

11.	Simulation of dynamics of flexible bodies using UM FEM	11-2
11.1.	Basic ideas and methods.....	11-2
11.1.1.	Introduction.....	11-2
11.1.2.	Kinematics	11-2
11.2.	Installation, preparing data, workflow.....	11-5
11.3.	Preparing data under ANSYS environment.....	11-8
11.4.	ANSYS-UM data exchange	11-9
11.5.	Wizard of flexible subsystems.....	11-11
11.5.1.	Animation window	11-12
11.5.2.	Control form.....	11-12
11.5.2.1.	General tab	11-13
11.5.2.2.	Solution tab	11-15
11.5.2.3.	Image tab.....	11-18
11.5.2.4.	Position tab.....	11-19
11.6.	Adding the flexible subsystem into a hybrid model	11-20
11.6.1.	Adding the flexible subsystem.....	11-20
11.6.2.	Flexible subsystem inspector	11-21
11.6.2.1.	General tab	11-21
11.6.2.2.	Position tab.....	11-21
11.6.2.3.	Solution tab	11-22
11.6.3.	Features of adding joints and forces.....	11-23
11.7.	Analysis of dynamics of flexible subsystem in model.....	11-24
11.7.1.	Object simulation inspector	11-24
11.7.1.1.	Simulation tab	11-24
11.7.1.2.	The Image tab.....	11-26
11.7.2.	Variables	11-27
11.7.2.1.	Coordinates	11-27
11.7.2.2.	Linear variables	11-27

11. Simulation of dynamics of flexible bodies using UM FEM

11.1. Basic ideas and methods

11.1.1. Introduction

UM FEM module is a set of software tools that are built-in **UM Input** and **UM Simulation** programs. The module gives a user a possibility to introduce flexible bodies under large displacements into a model of mechanical system. Flexible displacements are supposed to be small in the body-fixed frame of reference and could be described in terms of linear finite-element analysis (FEA). Introducing flexible bodies into a model of mechanical system is used for creating the more detailed models and obtaining more accurate results of simulation.

In some cases modeling the system with the help of rigid bodies only is too rough approximation of a real system. Then some bodies of the model should be considered as flexible, for example, car body and chassis of transport machines. Using flexible bodies to obtain more accurate solution (coordinates, accelerations) and widen its spectrum that might be important in some cases, for example, for analysis of vibrations and durability of machines.

UM FEM needs that **UM Subsystems** module is also being installed on your computer. As well as it is necessary that a FEA preprocessor and solver are available on your computer. The present **UM FEM** version supports import from **ANSYS** software version **5.5** and later.

It supposes that you have at least basic skills in using **ANSYS** software and have an idea of modal approach.

In this section some basic information concerning methods of simulation of flexible bodies in **UM FEM** is presented.

Mathematical model of a flexible body is based on using the following methods:

- subsystem technique,
- floating frame of reference method,
- finite-element method,
- Craig-Bampton method.

Every flexible body is considered as a separate subsystem that is why assembly of composite¹ model is similar to assembly of multibody model. Before assembly the preliminarily step of preparing the necessary data of FE-model of flexible bodies should take place. Flexible bodies/subsystems can interact with any other rigid or flexible bodies with the help of joints and force elements.

11.1.2. Kinematics

Kinematics of flexible bodies is described with the help of so called *floating frame of reference CS1*. Kinematical formulas are noted in this *floating frame of reference*. Position of certain point *K* of the flexible body in the global CS0 is defined as follows (Fig. 11.1):

$$\mathbf{r}_k^0 = \mathbf{r}_{01}^0 + \mathbf{A}_{01}(\boldsymbol{\rho}_k^1 + \mathbf{d}_k^1), \quad (11.1)$$

where \mathbf{r}_{01}^0 is radius vector of the origin of CS1 in CS0, \mathbf{A}_{01} is transformation matrix, $\boldsymbol{\rho}_k^1$ is radius vector of point *K* of undistorted flexible body in CS1, vector \mathbf{d}_k^1 presents elastic displacements of the point, superscript denotes the coordinate system in which vectors are given.

Elastic properties of the flexible bodies relatively to the CS1 are described with the help of finite-element method. The present **UM FEM** version supports import of data about flexible bodies from **ANSYS** software version **5.5** and later.

¹ Composite or hybrid model includes both rigid and flexible bodies

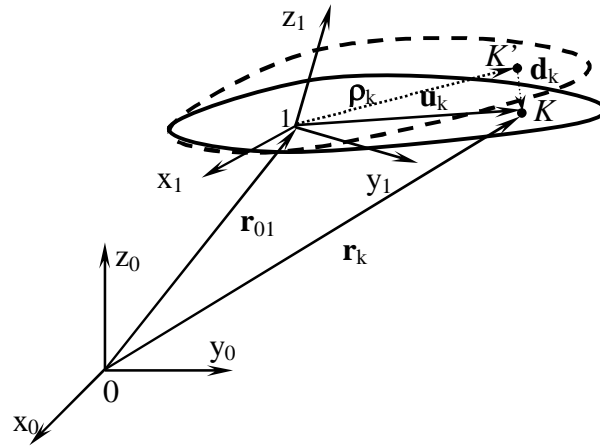


Figure 11.1. Floating frame of reference

Small elastic displacements are presented as a sum H of possible modes/shapes of flexible body:

$$\mathbf{x} = \sum_{j=1}^H \mathbf{h}_j w_j = \mathbf{H} \mathbf{w} , \tag{11.2}$$

where \mathbf{x} is nodal degrees of freedom of the flexible body, \mathbf{h}_j is the possible mode, w_j is the *modal* coordinate that describes flexible displacements correspond to mode j . The matrix \mathbf{H} is called *modal* matrix.

According to the Craig-Bampton method the *modal* matrix is formed as a combination of *eigenmodes* and *static modes*. The method consists of four steps.

- 1) Choice of *interface (boundary)* nodes of a finite-element scheme.
- 2) Successive calculation of *static* modes. Static modes are static shapes obtained by given each boundary d.o.f. a unit displacement while holding all other boundary d.o.f. fixed.
- 3) Calculation of *eigenmodes* while holding all *interface* nodes fixed;
- 4) Calculation of the mass matrix and the stiffness matrix, orthonormalization of the eigenmodes and static modes.

The short description of the each step is given below.

Choice of interface nodes. Flexible body/subsystem interacts with other bodies of the model via joints and force elements. It is recommended that every attachment point should be situated in the node of finite-element mesh. Very these nodes, where joints and force elements are attached to, should be chosen as *interface* nodes. Such an approach helps to create joint constrains correctly and quite accurate describe flexible displacements that determine force in force element.

It is necessary to choose *interface* nodes so as during calculation of each *static* mode the immobility of the subsystem was guaranteed.

Calculation of static modes. The number of static modes is equal to number of d.o.f. in *interface* nodes. During this procedure *interface* nodes are held fixed and static modes are obtained by given each interface d.o.f. a unit displacement/rotation.

Calculation of eigenmodes. Eigenmodes of flexible body are obtained from the solving the generalized eigenproblem:

$$(\mathbf{C} - \lambda \mathbf{M}) \mathbf{y} = 0, \tag{11.3}$$

where \mathbf{C} is the stiffness matrix, \mathbf{M} is the mass matrix, λ is the eigenvalue, \mathbf{y} is the eigenmode. If these matrices are of a full rank the equation (11.3) has N solutions, where N is the number of rows that correspond to nodal d.o.f. The mass matrix of the flexible subsystem may be formed based on shape functions of finite elements or may have a diagonal form as a result of using lumped model. A user determines number and shapes of used eigenmodes. As a rule a set of eigenmodes includes lower eigenmodes.

