UNIVERSAL MECHANISM 10

universal mechanism

Getting started

UM FEM

2024

Contents

Getting Started Using UM: Simulating Hybrid Models

The UM FEM additional module gives the user a possibility to create models of mechanical systems that include both rigid and elastic bodies, so called hybrid systems. Elastic displacements assumed to be rather small and describable by finite element method and linear theory.

This manual helps you to study main features of creating and analyzing hybrid systems using Universal Mechanism software. Detailed information about UM FEM you can find in the Chapter 11, file 11_UM_FEM.pdf of UM user's manual, which is available in the {UM Data}\MANUAL directory and in the Internet via this link: www.universalmechanism.com/download/90/eng/11_um_fem.pdf.

It is supposed that you already studied the gs_UM.pdf[1] manual, which is devoted to basics of UM modeling and know how to create new model, add new bodies and joints, generate and compile equations of motion **(UM Input)** and simulate mechanical systems **(UM Simulation)**.

The modal approach is used for simulation of dynamics of elastic bodies. This approach consists in presentation of elastic deformations with the help of a set of *eigenmodes* and *static modes*[2] . The approach assumes describing elastic bodies in terms of finite-element method in **ANSYS** software with subsequent export that data to **UM**. Thus, the necessary condition of using **UM FEM** is availability the **ANSYS** software for some preliminary analysis and calculations.

Every elastic body is considered as a separate subsystem. Data file of the elastic subsystem is a binary **input.fss** file. This file may be created with the help of **ANSYS_UM.EXE** program or with the help of Wizard of flexible subsystems in the **UM Input**. In the latter case **ANSYS_UM.EXE** creates intermediate **uminput.fum,** that contains input data for the **Wizard**.

After **ANSYS_UM.EXE** creates **input.fss** or **input.fum** files the subsequent preparing of the model is fulfilled with the help of Universal Mechanism. Since the data files about elastic body is exported from the ANSYS software and prepared by **ANSYS_UM** program ANSYS software is not used any more. Complete data flow from ANSYS to UM is shown in the eleventh part of UM user's manual (11_um_fem.pdf). Thus using **UM FEM** module is possible if ANSYS software is available on the user's computer.

Note.	(1) Before coming to the rest part of the manual please check if the **UM FEM** module is available on your computer. Run **UM Input** or **UM Simulation** and from the **Help** menu select **About…**. The list of available modules is shown in the **Configuration** section. (2) Please also check if the **ANSYS** software is available on your computer. If you do not have **ANSYS** on your computer you will have to leave some parts of this lesson, where working under **ANSYS** environment is considered. But nevertheless you will be able to complete the lesson using files prepared in advance.

[1] www.universalmechanism.com/download/90/eng/gs_um.pdf

[2] Please find more detailed information about static modes and eigenmodes in the eleventh part of UM user's manual (11_um_fem.pdf)

Copyright and trademarks

1. Slider-crank mechanism

Here the example model of the slider-crank mechanism (see Figure 1.1) is considered. There is **Slider_crank_all** model in the {UM Data}\SAMPLES\Flex directory. This model includes three slider-crank mechanisms. The difference between these models is in the way of representation of the con-rod. There are following cases:

- con-rod as a rigid body;
- con-rod as a system of eleven rigid bodies interconnected by revolution joints with damping and elasticity;
- con-rod as an elastic body according to UM FEM methodology, see Sect. 1.1.

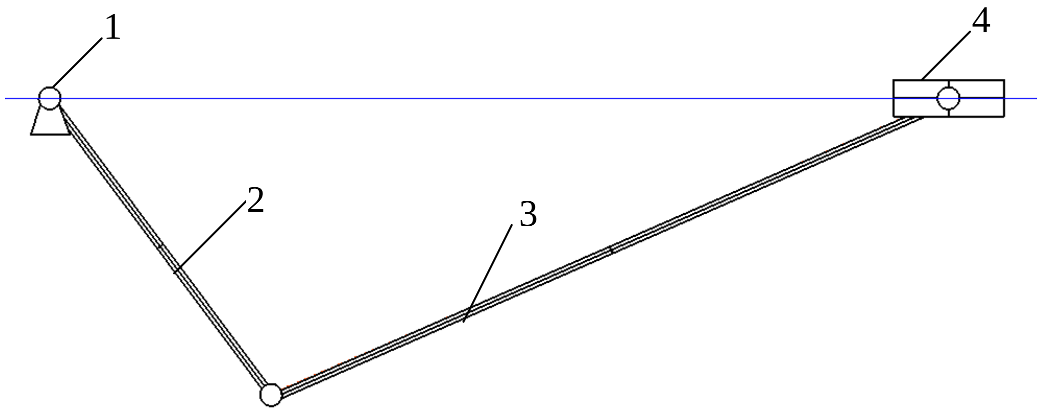


Figure 1.1. Slider-crank mechanism: 1 – base, 2 – crank, 3 – con-rod, 4 – slider.

The process of creating and simulating a hybrid model of the slider-crank mechanism with elastic con-rod is discussed in this section.

Preparing the model consists of the following steps:

1. describing FE model of the con-rod in **ANSYS**;
2. calculating elastic modes of the con-rod, saving data in UM format;
3. creating graphical objects;
4. describing bodies: crank and slider;
5. adding elastic con-rod;
6. creating joints and forces.

Steps 1-2 are done in under ANSYS environment, 3-6 – in UM.

Note.	UM uses subsystem technique to introduce elastic bodies into the model. Every elastic body is represented as a separate subsystem of **Linear FEM subsystem** type.

Create a directory for the future models, for example **{UM Data}\My Models** or **D:\models**. Within this section we address this directory as «.\». This directory will include two subdirectories:

- **flexbeam** for an elastic beam data;
- **slider_crank_fem** for the hybrid model.

You can read this manual more or less detailed. Please note the following remarks.

- If ANSYS software is available on your computer and you want to study all the data flow in details you should read this manual sequentially.
- If ANSYS software is not available or you want to omit the step of preparing data in ANSYS you can directly start from the Sect. 1.2.2 of this manual. Before that you should copy the {UM Data}\SAMPLES\Flex\flexbeam**\input.fum** to the **.\flexbeam** directory.
- You can omit all the steps of elastic body data preparing. Before that you should copy {UM Data}\SAMPLES\Flex\flexbeam**\input.fss** to the .**flexbeam** and start reading from the Sect. 1.3 of this manual.

1.1. Preparing ANSYS environment

We will use **ANSYS** software for preparing data for simulation of dynamics of elastic body. After creating FE model a calculation of the static and eigenmodes starts. Macro **um.mac** is used for such a calculation. Then **ANSYS_UM** program starts. This program translates data, that are produced by **um.mac** into **UM** format.

Copy the **um.mac** file from **{UM}\bin** to ANSYS default directory for macros. It is usually the **.\docu** directory in ANSYS 5.0, **.\apdl** in ANSYS 7.0 9.0 root directory. Otherwise you need to set search path with the ANSYS command

/PSEARCH,Path_to_macro

After preparing data the **um.mac** macros runs the external **ansys_um.exe** program for subsequent analysis of obtained data. The **ansys_um.exe** is situated in the **{UM}\bin** directory. You need to open the **um.mac** in any text editor and edit the path to the **ansys_um.exe** program in the last line of the macros. Set full path to the **ansys_um.exe** as the parameter of the **/sys** command. For example,

/sys,c:\um\bin\ansys_um.exe

Note 1.	If the full path to the **ansys_um.exe** program contains space(s) then use inverted commas. For example, */sys,"c:\universal mechanism\bin\ansys_um.exe"*
Note 2.	Path to the **ansys_um.exe** program should contain the Latin letters only.

1.2. Preparing con-rod as an elastic beam

As it mentioned above, preparing data for introducing elastic bodies into hybrid models contains the stage of solution of eigenvalues problem. There are two possible mathematical formulations of this problem:

- with diagonal mass matrix;
- with consistent mass matrix.

The {UM Data}\SAMPLES\Flex\flexbeam\input directory contains two subdirectories: **lumped** and **consistent**. The first one includes an ANSYS command file for the case of diagonal mass matrix, the second one – for consistent mass matrix.

In the manual we will consider the case with diagonal mass matrix.

1.2.1. Working under ANSYS environment

1. Copy the **flexbeam&mass21.ans** file from the {UM Data}\SAMPLES\Flex\flexbeam\input\lumped directory to the **.\flexbeam** directory. This file is the ANSYS command file, uses APDL language and describes the process of ANSYS model creation. This file also contains comments that explain every step of the process.
2. Run **ANSYS Interactive** and select the **.\flexbeam** directory as working directory and set **Working directory** to **.\flexbeam**, for example d:\models\flexbeam.
3. Run **ANSYS**. From the **File** menu select **Read Input from** and choose **.\flexbeam&mass21.ans.** Steel beam of 2 m length and square cross section with 2 cm width is created. Finite element model consists of 100 elements of BEAM4 type and 200 elements of MASS21 type. Two end nodes are automatically selected as **interface nodes**[1]. If you made all setting ANSYS environment correctly then the **um.mac** macros is started automatically and calculates 12 *static modes* and 10 *eigenmodes* of the beam.
4. If you changed path to the **ansys_um.exe** program in **um.mac** properly then **um.mac** runs **ansys_um.exe** automatically. Otherwise run the **{UM}\bin\ansys_um.exe** manually. The main window of **ansys_um** appears, Figure 1.2.
5. Point to the **General** tab. The **ANSYS results file (*.rst)** set to **.\flexbeam\flexbeam.rst**, **Target directory** set to **.\flexbeam**, see Figure 1.2.

[1] More detailed information about interface nodes you can find in the eleventh part of UM User's Manual

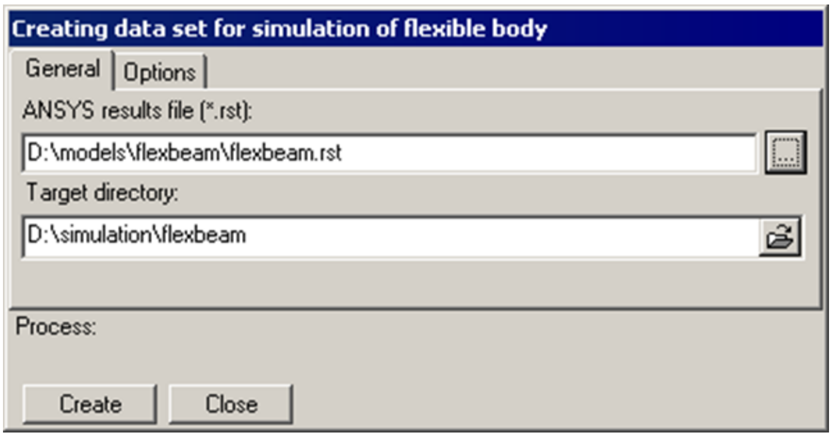


Figure 1.2. Main window of the ANSYS_UM program.

6. Point to the **Options** tab and turn off the **normalize modes** check box, Figure 1.3. This case corresponds to creating the intermediate **input.fum** file. On the successive step we will use the **Wizard of flexible subsystems** to convert the data into UM-compatible form.

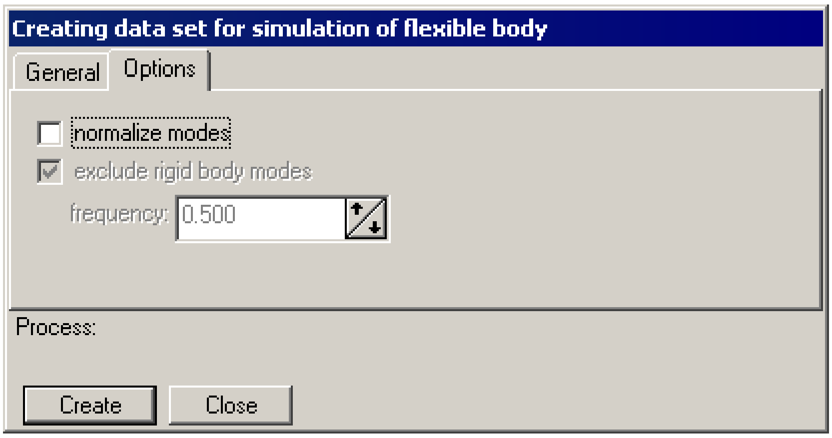


Figure 1.3.

Note. Using the **Wizard of flexible subsystems** is not necessary step of the creation of the model. However it seems to be very important for your understanding UM that you go through the **Wizard**.

It possible to prepare all necessary data with the help of **ANSYS_UM** program only. To do this you should turn on **modes normalize** and **exclude rigid body modes** check boxes and set frequency. In this case the **input.fss** file will be created. Please read eleventh part of UM User's Manual for more detailed information.

7. Click the **Create** button. Calculations will take some time. The **.\flexbeam\input.fum** file will be created as a result.
8. Click the **Close** button.

1.2.2. Wizard of flexible subsystems

During the next step we will use the wizard of flexible subsystems data. It is a tool for animation of elastic modes, and exclusion of some of them.

Note. Using the wizard of flexible subsystems data is not an obligatory phase. Preparing the data can be fulfilled with the help of **ansys_um** program. To do this point to the **Options** tab and turn on the **normalize modes** and **exclude rigid body modes** check boxes and set **frequency** value, Figure 1.3. Nevertheless now we will use the wizard of flexible subsystem data in order to familiarize you with it.

The intermediate **input.fum** file contains static modes and eigenmodes. To finish preparing data it is necessary to orthogonalize modes. It may be done directly in the **ansys_um** program and if necessary with the help of wizard of flexible subsystems data.

1. Run **UM Input** program (**uminput.exe**).
2. Click the **Tools | Wizard of flexible subsystems** menu item. The main window of the wizard of flexible subsystems data appears.
3. Click the  and select a file for the **Data file**, Figure 1.4, Figure 1.5.

Figure 1.4.

Wizard loads and shows the data, Figure 1.6. The **General** tab shows summary information about elastic subsystem, see Figure 1.6.

The **Position** tab (see Figure 1.7) is used for setting position and orientation of the elastic body. These transformations influence on the representation of the elastic body in the animation window of the wizard. Flexible body in the starting position coincides with X-axis that is not really comfortable to watch. Now we will shift the beam along Z axis with 0.3 m.

4. Point to the Position tab.
5. Set Shift | z to 0.3, see Figure 1.7.

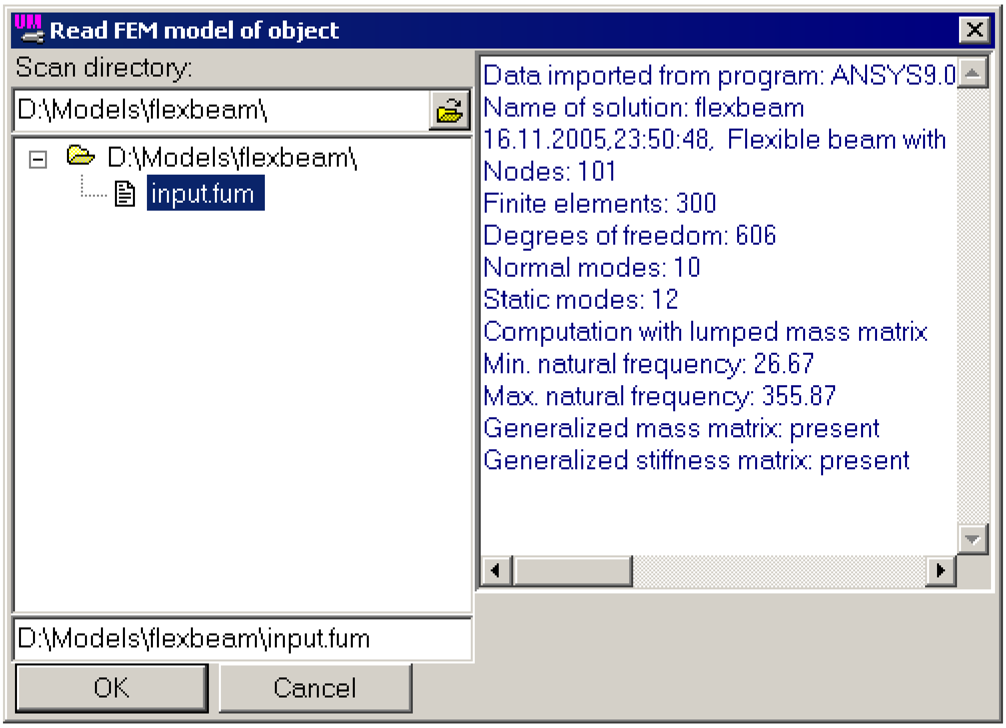


Figure 1.5.

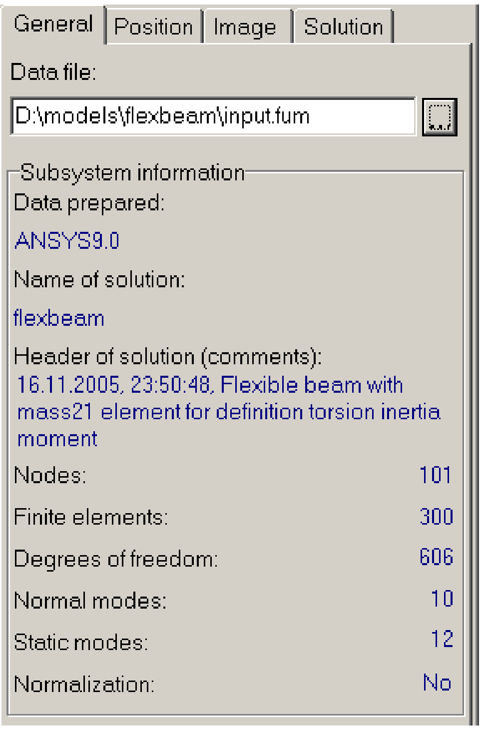


Figure 1.6.

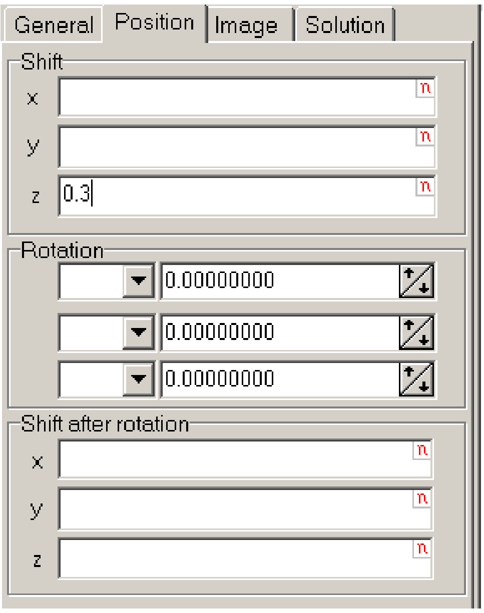


Figure 1.7.

Using the **Image** tab we can change graphical representation of the FE model. There are two modes of such a representation: simplified and full. During the full model status line shows the information about nodes and finite elements when mouse cursor is on it. However the full mode takes more CPU time to animate.

6. Set **Image** to **full**.
7. Turn off the **Image parameters | Draw nodes** check box.
8. Set the rest parameters according to the Figure 1.8.

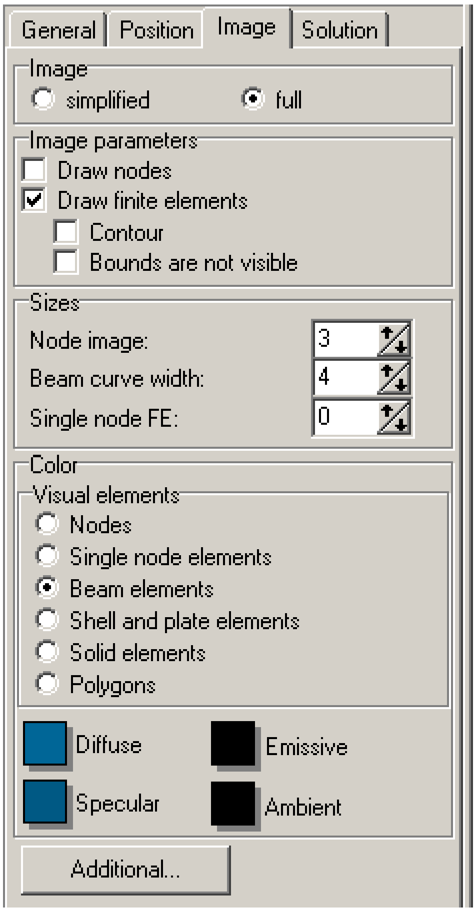


Figure 1.8.

Note. Single node finite elements of the **MASS21** type are used for setting moment of inertia of the body relative to the longitudinal axis. Set **Sizes | Single node FE** to **0** in order to hide such elements and make the image clearer.

The **Solution** tab gives you a possibility to animate modes of elastic subsystem. To start animation you should click the **Animate** button, see Figure 1.9. You can control this animation with the help of **Amplitude** and **Rate** track bars.

You can include/exclude any form from the final set of modes turning on/off the corresponding check boxes in the **Modes** tab. The more modes you include in the final solution and the more frequency these modes have the more accurate and time-consuming subsequent numerical integration you have. Generally it is recommended to turn on/off modes to keep a balance between solution accuracy and time efforts for it.

Thus, you can fulfill the only calculation in the ANSYS software with the maximum modes you will ever use (10 in this example) and then form various sets of modes with the help of the **Wizard of flexible subsystems** data.

Leave the initial set of modes without any changes.

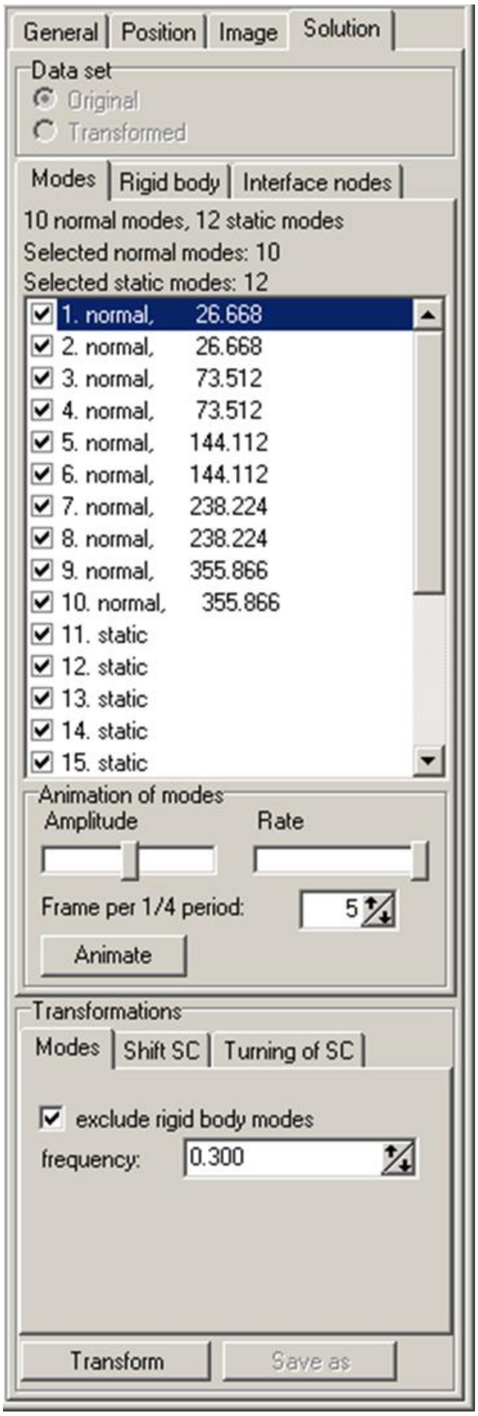


Figure 1.9.

9. Turn on the **Transformations | Exclude rigid body modes** (Figure 1.9).
10. Set **Transformations | Frequency** to **0.3** (Figure 1.9).
11. Click the **Transform** button and confirm this action in the subsequent dialog.

As a result the transformed set of modes of elastic body is created. In the case of successful execution of the transformation the following message appears, see Figure 1.10.

Figure 1.10.

Note. The initial set of modes includes rigid body modes, which should be excluded according to the used approach for simulation. Rigid body modes theoretically correspond to zero frequencies, but in fact because of using numerical methods and round-off errors these frequencies are small and close to zero but not exact zero.
In fact the **Transformations | Frequency** field indicates the threshold value and all frequencies that are less than this value are supposed to correspond to rigid body modes.

Now we need to save the transformed data set.

12. Point to Transformed in the Data set group, Figure 1.11.

Figure 1.11.

13. Click the **Save as** button. In the dialog set **Path to subsystem data** and click the **Save** button, see Figure 1.12. Please, note, that the latter directory will further serve as a subsystem name.

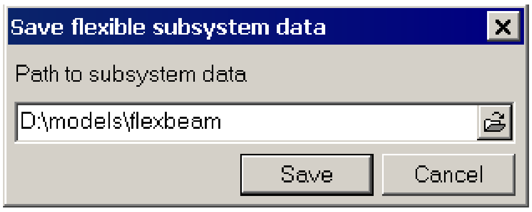


Figure 1.12.

Preparing the data for flexible subsystem is done.

1.3. Creating the model

The hybrid model of the slider-crank mechanism includes two rigid bodies, one elastic body and four joints.

Bodies:

- crank, 1 m length;
- con-rod, 2 m length;
- slider.

The crank and the slider are rigid bodies, con-rod is elastic subsystem (in terms of UM).

Joints:

- revolution joint between *Base0* and the crank, crank and the con-rod, and the con-rod and the slider;
- translational joint between slider and *Base0*.

1. Create a new model. Point the **File | New object** menu command or click the button. New constructor window appears.

1.3.1. Creating graphical objects

1. Load a graphical object from the **{UM Data}\graph**\Base1.umi file using  button or **Edit | Read from file…** menu item. Element «NoName» will be added to the list of graphic elements, see Figure 1.13.

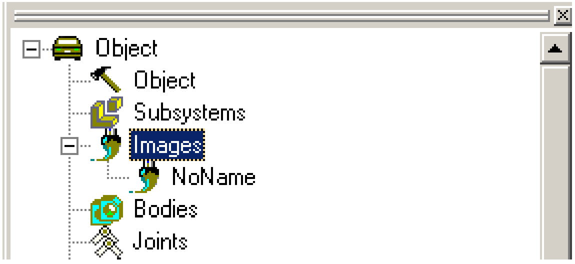


Figure 1.13.

2. Select this element and set name to **Base0** in the data inspector (Figure 1.14).

Figure 1.14.

3. Repeat these actions for **Crank1.umi** and **Slider1.umi** files, which are located in the directory **{UM Data}\graph**. Set the names **Crank** and **Slider** to created graphical objects correspondently.

Thus, three graphical objects are created.

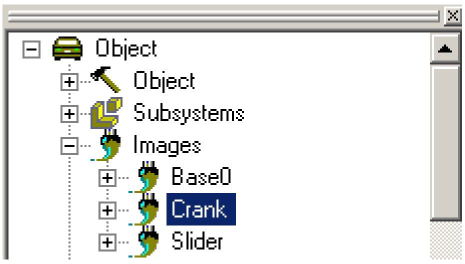


Figure 1.15.

