FIDESYS

strength analysis system

Version 1.6

User Guide

Contents

Introduction

At a glance

Fidesys Bundle is a software package for strength analysis. The package comprises the following types of analysis:

- Static
- Dynamic (transient)
- Buckling
- Modal

The package also includes a program ***Fidesys Viewer*** for visualization and analysis of the obtained results:

- Visualization of scalar and vector fields
- Graph
- Time dependency analysis

Getting Started

System requirements

CAE Fidesys from the beginning has been designed in such a way that the system requirements of the package are low: it can be run on an ordinary personal computer. If the computer has one or more multi-core processors calculations will be automatically parallelized on all cores. Starting version 1.5, calculation paralleization to several nodes connected to a local network or a cluster are available in the 64-bit version of the program package.

Fidesys Bundle software package has the following minimal requirements for software and hardware.

Hardware requirements

- CPU: Dual-core 1,7 GHz minimum
- RAM: 2GB minimum
- Free hard drive space: 5 GB
- Video card NVIDIA GeForce GTX 460 or faster
- Screen resolution: 1024x768 or higher

Operating system

The following Windows versions 32-*/64-bit are supported:

- Windows XP SP3;
- Windows Vista SP2;
- Windows 7;
- Windows 8;
- Windows Server 2008 (including R2);
- Windows Server 2012 (including R2).

The following Linux versions (only 64-bit) are supported:

- OpenSUSE 12.3;
- Red Hat EL 5.9, 6.4;
- Ubuntu Server 10.04;
- Ubuntu Desktop and Server 12.04;
- Debian 6.0.x;
- CentOS 6.5.

Spectral element method calculations and parallel calculations based on MPI technologies are available only in the 64bit version of the program package.

Installation

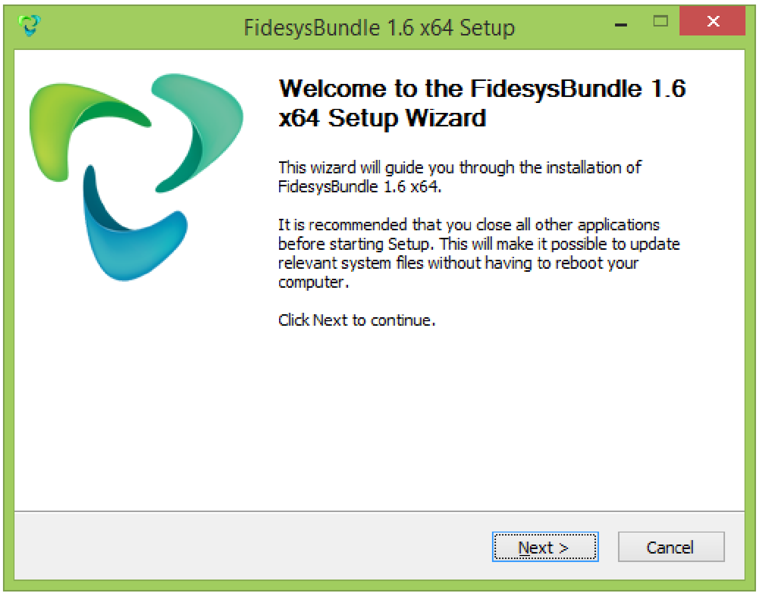
Microsoft Windows

The user installing the software on a computer must have administrator rights on that PC. Please, close all the ***Fidesys Bundle*** windows before installation if there's another version of CAE Fidesys installed.

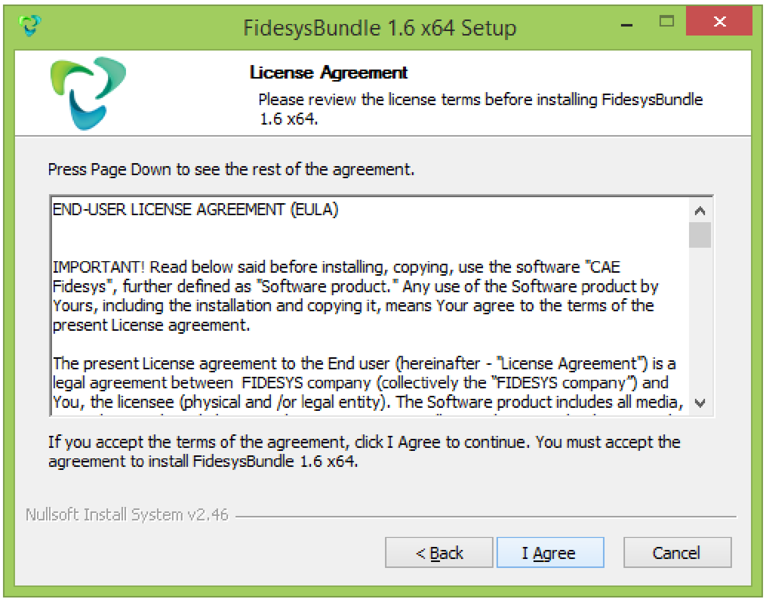
1. Download the CAE Fidesys installer from the site http://www.cae-fidesys.com/ru/download/login and run it for the architecture you are interested in (Windows x64 or Windows x32), or run the installation from the DVD-ROM.

If any other version of Fidesys Bundle is already installed on a computer, it will be suggested to remove it or to cancel the installation while starting the installation program.

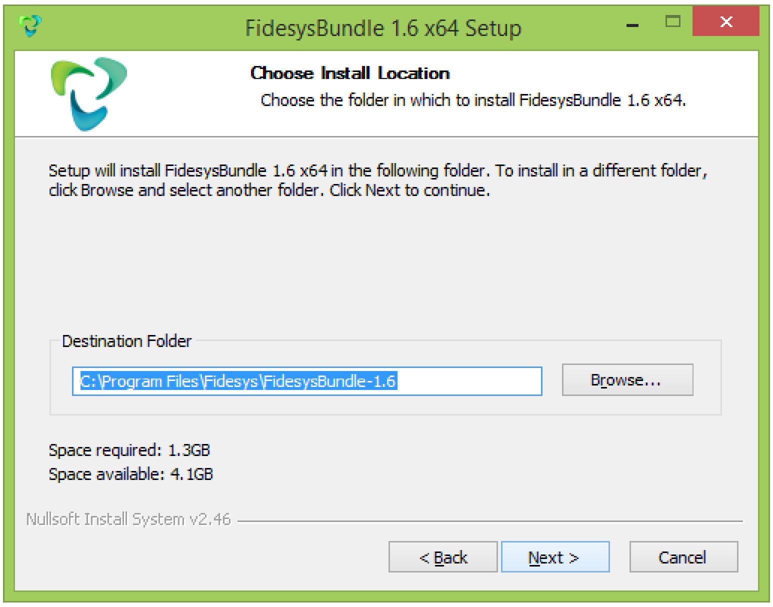
2. Click **Next** in a pop-up window.



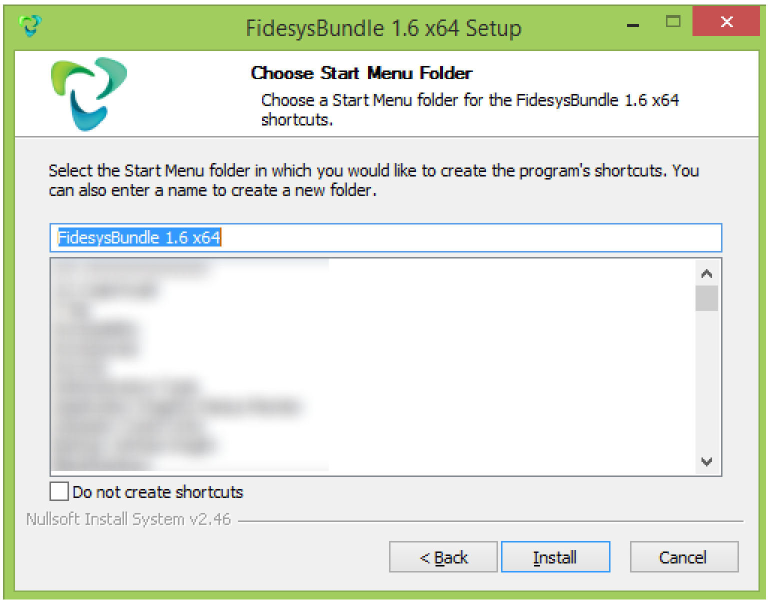
3. Please, read the license agreement. If you do not agree with any of its paragraphs, interrupt the installer by clicking **Cancel**. If you totally agree with its terms, click **Agree** to proceed the installation.



4. Select a folder for installation and click **Next**.



5. In the Start menu enter the name of the folder where a shortcut for running the program will be created. If you do not want to create a folder in the Start menu, choose **Do not create shortcuts**. Click **Install.**

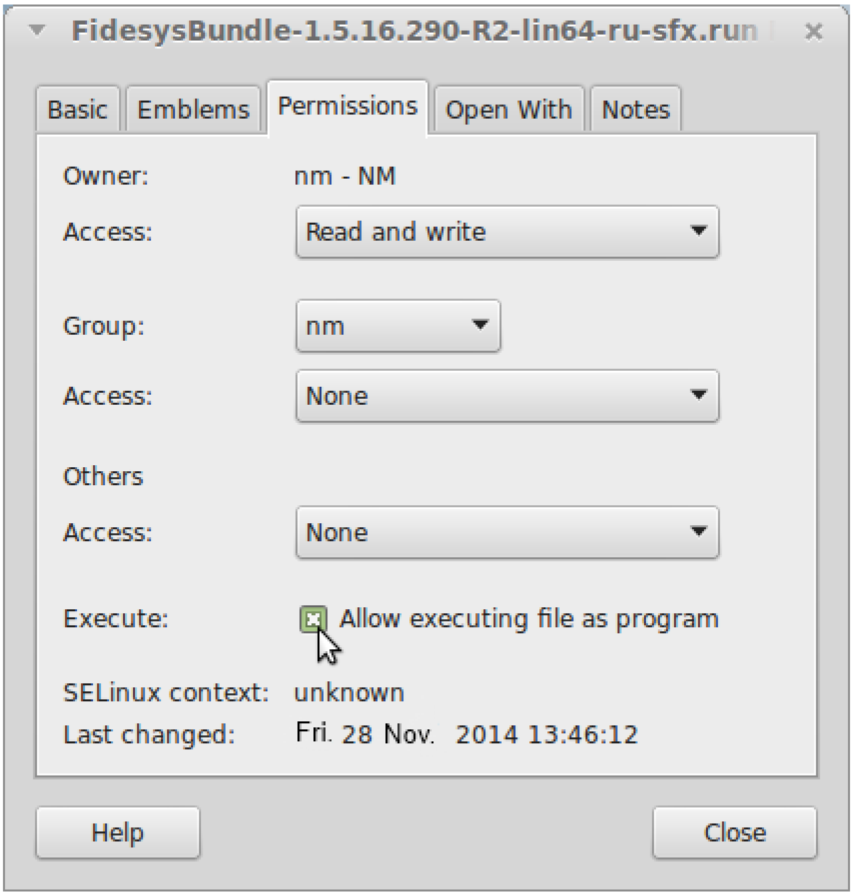


6. The process of installation may take some time. Click **Ready** after installing.

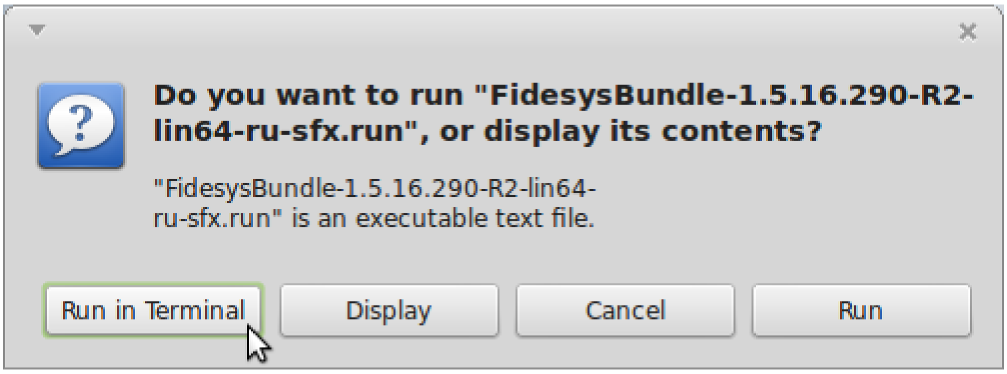
Linux

Only 64-bit Linux distribution kits are currently supported.

1. Please, download the CAE Fidesys file for Linux x64 from http://www.cae-fidesys.com/ru/download/login.
2. Right-click on the downloaded file and select **Properties** item from the contextual menu.
3. In the opened window, go to the tab **Permissions** and tick **Allow executing file as program**. Click **Close**.



4. Run the installer by double-clicking on the installer file. When the dialog box appears, click **Run in Terminal** button**:**



Activation and trial period

When you first run the preprocessor, the ***Fidesys Licensing*** window appears with a proposal to purchase a license or to activate a trial period.

Trial period

30-day trial period activates automatically on package installation. The trial period starts at the moment when application installation is completed. The trial period is intended for familiarization with the product and is not for any commercial calculations (related directly or indirectly to getting a profit out of them). The trial period can not be activated on a virtual machine, and a trial version is not dedicated to work with via remote desktop.

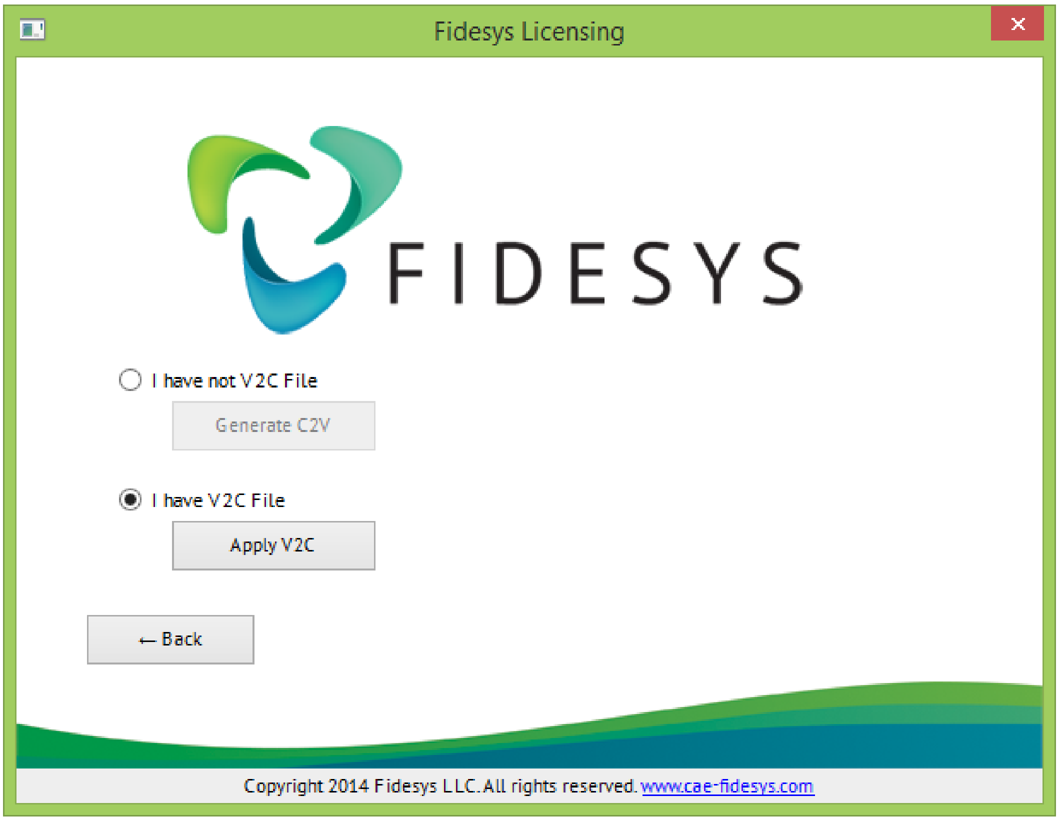
To activate a trial period, click the button **Trial period** in the start window.

As long as the program is running in trial mode, the ***Fidesys Licensing*** window will appear each time you launch it. Please, press the button **Try** to continue working in a trial mode.

Activation

To activate the product:

1. Press the button **Activate** in the ***Fidesys Licensing*** window.
2. Select **I do not have a V2C file** and click **Generate C2V.** Save File dialog window will be opened. Save the C2V file and send it to the organization where the product was purchased.
3. In response, you will get a file containing an activation key with V2C extension. Having received the V2C file, select **I have V2C file** and click **Apply V2C.** You will see an Open File dialog window. Specify a path to the received V2C file.



4. Your product is activated.

Information on the purchased license

On selecting **Help** → **About** in the Main Menu, you will see a window with the following information:

- The full software version number;
- License type and its expiration date;
- The list of features available in the purchased license.

Removing the software

The user removing the software must have administrator rights.

Please, finish all the running copies of the application before removing the software: both preprocessor (Fidesys) and postprocessor (Fidesys Viewer).

To remove the software, open Windows Control Panel and select **Programs and Features** (**Add or Remove Programs** in the earlier versions of Windows). Select *Fidesys Bundle #.#.#.# xNN* in the list of installed programs, where #.#.#.# are the four numbers standing for the number of the version and *xNN* is the architecture (x64 or x32). Right-click it and choose **Delete/Change**. Confirm your choice by clicking **Delete** in the opened window.

Removing the software does not involve removing its activation data.

Overview

Package structure

Fidesys Bundle comprises two main components:

- **Fidesys** — preprocessing and analysis (computational kernels).
- **Fidesys Viewer** — postprocessing and visualization of results.

Running the software

You can run the program in either of the following ways:

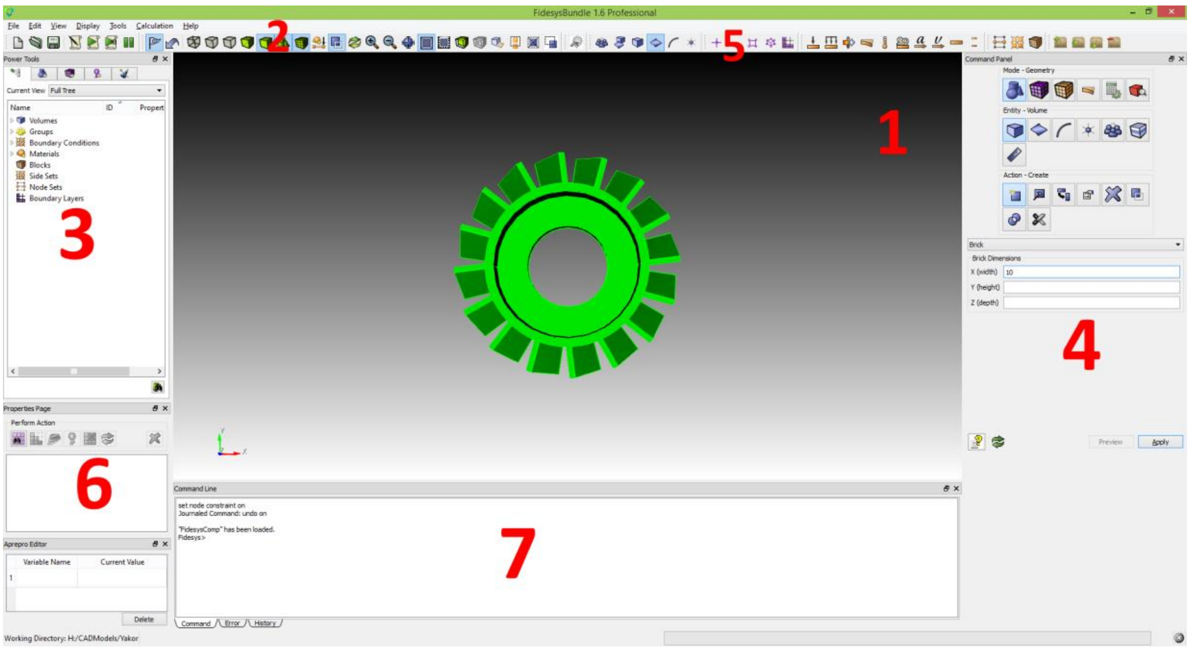
- Using the Start menu (if you chose creating shortcuts in it when installing): choose **Fidesys** in the folder where you installed the program (**FidesysBundle** by default).
- Using any file manager for Windows from the list where the program was installed (**C:\Program Files\Fidesys\FidesysBundle** by default): run the file **fidesys.exe** (it is in the folder **preprocessor\bin**).

Several copies of the program can be run on the same PC at a time.

If the license fee for the program is activated, after running the program you see its Main window. If the trial period is activated, a ***Fidesys Licensing*** window appears in which you should either click the button **Activate** in order to purchase a license or click **Try** to continue working in trial mode and go to the Main window.

Main Window

Fidesys Bundle has an intuitive graphic interface providing communication between the user and the software, and it allows the user to perform the full cycle of calculations step-by-step.



Workbench (1) displays the model and visual effects.

Main Menu (2) includes standard operations for working with files and projects, managing the visualization modes, panel display settings, help, and other functionality available in the drop-down lists of the menu.

Power Tools (3) comprise the Model Tree, as well as the tools for geometry and mesh analysis.

Command Panel (4) contains most of commands for working with the program. Panel display buttons are logically located, and it allows the user to perform the full cycle of calculations step-by-step.

Toolbar (5) comprises the buttons for calling the most frequently used commands while working with the program.

Properties Page (6) displays the properties of the selected object in the Workbench or in the Model Tree.

Console (7) is used for the input of commands of ***Fidesys Bundle*** and for the output of the messages for the user.

Software history

Version 1.6 R2

Released: April, 2015

Updates and improvements in the preprocessor

- The possibility to automatically process the calculation results of composites effective properties.
- The stability of the operations is increased.

Updates and improvements in the processor

- Elastoplastic deformation by the Drucker-Prager model
- Calculation of the effective properties of composite materials
- HPC and Dynamics modules are available in Standard and Professional

Updates and improvements in the postprocessor (3D-vizualization module)

- Programming interface operation based on Python Shell is improved.

Version 1.6

Released: February, 2015

- The support for importing geometry in the following CAD-formats is added:
  - SolidWorks;
  - Parasolid;
  - Pro/Engineer.
- The support of APREPRO (An Algebraic Preprocessor for Parameterizing Finite Element Analyses) is added
- New profiles of beam cross section are added:
  - Channel (C-shape);
  - Corner (L-shape);
  - Taurus (T-shape);
  - Z-shape;
  - Hollow rectangle.
- The possibility to set the boundary conditions using tabular and formular dependency on the coordinates and temperature for static analysis is added.
- The new generator of adaptive tetrahedral meshes is added.
- In accordance with the users wishes, the panel of nonlinear solver settings is changed.
- Contact problems in 3D and 2D
- Contact surface binding in 3D and the contact curves binding in 2D
- Modal analysis for prestressed bodies
- The automatic adaptive calculation of the loading steps size is added.
- The output of intermediate results and calculation log (textual information on the status of the calculation by stages) are added for nonlinear problems.
- Thermoelastic problems for 2nd order shell elements: TRISHELL6, SHELL8, SHELL9.
- The support for multiprocessor calculations based on MPI technology for the following calculations is added:
  - Spectral element method;

  - Plasticity for small and finite deformations;
  - Dynamic analysis for the explicit and implicit time schemes.
- The incorporation of plastic deformation using finite element method and spectral element method is modified taking into account:
  - Finite deformations;
  - Thermoelasticity.
- Dynamic analysis perfomance is improved
- Performance of spectral element method on hybrid meshes in buckling problems is improved
- The solution tolerance of static problem with shear loading using 4-node plane and 8-node volumetric elements is increased.
- Fidesys Viewer 1.6: focusing of model elements in the current position of the mouse pointer is added.
- Fidesys Viewer 1.6: strength analysis filter operation is improved.
- Fidesys Viewer 1.6: operation of the “Agreed resultants” filter for shell structures is improved.
- The compatibility with Windows 8 and Windows 8.1 is improved.
- The Progress Bar operation is improved.
- The Fidesys Bundle licensing system is added.

Version 1.5 R2

Released: July, 2014

- The ability to set the analytic spatial coordinates dependence of the boundary conditions is added.
- The possibility of producing graphs of the spatial coordinates and time dependency of the boundary conditions is added.
- The generation of console commands for setting up and running the calculation of the graphical interface widgets is added.
- Import/export of materials from the graphical interface are added.
- The package includes a new version of Fidesys Viewer with a number of improvements.

Version 1.5

Released: June, 2014

- Static analysis for elastoplastic material models
- Orthotropic materials
- Physically nonlinear hyperelastic materials: Mooney-Rivlin and Murnaghan.
- Calculations by spectral element method for hybrid meshes.
- The possibility of parallel calculations on one or more computers using the MPI technology (linear statics, modal analysis, buckling) is added.
- The calculation of the margin of safety in accordance with various strength theories.
- 8-node shell elements SHELL8.
- Console commands to set analysis parameters and to run the calculation.
- The ability to set the time dependence individually for each boundary condition in dynamic problems.
- Windows XP compatibility issues are fixed.

Version 1.4

Released: December, 2013

- Buckling problems
- Thermal conductivity and thermoelastic problems.
- Curvilinear finite elements.
- Geometrically nonlinear problems
- The spectral element method for linear and nonlinear two-and three-dimensional static problems and modal analysis
- Support for hybrid meshes
- Bug fixing

Version 1.3

Released: July, 2013

- Beam elements
- Shell elements
- Geometry creation of high-speed processes by the spectral element method is added.
- Static and dynamic nonlinear problems
- High-order finite elements
- Bug fixing

Version 1.2

Released: February, 2013

- Computational performance is improved.
- Plane-stress and plane-strain problems
- The translation into cylindrical and spherical systems is added in the postprocessor.
- The calculation of tensors invariants is added in the postprocessor.
- The visualization of the calculation results as contour lines is added in the postprocessor.

Version 1.1

Released: November, 2012

- Parameter setting and calculation launch from the Main Menu of the preprocessor are added.
- Dynamic transient problems
- Modal analysis
- Hexahedral meshes support
- The operation with projects and the calculation control system are added.
- The postprocessor’s localization support
- The postprocessor performance is improved.

Using the Program

Performing calculations with the use of ***Fidesys Bundle*** implies the following steps:

- Setting the geometry
- Meshing
- Setting boundary conditions
- Setting the material
- Starting calculation
- Visualizing and analyzing results

All of the steps except for the last one are made in preprocessor; the last step is made in postprocessor.

Geometry

Fidesys Bundle allows one to generate volume geometry on your own due to the built-in functionality, as well as to import 3D models created in different CAD-systems.

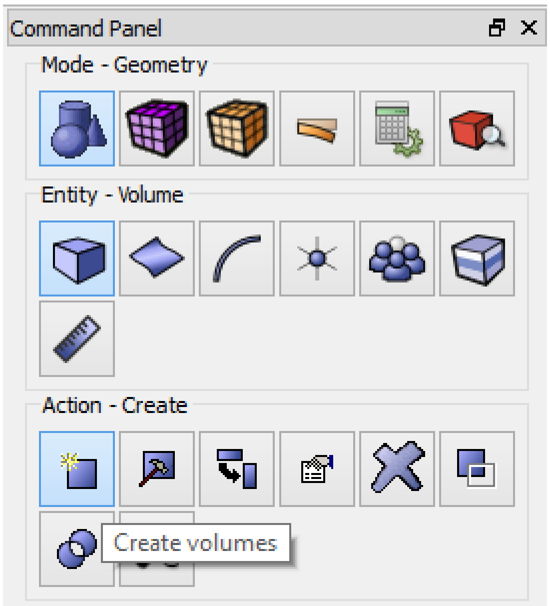
Geometry import

For geometry import choose **File** → **Import** in the Main menu. ***Fidesys Bundle*** supports the import of the following formats:

- ACIS (*.sat, *.sab);
- IGES (*.igs, *.iges);
- STEP (*.stp, *.step);
- AVS (*.avs);
- Genesis/Exodus (*.g, *.gen, *.e, *.exo);
- Facets (*.fac);
- STL Files (*.stl);
- Patran (*.pat, *.neu, *.out);
- Ideas (*.unv);
- Abaqus (*.inp);
- Fluent (*.msh);
- Nastran (*.bdf);
- Cubit (*.cub);
- Catia (*.CATPart, *.CATProduct, *.ncgm).

Geometry generation

For geometry generation ***Fidesys Bundle*** provides the user with large numbers of volume geometric elements (parallelepiped, cylinder, prism, cone, pyramid, sphere, torus). It also allows uniting the surfaces in closed volume bodies. For complex geometry generation you can use Boolean operations (Intersect, Subtract, Unite volumes) and different transformations of the object (Rotate, Move, Scale, Reflect). All of the described functionality is available on Command Panel in **Geometry** section.



Meshing

Fidesys Bundle supports the following types of the finite elements for meshes:

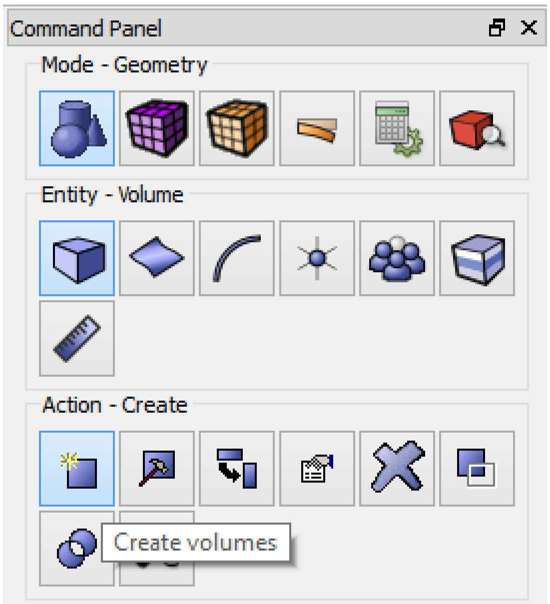
- 3D: 4-noded tetrahedron (TETRA/TETRA4), 10-noded tetrahedron (TETRA10), 8-noded hexahedron (HEX/HEX8), 20-noded hexahedron (HEX20), 27-noded hexahedron (HEX27), 5-noded pyramid (PYRAMID/PYRAMID5), 13-noded pyramid (PYRAMID13), 6-noded wedge (WEDGE/WEDGE6), 15-noded wedge (WEDGE15);
- 2D: 3-noded triangle (TRI/TRI3), 6-noded triangle (TRI6), 4-noded quadrilateral (QUAD/QUAD4), 8-noded quadrilateral (QUAD8), 9-noded quadrilateral (QUAD9);
- Shells: 3-noded triangle (TRISHELL/TRISHELL3), 4-noded quadrilateral (SHELL/SHELL4), 6-noded triangle (TRISHELL6), 8-noded quadrilateral (SHELL8), 9-noded quadrilateral (SHELL9);
- Beams: 2-noded beam (BEAM/BEAM2)

Fidesys Bundle supports the following types of the spectral elements for meshes (see the section ***Spectral element method***):

- 3D: 4-noded tetrahedron (TETRA/TETRA4), 10-noded tetrahedron (TETRA10), 8-noded hexahedron (HEX/HEX8), 20-noded hexahedron (HEX20), 27-noded hexahedron (HEX27), 5-noded pyramid (PYRAMID/PYRAMID5), 13-noded pyramid (PYRAMID13), 6- noded wedge (WEDGE/WEDGE6),15-noded wedge(WEDGE15);
- 2D: 3-noded triangle (TRI/TRI3), 6-noded triangle (TRI6), 4-noded quadrilateral (QUAD/QUAD4), 8-noded quadrilateral (QUAD8), 9-noded quadrilateral (QUAD9);

Volume mesh generation

Select volume mesh generation section on Command Panel (Mode — **Mesh**, Entity — **Volume**).



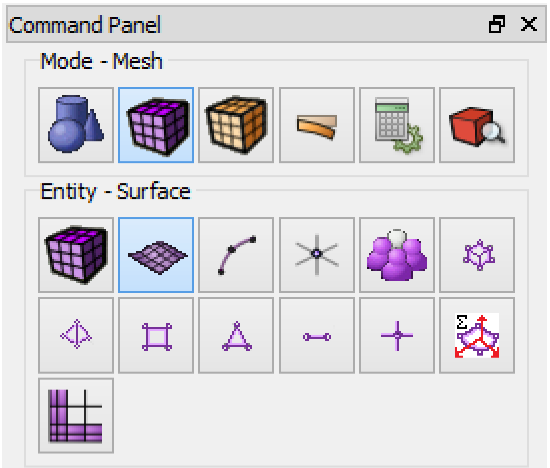
1. Specify the degree of mesh refinement (Action – **Intervals**) for each volume:
    - Select the volumes (specify their ID). You can enumerate several volumes using space after each of them. All of the volumes can be set by the command **all**.
    - Select the way of mesh generation (Auto, Approximate size, Geometry-adaptive, Interval or Sizing function).
    - Click **Apply Scheme**.
2. Specify the type of the elements for each volume:
    - Select the entities for mesh generation (specify their ID). You can enumerate several volumes using space after each of them. All of the volumes can be set by the command **all**;
    - Select meshing scheme (tetrahedral (Tetmesh) or hexahedral elements (Automatically calculate);
    - For tetrahedral mesh generation select the level of optimization (Extreme, Strong, Heavy, Standard, Medium, Light, or None) and set the checkboxes in front of the corresponding points, if you need to minimize the over-constrained and/or sliver tets.
    - Click **Apply Scheme;**
    - Click **Mesh.**

For complex geometry it is recommended to set the scheme of surface mesh generation first (triangular or quadrangular elements).

Surface mesh generation

To generate a surface mesh, follow these steps.

1. Select surface mesh generation section on Command Panel (Mode – **Mesh**, Entity – **Surface**).



2. Specify the degree of mesh refinement (Action – **Intervals**) for each surface:
    - Select surfaces (specify their ID). You can enumerate several surfaces using space after each of them. All of the surfaces can be set by the command **all**;
    - Select the way of mesh generation (Auto, Approximate size, Geometry-adaptive, Interval or Sizing function);
    - Click **Apply Scheme**.

3. Specify the type of the elements for each surface:
   - Select the entities for mesh generation (specify their ID). You can enumerate several surfaces using space after each of them. All of the surfaces can be set by the command **all**;
   - Select meshing scheme (triangular elements (Trimesh or TriDelaunay) or quadrangular elements (Automatically calculate);
   - Click **Apply Scheme**.

To generate an irregular mesh (e.g. make it finer in the vicinity of stress concentrators), you can add nodes on the boundaries near geometry features, as well as split curves, surfaces and volumes in the vicinity of the features.

Using the functionality available on Command Panel you can:

- Check the mesh quality (including checking the mesh quality of individual elements: volumes, surfaces, curves);
- Modify the generated mesh (Refine, Smooth, Cleanup);
- Renumber the elements and delete the generated mesh.

Setting material and element type

Element types

Fidesys Bundle makes analysis using the model made of linear isotropic material subject to Hook's law.

Fidesys Bundle supports the following types of the finite and spectral elements for meshes:

- 1st order 3D: 4-noded tetrahedron (TETRA/TETRA4), 8-noded hexahedron (HEX/HEX8), 5-noded pyramid PYRAMID5, 6-noded wedge WEDGE6;
- 2nd order 3D: 10-noded tetrahedron (TETRA10), 20-noded hexahedron (HEX20), 27-noded hexahedron (HEX27), 13-noded pyramid PYRAMID13, 15-noded wedge WEDGE15;
- 1st order 2D: 3-noded triangle (TRI/TRI3), 4-noded quadrilateral (QUAD/QUAD4);
- 2nd order 2D: 8-noded quadrilateral (QUAD8), 9-noded quadrilateral (QUAD9), 6-noded triangle (TRI6).

Fidesys Bundle supports the following types of the only finite elements for meshes:

- 1st order shell elements: 3-noded triangle (TRISHELL/TRISHELL3), 4-noded quadrilateral (SHELL/SHELL4);
- 2nd order shell elements: 6-noded triangle (TRISHELL6), 8-noded quadrilateral (SHELL8), 9-noded quadrilateral (SHELL9);
- Beam elements: 2-noded beam (BEAM/BEAM2).

***Fidesys Bundle* supports the following permissible combinations of elements in one model:**

- HEX/HEX8, TETRA/TETRA4, PYRAMID/PYRAMID5, WEDGE/WEDGE6, SHELL/SHELL4, TRISHELL/TRISHELL3, BEAM/BEAM2;
- HEX20, TETRA10, PYRAMID13, WEDGE15, SHELL8, TRISHELL6;
- HEX27, SHELL9;

- QUAD/QUAD4, TRI/TRI3;
- QUAD8/QUAD9, TRI6.

By default the surface element type is a shell element (SHELL 4 or TRISHELL3). Thus, in case of 2D problem, it is necessary to change element type to QUAD4 or TRI3.

In case of 3D problems the element type is HEX8 or TETRA4 by default.

By default the beam element type is BAR2 that is not supported in the current version of ***Fidesys Bundle***. Thus, in this case it is necessary to change element type to BEAM/BEAM2.

Note: you can find out the type of element and statistics on the specific model. For this purpose, please right-click on the model and select **Mesh info** from the drop-down list. The required information will be displayed in the Console.

Material groups

Fidesys Bundle supports the following materials:

- Hook's material;
- Orthotropic material;
- Mooney-Rivlin material;
- Murnaghan material.

For Mooney-Rivlin and Murnaghan materials, the following defining relations are used.

Mooney-Rivlin potential:

$$W = C_1(\overline{I}_1 - 3) + C_2(\overline{I}_2 - 3) - D(J-1)^2,$$

where D, C_1, C_2 are Mooney-Rivlin material constants.

Relation of D, C_1, C_2 and Poisson's ratio ν:

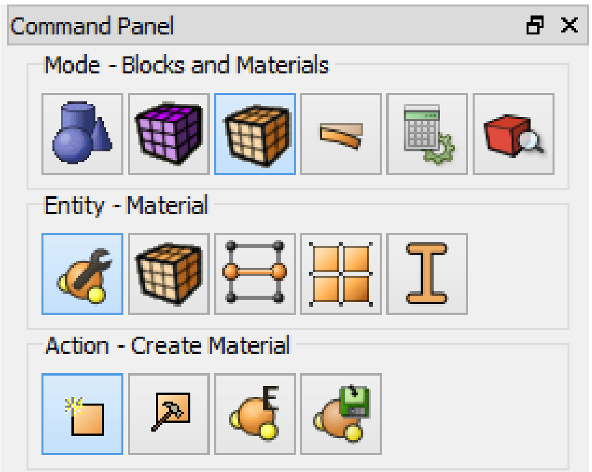
$$D = \frac{C_1 + C_2}{1-2\nu}.$$

Murnaghan potential:

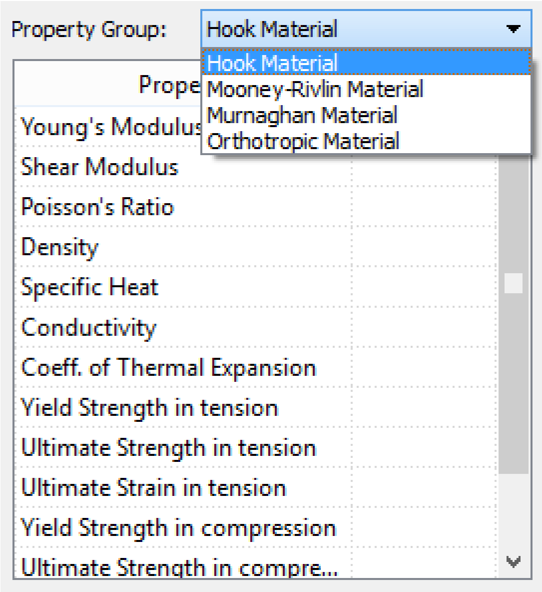
$$\overset{0}{\Sigma}_{0,n} = \lambda(\overset{0}{\varepsilon}\cdot\cdot I)I + 2G\overset{0}{\varepsilon} + 3C_3(\overset{0}{\varepsilon}\cdot\cdot I)^2 I + C_4(\overset{0}{\varepsilon}{}^2\cdot\cdot I)I + 2C_4(\overset{0}{\varepsilon}\cdot\cdot I)\overset{0}{\varepsilon} + 3C_5\overset{0}{\varepsilon}{}^2$$

where λ, G, C_3, C_4, C_5 are Murnaghan material constants.

To set the new material, select the setting material properties section on Command Panel (Mode – **Blocks and materials,** Entity – **Material,** Action – **Create material**).



Select a group of material properties from the drop-down list; specify the name of the material and the corresponding constants:



Note: In order to link the material and the model, **Block** is used**.**

Blocks operations

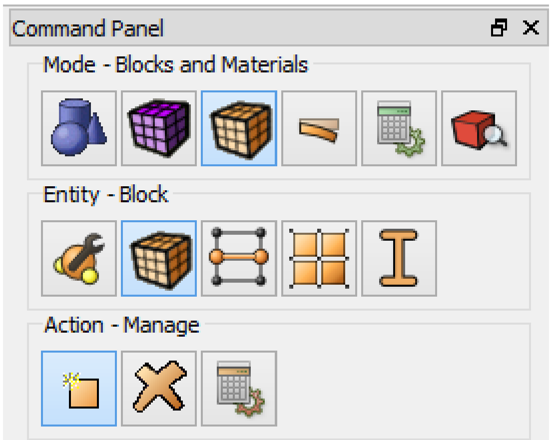
A block must contain an element type, ID and the name of the geometric model of the material.

The sequence of operations with blocks can be schematically represented as follows:

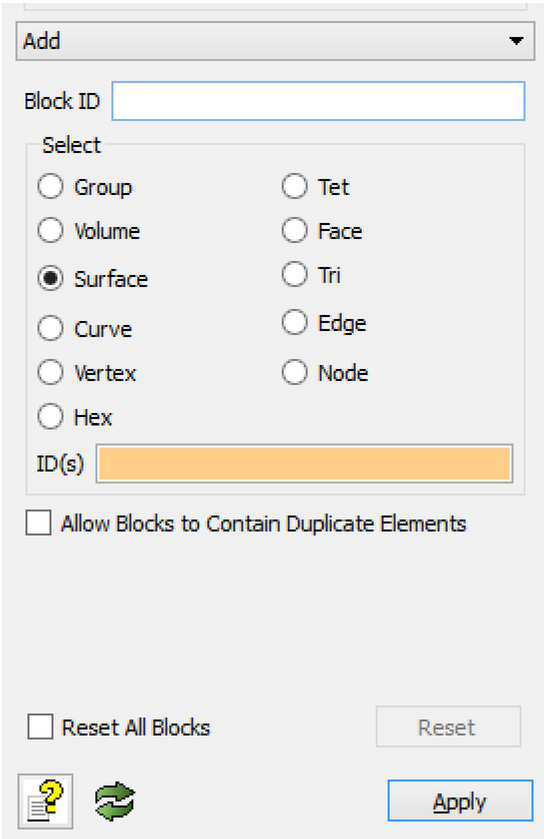
- To create block specifying geometric object ID;
- To assign the material to the block;
- To assign the element type to the block.

Let us consider these steps in detail.

1. To create a new block, please, go to Mode – **Blocks and materials**, Entity – **Block**, Action – **Create.**

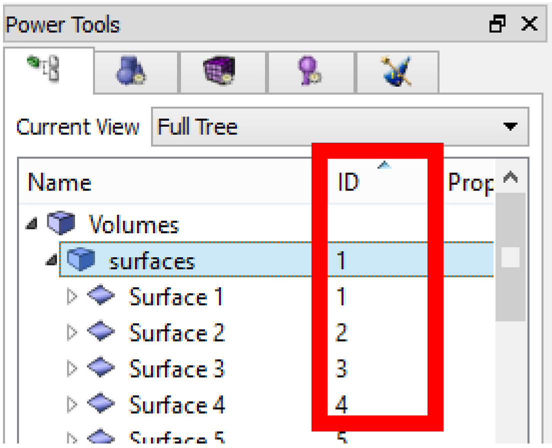


On the Panel below, select Action – **Add**, choose the entity type to be united into the block.



You can find out the ID of the geometrical entities united into the block as follows:

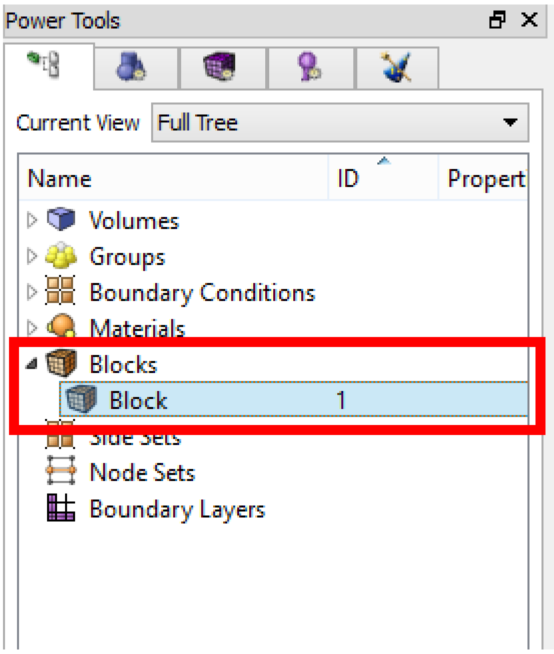
- in the Model Tree on the left;



- by clicking on geometrical objects you are interested in – their ID will automatically appear in the appropriate field.

The block ID field is filled automatically.
Click **Apply.**

Note: Created block must be displayed in the Model Tree on the left in the section **Blocks.**

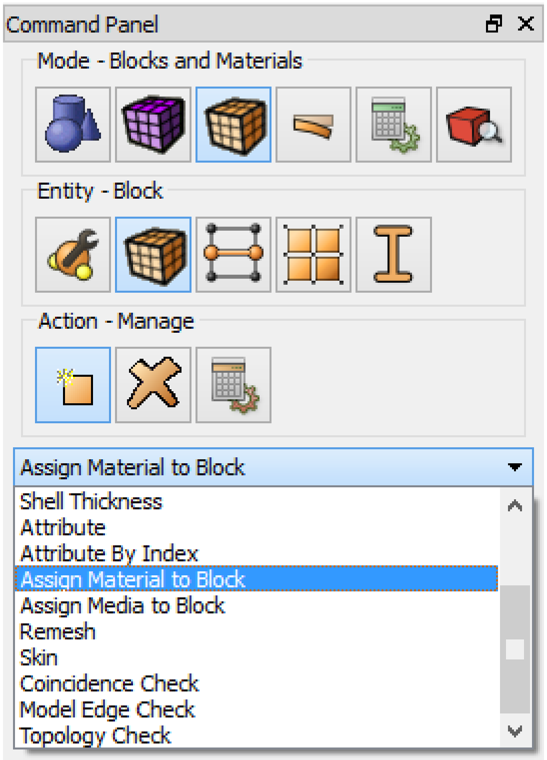


To look through the list of the geometric entities united into the block, enter in Command Line

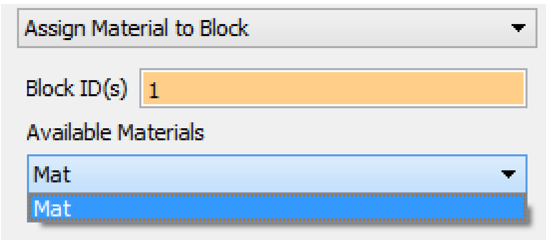
```
List block 1
```

In the Console, you will see the list of entities united into the block.

2. To assign the material to the block, select **Assign Material to Block** in the drop-down list.

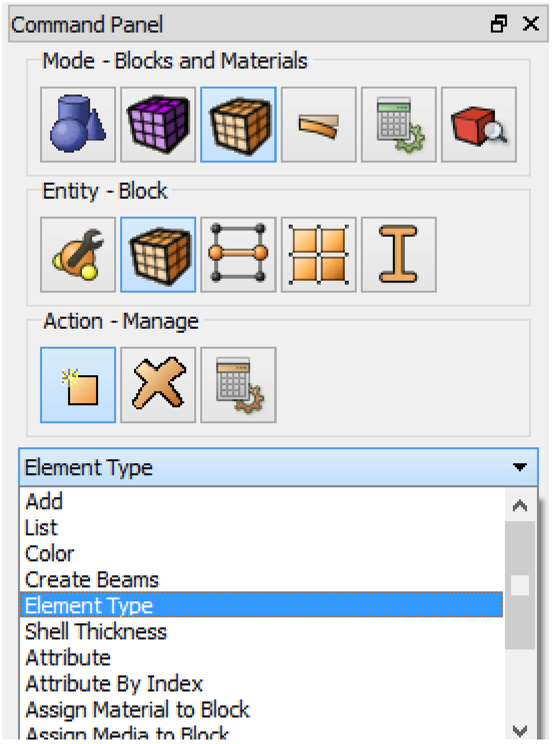


To enter the block ID, you can do it manually or click on the corresponding geometric entity. Available material is selected from the drop-down list.

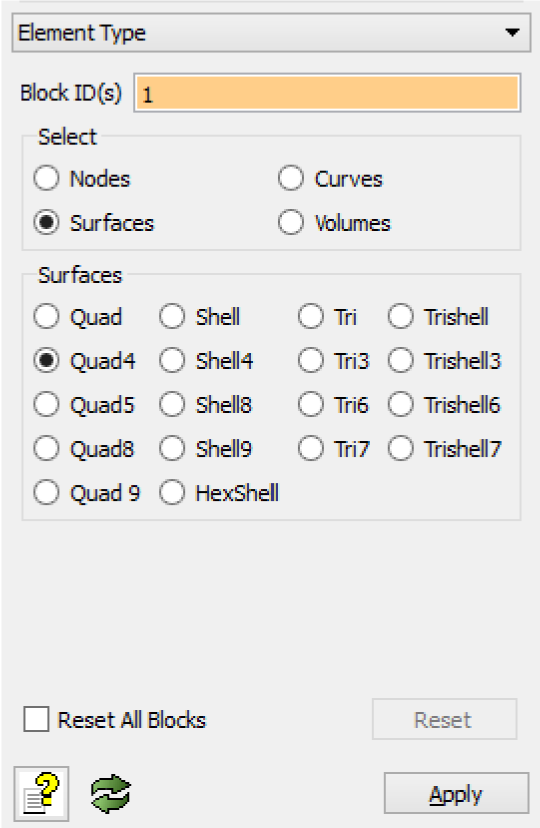


Click **Apply**.

3. To set the element type, select **Element Type** in the drop-down list.



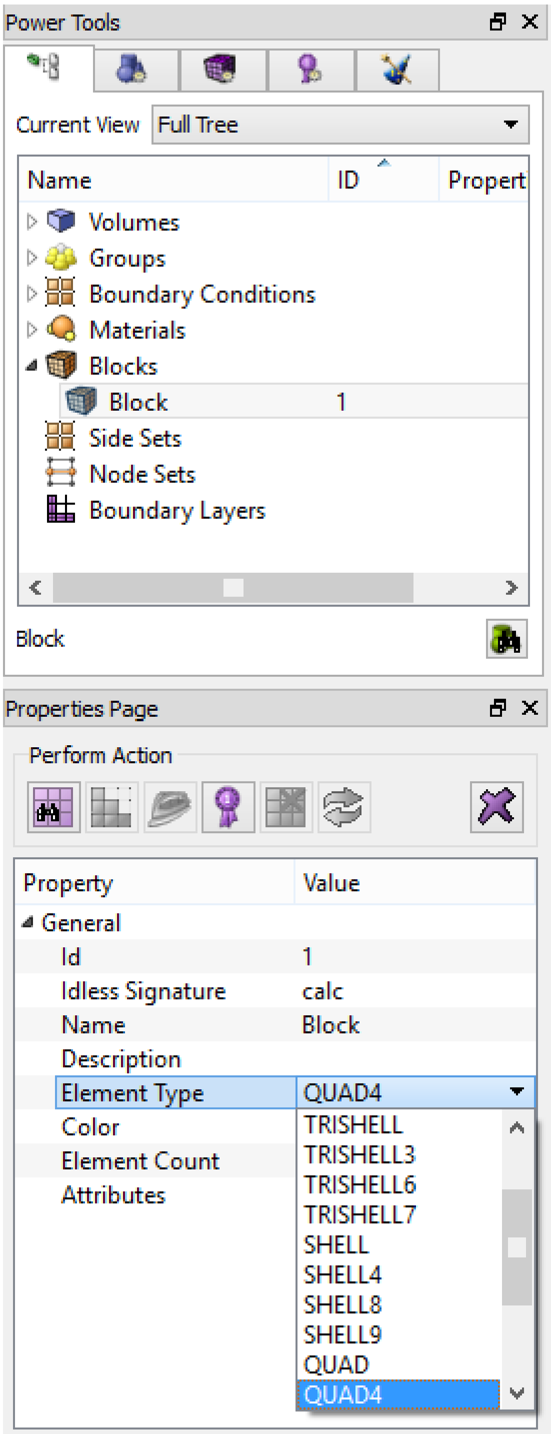
To enter the block ID, you can do it manually or click on the corresponding geometric entity. Available material is selected from the drop-down list.
Click **Apply.**


Element Type
Block ID(s) 1
Select
Nodes
Curves
Surfaces
Volumes
Surfaces
Quad
Shell
Tri
Trishell
Quad4
Shell4
Tri3
Trishell3
Quad5
Shell8
Tri6
Trishell6
Quad8
Shell9
Tri7
Trishell7
Quad 9
HexShell
Reset All Blocks
Reset
Apply

Changing element type chosen for the block

You can change the element type previously assigned to the block on the **Properties page** on the left.

For this purpose, select the required block in the Model Tree and specify the new element type below in the drop-down list on the property page.

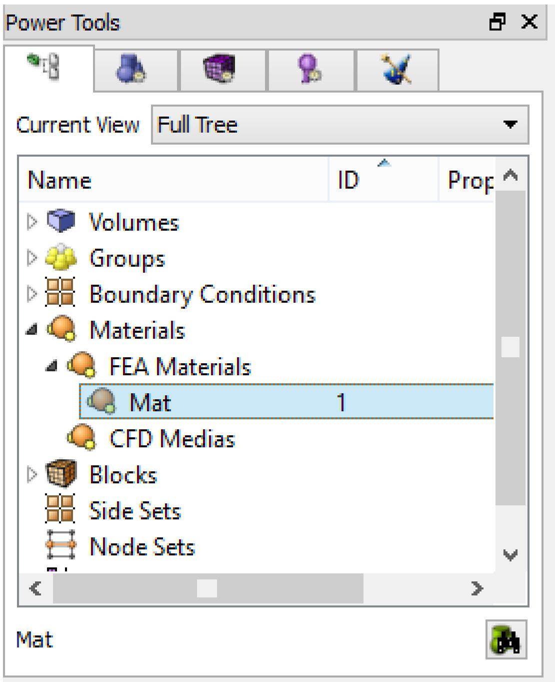


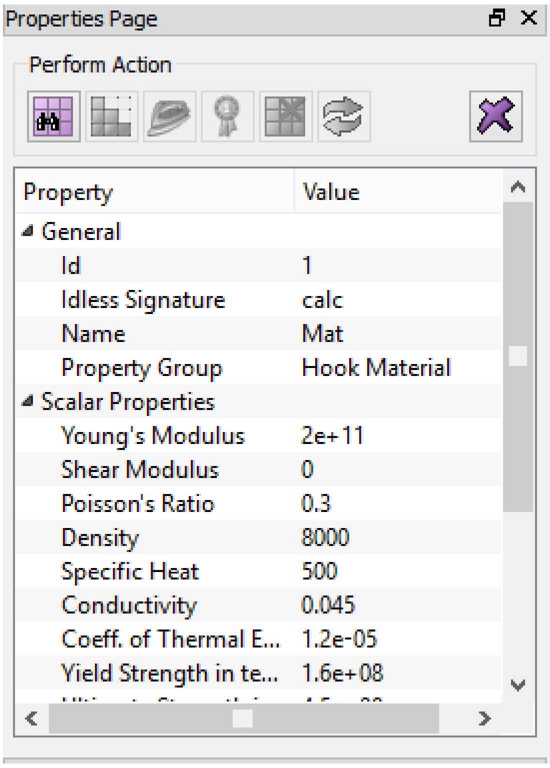
Changing material properties

You can change the material properties previously assigned to the block on the **Properties page** on the left.

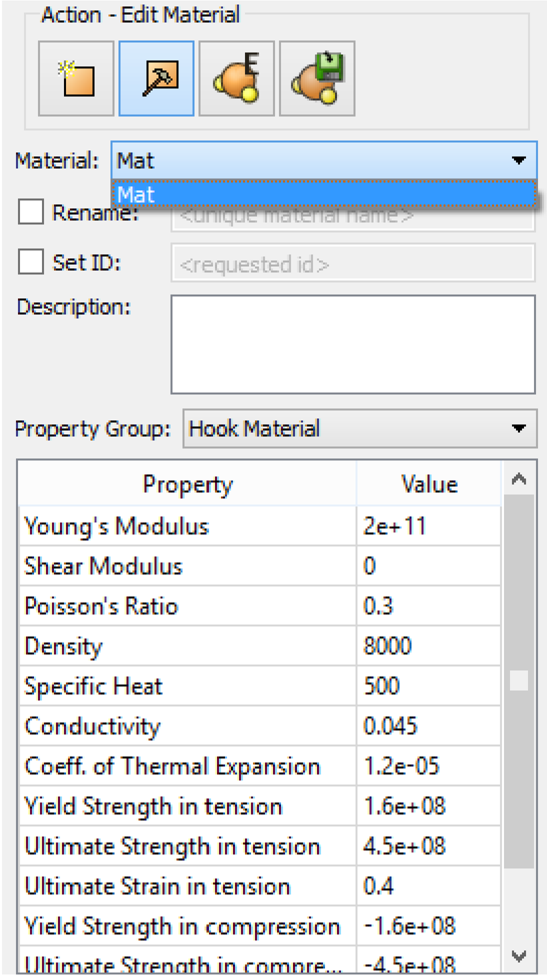
For this purpose, select the required material in the Model Tree and change the constants of the material properties page below. After the change, message on the material editing must appear on the command line. For example,

```
modify material 1 density 7800
```



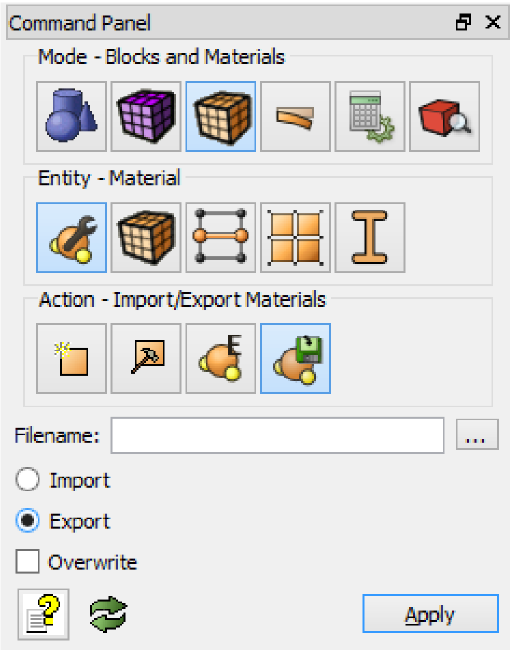


The material properties can be also changed by using Mode – **Blocks and materials**, Entity – **Material**, Action – **Change Material**).

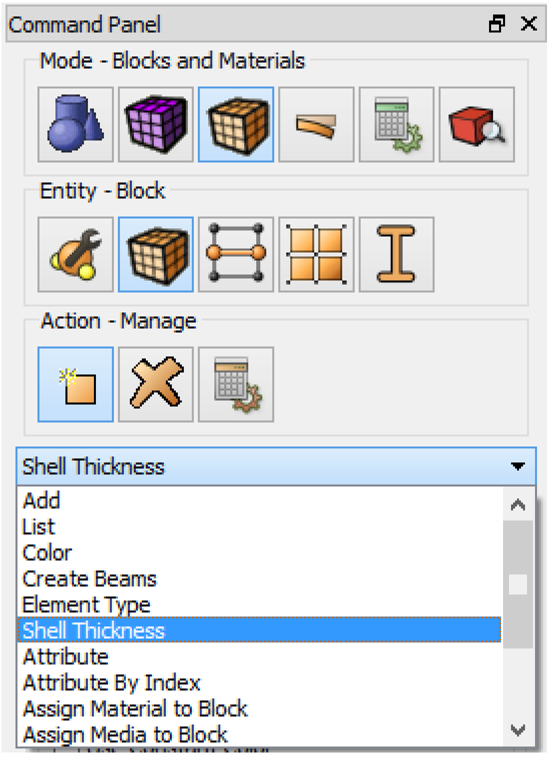


Import/Export Material

To import or to export the material, click on the section Import/Export Materials on the Command Panel (Mode – **Blocks and materials**, Entity – **Material**, Action – **Import/Export Materials**).



Moreover, ***Fidesys Bundle*** supports the import of the material in XML format.



Setting shell properties

Fidesys Bundle supports shell elements SHELL/SHELL4/SHELL8/SHELL9/TRISHELL/TRISHELL3/TRISHELL6.

To specify shell properties – thickness and loft factor – go to Mode – **Blocks and materials**, Entity – **Block**, Action – **Management.** Further in the drop-down list, select **Shell thickness.**

On the appeared panel, specify:

- Block(s) ID
- Shell thickness
- Loft Factor

Note: Loft Factor by default must be equal to 0.5.

3D shell cross section view is possible in the ***Fidesys Viewer*** postprocessor by clicking 3D-view button in the default string.

Setting beam properties

Fidesys Bundle supports beam elements BEAM/BEAM2.

To specify beam cross section using geometric features and moments of inertia, go to Mode – **Blocks and materials**, Entity – **Beam parameters.**

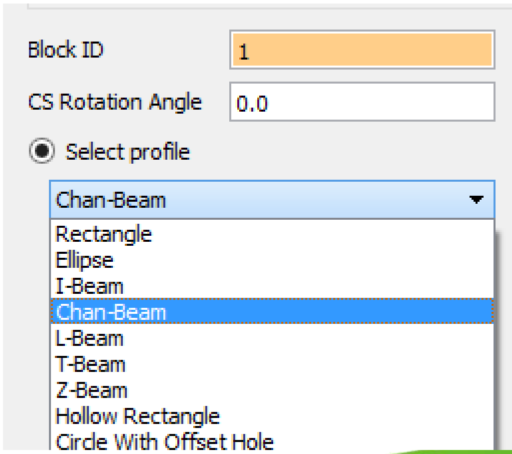
On the appeared Panel, specify:

- Block ID;
- Coordinate System Rotation Angle;
- cross section profile and the appropriate dimensions to it.

Click **Apply.**

Fidesys Bundle supports beam cross section of the following types:

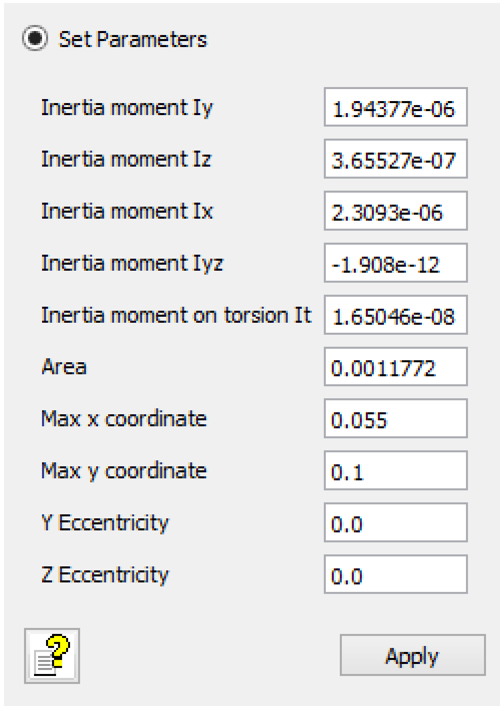
- Rectangle;
- Ellipse;
- I-Beam;
- Chan-Beam;
- T-Beam;
- Z-Beam;



- Hollow Rectangle;
- Circle With Offset Hole.

3D beam cross section view is possible in the ***Fidesys Viewer*** postprocessor by clicking 3D-view button in the default string after the calculation is complete.

To set the beam properties using inertia moments, change the checkbox **Cross Section Profile** to **Set parameters**. Then the fields for the inertia moments will be available for editing.

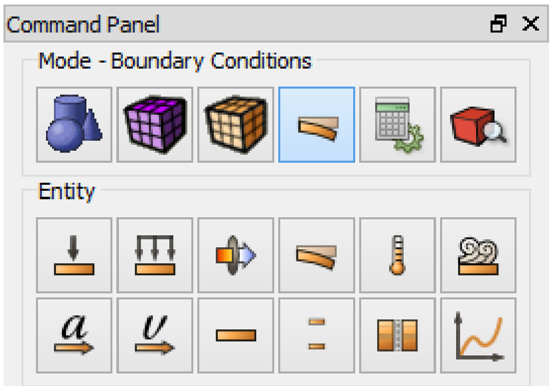


Setting boundary conditions

Types of boundary conditions

Fidesys Bundle supports boundary conditions of the following types:

- Point Force,
- Stress,
- Displacement,
- Heat Flux,
- Convection,
- Temperature,
- Acceleration,
- Speed,
- Contact Regions,
- Contact Pairs.



To set boundary conditions, follow these steps:

1. Select Mode— **Boundary conditions** on Command Panel.
2. Select Boundary Condition Type in **Entity** block.
3. Select Action — **Create**. Set the following parameters:
   - ID/Name (assign a new ID, enter a name using letters and/or numbers, or use the system assigned ID);
   - Entity where the boundary condition is applied (Volume, Surface, Curve, Edge, Vertex, Node, Nodeset, Element, Side, Sideset);
   - Entity ID(s) (point mouse cursor at the field Entity ID(s) and select the necessary entities with a mouse, their numbers will be entered into the field automatically. If you need to specify several entities, mark them holding down the Ctrl key);
   - Other parameters (Value, DOFs, etc.).
4. Click **Apply.**

Using the functionality available on Command Panel you can also see the list of Boundary Conditions, modify or delete the boundary condition you previously set.

Time/coordinate dependency

The time/coordinate dependency can be specified separately for each type of boundary conditions using tabular and formulaic dependencies.

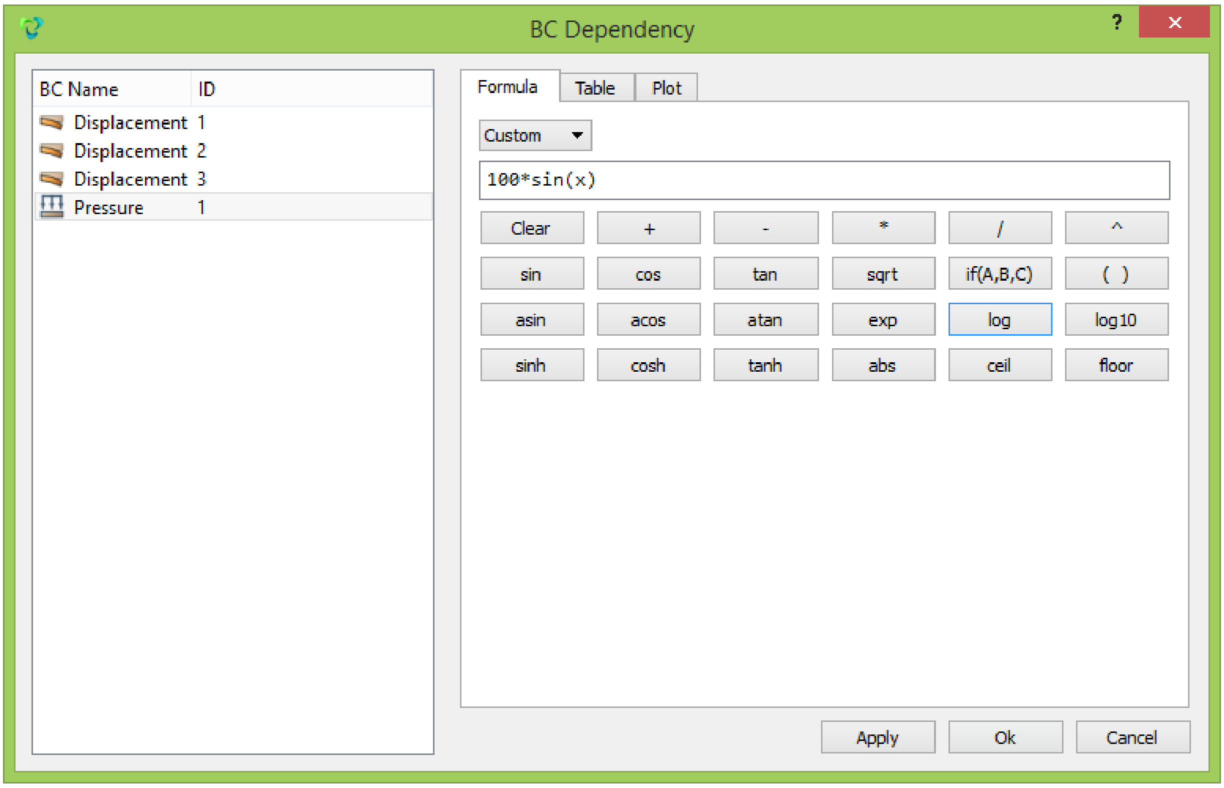
The boundary conditions must be set in advance (Mode – Boundary Conditions)..

To set the formulaic dependency on Command Panel, select the section of the boundary conditions (Mode - **Boundary Conditions**, Entity – **Time dependency**) and in the appeared form:

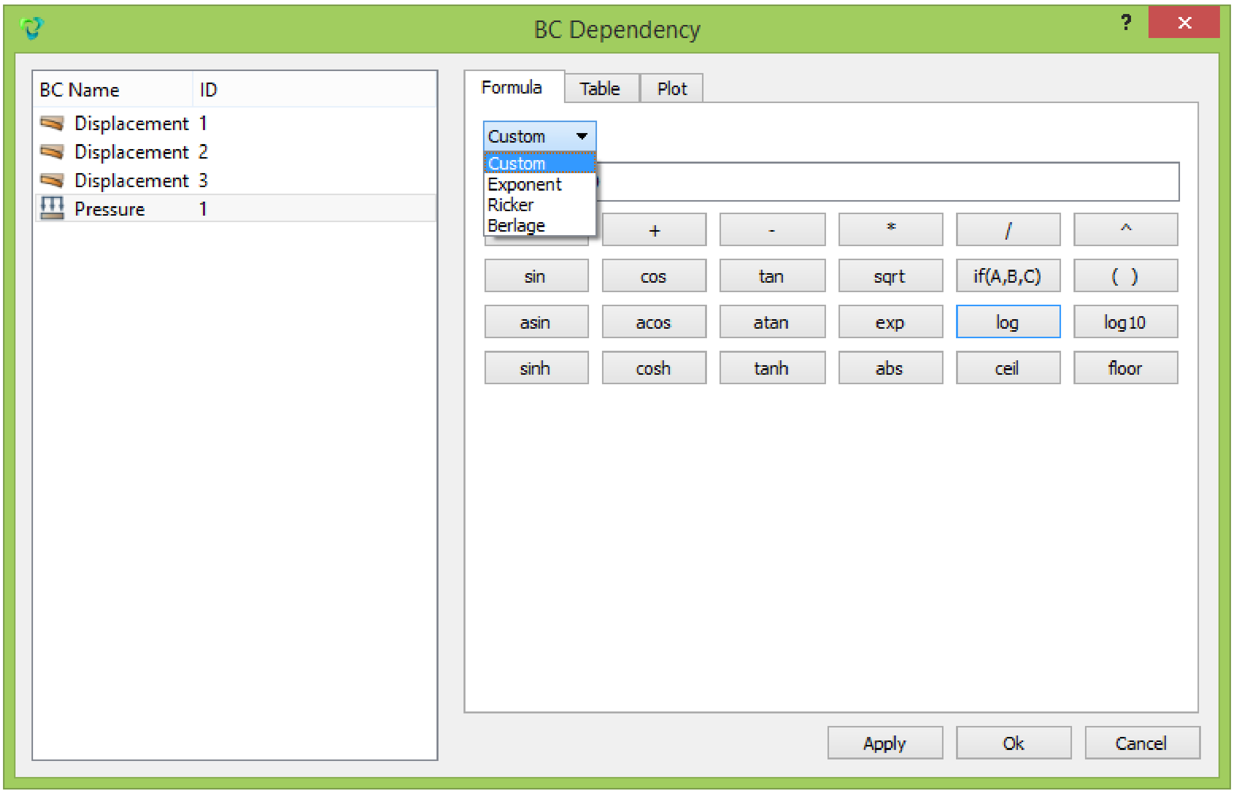
- Select BC Type;
- Select an individual component or an entire vector for time dependency application;
- Select Dependency Type (formula can be entered manually, you can use the standard formulae for the time dependency);
- Set Dependency Parameters.

Click **Apply.**

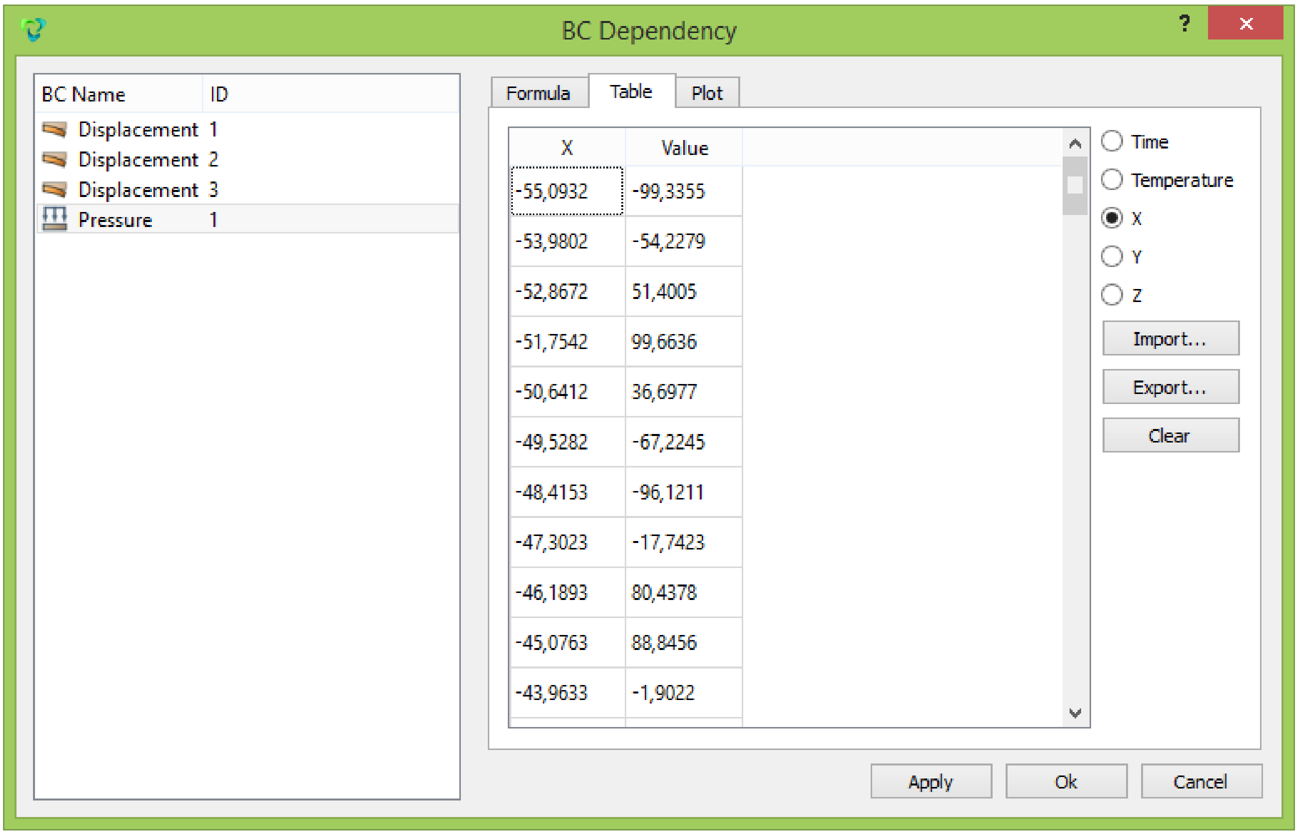
To view a tabular data or graphs plotted by a given formula, go to the corresponding tabs in the window BC Dependency. In addition, there is a possibility to export tabular data or to import new tables.



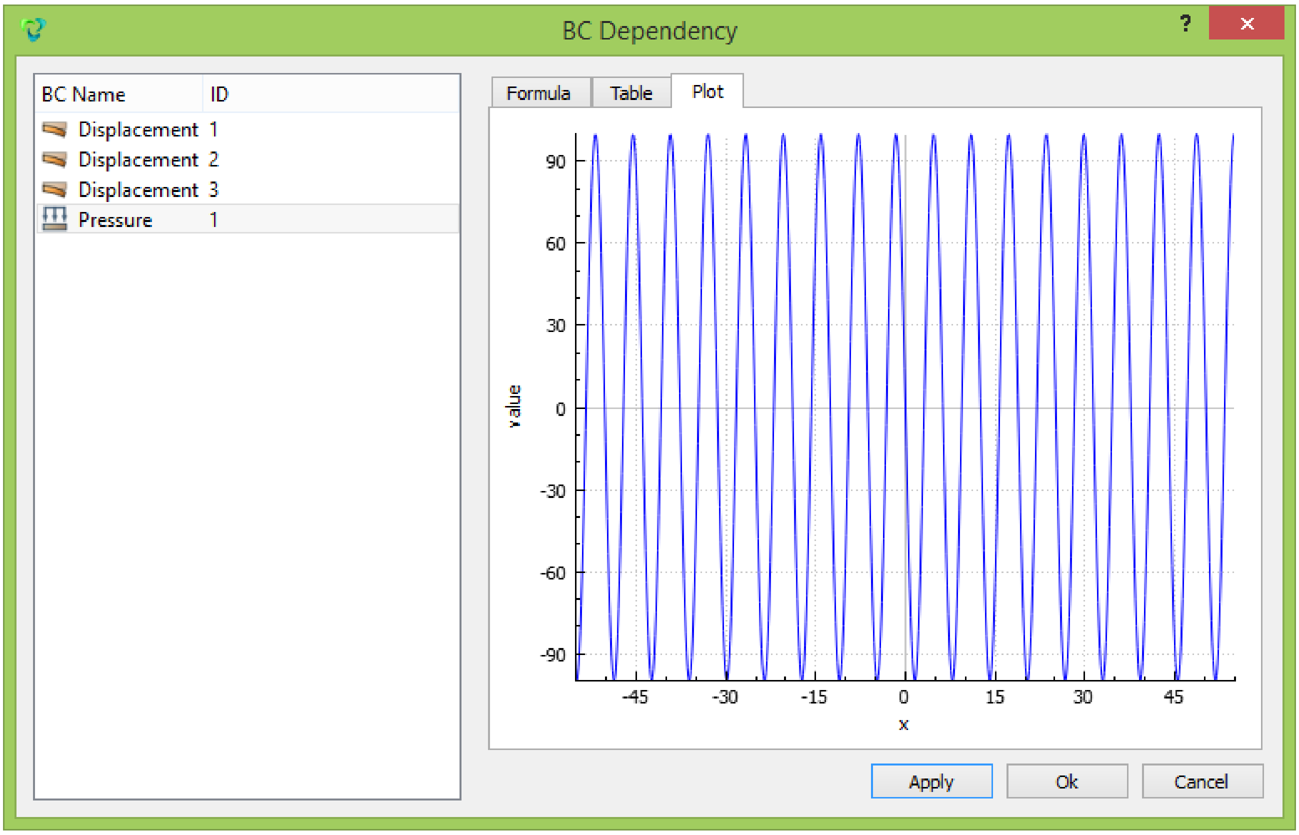
Here are standard formulae for the time dependency:



Viewing of the tabular data corresponding to the formula -100*sin(x):



Viewing of the graph corresponding to the formula -100*sin(x)

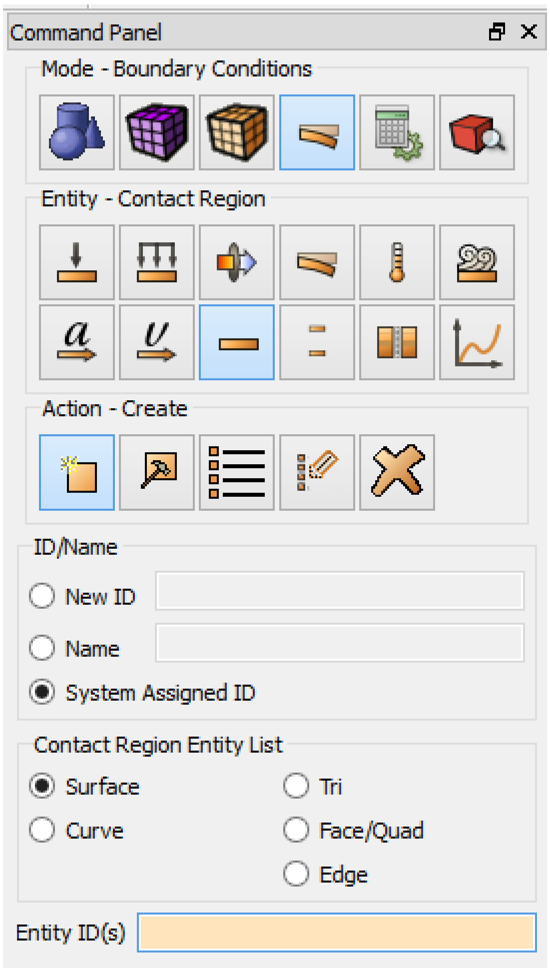


Setting contact interaction

Contact problems are highly nonlinear and require significant computer resources to be solved. Thus, to select the model resulting in the most effective solution, it is very important to understand the physical content of the problem. Two factors determine nonlinear nature of contact problems. Firstly, it is the contact area and consequently, the boundary conditions are unknown until the solutions. Secondly, it is necessary to take into account the friction in many contact problems. Effects related to the friction can result in poorly converging problems.

Contact region

To specify contact areas, choose the setting Contact area section (Mode – **Boundary Conditions**, Entity – **Contact region**).



The node-to-surface and node-to-curve contact interactions are realized in ***Fidesys Bundle 1.6***.

Note: if the contact conditions are not specified, parts in an assembly do not interact. The interaction of parts in the assembly at set contact areas is an obstacle to mutual penetration of parts and to the load transmission.

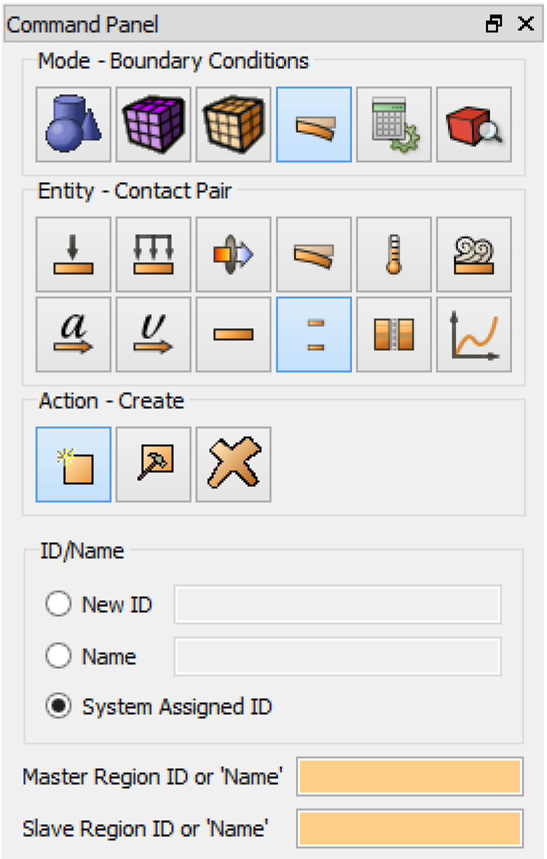
The areas for which it is planned to set the boundary conditions for the contact, should be allocated to individual surfaces for volume bodies or lines in the 2D case. Contact regions should be sufficient to make the bodies interaction process not come out their limits but it is recommended to minimize these regions to save computer resources.

Contact pair

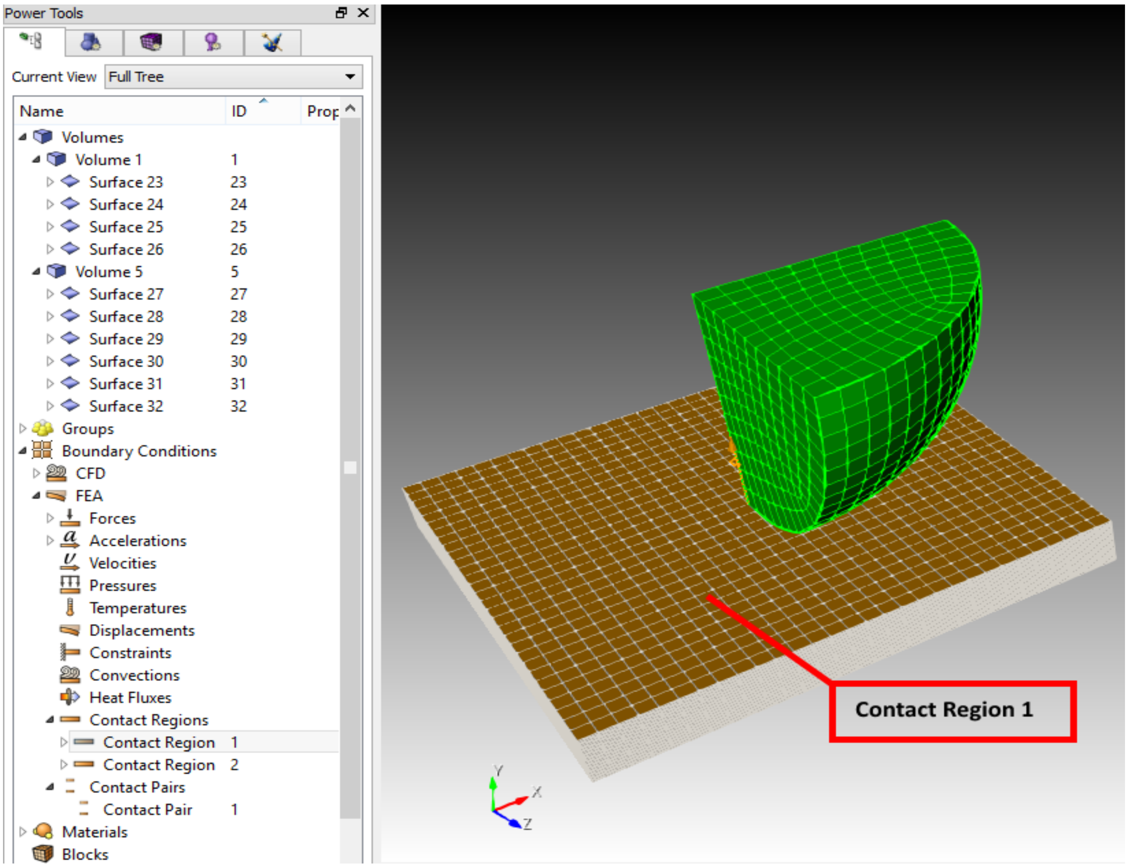
To set contact interaction, it is necessary to specify Contact pair (Mode — **Boundary conditions**, Entity — **Contact Pair**).

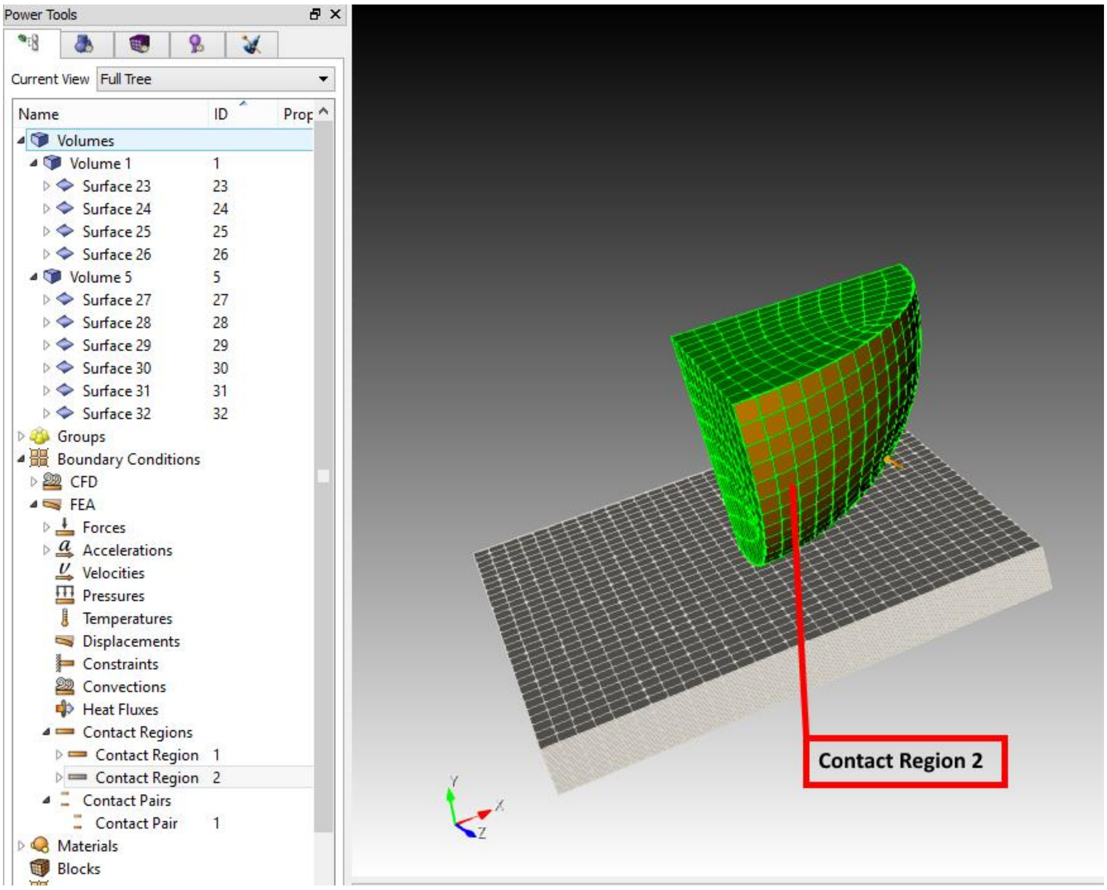
Please indicate the existing contact regions which will serve as the Master region and which will serve as the Slave region.

It should be noted that the Master region is modeled by surfaces and the Slave region – by nodes.

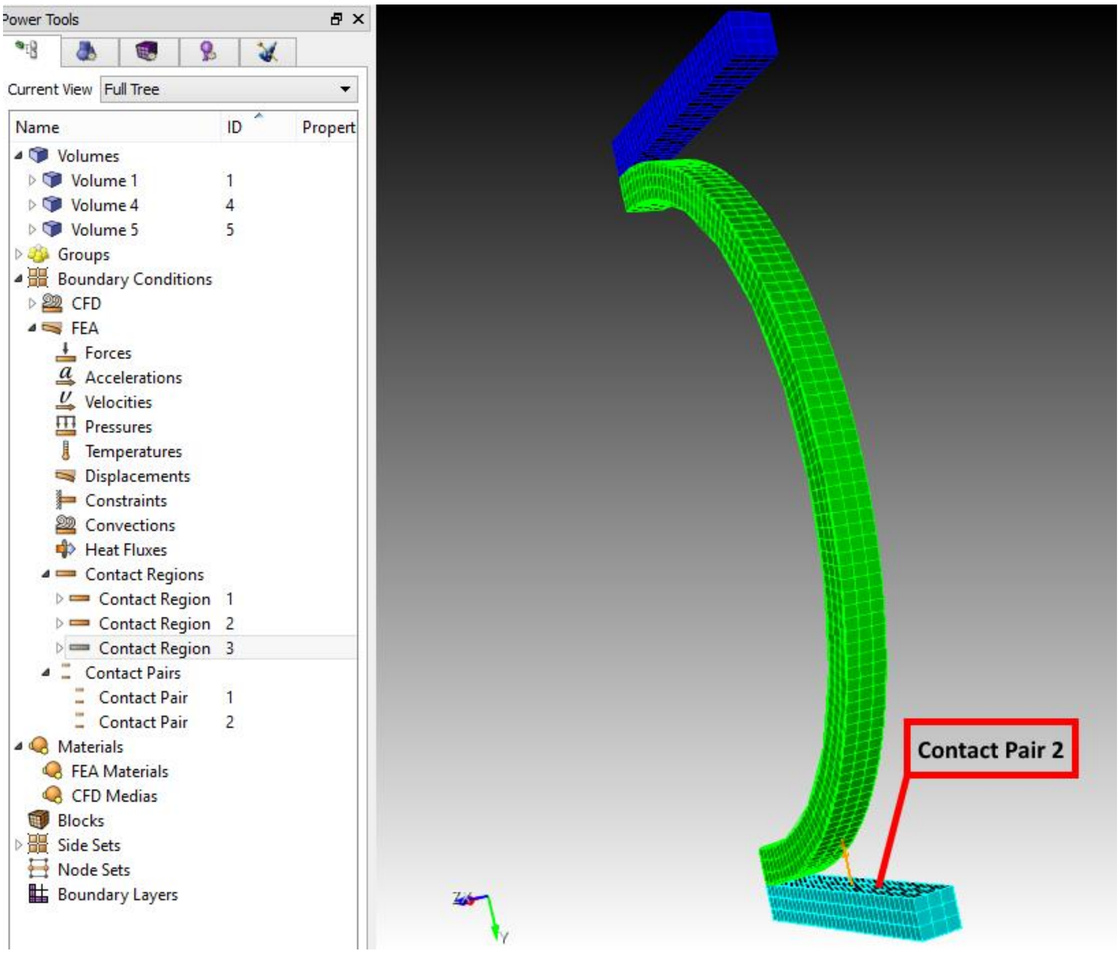


The contact regions and their numbers are displayed in the Model Tree on the left. For visualization, click on the name of the desired contact region in the Model Tree and it will be highlighted on the model.



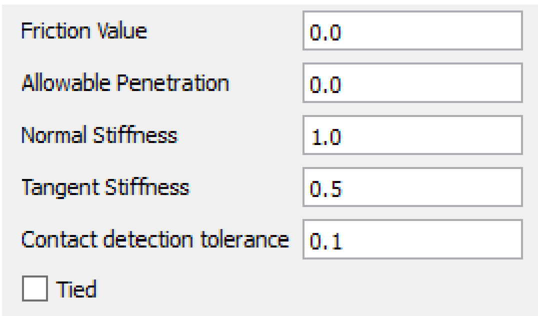


Individual number (ID) and a set of properties are ascribed to each contact pair. The number of contact pairs is unlimited. To visualize a created contact pair, click on the name of the required contact pair in the same Model Tree on the left. The selected pair will be highlighted in yellow on the model.



When you create a contact pair, please keep in mind that the choice of the Master and Slave regions may cause different modeling results and affect the solution tolerance.

In ***Fidesys Bundle 1.6***, the following contact pair settings are available:



To model the inseparable connection, please establish a tick **Tied,** then the Master and Slave regions engage in all directions until the end of the analysis if the contact has been set.

If the motion of a solid body is limited only by the contact conditions, it is important to ensure the interaction elements of the contact pair in the initial state. However, in some cases, it may be difficult to identify the interaction. This may occur in the following cases:

- the contours of the body can be quite complex and it is difficult to accurately determine the point at which the first contact happens;

- despite the fact that the geometric model is constructed without discontinuities, arising when meshing rounding errors can lead to the appearance of small gaps between the elements.

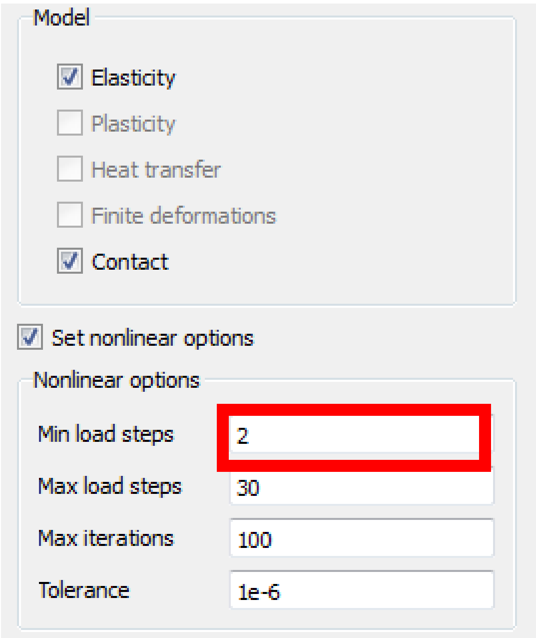
For similar reasons, too high initial penetration of the Master region into the Slave region may occur. In these cases, it may cause excessively large reactive forces in the contact elements which may **result in the divergence of solutions.** Therefore, the definition of the initial contact represents perhaps the most important meshing aspect for the contact analysis.

Contact algorithm

In ***Fidesys Bundle 1.6*** the contact algorithm Penalty Method is applied.

This method requires settings for both normal and tangential stiffnesses (see Contact pair settings). The main disadvantage of this method is that the penetration between the two surfaces depends on these stiffnesses. Higher stiffness values can reduce the penetration but can lead to ill-conditioning of the global stiffness matrix and to the poor convergence. Ideally, sufficiently high stiffnesses must be chosen for the contact penetration to stay sufficiently small. At the same time, sufficiently low stiffnesses provide better convergence of the problem.

In addition, when you run the task for the calculation, it is recommended to use at least 2 steps of loading in Nonlinear Solver Settings.



Element Types

Fidesys Bundle 1.6 computational algorithms allow simulating the contact without specifying an exact match of mesh nodes on the boundary. It does not require using any special finite elements in the contact area to describe the interaction details. In the process, it **is not required** to use any special finite elements in the contact area to describe the interaction details. Such approach makes it easy to set the conditions for interaction in contact or for tied surfaces.

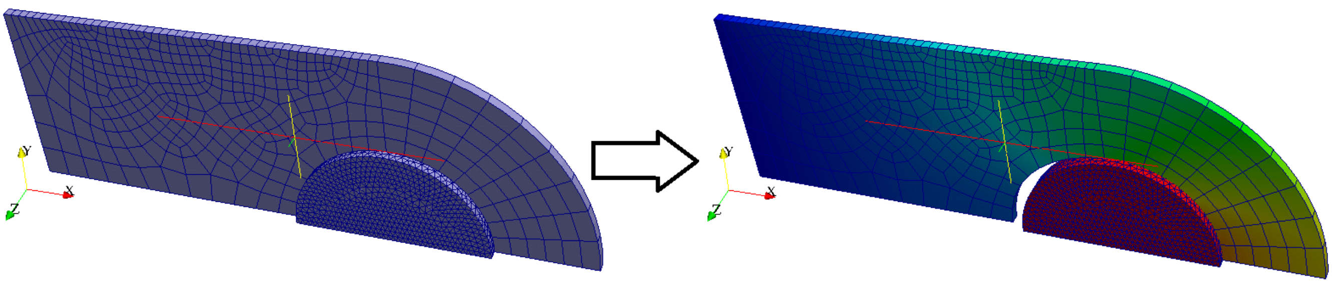
Fidesys Bundle 1.6 supports a solution of contact problems for the following already existing finite elements:

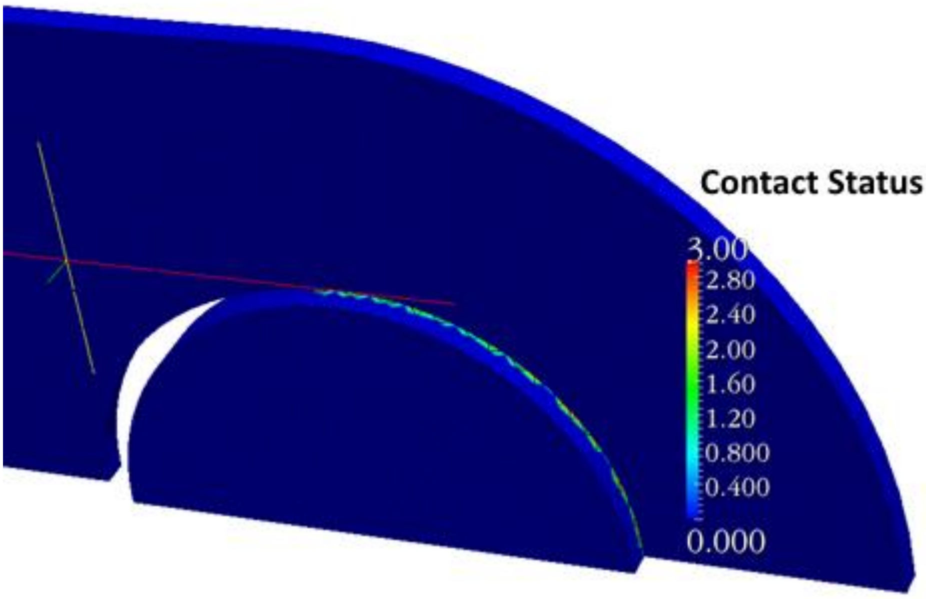
- 1st and 2nd order quadrilateral 2D elements QUAD/QUAD4/QUAD8/QUAD9;
- 1st and 2nd order triangular 2D elements TRI/TRI3/TRI6;
- 1st and 2nd order 3D hexahedron HEX/HEX8/HEX27;
- 1st and 2nd order 3D tetrahedron TETRA/TETRA4/TETRA10;
- 1st order 3D elements to construct hybrid meshes (pyramid PYRAMID/PYRAMID5 and wedge WEDGE/WEDGE6)

Contact Status

In ***Fidesys Viewer*** directly after the calculation the behavior of each contact element can be evaluated by the assigned status in the field **Contact Status**.
This field has one component that can assume the following values:

- STATUS = 0 – far;
- STATUS = 2 – contact with slip;
- STATUS = 3 – without slip (or without friction).



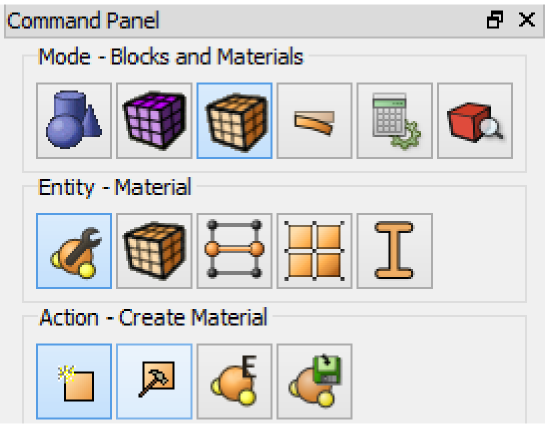


Setting the yielding model

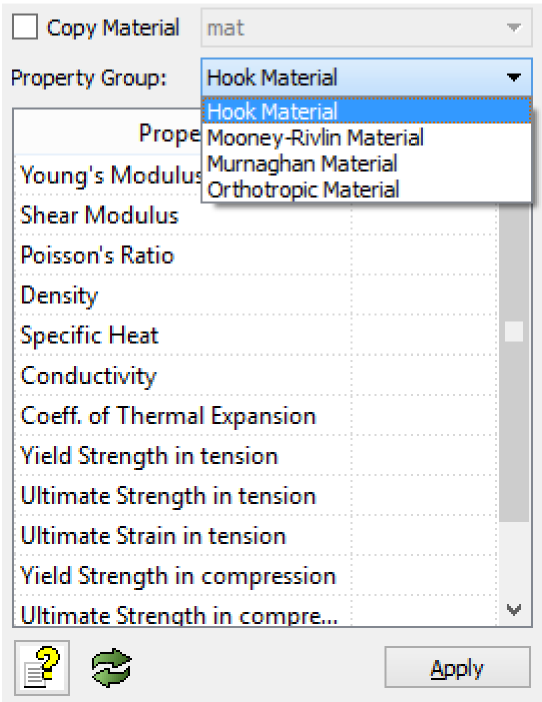
The choice of the correct model of the material plastic flux is very important to obtain a proper solution of the problem. Plasticity problems are nonlinear, therefore, they require substantial computer resources and solving problems with large plastic strains may take a long time. The Fidesys system of strength analysis for the Hook material realizes two criteria of transition into plasticity: the Mises criterion and the Drucker-Prager criterion. Problems are solved both for perfectly elastoplastic models and for models with linear hardening. It is currently implemented the approach taking into account finite strains in the elastic zone; the linear formulation of the problem is used in the zone of plastic flux.

Von Mises yield criterion

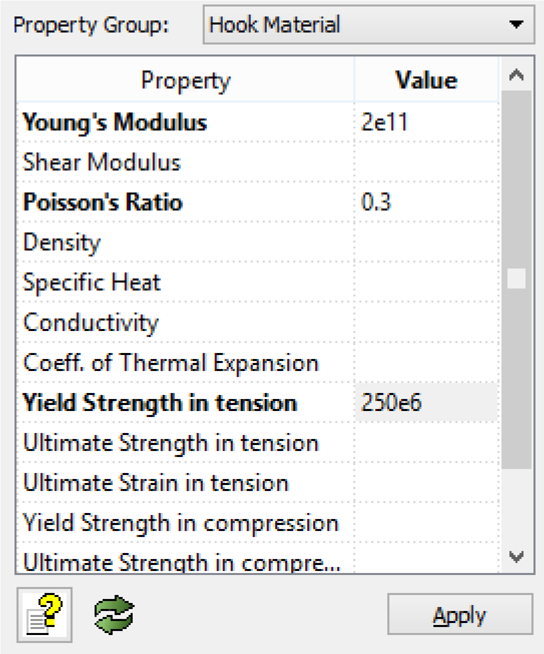
In order to add the Mises plasticity to the Hook material, please select the section for setting material properties on the Command Panel (Mode – **Blocks and materials,** Entity – **Material,** Action – **Create material**).



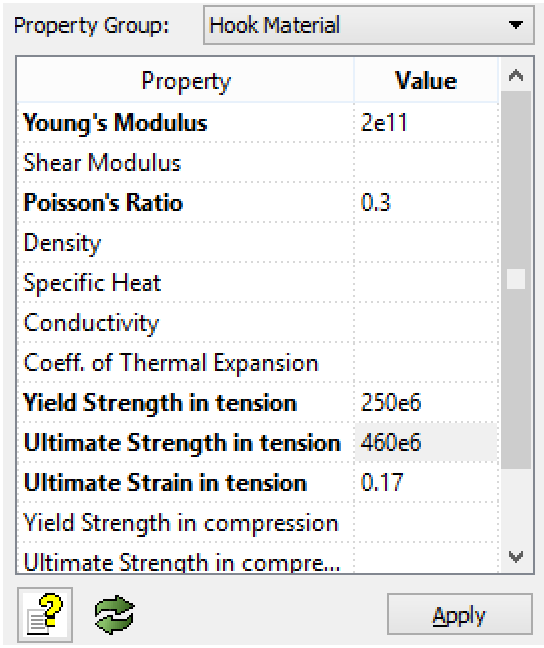
Choose the group of material properties from the drop-down list, specify the name of the material and the corresponding constants:



To create the model with the von Mises plasticity without hardening, set elastic properties of the Hook material as well as the yield strength in tension:

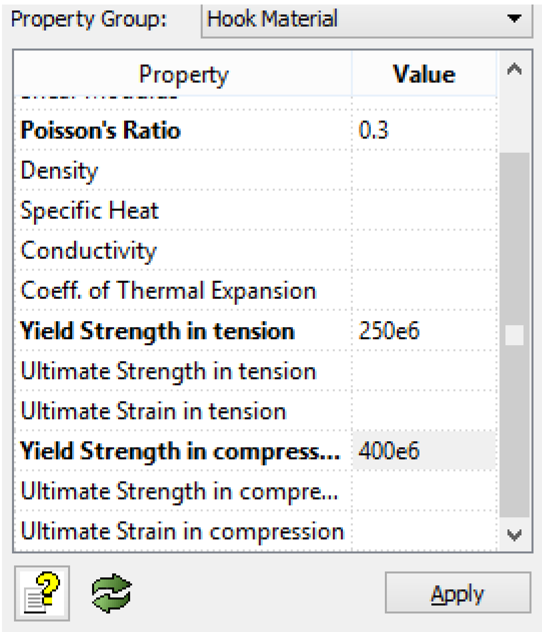


To create the Mises plasticity model with linear hardening, it is also necessary to enter the yield strength in tension and the ultimate strain in tension.

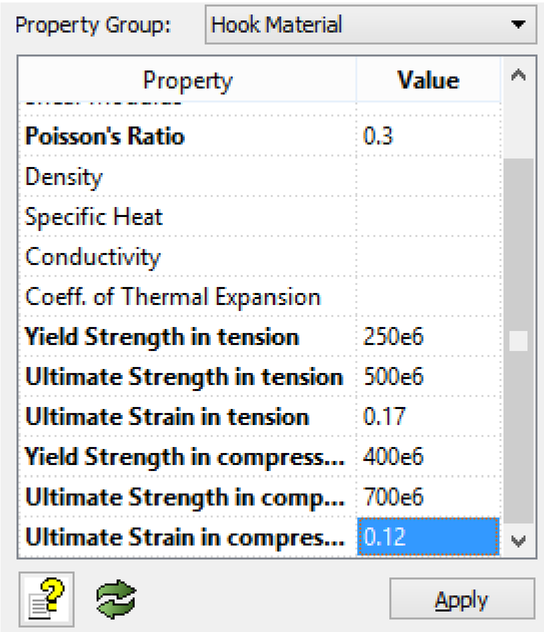


Drucker-Prager yield criterion

To set Drucker-Prager plastic model without hardening, it is necessary in addition to the elastic properties of the material to determine the yield strength in tension and yield strength in compression.



In order to create Drucker-Prager plasticity model with hardening, specify also the yield strength and ultimate strain in tension and ultimate strain in compression.



To operate with the Drucker-Prager plasticity model, it is necessary to enable the option of the ultimate strain when starting calculation.

Element types

Fidesys Bundle 1.6 supports the solution of elastoplastic problems for the following types of already existing finite elements:

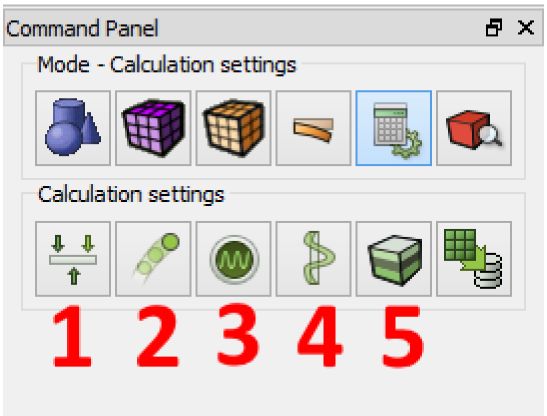
- 1st and 2nd order quadrilateral 2D elements QUAD/QUAD4/QUAD8/QUAD9;
- 1st and 2nd order triangular 2D elements TRI/TRI3/TRI6;
- 1st and 2nd order 3D hexahedron HEX/HEX8/HEX27;
- 1st and 2nd order 3D tetrahedron TETRA/TETRA4/TETRA10;
- 1st order 3D elements to construct hybrid meshes (pyramid PYRAMID/PYRAMID5 and wedge WEDGE/WEDGE6)

Starting calculation

Analysis types

Fidesys Bundle includes the following types of analysis:

- Static (1);
- Dynamic (transient) (2);
- Modal (3);
- Buckling (4);
- Composite materials effective properties calculation (5).



When starting calculation follow these steps:

1. Select Mode – **Calculation settings** on Command Panel.
2. Select the necessary type of analysis: Static, Dynamic, Modal, or Effective properties analysis.
3. Set the method of numerical analysis: Finite element method (by default) or Spectral element method (see the section **Spectral Element Method**).
4. Set the parameters of the type of analysis you chose: solver type, coordinate system, fields, scheme, time settings (for dynamic analysis), etc.
5. Click **Apply**.
6. Click **Start Calculation.**

After starting calculation its process will be displayed in the console. It will also output the messages for the user, including the errors in case of unsuccessful or incorrect finishing of the calculation. If the calculation is finished successfully, you will see the following message in the console: "*Calculation finished successfully at <date> <time>*".

All the calculations are made in Cartesian coordinate system by default. If necessary, you can also convert the results into cylindrical and spherical coordinate systems (use the appropriate filters in ***Fidesys Viewer***).

The dimension of the calculated problem is 2D or 3D. The following types of 2D problem are included:

- Plane stress;
- Plane strain.

Stress, strain and displacement fields are calculated by default. If necessary, you can also calculate principal stresses, strains, and Mises stress intensity (use the appropriate filters in ***Fidesys Viewer***).

The following types of solvers of linear systems (systems of linear algebraic equations (SLAE)) appearing while discretizing the problem, are available:

- Direct (LU)
- Iterative.

The following solvers for problems of modal analysis at systems of linear algebraic equations (SLAE) are available:

- Krylov- Schur;
- Arnoldi.

For dynamic load, one of the two calculation schemes can be used:

- Explicit
- Implicit.

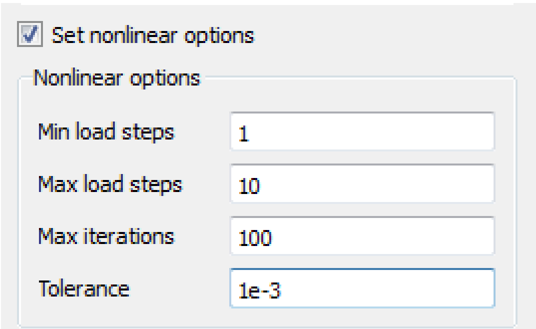
Models of problems

For the calculation, the following models of problems are available:

- elasticity;
- plasticity (for static analysis);
- thermal conductivity;
- ultimate strains (for static and dynamic analysis);
- contact.

To choose a model, the user selects the appropriate checkboxes. Selecting multiple checkboxes simultaneously allows setting various combinations of models. For example, the selection of the checkboxes Elasticity and Elasticity gives an elastoplastic model and the selection of the checkboxes Elasticity and Thermal conductivity gives a model of thermoelasticity..

To improve the convergence of nonlinear problems, use the following settings:



For nonlinear problems, convergence of iterations at each loading step can be checked in the file Convergence.txt. The file is downloaded into the folder that is created next to the file * .pvd which stores the calculation.

For effective performance of several calculations you can use the Calculations manager (see the section **Calculations manager**).

For visualization and analysis of the obtained results you can use the program ***Fidesys Viewer*** included into the package.

Calculations manager

Fidesys Bundle allows performing individual calculations (see the section ***Starting calculation***) as well as a cycle of calculations within one project.

To perform a cycle of calculations, create a new project. You can do this in one of the following ways:

- Choose **File** → **New Project** in the menu
- Click a button on Calculations manager toolbar
- Use the combination **Ctrl**+**Shift**+**N**

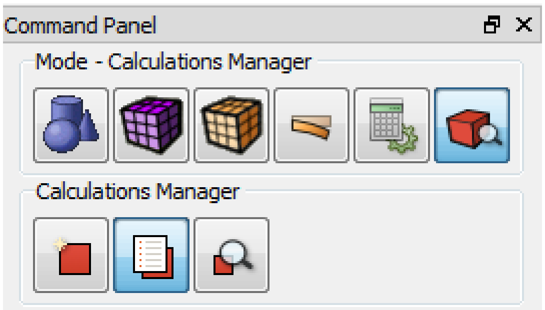
In a pop-up window select a name of the project, its location and a type of geometry generation for the model a cycle of calculations will be based on:

- If the model is imported (see the section ***Geometry generation***), select this model in the same pop-up window.
- If the model is created in the program with the use of geometric elements, select *Use current model*.

After creating the model, generate a mesh, set boundary conditions and material, set the required calculation settings (see the sections ***Meshing***, ***Setting boundary conditions***, ***Setting the material***, ***Starting calculation*** respectively), and start the calculation. In a pop-up window enter the name of the calculation (or leave the suggested by default), comments and settings of start and results visualization of the calculation. *(**Name** is the required field. The other settings are optional.)* Click **Add**. The calculation will be automatically added into the list of calculations within this project (in spite of its successful or unsuccessful performance).

To perform another calculation, modify some of the settings of the previous one (e.g. boundary conditions) and start the calculation. It will be automatically added into the list of calculations within this project again (in spite of its successful or unsuccessful performance).

To see your list of calculations, select the button **Calculations Manager** on Command Panel in the mode **Calculations Manager** and click **Open Calculation Manager.**



In a pop-up window you will see the list of calculations with their results (successful or unsuccessful finishing) and calculation settings (analysis, solver, fields for the output, etc.).

