T·FLEX ANALYSIS 16

Finite Element Analysis

User Manual

TOP SYSTEMS

T-FLEX ANALYSIS

USER MANUAL

TABLE OF CONTENTS

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 studies 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 grouped into three groups according to the studies types are available.

Static Analysis
    Static Strength
    Buckling
    Fatigue
    Express Analysis
Dynamic Analysis
    Frequency
    Forced Oscillation
    Mode Superposition
    Transitional Process
Thermal Analysis
    Stationary Thermal Process
    Nonstationary Thermal Process
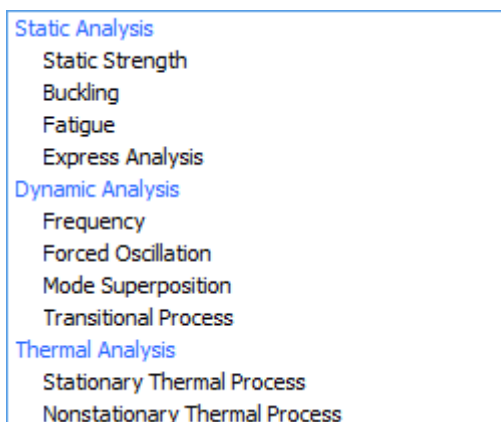
Finite element analysis studies types

Static analysis group includes four types of finite elements studies:

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.

Dynamic Analysis group offers the following type of finite element studies:

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 studies in nonlinear formulation.

Thermal analysis group allows to solve thermal studies.

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.

ABOUT 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 studies 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 studies 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 studies, 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 studies 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.
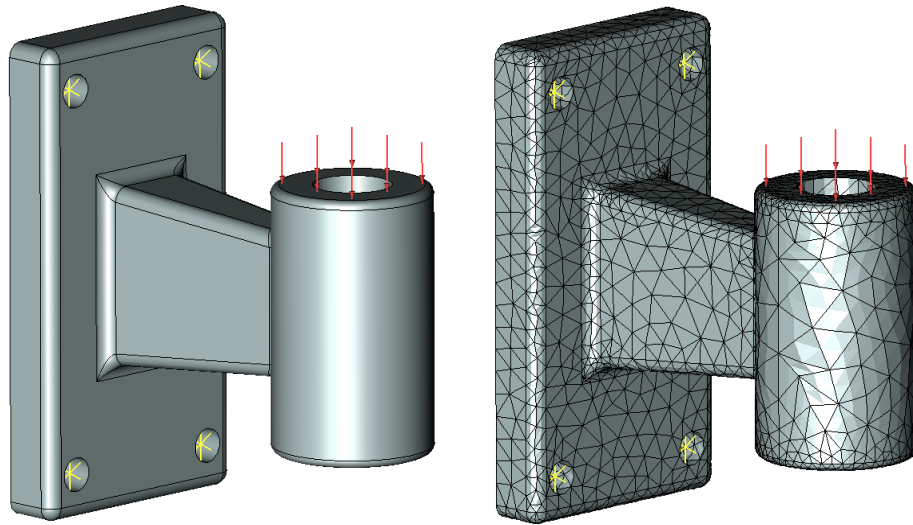
Original structure and its finite element discretization

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 studies). 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 studies 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.
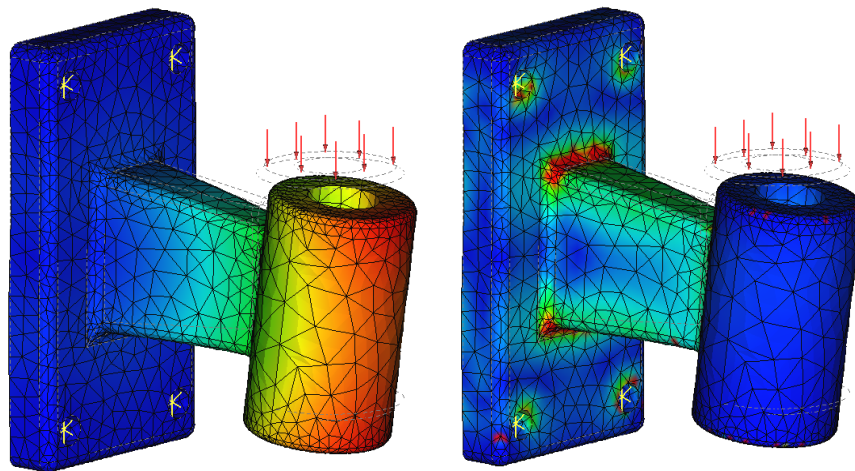
Results of finite element modeling (displacements and stresses)

The described algorithm of finite element modeling is applicable for solving various studies, 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 studies 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 studies 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 study 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 studies of higher dimensionality.

Computer parameters for work with T-FLEX Analysis

Minimal	
Processor	Pentium IV or similar

Hard drive space (for storing calculation results)	500 Mb
RAM	2 GB
Operating system	Windows Vista
Recommended	
Processor	Core i7 or similar
RAM	8 GB (and larger)
Operating system	Windows 7 x64, 8 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.

T-FLEX ANALYSIS SYSTEM INSTALLATION

In order to use the T-FLEX Analysis application, you need to have installed the T-FLEX CAD geometrical modeling system. Therefore, before installing T-FLEX Analysis system, first install T-FLEX CAD.
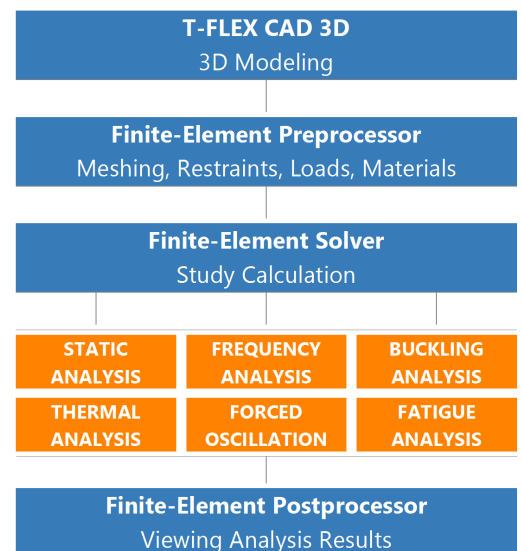
T-FLEX Analysis system is distributed with a protection from non-licensed use. To work with the system, you need a hardware key or software protection key.

STRUCTURAL ORGANIZATION OF T-FLEX ANALYSIS APPLICATION

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

T-FLEX CAD 3D
3D Modeling

Finite-Element Preprocessor
Meshing, Restraints, Loads, Materials

Finite-Element Solver
Study Calculation

STATIC ANALYSIS	FREQUENCY ANALYSIS	BUCKLING ANALYSIS
THERMAL ANALYSIS	FORCED OSCILLATION	FATIGUE ANALYSIS

Finite-Element Postprocessor
Viewing Analysis Results
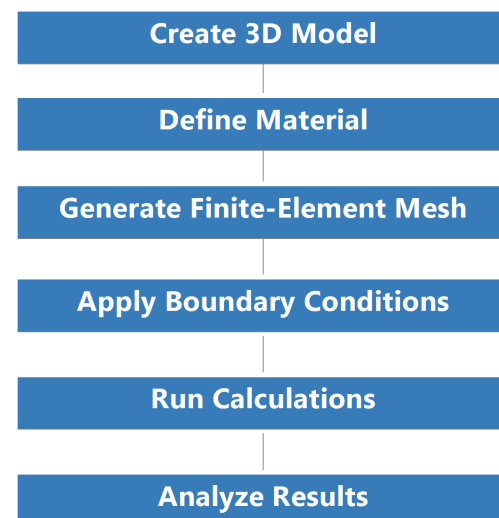
STEPS OF STRUCTURAL ANALYSIS

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.

Steps of Fenite Element Analysis

Create 3D Model
Define Material
Generate Finite-Element Mesh
Apply Boundary Conditions
Run Calculations
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.

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.
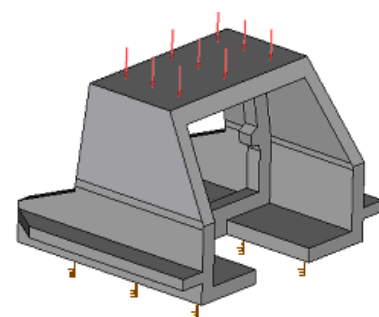
Step 1. Preparing Spatial Solid Model of a Part

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.

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

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.
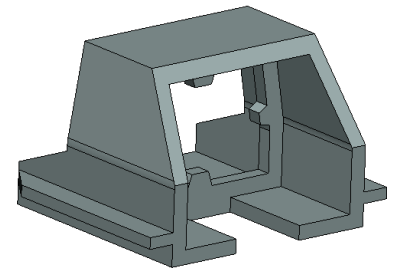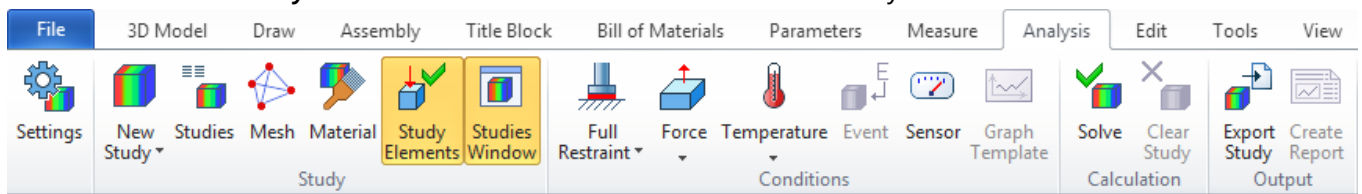
Step 2. Creating «Study»

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 analysis** for our model part.
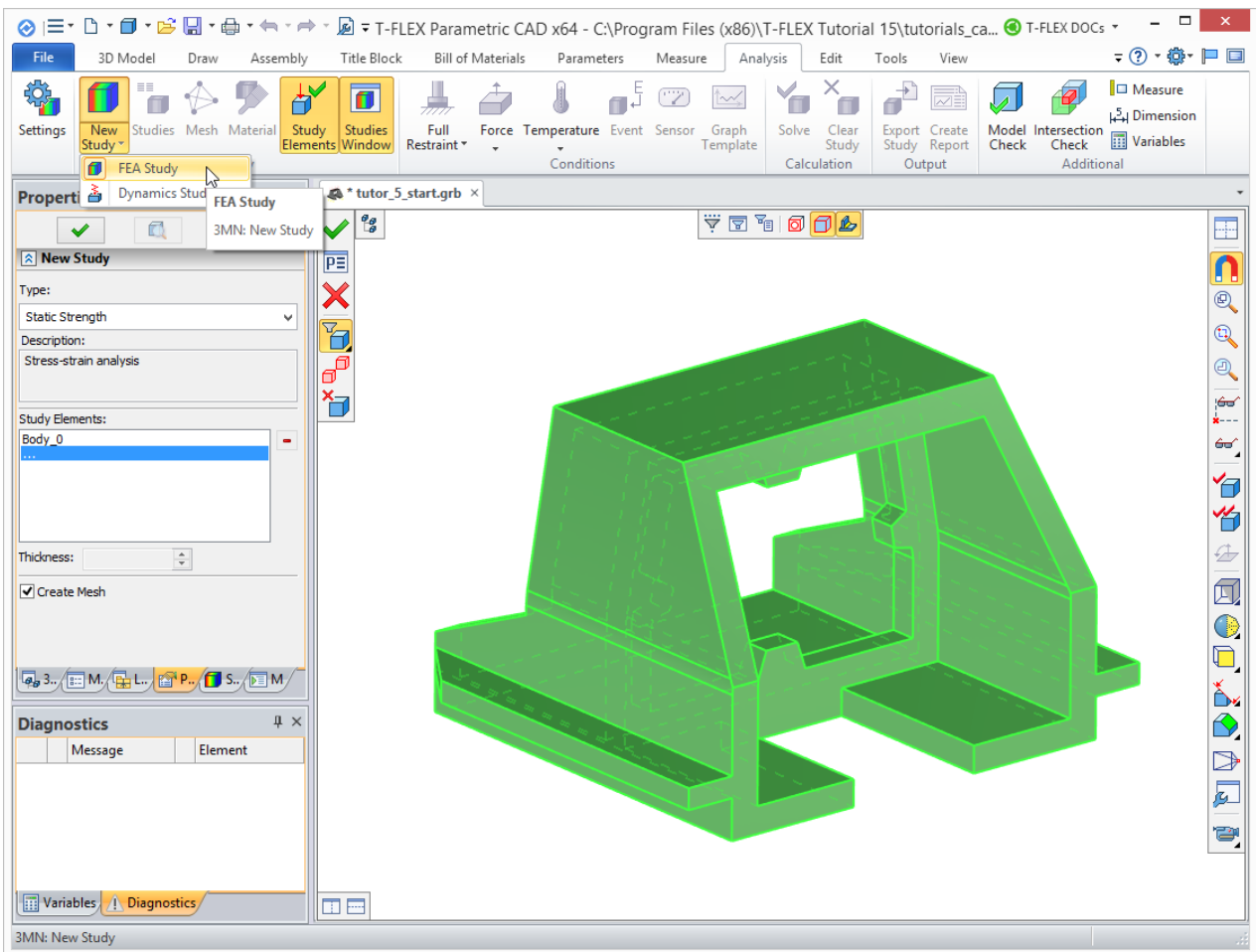

Creating finite element analysis study

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.
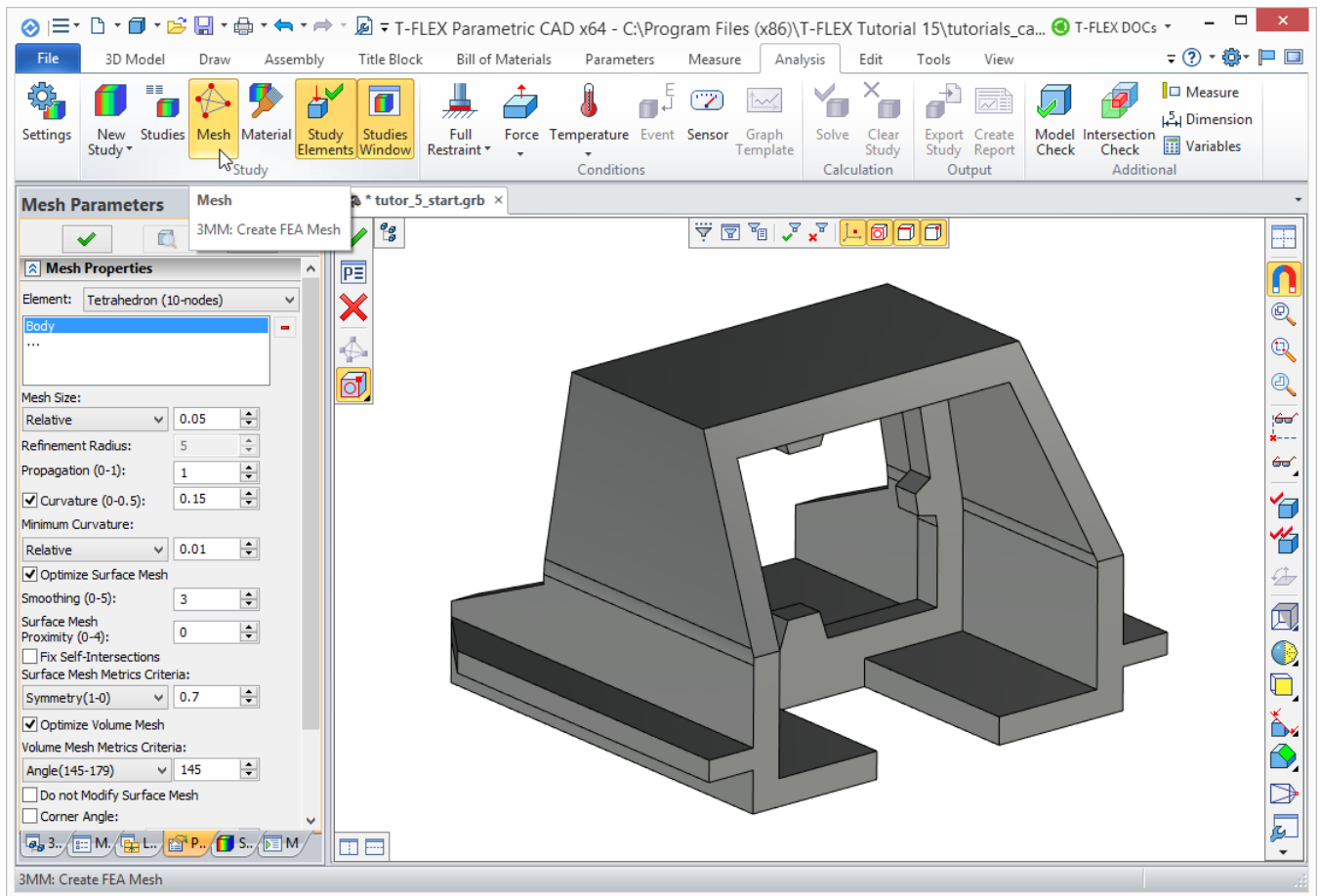
The **Analysis > Show Studies Window** command opens the study window, which displays the studies presented in the current document and their elements in a tree view.

The just created study becomes active. The newly created study elements and the issued Analysis commands will pertain to the active study.
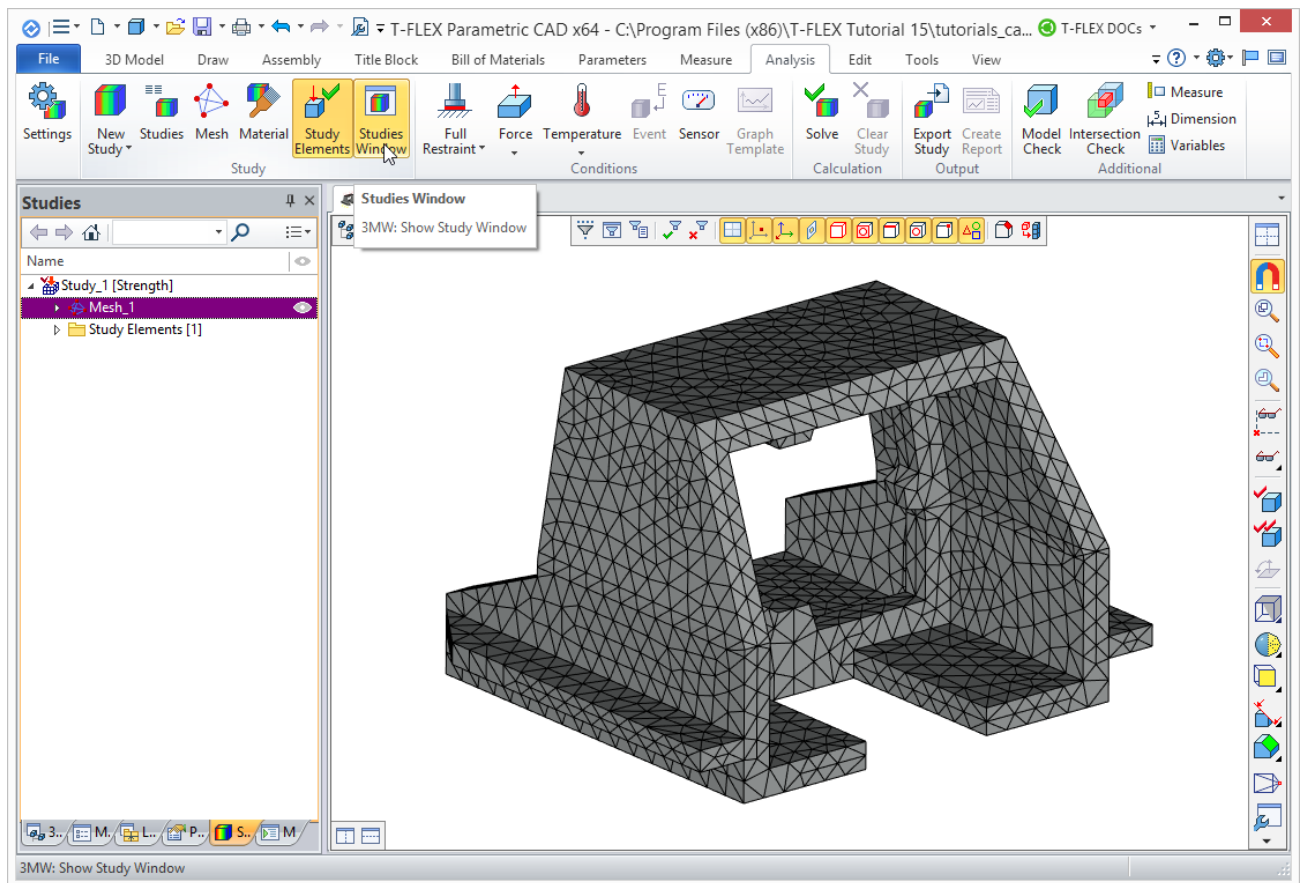
If there are many different studies in the document, then only one of them can be active. Switching an active study is done via the context menu accessed by 🖰 understudy name. The «Activate» command is provided for inactive studies.
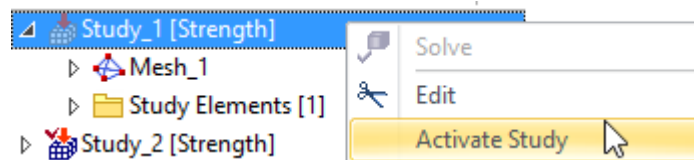
Step 3. Assigning Material

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 properties window. To check or modify the material in this case, called the operation's properties window from the context menu by 🖲 on the three-dimensional body, resulting from the operation, or on the operation name in the studies window.
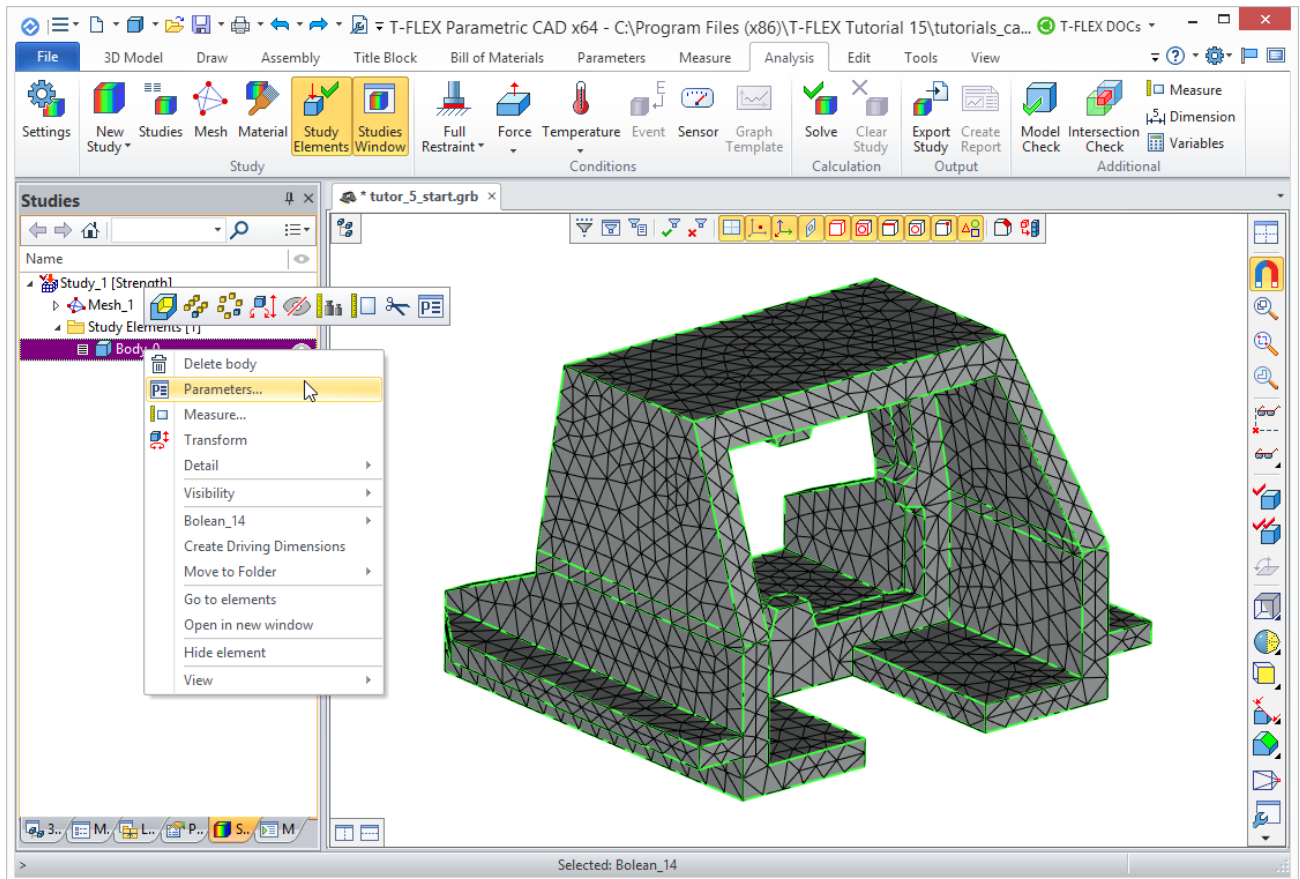
The operation's properties dialog lets you select a material from the standard T-FLEX CAD material library. If necessary, the user can add to the standard T-FLEX CAD materials database one's own materials and modify properties of any materials in this library.

> For detailed information on handling materials in T-FLEX CAD, refer to T-FLEX CAD documentation, the book «Three-Dimensional Modeling», «Materials» Chapter.
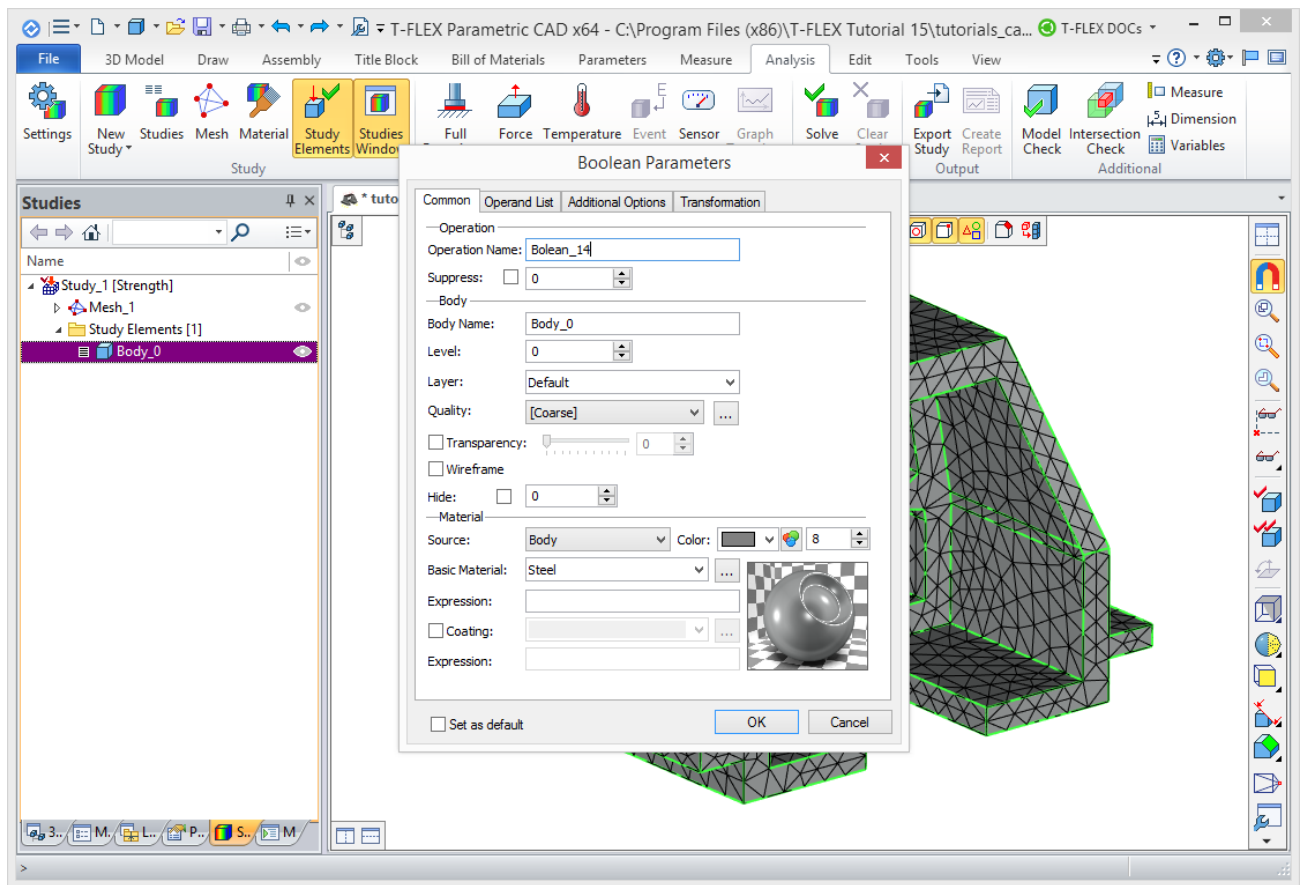
Access to the Analysis material database from the current study is provided by the command **Material** on the Ribbon or by the context menu of the studies tree, displayed in the studies window.
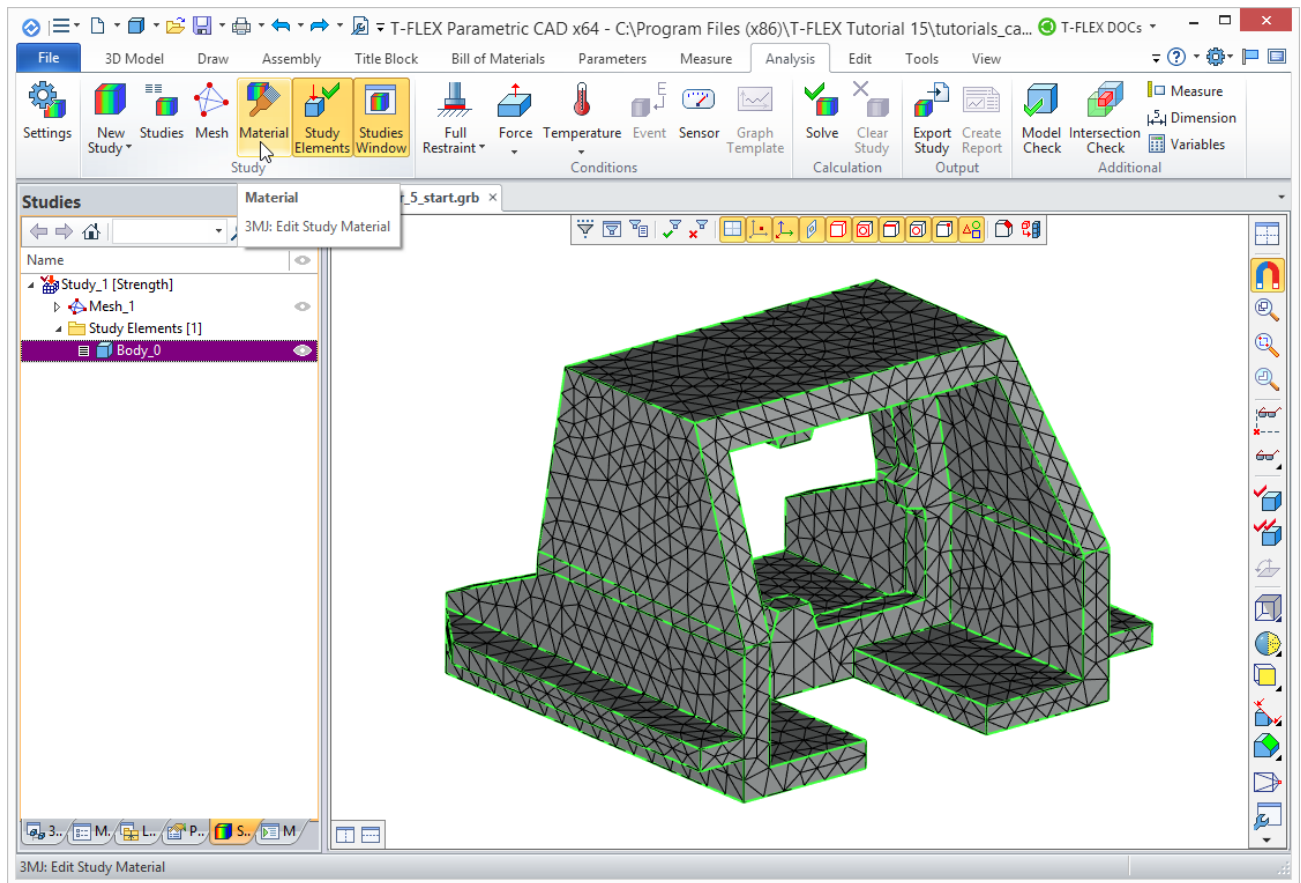
Let's assign the material «Steel/AISI 1020» from the T-FLEX Analysis materials database for the model under consideration.
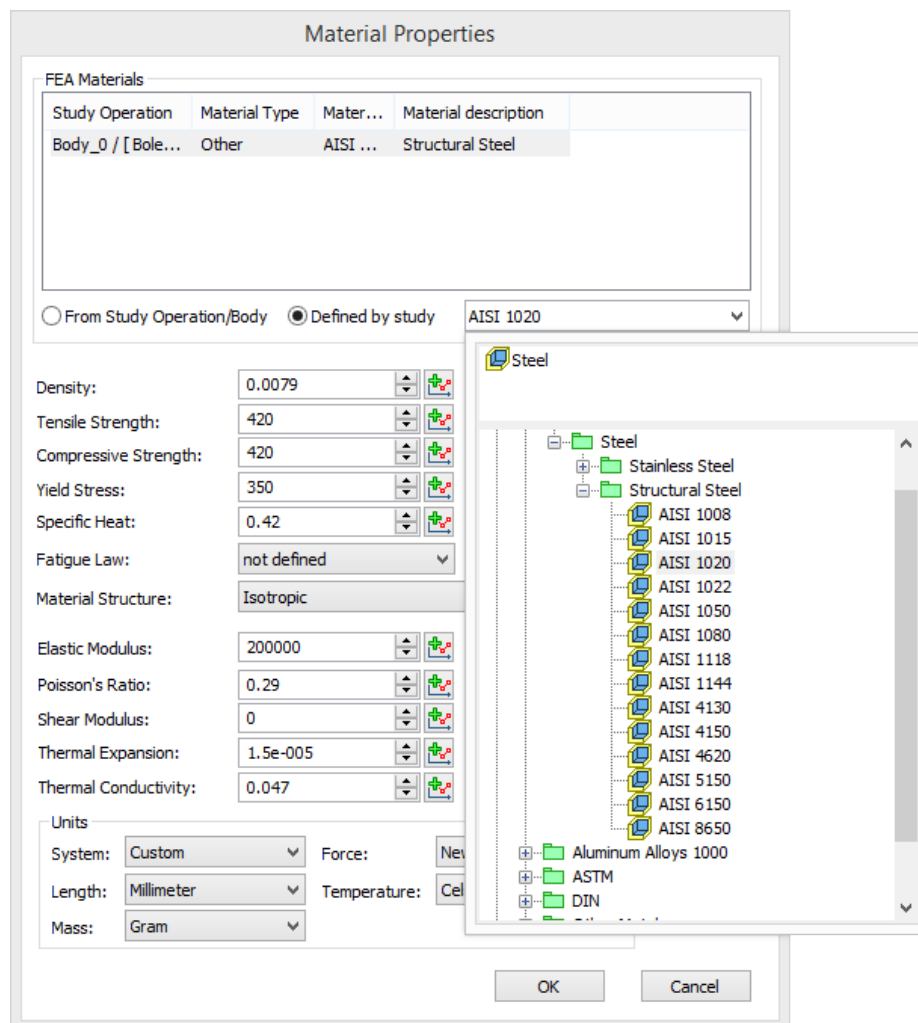
Step 4.1 Applying Boundary Conditions. Defining Restraints

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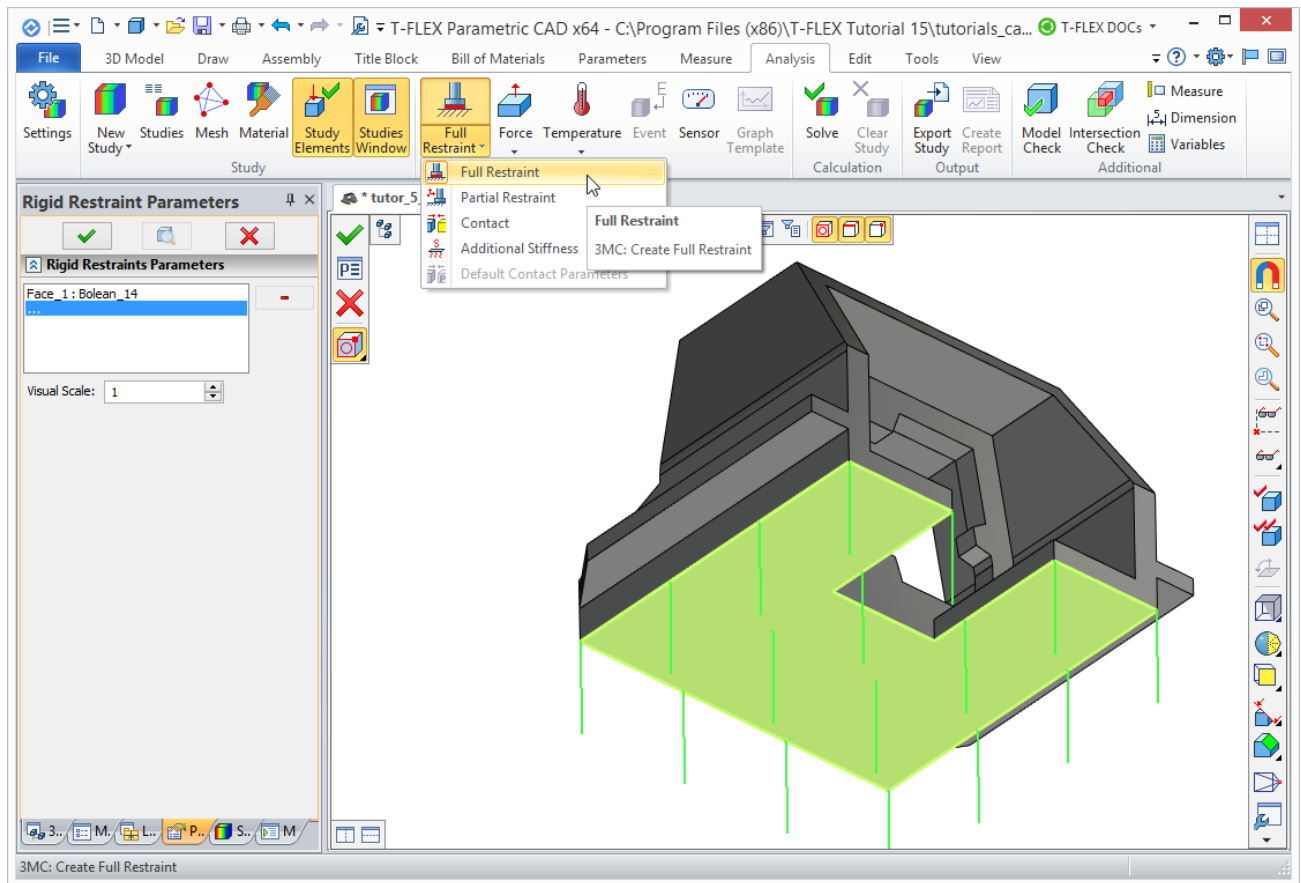
When defining boundary conditions, the finite element mesh gets automatically hidden in order to let you apply boundary conditions to elements of the three-dimensional solid model (faces, edges, vertices).

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.
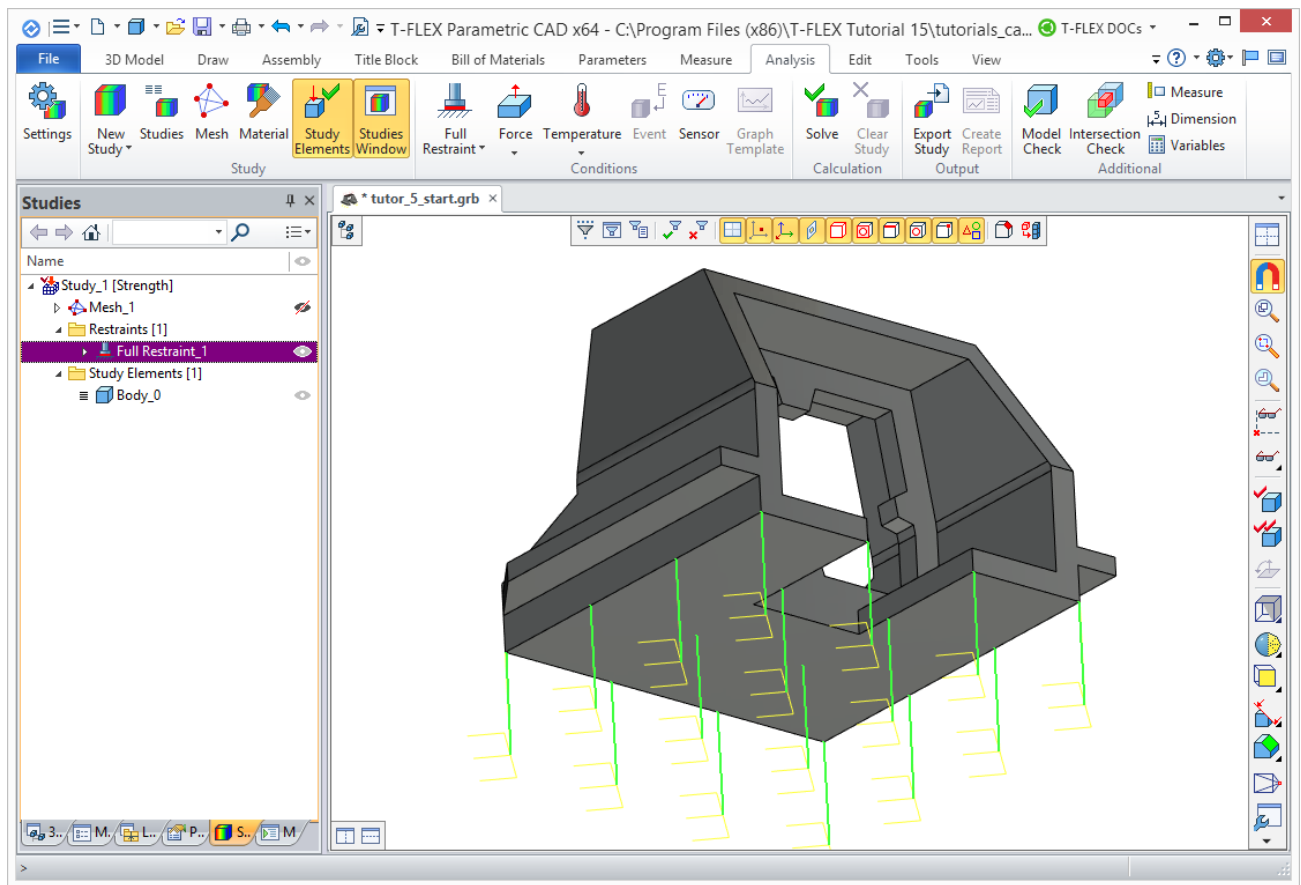
Step 4.2 Applying Boundary Conditions. Defining Loads

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 > Load > Force** command, select a face of the «Body», to which the load is applied. In the command's properties dialog, specify the force value in the «Value» field (550 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.
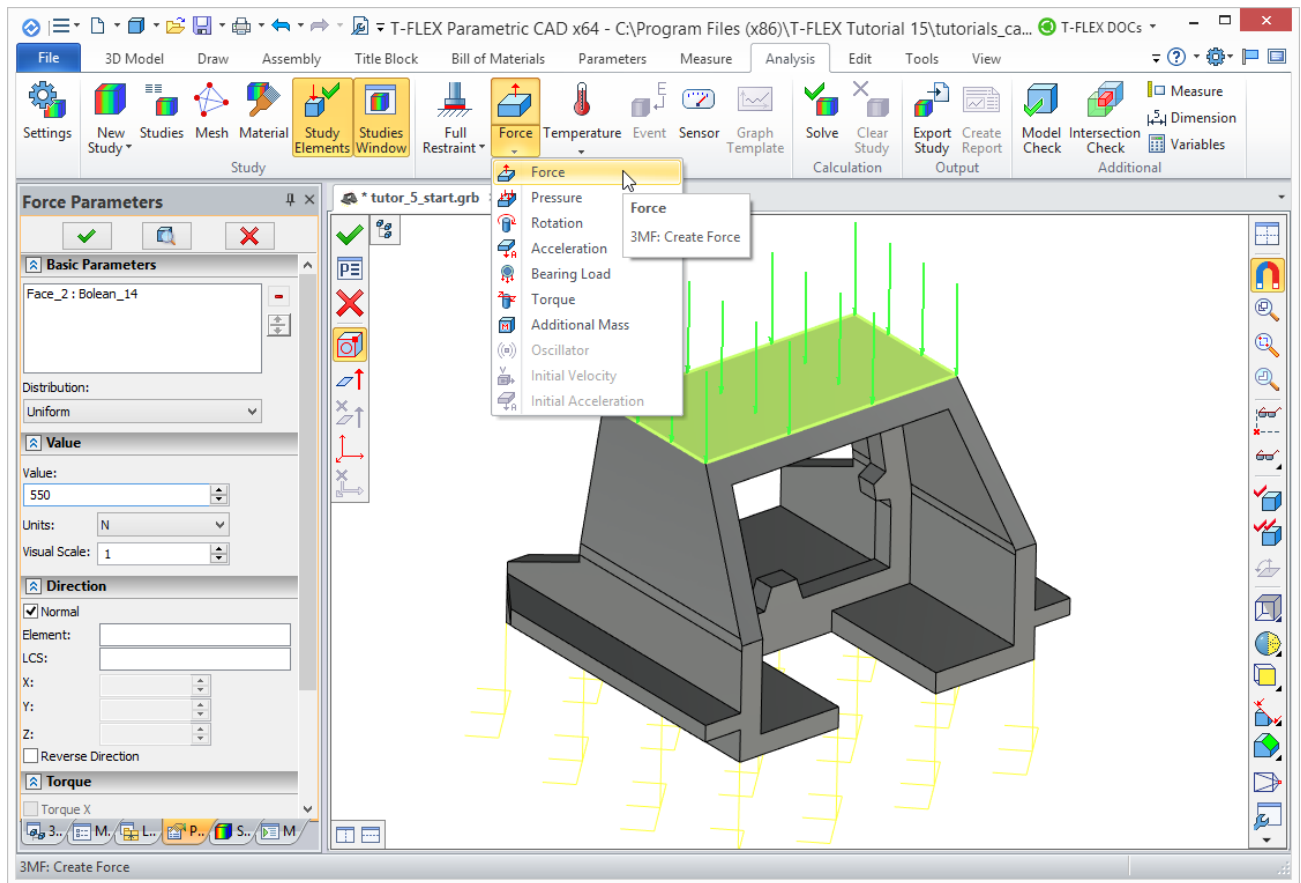
Upon completion of the loads creation command, the introduced loads are shown by special marks on the three-dimensional model of the part, applied to the appropriate model elements.
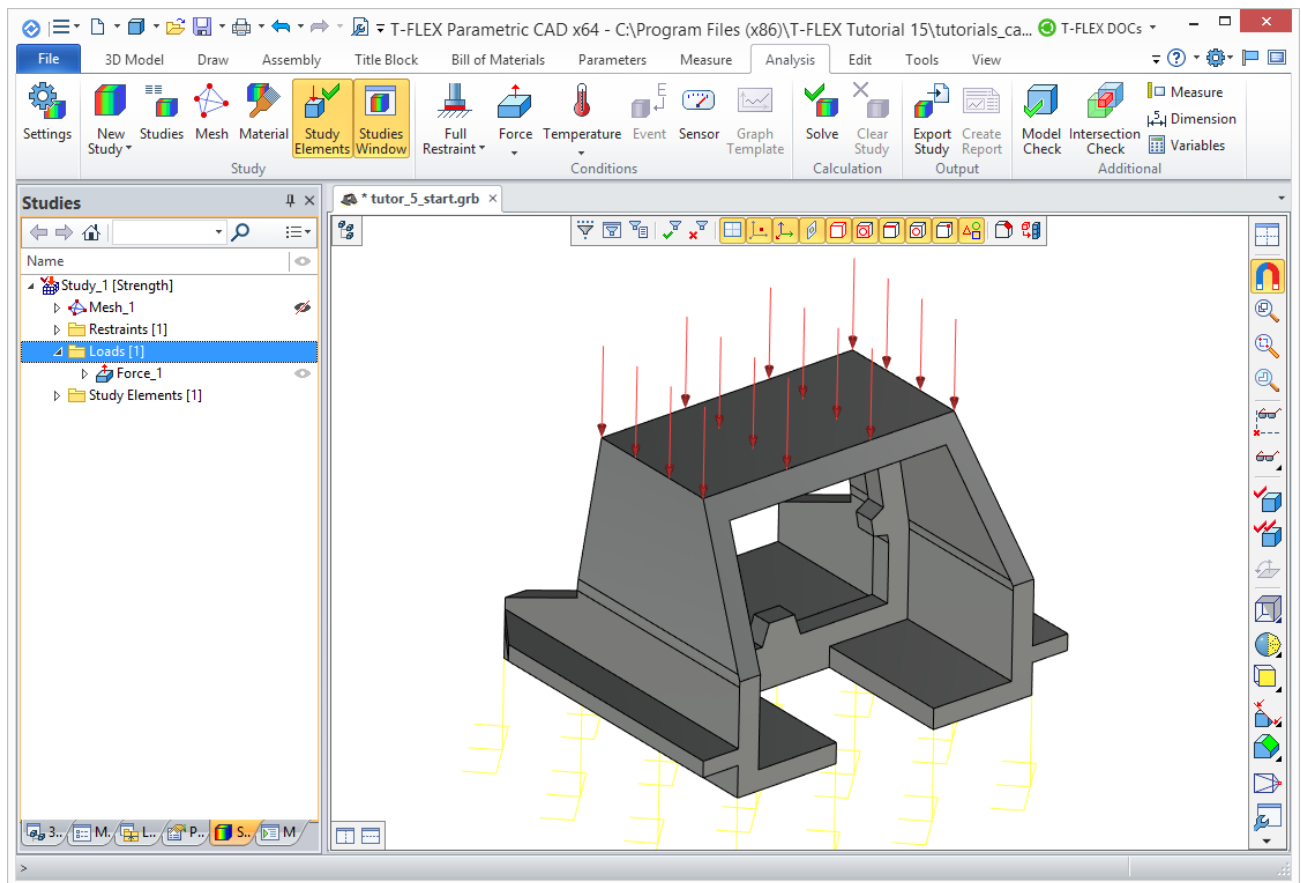
Upon a successful completion of the loads creation command, there are all four elements in the studies tree, required for running the calculations:

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

Step 5. 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.
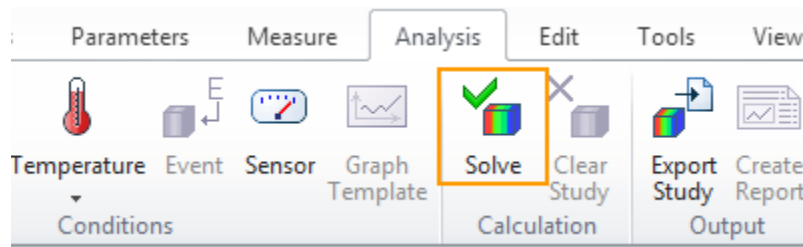
The «Solve» command can also be accessed from the context menu of the respective study in the studies tree displayed in the studies window.

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 properties 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.
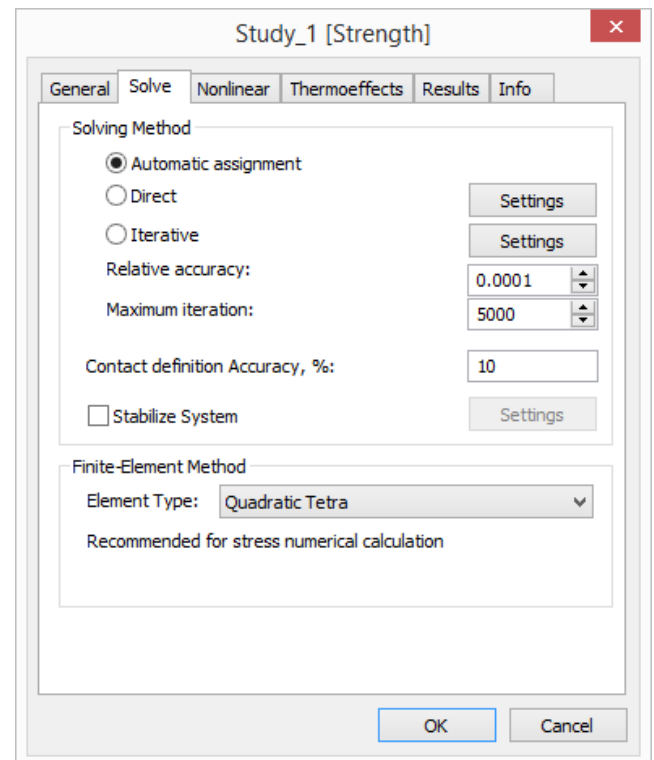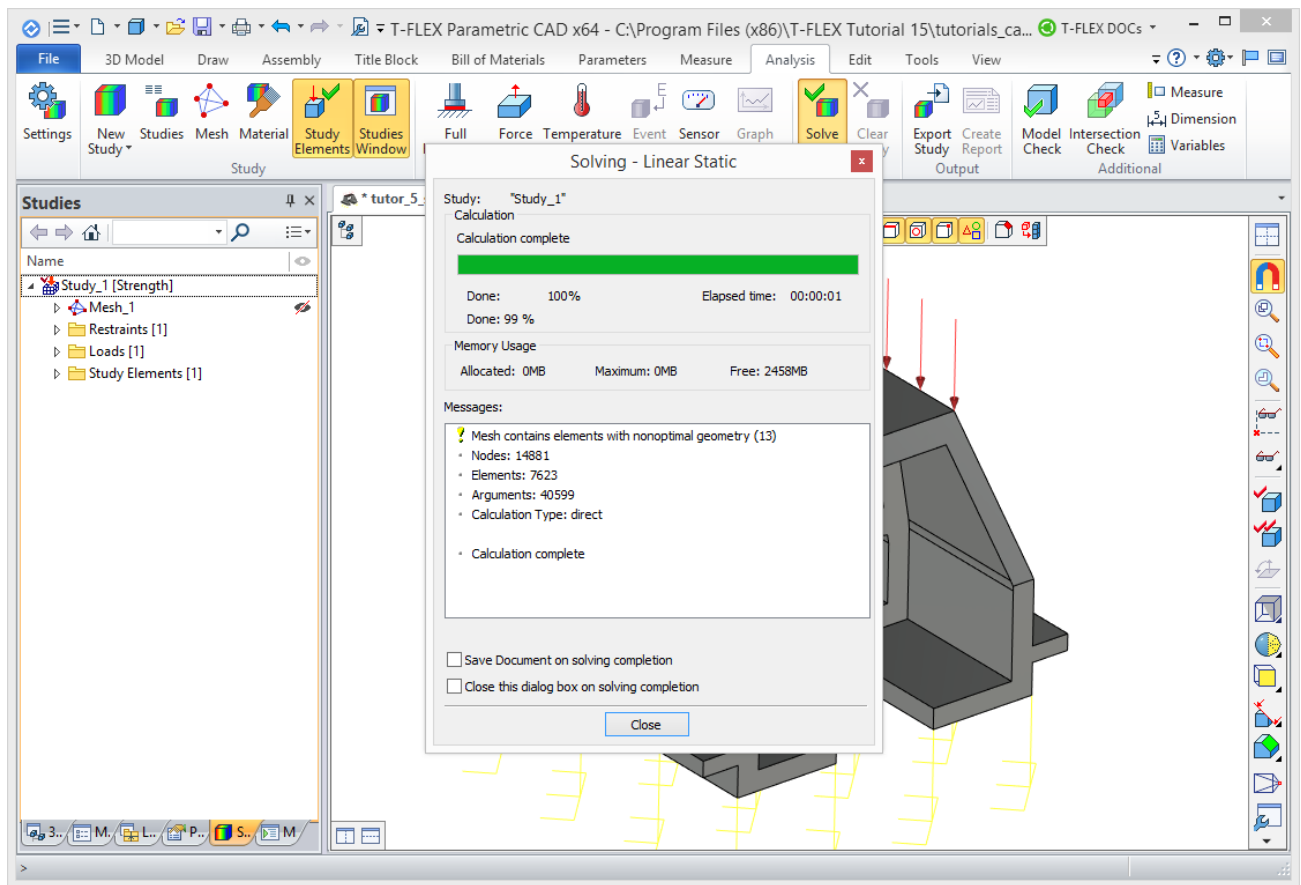
Step 6. Analyzing Calculation Results

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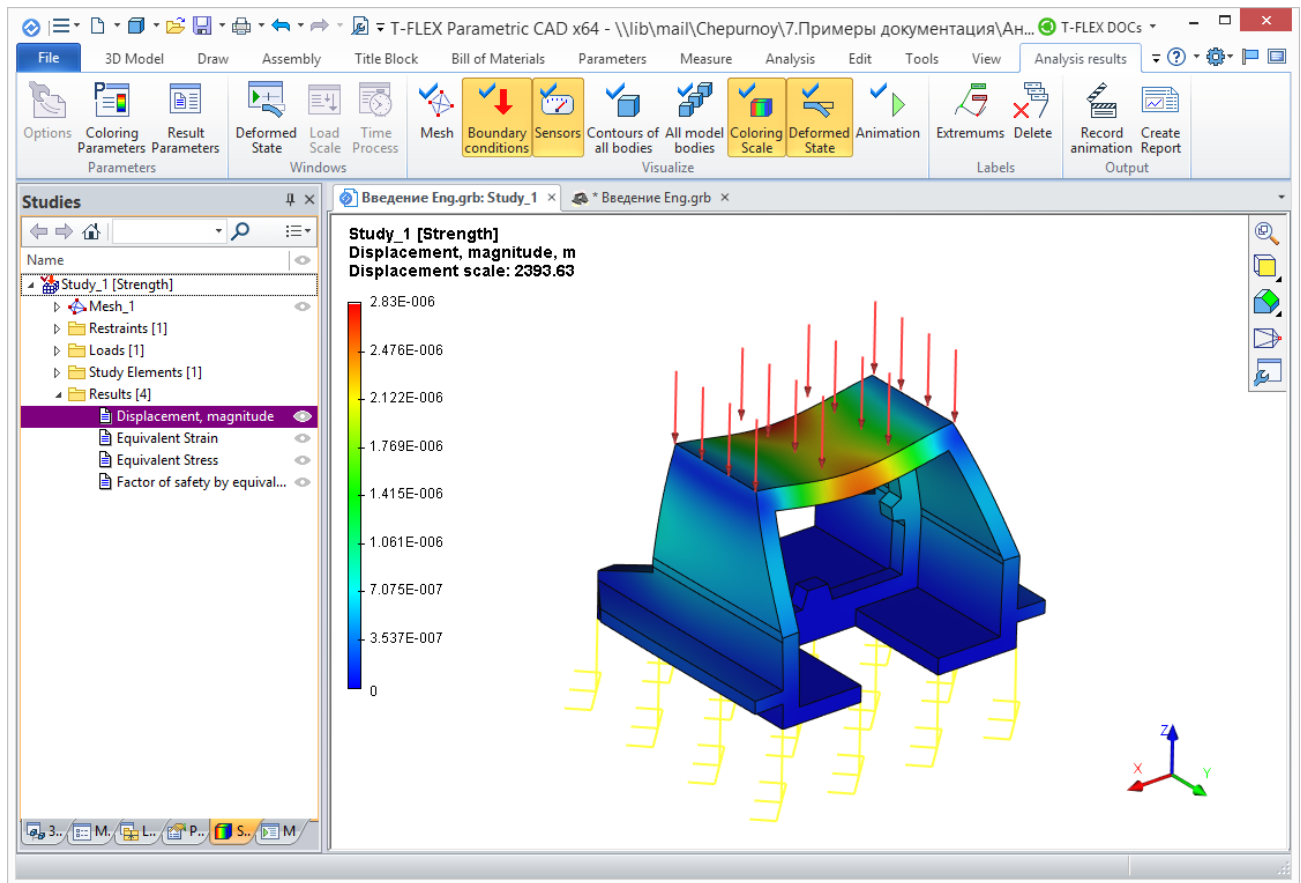
Meshed model display management. The functionality is accessed by double-clicking in the solution viewer window or by the context menu command **Properties.** The user can specify various modes of displaying calculation results – over the mesh, without mesh, with or without displaying contour of the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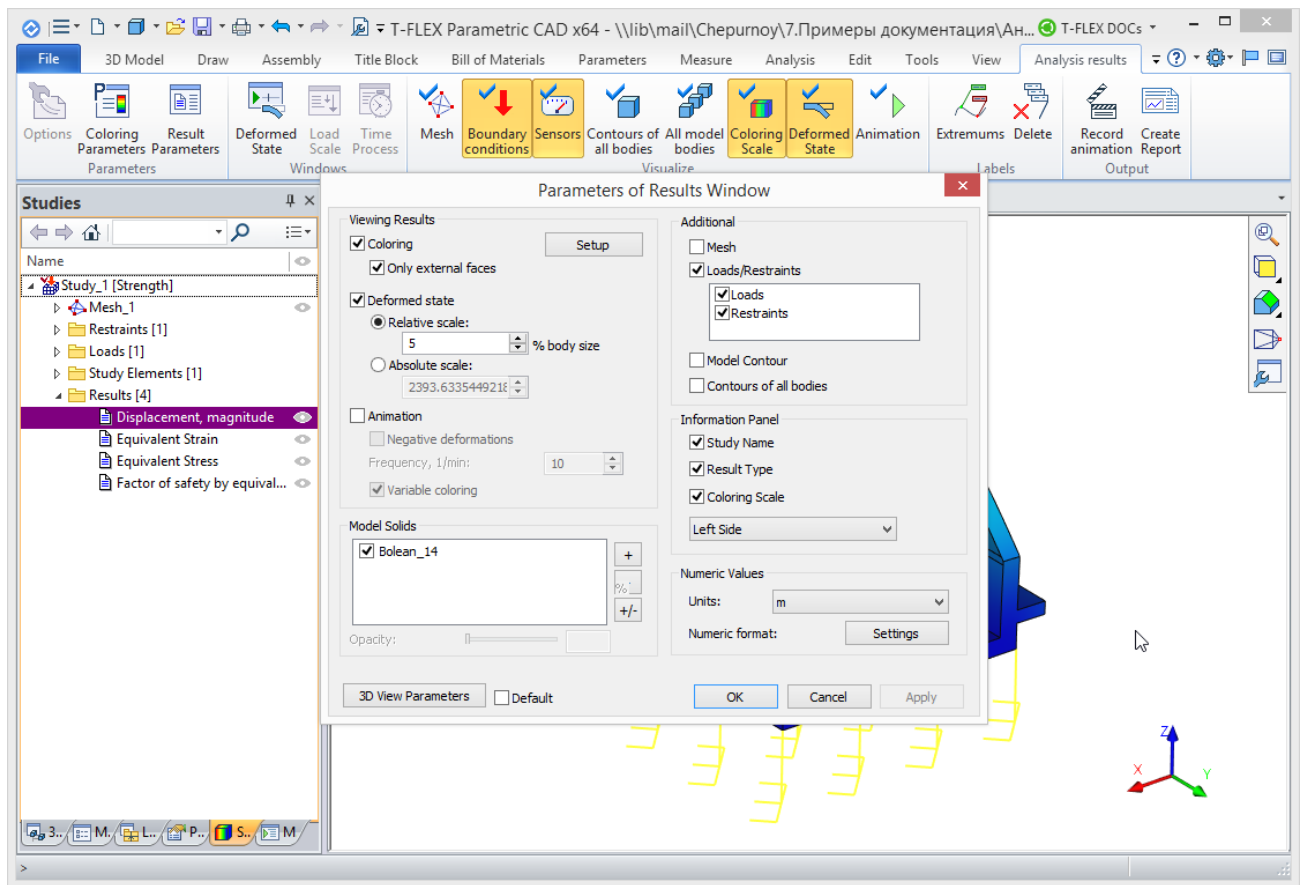
Animation parameter allows recreating the studied model behavior under a smoothly varying loading, with simultaneous display of stress or displacement fields corresponding to the varying load.

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.
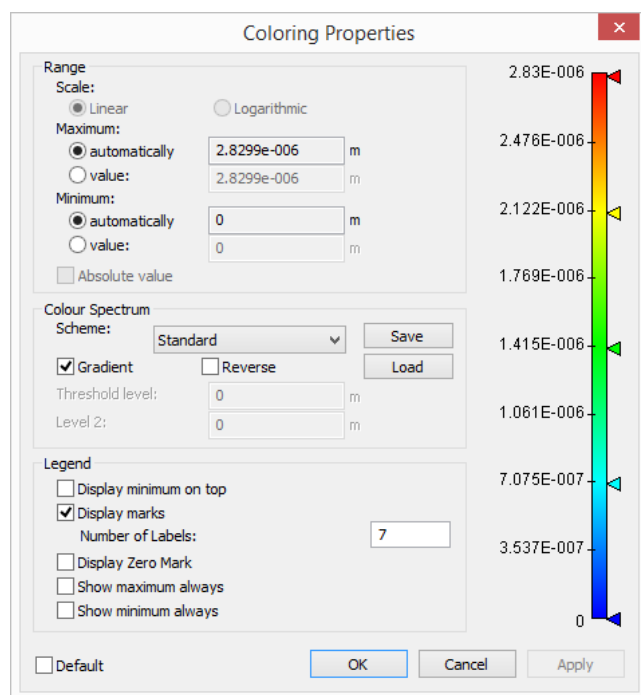
Dynamic result sampling. T-FLEX Analysis Postprocessor offers a convenient capability for sampling the result directly under the mouse pointer. The user needs to simply point the mouse at the location of interest on the meshed model, and the exact result value will be displayed at that spot. Sampling 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».
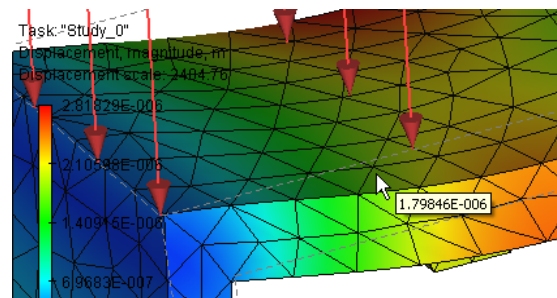
Creating report – results of a solved study can be saved as a separate electronic document. The dialog for generating a report of the active study is accessible via the Ribbon or from the **Report...** context menu item of the study selected in the studies tree.
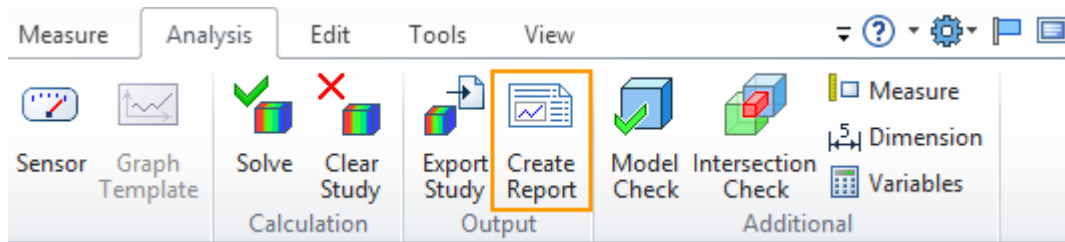
PREPARING FINITE ELEMENT MODEL (PREPROCESSOR)

The main purpose of the T-FLEX Analysis Preprocessor is preparing initial data for the physical study 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 study. 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 study 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.

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 studies 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.
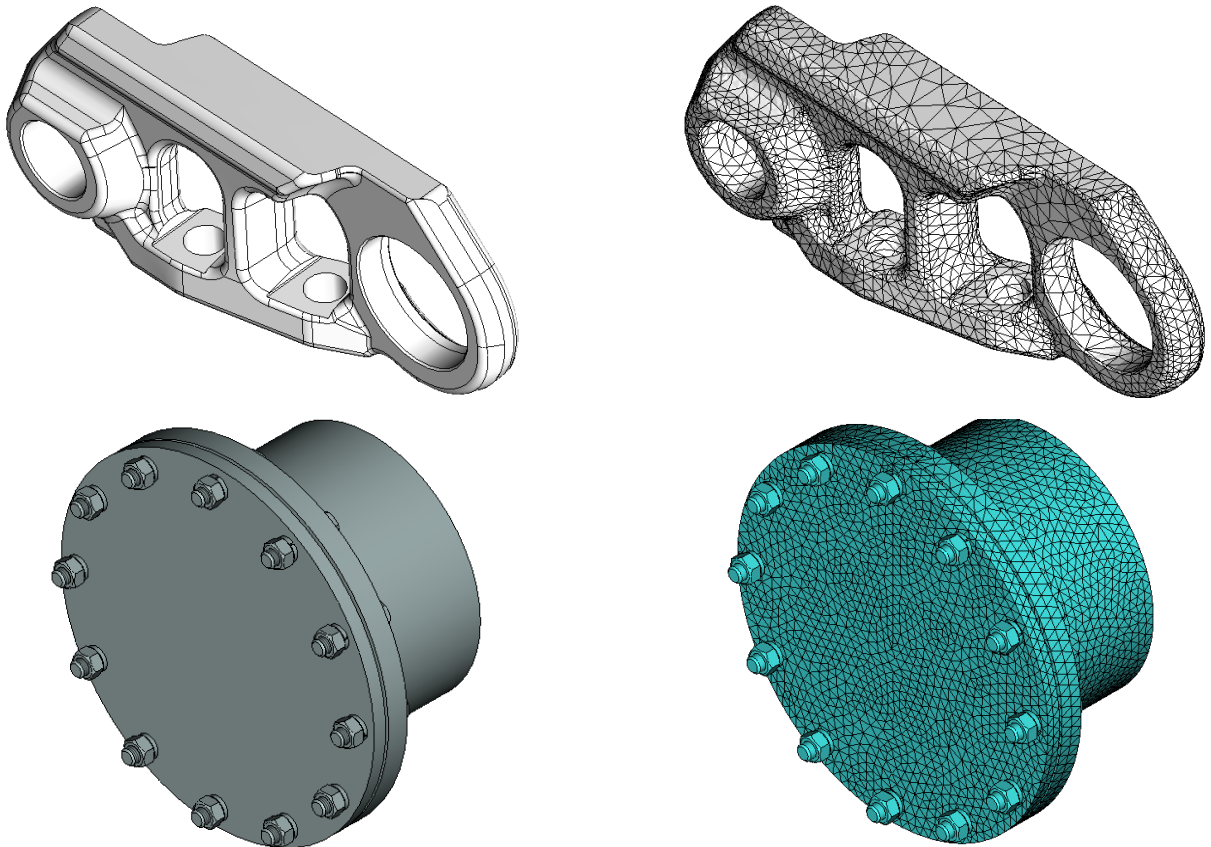
Typical mechanical engineering objects and their tetrahedral finite element models

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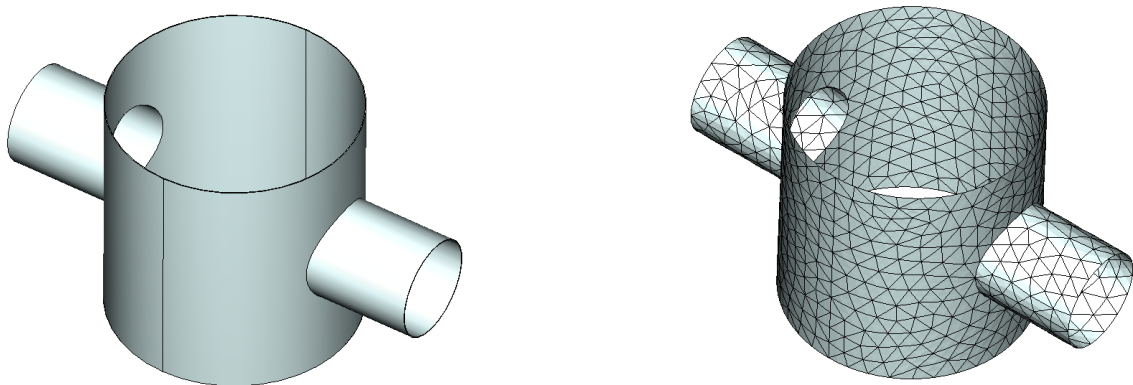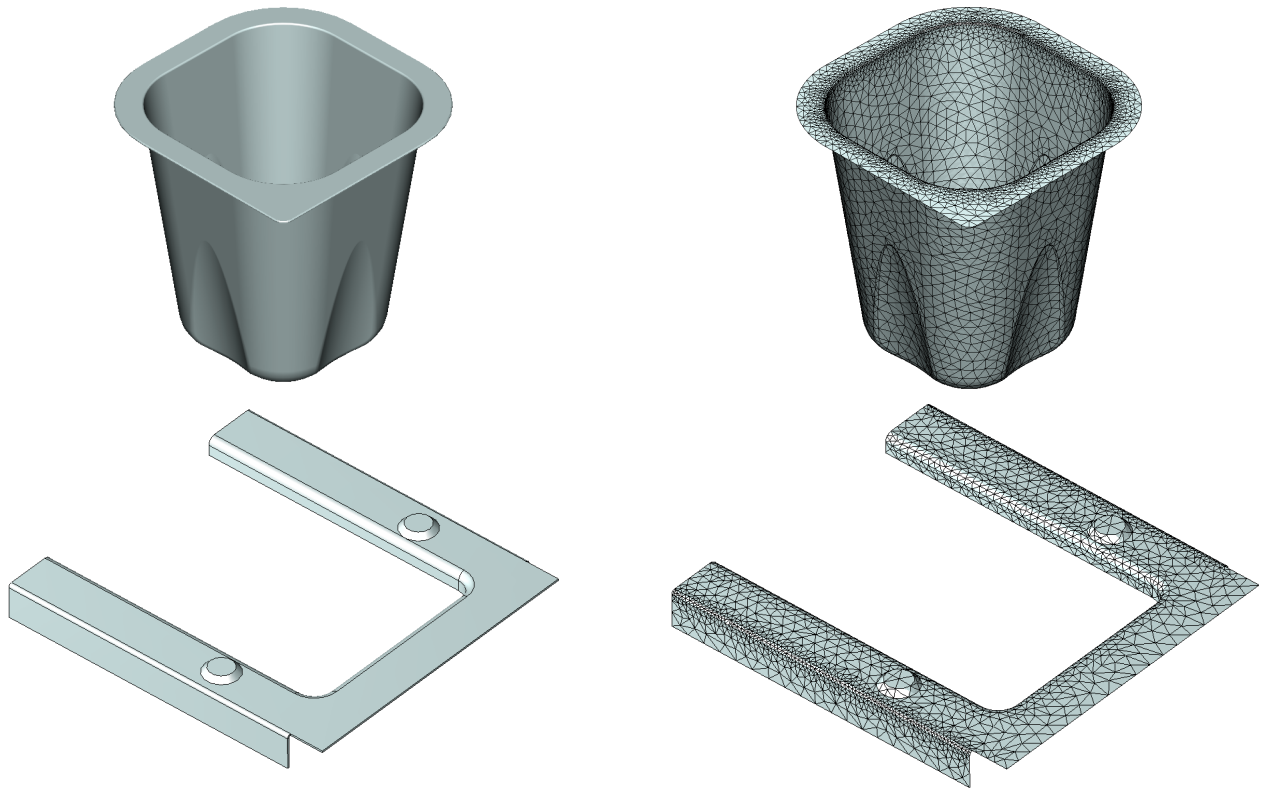
Examples of thin-walled structures and their laminar finite element models

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.
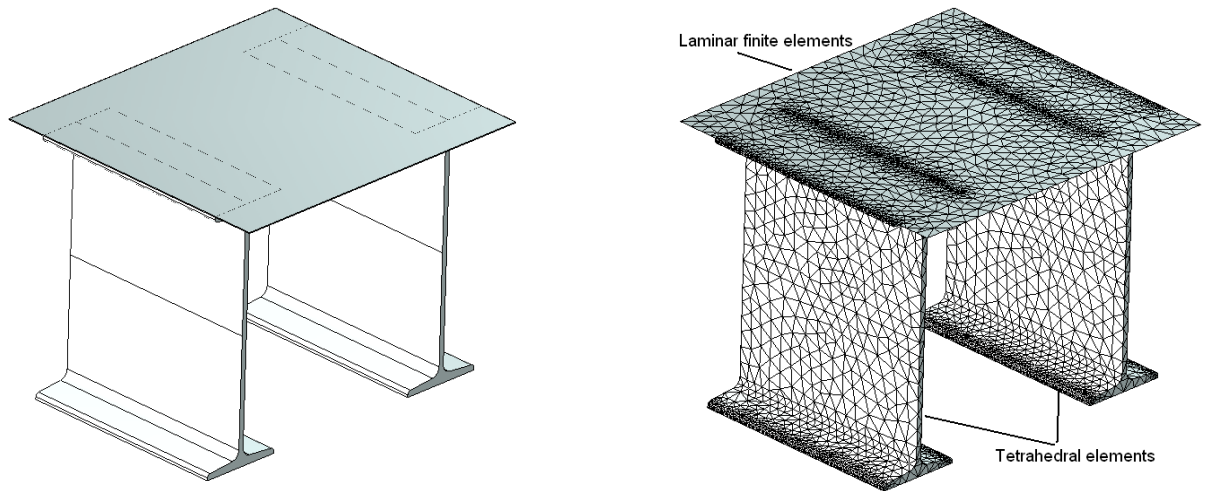
Examples of structures and their hybrid finite element models

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.
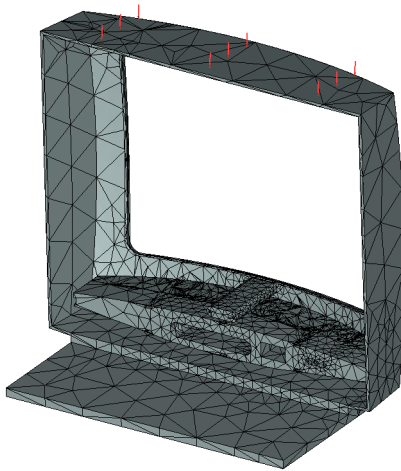
«Poor» mesh of finite element model   «Good» mesh of finite element model
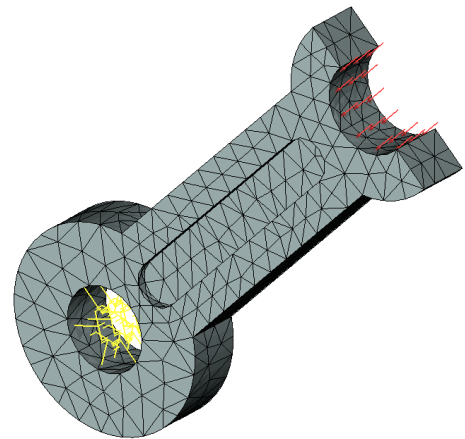
Thus, a user needs to control «quality» of the constructed finite element model based on a visual inspection or with the help of «Grid settings», aiming at possibly more uniform shape distribution of the elements involved in the mesh.
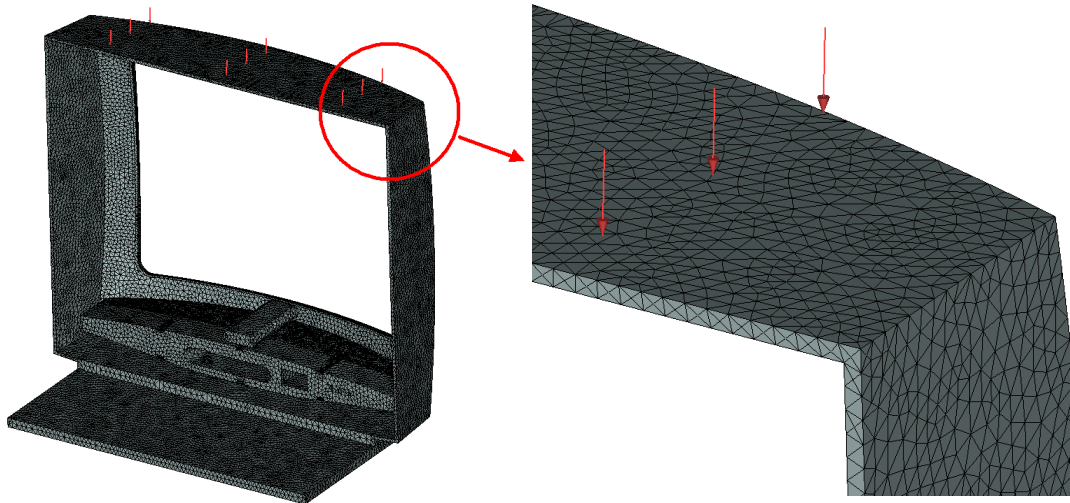
More adequate mesh obtained after using mesh parameters settings

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.
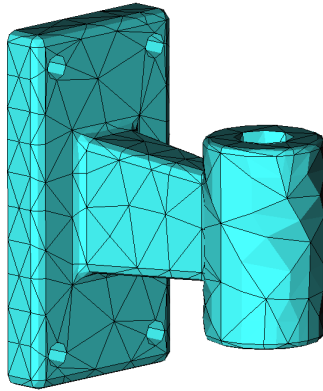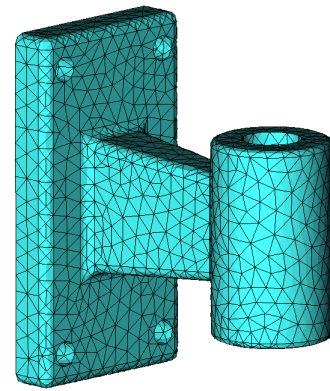
Relative size of 0.2	Relative size of 0.05

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 study being modeled.

TYPES AND ROLE OF 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 speed,

- Initial acceleration.

Specified in the study properties:

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

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

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

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

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

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

In the case of the **Thermal Analysis** study 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 load;
- «Thermal contact» constraint.

The essence of a physical study 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 study.

MANAGING «STUDIES», STUDIES MANAGEMENT COMMANDS

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.
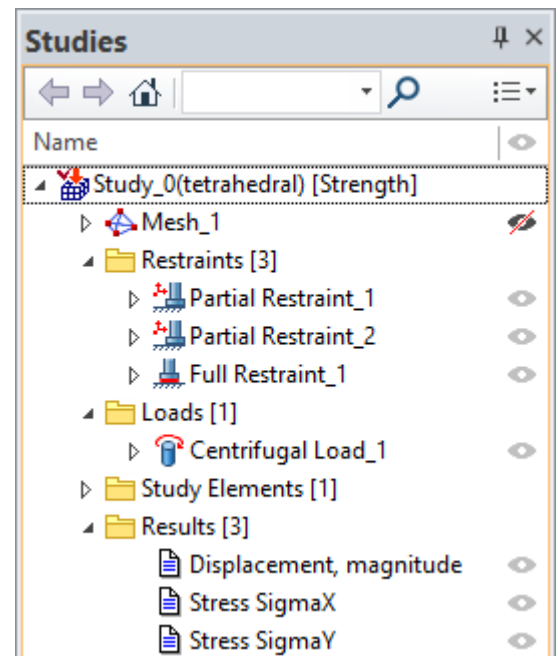
Several studies can be created in one document for running different calculations. The study currently being worked on is called active. The active study's icon has a red check in the studies window. To make another study active, use the context menu command «Activate».

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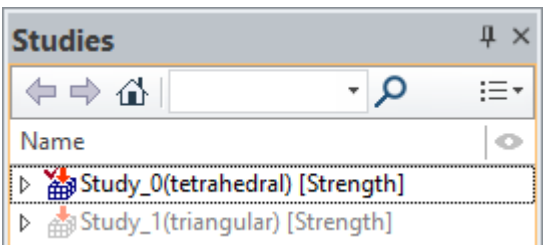
A special command is provided for managing the list of studies:

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.
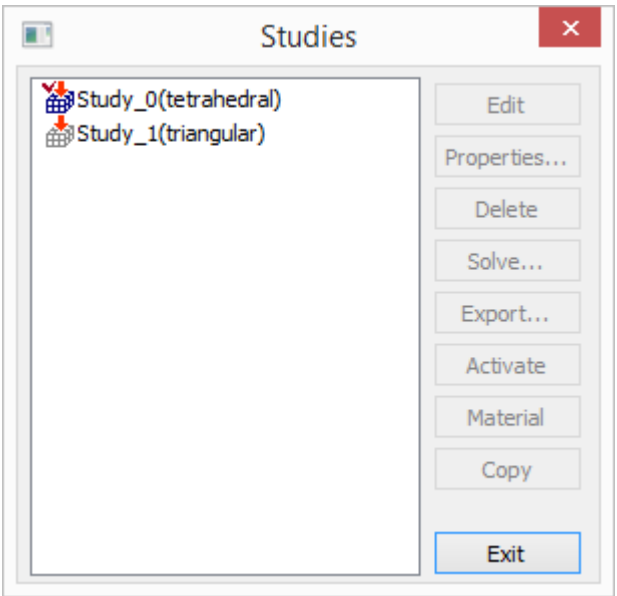
To create a new study, use the command:

Icon	Ribbon
	Analysis → Analysis → 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 properties window.

At this stage, all you can do is specifying the type of the new study. Once a study is created, its type can be changed only under the condition of clearing the study data (including the loss of the calculation results). A study is created based on one or several 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:
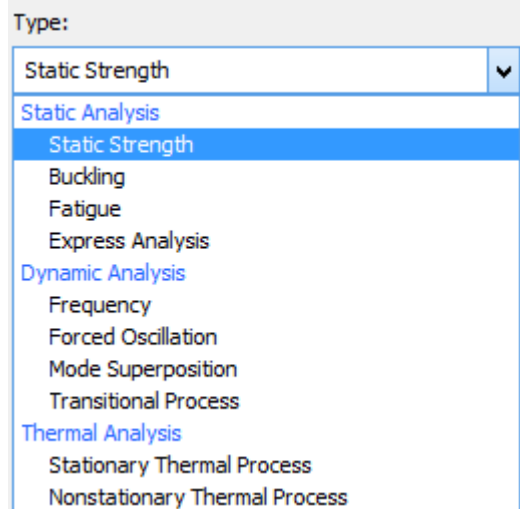
	<E>	Click to select Element (Body, Face)
	<S>	Select All Element

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».
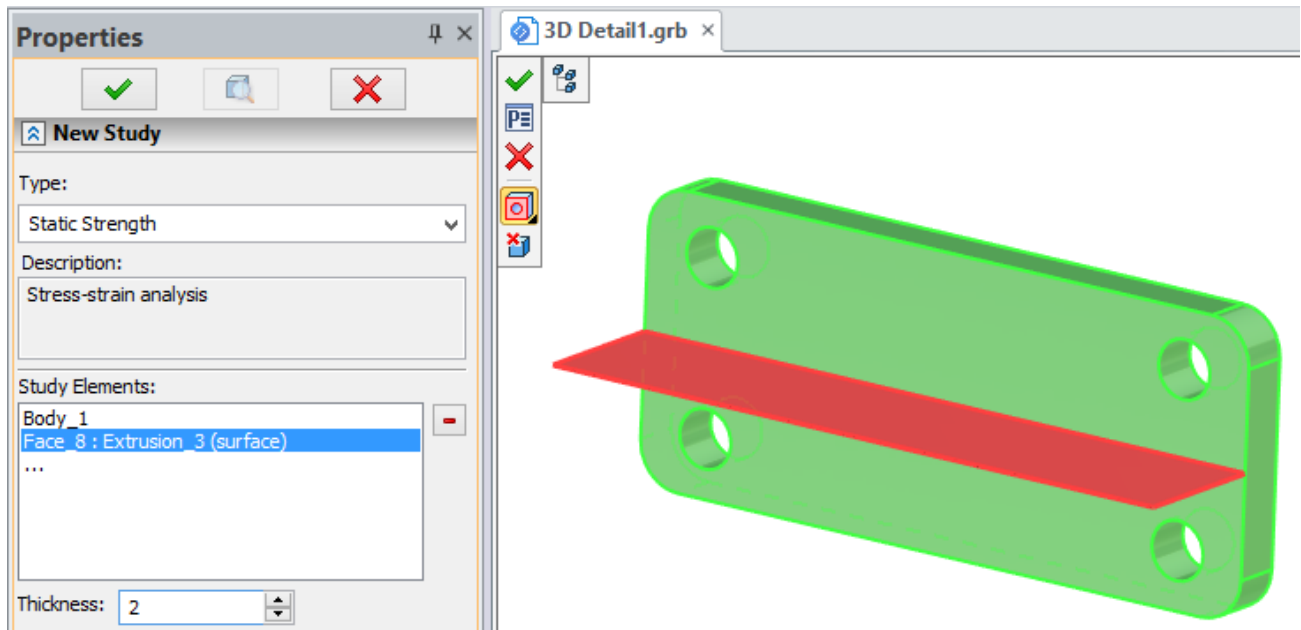
The system allows using several selected elements of the study in analysis, including elements of different types – bodies and faces (in this case the result is the so-called «hybrid model», consisting of shells and solid bodies). Given that, all elements of the study are treated as a single whole (similar 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 properties and solving methods – depending on the chosen type. Other study settings can be edited after its creation in the properties dialog. A study's properties 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 Properties of Studies

A number of similar properties exists in all types of studies, defined on the [**General**] and **[Results]** tabs in the parameters dialog.
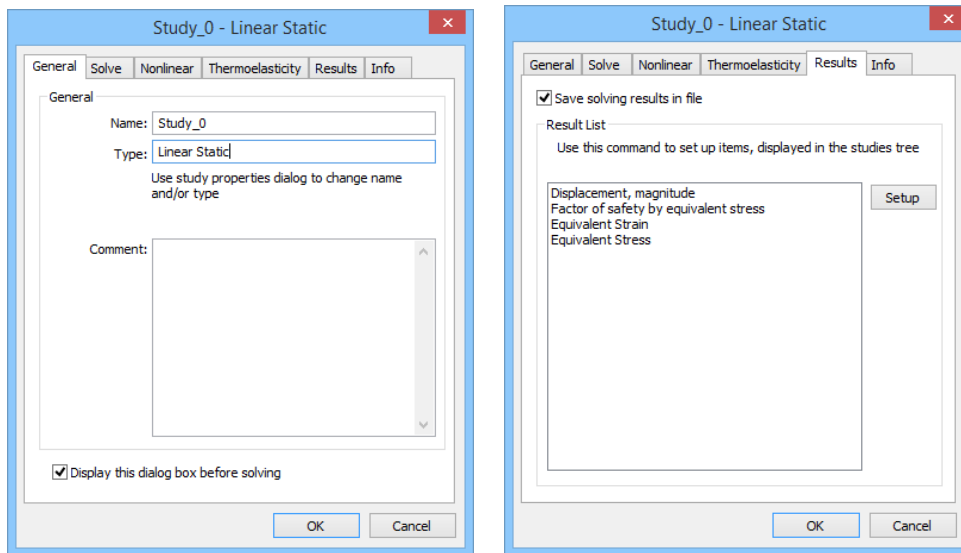
On the tab **[General]** the user can specify the study's name, modify its type (Static Analysis, Frequency Analysis, Stability Analysis, Thermal Analysis) and enter the comments. Comments are used for recording necessary explanations, and are output at the time of generating a report.

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

On the **[Results]** tab the option of saving the study'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 studies with large volumes of results, and also can speed up the saving of a document (this depends on operating system).

This list can be set up in the dialog accessible by clicking the **[Options]** button. 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.

Dialog for setting up the list of Results displayable in the studies tree

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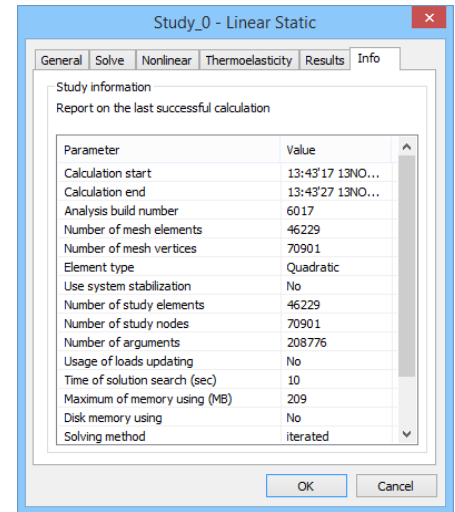
DEFINING MATERIAL

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.

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.

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 three-dimensional modeling guidebook. Additionally, there is an alternative approach to specifying a study's material properties.

Specifying an individual study's material is done by the command:

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

After calling the command, the dialog window appears.

By default, the switch is set in the «From Study Operation/Body» position. This means that material properties are inherited from the 3D model material. If the switch is moved to the «Other» position, then the input fields for the study's material properties become accessible. One can use the **[Select Material]** pull down list, which contains a set of predefined materials. After selecting a material, its properties are read and appear in the main dialog.
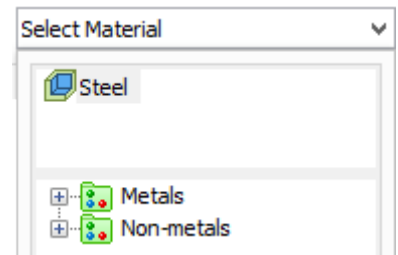
Located in the lower part of the dialog window are the fields for selecting units for main physical measures.

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 properties of the current study to consider temperature dependences.
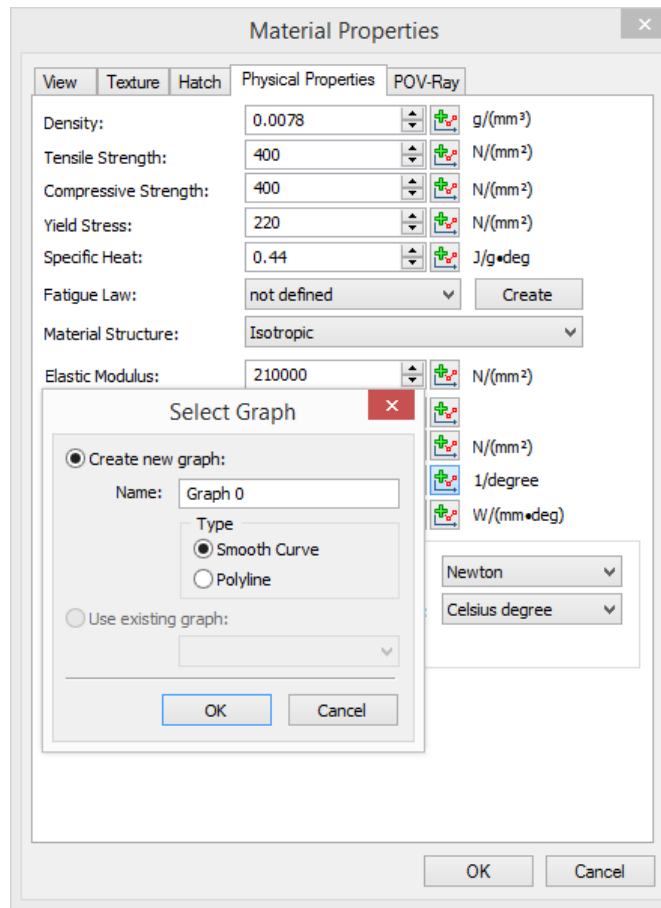
Specifying properties of anisotropic materials

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:

$$\left.\begin{array}{l}
\varepsilon_x = \dfrac{1}{E_1}\sigma_x - \dfrac{v_{21}}{E_2}\sigma_y - \dfrac{v_{31}}{E_3}\sigma_z, \\[2mm]
\varepsilon_y = -\dfrac{v_{12}}{E_1}\sigma_x + \dfrac{1}{E_2}\sigma_y - \dfrac{v_{32}}{E_3}\sigma_z, \\[2mm]
\varepsilon_z = -\dfrac{v_{13}}{E_1}\sigma_x - \dfrac{v_{23}}{E_2}\sigma_y + \dfrac{1}{E_3}\sigma_z, \\[2mm]
\gamma_{xy} = \dfrac{1}{G_{12}}\tau_{xy}, \; \gamma_{yz} = \dfrac{1}{G_{23}}\tau_{yz}, \; \gamma_{xz} = \dfrac{1}{G_{13}}\tau_{xz}
\end{array}\right\}$$

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:

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

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

$$G_{12} = \frac{E_1 E_2}{E_1\left(1 + 2v_{12}\right) + E_2}$$

To define an orthotropic material, it is required in the «Material properties» dialog, which can be opened with the «More...» button, to specify the structure of the material: «Orthotropic». After that, the group of parameters for specifying properties of an orthotropic material will appear.

Material Structure:	Orthotropic			
	X:	Y:	Z:	
Elastic Modulus:	200000	200000	200000	N/(mm²)
Poisson's Ratio:	0.25	0.25	0.25	
Thermal Expansion:	1.5e-005	1.5e-005	1.5e-005	1/degree
Thermal Conductivity:	0.047	0.047	0.047	W/(mm•deg)
	XY:	YZ:	XZ:	
Shear Modulus:	0	0	0	N/(mm²)

In this dialog, the following parameters are specified:

Elastic Modulus:	E_1, E_2, E_3
Poisson's Ratio:	v_{21}, v_{23}, v_{31}
Shear Modulus:	G_{12}, G_{23}, G_{13}

Coefficients of linear thermal expansion along the axes of the coordinate system: $\alpha_1, \alpha_2, \alpha_3$

Coefficients of heat conduction along the axes of the coordinate system: $\lambda_1, \lambda_2, \lambda_3$

The coordinate system specified for each body defines direction of axes of symmetry. The structure of the material: «Orthotropic» 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, it is required, in the context menu of the given body in the tree of the study, select the «Material coordinate system» option, and
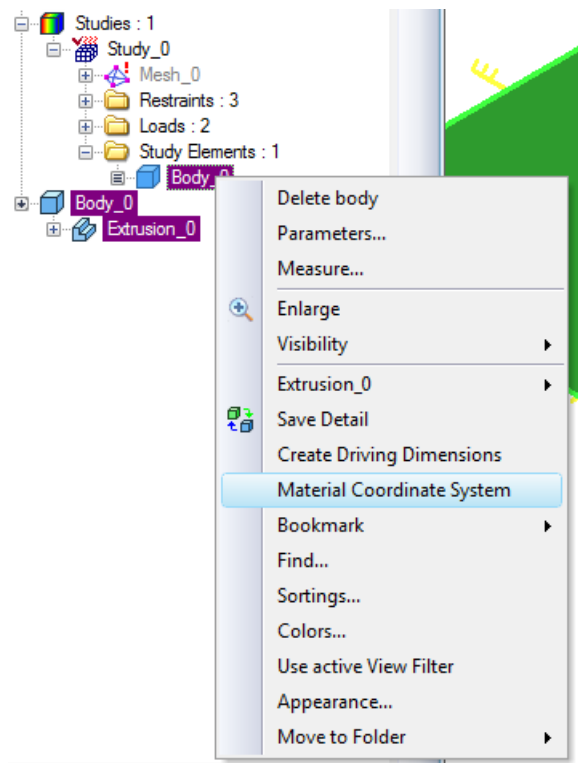
after that, indicate LCS. The selected LCS will be included into the list as the base coordinate system and along the direction of its axes will be selected the axes of symmetry of orthotropic or transversely isotropic bodies.
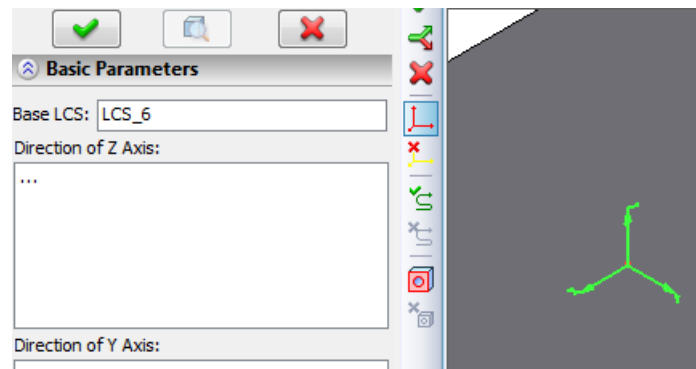
Moreover, if the anisotropic body is deformed or generated by extrusion, it is also possible to specify the law of the change of the direction of the Z-axis (for orthotropic or transversely isotropic bodies) and, in addition, of the Y-axis (for orthotropic body).
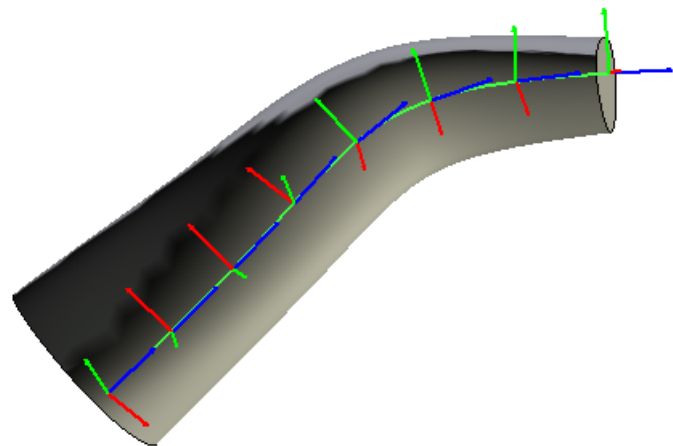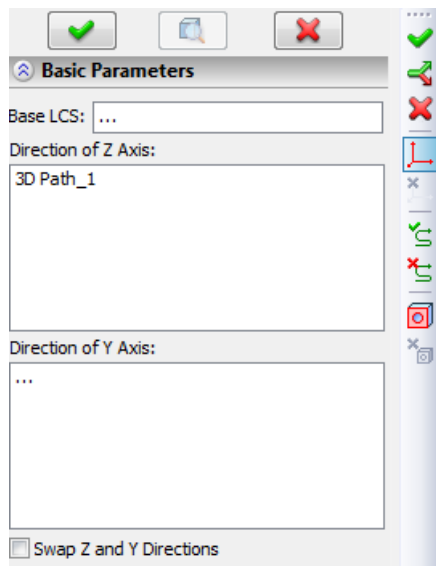
To specify the law of the change of the direction of the Z-axis, it is possible to select the 3D path (without specifying the base LCS) or several paths running together – in the latter case, the direction of the axis of symmetry at each point of the body will be determined according to the direction of the tangent in the nearest point of one of the paths.
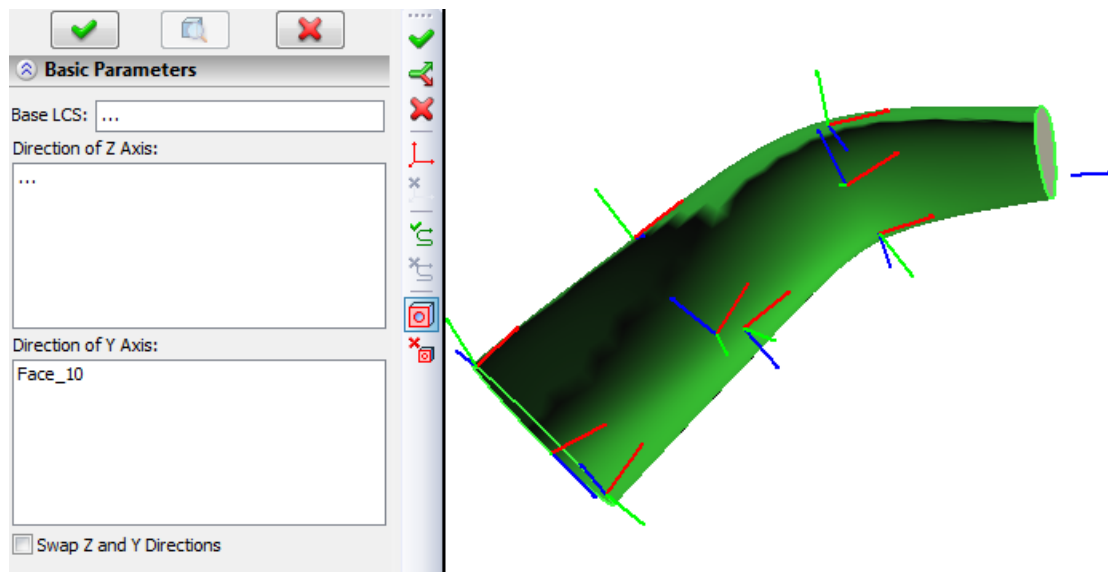
To specify the law of the change of the direction of the Y-axis, it is necessary to select one or several curvilinear or plane surfaces (without specifying the base LCS). The direction of the Y-axis at each point of the body will be determined according to the direction of the normal in the nearest point of the surfaces.
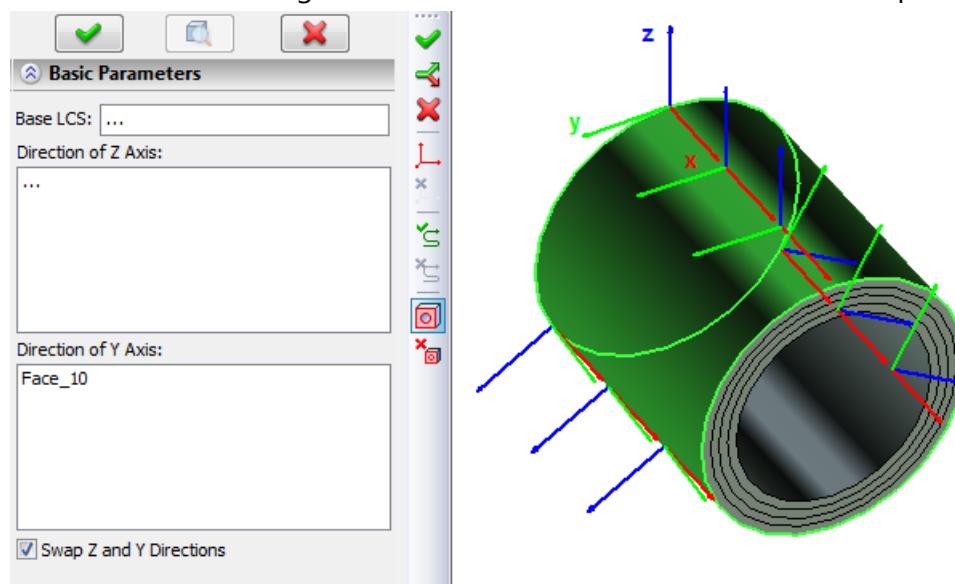
The «Swap Z and Y directions» switch serves for using instantaneous XY-axes, located in the plane tangent to the selected surface, as a plane of symmetry of a transversely isotropic body (see below). In this case, parameters E, ν, G will be related to the XY-plane, and parameters E', ν', G' – to all of the planes containing the Z-axis.