4. Select **Object** item in the tree of elements and set **Scene image** to **Base0**, see Figure 1.16.

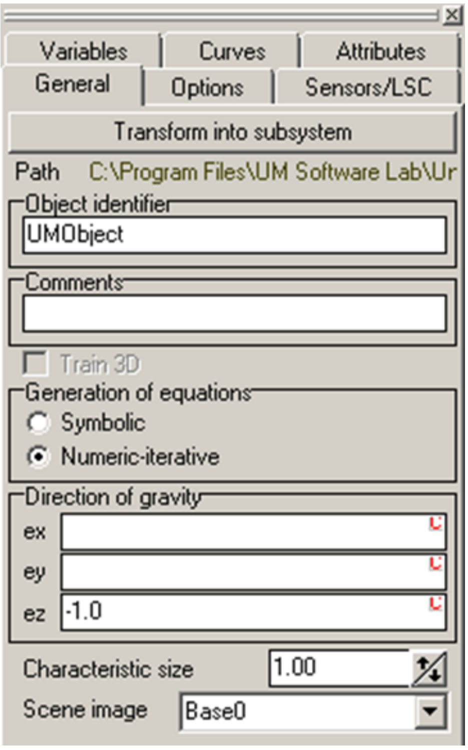

Variables
Curves
Attributes
General
Options
Sensors/LSC
Transform into subsystem
Path C:\Program Files\UM Software Lab\Ur
Object identifier
UMObject
Comments
Train 3D
Generation of equations
Symbolic
Numeric-iterative
Direction of gravity
ex
ey
ez -1.0
Characteristic size 1.00
Scene image Base0

Figure 1.16.

1.3.2. Creating rigid bodies

Here we create slider and crack as rigid bodies, set graphical objects for them and set their inertia parameters.

1. Select **Bodies** in the tree of elements.
2. Add two new bodies.
3. Rename bodies with **Slider** and **Crank** and set the correspondent graphical objects (Slider and Crank).
4. Select the **Parameters** tab and turn on the **Compute automatic** flag for the both of bodies. Inertia property of the bodies are computed automatically, see Figure 1.17.

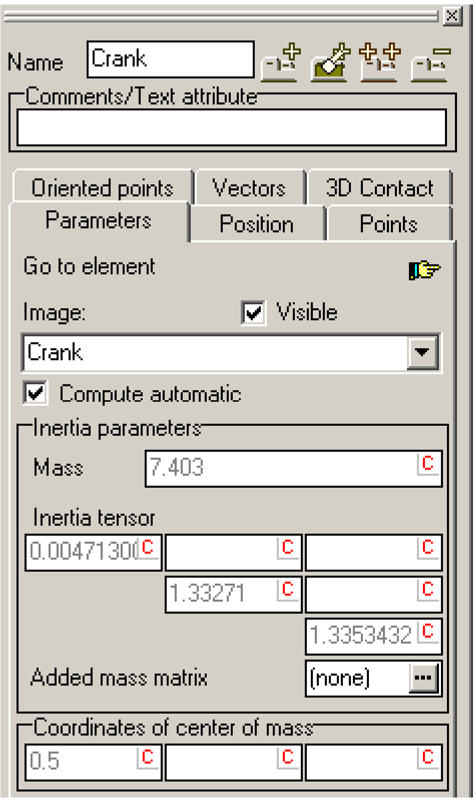


Figure 1.17.

1.3.3. Creating elastic subsystem

Now we introduce the elastic con-rod in the model. Every elastic body within a hybrid model is represented as elastic subsystem.

1. Select the **Subsystems** item of the tree of elements and create new subsystem using the  button.
2. In the **Type** select «**Linear FEM subsystem**» and choose the **.\flexbeam** directory in the open dialog window.
3. Set **Name** to **Con-rod FEM** (Figure 1.18).

After reading elastic subsystem data inspector looks like the wizard of flexible subsystem data described in the Sect. 2.1.4. There are following differences between wizard of flexible subsystem and the window of elastic subsystem data.

- You cannot changes set of modes in the window of elastic subsystem data since all data is already prepared.
- The **Position** tab influences to the real position and orientation of the elastic body in contrast to wizard of flexible subsystem where **Position** tab influences on the graphical representation of the body.

Elastic modes of the subsystem you can see using the **Solution | Modes** tab.

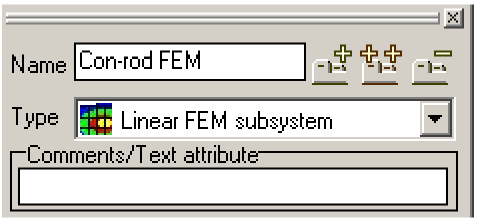


Figure 1.18.

1.3.4. Creating joints

Let’s create the first joint – revolution joint between *Base0* and the crank.

1. Select **Joints** item of the tree of elements. Add new joint.
2. Rename the joint to **jBase0_Crank**. Select **Rotational** type for the joint and set Y axis as **Joint vectors**, see Figure 1.19.

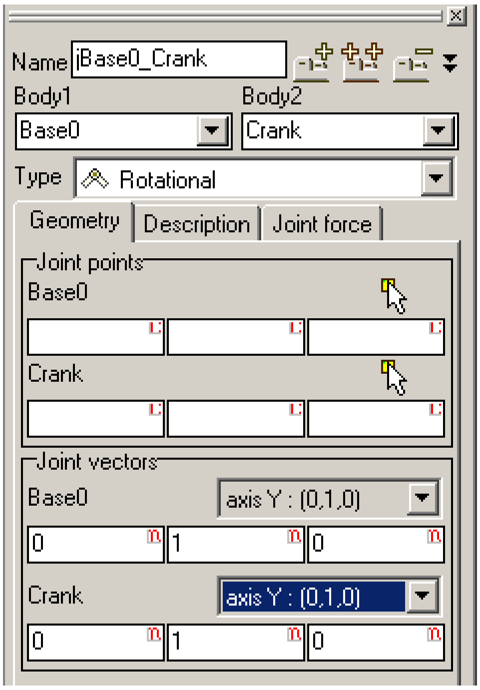


Figure 1.19.

3. Select the Joint force tab, set Joint torque to **Expression** and in the field Description of force set **F= torque-cdiss_crank*v**, see Figure 1.20. Press Enter. The window **Initialization of values** for new identifiers appears. Set identifiers value as follows: **torque = 100, cdiss_crank = 10**.

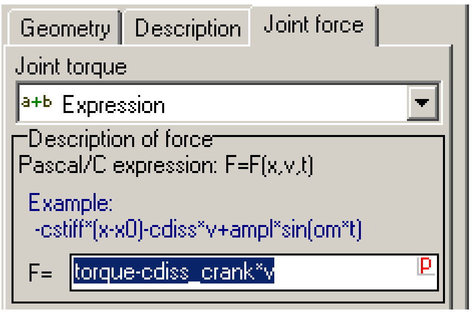


Figure 1.20.

4. Add the rest three joints as it is shown in the figure 1.21.

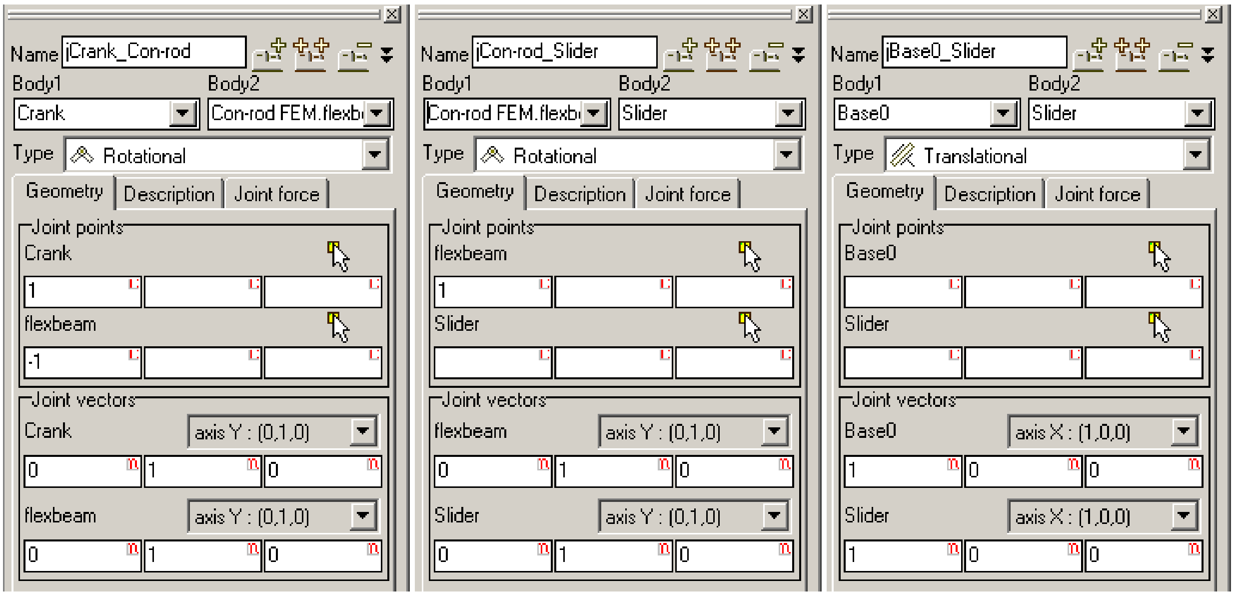


Figure 1.21.

1.3.5. Preparing for simulation

1. Save the model as **Slider_crank_fem (File | Save as…** menu command), see Figure 1.22.

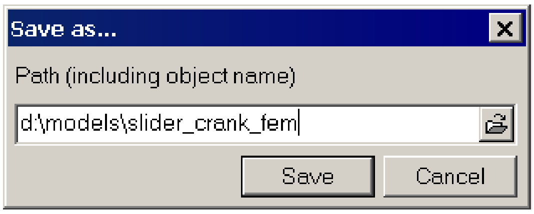


Figure 1.22.

2. Please check that **numeric-iterative** method of **generation of equations** is selected (see **Object | General** tab).

Now the model is ready for simulation.

1.4. Simulation

1. Use the menu command **Object | Simulation…** to run **UM Simulation** program. Main window of the **UM Simulation** program appears.

Let's obtain reaction forces in the joints **jCrank_Con-rod** and **jCon-rod_Slider**.

2. Open new **animation window**.
3. From the **Analysis** menu select **Simulation**. **Object simulation inspector** appears. Select the **FEM subsystems | Image** tab to set up animation parameters of the elastic con-rod as you want.

Now we will calculate initial conditions.

4. In the **Object simulation inspector** select the **Initial conditions tab**. Select the **Con-rod** subsystem in the drop down list, Figure 1.23. An anchor sign means that the correspondent degree of freedom is frozen. In this example it means that the elastic degrees of freedom will not be changed during calculation of initial position.

Note. If the **Initial condition** tab differs to the Figure 1.23 set the anchors manually.

5. Make sure that the **Autocalculation of constraint equations** mode is turned on (the  button should be pressed), otherwise press this button. Then calculate the initial conditions by clicking the button. Animation window shows the current position of the mechanism, Figure 1.24.

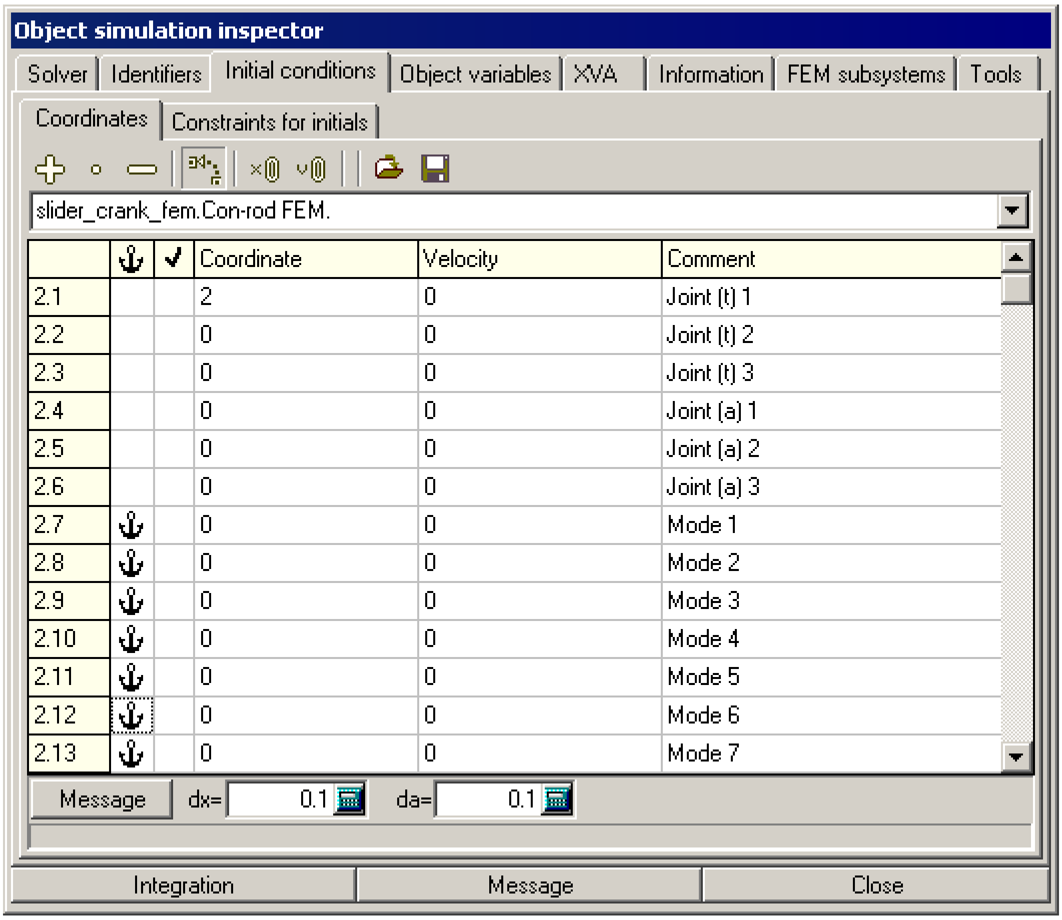


Figure 1.23.

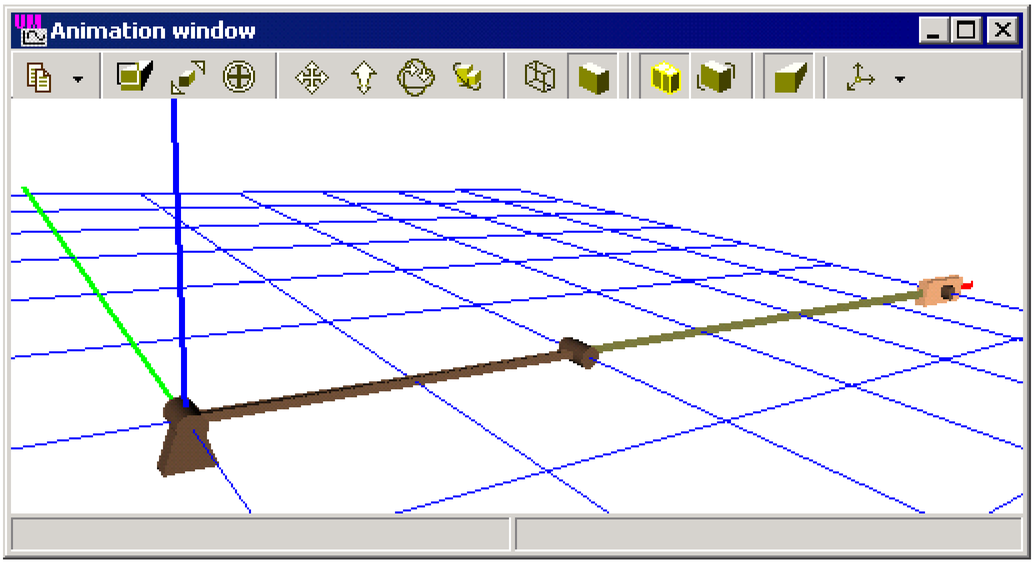


Figure 1.24.

6. Open new **graphical window (Tools | Graphical window** menu command).
7. Run **Wizard of variables** and create variables to determine absolute values of reaction forces in joints **jCrank_Con-rod jCon-rod_Slider** (Figure 1.25) and drag them to the graphical window.

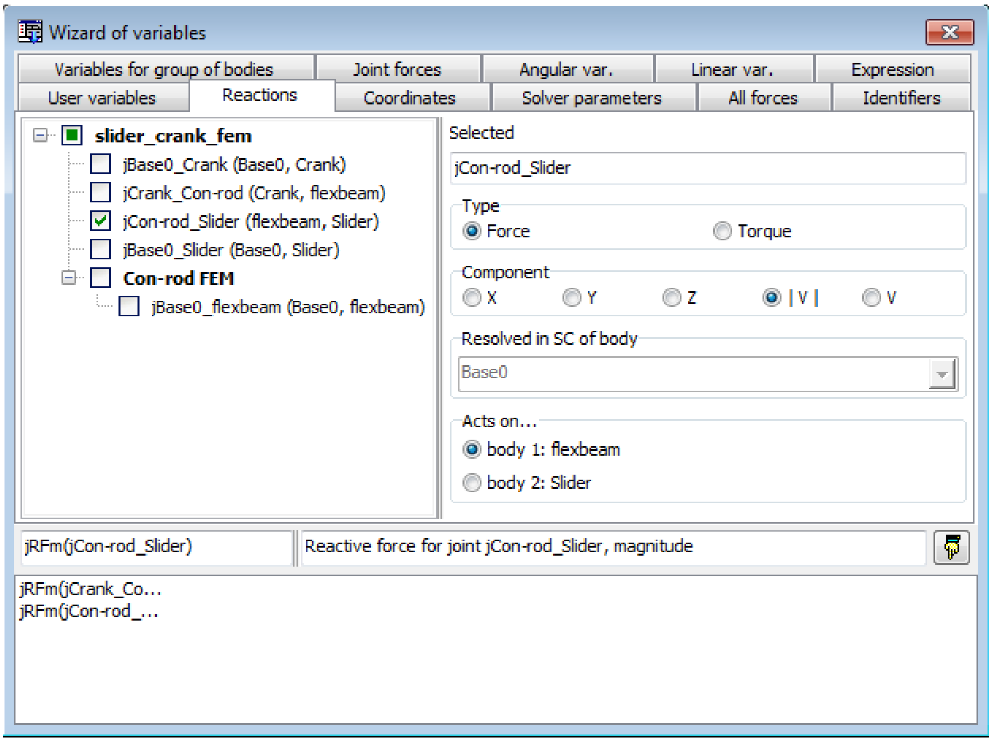


Figure 1.25.

8. Select the **Object simulation inspector** and point to the Solver tab. Set the following parameters:

- **Solver = Park,**
- **Type of solving = Range Space Method,**
- **Simulation time = 2.0.**
- **Step size for animation and data storage = 0.001.**
- **Error tolerance = 1E-7.**
- **Computing Jacobian matrices = on (always default).**
- **Block-diagonal matrices = off.**

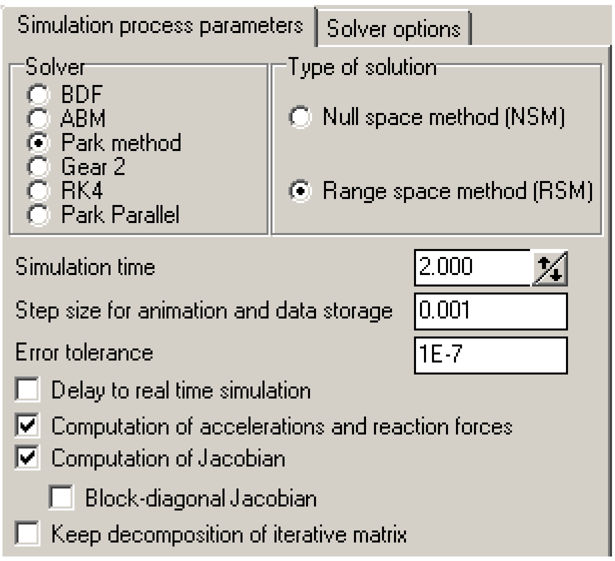


Figure 1.26.

9. Select the **FEM Subsystems | Simulation** tab and set up all options according to Figure 1.27.

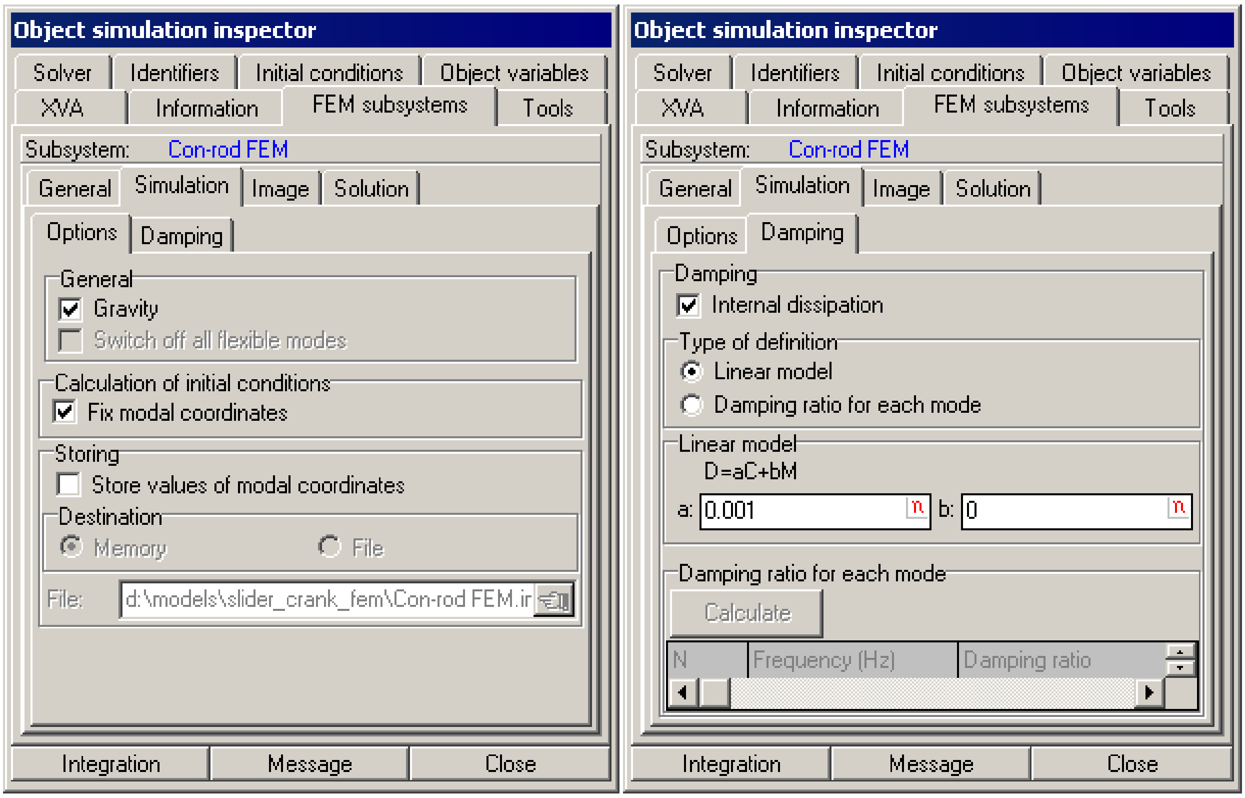


Figure 1.27.

10. Start simulation (**Integration** button).

You can see movement of the mechanism in the animation window (see Figure 1.28) and oscillograms of reaction forces in the graphical window (see Figure 1.29).

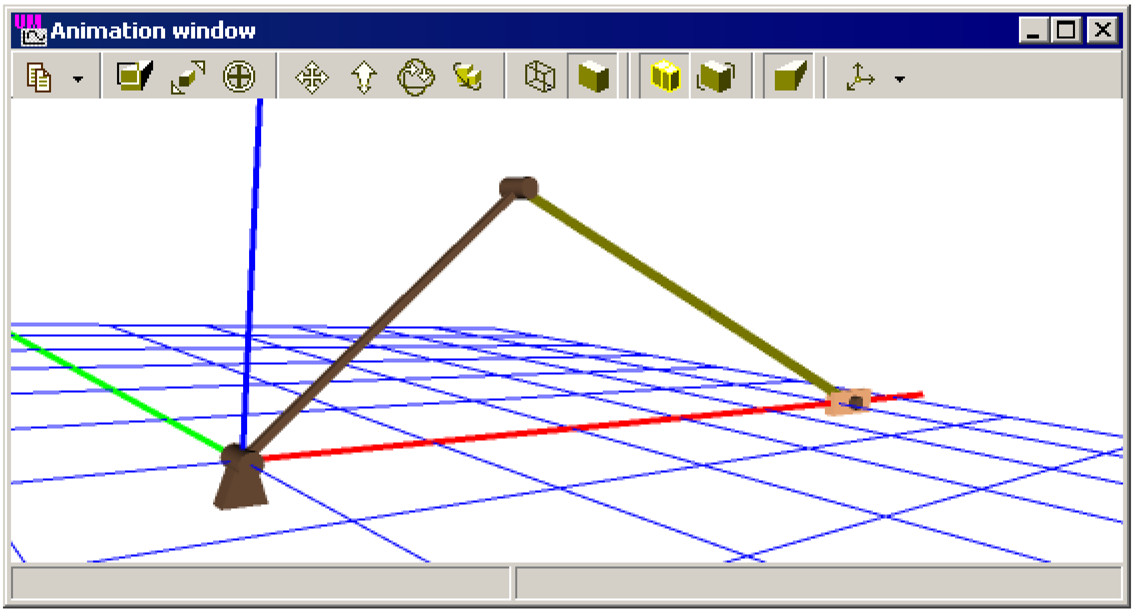


Figure 1.28. Animation window

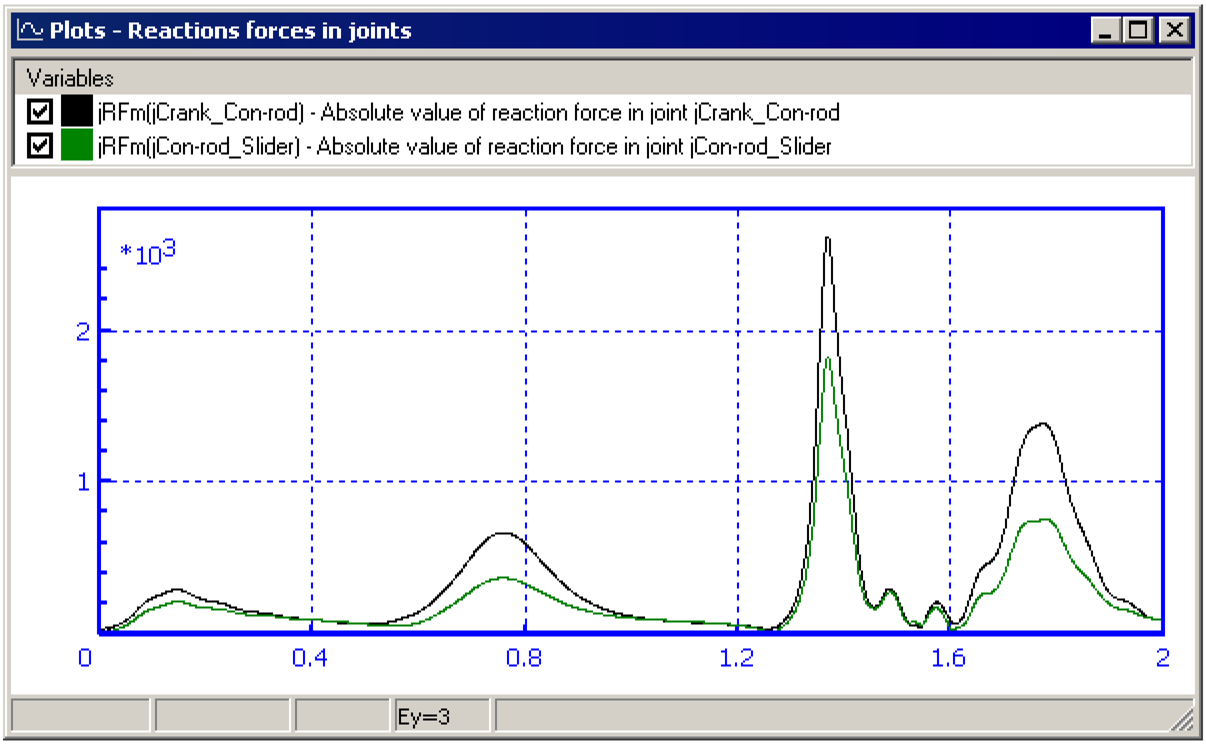


Figure 1.29. Graphical window

In order to estimate the influence of the elastic con-rod instead rigid one, open the {UM Data}\SAMPLES\Flex\Slider_crank_all model. Graphs of the reaction force are shown in the Figure 1.30.