You can save all the calculations in one of the following ways:

- Choose **File** → **Save Project** in the menu
- Click a button on Calculations manager toolbar
- Use the combination **Ctrl+Shift+S.**

To open the saved project, you can also use one of the three ways:

- Choose **File** → **Open Project** in the menu;
- Click a button on Calculations manager toolbar;
- Use the combination **Ctrl**+**Shift**+**O.**

When re-opening the saved project, all of the problem settings will be kept the same as they were in the last calculation you made before saving it within this project.

Only one project can be opened at the same time.

To close the project, you can also use one of the three ways:

- Choose **File** → **Close Project** in the menu
- Click a button on Calculations manager toolbar
- Use the combination **Ctrl+Shift+C**

You will be suggested to save the project so as not to lose any data.

Spectral element method

It is a unique feature of ***Fidesys Bundle*** that, in addition to the of finite element method (FEM) used by default, it enables calculations by spectral element method (SEM).

SEM brief description and advantages

Spectral element method (SEM) is a FEM modification where piecewise functions are used as basic functions consisting of high degree polynomials.

The main advantages of SEM in comparison to FEM:

1) High computational speed as there is no need to solve the system of linear algebraic equations due to diagonal form of mass matrix. The latter is obtained by specific quadrature formula for volume integration.

2) High precision of solution approximation at coarse meshing (low number of elements). The solution error is estimated as

$$\|[u]_h - u_h\| \leq C(N),$$

where

$$C(N) = C_2 h^N \text{ for FEM}$$

and

$$C(N) = C_1 h^N e^{-N} \text{ for SEM.}$$

C_1 *and* C_2 are constants, h is a characteristic element size, N is an element order, u_h is a numerical solution, $[u]_h$ is an exact solution in mesh nodes.

3) Ability of effective paralleling for OpenMP, MPI and CUDA.

SEM is most effective for the dynamic analysis using an explicit time scheme.

Here are the results of classical problem of wave propagation in 2D plate (size 1x1).

To achieve the computational error 2% and less, it is necessary to generate one of the following meshes:

a) 3-noded triangular mesh of 6 390 197 elements (characteristic element size is 4e-4)

b) 4-noded quadrilateral mesh of 1 640 961 elements (characteristic element size is 3e-3)

c) coarse spectral element mesh with 4th element order (only 16 elements are required).

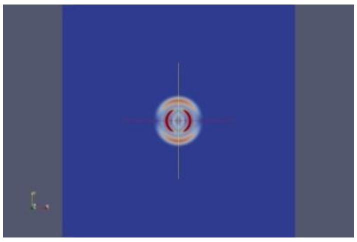


Fig.1. Displacements U at t_1

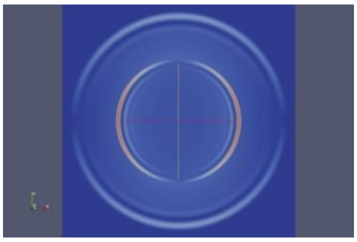
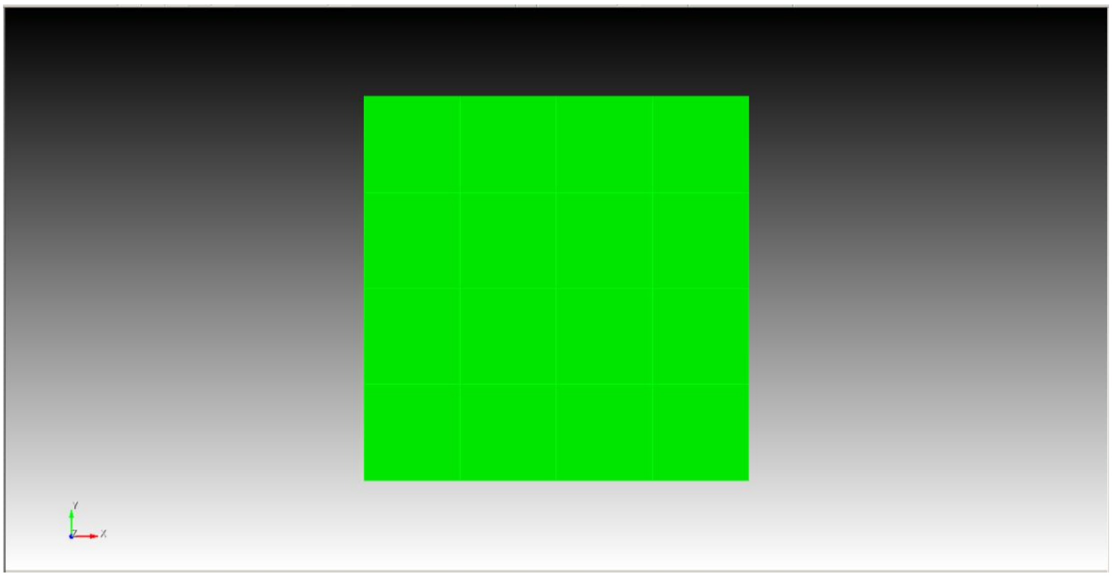
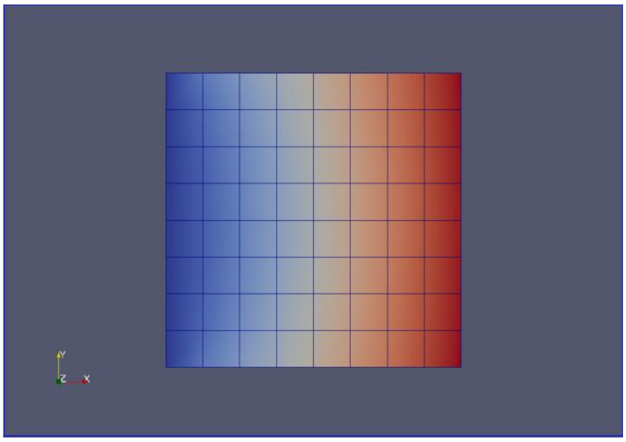
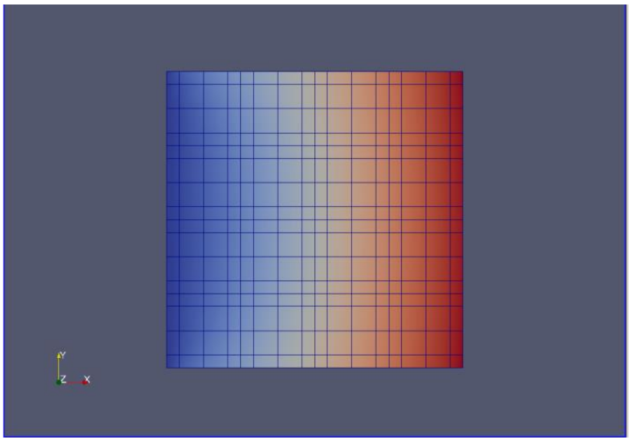
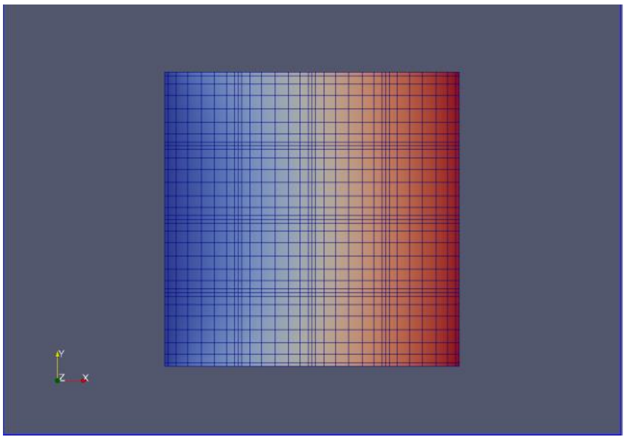
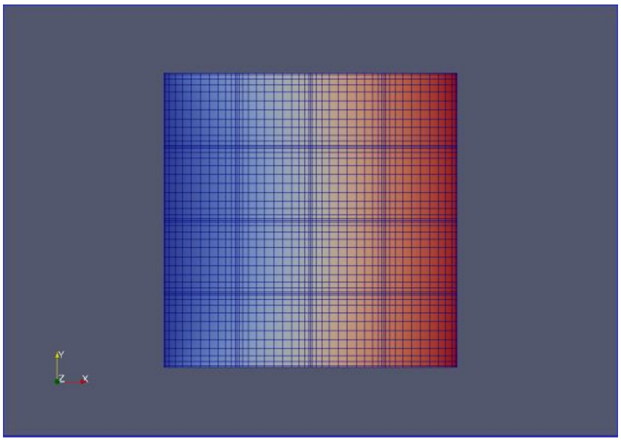


Fig.2. Displacements U at t_2

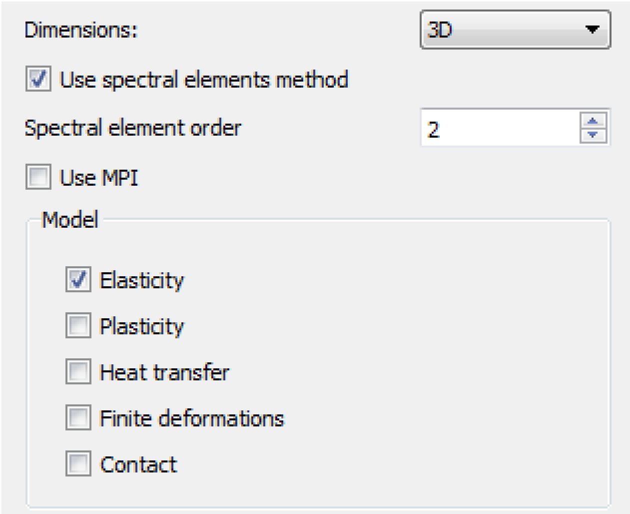
Here are the examples of computation results for different spectral element orders:



 2nd spectral element order	 4th spectral element order
 8th spectral element order	 12th spectral element order

SEM Usage

In order to use the method of spectral elements instead of the finite element method to solve the problem, select the checkbox **Use spectral elements method** in the general settings:



Parallel calculations on several computers using MPI technology

If you have a network of several computers with installed ***CAE Fidesys*** software MPI technology allows you to combine their computing capacity for parallel solution of the same problem.

MPI brief description and advantages

MPI technology currently represents a standard for parallel computing in distributed memory systems, i.e. those in which each processor has its own independent address space and communicates with the other processors via messages. MPI technology is more effective in solving problems with a large number of degrees of freedom because, on the one hand, it allows solving problems that do not fit in the computer memory and, on the other hand, large FEM or SEM problems require relatively low intensity of the messages exchange between the processors and thus they less load the network connection. This is particularly important for systems with distributed memory in which processors are connected by the common network with a capacity of 100 Mbit/s as if several computers in the office.

MPI implementation in Fidesys

Fidesys provides the ability to use MPI with the following types of calculations:

- Statics;
- Dynamics;
- Modal;
- Buckling.

Here are supported models to calculate via MPI:

- Elasticity;
- Elastoplasticity;
- Thermal conductivity;
- Thermoelasticity;
- Finite deformations.

MPI installation

Intel MPI installs and runs in conjunction with the installation of the ***Fidesys Bundle*** software package**.** If the Intel MPI version has already been installed on your computer and you do not want to replace it, please contact **Fidesys** customer support for instructions on how to install and configure it.

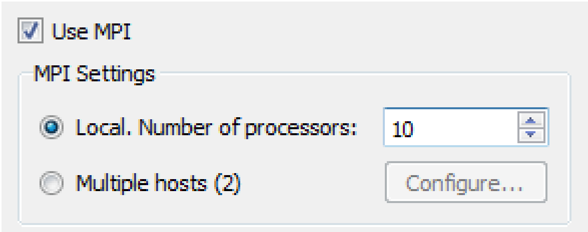
In order to use when calculating the MPInecessary to put a tick **Use MPI** in the Toolbar in the General settings of the selected calculation type. You will then see a special menu **MPI Settings** to specify needed parameters.

In the pop-up MPI settings, select parallelization mode:

a. Local – the calculation will be carried out on the local machine using a specified number of processors. The mode gives a gain in comparison to the calculations without MPI only for the local configuration with a large number of cores.
b. Multiple hosts. In this mode, the calculation will be launched on several computers.

MPI local usage

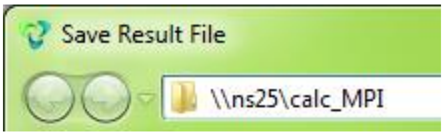
In order to use MPI locally on a single computer, you need to register at first (see below). Then go to the MPI Settings Panel, tick **Local** and select the number of processors in a special window. After this, you can start the calculation, no additional settings for MPI local use is not required.



MPI usage on several nodes

Requirements for the correct operation

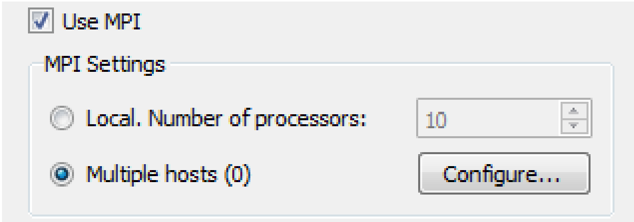
1. Make sure that the firewall settings on all computers allow correct operation of MPI.
2. It is recommended to disable the firewall on all computers involved in the parallel calculations.
3. Fidesys should be installed on the same path on all computers. This path **cannot** be network.
4. The path to FidesysCalc must be the same on all computers involved in the parallel calculations.
5. The working directory (the directory where the file .pvd and file folder of the calculation results are written) should be available at all nodes on the same path which **can** be network. The user on whose behalf the calculation is carried out must also have write access to the work directory in all nodes. To find out which way is the working directory, you can go to the Menu **Tools** → **Settings** → **Paths**, the string **Working Directory**. In other words, the calculation should be stored in a network folder, while the network path should be indicated in the save dialog:



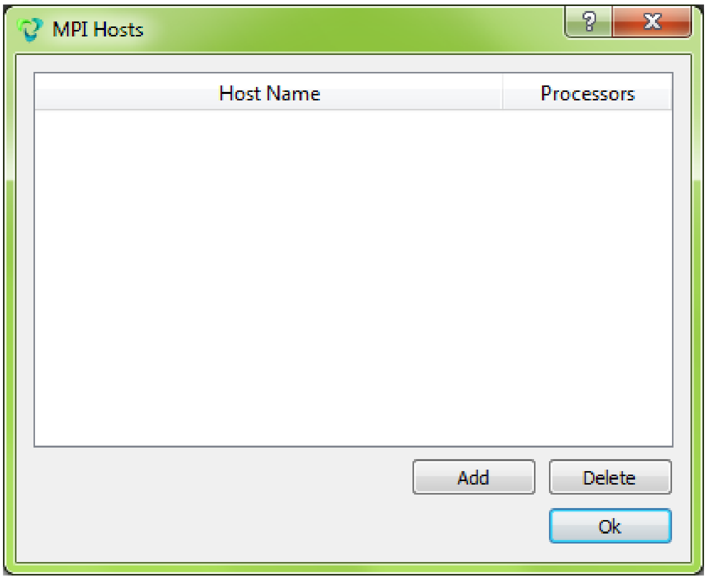
6. There are no special restrictions on the connection speed between the nodes but you should keep in mind that if the connection speed is very slow, the calculation using the MPI can take as much or more time than the calculation without MPI as all the time saved will be spent on the data exchange between nodes.
7. This software version has no limit on the number of used nodes.

MPI setting on several nodes

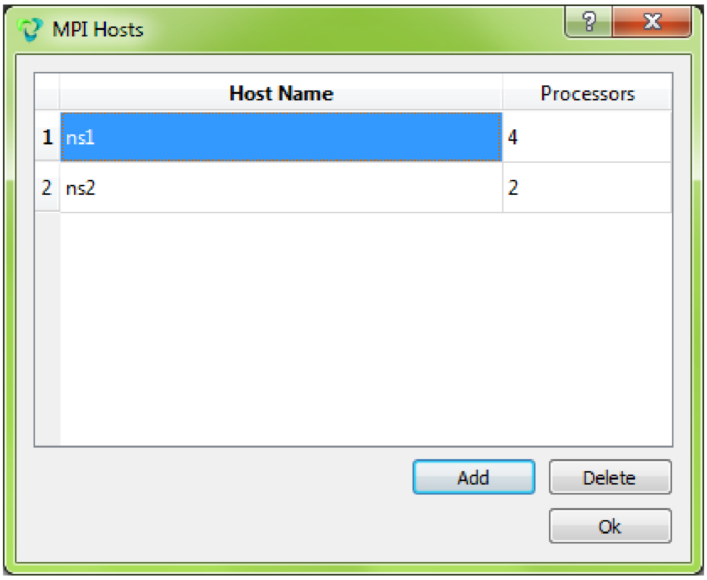
After making sure that all of the above requirements are met, go to the MPI settings panel (**Calculation Settings – Static – General – Use MPI)**. Put a checkbox next to the point **Multiple hosts** and click **Configure…**:



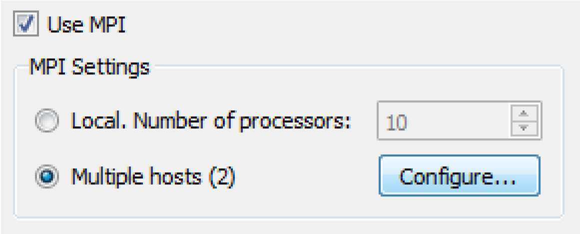
You will see the following window:



Using **Add** and **Delete**, add to the list all the hosts you use, in the field Name write the host name in the network, in the field Processors indicate the number of processors used on the host. After completing the list, click **Ok**.



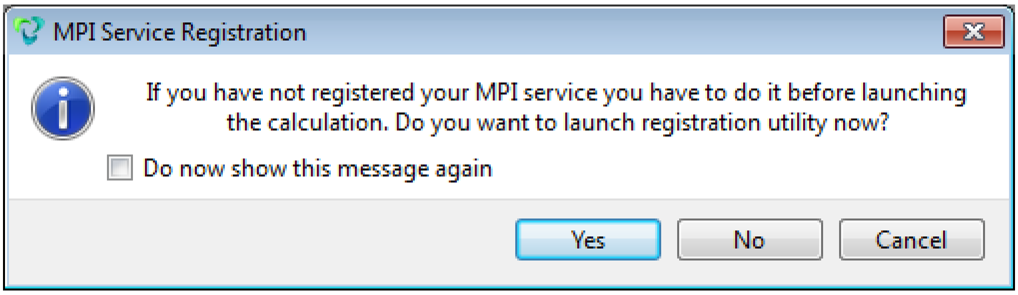
After this, the number of hosts indicated in parentheses after the words Multiple hosts on the MPI settings panel should change:



Now you can specify other calculation settings and run it as usual; it will be carried out using the MPI on several nodes.

Registration before the first usage

If you try to carry out the calculation using MPI for the first time, you should see the following window:



In order to register (without this step, the calculation is impossible), click **Yes**. You will see Windows terminal window which you will need to enter Windows username and password on behalf of which you launch the calculation using MPI.

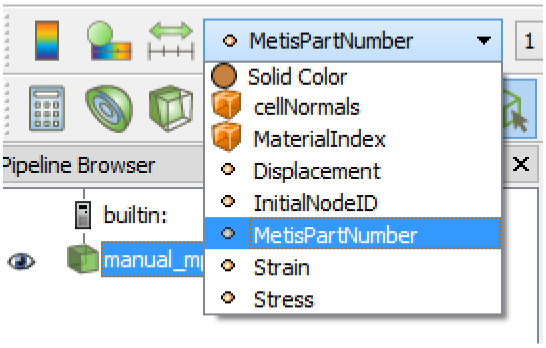
You can also register by running the Windows terminal window from the panel "Start" (to do this, type in the search box «cmd») and by typing the command **mpiexec –register** in the window. Then you need to enter login and password in the same way as when registering using a pop-up Fidesys window.

If you have already registered the service, tick **Do not show this message again.**

For more information, see the ***Intel MPI*** *documentation.*

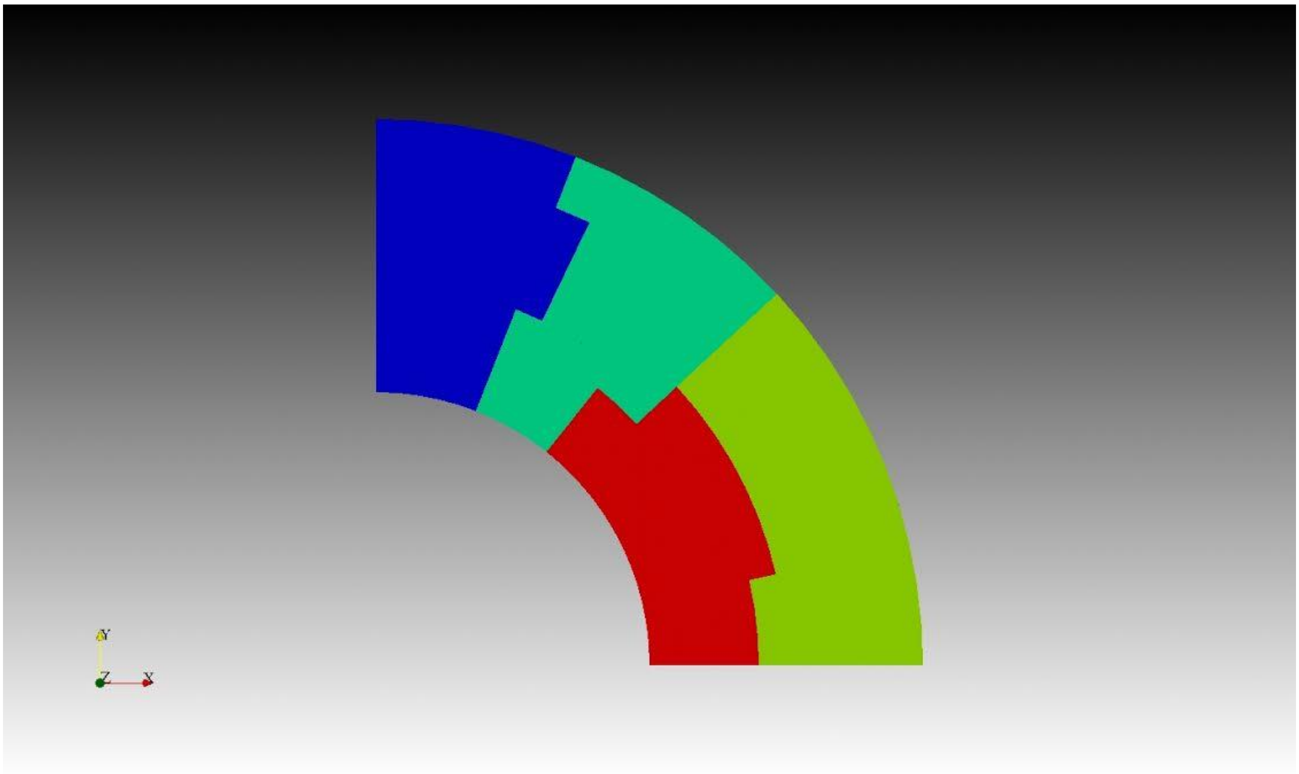
Overview of the calculation results

After performing the calculation using MPI, in the ***Fidesys Viewer*** postprocessor the new field **MPI Nodes** should appear that characterizes a partition on the specified earlier processors:



Calculation example using MPI

You can see an example of calculation on two computers in the picture below. Parts that are calculated on various computers are presented in different colors.



Heterogeneous materials effective property calculation

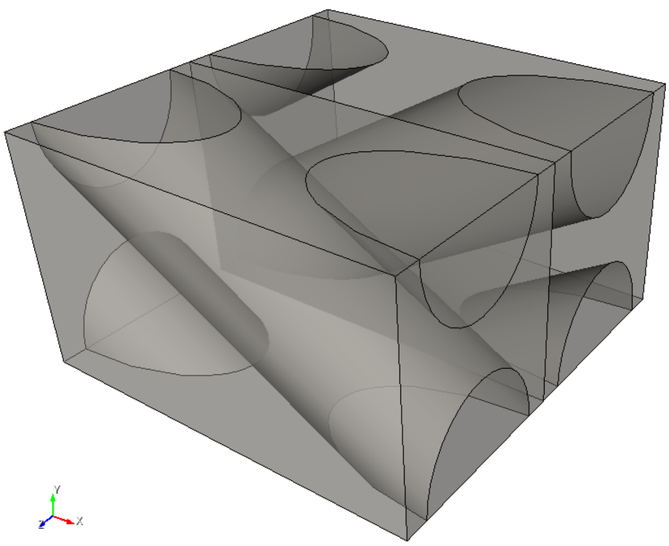
In ***Fidesys Bundle*** there is the possibility of calculating the effective properties of an heterogeneous material, for example, composite or porous material. The calculation is carried out **only in 3D case**.

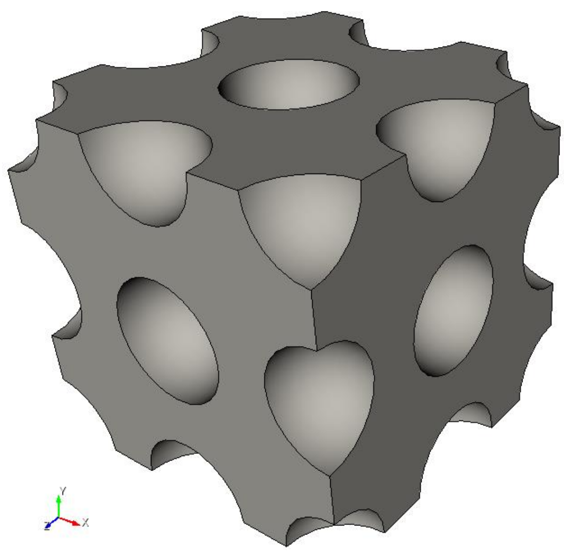
Geometry of the model for effective property calculation

A representative volume is a geometric model for calculating the effective properties of the material of nonperiodic structures, i.e. the volume by which behavior under deformation you can be judged on the behavior of the material in general. This typically means that the size of the representative volume should be approximately an order of magnitude greater than the characteristic pore size or the inclusions in the material. A periodicity cell may be a geometric model for the calculation of the effective properties of periodic structure material.

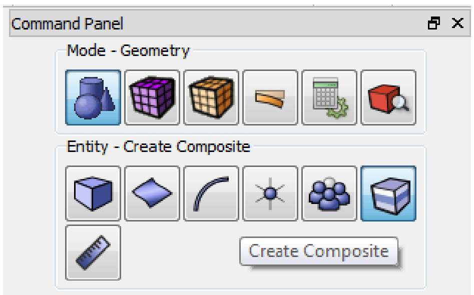
It is important that the geometric model for the calculation of the effective properties must always be a fragment of material «cut» out of it in the form of a **rectangular parallelepiped**. When calculating, this fragment should be positioned so that the edges of the parallelepiped were strictly parallel to the coordinate planes. Automatic checking of the model form and position to calculate the effective properties is not provided, this should be controlled by the User – otherwise the calculation can pass and be correctly completed but the results will be misleading.

Examples of valid models for the calculation of the effective properties are shown below. If the tested material is solid (left), then the model for calculating its effective properties must be a solid rectangular parallelepiped with edges parallel to the coordinate planes. If the material contains pores or cavities, then the model for the calculation must contain cavities that may come to the surface (as shown on the right).





Generation of periodicity cell geometry of some composite materials with periodic structure in ***Fidesys Bundle*** can be performed automatically. In the geometry control mode, there is a button «Create Composite» as shown below.



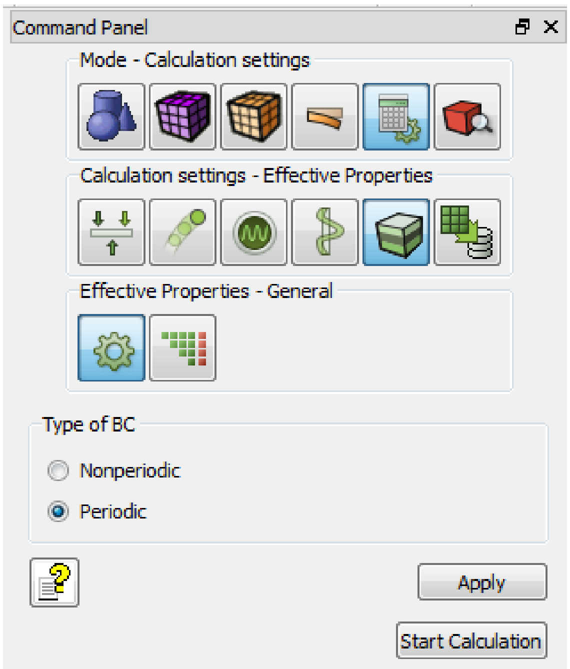
You can create periodicity cells of the following composite types

- Fiber-layered (two-layer) composite;
- single-layer fiber;
- single-layer fiber with shells;
- dispersed fiber reinforced (spherical inclusions);
- dispersed fiber reinforced with shells.

The user needs only to set the parameters of materials and click "Create" - the geometry will be generated automatically by means of the ***Fidesys Bundle*** interface. It stands to reason, the user can also create the geometry for the calculation manually by means of the interface or to import; the most important thing is that the geometric model for the calculation of the effective properties is «cut» out of the material in form of the rectangular parallelepiped with edges parallel to the coordinate system in the ***Fidesys Bundle*** interface.

Starting calculation

After creating the geometry, it is necessary to carry out the same actions as when calculating for static load: blocks creation, finite element mesh generation, material properties setting, etc, except for the boundary conditions application. To calculate the effective properties, it is unnecessary to apply the boundary conditions to the model: when calculating a number of boundary conditions types are automatically applied to the model sequentially; of the static load problem is solved for each type; results of all the problems are averaged and, as a result of averaging, the effective properties of the material are calculated. The user only needs to choose the type of boundary conditions: periodic or nonperiodic.



Periodic boundary conditions are preferred if the effective properties of the material of periodic structure are calculated and the periodicity cell serves as a model for the calculation. For example, if the material is a composite with matrix and inclusions, moreover, the stiffness of the inclusions is much higher than the one of the matrix and the inclusions are located on the surface of the model for the calculation – in this case it is necessary to use periodic conditions. If the effective properties of the material of irregular structure are studied and a representative volume is a model for the calculation, then the nonperiodic boundary conditions are preferred.

In ***Fidesys Bundle 1.6*** it is the SLAE direct solver that is available to calculate the effective properties.

Element types

Fidesys Bundle 1.6 supports the effective properties calculation for the following existing finite elements:

- 1st order 3D hexahedron HEX/HEX8;
- 1st order 3D tetrahedron TETRA/TETRA4.

Effective property calculation and its results

As mentioned, to calculate effective properties the model undergoes a series of strains. The following types of strains are used:

- tension (along each of the coordinate axes);
- shears (in each of the coordinate planes).

The strain magnitude is 0.2% for all types.

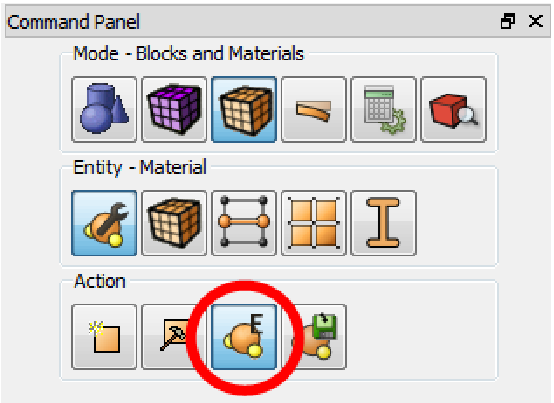
Effective properties are evaluated in the form of the generalized Hook's law:

$$\sigma_{ij} = C_{ijkl}\varepsilon_{kl}$$

The result of the calculation is effective elastic modules C_{ijkl} displayed to the command line and in the file called Cijkl.txt in the working directory. The modules are evaluated in the coordinate system where the calculation was carried out (which coordinate planes are parallel to the edges of calculation model).

Modules C_{ijkl} contain 21 independent constants – it is often more than it is necessary to describe effective properties of the tested heterogeneous material. That is why there is a possibility of the automatic conversion of the obtained effective elastic modules into constants of orthotropic, transversally isotropic or isotropic material. After completing the calculation of the effective properties, the window «Process effective properties data» opens. In the window, obtained effective elastic modules C_{ijkl} are shown at the bottom right in the form of a symmetric matrix sized 6x6 (the matrix part below the main diagonal is not displayed by the symmetry).

When the calculation is complete, the window opens automatically. If the user closes it, it can be re-opened in the mode **Blocks and materials** -> **Material Management** -> **Create the material using calculated effective properties**:



The user can assess whether the matrix with obtained C_{ijkl} to orthotropic materials with the satisfying tolerance. For the exact orthotropic material, the matrix should look as follows (where the letters X denote those components that can be nonzero).

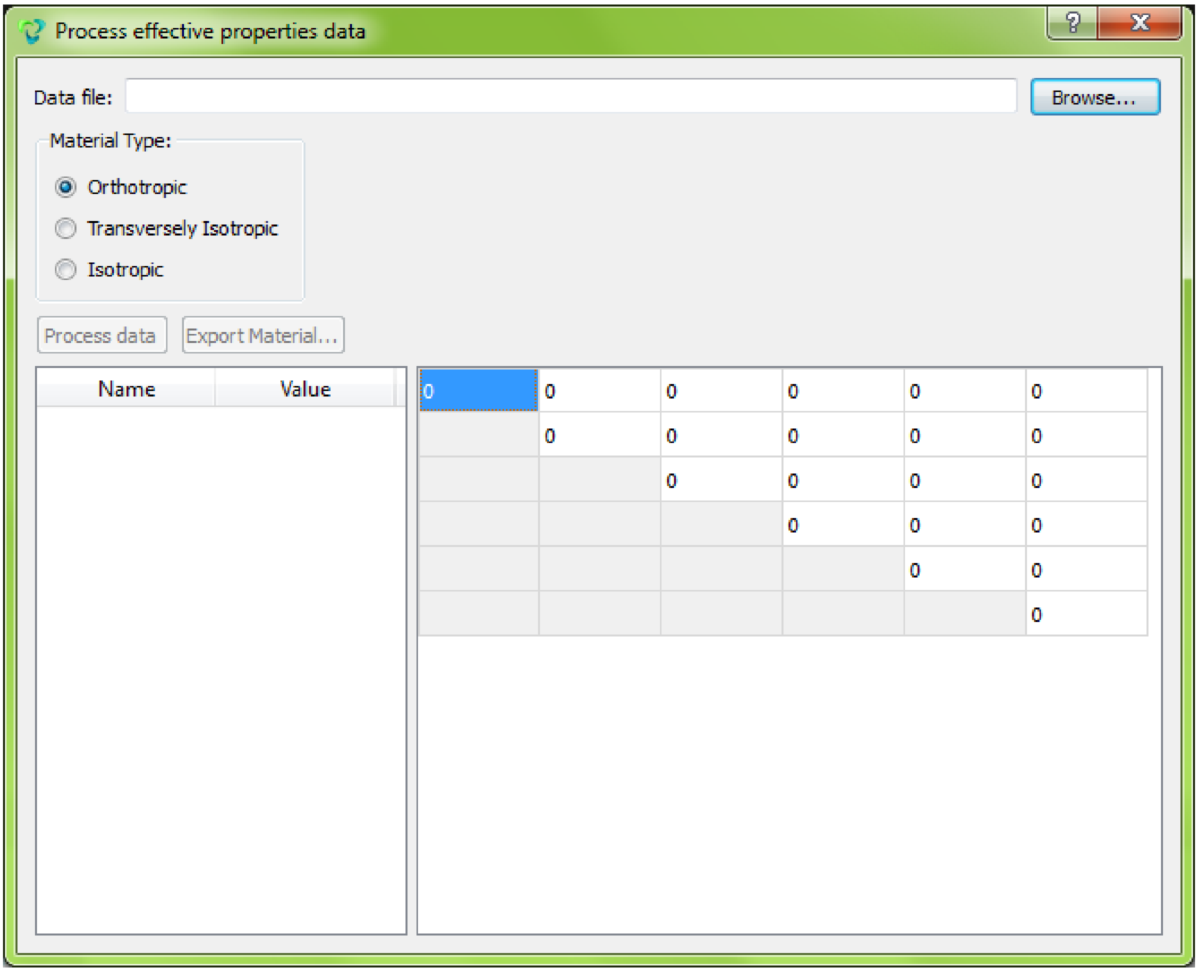
$$\begin{pmatrix} X & X & X & 0 & 0 & 0 \\ & X & X & 0 & 0 & 0 \\ & & X & 0 & 0 & 0 \\ & & & X & 0 & 0 \\ & & & & X & 0 \\ & & & & & X \end{pmatrix}$$

However, since the components of the matrix are the result of numerical calculation of effective properties - they tend to contain some errors. If, from the user's point of view, the matrix corresponds to orthotropic materials with acceptable tolerance, select the «Orthotropic» type of material and click «Process Data» resulting in the fact that nine constants of orthotropic material will be counted.

If orthotropic constants in the X and Y will coincide with acceptable tolerance the user can select the type of material «Transversely isotropic» and click «Process data». Five constants of the transversally isotropic material will be calculated.

But if orthotropic constants do not depend on the direction then you can select the type of material «isotropic» and again click «Process data». Two constants of an isotropic material – Young's Modulus and Poisson's Ratio – will be calculated.

The window exterior «Process effective properties data» is shown in the figure below.



If the processed material constants satisfy the user, the option to export the material into the file XML is available in the same window. You must select a name for the effective material and the name of the XML file into which it will be exported. When you click «Export Material...», the material with the specified name and obtained effective properties are first created whereupon all other created materials are exported into an XML file with the specified name. You can then import these materials from the created file.

If heterogeneous material which efficient properties are investigated is orthotropic, transversely isotropic or isotropic from empirical considerations; and the calculation results do not correspond to that – you should try to refine the mesh or to choose a model for the calculation in another way.

Results Visualization and Postprocessing

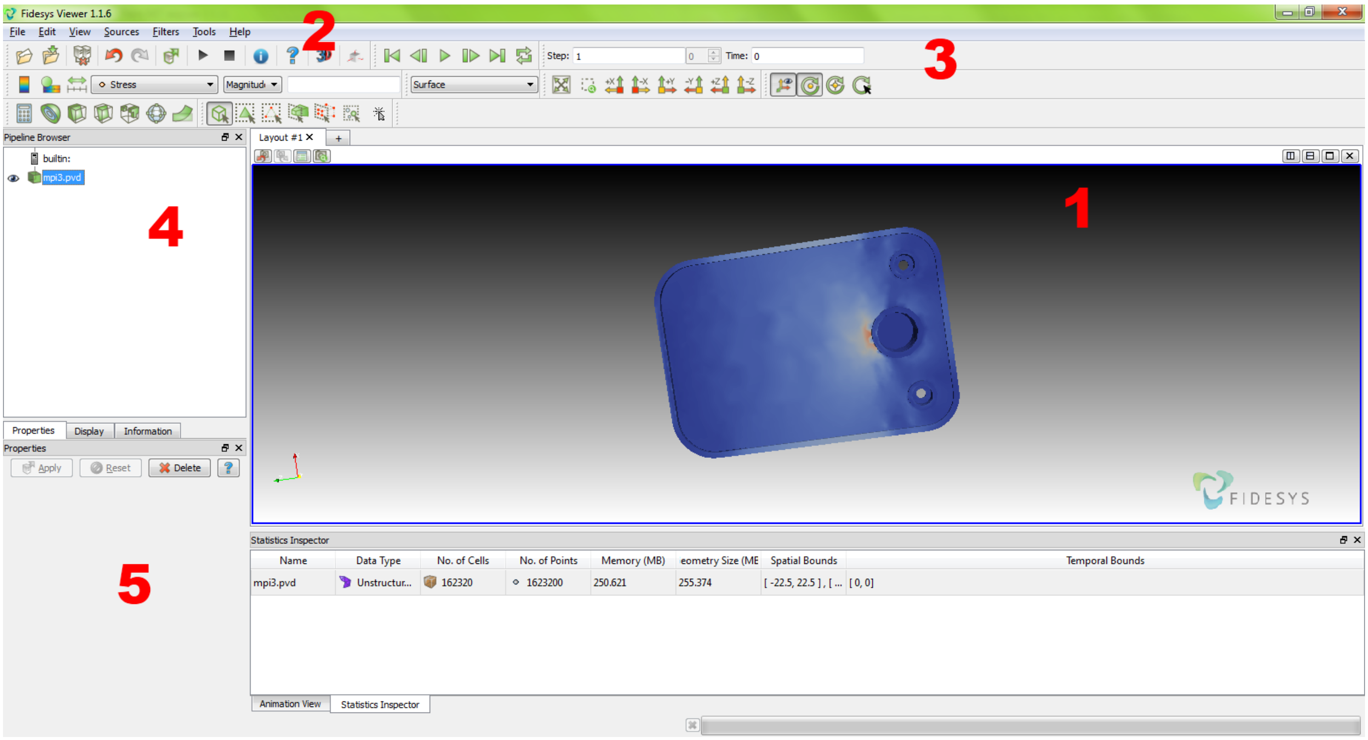
Fidesys Viewer at a glance

The program ***Fidesys Viewer*** is used for visualization and analysis of the obtained results:

- Visualization of vector and tensor fields
- Graph
- Time dependency analysis

Fidesys Viewer is included into the package ***Fidesys Bundle*** and does not require individual installation. To use ***Fidesys Viewer,*** the license is not required: the results of calculations obtained by using the ***Fidesys Bundle*** preprocessor are available for viewing in ***Fidesys Viewer*** even after the expiration of the license.

Main Window



Workbench (1) displays the model and visual effects.

Main Menu (2) includes standard operations for working with files and projects, managing the visualization modes, panel display settings, filters, tools, and help available in the drop-down lists of the menu.

Toolbar (3) comprises the buttons for calling the most frequently used commands while working with the program.

Pipeline Browser (4) includes the opened models and filters applied to them.

Properties Page (5) displays the properties of the selected object in the Workbench or in the Pipeline Browser.

Additional panels can be shown or hidden in the menu **View**.

Basics of the program

Fidesys Viewer allows you to view and analyze the results. Viewing and analysis are performed using multiple filters you can select the item **View** in the menu. Some of them are described below.

Display on the data field and legend model

Fields and components of display can be selected in the Toolbar:

You can also see the Color Map by clicking  in the Toolbar.

Selection

In order to select points or cells, use the following buttons in the Toolbar:

On-screen information display

Numerical results for the data fields can be viewed in the tab **Information.** If the entire model is in focus, the fields of the tab **Information** contain a range of data – from minimum to maximum value.

The values in points can be found using the filter Probe Location (**Filters** → **Alphabetical** → **Probe Location**). Then you must specify the viewing point coordinates. After applying the filter, data field values are displayed only for the specified point in the tab **Information**.

 To view the numerical results for the selected points is also possible by clicking **Point Information** on the Toolbar.

The values in the points/nodes/elements can be identified and viewed by using **Selection Inspector** (**View** → **Selection Inspector**).

Overview of the strained model

To view the strained model, select **Filters** → **Alphabetical** → **Warp By Vector.** In the Properties tab, you can select the display scale.

To quickly access the filter, click **Warp By Vector** on the top panel.

Spherical/cylindrical coordinate systems

To receive data from the spherical or cylindrical coordinate systems, select **Filters** → **Alphabetical** → **Coordinate systems**. Next, select the data field that you want to represent in new coordinates. After applying the filter, a new data field will appear in the tab Information, for example, Stress (spher.).

Graphing along straight line

To graph along a straight line, select **Filters** → **Alphabetical** → **Graph along a straight line.**

Specify coordinates of the beginning and end of the line. In the tab **View**, select the appropriate data field to display in the graph.

Graphing along curves

To graph along a curve, select nodes (see par. Selection) for which graph will be plotted. Next, use **Filters** → **Alphabetical** → **Extract selected** and then **Filters** → **Alphabetical** → **Show data**.

Graphing in time dependency

To plot a time dependency graph, you should allocate points of interest through the Allocation Inspector or by the button **Select points** in the standard string and then apply the filter **Filters** → **Alphabetical** → **Graph selected in time dependency.**

Mises stresses

To obtain Mises stress intensity select **Filters** → **Alphabetical** → **Invariants.** After applying the filter data field **Stress (Mises)** will appear in the tab Information. Quick access button is located in the *Fidesys Viewer* standard string*:*

Estimation of the mesh quality

To estimate the mesh quality, select **View** → **Filters** → **Alphabetical** → **Mesh Quality.** Specify the necessary settings in the tab **Properties.** After applying the filter, the new fields which analysis allows concluding about the mesh quality will appear in the tab **Information**.

Slice

To view the model slice, select **Filters** → **Alphabetical** → **Slice.** Specify the normal or the direction in which you want to make the slice.

Cross section

To view the model cross section, select **Filters** → **Alphabetical** → **Cross section.** Specify the normal or the direction in which you want to make the slice.

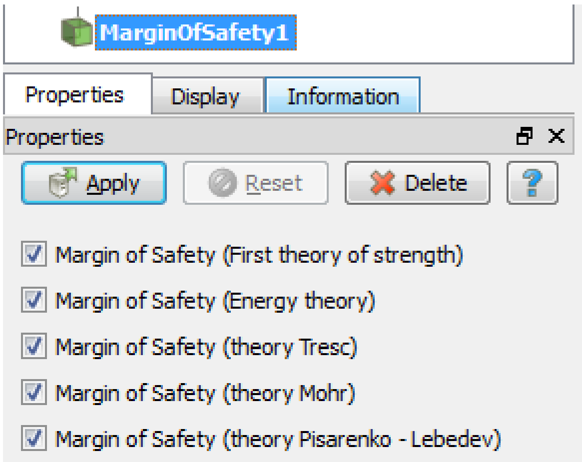
Beam and shell 3D-display

To view beams and shells in 3D in the *Fidesys Viewer* postprocessor, you can click on the button **3D** in the standard string.

Margin of Safety

To view the model cross section, select **Filters** → **Alphabetical** → **Margin Of Safety.** If the ultimate strength and yield strength were not specified when preprocessing, you should set them in the tab **Properties.** Margin of safety is calculated by the first theory of strength, energy theory, Tresca theory, Mohr's theory of failure, Pisarenko-Lebedev theory. Obtained values can be viewed in the tab **Information** in the new field

Margin Of Safety. The first component of the field is the margin of safety by the first theory of strength; the second is the margin of safety by the energy theory, etc.



Smoothed results

To smooth the obtained results, select **Filters** → **Alphabetical** → **Consistent results.** Select the field which numerical data are to be converted. After applying the filter a data field, for example, the **Stress (smoothed)**, will be displayed in the tab **Information.**

Data saving

To get numerical values of the obtained results, save the data in .csv format. Click **Ctrl+S** or select **File** → **Save** to do this. The saved file is an ordinary table of numerical data which can be opened in any text editor.

For dynamic problems, saving the model variation under deformation is available. Select **File** → **Save Animation** to do this.

Step-by-Step User Guide

Solving any problem using FIDESYS package includes 6 basic steps:

- Model generating
- Meshing
- Setting boundary conditions
- Setting the material
- Starting calculation
- Results analysis

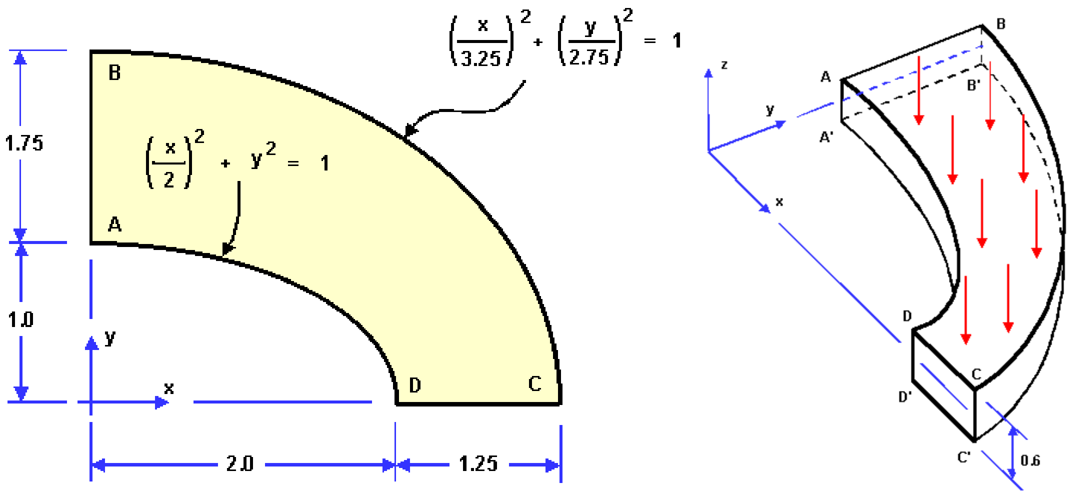
See some examples with step-by-step guide below.

Static analysis (3D)

NAFEMS test "Thick Plate Pressure", Test No LE10, Date/Issue 1990-06-15/2.

The problem of static load of an ellipsoid plate is being solved.

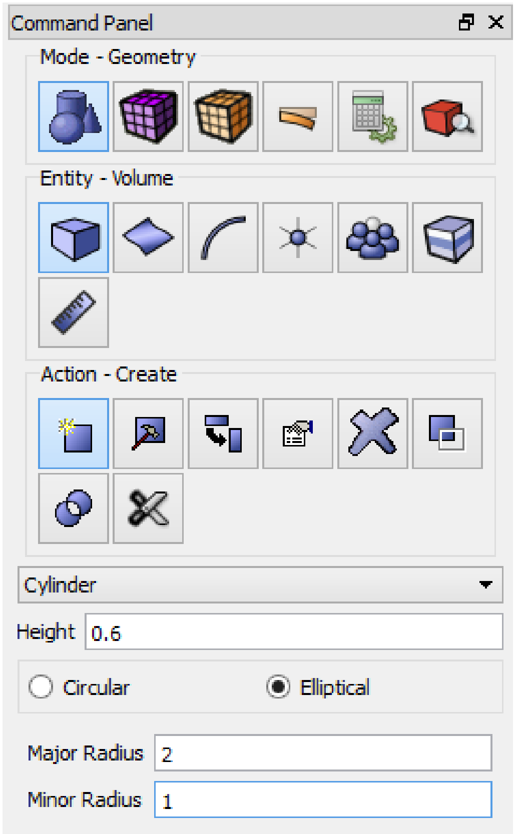
The pictures below represent a geometric model of the problem:



Displacements along the normal to the sides are constrained in the side slices of the plate. All of the points of the outer curvilinear surface are fixed in the XY plane. The outer curvilinear surface is fixed along the middle line of displacements along Z axis. The pressure to the upper side is 1 MPa. The material parameters are E = 210 hPa, v = 0.3.

Test pass criterion is the following: stress σ_{yy} at the point D is -5.38MPa to within 3%.

Geometry creation

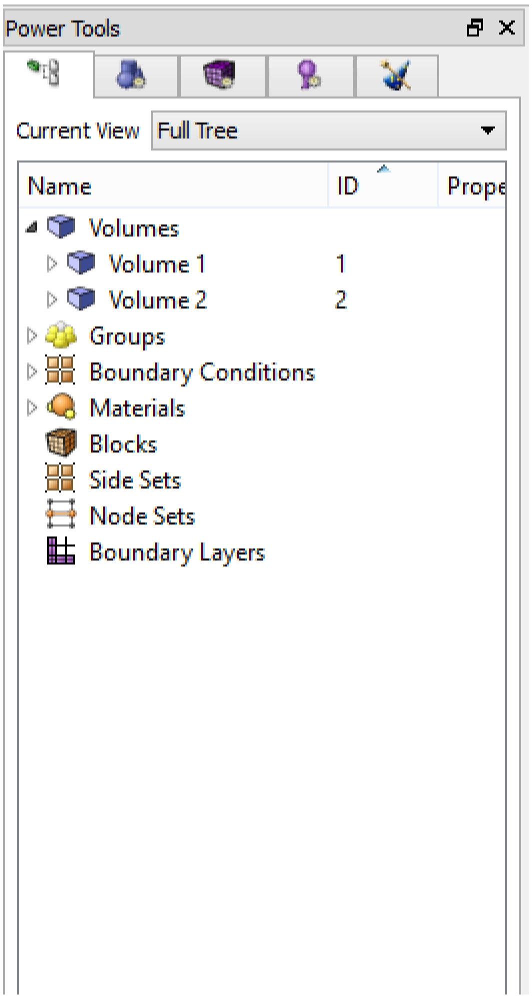


1. Create the first elliptic cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 0.6
- Cross section: Elliptical
- Major Radius: 2
- Minor Radius: 1

Click **Apply**.



2. Create the second elliptic cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 0.6;
- Cross section: Elliptical;
- Major Radius: 3.25;
- Minor Radius: 2.75.

Click **Apply**.

As a result, two generated entities are displayed in the Model Tree (Volume 1 and Volume 2):

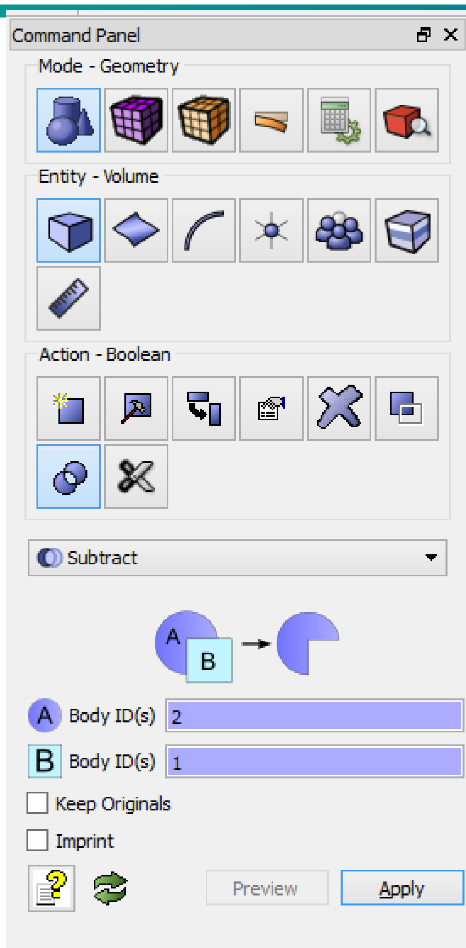
3. Subtract the first cylinder from the second one .

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

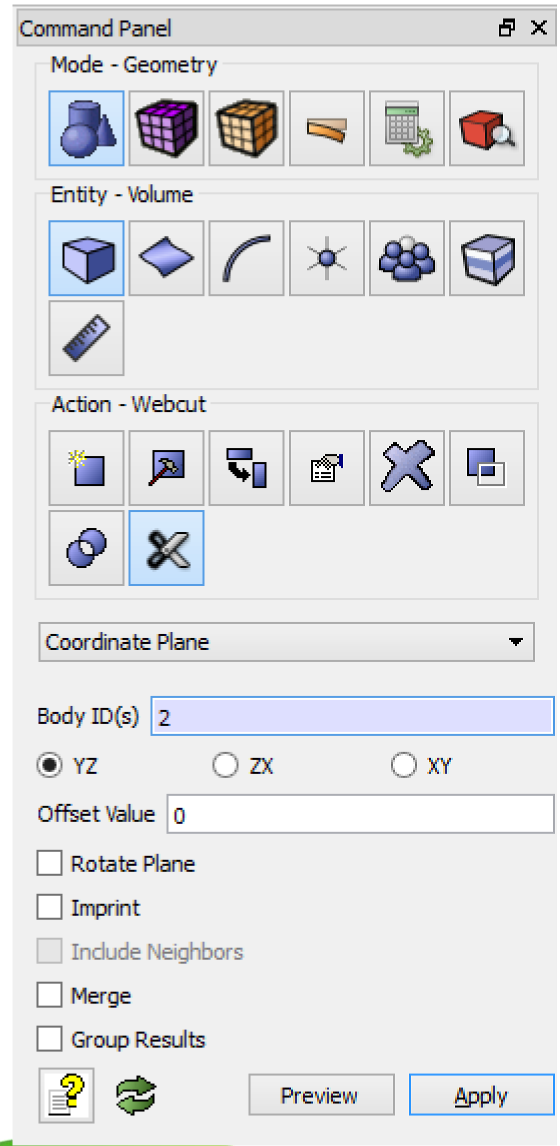
- Volume ID(s): 1 *(the volumes to be subtracted);*
- From Volume ID(s): 2 *(volumes from which other volumes will be subtracted);*
- Imprint.

Click **Apply**.

As a result, only one volume is displayed in the Model Tree (Volume 2).



4. Leave a quarter of a volume (symmetry of the problem).



Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

- Volume ID(s): 2 (*the volume to be webcut*);
- Webcut with: YZ Plane;
- Offset value: 0;
- Imprint.

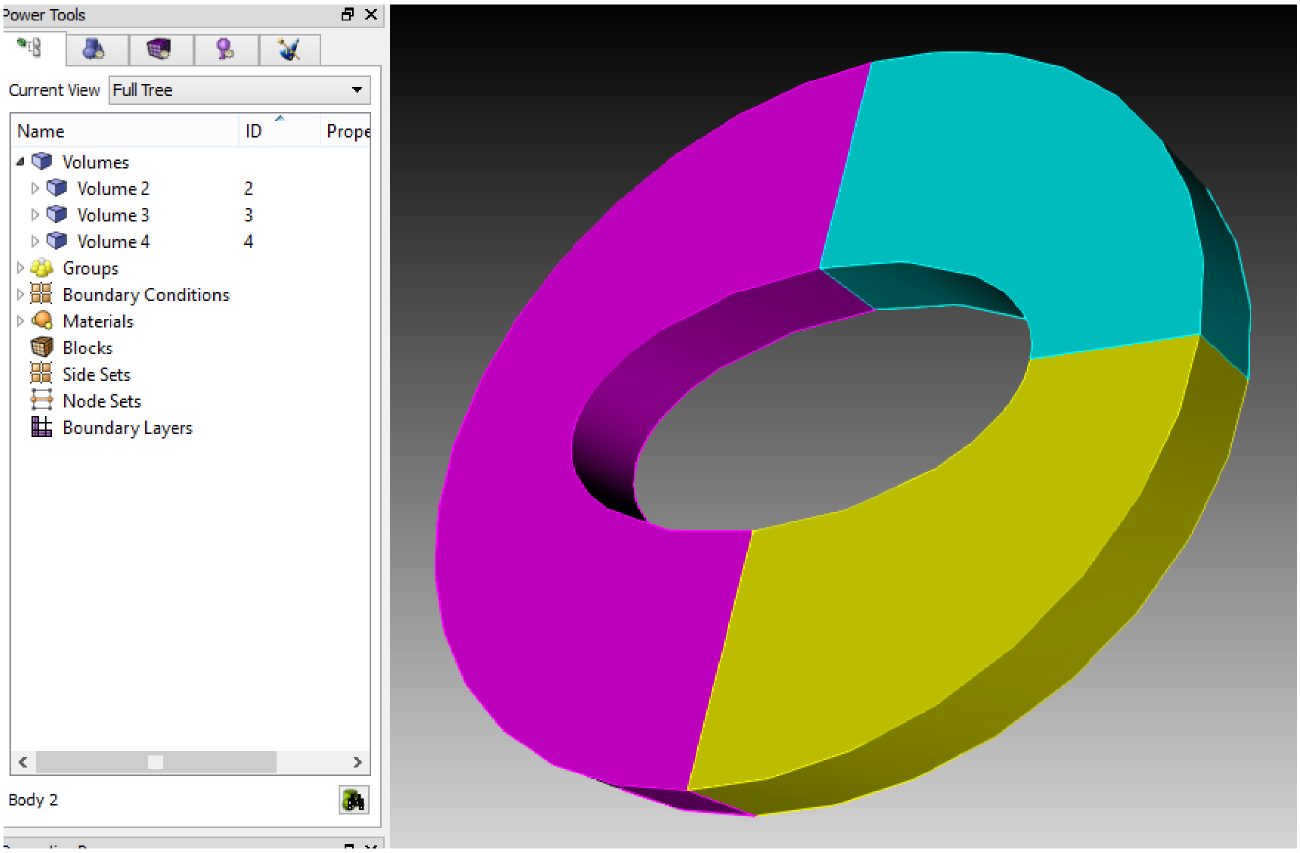
Click **Apply**.

Do the same for the ZX Plane:

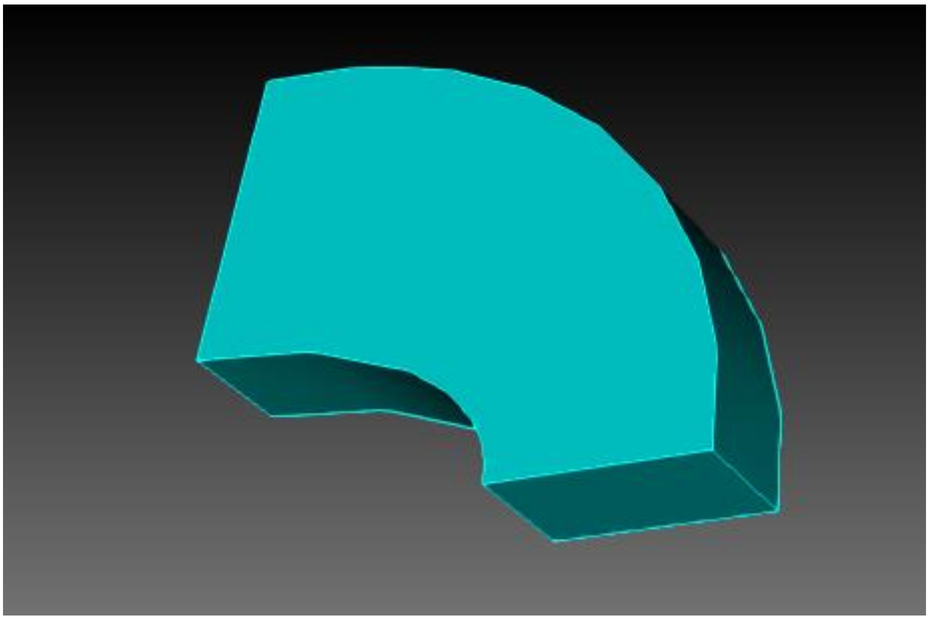
- Volume ID(s): 2 *(the volume to be webcut);*
- Webcut with: ZX Plane;
- Offset value: 0;
- Imprint.

Click **Apply**.

As a result, the original volume in the Model Tree is split into three (Volume 2, Volume 3 and Volume 4).



Delete the volumes 2 and 3. To do this, select these volumes in the Model Tree holding down Ctrl and click **Delete** in contextual menu. As a result, a quarter of the original volume is left (Volume 4):



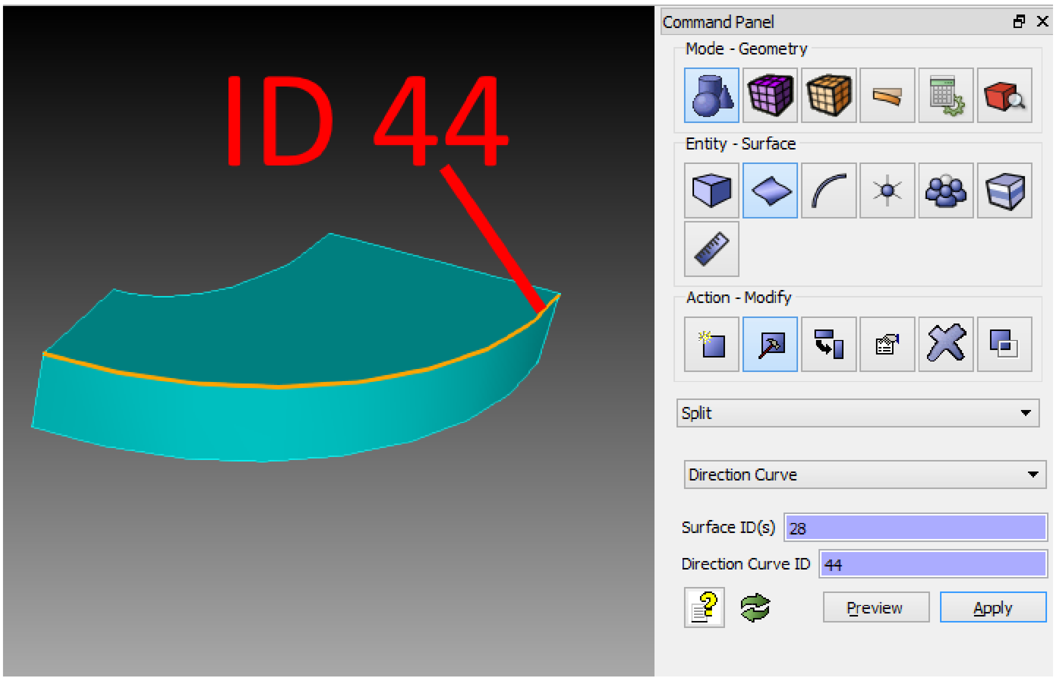
5. Split the outer curvilinear surface into two (it is necessary for restraining this surface from displacements along the middle line).

Select surface geometry modification section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Modify**). Select **Split** in the list of possible operations. Set the following parameters:

- Surface ID(s): 28 *(the surface to be split along the middle line);*
- Split Method: Direction Curve;
- Direction Curve ID: 44.

Click **Apply**.

As a result, the outer curvilinear surface is split into two, and two new surfaces (Surface 32 and Surface 33) are displayed in the Model Tree instead of Surface 28:



Meshing

It is necessary to calculate the value at the point D, that is why the generated mesh should be refined in the vicinity of the point.

1. Select meshing on curves section on Command Panel (Mode – **Mesh**, Entity – **Curve**, Action – **Mesh**). Specify the parameters of mesh refinement:

- Select Curves: 46 44 45 43 *(using space after each curve)*;
- Select the way of meshing: Equal;
- Select the checkbox Interval;
- Specify the number of intervals: 6

Click **Apply**.

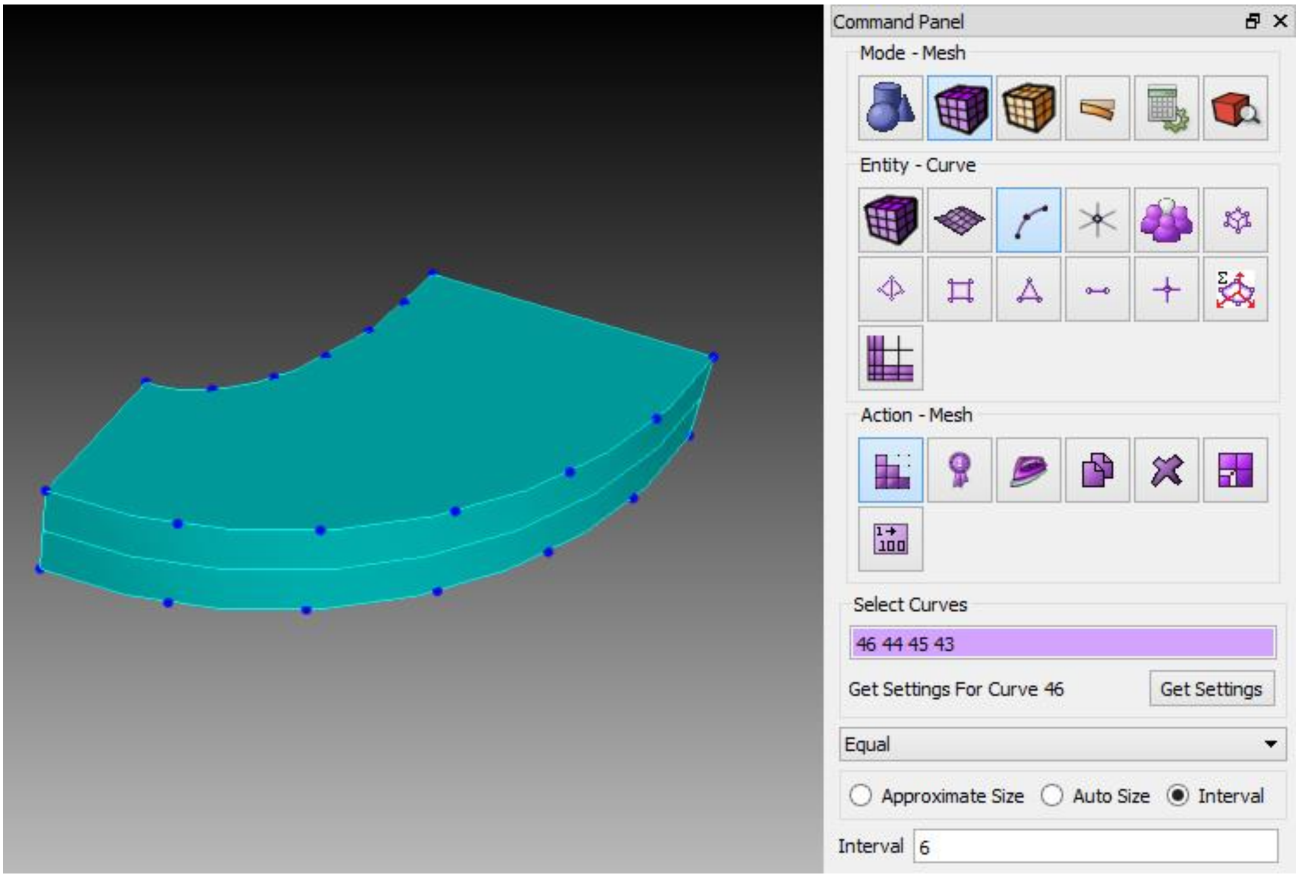
Click **Mesh**.

Select meshing on curves section on Command Panel (Mode – **Mesh**, Entity – **Curve**, Action – **Mesh**).

- Select Curves: 14 41 39 12 *(using space after each curve)*;
- Select the way of meshing: Equal;
- Select the checkbox Interval;
- Specify the number of intervals: 3;

Click **Apply**.

Click **Apply Scheme**

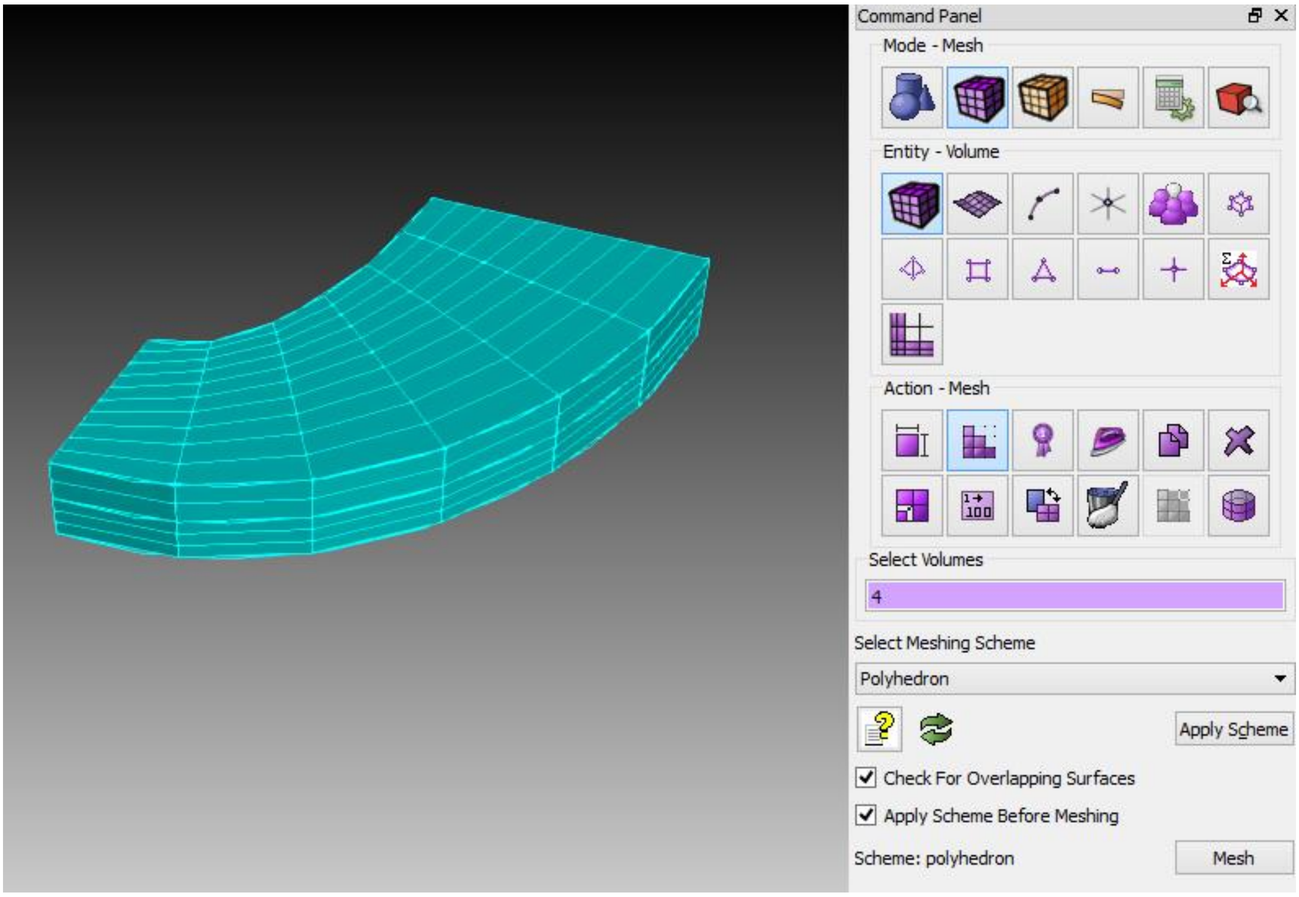


2. Select volume mesh generation section on Command Panel (Mode – **Mesh**, Entity – **Volume**, Action – **Mesh**).

- Select Entities to Mesh (specify their ID): 4 *(or by the command **all**)*;
- Select Meshing Scheme: Polyhedron.

Click **Apply Scheme**.

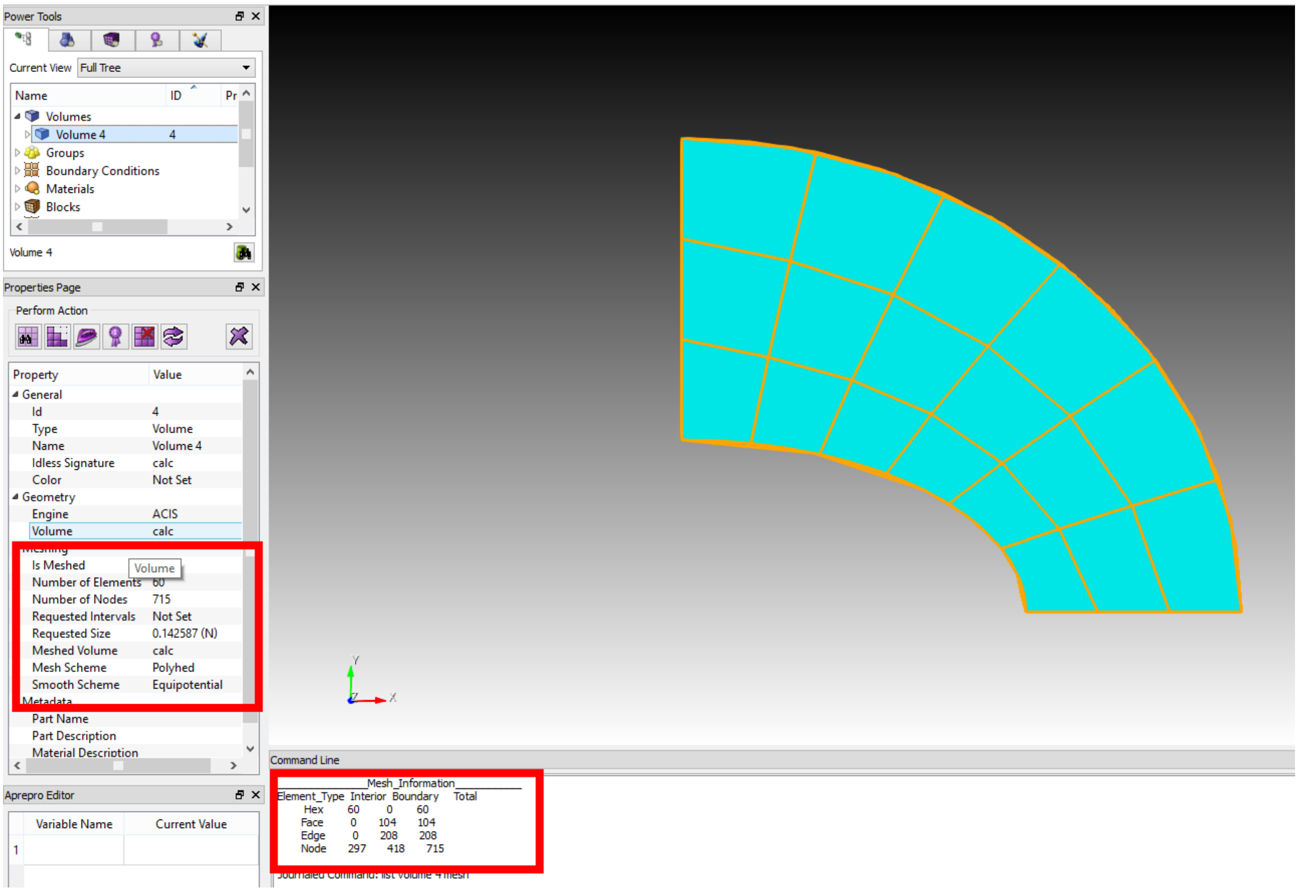
Click **Mesh**.



The resulting number of elements can be viewed in the Property Page by clicking on the inscription Volume 4 in the Model Tree on the left.

To view the mesh properties, you can follow these steps:

- Select the entire model
- Right-click on the model
- In the pop-up menu, select List Information – List Mesh Info
- Information on the mesh will be displayed in Command Line


Power Tools
Current View Full Tree
Name ID
Volumes
Volume 4 4
Groups
Boundary Conditions
Materials
Blocks
Volume 4
Properties Page
Perform Action
Property Value
General
Id 4
Type Volume
Name Volume 4
Idless Signature calc
Color Not Set
Geometry
Engine ACIS
Volume calc
Is Meshed
Number of Elements 60
Number of Nodes 715
Requested Intervals Not Set
Requested Size 0.142587 (N)
Meshed Volume calc
Mesh Scheme Polyhed
Smooth Scheme Equipotential
Part Name
Part Description
Aprepro Editor
Variable Name Current Value
Command Line
Mesh_Information
Element_Type Interior Boundary Total
Hex 60 0 60
Face 0 104 104
Edge 0 208 208
Node 297 418 715

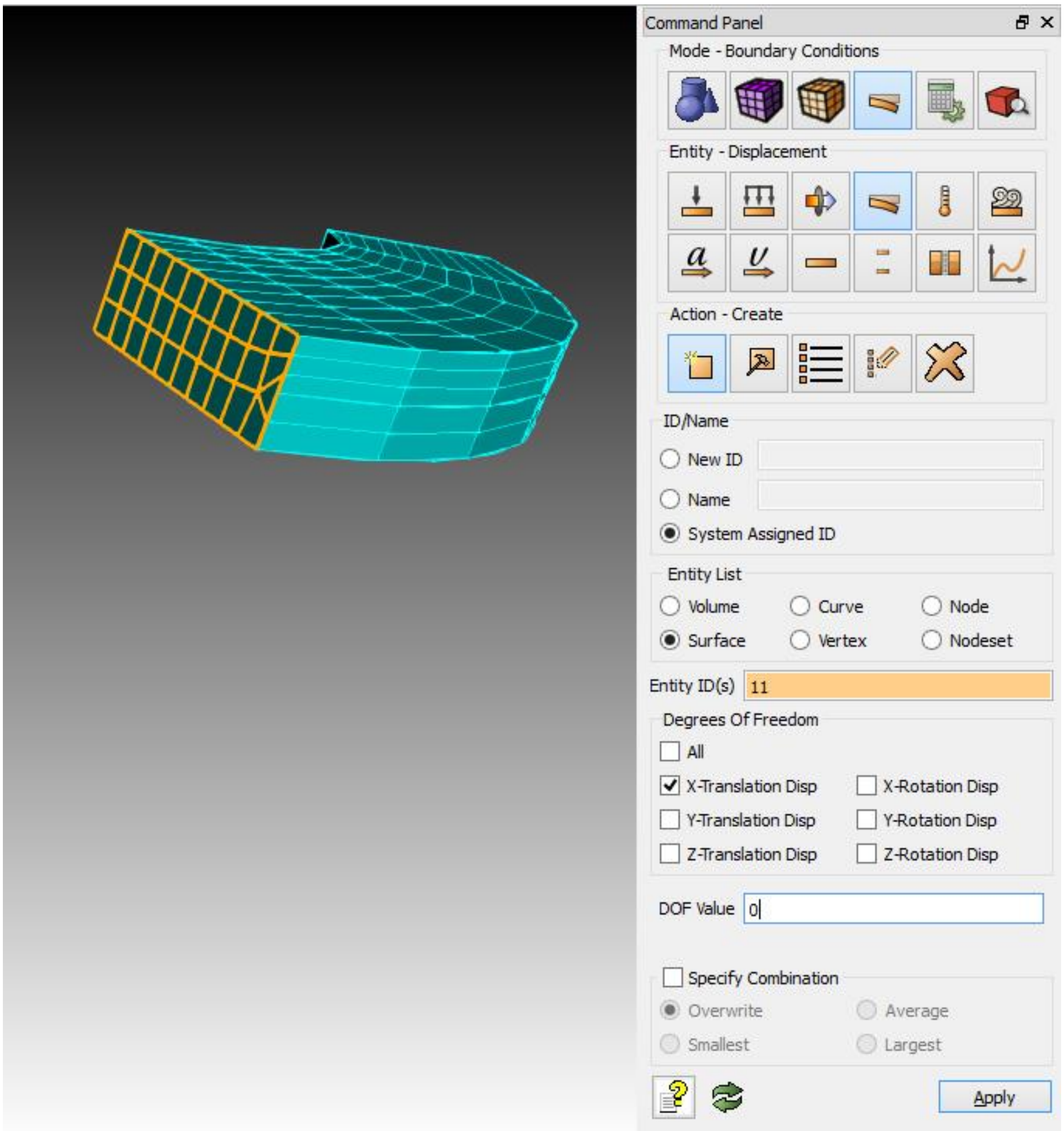
Setting boundary conditions

1. Fix one side (slice) along X axis.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID
- Entity List: Surface
- Entity ID(s): 11
- Degrees of Freedom: x-Translation
- DOF Value: 0

Click **Apply**.



2. Fix one side (slice) along Y axis.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID
- Entity List: Surface
- Entity ID(s): 27
- Degrees of Freedom: y-Translation
- DOF Value: 0

Click **Apply**.

3. Fix the outer curvilinear surface along X and Y axes.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID
- Entity List: Surface
- Entity ID(s): 32 33
- Degrees of Freedom: x-Translation and y-Translation
- DOF Value: 0

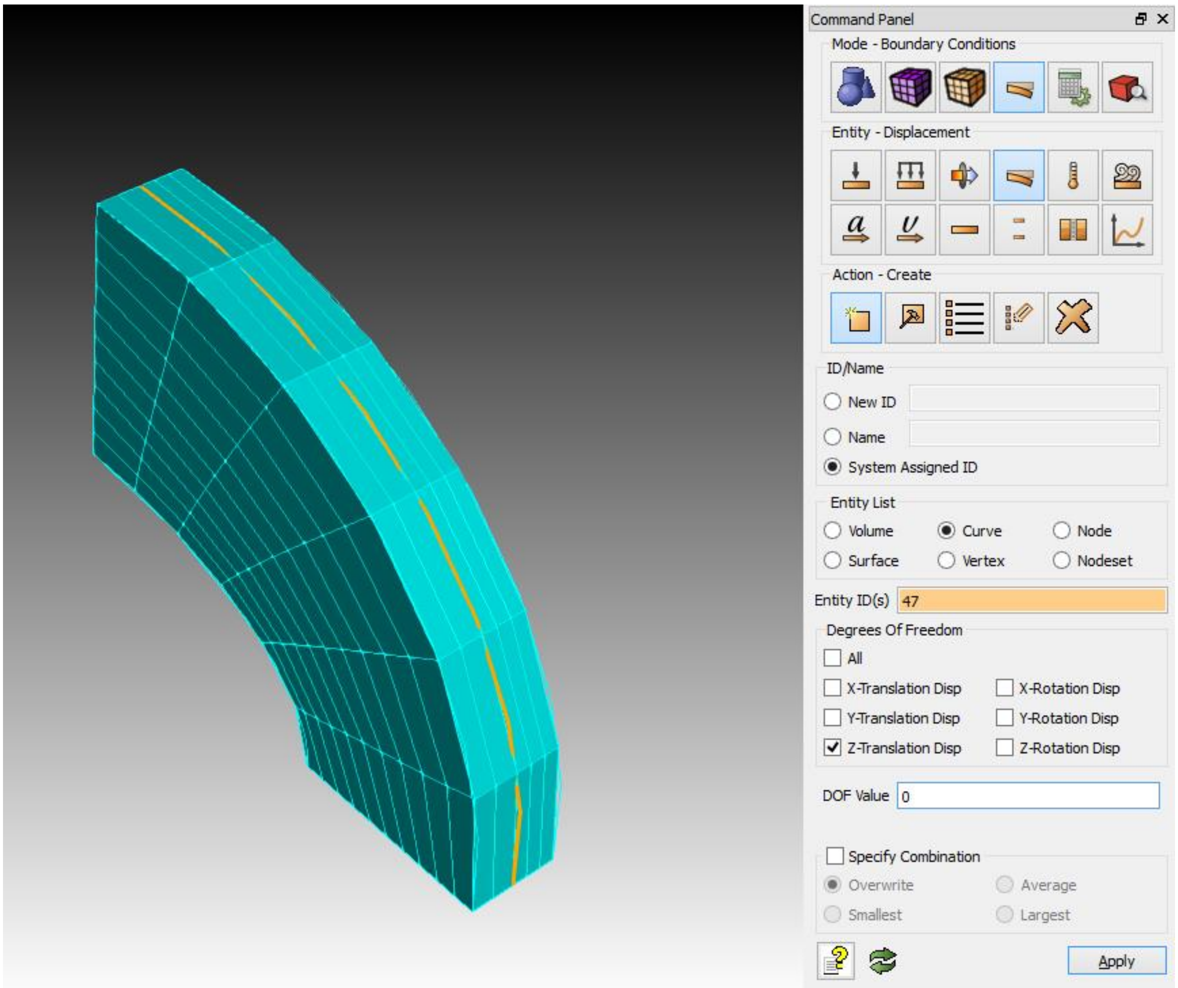
Click **Apply**.

4. Fix the middle line of the outer curvilinear side along Z axis.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID
- Entity List: Curve
- Entity ID(s): 47
- Degrees of Freedom: z-Translation
- DOF Value: 0

Click **Apply**.

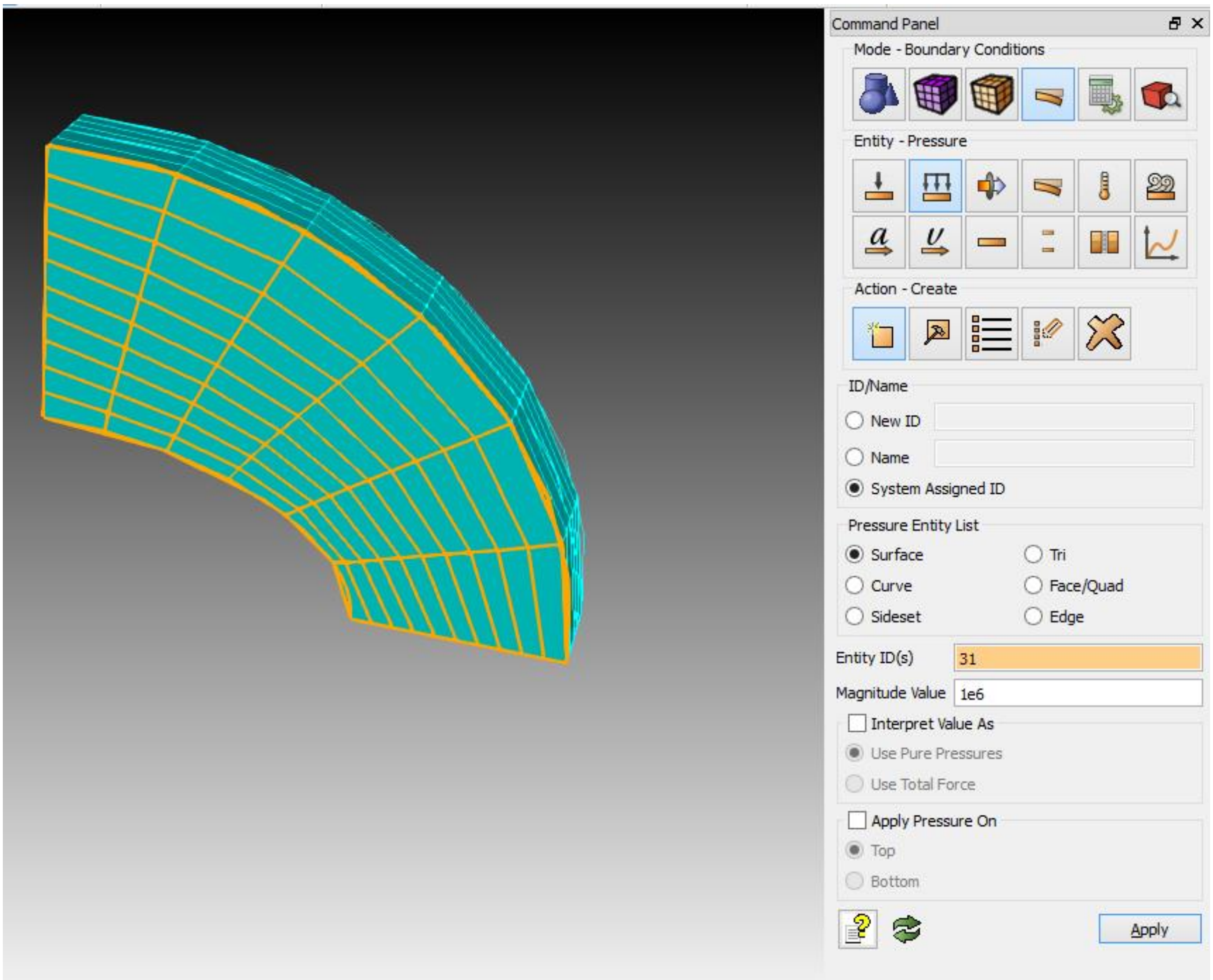


5. Apply pressure to the upper side.

Select Mode – **Boundary Conditions**, Entity – **Pressure**, Action – **Create**. Set the following parameters:

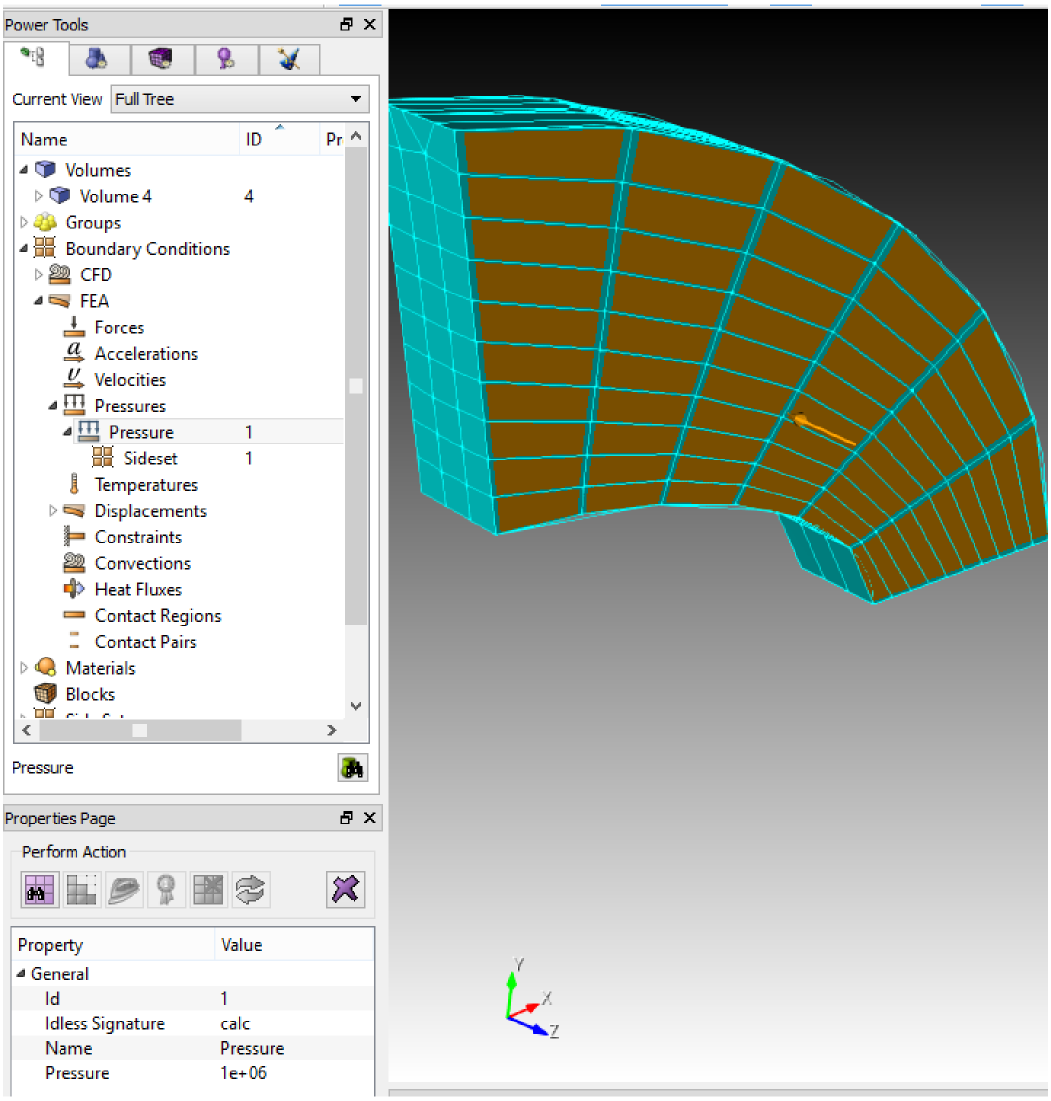
- System Assigned ID
- Entity List: Surface
- Entity ID(s): 31
- Magnitude Value: 1e6 *(an exponential number format using the Latin letter"e" is supported)*

Click **Apply**.



All applied boundary conditions must be displayed in the Model Tree on the left. In addition, the boundary conditions are available for editing from the Model Tree.

To view all the applied boundary conditions also click Show BC on the top panel.



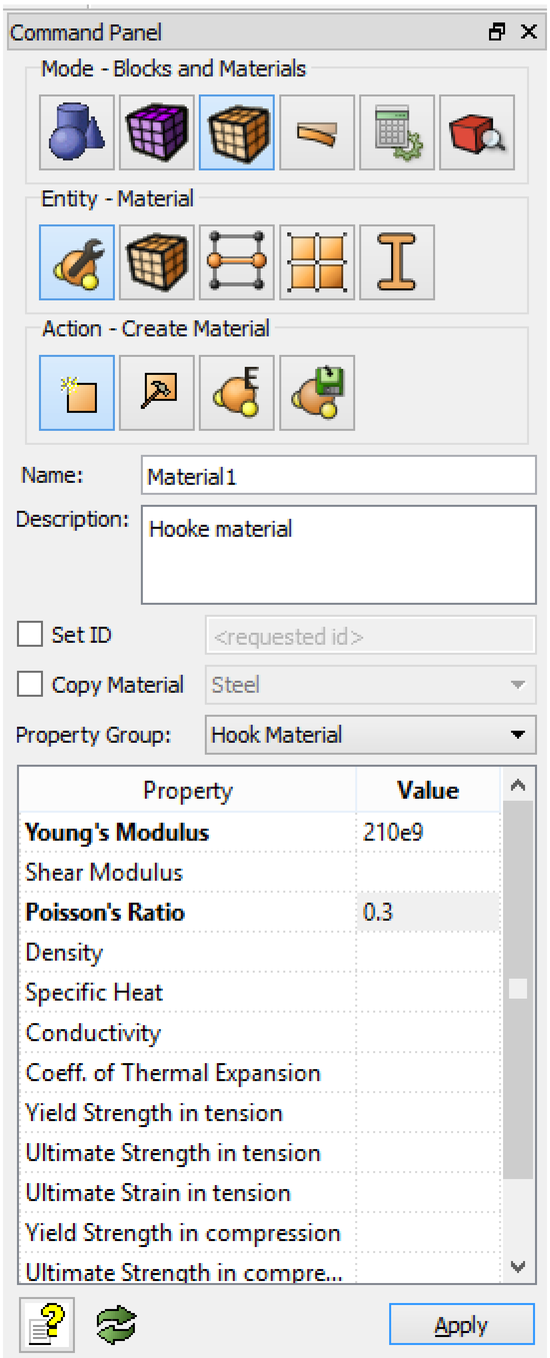
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block**, Action – **Create Material**). Set the following parameters:

- Name: Material1;
- Description: Hook material
- Properties group: Hook material;
- Young's Modulus: 210e9;
- Poisson's Ratio: 0.3.

Click **Apply**.

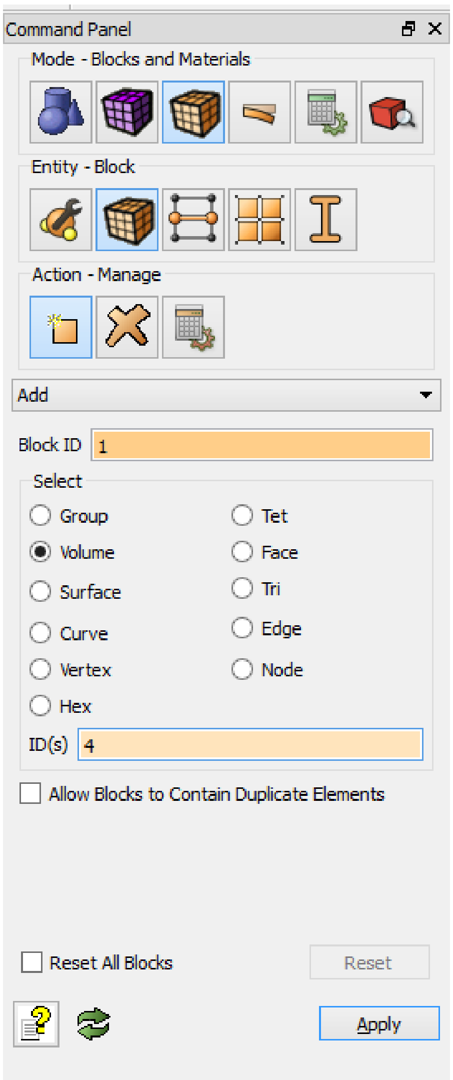


2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block**, Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Select the entities to be united into block: Volume;
- ID(s): 4 *(or by the command **all**)*.

Click **Apply**.

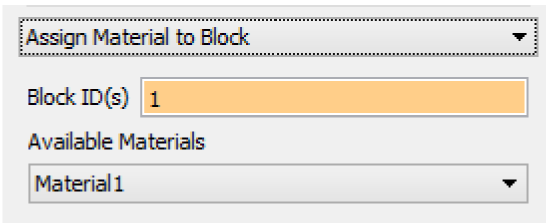


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block**, Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block ID(s): 1
- Select the previously created material in the list: FEA Material 1

Click **Apply**.

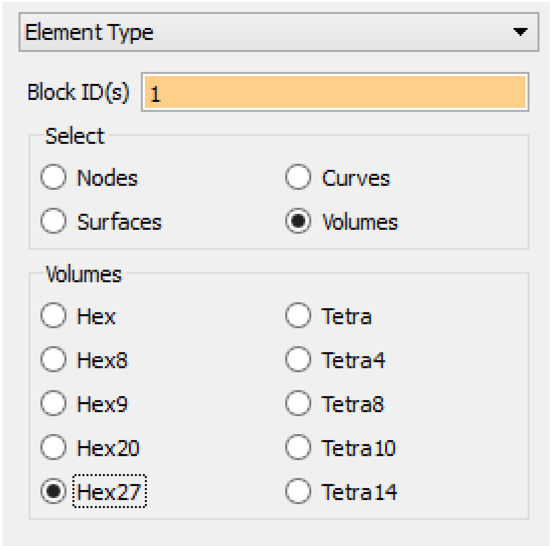


4. Assign the element type.

Select material properties setting section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block**, Action – **Manage**). Select **Element type** in the list of possible operations. Set the following parameters:

- Block ID(s): 1;
- Select: Volumes;
- Volumes: HEX27.

Click **Apply**.



Starting calculation

1. Set the type of the problem to be solved.

Select calculation settings section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select:

- Dimension: 3D;
- Model: Elasticity.

Click **Apply**.

2. Set the solver settings.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply**.

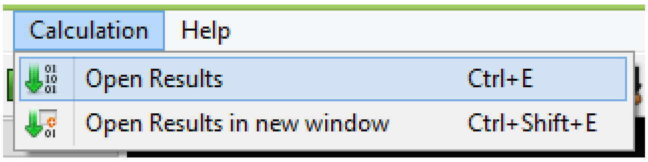
Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name.

4. If the calculation is finished successfully, you will see a message in the Console: *“Calculation finished successfully at <date> <time>”.*

Results analysis

1. Open the file with the results. You can do this in one of the three ways.
    - Press Ctrl+E.
    - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
    - Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.

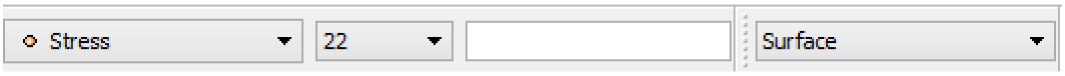



To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

2. Display the component σ_{yy} of the stress field and the mesh on the model.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Stress;
- Representation Component: 22.
- Surface with edges.



3. Select a point where you need to view the stress.

In the Main Menu, choose View – Allocation Inspector.

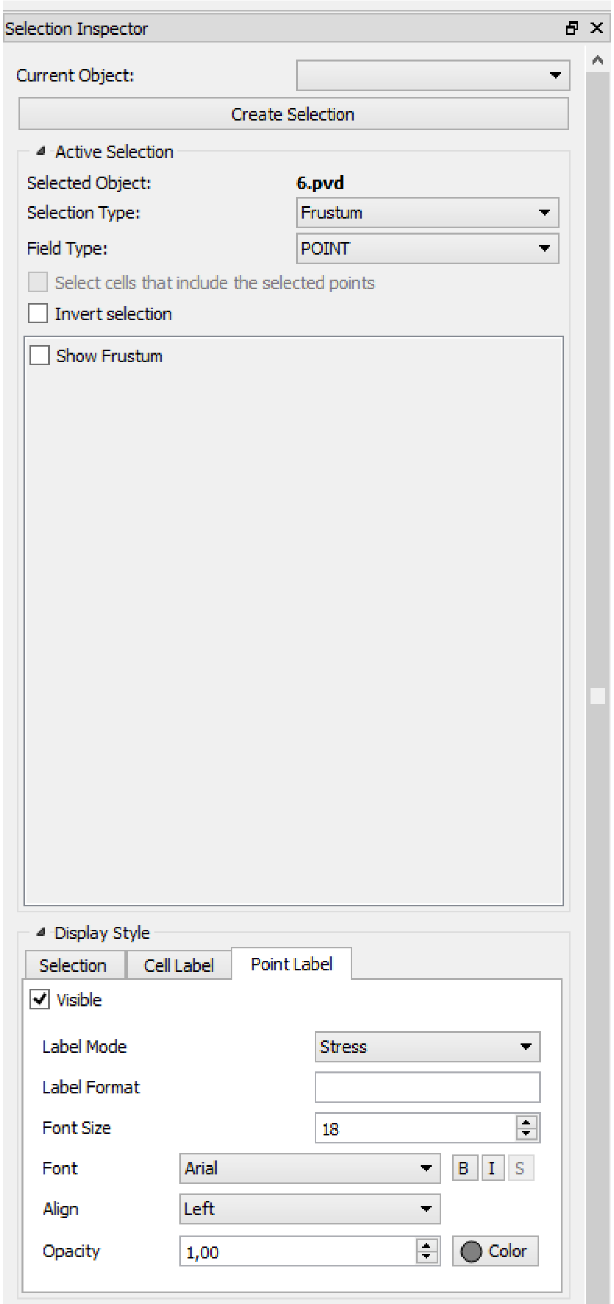
Select a point on the model by using **Select Points Through**.

Select a point D on the upper side. The node number (243) should be displayed in **Selection Inspector** on the right.

In Allocation Inspector, go to the tab **Point Tag** and select the following settings:

- Enable Visibility;
- Tag Mode: Stress;

As a result, Stress components at the point D is displayed at the picture.



4. View a numerical value σ_{yy} at the selected point D.

The difference between the obtained value -5.294807e+06 and the required one -5.380e+06 is 1.61%.

5. Download numerical data.

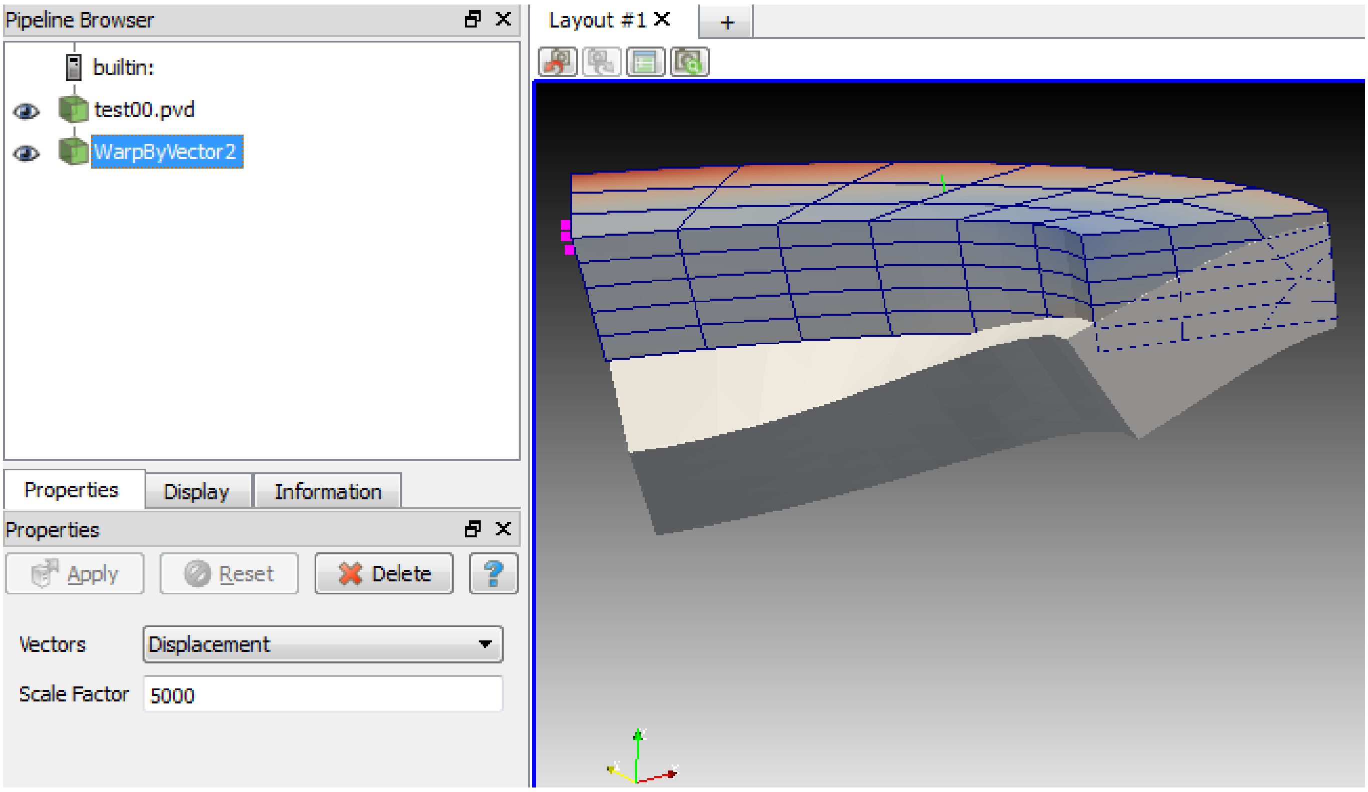
Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

6. You can see the way the body is deformed under the applied pressure.

Select the filter **Warp By Vector** to do this. Set the following parameters in the tab **Properties**:

- Vectors: Displacements;
- Scale Factor: 5000.

As a result, the deformed body is displayed at the picture. To see the original model, click near it in the Model Tree. The picture below shows the deformed (solid grey filling) and the original model (with the field of displacements distribution along Y axis).



Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd)

```
reset
set node constraint on
create Cylinder height 0.6 major radius 2 minor radius 1
create Cylinder height 0.6 major radius 3.25 minor radius 2.75
subtract body 1 from body 2 imprint
webcut body 2 with plane xplane offset 0 imprint
webcut body 2 with plane yplane offset 0 imprint
delete Body 3
delete Body 2
split surface 28 direction curve 44
curve 46 44 45 43 interval 6
curve 46 44 45 43 scheme equal
curve 46 44 45 43 interval 6
curve 46 44 45 43 scheme equal
mesh curve 46 44 45 43
curve 14 41 39 12 interval 3
curve 14 41 39 12 scheme equal
curve 14 41 39 12 interval 3
curve 14 41 39 12 scheme equal
mesh curve 14 41 39 12
volume 4 scheme Polyhedron
volume 4 scheme Polyhedron
```

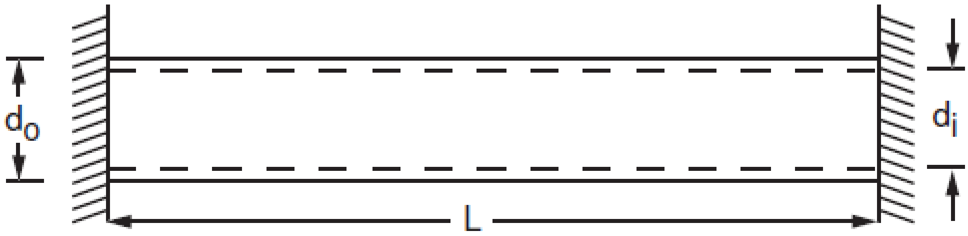
```
mesh volume 4
create displacement on surface 11 dof 1 fix 0
create displacement on surface 27 dof 2 fix 0
create displacement on surface 32 33 dof 1 dof 2 fix 0
create displacement on curve 47 dof 3 fix 0
create pressure on surface 31 magnitude 1e6
undo group begin
create material "material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "material 1" scalar_properties "MODULUS" 2.1e+11 "POISSON" 0.3
undo group end
set duplicate block elements off
block 1 volume 4
block 1 material 'material 1'
block 1 element type hex27
analysis type static elasticity dim3
spectralelement off
usempi off
solver method auto try_other off
solver method auto try_other off
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

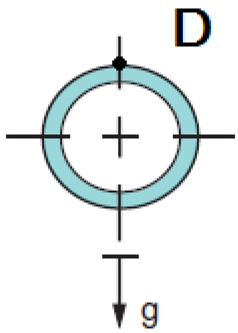
It is also possible to run the file *Example_1_Static_3D.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Static load (gravity force)

RJ. Roark, Formulas for Stress and Strain, 4th Edition? McGraw-Hill Book Co., Inc., New York, NY, 1965, pg 112, no. 33

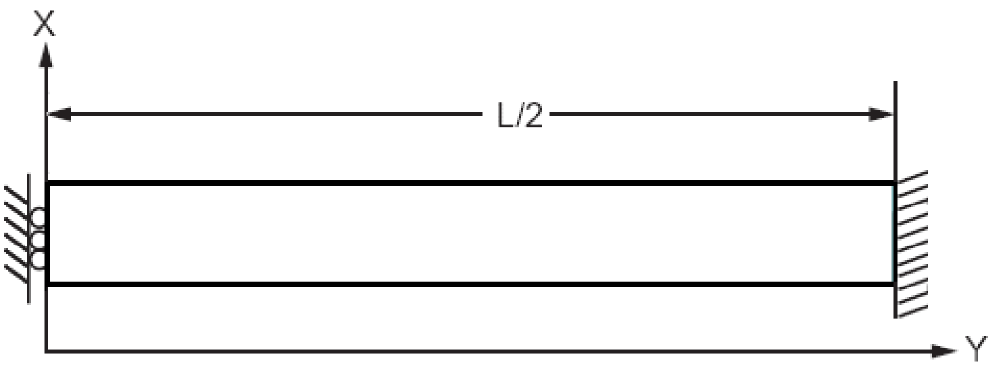
The problem of the tube bending of under gravity force is to be solved. The pictures below represent a geometric model of the problem:





The side edges are rigidly fixed on all displacements and rotations. Material parameters are E = 30e6 psi, v = 0.0, ρ=0.00073 lb-sec^2/in^4. The gravity force is defined via the acceleration g=386 in/sec^2. The geometrical dimensions of the model: L=200 in, d_0=2 in, d_i=1 in.

Due to the symmetry of the problem, half tube will now be considered (L/2).



Test pass criterion is the following: displacement in the center of the tube u_{yy} at the point D (0, d_0/2, 0) is -0.12529 within 3%.

Geometry creation

1. Create the first circular cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 100;
- Cross section: Circular;
- Radius: 1;

Click **Apply**.

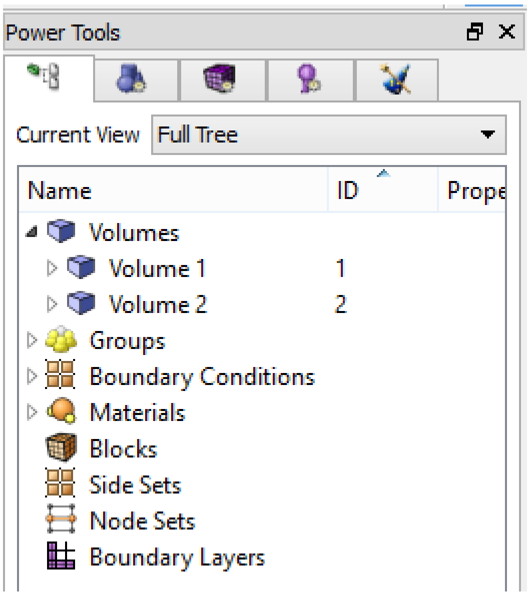
2. Create the second cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 100;
- Cross section: Circular;
- Major Radius: 0.5;

Click **Apply**.

As a result, two generated entities are displayed in the Model Tree (Volume 1 and Volume 2).



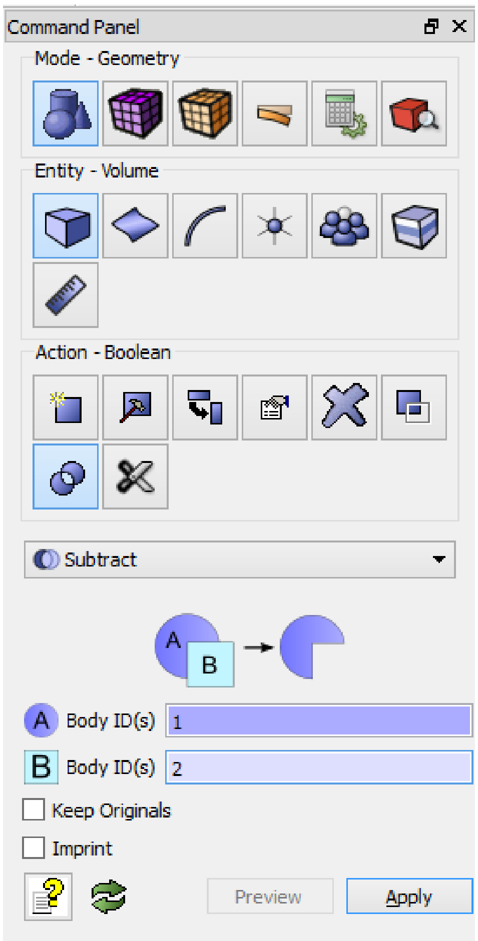
3. Subtract the first cylinder from the second one.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Volume ID(s): 1 *(the volumes to be subtracted)*;
- From Volume ID(s): 2 *(volumes from which other volumes will be subtracted)*;
- Imprint.

Click **Apply**.

As a result, only one volume is displayed in the Model Tree (Volume 1).



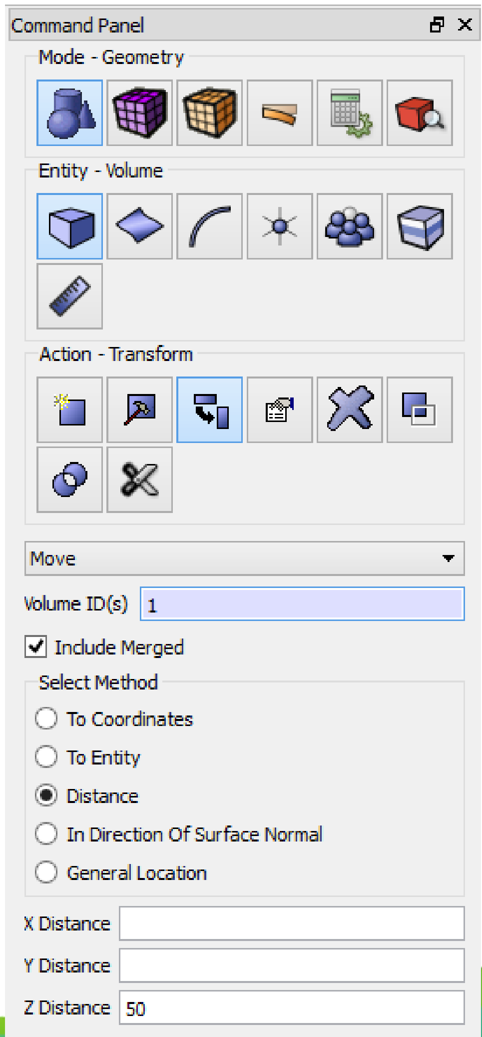
4. Place the volume to the coordinate origin.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Modify**). Select **Move** from the list of possible types of slices. Set the following parameters:

- Volumes ID: 1 *(the volume to be cut)*;
- Checkbox Distance;
- Distance along the Z axis: 50;

Click **Apply**.

Thus, the center of the left end of the tube is placed in the origin of coordinates.



Meshing

1. Set the approximate size of the elements.

   Select volume mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Volume**, Action – **Intervals**). Specify the approximate size of the elements:

   - Select volumes: 1;
   - Select Approximate size from the drop-down list;
   - Approximate size: 0.25;

Click **Apply**.

2. Select the way of mesh generation.

   Select volume mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Volume**, Action – **Mesh**).

   - Select volumes: 1;
   - Select meshing scheme: Polyhedron;

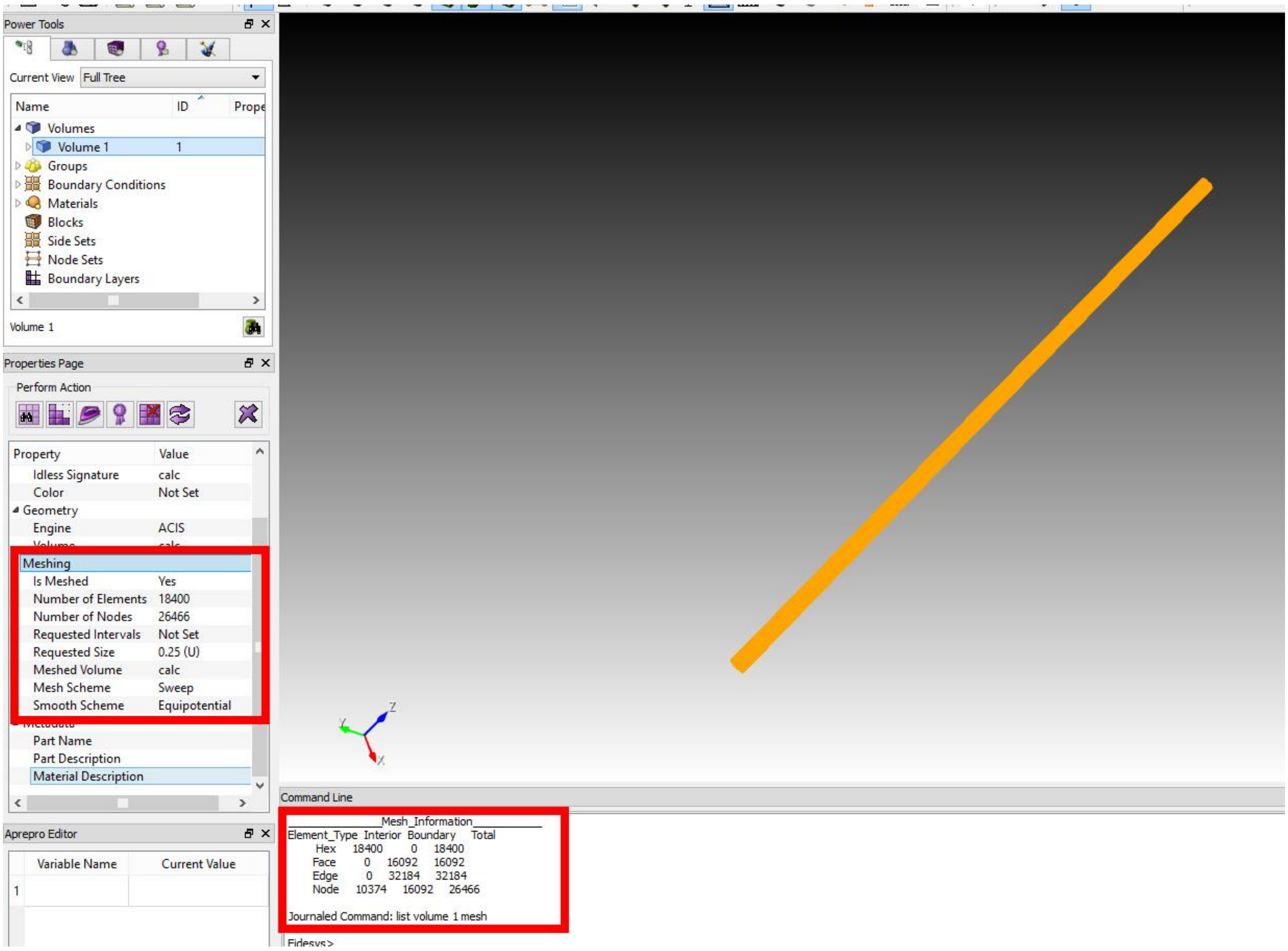
Click **Apply**.

Click **Mesh.**

The resulting number of elements can be found on the property page by clicking on Volume 1 in the Model Tree on the left.

To view the mesh properties, you can follow these steps:

- Select the entire model
- Right-click on the model
- In the pop-up menu, select List Information – List Mesh Info
- Information on the mesh will be displayed in Command Line


Power Tools
Current View Full Tree
Name ID Prope
Volumes
Volume 1 1
Groups
Boundary Conditions
Materials
Blocks
Side Sets
Node Sets
Boundary Layers
Volume 1
Properties Page
Perform Action
Property Value
Idless Signature calc
Color Not Set
Geometry
Engine ACIS
Meshing
Is Meshed Yes
Number of Elements 18400
Number of Nodes 26466
Requested Intervals Not Set
Requested Size 0.25 (U)
Meshed Volume calc
Mesh Scheme Sweep
Smooth Scheme Equipotential
Part Name
Part Description
Material Description
Aprepro Editor
Variable Name Current Value
Command Line
Mesh_Information
Element_Type Interior Boundary Total
Hex 18400 0 18400
Face 0 16092 16092
Edge 0 32184 32184
Node 10374 16092 26466
Journaled Command: list volume 1 mesh
Fidesys>

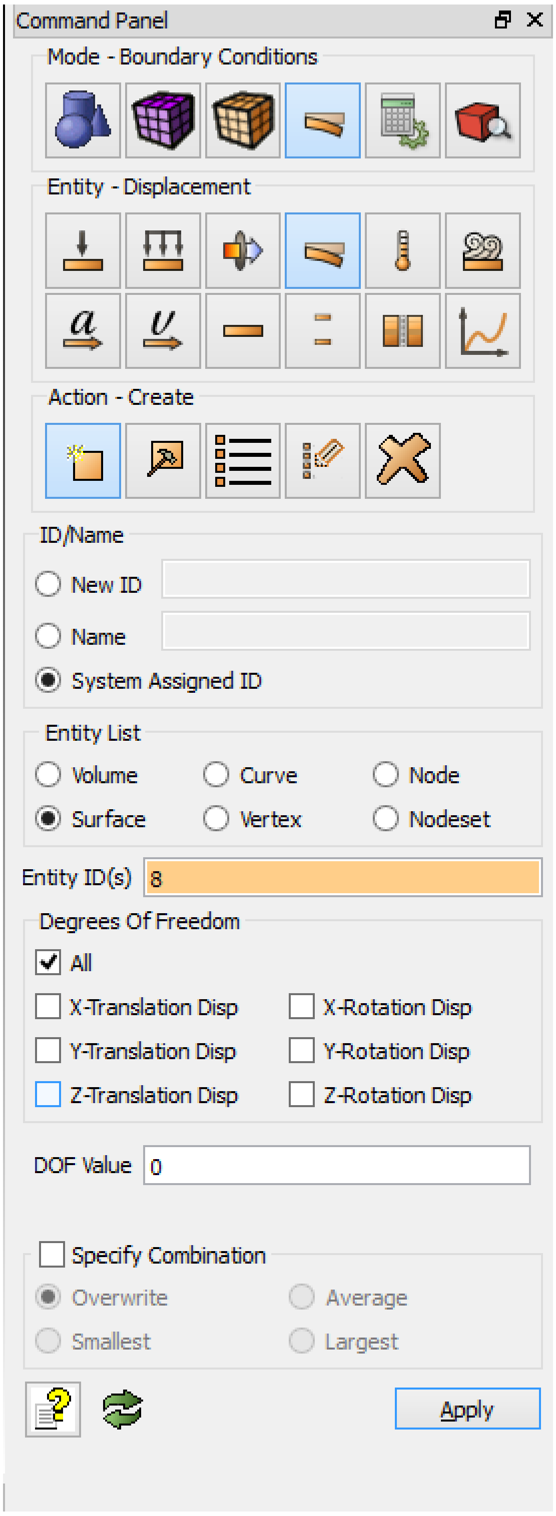
Setting boundary conditions

1. Fix the right lateral edge at all directions.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 8;
- Degrees of Freedom: All;
- DOF Value: 0.

Click **Apply**.



2. Fix the left lateral edge along X and Z axes by analogy.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 9;
- Degrees of Freedom: X-Component, Z-Component;
- DOF Value: 0.

Click **Apply**.

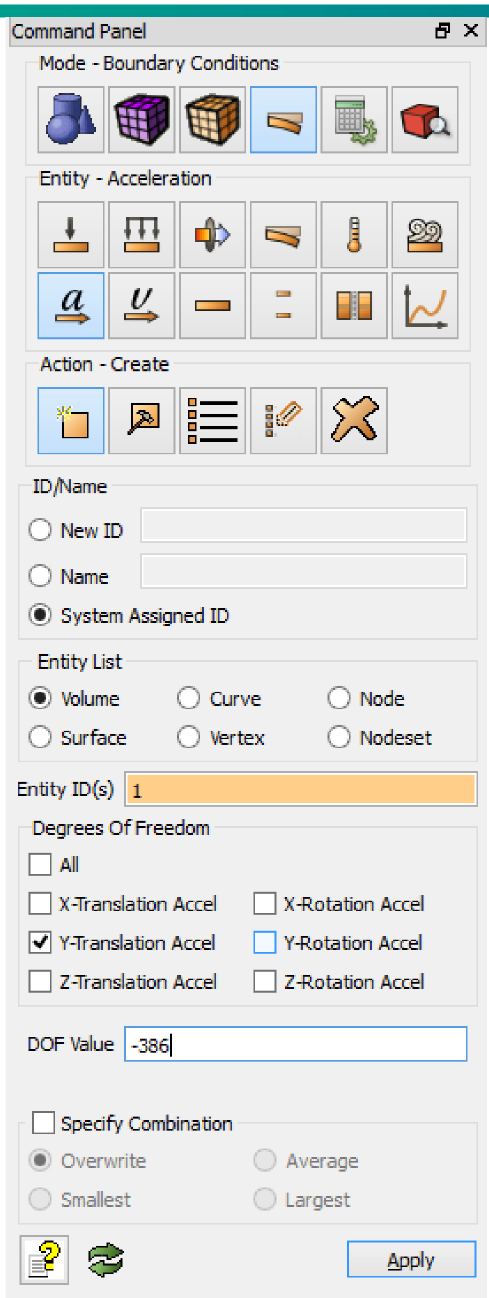
3. Set the gravity force using the acceleration.

Select Mode – **Boundary Conditions,** Entity – **Acceleration**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Volume;
- Entity ID(s): 1;
- Degrees of Freedom: Acceleration Y;
- DOF Value: -386.

Click **Apply**.

Note: The acceleration is set for mass forces taking into account the sign. In this case, the acceleration g acts in the negative direction of the Y axis.



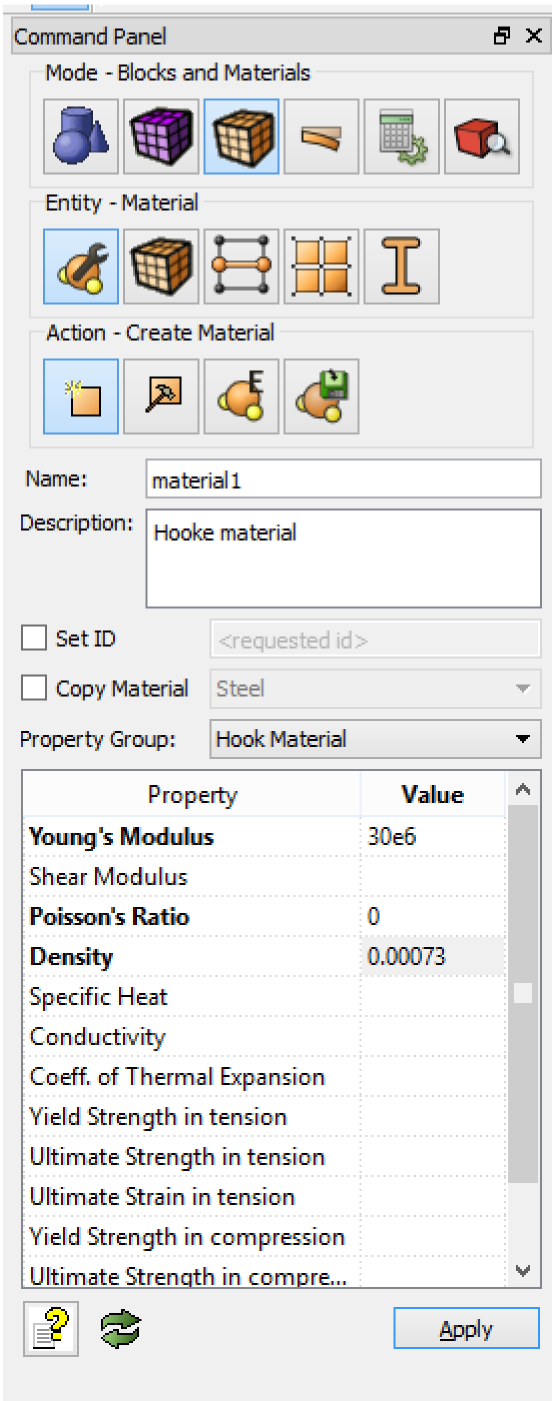
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 30e6;
- Poisson's Ratio: 0;
- Density: 0.00073.

Click **Apply**.

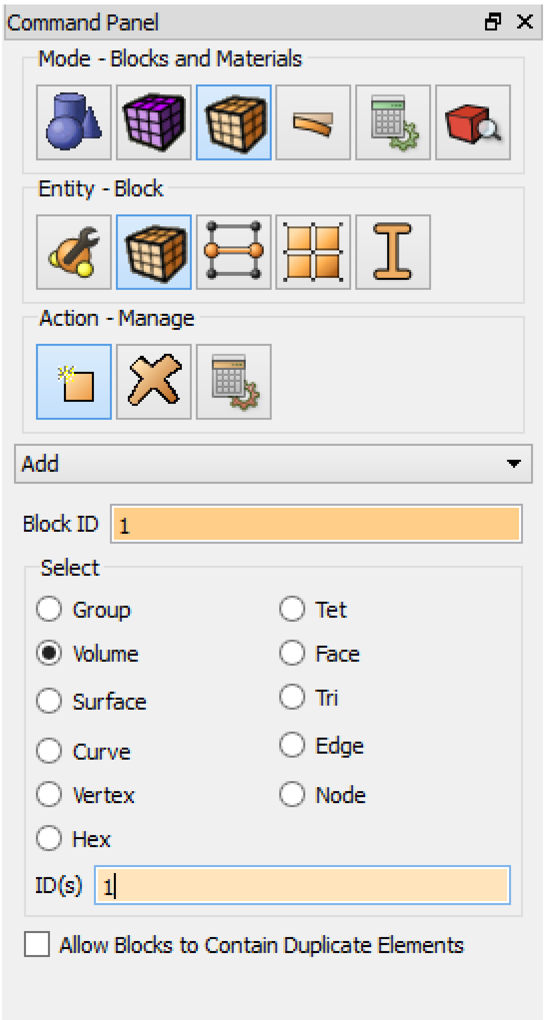


2. Create the block of one material type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Select entities to be united into the block: Volume;
- ID: 1 *(or by the command **all**).*

Click **Apply**.

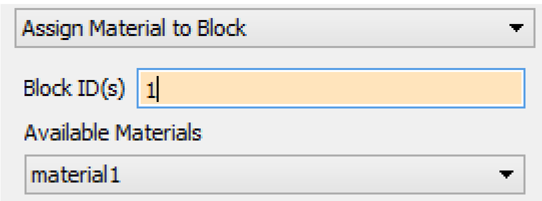


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

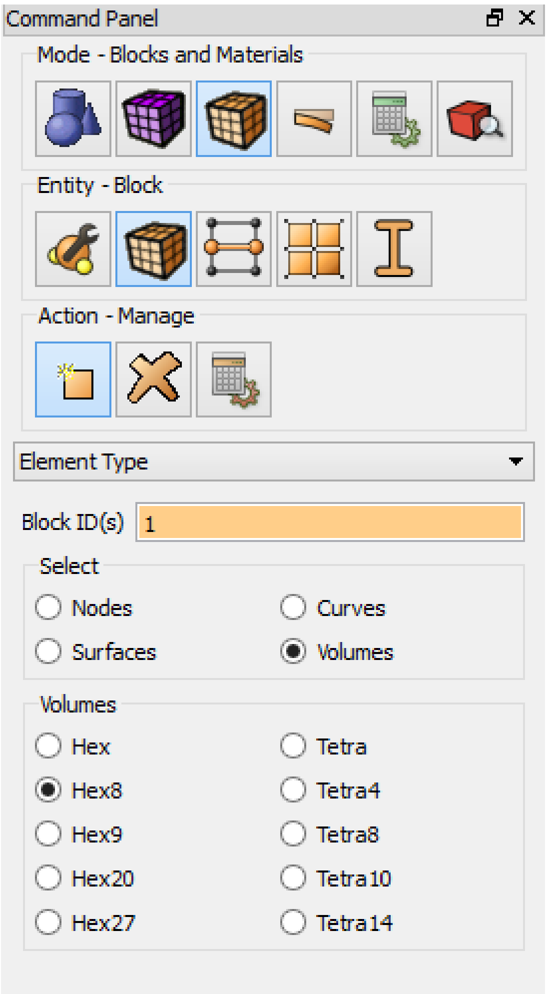


4. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Volumes;
- Volumes: HEX8.

Click **Apply**.



Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **General**). Select:

- Dimension: 3D;
- Model: Elasticity.

Click **Apply**.

2. Set the solver settings.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply**.

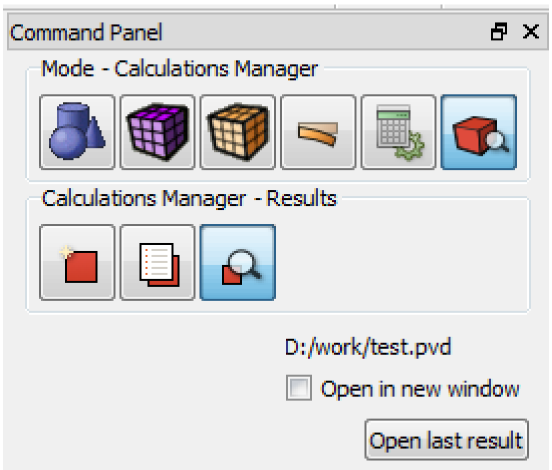
Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name.
4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis

1. Open the file with the results. You can do this in one of the three ways.

- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.



- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.

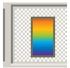
To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

2. Display the U_{yy} component of the displacement field on the model.

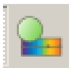
In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Displacement;
- Representation Component: 2.
- Surface.

3. Display the maximum Displacement U_{yy}.

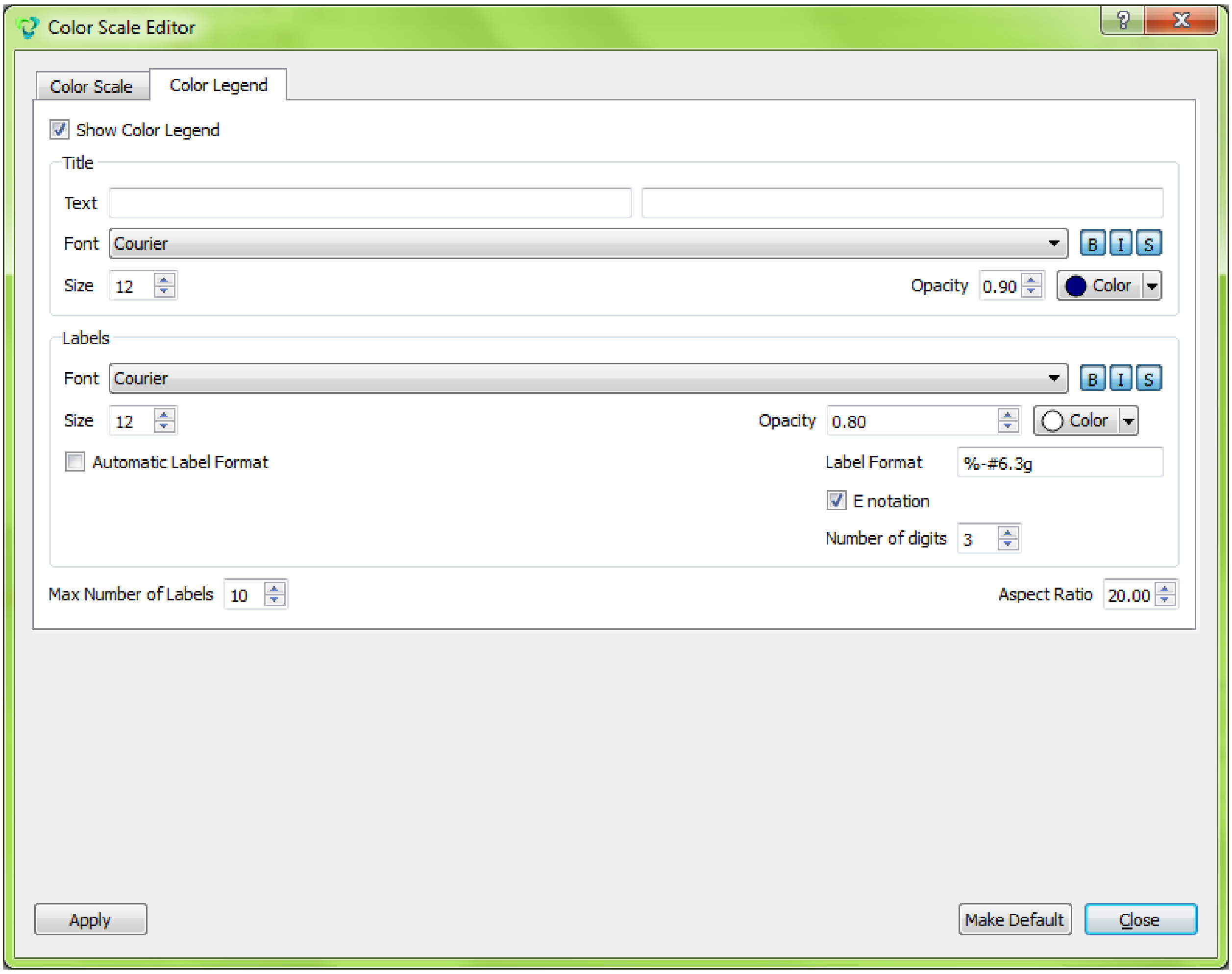


Display the legend.

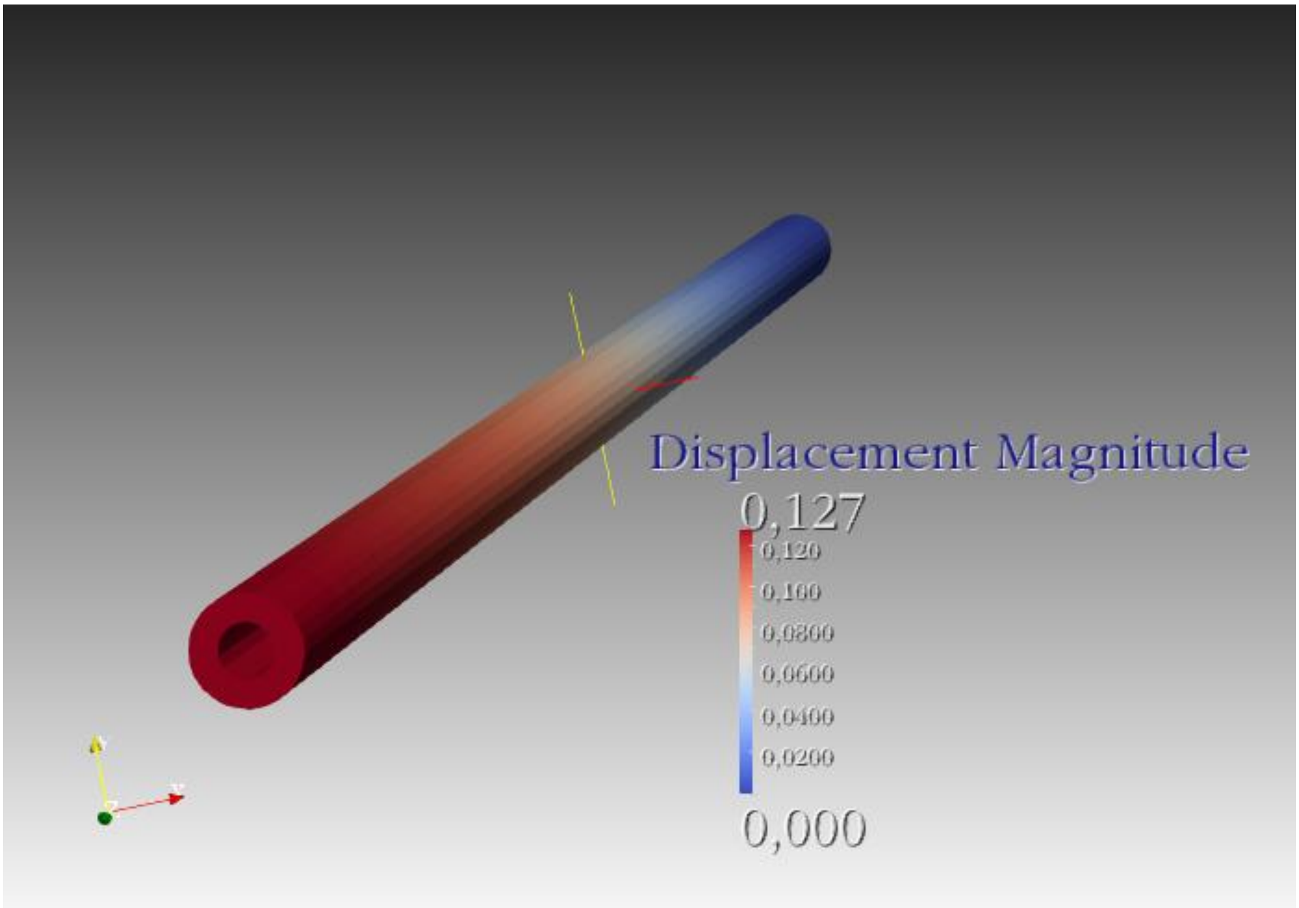


Next, go to the color scale settings by clicking the corresponding button near.

Specify the following settings in the pop-up form:



After applying the settings, you will see the following picture:



4. Check the maximum value U_{yy} at the selected point D.

   In the picture, it is the maximum in modulus Displacement (blue). It corresponds to -0.127222 in the color legend.

   The difference between the resulting value -0.127222 and the required -0.12524 is 1.56%.

5. Download numerical data.

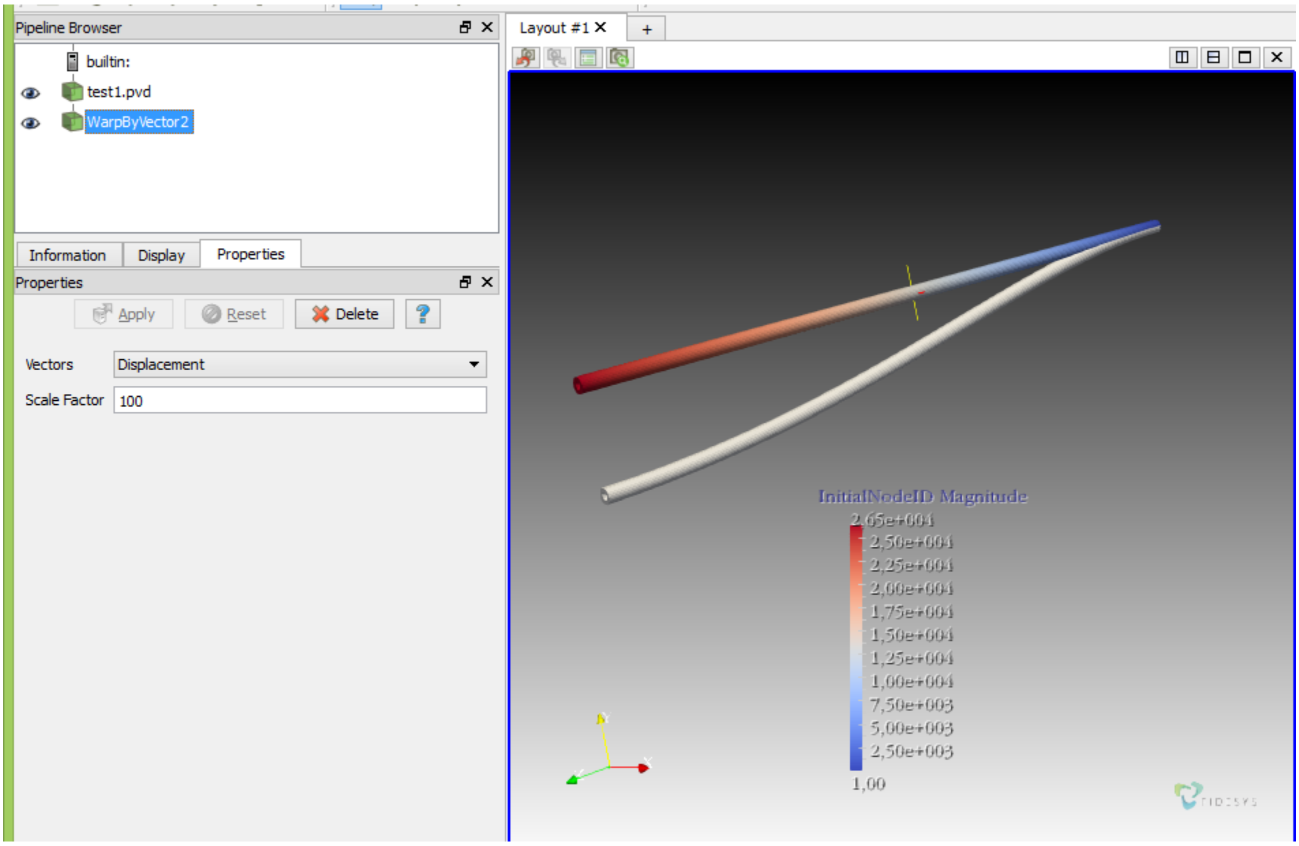
Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

6. You can see the way the body is deformed under the applied pressure.

To do this, select **Filters** → **Alpabatical** → **Warp By Vector.** Set the following parameters in the tab **Properties**::

- Vectors: Displacement;
- Scale Factor: 100.

As a result, the deformed body is displayed at the picture. To see the original model, click the button near the model in the Model Tree. The picture below shows the deformed (solid grey filling) and the original model (with the field of displacements distribution along Y axis).



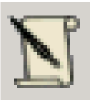
Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd)

```
reset
create Cylinder height 100 radius 1
create Cylinder height 100 radius 0.5
subtract body 2 from body 1
move Volume 1 preview z 50 include_merged
move Volume 1 z 50 include_merged
volume 1 size 0.25
volume 1 scheme Polyhedron
volume 1 scheme Polyhedron
mesh volume 1
create displacement on surface 8 dof all fix 0
create displacement on surface 9 dof 1 dof 3 fix 0
create acceleration on volume 1 dof 2 fix -386
undo group begin
create material "material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "material 1" scalar_properties "MODULUS" 3e+07 "POISSON" 0 "DENSITY"
0.00073
undo group end
set duplicate block elements off
block 1 volume 1
block 1 material 'material 1'
block 1 element type hex8
analysis type static elasticity dim3
```

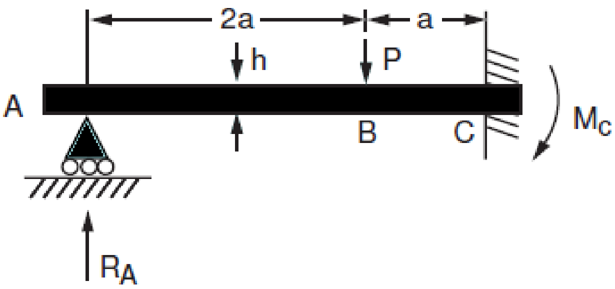
```
spectralelement off
usempi off
solver method auto try_other off
solver method auto try_other off
calculation start path " D:/FidesysBundle/calc/example.pvd"
```

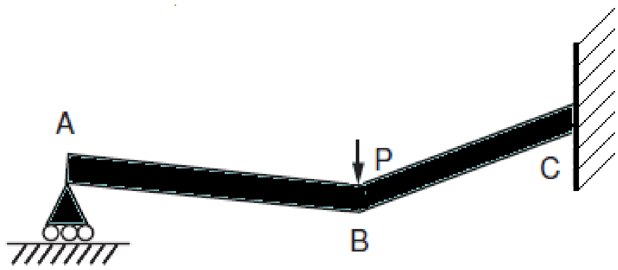


It is also possible to run the file *Example_2_Static_3D.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Static load (beam model, reaction forces)

S.H. Crandall, N.C. Dahl, An Introduction to the Mechanics of Solids, McGraw-Hill Book Co., Inc., New York, NY, 1959, pg. 389, ex. 8.9





The problem of static load of a square section beam is being solved. The picture represents a geometric model of the problem: a = 50 In, beam section 1 x 1 in. The boundary conditions are presented in the picture; the force applied at the point B is F_y = -1000 lb. The material parameters are E = 30e6 psi , v = 0.3.

Test pass criterion is the following: reaction force R_A at the point A (0,0,0) is 148.15 lb, reaction moment at the point C is 27778 in-lb within 1.5%.

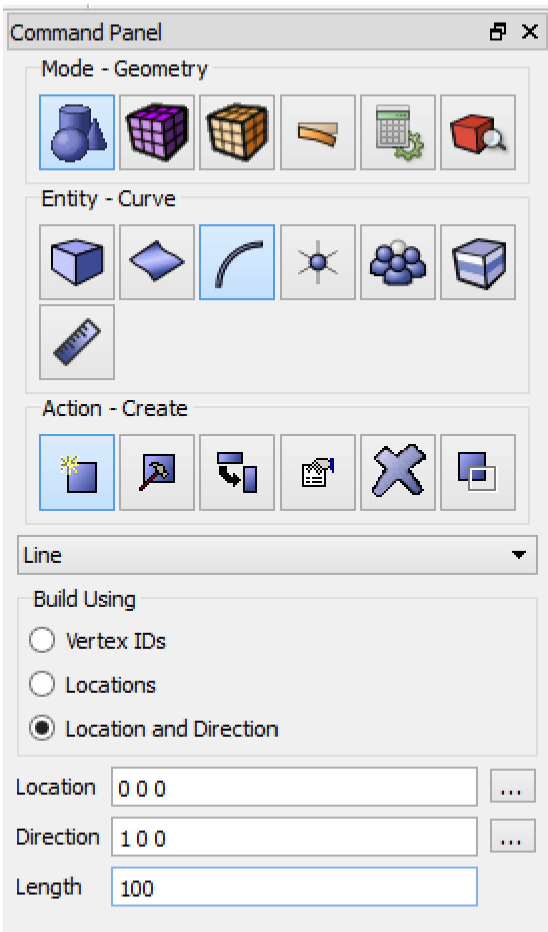
Geometry creation

1. Create a straight line 100 in length (segment AB).

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Curve**, Action — **Create**). Select **Line** in the list of geometric elements. Create it using **Location and Direction**. Set the following parameters:

- Location: 0 0 0 (*line origin*);
- Direction: 1 0 0 *(along X axis)*;
- Length: 100;

Click **Apply**.



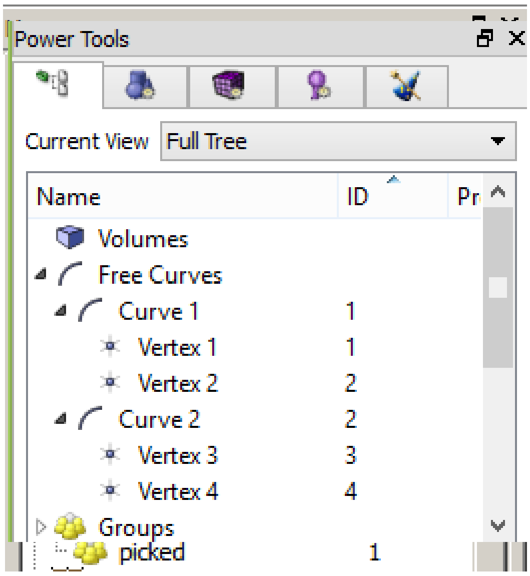
2. Create a straight line 50 in length (segment BC).

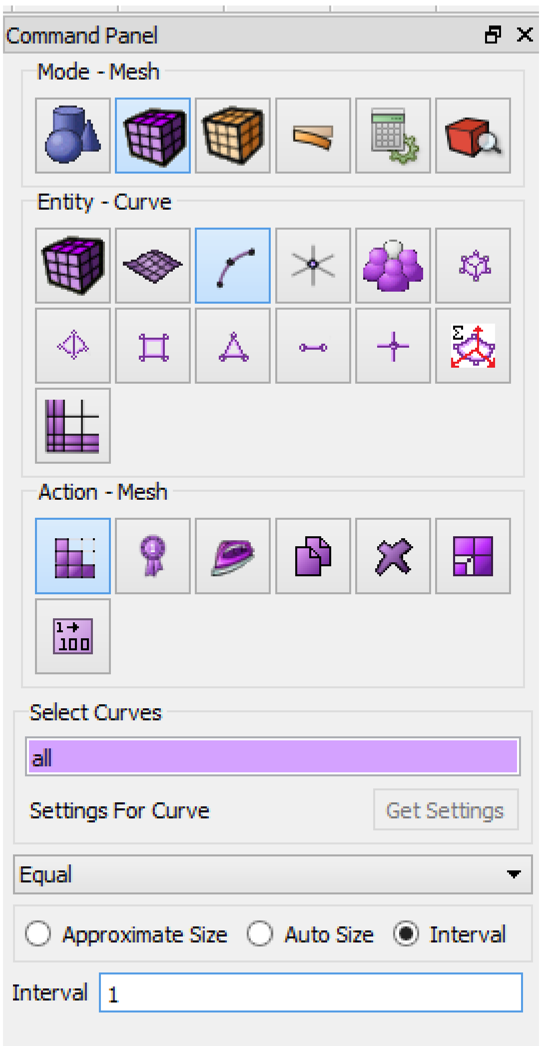
Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Curve**, Action — **Create**). Select **Line** in the list of geometric elements. Create it using **Location and Direction**. Set the following parameters:

- Location: 100 0 0 (*line origin*);
- Direction: 1 0 0 *(along X axis)*;
- Length: 50;

Click **Apply**.

As a result, in left side of the Model Tree there are two free curves having no common vertices.





3. Unite two vertices.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Vertex**, Action — **Merge**). Set the following parameters:

- Vertex ID: 2 3 (*using space after each of them*);

Click **Apply**.

Meshing

4. Select meshing on curves section on Command Panel (Mode — **Mesh,** Entity — **Curve**, Action — **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: all;
- Select the way of meshing: Equal;
- Select the meshing parameters: Interval;
- Interval: 1.

Click **Apply**.

Click **Mesh**.

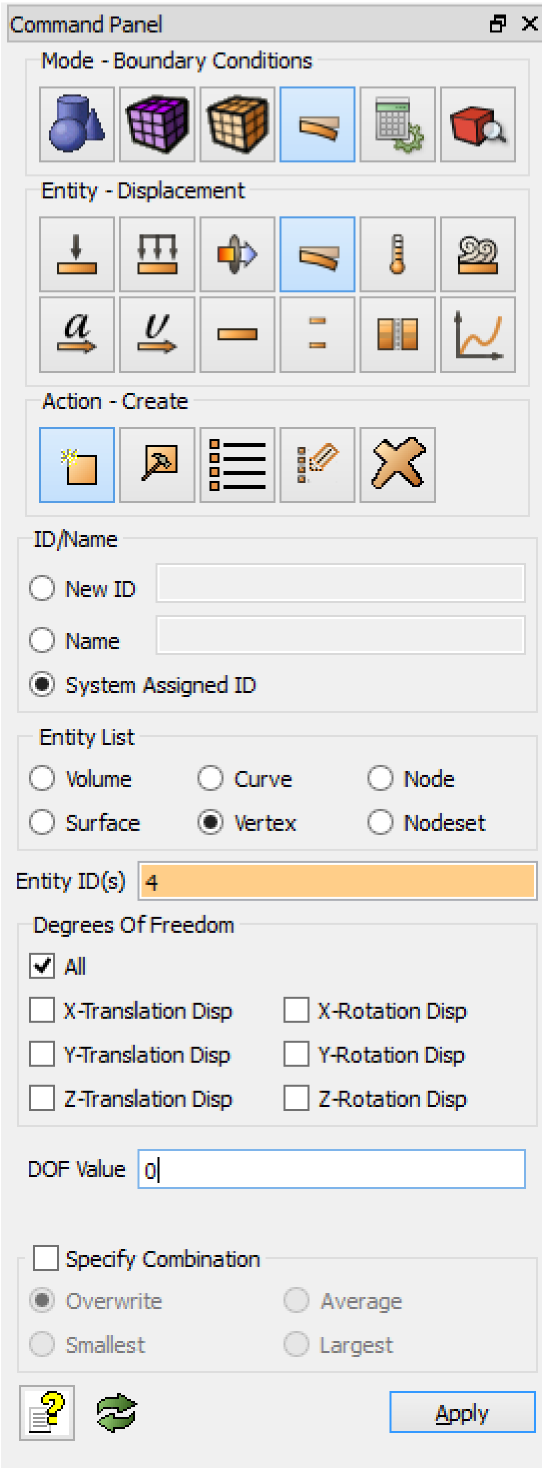
Setting boundary conditions

1. Fix the point C at all directions.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Vertex;
- Entity ID(s): 4;
- Degrees of Freedom: All;
- DOF Value: 0.

Click **Apply**.



2. Fix the point C at the Y and Z displacement.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Vertex;
- Entity ID(s): 1;
- Degrees of Freedom: Displacement Y, Displacement Z;
- DOF Value: 0.

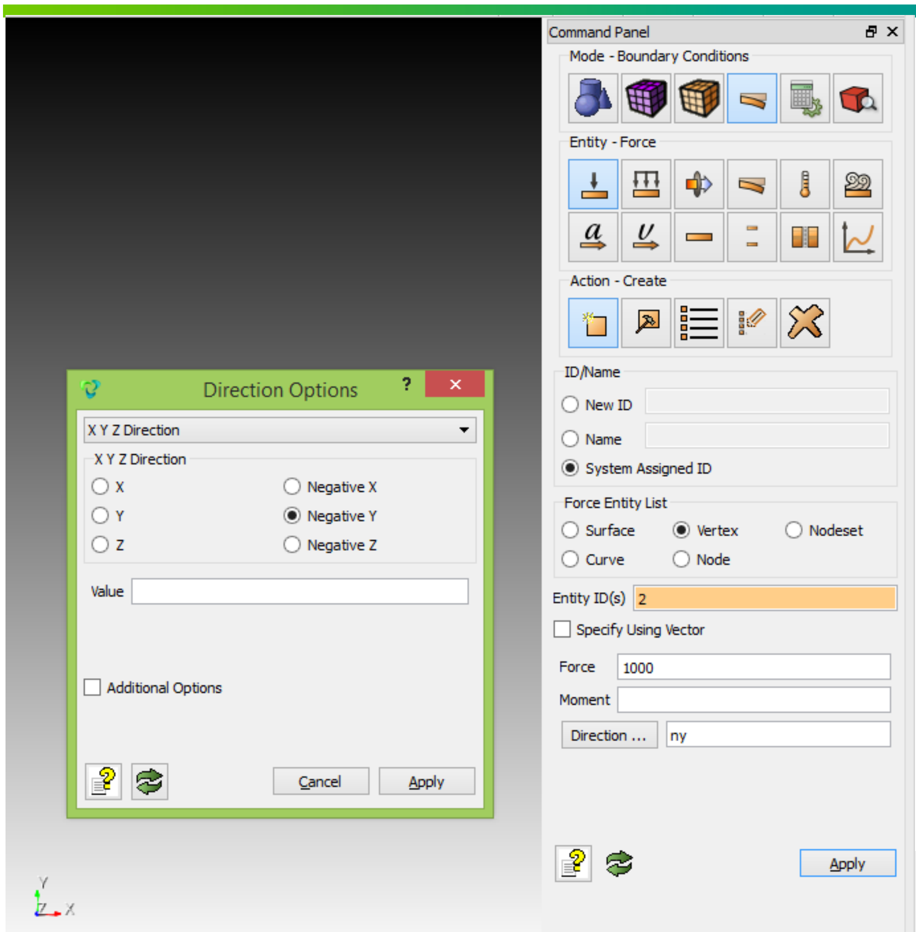
Click **Apply**.

3. Apply force at the point B.

Select Mode – **Boundary Conditions,** Entity – **Point Force**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Vertex;
- Entity ID(s): 2;
- Force: 1000;
- Click Direction…;
- In the pop-up window select: Negative Y.

Click **Apply**.



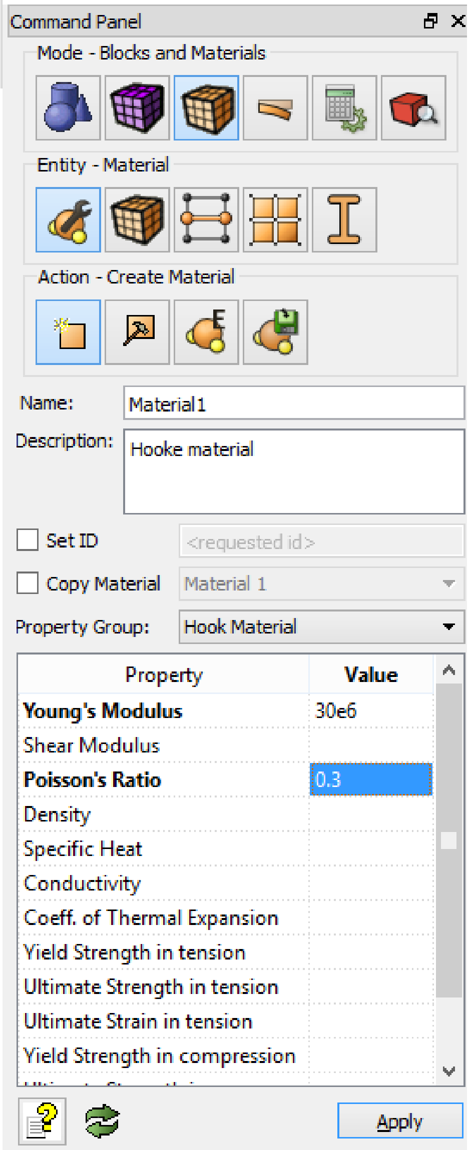
Setting material and element type

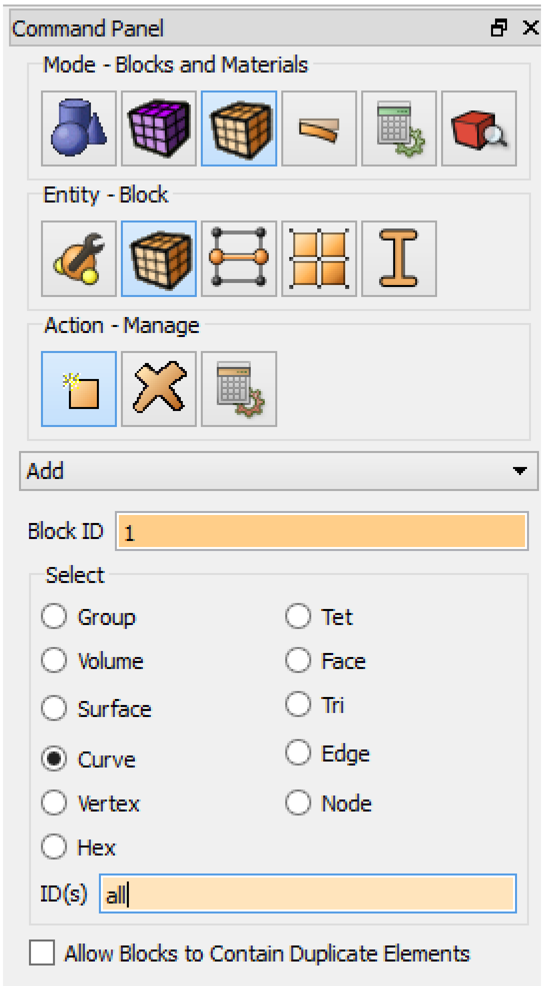
1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 30e6;
- Poisson's Ratio: 0.3

Click **Apply**.





2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Curve;
- ID: 1 2 *(or by the command **all**).*

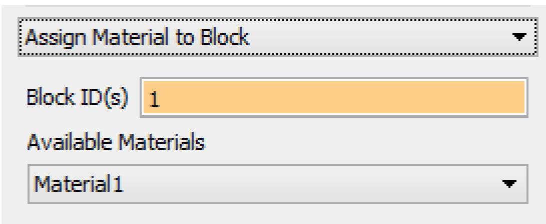
Click **Apply**.

3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

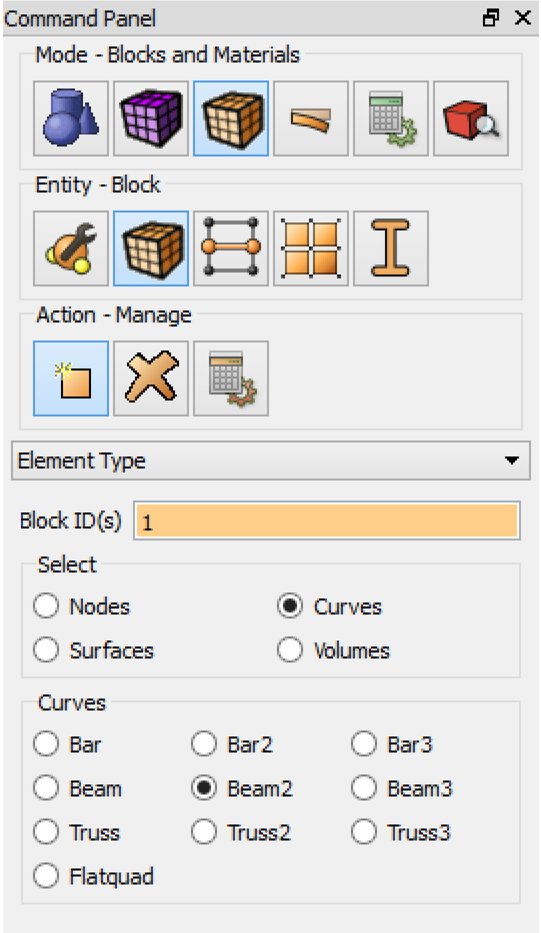


4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Curves;
- Element type: Beam2.

Click **Apply**.



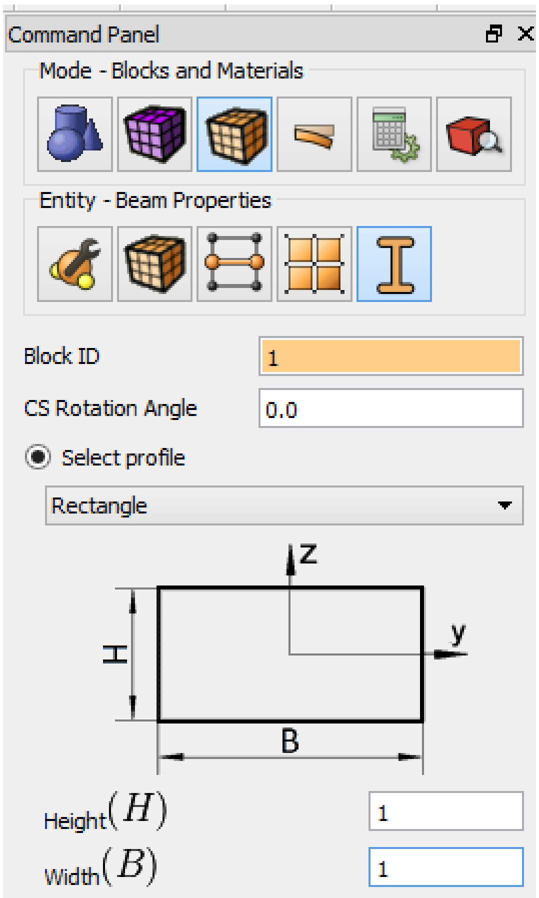
Setting beam cross section profile

5. Set beam parameters.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Beam Properties)**. Set the checkbox **Select profile**. Select **Rectangle** in the list of geometric elements. Specify the following parameters:

- Block ID: 1;
- Height (H): 1;
- Width (B): 1;

Click **Apply**.



Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **General**). Select:

- Dimension: 3D;
- Model: Elasticity.

Click **Apply**.

2. Set the solver settings.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply**.

3. Set the reaction force calculation

Go to the tab **Static** – **Output fields** and set the checkbox **Calculate nodal and reaction forces**.

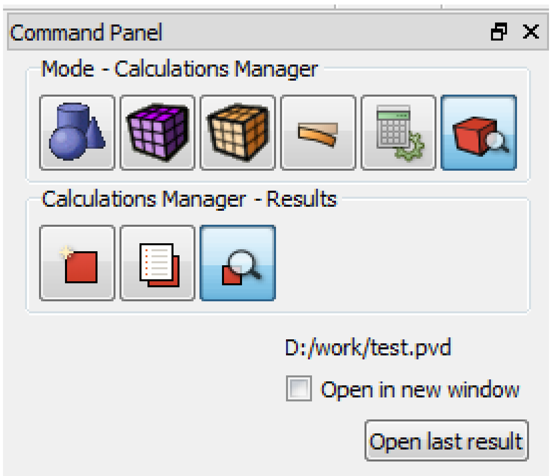
Click **Apply**.

Click **Start Calculation**.

Note: Without setting the checkbox **Calculate nodal and reaction forces,** the field is not calculated.

4. In a pop-up window select a folder to save the result and enter the file name.
5. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>".*

Results analysis

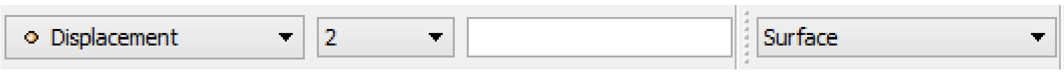


1. Open the file with the results. You can do this in one of the three ways.
   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (Mode — **Calculations Manager**, Calculations Manager — **Results**). Click **Open Results**.

2. Display the u_y component of the displacements field.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Displacement;
- Representation Component: 2.

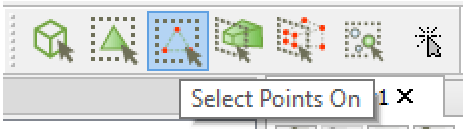


The field of displacements distribution along the Y axis wil be displayed on the model

3. Check the numerical value of the reaction force at the point A.

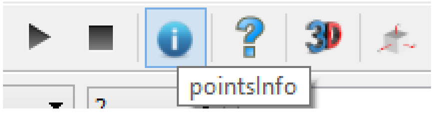
Display Component 2 of the Reaction Forces field.

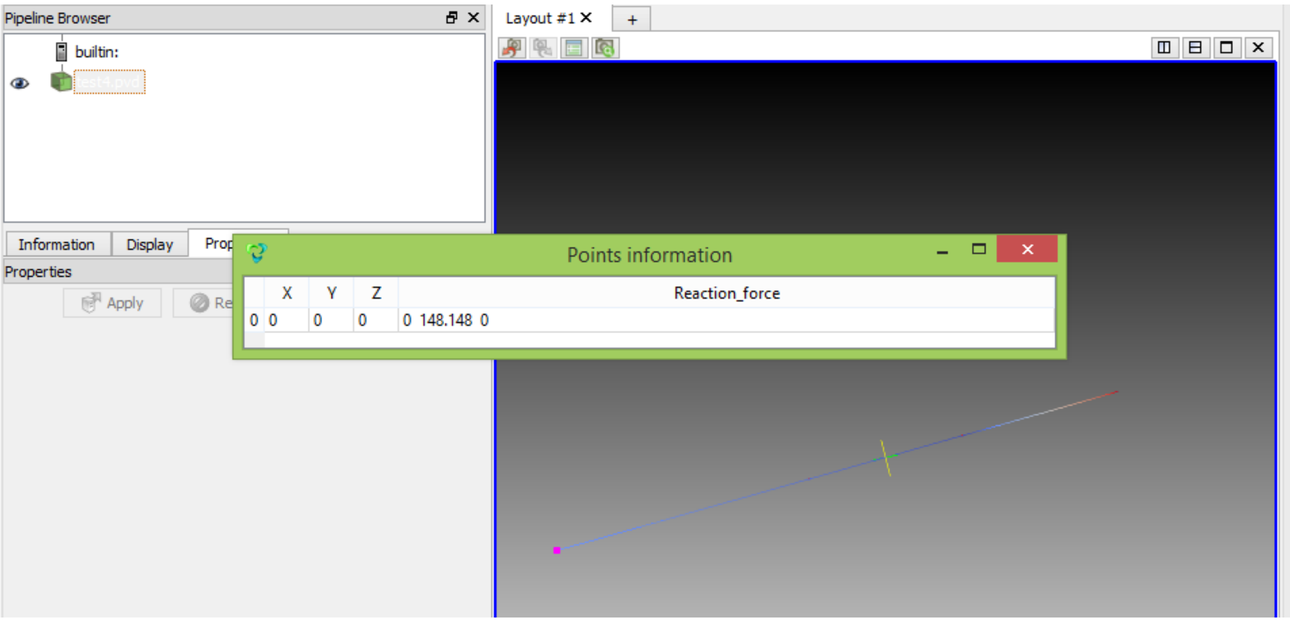
On the ***Fidesys Viewer*** Main Panel, click Select Points On surface.



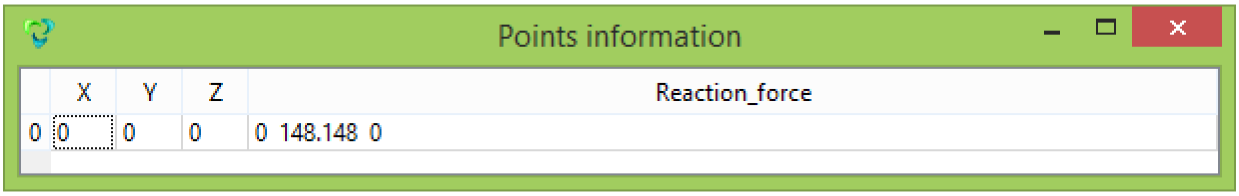
Select the limiting left point (point A) on the geometric model.

To quickly view the information at the fixed point, click **pointsInfo** on the Main Panel.





In the pop-up window, components of the reaction force at the selected point will be displayed.



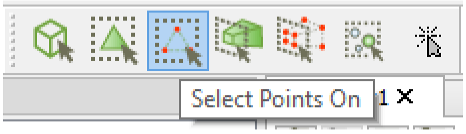
The difference between the resulting value 148.148 and the required 148.15 is less than 0.01%.

Please, do not close the window Points information.

4. Check the numerical value of reaction moments at the point C.

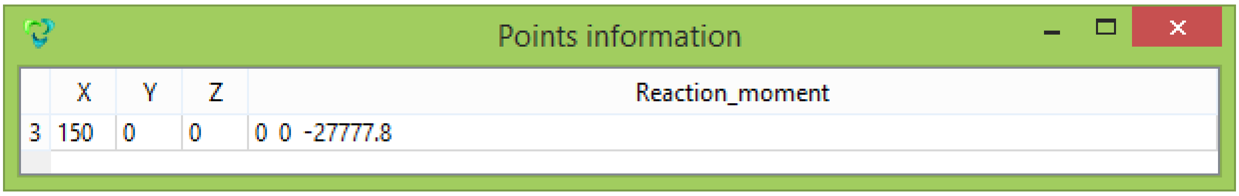
Display Component 3 of the Reaction_moment field.

On the ***Fidesys Viewer*** Main Panel, click Select Points On surface.



Select the limiting right point C on the geometric model.

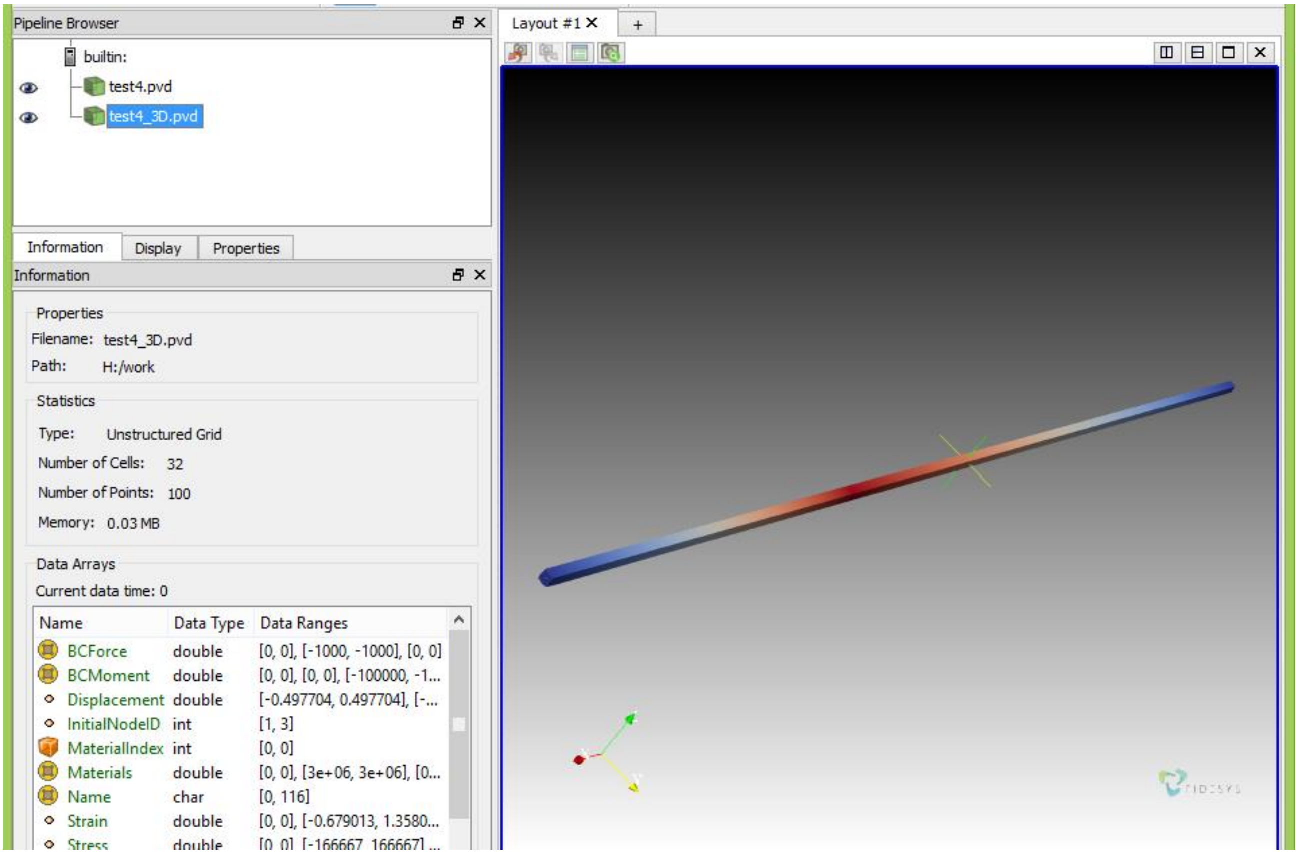
In the window Points information components of the reaction moment at the selected point will be displayed.



The difference between the resulting value -27777.8 and the required -27778 is less than 0.01%.

5. Open 3D-image of the beam.

To display 3D-view of the beam cross section, set the focus on the calculation title and click the button  in the Fidesys Viewer standard line.



 To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

6. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example3.pvd):

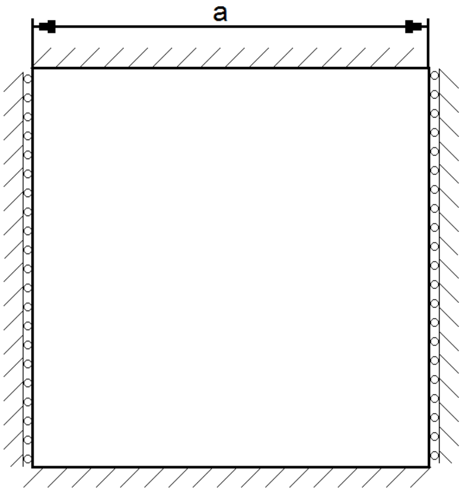
```
reset
set node constraint on
create curve location 0 0 0 direction 1 0 0 length 100
create curve location 100 0 0 direction 1 0 0 length 50
merge vertex 2 3
curve all interval 1
curve all scheme equal
curve all interval 1
curve all scheme equal
mesh curve all
create displacement on vertex 4 dof all fix 0
create displacement on vertex 1 dof 2 dof 3 fix 0
create force on vertex 2 force value 1000 direction ny
set duplicate block elements off
block 1 curve all
undo group begin
create material "Material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "Material 1" scalar_properties "MODULUS" 3e+06 "POISSON" 0.3
undo group end
block 1 material 'Material 1'
block 1 element type beam2
block 1 attribute count 14
block 1 attribute index 1 value 1 name 'A'
block 1 attribute index 2 value 0.140833 name 'It'
block 1 attribute index 3 value 0.166667 name 'Ix'
block 1 attribute index 4 value 0.0833333 name 'Iy'
block 1 attribute index 5 value 0 name 'Iyz'
block 1 attribute index 6 value 0.0833333 name 'Iz'
block 1 attribute index 7 value 0 name 'angle'
block 1 attribute index 8 value 0 name 'ey'
block 1 attribute index 9 value 0 name 'ez'
block 1 attribute index 10 value 0.5 name 'max_y'
block 1 attribute index 11 value 0.5 name 'max_z'
block 1 attribute index 12 value 0 name 'section_type'
block 1 attribute index 13 value 1 name 'geom_H'
block 1 attribute index 14 value 1 name 'geom_B'
analysis type static elasticity dim3
spectralelement off
usempi off
solver method auto try_other off
output nodalforce on midresults on
output nodalforce on midresults on
calculation start path 'D:/FidesysBundle/calc/example3.pvd'
```

It is also possible to run the file *Example_3_Static_Beam.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Static load (shell)

Тимошенко С.П. Войновский-Кригер С. Пластинки и оболочки – М.: Наука, 1966 г. – 636стр.

[Timoshenko S.P. Voynovskiy-Kriger S. Plates and shells, Nauka, Moscow, 1966, 636 pages [in Russian]]



The problem of static load of square shell which two sides are clamped and the other two are freely supported, is being solved.. The picture represents a geometric model of the problem: a = 1 m, shell thickness is 0.1 m. The boundary conditions are presented in the picture. The plate is loaded by uniform pressure of 10 kPa.

Test pass criterion is the following: the maximum deflection is 1.19e-6, moments M_x=252 N·m and M_y=332 N·m.

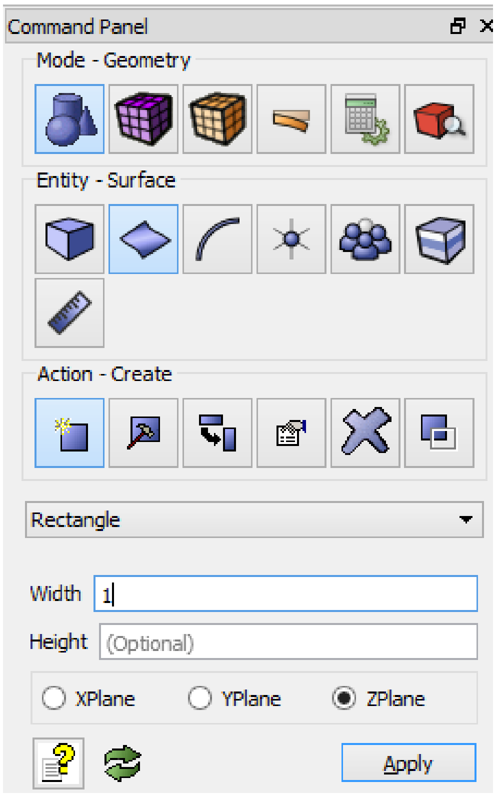
Geometry creation

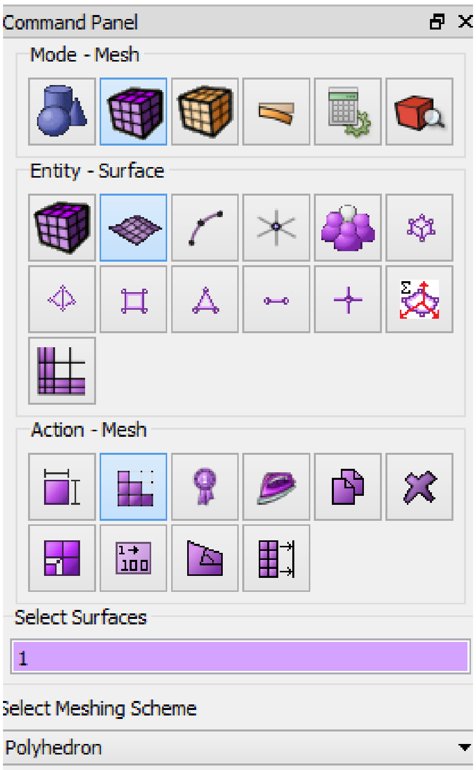
1. Create the square 1 m on side.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Create**). Select **Rectangle** in the list of geometric elements. Set the parameters:

- Width: 1;
- Height: Optional.

Click **Apply**.





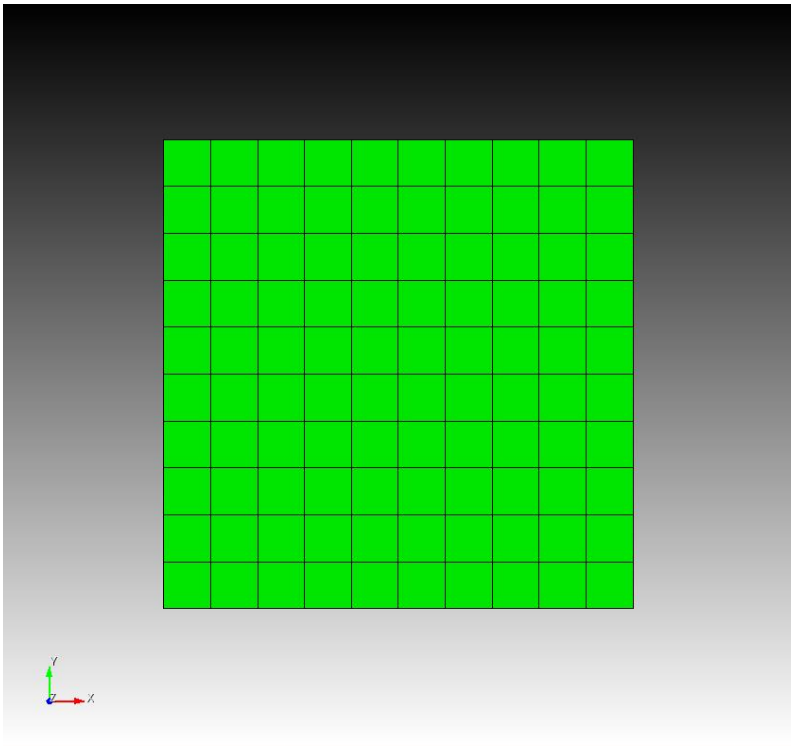
Meshing

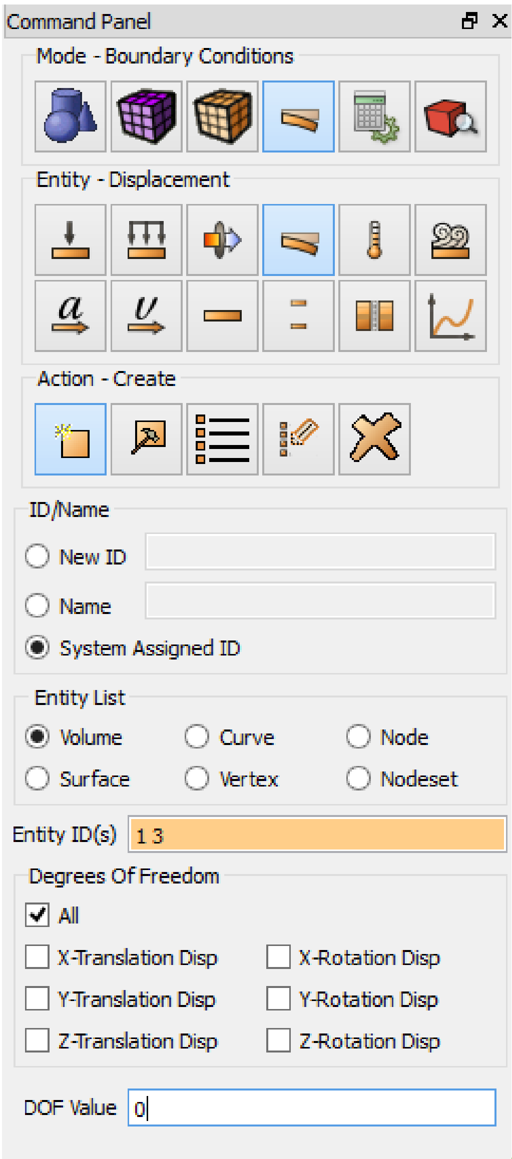
1. Select surface mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Meshing**). Specify the following parameters:

- Select surfaces: 1;
- Select meshing scheme: Polyhedron;

Click **Apply Scheme**.

Click **Mesh**.





Setting boundary conditions

1. Fix the two edges rigidly.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 1 3 *(or sequentially click on the top and bottom edges)*;
- Degrees of Freedom: All;
- DOF Value: 0.

Click **Apply**.

2. Fix the two other edges at displacements.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 2 4 *(or sequentially click on the right and left edges)*;
- Degrees of Freedom: X-Translation, Y-Translation, Z-Translation;
- DOF Value: 0.

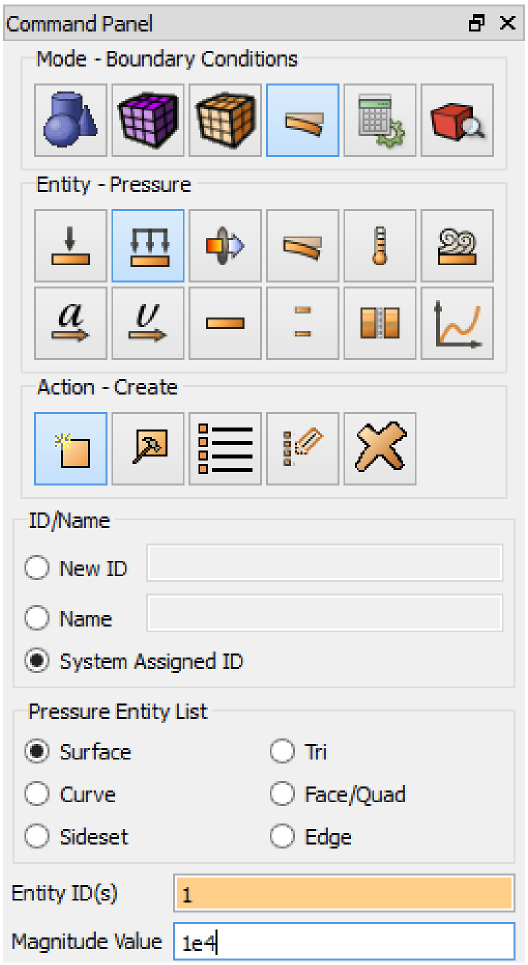
Click **Apply**.

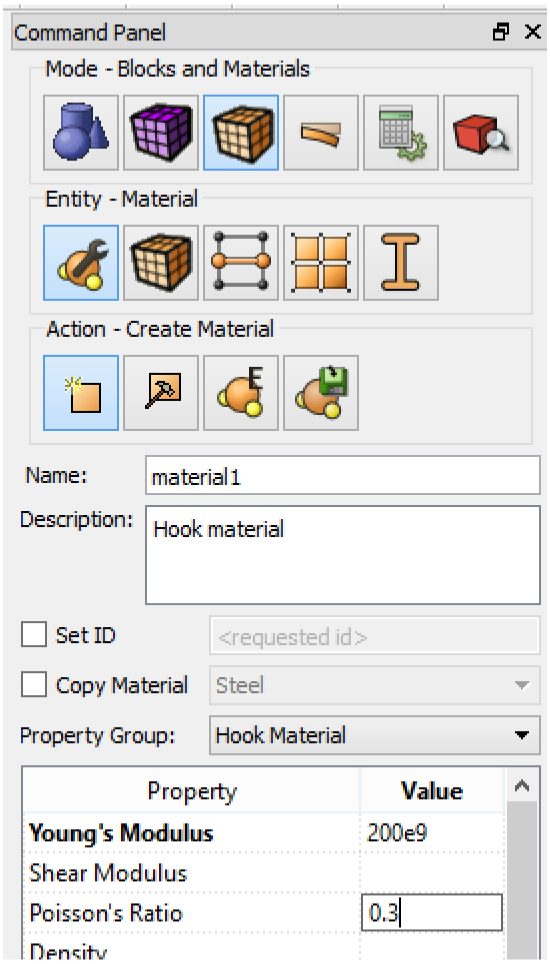
3. Apply the uniform pressure on the surface.

Select Mode – **Boundary Conditions,** Entity – **Pressure**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 1;
- Value: 1e4;

Click **Apply**.





Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 200e9;
- Poisson's Ratio: 0.3.

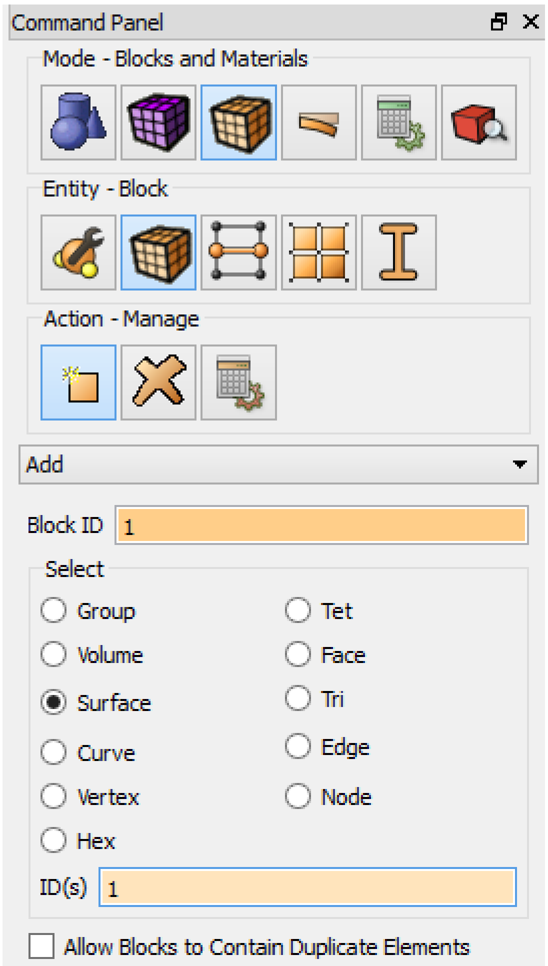
Click **Apply**.

2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Surface;
- ID: 1 *(or by the command **all**).*

Click **Apply**.

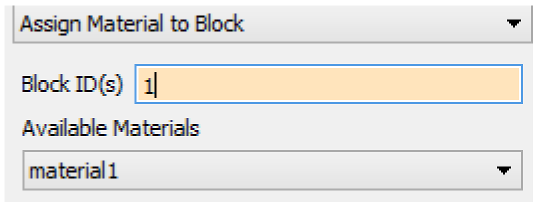


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

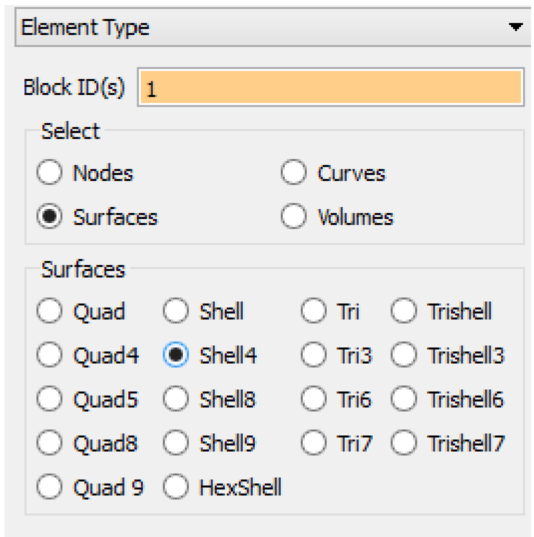


4. Assign the element type.

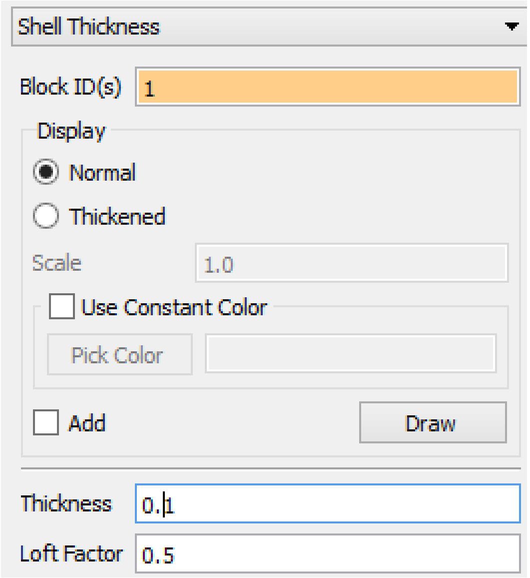
Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Surfaces;
- Element type: Shell4.

Click **Apply**.



Setting shell thickness



1. Set the shell thickness.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action – **Manage)**. Select in the list of possible operations **Shell thickness**. Set the following parameters:

- Block ID: 1;
- Thickness: 0.1;
- Loft Factor: 0.5;

Click **Apply**.

Starting calculation

2. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **General**). Select:

- Dimension: 3D;
- Model: Elasticity.

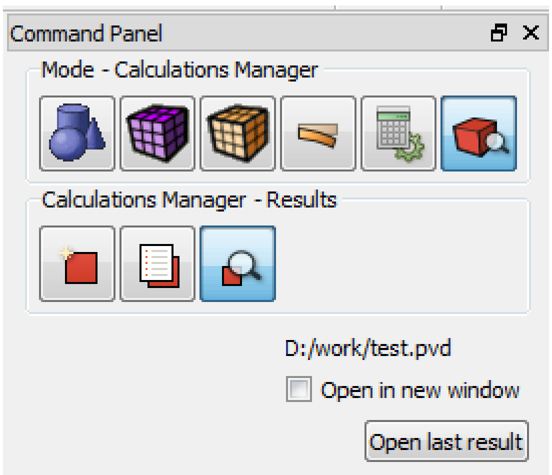
Click **Apply**.

3. Set the solver settings.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default. Click **Apply**.

4. Set the reaction force calculation

Go to the tab Static – **Output fields** and set the checkbox **Calculate nodal and reaction forces**.



Click **Apply**.

Click **Start Calculation**.

> **Note:** Without setting the checkbox **Calculate nodal and reaction forces,** the field is not calculated.

5. In a pop-up window select a folder to save the result and enter the file name.
6. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis

1. Open the file with the results. You can do this in one of the three ways.

- Click Ctrl+E.
- Select Calculation → Open Results in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.

2. Display the u_z component of the displacement field.

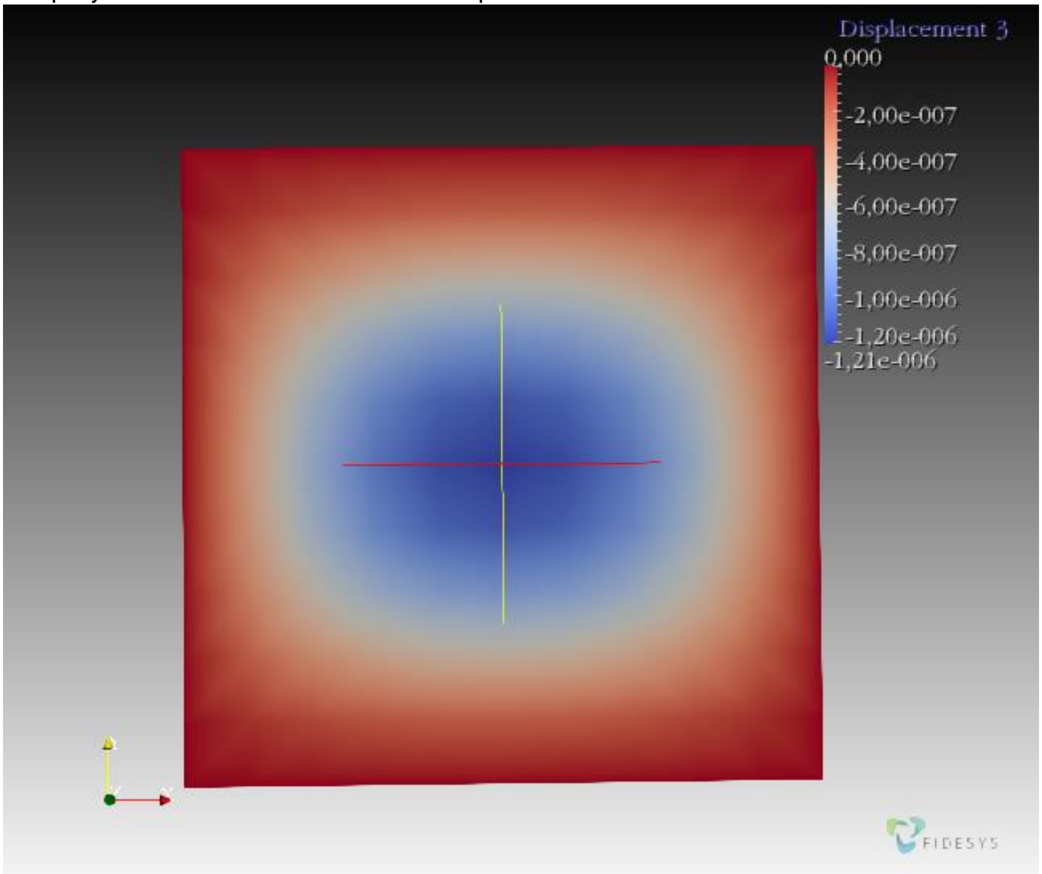
In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Displacement;
- Representation Component: 3.

The field of displacements distribution along the Z axis will be displayed on the model

3. Check the numerical value of the maximum displacement.

Display maximum Component 3 of the Displacement field.



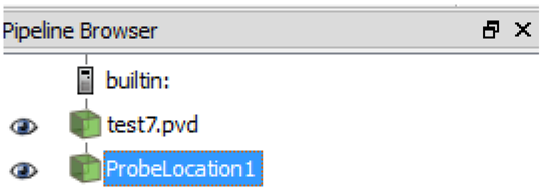
The difference between the resulting value -1.21e-6 and the required -1.19e-6 is 1.7%

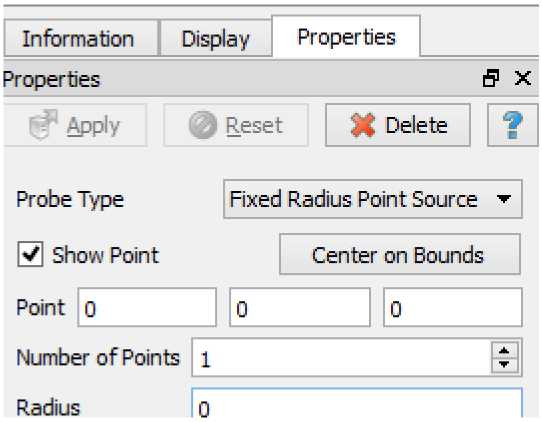
4. Check numeric values of moments in the center of the plate.

Display component 11 of the MomentsShell field.

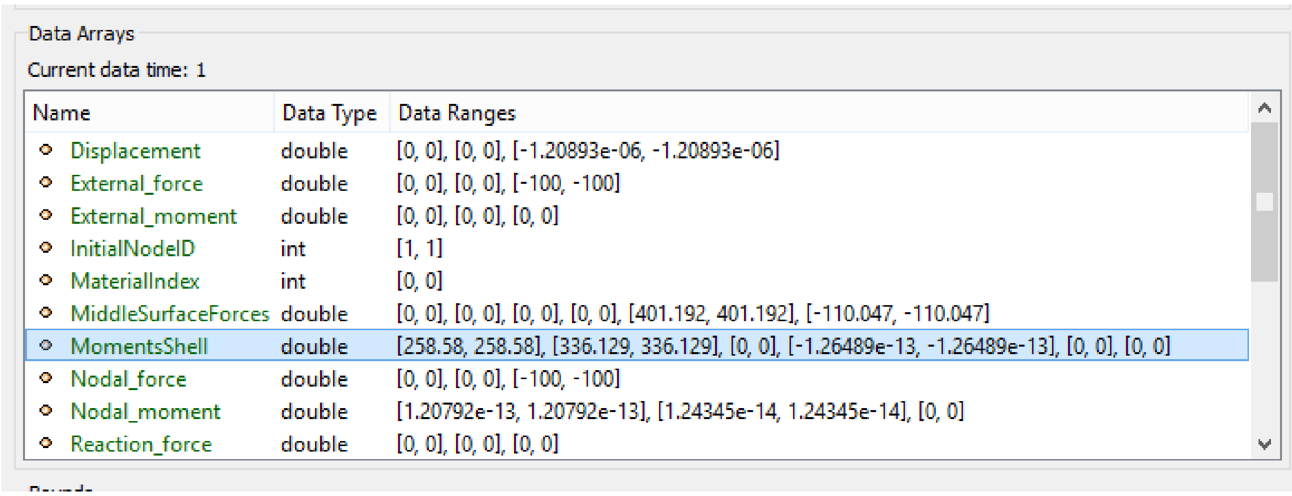
Select the filter **Probe Location** (Filters – Alphabetical – Probe Location) in the ***Fidesys Viewer*** Main Menu. In the tab **Properties** set the following values:

- Point: (0,0,0);
- Number of Points: 1;
- Radius: 0.



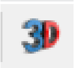


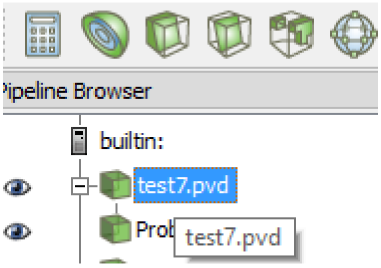
Go to the **Information** tab and look at the MomentsShell field.



The difference between the resulting values (M_x=258.58 and M_y=336.129) and the required (M_x=252 and M_y=332) is 2.6% and 1.2%, relatively.

5. Open 3D-image of the shell.

To display 3D-view of the beam cross section, set the focus on the calculation title and click the button  in the Fidesys Viewer standard line.



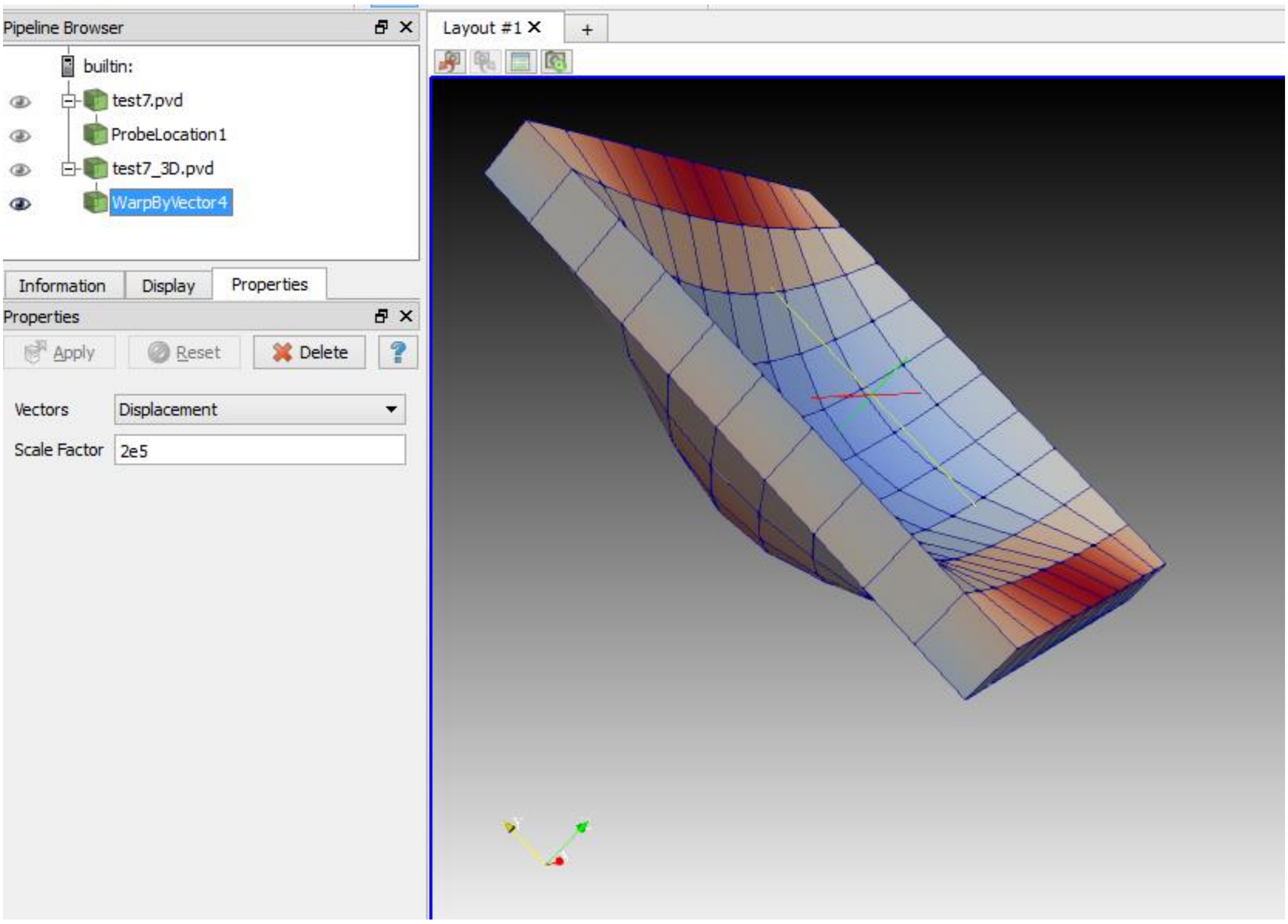
A new file example 4_3D.pvd will be opened and you will be able to apply various filters to it and to view its deformed view.

Choose the new file example_3D.pvd in the Model Tree and display Filters – Alphabetical – **Warp by Vector** for it with the following fields values

- **Vectors:** Displacement
- **Scale Factor:** 2e5

On the Toolbar, set once again the following parameters for the deformed type:

The first buckling mode will be displayed on the screen but the shell will be enveloped with thickness.



 To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

6. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK.** The saved file is an ordinary table of numerical data which can be opened in any text editor.

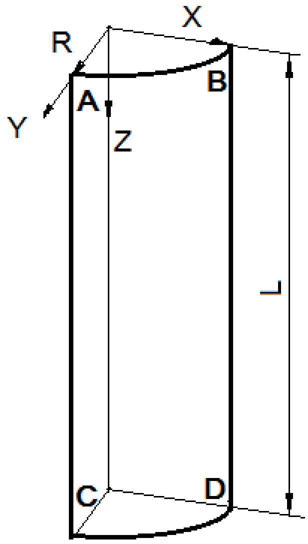
Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
create surface rectangle width 1 zplane
surface 1 scheme Polyhedron
surface 1 scheme Polyhedron
mesh surface 1
create displacement on curve 1 3 dof all fix 0
create displacement on curve 2 4 dof 1 dof 2 dof 3 fix 0
create pressure on surface 1 magnitude 1e4
undo group begin
create material "Material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "Material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3
undo group end
set duplicate block elements off
block 1 surface 1
block 1 material 'Material 1'
undo group begin
block 1 attribute count 2
block 1 attribute index 1 value 0.1
block 1 attribute index 2 value 0.5
undo group end
analysis type static elasticity dim3
spectralelement off
usempi off
solver method auto try_other off
output nodalforce on midresults on
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

It is also possible to run the file *Example_4_Static_Shell.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Hydrostatic pressure on cylinder (setting boundary conditions according to coordinates)



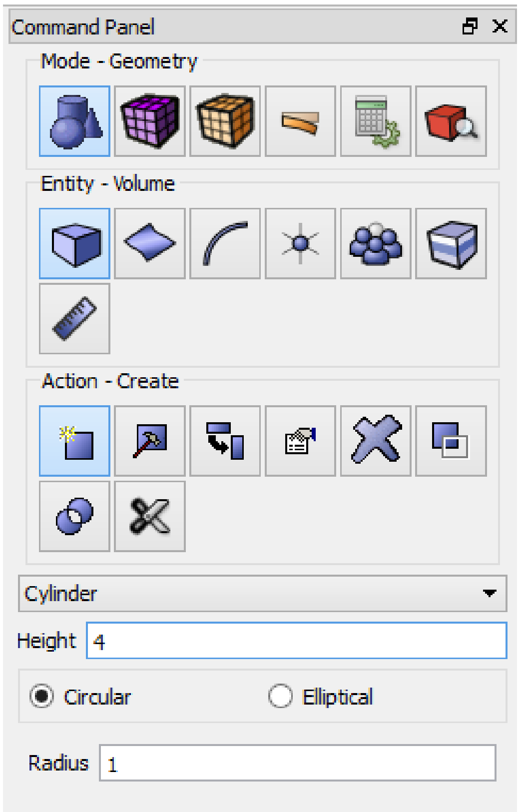
Societe Francaise des Mecaniciens, Guide de validation des progiciels de calcul de structures, (Paris, Afnor Technique,1990.) Test No. SSLS08/89. I-Deas Model Solution Verification Manual

The problem of hydrostatic load of the cylindrical shell is being solved. The picture represents a geometric model of the problem: radius 1 m, shell thickness 0.02 m. The shell is fixed on the condition of the symmetry. The plate is loaded by the pressure p = 20000·z/L Pa.

Test pass criterion is the following: displacement u_z at the point (0, R, L) is $2.86{\cdot}10^{-6}$ m.

Geometry creation

1. Create the cylinder of 1 m radius and 4 m high.



Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 4;
- Cross section: Circular;
- Radius: 1.

Click **Apply**.

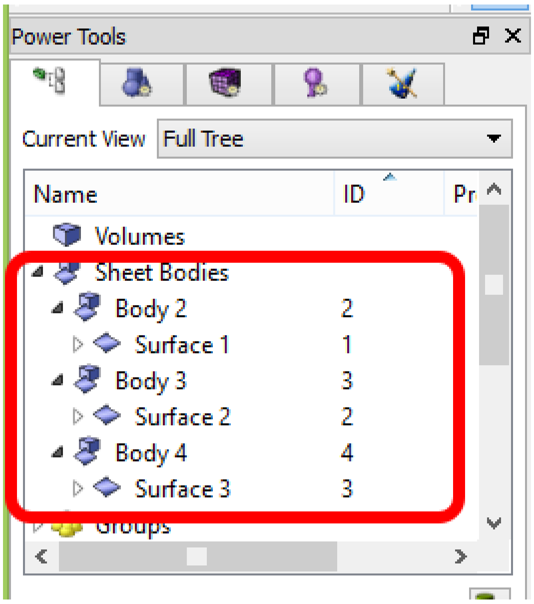
2. Get the cylindrical shell out of the volumeric cylinder.

Select the volume removing section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Delete**). Enter the number of the created volume – 1 into the field **Volume ID(s)**. Put a tick against **Keep lower geometry**.

Click **Apply**.

As a result, three plane bodies (Body 1, Body 2, Body 3) are obtained.

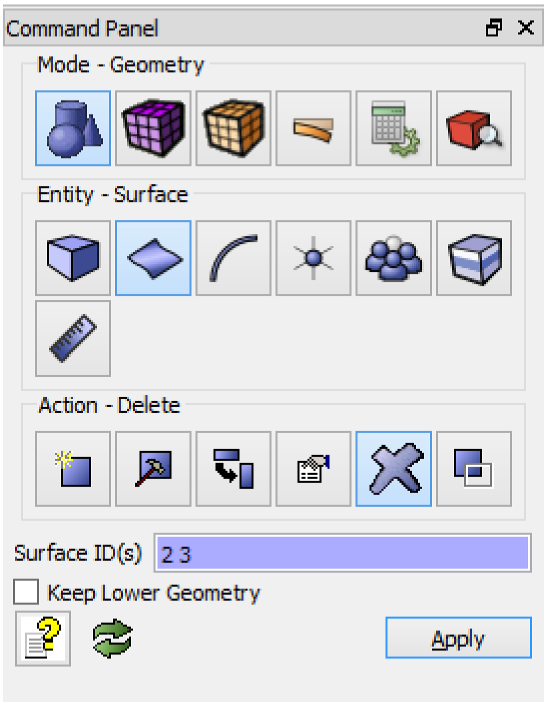
This will be displayed in the Model Tree.



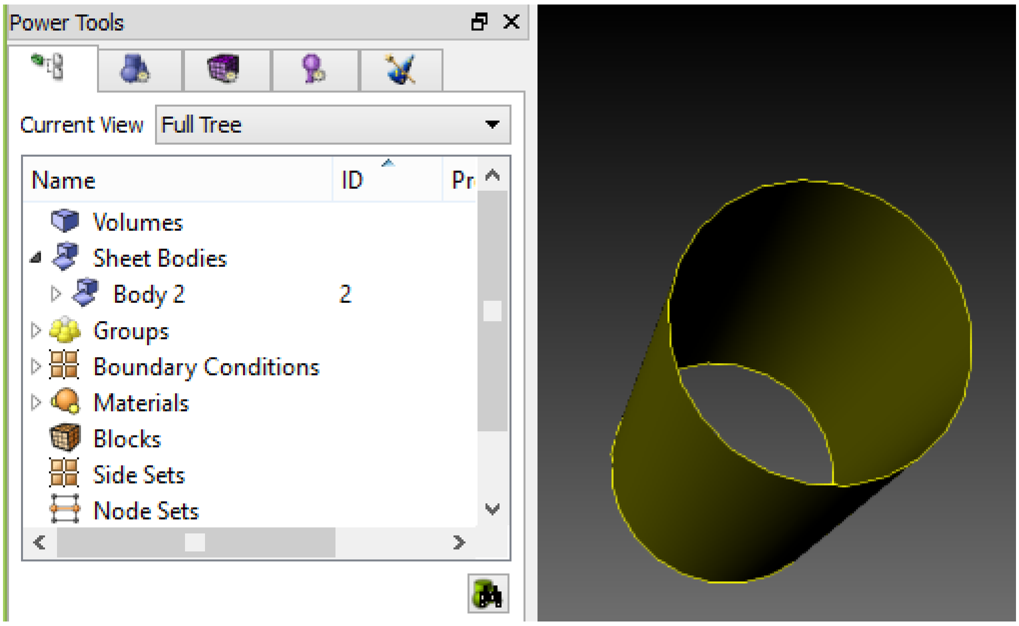
3. Delete side surfaces Surface 2 and Surface 3.

Select the surface removing section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Delete**). Enter numbers – 2 3 in the window **Surface ID(s)**.

Click **Apply**.



As a result, only the lateral cylindrical shell of 1 m radius and 4 m high will remain of the initial volume.



4. Leave a quarter of a shell (symmetry of the problem).

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Plane** in the list of possible webcut types. Set the following parameters:

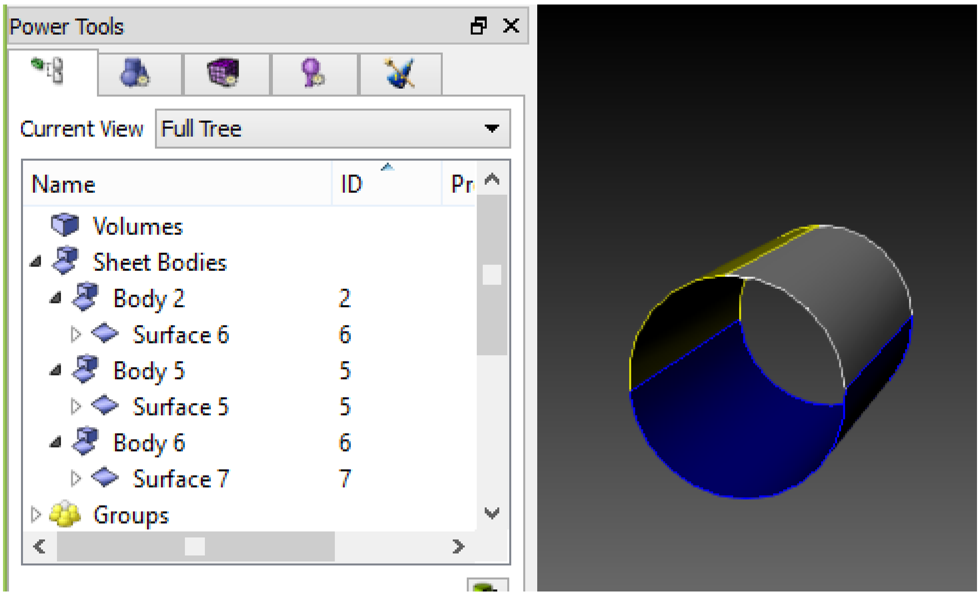
- Body ID: 2 (*the body to be webcut*);
- Webcut with: YZ Plane;
- Offset value: 0.

Click **Apply**.

Do the same for the ZX Plane.

- Body ID: 2 (the body to be webcut);
- Webcut with: ZX Plane;
- Offset value: 0.

Click **Apply**.

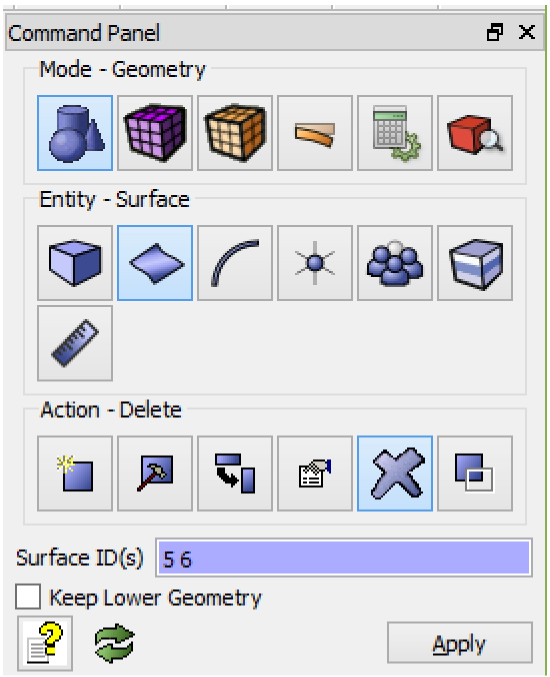


As a result, the original Body 2 in the Model Tree is split into three (Body 2, Body 5 and Body 6).

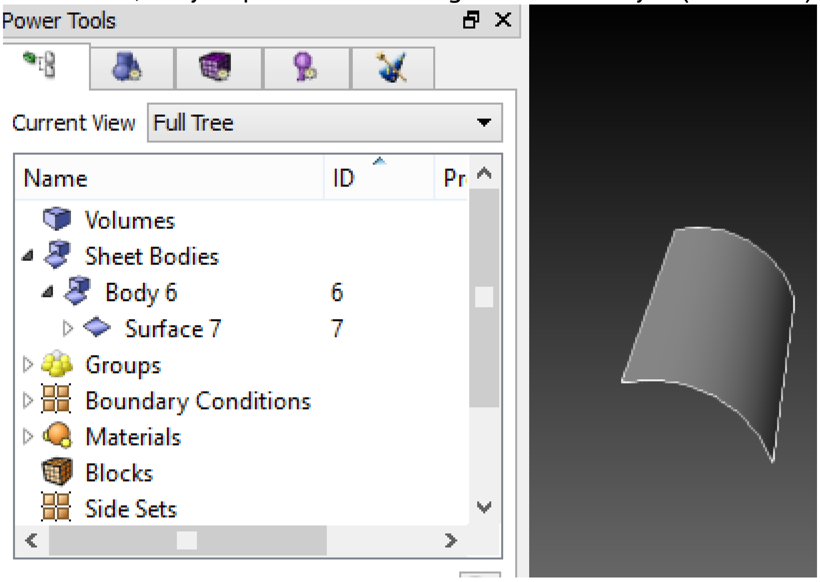
5. Delete surfaces Surface 5 and Surface 6.

Select the surface removing section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Delete**). Enter numbers – 5 6 into the window **Surface ID(s).**

Click **Apply**.



As a result, only a quarter of the original shell Body 6 (Surface 7) is left.

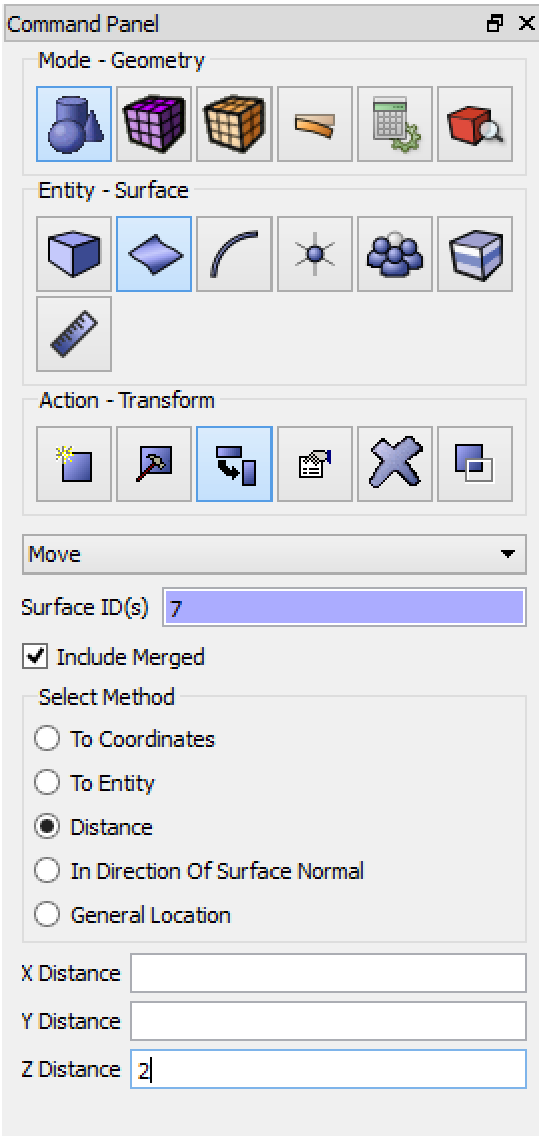


6. Move the surface to the coordinate origin.

Select surface geometry modification section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Modify**). Select **Move** in the list of possible webcut types. Set the following parameters:

- Surface: 7 (*the surface to be moved*);
- Checkbox Distance;
- Z Distance: 2.

Click **Apply**.



Meshing

1. Specify the parameters of mesh refinement.

Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**).

Split the cross-cut curves Surface 17 and Surface 18 into 10 elements.

- Select Curves: 17 18 (*or click the mouse while holding down the Ctrl key on contour of the cross-cut curves*);
- Select the way of meshing: Equal;
- Select splitting settings: Interval;
- Specify interval number: 10.

Click **Apply**.

Split longitudinal curves Curve 5 and Curve 16 into 20 elements.

- Select Curves: 5 16 (*or click the mouse while holding down the Ctrl key on contour of the longitudinal curves*);
- Select the way of meshing: Equal;
- Select splitting settings: Interval;
- Specify interval number: 20.

Click **Apply**.

2. Create the mesh.

Select the surface mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Intervals**).

- Select Surfaces to Mesh (specify their ID)): 7 (or by the command *all*);
- Select meshing scheme: Automatically Calculate.

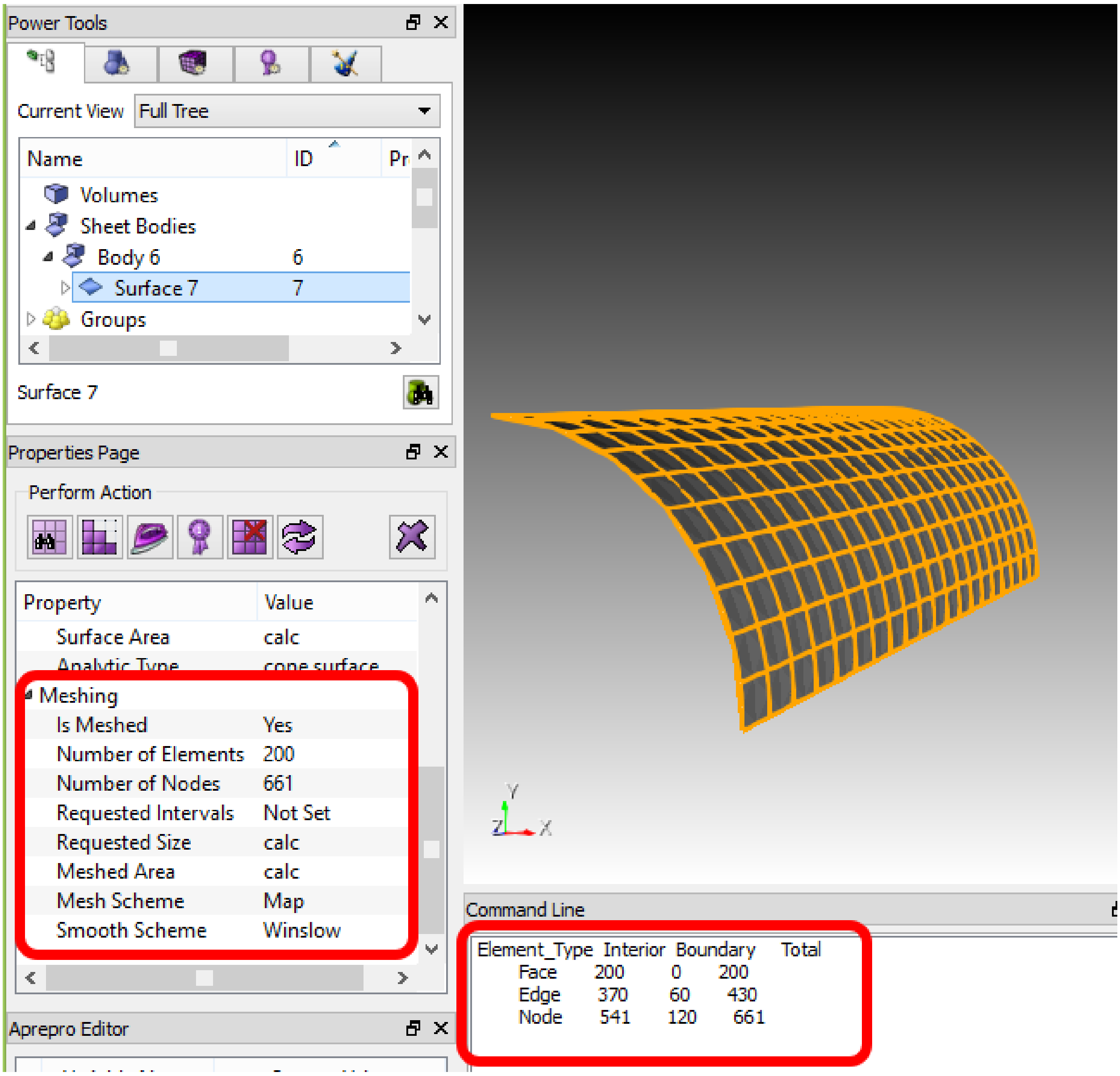
Click **Apply**.

Click **Mesh**.

The resulting number of elements can be viewed in the Property Page by clicking on the inscription Surface 7 in the Model Tree on the left.

To view the mesh properties, you can follow these steps:

- Select the entire model
- Right-click on the model
- In the pop-up menu, select List Information – List Mesh Info
- Information on the mesh will be displayed in Command Line


Power Tools
Current View Full Tree
Name ID Pr
Volumes
Sheet Bodies
Body 6 6
Surface 7 7
Groups
Surface 7
Properties Page
Perform Action
Property Value
Surface Area calc
Analytic Type cone surface
Meshing
Is Meshed Yes
Number of Elements 200
Number of Nodes 661
Requested Intervals Not Set
Requested Size calc
Meshed Area calc
Mesh Scheme Map
Smooth Scheme Winslow
Aprepro Editor
Command Line
Element_Type Interior Boundary Total
Face 200 0 200
Edge 370 60 430
Node 541 120 661

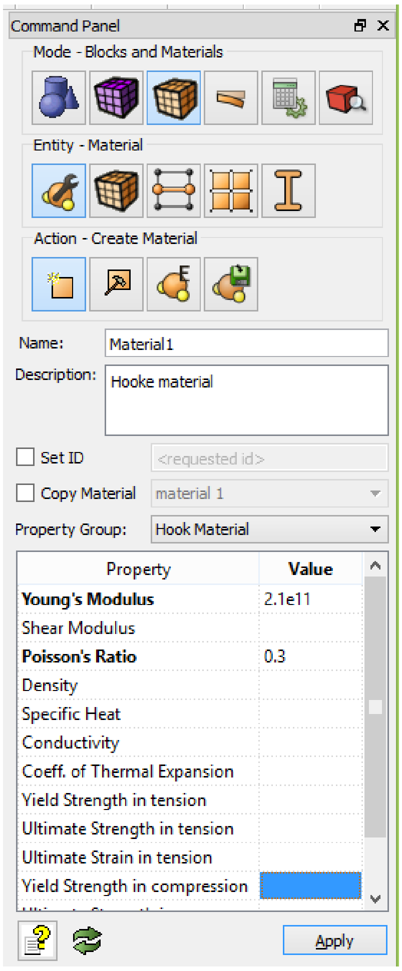
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Material**, Action — **Create Material**). Set the following parameters:

- Name: Material1;
- Property group: Hook material;
- Young's Modulus: 2.1e11;
- Poisson's Ratio: 0.3.

Click **Apply**.

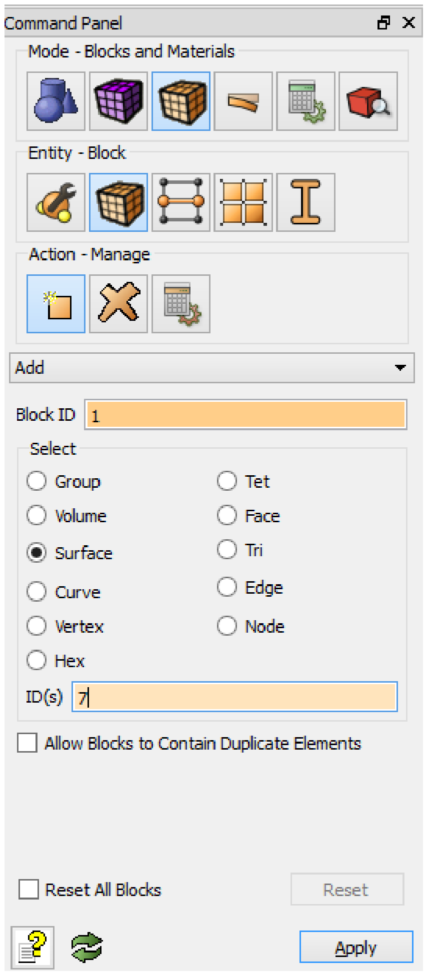


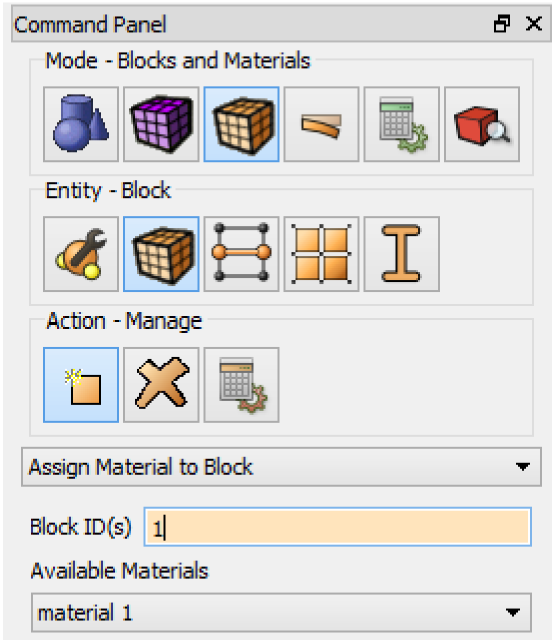
2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Select the entities to be united into block: Surface;
- ID: 7 (*or by the command all*).

Click **Apply**.



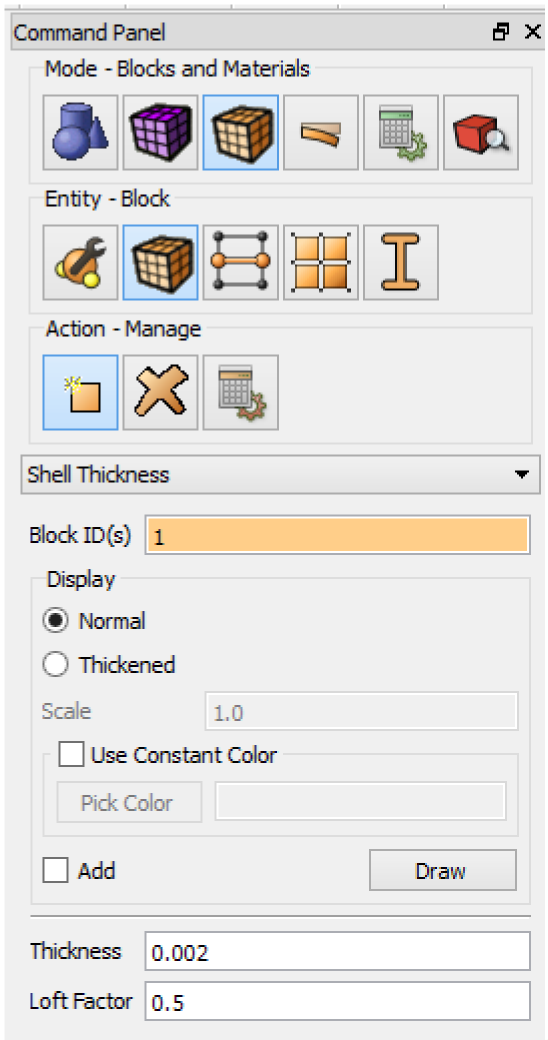


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.



4. Assign the shell thickness.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block**, Action – **Manage**). Select **Shell thickness** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Screen: normal;
- Thickness: 0.02;
- Loft Factor: 0.5.

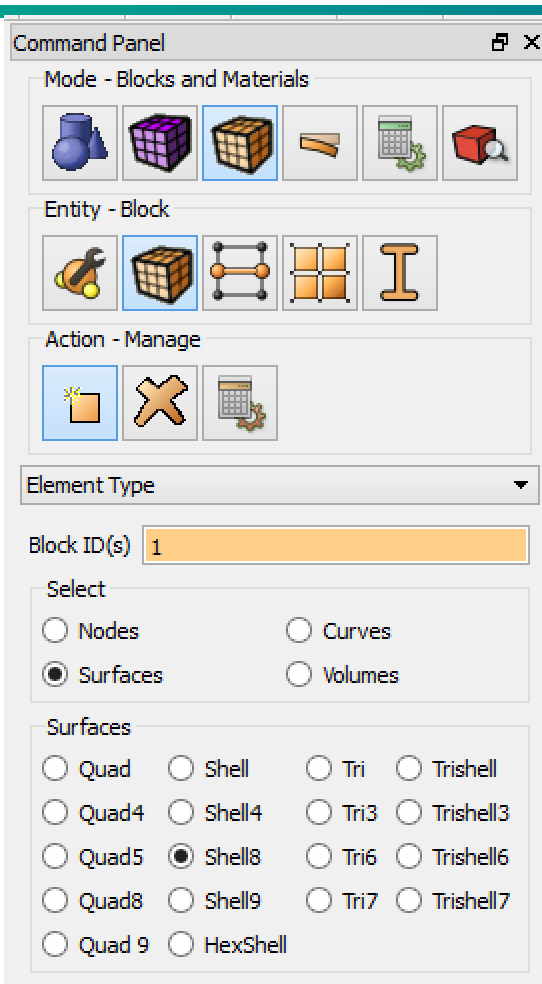
Click **Apply**.

5. Assign the element type to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Surfaces;
- Surfaces: Shell8.

Click **Apply**.



Setting boundary conditions

1. Fix the cross-cut curve Surface 17 by the symmetry condition.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 17 (*or click on the cross-cut curve*);
- Degrees of Freedom: Z-Translation; X-Rotation; Y-Rotation.

Click **Apply**.

2. Fix the longitudinal curve Curve 5 on the symmetry condition.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 5 (*or click on the longitudinal curve*);
- Degrees of Freedom: X-Translation; Y-Rotation; Z-Rotation.

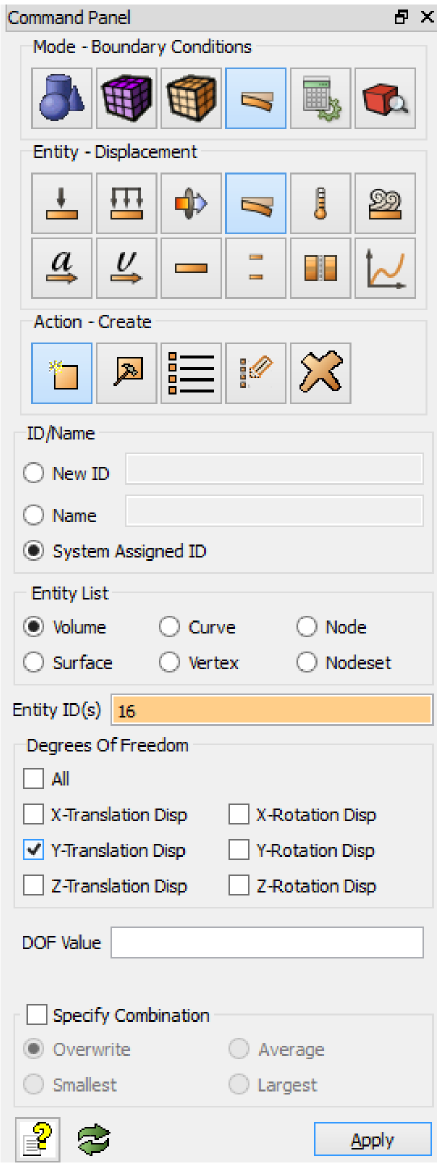
Click **Apply**.

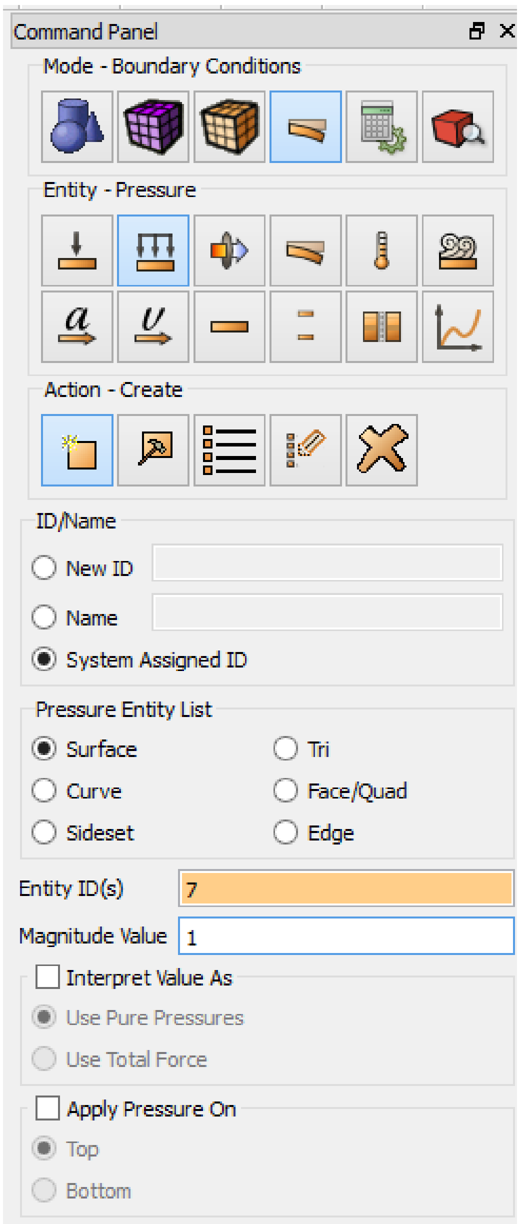
3. Fix the longitudinal curve Curve 16 by the symmetry condition.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 16 (*or click on the longitudinal curve*);
- Degrees of Freedom: Y-Translation; X-Rotation; Z-Rotation.

Click **Apply**.





4. Apply pressure to the cylinder inner surface with value of 1.

Select Mode – **Boundary Conditions,** Entity – **Pressure**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 7 (*or click on the cylinder surface*);
- Value: 1.

Click **Apply**.

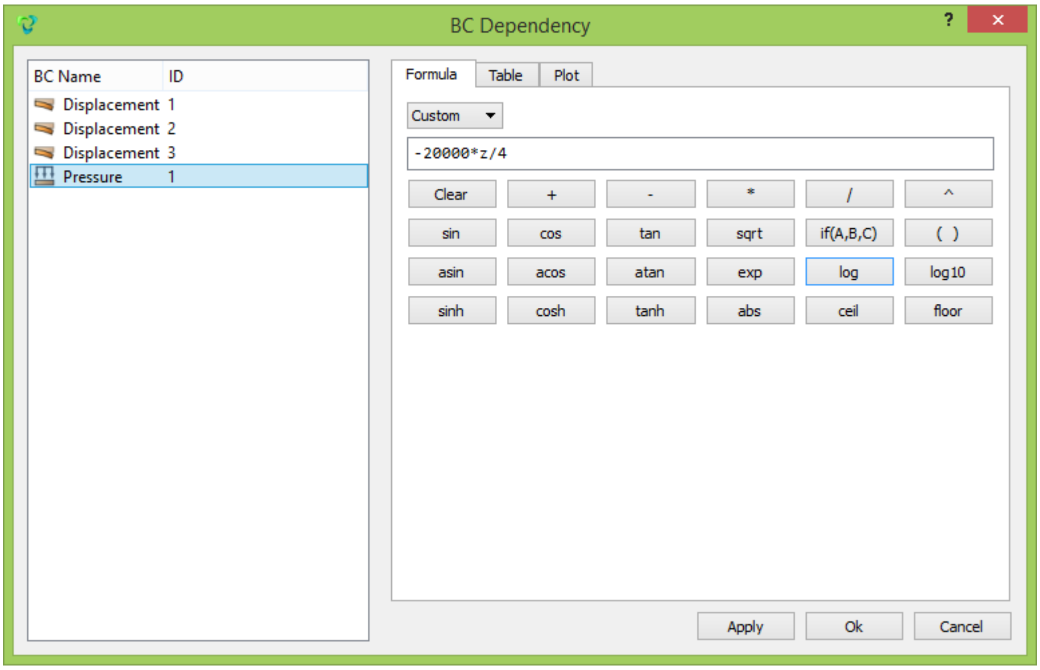
5. Set pressure dependency of the z-coordinate.

Select Mode — **Boundary Conditions,** Entity — **Set time/coordinates dependency of BC** on Command Panel.

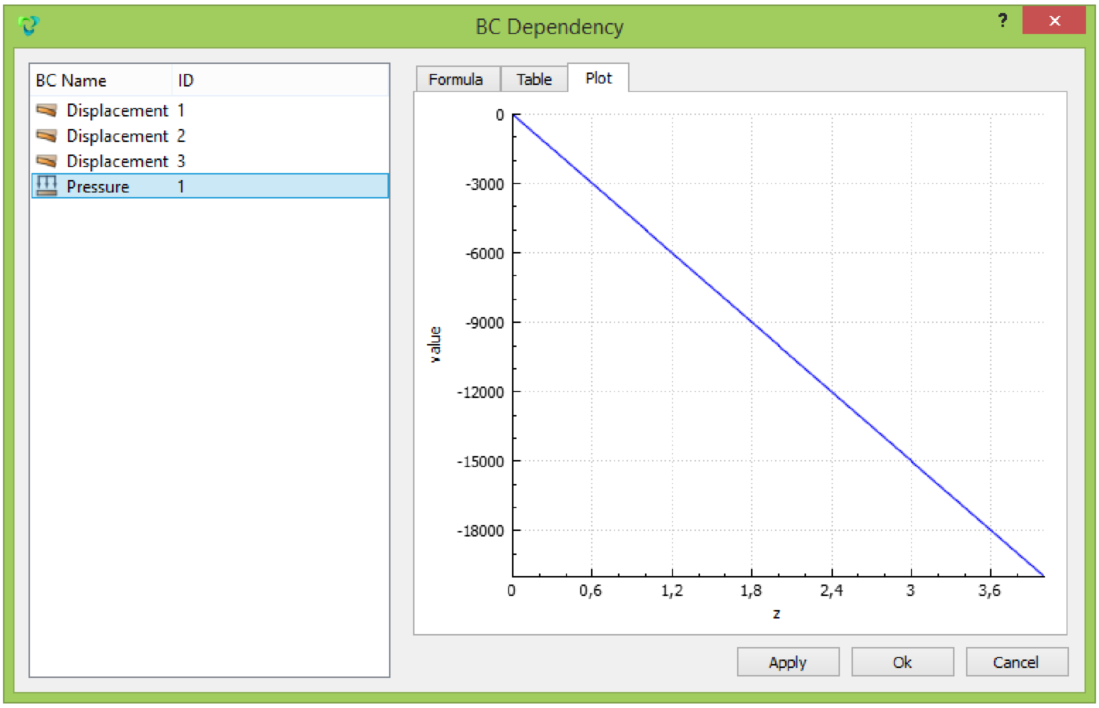
In the pop-up window **BC Dependency,** set the following parameters:

- BC name: Pressure 1;
- Select checkbox Formula, Manually;
- In the field below, enter -20000*z/4.

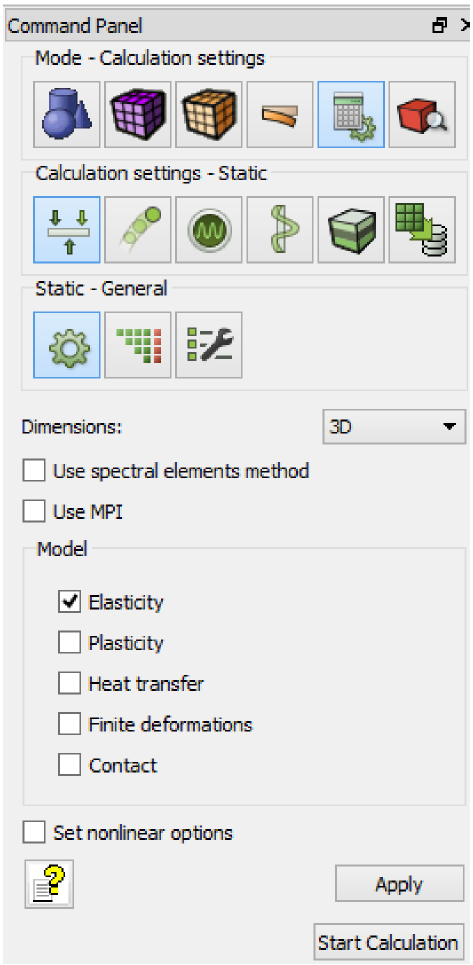
Click **Apply**.



To view the plotted graph, please, use the appropriate tab.



Starting calculation



1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select:

- Dimension: 3D;
- Model: Elasticity.

Click **Apply**.

Click **Start Calculation**.

In a pop-up window select a folder to save the result and enter the file name.

If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis

1. Open the file with the results. You can do this in one of the three ways.
   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (**Mode – Calculations Manager**, **Calculations Manager – Results**). Click **Open Results**.
2. Display the Uz component of the displacement field on the model.

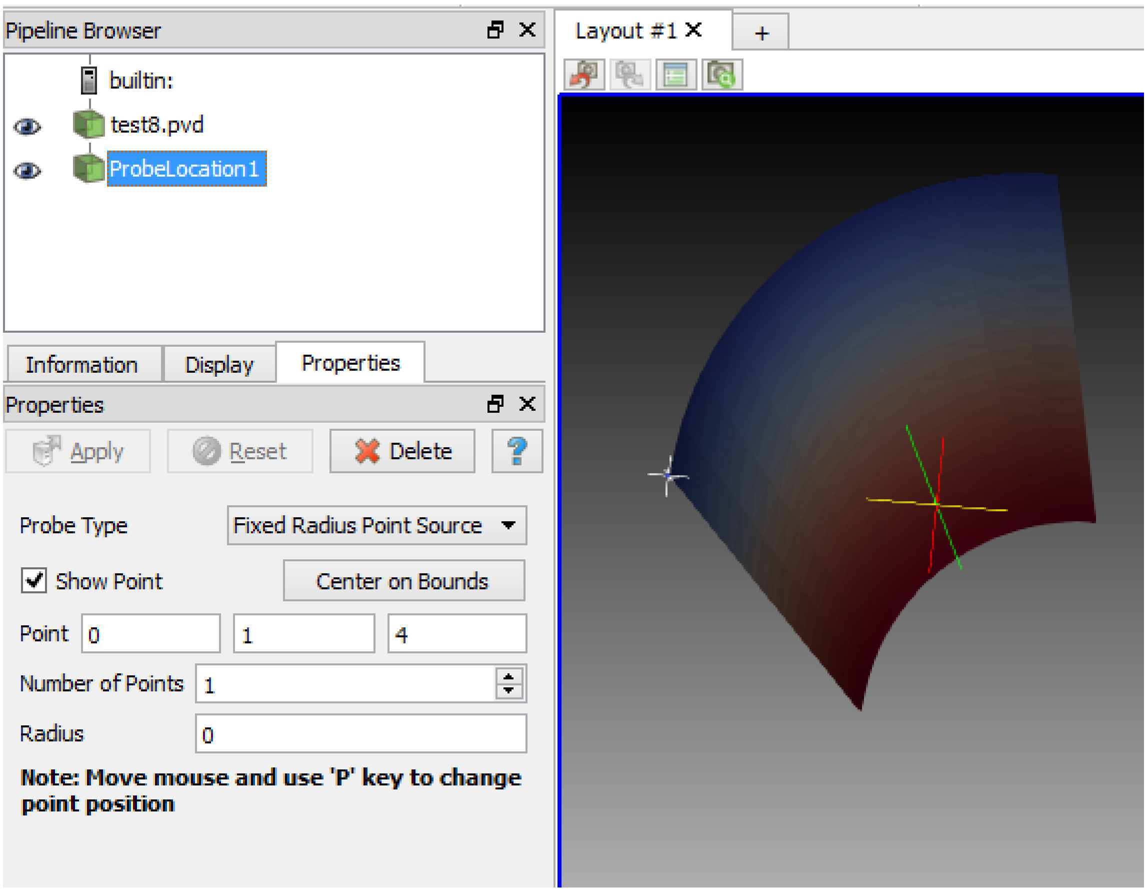
In **Fidesys Viewer** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Displacement;
- Representation Component: 3.

3. Compare the numerical value of the target displacement at the point (0,1,4) with the initial one of the source -2.86e-6.

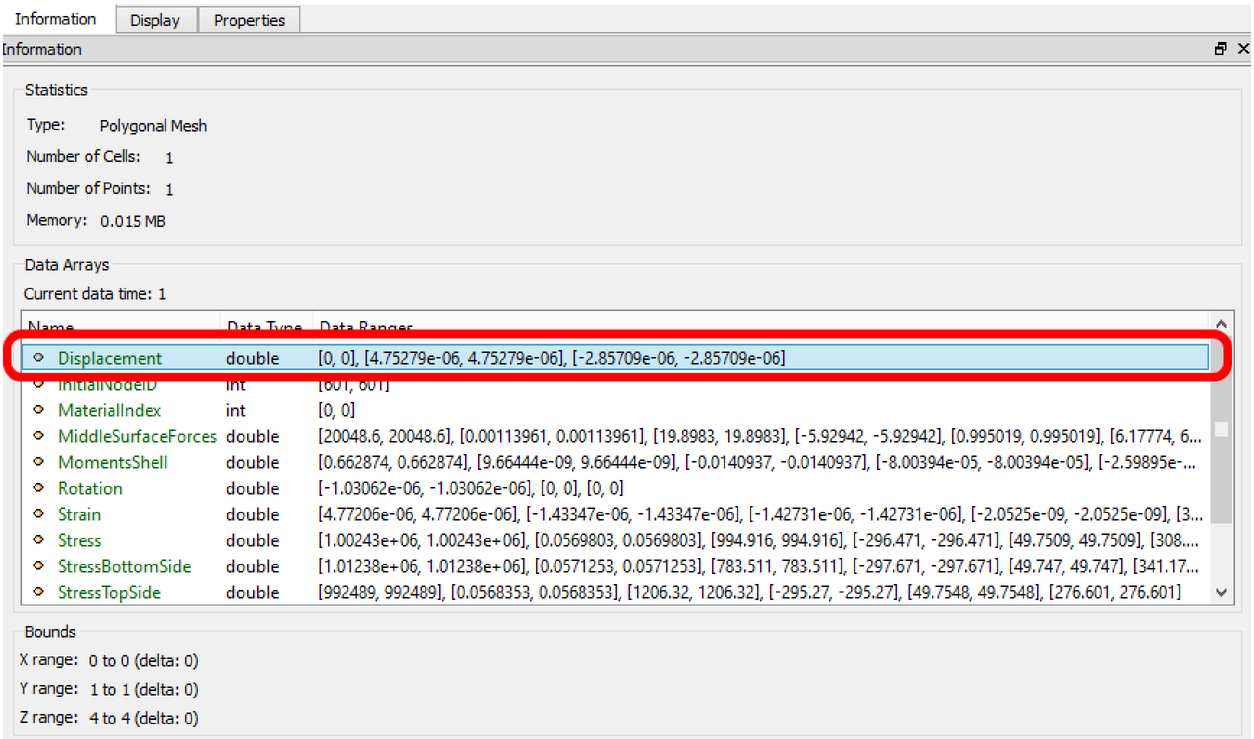
Select **Filters** → **Alphabetical** → **Probe Location**. In the tab Properties, set the following parameters for the filter:

- Point (0, 1, 4);
- Number of Points: 1;
- Radius: 0.



Go to the tab **Information**. We are interested in the field **Displacement** – the third component:

The difference between the resulting value -2.85709e-6 and the required -2.86e-6 is 0.1%.



You can see the way the body is deformed under the applied pressure.

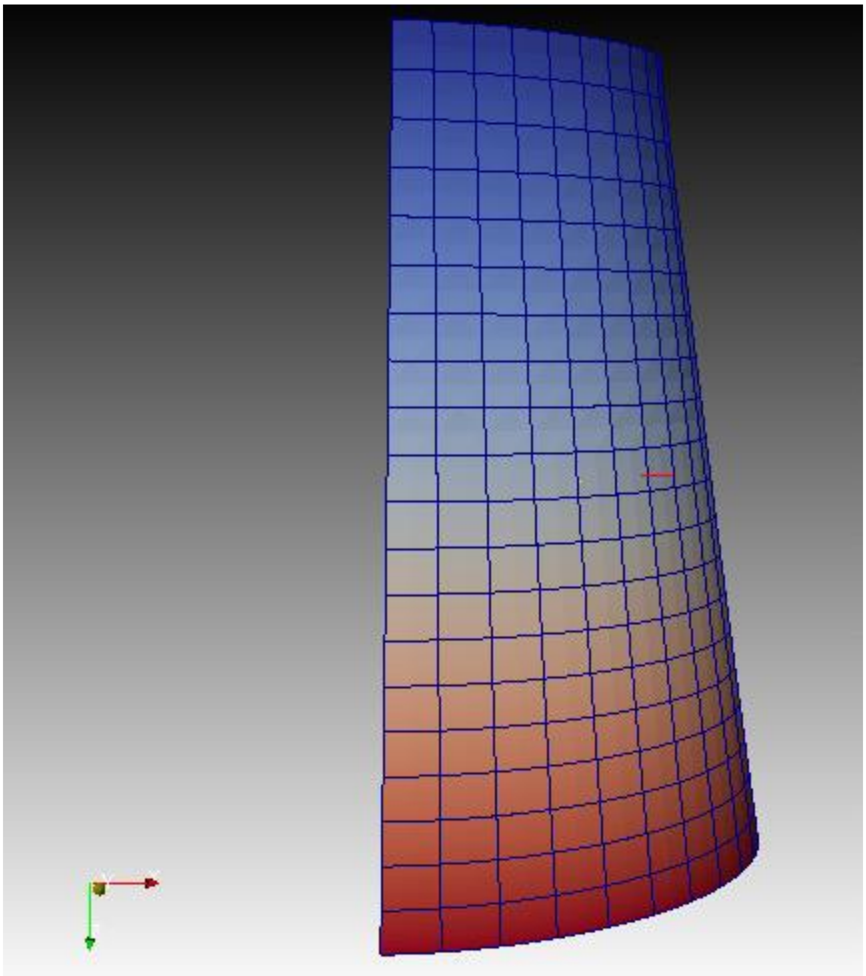
Select the filter **Warp By Vector** to do this. Set the following parameters in the tab **Properties**:

- Vectors: Displacement;
- Scale Factor: 1e5.

As a result, the deformed body is displayed at the picture.

Select the following display settings for the deformed view:

To see the original model, click the icon near the model in the Model Tree.



Consider the direction of the coordinate axes in the picture.

4. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

5. Get the original cylinder using reflection.

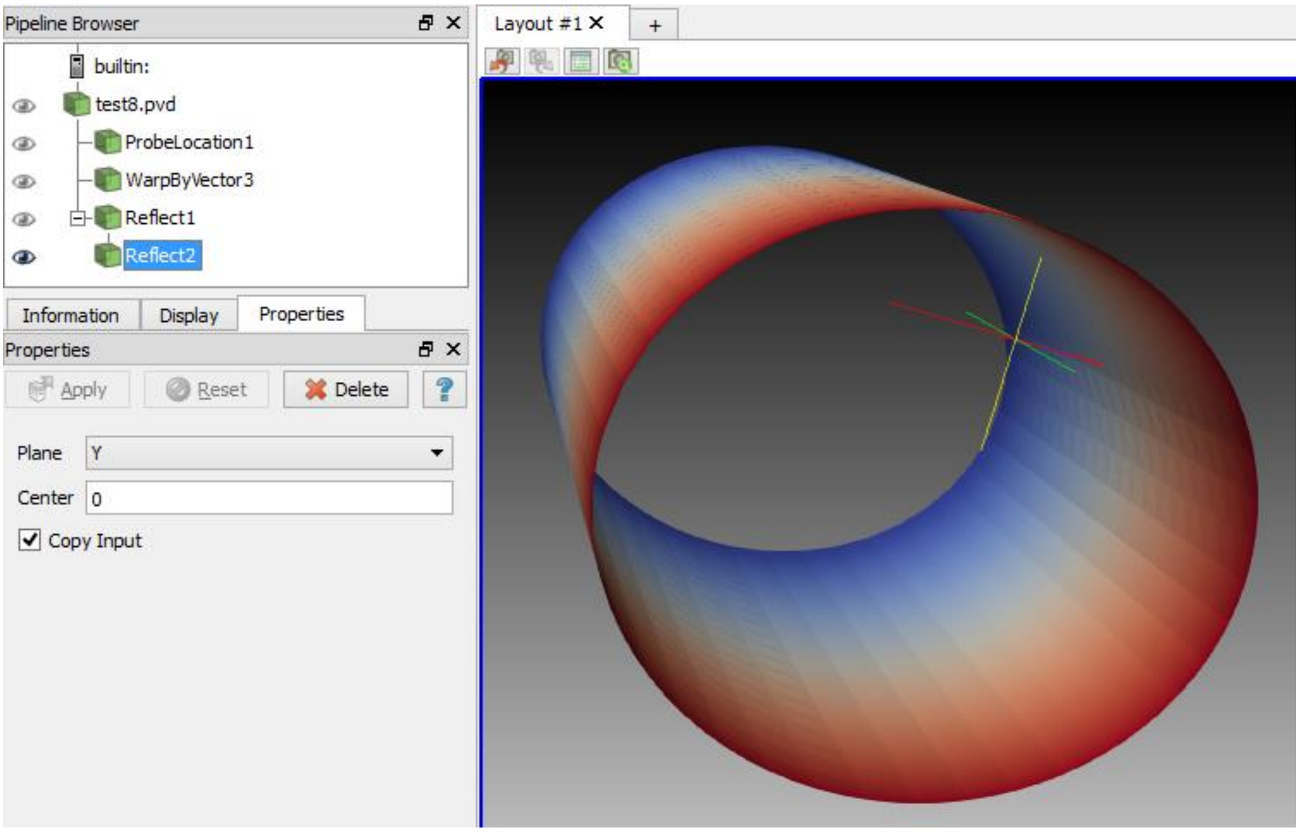
In the Model Tree, click on **test.pvd.** Then, in the Main Menu, select **Filters** → **Alphabetical** → **Reflect**. Set the following parameters in the tab Properties:

- Plane: X
- Center: 0.

In the Model Tree, select now **Reflect1** and then again **Filters** → **Alphabetical** → **Reflect**. Set the following parameters in the tab Properties:

- Plane: Y
- Center: 0.

You received completely original cylinder by reflecting ¼ part.



Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

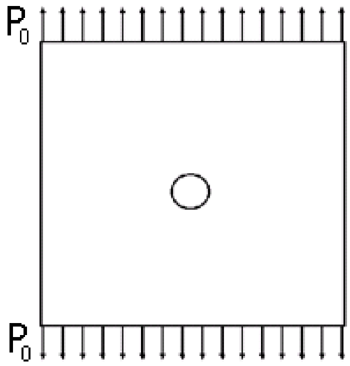
```
reset
set node constraint on
create Cylinder height 4 radius 1
delete volume 1 keep_lower_geometry
```

