1. 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 **11_UM_FEM.pdf** of UM user's manual, which is available in the **{um_root}\manual** directory and in the Internet via this link: http://www.umlab.ru/download/50/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 elastic subsystems** in the **UMInput.exe**. 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 (part11.pdf). Thus using **UM FEM** module is possible if ANSYS software is available on the user's computer.

---

[1] http://www.umlab.ru/download/50/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)

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 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.

Copyright and trademarks

2. Slider-crank mechanism

Here the example model of the slider-crank mechanism (see Fig. 2.1) is considered. There is **Slider_crank_all** model in the **{um_root}\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. 11.1.

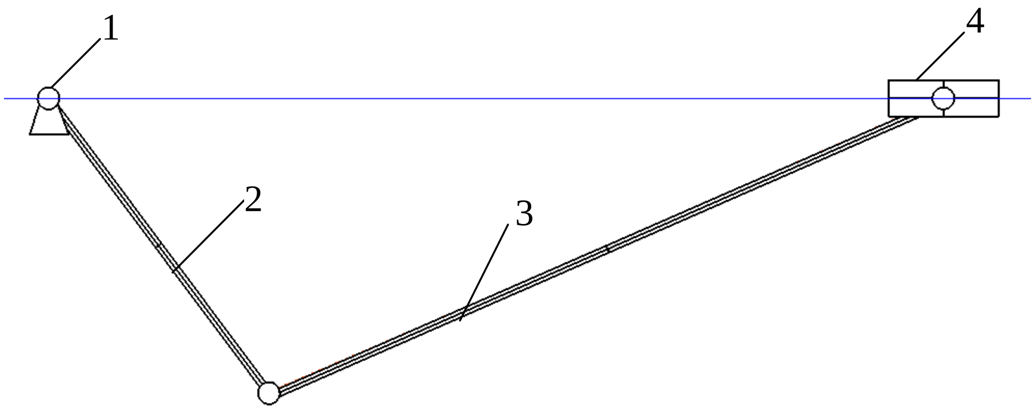


Figure 2.1. Slider-crank mechanism: 1 – base, 2 – crank, 3 – con-rod, 4 – slider.

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 FEA-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 are represented as a separate subsystem of **Linear FEM subsystem** type.

Create a directory for the future 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. 2.2.2 of this manual. Before that you should copy the **{um_root}\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_root}\Samples\Flex\flexbeam\input.fss** to the **.\flexbeam** and start reading from the sect. 2.3 of this manual.

2.1. Preparing ANSYS environment

We will use **ANSYS** software for preparing data for simulation of dynamics of elastic body. After creating FEA model a calculation of the static and eigen-modes 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_root}\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_root}\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.

2.2. 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 eigen-values problem. There are two possible mathematical formulations of this problem:

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

The **{um_root}\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.

2.2.1. Working under ANSYS environment

1. Copy the **flexbeam&mass21.ans** file from the **{um_root}\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_root}\bin\ansys_um.exe** manually. The main window of **ansys_um** appears, Fig. 2.2.
5. Point to the **General** tab. The **ANSYS results file (*.rst)** set to **.\flexbeam\flexbeam.rst**, **Target directory** set to **.\flexbeam**, see Fig. 2.2.

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

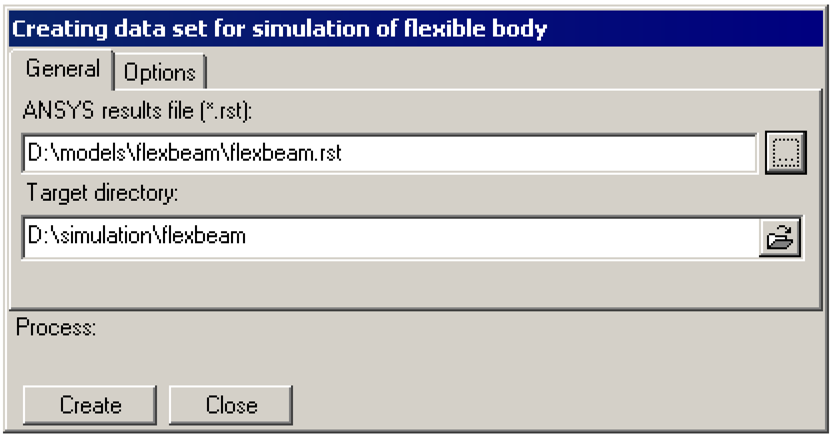


Figure 2.2. Main window of the **ANSYS_UM** program.

6. Point to the **Options** tab and turn off the **normalize modes** check box, Fig. 2.3. This case corresponds to creating the intermediate **input.fum** file. On the successive step we will use the **Wizard of elastic subsystems** to convert the data into UM-compatible form.

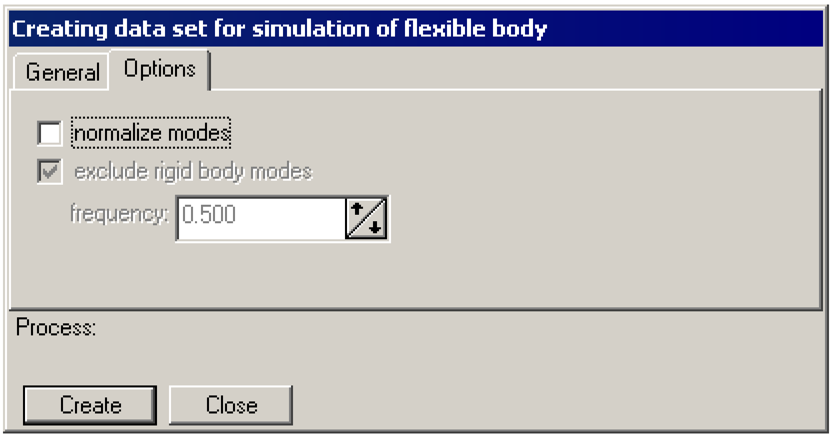


Figure 2.3.

Note. Using the **Wizard of elastic 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.

2.2.2. Wizard of elastic subsystems

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

Note. Using the wizard of flexible subsystem 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, Fig. 2.3. 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 subsystem data.

1. Run **UM Input** program **(uminput.exe)**.
2. Click the **Tools/Preparing flexible subsystems** menu item. The main window of the wizard of flexible subsystem data appears.
3. Click the  and select a file for the **Data file**, Fig. 2.4, 2.5.

Figure 2.4

Wizard loads and shows the data, Fig. 2.6. The **General** tab shows summary information about elastic subsystem, see Fig. 2.6.

The **Position** tab (see Fig. 2.7) 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 Fig. 2.7.

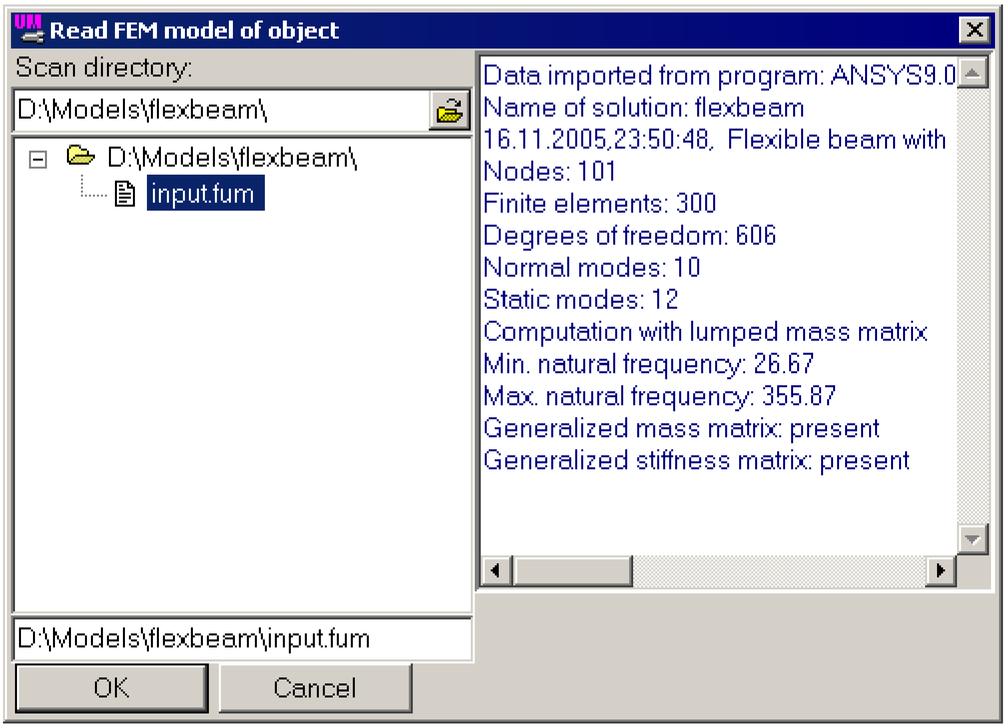


Figure 2.5

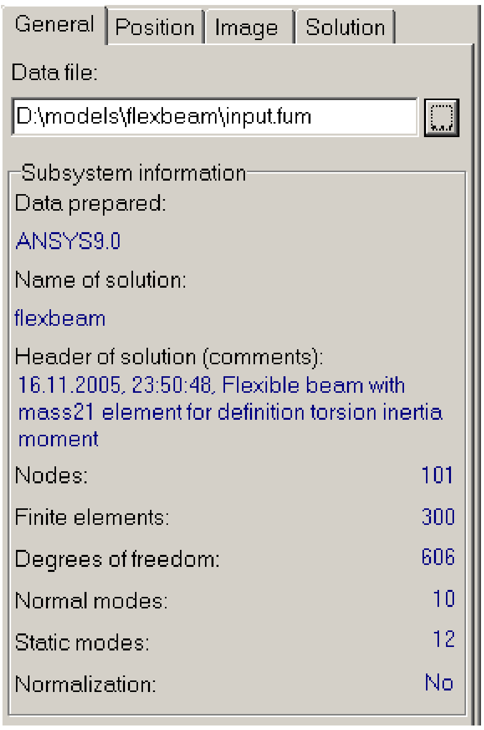


Figure 2.6

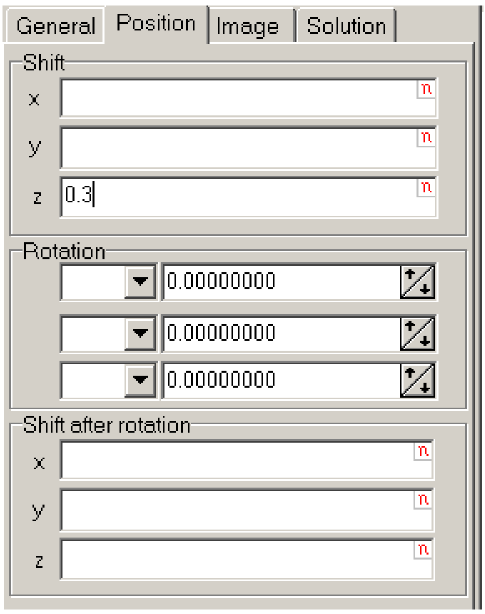


Figure 2.7

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 Fig. 2.8.

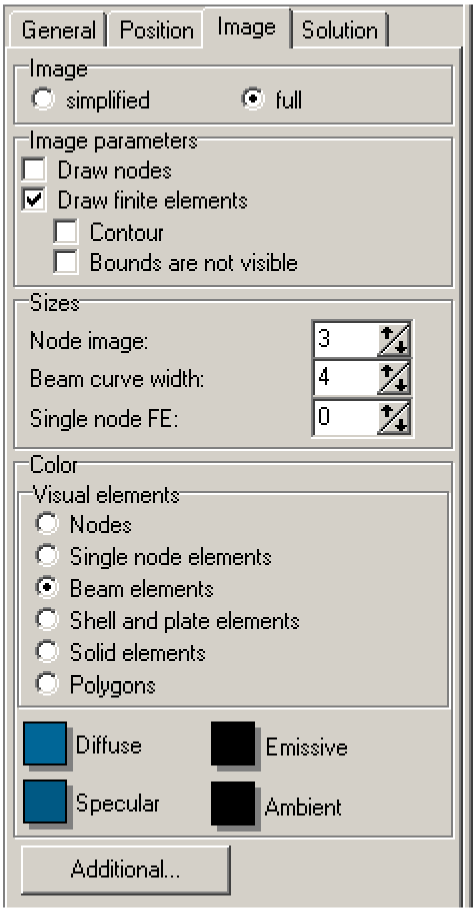


Figure 2.8

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 Fig. 2.9. 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.

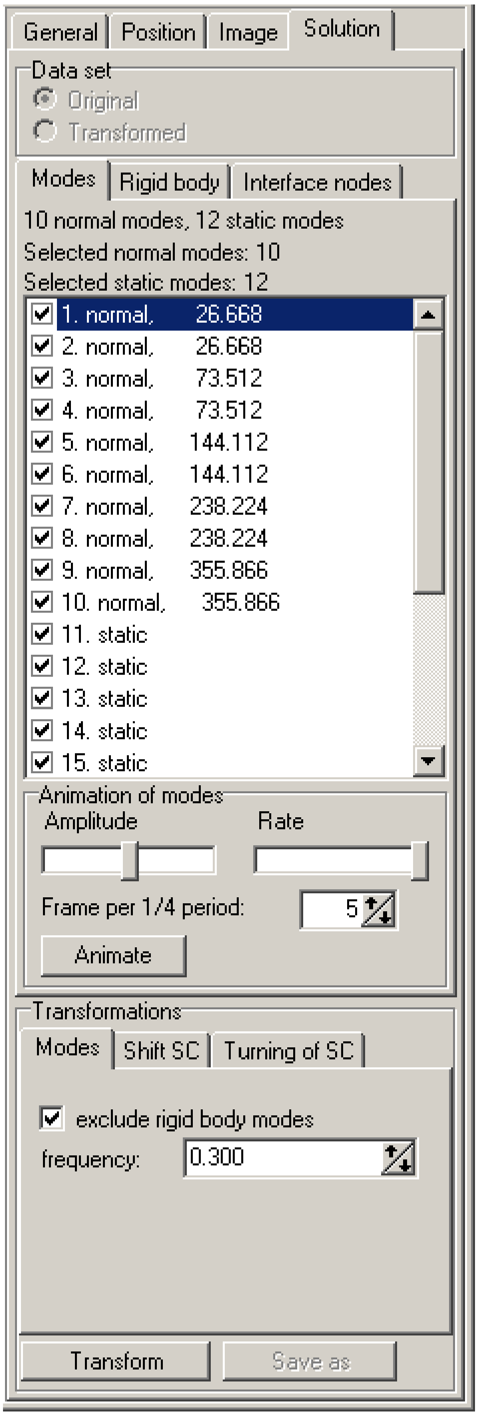

General
Position
Image
Solution
Data set
Original
Transformed
Modes
Rigid body
Interface nodes
10 normal modes, 12 static modes
Selected normal modes: 10
Selected static modes: 12
1. normal, 26.668
2. normal, 26.668
3. normal, 73.512
4. normal, 73.512
5. normal, 144.112
6. normal, 144.112
7. normal, 238.224
8. normal, 238.224
9. normal, 355.866
10. normal, 355.866
11. static
12. static
13. static
14. static
15. static
Animation of modes
Amplitude
Rate
Frame per 1/4 period:
5
Animate
Transformations
Modes
Shift SC
Turning of SC
exclude rigid body modes
frequency:
0.300
Transform
Save as

Figure 2.9

9. Turn on the **Transformations / exclude rigid body modes (**Fig. 2.9**)**.
10. Set **Transformations / Frequency** to **0.3 (**Fig. 2.9**)**.
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 Fig. 2.10.

