## 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*<sup>2</sup>. 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

<sup>&</sup>lt;sup>1</sup> <u>http://www.umlab.ru/download/50/eng/gs\_um.pdf</u>

<sup>&</sup>lt;sup>2</sup> 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**

This manual is prepared for informational use only, may be revised from time to time. No responsibility or liability for any errors that may appear in this document is supposed.

Copyright © 2008 Universal Mechanism Software Lab. All rights reserved.

All trademarks are the property of their respective owners.

1. GI	ETTING STARTED USING UM: SIMULATING HYBRID MODELS	1
2. SI	IDER-CRANK MECHANISM	4
2.1.	Preparing ANSYS environment	6
2.2.	Preparing con-rod as an elastic beam	7
2.2.1	Working under ANSYS environment	7
2.2.2	Wizard of elastic subsystems	10
2.3.	Creating the model	17
2.3.1	Creating graphical objects	18
2.3.2	Creating rigid bodies	20
2.3.3	Creating elastic subsystem	21
2.3.4	Creating joints	22
2.3.5	Preparing for simulation	24
2.4.	Simulation	25
3. EL	ECTRIC MOTOR ON ELASTIC PLATFORM	31
<ol> <li>3. El</li> <li>3.1.</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM	31
<ol> <li>EL</li> <li>3.1.1</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform Working under ANSYS environment	<b>31</b> <b>33</b> 34
<ol> <li>El</li> <li>3.1. 3.1.1 3.1.2</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform Working under ANSYS environment Wizard of elastic subsystems	<b>31</b> <b>33</b> 
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.2</li> </ul> </li> <li>3.2.</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform Working under ANSYS environment Wizard of elastic subsystems Creating the model and analyzing its dynamics	
<ol> <li>El</li> <li>3.1.1 3.1.2</li> <li>3.2. 3.2.1</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform Working under ANSYS environment Wizard of elastic subsystems Creating the model and analyzing its dynamics Introducing elastic platform	
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li></ul></li></ol>	ECTRIC MOTOR ON ELASTIC PLATFORM. Preparing elastic platform Working under ANSYS environment Wizard of elastic subsystems Creating the model and analyzing its dynamics Introducing elastic platform Attaching the elastic platform to a base	<b>31 33</b> 3435 <b>36</b> 363636
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.1</li> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> </ul> </li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform Working under ANSYS environment Wizard of elastic subsystems Creating the model and analyzing its dynamics Introducing elastic platform Attaching the elastic platform to a base Creating graphical elements	
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.1</li> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> </ul> </li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM Preparing elastic platform	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b>
<ol> <li>El</li> <li>3.1.1</li> <li>3.1.2</li> <li>3.2.</li> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM. Preparing elastic platform	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>44</b>
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.1</li> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> </ul> </li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM. Preparing elastic platform	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>44</b> <b>44</b>
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.1</li> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> <li>3.2.6</li> <li>3.2.6</li> <li>3.2.6</li> </ul> </li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM.         Preparing elastic platform         Working under ANSYS environment.         Wizard of elastic subsystems         Creating the model and analyzing its dynamics.         Introducing elastic platform         Attaching the elastic platform to a base.         Creating graphical elements         Force elements.         Model of electric motor         Adding motor to object as a subsystem         2.6.1.	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>40</b> <b>44</b> <b>44</b> <b>44</b> <b>44</b> <b>44</b>
<ol> <li>El</li> <li>3.1.1</li> <li>3.1.2</li> <li>3.2.</li> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> <li>3.2.7</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM.         Preparing elastic platform         Working under ANSYS environment.         Wizard of elastic subsystems         Creating the model and analyzing its dynamics.         Introducing elastic platform         Attaching the elastic platform to a base         Creating graphical elements         Force elements.         Model of electric motor         Adding motor to object as a subsystem         .6.1.         Setting angular velocity of the rotor         Electric motor and platform coupling by force elements.	<b>31 33 34 35 36 36 36 36 37 40 44 44 44 44 44</b>
<ol> <li>El</li> <li>3.1.1</li> <li>3.1.2</li> <li>3.2.</li> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> <li>3.2.7</li> <li>3.2.8</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM.         Preparing elastic platform         Working under ANSYS environment.         Wizard of elastic subsystems         Creating the model and analyzing its dynamics.         Introducing elastic platform         Attaching the elastic platform to a base.         Creating graphical elements         Force elements.         Model of electric motor         Adding motor to object as a subsystem         2.6.1.         Setting angular velocity of the rotor         Electric motor and platform coupling by force elements.         Preparing for simulation	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>40</b> <b>44</b> <b>44</b> <b>44</b> <b>44</b> <b>45</b> <b>46</b> <b>49</b> <b>50</b>
<ol> <li>El</li> <li>3.1.1         <ul> <li>3.1.1</li> <li>3.1.2</li> </ul> </li> <li>3.2.1         <ul> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> <li>3.2.6</li> <li>3.2.7</li> <li>3.2.8</li> <li>3.2.9</li> </ul> </li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM.         Preparing elastic platform         Working under ANSYS environment.         Wizard of elastic subsystems         Creating the model and analyzing its dynamics.         Introducing elastic platform         Attaching the elastic platform to a base.         Creating graphical elements         Force elements.         Model of electric motor         Adding motor to object as a subsystem         c.6.1. Setting angular velocity of the rotor         Electric motor and platform coupling by force elements.         Preparing for simulation	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>40</b> <b>44</b> <b>44</b> <b>44</b> <b>44</b> <b>50</b> <b>51</b>
<ol> <li>El</li> <li>3.1.1</li> <li>3.1.2</li> <li>3.2.</li> <li>3.2.1</li> <li>3.2.2</li> <li>3.2.3</li> <li>3.2.4</li> <li>3.2.5</li> <li>3.2.6</li> <li>3.2.7</li> <li>3.2.8</li> <li>3.2.9</li> <li>3.2</li> </ol>	ECTRIC MOTOR ON ELASTIC PLATFORM.         Preparing elastic platform         Working under ANSYS environment.         Wizard of elastic subsystems         Creating the model and analyzing its dynamics.         Introducing elastic platform         Attaching the elastic platform to a base.         Creating graphical elements         Force elements.         Model of electric motor         Adding motor to object as a subsystem         2.6.1. Setting angular velocity of the rotor         Electric motor and platform coupling by force elements.         Preparing for simulation         Simulation	<b>31</b> <b>33</b> <b>34</b> <b>35</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>36</b> <b>37</b> <b>40</b> <b>44</b> <b>44</b> <b>44</b> <b>44</b> <b>50</b> <b>51</b> <b>52</b>