By calling the same command for a body in the tree of the study, it is possible to infer which LCS is related to the given orthotropic body.

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:

$$\left. \begin{array}{c} \varepsilon_x = \dfrac{1}{E}\left(\sigma_x - v\sigma_y\right) - \dfrac{v'}{E'}\sigma_z, \\[2mm] \varepsilon_y = \dfrac{1}{E}\left(-v\sigma_x + \sigma_y\right) - \dfrac{v'}{E'}\sigma_z, \\[2mm] \varepsilon_z = -\dfrac{v'}{E'}\left(\sigma_x + \sigma_y\right) + \dfrac{1}{E'}\sigma_z, \\[2mm] \gamma_{xy} = \dfrac{1}{G}\tau_{xy}, \quad \gamma_{yz} = \dfrac{1}{G'}\tau_{yz}, \quad \gamma_{xz} = \dfrac{1}{G'}\tau_{xz} \end{array} \right\}$$

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

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

Material Structure:	Transversely Isotropic ▼		
	X, Y:	**Z:**	
Elastic Modulus:	210000	210000	N/(mm²)
Poisson's Ratio:	0.28	0.28	
Thermal Expansion:	0.000013	0.000013	1/degree
Thermal Conductivity:	0.043	0.043	W/(mm•deg)
	XZ, YZ:		
Shear Modulus:	0		N/(mm²)

To specify a transversely-isotropic material, it is required, in the «Material's properties» dialog opened with the «More...» button, 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: v, v' ;

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

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

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 with the **Tools > Materials** command of the menu.

2. In the «Material properties» dialog opened with the «More...» button, 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.

CONSTRUCTING MESH

For mesh manipulations, use the command:

Icon	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.
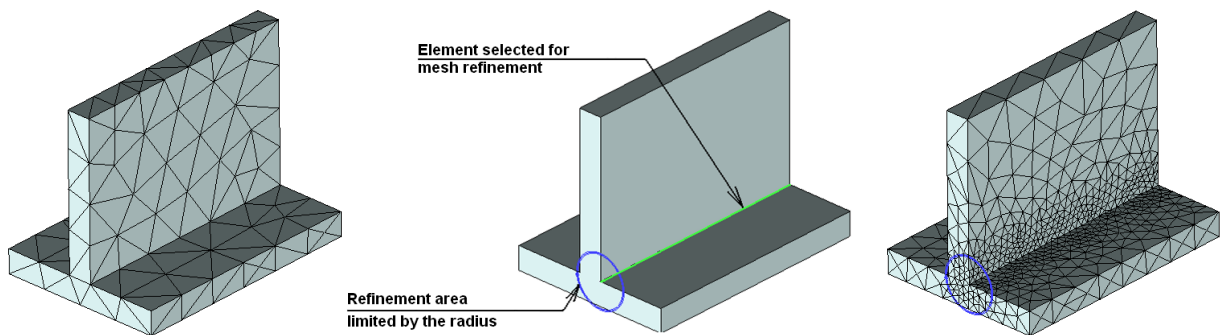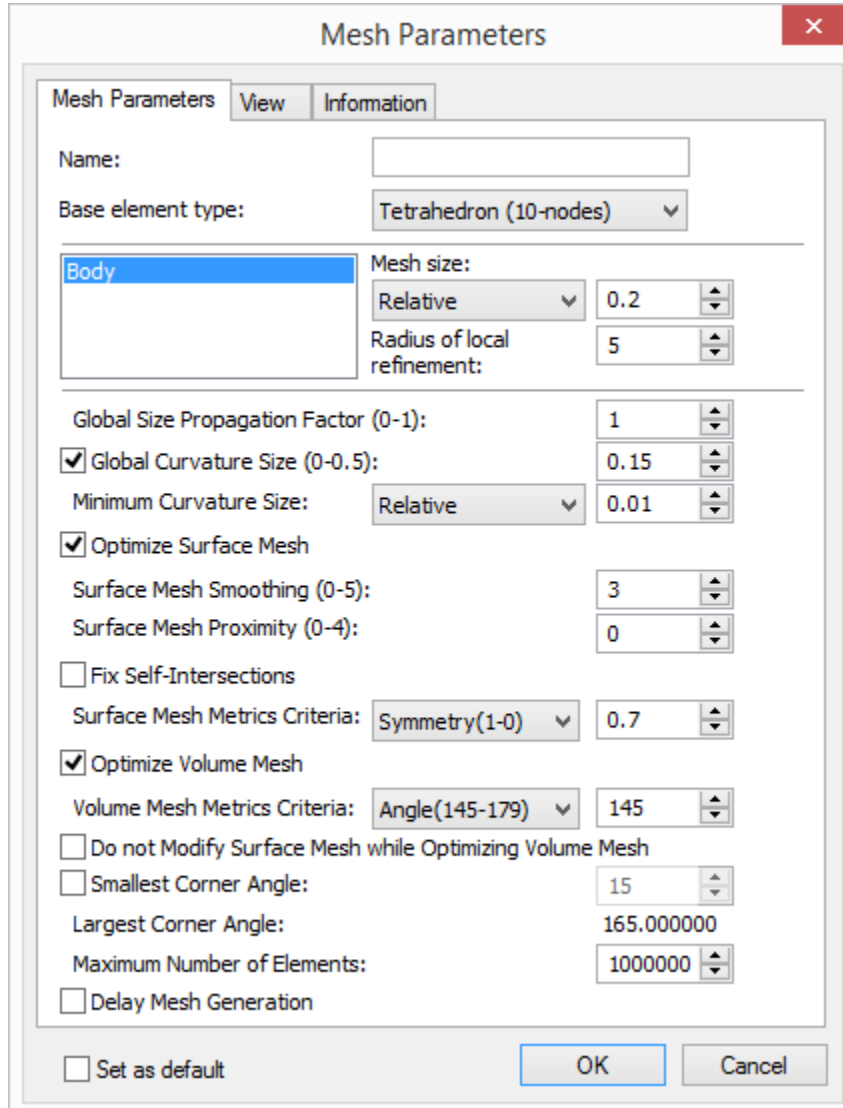
At the mesh calculation time, the system displays a tool window that tracks the progress of the generation process. The window has a **[Cancel]** button that allows terminating the mesh calculation process.
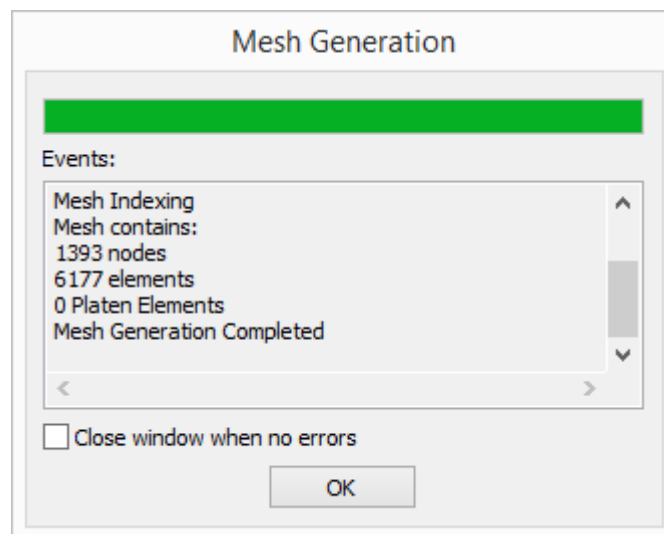
As the parametric model changes, the mesh may require an update. The system can automatically update the mesh, if the respective setting is made in the mesh parameters. Start the mesh update command manually from the context menu by right-clicking 🖱 the mesh in the studies window.

Mesh Parameters

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

There are two versions of finite elements used in the T-FLEX Analysis – straight-edged and curvilinear. The straight-edged finite element has nodes only at vertices, while the curvilinear element has intermediate nodes at the middle points of the edges (see the picture). Thus, tetrahedrons contain 4 or 10 nodes, and triangles – 3 or 6 nodes.
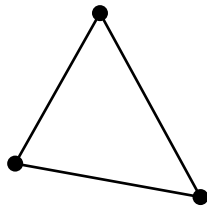
3-node straight-edged triangular finite element
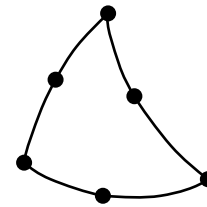
6-node curvilinear triangular finite element
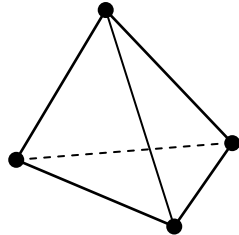
4-node straight-edged tetrahedral finite element
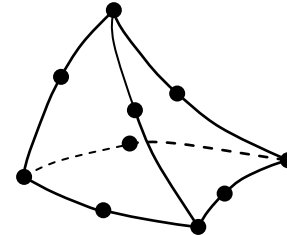
10-node curvilinear tetrahedral finite element

The use of curvilinear finite elements allows the user to more accurately approximate a complex geometry and obtain higher accuracy of the solution with the smaller number of elements.

Thus, for more accurate description of the complex geometry of the boundaries, it is necessary to use either a large number of the elements with the straight sides (edges), i.e., straight-edged finite elements, at the boundary, or use curvilinear finite 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 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.
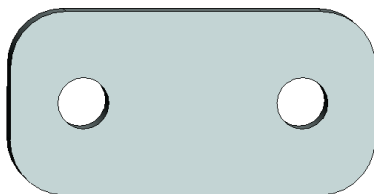
Original model
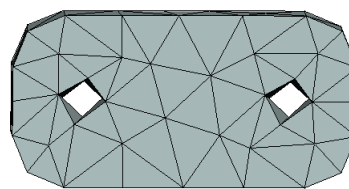
Mesh of straight-edged elements
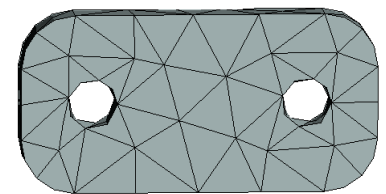
Mesh of curvilinear elements

Setting up mesh updating parameters is done by selecting a choice in the drop-down 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 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.
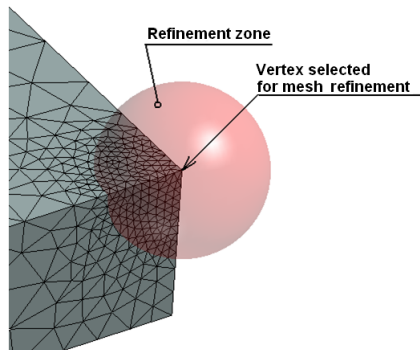
| Propagation factor = 1 | Propagation factor = 0.5 | Propagation factor = 0 |
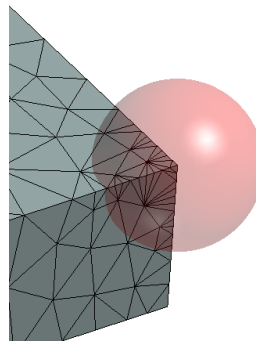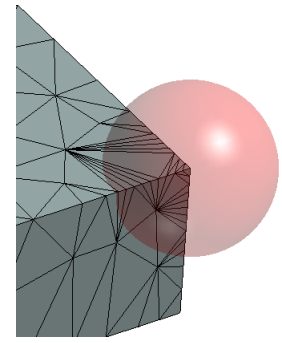
Refinement radius. This parameter can be set only for model elements selected for local mesh refinement. Refinement radius determines the size of the zone around the element, within which the mesh is constructed with enhanced individually specified properties – usually, finer meshing. The distance is 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 $0 \div 0.5$.
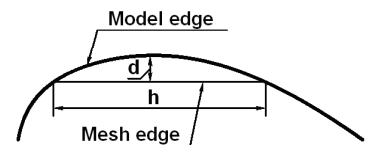
Minimum curvature. Works together with «Curvature ratio». This sets the ultimate minimum size of a curve segment, to which it can be divided. This parameter is introduced for limiting the number of mesh elements – this is because automatic subdivision of surfaces could go on indefinitely on some models,

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.
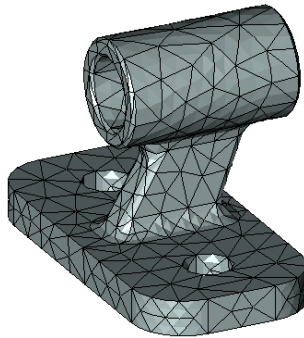
| Curve processing disabled | Curve processing enabled |
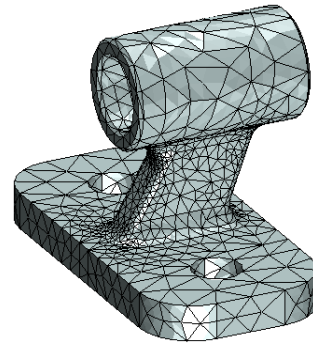
Optimization options provide control over the process of generating an improved-quality mesh. When the generator creates the mesh, it first calculates a preliminary mesh. After that, the program can apply certain manipulations to the obtained preliminary mesh in order to improve its quality. Those manipulations 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 ÷ 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 ÷ 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 |
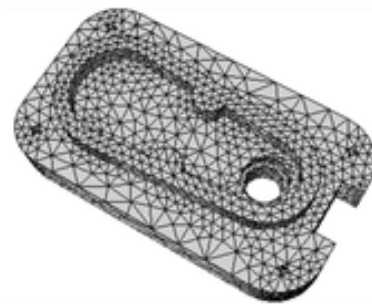
Fix Self-Intersections. Enables checking of the mesh for self-intersections after generation. If they are found the system tries to fix them automatically. Mesh self-intersections appear when coarse sizes are set for the model with highly curved geometry.

Surface/Volume mesh metrics criteria. 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 $145° \leq \theta < 180°$. The value by default is 145°.

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 properties 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.
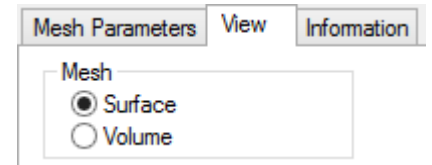
The volume mesh rendering shows the entire mesh, including its portions within the interior of the model's volume. In this mode, the system may experience a slowdown when rotating the 3D scene. In such a case, it would be best to use the wireframe mode for the 3D scene.

On the **«Information»** tab you can get information about certain properties 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.

Mesh Parameters		
Mesh Parameters	View	Information
Number of Vertices:		391

Volumetric Elements

Number of Tetrahedrons:	1583
Maximum Edge Length Relations:	8.5669
Minimum Angle between Edges:	12.5713
Maximum Angle between Edges:	156.4177
Minimum Radius Relations:	0.2252
Minimum Tetrahedron Volume:	77.4973

Shell Elements

Number of Shell Elements:	

☐ Set as default   OK   Cancel

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.

Mechanical Initial Conditions

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

Initial velocity

Use the following command to specify Initial Velocity boundary condition:

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

You need to select model elements to apply loads after the command call. Use automenu options for this purpose:

	<B>	Select Body
	<M>	Select All Solids

You can select several bodies or all bodies to specify their initial velocity.
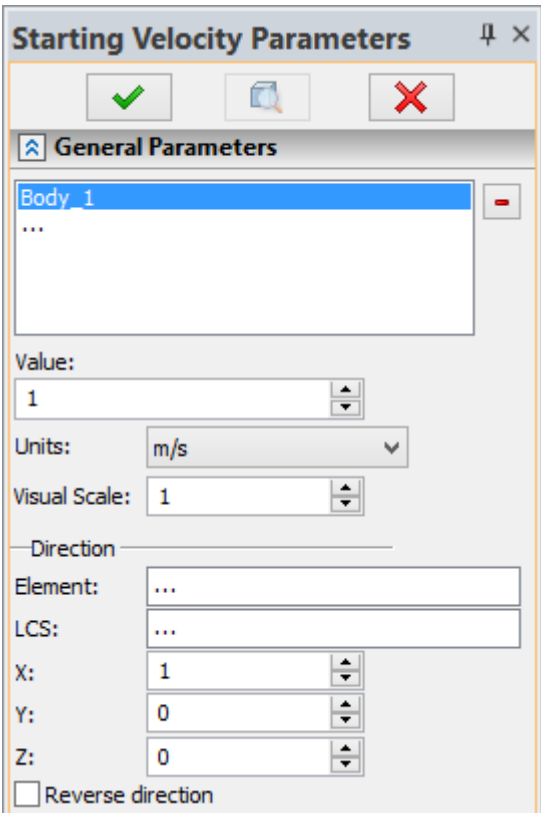
Value. Here you can insert the value of velocity.

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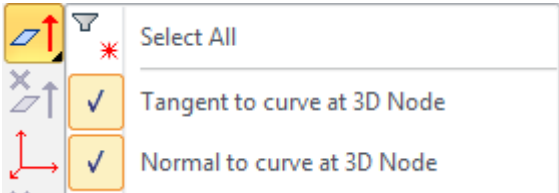
To set the direction of initial velocity using LCS use option:

	\<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. Use automenu options for this purpose:

	\<B\>	Select Body
	\<M\>	Select All Solids

You can select several bodies or all bodies to specify their initial velocity.
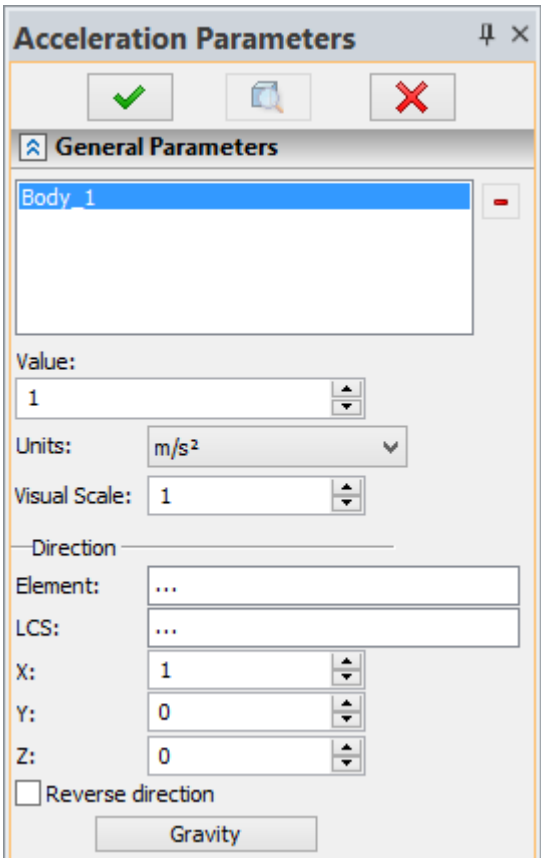
Value. Here you can insert the value of velocity.

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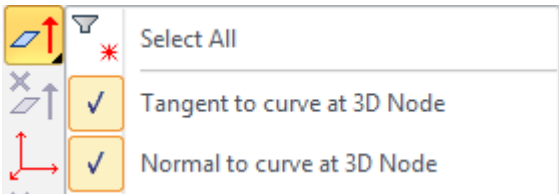
To set the direction of initial acceleration using LCS use option:

⌐	<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.

Direction

Element:	...
LCS:	LCS_1
X:	1
Y:	0
Z:	0
☐ Reverse direction	

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

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

The initial temperature condition is used for defining of the temperature in the nonstationary thermal study. The thermal load defines the temperature of the selected model elements at time zero.

The "default" value will be applied to the nodes of finite elements mesh. The nodes do not belong to the selected model elements. The value is specified in the **Study** dialog window on the **Parameters** tab. Study dialog window can be called from the context menu of the thermal study.
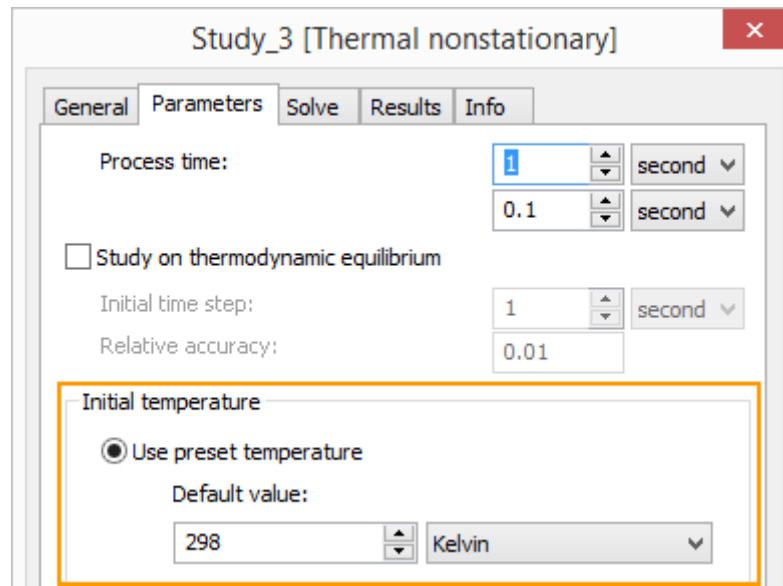
To specify the initial condition use command:

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

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

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

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

The selected elements are added to the list.

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

- Value,
- Units C, K, F.
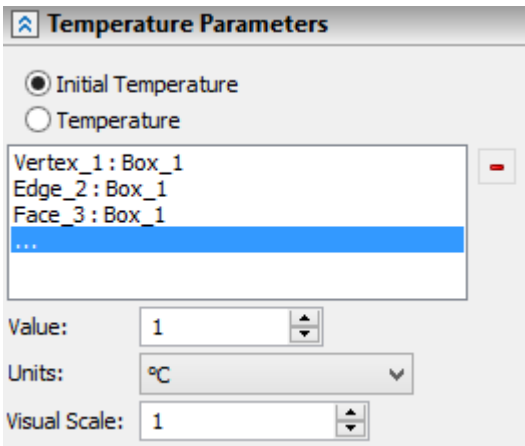
The initial temperature is shown in the 3D scene in the same way as the common *temperature* load.

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 three types of restraints: **full restraint**, **partial restraint** and **contact**. 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.

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 full restraint, use the command:

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

To specify a restraint, you need to select a model element. Faces, edges and vertices are available for selection. Upon selecting an element, the symbolic notation of the full restraint appears in the 3D window.

Symbolic notation of restraint

Face selected for restraint
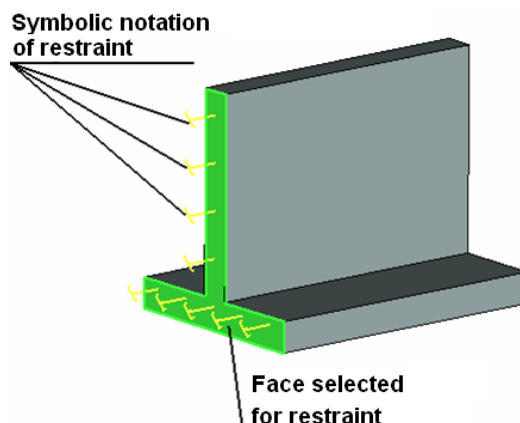
Partial Restraint

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 a partial restraint, use the command:

Пиктограмма	Лента
	Analysis → Conditions → Full Restraint → Partial Restraint
Клавиатура	Текстовое меню
<3ML>	Analysis > Restraint > Partial Restraint

To specify locations of a partial restraint, select an edge, face or vertex. 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.

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.
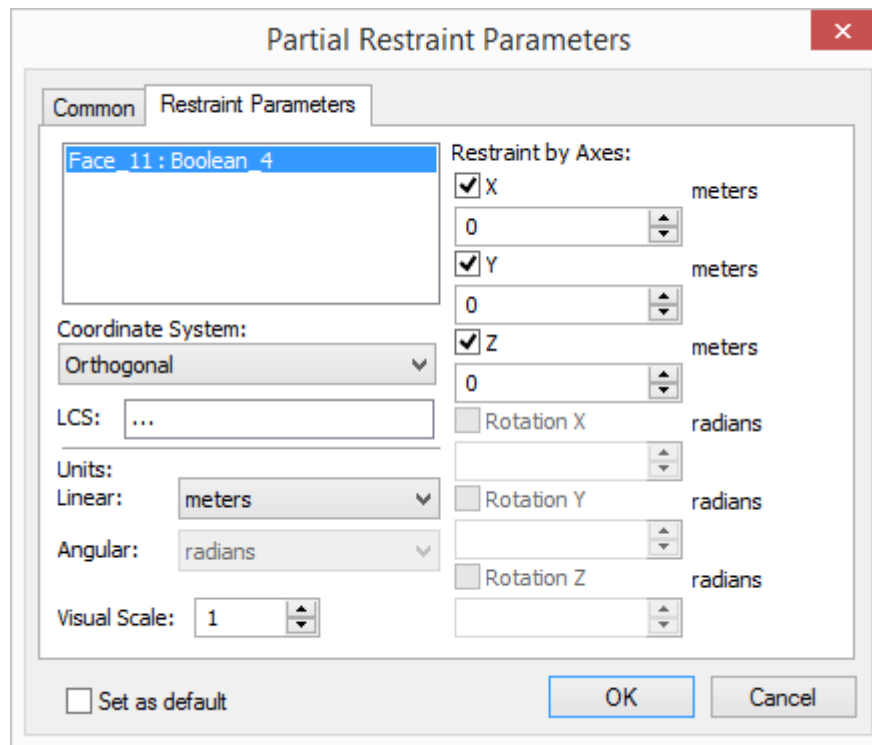
Parameters «*Rotation about X*», «*Rotation about Y*», «*Rotation about Z*» are required to specify rotations with respect to the axes of the coordinate system (local or global) to solve studies of plate (or shell) deformation. Given that, triangular elements must be used for discretization of the computational 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
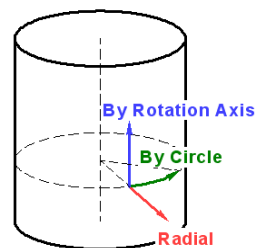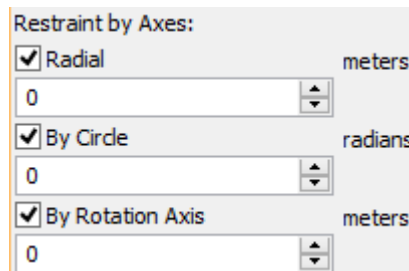
A spherical coordinate system allows constraining dimensions in:

- Radial
- Longitude
- Latitude
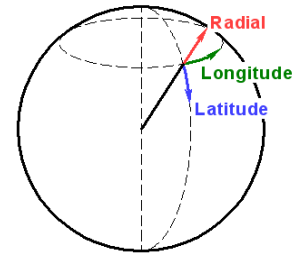
Shown on the picture is an example of a partial restraint on a surface in a cylindrical coordinate system. In this case, partial restraints are defined in the "circumferential" direction, whereas there are no restraints in the radial direction and along the rotation axis, meaning that the revolution 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 properties window. Static analysis will be performed with this condition.

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.

A typical order of steps for defining partial restraints is as follows:

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.

Contact

Contact restraints are needed in studies of contacting bodies. To define a contact, use the command:

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

To define a contact, you need to select the contacting faces of two bodies.

Next, select one of four contact types:

- Full Contact;
- No Contact;
- Touch;
- Hard Wall.

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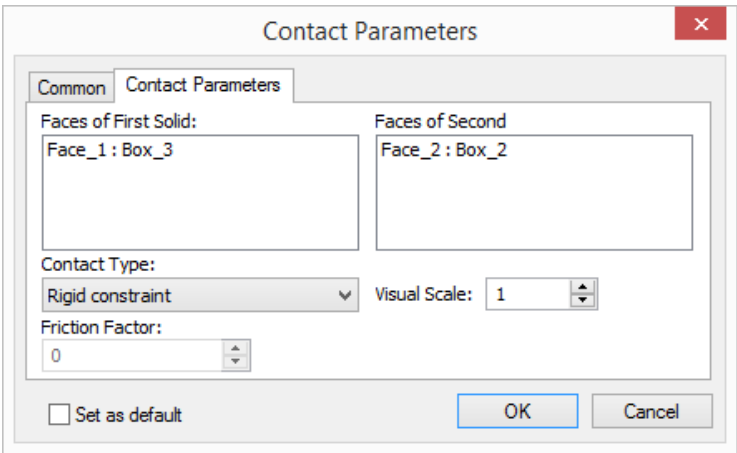
The dependency of a model behavior on various contact types can be illustrated with the following example. Two beams are kept together using a contact restraint, with one end fixed, and a distributed force acting normal on the top and side surfaces of the first beam.

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

Initial Model

Contact: rigid contact

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

Contact: touch

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: 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: rigid wall

It is possible to set default contacts.

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

A typical order of steps for defining the contact restraints is as follows:

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.

Elastic foundation

This type of restraints allows us to define elastic interaction on the boundary of a body. Elastic foundation 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 elastic foundation can be envisioned as a set of weightless springs of identical stiffness attached to the boundary of the body.

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

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
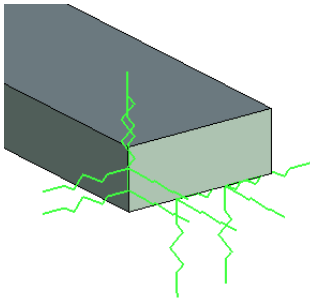
| | <K> | Cancel LCS selection |

If LCS is not set, the global coordinate system is used by default.

You can select the following parameters in the properties 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 «resultant» 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).
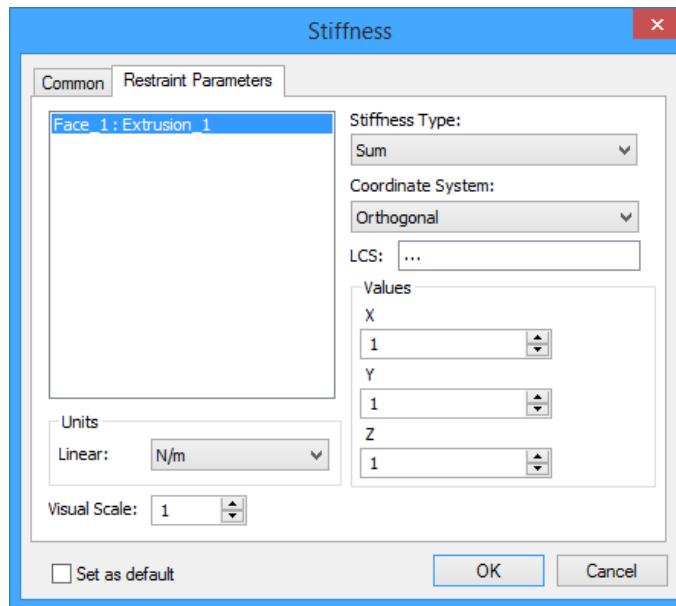
There is the following actions sequence when you set restraint of the additional stiffness type:

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.

DEFINING LOADS

Mechanical Loads

This type of loads is used when modeling the studies 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 studies of cyclic loading which take into account fatigue phenomena.

Force

Force is a type of loading used to specify *a concentrated load*, and also for specifying a *total magnitude* of distributed load.

To specify the load *Force*, use the command:

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

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 window properties, it is required to specify the load type:

- uniform;
- non-uniform.

Force as a Total Magnitude of 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.
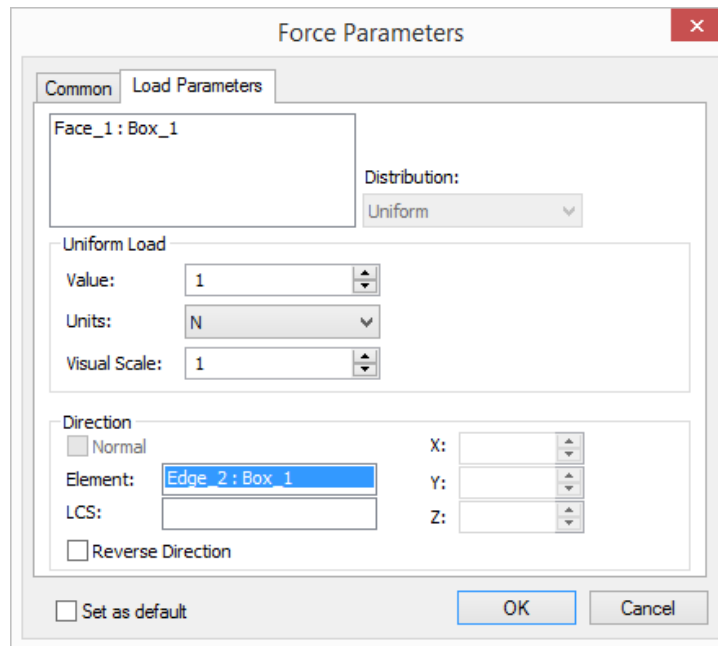
Numeric value. The numeric value is defined as a total equivalent magnitude of uniformly distributed along the edge length (over the face area) load.

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 define properties that are changing according to the temperature and time".

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

Load 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

![icon]	<K>	Unselect LCS

For specifying the load direction with the help of a 3D model, use the following automenu option:
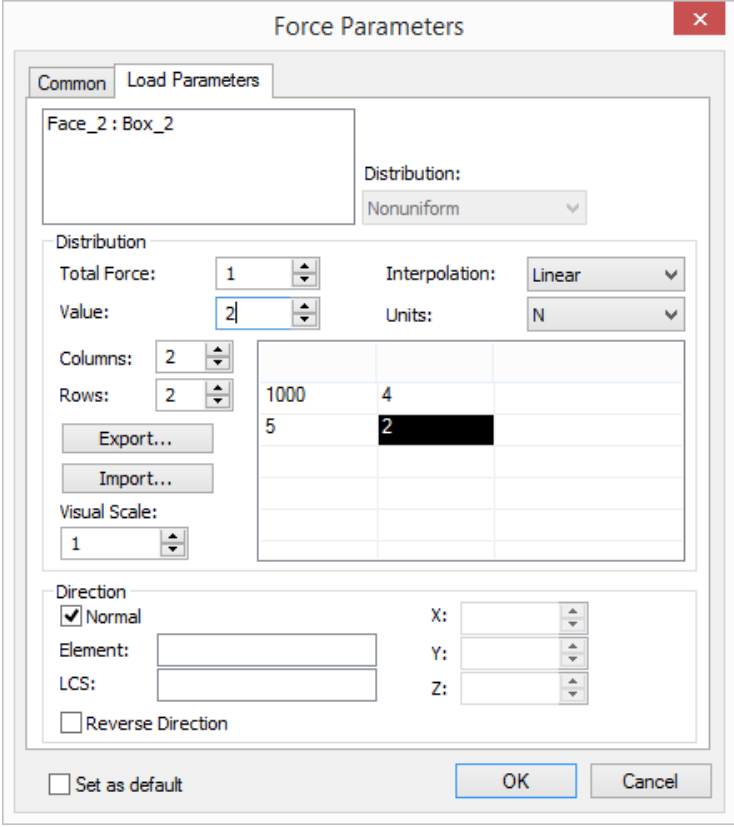
![icon]	<D>	Select direction

To cancel selection of direction, use the option:

![icon]	<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 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.
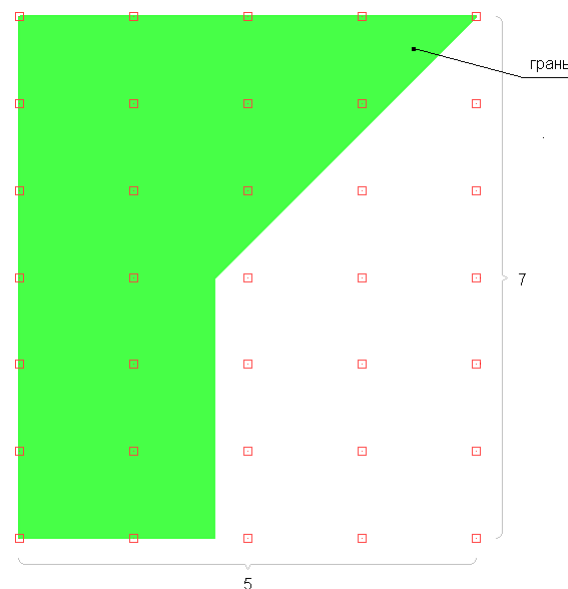
Example of specifying distribution law for non-uniform load

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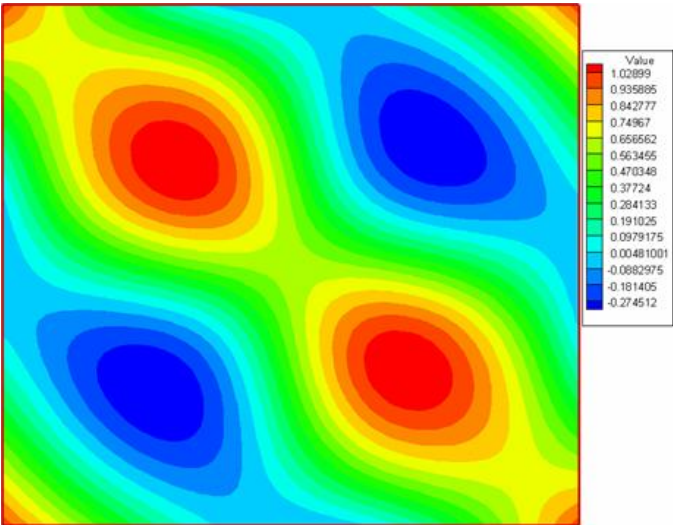
Example of non-uniform load distribution function obtained by spline construction
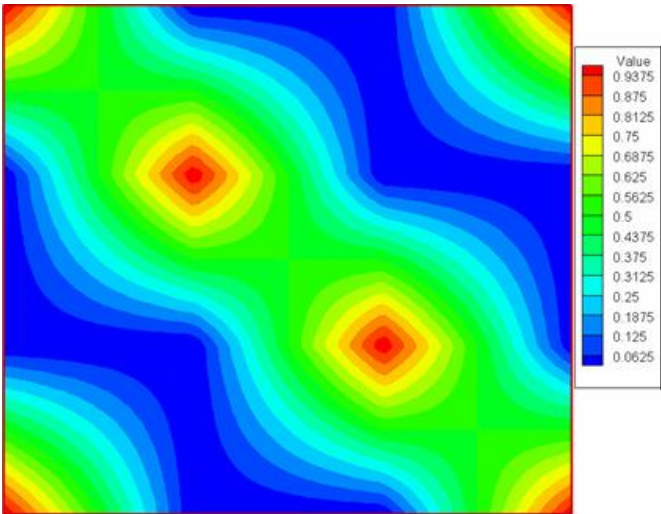
Example of non-uniform load distribution function obtained by using linear interpolation

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:
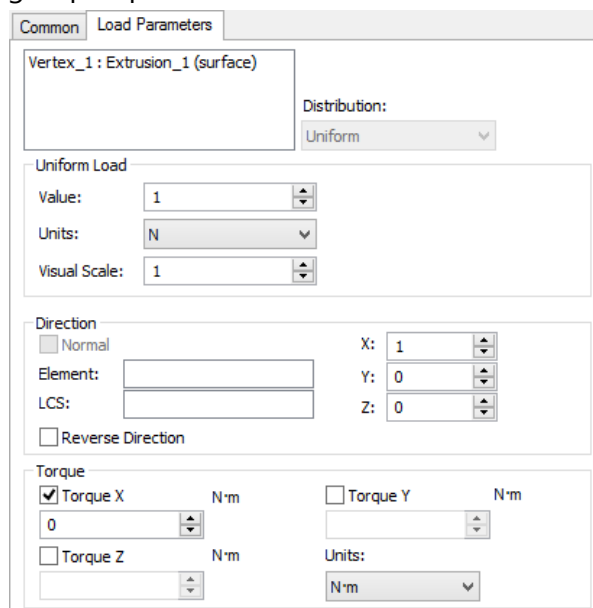
�◿↑	<D>	Select direction

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».

Bending Moments

In the dialog «Force parameters», in addition to parameters defining the load type, numeric value, units, load direction, etc., there is a group of parameters called «Moment».

With the help of parameters of this group, the user can specify bending moments with respect to the axes of the coordinate system (local or global) for solving studies of plate (or shell) deformation. It is important to note that computational domain was discretized with the triangular finite elements.

For the *Moment* the user can utilize the following units: N-m, lbf-in, kgf-cm.

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 the command «**Operation > Faces > Imprint Elements**» should be used.
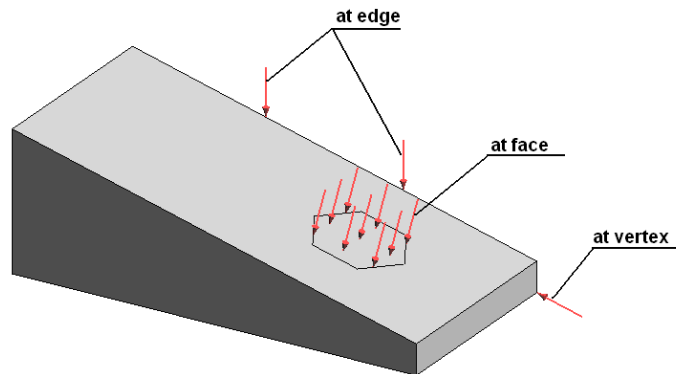
Specifying load «Force»

Typical sequence of steps for specifying the load «Force»:

1. Initiate 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

Pressure represents a loading type used for specifying a distributed load.

For specifying the load *Pressure,* use the command:

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

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 properties 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 properties, varying depending on time or temperature "

Units. For a load *Pressure*, applied to a face, the following units can be used: N/m^2, lbf/in^2, kgf/cm^2.

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/m^2, lbf/in^2, kgf/in^2.

Load 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 options:

	<C>	Select LCS

![icon]	<K>	Unselect LCS

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

![icon]	<D>	Select direction

To cancel selection of direction, use the option:

![icon]	<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 $p = \rho \cdot h$, where *h* – is the height of a liquid column of density *ρ*.
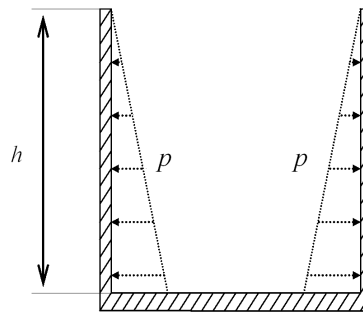
When defining *Pressure* as a hydrostatic load, it is necessary to specify the liquid density, units, and define the direction of load.
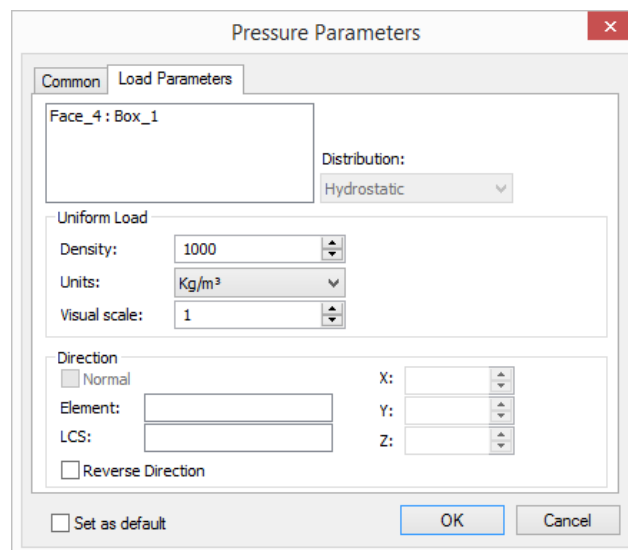
Direction of Load Change. The load *Hydrostatic Pressure* acts along the normal to the loaded face. It is required to specify direction of load change, that is, the direction along which the load is increased.

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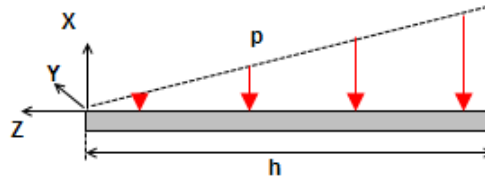
The zero value of the load corresponding to an upper level of liquid coincides in space with the coordinate *Z*=0 of local coordinate system determining direction of the load change.

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:
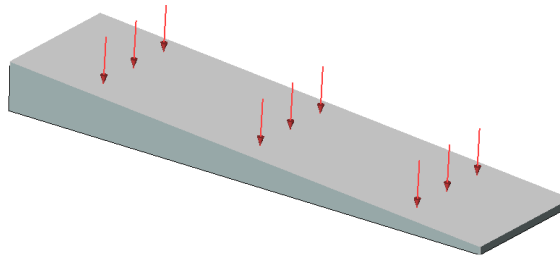
Specifying load «Pressure»

Typical sequence of steps for defining the load *Pressure*:

1. Initiate 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 represents a loading type used for simulating a centrifugal force, which arises upon uniform, or uniformly accelerated, rotation of an object.

For specifying the load *Rotation,* use the command:

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

After invoking this command, select one or several solid bodies for applying the load. With the help of automenu option:

	<B>	Select Body
	<M>	Select All Solids

select bodies of an analyzed model. They will be added to the list.

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

To cancel selection of axis of rotation, use the option:

	<C>	Cancel axis selection

In the load's properties 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^2 [rad/sec^2], degrees per second^2 [deg/sec^2], the number of revolutions per second^2 [Hz/sec], the number of revolutions per minute^2 [rpm^2].
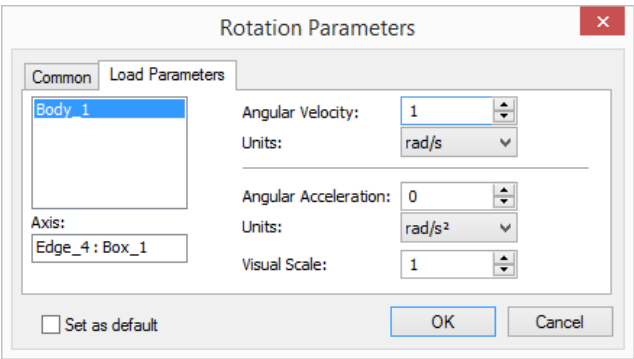
In the 3D scene the load *Rotation* is shown in the following way:
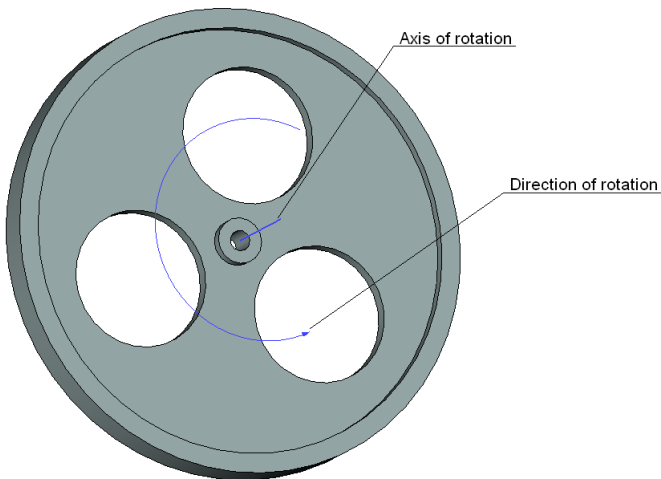
Specifying rotation

A typical sequence of steps for specifying the load «Rotation»:

1. Initiate command «Rotation» .

2. Specify axis of rotation.

3. Specify the magnitude and units for angular velocity and angular acceleration in the command's properties window.

4. Complete the command.

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.

To specify the load *Acceleration*, use the command:

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

After invoking this command, it is necessary to select the body (or several bodies) for applying the load. With the help of automenu option:

	<B>	Select Body
	<M>	Select All Solids

select the bodies of an analyzed model. They will be added to the list.

In the properties window specify:

- Value of load (you can specify dependence of acceleration from time for nonstationary studies. More information in "Using graphs to specify properties, varying depending on time or temperature" section);
  - Units: m/sec^2, cm/sec^2, in/sec^2;
  - 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».

For a quick specification of the gravity force, in the properties window there is a button **[Gravity force]**, which sets the value of acceleration equal to ~9.81 м/с^2 and sets the direction of load along the Z-axis of the global coordinate system equal to -1.

In the 3D scene, the load «Acceleration» is shown in the following way:
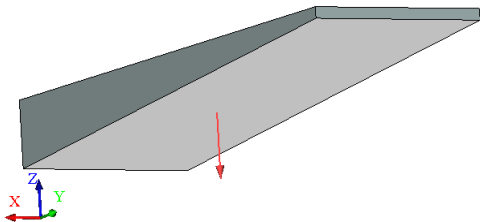
Specifying load «Acceleration»
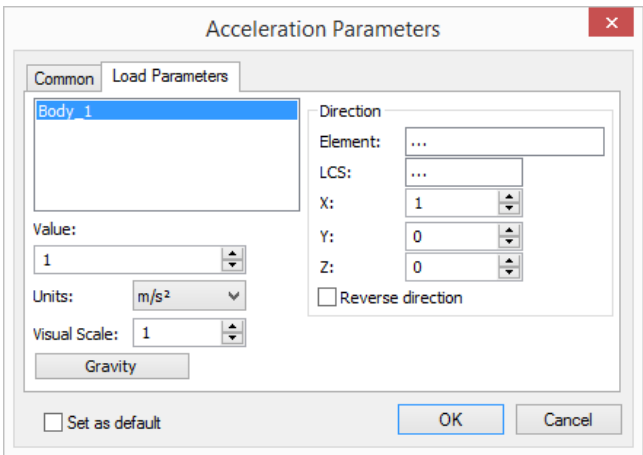
A typical sequence of steps to specify the load *Acceleration*:

1. Initiate command «Acceleration» .
2. Specify the load magnitude.
3. Specify the direction of load
4. Complete the command.

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).
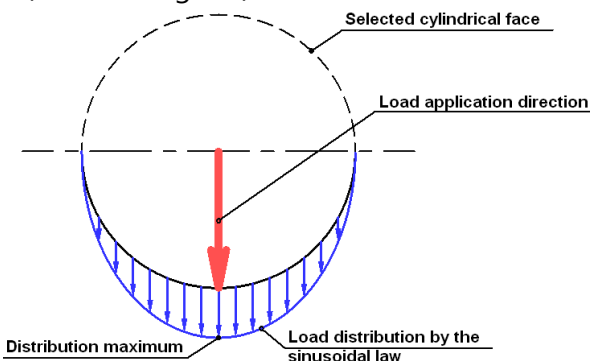
To specify *Bearing load*, use the command:

Icon	Ribbon
![icon]	Analysis → Conditions → Force → Bearing Load
Keyboard	Textual Menu
<3MB>	Analysis > Load > Bearing Force

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:

![icon]	<F>	Select cylindrical face

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

In the properties 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 properties, varying depending on time or temperature" section.

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

![icon]	<D>	Select direction

To cancel selection of direction, use the option:

![icon]	<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 load» is shown in the following way:
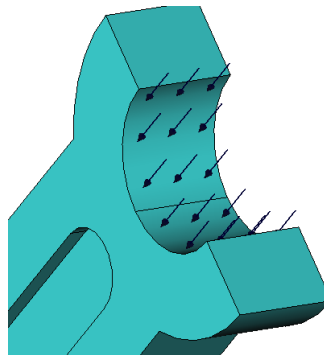
Specifying «Bearing force»

A typical sequence of step for specifying *Bearing load*:

1. Initialize the command «Bearing force» ![icon].
2. Select a cylindrical face or a set of faces.
3. Specify the magnitude of load.
4. Specify the load direction
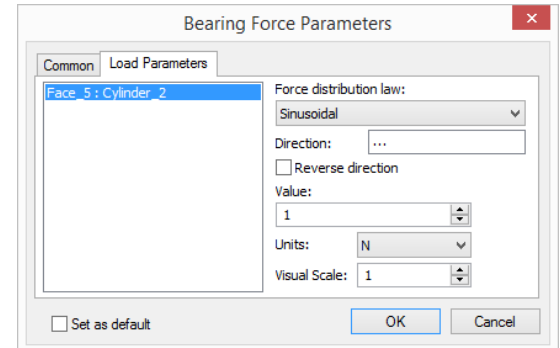
5. Complete the command.

Torque

Torque is the impact of a force moment of the specified magnitude distributed over the selected face.
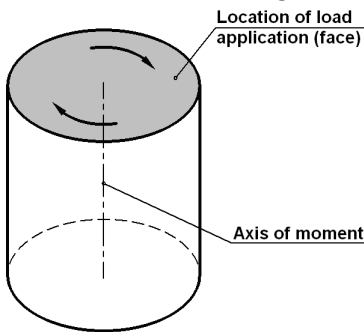
To specify *Torque*, use the command:

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

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 properties 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 properties, varying depending on time or temperature" section.

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

	<A>	Select axis of rotation
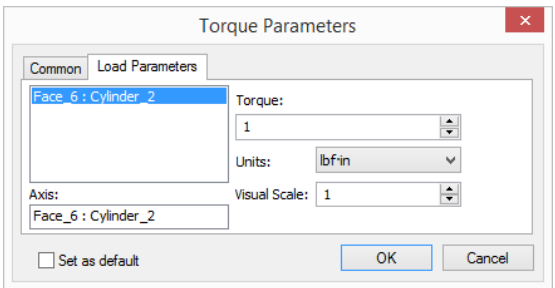
To cancel selection of axis rotation, use the option:

⬆	<C>	Cancel axis selection

In the 3D scene the load «Torque» is shown in the following way:
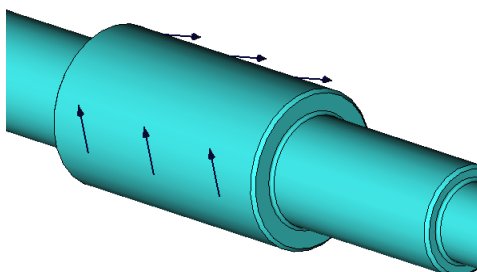
Specifying load «Torque»

A typical sequence of steps for specifying the load «Torque»:

1. Initialize 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

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 properties, varying depending on time or temperature" section.

To specify *the loading "oscillator"* use the command:

Icon	Ribbon
((■))	Analysis → Conditions → Force → Oscillator
Keyboard	**Textual Menu**
<3M3O>	Analysis > Load > Oscillator

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.

With the help of the option of the automenu:

	<B>	Select body
	<M>	Select all bodies

select the bodies of the analyzed model. They will be added to the list.

In the properties 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 $\dot{U}_m = U_m \cdot \omega_f$, mm/s, cm/s, m/s, in/s, ft/s;

acceleration $\ddot{U}_m = U_m \cdot \omega_f^2$, mm/s^2, cm/s^2, m/s^2, in/s^2, ft/s^2;

overloading $\ddot{U}_m / g = U_m \cdot \omega_f^2 / g$, times;

where ω_f – frequency of forced vibrations, rad/s; g – gravitational acceleration, m/s^2.

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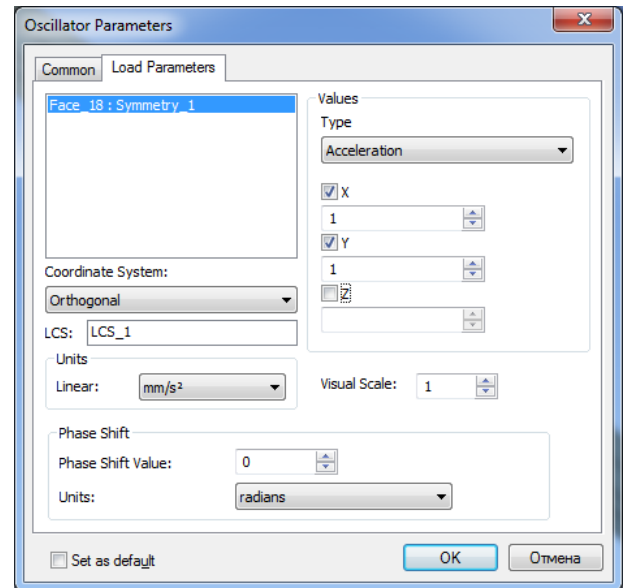
To work with the local coordinate system, use the following options:

![icon]	<C>	Select coordinate system
![icon]	<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:
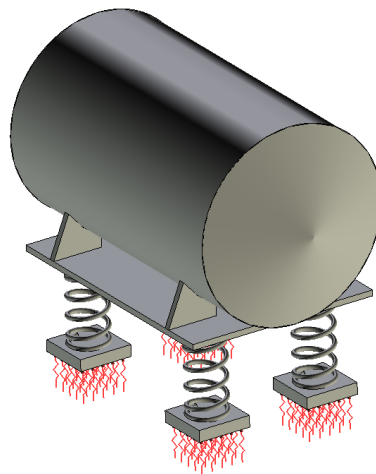
Displaying «Oscillator» loading

Typical order of actions when specifying the oscillator loading:

1. Initialize the «Oscillator» command ![icon].
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 properties window.
4. Specify phase shift and its units.
5. Complete the command.

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. In studies of static analysis and stability analysis this type of loading is used only in combination with the «Acceleration» load. This type of loading is used in studies of static analysis, stability analysis, natural frequency analysis and analysis of forced vibrations.
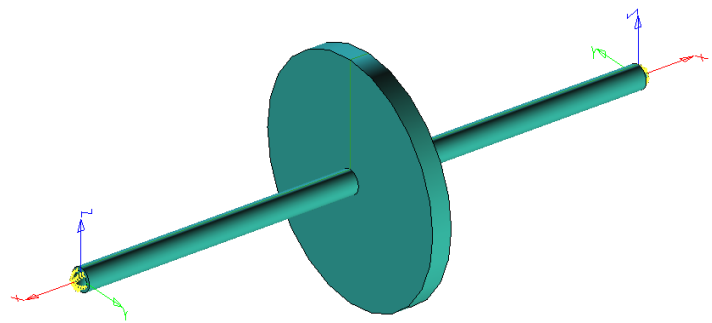
Initial full-size model
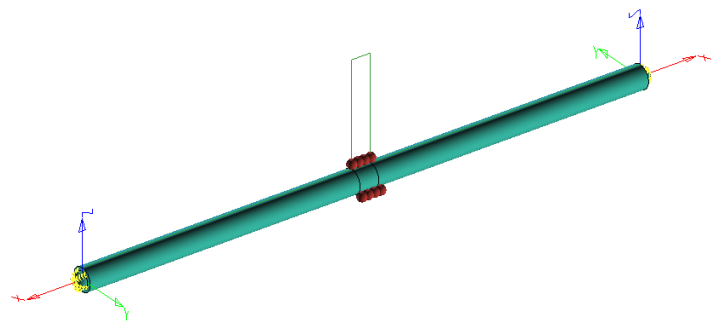
Replacement of the disk with the "Additional mass"

Icon	Ribbon
	Analysis → Conditions → Force → Additional mass
Keyboard	**Textual Menu**
<3M3M>	Analysis > Loading > 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.
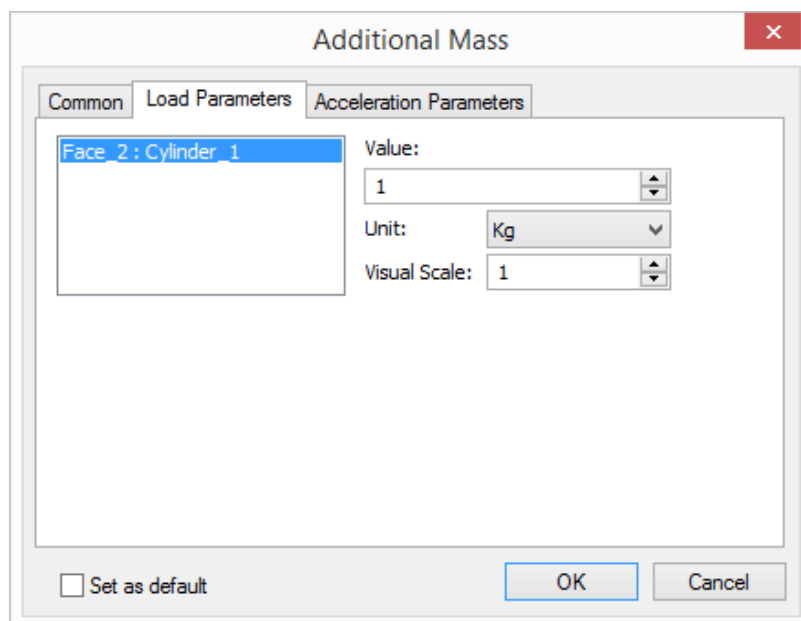
On the **Load Parameters** tab specify:

- 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 properties, varying depending on time or temperature" section.

- Units of measurement: kg, pounds;
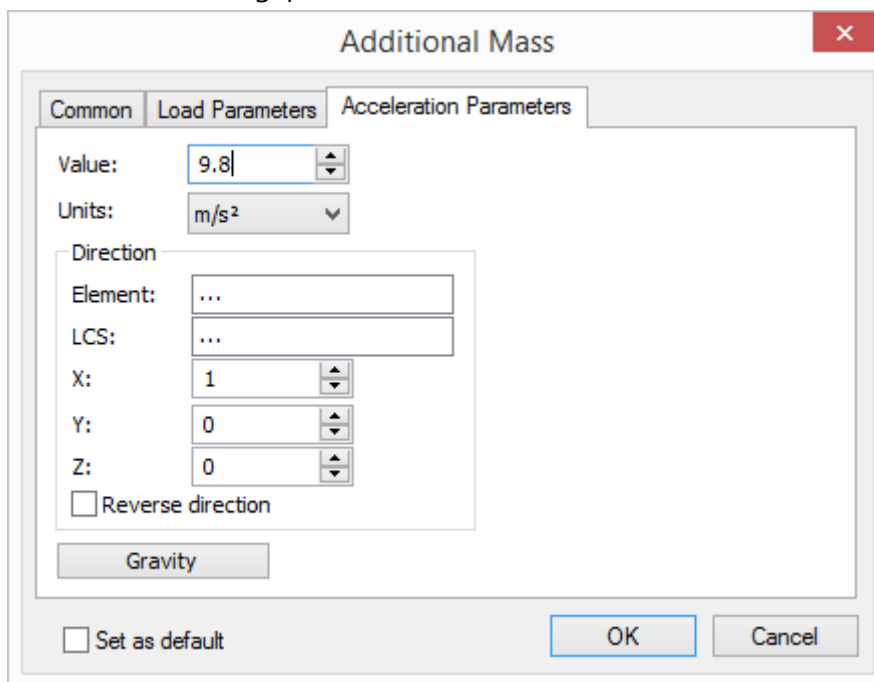
On the **Acceleration Parameters** tab specify:

- 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 studies, 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 order of actions upon specification of the «Additional mass» load:

1. Initialize 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.

THERMAL LOADS

This type of loading is used in the heat transfer studies.

Heat transfer is the process of transferring the heat from one region with higher temperature to the region with lower temperature.

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 studies.

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».

For specifying the load, use the command:

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

Temperature can be applied to a model body, a face, edge or vertex. For selecting elements of the model, use the automenu option:

🔲	<E>	Select Element for loading
🔲	<B>	Select Body
🔲	<M>	Select All Solids

Selected elements are added to the list.

In the properties 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 properties, varying depending on time or temperature" section);

* Units: K, C, F.

In the 3D scene the *Temperature* is shown in the following way:
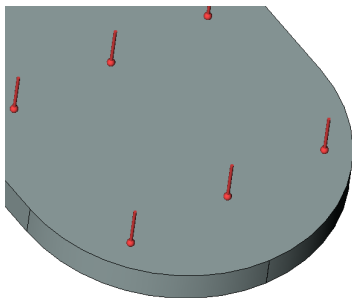
Specifying thermal load «Temperature»
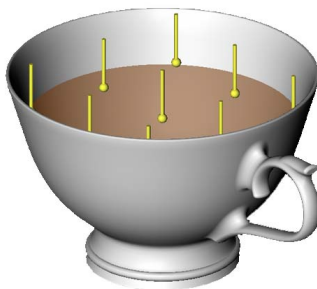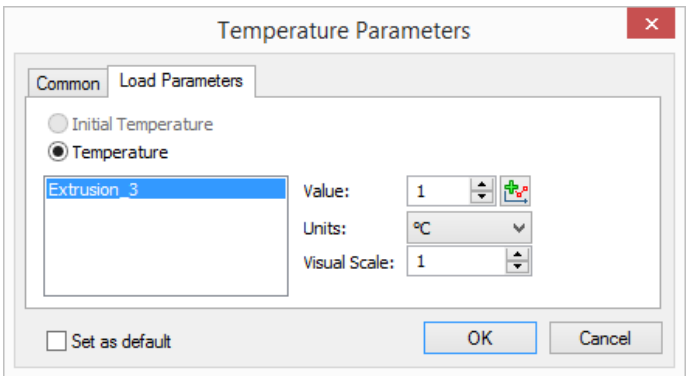
Specifying initial temperature in transient thermal analysis

A typical sequence of steps for specifying the thermal load is:

1. Initialize 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 Flux

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.

For specifying the load, use the command:

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

Heat flow can be applied to faces of the model. For selecting faces, use the automenu option:

🔲	<F>	Select Face

Selected elements are added to the list.

In the properties 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.
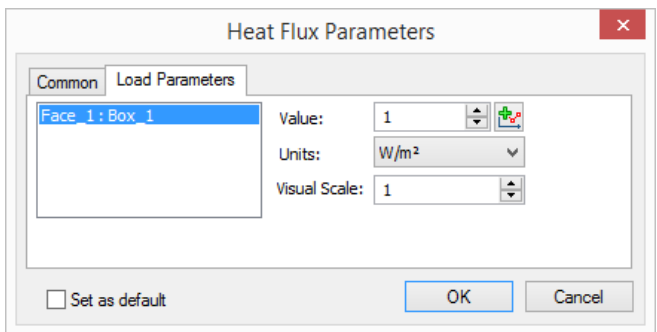
You can specify dependence of heat flow from time for **Nonstationary Thermal Process** study. More information in "Using graphs to specify properties, varying depending on time or temperature" section

In the 3D scene *Heat flow* is shown in the following way:
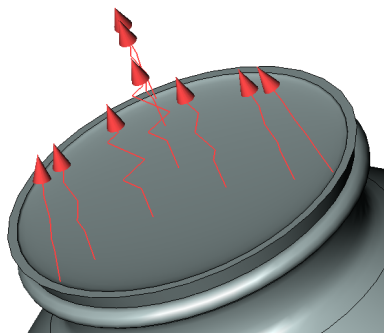
Specifying thermal load «Heat flow»

A typical sequence of steps for specifying «Heat flow»:

1. Initialize the command «Heat flow» ;
2. Select a face;
3. Specify the magnitude of load;
4. Specify units;
5. Complete the command.

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).

For specifying the load, use the command:

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

Heat power can be applied to a body, face, edge or a vertex of a model. For selecting elements of model use the automenu option:

	<E>	Select Element for loarding
	<B>	Select Body
	<M>	Select All Solids

Selected elements are added to the list.

In the properties window specify:

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

The negative value of this thermal load signifies that a body looses the energy.
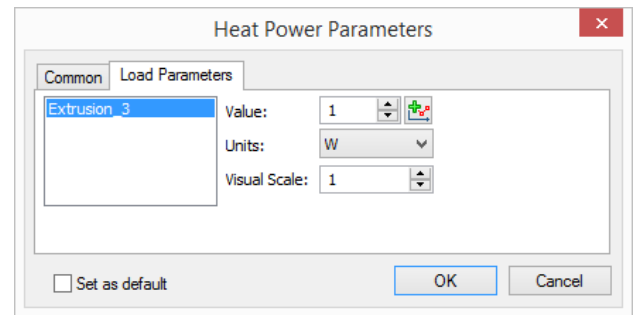
You can specify dependence of heat power from time for **Nonstationary Thermal Process** study. More information in "Using graphs to specify properties, varying depending on time or temperature" section

In the 3D scene *Heat power* is shown in the following way:
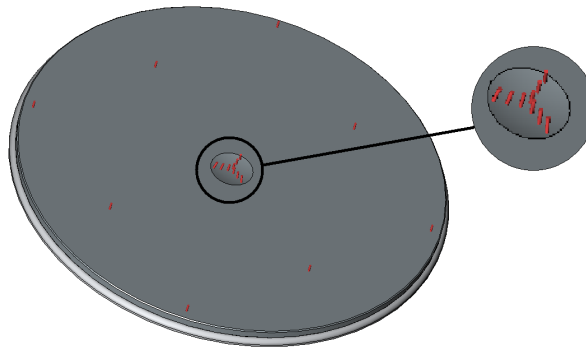
Specifying thermal load «Heat power»

A typical sequence of steps for specifying the load «Heat power»:

1. Initialize 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

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.

For specifying load *Convection* use the command:

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

Load *Convection* is defined for faces of the model. For selecting faces use the automenu option:

🔲	<F>	Select Face

Selected elements are added to the list.

In the properties 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].
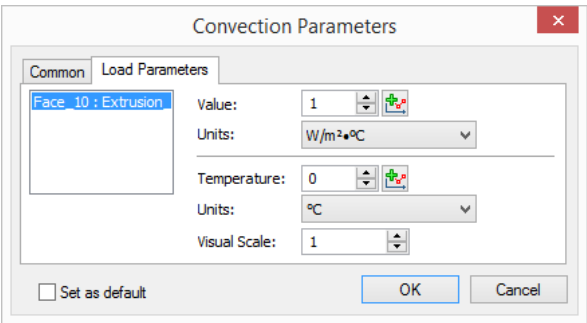
You can specify dependence of convection from time for **Nonstationary Thermal Process** study. More information in "Using graphs to specify properties, varying depending on time or temperature" section.

In the 3D scene the load «Convection» is shown in the following way:
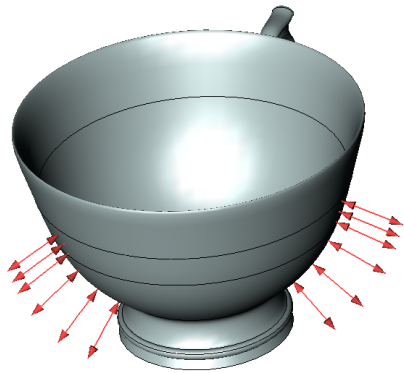
Specifying thermal load «Convection»

A typical sequence for specifying the load Convective heat transfer:

1. Initialize the command «Convection» 🔺.
2. Select a face.
3. Specify the heat transfer coefficient and temperature of the environment.
4. Complete the command.

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 study on heat transfer by radiation. They are applied to the bodies participating in the task.

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

For specifying *Radiation* use command:

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

Load *Radiation* is defined for faces of the model. For selecting faces use the automenu option:

◎	<E>	Select Element for loading

Selected elements are added to the list.

In the properties 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].
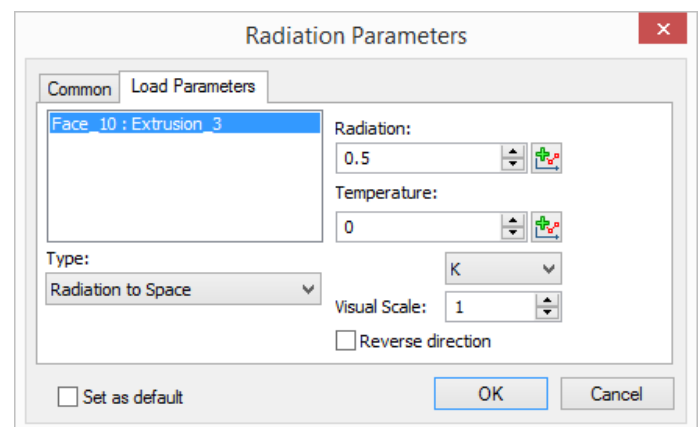
You can specify dependence of radiation factor from time for **Nonstationary Thermal Process** study. More information in "Using graphs to specify properties, varying depending on time or temperature" section

The load «Radiation» is shown in the 3D scene in the following way:
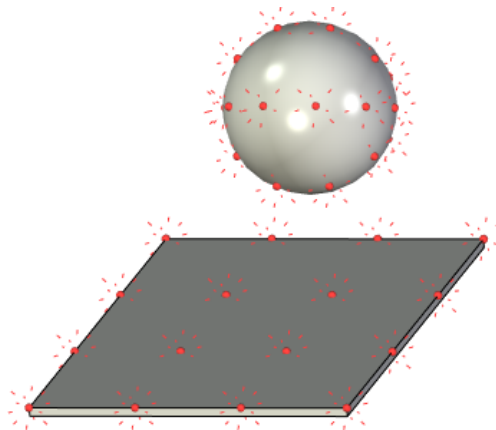
Specifying thermal load «Radiation»

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 properties.
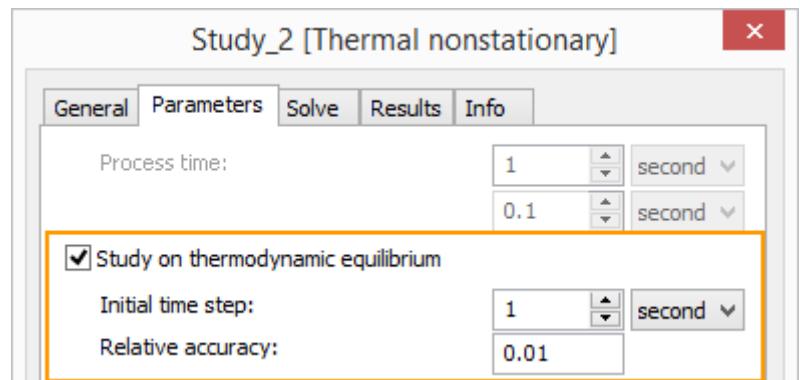
A typical sequence for specifying the load *Radiation*:

1. Initialize 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 (Thermal resistance)

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.
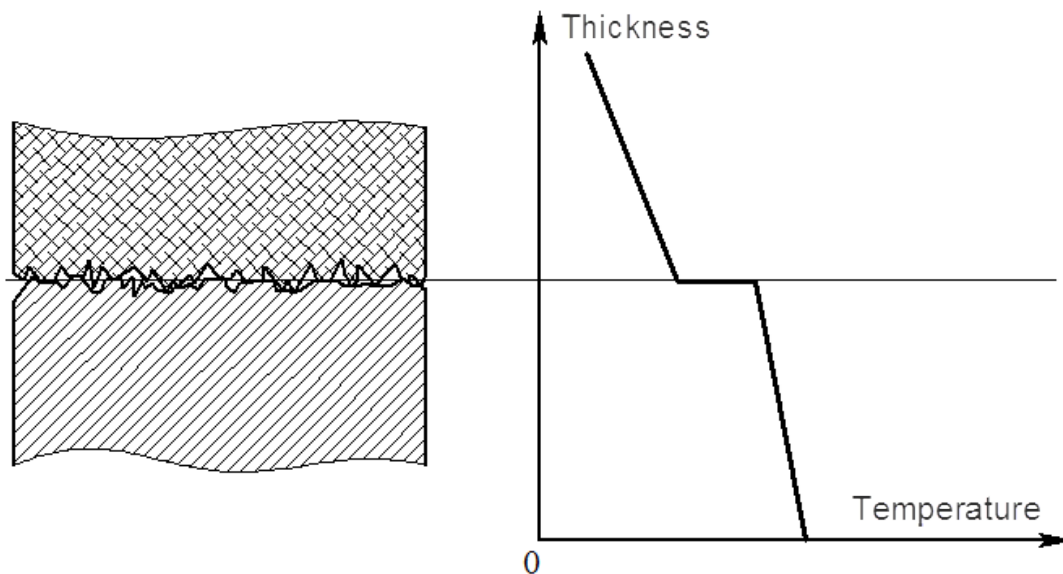
This physical phenomenon is called thermal resistance.

To specify the thermal resistance, use the command:

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

After the command is invoked, 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 (m2 ·°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 order of actions when specifying contact constraints is the following:

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 PROPERTIES, VARYING DEPENDING ON TIME OR TEMPERATURE

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 Load
- ✓ Torque
- ✓ Additional Mass
- ✓ Oscillator
- ✓ Temperature
- ✓ Convection
- ✓ Heat Flow
- ✓ Heat Power
- ✓ Radiation

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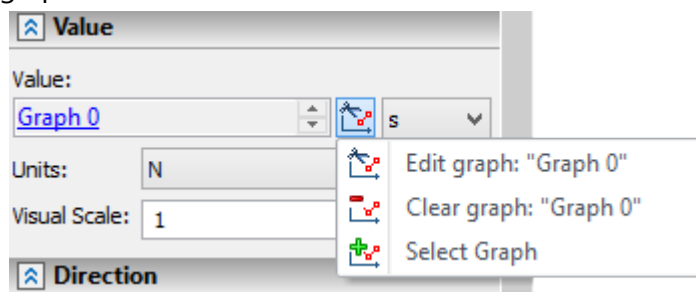
Pressing of button calls list that allows to edit, clear current graph or select a new graph. When the graph is deleted, it is unbound from the parameter but stored in the document. You need to use **PL: Edit Graph Functions** command to delete it from the document.

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,

✓ Convection,

✓ Heat Flow,

✓ Heat Power,

✓ 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 properties 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 **PL: 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 **PL: Edit Graph Functions** command to delete it from the document.

LOADS COMPENDIUM TABLE

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	-	Unit system, Heat Power
Convection	Face	-	Heat transfer coefficient, temperature of environment, units
Radiation	Face		Radiation type, emissivity, temperature of environment, units, radiation viewfactor of a face

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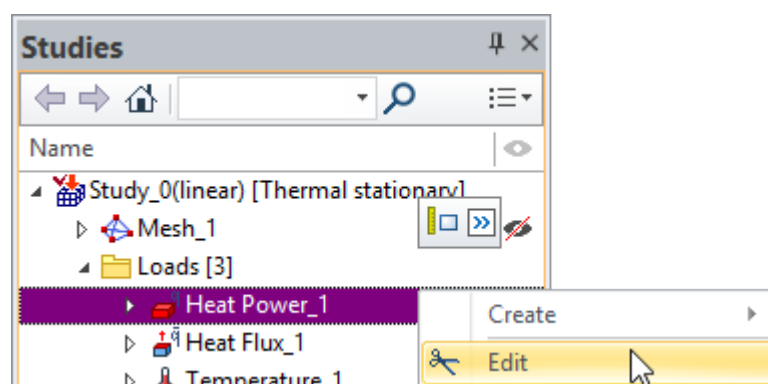
Further user actions of modifying loads or restraints are little different from the process of their creation. In the properties window, you can modify numerical parameters, while the appropriate automenu options (see above) let you cancel selection and then specify new defining model elements.

Upon finishing entering changes, confirm your actions by clicking ✅.

CUSTOMIZATION AND UTILITY COMMANDS

The user can define global settings of the T-FLEX Analysis system by the **Analysis > Settings** command.
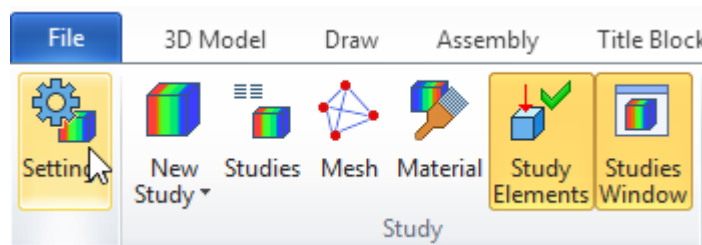
The following parameters are defined on the **[Processor]** tab:

Temporary directory – sets the path to the folder storing intermediate working data when solving systems of equations. By default, working files are stored in the folder defined by the Windows system variable "TEMP" (or "TMP") If necessary, the user can redefine this path.

System Resource group defines the following parameters:

Solver thread priority – allows the user to define the system priority of the modules responsible for solving systems of equations. The Windows operating system will distribute system resources in accordance to the specified priority, giving preference to a higher priority. For example, if planning on prolonged solving of a large study using hard drive memory, the user can specify in advance a priority below Normal, which would allow the user to simultaneously work in other Windows applications without much restriction.

Limit **memory usage**– allows the user to specify the size of RAM, upon exceeding which the system would start using hard drive memory for solving equations that usually takes considerably more time.

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

You can set the aspect ratio of finite elements (tetrahedrons and triangles) in which the system will consider them as "not optimal" or "incorrect" in the **Criteria of elements quality chec**k group. Incorrect elements are forced removed from the grid as they lead to singularities of equations system (inability to solve them). Usually, it is either almost flat or degenerate to the point tetrahedrons, or degenerate into a line triangular finite elements. The user can change these settings if necessary.

Display study properties dialog box before solving – when enabled, this control turns on the automatic launching of the study properties dialog box at initializing the **Analysis > Solve** command for all studies (the default setting).

Close solver window on completion - this control enables the mode of automatic hiding the information window displaying the process of solving systems of equations in all studies. By default, this mode is not set.

On the **[Postprocessor]** tab, the user can define global settings for viewing the results, which would affect all studies.

Main font – sets the default font for the textual information output in the visualizer window of the Postprocessor (study name, result type, etc.).

Scale font – sets the font of displaying numerical values on the color scale.

«**Value tooltip**» control enables the mode, in which a tooltip pops up with the interpolated result value corresponding to the location on the model in the Postprocessor window, aimed at by the mouse pointer.

«**Delay**» control allows specifying a time interval, after which the tooltip pops up.
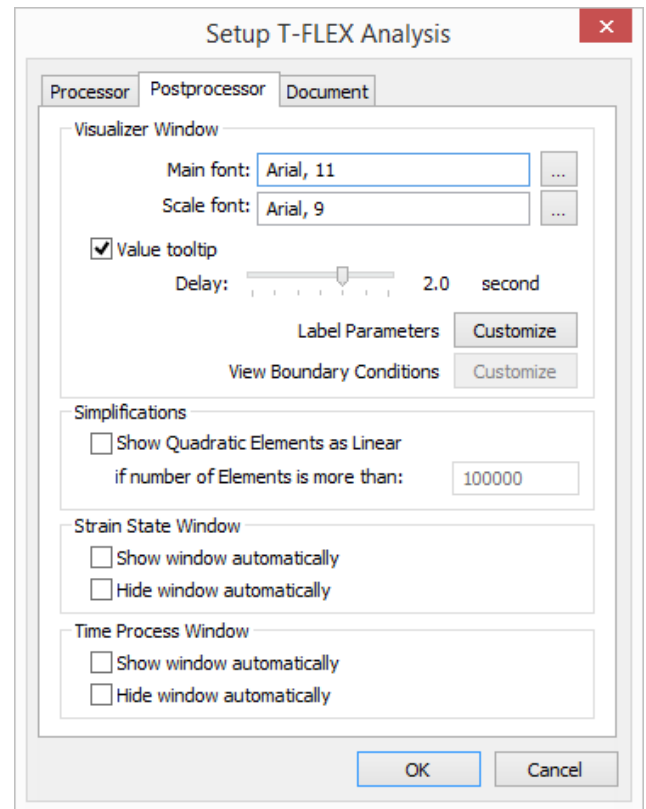
In the group «**Simplifications**» the user can specify the limit number of finite elements beyond which in the Post-processor window the calculation results are shown only for corner nodes of quadratic finite elements and omitted for the mid-side nodes (this does not affect extreme values). This mode allows the user significantly expedite downloading of results into the Postprocessor window for very large meshes (more than 10 000 000 elements).

Options «**Show window automatically**», «**Hide window automatically**» in groups «**Deformed state control window**» and «**Time process control window**» allow us to control the view of floating panels «Deformed state» and «Time process», respectively.

On the tab **[Document]** it is possible to activate the option of saving the calculation results in a separate external file. By default, the calculation results are saved in a main file of T-FLEX (extension .grb). When activating the external saving of results, the file with the name «Document name_Study name» (with extension .tfa) is created in the folder of the original document.

This option can be useful for solving studies with large amount of results, and also can speed up saving of the document (depends on the operating system).
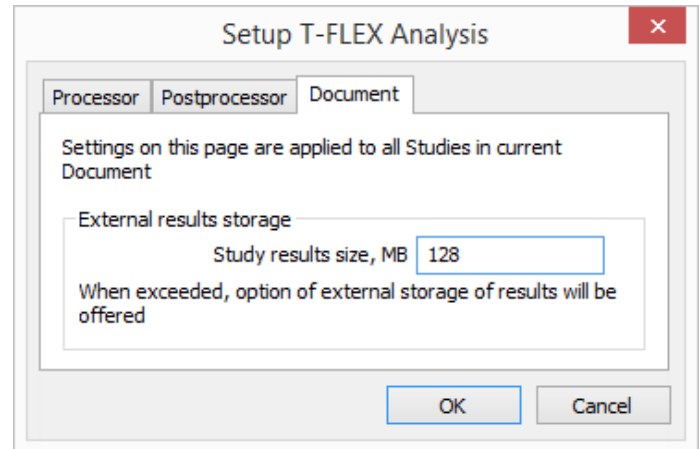
Working with the 3D Window when Preparing Study Elements

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 may 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**».
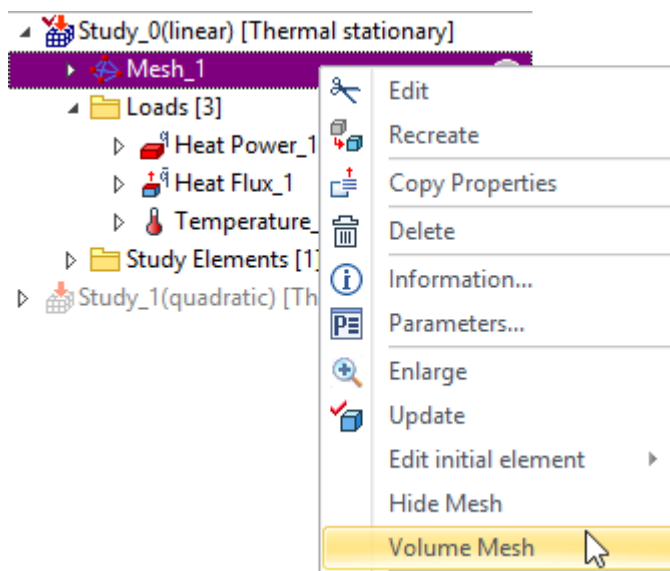
Specifics of Working with a Parametric Model

A T-FLEX CAD model is usually a parametric one. You can get full advantage of a parametric model in an analysis. All study elements (loads, restraints, mesh) related to the model can be automatically recalculated as a result of parametric modifications of the three-dimensional model, and you will not have to define them anew. It could be sometimes more convenient to manually update certain study elements that require significant computational resources (for example, meshes). To update any element of the study, you will need to select the command «**Update**» in the context menu invoked by pressing 🖰 on an element of the study.

If a parent element disappears (for example, a face with a force applied to it), then you may need to manually edit this study element to account for changes in the model.

After updating this study data, you will have to run calculations again to get up-to-date results.

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.
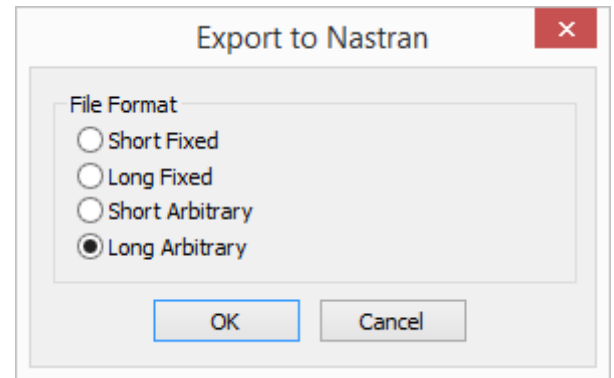
PROCESSING RESULTS (POSTPROCESSOR)

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 3D 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.

GENERAL PRINCIPLES OF WORKING WITH RESULTS

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.
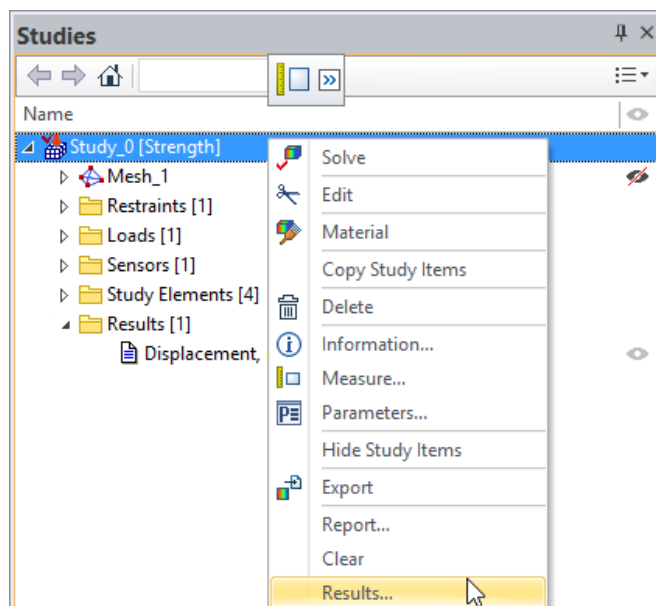
Invoking command «Customize results...»   Dialog for customizing the results list
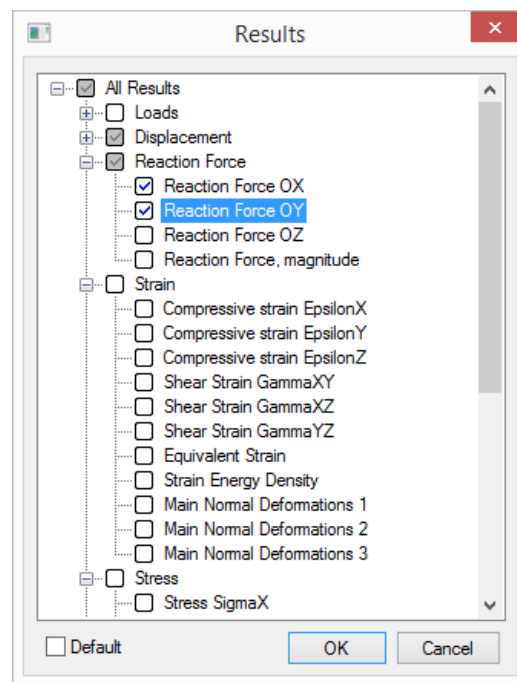
There are several ways to access results for viewing:

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 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.
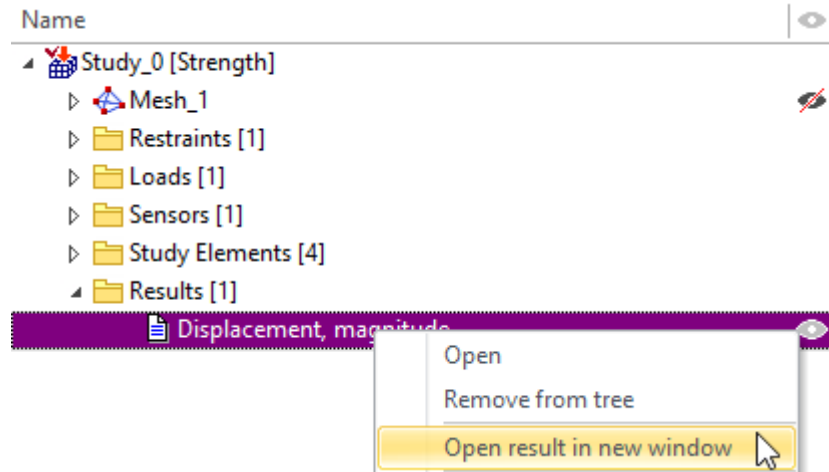
Context menu to open result

To delete a result from the studies tree, use the **Remove from tree** 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 **Results…**.

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.
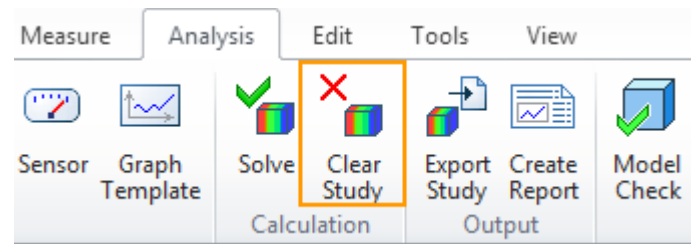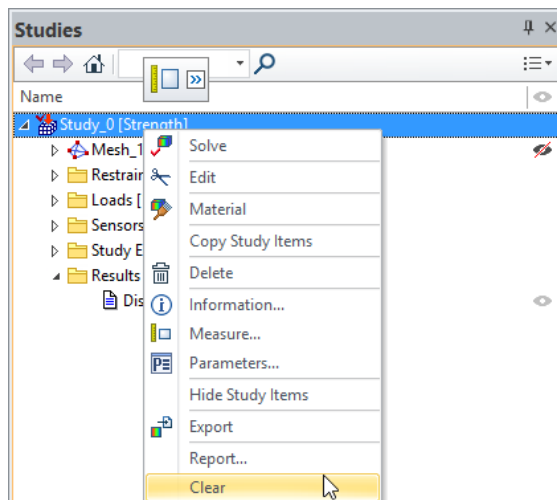
Invoking command for deleting all calculation results

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.

SETTINGS AND SERVICE COMMANDS OF CALCULATION RESULTS WINDOW

Customizing Calculation Results Window

The viewer's settings are accessed by double-clicking 🗗 🗗 in the results viewer window or from the context menu.
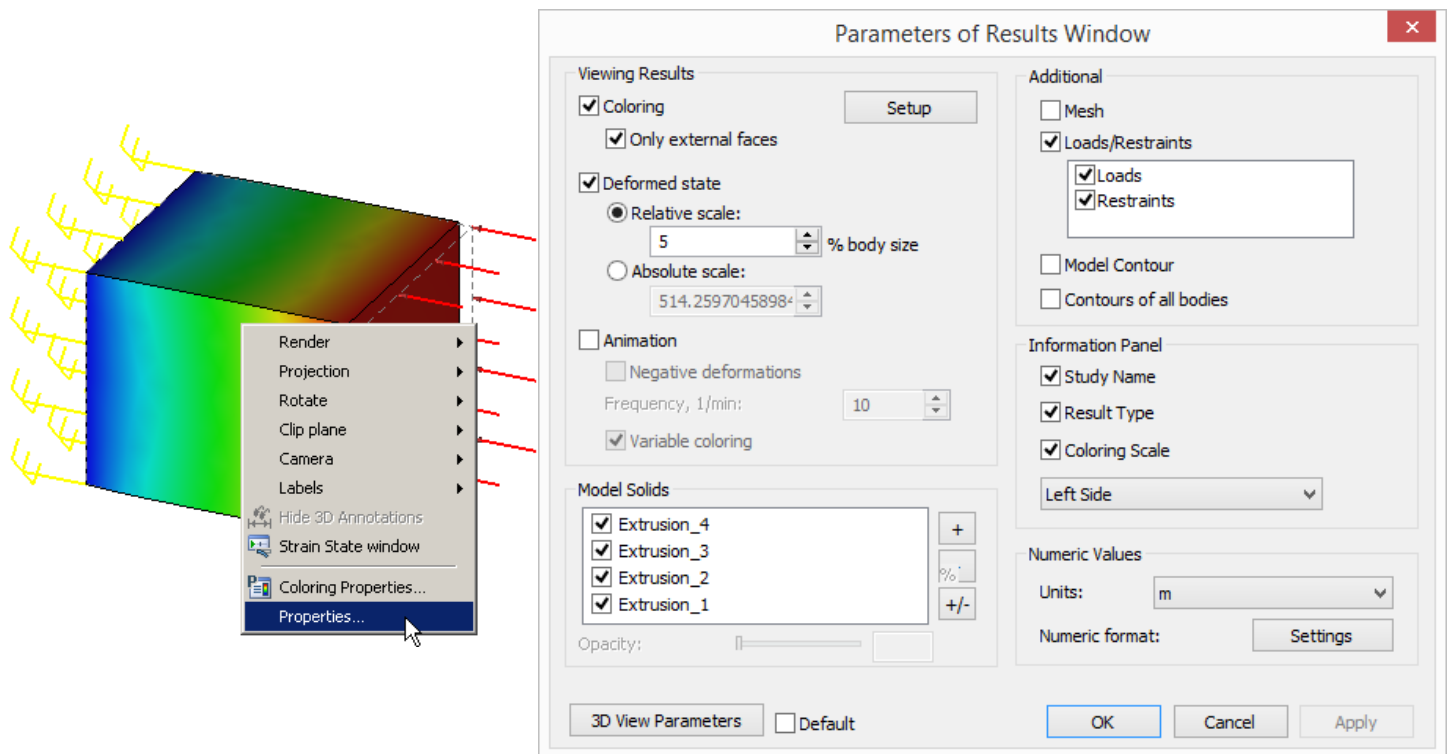
Dialog for customizing parameters of calculation results window

Parameters dialog of the calculation results window has five groups of settings.

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 on surface. Enables the mode of showing the mesh on the surface only. When the flag is off, the entire volume mesh is displayed.

Deformed 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.

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).
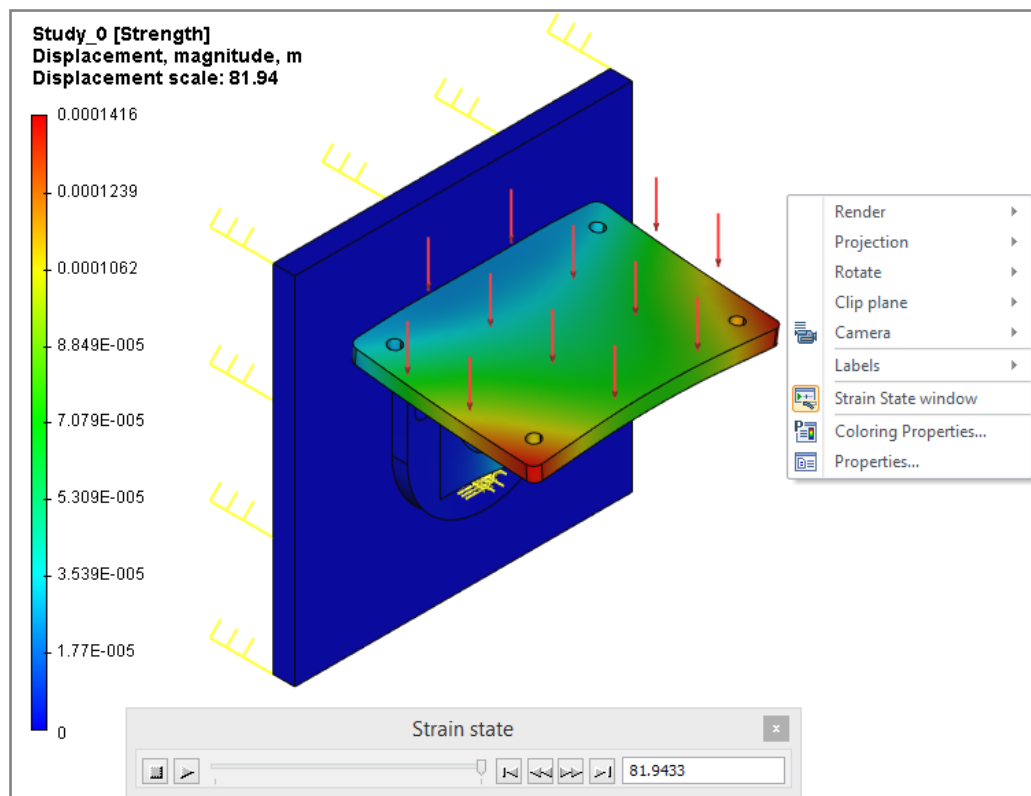
Use of the floating bar for controlling deformed state display and animation

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 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.

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".
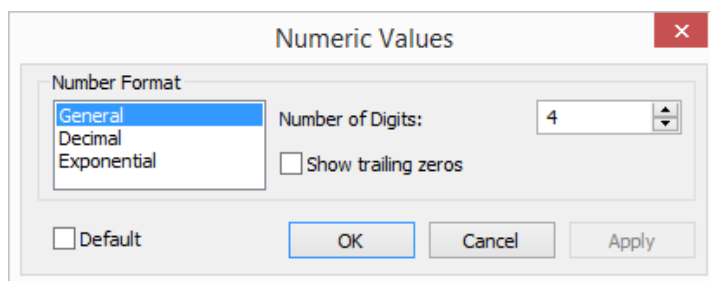
Color Scale Setup

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 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.
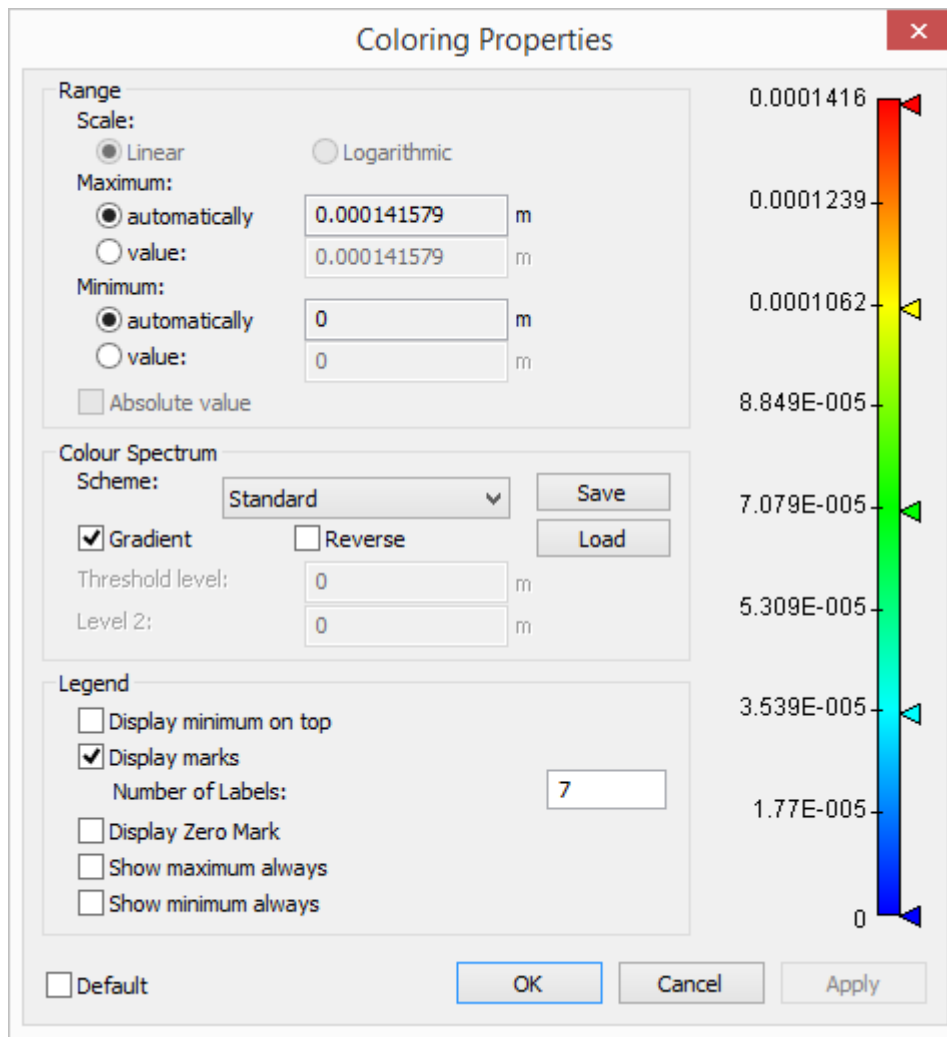
Scale options setup dialog

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 of parameters allows adjusting the number of colors in the color scale.

Scheme. There are seven predefined settings and one custom.

Standard. This option displays the scale of five main colors.

Gray-scale. 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.
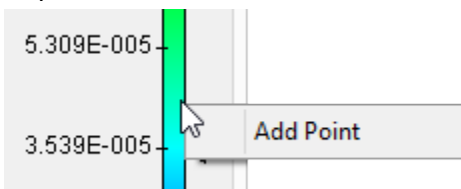
Gradient. This flag (enabled by default) serves to set smooth transition from one color to another in the color scale.

Invert. Serves to invert the color scale.

The **"Legend"** group provides the following controls:

Reverse. 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.

Show Marks. When this flag is enabled, the marks with numerical values will be displayed on the color scale in the results window.

Number of Marks. Sets the number of displayed marks.

Display Zero Mark. Serves to enable the display of the zero value mark on the color scale.

Always Show Maximum Value. When this flag is enabled, the maximum value string is displayed in the calculation results window.

Always Show Minimum Value. When this flag is enabled, the minimum value string is displayed in the calculation results window.
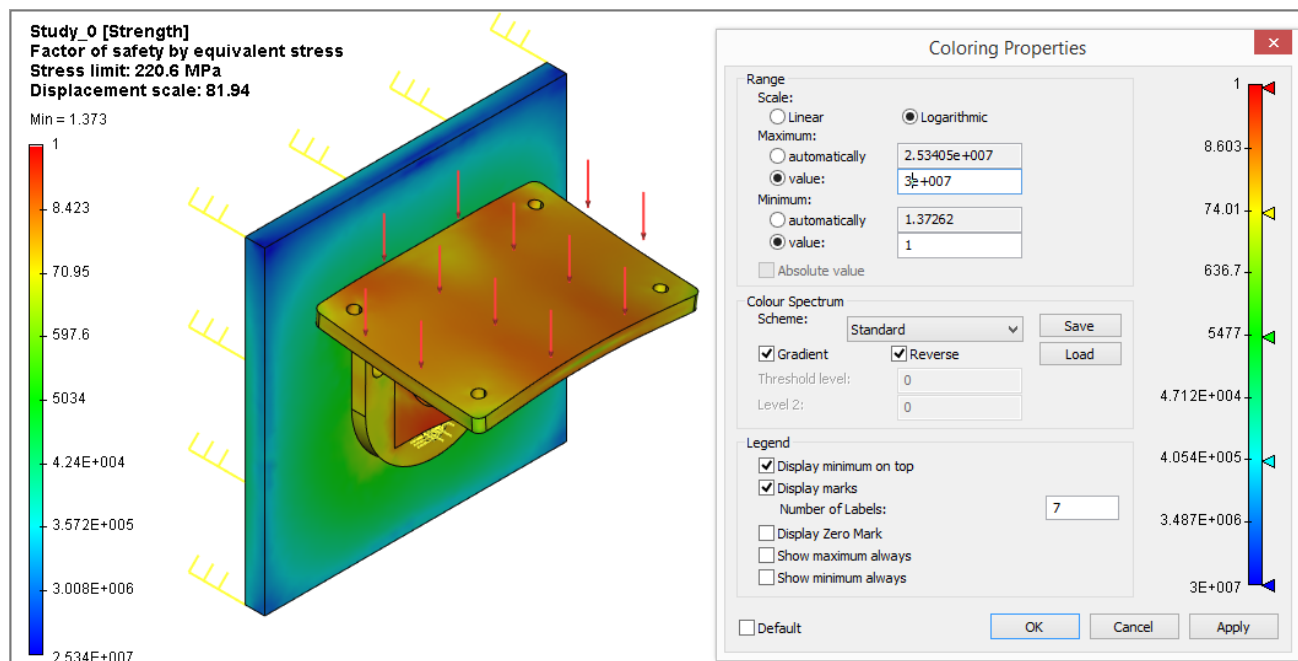
Setting the range of scale values for displaying results

Use of Sensors for Analysis of Results

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 study in a specified by the user point.

