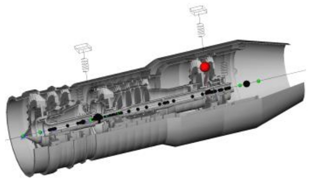
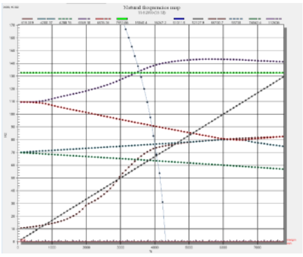
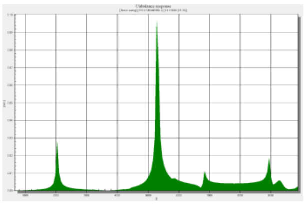
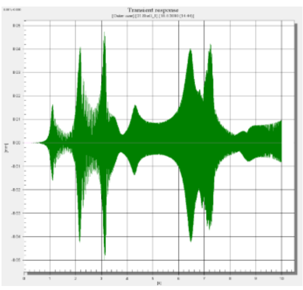
Engineering & Consulting Centre
Alfa-Transit

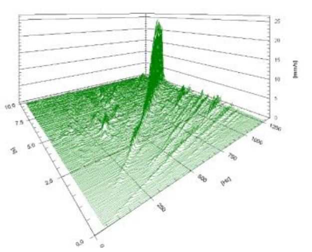




Simulation in Dynamics R4 (HowTo…)







DYNAMICS R4

TRANZIT

Contents

1. Modeling and calculation of rotor

1.1 Rotor simulation

Simulate the rotor presented in Fig. 1.1. Table 1.1 shows stiffness coefficients of links.

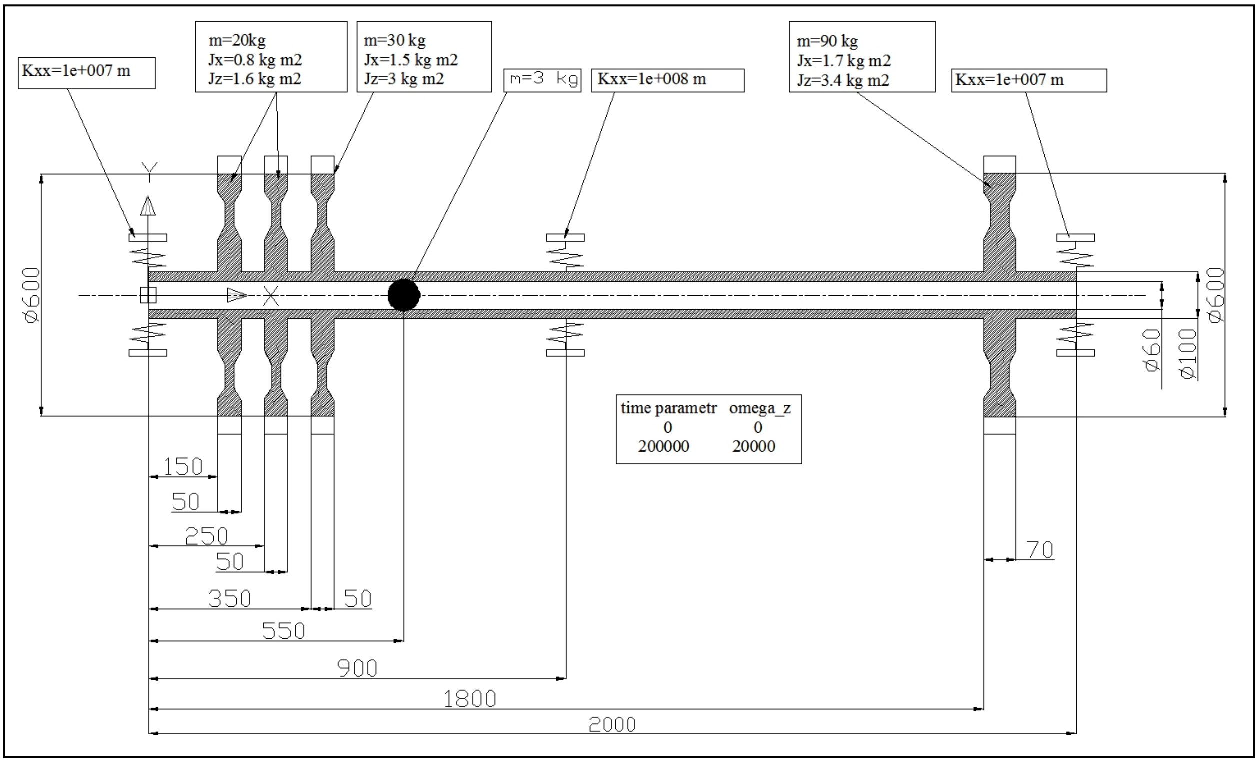


Fig. 1.1

Table 1.1

	Front support	Central support	Rear support
	N/m	N/m	N/m
$K_{XX}= K_{YY}$	1e+007	1e+008	1e+007

<u>Rotor simulation:</u>

a) When Dynamics R4 is launched, the main window of the program system is opened. To start the rotor building, the structural unit of the system must be assigned. Because the rotor is the simplest structural unit in this case, the subsystem (shaft) can be chosen in the elements library (Fig. 1.2). The element may be added by double click. The cursor may be also activated, pressing the left mouse button and transferring it into the area of the 2D model (this principle of operation may be used for all elements of structure and model).

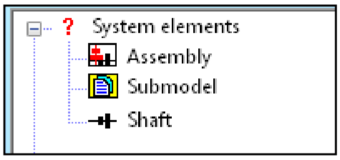


Fig. 1.2

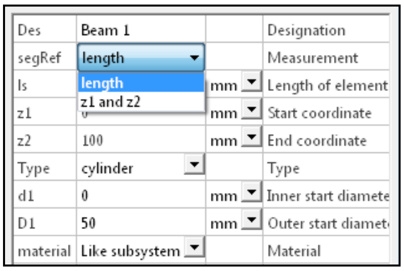


Fig. 1.3

b) Simulation. The first section of the rotor is cylindrical. The (Beam) element must be chosen in the elements bar and placed in subsystem. Assign the sizes of the element in correspondence with the drawing.

The extent of the element may be assigned either by the element length of by its start and end coordinates (in the coordinate system of the subsystem) (Fig. 1.3). Correspondingly, in the first case the ls parameter is used, in the second one – start and end coordinates z1 and z2. Inner and outer diameters are specified in the fields d1 and D1 correspondingly. If the unit is changed, the automatic conversion of the input value takes place. The rotor is simulated by the [Beam] and [Disk] elements. New beam elements are created if outer or inner diameter is changed.

c) To model bearings, links must be added into the model. The element [Link] should be chosen in the elements bar and added into the model. Its geometrical location (either by coordinates or attach it to link points), stiffness and damping matrixes are specified in the link characteristics (Fig. 1.4).

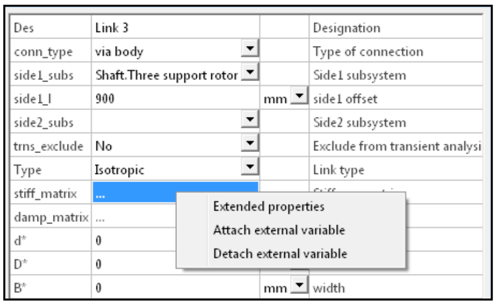


Fig. 1.4

To open the window of editing of the stiffness or damping matrix (Fig. 1.5) the right mouse button must be pressed in the field […] of “stiff_matrix” “damp_matrix” points correspondingly; the “Extended properties” point must be chosen in the context menu (Fig. 1.4).

In this example only K_{xx}=K_{yy} are assigned for stiffness matrixes of links. Let us input the parameters given in Table 1 into the matrixes of the corresponding supports. Note the necessity of change in units of input parameters. On default unit of displacement is [mm]. To input K_{xx} coefficient click twice the first cell in matrix and assign the value.

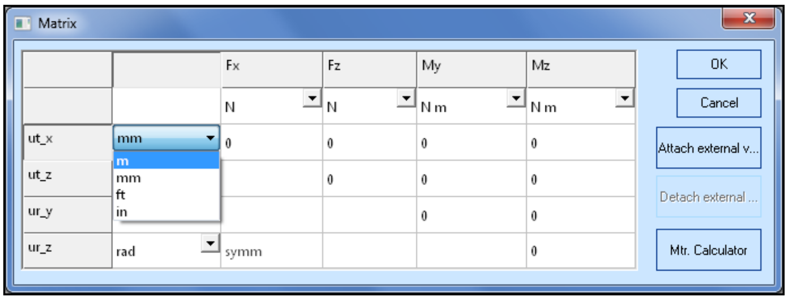


Fig. 1.5

d) Assignment of the rotor rotation. The element [Kinematic joint] must be added into the subsystem. Specifying the value in the [omega_z] parameter, the rotation of the subsystem will be constant. Generally the variable, having function of rotation change, must be assigned. In Dynamics R4 range of the model operating modes is specified through the abstract parameters t_pr1 and t_pr2 (velocity, experiment time, peak mode percentage). To specify the variable rotation, add

[Variables/Value variables] from the elements library to the model. The command [Extended properties] must be chosen among the characteristics of the added variable in the context menu of the […] field of the [value] parameter (Fig. 1.6).

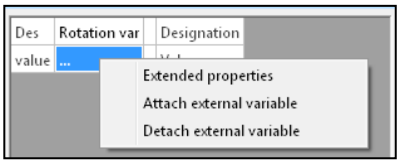


Fig. 1.6

The dialog of the variable editing is opened (Fig. 1.7). The graph is constructed by adding the intermediate points. Table 1.2 gives the examples of the main variants of rotating speed assignment.

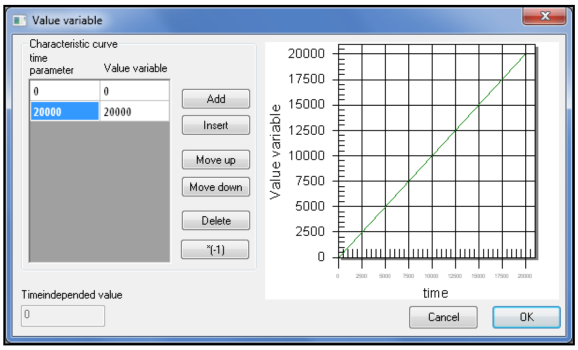


Fig. 1.7

Table 1.2

time parameter / Rotation var 0 / 0 20000 / 20000	[Time parameter] has physical meaning as value of rotating speed
time parameter / Rotation var 0 / 0 100 / 20000	Percentage from peak regime of 20000 rpm
time parameter / Rotation var 0 / 0 1 / 20000	Modified rotation
time parameter / Rotation var 0 / 0 5 / 20000	At correspondence of integration time in [Transient response], physical meaning is considered as time in seconds that the rotor needs to reach the peak mode

The created variable must be added to the parameter in the [Kinematic joint] element. The command [Attach external variable] (Fig. 1.8) must be chosen in the elements characteristics in the context menu of the parameter omega_z. The corresponding variable must be chosen the appearing dialog. Fig. 1.9 gives the model with the added element [Kinematic joint].

