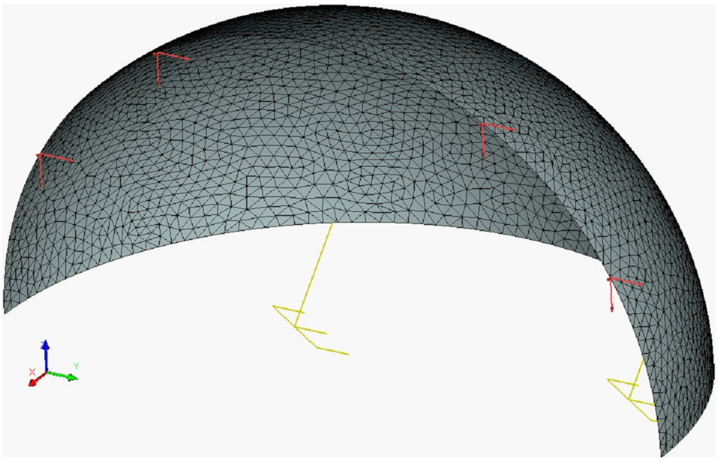
Natural Vibrations of Spherical Dome (+ Plate FE)

Consider a spherical dome of radius R, clamped along the contour (see figure)



Thickness of dome h is considerably smaller than its radius R.

Only one quarter of spherical surface was modeled. The bottom edge was fully restrained, the symmetry boundary conditions were applied to the side faces.



Finite element model of spherical shell with restraints

Let us use the following data: radius $R = 300mm$, thickness oieueia $h = 3mm$ ($\frac{R}{h} = 100$).

Elastic properties: $E = 2.1 \times 10^{11}\ Pa$, $\nu = 0.28$, $\rho = 7800 \frac{kg}{m^3}$.

Analytical solution of this problem is given by: $f = \frac{k_i \cdot \omega_0}{2\pi}$, $\omega_0 = \sqrt{\frac{E}{\rho R^2 (1-\nu^2)}}$,

where E – Young's modulus,

k_i – coefficient whose value for the first five natural frequencies is: 0.5457, 0.7377, 0.8563, 0.8598, 0.9034.

Thus, $f_1 = 1564.7\ Hz$, $f_2 = 2115.3\ Hz$, $f_3 = 2455.4\ Hz$, $f_4 = 2465.4\ Hz$, $f_5 = 2590.4\ Hz$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh number	**Finite element type**	**Number of nodes**	**Number of finite elements**
1	linear triangle (6 nodes)	2840	5510
2	quadratic triangle (6 nodes)	2840	5510
3	quadratic tetrahedron (10 nodes)	5507	16387

Table 2.
Result «Natural frequencies». Mesh №1

Natural frequency number *i*	**Numerical solution** **Natural frequency** λ_i ***, Hz***	**Analytical solution** **Natural frequency** f_i ***, Hz***	**Error** $\delta = \frac{\lvert f_i - \lambda_i \rvert}{\lvert f_i \rvert} \times 100\%$
1	1574.1	1564.7	0.59
2	2107.0	2115.3	0.39
3	2469.9	2455.4	0.59
4	2490.3	2465.4	1.01
5	2592.9	2590.4	0.10

Table 3.
Result «Natural frequencies». Mesh №2

Natural frequency number *i*	**Numerical solution** **Natural frequency** λ_i ***, Hz***	**Analytical solution** **Natural frequency** f_i ***, Hz***	**Error** $\delta = \frac{\lvert f_i - \lambda_i \rvert}{\lvert f_i \rvert} \times 100\%$
1	1573.8	1564.7	0.58
2	2105.4	2115.3	0.47
3	2466.7	2455.4	0.46
4	2488.5	2465.4	0.94
5	2586.7	2590.4	0.14

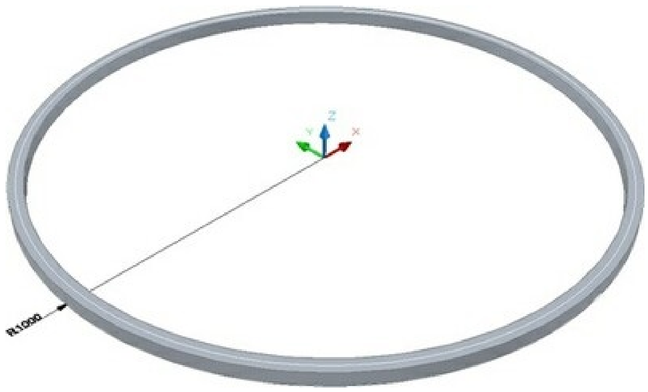
Table 4.
Result «Natural frequencies». Mesh №3

Natural frequency number *i*	**Numerical solution** **Natural frequency** λ_i ***, Hz***	**Analytical solution** **Natural frequency** f_i ***, Hz***	**Error** $\delta = \frac{\lvert f_i - \lambda_i \rvert}{\lvert f_i \rvert} \times 100\%$
1	1574.3	1564.7	0.61
2	2106.2	2115.3	0.43
3	2465.9	2455.4	0.43
4	2487.0	2465.4	0.88
5	2586.1	2590.4	0.17

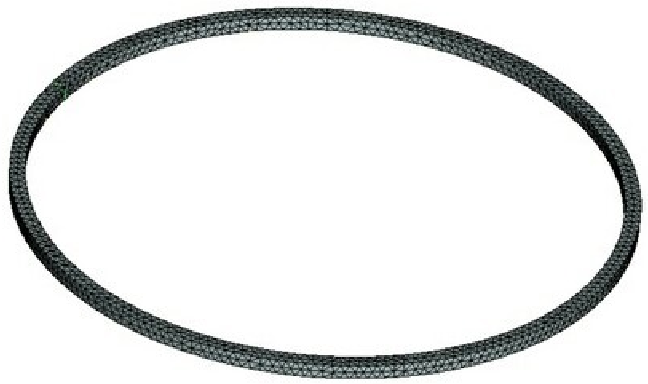
Conclusion:
The relative error of the numerical solution compared to the analytical solution did not exceed 1%.

Flexural Vibrations of a Circular Ring

Let us consider a circular ring. Radius R of the central line is equal to 1 m.



The square cross-section of the ring is much smaller than the radius R. The length of the side of the square is equal to 0.050 m.
The motion of the lower face is constrained along the normal.



Finite element model for indicated loads and constraints.

Material's properties: E = 2.0E+011 Pa, n=0.29, r = 7900 kg / m3.
Analytical solution can be found in the following way:

$$f_i = \frac{1}{2\pi}\sqrt{\frac{EJi^2\left(1-i^2\right)^2}{\rho a^2 R^4\left(1+i^2\right)}}, \quad J = \frac{a^4}{12}.$$

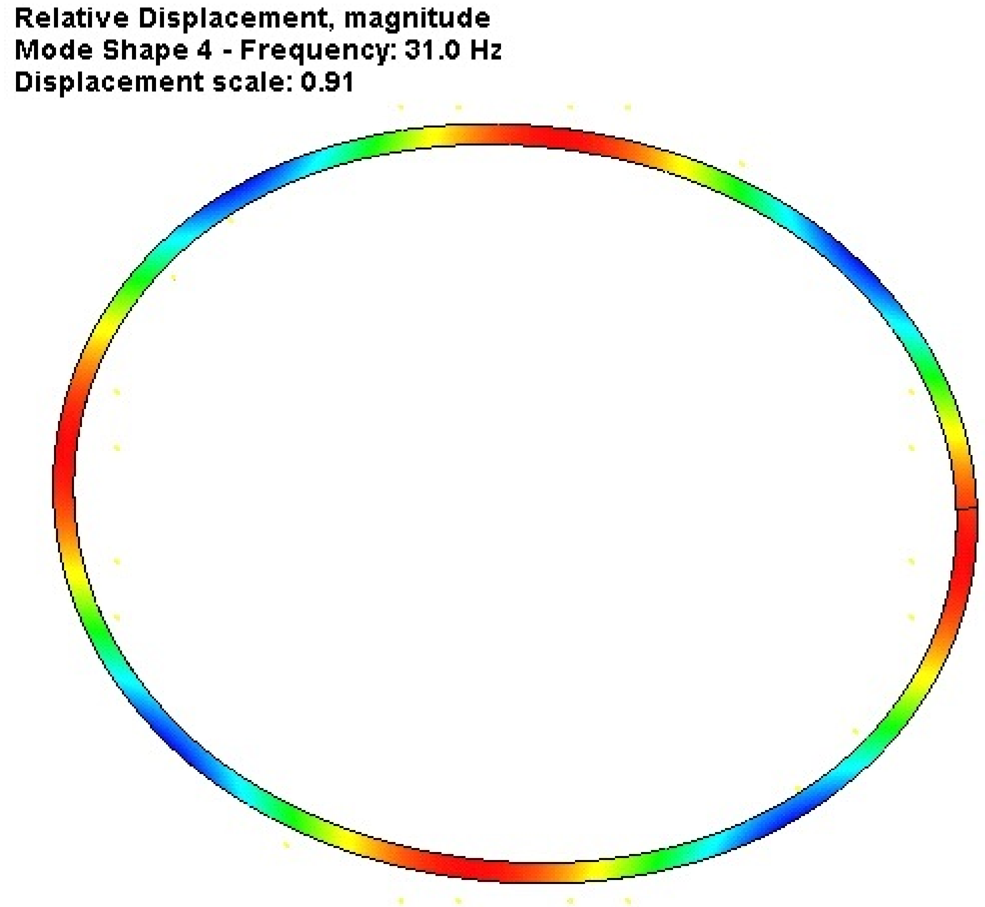
Therefore, f2 = 31.015 Hz , f3 = 87.723 Hz , f4 = 168.201 Hz, f5 = 272.017 Hz.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	1671	3722

Table 2. Result "Frequencies"

Numerical solution f_i^*, Hz	Analytical solution f_i, Hz	Error $\delta = 100\%*\| f_i^* - f_i\| / \| f_i \|$
31.0	31.015	0.05
87.5	87.723	0.25
167.3	168.201	0.54
269.5	272.017	0.9

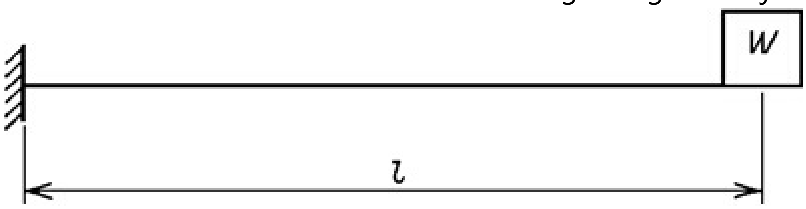


Fourth form of vibrations of a ring

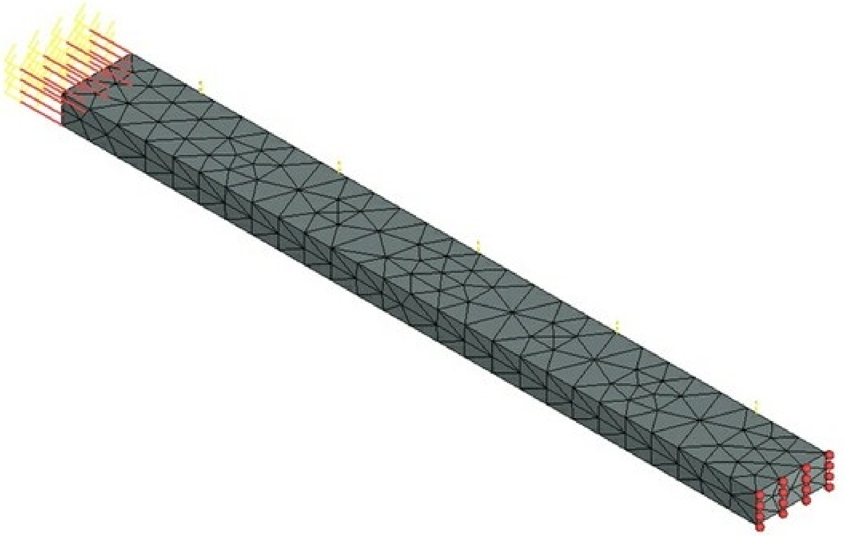
Conclusions:
The relative error of the numerical solution compared to the analytical solution did not exceed 0.9%.

Axial and Transverse Vibrational Frequencies of the Beam under the Weight

Let us consider a cantilever beam whose right edge is subjected to the action of the weight.



The length of the beam is L. The cross-section of the beam is a rectangle of width b and height h. Mass of the weight is M. Mass of the beam itself is m.
m = r F,
where F = b,h – dimensions of the cross-section, r - density of the material of the beam.



Finite element model for indicated loads and constraints.

Let L be equal to 0.5 м, b is equal to 0.02 m, h is equal to 0.05 m.
Material's characteristic's (steel): E = 2.1E+011 Pa, ν =0.28, ρ = 7800 kg / m3.
Mass of the weight M is 2·m·L kg (i.e., 7.8 kg).
Analytical solution can be obtained from the following formulas:
a) Frequency of longitudinal vibrations

$$\frac{f_A \cdot L\sqrt{\rho}}{\sqrt{E}} \cdot tg\left(\frac{f_A \cdot L\sqrt{\rho}}{\sqrt{E}}\right) = \frac{mL}{M}$$

b) Frequency of transverse vibrations

$$f_T = \frac{1}{2\pi}\sqrt{\frac{3EJ}{\left(M + \frac{33}{140}mL\right)L^3}},$$

$$J = \frac{hb^3}{12}.$$

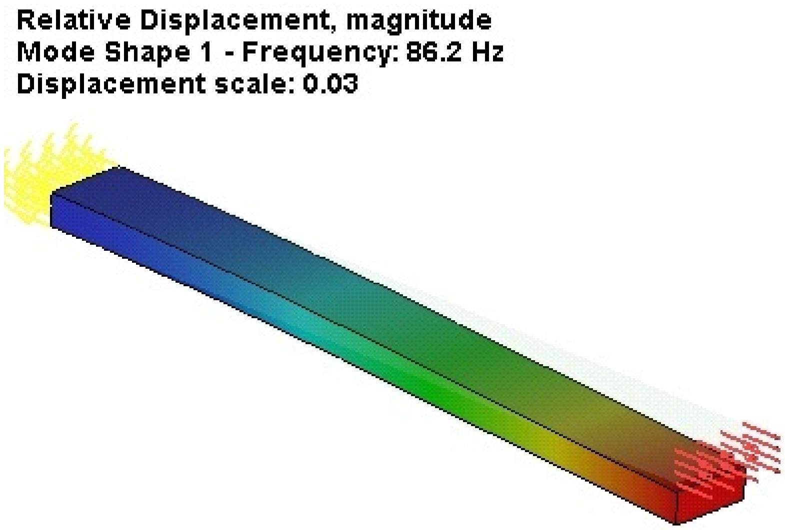
Therefore, fA = 1078.962 Hz, fT = 22.092 Hz.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	314	787

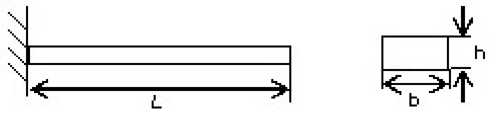
*Table 2. Result "Frequency"**

Numerical solution f^*, Hz	Analytical solution f, Hz	Error $\delta = 100\%*\| f_i^* - f_i\| / \| f_i \|$
22.252	22.092	0.72
1080.462	1078.962	0.14



First Natural Frequency of Cantilever Beam under the Action of Longitudinal Tensile Force

Let us consider a cantilever beam. The length of the beam is L. The cross-section of the beam is a rectangle of width b and height h.



The beam is fixed at the left edge and loaded with a longitudinal tensile force P at the right edge.
Let us assume the following input data: length L of the beam is 0.5 m, width b is 0.05 m, and height h is 0.02 m, the magnitude of the applied force P is 50000 N.
Material's characteristics (steel): E = 2.1E+011 Pa, ν = 0.28.
The sought value – first natural frequency of the beam.

Finite element model for indicated loads and constraints.

Analytical solution is expressed in the following way:

$$f_1^* = f_1 \cdot \sqrt{1+\frac{5PL^2}{14EJ}}$$

$$f_1 = \frac{1}{2\pi}\left(\frac{k_1}{L}\right)^2 \sqrt{\frac{EJ}{\rho F}},$$

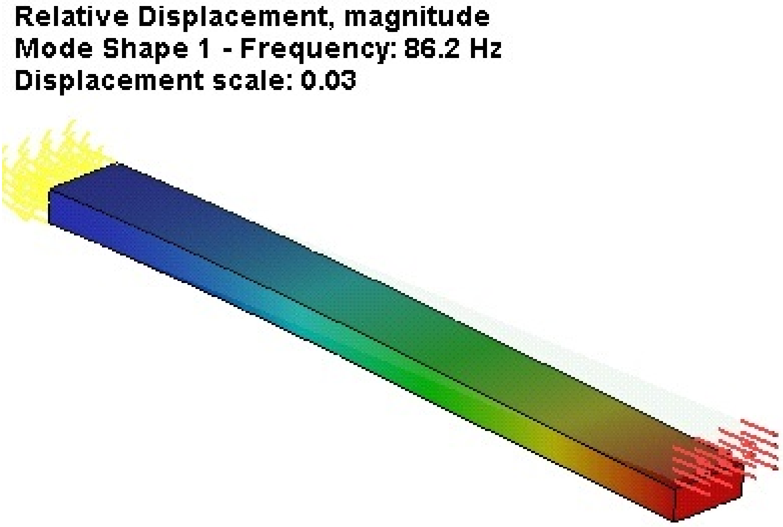
where f1 – first natural frequency of the beam, J –moment of inertia, r – density of the material, F – area of cross-section, k1 = 1.875.
Therefore, f1* = 85.804 Hz
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	314	787

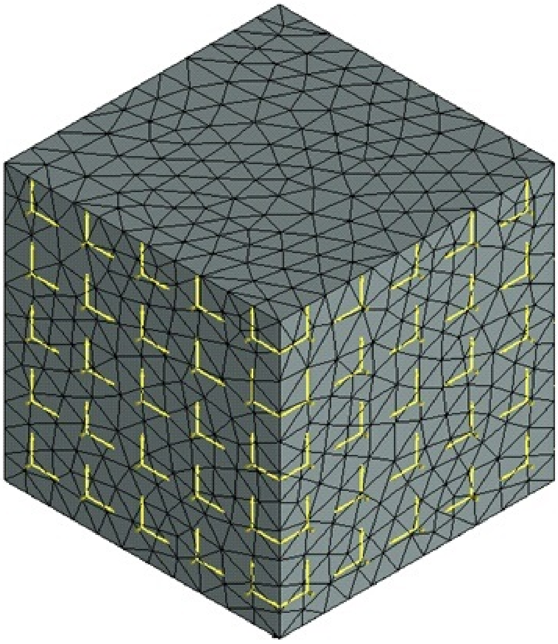
*Table 2. Result "Frequency"**

Numerical solution f^*, Hz	Analytical solution f, Hz	Error $\delta = 100\% * \| f_i^* - f_i \| / \| f_i \|$
86.218	85.804	0.48



First Natural Frequency of the System on an Elastic Foundation

Let us consider the mass that has a form of a cube located on an elastic foundation.



Finite element model for indicated loads and constraints.

The length of the side of the cube is equal to L. Let L be equal to 0.1 m.
Material's characteristics (steel): E = 2.1E+011 Pa, n=0.28, r = 7800 kg / m3.
Mass of the cube M is calculated from the following formula:
M= r L3
Therefore, M = 7.8 kg.
Stiffness of the spring k is equal to 1000 N/m.
Analytical solution can be obtained from the following formula:

$$f = \frac{1}{2\pi}\sqrt{\frac{k}{M}}$$

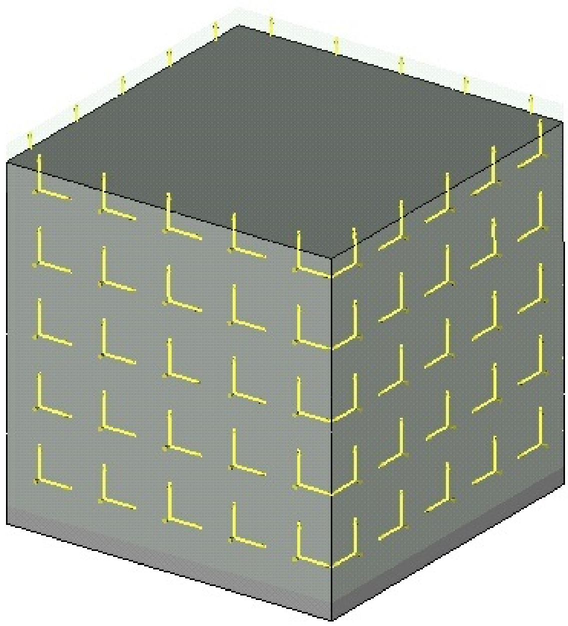
Therefore, f = 1.802 Hz.
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
quadratic tetrahedrons	1722	7505

*Table 2. Result "Frequency"**

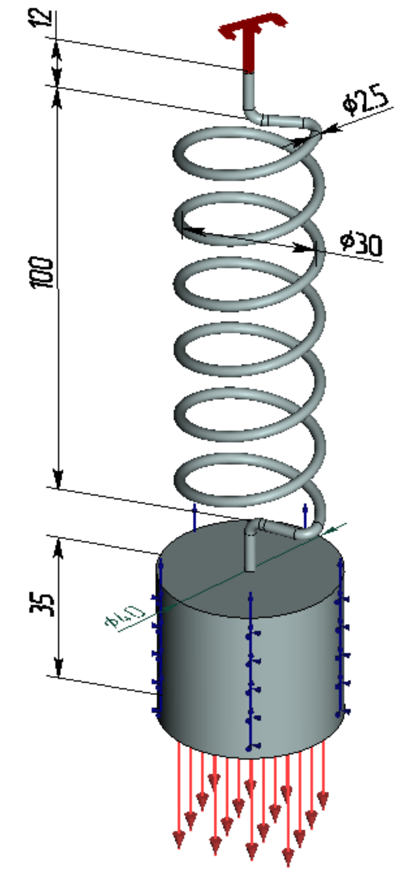
Numerical solution f^*, Hz	Analytical solution f, Hz	Error $\delta = 100\%*\| f_i^* - f_i\| / \| f_i \|$
1.802	1.802	0.00



Examples of Forced Vibrations Analysis Studies

Forced Vibrations of the Weight on a Spring (Excitation by Force)

Let us consider a weight of a cylindrical shape suspended at one end of the spring whose opposite end is fixed (see figure). The vertical load changing as a sinusoidal function is applied at the free edge of the spring.



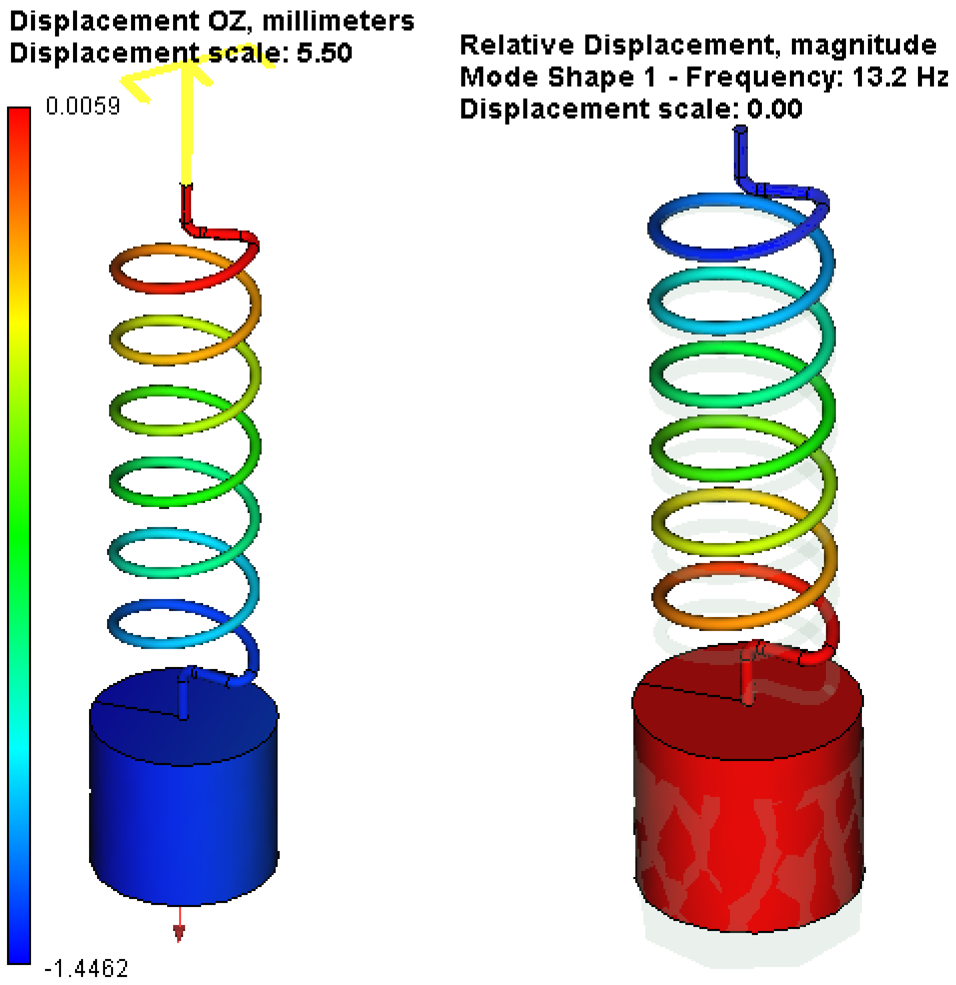
Let us assume the following input data: average diameter of the spring D = 30 mm, length of the spring is H = 100 mm, diameter of the wire is d = 2,5 mm, the number of spring's winds n = 6. Parameters of the weight: diameter Dw = 40 mm; height Hw = 35 mm, mass mw = 0,34306 kg, the amplitude of the load P = 10 N and it changes as a sinusoidal function.
Material's characteristics of the spring and weight: $E = 2.1\times10^{11}$ Pa; ν = 0,28; ρ = 7800 kg/m3; G = 8.203×10^{10} Pa.
It is required to determine the amplitude of vibrations of the weight for the first natural frequency and in the range of frequencies 4 – 20 Hz with an increment equal to 4 Hz. Rayleigh damping parameters: α = 0,02; β = 0,003531. Coefficient of damping for the frequency of the first mode is equal to 15% of the critical value.
The first natural vibrational frequency of the weight is obtained from the formula [1, p. 11 (f. 4)]:

$$f_*^{(1)} = \frac{1}{2\pi}\sqrt{\frac{g}{\Delta z_{st\ w}}},$$

where g – gravitational acceleration, m/s2; Δzst_w – static displacement of the free end of the spring caused by the mass of the weight. It can be found from the formula [2, p. 232, (f. 9.54)]:

$$\Delta z_{st\ w} = \frac{8m_w g\cdot D^3\cdot n}{d^4\cdot G},$$

where mw – mass of the weight, kg; D – average diameter of the spring, mm; n – number of spring's winds, d – diameter of the wire, mm, G – shear modulus, Pa.
The calculation performed from the last formula gives $\Delta z_{st\ w} = 1.361$ mm, finite element analysis gives the value $\Delta z_{st\ w} = 1.4462$ mm.



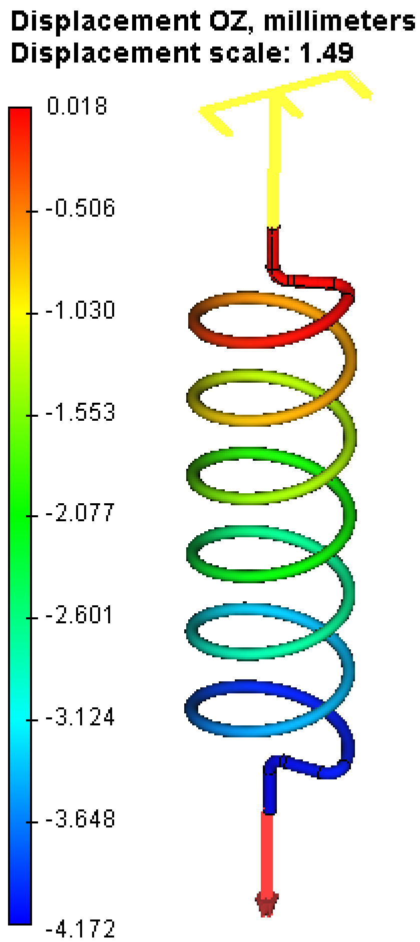
Now let us compute the first natural frequency: the calculation from the formula gives the value fc(1) =13.511 Hz, finite element analysis gives the value fc(1)=13.216 Hz.

Displacement of the free end of the spring subjected to a static loading can be found the formula [2, p. 232, (f. 9.54)]:

$$\Delta z_{st} = \frac{8P \cdot D^3 \cdot n}{d^4 \cdot G},$$

where P – axial force, N; D – average diameter of the spring, mm; n – number of spring's winds, d – diameter of the wire, mm, G – shear modulus, Pa.

Calculation performed from the formula given above yields the value $\Delta z_{st} = 4.045$ mm. Finite element static analysis gives $\Delta z_{st} = 4.172$ mm.



Now let us calculate the damped coefficient of amplitude magnification [1, p 74, (f.1.47)]:

$$\beta_D = \frac{1}{\sqrt{\left(1-\left(\frac{\omega_f}{\omega_c}\right)^2\right)^2 + \left(2\gamma\frac{\omega_f}{\omega_c}\right)^2}},$$

where $\omega_f = 2\pi \cdot f_f$, $\omega_c = 2\pi \cdot f_c$ - circular frequency of the applied force and the natural circular frequency, respectively; $\gamma = c / c_{cr} = 0.15$ - damping coefficient. At the resonance for fв = fc the amplitude magnification coefficient is

$$\beta_D = \frac{1}{2\gamma}$$

With the knowledge of the amplitude magnification coefficient, the amplitude Aв of the forced vibrations can be obtained from:

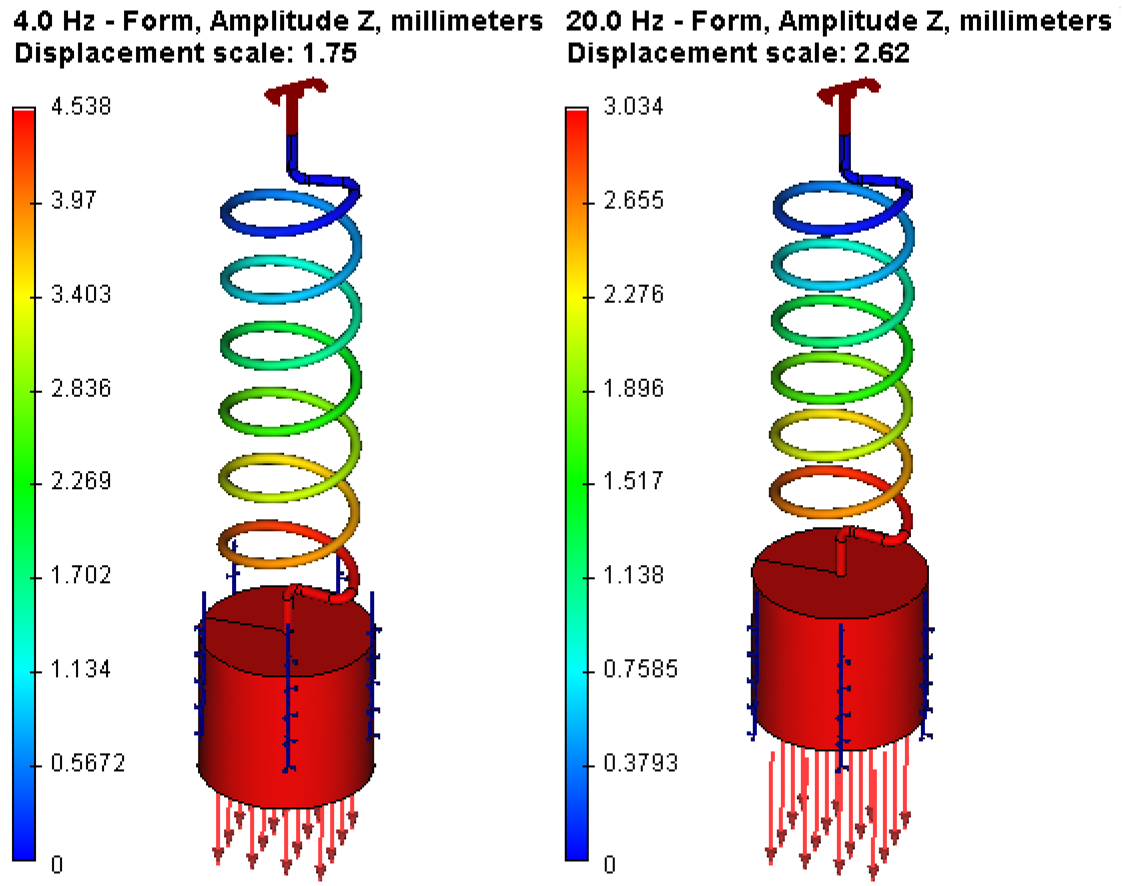
$$A_f = \beta_D \cdot \Delta z_{st}$$

Calculation performed for fв = 13.511 Hz from these formulas gives β_D = 3,333 and Aв = 13,482 mm. To perform analysis with the help of T-FLEX Analysis, let us determine the damping coefficients γ in terms of Rayleigh damping coefficients for the inertia α and stiffness β [1, 304, (f. 4.125)]:

$$\gamma = \frac{\alpha}{2\omega_c} + \frac{\beta\omega_c}{2}$$

For the frequency fc(1) = 13,511, this formula gives the value γ = 0,15.

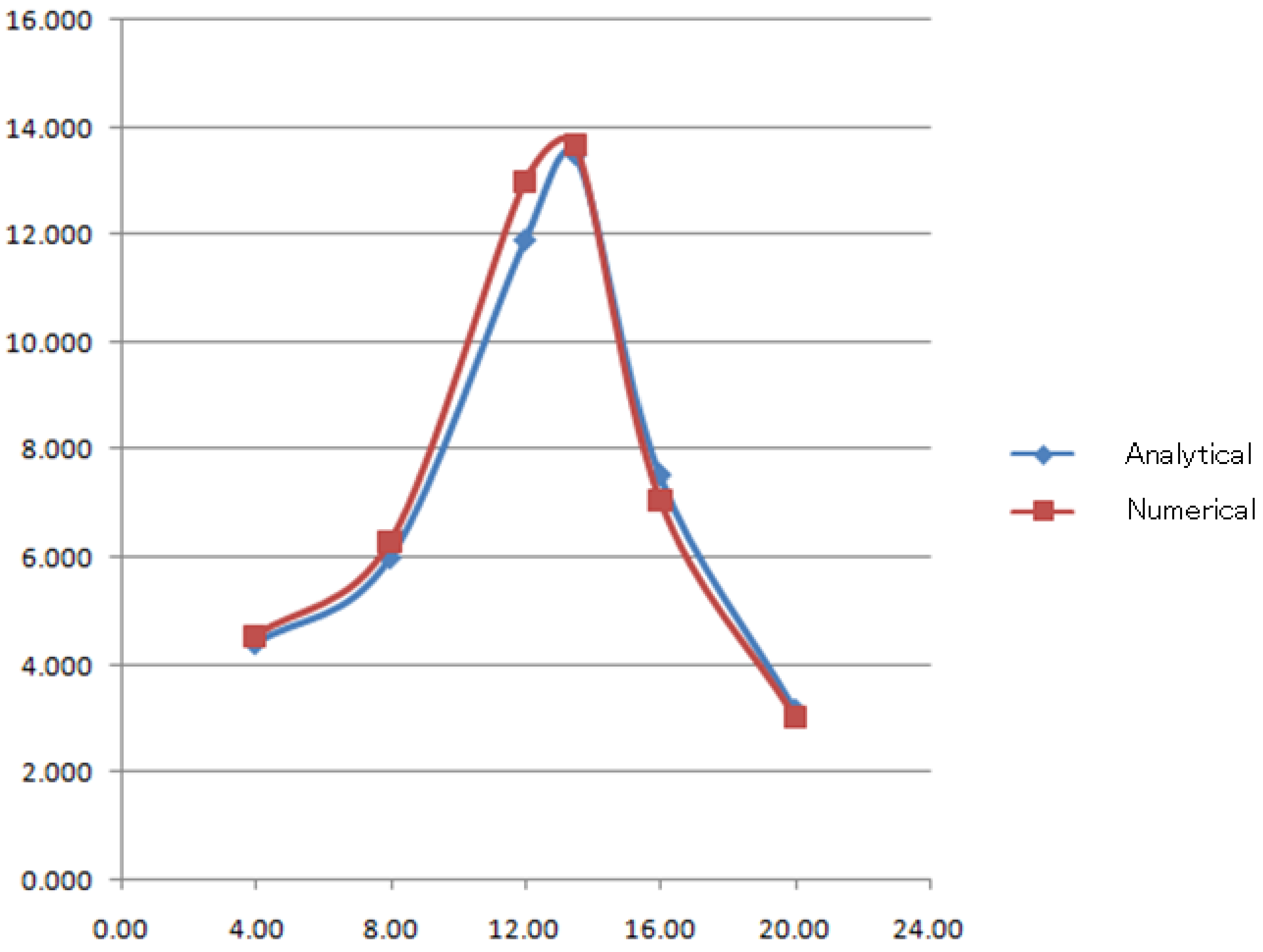
The value of the displacement amplitude can be determined using the «Displacements, amplitude, Z» result.



Results of the analysis are shown in the table.

Table 1. Parameters of finite element mesh and analysis results

Mesh parameters			
Finite element type	quadratic tetrahedron (10 nodes)		
Number of nodes	7614		
Number of finite elements	30846		
Calculation results			
Forced vibration frequency f_B, Hz	Numerical value, A_r^* ,mm	Analytical solution A_r , mm	Error $\delta_u = \frac{\lvert A_r - A_r^* \rvert}{A_r} \times 100\%$
4	4,538	4,412	2,852
8	6,295	6,007	4,792
12	12,98	11,896	9,117
13,511	13,65	13,482	1,245
16	7,068	7,536	6,209
20	3,034	3,182	4,643



Plot of comparison of results

Conclusion:

The relative error of the numerical solution compared to the analytical solution on the resonance frequency is 1.25%, however, in the neighbourhood of the resonance the error reached 10%.

Forced Vibrations of the Weight on a Spring (Kinematic Excitation)

Let us consider a cylindrical weight suspended at one end of the spring. The other end of the spring is fixed on a moving foundation that vibrates according to the sine law (see the Figure). The amplitude of vibrations of the foundation is equal to 1.0 mm.

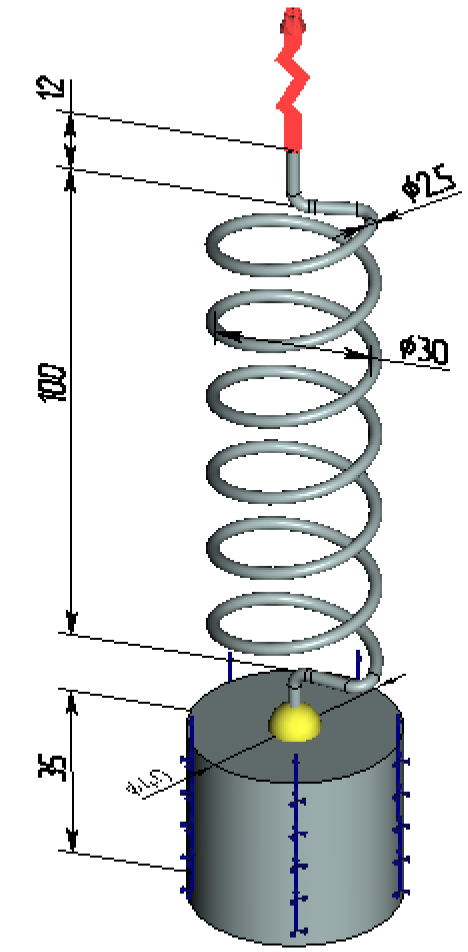


Figure 1 – A numerical model with boundary conditions and a sensor

Let us take the following input data: average diameter of the spring D = 30 mm, length of a spring H = 100 mm, diameter of a wire d = 2.5 mm, number of coils in a spring n = 6. Parameters of the weight: diameter Dw = 40 mm; height Hw = 35 mm, mass mr = 0.34306 kg.
Material properties of the spring and the weight: $E = 2.1 \times 10^{11}$ Pa; ν = 0.28; ρ = 7800 kg/m3;
G = 8.203×10^{10} Pa.
It is required to determine the amplitude of vibrations of the weight for the first natural frequency and in the range of frequencies 4 – 20 Hz with an increment equal to 4 Hz. Rayleigh damping parameters: α =1.25; β = 0.000297732. Coefficient of damping γ for the frequency of the first mode is equal to 2 % of the critical value.
To start FEM analysis, let us create the mesh with a size of an element equal to ~3 mm. On the top end of the spring let us apply the load «oscillator» with an amplitude, along the Z-axis, equal to 1.0 mm; the remaining amplitudes are set equal to zero (by default). The cylindrical face of the weight will be partially constrained with a possibility of displacement along the Z-axis.
The first natural frequency of the weight is determined by the formula [1, p. 11 (ф. 4)]:

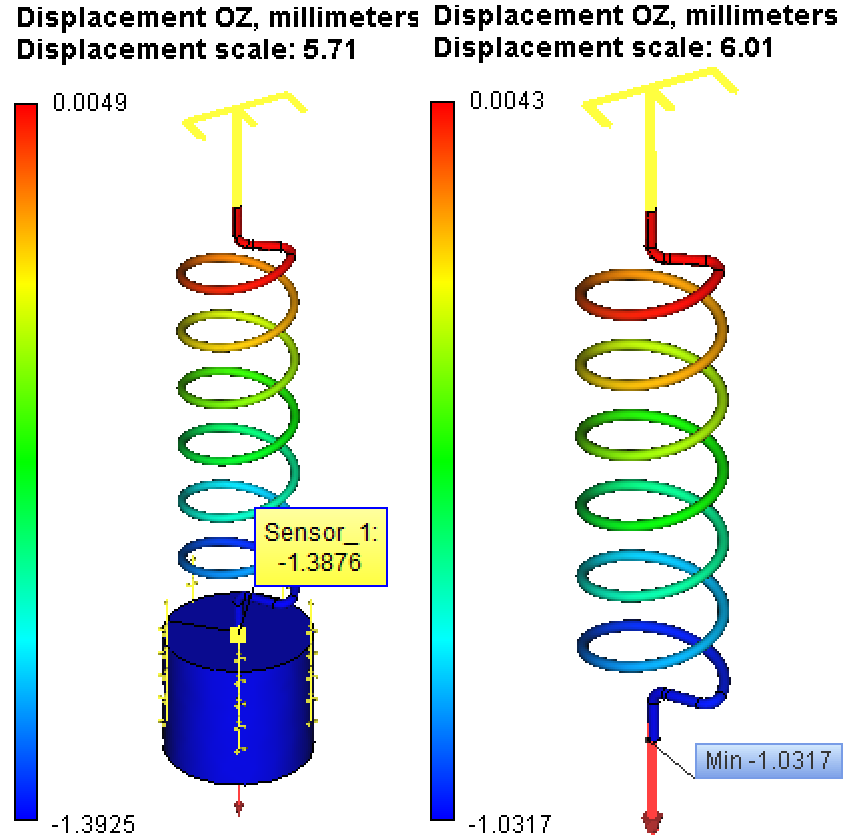
$$f_*^{(1)} = \frac{1}{2\pi}\sqrt{\frac{g}{\Delta z_{st\ g}}},$$

where g – gravitational acceleration, m/s2; Δzst_g – static displacement of the free end of the spring under the action of the weight. The latter can be determined by the formula [2, p. 232, (ф. 9.54)]:

$$\Delta z_{cm\ xp} = \frac{8 m_w g \cdot D^3 \cdot n}{d^4 \cdot G},$$

where mw – mass of the weight, kg; D – average diameter of the spring, mm; n – the number of coils in the spring, d – diameter of the spring, mm, G – shear modulus, Pa.

Calculations carried out by this formula give $\Delta z_{st\ r} = -1.361$ mm; FEM result is $\Delta z^{*}_{st\ r} = -1.388$ mm.



Now, let us calculate the first eigenfrequency: calculation by the formula gives the value fn(1)= 13.511 Hz, FEM calculation gives the value fn(1)=13.216 Hz.