Sensors are also used for extracting control data when solving optimization studies 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 studies 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.

Sensors can be created only for active study 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 properties window the sensor type is defined as a FEA point since the sensor is used for the study of the finite element analysis.

For specifying a point, where the sensor will be created, use the automenu option:

🔷	<V>	Select point
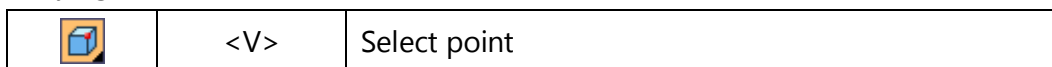
This point can be a vertex on a profile/path, vertex on a body, center of a curve/edge, middle point of an edge, center of a sphere/torus or a 3D node.
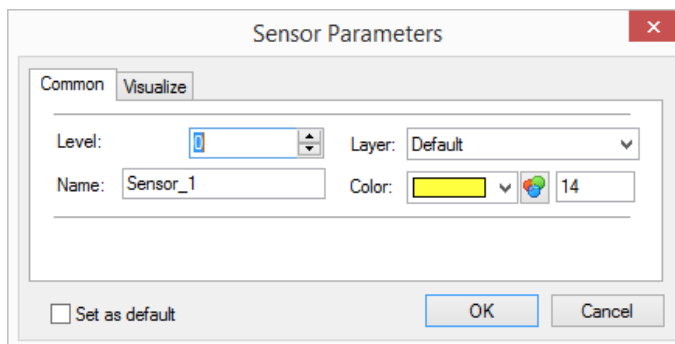
In the properties window other parameters of the sensor can be specified. As all T-FLEX objects, the sensor has general properties (name, color, layer, etc.). In addition, visualization parameters and the name can be defined for the sensor. By default, the sensors are given the name Sensor_n, where n 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 **ST: 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 3D Model Regeneration**» command, **<Shift-F7>**.

After creation of the sensor it will be displayed in the study's tree in the «Sensors» folder:
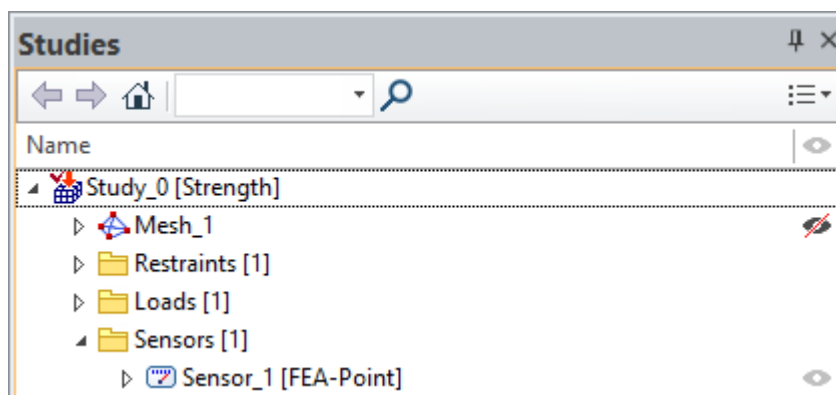
To do that, first, the user has to solve the finite element analysis study (even if the study has already been solved), and then from the context menu, invoked on selecting «Sensor» with a right mouse button in the window «Studies», call the command **Measure**.
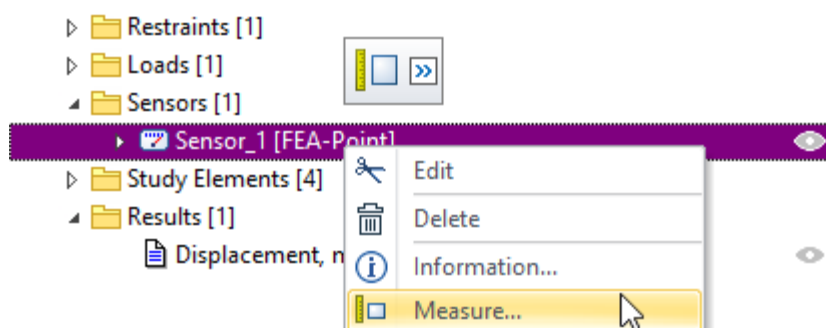
Measure dialogue:

- From the appeared properties 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]**.
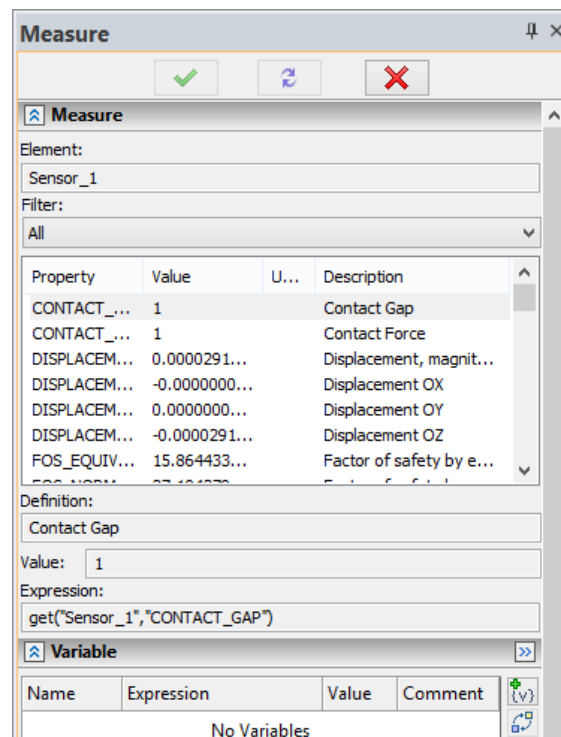
Using Graphs for Analyzing Results

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).

Graph template is created with the command:

Icon	Ribbon
	Analysis → Conditions → Graph Template
Keyboard	**Textual Menu**
<3GT>	Analysis > Graph Template

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.
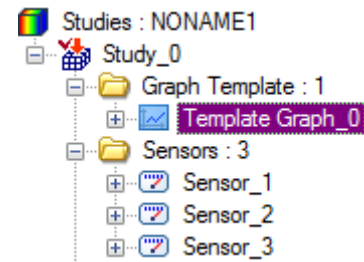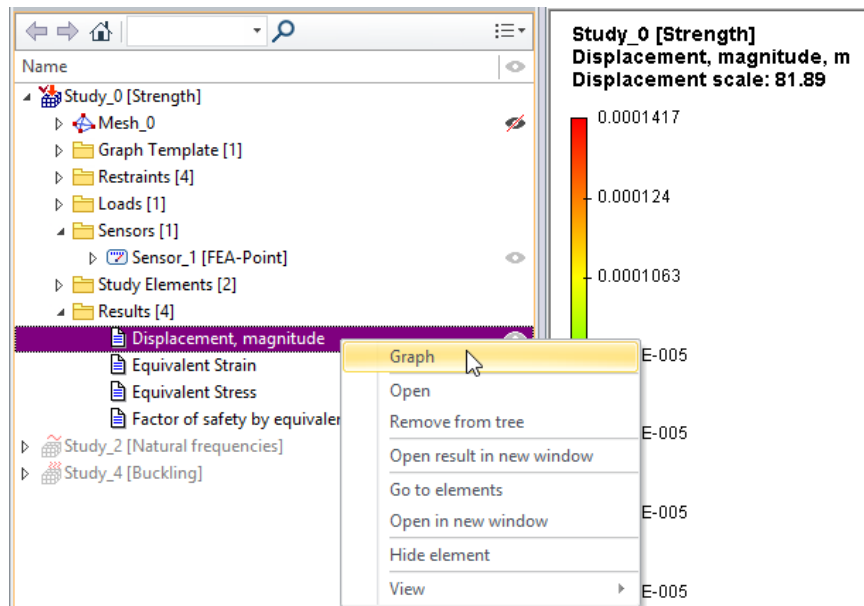
After that the graph for the given type of the results will be created. In the «Method» field the user can select one of the graph construction methods: by several sensors or by one sensor:

- 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.
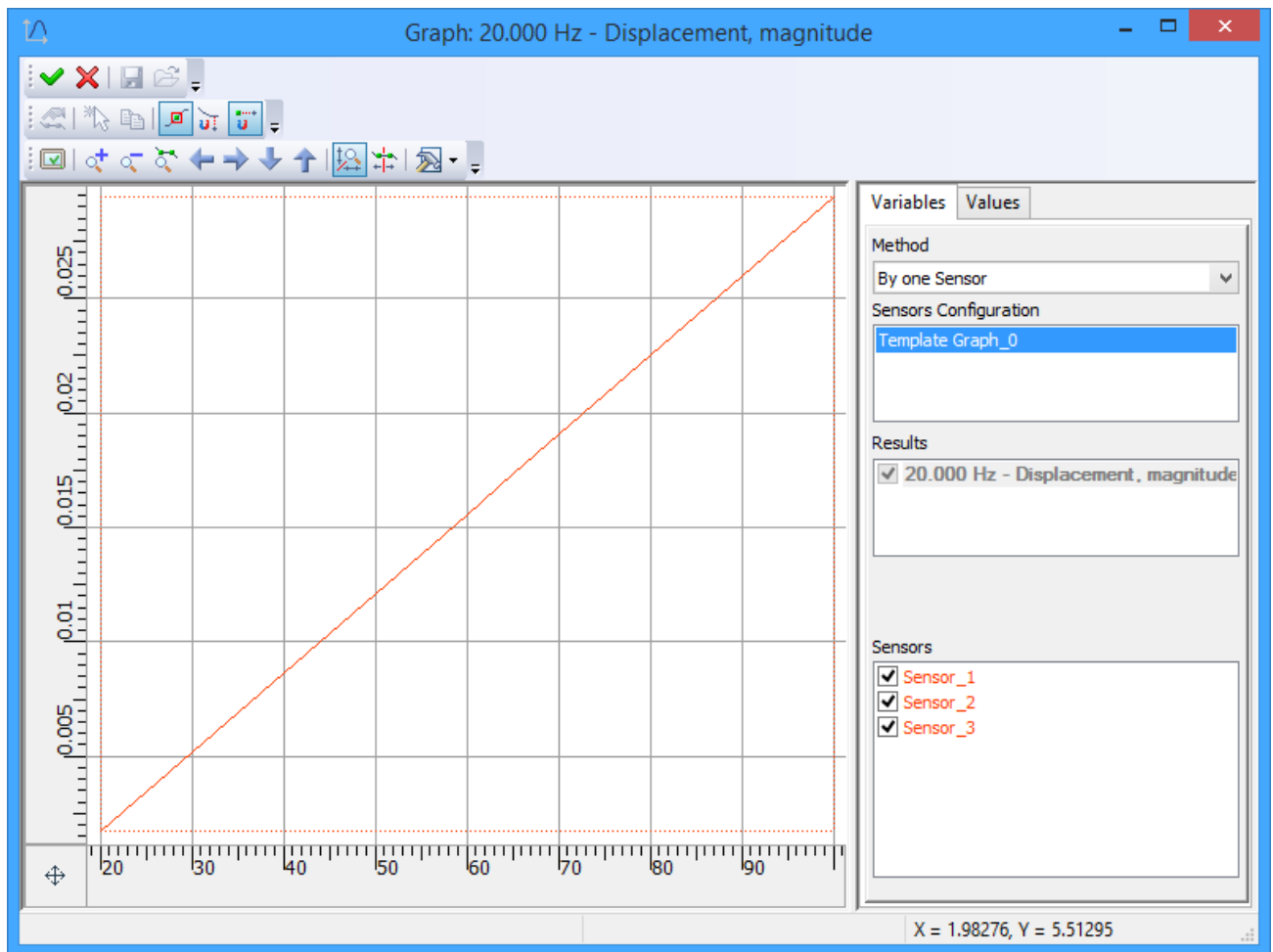
In the «Sensors» field the list of sensors related to the given graph template is shown.

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 studies 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 Reference Guide, the «Service 3D Tools and Entities» section.

Standard sequence of actions when creating a sensor:

1. Initialize the command «Create sensor».
2. Select 3D nodes, vertices for specifying sensor location.
3. If necessary, specify the sensor name and other general parameters.
4. Complete the command.

Standard sequence of actions when creating a graph template:

1. Initialize the command «Create graph template».
2. Select all or only the required sensors, if necessary correct their order while observing their location in the 3D scene window.
3. If necessary, specify the graph template name and other general parameters.
4. Complete the command.

Standard sequence of actions when creating a graph:

1. 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).
2. In the context menu select the «Graph» option.
3. In the graph edit window that appears select the graph construction method in the «Method» field and graph template in the «Sensors configurations» field.
4. 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 [PE].

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.
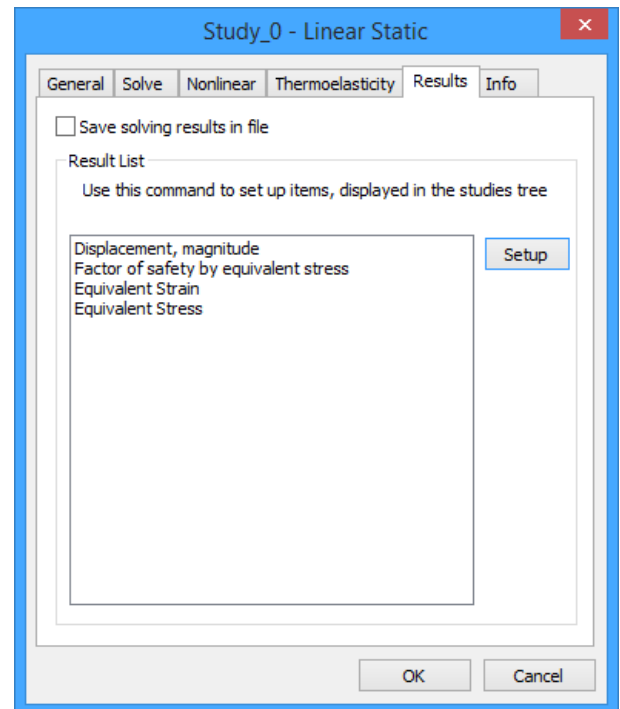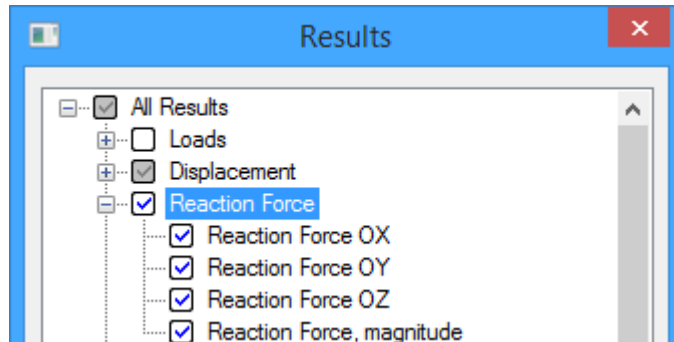
After the «Measurer» command from the result's context menu is invoked (click with a right mouse button 🖱 on the result) it is required to select elements of the model for application of the load.
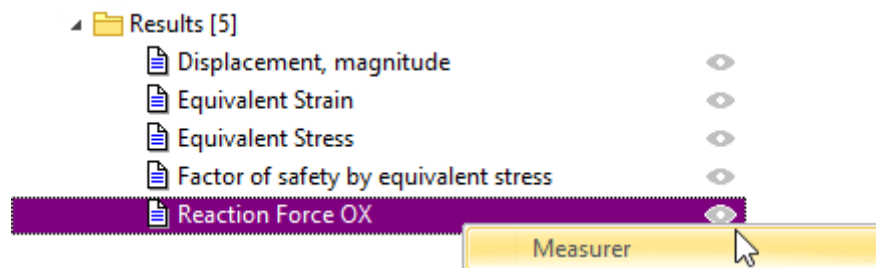
With the help of automenu, select faces, edges or vertices of the model being analyzed:

🔲	<E>	Select Element to Measure
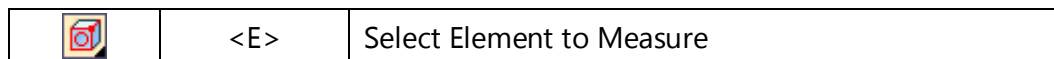
The selected elements will be added to the list.

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.
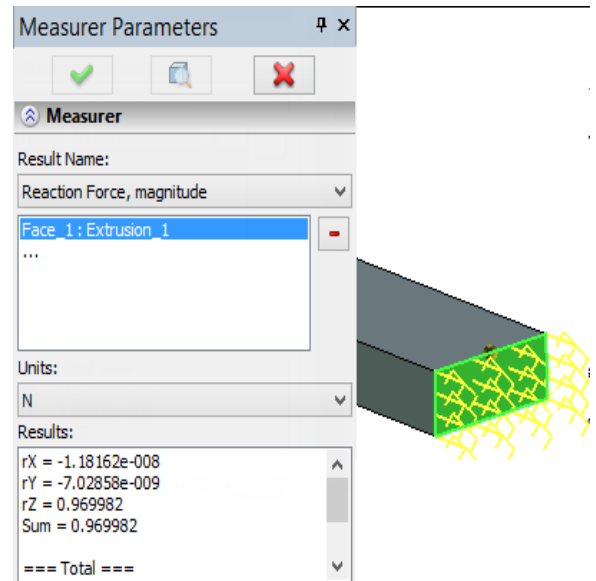
Standard sequence of actions when measuring a resultant reaction force value:

6. Enable display of the «Reaction Force» results in the study's tree.
7. Select faces, edges, vertices.
8. Specify units of measurement.
9. Complete the command.

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 study is solved successfully.

To construct a section, the user has to select the command **Clipping Plane > Activate** from the context menu that can be invoked by pressing the right mouse button in the **calculation results window**.

Study_0 [Strength]
Displacement, magnitude, m
Displacement scale: 426.26

2.933E-005

2.566E-005

2.2E-005

1.833E-005

1.466E-005

1.1E-005

7.332E-006

3.666E-006

0
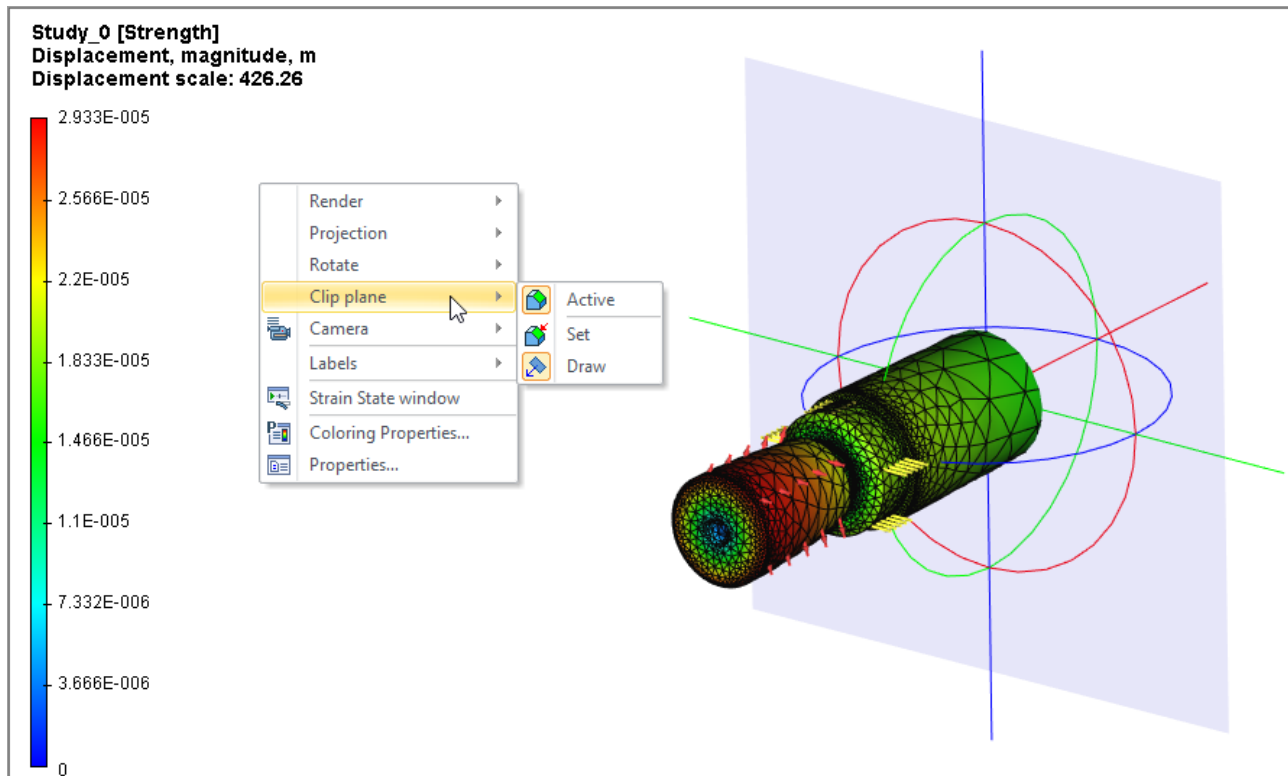
Then, the section plane has to be placed at a required position.

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.
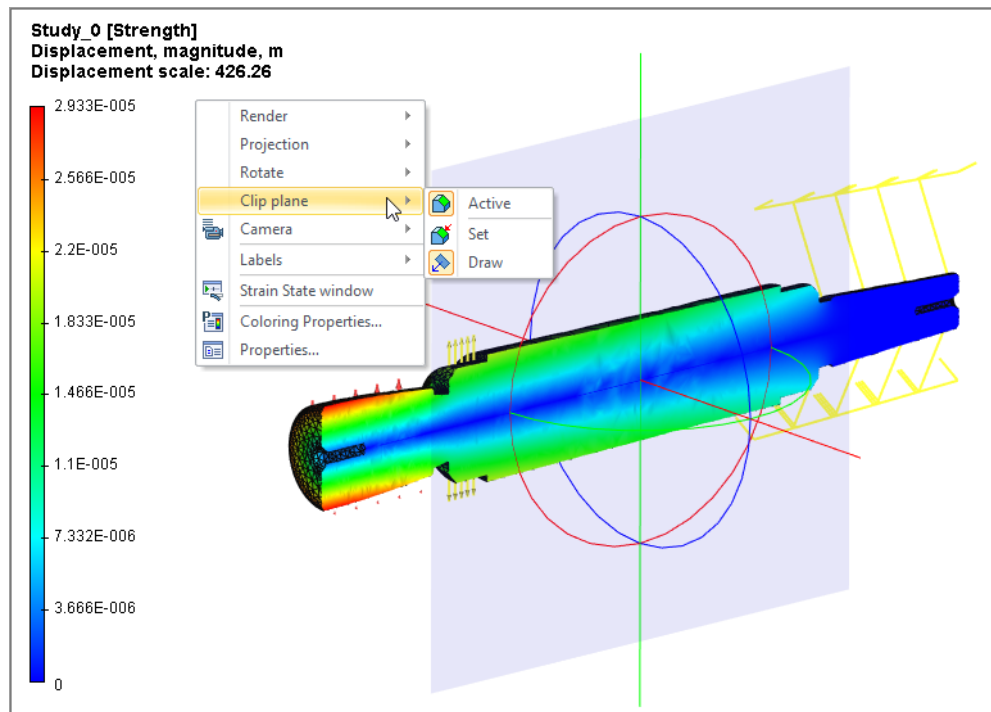
After fixing the position of the section plane, the user has to invoke the context menu, by pressing the right mouse button in the calculation results window, and select the command **Clipping Plane > Draw**. The section plane will be created.
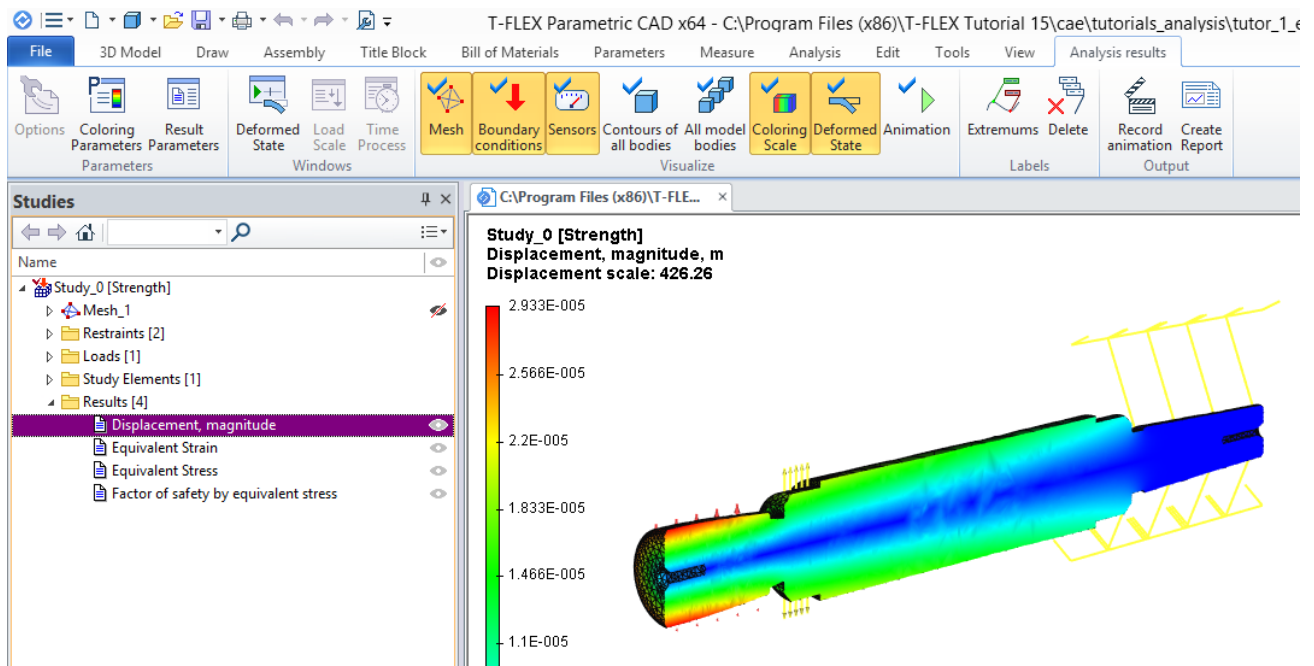
The capability of constructing a section of the finite element mesh is very important in case when the user needs to know the solution inside the structure.

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 > Report...**.
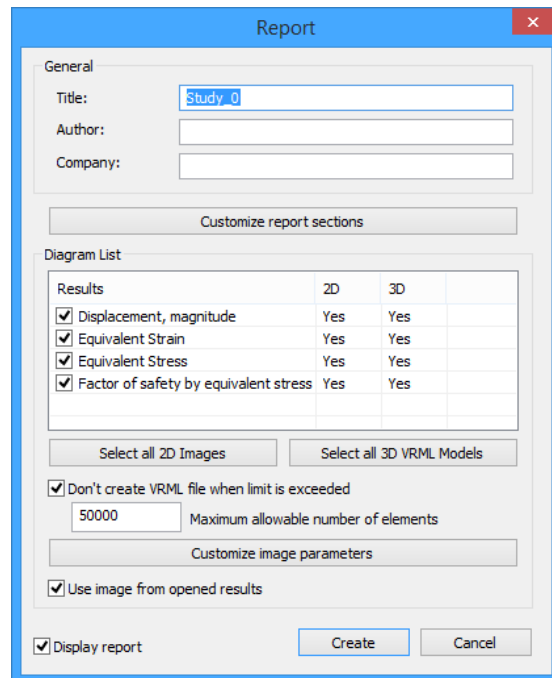
The report settings dialog can also be accessed from the context menu by right clicking 🖱 on the name of the selected study, via the command «**Report...**».

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 properties), and company information, which is also accessed from the document properties 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.

The flag **Don't create VRML file when limit is exceeded** cancels the creation of VRLM for models containing more than 50000 finite elements (default). 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
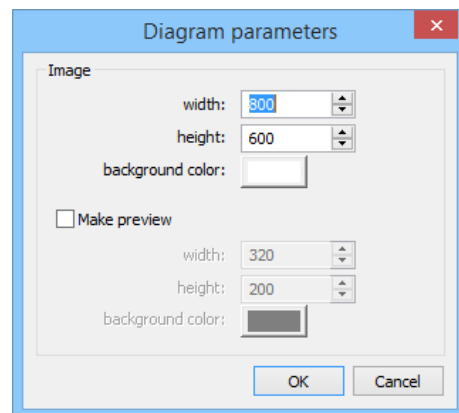
for viewing vrml-model via the Internet Explorer, it is necessary to use independent plugin, displaying vrml-model (for example, Cortona Vrml Client, http://www.cortona3d.com or another analog).

Use image from opened results 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»).

List of Tags for Generating Reports

Table 1

Tag	Tag value
$(TaskName)	Study title
$(TaskComment)	Comments to the study defined in the study properties 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
$(MaterialPuasson)	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 A 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.
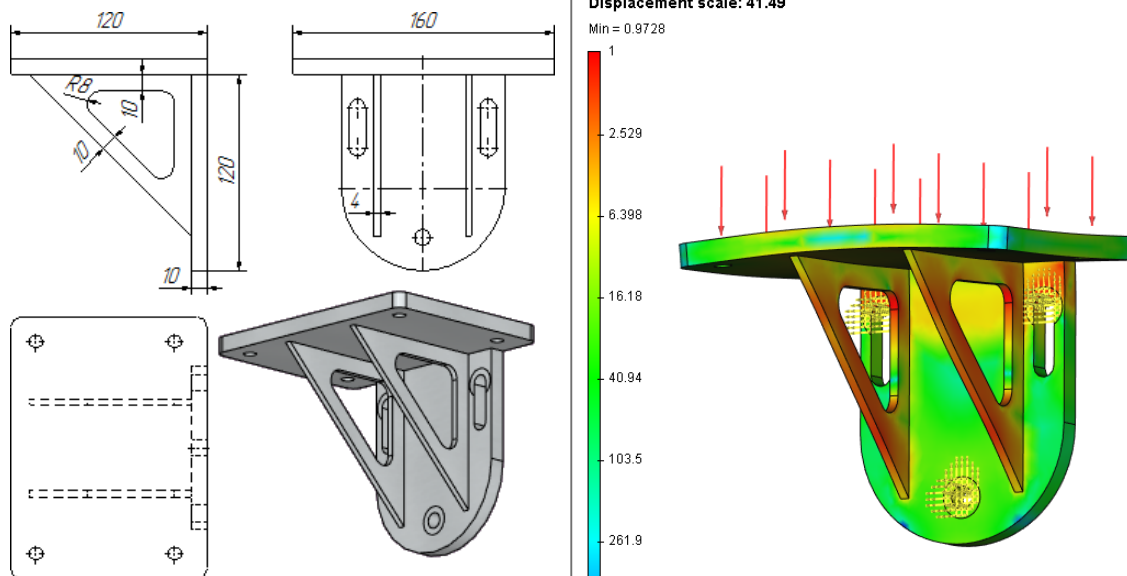
In this case, we are interested in the zones, in which the FS approaches critical values. We can scan the model using the mouse and pointing it at the places of interest, getting the result in the pop-up tooltip. However, for better visual representation of such zones, let's set up the color scheme as shown in the next figure.
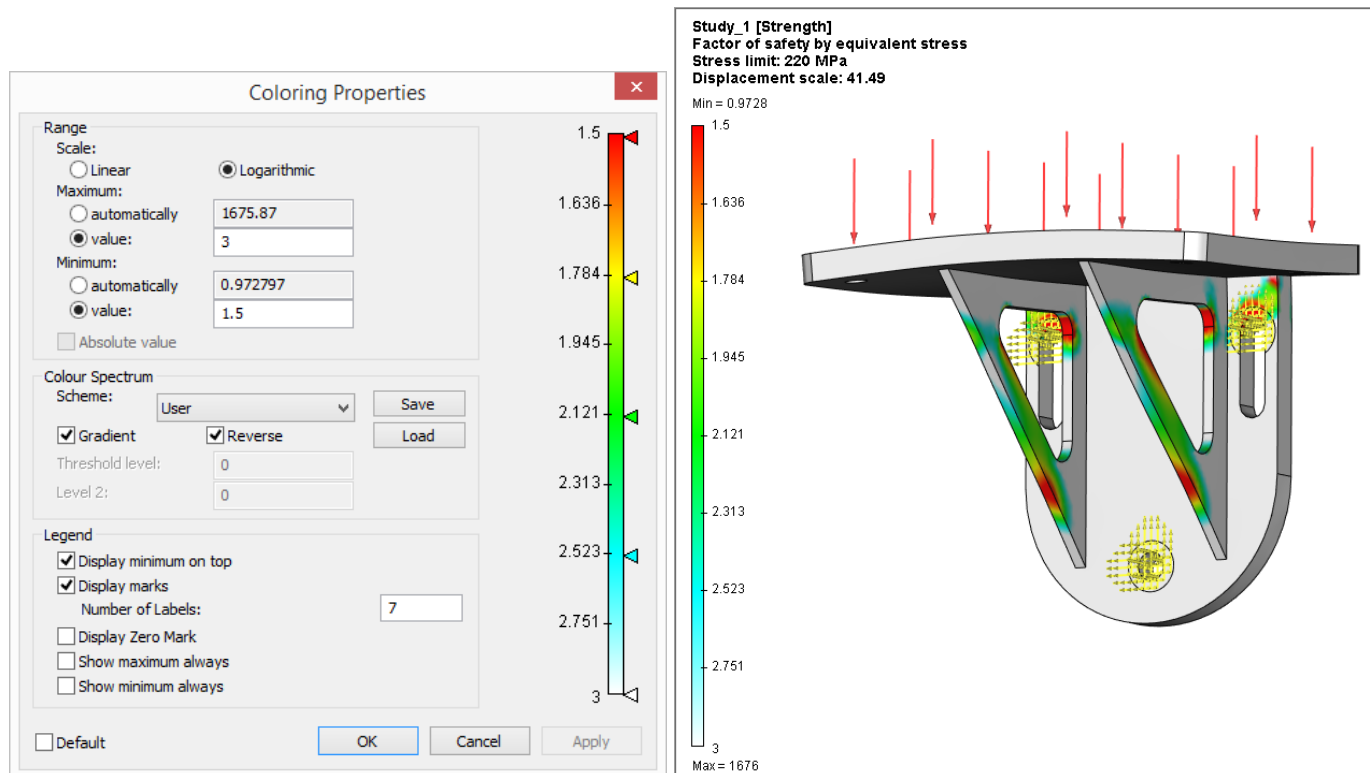
We specify the range of values of interest from 1.5 to 3. For convenience, we will correct color assignments. Anything greater than 3 will be displayed White – we are not interested in those zones for the time being. The critical zones with the value below 1.5 will be Red. The rest of the values in the 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.
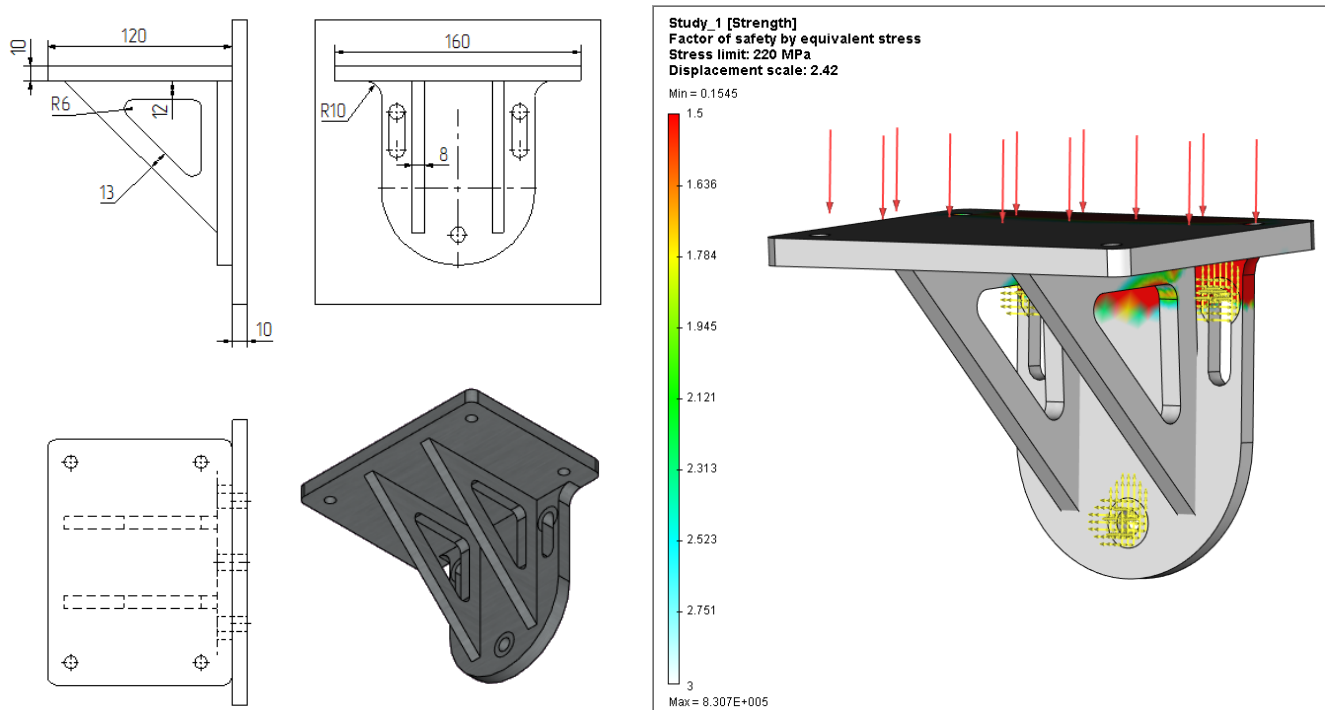
After updating the mesh and running calculations the second time, we can see that the factor of safety no longer reaches critical values in the «problematic» zones.

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.
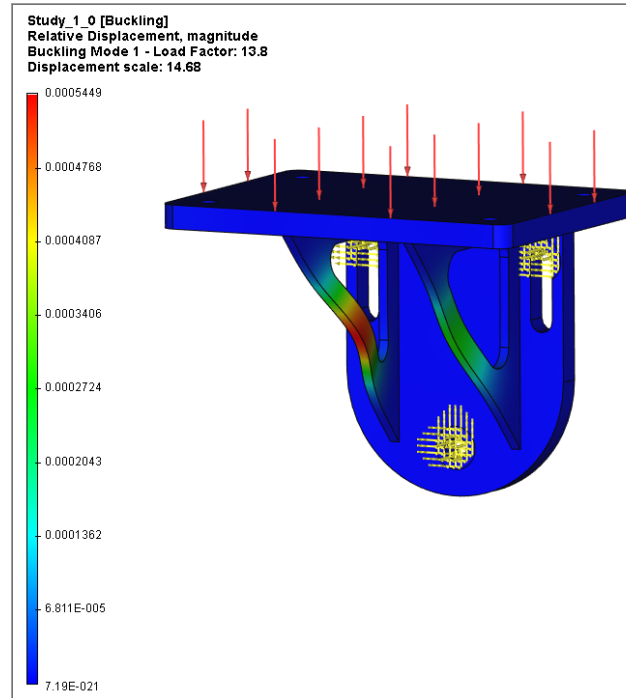
STATIC ANALYSIS

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 Ksafe.

$$\sigma \cdot K_{\text{safe}} \leq [\sigma]$$

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 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.
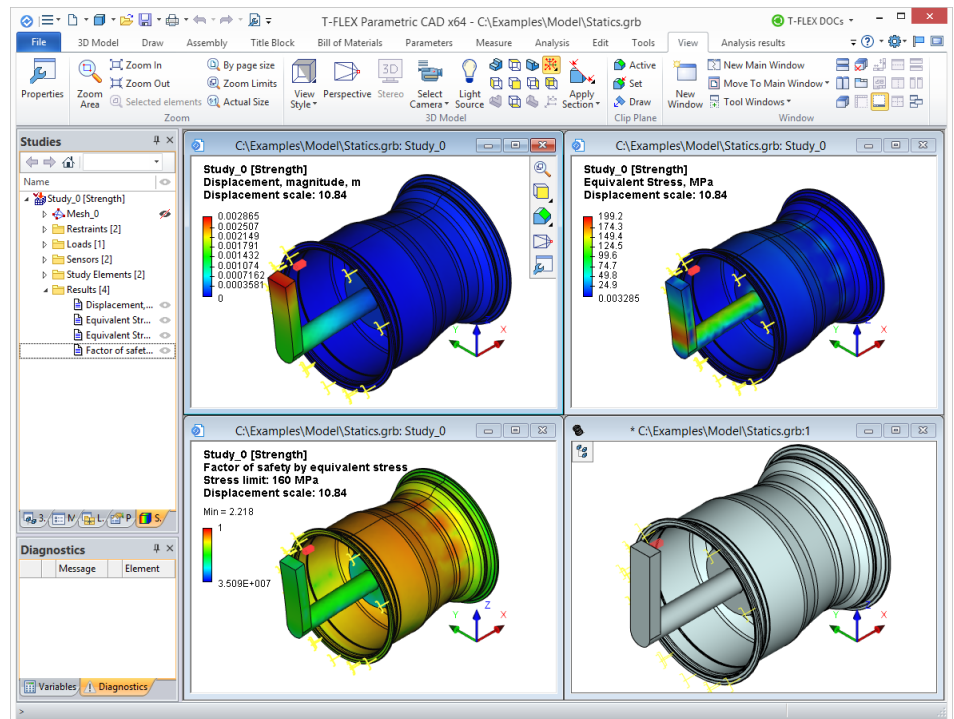
Details of Static Analysis Steps

STATIC ANALYSIS ALGORITM

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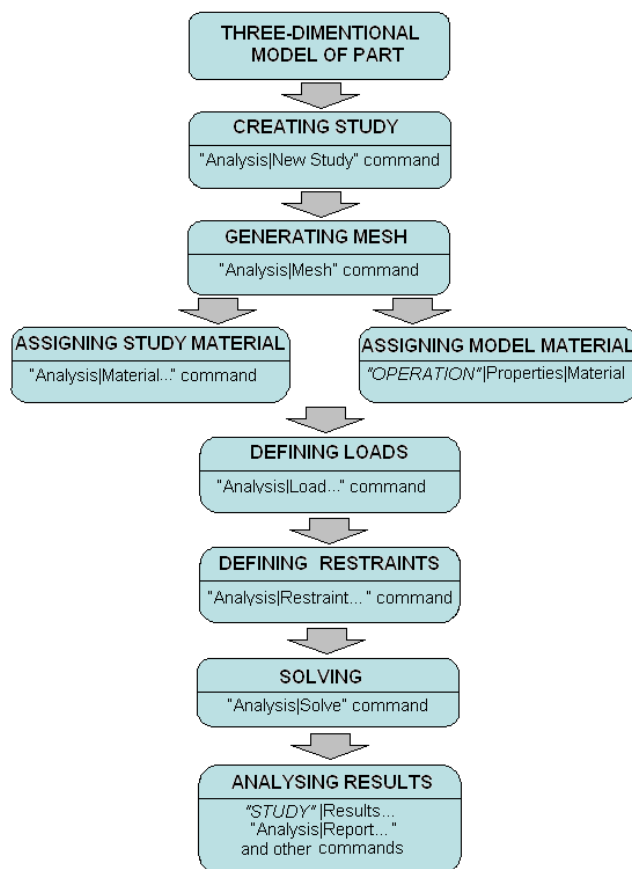
Icon	Ribbon
🟧	Analysis → Analysis → New Study → FEA Study
Keyboard	Textual Menu
<3MN>	Analysis > New Study > FEA Study

To perform a static analysis, when creating a study, the user specifies its type - "Static strength" in the command's properties 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 properties 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.
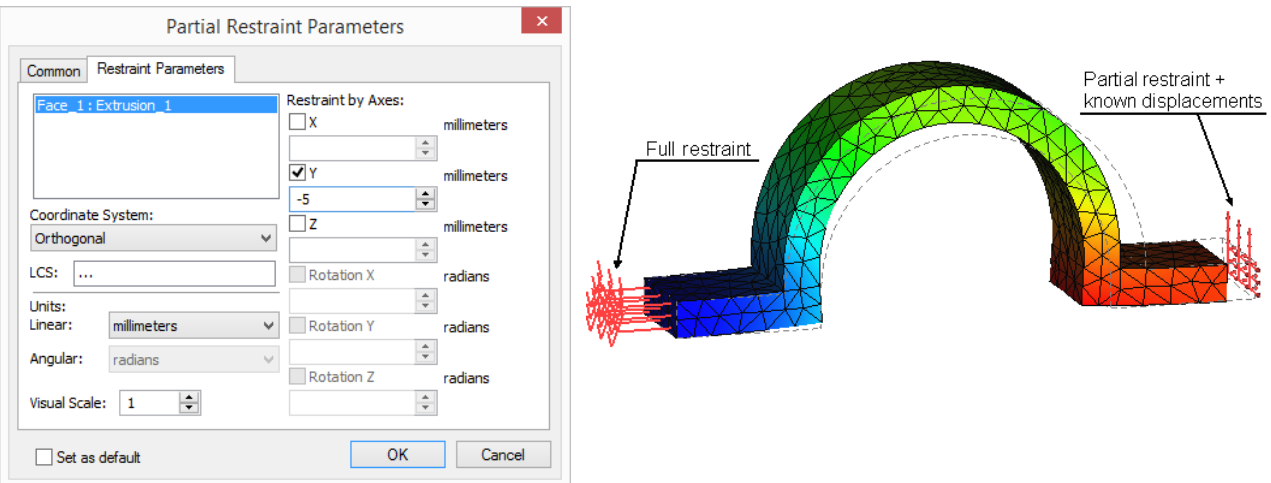
Example of using known displacements

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", "Moment"**). 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 tetrahedras in the finite element mesh.

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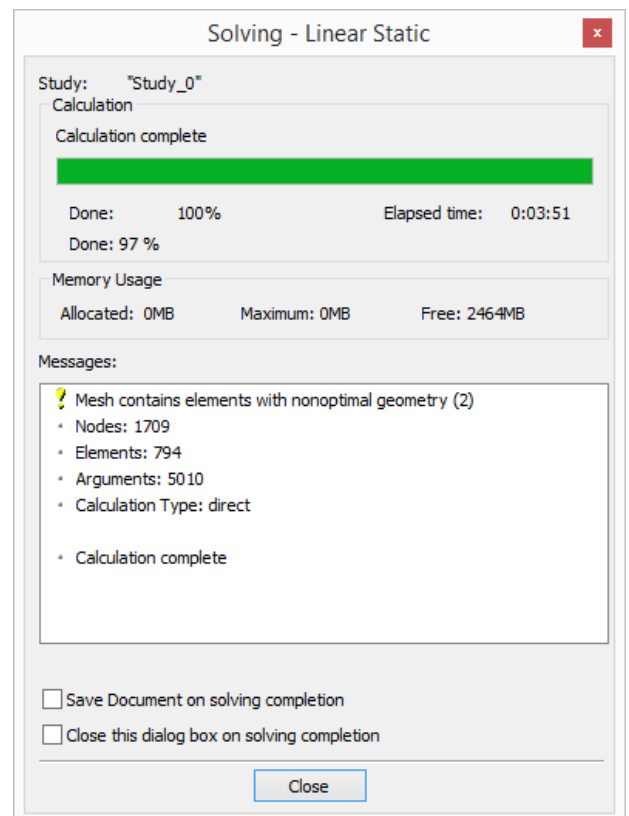
Calculation Type - the algorithm used for solving equations. Types of possible algorithms and their use are described in the section "Settings of Linear Statics Processor".

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. The **Memory usage** group shows the current state of memory and it shows if the computer being used is suitable for solving large studies. 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: $\Delta = \sqrt{x_i^2 + y_i^2 + z_i^2}$, 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:

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

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:

$$\sigma_I = \max\left(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|\right)$$

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:

$$K = \frac{[\sigma]}{\sigma_{eq}}$$

Safety factor by shear stresses is evaluated as:

$$K_\tau = \frac{[\sigma]}{2 \cdot \tau_{max}}, \tau_{max} = \frac{\sigma_1 - \sigma_3}{2}$$

Safety factor by normal stresses is evaluated as:

$$K_n = \frac{[\sigma]}{\sigma_1}$$

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

ε_1, ε_2, ε_3 - principal strains ($\varepsilon_1 \geq \varepsilon_2 \geq \varepsilon_3$).

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 $F_{m_i} = \sqrt{F_{x_i}^2 + F_{y_i}^2 + F_{z_i}^2}$ where F_{x_i} - x-component, F_{y_i} - y-component, F_{z_i} – 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.

Algorithm for Static Strength Evaluation Based on Modeling

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:

1. **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 correctness of applied loads and to assert correctness 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.
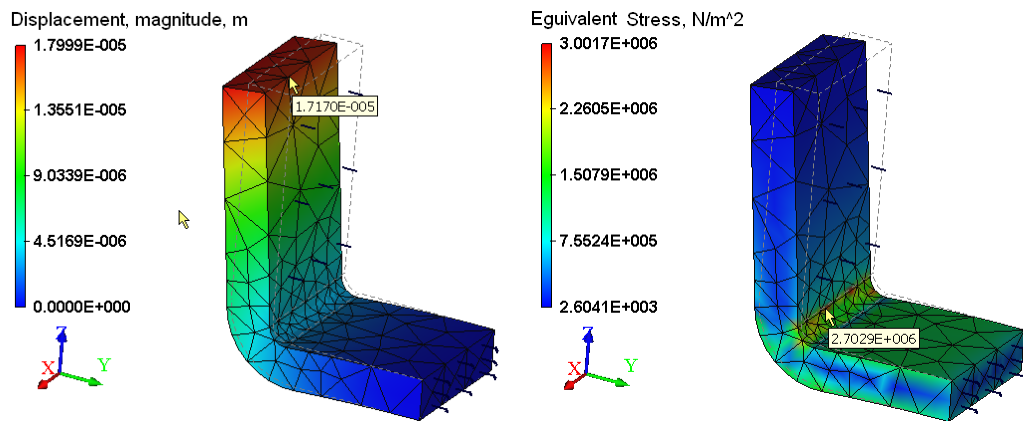
The diagram of absolute displacements and equivalent stresses

2. **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:

    a) Determine, at what locations and in which elements of the structure the largest stress develops;

    b) 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.

3. **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.

Settings of Linear and Nonlinear Statics Processor

The user-defined study properties are saved together with the document and are inherited upon copying a study. The main purpose of the study properties 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.

The [General] tab serves for defining descriptive properties 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.
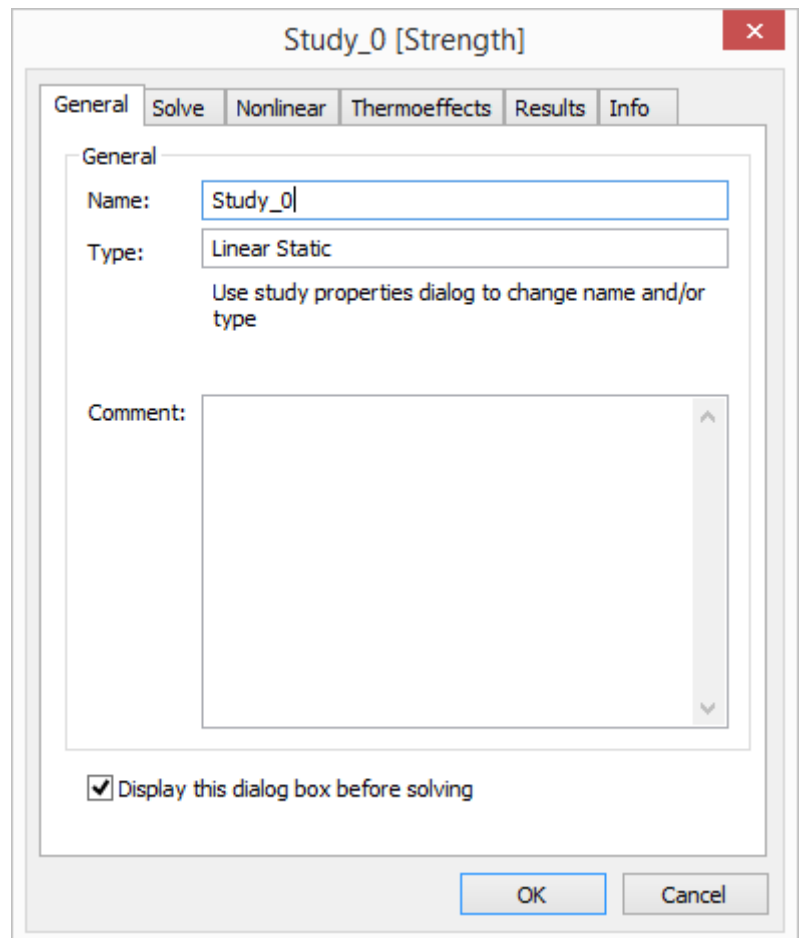
The [Solve] tab serves for defining processor properties for solving linear statics equations.

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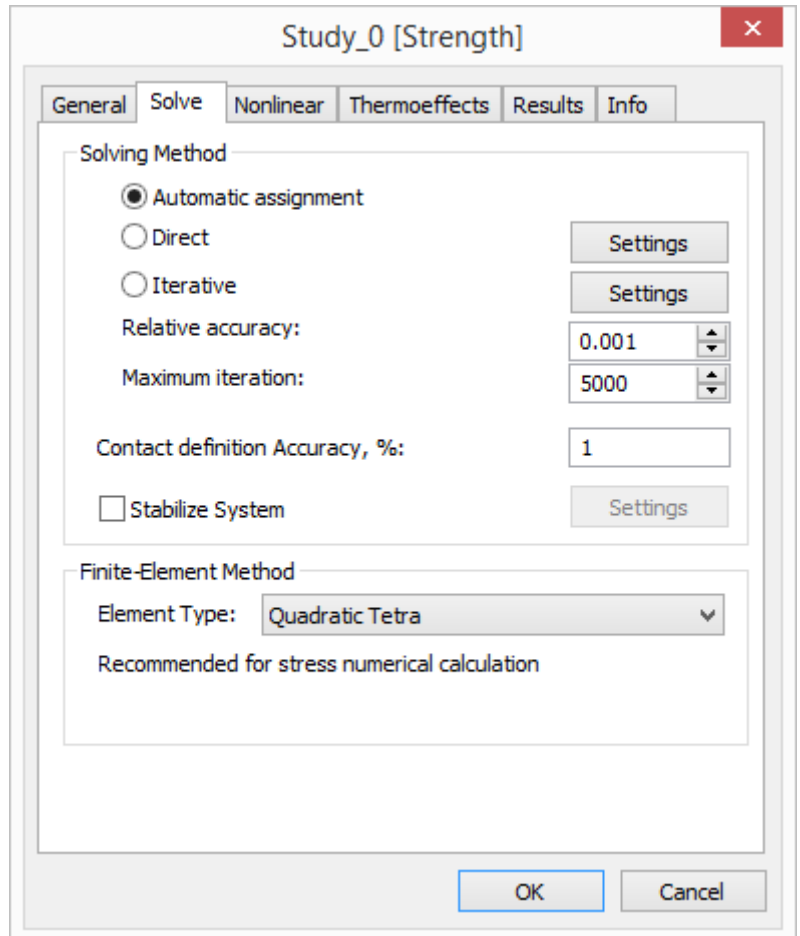
Iterated. The systems of equations are solved by iterative methods. This method is used by default for solving systems of equations, built based on a quadratic finite element. The following two options can be set for the iterative method: relative tolerance and the maximum number of iterations.

Relative miscalculation - 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) and random access memory of the computer system when solving SLAE by a direct or iterative method ([**Options**]).
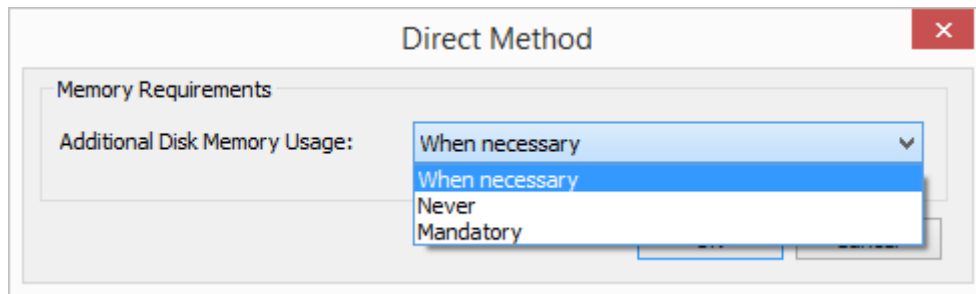
Dialog of settings for direct method of solving equations

There are three options for using additional disk memory: automatic, not available, 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 unknowns 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 large-scale studies 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).

precision.
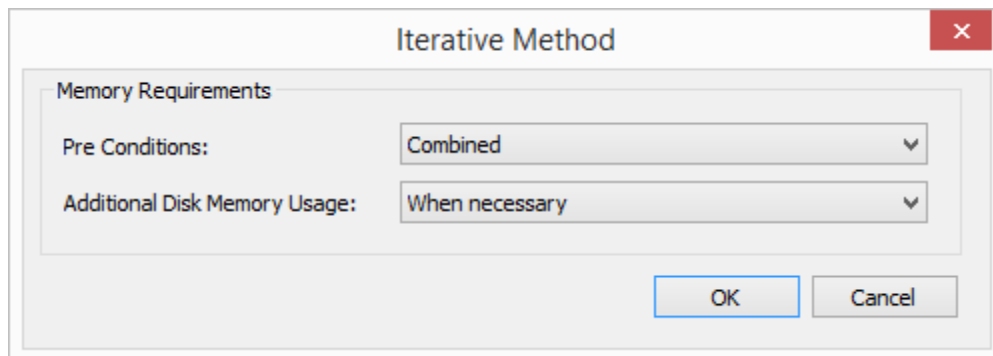
Dialog of settings for iterative method of solving equations

In the iterative method settings the user can select "**Preconditioner**" for the system of equations. *Preconditioner* 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 factorization – 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 unitary – 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 studies 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 the 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 study. 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
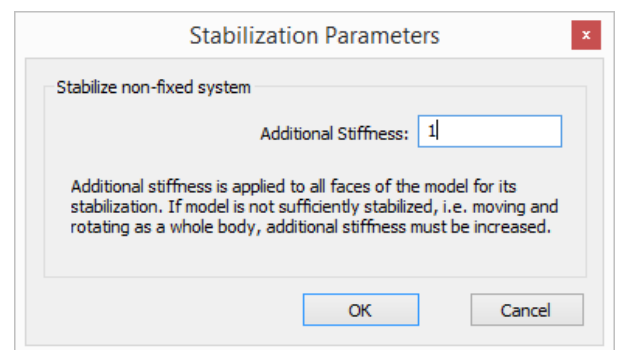
forces (do not cause additional reaction forces, for example, constrain displacements in those directions along which the force components are equal to zero or along axes of symmetry).
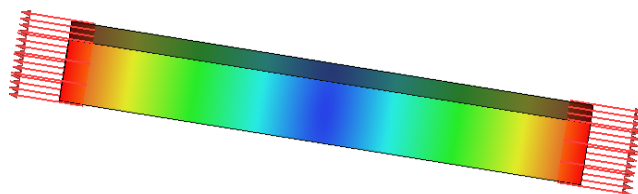
Analysis of a non-fixed model equilibrated by forces

The [**Settings**] button, located next to the «Stabilize the system» flag, allows us to select the magnitude of additional stiffness in N/m, suitable for conditions of the given study.

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.

Consider dependence of physical properties from temperature. When the flag is active dependence of material properties from to the temperature is taken into account.

The properties are set in the parameters for each material.

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 that were defined by the command **Analysis > Thermal Load > Temperature** are included in the static analysis.

Use heat-task 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
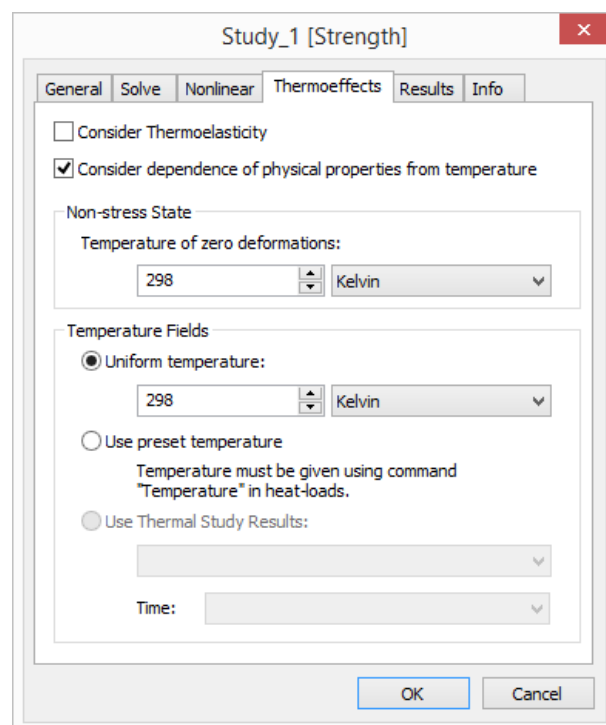
time instant, to which the solution pertains. Please note that certain conditions are to be met for using thermal analysis results as the initial temperature conditions:

1. 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:

   a) Create a study of the type "Thermal Analysis", generate a mesh, define boundary conditions, and run;

   b) Create a study copy using the **"Copy"** command;

   c) On the tab "General" in the study properties dialog, changed the study type to "Static Analysis".

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

2. The "Calculate using linear element" property 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.

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

Save solving results in file - enables the mode, in which all analysis results are saved in the file together with the model. This allows analyzing results of an earlier calculated and saved study without the need for a new calculation. Please note that saving calculation results in a document increases the size of the document file by approximately 4.5 to 5 Mbyte per a hundred thousand Degrees of Freedom.

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 studies 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 «**Account for stiffness change at large deflections**» 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.
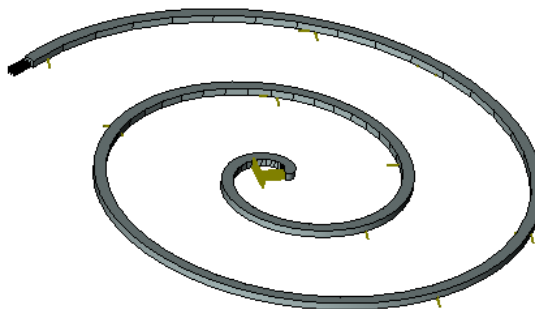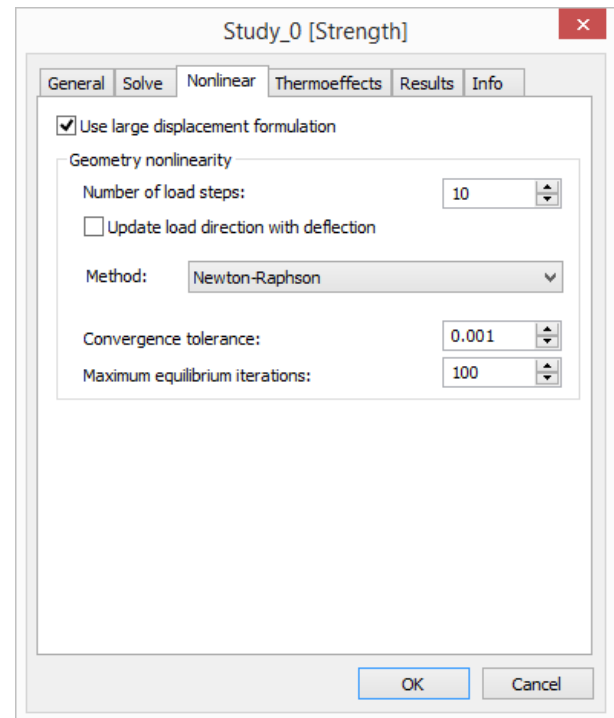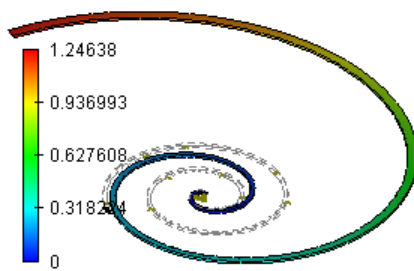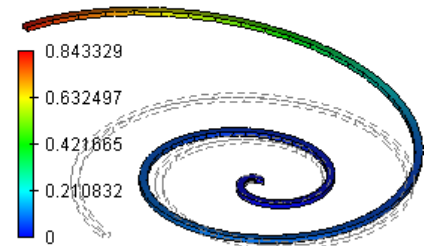
Linear analysis   Nonlinear analysis

Controls in the group **«Geometric nonlinearity»** allow the user to customize the process of solution of geometrically nonlinear studies.

For solving such studies, 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 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 does not become smaller than the prescribed tolerance.

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

Optimization Problem

Optimization study (or the study 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 study 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 study.

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 study of beam deflection. The beam clamped at both sides is subjected to the force 25 N.
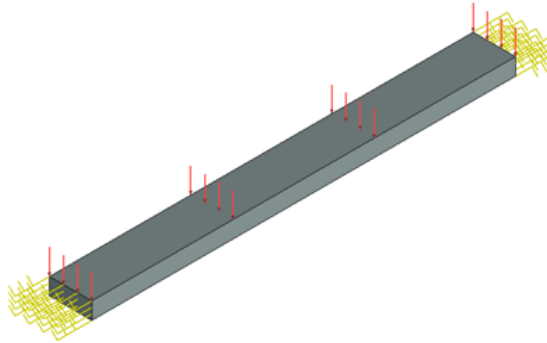
It is required to determine an admissible thickness of the beam at which the maximum value of the deflection in the direction coinciding with the direction of load does not exceed *1e-6 m* in absolute value. Length of the beam is equal to *500 mm*, width of the beam is *50 mm*, thickness of the beam is *20 mm*. The beam is manufactured from steal *AISI 1020*.

Let us first create the study 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 study, 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.
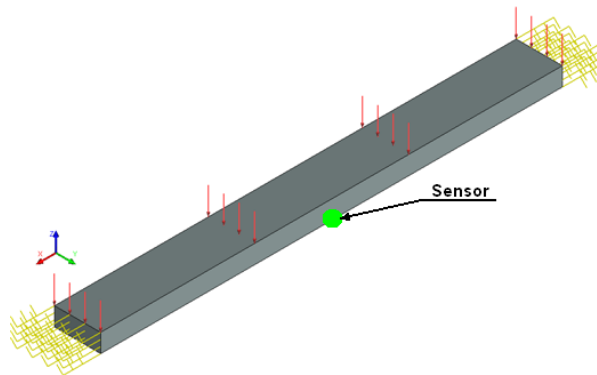
Next, after carrying out the static analysis of the study, let us create a variable with the help of a sensor. As a variable, we choose «Displacement OZ» and call it «*Displacement_Z*».
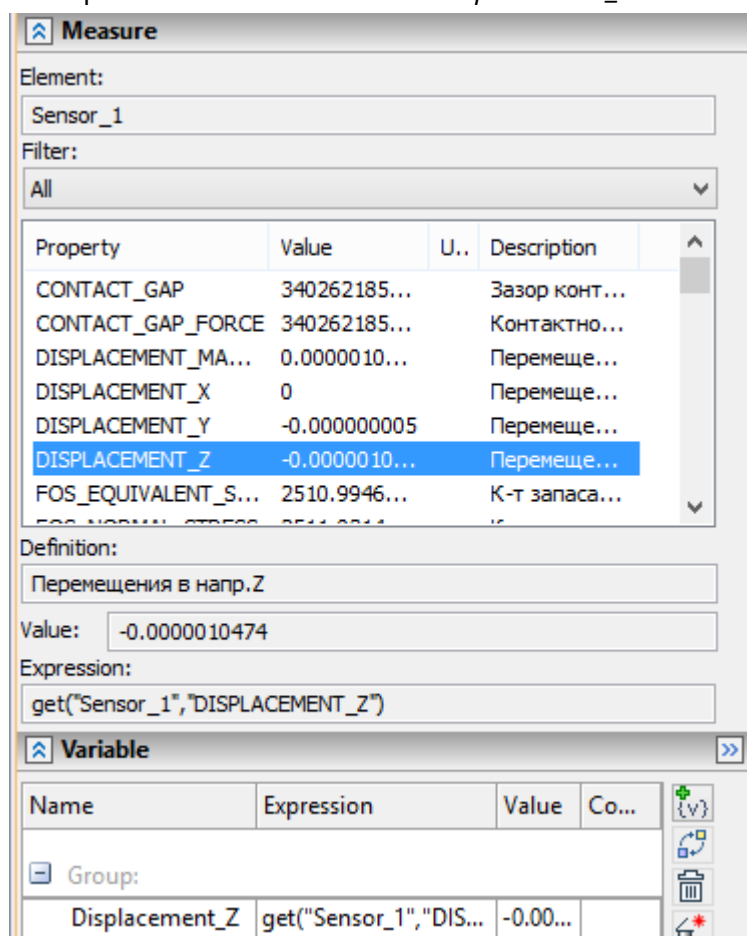
Let us create optimization study. The optimization study can be created with the help of command:

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.
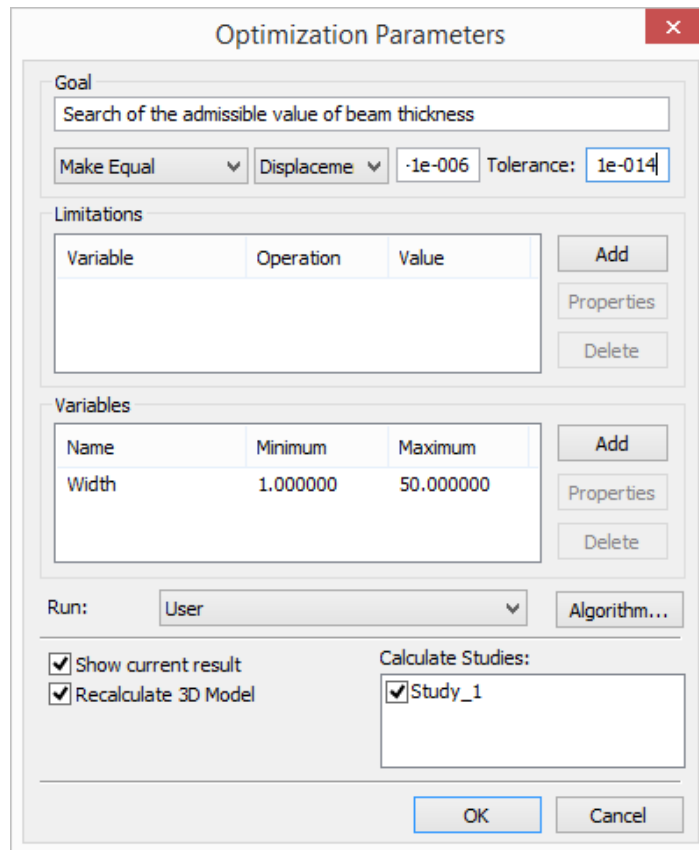
In the text line it is necessary to write down commentary for the current optimization study.

We will formulate the goal of the optimization study 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-6 m.* 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*.
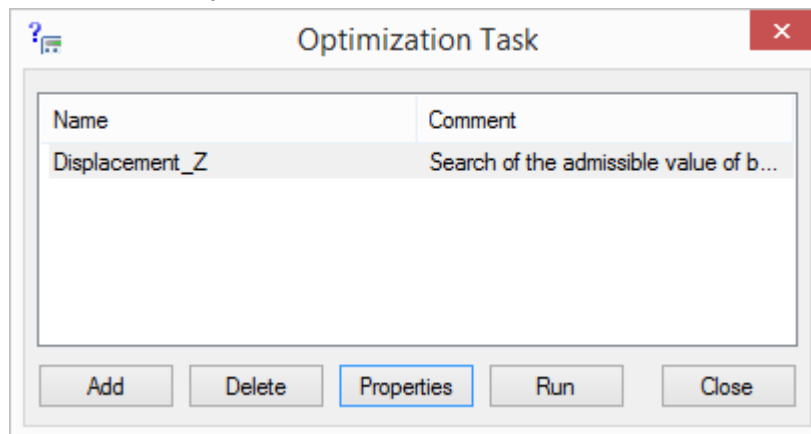
Since upon changing the beam thickness its deflection along the Z-axis also changes, while solving optimization study it will be necessary to recalculate 3D model and the static analysis study for each found value of the thickness. Thus, in the field «Calculate Studies» it is necessary to select the study whose results are used for searching the optimal value of the variable «thickness», and, also, activate the option «Recalculate 3D model».

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 **[Execute]**.

As a result of solving optimization study, we obtained the value of the beam thickness equal to 21.09*mm*, 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/m2] - is the ratio of stress with respect to relative strain $\sigma = E\varepsilon$ developing in a prism-shape specimen 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 longitudinal 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 specimen 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** σ_Π - the stress that is the upper bound in effect of purely elastic strain.
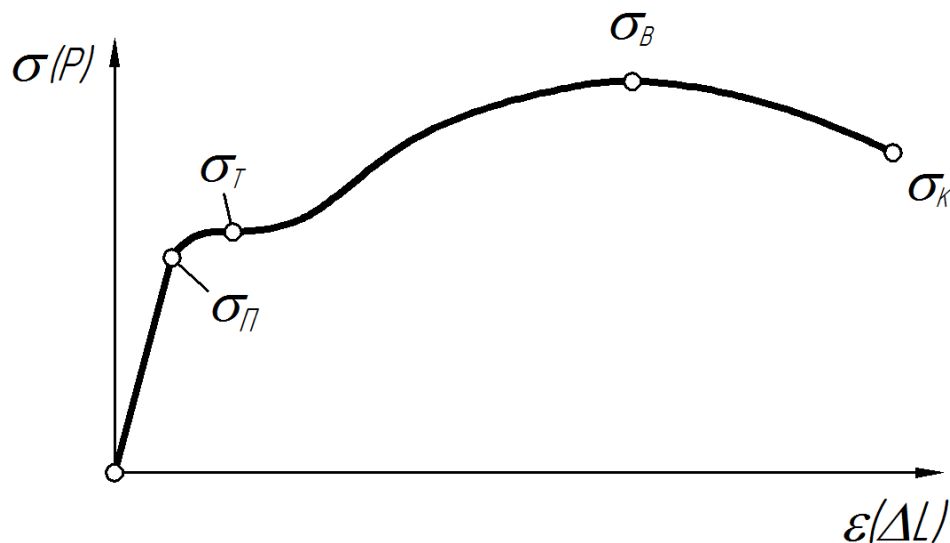
The stress (dilation) diagram for plastic materials
(for example, low-carbon steel)

> **Yield limit** σ_T. 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 [σ]**. 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** σ_r **(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 longitudinal direction, **µ**— the *Poisson's ratio*. For isotropic materials, the Poisson's ratio lies in the range $0 < \mu < 0,5$.

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	σ_T, MPa	σ_E, MPa	δ, %	ψ, %
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:

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

where l_0, F_0 -the length of the working part of the specimen and the area of the cross-section before strain; l_k - the length of the working part of the specimen after the rupture; F_k - 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 $\delta (\delta > 10\%)$ at the point of rupture are referred to as plastic materials; those referred as brittle are the materials with relative elongation of $\delta < 3\%$. For plastic materials, upon compressing to nearly the yield condition, the $\sigma = f(\varepsilon)$ 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 dilatational 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:

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

where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ - the longitudinal relative strain, $\gamma_{xz} = \gamma_{zx}$, $\gamma_{xy} = \gamma_{yx}$, $\gamma_{zy} = \gamma_{yz}$ - 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 ε_1, ε_2, ε_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 $\sigma_x, \sigma_y, \sigma_z$ - the compression-tension stress, $\tau_{xz} = \tau_{zx}$, $\tau_{xy} = \tau_{yx}$, $\tau_{zy} = \tau_{yz}$, - the shear stress.
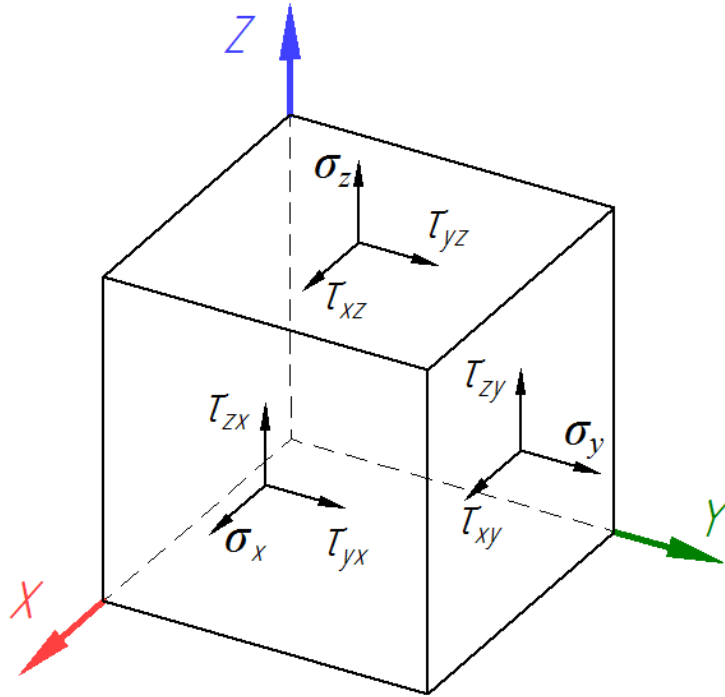
The stress state tensor components in an infinitesimal block

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:

$$\sigma^3 - J_1 \sigma^2 + J_2 \sigma - J_3 = 0 \text{, where}$$

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

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

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

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 $\sigma_{equiv} \leq [\sigma]$.

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 threshold. 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:

$$\boxed{\sigma^I_{equiv} \leq [\sigma]}$$

where $\sigma^I_{equiv} = |\sigma_1|$, if $\sigma_1 \geq |\sigma_3|$ and $\sigma^I_{equiv} = |\sigma_3|$, if $\sigma_1 \leq |\sigma_3|$.

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 threshold. For a plastic material, the strength criterion appears as $max|\varepsilon| \leq [\varepsilon]$, where $[\varepsilon] = \dfrac{[\sigma]}{E}$. If, for example,

$$max|\varepsilon| = \varepsilon_1 = \frac{1}{E}[\sigma_1 - v(\sigma_2 + \sigma_3)]$$

, then $\sigma^{II}_{equiv} = \sigma_1 - v(\sigma_2 + \sigma_3) \leq [\sigma]$.

For brittle materials, the strength criterion appears as:

$$\varepsilon_{max} \leq \left[\varepsilon_p\right] = \frac{\left[\sigma_p\right]}{E}, \quad \left|\varepsilon_{min}\right| \leq \left[\varepsilon_c\right] = \frac{\left[\sigma_c\right]}{E}$$

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 threshold.

The strength criterion appears as: $\tau_{max} = \dfrac{\sigma_1 - \sigma_3}{2}$, where $[\tau] = \dfrac{[\sigma]}{2}$. Consequently:

$$\sigma_{equiv}^{III} = \sigma_1 - \sigma_3 \leq [\sigma]$$

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 *ud*, 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:

$$[u_d] = \frac{1+\mu}{3E}\left(\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_1\sigma_3\right)$$

The strength criterion appears as *ud* $u_d \leq [u_d]$, where $[u_d] = \dfrac{1+\mu}{3E}[\sigma]$.

Consequently:

$$\sigma_{equiv}^{IV} = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_1\sigma_3} \leq [\sigma]$$

or

$$\sigma_{equiv}^{IV} = \frac{1}{\sqrt{2}}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_z - \sigma_x\right)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2)} \leq [\sigma]$$

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.
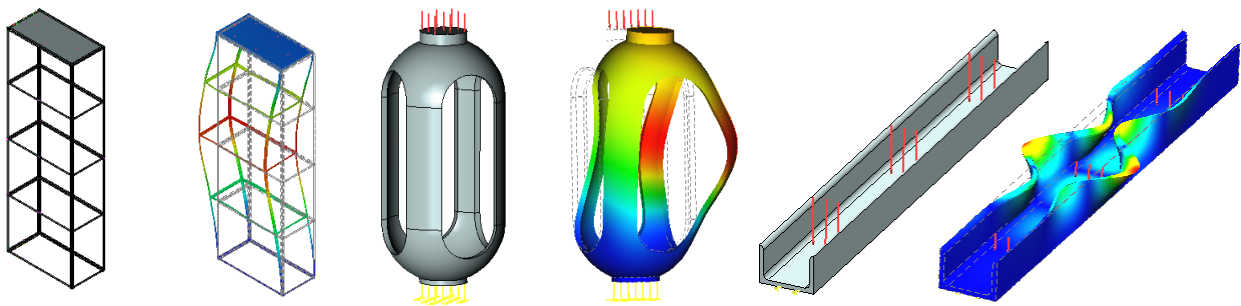
The buckling analysis module serves for conducting the so-called **initial buckling** structural study. The result of the study is a coefficient of the critical load, under which the structure may spring into a new equilibrium state, and the shape of the new equilibrium state corresponding to that load. 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.

Details of Buckling 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 Analysis. Therefore, in this chapter we will just mention what is special for the buckling analysis:

1. **Creating Study.** When creating a study, specify its type – «Buckling Analysis»

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.

3. **Running calculations.** Before running calculations, the user specifies computational algorithms and the number of buckling modes to be analyzed, in the study properties.

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.
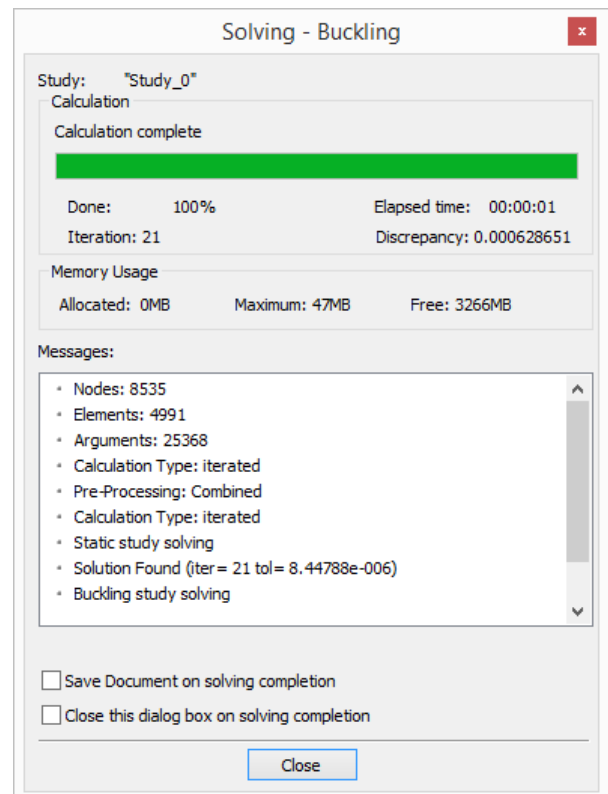
4. **Results.** The following are analysis results:

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.
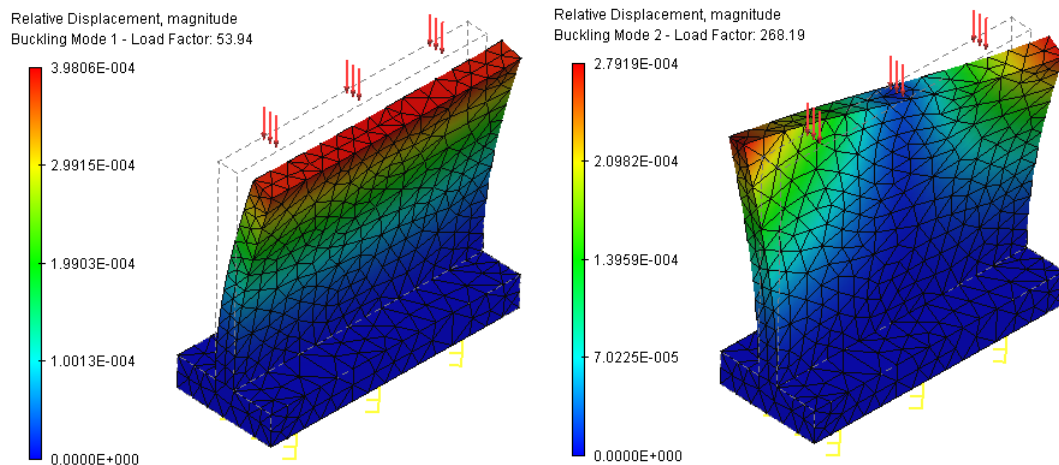
Buckling modes, corresponding to the first and second critical loads on the part

Algorithm for Buckling Analysis Based on Modeling

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.

Buckling Analysis Processor Settings

Upon initializing the «**Analysis|Solve**» command, the dialog of defining study properties appears by default, with several tabs to switch between. The user can change the default study properties. To access a study's properties, 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 properties are saved together with the document and are inherited upon copying a study. The main purpose of this study properties 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 **[General]** tab, you can define or modify the descriptive properties of the current study: the name, and the comment.

On the **[Solve]** tab, you can define processor properties for solving the equations.

The "**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: **When necessary**, **Never**, **Mandatory**.

Group "Finding state forms"

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 «**Calculate using linear element**». 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).

Parameters of temperature changes influence on the materials properties are specified on the **Thermoeffects** tab.
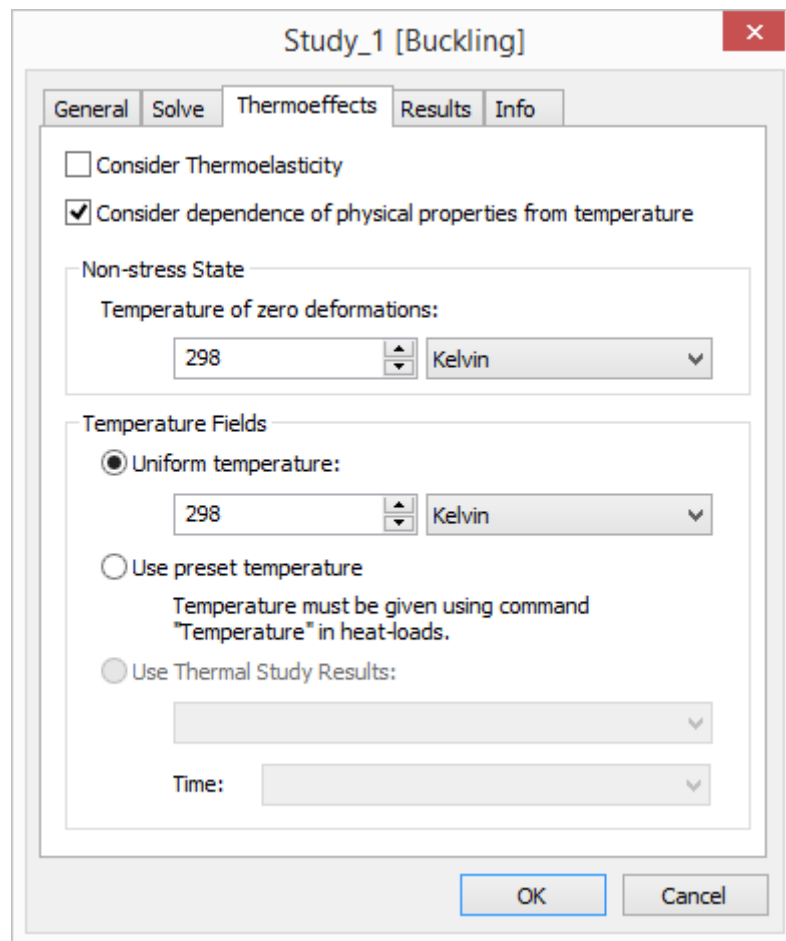
Consider Thermoelasticity. If the flag is enabled the influence of heat stress to the rigidity of construction and changes of critical load value will be considered. Otherwise, the condition of heat stress will not be considered.

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]** 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: $\sigma_m = \dfrac{\sigma_{max} + \sigma_{min}}{2}$, $\sigma_a = \dfrac{\sigma_{max} - \sigma_{min}}{2}$. 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:

$$R = \frac{\sigma_{min}}{\sigma_{max}}.$$

Fatigue Analysis

Stress cycle type	$R = \dfrac{\sigma_{min}}{\sigma_{max}}$	σ_{min}	σ_{max}	$\sigma_m = \dfrac{\sigma_{max} + \sigma_{min}}{2}$	$\sigma_a = \dfrac{\sigma_{max} - \sigma_{min}}{2}$	$2\sigma_a$
asymmetric	R_1	$R_1\sigma_{max}$	σ_{max}	$\dfrac{(1+R_1)\sigma_{max}}{2}$	$\dfrac{(1-R_1)\sigma_{max}}{2}$	$(1-R_1)\sigma_{max}$
pulsating	0	0	σ_{max}	$\dfrac{\sigma_{max}}{2}$	$\dfrac{\sigma_{max}}{2}$	σ_{max}
symmetric	-1	$-\sigma_{max}$	σ_{max}	0	σ_{max}	$2\sigma_{max}$

Asymmetric cycle (R= −0.2)

Symmetric cycle (R= −1)
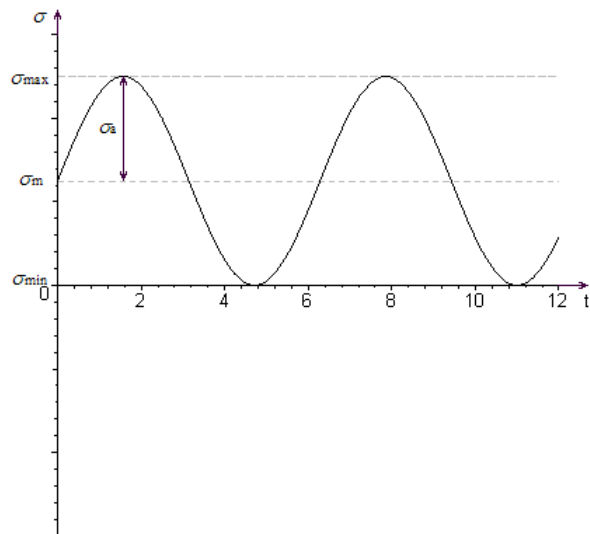
Pulsating cycle (R= 0)

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.
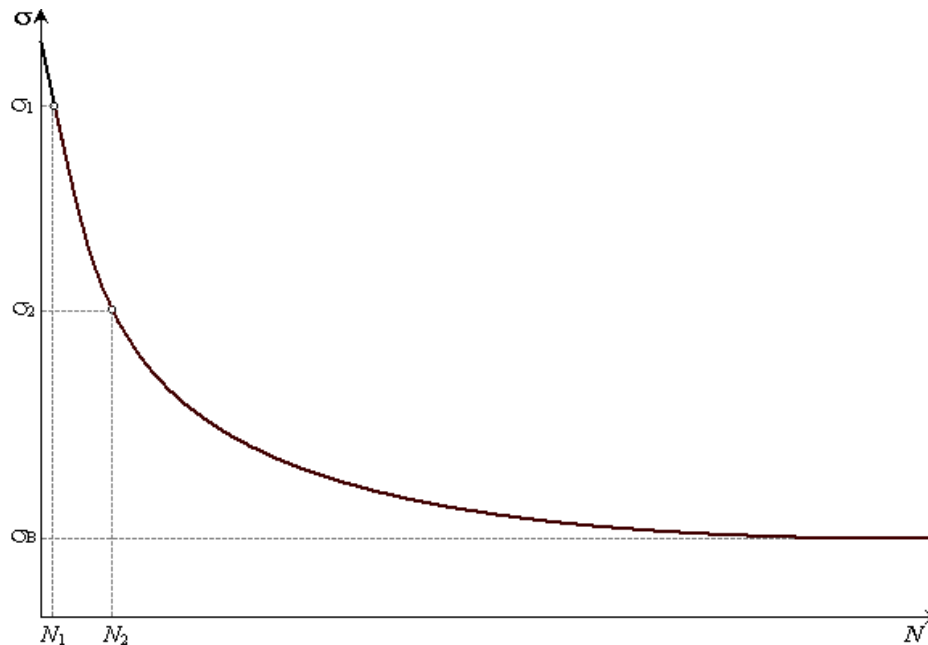
After a certain number of cycles the ordinates σ of the S-N curve practically remain constant. Because of this fact the number of cycles (during fatigue testing of the material) is bounded by a certain limit which is called the base number of cycles. If the material sustains the base number of cycles, then it is assumed that the stress in the material does not exceed the endurance limit σ_B.

For the same material we can construct the S-N curves for different stress cycles: symmetric (R=−1), pulsating (R=0), asymmetric. That is why the notion of the coefficient of S-N curve is introduced the value of which will coincide with the value of the asymmetry coefficient of the cycle for which the given curve is defined.

If, when carrying out the fatigue analysis, the cycle asymmetry coefficient defined by the user does not coincide with the coefficient of the S-N curve, selected for analysis, then when estimating the fatigue strength the stress adjustment is performed according to one of the three methods.

Stress Adjustment Methods

Let σ^* be the adjusted sign-alternating stress, σ_T – yield stress, σ_Π – ultimate strength. Then:

$$\sigma^* = \frac{\sigma_a}{1 - \left(\dfrac{\sigma_m}{\sigma_T}\right)}$$

Soderberg method:

$$\sigma^* = \frac{\sigma_a}{1 - \left(\dfrac{\sigma_m}{\sigma_\Pi}\right)}$$

Goodman method used for brittle materials:

$$\sigma^* = \frac{\sigma_a}{1 - \left(\dfrac{\sigma_m}{\sigma_\Pi}\right)^2}$$

Gerber method used for ductile materials:

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 $\sigma^* \leq \sigma_R$. Let us recall the general expressions for standard hypotheses of strength checking (by plasticity criterion) used for estimating the strength of structures:

Hypothesis of Tresca – St Venant (hypothesis of maximum shear stresses)

$$\sigma_T < (\sigma_1 - \sigma_3), \quad |\sigma_1| \geq |\sigma_2| \geq |\sigma_3|;$$

Hypothesis of Huber – Mises – Hencky (hypothesis of distortion energy)

$$\sigma_T < \sqrt{\frac{1}{2}\left((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2\right)};$$

Hypothesis of Gesta – Mohr (hypothesis of maximum principal stresses)

$$\sigma_T < \sigma_1, \quad |\sigma_1| \geq |\sigma_2| \geq |\sigma_3|,$$

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.

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 for the material from which the given workpiece will be made be specified.

The S-N curve can be selected from already existing curves or specified independently.

With the help of the command:

Icon	Ribbon
![icon]	Analysis → Conditions → Graph Template
Keyboard	**Textual Menu**
<3GT>	Analysis > Graph Template

it is possible to invoke 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 or a smooth curve) is determined when creating the graph.
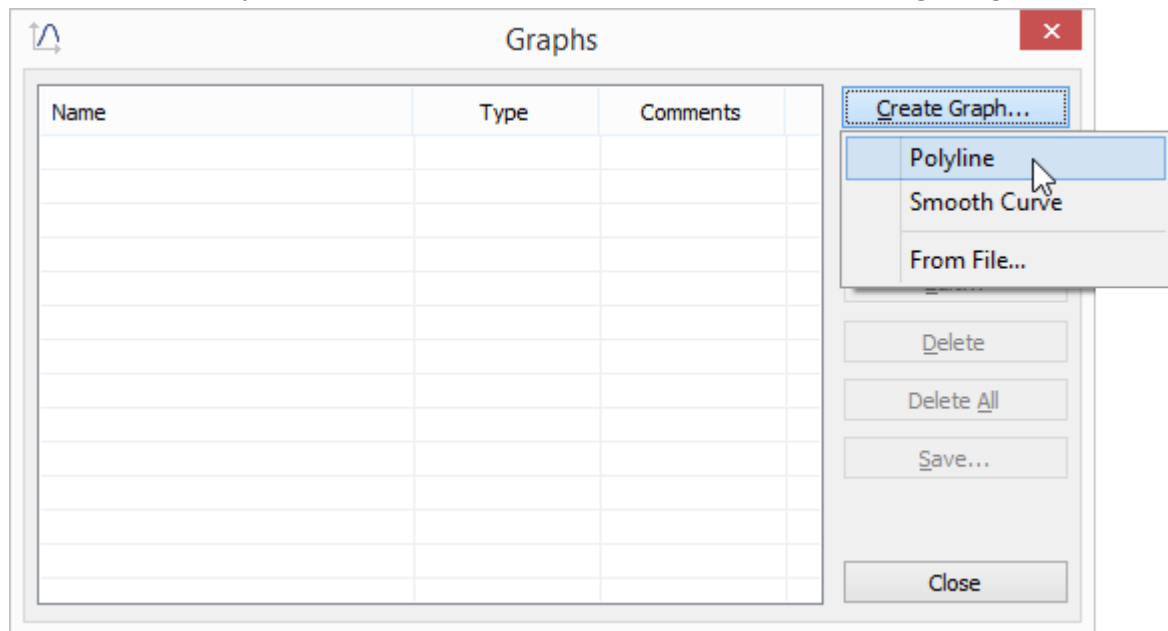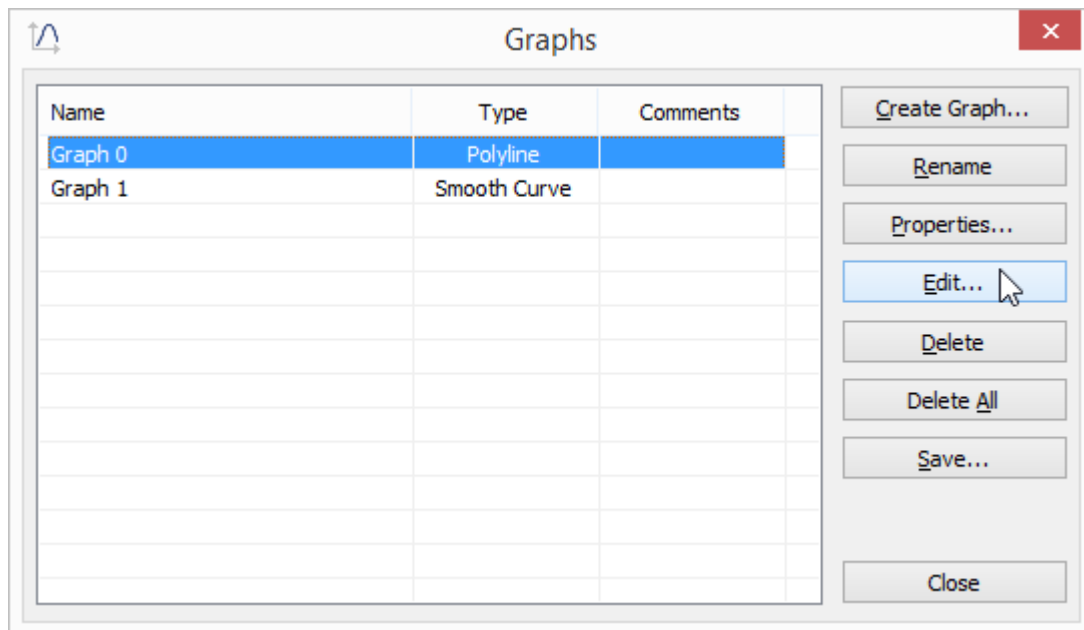
After the curve has been created (its type and name are defined), it is required to specify the nodes of the curve and the values of the function at these nodes.

Polyline
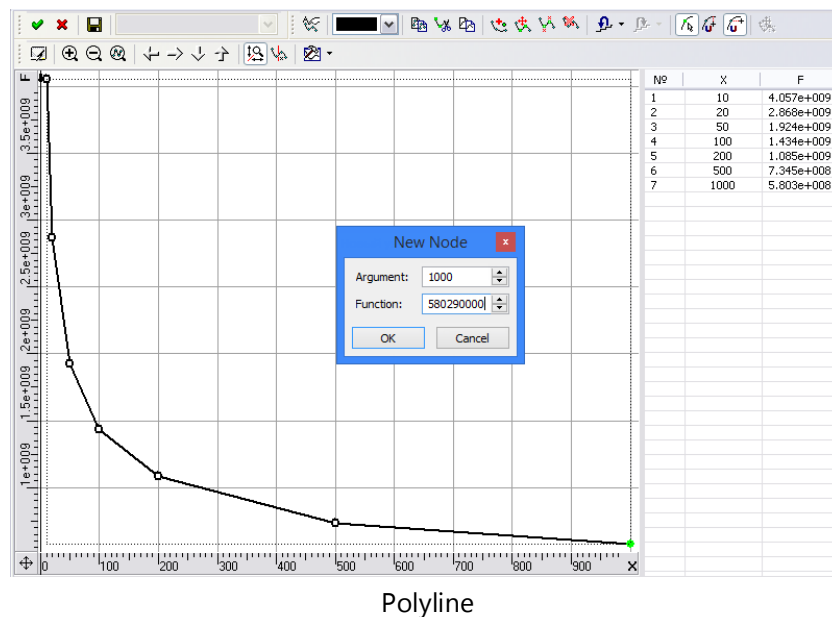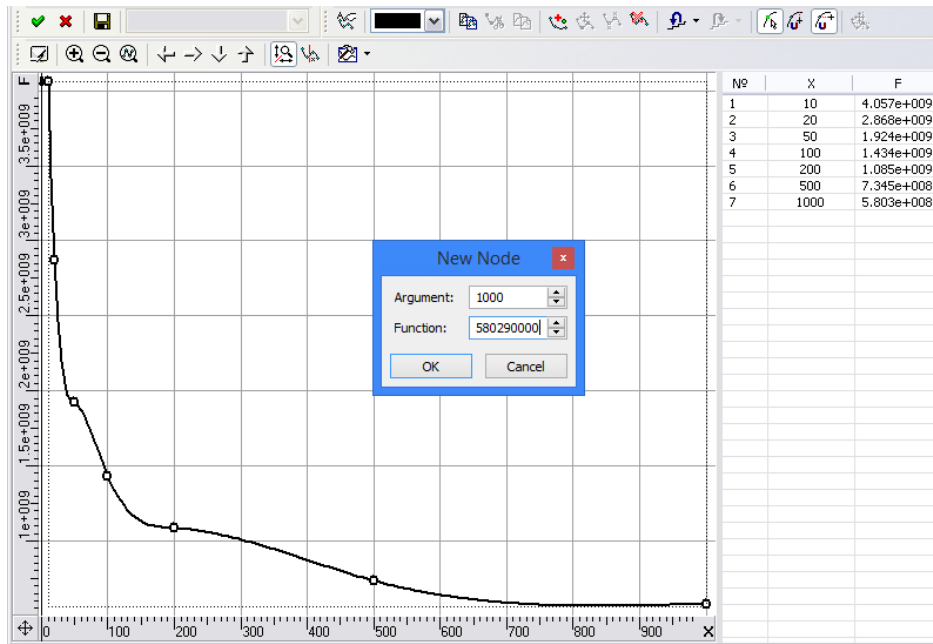
Smooth curve

The final appearance of the given S-N curve can be the following:
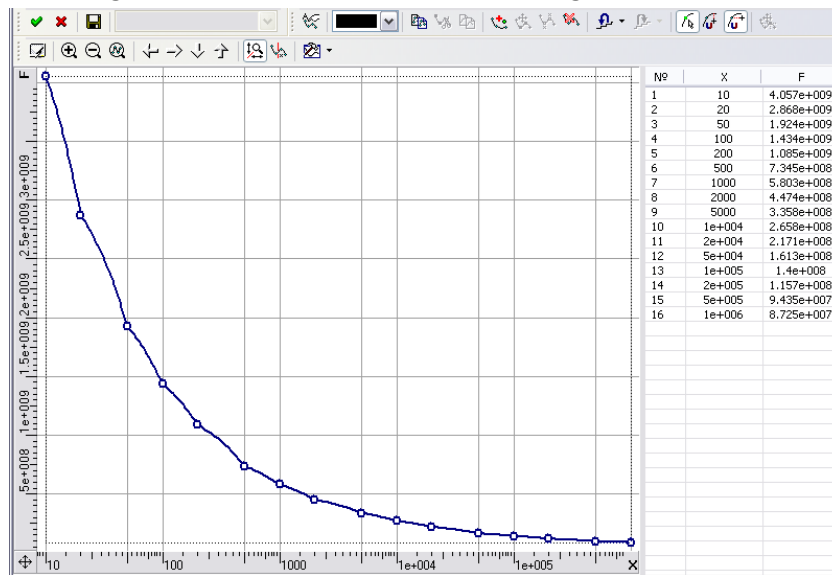
The graph of this curve can be saved into the external file for it to be used again later on in other fatigue calculations.