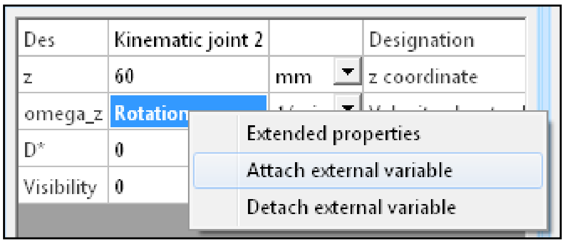


Fig. 1.8

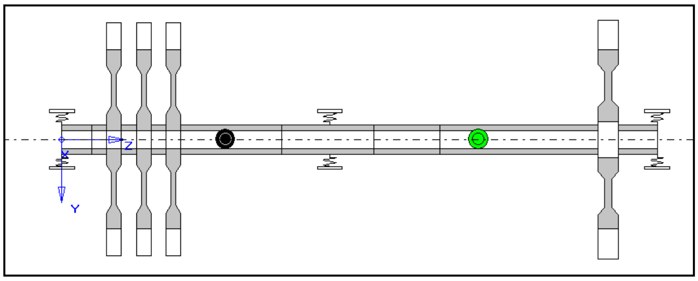


Fig. 1.9

1.2 Calculation of [Critical speeds] and [Natural frequency map] algorithms.

a) Natural frequencies and dynamic behavior of the rotor system are calculated using the methods of modal synthesis and analysis. The preliminary obtainment of set of frequencies and shapes of natural oscillations not taking into account damping and rotating speed in certain frequency range lies on its basis.

Basis calculation is also included into the model on default. Only frequency range (parameter of R_freq basis) is assigned. To start any calculation, click Start in upper part of the main algorithm window.

b) To calculate critical speeds, the corresponding algorithm must be added into the model. Add [Critical speeds] by double click from the library of the elements and algorithms. To start calculation, click Start.

в) To calculate natural frequency map, the corresponding algorithm must be added into the model. To start calculation, click Start.

1.3 Results comparison

Compare the obtained results with those presented below. There is basis (Fig. 1.10, for R_freq = 30000 1/min), critical speeds (Fig. 1.11), natural frequency map (Fig. 1.12).

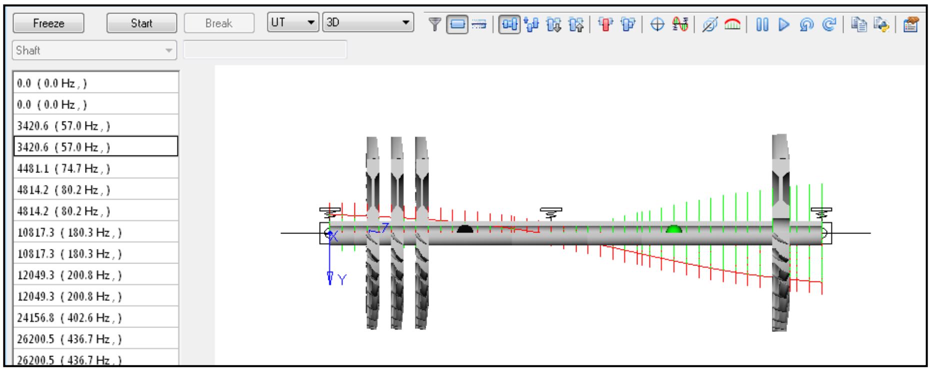


Fig. 1.10

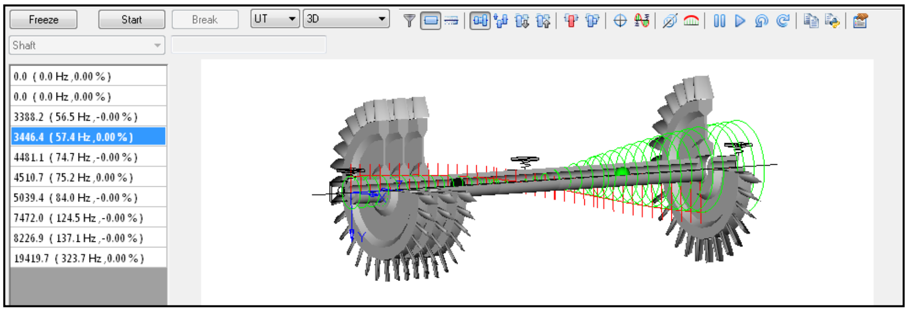


Fig. 1.11

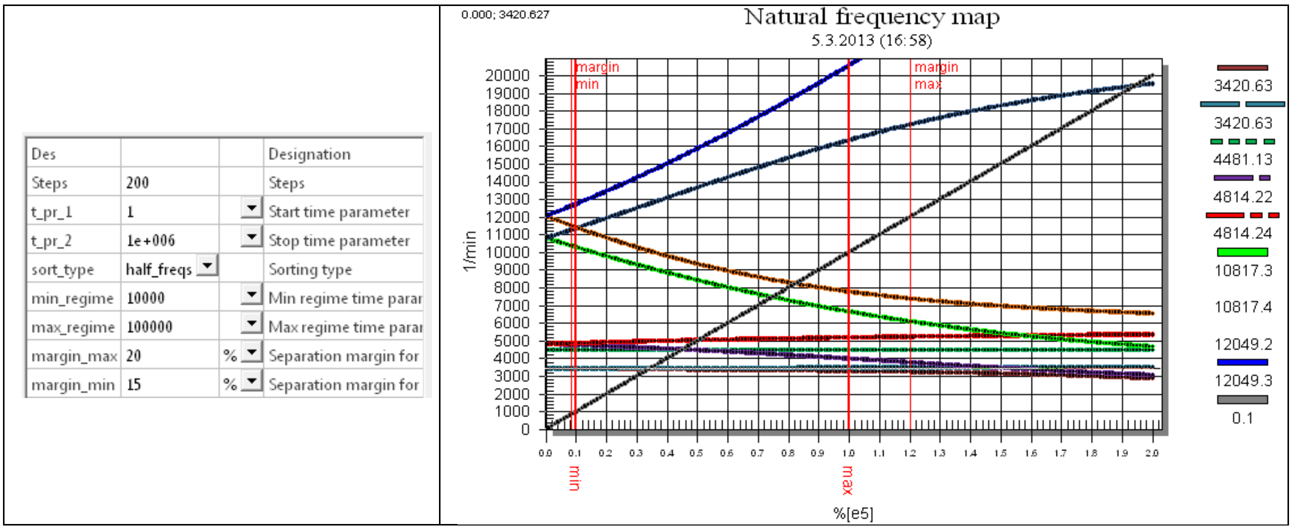


Fig. 1.12

Note: In the section of our web-site you may watch the video of creation and calculation of the detailed simple model. http://www.alfatran.com/movie.shtml

2. Modeling and calculation of assembly of rotor and case

On the basis of the presented model from the previous section, according to the given drawing (Fig. 2.1), the assembly of the rotor with the case should be modeled, critical speeds and natural frequency map should be calculated. The modeling order is presented below.

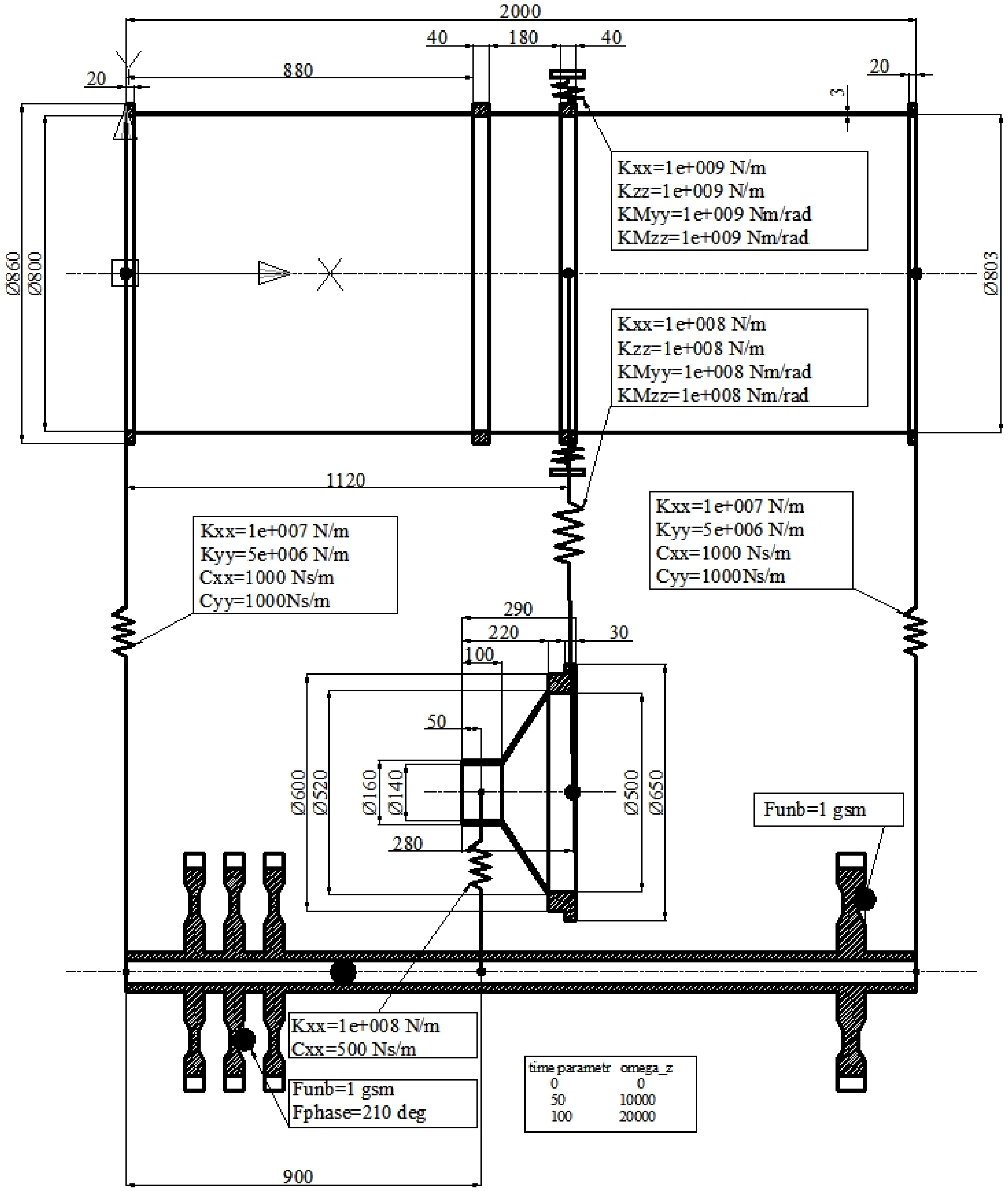


Fig. 2.1

2.1 Modeling

2.1.1 Cases modeling

In this model it is enough to use only subsystems as base units. But for complicated models it is recommended to structure the model using the [Subsystem] and [Submodel] elements.