Calculation of generalized matrices, orthonormalization of modes. Generalized mass and stiffness matrices are calculated using the modal matrix \mathbf{H} :

$$\bar{\mathbf{M}} = \mathbf{H}^T \mathbf{M} \mathbf{H}, \quad \bar{\mathbf{C}} = \mathbf{H}^T \mathbf{C} \mathbf{H}$$

where $\bar{\mathbf{M}}$ is the generalized mass matrix, $\bar{\mathbf{C}}$ is the generalized stiffness matrix.

The final step of the preparing set of modes is the orthonormalization of columns of the modal matrix based on eigenvalue problem solution with generalized mass and stiffness matrix:

$$(\bar{\mathbf{C}} - \lambda \bar{\mathbf{M}}) \bar{\mathbf{y}} = 0 \tag{11.4}$$

Transformed set of modes is formed based on the equation:

$$\bar{\mathbf{H}} = \mathbf{H} \bar{\mathbf{Y}} \tag{11.5}$$

Diagonal form of transformed generalized matrices leads to minimal CPU efforts during the integration of equations of motion. It is the basic advantage of such an approach. Another aim of such transformations is exclusion modes that correspond to movement of the flexible subsystem as a rigid body. It is necessary since movement the flexible subsystem as rigid one is defined by *floating frame of reference* CS1. Zero eigenvalues correspond to rigid body modes of flexible subsystem (11.4).

11.2. Installation, preparing data, workflow

UM FEM installation package includes the following items:

- macro file **um.mac** for ANSYS, which is written in APDL (ANSYS Parametric Design Language);
- stand alone program for data transformation **ansys_um.exe**;
- **wizard of flexible subsystems** built in **uminput.exe** program;
- software procedures for handling and simulation of dynamics of flexible bodies that are built in **uminput.exe** and **umsimul.exe**.

Simulation of dynamics of flexible bodies supposes the following steps to be done.

- 1) Creating the FEA model of the flexible body to analyze in the external FEA software.
- 2) Choosing the interface nodes, calculation of the *eigenmodes* and *static modes* according to Craig-Bampton method.
- 3) Exporting data from external FEA software and its transformation to UM format.
- 4) Including the flexible subsystem into hybrid model with the help of **UM Input** program.
- 5) Simulation of dynamics of the hybrid model with the help of **UM Simulation** program.

The whole workflow of the preparing input data for models that include flexible bodies is shown in Fig. 11.2. Let us consider basic steps of this procedure.

The first step is executed under ANSYS environment. According to instructions to ANSYS software the work directory and *JobName* are chosen. *JobName* is a name of all the files for certain FEA model.

After creating the FEA model and choosing interface nodes the macros **um.mac** is executed. This macros has commands for calculation of eigenmodes and static modes, as well as calculation and exporting mass and stiffness matrices. As a result of **um.mac** execution several files are created: standard ANSYS result file *JobName.rst*, *JobName.full* that contains matrices of a flexible body corresponded to fixed interface nodes, *JobName.free* that contains matrices of a free body, and *JobName.mlmp* with a diagonal mass matrix of a free body. In dependence of arguments of the **um.mac** the *JobName.mlmp* file may not be created.

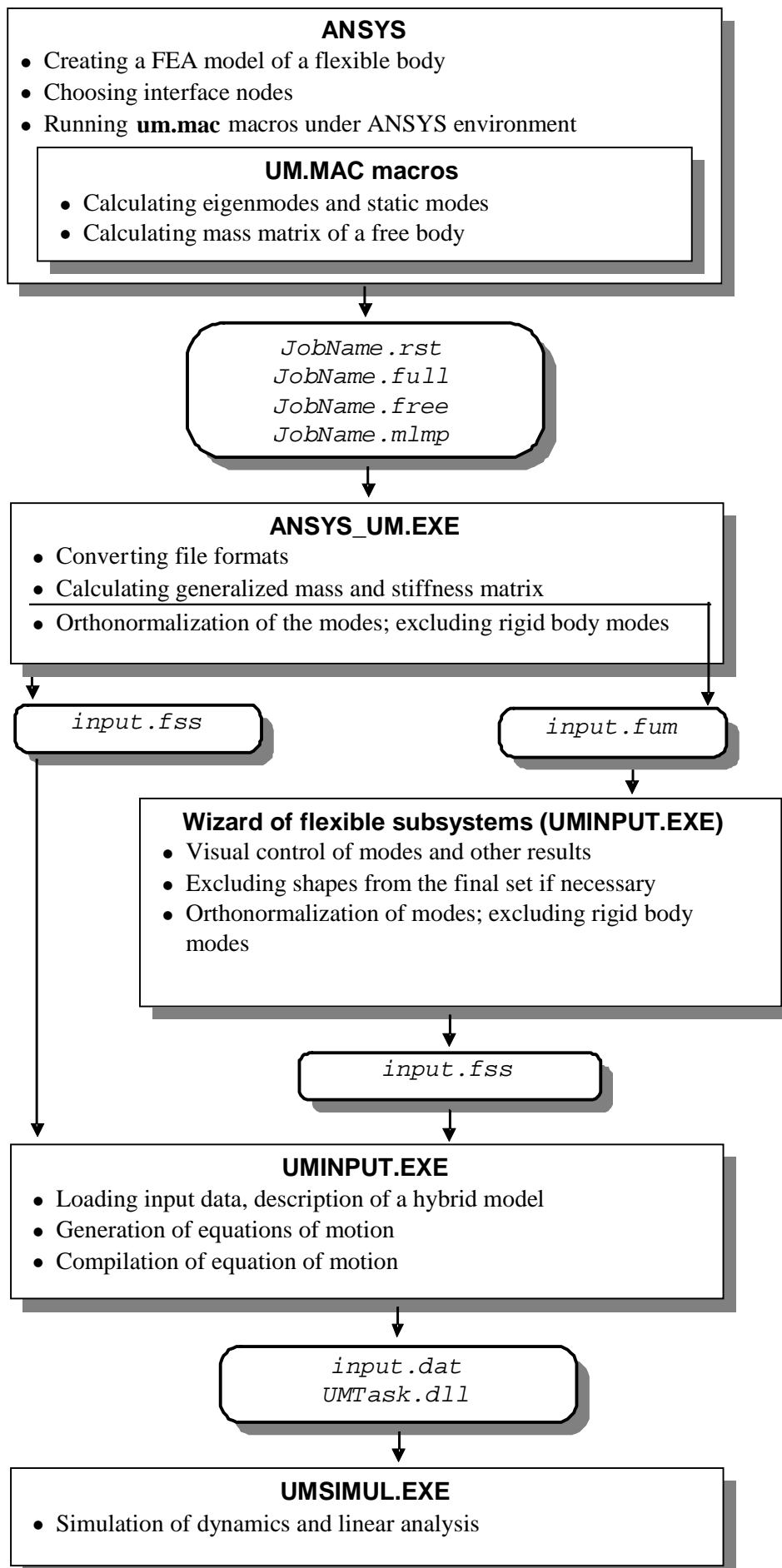


Figure 11.2. Data preparing workflow

After installation the **um.mac** file is situated in the **{um_root}\bin** directory. Copy the **um.mac** file to the directory that is selected as a default directory for the macro files in ANSYS. It is usually **.\docu** directory from the ANSYS root directory. Otherwise you should indicate the path to the **um.mac** file using **PSEARCH** command:

```
/PSEARCH, path_to_um.mac.
```

The second step of the data preparing is fulfilled in the **ansys_um.exe** program, which is situated in the **{um_root}\bin** directory. **Ansys_um.exe** may produce the final ready-to-use *input.fss* or *input.fum*, which contains intermediate data. The second way with *input.fum* often is more convenient for further analysis.