Figure 2.10

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, Fig. 2.11.

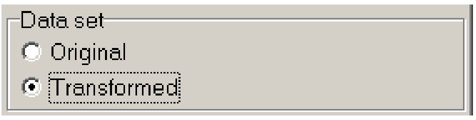


Figure 2.11

13. Click the **Save as** button. In the dialog set **Path to subsystem data** and click the **Save** button, see Fig. 2.12. Please, note, that the latter directory will further serve as a subsystem name.

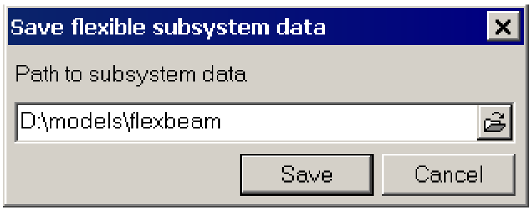


Figure 2.12

Preparing the data for flexible subsystem is done.

2.3. 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 MBS** menu command or click the  button. New constructor window appears.

2.3.1. Creating graphical objects

1. Load a graphical object from the **{um_root}\bin\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 Fig. 2.13.

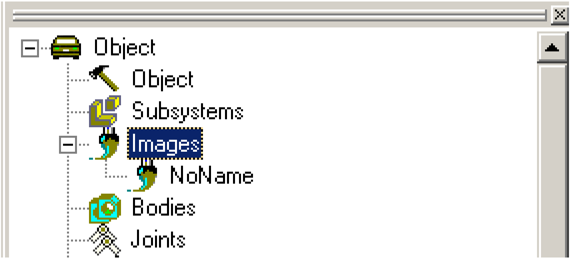


Figure 2.13

2. Select this element and set name to **Base0** in the data inspector (Fig. 2.14).

Figure 2.14.

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

Thus, three graphical objects are created.

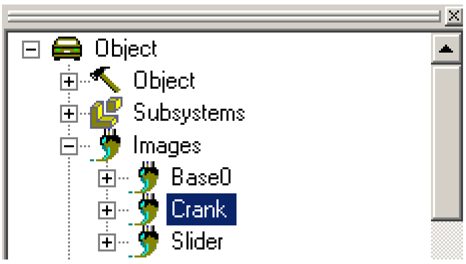


Figure 2.15

4. Select **Object** item in the tree of elements and set **Scene image** to **Base0**, see Fig. 2.16.

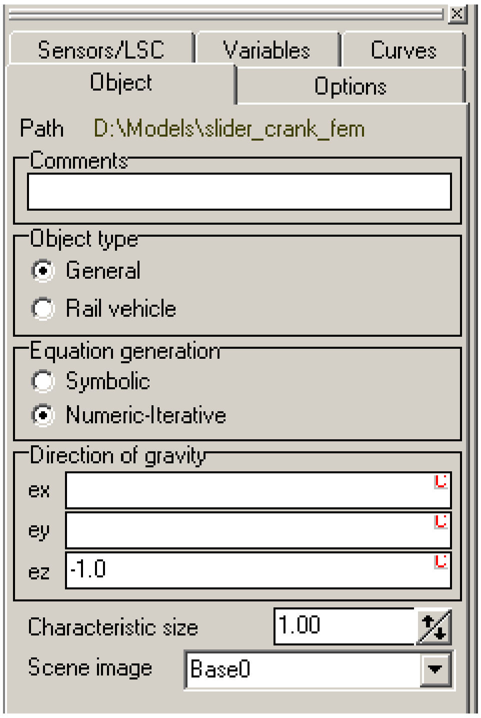


Figure 2.16

2.3.2. 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 Fig. 2.17.

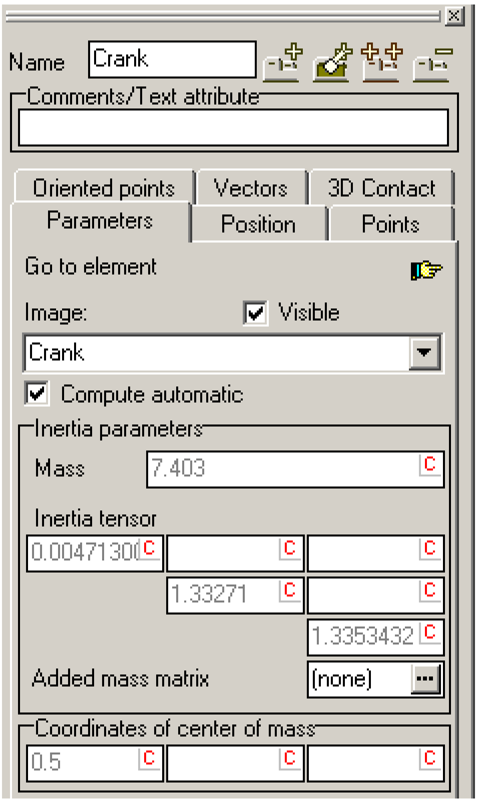


Figure 2.17.

2.3.3. Creating elastic subsystem

Now we introduce the elastic con-rid 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 (**Fig. 2.18.).

After reading elastic subsystem data inspector looks like the wizard of flexible subsystem data described in the sect. 2.2. 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.

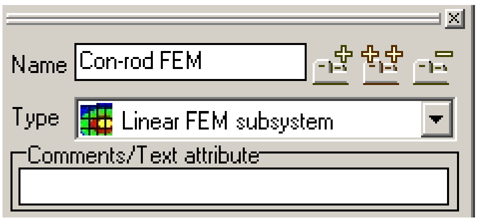


Figure 2.18.

2.3.4. 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 **Base0_Crank**. Select **Rotational** type for the joint and set **Y** axis as **Joint vectors**, see Fig. 2.19.

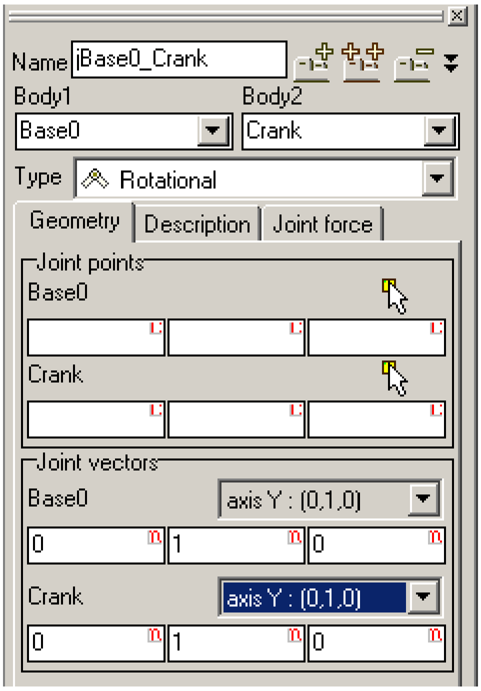


Figure 2.19.

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 Fig. 2.20. Press **Enter**. The window **Initialization of values** for new identifiers appears. Set identifiers value as follows: **torque = 100**, **cdiss_crank = 10**.

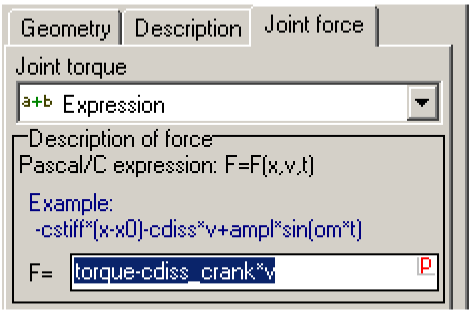


Figure 2.20.

4. Add the rest three joints as it is shown in the Fig. 2.21.

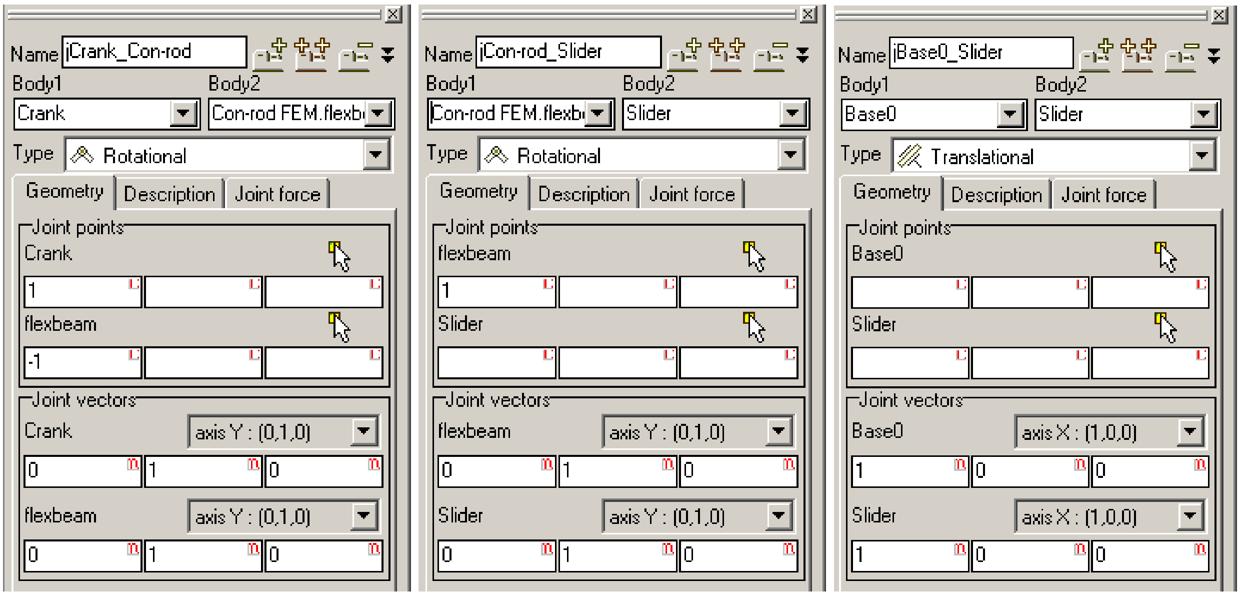


Figure 2.21.

2.3.5. Preparing for simulation

1. Save the model as **Slider_crank_fem** (**File/Save as** menu command), see Fig. 2.22.

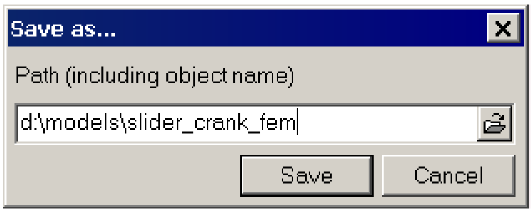


Figure 2.22.

2. Generate and compile equations of motion. Click the **Object/Generate equations** menu item. The new dialog window appears. Turn on the **Compile equations** flag. Change the **Output language** if necessary and click the **Generate** button (Fig. 2.23.).

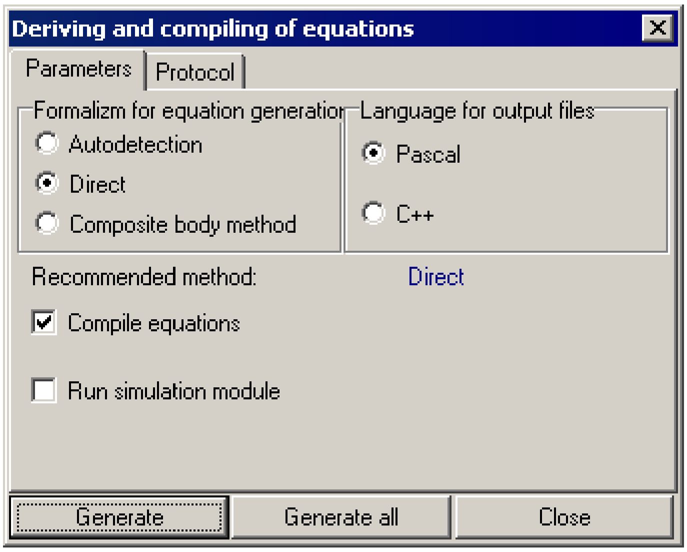


Figure 2.23.

Now the model is ready for simulation.

2.4. 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 **Crank_Con-rod** and **Con-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, Fig. 2.24. 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 Fig. 2.24 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, Fig. 2.25.

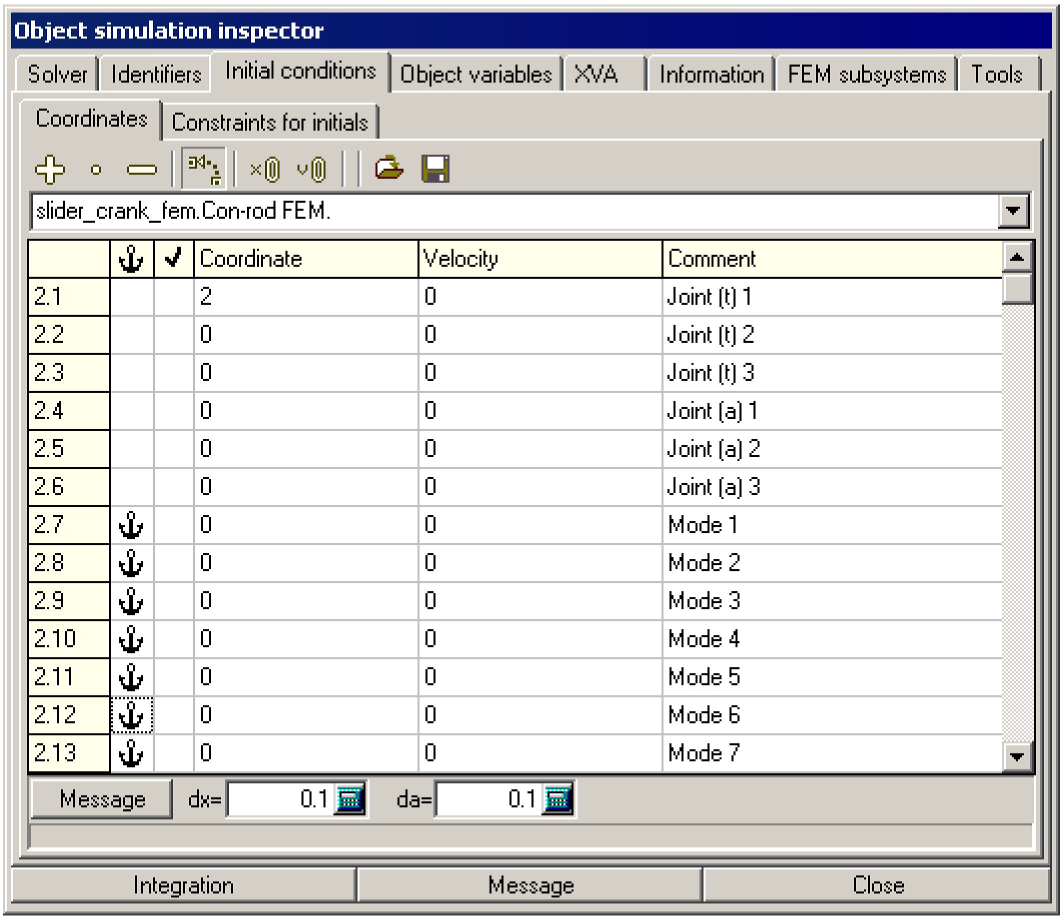


Figure 2.24.