For the consecutive spacing of the model, it is recommended to join the cases subsystems in the [Case] assembly, and the rotor subsystems – in the [Rotor] assembly. For this we add two corresponding assemblies to the model.

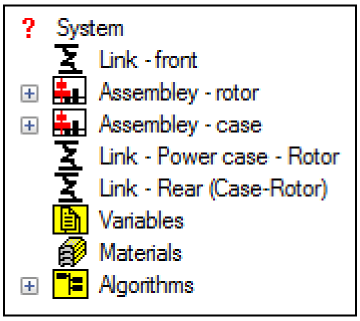


Fig. 2.2

The central pin and the case must be modeled in separate subsystems. For this we built two more subsystems: one – for the case (subsystem [Case]), the other one for central pin (subsystem [The central pin]) (Fig. 2.2). For easy work with them, the subsystems may be visually spaced using the [Spaced subsystems] button on the Toolbar (Fig. 2.3). Their spaced position is given in the window of the subsystem characteristics (Fig. 2.4). In this case all the coordinates and rotation will be equal to zero.

Fig. 2.3

Almost all structural elements of the case and the pin must be modeled using the shell element; meanwhile, long cylindrical elements must be created as compound (consisting from many identical smaller elements) in order to increase discretization, because the cs parameter (calculation step) in the window of the subsystem characteristics, that shows the length of the step of our subsystem's discretization, acts only on the [Beam] elements.

It is not recommended to divide conical shells. Note the fact that definition of the shell geometry (Fig. 2.5) differs from definition the beam geometry. Consider that the point of the support (link) attachment to the [Shell] element must be placed at the elements junction. So, when modeling the geometry of all structural elements of the assembly, provide for junctions in the points of the supports mounting; otherwise, the program gives an error message while calculating.

Des	Case		Designation
x	0	mm	x coordinate
y	0	mm	y coordinate
z	0	mm	z coordinate
eps_x	0	deg	Rotation about x axis
eps_y	0	deg	Rotation about y axis
eps_z	0	deg	Rotation about z axis
material	user defined		Material
E	2.1e+011	N/m2	Modulus of elasticity
Nue	0.3		Poisson's ratio
rho	7850	kg/m3	Density
Ln_dec	0		Logarithmic decrement
cs	10	mm	calculation step
Visibility	0		Element Visibility

Fig. 2.4

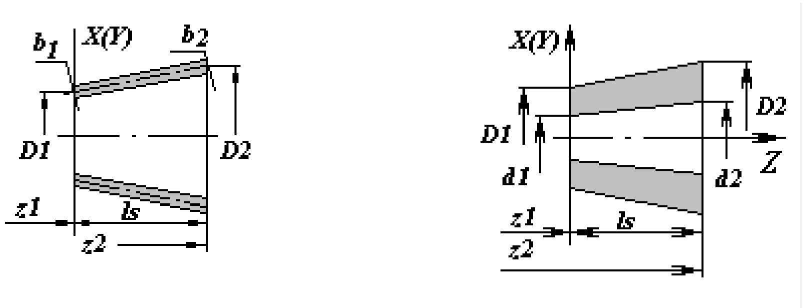


Fig. 2.5

2.1.2 Links and supports modeling

After definition of the geometry of the central pin and the case in the corresponding subsystems, the next step is to superimpose links onto the model (supports modeling). In the first part of the task, when modeling the rotor, the supports were mounted in the corresponding places with the corresponding values of the stiffness matrix. Now these supports must be connected with the case (the subsystem [Case]) and the central pin (the subsystem [The central pin]) and the missing ones should be completed.

Des	Link 1		Designation
conn_type	via body		Type of connection
side1_subs	Rotor.Sy		Side1 subsystem
side1_l	Rotor.System		offset
side2_subs	The central pin.System		subsystem
side2_l	Case.System		offset
trns_exclude	No		Exclude from transient analysis
Type	Full		Link type
stiff_matrix	...		Stiffness matrix
damp_matrix	...		Damping matrix
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Fig. 2.6

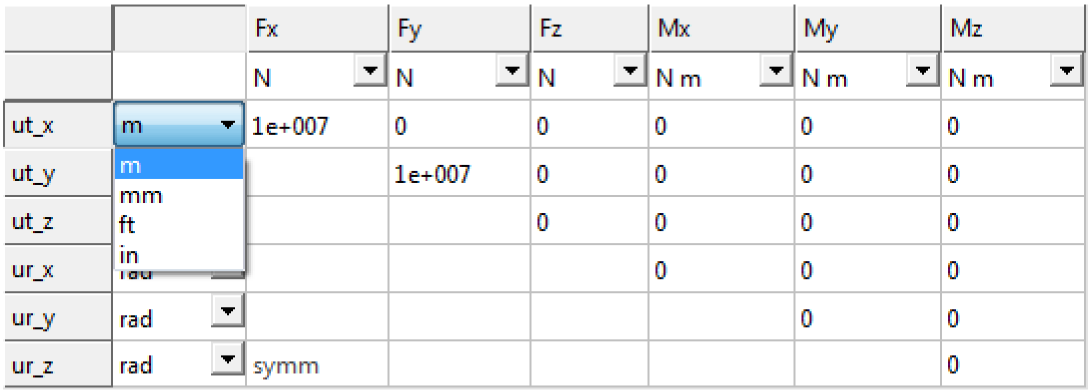
The missing supports are placed into one of the connected subsystems. In order to connect the supports with the corresponding subsystems, for the side2_subs parameter ([Subsystem for section 2 of the link]) of the window of the link characteristics, point the subsystem which must be connected (Fig. 2.6), give coordinates of the point of the link attachment into the corresponding subsystem below (parameter side2_l). For the side1_subs parameter ([Subsystem for section 1 of the link]) the subsystem, where the link was attached initially, is given automatically, in the «side1_l» line the coordinates of the point of the link attachment in the subsystem are specified.

Then it is necessary to fill the stiffness and damping matrixes. For this we choose the link type [Full] in the [Type] line. Clicking by the right mouse button in the [...] area of the stiff_matrix and damp_matrix parameters, choose the Extended properties item (Fig. 2.7). In the appearing dialog of the matrix editing, filling is done in correspondence with the values given in the drawing. To input the coefficients, choose the corresponding field in the matrix by double click and assign the value. Meanwhile, dimension given in your matrixes, must correspond to one specified in the drawing (Fig.

Des	Link 1		Designation
conn_type	via body		Type of connection
side1_subs	Rotor		Side1 subsystem
side1_l	0	mm	side1 offset
side2_subs	Case		Side2 subsystem
side2_l	0	mm	side2 offset
trns_exclude	No		Exclude from transient analysis
Type	Full		Link type
stiff_matrix	...		Extended properties
damp_matrix	...		Attach external variable
d*	0		Detach external variable
D*	0		
B*	0	mm	width

Fig. 2.7

2.8). The damping matrix is opened and filled in similar way.



		Fx	Fy	Fz	Mx	My	Mz
		N	N	N	N m	N m	N m
ut_x	m	1e+007	0	0	0	0	0
ut_y	m mm		1e+007	0	0	0	0
ut_z	ft			0	0	0	0
ur_x	in				0	0	0
ur_y	rad					0	0
ur_z	rad	symm					0

Fig. 2.8

2.1.3 Rotation assignment

To take into account gyroscopic moments in the analysis results, the rotor rotation must be assigned. As in the previous section, the rotor rotation must be assigned using the [Value variable]; difference is only that three control points will be used to define the rotor rotation speed (Fig. 2.9). For this clicking the [Add] button, we add one more line in the left field and change the value in correspondence with the assignment.

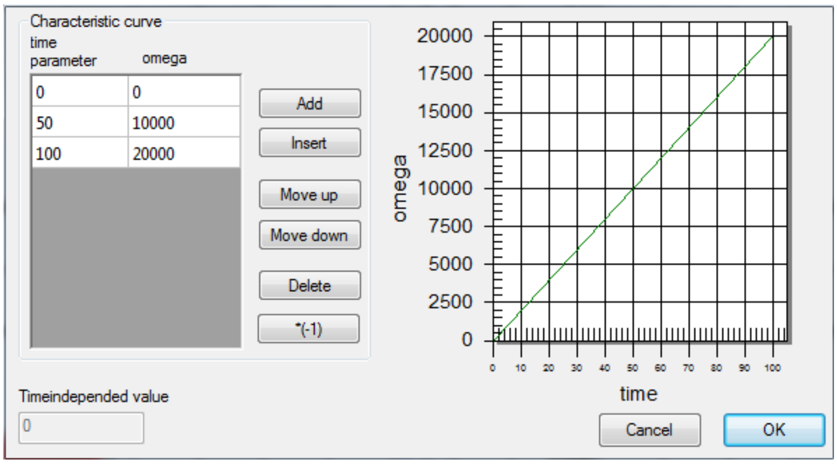


Fig. 2.9

2.1.4 Unbalances modeling

Note that two unbalances are added into the model. On the elements bar in the [Loads] section, the [Unbalance] element is chosen (Fig. 2.10) and dragged into the central window to activate a cursor. After adding unbalance to the rotor, the coordinates of the point application on the (z1) model axis are given in the log of the elements' characteristics; F unb- linear unbalance and F phase- phase of linear unbalance if it is necessary.

When modeling is finished, the model will be the following (Fig. 2.11).

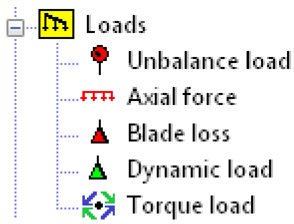


Fig. 2.10

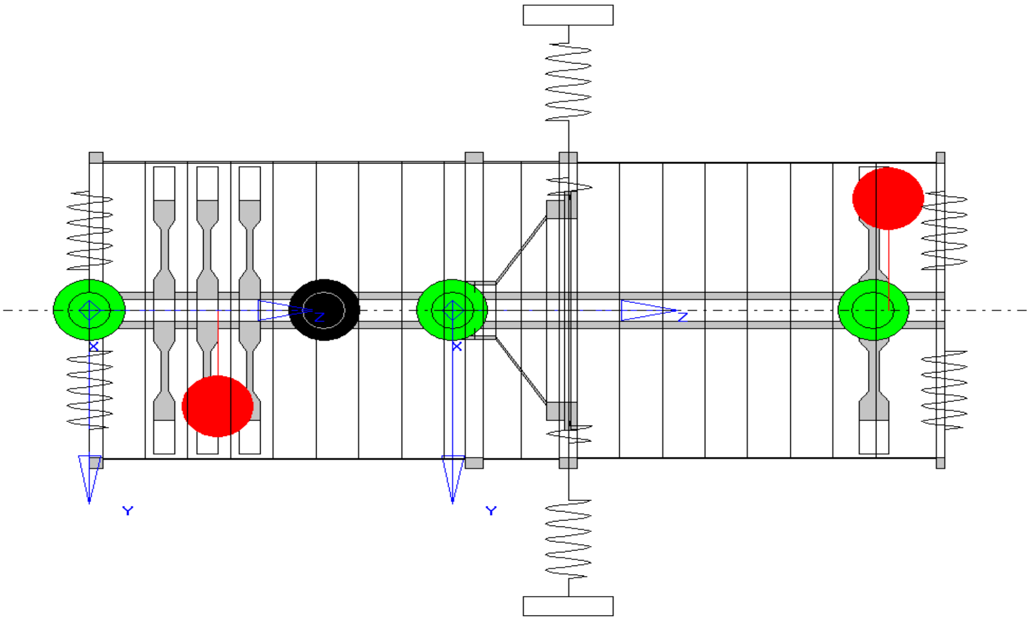


Fig. 2.11

2.2 Linear model analysis

Calculate basis (Fig. 2.12), critical speeds (Fig. 2.13) and natural frequency map (Fig. 2.14) for the built model. Compare the calculated values with those presented below. To remove warnings, it is enough to add zero rotation speeds.