Next it is required to attach the S-N curve to the material of the study. To do so, we select the body, invoke the «Body's properties» dialog (the «Properties» command) with the help of the context menu, go to the dialog window «Materials» (with the help of the «Materials» button). In the «Materials» dialog from the «Current model» list we select the desired material and press the «More» button. In the «Material's properties» dialog that appears we select the S-N curve in the «Fatigue rule» parameter.
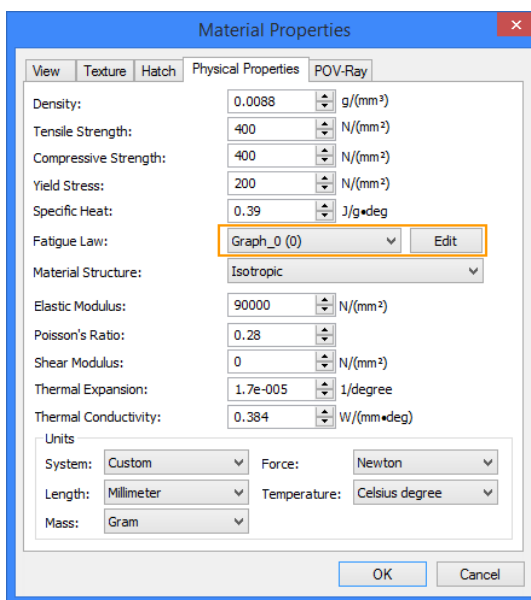
If 3D model is an assembly, i.e., it consists of 3D fragments, then in order to attach the S-N curve to the material of the fragment, we select 3D fragment in the tasks tree, then, with the help of the context menu, invoke the « 3D fragment Parameters» dialog (the «Properties» command), enable the «Current operation» option, go to the dialog window «Materials» (with the help of the [...] button).
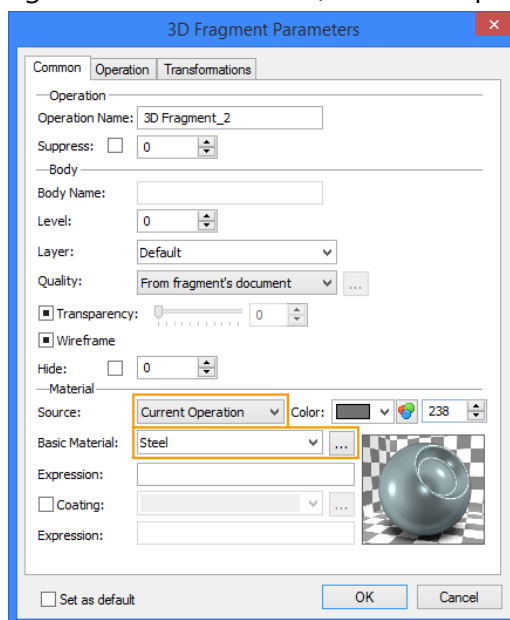
In the «Materials» dialog from the «Current model» list we select the desired material and press the «More» button. In the «Material's properties» dialog that appears we select the S-N curve in the «Fatigue rule» parameter.

Next, we create the Fatigue Analysis study. 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 properties 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:

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 loadings). Multi-event analysis allows us to estimate the action of several forces with different parameters of the cyclic loading (different number of cycles or non-coinciding types of the cyclic loading change). Each event is added to the «Events» folder in the tasks tree.
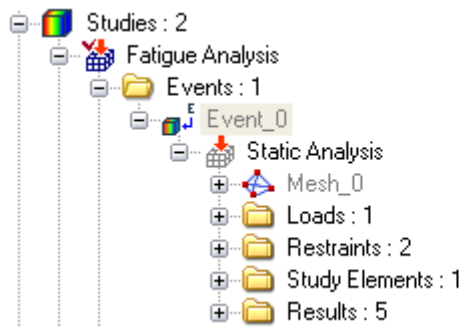
> It is important to note that to carry out a multi-event fatigue analysis, it is required that the finite element meshes of all statics studies included in the fatigue analysis coincide and the same body of each statics study be made of the same material.
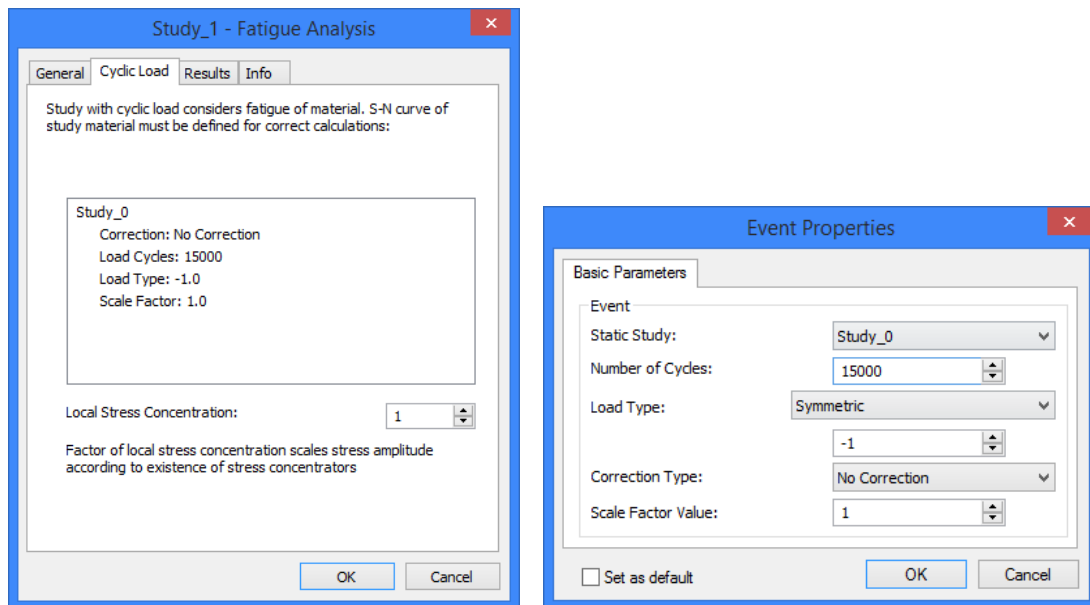
Then it is required to add the completed static analysis study and specify parameters of the cyclic loading: number of cycles, loading type, stress adjustment method (this is specified only in case if the cycle asymmetry coefficient R does not coincide with the coefficient of the given S-N curve), scale effect coefficient (stress scaling coefficient). Also in the «Parameters of study (fatigue analysis)» dialog on the «Cyclic recurrence» tab it is possible to specify the local stress intensity coefficient (by default this coefficient is equal to 1).

Fatigue Analysis Results

After completing the analysis, the new folder «Results» will appear in the tasks tree. The list of displayed results can be specified using the «Results» tab of the «Parameters of study (fatigue analysis)» dialog.
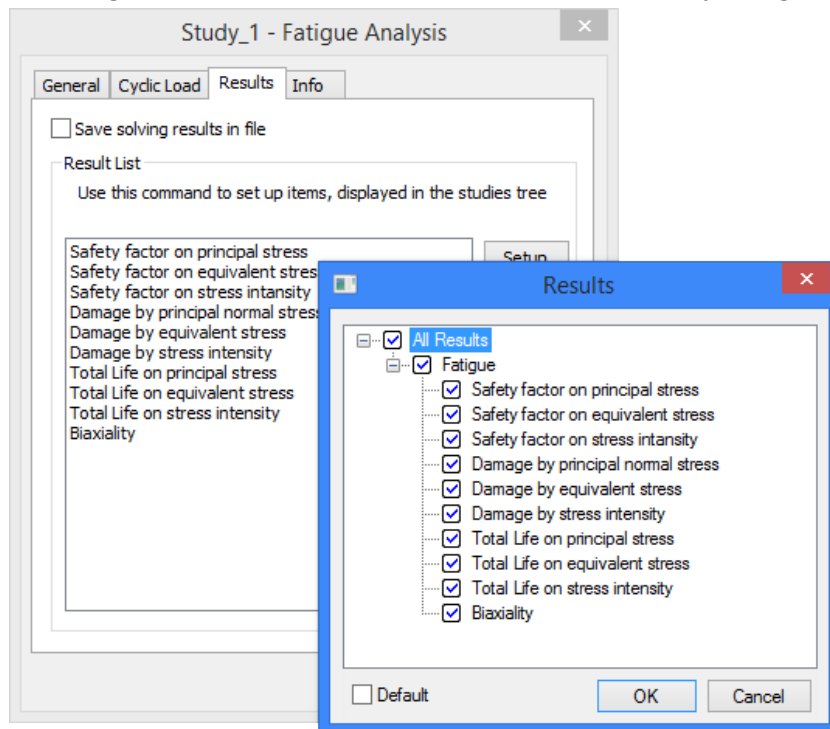
From the results of the fatigue analysis 10 results in total are accessible to the user, which can be divided into 4 groups.

The «Exhausted resource» group includes the following results:

- 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 n_1 cycles of sign-alternating stress S_1, n_2 cycles of sign-alternating stress S_2, n_3 cycles of sign-alternating stress S_3 ,..., n_k cycles of sign-alternating stress S_k, 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.

The «Life span» group (this type of the results is available only for a single-event analysis) includes the following results:

- Total life on principal stresses.
- Total life on equivalent stresses.
- Total life on stress intensity.

The «Life span» result shows the minimum number of cycles N_{min} required to induce fatigue damage.

The «Safety coefficient» group (this type of the results is available only for a single-event analysis) includes the following results:

- 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

$$K = \frac{\sigma_R}{|\sigma^*|}$$

calculated from the stresses obtained in the static analysis study σ: .

The stresses σ are evaluated according to the corresponding plasticity conditions:

hypothesis of distortion energy:
$$\sigma = \sqrt{\frac{1}{2}\left(\left(\sigma_1 - \sigma_2\right)^2 + \left(\sigma_2 - \sigma_3\right)^2 + \left(\sigma_1 - \sigma_3\right)^2\right)}$$
;

hypothesis of maximum shear stresses: $\sigma = \sigma_1 - \sigma_3$, $|\sigma_1| \geq |\sigma_2| \geq |\sigma_3|$;

- hypothesis of maximum principal stresses $\sigma = \sigma_1$.

The «Biaxiality» group (this type of the results is available only for a single-event analysis) includes the following results:

Biaxiality – the ratio of the smallest principal sign-alternating stress (different from 0) to the largest principal sign-alternating stress:

$$b = \frac{\sigma_3}{\sigma_1}, \quad |\sigma_1| \geq |\sigma_2| \geq |\sigma_3|.$$

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.

Workpiece Fatigue Strength Analyses Examples

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.

Single-event fatigue analysis

Let the shaft shown on the picture be subjected to the «Twisting moment» 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.
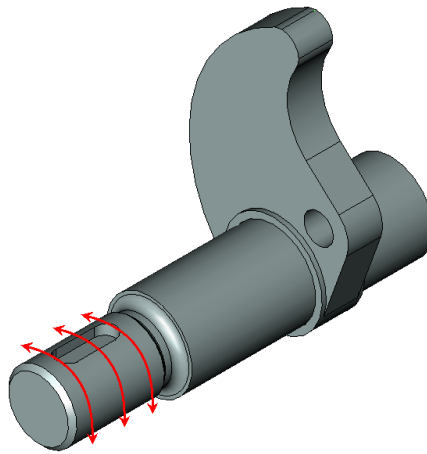
We first assume that the load is static (i.e., does not change with time) and perform the static strength analysis for the workpiece subjected to such a load.

Static Analysis
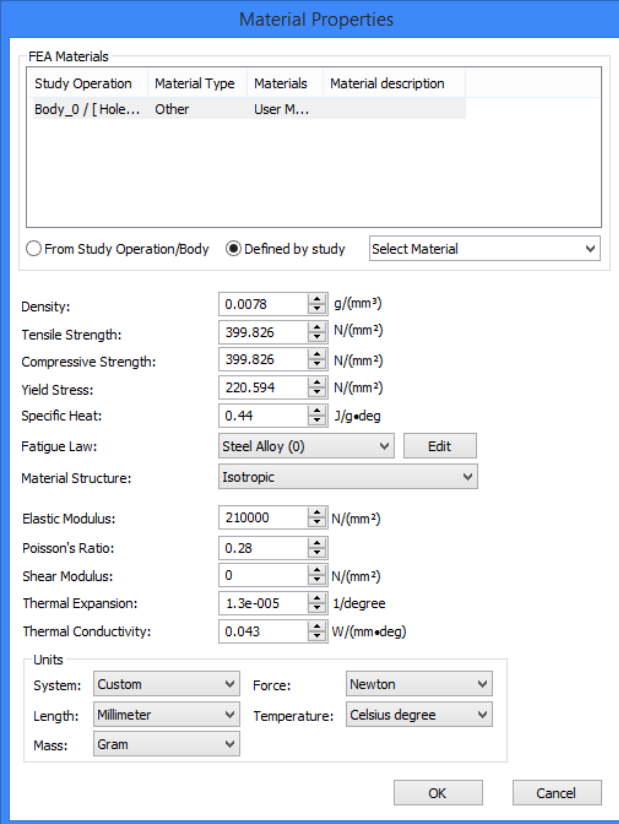
To solve Static Analysis study it is required to:

1. construct finite-element mesh;
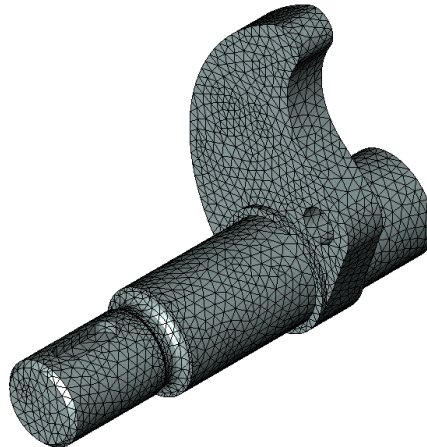
2. specify material of the study;

Later on when executing the fatigue strength analysis, it will be required to define the S-N curve for the material of the study. That is why the S-N curve for the material must be specified.

3. apply loads and specify constraints;

Magnitude of «Torque» is 50 N-m
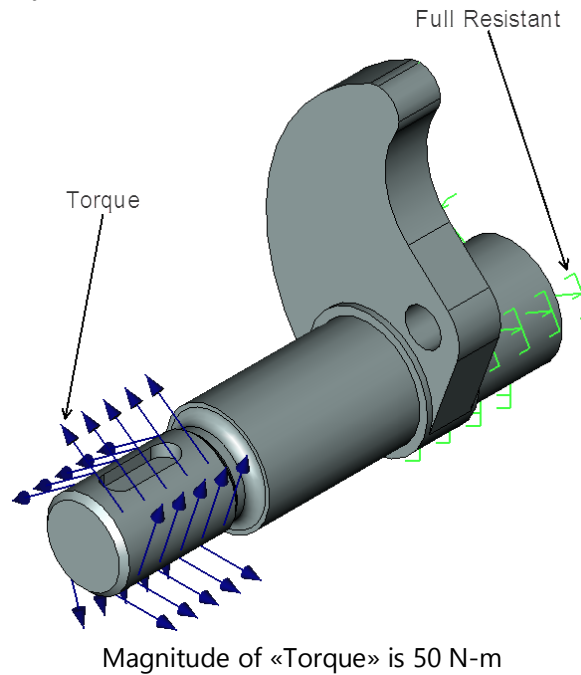
After performing the analysis, the obtained results must be analyzed. In particular we are interested in the safety coefficient. Let us consider, for example, «Safety coefficient by equivalent stresses».
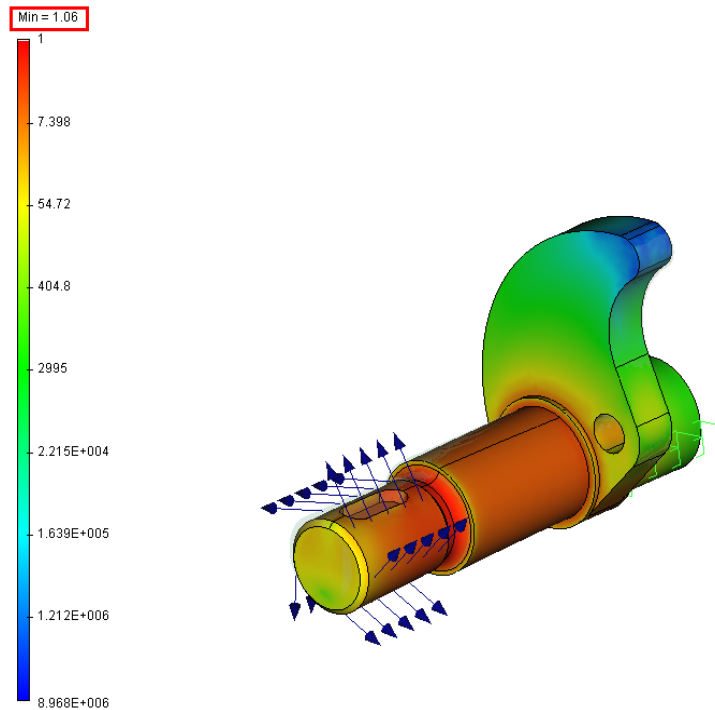
The minimum value of the safety coefficient is greater than 1, hence the workpiece will not be damaged for the static load. In the case it makes sense to consider the action of the cyclically varying loads applied to this workpiece.

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 «Twisting moment». 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 effect coefficient» (see below).

Let us create the Fatigue analysis study (**Analysis > New Problem > Finite-element analysis > Study**).

Let us create the event (**Analysis > Event**).

On the «Event's properties» tab we select the completed static study and specify parameters of the cyclic loading: number of cycles, loading type, stress adjustment method (specified 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 on the «Event Parameters» tab the local stress intensity factor can be specified (by default it is equal to 1).
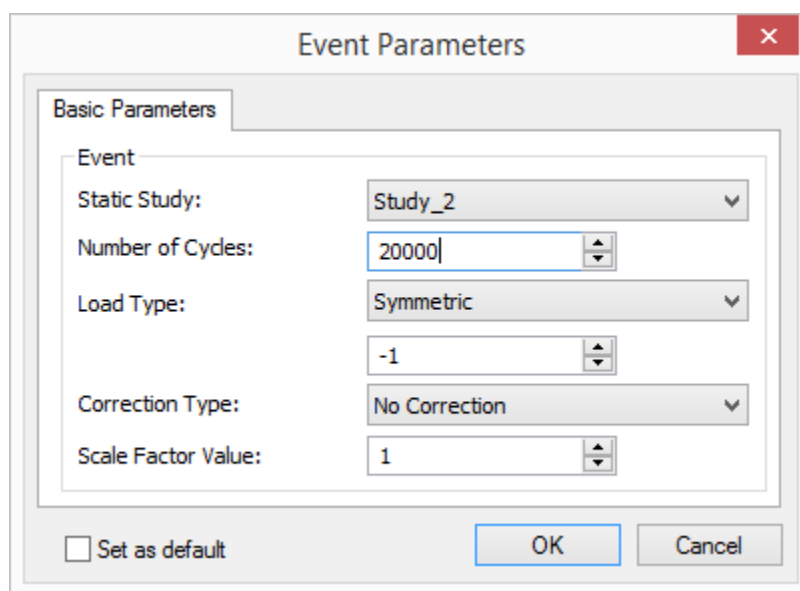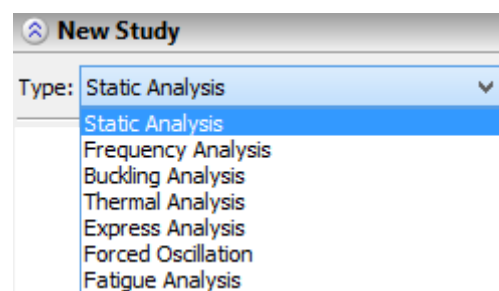
Before starting the analysis of the study, you can view the information about specified parameters of the study:
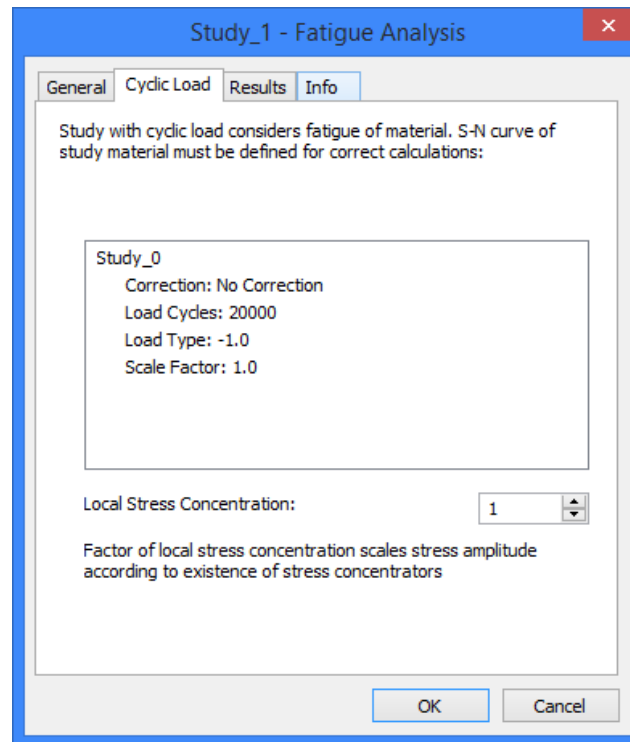
Results of Analysis

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.
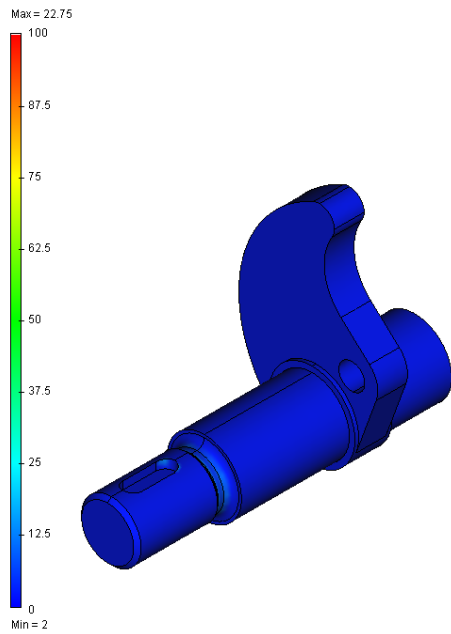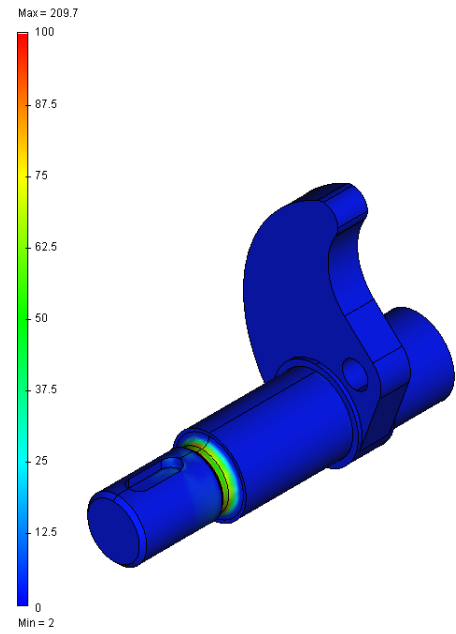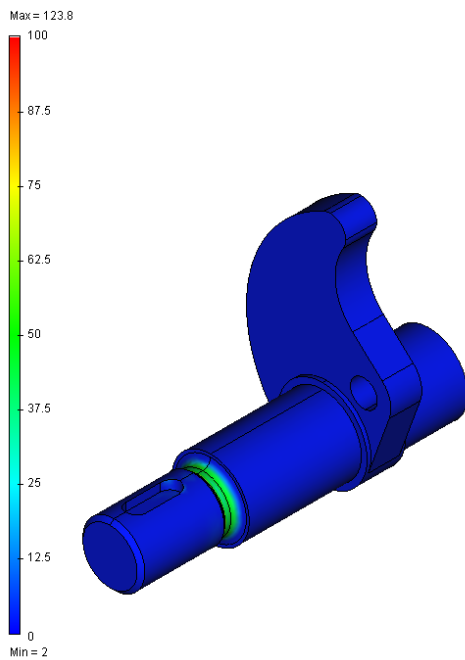
Damage by principal normal stress

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.
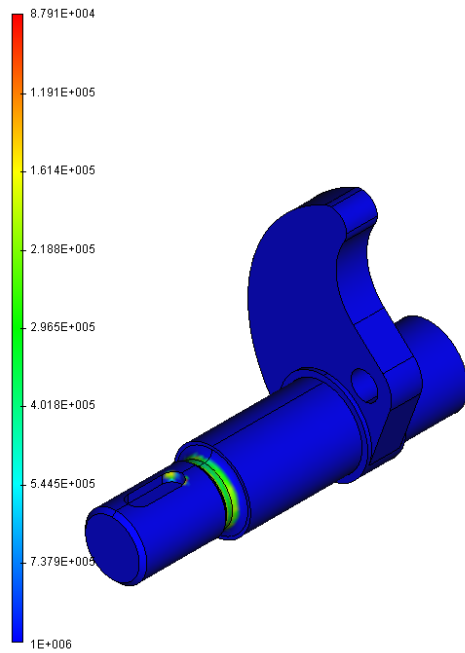
Total life on principal stresses
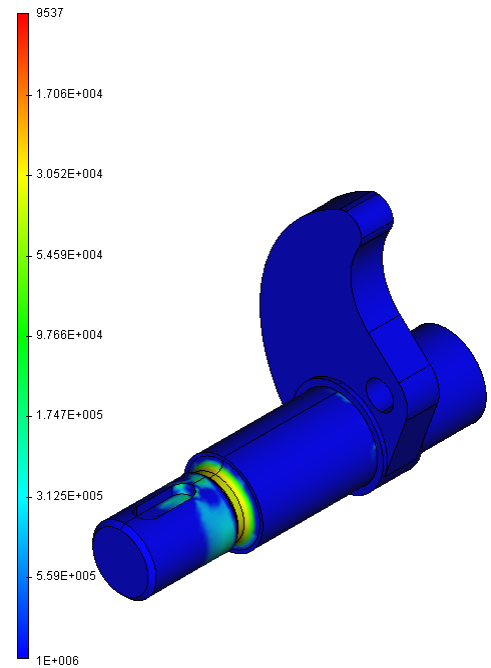
Total life on stress intensity.
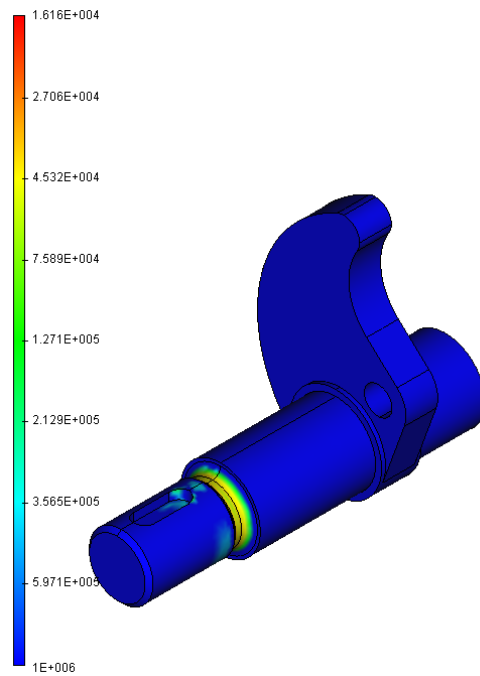
Total life on equivalent stresses

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.
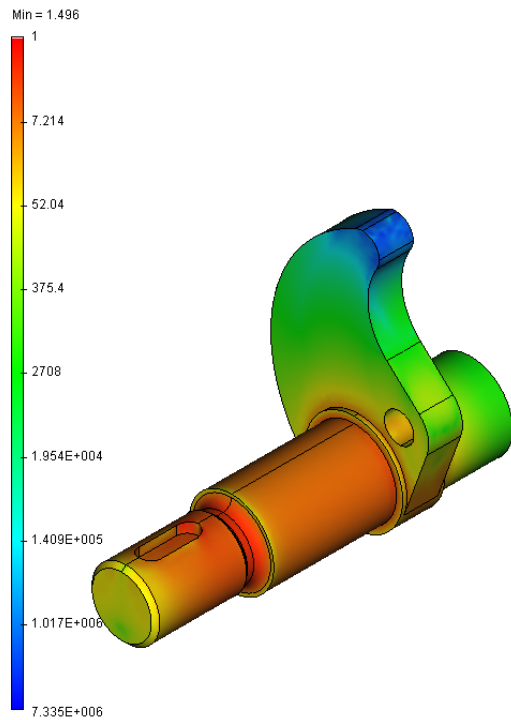
Safety factor on principal stresses
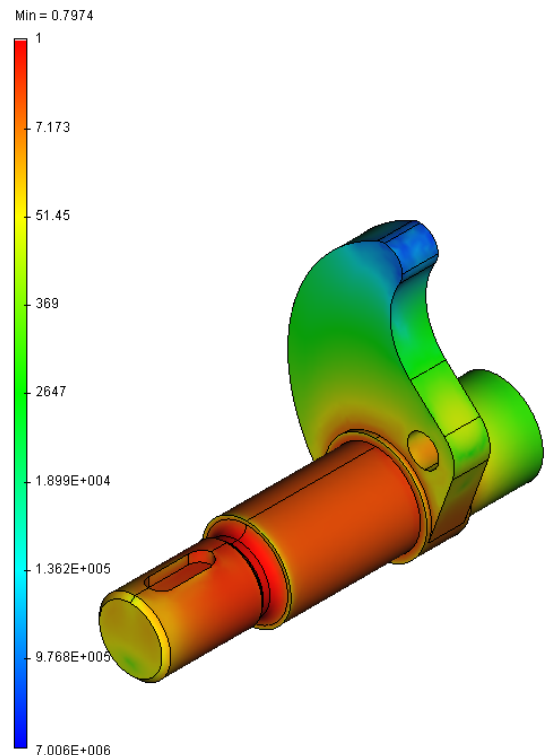
Safety factor on stress intensity
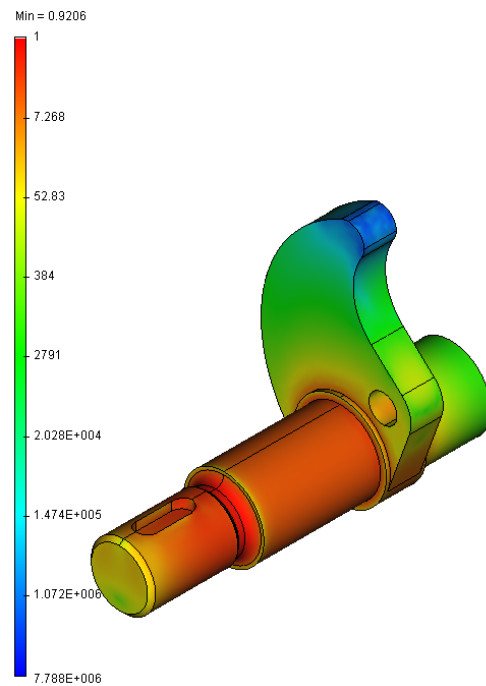
Safety factor on equivalent stresses

4. Biaxiality characterizes inequality of amplitudes of the principal stresses.
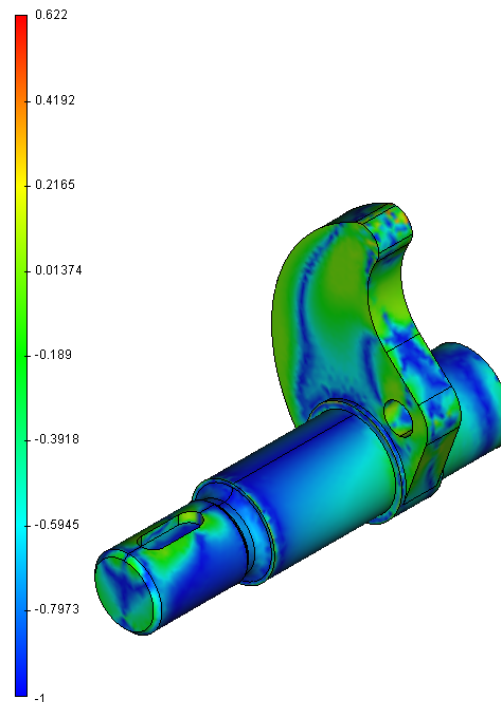
Multi-event fatigue analysis

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 «Twisting moment», and in addition a pulsating load «Force» is applied.
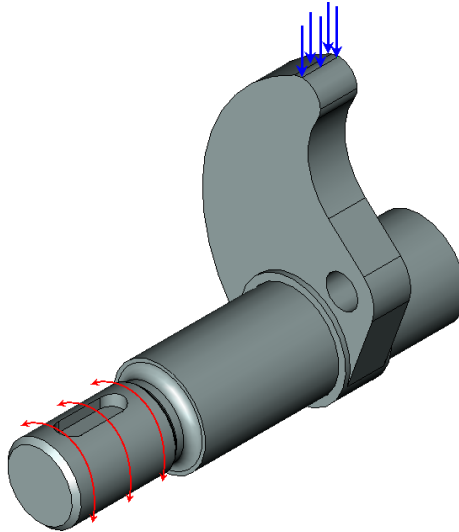
By the pulsating load «Force» in the given study we understand the maximum value of the load applied to the cam for one turn of the shaft.
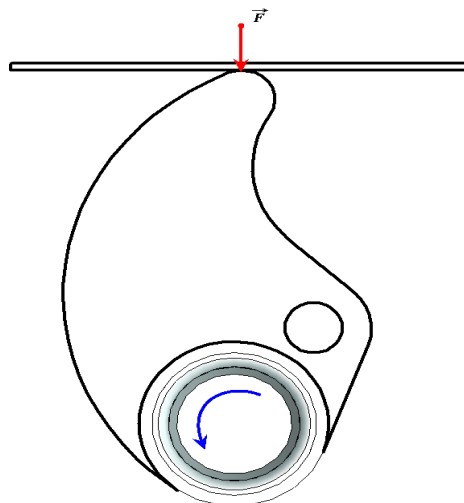
Both loads («Twisting moment» and «Force») are independent of each other. That is why the fatigue analysis will be multi-event.

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 «Twisting moment» 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 study with the «Twisting moment» 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:
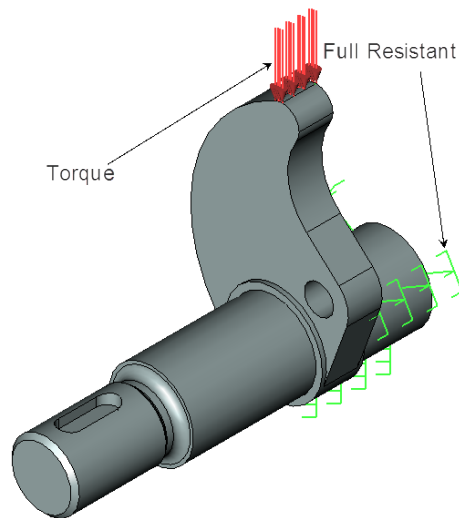
Magnitude of the «Force» load is equal to 3000 N

After the analysis has been completed, it is required to analyze the obtained results. Let us consider «Safety coefficient by equivalent stresses».
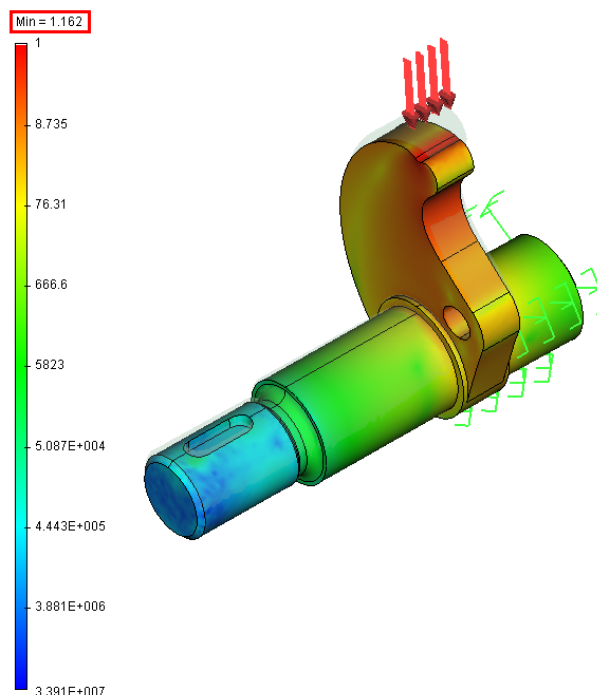
The minimum value of the safety coefficient is greater than 1, and hence, upon static loading the workpiece will not break down. Then it is meaningful to consider the action of the cyclically varying loads applied to this workpiece.

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: «Twisting moment» 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 «Twisting moment» load and for the «Force» load.

For the «Twisting moment» loading, we specify the same parameters as for the single-event analysis.

For the «Force» loading, on the «Event's properties» tab we select the completed static analysis study 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).
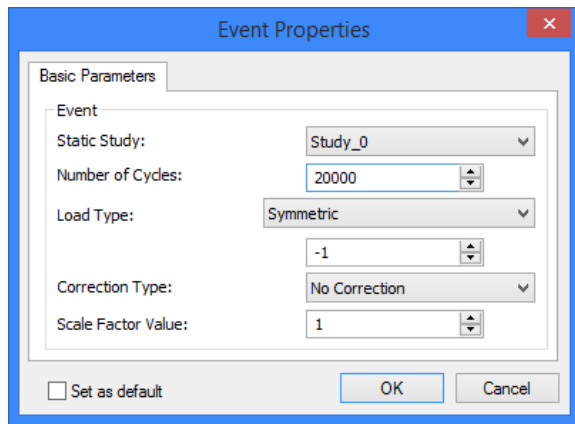
Parameters of the cyclic loading «Twisting moment»
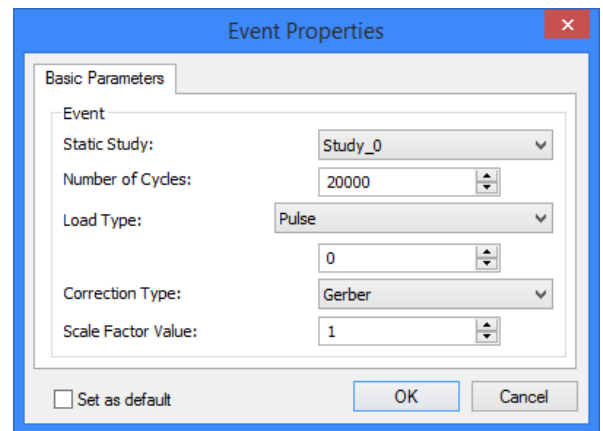
Parameters of the cyclic loading «Force»

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.
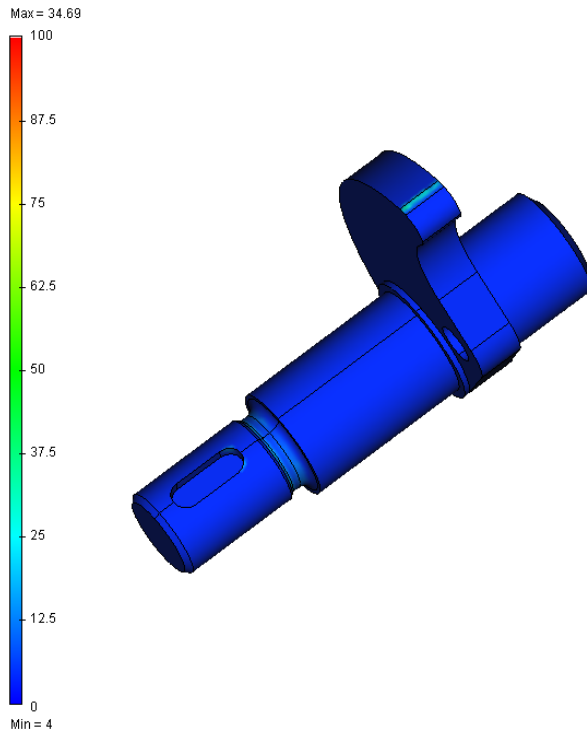
Damage by principal normal stress
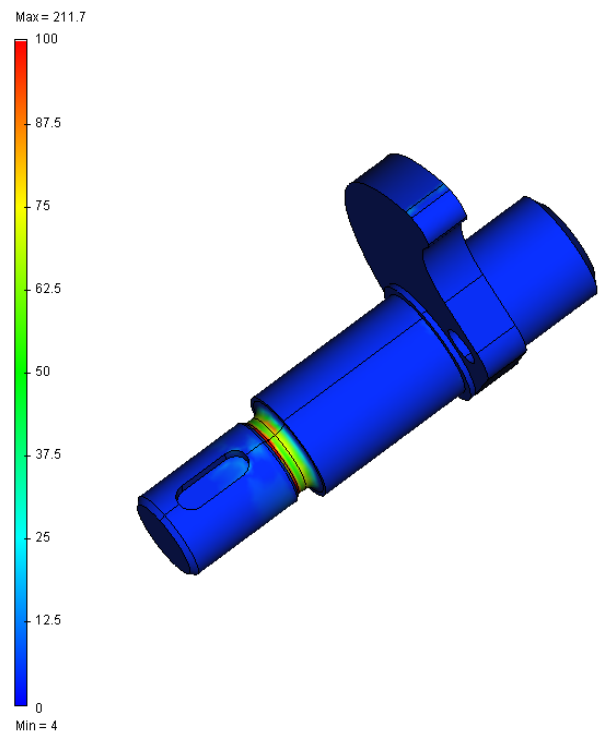
Damage by stress intensity
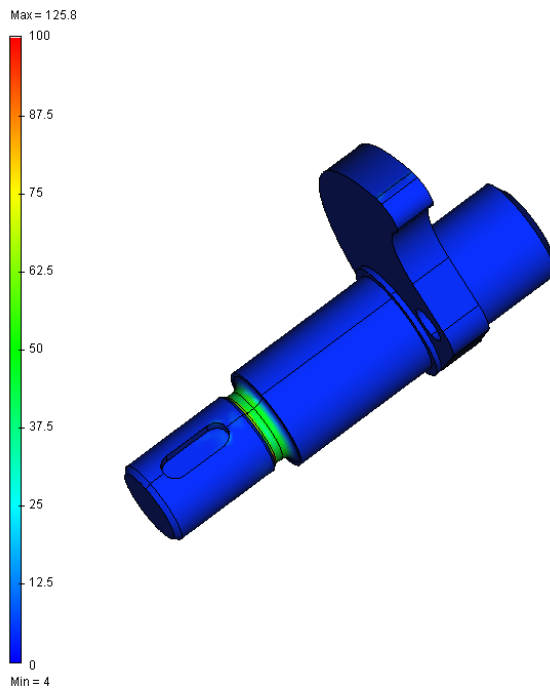
Damage by equivalent stress

Examples of Single-event Fatigue Analysis Results

1. «Damage»

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.
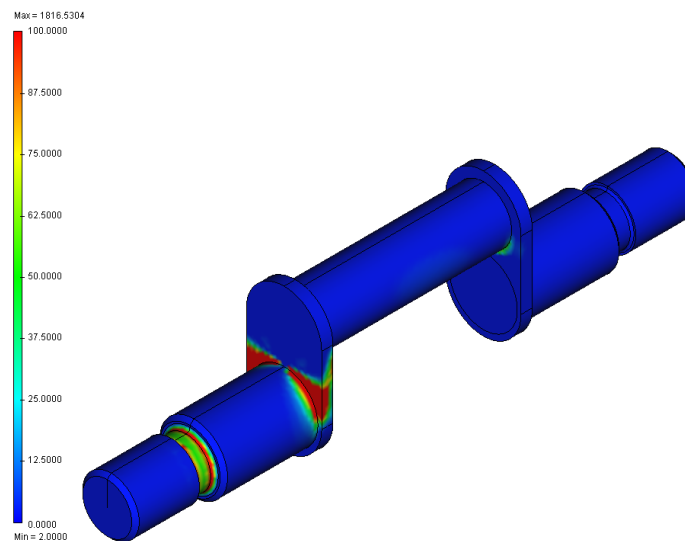
Damage by principal normal stress
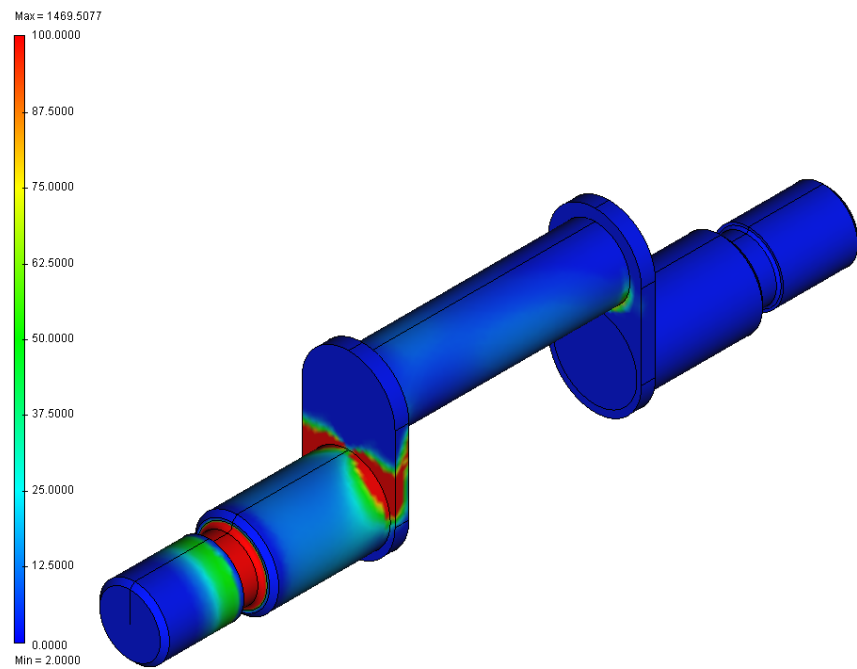
Damage by stress intensity
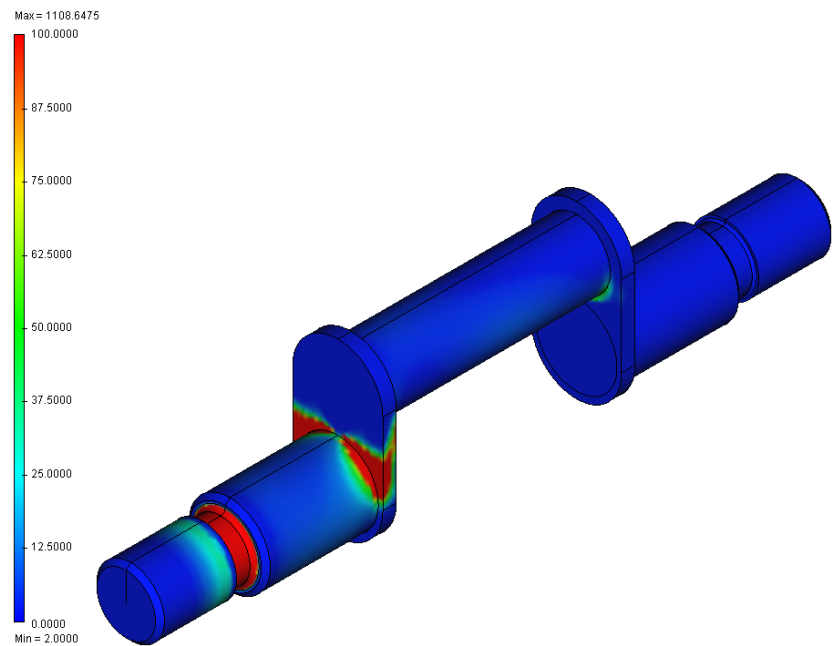
Damage by equivalent stress

2. «Total life»

Zones shown with red color have the smallest total life and with blue color – the largest.
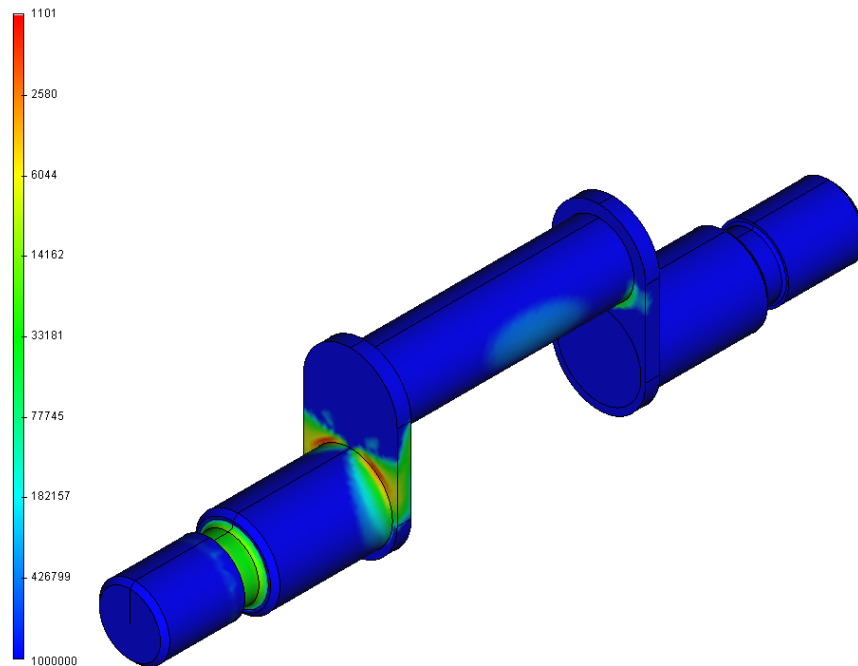
Total life on principal stresses
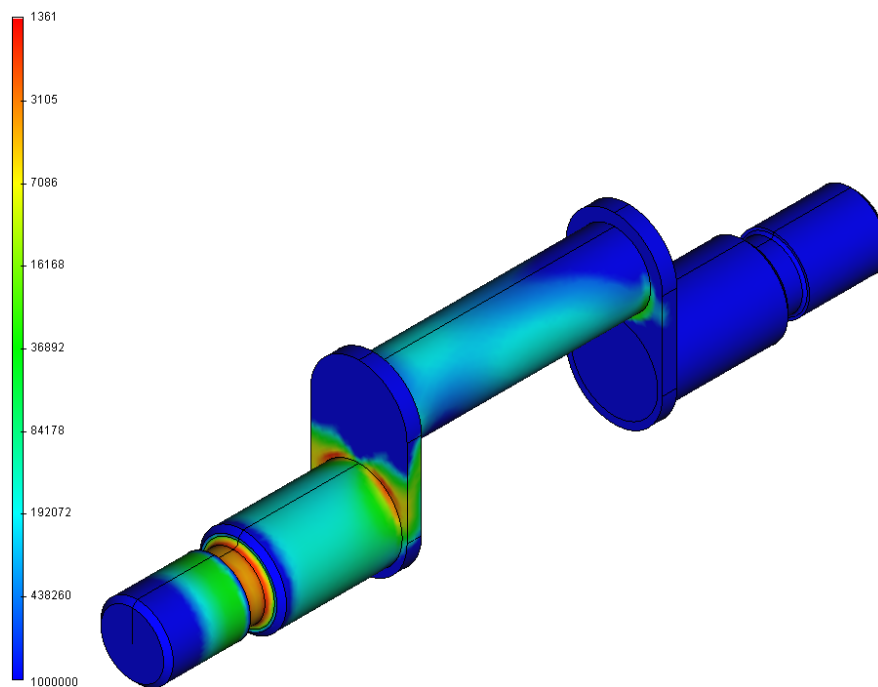
Total life on stress intensity
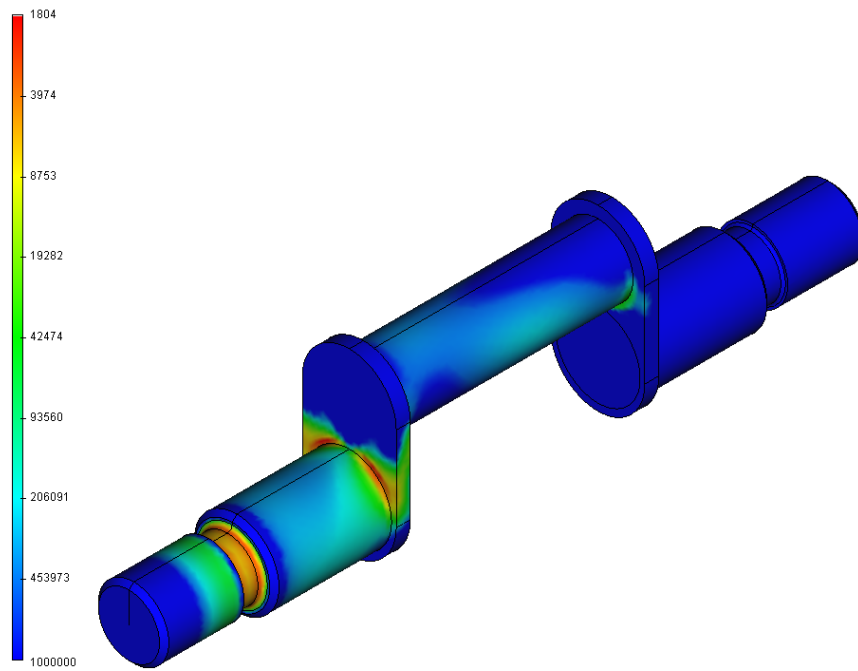
Total life on equivalent stresses

3. Safety factor. It shows the fatigue strength safety coefficient for the specified cyclic loading.
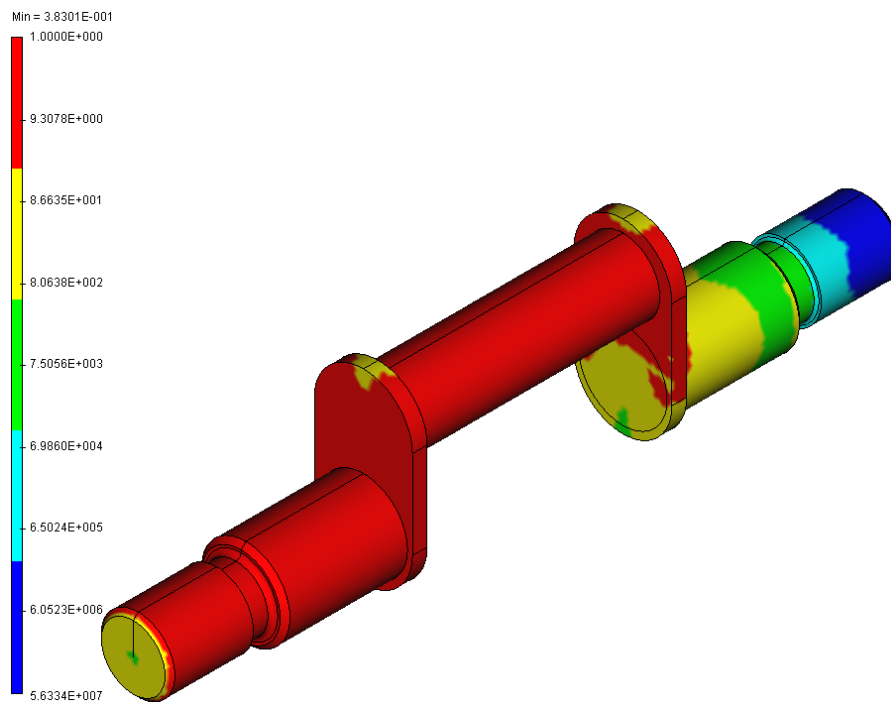
Safety factor on principal stresses

Min = 4.0787E-001

1.0000E+000

8.8588E+000

7.8477E+001

6.9521E+002

6.1587E+003

5.4559E+004

4.8332E+005

4.2816E+006

3.7930E+007

Safety factor on stress intensity

Min = 4.5868E-001

1.0000E+000

8.9669E+000

8.0406E+001

7.2099E+002

6.4651E+003

5.7972E+004

5.1983E+005

4.6612E+006

4.1797E+007
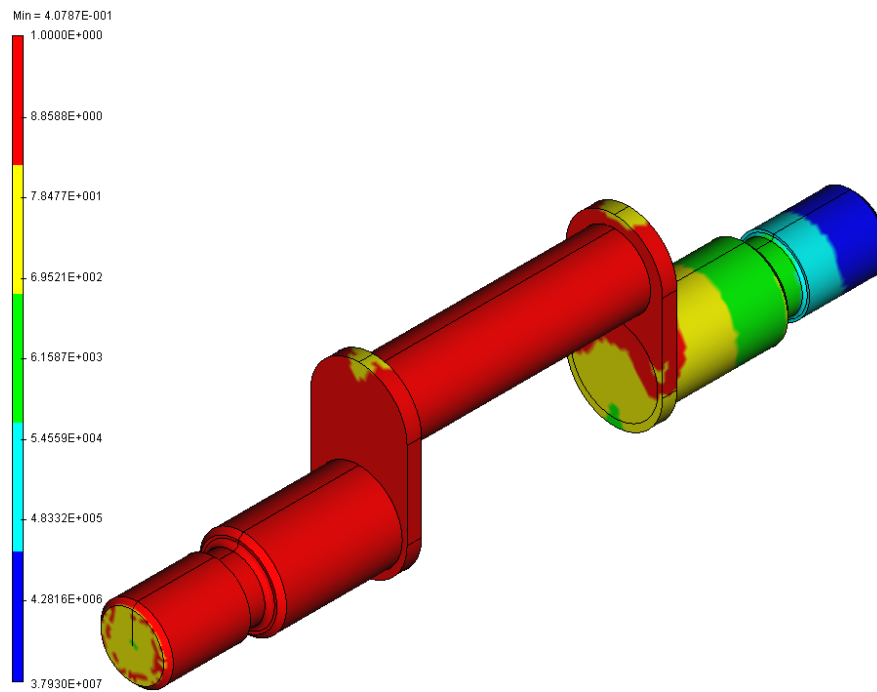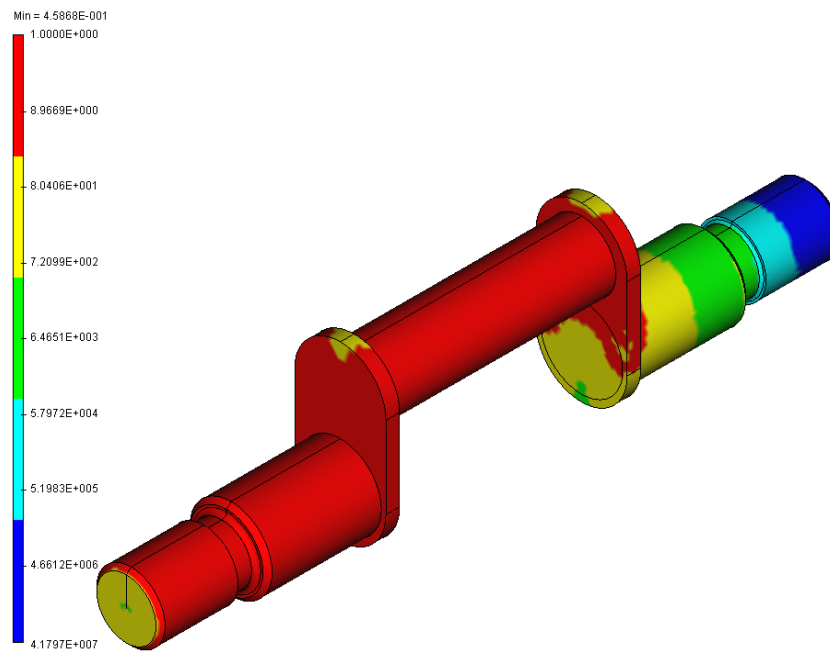
Safety factor on equivalent stresses

4. **Biaxiality.** It characterizes inequality of amplitudes of the principal stresses.
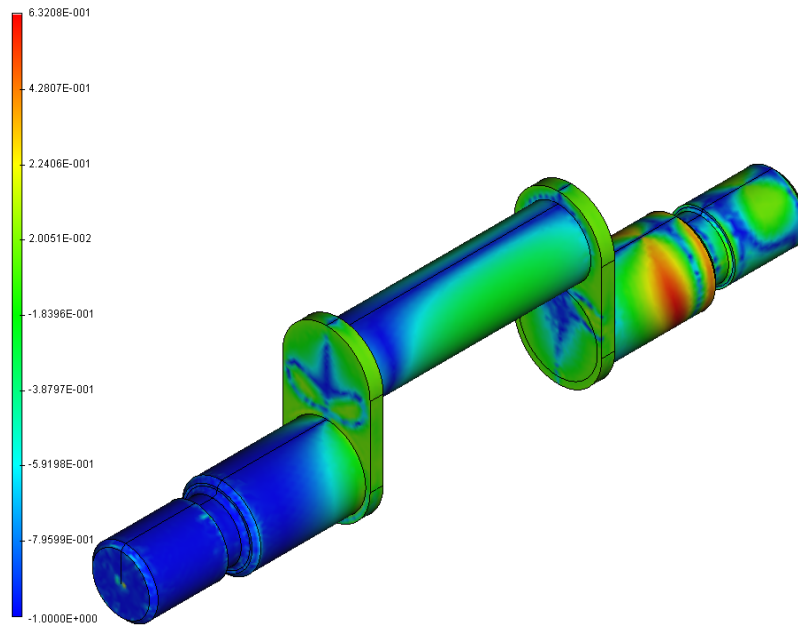
Examples of Multi-event Fatigue Analysis Results

The «Damage» result

Red-colored zones on the model correspond to the segments with the damage of 100% and signify insufficient stability to the action of cyclic loadings in these parts of the structure.
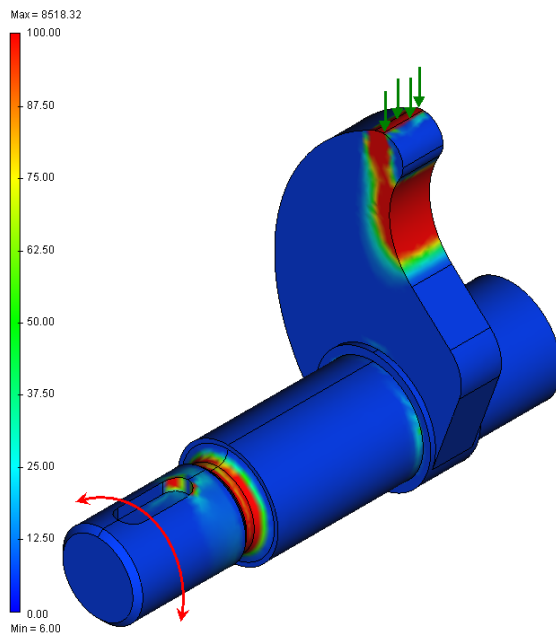
Damage by principal normal stress
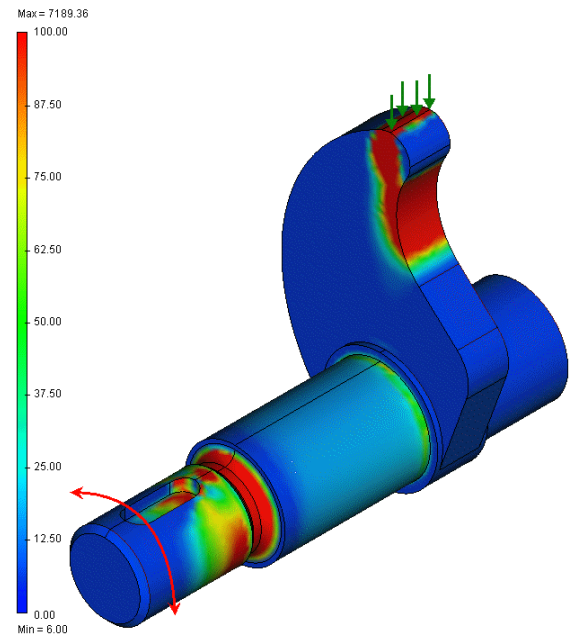
Damage by stress intensity
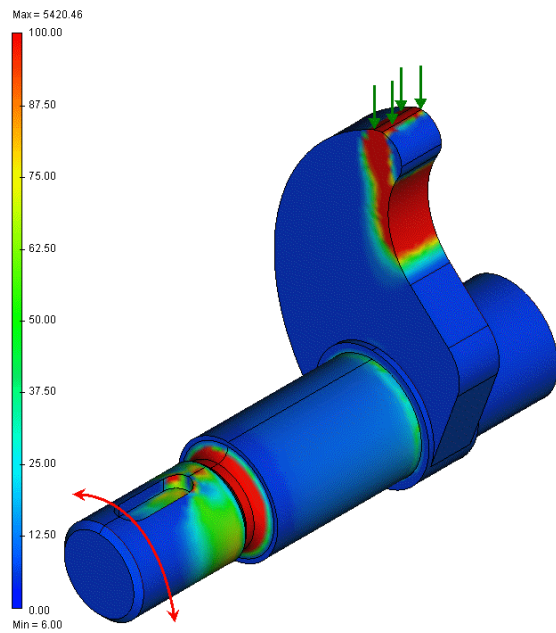
Damage by equivalent stress

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 3D" 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.

Express Analysis Limitations

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".

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 task 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:

$$f_i \notin \left[0.7 f_{min}^{\text{ex}} ; 1.3 f_{max}^{\text{ex}} \right]$$

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;

$f_{min}^{\text{ex}} ; f_{max}^{\text{ex}}$ – the lowest and the highest frequencies of the known range of external exciting vibration.
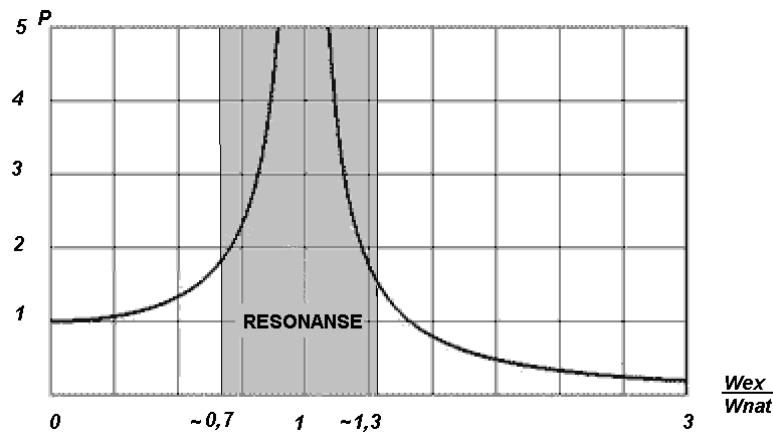
Variation in the amplitude magnification factor with respect to the natural frequency to the external exciting frequency ratio in a system with an insufficient damping

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.

Mode Shape 1 - Frequency: 104.6 Hz

Mode Shape 2 - Frequency: 151.7 Hz
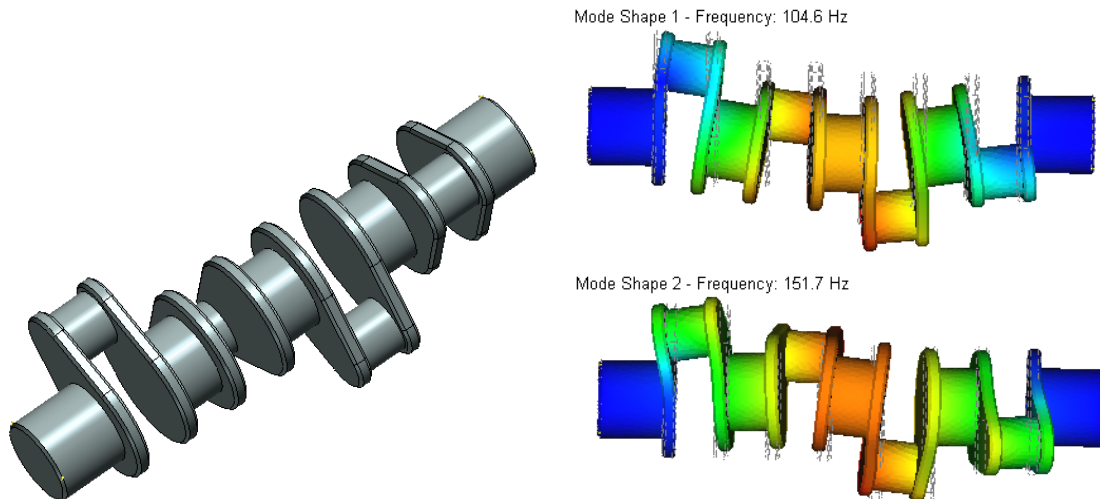
Thus, by calculating resonant frequencies at the design stage using the frequency analysis module and optimizing the part's mass-rigidity properties, the user can raise reliability of the structure being developed from the viewpoint of its vibro-stability and vibrational strength.

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:

1. **Creating Study.** When creating a study, specify the study type - "Frequency analysis" in the command's properties window.

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.

3. **Solving**. Before running calculations, the user should specify the number of natural frequencies and, if necessary, elaborate on the solution algorithm.

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 analyzing 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 properties.
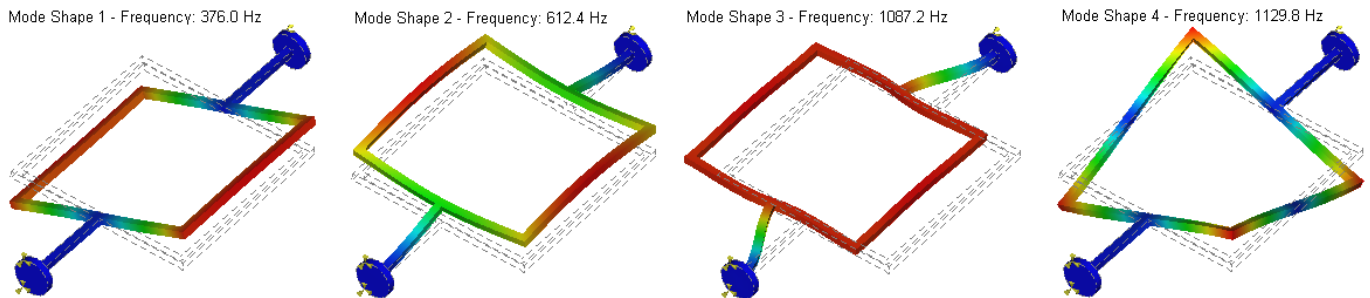
Consider also the convenience of animation for analyzing the pattern of the structure's motion at a certain frequency. Remember to enable the **Animation** option in the properties of the calculation results window (accessible by 🖵🖵 in the Postprocessor window) in order to have an animation, and specify the desired animation parameters.

Frequency Analysis Processor Settings

On the **[General]** tab, you can define or edit the descriptive attributes of the current study, as the name or a comment.

The **[Solve]** tab serves for defining processor properties for solving equations.

The **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: **When necessary**, **Never**, **Mandatory**.

In the group «**Frequency spectrum**» it is required to:

- 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 force 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.

In the group **Finite-Element Method** the user can set the **Calculate using linear element** 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. This 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.

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.

The **[Results]** 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.
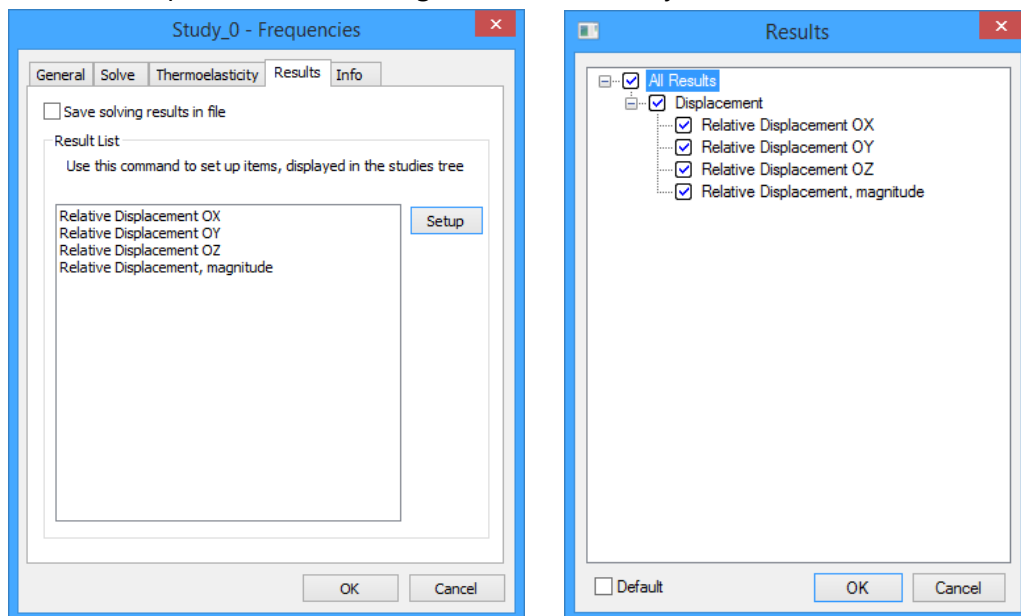
FORCED OSCILLATION

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:

$$[M]\{\ddot{U}\} + [K]\{U\} = \{F_0 \cos(\omega t + \varphi)\}_,$$

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:

$$u_i = e^{-nt}(C_1 \cos \omega_\partial t + C_2 \sin \omega_\partial t) + C_3 \cos \omega_s t + C_4 \sin \omega_s t$$
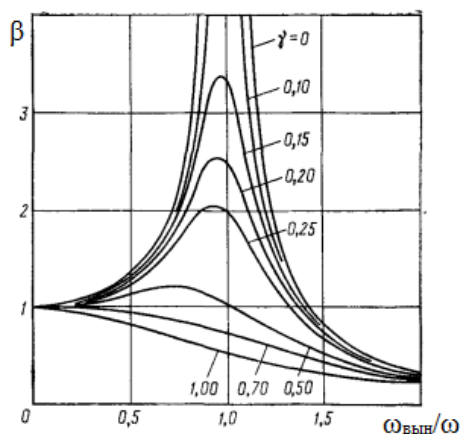
where ω_A – circular damped frequency;

ω_B – circular frequency of the driving force;

$2n = c/m$, 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_B.

To explain the influence of damping, let us consider a plot on which is shown the dependence of the

amplitude amplification factor $\beta = 1/\sqrt{\left(1-\omega_{e}^{2}/\omega^{2}\right)+\left(2\gamma\omega_{e}/\omega\right)^{2}}$ on the ratio of frequencies of the forced and free oscillations $\omega_{вын}/\omega$ for different values of the damping coefficient $\gamma = n/\omega_{â} = c/c_{êð}$,

where $c_{êð}$ – 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

$\omega_{e} \ll \omega$ and $\omega_{e} \gg \omega$ 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_j 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: $a = 2\omega_i\omega_j\dfrac{\gamma_j\omega_i - \gamma_i\omega_j}{\omega_i^2 - \omega_j^2}$ and $b = 2\dfrac{\gamma_i\omega_i - \gamma_j\omega_j}{\omega_i^2 - \omega_j^2}$. 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 *Um*.

Oscillation acceleration in nodes of the finite element mesh which are expressed in terms of the amplitudes *Um* as $\ddot{U}_m = U_m\omega_{вын}^2$.

- Oscillation overloads defined as the ratio of the oscillation acceleration to the gravitational acceleration $\ddot{U}_m/g$.

Special Features of Forced Oscillation Analysis Stages

The forced oscillation analysis is carried out in several stages. The sequence of user's actions for task 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:

1. **Creation of «Study»**. When creating a study we need to specify its type – «Forced oscillations» in the command's properties window.
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.

3. **Analysis execution**. Before execution of the analysis the user specifies, in the properties 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.
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

1. **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.
2. **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.

3. 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.
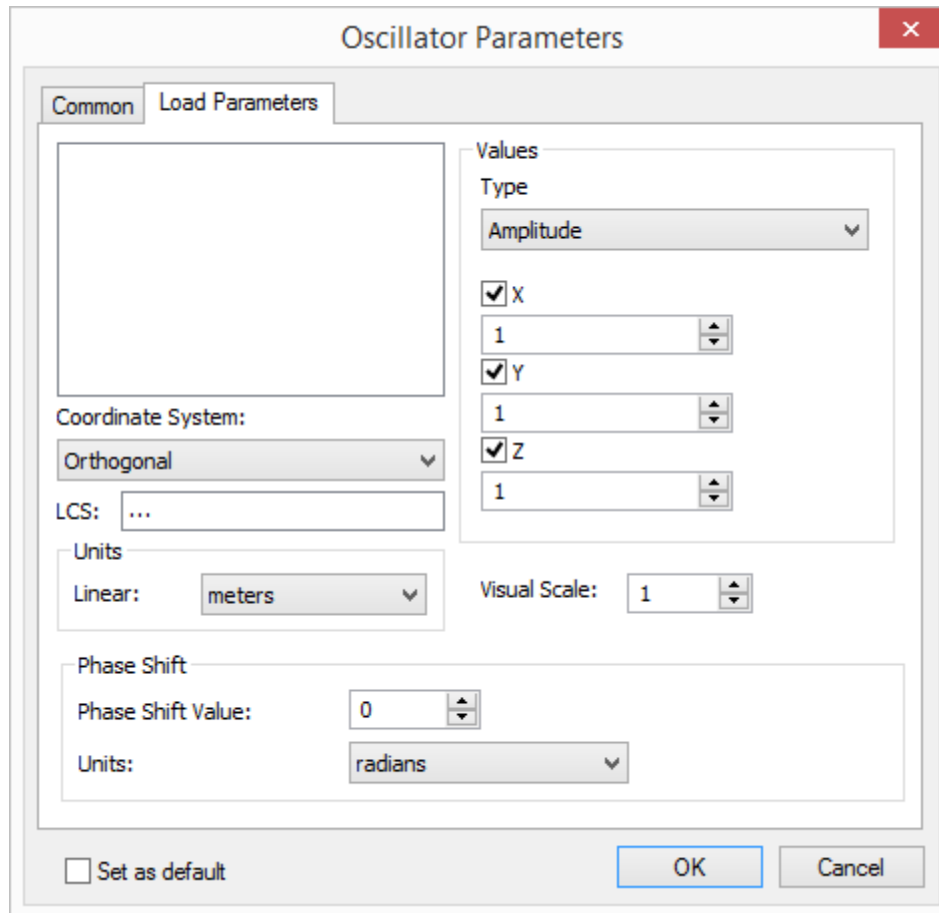
The type of kinematic loading is selected from the drop-down list and can be the following:

- 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 **[General]** tab we can define or change the descriptive properties of the current study: name, study type, comments.

On the **[Parameters]** tab we specify the main settings for forced oscillation analysis.

In the group of parameters **Frequency of external load** we specify the values for the frequencies of the external loads. New values of frequencies can be added in several ways:

1. 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.
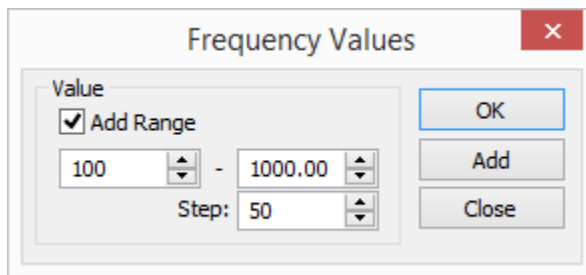
2. The **[Import]** button invokes the dialog window in which the values of the resonance frequencies can be imported from the results of the frequency analysis completed earlier.
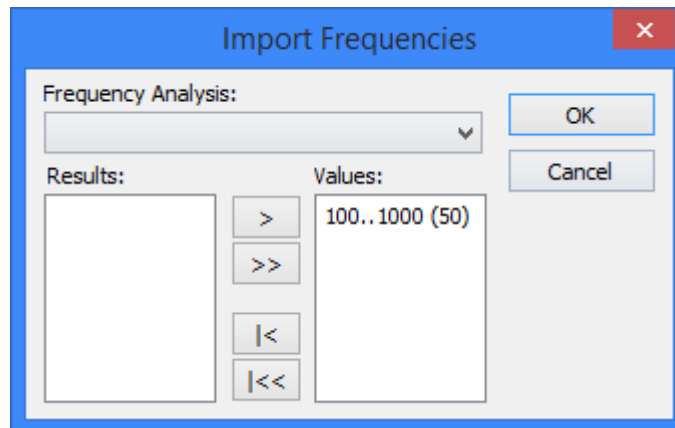
3. In case when it is required to execute the forced oscillation analysis only on all resonance frequencies, which were found in the earlier performed frequency analysis, enable the control element «**Use frequency analysis results**». The associated connection with the results of the selected frequency analysis is supported here, i.e., when the results of the frequency analysis are
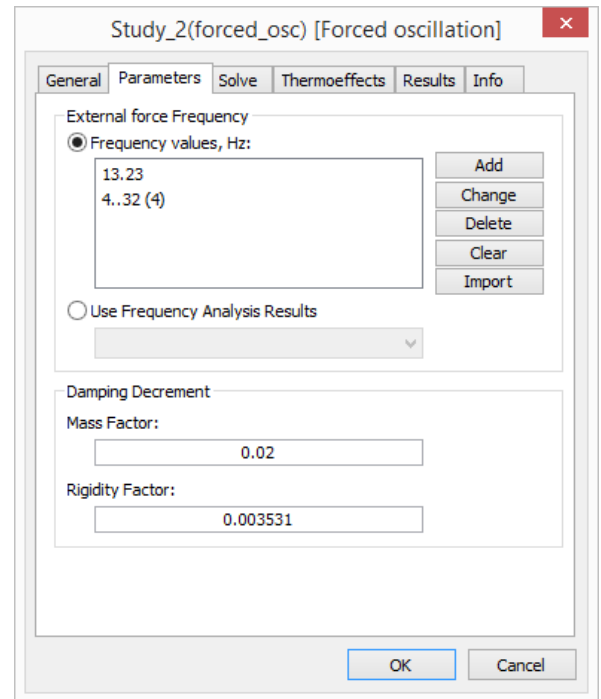
changed, the values of the updated natural frequencies will automatically be used in the forced oscillation analysis.

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 group of parameters «**Damping**» the structure's damping coefficients value is specified.

The **[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]** tab allows us to define the type of results, which will be displayed in the tree of tasks 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:
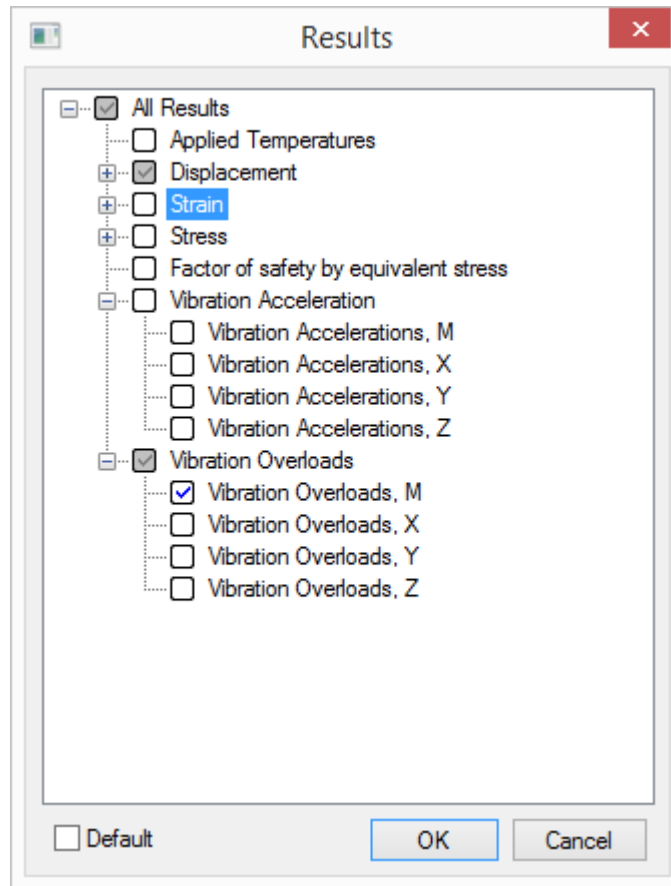
The «**Loads**» group includes the following results: the components and the magnitude of loads applied to the nodes of the finite element model. This type of results is considered as reference information.

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: $U_X = \sqrt{\mathrm{Re}\,U_X^2 + \mathrm{Im}\,U_X^2}$,

$U_Y = \sqrt{\mathrm{Re}\,U_Y^2 + \mathrm{Im}\,U_Y^2}$, $U_Z = \sqrt{\mathrm{Re}\,U_Z^2 + \mathrm{Im}\,U_Z^2}$, and also the magnitude of the displacement

$$U = \sqrt{U_X^2 + U_Y^2 + U_Z^2}$$.

- the real part of the displacement in the direction of the axes of the global coordinate system: *Re(UX)*, *Re(UY)*, *Re(UZ)*, and also the magnitude of the real part of the displacement $\mathrm{Re}\,U = \sqrt{\mathrm{Re}\,U_X^2 + \mathrm{Re}\,U_Y^2 + \mathrm{Re}\,U_Z^2}$.

- the imaginary part of the displacement in the direction of the axes of the global coordinate system: *Im(UX)*, *Im(UY)*, *Im(UZ)*, and also the magnitude of the imaginary part of the displacement $\mathrm{Im}\,U = \sqrt{\mathrm{Im}\,U_X^2 + \mathrm{Im}\,U_Y^2 + \mathrm{Im}\,U_Z^2}$.

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:

UXm, *UYm*, *UZm*, and also the magnitude of the amplitude $U_m = \sqrt{U_{Xm}^2 + U_{Ym}^2 + U_{Zm}^2}$.

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 $\varphi_{U_X} = arctg\left(\dfrac{\mathrm{Im}\,U_X}{\mathrm{Re}\,U_X}\right)$,

$\varphi_{U_Y} = arctg\left(\dfrac{\mathrm{Im}\,U_Y}{\mathrm{Re}\,U_Y}\right)$, $\varphi_{U_Z} = arctg\left(\dfrac{\mathrm{Im}\,U_Z}{\mathrm{Re}\,U_Z}\right)$, and also the magnitude of the phase angle $\varphi_U = arctg\left(\dfrac{\mathrm{Im}\,U}{\mathrm{Re}\,U}\right)$.

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 $\ddot{U}_m = U_m \omega_{вын}^2$ 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 $\ddot{U}_m / g$ 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:
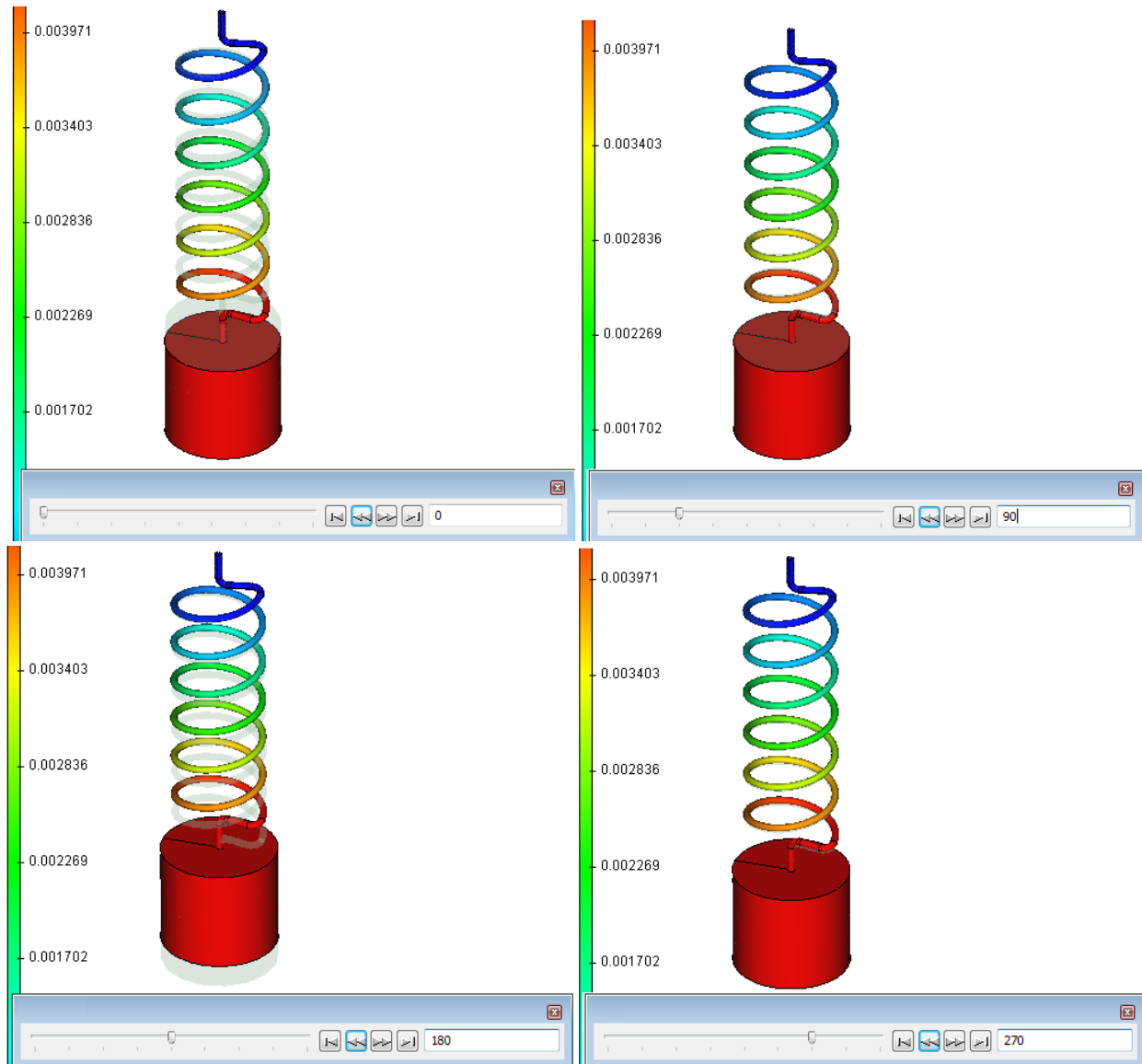
The phase value can be specified by translation of the slider or by inputting the value directly into the numeric field.

To obtain ARC (amplitude resonance curve), it is required to create at least one sensor (see the section of the Reference manual «Processing results (Postprocessor)/Using sensors and graphs for the results analysis») 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 aditional 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 spinup, 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.

$$\sigma_{dyn} \cdot SF \leq [\sigma]$$

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:

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{F(t)\},$$

(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,

$\dot{U}, \ddot{U}$ - 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:

1.   On the **Solve** tab in the "Use frequency analysis results" group select already calculated study of frequency analysis (study type: "Natural frequencies"). All natural modes from the selected study will be used for finding solution of mode superposition.

2.   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.
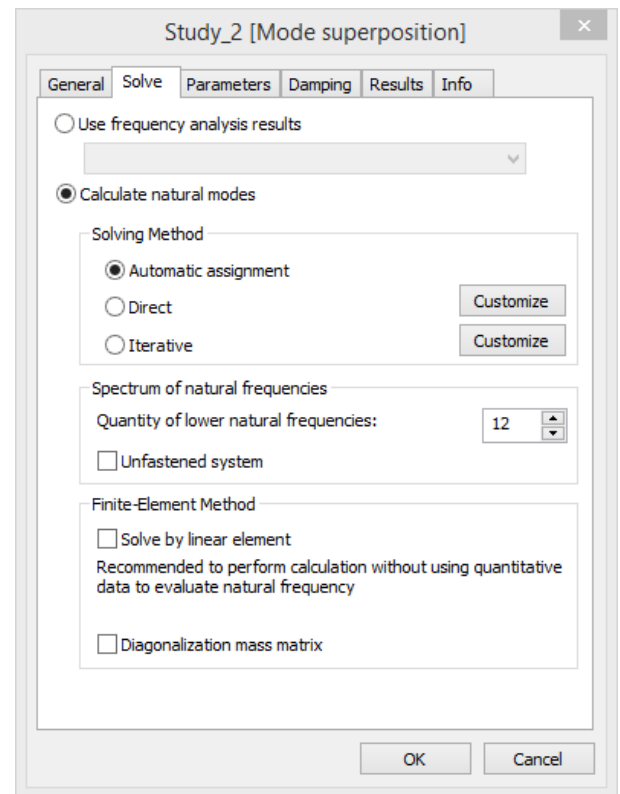
Dynamic Transitional Process

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 **Solve** tab repeat the same parameters for static analysis.
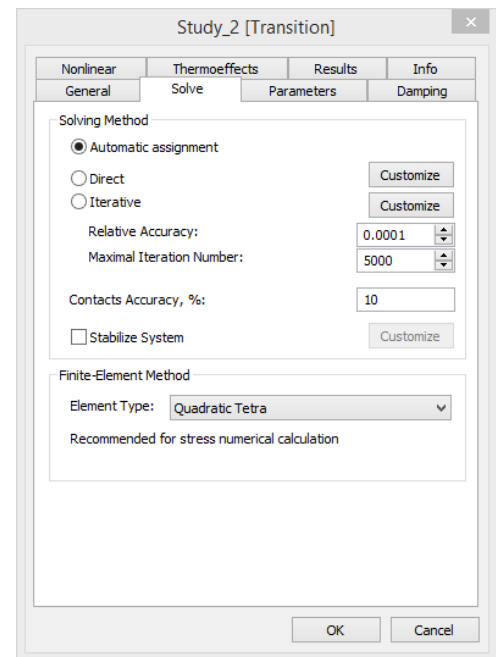
Parameters of Finite Time and Step of Modeling

User should specify the time step and finite time of modeling before running the dynamic calculation.

The parameters are set on the **Parameters tab.** 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:

$$\{\dot{U}(t+\tau)\} = \{\dot{U}(t)\} + \left[(1-\delta)\{\ddot{U}(t)\} + \delta\{\ddot{U}(t+\tau)\}\right]\tau \,,$$

$$\{U(t+\tau)\} = \{U(t)\} + \{\dot{U}(t)\}\tau + \left[(\frac{1}{2}-\alpha)\{\ddot{U}(t)\} + \alpha\{\ddot{U}(t+\tau)\}\right]\tau^2 \,,$$

where α, δ - parameters that determine the accuracy and stability of integration.

For α=1/6, δ=1/2 accelerations vary linearly within the interval of integration; preferred for linear tasks;

For α=1/4, δ=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: $\alpha > 1/8$.
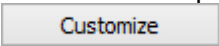
The Wilson method is based on the fact that the vector of accelerations and loads vary linearly in the interval $(t+\theta\tau)$, where $\theta \geq 1$, based on the following approximation of the accelerations:
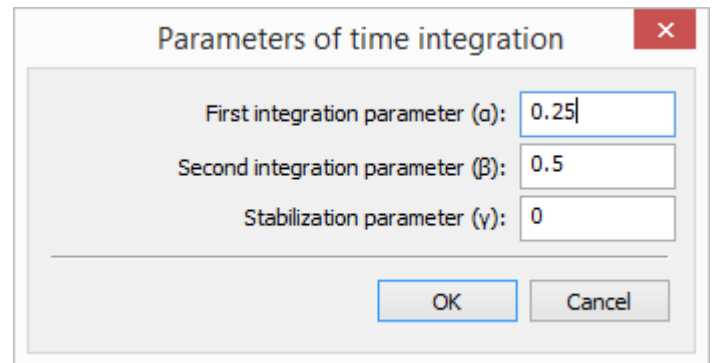
$$\{U(t+\Delta t)\} = \{U(t)\} + \frac{\Delta t}{\theta t}(\{U(t+\theta\tau)\} - \{U(t)\})$$

The expressions for velocities and displacements are received by integration of the equation:

$$\{\ddot{U}(t+\theta\tau)\} = \frac{6}{\theta^2\tau^2}(\{U(t+\theta\tau)\} - \{U(t)\}) - \frac{6}{\theta\tau}\{\dot{U}(t)\} - 2\{\ddot{U}(t)\}$$

$$\{\dot{U}(t+\theta\tau)\} = \frac{3}{\theta\tau}\left(\{U(t+\theta\tau)\} - \{U(t)\} - 2\{\dot{U}(t)\} - \frac{\theta\tau}{2}\{\ddot{U}(t)\}\right)$$

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.
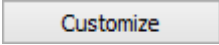
Parameters of time integration
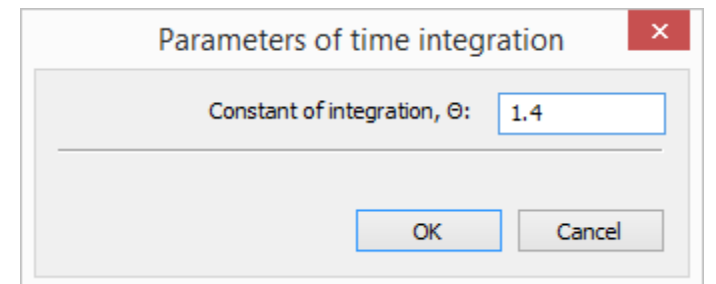
First integration parameter (a): 0.25

Second integration parameter (β): 0.5

Stabilization parameter (γ): 0

OK   Cancel

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.

Parameters of time integration

Constant of integration, Θ: 1.4
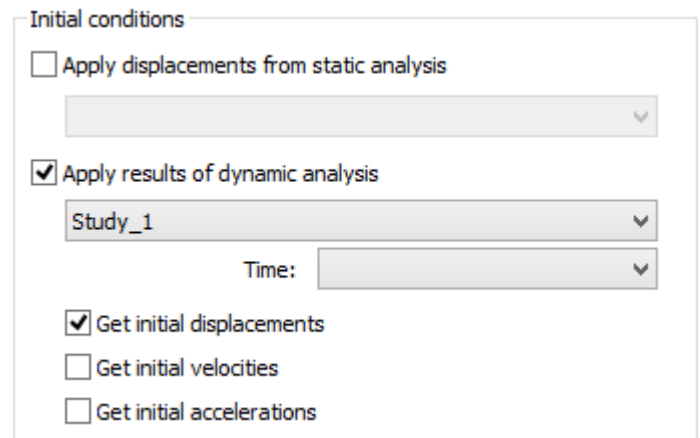
OK   Cancel

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 **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.
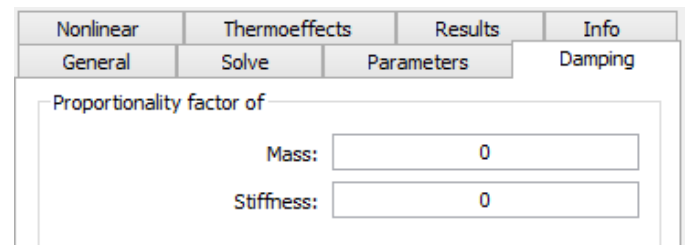
Damping Accounting

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 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 properties 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.

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:

- ✓ *Study creation*. When you create study you should select its type - Mode superposition or Transitional processes.

- ✓ *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.

  a. **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.

  b. 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.

✓ *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.
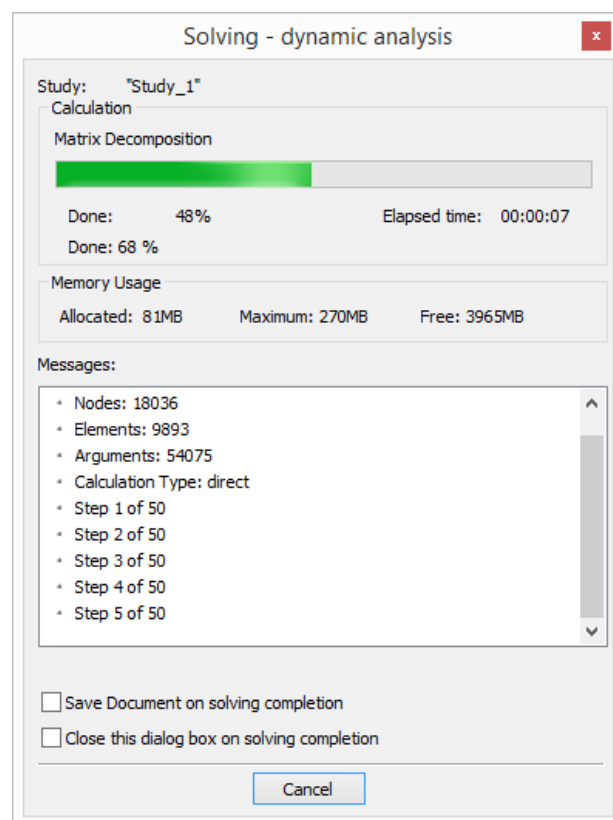
✓ **Solving results analysis.** After the solving is finished a new "Results" folder appears in the Studies window. Results specified on the Results tab in the Parameters dialog are displayed in the folder by default. Only 49 results grouped in 8 groups are available to the user after the dynamic analysis.

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.

4.   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.

5.   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 **3MN: 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.
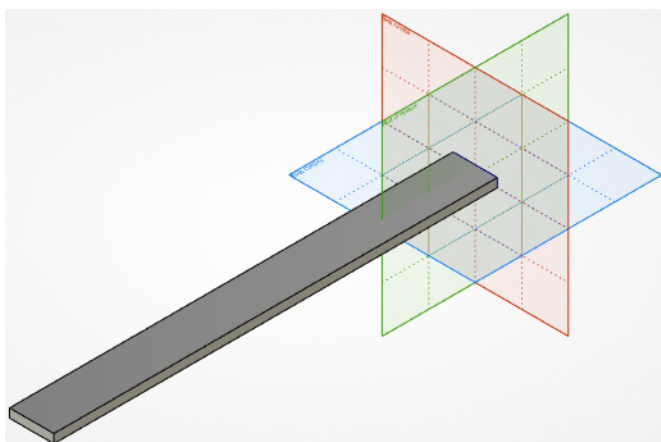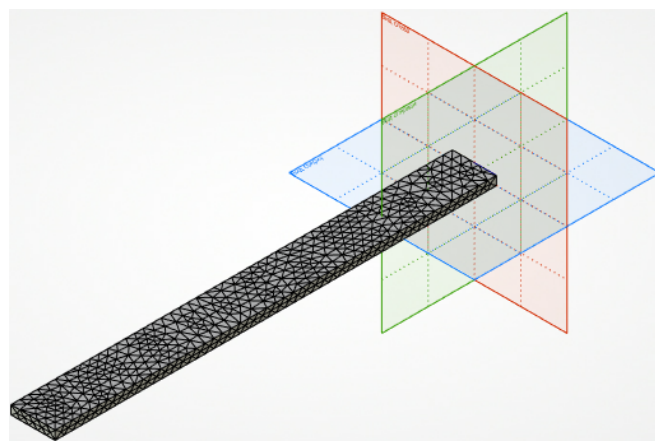
| 3D model | Created finite-element mesh |

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.
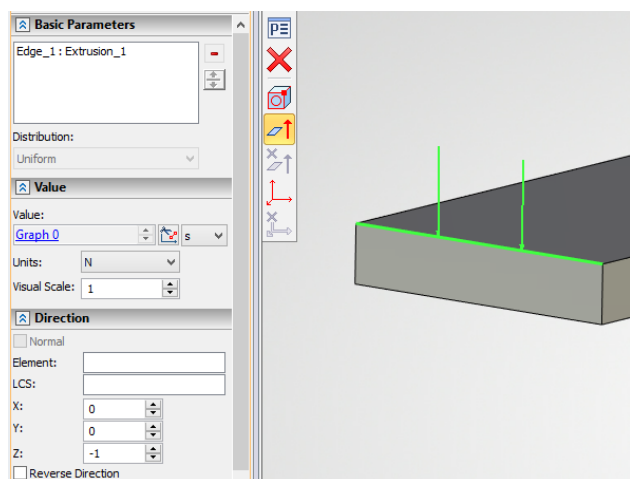
| Varying load to the face | Graph of varying load |
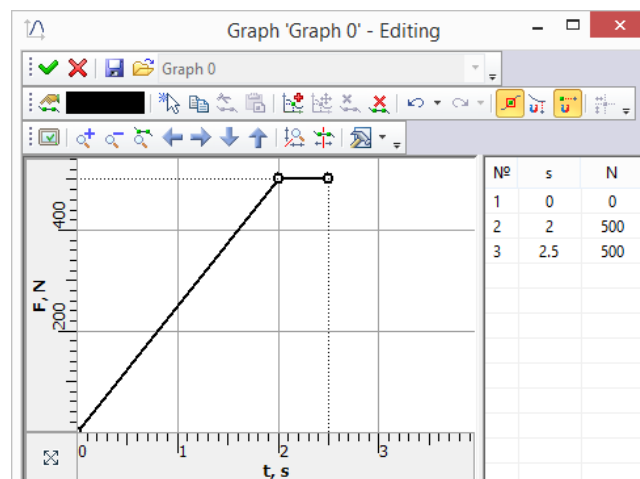
Step 3. Creation of sensors and graphs templates. To define the maximum deflection and velocity of the beam end it is necessary to create sensor using which the graph template is created:
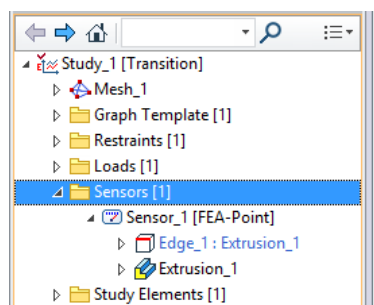
Sensor in the middle of the edge
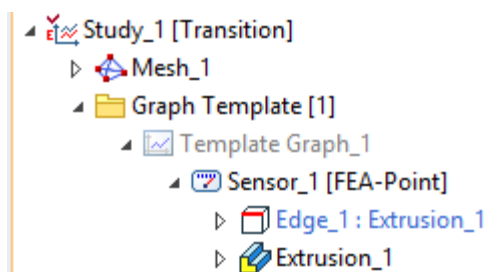
Graph template created using sensor in the study window

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.
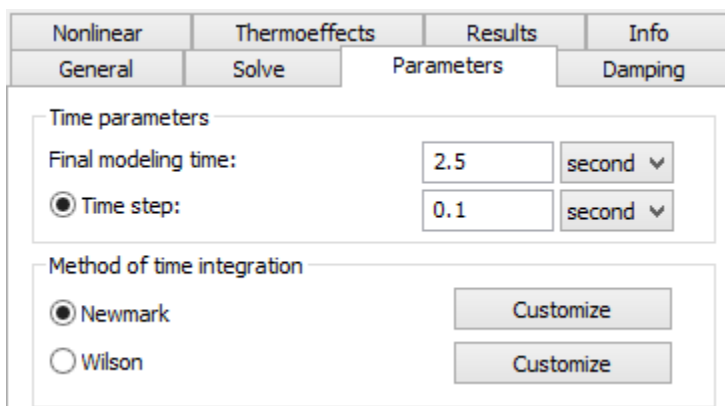
Specify step and time of the modeling

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.
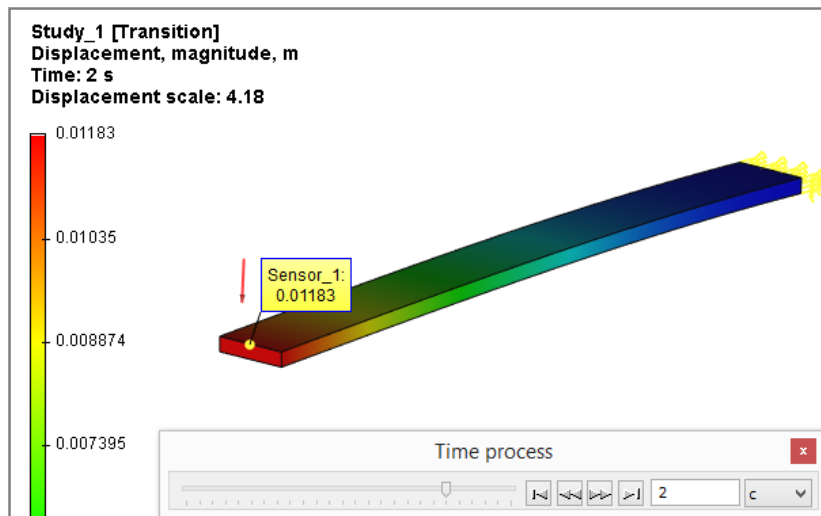
Displacements - time process

Deflection of the same beam under the constant force, received after static strength study calculation. The results practically coincide.