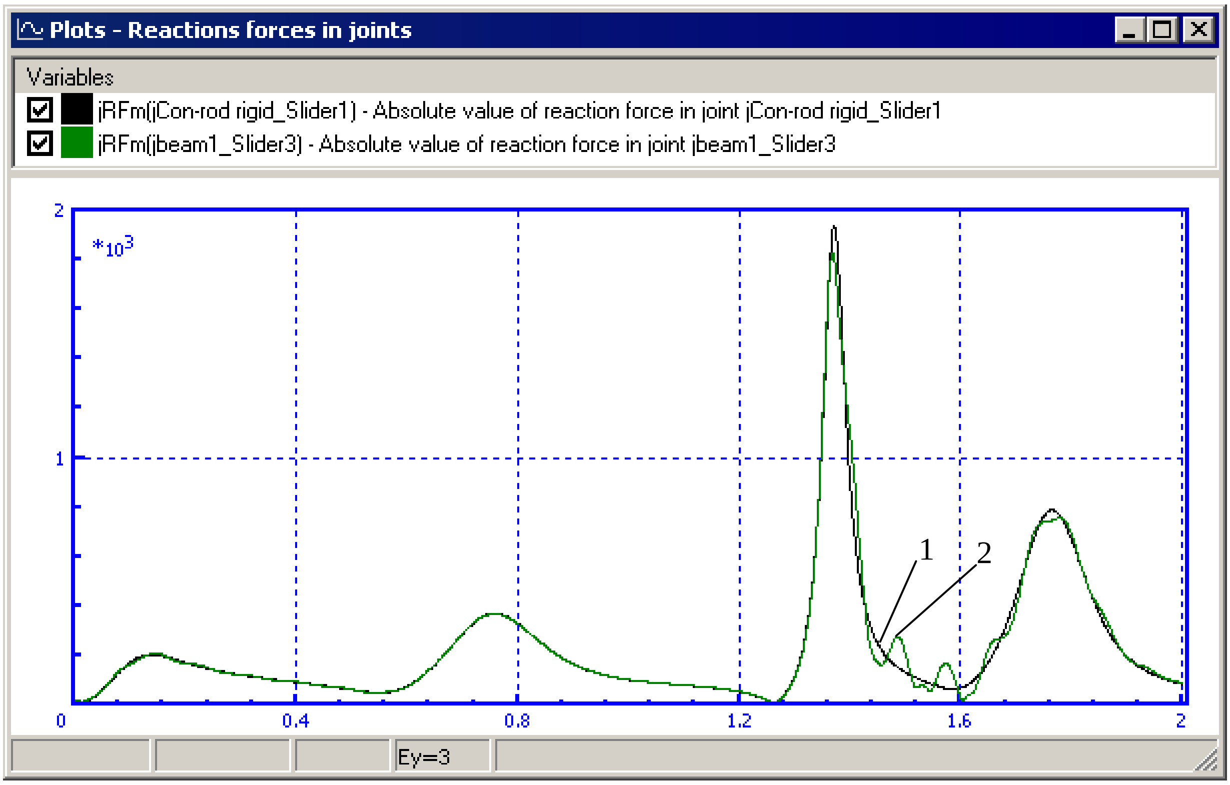


Figure 1.30. Reaction force in the Con-rod _Slider joint 1 – con-rod is a rigid body, 2 – con-rod is an elastic body.

Configuration file **example.icf**, which is situated in the **Slider_crank_all** directory, contains graphical windows with reaction forces in the rest joints of the model, as well as angular velocities of all cranks.

2. Electric motor on elastic platform

Let us consider step by step dynamical analysis of a mechanical system that consists of an electric motor and an elastic platform, Figure 2.1.

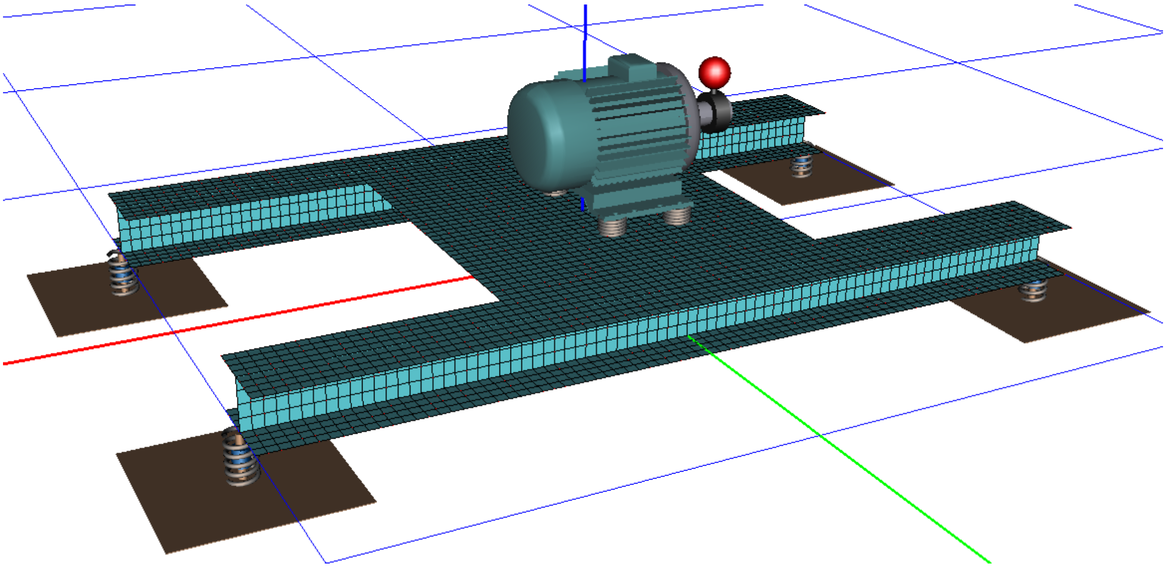


Figure 2.1.

The elastic platform is connected to a ground with the help of four visco-elastic linear force elements. The electric motor is included to a model as an external subsystem and is also connected with the help of four visco-elastic linear force elements, Figure 2.1. An eccentric is attached to a rotor of the electric motor. This eccentric produces forced oscillations of the platform.

Basic features of the description of the model and its dynamical analysis is considered in this section.

During the simulation we will analyze the following dynamical properties of the system:

- forces in the force elements;
- vertical displacements and accelerations of the platform in the center part under the motor.

Here we will simulate the following sequence of operation modes:

- running of the rotor from ω=0 up to its nominal angular velocity.
- operating duty;
- stop way – decreasing angular velocity of a rotor till ω=0.

Preparing the model includes the following steps:

- preparing data of the elastic platform;
- introducing FE model of the platform into the final UM-model;
- attaching the elastic platform to a ground;
- creating the model of the electric motor;
- introducing the electric motor into the final model as an external subsystem;
- attaching the electric motor to the platform with the help of visco-elastic elements.

Let us consider all of the described above steps in details. At that main attention will be put to the features that were not considered in the previous section.

It supposes that you already finished the previous section that is why some comments here are given shortly.

Please choose an existing or create a new directory for the future model. Within this section we will address this directory as «.\». Create two subdirectories:

- **.\Vibrostand** for the final composite model;
- **.\Vibrostand\Platform** for elastic platform.

2.1. Preparing elastic platform

In terms of Universal Mechanism software every elastic body is considered as a separate subsystem of **Linear FEM subsystem** type. Standard save file for such a subsystem is **input.fss** file. Preparing the elastic platform includes the following steps:

1. description the FE model of the platform in **ANSYS** software;
2. calculation of the elastic modes and export result from **ANSYS** in UM format.

There are two possible ways to fulfill the second step:

3. generate the **input.fss** file directly by **ANSYS_UM.EXE** program;
4. firstly generate the intermediate **input.fum** file by the **ANSYS_UM.EXE** and then complete data transformations with the help of **Wizard of flexible subsystems** that is a tool within the **UM Input** program. This wizard gives the user a possibility to visualize calculated elastic forms and exclude some modes from the final set of elastic modes (**input.fss**).

There are three files in the {UM Data}\SAMPLES\Flex\platform: **input.fss, input.fum** and **platformshell63.ans**.

- If you want to omit the step of preparing the data in **ANSYS** but familiarize yourself with **Wizard of flexible subsystems** you should copy the **{UM Data}\Samples\Flex\input.fum** file to the **.\platform directory** and go to the Sect. 2.1.4 of this manual.
- You may omit all the steps of creating the data of elastic platform, in this case you should copy the {UM Data}\SAMPLES\Flex\platform**input.fss** file to the **.\platform** directory and go to the Sect. 2.1.4 of this manual.

2.1.1. Working under ANSYS environment

Before you come to the next step please repeat all the steps from the Sect. 1.1.

Now we will create the FE model of the platform and export the data for the subsequent using them under UM environment.

5. Copy the **platformshell63.ans** file from the {UM Data}\SAMPLES\Flex\platform directory to the .\platform directory. This file contains APDL commands that automatize creating the FE model of the platform.
6. Run **ANSYS Interactive** and select the **.\platform** directory as a **working directory**.
7. Run **ANSYS**.
8. From the **File** menu select the **Read Input from** and open the **platformshell63.ans** file. As a result a steel platform that is consists of two beams of 1m length and a shelf between them.

This finite-element model includes 4224 elements of SHELL63 type. Width of all elements is 5 cm. You can open **platformshell63.ans** in any text editor and change some of parameters of the FE model, see comments in the body of this file. Four nodes, where the platform is connected with the ground, are selected as interfaced nodes. In the end the **um.mac** is run. If the **um.mac** is not run automatically you should run it manually, see Sect. 1.1. As a result of the **um.mac** execution 24 static modes and 10 eigenmodes are calculated.

9. If the path to the **ANSYS_UM.EXE** in the **um.mac** is set correctly (see Sect. 1.1), **ANSYS_UM.EXE** starts automatically. Otherwise run **ANSYS_UM.EXE** manually from the **{UM}\bin** directory.
10. Transform data according the 5-8 items of the Sect. 1.2.1.

2.1.2. Working under ANSYS Workbench environment

Run a **PlatformShell63Demo.ans** macro in **APDL** language to create a FE-model of a platform for a vibrostand in a classic **ANSYS** (**Mechanical APDL**) program. In a **Workbench** environment it is impossible to perform such a macro that is why to create a model of platform in **ANSYS Workbench** you should take the following steps.

11. Create a model in **ANSYS Mechanical APDL** (see Sect. 1.2.1. *"Working under ANSYS environment"*, p. 8). Before running the **PlatformShell63Demo.ans** macro, delete all the commands intended for import in the UM software:

```
NSEL,s,,,ALL
ESLN,s,0,ALL
CM,ESTRS,ELEM
ESEL,ALL
NSEL,ALL
KSEL,S,,,5
KSEL,A,,,11
KSEL,A,,,105
KSEL,A,,,111
NSLK,S
UM,10,1,1,1
```

Save changes. Run the changed macro using **File->Read Input From** commands. Create a file with the **Preprocessor -> Archive Model -> Write** archive. In the appeared **Write Geometry/Loads for Archiving** window choose the view of the archived information and a name of the file with the archive, Figure 2.2.

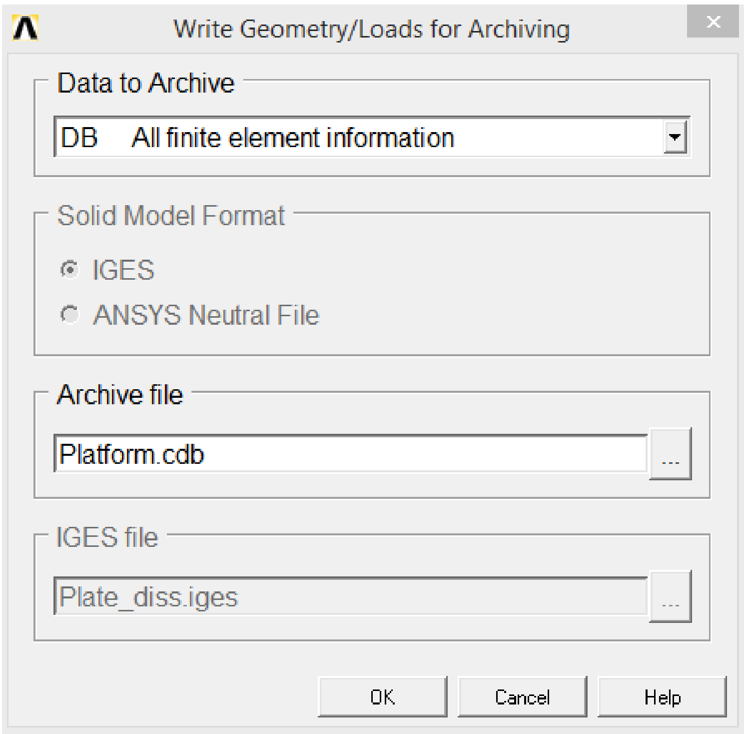


Figure 2.2. Creating an archive of a platform model

12. In the **ANSYS Workbench** environment on the taskbar in the **Component Systems** section choose **Finite Element Modeler** by double-clicking on it.

An empty object will appear in **Finite Element Modeler** in the **Project Schematic** window. Right-click on the **Model->Add Input Mesh->Browse** menu item. Choose a file with **.cdb** extension, created in the previous step, Figure 2.3. Refresh the model, right-click in the **Model** field then choose the **Update** in the dropdown list.

13. Choose a **Model** field of the scheme project then choose the name of the imported file (**Platform.cdb**) and choose **SI** system in the **Unit System** properties field, see Figure 2.4.

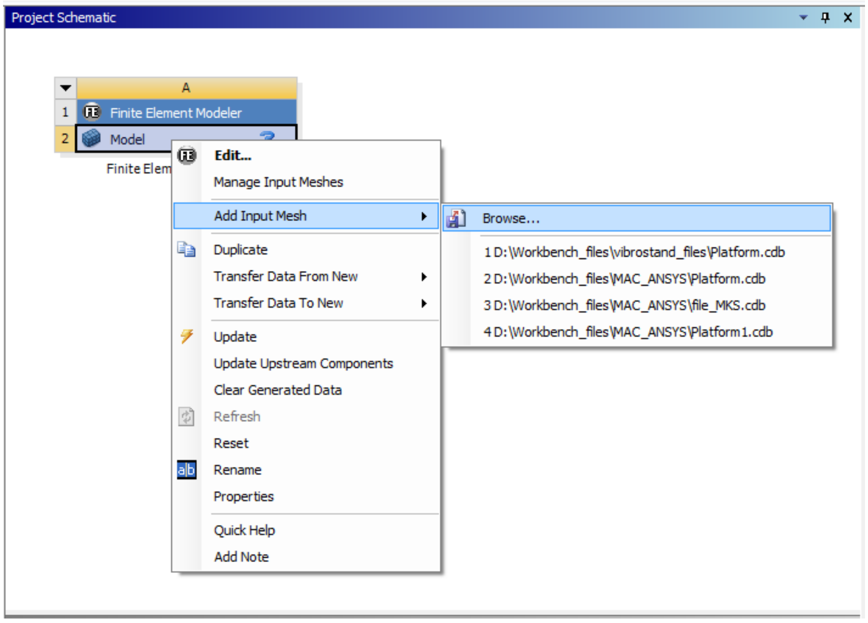


Figure 2.3.

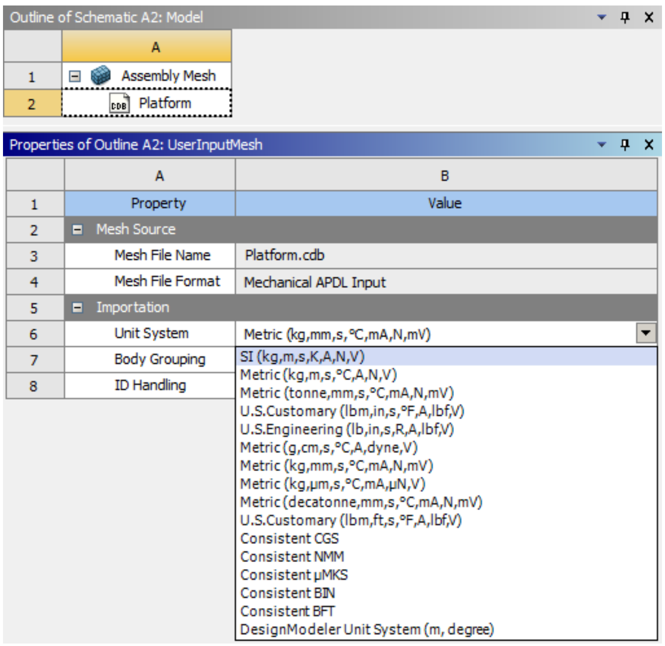


Figure 2.4. Choosing SI system

14. Create a project of modal analysis by double-clicking on a **Modal** component in the taskbar, Figure 2.5.

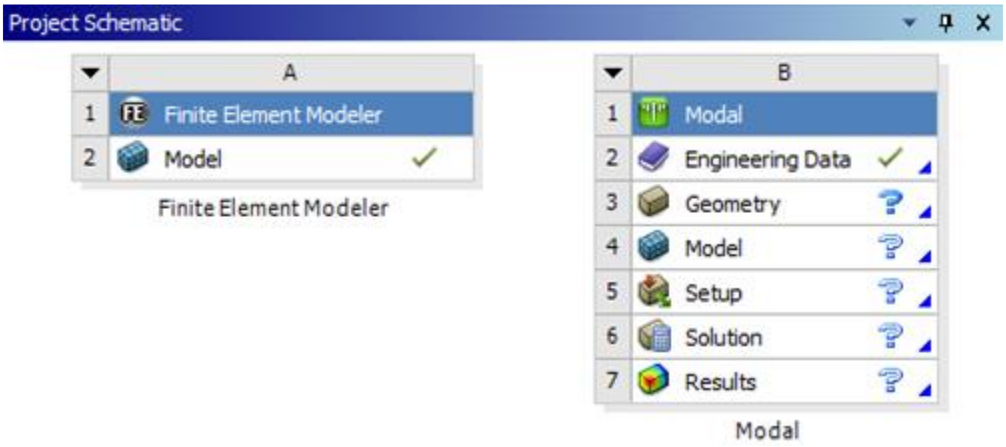


Figure 2.5.

15. Drag the **Model** field of a project in **Finite Element Modeler** on the **Model** field of modal analysis, created in the previous step, Figure 2.6.

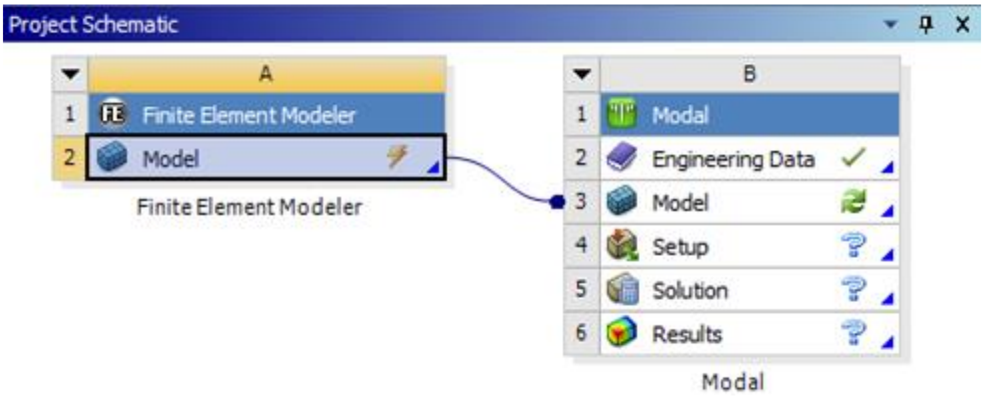


Figure 2.6.

Refresh the model in the project of modal analysis. Right-click on the **Model** field and choose **Update** in the dropdown list. Click **File->Save** to save the project.

16. Open the **Modal** project model by double-clicking on the **Model** field. The **Mechanical** module will be run, see Figure 2.7.
17. Add the **Commands** field in the project tree **Modal->Insert->Commands**. Set the commands in **APDL** language. For the downloaded model it is recommended to use the following commands:

```
NSEL,s,,,ALL
ESLN,s,0,ALL
CM,ESTRS,ELEM
ESEL,ALL
NSEL,ALL
NSEL,s,,,2435
NSEL,A,,,730
```

NSEL,A,,,2659

NSEL,A,,,958

UM,10,1,1,1

In these commands 2435, 730, 2659 and 958 are the numbers of the interface nodes, situated in the points with numbers 5, 11, 105 and 111, position of which is set in the **PlatformShell63Demo.ans** file. If the parameters of FE-mesh were changed, the numbers of the interface nodes can be different too.

There is more information about APDL language commands in the Chapter 11, which you can download using the following link: www.universalmechanism.com/download/90/eng/11_um_fem.pdf.

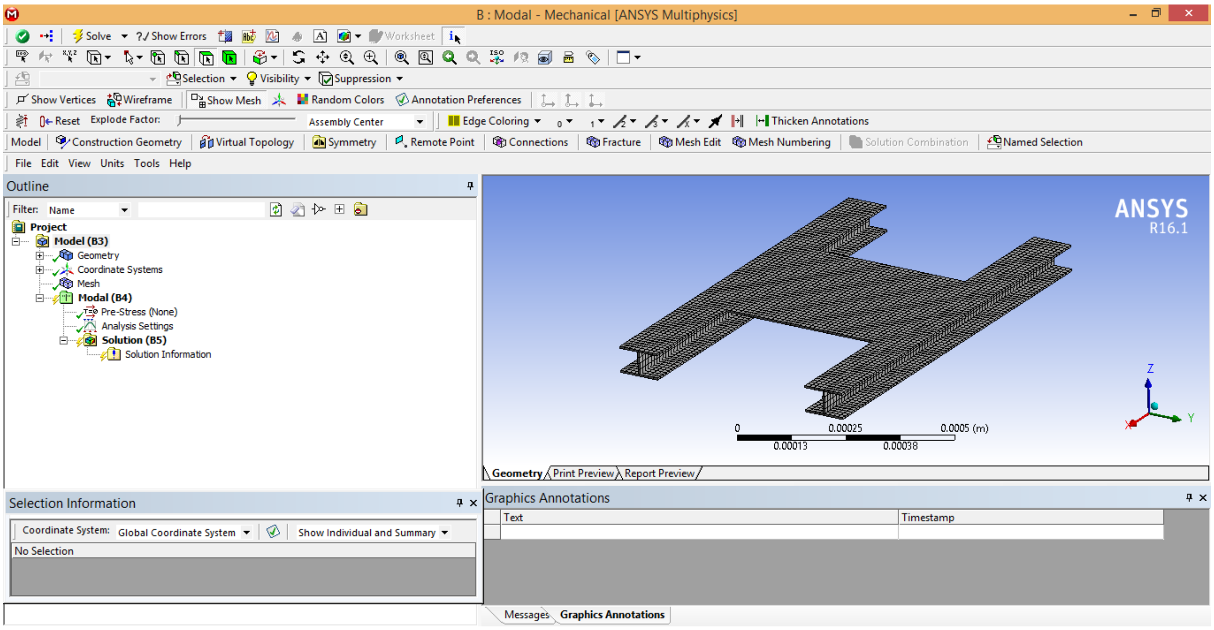


Figure 2.7. **Mechanical** module window with the model of platform

18. Run the solution process **Solution->Solve**. As a result the **.rst, .free** solution files will be situated in the folder of a **PROJECT_NAME\dp0\SYS\MECH** working project.
19. Run **ANSYS_UM.exe**, choose a file with **.rst** extension, create the **input.fum** file.

2.1.3. Preparing elastic platform data in FIDESYS software

In the UM program, each elastic body corresponds to a Linear FEM subsystem . The data file of the elastic subsystem is a binary file **input.fss**. Preparation of elastic platform data includes several steps described below: description of the platform by the finite element method in the **FIDESYS** program and **creation of files for the import in the UM software**; creation of an intermediate **input.fum** file with the **FIDESYS_UM.exe** program and the subsequent data transformation using the **Master of data preparation of elastic subsystems**, which is an integral part of the **UM Input** program. It provides the ability to view the calculated elastic forms of the subsystem, as well as exclude some forms from the resulting data set stored in the **input.fss** file.

The finite element model (FE model) prepared in the **FIDESYS** software is stored in a file with the **.fds** extension. The data on the FE model prepared using the **FIDESYS** program for import into the UM software are contained in the **geometry.vtk** files (contains data on the geometry of the object, the FE mesh), **res.cbm** (native and static forms), **M_CCS.hb** (mass matrix), **K_CCS.hb** (stiffness matrix).

The **geometry.vtk, res.cbm, M_CCS.hb, K_CCS** files are located in the {UM Data}\SAMPLES\Flex\vibrostand\platform directory, they **are created to import data into the UM software**, and the **platform.fds** file, **containing all the information about the platform's FE model in the FIDESYS software**, which should be used depending on the completeness of the study of this manual.

If the user wants to skip the stages of development and preparation of the FE model in the **FIDESYS** software, you can use the ready-made **geometry.vtk, res.cbm, M_CCS.hb, K_CCS.hb** files and go to Sect. 2.1.3.4.

The steps of creating a FE model in the **FIDESYS** program interface are presented in 2.1.3.1, the method of creating this FE model on the basis of **FIDESYS** software commands is shown in 2.1.3.2.

If the user wants to skip the steps of creating a FE model in the **FIDESYS** software, you can use the ready-made **platform.fds** file, which contains information about the FE model and interface nodes. In this case, you can go straight to Sect.2.1.3.3.

2.1.3.1. Working in the FIDESYS program interface

In the **FIDESYS** program, we will create a model of a steel platform consisting of two beams 1 m long, having an I-beam section, and a shelf that connects these beams. The width of the upper and lower shelves of the beam sections is 10 cm, the height of the sections is 6 cm. The length and width of the connecting shelf is 40 cm.

We will create the geometry of the model based on the plates. There are several options for creating planes in the **FIDESYS** program, for more information, see the user manual on the program's website[1]. In this manual, one of the ways is considered - the creation of a plane based on vertices. Select on the command panel **Mode - Geometry, Entity - Vertex**, set **Coordinates** in the drop-down list, enter the x, y, z coordinates of the four vertices of the future surface ([0.5; 0.3;0.06], [0.5; 0.25;0.06], [0.5; 0.25;0.06], [0.5; 0.3;0.06]), click **Apply** (Figure 2.8).

