UNIVERSAL MECHANISM 10

universal mechanism

User`s manual

Simulation of dynamics of flexible bodies

UM module for simulation of flexible bodies allows including finite-element models of flexible bodies into Universal Mechanism models. Issues of importing finite-element models, their interaction with other bodies as well as simulation of dynamics of hybrid systems are considered.

2024

Contents

1. Simulation of dynamics of flexible bodies using UM FEM

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 following FEA software:

- **ANSYS** software version **5.5** and higher;
- **MSC.NASTRAN 2005, MSC.NASTRAN 2007, MSC MD NASTRAN 2010, MSC.NASTRAN 2012, MSC.NASTRAN 2016, MSC.NASTRAN 2019;**
- **NX NASTRAN 8.0, NX NASTRAN 9.0, NX NASTRAN 12.**
- **ABAQUS 6.10-1.**
- **FIDESYS 3.0 and higher.**

It supposes that the user has at least basic skills in finite element analysis and using FE software, as well as has 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 composite[1] 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.

---

[1] *Composite* or *hybrid* model includes both rigid and flexible bodies

1.1.1. 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 (Figure 1.1):

$$\mathbf{r}_k^0 = \mathbf{r}_{01}^0 + \mathbf{A}_{01}(\boldsymbol{\rho}_k^1 + \mathbf{d}_k^1), \tag{1.1}$$

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

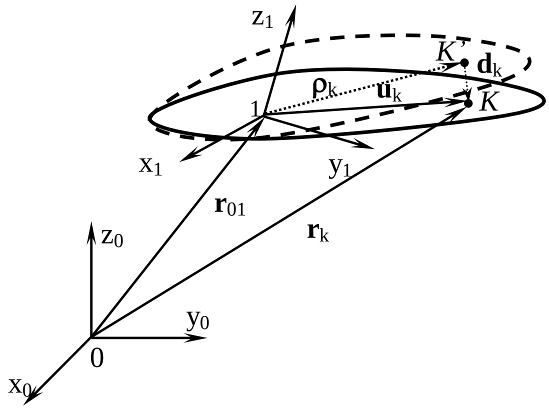


Figure 1.1. Floating frame of reference

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 external FE software products.

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

$$\mathbf{x} = \sum_{j=1}^{H} \mathbf{h}_j w_j = \mathbf{Hw}, \tag{1.2}$$

where $\mathbf{x}$ is nodal degrees of freedom of the flexible body, $\boldsymbol{h}_j$ 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:

$$(\mathbf{C} - \lambda\mathbf{M})\mathbf{y} = 0, \tag{1.3}$$

where **C** is the stiffness matrix, **M** is the mass matrix, $\boldsymbol{\lambda}$ is the eigenvalue, **y** is the eigenmode. If these matrices are of a full rank the equation (1.3) 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 **H**:

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

where $\bar{\boldsymbol{M}}$ is the generalized mass matrix, $\bar{\boldsymbol{C}}$ 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:

$$(\bar{\mathbf{C}} - \lambda\bar{\mathbf{M}})\bar{\mathbf{y}} = 0 \tag{1.4}$$

Transformed set of modes is formed based on the equation:

$$\bar{\mathbf{H}} = \mathbf{H}\bar{\mathbf{Y}} \tag{1.5}$$

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 the exclusion of modes that correspond to movement of the flexible subsystem as a rigid body. It is necessary since the movement of the flexible subsystem as a rigid one is defined by *floating frame of reference* CS1. Zero eigenvalues correspond to rigid body modes of flexible subsystem (1.4).

1.1.2. Calculation of stresses and strains

Let us consider the discrete expressions of elasticity theory used in the finite elements method:

$$\begin{gathered} \varepsilon_i^e = \mathbf{B}_i^e(\mathbf{x}_i^e)\mathbf{u}_i^e, \\ \boldsymbol{\sigma}_i^e = \mathbf{D}_i^e\varepsilon_i^e = \mathbf{D}_i^e\mathbf{B}_i^e\mathbf{u}_i^e, \end{gathered} \tag{1.6}$$

where $\boldsymbol{u}_i^e, \varepsilon_i^e, \boldsymbol{\sigma}_i^e$ are matrix-columns of nodal degrees of freedom of strains and stresses of *i-th* finite element, $\boldsymbol{B}_i^e$ is matrix expressing strain field of the finite element with the nodal displacement, $\boldsymbol{D}_i^e$ is elasticity matrix of the finite element which is generated according to Hooke's law, $\boldsymbol{x}_i^e$ is matrix-column of coordinates of finite elements nodes. Sizes of the matrices depend on finite element type.

If nodal displacements are represented as the sum (1.2), strains and stresses of a finite element can be represented by following expressions:

$$\begin{aligned} \varepsilon_i^e &= \mathbf{B}_i^e(\mathbf{x}_i^e)\mathbf{H}_i^e\mathbf{w} = \mathbf{B}_i^e(\mathbf{x}_i^e)\sum_{j=1}^{H}\mathbf{h}_{ji}^e w_j = \sum_{j=1}^{H}\mathbf{h}_{ji}^{e\varepsilon} w_j = \mathbf{H}_i^{e\varepsilon}\mathbf{w}, \\ \sigma_i^e &= \mathbf{D}_i^e\mathbf{B}_i^e(\mathbf{x}_i^e)\mathbf{H}_i^e\mathbf{w} = \mathbf{D}_i^e\mathbf{B}_i^e(\mathbf{x}_i^e)\sum_{j=1}^{H}\mathbf{h}_{ji}^e w_j = \sum_{j=1}^{H}\mathbf{h}_{ji}^{e\sigma} w_j = \mathbf{H}_i^{e\sigma}\mathbf{w}, \end{aligned} \tag{1.7}$$

where $\boldsymbol{h}_{ji}^e$ is the part of j-th mode which corresponds to nodal degrees of freedom of *i-th* finite element. Matrices-columns $\boldsymbol{h}_{ji}^{ez}$ and $\boldsymbol{h}_{ji}^{e\sigma}$ represent stresses and strains from nodal displacements of the finite element which are correspond to the mode $\boldsymbol{h}_{ji}^e$ when value of the modal coordinate w_j=1. These matrices-columns are called *element solutions*.

So far as $\boldsymbol{D}_i^e, \boldsymbol{B}_i^e, (\boldsymbol{x}_i^e)$ are constant matrix, they are not used for simulation after calculation of $\boldsymbol{h}_{ji}^{ez}$ and $\boldsymbol{h}_{ji}^{e\sigma}$. Therefore, stresses and/or strains can be calculated during integration of equations of motion of flexible body if stresses and/or strains modal matrices are calculated correspond to the expressions (1.7). Matrices-columns $\boldsymbol{h}_{ji}^{ez}$ and $\boldsymbol{h}_{ji}^{e\sigma}$ corresponded to the mode $\boldsymbol{h}_j$ of a flexible body are calculated by FEA software. Before using in UM software, they are transformed similarly to the matrices-columns $\boldsymbol{h}_j$ based on the expressions (1.4), (1.5).

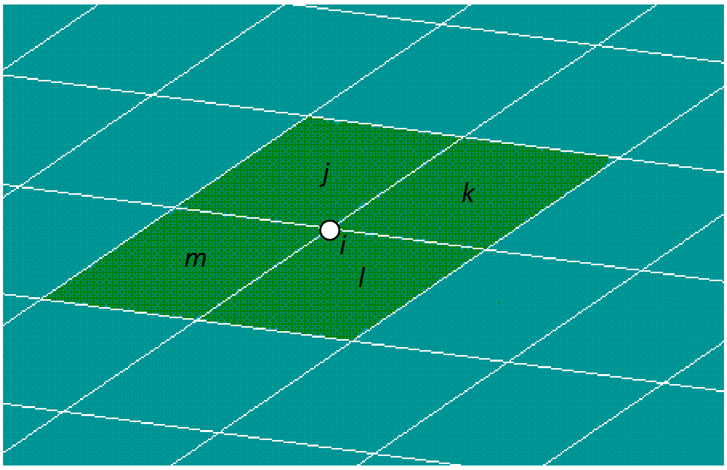


Figure 1.2. To example of calculation of nodal stresses

Nodal stresses or strains are calculated by FEA programs based on values which are calculated for elements including the concerned node. The simple averaging of values is often used. For example, if the node with index *i* is belonged to the four finite elements with the indices *j, k, l, m* (Figure 1.2), then nodal stress are calculated as

$$\sigma_i^n = \frac{\left(\sigma_{ji}^e + \sigma_{ki}^e + \sigma_{li}^e + \sigma_{mi}^e\right)}{4} = \frac{\sum_{b \in M_i} \sigma_{bi}^e}{N_i},$$

where σ_i^n is nodal stress, σ_{ji}^e is the stress components in the node i of the finite element with the index j, M_i is the set of indices of the finite elements including the node i, N_i is the count of finite elements including the node i.

UM imports solutions for elements. The nodal solutions are calculated as average values in the elements containing the node.

1.2. Installation, preparing data, workflow

UM FEM installation package includes the following items:

- software for data import from **ANSYS**:
  - macro file **um.mac** for ANSYS, which is written in APDL (*ANSYS Parametric Design Language*);
  - stand alone program for data transformation **ansys_um.exe**;
- software for data import from **MSC.NASTRAN**:
  - file **umfum.alt** with procedures which are written in DMAP language *(Direct Matrix Abstraction Program)*;
  - stand alone program for data transformation **nastran_um.exe**;
- software for data import from **ABAQUS** - **abaqus_um.exe**;
- **wizard of flexible subsystems** built in **UM Input** program;
- software procedures for handling and simulation of dynamics of flexible bodies that are built in **UM Input** and **UM Simulation** programs.

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.

Every step is considered in the next items. Data preparing in **ANSYS** is described in Sect. 1.2.1. "Exporting finite element model from ANSYS" p. 1-9, Sect. 1.2.2. "Exporting finite element model from MSC.NASTRAN", p. 1-26 is devoted to work in **MSC.NASTRAN**, Sect. 1.2.3. "Exporting finite element model from NX NASTRAN", p. 1-38 is devoted to **NX NASTRAN** and Sect. 1.2.4. "Model creation in ABAQUS environment and data exchange", p. 1-49 is devoted to **ABAQUS.**

1.2.1. Exporting finite element model from ANSYS

1.2.1.1. General information

The whole workflow of the preparing input data for models that include flexible bodies is shown in Figure 1.3. Let us consider basic steps of this procedure.

The first step is executed under **ANSYS** environment. According to the 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. The macro has commands for calculation of eigenmodes and static modes, as well as calculation and export of 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 that 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. For example, if *Beam* is the task name then files *Beam.rst*, *Beam.full* and *Beam.free* will be created in the working directory after calculations.

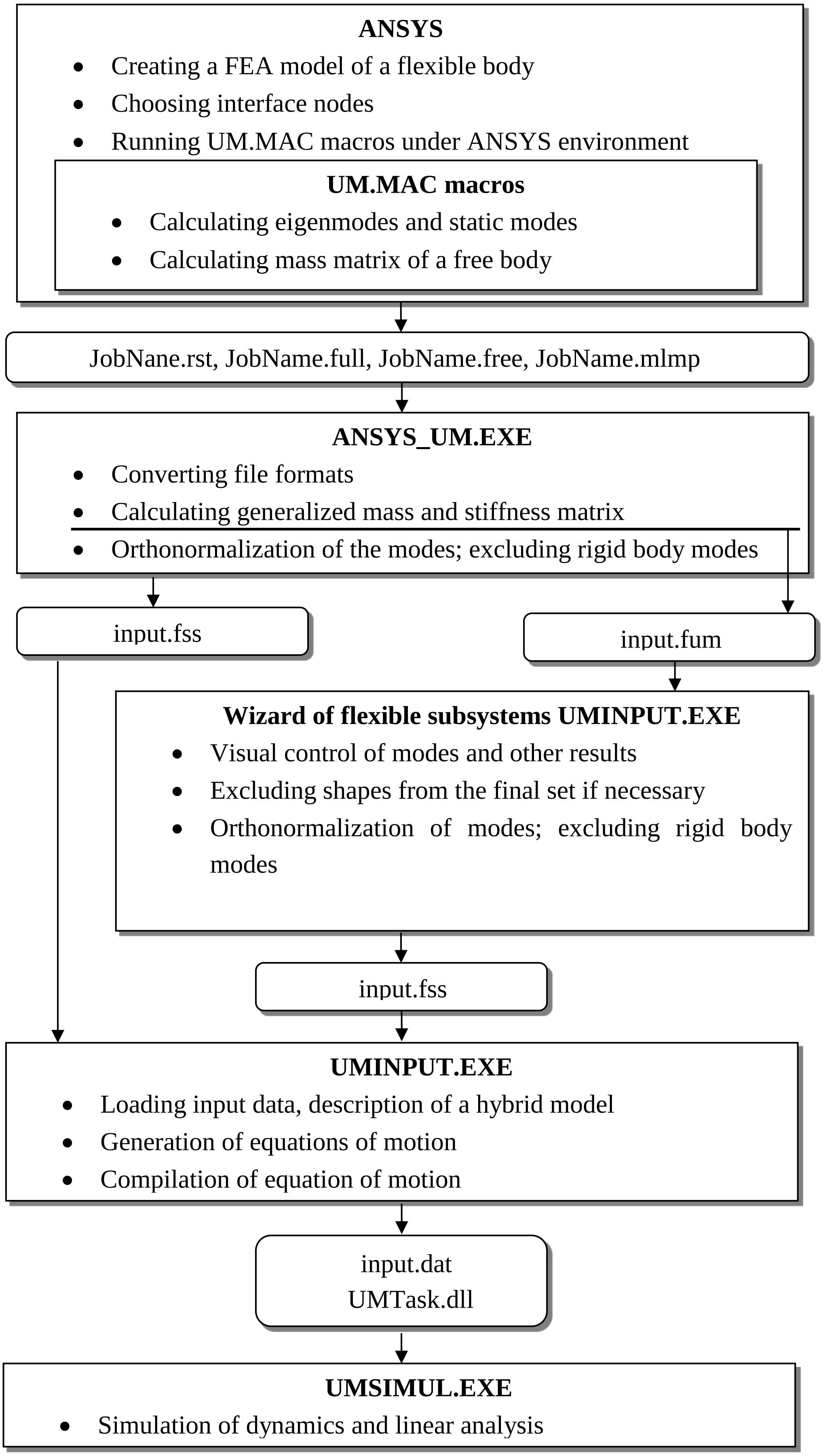


Figure 1.3. Data preparing workflow

After the installation of UM, 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:\um\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 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.

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 key points there (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,StressInclude,StrainInclude

NEForms is required number of eigenmodes correspond to the lowest eigenvalues of the flexible body;

WayM is a way of forming mass matrix:

0 means mass matrix, based on shape functions of finite elements;

1 means diagonal mass matrix.

StressInclude calculate stresses corresponding to modes of flexible subsystem

1 means 'yes'

0 means 'no'

StrainInclude calculate strains corresponding to modes of flexible subsystem

1 means 'yes'

0 means 'no'

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 appear 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.

1.2.1.2. Creating stress and strain sensors

The actions needed for creating sensors of stress and strains are similar. Therefore only stress sensors making is described in this item.

According to FE approach stresses are calculated in the nodes of finite elements. Nodal stress is average value calculating from all finite elements including given node. Just nodal stresses are usually interesting for constructions analysis. In order to calculate nodal stress in **UM**, data file input.fum should contain matrices-columns $\boldsymbol{h}_{ji}^{e\sigma}$ at least for one finite element including this node (see Sect. 1.1.2. *"Calculation of stresses and strains"*, p. 1-6). The nodes are called *sensors* if data for stresses calculation in these nodes is present.

Preparing of data which are needed for stresses calculation in **UM** can require a lot of CPU resources. For some models with greater number of nodes and finite elements, this calculation can be impossible if user tries to prepare data for all of nodes. Size of file which available for reading in **ANSYS_UM** and **UM Input** programs is limited by value of 2 Gb. At the same time, a researcher can interest in stresses only in some nodes which number is not great.

Let us consider two ways which allow calculating data only for selected nodes and thereby reduce computer resources requirement.

1.2.1.2.1. Choice of sensors in ANSYS

First way allows specifying a set of nodes for which stresses will be calculated and saved to result file JobName.rst by **ANSYS**. Before **um.mac** launch, the component ESTRS should be created using CM command.

Example. Create component for stresses or strains calculation in the nodes 17, 24, 138, 1235. The following sequence of APDL command is used.

```
NSEL,s,,,17         !create a new set of nodes which consists
                    !of one node with the number 17
NSEL,a,,,24         !add node 24 to the set
NSEL,a,,,138        !add node 138 to the set
NSEL,a,,,1235       !add node 1235 to the set
ESLN,s,0,all        !select finite elements which include
                    !selected nodes
CM,ESTRS,ELEM       !create component called ESTRS which
                    !consists of selected finite elements
ESEL,all            !select all finite elements
```

After execution **um.mac**, the result file will be contained data for stresses calculation in the nodes of elements belonging to ESTRS component only. Thus, size of file JobName.rst can be reduced and computation time can be shortened as compared with the full model. However note that the calculation in **ANSYS** is carried out once at the stage of data preparing and size of file JobName.rst is critical only if it exceed 2 Gb or under the deficit of disc space. Otherwise the second way can be used (see the next item).

Note.	Under the execution of **um.mac**, the component ESTRS including a set of selected finite elements is created. Data for the rest elements is not computed and not

included in the file input.fum. The last line in the example selects all elements. If model of flexible body consisted of not all elements which are made in **ANSYS**, this line should be edited. But making ESTRS component should be always precede selection of a set of finite element which are included in the file input.fum.

1.2.1.2.2. Choice of sensors in ANSYS_UM

The user can create the list of nodes-sensors. Data for them is selected by **ANSYS_UM** from file JobName.rst and included in the file input.fum (input.fss). The list of the sensors is described in the text file umsensors.lst which should be in the working directory of the task. The file has the following structure. The first line contains the comment starting with symbol «$»:

$ UM SENSORS NODES LIST

The second and following lines include numbers of the sensors. One number is per one line. The file can include comments which starting with symbol «$». It must be in separate lines. For example,

$ Second section

365

464

The Second line must contain one word «ALL» if all nodes must be selected.

UM includes the macro umsensors.mac which helps to create file *umsensors.lst*. Copy umsensors.mac to the **ANSYS** directory which contains macro files by default (for example, .\apdl for version 9 – 11). The command line is as follows:

umsensors,newfile,startnum,finishnum,stepnum

Newfile	0 means ‘create new list of nodes’; 1 means ‘add to existing list’.
startnum	is a starting number of node set., -1 includes all nodes of a finite element mesh to the set, other parameters are ignored, the list created earlier is deleted.
finishnum	is the finish number of the node set.
Stepnum	is a step of node number increment, 1 by default; The one number startnum is added to the set if parameters finishnum and stepnum are absent.

All nodes in which stresses (strains) can be calculated are selected if file *umsensors.lst* is absent in the task directory. If there is no data in the file JobName.rst for stresses calculation in elements which contain node N, inclusion of this number in *umsensors.lst* does not have effect. That is, file input.fum contains data for stresses calculation in the nodes which belong to intersection of sets ESTRS and *umsensors.lst*. The content of input.fum depending on prior actions when parameter *StressInclude* of **um.mac** is equal to 1 is presented in the Table 1.1.

Table 1.1

№	Component ESTRS is created	JobName.rst	umsensors.lst	input.fum (input.fss)
1.	no	all nodes	file is absent or all nodes are selected	all nodes
2.	no	all nodes	Umsensors	umsensors
3.	yes	ESTRS	file is absent or all nodes are selected	ESTRS
4.	yes	ESTRS	Umsensors	ESTRS ∩ umsensors

Simulation efficiency in **UM** directly depends on size of file input.fss. That is appreciably at multivariant calculations (see Sect. 6.2. «Scanning»). Therefore, it is recommended to prepare previously the file *umsensors.lst* if calculating of stresses in the every node of a model is not needed.

1.2.1.3. 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** macro and saves this data in UM format.

Input.fss file includes a 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.

Ansys_um includes the **General** and **Options** tabs, see Figure 1.4.

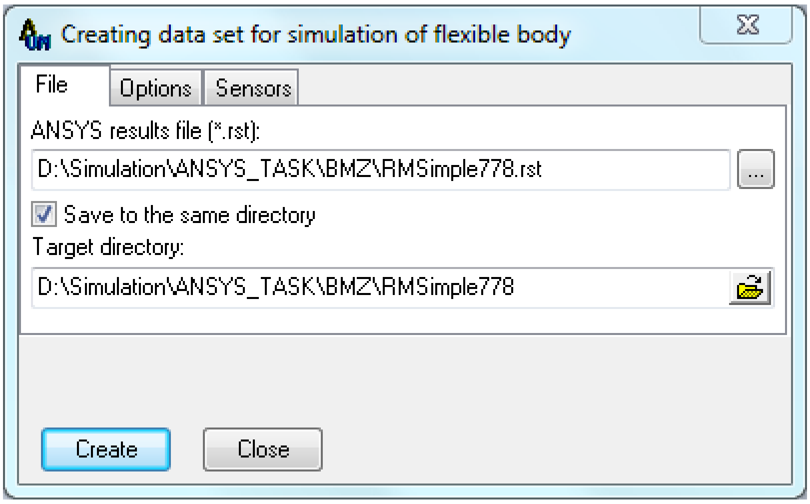


Figure 1.4.

The **General** tab (Figure 1.4) lets the user select a*.*rst* file and sets the target directory for saving *input.fum / input.fss* files. It is recommended to create this target directory in advance.

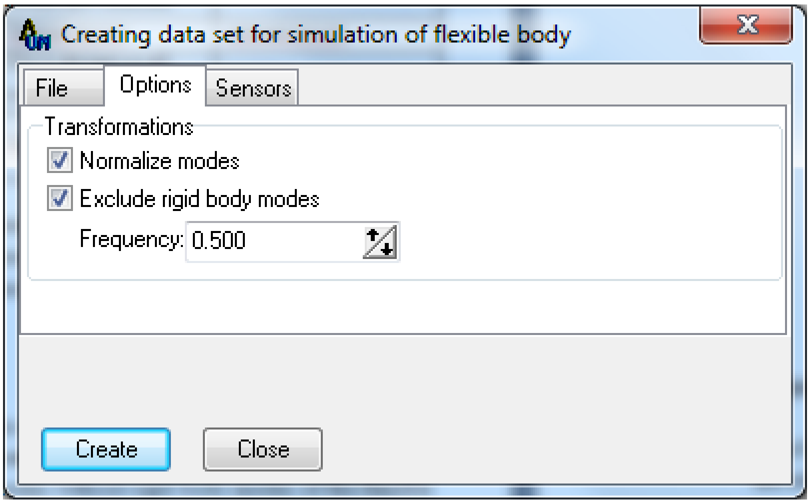


Figure 1.5.

The **Options** tab (Figure 1.5) defines structure of output files. The 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 the **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 leads 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 off **normalize modes** and then use the **Wizard of flexible subsystems**.

The **Sensors** tab includes control items for writing of data for calculation of stresses and strains. View of right part of the tab is defined by the way of choose of sensors. The left part are invariable.

If the directory of a task includes file umsensors.lst (see 1.3.2.2. *"Solution tab"*, p. 1-86), the tab looks like in Figure 1.6.

The check boxes of the **Include solutions for elements** group allow to include/exclude data for stresses and strains calculation to file input.fum. If the **delete list after transformation** check box is turned on then file *umsensors.lst* is deleted after creating *input.fum*. In most cases, it is recommended turn off this checkbox. Thus sensors list can be used many times.

Set of sensors can be edited via flags in the left part of each element of the list. The popup menu can be used for that (Figure 1.7).

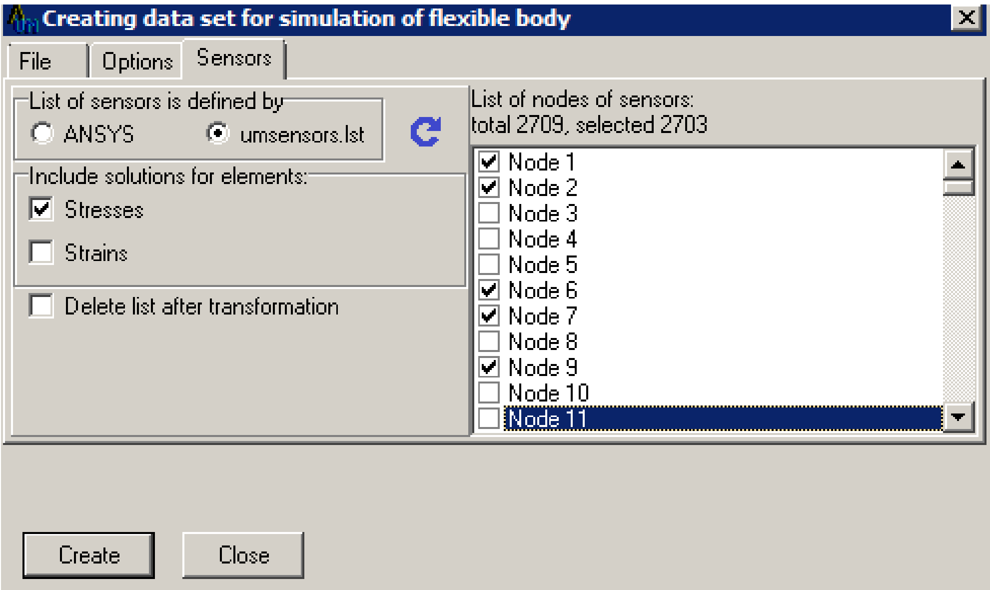


Figure 1.6. View of the Sensors tab under choose of nodes-sensors from file *umsensors.lst.*

Figure 1.7.

File *umsensors.lst* can be created, deleted or edited after run of **ansys_um**. Information on the form is refreshed by button  or by item **Refresh** of the popup menu.

If file *umsensors.lst* is absent in the task directory or read error occurred, the message presented in Figure 1.8 appears on the screen and no sensors are output to *input.fum*. The view of form after choosing of sensors in **ANSYS** is presented on Figure 1.9.

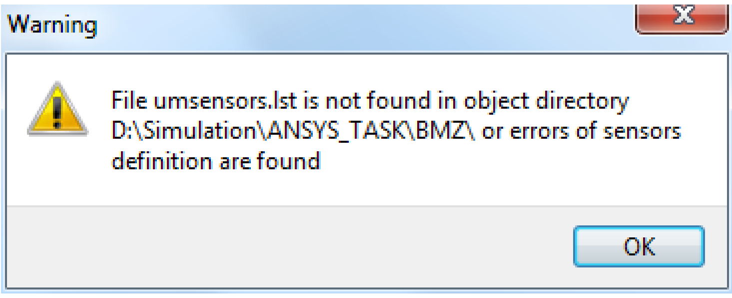


Figure 1.8.

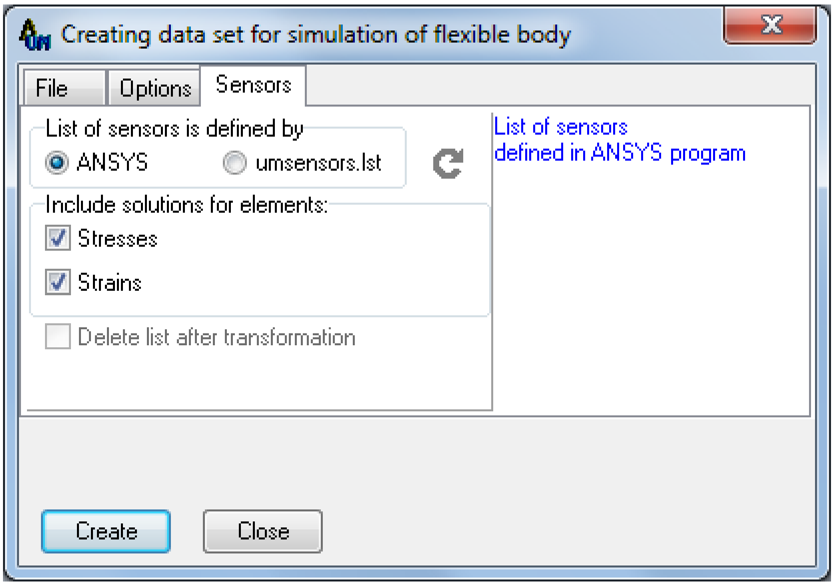


Figure 1.9. Sensors tab

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

1.2.1.4. Features of data import from ANSYS Workbench

ANSYS Workbench is a universal finite-element package, suitable for various classes of tasks (strength, thermal physics, fluid and gases dynamics, electromagnetism). This chapter explains how to prepare data of the mechanical FE models for dynamic studies in the UM software.

Let us note the most important points which are significant when importing flexible model from **ANSYS Workbench** software to UM software. To prepare the data the macro **um.mac**, which must be located in the directory **.\apdl** of the **ANSYS Workbench** environment, is used.

Note. SI system must be used, see Figure 1.10.

To import data in the UM software do the following steps.

The first step. Modal analysis should be set. To do this, choose **Modal** in the **Toolbox** taskbar by double-clicking the left mouse button. As a result the outline of a project of modal analysis will appear, see Figure 1.11.

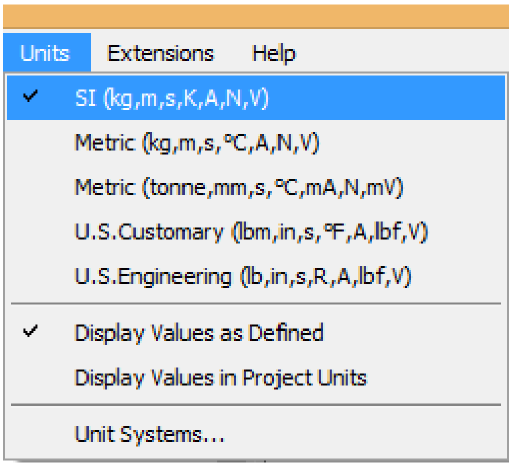


Figure 1.10. SI system

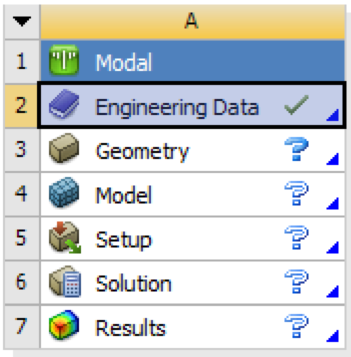


Figure 1.11. Project outline for modal analysis

In the **Design Modeler** module a geometrical rigid model of an object is built. Double-click the left mouse button on the **Geometry** field of the task project to call the module.

The second step. Creating a FE mesh in the **Mechanical** module, which can be activated by double-clicking on the left mouse button on the **Model** field, see Figure 1.12.

The third step. Adding commands in **APDL** language. According to the basic method of the flexible bodies dynamics modeling, a FE model should be described as a superelement, stress and strain sensors have to be chosen. The user is offered to perform all this actions in the **Workbench** environment with the help of the **APDL** commands.

Creation of rigid regions and description of a superelement, creation of stress and strain sensors, as well as the call of the **UM.MAC** macro are made in the **Commands** field in the element tree. The **Commands** field is added with the right mouse button **Modal** –> **Insert** –> **Commands**. In the **COMMANDS** field commands in the **APDL** language are set.

APDL language commands, used to prepare the import in UM are described in detail in Sect. 1.2.1.1. *"General information"*, p. 1-9 and Sect. 1.2.1.2.1. *"Choice of sensors in ANSYS"*, p. 1-14.

In Figure 1.13 you can see a window with commands described above without rigid regions for the platform.

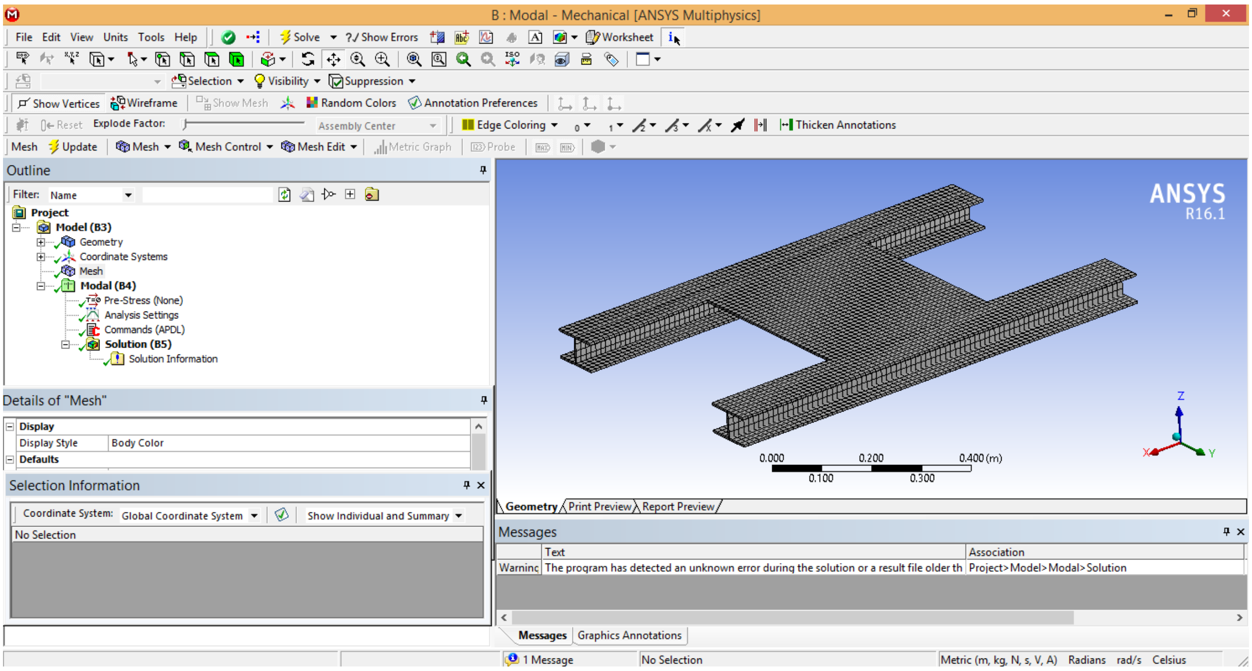


Figure 1.12. An example of a finite-element model of a platform in the **Mechanical** module

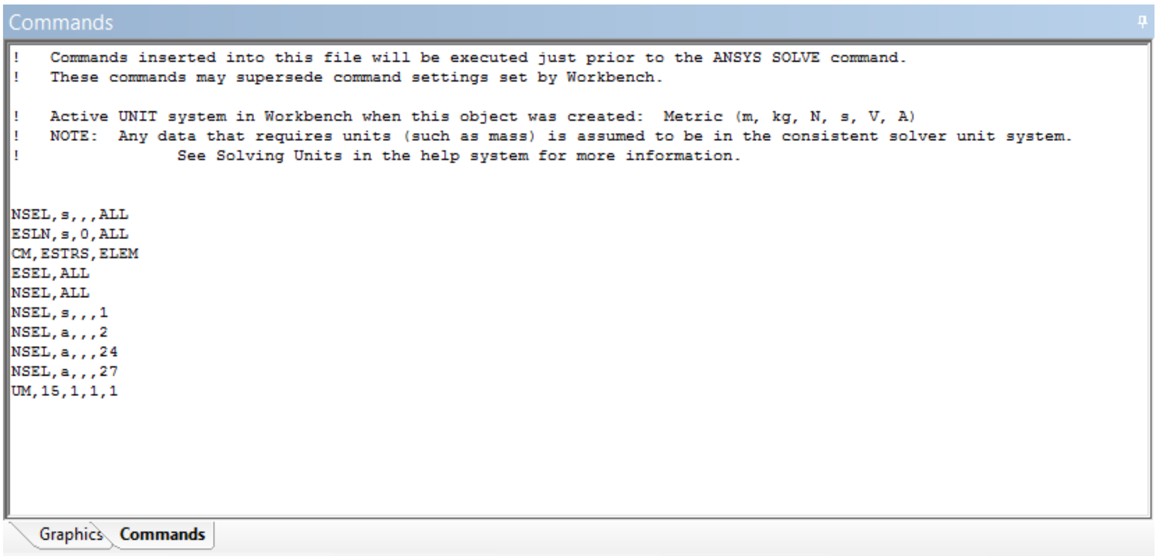


Figure 1.13. The **Commands** window with commands in **APDL**

In this example a set of strain and stress sensors which includes all nodes and elements of the FE model is prepared. Then the interface nodes with the numbers 1, 2, 24 and 27 are chosen and UM macro to calculate the first 15 eigenmodes, stresses and strains is run.

In some cases, for correct simulation results it is required to create **rigid regions** around the interface nodes.

The creation of rigid regions is done either interactively or with the help of the following **APDL** commands:

CERIG, MASTE, SLAVE, Ldof, Ldof2 ... Ldof5, where

CERIG is the command to create a rigid connection.

MASTE is the main node number,

SLAVE is the dependent node number,

Ldof, Ldof2 ... Ldof5 are the degrees of freedom, for which restrictions are imposed.

Note. Degrees of freedom with rigid regions must be valid for the used types of finite elements.

An interactive way to create a rigid region is connected with the creation of the **Point Mass** (mass point) and **Remote Points** elements. To create a **Point Mass** element in the element tree field choose **Geometry –> Insert –> Point Mass**. In the appeared window set in the **Geometry** field the position in which the mass point will be created, and in the **Mass** field, set up the **Point Mass** value (Figure 1.14).

Scope	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Geometry	No Selection
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change
Definition	
Mass	0. kg
Mass Moment of Inertia X	0. kg·m²
Mass Moment of Inertia Y	0. kg·m²
Mass Moment of Inertia Z	0. kg·m²
Suppressed	No
Behavior	Deformable
Pinball Region	All

Figure 1.14. Fields necessary to describe the **Point Mass** element

Right-click to choose the formed object **Point Mass->Promote to Remote Point**, see Figure 1.15.

In the appeared settings window of a created object **Point Mass – Remote Point** (Figure 1.16), in the **Geometry** field select the nodes that are included in the rigid region, in the **Location** field indicate the node with the main degrees of freedom (an interface node). In the **Behavior** field select **Rigid**, in the **DOF Selection** field set **Manual**. Set **Active** values to the fields of limited degrees of freedom **(X Component, Y Component, Z Component, Rotation X, Rotation Y, Rotation Z)**.

The fourth step. Right-click on the **Modal –> Solve** field of the element tree for the implementation of the solution, Figure 1.17. The results of the solution are in the folder of a working project **PROJECT_NAME\dp0\SYS\MECH**.

The fifth step. **Ansys_um.exe** program is run. How to work in it you can find in Sect. 1.2.1.3. *"ANSYS-UM data exchange"*, p. 1-16.

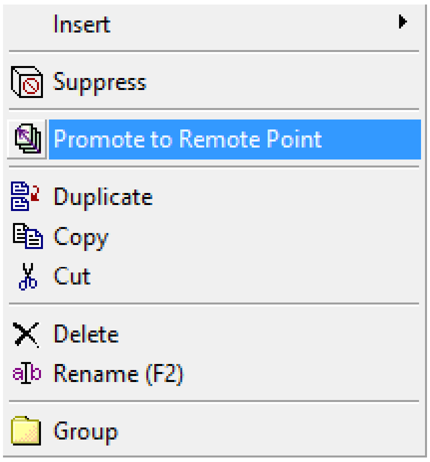


Figure 1.15.

Scope	
Scoping Method	Geometry Selection
Geometry	No Selection
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	10. m
Z Coordinate	1.e-002 m
Location	Click to Change
Definition	
Suppressed	No
Behavior	Rigid
Pinball Region	All
DOF Selection	Manual
X Component	Active
Y Component	Active
Z Component	Active
Rotation X	Active
Rotation Y	Active
Rotation Z	Active

Figure 1.16. Fields necessary to describe an object **Point Mass – Remote Point**

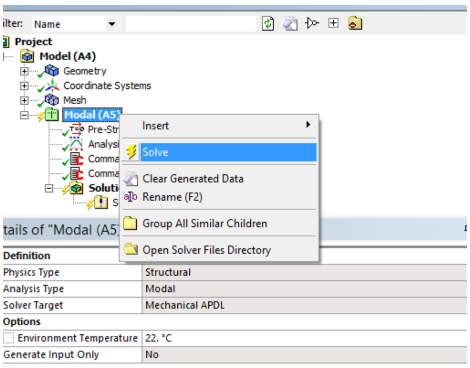


Figure 1.17. Starting solution

Consider the **case when the interface node is not located in the model area, but at some distance from it**. For example, Fig.

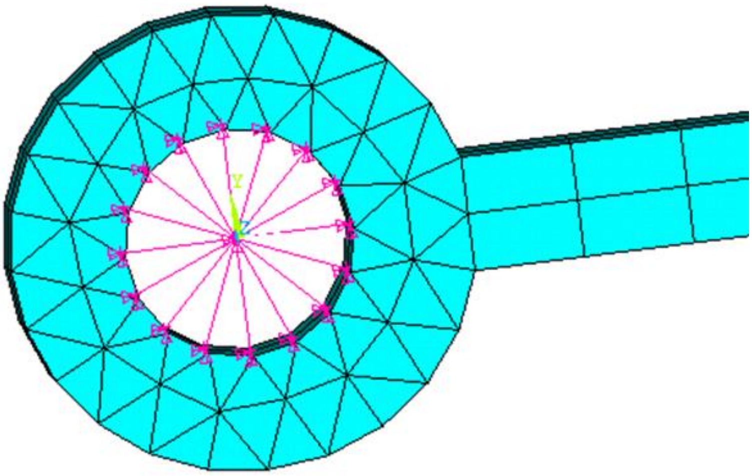


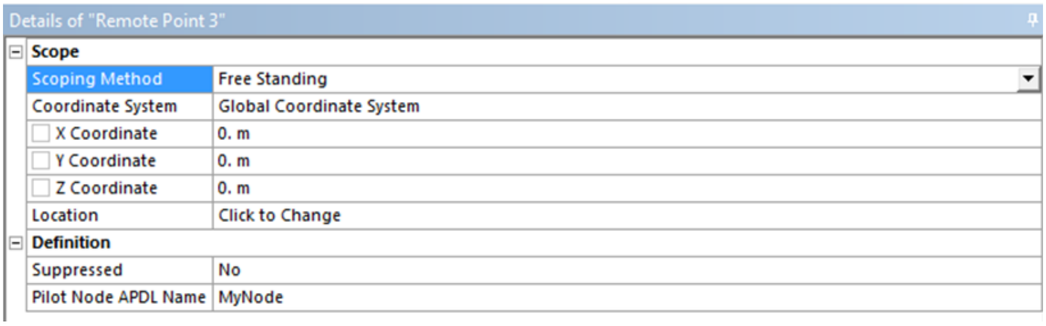
Figure 1.18. Constraint equations at the circular hole

In this example, you need create an interface node in the cylindrical hole of the model, place the mass element there, and create a rigid region around the interface node by linking it to the other nodes of the model.