Study_2 [Strength]
Displacement, magnitude, m
Displacement scale: 4.23
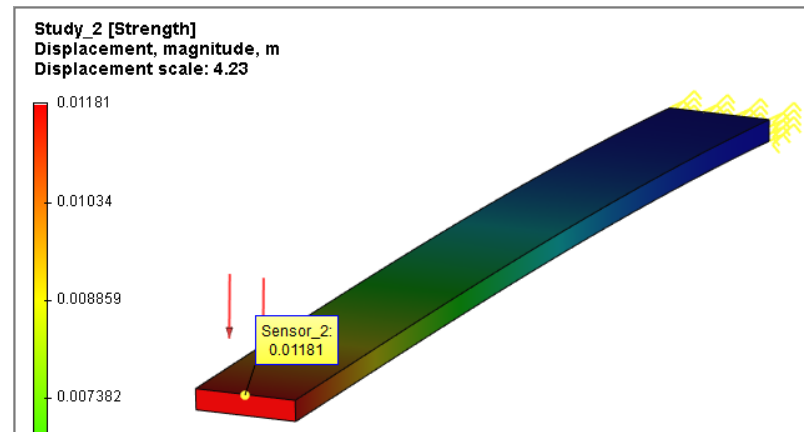
Comparison with the static displacement from the same load

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.
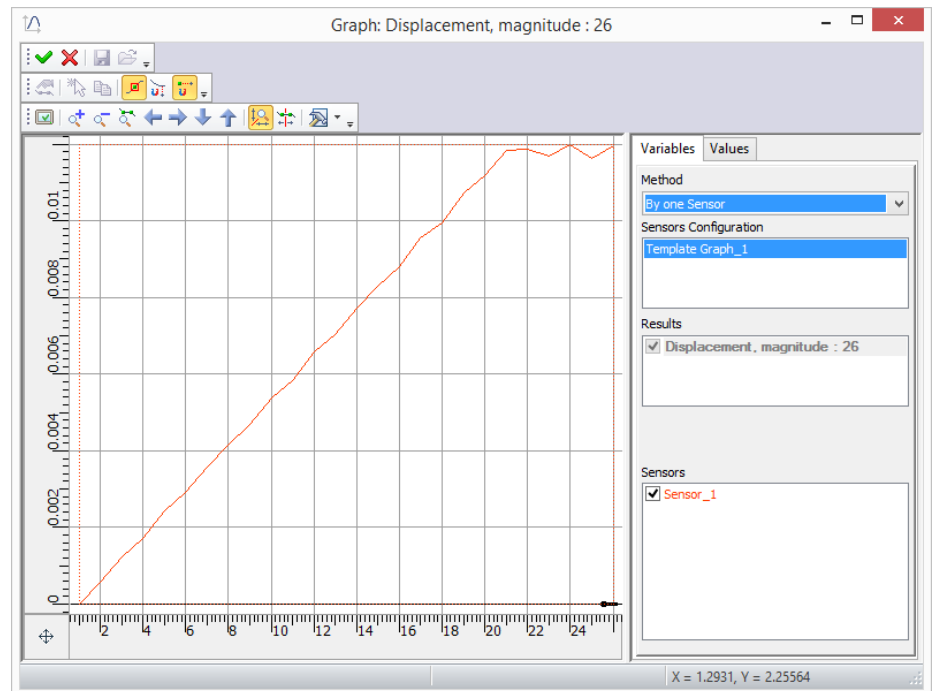
Displacements - time process

The maximal stresses can be shown on the stresses diagram. Display extremums.
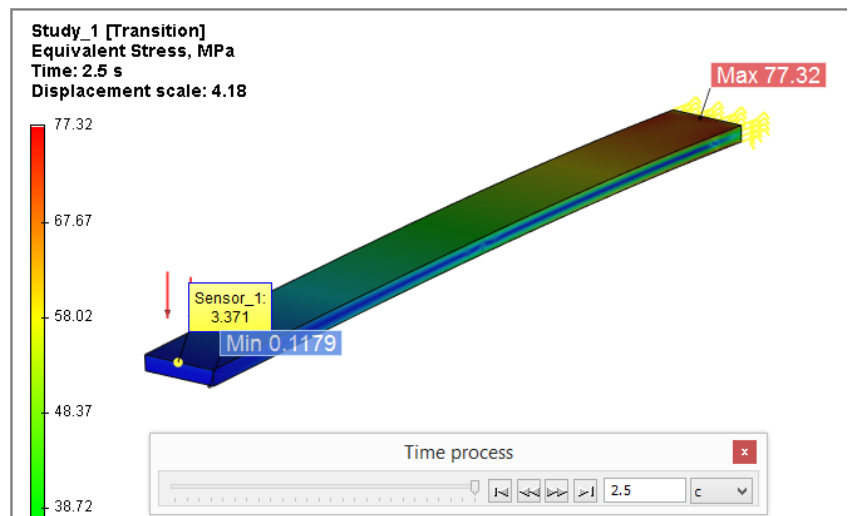
Dynamic stresses (peak)

Make sure that received dynamic stresses are close to the stresses received in the static calculation.
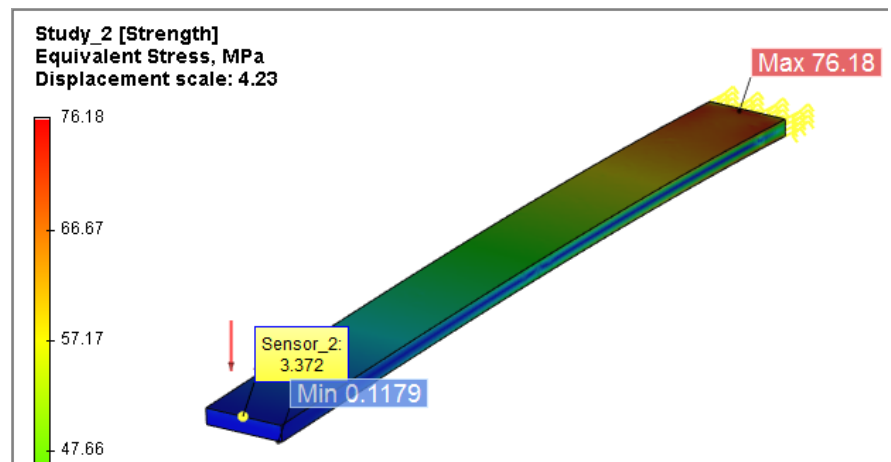
Comparison with static stresses

Mode Superposition

Calculate previous study using mode superposition (type: *Mode Superposition*).

Step 1. Creation or copying of the study. Use the 3MN: 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 properties not to apply twice the same loads and restraints.

Step 2. Calculation of natural modes.

Select Calculate natural modes on the **Solve** tab in the study properties. 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 properties on the **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.
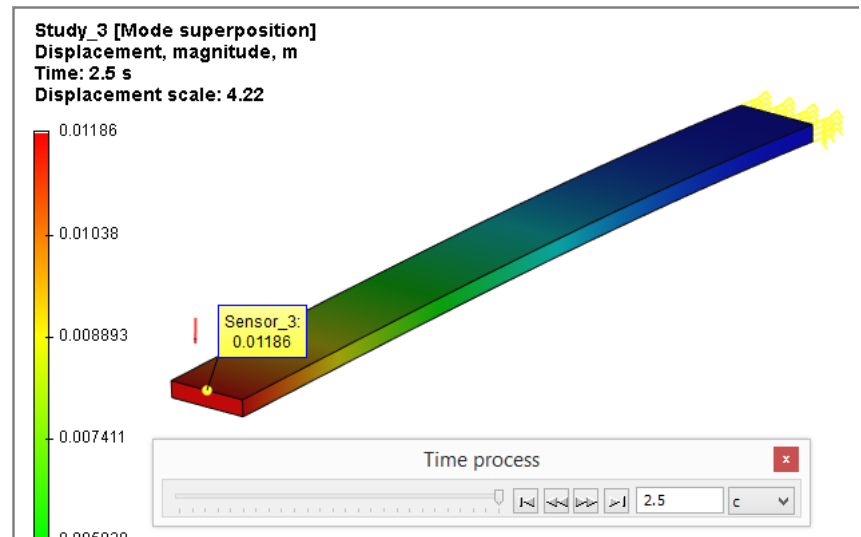
Displacements - time process

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.
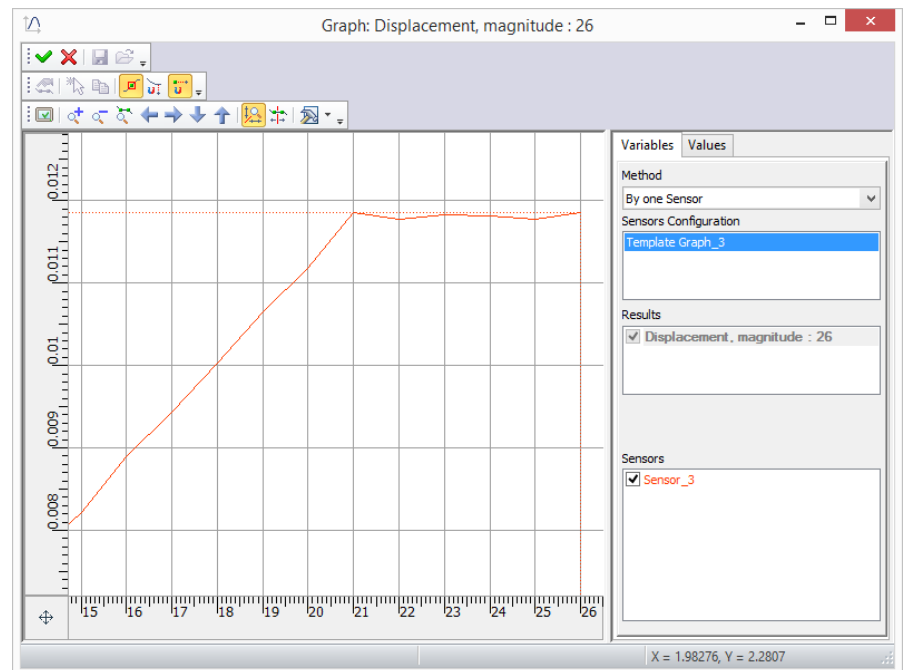
Displacements - time process

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.
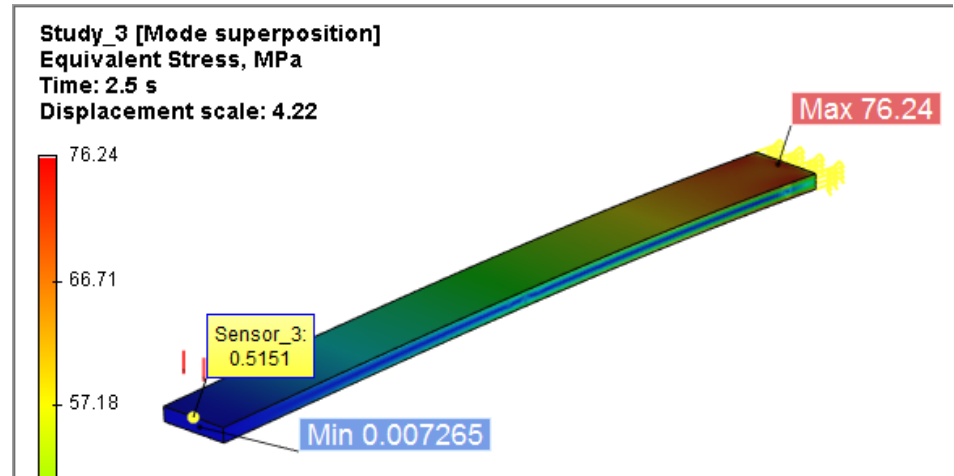
Dynamic stresses in the Mode superposition

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

THERMAL ANALYSIS

The thermal analysis module serves for solving heat transfer and thermal conduction studies. 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 studies:

- **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.
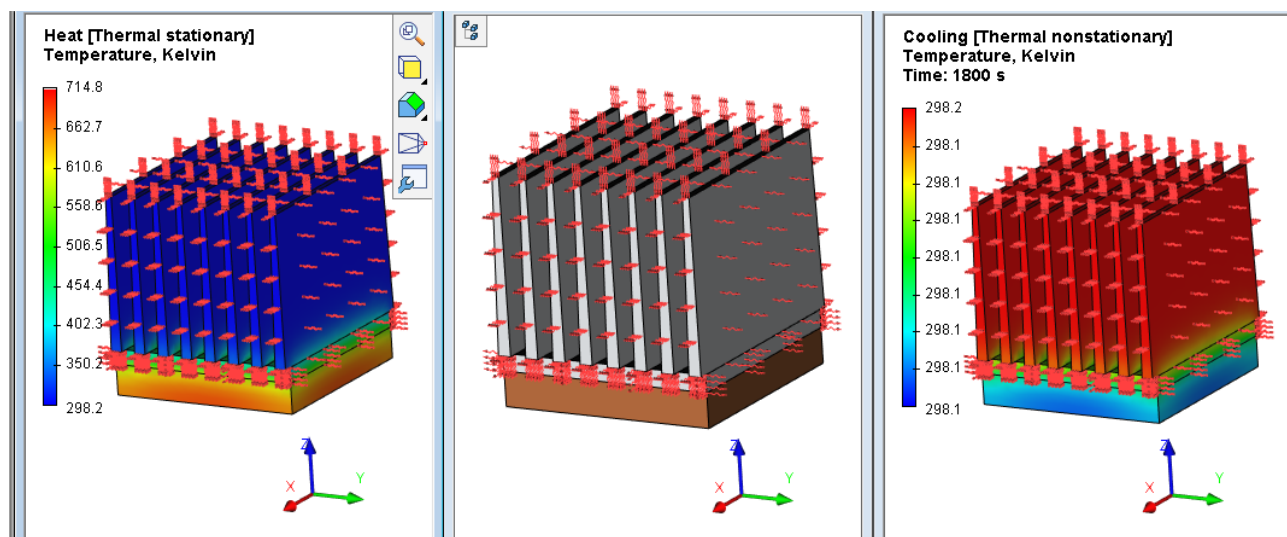
Examples of a stationary (left) and nonstationary (right) thermal processes

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.

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.

2. **Applying boundary conditions.** In the thermal analysis, the boundary conditions are represented by the boundary and initial temperatures, heat power sources, heat flux, and conditions of heat exchange between the model and environment – convection and radiation applied to the model.
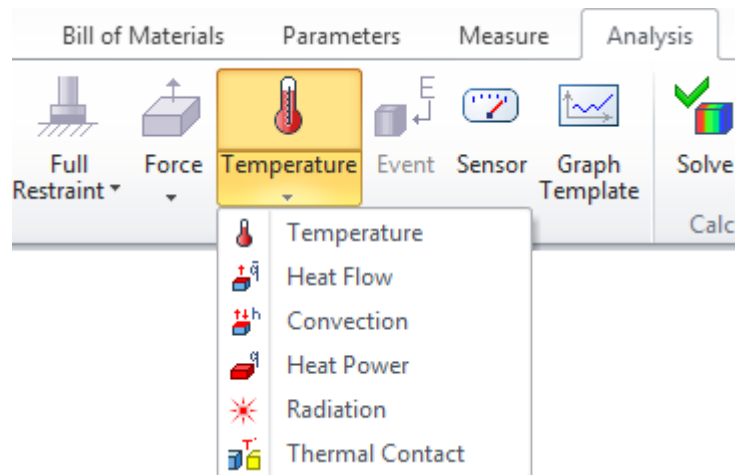
Commands for defining boundary conditions of thermal analysis

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.

3. **Solving**. Before running calculations, the user can specify adjust algorithms for solving systems of equations on the **[Solve]** tab.

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 **Nonstationary 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.
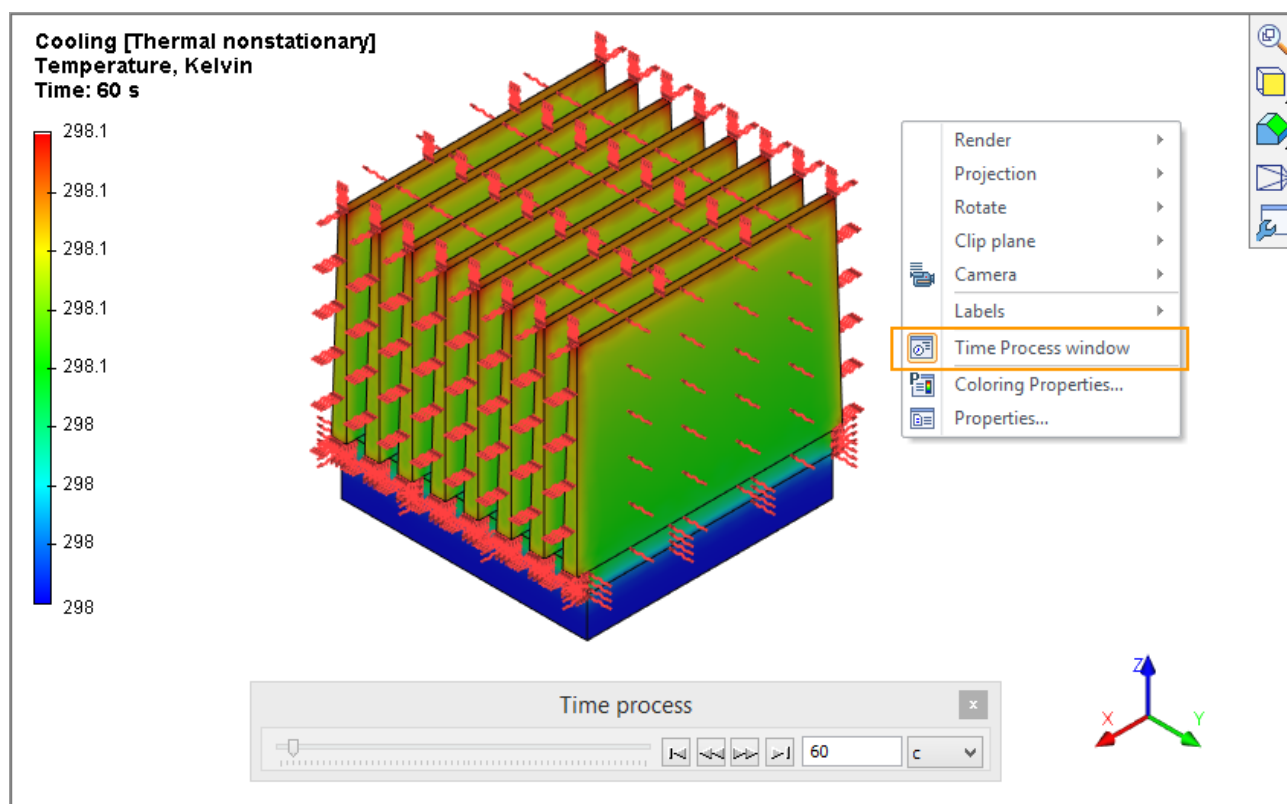
Use of the «Time process» window for managing access to the results of a Nonstationary Thermal Process study

Thermal Analysis Processor Settings

Stationary Thermal Process

The **General, Solve, Results** and **Info** tabs are the same for the both of thermal studies types.
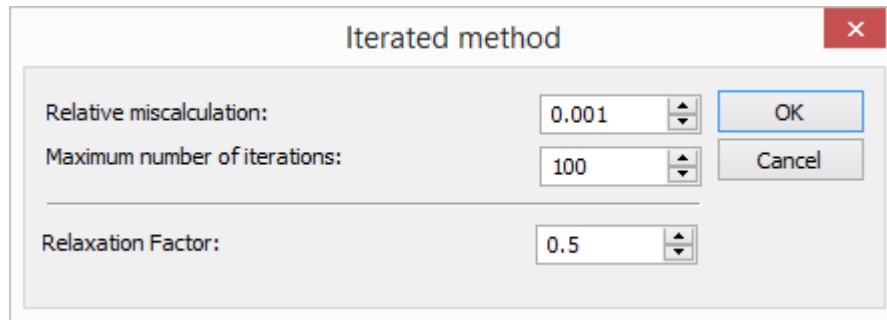
On the **[General]** tab, you can define or edit the descriptive attributes of the current study, as the name or a comment. The **[Solve]** tab contains settings for solving systems of algebraic equations, with their meanings similar to the settings of the «Static analysis» study (see the respective section). Note that the **Calculate using linear element** 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" calculation modes become available. When "Automatic" mode is chosen, the system assigns method of solution based on total number of equations.
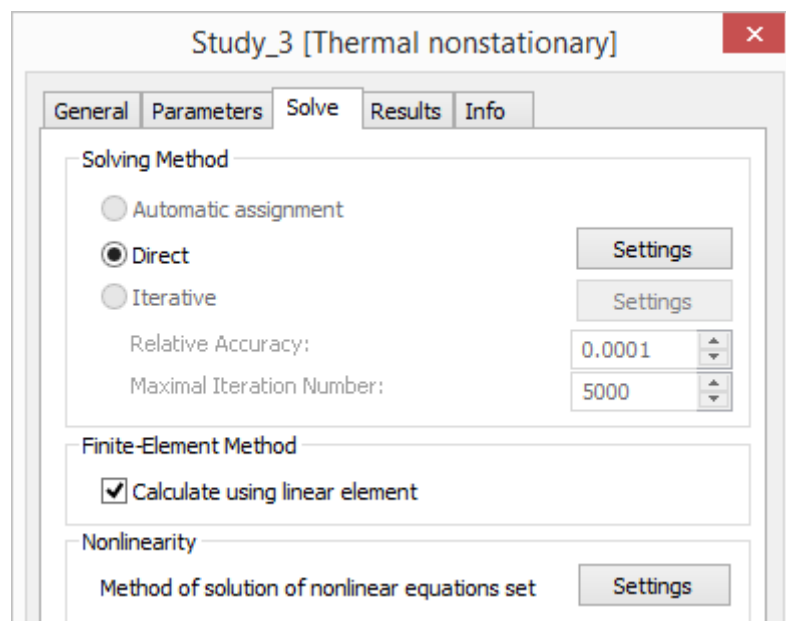
Thermal studies parameters dialogs

You can open the dialog of the iterated method of nonlinear equations set solution using the **[Settings]** button in the Nonlinearity group.
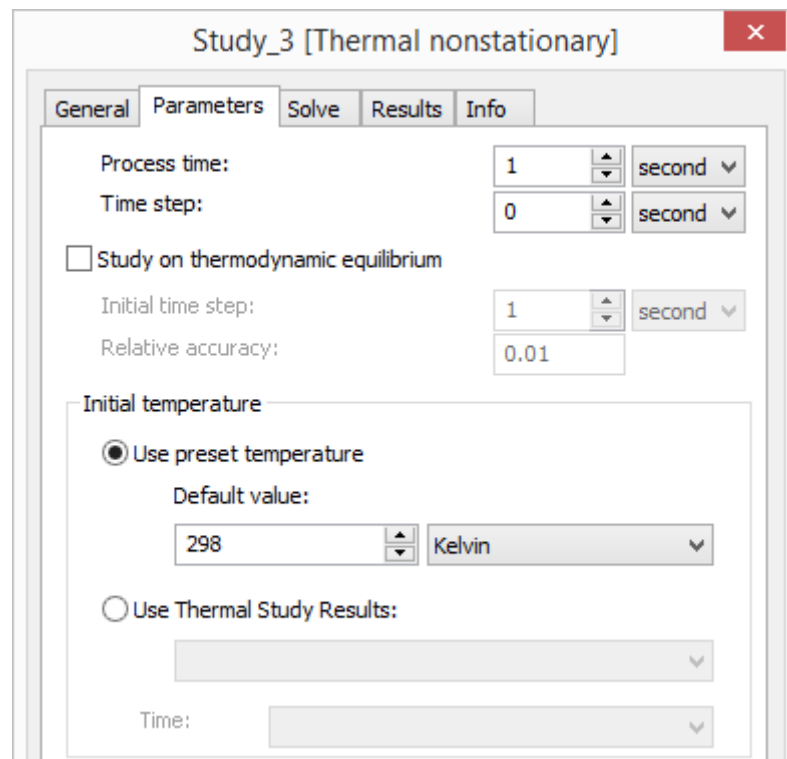
Nonstationary Thermal Processes

The user should set process time, time step and initial temperature using the **Parameters** tab before the calculation of the nonstationary thermal process study.

The **Study on thermodynamic equilibrium** study type is also available on the **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 heat-task 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:

1. 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:

   a) 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;

   b) create a study's copy using the **Copy Study Items** command;

   c) 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;

   d) define boundary conditions of a transient study in thermal analysis. On the **Parameters** tab of the study's properties, 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.

2. The **Calculate using linear element** property on the **Solve** 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.

Examples of Thermal Analysis Studies

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".
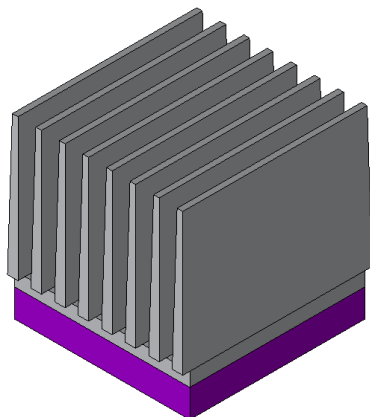
Three-dimensional model of the microchip with a passive cooling radiator
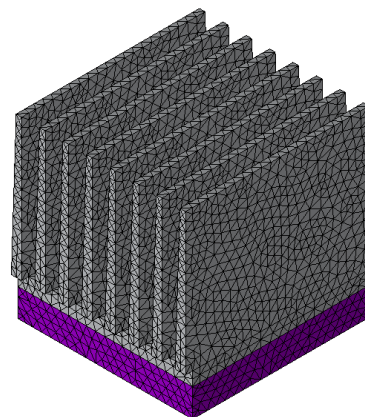
Resulting finite element mesh

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/(m^2 ·°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.
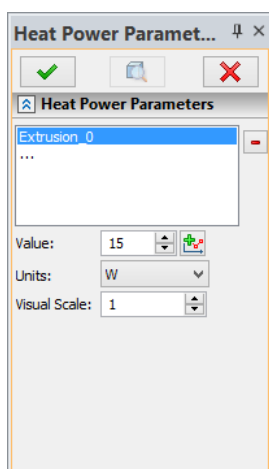
Defining "Heat Power" load
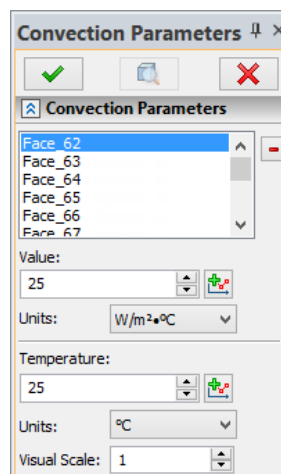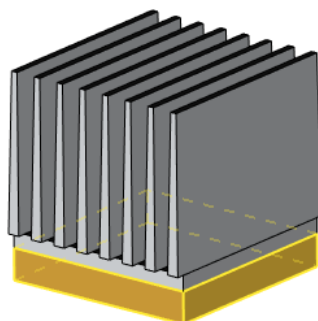
Defining "Convection" load
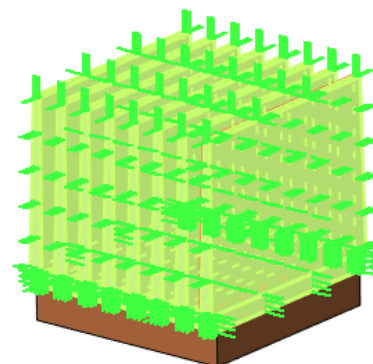
Step 3. Running calculations and analyzing results. We will start the thermal analysis by running the command **Analysis > Solve**. In the appearing dialog of the study's properties, set the «Steady state» option on the **[Parameters]** tab. Use the «Calculate using linear element» mode on the **[Solve]** 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.

Heating [Thermal stationary]
Temperature, Celsius
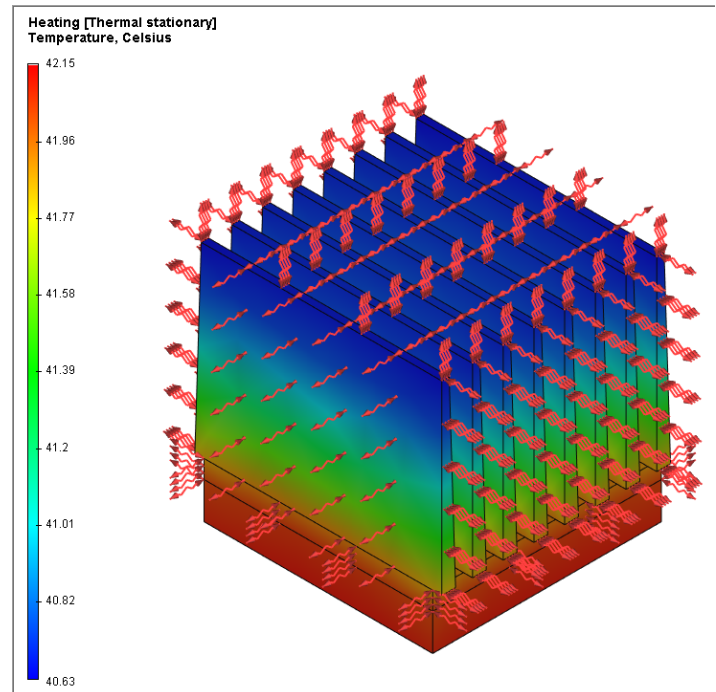
Calculating the Time of Heating up the Cooling Radiator. Transient Mode

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 **General** tab of the study's properties, change the study name to «Heating time».
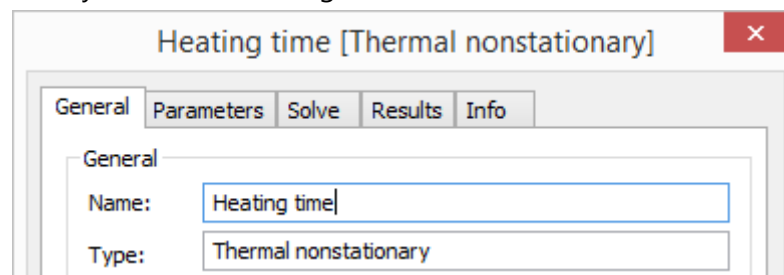
Step 2. Defining parameters of transient analysis. We change the study type to the "Nonstationary Thermal Process" using the **Edit** button from the context menu of the 3D model window. On the **[Parameters]** tab of the thermal analysis properties, we set the finite time– the modeling time of 30

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.
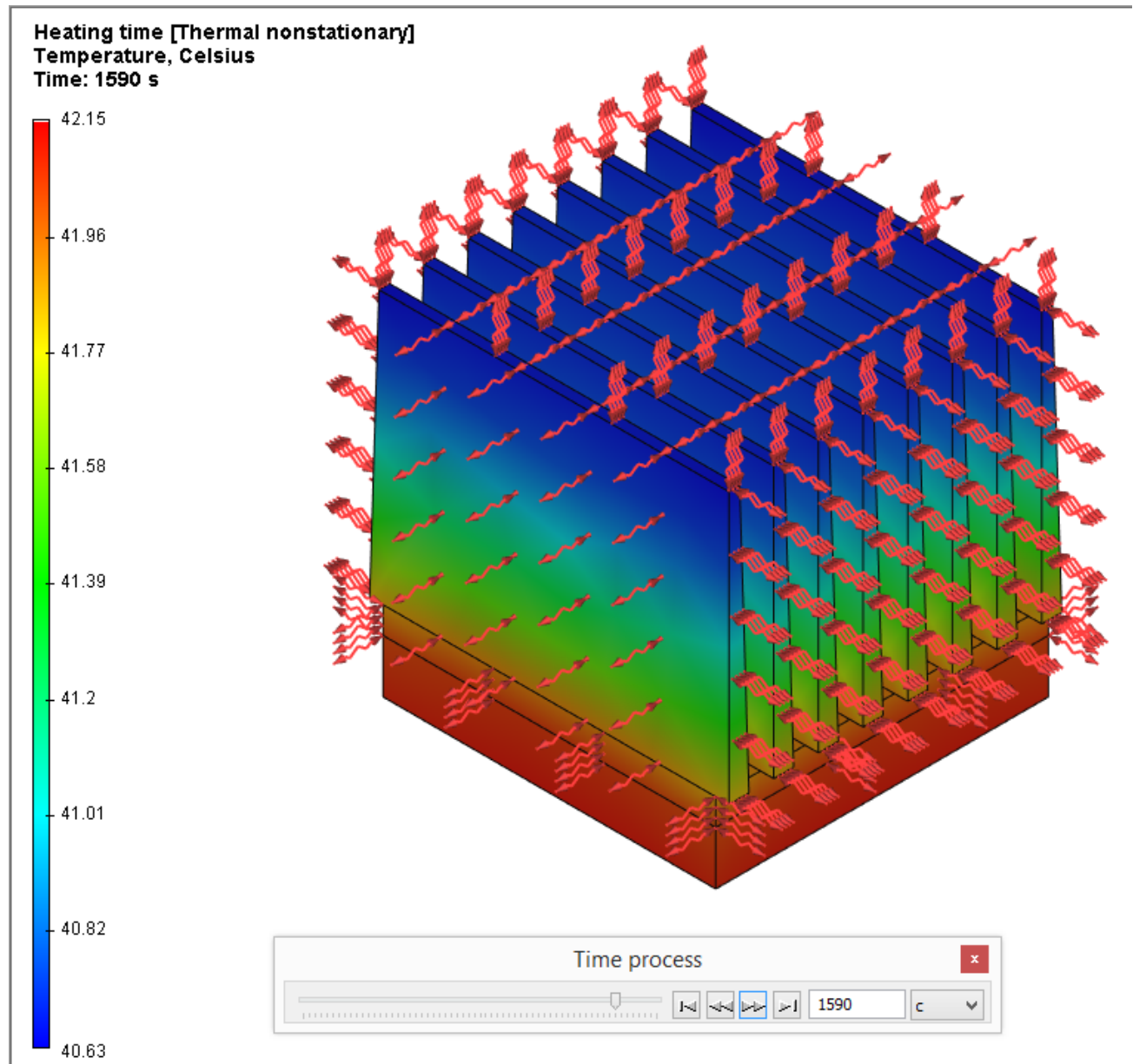
Result of thermal analysis at time 1590 sec (27 min.)

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. Adjusting boundary conditions. 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" study from the drop-down list.
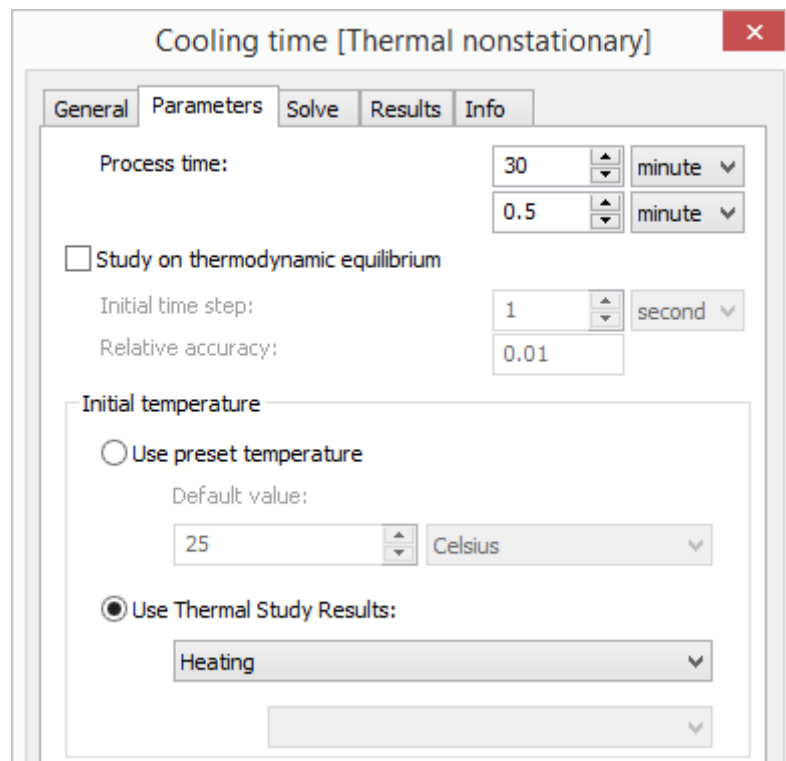
Setting up study parameters for calculating cooling process

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.
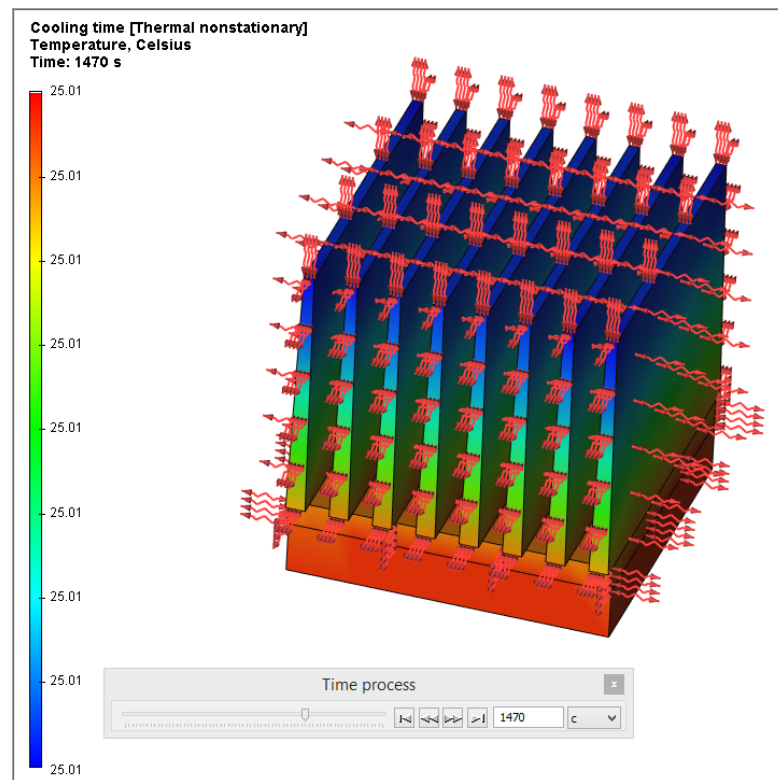
Device cooling calculation.
Temperatures distribution at 1470 seconds of the calculation time

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 study, 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 studies 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 studies that correspond to the given description. For several studies, 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 studies.

> **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.
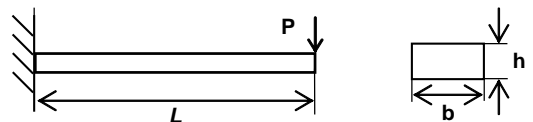
Sought is the maximum beam deflection.

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.
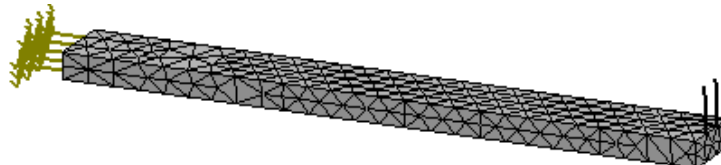
Finite-element model of the beam with loads and restraints

The analytical solution appears as:

$$w = \frac{Pl^3}{3EJ} = 1.1905 \times 10^{-2} \; m,$$

where P – is the force, l – the beam length, E – the material Young's modulus, $J = \dfrac{b \cdot h^3}{12}$ – 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

Table 2.

Result «Displacement»

Numerical Solution Displacement w^*, m	Analytical Solution Displacement w, m	Error $\delta = \dfrac{\lvert w - w^* \rvert}{\lvert w \rvert} \times 100\%$
1.1826E-002	1.1905E-002	0.66
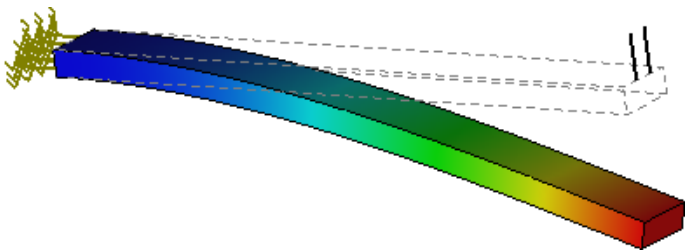
Displacements of beam points

Conclusions:

The relative error of the numerical solution compared to the analytical solution is equal to 0,7% for quadratic finite elements.
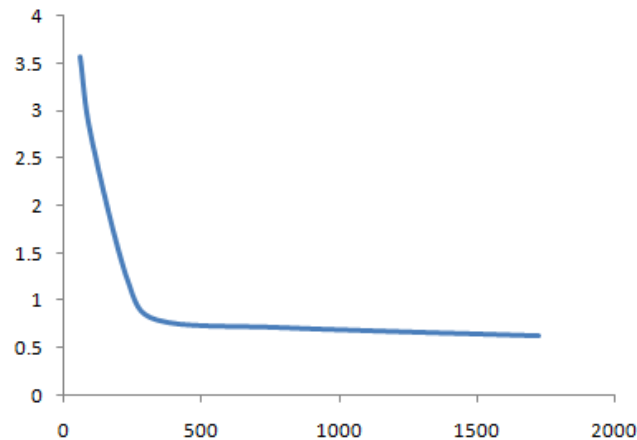
Dependence of the relative error on the number of finite elements

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.
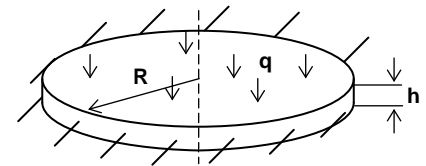
Assume that the plate radius is R=0.2 m, thickness h = 0.003 m, and the pressure q = 10 $\kappa N/m2$. Material characteristics assume default values: the Young's modulus E = 2.1E+011 Pa, the Poisson's ratio v = 0.28.

Next, we need to apply boundary conditions. The side surface of the plate will be fully restrained, whereas the free faces introduced after discarding ¾ of the plate are subjected to partial restraints in the direction 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.
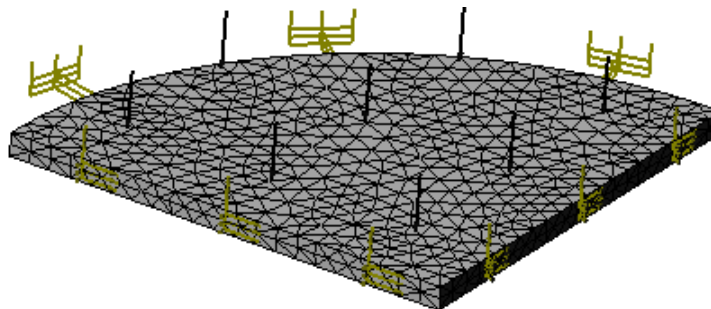
Finite element model of the plate with loads and restraints

There is an analytical solution to this study. The deflection at the plate center is calculated by the formula:

$$w = \frac{qR^4}{64D} = 4.8762 \times 10^{-4} \ m,$$

where $D = \frac{Eh^3}{12(1-v^2)}$ – flexural rigidity, q – is the pressure amount, R – the plate radius.

The stress on the plate contour is calculated by the formula:

$$\sigma = 0.75q\left(\frac{R}{h}\right)^2 = 3.3333 \times 10^7 \ Pa.$$

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 $\delta = \frac{\|w - w^*\|}{\|w\|} \times 100\%$
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 $\delta = \frac{\|\sigma - \sigma^*\|}{\|\sigma\|} \times 100\%$
3.0057E+007	3.3333E+007	9.83
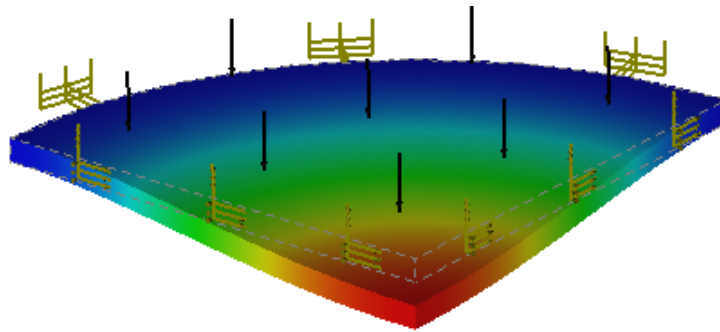
Displacements of plate points

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.
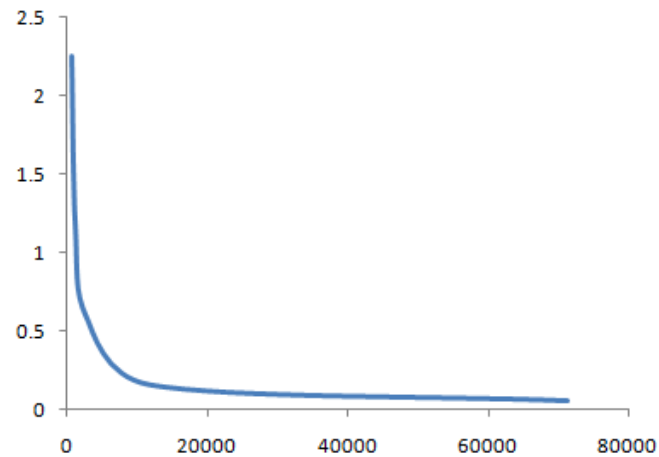
Dependence of the relative error on the number of finite elements

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 p_0 and external pressure p_1. 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 p_0=200 MPa,

Outer pressure p_1=120 MPa.

Assume the material properties as $E = 2.1 \cdot 10^{11}$ Pa, $v = 0.28$.

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.
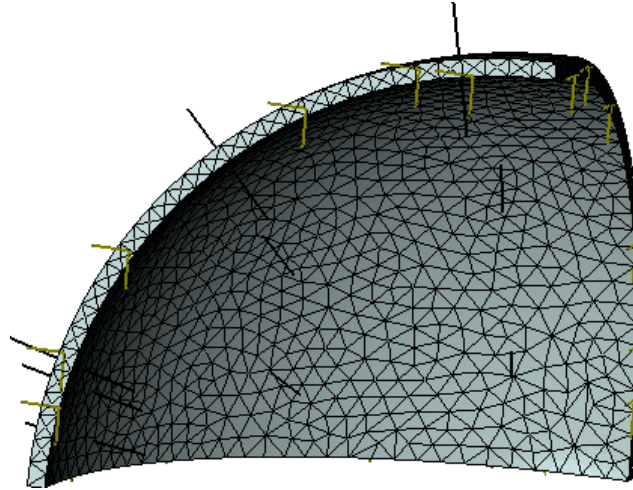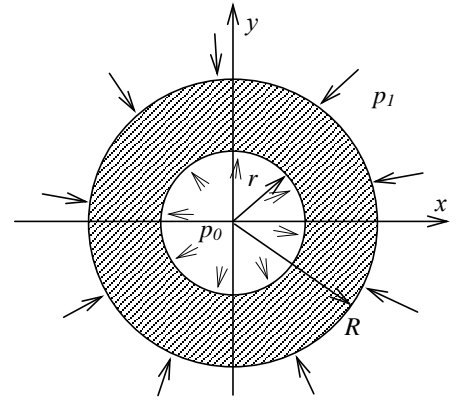
Finite element model of 1/8 of a sphere with loads and restraints

Displacement of internal surface of the sphere can be evaluated using the formula:

$$U = A \cdot r + \frac{B}{r^2} = 1.4063 \ mm,$$

$$A = \frac{\dfrac{P_1}{r^3} - \dfrac{P_0}{R^3}}{(2\mu + 3\lambda) \cdot \left(\dfrac{1}{R^3} - \dfrac{1}{r^3}\right)}, \quad B = \frac{P_1 - P_0}{4\mu \cdot \left(\dfrac{1}{R^3} - \dfrac{1}{r^3}\right)}, \quad \mu = \frac{E}{2 \cdot (1+v)}, \quad \lambda = \frac{E \cdot v}{(1+v) \cdot (1-2v)}.$$

where

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:

$$\sigma_{equiv}(\rho) = \sqrt{(\sigma_\rho(\rho))^2 + (\sigma_t(\rho))^2 - 2 \cdot \sigma_\rho(\rho) \cdot \sigma_t(\rho)}$$

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 $\delta = \dfrac{|w - w^*|}{|w|} \times 100\%$
1.4068E-003	1.4063E-003	0.03

Table 3.

Result «Equivalent stresses»

Numerical Solution Stress σ^*, $\dfrac{N}{m^2}$	Analytical Solution Stress σ, $\dfrac{N}{m^2}$	Error $\delta = \dfrac{|\sigma - \sigma^*|}{|\sigma|} \times 100\%$
1.1508E+009	1.1480E+009	0.24

Square Plate Subjected to Force at Center

Consider a rigidly supported plate subjected to a force applied at the center.

The study 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).
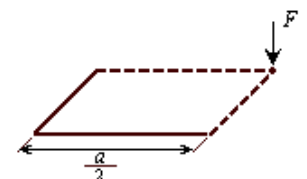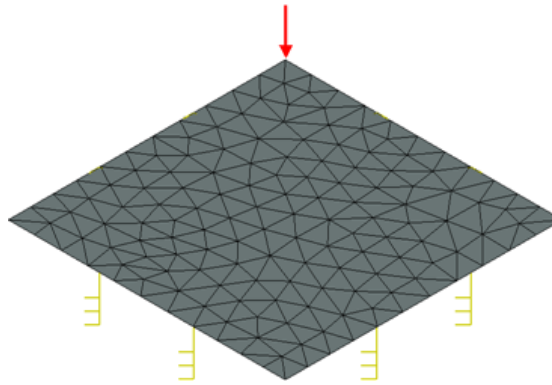
Finite element model of plate with loads and restraints

Analytical solution for the deflection at the point under the force is given by formula:

$$w = 0.0224 \frac{Pa^2}{D}, D = \frac{Eh^3}{12(1-v^2)}.$$

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: $E = 2.1 \times 10^{11} \ Pa$, $v = 0.28$.

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 $\delta = \frac{\|w - w^*\|}{\|w\|} \times 100\%$
1	5.3702E-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.
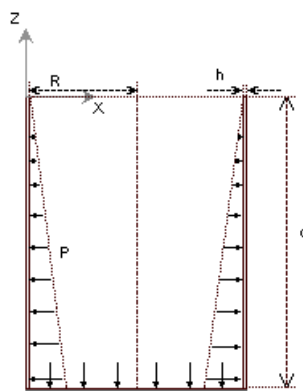
In most cases in practice the thickness of reservoir wall h is small compared to both the radius R, and the depth of the reservoir d. Taking into consideration this and the fact that the bottom of the reservoir does not experience any deformations, it is feasible to model the reservoir as the cylindrical shell whose bottom edge is clamped.
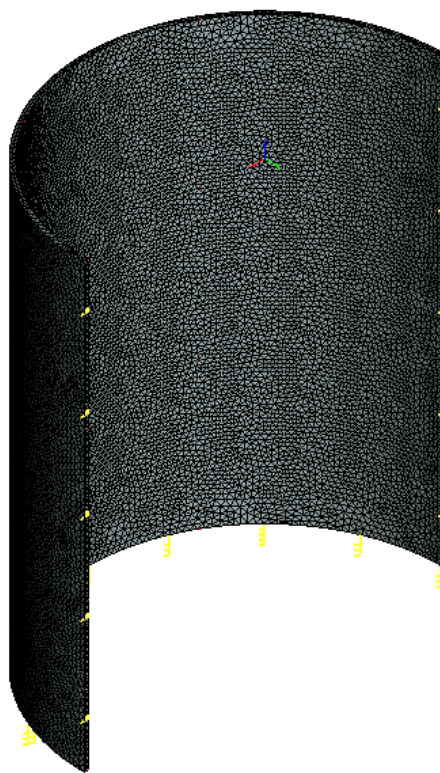
Finite element model of structure with loads and restraints

Analytical solution of the study 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:

$$\hat{z} = z + d, \quad C_1 = \frac{\rho g R^2 d}{Eh}, \quad C_2 = \frac{\rho g R^2}{Eh}\left(d - \frac{1}{\beta}\right), \quad \beta^4 = \frac{3(1 - v^2)}{R^2 h^2},$$

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: $E = 2.1 \times 10^{11} \, Pa$, $v = 0.28$.

Thus, $w = 6.1366 \times 10^{-7} \, m$ (maximum is at $\hat{z} = 0.134 \, m$ from the foundation – see the graph).
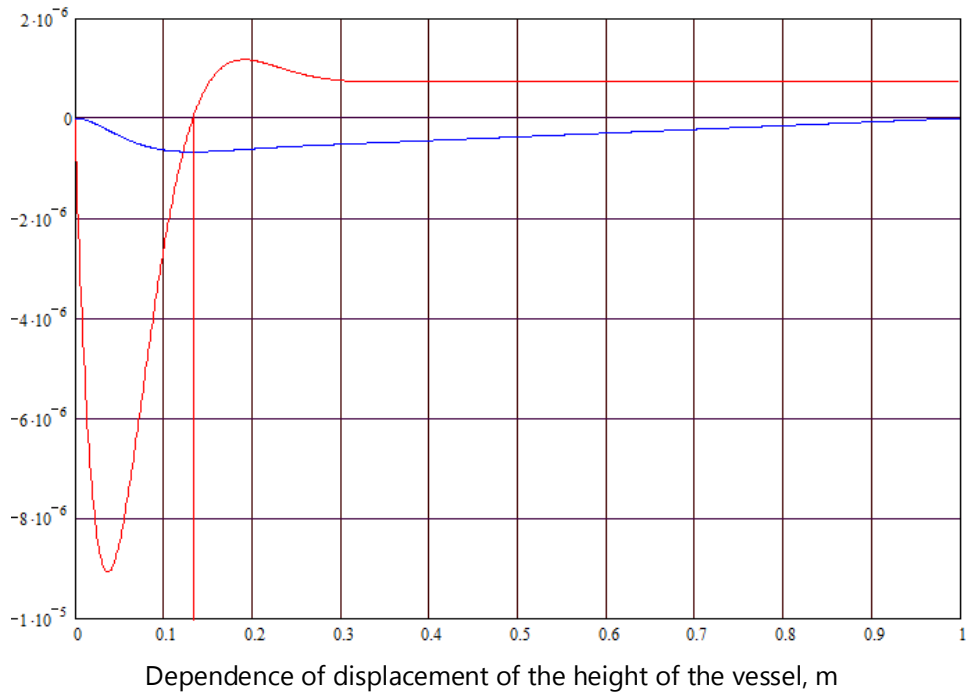
Dependence of displacement of the height of the vessel, m

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 $\delta = \dfrac{\left|w - w^*\right|}{\left|w\right|} \times 100\%$
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
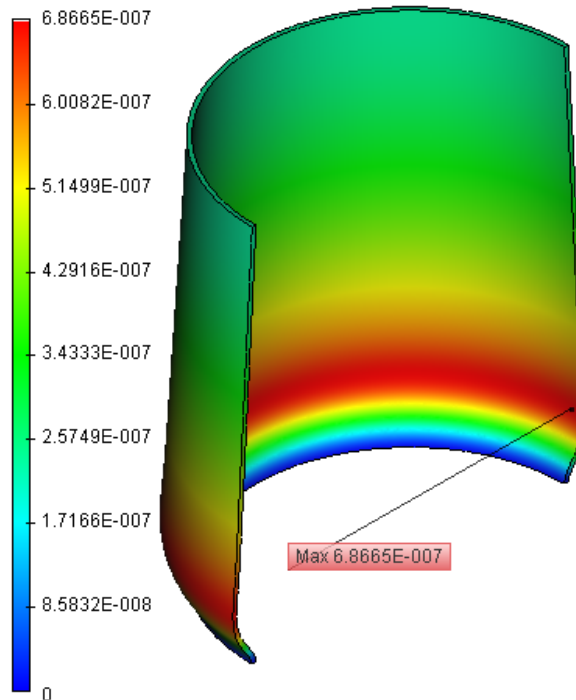
Conclusions:

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).
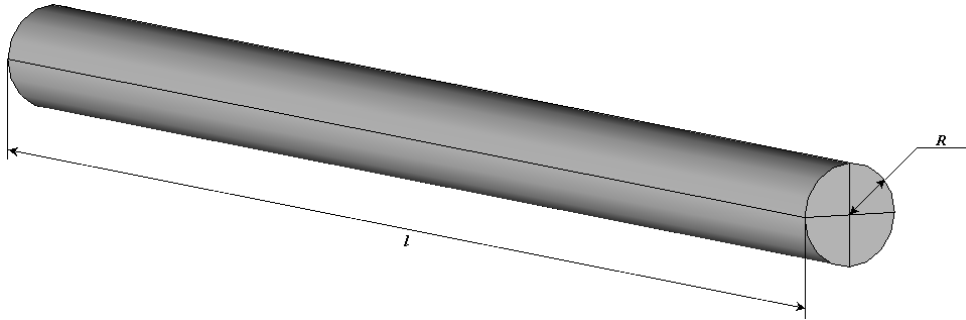
Select the coordinate system with the z-axis directed along the axis of the shaft, and the coordinate $z = 0$ located at the left edge of the shaft.

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.
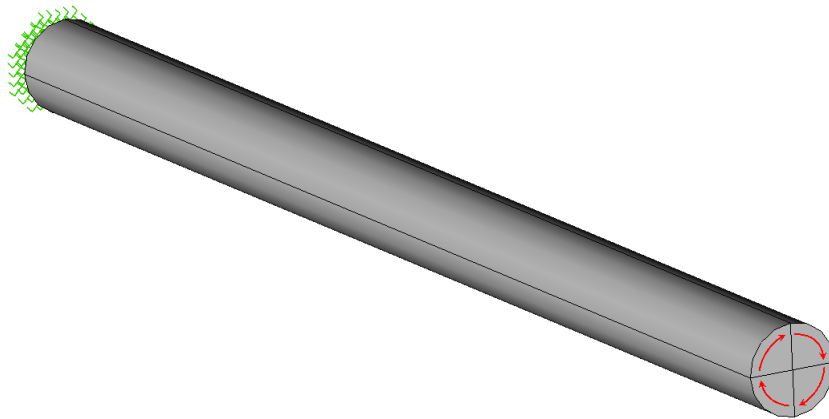
Let us use the following initial data: length l of the shaft is $0.6m$, radius of cross-section R of the shaft is $0.02m$, the magnitude of the applied torque is $\tau = 100\ N \cdot m$.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $v = 0.28$.

To find the angle of twist, let us use the following relation:

$$\varphi = \int\limits_{0}^{z} \frac{\tau}{GJ_p} dz + \varphi_0$$
,

where φ_0 – angle of twist of the cross-section $z = 0$, $G = \dfrac{E}{2(1+\nu)}$ – shear modulus, $J_p = \dfrac{\pi R^4}{2}$ – 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:

$$\varphi_{0.5l} = \frac{0.5\tau l}{GJ_p}$$

Thus, $\varphi_{0.5l} = 1.4551\times10^{-3}\,rad$.

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 study calculation*	*Number of Finite Elements*
quadratic tetrahedron (10 nodes)	*4509*	*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* $\delta = \dfrac{\lvert\varphi - \psi\rvert}{\lvert\varphi\rvert}\times100\%$
1.4547E-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 $1 м$, radius of cross-section of the bar R is equal to $0.02 m$.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $v = 0.28$, $\rho = 7800 \dfrac{kg}{m^3}$.

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 $\gamma = \rho \cdot g$, $g = 9.80665 \dfrac{m}{c^2}$.

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, $\Delta l = 1.8212 \times 10^{-7} m$; $\sigma = 3.8246 \times 10^4 \dfrac{N}{m^2}$ at $x = \dfrac{l}{2}$.

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 study 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 $\delta = \dfrac{\left|w - w^*\right|}{|w|} \times 100\%$
1.8177E-007	1.8212E-007	1.9410E-001

Table 3.

Result «Stress»

Numerical Solution Stress σ^*, $\dfrac{N}{m^2}$	Analytical Solution Stress σ, $\dfrac{N}{m^2}$	Error $\delta = \dfrac{\left|\sigma - \sigma^*\right|}{|\sigma|} \times 100\%$
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).
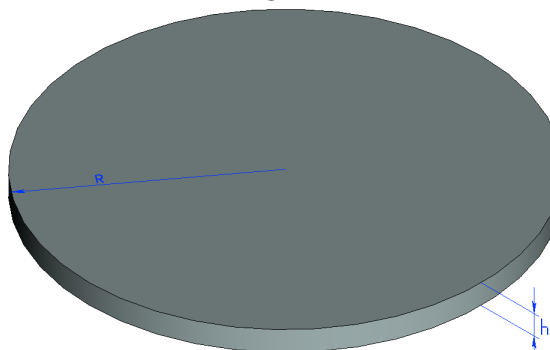
In analysis we consider only $^1/_4$ th part of the disc with application of symmetry conditions at the corresponding edges (restraint of displacements in the direction of the axis of the local coordinate system perpendicular to the plane of the edge).

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.
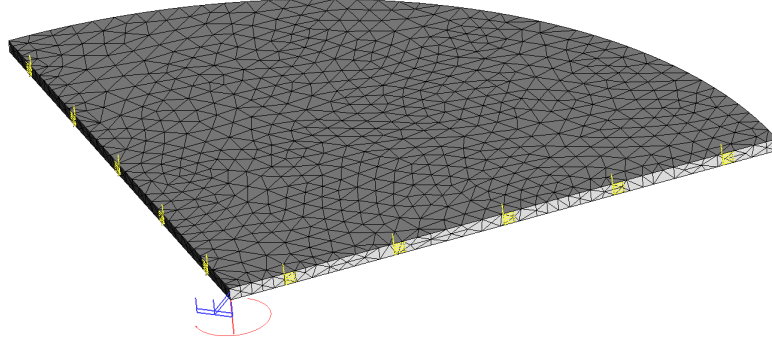
Let us use the following data: radius of disc R is equal to $0.457m$, thickness of disc h is equal to $0.01m$,

magnitude of angular velocity of rotation ω is equal to $300\dfrac{rad}{c}$.

Material characteristics: $E = 2.1 \times 10^{11} Pa$, $v = 0.28$, $\rho = 7800\dfrac{kg}{m^3}$

For this study, displacements u can be determined from the formula:

$$u = \frac{1}{E}\left((1-v)C_1 r - (1-v)C_2\frac{1}{r} - \frac{(1-v^2)}{8}\rho\omega^2 r^3 \right),$$

where constants $C_1 = \dfrac{3+v}{8}\rho\omega^2 R^2$, $C_2 = 0$ are determined from boundary conditions.

The maximum displacement u_{max} is expected to be at $r = R$, that is $u_{max} = \dfrac{(1-v)\rho\omega^2 R^3}{4E}$.

Stress components σ_r, σ_θ are found as:

$$\sigma_r = \frac{3+v}{8}\rho\omega^2\left(R^2 - r^2\right),$$

$$\sigma_\theta = \frac{3+v}{8}\rho\omega^2 R^2 - \frac{1+3v}{8}\rho\omega^2 r^2.$$

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, $u_{\max} = 5.7430\times10^{-5}m$, $\sigma_r = \sigma_\theta = \sigma = 6.0111\times10^7\dfrac{N}{m^2}$.

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 arguments	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 $\delta = \dfrac{\left|w - w^*\right|}{\left|w\right|}\times100\%$
5.8036E-005	*5.7430E-005*	*1.0390E+000*

Table 3.

Result «Stress»

Numerical Solution Stress σ^*, $\dfrac{H}{м^2}$	Analytical Solution Stress σ, $\dfrac{H}{м^2}$	Error $\delta = \dfrac{\left|\sigma - \sigma^*\right|}{\left|\sigma\right|}\times100\%$
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).
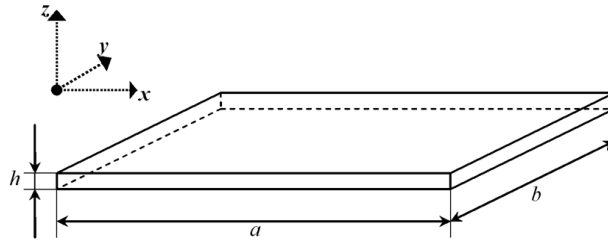
Thickness of plate h is considerably smaller than length of its sides a, b.

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 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=2.1×10^{11} Pa, $v = 0.28$.

Analytical solution of the study has the form:

$$w = \frac{q_0}{\pi^4 D\left(\dfrac{1}{a^2} + \dfrac{1}{b^2}\right)^2} \cdot \sin\left(\frac{\pi x}{a}\right)\cdot \sin\left(\frac{\pi y}{b}\right)$$

,

$$D = \frac{Eh^3}{12\left(1 - v^2\right)}$$

where – cylindrical stiffness of the plate.

The maximum deflection takes place at the center of the plate. Substituting $x = \dfrac{a}{2}$ and $y = \dfrac{b}{2}$ into the equation above we obtain:

$$w_{\max} = \frac{q_0}{\pi^4 D\left(\dfrac{1}{a^2} + \dfrac{1}{b^2}\right)^2}$$

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).
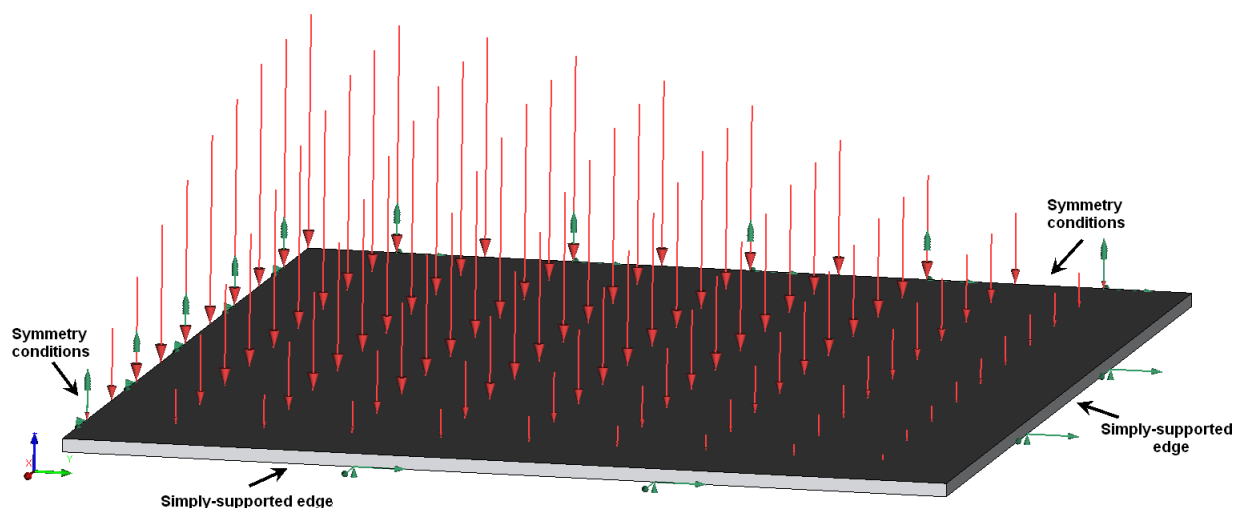
Model of plate with loads and restraints

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 arguments	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 $\delta = \dfrac{\left|w - w^*\right|}{\left|w\right|} \times 100\%$
1	1.8742E-005	1.9059E-005	1.671
2	1.8798E-005	1.9059E-005	1.37
3	1.8752E-005	1.9059E-005	1.61
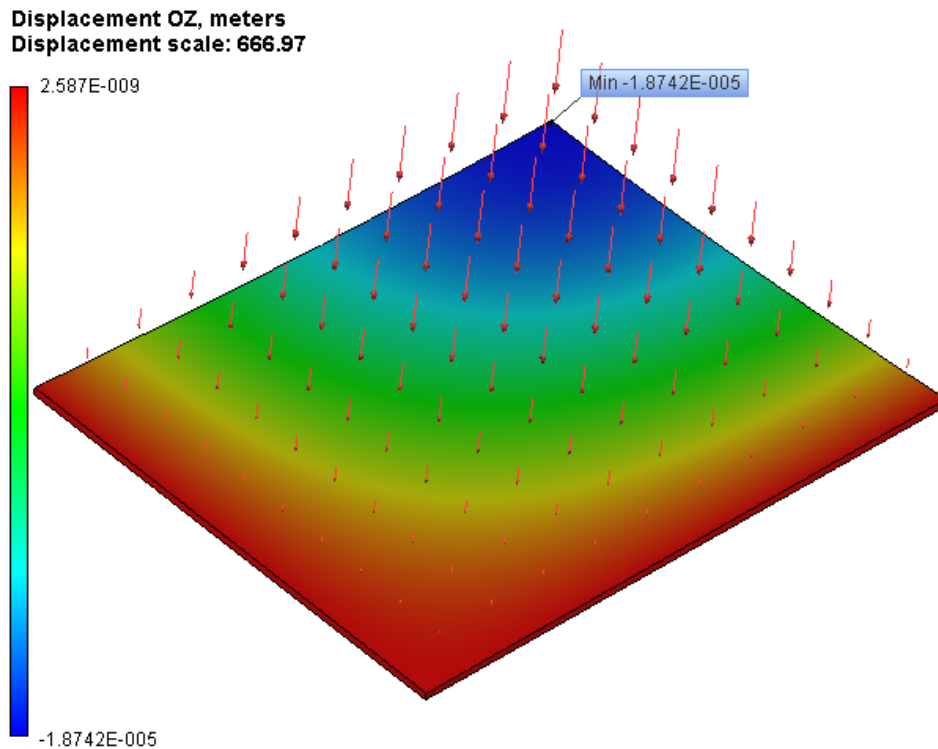
Conclusions:

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.
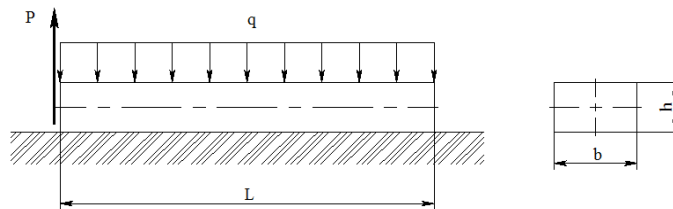
The beam is subjected to a uniformly distributed load q. In addition, the force P is applied at the left end-face of the beam.

Let us find the maximum displacement. Let us take the following input data: q_L = 200 N/m, P= 1000 N, L=5 m, b=0.05 m, h=0.02 m. Material properties (steel): E = 2.1E+011 Pa, ν = 0.28.
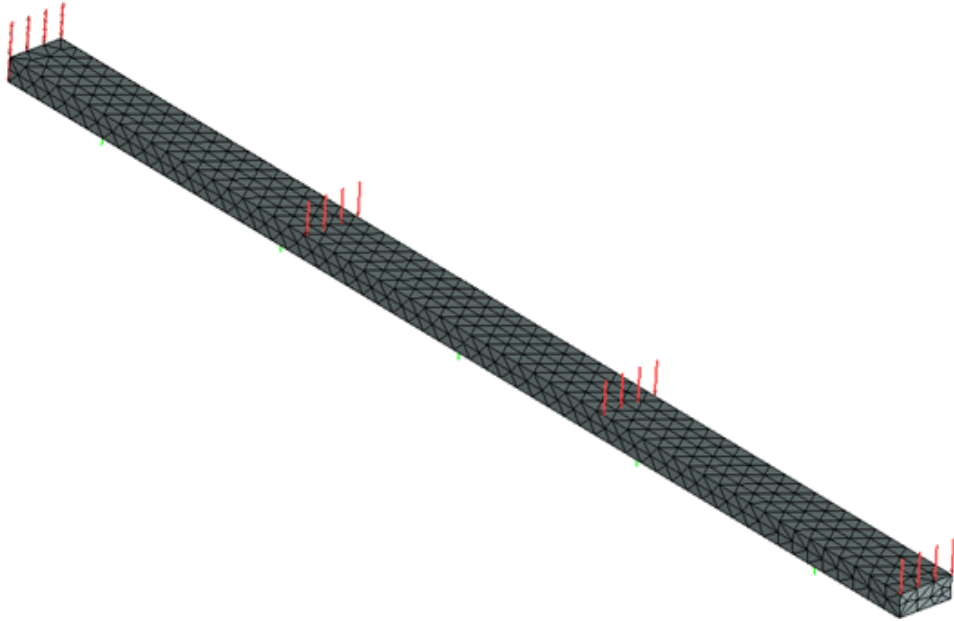
Finite-element model for specified load and constraints.

Analytical solution can be found by the formula:

$$w_0^{max} = (2\beta \cdot P - q_L) / k \cdot b,$$

where $J = bh^3 / 12$ – moment of inertia, k – stiffness coefficient of the supporting layer ($k= 3e+06$ N/m^3), $\beta = (k \cdot b/4E \cdot J)^{1/4} = 1.52136$. The edge effect can be observed up to a distance of $L_{кэ} \approx \pi/\beta = 1.381$ m measured from the left edge of the beam.

Therefore, $w_0^{max} = (2*1.52136 *1000-200)/(3*10^6*0.05) = 18.95147$ mm.

Deflection of the beam: $- q_L / k \cdot b = -1,333$ mm.

Before carrying out numerical calculations, let us determine the following quantities: area of the face over which the load q_L is distributed: $S=bL = 0.25$ m^2; therefore, pressure acting on this plane face is: $q= q_L*L/S=4000$ Pa. The input value of the total stiffness applied to the lower face is: $k_1=k*S=3*10^6*0.25= 750000$ 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** $\delta = 100\% \cdot |w^* - w| / |w|$
18.9523	18.9515	0.004

Results of calculations in T-FLEX Analysis (displacements along the Z-axis)
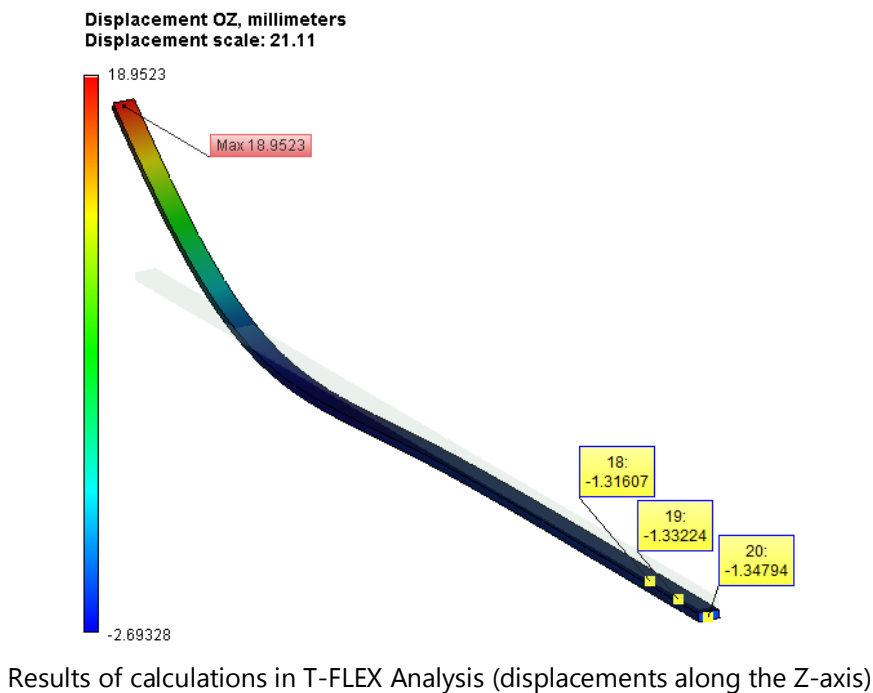
Plot of deflection of the semi-infinite beam (analytical solution)

Nº	X, i...	Y, meters
1	0	0.0189523
2	1	0.0111847
3	2	0.00500785
4	3	0.000872251
5	4	-0.00145842
6	5	-0.00247806
7	6	-0.00268996
8	7	-0.00249368
9	8	-0.00215626
10	9	-0.00182789
11	10	-0.00157402
12	11	-0.001408
13	12	-0.00131694
14	13	-0.00127898
15	14	-0.00127307
16	15	-0.00128318
17	16	-0.00129919
18	17	-0.00131607
19	18	-0.00133224
20	19	-0.00134794

Polyline (preview only), Nodes: 20   X = 19.4504, Y = -0.00182789
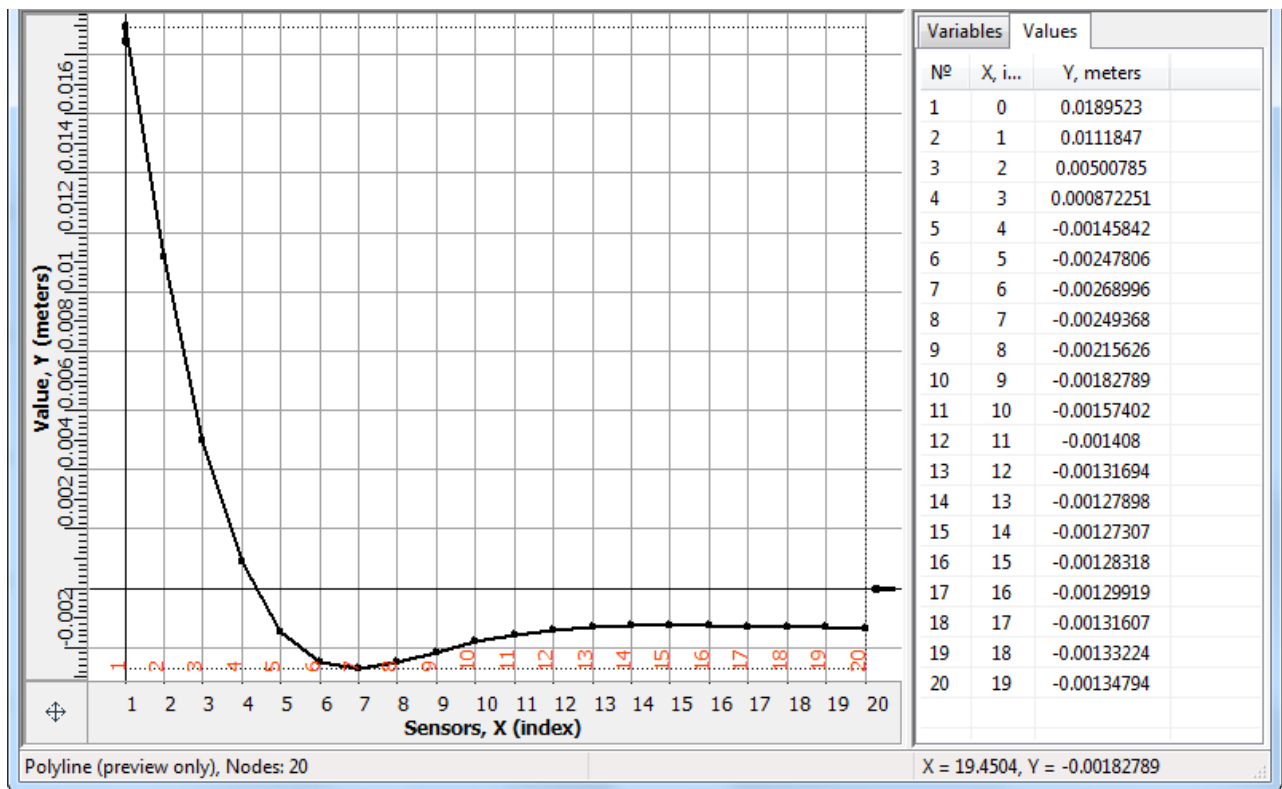
Plot of deflection of the semi-infinite beam (numerical solution)

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.
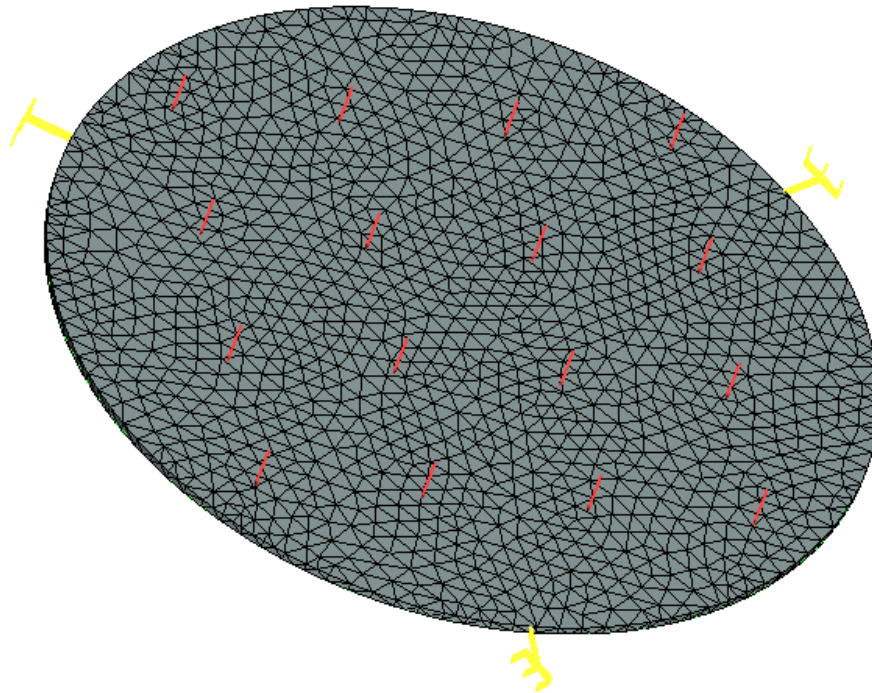
Finite element model for specified loads and constraints.

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 ν= 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 arguments	Number of finite elements
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

Displacement, magnitude, meters
Load factor: 1.00
Displacement scale: 10.90
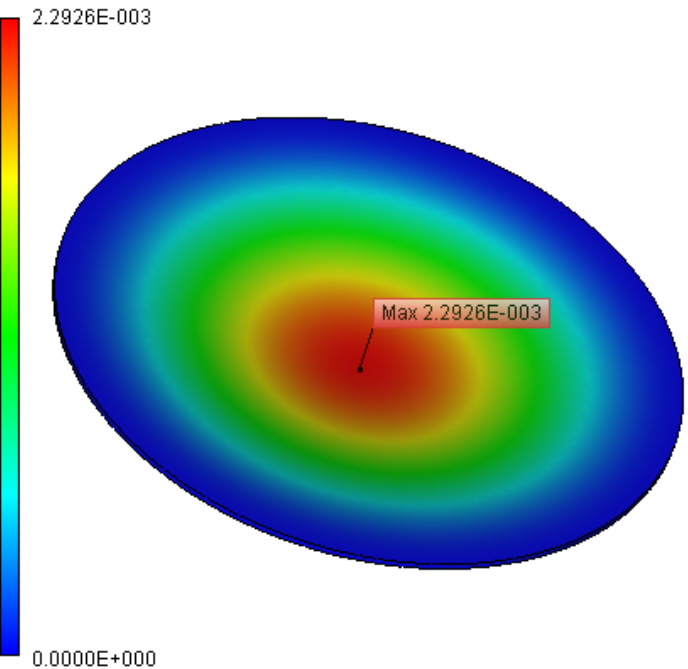
Conclusions:

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).
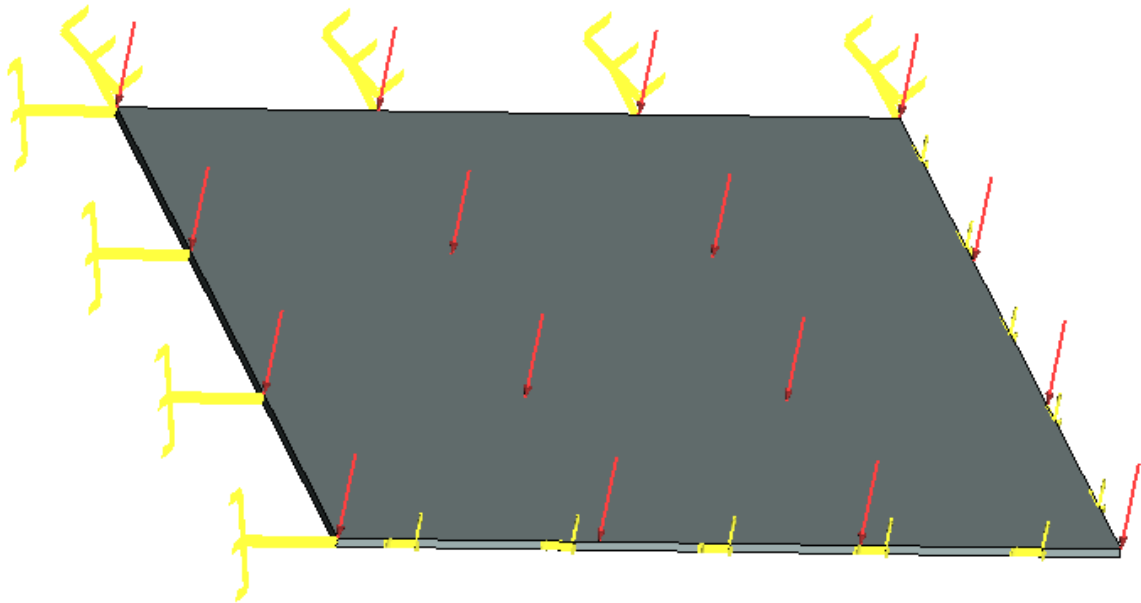
Finite element model with applied pressure and constraints

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, ν = 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

Displacement OZ, meters
Displacement scale: 203.68

2.4482E-010

Min -1.2274E-004

-1.2274E-004
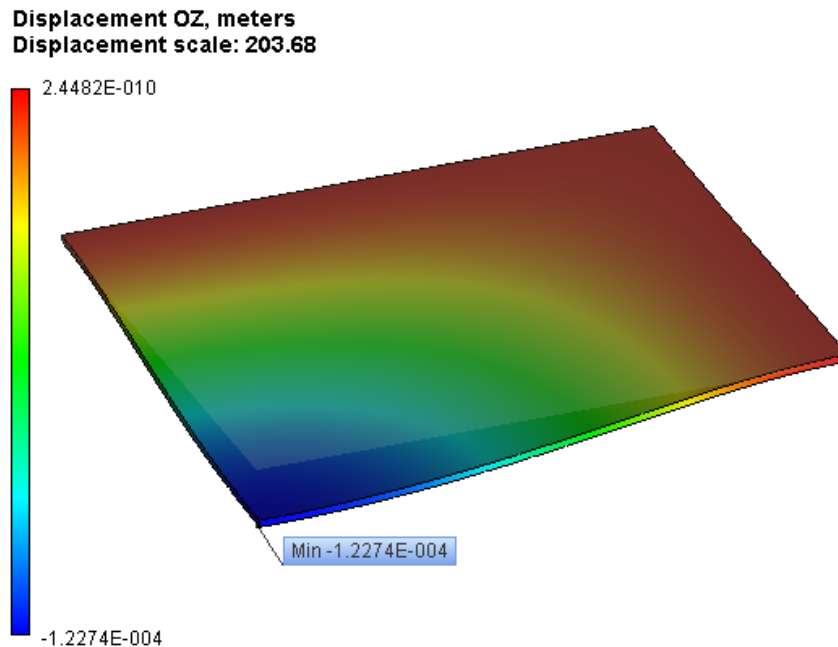
Conclusions:

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.
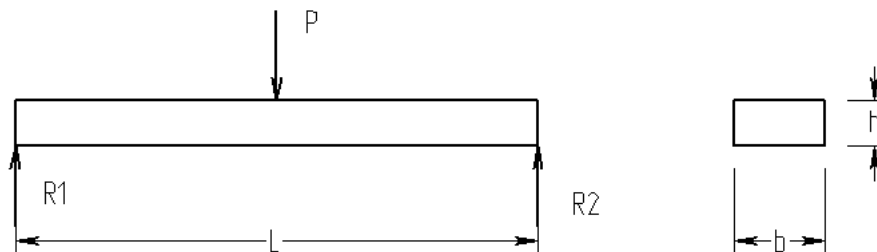
The value sought – maximum displacement of the beam.

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, $\boxed{\neq}$ 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.
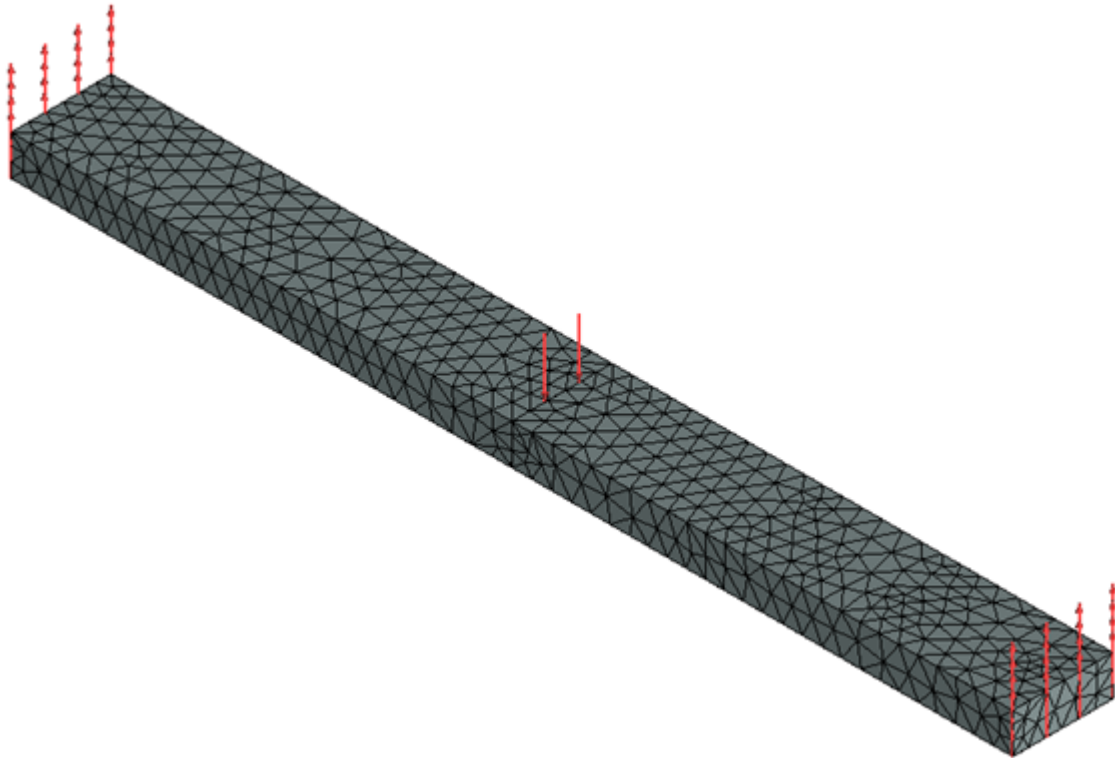
Finite element model for indicated loads and constraints.

Analytical solution is given by:

$$w = (P \cdot L^3) / (48 \cdot E \cdot J) = 3.720\text{E-}004 \text{ m}$$

where P – force, L – length of a beam, E – Young's modulus for the material, $J = b \cdot h^3 / 12$ – 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 arguments	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
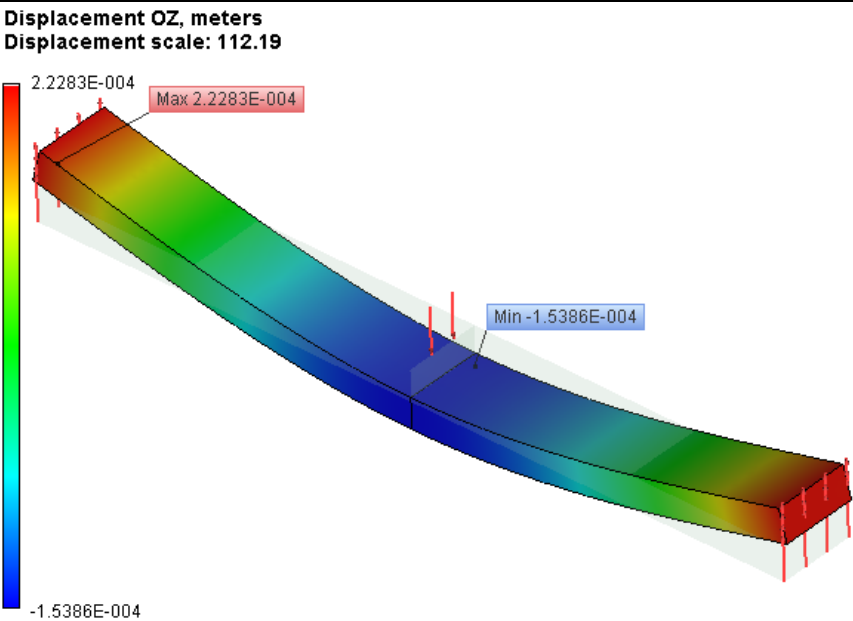
Conclusions: The relative error of the numerical solution compared to the analytical solution for displacements is equal to 1.3% for quadratic finite elements.Bending of a T-shape beam

Consider a T-shape beam.
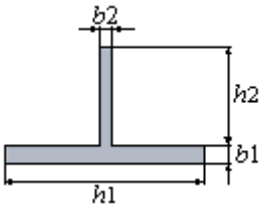
The length of the beam is equal to L. The beam is rigidly constrained at the left edge, and the force P is applied at the right edge.

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, ν = 0.28.

Finite element model for indicated loads and constraints.

Analytical solution of the study 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;

$$J_{Y1} = \frac{h_1 b_1^3}{12}, \quad J_{Y2} = \frac{b_2 h_2^3}{12};$$

$$A_1 = b_1 \cdot h_1, \quad A_2 = b_2 \cdot h_2;$$

z_{01}, z_{02} –distance between the axes Y_1 and Y, Y_2 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 arguments	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
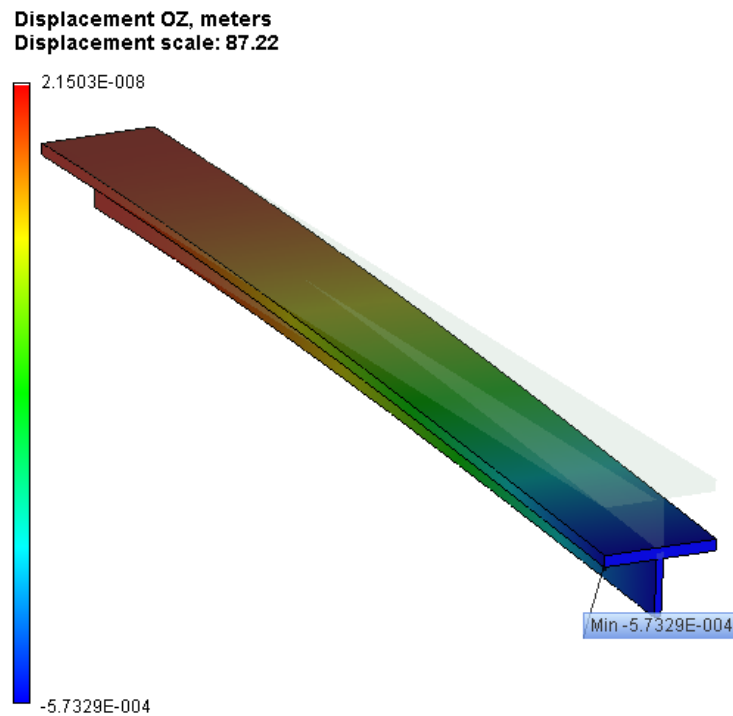
Displacement OZ, meters
Displacement scale: 87.22

2.1503E-008

Min -5.7329E-004

-5.7329E-004

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.
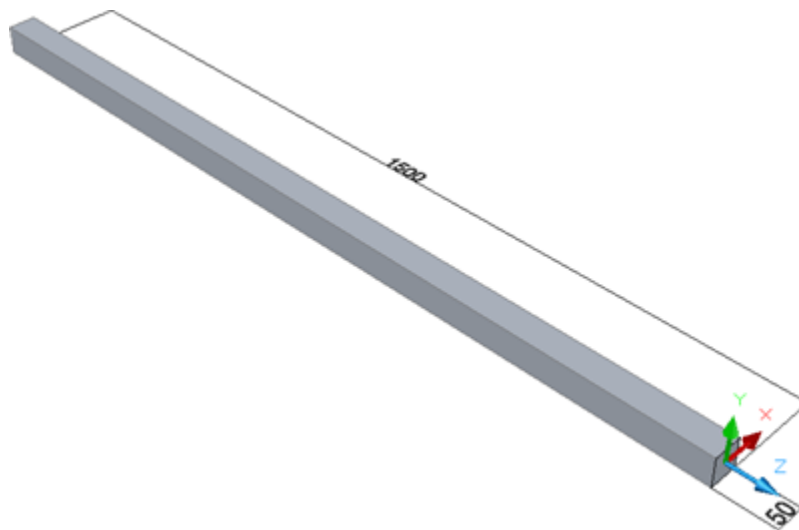
The beam is subjected to the torque M_t which is applied at the right edge perpendicularly. The left edge of the beam is rigidly fixed.

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 M_t is 1000 N-m.

Material's characteristics (steel AISI 1020): E = 2.0E+011 Pa, ν = 0.29.

In order to find the angle of twist, we use the following relationship:

$$\varphi = \frac{M_t L}{GJ_p},$$

where G=E/2(1+ν) – shear modulus, $J_p = \beta a^4$ –polar moment of inertia of a square section, β= 0.1406.

Therefore, φ= 2.2168E-002 rad.

The maximum deflection is calculated from the formula:

$$\Delta u = sin(\varphi) \cdot \frac{\sqrt{2}}{2} a$$

Therefore, Δu = 7.8371E-004 m.

The maximum shear stress τ can be obtained from the following formula:

$$\tau_{max} = \frac{M_t}{\alpha \cdot a^3}$$

where α= 0.208.

Therefore, τ= 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 arguments	Number of finite elements
quadratic tetrahedrons	91878	19876

Table 2. Result "Displacement"

Numerical solution Δu^*, m	Analytical solution Δu, m	Error $\delta = 100\% * \mid \Delta u^* - \Delta u \mid / \mid \Delta u \mid$
7.7647E-004	7.8371E-004	0.92

Table 3. Result "Shear stress"

Numerical solution $\square^*$, *Pa*	Analytical solution $\square$, *Pa*	Error $\delta = 100\% * \mid \square^* \square \mid / \mid \square \mid$
4.0591E+007	3.8462E+007	5.5

Displacement, magnitude, meters
Displacement scale: 136.60
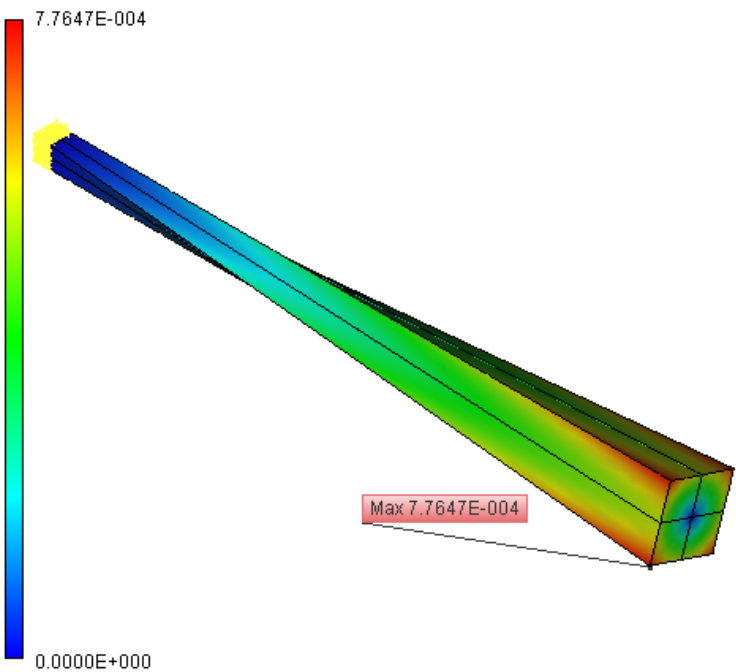
Conclusions: The relative error of the numerical solution compared to the analytical solution is equal to 0.92% for displacements and 5.5% for stresses when using quadratic finite elements.

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.
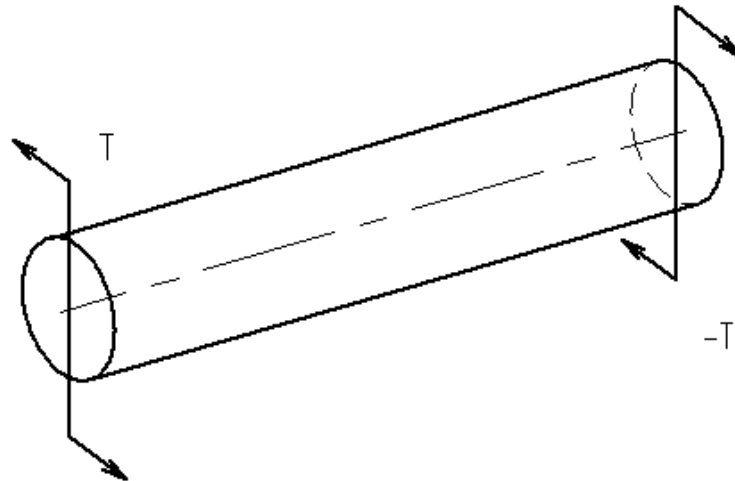
The sought value – maximum value of angle of twist.

Let us take T = 100 N*m, L = 0.5 m, d = 0.06 m.

Material's characteristics (steel): G = 8.203E+010 Pa, $\not= $ 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.
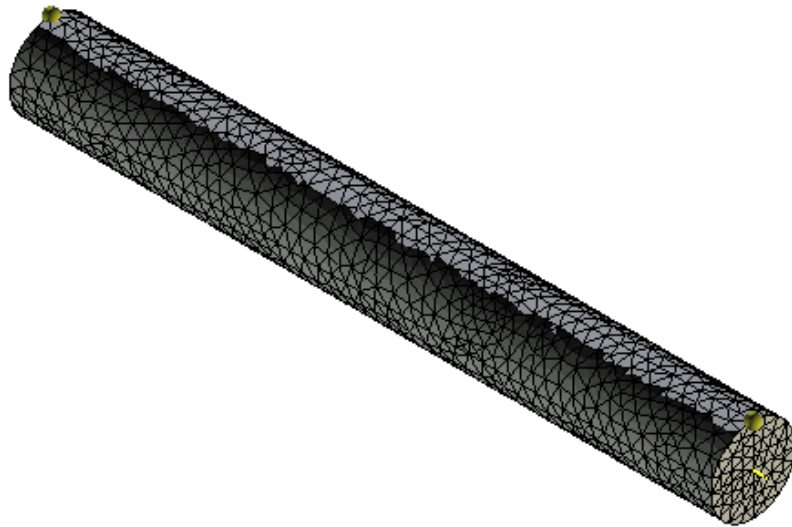
Finite element model for indicated loads and constraints.

Analytical solution is given by:

$\theta = (T*L)/(G*Jp) = 4.791E\text{-}004$ rad

$w = d*\sin(\theta/2) = 1.4372E\text{-}005$ 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,

$J_p = \pi d^4 / 32$ –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 arguments	Number of finite elements
quadratic tetrahedrons	69363	15348

Table 2. Result "Displacement"

Numerical solution w*, *m*	Analytical solution w, *m*	Error $\delta = 100\% * |w^* - w| / |w|$
1.4298E-005	1.4372E-005	0.513

Displacement, magnitude, meters
Displacement scale: 3482.33
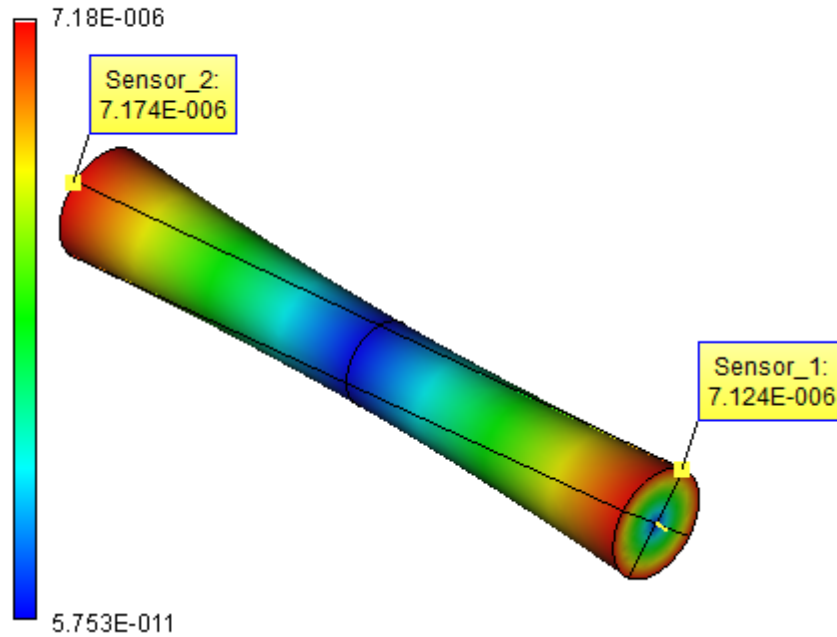
7.18E-006

Sensor_2:
7.174E-006

Sensor_1:
7.124E-006

5.753E-011

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.
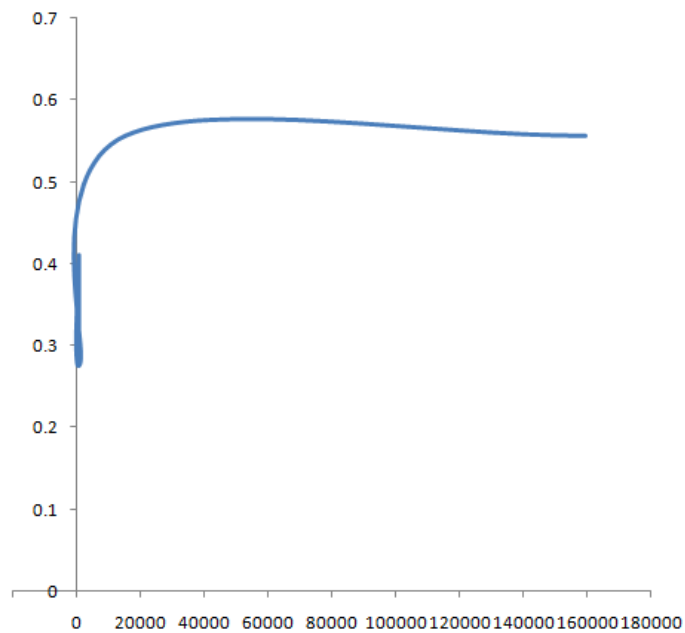
Dependence of relative error on the number of finite elements

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.
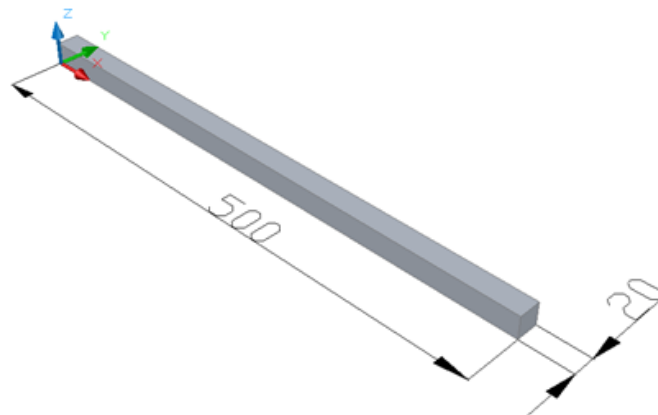
The sought value is the maximum displacement along the Z-coordinate.