# 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.

4

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<sup>1</sup>. 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.
- 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.

7

<sup>&</sup>lt;sup>1</sup> More detailed information about **interface nodes** you can find in the eleventh part of UM User's Manual

Creating data set for simulation of flexible body
General Options
ANSYS results file (*.rst):
D:\models\flexbeam\flexbeam.rst
Target directory:
D:\simulation\flexbeam
Process:
Create Close

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.

Creating data set for simulation of flexible body
General Options
normalize modes
frequency: 0.500
Process:
Create Close

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.

- 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.

9

#### 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.

Select data file	
Data file:	
D:\models\flexbeam\input.fum	

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

General Position Image Solution	
Data file:	
D:\models\flexbeam\input.fum	General Position Image Solution
Subsystem information Data prepared: ANSYS9.0 Name of solution: flexbeam Header of solution (comments): 16.11.2005, 23:50:48, Flexible beam with mass21 element for definition torsion inel moment Nodes: Finite elements: Degrees of freedom: Normal modes: Static modes: Normalization:	Intia     Shift       x     n       y     n       z     0.3       Rotation     0.00000000       101     0.00000000       300     Shift after rotation       606     ×       10     y       12     No

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.

General Position Image Solution
Image O simplified I full
Image parameters Draw nodes Draw finite elements Contour Bounds are not visible
Sizes       Node image:       3       Beam curve width:       4       Single node FE:
Color Visual elements O Nodes O Single node elements O Beam elements O Shell and plate elements O Solid elements O Polygons
Diffuse Emissive
Additional

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.