```
delete Surface 2 3
webcut body 2 with plane xplane offset 0 preview
webcut body 2 with plane xplane offset 0
webcut body 2 with plane yplane offset 0 preview
webcut body 2 with plane yplane offset 0
delete Surface 5 6
move Surface 7 preview z 2 include_merged
move Surface 7 z 2 include_merged
curve 17 18 interval 10
curve 17 18 scheme equal
curve 5 16 interval 20
curve 5 16 scheme equal
surface all size auto factor 5
mesh surface all
list Surface 7 mesh
undo group begin
create material "material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "material 1" scalar_properties "MODULUS" 2.1e+11 "POISSON" 0.3
undo group end
set duplicate block elements off
block 1 surface 7
block 1 material 'material 1'
block 1 element type shell8
undo group begin
block 1 attribute count 2
block 1 attribute index 1 value 0.02
block 1 attribute index 2 value 0.5
undo group end
block 1 element type shell8
create displacement on curve 17 dof 3 dof 4 dof 5 fix
create displacement on curve 5 dof 1 dof 5 dof 6 fix
create displacement on curve 16 dof 2 dof 4 dof 6 fix
create pressure on surface 7 magnitude 1
bcdep pressure 1 value '-20000*z/4'
analysis type static elasticity dim3
spectralelement off
usempi off
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

Static loading (2D, Kirsch problem)

Седов Л.И. "Механика сплошной среды, том 2". М.: Наука, 1970г., 568 стр.

[Sedov L.I., Continuum Mechanics, Volume 2, Nauka, Moscow, 1970, 568 pages [in Russian]]



2D problem of one-sided tension of a plate with a circular hole is regarded. The picture represents a geometric model of the problem: side length is 10 m, hole diameter is 0.5 m. The load P_0 = 1 MPa is applied on the edges.

In view of the symmetry of the problem, ¼ of the plate is considered.

Test pass criterion is the following: Stress σ_θ at the point E (0.25;0;0) is 3 MPa within 3%.

Geometry creation

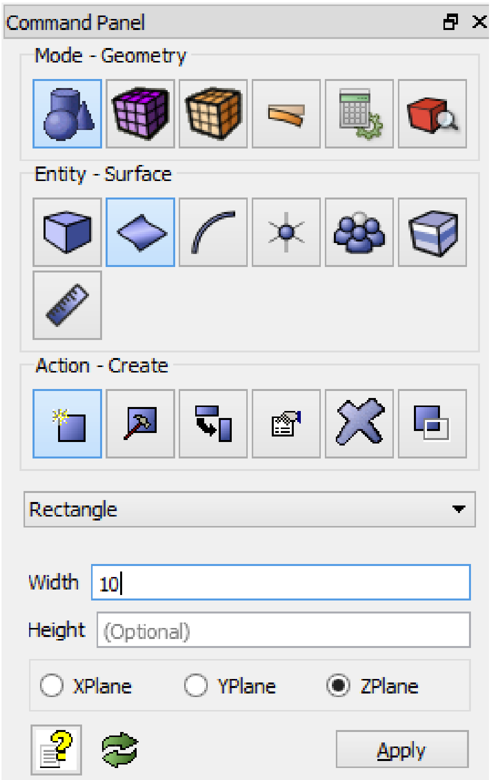
1. Create the square 10 on side.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity –**Surface**, Action – **Create**). Select Rectangle in the list of geometric elements. Set the parameters:

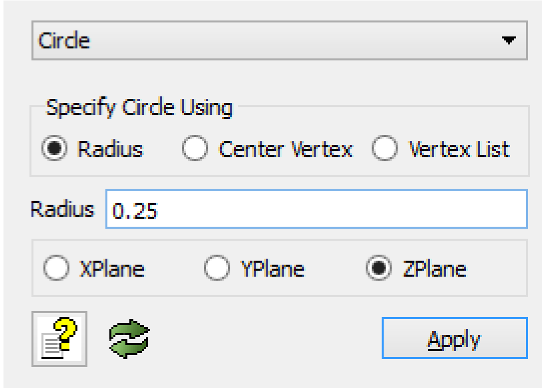
- Width: 10;
- Height: Optional.
- Z-plane

Click **Apply**.

Note: 2D problems for Quad/Quad4/Quad8/Quad8 and Tri/Tri3/Tri6 elements are to be solved if only the model is in the Z-plane.



2. Create the circle with a radius 0.25.

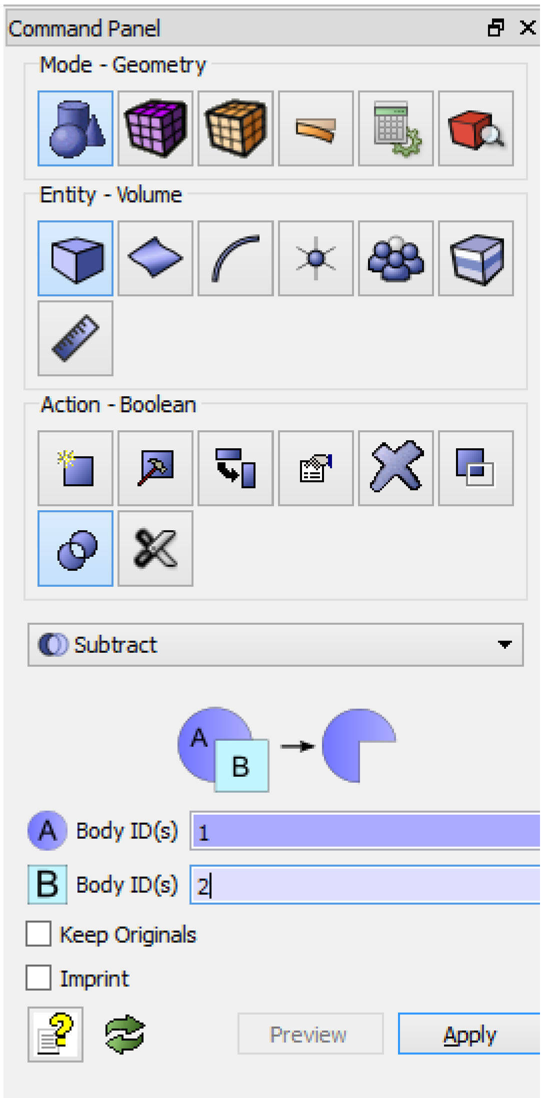


Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity –**Surface**, Action – **Create**). Select **Circle** in the list of geometric elements. Set parameters:

- Radius: 0.25;
- Z-plane.

Click **Apply**.

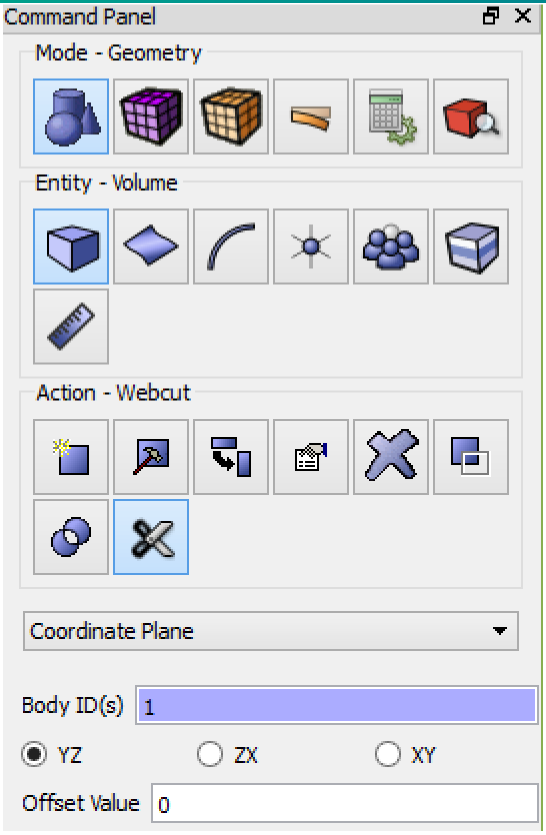
3. Subtract the circle from the square.



Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set parameters:

- Body ID: 1 *(or click on the square)*
- Subtract bodies (ID): 2 *(or click on the circle)*

Click **Apply**.



4. Leave a quarter of the plate.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

- Body ID: 1 *(or click on the plate)*;
- Webcut with: YZ Plane;
- Offset value: 0.

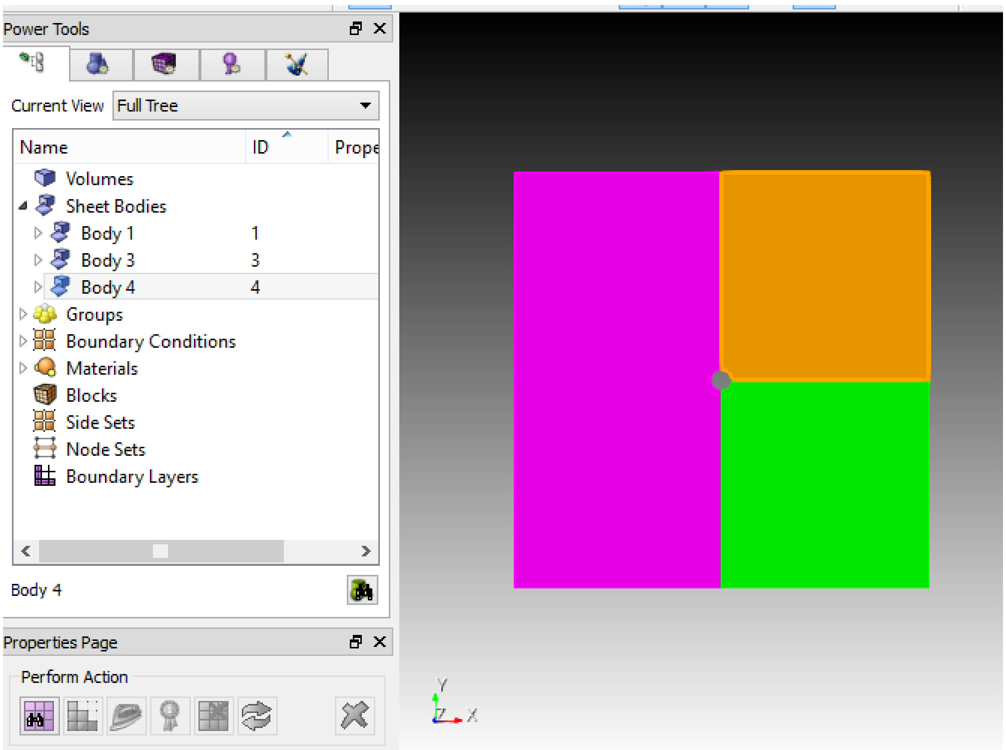
Click **Apply**.

Do the same for the ZX Plane. Set the following parameters:

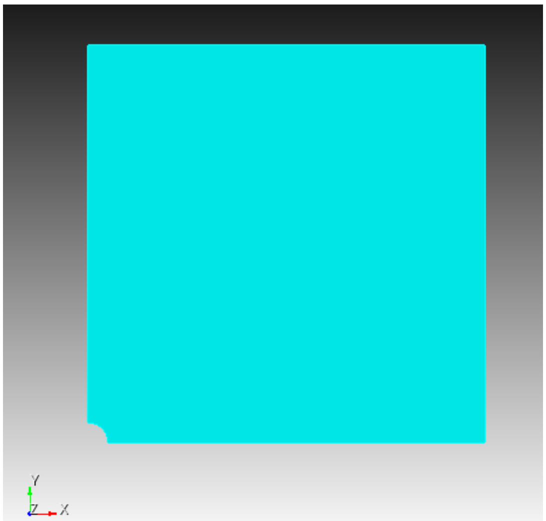
- Body ID: 1 *(or click on the right plate)*;
- Put the checkbox for ZX;
- Offset value: 0.

Click **Apply**.

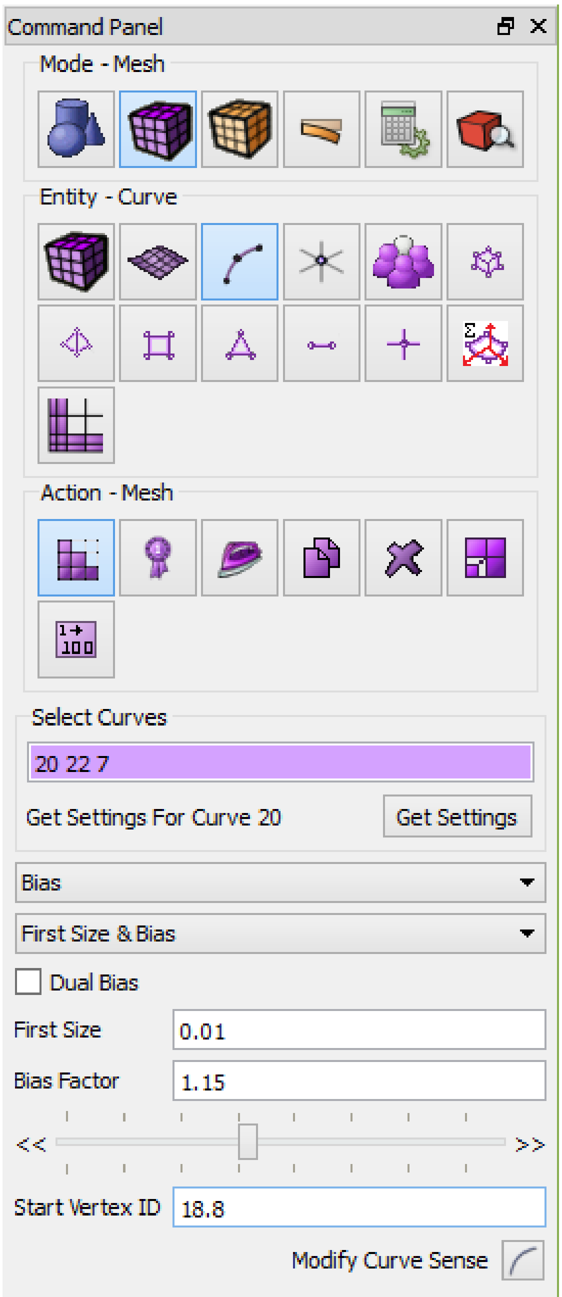
As a result, the original surface in the Model Tree is split into three (Body 1, Body 3 and Body 4).



Delete surfaces 1 and 3. To do this, holding down the Ctrl key, select the volumes in the Model Tree and click **Delete** in the context menu. As a result, a quarter of the original plate is left (Body 4):



Meshing



1. Select meshing on curves section on Command Panel (Mode — **Mesh,** Entity — **Curve**, Action — **Meshing**). Specify the following parameters:

- Select Curves: 22 20 7 *(or clicking on the bottom and left edges of the plate and on the circle outline while pressing the key Ctrl)*;
- Select meshing scheme: Bias;
- In the drop-down list below, select: First Size & Bias;
- The first size: 0.01;
- Bias Factor: 1.15;
- Start Vertex ID: 18 8.

Click **Apply size.**

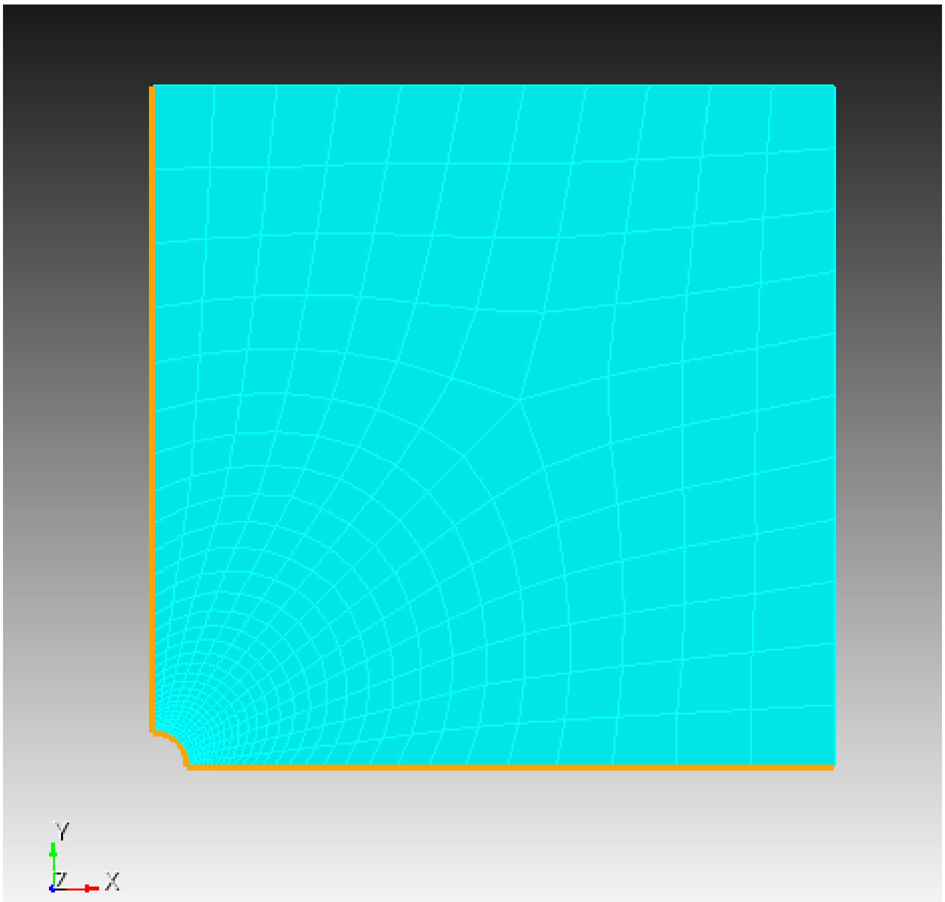
Click **Mesh.**

2. Select surfave mesh generation section on Command Panel (Mode — **Mesh,** Entity — **Surface**, Action — **Meshing**). Specify the following parameters:

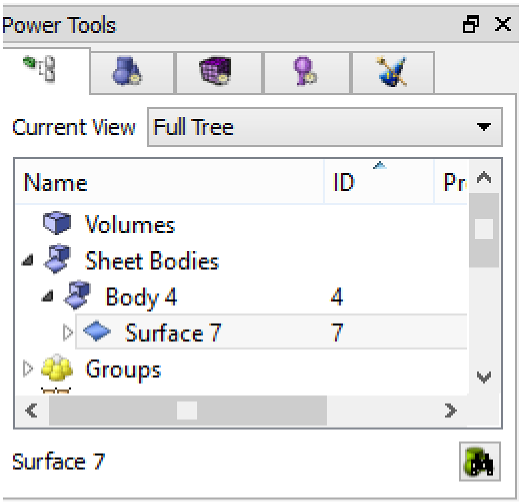
- Choosing surfaces: 7 *(or all)*;
- Select meshing scheme: Polyhedron;

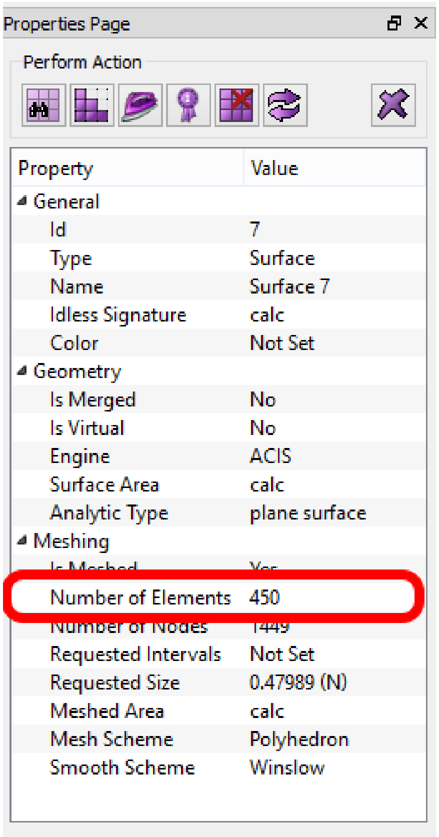
Click **Apply**.

Click **Mesh**.

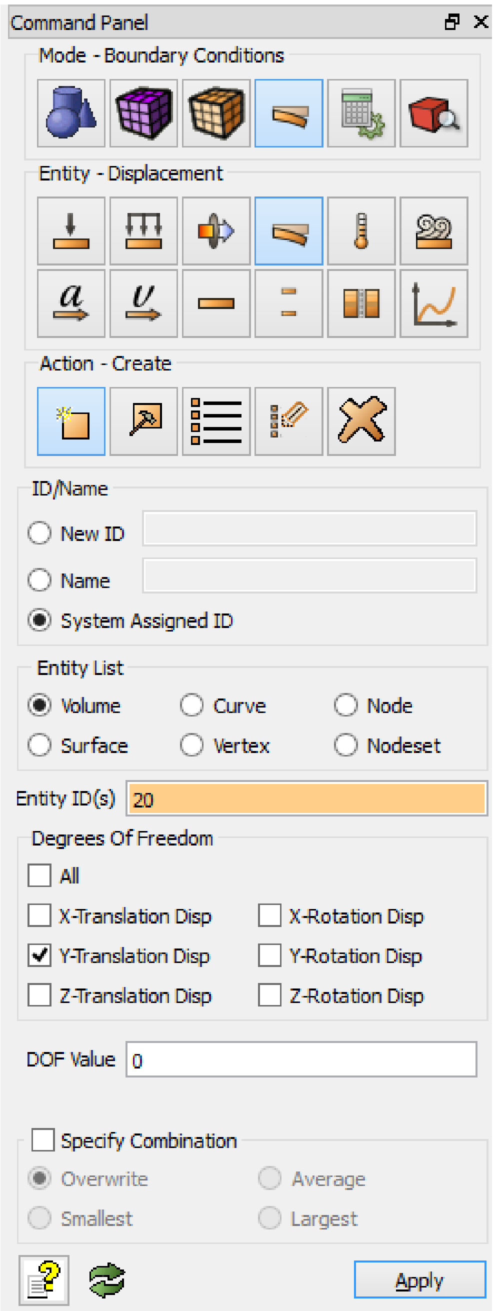


3. In the Model Tree, select Plane bodies – Body 4 – Surface 7. Below, on the Property Page you can see the number of elements for the generated mesh.





Setting boundary conditions



1. Fix the bottom edge by the symmetry condition.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 20 *(or click on the bottom edge)*;
- Degrees of Freedom: Y-Translation;
- DOF Value: 0.

Click **Apply**.

2. Fix the left edge on the symmetry condition.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 7 *(or click on the left edge)*;
- Degrees of Freedom: X-Translation;
- DOF Value: 0.

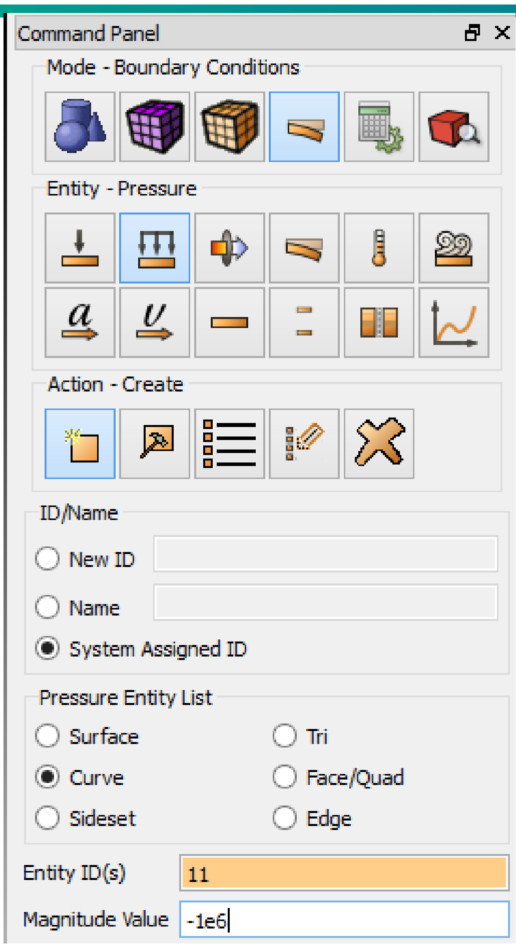
Click **Apply**.

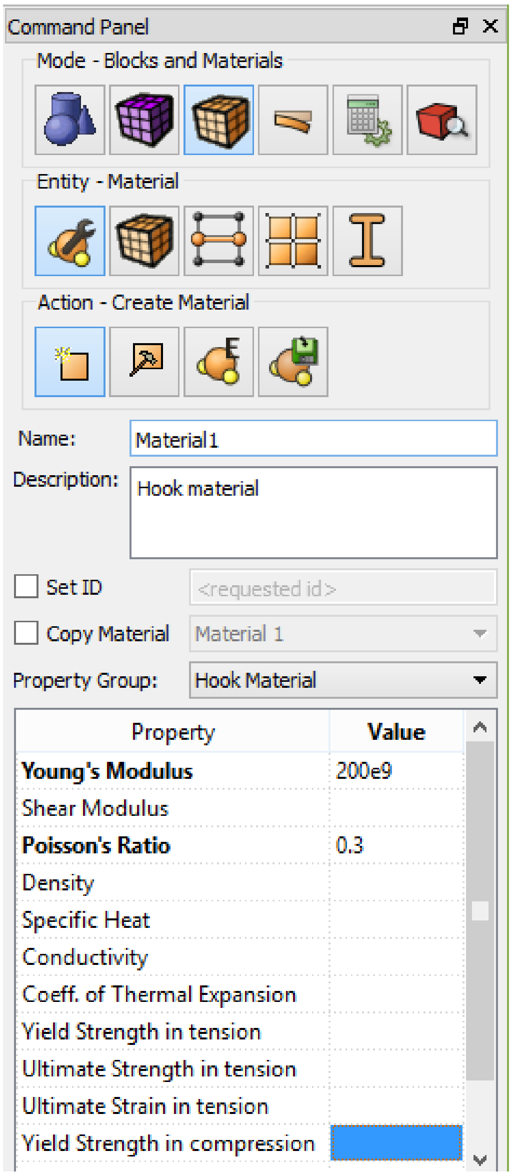
3. Apply uniform pressure to the surface.

Select Mode – **Boundary Conditions,** Entity – **Pressure**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 11;
- Value: -1e6;

Click **Apply**.





Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Description: Hook material
- Property group: Hook material;
- Young’s Modulus: 200e9;
- Poisson’s Ratio: 0.3.

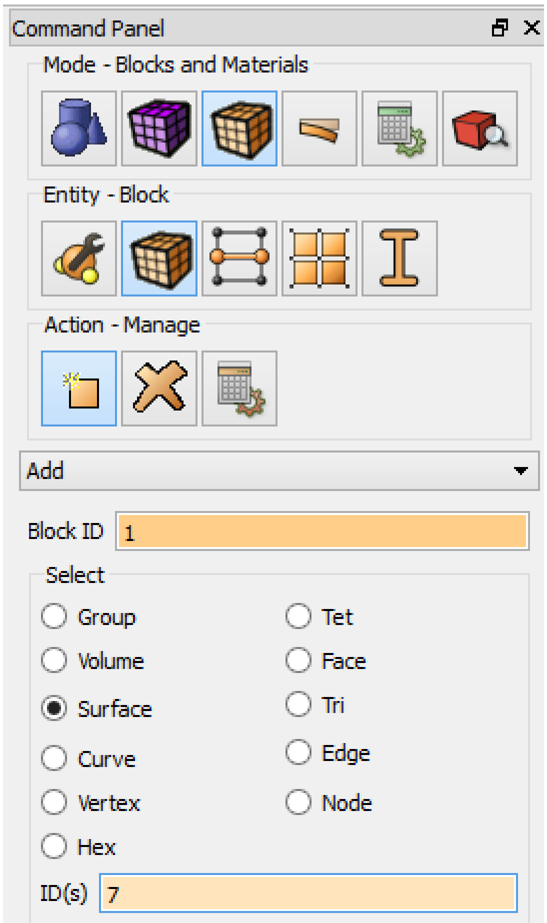
Click **Apply**.

2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Surface;
- ID: 1 *(or by the command **all**)*.

Click **Apply**.

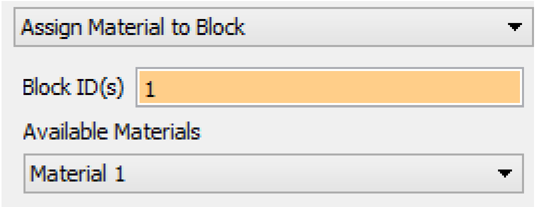


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

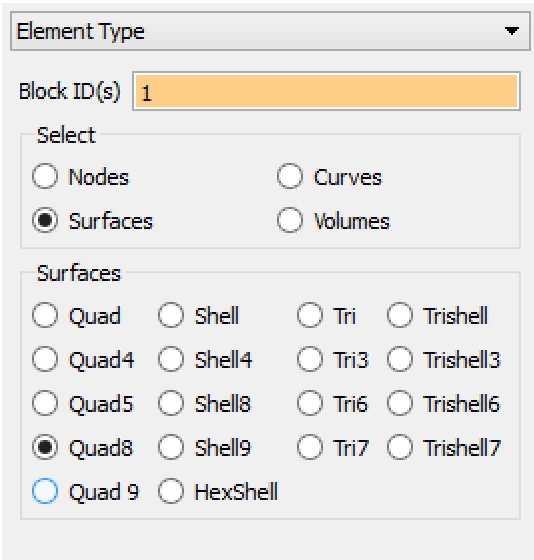


4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Surfaces;
- Element type: Quad8.

Click **Apply**.



Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select:

- Dimension: 2D;
- Type of plane problem: Plane deformed state;
- Model: Elasticity.

Click **Apply**.

2. Set the solver settings.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default. Click **Apply**.

3. Set the reaction force calculation

Go to the tab **Static** – **Output fields** and set the checkbox **Calculate nodal and reaction forces**.

Click **Apply**.

Click **Start Calculation**.

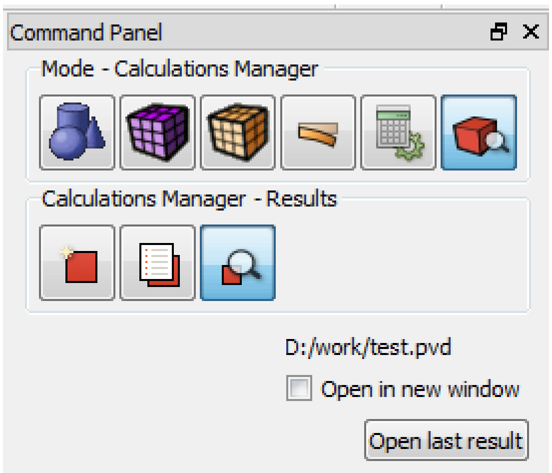
Note: Without setting the checkbox **Calculate nodal and reaction forces,** the field is not calculated.

4. In a pop-up window select a folder to save the result and enter the file name.
5. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>".*

Results analysis

1. Open the file with the results. You can do this in one of the three ways.

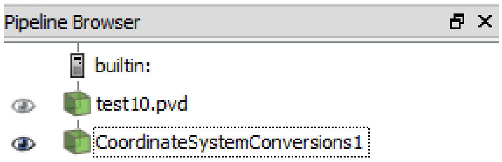
- Click Ctrl+E.
- Select Calculation → Open Results in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.

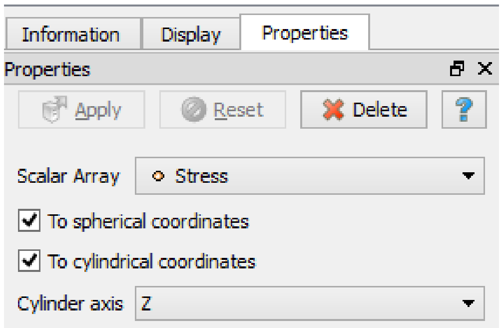


2. Display the σ_θ component of the stress field.

Select Filter → Alphabatical → Coordinate systems conversions. Select in the tab Properties:

- Scalar Array: Stress.
- Cylinder axis: Z

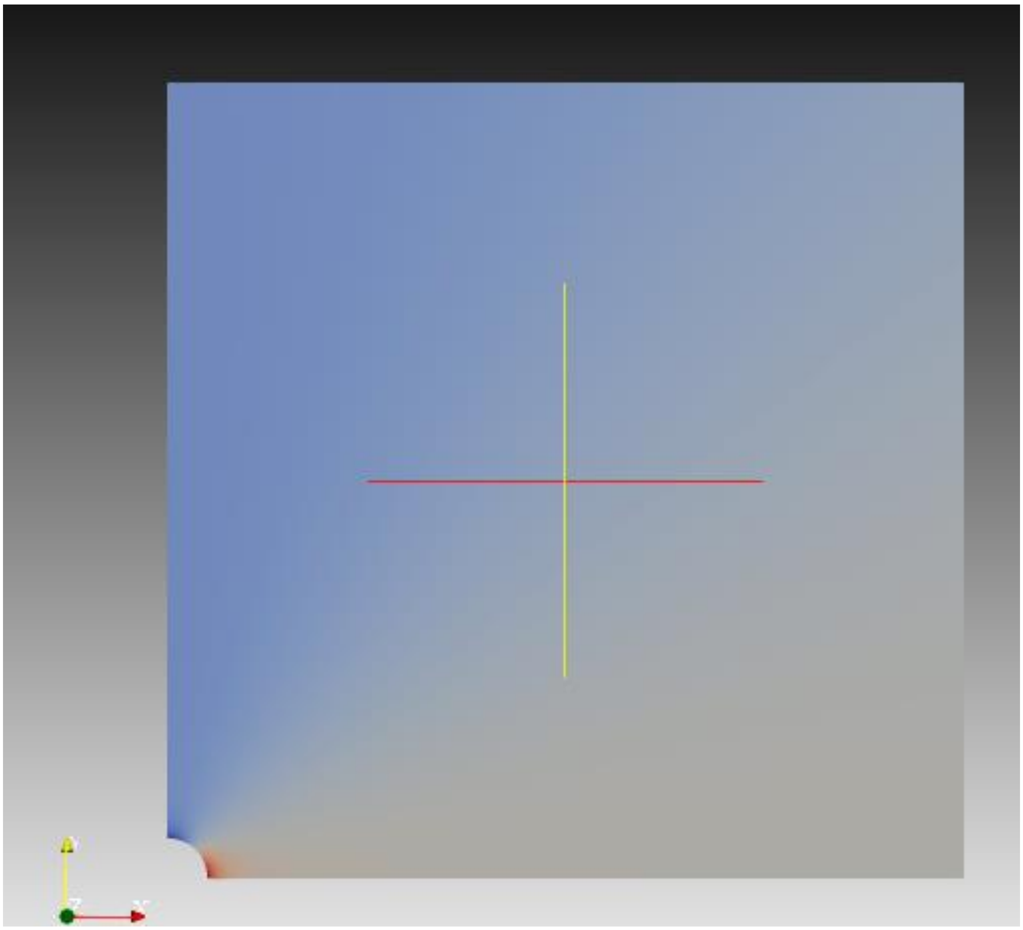




In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Stress cylindrical;
- Representation Component: FF.

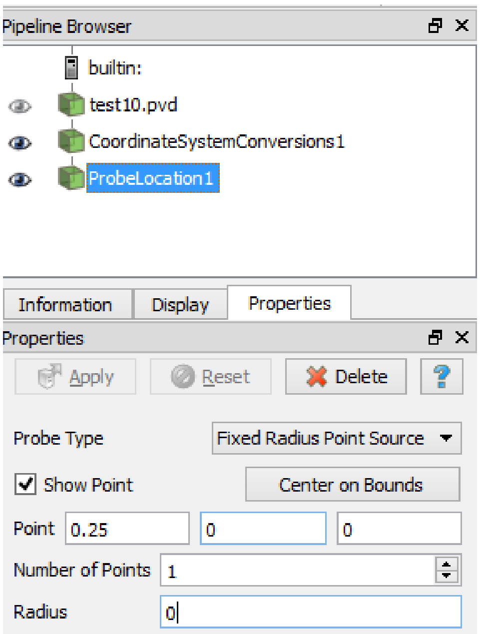
On the model stress distribution field σ_θ will be displayed in cylindrical coordinates.



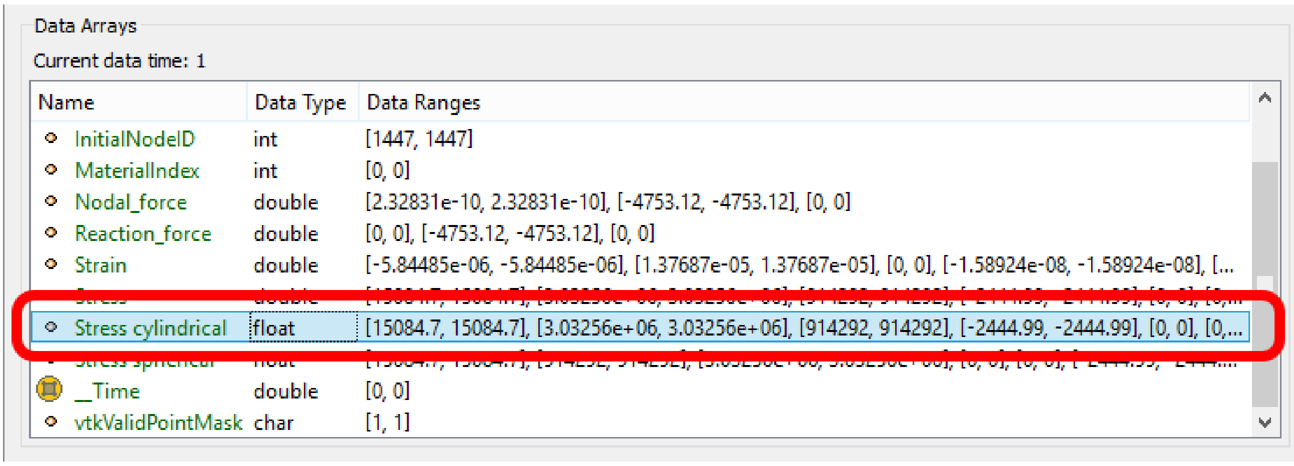
3. Check the numerical value of the target stress.

Click with the mouse on the name of the filter Coordinate systems in the Model Tree. Next, select Filter → Alphabetical → Probe Location. In the Properties tab for the filter, set the following parameters:

- Point (0.25, 0, 0);
- Number of Points: 1;
- Radius: 0.



Please, go to the Information tab. We are interested in the Stress field (cyl.) – the second component:


Data Arrays

Current data time: 1

Name	Data Type	Data Ranges
InitialNodeID	int	[1447, 1447]
MaterialIndex	int	[0, 0]
Nodal_force	double	[2.32831e-10, 2.32831e-10], [-4753.12, -4753.12], [0, 0]
Reaction_force	double	[0, 0], [-4753.12, -4753.12], [0, 0]
Strain	double	[-5.84485e-06, -5.84485e-06], [1.37687e-05, 1.37687e-05], [0, 0], [-1.58924e-08, -1.58924e-08], [...
Stress cylindrical	float	[15084.7, 15084.7], [3.03256e+06, 3.03256e+06], [914292, 914292], [-2444.99, -2444.99], [0, 0], [0,...
_Time	double	[0, 0]
vtkValidPointMask	char	[1, 1]

The difference between the resulting value 3.05077e+06 and the required 3e6 is 1.6%.

To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

4. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

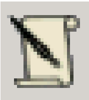
Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
create surface rectangle width 10 zplane
create surface circle radius 0.25 zplane
subtract body 2 from body 1
webcut body 1 with plane xplane offset 0
webcut body 1 with plane yplane offset 0
delete Body 1 3
curve 20 22 7 scheme bias fine size 0.01 factor 1.15 start vertex 18 8
mesh curve 20 22 7
surface 7 scheme Polyhedron
mesh surface 7
create displacement on curve 20 dof 2 fix 0
create displacement on curve 7 dof 1 fix 0
create pressure on curve 11 magnitude -1e+06
undo group begin
create material "Material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "Material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3
undo group end
set duplicate block elements off
block 1 surface all
block 1 material 'Material 1'
block 1 element type quad8
```

```
analysis type static elasticity dim2 planestrain
spectralelement off
usempi off
solver method auto try_other off
output nodalforce on midresults on
calculation start path "D:/FidesysBundle/calc/example.pvd"
```



It is also possible to run the file *Example_5_Static_2D.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Dynamic load (3D)

A dynamic problem of the wave propagation in a long rod is being solved. The problem has an analytic solution which sets the dependency between the velocity of wave distribution in the rod and the material parameters of the rod:

$$\alpha = \sqrt{\frac{\lambda + 2\mu}{\rho}}$$

where α is the wave velocity, ρ is the density,

$$\lambda = \frac{\upsilon E}{(1+\upsilon)(1-2\upsilon)},$$

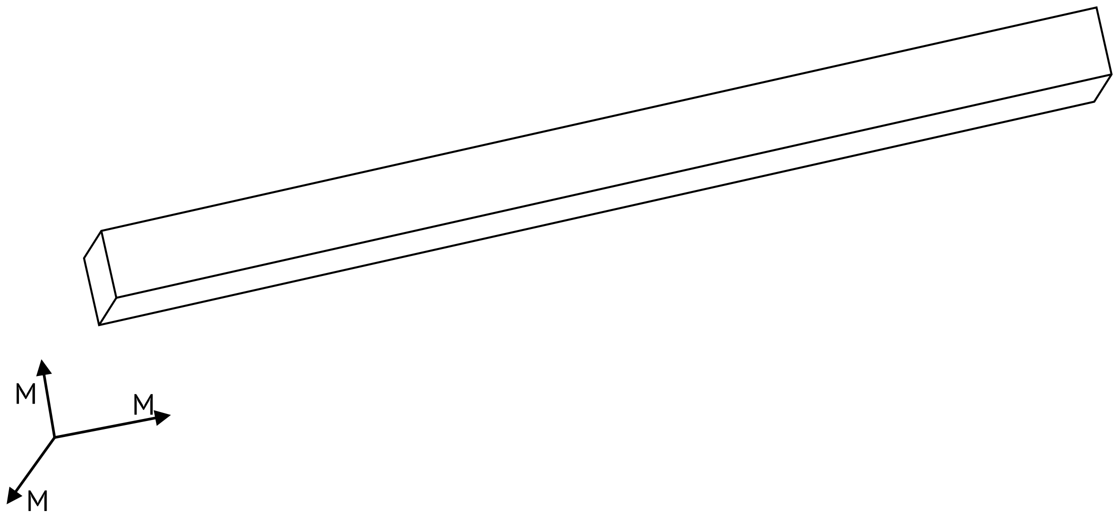
$$\mu = \frac{E}{2(1+\upsilon)}$$

E — Elastic Modulus(Young's Modulus),

υ — Poisson's Ratio.

At the values E = 200 hPa, ρ = 7900 kg/m^3 and υ = 0.3 the wave velocity according to analytic solution is 5717.41 m/s.

The picture below represents a geometric model of the problem:



The length of the rod is 100 m. Displacement along the normal to the rod's sides (parallel to the axis X) is constrained. The left butt is fixed along X axis. The pressure 100 MPa is suddenly applied to the right butt.

Test pass criterion is the following: stress σ_{yy} at the point D is -5.38MPa to within 1.5%.

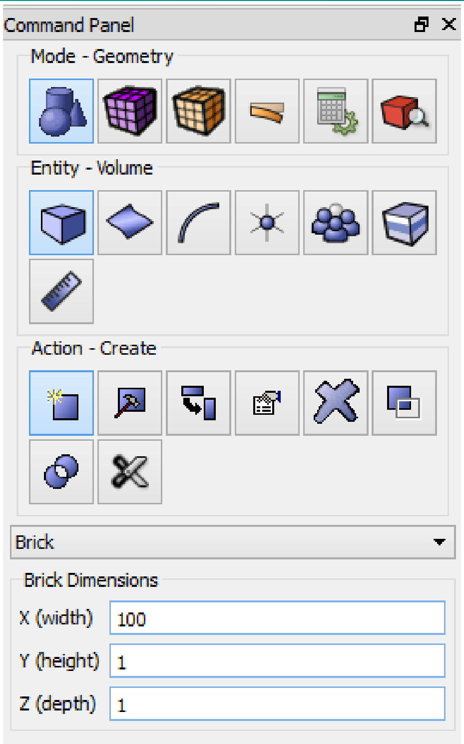
Geometry creation

1. Create a sliver parallelepiped.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Brick** in the list of geometric elements. Set the brick dimensions:

- Width: 100;
- Height: 1;
- Depth: 1.

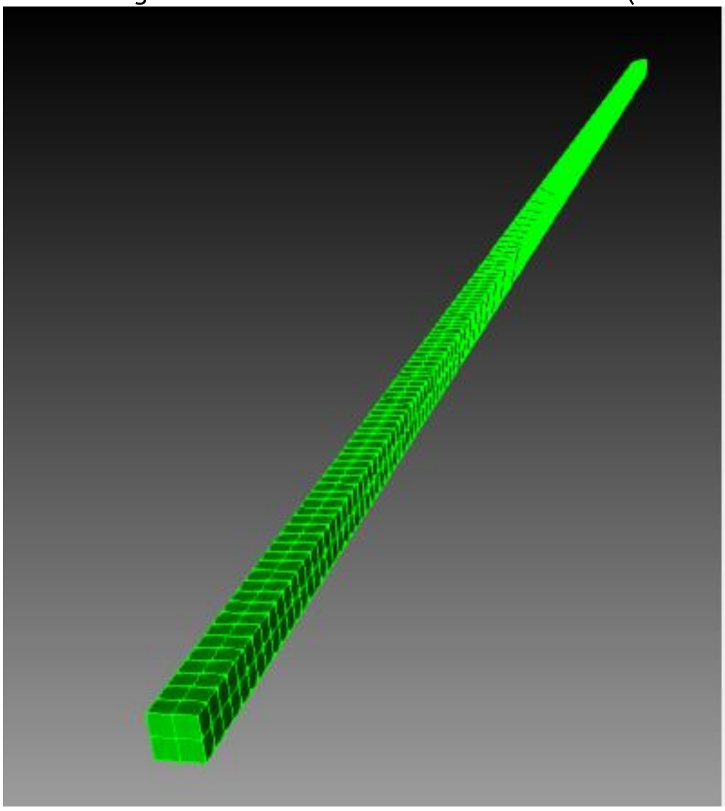
Click **Apply**.

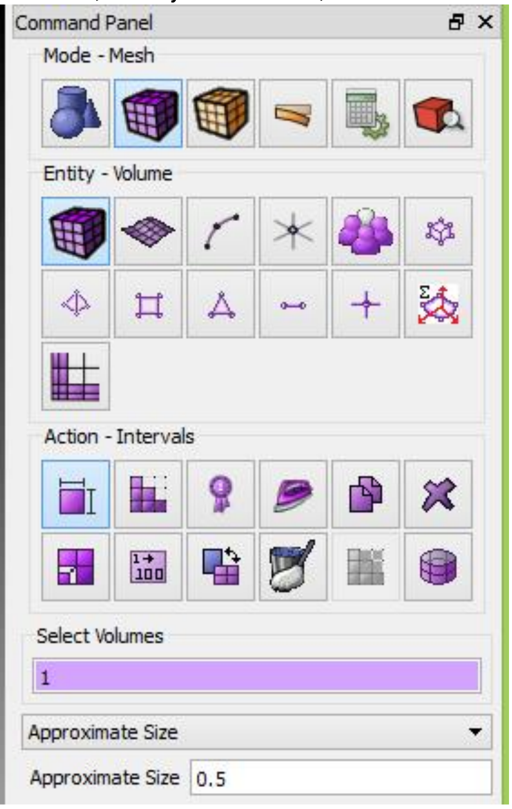


Meshing

1. Set the elements size.

Select volume mesh generation section on Command Panel (Mode — **Mesh,** Entity — **Volume**, Action — **Intervals**):





- Select Volumes (specify their ID): 1 *(or by the command **all**);*
- The way of meshing: Approximate Size;
- Approximate Size: 0.5

Click **Apply Scheme**.

Click **Mesh.**

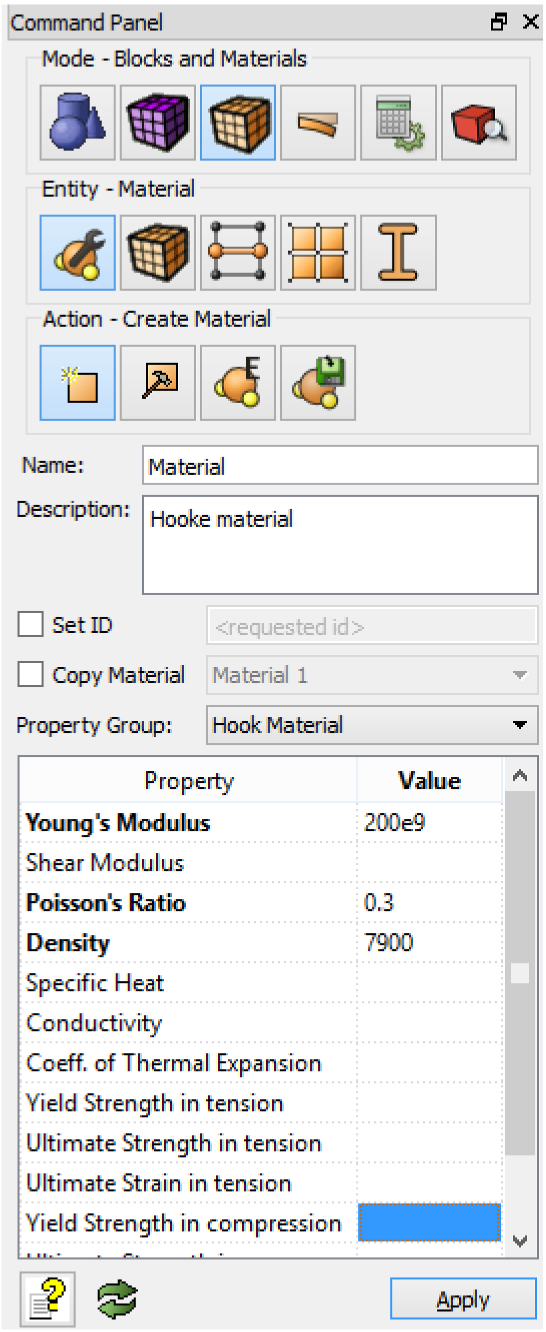
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Set the following parameters:

- Name: Material1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 200e9;
- Poisson's Ratio: 0.3.
- Density: 7900

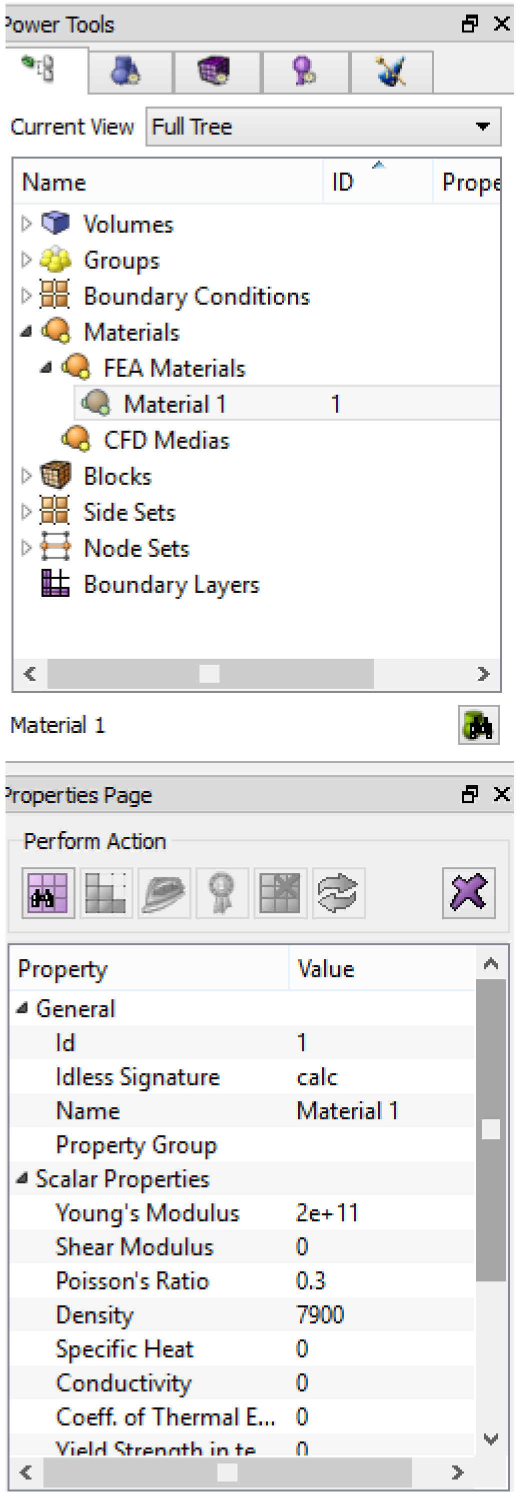
Click **Apply**.



On the left panel a new material Material1 is displayed. Click on it to display the following constants of materials on the **Properties page** below.

The parameters of materials on the properties page are available for editing. Left-click on the editing window opposite the inscription Density. Remove the 7900 value and enter the new value 7800. Click ENTER. After this a message about changing of constants of the material will be displayed in Command Line:

```
modify material 1 density 7800
```

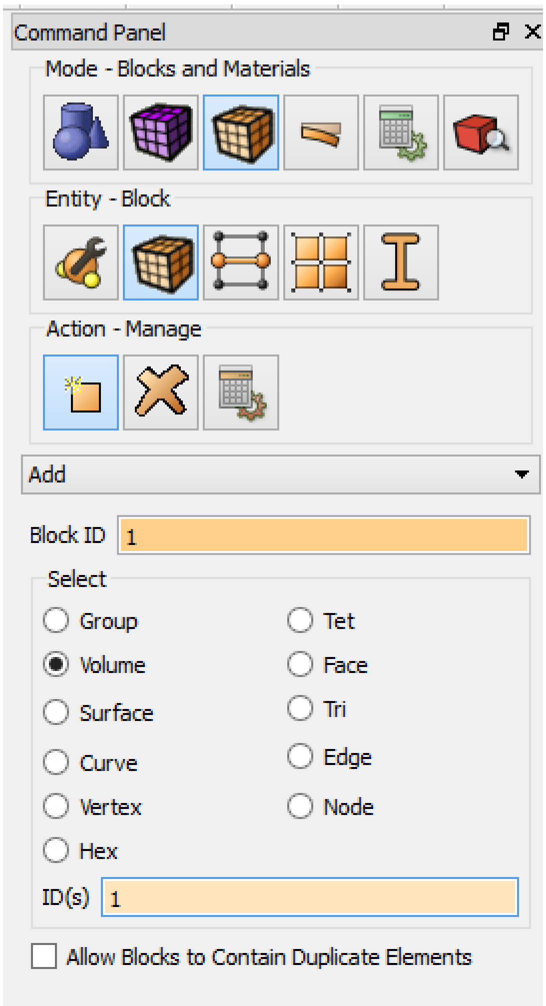


2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: 1 *(or by the command **all**).*

Click **Apply**.

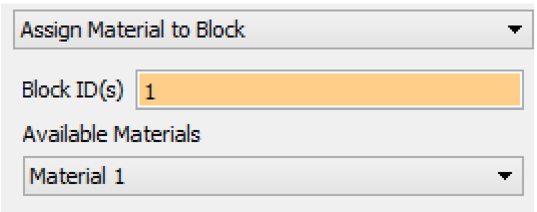


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

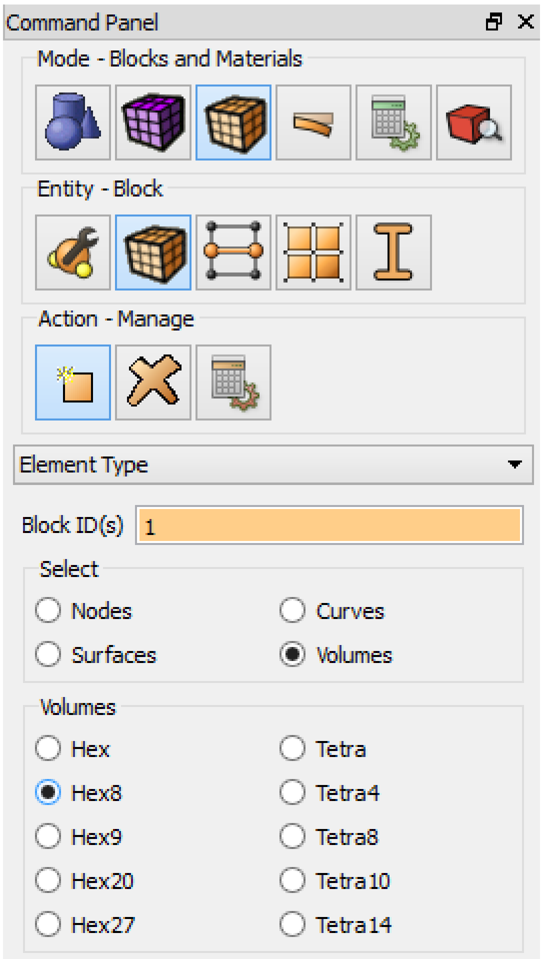


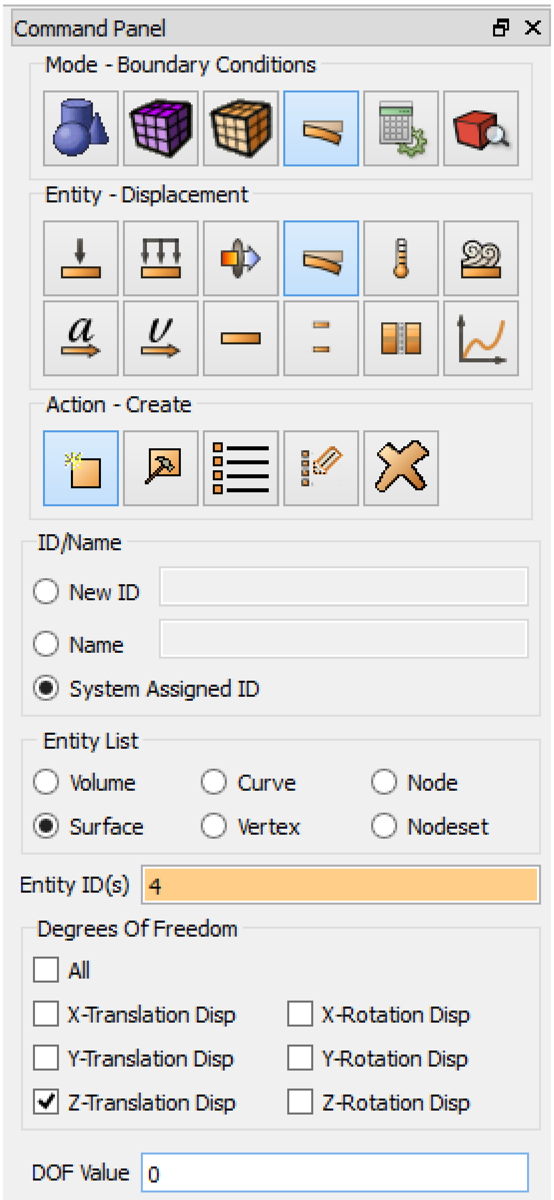
4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Volumes;
- Volumes: HEX8.

Click **Apply**.





Setting boundary conditions

1. Fix one side along X-axis.

Set Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 4 *(or click on the left butt of the beam)*;
- Degrees of Freedom: X-Component;
- DOF Value: 0.

Click **Apply**.

2. Fix two sides along Y axis.

The procedure is similar to the previous step. Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 3 5 *(or sequentially click on the top and bottom edges)*;
- Degrees of Freedom: Y-Component;
- DOF Value: 0.

Click **Apply.**

3. Fix another two sides along Z-axis.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 1 2 *(or sequentially click on two side edges)*;
- Degrees of Freedom: Z-Component;
- DOF Value: 0.

Click **Apply**.

4. Apply pressure to the right butt of the rod.

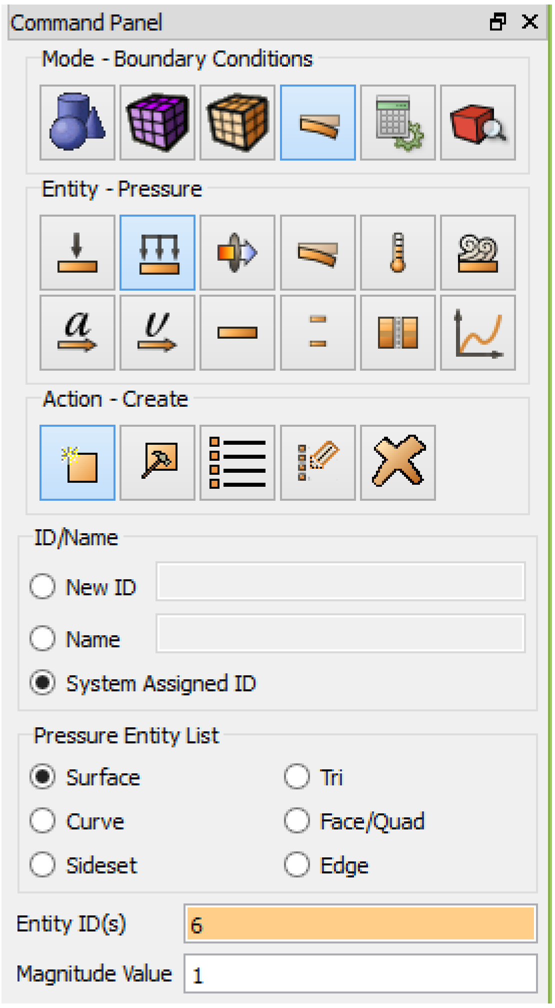
Select Mode – **Boundary Conditions,** Entity – **Pressure**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 6;

Value: 1.

Note: For the dynamic problems it is necessary to create the boundary condition (unit vector) before setting the time dependency.

Click **Apply.**



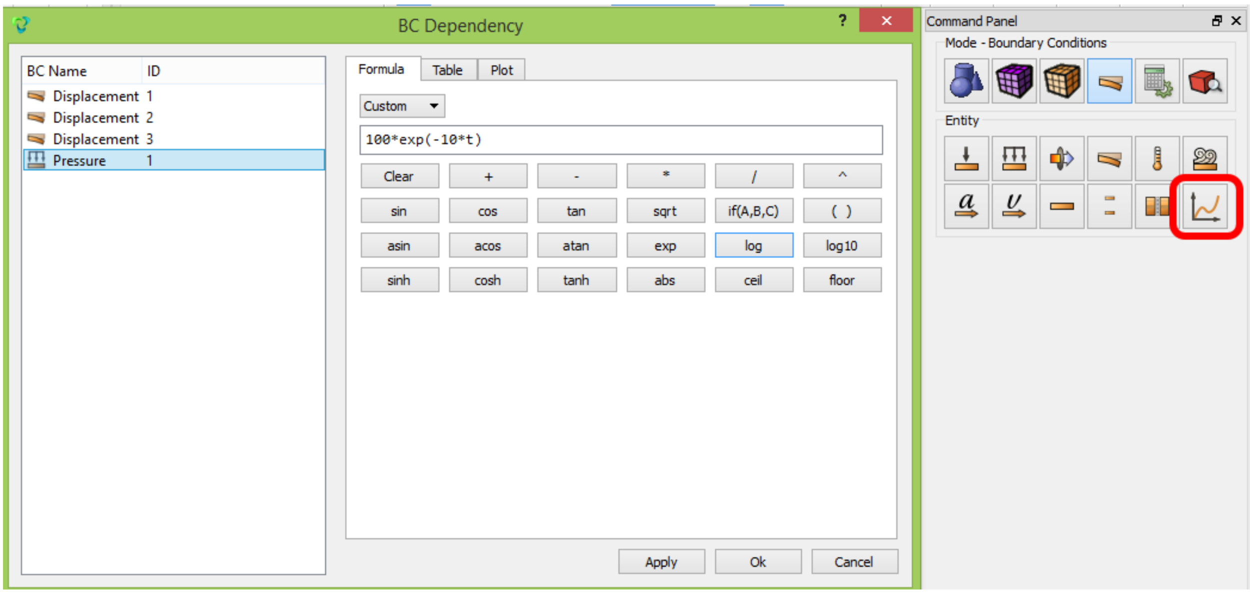
Setting time dependency

1. Creat time dependency.

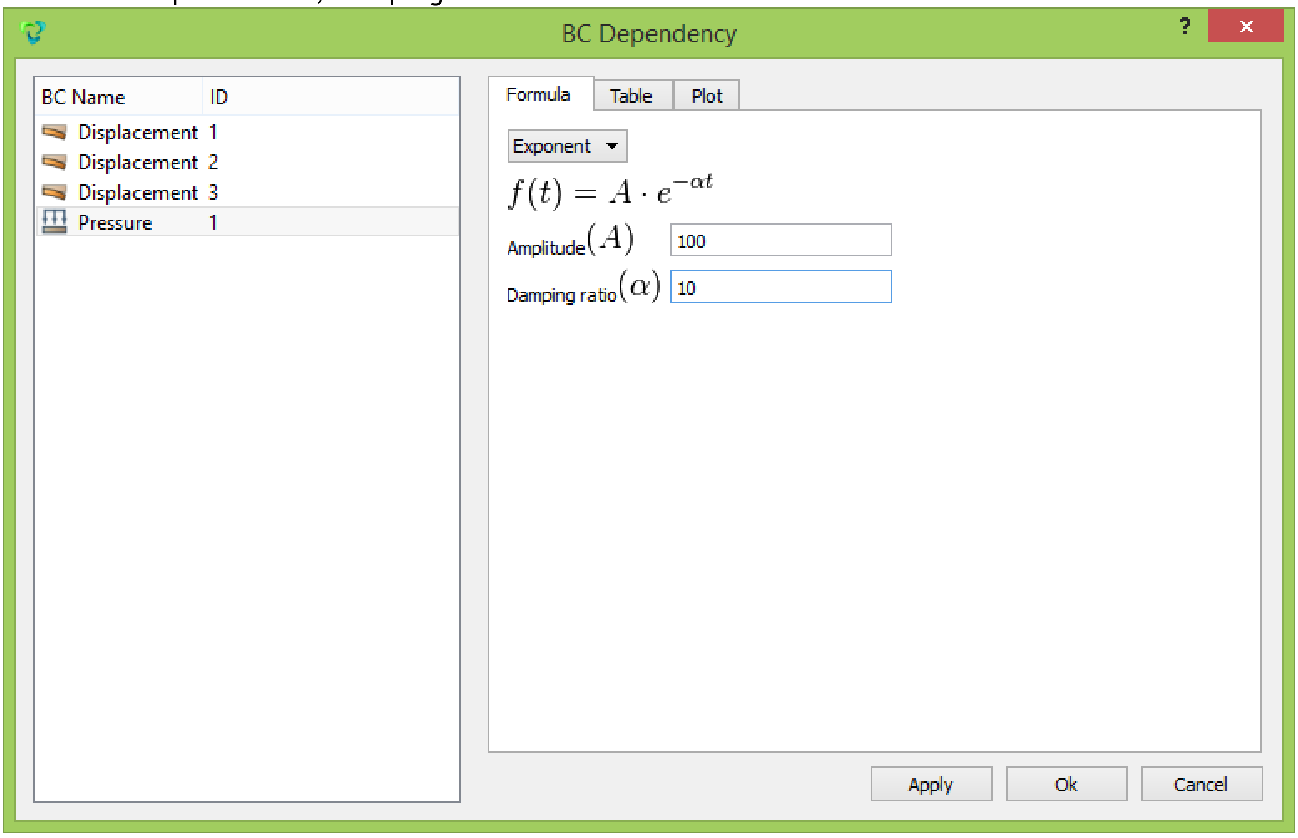
Select Mode – **Boundary Conditions,** Entity – **Creat time dependency** on Command Panel**.** In a pop-up window. Specify the following parameters:

- BC name: Pressure 1;
- Inlay Formula;
- From the drop-down list select: Manually;
- DOF Value: 100*exp(-10*t).

Click **Apply.**



This dependency can be entered using pre-installed functions. In the drop-down list below, select: Exponent and enter Amplitude 100, Damping ratio 10.



Note: After clicking Apply the following command should be entered in Command Line: *bcdep pressure 1 value "100*exp(-10*t)"*. The time dependency is not saved in general.cub-file, thus, it is advisable to keep this command separately for the following running of the saved calculation.

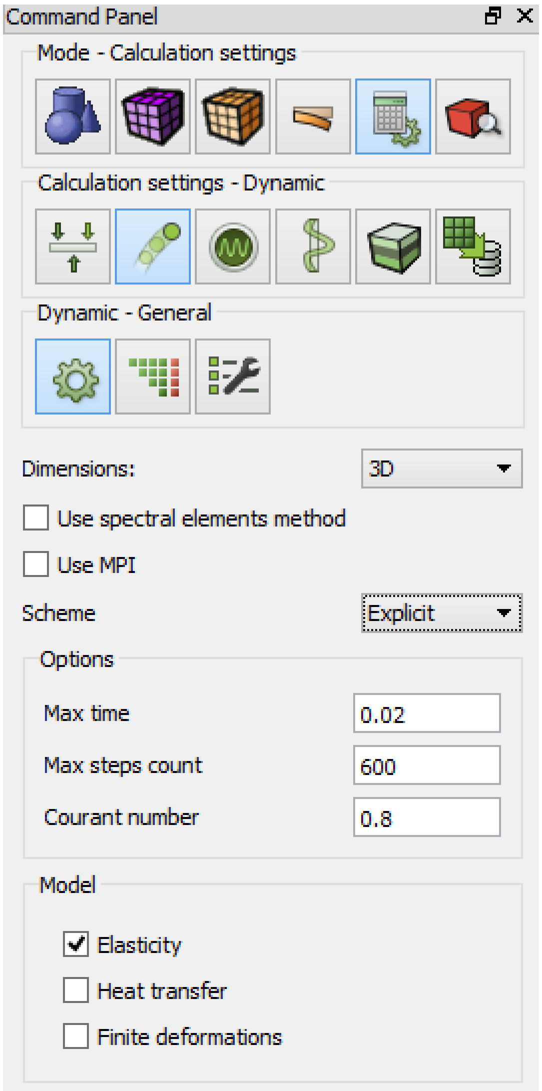
Starting calculation

1. Set the type of the problem to be solved.

Select calculation settings section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Dynamic**, Dynamic – **General**). Set the following calculation parameters:

- Dimension: 3D;
- Scheme: Explicit;
- Max time: 0.02;
- Max steps count: 600;
- Courant number: 0.8;
- Model: Elasticity

Click **Apply.**



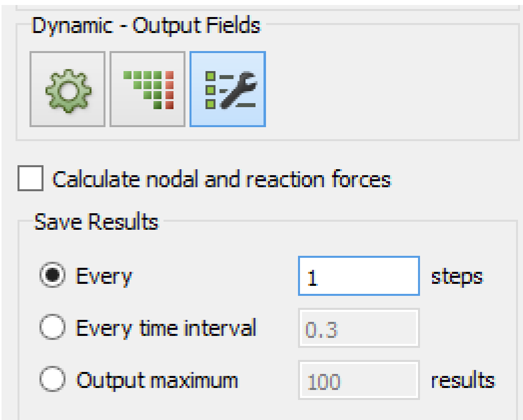
2. Set the parameters output field.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Dynamic**, Dynamic – **Output Fields**). Set the following calculation parameters:

- Output results: Every 1 step

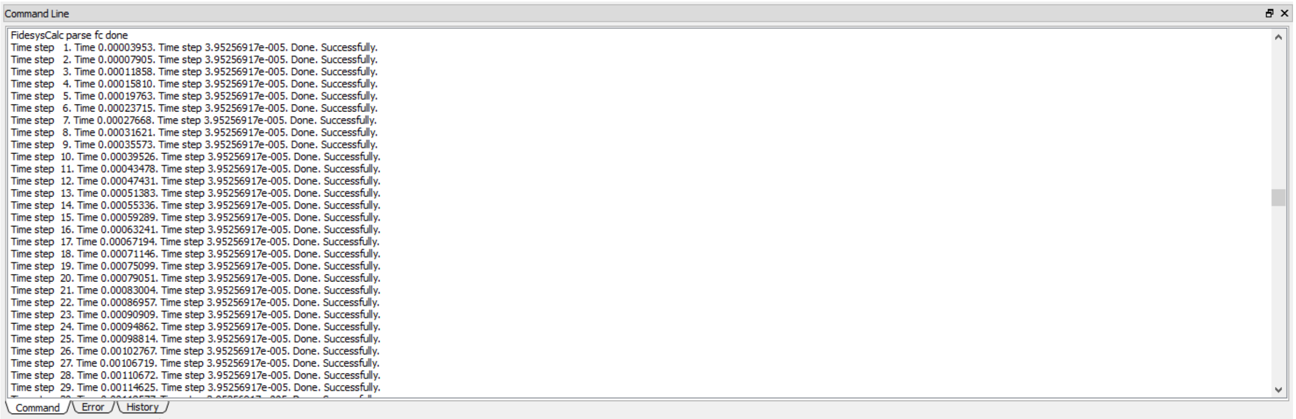
Click **Apply**.

Click **Start Calculation.**



3. In a pop-up window select a folder to save the result and enter the file name.

4. In the process of calculation the number of time step, current time for step and the time step for the step will be displayed in Command Line.



5. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

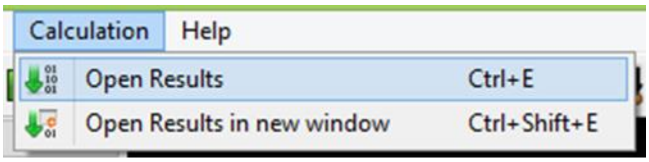
In this case, 506 steps were made to reach the predetermined maximum time of 0.02 seconds.

Results analysis

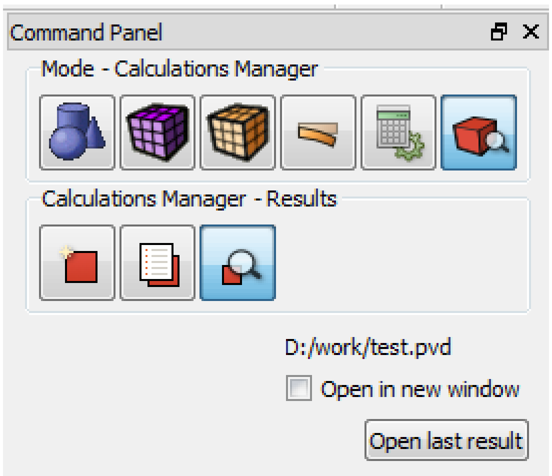
1. Open the file with the results. You can do this in one of the three ways.

   - Click Ctrl+E.

   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.



Select Calculations Manager on Command Panel (Mode — **Calculations Manager**, Calculations Manager — **Results**). Click **Open Results**.



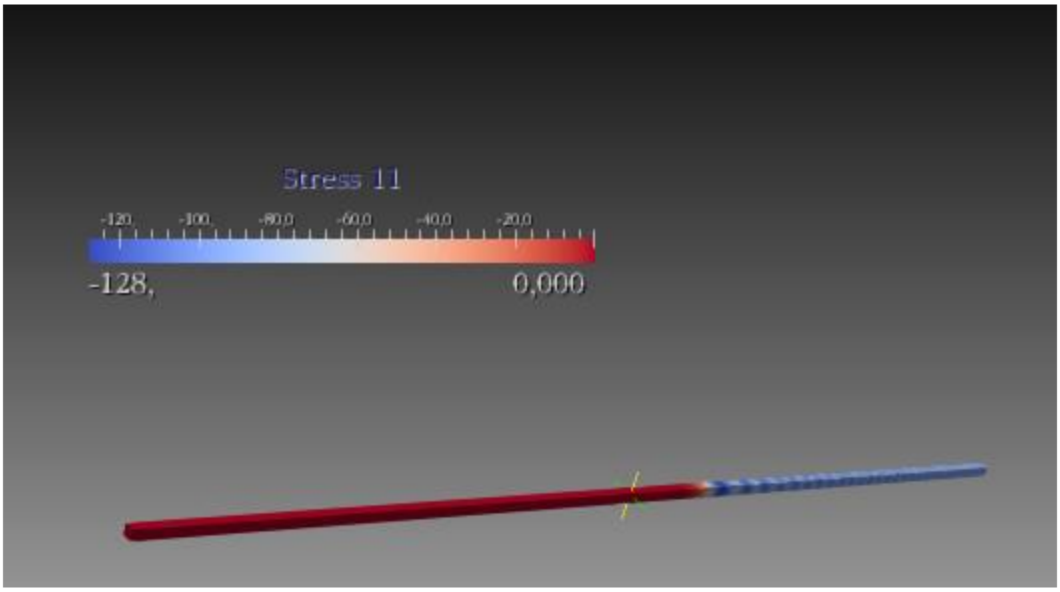
2. Display XX component of the stress field.

In a pop-up ***Fidesys Viewer*** window set the following parameters on Toolbar:

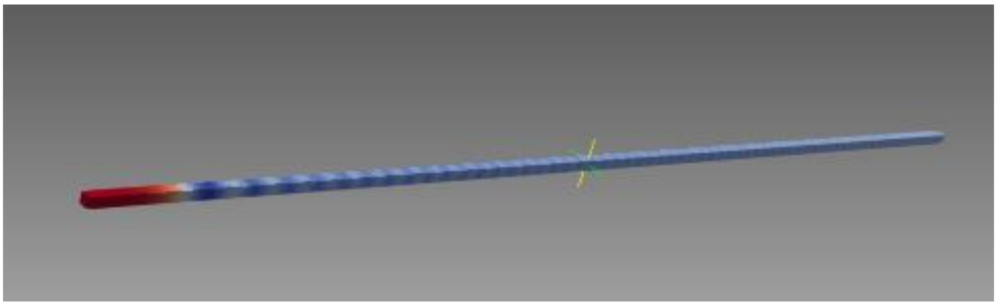
- Representation Mode: Surface;
- Representation Field: Stress;
- Representation Component: 11.

3. There is a menu on Toolbar which allows viewing animation. It consists of a cycle of solutions calculated for every moment of time. Click "Play" to start animation.

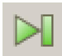
You will clearly see the way the deformation wave is distributing along the rod (see the picture below, screenshot is made for the time step with the number 200).

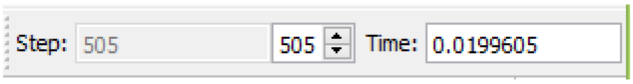


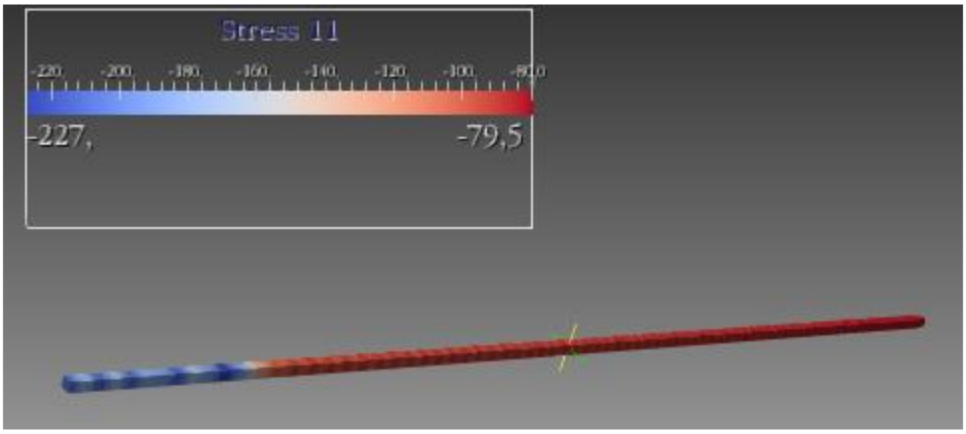
4. Fix the moment when the wave reaches the second end of the rod by clicking “Pause”.



Pay attention to the file number:

Define the total number of files by clicking “Last Frame” 





You can find the wave distribution velocity in the rod accounting for the modeling time indicated when setting calculation (0.02 s) accounting distance of the wave to the opposite beam side and the time which the wave needs to reach the beam side:

$$c = \frac{L}{t} = \frac{100}{\frac{0.02}{506} \cdot 432} \approx 5856.48$$

The difference between this value and the one obtained analytically is 2.43%.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd)

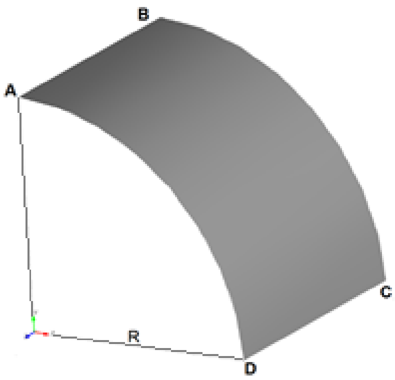
```
reset
set node constraint on
brick x 100 y 1 z 1
volume 1 size 0.5
volume 1 size 0.5
mesh volume 1
undo group begin
create material "Material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "Material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3 "DENSITY"
7900
undo group end
set duplicate block elements off
block 1 volume 1
block 1 material 'Material 1'
block 1 element type hex8
create displacement on surface 4 dof 1 fix 0
create displacement on surface 3 5 dof 2 fix 0
create displacement on surface 1 2 dof 3 fix 0
create pressure on surface 6 magnitude 1
bcdep pressure 1 value '100*exp(-10*t)'
analysis type dynamic elasticity dim3
dynamic scheme explicit maxtime 0.02 maxsteps 600 courant 0.8
spectralelement off
usempi off
output nodalforce off midresults on results everystep 1
output nodalforce off midresults on results everystep 1
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

It is also possible to run the *Example_6_Dynamic_3D.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Buckling (shell model)

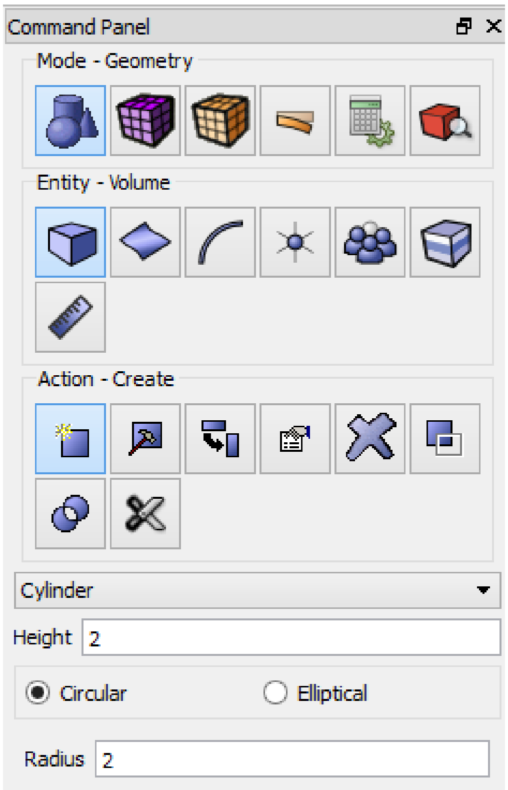
S.P. Timoshenko, J.M Manages "Theory of elastic stability" second edition. Dunod, 1966, 500 pages

The problem of cylindrical shell buckling under the pressure uniformly distributed over the entire surface is being solved.