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 ν = 0.28.
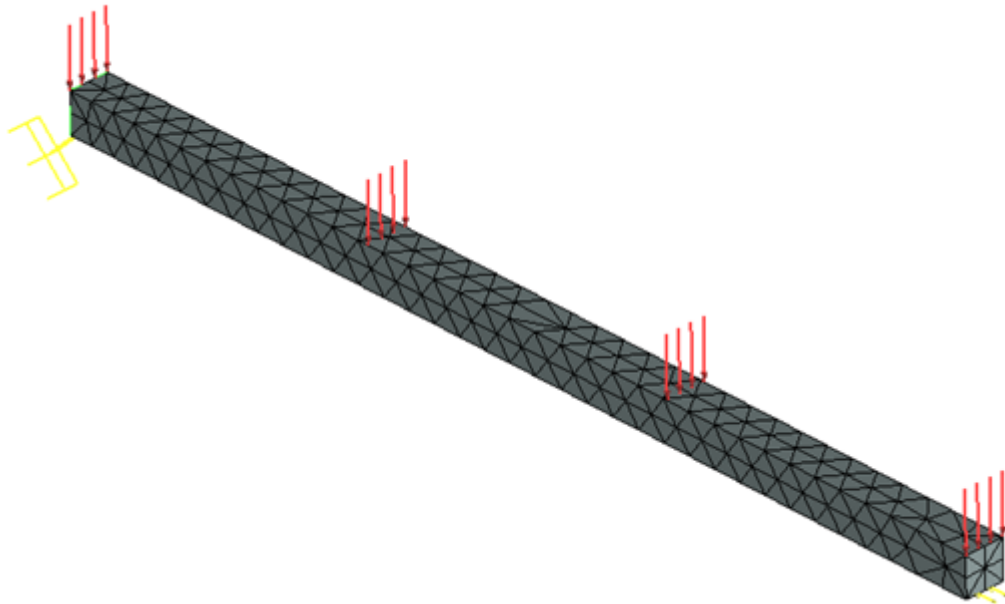
Finite element model for indicated loads and constraints

Analytical solution is calculated from the formula:

$$w = \frac{q \cdot a}{24\,EJ}\left(-x^4 + 2Lx^3 - L^3x\right),\ x \in [0, L]$$

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 $\mid w^* \mid, m$	Analytical solution $\mid w \mid, m$	Error δ =100%* $\mid w^* - w \mid / \mid w \mid$
1.752E-005	1.7439E-005	0.4

Displacement OZ, meters
Displacement scale: 1426.93

0.0000E+000

Min -1.7520E-005

-1.7520E-005
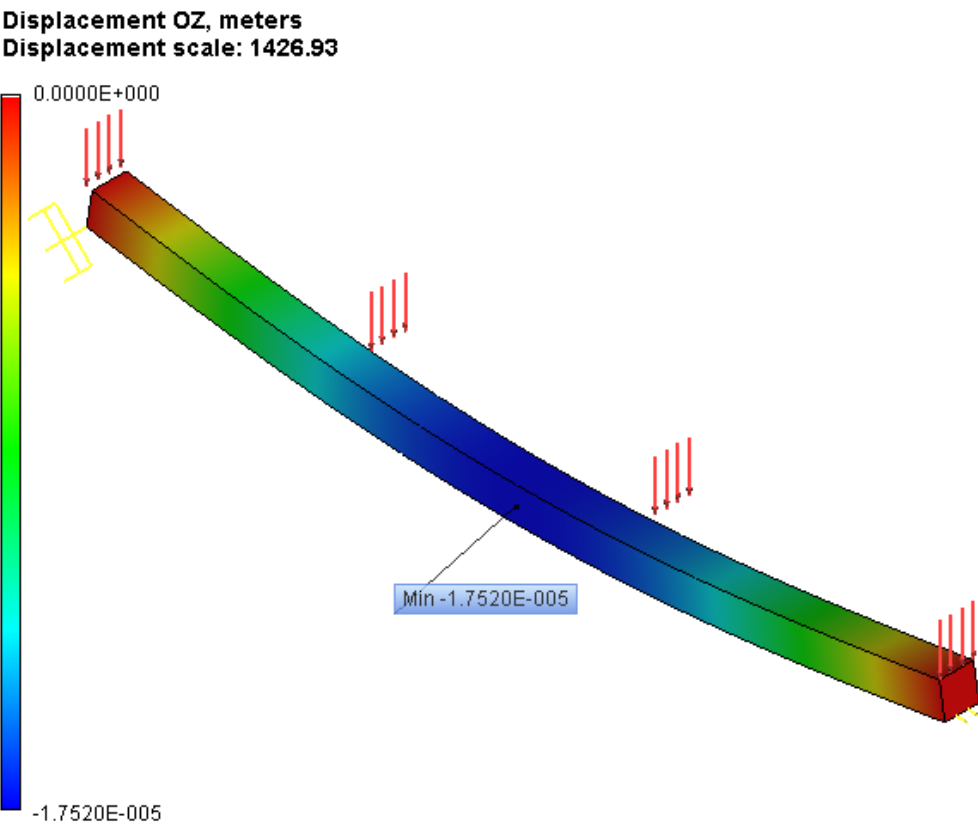
* Results of numerical solution depend on configuration of the mesh and can be somewhat different from those given in the table.

Deflection of a beam with the load

Let us consider a cantilever beam the right edge of which is loaded with the weight.
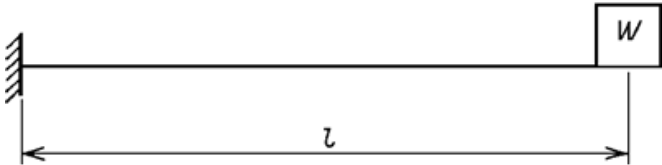
W

l

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*.

$m = \rho\, F$,

where $F = b$

h, - dimensions of the cross-section, ρ - density of the material of the beam.
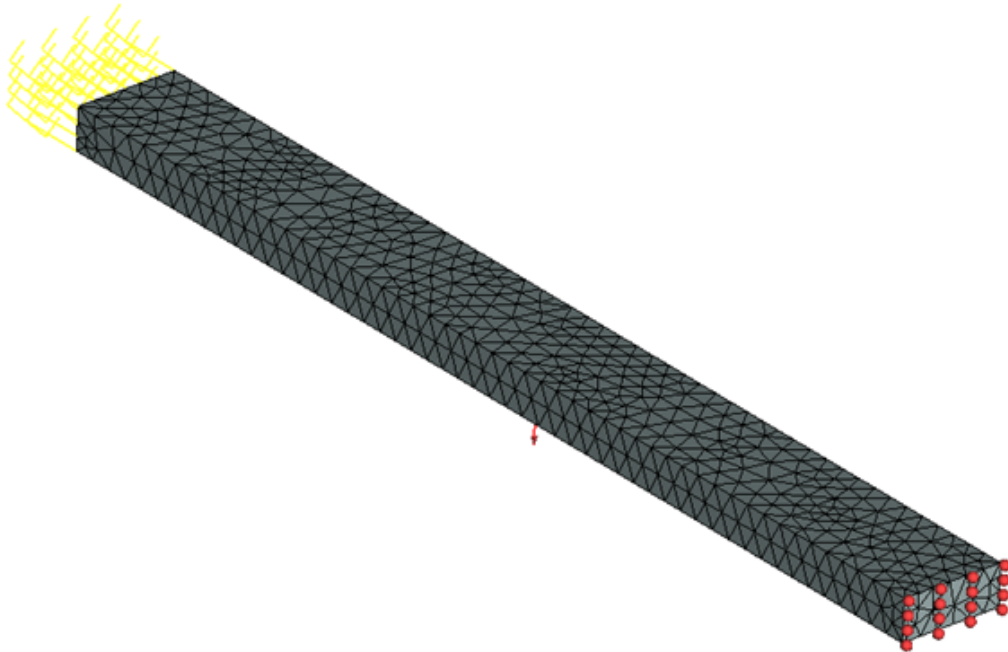
Finite element model for indicated loads and constraints.

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, ν=0.28, ρ = 7800 kg / m³.

Mass of the weight M is equal to 20mL kg (i.e., 78 kg).

Analytical solution can be obtained from formula:

$$\left.z\right|_{max} = \frac{gL^3}{3EJ}\left(M + \frac{33}{140}mL\right), \quad J = \frac{hb^3}{12}$$

.

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

Displacement OZ, meters
Displacement scale: 5.42
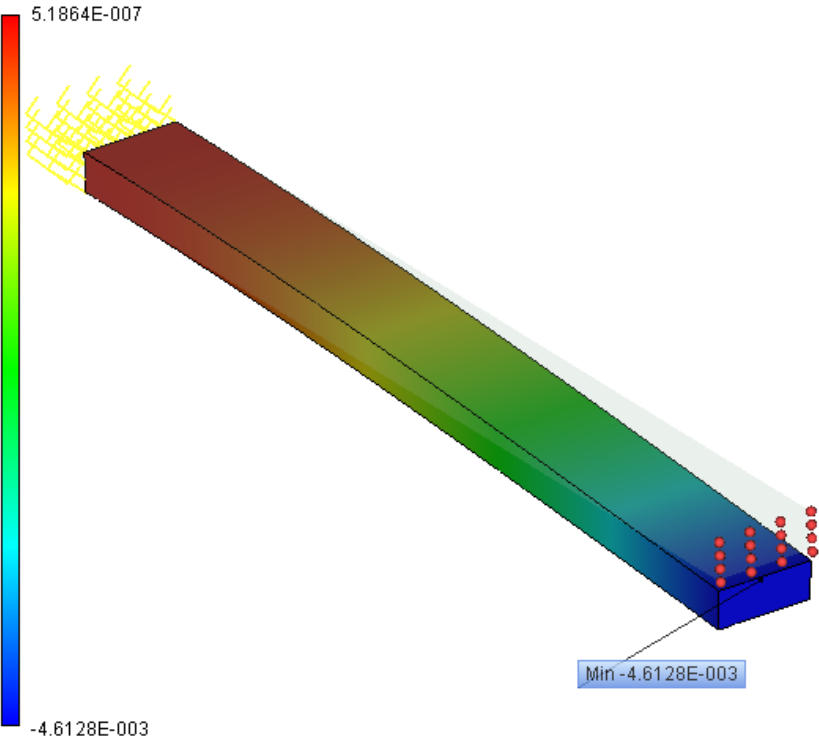
5.1864E-007

Min -4.6128E-003

-4.6128E-003

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.
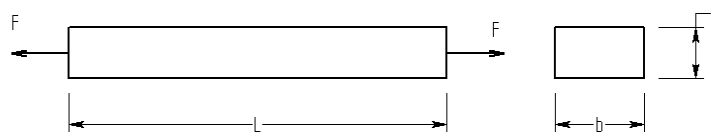
The sought value – maximum stretch.

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, � 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.
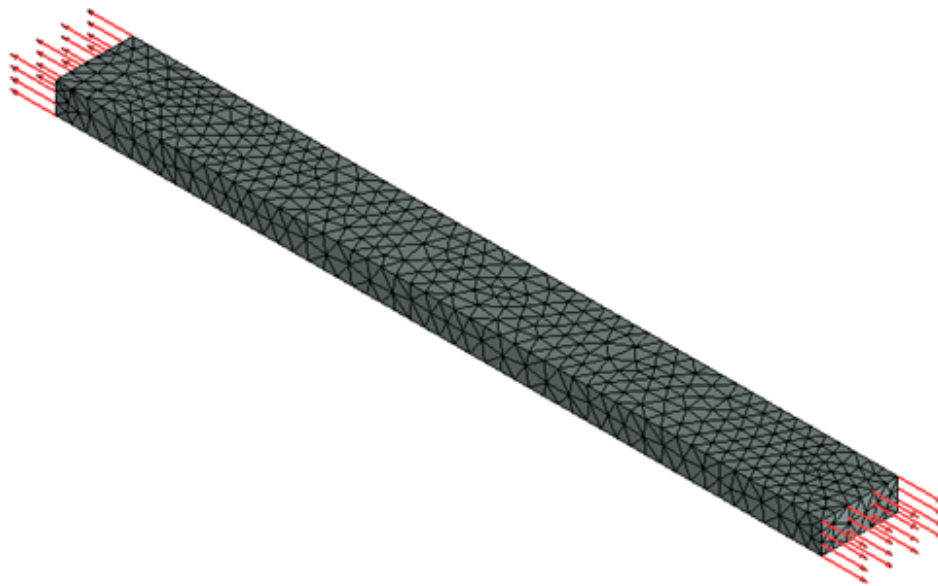
Finite element model for indicated loads and constraints.

Analytical solution is given by:

$$w = (F \cdot L) / (A \cdot E) = 2.381E\text{-}006 \text{ m}$$

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:

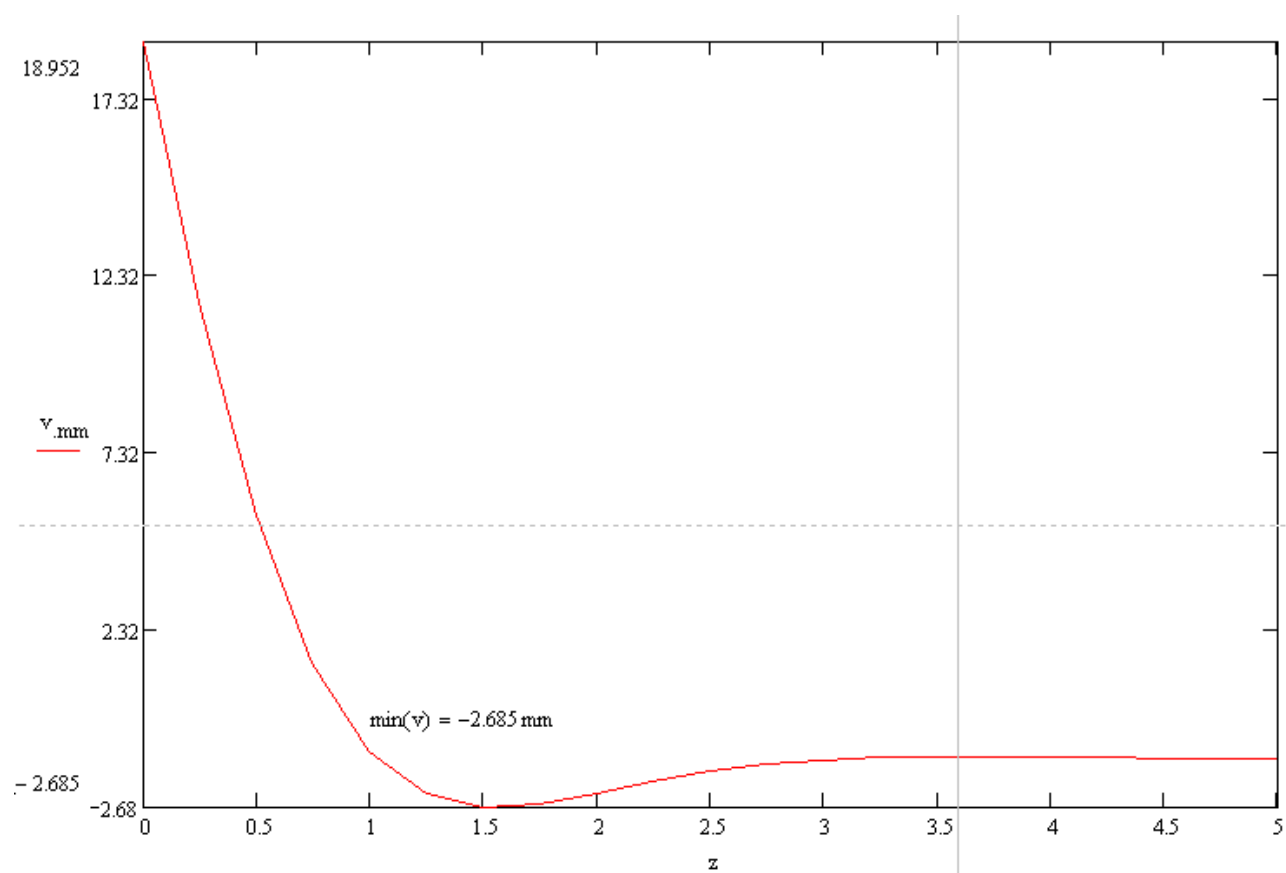(displacement is equal to (1.199E-006)+(1.182E-006)=2.381E-006 m)

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%* |0Y*\ - 0Y| / |0Y|
2.381E-004	2.381E-004	0.1
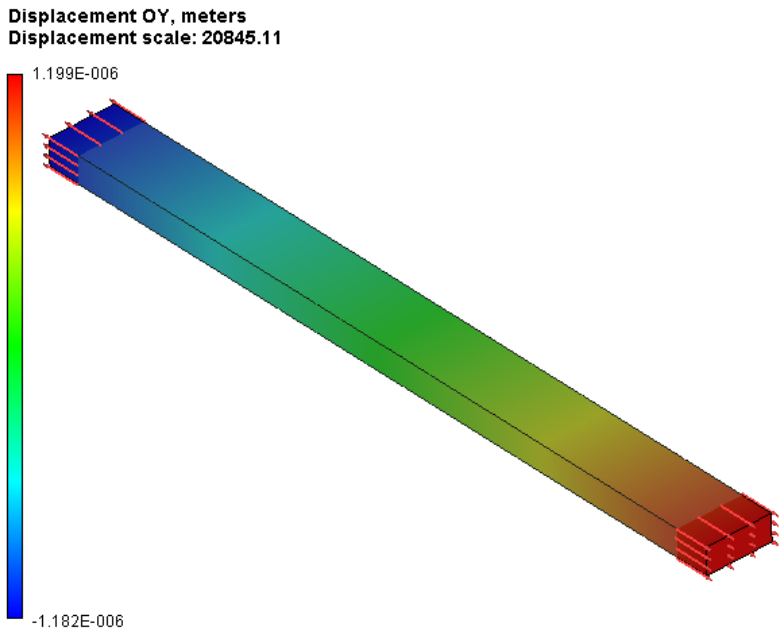
Displacement OY, meters
Displacement scale: 20845.11

1.199E-006

-1.182E-006

Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.1% for quadratic finite elements.

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 F_1, F_2 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 F_1=20000 N, F_2=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 E_1=5.59·1010 Pa, E_2=1.373·1010 Pa, E3=1.373·1010 Pa, shear modulus G_{12}=5.59·109 Pa, G_{23}=4.904·109 Pa, G_{31}=5.59·109 Pa, Poisson's ratios v_{12}=0.277, v_{23}=0.4, v_{31}=0.068. The principle axis of symmetry makes an angle α=45° with the horizontal axis.
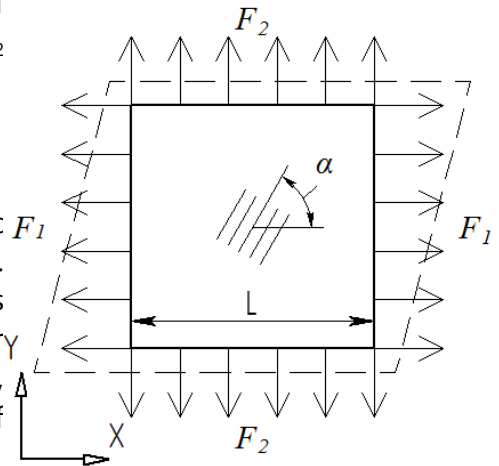
Let us place the plate in such a way that the principle axes of orthotropic symmetry coincide with the axes of the global coordinate system.

Apply partial constraint of the displacement along the Z-axis on the bottom face. To stabilize the model, let us apply the elastic foundation on the upper face of the plate with additional stiffness equal to 1 Н/м. We apply the normal loading of magnitude F1 on one pair of parallel lateral edges of the plate, and the normal loading of magnitude F_2 on another pair.
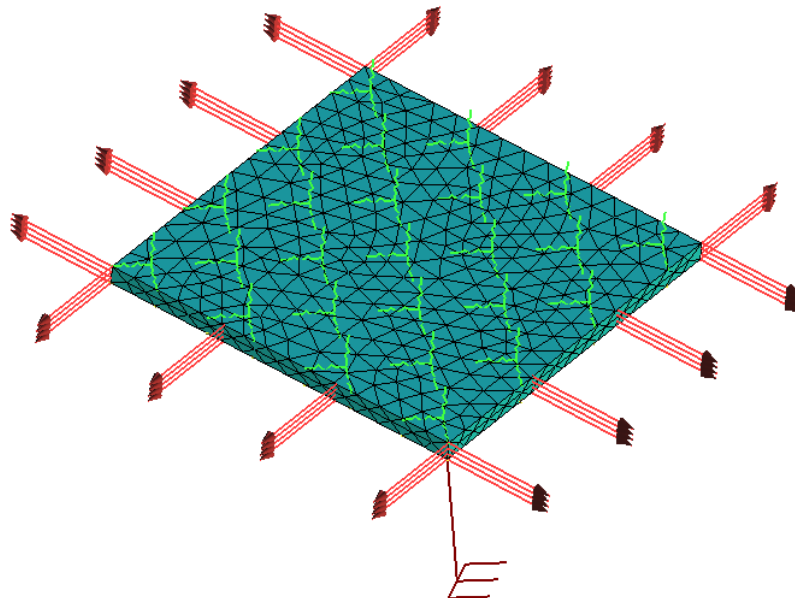
Finite element model of an orthotropic plate with the loads and constraints

With the help of the

«Analysis» command, let us carry out the static analysis of the plate. We obtain

the results in the for

m of deformations and stresses.
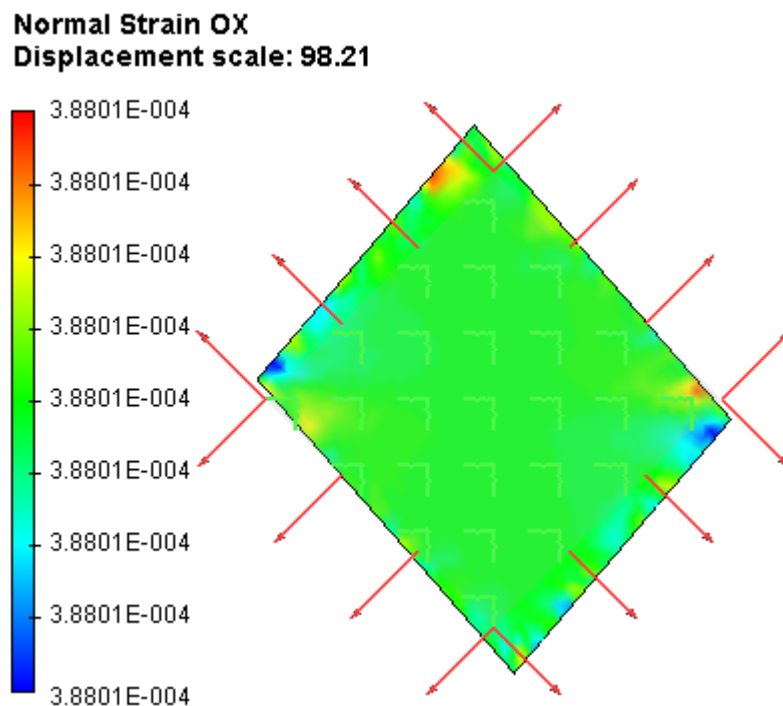
Deformations OX of an orthotropic plate

Normal Strain OY
Displacement scale: 98.21
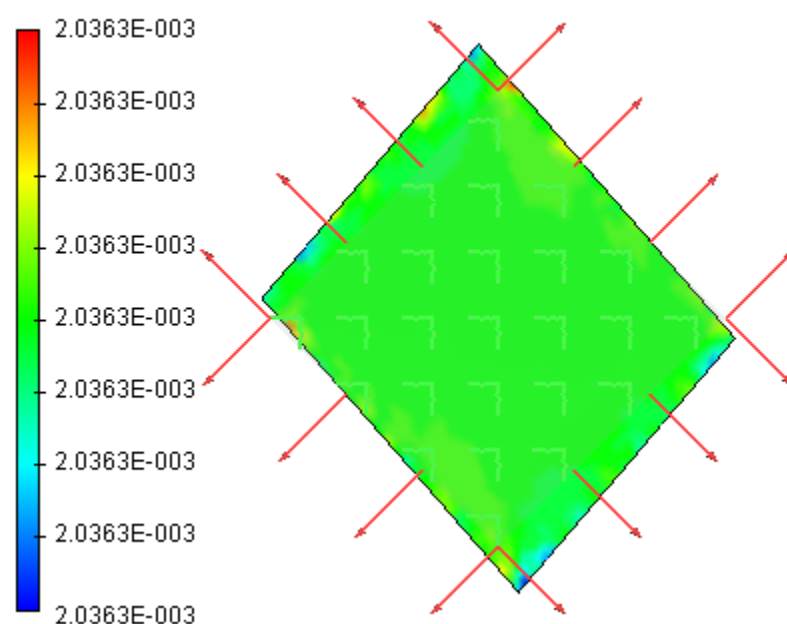
Deformations ОУ of an orthotropic plate

Normal Stress OX, N/m^2
Displacement scale: 98.21
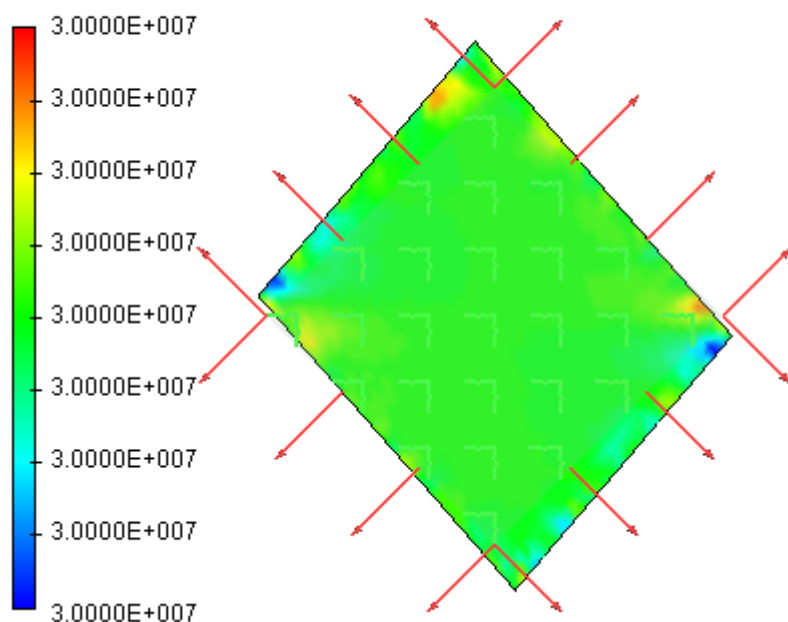
Stresses OX of an orthotropic plate

Normal Stress OY, N/m^2
Displacement scale: 98.21
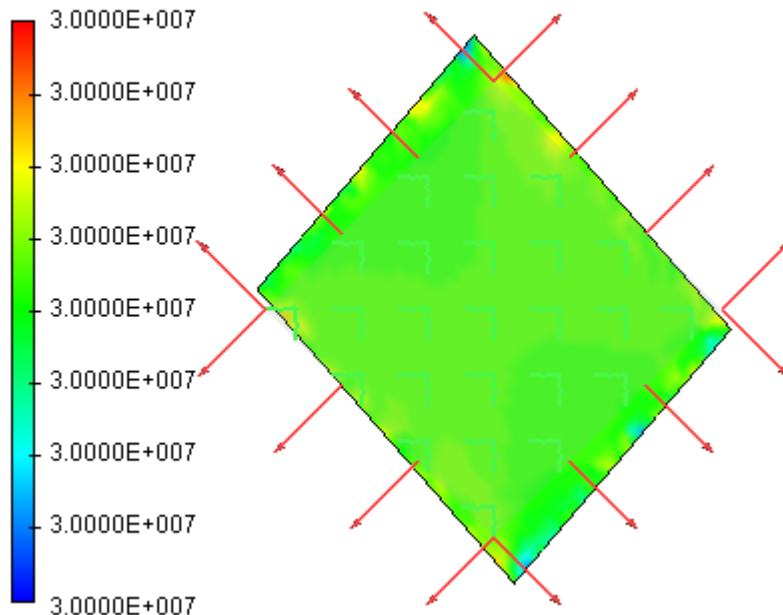
Stresses OY of an orthotropic plate

Main normal stress 1, N/m^2
Displacement scale: 98.21
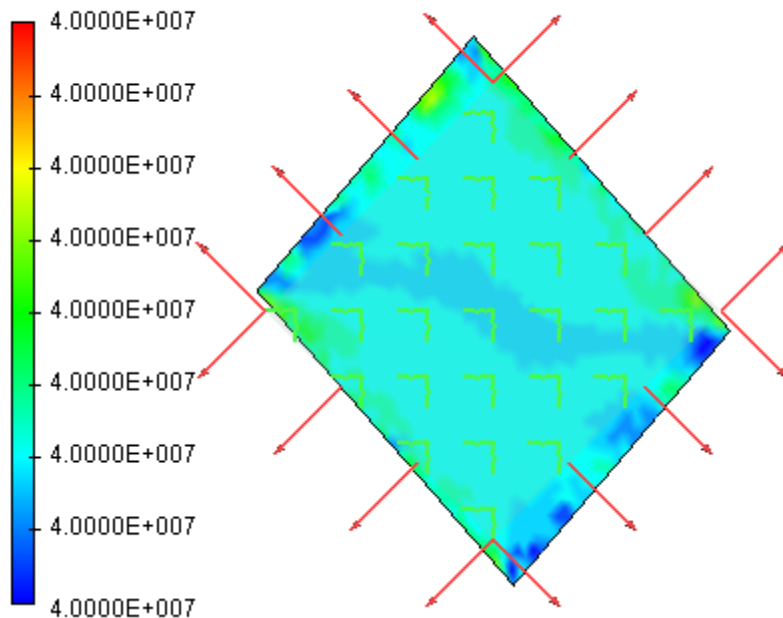
Principle stresses 1 of an orthotropic plate

Main normal stress 2, N/m^2
Displacement scale: 98.21

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

2.0000E+007

Principle stresses 2 of an orthotropic plate
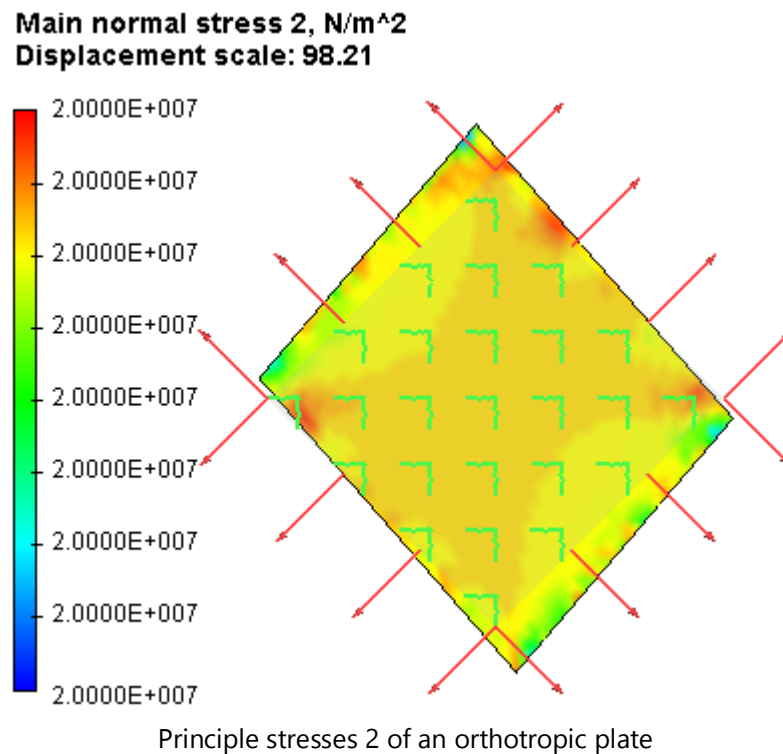
Conclusions:

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.
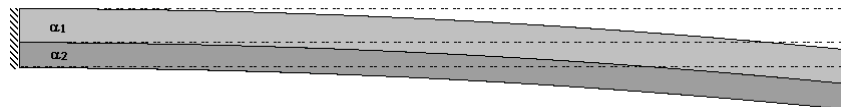
Let us compute how the curvature of the surface of seam of the bimetallic element changes depending on the temperature of heating.

Let the bimetallic element consist of plates having thicknesses $h_1 = 5\,mm$, $h_2 = 3\,mm$ and be made from different metals (aluminum $\alpha_1 = 2.4 \times 10^{-5}\,K^{-1}$, $E_1 = 6.9 \times 10^4\,N/mm^2$, $v_1 = 0.33$ and steel $\alpha_2 = 1.3 \times 10^{-5}\,K^{-1}$, $E_2 = 2.1 \times 10^5\,N/mm^2$, $v_2 = 0.28$). The plates have equal length $L = 250\,mm$ and width $b = 40\,mm$. The element is heated from $T_0 = 298.15\,K$ (or $T_0 = 25\,^0C$) to $T = 498,15\,K$ (or $T = 225\,^0C$).

Let $\dfrac{1}{\rho_0}$ – be the initial curvature of the seam's surface equal to 0; $\dfrac{1}{\rho}$ – the curvature of the seam's surface after heating. The change of the curvature can be found with the help of the following formula:

$$\frac{1}{\rho} - \frac{1}{\rho_0} = \frac{6(T - T_0)(\alpha_1 - \alpha_2)}{\left(\dfrac{\left(E_1 h_1^2 - E_2 h_2^2\right)^2}{E_1 E_2 h_1 h_2 (h_1 + h_2)} + 4(h_1 + h_2) \right)}.$$

The curvature of the seam's surface after heating of the given bimetallic element

$$\frac{1}{\rho} = 4{,}123 \times 10^{-4}; \quad \rho = 2425.43 \ mm.$$

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 study calculation	Number of finite elements
quadratic tetrahedron (10 nodes)	7932	165231	35667

Table 2.

Result «Curvature of seam's surface»

Surface Sij of separation of plates i and j	Numerical solution Curvature $\dfrac{1}{\rho_n}$, mm^{-1}	Analytical solution Curvature $\dfrac{1}{\rho_n}$, mm^{-1}	Error $\delta = \dfrac{\left\lvert \dfrac{1}{\rho} - \dfrac{1}{\rho_n} \right\rvert}{\left(\dfrac{1}{\rho} \right)} \times 100\%$
S12	125E-004	123E-004	0,049

Displacement OZ, meters
Displacement scale: 0.93

2.4017E-005

Sensor_9:
-3.4965E-003

Sen
-3.4149E-003

Ser
-3.4897E-003

Sensor_10:
-1.3390E-002

Sen
-1.3309E-002

Ser
-1.3384E-002

-1.3390E-002

250

5

3

R_{mean}=2424.012

K_{mean}=4.125*10^{-4}

R2435,04
K_1=4.106709x10^{-4}

R2412,98
K_2=4.144246x10^{-4}

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, v = 0.28, coefficient of thermal linear expansion α = 1.3E-005 K^{-1}.

The change of temperature ΔT is equal to 100°.
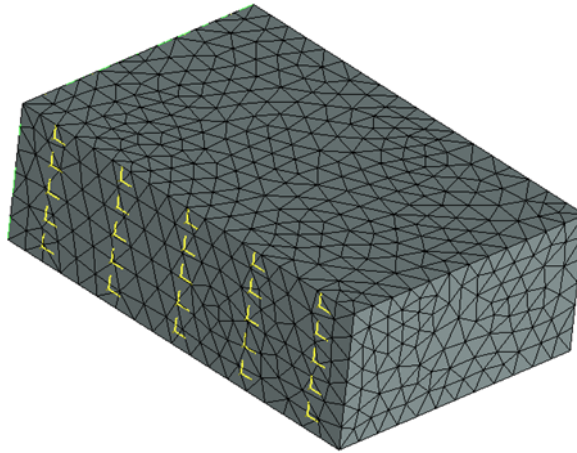
Analytical solution is calculated from the formulas:

$\Delta x = \alpha\ L\ \Delta T$

$\Delta y = \alpha\ b\ \Delta T$

$\Delta z = \alpha\ h\ \Delta T$

Therefore,

Δx = 2.60000000E-004 m

Δy = 3.90000000E-004 m

Δz = 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, *m*	Analytical solution, *m*	Error $\delta =100\%*|\Delta^* - \Delta |/| \Delta |$
2.60000001E-004	2.60000000E-004	0.27E-007
3.90000007E-004	3.90000000E-004	0.26E-007
1.300000004E-004	1.30000000E-004	0.30E-007
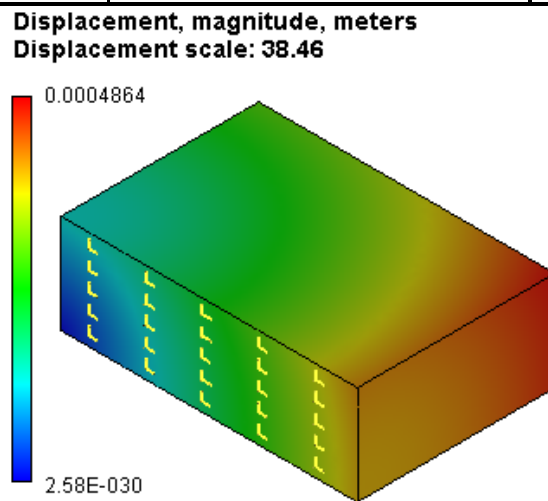
Displacement, magnitude, meters
Displacement scale: 38.46
0.0004864
2.58E-030

Conclusions:

The relative error of the numerical solution compared to the analytical solution for displacements is equal to 0.001% for quadratic finite elements.

EXAMPLE OF SOLVING STUDIES 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).
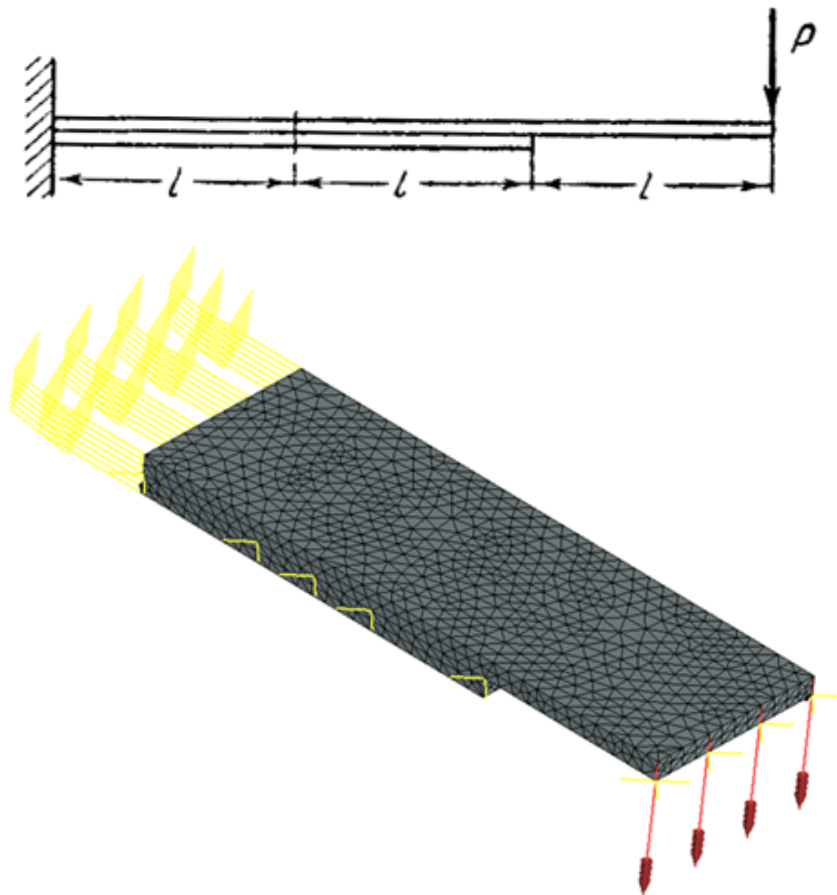
Finite element model for indicated loads and constraints.

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, ν= 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*h^3 / 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 δ=100%*|*Δz - Δz**|/|*Δz*|
5.4829E-004	5.6190E-004	2.42
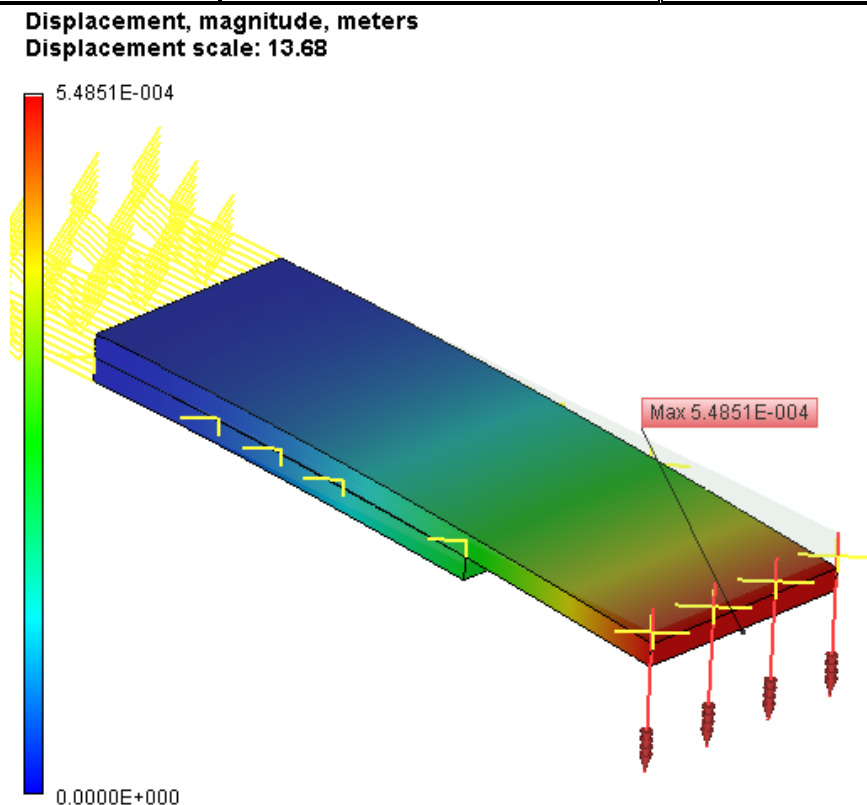
Conclusions: The relative error of the numerical solution compared to the analytical solution is equal to 2.4% for quadratic finite elements.

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.
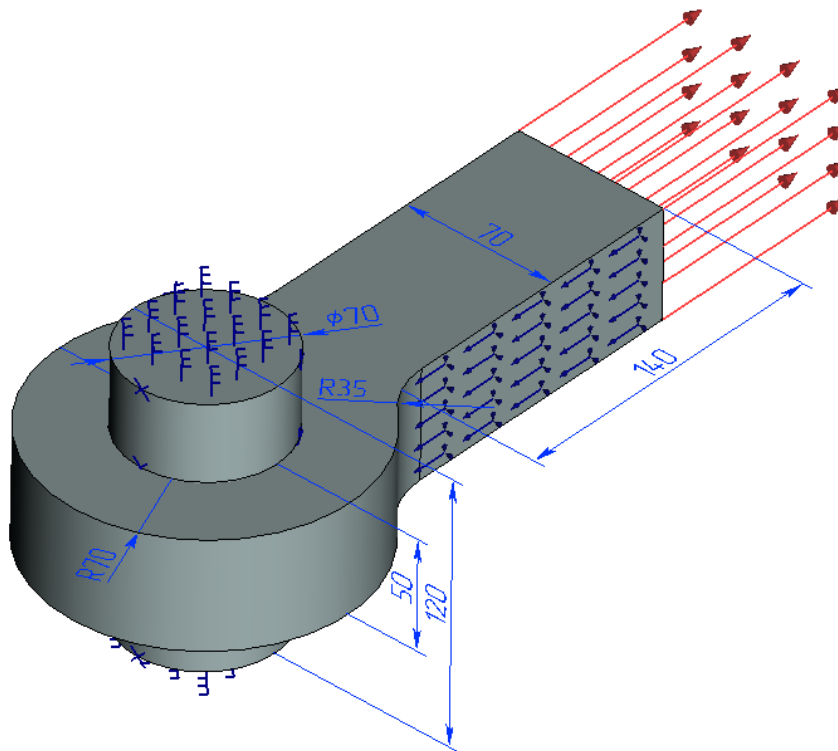
Let us take the following input data: diameter of the axis d = 30 mm, length of the axis H = 40 mm, external radius of the sleeve R = 25 mm, thickness of the sleeve h = 10 mm, width of the sleeve's bar b = 20 mm, length of the sleeve's bar b_1 = 40 мм, radius of rounding of the sleeve is r_1= 10 mm.

Material's characteristics: $E = 2.1 \times 10^{11}$ Pa, $v = 0.28$.

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 $\sigma = 1.029 \cdot 10^8$ 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 σ_1), we obtain the following results: the value of $\sigma_1 = 1.024 \cdot 10^8$ N/m^2.
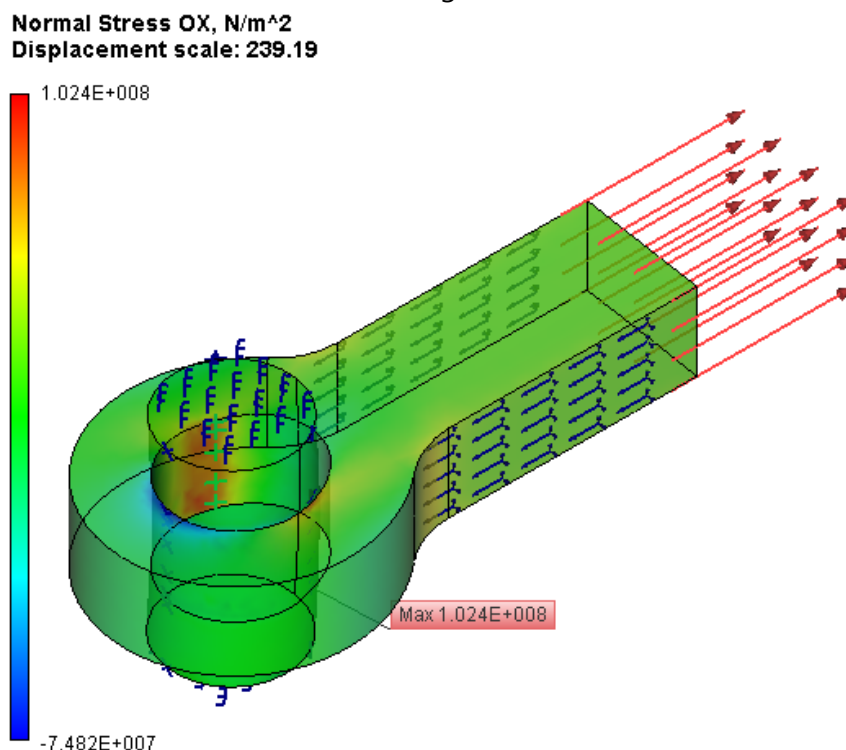
Results are shown in the table.

Table 1. Parameters of finite element mesh

Finite element type	Number of vertices	Number of finite elements
Quadratic tetrahedrons	2241	10368

Table 2.

Result of calculations, Stress σ, N/m2

Numerical solution, w^*	Analytical solution, w, m	Error $\delta_u = \dfrac{\left| w - w^* \right|}{w} \times 100\%$
$1.024 \cdot 10^8$	$1.223 \cdot 10^8$	19 %

EXAMPLES OF SOLVING BUCKLING STUDIES

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 study). 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.
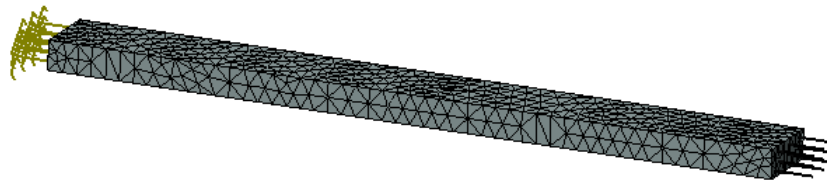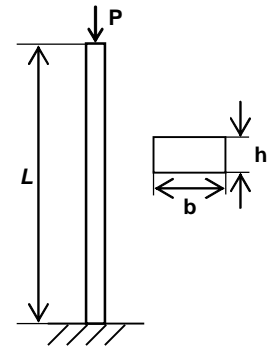
Finite element model of the beam for buckling analysis

The analytical solution to determine the critical load appears as:

$$P_{krit} = \frac{\pi^2 EJ}{(\mu l)^2} = 6.9087 \times 10^4 \ N$$

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, $\mu=2$.

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.

Numerical solution Critical load σ *krit*, $\dfrac{N}{m^2}$	Analytical solution Critical load p *krit*, $\dfrac{N}{m^2}$	Error $\quad \delta = \dfrac{\left| p_{krit} - \sigma_{krit} \right|}{\left| p_{krit} \right|} \times 100\%$
6.9341E+004	6.9087E+004	0.37
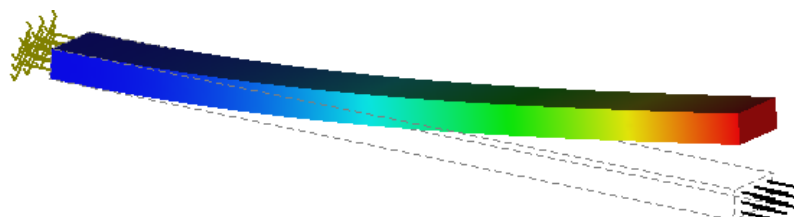
First buckling mode of the beam

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).
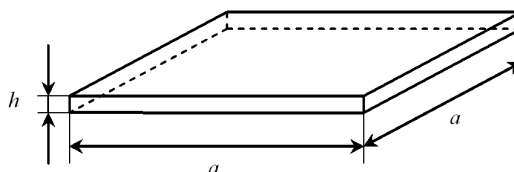
The thickness of plate h is much smaller than the length of its side a.

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.

clamped edge
simply-supported edge
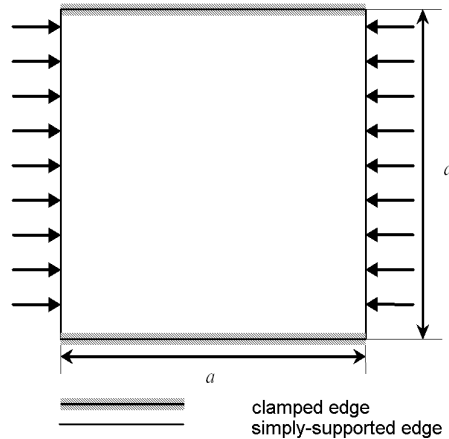
Let us use the following data: plate side length $a = 500mm$, thickness of plate $h = 3mm$, applied distributed force $P = 1\dfrac{N}{m^2}$.

Elastic properties are taken as: $E = 2.1 \times 10^{11}\ Pa$, $v = 0.28$.

Analytical solution for this study is given by: $\sigma_{crit} = K \dfrac{\pi^2 D}{a^2 h}$,

where $D = \dfrac{Eh^3}{12\left(1-v^2\right)}$ – cylindrical stiffness of plate, E – Young's modulus, K – coefficient whose value depends on the type of the supports of the plate edges (in this case $K = 7.69$).

Thus, $\sigma_{crit} = K \dfrac{\pi^2 D}{a^2 h} = 0.5188 \times 10^8 \left[\dfrac{N}{m^2}\right]$

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.

Mesh Number	Numerical solution Critical load q, $\dfrac{N}{m^2}$	Analytical solution Critical load σ_{krit}, $\dfrac{N}{m^2}$	Error $\delta = \dfrac{\left|\sigma_{krit} - q\right|}{\left|\sigma_{krit}\right|} \times 100\%$
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
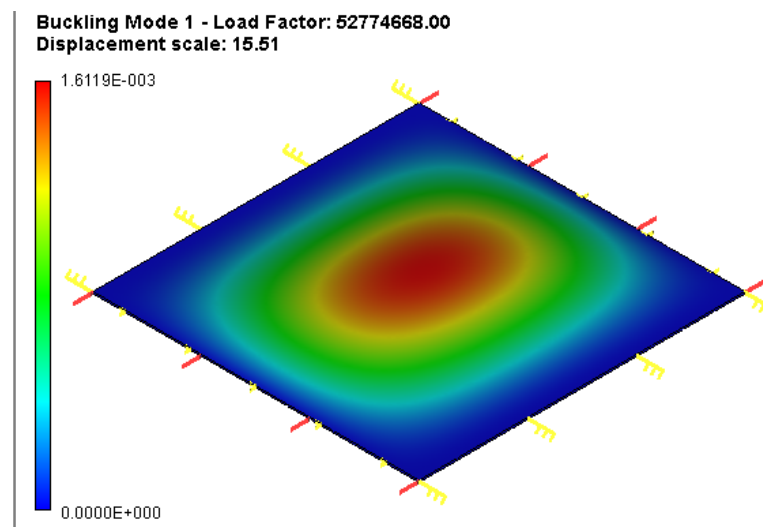
Buckling Mode 1 - Load Factor: 52774668.00
Displacement scale: 15.51

**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).
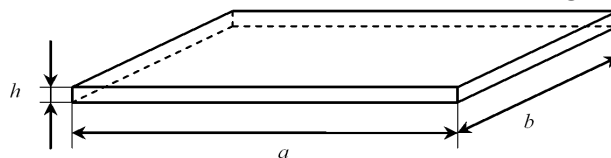
Plate thickness h is considerably smaller than two other dimensions a and b.

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.

clamped edge
simply-supported edge
free edge
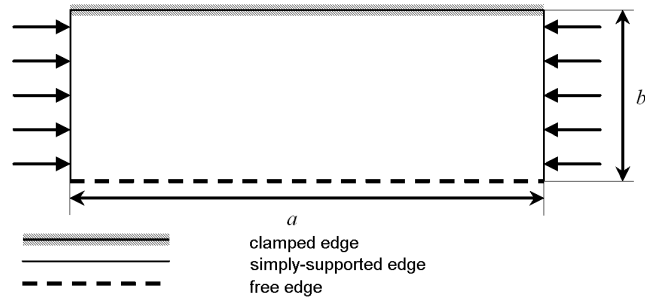
Let us use the following data: length and width of the plate $a = 800mm$, $b = 500mm$, plate thickness $h = 3mm$, applied distributed load $P = 1\dfrac{N}{m^2}$.

Elastic characteristics: $E = 2\times10^{11}\,Pa$, $v = 0.25$.

Analytical solution for this study is given by: $\sigma_{crit} = K\dfrac{\pi^2 D}{b^2 h}$,

where $D = \dfrac{Eh^3}{12(1-v^2)}$ – cylindrical stiffness of plate, E – Young's modulus, K – coefficient whose value depends on the type of the supports of the plate edges and the ratio $\dfrac{a}{b}$ (in this case $K = 1.33$).

Thus, $\sigma_{crit} = K\dfrac{\pi^2 D}{b^2 h} = 0.8401\times10^7 \left[\dfrac{N}{m^2}\right]$.

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, $\dfrac{N}{m^2}$	Analytical solution Critical load σ_{krit}, $\dfrac{N}{m^2}$	$\delta = \dfrac{\left|\sigma_{krit} - q\right|}{\left|\sigma_{krit}\right|} \times 100\%$ Error
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
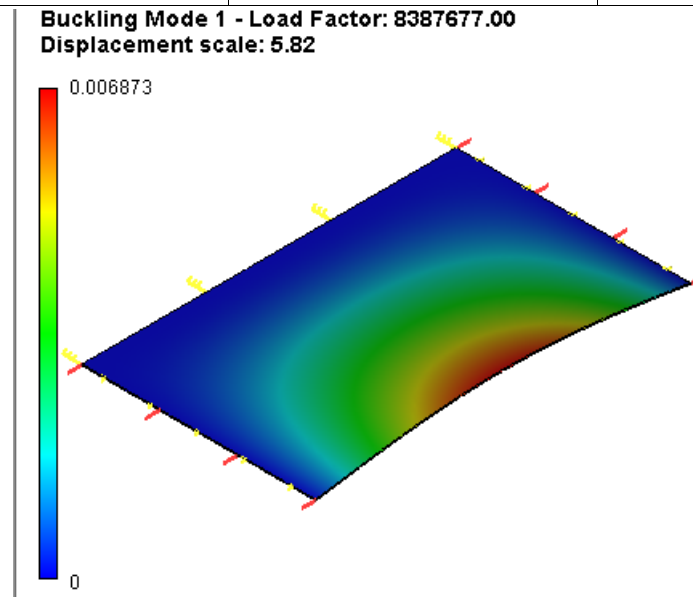
EXAMPLES OF FREQUENCY ANALYSIS STUDY

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.
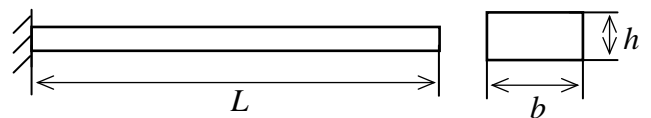
The material properties are: the Young's modulus $E = 2.1 \cdot 10^{11}$ Pa, the Poisson's ratio $\nu = 0.28$, the density $\rho = 7800 \ \dfrac{kg}{m^3}$.
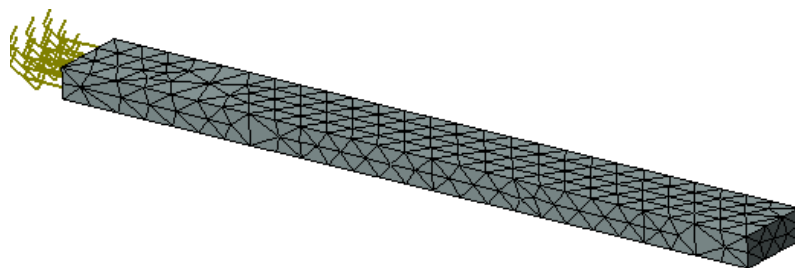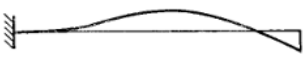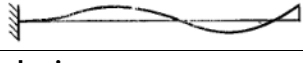
Finite element model of the beam with restraints

The analytical solution is given by:

$$f_i = \sqrt{\frac{E \cdot J}{\rho \cdot F} \frac{1}{2\pi} \left(\frac{k_i}{l} \right)^2}$$

where f_i - natural frequencies, E – the material Young's modulus, J – the moment of inertia, ρ – 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 $\rho = 7800 \frac{\kappa g}{m^3}$.
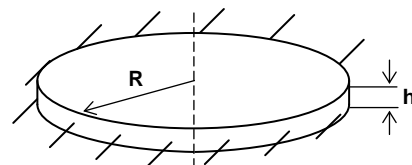
Due to the symmetry, we will consider the quarter of the plate and apply the appropriate boundary conditions.

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:

$$f = \frac{10.21}{R^2 \cdot 2\pi}\sqrt{\frac{D}{\rho \cdot h}} = 624.5 Hz$$,

where R – plate radius,

ρ – density of material, h – thickness of material, $\quad D = \dfrac{E \cdot h^3}{12 \cdot (1-v^2)}$ – flexural stiffness.
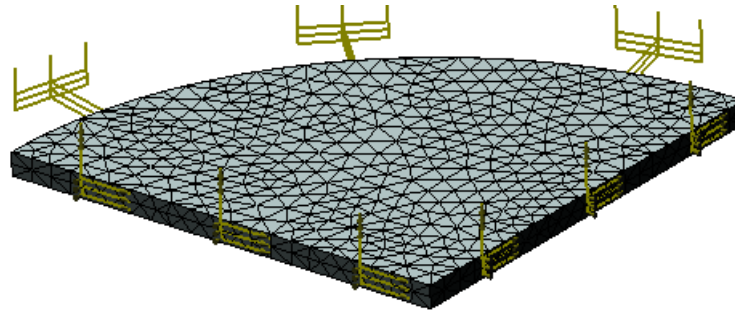
Finite element model of the plate with restraints

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	$\delta = \dfrac{\left|f_1 - \lambda_1\right|}{\left|f_1\right|} \times 100\%$ Error
1	623.0	624.5	0.24
2	621.5	624.5	0.48
3	622.1	624.5	0.39
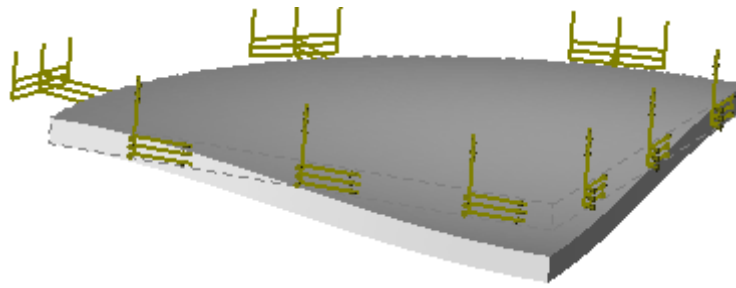
First vibration mode

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 (+shell FE)

Consider a spherical dome of radius R, clamped along the contour (see figure)
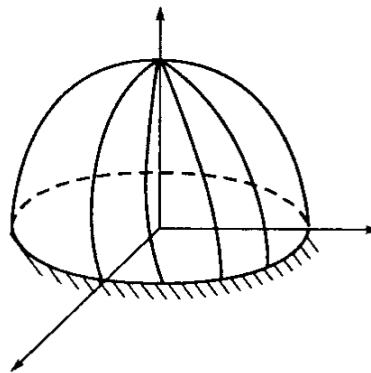
Thickness of dome h is considerably smaller than its radius R.

Only one quarter of spherical surface was modeled. The bottom edge was fully restrained, the symmetry boundary conditions were applied to the side faces.
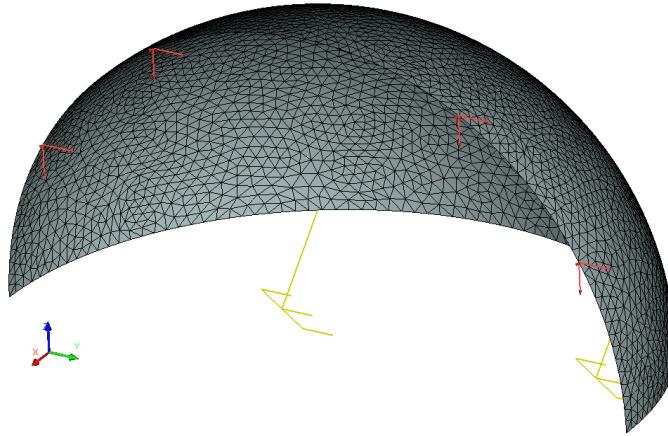
Finite element model of spherical shell with restraints

Let us use the following data: radius $R = 300mm$, thickness $h = 3mm$ ($\dfrac{R}{h} = 100$).

Elastic properties: $E = 2.1 \times 10^{11}\ Pa$, $v = 0.28$, $\rho = 7800\dfrac{kg}{m^3}$.

Analytical solution of this study is given by: $f_i = \dfrac{k_i \cdot \omega_0}{2\pi}$,

where $\omega_0 = \sqrt{\dfrac{E}{\rho R^2 \left(1 - v^2\right)}}$

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:
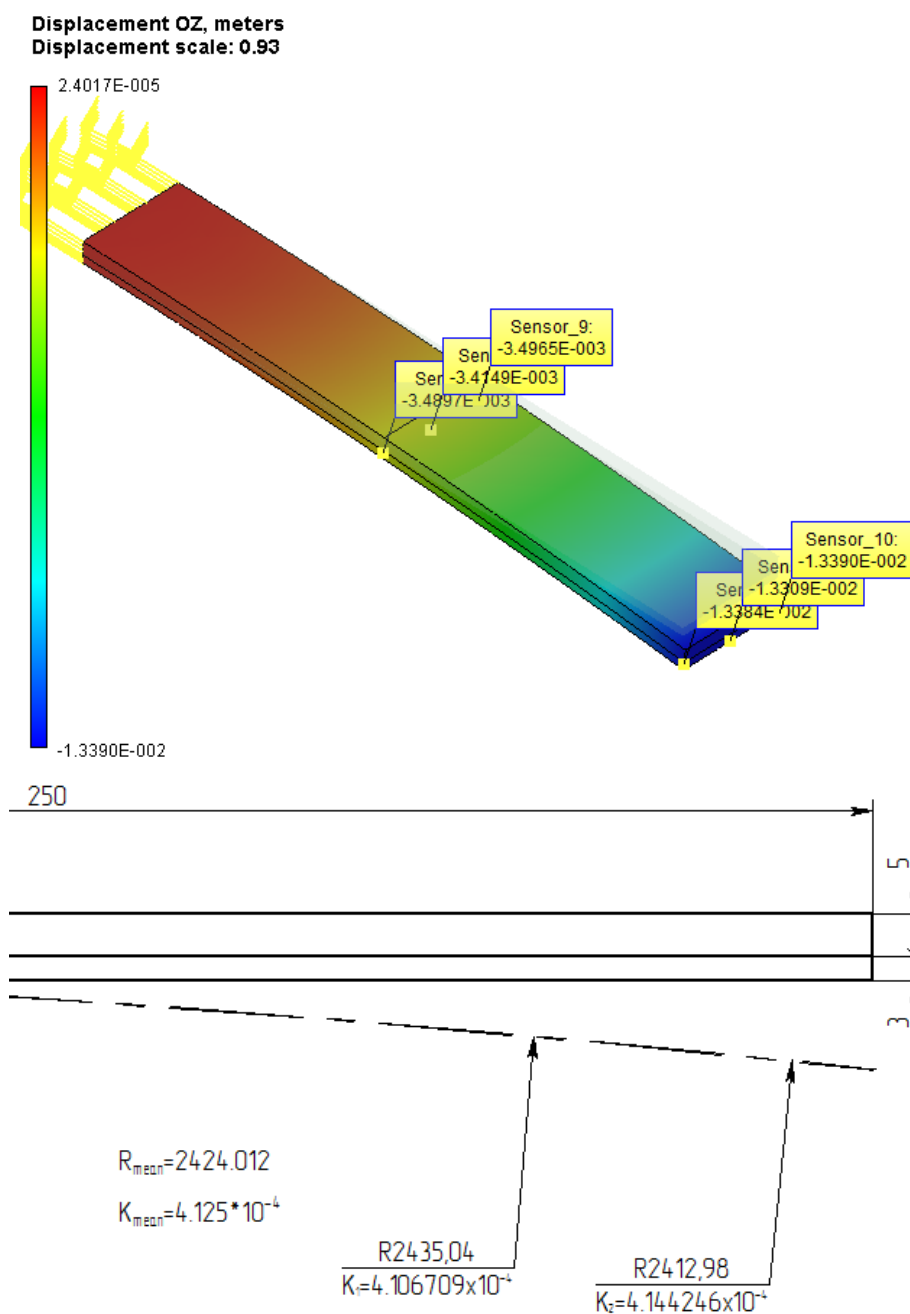
Table 1.

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	$\delta = \dfrac{\left\lvert f_i - \lambda_i \right\rvert}{\left\lvert f_i \right\rvert} \times 100\%$
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	$\delta = \dfrac{\left\lvert f_i - \lambda_i \right\rvert}{\left\lvert f_i \right\rvert} \times 100\%$
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	$\delta = \dfrac{\left\lvert f_i - \lambda_i \right\rvert}{\left\lvert f_i \right\rvert} \times 100\%$
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.
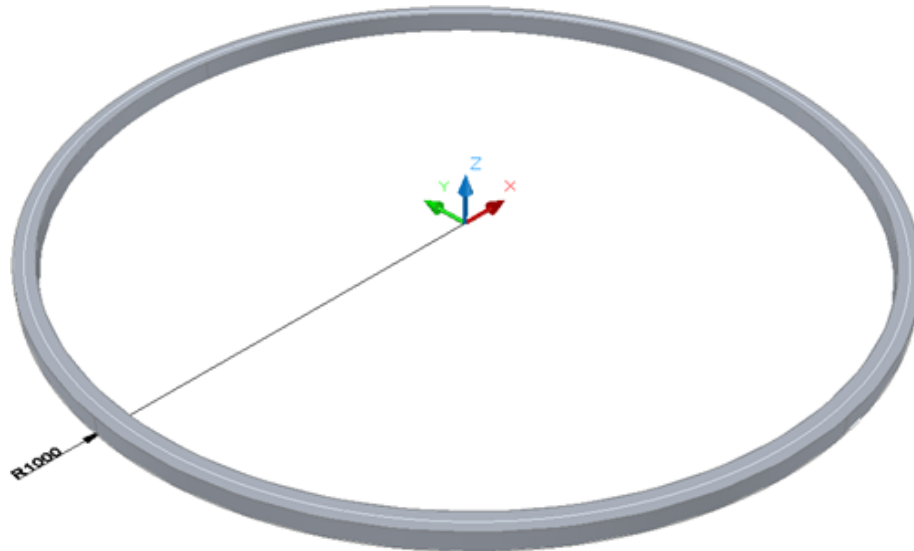
The square cross-section of the ring is much smaller than the radius R. The length of the side of the square is equal to 0.050 m.

The motion of the lower face is constrained along the normal.
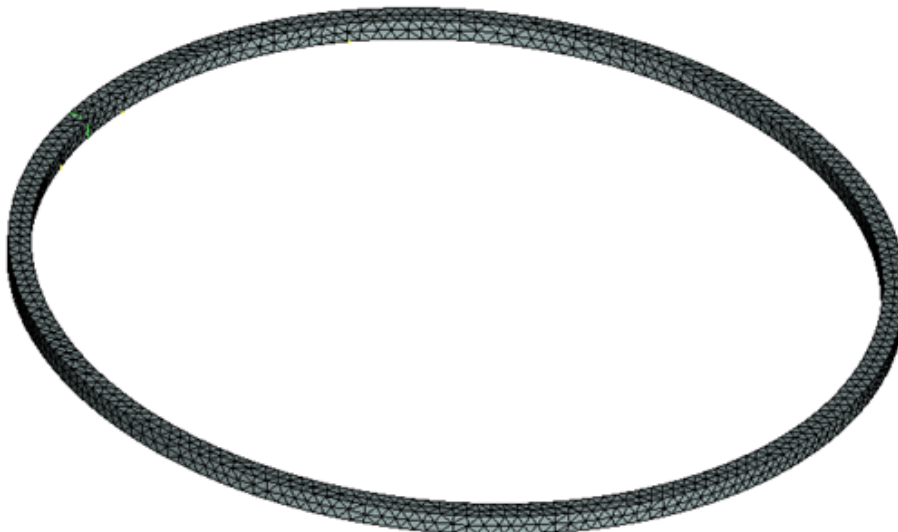
Finite element model for indicated loads and constraints.

Material's properties: E = 2.0E+011 Pa, ν=0.29, ρ = 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, f_2 = 31.015 Hz , f_3 = 87.723 Hz , f_4 = 168.201 Hz, f_5 = 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 δ = 100%*$\lvert f_i^* - f_i\rvert$ / $\lvert f_i\rvert$
31.0	31.015	0.05
87.5	87.723	0.25
167.3	168.201	0.54
269.5	272.017	0.9

Relative Displacement, magnitude
Mode Shape 4 - Frequency: 31.0 Hz
Displacement scale: 0.91
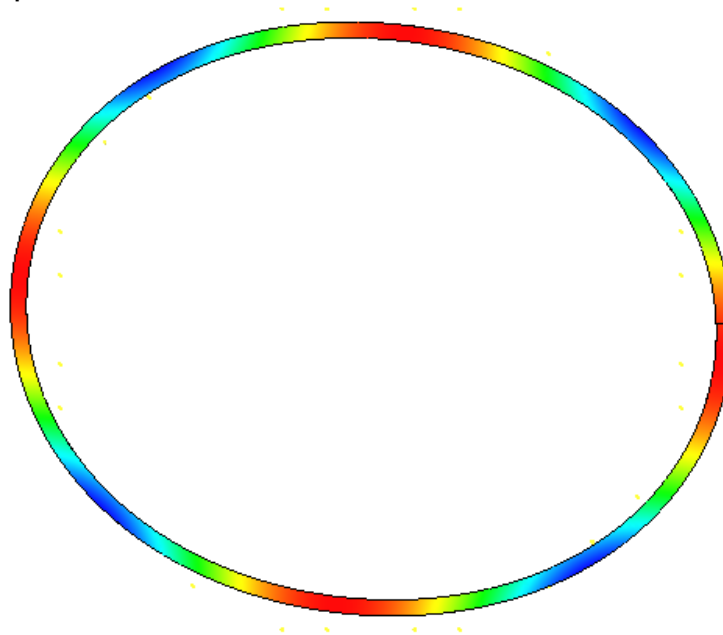
Fourth form of vibrations of a ring

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.
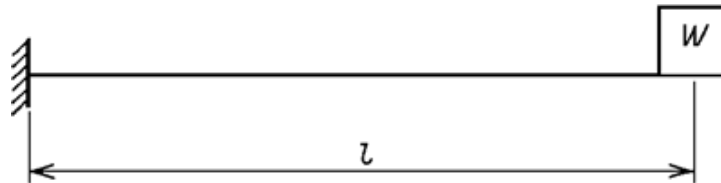
The length of the beam is L. The cross-section of the beam is a rectangle of width b and height h. Mass of the weight is M. Mass of the beam itself is m.

$m = \rho\, F,$

where $F = b,$

h – dimensions of the cross-section, ρ - density of the material of the beam.
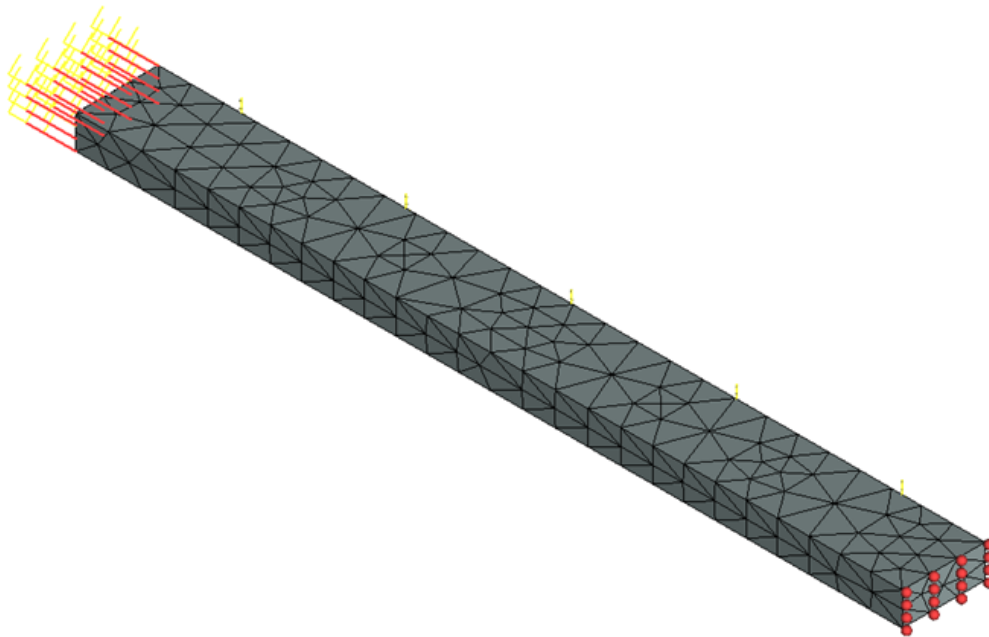
Finite element model for indicated loads and constraints.

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, f_A = 1078.962 Hz, f_T = 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

Relative Displacement, magnitude
Mode Shape 1 - Frequency: 22.3 Hz
Displacement scale: 0.07
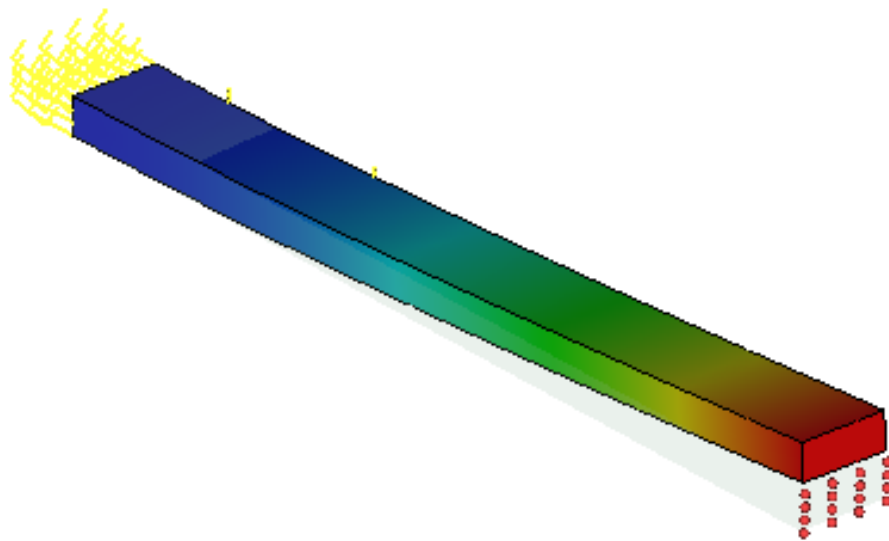
First natural frequency of cantilever beam under the action of longitudinal tensile force

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.
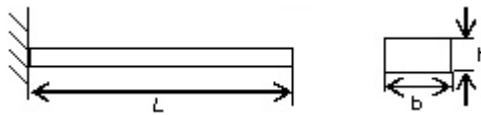
The beam is fixed at the left edge and loaded with a longitudinal tensile force P at the right edge.

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.
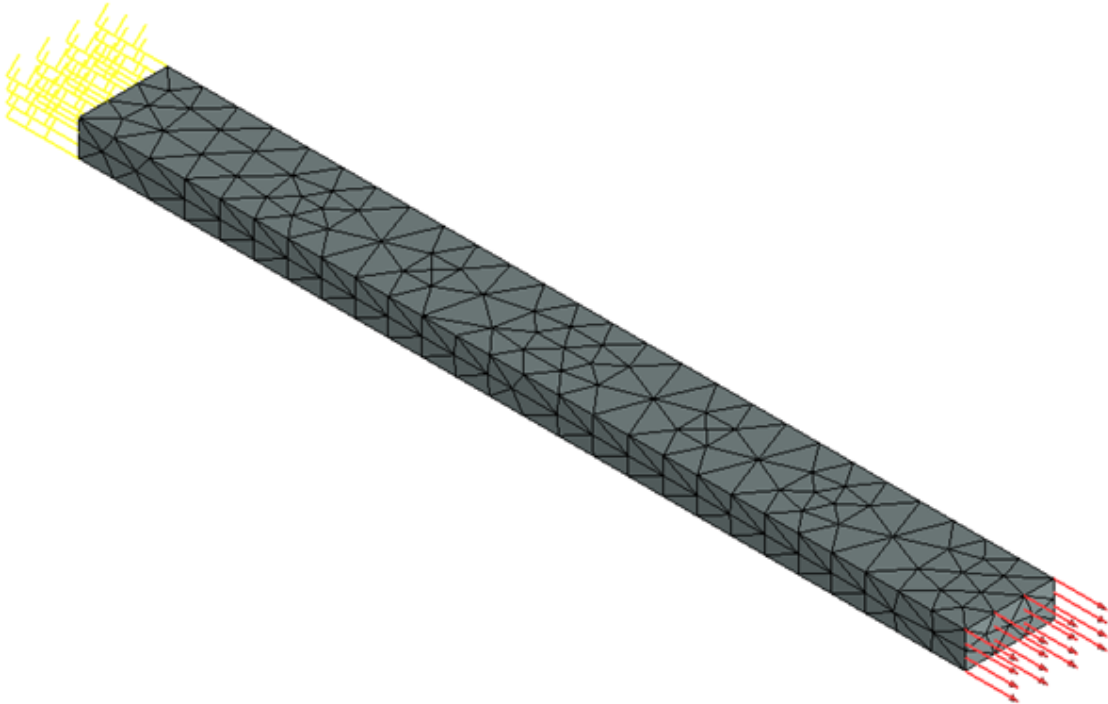
Finite element model for indicated loads and constraints.

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 f_1 – first natural frequency of the beam, J –moment of inertia, ρ – density of the material, F –area of cross-section, k_1 = 1.875.

Therefore, f_1^* = 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

Relative Displacement, magnitude
Mode Shape 1 - Frequency: 86.2 Hz
Displacement scale: 0.03
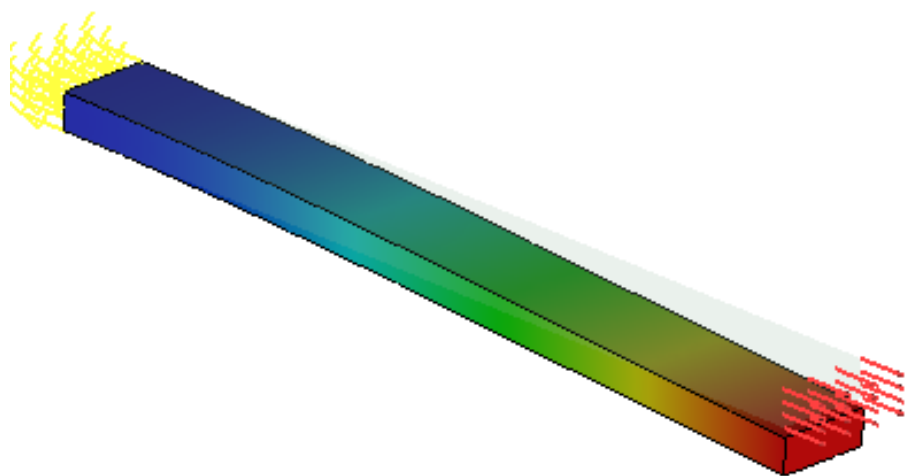
First natural frequency of the system on an elastic foundation

Let us consider the mass that has a form of a cube located on an elastic foundation.
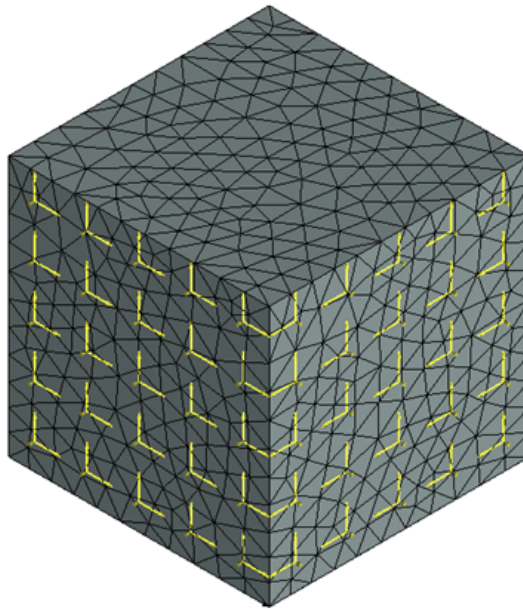
Finite element model for indicated loads and constraints.

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, ν=0.28, ρ = 7800 kg / m³.

Mass of the cube M is calculated from the following formula:

M= ρ 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
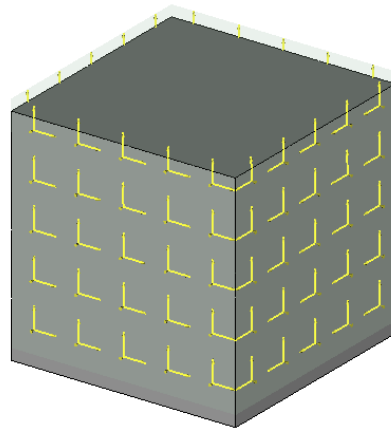
EXAMPLES OF FORCED VIBRATIONS ANALYSIS STUDIES

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.
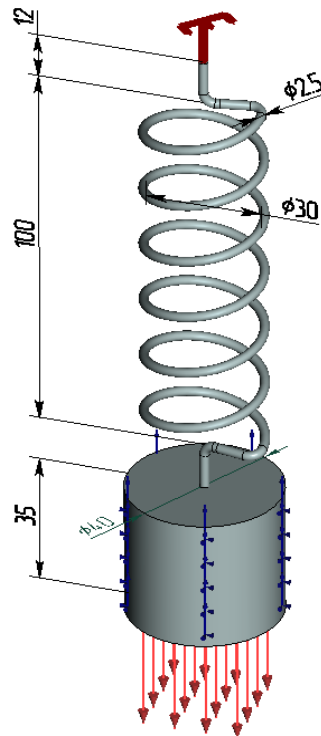
Let us assume the following input data: average diameter of the spring D = 30 mm, length of the spring is H = 100 mm, diameter of the wire is d = 2.5 mm, the number of spring's winds n = 6. Parameters of the weight: diameter D_w = 40 mm; height H_w = 35 mm, mass m_w = 0.34306 kg, the amplitude of the load P = 10 N and it changes as a sinusoidal function.

Material's characteristics of the spring and weight: $E = 2.1 \times 10^{11}$ 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 = 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)]:

$$f_c^{(1)} = \frac{1}{2\pi}\sqrt{\frac{g}{\Delta z_{st\_w}}},$$

where g – gravitational acceleration, m/s²; Δz_{st_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)]:

$$\Delta z_{st\_w} = \frac{8 m_w g \cdot D^3 \cdot n}{d^4 \cdot G},$$

where m_w – 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.

Displacement OZ, millimeters
Displacement scale: 5.50

0.0059

Relative Displacement, magnitude
Mode Shape 1 - Frequency: 13.2 Hz
Displacement scale: 0.00

-1.4462
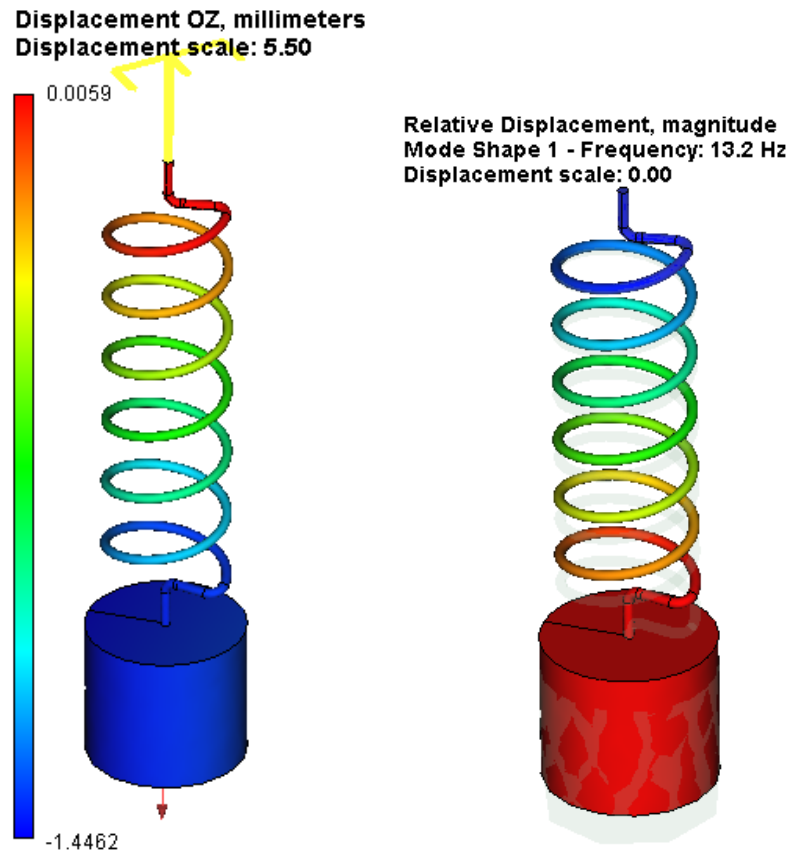
Now let us compute the first natural frequency: the calculation from the formula gives the value $f_c^{(1)}$=13.511 Hz, finite element analysis gives the value $f_c^{(1)}$=13.216 Hz.

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.

Displacement OZ, millimeters
Displacement scale: 1.49
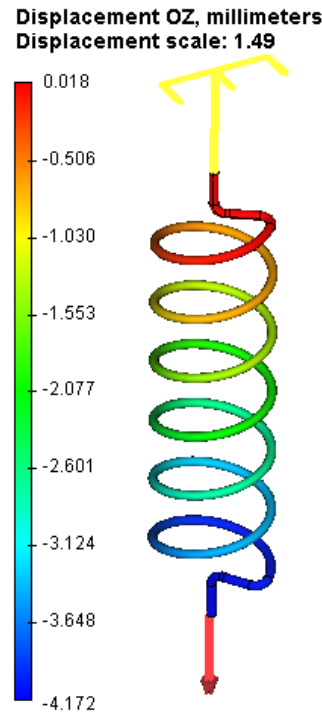
Now let us calculate the damped coefficient of amplitude magnification [1, p 74, (f.1.47)]:

$$\beta_D = \frac{1}{\sqrt{\left(1-\left(\frac{\omega_f}{\omega_c}\right)^2\right)^2+\left(2\gamma\frac{\omega_f}{\omega_c}\right)^2}},$$

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 $f_f = f_c$ the amplitude magnification coefficient is

$$\beta_D = \frac{1}{2\gamma}$$

With the knowledge of the amplitude magnification coefficient, the amplitude A_f of the forced vibrations can be obtained from:

$$A_f = \beta_D \cdot \Delta z_{st}$$

Calculation performed for f_B = 13.511 Hz from these formulas gives β_D = 3.333 and A_f = 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 $f_c^{(1)} = 13.511$, this formula gives the value $\gamma = 0.15$.

The value of the displacement amplitude can be determined using the «Displacements, amplitude, Z» result.

4.0 Hz - Form, Amplitude Z, millimeters
Displacement scale: 1.75

20.0 Hz - Form, Amplitude Z, millimeters
Displacement scale: 2.62
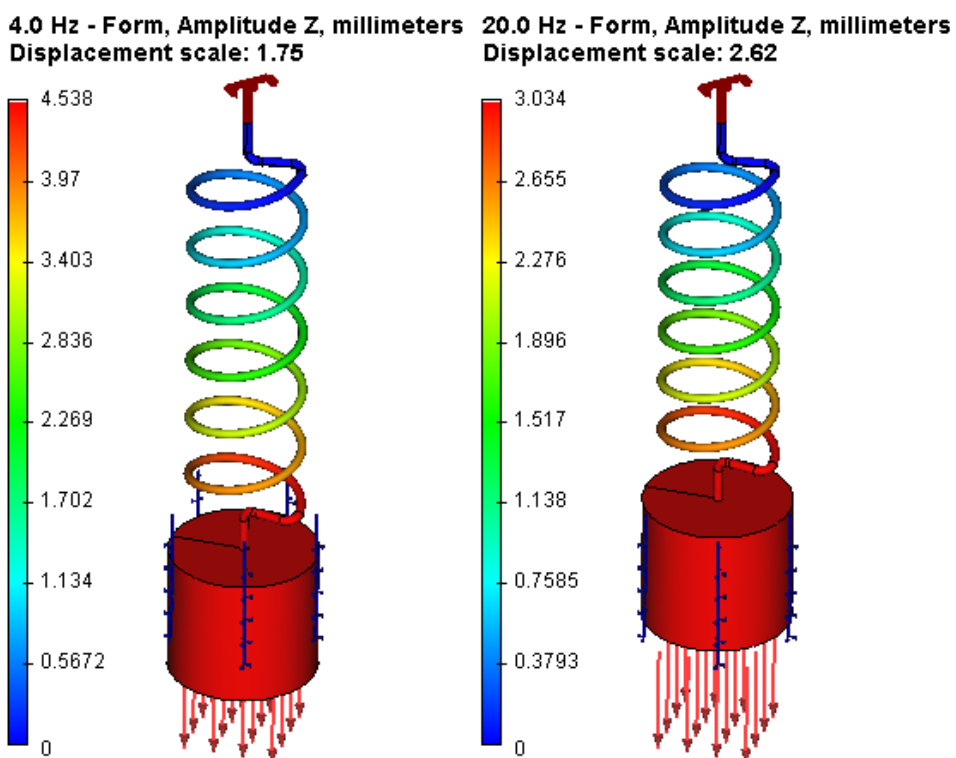
Results of the analysis are shown in the table.

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_f, Hz	Numerical value, A_f^*, mm	Analytical solution A_f, mm	Error $\delta_u = \dfrac{\left\lvert A_f - A_f^* \right\rvert}{A_f} \times 100\%$
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
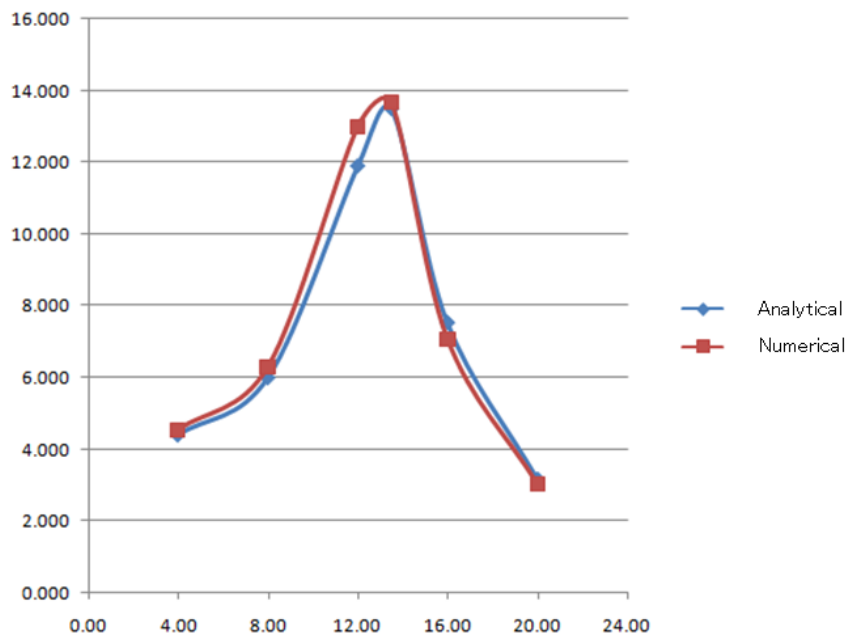
Plot of comparison of results

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.
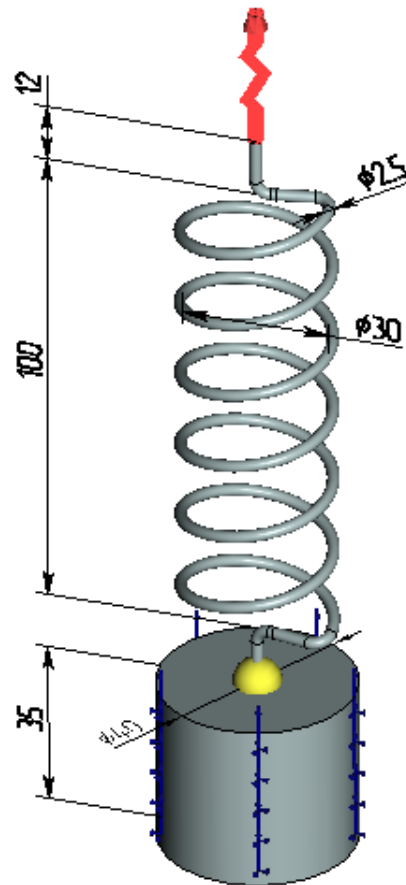
Figure 1 – A numerical model with boundary conditions and a sensor

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 D_w = 40 mm; height H_w = 35 mm, mass m_r = 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 (f. 4)]:

$$f_n^{(1)} = \frac{1}{2\pi}\sqrt{\frac{g}{\Delta z_{st\_g}}},$$

where g – gravitational acceleration, m/s²; Δz_{st_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, (f. 9.54)]:

$$\Delta z_{cm\_ep} = \frac{8 m_e g \cdot D^3 \cdot n}{d^4 \cdot G},$$

where m_w – 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 $\Delta z_{st\_g} = -1.361$ mm; FEM result is $\Delta z_{st\_g}^* = -1.388$ mm.

Displacement OZ, millimeters
Displacement scale: 5.71
0.0049
Sensor_1: -1.3876
-1.3925

Displacement OZ, millimeters
Displacement scale: 6.01
0.0043
Min -1.0317
-1.0317
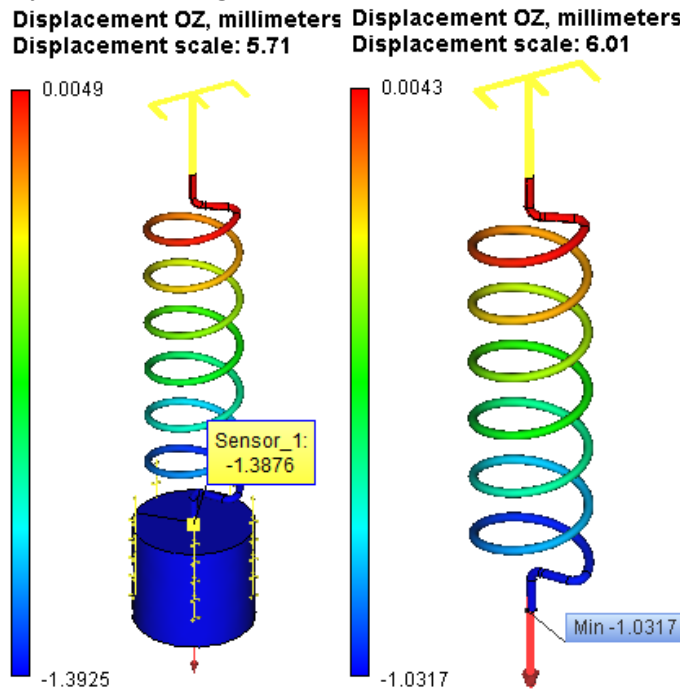
Now, let us calculate the first eigenfrequency: calculation by the formula gives the value $f_n^{(1)}$= 13.511 Hz, FEM calculation gives the value $f_n^{(1)}$=13.216 Hz.

Equivalent exciting force can be found from the displacement of the foundation attached to the upper end of the spring:

$$P = \frac{\Delta z_b d^4 \cdot G}{8D^3 \cdot n},$$

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 $P = 2.472$ N.

Now let us calculate the damped coefficient of the amplitude magnification [1, p. 74, (f.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 $f_f = f_n$ the maximum magnification coefficient is

$$\beta_d = \frac{1}{2\gamma}$$

Then, the amplitude A_f 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 $f_n^{(1)}$ = 25.00 Hz, this value is γ = 0.02;

The magnitude of the amplitude can be inferred from the result «Displacements, amplitude, Z»

4.0 Hz - Form, Amplitude Z, millimeters
Displacement scale: 7.21

13.2 Hz - Form, Amplitude Z, millimeters
Displacement scale: 0.31
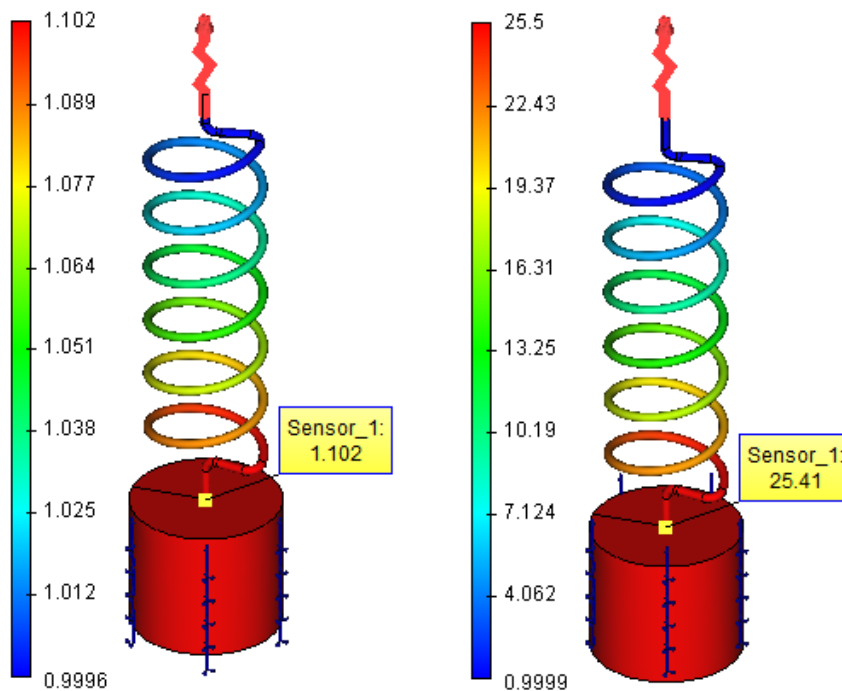
Results of calculations are shown in the Table.

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, $A_f{}^*$, mm	Analytical solution A_f, mm	Error $\delta = \dfrac{\left\vert A_f - A_f^* \right\vert}{A_f} \times 100\%$
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
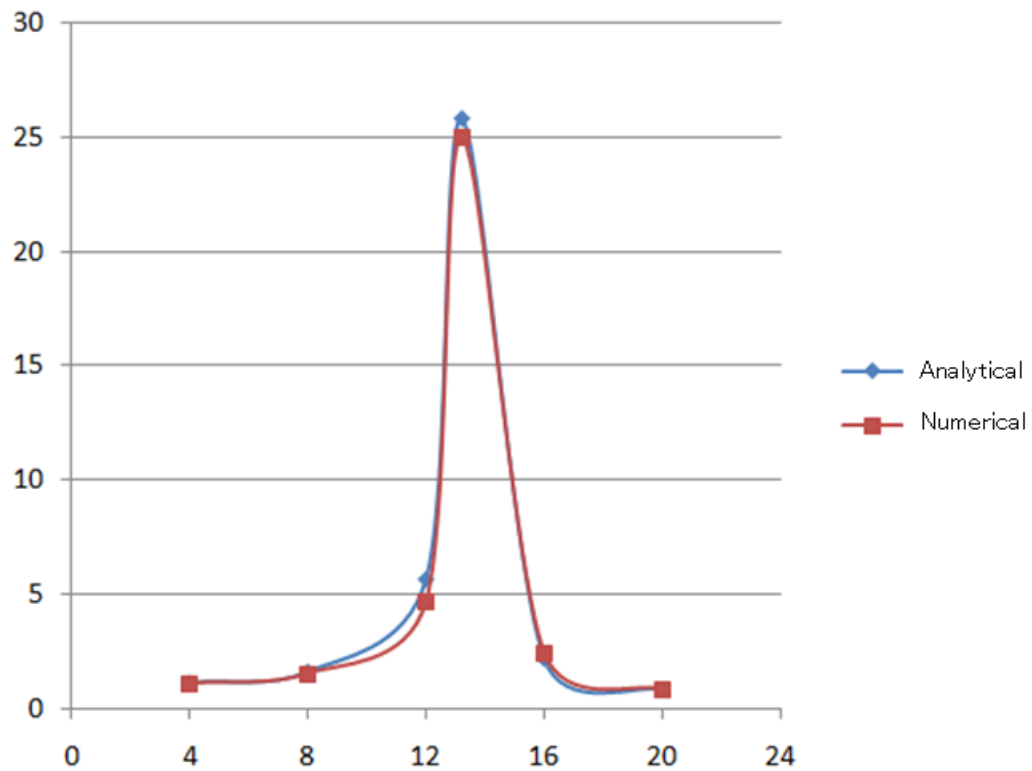
Plots of comparison of results

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 study 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).
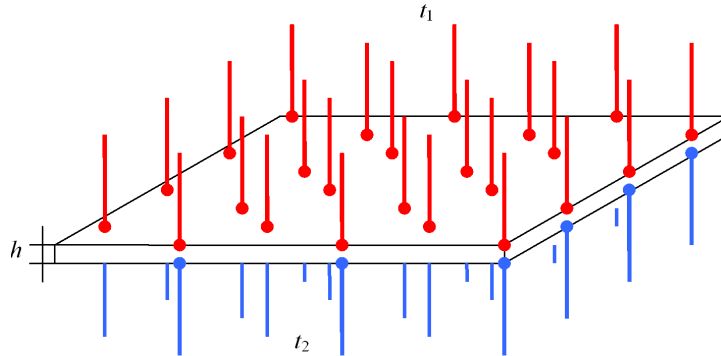
The change in temperature along the thickness of the plate h is defined by relation:

$$\frac{\partial T}{\partial z} = \frac{t_2 - t_1}{h}, t_1 < t_2$$

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 study, 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:

$$t_1 - t_{n+1} = f_1 R_1 + f_2 R_2 + \ldots + f_n R_n = (R_1 + R_2 + \ldots + R_n)f, R_i = \frac{h_i}{k_i}, i = 1,2,\ldots n$$

$$f = \frac{t_1 - t_{n+1}}{\dfrac{h_1}{k_1} + \dfrac{h_2}{k_2} + \ldots + \dfrac{h_n}{k_n}}$$

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: $k_1 = 200 \dfrac{W}{m \cdot K}$, $k_2 = 390 \dfrac{W}{m \cdot K}$, $k_3 = 43 \dfrac{W}{m \cdot K}$

Thus, $f = -7.6682 \times 10^5 \dfrac{W}{m^2}$, $t_2 = -R_1 f + t_1 = 299.9887\ K$, $t_3 = -(R_1 + R_2)f + t_1 = 319.6508\ K$.