#### 14

deneral rosition mage container
Data set
<ul> <li>Uriginal</li> <li>Transformed</li> </ul>
Modes Rigid body Interface nodes
10 normal modes, 12 static modes
Selected normal modes: 10
Selected static modes: 12
✓ I. normal, 26.668
✓ 2. normal, 20.000
✓ 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     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10     10
✓ IU. normal, 355.866
✓ 11. static
<ul> <li>✓ 12. static</li> <li>✓ 13. static</li> </ul>
✓ 14. static
🗹 15. static 📃 🗾
Animation of modes
Animation of modes Amplitude Rate
Animation of modes Amplitude Rate
Animation of modes Amplitude Rate Frame per 1/4 period: 5
Animation of modes Amplitude Rate Frame per 1/4 period: 5
Animation of modes Amplitude Rate Frame per 1/4 period: 5
✓     15. static       Animation of modes       Amplitude       Rate       Frame per 1/4 period:       5 ★       Animate
Interpretation of modes Amplitude Rate Frame per 1/4 period: 5 4 Animate Transformations Modes Shift SC Turning of SC
✓       15. static         Animation of modes         Amplitude       Rate         Frame per 1/4 period:       5 €         Animate         Transformations         Modes       Shift SC         Turning of SC
✓       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
✓       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
✓       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
✓       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
✓       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
Image: Static       Image: Static         Animation of modes         Amplitude       Rate         Frame per 1/4 period:       5 million         Animate       5 million         Transformations       Modes         Modes       Shift SC         Image: State       Image: State         Image: State       Image: St
Image: Static stati

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.

UM - Object data input 🛛 🗙
6 rigid body modes are deleted from the transformed data set
OK
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.

Data set		
• Transformed		
	0.1.1	

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.

Save flexible subsyst	em data	×
Path to subsystem date	a	
D:\models\flexbeam		Ē
	Save	Cancel
Einne	2.12	

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*.
- 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

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.



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

Name	Base0 🛷 - :북박북 - 🖵
Com	ents/Text attribute

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.



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

X
Sensors/LSC Variables Curves
Object Options
Path D:\Models\slider_crank_fem
Object type
<ul> <li>General</li> </ul>
C Rail vehicle
Equation generation
C Symbolic
<ul> <li>Numeric-Iterative</li> </ul>
Direction of gravity
ex 🔽
ey U
ez -1.0
Characteristic size 1.00
Scene image Base0 💌
,

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.

Name Crank <u>- 1</u> 🔏		
Oriented points Vectors 3D	Contact	
Parameters Position	Points	
Go to element	<b>1</b>	
Image: 🔽 Visible		
Crank	-	
Compute automatic		
Inertia parameters		
Mass 7.403	C	
Inertia tensor		
0.00471300	C	
1.33271 🚨	C	
1.33	53432 C	
Added mass matrix (non	e)	
Coordinates of center of mass		
0.5 C C	С	

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 12 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.

		=×
Name	Con-rod FEM 한 한호	
Туре	🌃 Linear FEM subsystem	•
Comr	ments/Text attribute	

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.
- Rename the joint to Base0\_Crank. Select Rotational type for the joint and set Y axis as Joint vectors, see Fig. 2.19.

	i <u>×</u>
Name (Base0_	_Crank 호 바람 - 드 ♥
Body1	Body2
BaseO	🔽 Crank 💽
Type \land Ro	tational 🗾
Geometry (	Description Joint force
Joint points	
BaseU	1.3
	L L L
Crank	 }
	C C C
Joint vector	8
Base0	axis Y : (0,1,0) 💌
0	n 1 n 0 n
Crank	axis Y : (0,1,0) 💌
0	n n n n n

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.



#### 2.3.5. Preparing for simulation

1. Save the model as **Slider\_crank\_fem** (**File/Save as** menu command), see Fig. 2.22.

Save as	×
Path (including object name)	
d:\models\slider_crank_fem	<u>A</u>
Save	Cancel

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.).

Deriving and compiling of equations			
Parameters Protocol			
Formalizm for equation generation C Autodetection C Direct C Composite body method	Canguage for output files C++		
Recommended method:	Direct		
Compile equations			
Run simulation module			
Generate Genera	ate all Close		

Figure 2.23.

Now the model is ready for simulation.

## 2.4. Simulation

 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.