Equivalent exciting force can be found from the displacement of the foundation attached to the upper end of the spring:

$$P = \frac{\Delta z_k d^4 \cdot G}{8D^3 \cdot n},$$

where P – equivalent exciting force, N; D – average diameter of the spring, mm; n – number of coils in the spring, d – diameter of the spring, mm, G – shear modulus, Pa.
Calculation by the aforementioned formulas gives us the value N.
Now let us calculate the damped coefficient of the amplitude magnification [1, p. 74, (ф.1.47)]:

$$\beta_d = \frac{1}{\sqrt{\left(1-\left(\frac{\omega_f}{\omega_n}\right)^2\right)^2 + \left(2\gamma\frac{\omega_f}{\omega_n}\right)^2}},$$

where $\omega_f = 2\pi \cdot f_f$, $\omega_n = 2\pi \cdot f_n$ - circular frequency of the exciting force and the natural circular eigenfrequency, respectively; $\gamma = c / c_{crit} = 0.02$ - damping coefficient. At the resonance when ff = fn the maximum magnification coefficient is

$$\beta_d = \frac{1}{2\gamma}$$

Then, the amplitude Af of the forced vibrations can be determined as:

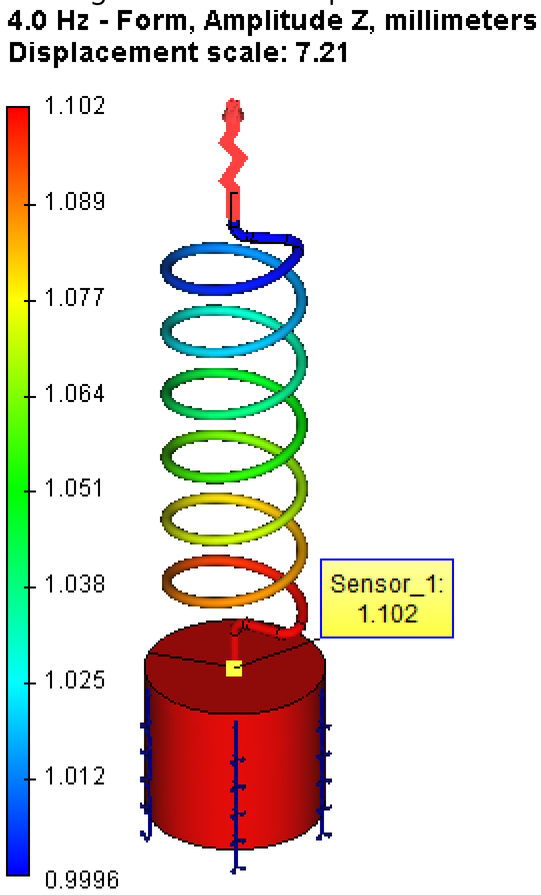
$$A_f = \beta_d \cdot \Delta z_{st}$$

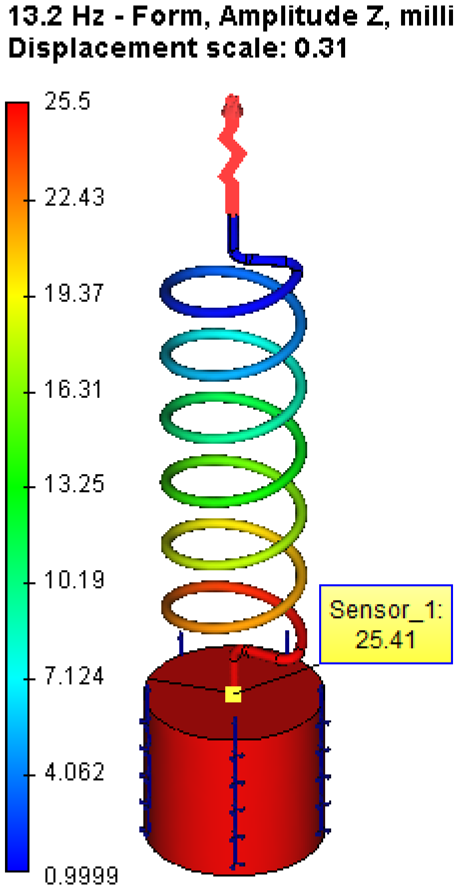
To perform calculation with the help of AutoFEM Analysis, let us determine the damping coefficients γ in terms of Rayleigh damping parameters by inertia α and by stiffness β [1, 304, (f.4.125)]:

$$\gamma = \frac{\alpha}{2\omega_n} + \frac{\beta\omega_n}{2}$$

For the frequency fn(1) = 25.00 Hz, this value is γ = 0.02;

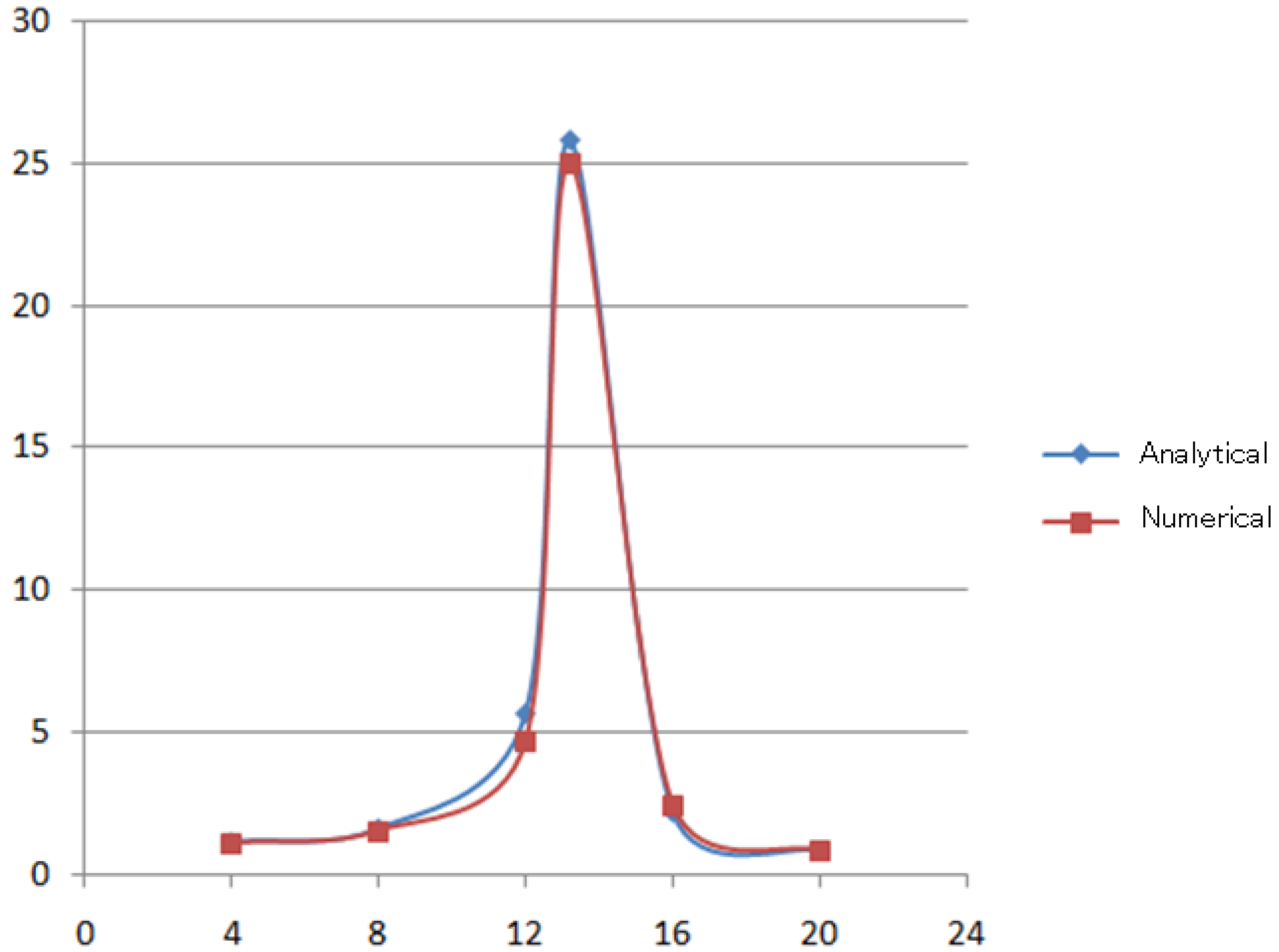
The magnitude of the amplitude can be inferred from the result «Displacements, amplitude, Z»





Results of calculations are shown in the Table.

Mesh parameters			
Finite element type	Quadratic tetrahedral (4 nodes)		
Number of nodes	7614		
Number of finite elements	30846		
Results of calculations			
Forcing frequency f_f, Hz	Numerical solution, A_r^*, mm	Analytical solution A_r, mm	Error $\delta = \frac{\lvert A_r - A_r^* \rvert}{A_r} \times 100\%$
4	1.096	1.102	0.554
8	1.538	1.584	2.907
12	4.671	5.633	20.5
13.126	25.41	25.00	1.66
16	2.466	2.169	-12.05
20	0.838	0.794	-5.17



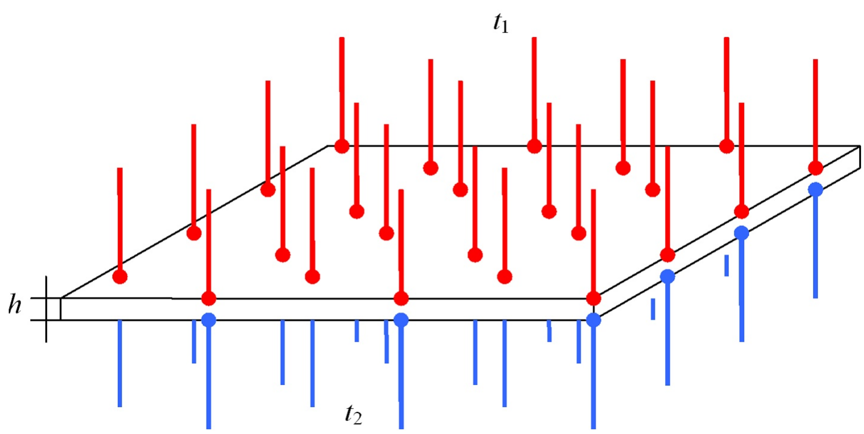
Plots of comparison of results

Conclusion:
The relative error of the numerical solution compared to the analytical solution for the resonance frequency is 1.66 %, however, in the neighbourhood of the resonance it varies from +20% to -12%.

Examples of Thermal Analysis Study

Steady-State Temperature

Consider a problem of steady-state flow of heat in a plate of thickness h with thermal conductivity k, the surface of which is held at temperatures t_1 and t_2 (see figure).



The change in temperature along the thickness of the plate h is defined by relation:

$$\frac{\partial T}{\partial z} = \frac{t_2 - t_1}{h}, t_1 < t_2$$

Thus, the heat flux at any point is equal to:

$$f = -k\frac{\partial T}{\partial z} = \frac{k(t_2 - t_1)}{h} = \frac{(t_1 - t_2)}{R}, R = \frac{h}{k}, t_1 < t_2$$

Now, let us assume that the plate is a composite, that is, consists of n layers with thicknesses $h_1, h_2, \ldots, h_n$ and coefficients of thermal conductivity $k_1, k_2, \ldots, k_n$, respectively. Then, the heat flux for each layer $f_i, i = 1,2,\ldots,n$ can be found from the formula:

$$f_i = -\frac{k_i(t_{i+1} - t_i)}{h_i} = \frac{(t_i - t_{i+1})}{R_i}, R_i = \frac{h_i}{k_i}, t_i < t_{i+1}, i = 1,2,\ldots,n$$

Let the layers have ideal thermal contact across the interfaces; then the heat flux will be continuous when going from one layer to another, and for this particular problem, it will be the same at any point (that is, $f_1 = f_2 = \ldots = f_n = f$). The change in temperature between the opposite external surfaces of the whole composite plate is equal to the sum of temperature changes in each single layer:

$$(t_1 - t_2) + (t_2 - t_3) + \ldots + (t_i - t_{i+1}) + \ldots + (t_n - t_{n+1}) = t_1 - t_{n+1}$$

Then:

$$t_1 - t_{n+1} = f_1 R_1 + f_2 R_2 + \ldots + f_n R_n = (R_1 + R_2 + \ldots + R_n) f, R_i = \frac{h_i}{k_i}, i = 1,2,\ldots n,$$

$$f = \frac{t_1 - t_{n+1}}{\frac{h_1}{k_1} + \frac{h_2}{k_2} + \ldots + \frac{h_n}{k_n}}$$

Let us use the following data: number of layers $n = 3$, length and width of each layer are $0.5m$ and $0.3m$ respectively, thicknesses of layers h_1, h_2, h_3 are equal to $0.007m$, $0.01m$ and $0.003m$. Applied temperatures t_1 and t_4 are equal to $273.15K$ (or 0^0C) and $373.15K$ (or 100^0C) respectively.

Coefficients of thermal conductivity: $k_1 = 200\frac{W}{m \cdot K}$, $k_2 = 390\frac{W}{m \cdot K}$, $k_3 = 43\frac{W}{m \cdot K}$

Thus, $f = -7.6682 \times 10^5 \frac{W}{m^2}$, $t_2 = -R_1 f + t_1 = 299.9887\ K$, $t_3 = -(R_1 + R_2) f + t_1 = 319.6508\ K$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Finite element type	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
linear tetrahedron (10 nodes)	3230	3230	13300

Table 2.
Result «Temperature»

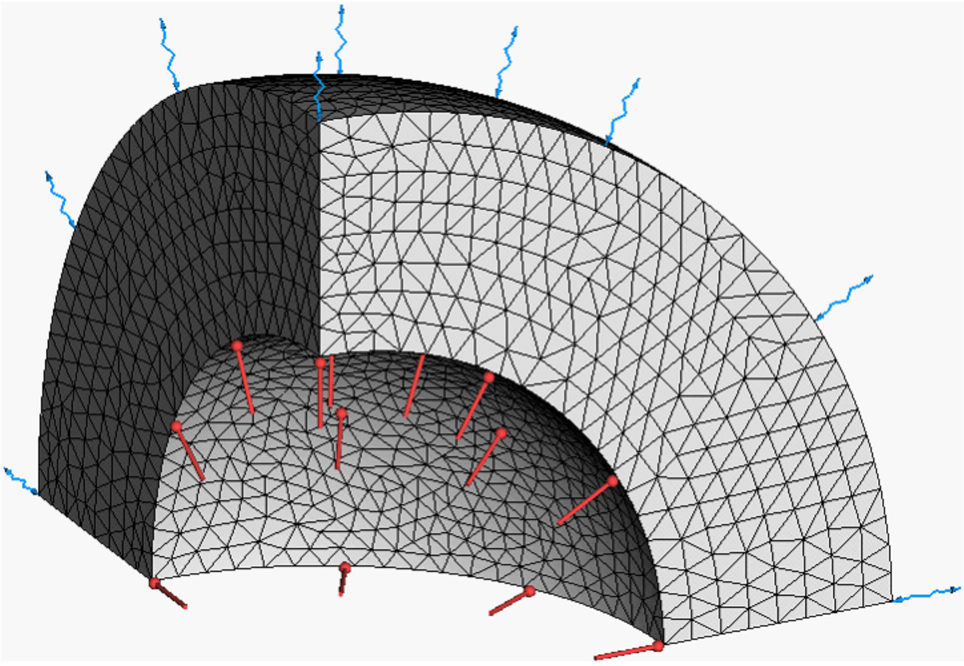
Surface S_{ij} of separation of layers *i* and *j*	**Numerical solution Temperature T^*, *K***	**Analytical solution Temperature *T*, *K***	**Error** $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
S_{12}	3.04589000E+02	3.04588978E+02	7.2188E-006
S_{23}	3.23015000E+02	3.23014753E+02	7.6569E-005

Table 3.
Result «Heat flux»

Numerical solution **Heat flux f^*, W/m^2**	**Analytical solution** **Heat flux f, W/m^2**	**Error** $\delta = \frac{\lvert f - f^* \rvert}{\lvert f \rvert} \times 100\%$
-7.18605000E+005	-7.18605212E+005	2.9461E-005

Flow of Heat in Sphere

Consider a hollow sphere with the internal radius r_1, external radius r_2, having constant coefficient of thermal conductivity λ. Internal surface of the sphere is held at temperature T_1. A heat exchange with the environment having temperature T_2 takes place on the external surface. Intensity of convective heat transfer is characterized by the heat transfer coefficient β.



Analytical solution of the problem has the form:

$$T = \frac{r_1 \cdot T_1 \cdot \left(\beta \cdot r_2^2 + r \cdot \left(1 - \beta \cdot r_2\right)\right) + \beta \cdot r_2^2 T_2 \cdot \left(r - r_1\right)}{r \cdot \left(\beta \cdot r_2^2 + r_1 \cdot \left(1 - \beta \cdot r_2\right)\right)}$$

For numerical calculation consider $\frac{1}{8}$ th part of the hollow sphere (see figure). On the lateral edges we specify symmetry conditions (the heat flux across the lateral edges is equal to 0).

Let us use the following data: internal radius of the sphere $r_1 = 150mm$, external radius of the sphere $r_2 = 250mm$. Coefficient of thermal conductivity λ of the material of the sphere is equal to $47\,\frac{W}{m \cdot K}$.

The temperature T_1 on the internal surface of the sphere is $373.15\ K$ (or $100\ ^0C$). The temperature of ambient environment T_2 is equal to $298.15\ K$ (or $25\ ^0C$), heat transfer coefficient β is equal to $100\,\frac{W}{m^2 \cdot {}^0C}$.

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

Mesh number	Finite element type	Number of main nodes	Number of nodes for problem calculation	Number of finite elements
1	quadratic tetrahedron (10 nodes)	4674	33357	21979
2	linear tetrahedron (10 nodes)	4674	4674	21979

Table 2.

Result «Temperature» at $r = \frac{3r_1 + r_2}{4} = 0.175\,м$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	Error $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
1	3.66138000E+002	3.66138033E+002	8.9883E-006
2	3.66166000E+002	3.66138033E+002	7.6384E-003

Table 3.

Result «Temperature» at $r = \frac{r_1 + r_2}{2} = 0.200\,м$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	Error $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
1	3.60879000E+002	3.60879058E+002	1.5959E-005
2	3.60901000E+002	3.60879058E+002	6.0803E-003

Table 4.

Result «Temperature» at $r = \frac{r_1 + 3r_2}{4} = 0.225\,м$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	Error $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
1	3.56788000E+002	3.56788743E+002	2.0837E-004
2	3.56824000E+002	3.56788743E+002	9.8816E-003

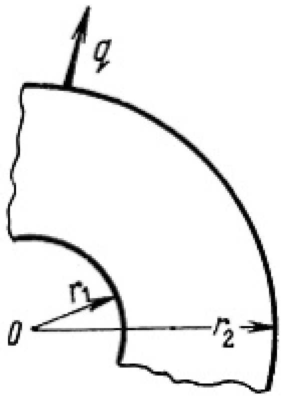
Table 5.

Result «Temperature» at $r = r_2 = 0.250\,m$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	Error $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
1	3.53482086E+002	3.53516492E+002	9.7325E-003
2	3.53476013E+002	3.53516492E+002	1.1450E-002

Thermal Conductivity of Cylindrical Wall

Consider infinitely long cylindrical wall (tube) with internal radius r_1, external radius r_2, having constant coefficient of thermal conductivity λ. Internal surface of the tube is held at temperature T_1. Inside the wall there are uniformly distributed sources of heat q_v. The heat generated in the wall is dissipated to the ambient environment through the external surface of the tube (see figure).



General solution of this problem has the form:

$T = C_1 \ln(r) + C_2 - \frac{q_v}{4\lambda} r^2$.

The constants C_1 and C_2 are determined from conditions prescribed on the internal ($r = r_1$)and external ($r = r_2$)surfaces of the tube: $T|_{r=r_1} = T_1$, $\lambda \frac{dT}{dr}\Big|_{r=r_2} = q$. Thus, $C_1 = \frac{r_2}{\lambda}\left(q + \frac{q_v r_2}{2}\right)$, $C_2 = T_1 - \frac{q r_2}{\lambda}\ln(r_1) - \frac{q_v}{4\lambda}\left(2r_2^2 \ln(r_1) - r_1^2\right)$.

Let us use the following data: internal radius of the tube $r_1 = 100mm$, external radius of the tube $r_2 = 250mm$, length of the tube $l = 1000mm$. Coefficient of the thermal conductivity λ, of material of the tube is equal to $43\frac{W}{m \cdot K}$.

Energy Q of sources of heat located inside the tube is equal to $4500\,W$. Since the sources of heat q_v are uniformly distributed over the volume of the tube, $q_v = \frac{Q}{\pi \cdot \left(r_2^2 - r_1^2\right) \cdot l} = 27283.705\frac{W}{m^3}$.

Specific heat flux on the external surface of the tube $q = -15000\frac{W}{m^2}$. Temperature T_1 on the internal surface of the tube is equal to $373.15\,K$ (or $100\,^0C$).

After carrying out calculation with the help of T-FLEX Analysis, the following results are obtained:

Table 1.
Parameters of finite element mesh

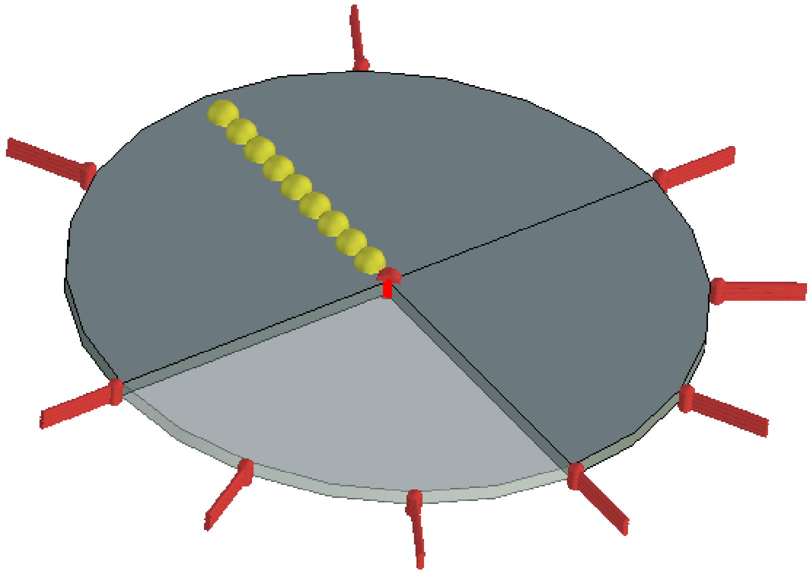
Mesh number	**Finite element type**	**Number of main nodes**	**Number of nodes for problem calculation**	**Number of finite elements**
1	quadratic tetrahedron (10 nodes)	13580	98620	66224
2	linear tetrahedron (10 nodes)	13580	13580	66224

Table 2.
Result «Temperature» at $r = r_2 = 0.250m$

Mesh number	**Numerical solution Temperature *T**, *K***	**Analytical solution Temperature *T*, *K***	**Error** $\delta = \frac{\lvert T - T^* \rvert}{\lvert T \rvert} \times 100\%$
1	3.03077942E+002	3.03081513E+002	1.1782E-003
2	3.03014069E+002	3.03081513E+002	2.2253E-002

Disk which is Heated along the Axis by a Distributed Heat Source and which has Constant Temperature on the External Cylindrical Surface

Let us consider a two-dimensional problem of a steady-state temperature distribution on the end-face cross-section of a disk. A heater of zero thickness (a line) with a power of P=100 W is situated on the axis of the disk, and on the periphery a constant temperature of 0 0C is held.
Parameters of a disk: metal disk with thickness d=5 mm, radius R=100 mm and thermal conductivity K=50 W/(m · 0C).



Numerical model with boundary conditions and sensors

Solution of this problem in which the source is regarded as distributed can be obtained by solving the problem for a point source of the heat. Let the origin of the coordinate system coincide with the center of the circular surface of a disk. The differential equation that has to be solved for a point source has the form:

$$\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)K=\rho\cdot\delta(P_0-P)$$

where ρ – density of a distributed heat source. For our case: ρ = P / d. Solution of this equation is a Green's function G (heat source function)

$$u=G(P,P_0)$$

$$G(P,P_0)=\frac{\rho}{2K\pi}\left(\ln\left(\frac{1}{r}\right)-\ln\left(\frac{R}{r_0\cdot r_1}\right)\right),\qquad P_0\neq 0$$

$$G(P,P_0)=\frac{\rho}{2K\pi}\left(\ln\frac{1}{r}-\ln\frac{1}{R}\right),\qquad \textit{otherwise}$$

$$P_0=P_0(x_0,y_0)$$

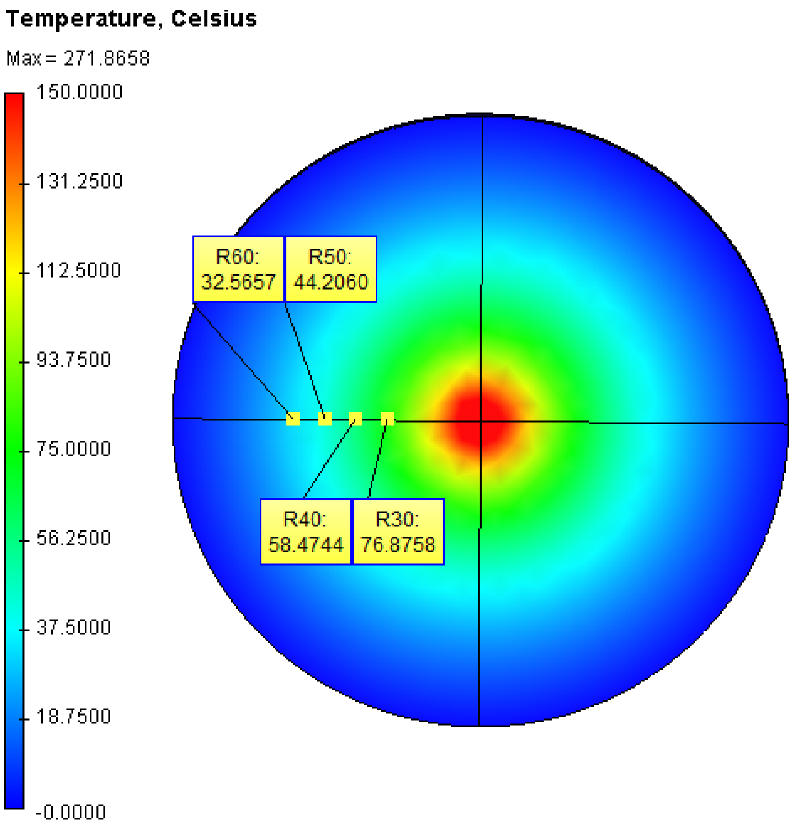
$$P=P(x,y)$$

$$r=\sqrt{(x-x_0)^2+(y-y_0)^2}$$

$$r_1=\sqrt{\left(x-x_0\frac{R^2}{r_0^2}\right)^2+\left(y-y_0\frac{R^2}{r_0^2}\right)^2}$$

$$r_0=\sqrt{x_0^{\,2}+y_0^{\,2}}$$

In this solution the zero point corresponds to the center of the disk. The Z-axis of the coordinate system, in which the solution is sought, is directed along the axis of the disk.



Temperature field on the end-face cross-section of the disk

Fig. 2 shows the distribution of the temperature field caused by a point source (quadratic element, relative size 0.02). Color scale was changed as compared to the default values: the upper limit was changed from 271.9 to 150 0C. This was done to better view the result away from the source of the heat in the center. Due to the symmetry of the range of values, the analytical solution of the problem with a point source coincides with the solution for a distributed heat source.
Let us compare analytical solution with the solution obtained from T-FLEX analysis. Analytical solution is calculated with the accuracy of up to 6 significant digits.

Table 1.
Parameters of a space mesh

Finite element type	Number of mesh nodes used in calculation	Number of elements in a mesh	Relative size
Tetrahedron, 4 nodes. Linear finite elements.	716	2123	0.04
Tetrahedron, 4 nodes. Quadratic finite element.	4247	2123	0.04

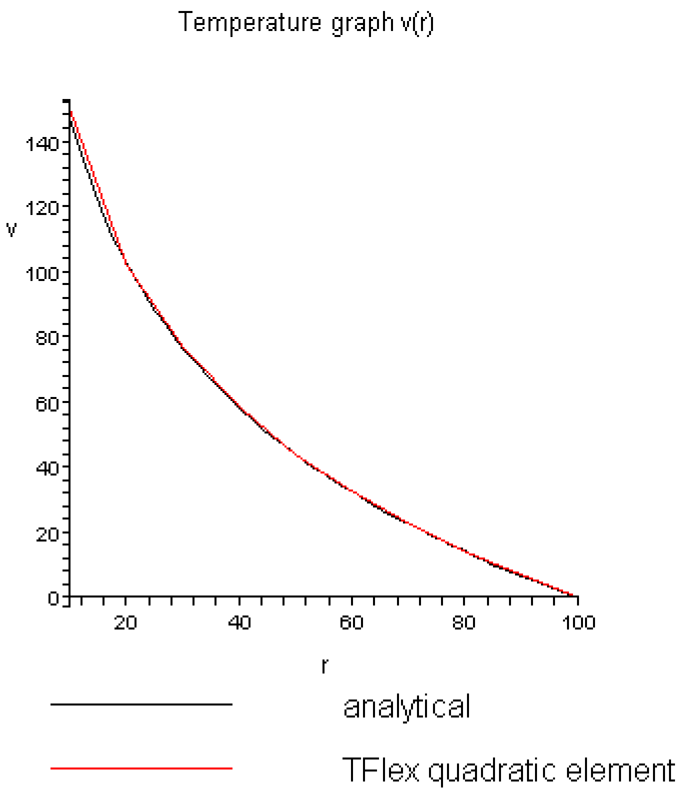
Table 2
Calculation using linear element

Distance from the center of the disk to the point of interest r, mm	30	40	50	60

Analytical solution, ${}^{0}C$	76.6472	58.3328	44.1271	32.5201
Numerical solution, ${}^{0}C$	76.8758	58.4744	44.2060	32.5657
Relative error, %	0.30	0.24	0.18	0.14

Table 3
Calculation using quadratic element

Distance from a center of the disk to the point of interest *r*, mm	30	40	50	60
Analytical solution, ${}^{0}C$	76.6472	58.3328	44.1271	32.5201
Numerical solution, ${}^{0}C$	76.6097	58.3095	44.1027	32.4983
Relative error, %	0.05	0.04	0.06	0.07



Dependence of steady-state temperature on radius (mm)

Conclusions:

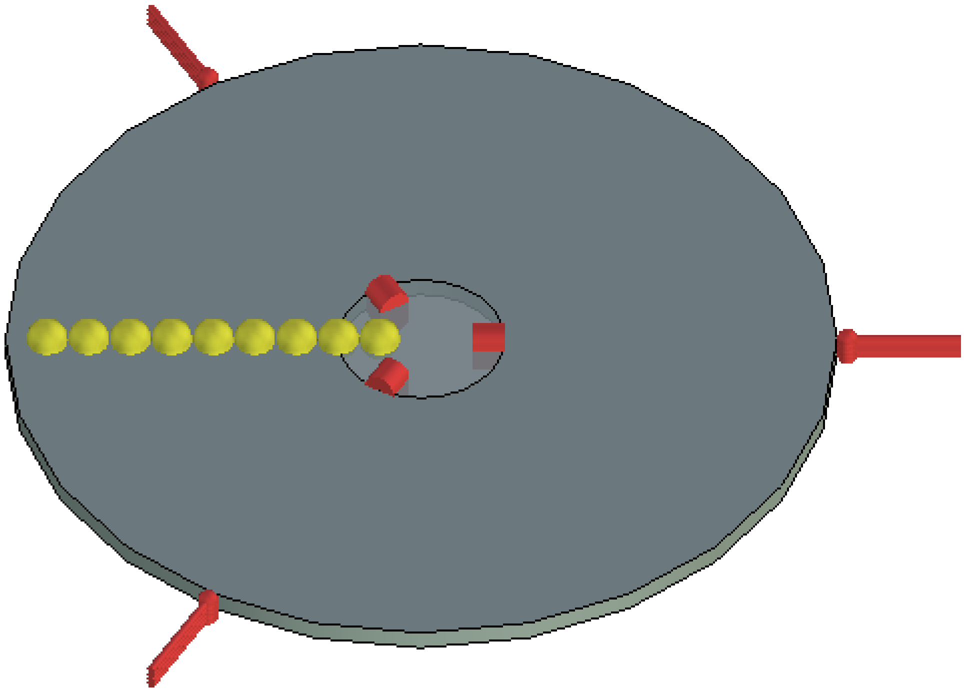
Relative error of the numerical solution as compared to the analytical solution did not exceed 1% when using linear element and 0.1% when using quadratic element on the edge of a disk. Accuracy of the calculation grows when moving away from the heat source because of the singularity of the analytical solution at the point of heat source application (at the center of a disk).

The plot of dependence of the temperature on the radius shows that analytical and numerical solutions practically coincide. This implies that distributions of temperature maximums and minimums are the same, and hence, calculation of thermal heat fluxes and power will be carried out with the same degree of accuracy.

Quadratic elements gave us one order of magnitude better convergence than linear elements. However, it required approximately 6 times more nodes/arguments for the solution.

Distributed Heat Source applied to the Cylindrical Surface inside the Disk

Let us consider a problem of steady-state distribution of the temperature at the end-face of the disk inside of which there is a heater in the form of a cylindrical surface with a radius rd=20 mm, power P=100 W; a constant temperature equal to 0 0C is maintained on the periphery.
Properties of a disk: metal disk of thickness d=5 mm, radius R=100 mm and thermal conductivity K=50 W/(m · 0C) – thermal conductivity of a disk inside and outside the cylindrical surface of the heater is the same.



Numerical model with boundary conditions and sensors

Solution of this problem, in which the source is considered as distributed, can be obtained by solving the problem for a point heat source (as in the previous example). Let the origin coincide with a center of a circular surface of a disk. The differential equation that is to be solved for a point source has the form:

$$\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)K = \rho\cdot\delta(P_0-P)$$

where ρ – density of distributed power. Solution of this equation is known. For our case: $\rho = P/(2\pi\cdot r_d\cdot d) = P/S$, where S – area of cylinder surface. By taking into account distribution of the power around the ring of the cylinder, let us express the temperature as a total power of all applied point sources by integrating the delta-function, on the right-hand side, over the point P0 . As a result, we obtain an equation:

$$\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)K = \rho\cdot\int_{\Omega'}\delta(P_0-P)d(P_0\in\Omega')$$

where Ω – is a set of points on the surface of a disk, Ω' – is a set of points that lie inside the surface of a disk and bounded by a ring of a cylinder on which the power is prescribed (see Fig. 1 – 2D flat ring inside the circle). Solution of this equation v is a linear combination (integral) of solutions for point sources:

$$v(P)=r_d\int_0^{2\pi}G_{II}(P,P_0(x(r_d,\theta),y(r_d,\theta)))d\theta$$

$$P_0(x(r_d,\theta),y(r_d,\theta))=P_0(r_d\cdot cos(\theta),r_d\cdot sin(\theta))$$

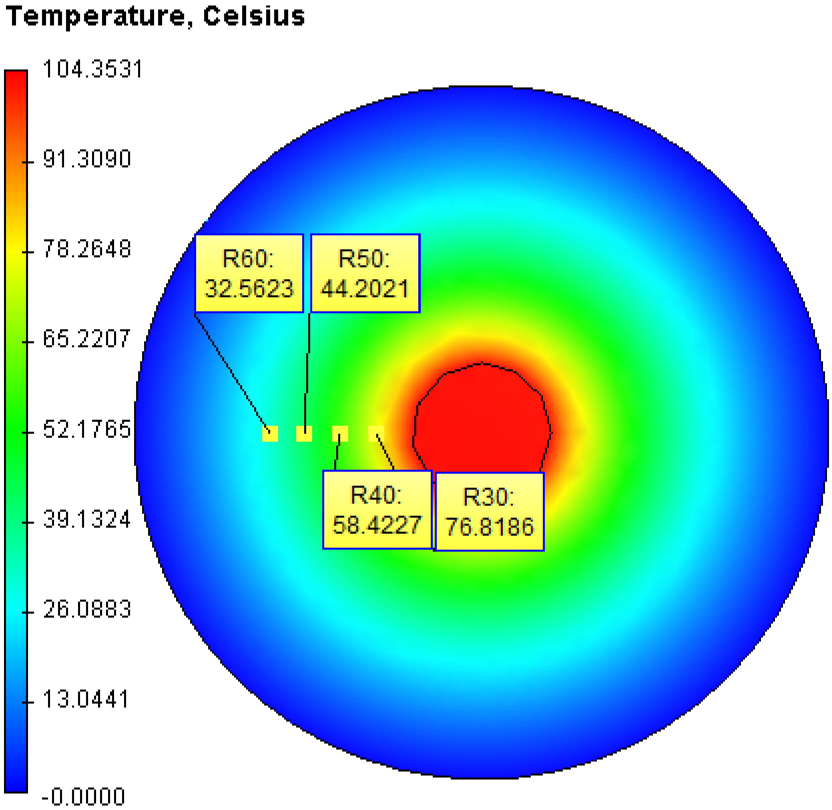
where GII – is a Green's function for a power defined by the formula $\rho = P/S$ for distribution on the surface. After integrating, we obtain:

$$v(P)=G_I(P,0), P\in\Omega\setminus\Omega'$$

$$v(P)=const, P\in\Omega'$$

$$0\leftrightarrow x=0, y=0$$

Here GI – is a Green's function for a power defined by the formula $\rho = P/d$ for a point source distributed on the edge. This implies that the magnitude of the temperature v will coincide with the temperature from a point source (located at the center of a circle) for points that lie outside with respect to the ring on which the distributed heat source is prescribed. In internal points, the temperature will take the value equal to some constant. Note that v – is a continuous function and that is why the value of the constant inside the embedded cylinder is always known.



Plot of a temperature field

Compare this plot with a field generated by the source distributed along the edge. We see that they coincide (the plot is made based on calculations with a quadratic element).

Let us compare a numerical solution obtained in T-FLEX analysis with the analytical solution.

Table 1

Parameters of finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of elements in a mesh	Relative size

Tetrahedron, 4 nodes. Linear finite element.	691	1998	0.06
Tetrahedron, 4 nodes. Quadratic finite element.	4060	1998	0.06

Table 2

Calculation with linear element

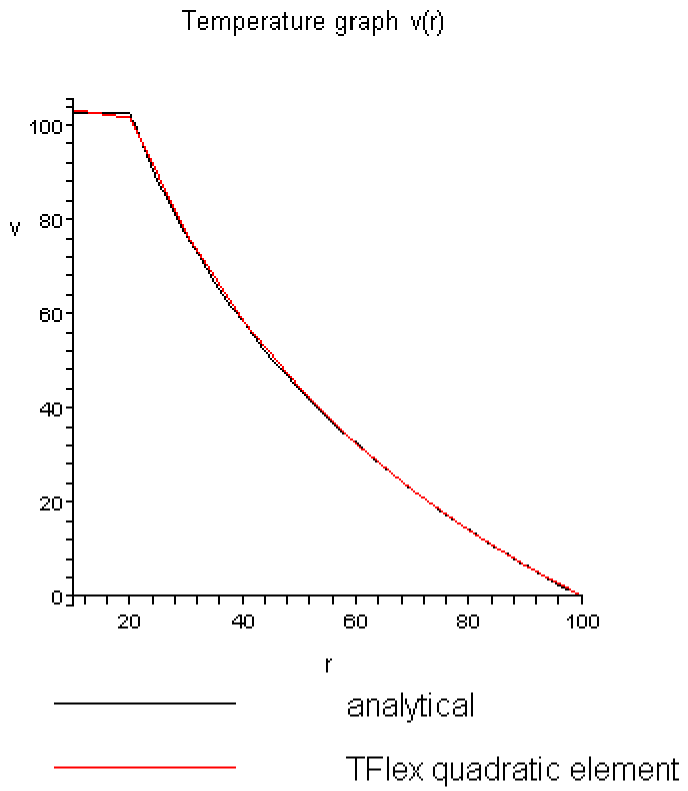
r – distance from the center of circular surface of a disk to the point of interest

Radial coordinate *r*, mm	30	40	50	60
Analytical solution, ^{0}C	76.6472	58.3328	44.1271	32.5201
Numerical solution, ^{0}C	77.2111	59.5685	44.6908	32.6473
Relative error, %	0.735708	2.118362	1.277446	0.391143

Table 3

Calculation with quadratic element

Radial coordinate *r*, мм	30	40	50	60
Analytical solution, ^{0}C	76.6472	58.3328	44.1271	32.5201
Numerical solution, ^{0}C	76.8186	58.4227	44.2021	32.5623
Relative error, %	0.223621	0.154115	0.169963	0.129766



Coordinate r is given in millimeters

Conclusions:

For a given problem we obtained a realistic picture of the temperature field. The relative error of the numerical solution compared to the analytical solution did not exceed 2% when using linear element and 0.2% when using quadratic element on the edge of a disk.

From the plot of dependence of temperature on the radial coordinate we can see that the analytical and numerical solution practically coincide. This implies that distributions of temperature maximums and minimums are the same, and hence, calculation of thermal heat fluxes and power will be carried out with the same degree of accuracy.

Quadratic elements turned out to be more accurate than linear elements but required 5.8 times more nodes.

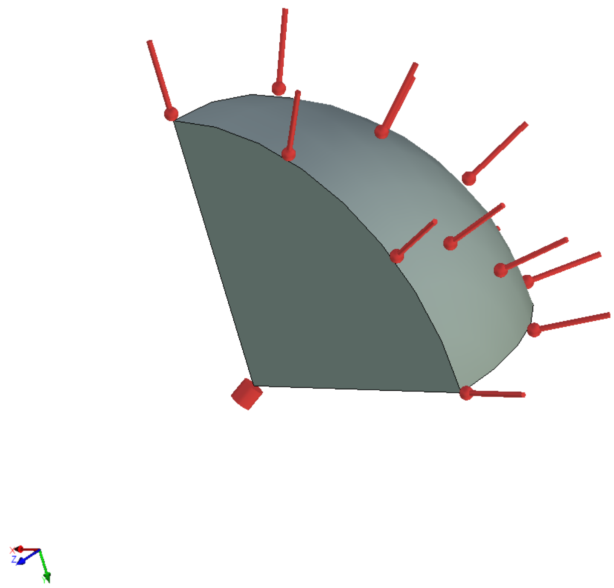
In addition, we note that modelling of a problem with distributed source by a point source was quite possible. We do not claim that it is always possible. However, for certain geometries this can be done. For example, in our case, the distributed heat source had a central symmetry.

Power of a Point Source inside the Sphere

Let us consider a problem of finding a temperature field inside the isotropic sphere with a point source in the center. The temperature is held constant on the spherical surface.

Radius of a sphere that bounds the body is R=0.1 m; the temperature on the surface of a sphere is t=20 0C. The center of the sphere coincides with the origin of the coordinate system. Let us assume that at the center of the sphere a heat point source of power P =500 W is prescribed. Coordinates of the point source are equal to P0=(0, 0, 0). Thermal conductivity is assigned the value equal to λ =1 W/(m·0C).

For modelling we consider a 1/8th part of the sphere. In the numerical model we apply the power of 500 W, and thus we divide it by 8, i.e., 500 / 8 = 62.5 W.



Numerical model with boundary conditions

For a given problem an exact solution at the point P=P(x, y, z) can be found as:

$$u(P)=\frac{\rho}{4\pi}\left(\frac{1}{r}-\frac{R}{r_0\cdot r_1}\right)+t, P_0\neq 0$$

$$u(P)=\frac{\rho}{4\pi}\left(\frac{1}{r}-\frac{1}{R}\right)+t,$$

$$P_0=P_0(x_0,y_0,z_0)$$

$$P=P(x,y,z)$$

$$r=\sqrt{(x-x_0)^2+(y-y_0)^2+(z-z_0)^2}$$

$$r_1=\sqrt{\left(x-x_0\frac{R^2}{r_0^2}\right)^2+\left(y-y_0\frac{R^2}{r_0^2}\right)^2+\left(z-z_0\frac{R^2}{r_0^2}\right)^2}$$

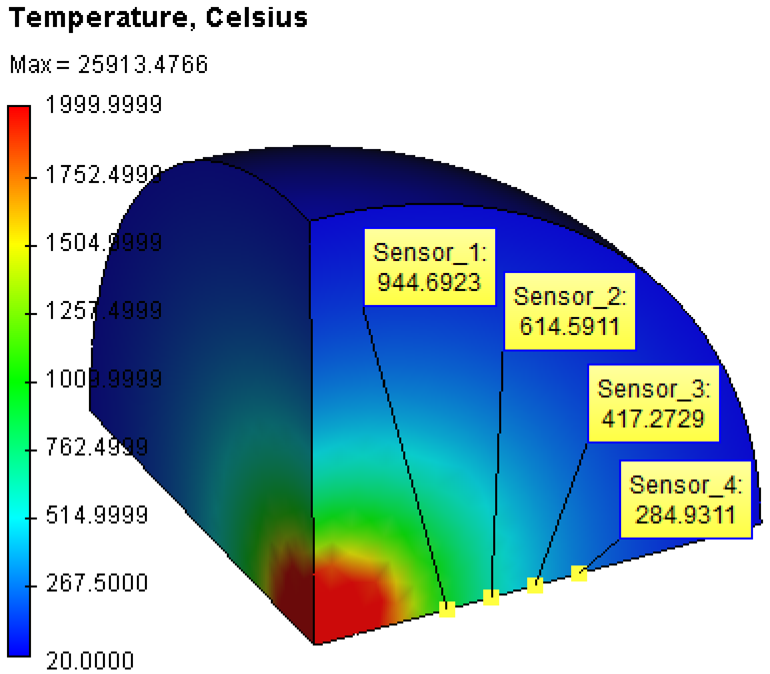
$$r_0=\sqrt{x_0^2+y_0^2+z_0^2}$$

The function u is a solution of the differential equation and can be written in the form:

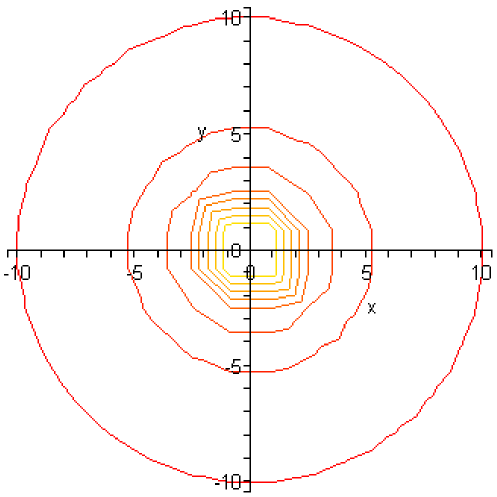
$$\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}=\rho\cdot\delta(P_0)$$

$$u(P)=t, P\in\partial\Omega$$

where δ is Dirac δ-function, Ω – domain of our sphere, $\partial\Omega$ – boundary of the sphere.



Temperature field inside the sphere obtained by the finite element analysis



Isothermal lines of the analytical solution plotted using Maple software

The temperature field obtained using T-FLEX Analysis is shown on «**Temperature field inside the sphere obtained by the finite element analysis**» picture. From "**Isothermal lines of the analytical solution plotted using Maple software**" picture we can see that the isothermal lines of the analytical solution in the diametrical plane of the sphere are not qualitatively different from those shown in «**Temperature field inside the sphere obtained by the finite element analysis**» picture.

Let us compare the numerical solution obtained in T-FLEX Analysis with the analytical solution. Analytical solution was obtained with the accuracy of up to six significant digits.

Table 1

Parameters of a finite element mesh

Finite element type	Number of mesh nodes	Number of finite elements	Relative size
Tetrahedron, 4 nodes. Linear finite element.	5244	23877	0.06
Tetrahedron, 4 nodes. Quadratic finite element.	5244	23877	0.06

Table 2

Calculation with linear element

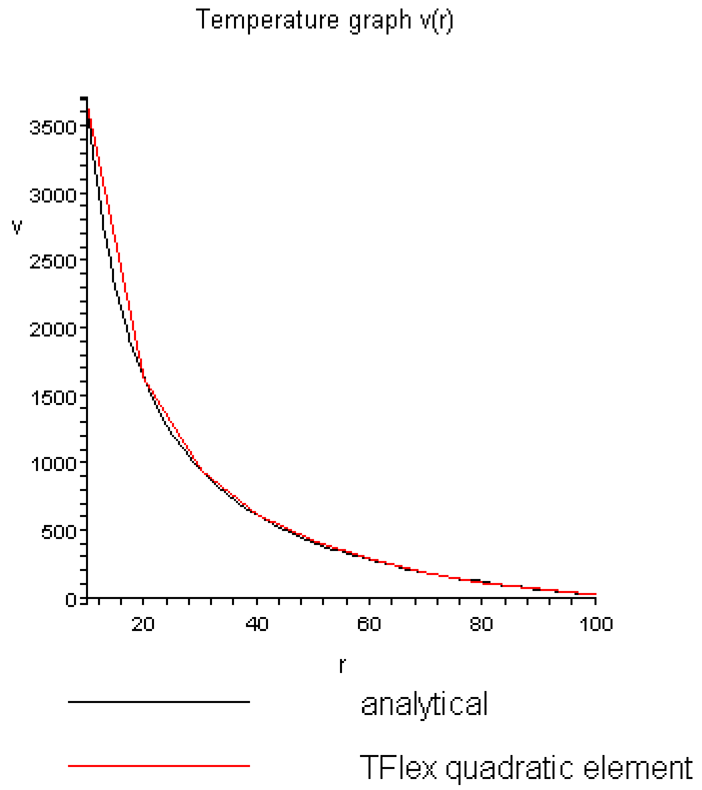
r – distance from the center of a sphere to a point of interest

Radial coordinate *r*, mm	30	40	50	60
Analytical solution, ^{0}C	948.403	616.831	417.887	285.258
Numerical solution, ^{0}C	970.479	631.200	424.348	290.662
Relative error, %	2.32770	2.32948	1.54611	1.894425

Table 3

Calculation with quadratic element

Radial coordinate *r*, mm	30	40	50	60
Analytical solution, ^0C	948.403	616.831	417.887	285.258
Numerical solution, ^0C	948.413	616.653	417.706	285.075
Relative error, %	0.001	0.03	0.04	0.06



Radial coordinate r is in millimeters

Conclusions:

The relative error of the numerical solution compared to the analytical solution did not exceed 2.3% for linear elements and 0.4% for quadratic elements.