The picture represents a geometric model of the problem: R = 2 m, L = 2 m, thickness h = 0.002 m. Due to the symmetry of the problem, the ¼ part of the cylinder is regarded. Constraints on the lines AB and CD are due to the conditions of symmetry; a uniformly distributed load on the surface is ABSD q = 1 kPa. The material parameters are E = 200 GPa, ν = 0.3.

It is necessary to compare the first three critical values.

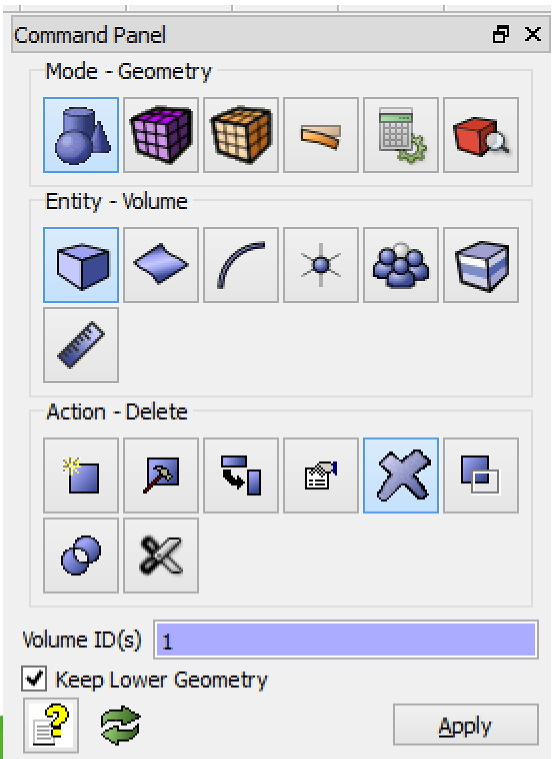


Geometry creation

1. Create a cylinder with radius of 2 m and length of 2 m.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Creat leaving **Circular** at the base. Set radius of 2 and height of 2

Click **Apply.**

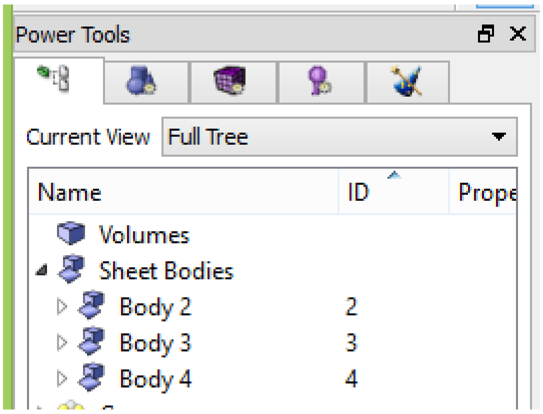


2. Get the cylindrical shell out of the volumeric cylinder.

Select the volume removing section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Delete**). Enter the number of the created volume – 1 into the field **Volume ID(s)**. Put a tick against **Keep lower geometry**.

Click **Apply.**

As a result, three plane bodies (Body 1, Body 2, Body 3) are obtained. This will be displayed in the Model Tree.

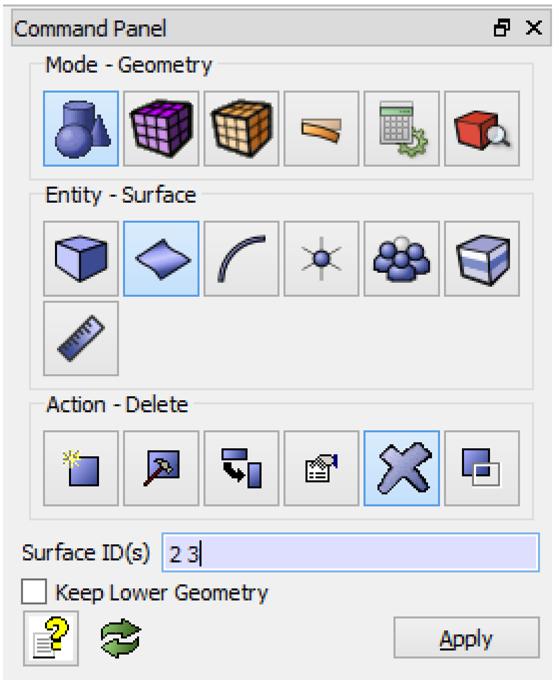


Delete side surfaces Body 3 and Body 4.

Select the surface removing section on Command Panel (Mode — **Geometry**, Entity — **Surface**, Action — **Delete**). Enter numbers 2 3 in the window **Surface ID(s)**.

Click **Apply.**

As a result, only the lateral cylindrical shell of 2 m radius and 2 m high will remain of the initial volume.

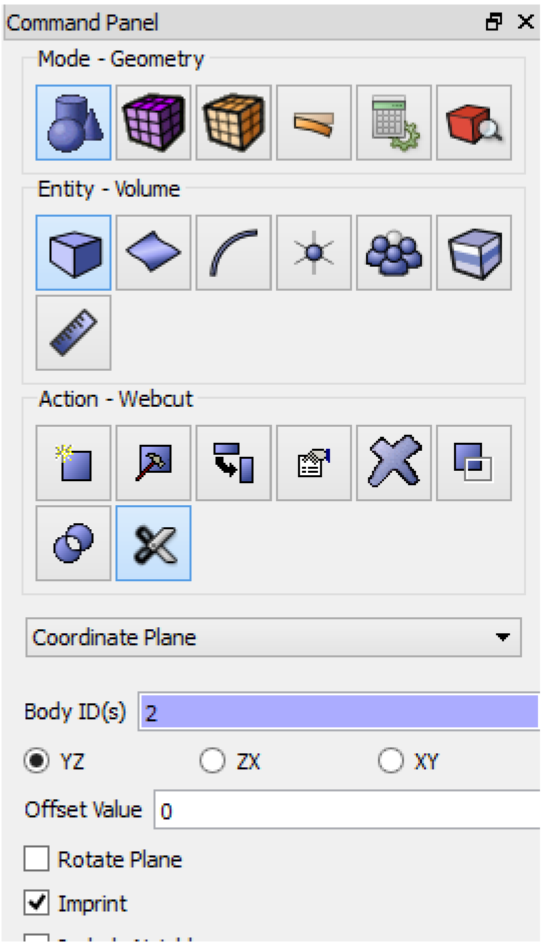


3. Leave a quarter of a shell (symmetry of the problem).

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Webcut**). Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

- Body ID: 2 (*the body to be webcut*);
- Webcut with: YZ Plane;
- Offset value: 0;
- Imprint.

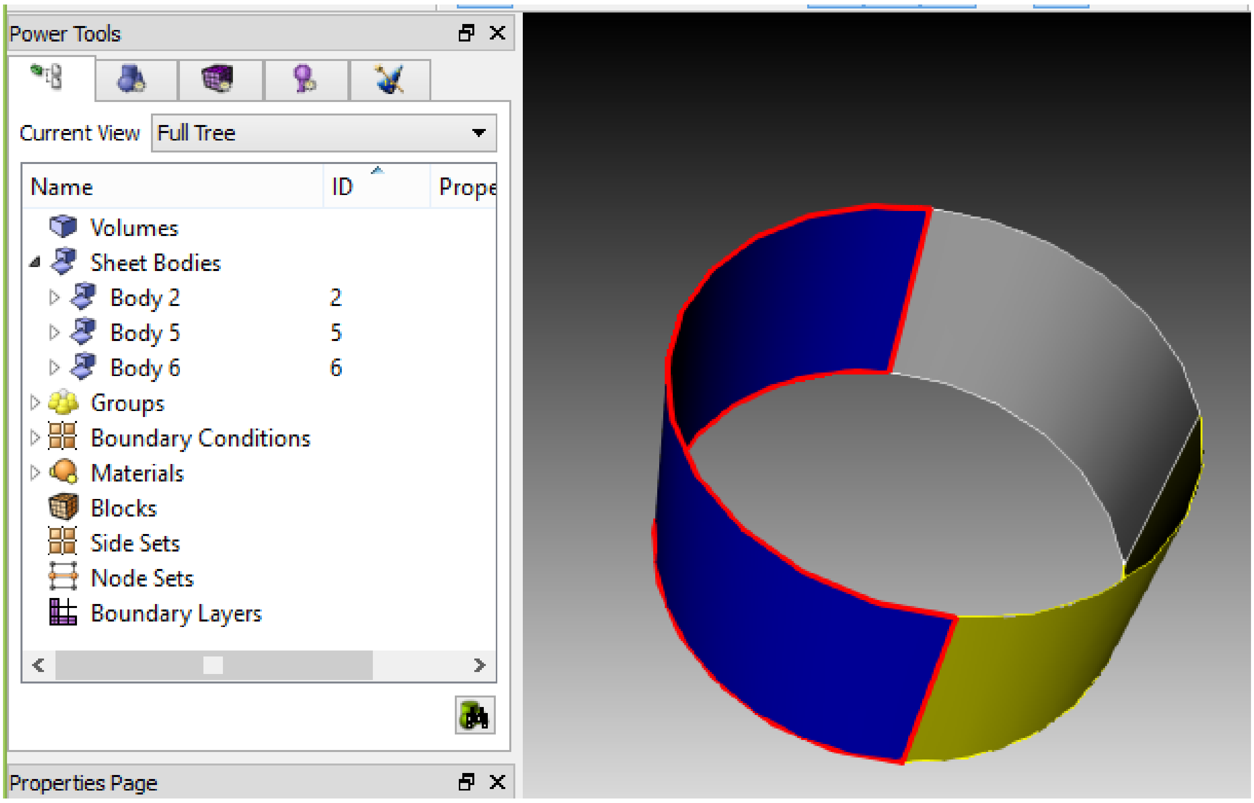
Click **Apply.**



Do the same for the ZX Plane:

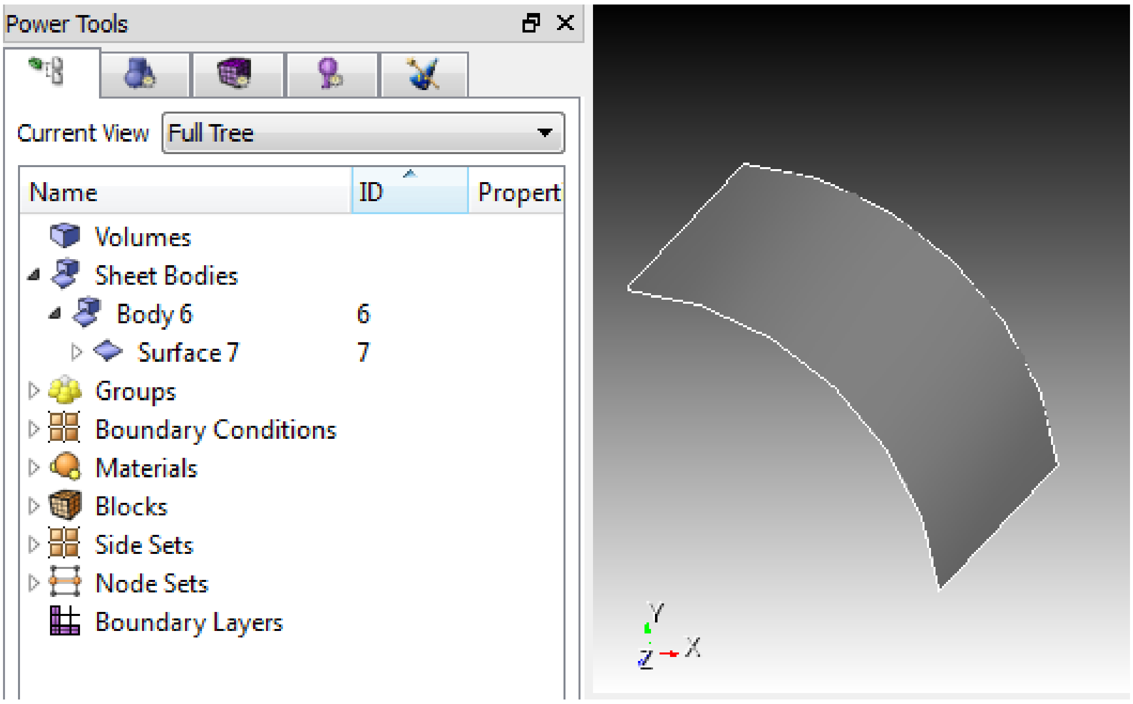
- Volume ID(s): 2 *(the volume to be webcut);*
- Webcut with: ZX Plane;
- Offset value: 0;
- Imprint.

Click **Apply.**



As a result, the original volume in the Model Tree is split into three (Body 2, Body 5 and Body 6).

Delete the bodies 2 and 5. To do this, select these bodies in the Model Tree holding down the Ctrl key and click **Delete** in contextual menu. As a result, a quarter of the original shell is left (Body 6):



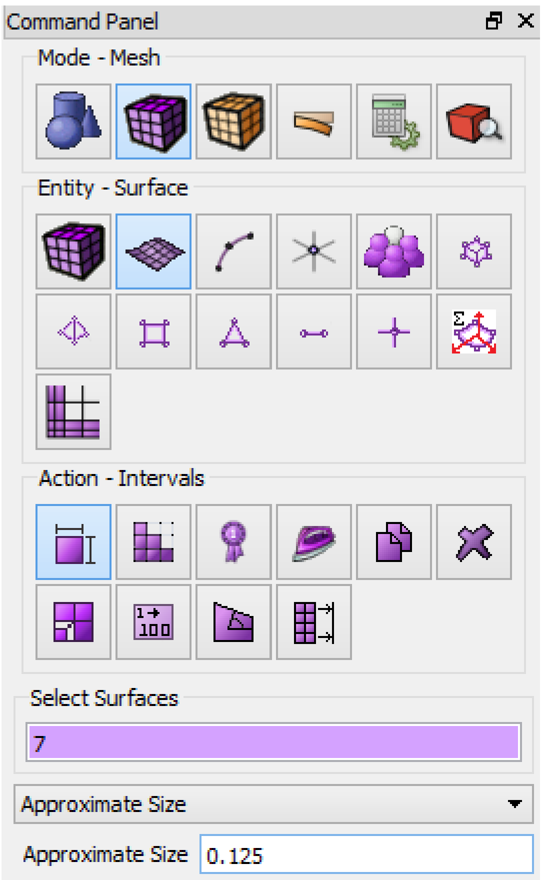
Meshing

1. Create a quadrangular mesh.

Select meshing on plane section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Intervals**). Specify the parameters of mesh refinement:

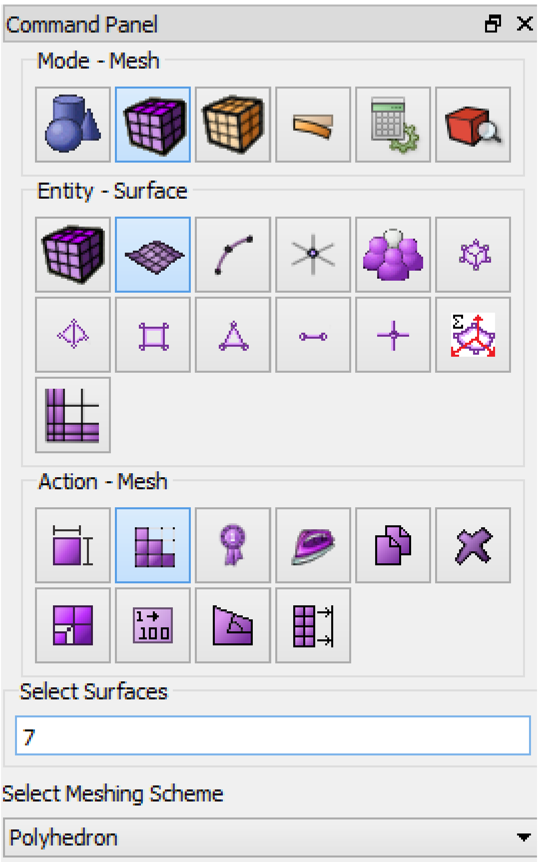
- Select surfaces: 7;
- The way of meshing: Approximate Size;
- Approximate Size: 0.125.

Click **Apply.**



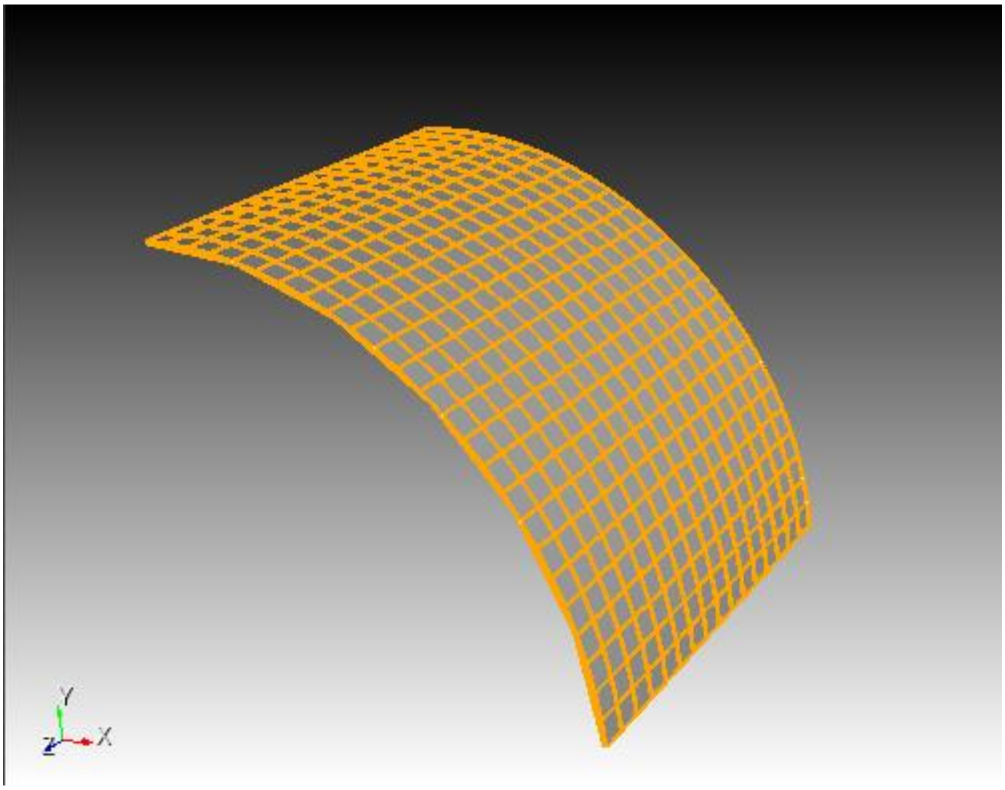
Select meshing on plane section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action –**Meshing**). Select meshing scheme:

- Select surfaces: 7;
- Select meshing scheme: Polyhedron;



Click **Apply**.

Click **Mesh**.



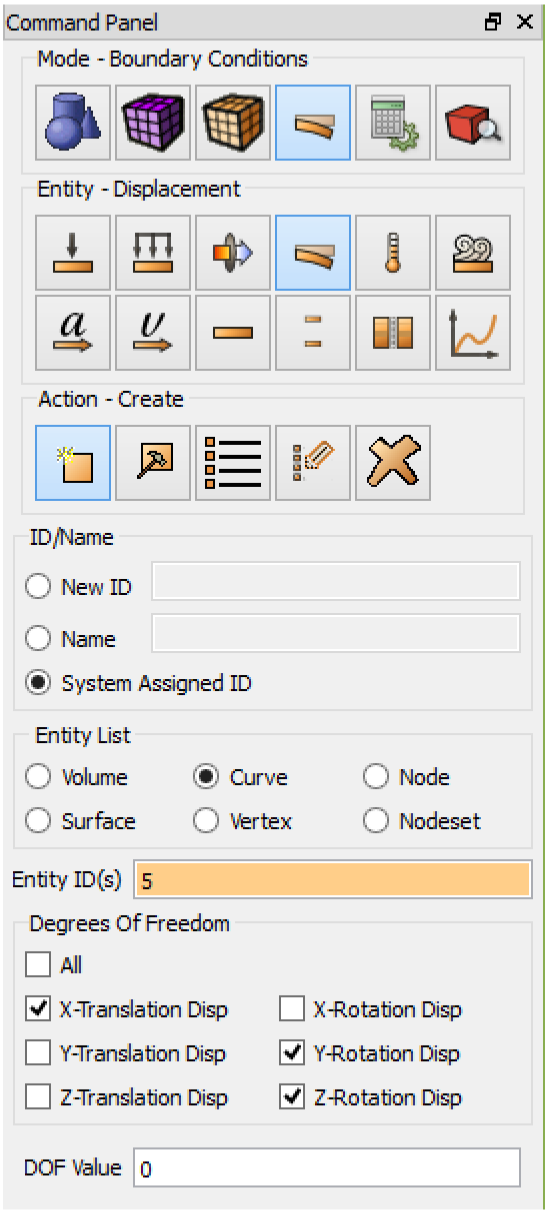
Setting boundary conditions

1. Fix the line AB on the conditions of symmetry.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entities ID: 5 (or click on the top line on a quarter of the shell);
- Degrees of Freedom: X-Translation, Y-Rotation, Z-Rotation;
- DOF Value: 0.

Click **Apply.**

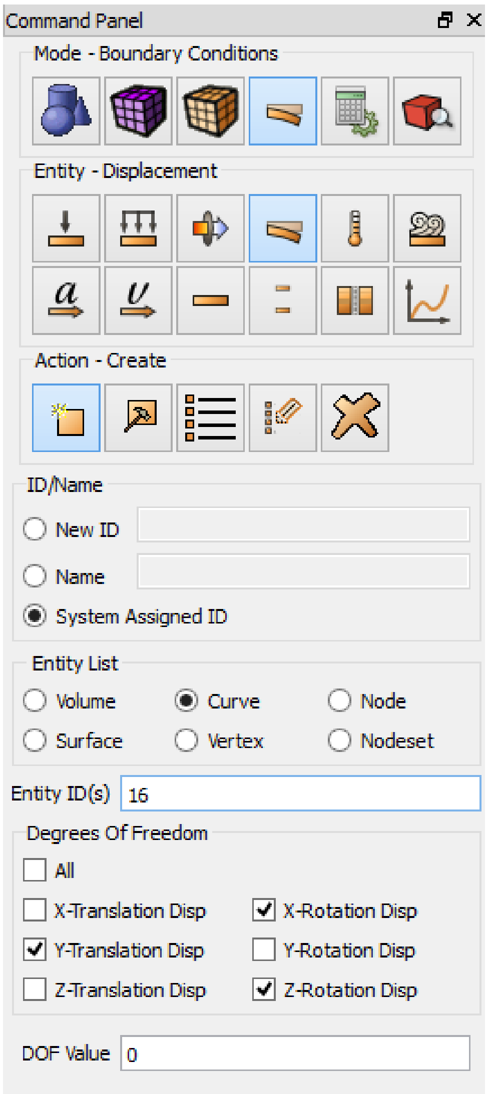


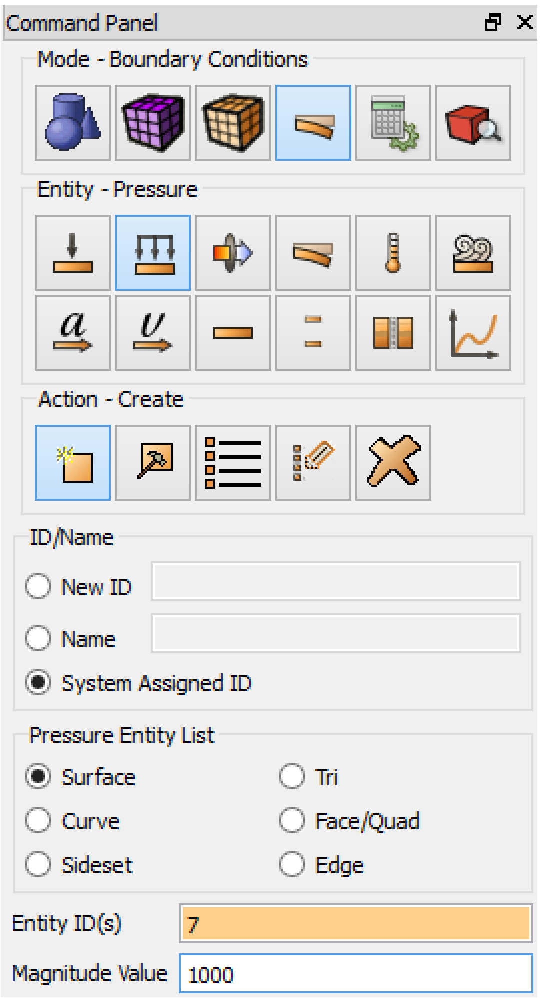
2. Fix the line CD of the conditions of symmetry.

Select Mode — **Boundary Conditions,** Entity — **Displacement**, Action — **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entities ID: 16 (or click on the lower line on a quarter of the shell);
- Degrees of Freedom: Y-Translation, X-Rotation, Z-Rotation;
- DOF Value: 0.

Click **Apply.**





Apply pressure to the entire surface of the shell.

Select Mode — **Boundary Conditions,** Entity — **Pressure**, Action — **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 7 (or click on the surface of the shell);
- Value: 1000.

Click **Apply.**

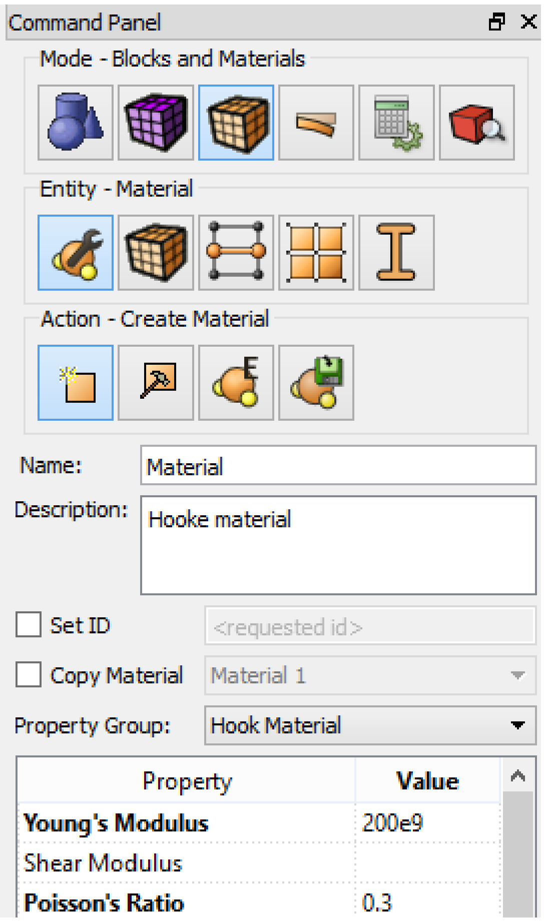
Setting material and element type

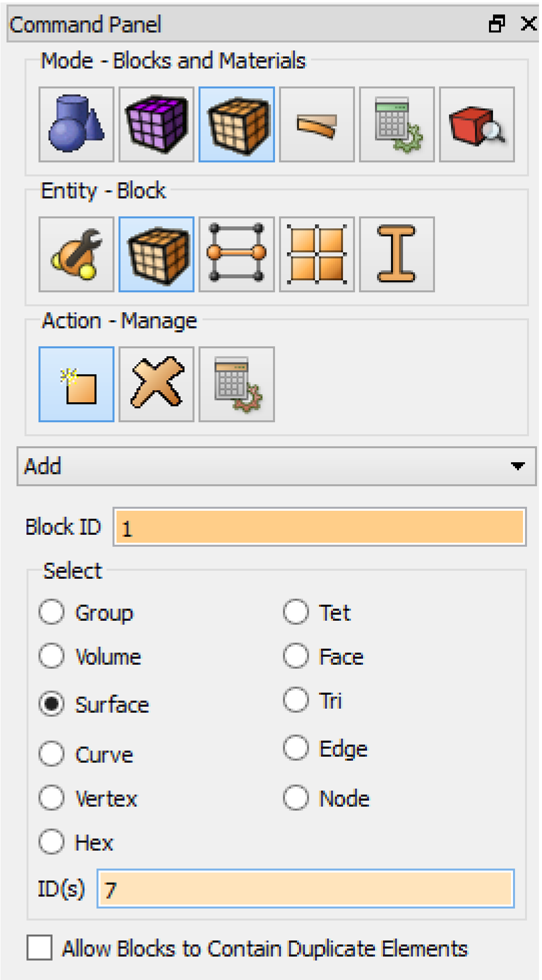
1. Create the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Material**, Action — **Create Material**). Set the following parameters:

- Name: Material 1;
- Property group: Hook material;
- Young's Modulus: 200e9;
- Poisson's Ratio: 0.3.

Click **Apply**.





2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: **Surface**;
- ID: 7 *(or by the command **all**)*.

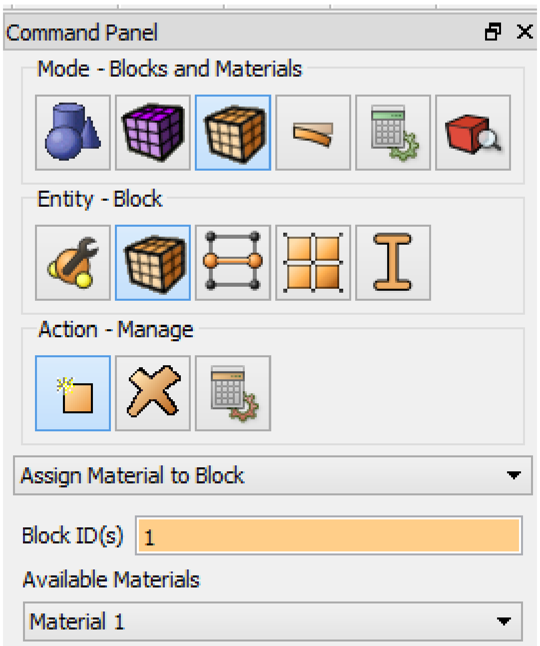
Click **Apply**.

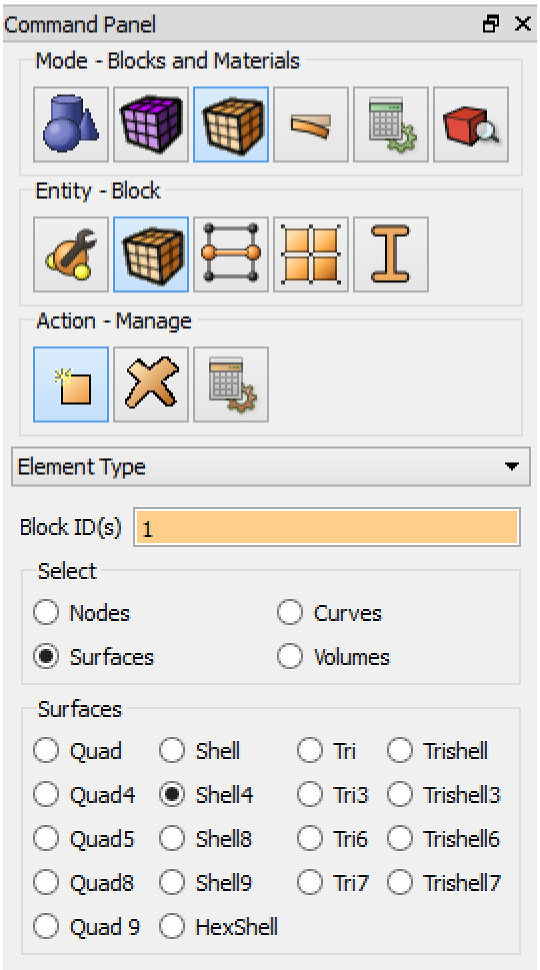
3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.





4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: **Surfaces**;
- Elements: Shell4.

Click **Apply**.

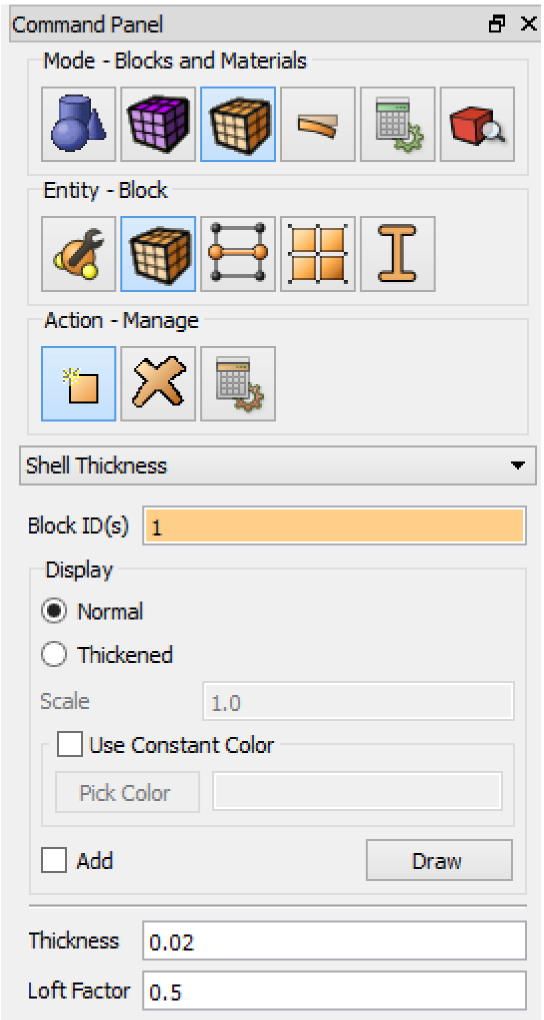
Setting shell thickness

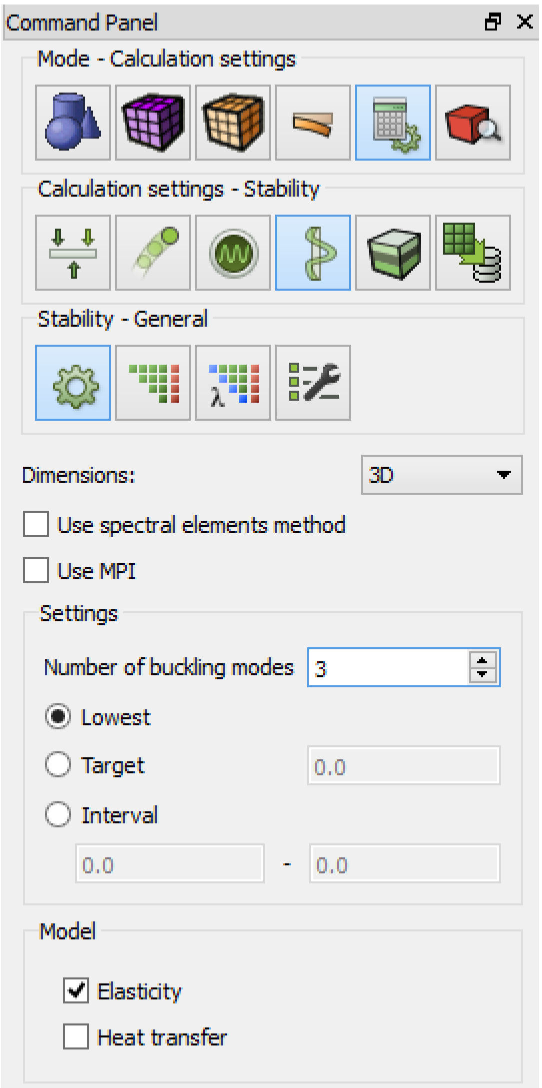
1. Set the shell thickness.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action – **Manage)**. Select **Shell thickness** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Thickness: 0.02;
- Loft Factor: 0.5;

Click **Apply**.





Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Stability**, Stability — **General**). Select 3 in the field **Number of buckling modes**. Leave other parameters by default. Click **Apply**. Click **Start calculation**.

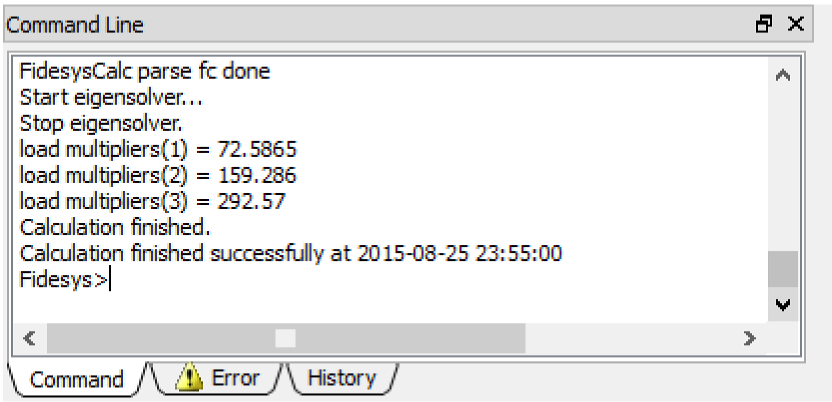
In a pop-up window select a folder to save the result and enter the file name.

If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis

1. Compare the obtained results.

The first three critical values are displayed in Command Line.

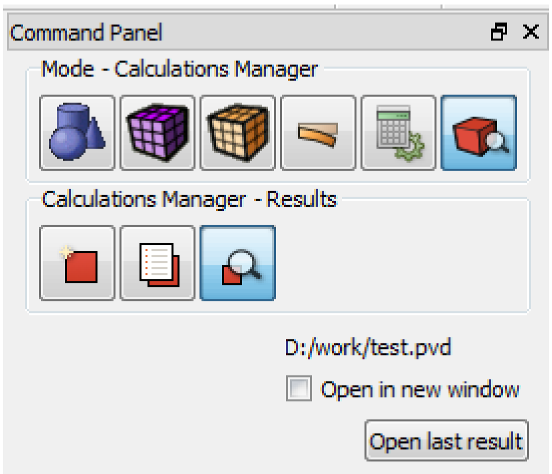


Compare the obtained results with those in the table:

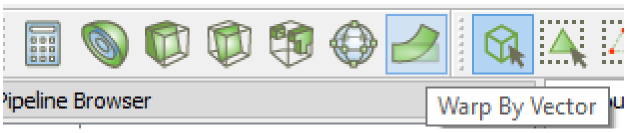
№	Theor. value	FIDESYS	
1	72.260	72.5865	0.45%
2	164.835	159.28	3.37%
3	293.040	292.571	0.16%

2. Open the file with the results. You can do this in one of the three ways.

- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



3. In a pop-up ***Fidesys Viewer*** window select a filter **Warp By Vector**.

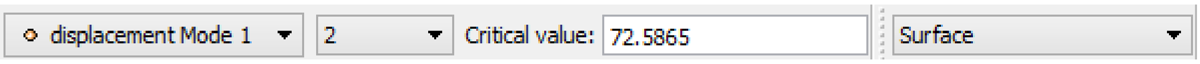


4. In a pop-up filter **Warp By Vector** in the tab **Properties**, set the following parameters:

- **Vectors:** Displacement Mode 1
- **Scale Factor:** 10

5. Display Displacements Mode 1.

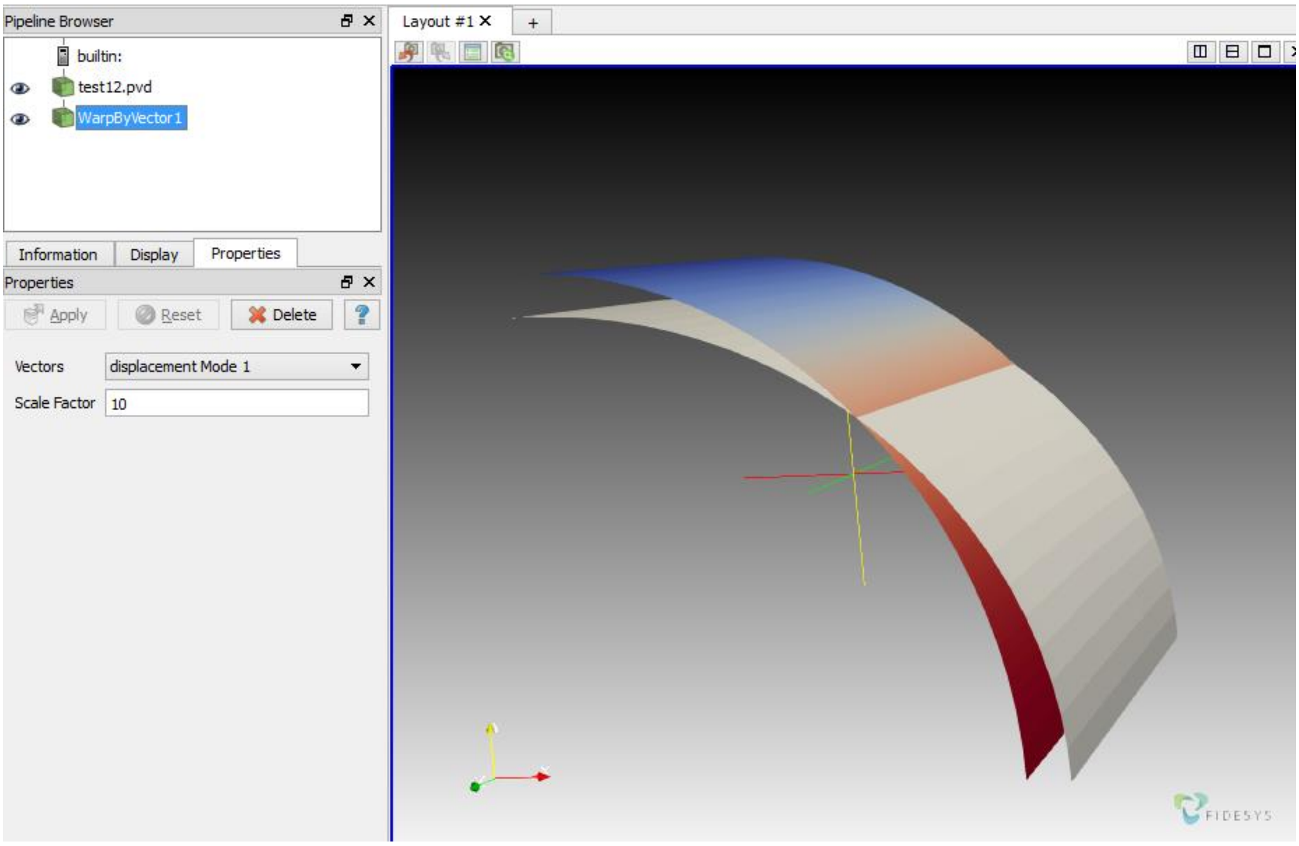
In ***Fidesys Viewer*** window set the following parameters on Toolbar:



Make sure that the first required critical value is displayed in the window **Critical value**.

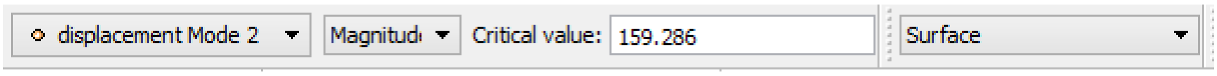
6. View results

As a result, the deformed body is displayed at the picture. To see the original model, click near the model in the Model Tree. The picture below shows the deformed (solid grey filling) and the original model (with the distribution field Displacements for Mode 1).



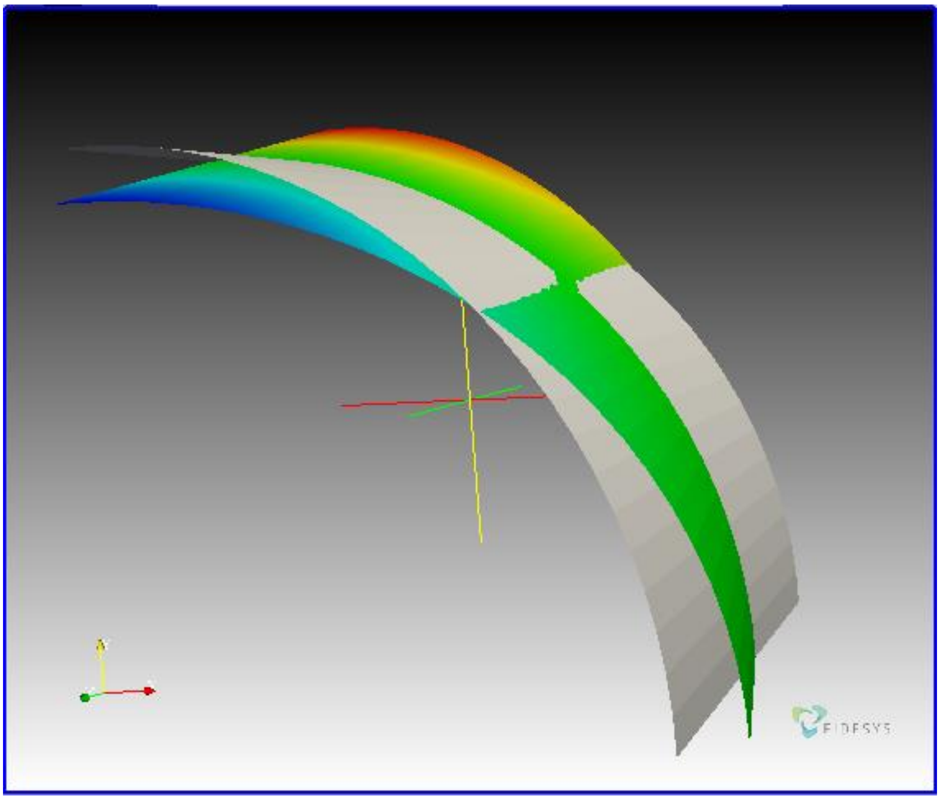
7. Select the filter **Warp By Vector** to do this. Set the following field value Displacements Mode 2 in the tab **Properties**
   - **Vectors:** Displacements Mode 2
   - **Scale Factor:** 10
8. Display Displacements Mode 2.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

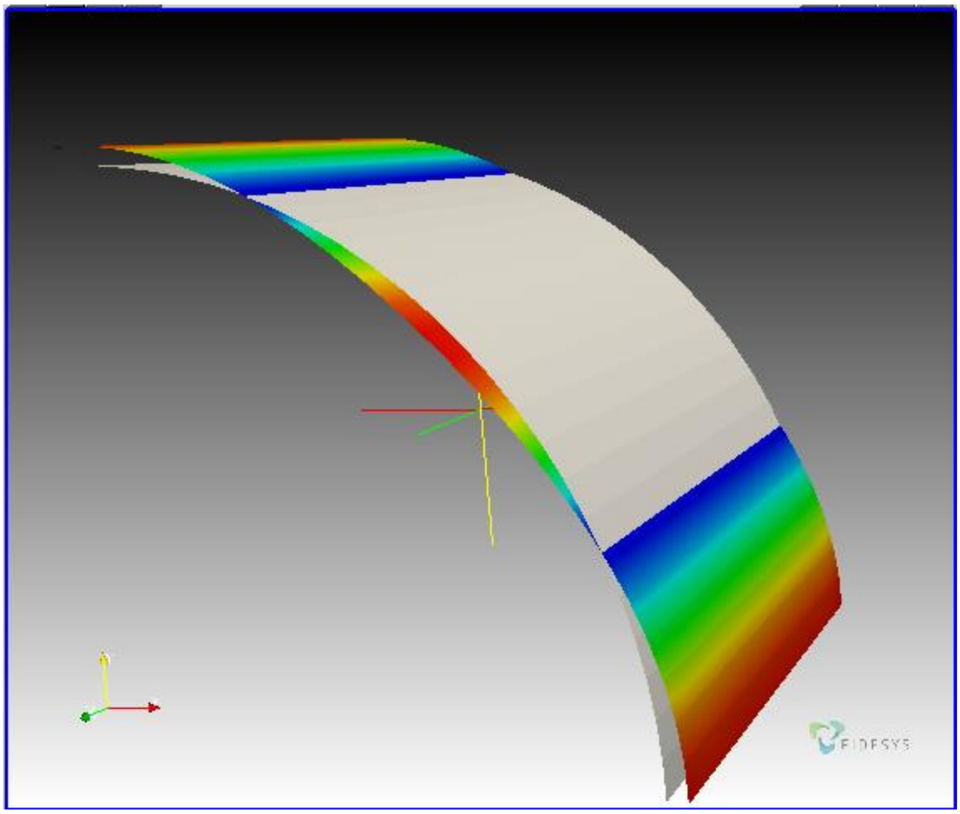


Make sure that the second required critical value is displayed in the window **Critical value**.

9. View results



10. Display Displacements Mode 3 in the same way, make sure that the third required critical value is displayed in the window **Critical value**.



11. Display the 3D-view of the model (shell with thickness).

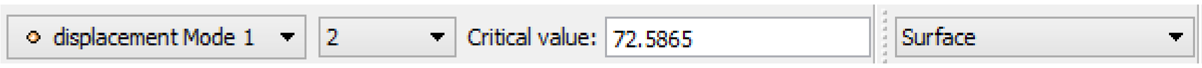
To do this, click on the name of the source file in the Model Tree. After this click 3D-view button in the default string.

The file* _3D.pvd with a 3D-image of the shell must be opened and you will be able to apply various filters to it and to view its deformed view.

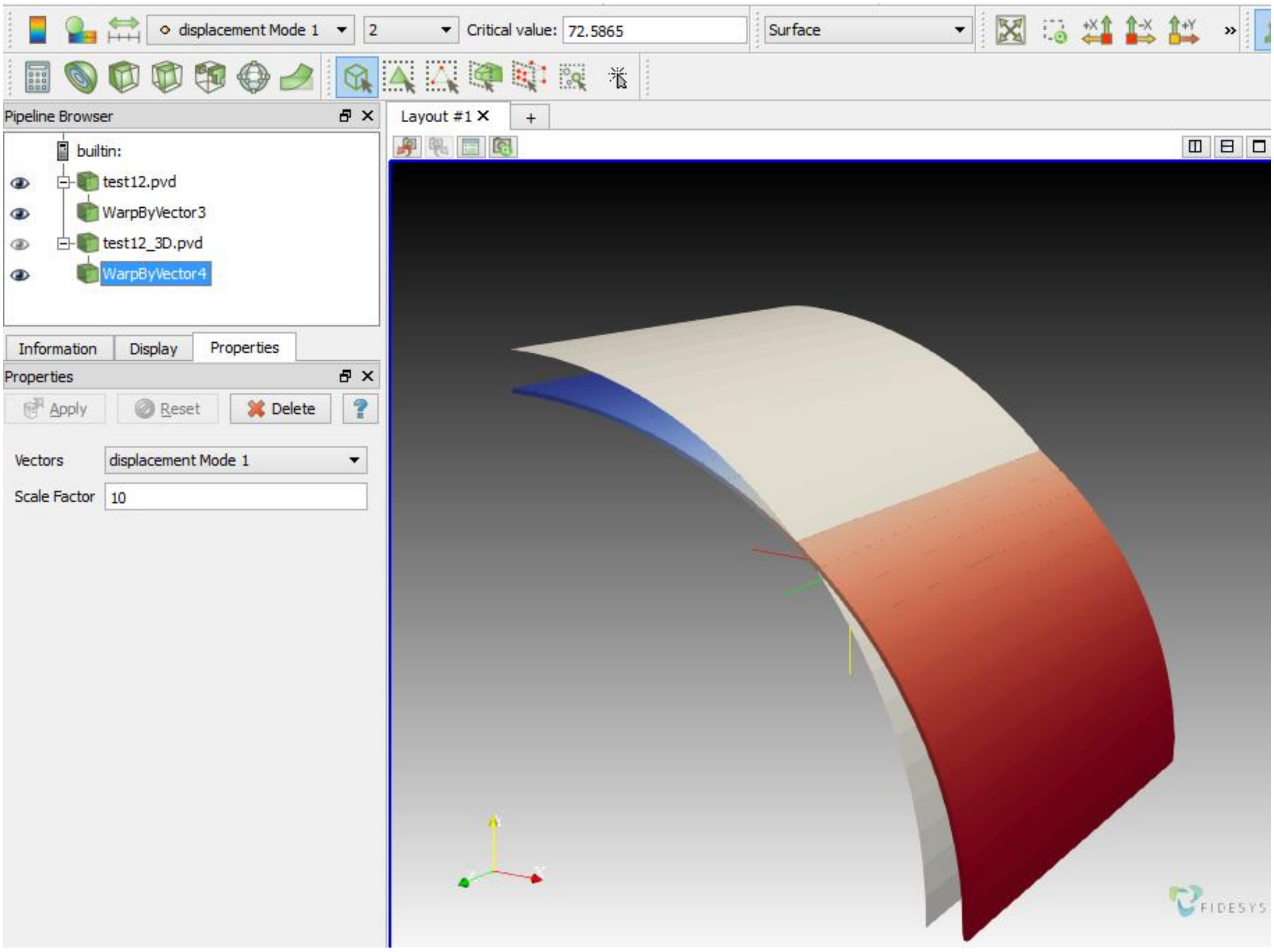
Choose the new file example_3D.pvd in the Model Tree and display Filters **Warp by Vector** for it with the following fields values:

- **Vectors:** Displacements for mode 1
- **Scale Factor:** 10

On the Toolbar, set once again the following parameters for the deformed type:



The first buckling mode will be displayed on the screen but the shell will be enveloped with thickness.

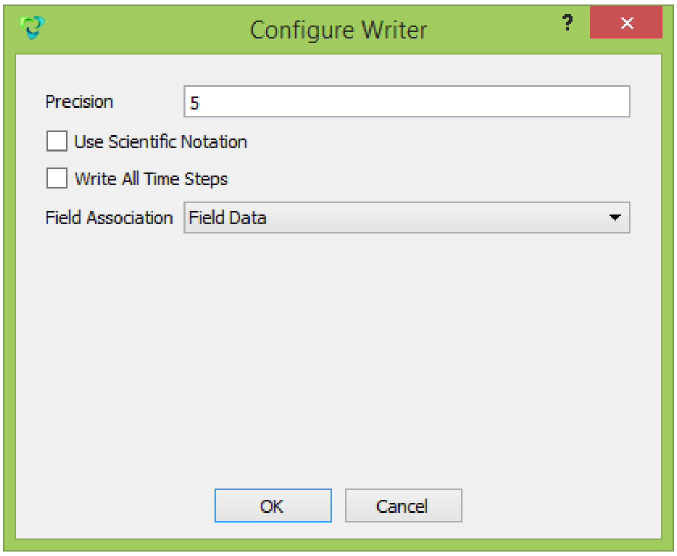


To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

12. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. In the pop-up window select:

- **Field Association**: Field Data



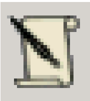
The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd)

```
reset
set node constraint on
create Cylinder height 2 radius 2
delete volume 1 keep_lower_geometry
delete Surface 3 2
webcut body 2 with plane xplane offset 0 imprint preview
webcut body 2 with plane xplane offset 0 imprint
webcut body 2 with plane yplane offset 0 imprint preview
webcut body 2 with plane yplane offset 0 imprint
delete Surface 5 6
surface 7 size 0.125
surface 7 scheme Polyhedron
mesh surface 7
create displacement on curve 16 dof 2 dof 4 dof 6 fix 0
create displacement on curve 5 dof 1 dof 5 dof 6 fix 0
create pressure on surface 7 magnitude 1000
undo group begin
create material "Material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "Material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3
undo group end
set duplicate block elements off
block 1 surface 7
block 1 material 'Material 1'
block 1 element type shell4
undo group begin
block 1 attribute count 2
block 1 attribute index 1 value 0.02
```

```
block 1 attribute index 2 value 0.5
undo group end
analysis type stability elasticity dim3
eigenvalue find 3 smallest
spectralelement off
usempi off
calculation start path "D:/FidesysBundle/calc/example.pvd"
```



It is also possible to run the file *Example_7_Stability_Shell.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Modal analysis (3D)

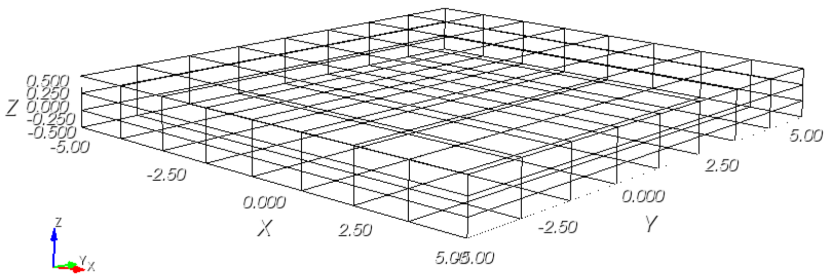
NAFEMS Selected Benchmarks for Natural Frequency Analysis "Simply Supported "Solid" Square Plate", Test No FV52.

The problem of modal analysis of a square plate is being solved.

The picture represents a geometric model of the problem and a mesh:

The size of the plate is 10 m x 10 m x 1 m. Displacements along z-axis are constrained for the edges of the plate bottom side. The material parameters are E = 200 hPa, ν = 0.3, ρ = 8000 kg/m^3.

Eigenmodes from 4 to 10 are to be compared.



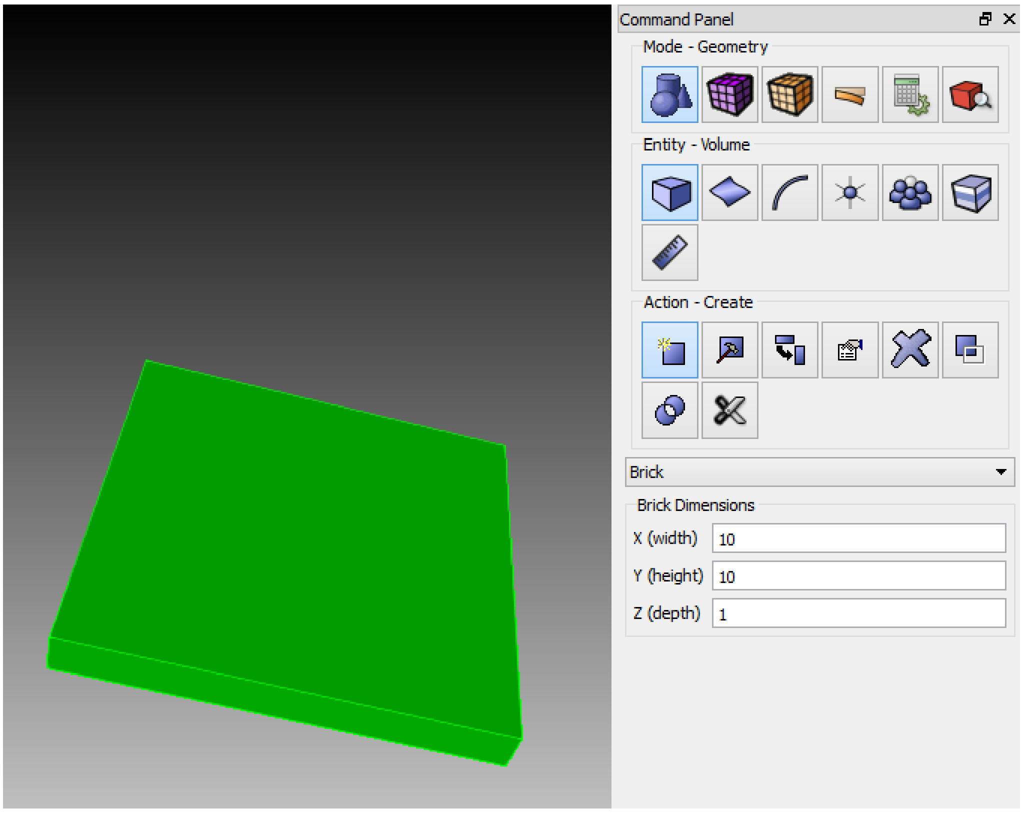
Geometry creation

1. Create the plate.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Brick** in the list of geometric elements. Set the brick dimensions:

- X (width): 10
- Y (height): 10
- Z (depth): 1

Click **Apply**.



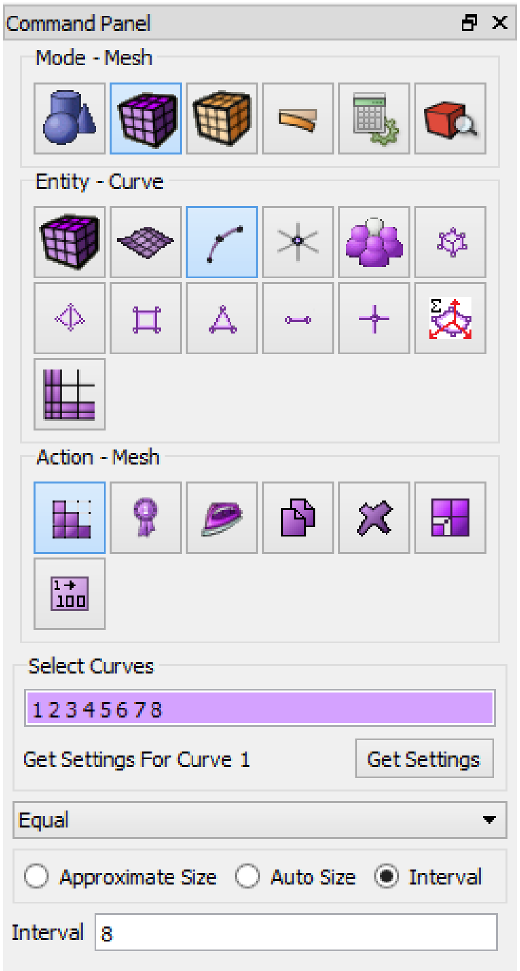
Meshing

A mesh of 8*8*3 linear hexahedral elements is to be generated (as shown at the picture with the problem setting).

1. Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: 1 2 3 4 5 6 7 8 *(using space after each of them)*;
- Select the way of meshing: Equal;
- Select splitting settings: Interval;
- Interval: 8 (see the figure)

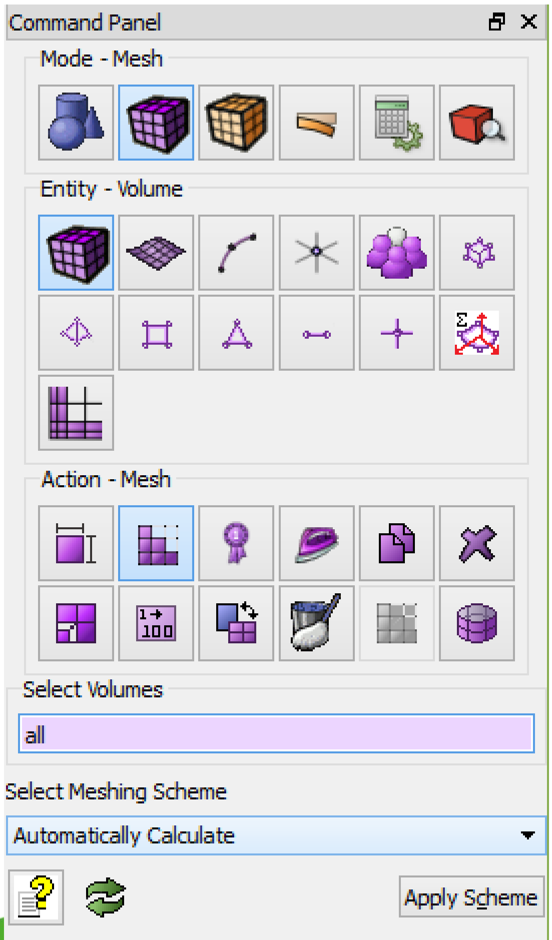
Click **Apply**.



2. Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: 9 10 11 12 *(using space after each of them)*;
- Select the way of meshing: Equal;
- Select splitting settings: Interval;
- Interval: 3.

Click **Apply.**

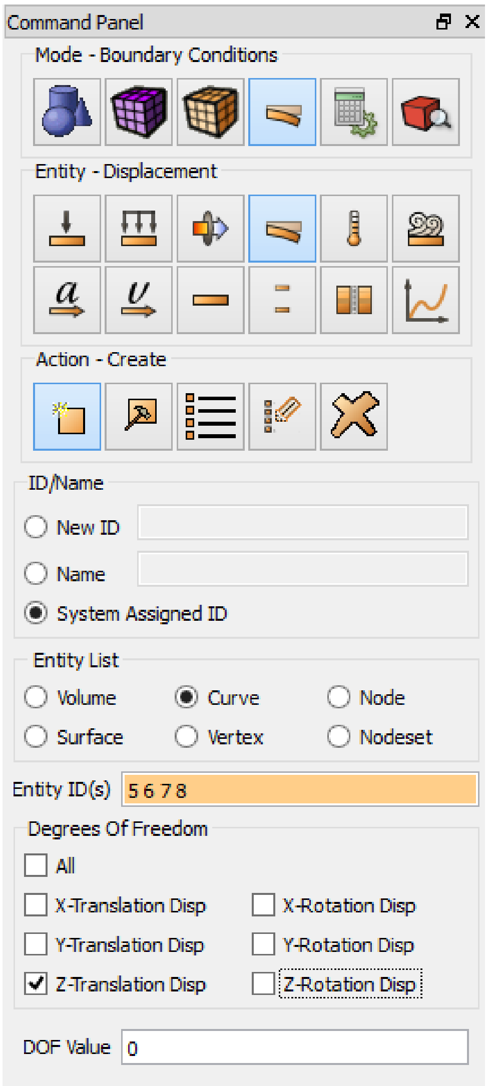


3. Select volume mesh generation section on Command Panel (Mode – **Mesh**, Entity – **Volume**, Action – **Mesh**).

- Select Entities to Mesh (specify their ID): 1 *(or by the command **all**)*
- Select Meshing Scheme: Automatically Calculate

Click **Apply Scheme**.

Click **Mesh**.



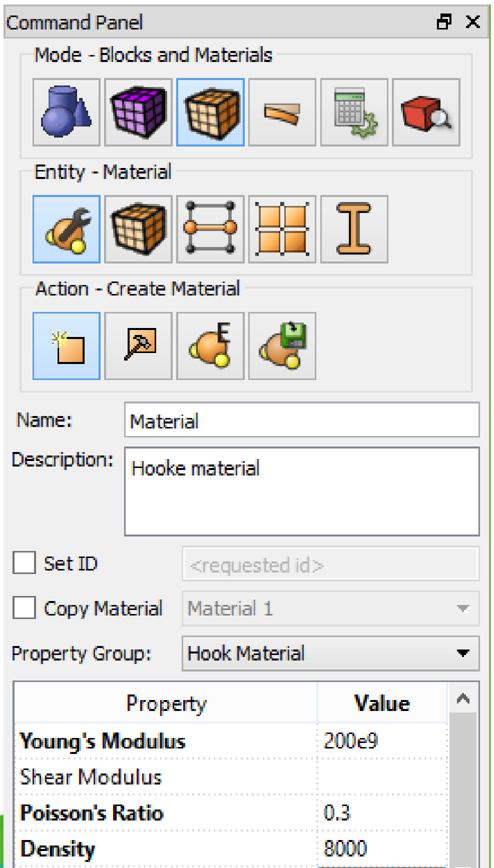
Setting boundary conditions

1. Fix the bottom side edges along Z.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID
- Entity List: Curve
- Entity ID(s): 5 6 7 8 *(using space after each of them)*
- Degrees of Freedom: Z-Translation
- DOF Value: 0

Click **Apply**.



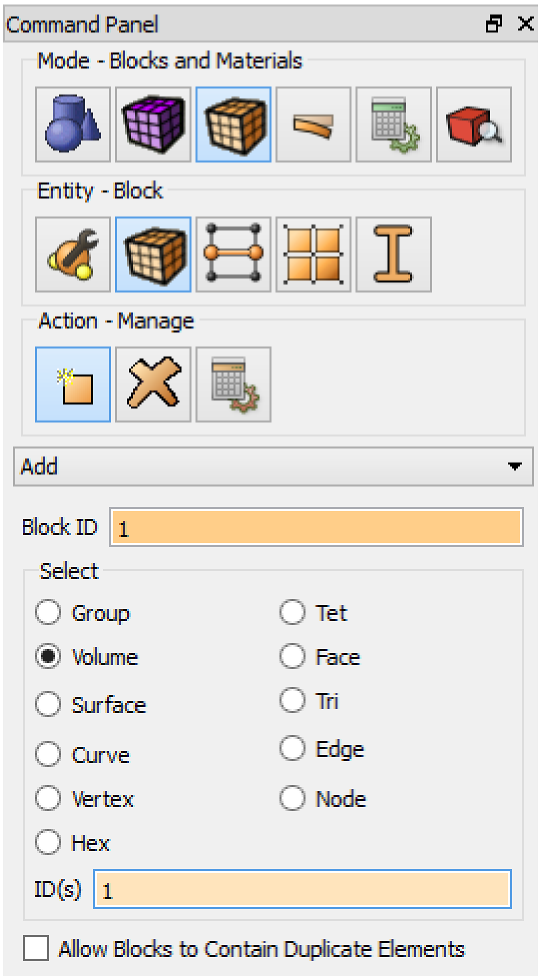
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Property group: Hook material;
- Young's Modulus: 200e9;
- Poisson's Ratio: 0.3;
- Density: 8000.

Click **Apply**.



2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: 1 *(or by the command **all**)*.

Click **Apply.**

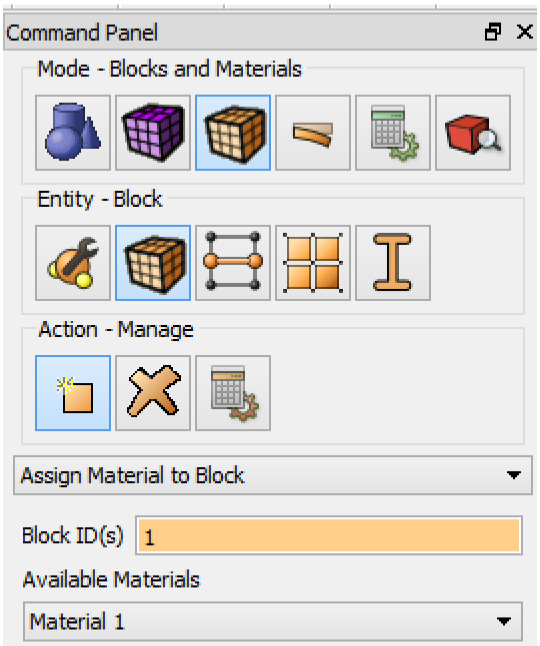
.

3. Assign the material to the block.

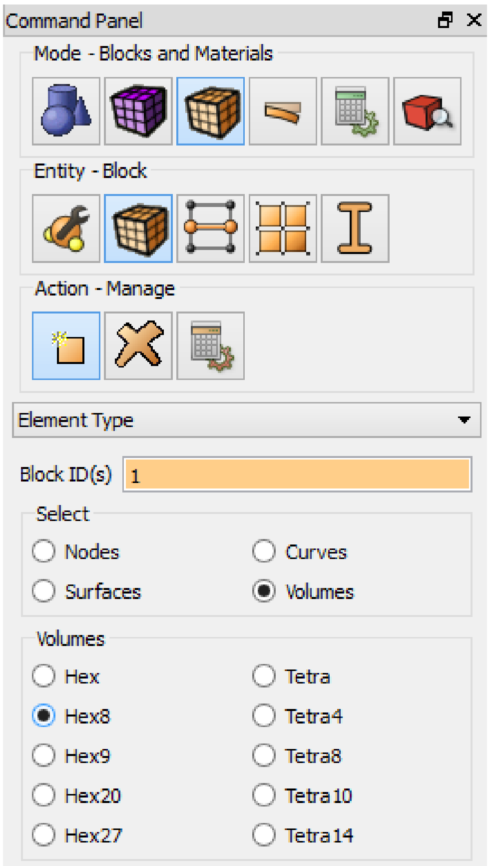
Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.



4. Assign the element type.



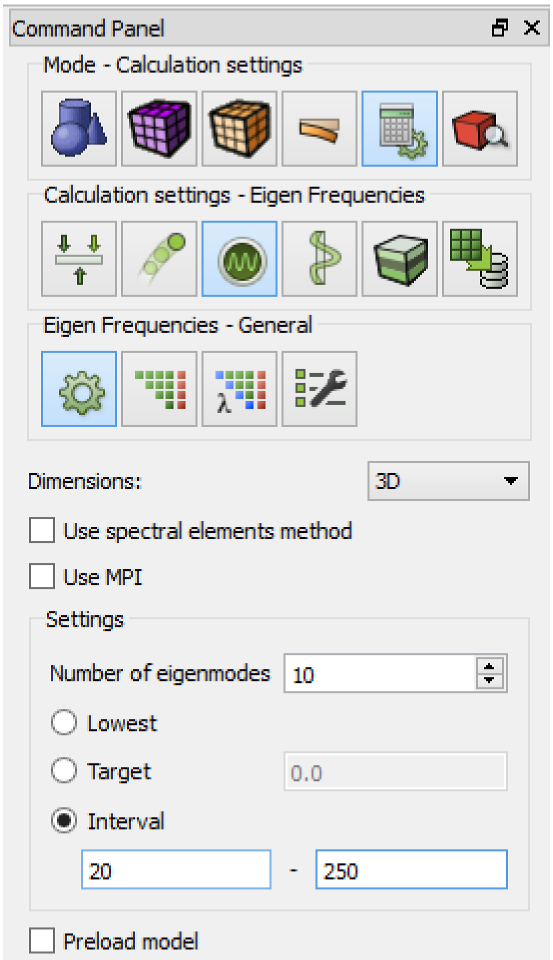
Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Volumes;
- Volumes: Hex8.

Click **Apply**.

Starting calculation

1. Set the type of the problem to be solved.



Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Eigen Frequencies**, Eigen Frequencies – **General**). Specify the following settings:

- Interval: 20 – 250.

Click **Apply**.

Click **Start Calculation**.

2. In a pop-up window select a folder to save the result and enter the file name.
3. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"* as well as the required eigen values and frequencies.

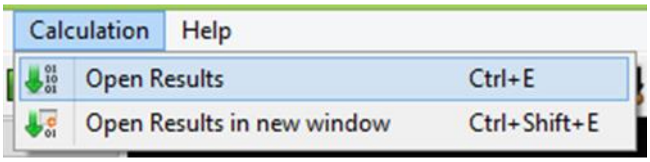
Results analysis

1. Compare the obtained results to those in the given table.

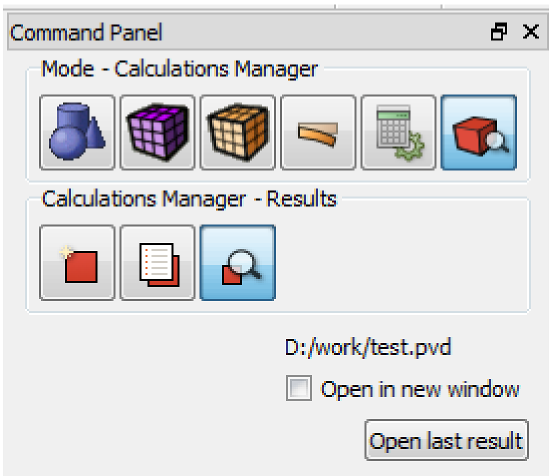
№	NAFEMS	FIDESYS	
		Value, Hz	Error
4	51.65	51.68	0.1%
5	132.73	132.75	0.0%
6	132.73	132.75	0.0%
7	194.37	194.38	0.0%
8	197.18	197.19	0.0%
9	210.55	210.55	0.0%
10	210.55	210.55	0.0%

2. Open the file with the results. You can do this in one of the three ways.

- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.



- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



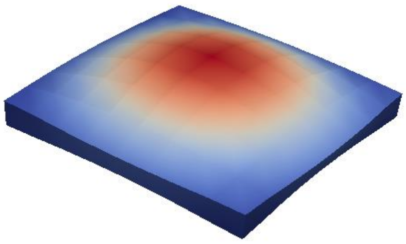
3. You can see the way the body is deformed.

Select a filter **Warp By Vector** to do this. Set the following parameters in the tab **Properties**:

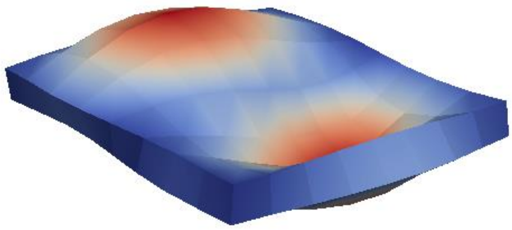
- Vectors: Eigenvalue_# *(# stands for the number of the eigenvalue)*
- Scale Factor: 700

As a result, the deformed body is displayed at the picture. To see the original model, click near it in the Model Tree. The picture below shows the deformed model at different eigenvalues.

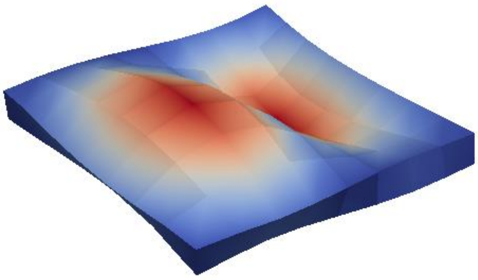
Mode 4



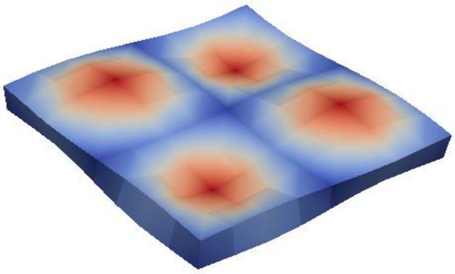
Mode 5



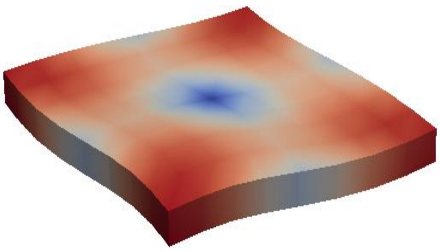
Mode 6



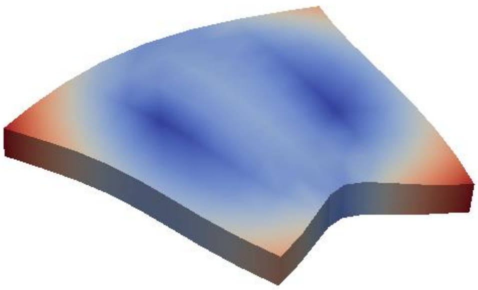
Mode 7



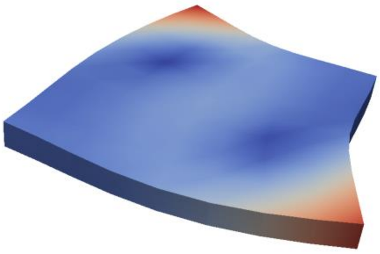
Mode 8



Mode 9



Mode 10



Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
brick x 10 y 10 z 1
curve 1 2 3 4 5 6 7 8 interval 8
```

```
curve 1 2 3 4 5 6 7 8 scheme equal
curve 9 10 11 12 interval 3
curve 9 10 11 12 scheme equal
volume 1 scheme Auto
volume 1 scheme Auto
mesh volume 1
create displacement on curve 5 6 7 8 dof 3 fix 0
undo group begin
create material "material 1 " property_group "CUBIT-FEA" description "Hook material"
modify material "material 1 " scalar_properties "MODULUS" 2e+11 "POISSON" 0.3 "DENSITY"
8000
undo group end
set duplicate block elements off
block 1 volume 1
block 1 material 'material 1'
block element type hex8
block 1 element type hex8
analysis type eigenfrequencies elasticity dim3
eigenvalue find 10 from 20 to 250 preload off
spectralelement off
usempi off
analysis type eigenfrequencies elasticity dim3
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

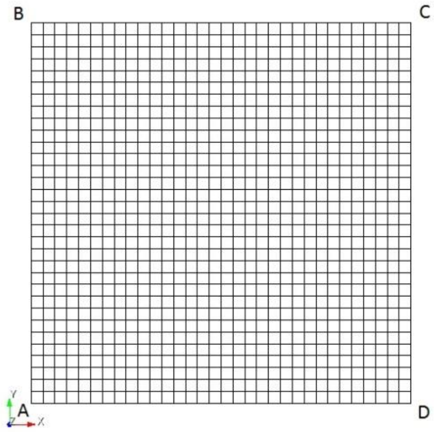
It is also possible to run the file *Example_8_EigenValue_3D.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file .

Modal analysis (shell model)

NAFEMS-Glasgow, BENCHMARK newsletter, Report No. E1261/R002, "Free Vibrations of a Simply-supported Thin Square Plate" , February 1989, p.21.

The problem of modal analysis of a square plate is being solved.

The size of the plate is 10 m x 10 m, the thickness is 0.05 m. X- and Y-Translation and Z-Rotation are constrained for all nodes of the plate. All the edges are constrained in Z-direction. The X-rotation is constrained for edges AB and CD. The Y-rotation is constrained for edges BC and AD. The material parameters are E = 200 hPa, ν = 0.3, ρ = 8000 kg/m^3.



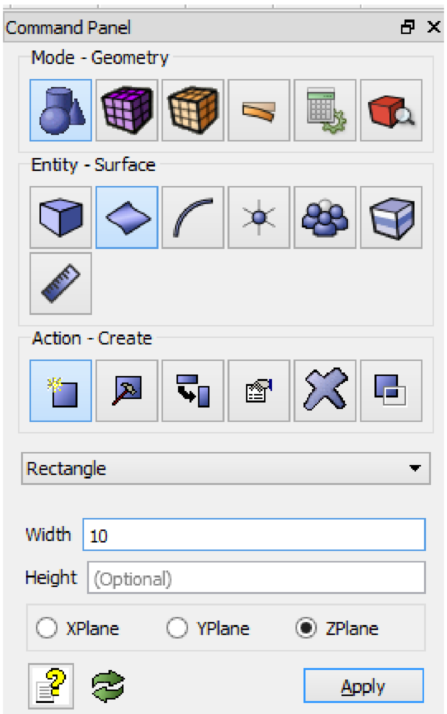
Eigenmodes from 1 to 8 are to be compared.

Geometry creation

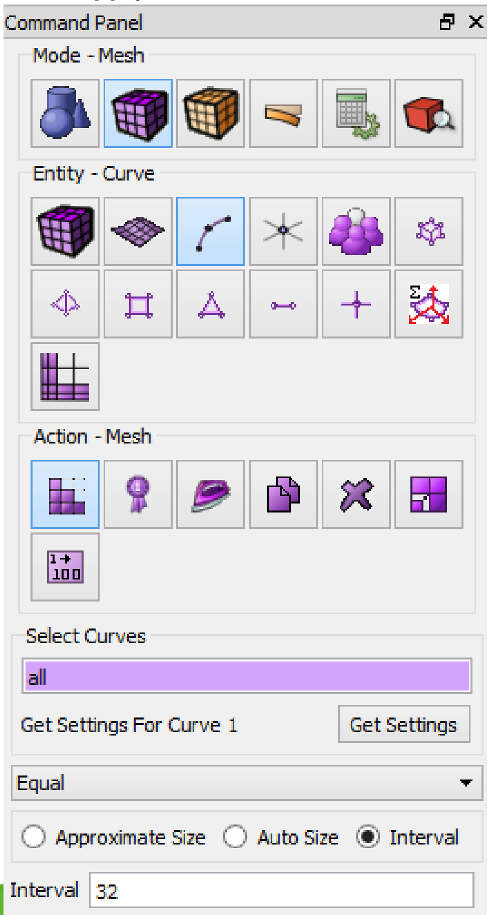
1. Create the plate.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Create**). Select **Rectangle** in the list of geometric elements. Set the brick dimensions:

- Width: 10;
- Location: ZPlane.



Click **Apply**.



Meshing

A mesh of 32*32 linear quadrilateral elements is to be generated (as shown at the picture with the problem setting.

1. Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: all;
- Select the way of meshing: Equal;
- Select splitting settings: Interval;
- Interval: 32 *(*see the figure*)*.

Click **Apply.**

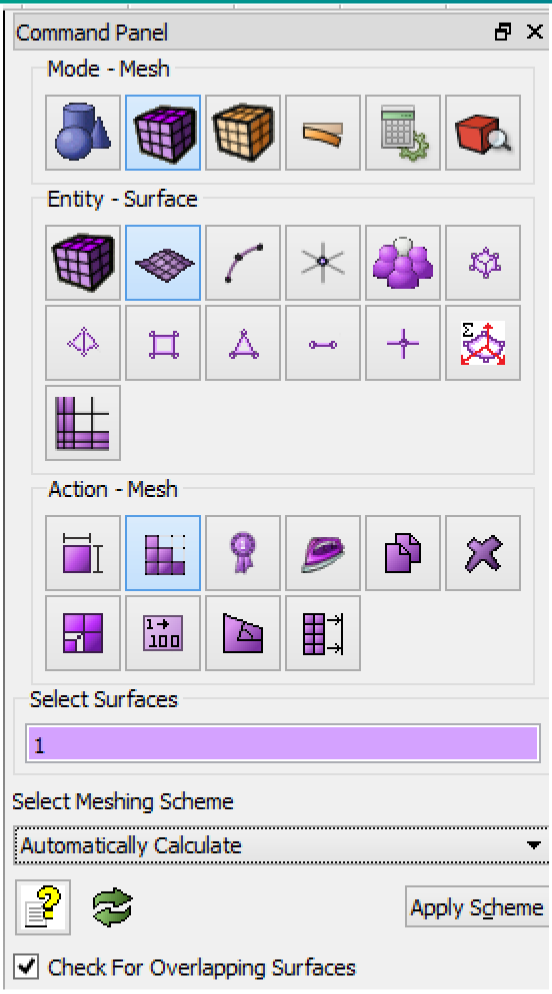
Click **Mesh**.

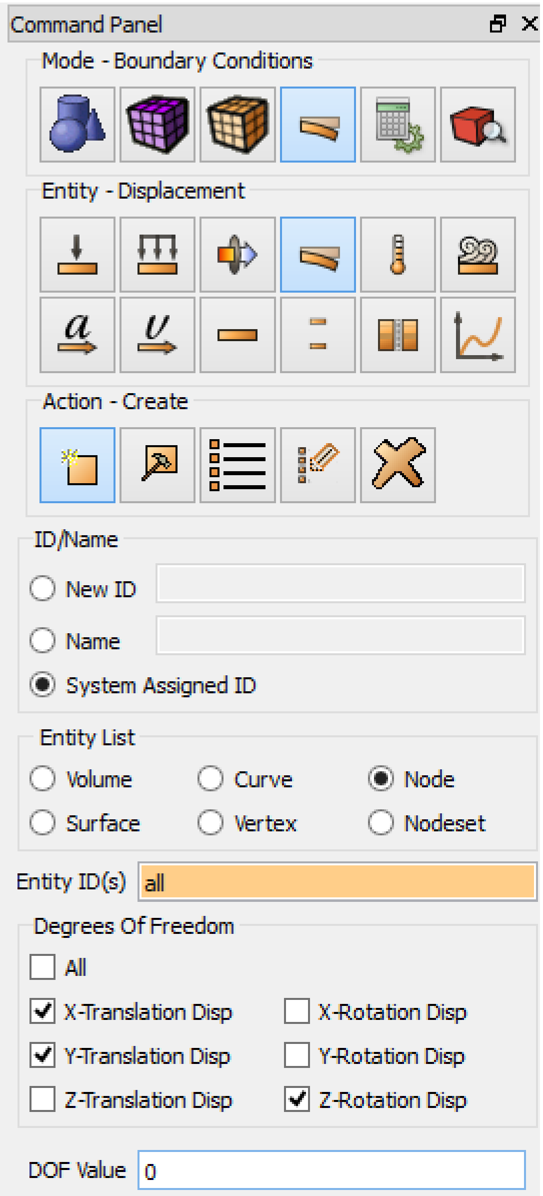
2. Select surface mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Meshing**).

- Select Entities to Mesh (specify their ID): 1 *(or by the command **all**)*
- Select Meshing Scheme: Automatically Calculate

Click **Apply Scheme**.

Click **Mesh**.





Setting boundary conditions

1. Fix the plate: X- and Y-Translations and Z-Rotations.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Node;
- Entity ID(s): all;
- Degrees of Freedom: X-Translation, Y-Translation and Z-Rotation;
- DOF Value: 0.

Click **Apply**.

2. Fix all the edges at the Z-direction.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create.** Set the following parameters:

- System Assigned ID;
- Entity List: Curves;

- Entity ID(s): all;
- Degrees of Freedom: Z-Translation;
- DOF Value: 0.

Click **Apply**.

3. Fix the edges AB and CD on X-Rotation.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Curves;
- Entity ID(s): 2 4 *(using space after each of them)*;
- Degrees of Freedom: X-Rotation;
- DOF Value: 0.

Click **Apply**.

4. Fix the edges BC and AD in Y-rotation.

Select Mode – **Boundary Conditions**, Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Curves;
- Entity ID(s): 1 3 *(using space after each of them)*;
- Degrees of Freedom: Y-Rotation;
- DOF Value: 0.

Click **Apply**.

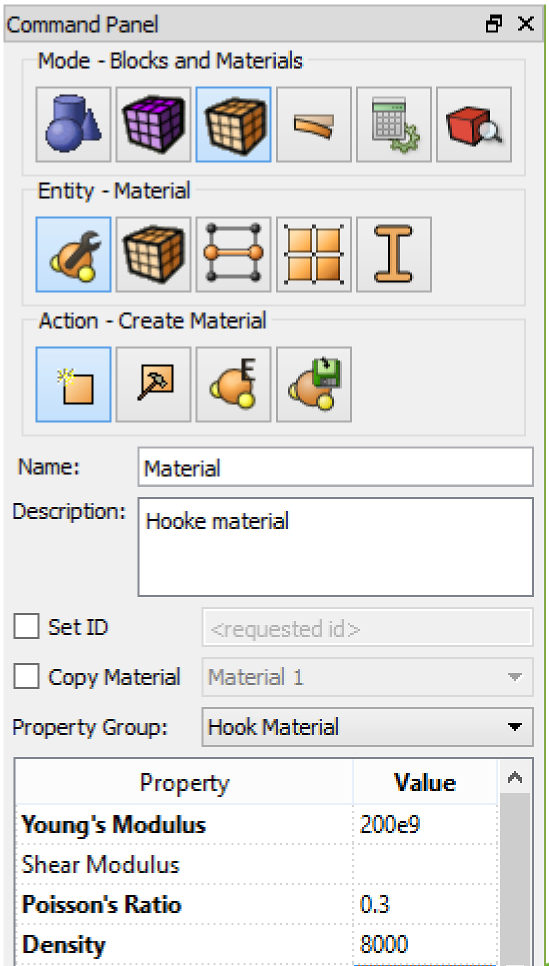
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Property group: Hook material;
- Young's Modulus: 200e9;
- Poisson's Ratio: 0.3;
- Density: 8000.

Click **Apply**.

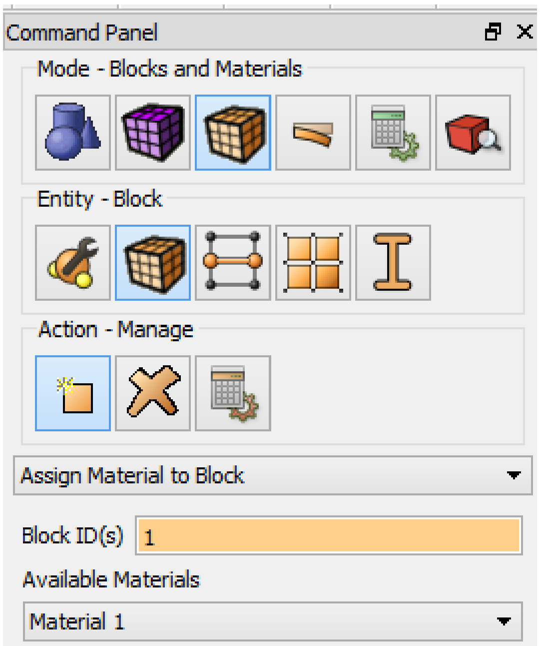


2. Create the block of one type of the material

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Surface;
- ID: 1 *(or by the command **all**)*.

Click **Apply.**



3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

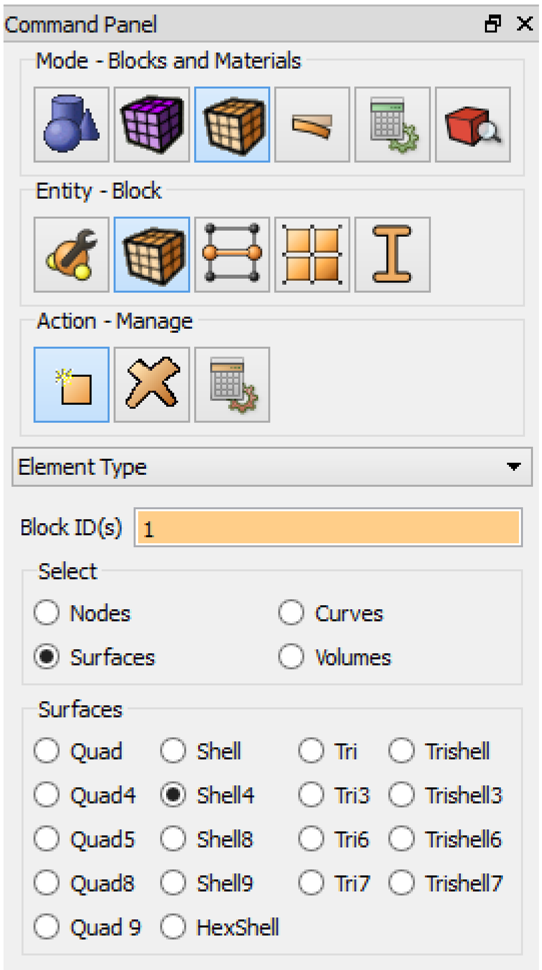
Click **Apply**.

4. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Surfaces;
- Surfaces: Shell4.

Click **Apply**.



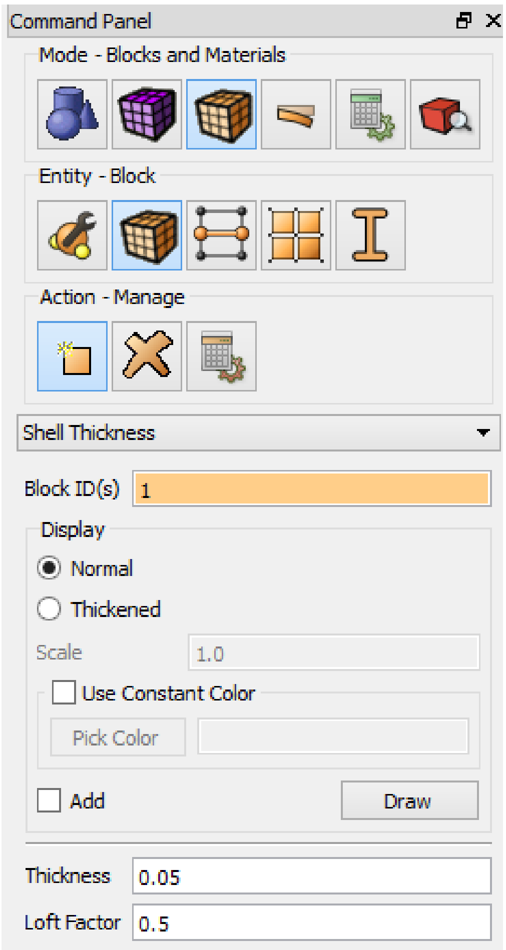
Setting shell thickness

1. Set the shell thickness.

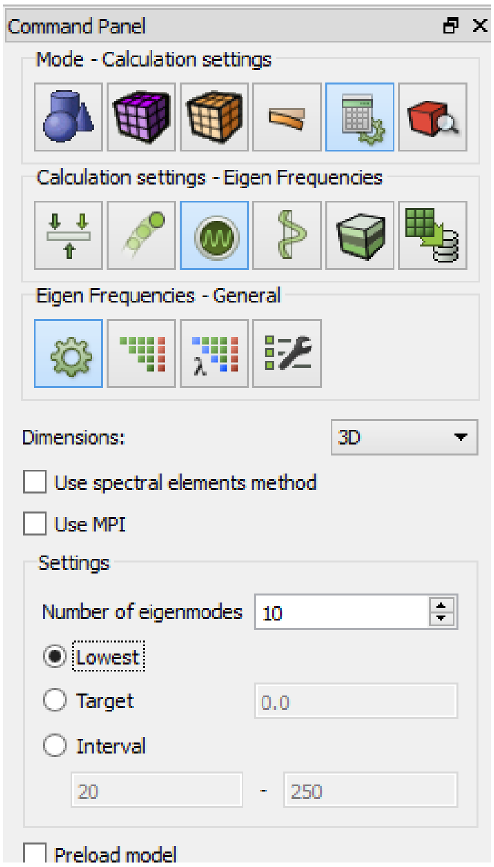
Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage)**. Select **Shell thickness** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Thickness: 0.05;
- Loft Factor: 0.5;

Click **Apply**.



Starting calculation



1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Eigen Frequencies**, Eigen Frequencies – **General**). Set the default settings.

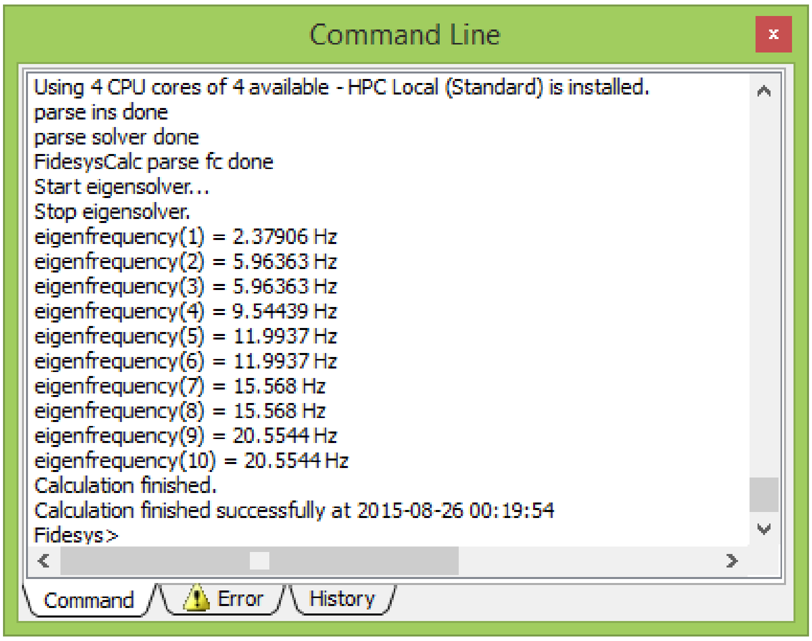
Click **Apply**.

Click **Start Calculation**.

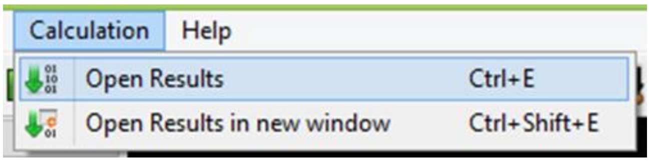
2. In a pop-up window select a folder to save the result and enter the file name.
3. If the calculation is finished successfully, you will see a message in the Console: "*Calculation finished successfully at <date> <time>*" as well as the required eigenvalues and frequencies.

Results analysis

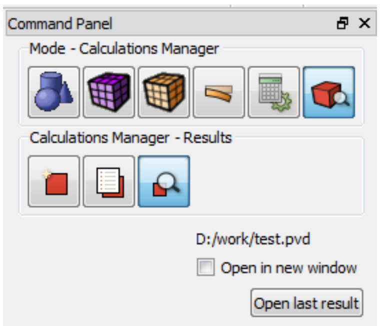
1. Compare the obtained results to those given in the picture.



2. Open the file with the results. You can do this in one of the three ways.



- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



3. You can see the way the body is deformed under the applied pressure.

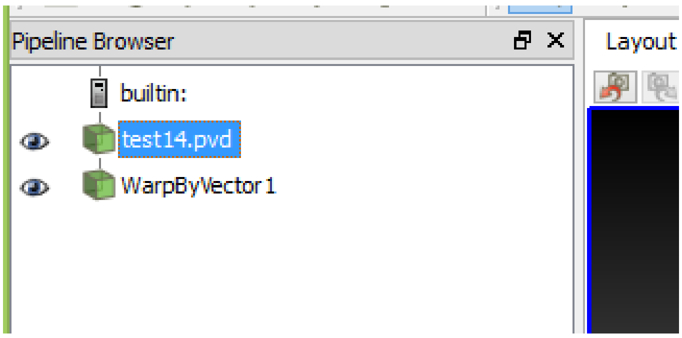
Select a filter **Warp By Vector** to do this. Set the following parameters in the tab **Properties**:

- Vectors: Eigenvalue_# *(# stands for the number of the eigenvalue)*;
- Scale Factor: 200.

As a result, the deformed body is displayed at the picture. To see the original model, click near it in the Model Tree. The picture below shows the deformed model at different eigenvalues.

4. Display the 3D-view of the model (shell with thickness).

To do this, click on the name of the source file in the Model Tree. After this click 3D-view button in the default string.



The file *_3D.pvd with a 3D-image of the shell must be opened and you will be able to apply various filters to it and to view its deformed view.