Object simulation inspector					
Solver Identifiers Initial conditions Object variables XVA Information FEM subsystems Tools					
Coordin	Coordinates Constraints for initials				
കം					
slider_c	rank	fem	n.Con-rod FEM.		<b></b>
	ŵ	1	Coordinate	Velocity	Comment
2.1			2	0	Joint (t) 1
2.2			0	0	Joint (t) 2
2.3			0	0	Joint (t) 3
2.4			0	0	Joint (a) 1
2.5			0	0	Joint (a) 2
2.6			0	0	Joint (a) 3
2.7	ψ		0	0	Mode 1
2.8	ψ		0	0	Mode 2
2.9	ψ		0	0	Mode 3
2.10	ψ		0	0	Mode 4
2.11	ψ		0	0	Mode 5
2.12	ψ		0	0	Mode 6
2.13	2.13 🖞 0 0 Mode 7 🔽				
Message dx= 0.1 🖬 da= 0.1 🖬					
	Integration Message Close			Close	

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.

📴 Wizard of variables	×
<ul> <li>Base0_Crank</li> <li>Base0_Crank</li> <li>Crank_Con-rod</li> <li>Con-rod_Slider</li> <li>Base_Slider</li> <li>Con-rod</li> </ul>	User       Expression       All forces       Joint force         Coordinates       Angular var.       Reaction F       Linear var.         Joint
jRFm(Con-rod_Slider)	🕣 Reactive force for joint Con-rod_Slider, magr 🕣 🐺 🚺
jRFm(Crank_Con-rod)	jRFm(Con-rod_Slider)

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.

Object simulation inspector	Object simulation inspector
Solver         Identifiers         Initial conditions         Object variables           XVA         Information         FEM subsystems         Tools	Solver         Identifiers         Initial conditions         Object variables           XVA         Information         FEM subsystems         Tools
XVA     Information     PEM subsystems     Fools       Subsystem:     Con-rod FEM       General     Simulation     Image       Options     Damping       General     Gravity       Switch off all flexible modes       Calculation of initial conditions       Image       Storing       Store values of modal coordinates       Destination       Image       Store values of modal coordinates	XVA       Information       PEM subsystems       Fools         Subsystem:       Con-rod FEM         General       Simulation       Image       Solution         Options       Damping         Internal dissipation       Type of definition         Type of definition       Internal dissipation         Type of definition       Internal dissipation         Damping ratio for each mode       Damping ratio for each mode         D=aC+bM       a:       0.001       m         b:       0       m
File:       d:\models\slider_crank_fem\Con-rod FEM.ir         Integration       Message         Close	Calculate           N         Frequency (Hz)         Damping ratio           Integration         Message         Close

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  $\omega = 0$  up to its nominal angular velocity.
- operating duty;
- stop way decreasing angular velocity of a rotor till  $\omega=0$ .

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.

- 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 D 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).

	<u> </u>
Name Platform <u>그</u> 한 한 한	-1
Type 🏭 Linear FEM subsystem	•
Comments/Text attribute	

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  $\stackrel{1}{\rightharpoonup}$  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 is at the lower panel (Fig. 3.4).

Description	Description GO position		

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)

Description GO position			
Parametric			
Parametric			
Comments			
GE position	Matorial		
CE position	Material		
Parameters	Color []		
Standard	▣		
Equation	Plane		
x=	Ellopsoid		
	Ring		
y=	Torus		
Z=	Cone		
	Paraboloid		
Parameter limits	Horp		
p1 0.0000 🏌	Molecule		
p2 0.0000 🏌	"Smooth cube"		
Gear			
Ende Add			
G al acret	C Side Proin		
<ul> <li>preconst</li> </ul>	i side Finin		

Figure 3.6.

7. Set parameter values as in Fig. 3.7

GE position	Material
Parameters	Color
Standard	Spring 💌
Equation	
x= (0.015+0.002	2*cos(p1))*cos(p2) 🖻
y= (0.015+0.002	2*cos(p1))*sin(p2) 📍
z= 0.02*sin(p1)+	0.03183*p2 📍
Parameter limits	
p1 0.0000 🏒	6.2832 🏂 5 🧏
p2 0.0000 🏒	31.415 🔨 10( 🏂
Closing	
Ends	Add
• p1=const	🔲 Side Pmin
C p2=const	🗖 Side Pmax
🗖 Close	E Bottom

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.