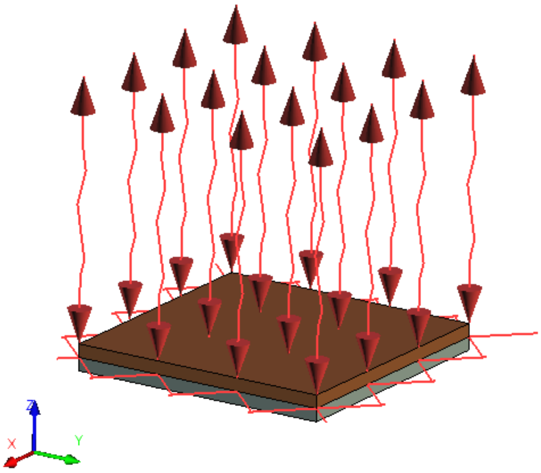
The method was proven to be effective for solving problems with point sources and singularities in the solution.

The plot of dependence of temperature on radius shows that analytical and numerical solutions practically coincided. This implies that the distributions of temperature maximums and minimums are the same, and hence, calculation of the heat fluxes and power will be carried out with the same degree of accuracy.

Quadratic elements were proven to be one order of magnitude more accurate than linear elements but the number of arguments was 6.75 higher. Furthermore note that near singular sources (concentrated on the edge or at the point), the relative error is higher than at the points more remote from the sources. This happens because the temperature at these sources is equal to infinity.

Temperature Field of a Thermal System of a Radiator and a Chip

Let us consider a system consisting of a radiator and a chip. The radiator has a thermal conductivity equal to Krad=390 W/(m·0C), and a chip has a conductivity equal to Kchip=50 W/(m · 0C). The chip is a source of heat which has a power of P=65 W. Along the entire interface between the radiator and the chip there is a thermal contact with the thermal resistance equal to R=2·10-4 m2 · 0C / W. The radiator diffuses the heat with the heat transfer coefficient equal to h=3000 W/(m2 · 0C) to the ambient environment that has a temperature of T0=20 0C. It is required to find the steady-state temperature distribution in the radiator and in the microcircuit. We assume the worst case scenario: the heat is dissipated only by the radiator, i.e., all other heat losses are ignored. The thickness of the radiator is b-a=1.5 mm. The thickness of the chip is a=1.5 mm. Both elements have rectangular shape with the volumes equal to Vrad=(b-a)·S and Vchip = a·S respectively, where S=900·10-6 m2 – area of the interface.



Numerical model with boundary conditions

Let us consider now the change in the field across the thickness of the system. Let z be the height measured from the foundation of the chip. Then the differential equation takes the form:

$$-K(z)\frac{\partial^2 u}{\partial z^2} = p\delta_{chip}(z), \quad 0<z<b$$

where δ chip – function of a heat source – microcircuit. In our case, this heat source – is a segment of length a. p – power distributed on this segment. If in the volume Vchip we apply the power P, then on the segment [0, a] p=P/Vchip.

K(z) – is a function of thermal conductivity that can be defined in the following way:

$$K(z)=\begin{cases} K_{rad}, z>a, \\ K_{chip}, z<a \end{cases}$$

For this equation, the boundary conditions have the form:

$$\begin{cases} -K_{rad}\dfrac{\partial u}{\partial z}(b)=h(u(b)-T_0), \\ \dfrac{\partial u}{\partial z}(0)=0 \end{cases}$$

For a point source of the heat, the solution of a problem with homogeneous boundary condition will take the form:

$$g(z,z_0)=-\frac{p}{2K(z)}\cdot|z-z_0|-\frac{p}{2K(z)}z-\frac{p}{2K(z)}z_0+C,$$

$$\frac{\partial g}{\partial z}(0,z_0)=0, z_0>0, z=0$$

Solution u for a heat source has the form:

$$u=\int_0^a g(z,z_0)dz_0=\begin{cases} a\cdot C_{rad}-\dfrac{p\cdot z\cdot a}{K_{rad}}, z>a, z\le b, \\ a\cdot C_{chip}-\dfrac{pz^2}{2K_{chip}}-\dfrac{pa^2}{2K_{chip}}, z\le a, z\ge 0 \end{cases}$$

Constants Crad and Cchip are determined from the following conditions:

$$\begin{cases} C_{chip} = C_{rad} - R \cdot p - p \cdot a\left(\frac{1}{K_{rad}} - \frac{1}{K_{chip}}\right) \\ C_{rad} = \frac{1}{a}\left(\frac{Pa}{h} + \frac{Pa^2}{K_{rad}} + T_0\right) \end{cases}$$

Let us locate the sensors of the temperature along the thickness; in 3D model they are located along the thickness of the plate. In the given points we will compare the numerical solution obtained using T-FLEX Analysis with the analytical solution.
Analytical solution of this problem is calculated with an accuracy of more than 6 significant digits.

Table 1

Parameters of finite element mesh

Finite element type	Number of mesh nodes used in calculation	Number of finite elements	Relative size
Tetrahedron, 4 nodes. Linear finite element	5957	3518	0.08
Tetrahedron, 6 nodes. Quadratic finite element	1030	3125	0.08

Table 2

Calculation with quadratic element

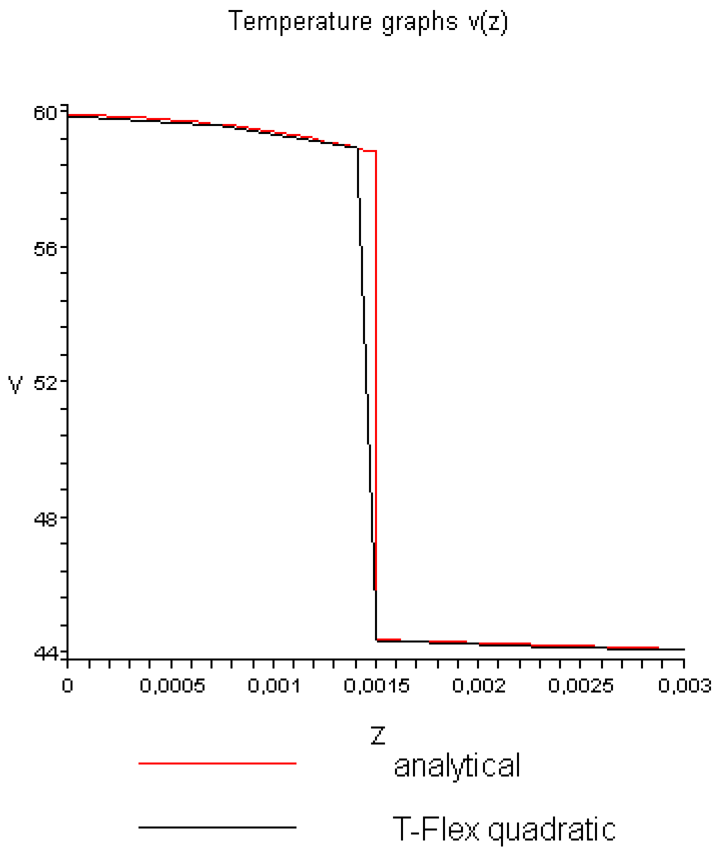
z – height from foundation of the chip in mm

Coordinate of a point z, mm	0	0.75	2.25	3
Analytical solution, ^{0}C	59.8796	59.6088	44.21296	44.07407
Numerical solution, ^{0}C	59.8796	59.6088	44.21297	44.07409
Relative error, %	3.34E-05	1.68E-05	2.26E-05	6.81E-05

Table 3

Calculation with linear element

Coordinate of a point z, mm	0	0.75	2.25	3
Analytical solution, ^{0}C	59.8796	59.6087	44.2129	44.0740
Numerical solution, ^{0}C	59.7295	59.3811	44.2086	44.0702
Relative error, %	0.25	0.38	0.0097	0.0086



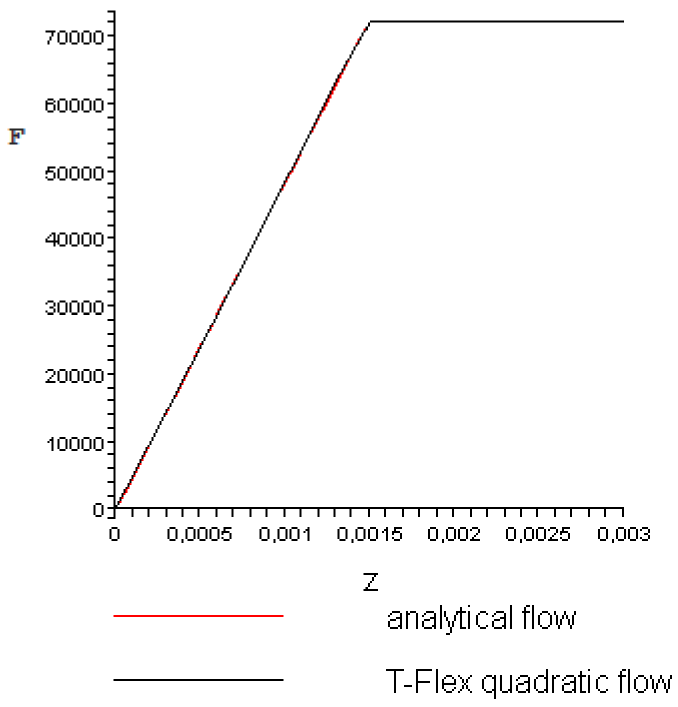
Coordinate z in meters, temperature in 0C.

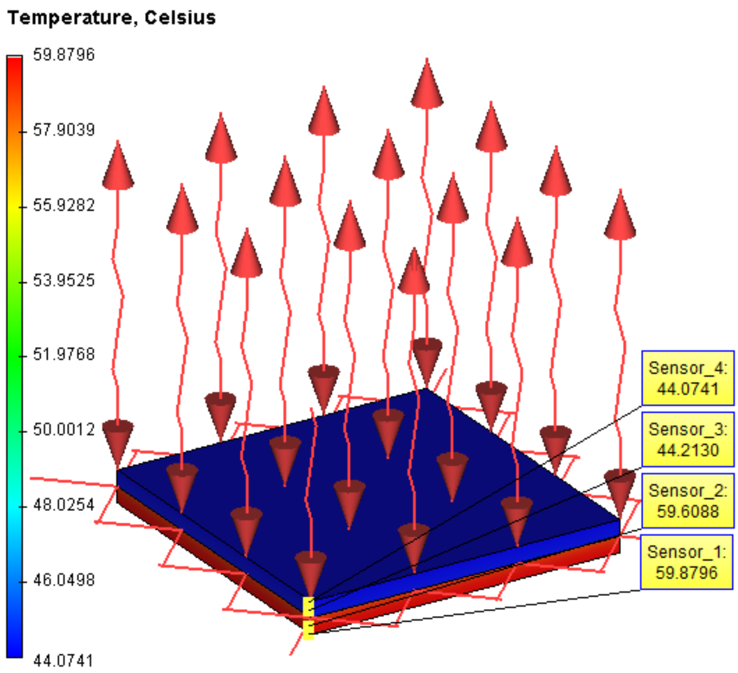
In addition, let us check the magnitude of the heat flux on the interface between the materials of the chip and radiator, and also on the upper boundary that diffuses the heat. The important fact for us is that the heat flux, unlike the temperature, is a continuous function. The expression for the heat flux is given below:

$$F = \begin{cases} -K_{rad}\dfrac{\partial u}{\partial z}, z > a, z \le b, \\ -K_{chip}\dfrac{\partial u}{\partial z}, z \le a, z \ge 0 \end{cases} = \begin{cases} p \cdot a, z > a, z \le b, \\ p \cdot z, z \le a, z > 0 \end{cases}$$

We can see from the analytical expression for the heat flux that inside the body of a radiator (to which the heat power is applied) the heat flux is equal to a constant value. On the boundary pa = pz for z=a, and hence the continuity of the heat flux is satisfied.

Heat flow graphs F(z), comparison to quadratic element





Results of numerical analysis with quadratic elements

Table 4

Calculation of heat flux F with linear element

Coordinate of a point z, mm	0.75	2.25	3
Analytical solution, W/m^2	36111.(1)*	72222.(2)	72222.(2)
Numerical solution, W/m^2	3.774E+4	7.167E+4	7.208E+4
Relative error, %	4.5	0.8	0.19

Table 5

Calculation of heat flux F with quadratic element

Coordinate of a point z, mm	0.75	2.25	3
Analytical solution, W/m^2	36111.(1)	72222.(2)	72222.(2)
Numerical solution, W/m^2	3.6111E+4	7.2222E+4	7.2222E+4
Relative error, %	$<10^{-5}$	$<10^{-4}$	$<10^{-4}$

**Digit enclosed in brackets signifies the period of a decimal, for example, 72222.(2)=72222.2222(2).*

Conclusions:

The relative error of the numerical solution compared to the analytical solution did not exceed 5% for linear elements and 10-4 for quadratic elements.

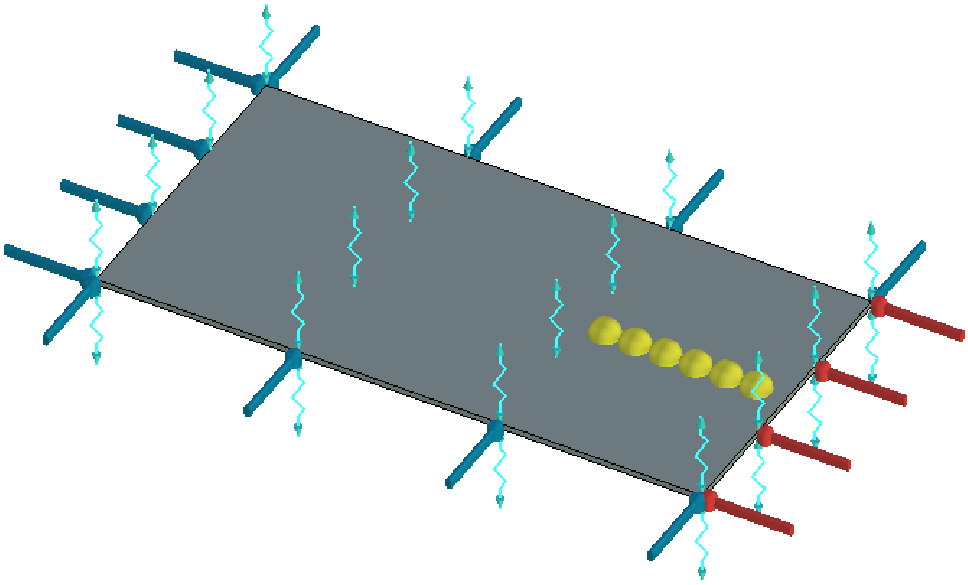
This problem was solved very accurately because solution was a piecewise continuous function with linear and quadratic parts.

We can see from Figure 2 that the plot showing the result obtained using quadratic elements coincided with the plot of the analytical solution. The difference between the analytical solution and the numerical solution obtained using linear elements was larger compared to the calculation performed using quadratic elements. This can be explained by the fact that the solution itself constitutes a quadratic function of temperature, and hence can be represented exactly using quadratic elements.

Orthotropic Graphite Plate under Steady State Temperature Regime

Let us consider now the problem of steady state temperature field distribution for a plate with orthotropic thermal conductivities of a crystalline material one of the edges of which is held at a constant temperature T1, and the remaining edges are at T2.

As an example, let us consider a rectangular graphite plate with the thermal conductivity of K1=278 W/(m · 0C) along the principle direction and K2=139 W/(m · 0C) in a perpendicular direction. The dimensions are: 200 mm x 100 mm. Parameters are a=200 mm, b=100 mm. The plate is elongated along the principle direction. Let us apply the temperature of T=80 0C to one of the edges of smaller size. In the steady-state regime the temperature is constant and does not depend on time. The temperature of 0 0C is applied to the remaining of the edges. Over the entire surface (from both sides) a heat exchange with the ambient environment takes place. The temperature of the ambient environment is 0 0C, the heat transfer coefficient is H=400 W/(m2 · 0C). The thickness of the plate is D=2 mm.



Numerical model with boundary conditions and sensors

For the given problem, the differential equation has the form:

$$K_1 \cdot \frac{\partial^2 u}{\partial x^2} + K_2 \cdot \frac{\partial^2 u}{\partial y^2} - \frac{2H}{D} u = 0$$

$$u(0,y) = 0$$

$$u(a,y) = 0$$

$$u(x,0) = t$$

$$u(b,y) = 0$$

This problem can be solved in the OXY-coordinate plane, on a rectangular domain [0,a]x[0,b], associated with the graphite plate.

Analytical solution of the problem is expressed in terms of Fourier series and has the form:

$$u(x,y) = \frac{2}{a} \sum_{n=1}^{\infty} \frac{\sin(n\pi\xi / a) \cdot sh\left((b-\zeta)\sqrt{k^2 + n^2\pi^2 / a^2}\right)}{sh\left(b\sqrt{k^2 + n^2\pi^2 / a^2}\right)} \int_0^a t \cdot \sin\left(\frac{n\pi x}{a}\right) dx$$

The isothermal lines of the obtained solution have the form shown on the Figure (obtained with Maple 9.5):

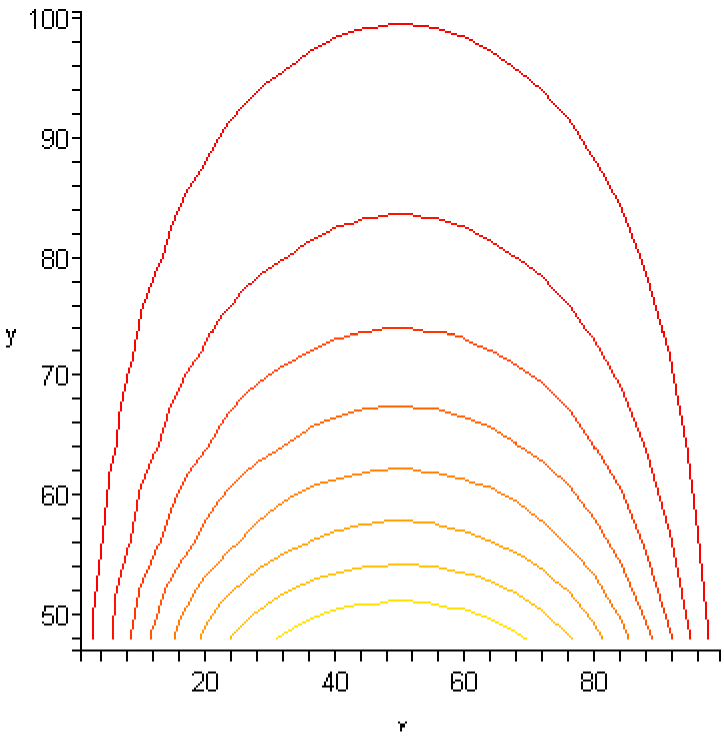


Fig. 2

We can see similar distribution of the temperature field obtained in T-FLEX (see Fig. 16.8-3).

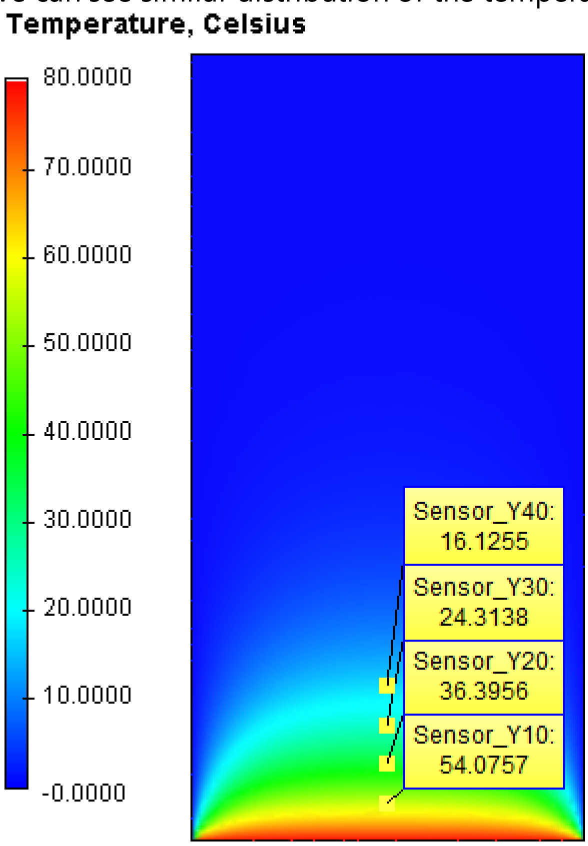
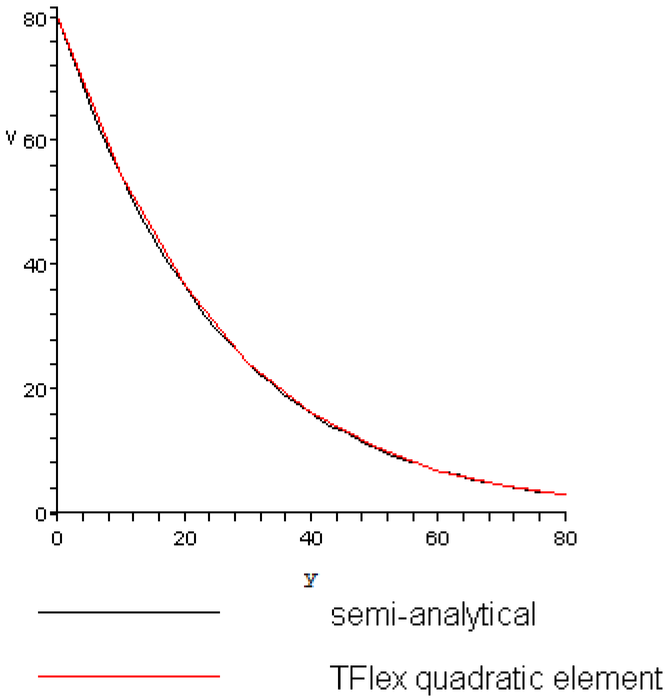


Fig. 16-8-3

We locate temperature sensors as shown on the Figure and we make a plot for them at x=50 mm, y=10,20,30,...60 mm. At these points we compare the plot with the analytical solution.

Temperature graphics v(y)



Coordinate y is in millimeters

Let us estimate this discrepancy numerically. Analytical solution was obtained with the accuracy of up to 6 digits.

Table 1

Parameters of a finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of finite elements	Relative error
Tetrahedron, 4 nodes. Linear finite element.	613	1756	0.05
Tetrahedron, 6 nodes. Quadratic finite element.	3591	1756	0.05

Table 2

Calculation with quadratic elements

y – height of a plate from a side with the applied temperature, x=50 mm.

Coordinates of a point y, mm	10	20	30	40
Analytical solution, ^{0}C	54.1929	36.4593	24.3452	16.1382
Numerical solution, ^{0}C	54.0757	36.3956	24.3138	16.1255

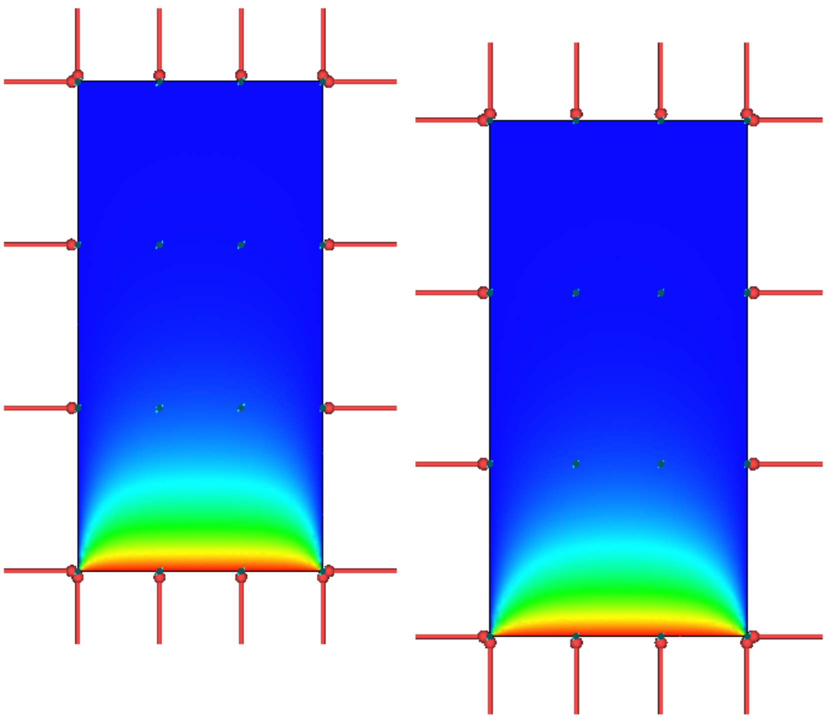
Relative error, %	0.22	0.19	0.14	0.07

Table 3

Calculation with linear elements

Coordinate of a point y, mm	10	20	30	40
Analytical solution, ^{0}C	54.1929	36.4593	24.3452	16.1382
Numerical solution, ^{0}C	54.6211	36.5994	24.2631	16.1600
Relative error, %	0.79	0.38	0.34	0.13

Notice that the points that are closer to the boundary do not have to be selected (200 terms in a sum is acceptable for 8-10 mm), since Fourier series will converge at these points with large oscillations. To obtain better accuracy, it is necessary to increase the number of terms in a partial sum of a series.
To visualize the effect of orthotropy, let us change, in T-FLEX, the coefficient of thermal conductivity along the OX-axis to K2=70 W/(m · 0C). Then the temperature distribution has the form:



*K2=70 W/(m * 0C) on the left and K2=139 W/(m * 0C) on the right*

We can see from picture that the isothermal lines shown on the left side will increase the slope if the thermal conductivity along the OX-axis is decreased.

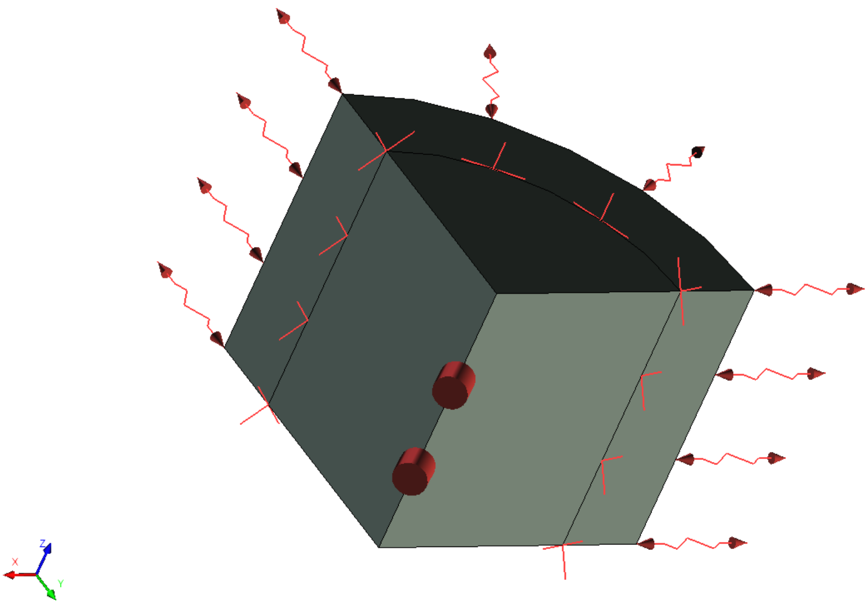
Conclusions:

The present method was proven to be effective when solving the problems with anisotropic distribution of temperatures. The existence of orthotropic properties did not affect the computational efficiency of the method.

The plot of dependence of the temperature on radius shows that the analytical and numerical solutions coincided from a practical point of view. This implies that distributions of temperature maximums and minimums are identical, and hence, the calculation of heat fluxes and power will be carried out with the same degree of accuracy.
The relative error of linear elements on average twice larger than the error of quadratic elements. However, the number of nodes for quadratic elements exceeded more than 5 times the number of nodes for linear elements.

Temperature Field of a System of Two Embedded Cylinders with a Thermal Resistance at the Interface

Let us consider a problem of distribution of temperature field of a system of two embedded cylindrical bodies with a thermal resistance on the contact surface. The radius of internal body is equal to r2=5 cm, and the radius of external body is equal to r1=7 cm. Both cylinders are located on the same level and have the height equal to D=7 cm. Zero heat flux is prescribed on the circular surfaces of these bodies. On the lateral surface of the external cylinder there is a heat exchange with an ambient environment with the heat transfer coefficient equal to H=120 W/(m2 · 0C) and the temperature of the ambient environment equal to T0=20 0C. Along the central segment of the cylinder along the entire height we prescribe the heat power P=200 W. On the interface between two cylinders we prescribe the thermal resistance of magnitude R=0.01 0C·m2/W. The thermal conductivity of the internal cylinder is equal to K2=50 W/(m · 0C), and external cylinder K1=50 W/(m · 0C).
We consider only one quarter of the entire cylindrical system of bodies. The symmetry condition can be satisfied by assuming zero heat flux on the rectangular boundaries of the structure. The power on the central segment must then be divided by four (distributed between two bodies). P1/4=50 W = P/4. An example of such construction is shown below.



Numerical model with boundary conditions

The differential equation for a point source has the form:

$$\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}K(x,y)\right)+\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial y}K(x,y)\right)=\rho\cdot\delta(x,y)$$

where ρ – density of distributed power. For our case: ρ = P / D. K – is a function of thermal conductivity of the material of the cylinders. K(x,y)=K1 if a point (x,y) belongs to the external cylinder, K(x,y)=K2 if a point belongs to the internal cylinder, δ – is a function of the heat source (called Dirac function in the literature). Solution of this differential equation is a Green's function G (heat source function). Equation is represented in the coordinates (x,y). But the solution will actually depend only on one variable – radial coordinate (distance from the segment with the applied power), and will have the form:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}K(r)\right)=\rho\cdot\delta(r)$$

$$K(r)=\begin{cases}K_1, r_2<r\le r_1\\ K_2, 0\le r\le r_2\end{cases}$$

The boundary condition for this problem corresponds to the heat transfer by convection

$$-K_1\frac{\partial u}{\partial r}(r_1)=H(u(r_1)-T_0)$$

Solution for this problem has the form:

$$u(r)=\begin{cases}\frac{\rho}{2\pi K_2}\ln|r|+C_2, 0\le r\le r_2\\ \frac{\rho}{2\pi K_1}\ln|r|+C_1, r_2<r\le r_1\end{cases}$$

where the constants C1 and C2 are determined from the condition of the temperature jump on the interface between two bodies and the boundary condition. Expressions that determine C1 and C2 have the form:

$$C_1=\frac{1}{H}\left(\frac{\rho}{2\pi r_1}+H\cdot T_0+H\rho\frac{\ln(r_1)}{2\pi K_1}\right)$$

$$C_2=C_1+R\frac{\rho}{2\pi r_2}+\frac{\rho\ln(r_2)}{2\pi}\left(\frac{1}{K_2}-\frac{1}{K_1}\right)$$

Condition of the temperature jump has the form:

$$R=\frac{\left|\lim\limits_{r\to r_2+}u(r)-\lim\limits_{r\to r_2-}u(r)\right|}{F}=\frac{|u(r_2+)-u(r_2-)|}{F}$$

where F – thermal heat flux on the boundary can be found by the formula

$$F=\lim_{r\to r_2-}-\frac{\partial u}{\partial r}K_2=\lim_{r\to r_2+}-\frac{\partial u}{\partial r}K_1$$

Let us compare the solution of T-FLEX Analysis with analytical solution of the problem.

Parameters of the finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of mesh elements	Relative size
Tetrahedron, 4 nodes. Linear finite element.	2730	12060	0.09
Tetrahedron, 6 nodes. Quadratic finite element.	18520	12427	0.09

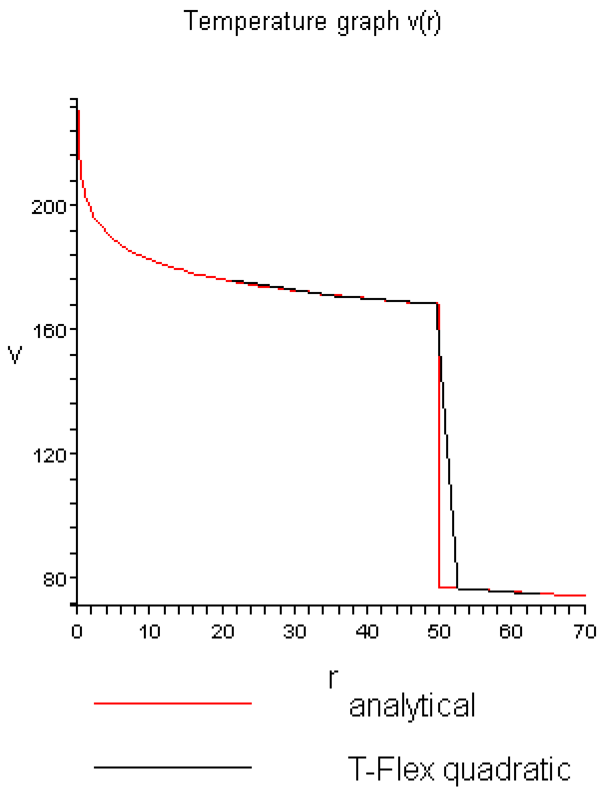
Calculation with linear element

Coordinate of a point $r=\|x\|\sqrt{2}$, mm	28.28427	42.42640	56.56854

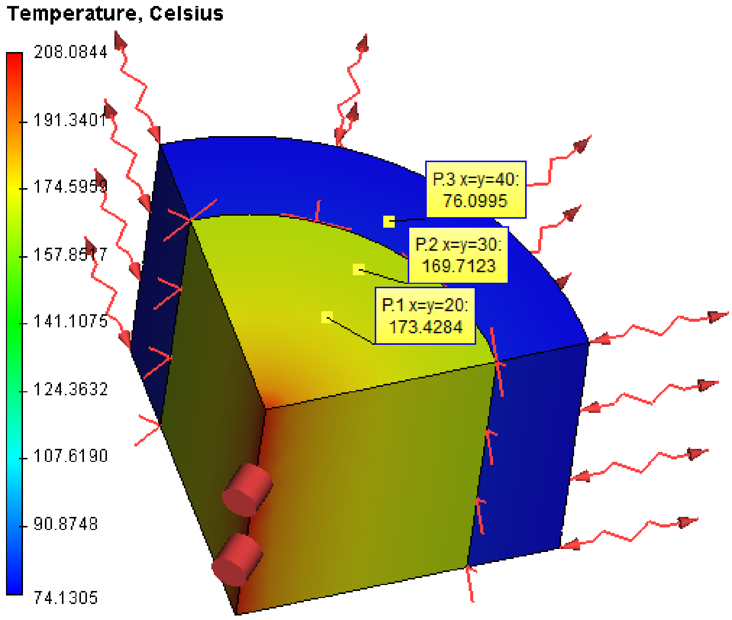
Coordinate $x=y$ of a point in the plane of a cylinder	20	30	40
Analytical solution, ^0C	173.3214	169.6338	76.07186
Numerical solution, ^0C	173.428	169.712	76.0995
Relative error, %	0.061504	0.046099	0.036334

Calculation with quadratic element

Coordinate of a point $r=\|x\|\sqrt{2}$, mm	28.28427	42.42640	56.56854
Coordinate $x=y$ of a point in the plane of a cylinder	20	30	40
Analytical solution, ^0C	173.3214	169.6338	76.07186
Numerical solution, ^0C	173.363	169.677	76.0716
Relative error, %	0.024002	0.025467	0.000342



Units along the axes: r is in millimeters, v in 0C



Temperature field

Notice that unlike the temperature, the heat flux does not have a jump on the interface between the materials, i.e., is a continuous function of space. It can be evaluated as:

$$F = -K\frac{\partial u}{\partial r} = \frac{1}{2\pi r}$$

Now let us compute a heat flux in T-FLEX Analysis.

Table 1

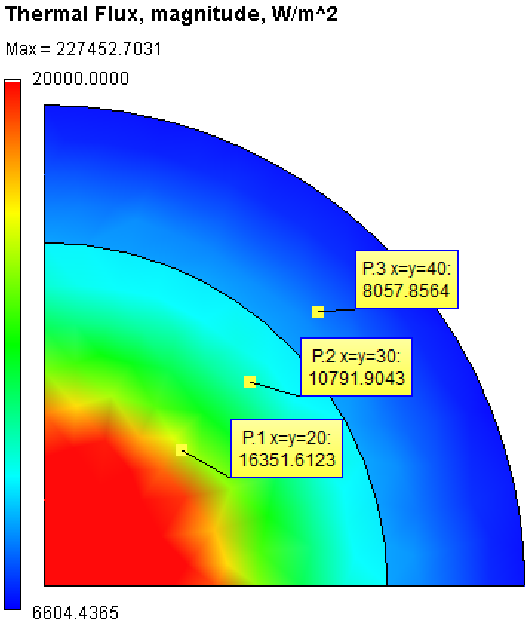
Calculation with linear elements

Coordinate of a point $r=\lvert x\rvert\sqrt{2}$, mm	28.28427	42.42640	56.56854
Coordinate x=y of a point in the plane of a cylinder.	20	30	40
Analytical solution, W/m^2	16077.0	10718.1	8038.53
Numerical solution, W/m^2	16351.6	10791.9	8057.86
Relative error, %	1.70803	0.688554	0.240466

Table 2

Calculation with quadratic elements

Coordinate of a point $r=\lvert x\rvert\sqrt{2}$, mm	28.28427	42.42640	56.56854
Coordinate x=y of a point in a plane of a cylinder	20	30	40
Analytical solution, W/m^2	16077.0	10718.1	8038.53
Numerical solution, W/m^2	16117.4	10733.0	8048.19
Relative error, %	0.251290	0.139017	0.120171



Picture of a heat flux

Conclusions:

The relative error of the numerical solution compared to the analytical solution did not exceed 1.7% for linear elements and 0.25% for quadratic elements.

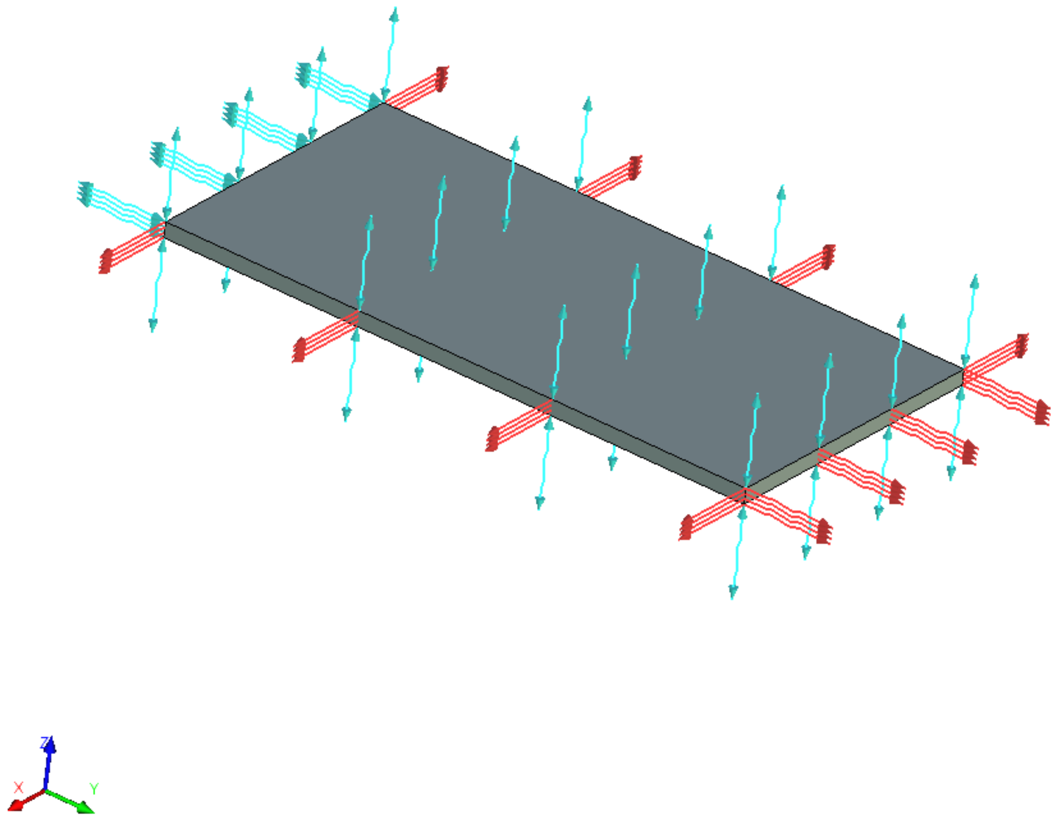
We confirmed the continuity of the thermal heat flux, physically important for the thermal systems.

At the neighbourhood of the concentrated source (point or segment) of the heat power, the errors of the linear and quadratic elements do not differ significantly. This is related to the fact that the temperature for such heat sources is unbounded. At a certain distance from them, the quadratic elements show superior accuracy compared to the linear elements.

Although the unbounded value of the temperature cannot represent a real physical model, it allows us to examine different heat sources, for example, thin wires or objects sufficiently remote from the domain of interest.

Heat Flux and Convection for Isotropic Plate

Let us consider a steel plate b=200 X a=100 mm of thickness D=5 mm with the thermal conductivity of K=50 W/(m 0C). Let us show that the steady-state heat transfer problem can also be solved for the case when on the boundary the temperature is not held constant (for correct modelling the temperature must be known in advance). Let us consider two types of loads: heat flux (equal to zero for some boundaries) and heat transfer by convection. On the surface of the plate (from both sides) we prescribe the heat exchange with ambient environment with the heat transfer coefficient C=40 W/(m2 · 0C) and the temperature of the ambient environment equal to zero. On the fourth side (of length 100 mm) we prescribe a convective heat transfer with the coefficient H=20 W/(m2 · 0C) and the temperature of the ambient environment equal to u0=10 0C.



Numerical model with boundary conditions

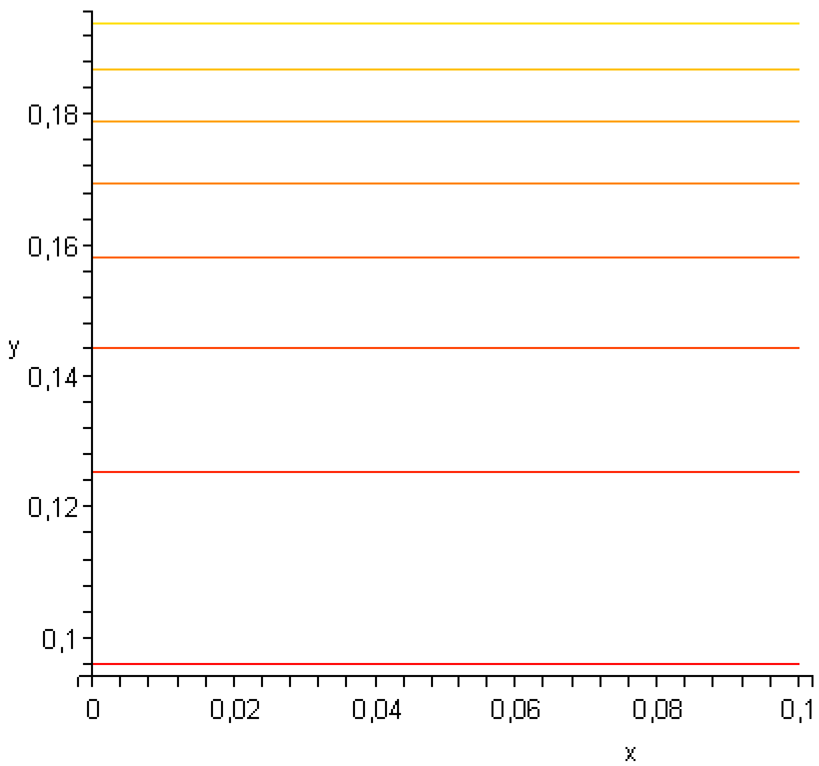
Let us locate the origin of the coordinate system at the corner of the plate, let the OY-axis be directed along the longer side of the plate, and convection will then be applied to the edge y=b. Then we will infer that two out of three edges, on which a zero heat flux is prescribed, correspond to the values x=0 and x=a. This means that the solution that we obtain must not depend on the coordinate x. Solution will have the form:

$$u(x,y) = V \cdot \left(e^{y\sqrt{L}} + e^{-y\sqrt{L}}\right),$$

$$L = C/(K \cdot D),$$

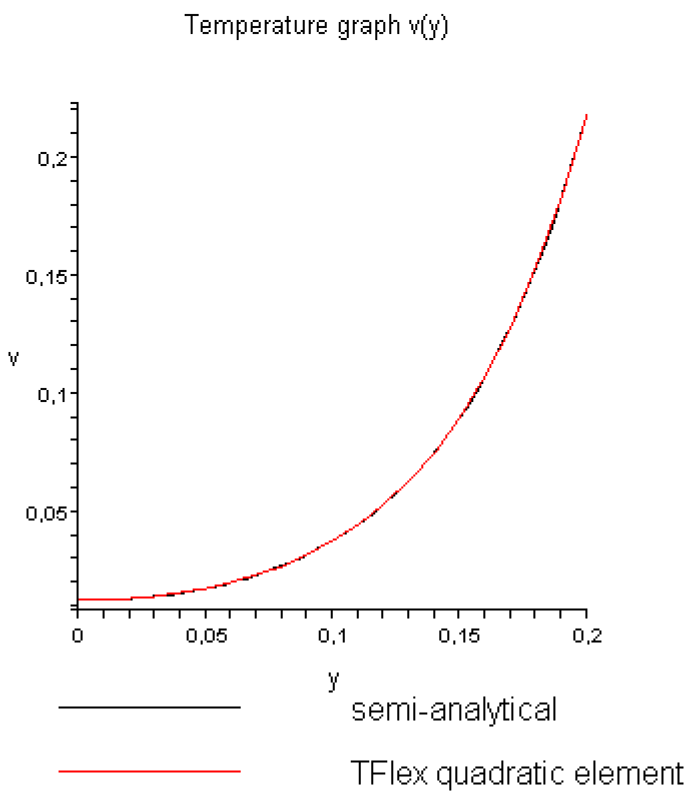
$$V = \frac{u_0}{\left(e^{b\sqrt{L}} - e^{-b\sqrt{L}}\right) - \frac{K}{H} \cdot \sqrt{L}\left(e^{b\sqrt{L}} + e^{-b\sqrt{L}}\right)}$$

Below we show the isothermal lines for the solution obtained using the Maple software.
Convection is applied for the coordinate Y=b.