1. Create a node at a point that does not belong to the model area.

Remote Point–> Insert–> Remote Point

In field **Scoping Method** set **Free Standing**. Enter coordinates X, Y и Z, and set the name of node at point in field **Pilot Node APDL Name** (in our case the name is **MyNode**).



Details of "Remote Point 3"

Scope	
Scoping Method	Free Standing
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change
Definition	
Suppressed	No
Pilot Node APDL Name	MyNode

2. Add a point mass**. Geometry –> Insert –> Point Mass**.

Set **Scoping Method–>Remote Point**. In field **Remote Point** select created **Remote point**. Enter mass and moment of inertia values.

Scope	
Scoping Method	Remote Point
Applied By	Remote Attachment
Remote Points	Remote Point
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Definition	
Mass	1.e-004 kg
Mass Moment of Inertia X	1.e-003 kg·m^2
Mass Moment of Inertia Y	1.e-003 kg·m^2
Mass Moment of Inertia Z	1.e-003 kg·m^2
Suppressed	No

3. In our case rigid regions are convenient to do by APDL commands in field Commands. Modal–>Insert–>Commands.

For example,

CERIG, MyNode, SLAVE, Ldof, Ldof2 ... Ldof5, where

CERIG is the command to create a rigid connection.

MyNode is the main node name,

SLAVE is the dependent node number,

Ldof, Ldof2 ... Ldof5 are the degrees of freedom, for which restrictions are imposed.

1.2.2. Exporting finite element model from MSC.NASTRAN

1.2.2.1. General information

Analysis of a finite element model is realized in **MSC.NASTRAN** by the means of procedures which are written on MSC.NASTRAN DMAP (Direct Matrix Abstraction Program) language. DMAP is a high-level language including compiler.

For solution of typical tasks, **MSC.NASTRAN** gives sets of procedures called *solution sequences* in the user guide of **NASTRAN**. For example, linear and nonlinear static analysis, modal analysis are typical tasks of **NASTRAN**. Type of analysis is selected via SOL operator. Predefined number of sequence is parameter of *SOL* operator.

For example,

SOL 101 is linear static analysis,

SOL 103 is modal analysis.

MSC.NASTRAN allows changing this sequences or writing new sequences using DMAP. Predefined operator sequences can be modified via ALTER operator which adds or deletes operators from standard procedures of **NASTRAN**. This opportunity of DMAP was used for development of procedures which import data to **Universal mechanism** software.

A flexible subsystem is created based on the superelement method. After description of a finite element model, a user should select interface nodes and create a superelement. Necessary data is imported during modal analysis of the superelement.

Rules of preparing data **MSC.NASTRAN** as well as sequence of using software for data import to **UM** are considered bellow step by step. The description of development and analysis of a model including a flexible subsystem imported from **MSC.NASTRAN** is contained in the guide *«Getting started: flexible bodies using UM FEM»*.

MSC.NASTRAN does not have visual development environment. As a rule, the program **MSC.PATRAN** is used for description of finite element model which is analyzed by **MSC.NASTRAN**. Therefore, becoming operations in **MSC.PATRAN** will be considered bellow under the description of necessary actions of user. Input file of **MSC.NASTRAN** is created by **MSC.PATRAN** automatically during analysis of a model. Screen copies with control elements of **MSC.PATRAN 2005** are presented bellow in the item Sect. 1.2.2.3. *"ANSYS-UM data exchange"*, p. 1-16. Dialog windows of others version of the program can be different.

1.2.2.2. Software modules and workflow

UM allows following software for data import from **MSC.NASTRAN**.

1. Module umfumYYYY.alt developed on DMAP programming language saves data to intermediate files geoms.op2 and matrix.op4 in DMAP format. YYYY in the module name is the version number of MSC.NASTRAN. For example, umfum2005.alt is intended for data import from MSC.NASTRAN 2005.

File geoms.op2 contains a finite element model (nodes, finite elements), flexible modes and data for stresses and strains calculations. File matrix.op4 includes generalized matrices of the model.

2. Program converter **NASTRAN_UM.EXE** loads files *geoms.op2 and matrix.op4* and generates *input.fum* file in **UM** format.
3. The main stages of making a flexible subsystem based on import from **MSC.NASTRAN** and its analysis in **UM** are presented on in Figure 1.19
4. Files *umfumYYYY.alt* and **NASTRAN_UM.EXE** are placed in the directory *.\bin* after installation of **Universal mechanism** software.

Note. **UM 9.0** supports importing data from **MSC.NASTRAN 2005, MSC.NASTRAN 2007, MSC MD NASTRAN 2010, MSC.NASTRAN 2012, MSC.NASTRAN 2016, MSC.NASTRAN 2019**. Importing data from other versions of **MSC.NASTRAN** was not tested.

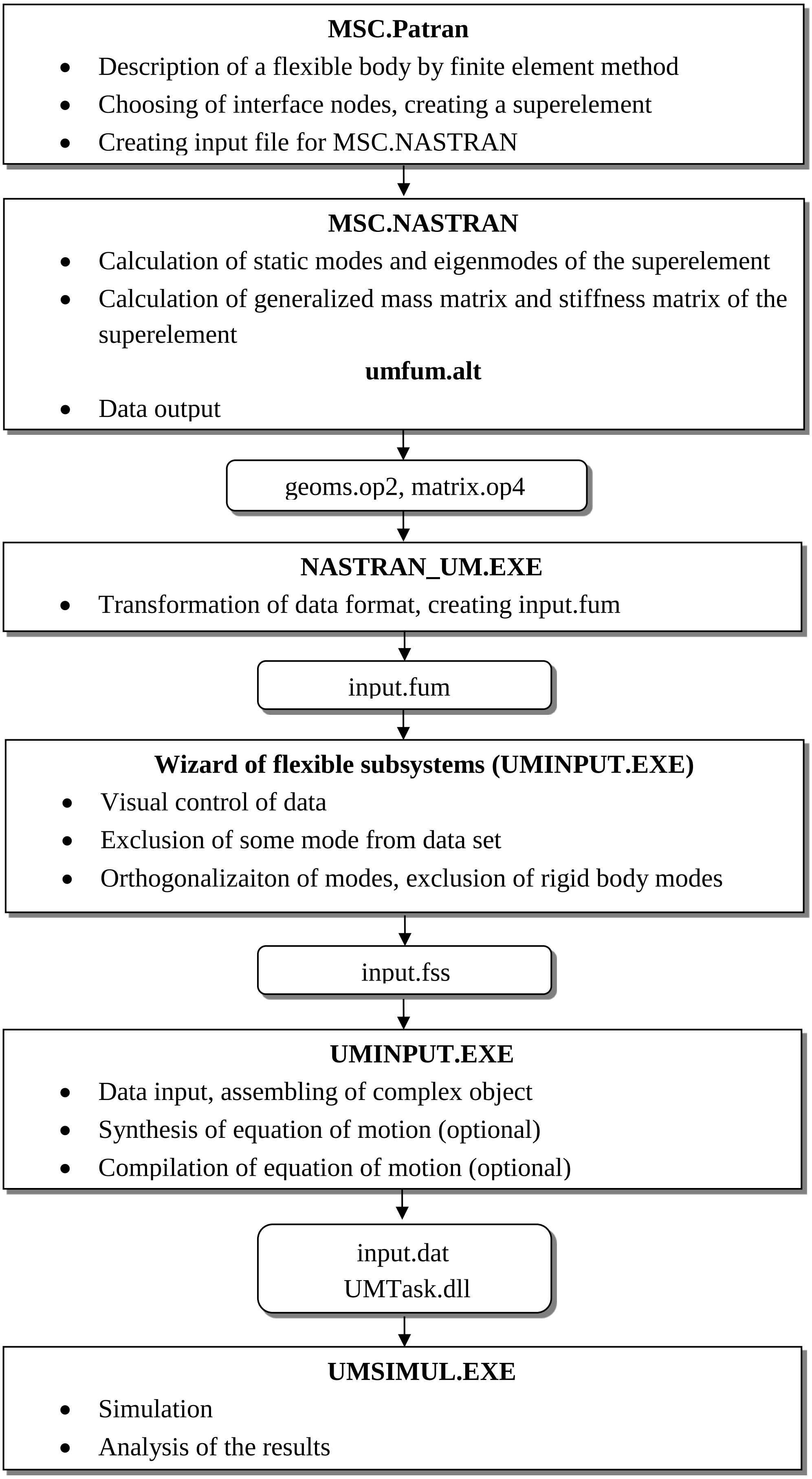


Figure 1.19. Creating of flexible subsystem using MSC.PATRAN/NASTRAN

1.2.2.3. Preparing data in MSC.PATRAN/NASTRAN environment

The main stages of creating a flexible subsystem

1. Making a finite element model of a considered object in **MSC.PATRAN**. The model should be described in the international system of units of measurement (SI). Finite element mesh should include nodes in the joint points and points of attachment of force elements to the subsystem in the complex object. Some features of preparing the finite element models are described in Sect. 1.3.1. *"Animation window"*, p. 1-83.
2. Choice of interface nodes in accordance with joint points and attachment points of force elements in the **UM** model and creating a superelement. This stage is implemented in **PATRAN** by the means of following actions.

a) Push the **Elements** button of the tools panel (Figure 1.20).

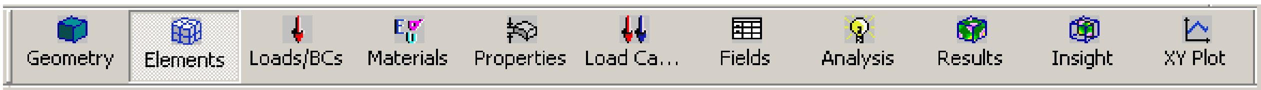


Figure 1.20. Tools panel of **MSC.PATRAN**

b) Carry out the following actions in the appeared dialog window (Figure 1.21).

- Select **Action: Create**.
- Select **Object: Superelement**.
- Enter the name of the superelement into the **Superelement Name** field.
- Push the **Select Boundary Nodes** button.

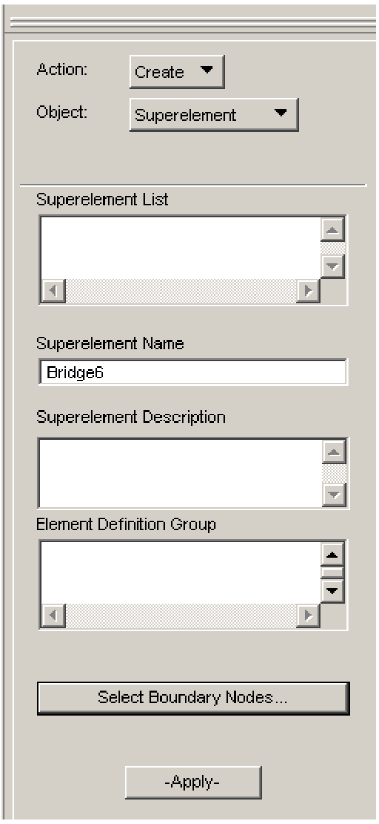


Figure 1.21. Creating of superelement in **MSC.PATRAN**

- Choose interface nodes by mouse in the window representing finite element model or input their numbers directly on the form (Figure 1.22). Push **OK** button.

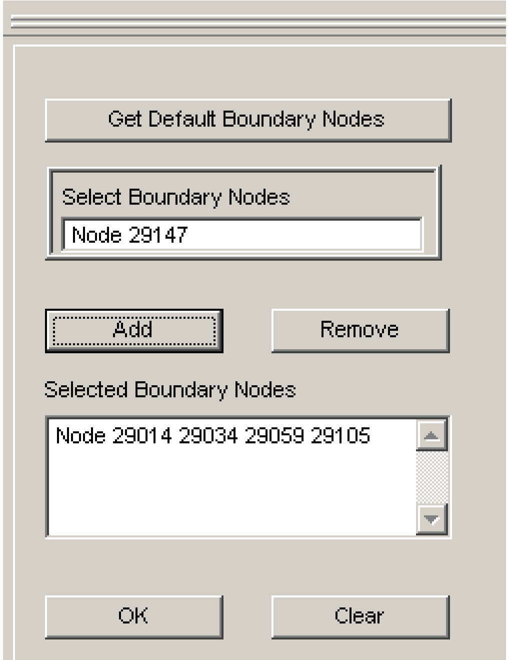


Figure 1.22. Choose of interface nodes **MSC.PATRAN**

- Push **Apply** button, Figure 1.21.

If errors are absent, the superelement with the name assigned in the **Superelement Name** field will be created.

Further, modal analysis of the superelement is executed. The operators which are included in the input file for data import as well as corresponding actions in **MSC.PATRAN** are described below.

3. Modal analysis of the superelement. Set type of analysis 103 (Normal modes) in the **Executive Control Section** of an input file **MSC.NASTRAN**.

SOL 103

In **MSC.PATRAN**, calculation parameters are defined via sequence of actions which is briefly described below.

Form calculation job.

The following actions are carried out before selection of the solution type.

- Push **Analysis** button of the tool panel.
- Execute the following actions on the appeared form on the right (Figure 1.23).
  - Choose **Action: Analyze**.
  - Choose **Object: Entire model**.
  - Choose **Method: Full Run**.
  - Assign name of the job in the field **Job Name**.

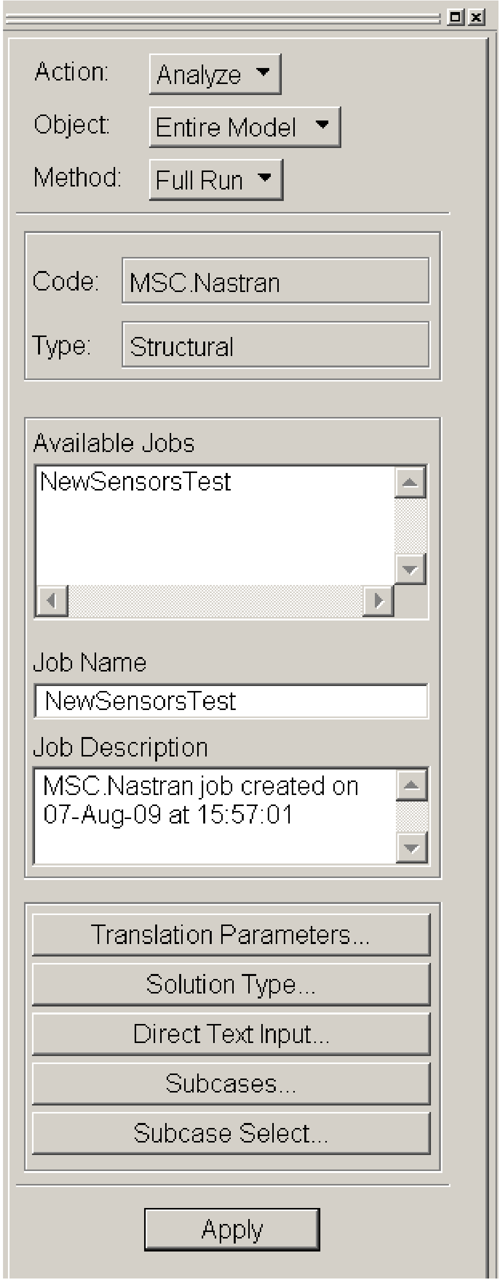


Figure 1.23. Model analysis form in MSC.PATRAN

Type of solution is selected on the form which appears after clicking the **Solution Type…** button, Figure 1.24.

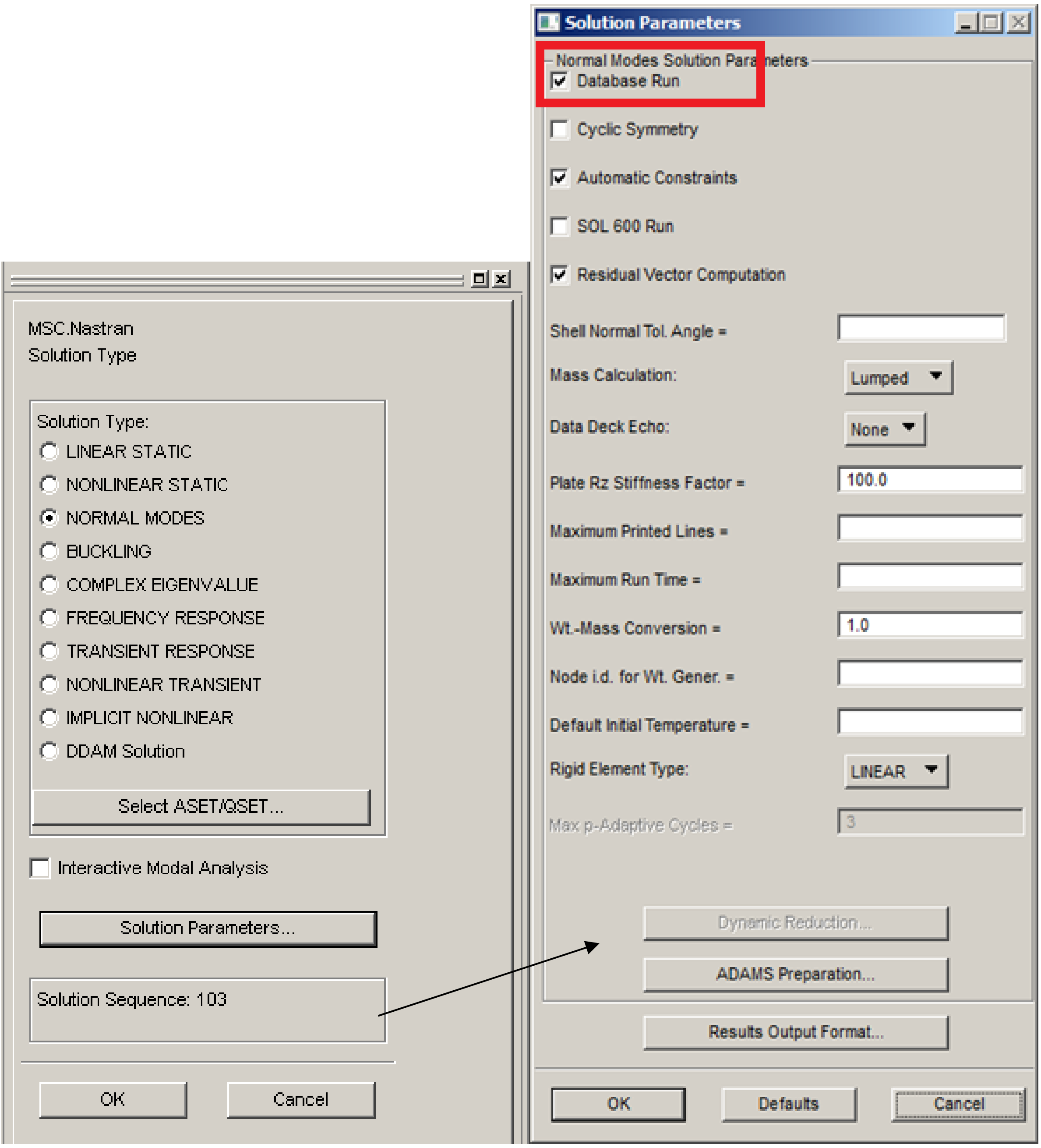


Figure 1.24. Choice of solution type in **MSC.PATRAN**

4. Assign of intermediate output files *geoms.op2* and *matrix.op4* in the **«File management»** section of an input file **.bdf* like it is shown below

ASSIGN OUTPUT2='geoms.op2' UNIT=13 FORM=UNFORMATTED

ASSIGN OUTPUT4='matrix.op4' UNIT=15 FORM=UNFORMATTED

In **MSC.PATRAN**, these lines should be written into **Analysis | Direct Text input… | File management** field.

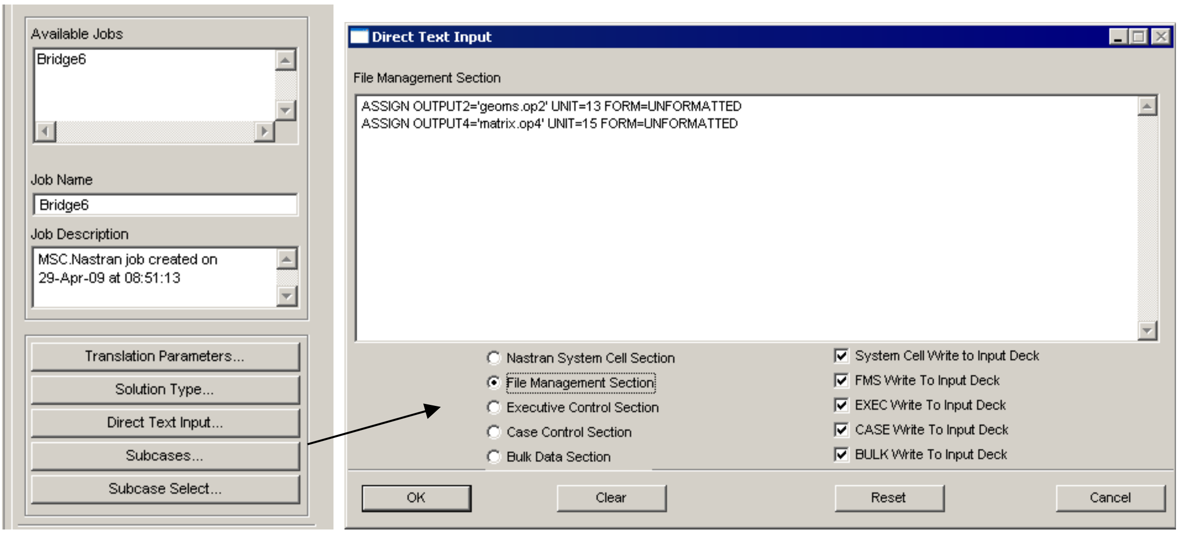


Figure 1.25. Assign of intermediate output files in MSC.PATRAN

5. Add the following line in the **Executive Control Section** in order to link up *umfumYYYY.alt* module.

include umfum2005.alt

This is example for **MSC.NASTRAN 2005**. For other versions of the program, specify the corresponding file *umfumYYYY.alt*.

In **MSC.PATRAN**, this line is written into the **Analysis | Direct Text input… | Executive Control Section** field.

File *umfumYYYY.dat* should be put to the directory which is included to list of directories for search of modules by **MSC.NASTRAN**. For example, the working directory of **MSC.PATRAN 2005** which is defined as parameter of shortcut default is subdirectory Temp of the Windows system directory. File *umfum2005.dat* can be put to this directory or to the directory *bin* of **MSC.NASTRAN**.

6. Assign SI units in the **Bulk Data Section**:

DTI, UNITS, 1, KG, NEWTON, METER, SECOND

In **MSC.PATRAN** this line is written into the **Analysis | Direct Text input… | Bulk Data Section** field.

7. Assign a number of eigenmodes of the superelement in the **Bulk Data Section**. The command lines similar to presented below are used.

SPOINT, 300001, THRU, 300030

SEQSET1,10,0, 300001,THRU,300030

Thirty eigenmodes of the superelement are required to be calculated in this example. As a rule, the eigenmodes corresponding to the lowest eigenvalues are calculated. Definition of the calculation parameters is described below in item 9.

The *SPOINT* operator defines scalar parameters called *scalar points* in the **MSC.NASTRAN** user’s guide. In this case, the scalar points are defined as modal coordinates of the superelement. They correspond to eigenmodes of the model fixed in interface nodes. Thirty scalar points are defined in the example. These points are numbered from 300001 to 300030. As a rule, numbers of the points are chosen greater than maximal number of nodes of the finite element model. That is, the numbers can be defined from 500001 to 500030. Coincidence numbers of scalar points and numbers of internal nodes of a superelement are not allowed.

SEQSET1 operator defines generalized coordinates of a superelement. In this example, the following parameters are used: parameter 10 is number of the superelement; parameter 0 defines scalar points as generalized coordinates of the superelement; the numbers 300001,THRU, 300030 are numbers of the scalar points assigned as generalized coordinates.

8. Other calculation parameters are defined with the help **Analysis | Subcases…** form.

Assign the name of a new set of parameters in the **Subcase Name** field.

9. Push **Subcase Parameters…** button and enter parameters of modal analysis into the fields of the appeared windows, Figure 1.26.

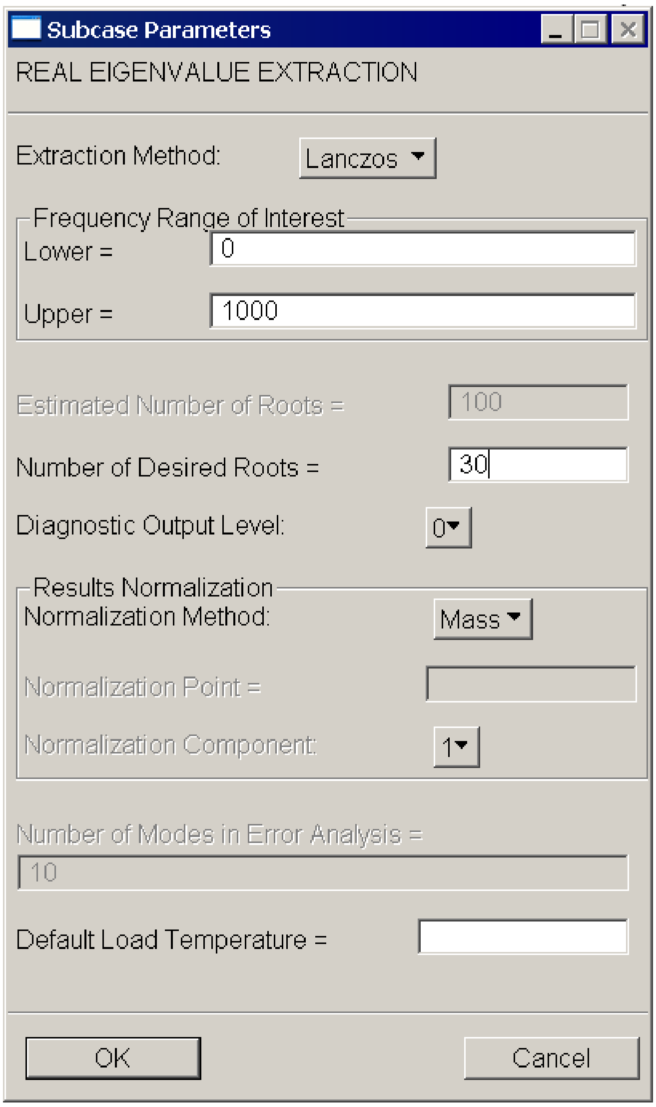


Figure 1.26. Parameters of modal analysis in **MSC.PATRAN** software.

In the presented example, it is required to calculate thirty eigenmodes corresponding to lowest eigenvalues in the range from 0 to 1000 Hertz by Lanczos method. The number of eigenmodes should be equal to the number of the scalar points defined in Step 7 by SPOINT operator. Calculated modes are normalized relatively mass matrix (M-norm). This normalization method should always be chosen under preparing data for export to **UM**.

10. Open **Output Requests…** form. One element should be in **Output Requests** list of this form:

VECTOR(SORT1,REAL)=ALL

Add this element from list **Select Result Type** if it was not added by default. Delete all other elements from the list if any.

11. In order to create stress or/and strain sensors, write the set of commands similar to following example into the **Subcase** section.

SET 501 = ALL

STRESS(CORNER)=501

SET 502 = 101,111,120 THRU 136, 170

STRAIN(CORNER) = 502

OUTPUT(POST)

SET 101 = ALL

SURFACE 11 SET 101

Let us briefly consider used commands.

SET defines a set of numbers of finite elements for which stresses or strains are calculated. 501 is the number of set. It can be chosen at will among numbers not defined earlier. Reused number as parameter SET command redefines set of finite element numbers.

STRESS calculates stresses in elements. The number of set is specified after the equality sign. The parameter CORNER in brackets is written if elements CQUAD4 are presented in the set. If this parameter is not specified, stresses are calculated only in the center of the elements CQUAD4.

STRAIN calculates strains in elements.

OUTPUT separates the commands of different types. The parameter POST starts output of stresses and strains.

SURFACE defines a surface for calculation of stresses and strains. The command is used for a set of shell elements. If calculation of stresses in solid elements is needed, the VOLUME command is used. For example,

VOLUME 11 SET 101

In this example, the set 501 includes all finite elements; the set 502 consists of finite elements 101, 111, from 120 to 136 and 170.

In order to create a sensor in a node in **UM** software, all elements which include this node should be specified in the set for STRESS or STRAIN command.

In **MSC.PATRAN**, the commands for creating of sensors are written in the field **Analysis | Subcases | Direct text input**.

12. Select superelement created in p. 2.2 in the **Available Superelements** list on the form which is called by **Select Superelements…** button (Figure 1.27). Press the **OK** button.
13. Press the **Apply** button of the **Subcases** form. In the issue, new set of parameters (subcase) for calculation of the superelement will be created. In the next time it should be selected in the **Subcase Select** form which is called on the **Analysis** form.
14. Select the created subcase.
15. Calculate the model in **MSC.NASTRAN** by the **Apply** button on the **Analysis** form. If the calculation is successful, files geoms.op2 and matrix.op4 will be created in the working di-

rectory. Diagnostics messages are outputted in the file JobName.f06. This information can be useful if there are calculation errors.

16. The job is saved in the database (*db* file) and it can be selected next time in the list of **Analysis** form.

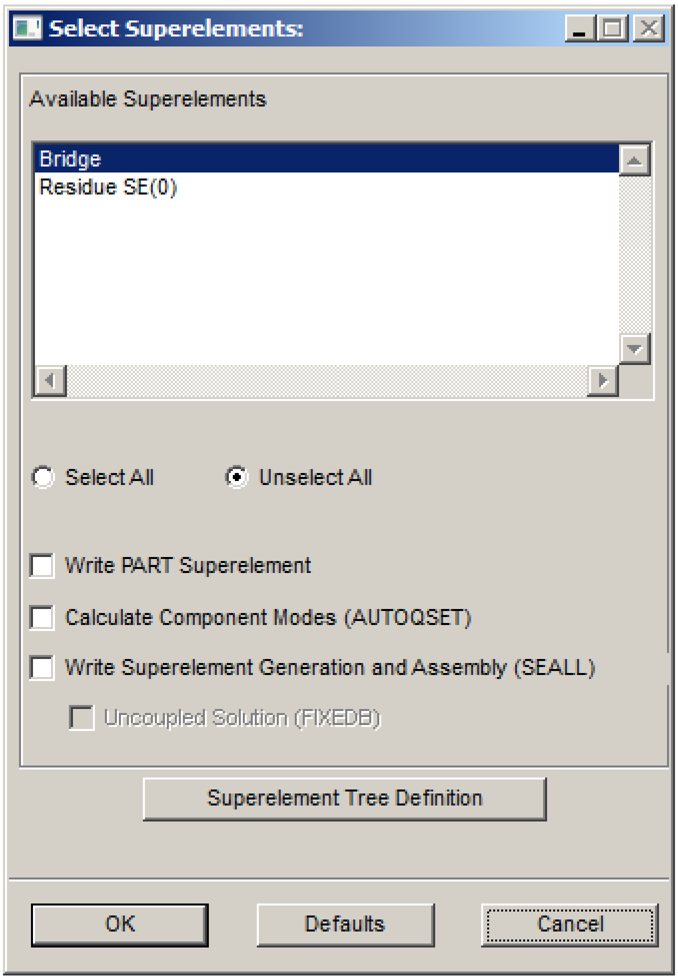


Figure 1.27. Window of **Select Superelements…**

1.2.2.4. MSC.NASTRAN to UM data exchange

Run **NASTRAN_UM.EXE** program for making input.fum file. Select geoms.op2 in the working directory of **MSC.NASTRAN** and specify target directory for input.fum. The flags **Stresses** and **Strains** enable/disable the output of corresponded data to file input.fum, Figure 1.28. This data, of course, must be calculated by **MSC.NASTRAN** using commands specified in Sect. 1.2.2.3. *"ANSYS-UM data exchange"*, p. 1-16, Step 11 of the list.

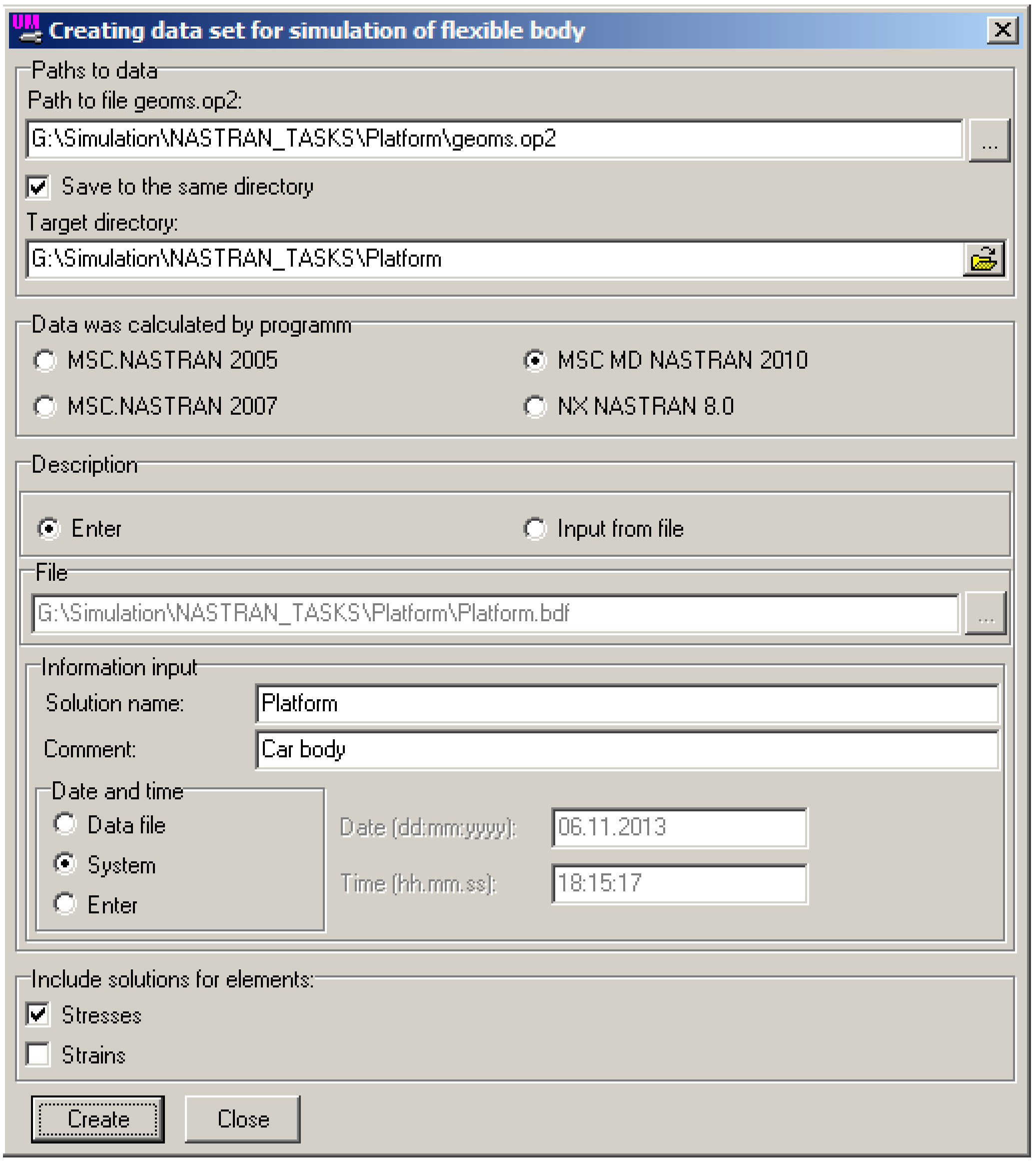


Figure 1.28. Window of NASTRAN_UM.exe program

Solution name, comment and date of calculation can be inputted into corresponded fields of the form or can be read from files *JobName.bdf* or *JobName.f06*. This data is attributes of an **UM** model which are written to file input.fum.

Perform data transformation by the **Create** button. If it is successful, file input.fum will be created in the target directory. Further work with this file is described below in Sect. 1.2.3. *"Exporting finite element model from MSC.NASTRAN"*, p. 1-26.

1.2.3. Exporting finite element model from NX NASTRAN

1.2.3.1. General information

Export of finite element models from **NX NASTRAN** is implemented using DMAP (Direct Matrix Abstraction Program) language. DMAP is a high-level language including compiler.

For solution of typical tasks **NX NASTRAN** gives sets of procedures called *solution sequences* in the user guide of **NX NASTRAN**. For example, linear and nonlinear static analysis, modal analysis are typical tasks of **NASTRAN**. Type of analysis is selected via SOL operator. Predefined number of sequence is parameter of SOL operator.

For example,

SOL 101 is linear static analysis,

SOL 103 is modal analysis.

NX NASTRAN allows changing this sequences or writing new sequences using DMAP. Predefined operator sequences can be modified via ALTER operator which adds or deletes operators from standard procedures of **NASTRAN**. This opportunity of DMAP was used for development of procedures which import data to **Universal mechanism** software.

A flexible subsystem is created based on the superelement method. After description of a finite element model, a user should select interface nodes and create a superelement. Necessary data is imported during modal analysis of the superelement.

Rules of preparing data **NX NASTRAN** as well as sequence of using software for data import to **UM** are considered bellow step by step. The description of development and analysis of a model including a flexible subsystem imported from **MSC.NASTRAN** is contained in the guide *«Getting started: flexible bodies using UM FEM»*.

NX NASTRAN does not have visual development environment. As a rule, the program **FEMAP** is used for description of finite element models for **NX NASTRAN**. Therefore below **FEMAP** commands and operations will be considered. Input file for **MSC.NASTRAN** is created by **FEMAP** automatically during analysis of a model. Screen copies with control elements of **FEMAP 10.3.0** are presented below. Dialog windows of others version of the program might be different.

1.2.3.2. Software modules and workflow

UM provides the following software for data import from **NX NASTRAN**.

1. Module umfumNX**.alt is developed in DMAP language. It saves data to intermediate files geoms.op2 and matrix.op4 in DMAP format. File geoms.op2 contains a finite element model (nodes, finite elements), flexible modes and data for stresses and strains calculations. File matrix.op4 includes generalized matrices of the model.
2. Converter **NASTRAN_UM.EXE** loads files geoms.op2 and matrix.op4 and generates input.fum file in **UM** format.
3. The main stages of preparing a flexible subsystem based on import from **MSC.NASTRAN** and its analysis in **UM** are presented on in Figure 1.29.

4. File umfumNX**.alt and **NASTRAN_UM.EXE** are placed in the directory .\bin after installation of **Universal mechanism** software.

Note.	**UM 9.0** supports importing from **NX NASTRAN 8.0** and **NX NASTRAN 9.0, NX NASTRAN 12** Importing data from other versions of **NX NASTRAN** was not tested.

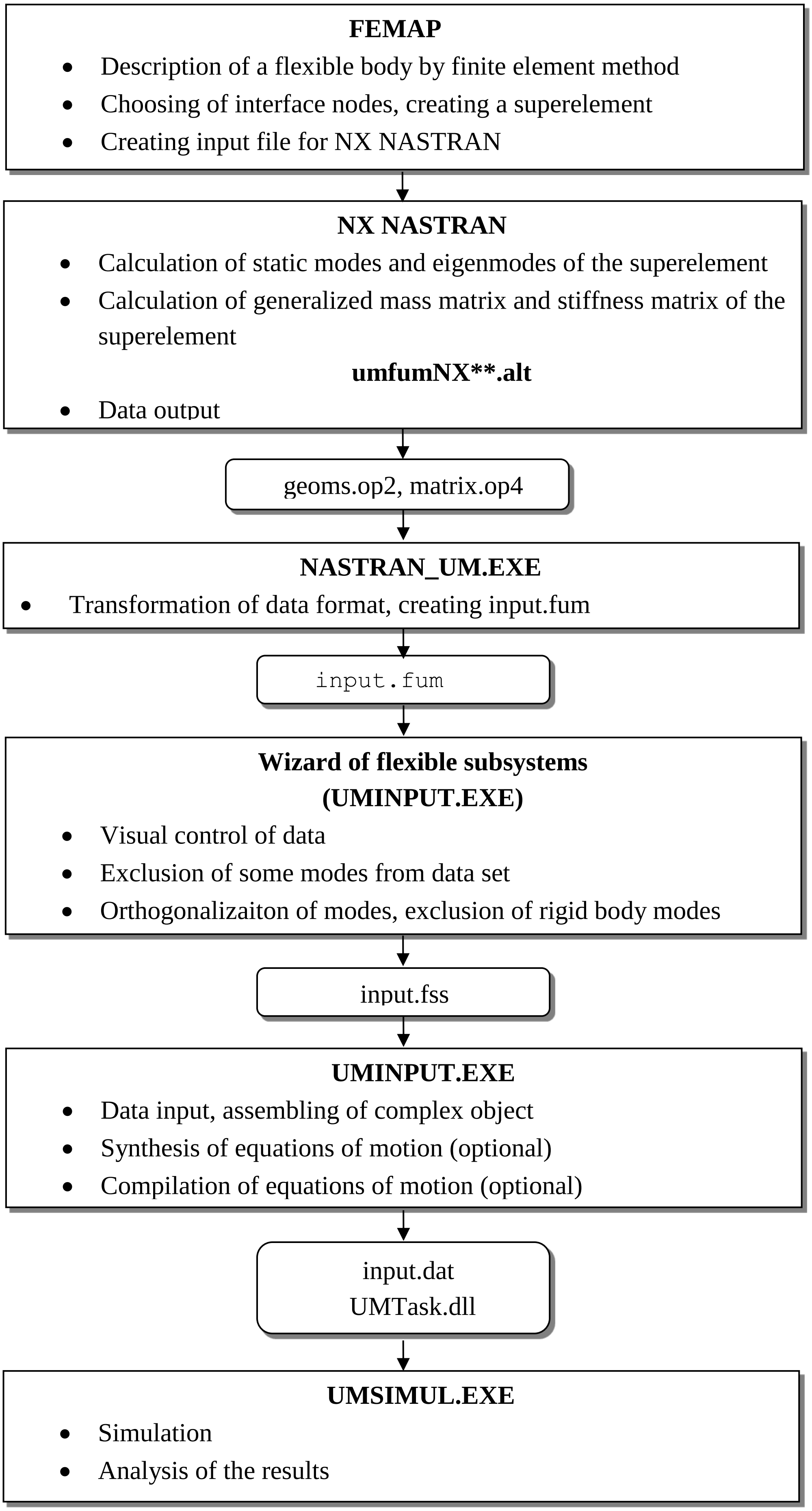


Figure 1.29. Creating of flexible subsystem using MSC.PATRAN/NASTRAN

1.2.3.3. Preparing data in NX NASTRAN/FEMAP environment

The main stage of creating a flexible subsystem

1. Making finite element model of a considered object in **MSC.PATRAN**. The model should be described in the international system of units of measurement (SI). Finite element mesh should include nodes in the joint points and points of attachment of force elements to the subsystem in the complex object. Some feature of preparing the finite element models are described in Sect. 1.3.1. *"Animation window"*, p. 1-83.
2. Choice of interface nodes in accordance with joint points and attachment points of force elements in the **UM** model and creating a superelement. This stage is implemented in **FEMAP** by the means of *Constraints*. Create new constraint set using **Model** → **Constraint** → **Create/Manage Set named**, for example, *InterfaceNodes*, see Figure 1.30. Then use **Model** → **Constraint** → **Nodal** command to fix all degrees of freedom for interface nodes, see Figure 1.31.