Basis:

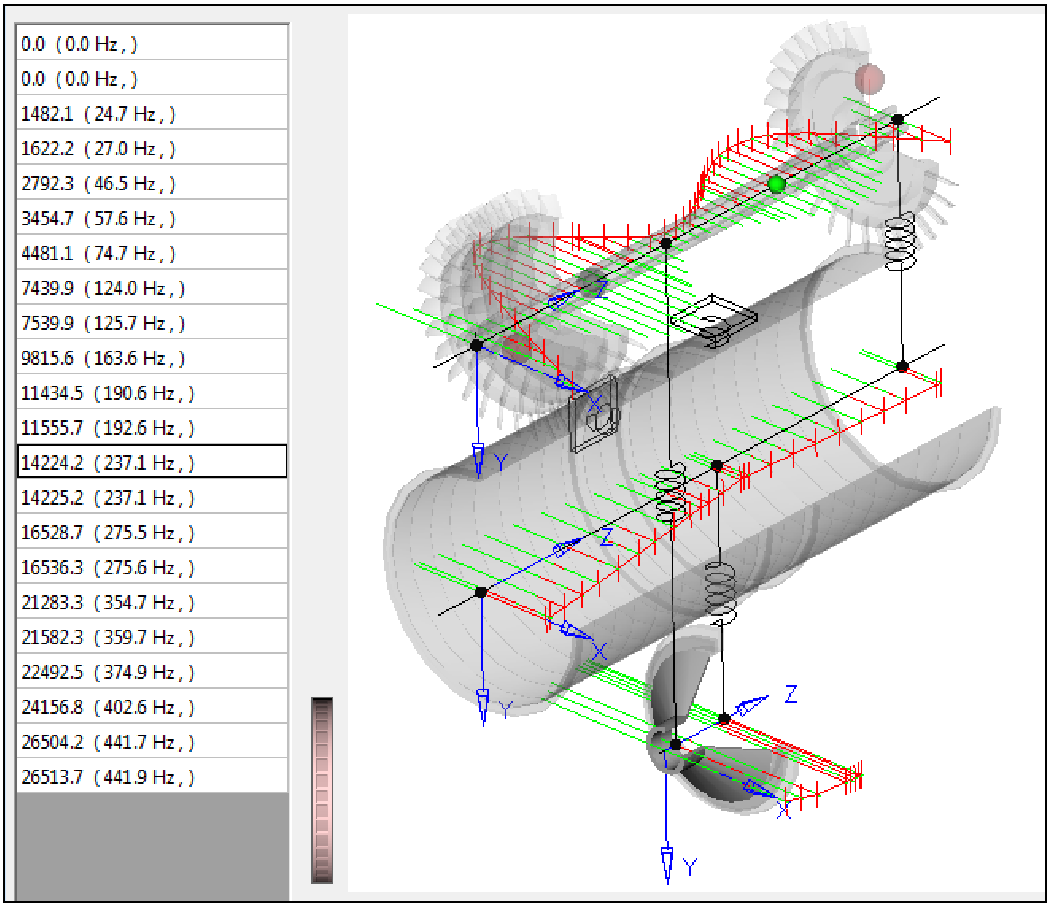


Fig. 2.12

Critical speeds:

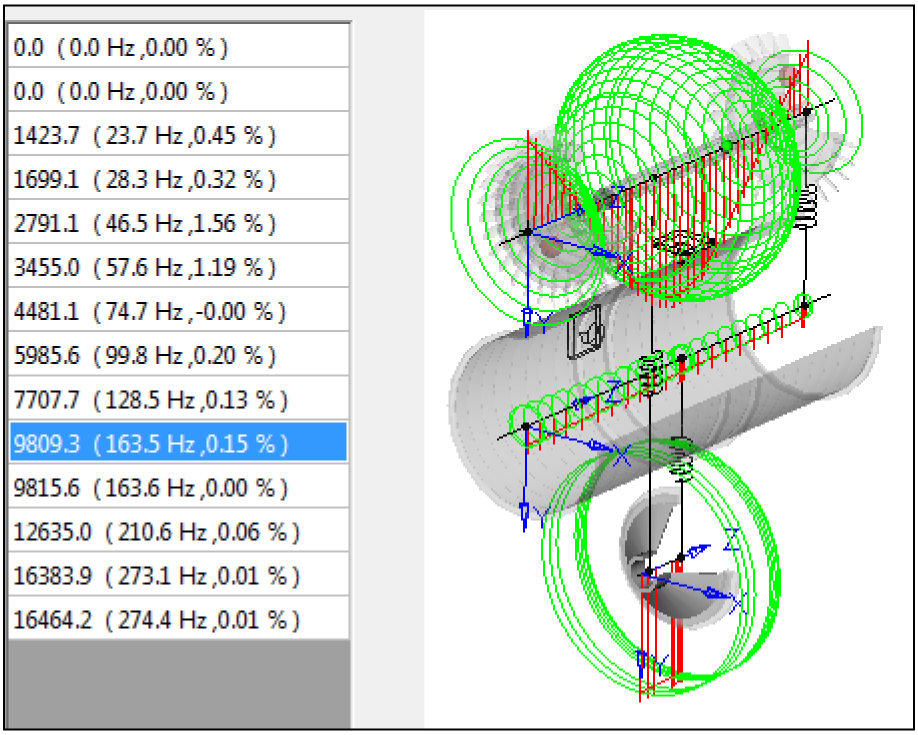


Fig. 2.13

Natural frequency map:

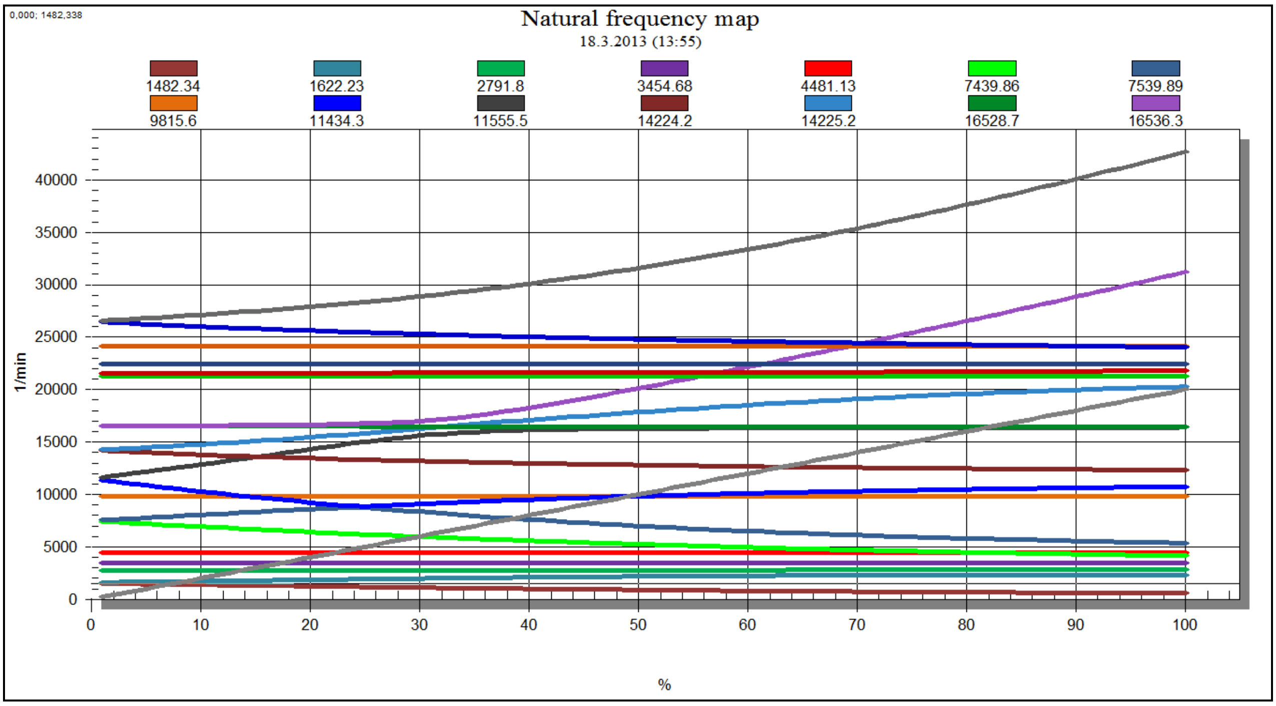


Fig. 2.14

3. Recommendations for cases modeling using FEM

When modeling cases (pylons, other complex beam elements), it is necessary to take into consideration unequal change in the beam cross section. For this purpose the model should be divided into sections. Let this consider by example of a simple beam.

In some standards (for example, API) for modeling of complex section it is suggested to use beam elements with selected diameter and density for coincidence of mass-inertia characteristics. Errors on stiffness determination are considered acceptable in this case. It should be noticed that such methodology gives the minimum error only for bending oscillations.

3.1 Model data preparation

3.1.1 Obtainment of mass-inertia characteristics of detail

It is necessary to model the structure of our interest (in this case it is thin-walled beam of the rectangular cross section (Fig. 3.1)). If the structure is fixed not only at the boundary parts, it should be divided into separate sections in the places of additional links connection; mass-inertia characteristics of these sections halves, at which the beam will be divided, should also be obtained. In this case the beam is divided into 3 sections (2064 mm, 649 mm, 1980 mm), mass-inertia characteristics are obtained for halves of every section, Table 3.1. Material properties are: Rho= 4500 kg/m3, Nue=0.3, E= 1.1e+11 N/m2.