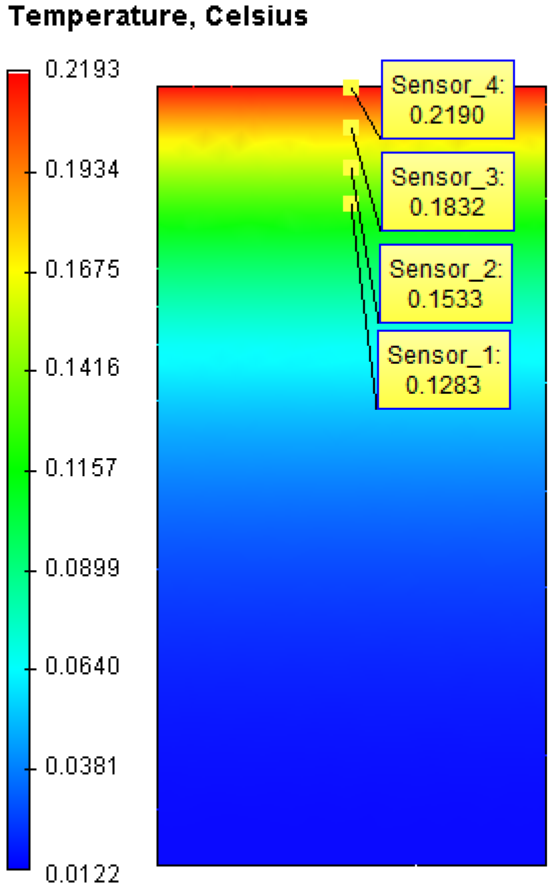


Isothermal lines for the solution obtained using the Maple software. Convection is applied to the edge y=b. When y=0 the thermal heat flux is zero

Comparison with the temperature field obtained using T-FLEX Analysis is shown below.



Units along the y-axis are in meters



Let us compare the numerical solution and the solution obtained in the system T-FLEX Analysis.

Table 1

Parameters of a finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of elements	Relative size
Tetrahedron, 4 nodes. Linear finite element.	742	1972	0.05
Tetrahedron, 6 nodes. Quadratic finite element.	4195	1972	0.05

Table 2

Calculation with quadratic element

y – height of the plate from the side with the applied temperature

Coordinate of a point y, mm	200	190	180	170
Analytical solution, ^{0}C	0.219050	0.183231	0.153292	0.128271
Numerical solution, ^{0}C	0.218979	0.183186	0.153265	0.128248
Relative error, %	0.032412	0.024559	0.017613	0.017931

Table 3

Calculation with linear element

Coordinate of a point y, mm	200	190	180	170
Analytical solution, ^{0}C	0.219050	0.183231	0.153292	0.128271
Numerical solution, ^{0}C	0.218785	0.183406	0.153406	0.128254
Relative error, %	0.120976	0.095507	0.074367	0.013253

Conclusions:

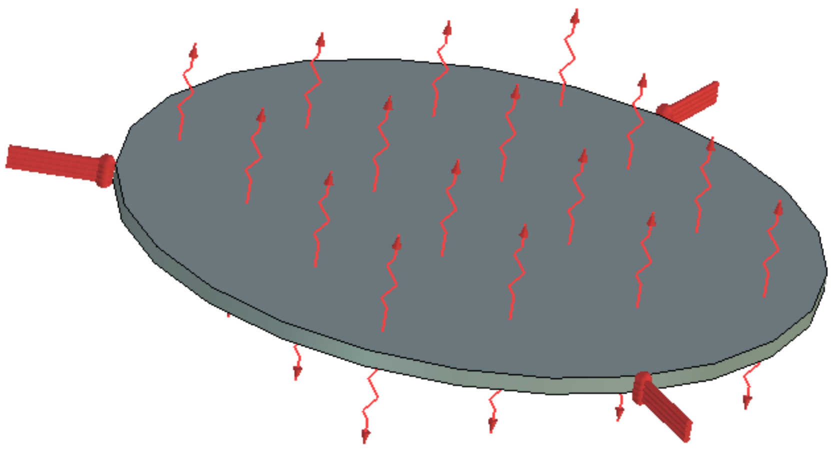
The relative error of the numerical solution compared to the analytical solution did not exceed 0.12% for linear element and 0.03% for quadratic element. This implies that for the given problem we obtained a solution with up to 3 significant digits of accuracy. The problem was solved almost exactly with minimum computational cost.

Quadratic elements showed the relative error twice smaller than the linear elements, however, they required almost 5.8 times more nodes.

Thermal Heat Flux in an Isotropic Disk

Let us consider now the problem of heating of a circular plate with the sources of heat distributed over the surface. Let us prescribe the heat flux from both sides. Around the plate along the edges we will prescribe the constant temperature.

As an example we consider a thin plate with the thermal conductivity K=75 W/(m · 0C) . The radius of the plate is R=100 mm, thickness D=5 mm. The value of the density of the heat flux is F=60 W/m2. Temperature of the edges around the plate is T=20 0C.



Numerical model with boundary conditions

Picture shows the T-FLEX model of such a disk. At a distance from the center of the disk r=0, 10, 20, ..., 60 mm on the surface we install the sensors of the temperature and compare the results.

The equation that must be solved for a surface of a disk has the following form:

$$\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)K=\frac{2F}{D}$$

At the right side of the equation we have the volumetric density of the heat source (equivalent to a surface density for a plane problem).

Boundary conditions:

$$u(P)=T, P\in\partial\Omega\setminus(U_+\cup U_-)$$

$$K\frac{\partial u}{\partial z}(P)=\mathrm{F}, \mathrm{P}\in U_+$$

$$K\frac{\partial u}{\partial z}(P)=-\mathrm{F}, \mathrm{P}\in U_-$$

where U- and U+ - are respectively lower and upper sides of the plate. Analytical solution can be expressed in terms of the function of the heat source G(P,P0).

$$G(P,P_0)=\frac{1}{2\pi}\left(\ln\left(\frac{1}{r}\right)-\ln\left(\frac{R}{r_0\cdot r_1}\right)\right), \mathrm{P}_0\neq 0$$

$$G(P,P_0)=\frac{1}{2\pi}\left(\ln\frac{1}{r}-\ln\frac{1}{R}\right), \mathrm{P}_0=0$$

$$\mathrm{P}_0=\mathrm{P}_0(\mathrm{x}_0, y_0)$$

$$\mathrm{P}=\mathrm{P}(\mathrm{x}, \mathrm{y})$$

$$r=\sqrt{(x-\mathrm{x}_0)^2+(y-y_0)^2}$$

$$r_1=\sqrt{\left(x-\mathrm{x}_0\frac{R^2}{r_0^2}\right)^2+\left(y-y_0\frac{R^2}{r_0^2}\right)^2}$$

$$r_0=\sqrt{\mathrm{x}_0{}^2+y_0{}^2}$$

This function is a solution of the Laplace equation with the singular right-hand side (at the right side we have Dirac function) of unit power.

$$\frac{\partial^2 G}{\partial x^2}+\frac{\partial^2 G}{\partial y^2}=\delta(P,P_0)$$

$$G(P,P_0)=0, P\in\partial\Omega\setminus(\tilde{A}_+\cup\tilde{A}_-)$$

$$G(P,P_0)=G(P_0,P), \forall P,P_0\in\Omega$$

Solution of Poisson's equation can be expressed in terms of the source function as:

$$u(P)=\int_\Omega f(P_0)G(P,P_0)dP_0,$$

$$\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}=f, f=\frac{2F}{D\cdot K}=const$$

Analytical solution was calculated in the Maple 9.5 system by the method of numerical integration _d01ajc (Gauss integration using 10 points and Croncord integration using 21 points). We used 6 significant digits of accuracy in the analytical solution to compare the results.

Table 1

Parameters of finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of elements	Relative error
Tetrahedron, 4 nodes. Linear finite element.	322	914	0.09
Tetrahedron, 6 nodes. Quadratic finite element.	1876	914	0.09

Table 2

Calculation with quadratic element

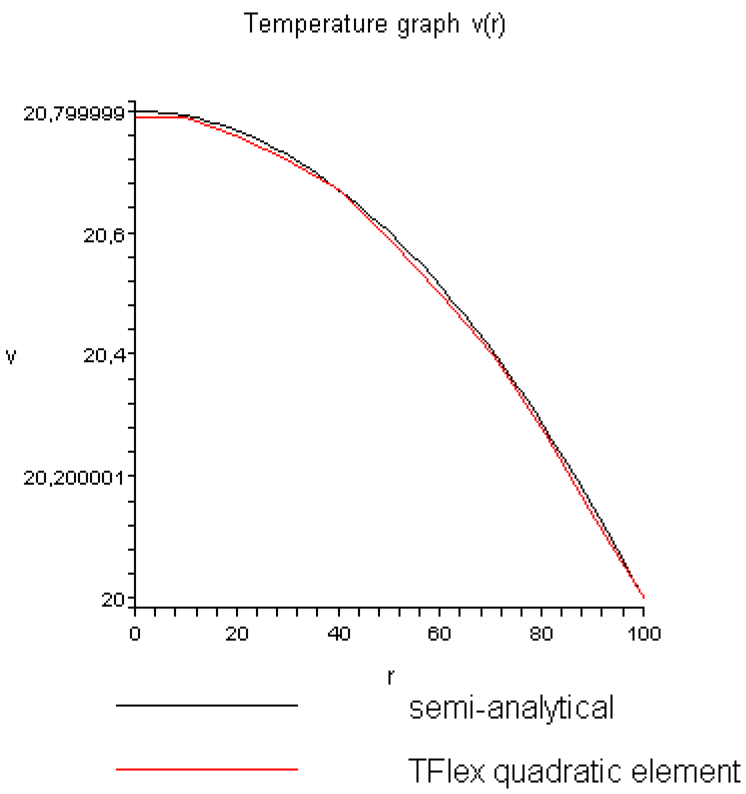
r – radial coordinate at which the sensor is located

r, mm	0	20	40	60
Analytical solution, ^0C	20.8000	20.7610	20.6720	20.5120
Numerical solution, ^0C	20.7973	20.7653	20.6692	20.5092
Relative error, %	0.012980	0.020711	0.013544	0.013651

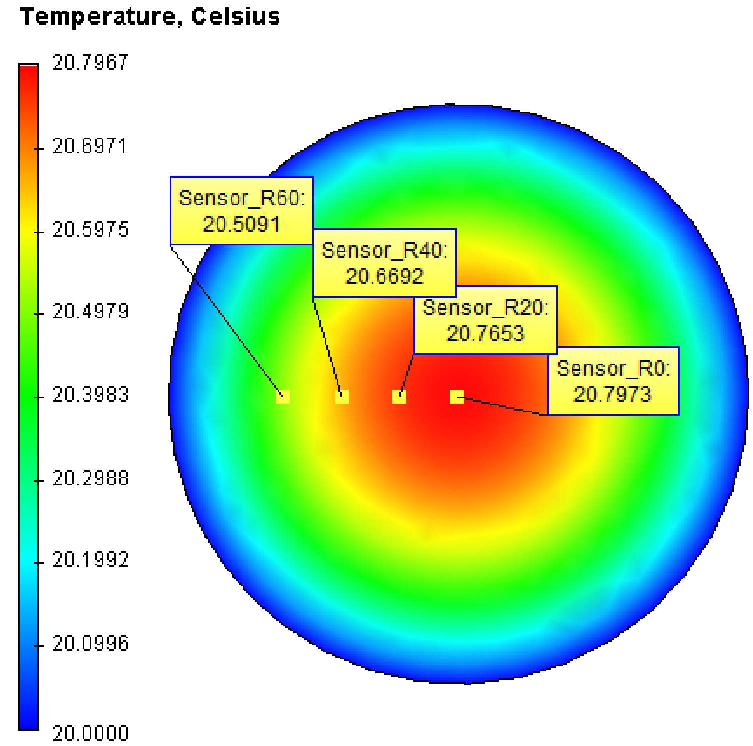
Table 3

Calculation with linear element

r, mm	0	20	40	60
Analytical solution, ^0C	20.8000	20.7610	20.6720	20.5120
Numerical solution, ^0C	20.7939	20.7609	20.6653	20.5049
Relative error, %	0.029326	0.000481	0.032410	0.034614



Units of radius measured from center of the disk; r is mm.



Temperature field

Conclusions:

The relative error of the numerical solution compared to the analytical solution did not exceed 0.03% for linear elements and 0.02% for quadratic elements, i.e., the solutions were obtained very accurately with up to 4 significant digits of accuracy and low computational cost.

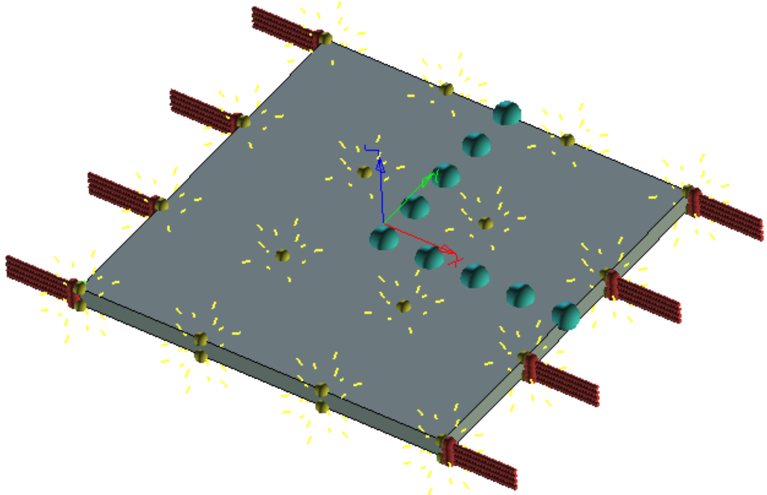
On average, the relative error of quadratic elements turned out to be twice lower than for linear elements. At the same time, the number of nodes we used for calculation with quadratic elements was 6 times larger.

It is important to note that at a point r=20 mm the solution with linear elements that we obtained turned out to be more accurate. This was really possible because of the convergence properties of the solution in the finite element method (convergence in a sense of integral norm and smaller number of points/arguments plays a role here). However, in general it is impossible to predict appearance and location of these points.

Radiation of a Plate into External Environment

Let us consider now a problem of calculation of the steady-state temperature field for an infinite flat plate that radiates to the external environment. On the edges of the plate (along the length), we maintain the constant temperature of T=500 0K (we assume that for this temperature the effect of radiation for this plate will be significant). Radiation will take place from the surface of the plate at both sides of the plate to the external environment with the temperature equal to Text=293 K. Now assuming the steady-state, we determine the temperature field on the surface of the plate at control points that are designated by the sensors (located along the OX-axis).

Characteristics of the plate: thickness d=5 mm; width l=100 mm; thermal conductivity K=50 W/(m * 0K); emissivity α =1. We maintain the zero heat flux along the boundary of a width of the plate.



Numerical model with boundary conditions and sensors

Let us consider the differential equation for this problem. Since the thermal heat flux on the opposite edges of the plate is equal to zero, the temperature field will be changing only along the length of the plate. By selecting the coordinate system such that the OX-axis is directed along the width of the plate and the OY-axis along the length, we will obtain the solution that depends only on the width of the plate. We will place the origin of the coordinate system O at the center of the plate. The equation will then take the form:

$$\frac{\partial^2 u}{\partial x^2} = C \cdot \left(u^4 - T_{ext}^4\right)$$

$$u(a) = u(b) = T$$

where the constant C is determined by taking into account the width of the plate as:

$$C = \frac{2 \cdot \alpha \cdot \sigma}{K \cdot d},$$

$$\sigma = 5.67 \cdot 10^{-8}$$

where σ – Boltzmann constant.

The analytical solution for this problem can be found as a Taylor's series expansion. It can also be obtained numerically in the system Maple 9.5. We will consider the value along the length only up to the middle point, i.e., the selected origin of the coordinate system, since the solution is an even function.

Let us represent now the analytical solution for the problem as a series. We note that $\frac{\partial^2 u}{\partial x^2} = C \cdot \left(u^4 - T_{ext}^4\right) > 0$ since from the physical point of view the plate could not cool down to the temperature below the temperature of the ambient environment. Hence, $\partial u/\partial x$ grows in a monotonous manner from $-\infty$ to $+\infty$ and that means that it has a single point of intersection with the OX-axis which will be the point of minimum of the solution. In the neighbourhood of this point we expand the solution in a series:

$$u(x_0 + \Delta x) = u(x_0) + \frac{\partial u}{\partial x}(x_0) \cdot \Delta x + \frac{\partial^2 u}{\partial x^2}(x_0) \cdot \frac{\Delta x^2}{2} + \ldots + \frac{\partial^n u}{\partial x^n}(x_0) \cdot \frac{\Delta x^n}{n!} + \ldots$$

By differentiating the given equation, we will obtain the expressions for the higher-order derivatives:

$$\frac{\partial^3 u}{\partial x^3} = d\left(\frac{\partial^2 u}{\partial x^2}\right) / dx = \frac{\partial\left(u^4\right)}{\partial x} = 4 \cdot u^3 \cdot \frac{\partial u}{\partial x},$$

$$\frac{\partial^4 u}{\partial x^4} = d\left(4 \cdot u^3 \cdot \frac{\partial u}{\partial x}\right) / dx = 4 \cdot u^3 \cdot \frac{\partial^2 u}{\partial x^2} + 12 \cdot \left(u \cdot \frac{\partial u}{\partial x}\right)^2 = \left[\frac{\partial^2 u}{\partial x^2} = C\left(u^4 - T_{ext}^4\right)\right] =$$

$$= 4C \cdot u^7 - 4C \cdot u^3 T_{ext}^4 + 12 \cdot \left(u \cdot \frac{\partial u}{\partial x}\right)^2$$

and so on. It is obvious that all derivatives can be represented in the form of a polynomial F of function u and the values of its first derivative

$$\frac{\partial^k u}{\partial x^k}(x)=F\left(u(x),\frac{\partial^{k-1}u}{\partial x^{k-1}}(x),\frac{\partial^{k-2}u}{\partial x^{k-2}}(x),\dots,\frac{\partial u}{\partial x}(x),T\right)=F\left(u(x),\frac{\partial u}{\partial x}(x),T=const\right)$$

For all odd derivatives, all the terms of the polynomial F will contain as a factor the first derivative and therefore at the point of extremum we have $\frac{\partial^{2k+1}u}{\partial x^{2k+1}}(x_0)=0$. Hence the solution will take the form

$$u(x_0+\Delta x)=\sum_{k=0}^{\infty}\frac{\partial^{2k}u}{\partial x^{2k}}(x_0)\cdot\frac{\Delta x^{2k}}{(2k)!}$$

It is obvious from the symmetry of boundary conditions that the point x0 coincides with the selected zero O of the coordinate system.

There is no simple analytical solution for $\frac{\partial^{2k}u}{\partial x^{2k}}(x_0)$ since on each step of calculation of the sum of the series, we need to store all the coefficients of the polynomial F to evaluate the derivative for the next step. That is why we used standard ways of analytical evaluation of the derivative via the system Maple 9.5.

The value of u at a point x0 was obtained in the following way: we select partial sums of the Taylor's series at a point x0 +l/2 and equate them to the boundary value of the temperature T. Solution of these equations can be found numerically and, moreover, only the real positive roots were of interest. As a result we obtained u(0) = 467.4671303. Let us compare the obtained solution with the analytical solution.

Table 1

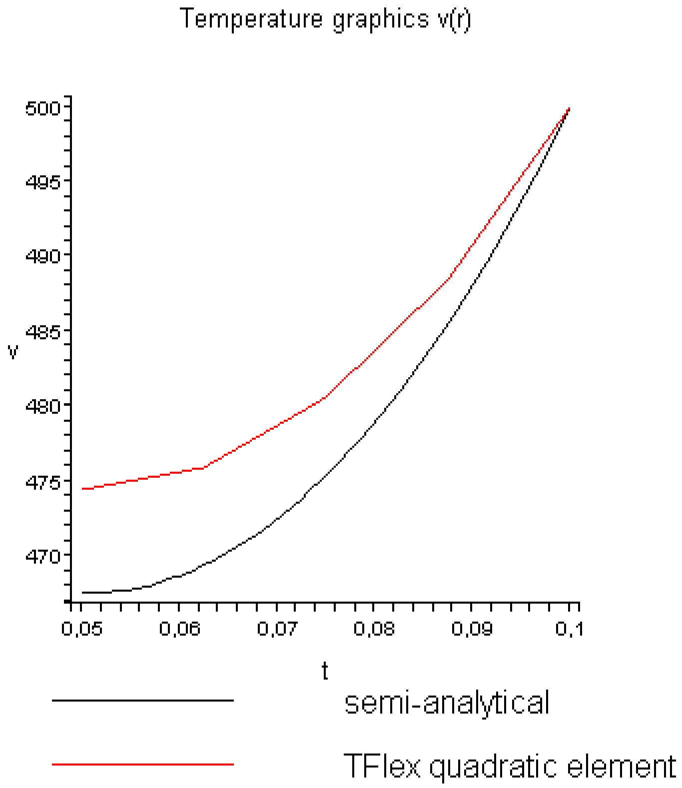
Calculation with linear element

Coordinate Y of a point of interest (mm)	37.5	25	12.5	0
Analytical solution (^{0}K)	485.442	475.357	469.425	467.467
Linear element (^{0}K)	488.622	480.737	476.017	474.401
Relative error, %	0.655073	1.13178	1.40427	1.48331

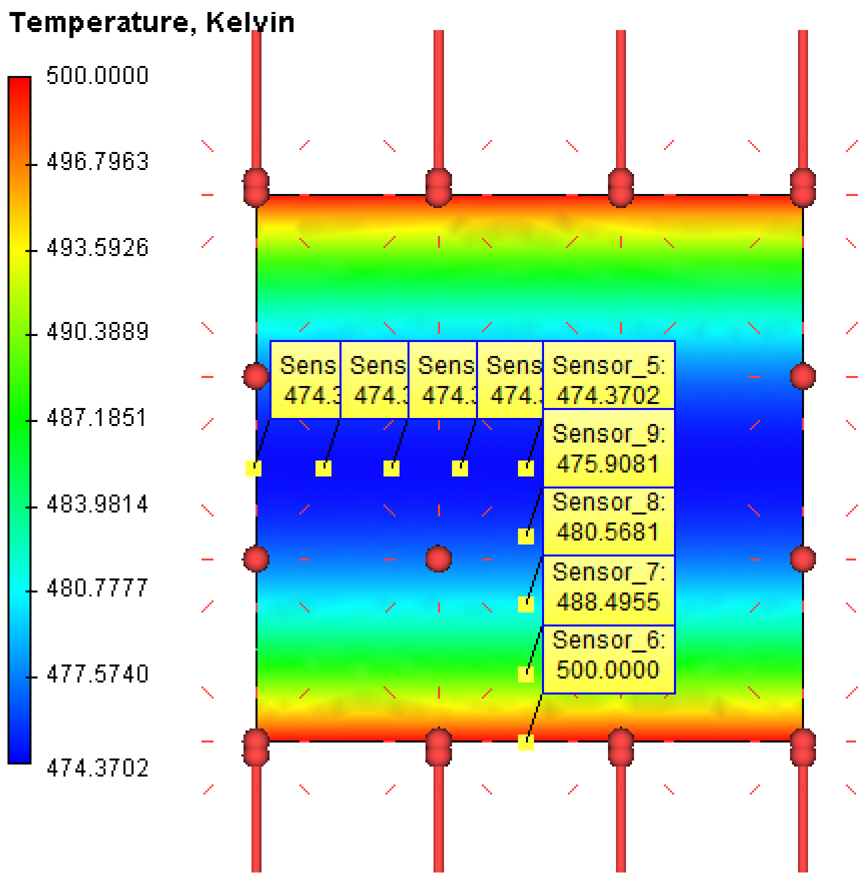
Table 2

Calculation with quadratic element

Coordinate Y of a point of interest (mm)	37.5	25	12.5	0
Analytical solution (^{0}K)	485.442	475.357	469.425	467.467
Quadratic element (^{0}K)	488.496	480.568	475.908	474.37
Relative error, %	0.629117	1.09622	1.38105	1.47668



Plot of temperature distribution along the width of the plate



Plot of a temperature field and the sensors

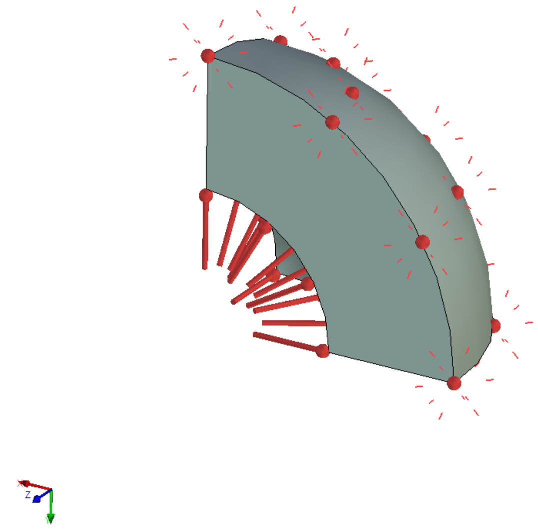
Conclusions:

The relative error of the numerical solution compared to the analytical solution was equal to 0.6% (on the axis of the plate) and reached 1.5% (on the edge of the plate). Notice that when the mesh is

refined, convergence of the numerical solution to the analytical solution is slower for problems with radiation since the problem is nonlinear.
As in the example with radiation of a hollow sphere to the ambient environment, we make a Conclusion that when solving a nonlinear problem it does not matter which elements are used for calculation: linear or quadratic.

Radiation of the Surface of a Hollow Sphere

Let us consider a problem of determination of the steady-state temperature field of the hollow sphere, on the internal surface of which we maintain a constant temperature T1 =600 0K, and the external surface radiates to the ambient environment. The ambient environment has the temperature Tокр=290 0K. For this temperature the effect of radiation significantly changes the temperature inside the volume of the body. We can determine the temperature field at the control points with radii R123.
Characteristics of the sphere: external radius R=100 мм; internal radius R/2=50 mm; the thermal conductivity K=50 W/(m · 0K); emissivity of the spherical surface is equal to α =1.
Let us consider a 1/8th part of the entire sphere. On the lateral surfaces of the sphere we prescribe the zero heat flux.



Numerical model with boundary conditions

Since the material is isotropic, solution of the problem will depend only on the radius. If the origin of the spherical coordinate system is placed at the center of the sphere, we will obtain the equation with boundary conditions:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\cdot\frac{\partial u}{\partial r}\right)=0$$

$$u(R/2)=T_1$$

$$K\frac{\partial u}{\partial r}(R)=\alpha\cdot\sigma\cdot\left(T_{amb}^4-u(R)^4\right)$$

The solution of the equation has the form:

$$u=C_1\cdot ln(r)+C_2$$

where the constants C1 and C2 are determined from the boundary conditions in the following way:

$$C_1=\frac{\alpha\cdot\sigma\cdot R}{K}\left(T_{amb}^4-(T_1+C_1\cdot\ln(2))^4\right)$$

$$C_2=T_1-C_1\cdot\ln\left(\frac{R}{2}\right)$$

The first equation (for the given parameters) has two real roots. For the roots: C1 = -4483.400605 and C2 = -12831.06789 the solution on the boundary has the negative temperature on the boundary. For Kelvin scale it is impossible. Therefore, only the roots C1 = -13.02949201 and C2 = 560.9671302 satisfy this equation .
Let us compare the numerical solution with the obtained analytical solution at points with radii R123=0.0707106; 0.0848528; 0.0989949 m (coordinates of the sensors at these points: X123=50; 60; 70 mm and Y123=50; 60; 70 mm).

Table 1

Parameters of the mesh

Finite element type	Number of mesh nodes used in calculation	Number of elements	Relative error
Tetrahedron, 4 nodes. Linear finite element.	1562	6823	0.09
Tetrahedron, 6 nodes. Quadratic finite element.	10594	6823	0.09

Table 2

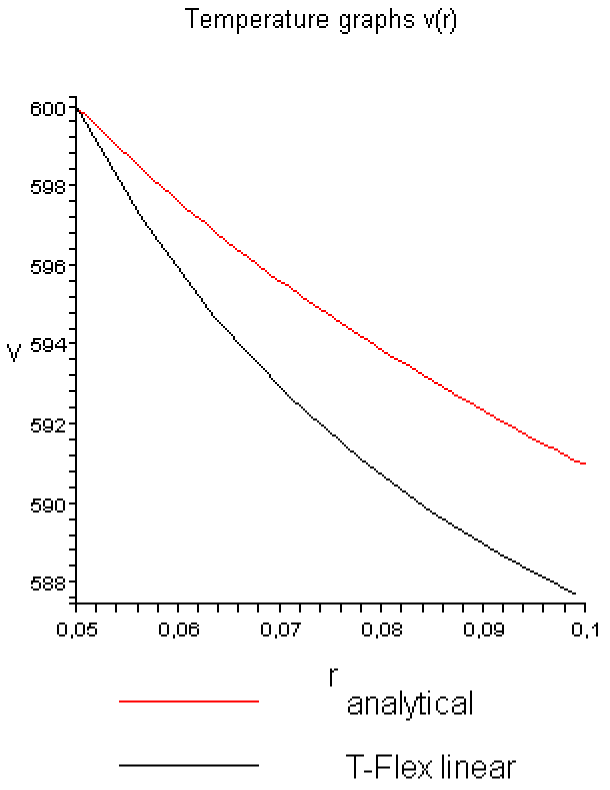
Calculation with linear element

Radius of a point of interest, m	0.0707106	0.0848528	0.0989949
Analytical solution, ^{0}K	595.4843221	593.1087648	590.9686443
Numerical solution, ^{0}K	592.7	589.8	587.7
Relative error, %	0.47	0.57	0.6

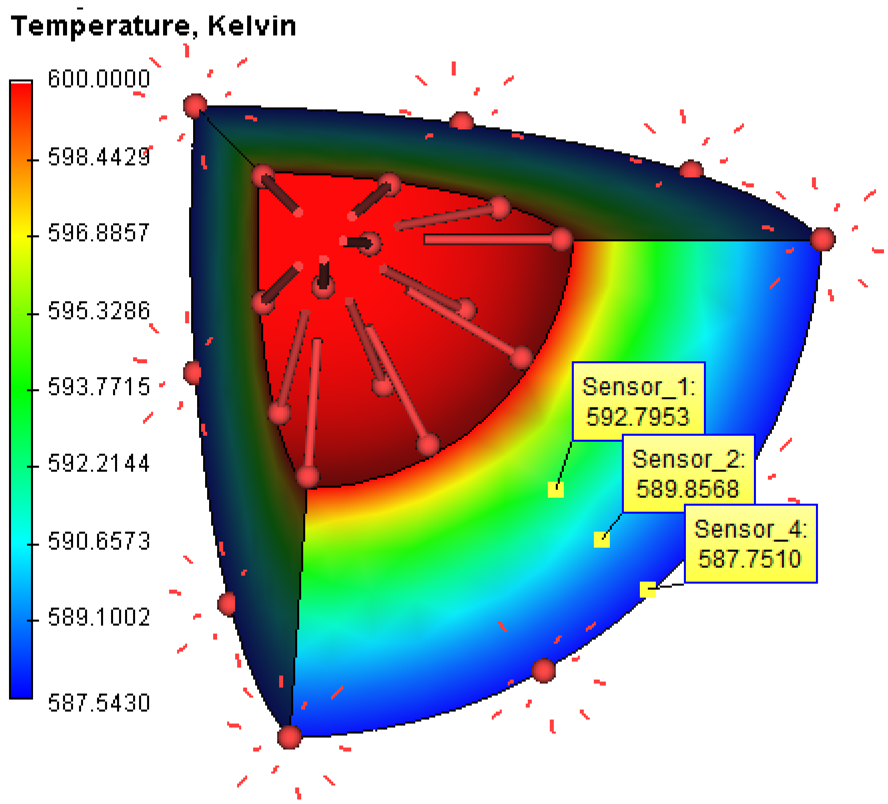
Table 3

Calculation with quadratic element

Radius of a point of interest, m	0.0707106	0.0848528	0.0989949
Analytical solution, ^{0}K	595.4843221	593.1087648	590.9686443
Quadratic element, ^{0}K	592.8	589.9	587.8
Relative error, %	0.47	0.56	0.54



Plot of dependence of the temperature V on radius r in the spherical coordinate system



Temperature field plotted from the results of the finite element analysis with the readings of the sensors

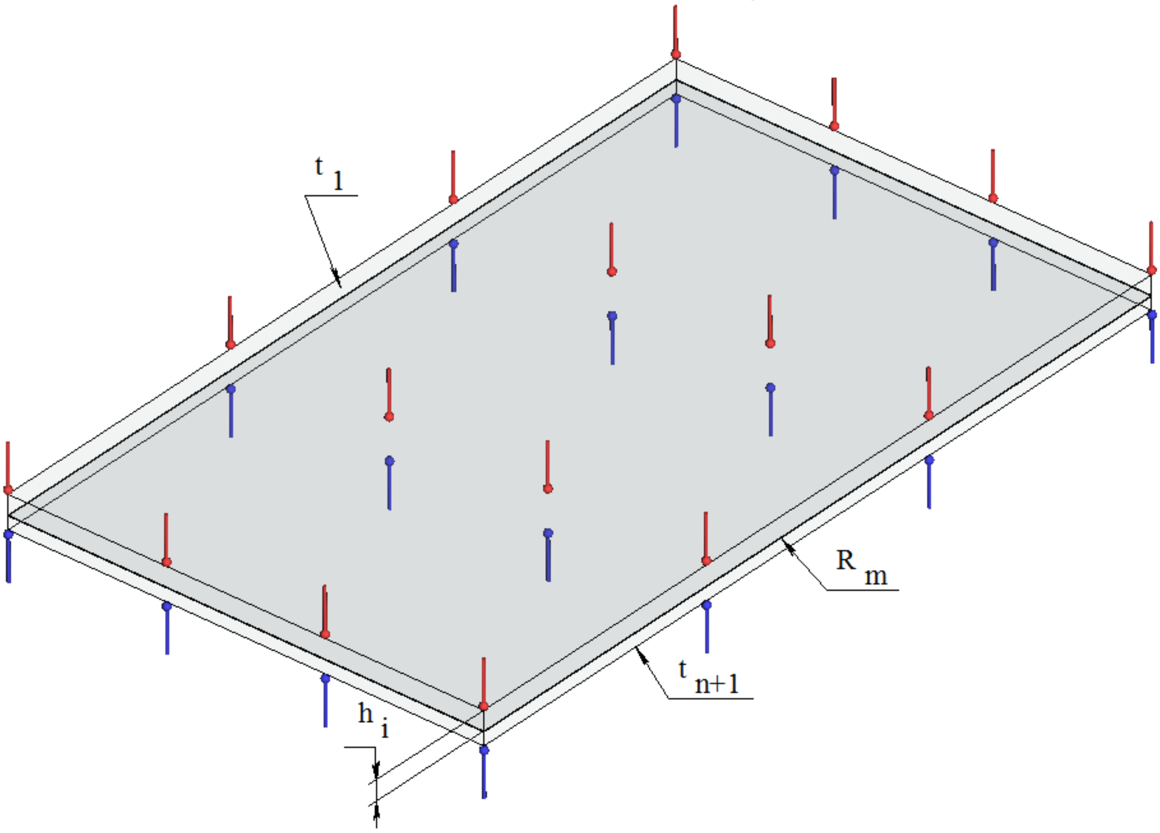
Conclusions:

The relative error of the calculations approaches 0.5%. Notice that when the mesh is refined, the numerical solution approaches the analytical solution slower because of the nonlinearity of the problem itself.

As we see, when computing the radiation, the relative error is practically the same for linear and quadratic elements. This can be explained by the error in the solution of the nonlinear equations. Thus, when computing radiation, it makes sense to use only linear elements because of smaller computational cost.

Thermal Resistance of a Flat Plate

Let us consider a problem of the steady-state heat flow in the composite plate of thickness $\sum h_i$ with coefficients of thermal conductivity k_i, whose upper and lower surfaces are maintained at temperatures t_1 and t_{n+1} , and between the plates with numbers m-1 and m+1 there is a thermal contact with the specific thermal resistance R_m (see the figure).



For each layer $i=1,2,\dots,n$ of the composite plate composed of n layers with thicknesses $h_1,h_2,\dots,h_n$ and coefficients of thermal conductivity $k_1,k_2,\dots,k_n$, respectively, the change in temperature and thermal flow across the thickness $f_i, i=1,2,\dots,n$ can be obtained from formula:

$$f_i=-k_i\frac{\partial T}{\partial z}=-\frac{k_i(t_{i+1}-t_i)}{h_i}=\frac{(t_i-t_{i+1})}{R_i},\ R_i=\frac{h_i}{k_i},t_i<t_{i+1},i=1,2,\dots,n$$

Let us assume that all layers except the two are in the state of ideal thermal contact across the interfaces and between those two layers with numbers m-1 and m+1 let us place a thermal resistance R_m. Then the thermal heat flow will be continuous upon transition from one layer to another, and in the given case, it will be equal at all points (that is, $f_1=f_2=\dots=f_n=f$). The change of temperature between opposite surfaces of the entire composite plate will be equal to the sum of temperature changes in individual layers:

$$(t_1-t_2)+(t_2-t_3)+\dots+(t_i-t_{i+1})+\dots+(t_n-t_{n+1})=t_1-t_{n+1}$$

Then

$$t_1-t_{n+1}=f_1R_1+f_2R_2+\dots+f_nR_n=(R_1+R_2+R_m+\dots+R_n)f,\ R_i=\frac{h_i}{k_i},i=1,2,\dots n$$

,

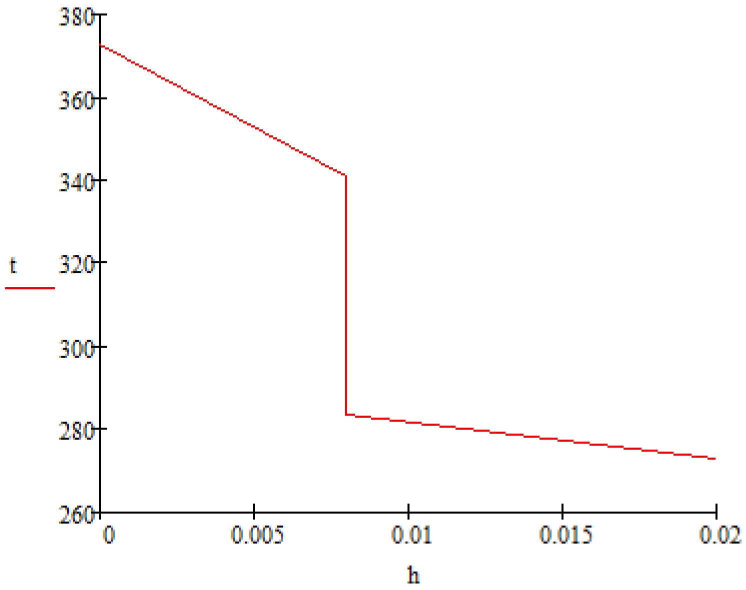
$$f=\frac{t_1-t_{n+1}}{\frac{h_1}{k_1}+\frac{h_2}{k_2}+\ldots R_m+\ldots+\frac{h_n}{k_n}}$$

Let us assume the following input data: the number of layers $n=2$, the length and width of each layer is equal to $500\ mm$ and $250\ mm$, respectively; thicknesses of the layers h_1, h_3 are equal to $8\ mm$, $12\ mm$. The applied temperatures t_1 and t_4 are equal to $373^0 K$ and $273^0 K$, respectively.

Coefficients of thermal conductivity: $k_1=43\frac{W}{m\cdot\deg}$, $k_3=200\frac{W}{m\cdot\deg}$

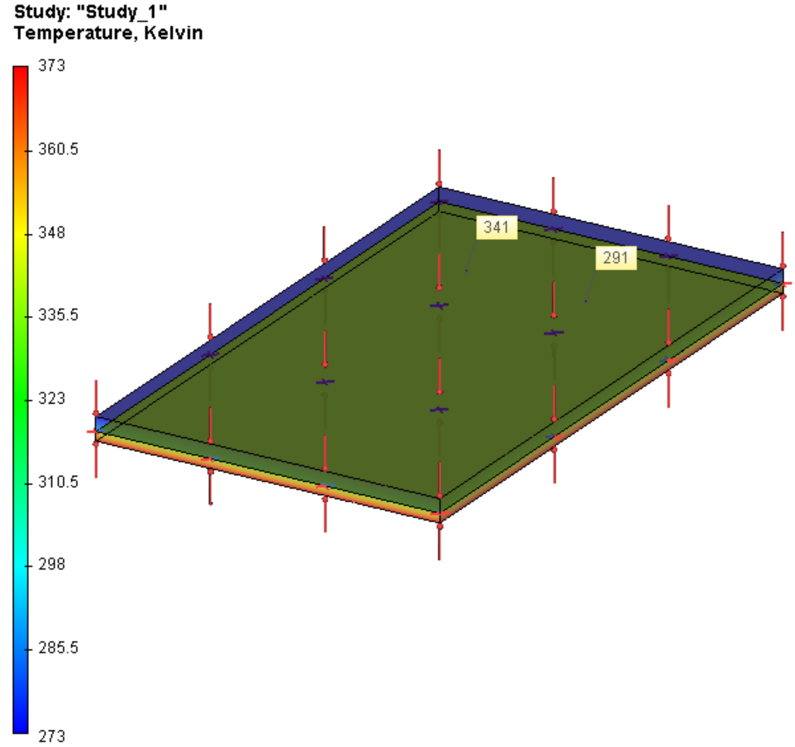
Thermal resistance of the contact surface: $R_2=3.33\times10^{-4}\frac{m^2\cdot\deg}{W}$ (approximately, equivalent to the thermal resistance of the flat layer of air of thickness 0,05 mm with $k_2=0.15\frac{W}{m\cdot\deg}$)

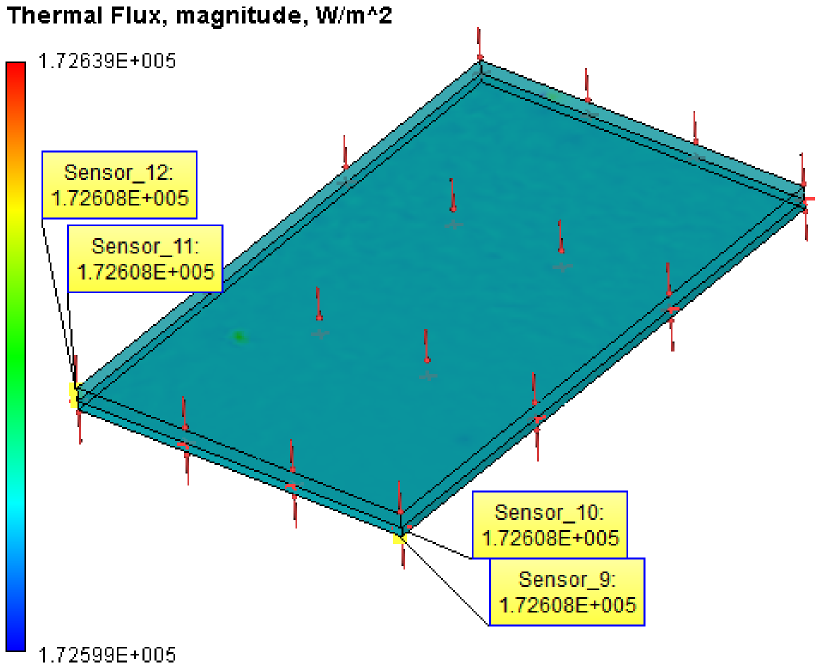
Therefore, $f=1.72599\times10^5\frac{W}{m^2}$, $t_2=-R_1 f+t_1=-1.860465\times10^{-4}\cdot1.72599\times10^5+373=340.88\ K$,
$t_3=-(R_1+R_2)f+t_1=-(1.860465\times10^{-4}+3.33\times10^{-4})\cdot1.72599\times10^5+373=283.35\ K$.



After calculations are carried out with the help of T-FLEX, we obtain the following results:

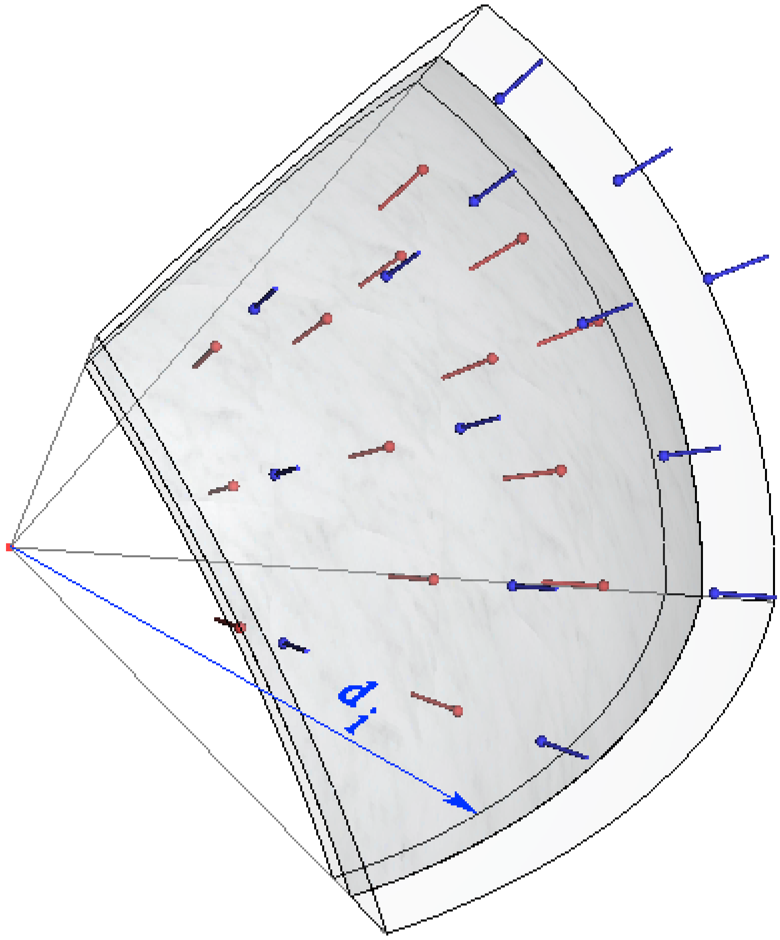
Mesh parameters			
Finite element type:	quadratic tetrahedron (10 nodes)		
Number of main nodes:	68155	Number of nodes for problem calculation:	68155
Number of finite elements	44043		
Calculation results			
	Numerical solution, w^*	Analytical solution w	Error $\delta_w=\frac{\lvert w-w^*\rvert}{w}\times100\%$
Heat flow, $\frac{W}{m^2}$	3.00×10^5	1.75631×10^5	0.71×10^2 %
Temperature, t_2, K	341.00	340.88	0.03 %
Temperature, t_3, K	291.00	283.35	2.7 %





Thermal Resistance of a Sphere

Let us consider a problem of steady state heat flow of a composite spherical plate of thickness $\sum 0{,}5\Delta d_i$ with coefficients of thermal conductivity k_i, whose upper and lower surfaces are maintained at temperatures t_1 and t_{n+1}, and between the layers with numbers m-1 and m+1 there is a thermal contact with specific thermal resistance R_m (see the figure).