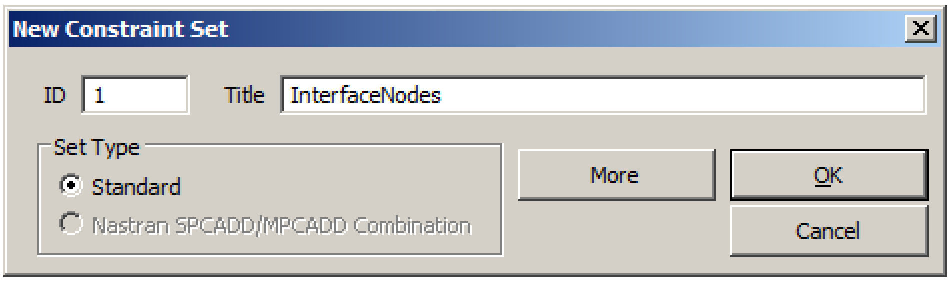


Figure 1.30. Creating new constraint set in FEMAP

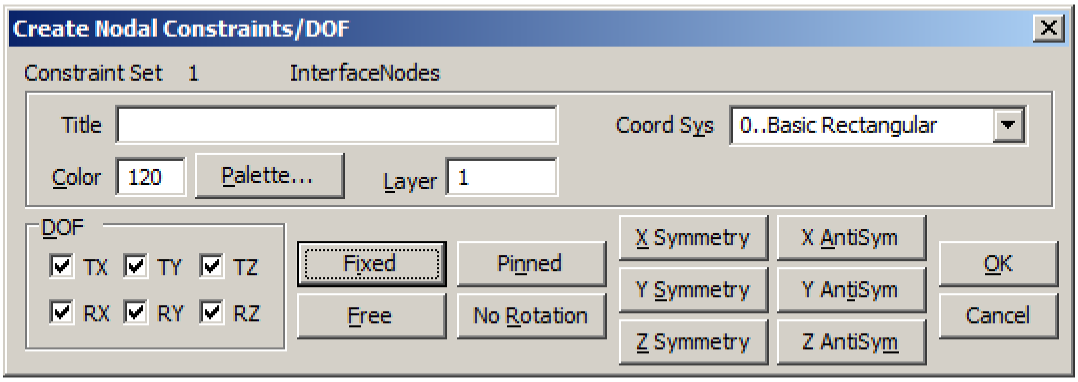


Figure 1.31. Creating nodal constrains in FEMAP

3. Create a new analysis set using **Model** → **Analysis** command, Figure 1.32.
   (1) Input the title of the analysis set and select **2..Normal Modes/Eigenvalue** in **Analysis Type** field, Figure 1.33.
   (2) Assign the intermediate output files *geoms.op2* and *matrix.op4* as it is shown below, Figure 1.34.

ASSIGN OUTPUT2='geoms.op2' UNIT=13 FORM=UNFORMATTED
ASSIGN OUTPUT4='matrix.op4' UNIT=15 FORM=UNFORMATTED

   (3) Add the following line to window shown in Figure 1.35 in order to link up umfumNX**.alt module.

include umfumNX**.alt

File umfumNX**.alt should be placed into the directory where it can be found by **NX NASTRAN**.

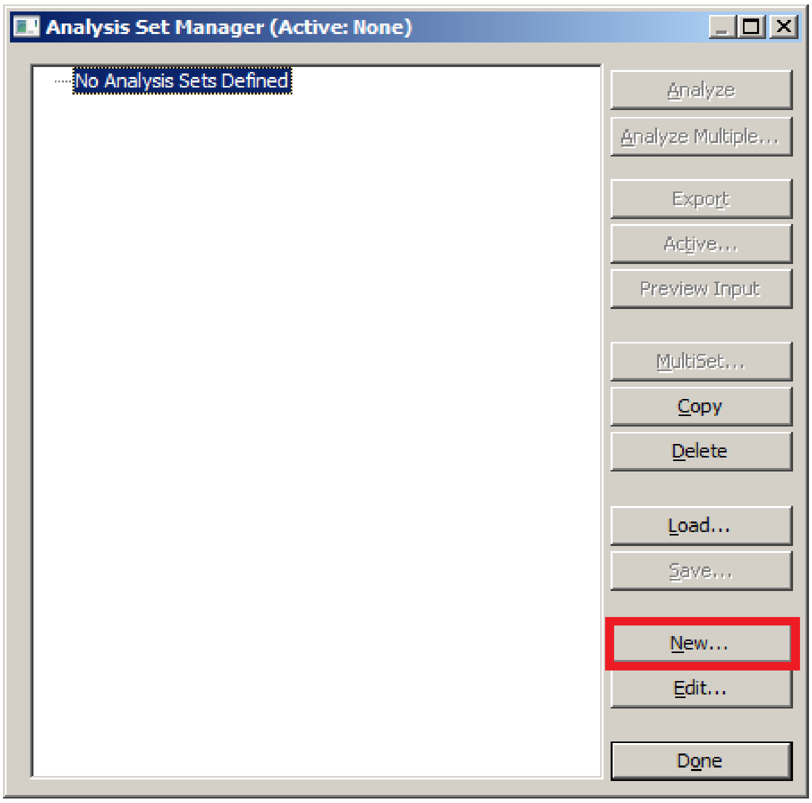


Figure 1.32. Creating a new analysis set in FEMAP

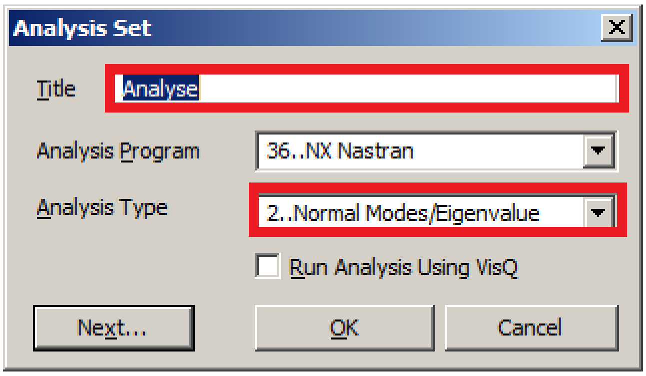


Figure 1.33. Assignment of analysis type in FEMAP

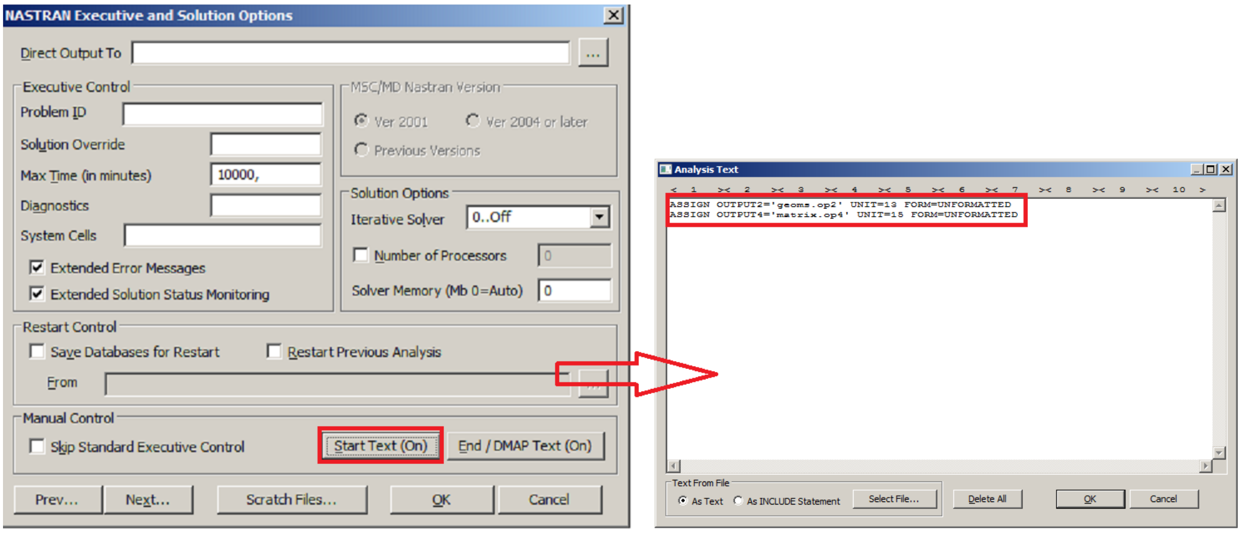


Figure 1.34. Assignment of intermediate output files geoms.op2 and matrix.op4

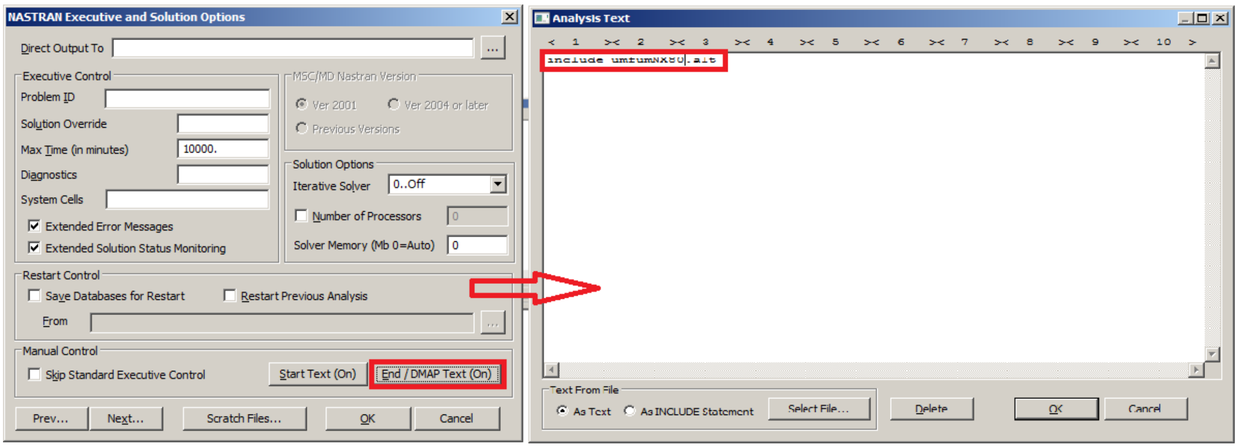


Figure 1.35. Linking umfumNX**.alt module

(4) The number of needed eigenmodes should be set in the window that is shown in Figure 1.36. For that use the following commands.

SPOINT, 300001, thru, 300030

QSET1, 0, 300001, thru, 300030

Thirty eigenmodes of the superelement are required to be calculated in this example. As a rule, the eigenmodes corresponding to the lowest eigenvalues are calculated.

The SPOINT operator defines scalar parameters called *scalar points* in the **NX NASTRAN** user's guide. In this case, the scalar points are defined as modal coordinates of the superelement. They correspond to eigenmodes of the model fixed in interface nodes. Thirty scalar points are defined in the example. These points are numbered from 300001 to 300030. As a rule, numbers of the points are chosen greater than maximal number of nodes of the finite element model. That is, the numbers can be defined from 500001 to 500030. Coincidence numbers of scalar points and numbers of internal nodes of a superelement are not allowed.

SEQSET1 operator defines generalized coordinates of a superelement. More detailed information about command mentioned above one can find in the **NX NASTRAN** user's manual.

(5) Set parameters in **NASTRAN Modal Analysis** dialog window, Figure 1.37.

In the presented example, it is required to calculate thirty eigenmodes corresponding to lowest eigenvalues in the range from 0 to 1000 Hertz by Lanczos method. The number of eigenmodes should be equal to the number of the scalar points defined in Step 7 by SPOINT operator. Calculated modes are normalized relatively mass matrix (M-norm). This normalization method should always be chosen under preparing data for export to **UM**.

(6) Open the dialog window that is shown in Figure 1.38 and copy the following command:

VECTOR(SORT1,REAL)=ALL

This command defines the output format of displacement vector.

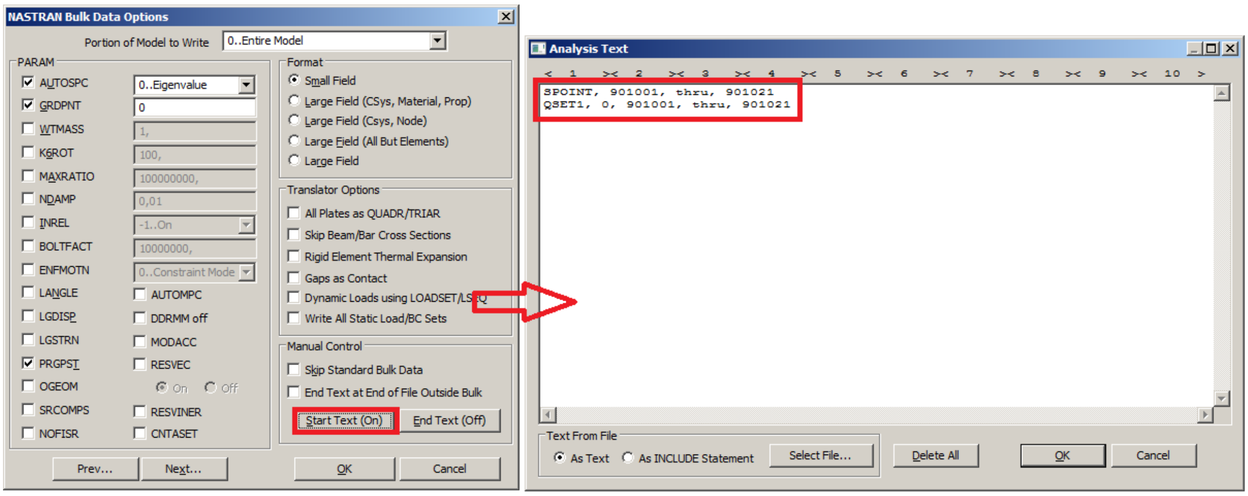


Figure 1.36. Scalar points

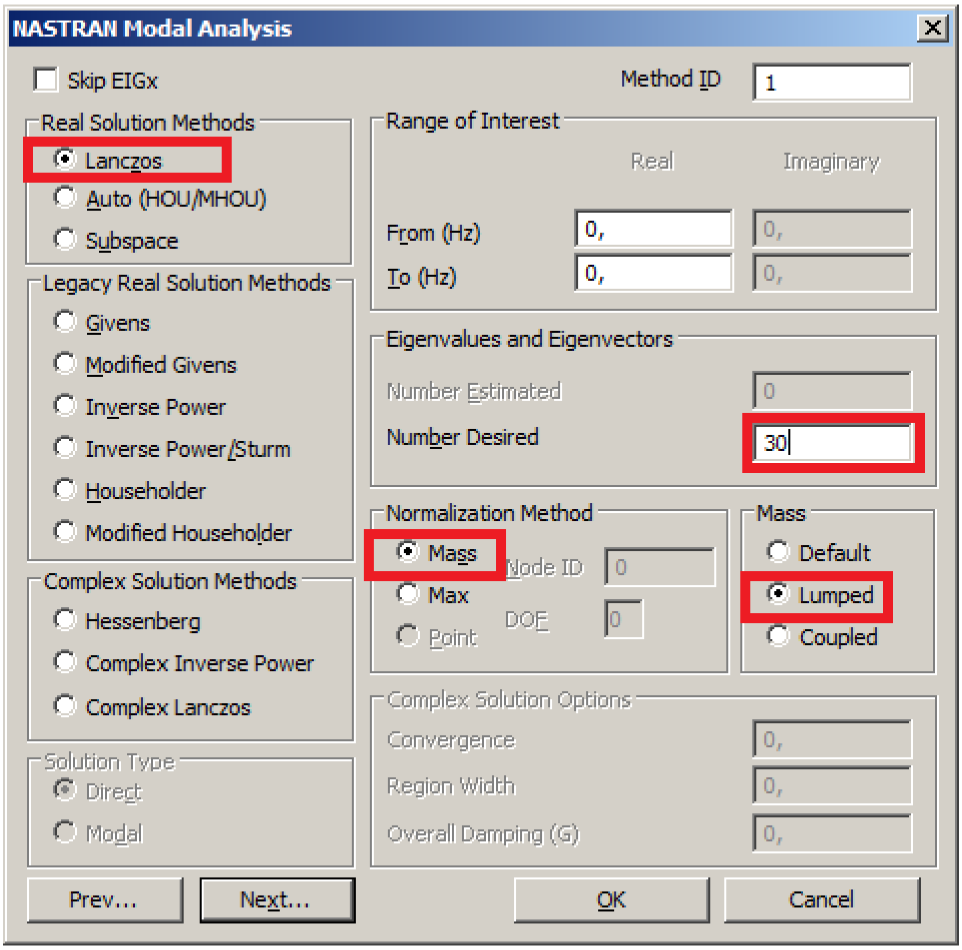


Figure 1.37. Parameters of modal analysis

(7) In order to create stress or/and strain sensors, place the following list of commands after VECTOR command, see Figure 1.38.

SET 501 = ALL

STRESS(CORNER)=501

SET 502 = 101,111,120 THRU 136, 170

STRAIN(CORNER) = 502

OUTPUT(POST)

SET 101 = ALL

SURFACE 11 SET 101

Let us briefly consider used commands.

SET defines a set of numbers of finite elements for which stresses or strains are calculated. 501 is the number of set. It can be chosen at will among numbers not defined earlier. Reused number as parameter SET command redefines set of finite element numbers.

STRESS calculates stresses in elements. The number of set is specified after equals sign. The parameter CORNER in brackets is written if elements CQUAD4 are presented in the set. If this parameter is not specified, stresses are calculated only in the center of the elements CQUAD4.

STRAIN calculates strains in elements.

OUTPUT separates the commands of different types. The parameter POST starts output of stresses and strains.

SURFACE defines a surface for calculation of stresses and strains. The command is used for set of shell elements. If calculation of stresses in solid elements is needed, the VOLUME command is used. For example,

VOLUME 11 SET 101

In this example, the set 501 includes all finite elements; the set 502 consists of finite elements 101, 111, from 120 to 136 and 170.

In order to create a sensor in a node in **UM** software, all elements which include this node should be specified in the set for STRESS or STRAIN command.

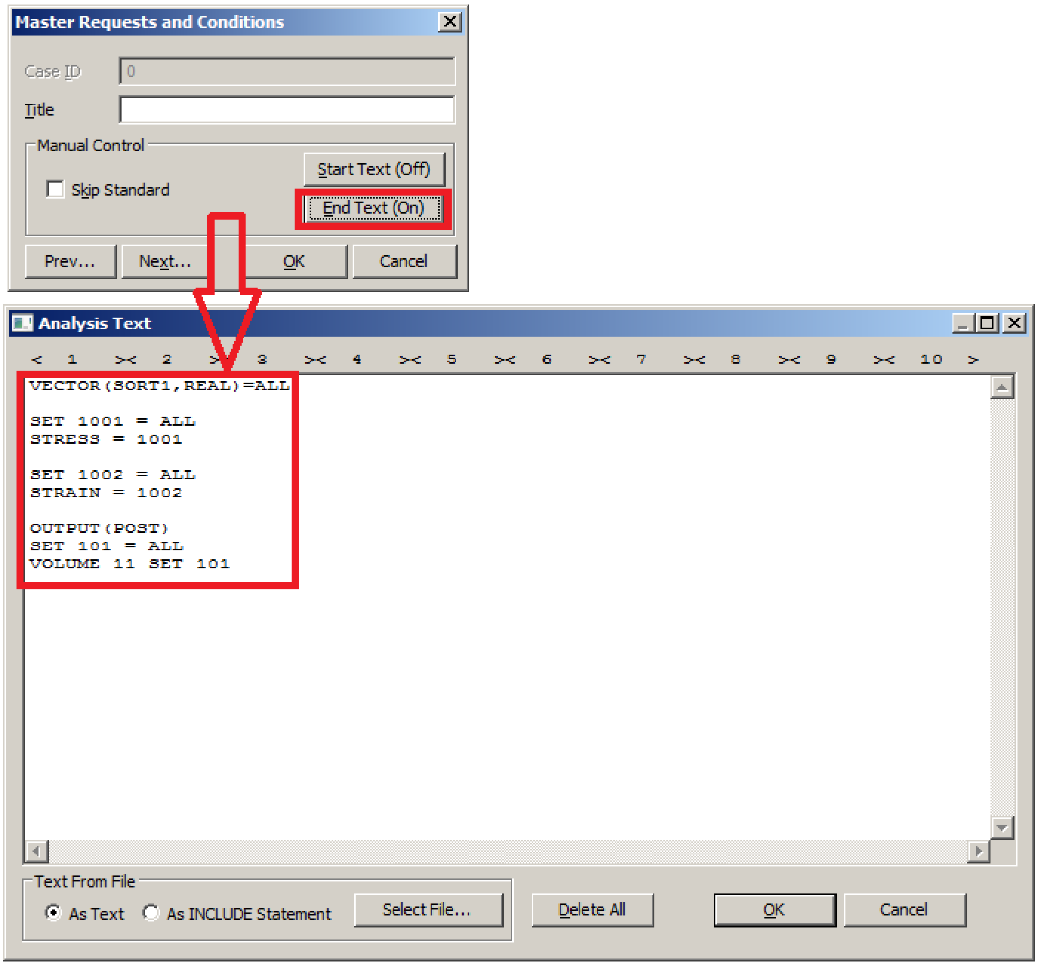


Figure 1.38. Stress/strain sensors

(8) Click **Next** to come to **Boundary** Conditions dialog window. Select **0..None** in **Constraints** list, Figure 1.39. Select the set of interface nodes described above in **Master (ASET)** field. Click **OK** to finish preparing **Analysis Set**. Later use the **Model Info** panel to select this analysis set.

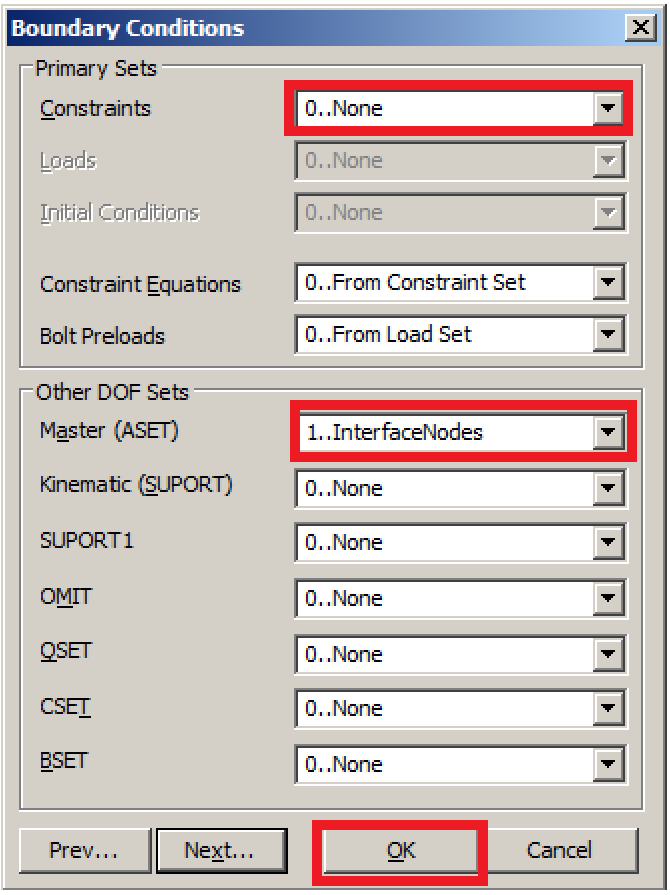


Figure 1.39. Define boundary conditions

4. Select the just created analysis set.
5. Click **Analyze** to run the analysis set. If the calculation is successful, files geoms.op2 and matrix.op4 will be created in the working directory. Diagnostics messages are outputted in the file JobName.f06. This information can be useful if there are calculation errors.

1.2.3.4. NX NASTRAN to UM data exchange

Run **NASTRAN_UM.EXE** program for making input.fum file. Select geoms.op2 in the working directory of **NX NASTRAN** and specify target directory for input.fum. The flags **Stresses** and **Strains** enable/disable output of corresponded data to file input.fum, Figure 1.28. This data, of course, must be calculated by **MSC.NASTRAN** using commands specified in the Sect. 1.2.2.3. *"ANSYS-UM data exchange"*, p. 1-16, Step 3.7 of the list.

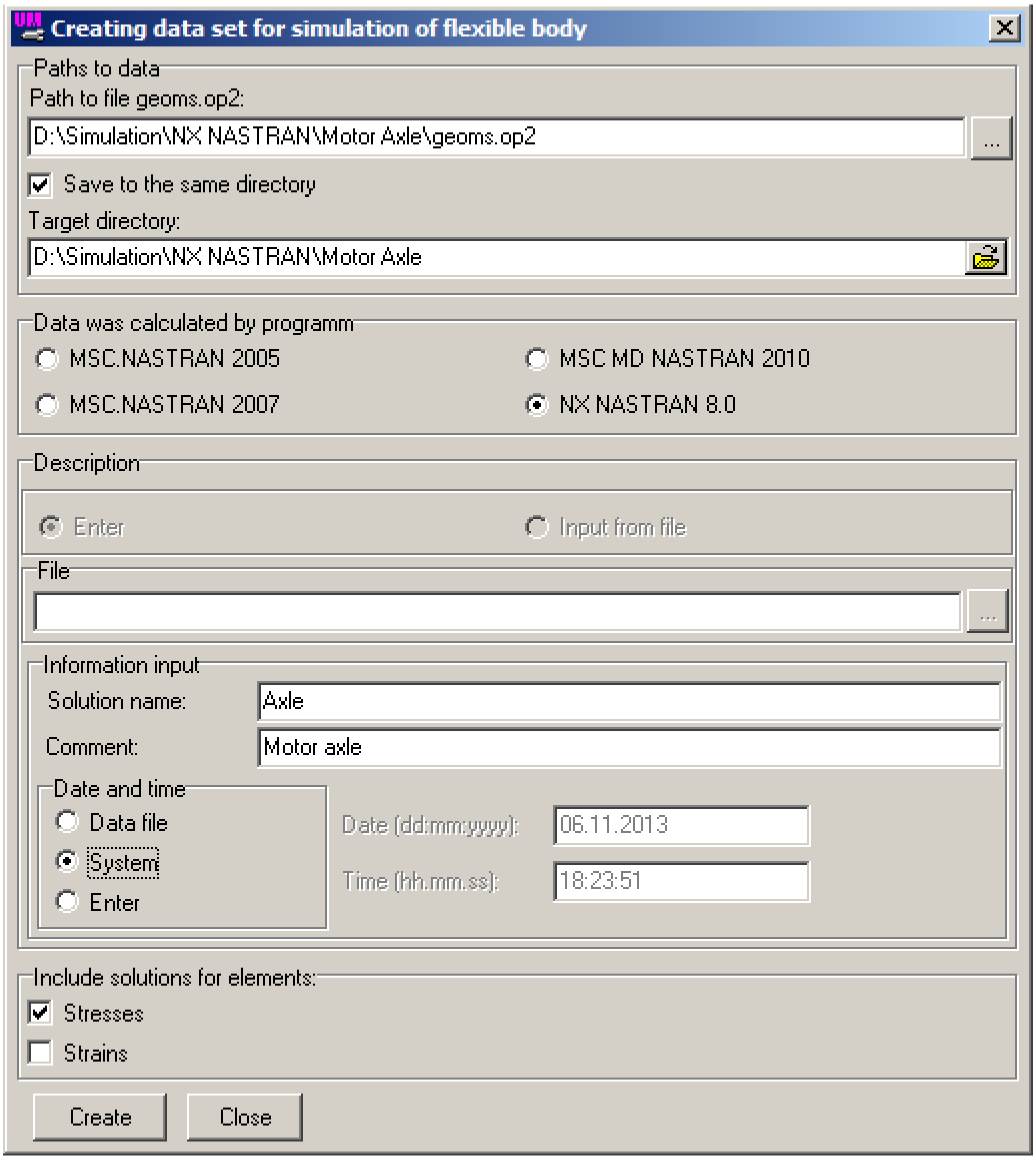


Figure 1.40. NASTRAN_UM.exe main window

Solution name, comment and date of calculation can be inputted into corresponded fields of the form or can be read from files JobName.bdf or JobName.f06. This data is attributes of an **UM** model which are written to file input.fum.

Perform data transformation by the **Create** button. If it is successful, file input.fum will be created in the target directory. Further work with this file is described below in Sect. 1.2.6. *"Exporting finite element model from NX NASTRAN"*, p. 1-38.

1.2.4. Model creation in ABAQUS environment and data exchange

1.2.4.1. General information

Figure 1.41 represents a complete cycle of preparation of source data using the ABAQUS program and analysis of the model in UM.

Elastic subsystem is created on the basis of the superelement method. After development of the finite element model, user should choose the interface nodes and create the superelement. The necessary data is imported during the modal analysis of the superelement.

Now we will describe the rules for preparing source data in ABAQUS and also the structure and sequence of the software use to import data in **Universal mechanism**.

Step-by-step description of the development and model analysis, which includes imported elastic subsystem, is listed in the manual *"Getting started to work in "Universal mechanism software package": a module for simulation of elastic bodies"*.

1.2.4.2. Software structure, import scheme

For importing from **ABAQUS software package UM** provides the **ABAQUS_UM.EXE** conversion program that reads the file *.fil and generates an input.fum file, that is, saves the data in the **UM** format.

The **ABAQUS_UM.EXE** conversion program after installing the universal mechanism program is located in the directory .\bin. Now we will describe in detail the preparation of the data.

Note.	**UM 2023** supports data import only from **ABAQUS 6.12-1**. The import data on deformations and stresses from the **ABAQUS 6.12-1** program is not supported.

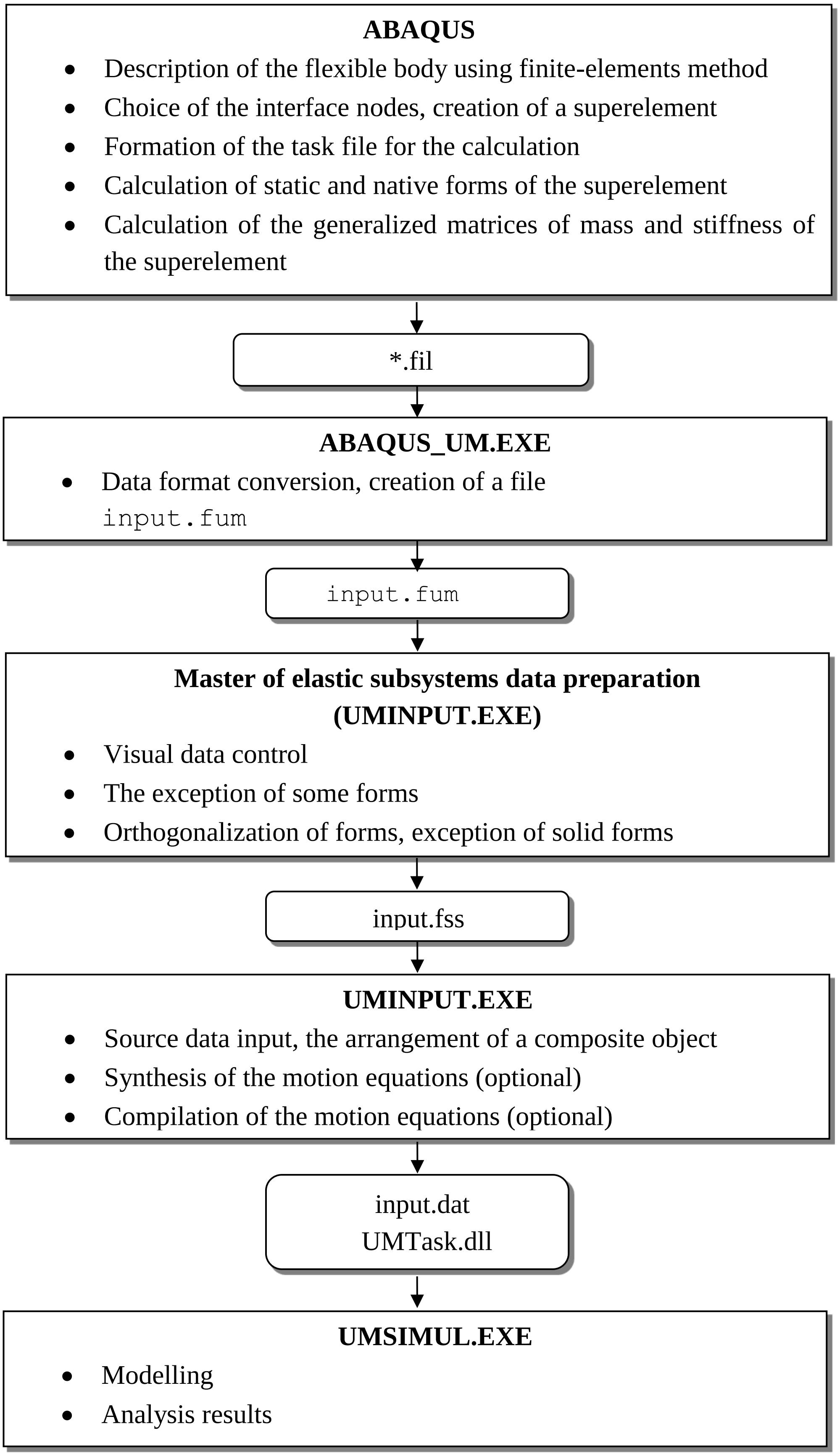


Figure 1.41. Creating a flexible subsystem using ABAQUS

1.2.4.3. Preparing data in ABAQUS environment

The basic steps of creating elastic subsystem

1. The creation of the finite element model of the analyzed object in the **ABAQUS** program. The model is described in the SI-system of units. The finite element mesh must contain nodes in the hinge points and points of attachment of the strength elements. Some features of creation of the finite element model are described in Sect. 1.2.6. *"Some* features related to preparing FE models*"*, p. 1-72.
2. To create a step of calculating the natural frequencies in the **ABAQUS** program go to the **STEP** module (Figure 1.42).

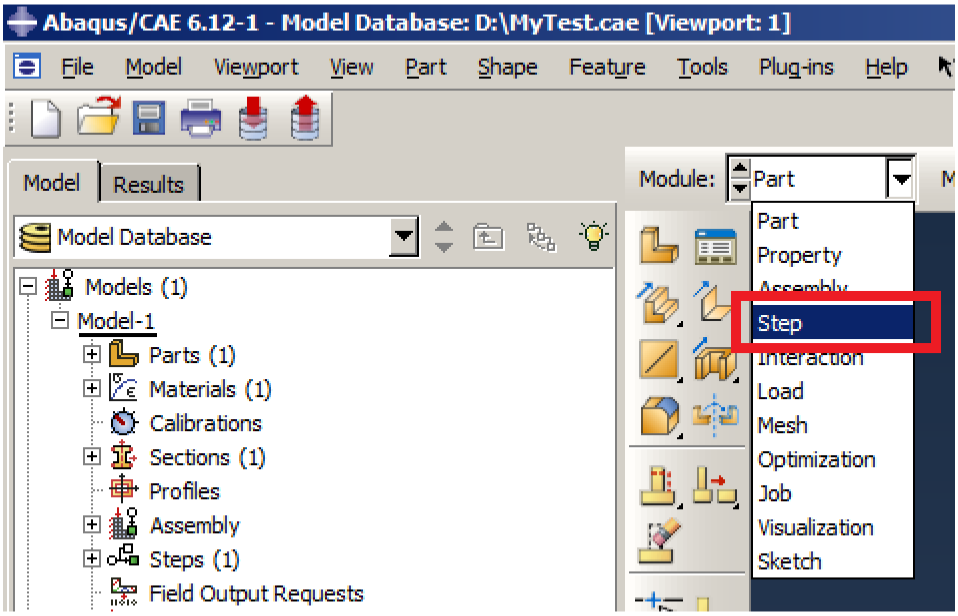
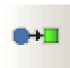


Figure 1.42. How to go to the **STEP** module in the **ABAQUS** program

Using the **Create Step** button  (or double-click on the element  of the model tree or by using **Step** → **Create** menu command), create a new calculation step. As a result, **Create Step** window should appear (Figure 1.43), where the initial calculation step is defined. In this window set the name of the step of calculation as Frequency, in the **Procedure type** drop-down list select **Linear perturbation** and choose **Frequency** as a calculation step.

Click the **Continue** button to go to the parameters window of the **Edit Step** calculation step, (Figure 1.44). In the **Edit Step** window on the **Basic** tab in the **Description** column set the name of the current task as Frequency and specify the number of natural frequencies. On the **Other** tab of the **Edit Step** (Figure 1.45) select the value of parameter **Normalize eigenvectors by** equal to **Mass**. Complete the assignment of parameters of the calculation of natural frequencies by pressing the **OK** button.