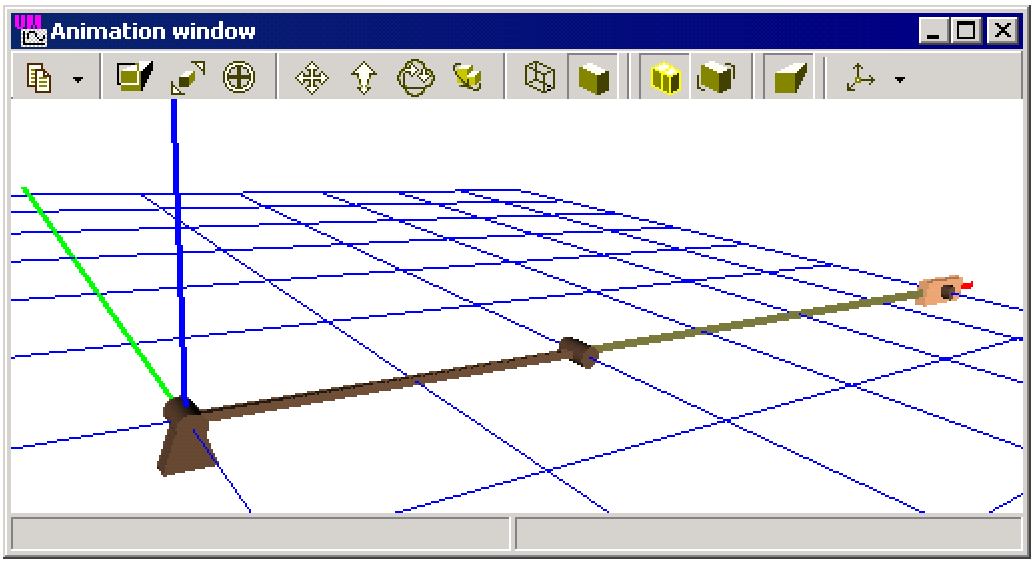


Figure 2.25.

6. Open new **graphical window** (**Tools/Graphical window** menu command).
7. Run **Wizard of variables** and create variables for reaction forces according to Fig. 2.26 and drag them to the graphical window.

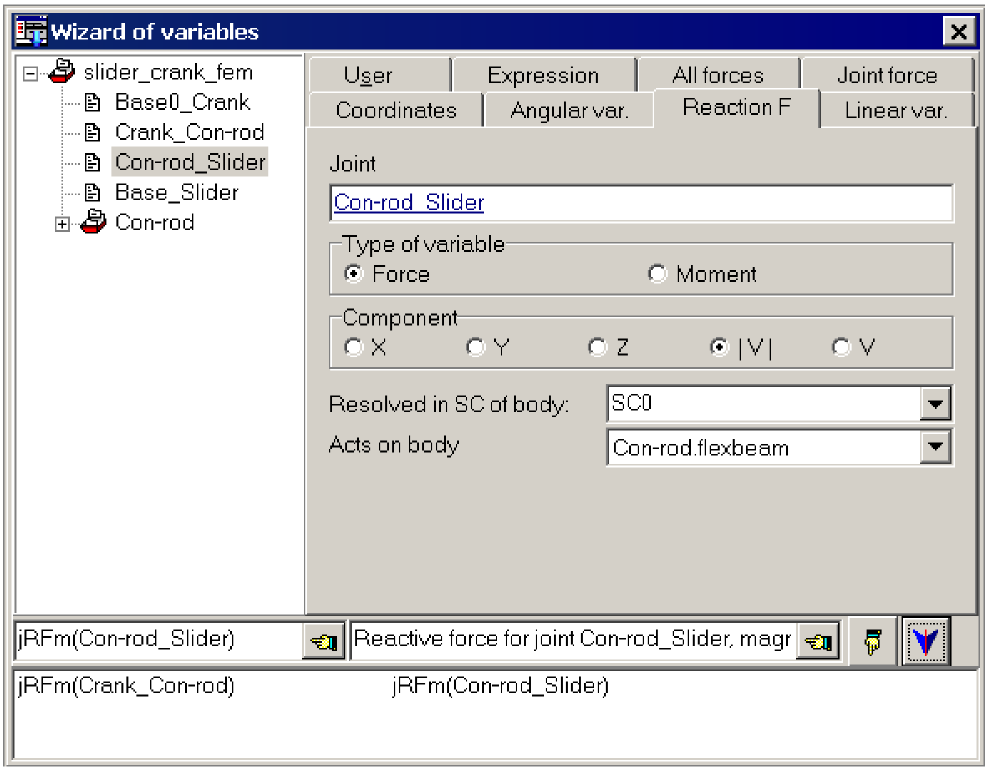


Figure 2.26.

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** = **0.001**.
    - **Error tolerance** = **1E-7**.
    - **Computing Jacobian matrices** = **on (always default)**.
    - **Block-diagonal matrices** = **off**.

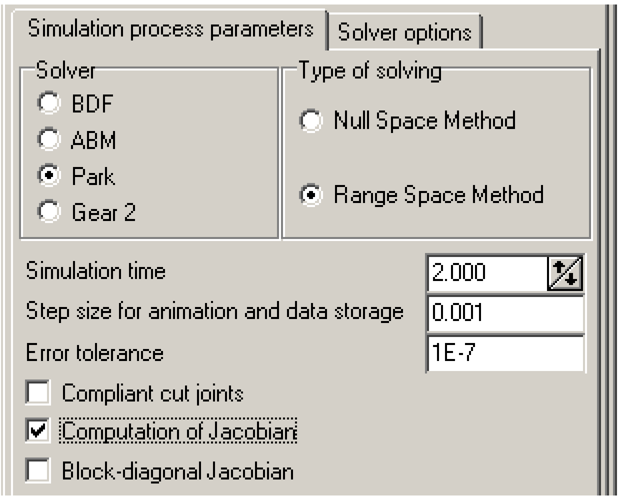


Figure 2.27.

9. Select the **FEM Subsystems/Simulation** tab and set up all options according to Fig. 2.28.

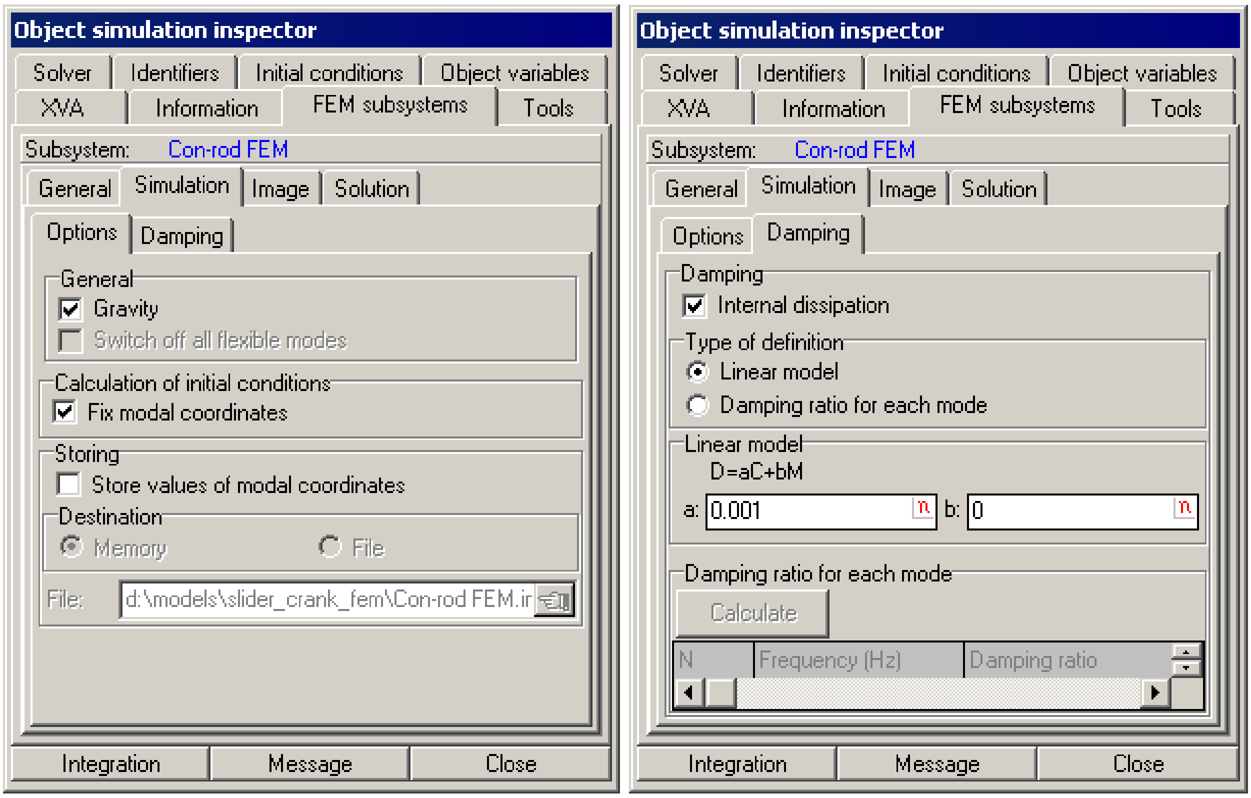


Figure 2.28.

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

You can see movement of the mechanism in the animation window (see Fig. 2.29) and oscillograms of reaction forces in the graphical window (see Fig. 2.30).

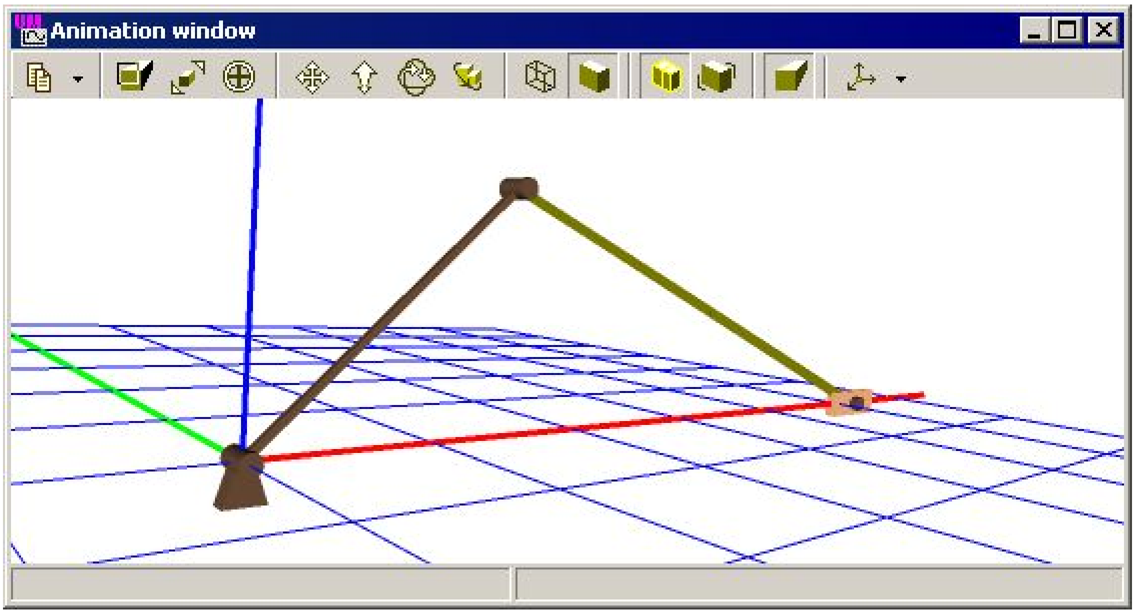


Figure 2.29. Animation window

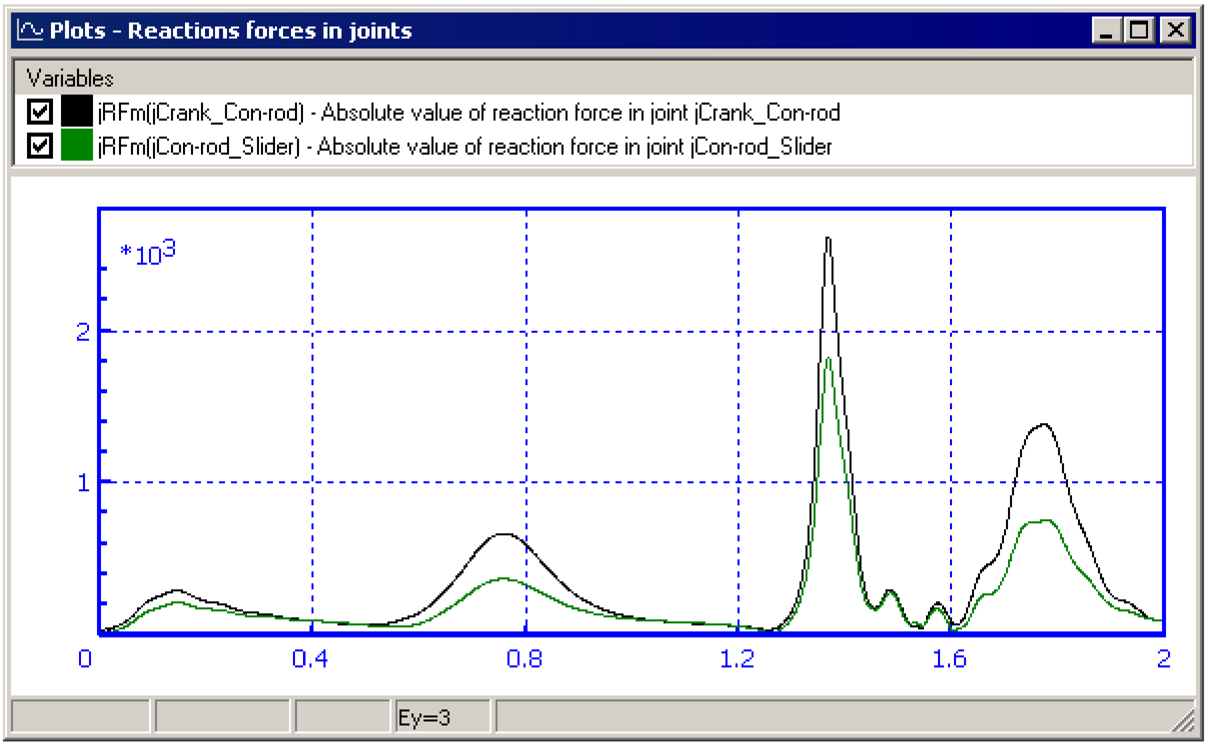


Figure 2.30. Graphical window

In order to estimate the influence of the elastic con-rod instead rigid one, open the **{um_root}\Samples\Flex\Slider_crank_all** model. Graphs of the reaction force are shown in the Fig. 2.31.