For each layer $i=1,2,...,n$ of the composite plate composed of n layers with diameters $d_1, d_2, ..., d_n$ and coefficients of thermal conductivity $k_1, k_2, ..., k_n$, respectively, the change of temperature and thermal heat flow across the thickness of the plate $f_i, i=1,2,...,n$ can be determined from formula:

$$f_i = -k_i \frac{\partial T}{\partial z} = -\frac{2\pi k_i (t_{i+1} - t_i)}{\left(\frac{1}{d_i} - \frac{1}{d_{i+1}}\right)} = \frac{(t_i - t_{i+1})}{R_i}, R_i = \frac{\left(\frac{1}{d_i} - \frac{1}{d_{i+1}}\right)}{2\pi k_i}, t_i < t_{i+1}, i = 1,2,...,n$$

Let us assume that all layers except the two are in the state of ideal thermal contact across the interfaces and between those two layers with numbers m-1 and m+1 let us place a thermal resistance R_m. Then the thermal heat flow will be continuous upon transition from one layer to another, and in the given case, it will be equal at all points (that is, $f_1 = f_2 = ... = f_n = f$). The change of temperature between opposite surfaces of the entire composite plate will be equal to the sum of temperature changes in individual layers:

$$(t_1 - t_2) + (t_2 - t_3) + ... + (t_i - t_{i+1}) + ... + (t_n - t_{n+1}) = t_1 - t_{n+1}$$

Then

$$t_1 - t_{n+1} = f_1 R_1 + f_2 R_2 + ... + f_n R_n = (R_1 + R_2 + R_m + ... + R_n) f, \ i = 1,2,...n,$$

$$f = \frac{t_1 - t_{n+1}}{\frac{\left(\frac{1}{d_1} - \frac{1}{d_2}\right)}{2\pi k_1} + \frac{\left(\frac{1}{d_2} - \frac{1}{d_3}\right)}{2\pi k_2} + ... R_m + ... + \frac{\left(\frac{1}{d_n} - \frac{1}{d_{n+1}}\right)}{2\pi k_n}}$$

Let us assume the following input data: the number of layers $n=2$, diameters of layers d_1, d_2, d_3 are equal to 350, 380, 420 mm, respectively. Applied temperatures t_1 and t_4 are equal to $473^0 K$ and $273^0 K$ respectively.

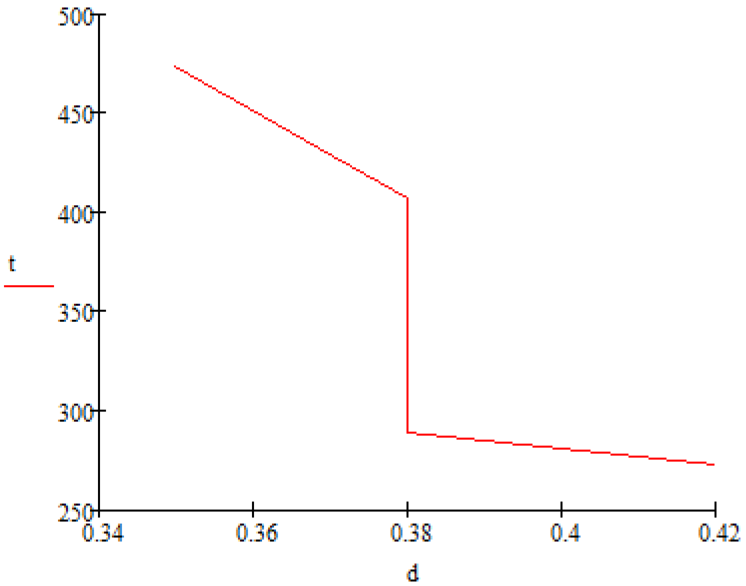
Coefficients of thermal conductivity: $k_1 = 43\,\dfrac{W}{m\cdot \deg}$, $k_3 = 200\,\dfrac{W}{m\cdot \deg}$ (steel and aluminum alloy).

Thermal resistance of the contact surface $R_2 = 1.5\times 10^{-3}\,\dfrac{m^2\cdot \deg}{W}$ (approximately equivalent to the thermal resistance of the spherical layer of air of thickness 0,1 mm with $k_2 = 0.15\,\dfrac{W}{m\cdot \deg}$)

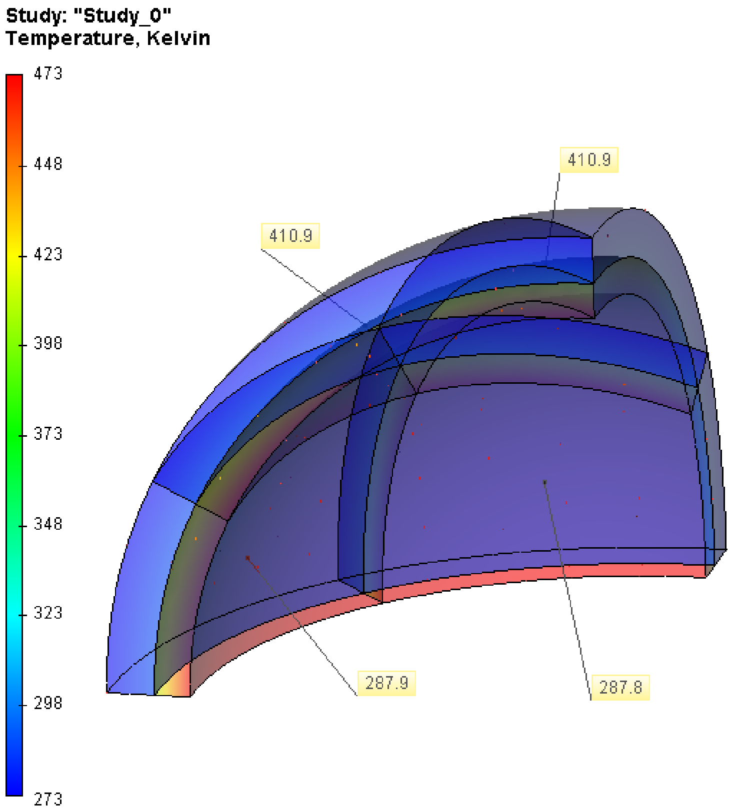
Therefore, $f = 7.8917\times 10^4\,\dfrac{W}{m^2}$,

$t_{21} = -R_1 f + t_1 = -4.174\times 10^{-4}\cdot 2.352\times 10^4 + 473 = 374.836\ K$,

$t_3 = -(R_1 + R_2) f + t_1 = -(8.349\times 10^{-4} + 1.5\times 10^{-3})\cdot 7.8917\times 10^4 + 473 = 288.7\ K$.



After calculations are carried out with the help of T-FLEX, we obtain the following results (at the lower and upper sides of the contact surface, respectively):



Results of finite element analysis (temperature)

Mesh parameters			
Finite element type	quadratic tetrahedron (10 nodes)		
Number of main nodes:	22115	Number of nodes for problem calculation:	22115
Number of finite elements	101217		
Results of calculation			
	Numerical solution, w^*	Analytical solution w	Error $\delta_w = \frac{\lvert w-w^*\rvert}{w}\times 100\%$
Heat flow, $\frac{W}{m^2}$	1.08×10^5	7.8917×10^4	0.36×10^2 %
Temperature, t_2, K	410.9	407.1	1%
Temperature, t_3, K	287.8	288.7	−0.3 %

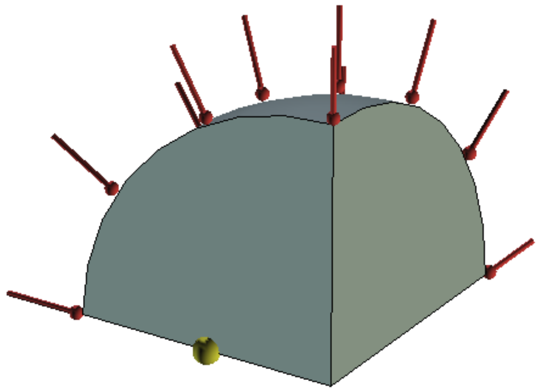
Mesh parameters			
Finite element type:	Linear tetrahedron (4 nodes)		
Number of main nodes:	22107	Number of nodes for problem calculation:	22107
Number of finite elements	101190		
Results of calculation			
	Numerical solution, w^*	Analytical solution w	Error $\delta_w = \frac{\lvert w-w^*\rvert}{w}\times 100\%$
Heat flow, $\frac{W}{m^2}$	1.08×10^5	7.8917×10^4	0.36×10^2 %
Temperature, t_2, K	410.9	407.1	1%
Temperature, t_3, K	287.8	288.7	−0.3 %

Non-Stationary Temperature Field for an Isotropic Sphere

Let us consider a problem of determination of a temperature field inside an isotropic sphere in the volume of which we prescribe the initial temperature Tstart=20 0C, and which is heated along the external surface where we prescribe the constant temperature Tborder=100 0C (for example, the sphere at room temperature was put down to the boiling water). Let us determine the temperature at any point of the sphere (located at a distance r from the center of the sphere) in time increments equal to Δ t1,2,3 = 10, 20, 30 seconds.

The sphere has the following parameters: radius a =100 mm, density of the material 7800 kg / m3, specific heat c = 440 J / (kg ·0C), thermal conductivity K=50 W / (m · 0C). Several alloys of steel have similar thermal characteristics.

For numerical modelling we consider a 1/8th part of the entire sphere. Conditions of symmetry are enforced by prescribing zero heat flux on the boundary surfaces of the sphere which have a center of the sphere as a vertex.



Numerical model with boundary conditions and a sensor located at a coordinate r=50 mm

Figure above shows the model of this body. A temperature sensor is shown with yellow color. It is located half radius away from the center.

Let v be the desired solution (temperature field). Then, by taking into account that the solution does not depend on the angles of rotation of the vector emanating from the center of the sphere (symmetry condition), we can perform the change of variables in the form v=r·u, where r – distance from the center of the sphere, and u – some function. After this change of variables, we obtain the equation for u

$$\frac{\partial u}{\partial t} = \frac{K}{c \cdot \rho} \cdot \frac{\partial^2 u}{\partial r^2}$$

where t – is time for cooling/heating of the solid body. Boundary conditions for u:

$$u(r,0) = r \cdot f(r) = T_{start} \cdot r, \quad t = 0$$

$$u(a,t) = r \cdot \varphi(t) = T_{border} \cdot r, \quad r = a$$

By solving this problem by the method of separation of variables, we obtain the expression for v:

$$v(r,t) = \frac{2}{ar}\sum_{n=1}^{\infty} A_n \cdot e^{-\chi\left(\frac{\varkappa^2\pi^2}{a^2}\right)t} \cdot \sin\left(\frac{n\pi r}{a}\right)$$

where χ=K/(c·ρ) is the coefficient of temperature conductivity. The weights An in the expansion are determined by the formula:

$$A_n = \int_0^a r' \cdot f(r')\sin\left(\frac{\pi n r'}{a}\right)dr' - n\pi\chi(-1)^n \int_0^t e^{\chi\left(\frac{\varkappa^2\pi^2}{a^2}\right)\lambda} \cdot \varphi(\lambda)d\lambda =$$

$$= \frac{a^2}{n\pi}(-1)^{n+1} \cdot \left\{T_{start} + T_{border}\left(e^{\chi\left(\frac{\varkappa^2\pi^2}{a^2}\right)t} - 1\right)\right\}$$

Let us compare the numerical solution with this semi-analytical solution.

Table 1

Parameters of the finite element mesh

Finite element type	Number of mesh nodes used in calculations		Number of mesh elements	Relative size
	Linear	Quadratic		

	ele me nts	element s		
Tetrahedron, 4 nodes	177 6	11966	7711	0.09

Table 2

Parameters of time discretization

Total calculation time (sec)	Time step (sec)	Number of time layers
120	1	120

Since the alternative solution (with which we compare the solution of T-FLEX) was obtained by the semi-analytical approach (by extraction of partial sums of the series), it is required to determine the number of significant digits which can be used for comparison with analytical solution. Table given below shows with what accuracy the calculations were carried out for obtaining solution by the series expansion and for construction of the solution plot. Conclusion about the number of significant digits in the analytical solution can be made by the indicator of the relative change in the solution and the fact that our series always converges.

Table 3

Parameters of calculation of semi-analytical solution obtained by series expansion:

Number of terms in a series, N	Time in sec. t, (temperature depends on time)	Value of the temperature, $u_n(t)$, in ^{0}C.	Value of the temperature, when N is doubled, $u_{2n}(t)$, in ^{0}C.	Relative change $\delta = \frac{\lvert u_n - u_{2n} \rvert}{\lvert u_n \rvert} \cdot 100\%$
500	10	20.48013866	20.41647716	0.31
	20	26.07028450	26.00662298	0.24
	30	34.46547534	34.40181384	0.18
7000	10	20.53470594	20.53925332	0.02
	20	26.12485180	26.12939918	0.01
	30	34.52004264	34.52459002	0.01

For numerical calculation we take the results with N=7000. For construction of plots we use the results with N=500.

Tables that contain the values of temperatures at a point r=50 mm (midradius) in 0C are presented below.

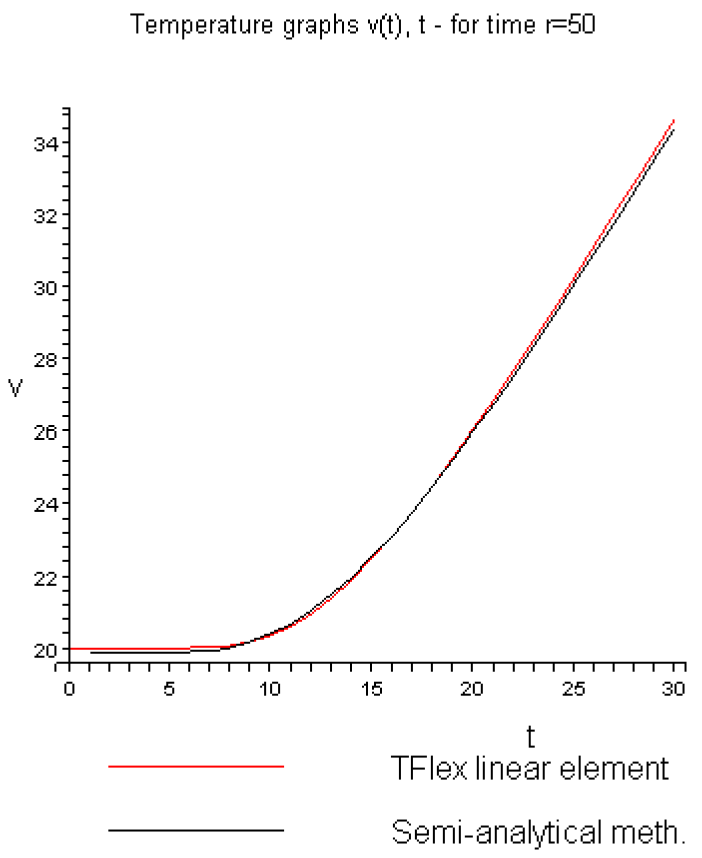
Table №1 for calculation with linear element, 4 significant digits.

Calculation time t, sec	10	20	30
Analytical solution, ^{0}C	20.53	26.12	34.52
Numerical solution, ^{0}C	20.37	26.08	34.67

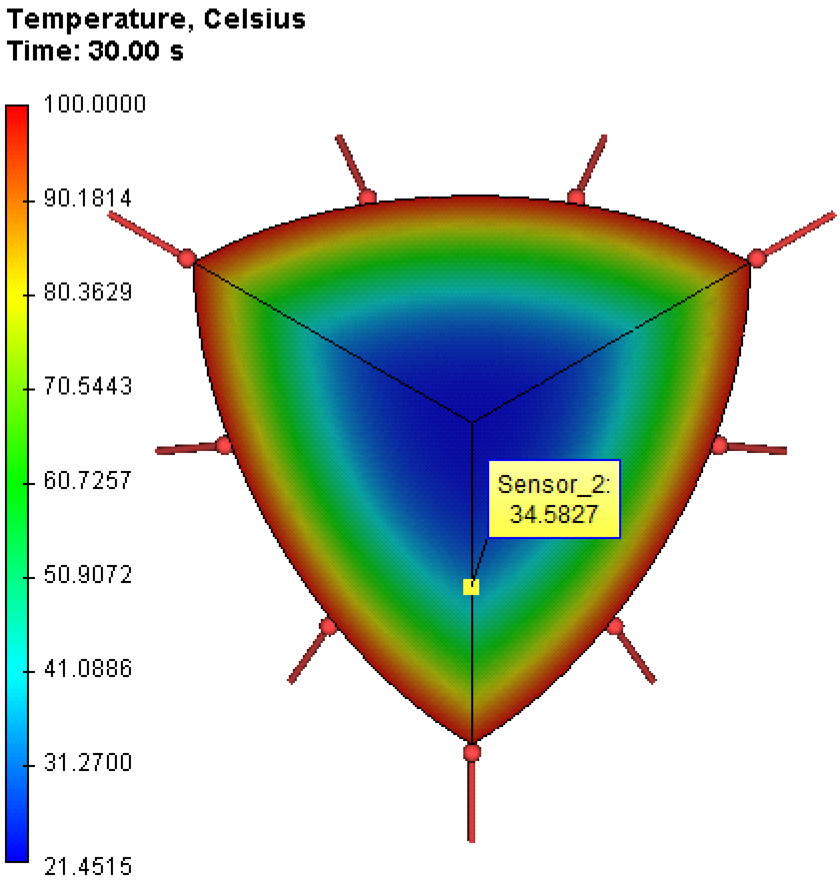
Relative error, %	0.7793	0.1531	0.4345

Table №2 for calculation with quadratic element, 4 significant digits.

Calculation time *t*, c	10	20	30
Analytical solution, ^0C	20.53	26.12	34.52
Numerical solution, ^0C	20.61	26.16	34.58
Relative error, %	0.3896	0.1531	0.1738



Plot of dependence of temperature V(t) at a point r=50 mm on time t

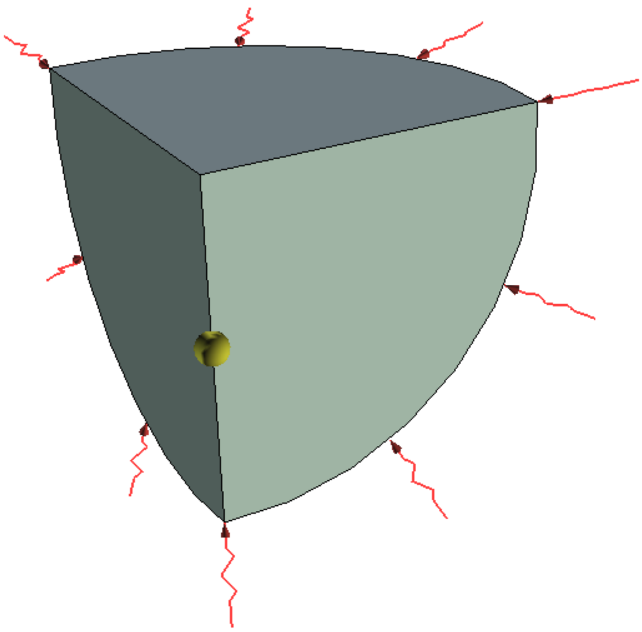


Temperature field in 30 seconds

Conclusions:
We obtained realistic picture of the temperature field. The relative error of the numerical solution compared to the analytical solution does not exceed 0.8% with linear elements and 0.4% with quadratic elements. The calculation error is stable in time and does not grow significantly when the computational time is increased. Plot of dependence of temperature on time shows that analytical and numerical solutions practically coincided.
The calculation error with quadratic elements is smaller than that with linear elements on a small interval of time.

Non-Stationary Temperature Field in an Isotropic Sphere with the Heat Transfer on the Surface

Let us consider the problem of a temperature determination inside an isotropic sphere, from which the heat is being removed, if in the volume of the sphere we prescribe the initial temperature Tstart = 80 °C, and on the surface we prescribe the heat flux of magnitude F=-800 W/(m2 °C). The minus sign implies that the sphere looses the heat. Let us determine the temperature at any point of the sphere in equal increments of time Δ t1,2,3 =20, 60, 90, 120 sec.
Parameters of the sphere: radius a=100 mm, density of the material 7800 kg/m3, specific heat c = 480 J / (kg· °C), thermal conductivity K=150 W / (m· °C).
For numerical modelling we consider a 1/8th part of the entire sphere. Conditions of symmetry are enforced by prescribing zero heat flux on the boundary surfaces of the sphere that have a center of the sphere as a vertex.



Numerical model with boundary conditions

Figure above shows the model of this body. Let v be the desired solution (temperature field). Then, by taking into account that the solution does not depend on the angles of rotation of the vector emanating from the center of the sphere (symmetry condition), we can perform the change of variables in the form v=r·u and obtain the equation for u

$$\frac{\partial u}{\partial t} = \frac{K}{c \cdot \rho} \cdot \frac{\partial^2 u}{\partial r^2}$$

where t – time for heating/cooling of the solid body. Boundary conditions for u are expressed in the following form:

$$u(r,0) = r \cdot f(r) = T_{start} \cdot r, \quad t = 0$$

$$K \frac{\partial u}{\partial r}(a,t) = F, \quad r = a$$

where f(r) – initial distribution of the temperature. Analytical solution of this problem is given below.

$$v(r,t) = \frac{3Ft}{\rho c a} + \frac{F(5r^2 - 3a^2)}{10Ka} - \frac{2Fa^2}{Kr} \sum_{n=1}^{\infty} \frac{\sin(r\alpha_n / a)}{\alpha_n^2 \sin(\alpha_n)} \cdot e^{-\chi \alpha_n^2 t / a^2} + T_{start}$$

where χ =K/(c·ρ) is the coefficient of temperature conductivity. The coefficients αn are determined as the roots of the equation (only positive roots):

$$tg(\alpha_n) = \alpha_n$$

Analytical solution was obtained with the accuracy of up to 6 significant digits.

Table 1

Parameters of the mesh

Finite element type	Number of mesh nodes used in calculations		Number of mesh elements	Relative error
	Linear elements	Quadratic elements		
Tetrahedron, 4 nodes	1776	11966	7711	0.09

Table 2

Parameters of time discretization

Total calculation time (sec)	**Time step (sec)**	**Number of time steps**
120	1	120

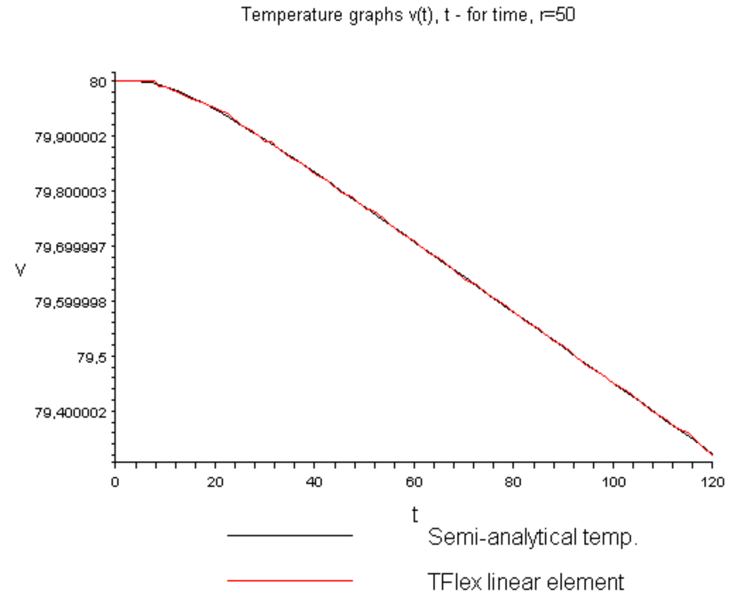
Tables of the values of temperature in 0C at a point located at a distance of 50 mm from the center are given below.

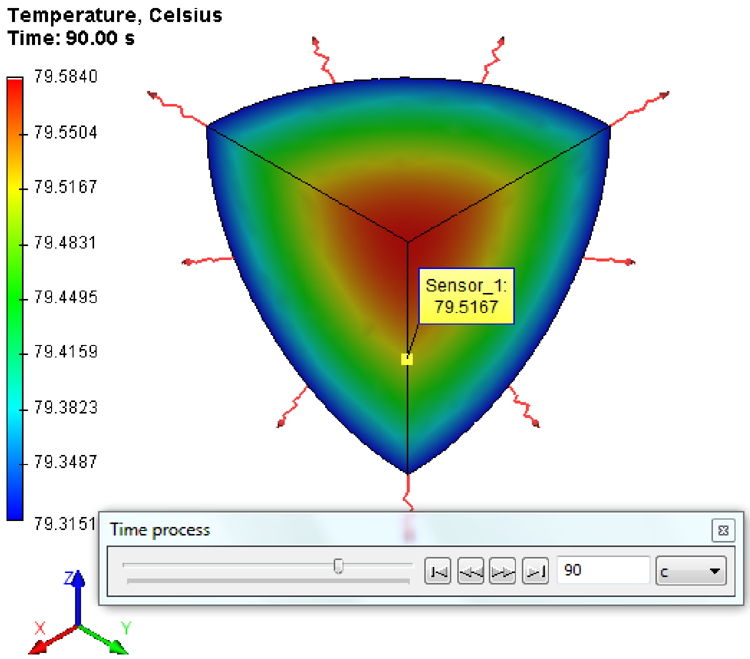
Table №1 for calculation with linear element

Calculation time t, sec	20	60	90	120
Analytical solution, ^0C	79.9481	79.7080	79.5163	79.3240
Numerical solution, ^0C	79.9491	79.7086	79.5167	79.3242
Relative error, %	0.00125	0.00068	0.00044	0.00013

Table №2 for calculation with quadratic element

Calculation time t, sec	20	60	90	120
Analytical solution, ^0C	79.9481	79.7080	79.5163	79.3240
Numerical solution, ^0C	79.9479	79.7075	79.5155	79.3231
Relative error, %	0.00035	0.00070	0.00107	0.00126





Temperature field in 90 seconds (linear elements)

Conclusions:

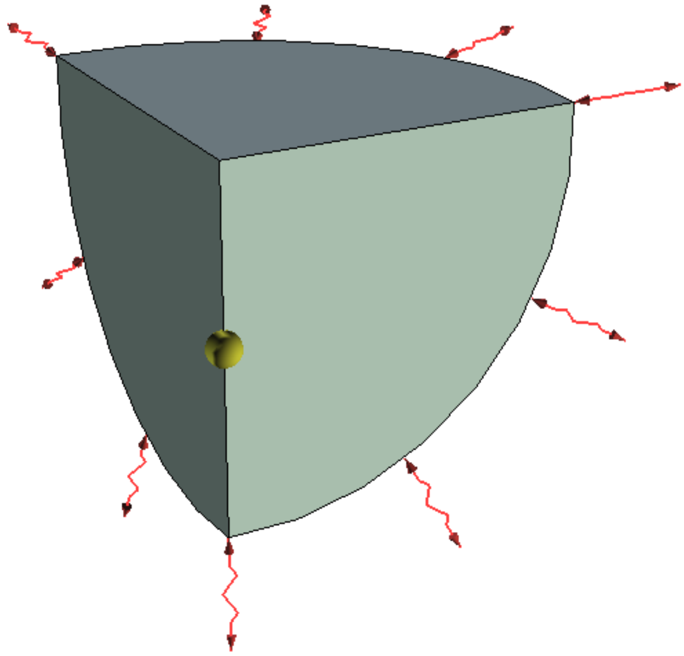
The relative error of the numerical solution compared to the analytical solution is smaller than 0.01 %. The calculation error is stable in time and does not grow significantly when the computational time is increased. Plot of dependence of temperature on time shows that analytical and numerical solutions practically coincided.

When using quadratic elements the number of nodes is significantly larger than that for linear elements. Hence, with time (for each new time layer), quadratic elements accumulate larger error than linear elements. As we see, in 20 seconds, i.e., for relatively small time interval for our problem, quadratic elements are more accurate than linear elements, but on the significantly larger time interval the calculation error with quadratic elements became even larger.

Non-Stationary Temperature Field in an Isotropic Sphere with the Heat Exchange on the Surface

Let us consider the problem of determination of the temperature at any point inside the sphere in equal increments of time $\Delta t_{1,2,3}$ = 30, 40, 60 sec if we prescribe the initial temperature T_{start}=60 ^{0}C inside the sphere, and on the boundary of the sphere we prescribe the heat exchange by convection with the heat transfer coefficient H=300 W/(m$^{2}\cdot{}^0$C) (the sphere that had a temperature of 60 ^{0}C inside was put down to cold sea water at a temperature of zero degrees).

Parameters of the sphere: radius a=100 mm, density of the material 7800 kg / m^3, specific heat c=440 J / (kg $\cdot{}^0$C), thermal conductivity K=50 W / (m $\cdot$ ^{0}C). For numerical modelling we consider a 1/8th part of the sphere. Conditions of symmetry are enforced by prescribing zero heat flux on the boundary surfaces of the sphere that have a center of the sphere as a vertex.



Numerical model with boundary conditions

Figure above shows the model of the solid body. Let v be the desired solution (temperature field). Then, by taking into account that the solution does not depend on the angles of rotation of the vector emanating from the center of the sphere (symmetry condition), we can perform the change of variables in the form $v=r\cdot u$ and obtain the equation for u

$$\frac{\partial u}{\partial t} = \frac{K}{c \cdot \rho} \cdot \frac{\partial^2 u}{\partial r^2}$$

where t – is the time for cooling/heating of the solid body. Boundary conditions for u are expressed in the following form:

$$u(r,0) = r \cdot f(r) = T_{start} \cdot r, \quad t = 0$$

$$\frac{\partial u}{\partial r}(a,t) + \left(h - \frac{1}{a} \right) u = 0, \quad r = a$$

where $f(r)$ – initial distribution of the temperature, and the coefficient h = H/K.

Solving this problem by the method of separation of variables, we obtain the expression for *v*:

$$v(r,t)=\frac{2hT_{start}}{r}\sum_{n=1}^{\infty}A_n\cdot e^{-\chi\cdot\alpha_n^2\cdot t}\cdot\sin(r\cdot\alpha_n)$$

where $\chi=K/(c\cdot\rho)$ is the coefficient of temperature conductivity. The expansion coefficients A_n and the values α_n can be determined by the formula:

$$A_n=\sin(a\cdot\alpha_n)\cdot\frac{a^2\alpha_n^2+(ah-1)^2}{\alpha_n^2\left(a^2\alpha_n^2+ah(ah-1)\right)}$$

and

$$a\cdot\alpha_n\cdot ctg\,(a\cdot\alpha_n)+ah-1=0$$

i.e., α_n – are the roots of the last equation. Now let us compare the numerical solution with the analytical solution. The analytical solution was obtained with the accuracy of up to 6 significant digits.

Table 1

Parameters of a finite element mesh

Finite element type	Number of mesh nodes used in calculations		Number of finite elements	Relative error
	Linear element	Quadratic element		
Tetrahedron, 4 nodes	1776	12078	7809	0.09

Table 2

Parameters of time discretization

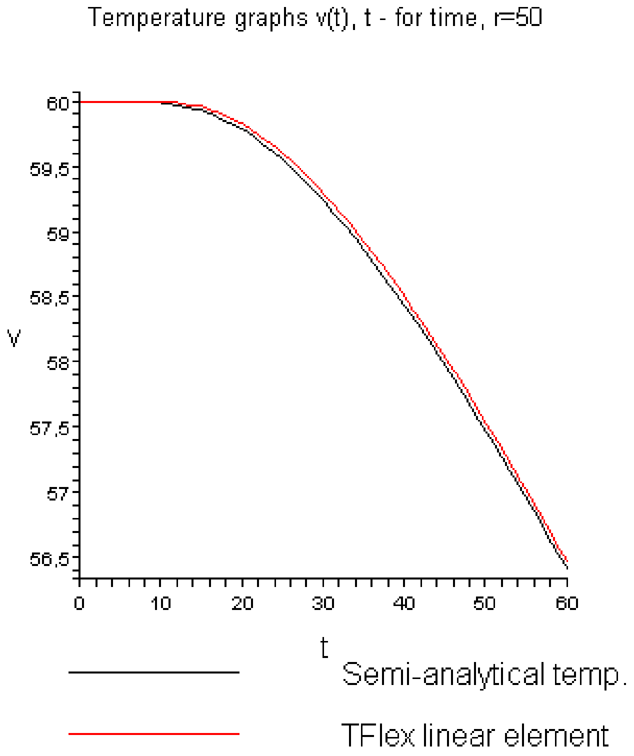
Total calculation time (sec)	Time step (sec)	Number of time layers
60	1	60

Tables of values of temperatures in ^0C at a point located from the center at a distance of 50 mm.

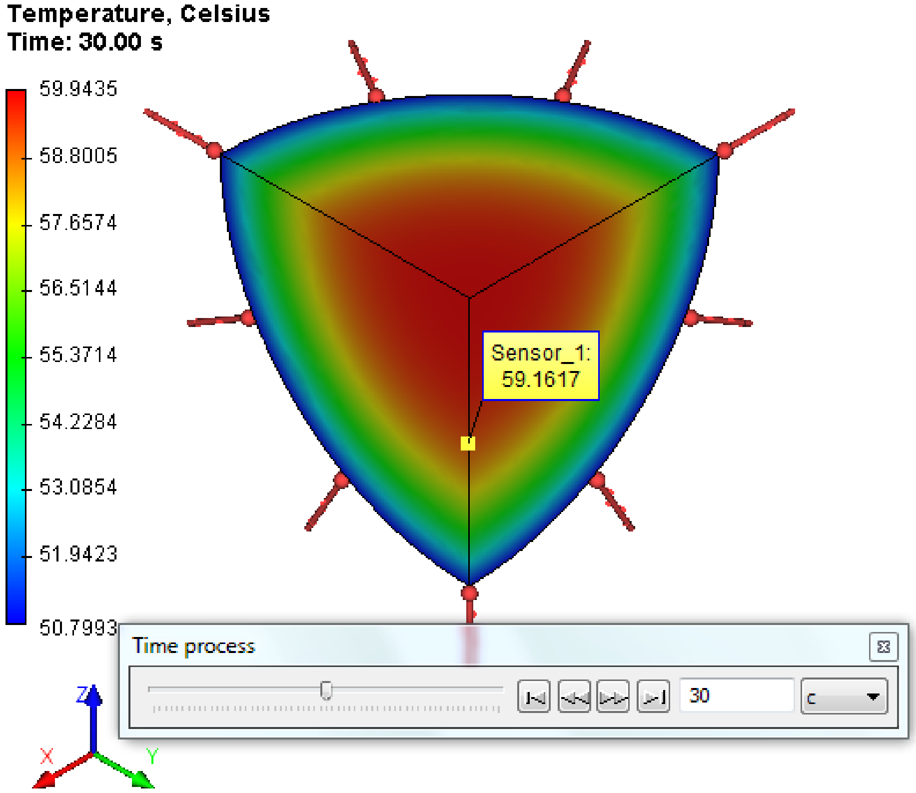
Table №1 for calculation with linear element			
Calculation time *t*, c	30	40	60
Analytical solution, ^0C	59.2381	58.4375	56.4123
Numerical solution, ^0C	59.2223	58.4072	56.3901
Relative error, %	0.03	0.05	0.035

Table №2 for calculation with quadratic element			
Calculation time *t*, c	30	40	60
Analytical solution, ^0C	59.2381	58.4375	56.4123

Numerical solution, ^0C	59.1617	58.3415	56.3222
Relative error, %	0.11	0.15	0.15



Plot of dependence of temperature V(t) on time t at a point r=50 mm.



Plot of the temperature field in 30 seconds (linear element)

Conclusions:

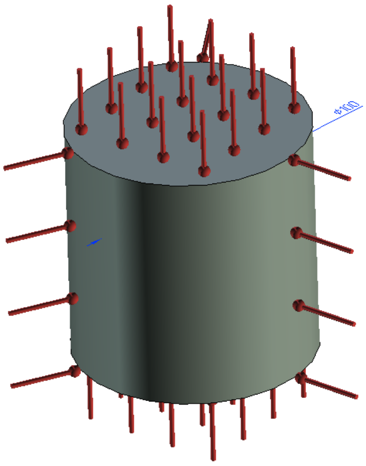
We confirmed the numerical efficiency of the method. The relative error of the numerical solution compared to the analytical solution was smaller than 0.5%, which guarantees two significant digits accuracy for relatively small computational expenses of memory and time. The calculation error is stable in time and does not grow significantly when the computational time is increased. Plot of dependence of temperature on time shows that analytical and numerical solutions practically coincided.

The calculation error is significantly smaller for quadratic elements than for linear elements, however, the time rate of growth of the error is somewhat larger for quadratic elements.

Non-Stationary Temperature Field in an Isotropic Cylinder

Let us consider a problem of cooling of a cylindrical body with isotropic properties that has an initial temperature of t0=60 0C inside the volume. The temperature equal to zero is maintained on the boundary of the body; cooling time does not exceed 20 sec. Let us determine the temperature in control points 1, 2, 3 that have the following coordinates in the cylindrical coordinate system (the origin of the coordinate system is located in the center of the cylinder): r1=25 mm, h1=25 mm; r2=25 mm, h2= - 25 mm; r3=30 mm, h3=0 mm at the following moments of time t1,2,3=2; 10; 20 sec.

Geometric and physical parameters of the body: height of the cylinder H =100 mm, radius of the cylinder a =50 mm. Density ρ =7.7 g/cm3, specific heat c =460 J/(kg*0C), thermal conductivity K = 40 W/(m * 0C).



Numerical model with boundary conditions

Let us look for the solution in the cylindrical coordinate system. The center of the coordinate system is located at the center of the cylinder in the middle cross-section (the total height - h), and the axis from which the distance is measured (radius in the cylindrical coordinate system – r) coincides with the axis of the cylinder. Denote l=H/2. Then the solution of the equation takes the form:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r \cdot \frac{\partial u}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 u}{\partial \varphi^2}+\frac{\partial^2 u}{\partial z^2}=\chi\frac{\partial u}{\partial t},$$

$$\frac{K}{c \cdot \rho}=\chi$$

Boundary conditions for this equation have the following form:

$$u(r,\varphi,z)=f(r,\varphi,z), t=0$$

$$u(t)=0, (r,\varphi,z)\in\{(r,\varphi,z): r=a\}\cup\{(r,\varphi,z): h=\pm l\}$$

In the boundary conditions we imply that f – some distribution of the temperature inside the body at initial moment of time, u(t)=0 on the surface of the cylindrical body during the entire period of time. We notice that in our case f – is a constant.

Let us represent the solution of the given equation in the form of a series in harmonic and Bessel functions for more general case when f is not a constant.

$$u(r,\varphi,z,t)=\sum_{\mu}\sum_{m=1}^{\infty}\sum_{n=0}^{\infty} e^{-t\left(\mu^2+\frac{m^2\pi^2}{4l^2}\right)t} J_n(\mu\cdot r)\cdot\sin\frac{m\pi(z+l)}{2l}\times$$
$$\times\left(A_{\mu,m,n}\cos(n\varphi)+B_{\mu,m,n}\sin(n\varphi)\right)$$

where Jn (r)– Bessel function, parameter μ is a root of the equation Jn (aμ) =0

The coefficients A and B can be calculated according to the formulas shown below:

$$A_{\mu,m,n}=\frac{2}{\pi a^2 l\{J_n'(\mu a)\}^2}\int_{-\pi}^{\pi}\cos(n\varphi)\int_{-l}^{l}\sin\frac{m\pi(z+l)}{2l}\int_{0}^{a} rJ_n(\mu r)\cdot f(r,\varphi,z)drdzd\varphi$$

$$B_{\mu,m,n}=\frac{2}{\pi a^2 l\{J_n'(\mu a)\}^2}\int_{-\pi}^{\pi}\sin(n\varphi)\int_{-l}^{l}\sin\frac{m\pi(z+l)}{2l}\int_{0}^{a} rJ_n(\mu r)\cdot f(r,\varphi,z)drdzd\varphi$$

In our simple case when f = t0 from the entire series over n we have only the first term left corresponding to n=0. But expansion over m and μ must be considered. The simplified form of the solution can be written as:

$$u(r,\varphi,z,t)=\sum_{\mu}\sum_{m=1}^{\infty}A_{\mu,m}e^{-t\left(\mu^2+\frac{m^2\pi^2}{4l^2}\right)t}J_0(\mu\cdot r)\cdot\sin\frac{m\pi(z+l)}{2l}$$

where

$$A_{\mu,2m+1}=\frac{8\cdot t_0}{\pi\mu a J_1(\mu a)(2m+1)},$$
$$A_{\mu,2m}=0$$

i.e., in the sum over m we retain only the odd coefficients.

Let us compare the value of the temperature at a fixed point at different moments of time with the solution obtained by the finite element method. The point will be selected at a sufficiently large distance from the main axis of the cylinder, for better convergence of the series in the analytical solution (the equation has a singularity of the type 1/r which significantly affects the convergence of the series of Bessel functions).

Let us compare the solution of the T-FLEX analysis with the analytical solution. For semi-analytical method of the solution (extraction of partial sum of the series) we obtained the solution with the accuracy exceeding 6 significant digits.

Table 1

Parameters of a finite element mesh

Finite element type	Number of mesh nodes used in calculations	Number of finite elements	Relative size
Tetrahedron, 4 nodes. Linear finite element.	2473	11209	0.09
Tetrahedron, 6 nodes. Quadratic finite element.	16945	11209	0.09

Parameters of time discretization

Total computational time (sec)	Time step (sec)	Number of time layers
20	0.5	40

Table 2

Table of control points:

Point number	1	2	3
r, mm	25	25	30
h, mm	25	-25	0

Let us examine the value of the temperature at the moments of time: 2, 10, 20 sec.

Temperature at control points in 0C is given below.

Table 3

Calculation with linear elements

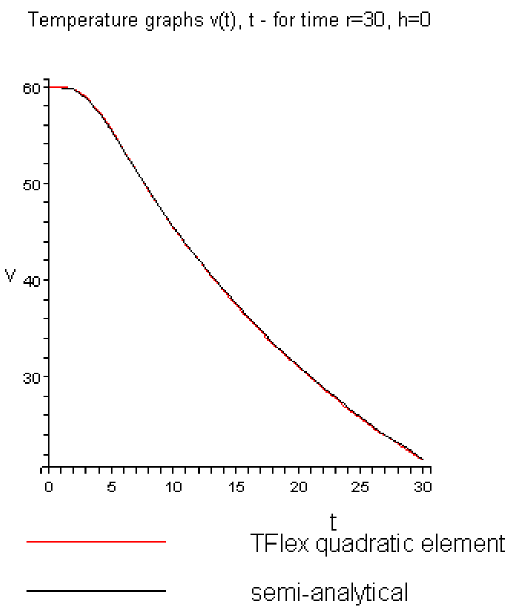
For time t=2 sec			
Control point number	1	2	3
Analytical solution, ^{0}C	59.9710	59.9710	59.7728
Numerical solution, ^{0}C	60.8509	60.8372	59.3652
Relative error, %	1.2	1.4	0.5
For time t=10 sec			
Control point number	1	2	3
Analytical solution, ^{0}C	46.7052	46.7052	45.5215
Numerical solution, ^{0}C	44.7267	44.8331	43.8220
Relative error, %	4.2	4.05	3.5
For time *t*=20 sec			
Control point number	1	2	3
Analytical solution, ^{0}C	29.5316	29.5316	31.1765
Numerical solution, ^{0}C	27.9843	28.2911	30.2286
Relative error, %	5.4	4.1	3.0

Table 4

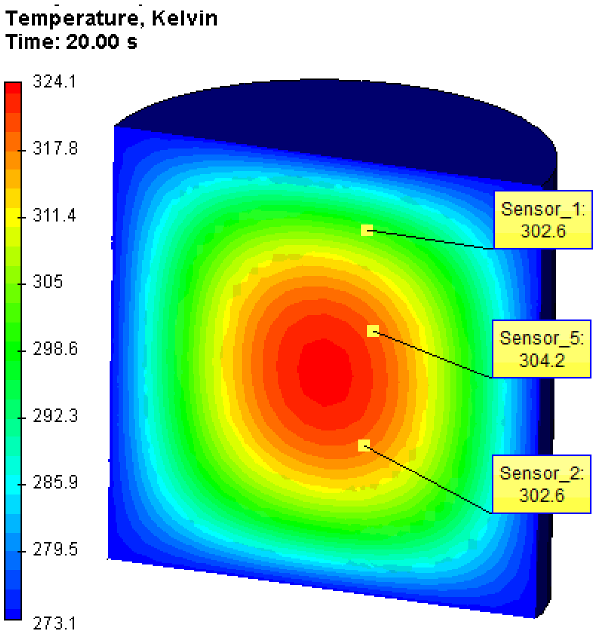
Calculation with quadratic elements

For time t=2 sec			
Control point number	1	2	3

Analytical solution, ^{0}C	59.9710	59.9710	59.7728
Numerical solution, ^{0}C	59.8491	59.8202	59.7568
Relative error, %	0.20	0.25	0.03
For time t=10 sec			
Control point number	1	2	3
Analytical solution, ^{0}C	46.7052	46.7052	45.5215
Numerical solution, ^{0}C	46.6162	46.6866	45.3993
Relative error, %	0.19	0.04	0.27
For time *t*=20 sec			
Control point number	1	2	3
Analytical solution, ^{0}C	29.5316	29.5316	31.1765
Numerical solution, ^{0}C	29.4206	29.4812	31.0185
Relative error, %	0.37	0.17	0.5



Calculation of temperature v in 0C, time t in sec, r and h – third control point (in mm)



View of the temperature field in 30 seconds

Conclusions:

For the given problem we obtained a realistic temperature field. The relative error of the numerical solution compared to the analytical solution did not exceed 5% for linear and 0.5% for quadratic element (on the time interval 20 sec). The method turned out to be effective for solution of the problems with complex geometry. The calculation error is stable in time and does not grow significantly when the computational time is increased. Plot of dependence of temperature on time shows that analytical and numerical solutions practically coincided.

For complex geometry the quadratic elements were proven to be superior to linear elements. However, it should be remembered that on the large intervals of time the quadratic elements accumulate larger error than linear elements during the last time layers.

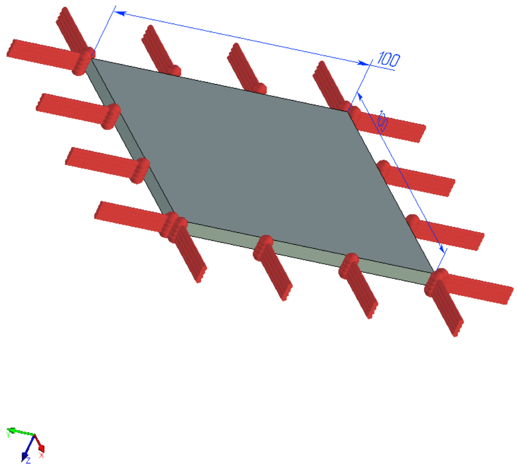
Non-Stationary Temperature Field in an Orthotropic Plate

Let us compare a two-dimensional problem of cooling of a graphite plate with orthotropic properties. The initial temperature of the plate is t0=60 0C. The temperature on the boundary of the plate is maintained equal to zero. The plate cools down during the period of 20 seconds. Let us determine the temperature field at the moments of time t1-4 = 2, 5, 10, 60 sec. Calculation of the temperature field will be conducted for control points 1-5 with coordinates (Xi,Yi):

i	1	2	3	4	5
X, mm	20	20	80	80	50
Y, mm	20	80	20	80	50

The material coordinate system of the body is chosen such that the origin of the coordinate system coincides with the angle of the plate, and the base direction is selected along the OY-axis.

Properties of the orthotropic plate: density p =2.5 g/cm3, specific heat c=840 J/(kg•0C), coefficients of thermal conductivity: K1=0.139 W/(mm • 0C) (along the OX-axis);x K2=0.278 W/(mm • 0C) (along the OY-axis – base direction). The plate has a rectangular shape 100 100 mm.