38

#### Universal Mechanism 5.0

39

#### Getting Started: UM FEM

Description GO position	Description GO position
Cone	Cone Cone
Cone -1	Cone
Comments	Comments
GE position Material	GE position Material
Parameters Color	Parameters Color
Radius R2 0.005	Radius R2 0.01
Radius R1 0.005 C	Radius R1 0.01 C
Height h 🛛 🚺	Heighth 0.4 🧧
Number of points	Number of points
Generatrix 2	Generatrix 2
Angles 0 1/2 0.00	Angles 0.00 🌠 0.00 🌠
Closing (none)	Closing (none)
a)	b)
Figure	e 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)

Cone Cor	ie i
Type 🥭 Cone	Image: 1 = 1 Im
Comments/Te	xt attribute
GE positio	
Paramete	ers Color
🔲 Hide	Assign to all GE
Diffuse	Emissive
Specular	Ambient
Assign color fr	om list:
	•
Shininess 🔳	
Visible side O Both	© Front O Back
🔲 Wired	
Width of curve	es 1 1
Fi	gure 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  $\stackrel{1}{\rightharpoonup}$  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;

X
Name SpringFL <u>그</u> 한 그
Comments/Text attribute
Body1 Body2
Base0 🗾 Platform.Platform 🗾
Type 🔰 Viscous-elastic 💌
GO Spring 💌
Position Parameters
Compute for the 2nd body
Automatic computation for 2nd body
Body1 Body2
System of coordinates at pt. A (SCA)
BeamLeng -WidthShe -0.05
.00000000
▼ 0.00000000 14
0.0000000
Point B1 - the end of element:
¶չ BeamLeng <sup>C</sup> -WidthShe <sup>C</sup> Ը

Figure 3.10

Body1 Body2		
System of coordinates at pt. B2 (SCB2)		
KaamLen <sup>©</sup> -WidthSh <sup>©</sup> <sup>©</sup>		
▼ 0.00000000		
0.0000000 🔀		
<b>_</b>	0.00000000	

Figure 3.11

🚆 Initialization of values 💦 💦 🔀		
Identifier	Value	Comment
beamlength	1.0	
Accept	Add to the sheet:	

Figure 3.12

5. Let us introduce a stiffness matrix of the element. Select **Parameters** tab. Click the **D** 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).

Position Parame	eters	
Stationary force		
C	C	
<ul> <li>Linear</li> </ul>	O Bilinear	
<ul> <li>Linear</li> <li>Stiffness matrix</li> </ul>	O Bilinear	

Figure 3.13

👑 Element matrix	×
Elements	
-coordinate-coordinate	coordinate-angle
angle-coordinate	angle-angle
	OK Cancel

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  $\stackrel{\text{the}}{=}$  button.
- 2. Rename the new element as **DamperFL** (forward, left), set the element type **Dissipative** and set GO to **Damper** (Fig. 3.15).

Comments/Text attribute				
Body1 Body2				
Base0 🛛 🚽 Platform.Platform 💌				
Type 💉 Dissipative				
GO Damper 💌				
Position Parameters				
Body1 Body2				
System of coordinates at pt. A (SCA)				
ReamLen <sup>®</sup> -WidthShe <sup>®</sup> -0.05 <sup>©</sup>				
0.0000000				
0.0000000 🕺				
0.0000000				

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  $\square$  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**}**SamplesFlexelectricmotor** model (Fig. 3.16).

🖳 Open object	×
Scan directory:	
C:\Program Files\UM Software Lab\ 🗃	
<ul> <li>□ C:\Program Files\UM Software L</li> <li>□ I electricmotor</li> <li>□ Slider_crank_all</li> <li>□ Slider_crank_fem</li> <li>□ vibrostand</li> </ul>	
<u>۱</u>	
C:\Program Files\UM Software Lab\um!	
OK Cancel	Accept as default

Figure 3.16

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

<u>×</u>		
Name Electricmotor <u>- 한 한호</u>		
Type 📳 included 💌		
Comments/Text attribute		
Edit subsystem		
General Position Identifiers		
Translation		
×		
y C		
z 0.13		
X -90.0000000 1		
▼ 0.0000000 14		
0.0000000		
Translation after rotation		
×		
уС		
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.