Using Console Interface

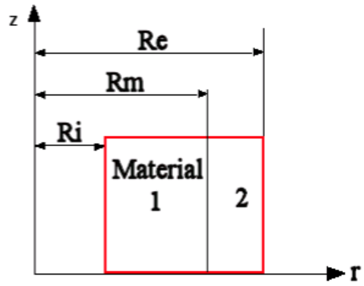
For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
create surface rectangle width 10 zplane
curve all interval 32
curve all scheme equal
surface 1 scheme Auto
surface 1 scheme Auto
mesh surface 1
create displacement on node all dof 1 dof 2 dof 6 fix 0
```

```
create displacement on curve all dof 3 fix 0
create displacement on curve 2 4 dof 4 fix 0
create displacement on curve 1 3 dof 5 fix 0
undo group begin
create material "material 1" property_group "CUBIT-FEA" description "Hook material"
modify material "material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3 "DENSITY"
8000
undo group end
set duplicate block elements off
block 1 surface 1
block 1 material 'material 1'
block 1 element type shell4
undo group begin
block 1 attribute count 2
block 1 attribute index 1 value 0.05
block 1 attribute index 2 value 0.5
undo group end
analysis type eigenfrequencies elasticity dim3
eigenvalue find 10 smallest preload off
spectralelement off
usempi off
analysis type eigenfrequencies elasticity dim3
eigenvalue find 10 smallest preload off
spectralelement off
usempi off
calculation start path 'D:/FidesysBundle/calc/example.pvd'
```

It is also possible to run the file *Example_9_Eigenvalue_Shell.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

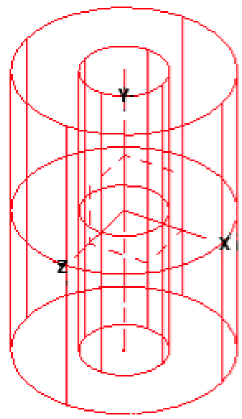
Setting heat transfer (3D, working with two blocks)



The 3D problem of a hollow two-material cylinder which inner and outer surfaces undergo the convection is being solved.

The pictures represent a geometric model of the problem:

The inner radius of the cylinder Ri = 0.30 m, the middle radius of the cylinder (at the place of material changing) Rm = 0.35 m, the external radius of the cylinder Re = 0.37 m.



Convective heat exchange with internal temperature Ti = 70 ° C and coefficient hi = 150 W/ m^2/°C occurs on the inner surface of the cylinder. Convective heat exchange with exterior temperature T_e = -15 °C and coefficient h_e = 200 W/ m^2/°C occurs on the outer surface of the cylinder.

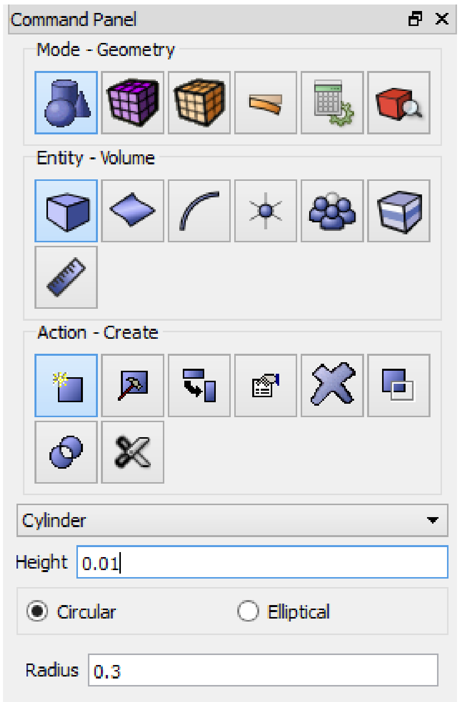
Materials are isotropic. The material heat transfer 1 is V_1 = 40 W/(m·°C). The material heat transfer 2 is V_2 = 20 W/(m·°C).

Test pass criterion is the following:

at the point (0.3, 0, 0) heat flux ϕ = 6687 W/ m^2 is within 1%.

Geometry creation

1. Create the first cylinder.



Select volume geometry generation section on Command Panel (Mode – **Geometry**, Object – **Volume**, Action – **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 0.01;
- Circular;
- Radius: 0.3.

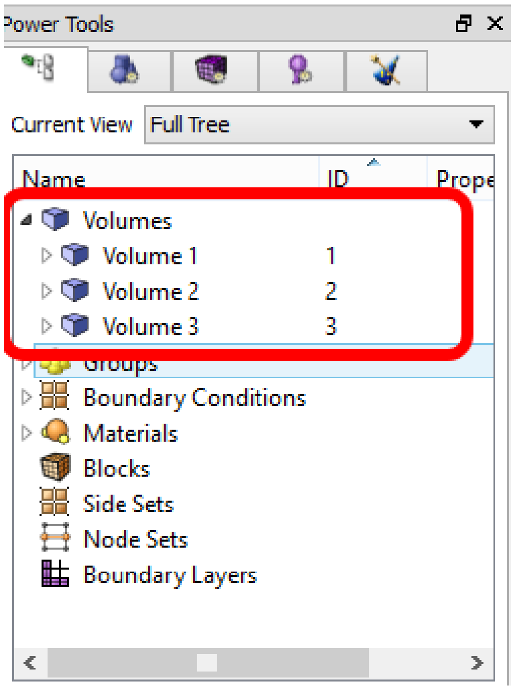
Click **Apply.**

2. Create the second cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 0.01;
- Circular;
- Radius: 0.35.

Click **Apply.**



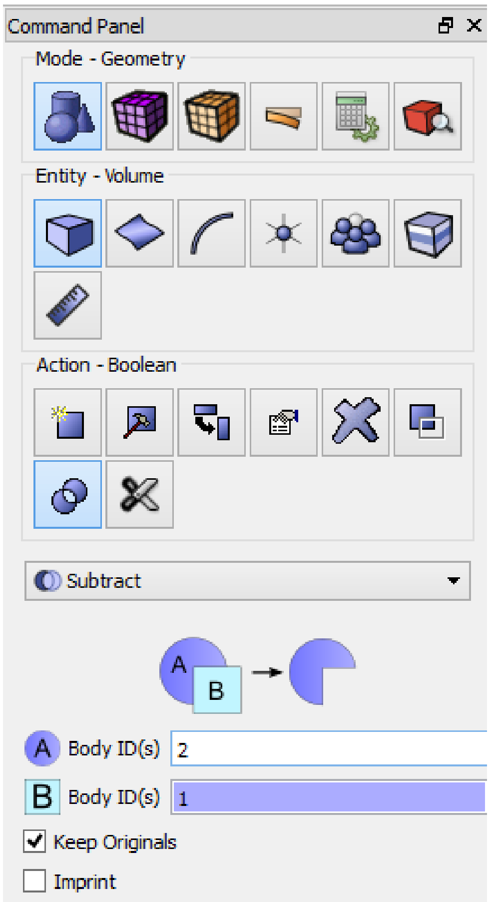
3. Create the third cylinder.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 0.01;
- Circular;
- Radius: 0.37.

Click **Apply**.

As a result, three generated entities are displayed in the Model Tree (Volume 1, Volume 2 and Volume 3).



4. Subtract the first cylinder from the second one.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Body ID: 2 *(volumes from which other volumes will be subtracted)*;
- Substract bodies (ID): 1 *(the volumes to be subtracted)*;
- Keep Originals.

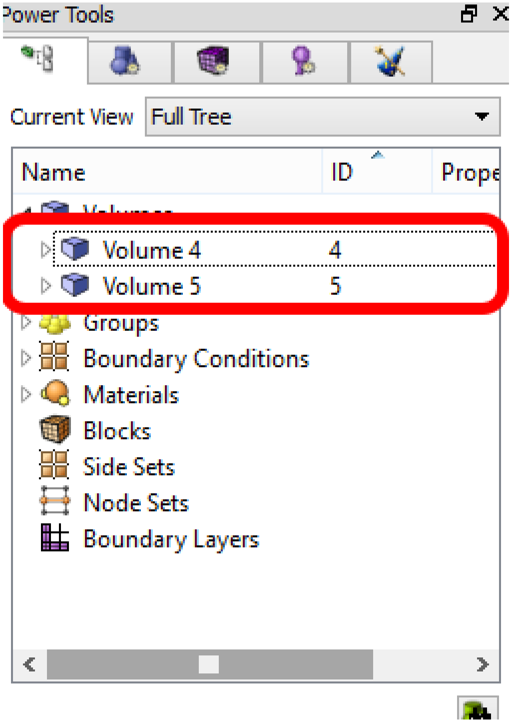
Click **Apply**.

5. Subtract the second cylinder from the third one.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Body ID: 3 *(volumes from which other volumes will be subtracted)*;
- Substract bodies (ID): 2 *(the volumes to be subtracted)*;
- Keep Originals.

Click **Apply.**



As a result, five generated entities are displayed in the Model Tree: Volume 1, Volume 2, Volume 3, Volume 4 and Volume 5. Delete the thirst three bodies by right-clicking and selecting Delete.

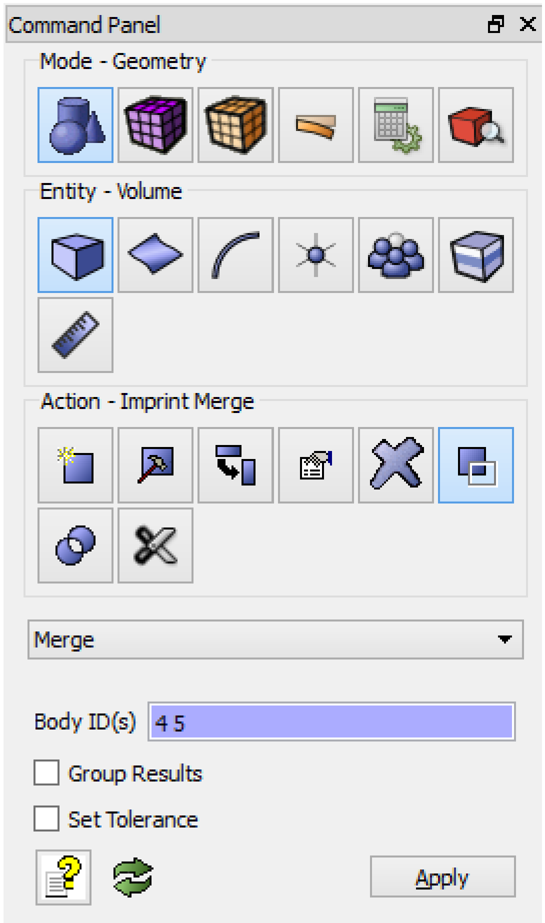
Two entities: Volume 4 and Volume 5 are left in the Model Tree.

6. Unite obtained objects.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Impirint And Merge**). Select **Merge** in the list of operations. Set the following parameters:

- Body ID: 4 5 *(the volumes to be united)*.

Click **Apply**.

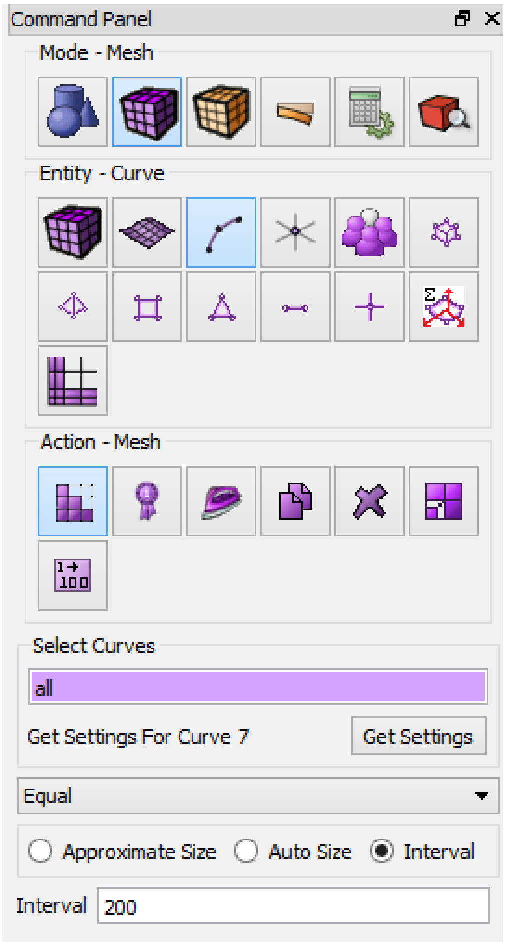


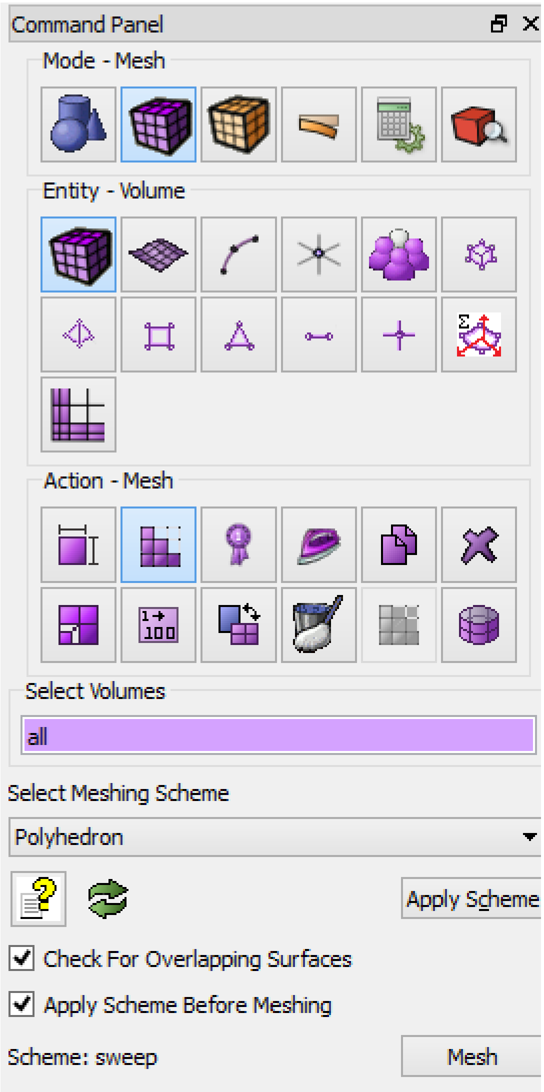
Meshing

1. Select meshing on curves section on Command Panel (Mode — **Mesh,** Entity — **Curve**, Action — **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: all *(mesh will be creat on all the curves)*;
- Select the way of meshing: Equal;
- Select the meshing parameters: Interval;
- Interval: 200.

Click **Apply**.





2. Select volume mesh generation section on Command Panel (Mode — **Mesh**, Entity — **Volume**, Action — **Mesh**).

- Select volumes: all *(mesh will be creat on all the volumes)*;
- Select meshing scheme: Polyhedron.

Click **Apply Scheme**.

Click **Mesh**.

Setting material and element type

1. Create Material 1.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Enter the name for the material. Set the following parameters:

- Coeff. of Thermal Expansion: 40.

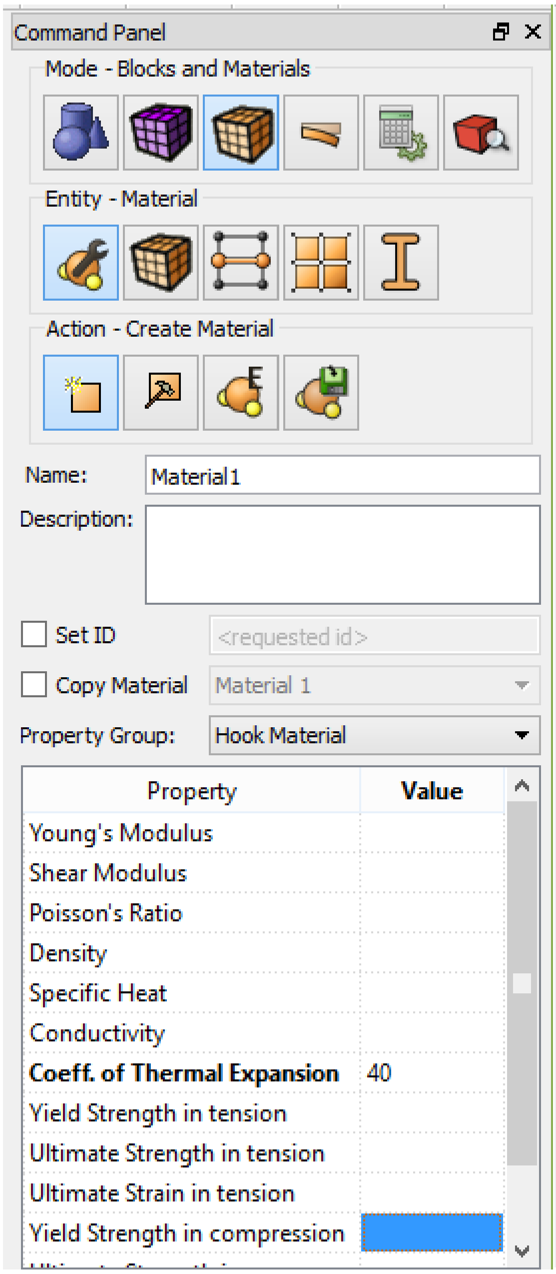
Click **Apply**.

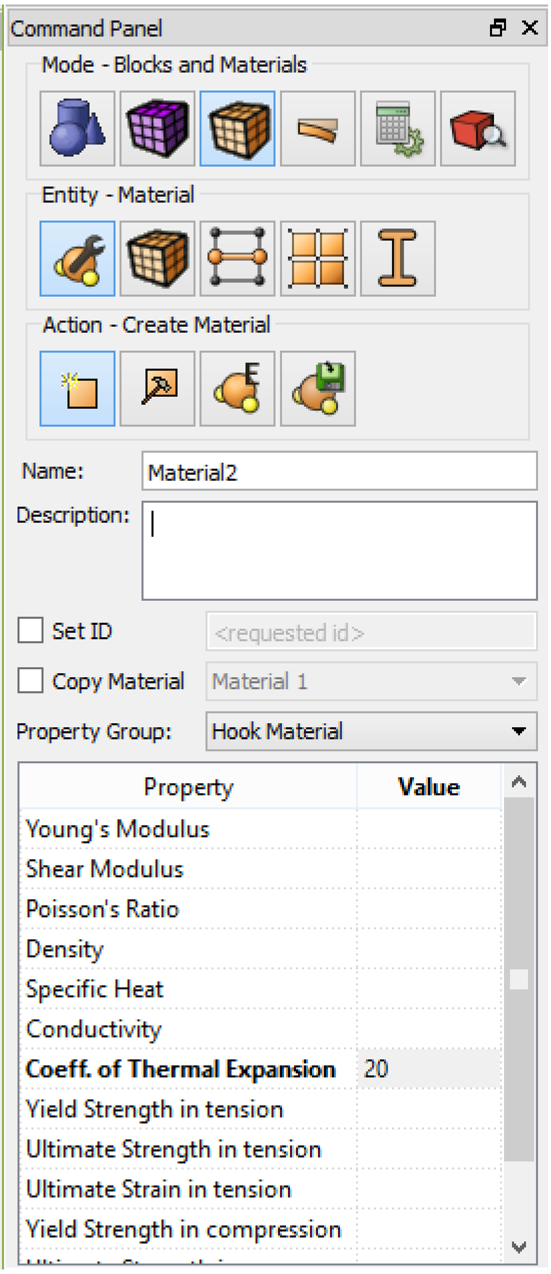
2. Create Material 2.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Enter the name for the material. Set the following parameters:

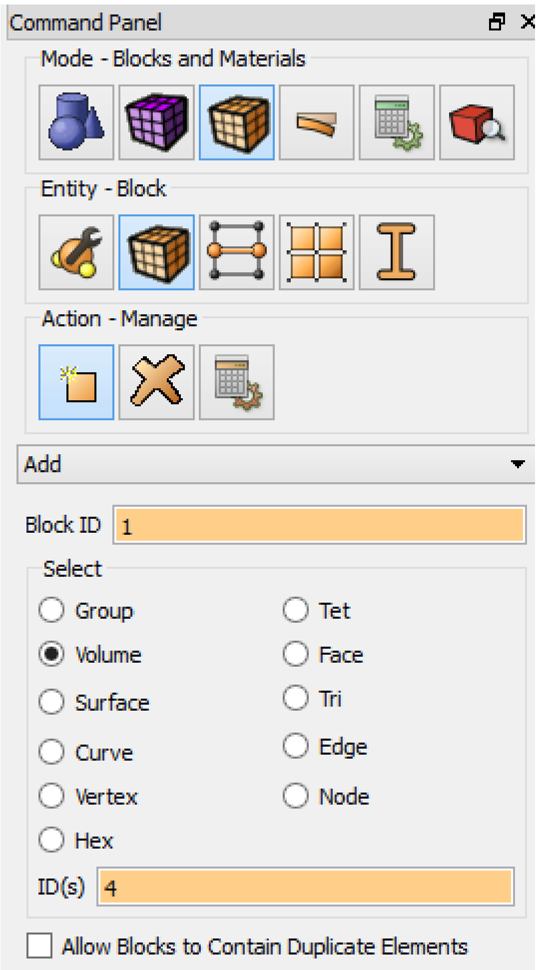
- Coeff. of Thermal Expansion: 20.

Click **Apply**.





3. Create Block 1.

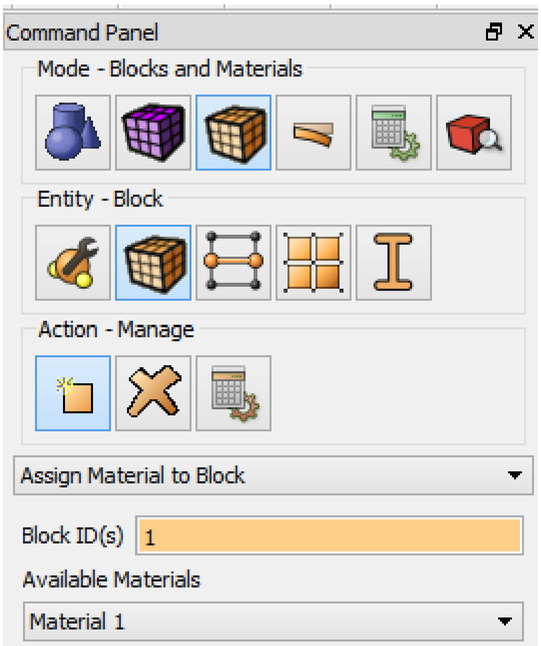


Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: 4.

Click **Apply**.

4. Assign the material to block 1.



Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

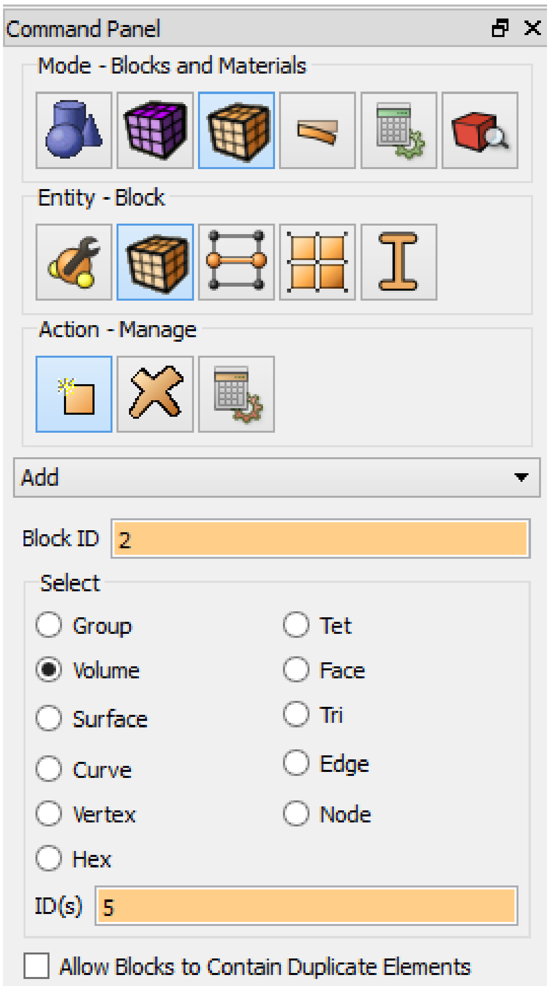
Click **Apply**.

5. Create Block 2.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 2;
- Entity type to be united into the block: Volume;
- ID: 5.

Click **Apply**.

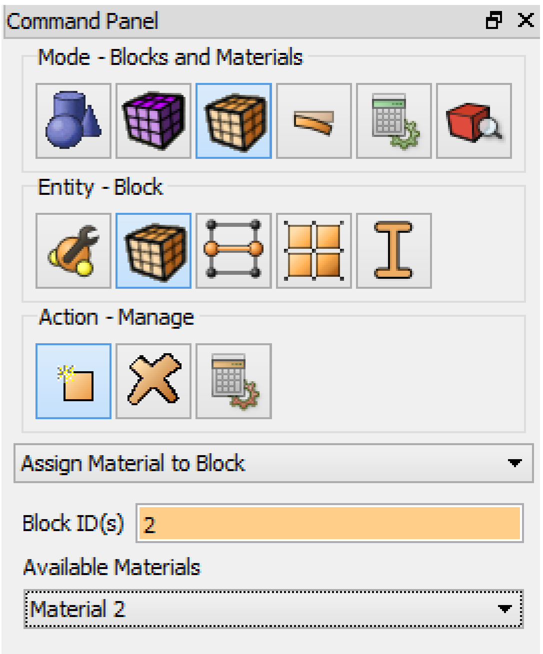


6. Assign the material to block №2.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 2;
- Select the previously created material in the list: Material 2.

Click **Apply**.

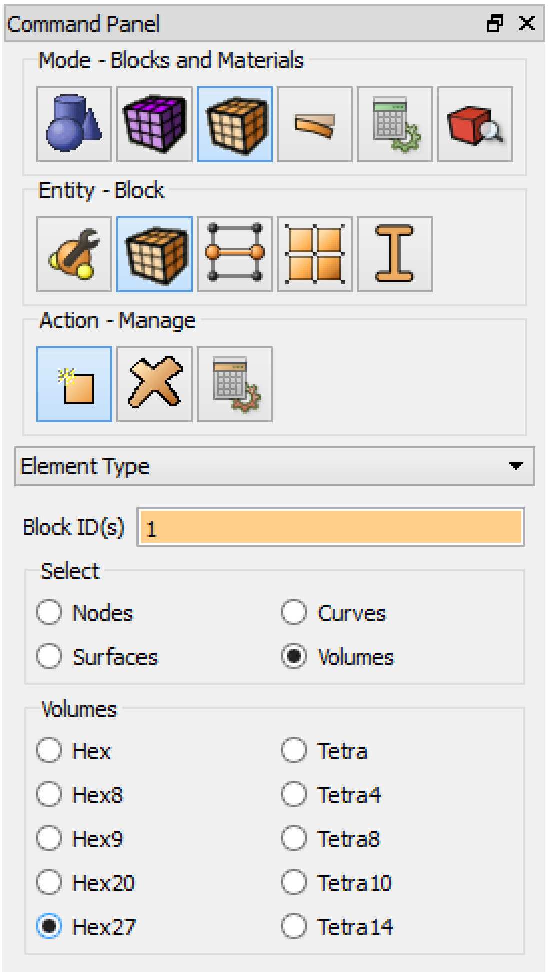


7. Assign the element type to the both of blocks.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: all;
- Select: Volumes;
- Volumes: Hex27.

Click **Apply**.



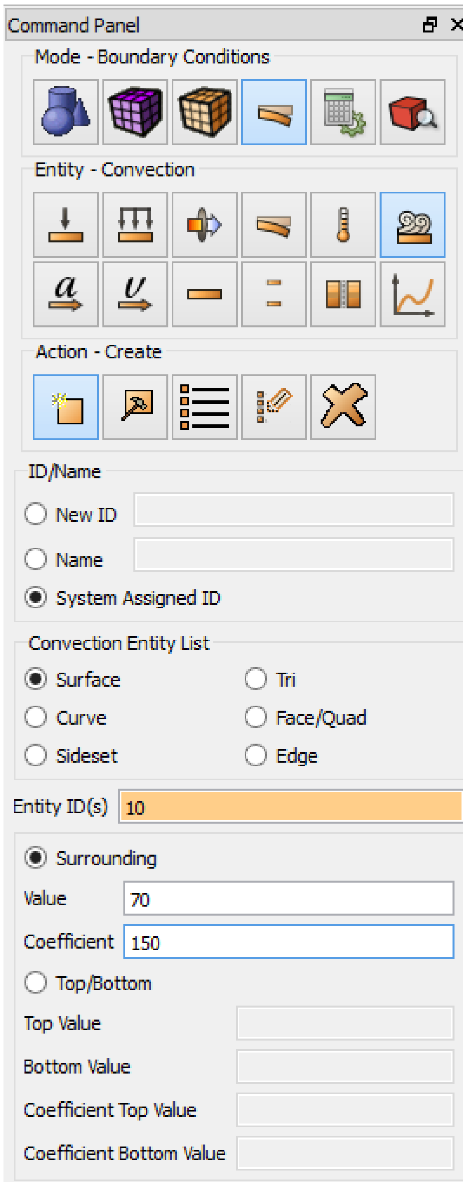
Setting boundary conditions

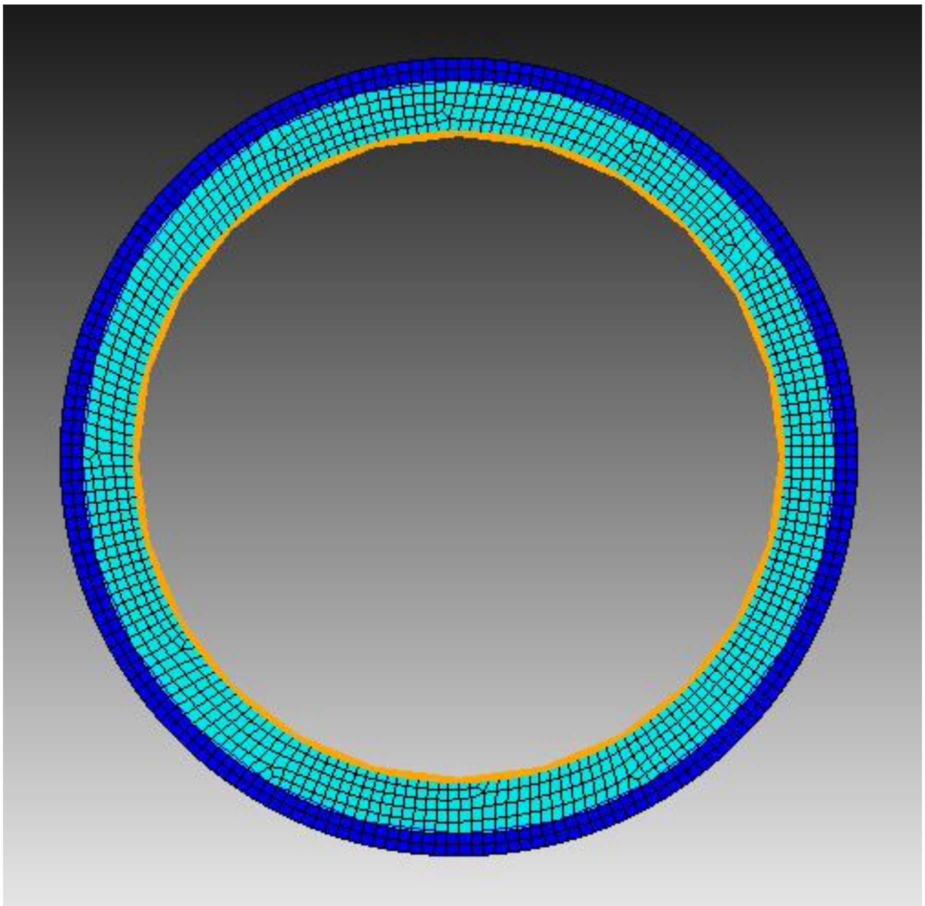
1. Set the process of convective heat exchange on the inner surface of the cylinder.

Select Mode — **Boundary Conditions,** Entity — **Convection**, Action — **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 10;
- Select the way of parameters setting: Surrounding;
- Value: 70;
- Coefficient: 150.

Click **Apply**.



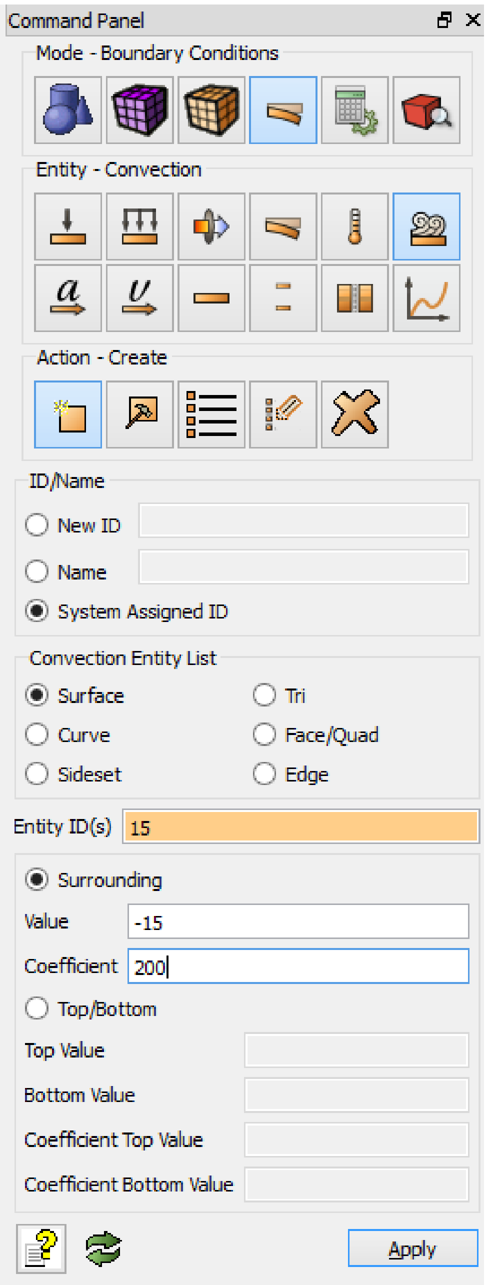


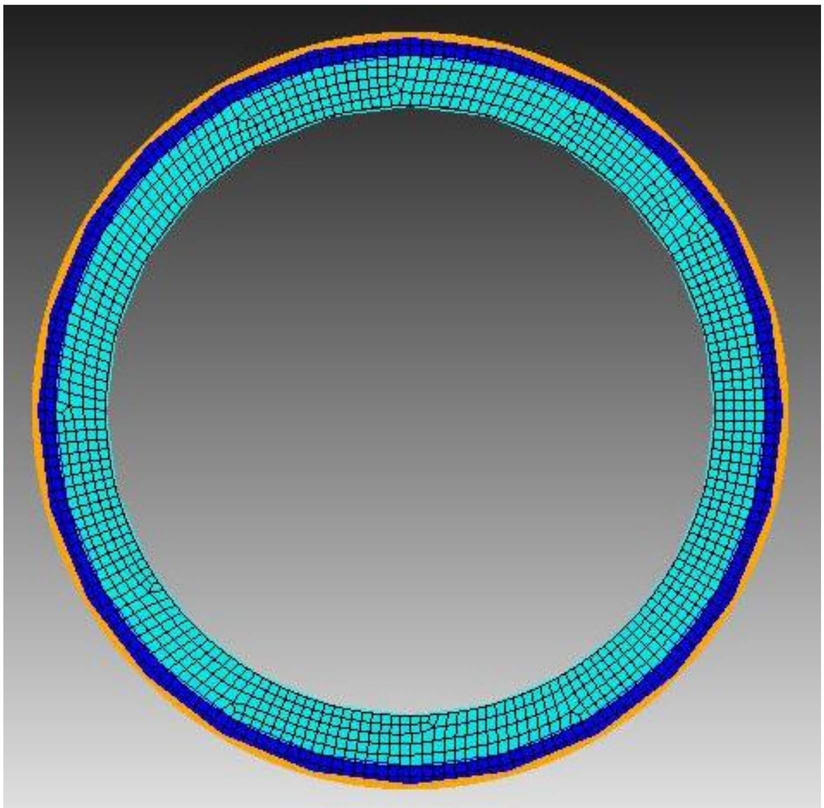
2. Set the process of convective heat exchange on the outer surface of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Convection**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 15;
- Select the way of parameters setting: Surrounding;
- Value: -15;
- Coefficient: 200.

Click **Apply**.



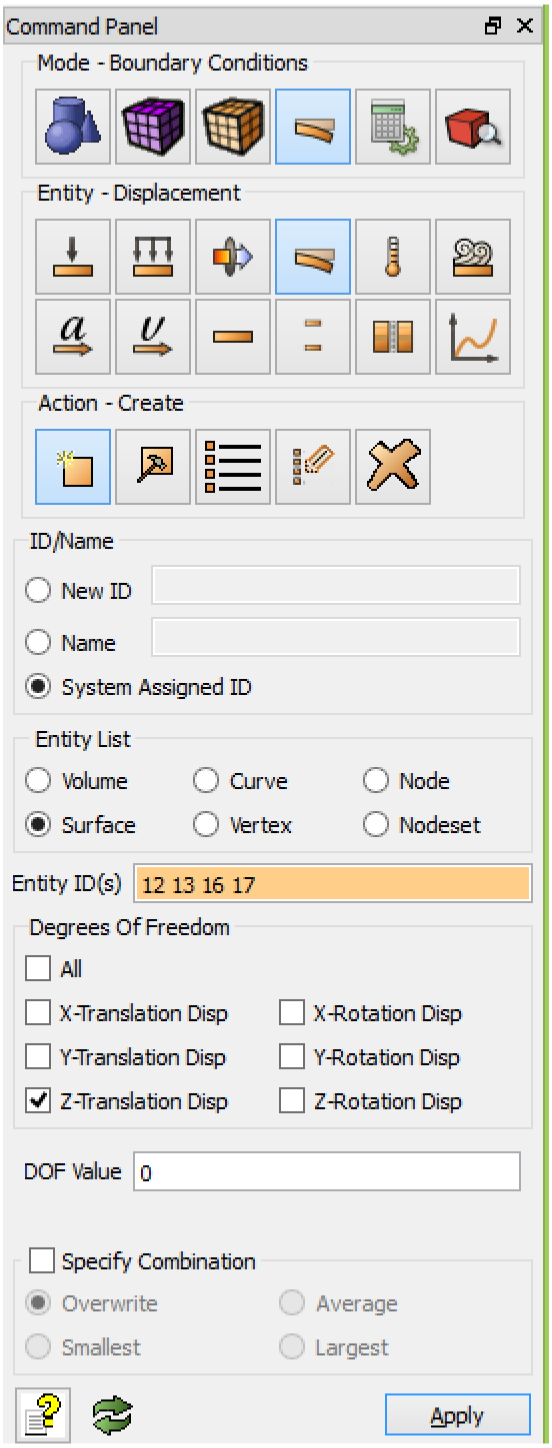


3. Fix the base of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entities ID: 12 13 16 17 *(using space after each of them)*;
- Degrees of Freedom: Z-Translation;
- DOF Value: 0.

Click **Apply**.

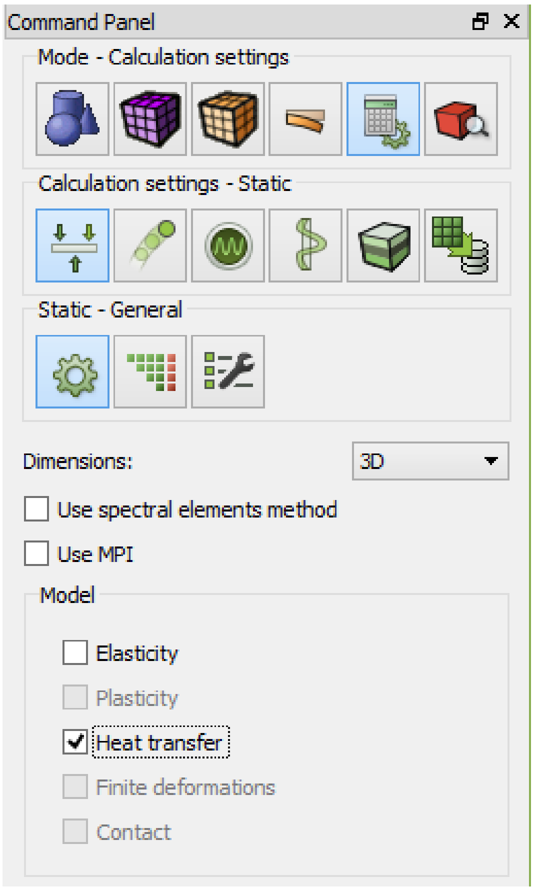


Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **General**). Select Dimension – **3D**. Untick next to the item **Elasticity**. Tick next to the item **Heat transfer**.

Click **Apply**.



2. Set the solver settings.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply**.

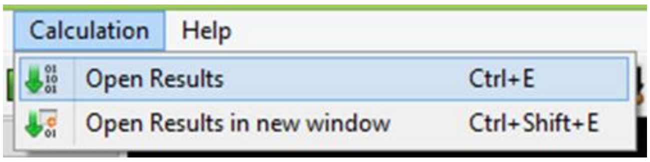
Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name.

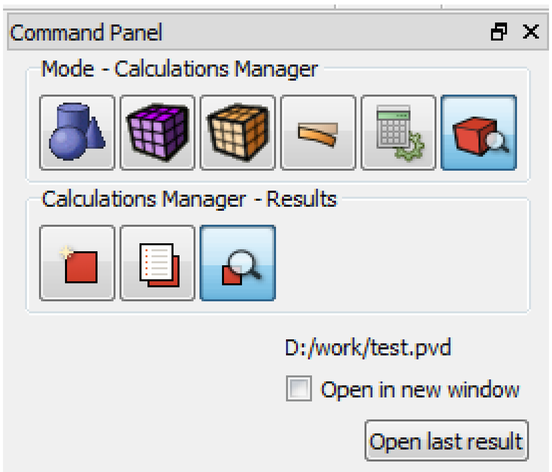
4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>".*

Results analysis

1. Open the file with the results. You can do this in one of the three ways.



- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode — **Calculations Manager**, Calculations Manager — **Results**). Click **Open Results**.



2. Display the component of the heat flux.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: HeatFlux.

To display the color legend scale, click the button **Switch the color legend visibility** on Command Panel.

3. Select a point where you need to view the heat flux.

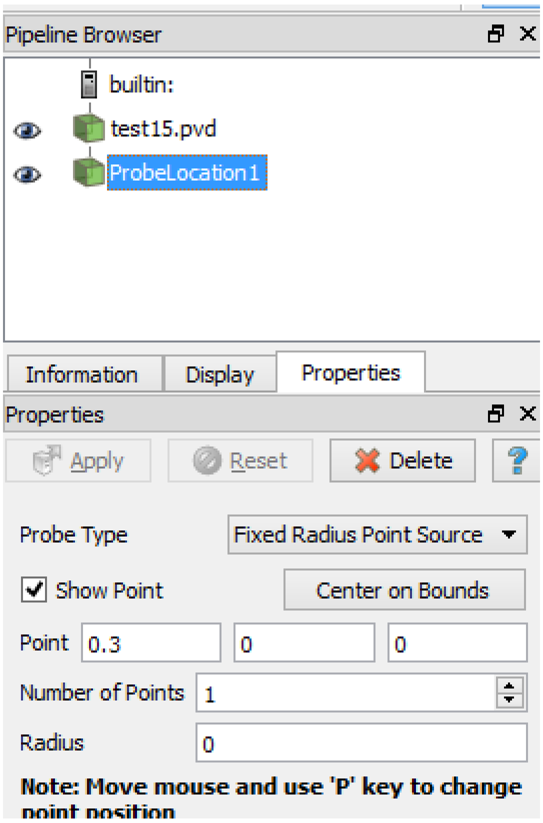
In the Main Menu, select the filter **Probe Location**. In the tab **Properties** set the coordinates of the point A where you need to view the stress:

- Show Point;
- Point (coordinates): 0.3 0 0;
- Number of Points: 1;
- Radius: 0.

Click **Apply**.

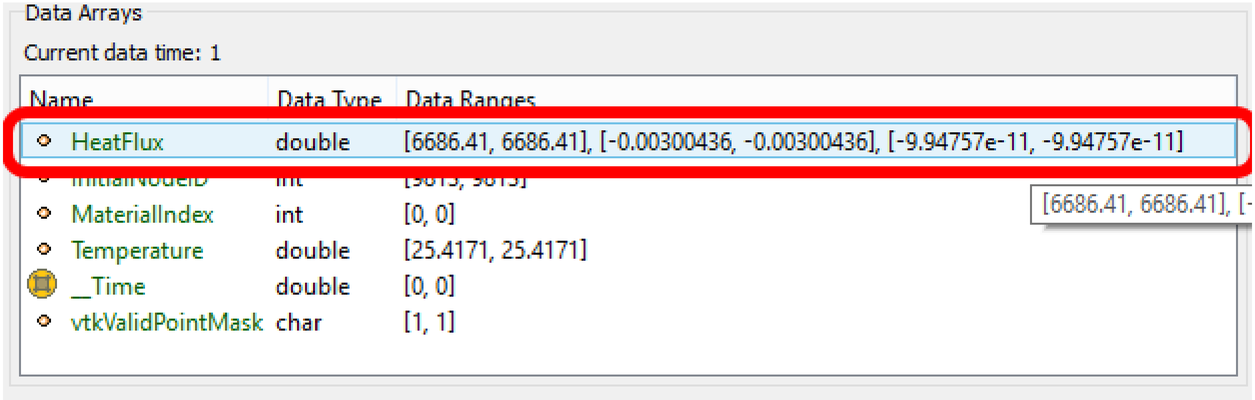
To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

As a result, point A is displayed at the picture.



4. View a numerical value of the heat flux φ at the selected point A.

See the heat flux values in the line **HeatFlux** in the tab **Information** in the field **Data Arrays**.



The heat flux value is calculated using the following formula:

$$\sqrt{{\varphi_x}^2 + {\varphi_y}^2 + {\varphi_z}^2} = \sqrt{6686.41^2 + (-0.00302395)^2 + (8.02105\mathrm{e}-05)^2} = 6686.41$$

The difference between the obtained value 6686.41 and the required one 6 687 is 0.01%.

5. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

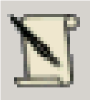
Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
create Cylinder height 0.01 radius 0.3
create Cylinder height 0.01 radius 0.35
create Cylinder height 0.01 radius 0.37
subtract body 1 from body 2 keep
subtract body 2 from body 3 keep
delete body 1 2 3
merge volume 4 5
curve all interval 200
curve all scheme equal
volume all scheme Polyhedron
mesh volume all
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "CONDUCTIVITY" 40
create material "Material 2" property_group "CUBIT-FEA"
modify material "Material 2" scalar_properties "CONDUCTIVITY" 20
block 1 volume 4
block 1 material 'Material 1'
block 2 volume 5
```

```
block 2 material 'Material 2'
block all element type HEX27
create convection on surface 10 surrounding 70 coefficient 150
create convection on surface 15 surrounding -15 coefficient 200
create displacement on surface 12 13 16 17 dof 3 fix 0
analysis type static heattrans dim3
calculation start path " D:/FidesysBundle/calc/example.pvd"
```

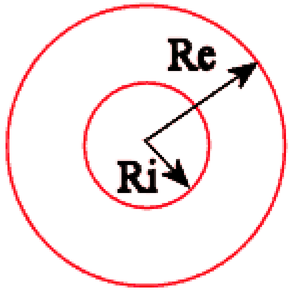


It is also possible to run the file *Example_10_Static_3D_Conduction.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Setting heat transfer (2D)

2D problem of a hollow cylinder under the convection is being solved.

The picture below represents a geometric model of the problem:



The inner radius of the cylinder is R_i = 0,30 m, the external radius is R_e = 0,391 m.

Convective heat exchange with internal temperature T_i = 500 °C and coefficient h_i = 150 W/ m^2/°C occurs on the inner surface of the cylinder. Convective heat exchange with exterior temperature T_e = 20 °C and coefficient h_e = 142 W/ m^2/°C occurs on the outer surface of the cylinder.

The material is isotropic. The material parameters are E=210 GPa, ν =0.3, V=40 W/(m · °C).

Test pass criterion: at the point (0,3;0;0) temperature T = 272.3 °C, heat flux φ = 3.416E4 W/m^2 are within 1 %.

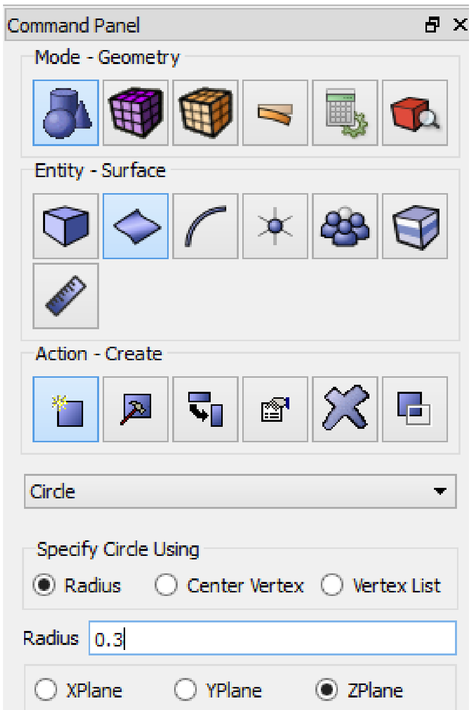
Geometry creation

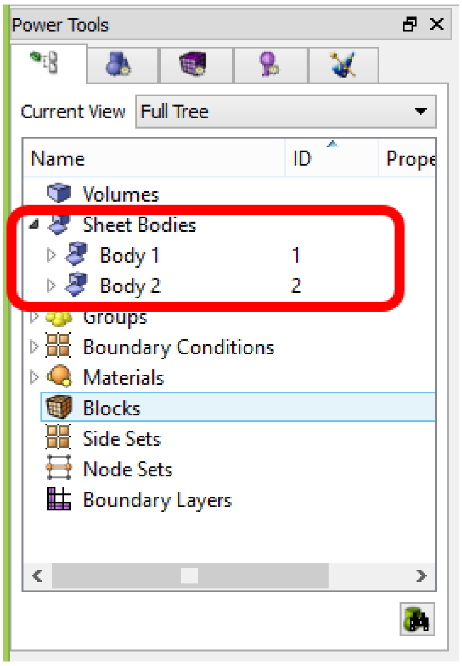
1. Create the first circle.

Select 2D geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Surface**, Action — **Create**). Select **Circle** in the list of geometric elements. Specify the circle dimensions:

- Radius: 0.3.

Click **Apply**.





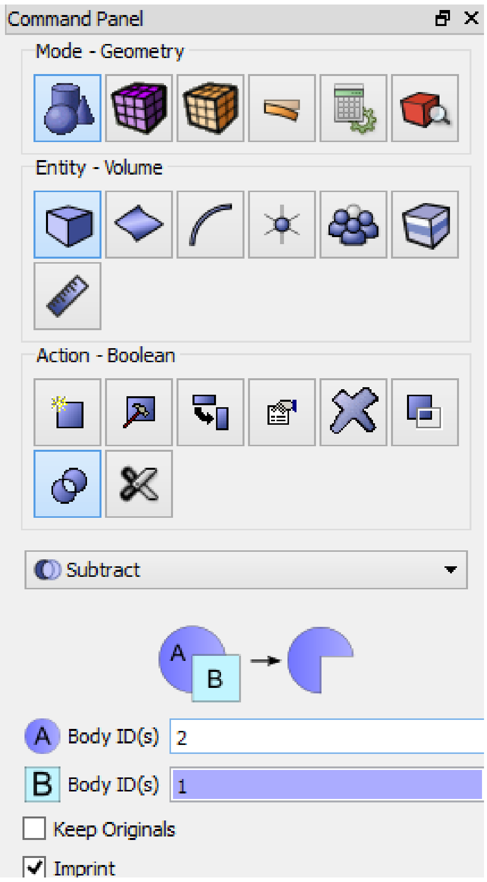
2. Create the second circle.

Select 2D geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Surface**, Action — **Create**). Select **Circle** in the list of geometric elements. Specify the circle dimensions:

- Radius: 0.391.

Click **Apply.**

As a result, two generated entities are displayed in the Model Tree (Body 1 and Body 2).



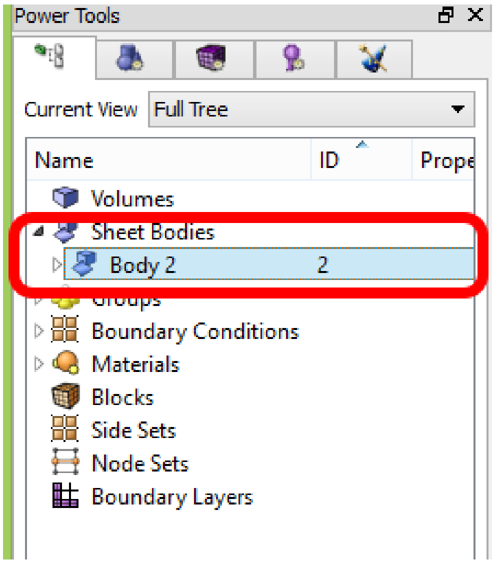
3. Subtract the first circle from the second one.

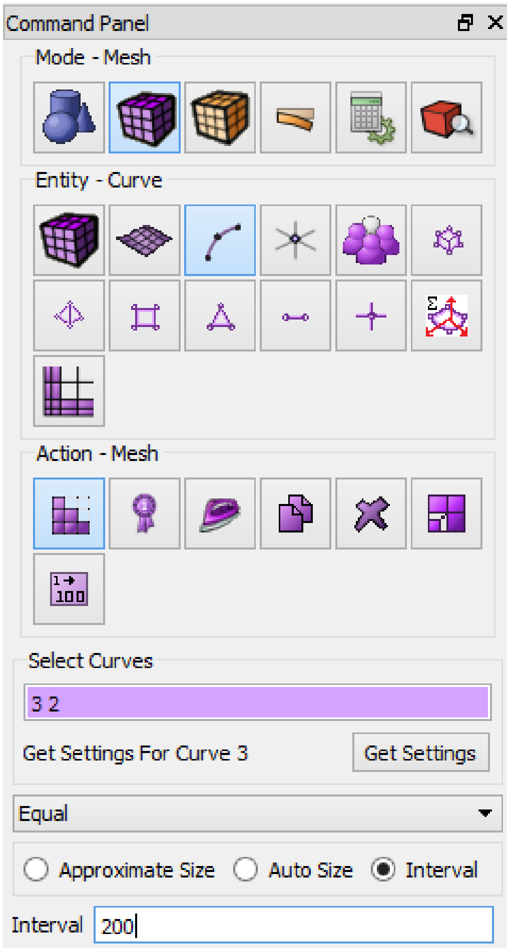
Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Body ID: 2 *(volumes from which other volumes will be subtracted)*;
- Substract bodies (ID): 1 *(the volumes to be subtracted)*;
- Imprint.

Click **Apply**.

As a result, only one entity is displayed in the Model Tree (Body 2).



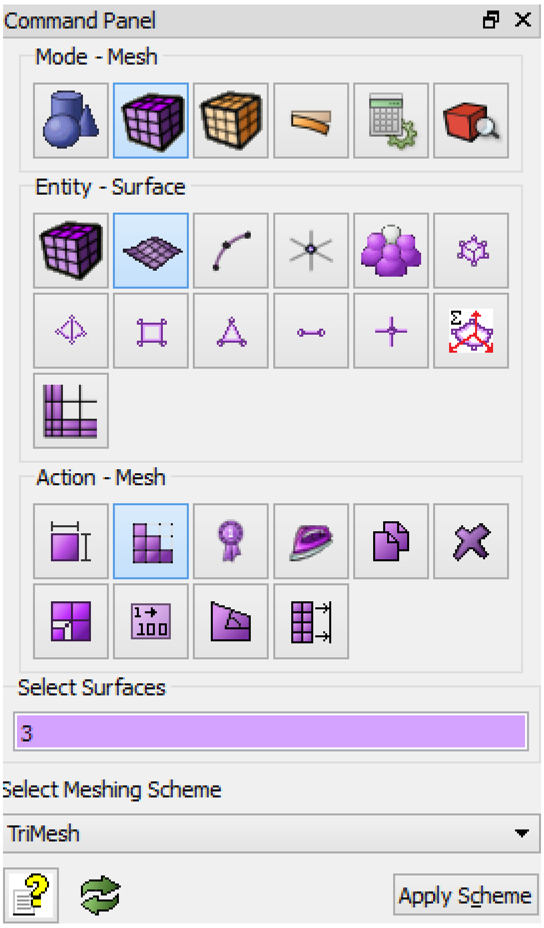


Meshing

1. Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**). Specify the parameters of mesh refinement:

- Select Curves: 3 2 *(using space after each of them)*;
- Select the way of meshing: Equal;
- Select the meshing parameters: Interval;
- Interval: 200.

Click **Apply**.



2. Select surface mesh generation section on Command Panel (Mode — **Mesh,** Entity — **Surface**, Action — **Meshing**).

- Select surfaces (specify their ID): 3 (*or by the command **all***);
- Select meshing scheme: TriMesh.

Click **Apply Scheme**.

Click **Mesh**.

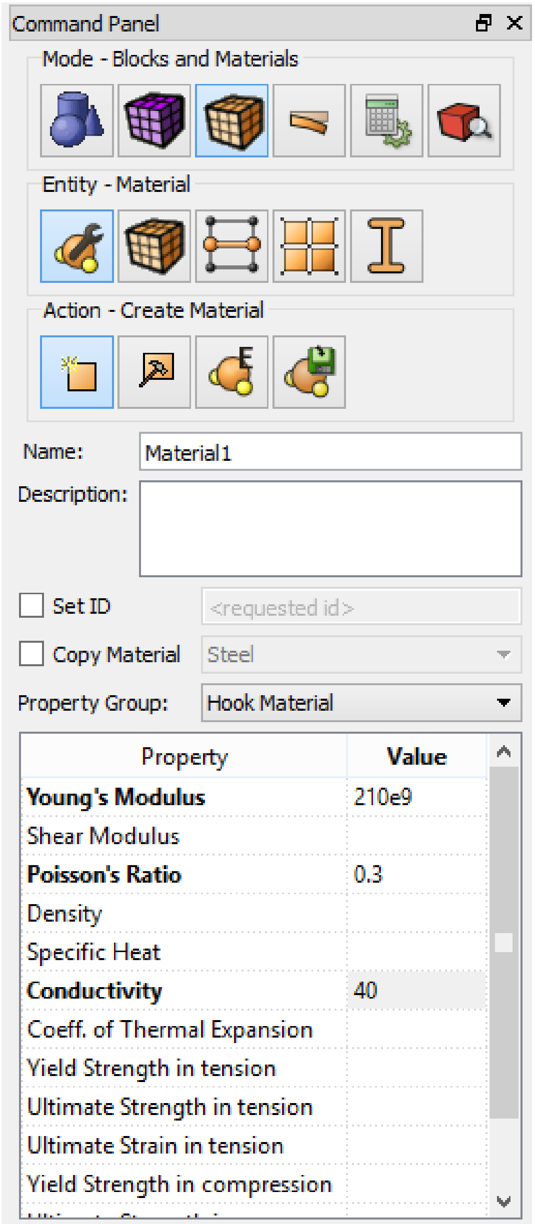
Setting material and element type

1. Create the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Material**, Action — **Create Material**). Enter the name for the material. Set the following parameters:

- Conductivity: 40.

Click **Apply**.

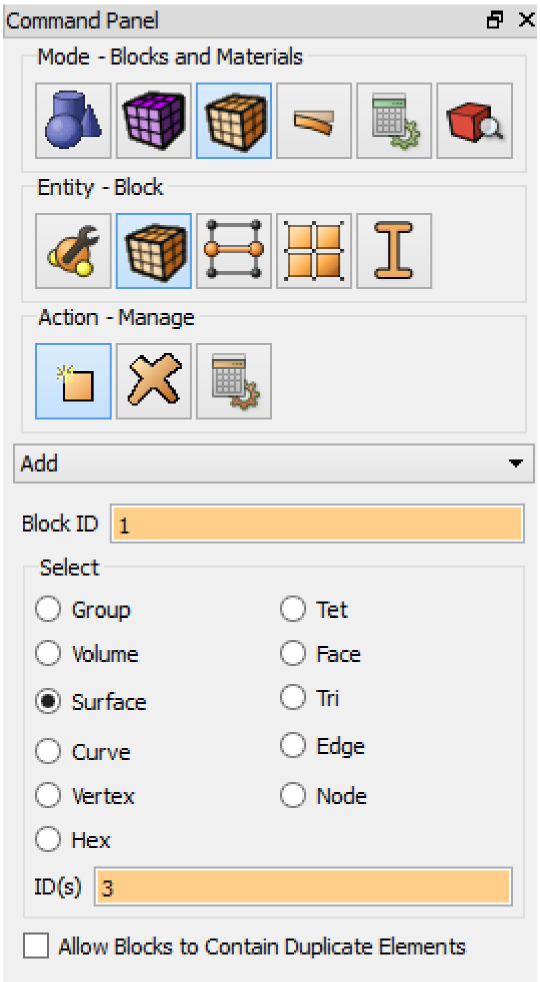


2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Surface;
- ID: 3 *(or by the command **all**).*

Click **Apply**.

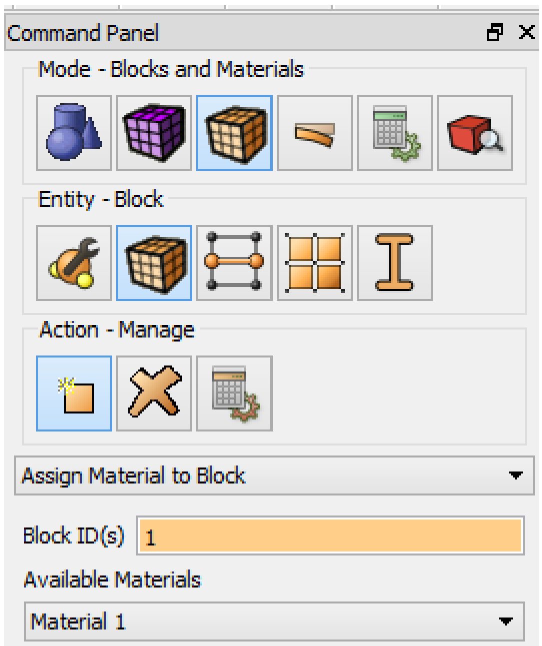


3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

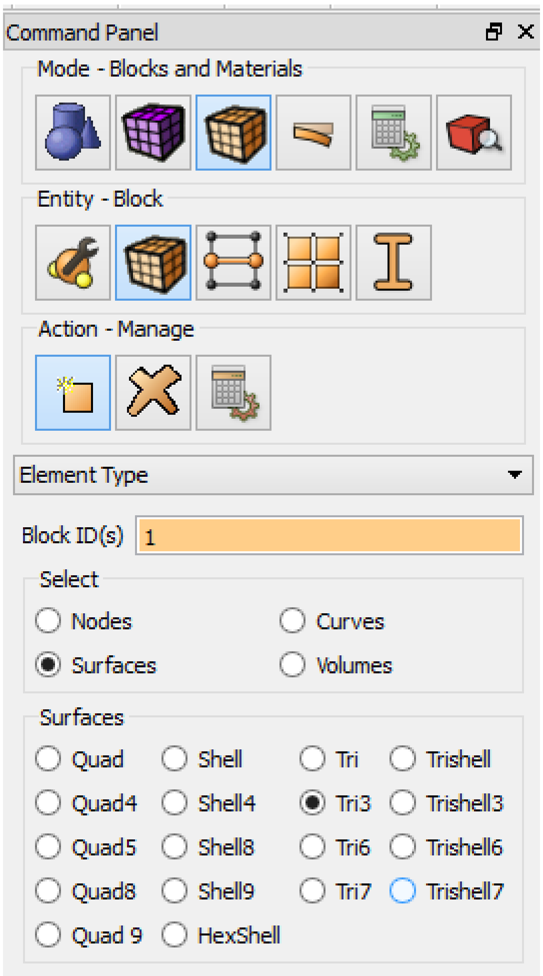
- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.



4. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:



- Block(s) ID: 1;
- Select: Surfaces;
- Surfaces: Tri3.

Click **Apply**.

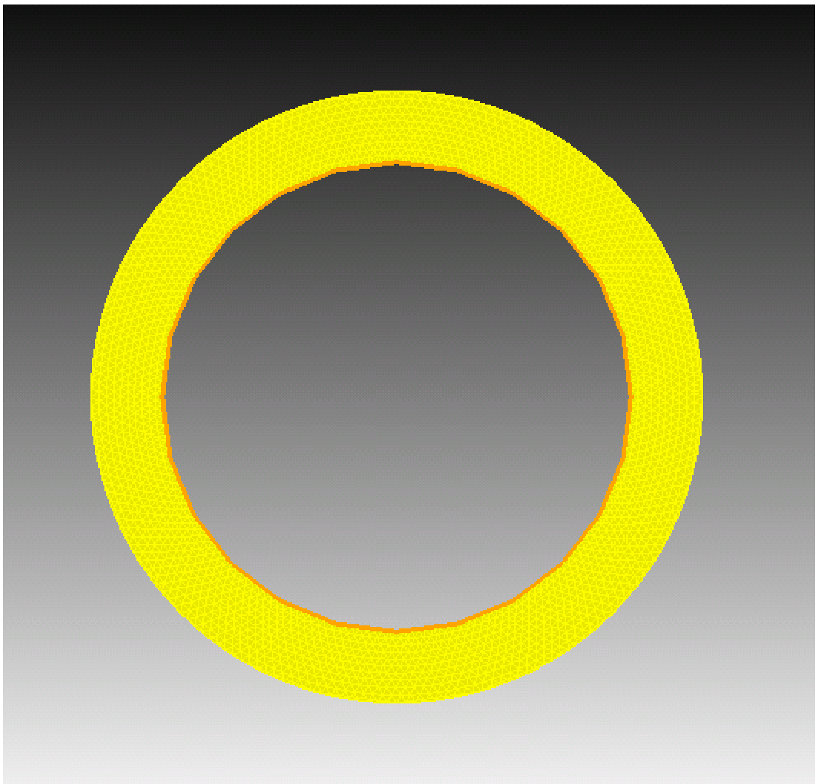
Setting boundary conditions

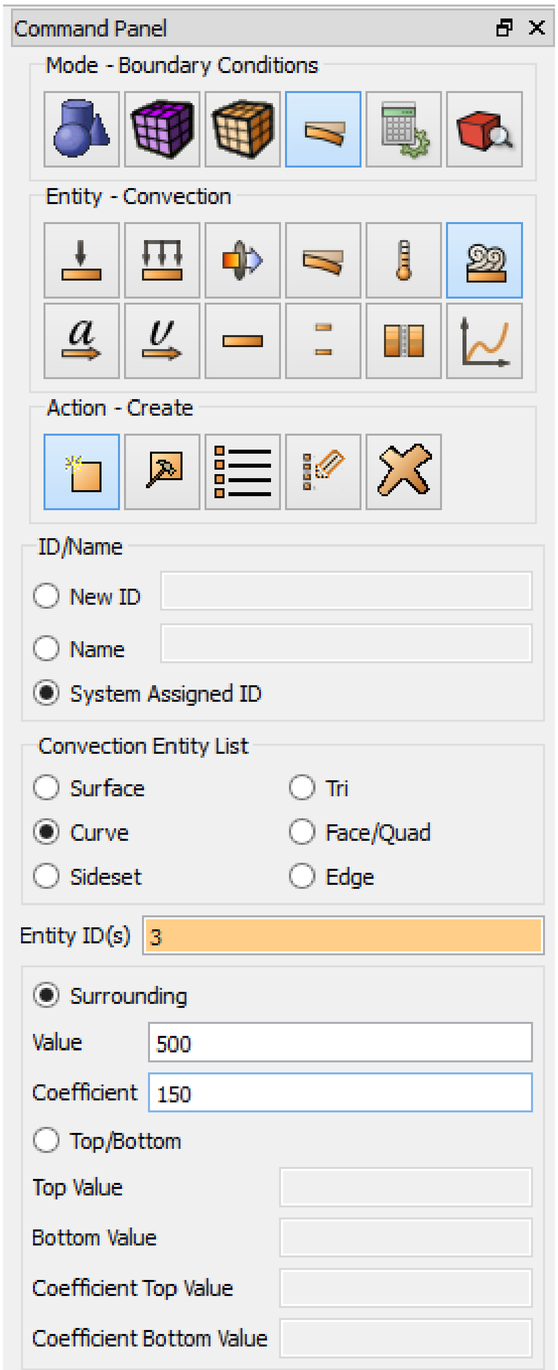
1. Set the process of convective heat exchange on the inner surface of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Convection**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID: 10;
- Select the way of parameters setting: Surrounding;
- Value: 500;
- Coefficient: 150.

Click **Apply**.



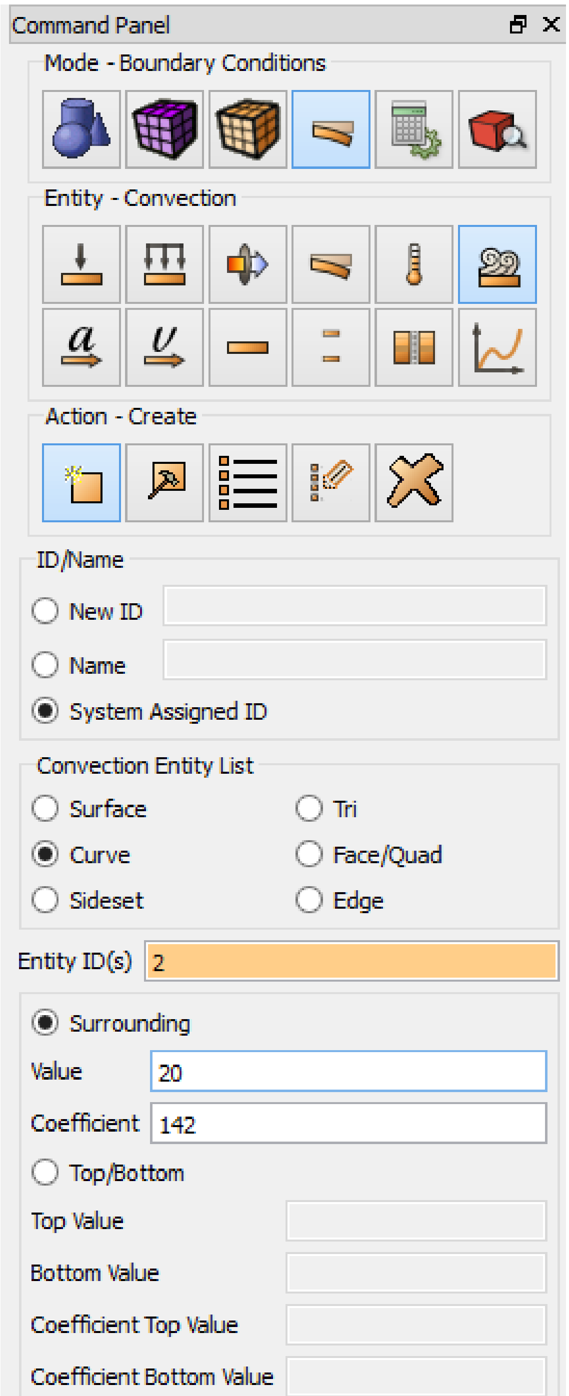


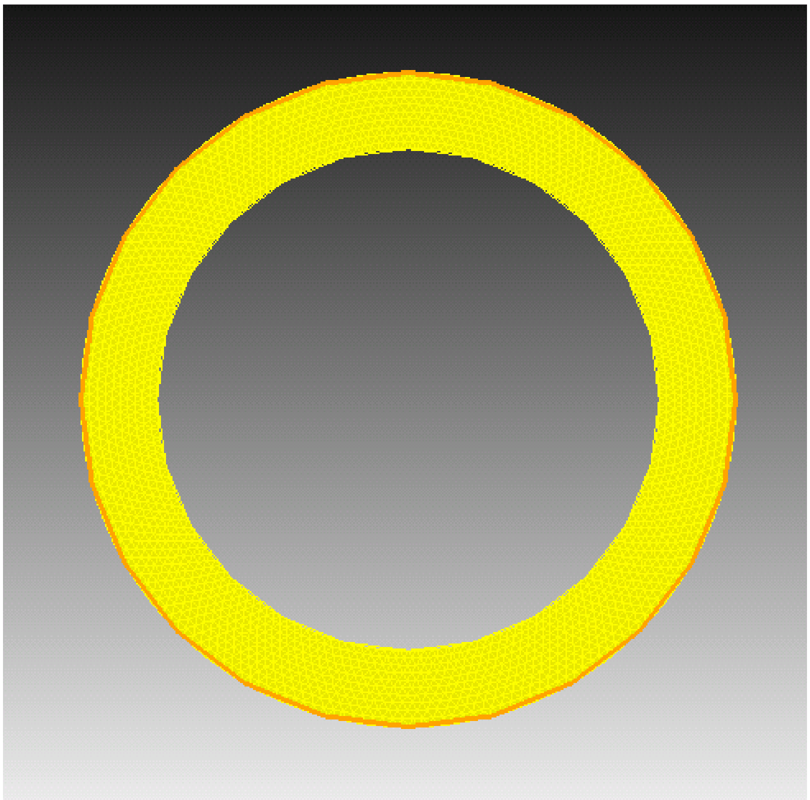
2. Set the process of convective heat exchange on the outer surface of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Convection**, Action – **Create** on Command Panel**.** Set the following parameters:

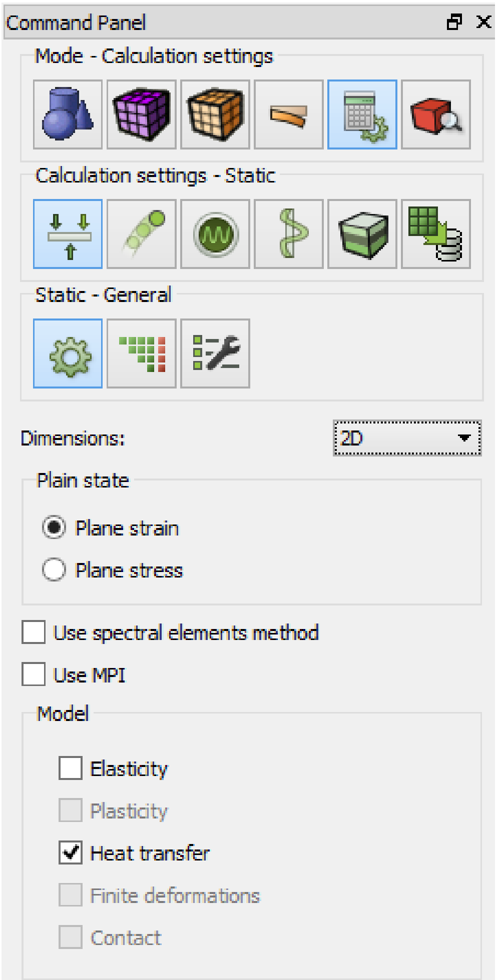
- System Assigned ID;
- Entity List: Curve;
- Entity ID: 2;
- Select the way of parameters setting: Surrounding;
- Value: 20;
- Coefficient: 142.

Click **Apply**.





Starting calculation



1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select Dimension – **2D**, Type of plane problem: Plane deformed state. Untick next to the item **Elasticity**. Tick next to the item **Heat transfer**.

Click **Apply**.

2. Set the solver settings.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

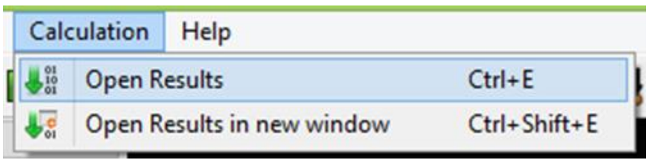
Click **Apply**.

Click **Start Calculation**.

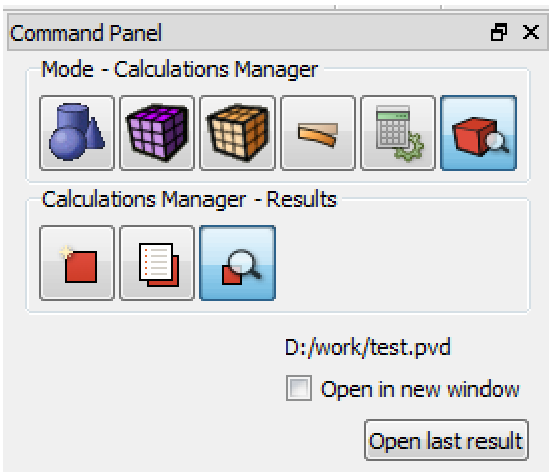
3. In a pop-up window select a folder to save the result and enter the file name.

4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis



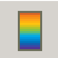
1. Open the file with the results. You can do this in one of the three ways.
   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



2. Display the component of temperatures.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: Temperature.



To display the color legend scale, click the button **Switch the color legend visibility** on Command Panel.

3. Select a point where you need to check the temperature.

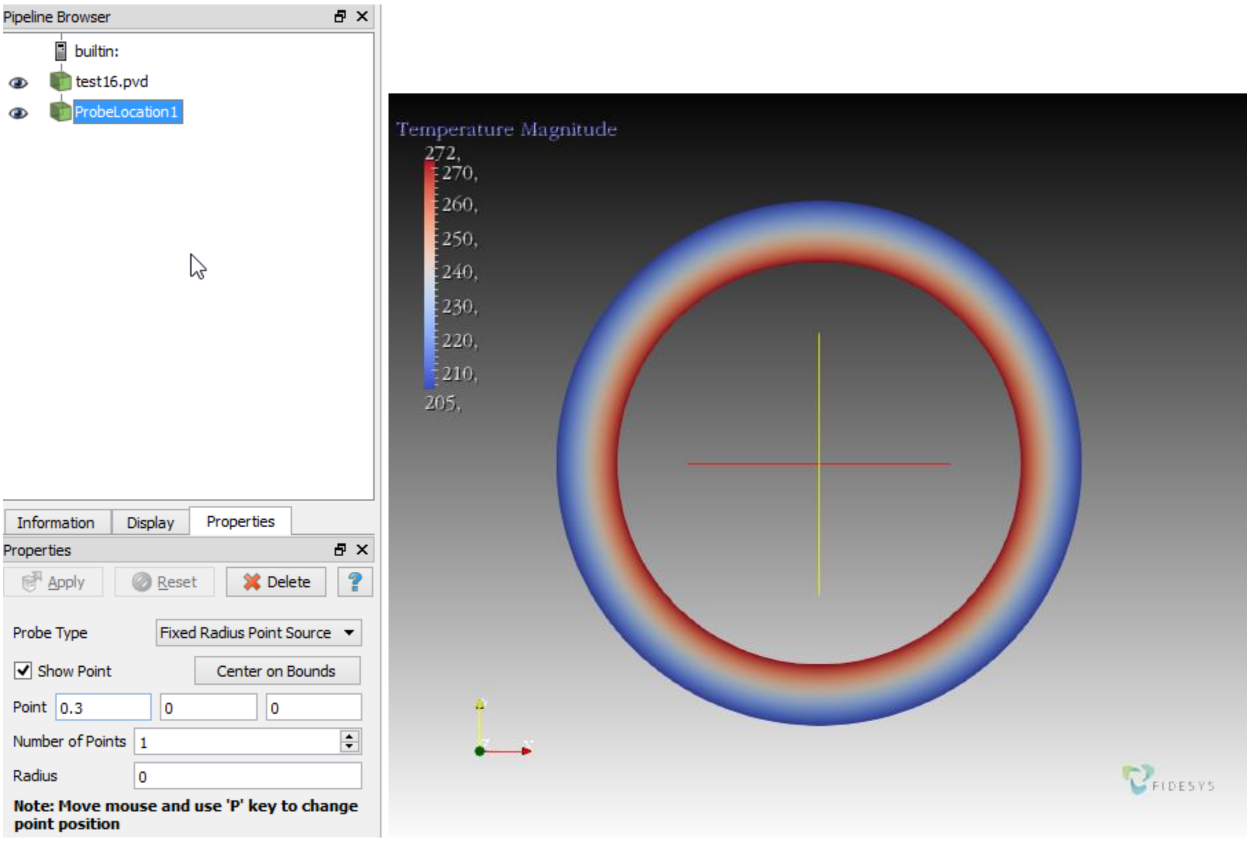
In the Main Menu, select the filter **Probe Location**. In the tab **Properties** set the coordinates of the point A where you need to view the stress:

- Show Point;
- Point (coordinates): 0.3 0 0;
- Number of Points: 1;
- Radius: 0.

Click **Apply**.

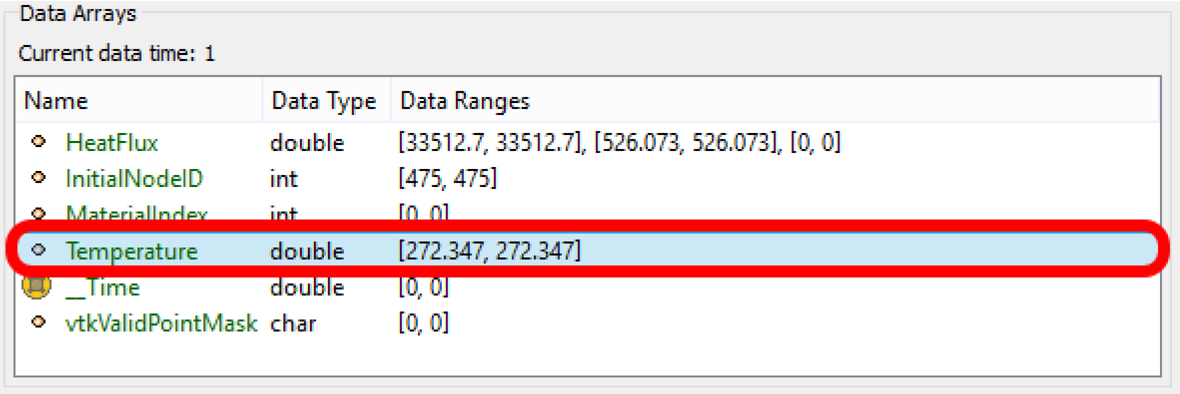
To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

As a result, point A is displayed at the picture.



4. Check a numerical temperature value T at the selected point A.

See the Temperature value in the line **Heat flux** in the tab **Information** in the field **Data Arrays**.


Data Arrays

Current data time: 1

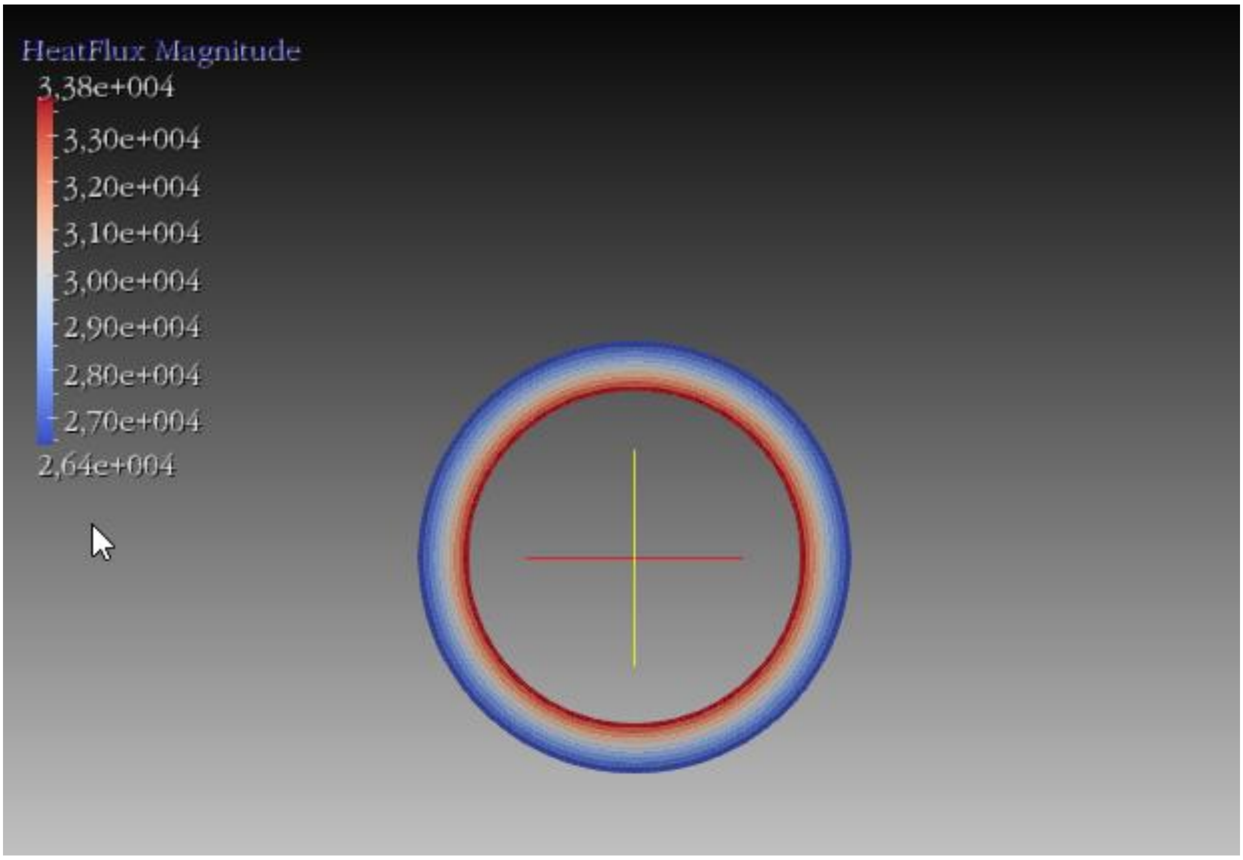
Name	Data Type	Data Ranges
HeatFlux	double	[33512.7, 33512.7], [526.073, 526.073], [0, 0]
InitialNodeID	int	[475, 475]
MaterialIndex	int	[0, 0]
Temperature	double	[272.347, 272.347]
__Time	double	[0, 0]
vtkValidPointMask	char	[0, 0]

The difference between the obtained value 272.347 and the required one 272.3 is 0.02%.

5. Display the component of the heat flux.

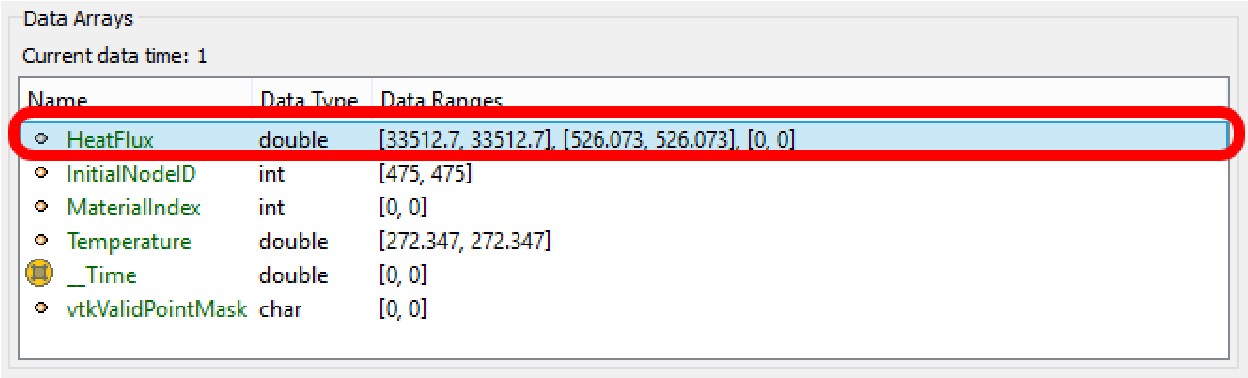
In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Mode: Surface;
- Representation Field: HeatFlux.



6. View a numerical value of the heat flux φ at the selected point A.

You can see the heat flux values in the line **HeatFlux** in the tab **Information** in the field **Data Arrays**..



The heat flux value is calculated using the following formula:

$$\sqrt{{\varphi_x}^2 + {\varphi_y}^2} = \sqrt{33512.7^2 + 526.073^2} = 33516.82882$$

The difference between the obtained 33516.82882 and the required one 3.416E4 is 1.88%.

7. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

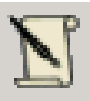
Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
create surface circle radius 0.3 zplane
create surface circle radius 0.391 zplane
subtract body 1 from body 2 imprint
curve 3 2 interval 200
curve 3 2 scheme equal
surface 3 scheme TriMesh
mesh surface 3
undo group begin
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "MODULUS" 2.1e+11 "POISSON" 0.3
"CONDUCTIVITY" 40
undo group end
set duplicate block elements off
block 1 surface 3
block 1 material 'Material 1'
block 1 element type tri3
create convection on curve 3 surrounding 500 coefficient 150
create convection on curve 2 surrounding 20 coefficient 142
analysis type static heattrans dim2 planestrain
```