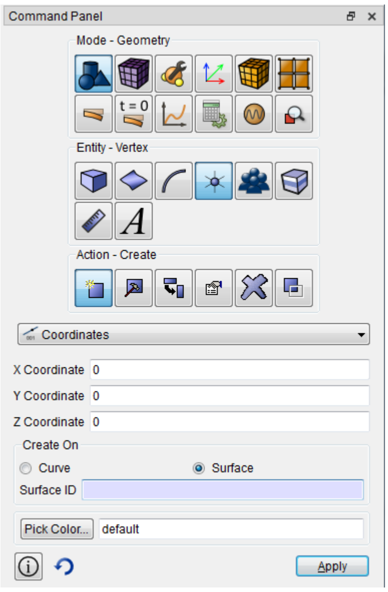


Figure 2.8

As a result, 4 points will appear in the animation window. Then select **Entity - Surface**, **Action - Create**, in the drop-down list - **Vertex ID(s)**, enter 4 vertices, click **Apply** (Figure 2.9). As a result, the first surface is created.

[1] https://cae-fidesys.com/documentation/

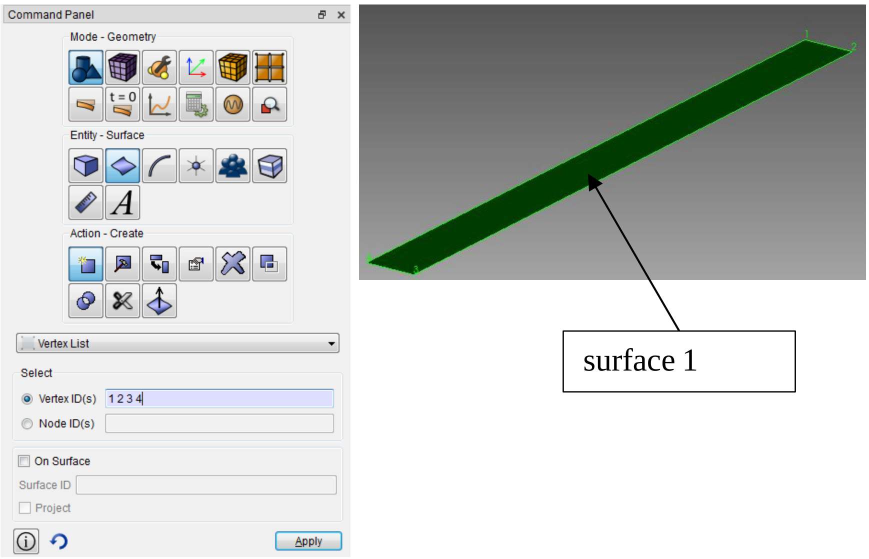


Figure 2.9

By entering the coordinates of two more vertices [-0.5;-0.25;0] and [0.5;-0.25;0], we will create surface 2 (Figure 2.10).

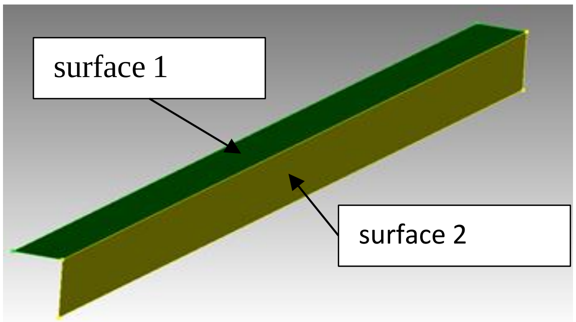


Figure 2.10

Using the method described above, you can create all the surfaces of the model, and to simplify the work, you can use copying, rotating and moving an already created surface (Figure 2.11). Create a surface 3 by copying with moving along the Z axis by -0.06 m of surface 1 (**Mode - Geometry, Entity - Surface, Action - Create, Copy and Transform, Move**, in the z coordinates field we specify z = -0.06), and the surface 4 by copying with rotation around the curve 2 by 180 degrees of the surface 1, the surface 5 by copying with the movement of the surface 4 (Figure 2.12).

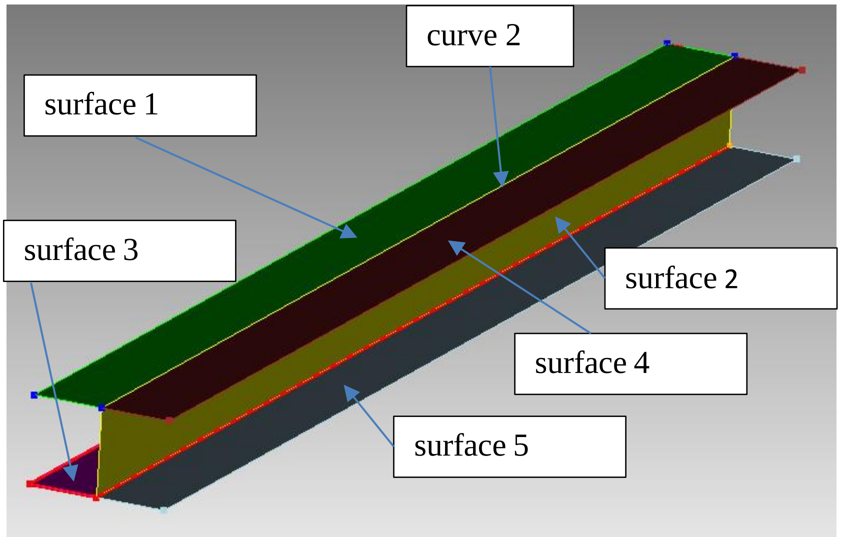


Figure 2.11

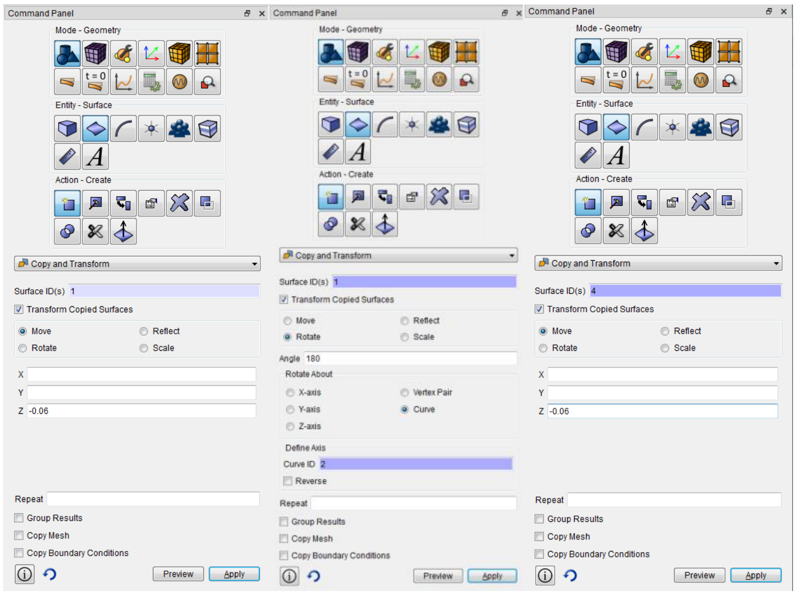


Figure 2.12. Creating a surface by copying and moving

At this stage, one of the two beams of the I-beam section has been obtained. Let's create a second beam (Figure 2.13) by copying the surfaces 1, 2, 3, 4, 5 of the first beam along the Y axis by 0.5 m (**Mode - Geometry**, **Entity - Surface**, **Action - Create**, **Copy and Transform**, **Move**, in the coordinates field we specify y = 0.5).

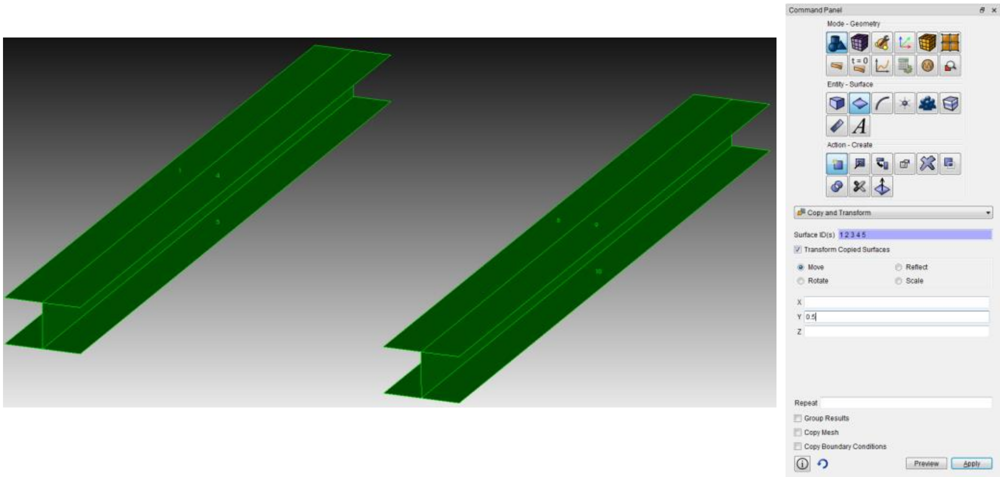


Figure 2.13

Let's create the middle surface of the platform by its 4 vertices (Figure 2.13). Select **Mode - Geometry**, **Entity - Vertex**, **Action - Create**, set **Coordinates** in the drop-down list, enter the x, y, z coordinates of each vertex of the future surface ([-0.2;-0.2;0.06], [0.2;-0.2;0.06], [0.2;0.2;0.06], [-0.2;0.2;0.06]), click **Apply**. Then select **Entity - Surface, Action - Create**, in the drop-down list - **Vertex list**, enter 4 created vertices (Figure 2.14).

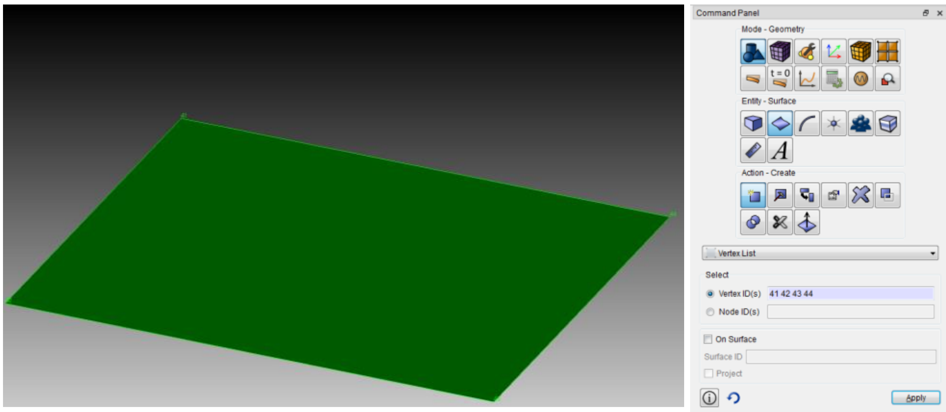


Figure 2.14

To get the desired mesh, combine several surfaces into one. To do this, select **Mode - Geometry, Entity - Surface, Action - Boolean** on the command panel, select **Unite**, specify the numbers of the middle surface of the platform (**11**) and two neighboring surfaces (**6** and **4**), click **Apply** (Figure 2.15).

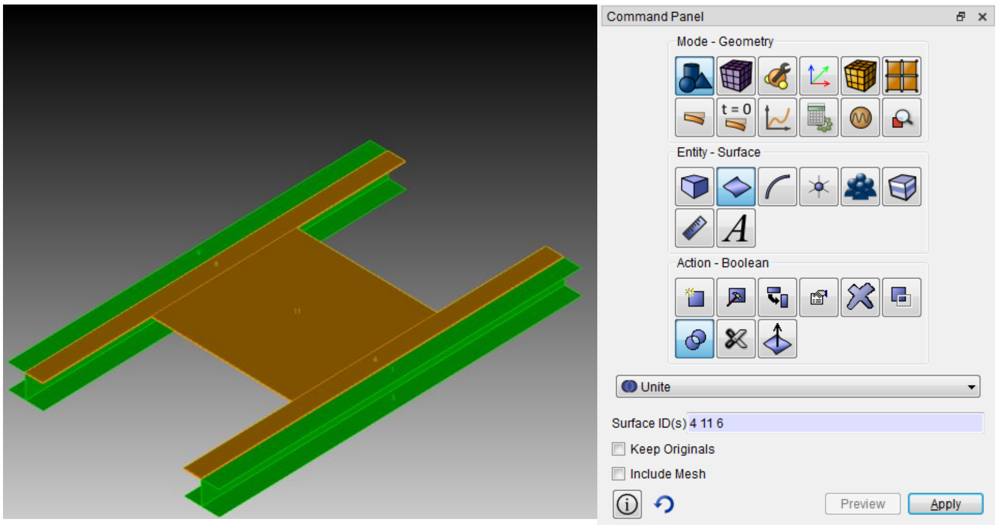


Figure 2.15

The result of the merge will be a surface 4 (Figure 2.16).

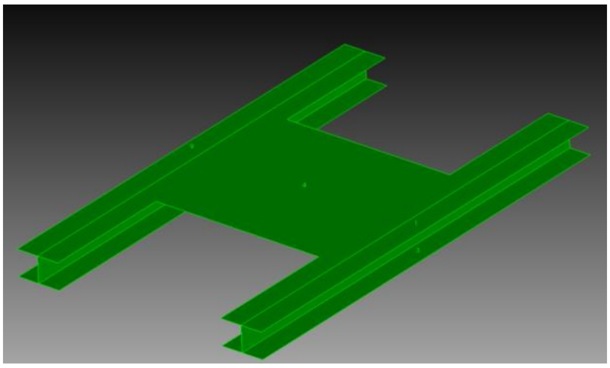


Figure 2.16

To exclude duplicated curves from the geometric model, the presence of which is caused by copying planes, combine the repeating curves **Mode - Geometry, Entity - Curve, Action - Merge**, in the **Curve ID(s)** field, type **all**, click **Apply** (Figure 2.17). The geometric model is ready.

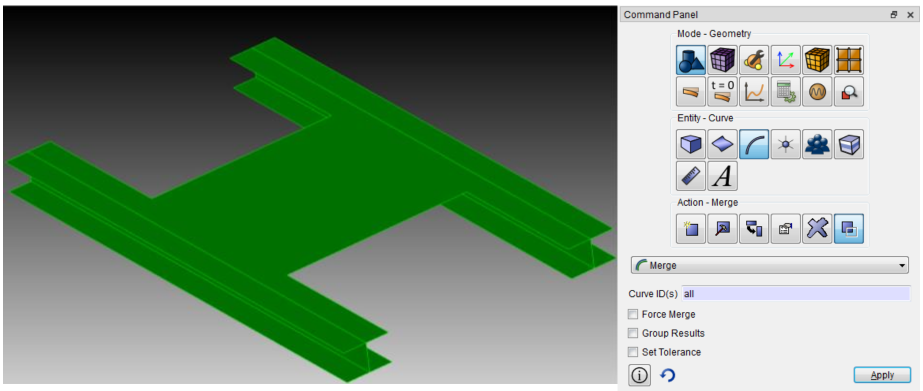


Figure 2.17

Let's create a material named **Steel**. On the command panel, select **Mode - Material**, object - **Material Management**. Let's give it the **Young's modulus** properties 2e11 Pa, **Poisson ratio** 0.3, **Density** 7850 kg/m3. Click **Apply**, the created material with the name **Steel** (Figure 2.18) should appear in the list of materials.

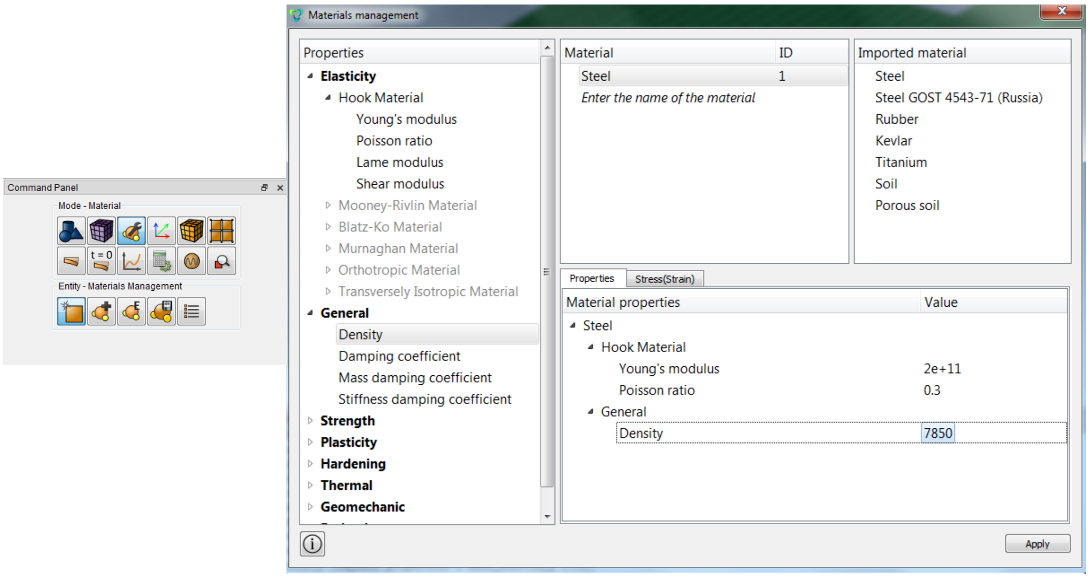


Figure 2.18

Creating a block with the inclusion of all created surfaces in it, select **Mode -Blocks, Entity - Block, Action - Add**, select **Surface** from the list of entities, type **all** (Figure 2.19) in the **Entity ID(s)** field.

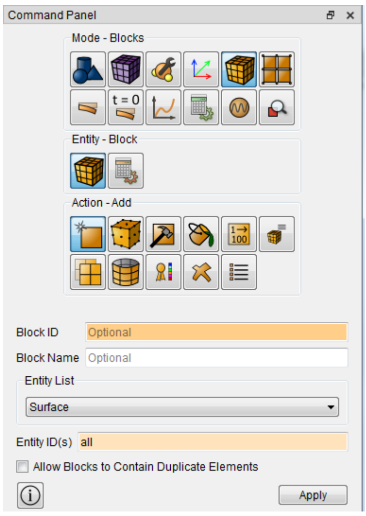


Figure 2.19

Now you need to assign shell properties to the created block (Figure 2.20). Select **Mode - Blocks**, **Entity - Block**, **Action - Block properties/parameters**, select the created block, **Category - Sell, Order** 1, click **Set Shell Properties** (Figure 2.20 on the left). In the appeared window **Set Shell Properties** (Figure 2.20 on the right), enter **Thickness** 0.005 m and select the **Steel** material, click **Apply**. Then click **Apply** in the window for setting block properties (Figure 2.20 on the left).

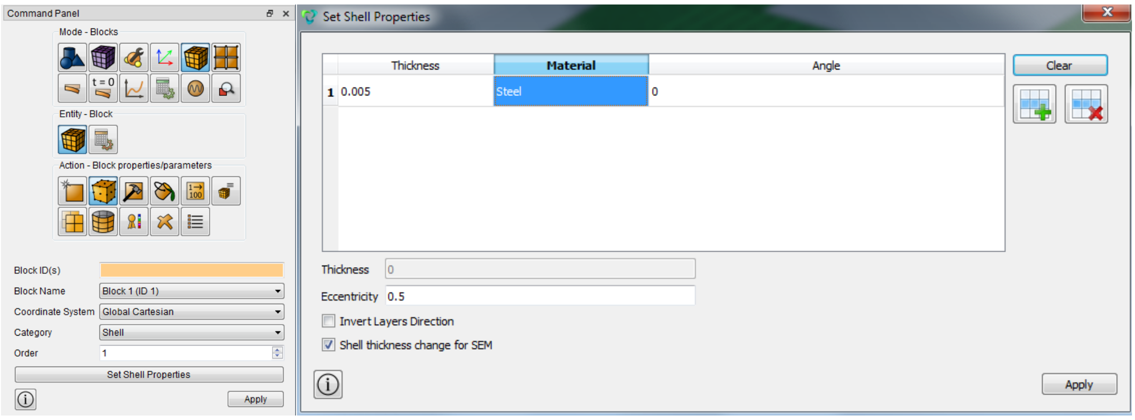


Figure 2.20

Let's set the grid construction scheme. When creating a finite element scheme of the platform, nodes are provided at the attachment points of the force elements, through which the electric motor is located. In order to get the desired FE grid, we determine the size of the finite element by specifying the number of elements for each curve (Figure 2.21).

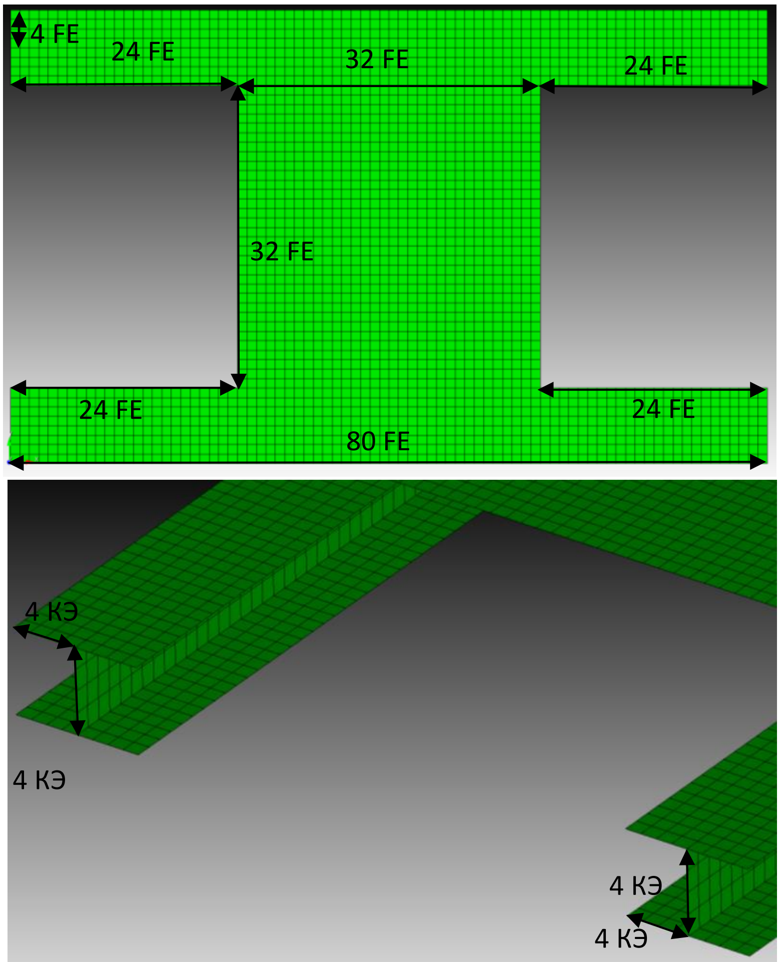


Figure 2.21

Select **Mode - Mesh, Entity - Curve, Action - Mesh**, put the cursor in the **Select Curves** field, then select the desired curves in the graphical window, select **Equal** in the drop-down list, set **Approximate Size**, enter the desired number of partitions in the **Approximate Size** field (Figure 2.22 - Figure 2.25).

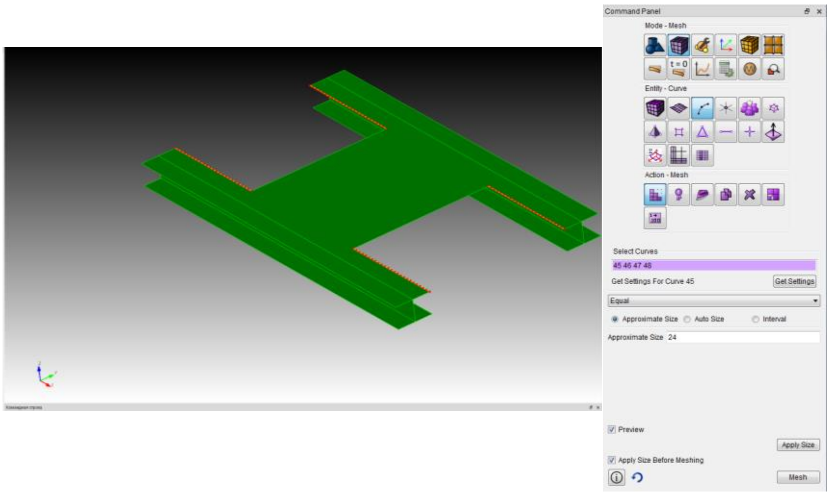


Figure 2.22

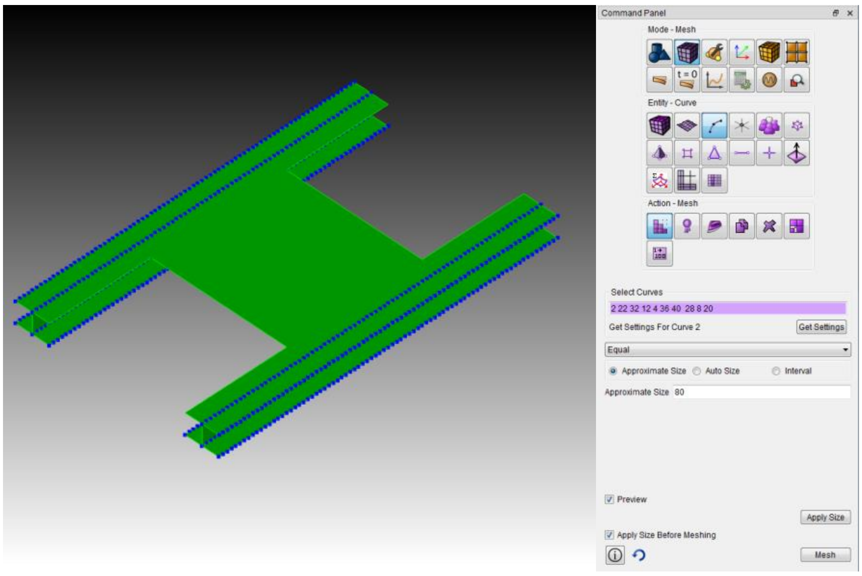


Figure 2.23

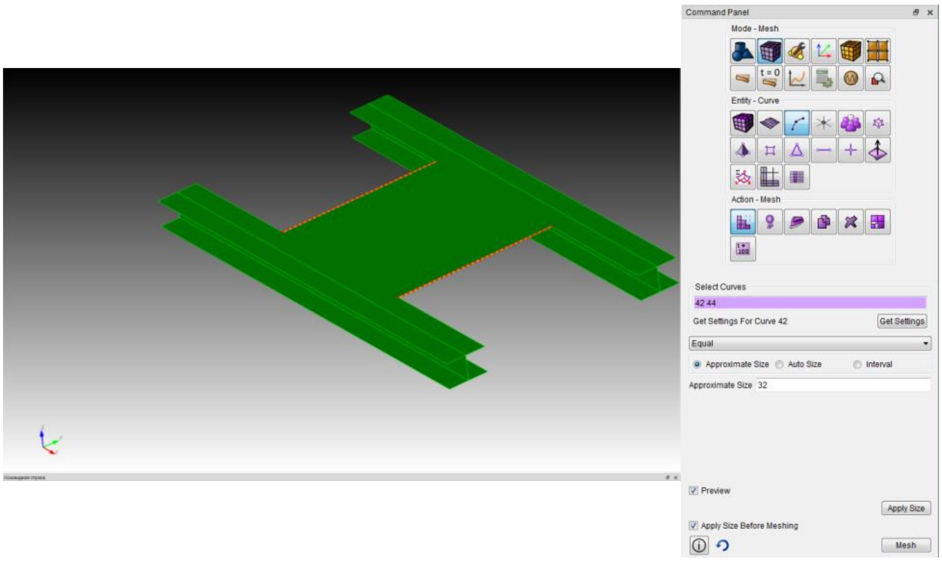


Figure 2.24

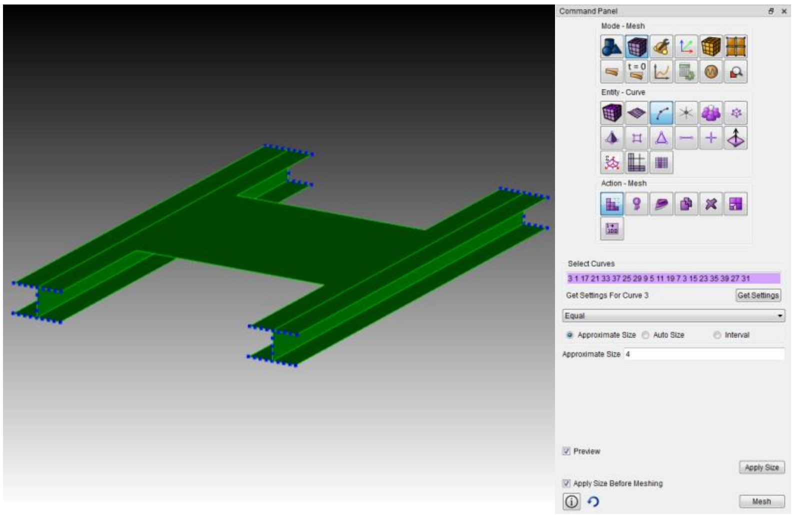


Figure 2.25

To create a surface FE grid with the specified scheme, select **Mode -Mesh, Entity - Surface, Action - Mesh**, select 4 meshing scheme **SubMap** for the combined surface, click **Apply Scheme, Mesh**, for the remaining surfaces (1, 2, 3, 5, 7, 8, 9, 10) choose the meshing scheme **Map**, then click **Apply Scheme, Mesh**. (Figure 2.26).