N⁰	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+
	tbraking	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

×						
Name iRotor->Body <u>그</u> 로 한말 <u>그</u> 로 주						
Body1 Body2						
Rotor Body 🗾						
Type 📜 Generalized 💌						
TC RCy RTx TC						
ET type 5- rt (rotational t-function)						
Comments/Text attribute						
Transformation vector						
axis X : (1,0,0) ▼						
ex 1						
ey O n						
ez 0 n						
C Expression						
C Function C File						
et e ef 🖾						
T Function of time						
tstart 0						
tstart+tspeed (omega/tspeeding_up)*sqr(t-tstart)/						
tstart+tspeed (omega/tspeeding_up)*sqr(tspeedir						
tstart+tspeed (omega/tspeeding_up)*sqr(tspeedir						
100 (omega/tspeeding_up)*sqr(tspeedir						

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.

X
Name SpringMotorBL - 앞 박맞 - 드
Comments/Text attribute
Body1 Body2
Electricmotor.Body Platform.Platform
Type 🔰 Viscous-elastic 💌
GO Spring
Position Parameters
Compute for the 2nd body
Automatic computation for 2nd body
Body1 Body2
System of coordinates at pt. B2 (SCB2)
¶∑ -0.1+0.01 <sup>€</sup> -0.069+0.( <sup>©</sup> 0.06 <sup>©</sup>
× 🔽 -90.00000000 🕺
• 0.0000000 1
0.0000000 🕺

Figure 3.20

Table 1 contains coordinates of attachment points of elastic and damping force elements realizing the coupling.

Table 1

Force element	Electric motor. Body			Platform.Platform		
	Х	Y	Z	Х	Y	Z
SpringMotorFL,	0.0156-	0.053	-0.069+	0.0156-	-0.069+	0.06
DamperMotorFL	0.015		0.015	0.015	0.015	
SpringMotorFR,	0.0156-	0.053	0.1-0.015	0.0156-	0.1-0.015	0.06
DamperMotorFR	0.015			0.015		
SpringMotorBL,	-0.1+	0.053	-0.069+	-0.1+	-0.069+	0.06
DamperMotorBL	0.015		0.015	0.015	0.015	
SpringMotorBR,	-0.1+	0.053	0.1-0.015	-0.1+	0.1-0.015	0.06
DamperMotorBR	0.015			0.015		

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.

🚆 Element matrix		×
Elements		
coordinate-coordinate	coordinate-angle	
cStifflateral 🔍 🔍		<u>0</u>
CStifflongitudin 🔍 🔍		<u>0</u>
CStifflateral		<u>a</u>
angle-coordinate	angle-angle	
	٩	۵
		0.
		0
	ОК	Cancel

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  $\mathbf{nu} = 1620$  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 **b** 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.

💑 Linear analysis	×						
Equilibrium Freque	encies Root locus Initial conditions Identifiers Options						
Dependence on pa	arameter						
Identifier	<b>_</b>						
Limits	0 🔜						
	0 🔟						
Discretization	1 1						
✓ Keep coordinate	es and identifiers						
	1						
Variable: value of identifier							
	Compute						
Parameters of proc	ess 2.7010E.10						
Process: Laiculati	on over e=3.7613E-16						

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 **I** 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.

💑 Linear analysis 🛛 🛛 🗙							
Initial conditions I Identifiers Dptions							
	Equilibrium	Freque	encies	Root locus			
_Cc	ompute:		Anima	tion of modes			
•	Natural frequenci	es (Hz)		Amplitude			
C	Eigenvalues						
	Re	lm ]	-	Rate			
1	21.5711						
2	24.1235						
3	28.3353			Animate			
4	53.2315						
5	56.1522						
6	68.8842						
7	71.8617						
8	72.7077						
9	89.8306						
10	108.244		-				
	Quit Interrupt						

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.