```
spectralelement off
usempi off
solver method auto try_other off
solver method auto try_other offcalculation start path "D:/FidesysBundle/calc/example.pvd"
```



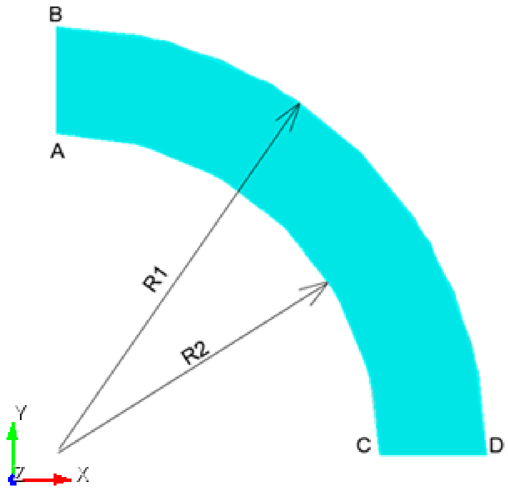
It is also possible to run the file *Example_11_Static_2D_Conduction.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Setting thermoelasticity (2D)

Боли Б.,Уэйнер Дж. Теория температурных напряжений. М.: Мир, 1964, p. 259.

[Boley B, Weiner J., Theory of thermal stresses (translated from English), Mir publ, Moscow, 1964, p.259 [in Russian]]

The problem of static temperature load of the hollow cylindrical body of external radius R1 = 4 m and inner radius R2 = 3 m is being solved. Temperature effect applied to the inner edge of the cylinder AC is constant and equal to T1 = 30°C. Temperature effect applied to the inner edge of the cylinder BD is constant and equal to T2 = 100°C.



Edges AB and CD are fixed on the symmetry condition:

- Zero X-Translation on the line AB;
- Zero Y-Translation on the line DC.

The material parameters are Young's Modulus E = 200 GPa, Poisson's Ratio = 0.3, thermal expansion μ = 0.0001 1/ °C.

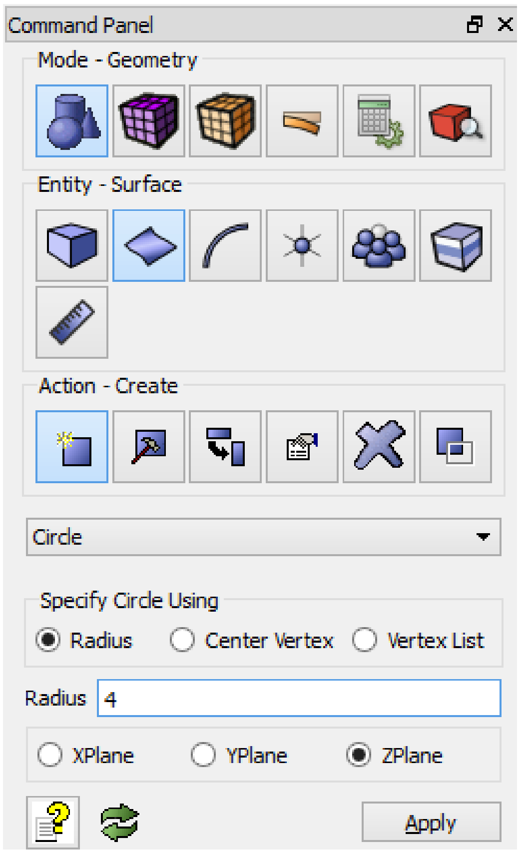
Test pass criterion is the following: Displacement u_x at the point (3, 0, 0) is 0.0205014 m within 1%.

Geometry creation

1. Create Circle 1.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Create**). Select **Circle** in the list of geometric elements. Specify the circle dimensions:

- Radius: 4;
- Location: ZPlane.



Click **Apply**.

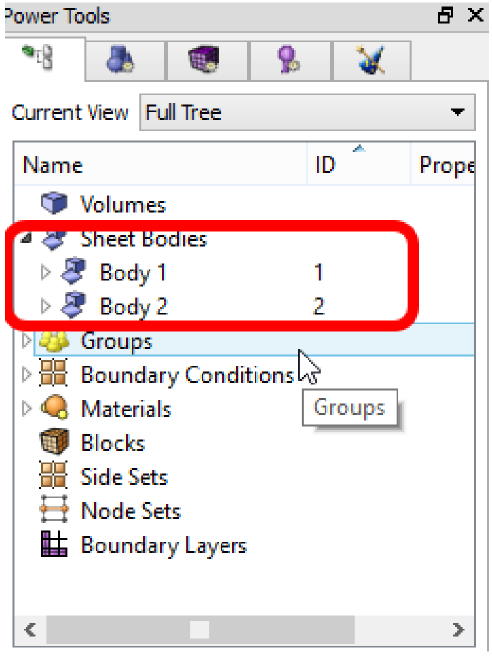
2. Create Circle 2

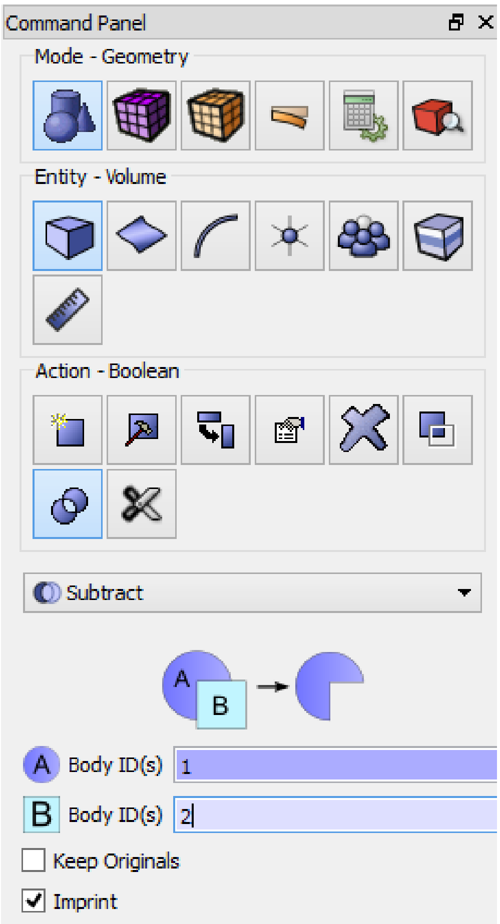
Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Surface**, Action – **Create**). Select **Circle** in the list of geometric elements. Specify the circle dimensions:

- Radius: 3;
- Location: ZPlane.

Click **Apply**.

As a result, two generated entities are displayed in the Model Tree (Body 1 and Body 2).





3. Subtract the first circle from the second one.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Body ID: 2 *(volumes from which other volumes will be subtracted)*;
- Substract bodies (ID): 1 *(the volumes to be subtracted)*;
- Imprint.

Click **Apply**

As a result, only one volume is displayed in the Model Tree (Body 1).

4. Leave a quarter of a surface (symmetry of the problem).

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Coordinate plane** in the list of possible webcut types. Set the following parameters:

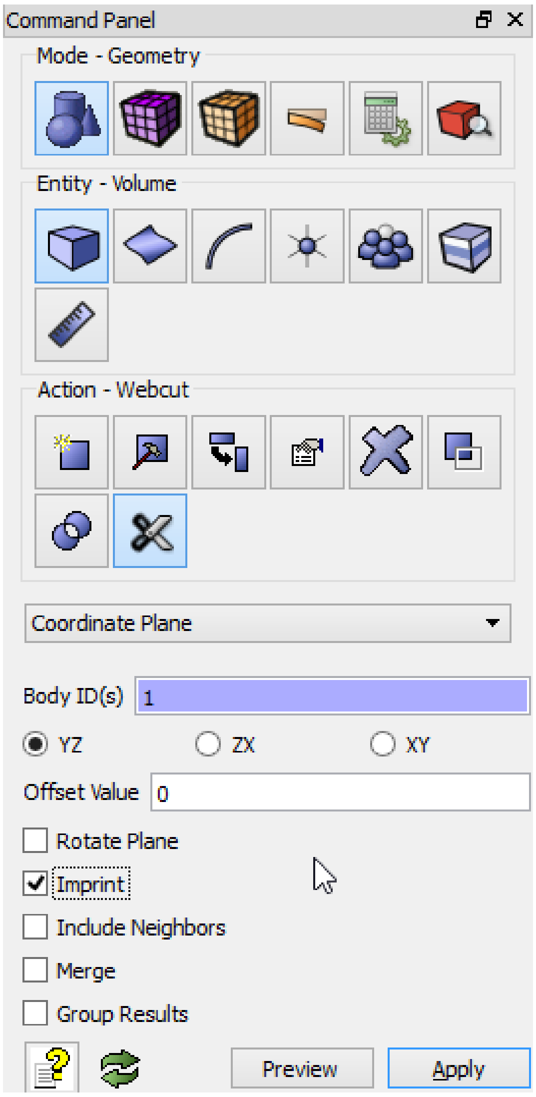
- Body ID: 1 *(the surface to be webcut)*;
- Webcut with: YZ Plane;
- Offset value: 0;
- Imprint.

Click **Apply.**

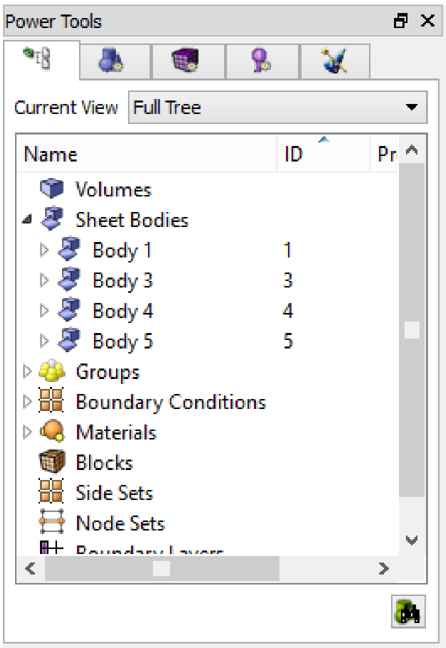
Do the same for the ZX Plane:

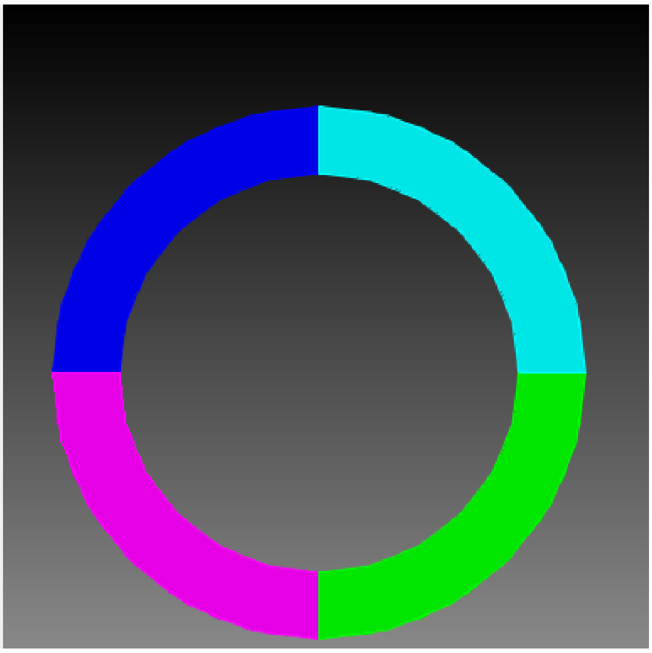
- Body ID: 1 *(the surfaces to be webcut)*;
- Webcut with: ZX Plane;
- Offset value: 0;
- Imprint.

Click **Apply**.

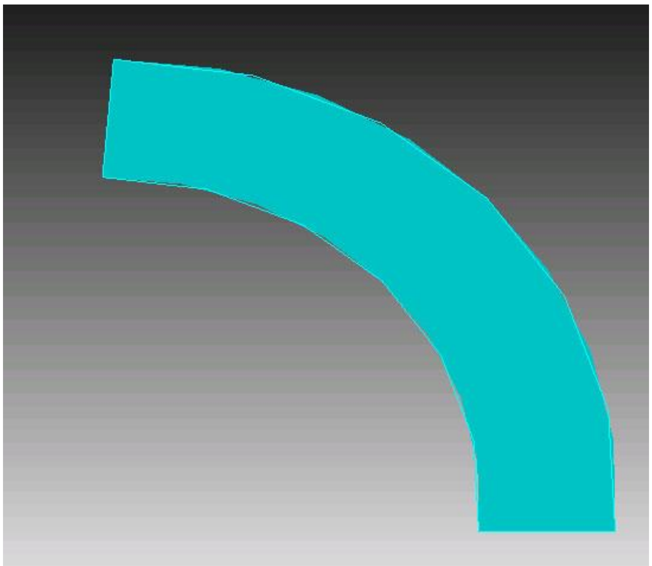


As a result, the original plane in the Model Tree is split into three (Body 1, Body 3, Body 4).





Delete the bodies 1, 3 and 5. To do this, right-click on these planes in the Model Tree and click **Delete** in contextual menu. As a result, a quarter of the original plane is left (Body 4):

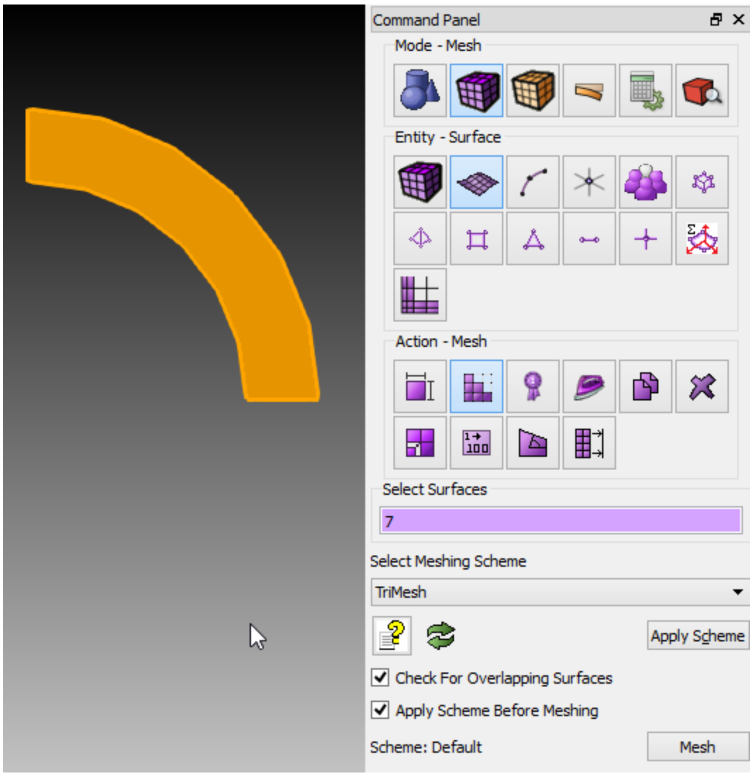


Meshing

1. Select meshing on planes section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Meshing**). Select meshing scheme:

- Select surfaces (specify their ID): 7 *(*or by the command **all***)*;
- Select meshing scheme: TriMesh;

Click **Apply Scheme**.

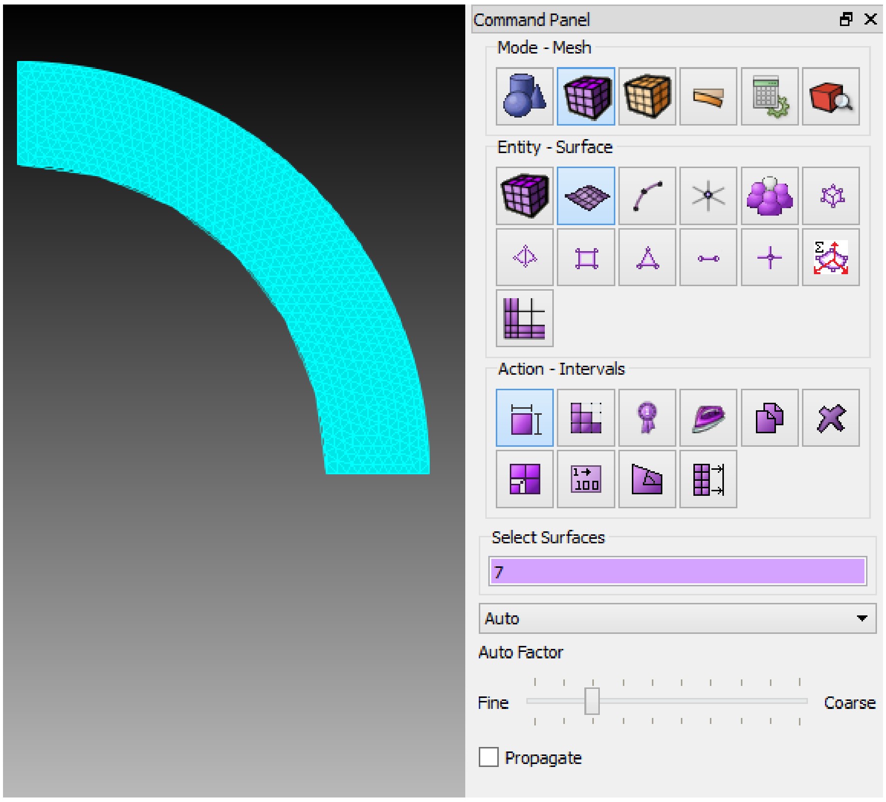


2. Select meshing on planes section on Command Panel (Mode – **Mesh,** Entity – **Surface**, Action – **Intervals**). Select the meshing interval:

- Select surfaces (specify their ID): 7 *(*or by the command **all***)*;
- Select meshing: Automatically;
- Move the cursor of the automatical multiplier to the third position from the left side.

Click **Apply**.

Click **Mesh**.



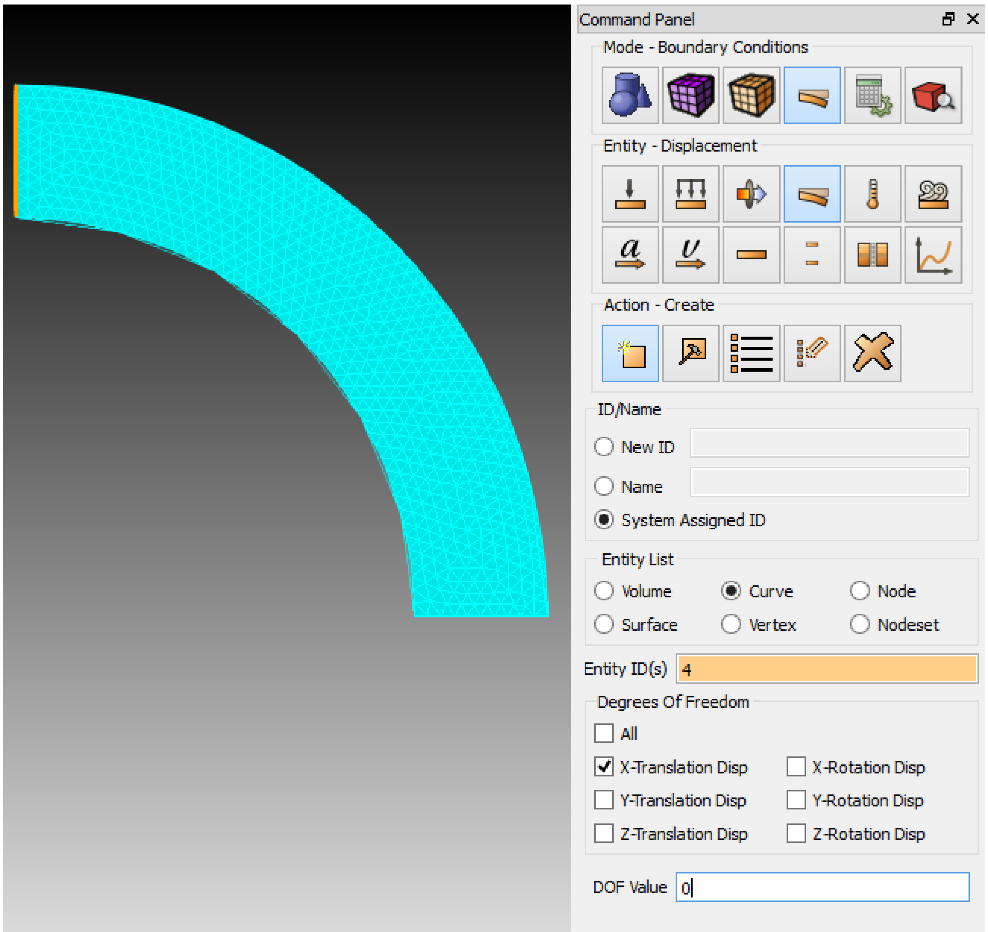
Setting boundary conditions

1. Fix one side line in the X direction.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entities ID: 4;
- Degrees of Freedom: X-Translation;
- DOF Value: 0.

Click **Apply**.



2. Fix one side line in the Y direction.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entities ID: 15;
- Degrees of Freedom: Y-Translation;
- DOF Value: 0.

Click **Apply**.

3. Set the the thermal stress on the inner curve of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Temperature**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entities ID: 17;
- Temperature value: 30.

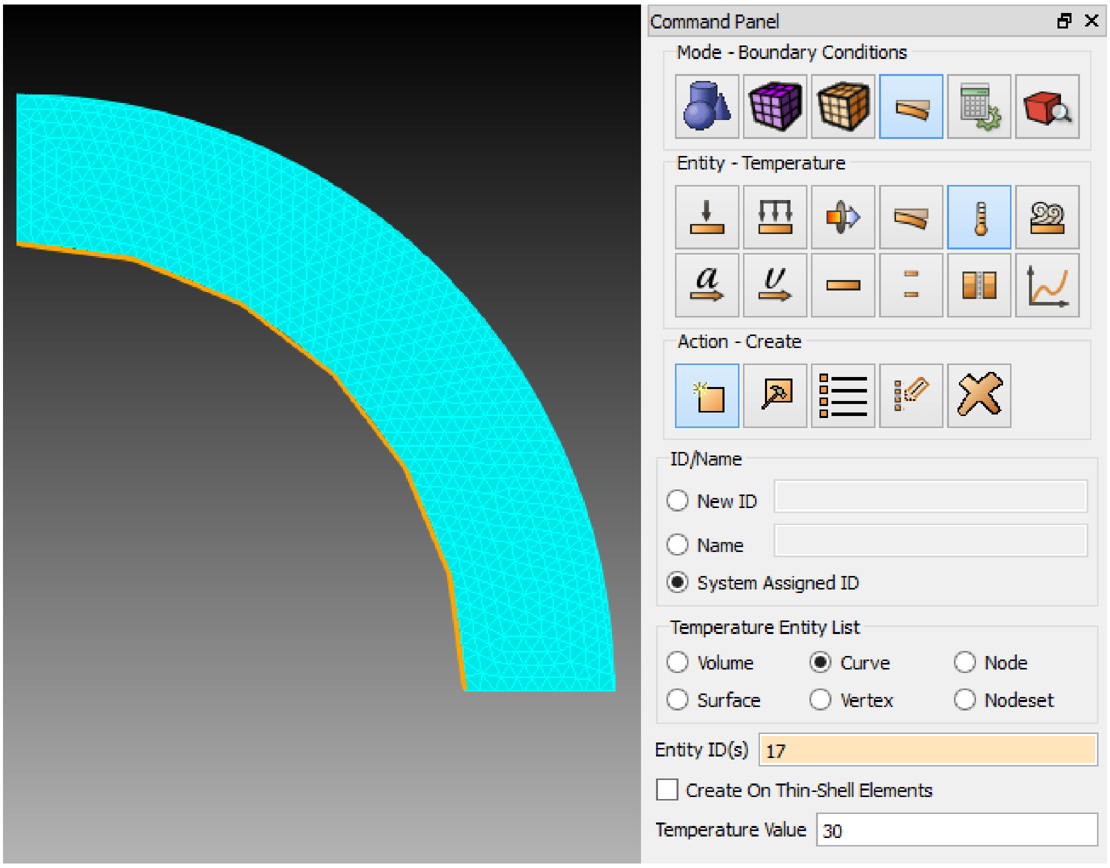
Click **Apply**.

4. Set the thermal stress on the outer curve of the cylinder.

Select Mode – **Boundary Conditions,** Entity – **Temperature**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID: 16;
- Temperature value: 100.

Click **Apply**.



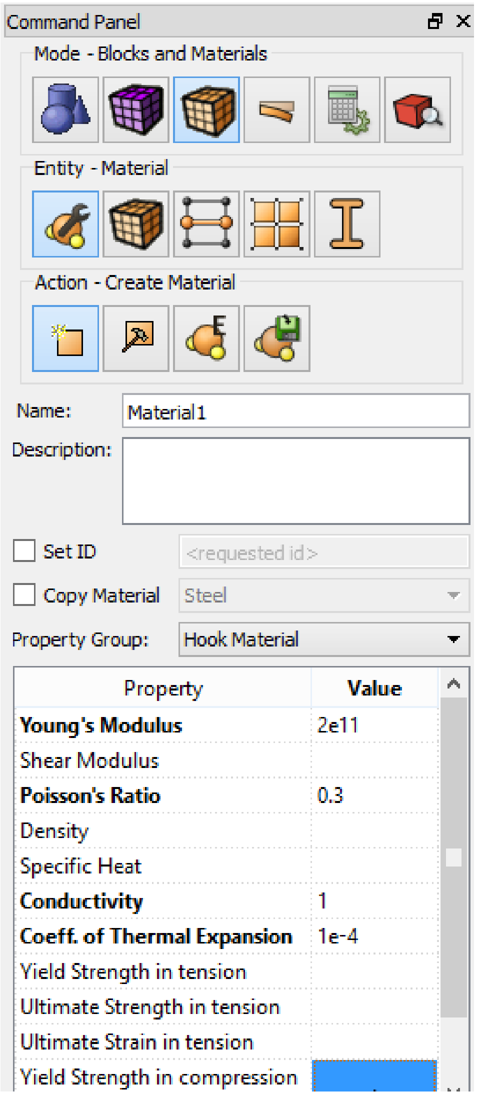
Setting material and element type

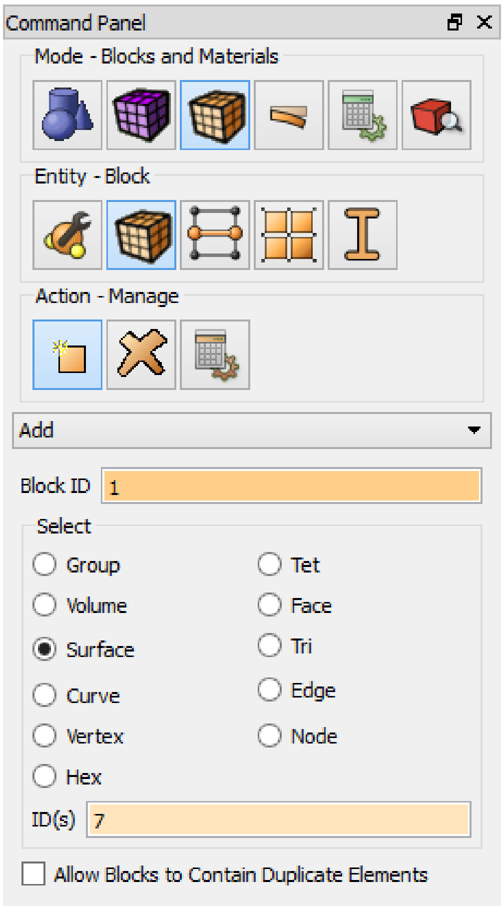
1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Enter the name for the material. Set the following parameters:

- Young's Modulus: 2e11;
- Poisson's Ratio: 0.3;
- Heat transfer: 1;
- Coeff. of Thermal Expansion: 1e-4.

Click **Apply**.





2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1
- Entity type to be united into the block: Surface;
- ID: 7 *(or by the command **all**).*

Click **Apply**.

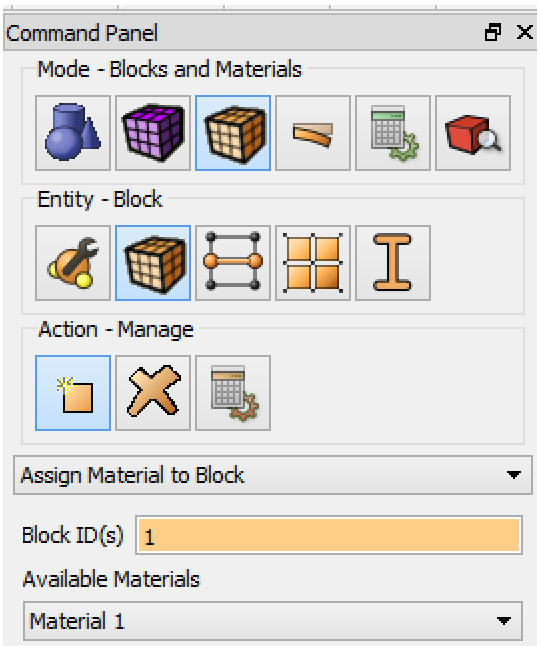
3. Assign the material to the block.

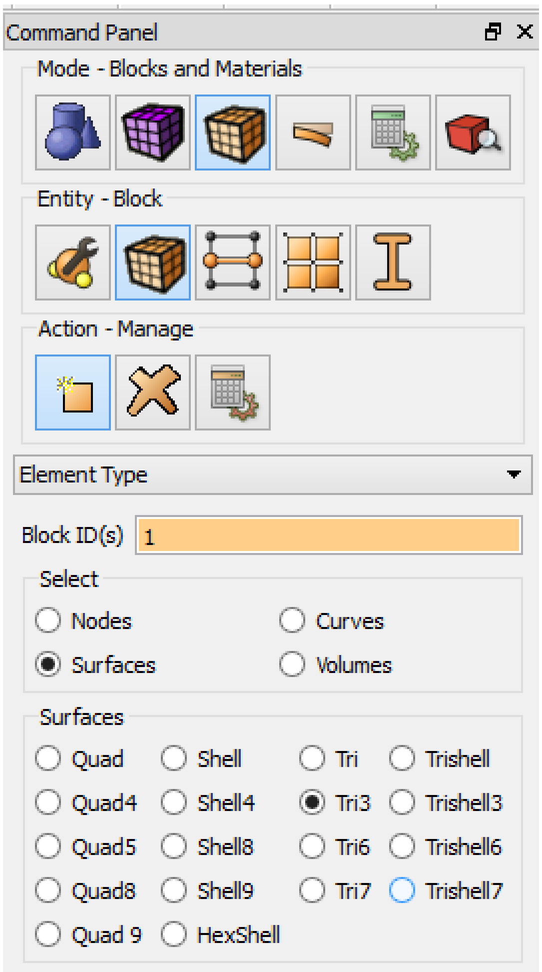
Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations.

Set the following parameters:

- Block(s) ID: 1;.
- Select the previously created material in the list: Material 1.

Click **Apply**.





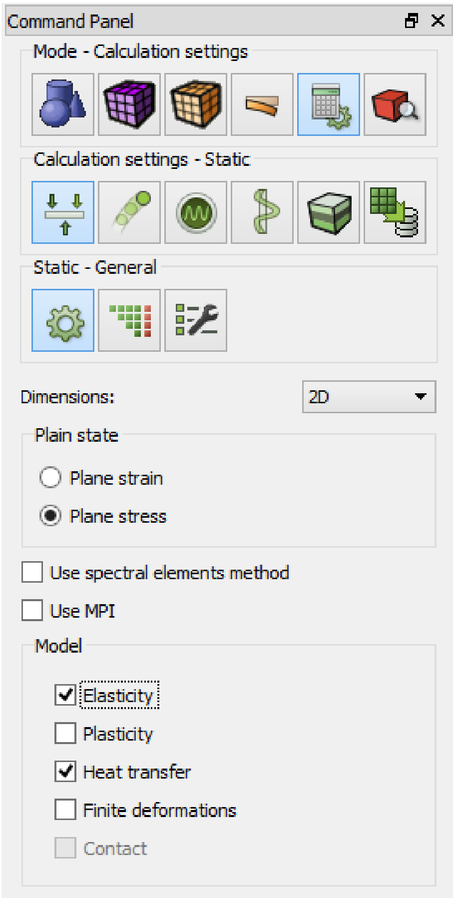
4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Surfaces;
- Surfaces: Tri3.

Click **Apply**.

Starting calculation



1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **General**). Select Dimension – **2D**, Type of plane problem – **Plane stress**. Tick next to the item **Heat transfer**.

Click **Apply**.

2. Set the solver settings.

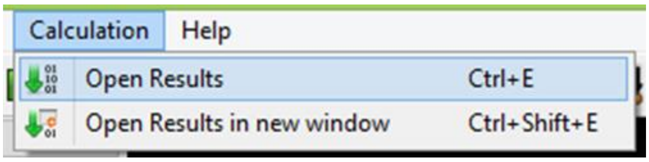
Select calculation setting section on Command Panel (Mode — **Calculation settings**, Calculation settings — **Static**, Static — **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply**.

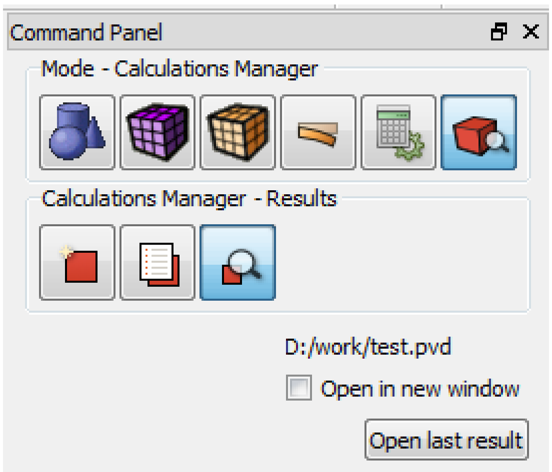
Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name.
4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>".*

Results analysis



1. Open the file with the results. You can do this in one of the three ways.
   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (Mode — **Calculations Manager**, Calculations Manager — **Results**). Click **Open Results**.



2. Display the u_R component of the displacements field.

Set the following parameters on ***Fidesys Viewer*** Toolbar:

- Representation Mode: Surface With Edges;
- Representation Field: Displacement;
- Representation Component: 1.

As a result, the displacement field u_x is displayed.

3. Select a point where you need to view the Displacement.

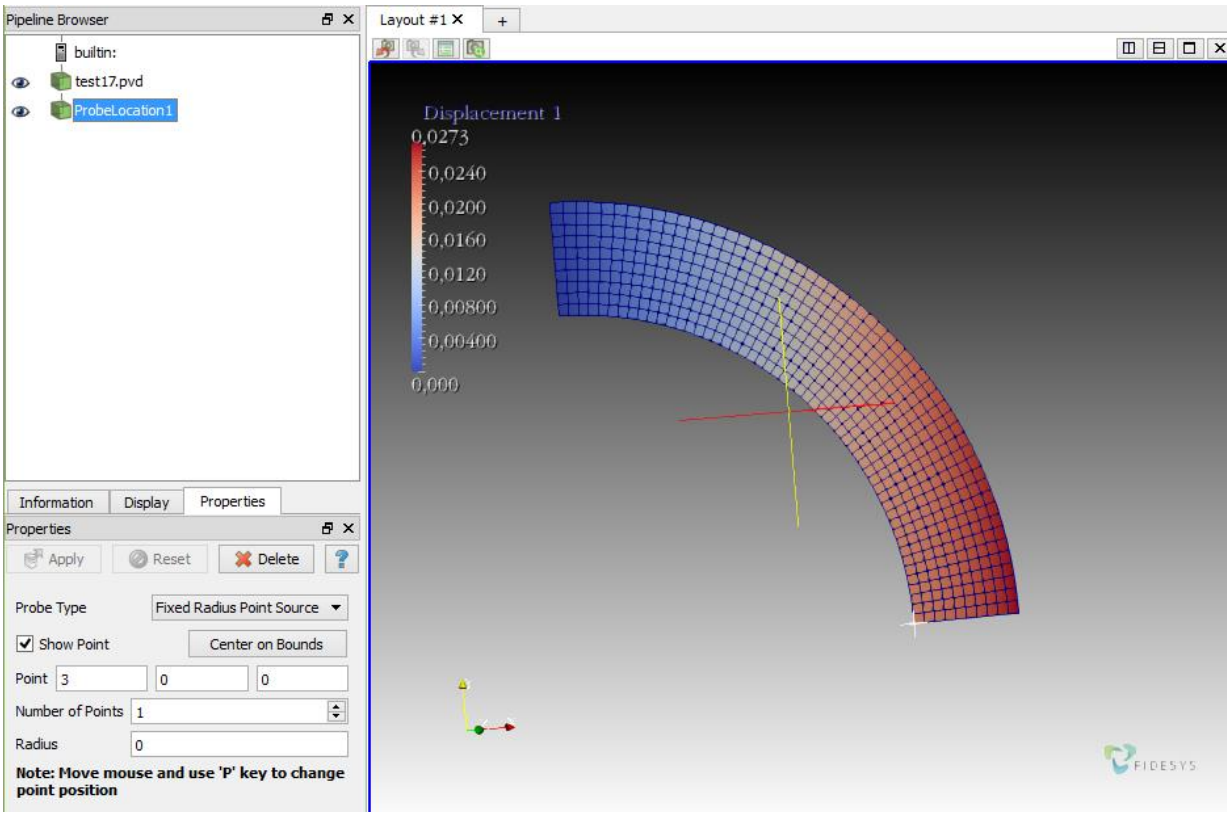
In the Main Menu, select the filter **Probe Location** in the tab **Filters** → **Alphabetical.** In the tab **Properties** set the coordinates of the point E where you need to view the Displacement:

- Show Point;
- Point (coordinates): 3 0 0;
- Number of Points: 1;
- Radius: 0.

Click **Apply**.

 To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

As a result, point C is displayed at the picture.



4. View a numerical valueof U_x at the selected point C.

You can see the required value in the tab **Information** in the field **Data Arrays** (the first value in the line **Displacement** corresponds to the component u_x).

Data Arrays

Current data time: 1

Name	Data Type	Data Ranges
Displacement	double	[0.0204912, 0.0204912], [0, 0], [0, 0]
HeatFlux	double	[-79.784, -79.784], [-1.16049, -1.16049], [0, 0]
InitialNodeID	int	[1, 1]
MaterialIndex	int	[0, 0]
Strain	double	[0.0024427, 0.0024427], [0.00675753, 0.00675753], [-0.00102959, -0.00102959], [-6.27743e-05, -6....
Stress	double	[1.12887e+07, 1.12887e+07], [6.75108e+08, 6.75108e+08], [0, 0], [-9.65758e+06, -9.65758e+06], ...
Temperature	double	[30, 30]
_Time	double	[0, 0]
vtkValidPointMask	char	[0, 0]

The difference between the obtained 0.0204953 and the required one 0.0205014 is 0.03%.

5. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+C**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
set node constraint on
create surface circle radius 4 zplane
create surface circle radius 3 zplane
subtract body 2 from body 1 imprint
webcut body 1 with plane xplane offset 0 imprint preview
webcut body 1 with plane xplane offset 0 imprint
webcut body 1 with plane yplane offset 0 imprint preview
webcut body 1 with plane yplane offset 0 imprint
delete Body 1
delete Body 3
surface 7 size auto factor 3
surface 7 size auto factor 3
mesh surface 7
create displacement on curve 4 dof 1 fix 0
create displacement on curve 15 dof 2 fix 0
create temperature on curve 17 value 30
create temperature on curve 16 value 100
undo group begin
```

```
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "MODULUS" 2e+11 "POISSON" 0.3
"CONDUCTIVITY" 1 "THERMAL_EXPANSION" 0.0001
undo group end
set duplicate block elements off
block 1 surface 7
block 1 material 'Material 1'
block 1 element type tri3
analysis type static elasticity heattrans dim2 planestress
spectralelement off
usempi off
solver method auto try_other off
solver method auto try_other off
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

It is also possible to run the file *Example_12_Static_2D_Termoelasticity.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Temperature load stability (beam model)

S.P. Timoshenko, J.M Manages "Theory of elastic stability" second edition. Dunod, 1966, 500 pages

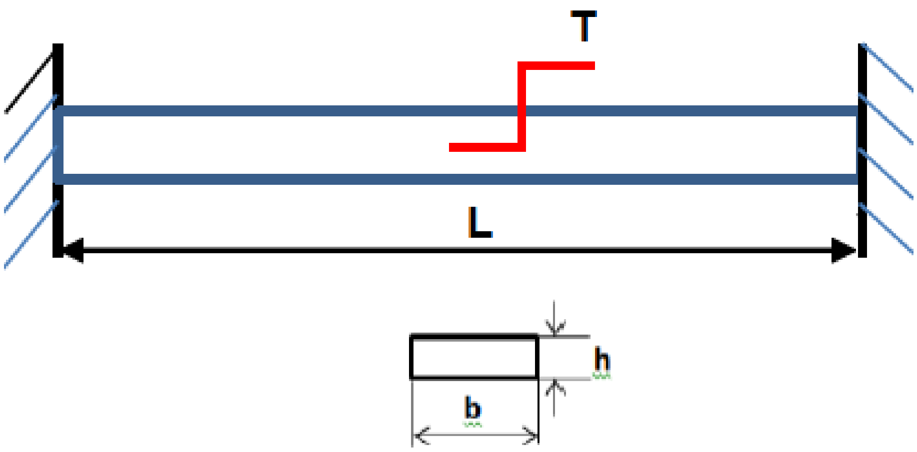
The problem of stability of a body clamped at butts and subjected to the uniform temperature heat is being solved.

The picture represents a geometric model of the problem.

A beam length is L = 200 mm. Rectangular cross section: width – 10 mm, thickness – 1 mm. Both ends of the beam are rigidly fixed at all displacements and rotations.

The material is isotropic. Elastic modulus is E = 200 000 N/mm^2, Poisson's Ratio is ν = 0.33, the coefficient of thermal expansion is α=11.7·10^{-6} C^{-1}.

Test pass criterion is the following:
critical temperature is T_{crit} = 7.028° C within 1% .



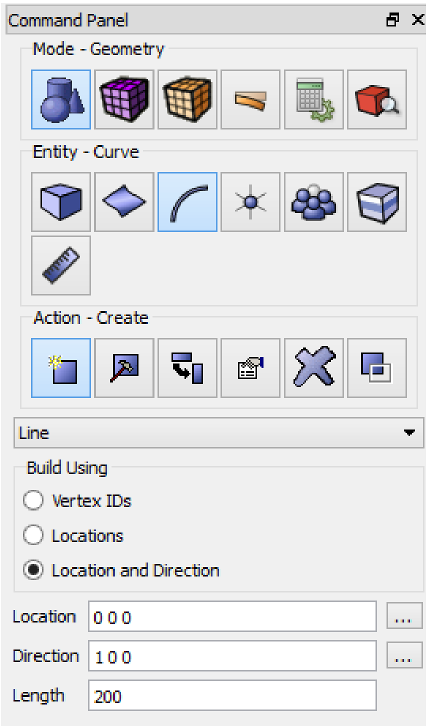
Geometry creation

1. Create the line.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Curve**, Action – **Create**). Select **Circle** in the list of geometric elements. Create using **Location and Direction**. Set Parameters:

- Location: 0 0 0;
- Direction: 1 0 0;
- Length: 200.

Click **Apply**.

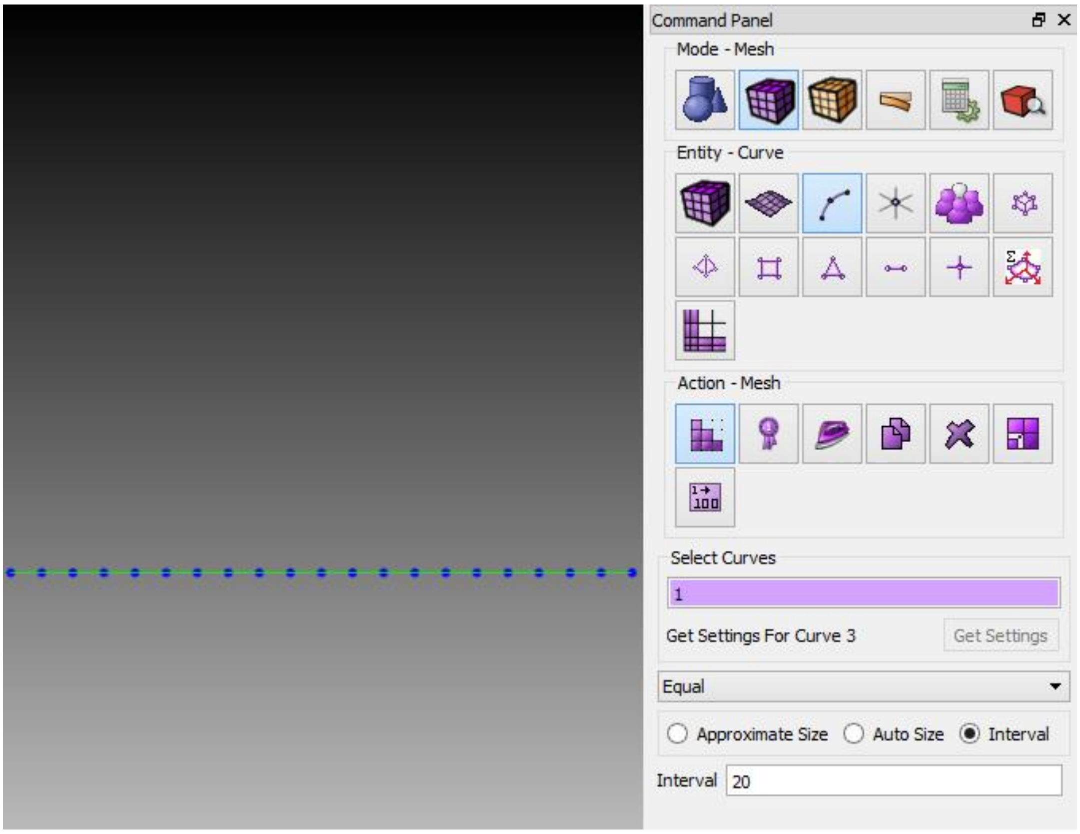


Meshing

1. Select meshing on curves section on Command Panel (Mode – **Mesh,** Entity – **Curve**, Action – **Meshing**). Specify the parameters of mesh refinement:
    - Select Curves: 1;
    - Select the way of meshing: Equal;
    - Select the meshing parameters: Interval;
    - Interval: 20.

Click **Apply**.

Click **Mesh**.



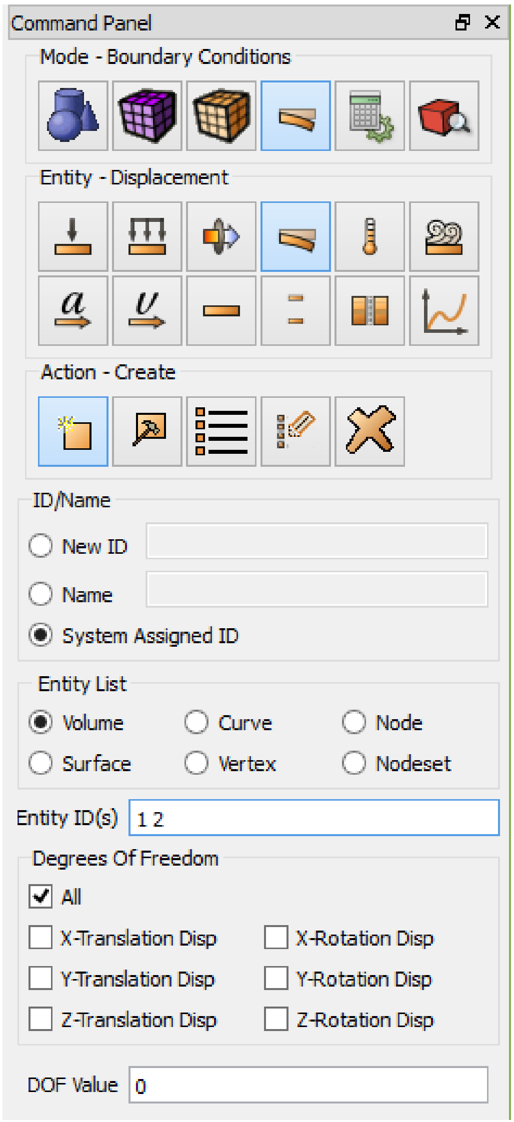
Setting boundary conditions

1. Fix beam ends.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Vertex;
- Entity ID(s): 1 2 *(using space after each of them)*;
- Degrees of Freedom: All;
- DOF Value: 0.

Click **Apply**.

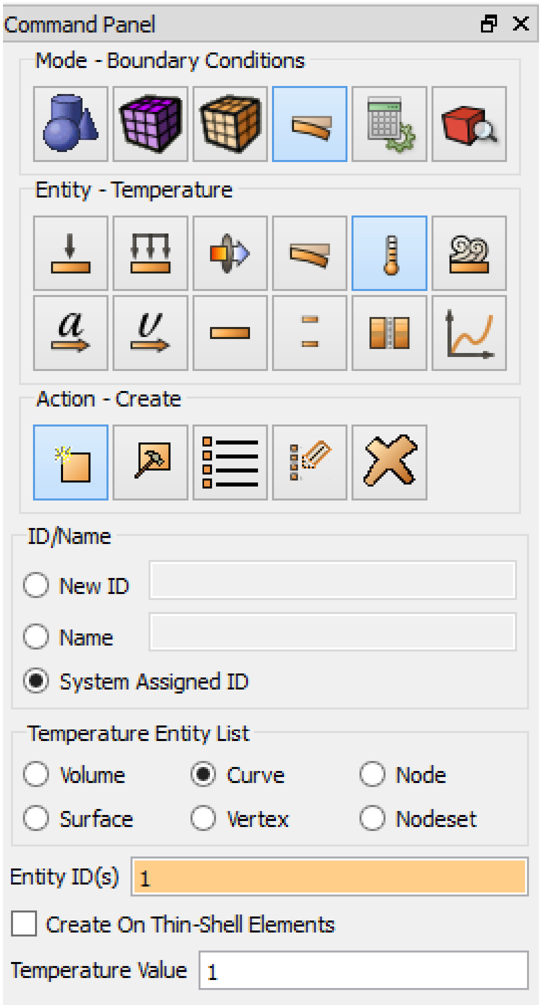


2. Set the temperature load.

Select Mode – **Boundary Conditions,** Entity – **Temperature**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Curve;
- Entity ID(s): 1;
- Temperature value: 1.

Click **Apply**.



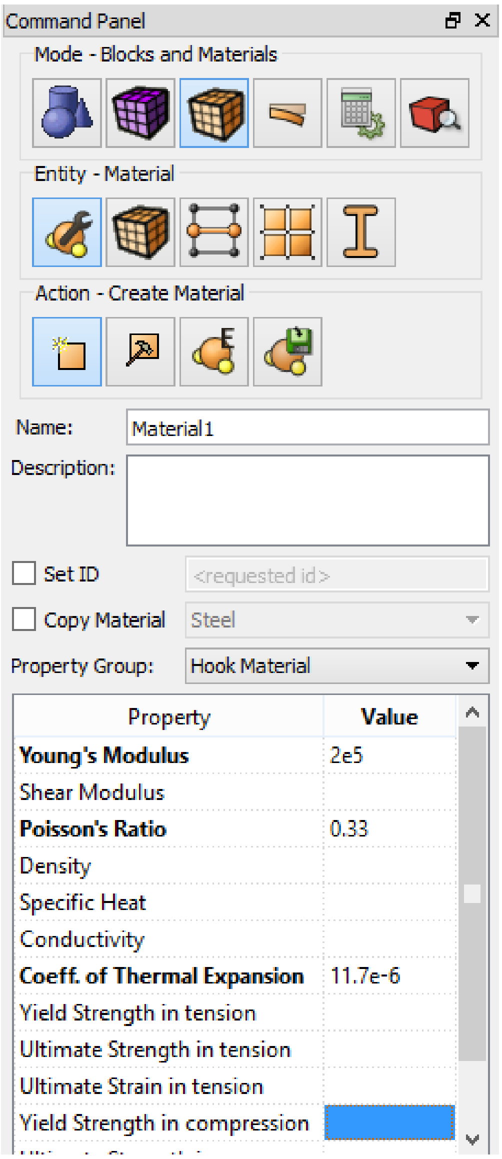
Setting material and element type

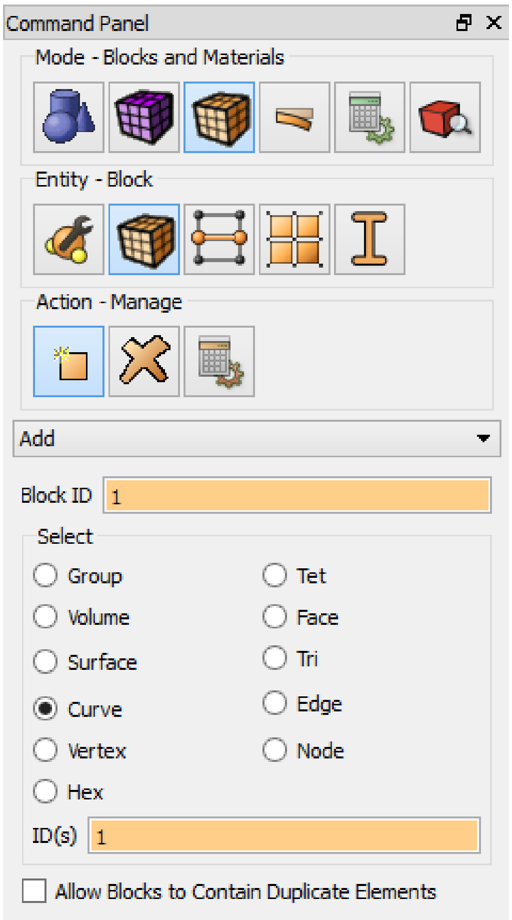
1. Create the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Material**, Action — **Create Material**). Enter the name for the material. Set the following parameters:

- Young's Modulus: 2e5;
- Poisson's Ratio: 0.33;
- Coeff. of Thermal Expansion: 11.7e-6.

Click **Apply**.





2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Curve;
- ID: 1 *(or by the command **all**)*.

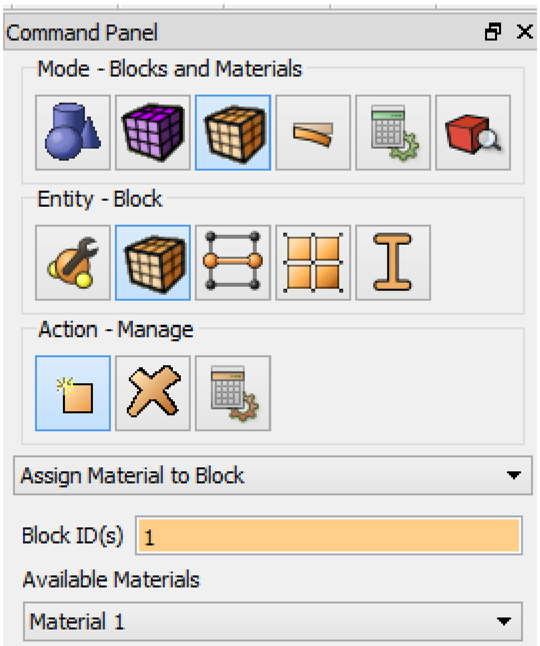
Click **Apply**.

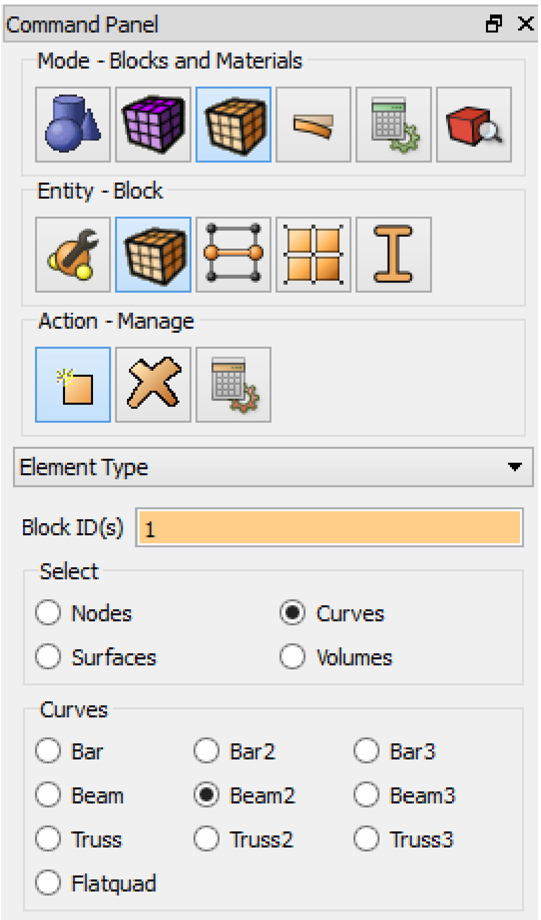
3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.





4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Curves;
- Curves: Beam2.

Click **Apply**.

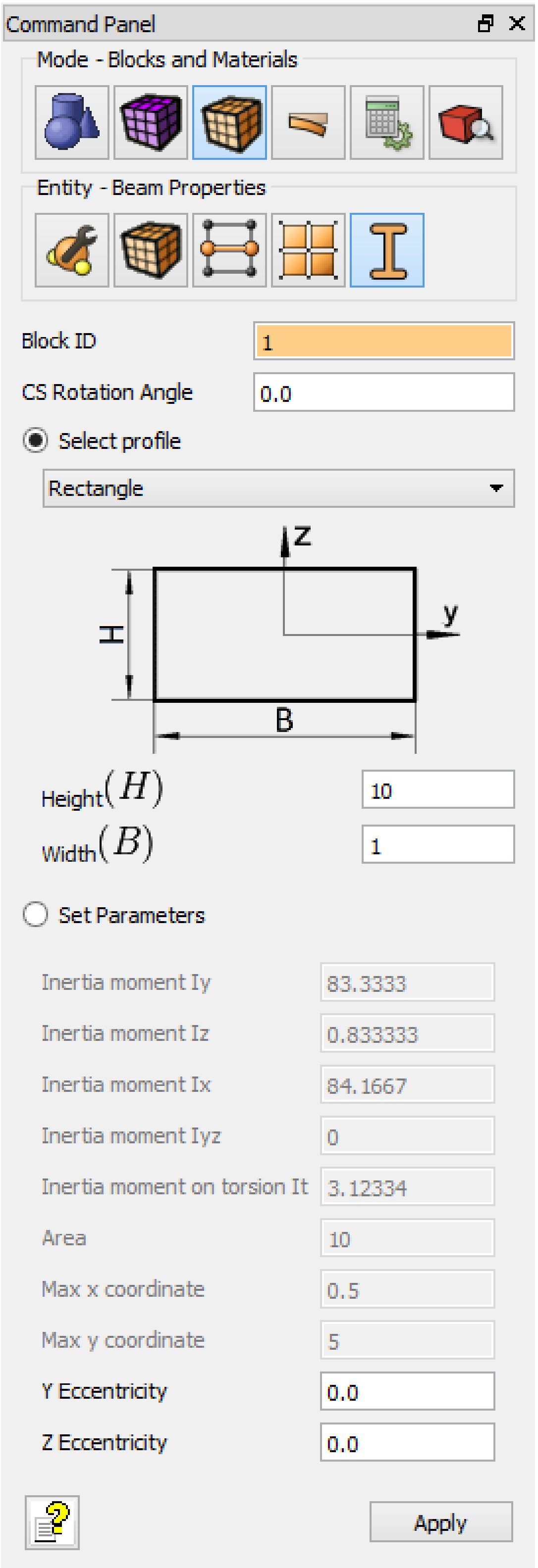
Setting beam cross section profile

1. Set the parameters of the beam.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Beam parameters)**. Select the checkbox at Cross section profile. Select **Rectangle** from the list of possible profiles. Set the following parameters:

- Block ID: 1;
- Height (H): 10;
- Width (B): 1.

Click **Apply**.

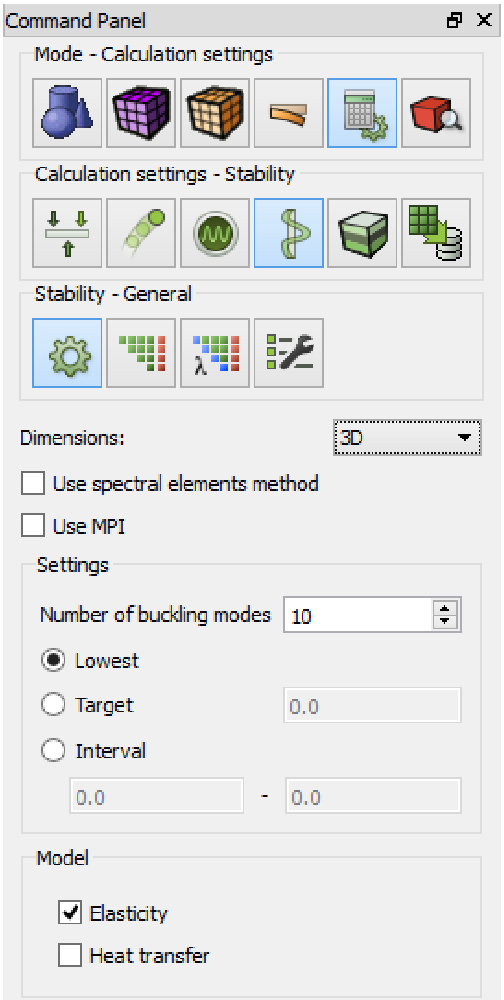


Starting calculation

1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Stability**, Stability – **General**). Leave all the settings by default.

Click **Apply**.



2. Set the solver settings.

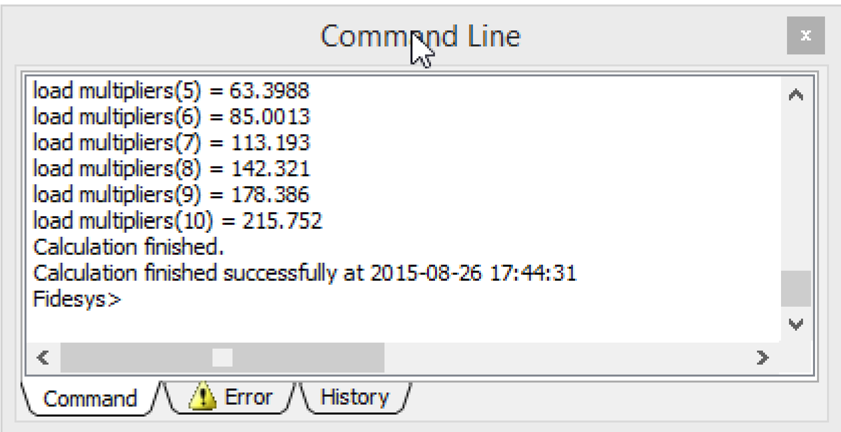
Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Stability**, Stability ость – **Solver**). Select the solver method (direct or iterative) and set Convergence Parameters in case of choosing an iterative one. You can also leave all the settings by default.

Click **Apply.**

Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name.

4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"* as well as the required values of the critical value.

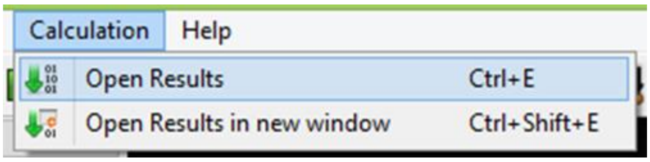


Results analysis

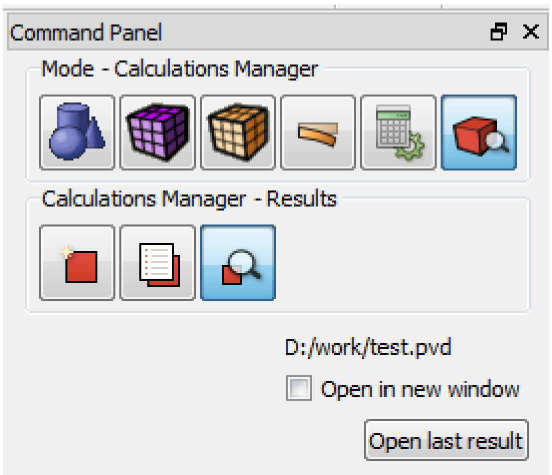
1. Compare the obtained results to those given in the table.

№	Analitical solution	FIDESYS	
		Value, ° C	**Error**
1	7.028	7.02982	0.03%

2. Open the file with the results. You can do this in one of the three ways.



- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



3. Display the 3D-view of the model (beam with thickness).

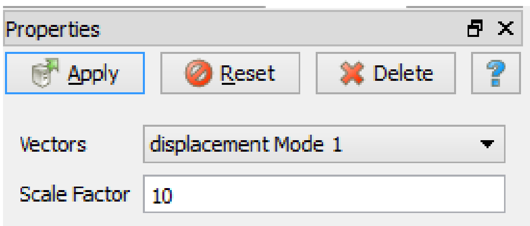
To do this, click on the name of the source file in the Model Tree in the pop-up ***Fidesys Viewer*** window. After this click 3D-view button in the default string .

The file *_3D.pvd with a 3D-image of the beam must be opened and you will be able to apply various filters to it and to view its deformed view.

4. Select a filter **Warp By Vector**.

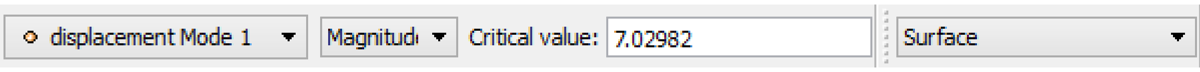
Choose the new file example Beam_Stability_3D.pvd in the Model Tree and display Filters **Warp by Vector** for it with the following field values :

- Vectors: displacements Mode 1
- Scale Factor: 10



5. Display displacements Mode 1.

In **Fidesys Viewer** window set the following parameters on Toolbar:

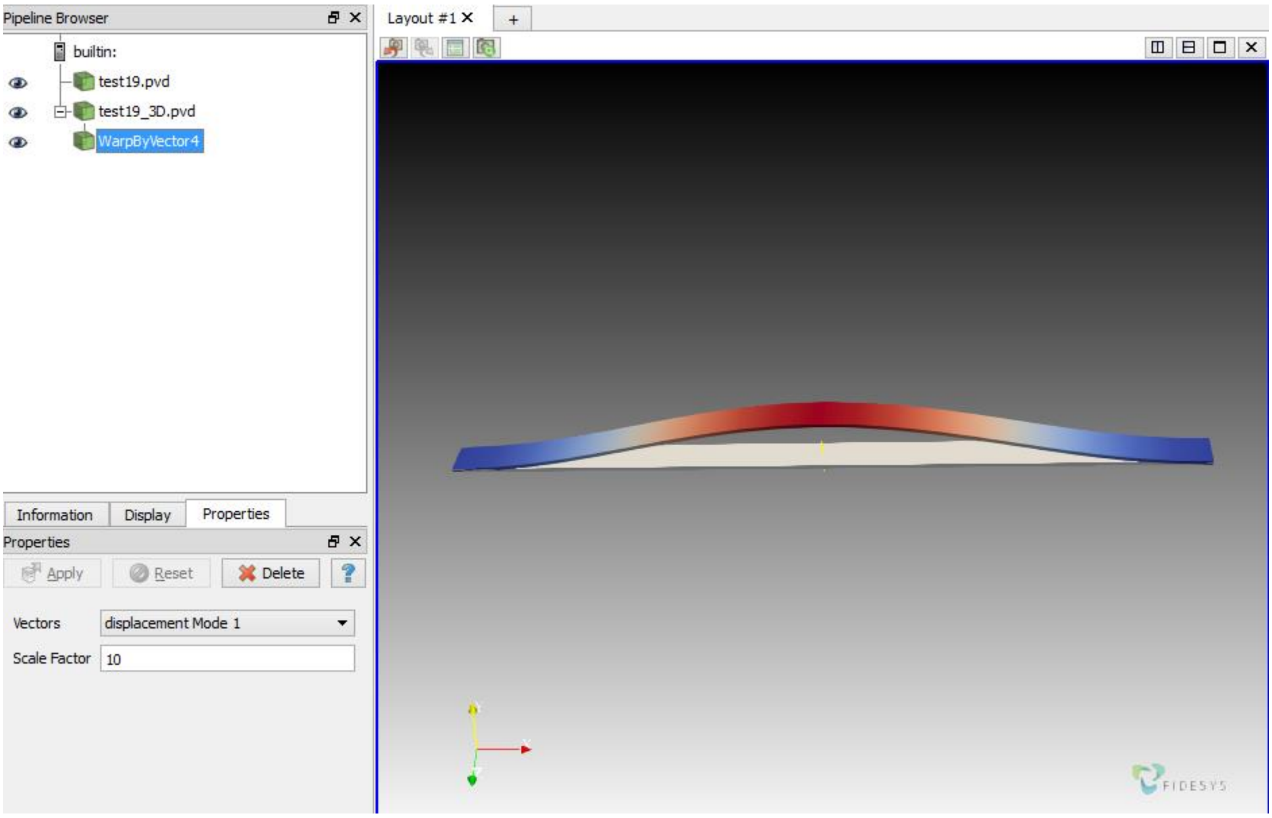


Make sure that the first required critic temperature value is displayed in the window **Critical value**.

To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

6. View results.

As a result, the deformed body is displayed at the picture. To see the original model, click near the model in the Model Tree. The picture below shows the deformed (solid grey filling) and the original model (with the field of displacements distribution for Mode 11).



7. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

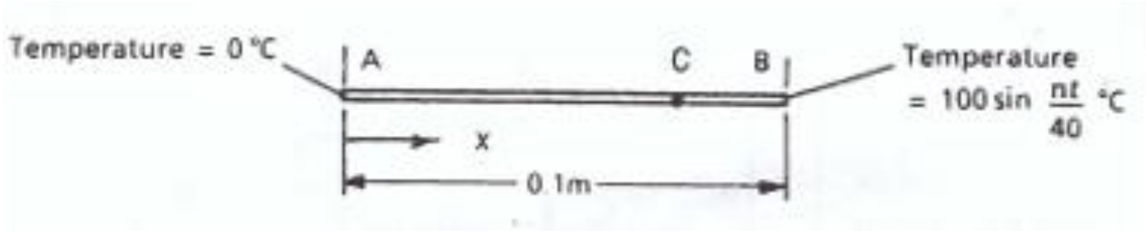
```
reset
set node constraint on
create curve location 0 0 0 direction 1 0 0 length 200
curve 1 interval 20
curve 1 scheme equal
curve 1 interval 20
curve 1 scheme equal
mesh curve 1
create displacement on vertex 1 2 dof all fix 0
create temperature on curve 1 value 1
undo group begin
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "MODULUS" 2e+05 "POISSON" 0.33
"THERMAL_EXPANSION" 1.17e-05
undo group end
set duplicate block elements off
block 1 curve 1
block 1 material 'Material 1'
block 1 element type beam2
block 1 attribute count 14
block 1 attribute index 1 value 10 name 'A'
block 1 attribute index 2 value 3.12334 name 'It'
block 1 attribute index 3 value 84.1667 name 'Ix'
block 1 attribute index 4 value 83.3333 name 'Iy'
block 1 attribute index 5 value 0 name 'Iyz'
block 1 attribute index 6 value 0.833333 name 'Iz'
block 1 attribute index 7 value 0 name 'angle'
block 1 attribute index 8 value 0 name 'ey'
block 1 attribute index 9 value 0 name 'ez'
block 1 attribute index 10 value 0.5 name 'max_y'
block 1 attribute index 11 value 5 name 'max_z'
block 1 attribute index 12 value 0 name 'section_type'
block 1 attribute index 13 value 10 name 'geom_H'
block 1 attribute index 14 value 1 name 'geom_B'
analysis type stability elasticity dim3
eigenvalue find 10 smallest
spectralelement off
usempi off
solver method auto try_other off
solver method auto try_other offcalculation start path "D:/FidesysBundle/calc/example.pvd"
```

It is also possible to run the file *Example_13_Stability_Temperature_Beam.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Dynamic load: nonsteady heat transfer (3D, implicit scheme)

The 3D problem of 1D nonsteady heat transfer inside a beam is being solved.

The picture below represents a geometric model of the problem:



The beam length is 0.1 m, square cross section is 0.01x0.01 m. The temperature at the point A is T_A = 0 °C, the temperature at the point B varies harmonically: $T_B = 100 \sin\frac{\pi t}{40}$ °C. The material parameters are isotropic, V = 35 W/(m·°C), C = 440.5 J/(kg ·°C), ρ = 7 200 kg/m^3.

Test pass criterion is the following: temperature T at the point C (0.8;0;0) at time t = 32c is 36.60°C within 2%.

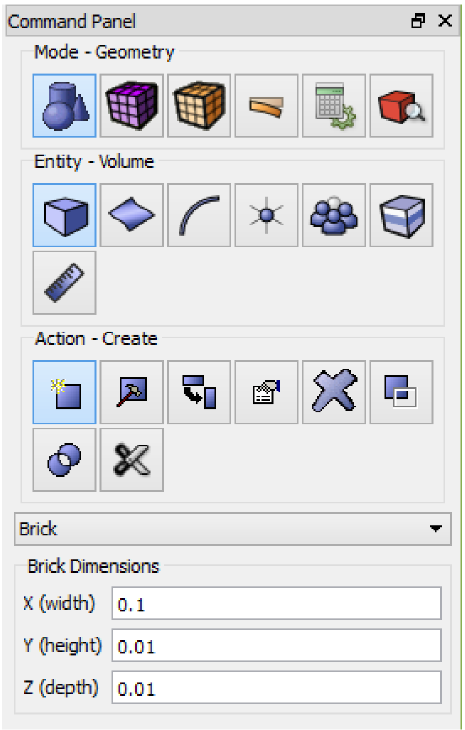
Geometry creation

1. Create the sliver parallelepiped.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Create**). Select **Brick** in the list of geometric elements. Set the brick dimensions:

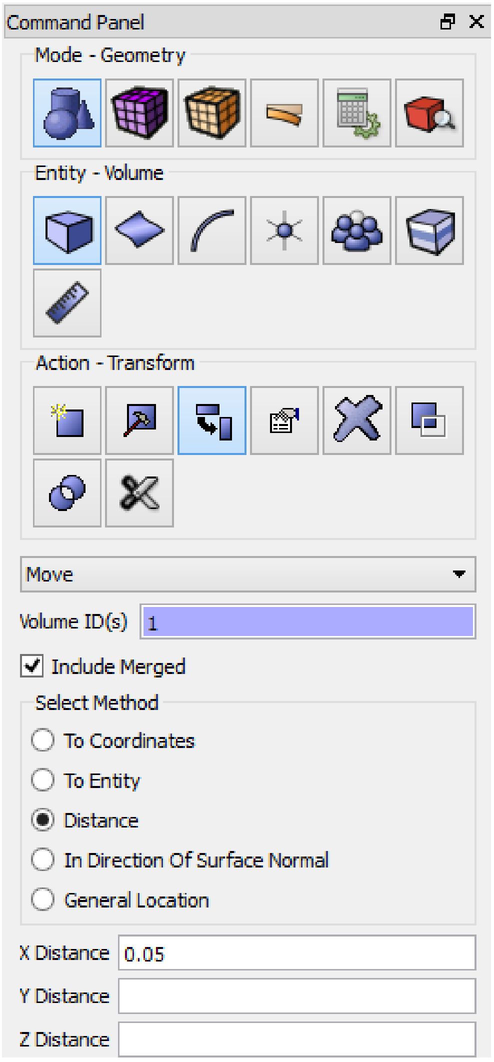
- Width: 0.1;
- Height: 0.01;
- Depth: 0.01.

Click **Apply**.



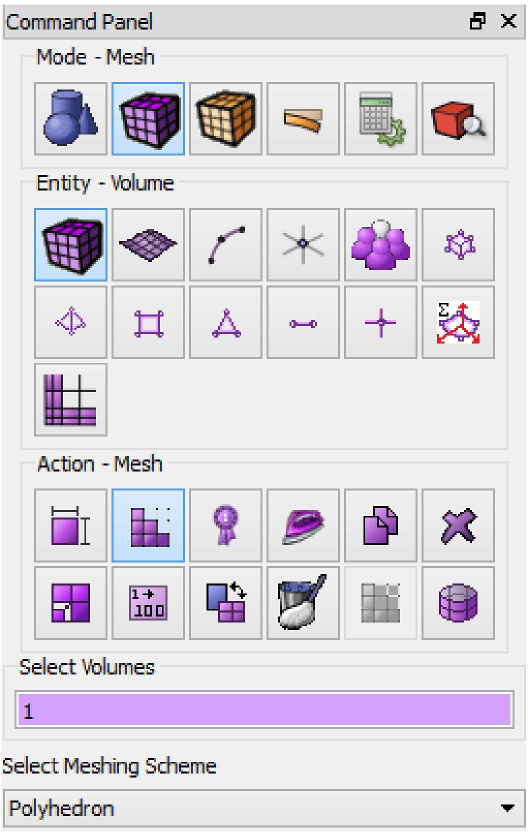
2. Combine left edge of the beam with the origin of coordinates.

Set the following parameters: Select volume geometry modification section on Command Panel (Mode — **Geometry**, Entity — **Surface**, Action — **Transform**). Select **Move** in the list of possible webcut types. Set the following parameters:



- Volume: 1;
- Select method: Distance;
- X Distance: 0.05.

Click **Apply**.



Meshing

1. Creat the mesh of hexahedrons.

Select volume mesh generation section on Command Panel (Mode — **Mesh,** Entity — **Volume**, Action — **Mesh**):

- Select Volumes (specify their ID): 1 *(or by the command **all**);*
- The way of meshing: Polyhedron.

Click **Apply Scheme**.

Click **Mesh**.

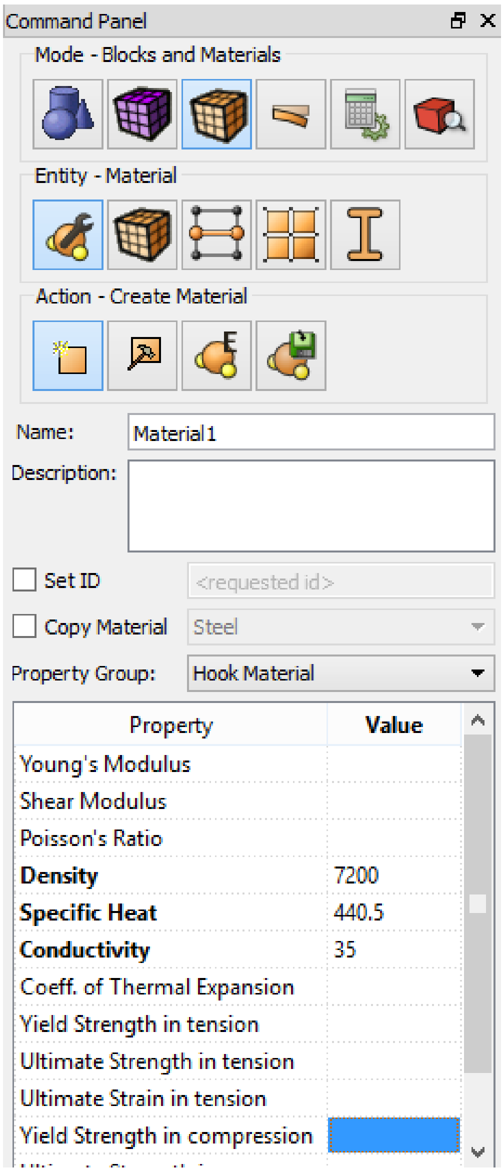
Setting material and element type

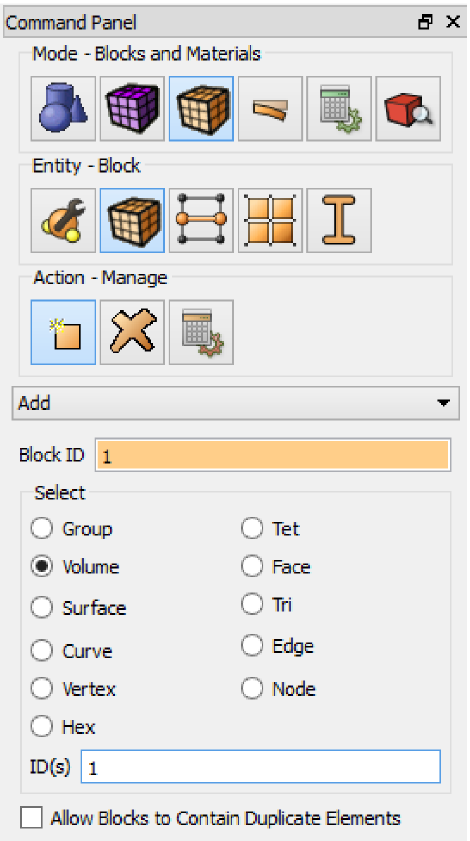
1. Create the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Material**, Action — **Create Material**). Enter the name for the material. Set the following parameters:

- Density: 7200;
- Specific Heat: 440.5;
- Conductivity: 35.

Click **Apply**.





2. Create a block of one type of the material.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: 1 *(*or by the command **all***)*.

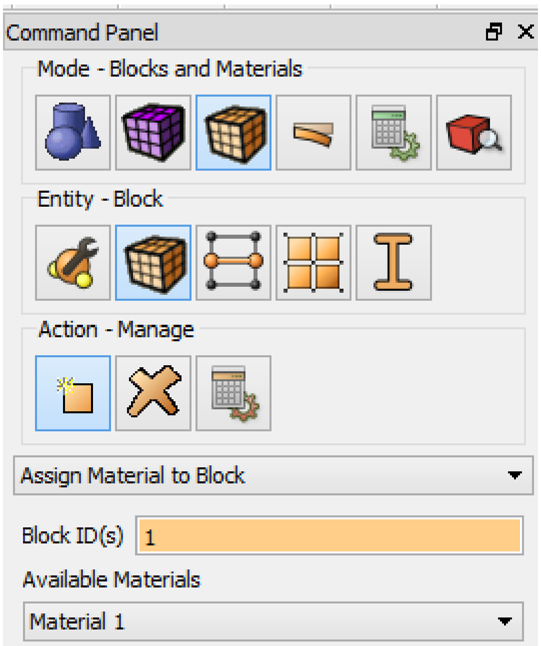
Click **Apply**.

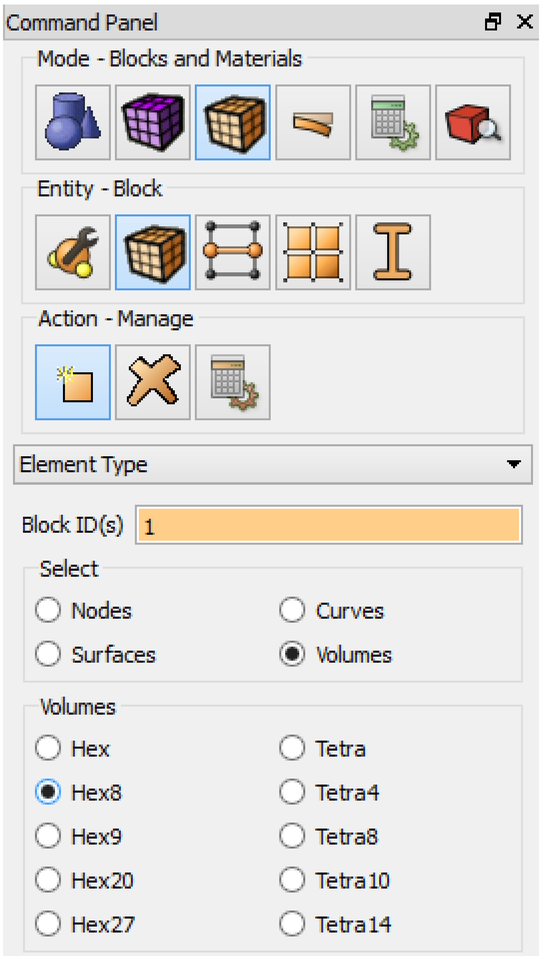
3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.





4. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Volumes;
- Volumes: Hex8.

Click **Apply**.

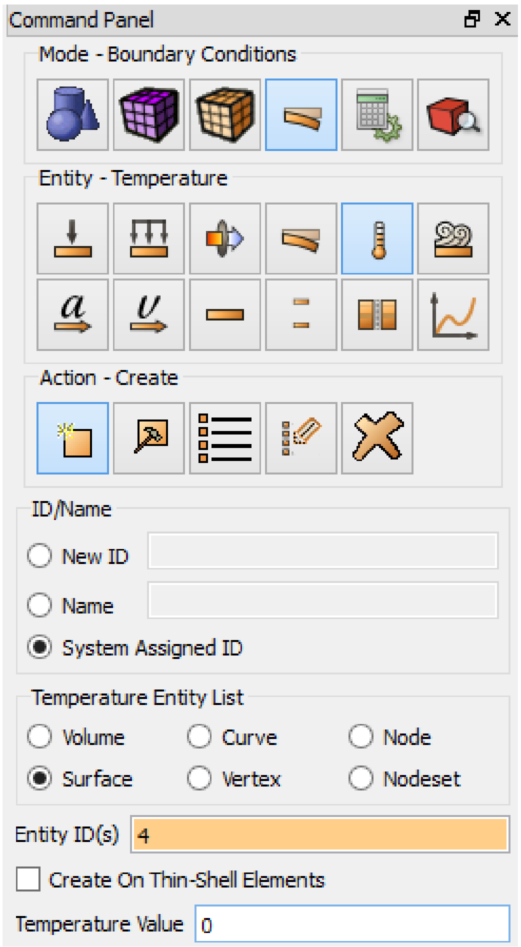
Setting boundary conditions

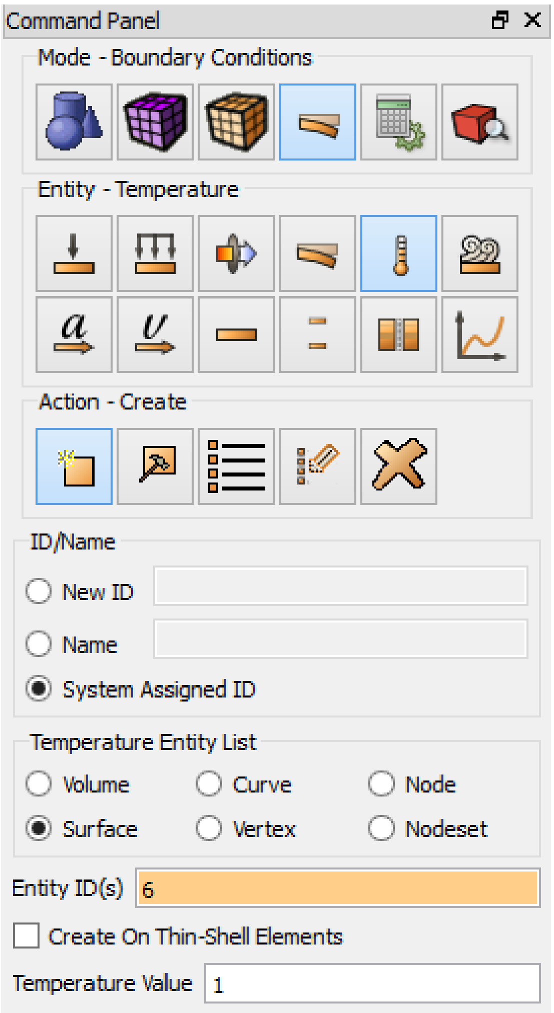
1. Set the value of temperature applied to the left side of the beam.

Select Mode – **Boundary Conditions,** Entity – **Temperature**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 4;
- Temperature Value: 0.

Click **Apply**.





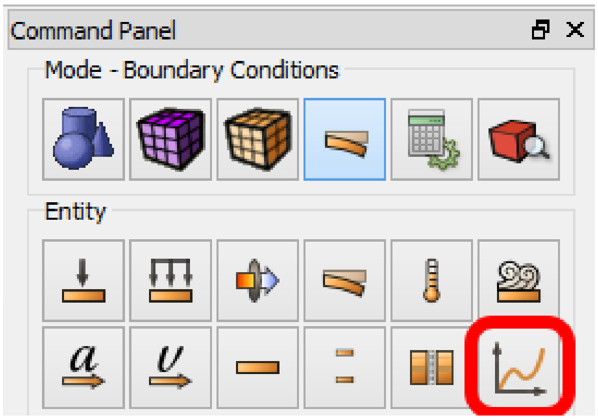
2. Set the value of temperature applied to the right side of the beam.

Select Mode – **Boundary Conditions,** Entity – **Temperature**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 6;
- Temperature Value: 1.

Click **Apply**.

Setting time dependency of boundary conditions

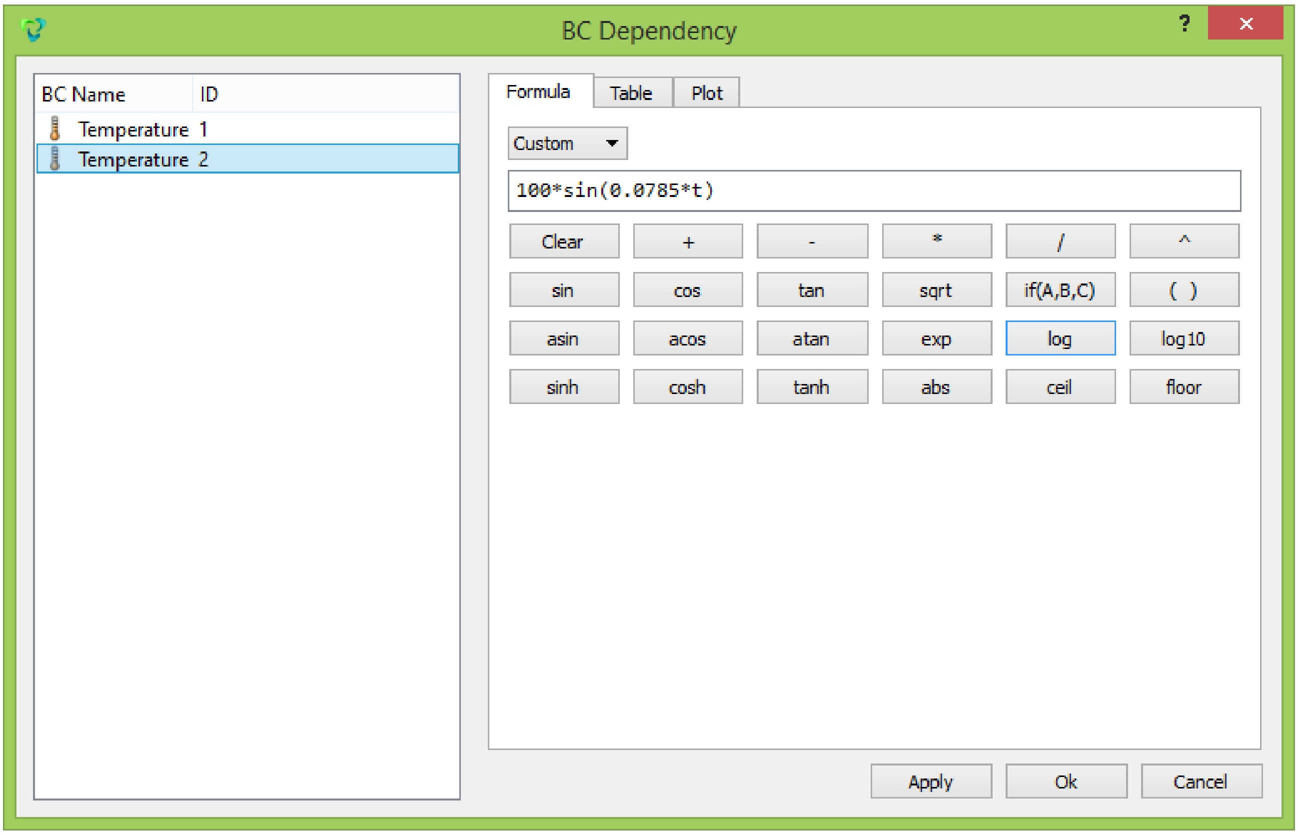


3. Set time dependency of the temperature applied to the right edge of the beam.

Select Mode – **Boundary Conditions**. Click on the button **Time dependency** on Command Panel. The pop-up menu with the settings will be opened. On the left panel, select BC for which the time dependency will be set: **Temperature 2**. Put a checkbox next to the item **Formula**. Set the following parameters:

- Time dependency type: Manually;
- Enter formula: 100*sin(0.0785*t).

Click **Ok**.



Starting calculation

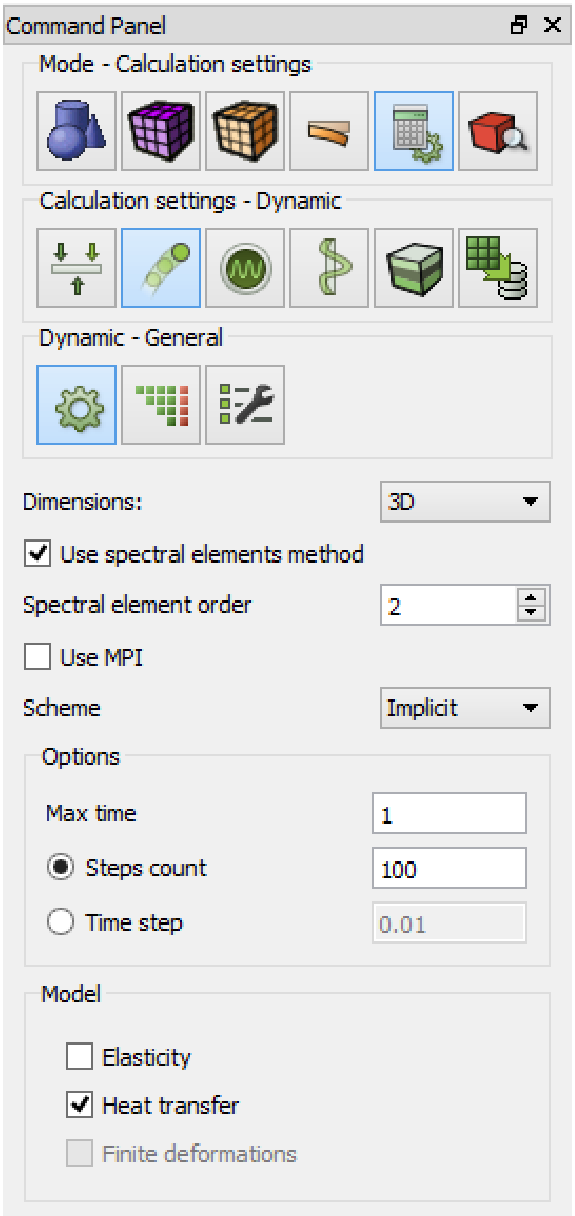
4. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Dynamic**, Dynamic – **General**). Set the following calculation parameters:

- Dimension: 3D;
- Spectral element method;
- Spectral element order: 2;
- Elasticity: untick;
- Heat transfer: tick;
- Scheme: Implicit;
- Max time: 32;
- Steps count: 100.

Click **Apply.**

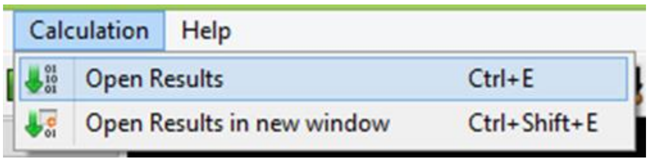
Click **Start Calculation**.



5. In a pop-up window select a folder to save the result and enter the file name.
6. If the calculation is finished successfully, you will see a message in the Console: “*Calculation finished successfully at <date>* <time>”.

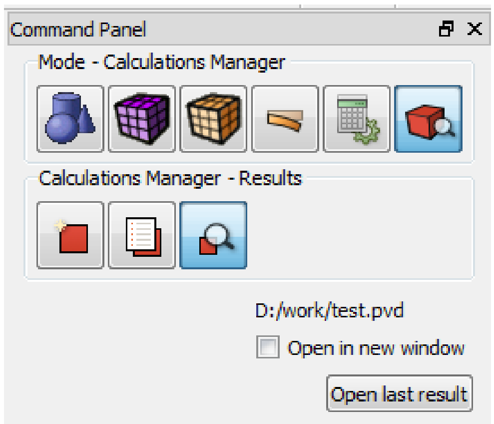
Results analysis

1. Open the file with the results.



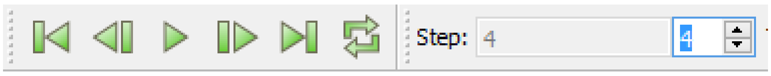
You can do this in one of the three ways:

- Click Ctrl+E.
- Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
- Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



You can see the calculation results in the pop-up **Fidesys Viewer** window.

2. There is a menu on Toolbar which allows viewing animation. It consists of a cycle of solutions calculated for every moment of time. Click “Last Frame” to see the model in time moment t = 32°C.



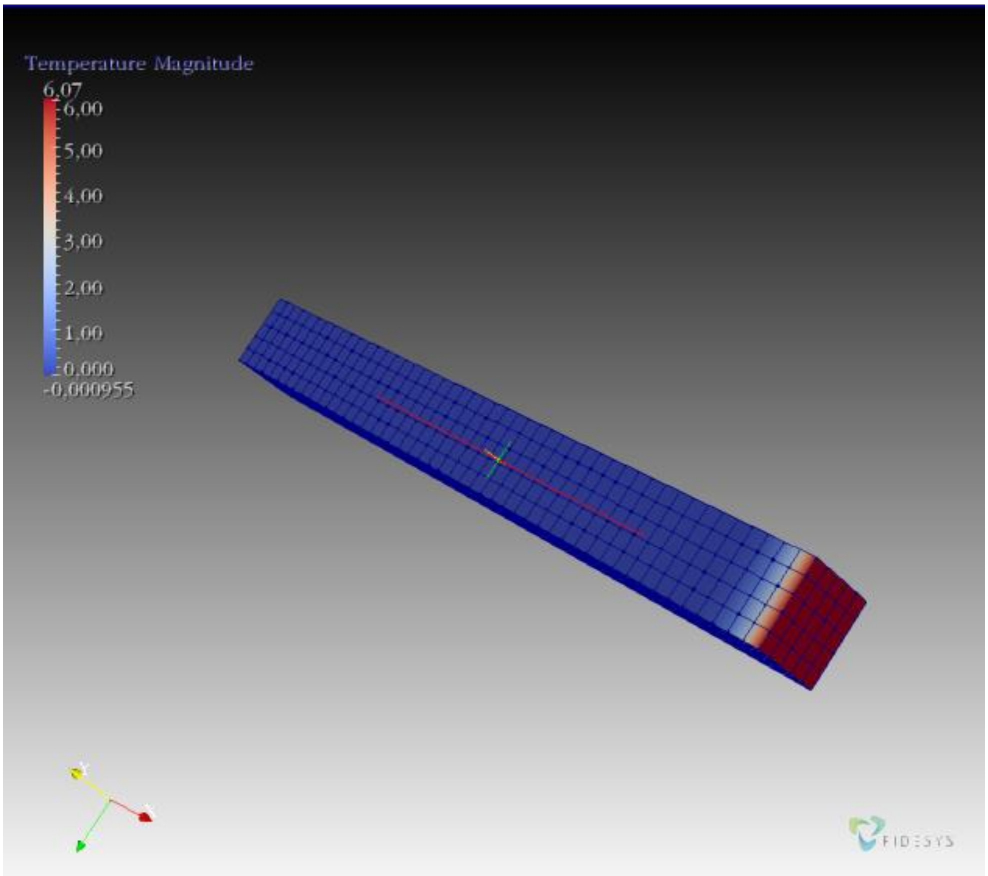
3. Display the component of the temperature.

In Fidesys Viewer window set the following parameters on Toolbar:

- Representation Field: Temperature;
- Representation Mode: Surface With Edges.

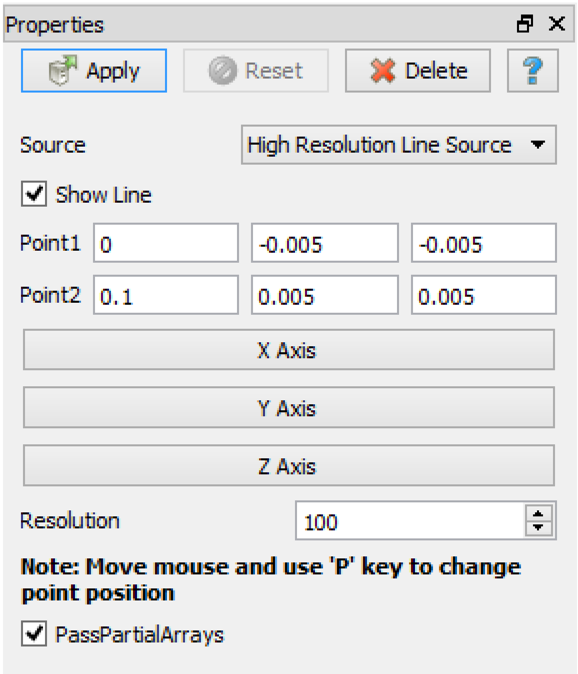
The model displays the mesh resulting from application of the spectral element method and the field of temperature distribution.

To display the color legend scale, click the button **Switch the color legend visibility** on Command Panel.



4. To graph along one of the beam edges.

Select the filter **Plot Over Line** in the Main Menu. Set the coordinates of the points defining the line In the tab **Properties**:



- Source: High Resolution Line;
- Show Line;
- Point 1 (coordinates): 0 -0.005 0.005;
- Point 2 (coordinates): 0.1 -0.005 0.005;
- Resolution: 100;
- PassPartialArrays

Click **Apply**.

 To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

Click on the graph window appeared on the right side of the screen.

5. Display temperature change on the graph.

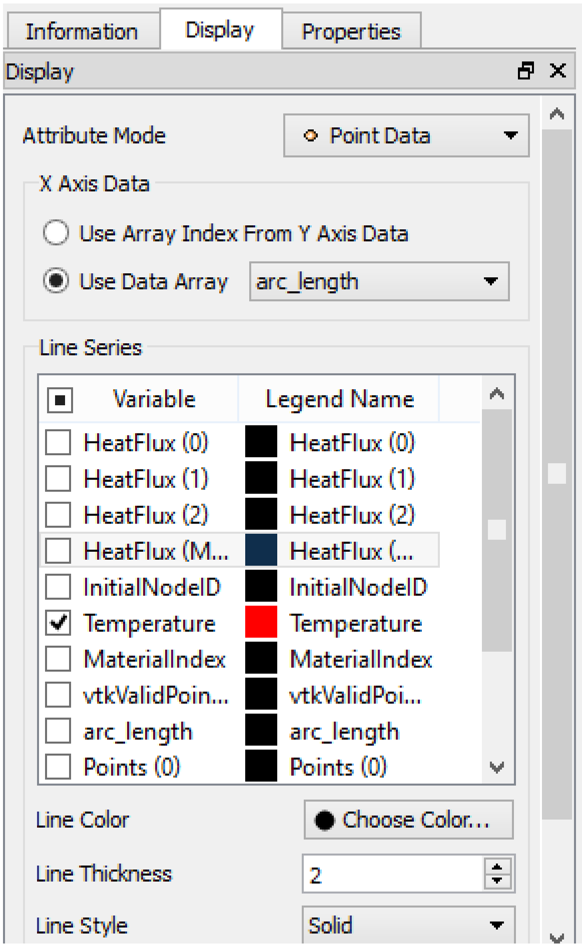
Click on the graph window, go to the tab "Display" in the filter control panel.

Set the Attribute Mode – Point Data

Next, in the field “Line Series”, set up labels against the parameters that you want to display on the graph.

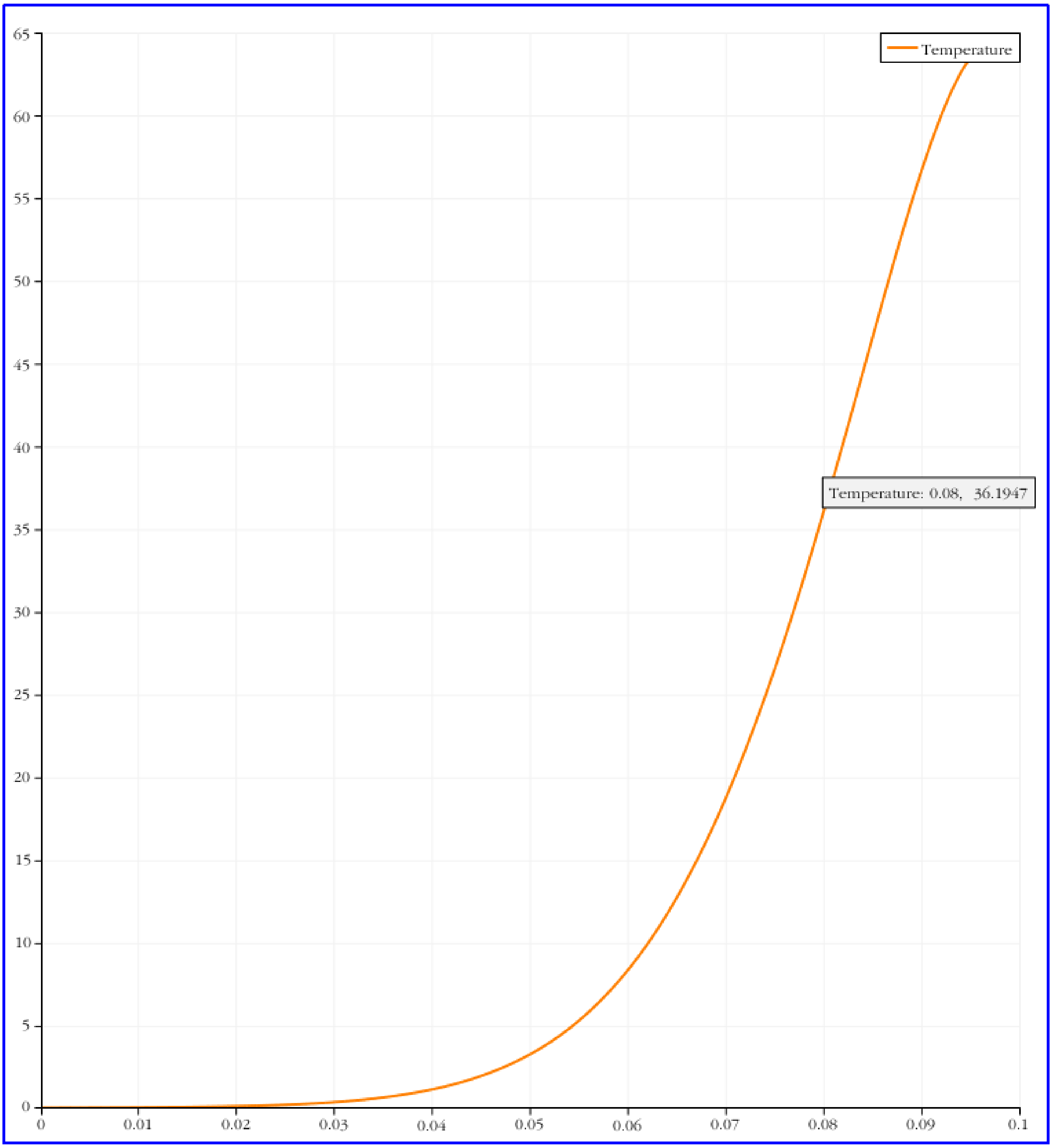
Untick all the options except Temperature.

The temperature dependency at points belonging to the beam edge and the coordinates of these point coordinates are displayed on the graph.



6. Check the numerical temperature value T at the point (0.08;0;0).

Move the cursor to the required point on the graph. You can see a tool tip with the temperature value.



The difference between the obtained value 36.1947 and the required one 36.60 is 1.11%.

7. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd):

```
reset
brick x 0.1 y 0.01 z 0.01
move volume 1 x 0.05
volume all size auto factor 5
mesh volume all
undo group begin
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "DENSITY" 7200 "SPECIFIC_HEAT" 440.5
"CONDUCTIVITY" 35
undo group end
set duplicate block elements off
block 1 volume 1
block 1  material 'Material 1'
block 1 element type HEX8
create temperature  on surface 4  value 0
create temperature  on surface 6  value 1
bcdep temperature 2 value '100*sin(0.0785*t)'
analysis type dynamic heattrans dim3
dynamic scheme implicit maxtime 32 maxsteps 100
spectralelements on order 2
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

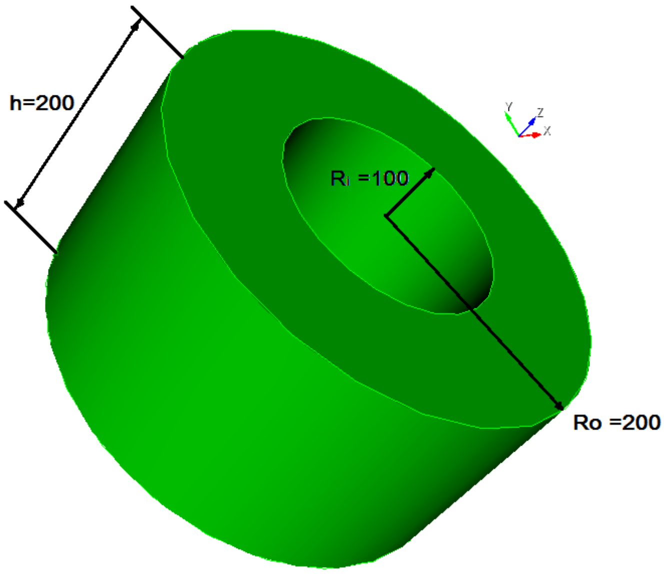
It is also possible to run the file *Example_14_Dynamic_3D_Conduction_Spectr.jou*, by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

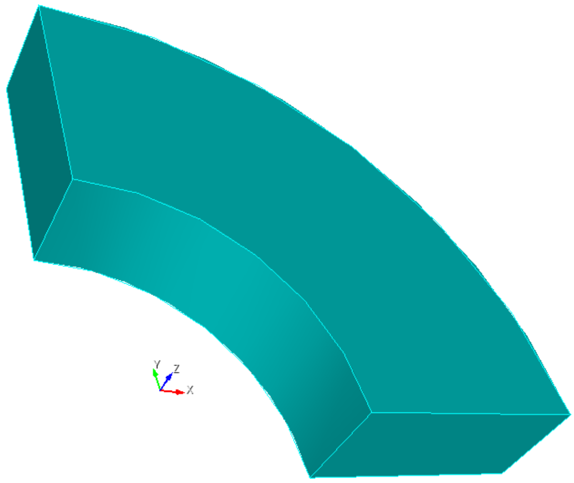
Static load considering plasticity (3D)

NAFEMS Understanding Non-linear Finite Element Analysis Through Illustrative Benchmarks, Pressured cylinder plasticity benchmark p.51.

The problem of static load of a cylinder with perfect plasticity is being solved.

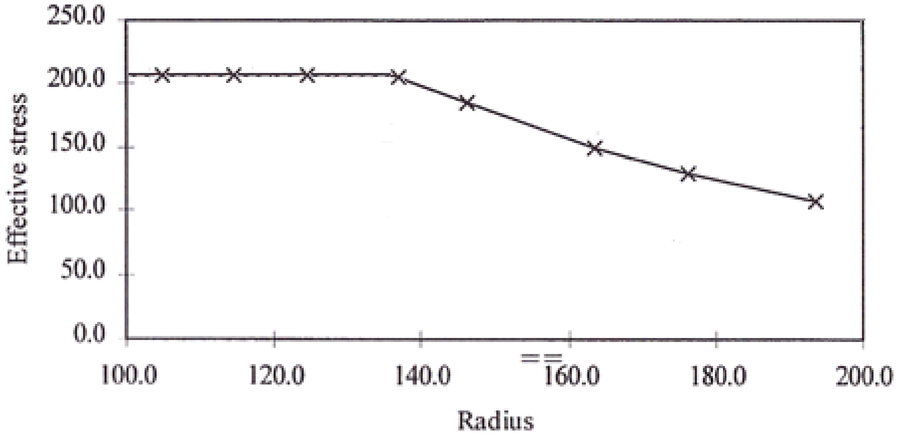
The pictures below represent a geometric model of the problem:





Due to the symmetry, 1/8 part of the cylinder is considered. The model is also fixed on the symmetry condition. The pressure of 140 Pa is applied to the inner edge. The material parameters are Young's Modulus E = 207000 Pa, Poisson's Ratio v = 0.3, yield strength σ_y=208. In the process of solving, the load is split into 14 steps of loading.

Test pass criterion is the following: stress graphs σ_{Mises} for P=140 coincide with the shown in the picture:



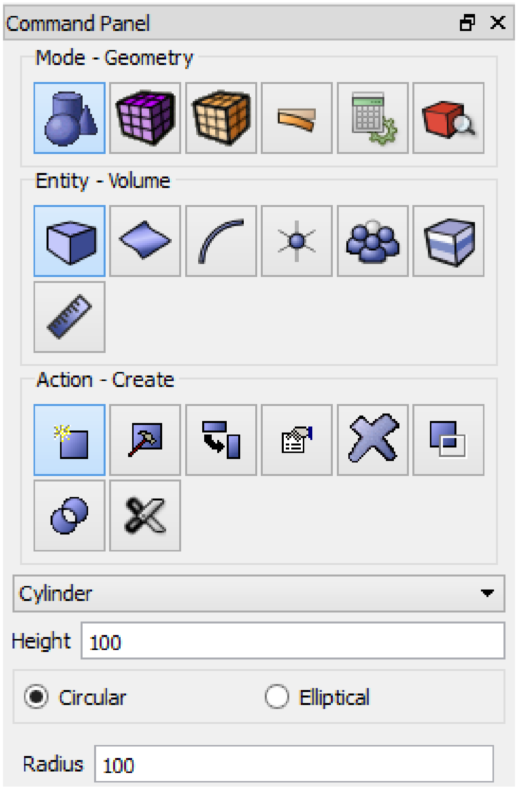
Geometry creation

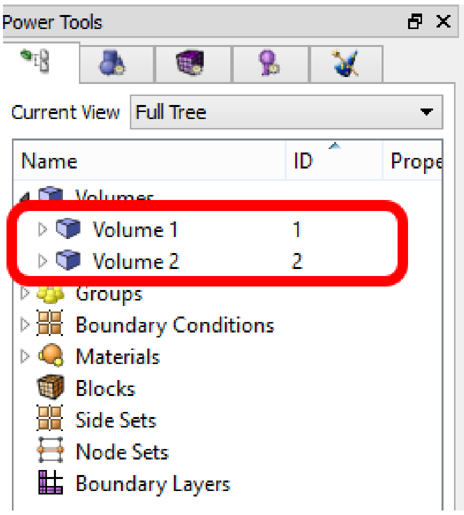
1. Create the first cylinder.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 100;
- Cross section: Circular;
- Radius: 100;

Click **Apply**.





2. Create the second cylinder.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Create**). Select **Cylinder** in the list of geometric elements. Specify the cylinder dimensions:

- Height: 100;
- Cross section: Circular;
- Radius: 200;

Click **Apply**.

As a result, two generated entities are displayed in the Model Tree (Volume 1 and Volume 2).

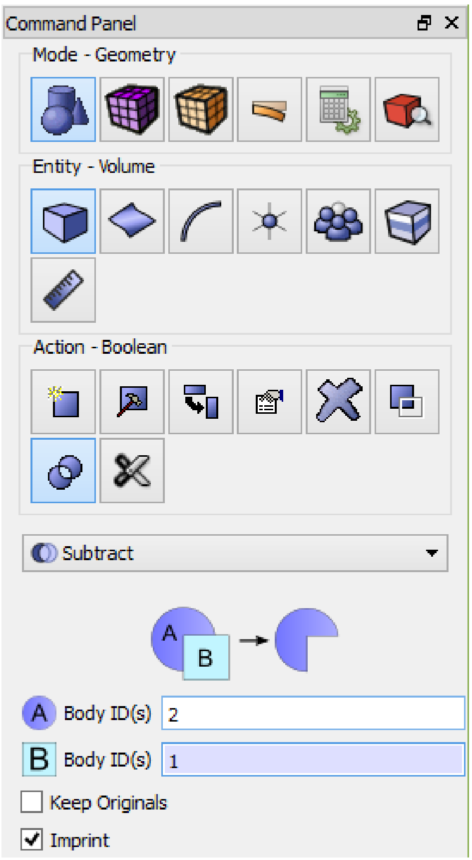
3. Subtract the first cylinder from the second one.

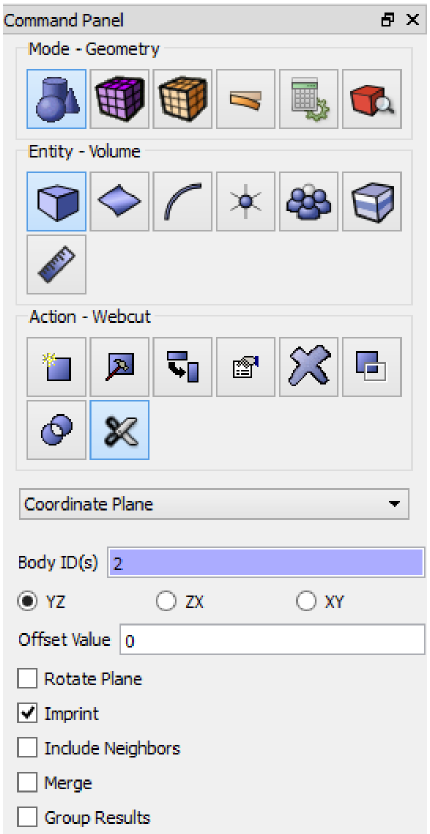
Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Boolean**). Select **Subtract** in the list of operations. Set the following parameters:

- Body ID: 2 *(volumes from which other volumes will be subtracted)*;
- Substract Body ID(s): 1 *(the volumes to be subtracted)*;
- Imprint.

Click **Apply**.

As a result, only one volume is displayed in the Model Tree (Volume 2).





4. Leave a quarter of a volume (symmetry of the problem).

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

- Volume ID(s): 2 (*the volume to be webcut)*;
- Webcut with: YZ Plane;
- Offset value: 0;
- Imprint.

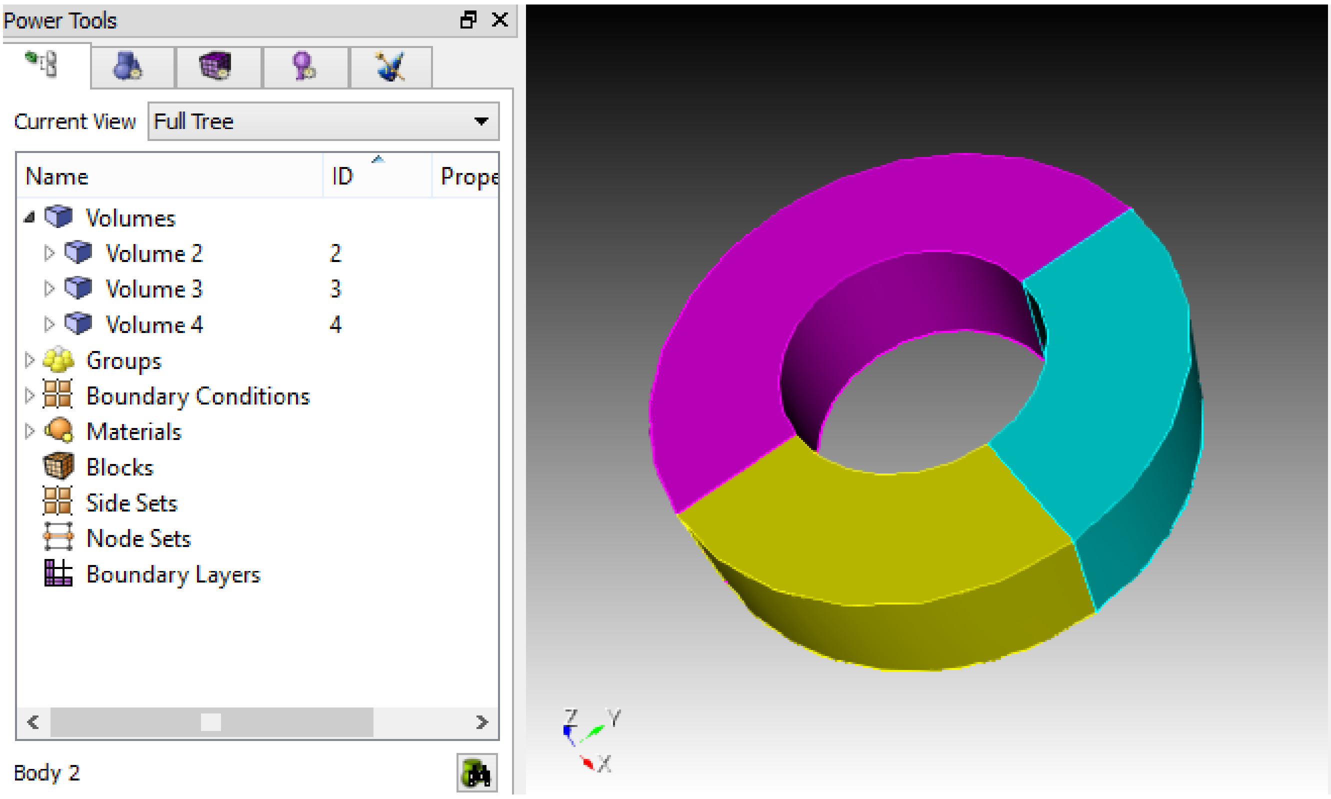
Click **Apply**.

Do the same for the XZ Plane:

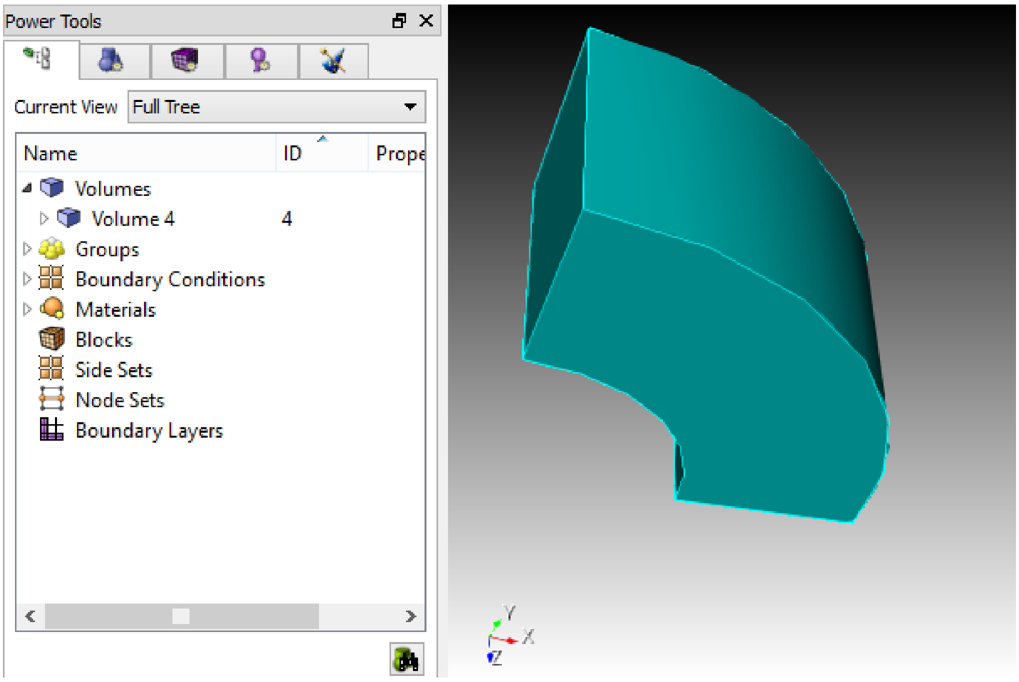
- Volume ID(s): 2 *(the volume to be webcut);*
- Webcut with: ZXPlane;
- Offset value: 0;
- Imprint.

Click **Apply**.

As a result, the original volume in the Model Tree is split into three (Volume 2, Volume 3 and Volume 4).



Delete the volumes 2 and 3. To do this, select these volumes in the Model Tree holding down Ctrl and click **Delete** in contextual menu. As a result, a quarter of the original volume is left (Volume 4):

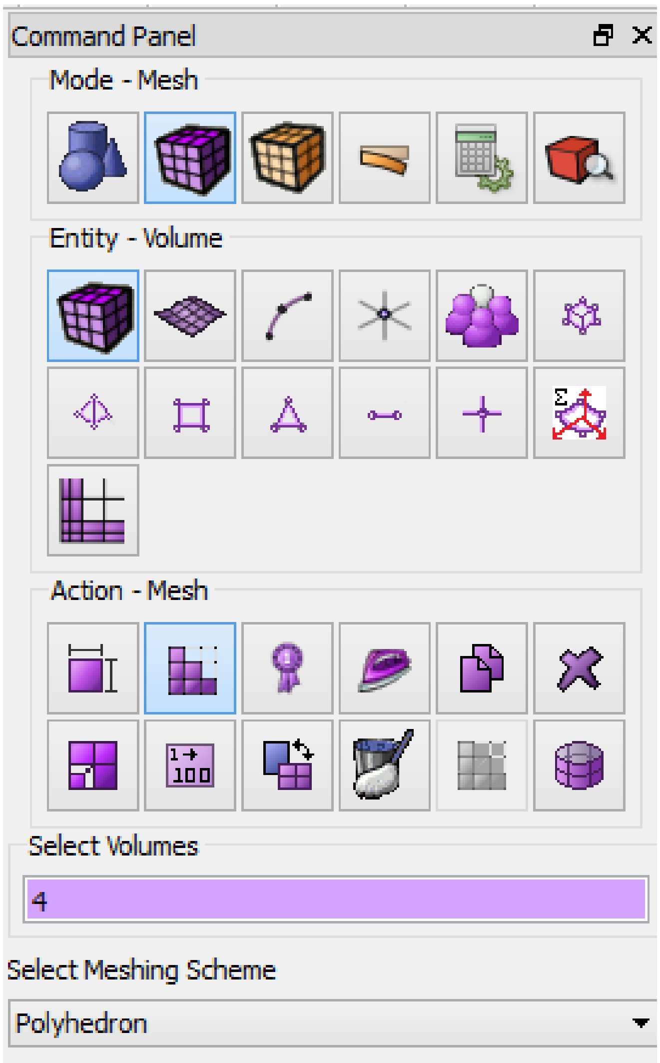


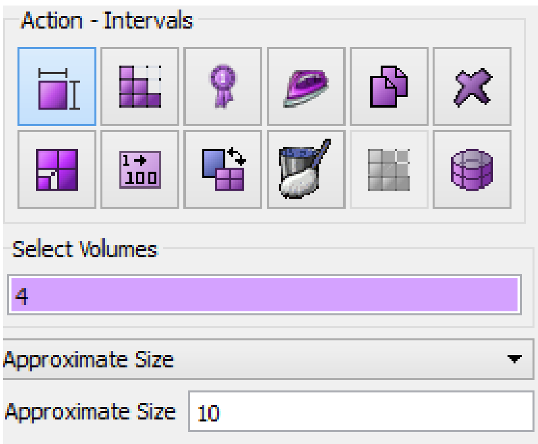
Meshing

1. Select volume mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Volume**, Action – **Meshing**). Specify the following parameters:

- Select volumes: 4;
- Select meshing scheme: Polyhedron;

Click **Apply Scheme**.





2. Go to the section Intervals on Command Panel (Mode – **Mesh,** Entity – **Volume**, Action – **Intervals**). Specify the following parameters:

- Select volumes: 4;
- Select meshing scheme: Approximate size;
- Approximate size: 10

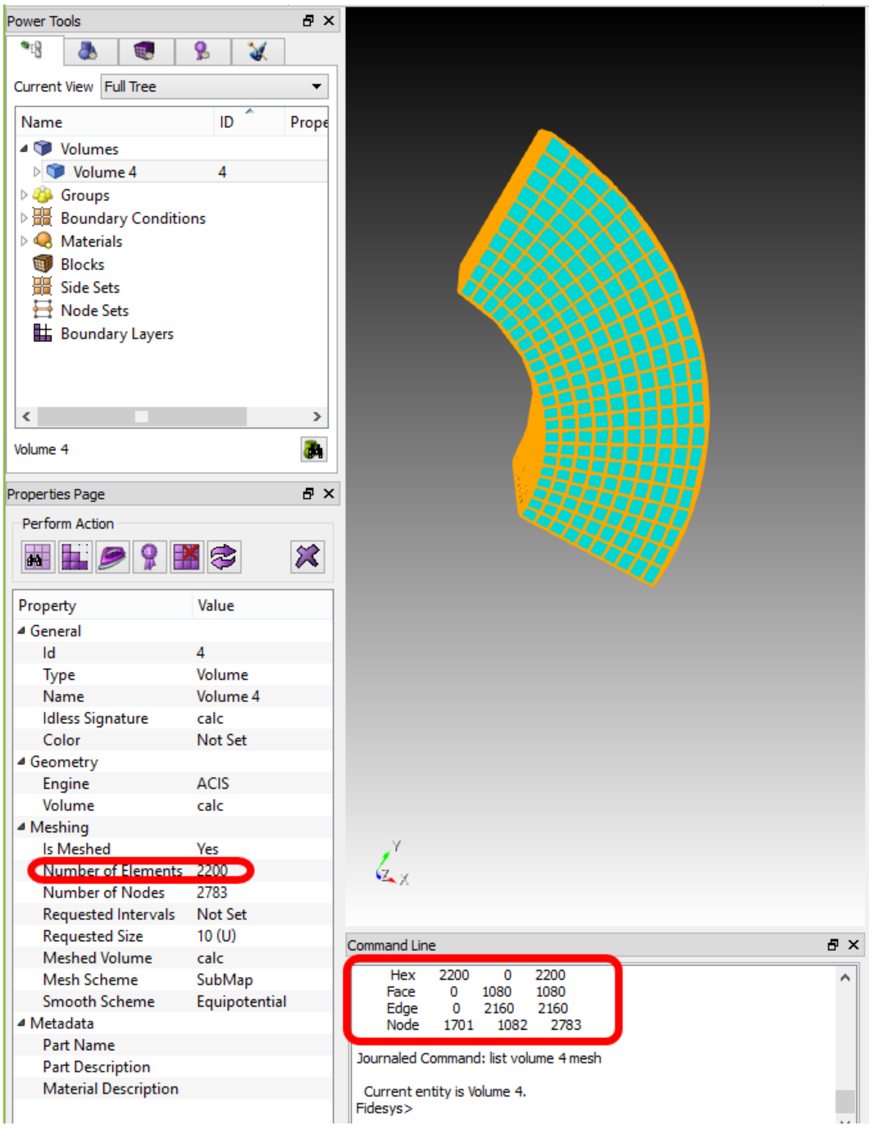
Click **Apply**.

Click **Mesh**.

The resulting number of elements can be viewed in the Property Page by clicking on the inscription Volume 4 in the Model Tree on the left.

To view the mesh properties, you can follow these steps:

- Select the entire model
- Right-click on the model
- In the pop-up menu, select List Information – List Mesh Info
- Information on the mesh will be displayed in Command Line



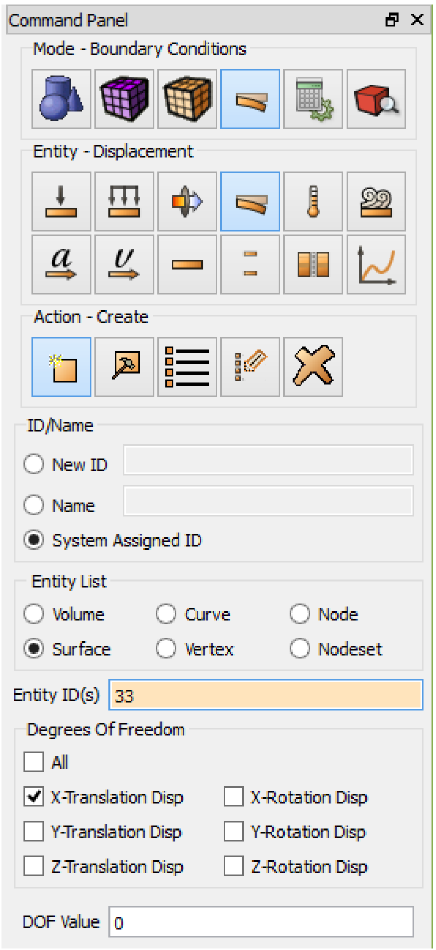
Setting boundary conditions

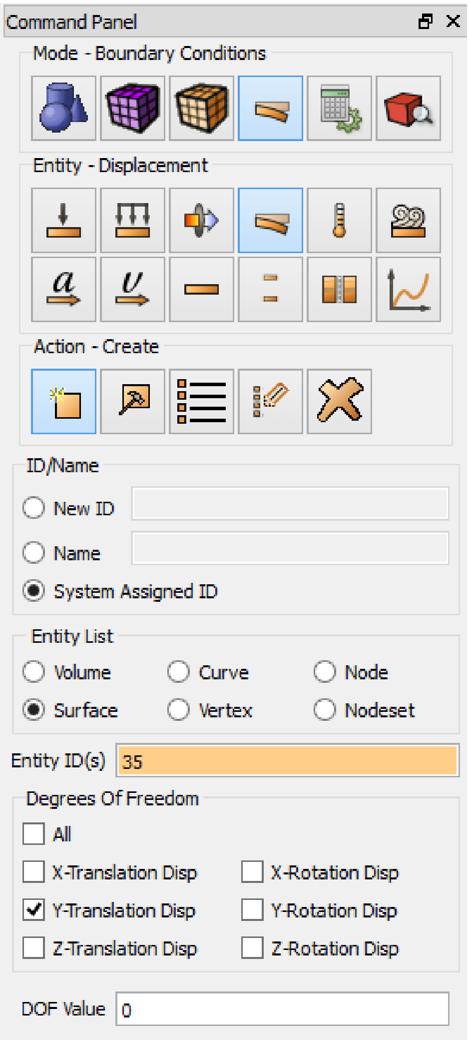
1. Fix one side (slice) along X axis.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel**.** Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 11 *(or click on the left slice)*;
- Degrees of Freedom: X-Component;
- DOF Value: 0.

Click **Apply**.





2. Fix one side (slice) along Y axis by analogy.

Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 27 *(or click on the bottom slice)*;
- Degrees of Freedom: Y-Translation;
- DOF Value: 0.

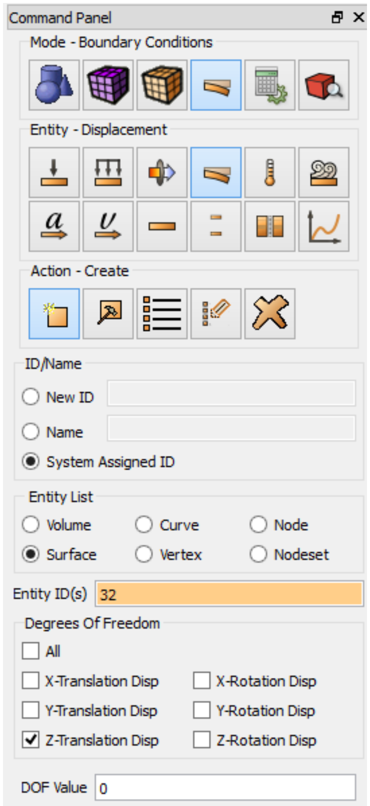
Click **Apply**.

3. Fix the bottom surface along Z axis by analogy.

   Select Mode — **Boundary Conditions** on Command Panel. Set the following parameters:

   - System Assigned ID;
   - Entity List: Surface;
   - Entity ID(s): 29 *(or clicking on the bottom edge of the model)*;
   - Degrees of Freedom: Z-Translation;
   - DOF Value: 0.

Click **Apply**.

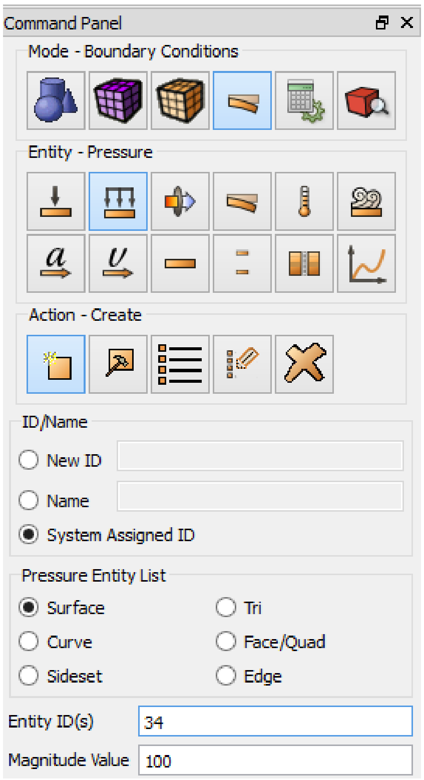


4. Apply pressure to the inner edge.

Select Mode — **Boundary Conditions,** Entity — **Pressure**, Action — **Create** on Command Panel**.** Set the following parameters:

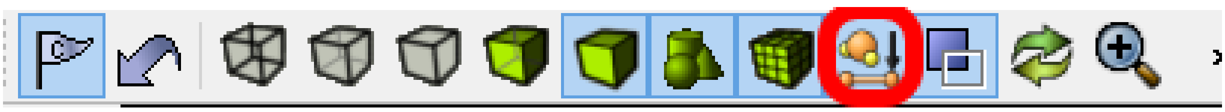
- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 30 *(or click on the inner surface of the cylinder)*;
- Value: 160

Click **Apply**.



All the applied boundary conditions must be displayed in the Model Tree on the left. Besides, the boundary conditions are available for editing from the Model Tree.

To view all the applied boundary conditions, click Show BC in the top panel.



Setting material and element type

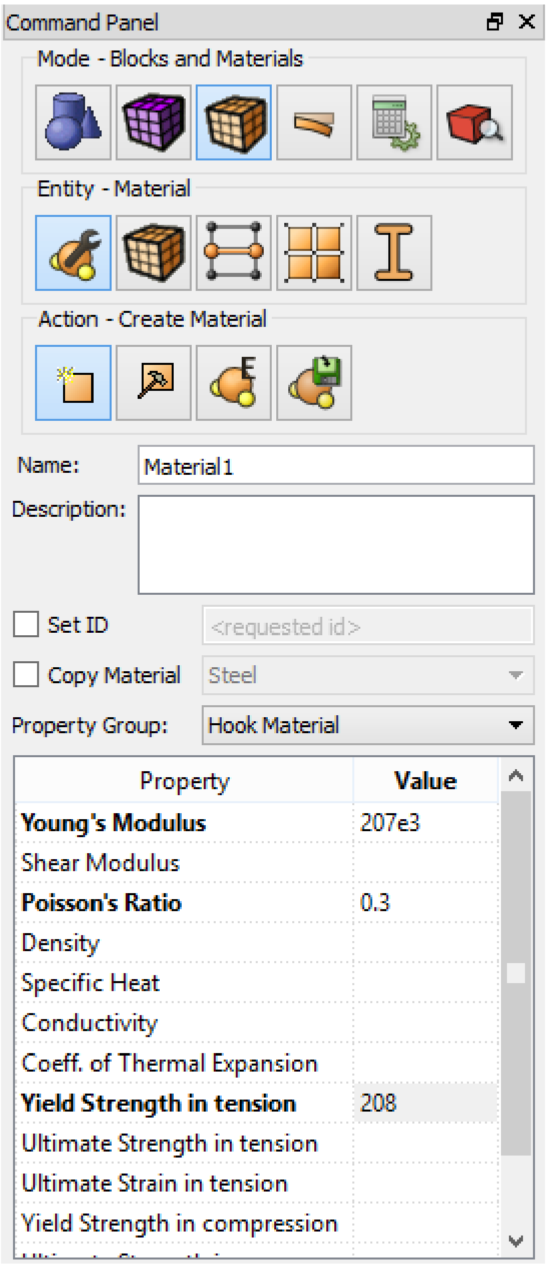
1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Material**, Action – **Create Material**). Set the following parameters:

- Name: Material1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 207e3;
- Poisson's Ratio: 0.3;
- Yield strength: 208.

Click **Apply**.

Note: Since the case of perfect plasticity is regarded, the yield strength and ultimate strain are not introduced.

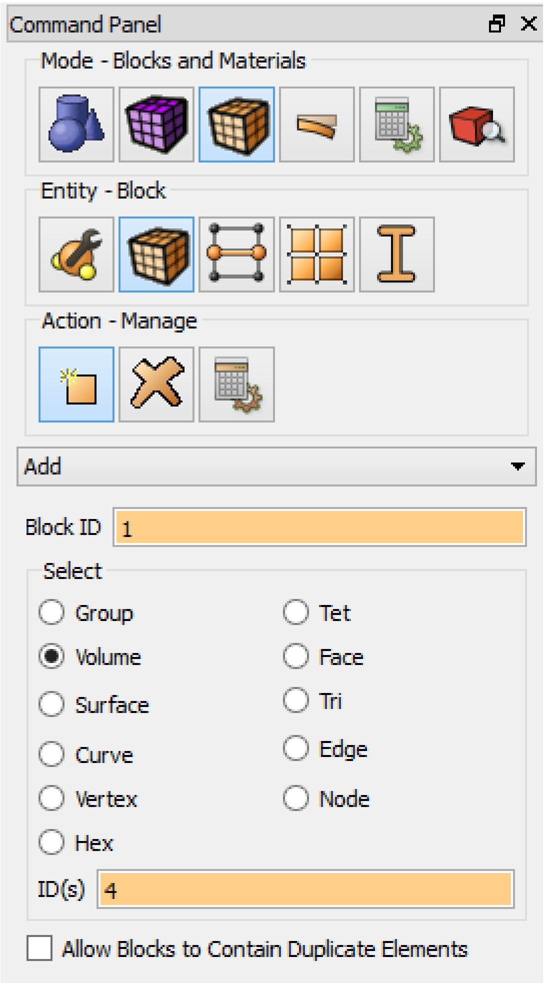


2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

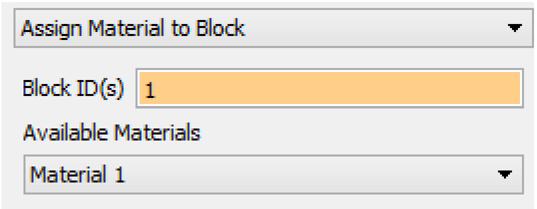
- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: 4 *(or by the command **all**)*.

Click **Apply**.



3. Assign the material to the block.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

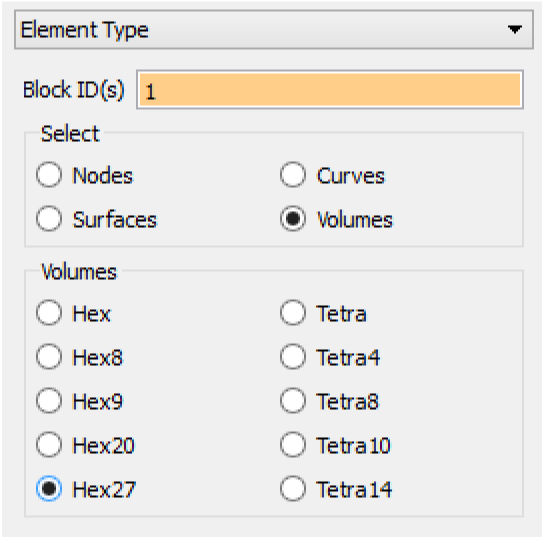


- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.

4. Assign the element type.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:



- Block(s) ID: 1;
- Select: Volumes;
- Volumes: HEX27.

Click **Apply**.

Starting calculation

1. Set the type of the problem to be solved.

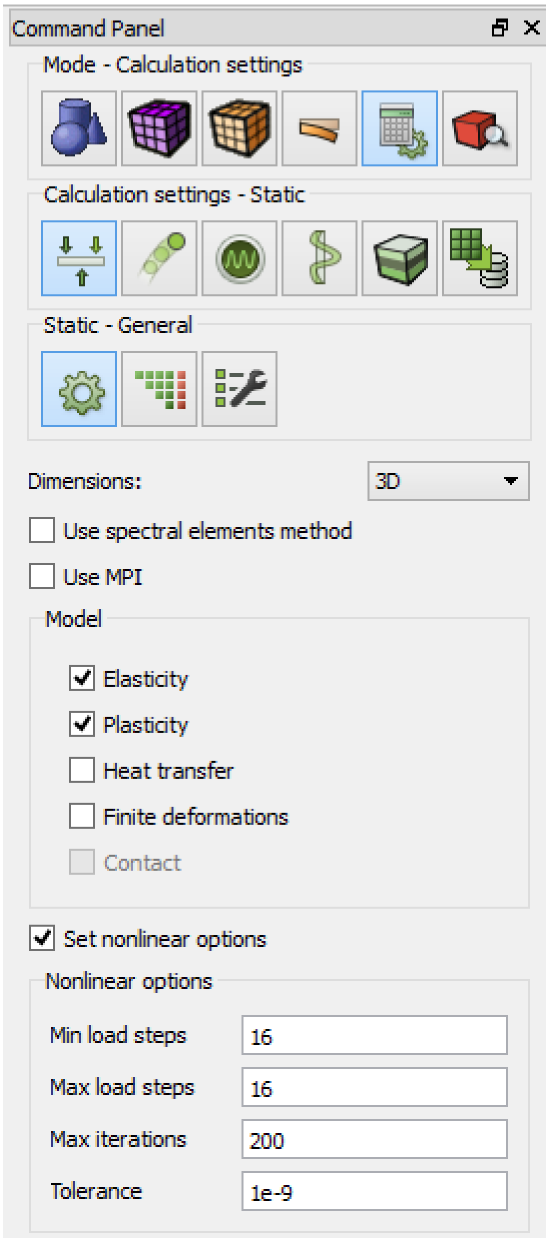
Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select:

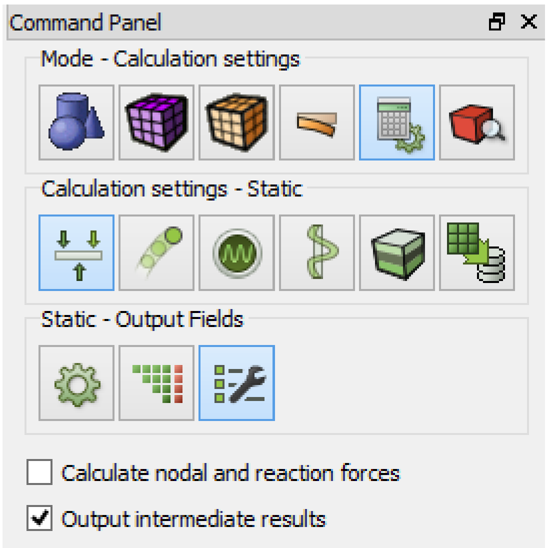
- Dimension: 3D;
- Model: Elasticity;
- Model: Plasticity.
- Nonlinear solver settings. Specify the following parameters:
- Maximum number of iterations: 200;
- The number of loading steps: 16;
- Tolerance: 1e-9.

Click **Apply**.

2. Set up Output Fields.

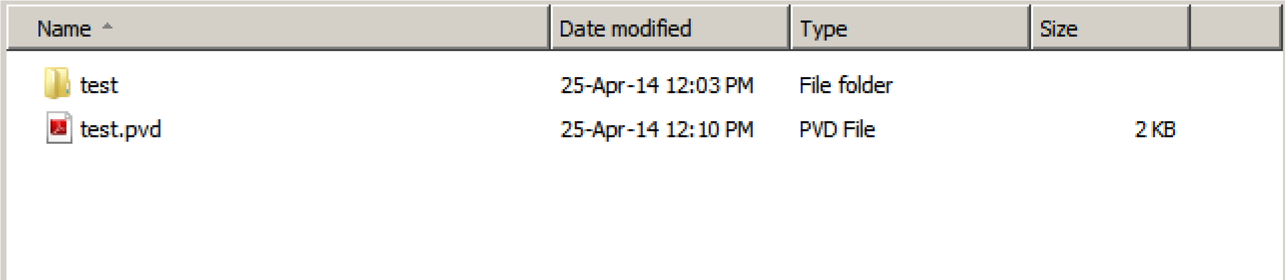
Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **Output Fields**). For nonlinear problems, results output is available at each loading step. Make sure that the checkbox Output intermediate results is on.





Click **Start Calculation**.

3. In a pop-up window select a folder to save the result and enter the file name. In this case, save the calculation in the file test.pvd.
4. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.
5. For nonlinear problems, convergence of iterations at each step of loading can be checked in the file Convergence.txt. The file is downloaded into the folder *test* which is created next to the file test.pvd.

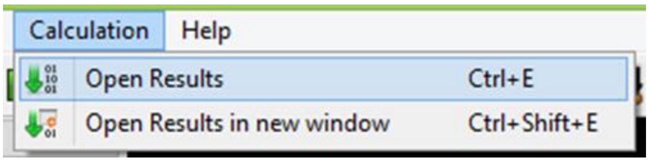


Open the file test\Convergence.txt:

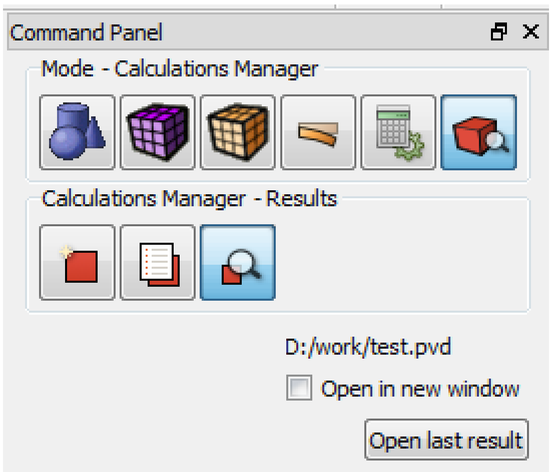
Make sure that the convergence with the set tolerance is reached at each step of loading.

```
           CONVERGENCE ITERATIONS

             LoadTime   Residual
      0.071429   50700.241485535989000
        0.071429   0.000000000915691
        -----------------------------
      0,142857   50700,241485575054000
        0,142857   0,000000001017351
        -----------------------------
      0,214286   50700,241485579267000
        0,214286   0,000000001169061
        -----------------------------
      0,285714   50700,241485585524000
        0,285714   0,000000001352515
        -----------------------------
      0,357143   50700,241485593193000
        0,357143   0,000000001542185
        -----------------------------
      0,428571   50700,241485601844000
        0,428571   0,000000001740132
        -----------------------------
      0,500000   50700,241485610350000
        0,500000   0,000000001895954
```

Results analysis



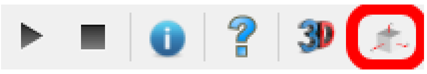
1. Open the file with the results. You can do this in one of the three ways.
   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (Mode – **Calculations Manager**, Calculations Manager – **Results**). Click **Open Results**.



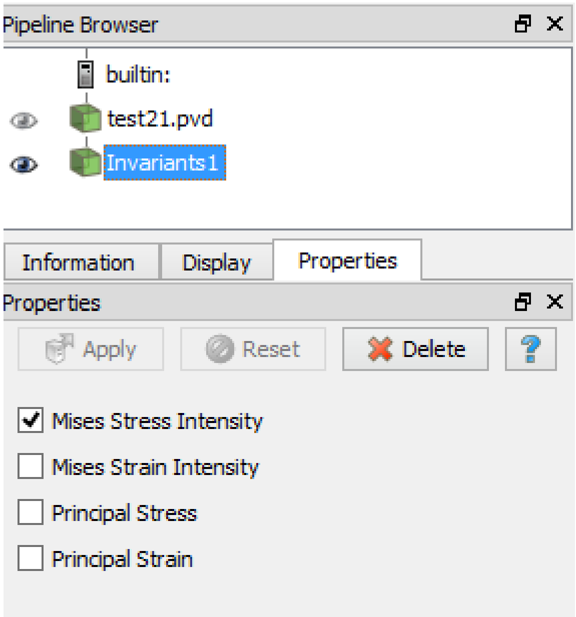
 To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

2. Display the Stress field σ_{Mises} and the mesh on the model.

To do this, invoke the filter Invariants by the quick access button on the Fidesys Viewer panel

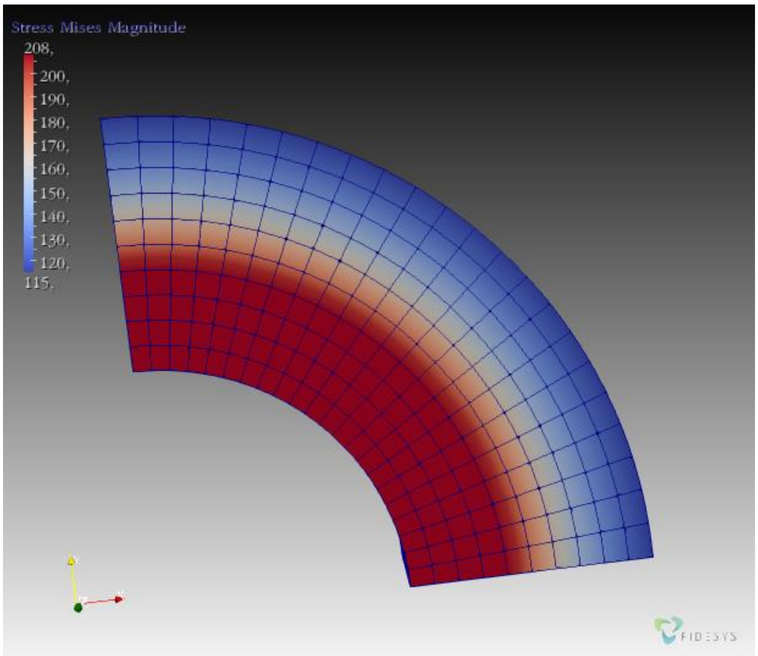


Leave only the checkbox Stress intensity for this filter in the tab Properties in the Model Tree.



On the ***Fidesys Viewer*** toolbar, set the following parameters:

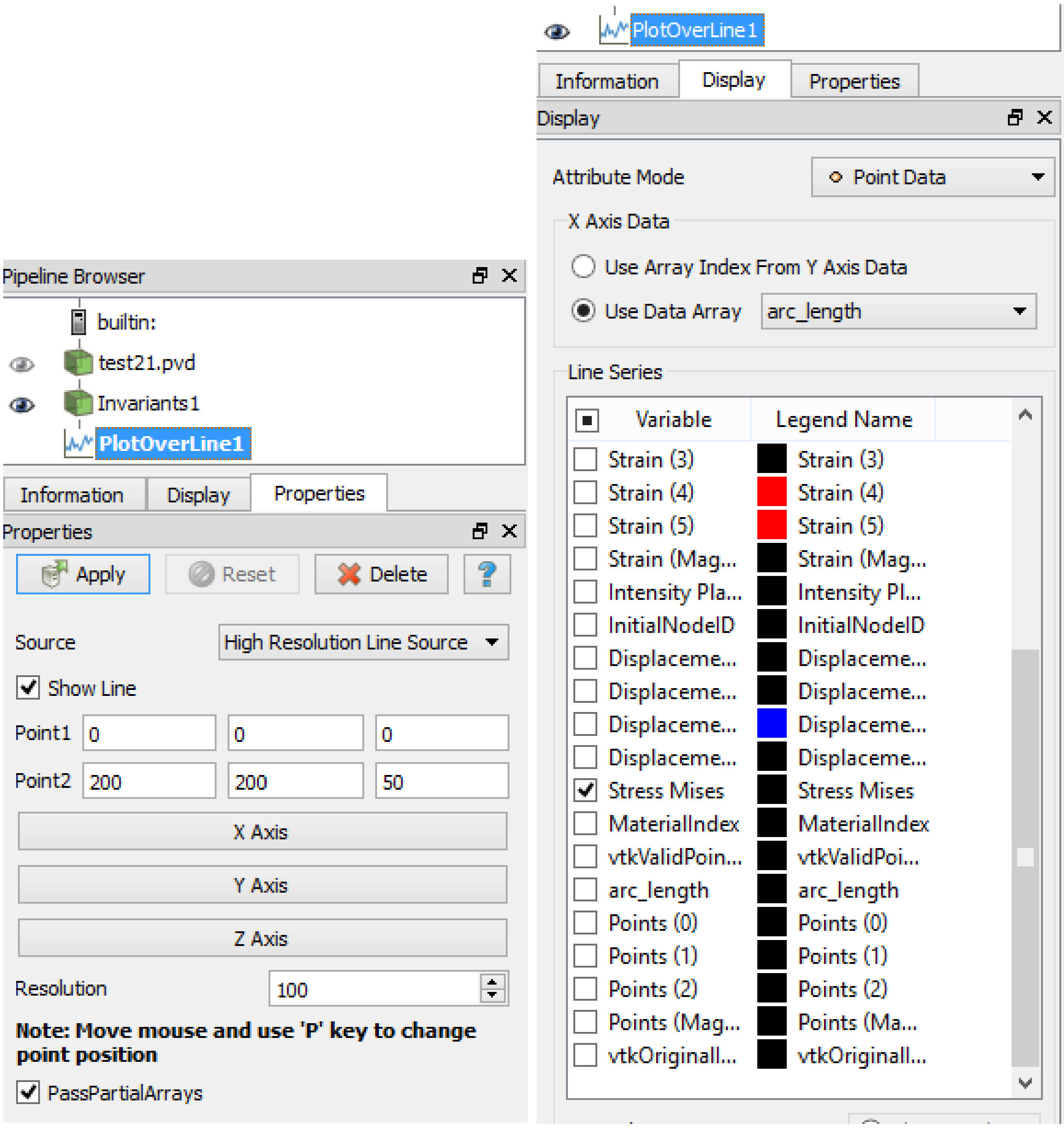
- Representation Mode: Surface With Edges;
- Representation Field: Stress (Mises);
- Surface With Edges.