Ansys_um.exe can be run automatically right from the **um.mac** or manually. To run **ansys_um.exe** automatically you should open the **um.mac** and edit the last line with the **/sys** command. The argument of this command should be the correct path to the **ansys_um.exe** program. For example,

```
/sys, c:\um50\bin\ANSYS_UM.exe
```

Note. ANSYS ignores the **/sys** command if it contains spaces. In order to run **ansys_um.exe** from the **um.mac** you should copy **ansys_um.exe** and **um.rsc** from the **{um_root}\bin** to the directory without spaces in path.

The **Wizard of flexible subsystems (UM Input** program) gives a user additional possibilities for preparing data. Using the *input.fum* the **Wizard** let the user visually control the calculated modes, exclude some modes if necessary and fulfill all the transformations for creating the final *input.fss*.

Further work flow is very similar to modeling with usual subsystems. The *input.fss* file is the standard data file for the flexible subsystem just as *input.dat* is a data file for the whole model. A name of directory that contains *input.fss* is considered as a name of the flexible subsystem.

Describing a hybrid model should be done within **UM Input** program. This program generates a data file of the hybrid model *input.dat*, generates equations of motion and compiles them as *UMTask.dll* file, which is used in the **UM Simulation** program for numerical solving these equations.

11.3. Preparing data under ANSYS environment

Preparing data under ANSYS environment includes three basic steps. Let us consider them more detailed.

1. Describing the flexible body in terms of ANSYS according ANSYS User's Guide.

Note. It is necessary to use *System International* for all units. Use the command
/UNITS,SI

Note. During the preparing the data and creating a FE-mesh it is necessary to provide creating the nodes of FE-mesh in joint points and points of attaching force elements. To create nodes of FE-mesh with specified coordinates in body-fixes reference frame you can set there key points (*K* command), hardpoints (*HPTCREATE* command) or with the help of choosing appropriate parameters of automatic generation of FE-mesh.

2. Selection of the *interface nodes* of the FE-mesh of the flexible body with the help of sequence of *NSEL* commands or combination of the *KSEL* and *NSLK* commands. For example, you can select *interface nodes* as following:

- NSEL,s,,,1,10,1 !selection of a new set of nodes
!from #1 to #10, step 1
- NSEL,s,,,1 !selection of a new set of nodes that includes
!one node #1
NSEL,a,,,385 !add one more node #385 to the set of
!selected nodes
- KSEL,s,,,1 !creating the a set of key points consisted of
!one key point #1
KSEL,a,,,23 !add the #23 key point to the set of
!selected key points
NSLK,S !selection of a new set of nodes associated
!with selected key points

3. Running the **um.mac** macro-command from the ANSYS command line:

```
UM,NEForms,WayM
```

- NEForms - required number of eigenmodes correspond to the lowest eigenvalues of the flexible body;
- WayM - way of forming mass matrix:
0 – mass matrix, based on shape functions of finite elements;
1 – diagonal mass matrix.

Note. If you suppose to run series of calculations with various numbers of eigenmodes it is recommended that you set maximal number of eigenmodes in NEForms . Once exporting here all the eigenmodes that could be used for all of the calculations you will be able to remove some of them later using the **Wizard of flexible subsystems**.

Note. The message «6 RIGID modes defined but only 5 total modes requested. Solution not interesting» might appears during running UM macros. It is not a critical error and is connected with calculation of mass matrix of a free body. Close the error message and go on working.

11.4. ANSYS-UM data exchange

The ANSYS_UM.EXE is considered in this section and used for importing data from ANSYS to UM. It converts data that made by **um.mac** macros and saves this data in UM format.

Input.fss file includes set of transformed modes without rigid body modes of the flexible subsystem.

UM Input program directly loads and supports this file for describing flexible subsystems.

Input.fum includes intermediate data, that could be transformed with the help of **Wizard of flexible subsystem**. This file contains static modes, eigenmodes and the generalized mass matrix.

Files **.fum* and **.fss* are of the same structure and contain information about modes and matrices as well as information about FE-mesh, nodes and elements.

Ansyz_um includes the **General** and **Options** tabs, see Fig. 11.3.

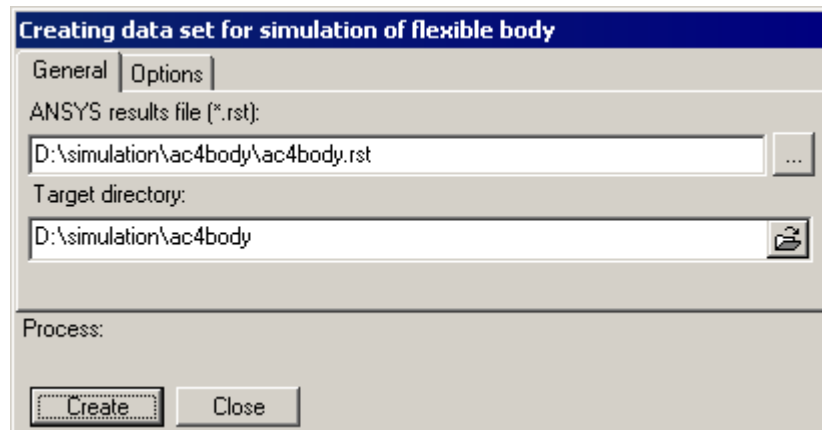


Figure 11.3.

The **General** tab (Fig. 11.3) let the user select a **.rst* file and set the target directory for saving *input.fum* / *input.fss* files. It is recommended to create this target directory in advance.

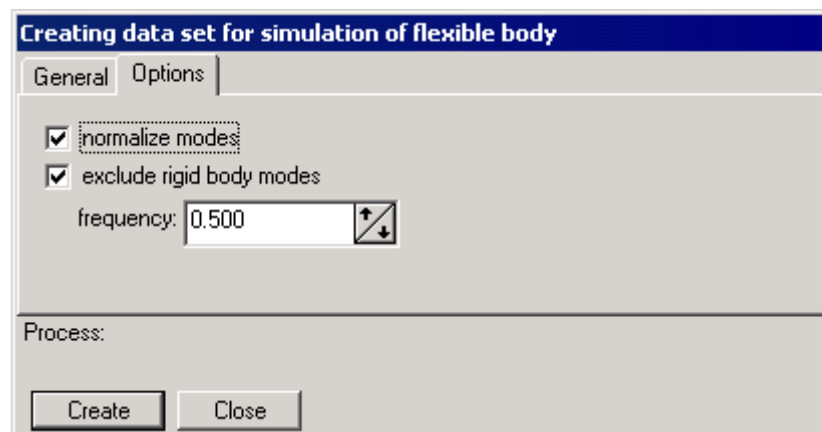


Figure 11.4.

The **Options** tab (Fig. 11.4) defines structure of output files. Following variants are possible.

- Set of static modes and eigenmodes of the flexible subsystem, the generalized mass and the stiffness matrix (transformations 11.4, 11.5 are omitted). To prepare such a set of data turn off **normalize modes** flag, the rest control elements are not enabled.
- Set of transformed modes that includes modes of motion as rigid body. To prepare such a set of data turn on the **normalize modes** flag and turn off the **exclude rigid body modes** flag. In this case to finish preparing data it is necessary to exclude rigid body modes later in **Wizard of flexible subsystems**.
- Set of transformed modes, without rigid body modes. Turn on **normalize modes** and **exclude rigid body modes** flags. In this case a value in the **frequency** box defines maximum module of natural

frequency that correspond to rigid body motion. This variant of setting lead to creating the files which can be directly loaded in **UM Input** and **UM Simulation** programs.

- Notes.**
- 1) Severe solution provides zero eigenvalues for eigenmodes that correspond to free body motion. However round-off errors during numerical solution lead to appearing the small non-zero eigenvalues in the spectrum of a problem. Here the **frequency** option is used.
 - 2) When coupled mass matrix is used the latter variant is obligatory.
 - 3) If you use a diagonal mass matrix it is recommended to use the variant with turned on **normalize modes** and then use **Wizard of flexible subsystems**.