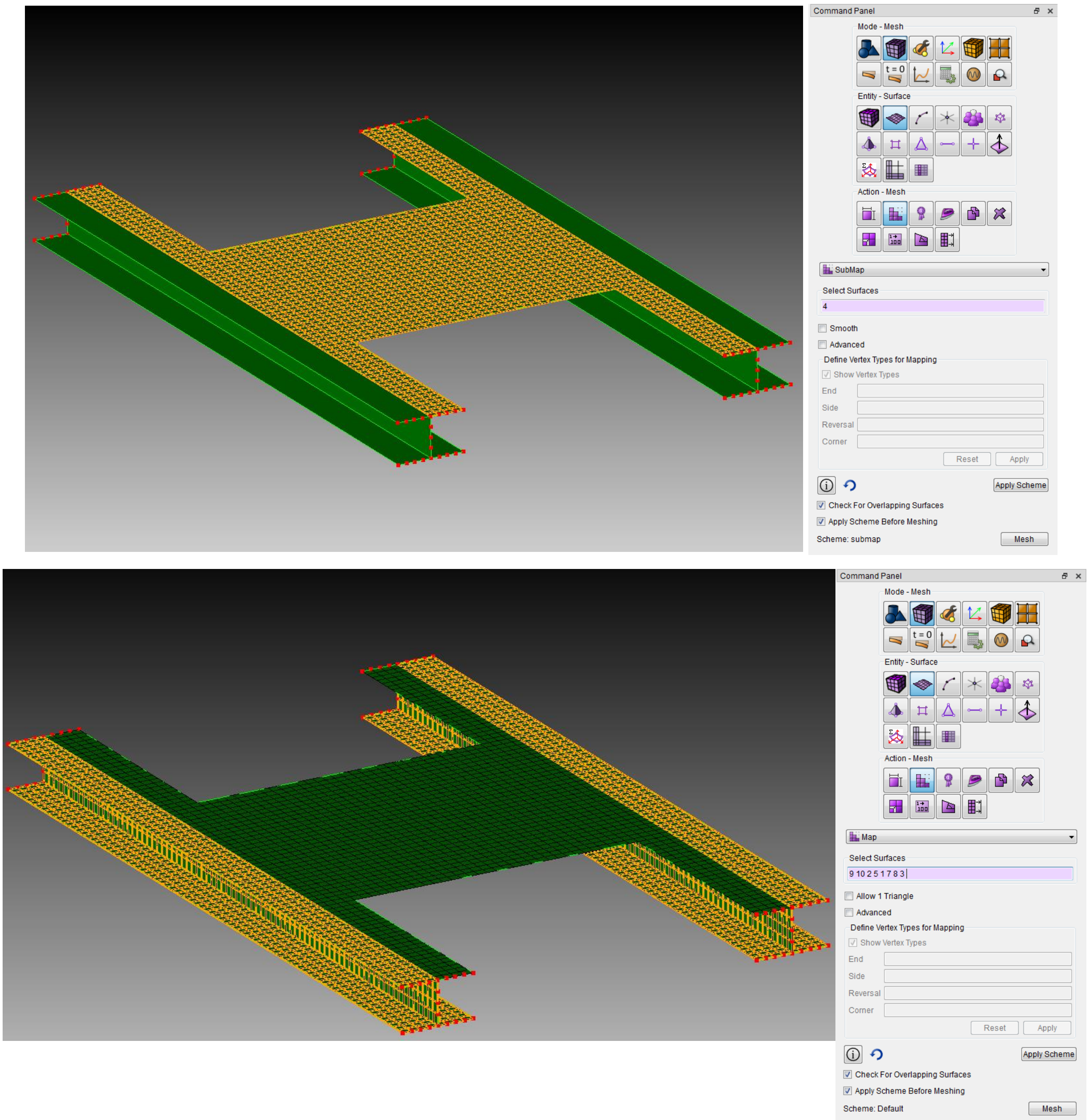


Figure 2.26

As a result, we obtain a finite element model including 4224 elements of the SHELL type, the thickness of all elements is 5 mm.

4 nodes are selected as interface nodes at the points of attachment of the platform to the base. The definition of the idea and description of the purpose of interface nodes are given in Chapter 11 of the UM User Manual (11_UM_FEM.pdf). Create a set of interface nodes **Mode - Manage Set, Entity - Nodeset, Action - Create nodeset**, enter **NodesetName**, select interface nodes (Figure 2.27).

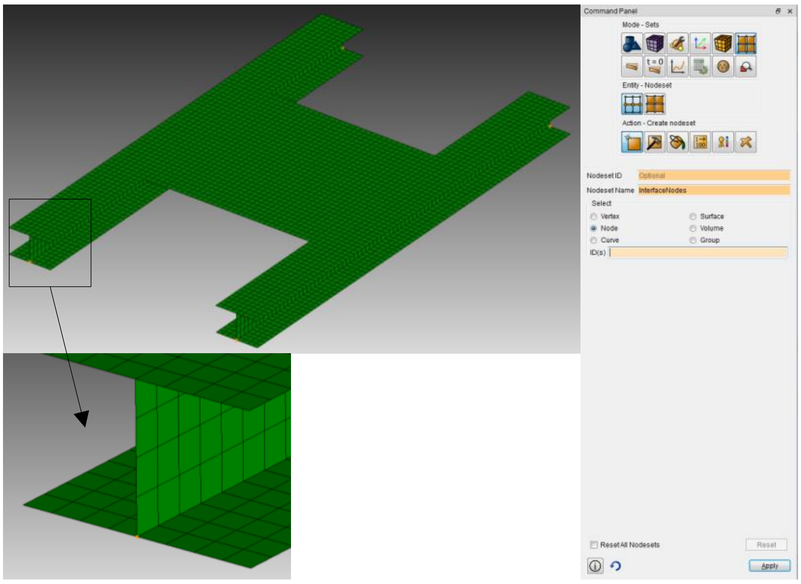


Figure 2.27

Finite element model of the platform is ready.

2.1.3.2. Working in the FIDESYS software using command-line commands

The operation of the FIDESYS program is also controlled by entering commands in the command line (Figure 2.28).

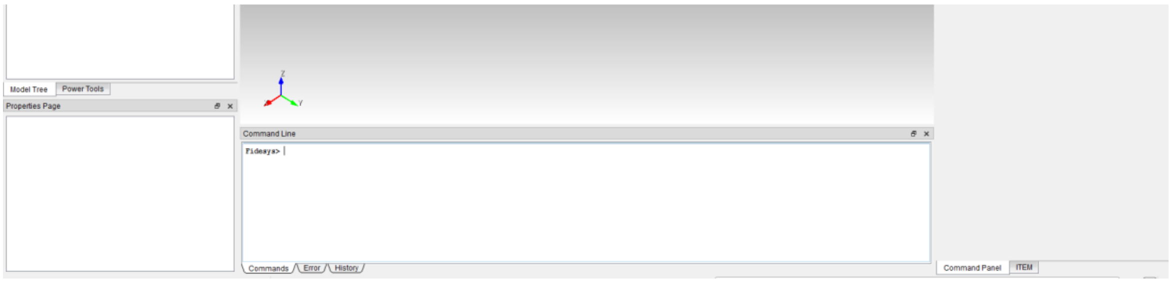


Figure 2.28

Thus, all the actions described in 2.1.3.1 can be performed by a set of commands. Below there is a list of commands that specify the creation of a finite element model of the platform with interface nodes. If you want to quickly create a platform model, copy and paste the following commands into the command line window (you can copy and paste all the lines at once).

```
create vertex -0.5 -0.3 0.06
create vertex -0.5 -0.25 0.06
create vertex 0.5 -0.25 0.06
create vertex 0.5 -0.3 0.06
create surface vertex 1 2 3 4
create vertex -0.5 -0.25 0
create vertex 0.5 -0.25 0
create surface vertex 5 2 3 6
Surface 1 copy move z -0.06
Surface 1 copy rotate 180 about vertex 2 3 nomesh
Surface 4 copy move z -0.06 nomesh
```

```
Surface 1 2 3 4 5 copy move y 0.5 nomesh
create vertex -0.2 -0.2 0.06
create vertex 0.2 -0.2 0.06
create vertex 0.2 0.2 0.06
create vertex -0.2 0.2 0.06
create surface vertex 41 42 43 44
unite surface 4 11 6
merge curve all
create material 1
modify material 1 name 'Steel'
modify material 1 set property 'MODULUS' value 2e+11
modify material 1 set property 'POISSON' value 0.3
modify material 1 set property 'DENSITY' value 7850
set duplicate block elements off
block 1 add surface all
create shell properties 1
modify shell properties 1 layer count 1
modify shell properties 1 layer 1 thickness 0.005
modify shell properties 1 layer 1 material 1
modify shell properties 1 layer 1 angle 0
modify shell properties 1 layer 1 cs 1
modify shell properties 1 eccentricity 0.5
modify shell properties 1 layer direction normal
modify shell properties 1 thickness_change on
block 'Block 1' element shell order 1
block 'Block 1' shell properties 1
curve 46 47 48 45 interval 24
curve 46 47 48 45 scheme equal
curve 2 22 32 12 4 36 40 28 8 20 interval 80
curve 2 22 32 12 4 36 40 28 8 20 scheme equal
curve 42 44 interval 32
curve 42 44 scheme equal
curve 13 1 17 21 33 37 25 29 9 5 11 19 7 3 15 23 35 39 27 31 interval 4
curve 13 1 17 21 33 37 25 29 9 5 11 19 7 3 15 23 35 39 27 31 scheme equal
surface 4 submap smooth off
surface 4 scheme submap
mesh surface 4
surface 9 10 2 5 1 7 8 3 scheme map
surface 9 10 2 5 1 7 8 3 scheme map
mesh surface 9 10 2 5 1 7 8 3
nodeset 1 add vertex 28 6 25 5
nodeset 1 name "InterfaceNodes"
```

As a result of executing the commands, a finite element model of the platform with interface nodes will be created, which can be prepared for import, according to 2.1.3.3.

2.1.3.3. Preparing the model for import into the UM software.

To prepare a model for import click **Mode - Calculation Settings**, **Calculation Settings - External Integration MBD**, choose **Number of eigenmodes** - 10, set **OutputFormat - Binary**, нажимаем **Apply**, then **Start Calculation (**Figure 2.29**)**.

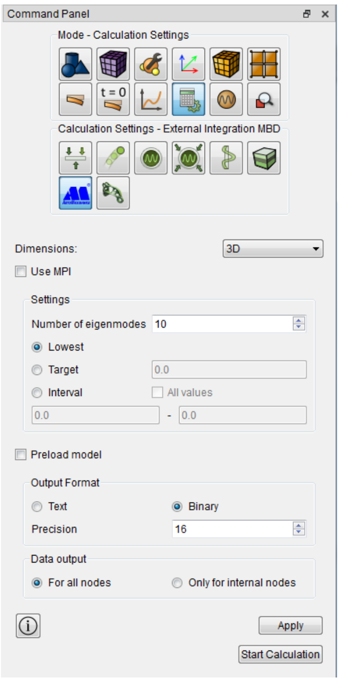


Figure 2.29

Specify the path to save the task (Figure 2.30).

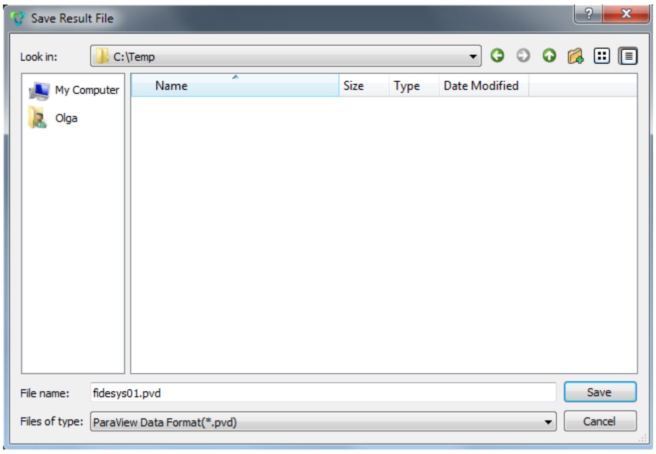


Figure 2.30

After the calculation is completed, files with the results will be received with the resuts: **geometry.vtk, res.cbm, M_CCS.hb, K_CCS.hb** containing 24 static forms and 10 eigenforms corresponding to 10 lower eigenfrequencies, mass and stiffness matrices.

Then run **FIDESYS_UM.exe**.

2.1.3.4. Data exchange with FIDESYS software

Run the **FIDESYS_UM.EXE** converter program to create **input.fum** file. Select the **geometry.vtk** file in the task directory and specify the save directory for the **input.fum** file.

It's important! Files **res.cbm**, **M_CCS.hb**, **K_CCS.hb must be located in the same directory as the geometry.vtk file**. Assignment of all fields of the **FIDESYS_UM.EXE** program window is clear by their names (Figure 2.31).

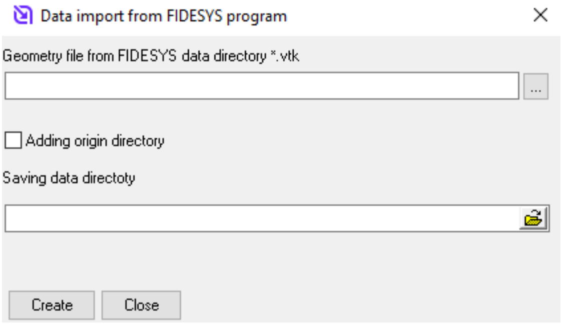


Figure 2.31

Perform the data conversion with the **Create** button. If successful, the **input.fum** file will be created in the directory specified in the **Directory to save data** field.

2.1.4. Wizard of flexible subsystems

Working with the **Wizard of flexible subsystems** is described in the Sect. 1.2.2. Now you should repeat all the instructions from the Sect. 1.2.2. Use the **.\platform\input.fum** as an input file for the **Wizard**. Please, note, that the **.\platform\input.fss** file should be created after all.

2.2. Creating the model and analyzing its dynamics

Now we will create a new model. From the **File** menu select **New object** or click the  button.

2.2.1. Introducing elastic platform

1. Select **Subsystems** item in the tree of elements. Create a new subsystem by clicking  button.
2. Set **Type** to **Linear FEM subsystem**. New open dialog appears. In this dialog select the **.\platform** directory.

You can see elastic modes using the **Amplitude** and **Rate** track bars on the **Solution | Modes** tab.

3. Set **Name** to **Platform** (Figure 2.32).

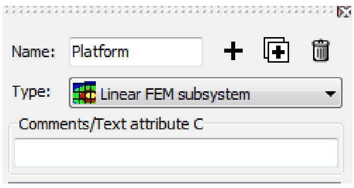


Figure 2.32.

2.2.2. Attaching the elastic platform to a base

Platform is attached to a ground with the help of four visco-elastic force elements that are situated at the edges of the platform. Firstly we will create graphical objects for force elements and then create force elements themselves.

2.2.3. Creating graphical elements

Now we will create graphical object for elastic force elements.

1. Select **Images** in the tree of elements.
2. Add new graphic object (GO) by clicking the + button.
3. Set name of the new GO to **Dampfer** (Figure 2.33).

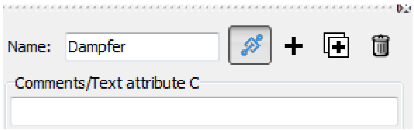


Figure 2.33.

4. Click the  button and select parameters of Bipolar GO (Figure 2.34)

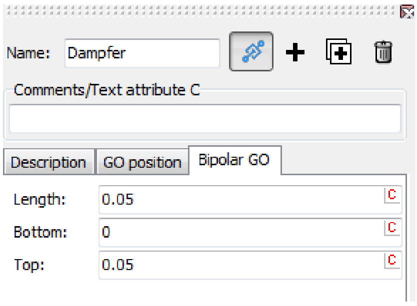


Figure 2.34

5. Add a new graphic element (GE) by clicking the + at the lower panel (Figure 2.35).

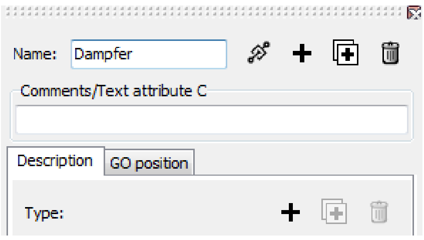


Figure 2.35.

6. Select **Spring** type in the pull-down menu (Figure 2.36).

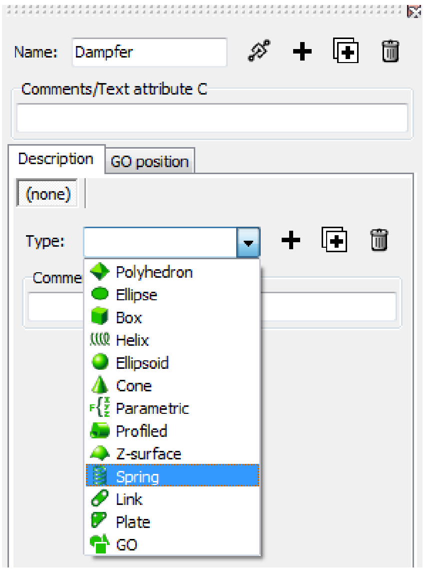


Figure 2.36.

7. Set **Spring** parameters as in Figure 2.37.

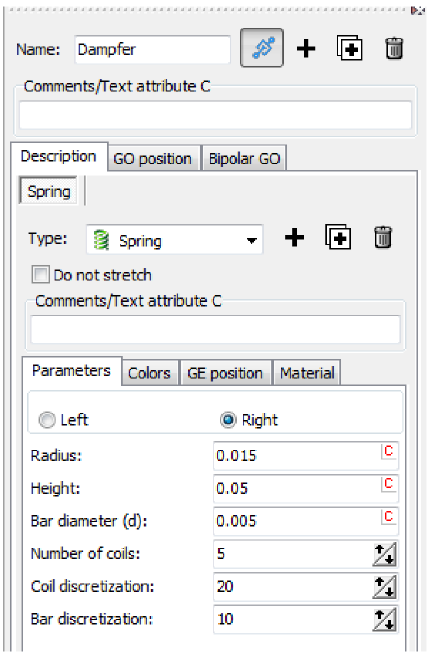


Figure 2.37.

8. Add a new GE to the GO. Set its type as **Cone** and parameters as in figure 2.38a.

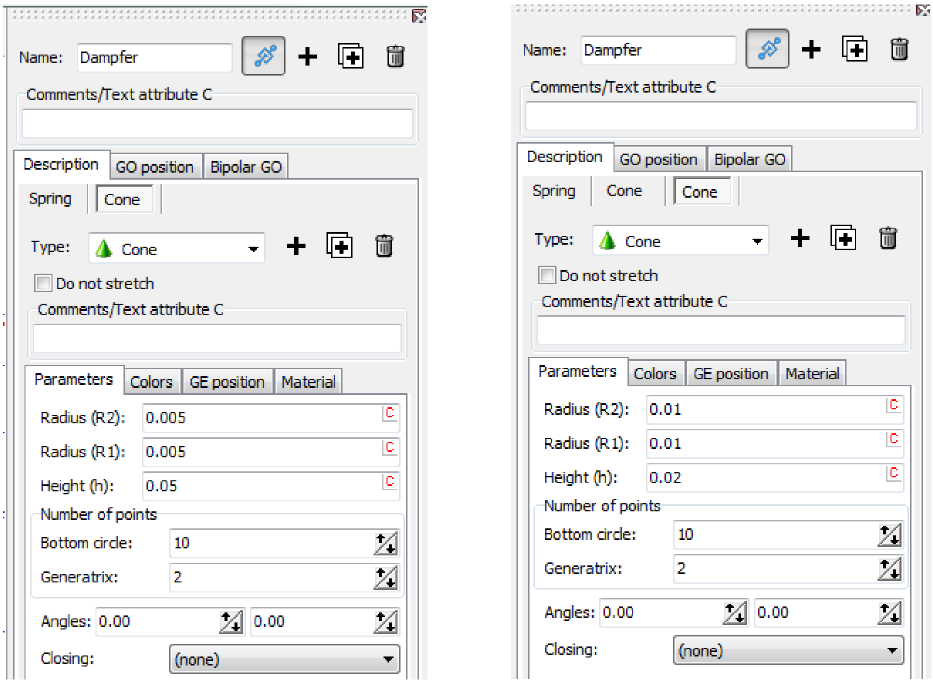


Figure 2.38. a), b)

9. Add the second GE **Cone** and set its parameters as in Figure 2.38b. Go to the **GE position** tab of the second GE **Cone** (Figure 2.39) and shift the element on **0.015** along Z axis (the **Translation** | z box).

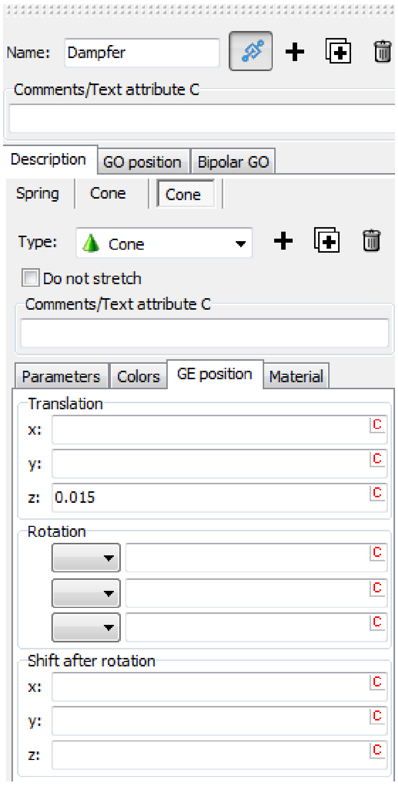
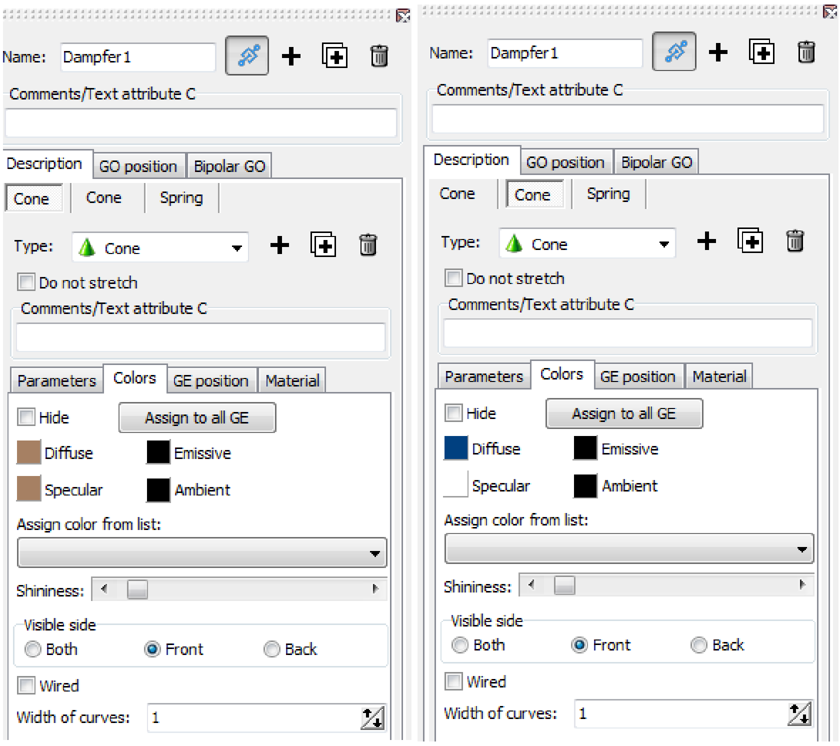


Figure 2.39.

10. Set the diffuse and specular components of the **Cones** color by **Diffuse** and **Specular** button on the Color tab (Figure 2.40a - **first Cone**, Figure 2.40b - **second Cone**).



a b

Figure 2.40.

The images are created. Let us continue with the force elements.

2.2.4. Force elements

Let us introduce several identifiers to set the attachment points:

- **BeamLength** – the length of platform beams;
- **WidthShelf** – the width of connecting shelf;
- **WidthBeamShelfLow** – the width of lower shelf of beam section.

Let us start with the elastic element on the front left end of the platform beam.

1. Select **Linear forces** in the object element list.
2. Add a new force element by clicking the + button.
3. Rename it as **Dampfer_FL** (forward, left), set element type Viscous-elastic, interacting bodies **Base0 Platform.Platform** as well as the Spring_Dampfer GO (Figure 2.41).
4. Set coordinates of element attachment points to the first body **Base0:**

BeamLength/2, WidthShelf/2 + WidthBeamShelfLow/2, – 0.05;

Initialize values of identifiers as (Figure 2.43)

BeamLength=1.0, WidthShelf=0.4, WidthBeamShelfLow=0.1

5. Coordinates of the element end point in undeformed state in system of coordinates of the first body, Figure 2.41:

BeamLength/2, WidthShelf/2 WidthBeamShelfLow/2, 0.