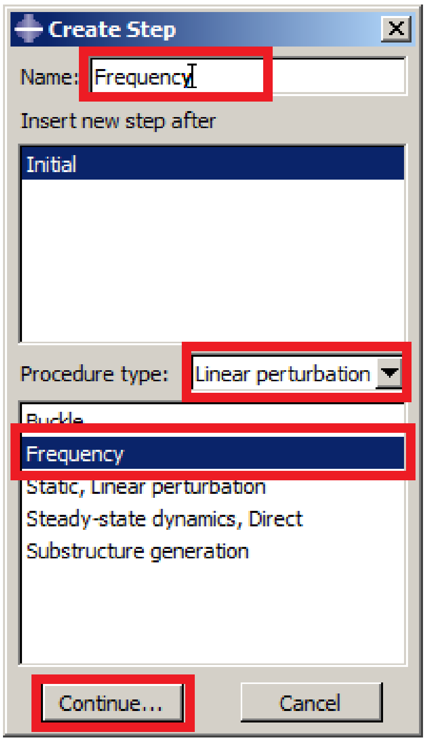


Figure 1.43. Creating **Frequency** calculation step

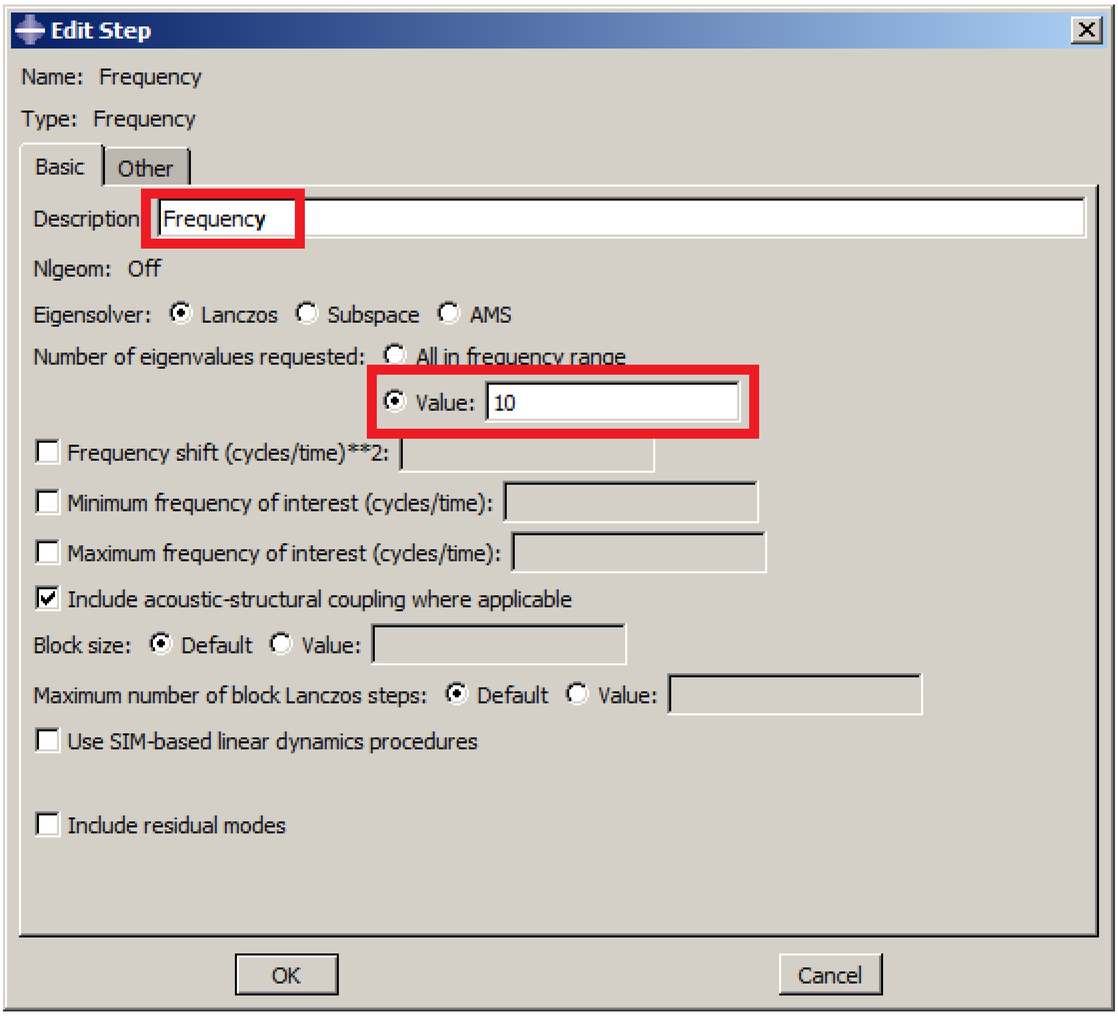


Figure 1.44. Setting of parameters for calculating natural frequencies

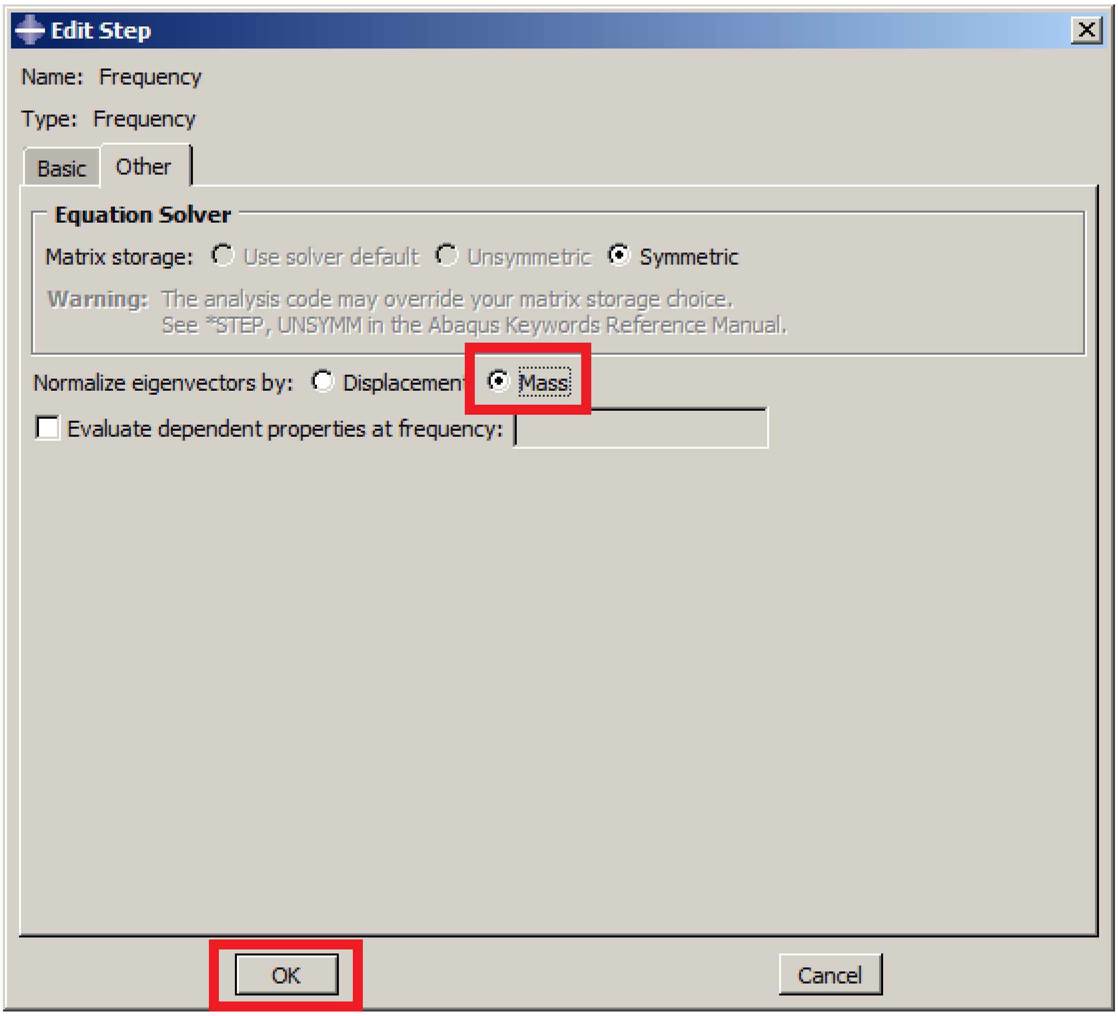


Figure 1.45. Setting of parameters for calculating natural frequencies (continuation)

Then in the interface nodes set the boundary conditions for the step of calculating Frequency. To do this click twice on the item **BCs** in the model tree in the section of the created Frequency calculation step (Figure 1.46).

In the appeared window **Create Boundary Condition** choose the category of boundary conditions **Mechanical** boundary conditions **Displacement/Rotation** and push **Continue** button (Figure 1.47). Then you need to either choose the interface nodes using mouse in the graphic window, or use a pre-generated set of nodes **Set**. Then click the **Done** button at the bottom of the graphic window (Figure 1.48).

In the appeared **Edit Boundary Condition** window choose the fixed degrees of freedom (Figure 1.49).

Note. Fixed degrees of freedom in the **Edit Boundary Condition** window should be acceptable for the types of finite-elements used in model.

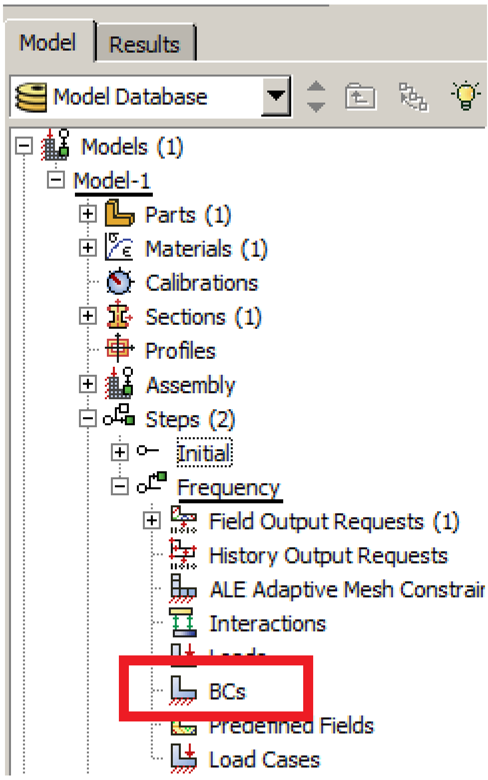


Figure 1.46. How to create boundary conditions for *Frequency* calculation step

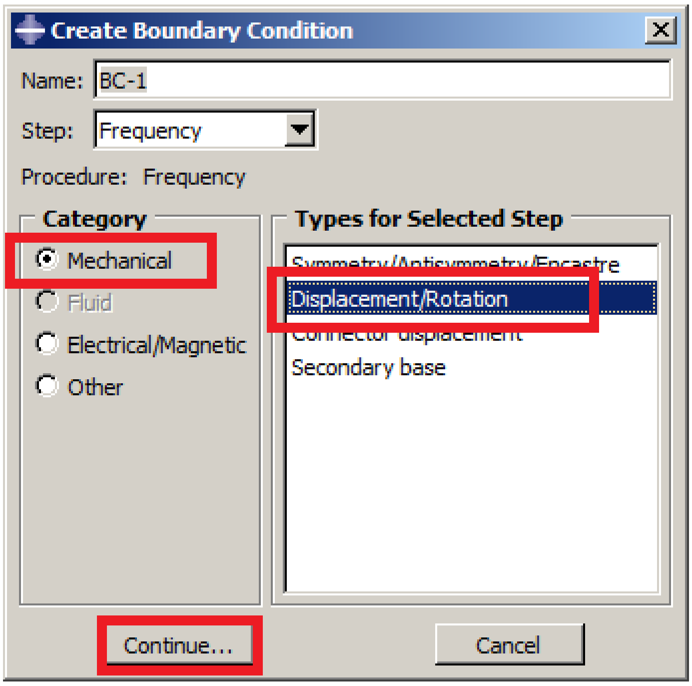


Figure 1.47. How to create boundary conditions for *Frequency* calculation step (continuation)

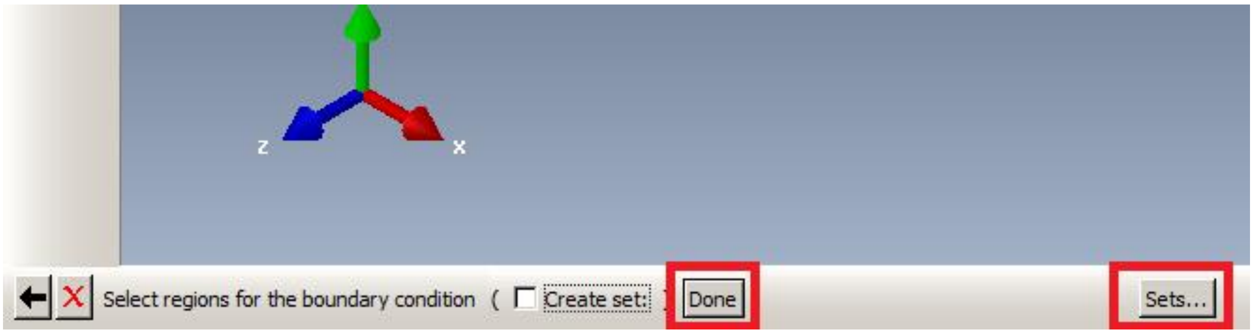


Figure 1.48. How to create boundary conditions for *Frequency* calculation step (continuation)

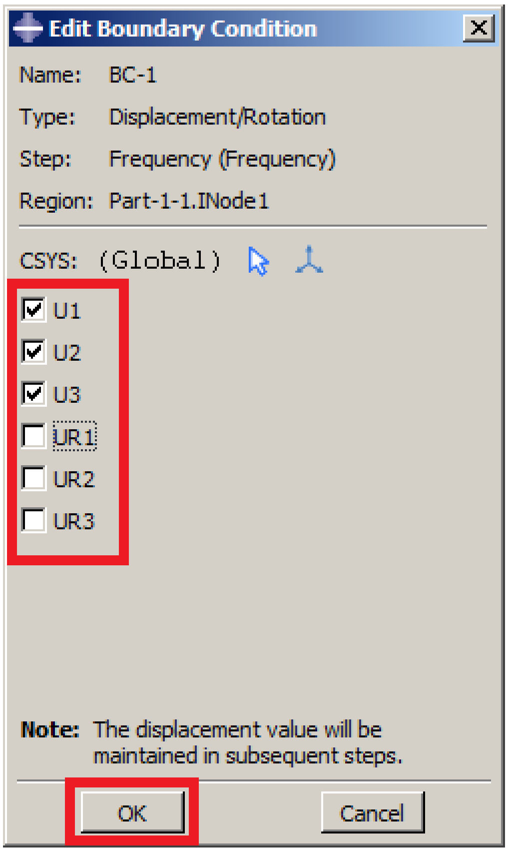
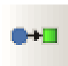


Figure 1.49. How to create boundary conditions for *Frequency* calculation step (continuation)

3. To create calculation step the generation of the superelement click the **Create Step** button again  (or use double click on the model tree element 

, or a menu command **Step** → **Create**).

In the appeared **Create Step** window, (Figure 1.50) enter the name of the calculation step Substructure generation, then in the drop-down list **Procedure type** select **Linear perturbation**, specify the calculation step **Substructure generation** and click **Continue**.

In the **Edit Step** window on the **Basic** tab (Figure 1.51) in the **Description** column specify the name of the problem being solved as Substructure generation, and in the **Substructure identifier** column you should enter a random number, such as 1. This means that the created superement will have the ID Z1. Set the tab **Evaluate recovery matrix** in position **True** and activate the **Whole model** field.

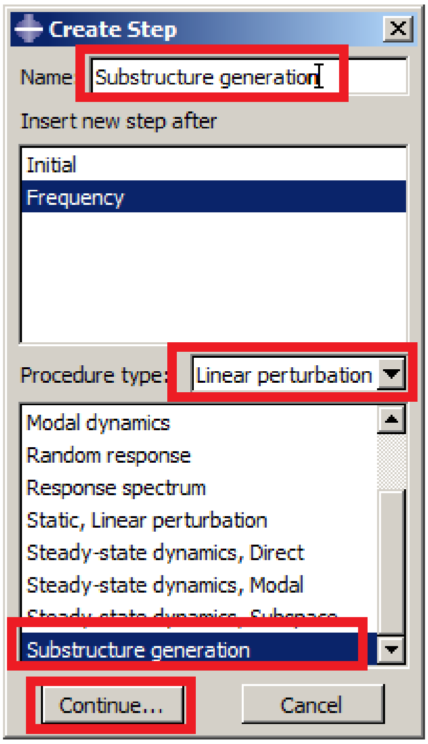


Figure 1.50. Creating the Substructure generation calculation step

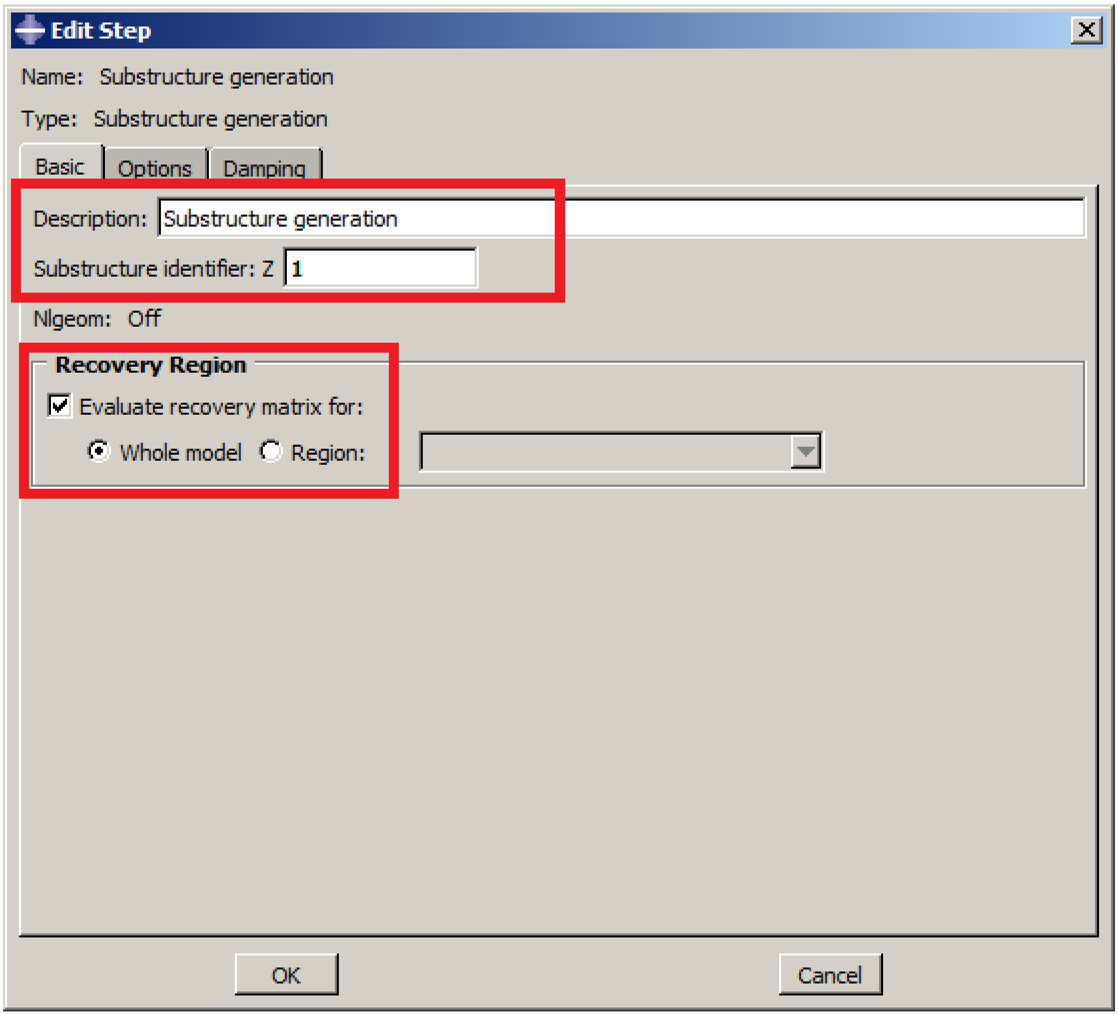


Figure 1.51. Creating calculation parameters of the superelement

On the **Options** tab of the **Edit Step** window (Figure 1.52) set the **Compute redused mass matrix** and **Specify redained eigenmodes by** checkboxes and select **Mode range**. Then specify the eigenforms, which were used to build the superelement. In this particular case in the figure, for example, it is shown that the eigenforms are used from 1 to 10 with the step 1.

On the **Damping** tab of the **Edit Step** window you don't need to point at anything, click **OK**.

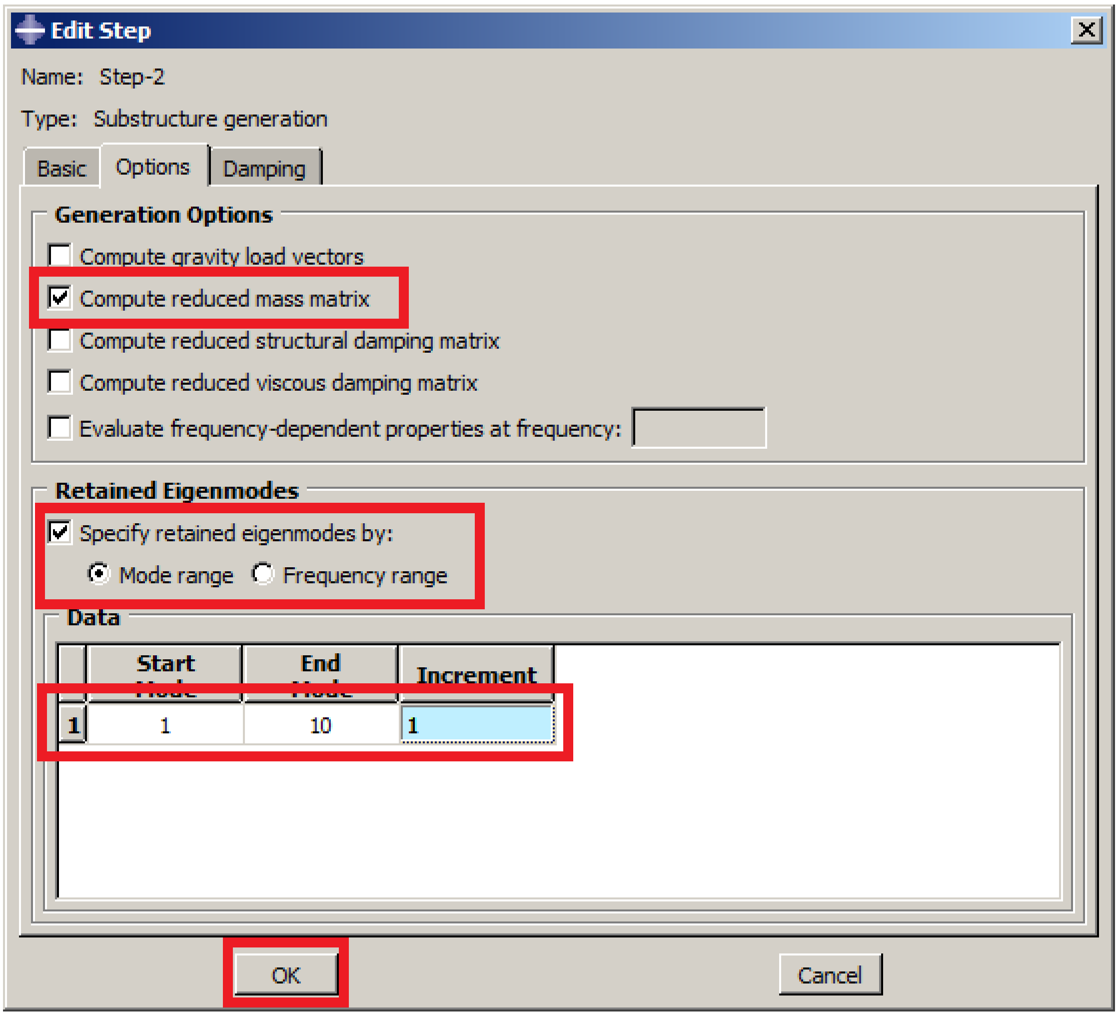


Figure 1.52. Creating calculation parameters of the superelement (continuation)

Then set the boundary conditions for the **Substructure generation** calculation step in the interface nodes. To do that in the model tree in the **Substructure generation** section click twice on the item **BCs** (Figure 1.53).

In the appeared window **Create Boundary Condition** choose the category of boundary conditions **Mechanical**, type for selected step **Retained nodal dofs** and click **Continue** (Figure 1.54). Then you must either select the interface nodes with the mouse in the graphics window or use a pre-formed node **Set**. Then click the **Done** button at the bottom of the graphics window (Figure 1.55).

In the appeared window **Edit Boundary Condition** choose the fixed degrees of freedom (Figure 1.56).

Note. The fixed degrees of freedom in the **Edit Boundary Condition** window have to be acceptable for the types of finite elements, used in the model.

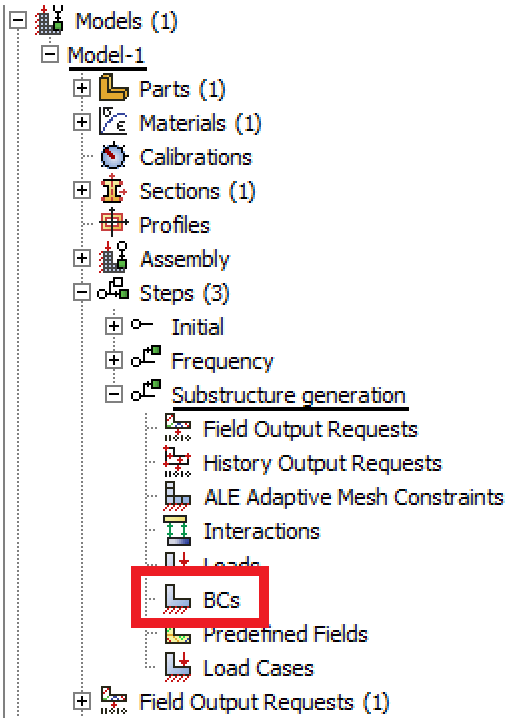


Figure 1.53. Setting boundary conditions for **Substructure generation** calculation step

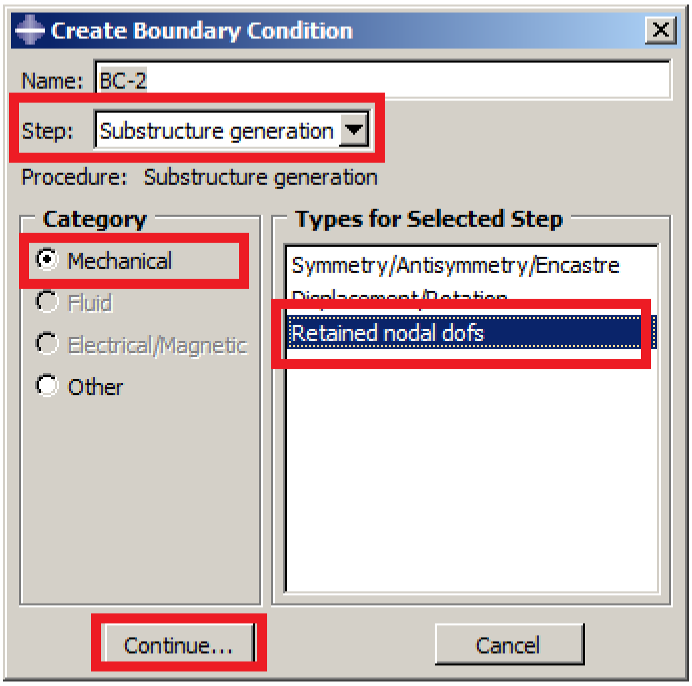


Figure 1.54. Setting boundary conditions for **Substructure generation** calculation step (continuation)

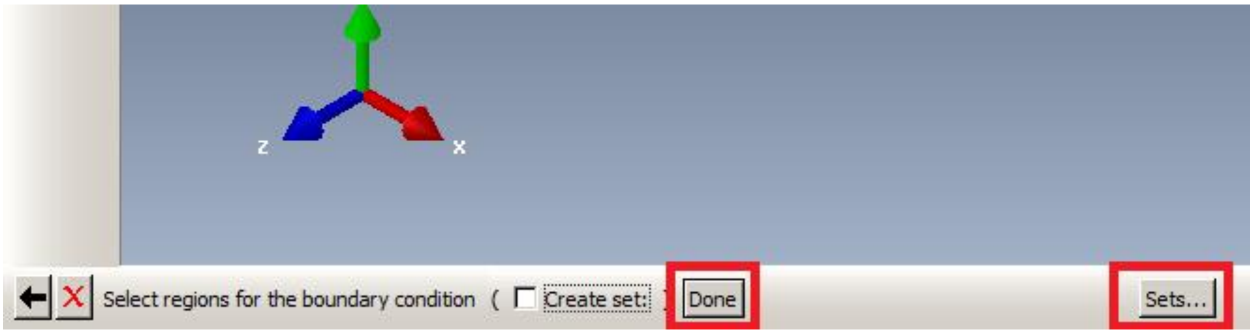


Figure 1.55. Setting boundary conditions for **Substructure generation** calculation step (continuation)

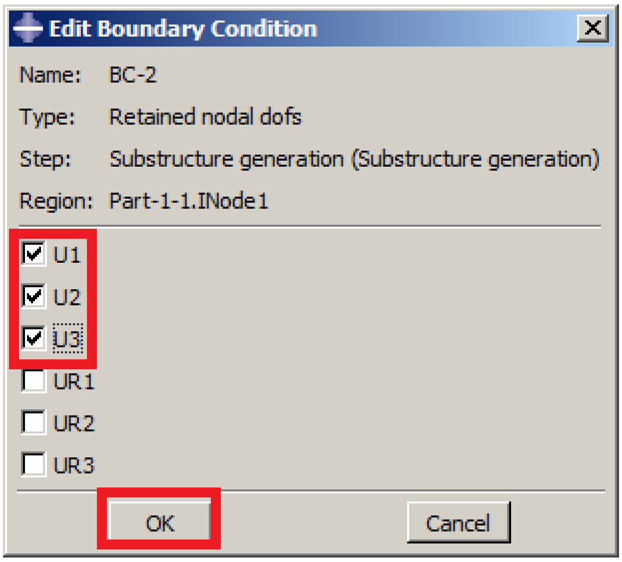


Figure 1.56. Setting boundary conditions for **Substructure generation** calculation step (continuation)

4. To create an analysis project in the project tree click twice on the **Job** item (Figure 1.57) or right-click and select **Create Job**. Click **Continue** in the appeared window **Create Job** (Figure 1.58).

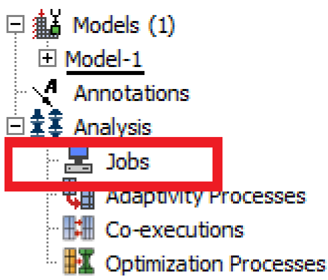


Figure 1.57. Creating analysis project

An edit job window appears (Figure 1.59). To accept the default parameters click **OK**.

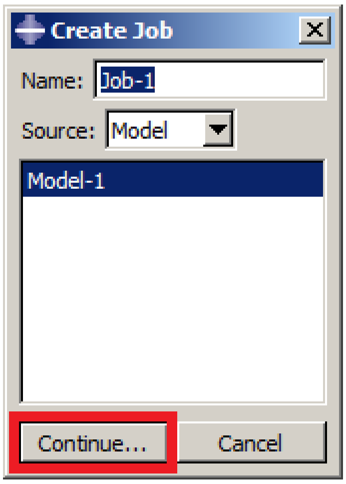


Figure 1.58. Create Job window

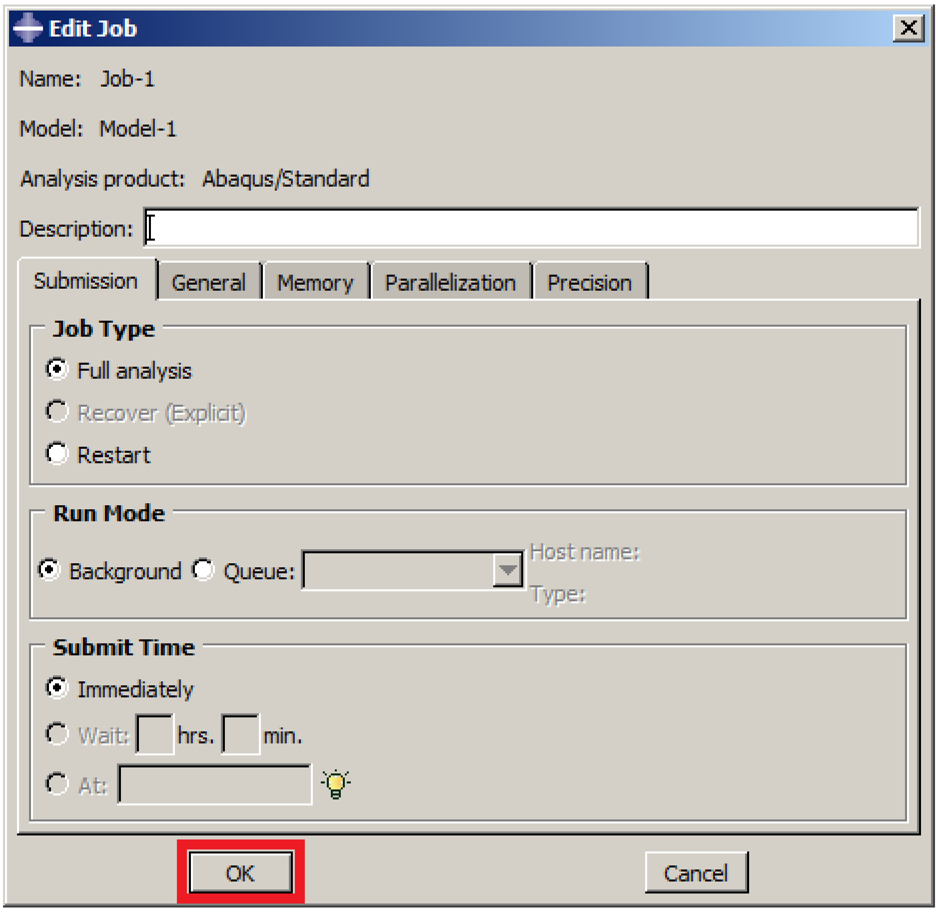


Figure 1.59. Edit Job window

Right-click on **Job** in the model tree and choose the item **Manager** in the context menu (Figure 1.60).

In the appeared **Job Manager** window click the **Write Input** (Figure 1.61) button. In the working directory of the **ABAQUS** program a file will be created with the name of the project for analyzing (in our example it is Job-1) and the inp extension. To change the current working directory of the **ABAQUS** program use the menu command **File → Set Work Directory** (Figure 1.62).

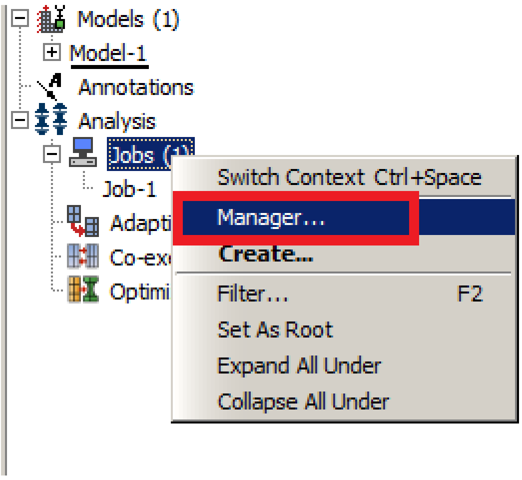


Figure 1.60. Starting Job Manager

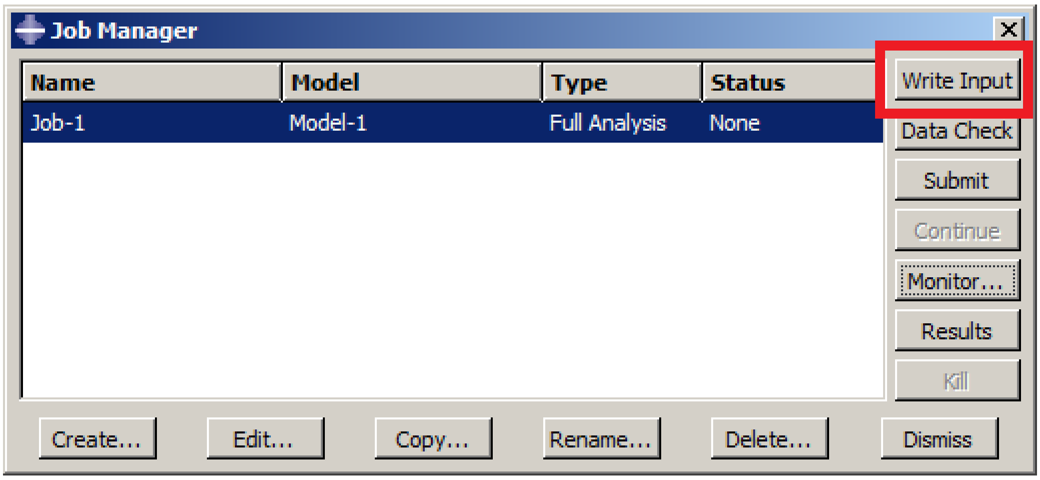


Figure 1.61. Job Manager window

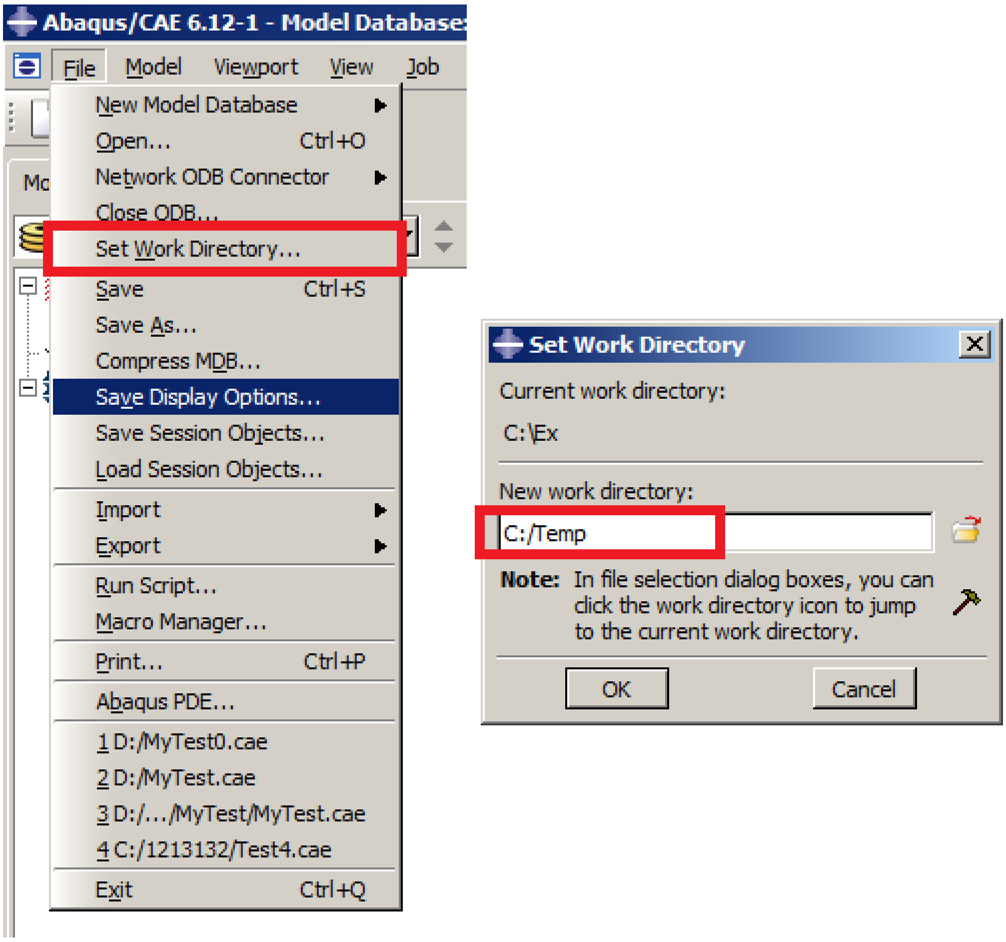


Figure 1.62. How to change the current work directory in the **ABAQUS** program

Then you need to change the generated *. inp file by opening it in a text editor.

To output eigenforms, nodes and elements at the stage of description of the parameters of calculation of the natural frequencies before the last line *END STEP you need to enter the command of the matrix output in the output file *.fil.

** STEP: Frequency

…

…

…

*ELEMENT MATRIX OUTPUT, MASS=YES, ELSET=Part-1-1__PickedSet2

*NODE FILE

U

*END STEP

At the end of the file before the last line (*End Step) you need to enter the command to output matrices of mass and stiffness in the output file *.fil. The last lines in the Job-1.inp file should look like this:

** STEP: Superelement

…

…

…

*SUBSTRUCTURE MATRIX OUTPUT, STIFFNESS=YES, MAS=YES, RECOVERY=YES
*END STEP

Note. The ELSET parameter takes the value corresponding to the name of the set which includes all the FE models (in the example above it is Part-1-1__PickedSet2). The name of the sample that includes all the finite elements in the file Job-1.inp. is already present because the program creates and uses it, for example, when generating the finite elements. Below are the lines from the file Job-1.inp., which specify a set of all created FE models from 1 to 60 with step 1, the name of the assembly is Part-1-1__PickedSet2.
*Elset, elset=Part-1-1__PickedSet2, generate
1, 60, 1
Therefore in the output matrixes command in the output file replace the value of the ELSET parameter on the value which is used in Your *.inp file in the generation of finite elements.

After all changes save the *.inp file.

To run the *.inp task file for the calculation in the job manager, click **Create** in the dialog box **Create Job** that appears, select *.inp file and click **Continue** (Figure 1.63). Edit job window will appear (Figure 1.59). Click **OK** to accept all default settings.

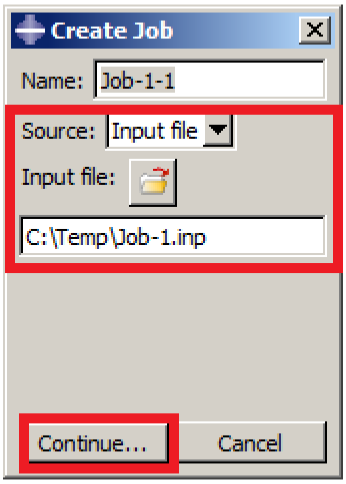


Figure 1.63. Create job window

Make the project, based on the modified *.inp file, active and run the task for simulation by clicking **Submit** in the Job Manager (Figure 1.64). **Abaqus** in the working process allows monitoring when carrying out the calculation. To do this, click the **Monitor** button in Job Manager. In the appeared dialog window at the **Errors** and **Warnings** tabs you can view the critical errors and warnings that may occur during the calculation.

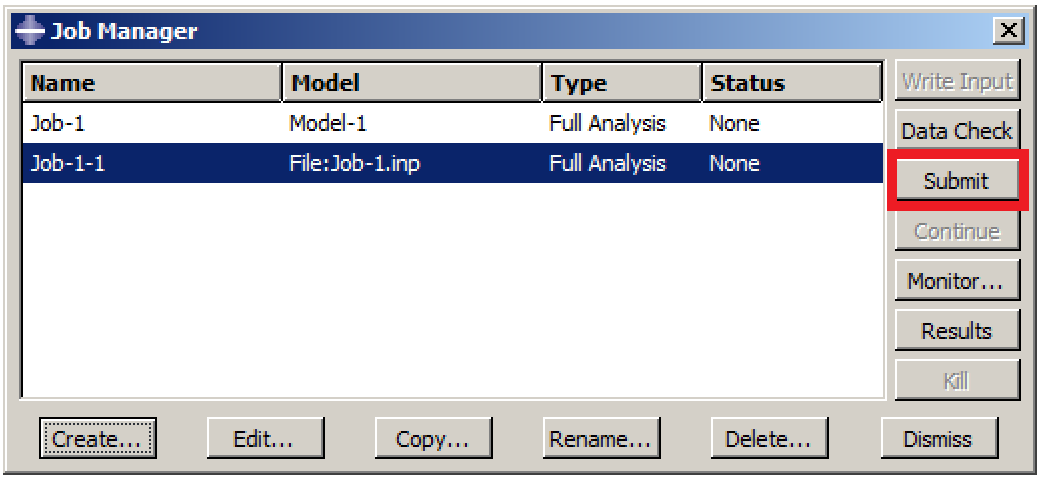


Figure 1.64. Calculation running

Upon successful completion of the calculation in the working directory *.fil file will be created which contains the data to import in the **UM**.

1.2.4.4. Data exchange with ABAQUS

Run the converter **ABAQUS_UM.EXE** to create a file input.fum. Select the *.fil file in the appropriate directory and specify the directory in which to save the file input.fum. The purpose of all fields is clear by their names.