Load a *.rst file, select the suitable options and click **Create** button. Preparing output data takes some time that depend on number of nodes, static modes and eigenmodes. Correspondent message informs you about results of calculation.

11.5. Wizard of flexible subsystems

Wizard of flexible subsystems (Fig. 11.5) is implemented as a tool within **UM Input** program and is aimed for flexible subsystems data control and transformation. **Wizard** is also can be used for excluding some shapes from the final solution as well as for orthonormalization of modes and excluding rigid body modes. Functions of the **wizard** are quite similar to the **ANSYS_UM** program.

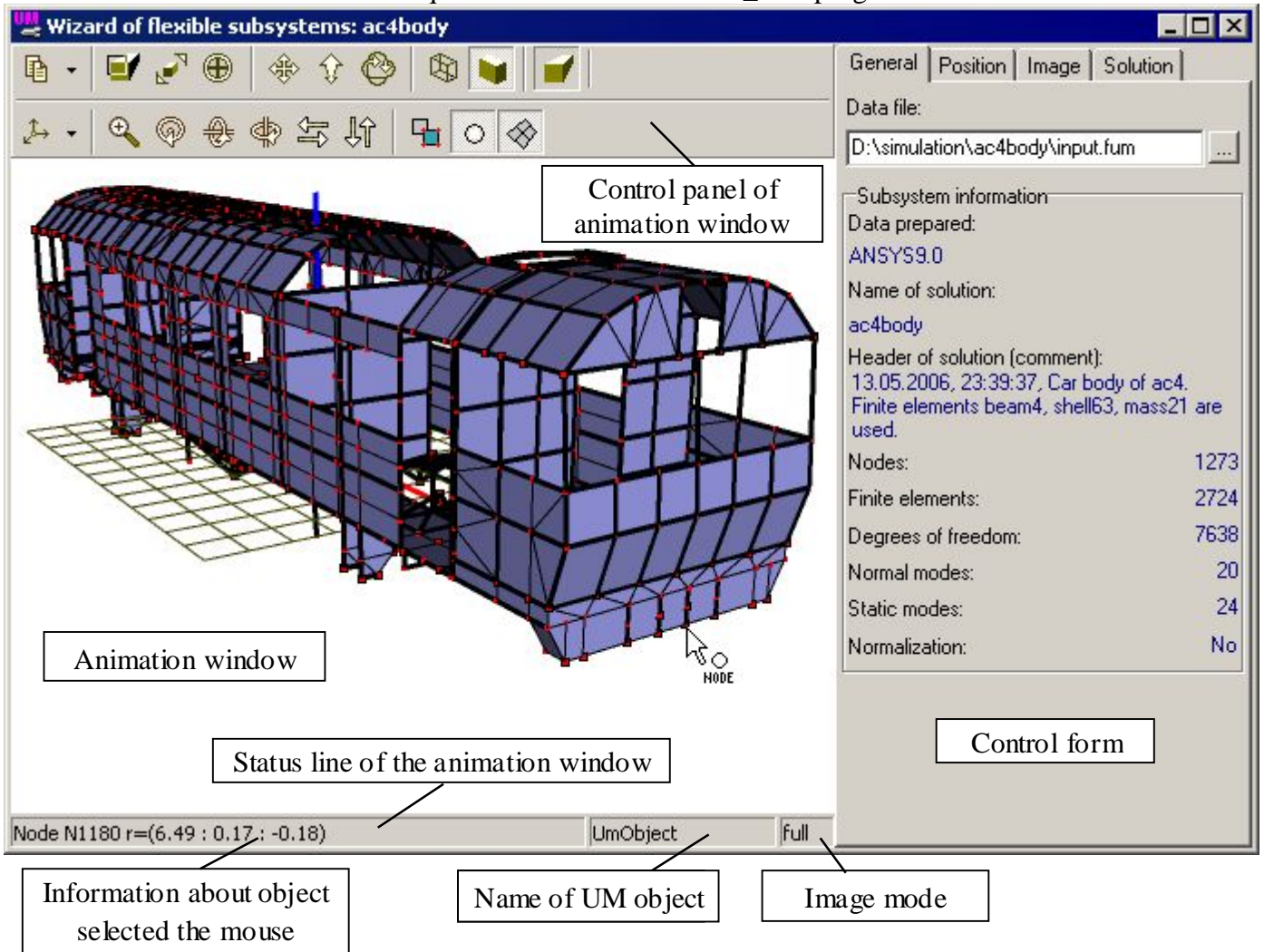
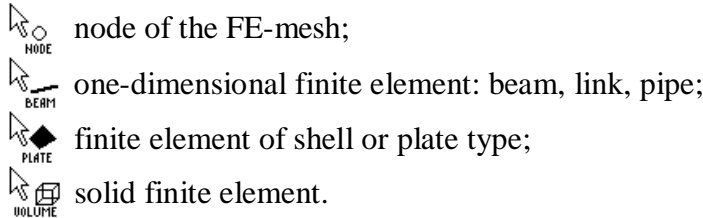


Figure 11.5. Wizard of flexible subsystems

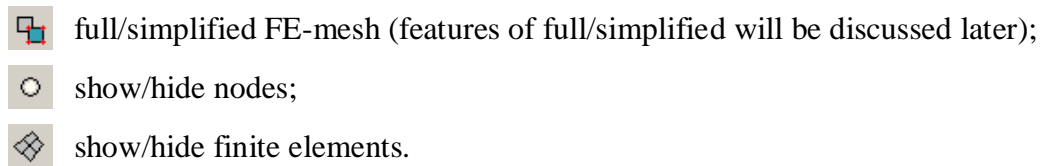
11.5.1. Animation window

An animation window of the **wizard of flexible subsystems** is aimed for visualization of the FE-mesh, nodes and elements, as well as animation of flexible modes. Common functions of animation window are discussed in the Sect. 3.3.1.2. Here we will discuss additional functions of the animation window concerning visualization of the FE-mesh.

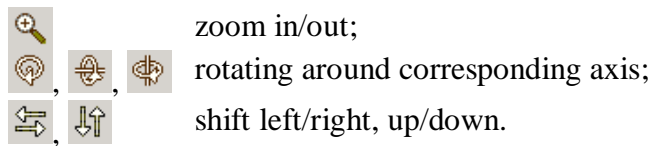
- Mouse cursors:



- Status line shows information about current selected node or element.
- Additional buttons that affect for FE-mesh visualization:



Besides standard buttons animation window has additional positioning buttons:



Use the context menu to hide/show tool panels, see Fig. 11.6.).

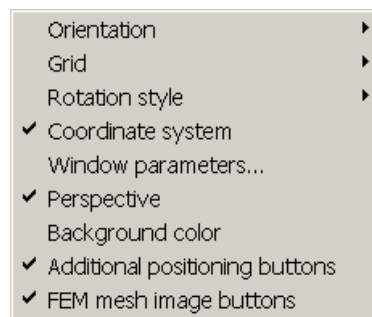


Figure 11.6.

11.5.2. Control form

Control form includes four tabs.

- General tab shows general information about the flexible subsystem.
- Position tab let the user a possibility to change the flexible body position and orientation.
- Image tab contains control elements, which are used for definition of graphical representation of the flexible subsystem.
- Solution tab show information about current solution with the help of two descendant tabs:

- **Modes** tab shows information about calculated static modes and eigenmodes, as well as control elements for excluding some modes and their orthonormalization;
- **Rigid body** shows inertia parameters of the flexible subsystem as a rigid body.