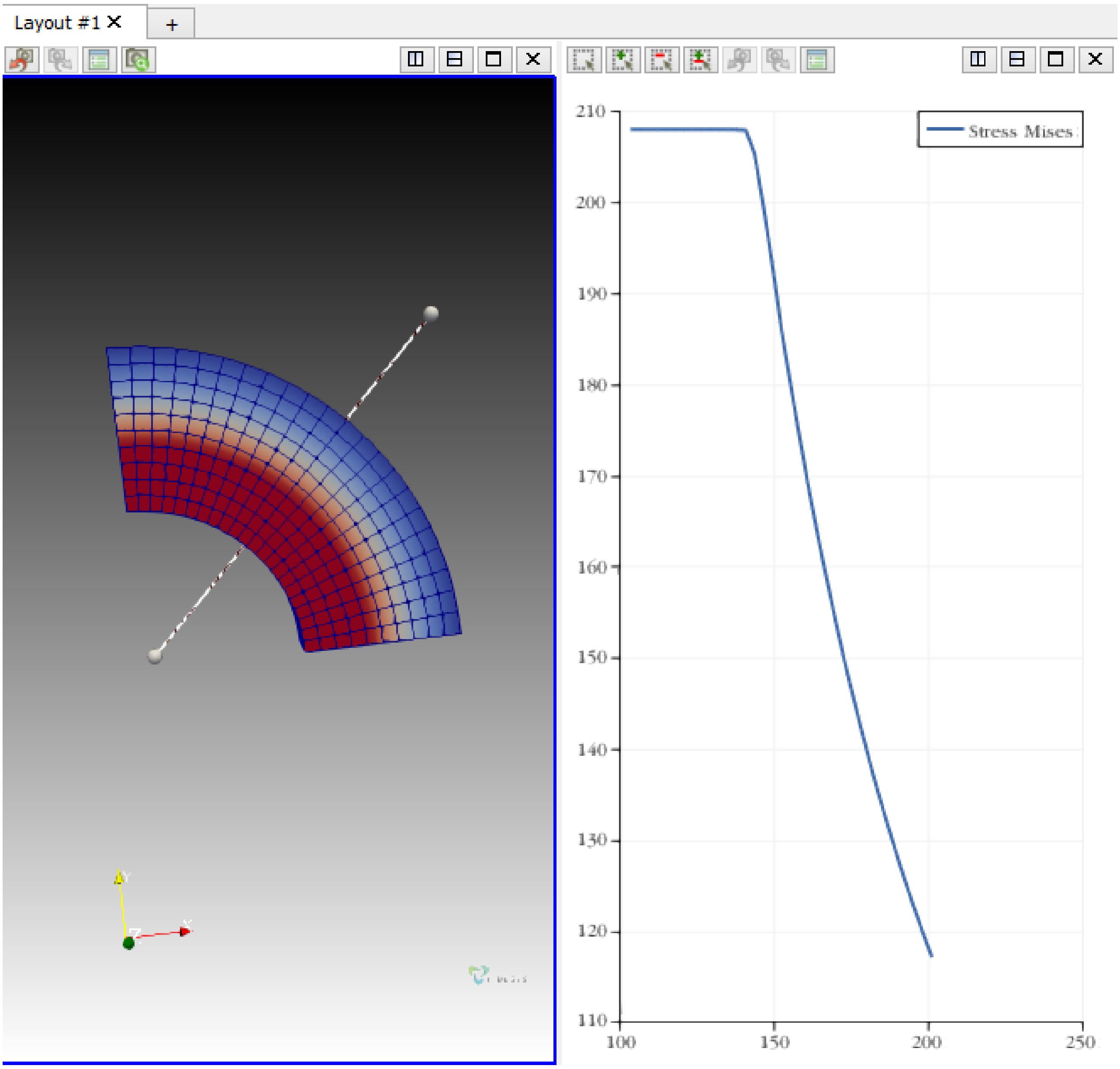
3. Build the Stress distribution (Mises) along the radius of the cylinder.

To do this, open **Filter – Alphabetical– Plot over line**.

In the tab Properties for the filter Graph along the line1, set parameters as it is shown in the picture below. After this, go to the tab Display and select only the field Stress_Mises for display:



As a result, the distribution graph of the stress field (Mises) along the radius of the cylinder will appear on the graph to the right.



4. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/example.pvd)

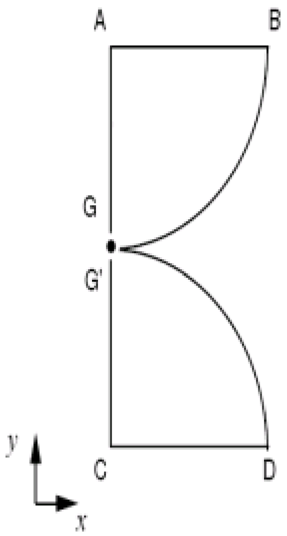
```
reset
create Cylinder height 100 radius 200
create Cylinder height 100 radius 100
subtract body 2 from body 1
webcut body 1 with plane xplane offset 0
webcut body 1 with plane yplane offset 0
webcut body 4 with plane zplane offset 0
delete Body 5
delete Body 1
delete Body 3
volume 4 scheme Auto
volume 4 size 10
mesh volume 4
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "MODULUS" 207e3 "POISSON" 0.3
"YIELD_STRENGTH" 208
set duplicate block elements off
block 1 volume 4
block 1 material 'Material 1'
create displacement on surface 32 dof 3 fix
create displacement on surface 33 dof 1 fix
create displacement on surface 35 dof 2 fix
create pressure on surface 34 magnitude 140
block 1 element type HEX27
analysis type static elasticity plasticity dim3
nonlinearopts maxiters 1000 loadsteps 10 tolerance 1e-3
calculation start path "\\ns25\calc_MPI\nointerface1.pvd"
calculation start path "D:/FidesysBundle/calc/example.pvd"
```

It is also possible to run the file *Example_15_Plasticity_3D.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Contact interaction modeling (3D)

G. DUMONT: "Method of the active stresses applied to the unilateral contact" Note HI-75/93/016 (http://www.code-aster.org/V2/doc/v10/en/man_v/v1/v1.01.246.pdf)

The Hertz problem for the two hemispheres is being solved. The pictures below represent a geometric model of the problem:



Due to the symmetry, ¼ part of the hemispheres is considered. The model is also fixed due to the condition of symmetry. On the upper side of the first hemisphere Displacement Uy = -2 mm is applied; Displacement Uy = 2 mm is applied on the bottom side of the second hemisphere. The material parameters are Young's Modulus E = 2e4 MPa, Poisson's Ratio v = 0.3.

Test pass criterion is the following:

at the point G σ_{yy} = -2798.3 MPa

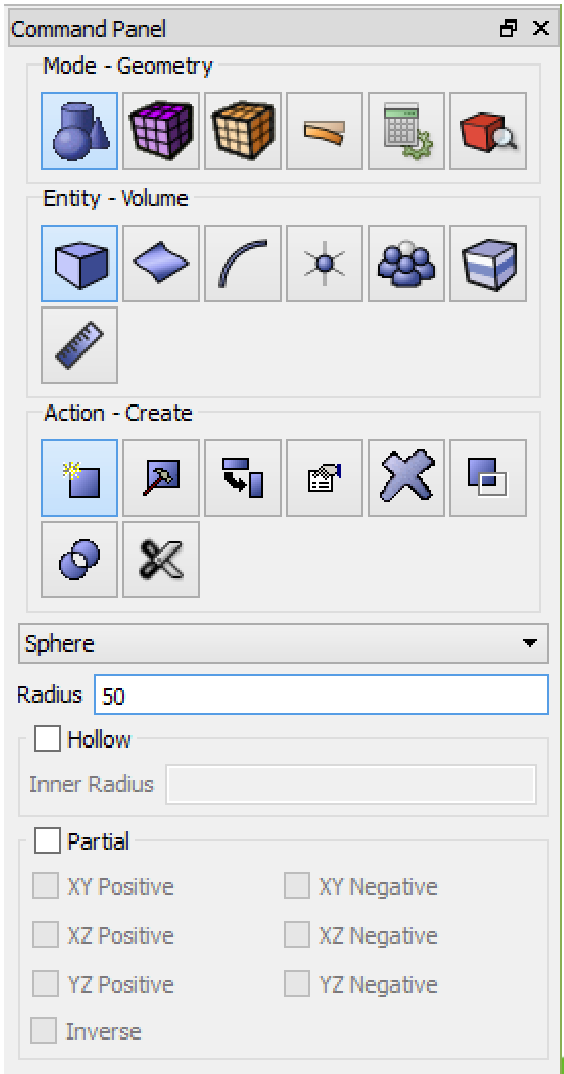
Geometry creation

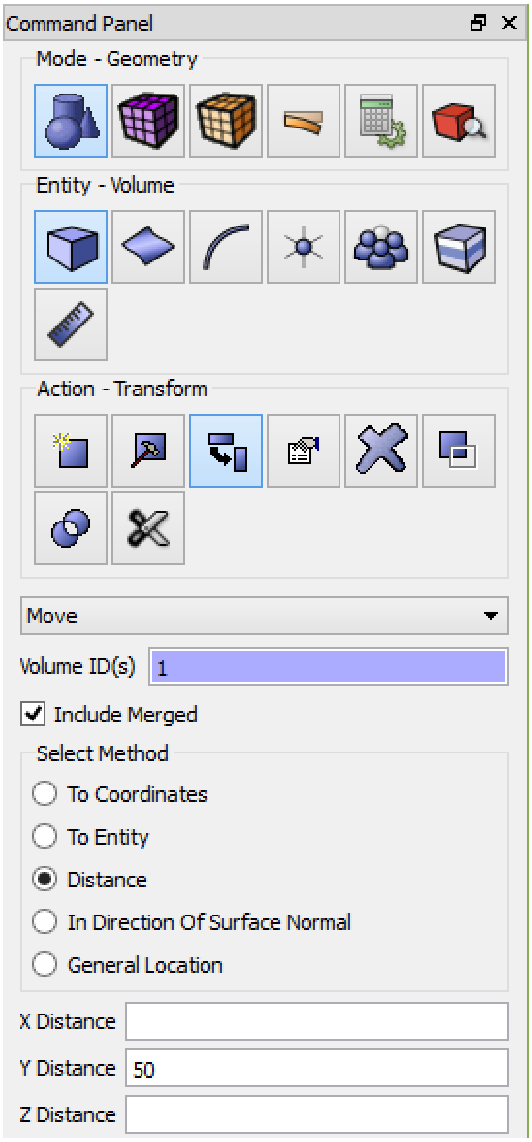
1. Create the first sphere.

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Create**). Select **Sphere** in the list of geometric elements. Set the sphere dimensions:

- Radius: 50;

Click **Apply**.





2. Move the created sphere along the Oy axis.

Select Mode — **Geometry**, Entity — **Volume**, Action — **Transform** on Command Panel. Set the following parameters:

- In the drop-down list, select: Move;
- Volume (ID): 1 (*or click with a mouse on the created sphere*);
- Select method: Distance;
- Distance along Y: 50.

3. Create the second sphere.

Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Volume**, Action — **Create**). Select **Sphere** in the list of geometric elements. Set the sphere dimentions:

- Radius: 50;

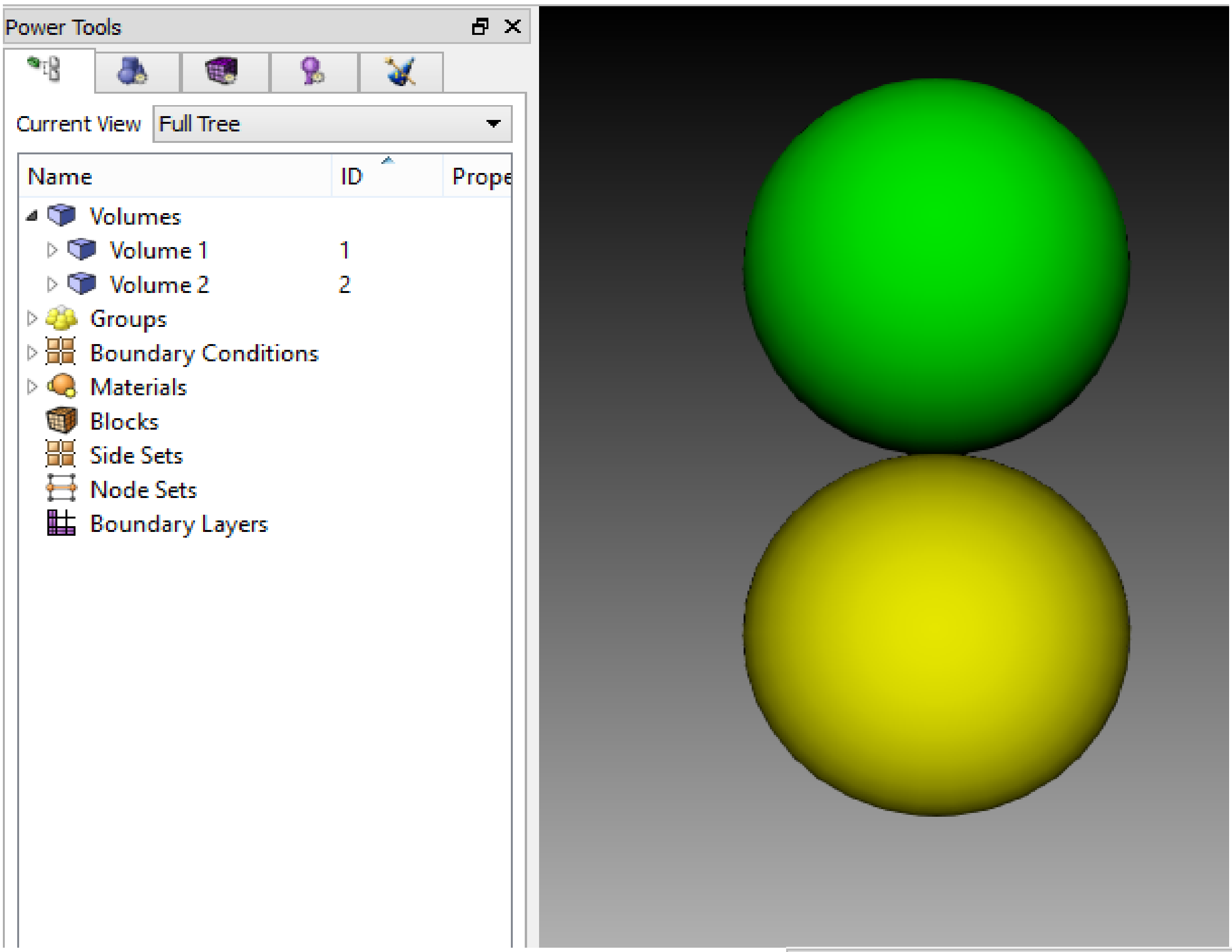
Click **Apply**.

4. Move the second sphere along the Oy axis.

Select Mode — **Geometry**, Entity — **Volume**, Action — **Transform** on Command Panel. Set the following parameters:

- In the drop-down list, select: Move;
- Volume (ID): 1 (*or click with a mouse on the created sphere*);
- Select method: Distance;
- Distance along Y: -50.

As a result, two generated entities are displayed in the Model Tree (Volume 1 and Volume 2). At the same time, both spheres contact at the point (0,0,0).



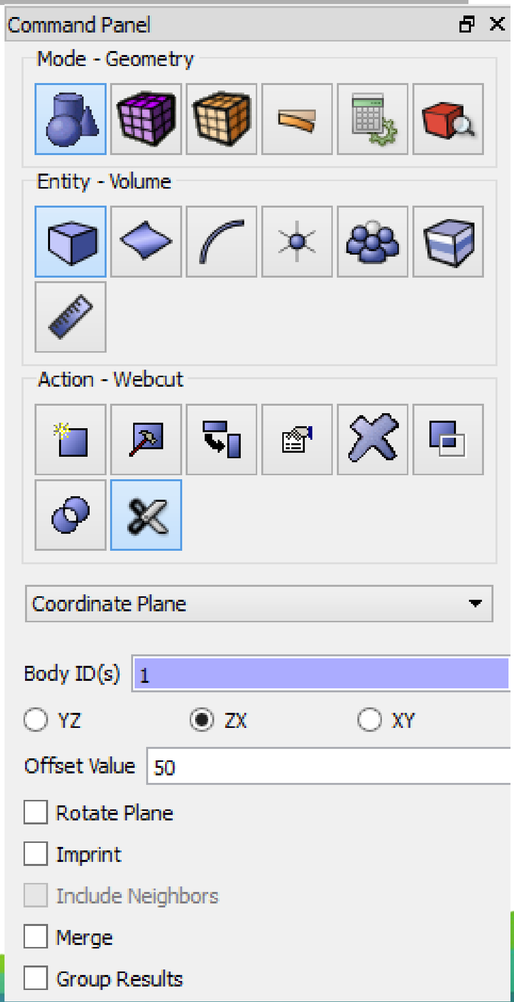
5. Leave half of the first sphere.

To do this, cut the first (upper) sphere by the ZX plane on center. Select Mode – **Geometry**, Entity – **Volume**, Action – **Webcut** on Command Panel. Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

- Body ID: 1 *(or click with a mouse on the upper sphere)*;
- Set the ZX checkbox;
- Offset value: 50.

Click **Apply**.

As a result, the upper sphere must be split into two parts.



6. Leave half of the second sphere.

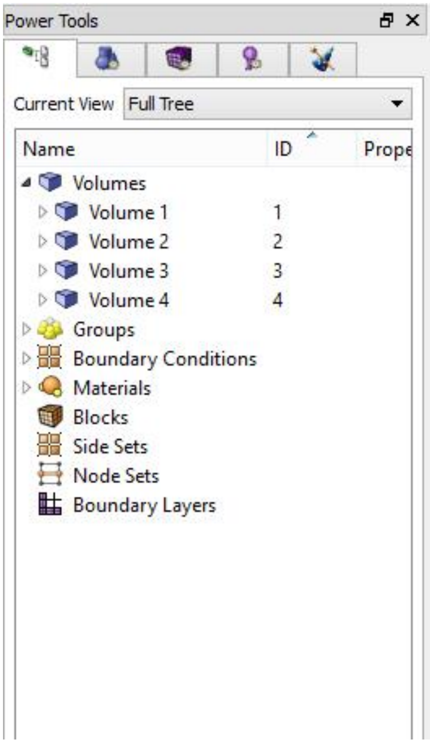
To do this, cut the second (upper) sphere by the ZX plane on center. Select Mode – **Geometry**, Entity – **Volume**, Action – **Webcut** on Command Panel. Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

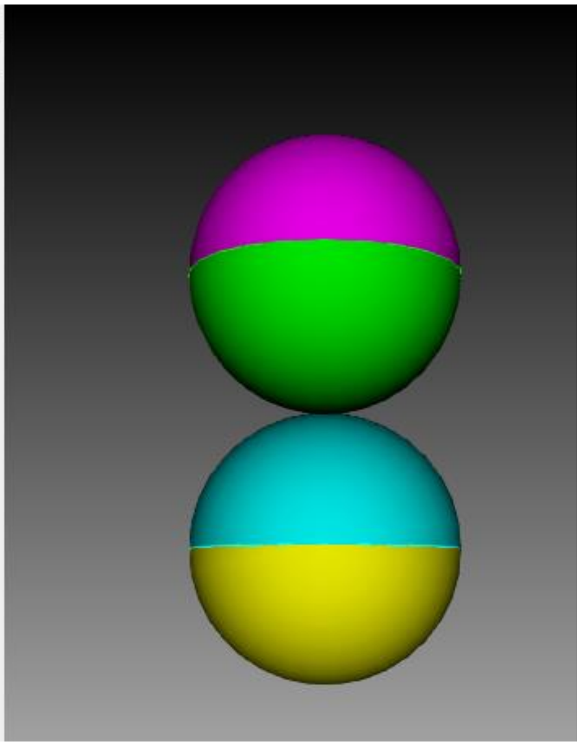
- Body ID: 2 *(or click with a mouse on the upper sphere)*;
- Set the ZX checkbox;
- Offset value: -50.

Click **Apply**.

As a result, the bottom sphere must be split into two parts.

Four resulting volumes must be displayed in the Model Tree on the left.



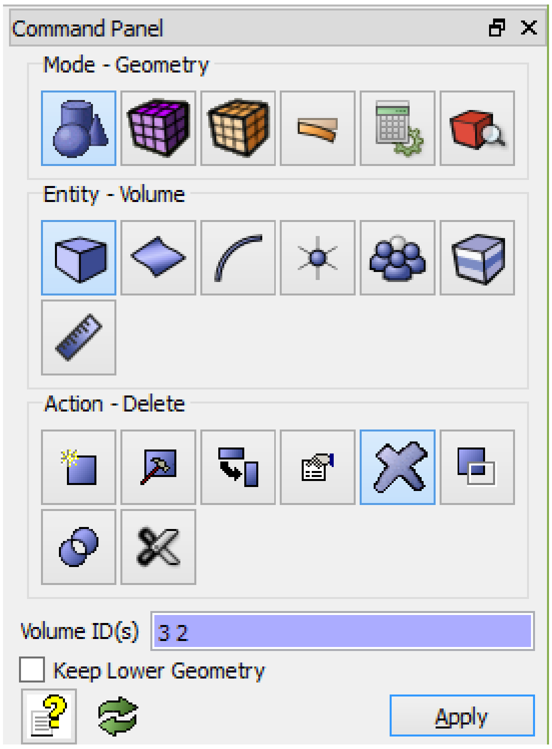


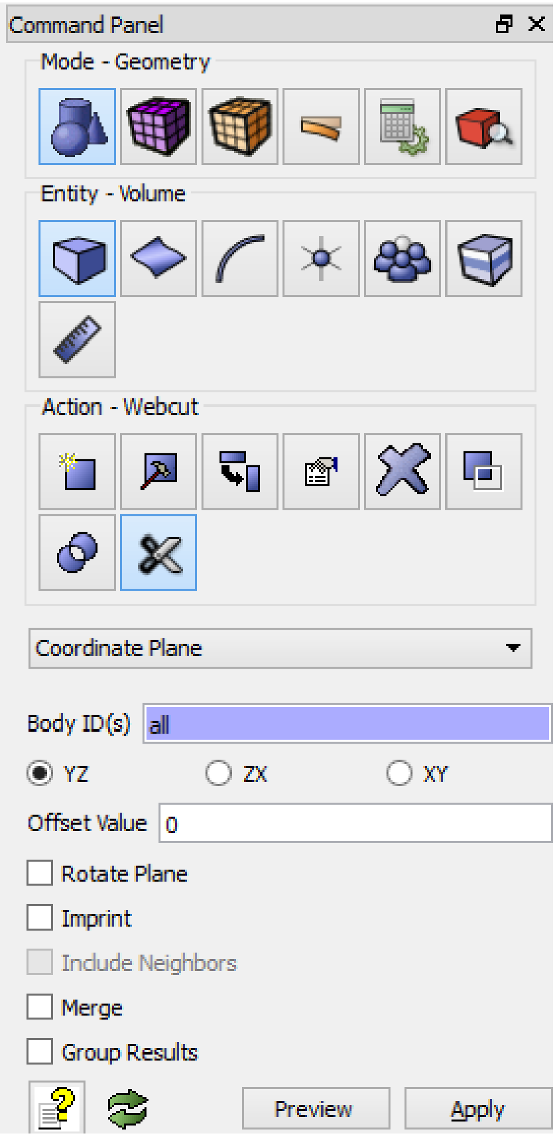
7. Remove the excess volumes.

It is necessary to remove the uppermost and the bottommost volumes. Select Mode – **Geometry**, Entity – **Volume**, Action – **Delete** on Command Panel. Set the following parameters:

- Body ID: 3 2 *(or click on the uppermost and the bottommost volumes holding down CTRL)*;

Click **Apply**.





8. Leave a quarter of spheres (symmetry of the problem).

Select volume geometry generation section on Command Panel (Mode – **Geometry**, Entity – **Volume**, Action – **Webcut**). Select **Coordinate Plane** in the list of possible webcut types. Set the following parameters:

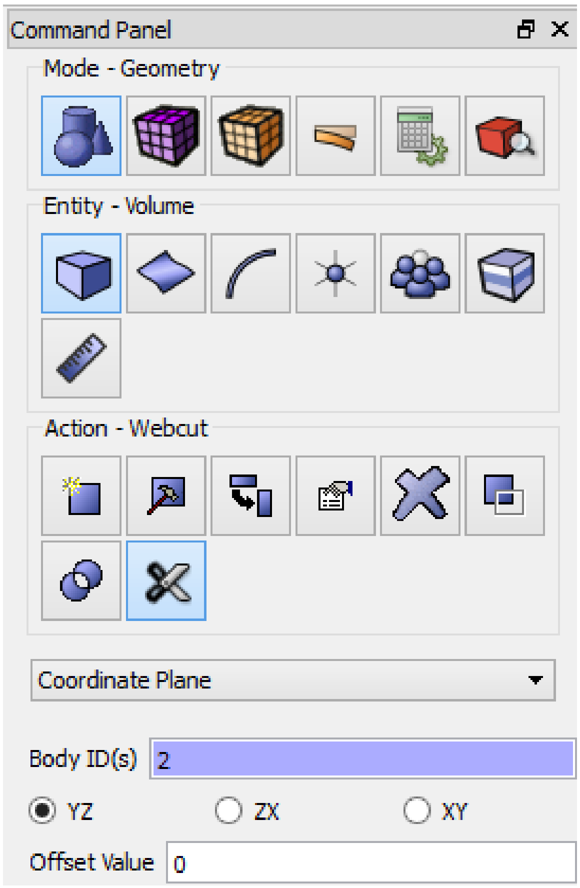
- Volume ID(s): all *(the volumes to be webcut)*;
- Webcut with: YZ Plane;
- Offset value: 0.

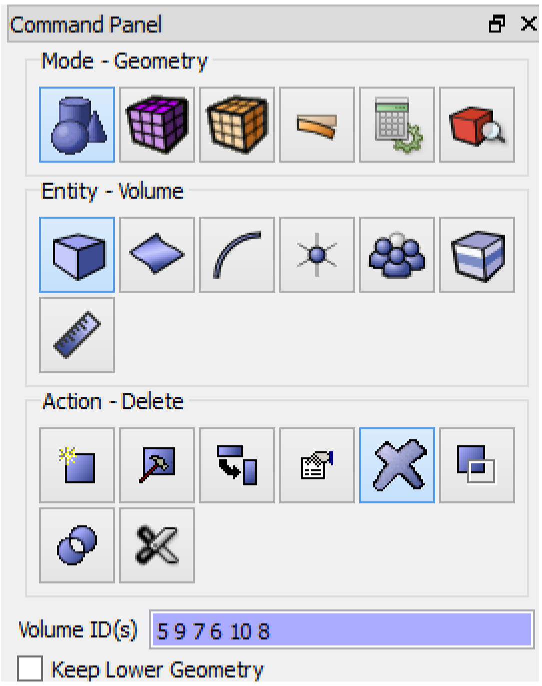
Click **Apply**.

Do the same for the XZ Plane:

- Volume ID(s): all *(the volumes to be webcut);*
- Webcut with: XZ Plane;
- Offset value: 0;
- Imprint.

Click **Apply**.





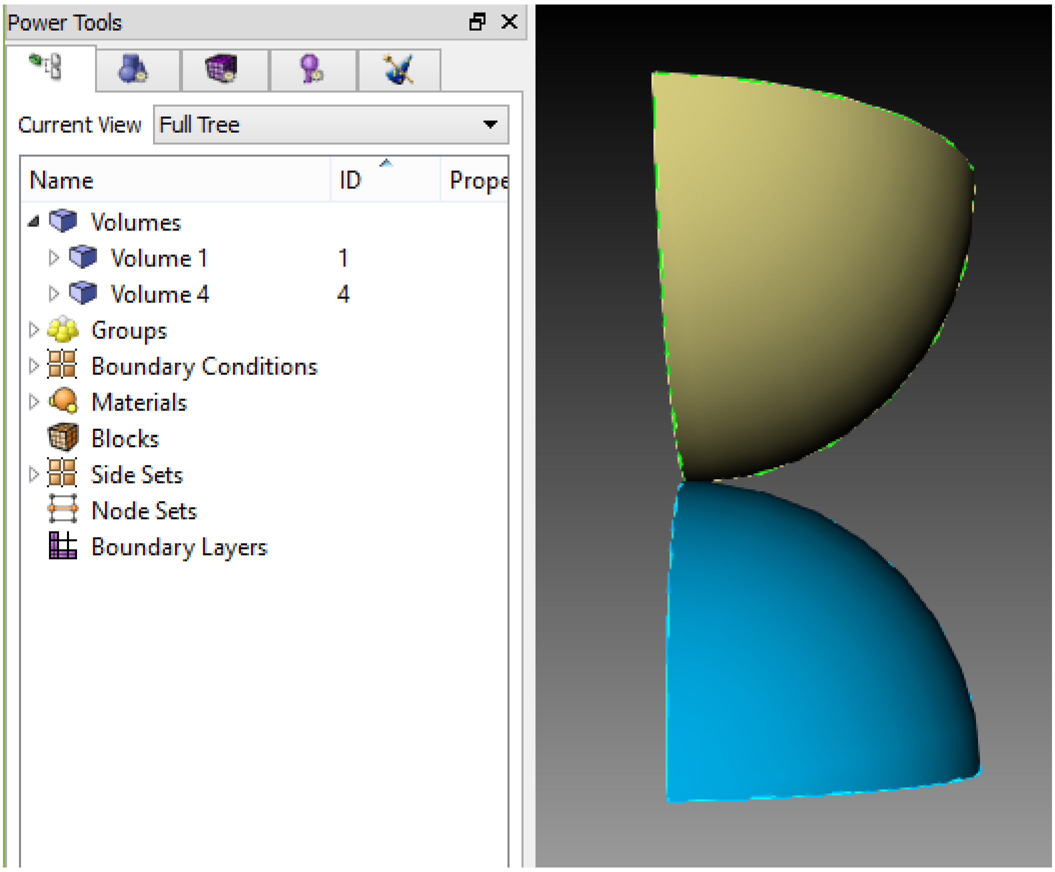
9. Remove the excess volumes.

Select Mode — **Geometry**, Entity — **Volume**, Action — **Delete** on Command Panel. Set the following parameters:

- Body ID: 5 9 7 6 10 8 (*or by holding down CTRL, click successively on all volumes, except those that are in the first and eighth coordinate quarter*);

Click **Apply**.

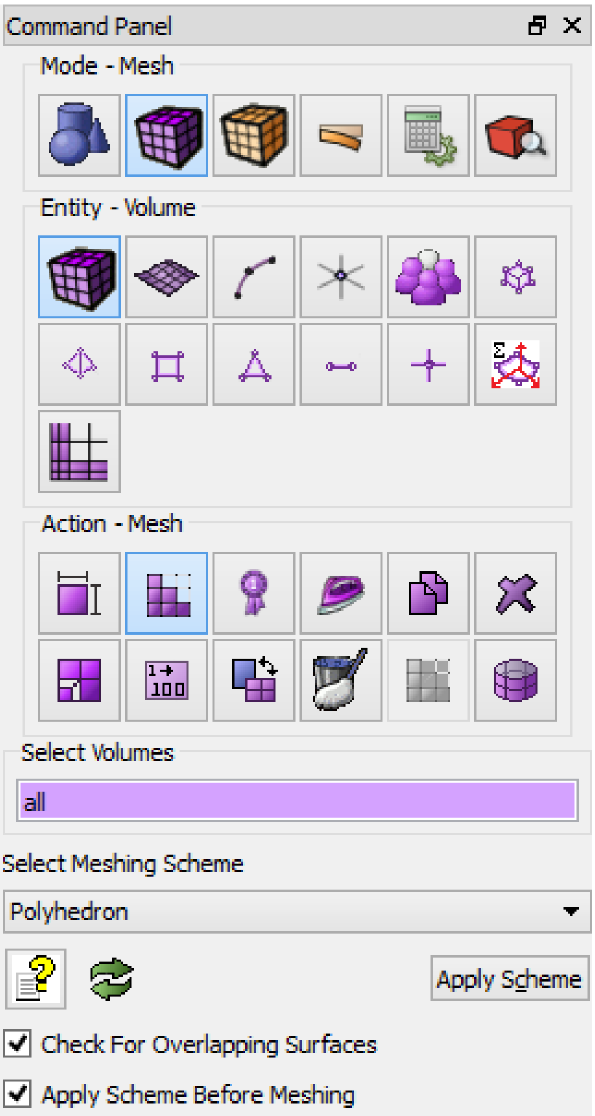
As a result, two volumes must be left in the Model Tree on the left – Volume 1 and Volume 4.

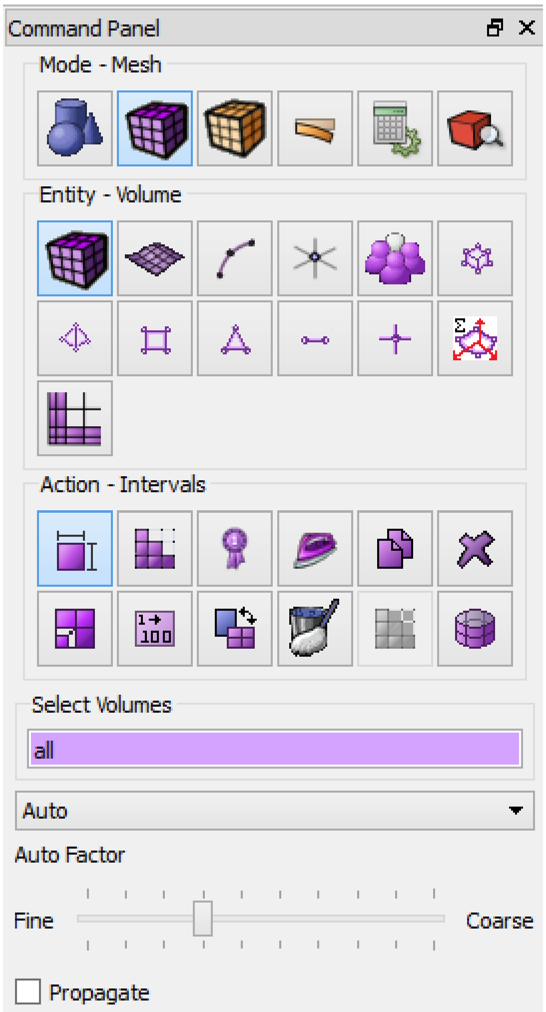


Meshing

1. Select volume mesh generation section on Command Panel (Mode – **Mesh,** Entity – **Volume**, Action – **Meshing**). Specify the following parameters:
   - Select volumes: *all*;
   - Select meshing scheme: Polyhedron;

Click **Apply Scheme**.



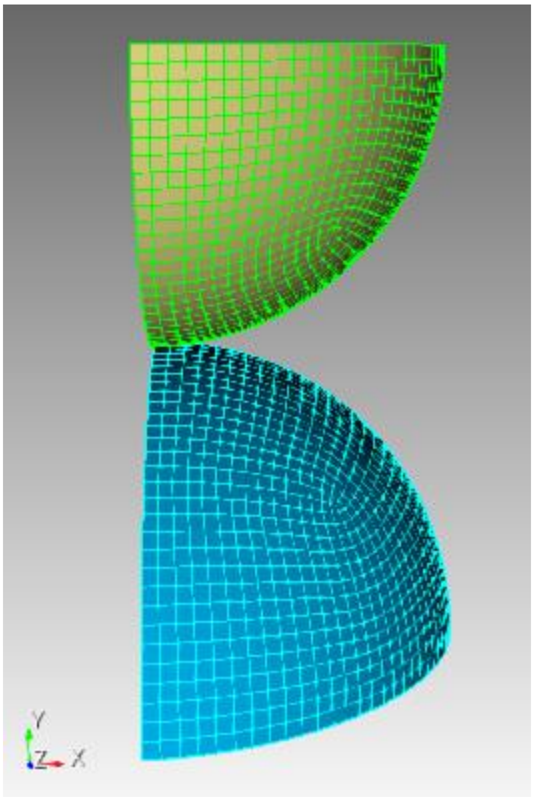


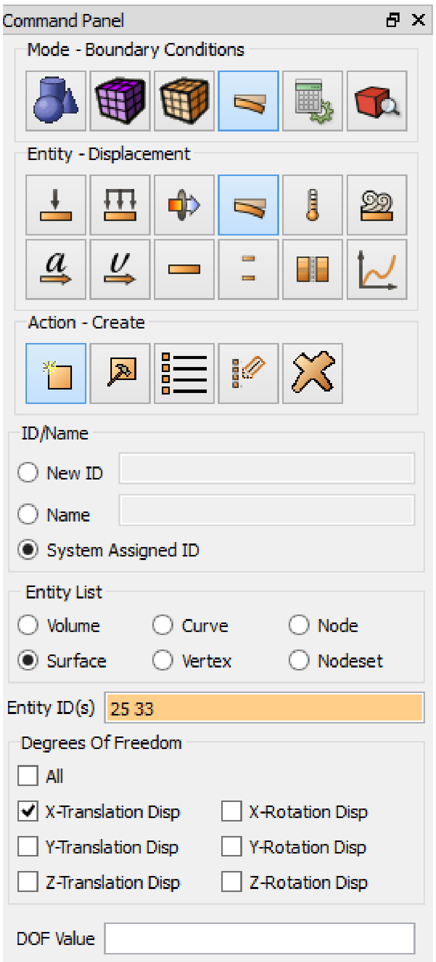
2. On Command Panel, go to the section Intervals (Mode – **Mesh,** Entity – **Volume**, Action – **Intervals**). Specify the following parameters:
   - Select volumes: all;
   - Select meshing scheme: Automatically calculate;
   - Move the slider one scale division to the left

Click **Apply**.

Click **Mesh**.

The result of the actions should be the following model partition into finite elements:





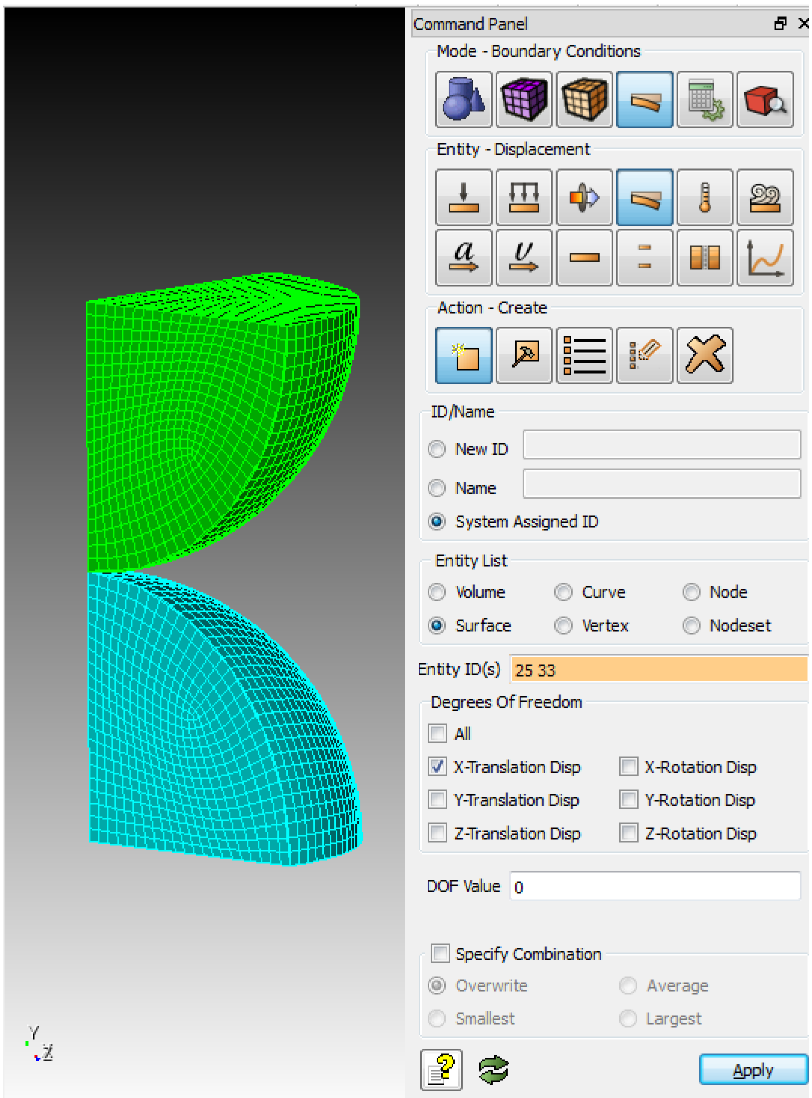
Setting boundary conditions

1. Fix side edges at direction X.

Select Mode – **Boundary Conditions,** Entity – **Displacement**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 25 33 *(or select surfaces with the mouse as it is shown in the picture below)*;
- Degrees of Freedom: X-Translation Disp;
- DOF Value: 0.

Click **Apply**.

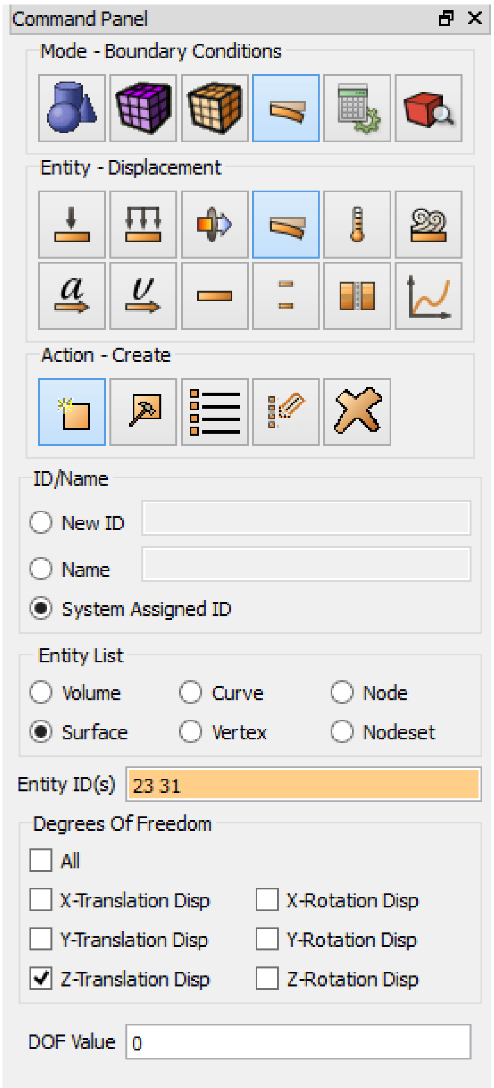


2. Fix side edges at direction Z by analogy

Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 23 31 *(or select surfaces with the mouse as it is shown in the picture below)*;
- Degrees of Freedom: Z-Translation Disp;
- DOF Value: 0.

Click **Apply**.

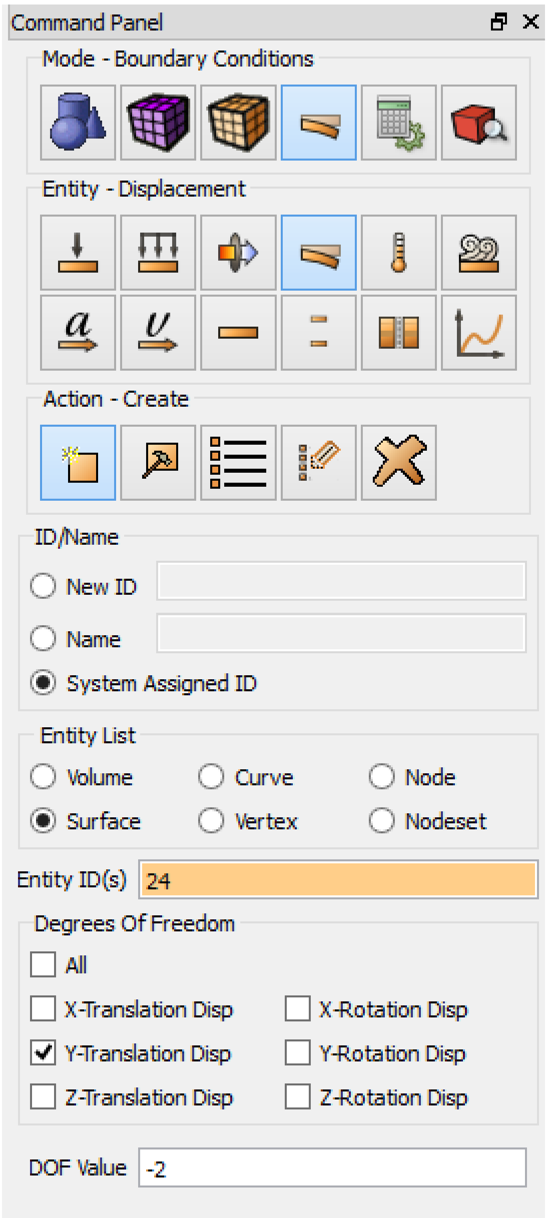


3. Set the displacement Uy = -2 on the upper edge of the first hemisphere by analogy

Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 24 *(or click on the upper edge of the upper hemisphere)*;
- Degrees of Freedom: Y-Translation;
- DOF Value: -2.

Click **Apply**.



4. Set the displacement Uy = 2 on the bottom edge of the second hemisphere by analogy

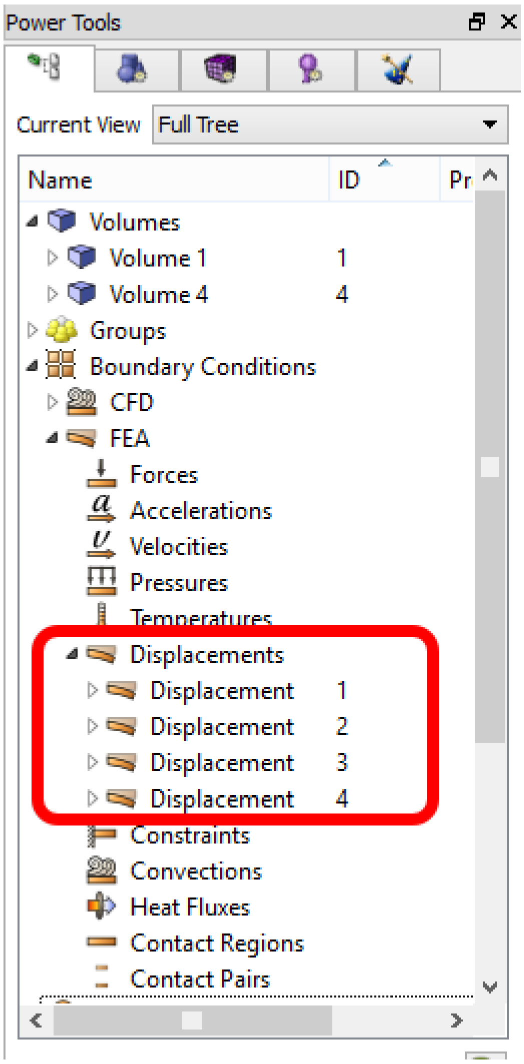
Set the following parameters:

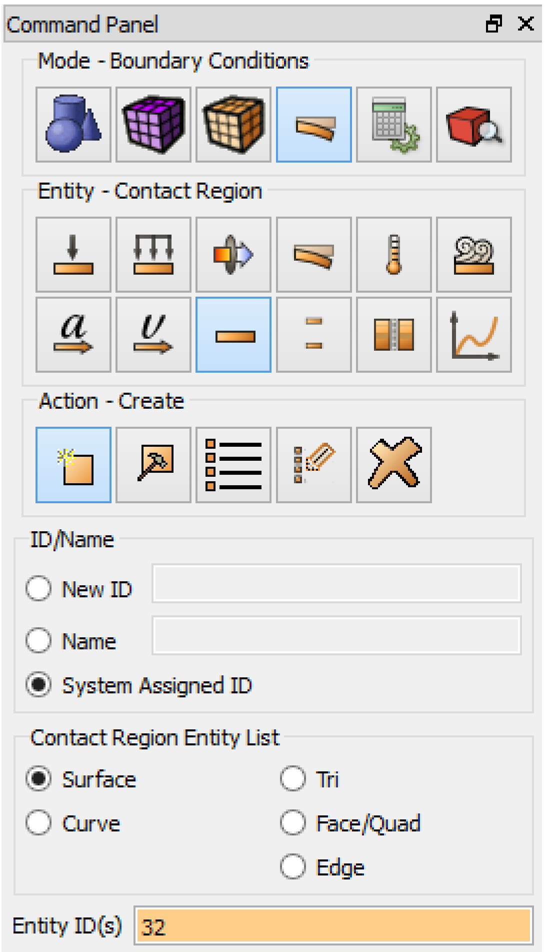
- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 34 *(or click on the bottom edge of the bottom hemisphere)*;
- Degrees of Freedom: Y-Translation Disp;
- DOF Value: 2.

Click **Apply**.

All the applied boundary conditions should be displayed in the Model Tree on the left. In addition, the boundary conditions can be edited from the Model Tree.

To view all the applied boundary conditions, please click Show BC on the top panel.





Setting contact interaction

1. Set Contact Region 1.

Select Mode – **Boundary Conditions,** Entity – **Contact Region**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;
- Entity List: Surface;
- Entity ID(s): 32 *(or select with the mouse the convex surface of the bottom hemisphere)*;

Click **Apply**.

2. Set Contact Region 2.

Select Mode – **Boundary Conditions,** Entity –**Contact Region**, Action – **Create** on Command Panel. Set the following parameters:

- System Assigned ID;

- Entity List: Surface;
- Entity ID(s): 26 *(or select with the mouse the convex surface of the upper hemisphere)*;

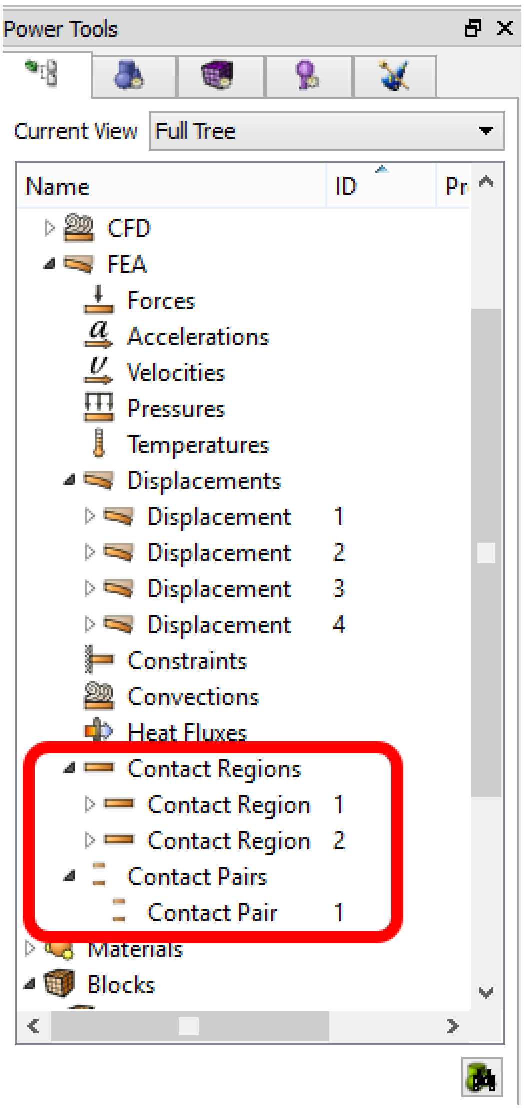
Click **Apply**.

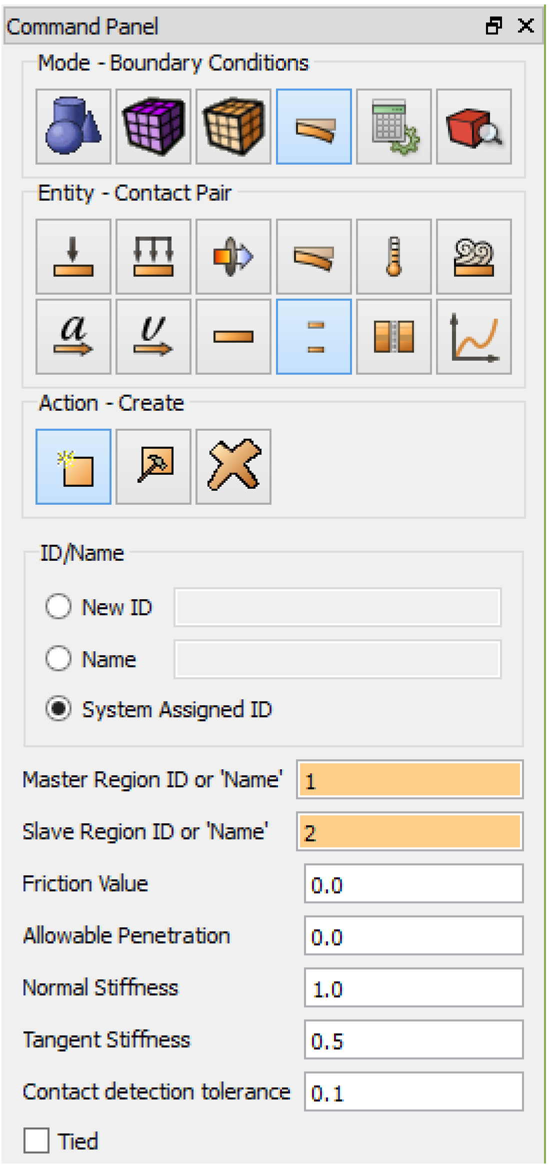
3. Set the Contact pair.

Select Mode — **Boundary Conditions,** Entity — **Contact Pair**, Action — **Create.** Set the following parameters:

- System Assigned ID;
- ID/Master region name: 1 *(the bottom hemisphere)*;
- ID/Slave region name: 2 *(the upper hemisphere)*;
- Keep all other settings of the contact pair by default

Click **Apply**.





Contact Regions and Contact Pair should be displayed in the Model Tree on the left.

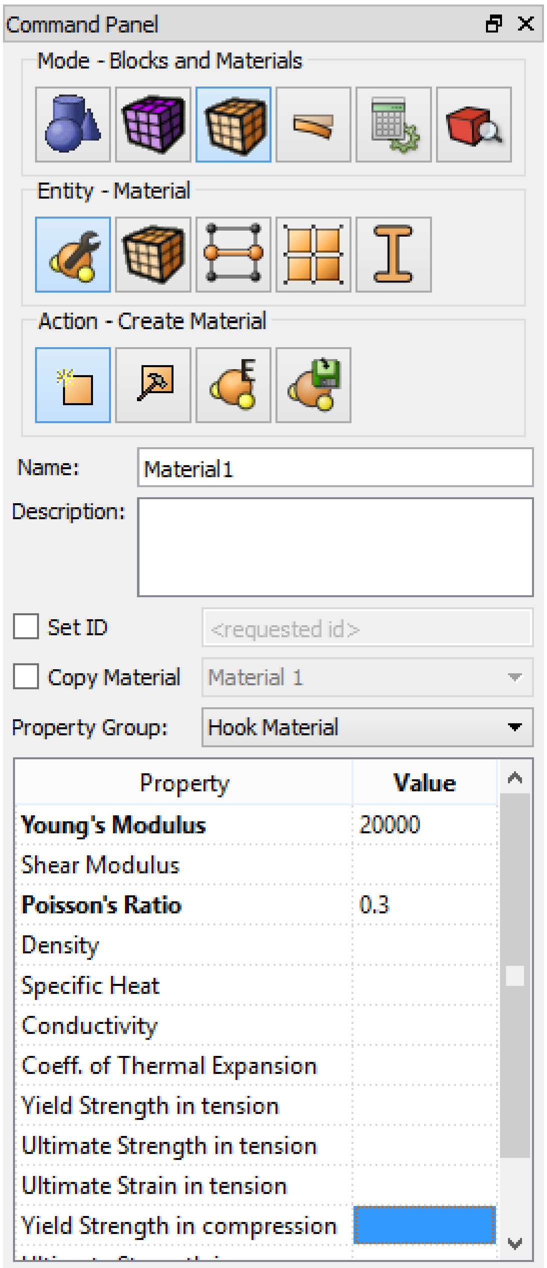
Setting material and element type

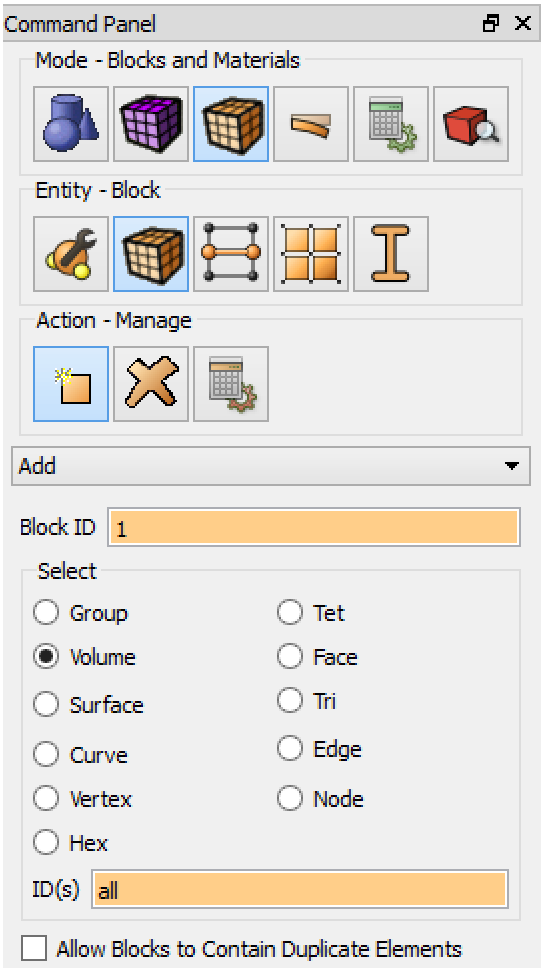
1. Create the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material 1;
- Description: Hook material
- Property group: Hook material;
- Young's Modulus: 20 000;
- Poisson's Ratio: 0.3;

Click **Apply**.





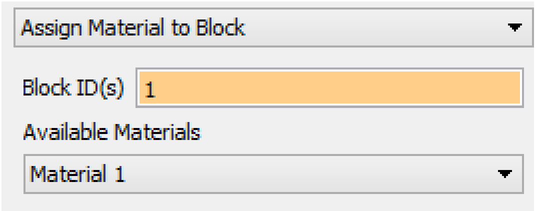
2. Create the block of one type of the material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Add** in the list of possible operations. Set the following parameters:

- Block ID: 1;
- Entity type to be united into the block: Volume;
- ID: all.

Click **Apply**.

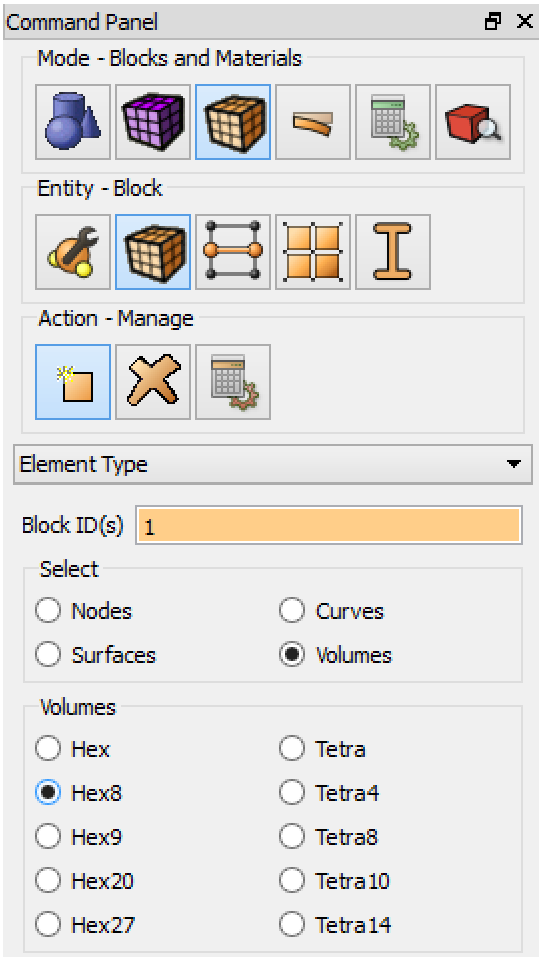
3. Assign the material to the block.



Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material 1.

Click **Apply**.



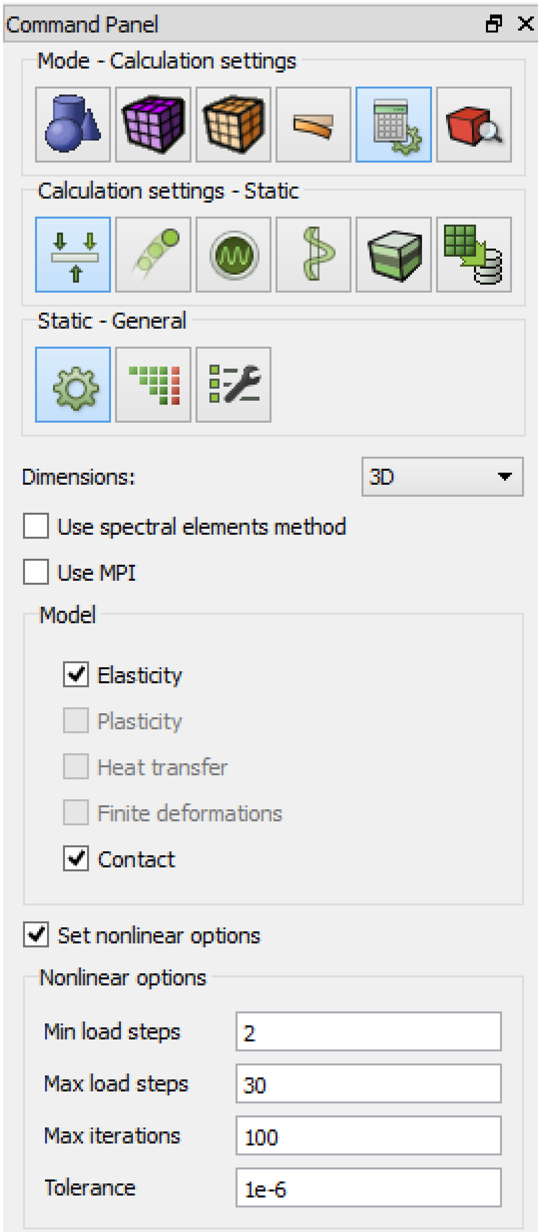
4. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select: Volumes;
- Volumes: HEX8.

Click **Apply**.

Starting calculation



1. Set the type of the problem to be solved.

Select calculation setting section on Command Panel (Mode – **Calculation settings**, Calculation settings – **Static**, Static – **General**). Select:

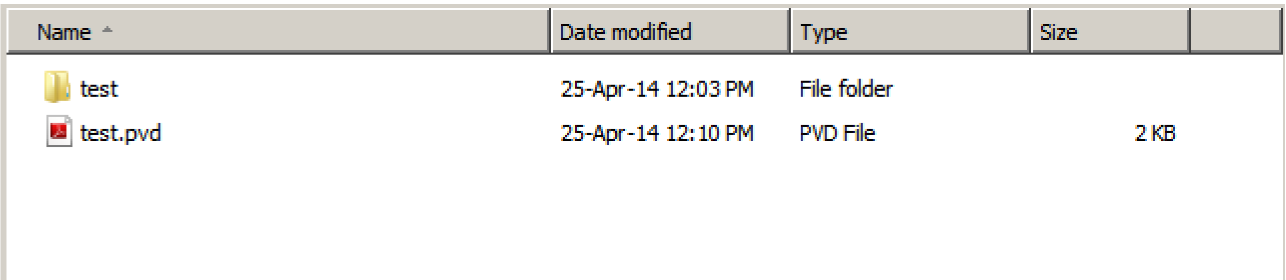
- Dimension: 3D;
- Model: Elasticity;
- Model: Contact.
- Nonlinear solver settings: Leave the default values.

Click **Apply**.

Click **Start Calculation**.

2. In a pop-up window select a folder to save the result and enter the file name. In this case, save the calculation in the file test.pvd.
3. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

For non-linear problems, convergence of iterations at each loading step can be checked in the file Convergence.txt. The file is downloaded into the folder *test* that is created next to the file test .pvd.

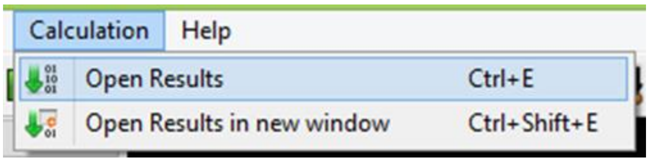


Open the file test\Convergence.txt:

Make sure that the convergence with a specified tolerance is reached at each step of loading.

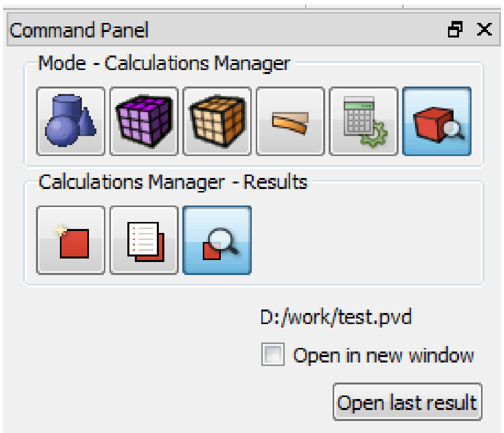
```
CONVERGENCE ITERATIONS

CONVERGENCE ITERATIONS

     LoadTime   Residual
0.500000   234474484.859693200000000
    0.500000   0.000000001649635
    ----------------------------
1.000000   807631243.628153320000000
  1.000000   463217.315709784280000
   1.000000   20885.849428797028000
    1.000000   557.717737451076910
    ---------------------------
```

Results analysis



1. Open the file with the results. You can do this in one of the three ways.

   - Click Ctrl+E.
   - Select **Calculation** → **Open Results** in the Main Menu. Click **Open last result**.
   - Select Calculations Manager on Command Panel (Mode — **Calculations Manager**, Calculations Manager — **Results**). Click **Open Results**.

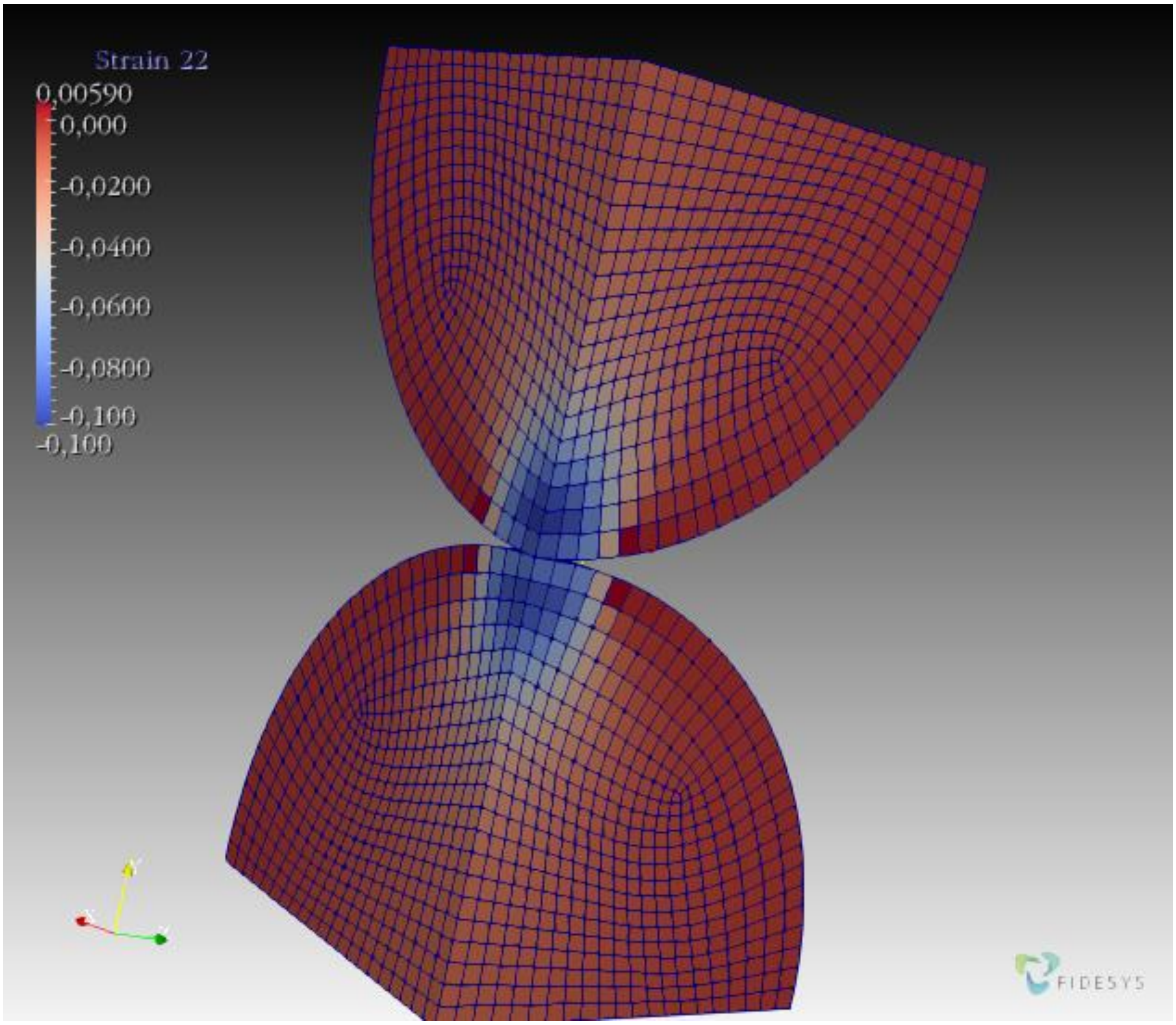


To apply all of the filters changes automatically, click **Apply changes to parameters automatically** on Command Panel.

2. Display the stress field σ_{yy} and the mesh on the model.

In ***Fidesys Viewer*** window set the following parameters on Toolbar:

- Representation Field: Strain;
- Component: 22;
- Representation Mode: Surface With Edges.



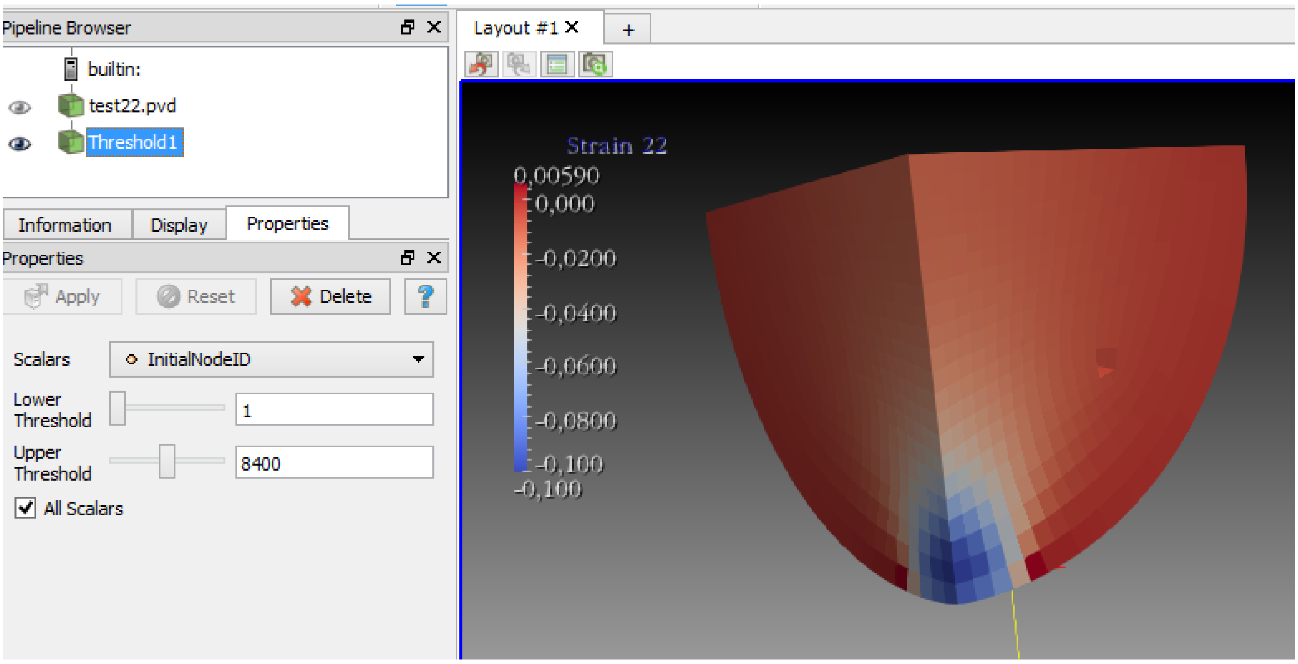
3. Define the stress at the specified point G.

This point belongs to the upper hemisphere, therefore the the lower hemisphere should be cut off.

Open **Filter – Alphabetical – Threshold** to do this. Or click on the appropriate button on the Fidesys Viewer top panel.

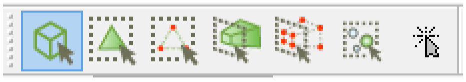
Set the following parameters for the filter Threshold in the tab Properties:

- Scalars: InitialNodeID;
- Lower threshold: Leave unchanged;
- Upper threshold: 8400.

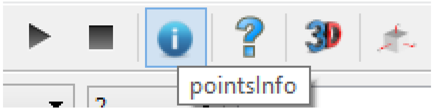


Thus, only the upper hemisphere is left.

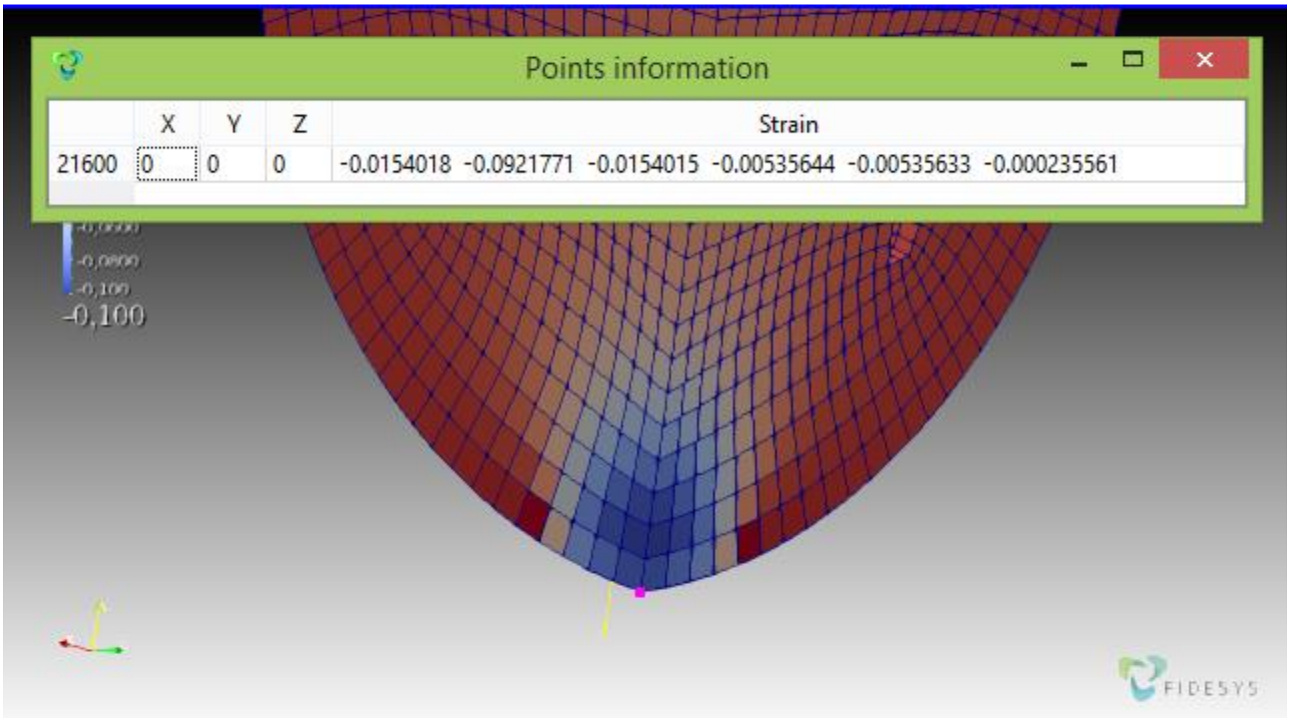
Select the bottom point of the hemisphere using the appropriate Fidesys Viewer tools.



Use the button Point information



As a result, the components of the Stress field for the selected point G should be displayed in the pop-up window Point information.

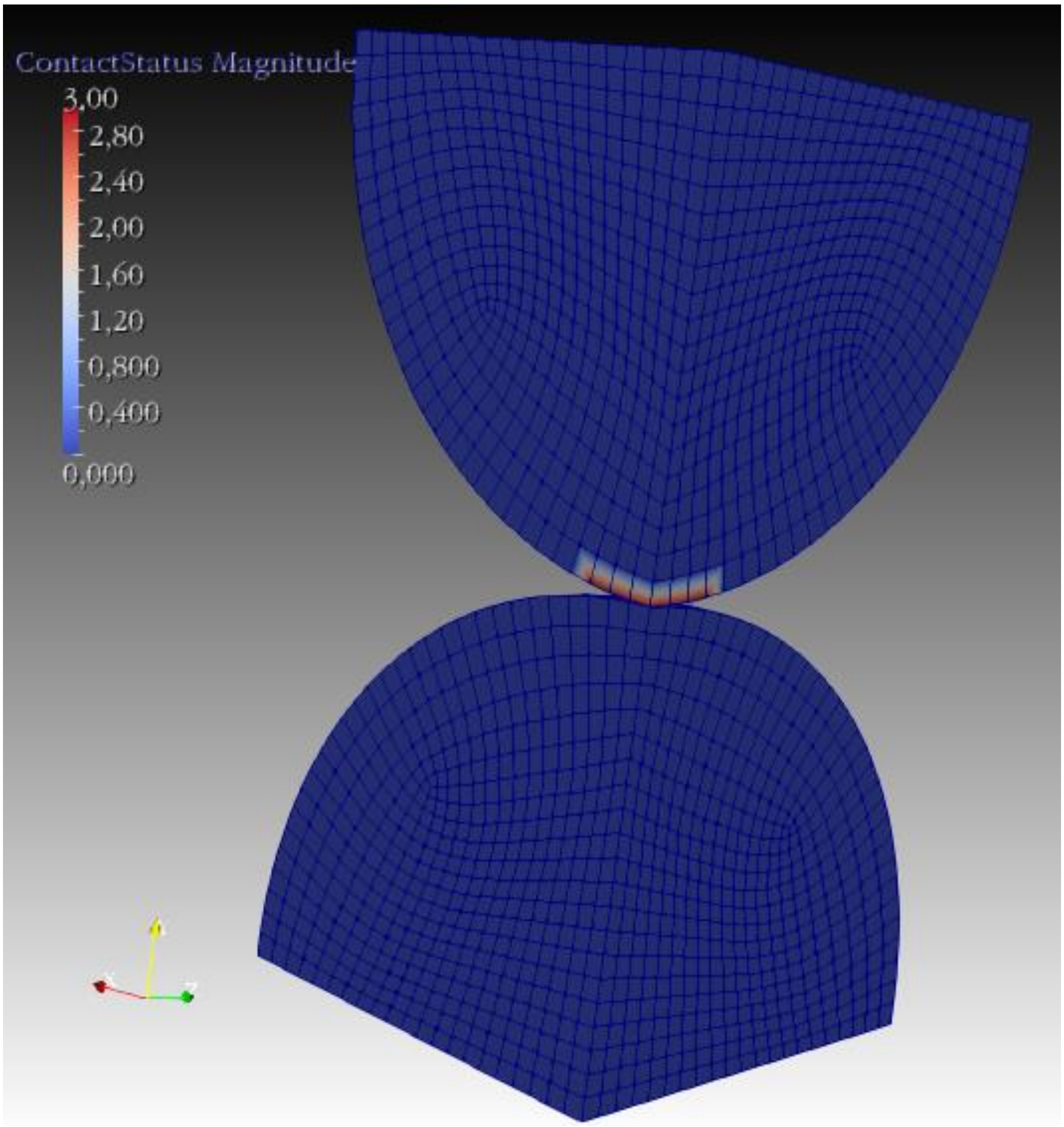


Compare the value of the second stress field component and the original value -2798.3 MPa. The difference between the resulting value -2837.1 and the original one is 1.4%.

4. Display the Contact Status field and the mesh on the model.

In the Model Tree on the left, please, change back focus on the calculation name test.pvd. Set the following parameters on Toolbar:

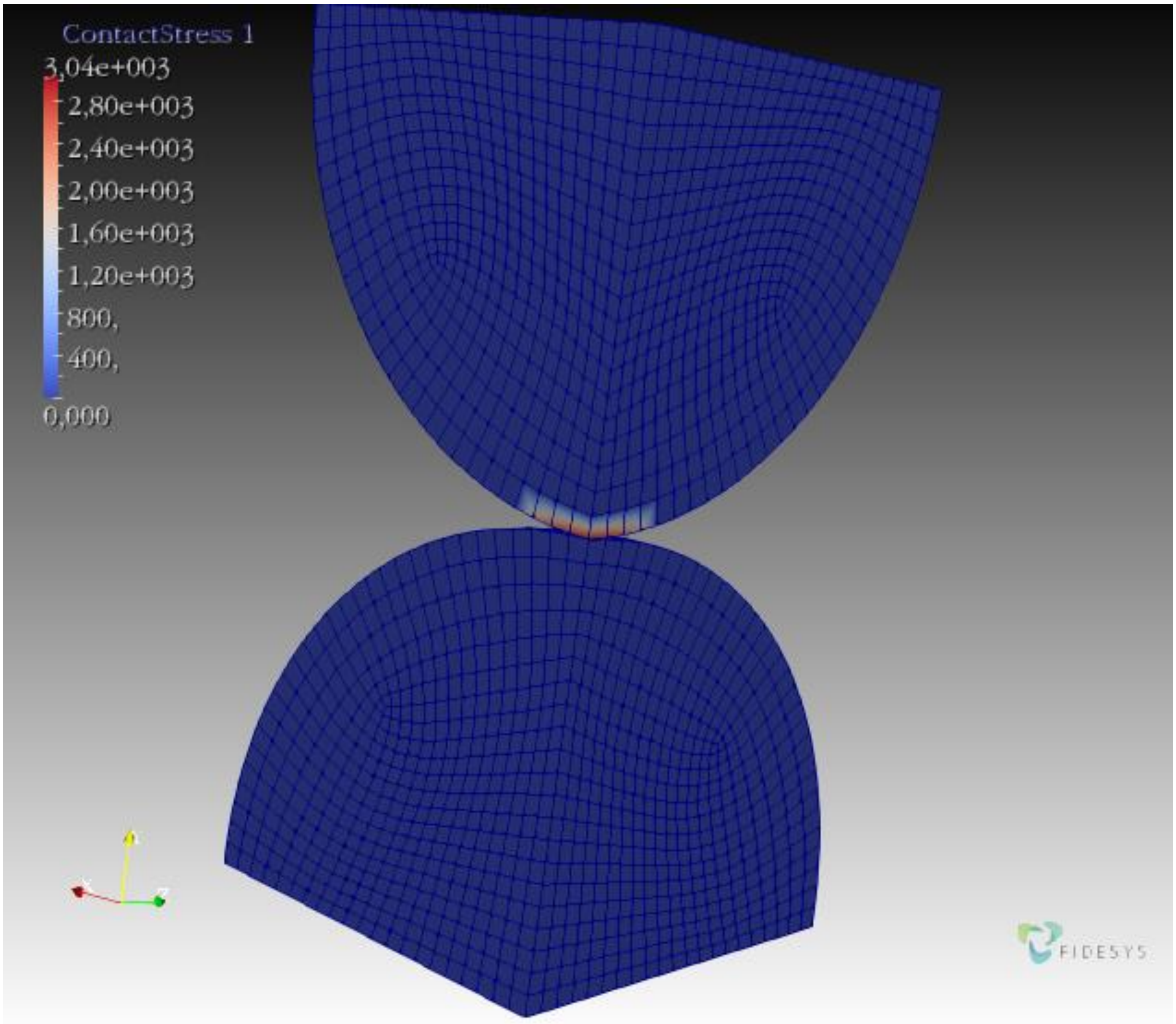
- Representation Field: Contact Status;
- Component: 1;
- Representation Mode: Surface With Edges.



5. Display the Stress field in contact and the mesh on the model.

In Fidesys Viewer window set the following parameters on Toolbar:

- Representation Field: ContactStress;
- Component: 11;
- Representation Mode: Surface With Edges.



6. Download numerical data.

Select **File** → **Save Data** in the Main Menu or click **Ctrl+S**. Enter the file name (*.csv format), leave it by default. Click **OK**. The saved file is an ordinary table of numerical data which can be opened in any text editor.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.

Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/test.pvd)

```
reset
create sphere radius 50
create sphere radius 50
move Volume 1 y 50 include_merged
move Volume 2 y -50 include_merged
webcut body all with plane yplane offset 50
webcut body all with plane yplane offset -50
delete volume 3 2
webcut body all with plane xplane offset 0
webcut body all with plane zplane offset 0
delete volume 5 9 7 6 10 8
volume all scheme Polyhedron
volume all size auto factor 4 #14850
mesh volume all
create displacement on surface 23 31 dof 3 fix 0
create displacement on surface 25 33 dof 1 fix 0
create displacement on surface 24 dof 2 fix -2
create displacement on surface 34 dof 2 fix 2
create contact region on surface 32
create contact region on surface 26
create contact pair 1 master contact region 1 slave contact region 2 friction 0.0
tolerance 0.0 tied off
cpairdata id 1 normal_stiffness 1.0 tangent_stiffness 0.5 detection_tolerance 0.1
create material "Material 1" property_group "CUBIT-FEA"
modify material "Material 1" scalar_properties "MODULUS" 2e+04 "POISSON" 0.3
block 1 volume all
block 1 material 'Material 1'
block 1 element type hex8
analysis type static elasticity contact dim3
nonlinearopts maxiters 100 minloadsteps 2 maxloadsteps 30 tolerance 1e-6
spectralelement off
usempi off
calculation start path "D:/FidesysBundle/calc/test.pvd"
```

It is also possible to run the file *Example_16_Contact_3D.jou* by selecting Journal Editor on Toolbar. In a pop-up window of the main menu select **File** → **Open** and open the necessary journal file.

Composite effective properties calculation

Кристенсен Р. Введение в механику композитов. – М., «Мир», 1982. – 334 с.

[Christensen R., Mechanics of composite materials (translated from English), Mir publ, Moscow, 1982, 334 pages [in Russian]]

The problem of finding effective material properties for the two-layer fiber-layered composite is being solved. The model has the following parameters: cord diameter is 6.0 mm, cord inclination angle is 30°, cord step is 8.0 mm, layer thickness is 16.0 mm. Thus, geometry is generated automatically via the interface with the specified parameters. The boundary conditions are periodic. The material properties of cords (block 1): Young's Modulus 200000, Poisson's Ratio 0.25. The material properties of the matrix (block 2): Young's Modulus 2.0, Poisson's Ratio 0.49.

Test pass criterion is the following: as a result of solving, the following material constants are obtained:

C_1111 = 24 852.4 MPa, C_1122 = 8 281.54 MPa, C_2222 = 2 763.12 MPa, C_1212 = 8 283.5 MPa within 3%.

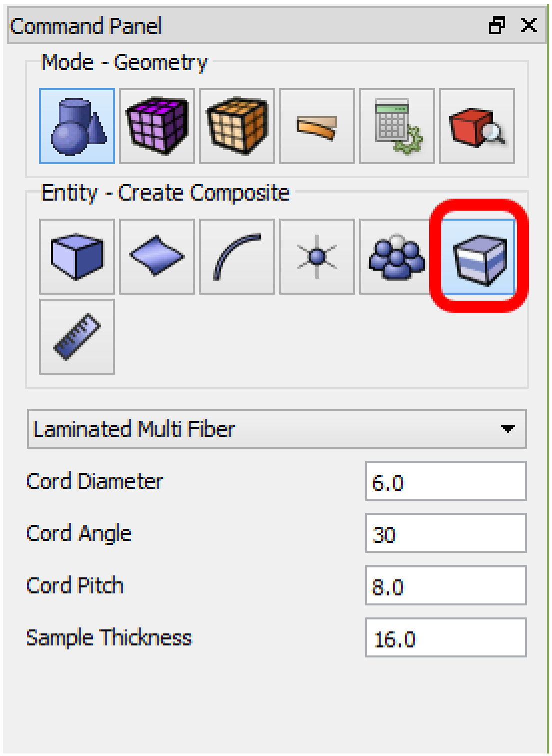
Geometry creation

1. Create the composite.

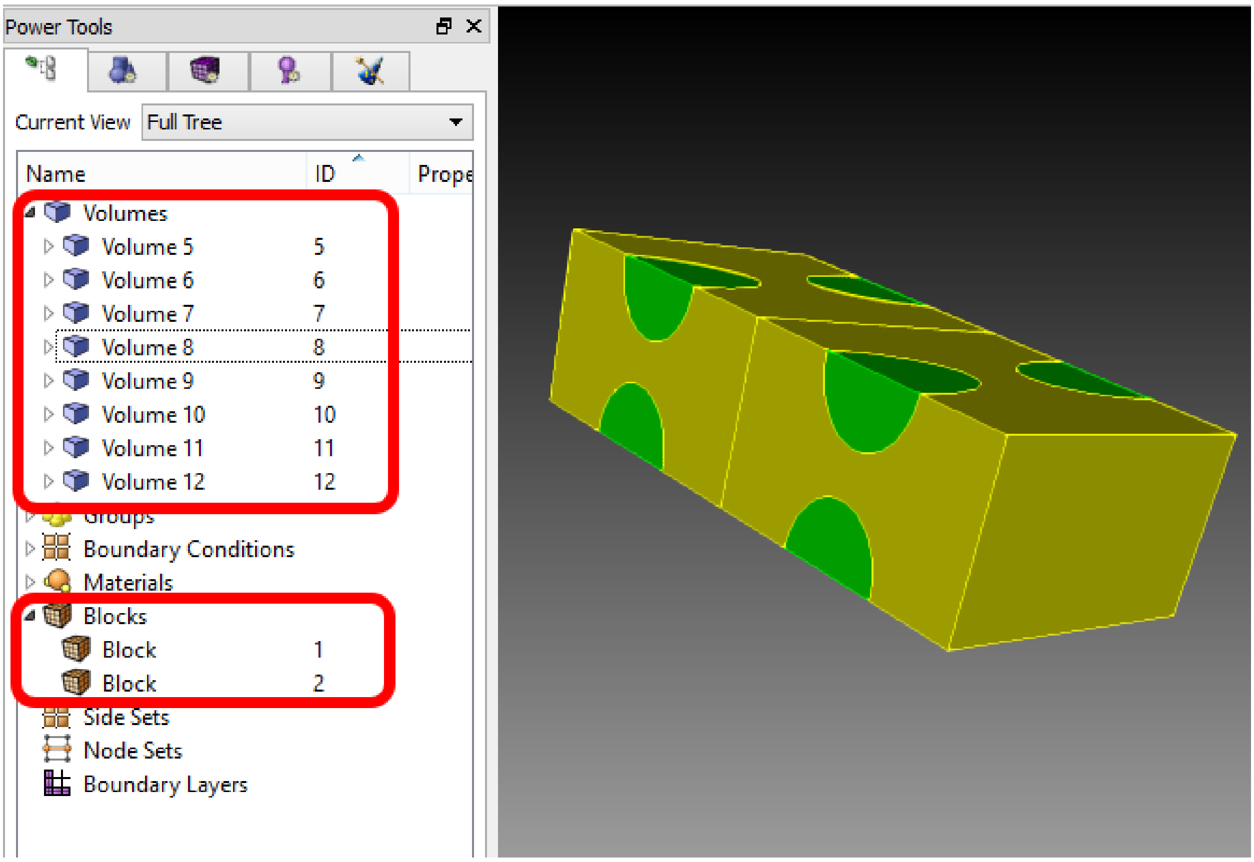
Select volume geometry generation section on Command Panel (Mode — **Geometry**, Entity — **Create composite**). Select **Laminated Multi Fiber** from the drop-down list. Set the following parameters:

- Cord Diameter: 0.6;
- Cord Angle: 30 (degrees);
- Cord Pitch: 8;
- Sample Thickness: 16.

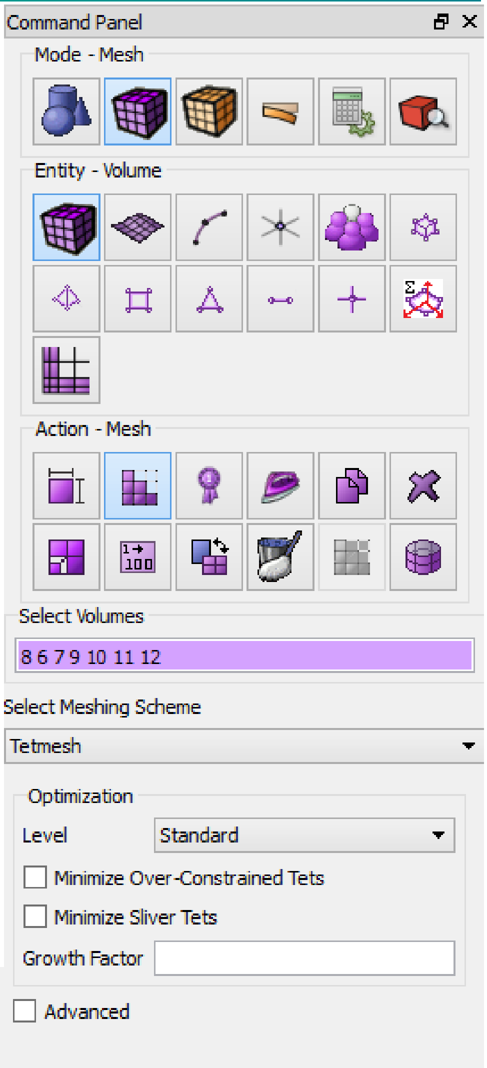
Click **Create**.



As a result, the geometry for the two-layer composite with the specified parameters is automatically generated. On the left in the Model Tree, all generated volumes and the corresponding blocks (Block 1 – thread, Block 2 – matrix) should be displayed.



Meshing



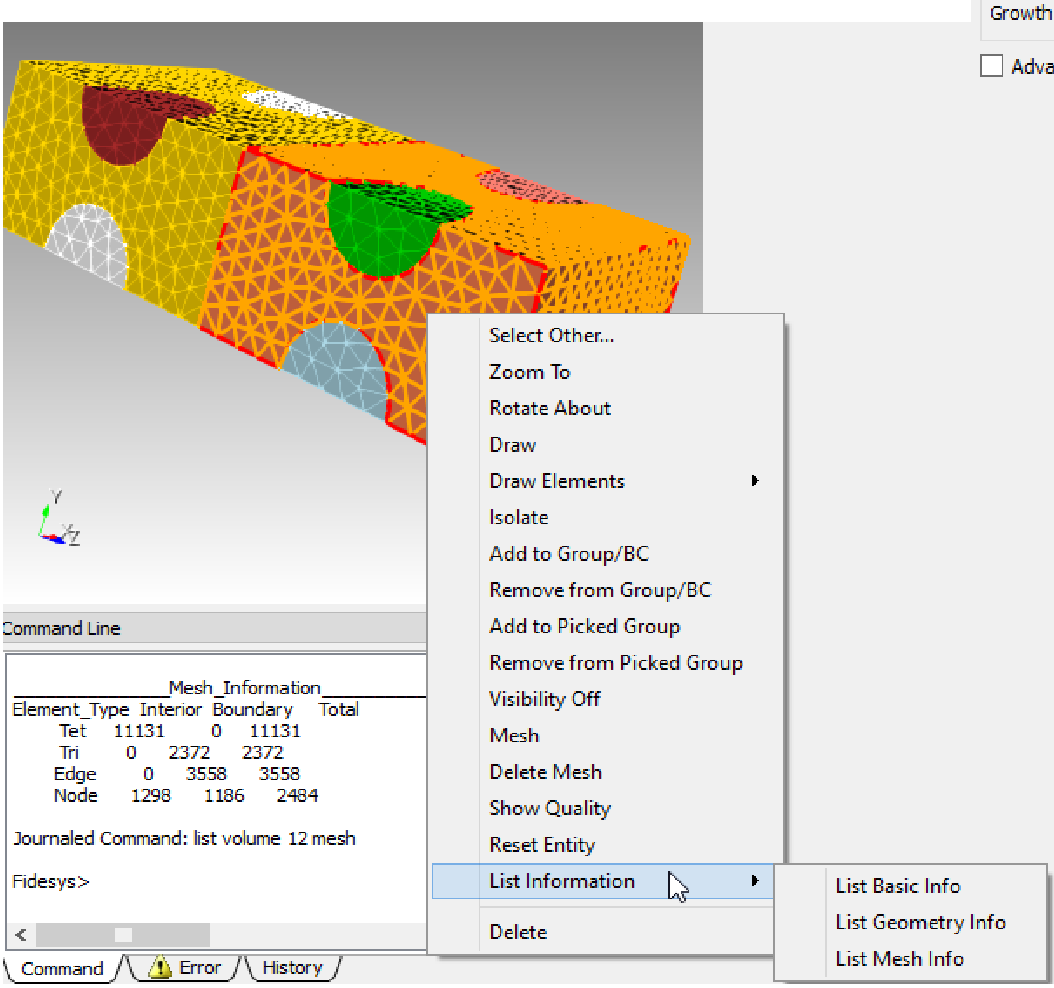
1. Select volume mesh generation section on Command (Mode — **Mesh**, Entity — **Volume**, Action — **Meshing**).
   - Select volumes (specify their ID): 5 6 7 8 9 10 11 12 *(or by the command **all**)*;
   - Select meshing scheme: Tetmesh;
   - Keep all other settings of the contact pair by default.

Click **Apply Scheme**.

Click **Mesh**.

To see the resulting number of items, you can do the following:

- Select the entire model (by left-clicking and pressing the *Ctrl* key);
- Right-click on the model;
- In the pop-up menu, select **List Information – List Mesh Info;**
- Information on the mesh will appear in Command Line



Setting material and element type

1. Create the cord material.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material1;
- Property group: Hook material;
- Young's Modulus: 200 000;
- Poisson's Ratio: 0.25.

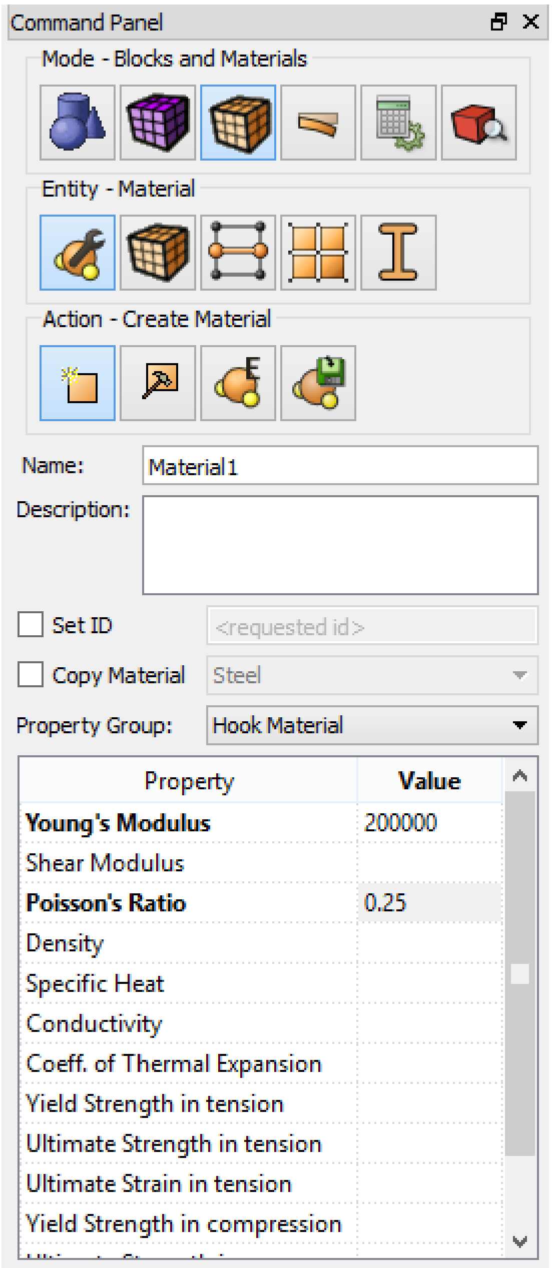
Click **Apply**.

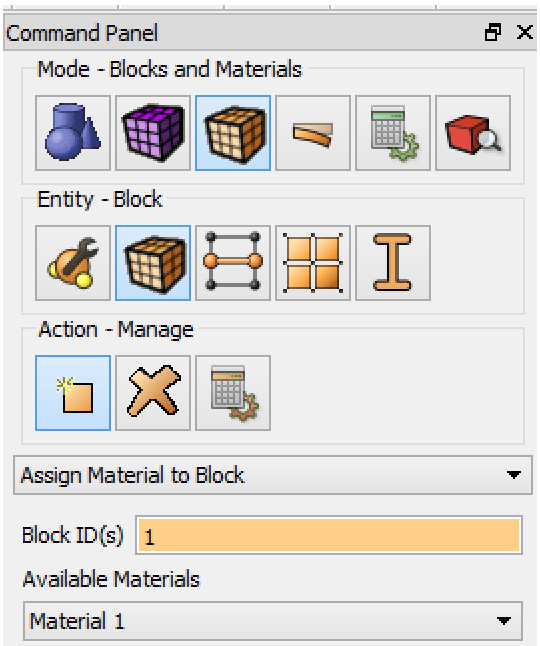
2. Create the material of matrix by analogy.

Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity –**Material**, Action – **Create Material**). Set the following parameters:

- Name: Material2;
- Property group: Hook material;
- Young's Modulus: 2;
- Poisson's Ratio: 0.49.

Click **Apply**.





3. Assign the thread material (Material1) to Block 1.

Block 1 was automatically created when generating the geometry for composite. Select setting the material properties section on Command Panel (Mode – **Blocks and Materials**, Entity – **Block,** Action – **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1;
- Select the previously created material in the list: Material1.

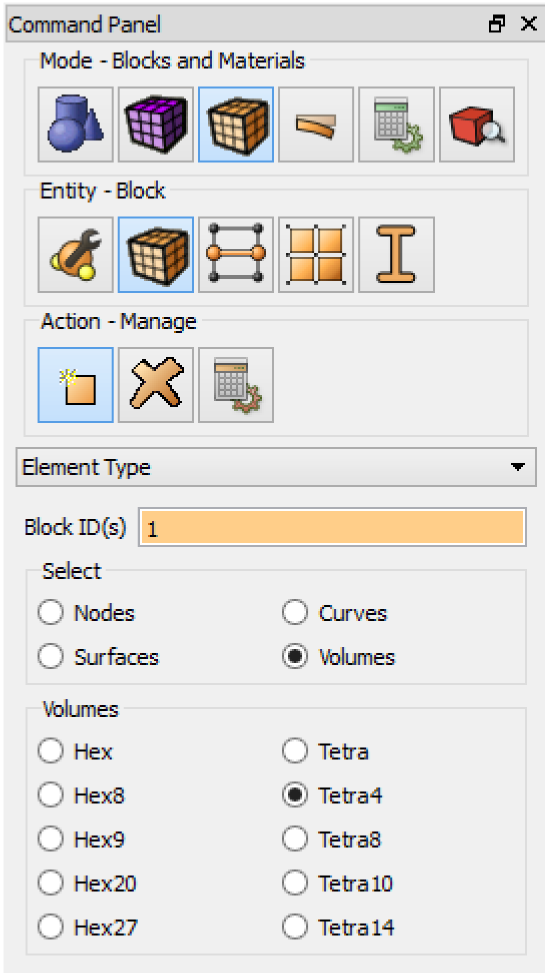
Click **Apply**.

4. Assign the matrix material (Material2) to Block 2.

Block 2 was automatically created when generating the geometry for composite. Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Assign Material to Block** in the list of possible operations. Set the following parameters:

- Block(s) ID: 2;
- Select the previously created material in the list: Material2

Click **Apply**.

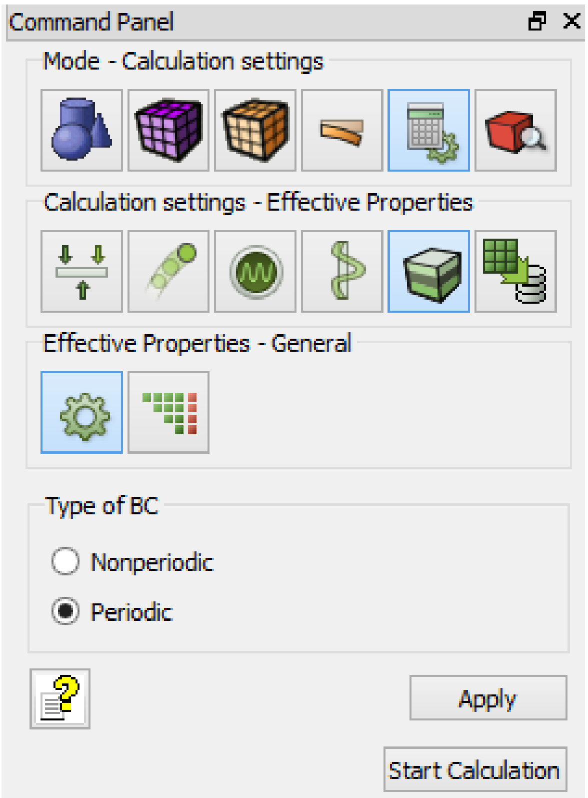


5. Assign the element type.

Select setting the material properties section on Command Panel (Mode — **Blocks and Materials**, Entity — **Block,** Action — **Manage**). Select **Element Type** in the list of possible operations. Set the following parameters:

- Block(s) ID: 1 2 (*or all*);
- Select: Volumes;
- Volumes: TETRA4.

Click **Apply**.



Setting boundary conditions and Starting calculation

1. Type of boundary conditions is set directly in the the calculation settings. Select Mode — **Calculation Settings**, Calculation Settings — **Effective properties**, Effective properties — **General** on Command Panel.

Select **Type of BC** on the Panel:

- Periodic.

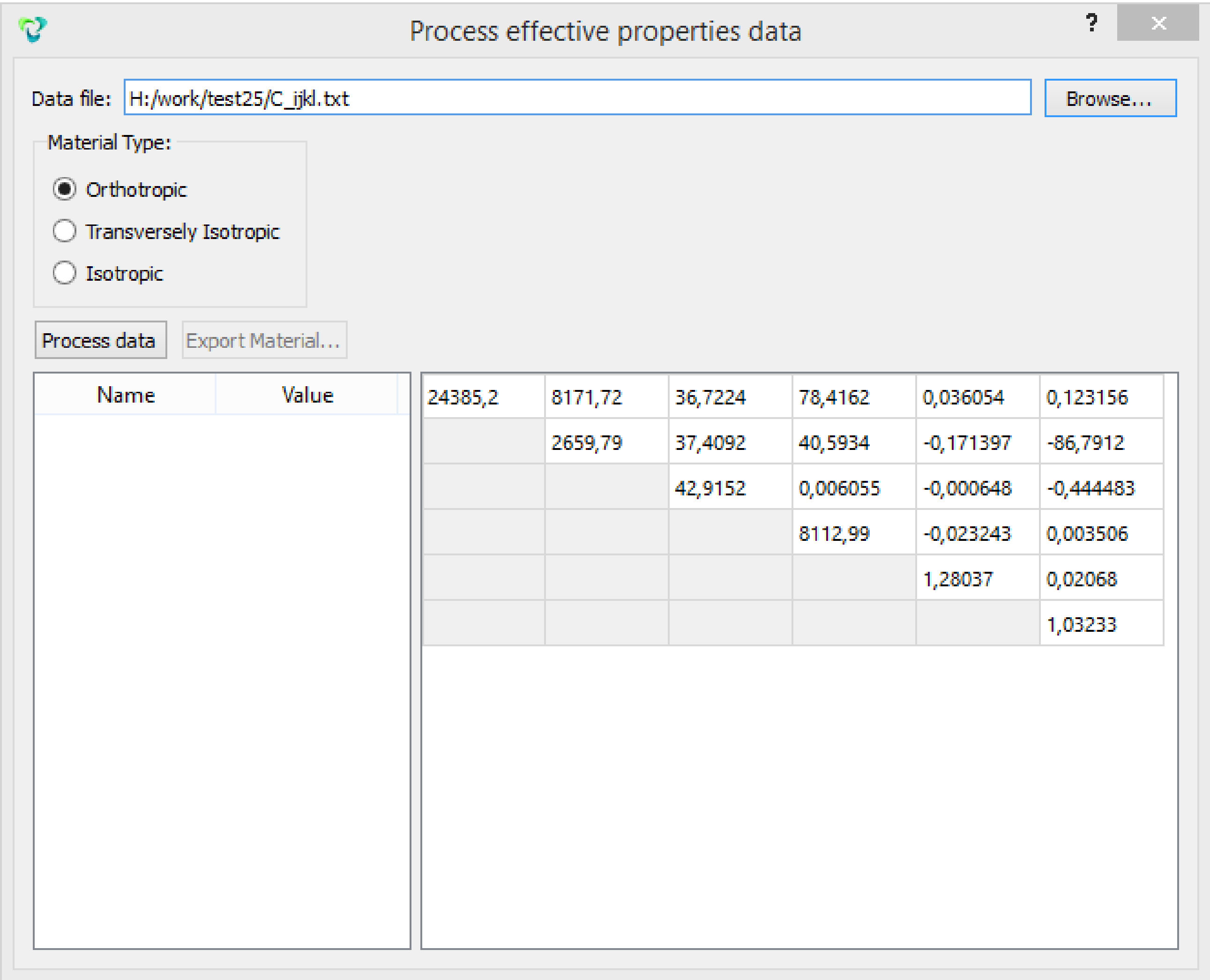
Click **Apply**.

Start the calculation by clicking **Start calculation**.

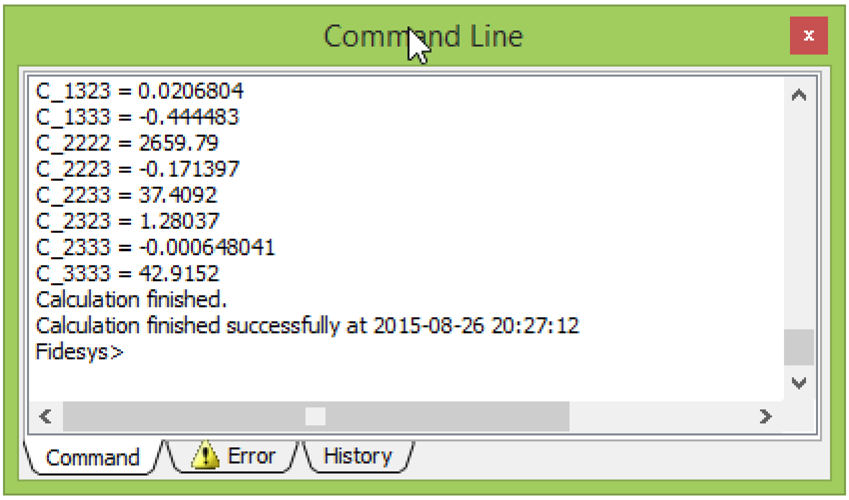
2. If the calculation is finished successfully, you will see a message in the Console: *"Calculation finished successfully at <date> <time>"*.

Results analysis

1. Calculation results of the composite effective properties will be displayed in the window **Process effective properties data**.



The calculation results are also displayed in Command Line.



2. Define the required material constants.

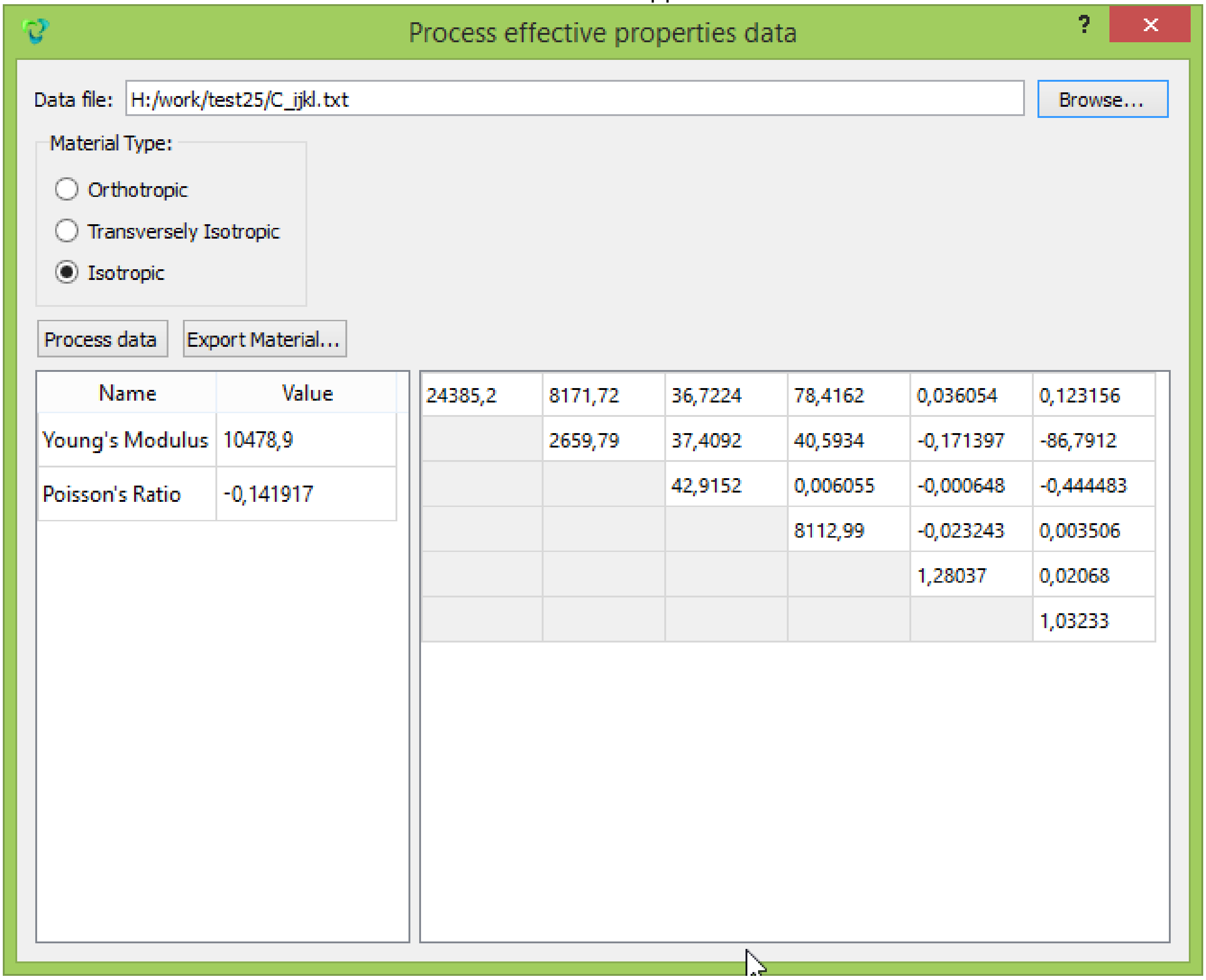
   - The difference between C_1111 = 24 496 and the original one 24 852.4 MPa is 1.5%;
   - The difference between C_1122 = 8 227.62 and the original one 8 281.54 MPa is 0.7%;
   - The difference between C_2222 = 2 825.56 and the original one 2 763.12 MPa is 2.2%;
   - The difference between C_1212 = 8 271.82 and the original one 8 283.5 MPa is 0.1%.

3. Process the received data.

   In the window **Process effective properties data**, select:
   - Material Type: Isotropic

   Click **Process data**.

   In the left column new material constants should appear.



4. Export of calculation results.

   To export the material into the XML file, select the button **Export Material...**.
   You must select a name for effective material and a name of the XML file to export to.

Using Console Interface

For geometry generation, meshing, setting boundary conditions and materials you can use Console Interface. The following program code allows performing the steps of the above-described guide, you only need to **manually specify the full path and name of the file to be saved**.
Here is an example of the working script (the file is saved in D:/FidesysBundle/calc/test.pvd)

```
reset
create brick width 16 depth 9.2376 height 16
create cylinder height 50.4752 radius 3
volume 2 rotate 90.0 about y
volume 2 rotate 30 about z
volume 2 move y -9.2376
create cylinder height 50.4752 radius 3
volume 3 rotate 90.0 about y
volume 3 rotate 30 about z
volume 3 move y 0
create cylinder height 50.4752 radius 3
volume 4 rotate 90.0 about y
volume 4 rotate 30 about z
volume 4 move y 9.2376
intersect volume 1 2 keep
intersect volume 1 3 keep
intersect volume 1 4 keep
delete volume 2
delete volume 3
delete volume 4
subtract volume 5 6 7 from volume 1 keep
delete volume 1
volume all move z 8
volume all move z 16 copy
volume 9 10 11 12 reflect 1.0 0.0 0.0
imprint volume all
merge volume all
block 1 volume 5 6 7 9 10 11
block 2 volume 8 12
volume all scheme Tetmesh
set tetmesher interior points on
set tetmesher optimize level 3 overconstrained off sliver off
set tetmesher boundary recovery off
volume all tetmesh growth_factor 1.0
delete mesh volume all propagate
volume all scheme Tetmesh
set tetmesher interior points on
set tetmesher optimize level 3 overconstrained off sliver off
set tetmesher boundary recovery off
volume all tetmesh growth_factor 1.0
mesh volume all
create material "Material1" property_group "CUBIT-FEA"
modify material " Material1" scalar_properties "MODULUS" 2e+05 "POISSON" 0.25
create material " Material2" property_group "CUBIT-FEA"
modify material " Material2" scalar_properties "MODULUS" 2 "POISSON" 0.49
block 1 material ' Material1'
block 2 material ' Material2'
analysis type effectiveprops elasticity dim3
nonlinearopts maxiters 100 minloadsteps 1 maxloadsteps 10 tolerance 1e-6
spectralelement off
usempi off
periodicbc on
calculation start path 'D:/FidesysBundle/calc/test.pvd'
```

Contacts

http://www.cae-fidesys.com

support@cae-fidesys.com

+7 (495) 930-87-53