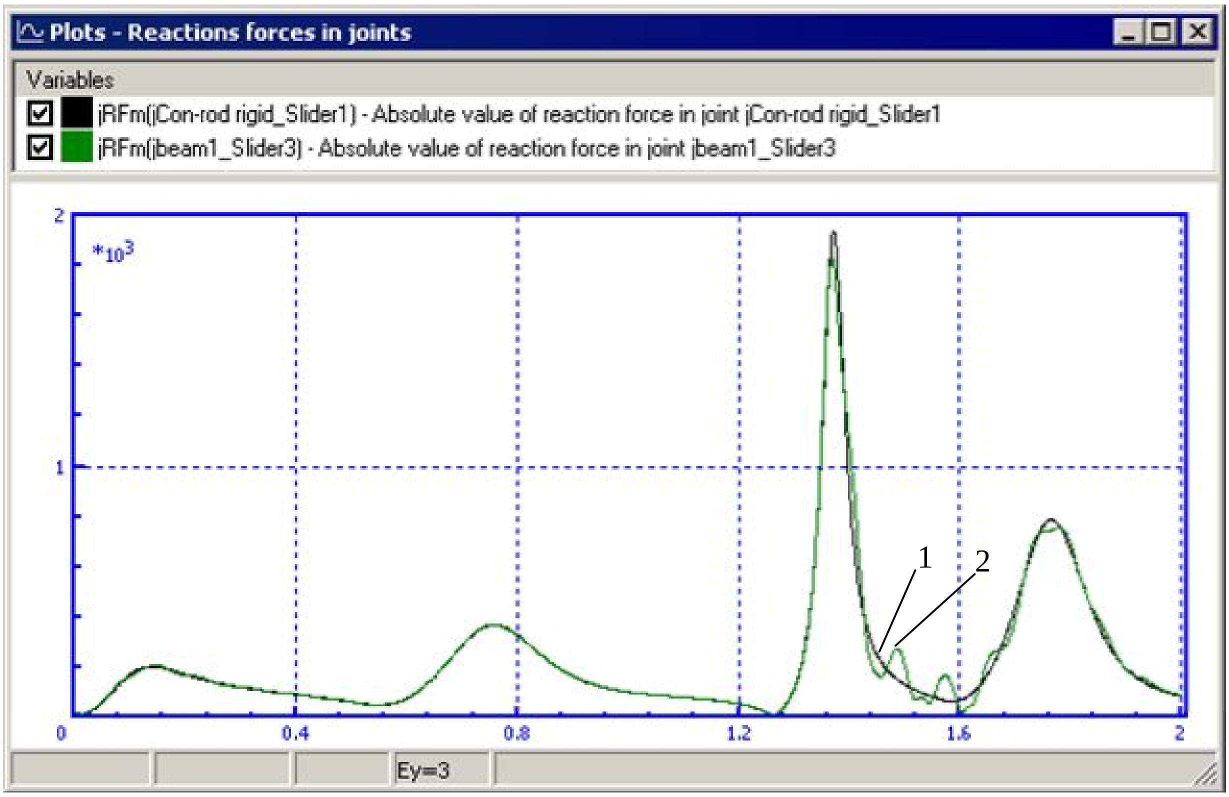


Figure 2.31. Reaction force in the Con-rod _Slider joint
1 – con-rod is a rigid body, 2 – con-rod is an elastic body.

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.

3. 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, Fig. 3.1.

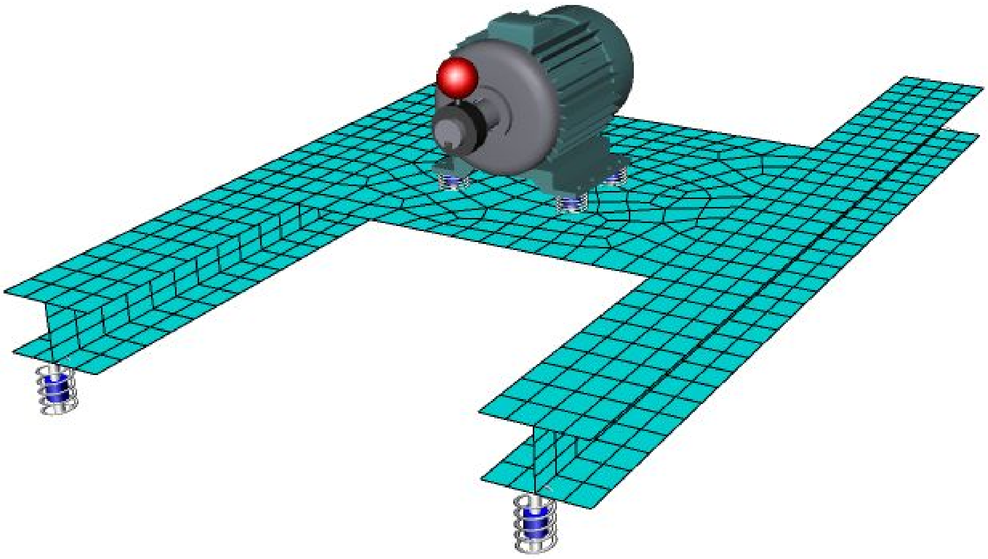


Figure 3.1.

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, Fig. 3.1. 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 FEA-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.

3.1. 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 FEA 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:

1) generate the **input.fss** file directly by **ANSYS_UM.EXE** program;
2) firstly generate the intermediate **input.fum** file by the **ANSYS_UM.EXE** and then complete data transformations with the help of **Wizard of elastic 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_root}\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 elastic subsystems** you should copy the **{um_root}\Samples\Flex\input.fum** file to the **.\platform** directory and go to the sect. 3.1.2 of this manual.
- You may omit all the steps of creating the data of elastic platform, in this case you should copy the **{um_root}\Samples\Flex\platform\input.fss** file to the **.\platform** directory and go to the sect. 3.2 of this manual.

3.1.1. Working under ANSYS environment

Before you come to the next step please repeat all the steps from the sect. 2.1.

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

1. Copy the **platformshell63.ans** file from the **{um_root}\Samples\Flex\platform** directory to the **.\platform** directory. This file contains APDL commands that automatize creating the FEA model of the platform.
2. Run **ANSYS Interactive** and select the **.\platform** directory as a **working directory**.
3. Run **ANSYS**.
4. 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 886 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 FEA 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. 2.1. As a result of the **um.mac** execution 24 *static modes* and 10 *eigenmodes* are calculated.

5. If the path to the **ANSYS_UM.EXE** in the **um.mac** is set correctly (see Sect. 2.1), **ANSYS_UM.EXE** starts automatically. Otherwise run **ANSYS_UM.EXE** manually from the **{um_root}\bin** directory.
6. Transform data according the 5-8 items of the Sect. 2.2.1.

3.1.2. Wizard of elastic subsystems

Working with the **Wizard of elastic subsystems** is described in the Sect. 2.2.2. Now you should repeat all the instructions from the Sect. 2.2.2. 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.

3.2. Creating the model and analyzing its dynamics

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

3.2.1. 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** (Fig. 3.2).

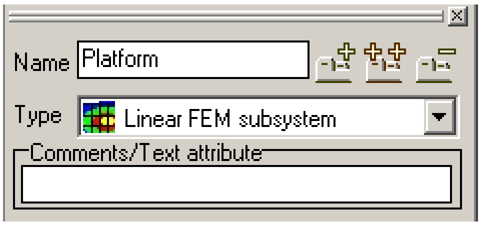


Figure 3.2.

3.2.2. 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.

3.2.3. 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 **Spring** (Fig. 3.3).

Figure 3.3.

4. Add a new graphic element (GE) by clicking the  at the lower panel (Fig. 3.4).

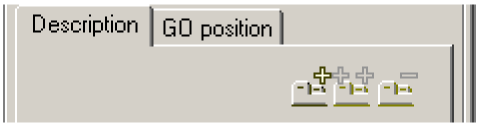


Figure 3.4.

5. Select **Parametric** type in the pull-down menu (Fig. 3.5).

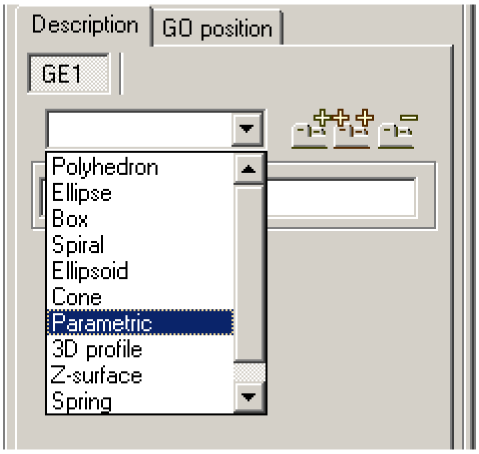


Figure 3.5.

6. Select **Spring** in the list of the standard parametric GE (Fig. 3.6)

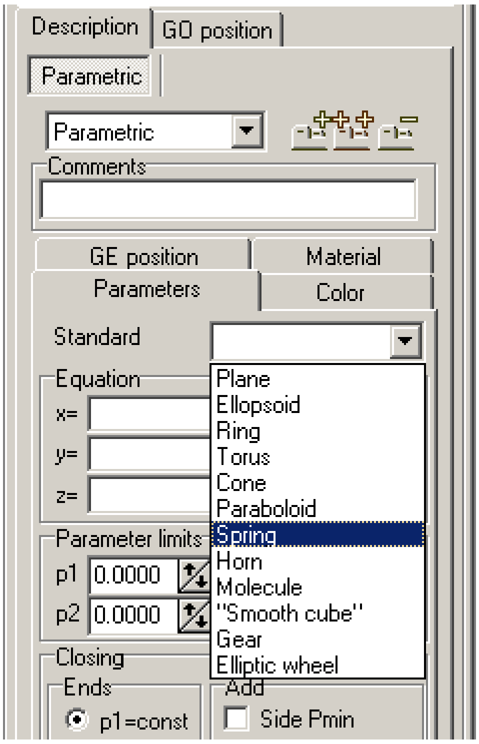
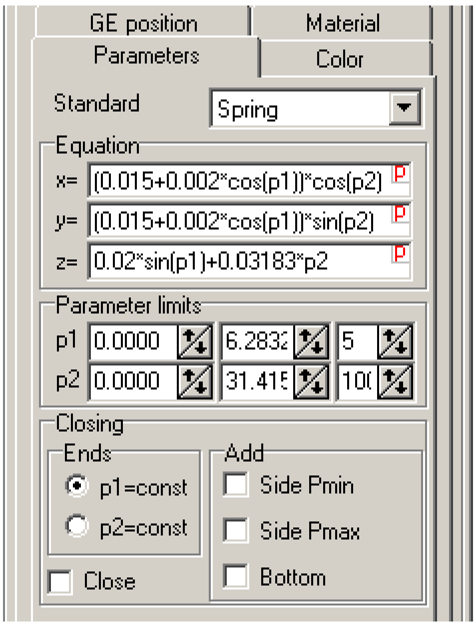


Figure 3.6.

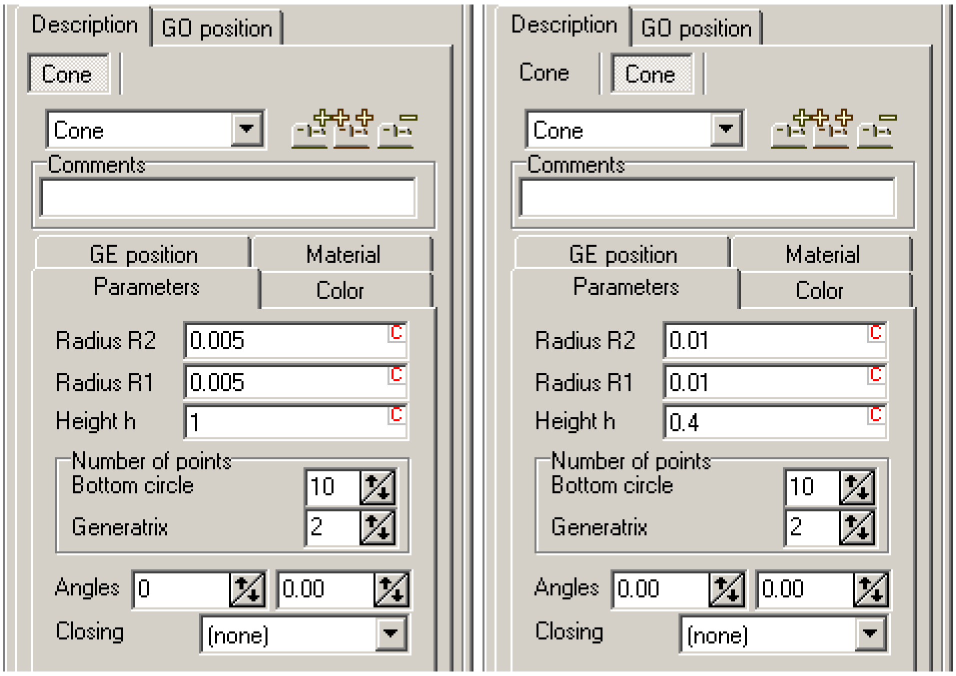
7. Set parameter values as in Fig. 3.7



Fugure 3.7.

Let us add now a GE for the damping force element.

1. Add a new GO
2. Rename it as **Damper**.
3. Add a new GE to the GO.
4. Set its type as **Cone** and parameters as in **Fig**. 3.8a.



a) b)

Figure 3.8.