11.5.2.1. General tab

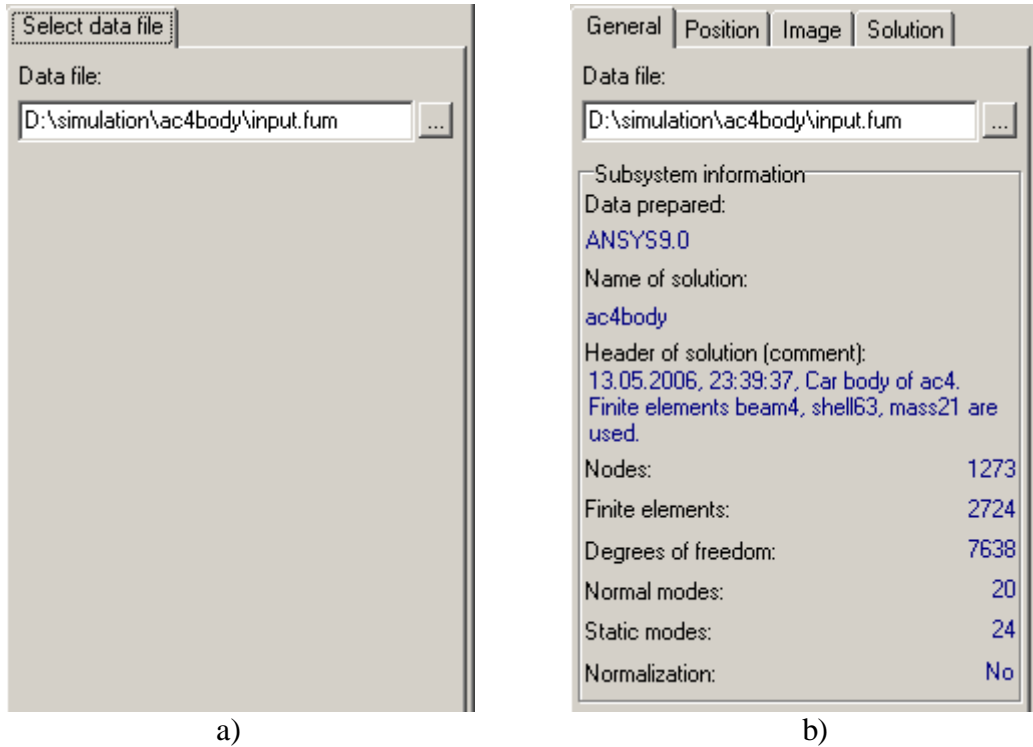


Figure 11.7. General tab

When you just started the **Wizard of flexible subsystems** it has the only tab “**Select data file**”, see Fig. 11.7a. The **Data file** box contains the last loaded file, if the **Wizard** starts at the first time this field is empty. Use the **...** button to start the open dialog, see Fig. 11.8.

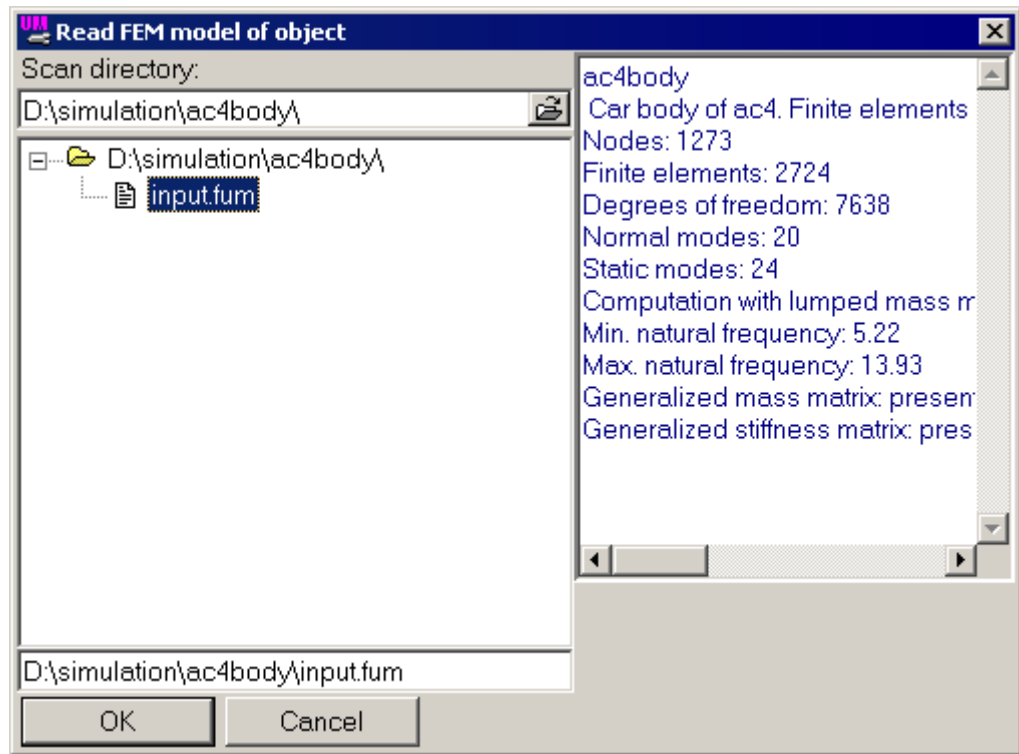


Figure 11.8.

This dialog has the following features:

- you can load only *.fum files in this dialog;
- the right panel shows summary text information about the selected flexible subsystem.

After loading the *.fum file the **General** tab looks like it is shown in Fig. 11.7b.

11.5.2.2. Solution tab

Let us consider basic features of the **Solution** tab, see Fig. 11.9.

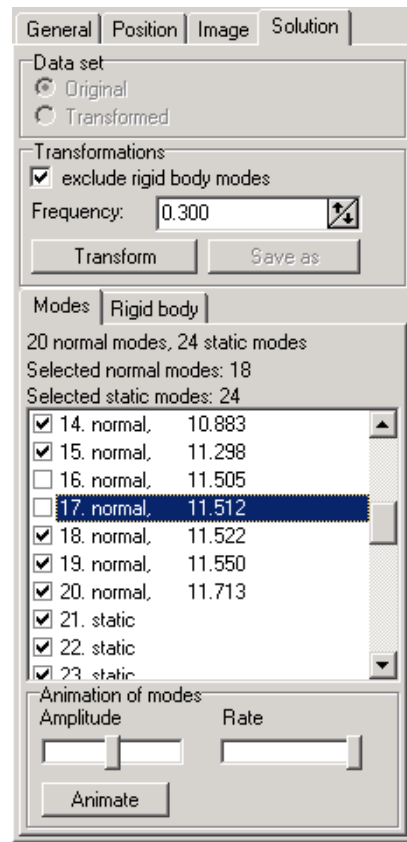


Figure 11.9.

There are two tabs in the bottom part of the **Solution** tab: **Modes** and **Rigid body**.

- The **Modes** tab includes the following components.
 - Summary text information
 - List of modes. Each list item corresponds to a mode of the flexible subsystem and shows the following information:
 - index of the mode in the set of modes;
 - type of mode: static or eigenmode;
 - eigenvalue for the eigenmode.

Every list item has a check box, which is shown if the mode is included to the transformed set of modes.

The list of modes is filled according to the following rules:

- eigenmodes according their eigenvalues;
- static modes according to the order of their calculation in the FE program (ANSYS).

The list of modes shows the complete set of calculated eigenmodes and static modes and shows the flexible mode in the animation window. Besides that the list of modes gives the user a possibility to create new reduced set of modes as a subset of already calculated modes. You should turn on the check boxes that correspond to the modes you are intended to add to the new subset of modes.

- The **Animation of modes** group contains the following elements.
 - The **Animate** button starts animation of modes.
 - Track bars **Amplitude** and **Rate** set the scale factor and the frames per second for animation correspondently.

Note. Maximal number of frames per second depends on number of nodes and finite elements of the flexible subsystem, computer performances and settings of graphical representation of the flexible subsystem in the **Image** tab.