Numerical model with boundary conditions

Differential equation has the form:

$$K_1 \cdot \frac{\partial^2 u}{\partial x^2} + K_2 \cdot \frac{\partial^2 u}{\partial y^2} = c \cdot \rho \cdot \frac{du}{dt}$$

$$u(x, y, 0) = \varphi(x, y), t = 0, (x, y) \in \Omega$$

$$u(t, x, y) = 0, (x, y) \in \partial\Omega$$

where $\partial\Omega$ – the boundary of the numerical domain. Analytical solution of the problem has the form:

$$u(x, y, t) = \sum_{n=1}^{\infty}\sum_{m=1}^{\infty} A_{m,n} \cdot e^{\lambda(m,n) t/(\rho \cdot c)} \cdot \sin\left(\frac{\pi \cdot my}{l_y}\right) \sin\left(\frac{\pi \cdot nx}{l_x}\right)$$

where

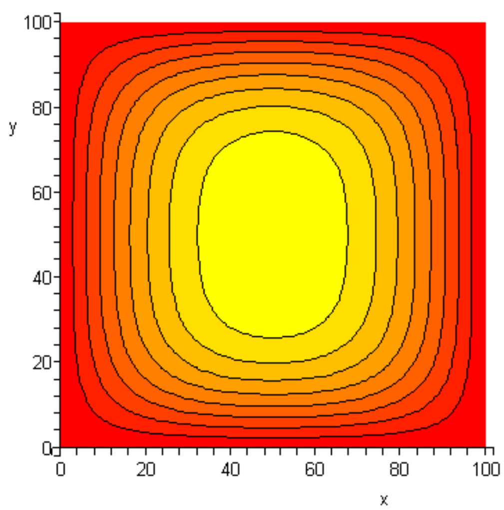
$$\lambda(m,n) = -\left(\frac{\pi \cdot m}{l_y}\right)^2 \cdot K_2 - \left(\frac{\pi \cdot n}{l_x}\right)^2 \cdot K_1$$

$$A_{m,n} = \frac{2}{l_x}\int_{x_0}^{x_1}\frac{2}{l_y}\int_{y_0}^{y_1} \varphi(x, y) \cdot \sin\left(\frac{\pi \cdot nx}{l_x}\right) \cdot \sin\left(\frac{\pi \cdot my}{l_y}\right) dxdy$$

$$l_x = x_1 - x_0,$$

$$l_y = y_1 - y_0$$

In our case x1=100, y1=100, x0=y0=0. We used n=m=60 terms in the series expansion.
The shape of the temperature field in 2 seconds is shown below:



View of the temperature field at the moment of time t1=2 sec (analytical solution)

As shown on the Figure, the field is stretched along the OY-axis. The same picture of the field on the surface of the plate can be observed in T-FLEX as well.
The system of coordinates on the picture corresponds to the system of coordinates on the surface of the plate.

Table 1

Parameters of finite element mesh

Finite element type	Number of mesh nodes used in calculation		Number of elements in a mesh	Relative error
	Linear elements	Quadratic element		
Tetrahedron, 4 nodes	2510	196132	132680	0.02

Table 2

Parameters of time discretization

Total calculation time (sec)	Time step (sec)	Number of time layers
60	0.5	120

Table 3

Table of control points:

Point number	1	**2**	**3**	**4**	**5**
X, mm	20	20	80	80	50
Y, mm	20	80	20	80	50

At the moments of time: 2, 10, 20 sec we will look at the value of the temperature.
The number of significant digits of the analytical solution is equal to 6.
Temperatures at control points in 0C are given below.

Table 4

Calculation with linear element

For time t=2 sec					
Point number	1	2	3	4	5
Analytical solution, ^{0}C	28.8050	28.8050	28.8050	28.8050	56.1856
Numerical solution, ^{0}C	28.4692	28.7457	28.5992	28.4386	56.018
Relative error, %	1.16577	0.20586	0.714459	1.272001	0.298297
For time t=5 sec					

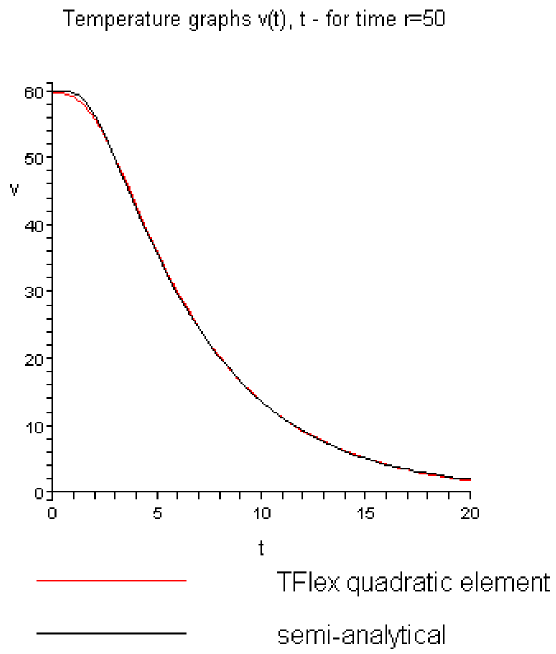
Point number	1	2	3	4	5
Analytical solution, ^0C	13.1503	13.1503	13.1503	13.1503	35.5560
Numerical solution, ^0C	12.8375	12.7288	12.7474	12.8277	35.5369
Relative error, %	2.37865	3.20525	3.063808	2.453176	0.053718
For point t=10 sec					
Point number	1	2	3	4	5
Analytical solution, ^0C	4.74820	4.74820	4.74824	4.74824	13.6788
Numerical solution, ^0C	4.61898	4.58004	4.58856	4.613	13.3372
Relative error, %	2.72145	3.54155	3.36293	2.848213	2.497295

Table 5

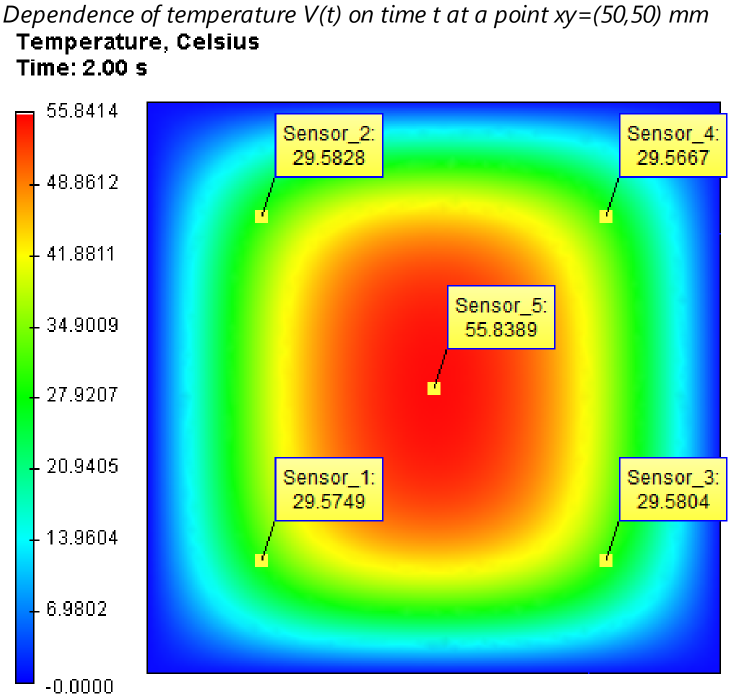
Calculation with quadratic element

For time t=2 c					
Point number	1	2	3	4	5
Analytical solution, ^0C	28.8050	28.8050	28.8050	28.8050	56.1856
Numerical solution, ^0C	29.6266	29.6242	29.6232	29.6237	55.8254
Relative error, %	2.85228	2.84395	2.840479	2.842215	0.64109
For time t=5 sec					
Point number	1	2	3	4	5
Analytical solution, ^0C	13.1503	13.1503	13.1503	13.1503	35.5560
Numerical solution, ^0C	13.1271	13.127	13.1269	13.1274	35.8344
Relative error, %	0.17642	0.17718	0.177943	0.174141	0.78299
For time t=10 sec					
Point number	1	2	3	4	5
Analytical solution, ^0C	4.74820	4.74820	4.74824	4.74824	13.6788
Numerical solution, ^0C	4.74935	4.74938	4.74935	4.74954	13.7008
Relative error, %	0.02422	0.02485	0.023377	0.027379	0.160833

For time 60 sec. the results are not shown since the plate has practically cooled down to 0 0C.



Dependence of temperature V(t) on time t at a point xy=(50,50) mm



View of the temperature field at the moment of time t1=2 sec (from the results of finite element analysis)

Conclusions:

For the given problem we obtained a realistic picture of the field. The relative error of the numerical solution compared to the analytical solution did not exceed 3.5% for linear element and 3% for quadratic element (on the time interval 20 sec). The calculation error is stable in time and does not grow significantly when the computational time is increased. Plot of dependence of temperature on time shows that analytical and numerical solutions practically coincided.

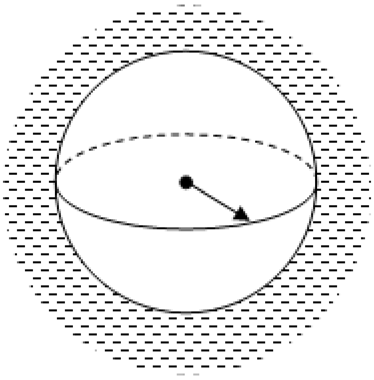
As was already shown in the example with the heat flux on the surface of the sphere: for calculation over the large intervals of time it is preferable to use linear elements since because of the smaller number of nodes the error on the time layers is accumulated slower.

In our case thermal conductivities along the main axes of the plate are sufficiently high which implies that it will cool down very fast. That is the reason why the quadratic elements on sufficiently small intervals of time give more accurate results. The error of the solution for the non-stationary problems does not exceed 3.5 % for FE with linear elements and 3 % with quadratic elements.

**Remark – for calculation on the time interval 20 sec, change the calculation time.*

Sphere with Variable Thermal Conductivity, which is Heated from the Center

Let us consider a sphere in the center of which a point source of heat with power r exists and convective heat exchange with the environment of T temperature takes place. Convective heat exchange factor α does not depend on temperature. Thermal conductivity factor depends on the temperature: $\lambda(u) = \lambda_1(1 + \lambda_2 u)$, where u - is a temperature in the sphere point. Let us find stationary temperature distribution in the sphere.



Computation scheme

According to the symmetry, the temperature changes only in the radial direction. Let us write down a statement of the study.

Differential equation:

$$\frac{1}{r^2}\frac{d}{dr}\left[\lambda r^2 \frac{du}{dr}\right] = -\,q(r),$$

Boundary conditions (convective heat exchange):

$$r = R:\ \ \lambda\frac{du}{dr} + \alpha[u - T] = 0$$

Here $q(r)$ - volumetric density of the heat sources power.

In the case of constant thermal conductivity factor λ and convective heat exchange α:

$$u(r) = \frac{1}{\lambda}\int_r^R \left\{\frac{1}{\eta^2}\int_0^\eta \xi^2 q(\xi)d\xi\right\}d\eta + \frac{1}{\alpha R^2}\int_0^R \xi^2 q(\xi)d\xi + T,$$

here x and h are integration variables.

In this case, the point source of heat can be presented using density of heat sources using d-Dirac function: $q(r) = \rho\delta(r)$, where $\delta(r) = \begin{cases} 0, & r \neq 0 \\ \infty, & r = 0 \end{cases}$

Moreover

$$\iiint_{V_r} \delta dV = 4\pi\int_0^r \xi^2 \delta(\xi) d\xi = 1.$$

Here the integration area Vr - sphere of r radius.

Then the stationary field of temperatures in the sphere with a point source of heat power and with constant factors of thermal conductivity and convective heat exchange will look like:

$$u(r) = \frac{\rho}{4\pi}\left[\frac{1}{\lambda}\left(\frac{1}{r} - \frac{1}{R}\right) + \frac{1}{\alpha R^2}\right] + T$$

Let us return to the case with a linear dependence of the thermal conductivity on temperature. Let us rewrite the study statement:

differential equation:

$$\frac{1}{r^2}\frac{d}{dr}\left[r^2\frac{dv}{dr}\right] = -\frac{2\lambda_2}{\lambda_1}q(r),$$

boundary conditions (convective heat exchange):

$$r = R: \quad \frac{\lambda_1}{2\lambda_2}\frac{dv}{dr} + \alpha\left[\frac{\sqrt{v} - 1}{\lambda_2} - T\right] = 0$$

where $v(r) = \left(1 + \lambda_2 u(r)\right)^2 = \left[\frac{\lambda(u)}{\lambda_1}\right]^2$.

The solution of this study will be the function:

$$v(r) = \frac{2b}{\lambda_0}\int_r^R \left\{\frac{1}{\eta^2}\int_0^\eta \xi^2 q(\xi)d\xi\right\} d\eta + \left[\frac{b}{\alpha R^2}\int_0^R \xi^2 q(\xi)d\xi + Tb + 1\right]^2.$$

The function of stationary temperature:

$$u(r) = \frac{\sqrt{v(r)} - 1}{\lambda_2}.$$

In the case of a point source of heat:

$$v(r) = \frac{\lambda_2\rho}{2\pi\lambda_1}\left(\frac{1}{r} - \frac{1}{R}\right) + \left[\frac{\lambda_2\rho}{4\pi R^2\alpha} + T\lambda_2 + 1\right]^2.$$

For the calculations we accept initial data:

$\rho = 500\,W \rho = 500\,W$ - concentrated heat power;

$\lambda_1 = 87{,}77\ \frac{W}{m \cdot °C}$, $\lambda_2 = -1.103 \cdot 10^{-3}\frac{1}{°C}$ - factors that define thermal conduction (Steel 20);

R=0.1 m - radius of the sphere;

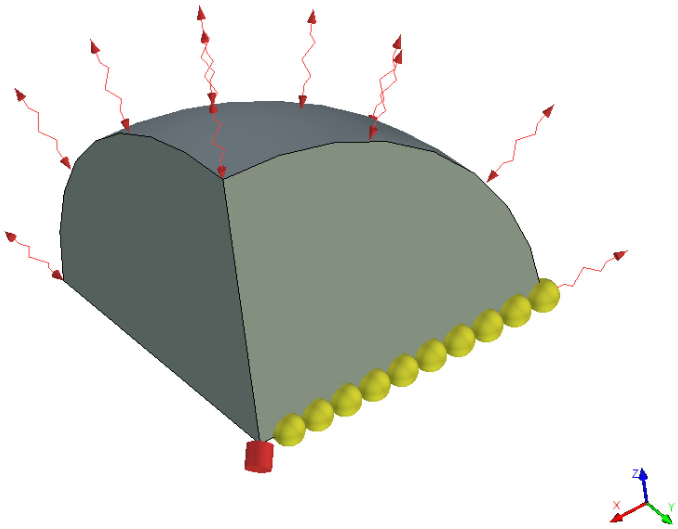
T=20 °C - temperature of the environment;

$\alpha = 1000\ \frac{W}{m^2 \cdot °C}$ - factor of the convective heat exchange (water)

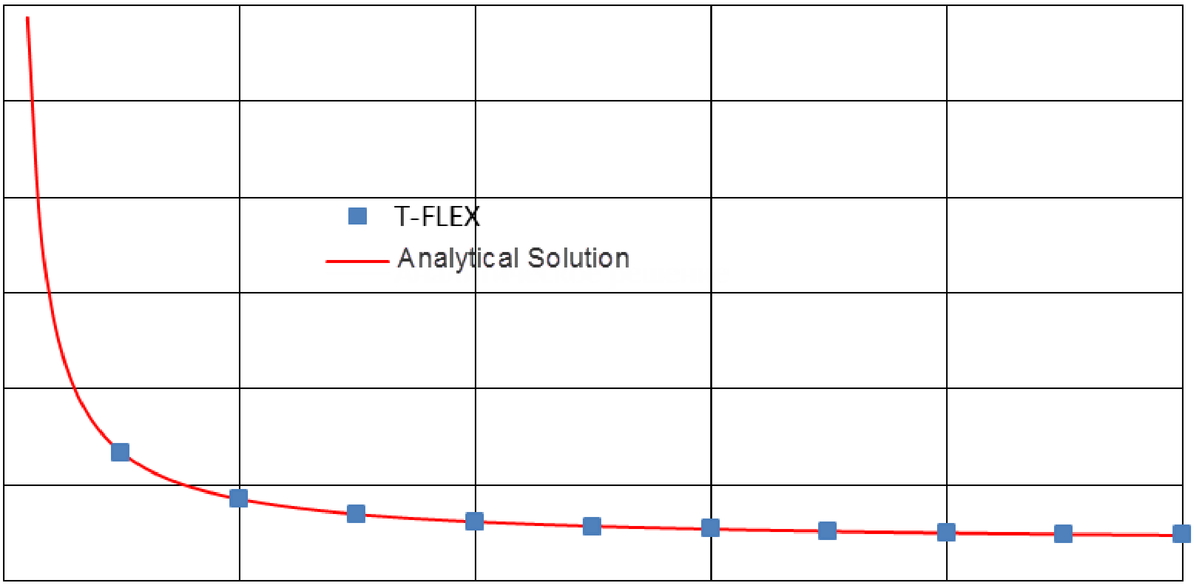
Let us consider 1/8 of the sphere for the modeling. In the calculation model, the applied power of 500 W is divided by 8, i.e. 500/8 = 62.5 W.

Parameters of division are given in the table

Type of finite element	Number of mesh nodes	Number of finite elements	Relative size
Tetrahedron, linear	5399	24481	0.06
Tetrahedron, quadratic	36785	24481	0.06


Calculation model in T-FLEX

Graph of temperature distribution in radial direction for analytical solution and temperature in 10 points, found using T-FLEX Analysis are shown on the picture.



Dependence of temperature u (°C) from the distance to the center of the sphere r (m)

Numerical values of the temperature in 10 points, taken from the analytical solution and results of calculation in T-FLEX Analysis are presented in the table.

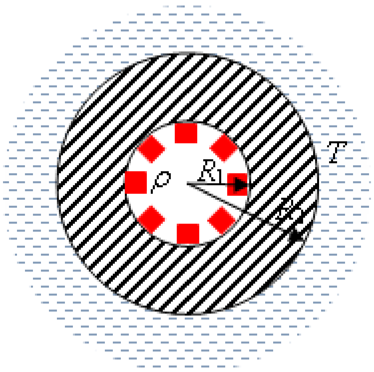
r, m	0.01	0.02	0.03	0.04	0.05	0.06	0.07	0.08	0.09	0.1
Analytics	66.93	42.81	34.91	30.99	28.65	27.09	25.98	25.14	24.5	23.98
Linear element	66.33	43.08	34.82	30.97	28.64	27.09	25.98	25.15	24.5	23.98
Relative error, %	0.9	0.63	0.26	0.06	0.03	0	0	0.04	0	0
Quadratic element	67.23	42.82	34.91	30.99	28.65	27.09	25.98	25.15	24.5	23.98
Relative error, %	0.45	0.02	0	0	0	0	0	0.04	0	0

Conclusion:

From the comparison of the results it is clear that the temperature field is approximated by a linear element rather accurately (maximum error 0.9%). The error increases when approaching the center of the sphere. This is because the analytic solution is singular (equal to infinity) in the center. On the boundary of the sphere, the temperature is equal to 23.98 °C, which is slightly higher than the temperature of the medium, equal to 20 °C.

Hollow Sphere with Variable Thermal Conductivity, Heated from Within

Let us consider a hollow sphere with inner radius R1 and outer radius R2. At the inner boundary, heat sources of power r uniformly distributed over the area of the sphere of radius R1 (or heat flux $q = \rho/\left(4\pi R_1{}^2\right)$) influence, and on the outer boundary convective heat exchange occurs with the environment with temperature T. Convective heat exchange factor α does not depend on temperature.



Computation scheme

Thermal conductivity factor depends on the temperature: $\lambda(u) = \lambda_1(1 + \lambda_2 u)$, where u - temperature in the sphere point. Let us find stationary temperature distribution in the sphere.

According to the symmetry, the temperature changes only in radial direction. Let us write down the statement of the study:

Differential equation:

$$\frac{d}{dr}\left[\lambda r^2 \frac{du}{dr}\right] = 0,$$

Boundary condition on the inner surface of the sphere (specified heat flow):

$$r = R_1: \ \lambda \frac{du}{dr} + q = 0.$$

Boundary conditions on the outer surface of the sphere (convective heat exchange):

$$r = R_2: \ \lambda \frac{du}{dr} + \alpha[u - T] = 0.$$

In the case of constant thermal conductivity factors λ and convective heat exchange α (linear solution):

$$u_{linear}(r) = \frac{qR_1^{\ 2}}{\lambda}\left[\frac{1}{r} - \frac{1}{R_2} + \frac{\lambda}{\alpha}\frac{1}{R_2^{\ 2}}\right] + T$$

If the thermal conductivity factor linearly depends on temperature the statement of the study can be rewritten as follows:

Differential equation:

$$\frac{d}{dr}\left[r^2\frac{dv}{dr}\right] = 0,$$

Boundary condition on the inner surface of the sphere (specified heat flow):

$$r = R_1: \ \frac{\lambda_0}{2b}\frac{dv}{dr} + q = 0.$$

Boundary conditions on the outer surface of the sphere (convective heat exchange):

$$r = R_2: \ \frac{\lambda_1}{2\lambda_2}\frac{dv}{dr} + \alpha\left[\frac{\sqrt{v} - 1}{\lambda_2} - T\right] = 0,$$

where $v(r) = \left(1 + \lambda_2 u(r)\right)^2 = \left[\frac{\lambda(u)}{\lambda_1}\right]^2$.

The solution of this study will be the function:

$$v(r) = -\frac{C_1}{r} + C_2,$$

Where

$$C_1 = -\frac{2q\lambda_2 R_1^{\ 2}}{\lambda_1}, \quad C_2 = \left[1 - \frac{\lambda_1}{2\alpha}\frac{C_1}{R_2^{\ 2}} + \lambda_2 T\right]^2 + \frac{C_1}{R_2}.$$

The function of the stationary temperature (non-linear solution):

$$u_{nonlinear}(r) = \frac{\sqrt{v(r)} - 1}{\lambda_2}.$$

We accept initial data for the calculations:

$\rho = 3000\,W$ - heat power, which is uniformly distributed over the inner spherical boundary;

$\lambda_1 = 87{,}77 \frac{W}{m \cdot {}^{\circ}C}$, $\lambda_2 = -1.103 \cdot 10^{-3} \frac{1}{{}^{\circ}C}$ - factors, which define variable thermal conductivity (for non-linear solution) (steel 20);

$\lambda = 54.60 \frac{W}{m \cdot {}^{\circ}C}$ - thermal conductivity factor for a linear solution;

R1=10 mm, R2=30 mm – inner and outer radius of the sphere;

T=20 $^{\circ}C$ - temperature of the environment;

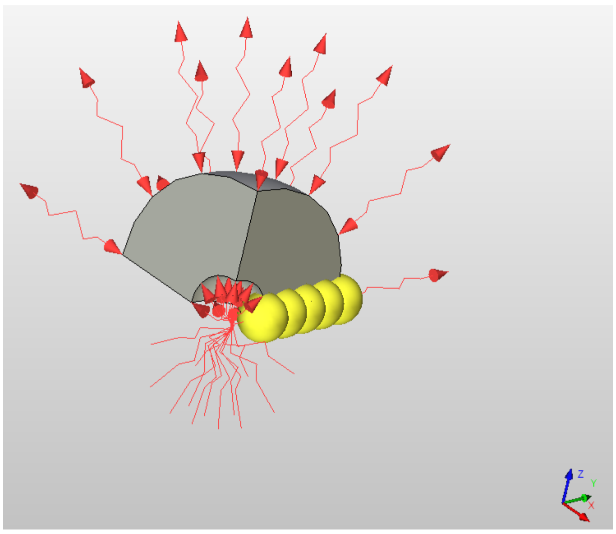
$\alpha = 1500 \dfrac{W}{m^2 \cdot {}^\circ C}$ - convective heat exchange factor.

Let us consider 1/8 of the sphere for the modeling. In the calculation model the applied power of 3 kW is divided by 8, i.e. 3000/8 = 62.5 W. Density of the heat flow equivalent to the specified power

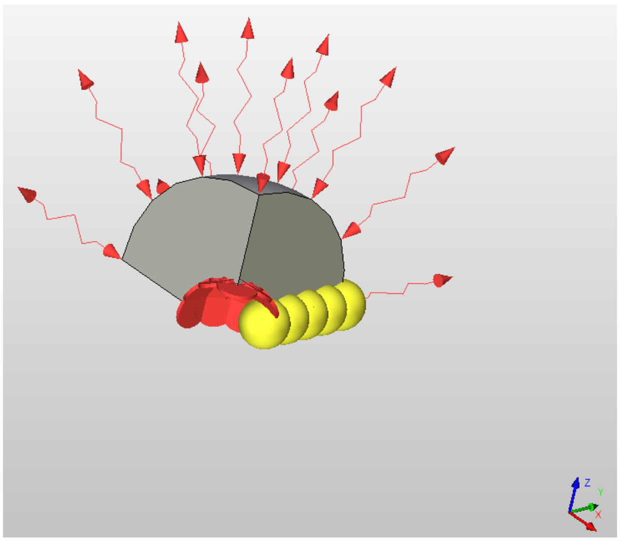
$$q = \rho / \left(4\pi R_1{}^2\right) = 2{,}387 \cdot 10^6\ W/m^2$$

Parameters of division are given in the table

Type of finite element	Number of mesh nodes	Number of finite elements	Relative size
Tetrahedron, linear	5229	23512	0.06
Tetrahedron, quadratic	35512	23512	0.06



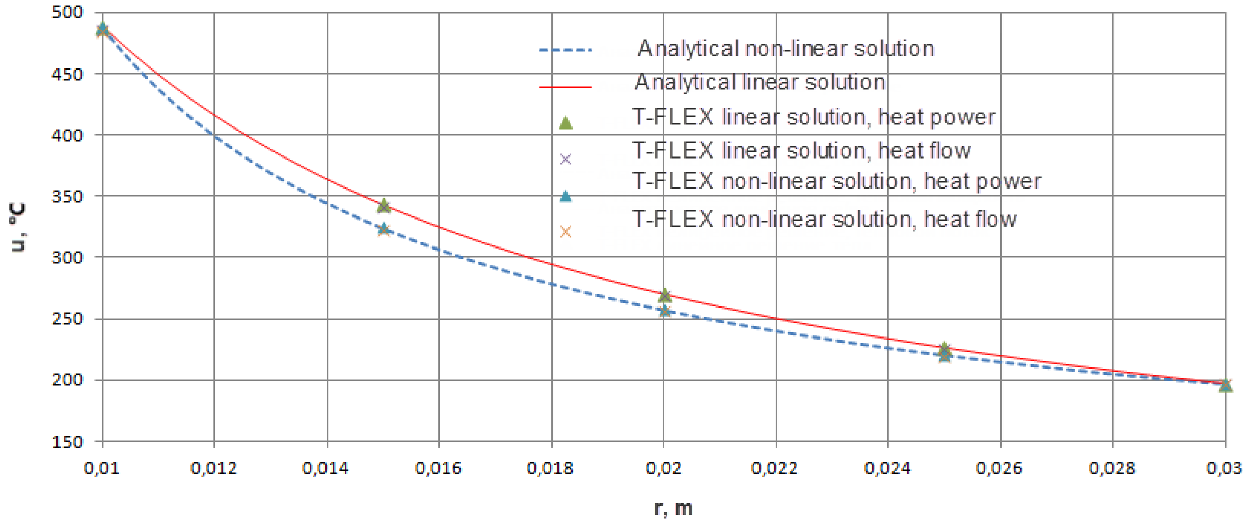
a) heat flow is specified



b) heat power is specified

Computation scheme

Graphs of temperature distribution in the radial direction for analytical solution and temperature in 5 points, found using T-FLEX Analysis are shown on the picture. Two solutions are presented here: thermal conductivity factor does not depend on the temperature (linear solution) and thermal conductivity factor depends on the temperature linearly (non-linear solution). Two equivalent variants of applying a thermal load to the inner boundary of the sphere were used in T-FLEX Analysis: a) heat power and b) heat flow. Linear (4-node) finite element was used when creating graphs.



Dependence of temperature u (°C) from the distance to the center of the sphere r (m)

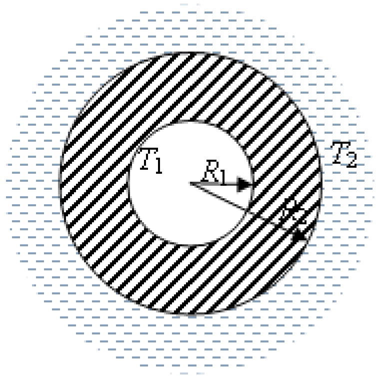
Numerical values of the temperature in 5 points, taken from the analytical solution and results of calculation in T-FLEX Analysis are presented in the table. Here, δ_{max} – maximal relative error of T-FLEX Analysis solutions for five points in comparison with the corresponding analytical solution.

Conclusions:

- Maximal relative deviation of non-linear solution from linear is 5,4%. Moreover, the study parameters were chosen so that this deviation was as much as possible. With other reasonable parameters of the study, the linear and nonlinear solutions practically coincide. Consequently, in the majority of real problems, it makes no sense to take into account the dependence of the thermal conductivity on temperature. It can be considered constant and equal to its average value in the range of the reviewed study temperatures.
- Temperatures calculated in the T-FLEX Analysis model using heat power were closer to the analytical solution in comparison with the temperatures in the model that defines heat flow both for linear and for nonlinear solutions. It can be explained by the fact that the area of the inner boundary approximated by finite elements is less than the real. Consequently, in the integral sense, the boundary condition on the inner sphere for a model with thermal power is fulfilled exactly, and for the model with a heat flux, it is fulfilled approximately. In addition, even when a linear element is replaced by a quadratic in a model with a heat flux, the results do not reach the accuracy obtained in the model with a thermal power with a linear element.

Hollow Sphere, which is Heated from within, with Variable Thermal Conductivity and Convection

Let us consider a hollow sphere with inner radius R1 and outer radius R2. Constant temperature T1 is specified on the inner boundary, convective heat exchange with the environment at temperature T2 occurs on the outer boundary.



Computation scheme

Convective heat exchange factor α linearly depends on the average temperature of the boundary layer $\tau = \frac{1}{2}(u(R_2) + T_2)$, $\alpha(\tau) = \alpha_1(1 + \alpha_2\tau)$, where $u(R_2)$ - wall temperature, α_1 and α_2 - constant values. Thermal conductivity factor depends on the temperature: $\lambda(u) = \lambda_1(1 + \lambda_2 u)$, where u - temperature in the sphere point, λ_1 and λ_2 - constant values. Let us find stationary temperature distribution in the sphere.

According to the symmetry, the temperature changes only in radial direction. Let us write down the statement of the study:

Differential equation:

$$\frac{d}{dr}\left[\lambda r^2 \frac{du}{dr}\right] = 0,$$

Boundary condition on the inner surface of the sphere (specified temperature):

$$r = R_1: \ u = T_1$$

Boundary conditions on the outer surface of the sphere (convective heat exchange):

$$r = R_2: \ \lambda \frac{du}{dr} + \alpha[u - T_2] = 0.$$

Let us consider particular cases of the study.

1. Thermal conductivity λ and convective heat exchange α factors are constant (linear study)

The solution appears as:

$$u_1(r) = \frac{\alpha(T_2 - T_1)R_1R_2}{\lambda\frac{R_1}{R_2} + \alpha(R_2 - R_1)}\left(\frac{1}{R_1} - \frac{1}{r}\right) + T_1.$$

2. The thermal conductivity factor λ is constant and the convective heat exchange factor α depends on the average temperature of the boundary layer linearly.

An overall solution of the differential equation appears as:

$$u_2(r) = -\frac{C_1}{r} + C_2.$$

Constants of integration are determined by the boundary conditions, which can be reduced to the quadratic equation with respect to C_1:

$$aC_1^2 + bC_1 + c = 0,$$

where

$$a = \frac{\alpha_1\alpha_2}{2}\left(\frac{1}{R_1} - \frac{1}{R_2}\right)^2, \qquad b = \frac{\lambda}{R_2^2} + \alpha_1\left(\frac{1}{R_1} - \frac{1}{R_2}\right)(1 + \alpha_2 T_1),$$

$c = \alpha_1\left[T_1 - T_2 + \frac{\alpha_2}{2}\left(T_1{}^2 - T_2{}^2\right)\right].$

Then

$C_1 = \frac{-b + \sqrt{b^2 - 4ac}}{2a},\ C_2 = T_1 + \frac{C_1}{R_1}.$

The study can be solved differently, avoiding substituting the general solution into a nonlinear boundary condition. The solution is obtained using the iterative method:

An initial approximation of the convective heat exchange factor is specified: $\alpha^{(0)} = \alpha(T_2)$;

Temperature of the wall on the first iteration if determined from the linear solution: $T_{\text{ст}}^{(1)} = u_1(R_2)$ at $\alpha = \alpha^{(0)}$;

Convective heat exchange factor is calculated on the first iteration $\alpha^{(1)} = \alpha(\tau^{(1)})$, where $\tau^{(1)} = \frac{1}{2}\left(T_{\text{ст}}^{(1)} + T_2\right)$;

Repeat these steps for the i-th iteration:

$T_{\text{ст}}^{(i)} = u_1(R_2)\big|_{\alpha = \alpha^{(i-1)}},\ \tau^{(i)} = \frac{1}{2}\left(T_{\text{ст}}^{(i)} + T_2\right),\ \alpha^{(i)} = \alpha(\tau^{(i)});$

The iteration process is stopped when $\left|T_{\text{ст}}^{(i)} - T_{\text{ст}}^{(i-1)}\right| < \varepsilon$, where ε - specified maximum permissible absolute error.

Generally, for $\varepsilon = 0{,}1$ °C six iterations are sufficient.

3. Thermal conductivity factor λ linearly depends on temperature and the convective heat exchange factor α is constant.

In the case, the statement of the study can be rewritten as follows:

Differential equation:

$$\frac{d}{dr}\left[r^2\frac{dv}{dr}\right] = 0,$$

Boundary condition on the inner surface of the sphere (specified temperature):

$$r = R_1:\ \ v = [1 + \lambda_2 T_1]^2,$$

Boundary conditions on the outer surface of the sphere (convective heat exchange):

$$r = R_2:\ \ \frac{\lambda_1}{2\lambda_2}\frac{dv}{dr} + \alpha\left[\frac{\sqrt{v} - 1}{\lambda_2} - T_2\right] = 0,$$

where $v(r) = \left(1 + \lambda_2 u(r)\right)^2 = \left[\frac{\lambda(u)}{\lambda_1}\right]^2$.

The general solution of the differential equation:

$$v(r) = -\frac{C_1}{r} + C_2,$$

Constants of integration are determined by substitution of the general solution in the boundary conditions that leads to the square equation with regard to C_1: $aC_1{}^2 + bC_1 + c = 0$. Here:

$$a = \left[\frac{\lambda_1}{2\alpha}\frac{1}{R_2{}^2}\right]^2, \qquad b = \frac{1}{R_2} - \frac{1}{R_1} - \frac{\lambda_1}{\alpha}\frac{1}{R_2{}^2}(1 + \lambda_2 T_2),$$

$c = \lambda_2(T_2 - T_1)\big(2 + \lambda_2(T_1 + T_2)\big).$

Then

$$C_1 = \frac{-b - \sqrt{b^2 - 4ac}}{2a},\ C_2 = \frac{C_1}{R_1} + (1 + \lambda_2 T_1)^2.$$

The temperature is expressed using the function $v(r)$:

$$u_3(r) = \frac{\sqrt{v(r)} - 1}{\lambda_2}.$$

4. The thermal conductivity factor λ linearly depends on the sphere temperature and the convective heat exchange factor α linearly depends on the average temperature of the boundary layer.

A solution of the study can be received by an iterative method that is described in item 2 if we replace the function $u_1(r)$ by the function $u_3(r)$.

For the calculations, we accept initial data:

$T_1 = 200\ °C$ - temperature on the inner spherical boundary;

$T_2 = 20\ °C$ - temperature of the environment;

$\lambda_1 = 87{,}77\ \frac{W}{m \cdot °C}$, $\lambda_2 = -1.103 \cdot 10^{-3}\ \frac{1}{°C}$ - parameters that define variable thermal conductivity factor in the studies, which consider dependence of λ from temperature (steel 20);

$\lambda = 77.12\ \frac{W}{m \cdot °C}$ - thermal conductivity factor in the studies where λ not depends on the temperature (considered equal to the thermal conductivity factor of the material with the temperature equal to $(T_1 + T_2)/2$);

$\alpha_1 = 100\ \frac{W}{m^2 \cdot °C}$, $\alpha_2 = 0.1\ \frac{1}{°C}$ - parameters that specify variable thermal conductivity factor in the studies, which do not consider dependence of α from the average temperature of the boundary layer;

$\alpha = 750\ \frac{W}{m^2 \cdot °C}$ - convective heat exchange factor, where α does not depend on temperature (assumed equal to the convective heat exchange factor at an average temperature of the boundary layer equal to $(T_1 + 3T_2)/4$).

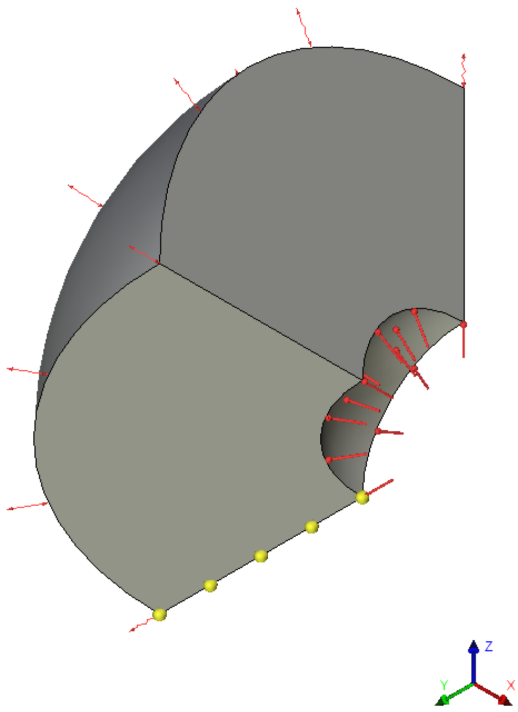
R1=100 mm, R2=300 mm – inner and outer radius of the sphere;

There were ten iterations for the function of temperature of the fourth study ($\alpha^{(10)} = 485.05\ \frac{W}{m^2 \cdot °C}$) (variable thermal conductivity and convection)

Let us consider 1/8 of the sphere for the modeling.

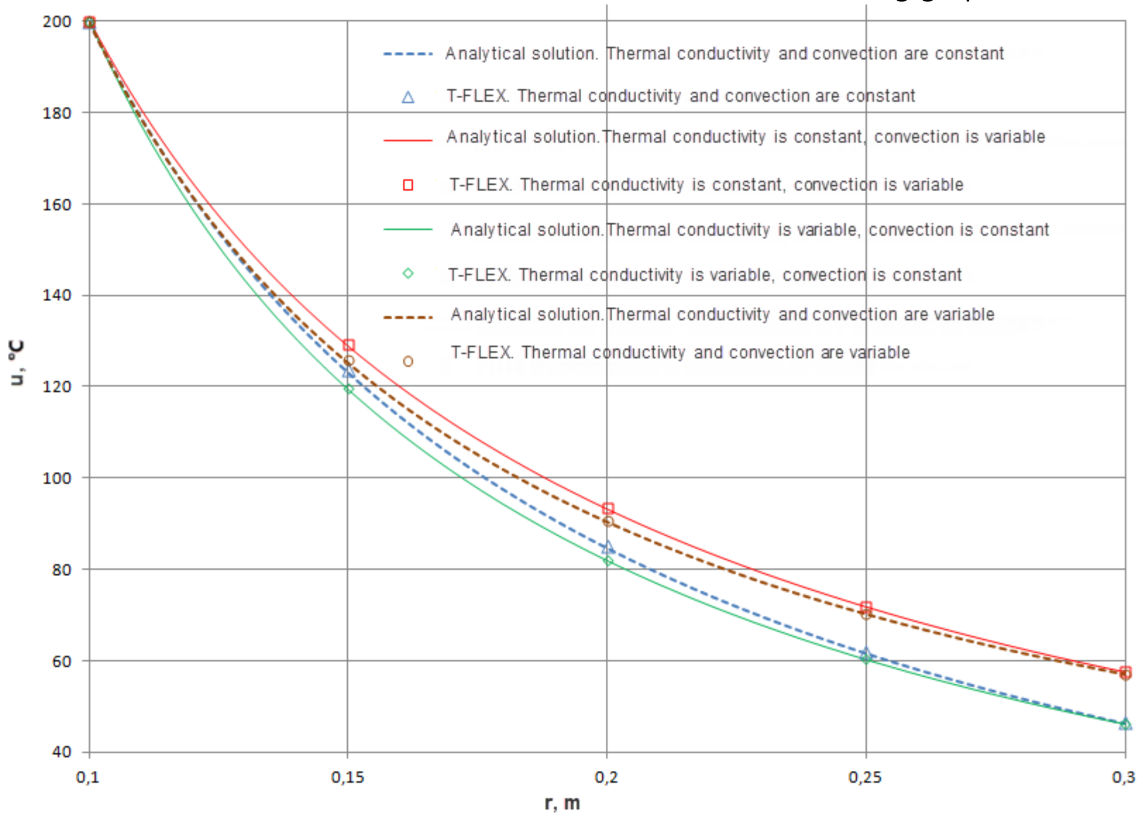
Parameters of division are given in the table

Type of finite element	Number of mesh nodes	Number of finite elements	Relative size
Tetrahedron, linear	5228	23502	0.06
Tetrahedron, quadratic	35500	23502	0.06



Calculation model in T-FLEX

Graphs of temperature distribution in the radial direction for analytical solution and temperature in 5 points, found using T-FLEX Analysis are shown on the picture. Four studies that described above are presented here. Linear (4-node) finite element was used when creating graphs.



Dependence of temperature u (° C) from the distance to the center of the sphere r (m)

Numerical values of the temperature in 5 points, taken from the analytical solution and results of calculation in T-FLEX Analysis are presented in the table. Here, δ max – the maximum relative error by five points.

T-FLEX Analysis in comparison with the corresponding analytical solution.

r, m	Temperature, °C											
	α=const, λ=const			$\alpha\neq$ const, λ=const			α=const, $\lambda\neq$ const			$\alpha\neq$ const, $\lambda\neq$ const		
	analytical solution	T-FLEX		analytical solution	T-FLEX		analytical solution	T-FLEX		analytical solution	T-FLEX	
		Linear element	Quadratic element		Linear element	Quadratic element		Linear element	Quadratic element		Linear element	Quadratic element
0.1	200	200	200	200	200	200	200	200	200	200	200	200
0.15	123.17	123.67	122.82	128.77	129.22	128.46	119.21	119.76	118.87	125.23	125.71	124.9
0.2	84.751	85.058	84.517	93.157	93.42	92.953	81.774	82.126	81.568	90.405	90.685	90.193
0.25	61.701	61.949	61.544	71.789	71.993	71.659	60.108	60.384	59.974	70.208	70.412	70.064
0.3	46.335	46.497	46.229	57.543	57.667	57.464	45.967	46.15	45.875	57.01	57.128	56.911
δ max, %	-	0.4081	0.2845	-	0.3505	0.2459	-	0.4649	0.2809	-	0.387	0.2598

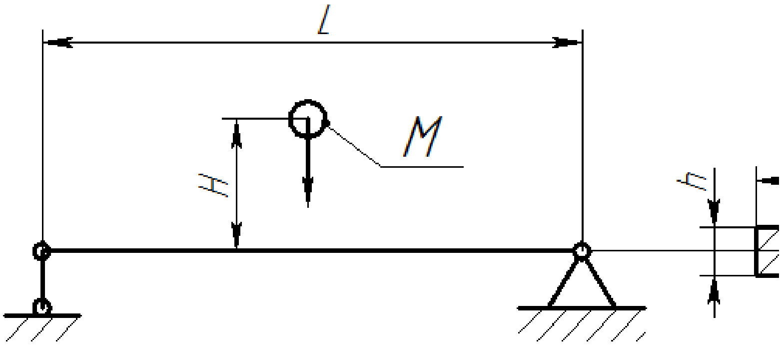
Conclusions:

1. The results show that the dependence of thermal conductivity factor from temperature has no significant effect on the temperature: the maximum difference between solutions of the second ($\alpha\neq$ const, $\lambda\neq$ const) and the fourth ($\alpha\neq$ const, λ=const) studies is 3,5 °C. On the other hand, the effect of the convective heat exchange factor from temperature is more noticeable: the maximum difference between solutions of the fourth ($\alpha\neq$ const, $\lambda\neq$ const) and the third (α=const, $\lambda\neq$ const) studies is 11 °C.
2. The maximal relative error of the calculation results in T-FLEX Analysis of the four considered studies is 0,46% according to the corresponding analytical solutions. This indicates high accuracy of the calculation.

Examples of Dynamic Studies Calculations

Bending of a Simply -supported Beam Under Impact in the Middle

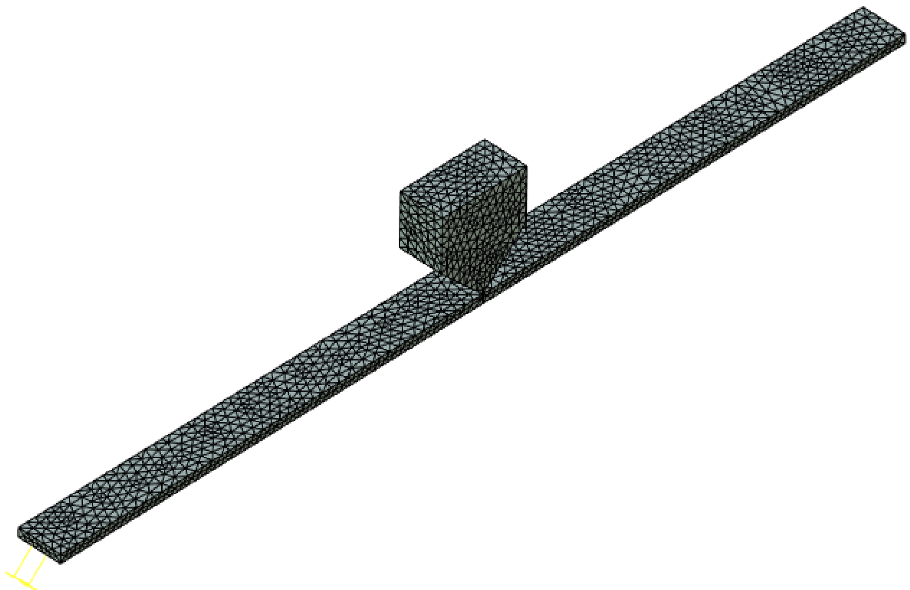
Let us consider a simply-supported beam of L length on which the burden of M mass falls from the H height and impacts in the middle. Beam cross-section is a rectangle of b width and h height. We can neglect the beam mass.



It is necessary to define the beam maximum bending.

Let us consider L=1000 mm, b=50 mm, h=10 mm. The material properties are set by default. Modulus of elasticity E = 2.1E+011 Pa, Poisson ratio v = 0.28. The mass of burden is M = 50,85 kg. The height of burden fall is H = 5 mm.

Create Transitional processes study. The displacement of the lower edge of the left butt end is restricted by Z axis, the lower edge is fixed for the right butt. The load Acceleration of gravity is applied to the burden (as the beam mass can be neglected). The initial velocity of 313.209 m/s that corresponds to 5 mm of fall height is applied to the burden.