6. Select **Body2** tab. Set coordinates of element attachment points to the second body **Platform.Platform** (Figure 2.42):

BeamLength/2, WidthShelf/2 + WidthBeamShelfLow/2, 0;

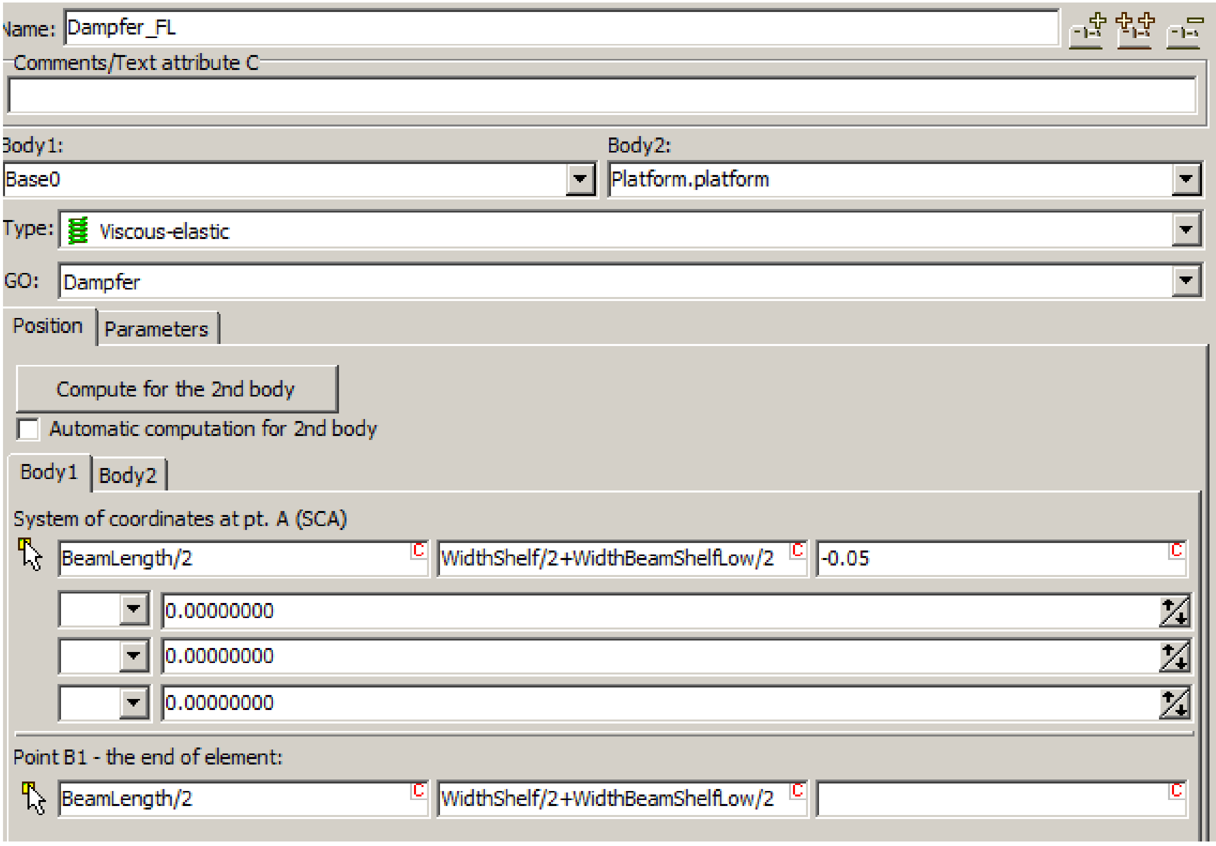


Figure 2.41.

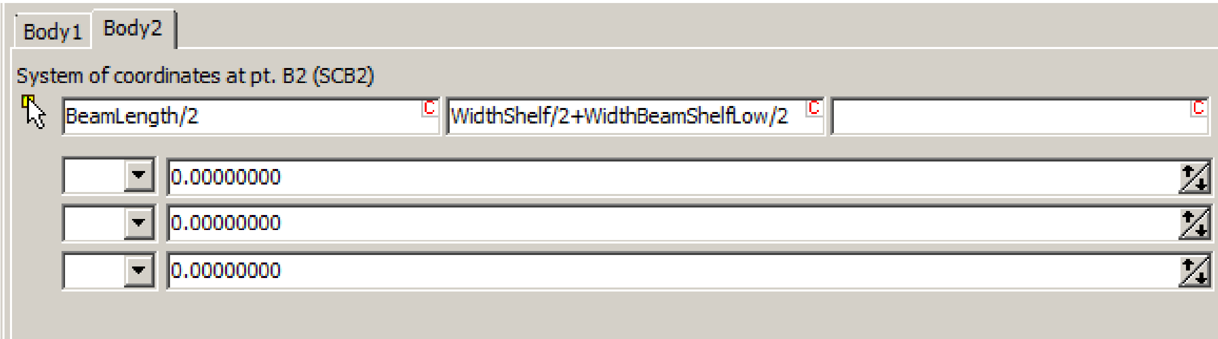


Figure 2.42.

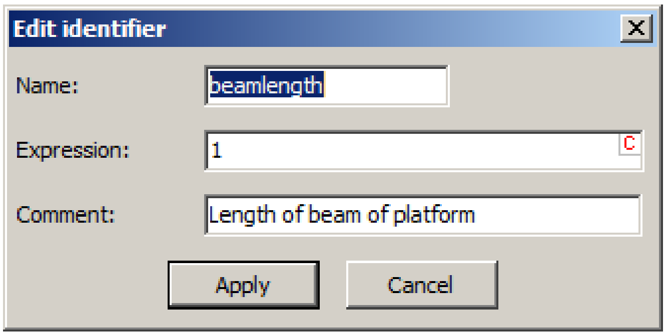


Figure 2.43.

7. Let us introduce a stiffness matrix of the element. Select **Parameters** tab. Click the button in the **Stiffness matrix** box (Figure 2.44), set diagonal elements of the matrix corresponding to the translational degrees of freedom (Figure 2.45), and click OK. Set the following identifier values: **cxx=1e+6, cyy=1e+6, czz=1e+6** (N/m).

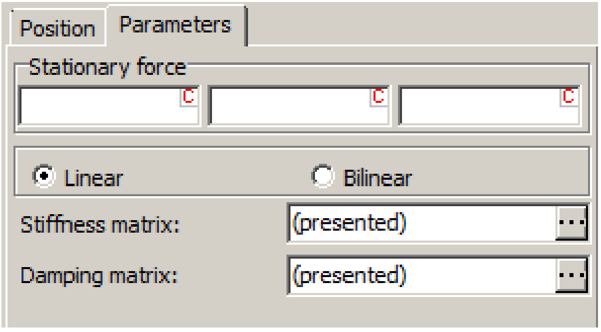


Figure 2.44.

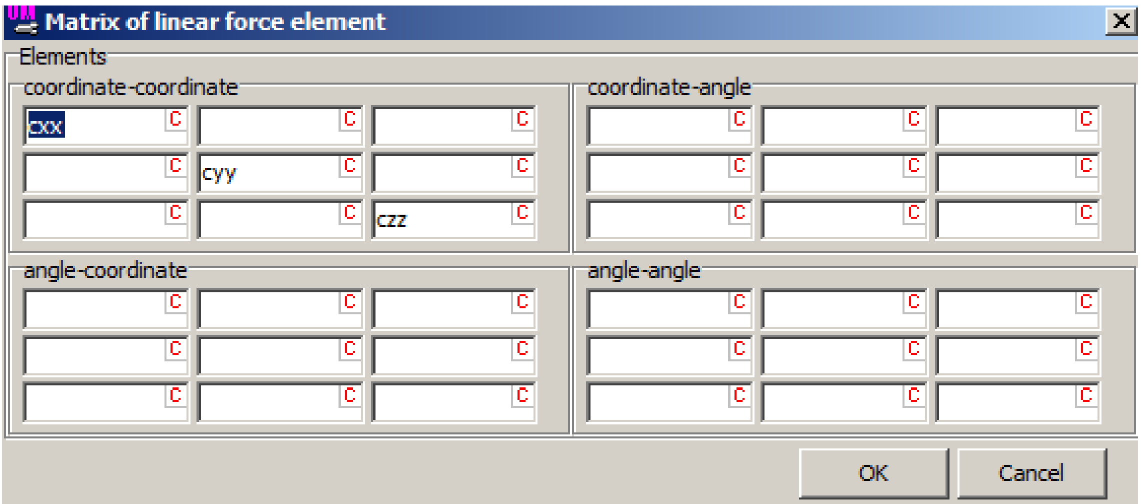


Figure 2.45.

The elastic part of force element is described.

Now let us describe damping part of the front left element.

Let us set dissipative matrix of the element. Select **Parameters** tab (Figure 2.44). Click the button in the **Dissipative matrix** box, set the diagonal elements of the matrix corresponding to the translational degrees of freedom **dxx, dyy, dzz** , and click **OK (** Figure 2.46**)**. Set the following identifier values **dxx=1E3, dyy=1E3, dzz=1E3** (Ns/m).

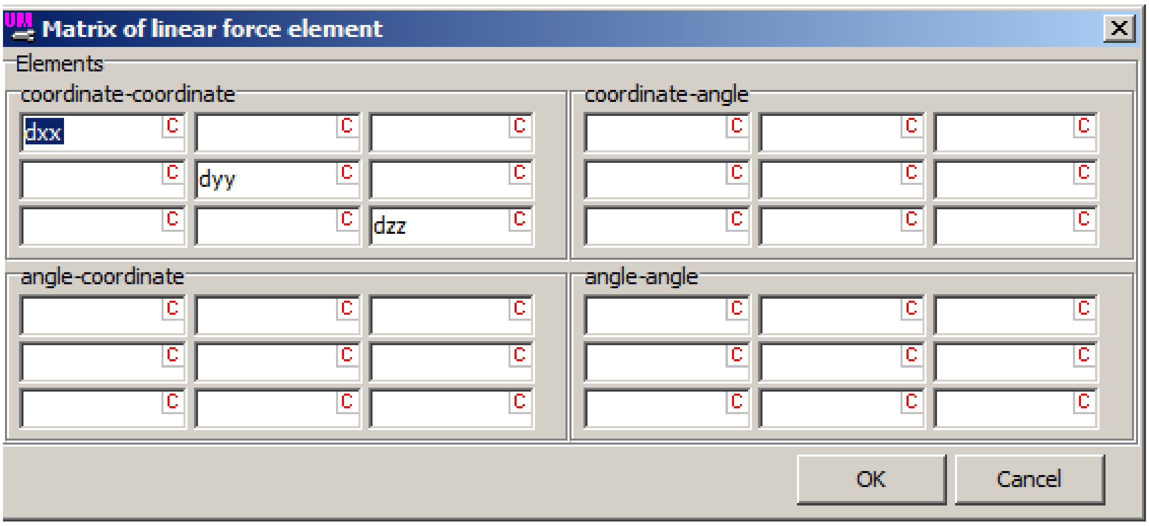


Figure 2.46

Damping part of element is described.

Create the rest three pairs of force element quite similar to the previous ones. Use the  button to copy the description. Do it in the following manner.

8. Select previously described element of the necessary type, e.g. Spring_Dampfer_FL in the case of a new elastic element.
9. Click the  button to create a copy.
10. Rename the copy, e.g. **Dampfer_FR** (forward, right).
11. Correct coordinates of attachment points. For the **Dampfer_FR** element we have **Base0**:

BeamLength/2, -WidthShelf/2 - WidthBeamShelfLow/2, –0.05;

Platform.Platform:

BeamLength/2, -WidthShelf/2 - WidthBeamShelfLow/2, 0.0;

Coordinates of the element end point in undeformed state in system of coordinates of the first body:

BeamLength/2, -WidthShelf/2 - WidthBeamShelfLow/2, 0.0 (Figure 2.41).

Thus, the full list of force elements connecting the platform with the base must include the following elements: **Dampfer_FL, Dampfer_FR, Dampfer_BL, Dampfer_BR**.

2.2.5. Model of electric motor

We shall not create the model but use the ready model of an electric motor located in the {UM Data}\SAMPLES\Flex\electricmotor directory.

2.2.6. Adding motor to object as a subsystem

1. Select the **Subsystems** tab in the element list. Add a new subsystem by the + button.
2. Select its type **Included** and open the {UM Data}\SAMPLES\ Flex\electricmotor model (Figure 2.47).

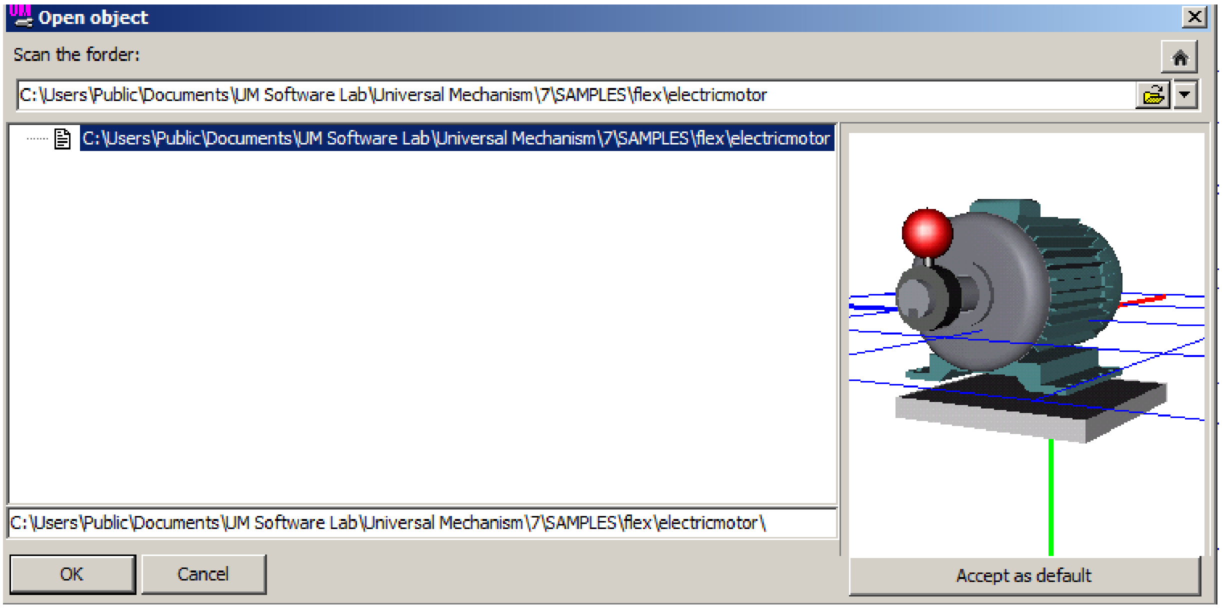


Figure 2.47.

3. Rename the subsystem as **Electricmotor**.
4. Set the subsystem location as in Figure 2.48.

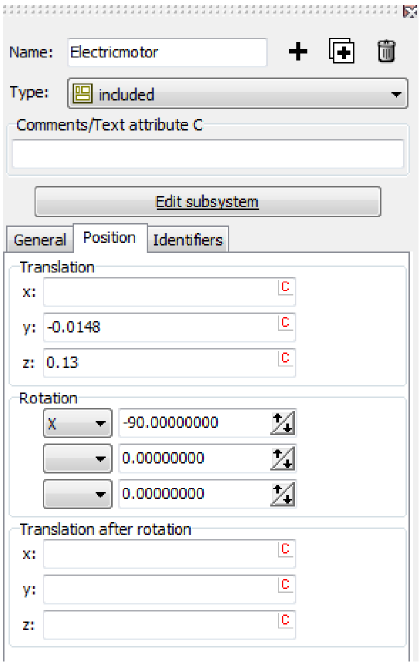

Name:
Electricmotor
Type:
included
Comments/Text attribute C
Edit subsystem
General
Position
Identifiers
Translation
x:
y:
-0.0148
z:
0.13
Rotation
X
-90.00000000
0.00000000
0.00000000
Translation after rotation
x:
y:
z:

Figure 2.48.

2.2.6.1. Setting angular velocity of the rotor

Let us set the law for angular velocity of the rotor as it shown in Figure 2.49. Here we can see three modes: speeding up, a working mode and a braking mode. During speeding up and braking angular acceleration is constant and angular velocity changes linearly, see Figure 2.49. The law from Figure 2.49 is parameterized with the help of six identifiers, see Table 1.

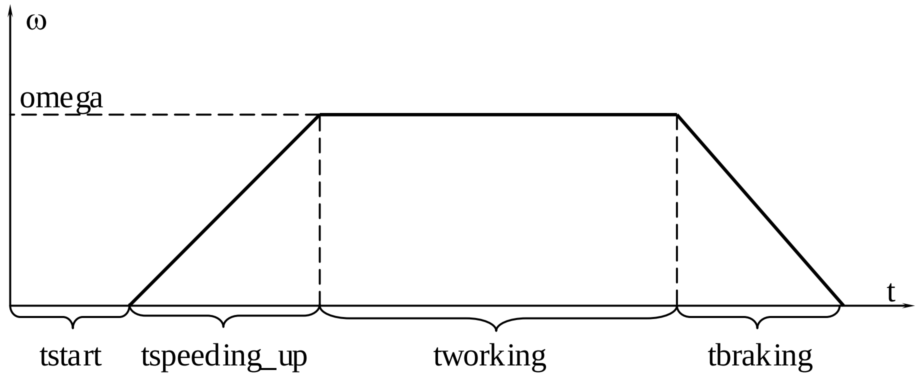


Figure 2.49. Angular velocity of the rotor

Table 1

Identifiers

	Identifier	Meaning
1	Nu	Nominal angular velocity of the rotor, revolutions per minute (r.p.m.)
2	omega	Nominal angular velocity of the rotor, rad/s
3	tstart	Time before speeding up, s
4	tspeeding_up	Time of speeding up mode, s
5	tworking	Time of working mode, s
6	tbraking	Time of braking mode, s

1. Click the **Edit subsystem** button to edit the **Electricmotor** subsystem, see Figure 2.48. New object constructor for the **Electricmotor** appears.
2. Select **Joints | jRotor->Body** in the tree of elements. It is a joint of the **Generalized** type.
3. In the **Inspector** window in the right part select the RTx elementary transformation (Figure 2.50.) This time function is set as **time-table** of 5 rows, see Table 2 and Figure 2.50.

Table 2

Time-table for the rotor

№	**Time interval**	**Expression**
1	Tstart	0
2	tstart+tspeeding_up	(omega/tspeeding_up)*sqr(t-tstart)/2
3	tstart+tspeeding_up+tworking	(omega/tspeeding_up)*sqr(tspeeding_up)/2+ omega*(t-tstart-tspeeding_up)
4	tstart+tspeeding_up+tworking+	(omega/tspeeding_up)*sqr(tspeeding_up)/2+ ome-

	tbraking	ga*tworking+omega*(t-tstart-tspeeding_up-tworking)-(omega/tbraking)* sqr(t-tstart-tspeeding_up-tworking)/2
5	100	(omega/tspeeding_up)*sqr(tspeeding_up)/2+ omega*tworking+omega*(tworking)-(omega/tbraking)*sqr(tbraking)/2

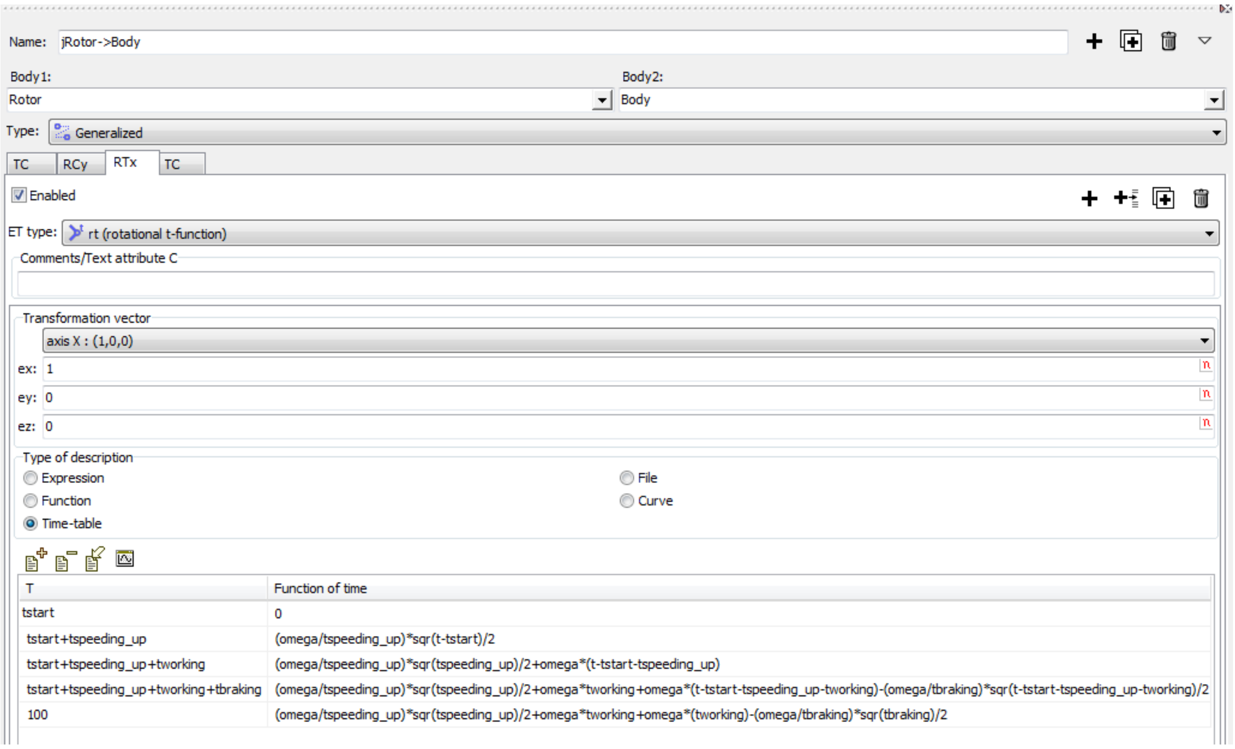


Figure 2.50.

4. Close the constructor window of the **Electricmotor** and come back to the composite model.

2.2.7. Electric motor and platform coupling by force elements

Coupling the electric motor and the platform can be set quite similar to attaching the platform to the base. **Electricmotor.Body** and **Platform.Platform** are interacting bodies. An example of description of an elastic force element is shown in Figure 2.51.

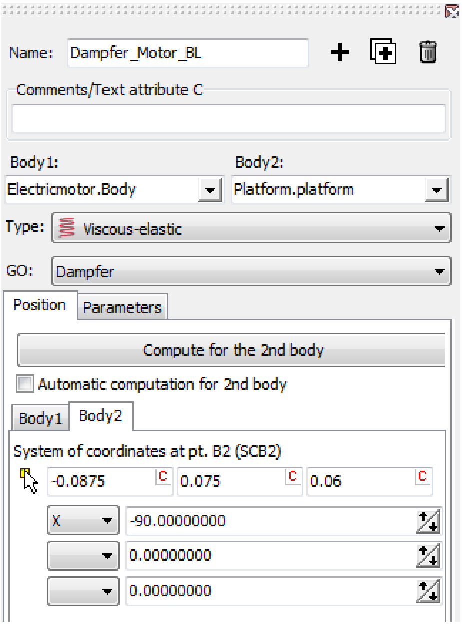


Figure 2.51.

Table 3 contains coordinates of attachment points of elastic and damping force elements realizing the coupling.

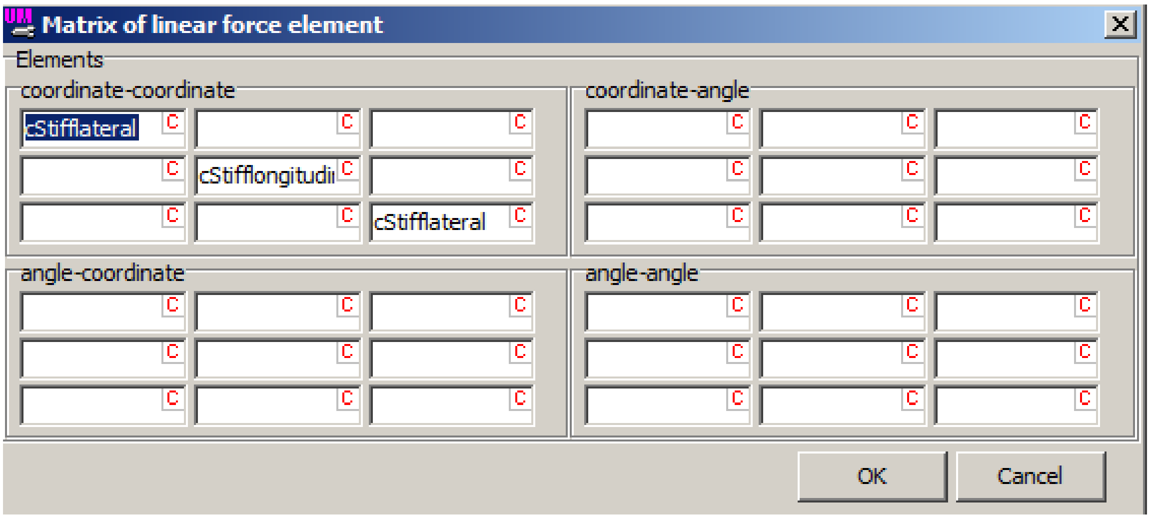
Table 3.

Force element	Electricmotor.Body			Platform.Platform		
	X	Y	Z	X	Y	Z
DamperMotorFL,	0	0.05	-0.06	0	0.075	0.06
DamperMotorFR	0	0.05	-0.06	0	-0.075	0.06
DamperMotorBL	-0.0875	0.05	0.0899	-0.0875	0.075	0.06
DamperMotorBR	-0.0875	0.05	0.0899	-0.0875	-0.075	0.06

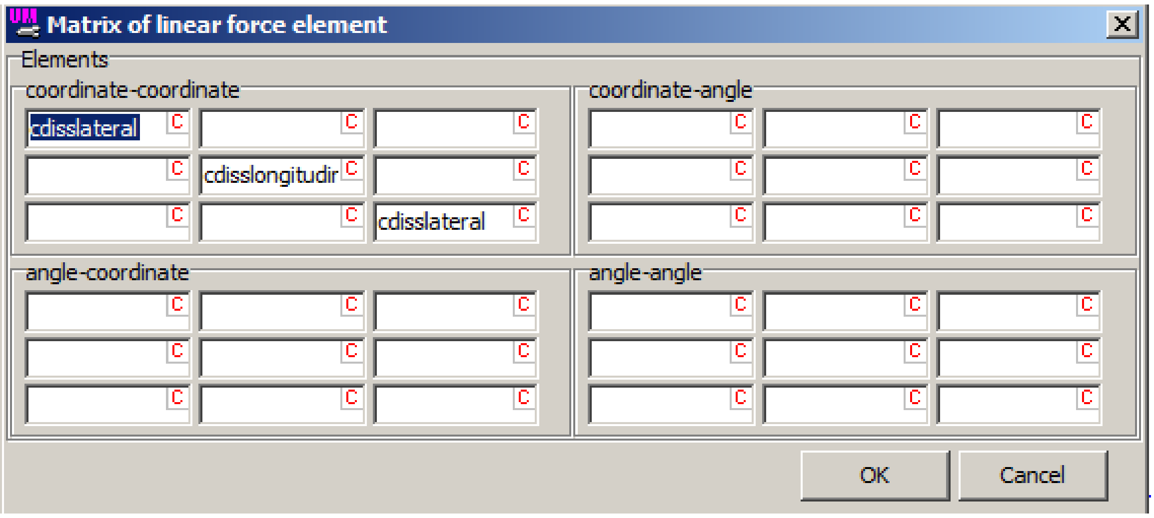
Coordinates **X, Z** of the end points of elastic element in undeformed state coincides with **Electricmotor.Body, Y=0.07**.