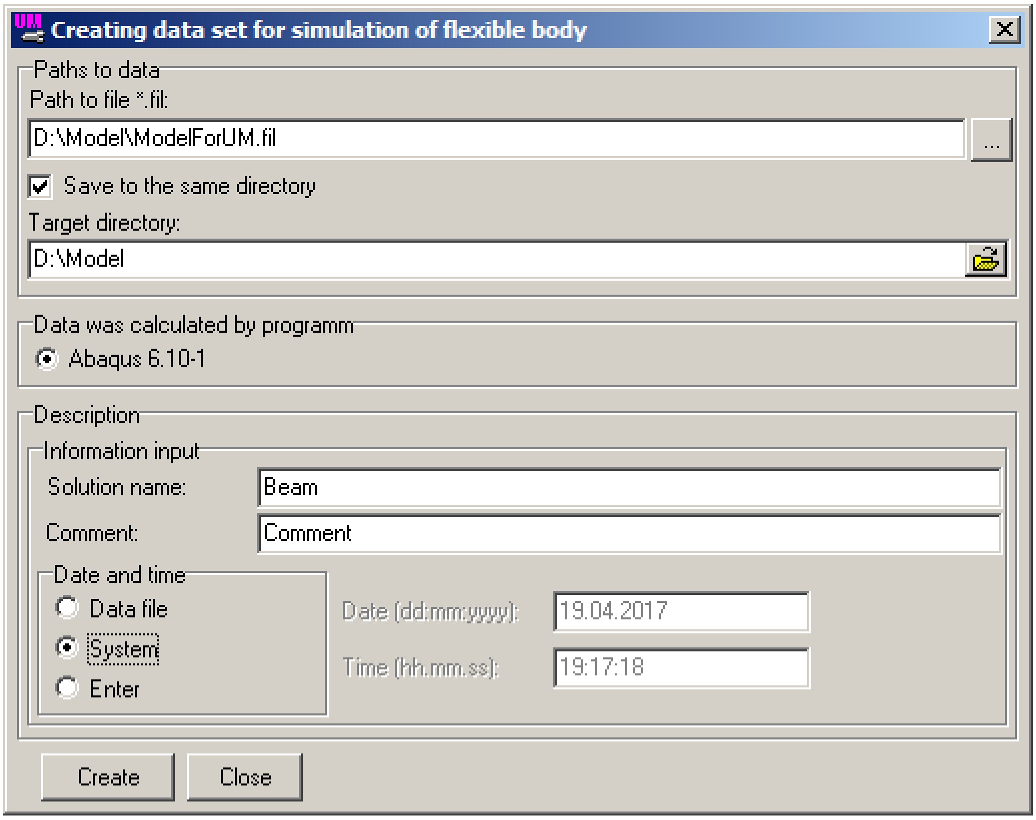


Figure 1.65. ABAQUS_UM.exe window

In the corresponding dialog box fields you can set the solution name, the comment and the date of calculation. These data are attributes of the **UM** model which are being written to the input.fum file.

Use the **Create** button to perform the data conversion. If it was successfully, the file input.fum will be created in the directory specified in the **Target directory**. Further work with this file is described below in Sect. 1.3 "Wizard of flexible subsystems", p. 1-82.

1.2.5. Model creation in FIDESYS program and data exchange

1.2.5.1. General information

Figure 1.66 presents a full cycle of preparation of initial data using the **FIDESYS** program and model analysis in **UM**.

The elastic subsystem is created based on the superelement method. After developing the finite element model, user must select the interface nodes and create a superelement. The necessary data is imported during the modal analysis of the superelement.

Next, let us describe the rules for preparing source data in **FIDESYS**, as well as the order of using software to import data into a Universal mechanism.

A step-by-step description of the development and analysis of a model that includes an imported elastic subsystem is given in the manual "Starting to work in the Universal Mechanism software package: elastic body modeling module".

1.2.5.1.1. The composition of the software, the scheme of import from the FIDESYS software

UM software provides a converter program **FIDESYS_UM.EXE**, which reads **geometry.vtk**, **res.cbm**, **M_CCS.hb**, **K_CCS.hb** files and generates **input.fum** file, that is, saves the data in UM format.

FIDESYS_UM.EXE converter program after installing the Universal mechanism software is located in the directory .\bin. Next, we will describe in detail the preparation of the data.

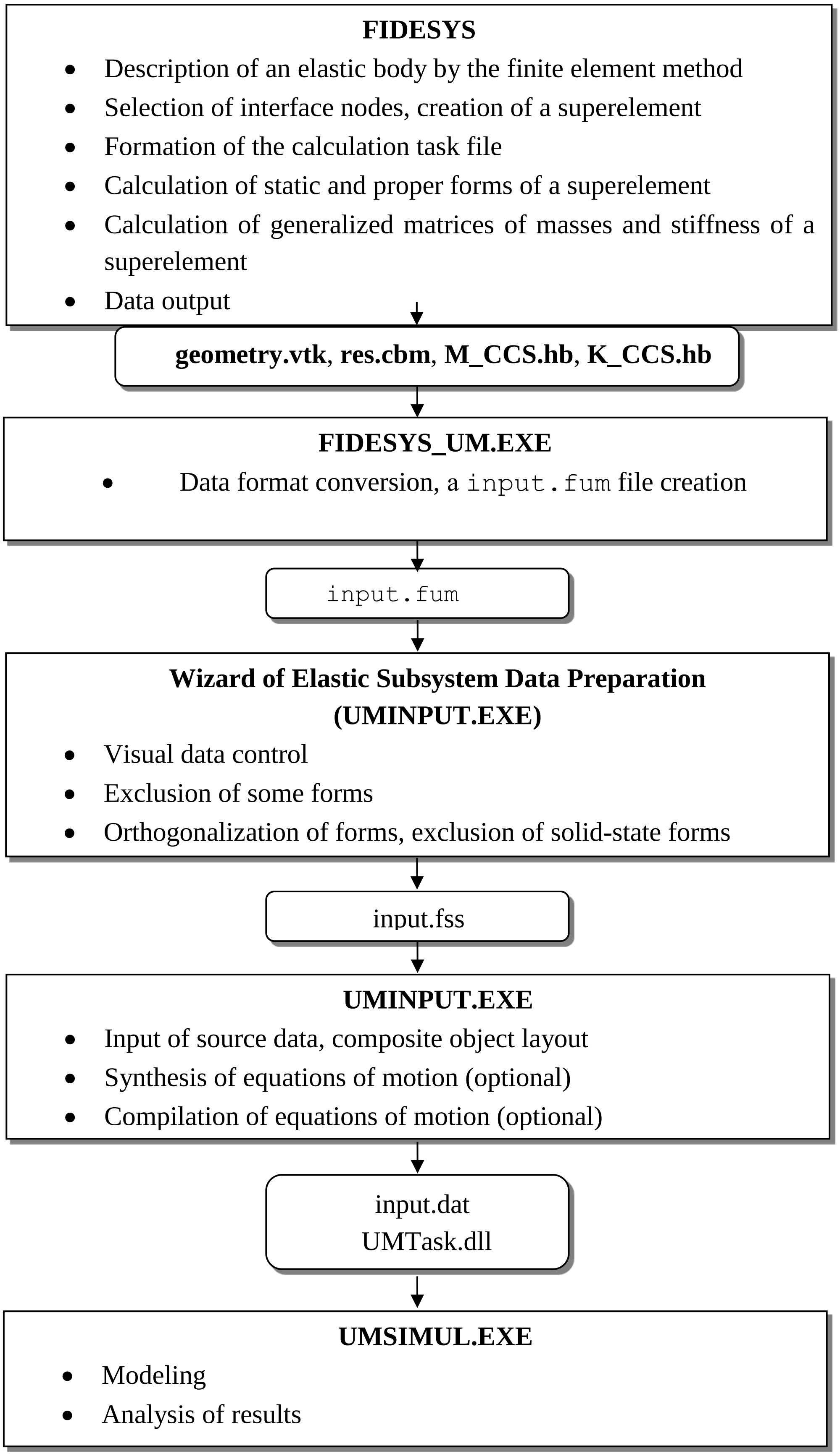


Figure 1.66. Creating an elastic subsystem using FIDESYS

1.2.5.1.2. The main stages of creating an elastic model in the FIDESYS software

The **FIDESYS** User Manual is available for downloading on the program's website[2]. The model must be described in the SI system of units, and this must also be taken into account when specifying the elastic properties of the material. The finite element mesh must contain nodes at the joint points and attachment points of the force elements. Some features of creating a finite element model are described in Sect. 1.2.6. "Some features related to the preparation of FE models", p. 1-72.

After creating the FE model, it is required to specify the interface nodes of the superelement used in the calculation. To do this, select **Mode - Set, Entity - Create nodeset, Action - Create nodeset** on the command panel. Enter **NodesetName**, enter numbers in the **ID(s)** field, or by activating this field, select nodes with the mouse in the animation window (for several nodes by pressing Ctrl). After selecting the necessary nodes, click **Apply** (Figure 1.67).

Figure 1.67

You can view the created node sets in the model tree (Figure 1.68).

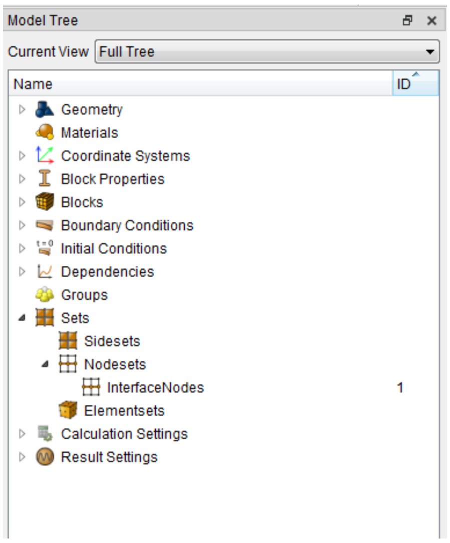


Figure 1.68. Model Tree

---

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

On the Command Panel we set the settings for the calculation: specify Number of eigenmodes, Output Format - binary (Figure 1.69). Click the **Apply** button, then **Start Calculation**.

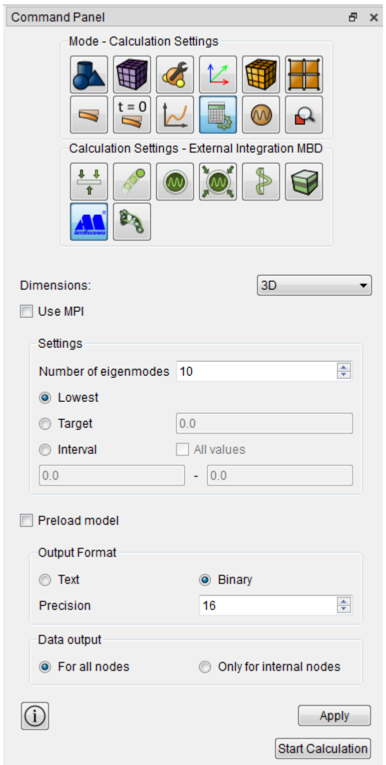


Figure 1.69

We set a name and choose a place to save the files with the results, select **Save** (Figure 1.70).

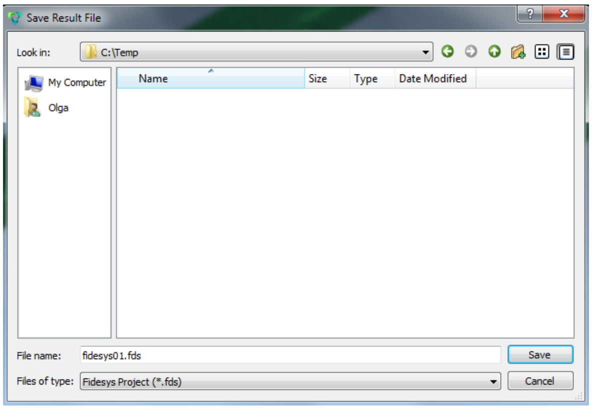


Figure 1.70

1.2.5.1.3. Lumped masses and rigid regions in FIDESYS

When preparing the FE model, it is necessary to provide six degrees of freedom in the interface node and the distribution of forces across its location. Six degrees of freedom in the interface node can be provided by adding a concentrated (nodal) mass to this node. For the correct distribution of forces around the interface node, it is necessary to build a rigid region around it.

In this chapter, we will show how to create concentrated masses in interface nodes and rigid regions in the **FIDESYS** software.

To add a lumped mass to a node, you must first create a block by including nodes in it (in which the lumped masses will be located), then specify the properties of the lumped mass for this block.

Choose **Mode - Blocks**, **Entity - Block**, **Action - Add**, **Entity List - Node**, in the **Entity ID(s)** field we specify the numbers of nodes in which the lumped masses will be located, press **Apply** (Figure 1.71). A block including 3 nodal masses has been created.

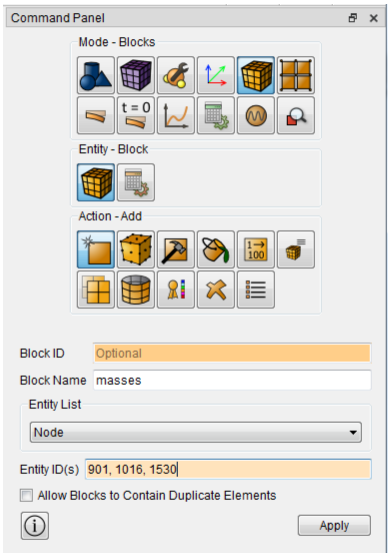


Figure 1.71

Next, for the created block, you need to set the properties of the lumped mass: **Mode - Blocks**, **Entity - Block**, **Action - Block properties/parameters**, choose the block created earlier, **Category - LumpMass**, press **Set Sphere Element Properties** (Figure 1.72). In the appeared window we set the masses and moments of inertia along the axes.

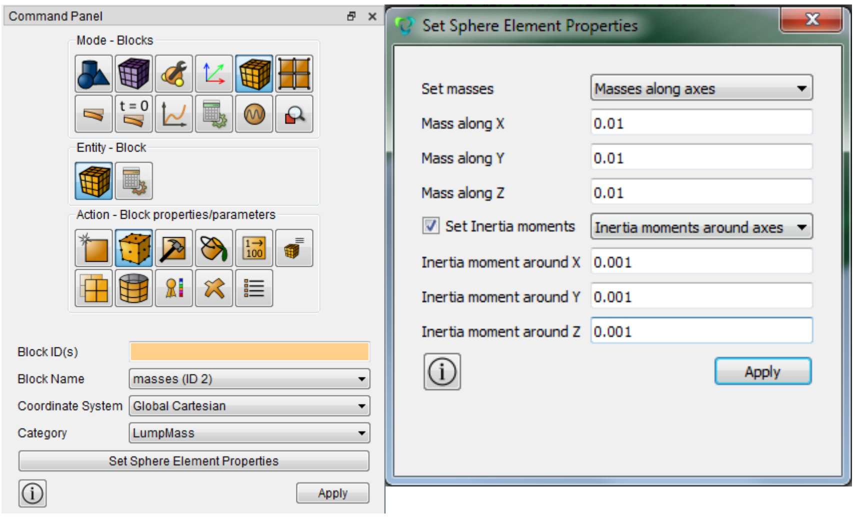


Figure 1.72

To create a rigid region, select **Mode - Boundary Conditions**, **Entity - Coupling Constraint**, **Action - Create**, from the drop-down **Entity List** (and Master Entity, and Slave Entity) choose **Node**, the in the field **Entity ID(s)** enter the number of one master node and several slave nodes (Figure 1.73).

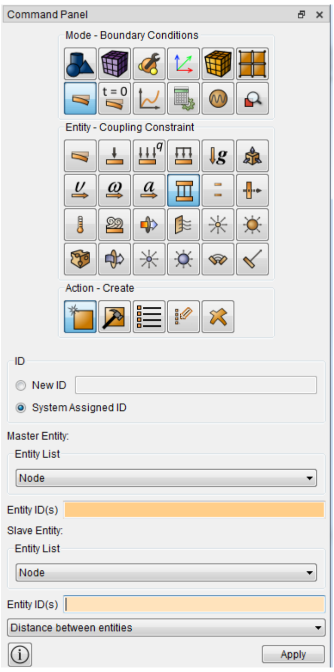


Figure 1.73

More detailed information about lumped (node) masses and rigid regions you may find in Sect. 1.2.6.1

1.2.5.1.3.1.1. FIDESYS UM Data Exchange

Run **FIDESYS_UM.EXE** converter program (Figure 1.74) to create **input.fum** file. Choose **geometry.vtk** file in the appropriate directory and specify the file save directory **input.fum**.

Note: **res.cbm**, **M_CCS.hb**, **K_CCS.hb files must be located in the same directory as the geometry.vtk file.** The assignment of all fields is clear by their names (Figure 1.74).

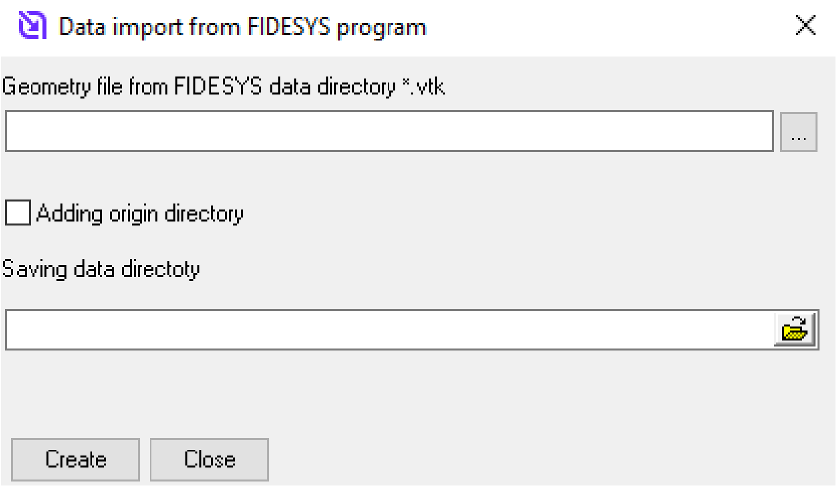


Figure 1.74. FIDESYS_UM.exe program window

Perform the data convertation by clicking **Create**. If successful, the **input.fum** file will be created in the directory specified in the **Directory to save data** field. Further work with this file is described below in Sect. 1.3. "Wizard of flexible subsystems" p. 1-82.

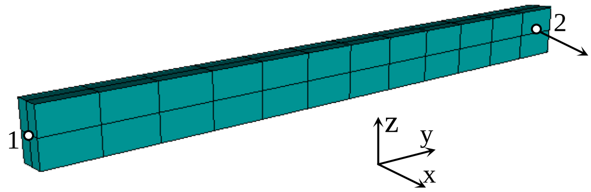
1.2.6. Some features related to preparing FE models

Let us consider several features that should be taken into account during preparing the finite element models of the flexible bodies.

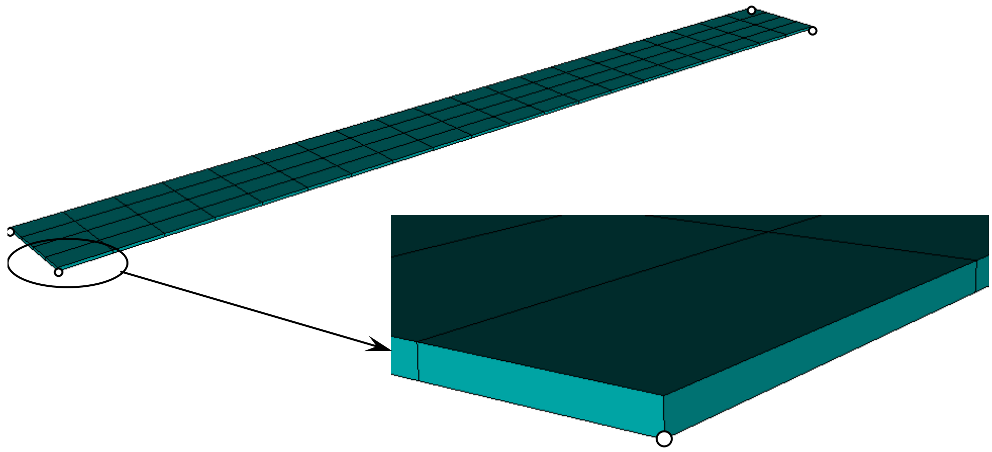
1.2.6.1. Selection of the interface nodes

The following rules should be taken into account for choosing interface nodes.

1. Interface node should be chosen so as to prevent motion of the flexible subsystem as a rigid body during calculation of static modes. Otherwise the model will be mathematically incorrect and cannot be described with the help of Craig-Bampton approach. Figure 1.75 illustrates incorrect (a) and correct (b) sets of the interface nodes.



(a) Incorrect set of interface nodes



(b) Correct set of interface nodes

Figure 1.75. Interface nodes. Positions of interface nodes are marked

Both models consist of 8-node solid elements that have 3 translational degrees of freedom in each node. In **ANSYS** the correspondent type of the finite elements is named as **SOLID45**, and in **NASTRAN**[3] it is named as **HEXA**.

Two interface nodes in Figure 1.75a do not provide immobility of the model as rigid body during calculation of static nodes for lateral displacements along *x* or *z* axes. For example, lateral displacement of node 2 along *x*-axis the flexible body can rotate around *z*-axis. There are no rotational degrees of freedom in the nodes, so fixing the node 1 does not prevent such a rotation.

Selection of four interface nodes for a plate in Figure 1.75b allows correctly calculating all static modes.

2. All applied forces, besides contact ones, can be applied in UM at the nodes of FE mesh only. To increase the simulation accuracy it is recommended to select all nodes, where any external force is applied to, as interface nodes. Influence of the applied force on the state of the flexible body directly depends on the presence of the correspondent degrees of freedom in the node. Let us consider the model that is shown in Figure 1.76. A plate consists of solid finite elements and interacts with the support via four linear force elements that are attached at the interface nodes. Lateral or longitudinal displacements of the plate lead to non-zero torques in the force element. Absence of the rotational degrees of freedom in the nodes of FE-mesh leads to ignoring arisen torque components during simulation. In other words, angular stiffness of that springs does not influence on simulation results. So the model will be *physically incorrect* if a real or designed prototype undergoes rather big torques. To simulate such a model correctly it needs to add some additional components into FE-model to provide 6 d.o.f. in each interface node. It will be considered in details below.

[3] Here and below **NASTRAN** means both supported **MSC.NASTRAN** and **NX NASTRAN**

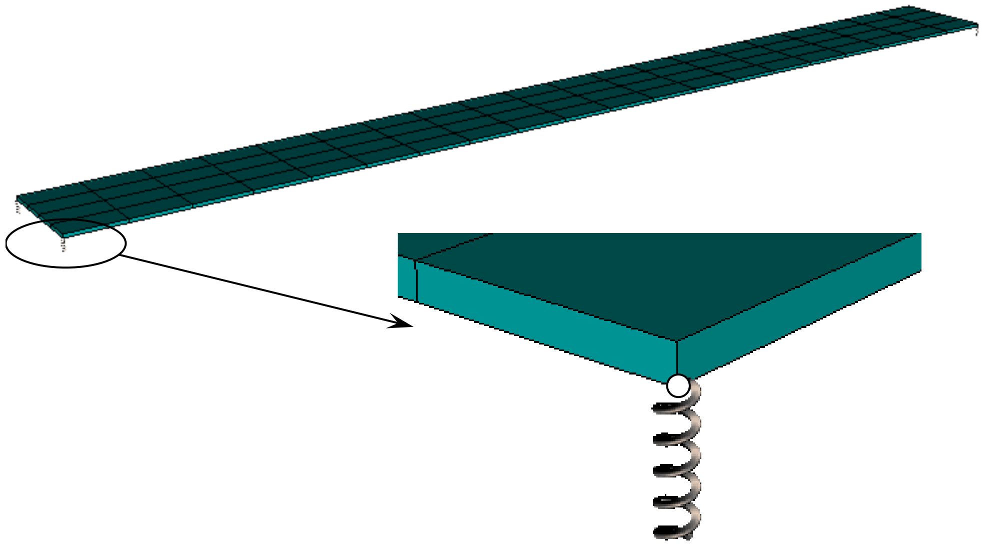


Figure 1.76. Sample of UM model with flexible subsystem

3. All that is written above about force elements is also directly related to reaction forces in joints. For example, if one fixes the plate in Figure 1.76 in two interface nodes at its corners, the plate will freely rotate around the axis that connects both nodes. In such a model each node can be considered as a spherical joint.
4. In the reality, any force that acts on the flexible body is transmitted via some area around the ideal attachment point. It means that it would be more realistic model if the force is distributed among several neighboring nodes in comparison with concentrated force application.
5. If the interface node does not have 6 d.o.f., it normally means that constraint equations cannot be generated correctly or even cannot be generated at all.
6. Selection of closely situated nodes as interface ones leads to high frequencies in the final solution.

Let us consider some steps that will help us to provide 6 d.o.f. in the interface nodes and to distribute a force via some area.

1. Adding mass point and constraints.

Six degrees of freedom is introduced with the help of a mass point finite element with non zero moments of inertia. It is **MASS21** element in **ANSYS** (keyopt(3)=0) and **CONM2** in **NASTRAN**.

In the **ABAQUS** program to generate the nodal mass you can use the command in the main menu **Special** → **Inertia** → **Create** of the **Property** module (Figure 1.77). In the appeared **Create Inertia** window select the type **Point Mass/Inertia**, click **Continue**, select using the mouse in the graphics window the points, in which the nodal masses will be, or select a set of nodes, which contains nodes in which the nodal masses will be. Then in the appeared **Edit Inertia** window (Figure 1.78) specify the inertia parameters of the nodal mass and click **OK**.

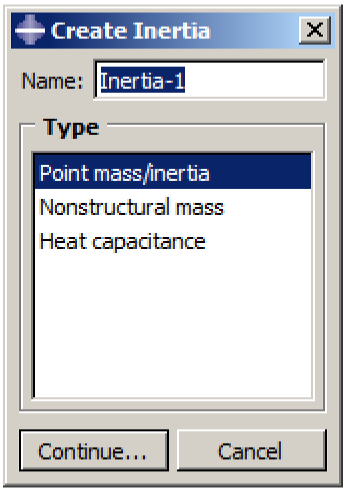


Figure 1.77. Creating nodal mass in **ABAQUS**

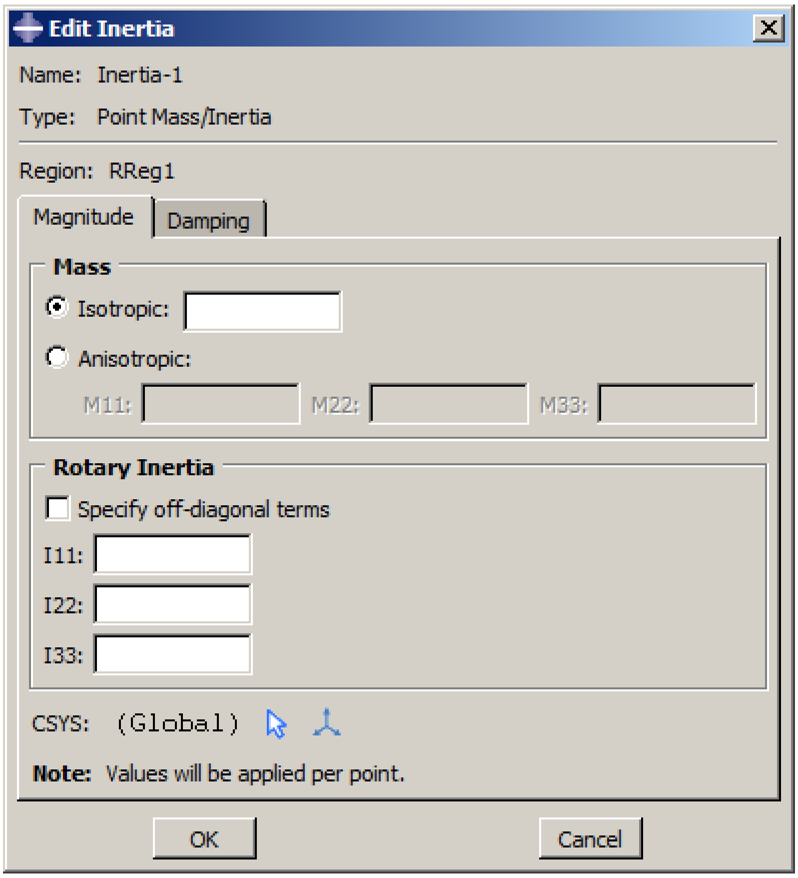


Figure 1.78. Edit Inertia window

Such a mass point should have very small inertia parameters relatively to inertia of the whole flexible body. After adding the mass point element to the FE-mesh, angular degrees of freedom in the mass point are not connected with the rest degrees of freedom of the flexible body. In order to interconnect angular degrees of freedom with the translational ones the constraint equations should be added to the model.

ANSYS

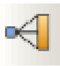
Use the following menu command

Preprocessor > Coupling/Ceqn > Rigid Region or command

CERIG, MASTER, SLAVE, UXYZ,

where **MASTER** is the number of the interface node, **SLAVE** is the node that degrees of freedom are eliminated, **UXYZ** means that all translational degrees of freedom are eliminated. Use **SLAVE=ALL** to eliminate degrees of freedom for all selected nodes. One can also use **CE** command to create constraint equations. Using the constraint equations are fully described in ANSYS user's manual.

ABAQUS

In the **ABAQUS** program it is possible to use the **Constraints**  button from the **Interaction** module. When you click **Create Constraint**, dialog window **Create Constraint** appears (Figure 1.79) in which you have to select the **MPC Constraint** type and click **Continue**. Then you must select an interface node or set of nodes, including the interface node, and then select the dependent nodes or set of nodes, including the dependent nodes. Finally, the **Edit Constraint** window (Figure 1.80) should appear in which you have to choose a **MPC Type** equal to **Beam** and click **OK**.

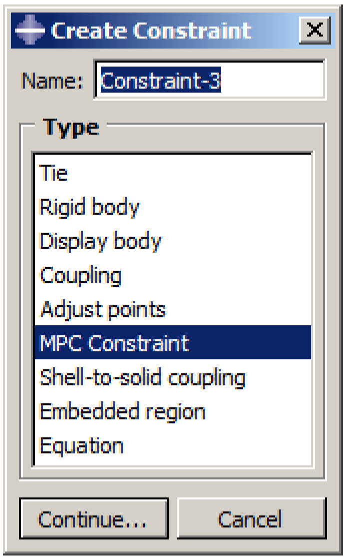


Figure 1.79. Create Constraint dialog window

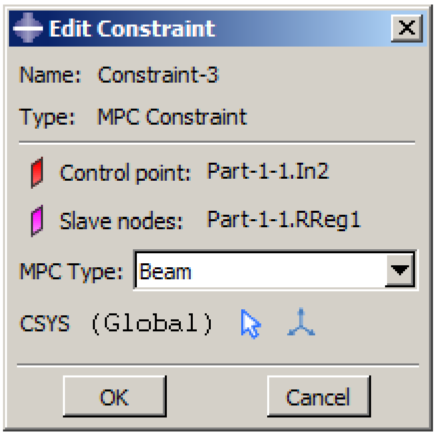


Figure 1.80. Edit Constraint dialog window

NASTRAN

Constraint equations in NASTRAN are introduced by RBE2 element

```
RBE2 EID GN CM GM1 GM2 GM3 …,
```

where **EID** is the element number, **GN** is the number of node with independent degrees of freedom, **CM** are numbers of eliminating degrees of freedom (1 to 6 without spaces), **GMi** are numbers of nodes whose degrees of freedom are eliminating. For example, the following line in NASTRAN

```
RBE2     163     62      123456  61      103     185
```

introduces constraints equations with number 163, 62 is the number of the independent node, with which nodes 61, 103, 185 are rigidly connected.

To add such an element in MSC.PATRAN one can click **Elements** button in the tool panel, see Figure 1.81. Use the following parameters in the dialog window:

Action: Create

Object: MPC (Multipoint Constraints)

Type: RBE2

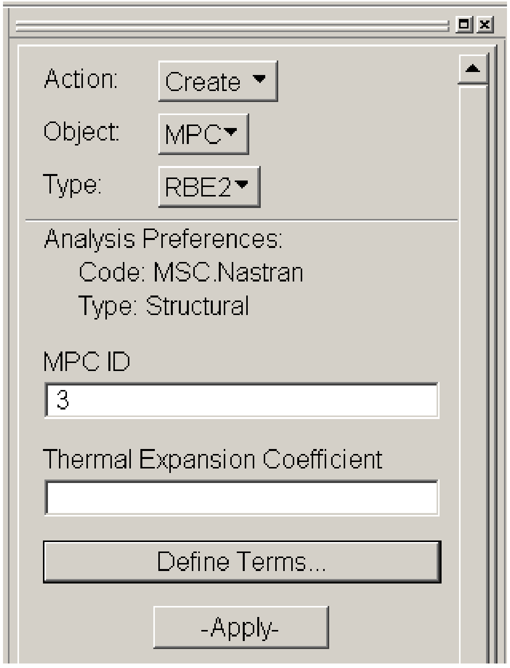


Figure 1.81. Constraints in MSC.PATRAN

Click **Define Terms…** to define constrained nodes. Hereby the rigid region with 6 d.o.f. is created.

Let us consider one more case of using constraint equations for simulation of the crank-and-rod mechanism (Figure 1.82) with a flexible rod (Figure 1.83).

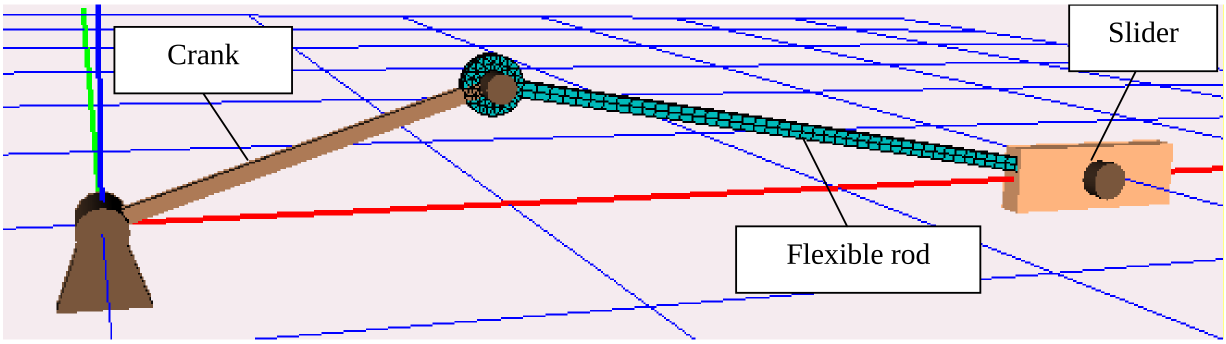


Figure 1.82. Crank-and-rod mechanism with a flexible rod

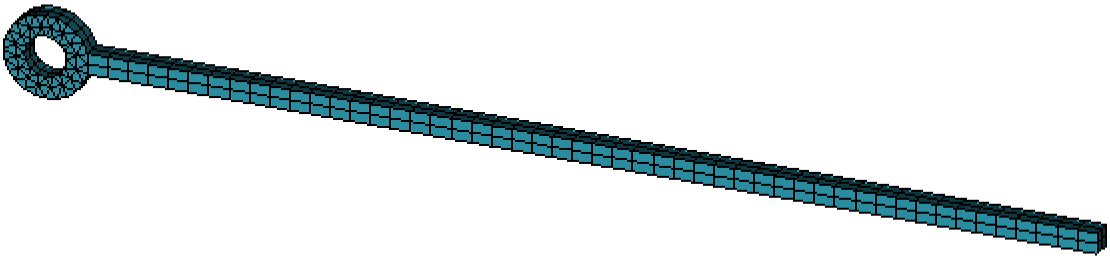


Figure 1.83. FE-mesh of the rod

Two interface nodes should be defined for the rod. The first interface node should be placed in the center of the cross section at the end where the rod connects the slider, Figure 1.84. The second interface node should be placed on the rotation axis in the center of the circular hole at another end of the rod, Figure 1.85.

1. Adding the mass point and constraint equations

As it already described above, let us add the mass point in the first interface node and then add constraint equations to create a rigid region that involves all nodes that lie at the end of the rod, Figure 1.84.

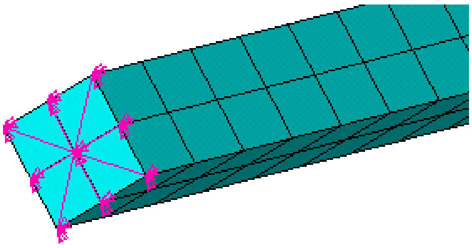


Figure 1.84. Constraint equations at the end of the rod

Then the following steps should be done at the other end of the rod, Figure 1.85:

- add the node at the rotation axis;
- add the mass point at the rotation axis;
- add constraint equations between the central node and all nodes at the inner cylindrical surface.

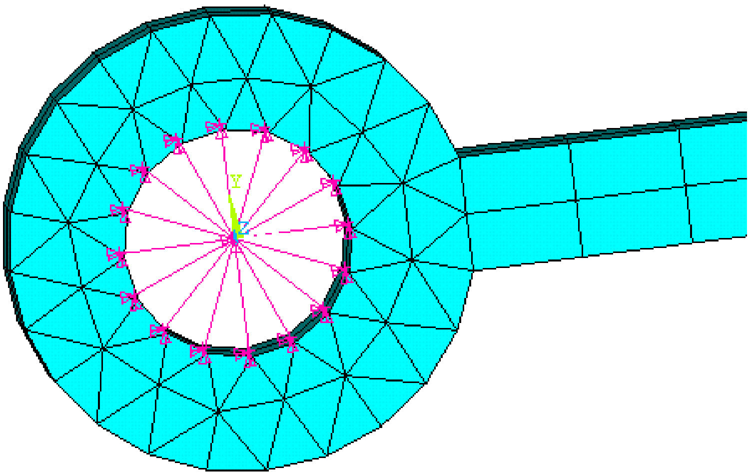


Figure 1.85. Constraint equations at the circular hole

2. Adding beam elements.

There is an alternative approach to correctly introduce interface nodes for the rod. This approach assumes creating beam elements to introduce 6 d.o.f. in the interface node and transfer the applied force though several nodes. In ANSYS one can use, for example, elements of BEAM4 type.

It is recommended that beams should be of rather small mass and rather high stiffness. Otherwise eigenmodes and frequencies might be changed. Required beam parameters are achieved by using special characteristics of material and the cross section.

If you use beam elements you do not need to create a mass point in the interface node since every node of a beam element has 6 d.o.f.

Using beam elements are shown in Figure 1.86, Figure 1.87.

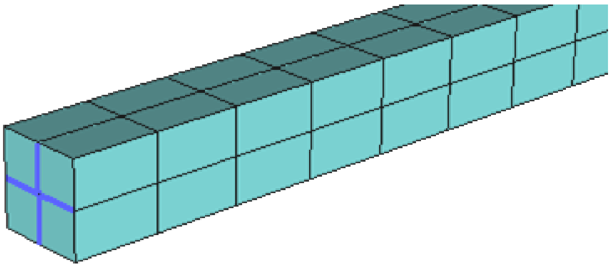


Figure 1.86. Beam elements at the end of the rod

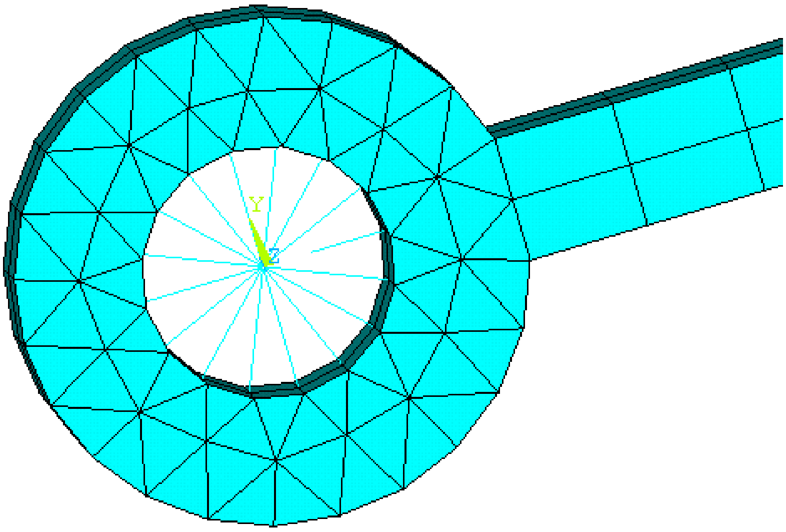


Figure 1.87. Beam elements in the circular hole

1.2.6.2. Normals for shell elements

External FE software computes and saves calculation results including data for calculation of stress and strains to intermediate files and then the specific convertor creates input.fum or input.fss files, see Sect. 1.2.1. *"Exporting finite element model from ANSYS"*, p. 1-9, and Sect. 1.2.2. "*Exporting finite element model from MSC.NASTRAN*", p. 1-26. Count of stress components in the node is explicitly determined by the type of the finite element. In some cases the count of stress components depends on the program settings. For example, **ANSYS** SHELL63 element has two stress components on the top and bottom surfaces at *keyopt(11)=0*; the same element has three stress components: on top, bottom and middle surfaces at *keyopt(11)=2*.

UM calculates nodal stress as a mean stresses among all elements containing the target node. However UM does not control normals to elements. To get correct stress estimation for shell elements the use should be sure that all shell elements around the inspected node have the same normal direction. In other words, top and bottom surfaces for the shell elements should coincide. It should be checked directly in FE software you use.

Let us consider the following example of calculation of average stresses on the top surface of the plate, Figure 1.88. The plate is developed in **ANSYS** and consists of SHELL63 finite elements. Top surface is defined by a normal direction. Normal to the selected element has (0; 0; **-1**) components. Whereas normals to other elements have the opposite direction (0; 0; **1**). Correct average stress estimation in node 77 supposes calculation of stresses at top surface of elements 1, 2, 3 and bottom surface of the element 3 and vice-versa.

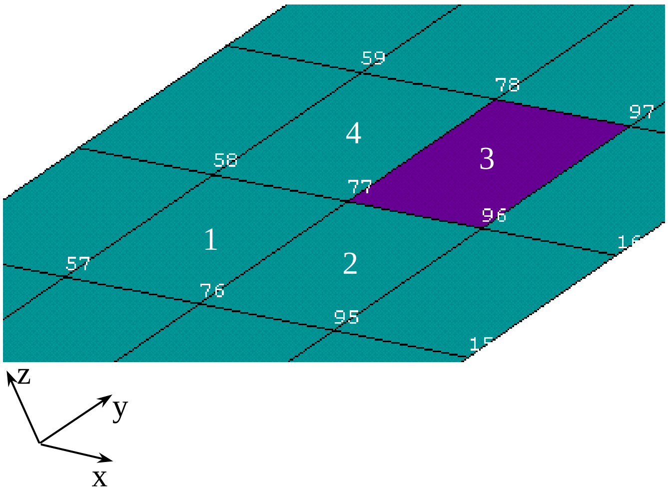


Figure 1.88. On calculation of mean stresses on surfaces of shell elements

The situation considered above is not checked by UM automatically. User should check normal directions at the stage of the model development in external FE software products (ANSYS, MSC.NASTRAN, NX NASTRAN).

You can inverse the normal direction in **ANSYS** using the following menu command *Preprocessor > Modeling > Move / Modify >*
Reverse Normals > of Shell Elements or use ENSYM command.