Calculated model with loads and restraints

Finite-element model of the beam with loads and restraints

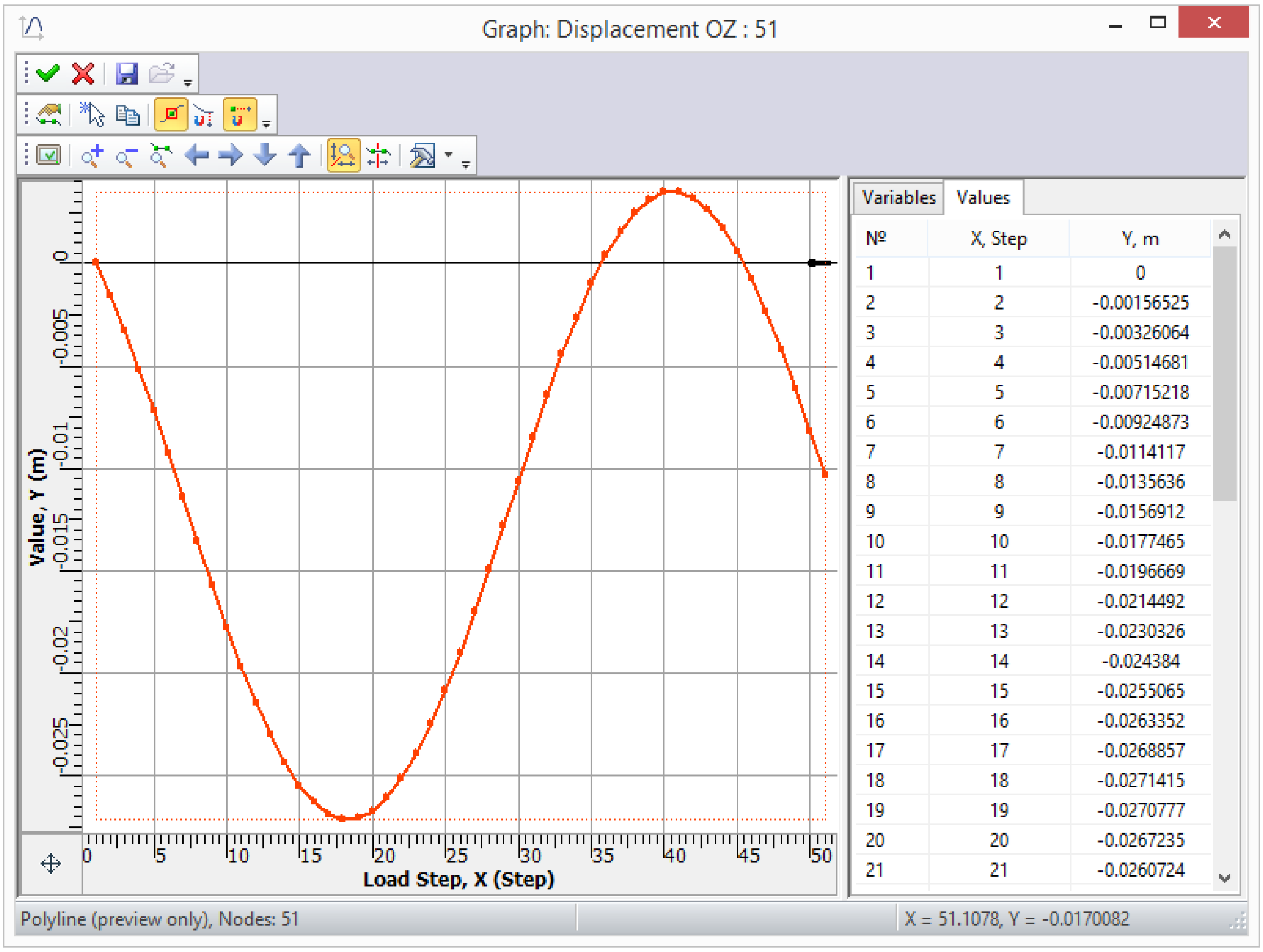
An analytical solution is the following:

$$w = w_{st} + \sqrt{w_{st}^2 + \frac{V_0^2 w_{st}}{g}} = 27{,}997\,mm \quad ,$$

$$w_{st} = \frac{Mgl^3}{48EJ} = 11{,}877\,mm$$

where $J = \frac{b \cdot h^3}{12}$; - a moment of inertia of the section.

Let us find the step on which the maximum deviation by graph appears after calculation using T-FLEX Analysis:



The following results are obtained at the current time step:

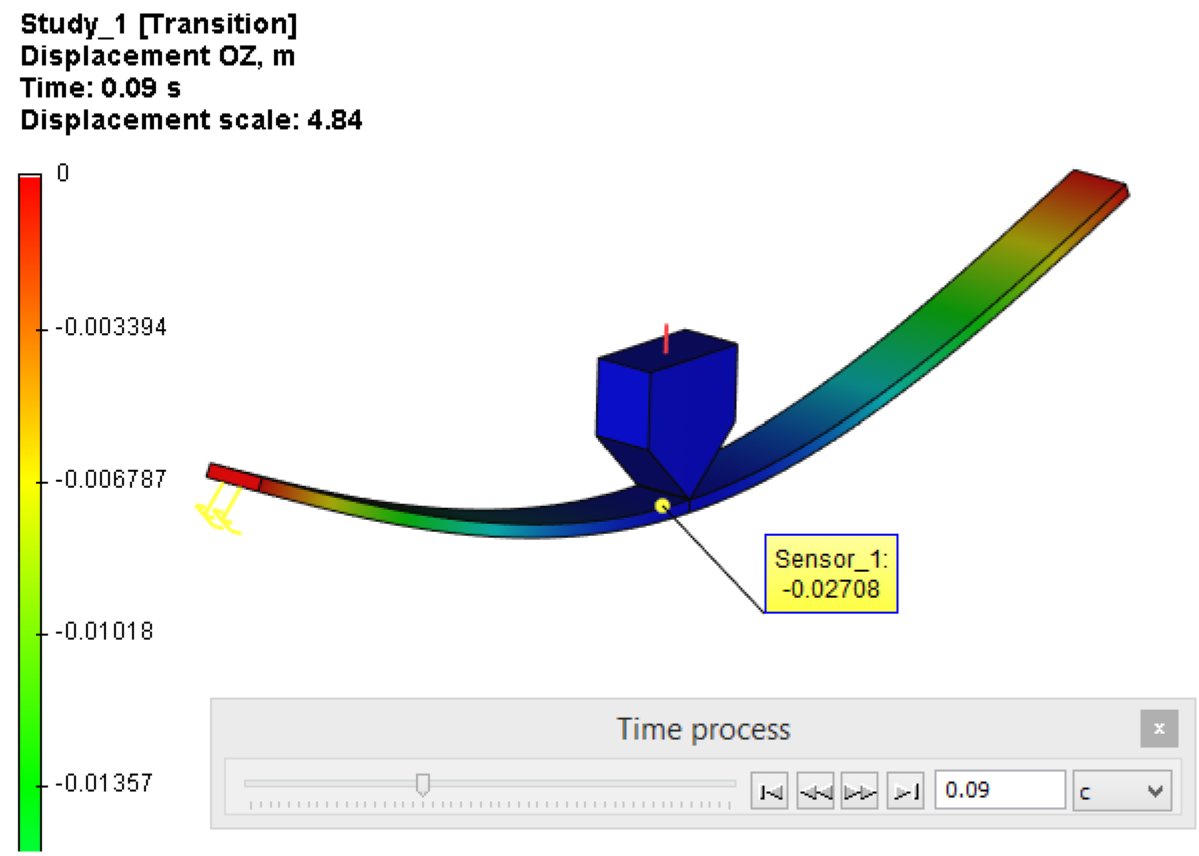


Table 1.
Parameters of finite-element mesh

Type of finite elements	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	2946	9857

Table 2. Parameters of temporal discretization

Total calculation time (s)	Time step (s)	Number of time layers
0.25	0.005	50

Table 3. "Displacement Z" result

Numerical solution w^*, *mm*	Analytical solution $\tilde{w}$, *mm*	**Error** =100% $(w^* - \tilde{w}) / \tilde{w}$
27.142	27.997	3.05

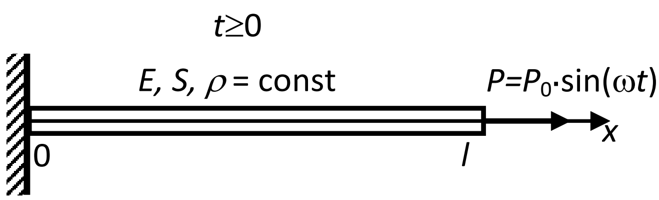
Conclusion:

The relative error of numerical solution according to the analytical is less than 3,05%.

Beam loaded with Sinusoidal Force on the Free End

Let us review the study of longitudinal

vibrations of steel beam. Its left end is fixed. Longitudinal force is applied to the right end starting from time t = 0. The force changes under the law P=P0•sin(wt).



It is necessary to define the maximum displacement of the beam end.

Beam length l=0,5 m. Cross-section is rectangular with height h=50 mm and width b= 20 mm. Modulus of elasticity E=200 GPa, Poisson ratio n=0,29, densityr=7900 kg/m 3. Amplitude and frequency of external force are correspondingly equal P0=10 kN and $V_0 = \frac{\omega}{2\pi} = 2\text{kHz}$.

Displacements of the beam are determined by the formula:

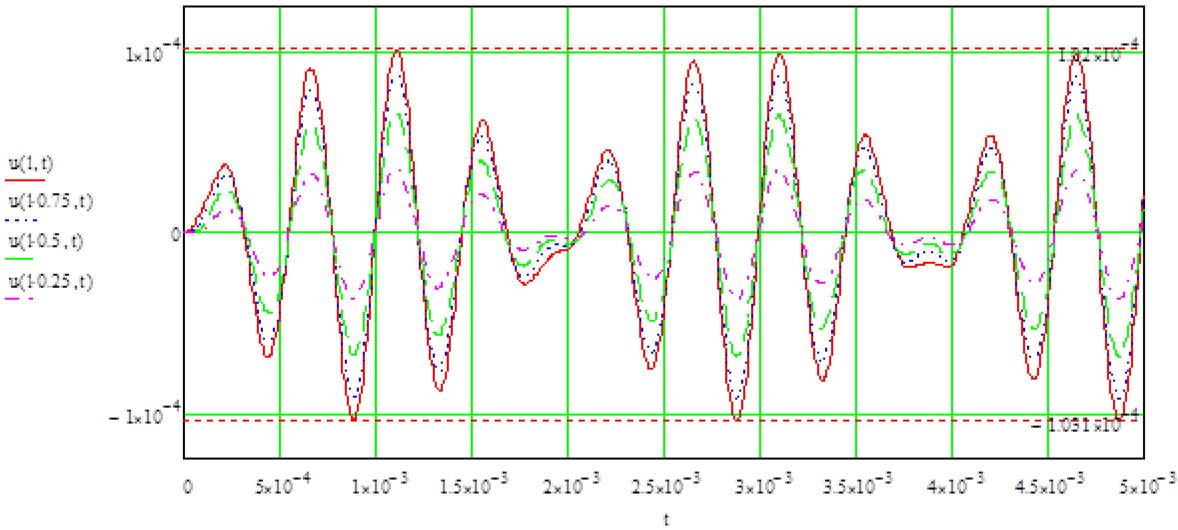
$$u(x,t) = U(x,t) + \sum_{n=0}^{\infty} b_n \sin\frac{(2n+1)\pi x}{2l} \sin\frac{(2n+1)\pi at}{2l}.$$

Here:

$$U(x,t) = \frac{aP_0}{ES\omega}\frac{\sin\frac{\omega}{a}x}{\cos\frac{\omega}{a}l}\sin\omega t, \quad b_n = -\frac{4}{(2n+1)\pi a}\int_0^l U_t(z,0)\sin\frac{(2n+1)\pi z}{2l}dz,$$

$$a = \sqrt{\frac{E}{\rho}}, \quad S = h \cdot b.$$

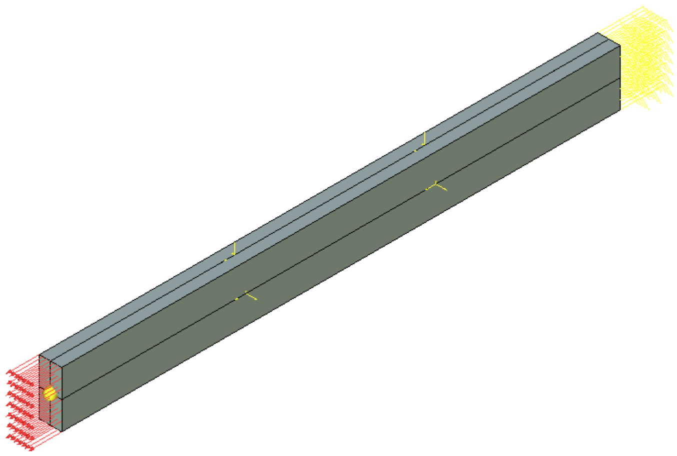
Limited by six terms of series (n=0...5) we create a graph of sections displacements from time t on ten intervals of external force. The sections have the following coordinates x=l (right beam end), x=0.75l, x=0.5l and x=0.25l .



Displacements of beam sections

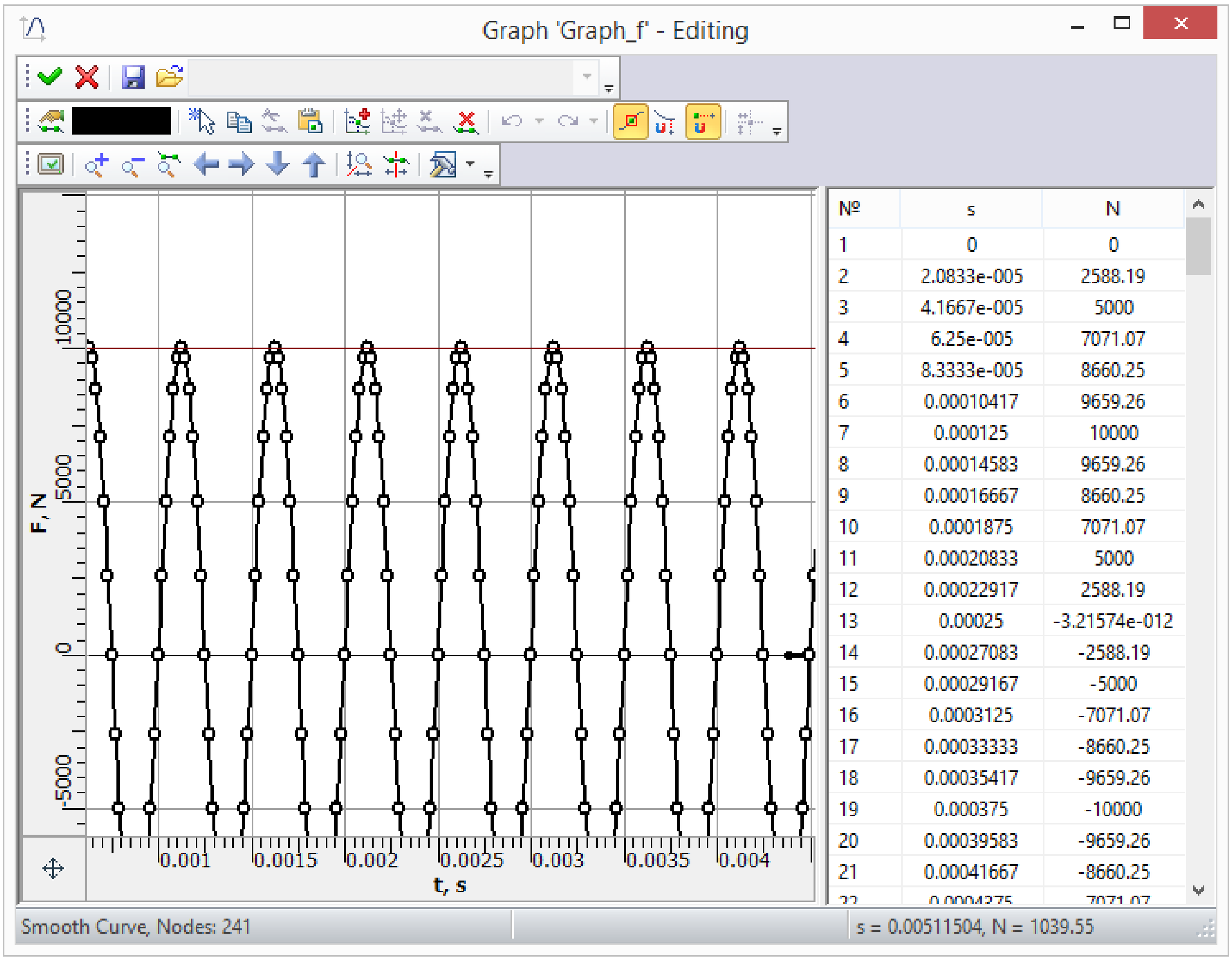
Maximum displacements appear on the right end of the beam and are under tension 1.02 10-4 m, under compression -1.031 10-4 m.

Let us calculate T-FLEX Analysis study: We create two studies, one Transitional processes, another Mode superposition with the same loads and restraints.



Calculated model with loads and restraints

We create a full restraint on the left butt end of the beam. The right end will stay free. We apply distributed force to the free end and specify its value using graph:



Load graph

For both of the studies, we set the finite modeling time 0,005 s, the time step of integration 5•10-6 s. The method of time integration: Newmark. We set a number of the lower natural frequencies in the Mode superposition study: 5.

Table 1.
Parameters of finite-element mesh

Type of finite elements	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	317	1018

Table 2. Parameters of temporal discretization

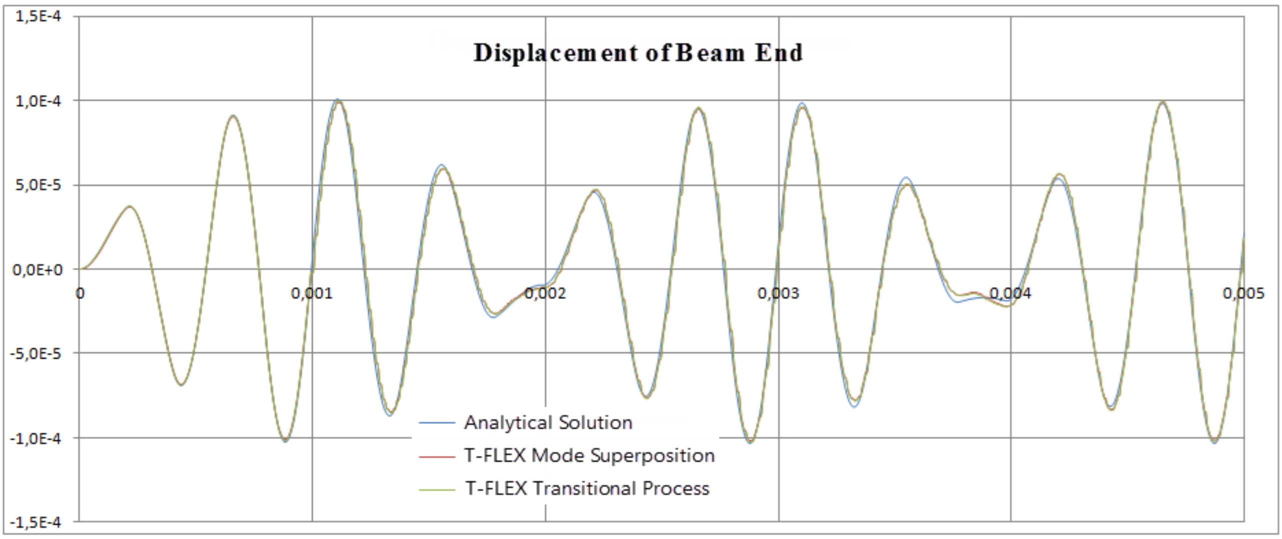
Total calculation time (s)	Time step (s)	Number of time layers
0.005	$5 \cdot 10^{-6}$	1001

Table 3. Transitional processes, "Displacement Z" result

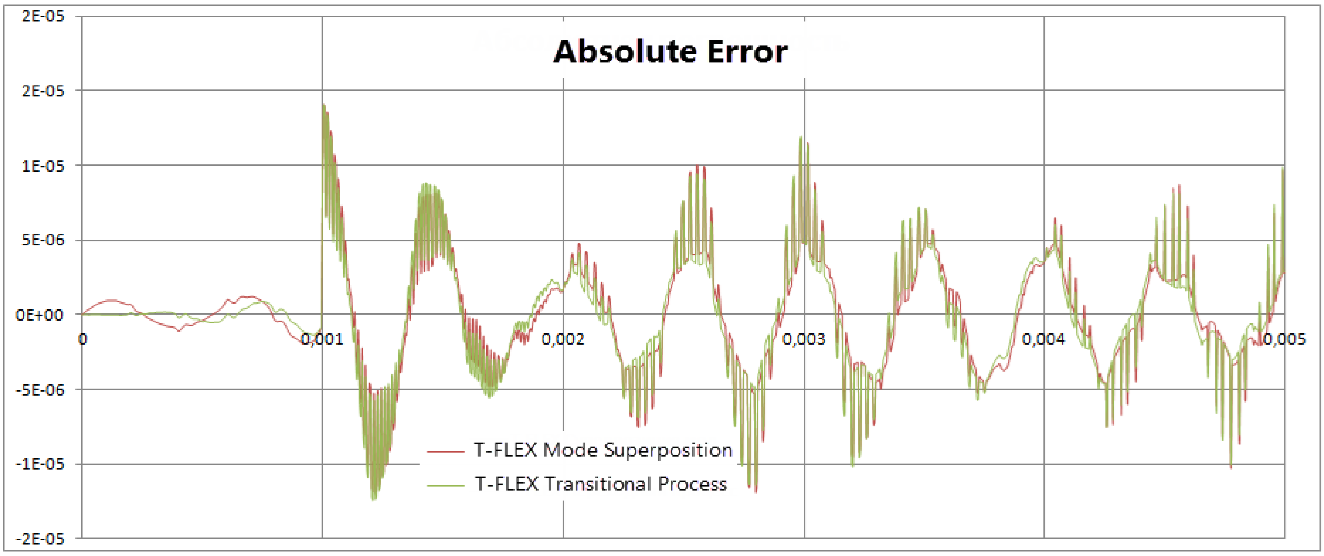
Numerical solution *w** , *mm*	Analytical solution *w* , *mm*	**Error** $=100\% \, (w^* - w) / w$
$1{,}0259 \cdot 10^{-4}$	$1{,}02 \cdot 10^{-4}$	0.51

Table 4. Mode superposition, "Displacement Z" result

Numerical solution *w** , *mm*	Analytical solution *w* , *mm*	**Error** $=100\% \, (w^* - w) / w$
$1{,}0159 \cdot 10^{-4}$	$1{,}02 \cdot 10^{-4}$	1.48



Analytical solution and calculation results

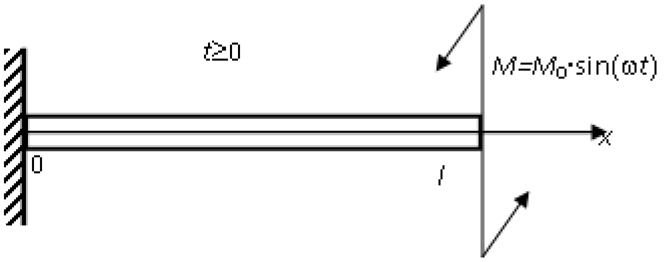


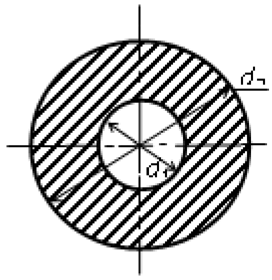
Difference between analytical solution and T-FLEX solution

Conclusion:

The maximum displacement of the free end of the beam, found using T-FLEX Analysis is: for the Transitional process 1.0259 10-4 m (relative error 0.51%), for the Mode superposition1.0159 10-4 m (relative error 1,48%).

Hollow Shaft Loaded with Sinusoidal Torque on the Free End





Let us review the study of torsional oscillation of steel shaft. Its left end is fixed. A torsional couple of forces is applied to the right end starting from time t = 0. The forces torque changes under the law M=M0•sin(wt). Shaft length l = 1 m. The cross-section is annular with inner diameter d1=50 mm and outer diameter d2= 80 mm. Modulus of elasticity E=200 GPa, Poisson ratio n=0,29, density r=7900 kg/m 3. Amplitude and frequency of external torque are correspondingly equal M0=1kN Чm and

$V_0 = \dfrac{\omega}{2\pi} = 500\text{Hz}$. It is necessary to find the maximum torsion angle and the maximum shearing stresses in the shaft.

Studies of longitudinal vibrations and torsion oscillations of beams are mathematically similar. That's why we can use the solution of longitudinal vibrations of the beam to the left end of which the force changing under the law P=P0•sin(wt) is applied to the torsion angle j finding:

$$\varphi(x,t) = \Phi(x,t) + \sum_{n=0}^{\infty} b_n \sin\frac{(2n+1)\pi x}{2l} \sin\frac{(2n+1)\pi a t}{2l}.$$

Here:

$$\Phi(x,t) = \frac{aM_0}{GJ_p\omega}\frac{\sin\frac{\omega}{a}x}{\cos\frac{\omega}{a}l}\sin\omega t,$$

$$b_n = -\frac{4}{(2n+1)\pi a}\int_0^l \Phi_t(z,0)\sin\frac{(2n+1)\pi z}{2l}dz,$$

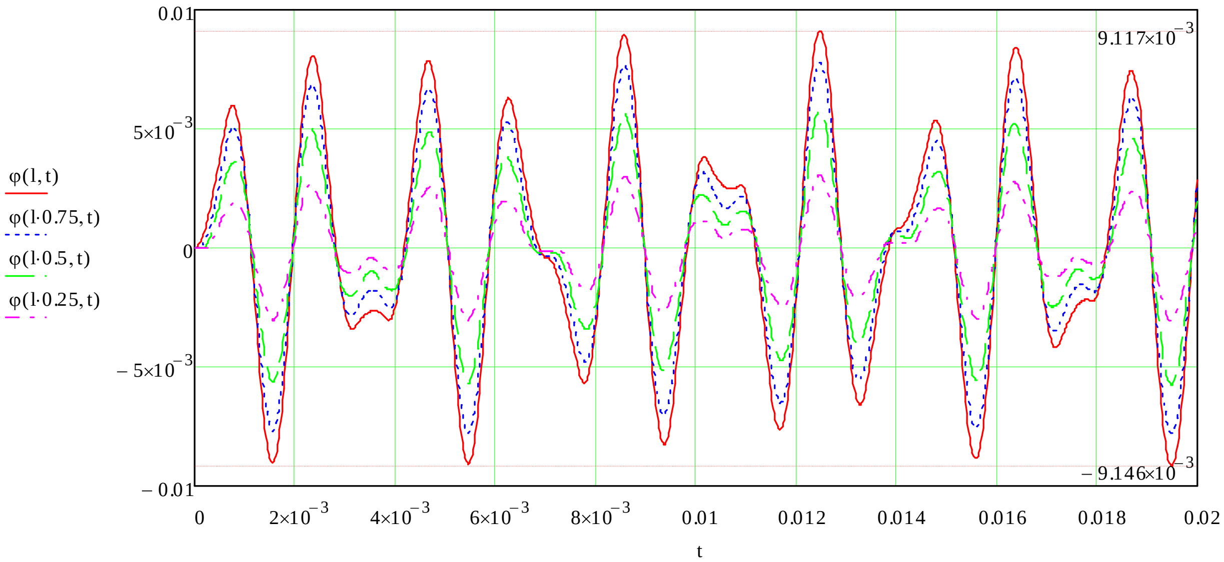
$$a = \sqrt{\frac{G}{\rho}}, G = \frac{E}{2(1+\nu)}, \ J_p = \frac{\pi d_2^4}{32}\left(1-\left(\frac{d_1}{d_2}\right)^4\right).$$

The maximal shearing stress in the section (appears on the outer circumference) is calculated using the formula:

$$\tau^{max} = G\frac{d_2}{2}\cdot\frac{d\varphi}{dx}.$$

It is positive if its torque is directed counter-clockwise from the outer normal line side.

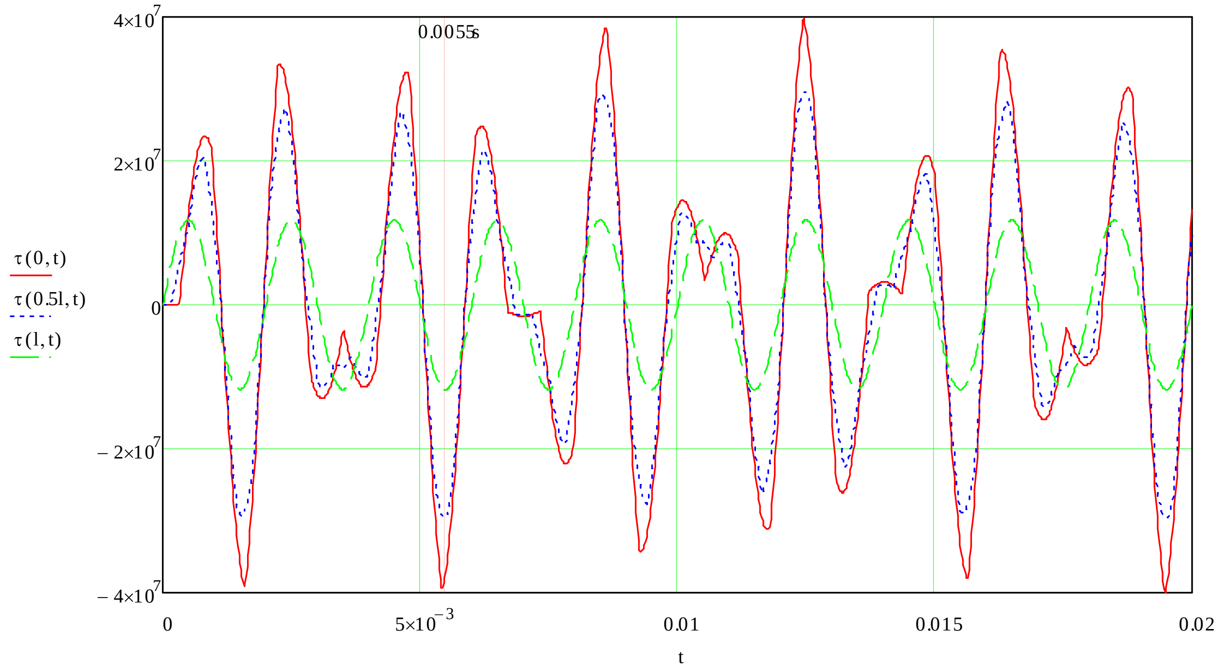
Limited by the eleven terms of series (n=0…10), we create a graph of angles of rotation of sections with coordinates x=l (right end of the shaft), x=0.75l, x=0.5l и x=0.25l according to the time t on ten intervals of outer torque.



Graphs of angles of rotation of four sections according to the time

The maximal angle of rotation is on the right end of the shaft and is 9.146 10-3 rad.

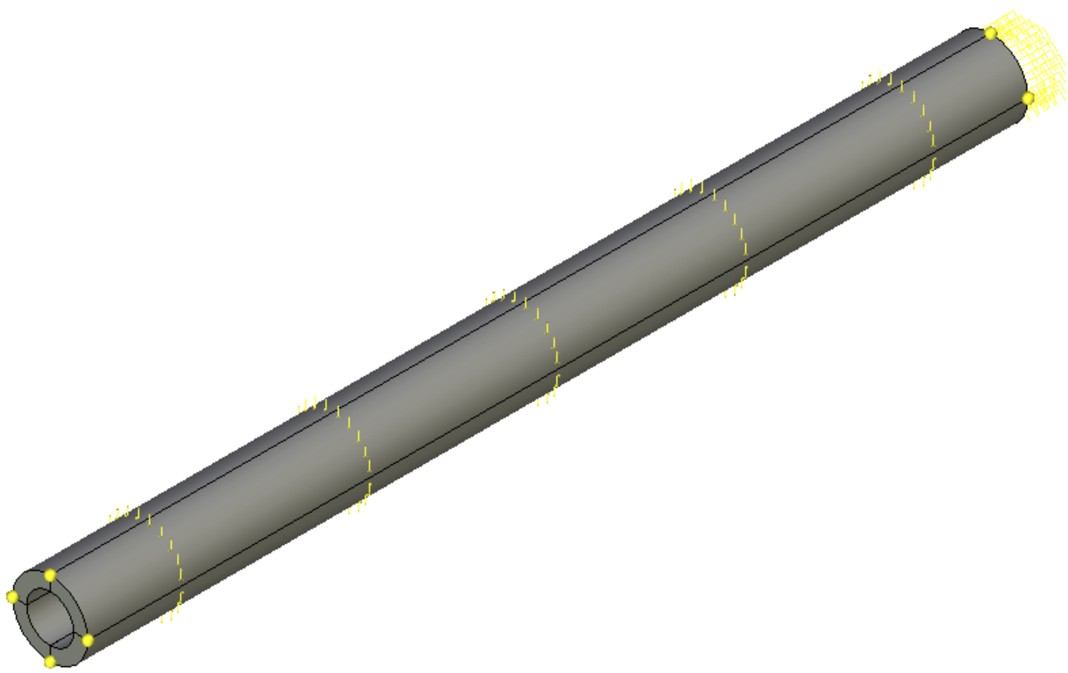
Let us create graphs of maximal shearing stresses in the section for three sections with coordinates: x=0 (closing), x=0.5l and x=l (right end of the shaft) according to the t time on ten intervals of outer torque.



Graphs of maximal shearing stresses in the section according to the time.

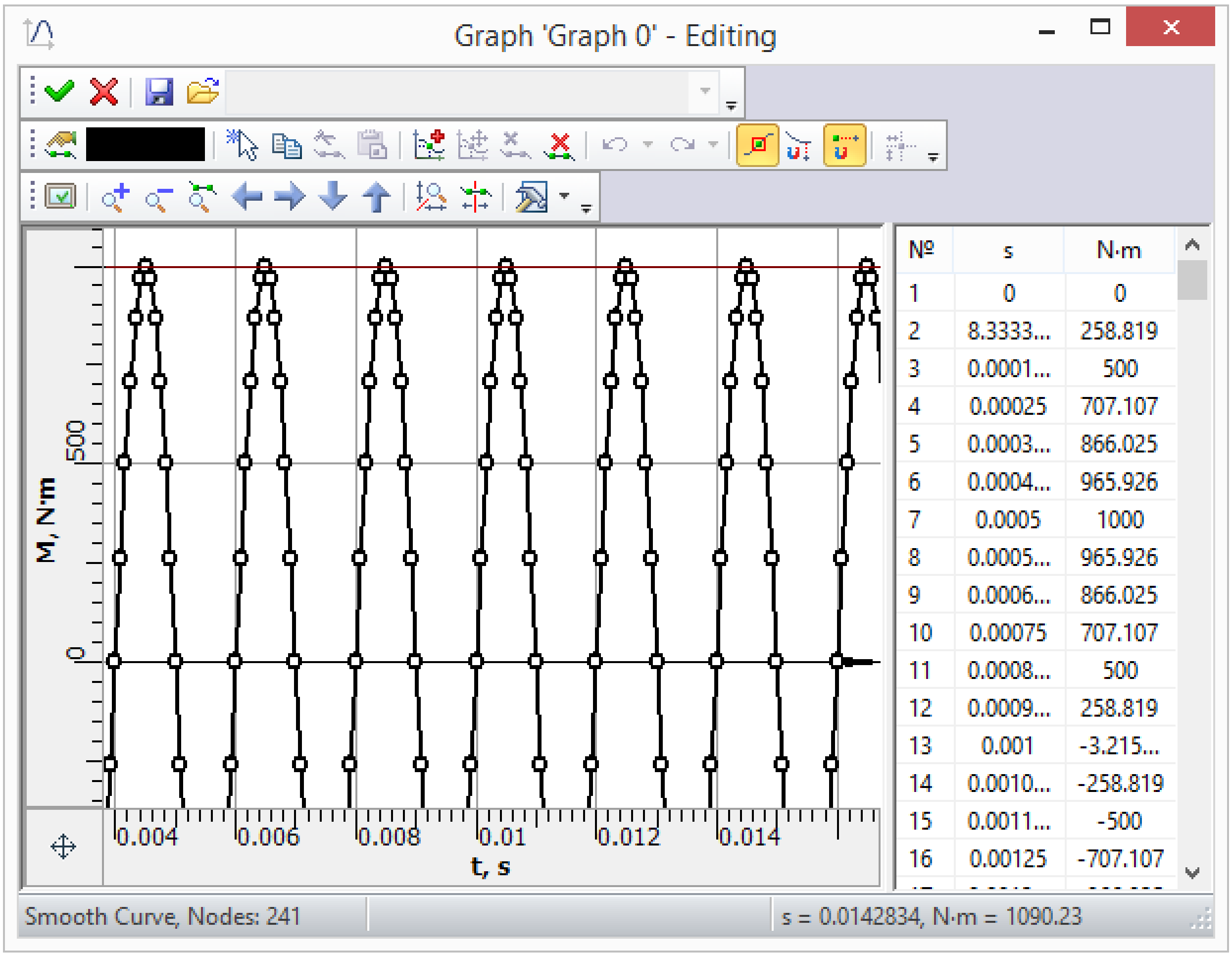
Maximal shearing stress is in the closing (x=0) and is 39,9 MPa modulo.

Let us calculate T-FLEX Analysis study: We create two studies, one Transitional processes, another - Mode superposition with the same loads and restraints.



Calculated model with loads and restraints

We create a full restraint on the left butt end of the shaft. The right end will stay free. To exclude the shaft bending we set kinematic restrictions on the radial displacements (radial restrictions are equal to zero) for the outer and inner cylindrical surfaces- for that purpose we apply partial restraints in the cylindrical coordinate system. Displacements by radius and axis are restricted, displacements around the circumference are permitted, Z axis of the cylindrical coordinate system is directed along the shaft axis. On the free end we apply torque load, the value of the load is specified using graph:

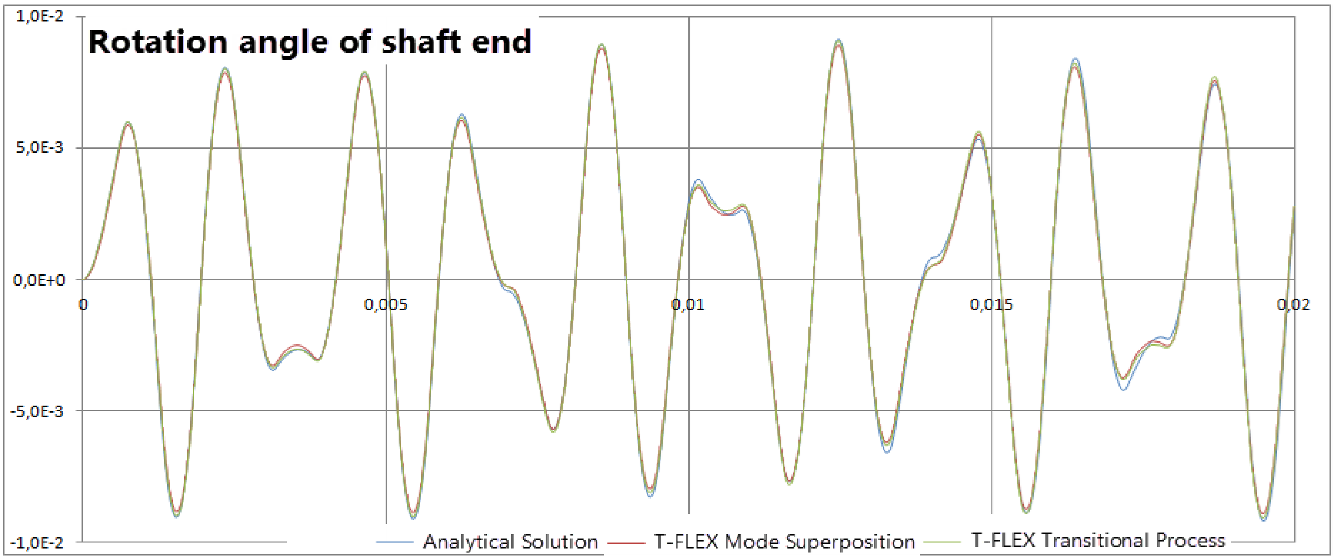


Load graph

For both of the studies, we set the finite modeling time 0,02 s, the time step of integration 1•10-5 s. The method of time integration: Newmark. We set a number of the lower natural frequencies in the Mode superposition study: 5.

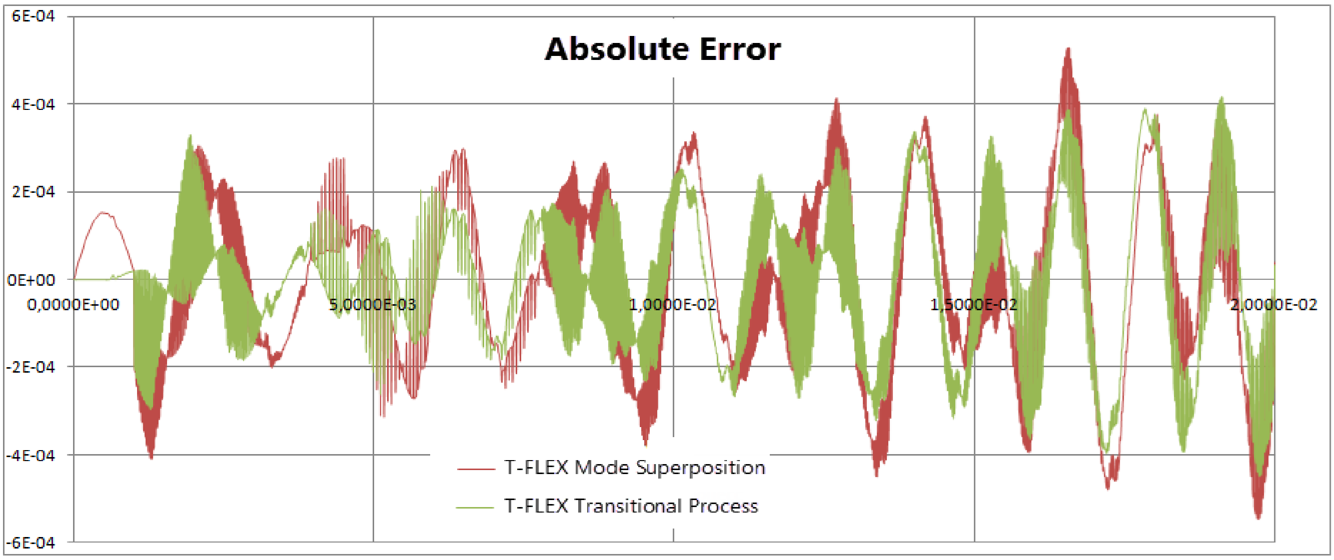
The angle of rotation of the section on the free end of the shaft is calculated by the formula $\varphi = 2u_y^B/d_2$, where u_y^B - projection of point B displacement on the 0 y-axis (found from the calculation in T-FLEX Analysis).

On the figure below, you may see the dependences graphs of free shaft end angle of rotation from time for an analytical solution, a numerical solution using Transitional process and Mode superposition.



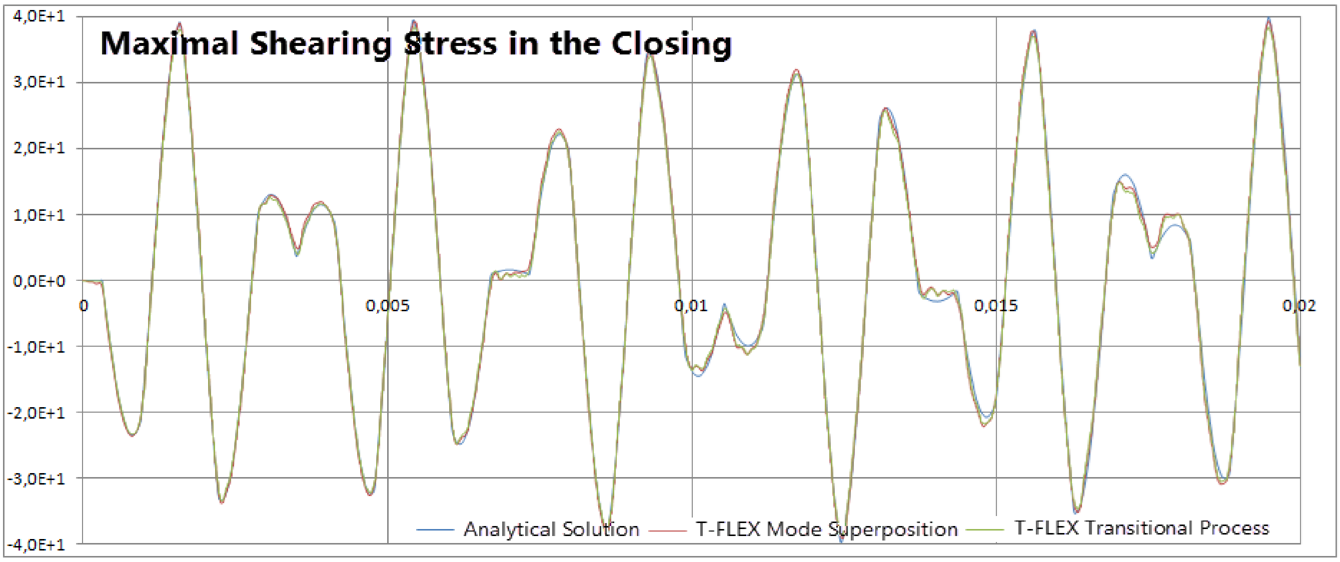
Graphs of rotation angle of free shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below, the difference between analytical solution and solutions by two T-FLEX Analysis methods are shown.