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 study 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	$\delta = \dfrac{\left|T - T^*\right|}{\left|T\right|} \times 100\%$ Error
$S12$	3.04589000E+02	3.04588978E+02	7.2188E-006
$S23$	3.23015000E+02	3.23014753E+02	7.6569E-005

Result «Heat flux»

Numerical solution Heat flux f *, W/m^2	Analytical solution Heat flux f , W/m^2	Error $\delta = \dfrac{\left\| f - f^* \right\|}{\left\| f \right\|} \times 100\%$
-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.
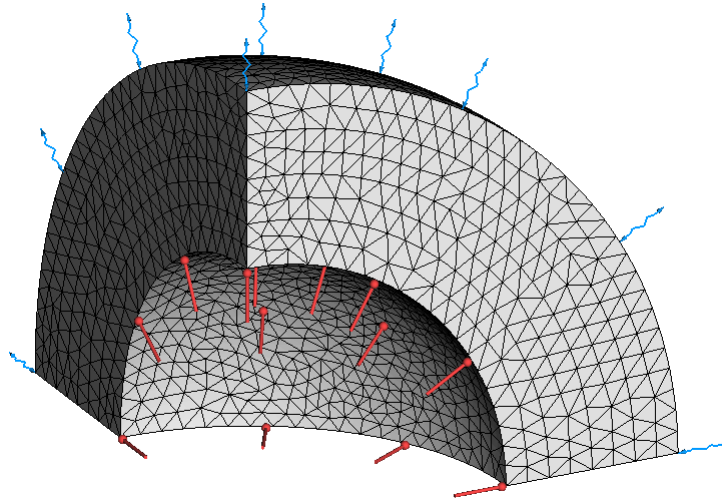
Analytical solution of the study has the form:

$$T = \frac{r_1 \cdot T_1 \cdot \left(\beta \cdot r_2^2 + r \cdot (1 - \beta \cdot r_2) \right) + \beta \cdot r_2^2 T_2 \cdot (r - r_1)}{r \cdot \left(\beta \cdot r_2^2 + r_1 \cdot (1 - \beta \cdot r_2) \right)}$$

For numerical calculation consider $\dfrac{1}{8}$ 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 $r_1 = 150 mm$, external radius of the sphere $r_2 = 250 mm$. Coefficient of thermal conductivity λ of the material of the sphere is equal to $47 \dfrac{W}{m \cdot K}$.

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 $25\,{}^{0}C$), heat transfer coefficient β is equal to

$$100\,\frac{W}{m^2 \cdot {}^{0}C}.$$

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 study 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 $r = \dfrac{3r_1 + r_2}{4} = 0.175\,м$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	$\delta = \dfrac{\left|T - T^*\right|}{\left|T\right|} \times 100\%$ Error
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 $r = \dfrac{r_1 + r_2}{2} = 0.200\,м$

Mesh number	Numerical solution Temperature T^*, K	Analytical solution Temperature T, K	$\delta = \dfrac{\left|T - T^*\right|}{\left|T\right|} \times 100\%$ Error
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 $r = \dfrac{r_1 + 3r_2}{4} = 0.225\,м$

Mesh number	Numerical solution Temperature $T*$, K	Analytical solution Temperature T, K	Error $\delta = \dfrac{\left|T - T^*\right|}{\left|T\right|} \times 100\%$
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 $\delta = \dfrac{\left|T - T^*\right|}{\left|T\right|} \times 100\%$
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).
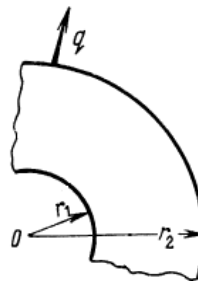
General solution of this study has the form:

$$T = C_1 \ln(r) + C_2 - \frac{q_v}{4\lambda} r^2.$$

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: $T|_{r=r_1} = T_1$, $\lambda \dfrac{dT}{dr}\Big|_{r=r_2} = q$. Thus, $C_1 = \dfrac{r_2}{\lambda}\left(q + \dfrac{q_v r_2}{2}\right)$,

$$C_2 = T_1 - \frac{qr_2}{\lambda}\ln(r_1) - \frac{q_v}{4\lambda}\left(2r_2^2\ln(r_1) - r_1^2\right).$$

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 $43\dfrac{W}{m \cdot K}$.

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, $q_v = \dfrac{Q}{\pi \cdot \left(r_2^2 - r_1^2\right) \cdot l} = 27283.705\dfrac{W}{m^3}$.

Specific heat flux on the external surface of the tube $q = -15000\dfrac{W}{m^2}$. 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 meshes

Mesh number	Finite element type	Number of nodes	Number of nodes for study 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 $r = r_2 = 0.250\,м$

Mesh number	Numerical solution Temperature $T *$, K	Analytical solution Temperature T, K	Error $\delta = \dfrac{\left|T - T^*\right|}{|T|}\times 100\%$
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 study 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 ^{0}C is held.

Parameters of a disk: metal disk with thickness d=5 mm, radius R=100 mm and thermal conductivity K=50 W/(m · ^{0}C).
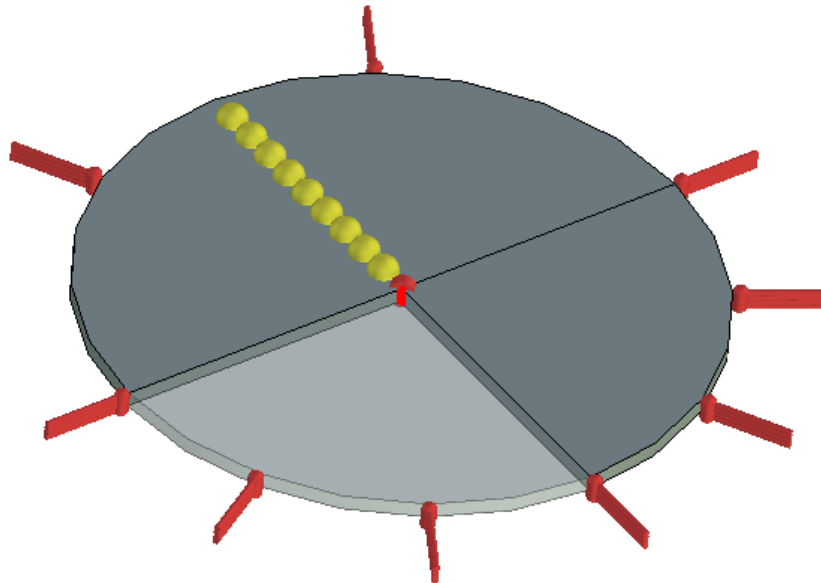
Numerical model with boundary conditions and sensors

Solution of this study in which the source is regarded as distributed can be obtained by solving the study 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)

$$u = G(P, P_0)$$

$$G(P, P_0) = \frac{\rho}{2K\pi}\left(\ln\left(\frac{1}{r}\right) - \ln\left(\frac{R}{r_0 \cdot r_1}\right)\right), \qquad P_0 \neq 0$$

$$G(P, P_0) = \frac{\rho}{2K\pi}\left(\ln\frac{1}{r} - \ln\frac{1}{R}\right), \qquad otherwise$$

$$P_0 = P_0(x_0, y_0)$$

$$P = P(x, y)$$

$$r = \sqrt{(x - x_0)^2 + (y - y_0)^2}$$

$$r_1 = \sqrt{\left(x - x_0\frac{R^2}{r_0^2}\right)^2 + \left(y - y_0\frac{R^2}{r_0^2}\right)^2}$$

$$r_0 = \sqrt{x_0^2 + y_0^2}$$

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, Celsius

Max = 271.8658
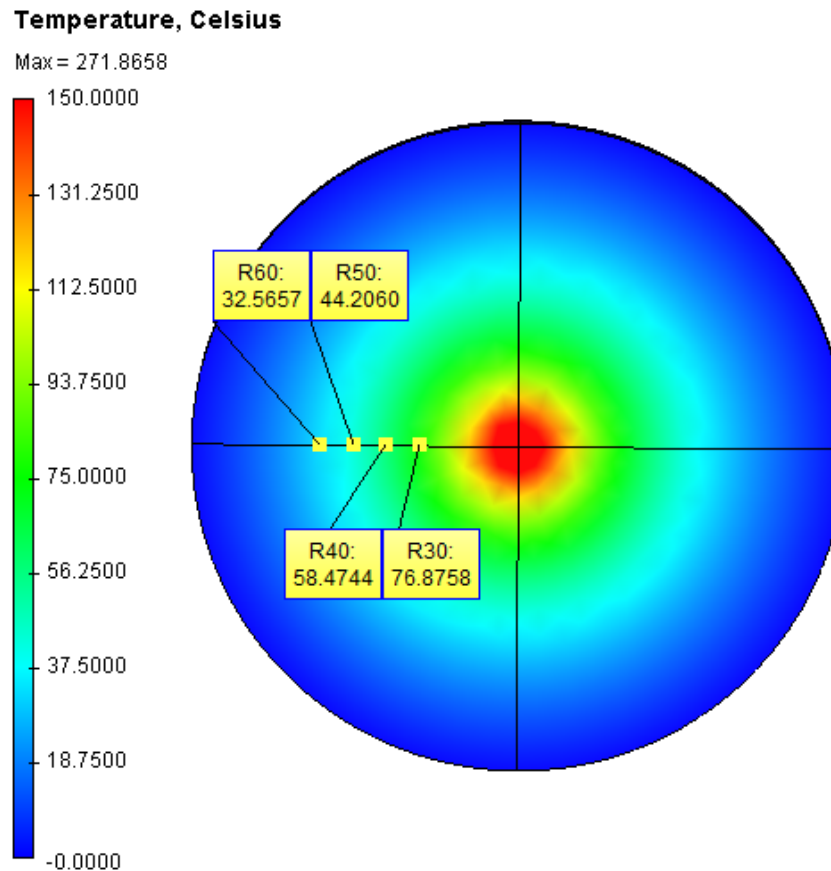
Temperature field on the end-face cross-section of the disk

Picture 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 ^{0}C. 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 study 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

Temperature graph v(r)
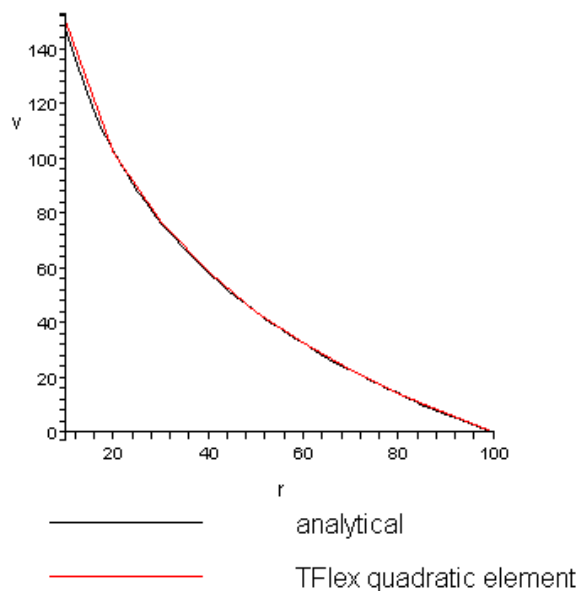
Dependence of steady-state temperature on radius (mm)

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 study 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 ^{0}C 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 · ^{0}C) – thermal conductivity of a disk inside and outside the cylindrical surface of the heater is the same.
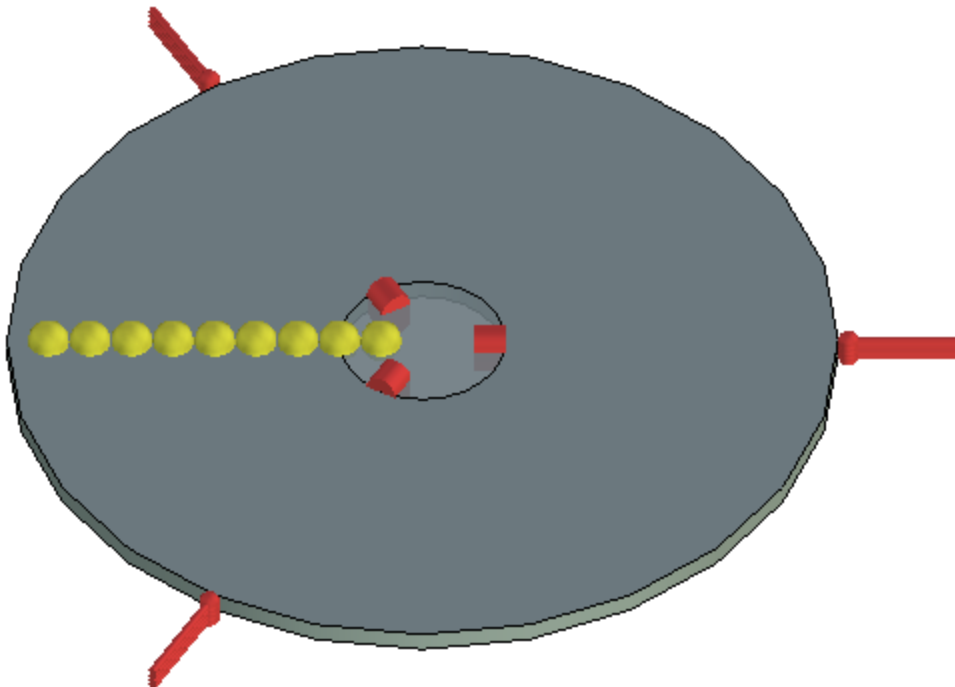
Numerical model with boundary conditions and sensors

Solution of this study, in which the source is considered as distributed, can be obtained by solving the study 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[1]. 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 Picture – 2D flat ring inside the circle). Solution of this equation v is a linear combination (integral) of solutions for point sources:

$$v(P) = r_d \int_0^{2\pi} G_{II}\left(P, P_0\left(x(r_d, \theta), y(r_d, \theta)\right)\right)d\theta$$

$$P_0\left(x(r_d, \theta), y(r_d, \theta)\right) = P_0\left(r_d \cdot \cos(\theta), r_d \cdot \sin(\theta)\right)$$

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:

$$v(P) = G_I(P, 0), P \in \Omega \backslash \Omega'$$

$$v(P) = const, P \in \Omega'$$

$$0 \leftrightarrow x = 0, y = 0$$

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 <u>coincide</u> 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.

---

[1] See example of a distributed power on the edge.
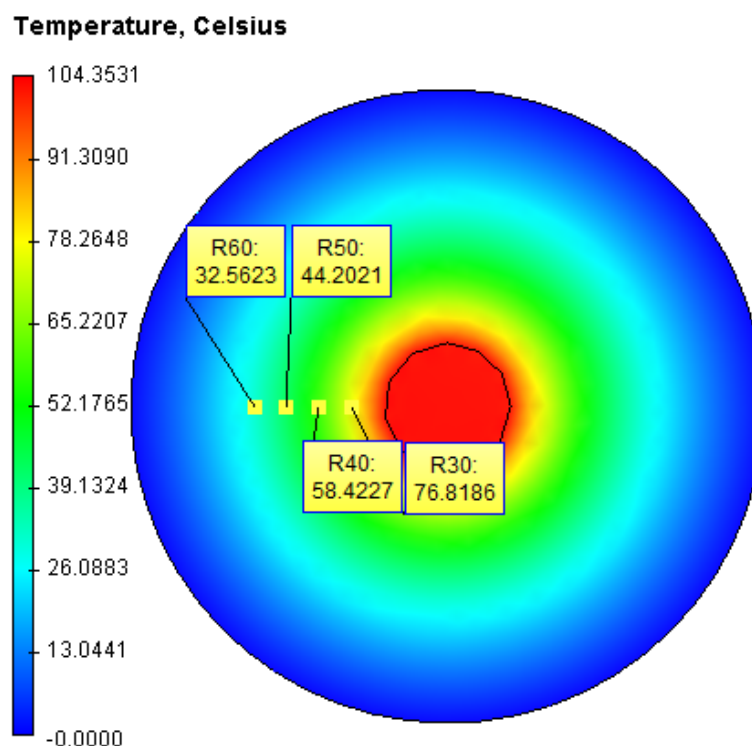
Plot of a temperature field

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

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

Temperature graph v(r)

Coordinate r is given in millimeters
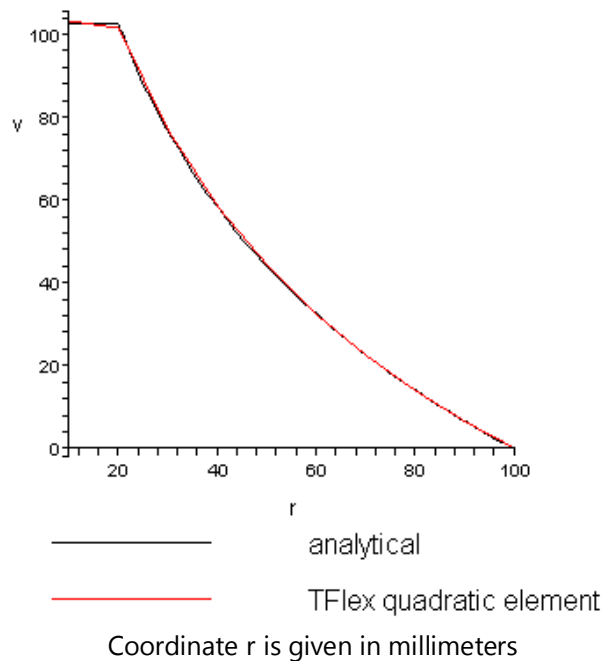
Conclusions:

For a given study 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 study 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 study 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 $t=20\ ^0$C. 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 $\rho=500$ W is prescribed. Coordinates of the point source are equal to $P0=(0, 0, 0)$. Thermal conductivity is assigned the value equal to $\lambda=1$ W/(m·^{0}C).

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.
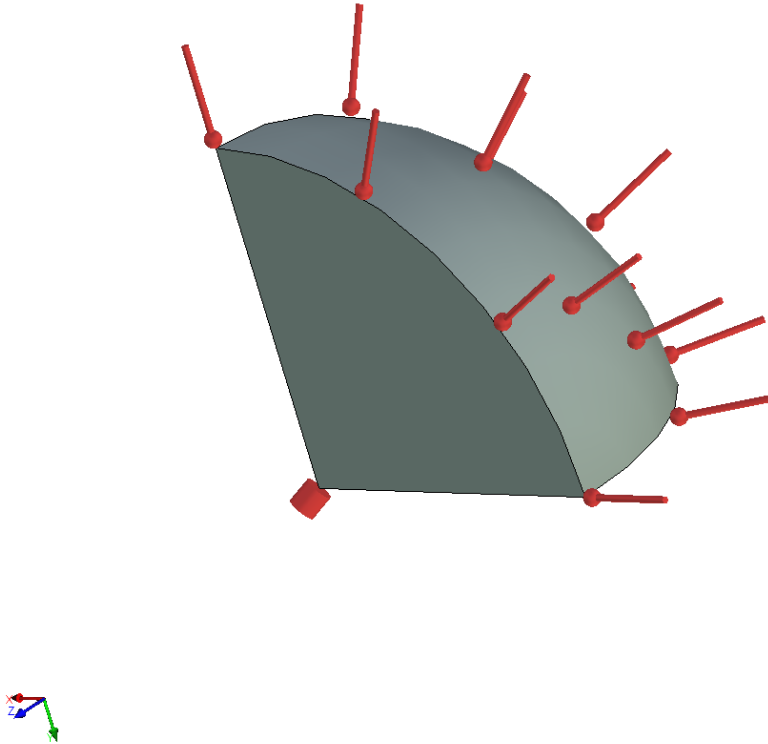
Numerical model with boundary conditions

For a given study 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.
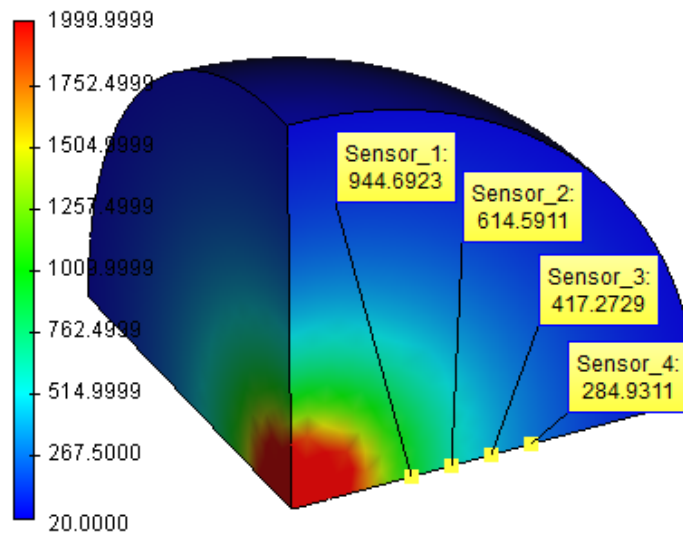
Temperature field inside the sphere obtained by the finite element analysis
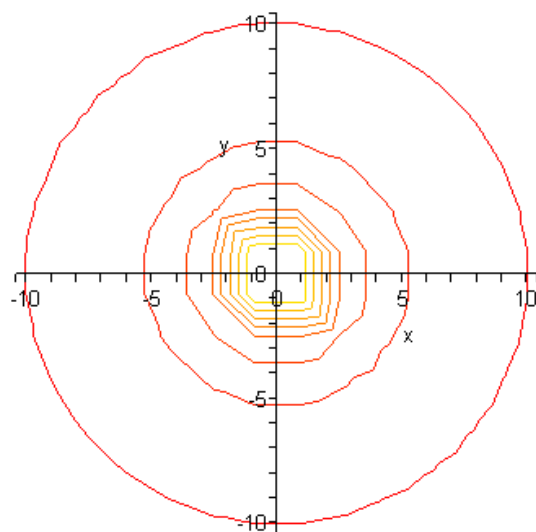
Isothermal lines of the analytical solution plotted using Maple software

The temperature field obtained using T-FLEX Analysis is shown on рисунке 6.6-2. From рисунке 6.6-3 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 рисунке 6.6-2.

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, ^{0}c	948.403	616.831	417.887	285.258
Numerical solution, ^{0}c	948.413	616.653	417.706	285.075
Relative error, %	0.001	0.03	0.04	0.06
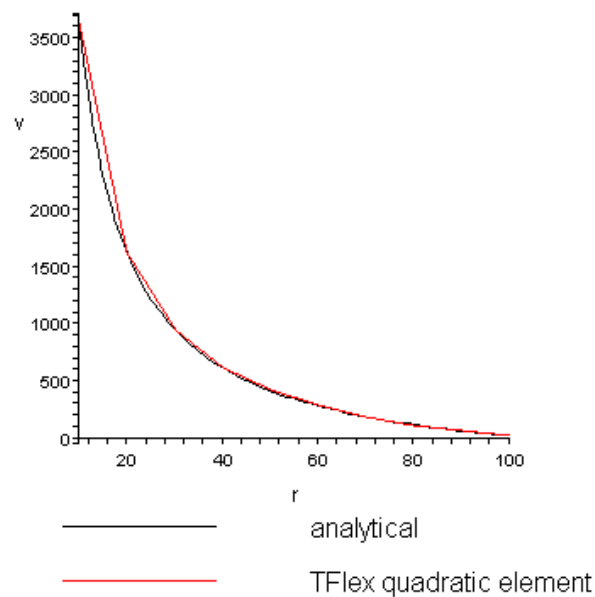
Radial coordinate r is in millimeters

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 studies 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·^{0}C), and a chip has a conductivity equal to $Kchip=50$ W/(m · ^{0}C)[2]. 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 $R=2 \cdot 10^{-4}$ m^2 · ^{0}C / W. The radiator diffuses the heat with the heat transfer coefficient equal to $h=3000$ W/(m^2 · ^{0}C) to the ambient environment that has a temperature of $T0=20$ ^{0}C. 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 $Vrad=(b-a) \cdot S$ and $Vchip = a \cdot S$ respectively, where $S=900 \cdot 10^{-6}$ m^2 – area of the interface.
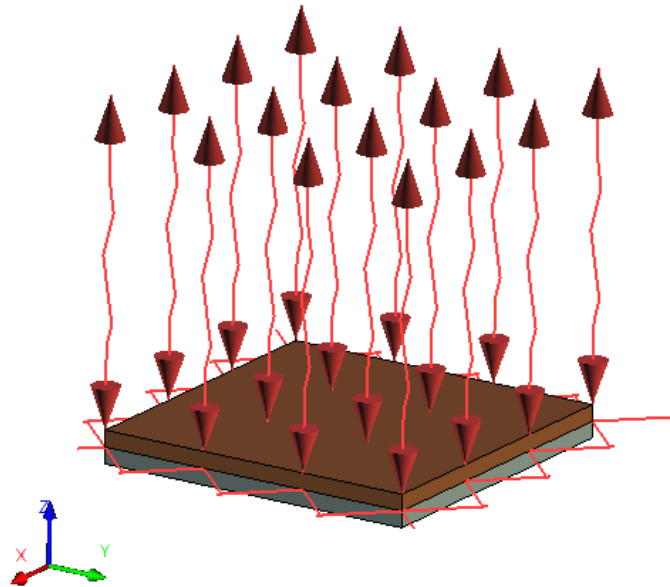
Numerical model with boundary conditions

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:

---

[2] Thermal conductivities of copper (radiator) and steel (microcircuit). Thermal conductivity of the chip was approximated by conductivity of the steel.

$$-K(z)\frac{\partial^2 u}{\partial z^2} = p\delta_{chip}(z), \quad 0 < z < b$$

where *δchip* – 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:

$$\begin{cases} -K_{rad}\frac{\partial u}{\partial z}(b) = h(u(b) - T_0), \\ \frac{\partial u}{\partial z}(0) = 0 \end{cases}$$

For a point source of the heat, the solution of a study with homogeneous boundary condition will take the form: [3]

$$g(z, z_0) = -\frac{p}{2K(z)} \cdot |z - z_0| - \frac{p}{2K(z)} z - \frac{p}{2K(z)} z_0 + C,$$

$$\frac{\partial g}{\partial z}(0, z_0) = 0, z_0 > 0, z = 0$$

Solution *u* for a heat source has the form:

$$u = \int_0^a g(z, z_0)dz_0 = \begin{cases} a \cdot C_{rad} - \frac{p \cdot z \cdot a}{K_{rad}}, z > a, z \le b, \\ a \cdot C_{chip} - \frac{pz^2}{2K_{chip}} - \frac{pa^2}{2K_{chip}}, z \le a, z \ge 0 \end{cases}$$

Constants *Crad* and *Cchip* are determined from the following conditions:

$$\begin{cases} C_{chip} = C_{rad} - R \cdot p - p \cdot a\left(\frac{1}{K_{rad}} - \frac{1}{K_{chip}}\right) \\ C_{rad} = \frac{1}{a}\left(\frac{Pa}{h} + \frac{Pa^2}{K_{rad}} + T_0\right) \end{cases}$$

[3] Solution was obtained on the basis of the Green's function. Its form is described in the book by Ivanchenko D. and Sokolov A. Classical field theory, Moscow 1951, Leningrad. p. 39

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 study 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

Temperature graphs v(z)
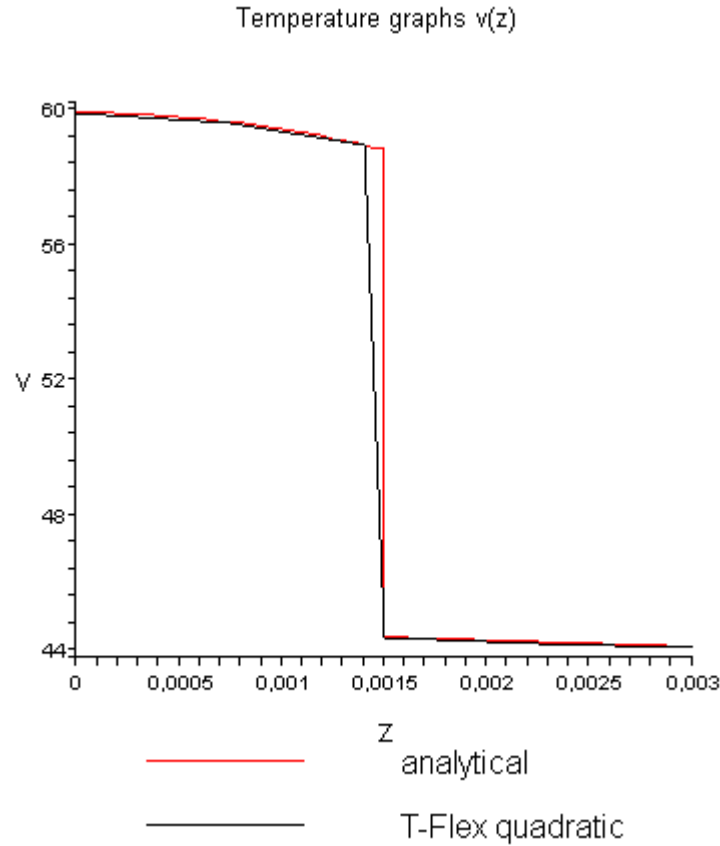
Coordinate z in meters, temperature in 0C.

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
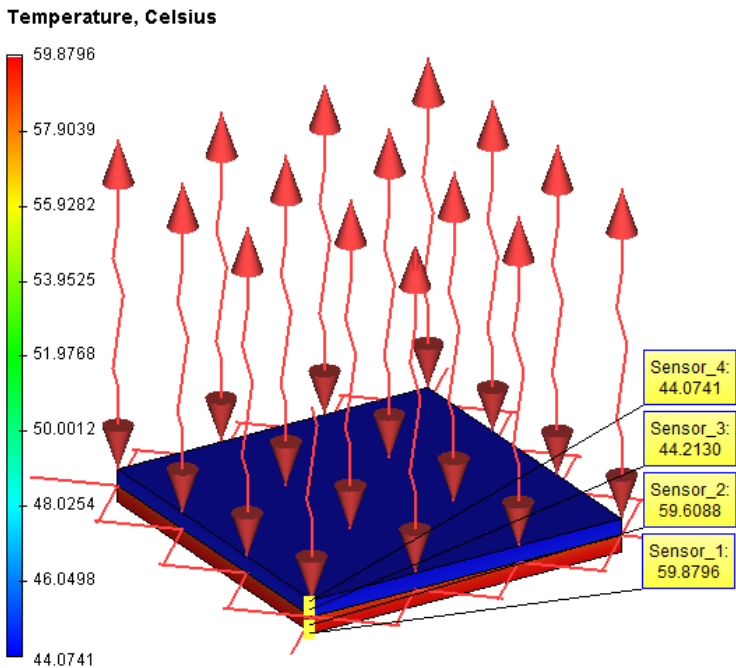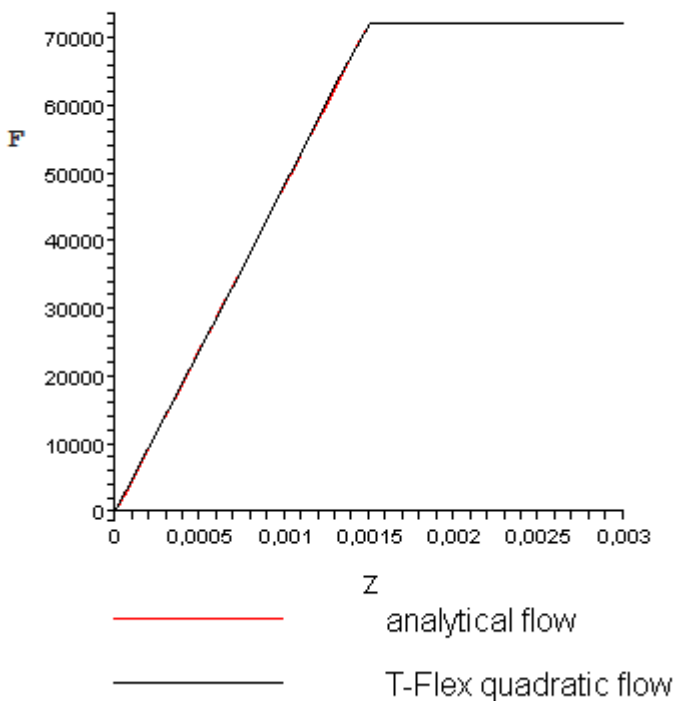
Results of numerical analysis with quadratic elements

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 study 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 study 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 T_1, and the remaining edges are at T_2.

As an example, let us consider a rectangular graphite plate with the thermal conductivity of K_1=278 W/(m $\cdot$ ^{0}C) along the principle direction and K_2=139 W/(m $\cdot$ ^{0}C) 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 ^{0}C 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 ^{0}C 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 ^{0}C, the heat transfer coefficient is H=400 W/(m$^2 \cdot$ ^{0}C). The thickness of the plate is D=2 mm.
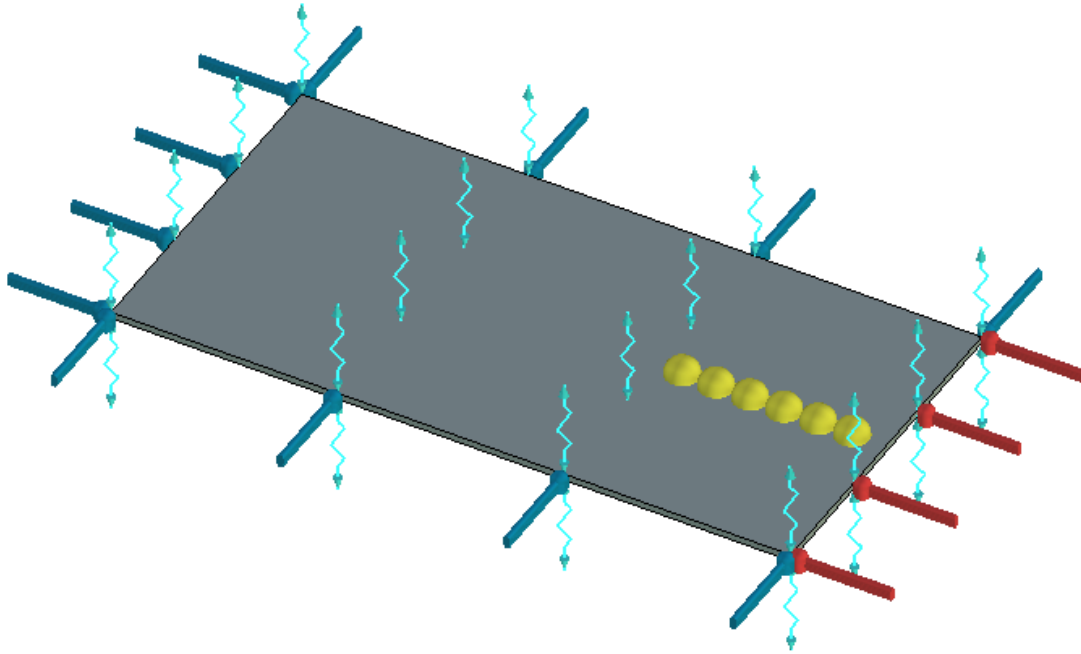
Numerical model with boundary conditions and sensors

For the given study, 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 study 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 study is expressed in terms of Fourier series and has the form[4]:

---

[4] Source: H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, The «Science» publishing house, M. 1964, p. 171.

$$u(x,y) = \frac{2}{a} \sum_{n=1}^{\infty} \frac{\sin(n\pi\xi/a) \cdot sh\left[(b-\zeta)\sqrt{k^2+n^2\pi^2/a^2}\right]}{sh\left(b\sqrt{k^2+n^2\pi^2/a^2}\right)} \Bigg|_0^a t \cdot \sin\left(\frac{n\pi x}{a}\right) dx$$

The isothermal lines of the obtained solution have the form shown on the Figure (obtained with Maple 9.5):
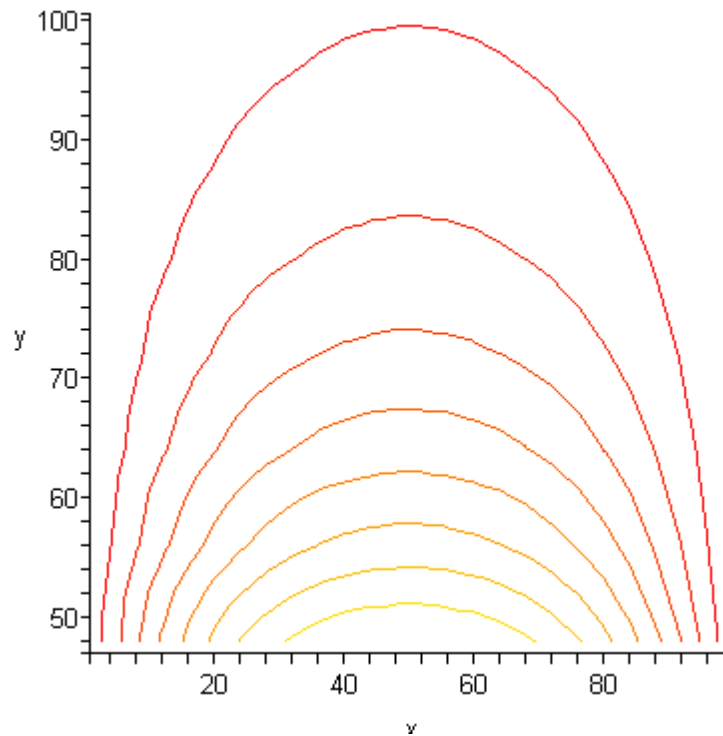
We can see similar distribution of the temperature field obtained in T-FLEX (see Picture below).

Temperature, Celsius
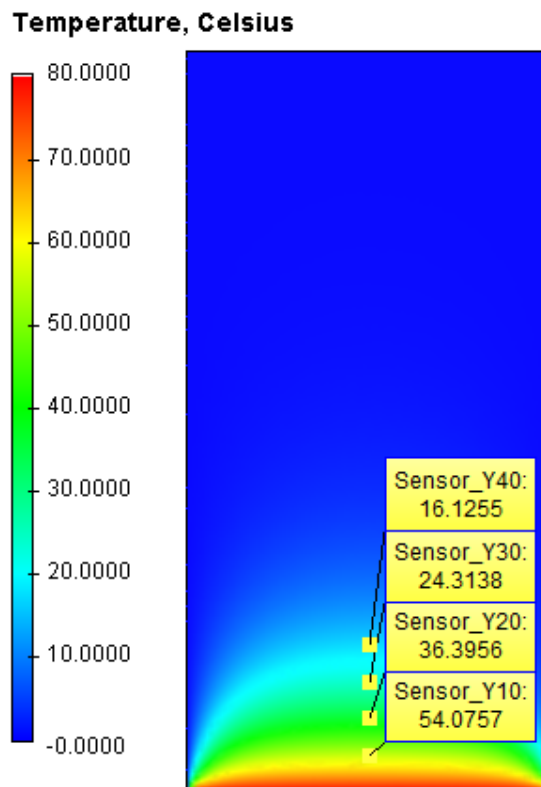
We locate temperature sensors as shown on the Figure and we make a plot for them at x=50 mm, y=10,20,30,...60 mm. At these points we compare the plot with the analytical solution.

Temperature graphics v(y)

Coordinate y is in millimeters
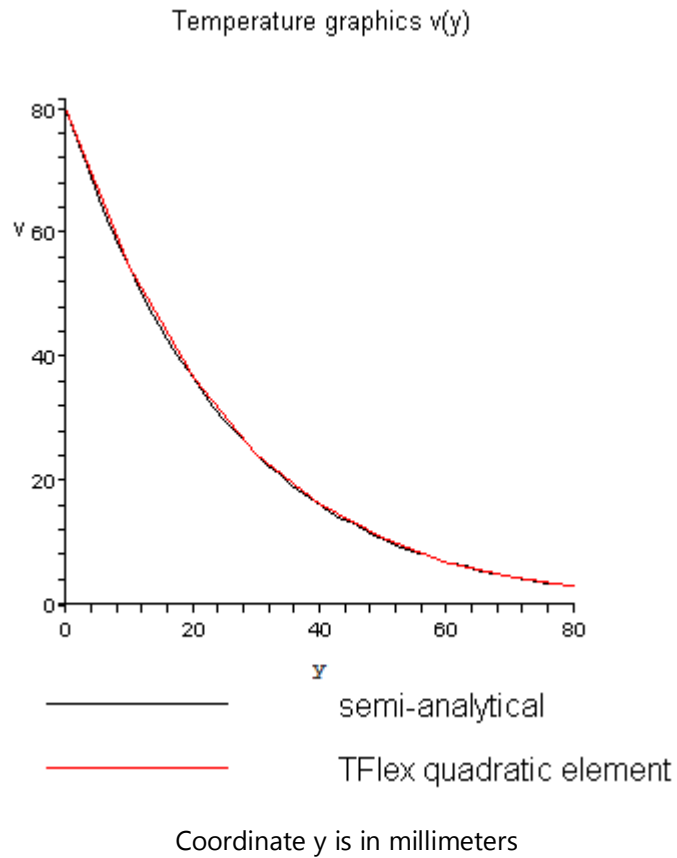
Let us estimate this discrepancy numerically. Analytical solution was obtained with the accuracy of up to 6 digits.

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 K_2=70 W/(m · ^{0}C). Then the temperature distribution has the form:
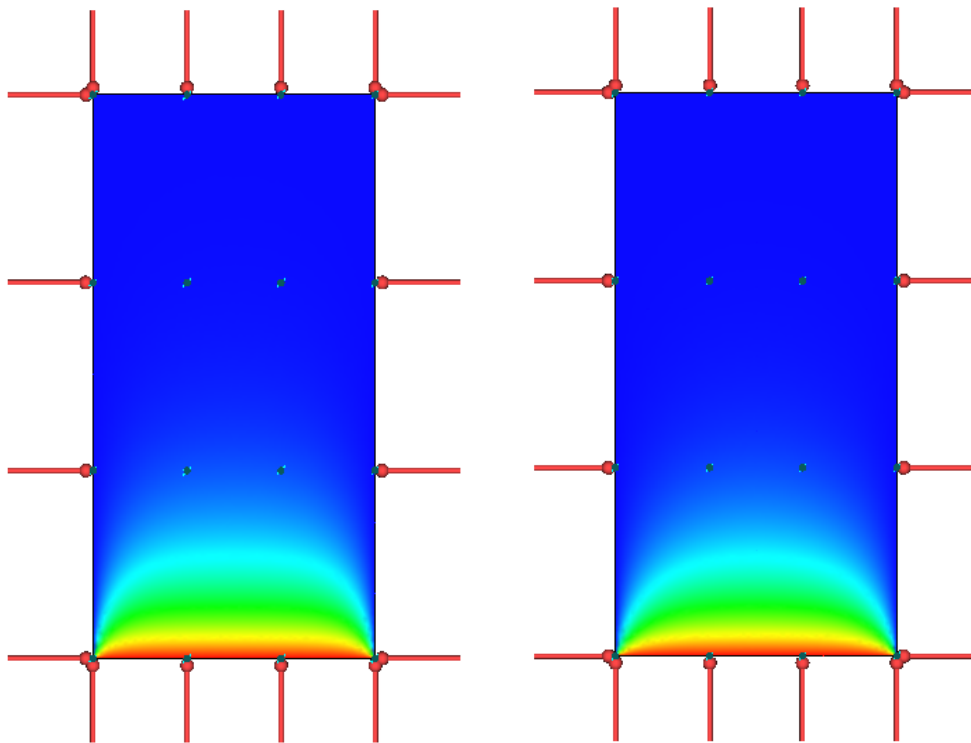
5 K2=70 W/(m * 0C) on the left and K2=139 W/(m * 0C) on the right

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 studies 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 study 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/(m$^2 \cdot$ ^{0}C) and the temperature of the ambient environment equal to $T0=20$ ^{0}C. 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 $R=0.01$ ^{0}C·m^2/W. The thermal conductivity of the internal cylinder is equal to $K2=50$ W/(m · ^{0}C), and external cylinder $K1=50$ W/(m · ^{0}C)[5].

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.

---

[5] Here we intentionally select equal K_1 and K_2 to show the influence, on the temperature jump on the interface, of only the thermal resistance but not the difference in the thermal conductivities. However, analytical solution given below includes the case when the thermal conductivities of the materials are not equal.
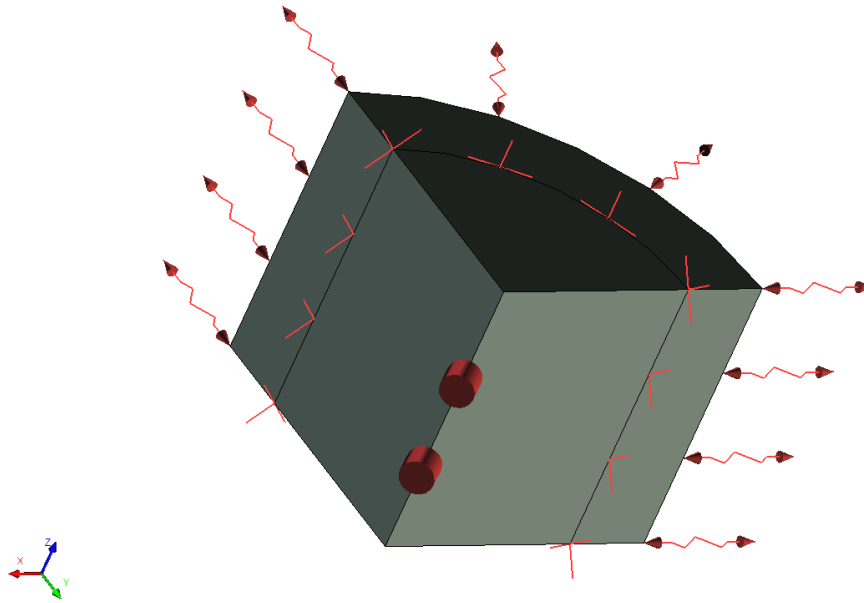
Numerical model with boundary conditions

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: $\rho = P / D$. 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[6]:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}K(r)\right) = \rho \cdot \delta(r),$$

$$K(r) = \begin{cases} K_1, r_2 < r \leq r_1 \\ K_2, 0 \leq r \leq r_2 \end{cases}$$

The boundary condition for this study corresponds to the heat transfer by convection

---

[6] Tikhonov A. N., Samarsky A. A. Equations of mathematical physics. P. 282 – form of a differential equation is obtained

$$-K_1 \frac{\partial u}{\partial r}(r_1) = H\big(u(r_1) - T_0\big)$$

Solution for this study has the form[7]:

$$u(r) = \begin{cases} \dfrac{\rho}{2\pi K_2} \ln|r| + C_2, & 0 \le r \le r_2 \\[2ex] \dfrac{\rho}{2\pi K_1} \ln|r| + C_1, & r_2 < r \le r_1 \end{cases}$$

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:

$$R = \frac{\left| \lim\limits_{r \to r_2+} u(r) - \lim\limits_{r \to r_2-} u(r) \right|}{F} = \frac{\left| u(r_2+) - u(r_2-) \right|}{F}$$

where *F* – thermal heat flux on the boundary can be found by the formula

$$F = \lim\limits_{r \to r_2-} -\frac{\partial u}{\partial r} K_2 = \lim\limits_{r \to r_2+} -\frac{\partial u}{\partial r} K_1$$

Let us compare the solution of T-FLEX Analysis with analytical solution of the study.

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

---

[7] Tikhonov A. N., Samarsky A. A. Equations of mathematical physics. P. 283 – the general solution is obtained

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, ^{0}C	173.3214	169.6338	76.07186
Numerical solution, ^{0}C	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, ^{0}C	173.3214	169.6338	76.07186
Numerical solution, ^{0}C	173.363	169.677	76.0716
Relative error, %	0.024002	0.025467	0.000342

Temperature graph v(r)
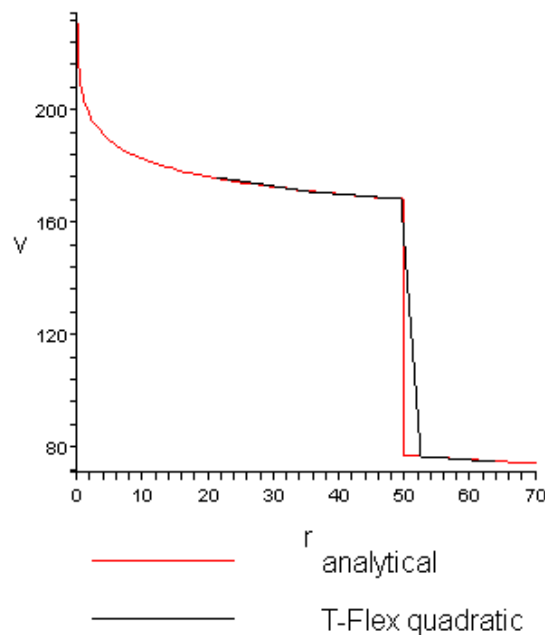
Units along the axes: r is in millimeters, v in 0C
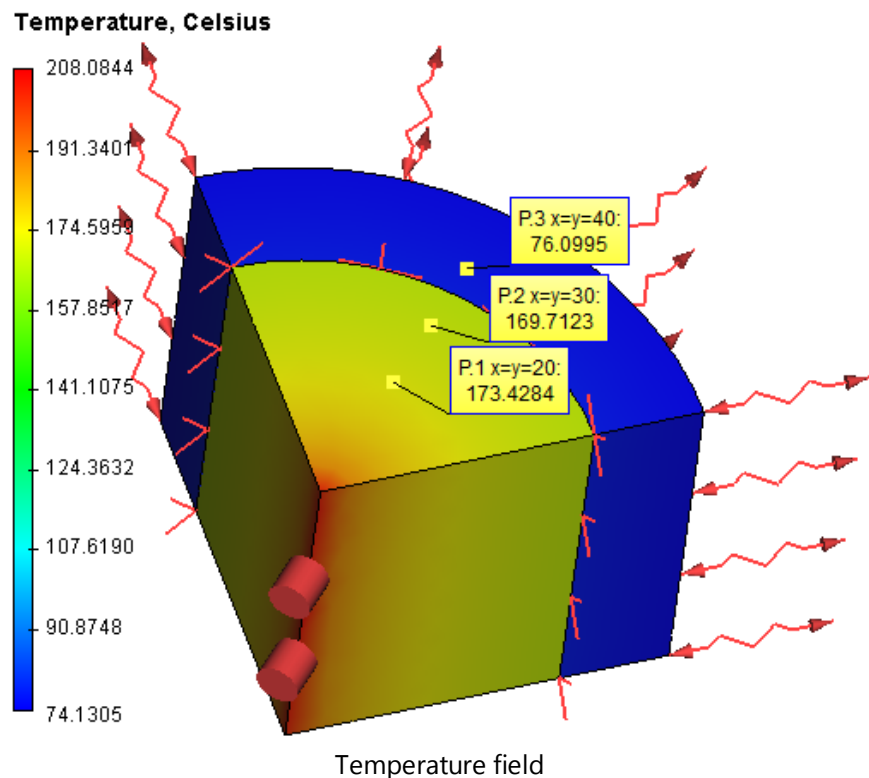
Temperature field

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

	28.28427	42.42640	56.56854
Coordinate of a point $r = |x|\sqrt{2}$, mm			
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

Coordinate of a point $r = |x|\sqrt{2}$, 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

Thermal Flux, magnitude, W/m^2

Max = 227452.7031

20000.0000

P.3 x=y=40:
8057.8564

P.2 x=y=30:
10791.9043

P.1 x=y=20:
16351.6123

6604.4365
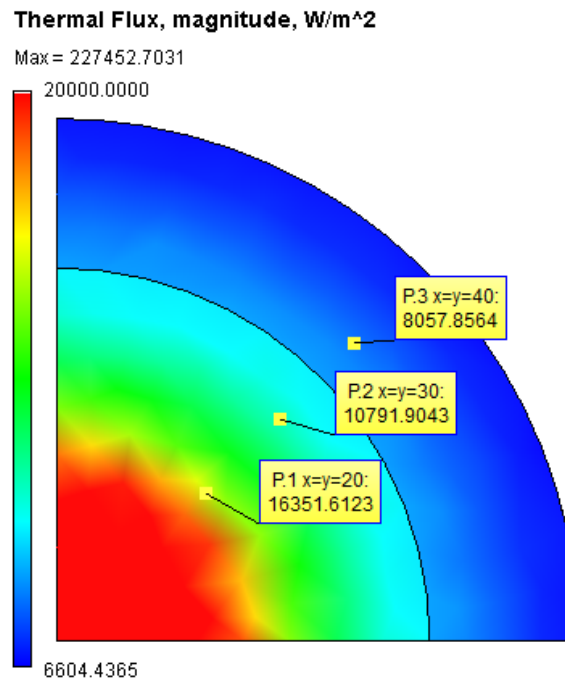
Picture of a heat flux

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 ^{0}C). Let us show that the steady-state heat transfer study 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/(m^2 · ^{0}C) 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/(m^2 · ^{0}C) and the temperature of the ambient environment equal to $u0$=10 ^{0}C.
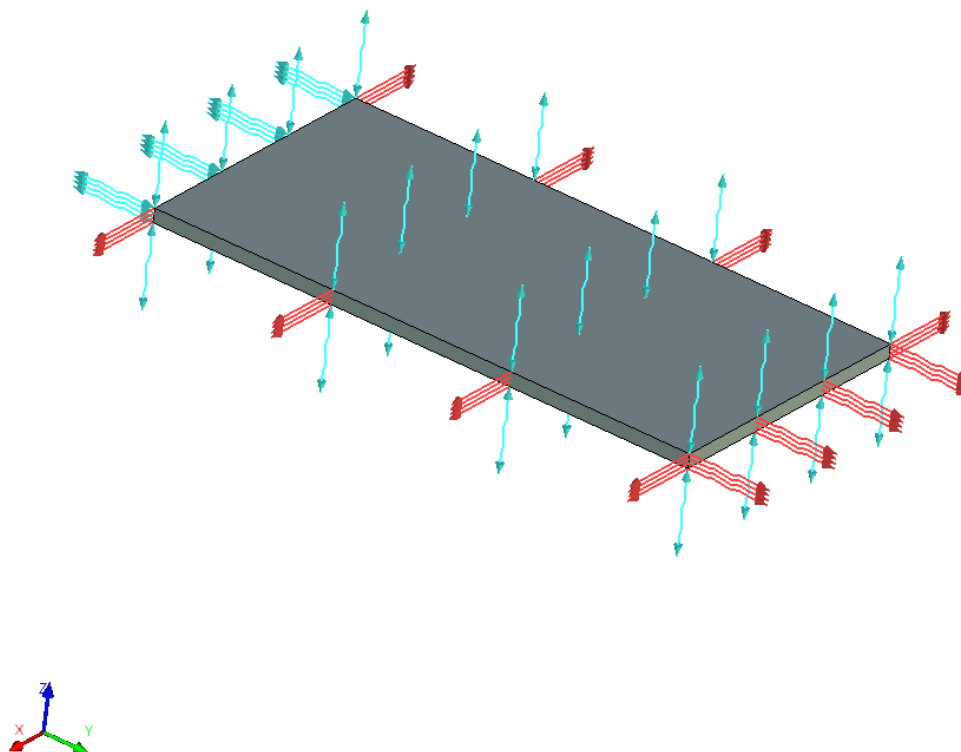
Numerical model with boundary conditions

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:

$$u(x, y) = V \cdot \left(e^{y\sqrt{L}} + e^{-y\sqrt{L}} \right),$$

$$L = \frac{C}{(K \cdot D)},$$

$$V = \frac{u_0}{\left(e^{b\sqrt{L}} - e^{-b\sqrt{L}} \right) - \frac{K}{H} \cdot \sqrt{L} \left(e^{b\sqrt{L}} + e^{-b\sqrt{L}} \right)}$$

Below we show the isothermal lines for the solution obtained using the Maple software.

Convection is applied for the coordinate Y=b.
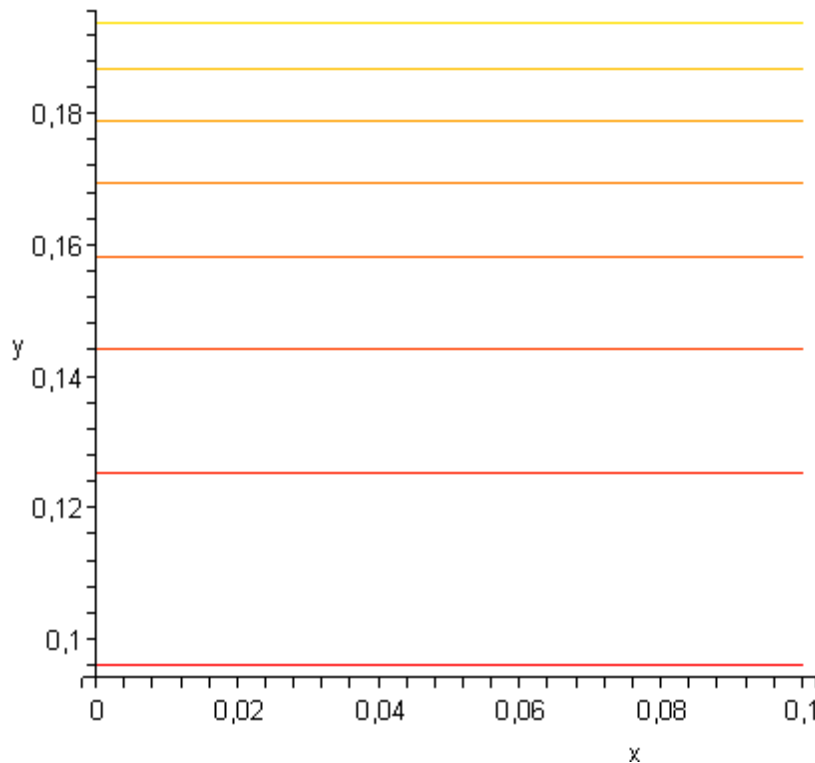
Isothermal lines for the solution obtained using the Maple software. Convection is applied to the edge y=b. When y=0 the thermal heat flux is zero

Comparison with the temperature field obtained using T-FLEX Analysis is shown inFig. 6.10-4.
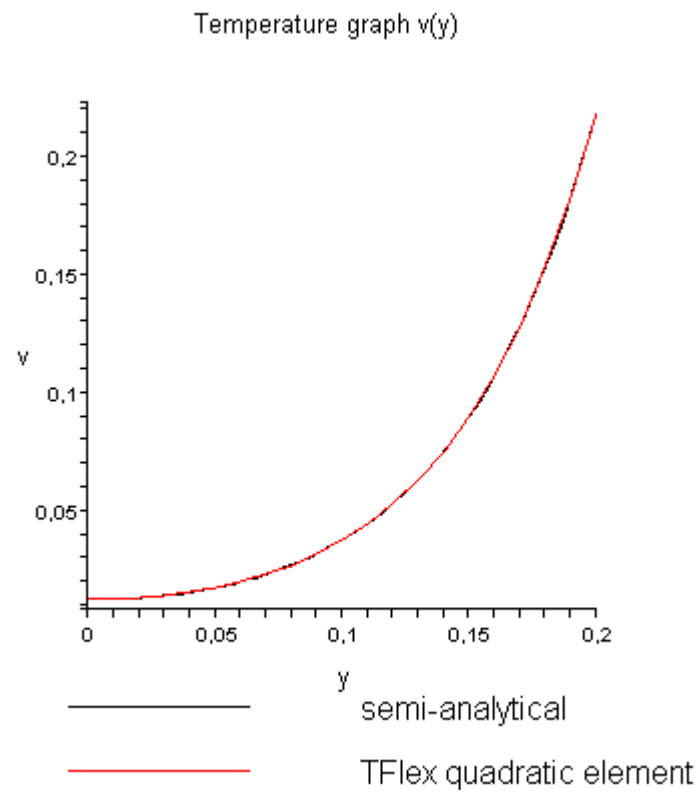
Units along the y-axis are in meters

Temperature, Celsius
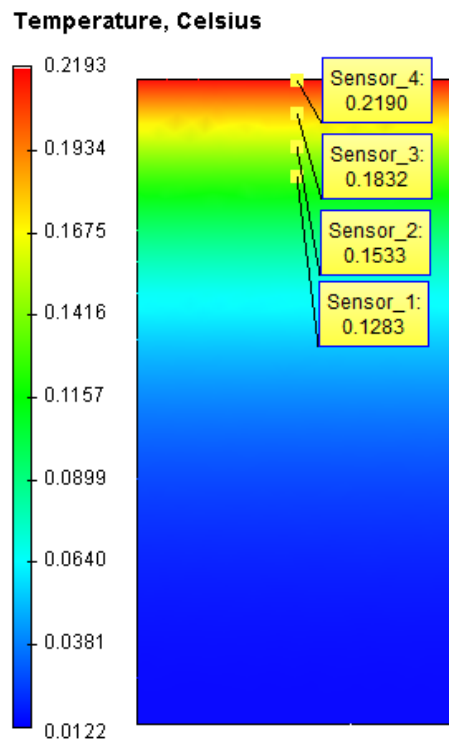
Let us compare the numerical solution and the solution obtained in the system T-FLEX Analysis.

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 study we obtained a solution with up to 3 significant digits of accuracy. The study 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 study 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 · ^{0}C) . 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/m^2. Temperature of the edges around the plate is T=20 ^{0}C.
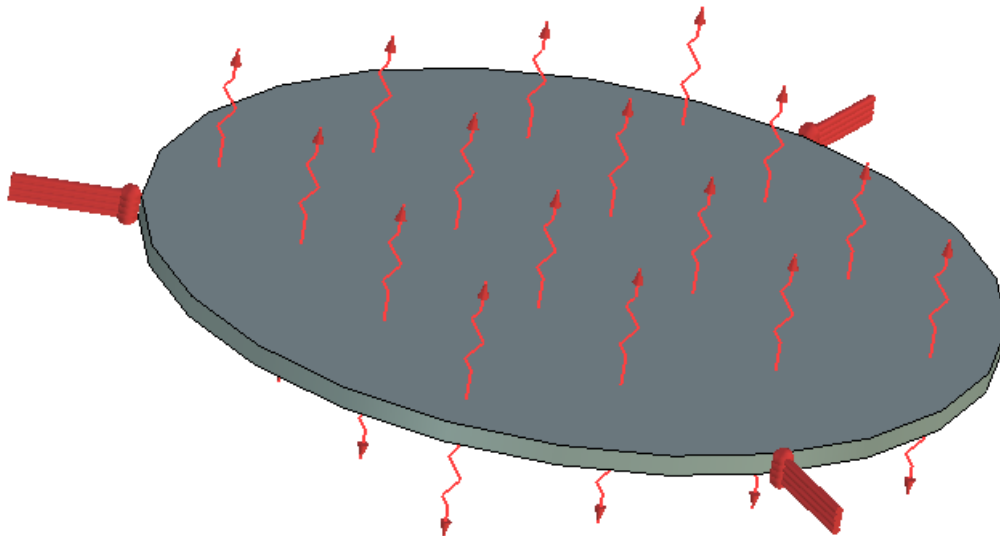
Numerical model with boundary conditions

Figure 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[8]:

$$\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)K = \frac{2F}{D}$$

At the right side of the equation we have the volumetric density of the heat source (equivalent to a surface density for a plane study).

Boundary conditions:

$$u(P) = T, P \in \partial\Omega \setminus (U_+ \cup U_-)$$

$$K\frac{\partial u}{\partial z}(P) = F, P \in U_+$$

$$K\frac{\partial u}{\partial z}(P) = -F, P \in U_-$$

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[9] G(P,P₀).

$$G(P, P_0) = \frac{1}{2\pi}\left(\ln\left(\frac{1}{r}\right) - \ln\left(\frac{R}{r_0 \cdot r_1}\right)\right), P_0 \neq 0$$

$$G(P, P_0) = \frac{1}{2\pi}\left(\ln\frac{1}{r} - \ln\frac{1}{R}\right), P_0 = 0$$

$$P_0 = P_0(x_0, y_0)$$

$$P = P(x, y)$$

$$r = \sqrt{(x - x_0)^2 + (y - y_0)^2}$$

$$r_1 = \sqrt{\left(x - x_0\frac{R^2}{r_0^2}\right)^2 + \left(y - y_0\frac{R^2}{r_0^2}\right)^2}$$

$$r_0 = \sqrt{x_0^2 + y_0^2}$$

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:

---

[8] Poisson's equation

[9] In the literature this function is also called Green's function.

$$u(P) = \int_{\Omega} f(P_0) G(P, P_0) dP_0,$$

$$\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} = f, f = \frac{2F}{D \cdot K} = const$$

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, ^{0}C	20.8000	20.7610	20.6720	20.5120
Numerical solution, ^{0}C	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, ^{0}C	20.8000	20.7610	20.6720	20.5120
Numerical solution, ^{0}C	20.7939	20.7609	20.6653	20.5049
Relative error, %	0.029326	0.000481	0.032410	0.034614

Temperature graph v(r)
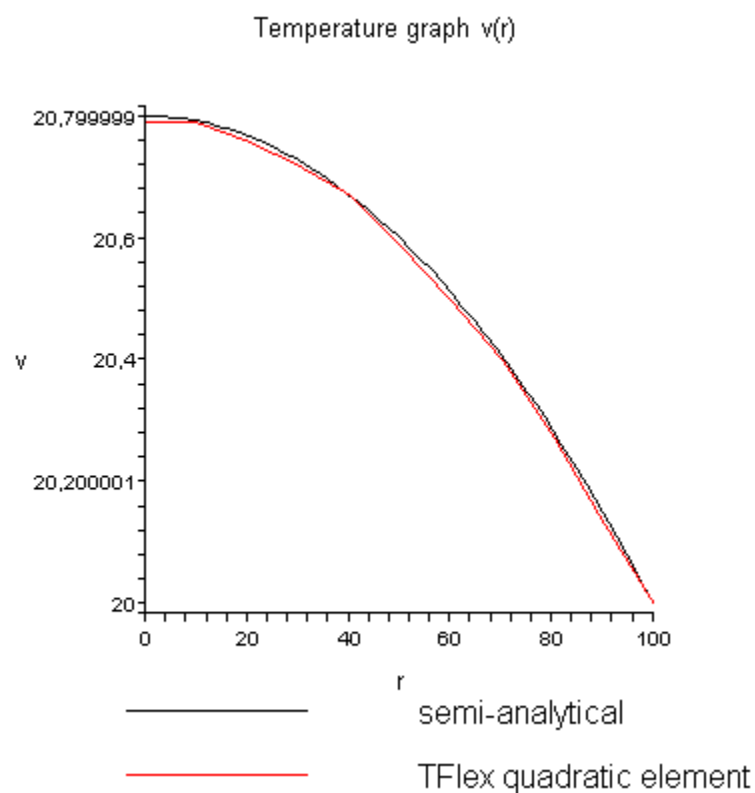
Units of radius measured from center of the disk; r is mm.
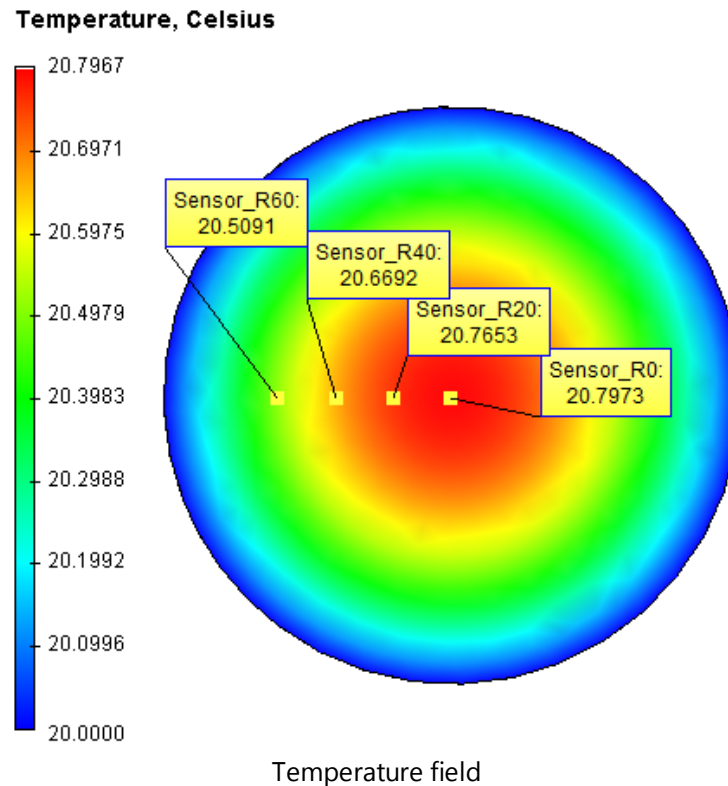
Temperature, Celsius

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 study 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 ^{0}K (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 T_{ext}=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 * ^{0}K); emissivity α=1. We maintain the zero heat flux along the boundary of a width of the plate.
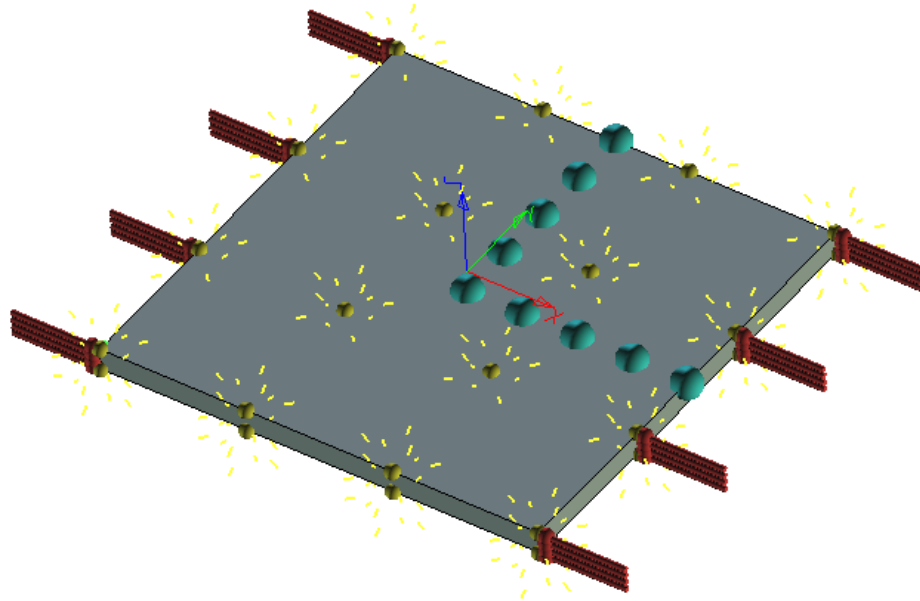
Numerical model with boundary conditions and sensors

Let us consider the differential equation for this study. 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 study can be found as a Taylor's series expansion. It can also be obtained numerically in the system Maple 9.5[10]. 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[11].

Let us represent now the analytical solution for the study 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:

$$u(x_0 + \Delta x) = u(x_0) + \frac{\partial u}{\partial x}(x_0) \cdot \Delta x + \frac{\partial^2 u}{\partial x^2}(x_0) \cdot \frac{\Delta x^2}{2} + \dots + \frac{\partial^n u}{\partial x^n}(x_0) \cdot \frac{\Delta x^n}{n!} + \dots$$

By differentiating the given equation, we will obtain the expressions for the higher-order derivatives:

$$\frac{\partial^3 u}{\partial x^3} = d\left(\frac{\partial^2 u}{\partial x^2}\right)/dx = \frac{\partial(u^4)}{\partial x} = 4 \cdot u^3 \cdot \frac{\partial u}{\partial x},$$

$$\frac{\partial^4 u}{\partial x^4} = d\left(4 \cdot u^3 \cdot \frac{\partial u}{\partial x}\right)/dx = 4 \cdot u^3 \cdot \frac{\partial^2 u}{\partial x^2} + 12 \cdot \left(u \cdot \frac{\partial u}{\partial x}\right)^2 = \left[\frac{\partial^2 u}{\partial x^2} = C\left(u^4 - T_{ext}^4\right)\right] =$$

$$= 4C \cdot u^7 - 4C \cdot u^3 T_{ext}^4 + 12 \cdot \left(u \cdot \frac{\partial u}{\partial x}\right)^2$$

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 $\dfrac{\partial^{2k+1} u}{\partial x^{2k+1}}(x_0) = 0$. Hence the solution will take the form

$$u(x_0 + \Delta x) = \sum_{k=0}^{\infty} \frac{\partial^{2k} u}{\partial x^{2k}}(x_0) \cdot \frac{\Delta x^{2k}}{(2k)!}$$

---

[10] Solution was obtained by two methods and the difference was 1e-8 % at the points indicated below

[11] From the physical point of view it is obvious because we have equal temperature along the edges and the material is isotropic. In the center we will have the point of minimum since the body looses heat.

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 $\dfrac{\partial^{2k} u}{\partial x^{2k}}(x_0)$ 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[12].

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

---

[12] In the cycle over the number of the terms in the series we performed differentiation and substitution of the fourth power of the solution instead of second derivative $\dfrac{\partial^{2k} u}{\partial x^{2k}}(x)$ and of zero instead of the first derivative into the final expression.
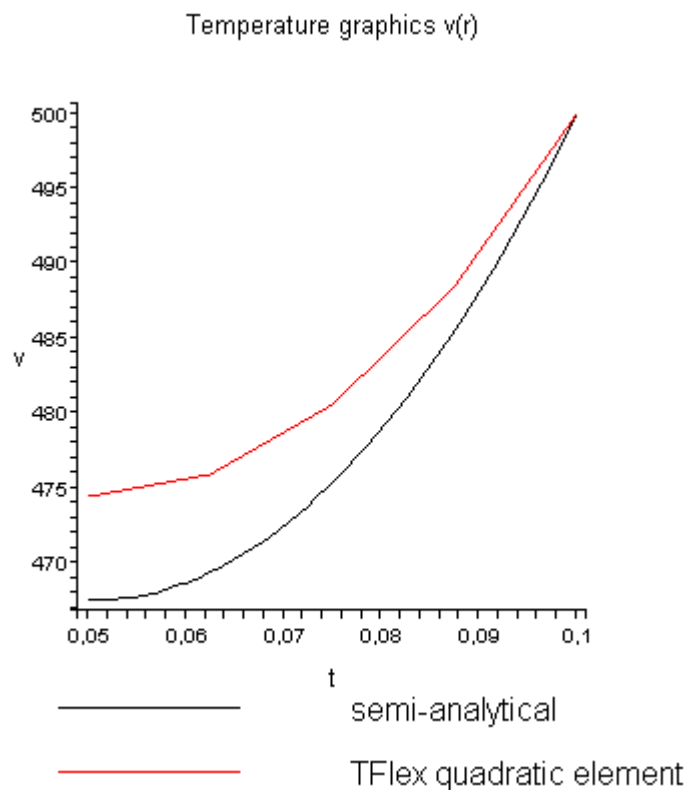
Plot of temperature distribution along the width of the plate
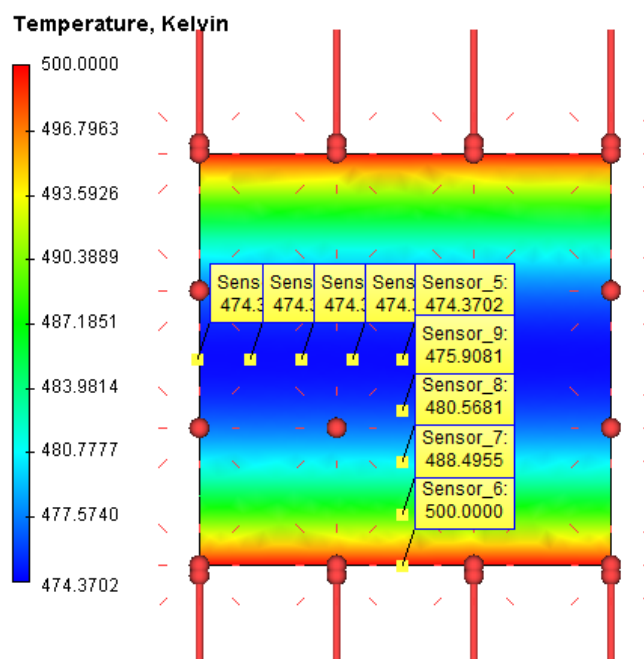
Plot of a temperature field and the sensors

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 studies with radiation since the study 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 study 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 study 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 ^{0}K, and the external surface radiates to the ambient environment. The ambient environment has the temperature $Tокр$=290 ^{0}K. 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 R_{123}.

Characteristics of the sphere: external radius R=100 мм; internal radius R/2=50 mm; the thermal conductivity K=50 W/(m · ^{0}K); 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.
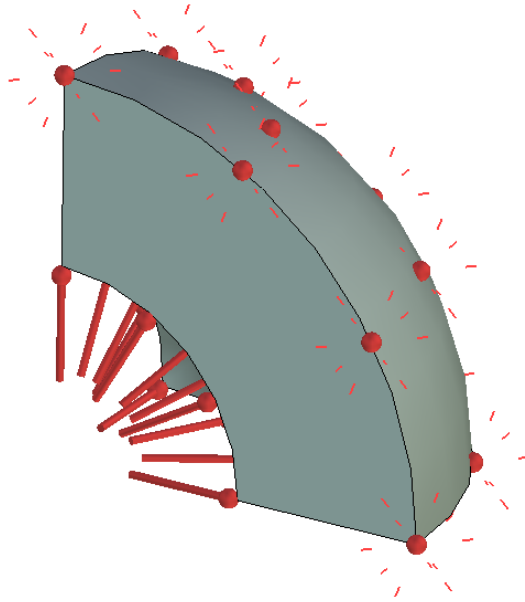
Numerical model with boundary conditions

Since the material is isotropic, solution of the study 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:

$$C_1 = \frac{\alpha \cdot \sigma \cdot R}{K}\left(T_{amb}^4 - \left(T_1 + C_1 \cdot \ln(2)\right)^4\right)$$

$$C_2 = T_1 - C_1 \cdot \ln\left(\frac{R}{2}\right)$$

The first equation (for the given parameters) has two real roots[13]. 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 [14].

Let us compare the numerical solution with the obtained analytical solution at points with radii R_{123}=0.0707106; 0.0848528; 0.0989949 m (coordinates of the sensors at these points: X_{123}=50; 60; 70 mm and Y_{123}=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

---

[13] Roots were found with the help of the Maple 9.5 system

[14] Difference between the obtained roots was less than 1e-10
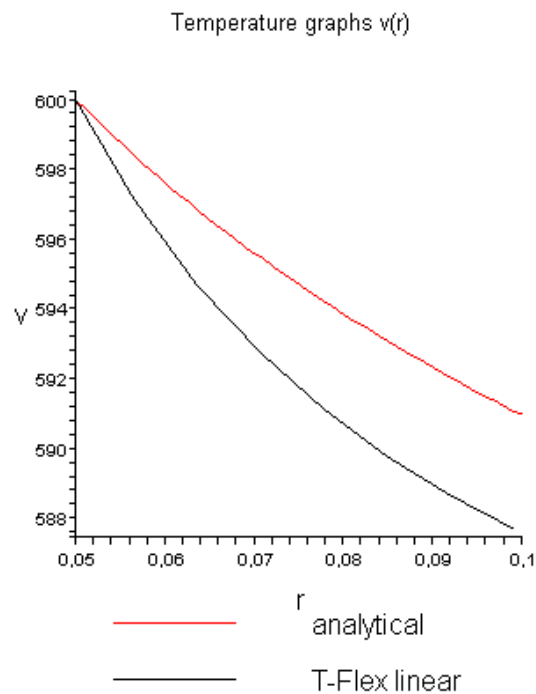
Plot of dependence of the temperature V on radius r in the spherical coordinate system
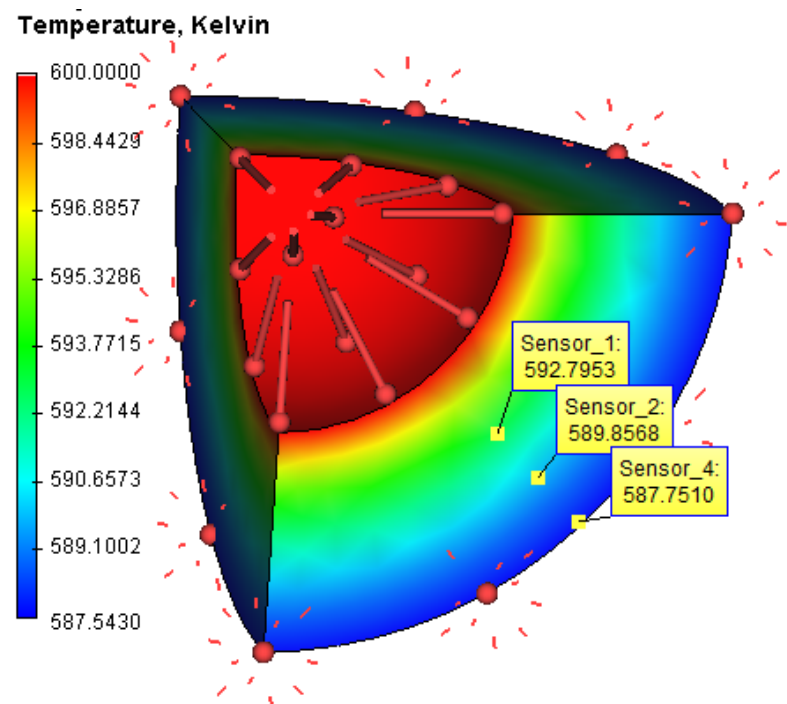
Temperature field plotted from the results of the finite element analysis with the readings of the sensors

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 study 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 study 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).
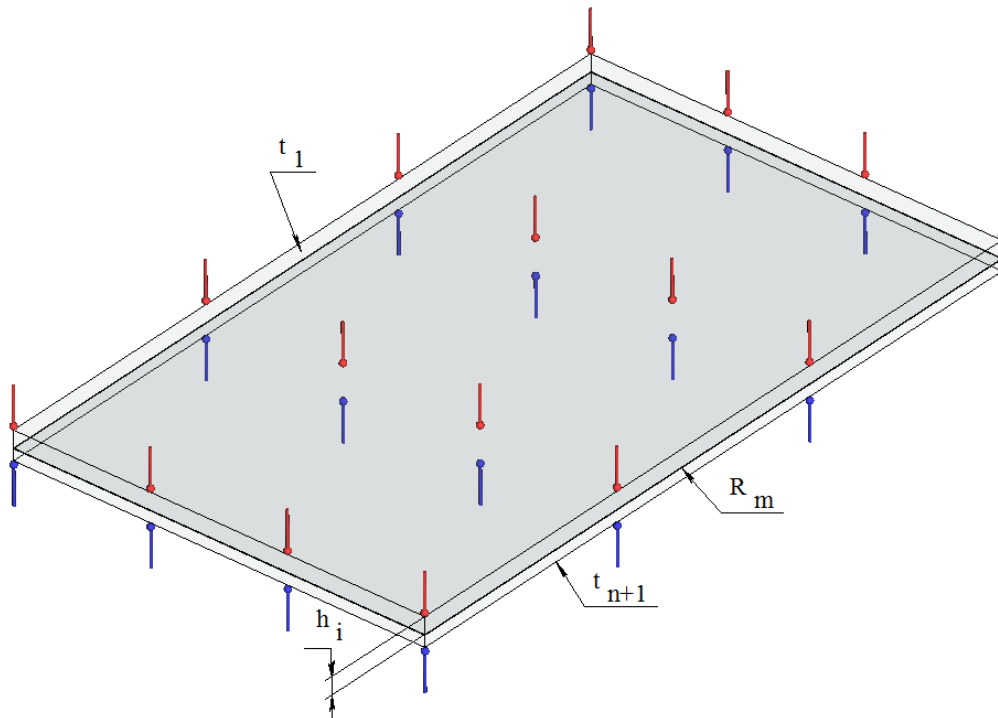
For each layer $i = 1,2,\ldots,n$ of the composite plate composed of n layers with thicknesses $h_1, h_2, \ldots, h_n$ and coefficients of thermal conductivity $k_1, k_2, \ldots, k_n$, respectively, the change in temperature and thermal flow across the thickness $f_i, i = 1,2,\ldots,n$ can be obtained from formula:

$$f_i = -k_i \frac{\partial T}{\partial z} = -\frac{k_i (t_{i+1} - t_i)}{h_i} = \frac{(t_i - t_{i+1})}{R_i}, \; R_i = \frac{h_i}{k_i}, \; t_i < t_{i+1}, \; i = 1,2,\ldots,n$$

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, $f_1 = f_2 = \ldots = f_n = f$). The change of temperature between opposite surfaces of the entire composite plate will be equal to the sum of temperature changes in individual layers:

$$(t_1 - t_2) + (t_2 - t_3) + \ldots + (t_i - t_{i+1}) + \ldots + (t_n - t_{n+1}) = t_1 - t_{n+1}$$

Then

$$t_1 - t_{n+1} = f_1 R_1 + f_2 R_2 + \ldots + f_n R_n = (R_1 + R_2 + R_m + \ldots + R_n)f, \; R_i = \frac{h_i}{k_i}, \; i = 1,2,\ldots n$$

$$f = \frac{t_1 - t_{n+1}}{\dfrac{h_1}{k_1} + \dfrac{h_2}{k_2} + \ldots R_m + \ldots + \dfrac{h_n}{k_n}}$$

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: $k_1 = 43 \dfrac{W}{m \cdot \deg}$, $k_3 = 200 \dfrac{W}{m \cdot \deg}$

Thermal resistance of the contact surface: $R_2 = 3.33 \times 10^{-4} \dfrac{m^2 \cdot \deg}{W}$ (approximately, equivalent to the thermal resistance of the flat layer of air of thickness 0,05 mm with $k_2 = 0.15 \dfrac{W}{m \cdot \deg}$)

Therefore, $f = 1.72599 \times 10^5 \dfrac{W}{m^2}$, $t_2 = -R_1 f + t_1 = -1.860465 \times 10^{-4} \cdot 1.72599 \times 10^5 + 373 = 340.88 \, K$,

$t_3 = -(R_1 + R_2)f + t_1 = -(1.860465 \times 10^{-4} + 3.33 \times 10^{-4}) \cdot 1.72599 \times 10^5 + 373 = 283.35 \, K$.
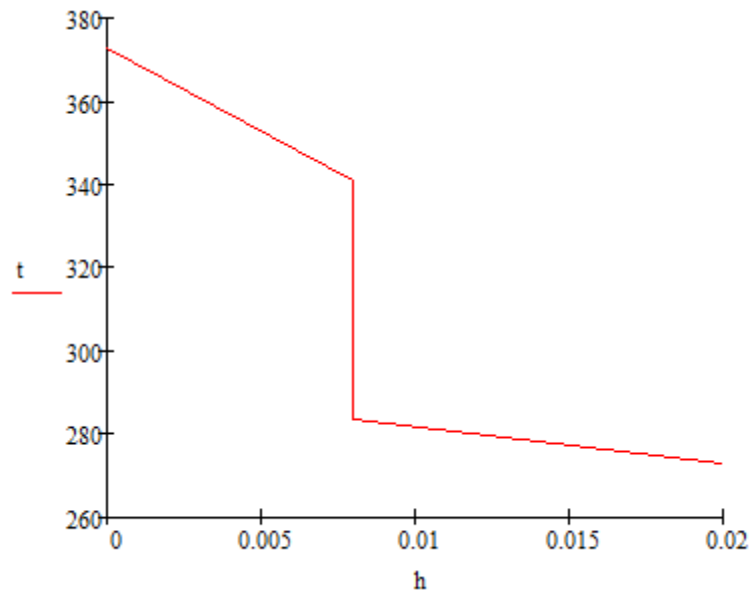
After calculations are carried out with the help of T-FLEX, we obtain the following results:

Mesh parameters			
Finite element type:	quadratic tetrahedron (10 nodes)		
Number of main nodes:	68155	Number of nodes for study calculation:	68155
Number of finite elements	44043		
Calculation results			
	Numerical solution, w^*	Analytical solution w	Error $$\delta_u = \frac{\left| w - w^* \right|}{w} \times 100\%$$
Heat flow, $\dfrac{W}{m^2}$	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 %

Study: "Study_1"
Temperature, Kelvin

Thermal Flux, magnitude, W/m^2
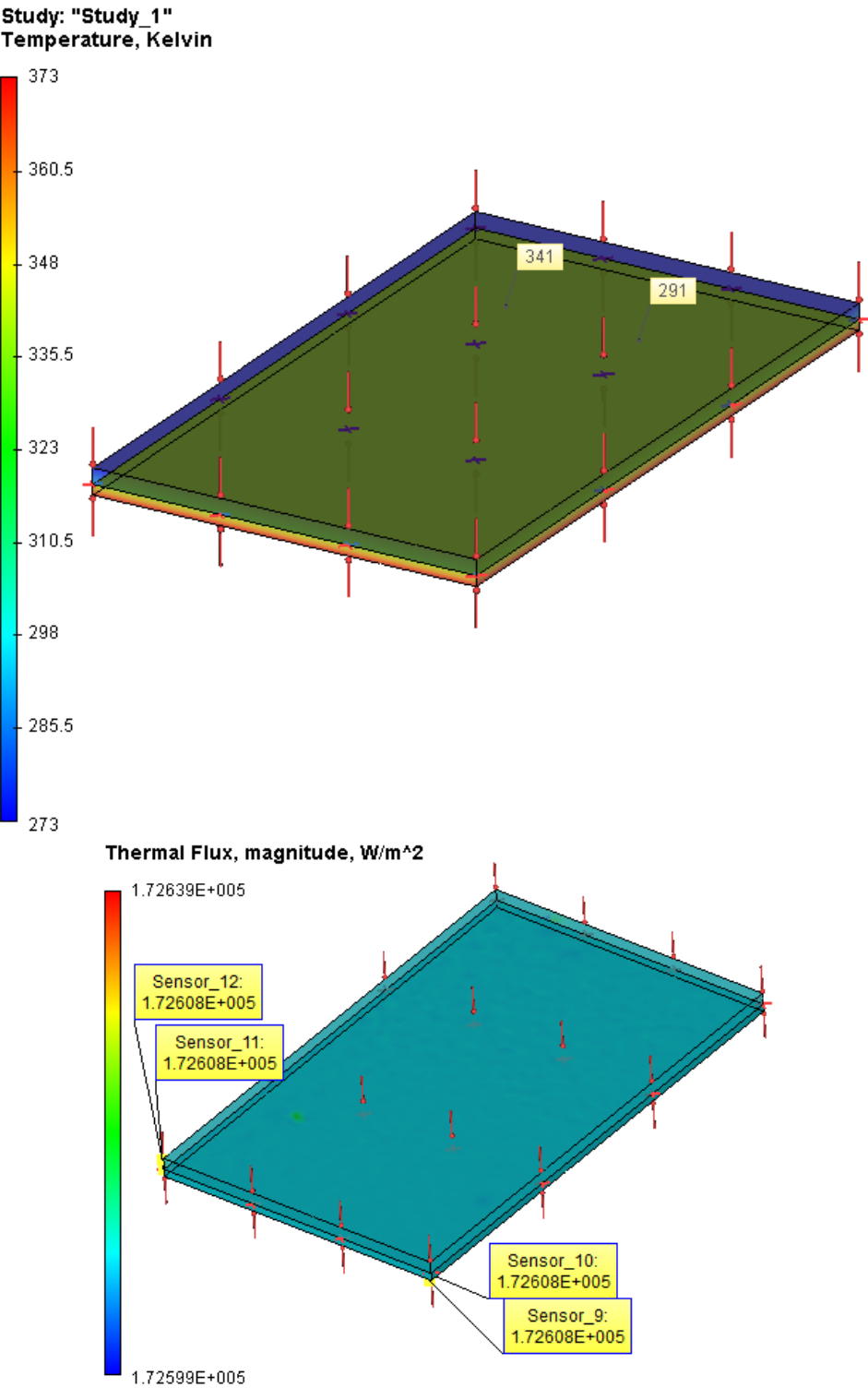
Thermal resistance of a sphere

Let us consider a study 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).
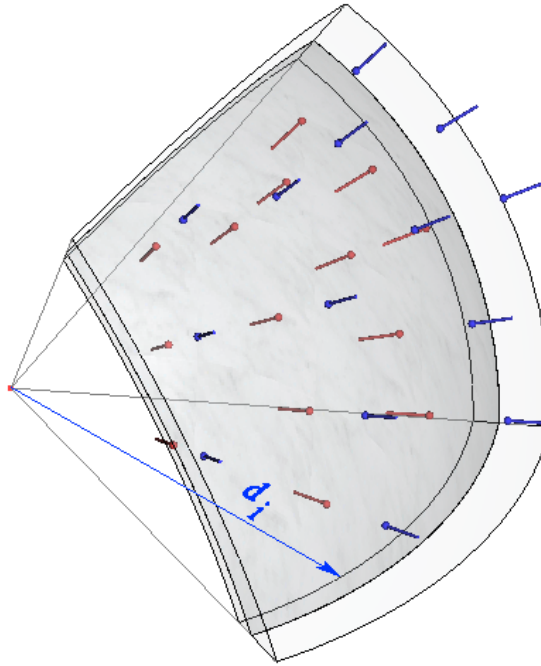
For each layer $i = 1,2,\ldots,n$ of the composite plate composed of n layers with diameters $d_1, d_2, \ldots, d_n$ and coefficients of thermal conductivity $k_1, k_2, \ldots, k_n$, respectively, the change of temperature and thermal heat flow across the thickness of the plate $f_i, i = 1,2,\ldots,n$ can be determined from formula:

$$f_i = -k_i \frac{\partial T}{\partial z} = -\frac{2\pi k_i (t_{i+1} - t_i)}{\left(\dfrac{1}{d_i} - \dfrac{1}{d_{i+1}}\right)} = \frac{(t_i - t_{i+1})}{R_i}, \; R_i = \frac{\left(\dfrac{1}{d_i} - \dfrac{1}{d_{i+1}}\right)}{2\pi k_i}, \; t_i < t_{i+1}, \; i = 1,2,\ldots,n$$

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, $f_1 = f_2 = \ldots = f_n = f$). The change of temperature between opposite

surfaces of the entire composite plate will be equal to the sum of temperature changes in individual layers:

$$(t_1 - t_2) + (t_2 - t_3) + \ldots + (t_i - t_{i+1}) + \ldots + (t_n - t_{n+1}) = t_1 - t_{n+1}$$

Then

$$t_1 - t_{n+1} = f_1 R_1 + f_2 R_2 + \ldots + f_n R_n = (R_1 + R_2 + R_m + \ldots + R_n)f, \quad i = 1,2,\ldots n$$

$$f = \frac{t_1 - t_{n+1}}{\dfrac{\left(\dfrac{1}{d_1} - \dfrac{1}{d_2}\right)}{2\pi k_1} + \dfrac{\left(\dfrac{1}{d_2} - \dfrac{1}{d_3}\right)}{2\pi k_2} + \ldots R_m + \ldots + \dfrac{\left(\dfrac{1}{d_n} - \dfrac{1}{d_{n+1}}\right)}{2\pi k_n}}$$

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, $f = 7.8917 \times 10^4 \dfrac{W}{m^2}$,

$$t_2 = -R_1 f + t_1 = 8.349 \times 10^{-4} \cdot 7.8917 \times 10^4 + 473 = 407.1 \, K$$

$$t_3 = -(R_1 + R_2)f + t_1 = -(8.349 \times 10^{-4} + 1.5 \times 10^{-3}) \cdot 7.8917 \times 10^4 + 473 = 288.7 \, K.$$
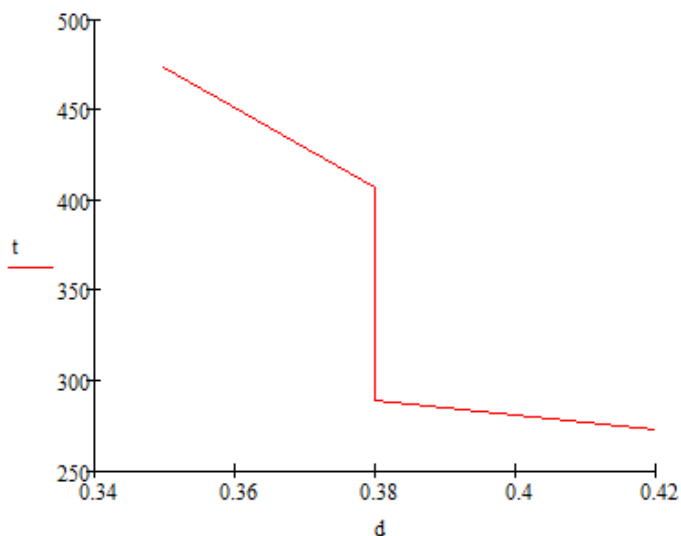
After calculations are carried out with the help of T-FLEX, we obtain the following results (at the lower and upper sides of the contact surface, respectively):
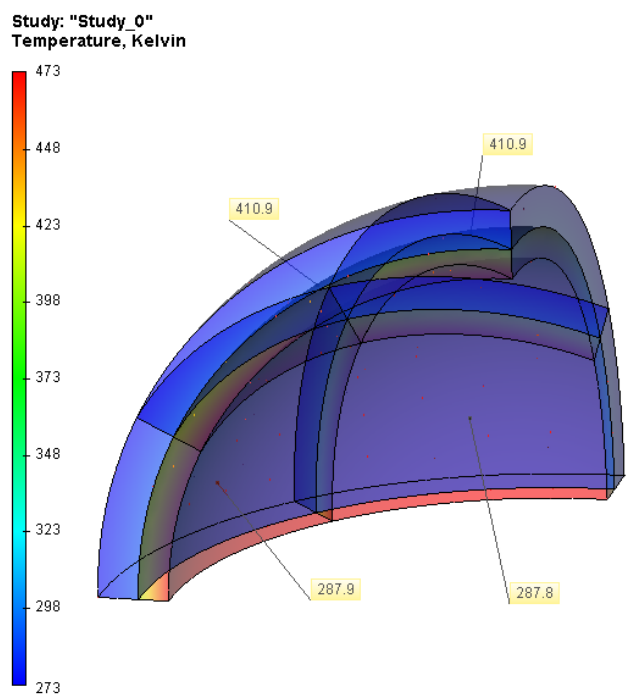
Results of finite element analysis (temperature)

Mesh parameters		
Finite element type	quadratic tetrahedron (10 nodes)	
Number of main nodes:	22115	Number of nodes for study calculation:	22115

Number of finite elements	101217		
Results of calculation			
	Numerical solution, w^*	Analytical solution w	Error $\delta_u = \dfrac{\left|w-w^*\right|}{w} \times 100\%$
Heat flow, $\dfrac{W}{m^2}$	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 study calculation:	22107
Number of finite elements	101190		
Results of calculation			
	Numerical solution, w^*	Analytical solution w	Error $\delta_u = \dfrac{\left|w-w^*\right|}{w} \times 100\%$
Heat flow, $\dfrac{W}{m^2}$	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	287.9	288.7	-0.3 %

Non-stationary temperature field for an isotropic sphere

Let us consider a study of determination of a temperature field inside an isotropic sphere in the volume of which we prescribe the initial temperature *Tstart*=20 ^{0}C, and which is heated along the external surface where we prescribe the constant temperature *Tborder*=100 ^{0}C (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 $\Delta t1,2,3 = 10$, 20, 30 seconds.

The sphere has the following parameters: radius a =100 mm, density of the material 7800 kg / m^3, specific heat c = 440 J / (kg ·^{0}C), thermal conductivity K=50 W / (m · ^{0}C). 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.[15]
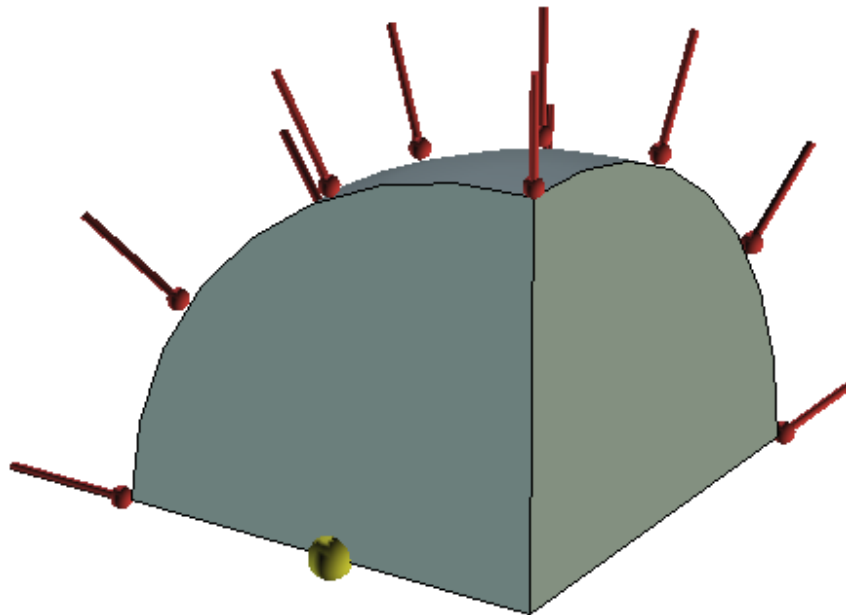
Numerical model with boundary conditions and a sensor located at a coordinate r=50 mm

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 $v=r \cdot u$, where r – distance from the center of the sphere, and u – some function. After this change of variables, we obtain the equation for u

---

[15] On the boundaries where no boundary conditions are specified, the zero heat flux condition is satisfied automatically.

$$\frac{\partial u}{\partial t} = \frac{K}{c \cdot \rho} \cdot \frac{\partial^2 u}{\partial r^2}$$

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 study by the method of separation of variables, we obtain the expression for v:[16]

$$v(r,t) = \frac{2}{ar} \sum_{n=1}^{\infty} A_n \cdot e^{-\chi\left(\frac{n^2\pi^2}{a^2}\right)t} \cdot \sin\left(\frac{n\pi r}{a}\right)$$

where $\chi = K/(c \cdot \rho)$ is the coefficient of temperature conductivity. The weights An in the expansion are determined by the formula:

$$A_n = \int_0^a r' \cdot f(r')\sin\left(\frac{\pi n r'}{a}\right)dr' - n\pi\chi(-1)^n \int_0^t e^{\chi\left(\frac{n^2\pi^2}{a^2}\right)\lambda} \cdot \varphi(\lambda)d\lambda =$$

$$= \frac{a^2}{n\pi}(-1)^{n+1} \cdot \left\{ T_{start} + T_{border}\left(e^{\chi\left(\frac{n^2\pi^2}{a^2}\right)t} - 1\right) \right\}$$

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 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 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

---

[16] H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, The «Science» Publishing house, Moscow 1964, p. 230

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 $\delta = \dfrac{\left|u_n - u_{2n}\right|}{\left|u_n\right|} \cdot 100\%$
	10	20.48013866	20.41647716	0.31
500	20	26.07028450	26.00662298	0.24
	30	34.46547534	34.40181384	0.18
	10	20.53470594	20.53925332	0.02
7000	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 ^{0}C 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, ^{0}C	20.53	26.12	34.52
Numerical solution, ^{0}C	20.61	26.16	34.58
Relative error, %	0.3896	0.1531	0.1738

Temperature graphs v(t), t - for time r=50
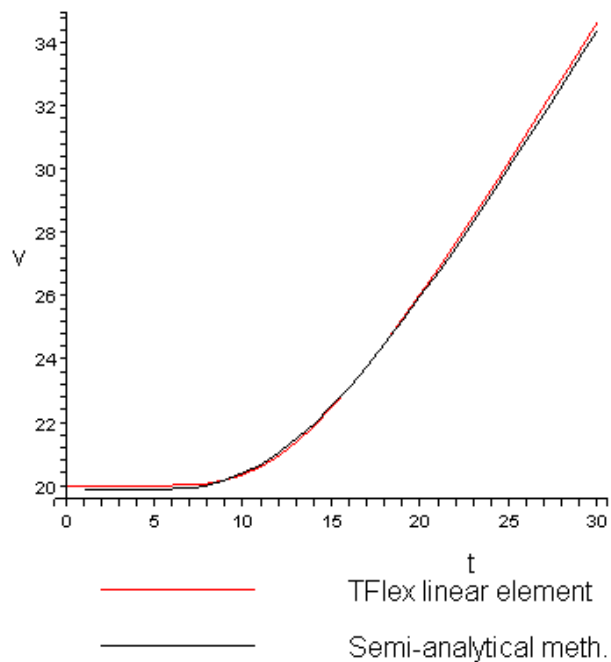
Plot of dependence of temperature V(t) at a point r=50 mm on time t
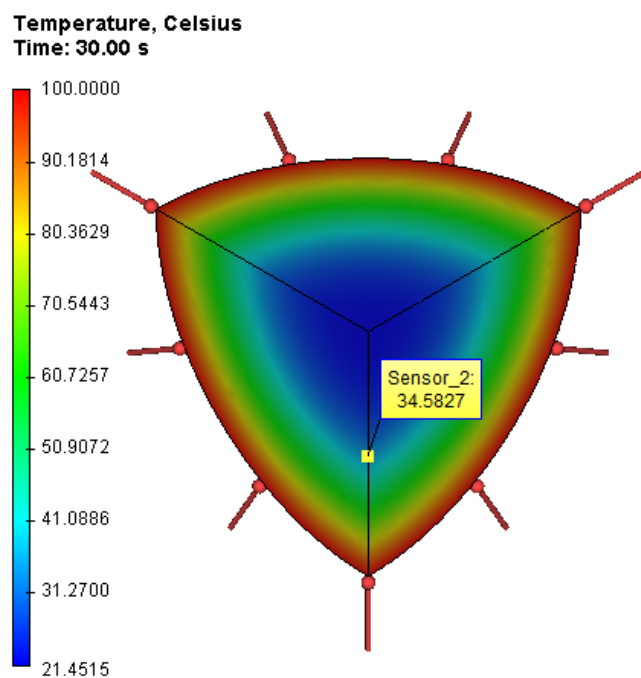
Temperature field in 30 seconds

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 study 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/(m^2 °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 *Δt1,2,3* =20, 60, 90, 120 sec.