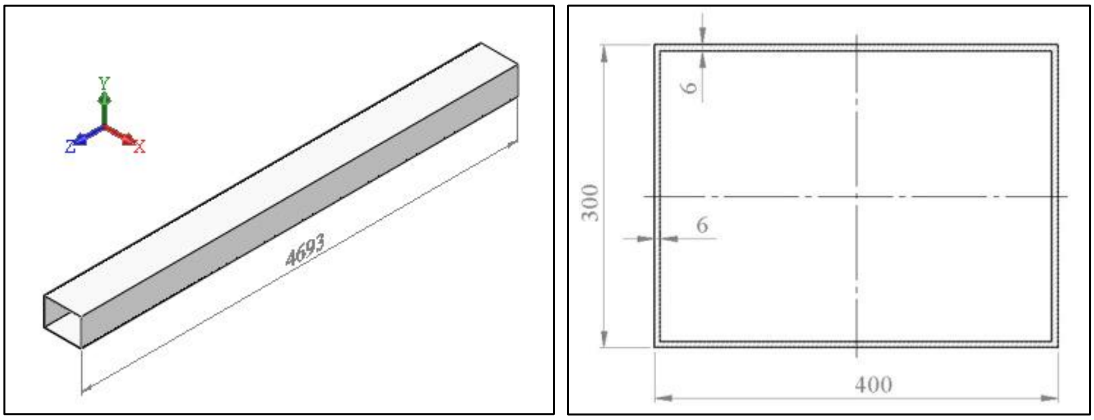


Fig. 3.1

Table 3.1

characteristics of 1 section half			characteristics of 3 section half		
mass	38.102	kg	mass	36.551	kg
Ixx =	14.116	kg*m2	Ixx =	12.506	
Iyy =	14.441		Iyy =	12.506	kg*m2
Izz =	1.503		Izz =	1.442	
characteristics of 2 section half			characteristics of complete beam		
mass	11.981	kg	mass	173.269	kg
Ixx =	0.606	kg*m2	Ixx =	320.689	
Iyy =	0.708		Iyy =	322.169	kg*m2
Izz =	0.473		Izz =	6.837	

3.1.2 Stiffness matrixes

Let us consider flexibility matrixes obtainment by example of the shaft section shown in

. To obtain flexibility matrixes it is necessary to import the 3D model in the finite-element system and divide it into finite elements (Fig. 3.2). For this purpose it is preferable to export the 3D model from CAD-system into the Parasolid (*.x_t) format, and then import it as a solid body (Solid). The model should be oriented in correspondence with the coordinate system in Dynamics R4. Material characteristics are set in the finite-element system. After that it is necessary to superimpose the detailed finite element mesh for static loading. It should be noticed that for this task it is necessary to use a hexagonal type of the finite element mesh.

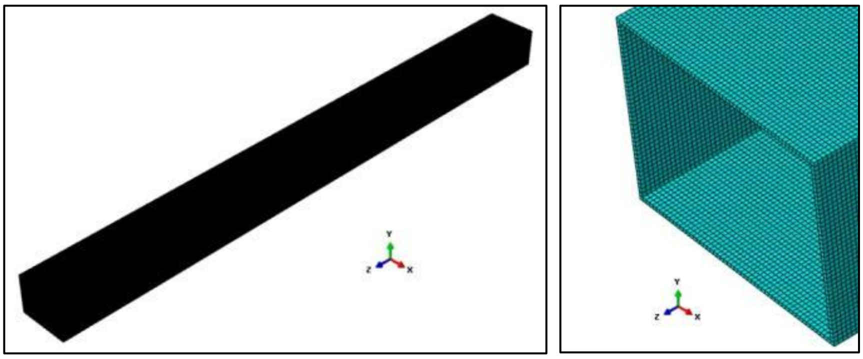


Fig. 3.2

Fixing is done along the whole plane of the left boundary section, the right one is loaded in turn by unit forces and moments. Planes of loading and fixing are rigid. Fig. 3.3 shows the loading scheme.

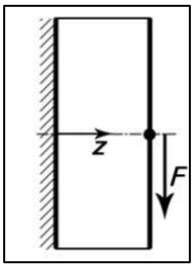


Fig. 3.3

Flexibility matrix is obtained for every section separately by successive measurement of displacements and rotations about all the axes for all loading steps, Table 3.2. The important values are highlighted in yellow.

Table 3.2

Flexibility of 1 section (2064 mm)

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1,4020E-04	-4,9390E-16	3,6073E-19	3,0855E-19	9,4016E-08	1,2950E-20
Fy [N]	-2,6646E-15	2,1661E-04	3,4908E-18	-1,4581E-07	-1,7622E-18	-1,8059E-20
Fz [N]	5,5935E-16	-9,2285E-16	2,1837E-06	7,3859E-19	3,2864E-19	3,7389E-21
Mx [N mm]	2,4176E-18	-1,4581E-07	2,1646E-18	1,4129E-10	2,0042E-21	2,1793E-21
My [N mm]	9,4016E-08	1,8857E-19	7,0670E-22	-2,0509E-22	9,1101E-11	1,1165E-22
Mz [N mm]	-1,0533E-18	3,8615E-19	-9,8697E-22	-2,4166E-22	-7,3370E-22	1,9845E-10

Flexibility of 2 section (649 mm)

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	7,4042E-06	-8,3196E-19	2,2237E-20	1,4946E-21	9,2305E-09	-1,0930E-21
Fy [N]	-2,5318E-18	1,1125E-05	3,4372E-20	-1,4280E-08	-4,6818E-21	-1,8790E-22
Fz [N]	7,6859E-19	1,1723E-18	6,8023E-07	-2,0944E-21	1,2572E-21	1,5724E-21
Mx [N mm]	9,4846E-20	-1,4280E-08	-3,6105E-20	4,4007E-11	2,6693E-22	-3,9820E-23
My [N mm]	9,2305E-09	-4,6074E-21	-4,8881E-22	8,8405E-24	2,8445E-11	1,8670E-24
Mz [N mm]	-4,9498E-21	-2,3441E-22	-3,4289E-23	-1,6717E-24	-9,9100E-24	6,1943E-11

Flexibility of 3 section (1980 mm)

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1,2458E-04	-3,5211E-16	-5,8379E-18	2,3012E-19	8,6507E-08	9,3650E-20
Fy [N]	-1,7670E-15	1,9242E-04	-7,4964E-19	-1,3416E-07	-1,1185E-18	-2,4404E-19
Fz [N]	3,0684E-16	5,5410E-15	2,0944E-06	-3,6471E-18	1,7957E-19	4,9371E-19
Mx [N mm]	2,3285E-18	-1,3416E-07	1,5128E-18	1,3551E-10	1,7179E-21	1,3087E-21
My [N mm]	8,6507E-08	-3,3001E-18	-6,0897E-21	2,1482E-21	8,7381E-11	-5,5306E-23
Mz [N mm]	-9,6214E-19	9,3567E-20	1,0536E-21	-7,0143E-23	-6,4075E-22	1,9035E-10

Flexibility of complete beam

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1,5475E-03	-1,4478E-13	-4,9637E-17	4,1640E-17	4,8692E-07	5,1521E-18
Fy [N]	-4,9127E-13	2,3995E-03	-9,5813E-17	-7,5563E-07	-1,4473E-16	-8,4326E-18
Fz [N]	1,6749E-13	-5,4992E-13	4,9770E-06	1,7779E-16	4,7955E-17	5,2527E-18
Mx [N mm]	1,1936E-16	-7,5563E-07	1,2867E-17	3,2202E-10	3,8903E-20	1,3052E-20
My [N mm]	4,8692E-07	7,7896E-17	-1,4579E-19	-2,7019E-20	2,0751E-10	-2,1262E-21
Mz [N mm]	-1,0487E-16	4,6829E-17	-1,5799E-20	-1,3606E-20	-3,1172E-20	4,5176E-10

Then it is necessary to invert the obtained flexibility matrixes in order to get the stiffness matrixes (Table 3.3) – they will be used at links modeling in Dynamics R4. Inversion may be done in Dynamics R4 matrix calculator: the “Service” menu -> “Matrix calculator”

Table 3.3

Stiffness matrix of 1 section

	Fx [N]	Fy [N]	Fz [N]	Mx [N mm]	My [N mm]	Mz [N mm]
ut_x [mm]	2,3164E+04	5,3118E-08	3,9098E-09	-3,0468E-05	-2,3905E+07	1,1938E-05
ut_y [mm]	1,7003E-07	1,5121E+04	-1,5492E-05	1,5604E+07	-2,2628E-04	-1,6998E-04
ut_z [mm]	-2,3358E-06	1,1122E-06	457944	-1,2461E-03	7,5852E-04	-8,6279E-06
ur_x [rad]	1,1820E-04	1,5604E+07	-2,3004E-02	2,3182E+10	-3,3012E-01	-2,5315E-01
ur_y [rad]	-2,3905E+07	-5,0987E-05	-7,5873E-06	5,1330E-02	3,5647E+10	-1,8496E-02
ur_z [rad]	3,4565E-05	-1,0420E-05	2,2776E-06	-2,1349E-03	4,9124E-03	5,0391E+09

Stiffness matrix of 2 section

	Fx [N]	Fy [N]	Fz [N]	Mx [N mm]	My [N mm]	Mz [N mm]
ut_x [mm]	226813	-7,5905E-09	-6,0304E-08	4,6194E-06	-7,3601E+07	6,2206E-06
ut_y [mm]	-5,7921E-08	154068	2,6458E-06	4,9995E+07	-4,2500E-04	3,2607E-05
ut_z [mm]	-1,2025E-07	-1,1158E-07	1,4701E+06	3,3756E-05	-2,5955E-05	-3,7317E-05
ur_x [rad]	-6,1203E-05	4,9995E+07	2,0647E-03	3,8947E+10	-3,3739E-01	2,5189E-02
ur_y [rad]	-7,3601E+07	1,1880E-05	4,4831E-05	-5,5054E-03	5,9039E+10	-3,0782E-03
ur_z [rad]	6,3495E-06	1,9323E-06	8,1378E-07	1,2403E-03	3,5640E-03	1,6144E+10

Stiffness matrix of 3 section

	Fx [N]	Fy [N]	Fz [N]	Mx [N mm]	My [N mm]	Mz [N mm]
ut_x [mm]	2,5679E+04	-4,4694E-07	-2,3412E-09	-8,3072E-05	-2,5423E+07	-2,0020E-05
ut_y [mm]	1,6453E-08	16777	-1,1991E-05	1,6609E+07	-1,2808E-04	-9,2687E-05
ut_z [mm]	-1,5824E-06	-1,5463E-05	477458	-2,4586E-03	5,8537E-04	-1,2384E-03
ur_x [rad]	-1,0267E-04	1,6609E+07	-1,7201E-02	2,3823E+10	-1,5410E-01	-1,4250E-01
ur_y [rad]	-2,5423E+07	6,6777E-04	3,5593E-05	1,2387E-01	3,6612E+10	2,3145E-02
ur_z [rad]	4,4221E-05	-2,1264E-06	-2,6427E-06	6,1414E-04	-5,2553E-03	5,2535E+09

Stiffness matrix of complete beam

	Fx [N]	Fy [N]	Fz [N]	Mx [N mm]	My [N mm]	Mz [N mm]
ut_x [mm]	2,4694E+03	3,1968E-07	-1,4511E-07	-5,5372E-05	-5,7945E+06	-5,5435E-05
ut_y [mm]	3,3812E-07	1,5964E+03	-9,6537E-06	3,7459E+06	-3,8220E-04	-7,8422E-05
ut_z [mm]	-2,7272E-05	4,2575E-05	200926	-1,1031E-02	1,7560E-02	-2,3362E-03
ur_x [rad]	5,7808E-04	3,7459E+06	-3,0681E-02	1,1895E+10	-9,7379E-01	-2,7373E-01
ur_y [rad]	-5,7945E+06	-8,6164E-04	4,8167E-04	2,7258E-01	1,8416E+10	1,5276E-01
ur_z [rad]	1,7340E-04	-5,2660E-05	7,0268E-06	-3,0040E-02	-7,4366E-02	2,2136E+09

3.2 Modeling

Let us build the model in Dynamics R4 (Fig. 3.4). Every section is represented by a link between two subsystems, each one has mass and link point. For example, if there was one section, it would be simulated by two subsystems spaced from each other to the distance equal to the section length. Each subsystem would have mass equal to the half of the section mass and link point. There would be a link with the given stiffness matrix identical to the stiffness matrix of the complete beam.