1.3. Wizard of flexible subsystems

Wizard of flexible subsystems (Figure 1.89) 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 modes from the final solution as well as for orthonormalization of modes and excluding rigid body modes. Functions of the **wizard** are quite similar to program-convertors **ANSYS_UM** and **NASTRAN_UM**.

Use menu command **UM Input / Tools / Wizard of flexible subsystems** to open the window of the wizard.

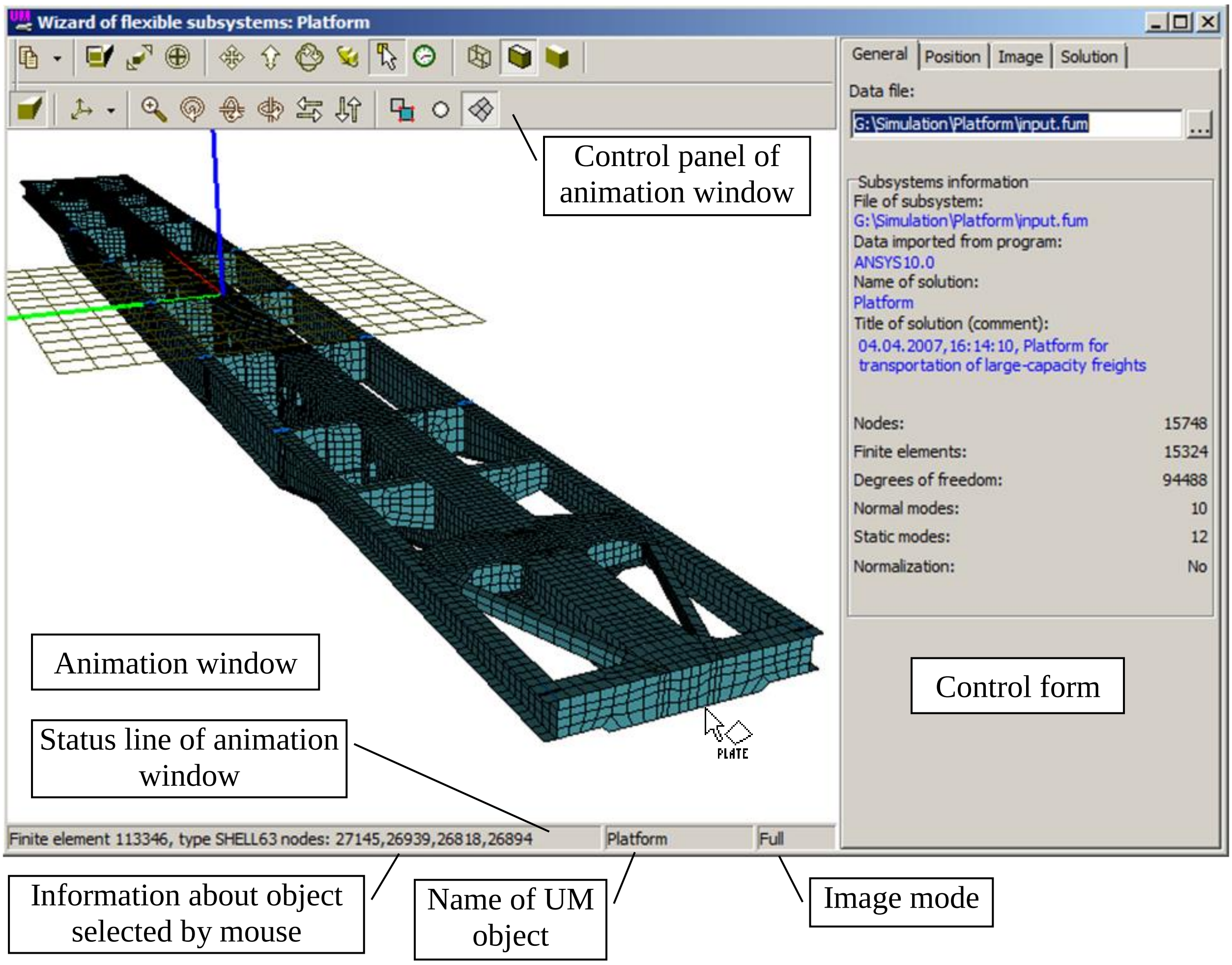


Figure 1.89. Wizard of flexible subsystems

1.3.1. 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. 1.3.1. *"Animation window"*, p. 1-83. Here we will discuss additional functions of the animation window concerning visualization of the FE-mesh.

Mouse cursors:

- node of the FE-mesh;
- one-dimensional finite element: beam, link, pipe;
- finite element of shell or plate type;
- solid finite element.

Status line shows information about current selected node or element.

Additional buttons that affect for FE-mesh visualization:

- full/simplified FE-mesh (features of full/simplified will be discussed later);
- show/hide nodes;
- show/hide finite elements.

Besides standard buttons animation window has additional positioning buttons:

- zoom in/out;
- , , rotating around corresponding axis;
- , shift left/right, up/down.

Use the context menu to hide/show tool panels, see Figure 1.90.

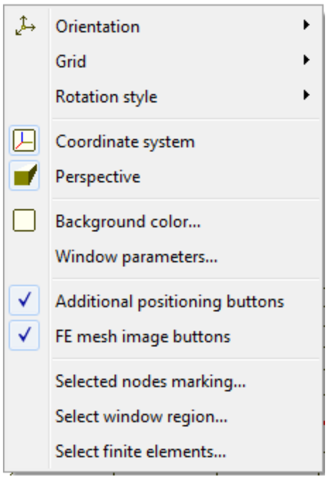


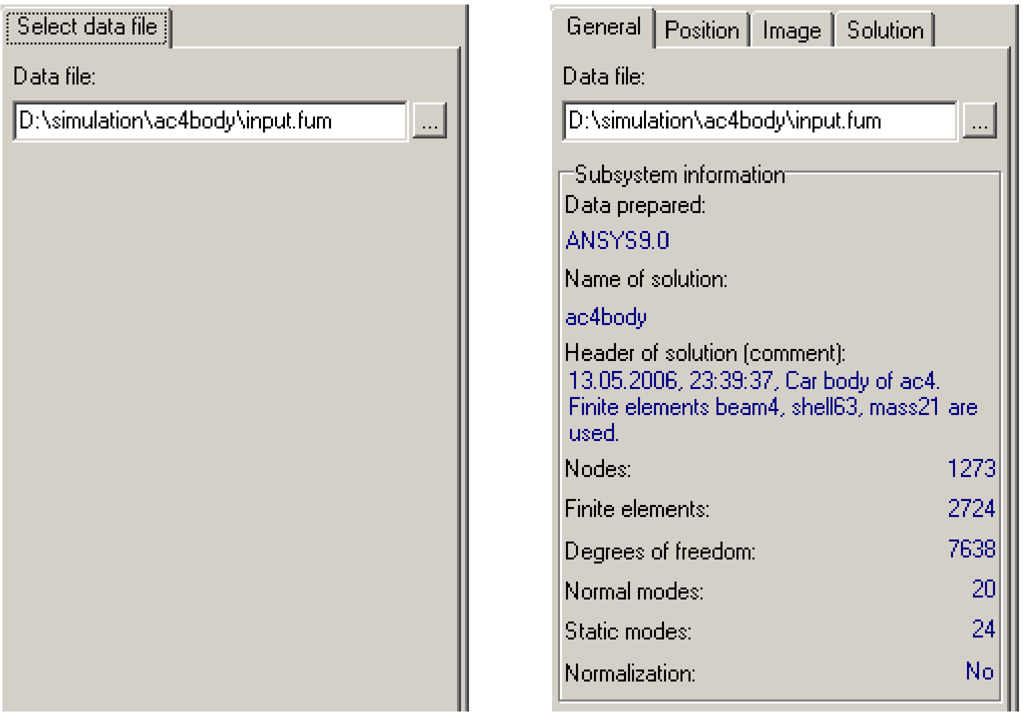
Figure 1.90.

1.3.2. Control panel

Control panel 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.

1.3.2.1. General tab



a) b)

Figure 1.91. General tab

When you just started **Wizard of flexible subsystems** it has the only tab **"Select data file"**, see Figure 1.91a. **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 Figure 1.92.

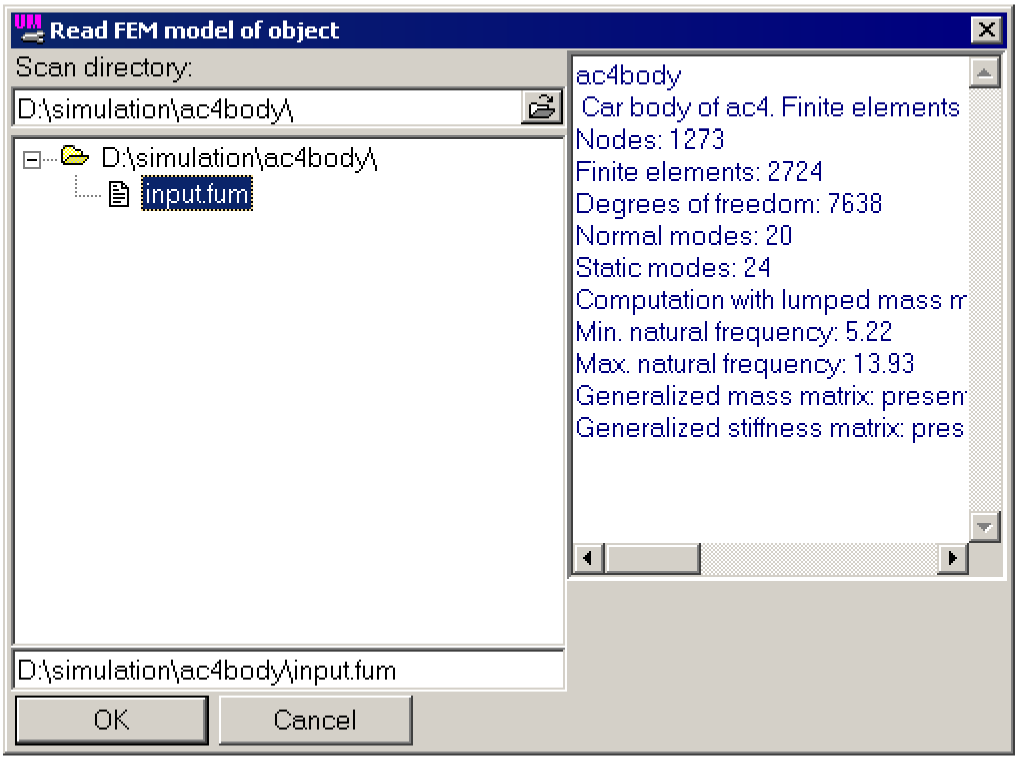


Figure 1.92.

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 Figure 1.91b.

1.3.2.2. Solution tab

Let us consider basic features of the **Solution** tab, see Figure 1.93.

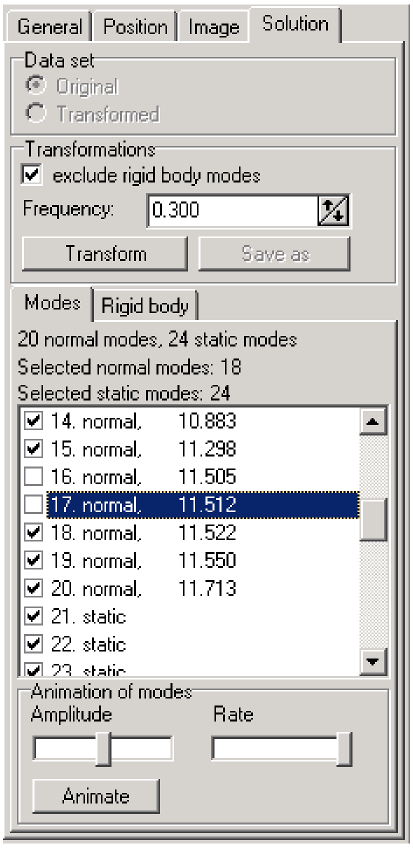


Figure 1.93.

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 Figure 1.94) shows information about position of center of gravity, mass and moments of inertia of the flexible subsystem.

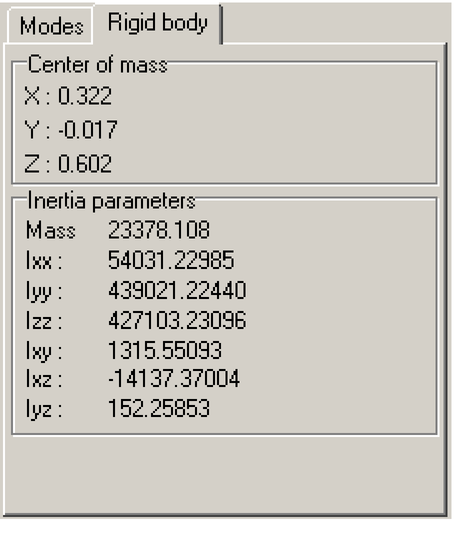


Figure 1.94.

- 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. 1.2.1.2. *"Creating stress and strain sensors"*, p. 1-14.
  - The **Save as** button is aimed for saving the orthonormalized set of modes. Input the full path to the subsystem in the save dialog (Figure 1.95).

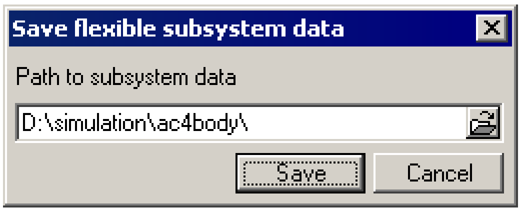


Figure 1.95.

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.

1.3.2.3. Image tab

This tab is intended for describing the graphical image of the FE-mesh of the flexible subsystem in an animation window, see Figure 1.96.

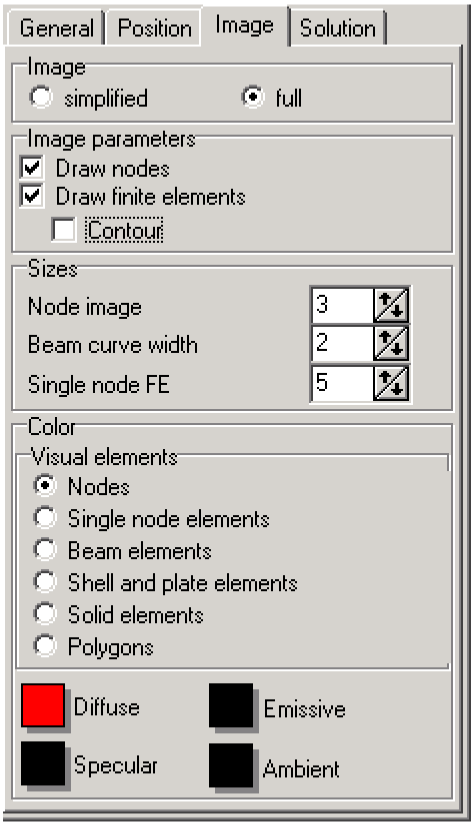


Figure 1.96.

- 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.

1.3.2.4. 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.

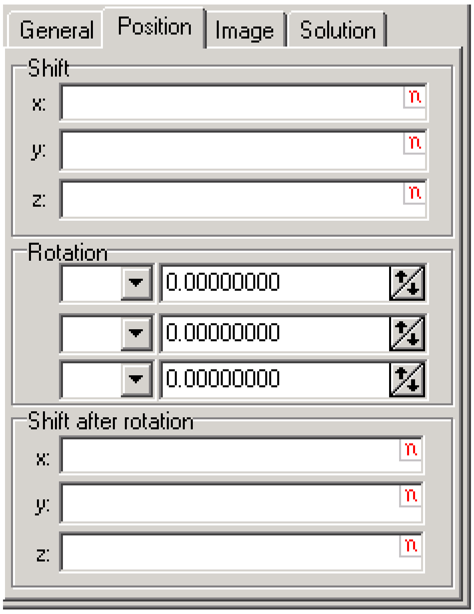


Figure 1.97.

1.4. New wizard of flexible subsystems – UM FEM Wizard

Starting from the 10 version **Universal Mechanism software** includes a new wizard of elastic subsystems **UM FEM Wizard**, which is implemented as a separate application. The executable file **UMFEMWizard.exe** is located in the **UM** executable directory (by default it is *C:\Program Files\UM Software Lab\Universal Mechanism\10\bin*). The master can be launched from the **UM Input** main menu**, Tools - Wizard of elastic subsystems (new),** similar to the previous version (Figure 1.98), as well as from a shortcut, file manager or command line. **UM FEM Wizard** does not interact with the **UM Input** program, the menu call is implemented simply as an additional feature.

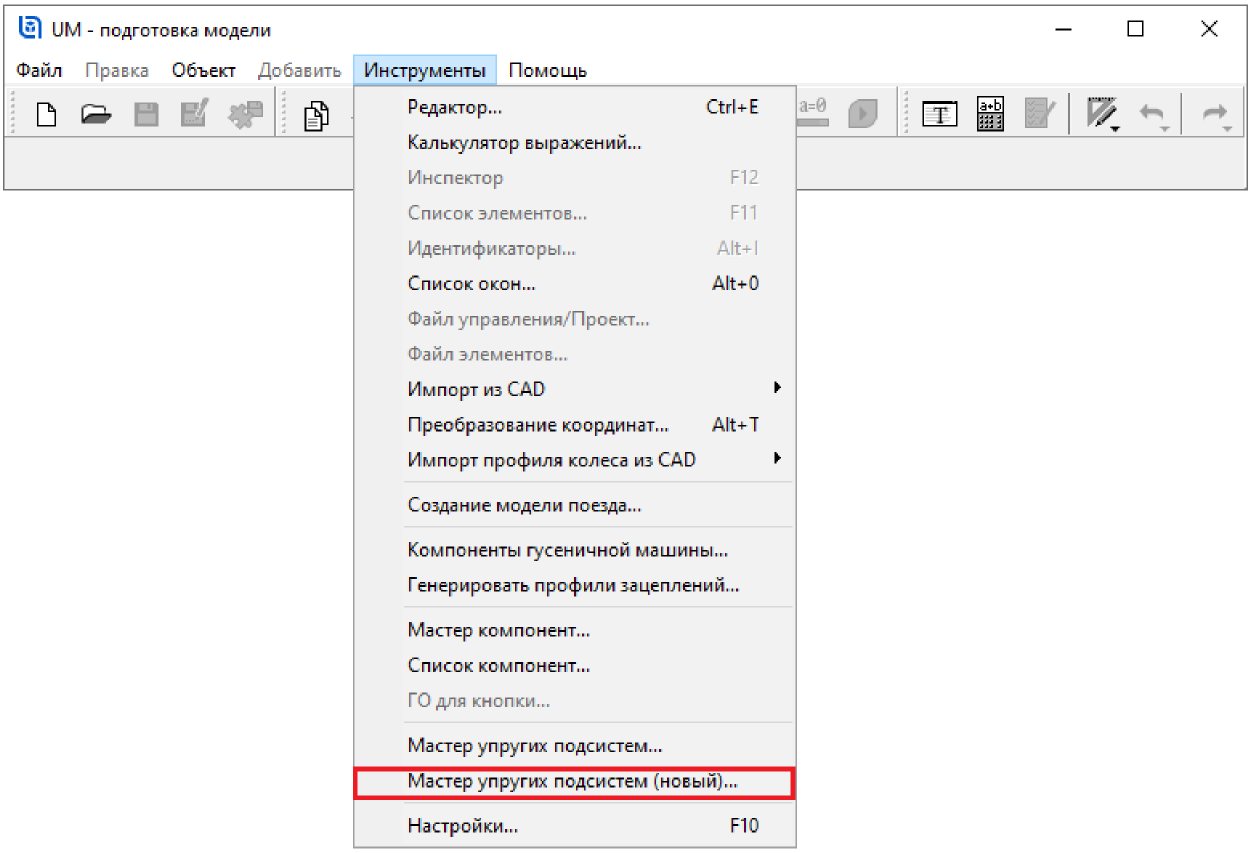


Figure 1.98

The view of the new **Wizard of flexible subsystems** window is shown in Figure 1.99. The main differences from the previous version are as follows.

1. The wizard contains a new animation window developed on the basis of a new more productive graphics core. Working with large models has become much more comfortable, the problems with mouse sensitivity - getting information about the object when pointing the cursor at it (that occurred when using the previous version of the window) have been solved. The selection of multiple finite elements in the window has been implemented, which is used, in particular, for assigning properties to them.

2. The wizard contains a form for assigning properties to finite elements. This is necessary for dynamic stress calculations using the FE library (see Sect. 11.1.2.2.2), since importing FE properties from external programs is not always possible. In the current version only isotropic

materials are realized. Note that for correct calculation of stresses it is necessary to set properties that coincide with the properties of finite elements entered in the FE program before calculation of the superelement (before exporting to the UM).

These and other smaller changes to the **Wizard** are described below.

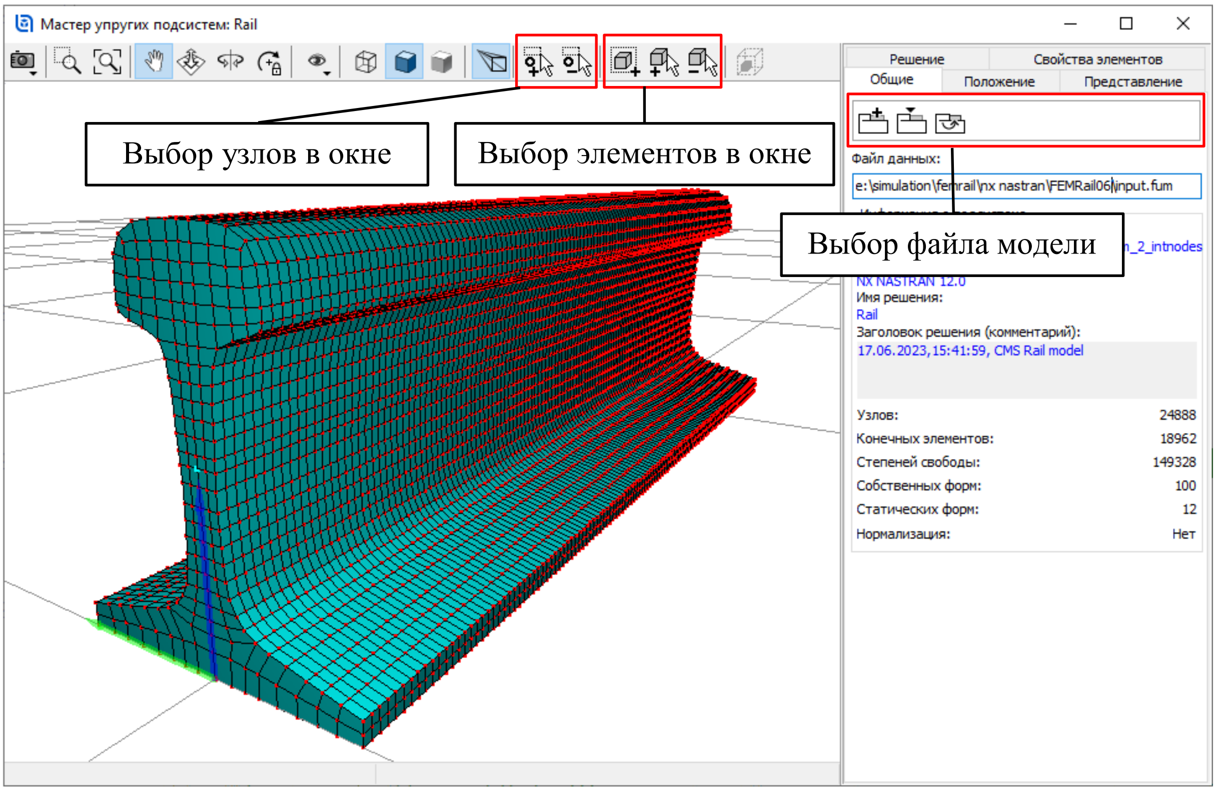


Figure 1.99. View of the new **Wizard of flexible subsystems** window

1.4.1. Selecting data file

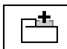
The **wizard** can load files of three types.

input.fum – source file of the flexible subsystem. In addition to the transformations described above, it is possible to enter and save finite element properties for dynamic stress calculations.

input.fss – a file with transformed shapes of the elastic subsystem. It can also be supplemented with finite element properties.

**.mnf* – file of flexible subsystem in MSC.ADAMS program format. It is loaded only for visual control of initial data, including animation of flexible forms.

On the panel of data file selection there are three buttons, by pressing which you can select the file using different forms of dialog.

	Select a file using the Windows dialog box (Figure 1.100)
	Select a file from the recently opened files (Figure 1.101)
	Select a file using the UM dialog box, which directory tree includes only directories containing flexible subsystem files (Figure 1.102)

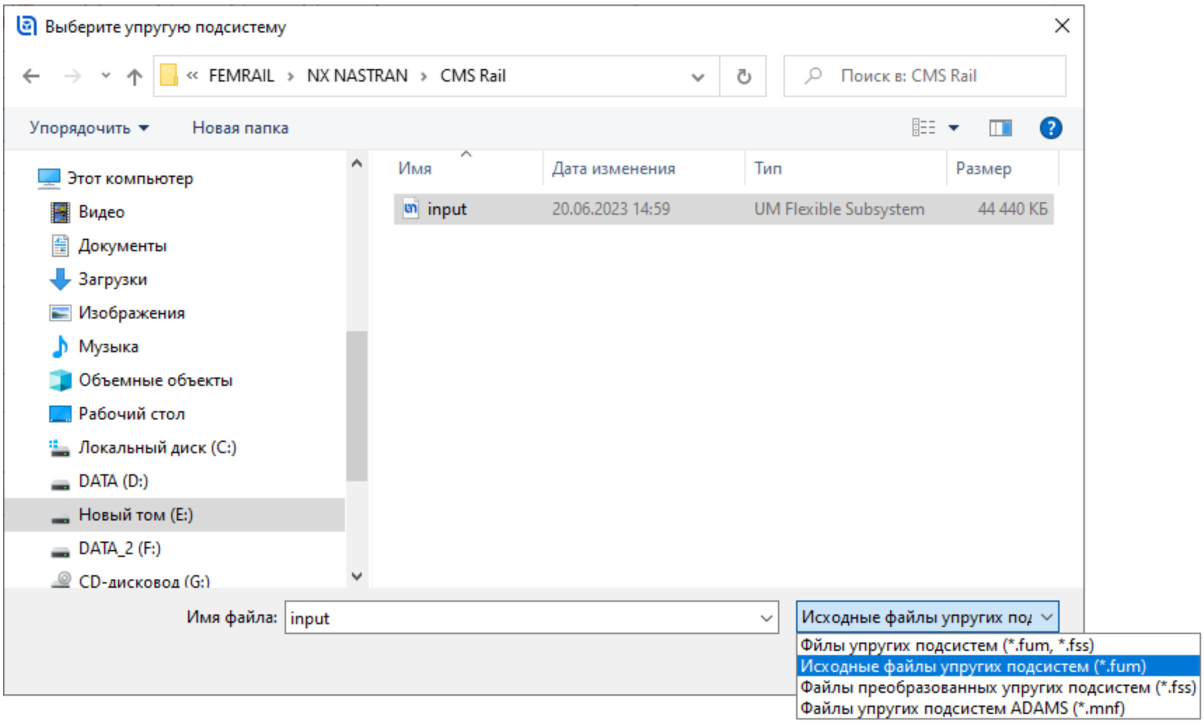


Figure 1.100. Selecting an flexible subsystem file using the Windows dialog box

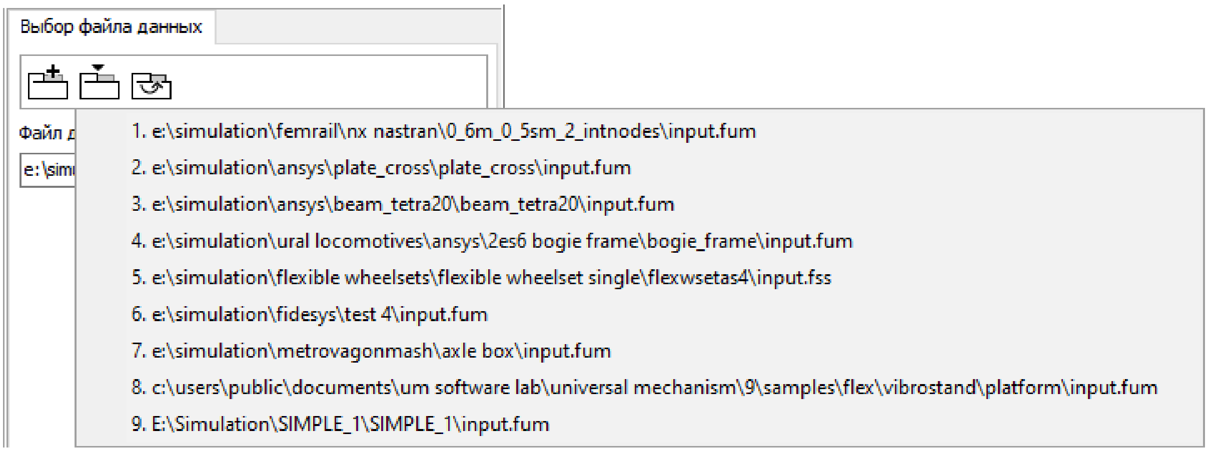


Figure 1.101. Select a file from the recently opened files

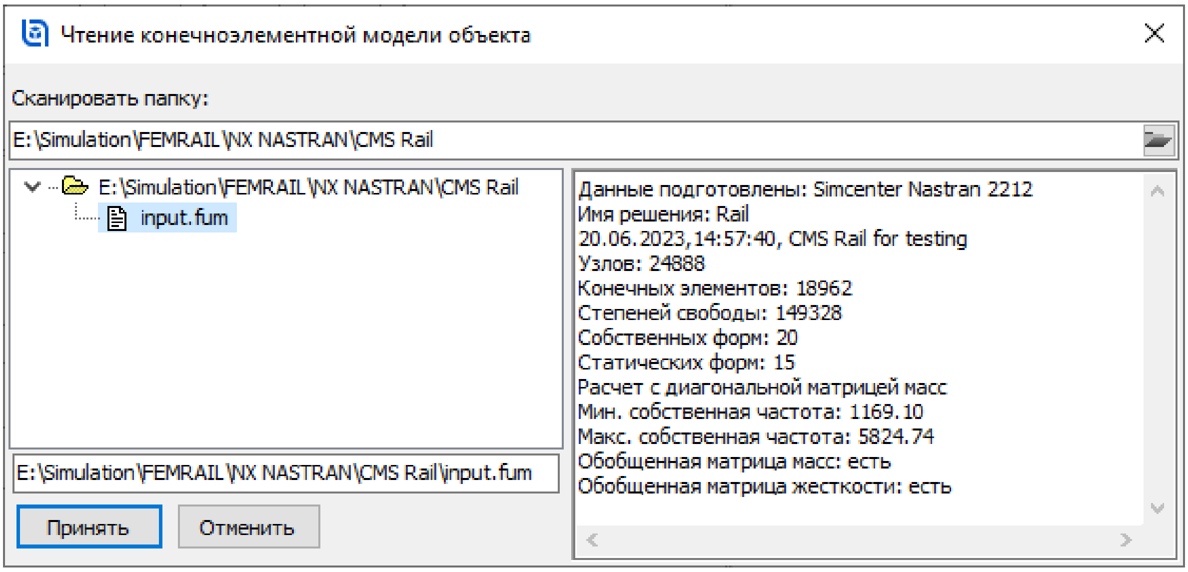


Figure 1.102. Selecting a file using the UM dialog box

1.4.2. Selecting objects in the animation window

In the new animation window, you can select multiple nodes and finite elements. This is done by activating the corresponding window mode in situations where this is required, e.g. from the node selection form (see Sect. 11.3.3) or the element selection form (Sect. 11.4.4).

1.4.2.1. Selection of nodes

Nodes can be selected one by one by clicking the mouse, or as a group using the frame. Lists of selected nodes are formed, for example, in the process of creating stress sensors in hazardous areas when calculating the loading and durability of the flexible subsystem.

Selecting nodes by frame is available if the following buttons are active.

 Select nodes with a frame.

 Unselect nodes with a frame.

To select by frame it is necessary to press the first button on the panel, press the left mouse button and, holding it, select a rectangular area on the screen, in which the selected nodes fall, and then release the mouse button (Figure 1.103). All nodes that fall into the selected area will be added to the list of selected nodes. The frame selection mode is deactivated after that, the button on the panel becomes inactive. For the next selection it is necessary to repeat all actions.

Similar actions after pressing the Cancel selection button lead to deleting of previously selected nodes in the frame area from the list.

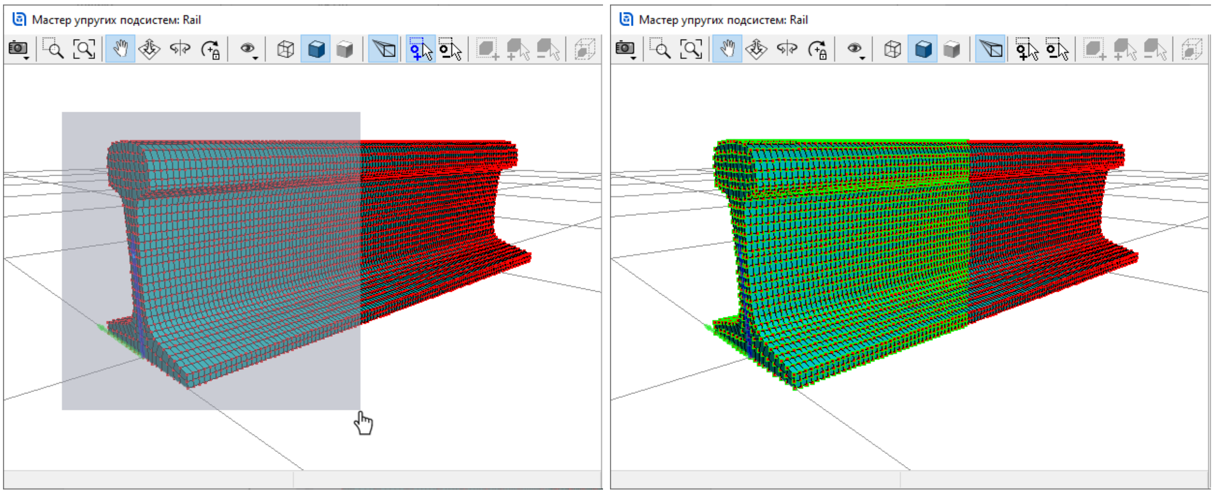
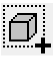


Figure 1.103. Selecting a group of nodes with a frame in the animation window (left) and displaying the selection result (right)

1.4.2.2. Selection of finite elements

You can select the finite elements one by one by mouse click, or in a group by using a frame. Finite element lists are created to assign properties to them, to create coloring variables if you do not want to color the entire pattern, and in other cases.

The selection is available when the following buttons are active.

 Highlight the finite elements with a frame.

 Select finite elements by clicking.

 Unselect finite elements by clicking.

To select with a frame it is necessary to press the button on the panel, press the left mouse button and, holding it, select a rectangular area on the screen, in which the selected elements fall, and then release the mouse button (Figure 1.104). The elements, all nodes of which fall into the selected area, will be added to the list. The frame selection mode is deactivated after that, the button on the panel becomes inactive. For the next selection it is necessary to repeat all actions.

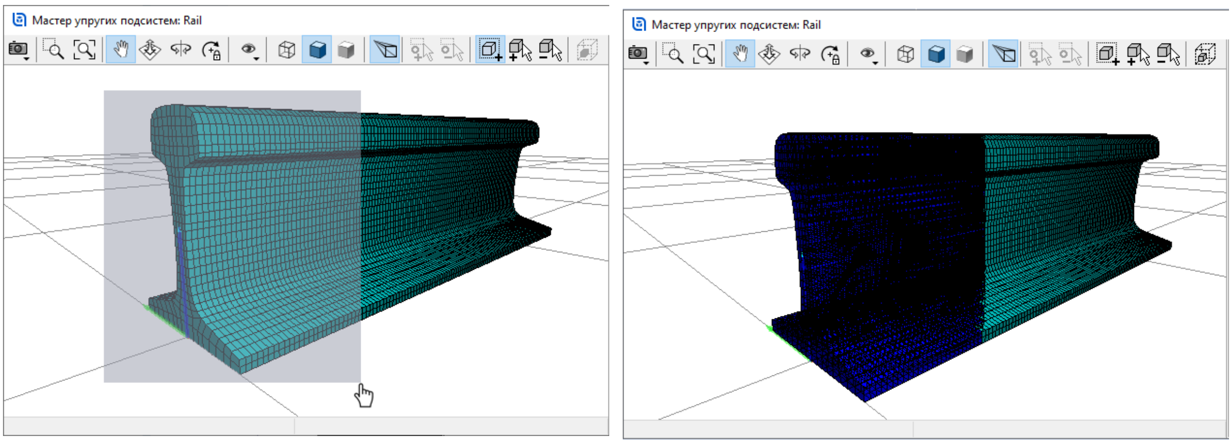


Figure 1.104. Selecting a group of finite elements with a frame in the animation window (left) and displaying the selection result (right).

The modes of selecting and deselecting items by clicking the corresponding button on the panel, after which you can add or remove an item from the list by clicking on its image. The modes are deactivated by pressing the button on the panel again.

The mode of canceling selection by frame has not been realized yet.

1.4.3. Entering finite element properties

To calculate dynamic stresses using the **UM** finite element library (see Sect. **Ошибка! Источник ссылки не найден.**), it is necessary to assign properties to the elements, since importing properties is most often not available. The following properties can be entered in **UM 2023** (10):

- isotropic material for volumetric (three-dimensional) finite elements;
- isotropic material and constant thickness for plate and shell elements.

If the FE model includes anisotropic materials, the stress calculation in **UM** will be incorrect.

Properties are entered using the tab **Element Properties** of the control window for groups of elements, which can include from one to all elements of the subsystem, (Figure 1.105). The panel at the top of the tab contains the following buttons for entering and editing properties.

 add property;

 delete property;

 copy property;

 add material from the directory - calls the **Material Directory (**Figure 1.106**);**

 edit list of elements - opens the window for editing the list of finite elements (Figure 1.108).

 read properties from the **.prp* file, in the current solution properties are assigned by the finite element number;

 write the properties to the ****.prp*** file;

 write down an flexible model with modified properties.

Comments.

1. UM software contains an editable material directory from which one of the elements in the list can be selected and assigned as a property to a finite element group. Detailed description of working with the directory is given in Sect. 13.3.1.3.1 of the User Manual for the fatigue life calculation module.

2. Working with the edit window of the finite element list is described below in Sect. 1.4.4.

3. Writing and reading properties is implemented to quickly assign them to the same or similar subsystems. After entering properties to one subsystem, save them to the **.prp* file (properties), then open a similar subsystem with unassigned properties, go to the **Element Properties** tab and read the previously written file.

4. The model with changed properties is written to the initial file, before that a backup file *input.fum.N* is created, where *N* is the sequence number of the copy, for example, *input.fum.1*.

5. The property name is edited after double-clicking on the corresponding item in the **Item Properties** list.

6. If the entire subsystem has the same properties, there is no need to form a finite element group, the **All elements** checkbox should be enabled.