5. Add the second GE **Cone** and set its parameters as in Fig. 3.8b
6. Go to the **GE position** tab and shift the element on **0.3** along Z axis (the **Translation | z** box**)**.
7. Set the diffuse component of the GE color by **Diffuse** button on the **Color** tab (Fig. 3.9)

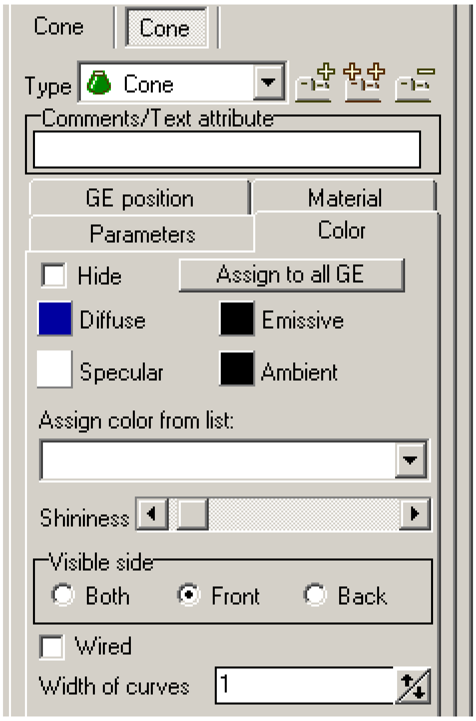


Figure 3.9

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

3.2.4. 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 **SpringFL** (forward, left), set element type **Elastic**, interacting bodies **Base0-Platform.Platform** as well as the **Spring** GO (Fig. 3.10).
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 (Fig. 3.12)

   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, Fig. 3.10:

   BeamLength/2, -WidthShelf/2-WidthBeamShelfLow/2, 0.
6. Select **Body2** tab. Set coordinates of element attachment points to the second body **Platform.Platform** (Fig. 3.11):

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

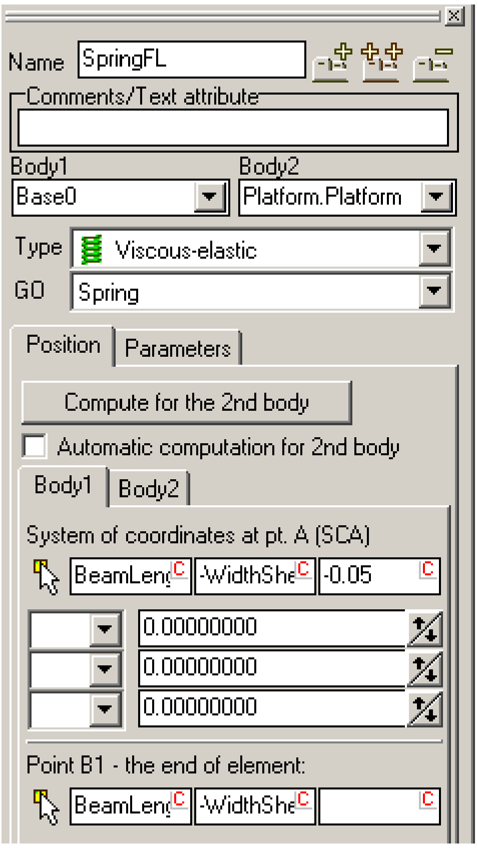


Figure 3.10

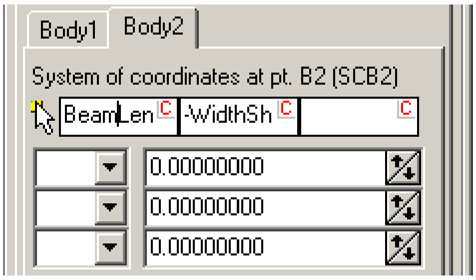


Figure 3.11

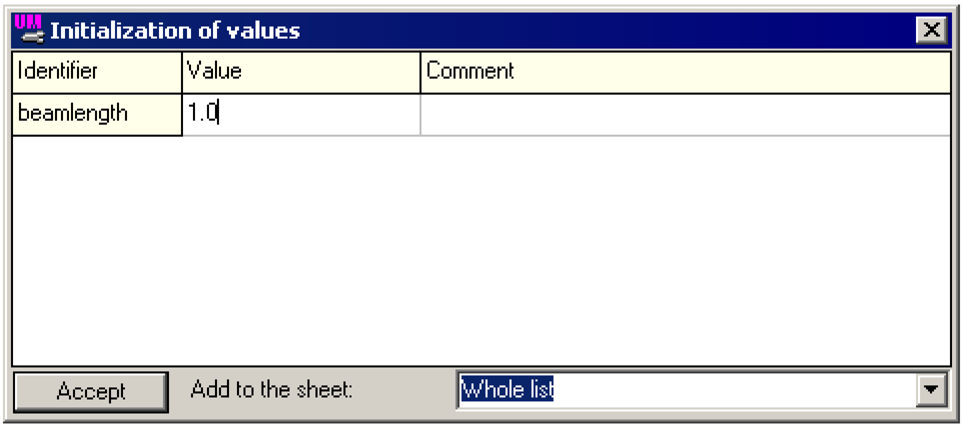


Figure 3.12

5. Let us introduce a stiffness matrix of the element. Select **Parameters** tab. Click the button in the **Stiffness matrix** box (Fig. 3.13), set diagonal elements of the matrix corresponding to the translational degrees of freedom (Fig. 3.14), and click **OK**. Set the following identifier values: **cxx=1e+6, cyy=1e+6**, **czz=1e+6** (N/m).

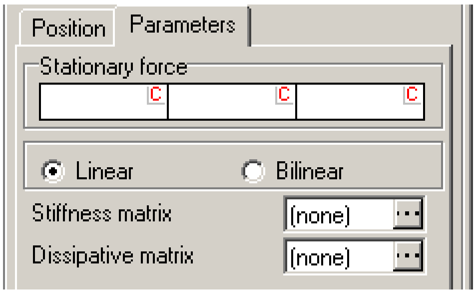


Figure 3.13

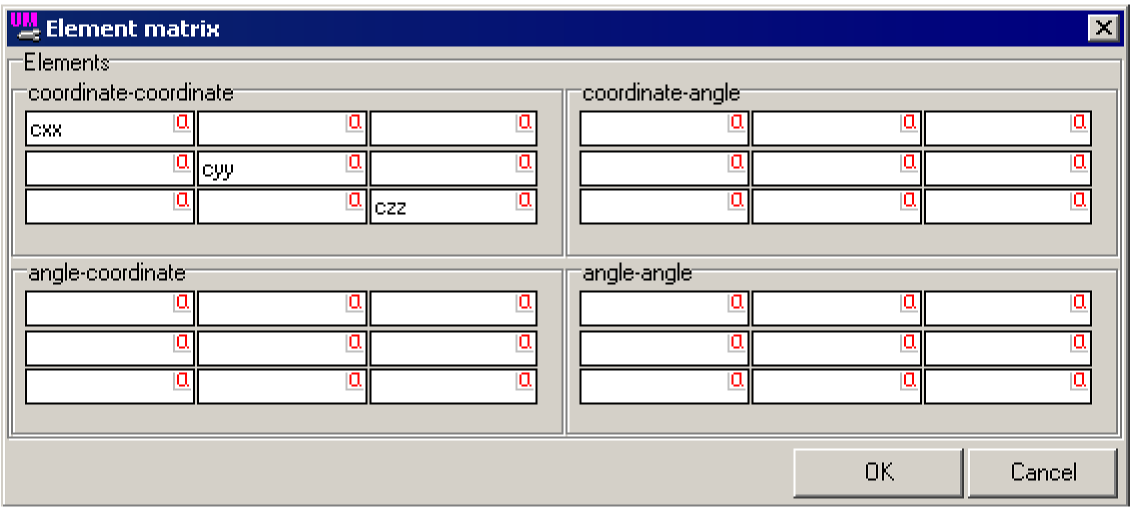


Figure 3.14

The elastic force element is described.

Now let us describe the front left damping element.

1. Copy the linear force element by the button.
2. Rename the new element as **DamperFL** (forward, left), set the element type **Dissipative** and set GO to **Damper (**Fig. 3.15).

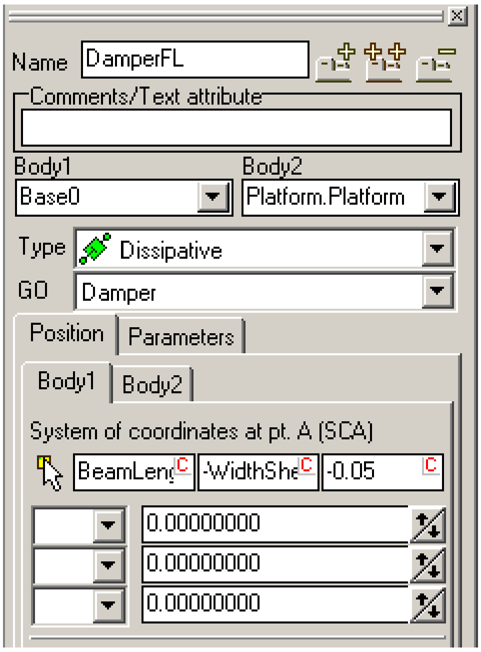


Figure 3.15

4. Let us set dissipative matrix of the element. Select **Parameters** tab. 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**. Set the following identifier values **dxx=1E3, dyy=1E3**, **dzz=1E3** (Ns/m).

Damping 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.

1. Select previously described element of the necessary type, e.g. **SpringFL** in the case of a new elastic element.
2. Click the button to create a copy.
3. Rename the copy, e.g. **SpringFR** (forward, right).
4. Correct coordinates of attachment points. For the **SpringFR** 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 (Fig. 3.10)

Thus, the full list of force elements connecting the platform with the base must include the following elements: **SpringFL, DamperFL, SpringFR, DamperFR, SpringBL, DamperBL, SpringBR, DamperBR**.

3.2.5. Model of electric motor

We shall not create the model but use the ready model of an electric motor located in the **{um_root}\Samples\Flex\electricmotor** directory.

3.2.6. 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_root}\Samples\Flex\electricmotor** model (Fig. 3.16).

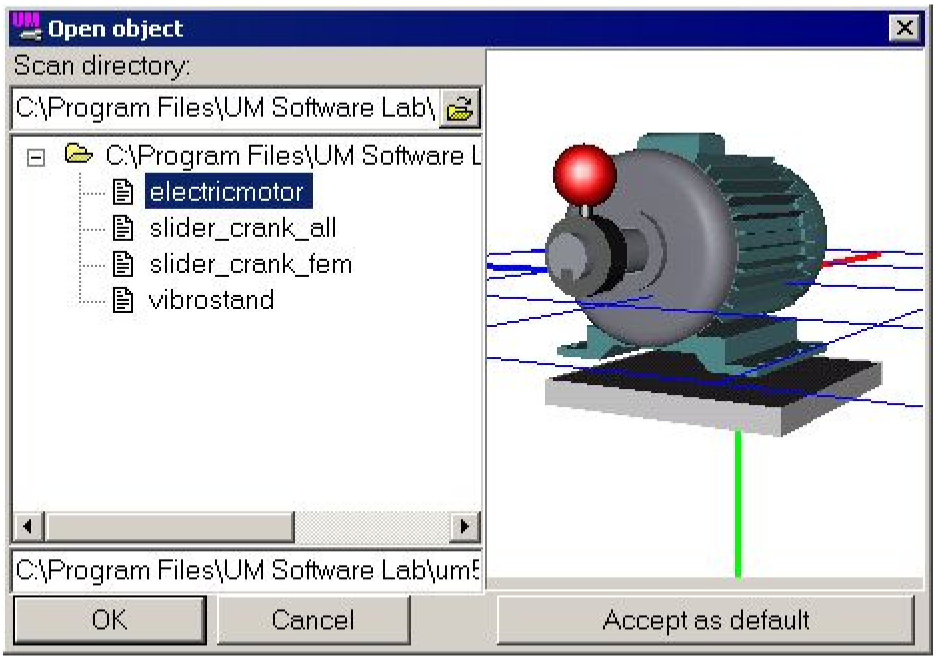


Figure 3.16

3. Rename the subsystem as **Electricmotor**.
4. Set the subsystem location as in Fig. 3.17.

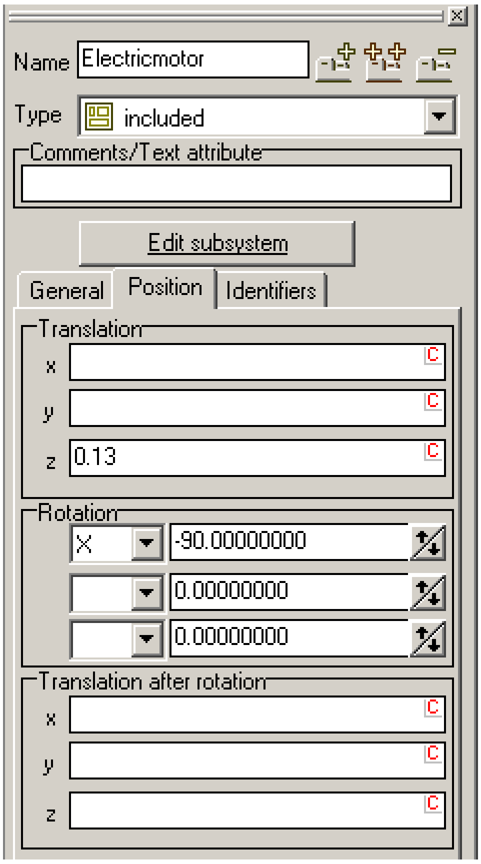

Name
Electricmotor
Type
included
Comments/Text attribute
Edit subsystem
General
Position
Identifiers
Translation
x
y
z
0.13
Rotation
X
-90.00000000
0.00000000
0.00000000
Translation after rotation
x
y
z

Figure 3.17

3.2.6.1. Setting angular velocity of the rotor

Let us set the law for angular velocity of the rotor as it shown in Fig. 3.18. 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 Fig. 3.18. The law from Fig. 3.18 is parameterized with the help of six identifiers, see table 1.

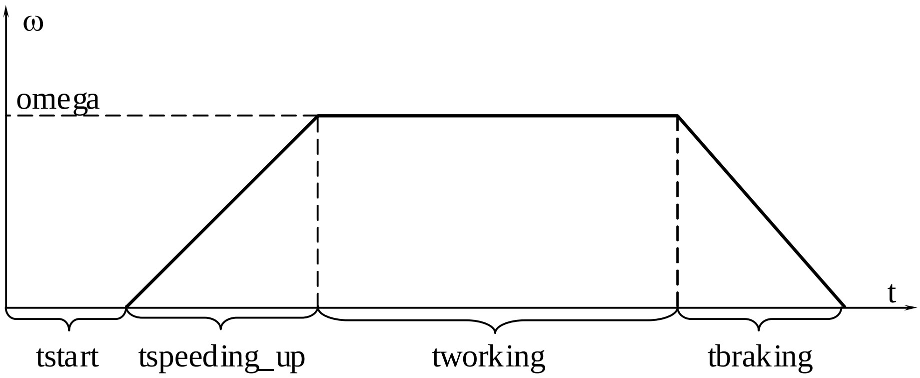


Fig. 3.18. Angular velocity of the rotor

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 Fig. 3.17. 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 (Fig. 3.19). This time function is set as **time-table** of 5 rows, see table 2 and Fig. 3.19.