- The **Rigid body** tab (see Fig. 11.10) shows information about position of center of gravity, mass and moments of inertia of the flexible subsystem.

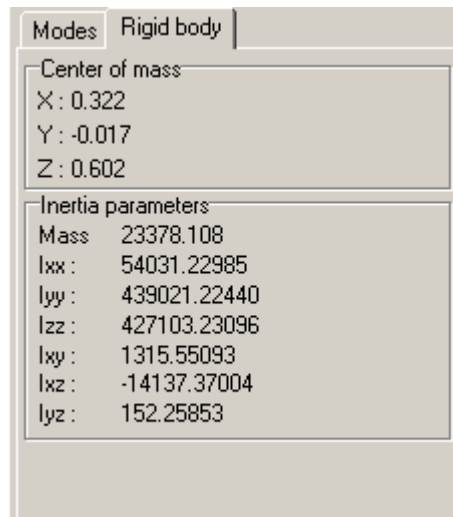


Figure 11.10.

- The **Data set** radio group defines original or transformed data set as a current one. This group is enabled if the transformed data set is not created.
- **Transformations** group is aimed for orthonormalization of selected flexible modes.
 - Click **Transform** to start transformation process. A new set of flexible modes is created, the source flexible modes is not changed. In the case of successful transformation the **Data set** radio group becomes enabled. Set **Data set** to **Transformed** to control the transformed modes visually.
 - The **exclude rigid body modes** flag mostly should be turned on. Turn off the flag when there is no possibility to exclude rigid body modes automatically. In this case you should exclude rigid body modes manually.
 - The minimal frequency that corresponds to rigid body modes is set in the **Frequency** box, see Sect. 11.4.
 - The **Save as** button is aimed for saving the orthonormalized set of modes. Input the full path to the subsystem in the save dialog (Fig. 11.11).

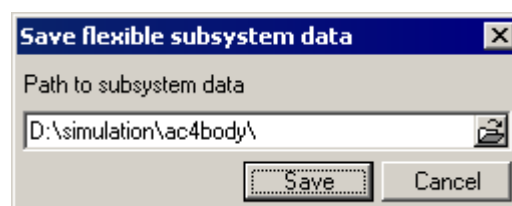


Figure 11.11.

A new *input.fss* file will be created in the specified directory. To create another data set you should select the **Original** in the **Data set** group and then prepare the new data set.

11.5.2.3. Image tab

This tab is intended for describing the graphical image of the FE-mesh of the flexible subsystem in an animation window, see Fig. 11.12.

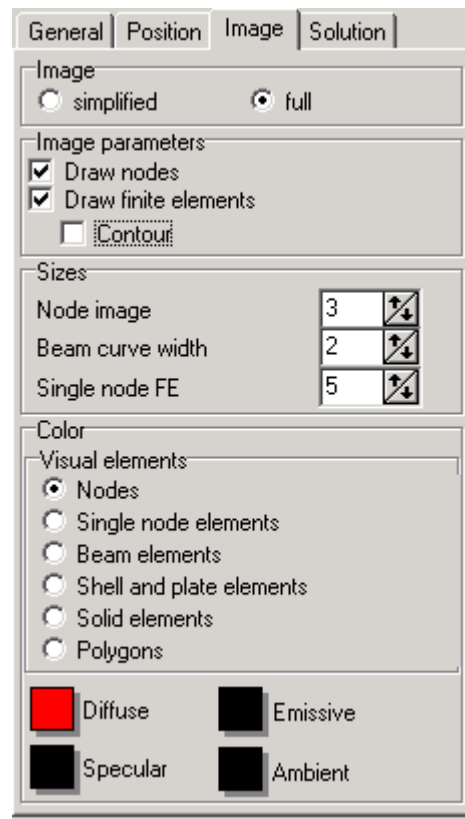


Figure 11.12.

- You can choose **simplified** or **full** options in the **Image** group.

In the **full** mode you can see information about each node and element in the status line of the window when selecting them with the help of a mouse.

If you select the **simplified** mode then the flexible subsystem is shown as one graphical element with all nodes of the FE-mesh. Information about nodes and elements under the mouse is not available in this mode. Nodes are not drawn and flexible subsystem has **Polygons** color, see **Color** group. **Simplified** mode is intended for big FE-meshes and helps to significantly increase animation rate.

11.5.2.4. Position tab

With the help of this tab you can change position and orientation of the flexible subsystem relative to basic inertial system of coordinates *SC0*. All these data influence on the representation of a graphical image of flexible subsystem in the animation window.

The next step of creating the hybrid (rigid + flexible bodies) model is describing the interactions of flexible subsystem and other bodies of the model. Use the **UM Input** program for that.

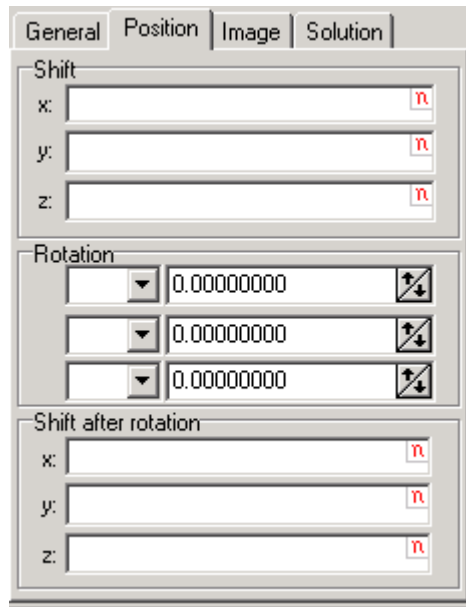



Figure 11.13.

11.6. Adding the flexible subsystem into a hybrid model

11.6.1. Adding the flexible subsystem

Select the **Subsystems** item in the list of elements in the left part of the constructor window and add a new subsystem by the  button, see Fig. 11.14. Choose the **Linear FEM subsystem** item from the drop-down list and then select the flexible subsystem (*input.fss*) in the open dialog. After loading the flexible subsystem input a name for the new subsystem.

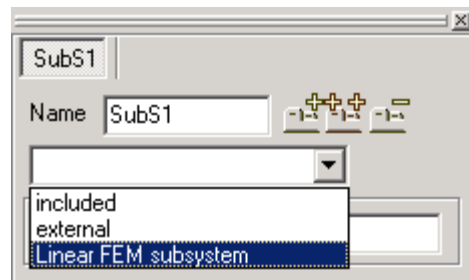


Figure 11.14.

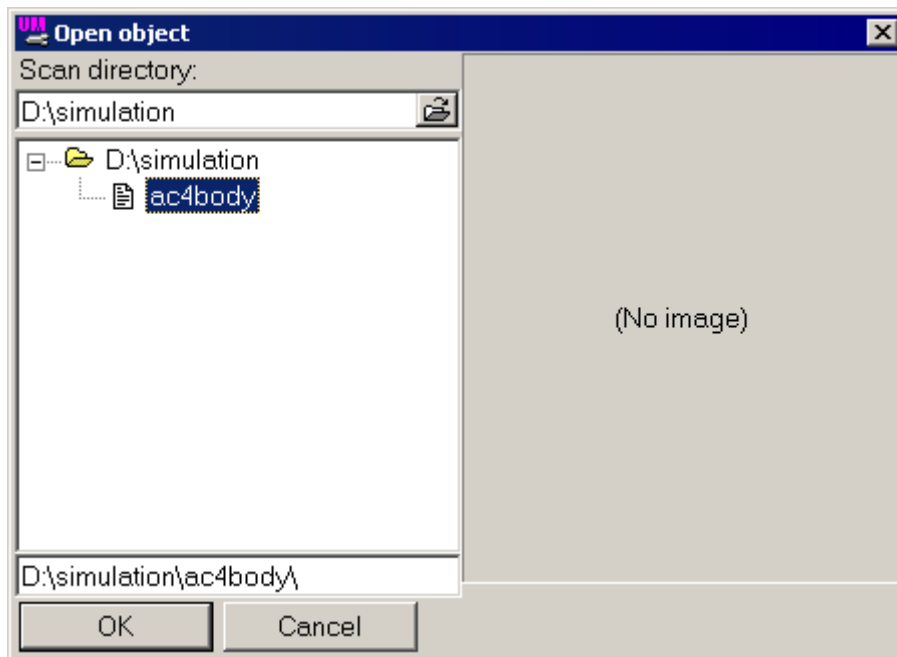


Figure 11.15.