7. The group can contain elements of different types, the **Plate thickness** parameter for volume elements is ignored.

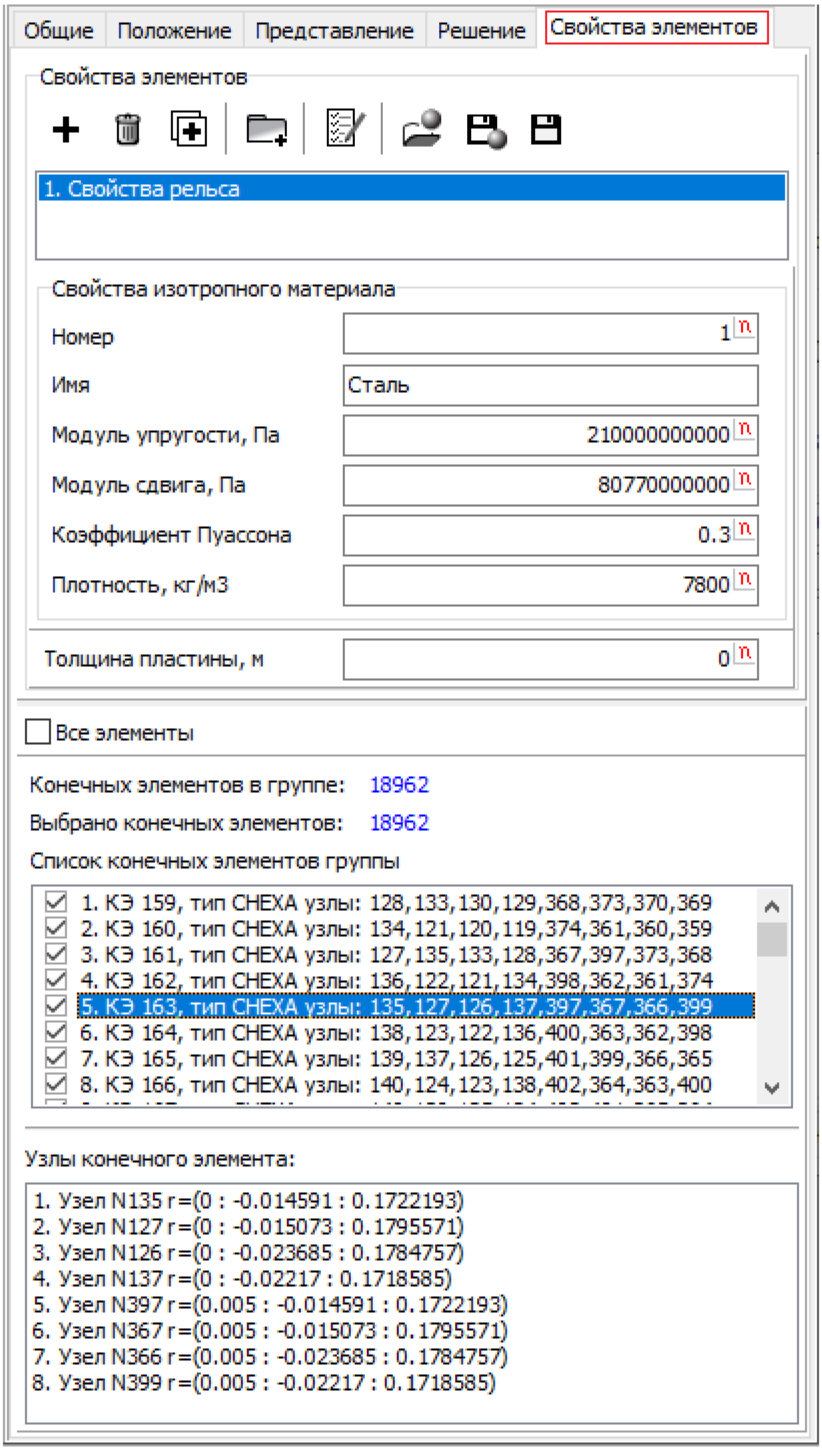


Figure 1.105. Control window **element properties** tab

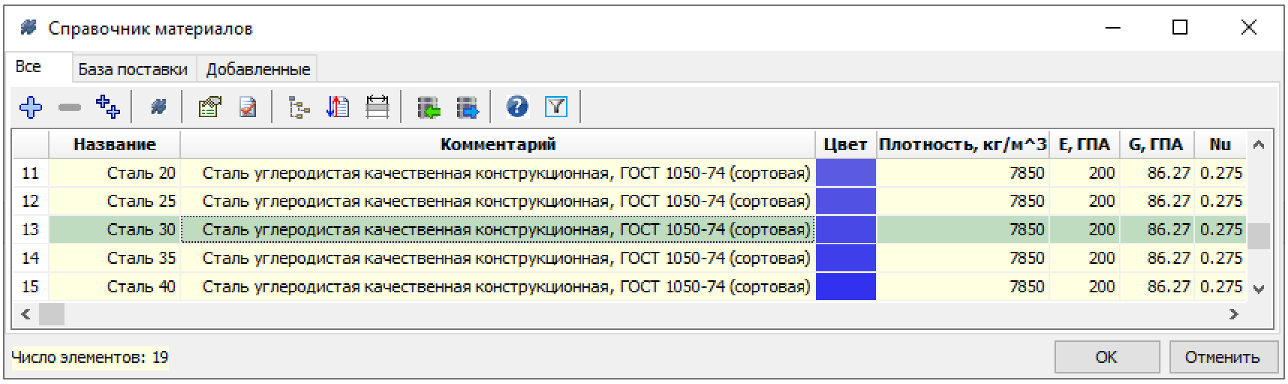


Figure 1.106. **UM Material Directory** window

1.4.4. Window for editing the finite element list

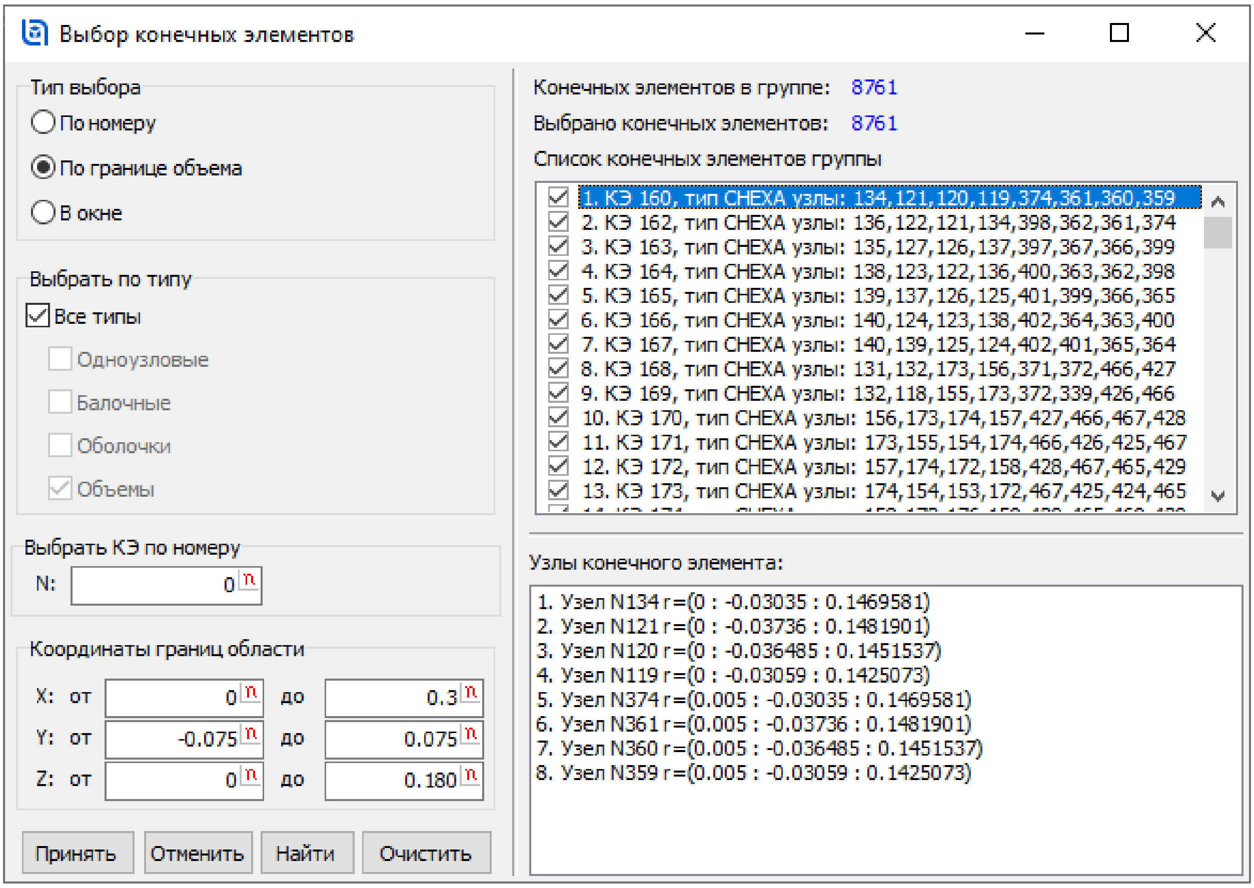


Figure 1.107. Window for editing the finite element list

Finite elements can be selected in one of three ways, which is specified by the current radio group value **Selection Type**:

- By number;
- By volume boundary;
- In window.

To select by number, enter the element number in the **N** field and press ENTER. If the subsystem contains a FE with such number and it is absent in the list, i.e. it was not selected earlier, it will be added to the list. Otherwise, a message will be displayed.

If you select the **By volume boundary** option, the **Area boundary coordinates** fields are filled with the subsystem dimensions - the minimum and maximum coordinate values among all nodes. These values define a parallelepiped faces of which are parallel to the coordinate axes of the subsystem. The value of any field can be edited, after that you should press the **Find** button. The found finite elements, not selected earlier, will be added to the list. An element is selected if all nodes belonging to it lie in a given volume which is a parallelepiped.

The selection can be made taking into account the FE dimension, the filter is set by the **Select by type** checkbox group. To select elements of certain dimensions, uncheck the **All types** checkbox and set the required dimensions.

If the selection type **In window** is set, the finite elements can be selected in the animation window as described in Sect. 1.4.2.2.

1.4.4.1. Context menu of the group finite element list

Click right mouse button to call the context menu (Figure 1.108). Only the first menu item **Show current element in the window** requires explanation. It works as an on/off switch, changing its state to the opposite at each selection. When the menu item is on, the current list element is selected in the window and depth control is turned off for it. That is, it will be visible even if it is inside the body (Figure 1.109).

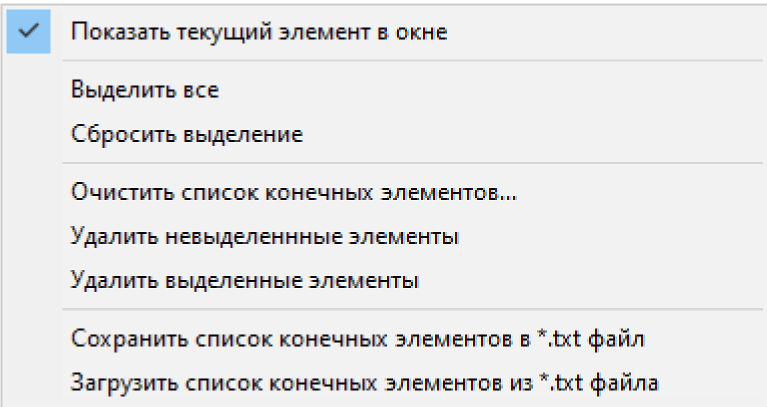


Figure 1.108. Context menu of the group finite element list

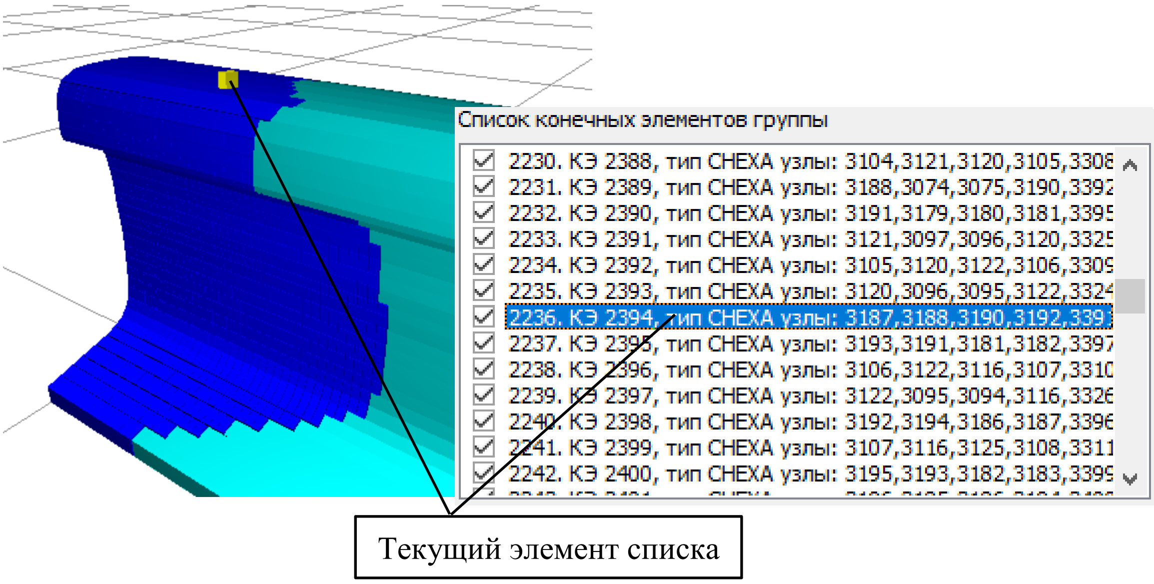


Figure 1.109. Display of the current element of the group list

1.4.5. Solution tab of the control window

Let's describe the differences between the current version of the **Solution** tab (Figure 1.110) and the previous one (See Sect. 1.3.2.2).

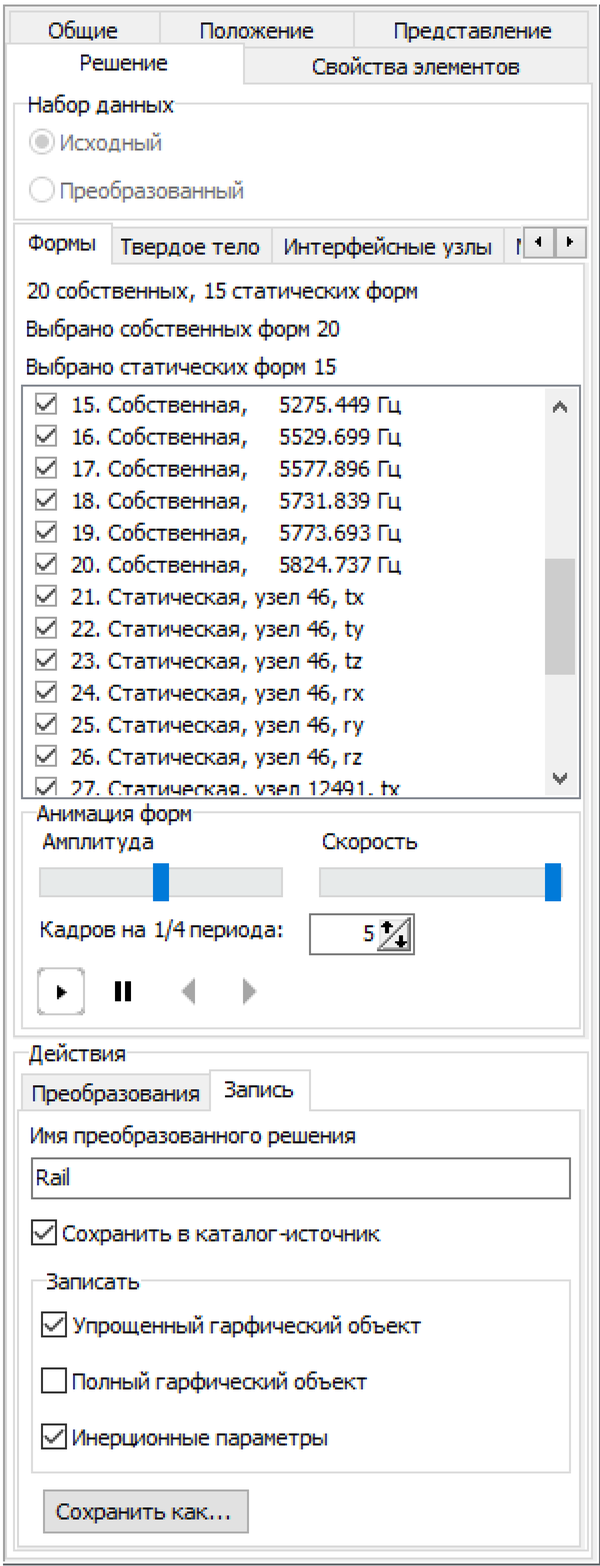
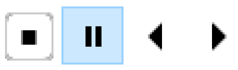


Figure 1.110. **Solution** tab

1. The elements of the form list corresponding to static forms contain in their names the number of the interface node and the degree of freedom for which they are computed: *tx, ty, tz* - translational, *rx, ry, rz* - rotational.

2. Buttons for controlling the animation of forms have appeared. Here are their states and purpose.

	Animation is not started, start by left button **Start**.
	The animation is running, you can stop it with the left **Stop** button or pause it with the **Pause** button, second from the left.
	Pause mode, **Left**, **Right** buttons can be used to change the image frame by frame. Pressing the **Pause** button again resumes the animation, the left **Stop** button stops it.

3. Graphical object and inertial parameters of the subsystem can now be written to the file *input.fss* or *input.fum* at the user's request. To do this, it is necessary to enable the corresponding checkboxes. Otherwise, the graphical object, as before, will be calculated at the first loading of the subsystem and written to the file *grinfo.fss*, and inertial parameters - at the first start of integration or linear analysis and written to the file *inertia.fss*. Both files are located in the flexible subsystem catalog. For large subsystems the mentioned calculations can take lots of time, so it seems more convenient to run them together with transformations of shapes in the **Wizard**. Note that in this case the size of *input.fss* file increases insignificantly.

4. The **Save as...** button is always available, allowing you to save the *input.fum* file with entered properties, modified number of selected eigenforms without transformations, add a graphical object. Inertial parameters are calculated only for the transformed solution. Data modification is not controlled, i.e. for the original subsystem it is possible to overwrite, for example, to another directory.

1.4.6. Control window view tab

The tab has changed, the control elements are grouped differently, but their functionality remains the same. The current view of the sub-tabs is shown in the figure below.

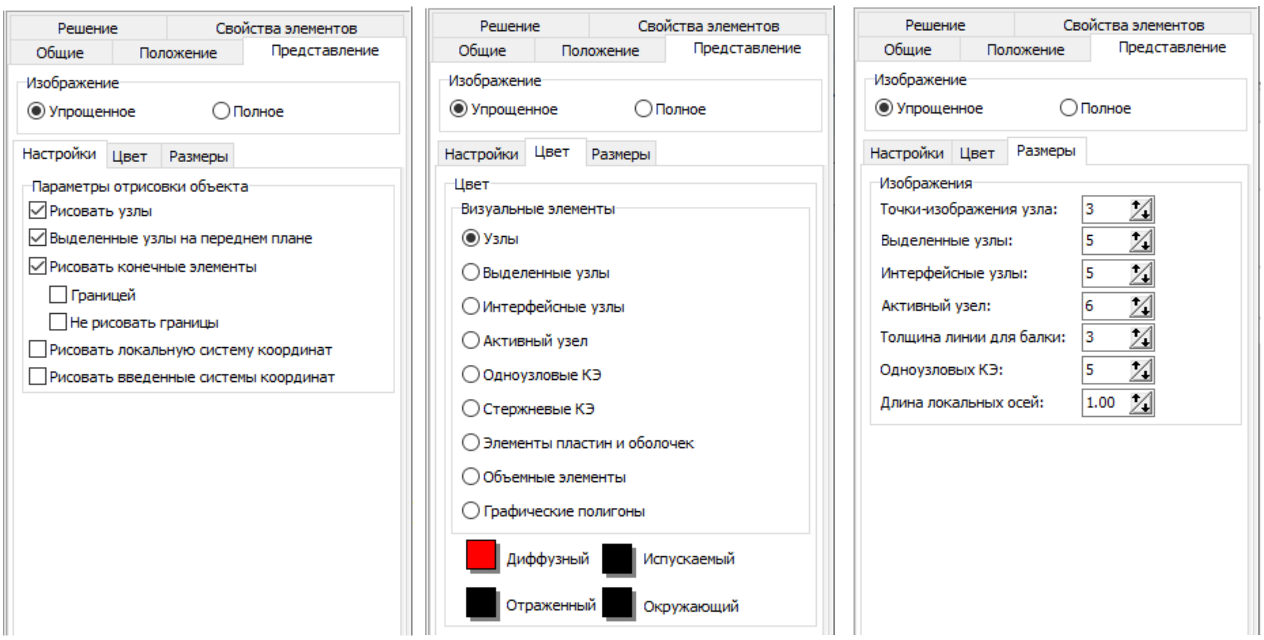


Figure 1.111. Control window **View** tab

1.5. Adding the flexible subsystem into a hybrid model

1.5.1. 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 Figure 1.112. Choose the **Linear FE 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.

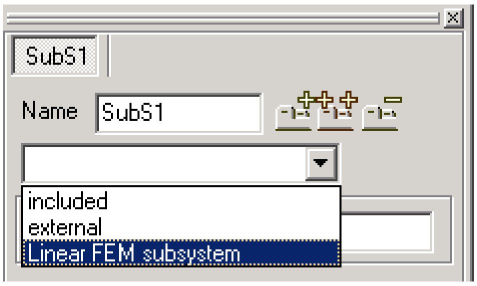


Figure 1.112.

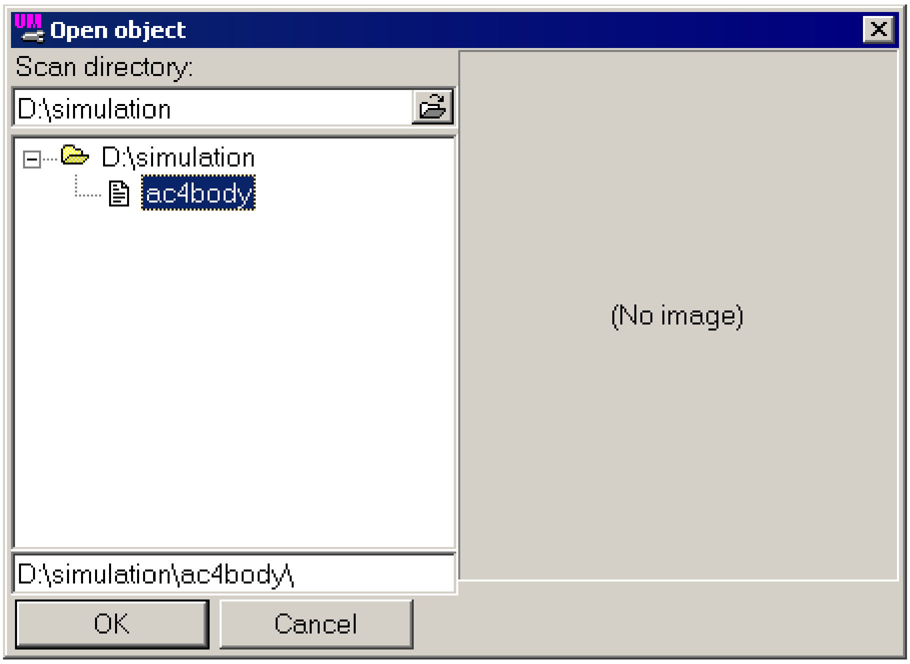


Figure 1.113.

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.

1.5.2. 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.

1.5.2.1. General tab

The **General** tab contains some additional boxes, Figure 1.114.

- **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.

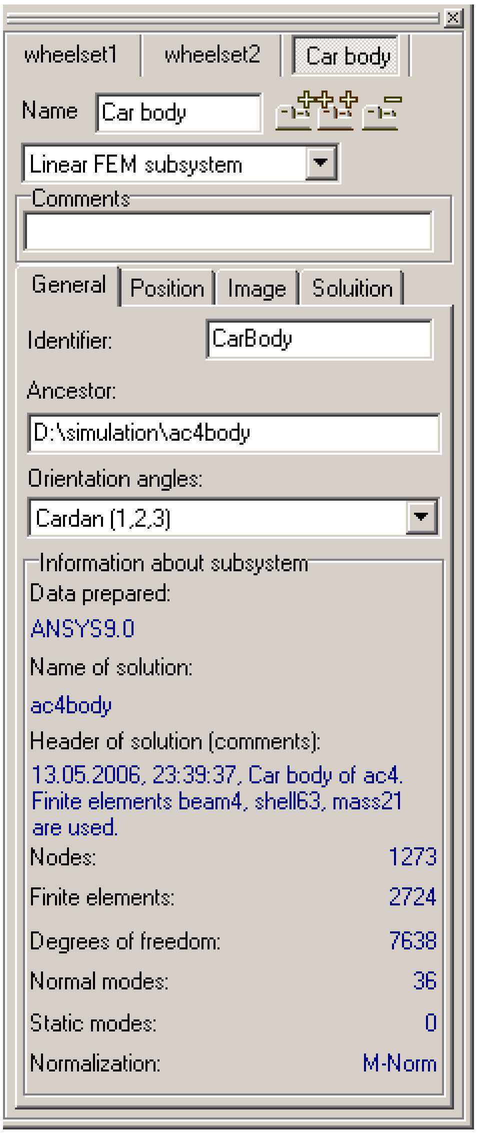


Figure 1.114.

1.5.2.2. 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.

1.5.2.3. 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.

1.5.2.4. Image tab

Using the tool panel buttons in the **UM Input** main window makes sense for **Simplified** image for flexible subsystem only. **Simplified** or **full** image of the flexible subsystem is defined at the **Image** tab, see Figure 1.115, Figure 1.116.

Figure 1.115. Graphical modes in **UM Input** window. Fragment of the tool panel.

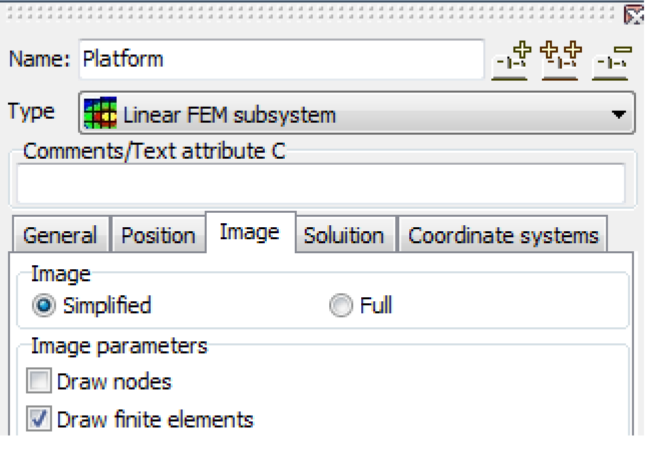


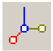
Figure 1.116. Image tab of the flexible subsystem (**UM Input**).

1.5.2.5. Coordinate systems tab

Using this tab the user can create additional frames of reference to transform stresses and strain that are calculated relatively local frame of reference of the flexible subsystem. In fact, the oriented points, which are described in Sect. 1.5.2.5. *"Coordinate systems tab"*, p. 1-106 of the 3rd chapter of UM User's Manual, are created.

Let us consider an example of using the **Coordinate systems** tab. Let us imagine that we need to estimate axial stresses on the inclined beam of a railway bridge, Figure 1.117. Longitudinal axis of the inclined beam coincides with no one axis of the local frame of reference of the flexible subsystems.

Let us create an additional **Coordinate system** relative to the local frame of reference of the flexible subsystems. Let us orient x-axis along to the longitudinal axis of the beam. Click  button to create new **coordinate system**. Position of the origin and orientation of the new coordinate system are expressed in the local frame of reference of the flexible subsystem in the fields of the dialog window, Figure 1.118.

There is an alternative way to define a new **coordinate system**. Click **Get by 3 points** button  and step by step visually define the node for origin, another node to determine the X-direction and the third node to determine XY-plane, Figure 1.119. The process of selecting the nodes of the FE-mesh is shown in Figure 1.119a and the results are shown in Figure 1.119b. X-axis is depicted along longitudinal beam axis.

As soon as the new **Coordinate system** is defined one can express stresses and strains in the nodes of the inclined beam in projection on newly defined **coordinate system**. To create new variable use **UM Simulation / Tools / Wizard of variables**.

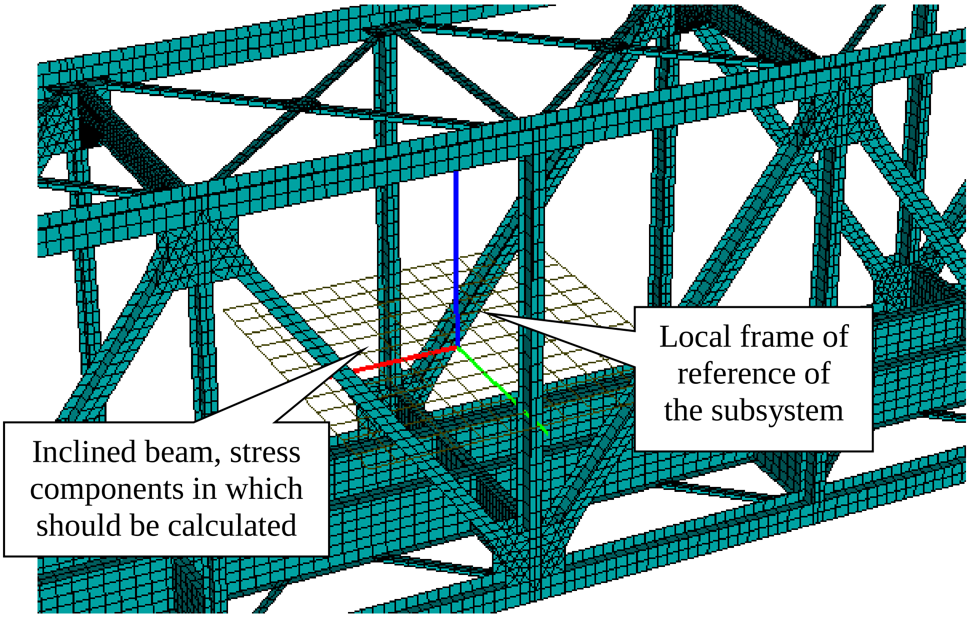


Figure 1.117. Fragment of the railway bridge with inclined beam

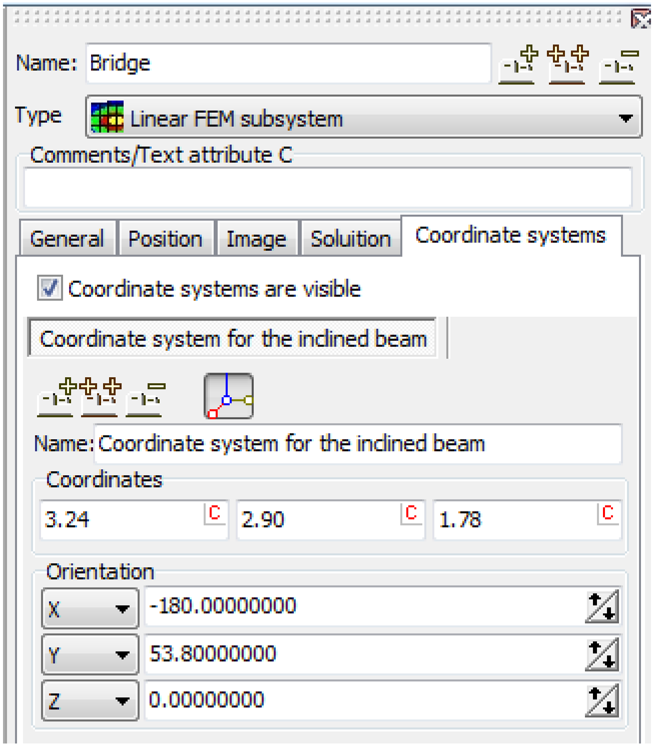
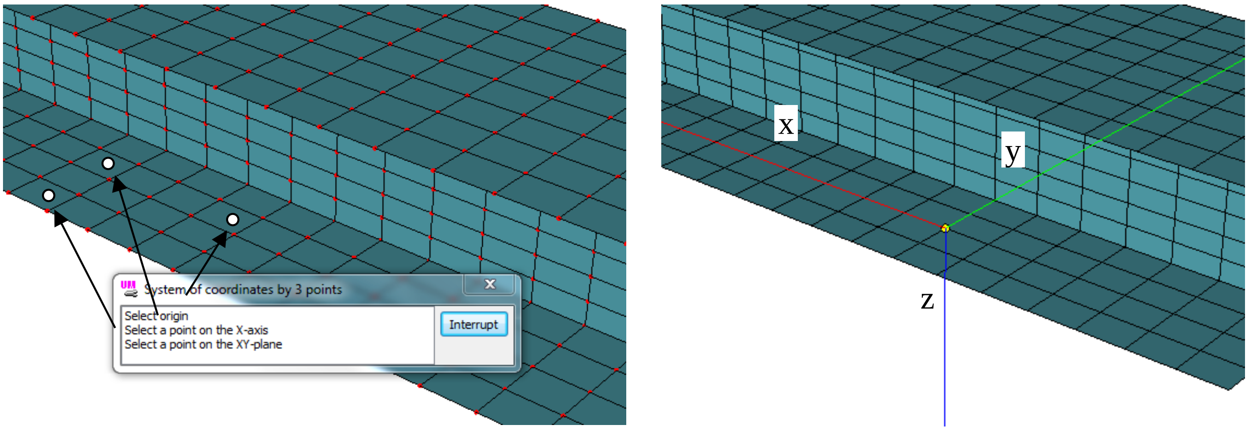


Figure 1.118. **Coordinate systems** tab



b)

Figure 1.119. Creating coordinate system by three points

1.5.3. 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. 1.5.3. *"Features of adding joints and forces"*, p. 1-108.

The following types of joints are supported for the flexible subsystems:

- rotational;
- translational;
- 6 degrees of freedom;

- generalized;
- rod.

The following types of force elements are supported:

- bipolar forces of all kinds;
- linear force elements;
- contact forces **Points-Plane** type (with some restrictions, see below) and **Flexible body – Flexible body** type.
- T-Forces.

Below 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 Figure 1.120.

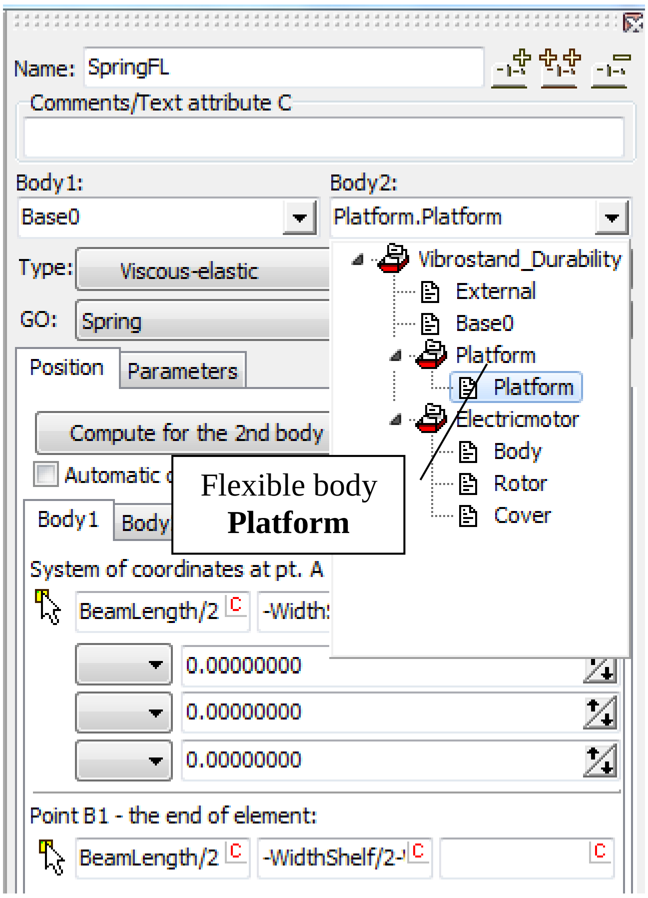


Figure 1.120. Selecting flexible bodies

- 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. It might lead to relatively significant inaccuracy of the simulation results.

- Joint point of force attachment point may be selected visually after clicking  button. Along with coordinates of the attachment point the body itself will be selected or changed.
- If the flexible body was already selected as one of interacting bodies when clicking  the following popup menu appears, Figure 1.121.

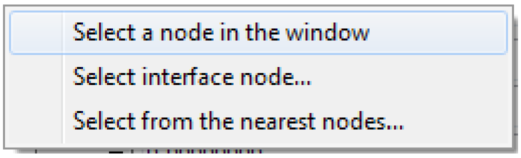


Figure 1.121.

- Use **Select from the nearest nodes** menu item to observe the list of the five nearest nodes to the point that coordinates are given in the files for force element or joint. Each line includes the number of node, its coordinates and the distance **(d)** to the target point, see Figure 1.122.

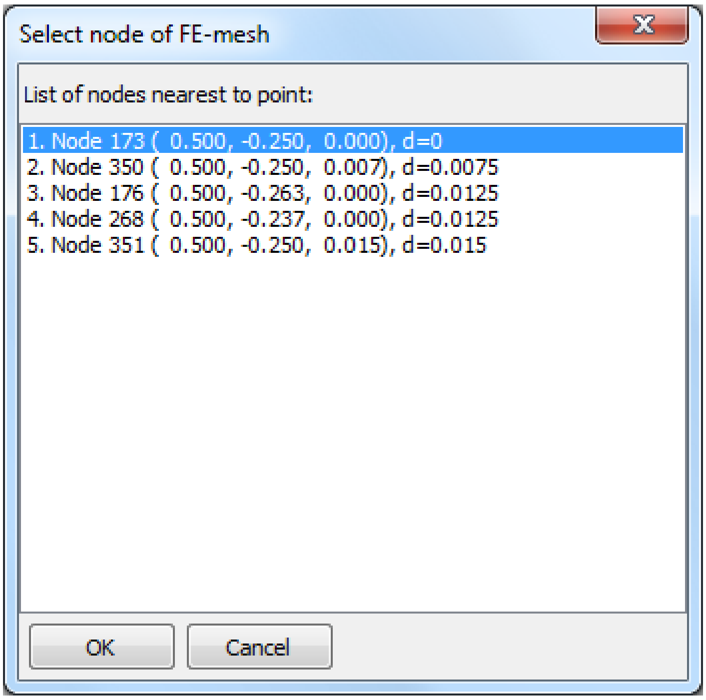


Figure 1.122. List of nearest nodes

1.5.3.1. Features of defining the contact forces for flexible bodies

A flexible body can interact with other bodies of the model via contact forces of the following two types.

1. **Points-Plane** contact force. Description of this force element is considered in Sect. 3.4.9.3.1 of UM User's Manual. Contact points are described for one body and the contact plane is described for another body. The flexible body can be only the body for assigning contact points only. Contact plane cannot be assigned with the flexible body.

2. Contact force of **Flexible body – Flexible body** type. Mathematical model of the force is based on the point-plane contact model, Sect. 2.4.6.1 of UM User's Manual, and is modified to consider features of flexible bodies. To describe this type of contact force the user only needs to specify flexible bodies.
3. Contact points are assigned with the first body that interacts with the second body that is considered as a set of triangular plane elements. The whole surface of the second body is considered automatically. If the finite element face that is lies on the surface is triangular then contact polygon coincide with the face. Otherwise the face of the finite element is divided into several triangular polygon implicitly. At each integration time step each contact point interacts with the only contact polygon. The contact point cannot interact with several contact polygons simultaneously. For each *contact point – contact polygon* pair the *point – plane* mathematical contact model is used. Contact force acts on the first body at the contact point and is reduce to nodal forces for the second body, Figure 1.124. There are no torque components in the contact.

Note. Simulation results significantly depend on smoothness of the shape of the FE mesh of the second body. Finite element borders are treated as, in fact, borders of contact planes. The FE-mesh with rather large finite elements may lead to stepwise changing of the contact forces whilst moving the contact point from one finite element to another one. It is recommended to keep the size of finite elements small enough to keep rather smooth contact forces.

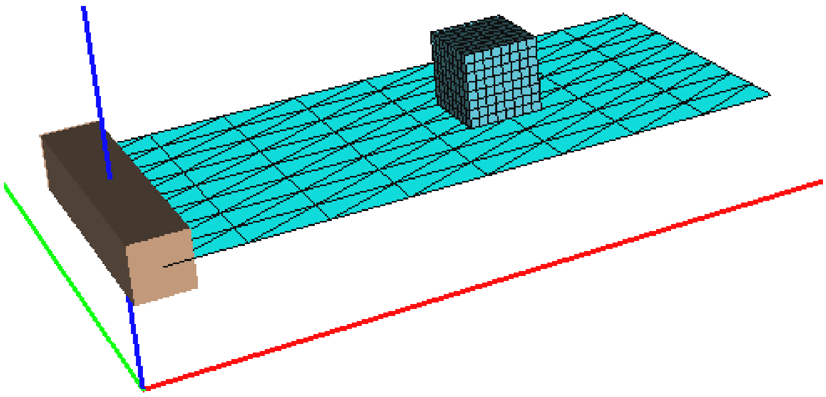


Figure 1.123. Contact of two flexible bodies

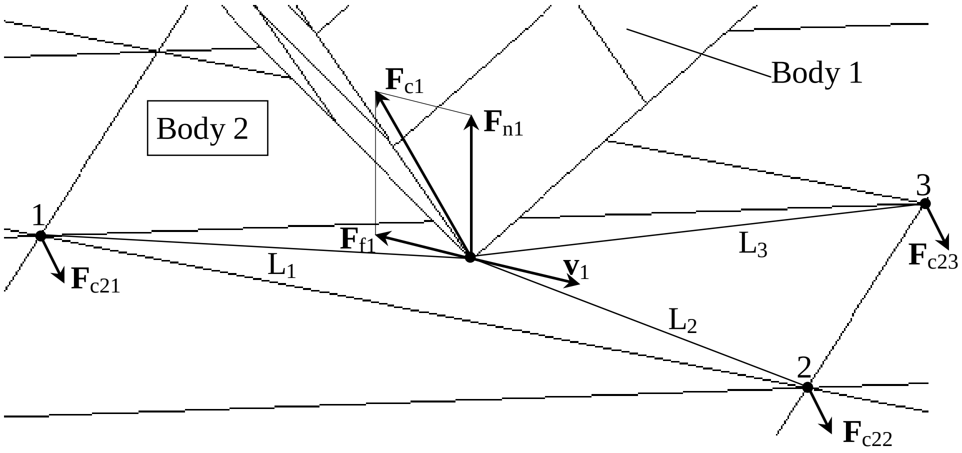


Figure 1.124. On the algorithm of contact force calculation

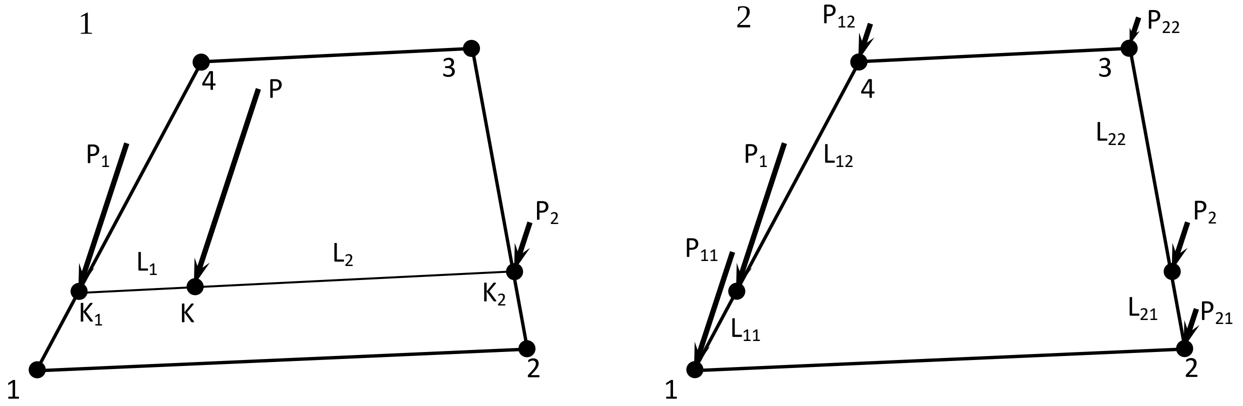


Figure 1.125. On transformation of the arbitrary applied force to nodal forces

The algorithm of the reduction of the arbitrary applied force to nodal forces has two stages. At the first stage, force **P** applied at point **K** is reduced to equivalent pair of forces applied at points $\mathbf{K}_1$ and $\mathbf{K}_2$. At the second stage force $\mathbf{P}_1$ is reduced to $\mathbf{P}_{11}$ and $\mathbf{P}_{12}$ and $\mathbf{P}_2$ is reduced to $\mathbf{P}_{21}$ and $\mathbf{P}_{22}$. Forces $\mathbf{P}_{12}$ and $\mathbf{P}_{22}$ are summarized for the triangular elements. The following equation is satisfied:

$$\mathbf{P}_1 L_1 = \mathbf{P}_2 L_2, \mathbf{P}_{11} L_{11} = \mathbf{P}_{12} L_{12}, \mathbf{P}_{21} L_{21} = \mathbf{P}_{22} L_{22}.$$

Let us consider a dialog window for setting parameters for Flexible body – Flexible body contact force element.