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+ tbraking	(omega/tspeeding_up)*sqr(tspeeding_up)/2+ omega*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

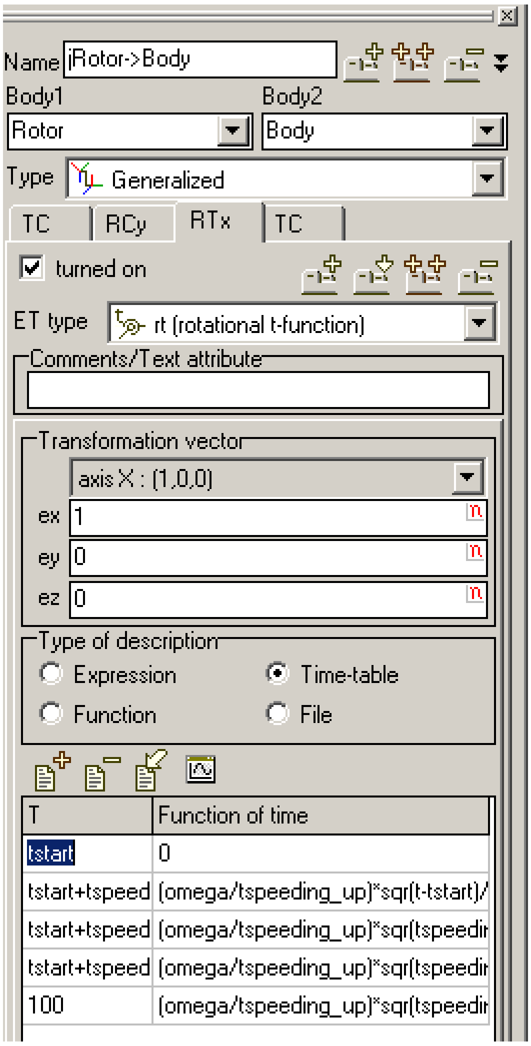


Figure 3.19

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

3.2.7. 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 Fig. 3.20.

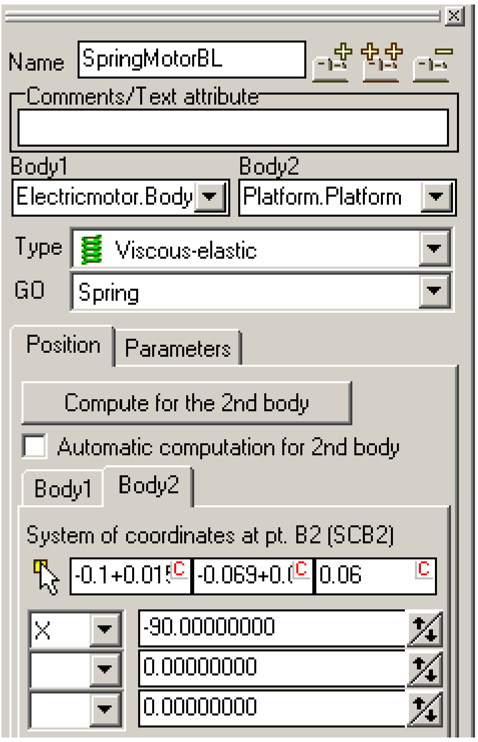


Figure 3.20

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

Table 1

Force element	Electricmotor.Body			Platform.Platform		
	X	Y	Z	X	Y	Z
SpringMotorFL, DamperMotorFL	0.0156-0.015	0.053	-0.069+0.015	0.0156-0.015	-0.069+0.015	0.06
SpringMotorFR, DamperMotorFR	0.0156-0.015	0.053	0.1-0.015	0.0156-0.015	0.1-0.015	0.06
SpringMotorBL, DamperMotorBL	-0.1+0.015	0.053	-0.069+0.015	-0.1+0.015	-0.069+0.015	0.06
SpringMotorBR, DamperMotorBR	-0.1+0.015	0.053	0.1-0.015	-0.1+0.015	0.1-0.015	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 (Fig. 3.20), 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 as it is shown in Fig. 3.21.

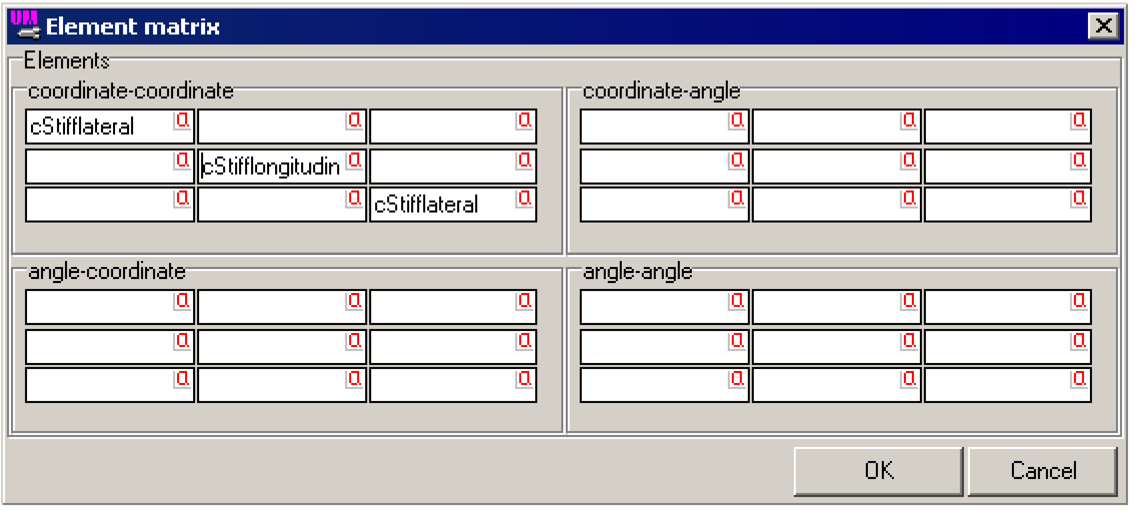


Figure 3.21

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

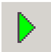
3.2.8. Preparing for simulation

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.

3.2.9. 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, **Tools/Animation window**.
3. Use the **Analysis | Simulation** menu command to open the **Object simulation inspector.**
4. Use the **FEM Subsystems | Image** tab of the **Object simulation inspector** to change the flexible platform image if necessary.

3.2.9.1. 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 **Linear analysis** or press the **F8** key. Window of linear analysis appears.
3. Select the **Equilibrium** tab. Turn on the **Keep coordinates and identifiers** check box. Start the calculation by the **Compute** button, Fig. 3.22.

Calculation process might take some time.

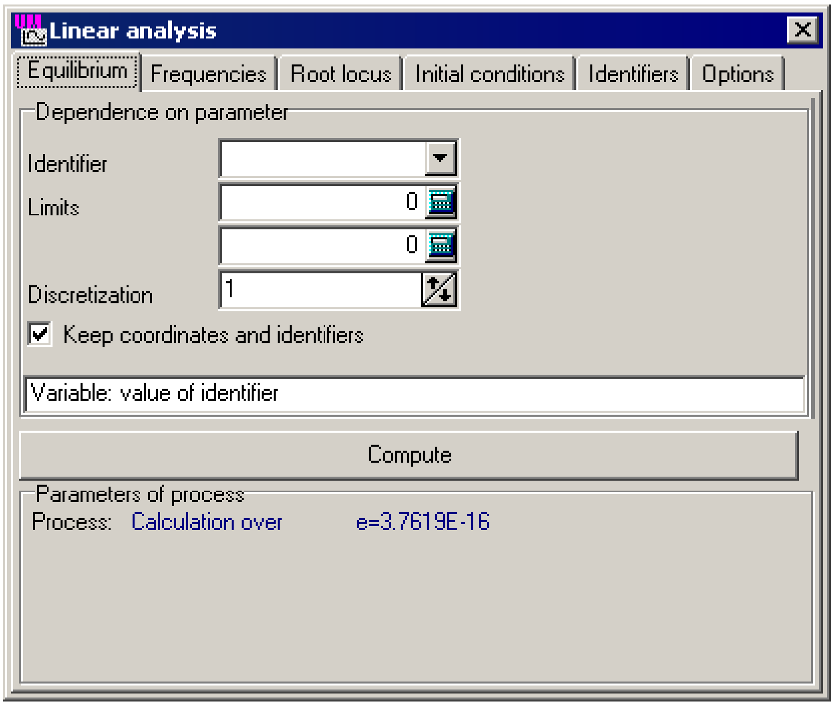


Figure 3.22

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** tab. Natural frequencies of the model are calculated automatically, Fig. 3.23.
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 **Animate** button. Now you can see that the animation window shows any selected eigenmode of the model. You can control the **Amplitude** and **Rate** of eigenmode animation. To stop animation click the **Stop** button.
7. Close the window of **Linear analysis**.

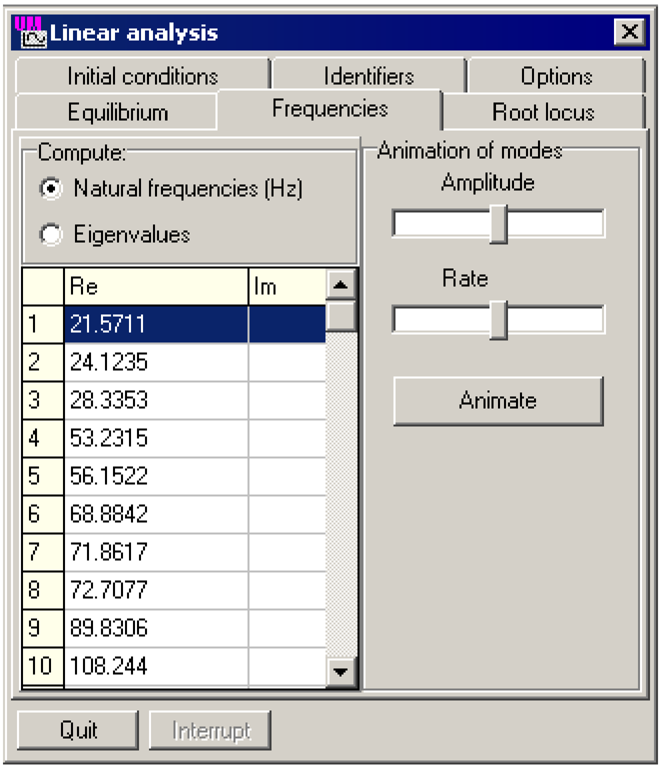


Figure 3.23

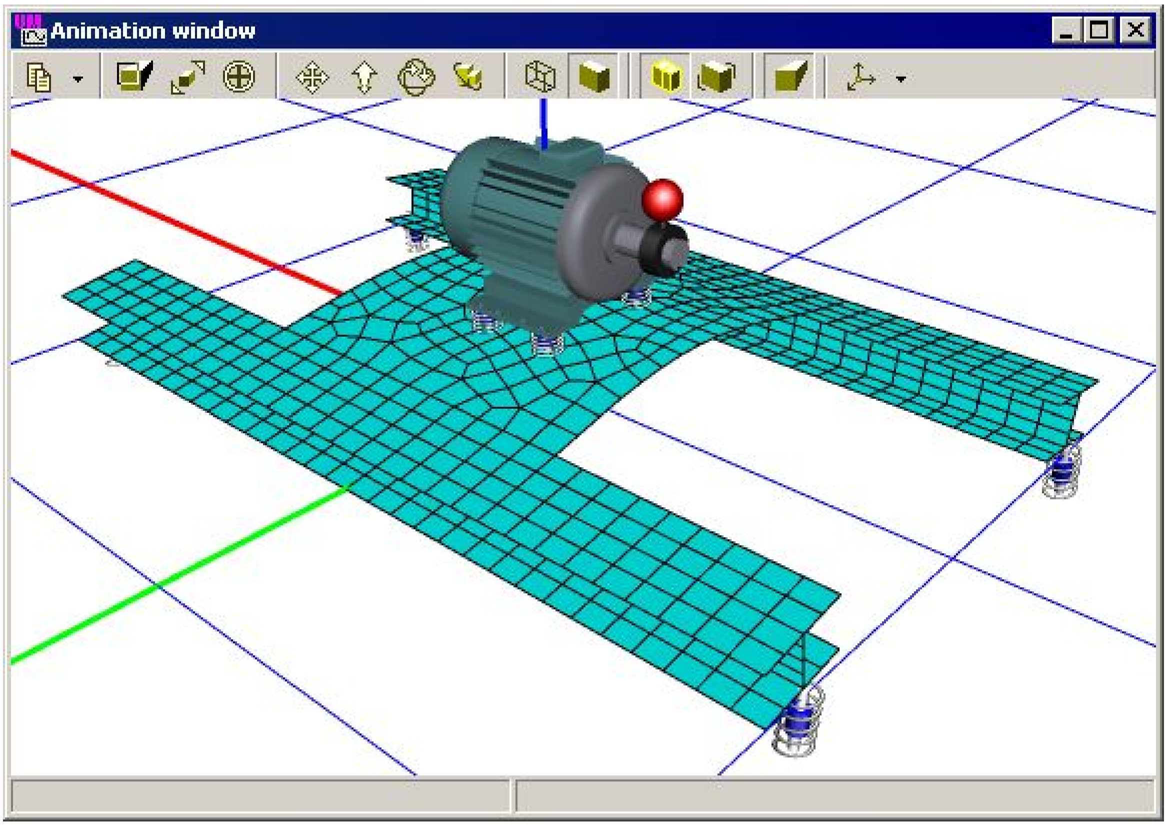


Figure 3.24. Animation of second eigenmode, 24.11 Hz

3.2.9.2. 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 **SpringMotorBR, DamperMotorBR**, Fig. 3.25.

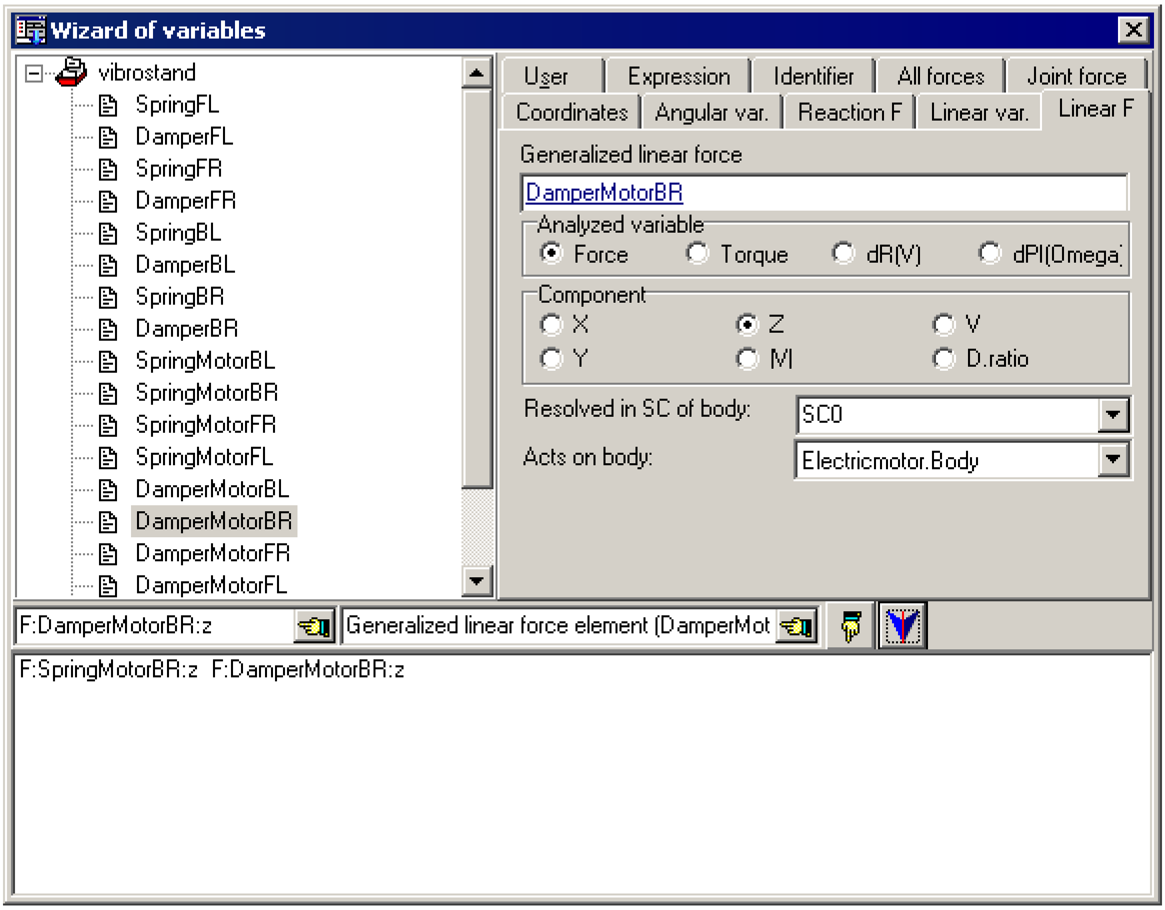


Figure 3.25

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 FEM-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 Fig. 3.26.