Note. After adding the flexible subsystem a fictitious rigid body and a 6 d.o.f. joint are automatically created. The fictitious body has a name of solution, see **General** tab. The fictitious joint is not included into the list of elements. The fictitious rigid body and joint are introduced into the model for uniform creating joint and force elements between flexible subsystem and the rest part of the mechanical system.

11.6.2. Flexible subsystem inspector

After loading the flexible subsystem its parameters are shown in the inspector window. This window is similar to **Wizard of flexible subsystems**. Let us consider basic distinctions of this window.

11.6.2.1. General tab

The **General** tab contains some additional boxes, Fig. 11.16.

- **Identifier** is used during the programming under UM environment. Syntax rules for identifiers are given in sect. 3.3.2.3.2.
- **Ancestor** shows path to the flexible subsystem source data.
- **Angles of orientation** (sequence of angles) determine orientation of the flexible subsystem.

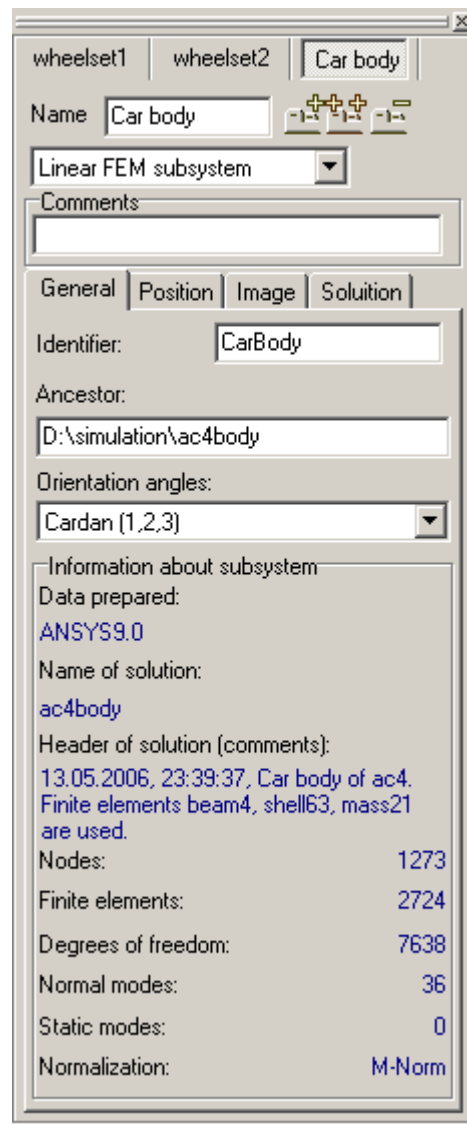


Figure 11.16.

11.6.2.2. Position tab

This tab let the user a possibility to determine initial position of the flexible subsystem in the basic inertial frame of reference according to design of mechanical system.

11.6.2.3. Solution tab

This tab is aimed for informational purposes only. Here you cannot change the set of modes, but you can animate all modes in the current solution.

11.6.3. Features of adding joints and forces

After adding the new flexible subsystem it is necessary to describe attached joints and force elements. Basics of creating joints and force elements are given in the Sect. 3.4.6. Here we will consider some basic distinctions.

- When you describe a joint or a force element for the flexible subsystem you should select a fictitious rigid body that has the solution name as one of the body in the joint/force, see Fig. 11.17.
- A joint or a force element can be attached in a node of the FE-mesh only. You should keep in mind this fact during creating FE-mesh of the flexible body and specially create nodes so as it would be a node for each attached joint or force element. If there is no a node in the point where a joint/force should be attached to the flexible body, program will find the nearest node for this joint/force.
- In the present UM version forces should act at the body-fixed points.

Describing interaction between the flexible subsystem and the rest part of the model is the final step of creating hybrid models. Then you should generate and compile equations of motion (Sect. 3.6) and can start simulation.

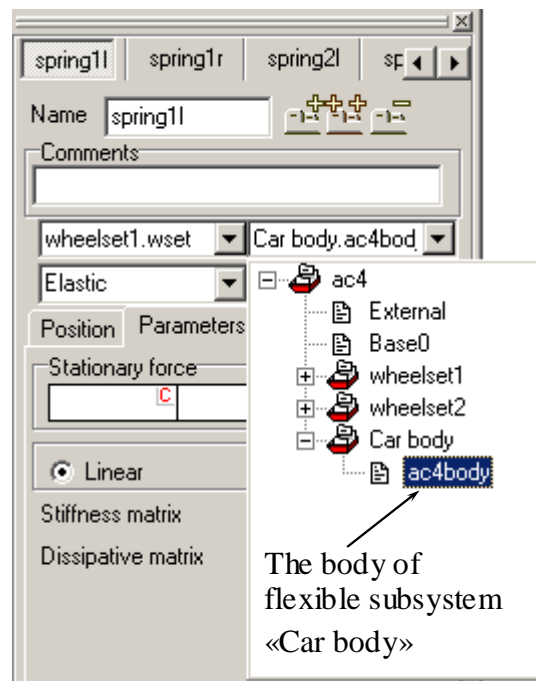


Figure 11.17.

11.7. Analysis of dynamics of flexible subsystem in model

Practically all tools, which are available in UM for rigid body modeling, support flexible bodies. The working procedure with the **UM Simulation** program is described in Chapter 4. Let us consider simulation features relating to the presence of flexible subsystems in a UM model.

Let us consider the **UM Simulation** tools for working with flexible subsystems.

11.7.1. Object simulation inspector

If a model contains flexible subsystems then the **FEM Subsystems** tab appears on the **Object simulation inspector**. The tab has the **Simulation**, **Image** and **Solution** tabs, Fig. 11.18.

If the model contains more than one flexible subsystem then the list with all subsystems appears. The user can select a subsystem from the list to set its parameters.

11.7.1.1. Simulation tab

The **Simulation** tab contains the following interface elements:

- The **Gravity** flag allows switching on/off gravity. It may be necessary when a subsystem does not have large displacement.
- The **Switch off all flexible modes** flag allows modeling a flexible subsystem as a rigid body. In this case the subsystem has only six joint coordinates as a free body. The flag is enabled if the subsystem interacts with the object by means of force elements only.