Parameters tab is the same for all types of contact forces, see Sect. 3.4.8.3.

Geometry tab contains the list of contact nodes of the first body and the flag that allows making the second body invisible whilst selecting nodes on the first one. Selection of nodes window is used for defining the contact nodes, Figure 1.126. Click  to add new nodes. Nodes, selected during the current and preceding sessions are marked in different colors, Figure 1.127.

Contact node can be *active* or *inactive*. Turn on/off the correspondent check boxes in **Selection of nodes** window to make nodes active/inactive, Figure 1.126. Inactive nodes do not take part in the contact interaction. Turning off the flag in mechanical sense is equal to removing the node from the list. This possibility helps the user to change the list of contact nodes relatively quick.

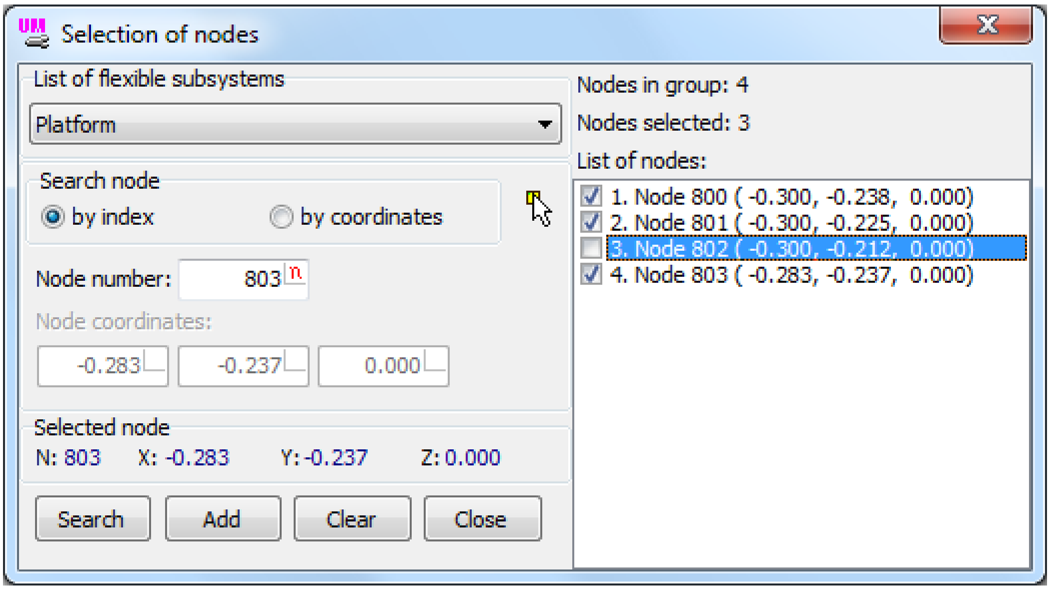


Figure 1.126. Selection of nodes

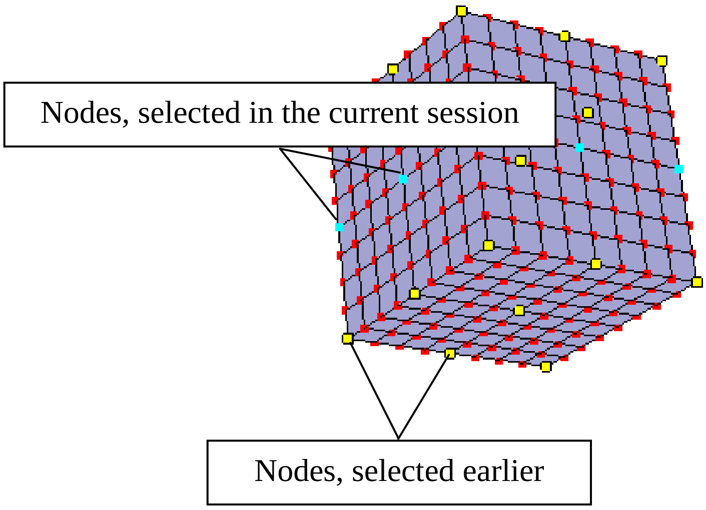


Figure 1.127. Selection of contact nodes

Let us consider the interaction of the flexible cube (body1) with the flexible plate (body2), Figure 1.123. Simulation accuracy increases along with increasing the number of contact points on the cube. However, increasing the number of contact points on the cube makes the simulation time longer. Practically there is some minimal number of contact points that can be considered as a practical optimum – increasing the number of contact points does not influence on simulation results significantly.

1.6. 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.

There are several special tools.

- Export of flexible displacements to ANSYS.
- Data preparing for durability analysis with the help of **UM Durability** module.

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

1.6.1. Special tools

1.6.1.1. Export of flexible displacements to ANSYS

UM uses the modal approach to simulate stress-strain state of the flexible body during simulation of dynamics of mechanical system. Variables created with the help of **Wizard of variables** can represent stresses and strains, as well as, all kinematical performances as time histories in graphical windows.

Since the modal approach is an approximation of the FE analysis – a rather accurate approximation, but approximation anyway – it loses some accuracy. There is an alternative way to estimate stresses and strains of the full FE model directly in **ANSYS**. It might be an additional stage of the applied research that goes after stress and strain analysis in UM or even separate stage of stress and strain analysis of the flexible model without preliminary analysis in UM.

For stress and strain analysis of the full FE scheme in **ANSYS** boundary conditions saved as **UM Simulation** are used. Boundary conditions are saved as *.ald (**A**NSYS **L**oa**d**) files. Term 'Load' in ANSYS has several meaning including nodal displacements, external forces, gravity forces, pressure, temperature, etc. Files *.ald contains nodal displacements of flexible subsystem that is one of supported kinds of loads in ANSYS. The separate line in APDL language is generated for each degree of freedom. The following command

D,863,UZ,0.0001472

sets 0.0001472 displacement in Z direction (UZ component) of node 863. D is APDL *DOF constraint* command.

ANSYS Load files can be generated for one or several time points. The user selects that time points based on preliminary analysis. Generally several time points with the worst load conditions should be considered.

Note. Generation of ANSYS Load file does not need importing stress and strain data from ANSYS. If it is planned to perform stress analysis directly in ANSYS without calculation of stresses in UM, in such a case stress sensors in ANSYS can be avoided.

Steps that should be done to generate ANSYS Load file in UM software are described in Sect. 1.6.3.4. *"Export flexible displacements to ANSYS"*, p. 1-125.

Working with *.ald files in ANSYS supposes the following steps.

1. Load previously created FE model.
2. Delete all existing loads, constraints and boundary conditions. Free of constraints object should be considered.

Note. It is recommended to save FE model in ANSYS as separate *.db file as soon as it is ready, prior to running macro to export it to UM.

3. Load *.ald files with the help of **File | Read Input from…** menu item.
4. Run static analysis, for instance, with the help of APDL command
   ANTYPE,STATIC
5. Analyze simulation results.

1.6.1.2. Preparing data for UM Durability

Universal Mechanism has a built in **UM Durability** module for prediction of damage sum accumulation, see 13 chapter of UM User's Manual. An important part of durability analysis is obtaining the time histories of stresses of flexible body. Such data can be prepared with the help of **UM FEM**.

Nodal stresses in I node are calculated according to formula 11.6. (see Sect. 1.1.2. *"Calculation of stresses and strains"*, p. 1-6):

$$\boldsymbol{\sigma}_i^e = \mathbf{H}_i^{e\sigma}\mathbf{w},$$

where modal matrix $\mathbf{H}_i^{e\sigma}$ is constant. Therefore time histories of stresses can be computed based on relation $\mathbf{w} = \mathbf{w}(t)$, where column matrix w is equal to number of modal coordinates.

UM FEM allows a user to save time histories of modal coordinates to file. Later on **UM Durability** will be able to recover stresses in flexible body having modal matrix $\mathbf{H}_i^{e\sigma}$and column matrix **w**. Modal coordinates are saved to **.tmc* and **.imc* files. Files *.tmc (**t**itle of **m**odal **c**oordinates) have header information in text format. Header information includes the following issues:

1. name of **UM** model;
2. name if the flexible subsystem;
3. path to *.fss file that describes the flexible subsystem;
4. date of *.fss file generation (in a packed form);
5. date of *.tmc file generation (in a packed form);
6. number of nodes of FE mesh;
7. number of finite elements;
8. number of modal coordinates.

Let us consider *.tmc file format.

```
with FEASubSystem;
 ObjectName=PlatformCar;
 name=Platform;
 path=d:\Simulation\PlatformCar\platform_FEA;
 PackDateSolution=20061007;
```

PackTimeSolution=155059;
NodesCount=15748;
FECount=15324;
MCCount=88;

Files *.imc (**i**ntegration, **m**odal **c**oordinates) are binary and contain modal coordinates. Each record has the following format:

$$(t_i, w_1, w_2, ..., w_n),$$

where t_i is time, $w_1, ..., w_n$ are values of modal coordinates at t_i, n is number of modal coordinates. Time step for saving modal coordinates to *.imc is set in **Step size for animation and data storage** in **Object simulation** inspector, see Figure 1.128. The smaller the step size and bigger a number of modal coordinates the bigger the *.imc file size. Recommended step size is in between 0.001…0.02 s.

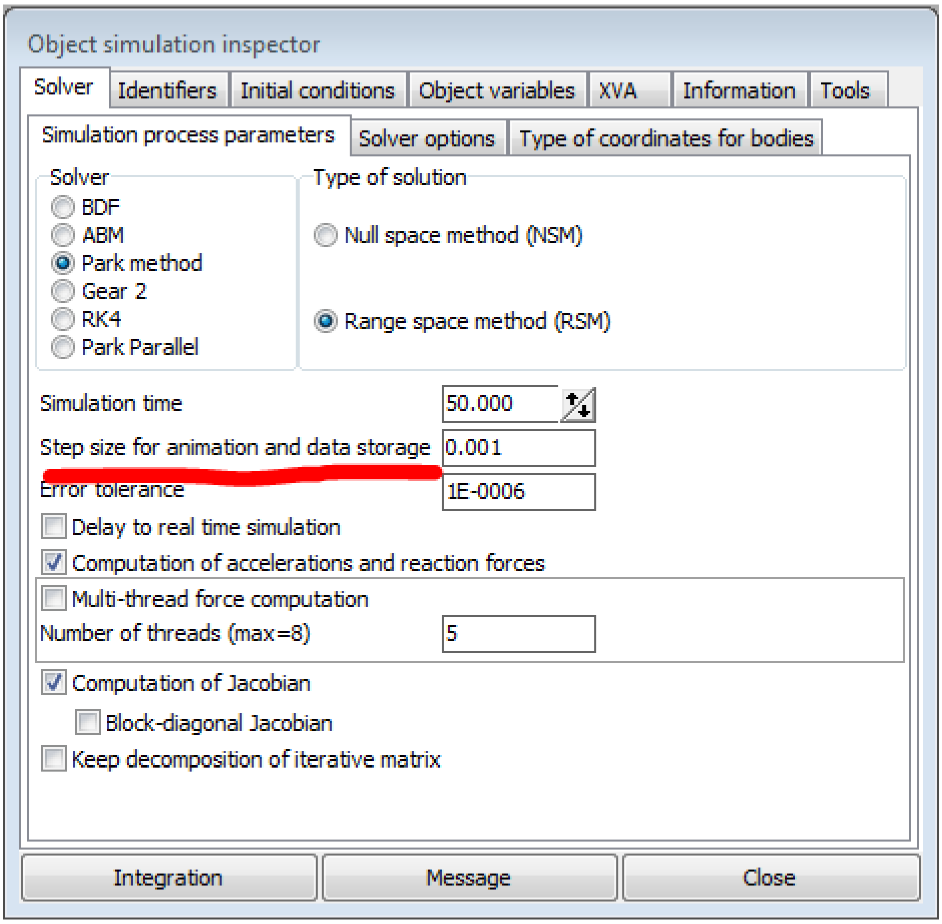


Figure 1.128. Step size for modal coordinate storage

Note. No stress sensors are needed for generation of *.imc file. If no stress are expected be directly analyzed, the model of the flexible subsystem may not contain stress sensors. It would make the file size smaller. Such a model would be faster to simulate and requires less RAM. However UM Durability would not be able to recover stresses without data for stress sensors.

Sometimes it is recommended to export to UM two flexible subsystems. The first one without stress sensors for generation *.imc files, and the second one with stress sensors to recover stresses based on *.imc files later on. All other parameters of flexible models should be the same.

Example.

Let us fulfill durability analyses of the railway flat platform. UM model *FlatCar* is used. Flexible subsystem is named as *Platform*. File *input.fss* is situated at the following folder:

d:\Simulation\FlatCar\platform_FEA.

As it was described above let use prepare two input.fss files: with and without stress sensors.

1. Firstly, copy the input.fss without stress sensors to *d:\Simulation\FlatCar\platform_FEA* and generate a number of *.imc files to correctly represent real load conditions of the flat car.
2. Secondly, copy input.fss with stress sensors to the same directory *d:\Simulation\FlatCar\platform_FEA* and replace the old file. Run durability analysis.

It might be practically useful of the flexible model is big enough and has many stress sensors so as it takes too much time every time to handle with rather big model. If the model is rather small one and is comfortable to work with it is recommended to use the only model with all necessary stress sensors.

1.6.2. 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, Figure 1.129.

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.

1.6.2.1. Simulation tab

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

- **Gravity** flag allows switching on/off gravity. It may be necessary when a subsystem does not have large displacement.
- **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.
- **Fix modal coordinates** flag prevents changing modal coordinates during calculation of initial conditions. Let us consider an example. Let us imagine two bodies, one or two of them are flexible. Bodies are interconnected by a rotational joint. To calculate initial conditions of the simulated object the constraint equations are solved. Joint points of two bodies are place in the same position. There are several possible ways to place both points to the same position, including flexible deformation of the flexible body (bodies). To prevent such a situa-

tion you should set zero to all coordinates for the flexible body at **Initial conditions / Coordinates** tab sheet of **Object simulation inspector** and turn on **Fix modal coordinates** flag. After that UM will consider and treat flexible body as rigid one and will solve constraint equations simply moving and rotating the flexible body as rigid one.

- **Store values of modal coordinates**. It helps you to prepare the ANSYS Load file and fatigue analysis with the help of **UM Durability**, Sect. 1.6.1.1. *"Export of flexible displacements to ANSYS"*, p. 1-114, Sect. 1.6.1.2. *"Preparing data for UM Durability"*, p. 1-115. To create ANSYS Load file both **Memory** and **File** options are possible. Durability analysis needs the modal coordinates to be saved to a **file**. You can assign the default file name by clicking Ctrl+ENTER or button. Default directory is the directory of the flexible subsystem.

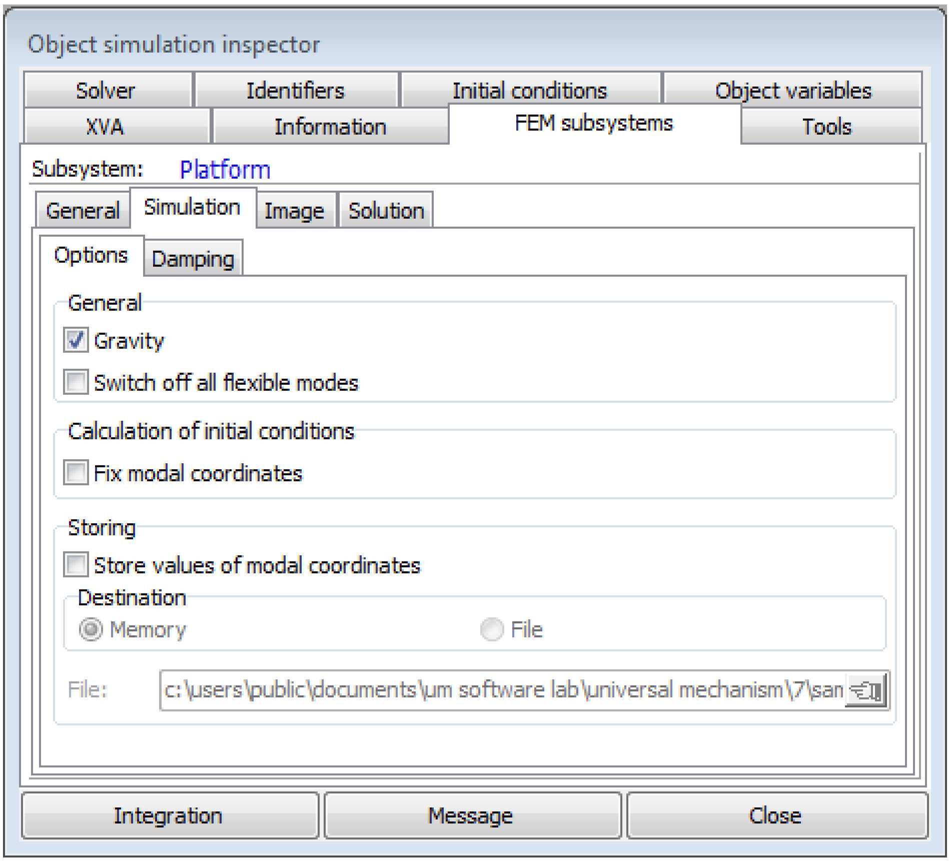


Figure 1.129. Object simulation inspector

- 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.

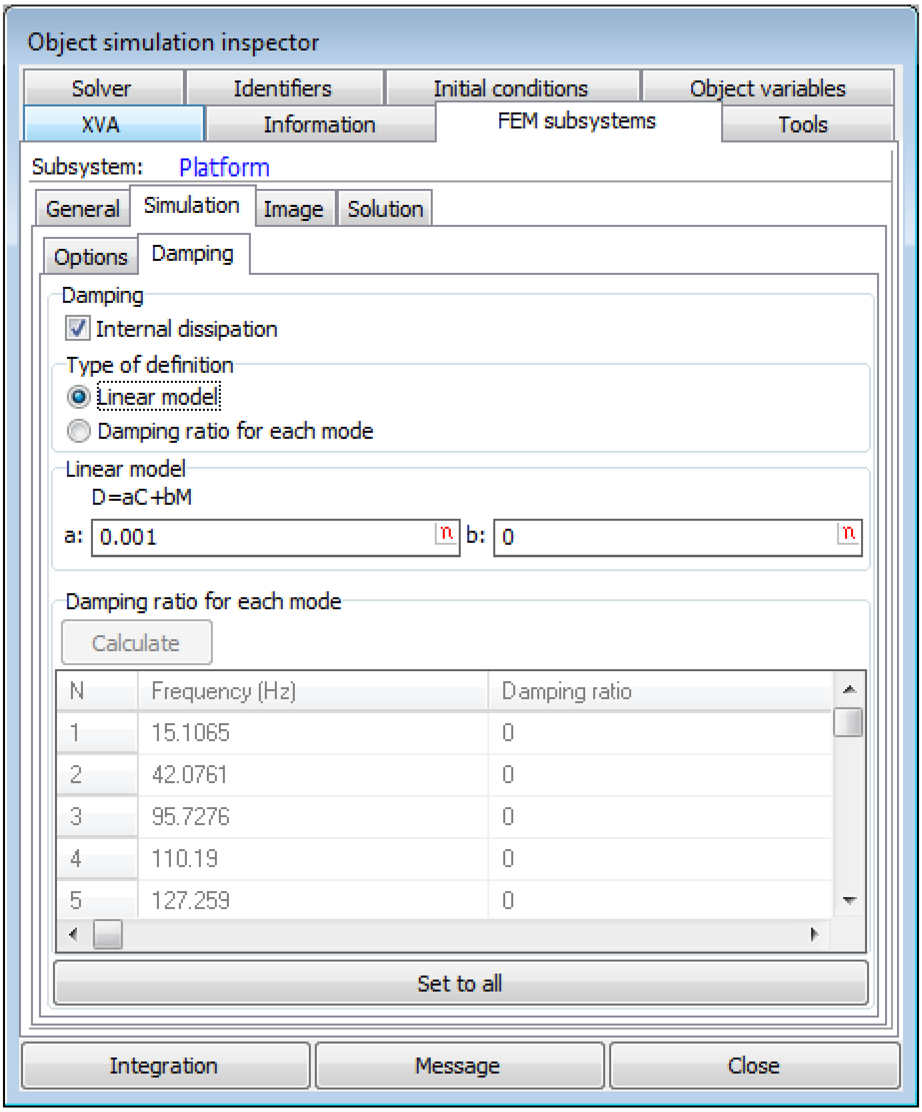


Figure 1.130. Object simulation inspector / FEM Subsystem / Simulation / Damping

- When **Damping ratio for each mode** is checked the dissipation for every mode as a damping ratio, Figure 1.131. 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 (Figure 1.132). Use the popup menu to show this dialog. The frequency interval can be set by indices of modes (Figure 1.132a) or by the value of the frequency (Figure 1.132b).

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 (Figure 1.129). You can firstly set values of coefficients of linear dissipation model **a** and **b** (Figure 1.133) and then to click the **Calculate** button to calculate damping ratios for each mode. Values of damping ratios are available to edit.

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

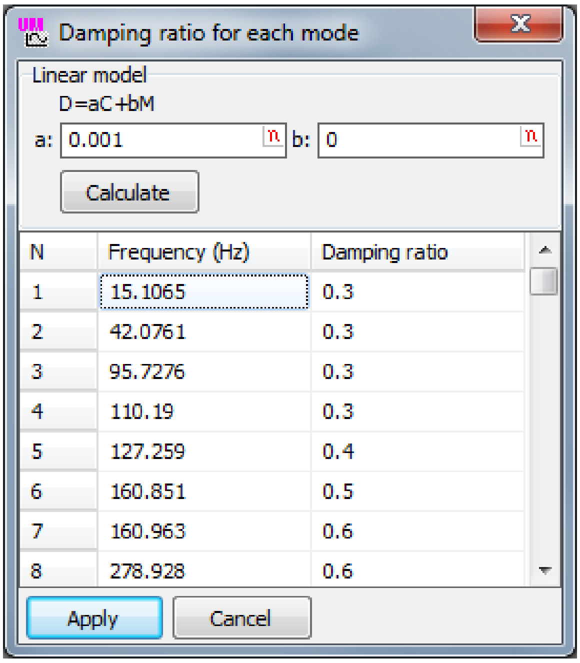
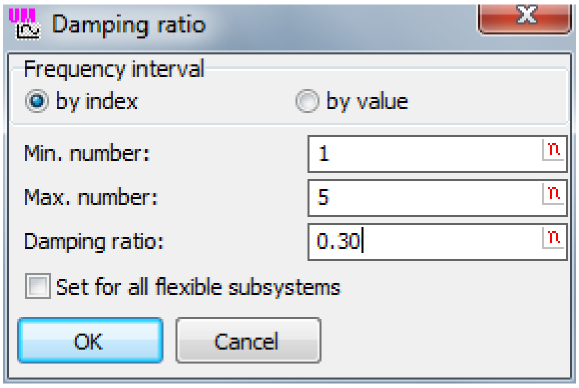
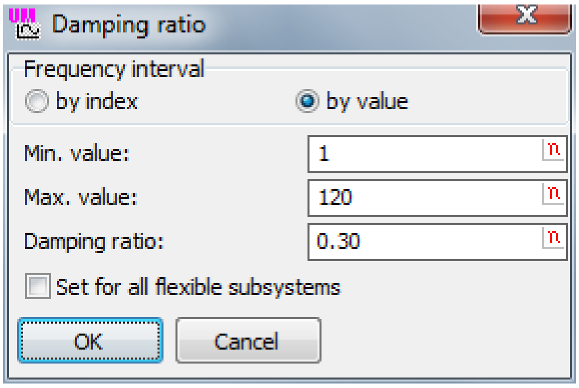


Figure 1.131. Damping ratio



a)



b)

Figure 1.132. Damping ratio

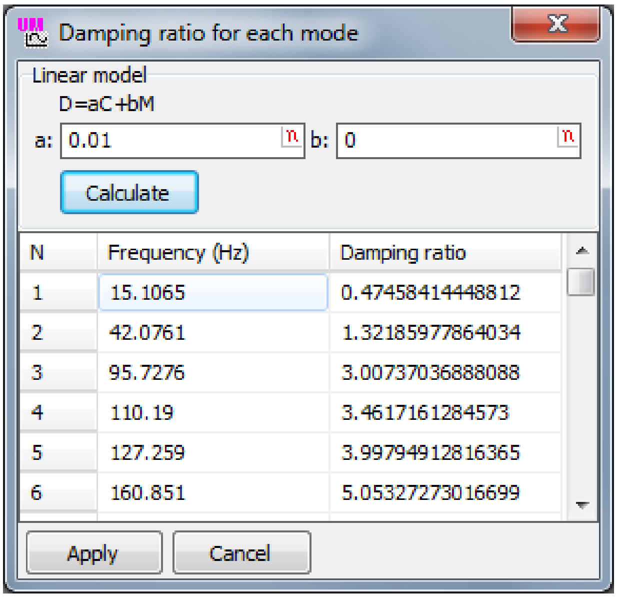


Figure 1.133. Linear model of damping ratio

1.6.2.2. 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. 1.3.2.3. *"Image tab"*, p. 1-89.

1.6.2.3. Solution tab

Solution tab is identical to the corresponding tab of **Object constructor** that is described in details in Sect. 1.3.2.3. *"Image tab"*, p. 1-89.

1.6.3. Variables

You should use the **Wizard of variables** to create new variables to analyze, see Sect. 1.3.2. *"Control panel"*, p. 1-83. Here we will discuss some features, which are connected with simulation of flexible subsystems.

1.6.3.1. Coordinates

In the list of coordinates of the **Wizard of variables** you can see two groups of elements that correspond to flexible subsystem, Figure 1.134. 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).

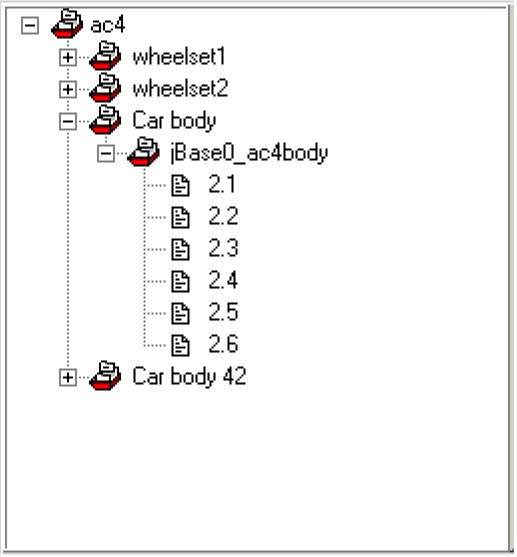


Figure 1.134. Coordinates of the model

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

1.6.3.2. 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 Figure 1.135.

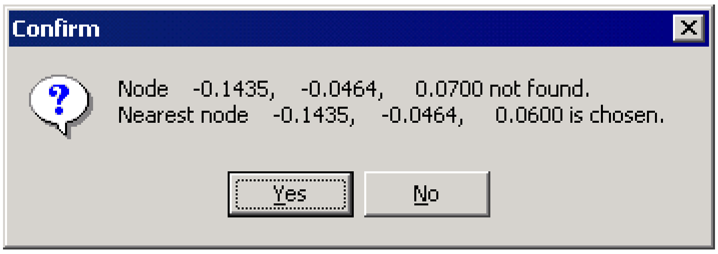


Figure 1.135.

1.6.3.3. Stresses and strains

Variables that are used for estimation of stress-strain state of the flexible body are created at **Wizard of variables / FE Sensors** tab sheet, Figure 1.136.

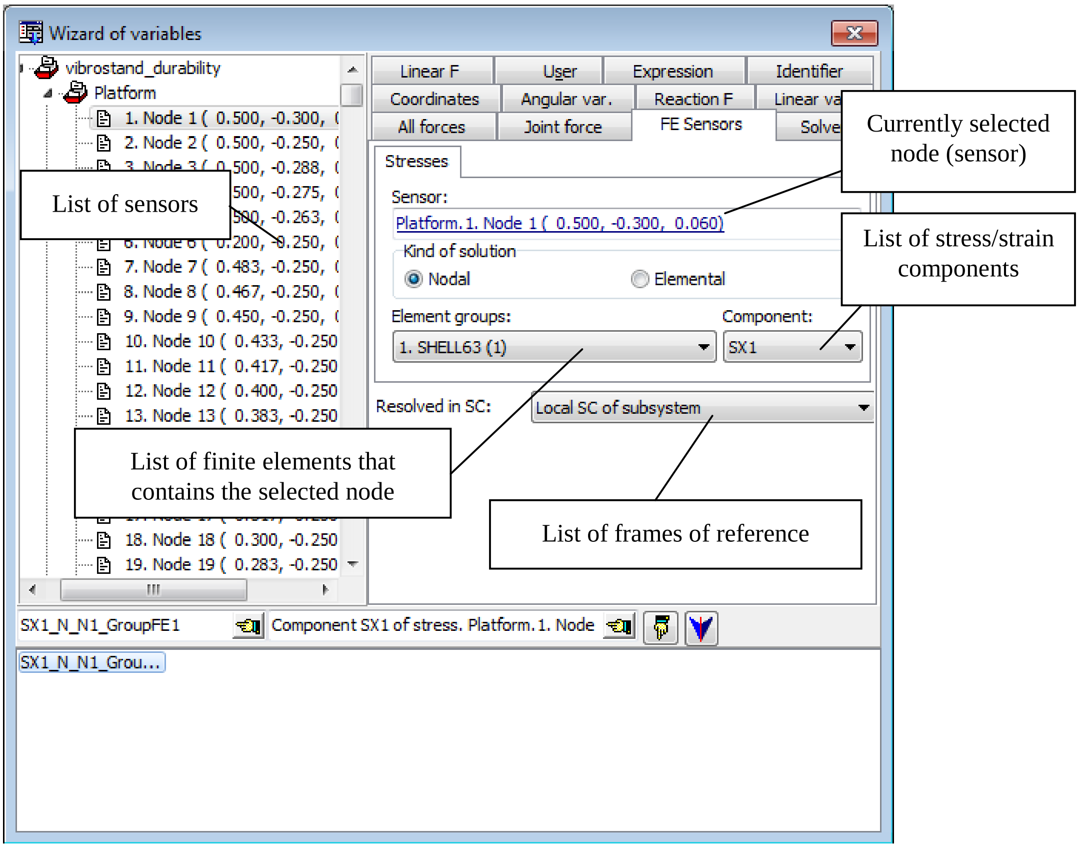


Figure 1.136. FE Sensors tab of Wizard of variables

FE Sensors tab sheet contains two subsidiary tabs **Stresses** and **Strains**. Since **Stresses** and **Strains** tab sheets are identical let us consider the **Stresses** tab only.

Note. **FE Sensors** tab sheet or one of the subsidiary tabs will be invisible if *input.fss* file that describes the flexible subsystem does not contain data for recovering stresses or strains correspondingly. Data export procedure is considered in Sect. 1.2.1.2. *"Creating stress and strain sensors"*, p. 1-14, Sect. 1.2.2.3. *"Preparing data in MSC.PATRAN/NASTRAN environment"*, p. 1-29.

To create a stress or strain variable the user has to go through the following steps.

1. Select the **Stresses** or **Strains** tab sheet. Please note, that the lists of the nodes (sensors) for both tabs generally can be different.
2. Select the node (sensor) in the list. If the loaded model has several flexible subsystems, then the tree of nodes includes all of flexible subsystems with their nodes (sensors). If there are many sensors in the model you can use **Search node** tool, Figure 1.137. **Search node** helps you to find a node by its index (number) or by its coordinates in the local frame of reference of the flexible body. Enter node **Index** or **Coordinates**, click **Search** and **Apply** if the interesting node is found.

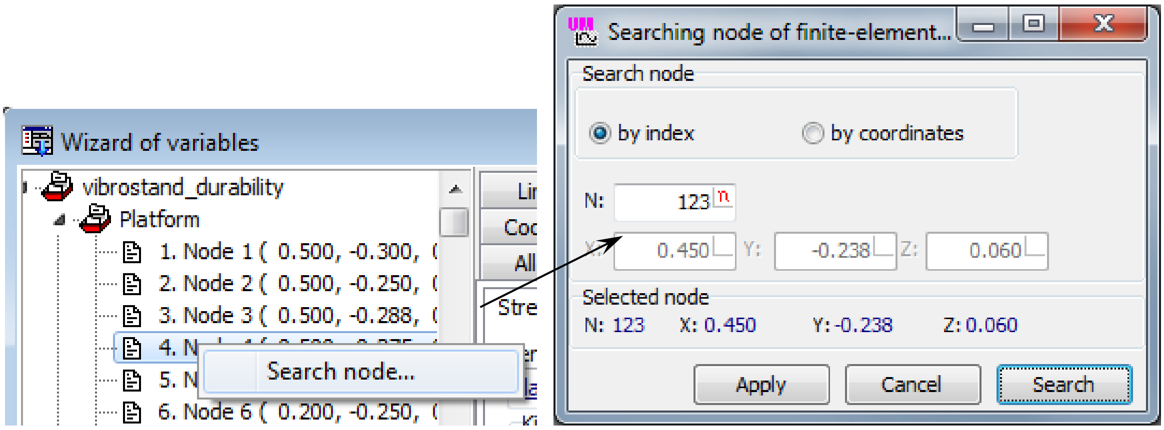


Figure 1.137. Search node

3. Select **Kind of solution: Nodal** or **Elemental**. Nodal stresses and strains are calculated as mean values for all elements that contains the selected node, see Sect. 1.1.2. *"Calculation of stresses and strains"*, p. 1-6. If the node belongs to finite elements of different types, the nodal solutions can be calculated for each type of finite element, see **Element groups** list.
4. Select the **Component** of the solution. The list of components contains elements of stress tensor, principal stresses and von Mises stresses. Components for the top and bottom surfaces are available for shell elements. Top and bottom surfaces are defined by normal direction. Captions of the stress components correspond to its conventional captions in the FE software product where the model was developed and exported to UM.

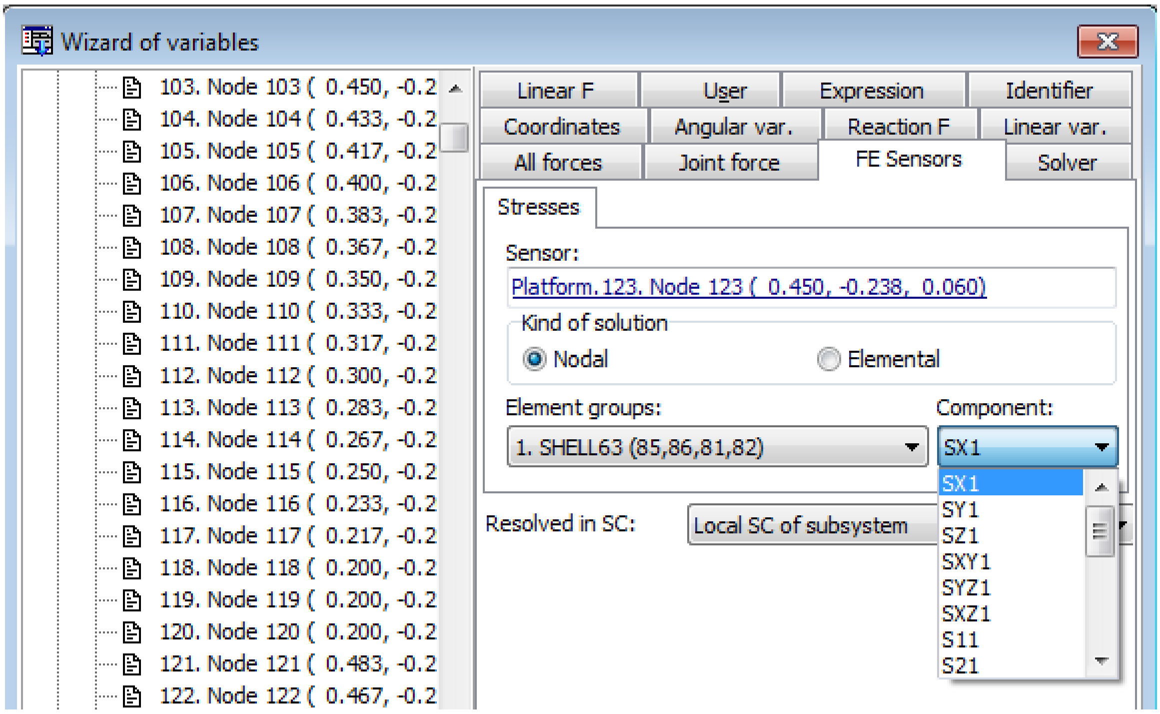


Figure 1.138. Stress/strain component list

For example, in **ANSYS** stress components are named SX, SY, SZ, SXY, SYZ, SXZ, S1, S2, S3, SEQV, where S1..S3 are principal stresses, SEQV is von Mises stresses. If shell elements are chosen, then component names are complemented by number 1 for the top surface and 2 for the bottom surface, for examples S12 would be a maximal principal stress on the bottom surface and SEQV1 would be von Mises stress on the top surface.

1.6.3.4. Export flexible displacements to ANSYS

Flexible displacements are exported into **ANSYS** software with the help of the dialog window shown in Figure 1.140. Click tool button in the tool panel of **UM Simulation**. If you cannot find the button you probably need to turn on the **Flexible** subsystems menu command, see Figure 1.139.

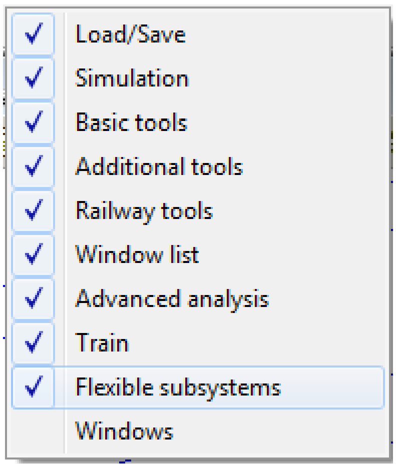


Figure 1.139. Popup menu of **UM Simulation** tool panels

To save a flexible displacement for the flexible body the following steps should be done.

1. Turn on **Store values of modal coordinates** flag in **Object simulation inspector / FEM Subsystems / Simulation / Options** tab sheet, see Figure 1.129.
2. Open **Wizard of variables** and create variables that will be treated as criteria for time point selection for saving load file.
3. Copy created variables to a graphical window.
4. Run simulation.
5. Open **Export of load files for flexible subsystems** dialog window, see Figure 1.140. Copy variables from the graphical windows to **Export of load files / List of variables** group.
6. In **Subsystem** drop down lost select the flexible subsystem you are interesting in.
7. Select some specific variable in **Export of load files / List of variables** group. Panel **Analysis of variable** shows some variables-related information like name of variable, minimal and maximal values of variable and the correspondent time points. There is also an edit box where the user can input the time what the ANSYS load file should be generated for.
8. Select the desired **Criterion** and **File name** and click **Save**. Click Ctrl+Enter keys or button to generate default file name. Default directory for the load file is the directory of the flexible subsystem. Default file name includes flexible subsystem name, variable name and the selected criterion.

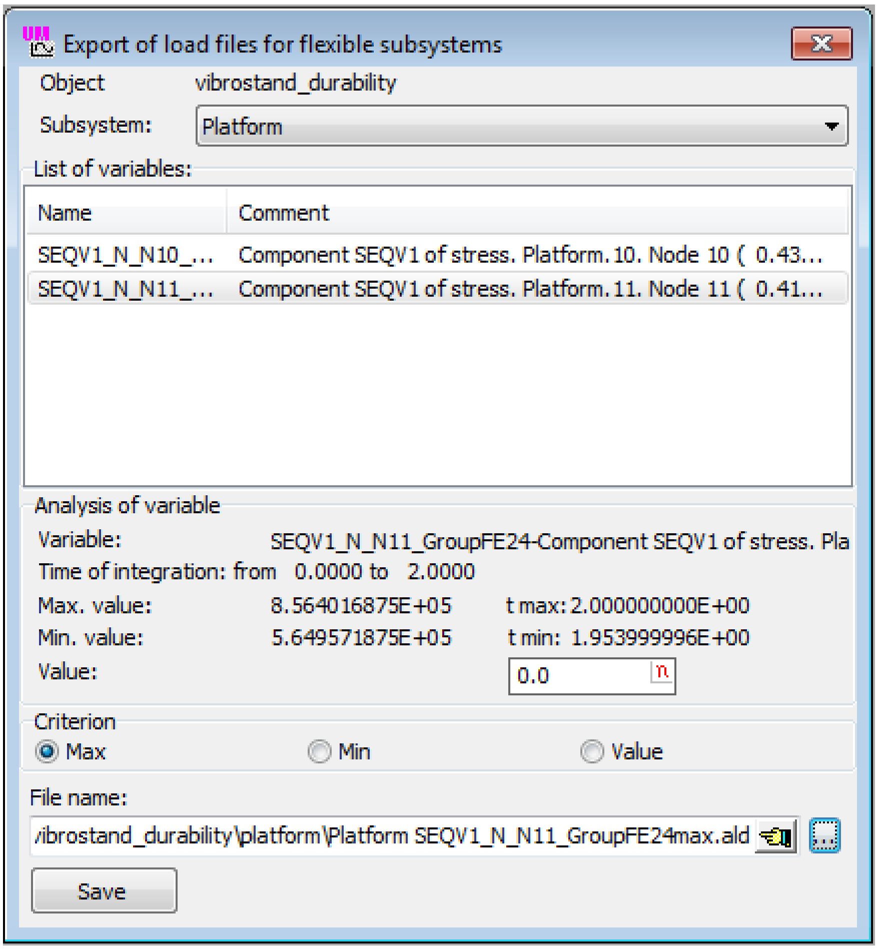


Figure 1.140. Generation of ANSYS Load file

Note.	Modal coordinates are calculated for time steps with an interval that is described in **Object simulation inspector / Solver / Simulation process parameters / Step size for animation and data storage** field, see Figure 1.128. The closest time point to specified by **Value** field is stored.
Note.	You can select any arbitrary time point simply analyzed plots of variables in graphical windows.