Please draw attention to the rotation on -90 degrees about the **X** axis (Figure 2.51), to make the orientation of SC of the force element coinciding with the SC of the **Electricmotor.Body**.

Set the stiffness matrices of elastic force element (a) and dissipative matrix of the element (b) as it is shown in Figure 2.52.



a)



b)

Figure 2.52.

Initialize the identifiers as **cStifflateral=1.0E6**, **cStifflongitudinal=1.0E6**. The corresponding values for the damping elements are **cDisslateral=1.0E3, cDisslongitudinal=1.0E3.**

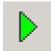
2.2.8. Preparing for simulation

1. Save the model as **Vibrostand** with the help of the main menu or the corresponding button.
2. Generate and compile equations of motion if equations are generated in symbolic form.

If no errors detected, the model is ready for simulation.

2.2.9. Simulation

Let us compute the vertical components of forces in force elements coupling the electric motor and the platform, when the rotor of the motor rotates with the constant angular velocity **nu** = 1620 r.p.m. As an example consider the rear right pair of elements. Let us compute displacements and accelerations of a center of plate under the electric motor as well.

1. Run the **UM Simulation** with the **F9** key or by clicking the  button on the tool panel.
2. Open a new animation window to visualize the simulation process,
3. **Tools | Animation window…**.
4. Use the **Analysis | Simulation…** menu command to open the **Object simulation inspector**.
5. Use the **FEM Subsystems | Image** tab of the **Object simulation inspector** to change the flexible platform image if necessary.

2.2.9.1. Calculating the equilibrium position and natural frequencies

Let us calculate the equilibrium position of the stand.

1. If the **Objection simulation inspector** is active close it by the Close button.
2. From the **Analysis** menu select **Static and linear analysis…** or press the F8 key. Window of linear analysis appears.
3. Select the **Equilibrium** tab. Start the calculation by the **Run computations** button, Figure 2.53. Calculation process might take some time.

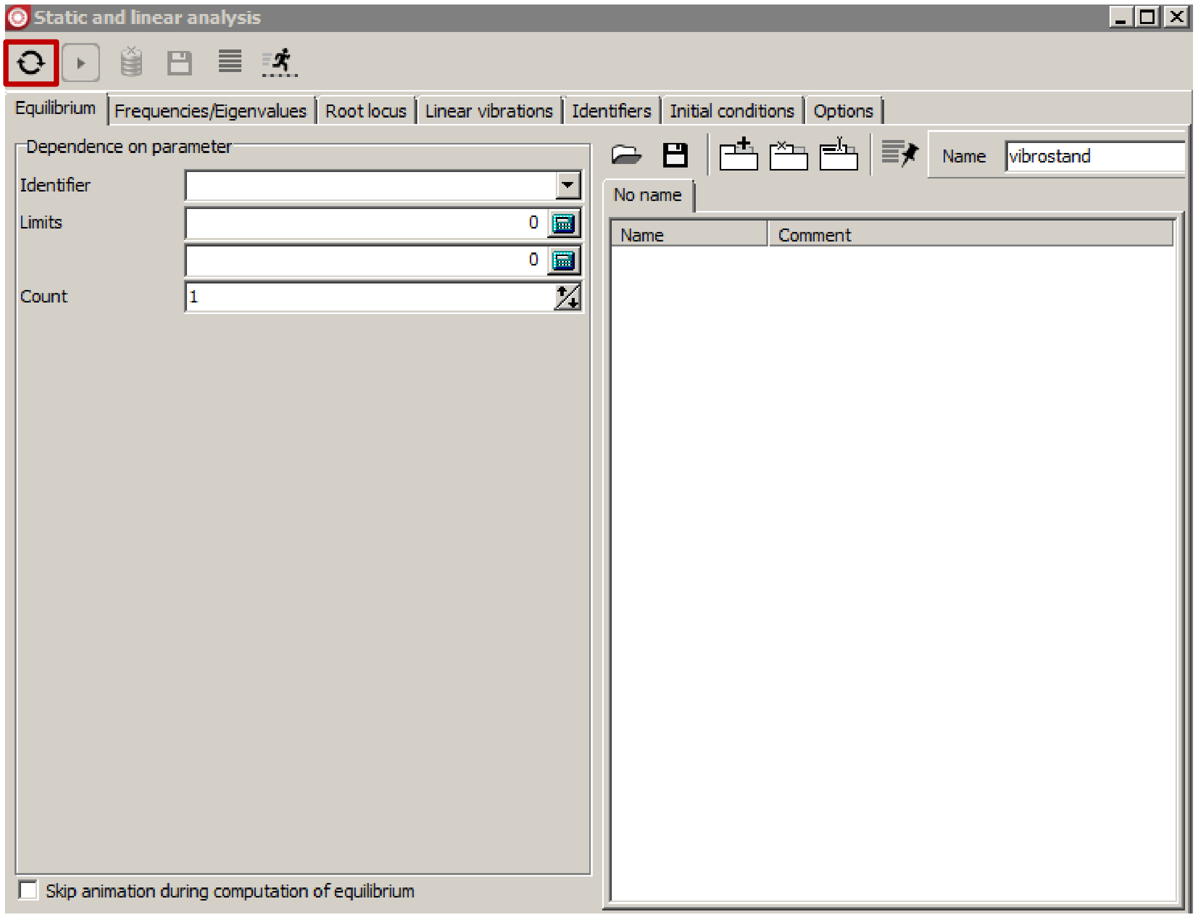


Figure 2.53.

Now we need to save current coordinates, which correspond to the found equilibrium position, to a file of initial conditions.

4. Select the **Initial conditions** tab. Click the  button and save current initial conditions to the **equilibrium.xv** file.

Note. Just found values of coordinates correspond to equilibrium position are correct for the current values of identifiers of the model only. Any changes of identifiers will lead that found above set of coordinates will not correspond to equilibrium position any more. In such a case you need to repeat the calculation of equilibrium position.

5. Select the **Frequencies/Eigenvalues** tab. Natural frequencies of the model are calculated after selecting button , Figure 2.54. You can change **Amplitude** and **Rate** of the animation of vibrations.

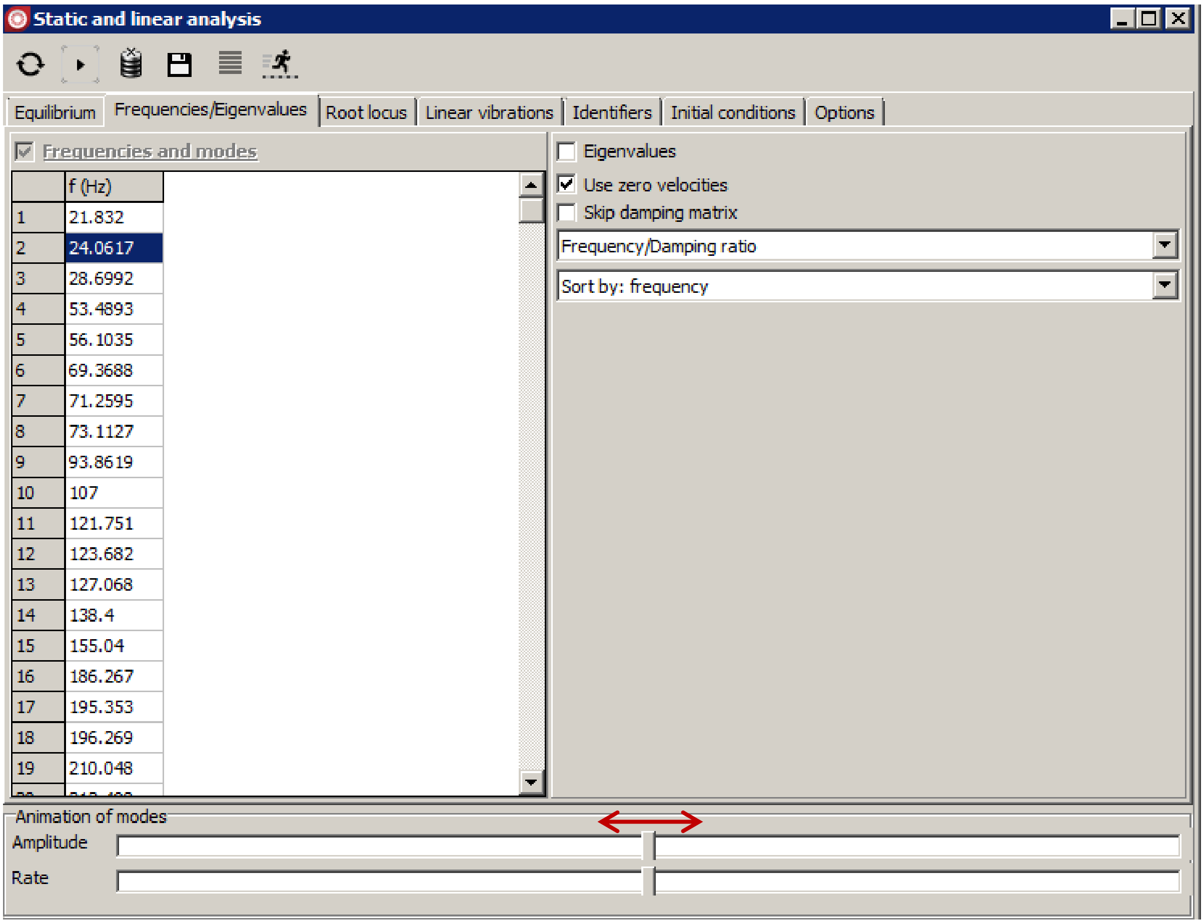
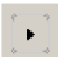


Figure 2.54.

6. You can see eigenmodes of the model in the animation window. To see an eigenmode just select it in the list and click the **Run** button . Now you can see that the animation window shows any selected eigenmode of the model. For example, animation of second eigenmode, 24.0617 Hz is shown at Figure 2.55. To stop animation click the **Stop** button .

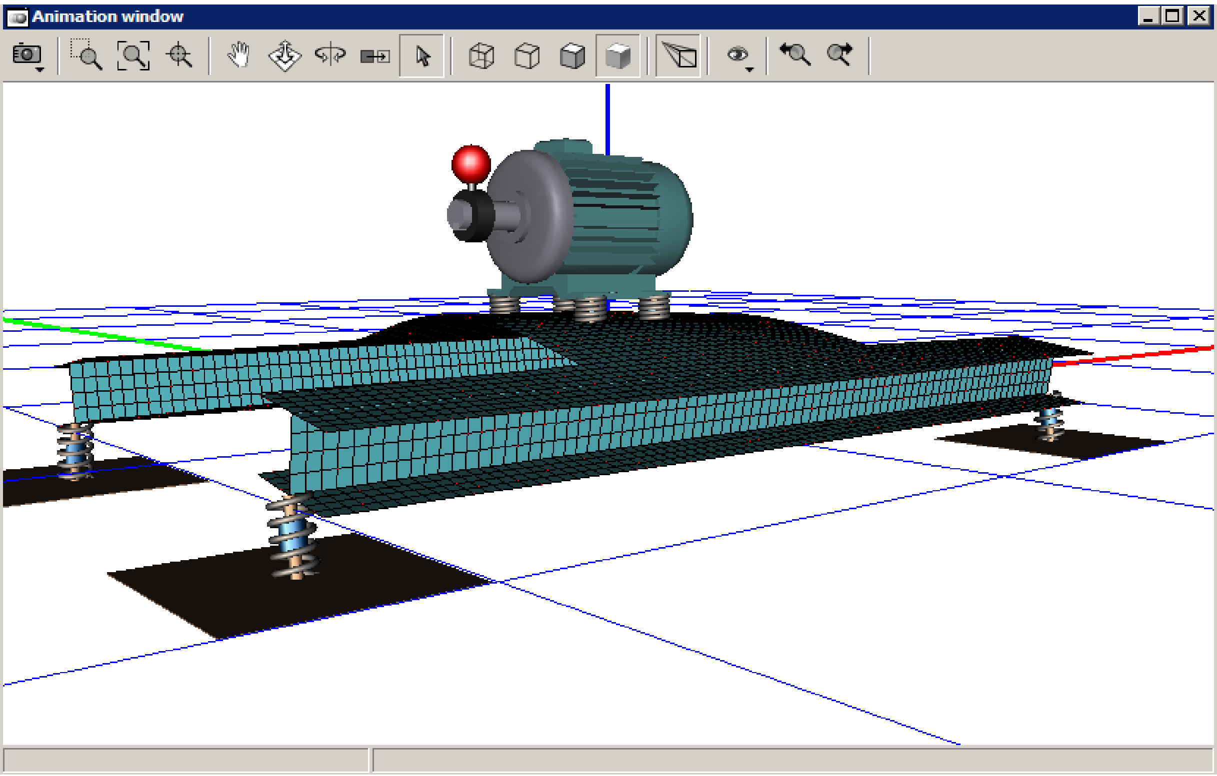


Figure 2.55. Animation of second eigenmode, 24.0617 Hz

Close the window of **Linear analysis**.

2.2.9.2. Integration of equations of motion

1. Open the **Wizard of variables** (the **Tools | Wizard of variables menu command**) and create variables for **Z** components of linear force elements **DamperMotorBR** Figure 2.56.

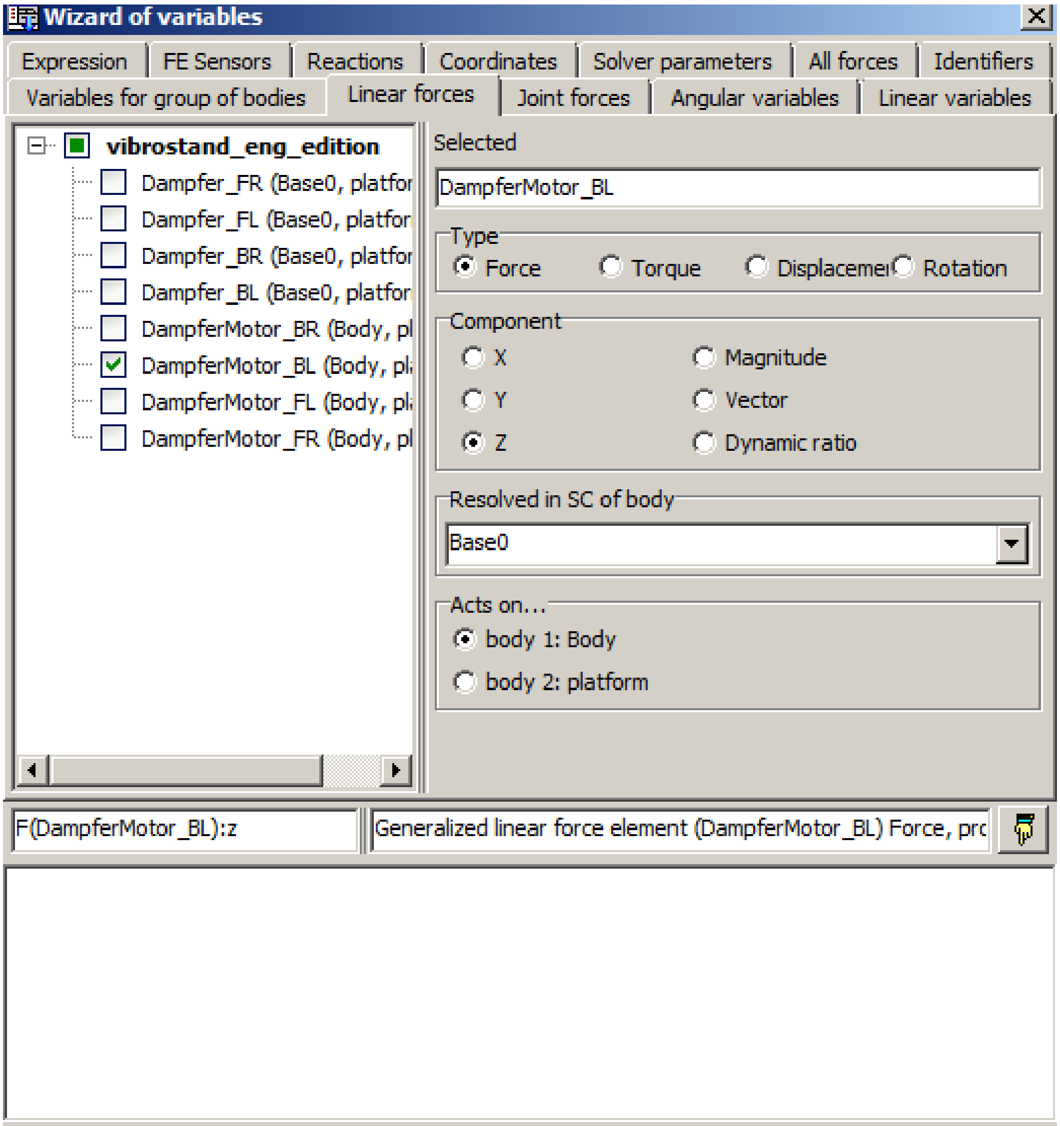


Figure 2.56.

2. Open a new graphical window (the **Tools | Graphical window…** menu command).
3. Drag the created variables into the graphical window by the mouse.
4. Let us select some node of the FE model where we will calculate Z components of position and acceleration. If the animation window does not show nodes of FE mesh, select the **FEM subsystems | Image**. Set **Image** to **full**. Turn on the **Image | Draw nodes** check box. Set non-zero value in Node image, for example 3, see Figure 2.57.

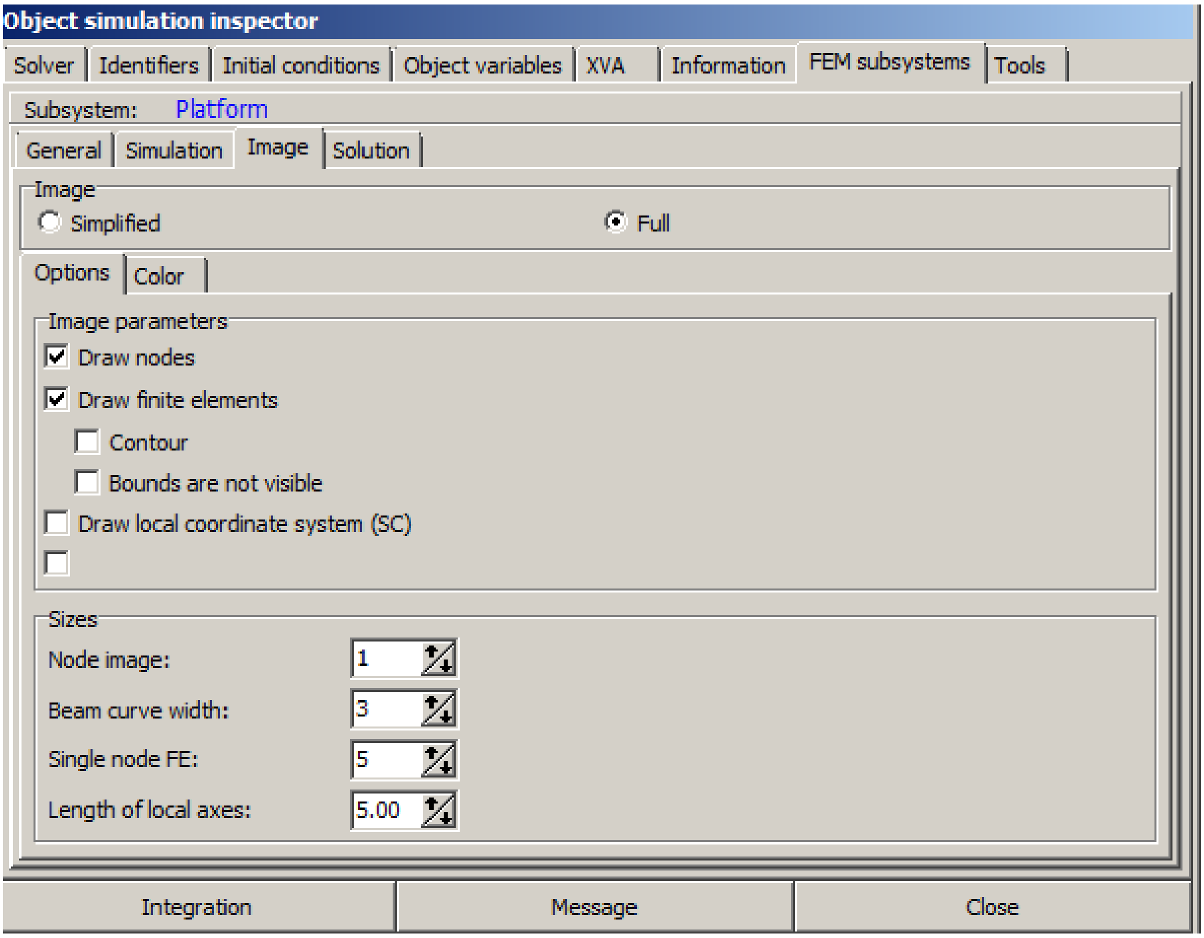


Figure 2.57.

Now we will plot oscillograms of a position and acceleration of some arbitrary node of the platform.

5. Select **Wizard of variables** and create two variables for calculation Z projections of position and acceleration of the node 3941 with approximate coordinates (0.05; 0; 0.06), see Figure 2.58, Figure 2.59.

Note. You can plot position and acceleration of any node you want. The only information you need is coordinates of the node. To get them point the mouse to the node in an animation window and you can see its coordinates in the status bar of the window, see Figure 2.58.

6. Select **Wizard of variables** and create variable for calculation stress component SX1 of node 3941 with approximate coordinates (0.05; 0; 0.06), see Figure 2.60.
7. Create two new graphical windows (**Tools | Graphical window…**) and drag and drop just created variables to these windows separately.

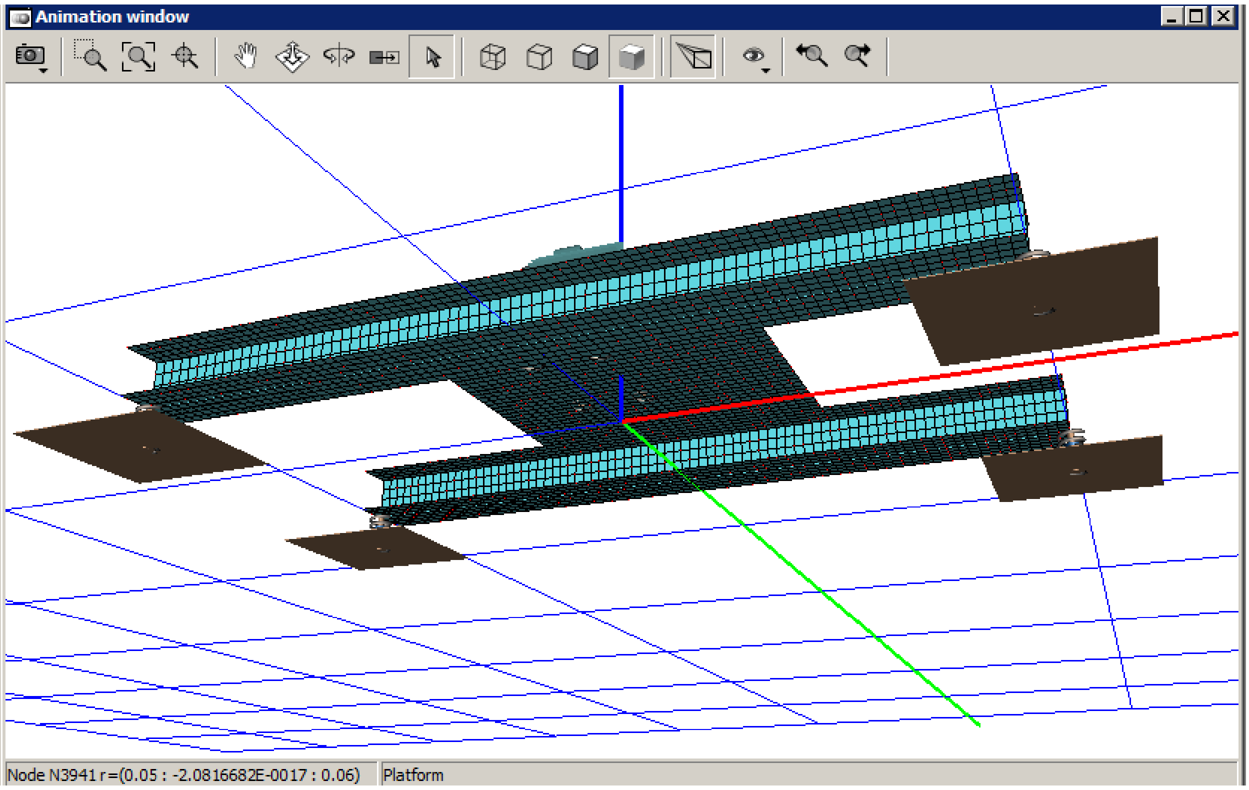

Animation window
Node N3941 r=(0.05 : -2.0816682E-0017 : 0.06)
Platform

Figure 2.58.

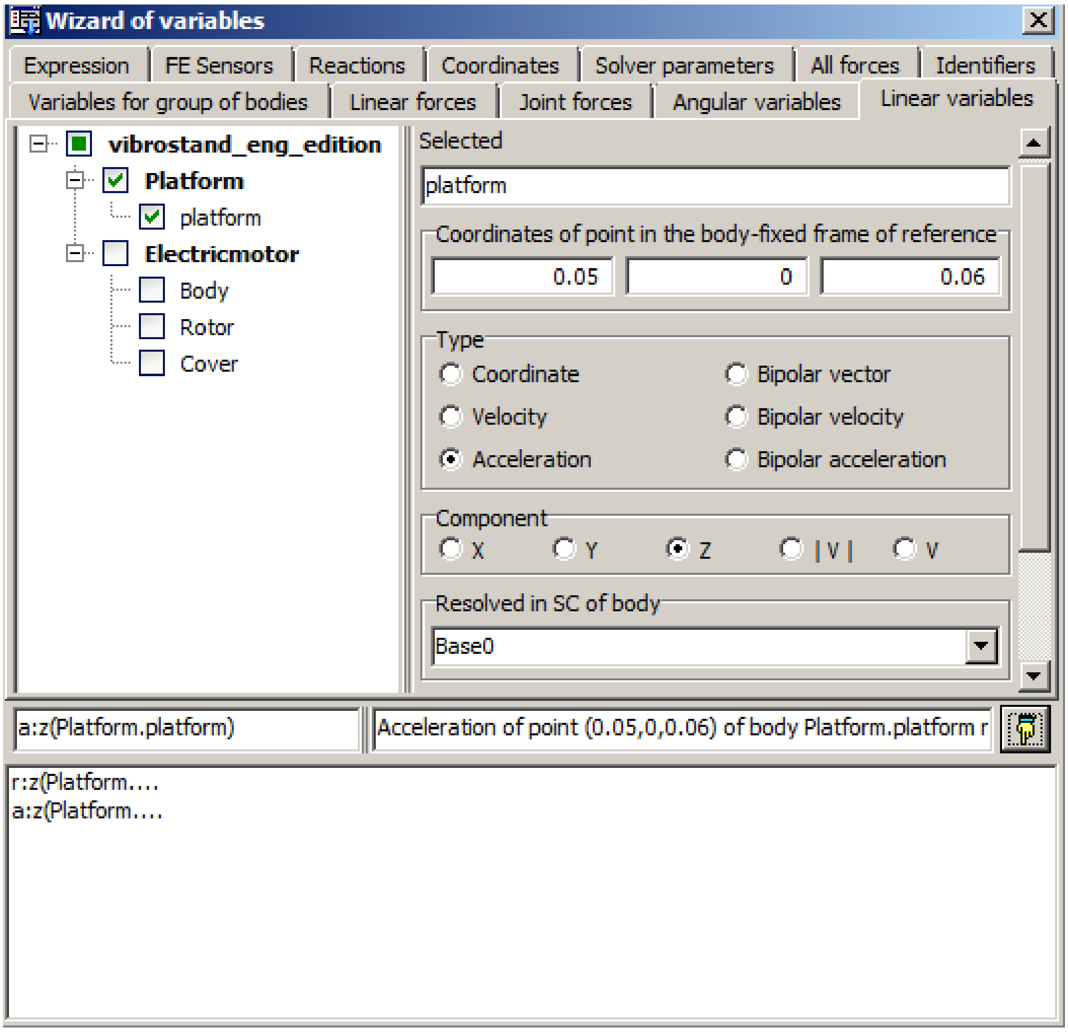

Wizard of variables
Expression
FE Sensors
Reactions
Coordinates
Solver parameters
All forces
Identifiers
Variables for group of bodies
Linear forces
Joint forces
Angular variables
Linear variables
vibrostand_eng_edition
Platform
platform
Electricmotor
Body
Rotor
Cover
Selected
platform
Coordinates of point in the body-fixed frame of reference
0.05
0
0.06
Type
Coordinate
Velocity
Acceleration
Bipolar vector
Bipolar velocity
Bipolar acceleration
Component
X
Y
Z
| V |
V
Resolved in SC of body
Base0
a:z(Platform.platform)
Acceleration of point (0.05,0,0.06) of body Platform.platform r
r:z(Platform....
a:z(Platform....