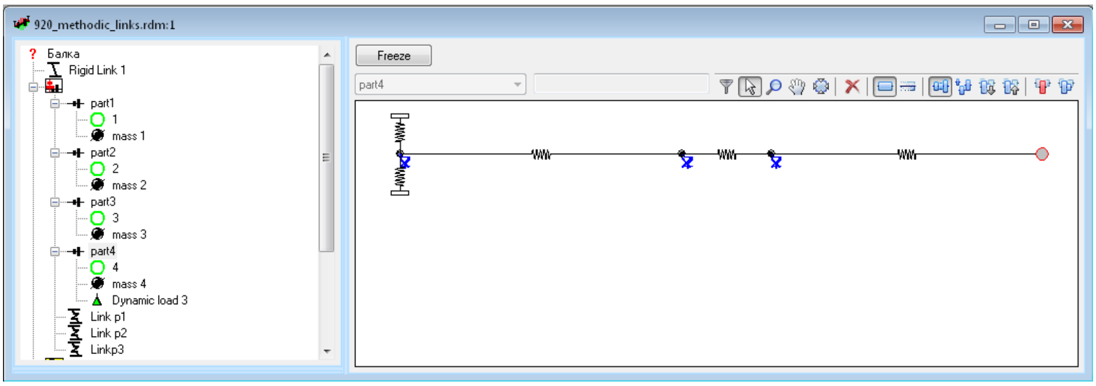


Fig. 3.4

It should also be noticed that at connection of links between subsystems, the loading section should be attached to the "side1_c_point "parameter, fixing section – to the "side2_c_point" parameter.

As in our case the beam has two intermediate sections, to each of which two sections will be related at once, so characteristics of the lumped mass in this section (mass and inertia moments) must be summed up from the characteristics of the halves of the corresponding sections (Table 3.4).

Table 3.4

mass characteristics between 1 and 2 sections			mass characteristics between 2 and 3 sections		
mass	50,083	kg	mass	48,532	kg
Ixx =	14,722	kg*m2	Ixx =	13,112	kg*m2
Iyy =	15,149		Iyy =	13,214	
Izz =	1,976		Izz =	1,915	

In links between subsystems we set stiffness matrixes obtained in FEM. Dynamics R4 has the following feature – if the values in flexibility matrix have high accuracy (the number of decimal digits more than 4 at exponential number format), then after setting stiffness matrix, the repeated opening of the corresponding window with matrix should be avoided. If it is necessary, the window should be closed by pressing the button "Cancel". Otherwise, Dynamics R4 keeps just 4 decimal digits.

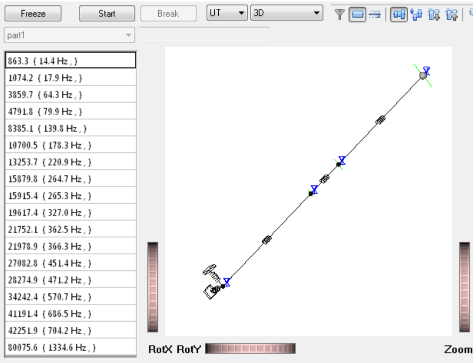


Fig. 3.5

After building the model, we calculate basis (Fig. 3.5). Every unfixed point has 6 freedom degrees, so it may be necessary to extend the basis range in order to correspond the frequency number with the number of freedom degrees. Unfixed generalized element has 2 stations, each one has 6 freedom degrees. Correspondingly, unfixed generalized element has 12 freedom degrees, fixed one on the left– 6, the whole section – 18. In this calculation the basis range is up to 100 000 1/min.

3.3 Model check

To calculate flexibility matrix it is necessary to add the "dynamic load" element into the model, placed it into the subsystem corresponding to the right boundary station as for loading of the whole beam. As dynamic load we set the unit force along X-axis. When setting the unit moment, it is necessary to determine dimension (N*mm) in correspondence with dimension set in the FEM-system and stiffness matrix of links. It is necessary to add [Transient response] in the algorithms, choose displacement along X-axis and press the button [Start]. When calculation is finished, we choose the last section [Closing Section], select any point on the horizontal part of the plot and press the "A" button on the Toolbar – Flexibility of the chosen section depending on the given force in dynamic loading will be written in the [Log]. The log line, selected in Fig. 3.6, is the first line of flexibility matrix.

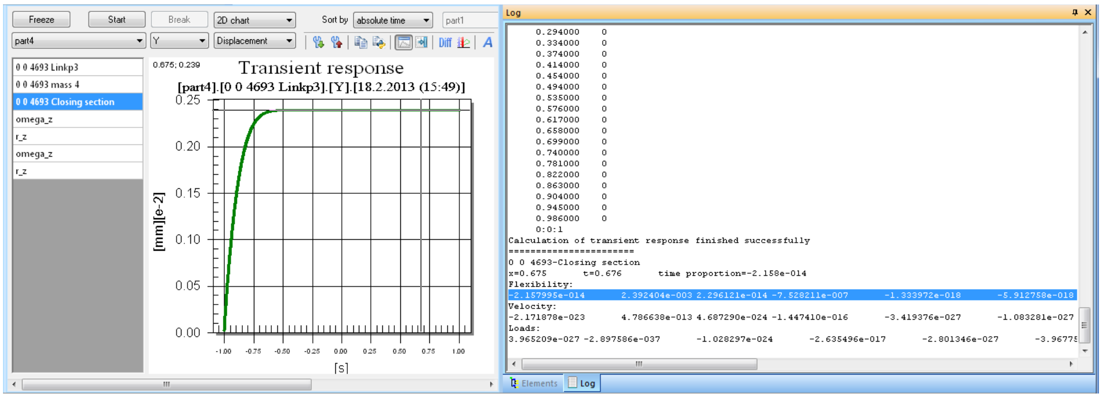


Fig. 3.6

Fig. 3.7 gives the settings of nonlinear analysis.

Note: when there is no horizontal section on the plot, it is necessary to vary damping at initial integration time (“init_damping” in the characteristics of [Transient response]).

After obtainment of the first line of flexibility matrix (radial flexibility relatively X-axis), in dynamic load we zero force along X and set the unit force along Y axis, in transient analysis calculation we change from displacement along X to displacement along Y and obtain the next line. Similar actions are also necessary for flexibility relatively Z axis. In order to obtain moment flexibility, it is necessary to set the unit moment in dynamic load and rotation about the corresponding axis in transient response calculation.

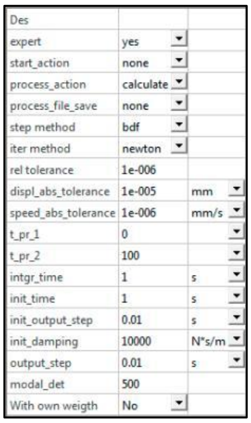


Fig. 3.7

To make a conclusion about correctness of modeling and calculation, it is necessary to compare the flexibility matrixes of the complete beam obtained in FEM and Dynamics R4, getting error less than 5% (Table 3.5).

Table 3.5

Flexibility of complete beam (FEM)

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1,5475E-03	-1,4478E-13	-4,9637E-17	4,1640E-17	4,8692E-07	5,1521E-18
Fy [N]	-4,9127E-13	2,3995E-03	-9,5813E-17	-7,5563E-07	-1,4473E-16	-8,4326E-18
Fz [N]	1,6749E-13	-5,4992E-13	4,9770E-06	1,7779E-16	4,7955E-17	5,2527E-18
Mx [N mm]	1,1936E-16	-7,5563E-07	1,2867E-17	3,2202E-10	3,8903E-20	1,3052E-20
My [N mm]	4,8692E-07	7,7896E-17	-1,4579E-19	-2,7019E-20	2,0751E-10	-2,1262E-21
Mz [N mm]	-1,0487E-16	4,6829E-17	-1,5799E-20	-1,3606E-20	-3,1172E-20	4,5176E-10

Flexibility of complete beam (Dynamics)

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1,5443E-03	-2,1577E-14	-6,5898E-16	4,0391E-18	4,8558E-07	4,0274E-19
Fy [N]	-2,1580E-14	2,3924E-03	2,2961E-14	-7,5282E-07	-1,3340E-18	-5,9128E-18
Fz [N]	-5,4496E-16	1,5252E-14	4,9583E-06	-4,8640E-18	-2,2283E-19	4,9902E-19
Mx [N mm]	4,0398E-18	-7,5282E-07	-7,1949E-18	3,2081E-10	-1,2667E-21	3,4482E-21
My [N mm]	4,8558E-07	-1,3339E-18	-2,5753E-19	-1,2667E-21	2,0693E-10	5,8213E-23
Mz [N mm]	4,0274E-19	-5,9128E-18	4,9902E-19	3,4482E-21	5,8213E-23	4,5074E-10

Comparison

	ut_x [%]	ut_y [%]	ut_z [%]	ur_x [%]	ur_y [%]	ur_z [%]
Fx [%]	0,21				0,28	
Fy [%]		0,30		0,37		
Fz [%]			0,37			
Mx [%]		0,37		0,38		
My [%]	0,28				0,28	
Mz [%]						0,23

Table 3.6 compares mass-inertia characteristics of the models built in CAD-system and Dynamics.