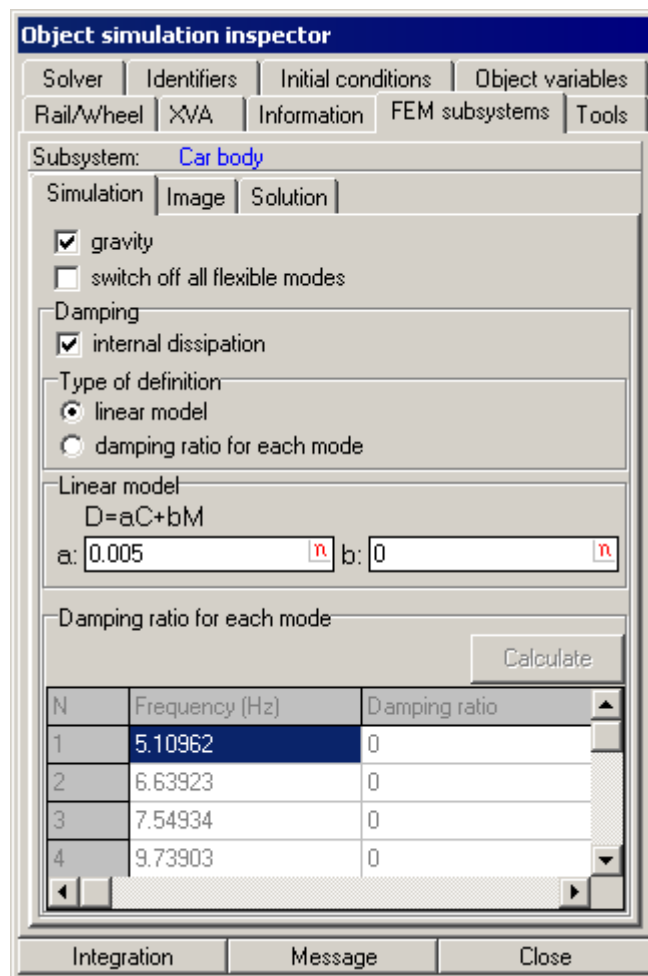


Figure 11.18. Object simulation inspector

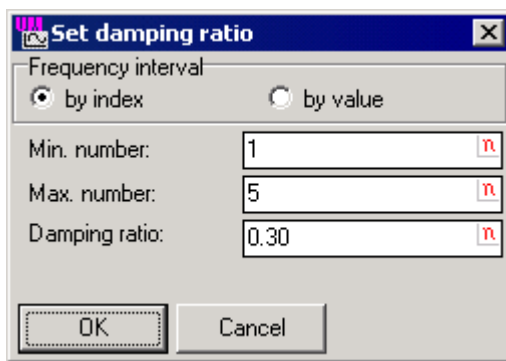
- The **Damping** group determines internal dissipation, choosing the mathematical model of dissipation and parameters of the model. Let us consider the **Type of definition** group:
 - **Linear model** allows setting the dissipation matrix as a sum of mass and stiffness matrices multiplied by ratios. User can set the values of ratios at his or her discretion.

N	Frequency (Hz)	Damping ratio
1	5.10962	0.3
2	6.63923	0.3
3	7.54934	0.3
4	9.73903	0.5

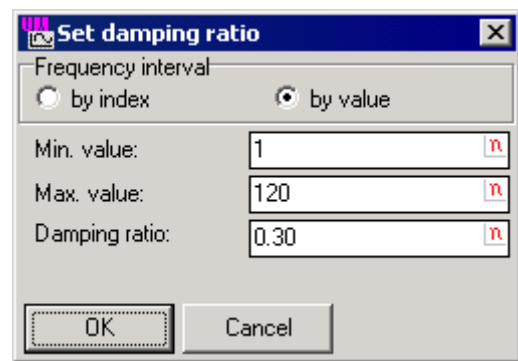
Figure 11.19. Damping ratio

- When **Damping ratio for each mode** is checked the dissipation for every mode as a damping ratio, Fig. 11.19. The critical damping ratio equal to 1 and separates non-oscillatory motion from oscillatory motion.

The values of damping ratio for the given range of flexible modes can be assigned with the help of the dialog window (Fig. 11.20). Use the popup menu to show this dialog. The frequency interval can be set by indices of modes (Fig. 11.20a) or by the value of the frequency (Fig. 11.20b).



a)



b)

Figure 11.20. Damping ratio

Since the flexible modes of a subsystem are orthonormal, values of damping ratio could be calculated based on the linear damping model. For that it is necessary to click the **Calculate** button on the **Simulation** tab (Fig. 11.18). You can firstly set values of coefficients of linear dissipation model **a** and **b** (Fig. 11.21) and then to click the **Calculate** button to calculate damping ratios for each mode. Values of damping ratios are available to edit.

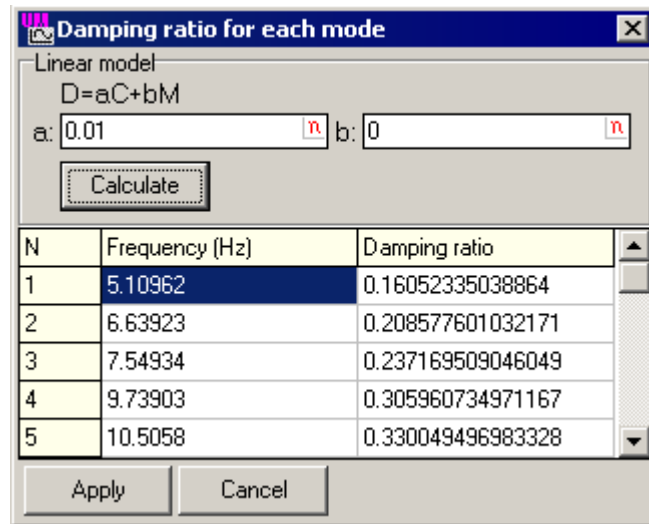


Figure 11.21. Linear model of damping ratio

To accept the calculated values for the subsystem click the **Apply** button. The **Cancel** button closes the window without changing the model parameters.

11.7.1.2. The Image tab

The **Image** tab is identical to the corresponding tabs of **Wizard of flexible subsystems** and data inspector in **UM Input** program and is described in details in Sect. 11.5.2.3.

11.7.2. Variables

You should use the **Wizard of variables** to create new variables to analyze, see Sect. 4.3.2. Here we will discuss some features, which are connected with simulation of flexible subsystems.

11.7.2.1. Coordinates

In the list of coordinates of the **Wizard of variables** you can see two groups of elements that correspond to flexible subsystem, Fig. 11.22. The first group includes six d.o.f. of a flexible subsystem as a free body (coordinates 2.1-2.6) and the second one includes complete set of generalized coordinates of the flexible subsystem (starting with coordinate 2.7).

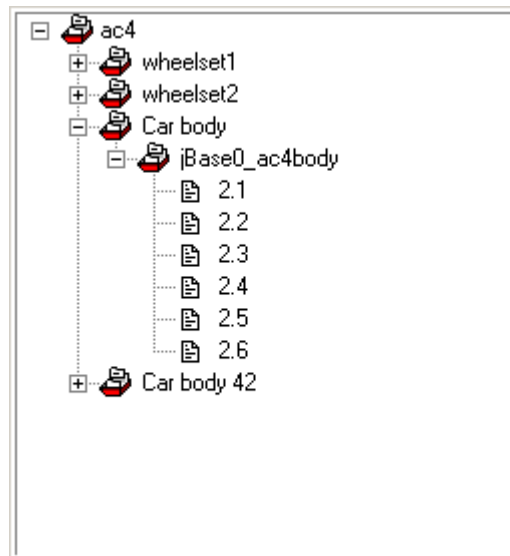


Figure 11.22. Coordinates of the model

Note. Modal coordinates are dimensionless variables. That is why make sense only comparative analysis of modal coordinates between each other.

11.7.2.2. Linear variables

Please note that kinematical variables (coordinates, velocities and accelerations) are available for nodes of finite-element mesh only. If there is no node in the indicated point then the nearest node will be chosen and the user will have the correspondent message, see Fig. 11.23.

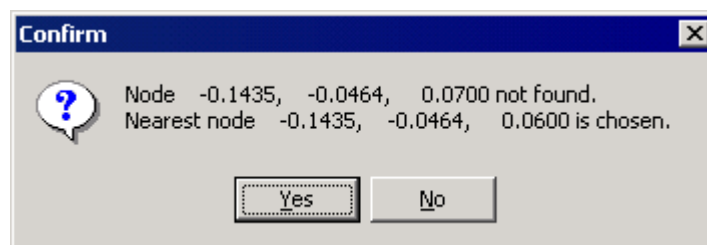


Figure 11.23.