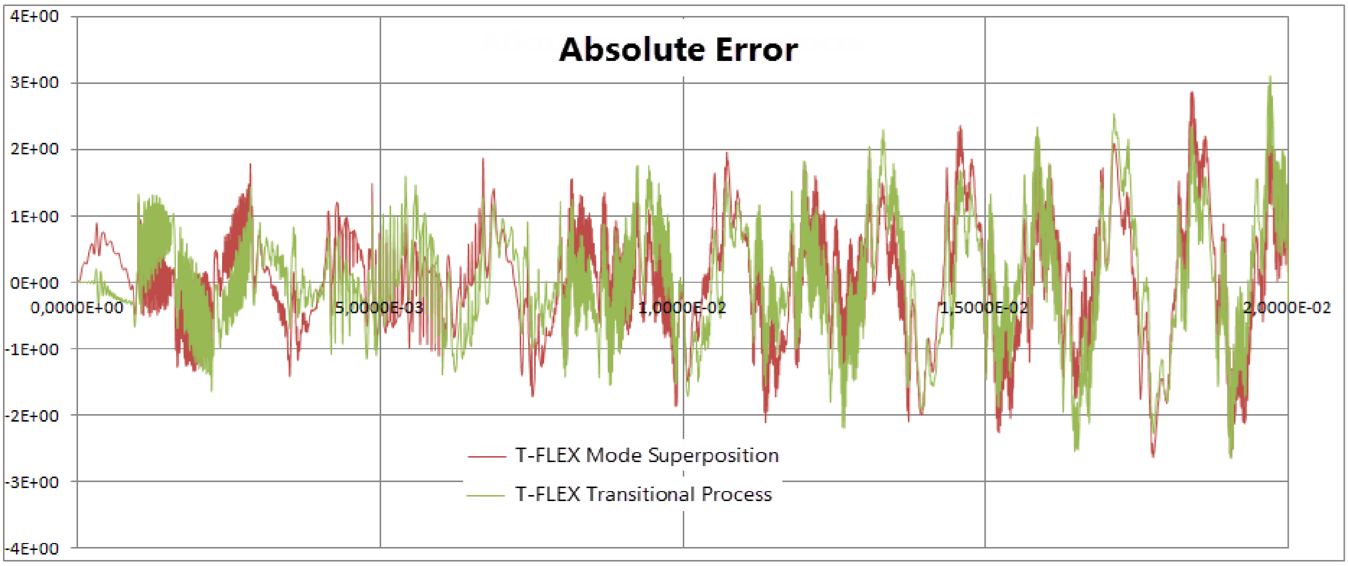
Graphs of differences of rotation angle of shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below, the graphs of shearing stress τ_{xy} in the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown. Positive directions of the shearing stress of analytical solution τ^A and considered in the theory of elasticity τ^A_{xy} (by T-FLEX Analysis) are shown on the figure "**Calculated model with loads and restraints**".



The graphs of shearing stress τ_{xy} *in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.*

On the figure below, the graphs of shearing stress τ_{xy} differences in the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown.



The graphs of shearing stress τ_{xy} *differences in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.*

Table 1.
Parameters of finite-element mesh

Type of finite elements	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	595	1970

Table 2. Parameters of temporal discretization

Total calculation time (s)	Time step (s)	Number of time layers
0.02	$1{\cdot}10^{-5}$	2001

Table 3. Transitional processes, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$9{,}0662 \cdot 10^{-3}$	$9{,}146 \cdot 10^{-3}$	0.88

Table 4. Mode superposition, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$8{,}9169 \cdot 10^{-3}$	$9{,}146 \cdot 10^{-3}$	1.48

Table 5. Transitional processes, maximal shearing stress

Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
38.434	39.9	3.68

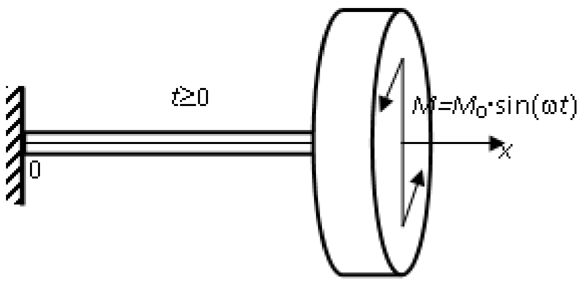
Table 6. Mode superposition, maximal shearing stress

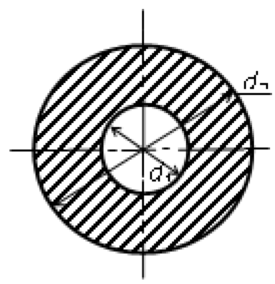
Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
39.210	39.9	1.73

Conclusion:

The maximal angle of rotation of shaft free end calculated using T-FLEX Analysis is: for the Transitional process 9,0662•10-3 rad (relative error 0.88%), for the Mode superposition $8{,}9169 \cdot 10^{-3}$ rad (relative error 2.51%). The maximum shearing stress at the A point found using T-FLEX Analysis is: for the Transitional process 38,434 MPa (relative error 3.68%), for the Mode superposition 39,210 MPa (relative error 1,73%).

Shaft with a Wheel Loaded with Sinusoidal Torque





Let us review the study of torsional oscillation of steel shaft on the end of which the wheel is attached. The left end of the shaft is fixed. To the right end, the wheel is attached. The wheel diameter is D = 200 mm and its thickness h = 30 mm. Couple of torsional forces which changes under the law M(t) =M0•sin(wt) is applied to the wheel from the point in time t=0. Shaft length l = 1 m. The cross-section is annular with inner diameter d 1=50 mm and outer diameter d 2= 80 mm. The wheel and the shaft have the same material. Modulus of elasticity E=200 GPa, Poisson ratio n=0,29, density r=7900 kg/m 3. Amplitude and frequency of external torque are correspondingly equal M0=1kNЧm and

$V_0 = \dfrac{\omega}{2\pi} = 200\text{Hz}$. Let us find the maximum torsion angle and the maximum shearing stresses in the shaft.

The study can be solved using methods described in the book.
First of all, it is necessary to find normal basis function - eigenfunctions of the free oscillations of the shaft with the wheel task:

$$X_n(x) = \sin\left(\frac{\mu_n b}{l}\right), \qquad n = 1,2,3, \ldots$$

Here $b = \sqrt{\frac{G}{\rho}}$, $G = \frac{E}{2(1+\nu)}$ - shear modulus 6 μ_n - positive roots of the equation $\mathrm{tg}(\mu) = \frac{J_p \rho l}{J} \cdot \frac{1}{\mu}$,
$J_p = \frac{\pi d_2^4}{32}\left(1 - \left(\frac{d_1}{d_2}\right)^4\right)$ - a polar moment of inertia of the shaft section, $J = \rho h \frac{\pi D^4}{32}$ - wheel moment of inertia.

Then the angle of rotation of the shaft is recorded in normal coordinates:

$$\theta(x,t) = \sum_{n=1}^{\infty} \varphi_n(t) \cdot X_n(t).$$

Then the Lagrange equations are formulated. After their calculation the normal coordinates are defined:

$$\varphi_n(t) = \frac{X_n(l) M_0 \mu_n}{A_n \rho J_p l b} \int_0^t \sin(\omega\tau) \sin\left(\frac{\mu_n b}{l}(t-\tau)\right) d\tau.$$

Here $A_n = \int_0^l \left[\frac{dX(x)}{dx}\right]^2 dx$.

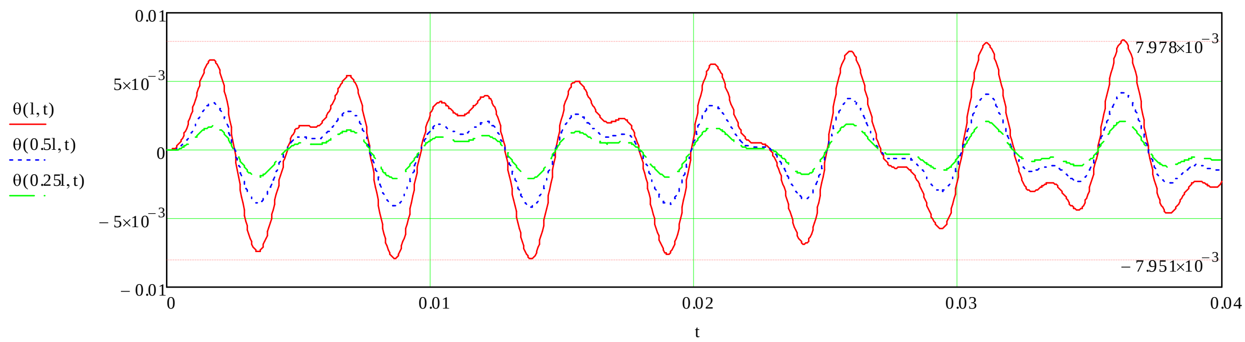
The maximal shearing stress in the section (appears on the outer circumference) is calculated using the formula:

$$\tau^{max} = G\frac{d_2}{2} \cdot \frac{d\theta}{dx}.$$

It is positive if its torque is directed counter-clockwise from the outer normal line side.

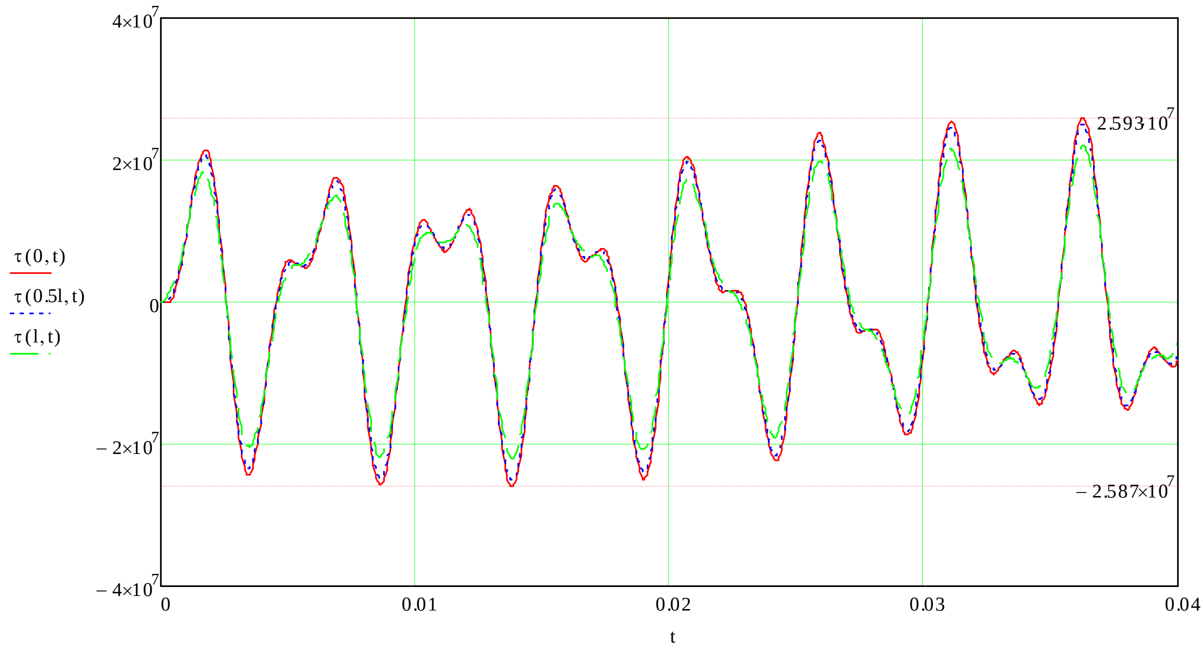
Let us limit ourselves by ten terms of series. With the given parameters, the maximal angle of rotation is on the shaft end with the wheel and is $7.978 \cdot 10^{-3}$ rad.

The graph of angles of rotation of sections with coordinates x=l (shaft end with the wheel) x=0.5l, and x=0.25 l according to the time t on eight intervals of outer torque is presented on the picture.



Graphs of angles of rotation of three sections according to the time

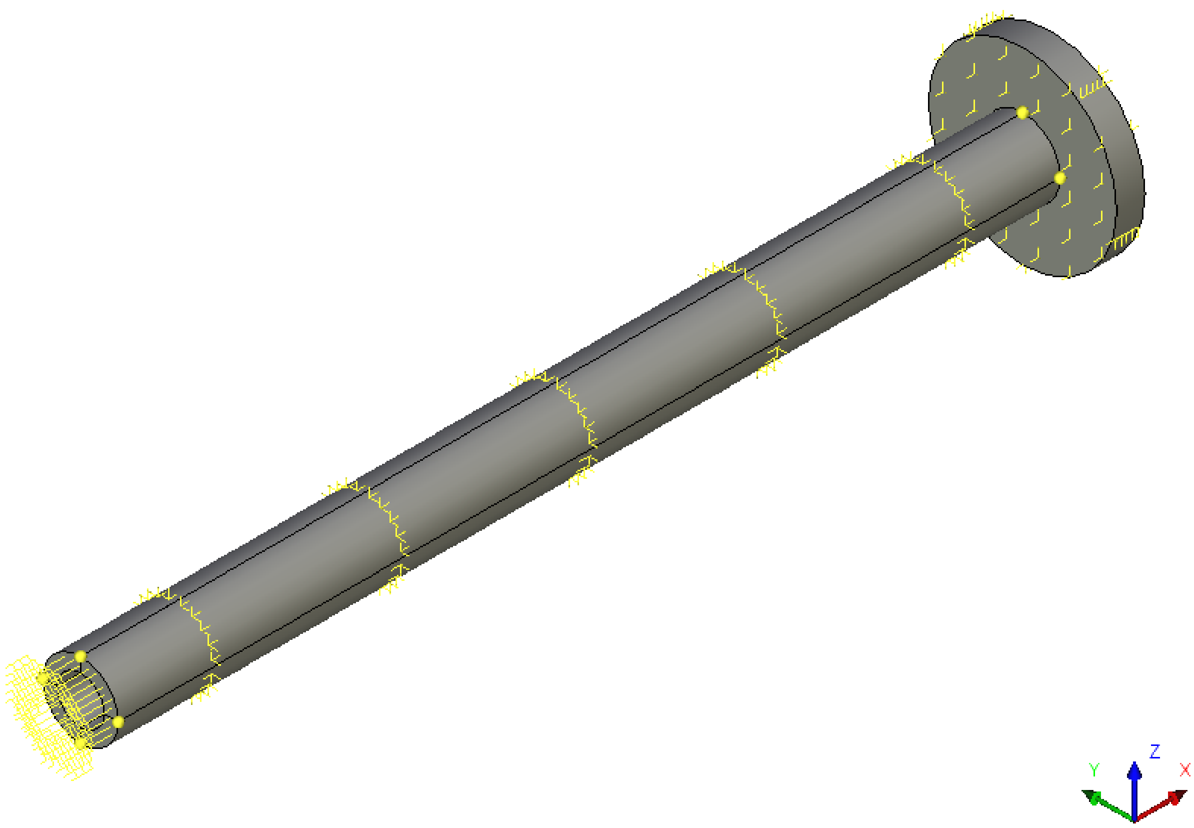
Let us create graphs of maximal shearing stresses in the section for three sections with coordinates: x=0 (closing), x=0.5l and x=l (shaft end with the wheel) according to the t time on eight intervals of outer torque.



Graphs of maximal shearing stresses in the section according to the time.

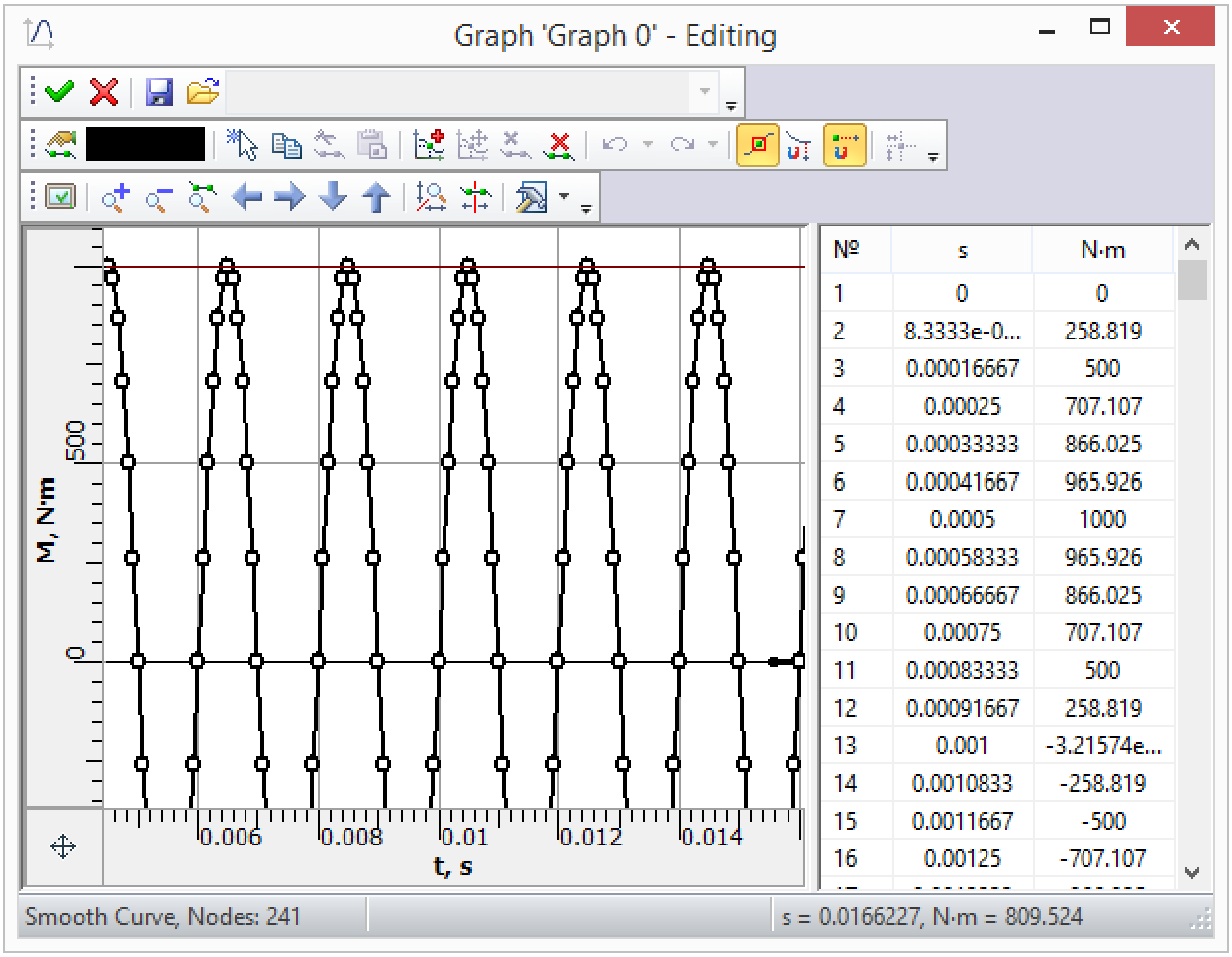
Maximal shearing stress is in the closing (x=0) and is 25,9 MPa modulo.

Let us calculate T-FLEX Analysis study: We create two studies, one Transitional processes, another Mode superposition with the same loads and restraints.



Calculated model with loads and restraints

We create a full restraint on the left butt end of the shaft. The right end will stay free. To exclude the shaft bending we set kinematic restrictions on the radial displacements (radial restrictions are equal to zero) for the outer and inner cylindrical surfaces and for all wheel faces - for that purpose, we apply partial restraints in the cylindrical coordinate system. Displacements by radius and axis are restricted, displacements around the circumference are permitted, Z axis of the cylindrical coordinate system is directed along the shaft axis. We set restrictions on the displacements along 0x axis (the axial displacements are equal zero) to exclude longitudinal vibrations on the outer cylindrical surface and to the outer circular surface of the wheel. For that purpose, we set partial restraint in the rectangular coordinate system with restraint only by X axis. On the free end we apply torque load, the value of the load is specified using graph:

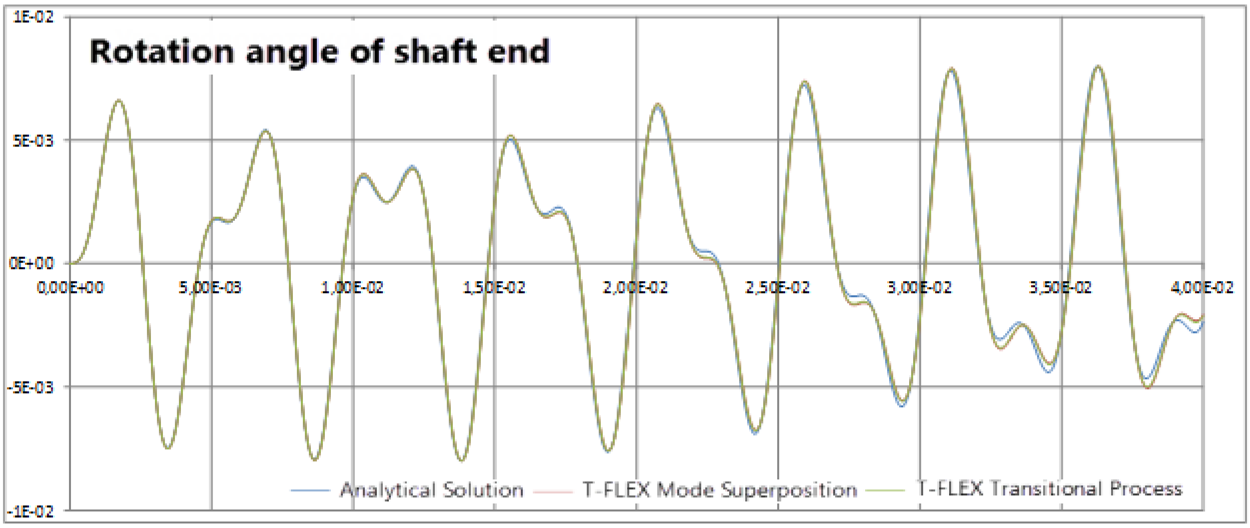


Load graph

For both of the studies, we set the finite modelling time 0,04 s, the time step of integration 2,5•10-5 s. The method of time integration: Newmark. We set a number of the lower natural frequencies in the Mode superposition study: 15.

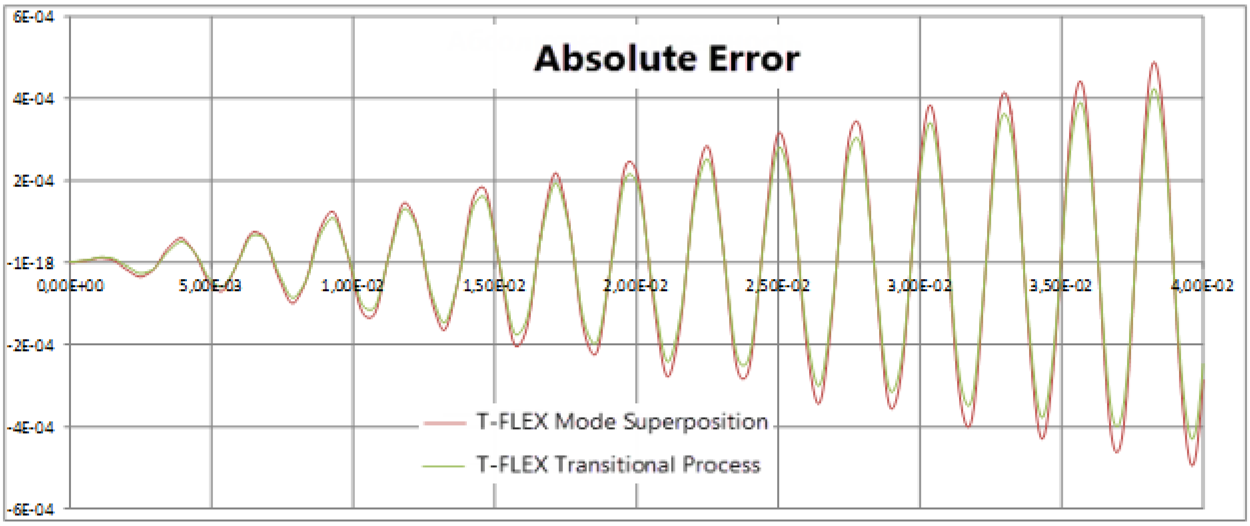
The angle of rotation of the section on the free end of the shaft with the wheel is calculated by the formula $\varphi = 2u_y^B/d_2$, where u_y^B - projection of point B displacement on the 0 y-axis (found from the calculation in T-FLEX Analysis).

On the figure below, you may see the dependences graphs of free shaft end with the wheel angle of rotation from time for an analytical solution, a numerical solution using Transitional process and Mode superposition.



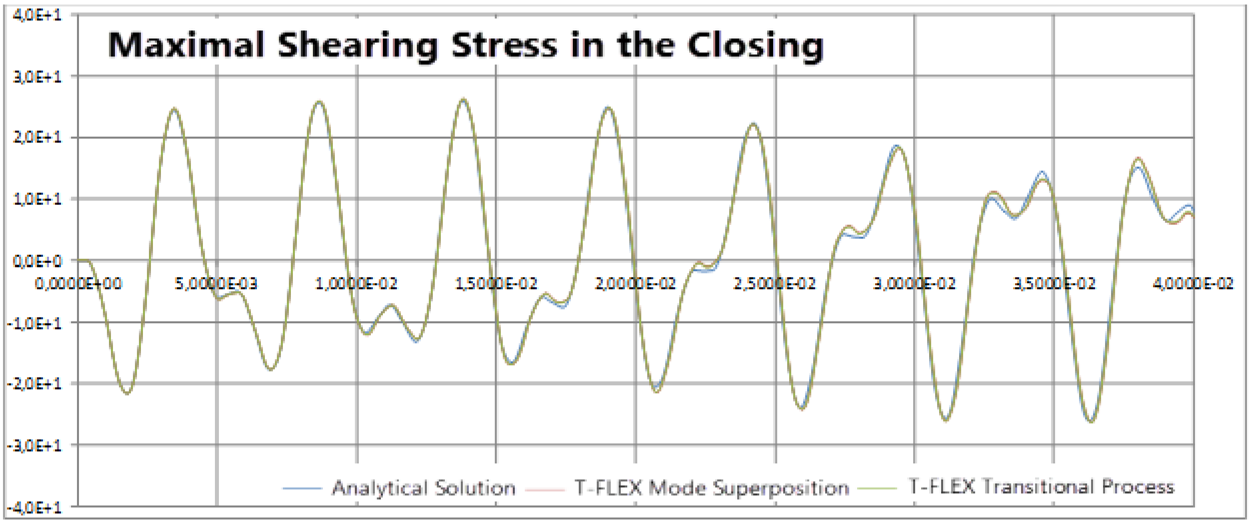
Graphs of rotation angle of free shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below, the difference between analytical solution and solutions by two T-FLEX Analysis methods are shown.



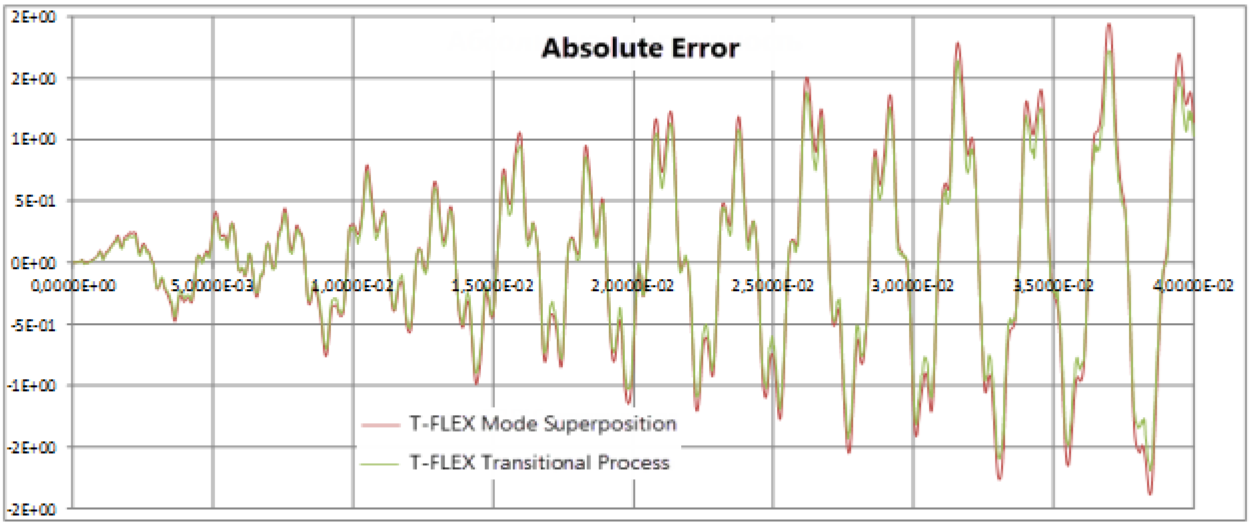
Graphs of differences of rotation angle of shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below the graphs of shearing stress τ_{xy} in the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown. Positive directions of the shearing stress of analytical solution τ^A and considered in the theory of elasticity τ^A_{xy} (by T-FLEX Analysis) are shown on the figure “Calculated model with loads and restraints”.



The graphs of shearing stress τ_{xy} *in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.*

On the figure below, the graphs of shearing stress τ_{xy} differences in the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown.



The graphs of shearing stress τ_{xy} *differences in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.*

Table 1.
Parameters of finite-element mesh

Type of finite elements	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	602	1932

Table 2. Parameters of temporal discretization

Total calculation time (s)	Time step (s)	Number of time layers
0.04	$2{,}5 \cdot 10^{-5}$	1601

Table 3. Transitional processes, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$7{,}9763 \cdot 10^{-3}$	$7{,}978 \cdot 10^{-3}$	0.15

Table 4. Mode superposition, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$7{,}9732 \cdot 10^{-3}$	$7{,}978 \cdot 10^{-3}$	0.08

Table 5. Transitional processes, maximal shearing stress

Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
26.270	25.9	1.30

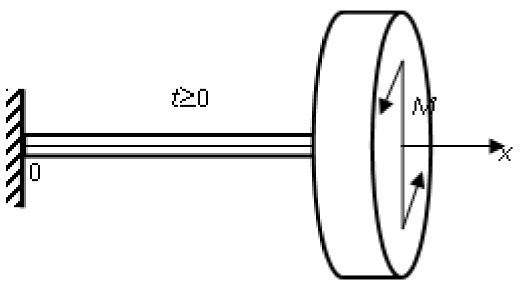
Table 6. Mode superposition, maximal shearing stress

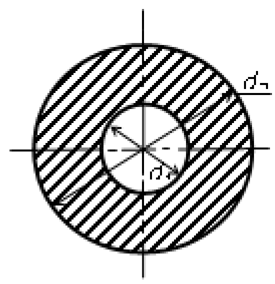
Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
26.318	25.9	1.49

Conclusion:

The maximal rotation angle of the shaft end with the wheel calculated using T-FLEX Analysis is: for the Transitional process 7,9763•10-3 rad (relative error 0.15%), for the Mode superposition 7,9732•10-3 rad (relative error 0.08%). The maximum shearing stress at the A point found using T-FLEX Analysis is: for the Transitional process 26,270 MPa (relative error 1.30%), for the Mode superposition 26,318 MPa (relative error 1,49%).

Shaft with the Wheel Loaded with the Torque that linearly increasing to the Constant Value

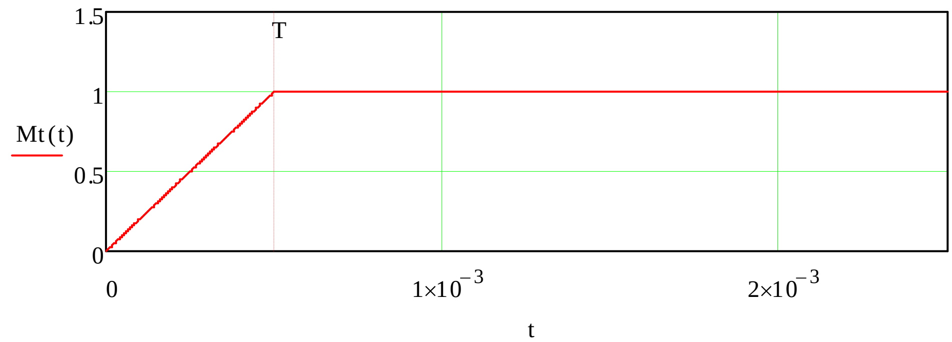




Let us review the study of torsional oscillation of steel shaft on the end of which the wheel is attached. The left end of the shaft is fixed. The wheel is attached to the right end. The wheel diameter is D = 200 mm and its thickness h = 30 mm. A couple of torsional forces is applied to the shaft from the point in time t=0. The torque of the torsional forces changes under the law:

$$M(t) = \begin{cases} M_0 \dfrac{t}{T}, \text{if } \ t < T \\ M_0, \mathit{if} \ \ t \geq T \end{cases}$$

,where M_0 = 1 kNm, T=0,5·10-3sec



Graph of torque of the forces couple from the time

Shaft length l = 1 m. The cross-section is annular with inner diameter d 1=50 mm and outer diameter d2= 80 mm. The wheel and the shaft have the same material. Modulus of elasticity E=200 GPa, Poisson ratio n=0,29, densityr=7900 kg/m 3. Amplitude and frequency of external torque are correspondingly equal M0=1kN Чm and $V_0 = \frac{\omega}{2\pi} = 200\text{Hz}$. Let us find the maximum torsion angle and the maximum shearing stresses in the shaft.

The study can be solved using methods described in the book [15].

First, it is necessary to find normal basis function - eigenfunctions of the free oscillations of the shaft with the wheel task:

Here , - shear modulus 6 - positive roots of the equation , - a polar moment of inertia of the shaft section, - wheel moment of inertia.

Then the angle of rotation of the shaft is recorded in normal coordinates:

Then the Lagrange equations are formulated. After their calculation the normal coordinates are defined:

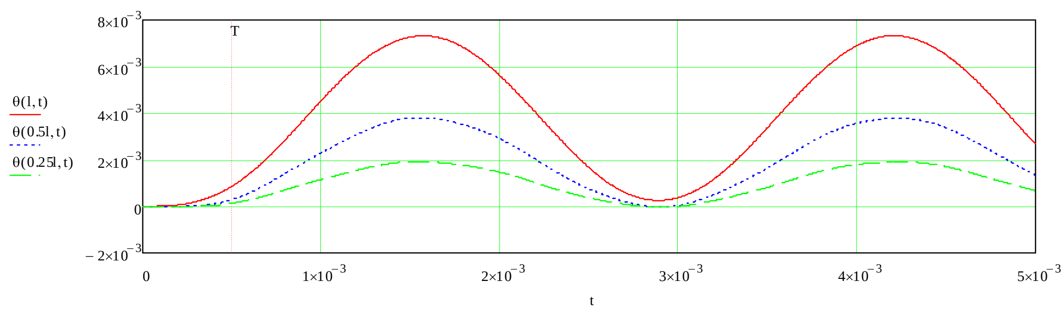
Here .

The maximal shearing stress in the section (appears on the outer circumference) is calculated using the formula:

It is positive if its torque is directed counter-clockwise from the outer normal line side.

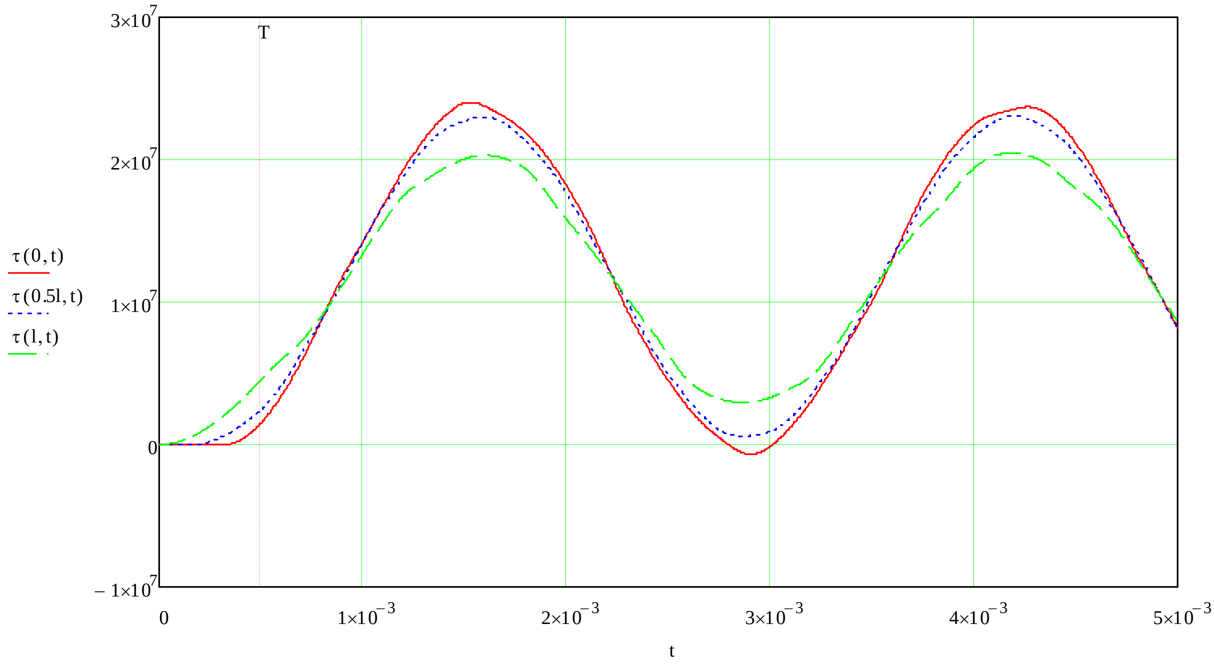
Let us limit ourselves by ten terms of series. With the given parameters, the maximal angle of rotation is on the shaft end with the wheel and is rad.

The graph of rotation angles of sections with coordinates x=l (shaft end with the wheel) x=0.5l, and x=0.25 l according to the time t.



Graphs of angles of rotation of three sections according to the time

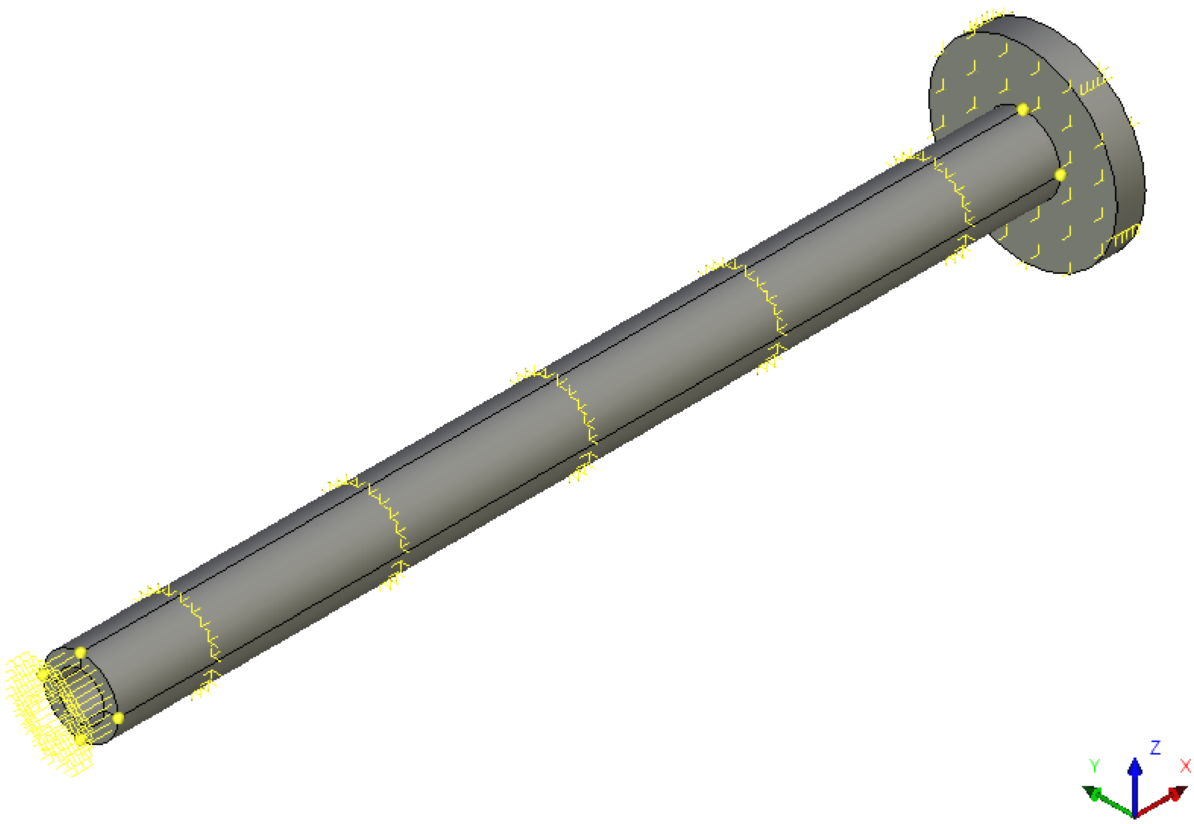
Let us create graphs of maximal shearing stresses in the section for three sections with coordinates: x=0 (closing), x=0.5l and x=l (shaft end with the wheel) according to the t time.



Graphs of maximal shearing stresses in the section according to the time.

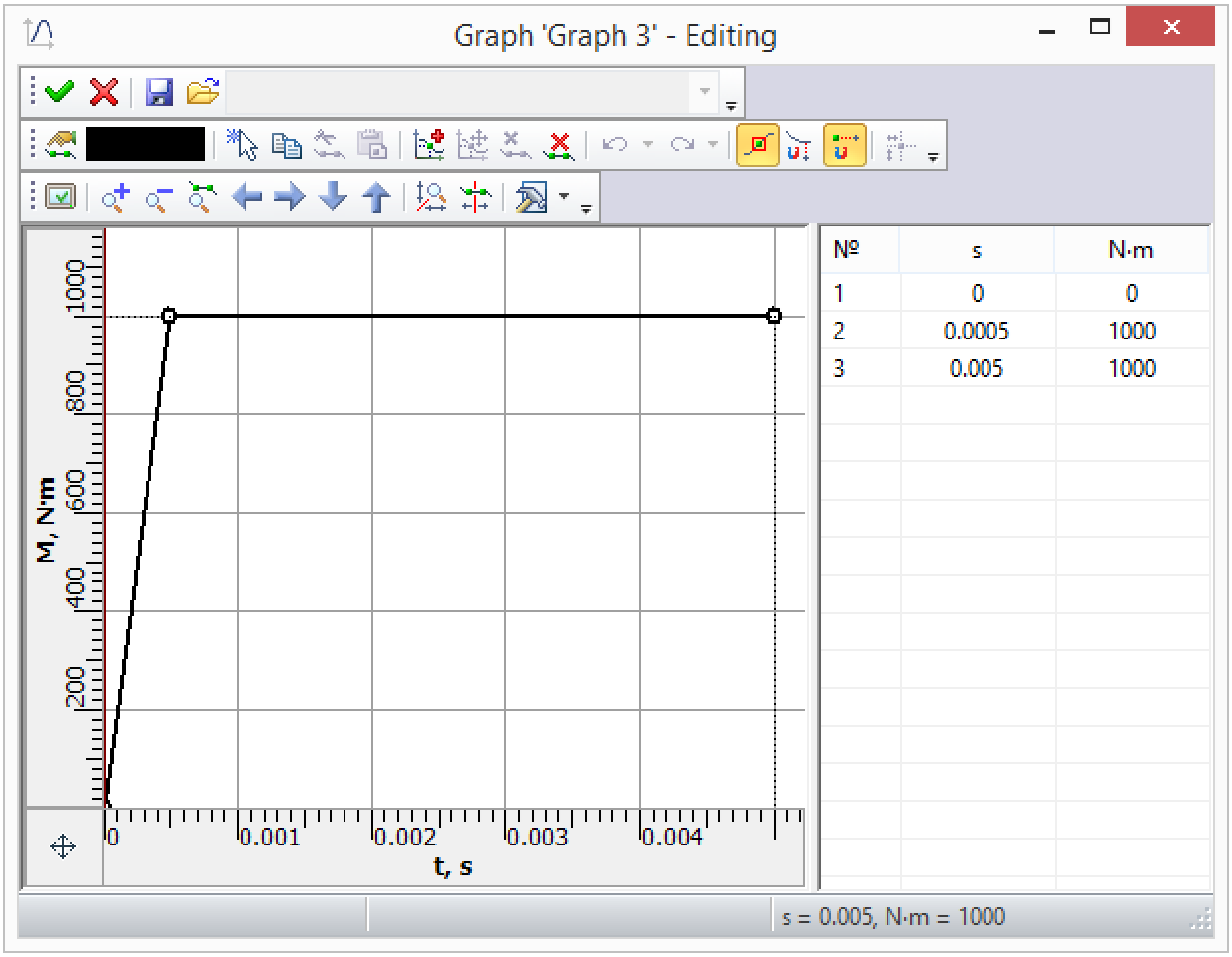
Maximal shearing stress is in the closing (x=0) and is 24,0 MPa modulo.

Let us calculate T-FLEX Analysis study: We create two studies, one Transitional processes, another Mode superposition with the same loads and restraints.



Calculated model with loads and restraints

We create a full restraint on the left butt end of the shaft. The right end will stay free. To exclude the shaft bending we set kinematic restrictions on the radial displacements (radial restrictions are equal to zero) for the outer and inner cylindrical surfaces and for all wheel faces - for that purpose, we apply partial restraints in the cylindrical coordinate system. Displacements by radius and axis are restricted, displacements around the circumference are permitted, Z axis of the cylindrical coordinate system is directed along the shaft axis. We set restrictions on the displacements along 0x axis (the axial displacements are equal zero) to exclude longitudinal vibrations on the outer cylindrical surface and to the outer circular surface of the wheel. For that purpose, we set partial restraint in the rectangular coordinate system with restraint only by X axis. On the free end we apply torque load, the value of the load is specified using graph:



Load graph

For both of the studies, we set the finite modeling time 0,005 s, the time step of integration 5•10-5 s. The method of time integration: Newmark. We set a number of the lower natural frequencies in the Mode superposition study: 15.

The angle of rotation of the section on the free end of the shaft with the wheel is calculated by the formula , where - projection of point B displacement on the 0 y-axis (found from the calculation in T-FLEX Analysis).

On the figure below, you may see the dependences graphs of free shaft end with the wheel angle of rotation from time for an analytical solution, a numerical solution using Transitional process and Mode superposition.

Graphs of rotation angle of free shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below, the difference between analytical solution and solutions by two T-FLEX Analysis methods are shown.

Graphs of differences of rotation angle of the shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below the graphs of shearing stress at the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown. Positive directions of the shearing stress of analytical solution and considered in the theory of elasticity (by T-FLEX Analysis) are shown on the figure “Calculated model with loads and restraints”.

The graphs of shearing stress in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.

On the figure below, the graphs of shearing stress differences in the point A using analytical solution and numerical solutions by two methods in T-FLEX Analysis are shown.

The graphs of shearing stress differences in the point A using analytical solution and two numerical solutions by two methods in T-FLEX Analysis are shown.

Table 1.
Parameters of finite-element mesh

Type of finite elements	Number of nodes	Number of finite elements
quadratic tetrahedron (10 nodes)	602	1932

Table 2. Parameters of temporal discretization

Total calculation time (s)	Time step (s)	Number of time layers
0.04	$2{,}5 \cdot 10^{-5}$	1601

Table 3. Transitional processes, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$7{,}3173 \cdot 10^{-3}$	$7{,}326 \cdot 10^{-3}$	0.11

Table 4. Mode superposition, maximal torsion angle

Numerical solution *, *rad*	Analytical solution , *rad*	**Error** =100% (* -) /
$7{,}3173 \cdot 10^{-3}$	$7{,}326 \cdot 10^{-3}$	0.11

Table 5. Transitional processes, maximal shearing stress

Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
24.258	24.0	1.09

Table 6. Mode superposition, maximal shearing stress

Numerical solution *, *MPa*	Analytical solution , *MPa*	**Error** =100% (* -) /
24.249	24.0	1.05

Conclusion:

The maximal rotation angle of the shaft end with the wheel calculated using T-FLEX CAD is 7 rad (relative error 0,11%) for the Transitional process and Mode superposition. The maximum shearing stress at the A point found using T-FLEX Analysis is: for the Transitional process 24,248 MPa (relative error1,09%), for the Mode superposition MPa (relative error 1,05%).

Contact us to request information about T-FLEX software, our Academic Program, or if you have ideas on cooperating with Top Systems

www.tflex.com/mail

Contact Us

TOP SYSTEMS

www.tflex.com

(7-499) 973-20-34
(7-499) 973-20-35