🛱 Wizard of variables	×
→       PringFL         →       ⇒         >       > DamperFL         →       ⇒         >       > DamperFR         →       ⇒         >       > DamperFR         →       ⇒         >       > DamperFR         →       ⇒         >       > DamperBL         →       ⇒         >       > DamperBR         →       > SpringMotorBR         →       > SpringMotorFR         →       > DamperMotorBL         →       > DamperMotorFR         →       > DamperMotorFR         →       > DamperMotorFR	Uger       Expression       Identifier       All forces       Joint force         Coordinates       Angular var.       Reaction F       Linear var.       Linear F         Generalized linear force       DamperMotorBR         Analyzed variable <ul> <li>Force</li> <li>Torque</li> <li>dR(V)</li> <li>dPl(Omega)</li> </ul> Component <ul> <li>X</li> <li>Z</li> <li>V</li> <li>MI</li> <li>D.ratio</li> </ul> Resolved in SC of body: <ul> <li>SC0</li> <li>Acts on body:</li> <li>Electricmotor.Body</li> <li>Image:</li> </ul>
F:DamperMotorBR:z	ar force element (DamperMot 🕣 🐬 💓
F:SpringMotorBR:z F:DamperMotorBR:z	

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.

Object simulation inspector
Solver   Identifiers   Initial conditions   Object variables
XVA Information FEM subsystems Tools
Subsystem: Platform
General Simulation Image Solution
Image C simplified
Image parameters ✓ Draw nodes ✓ Draw finite elements Contour ■ Bounds are not visible
Sizes Node image: 3 1/4 Beam curve width: 3 1/4 Single node FE: 5 1/4

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.

Universal Mechanism 5.0



57

Figure 3.27.

📭 Wizard of variables	×
<ul> <li>→ Description</li> <li>→ Description</li> <li>→ Description</li> <li>→ Description</li> <li>→ Description</li> </ul>	User Expression Identifier All forces Joint force Coordinates Angular var. Reaction F Linear var. Linear F Body Platform.Platform -0.048 0.007 0.06 Relative to body SCO Type Resolved in SC of body: C r C r12 C v C v12 C a C a12 C X C Y C Z C  V  C V
a:z(Platform)	point (-0.048,0.007,0.06) of b 📶 🐬 💓
r:z(Platform) a:z(Platform)	

- 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.

Object simulation inspector							
XVA Solver	Inform Identifier	ation s   Ini	FEM subs	ystems Dbjed	Tools ct variables		
Simulation process parameters       Solver options         Solver       Type of solving         C BDF       C Null Space Method         C Park       C Range Space Method         C RK4       Range Space Method							
O RK4         Simulation time         Step size for animation and data storage         Error tolerance         Delay to real time simulation         Computation of Jacobian         Block-diagonal Jacobian         Keep decomposition of iterative matrix							
Integra	ation	M	lessage	(	Close		

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).

## Universal Mechanism 5.0 Getting Started: UM FEM Object simulation inspector Object simulation inspector Solver Identifiers Initial conditions Object variables Solver | Identifiers | Initial conditions | Object variables

XVA Information FEM subsystems Tools	XVA Information FEM subsystems Tools				
Subsystem: Platform	Subsystem: Platform				
General Simulation Image Solution	General Simulation Image Solution				
Options Damping	Options Damping				
General ✓ Gravity ─ Switch off all flexible modes Calculation of initial conditions	Damping ✓ Internal dissipation Type of definition • Linear model				
🔲 Fix modal coordinates	Damping ratio for each mode				
Storing Store values of modal coordinates Destination	Linear model D=aC+bM a: 0.001 n b: 0 n				
Memory     File  File: d:\models\vibrostand\Platform.imc	Damping ratio for each mode Calculate				
Integration Message Close	Integration Message Close				

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

59

Object simulation inspector								
Informa	ation	FEM subsystems		[	Tools			
Solver	Identifiers	Initial conditions	s   O	bject variables	XVA			
🗁 🔒 🖹 🖿	🗢 🕞 🖺 🐚 vibrostand Electricmotor.							
Whole list								
Name	Expression	Value	Comment					
cStifflateral	1.00000E+0006		Lateral stiffn	ess of mount elen	nent of electricmot			
cStifflongitudinal	1.00000E+0006		Longitudinal	stiffness of moun	t element of electr			
cdisslateral	1000		Lateral dissip	pation of mount el	ement of electricm			
cdisslongitudinal	1000		Longitudinal dissipation of mount element of ele					
nu	1620		Frequency of rotor rotation, r.p.m.					
omega	nu*2*pi/60	169.646	Angular velocity of rotor rotation					
tstart	0.5		Time of the l	beginning of run				
tspeeding_up	2		Time of run					
tworking	3		Working time					
tbraking	4		Time of running-out					
Integra	Integration Message Close							

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.



#### Getting Started: UM FEM





61



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.