Table 3.6

	Parameter		Value
CAD	Mass		173.269
	Inertia moments	I_{xx}, kg*m^2	320.689
		I_{yy}, kg*m^2	322.169
		I_{zz}, kg*m^2	6.837
Dynamics R4	Mass		173.268
	Inertia moments	I_{xx}, kg*m^2	476.016
		I_{yy}, kg*m^2	477.496
		I_{zz}, kg*m^2	6.836

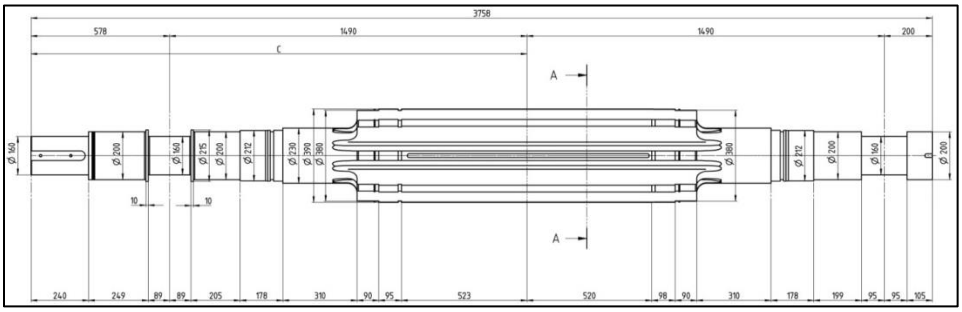
4. Recommendations to model section of rotor shaft using FEM

Shaft sections, stiffness characteristics of which cannot be modeled by [Beam] and [Shell] elements in Dynamics R4, are modeled with the [Generalized element] element. At modeling a body of rotation with its help it is necessary to take into account the following limitations: the detail should be isotropic at mass-inertia and stiffness characteristics. It is possible to set only isotropic flexibility matrix in the element. Orthotropic inertia moments may be also set but such element is impossible to use at modeling of shafts with rotation.

In some standards (for example, API RP684) in order to model complex sections it is suggested to use beam elements with matched diameter and density for coincidence of mass- inertia characteristics. Errors on stiffness characteristics determination are considered acceptable. It should be noticed that such methodology is recommended for determination of bending vibrations.

4.1 Data preparation for model

4.1.1 Obtainment of mass-inertia shaft characteristics



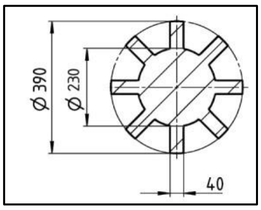


Fig. 4.1

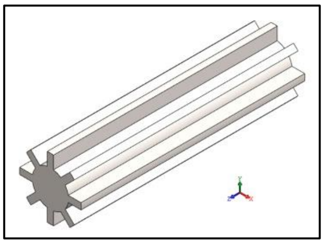


Fig. 4.2

Usually initial data for modeling are the drawing of the shaft and section of the part of interest, Fig. 4.1(grooves of the instrument entry and butts rounding are simplified).

From these drawings it is necessary to build 3D model of the central shaft section (Fig. 4.2) in CAD-system and obtain its mass-inertia characteristics in the centre of mass (Table 4.2), set the corresponding material (here TStE 420N, Table 4.1). These data should be used to verify the model.

Table 4.1

Parameter	Value
Modulus of elasticity	2.1e+011 N/m^2
Poisson's ratio	0.28
Mass density	7800 kg/m^3

Table 4.2

Parameter		Value
Mass		742.479 kg
Inertia moments	I_{xx}	129.058 kg*m^2
	I_{yy}	129.058 kg*m^2
	I_{zz}	9.997 kg*m^2

4.1.2 Flexibility matrixes obtainment

Let us consider flexibility matrixes obtainment by example of section of the shaft, given in Fig. 4.1 To obtain flexibility matrix it is necessary to import the 3D model in the finite-element system and divide it into finite elements, Fig. 4.3. It is recommended to export 3D model from CAD-system into Parasolid (*.x_t) format and then to import it as a solid body (Solid). The model should be oriented in correspondence with the coordinate system in Dynamics R4. Material characteristics are set in the finite-element system. After that the detailed finite element mesh for static loading is superimposed. In this case there is no task of optimal (speed-accuracy) partition. In this case the model was divided automatically into 111044 elements. It should be noticed that for such task the hexagonal type of finite element mesh must be used.

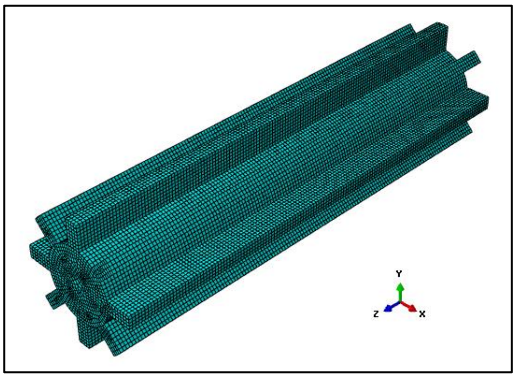


Fig. 4.3

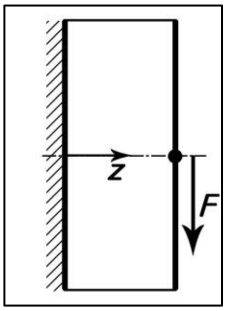


Fig. 4.4

Fixing performed at the whole plane of the left boundary station, right one is loaded in turn by unit forces and moments. Planes of loading and fixing are rigid. Fig. 4.4 shows the scheme of loading.

Flexibility matrix is obtained by successive measurement of displacements and rotations along all axes for all loading steps, Table 4.3. This matrix will be used to check the model in Dynamics R4.

Table 4.3

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1.0241E-05	-1.4373E-10	5.1529E-13	1.6650E-13	1.0510E-08	2.6945E-13
Fy [N]	-1.4373E-10	1.0241E-05	2.5419E-13	-1.0511E-08	-1.6650E-13	-1.0717E-12
Fz [N]	5.1530E-13	2.5419E-13	9.9417E-08	-3.5903E-16	7.2782E-16	-1.3094E-21
Mx[N mm]	1.6650E-13	-1.0511E-08	-3.5904E-16	1.4845E-11	2.3517E-16	2.0710E-24
My[N mm]	1.0510E-08	-1.6650E-13	7.2781E-16	2.3517E-16	1.4845E-11	-1.7502E-24
Mz[N mm]	2.6945E-13	-1.0717E-12	-5.0445E-23	-5.6205E-24	-1.2285E-22	5.2566E-11

4.1.3 Features of lengthy shaft section

For correct modeling of bending vibrations of lengthy section it is necessary to divide the whole section into several segments. In this case length of the whole section is 1416 mm, let us divide it into 12 segments of 118 mm length each one.

We obtain flexibility matrix of the segment, as it was described above. When the segment model is divided into finite elements, mesh density should be the same as at division of the complete shaft section. It simplifies verification. The number of finite elements, the model was divided into, accounted for 9312. Fig. 4.5 shows the segment. Table 4.4 gives flexibility matrix.

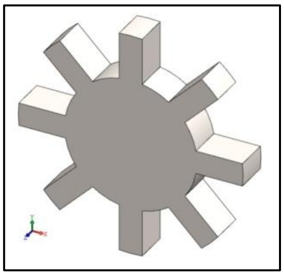


Fig. 4.5

Table 4.4

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	3.1937E-08	-1.5548E-13	1.9949E-14	2.4542E-15	6.8619E-11	6.6410E-15
Fy [N]	-1.5548E-13	3.1935E-08	1.8739E-14	-6.8620E-11	-2.4542E-15	-4.1299E-14
Fz [N]	1.9949E-14	1.8739E-14	7.4636E-09	-3.1761E-16	3.3812E-16	-1.1878E-23
Mx [N mm]	2.4542E-15	-6.8620E-11	-3.1761E-16	1.1631E-12	4.1596E-17	1.2175E-27
My [N mm]	6.8619E-11	-2.4542E-15	3.3812E-16	4.1596E-17	1.1630E-12	8.8888E-26
Mz [N mm]	6.6410E-15	-4.1299E-14	-1.4249E-23	-1.1447E-26	1.0925E-25	3.1851E-12

When [Generalized element] is used, there are 2 methods to set mass-inertia characteristics. When the first one is used, it is enough to know masses and inertia moments in the centre of mass of the segment (LG is the parameter to set the coordinates of the centre of mass relatively the segment beginning). It should be used if there is no possibility to use the second method. When the second method is used, masses and inertia moments are set on the boundary stations of the segment. To obtain them, it is necessary to model the segment half and to obtain the mass-inertia characteristics not in the centre of mass but on the boundary section. In our case length of the segment half is equal to 59 mm. Table 4.5 gives its mass-inertia characteristics.

Table 4.5

Parameter		Value	Units
Mass (m)		30.937	kg
Inertia moments	I_{xx}	0.244	$kg*m^2$
	I_{yy}	0.244	
	I_{zz}	0.417	

4.2 Modeling

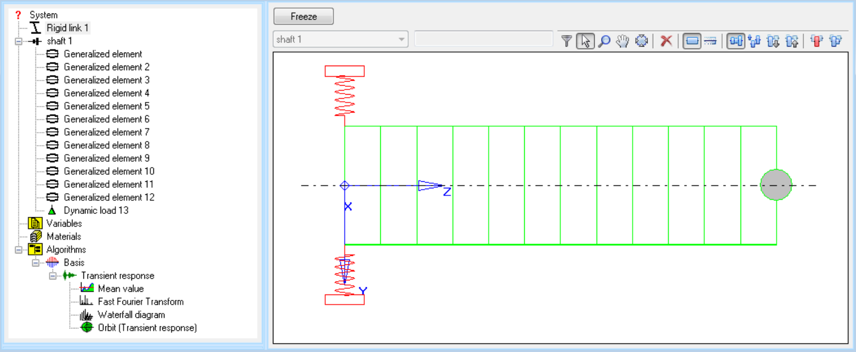


Fig. 4.6

We build the new model. Then we add the subsystem (Fig. 4.6), where we create the [Generalized element] of 118 mm length with mass-inertia characteristics in boundary sections, obtained before for the segment of 59 mm (Fig. 4.7), and flexibility matrix obtained in FEM for the segment of the shaft section. Fig. 4.8 shows the schematic diagram of the generalized element. Copying flexibility matrix, beforehand it is necessary to specify the corresponding size of moment coefficients (in this case N*mm) in Dynamics.

Note: In Dynamics R4 when editing matrixes cells, rounding up to 4 decimal digits happens. If data in FEM are obtained with higher accuracy and it is required to minimize error at verification, this should be taken into account. In dynamics tasks such rounding does not bring any significant error. In case if it is necessary to look over the matrix and to save accuracy of entry, editing window should be closed using the “Cancel” button.