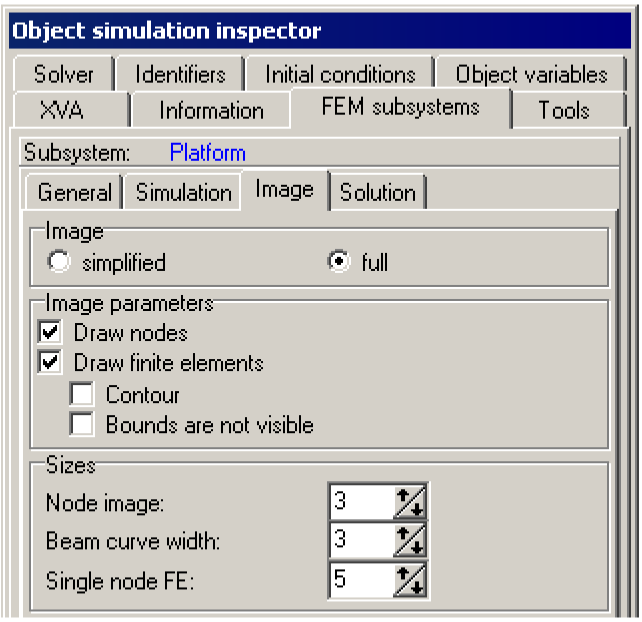


Figure 3.26.

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 956 with approximate coordinates (-0.048; 0.007; 0.06), see Fig. 3.27, 3.28.

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 Fig. 3.27.

6. Create two new graphical windows (**Tools/Graphical window**) and drag and drop just created variables to these windows separately.

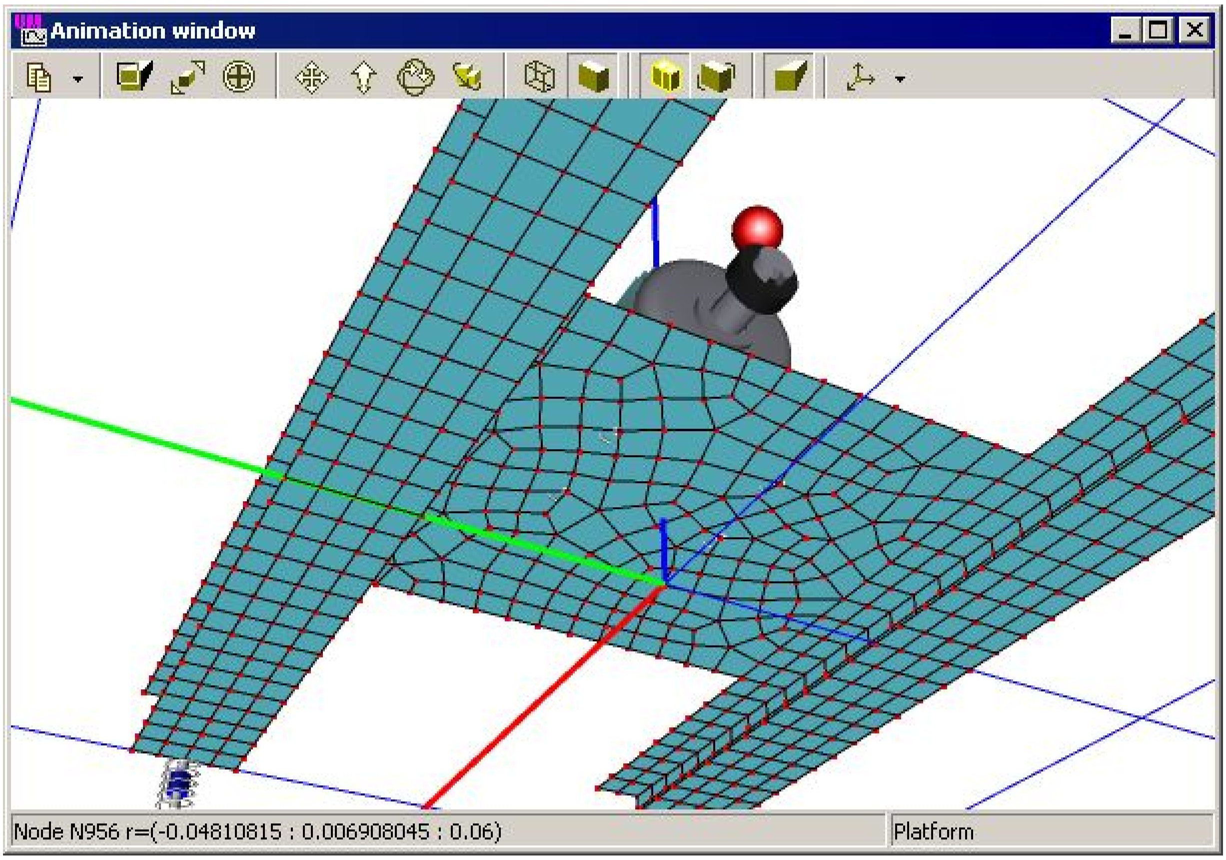


Figure 3.27.

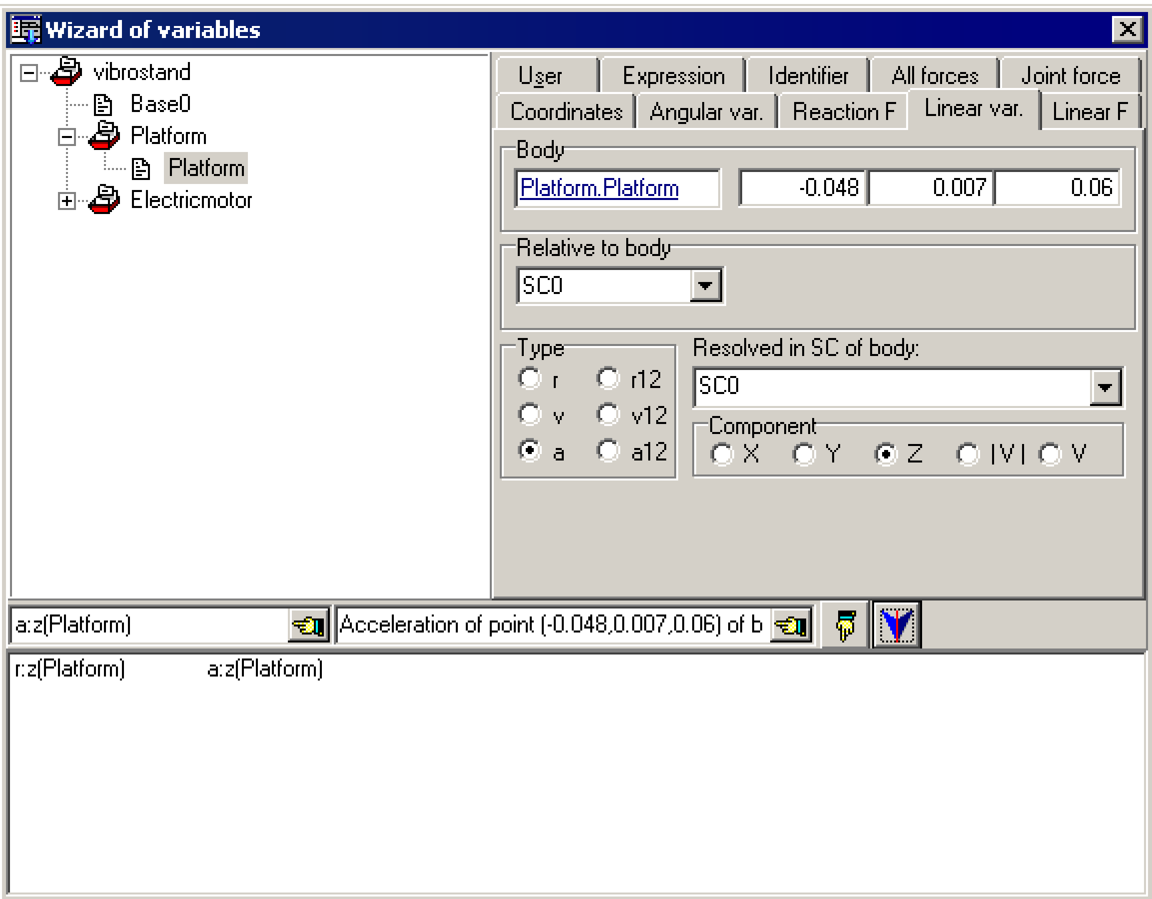


Figure 3.28.

7. Set the solver parameter on the **Solver** tab of the inspector as in Fig. 3.29:
   - **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.**

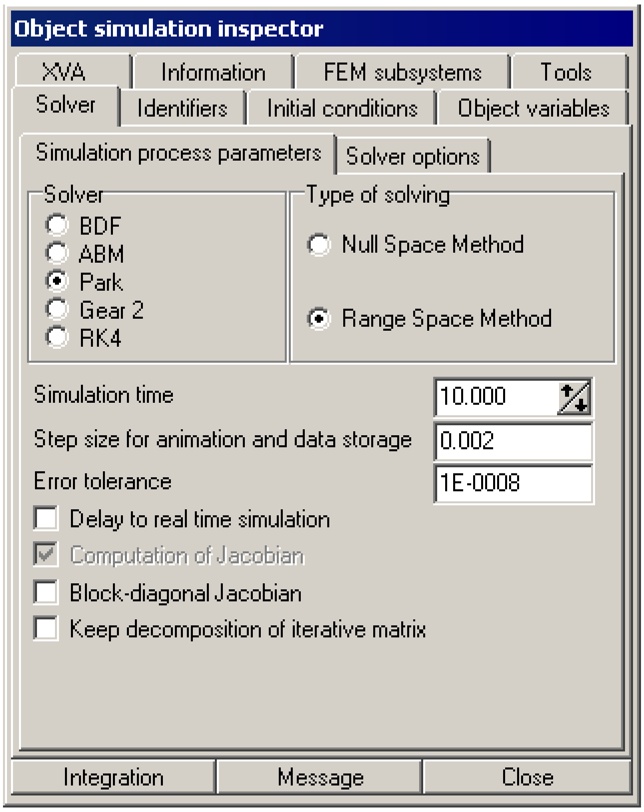


Figure 3.29

8. On the **FEM subsystems | Simulation** tab switches **gravity**, **internal dissipation** as well as **linear model** should be **ON**. Set **a=0.001**, **b=0** (Fig. 3.30).

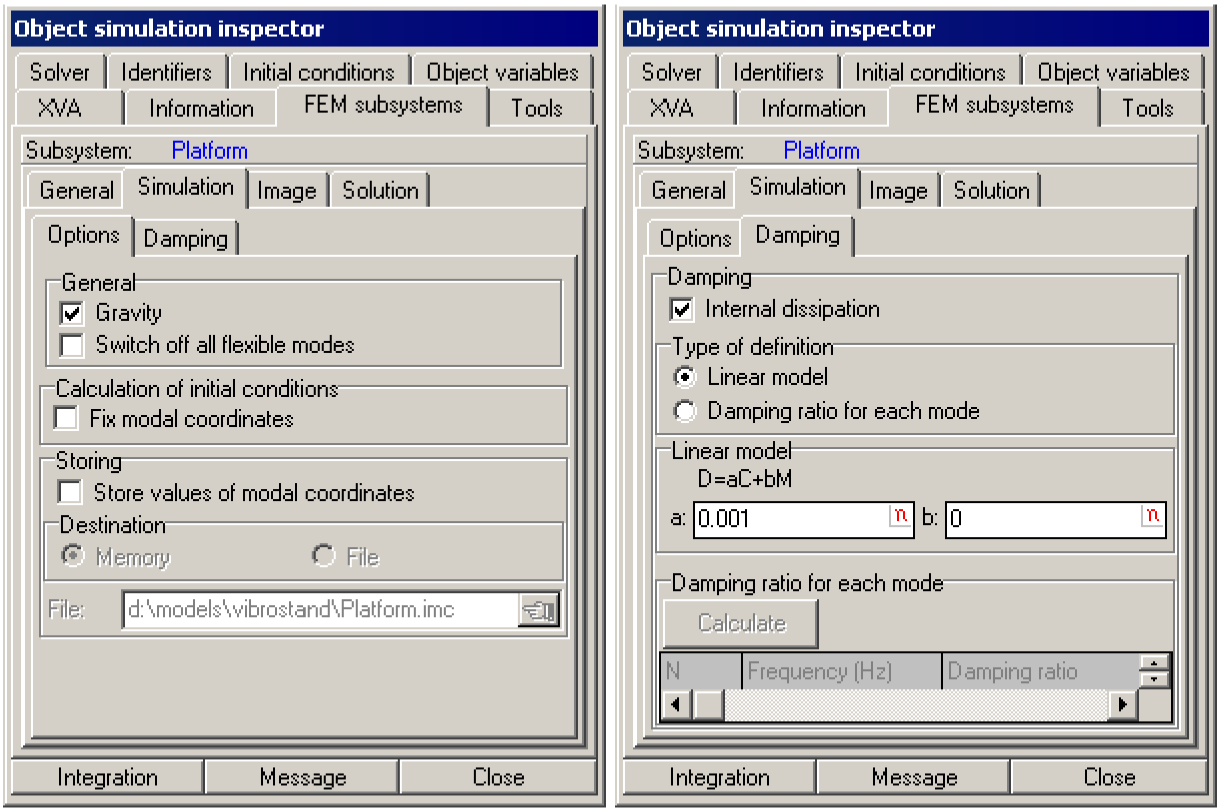


Figure 3.30.

9. Select the **Identifiers** tab in the **Object simulation inspector**. Select the **Vibrostand.Electricmotor** from the pull-down list of subsystems. Set the following values (Fig. 3.31):
   - **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.