Figure 2.59.

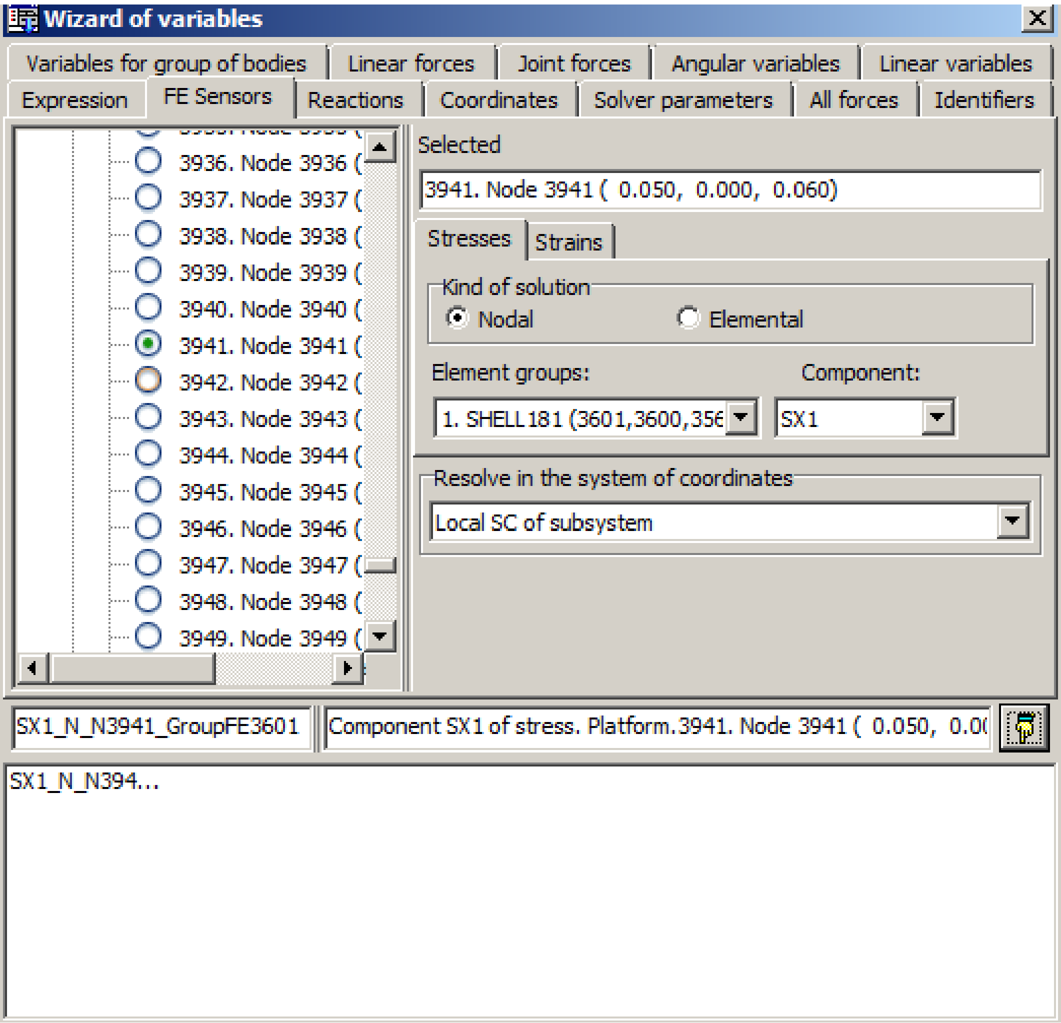


Figure 2.60.

8. Set the solver parameter on the **Solver** tab of the inspector as in Figure 2.61:

- **Solver = Park;**
- **Type of solving = Range Space Method (RSM);**
- **Simulation time = 10.0;**
- **Step size = 0.002;**
- **Error tolerance = 1E-8;**
- **Computing Jacobian Matrices = ON (always for flexible subsystems);**
- **Block-diagonal matrices = OFF.**

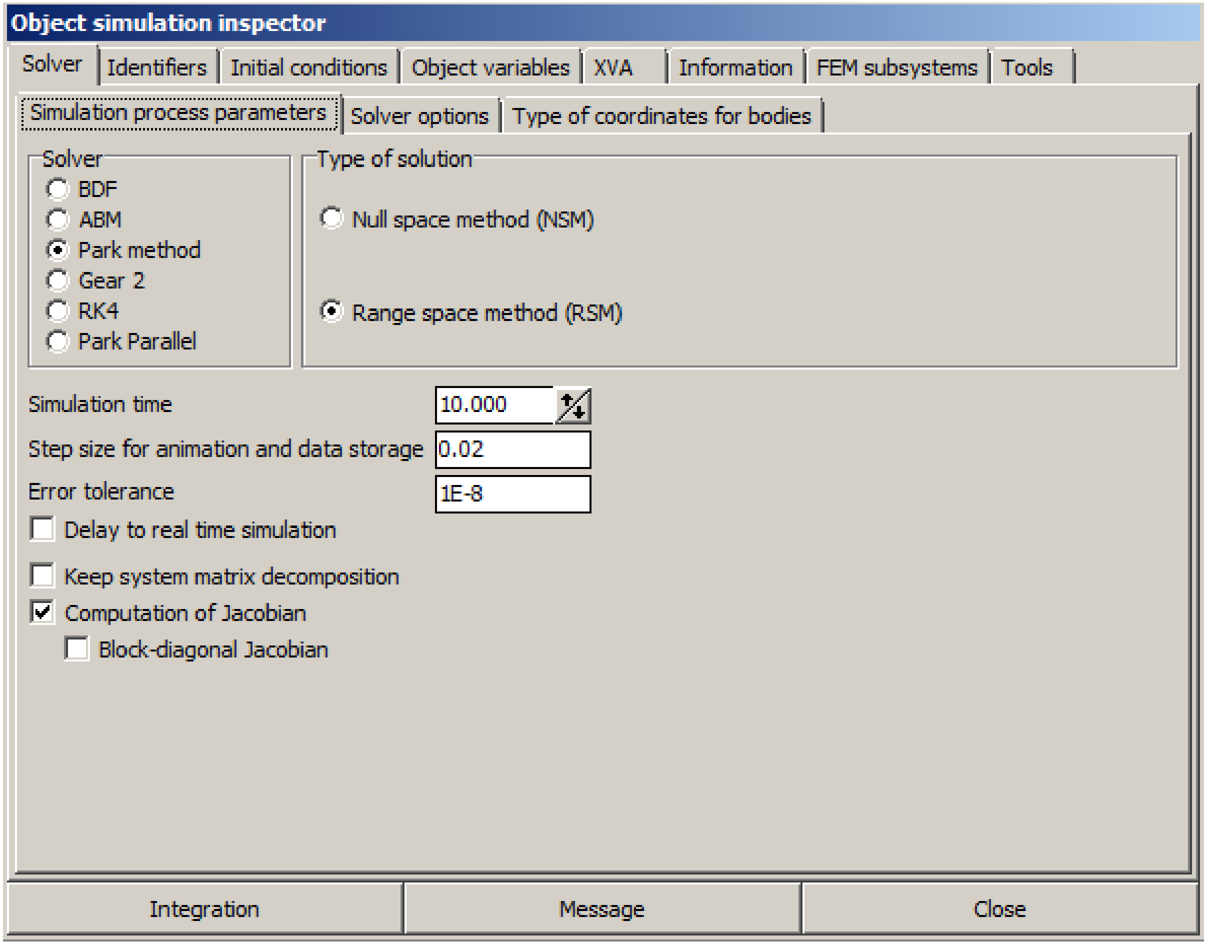


Figure 2.61.

9. On the FEM subsystems | Simulation tab switches gravity, internal dissipation as well as linear model should be ON. Set a=0.001, b=0 (Figure 2.62).

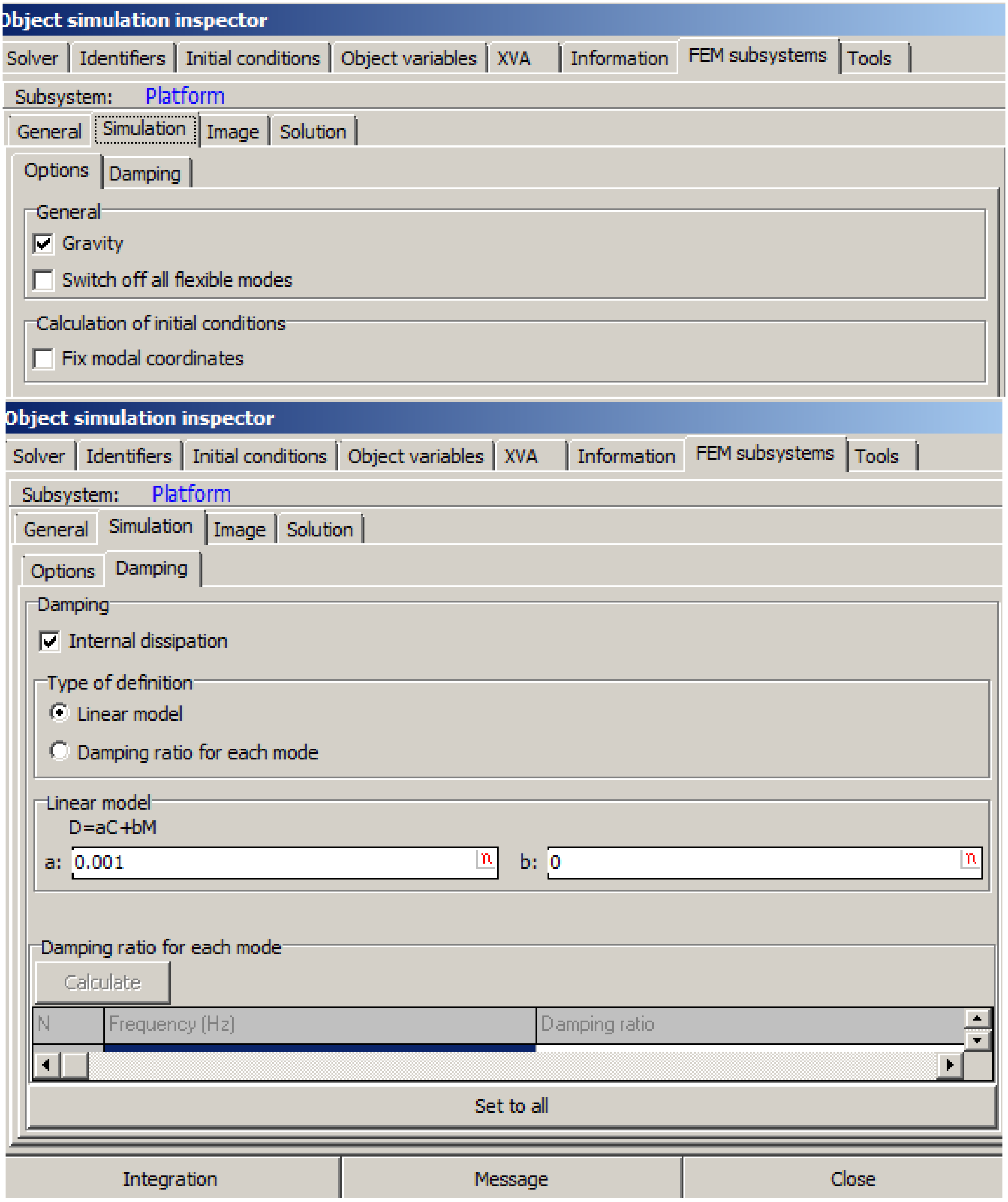


Figure 2.62.

10. Select the **Identifiers** tab in the **Object simulation inspector**. Select the **Vibrostand.Electricmotor** from the pull-down list of subsystems. Set the following values (Figure 2.63):

- **nu=1620** (27 revolutions per second);
- **tstart=0.5;**
- **tspeeding_up=2;**
- **tworking=3;**

- **tbraking=4.**

Note. Rotational speed of the rotor exceeds two first natural frequencies of the **vibrostand** that is why there will be resonance conditions during speeding-up the rotor.

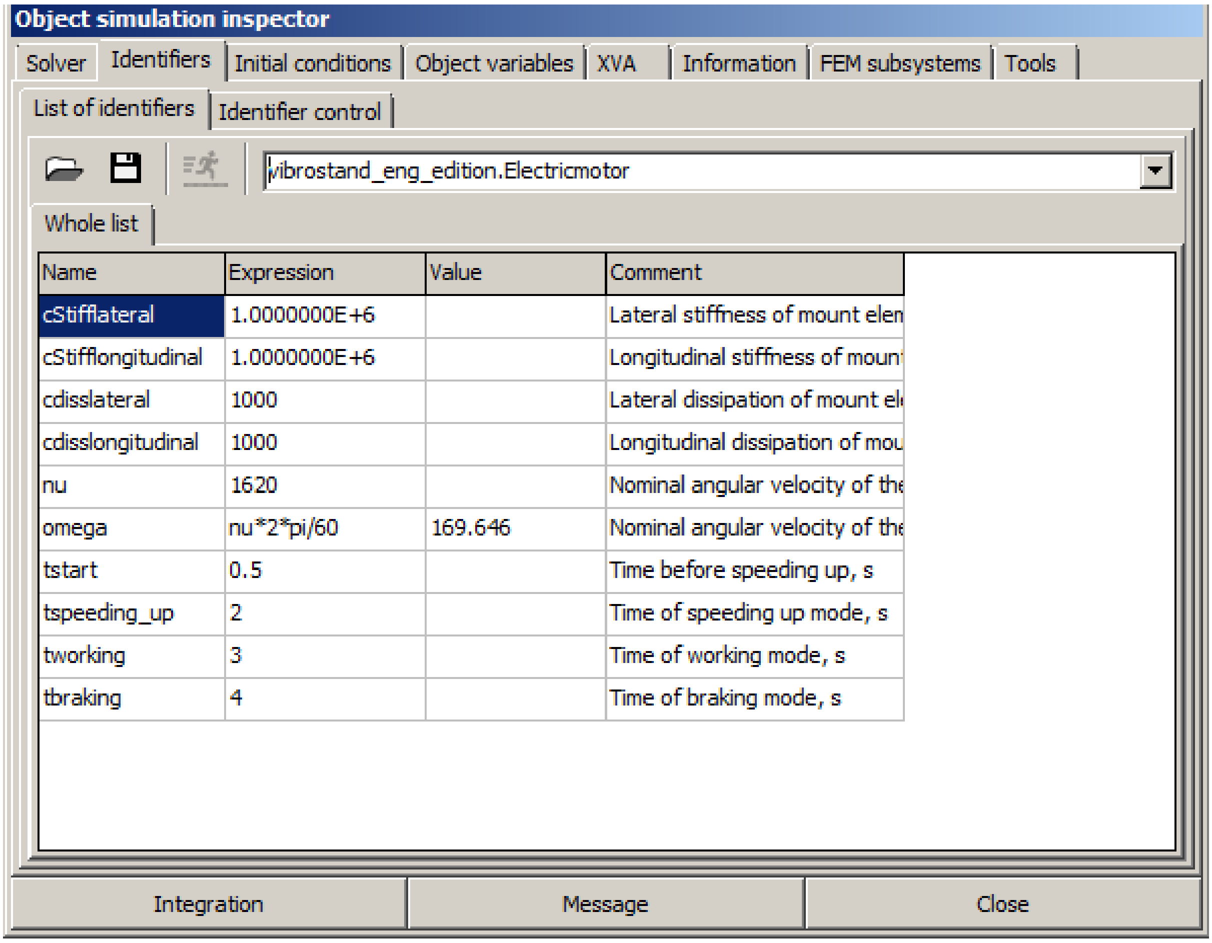
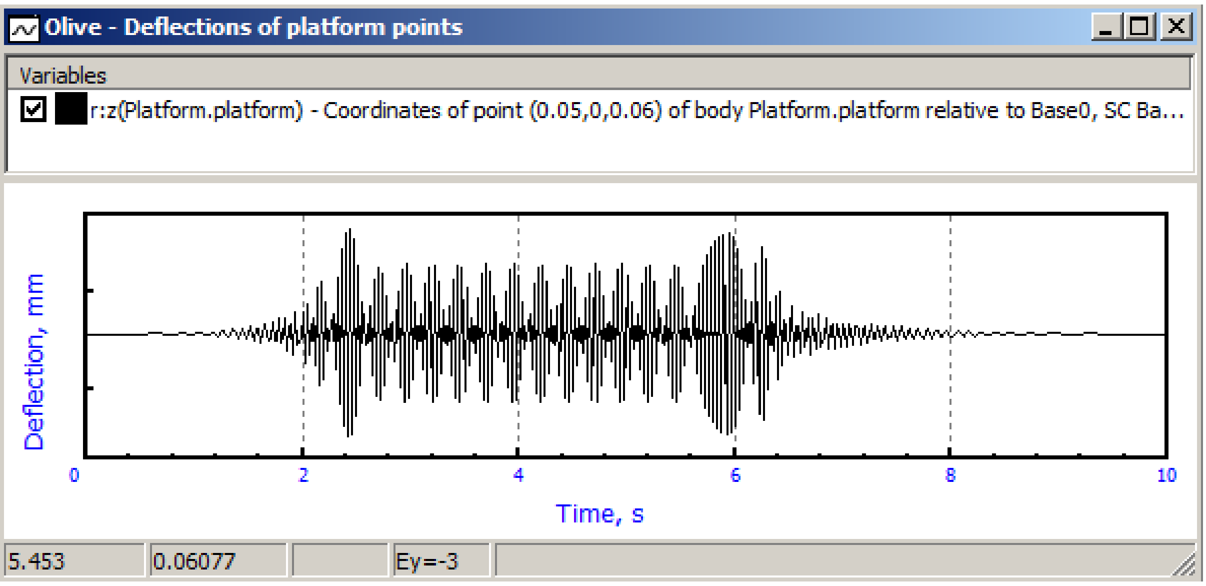
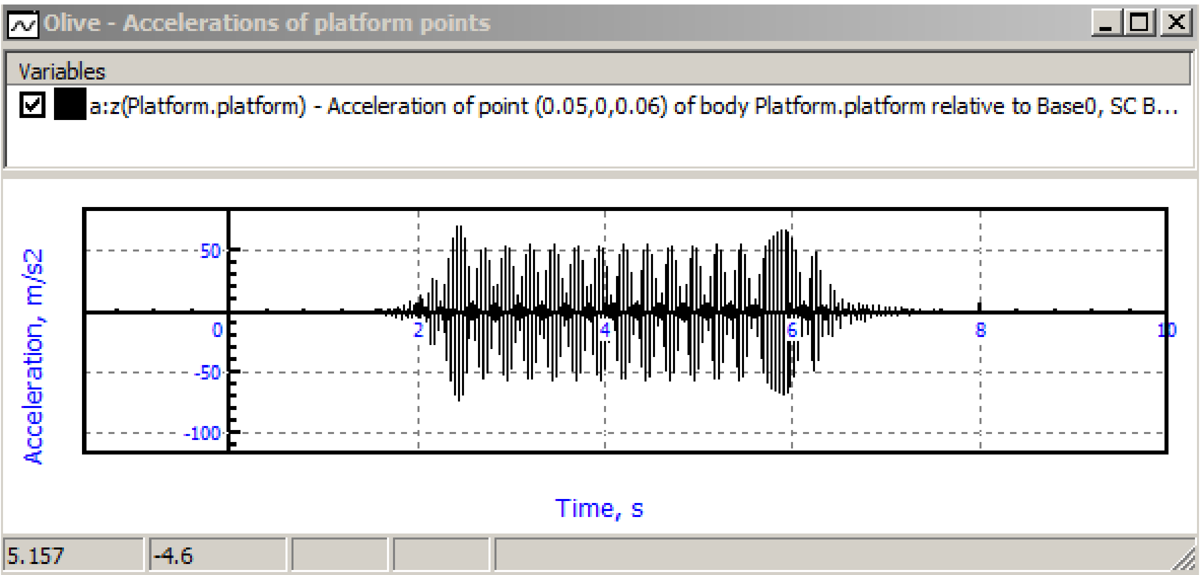
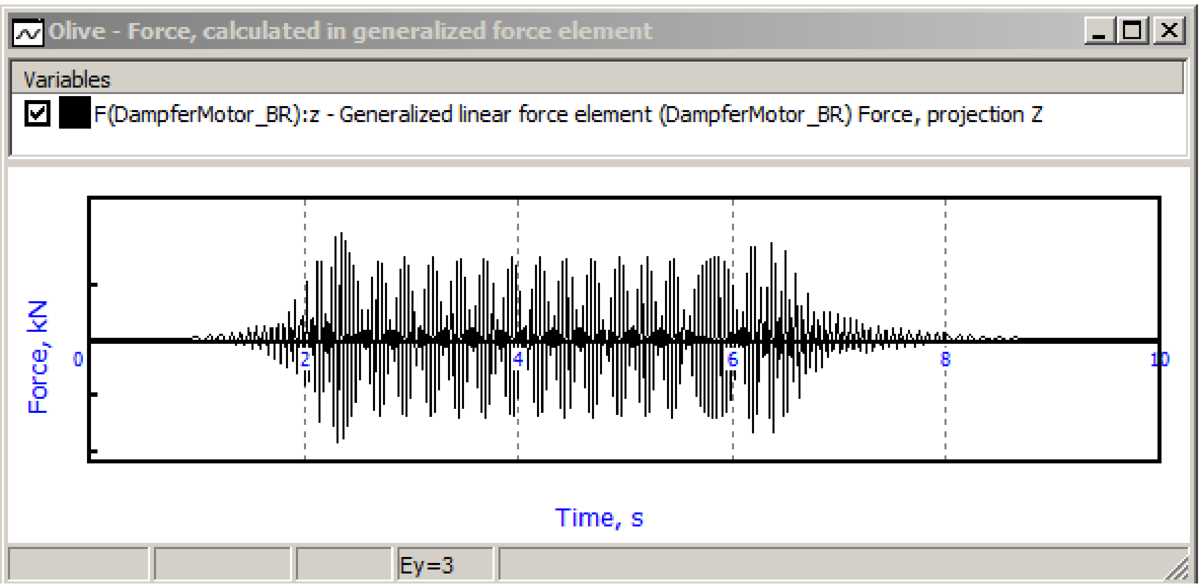


Figure 2.63.

11. Start the simulation process by the **Integration** button on the bottom part of the inspector. Figure 2.64 depicts some simulation results.


Olive - Deflections of platform points
Variables
r:z(Platform.platform) - Coordinates of point (0.05,0,0.06) of body Platform.platform relative to Base0, SC Ba...
Deflection, mm
0
2
4
6
8
10
Time, s
5.453
0.06077
Ey=-3


Olive - Accelerations of platform points
Variables
a:z(Platform.platform) - Acceleration of point (0.05,0,0.06) of body Platform.platform relative to Base0, SC B...
Acceleration, m/s2
50
0
-50
-100
2
4
6
8
10
Time, s
5.157
-4.6


Olive - Force, calculated in generalized force element
Variables
F(DampferMotor_BR):z - Generalized linear force element (DampferMotor_BR) Force, projection Z
Force, kN
0
2
4
6
8
10
Time, s
Ey=3

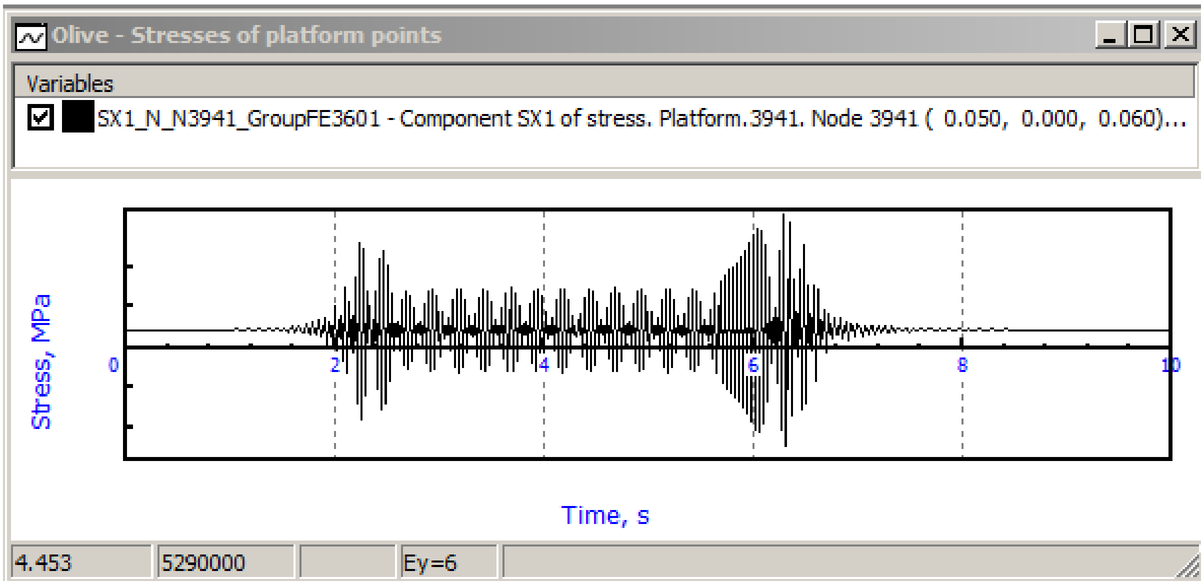


Figure 2.64.

To estimate the influence of the platform flexibility, the following operations could be done.

12. The option **switch off all flexible modes** should be on (Figure 2.62).
13. Run simulation.
14. Copy variables in graphical windows as static using popup menus (contact menu in a graphical window, **Copy as static variables** menu item).
15. Change the **option switch of all flexible modes** to off (Figure 2.62).
16. Repeat the simulation.
17. Compare simulation results.