Parameters of the sphere: radius *a*=100 mm, density of the material 7800 kg/m^3, 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.[17]
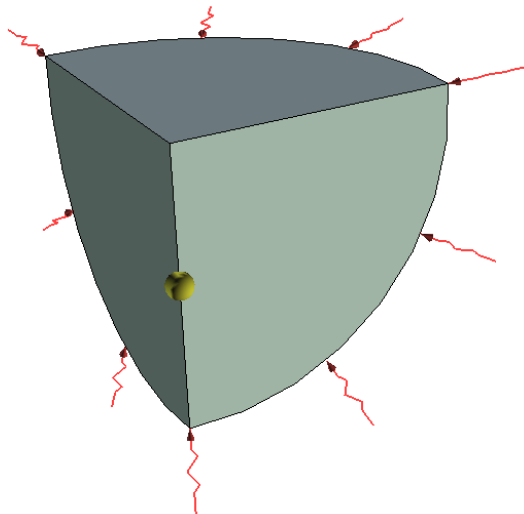
Numerical model with boundary conditions

---

[17] On the boundaries where no boundary conditions are specified, the zero heat flux condition is satisfied automatically.

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 study is given below.[18]

$$v(r,t) = \frac{3Ft}{\rho ca} + \frac{F(5r^2 - 3a^2)}{10Ka} - \frac{2Fa^2}{Kr}\sum_{n=1}^{\infty}\frac{\sin(r\alpha_n/a)}{\alpha_n^2 \sin(\alpha_n)} \cdot e^{-\chi \cdot \alpha_n^2 \cdot t/a^2} + T_{start}$$

where *χ=K/(c·ρ)* is the coefficient of temperature conductivity. The coefficients α_n 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

---

[18] H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, The «Science» publishing house, Moscow 1964, p. 234

Authors derived the solution for zero initial temperature. The constant T_{start} must be added to their solution

Tables of the values of temperature in ^{0}C 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, ^{0}C	79.9481	79.7080	79.5163	79.3240
Numerical solution, ^{0}C	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, ^{0}C	79.9481	79.7080	79.5163	79.3240
Numerical solution, ^{0}C	79.9479	79.7075	79.5155	79.3231
Relative error, %	0.00035	0.00070	0.00107	0.00126
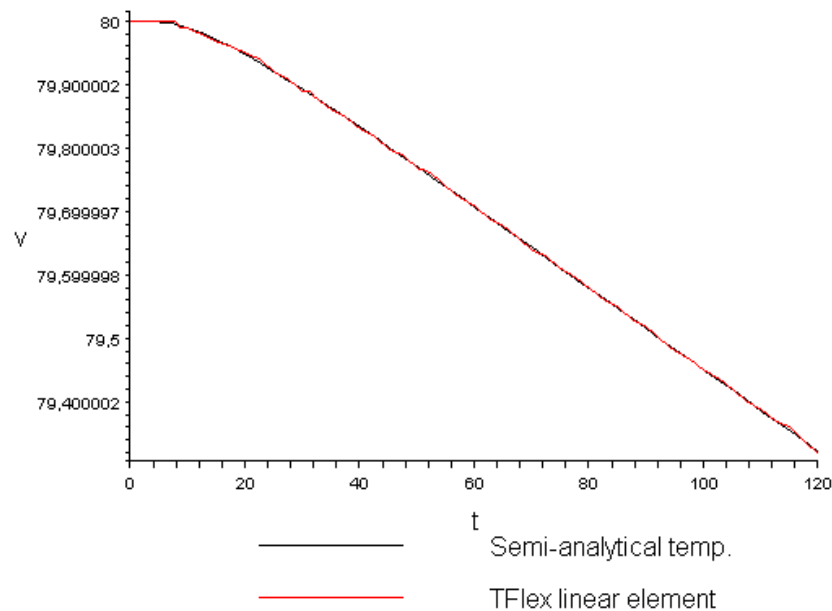
Plot of dependence of temperature V(t) on time t at a point r=50 mm
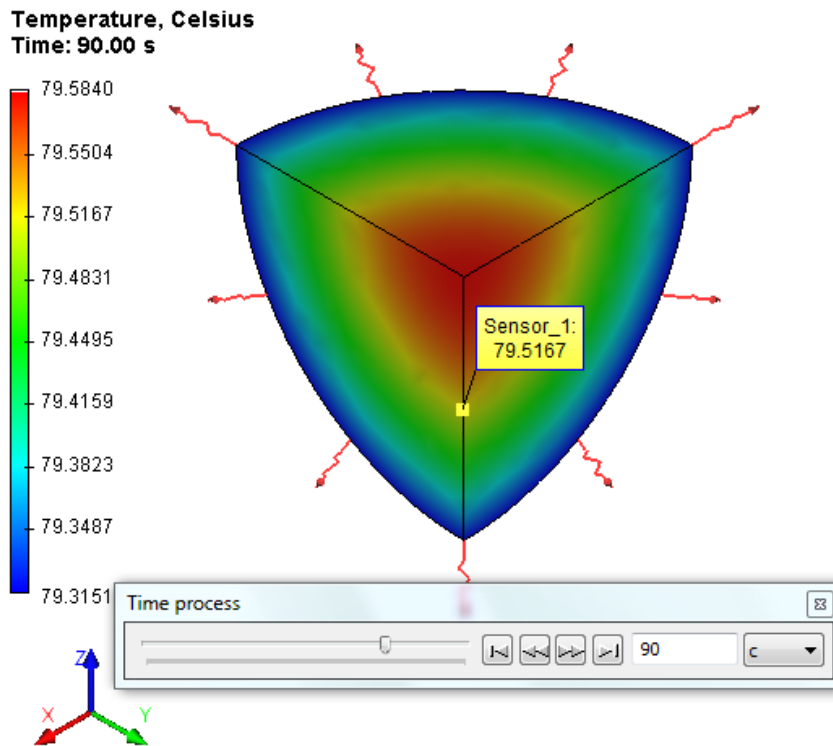
Temperature field in 90 seconds (linear elements)

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 study, 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 transfer on the surface

Let us consider the study of determination of the temperature at any point inside the sphere in equal increments of time $\Delta t1,2,3 = 30, 40, 60$ sec if we prescribe the initial temperature $Tstart=60\ ^0C$ inside the sphere, and on the boundary of the sphere we prescribe the heat transfer by convection with the heat

transfer coefficient H=300 W/(m$^{2 \cdot 0}$C) [19] (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.[20]
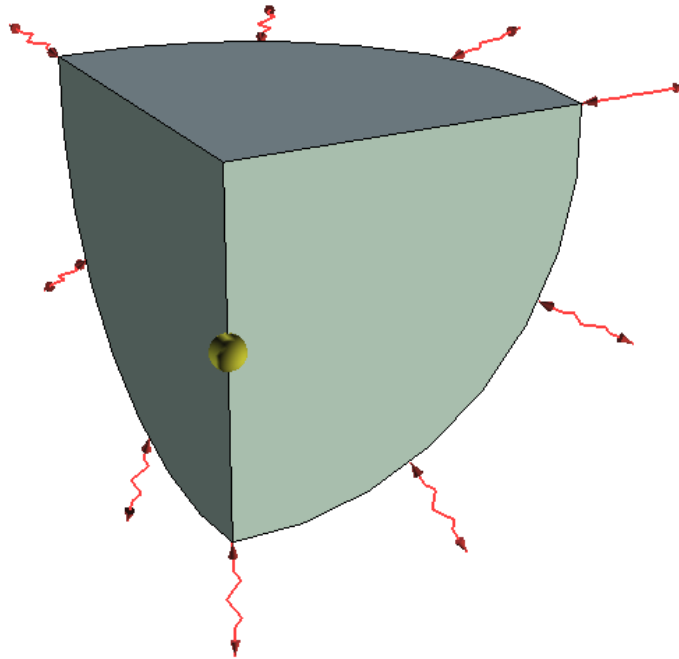
Numerical model with boundary conditions

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:

---

[19] The heat transfer coefficient for water in rest condition was taken from the website http://www.helpw.ru/Teplotdach.php

[20] On the boundaries where no boundary conditions are specified, the zero heat flux condition is satisfied automatically

$$u(r,0) = r \cdot f(r) = T_{start} \cdot r, \quad t = 0$$

$$\frac{\partial u}{\partial r}(a,t) + \left(h - \frac{1}{a}\right)u = 0, \quad r = a$$

where $f(r)$ – initial distribution of the temperature, and the coefficient $h = H/K$.

Solving this study by the method of separation of variables, we obtain the expression for v:[21]

$$v(r,t) = \frac{2hT_{start}}{r}\sum_{n=1}^{\infty} A_n \cdot e^{-\chi \cdot \alpha_n^2 \cdot t} \cdot \sin(r \cdot \alpha_n)$$

where $\chi = K/(c \cdot \rho)$ is the coefficient of temperature conductivity. The expansion coefficients An 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.

---

[21] H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, The «Science» publishing house, Moscow 1964, p. 234

Table №1 for calculation with linear element

Calculation time t, c	30	40	60
Analytical solution, ^{0}C	59.2381	58.4375	56.4123
Numerical solution, ^{0}C	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, ^{0}C	59.2381	58.4375	56.4123
Numerical solution, ^{0}C	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.

Temperature, Celsius
Time: 30.00 s
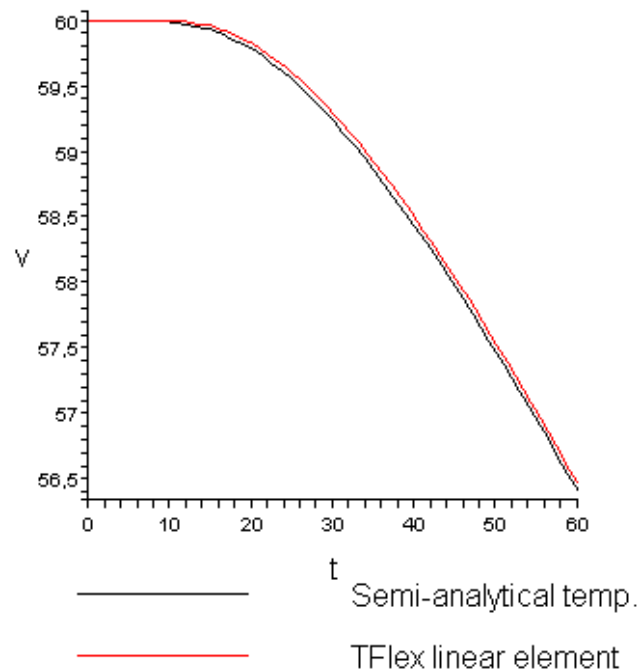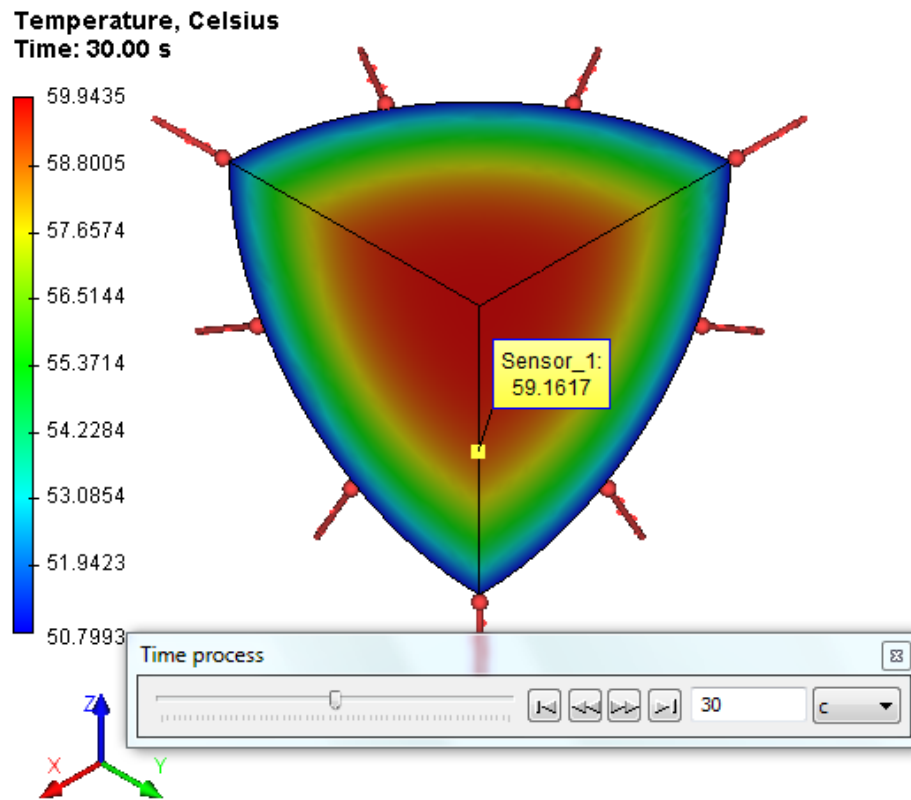
Plot of the temperature field in 30 seconds (linear element)

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 [22]

Let us consider a study of cooling of a cylindrical body with isotropic properties that has an initial temperature of $t_0=60$ ^{0}C 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

---

[22] This example is taken from the book by H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, Moscow. The "Science" publishing house. P. 225

coordinate system is located in the center of the cylinder): r_1=25 mm, h_1=25 mm; r_2=25 mm, h_2= -25 mm; r_3=30 mm, h_3=0 mm at the following moments of time $t_{1,2,3}$=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/cm^3, specific heat c =460 J/(kg*^{0}C), thermal conductivity K = 40 W/(m * ^{0}C).
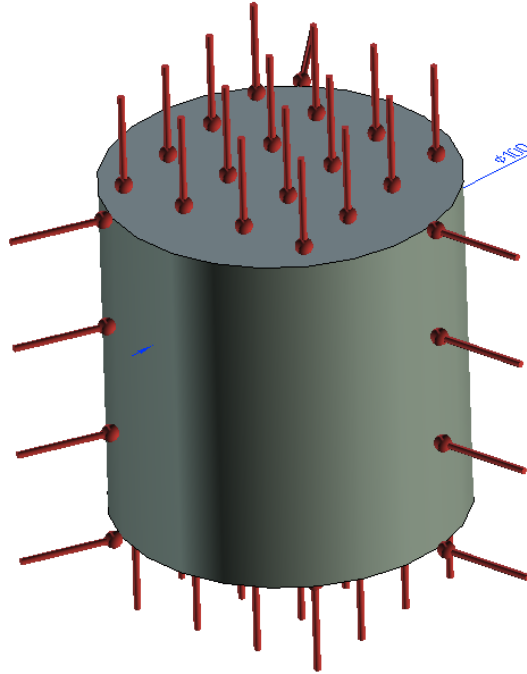
Numerical model with boundary conditions

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.

$$u(r,\varphi,z,t) = \sum_{\mu}\sum_{m=1}^{\infty}\sum_{n=0}^{\infty} e^{-\chi\left(\mu^2 + \frac{m^2\pi^2}{4l^2}\right)t} J_n(\mu \cdot r) \cdot \sin\frac{m\pi(z+l)}{2l} \times$$

$$\times \left(A_{\mu,m,n}\cos(n\varphi) + B_{\mu,m,n}\sin(n\varphi)\right)$$

where Jn (r)– Bessel function[23], parameter μ is a root of the equation $J_n(a\,\mu) = 0$

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 $f = t_0$ 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:

$$u(r,\varphi,z,t) = \sum_{\mu}\sum_{m=1}^{\infty} A_{\mu,m} e^{-\chi\left(\mu^2 + \frac{m^2\pi^2}{4l^2}\right)t} J_0(\mu \cdot r) \cdot \sin\frac{m\pi(z+l)}{2l}$$

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.

---

[23] H. Carslaw, J. Jaeger «Conduction of Heat in Solids», in Russian, Moscow, The "Science" publishing house, p. 478

Table 1

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 ^{0}C 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

Temperature graphs v(t), t - for time r=30, h=0

TFlex quadratic element

semi-analytical
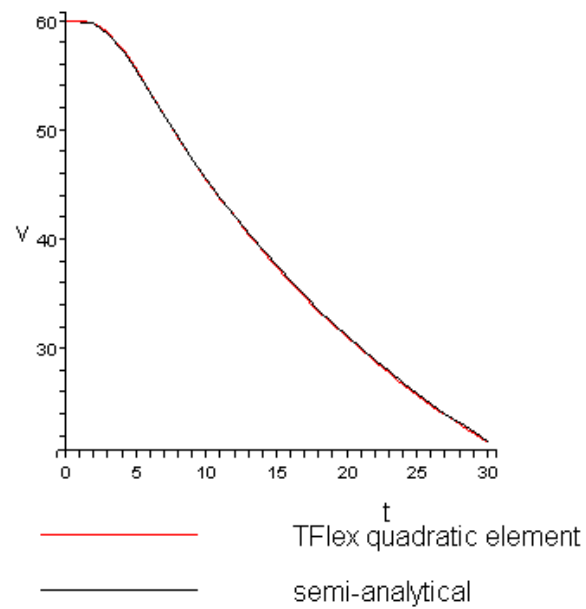
Calculation of temperature v in 0C, time t in sec, r and h – third control point (in mm)

Temperature, Kelvin
Time: 20.00 s

324.1

317.8

311.4

305

298.6

292.3

285.9

279.5

273.1

Sensor_1:
302.6

Sensor_5:
304.2

Sensor_2:
302.6
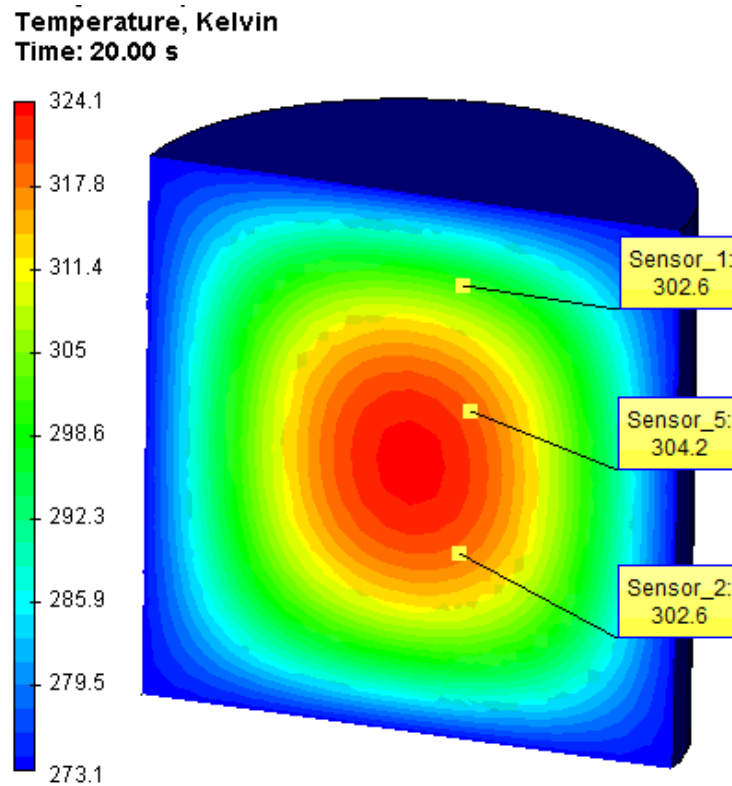
View of the temperature field in 30 seconds

Conclusions:

For the given study 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 studies 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 study of cooling of a graphite plate with orthotropic properties. The initial temperature of the plate is t_0=60 ^{0}C. 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 t_{1-4} = 2, 5, 10, 60 sec. Calculation of the temperature field will be conducted for control points 1-5 with coordinates (X_i, Y_i):

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 ρ=2.5 g/cm^3, specific heat c=840 J/(kg•^{0}C), coefficients of thermal conductivity: K_1=0.139 W/(mm • ^{0}C) (along the OX-axis); K_2=0.278 W/(mm • ^{0}C) (along the OY-axis – base direction). The plate has a rectangular shape 100 × 100 mm.
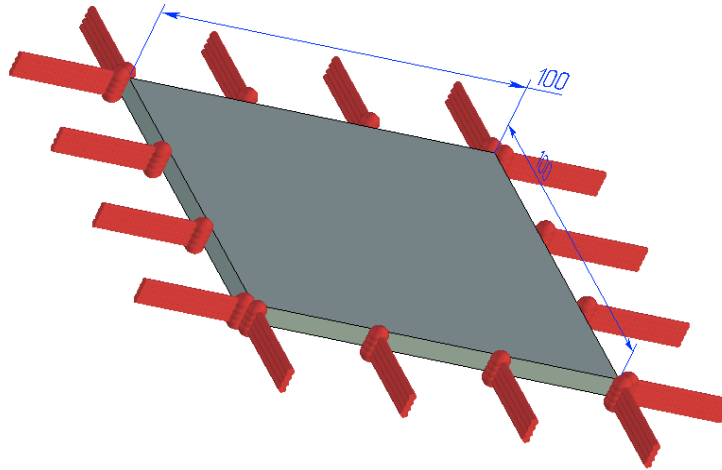
Numerical model with boundary conditions

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 study has the form[24]:

$$u(x, y, t) = \sum_{n=1}^{\infty} \sum_{m=1}^{\infty} A_{m,n} \cdot e^{\lambda(m,n)t/(\rho \cdot c)} \cdot \sin\left(\frac{\pi \cdot my}{l_y}\right) \sin\left(\frac{\pi \cdot nx}{l_x}\right)$$

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[25]:

---

[24] Solution was obtained by the method of separation of variables. Description of this method is contained in the book by Tikhonov A. N., Samarsky A. A. Equations of mathematics physics, p. 200.

[25] Analytical solution and the plot are built in the Maple 9.5 system
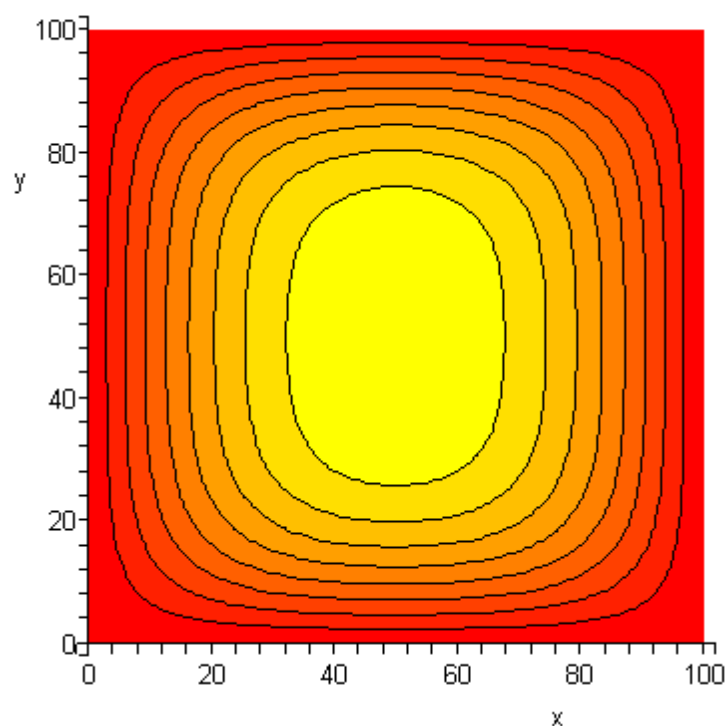
View of the temperature field at the moment of time t1=2 sec (analytical solution)

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

Total calculation time (sec)	Time step (sec)	Number of time layers
60	0.5	120

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 ^{0}C are given below.

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, ^{0}c	13.1503	13.1503	13.1503	13.1503	35.5560
Numerical solution, ^{0}c	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, ^{0}c	4.74820	4.74820	4.74824	4.74824	13.6788
Numerical solution, ^{0}c	4.61898	4.58004	4.58856	4.613	13.3372
Relative error, %	2.72145	3.54155	3.36293	2.848213	2.497295

Table 5

For time t=2 c

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	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, ^{0}c	13.1503	13.1503	13.1503	13.1503	35.5560
Numerical solution, ^{0}c	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, ^{0}c	4.74820	4.74820	4.74824	4.74824	13.6788
Numerical solution, ^{0}c	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 ^{0}C.

Temperature graphs v(t), t - for time r=50
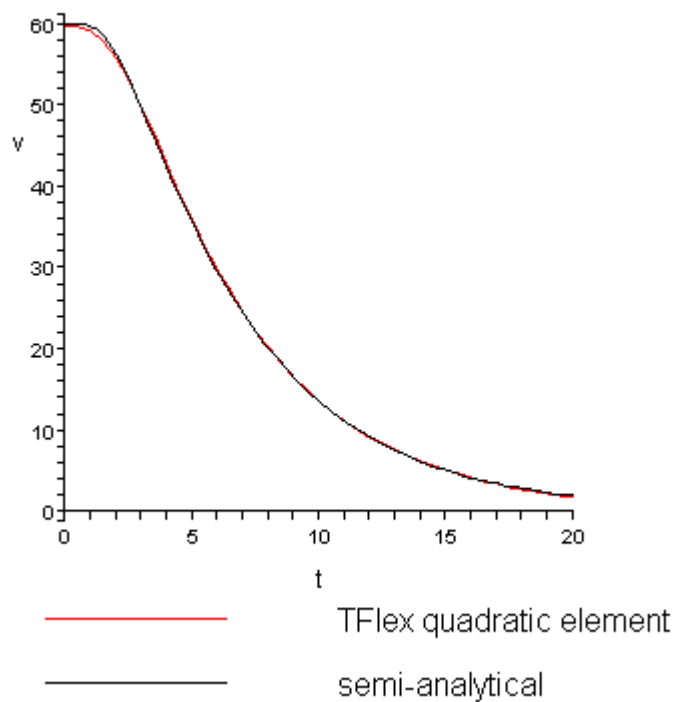
Dependence of temperature V(t) on time t at a point xy=(50,50) mm

Temperature, Celsius
Time: 2.00 s
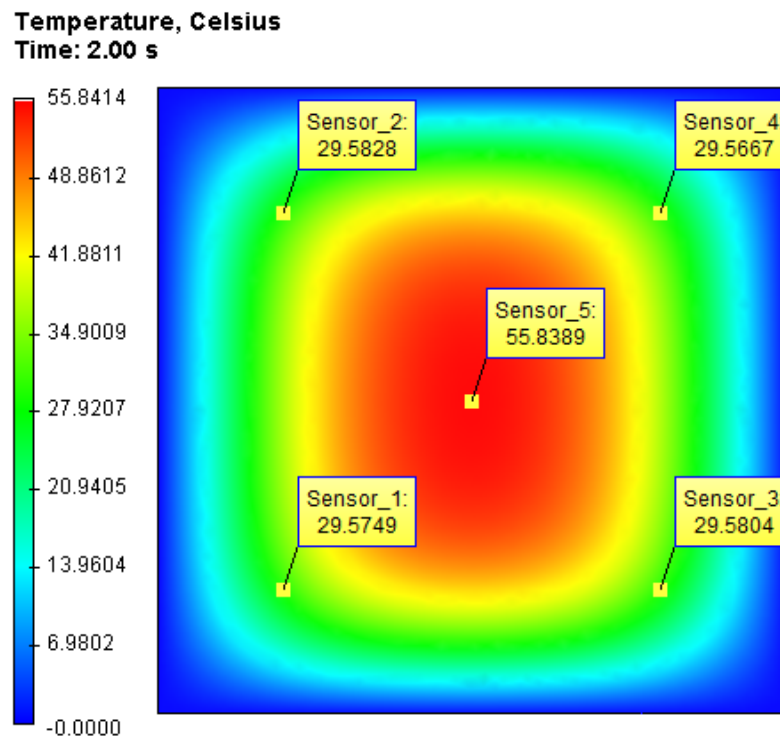
View of the temperature field at the moment of time t1=2 sec (from the results of finite element analysis)

Conclusions:

For the given study 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 studies 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.

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.
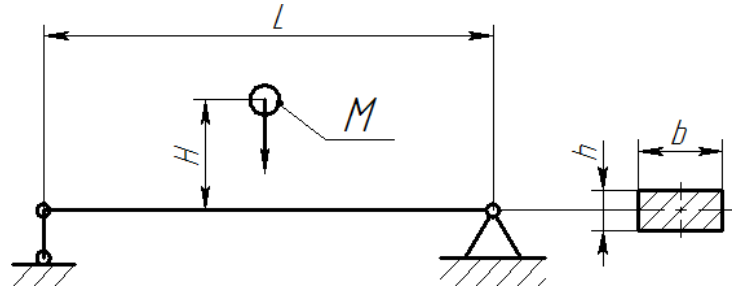
It is necessary to define the beam maximum bending.

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.
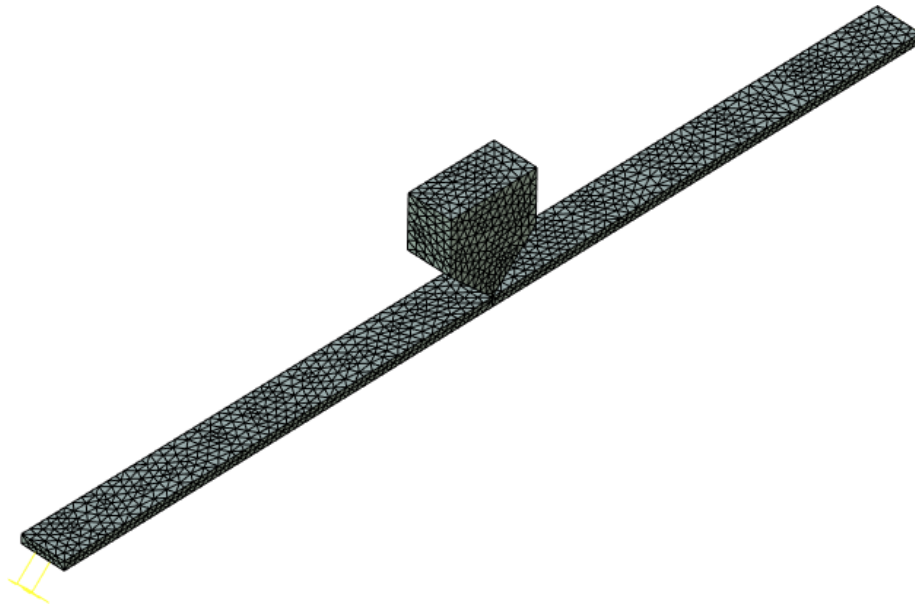
Calculated model with loads and restraints

Finite-element model of the beam with loads and restraints

An analytical solution is the following:

$$w = w_{st} + \sqrt{w_{st}^2 + \frac{V_0^2 w_{st}}{g}} = 27{,}997 mm$$

$$w_{st} = \frac{Mgl^3}{48EJ} = 11{,}877 mm$$

,

where $J = \dfrac{b \cdot h^3}{12}$ - 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:

Graph: Displacement OZ : 51

№	X, Step	Y, m
1	1	0
2	2	-0.00156525
3	3	-0.00326064
4	4	-0.00514681
5	5	-0.00715218
6	6	-0.00924873
7	7	-0.0114117
8	8	-0.0135636
9	9	-0.0156912
10	10	-0.0177465
11	11	-0.0196669
12	12	-0.0214492
13	13	-0.0230326
14	14	-0.024384
15	15	-0.0255065
16	16	-0.0263352
17	17	-0.0268857
18	18	-0.0271415
19	19	-0.0270777
20	20	-0.0267235
21	21	-0.0260724

Polyline (preview only), Nodes: 51   X = 51.1078, Y = -0.0170082
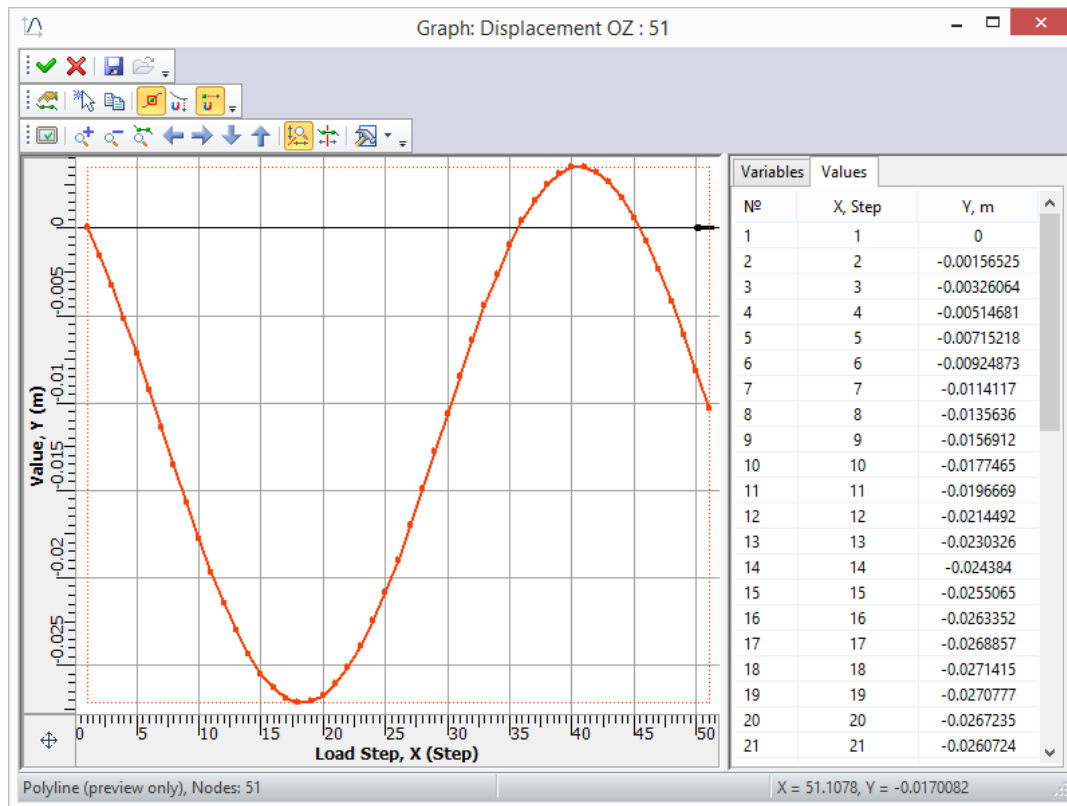
The following results are obtained at the current time step:
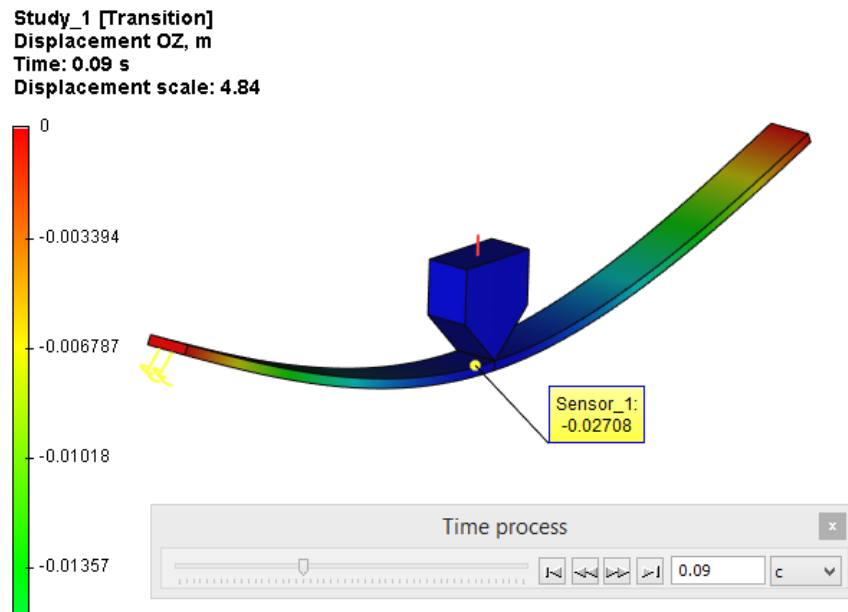
Study_1 [Transition]
Displacement OZ, m
Time: 0.09 s
Displacement scale: 4.84

Sensor_1:
-0.02708

Time process

0.09

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 w , mm	**Error** δ =100% $(w^* - w)$ / 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 $P=P_0 \cdot \sin(\omega t)$.

$t \geq 0$

$E,\ S,\ \rho = \text{const}$

$P=P_0 \cdot \sin(\omega t)$

0

l

x
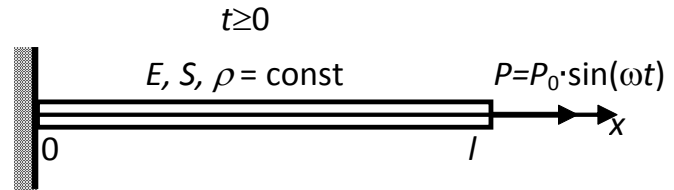
It is necessary to define the maximum displacement of the beam end.

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 ν=0,29, density ρ=7900 kg/m ^{3}. Amplitude and frequency of external force are correspondingly equal P_0=10 kN and $V_0 = \dfrac{\omega}{2\pi} = 2\text{kHz}$.

Displacements of the beam are determined by the formula:

$$u(x,t) = U(x,t) + \sum_{n=0}^{\infty} b_n \sin \frac{(2n+1)\pi x}{2l} \sin \frac{(2n+1)\pi a t}{2l}.$$

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*=0.75*l*, *x*=0.5

l and *x*=0.25*l* .
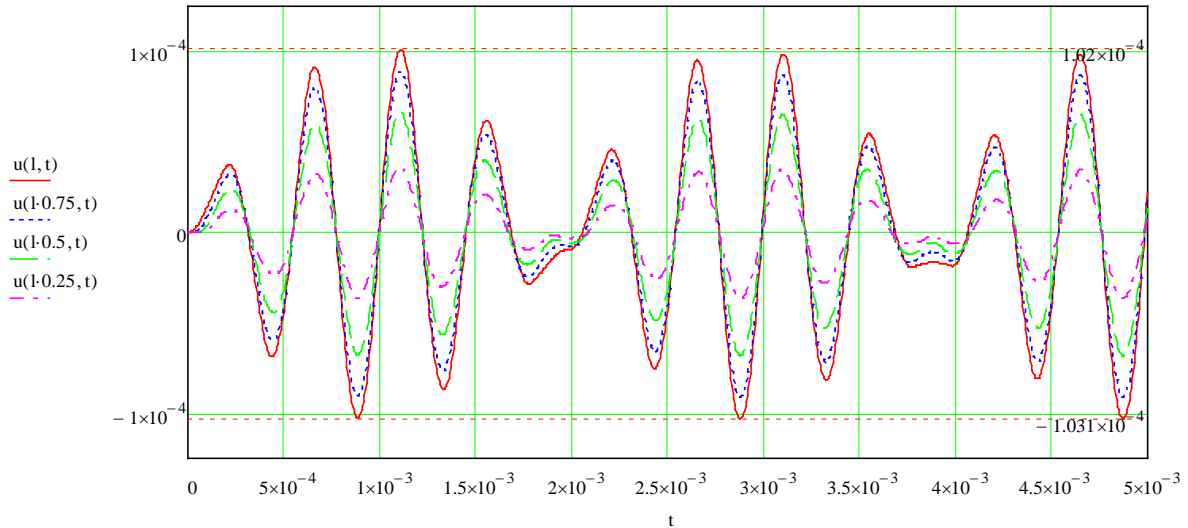
Displacements of beam sections

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:
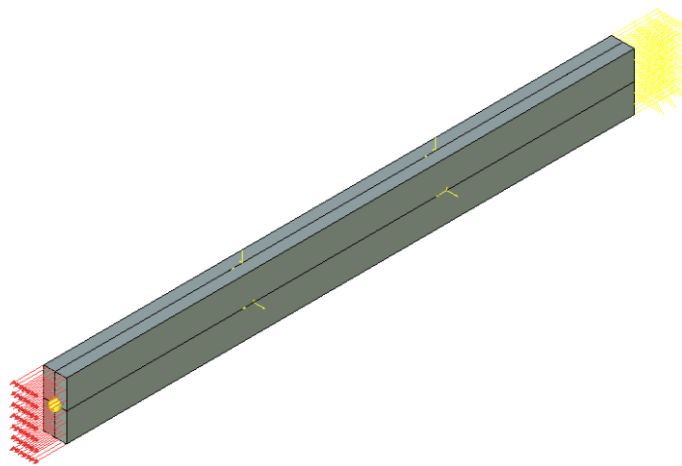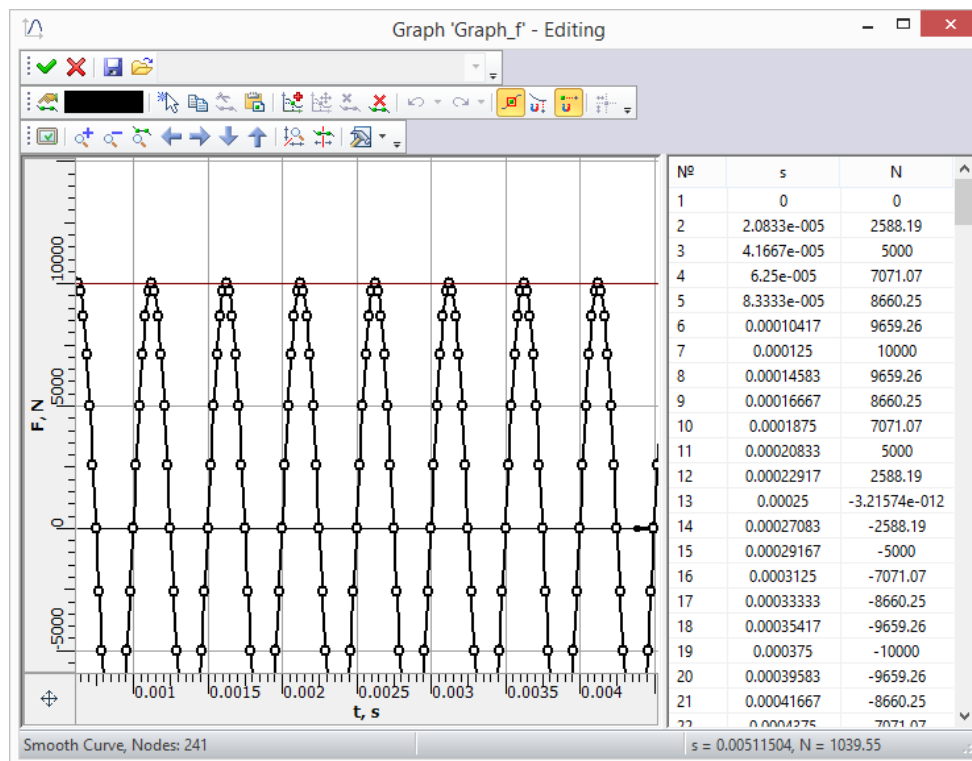

Load graph

For both of the studies, we set the finite modeling time 0,005 s, the time step of integration $5 \cdot 10^{-6}$ 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** δ =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** δ =100% $(w^* - w) / w$
$1{,}0159 \cdot 10^{-4}$	$1{,}02 \cdot 10^{-4}$	1.48
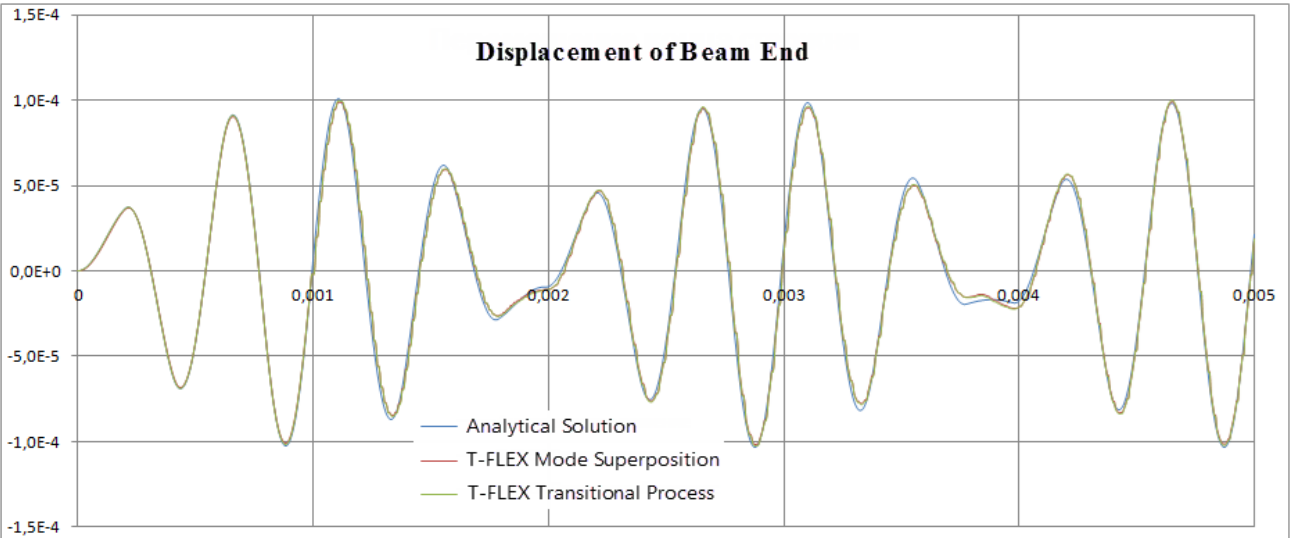
Analytical solution and calculation results
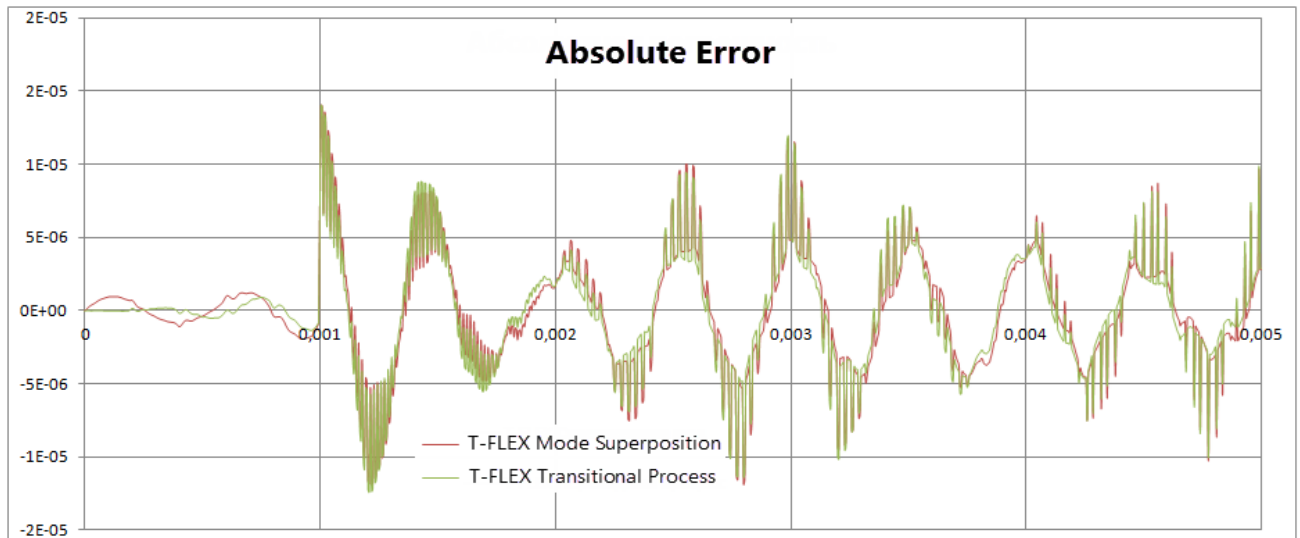
Difference between analytical solution and T-FLEX solution

Conclusion: The maximum displacement of the free end of the beam, found using T-FLEX Analysis is: for the Transitional process $1.0259 \cdot 10^{-4}$ m (relative error 0.51%), for the Mode superposition $1.0159 \cdot 10^{-4}$ m (relative error 1,48%).

Hollow shaft loaded with sinusoidal torque on the free end.

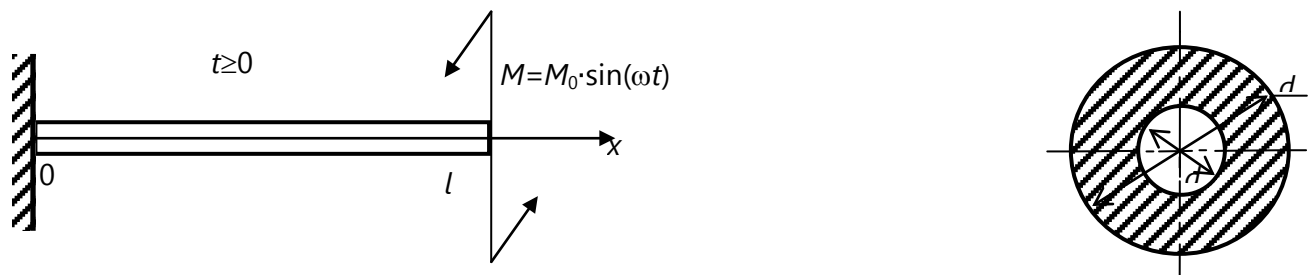Let us review the study of torsional oscillation of steel shaft. Its left end is fixed. A torsional couple of forces is applied to the right end starting from time $t = 0$. The forces torque changes under the law $M=M_0 \cdot \sin(\omega t)$. Shaft length $l = 1$ m. The cross-section is annular with inner diameter $d_1 = 50$ mm and outer diameter $d_2 = 80$ mm. Modulus of elasticity $E=200$ GPa, Poisson ratio $\nu=0{,}29$, density $\rho=7900$ kg/m^3. Amplitude and frequency of external torque are correspondingly equal $M_0=1$kN $\cdot$m and

$$V_0 = \frac{\omega}{2\pi} = 500\text{Hz}$$

. 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 $P=P_0 \cdot \sin(\omega t)$ is applied to the torsion angle φ 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:

$$\Phi(x,t) = \frac{aM_0}{GJ_p\omega} \frac{\sin\frac{\omega}{a}x}{\cos\frac{\omega}{a}l} \sin\omega t,$$

$$b_n = -\frac{4}{(2n+1)\pi a} \int_0^l \Phi_t(z,0) \sin\frac{(2n+1)\pi z}{2l} dz,$$

$$a = \sqrt{\frac{G}{\rho}}, \quad G = \frac{E}{2(1+v)}, \quad J_p = \frac{\pi d_2^4}{32}\left(1 - \left(\frac{d_1}{d_2}\right)^4\right).$$

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.
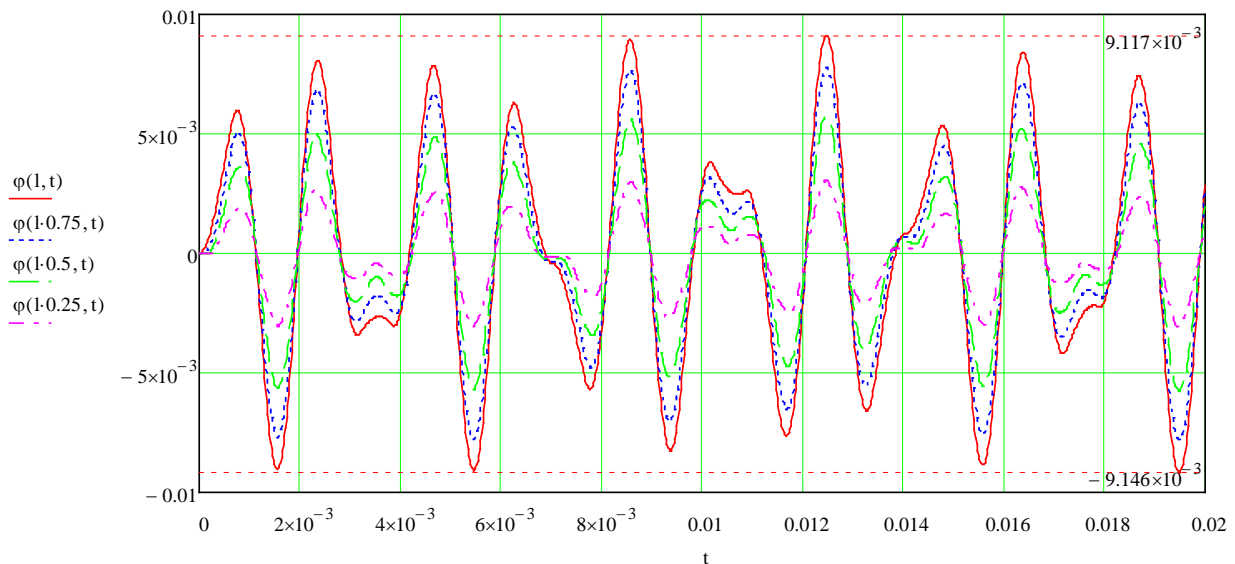
Graphs of angles of rotation of four sections according to the time

The maximal angle of rotation is on the right end of the shaft and is $9.146 \cdot 10^{-3}$ 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.
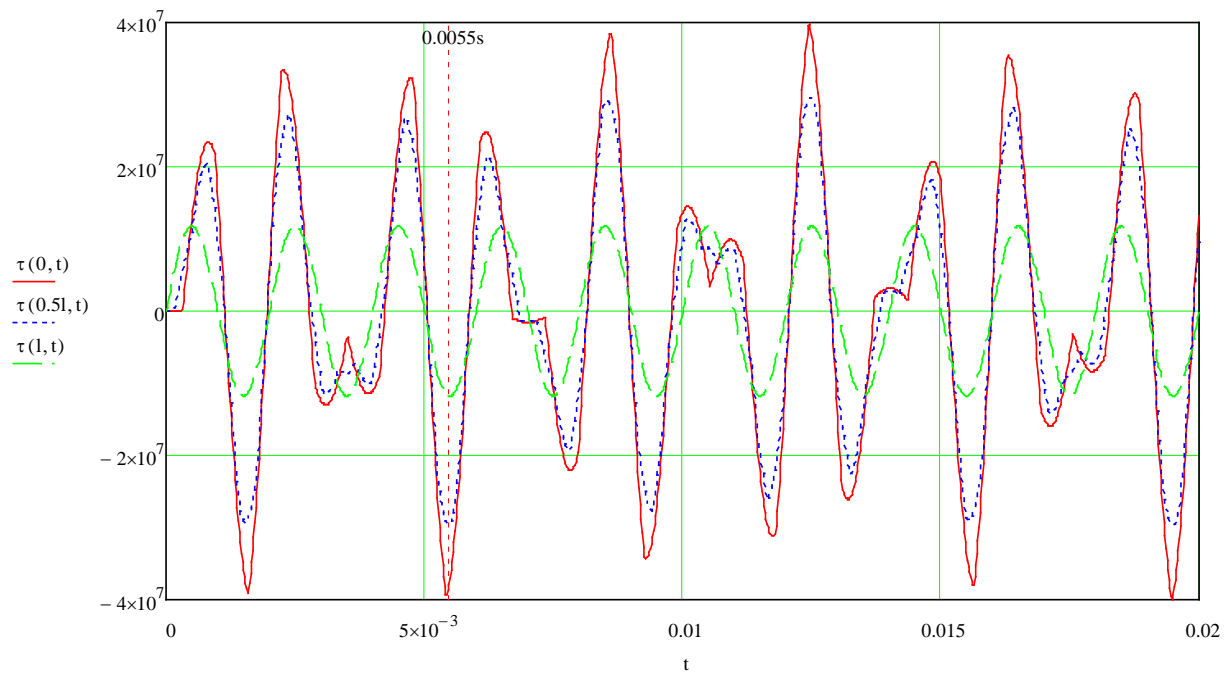
Graphs of maximal shearing stresses in the section according to the time.

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
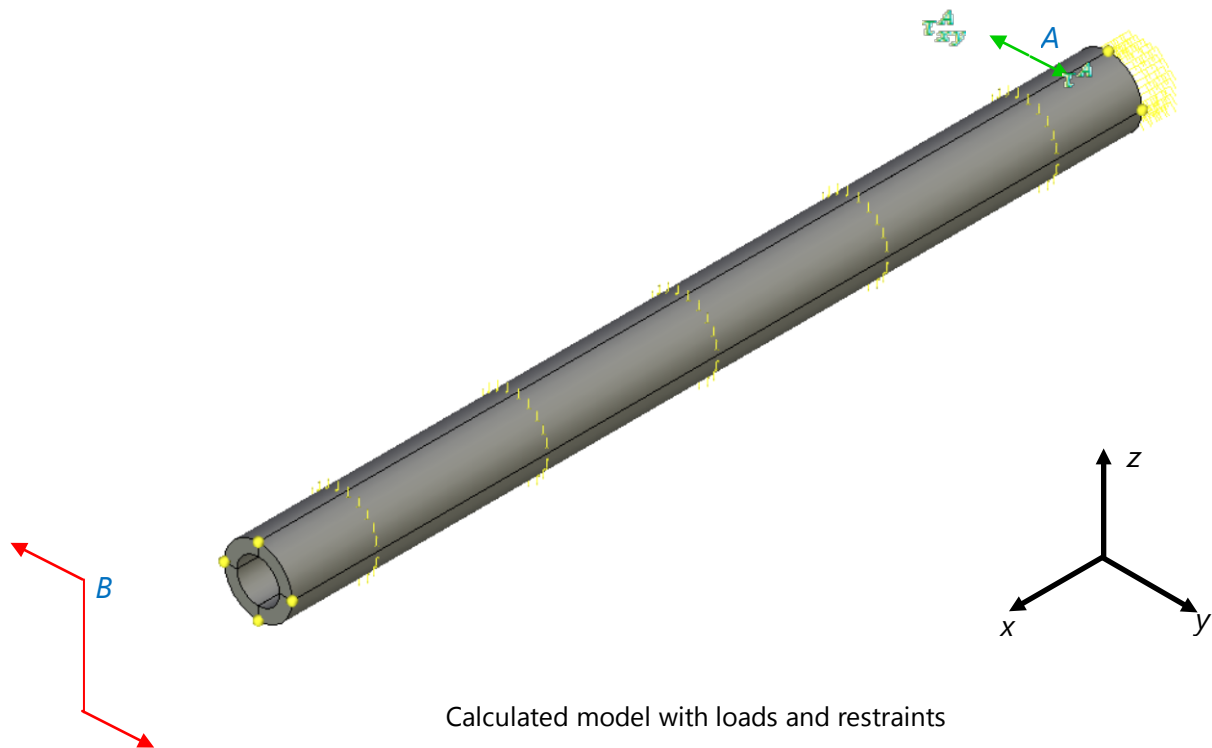
We create a full restraint on the left butt end of the shaft. The right end will stay free. To exclude the shaft bending we set kinematic restrictions on the radial displacements (radial restrictions are equal to zero) for the outer and inner cylindrical surfaces- for that purpose we apply partial restraints 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:
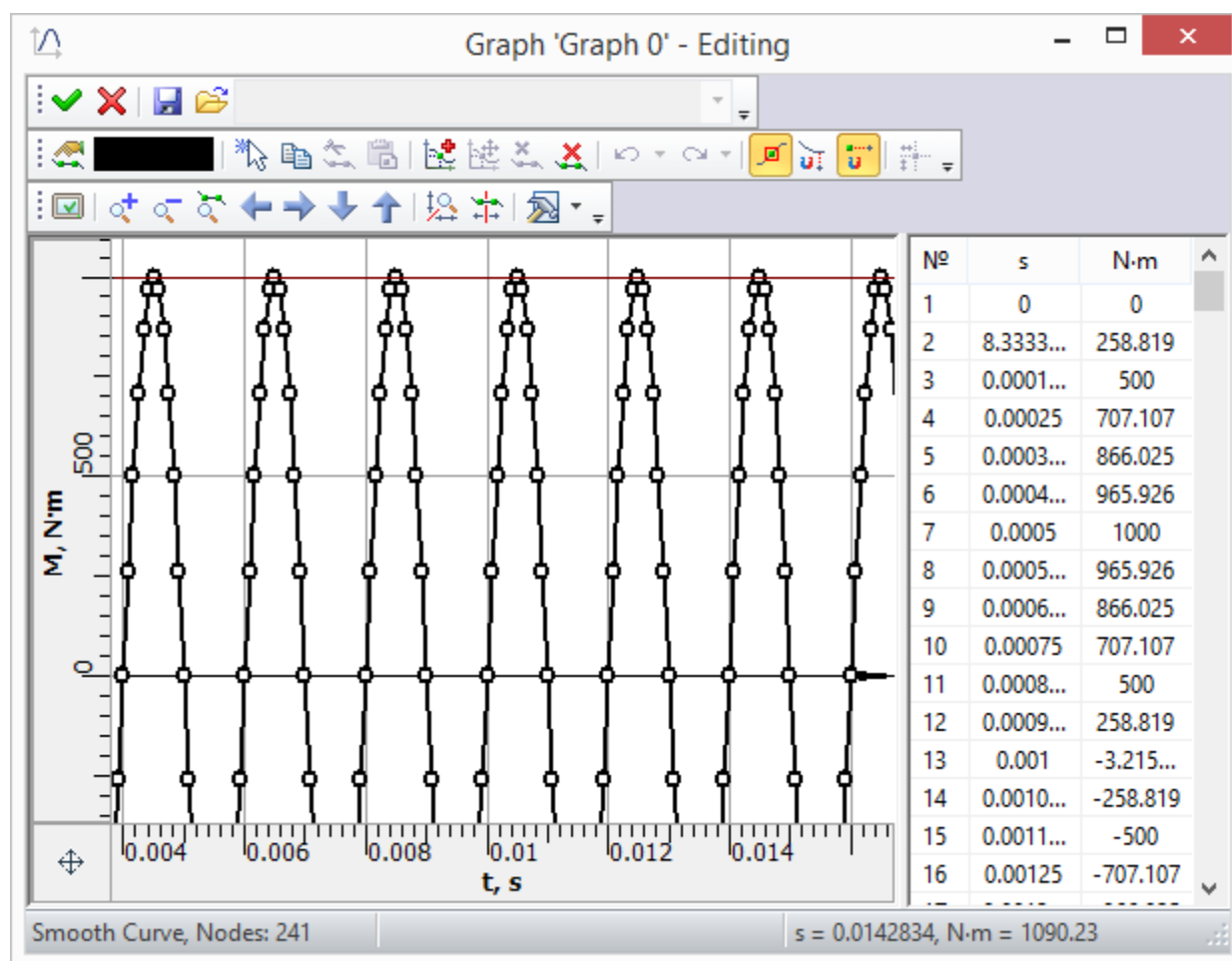
Load graph

For both of the studies, we set the finite modeling time 0,02 s, the time step of integration $1 \cdot 10^{-5}$ 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_{xy}^A (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 φ*, *rad*	Analytical solution φ , *rad*	**Error** δ =100% (φ* - φ) / φ
$9{,}0662 \cdot 10^{-3}$	$9{,}146 \cdot 10^{-3}$	0.88

Table 4. Mode superposition, maximal torsion angle

Numerical solution φ*, *rad*	Analytical solution φ , *rad*	**Error** δ =100% (φ* - φ) / φ
$8{,}9169 \cdot 10^{-3}$	$9{,}146 \cdot 10^{-3}$	1.48

Table 5. Transitional processes, maximal shearing stress

Numerical solution τ*, *MPa*	Analytical solution τ, *MPa*	**Error** δ =100% (τ* - τ) / τ
38.434	39.9	3.68

Table 6. Mode superposition, maximal shearing stress

Numerical solution τ*, *MPa*	Analytical solution τ, *MPa*	**Error** δ =100% (τ* - τ) / τ
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 $9{,}0662 \cdot 10^{-3}$ 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.
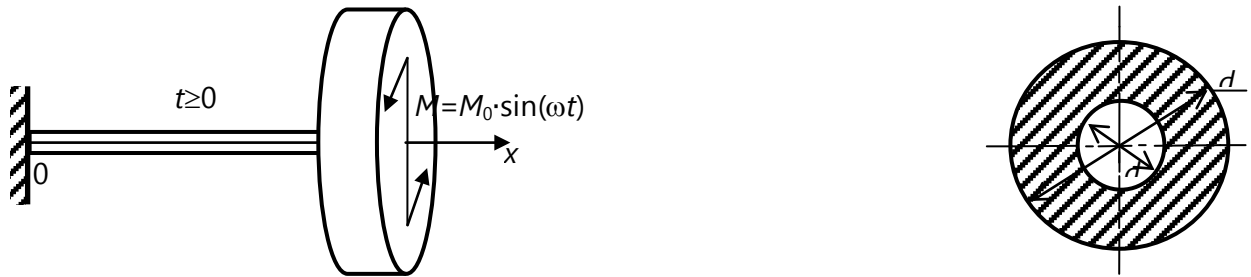
Let us review the study of torsional oscillation of steel shaft on the end of which the wheel is attached. The left end of the shaft is fixed. To the right end, the wheel is attached. The wheel diameter is $D = 200$ mm and its thickness $h = 30$ mm. Couple of torsional forces which changes under the law $M(t)=M_0 \cdot \sin(\omega t)$ 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 $\nu=0{,}29$, density $\rho=7900$ kg/m^3. Amplitude and frequency of external torque are correspondingly equal $M_0=1$kN·m 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:

$$X_n(x) = \sin\left(\frac{\mu_n b}{l}\right), \qquad n = 1,2,3,\dots$$

Here $b = \sqrt{\frac{G}{\rho}}$, $G = \frac{E}{2(1+\nu)}$ - shear modulus 6 μ_n - positive roots of the equation $\text{tg}(\mu) = \frac{J_p \rho l}{J} \cdot \frac{1}{\mu}$, $I_p = \frac{\pi d_2^4}{32}\left(1 - \left(\frac{d_1}{d_2}\right)^4\right)$ - 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.
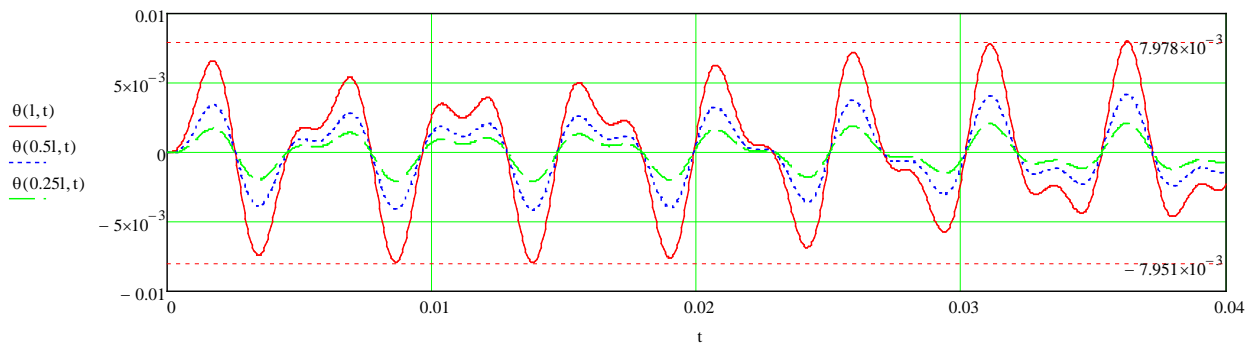
Graphs of angles of rotation of three sections according to the time

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.
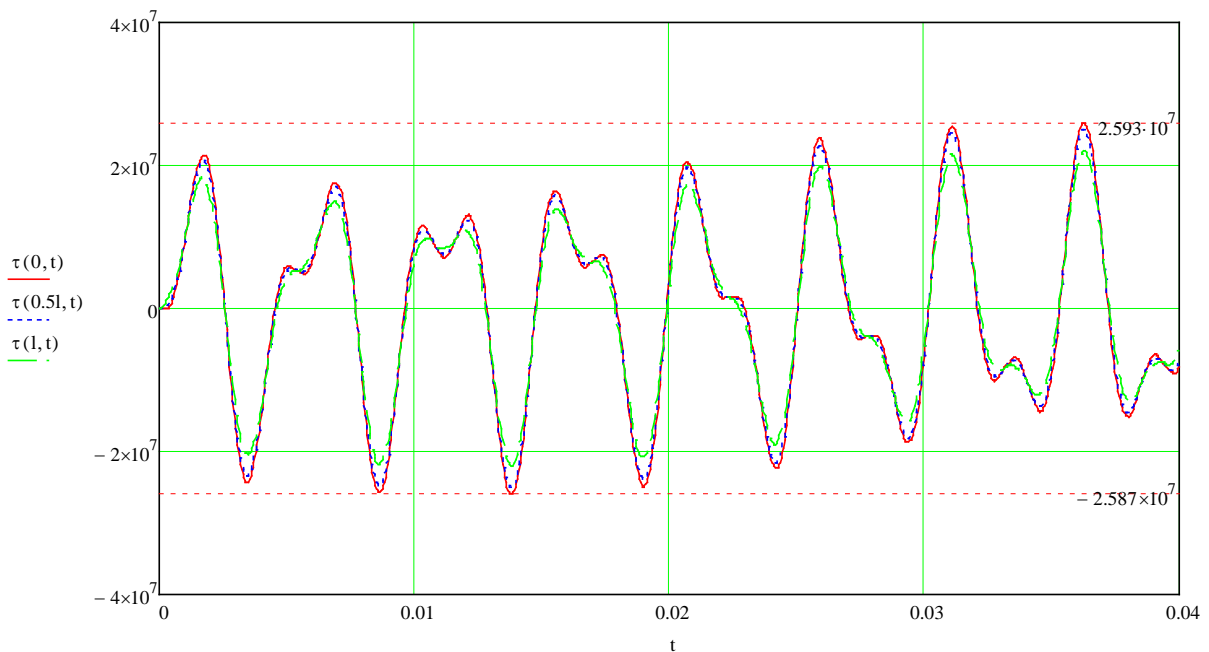
Graphs of maximal shearing stresses in the section according to the time.

Maximal shearing stress is in the closing (*x*=0) and is 25,9 MPa modulo.

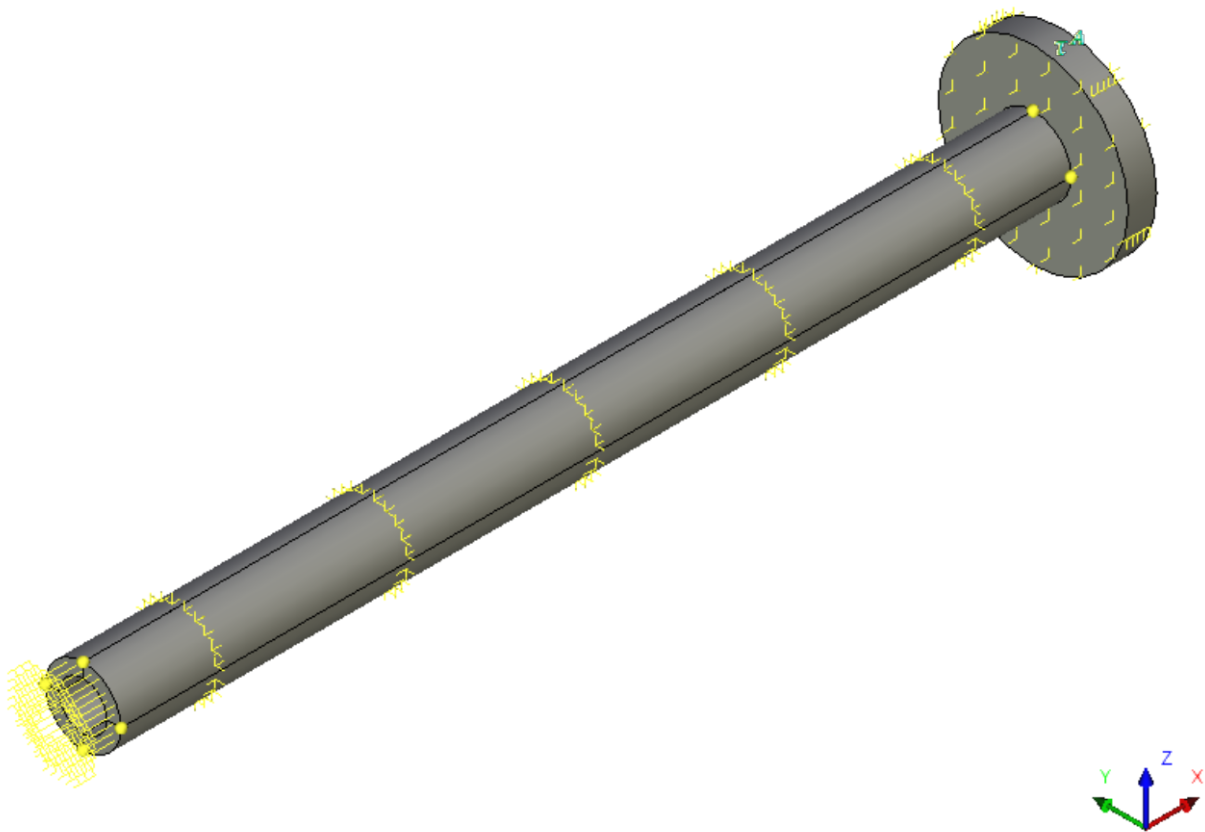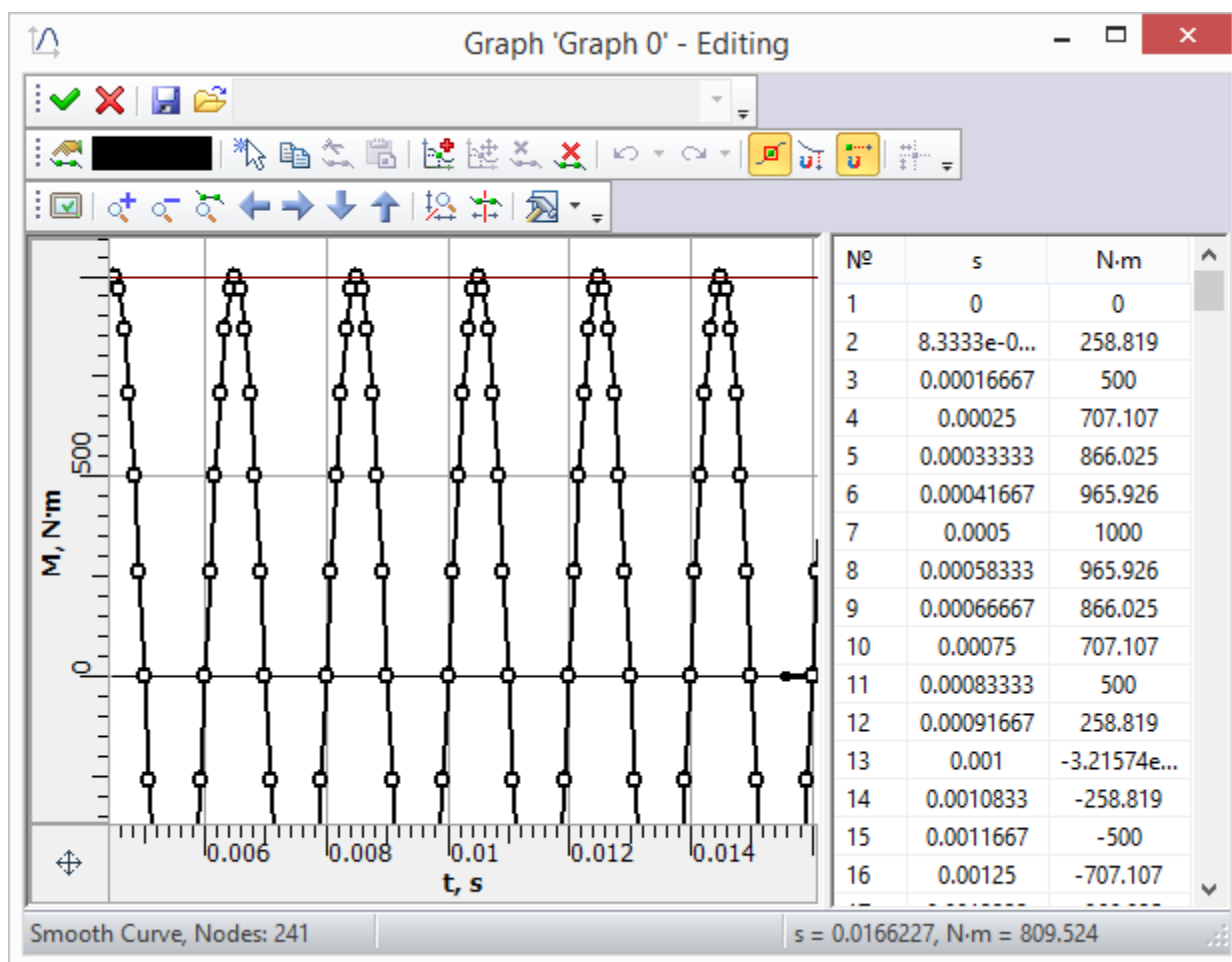Let us calculate T-FLEX Analysis study: We create two studies, one Transitional processes, another Mode superposition with the same loads and restraints.

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 0*x* 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:

№	s	N·m
1	0	0
2	8.3333e-0...	258.819
3	0.00016667	500
4	0.00025	707.107
5	0.00033333	866.025
6	0.00041667	965.926
7	0.0005	1000
8	0.00058333	965.926
9	0.00066667	866.025
10	0.00075	707.107
11	0.00083333	500
12	0.00091667	258.819
13	0.001	-3.21574e...
14	0.0010833	-258.819
15	0.0011667	-500
16	0.00125	-707.107

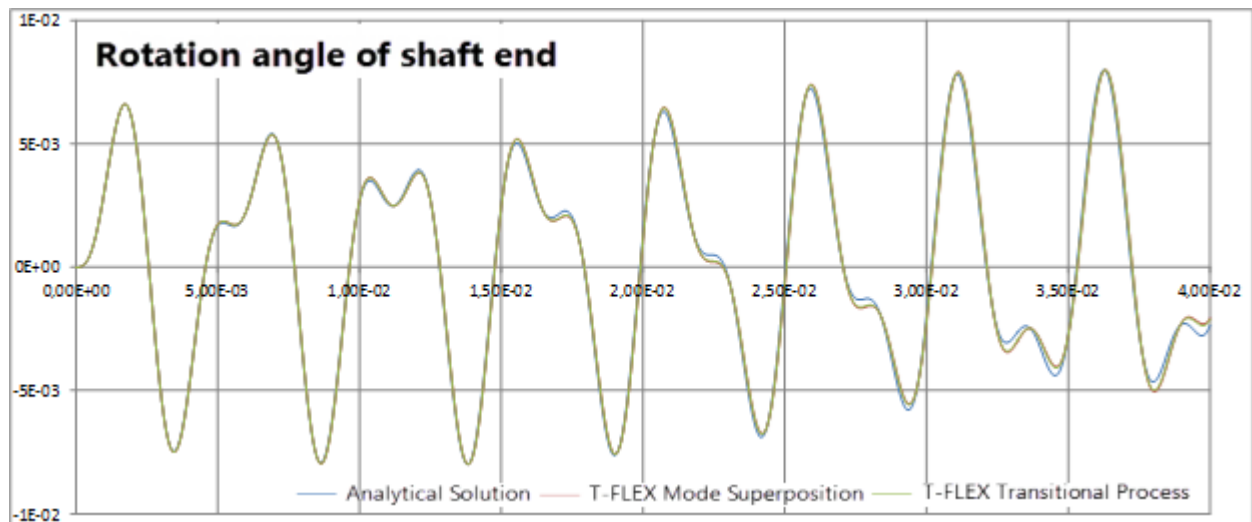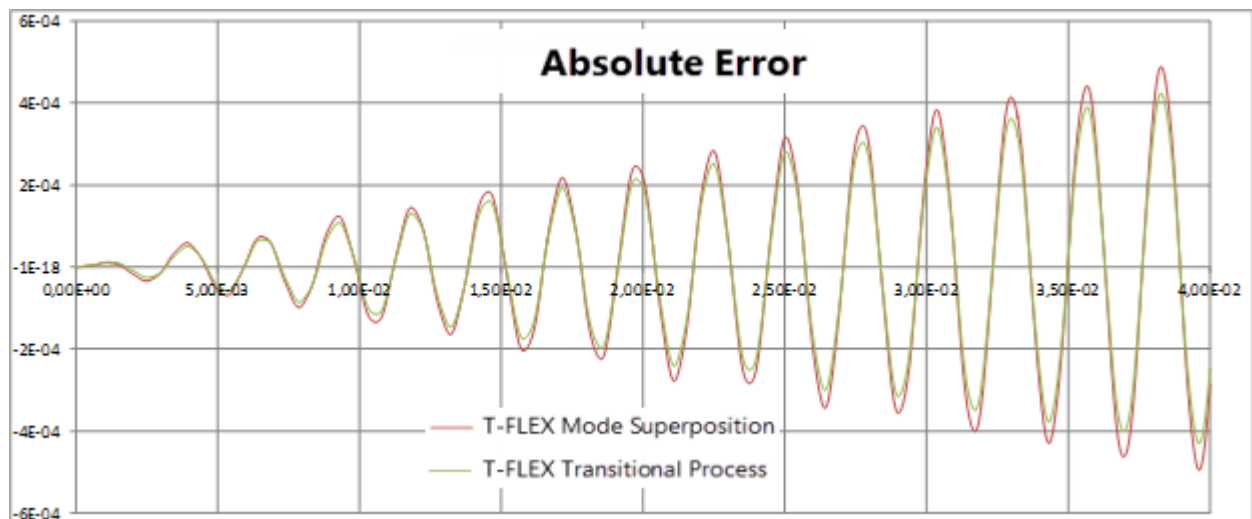Smooth Curve, Nodes: 241   s = 0.0166227, N·m = 809.524

Load graph

For both of the studies, we set the finite modelling time 0,04 s, the time step of integration $2,5 \cdot 10^{-5}$ 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).
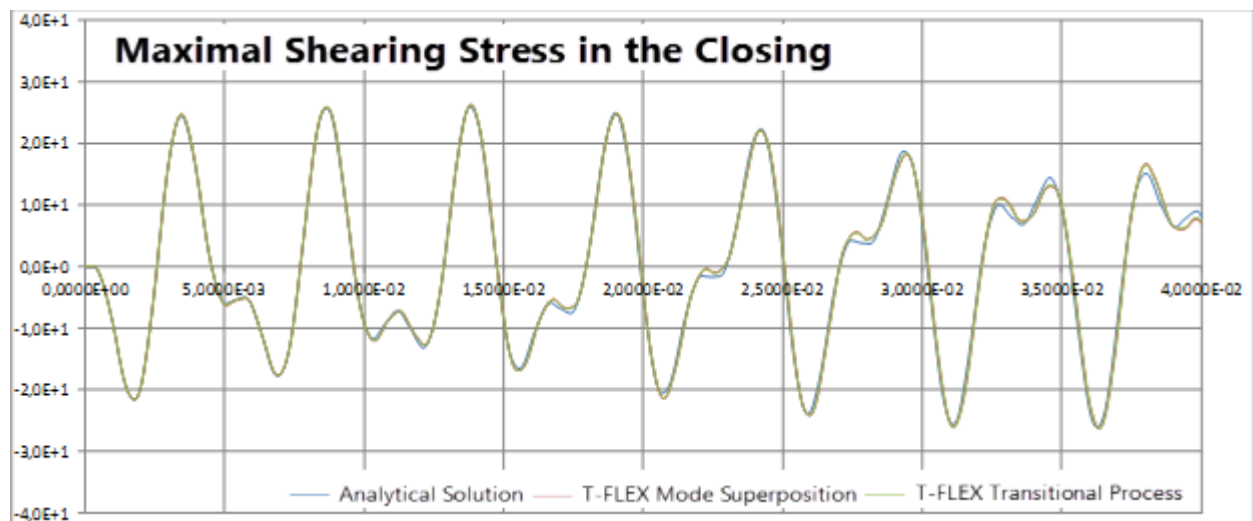
On the figure below, you may see the dependences graphs of free shaft end with the wheel angle of rotation from time for an analytical solution, a numerical solution using Transitional process and Mode superposition.

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 φ^*, *rad*	Analytical solution φ, *rad*	**Error** $\delta = 100\% (\varphi^* - \varphi) / \varphi$
$7{,}9763 \cdot 10^{-3}$	$7{,}978 \cdot 10^{-3}$	0.15

Table 4. Mode superposition, maximal torsion angle

Numerical solution φ^*, *rad*	Analytical solution φ, *rad*	**Error** $\delta = 100\% (\varphi^* - \varphi) / \varphi$
$7{,}9732 \cdot 10^{-3}$	$7{,}978 \cdot 10^{-3}$	0.08

Table 5. Transitional processes, maximal shearing stress

Numerical solution τ^*, *MPa*	Analytical solution τ, *MPa*	**Error** $\delta = 100\% (\tau^* - \tau) / \tau$
26.270	25.9	1.30

Table 6. Mode superposition, maximal shearing stress

Numerical solution τ^*, *MPa*	Analytical solution τ, *MPa*	**Error** $\delta = 100\% (\tau^* - \tau) / \tau$
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 $7{,}9763 \cdot 10^{-3}$ rad (relative error 0.15%), for the Mode superposition $7{,}9732 \cdot 10^{-3}$ 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.
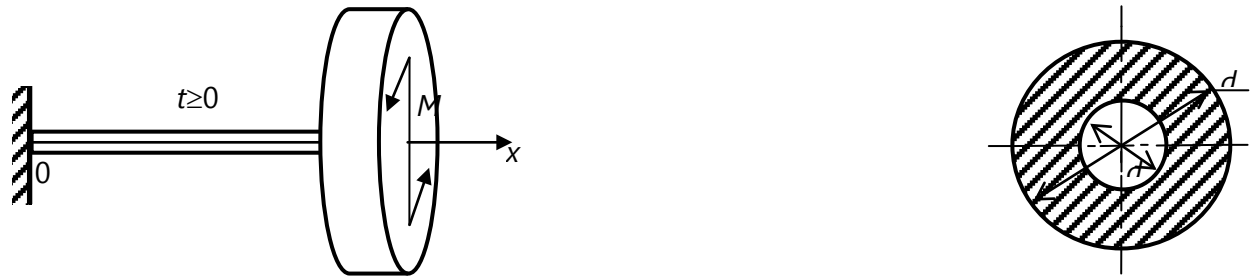
The wheel is attached to the right end. The wheel diameter is D = 200 mm and its thickness h = 30 mm. A couple of torsional forces is applied to the shaft from the point in time $t=0$. The torque of the torsional forces changes under the law:

$$M(t) = \begin{cases} M_0 \dfrac{t}{T}, & \text{if } t < T \\ M_0, & \text{if } t \geq T \end{cases}$$

,where M_0 = 1 kNm, T=0,5·10⁻³sec
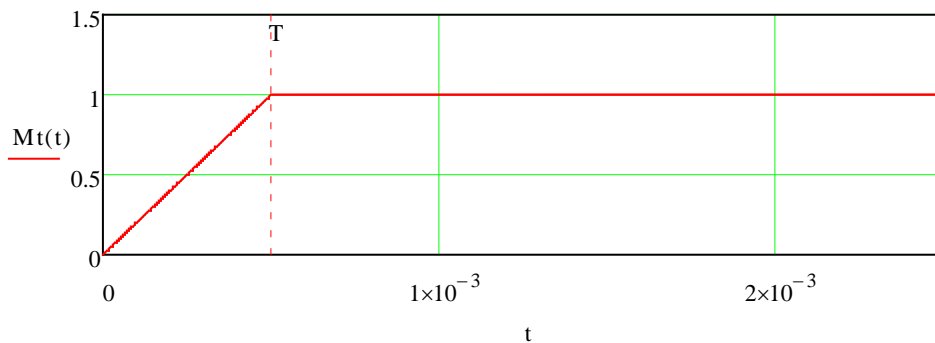
Graph of torque of the forces couple from the time

Shaft length l = 1 m. The cross-section is annular with inner diameter d_1=50 mm and outer diameter d_2= 80 mm. The wheel and the shaft have the same material. Modulus of elasticity E=200 GPa, Poisson ratio v=0,29, densityρ=7900 kg/m³. Amplitude and frequency of external torque are correspondingly equal M_0=1kN·m 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:

$$X_n(x) = \sin\left(\frac{\mu_n b}{l}\right), \qquad n = 1,2,3,\ldots$$

Here $b = \sqrt{\frac{G}{\rho}}$, $G = \frac{E}{2(1+\nu)}$ - shear modulus 6 μ_n - positive roots of the equation $tg(\mu) = \frac{J_p \rho l}{J} \cdot \frac{1}{\mu}$,

$I_p = \frac{\pi d_2^4}{32}\left(1 - \left(\frac{d_1}{d_2}\right)^4\right)$ - 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:

$$\varphi_n(t) = \frac{X_n(l)\mu_n}{A_n \rho J_p l b}\int_0^t M(\tau)\sin\left(\frac{\mu_n b}{l}(t-\tau)\right)d\tau.$$

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 $7.326 \cdot 10^{-3}$ 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*.
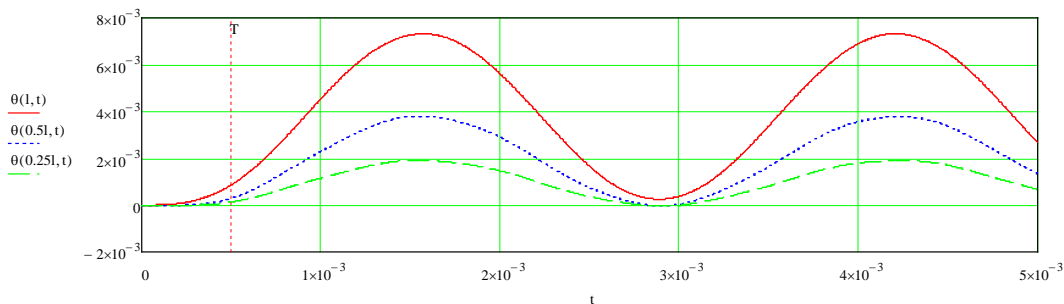
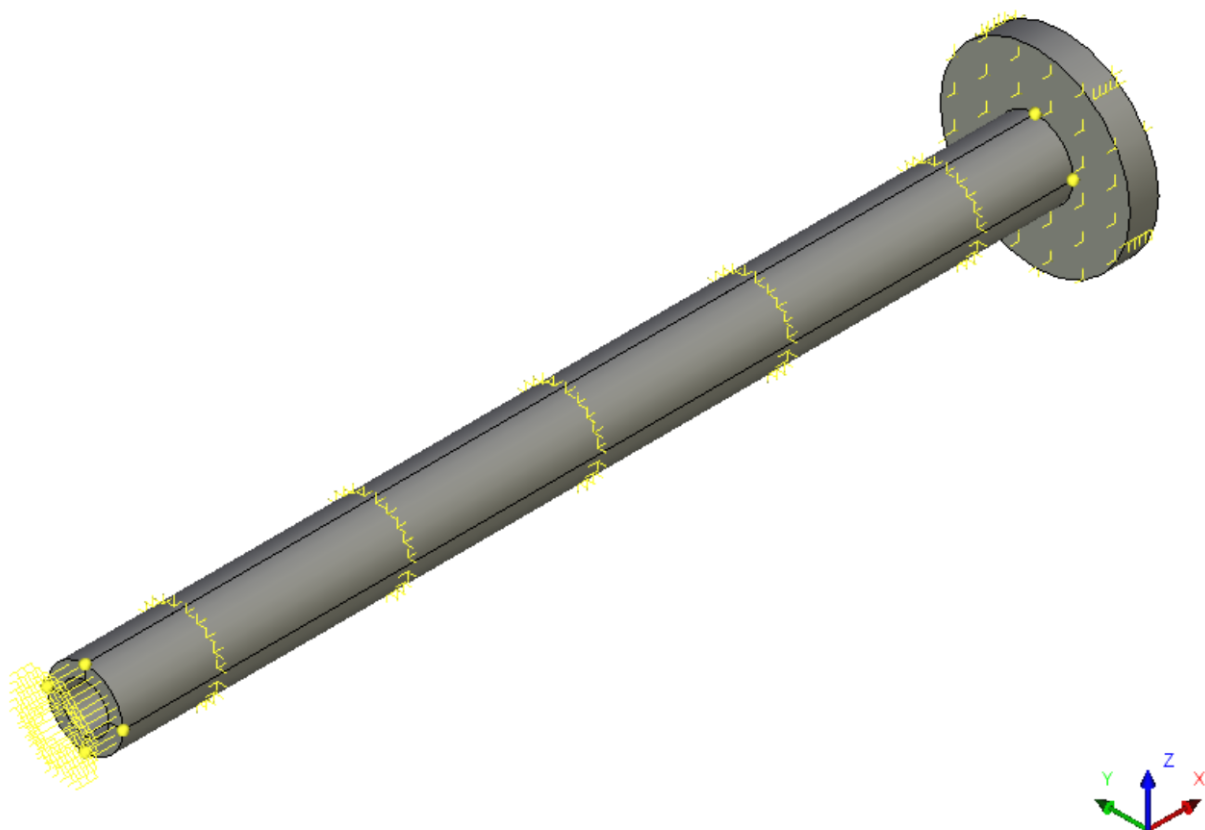Graphs of angles of rotation of three sections according to the time

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.
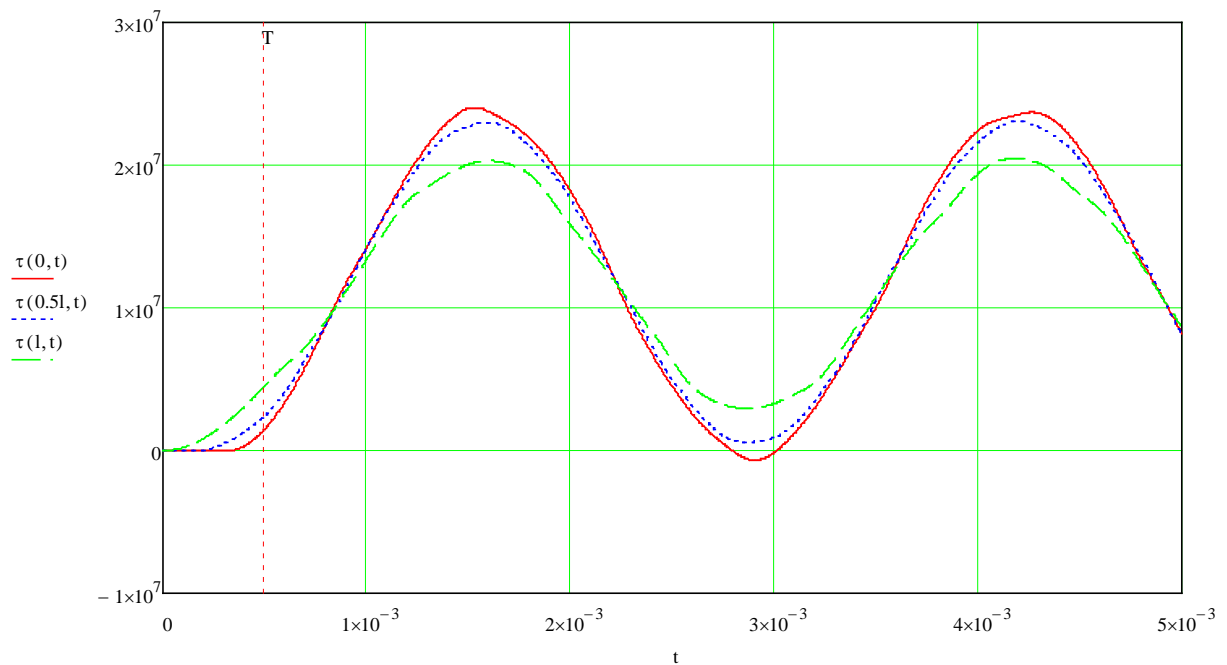
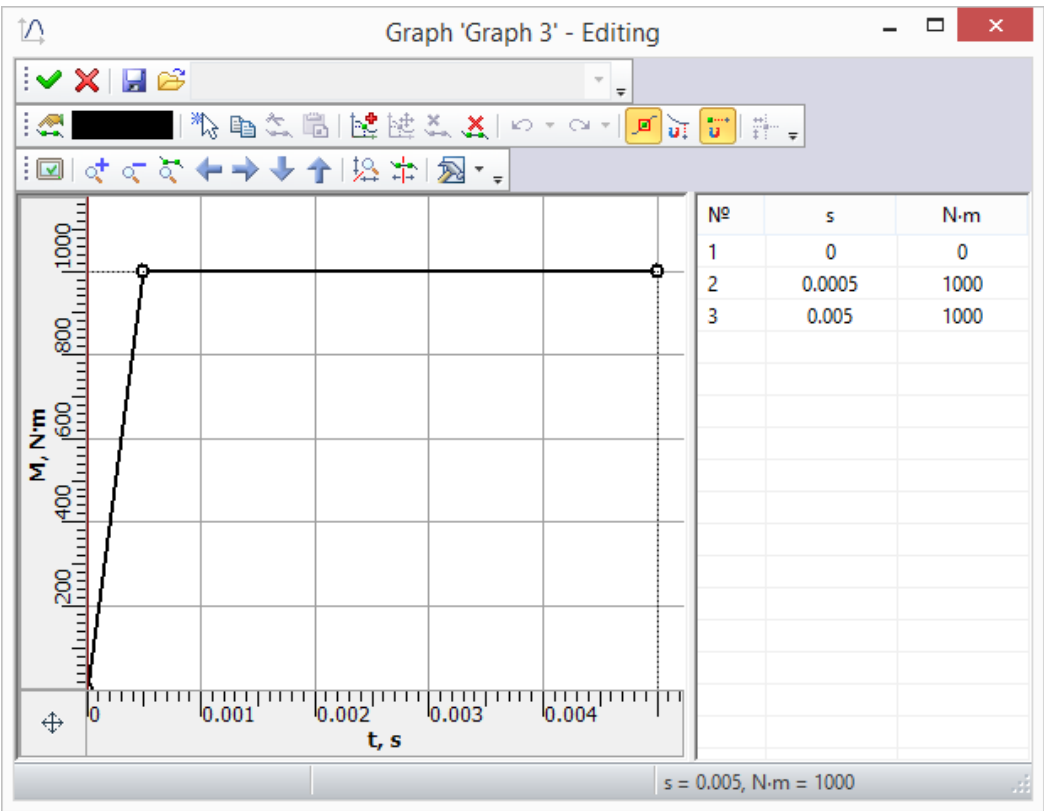Graphs of maximal shearing stresses in the section according to the time.

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:
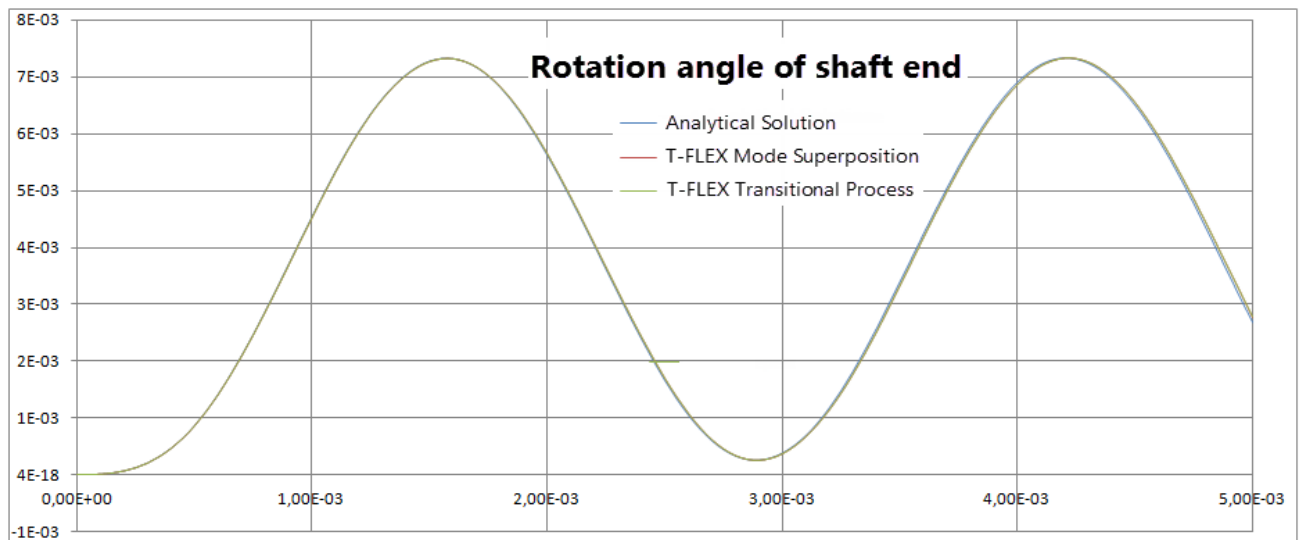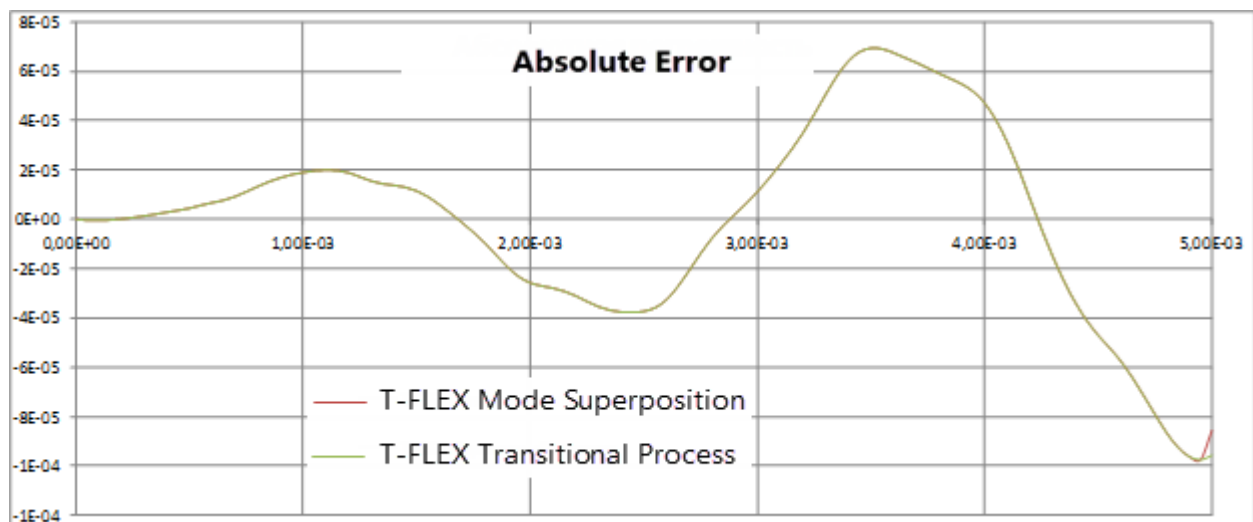
Load graph

For both of the studies, we set the finite modeling time 0,005 s, the time step of integration $5 \cdot 10^{-5}$ 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.
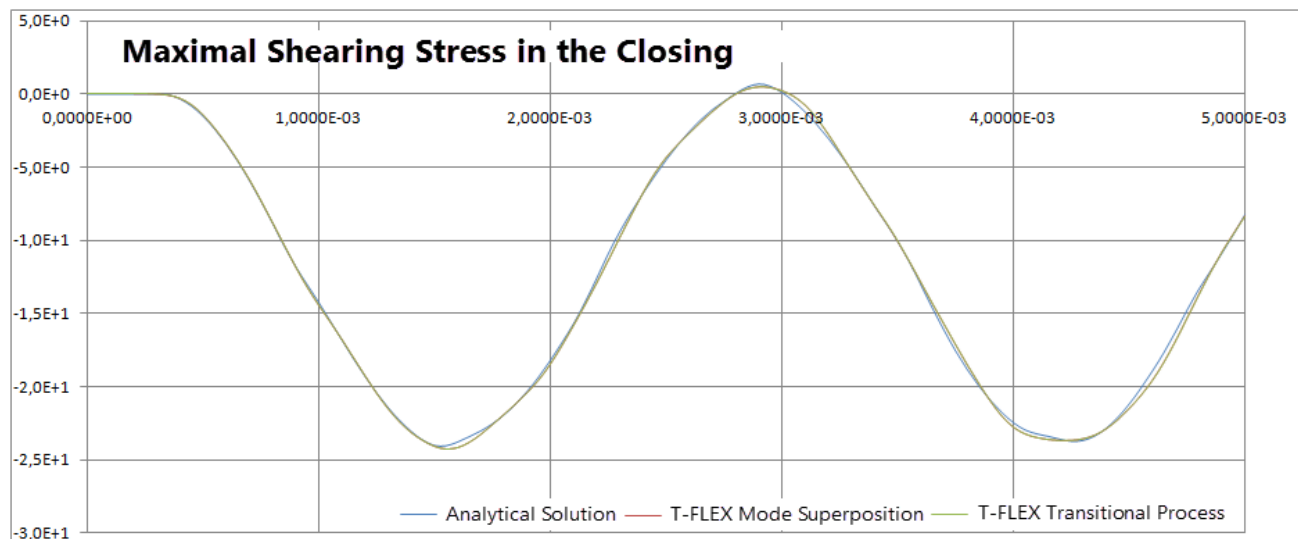
Graphs of rotation angle of free shaft end using analytical solution and solutions by two T-FLEX methods

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 the shaft end using analytical solution and solutions by two T-FLEX methods

On the figure below the graphs of shearing stress τ_{xy} 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 τ^A and considered in the theory of elasticity τ_{xy}^A (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 φ^*, *rad*	Analytical solution φ, *rad*	**Error** $\delta = 100\% (\varphi^* - \varphi) / \varphi$
$7{,}3173 \cdot 10^{-3}$	$7{,}326 \cdot 10^{-3}$	0.11

Table 4. Mode superposition, maximal torsion angle

Numerical solution φ^*, *rad*	Analytical solution φ, *rad*	**Error** $\delta = 100\% (\varphi^* - \varphi) / \varphi$
$7{,}3173 \cdot 10^{-3}$	$7{,}326 \cdot 10^{-3}$	0.11

Table 5. Transitional processes, maximal shearing stress

Numerical solution τ^*, *MPa*	Analytical solution τ, *MPa*	**Error** $\delta = 100\% (\tau^* - \tau) / \tau$
24.258	24.0	1.09

Table 6. Mode superposition, maximal shearing stress

Numerical solution τ^*, *MPa*	Analytical solution τ, *MPa*	**Error** $\delta = 100\% (\tau^* - \tau) / \tau$
24.249	24.0	1.05

Conclusion:

The maximal rotation angle of the shaft end with the wheel calculated using T-FLEX CAD is $7{,}3173 \cdot 10^{-3}$ 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 $24{,}249$ MPa (relative error 1,05%).