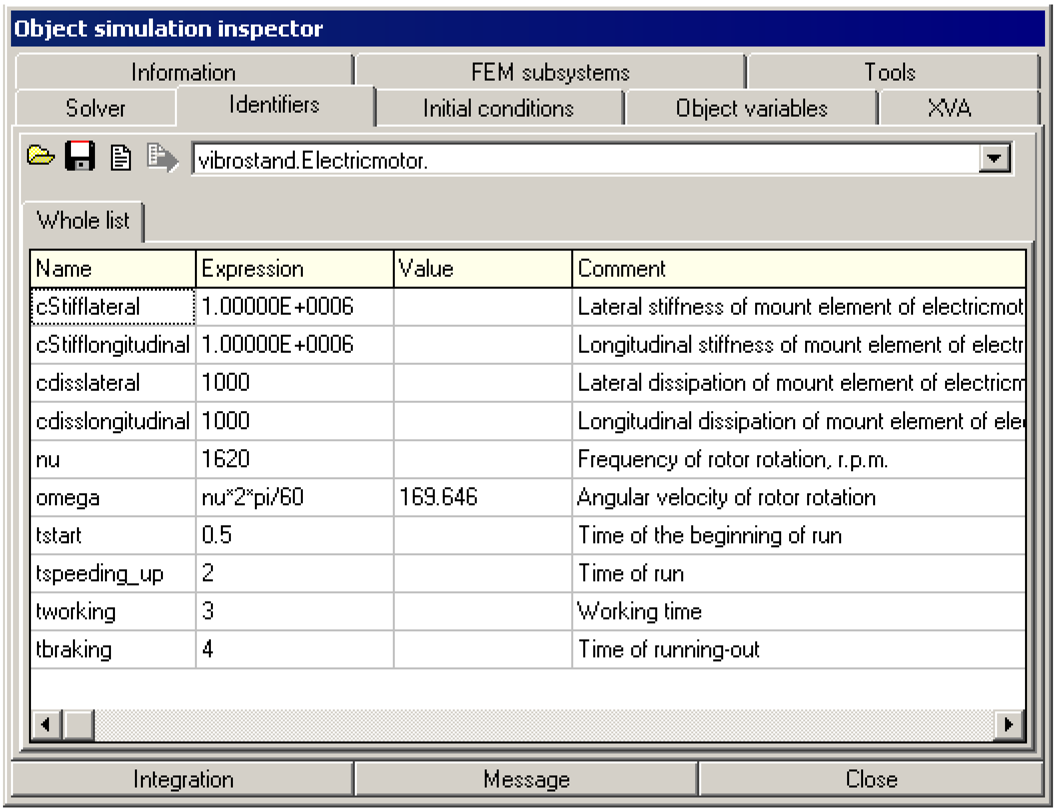
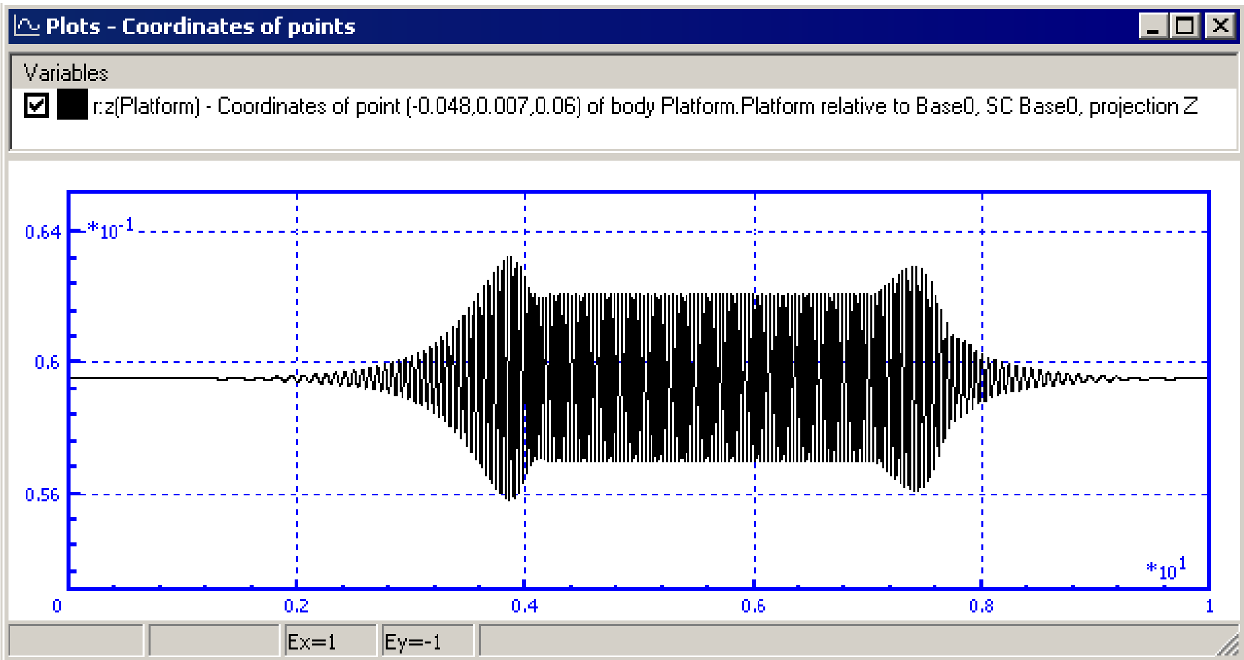
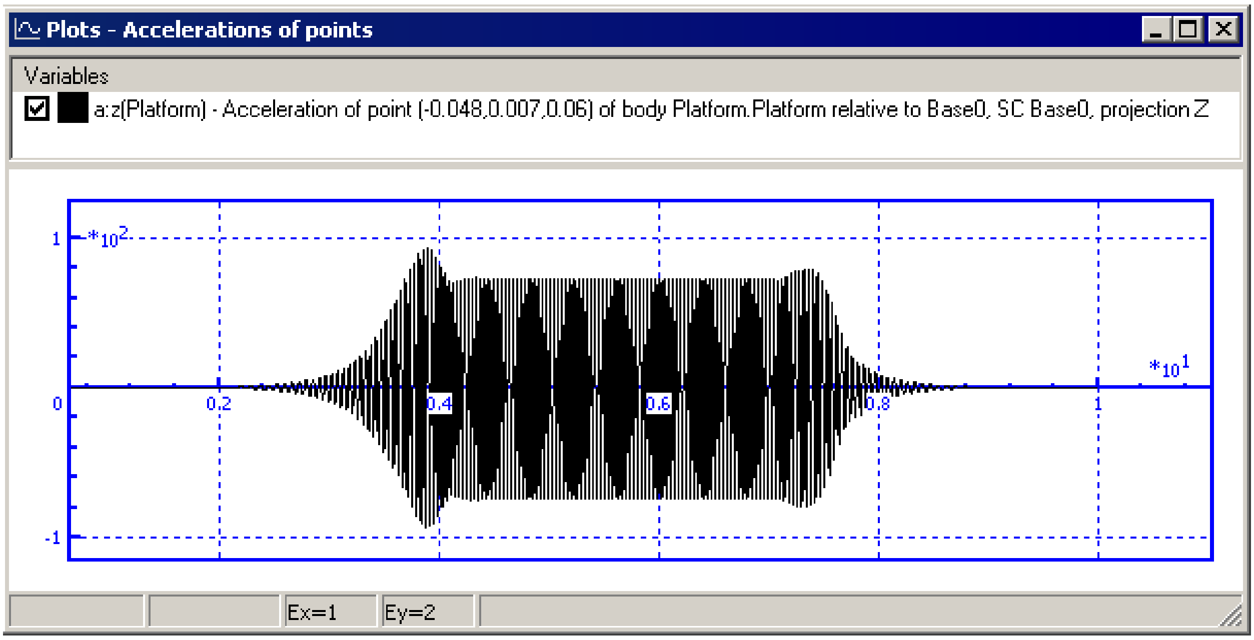


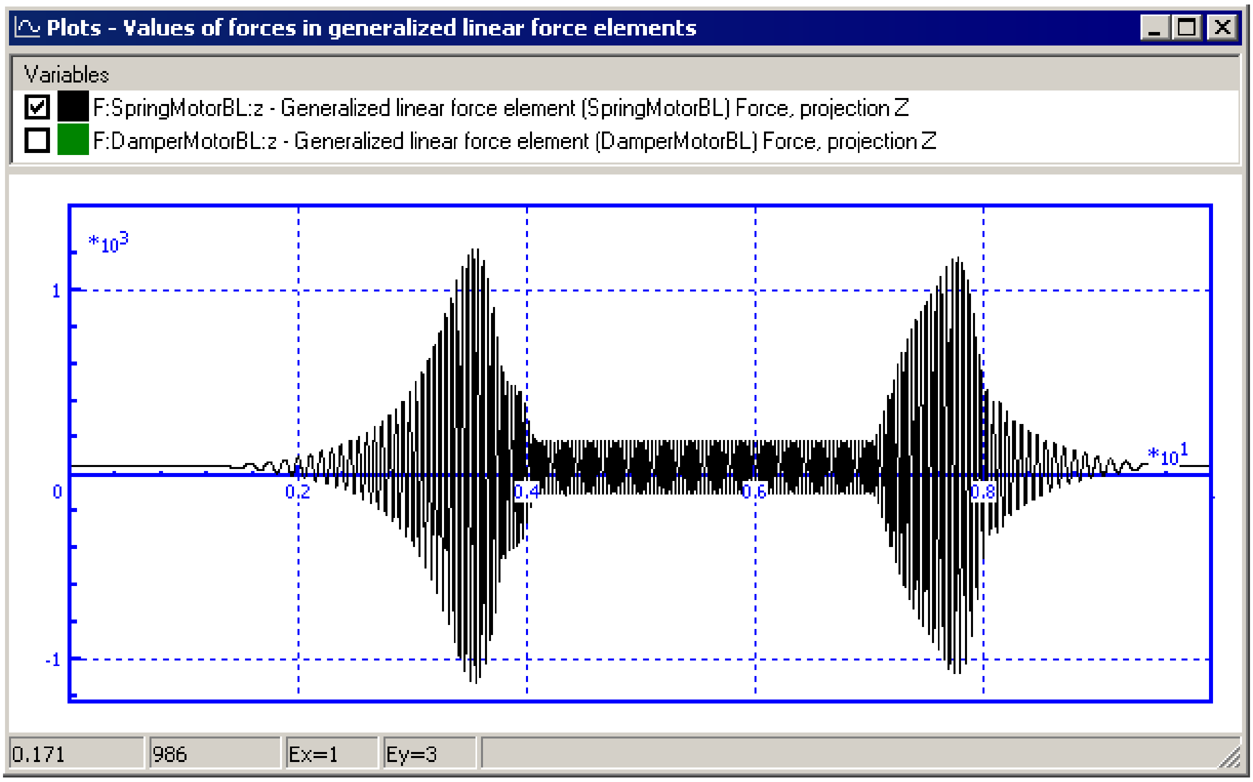
Figure 3.31.

10. Start the simulation process by the **Integration** button on the bottom part of the inspector.

Fig. 3.32 depicts some simulation results.




Plots - Accelerations of points
Variables
a:z(Platform) - Acceleration of point (-0.048,0.007,0.06) of body Platform.Platform relative to Base0, SC Base0, projection Z
*10^2
*10^1
1
0
-1
0.2
0.4
0.6
0.8
1
Ex=1
Ey=2


Plots - Values of forces in generalized linear force elements
Variables
F:SpringMotorBL:z - Generalized linear force element (SpringMotorBL) Force, projection Z
F:DamperMotorBL:z - Generalized linear force element (DamperMotorBL) Force, projection Z
*10^3
*10^1
1
0
-1
0.2
0.4
0.6
0.8
0.171
986
Ex=1
Ey=3

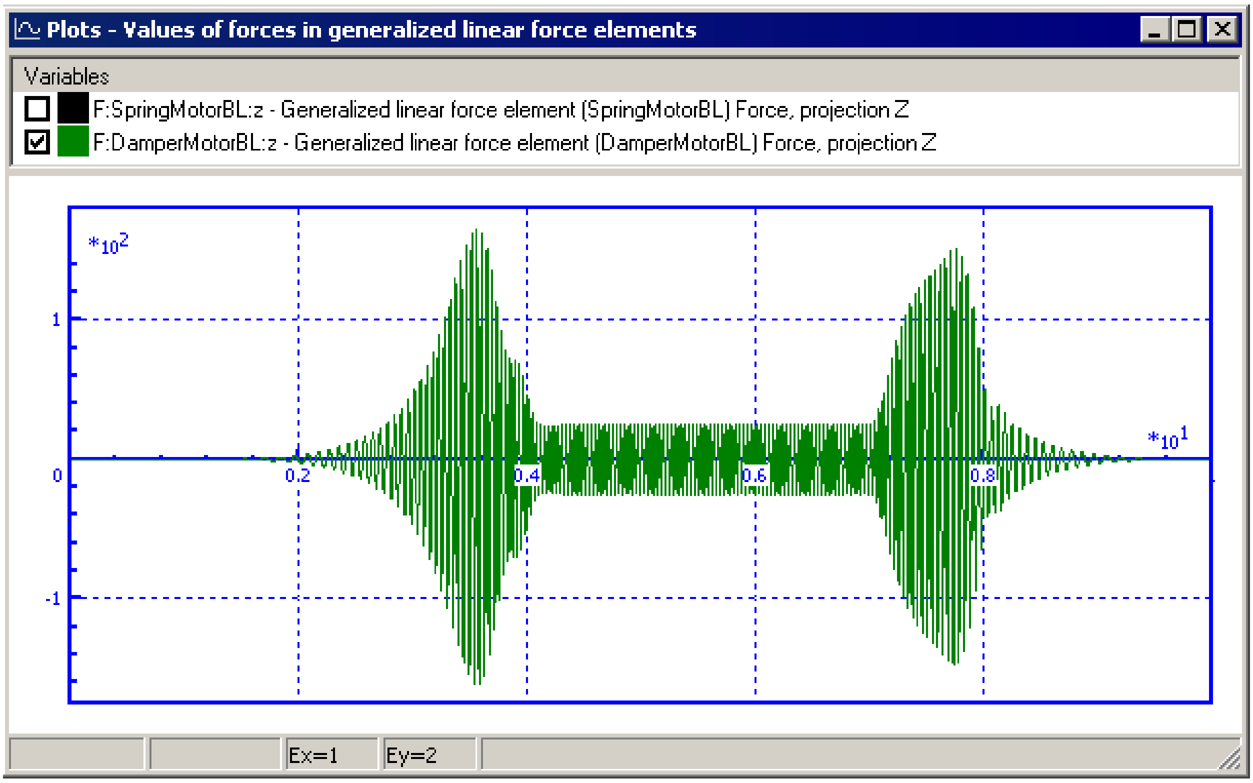


Figure 3.32

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

1. The option **switch off all flexible modes** should be on (Fig. 3.30).
2. Run simulation.
3. Copy variables in graphical windows as static using popup menus (contact menu in a graphical window, **Copy as static variables** menu item).
4. Change the option **switch of all flexible modes** to off (Fig. 3.30).
5. Repeat the simulation.
6. Compare simulation results.