Des	Generalized element 3		Designation
segRef	z1 and z2		Measurement
ls	0	mm	Length of element
z1	118	mm	Start coordinate
z2	118	mm	End coordinate
type	general		Type of element
type	Orthotropic		Type
inertia_cos	boundary		Coordinate system fo
m1	30.937	kg	Mass1
Jx1	0.244	kg m2	Jx1
Jy1	0.244	kg m2	Jy1
Jz1	0.417	kg m2	Jz1
S1	0.912649	kg m	Static moment of iner
m2	30.937	kg	Mass2
Jx2	0.244	kg m2	Jx2
Jy2	0.244	kg m2	Jy2
Jz2	0.417	kg m2	Jz2
S2	0.912649	kg m	Static moment of iner

Fig. 4.7

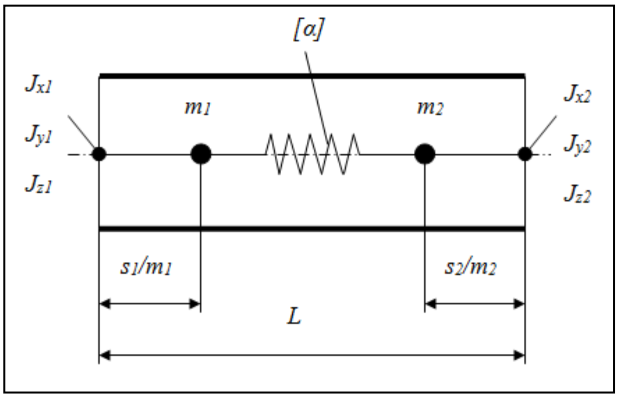


Fig. 4.8

On the schematic diagram of generalized element Jx1=Jx2, Jy1=Jy2, Jz1=Jz2, m1=m2 – inertia moments and segment mass of the shaft section of 59 mm length. Static inertia moments S1 and S2 are used for more detailed specification of mass distribution at the invariable number of sections. Static inertia moment is equal to product of an arm to mass. The arm may be obtained by the following way:

1) In dynamics build the new model with the beam segment of 118 mm length, 390 mm diameter, material - – TStE 420N .
2) Create the log, retrieve static inertia moment (S1 or S2) from it.
3) In calculator divide static inertia moment by the beam element mass (m1 or m2) and obtain the arm of static moment.
4) Multiply mass of the segment half (Table 4.6) by the obtained arm and write down the result in S1 and S2 fields in the generalized element.

When the unit generalized element is specified, it is necessary to calculate its basis. If basis is calculated without mistakes at sufficient frequency range, the generalized element is set correctly. We copy this element 11 times one by one.

After that we set Rigid link on the left end of the section, on the right– dynamic load of general type.

We calculate basis. Algorithms using methods of reduction of modal synthesis give maximal accuracy at basis calculation with the complete set of frequencies. In simple models, used for verifications, it is possible to get the complete basis. Every unfixed point has 6 degrees of freedom, so it is necessary to extend the basis range in order to correspond the number of frequencies with the number of freedom degrees. The unfixed generalized element has 2 points, each one having 6 degrees of freedom. Correspondingly, the unfixed generalized element has 12 degrees of freedom, the fixed one on the left – 6, in this case, the complete section – 72. The basis range of up to 700 000 1/min was chosen for this calculation.

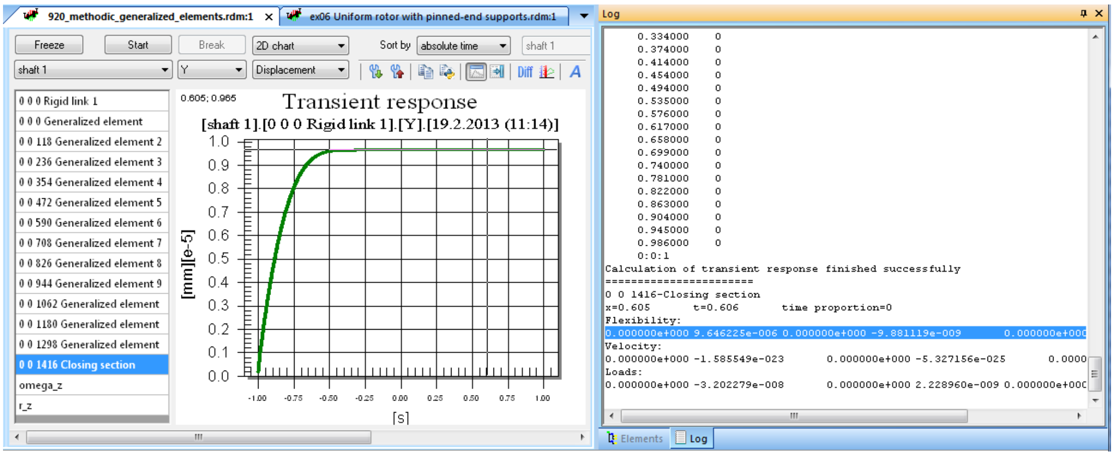
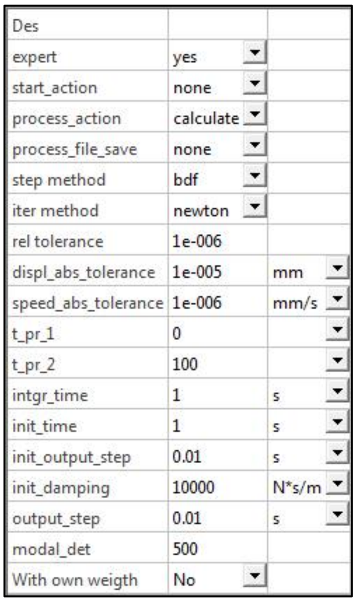


Fig. 4.9

In dynamic load we set unit force along X. When setting unit moment, it is necessary to specify the size (N*mm), in correspondence with the size of results obtained in FEM-system. It is necessary to add “Transient response” in algorithms, choose displacement along X axis and press «Start» button. When calculation is finished, we choose the last section (“Closing Section”), select any point on the horizontal part of the graph and press the “A“ on the Toolbar – in the log Flexibility of the chosen section depending on given force will be written in dynamic load. The log line highlighted in Fig. 4.9, is the first line of flexibility matrix. Fig. 4.10 shows settings of transient response.

Note: if there is no horizontal part on the graph, it is necessary to vary damping in initial integration time (“init_damping” in “transient response” characteristics).



Des		
expert	yes	
start_action	none	
process_action	calculate	
process_file_save	none	
step method	bdf	
iter method	newton	
rel tolerance	1e-006	
displ_abs_tolerance	1e-005	mm
speed_abs_tolerance	1e-006	mm/s
t_pr_1	0	
t_pr_2	100	
intgr_time	1	s
init_time	1	s
init_output_step	0.01	s
init_damping	10000	N*s/m
output_step	0.01	s
modal_det	500	
With own weigth	No	

Fig. 4.10

When the first line of flexibility matrix (radial flexibility along X axis) is obtained, in dynamic load we zero force along X and set unit force along Y, in calculation of nonlinear analysis we change displacement along X by displacement along Y and obtain the next line. Similar steps are also necessary for axial flexibility along Z-axis. To measure flexibility at angular freedom degrees, it is important to set in dynamic loads the necessary unit moment and in calculation of transient response – rotation relatively the corresponding axis.

Table 4.6 shows the flexibility matrix of the given model and its comparison with the one obtained in the 2 point. In this table the corrected torsion flexibility is presented.

Table 4.6

		ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Finite-element model	Fx [N]	1.0241E-05	-1.4373E-10	5.1529E-13	1.6650E-13	1.0510E-08	2.6945E-13
	Fy [N]	-1.4373E-10	1.0241E-05	2.5419E-13	-1.0511E-08	-1.6650E-13	-1.0717E-12
	Fz [N]	5.1530E-13	2.5419E-13	9.9417E-08	-3.5903E-16	7.2782E-16	-1.3094E-21
	Mx [N mm]	1.6650E-13	-1.0511E-08	-3.5904E-16	1.4845E-11	2.3517E-16	2.0710E-24
	My [N mm]	1.0510E-08	-1.6650E-13	7.2781E-16	2.3517E-16	1.4845E-11	-1.7502E-24
	Mz [N mm]	2.6945E-13	-1.0717E-12	-5.0445E-23	-5.6205E-24	-1.2285E-22	5.2566E-11
Dynamics R4	Fx [N]	9.6464E-06	0.0000E+00	0.0000E+00	0.0000E+00	9.8813E-09	0.0000E+00
	Fy [N]	0.0000E+00	9.6462E-06	0.0000E+00	-9.8811E-09	0.0000E+00	0.0000E+00
	Fz [N]	0.0000E+00	0.0000E+00	6.9901E-09	0.0000E+00	0.0000E+00	0.0000E+00
	Mx [N mm]	0.0000E+00	-9.8811E-09	0.0000E+00	1.3956E-11	0.0000E+00	0.0000E+00
	My [N mm]	9.8813E-09	0.0000E+00	0.0000E+00	0.0000E+00	1.3957E-11	0.0000E+00
	Mz [N mm]	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	3.1204E-12
Error		ut_x [%]	ut_y [%]	ut_z [%]	ur_x [%]	ur_y [%]	ur_z [%]
	Fx [%]	5.802288969	100	100	100	5.984791719	100
	Fy [%]	100	5.805153268	100	5.988173229	100	100
	Fz [%]	100	100	7.031154639	100	100	100
	Mx [%]	100	5.988173229	100	5.988320716	100	100
	My [%]	5.984991523	100	100	100	5.984864389	100
	Mz [%]	100	100	100	100	100	5.936263782

Error in modeling of torsion flexibility of the shaft with nonround section is conditioned by the fact that it is not additive for such crosssection. In order to obtain quite exact result at torsion flexibility relatively Z axis, it is necessary to do the following: to divide torsion flexibility of complete shaft section calculated in FEM, into the segments. The obtained value should be included into the corresponding field in flexibility matrix of the generalized element.

Table 4.7 gives comparison of mass-inertia characteristics, obtained for the complete shaft section in CAD-system and in Dynamics R4.

Table 4.7

	Parameter		Value
CAD	Mass		742.479 kg
	Inertia moments	I_{xx}	129.058 kg*m^2
		I_{yy}	129.058 kg*m^2
		I_{zz}	9.997 kg*m^2
Dynamics R4	Mass		742.488 kg
	Inertia moments	I_{xx}	131.472 kg*m^2
		I_{yy}	131.472 kg*m^2
		I_{zz}	10.008 kg*m^2

5. Conversion of [Rigid link] elements

As a result of incorrect modeling in complex models at active use of the [Rigid link] elements at basis calculation, shapes can be computed with improper orthogonality. Quite significant values in orthogonality matrix (1e-5…1) for nondiagonal elements testify this.

This may take place, for example, at refixing in several stations. Several rigid links match one or close stations. To check reasons of improper orthogonality, functionality of automatic conversion of [Rigid link] elements by equivalent elastic elements [Link] was included into the program. This allows improving orthogonality of mode shapes decreasing computational problems at building systems of equations.

As substitute, an equivalent elastic link is created. Meanwhile the previous link is deleted. Deletion of initial rigid links may be cancelled using Undo/Redo.

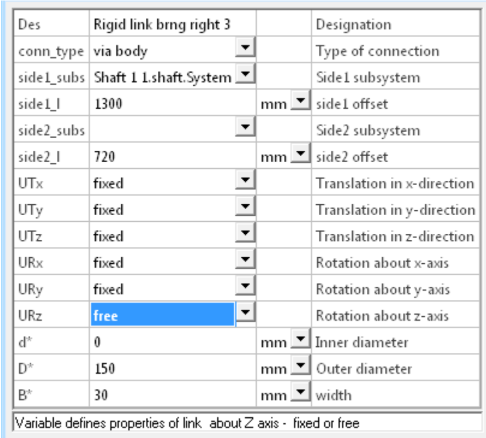


Fig. 5.1

When converting links, their attachment to subsystems is taken into consideration; there are visualization parameters d*, B*, D* and fixing type (Fig. 5.1).

In an equivalent elastic link the big stiffness is assigned only for those degrees of freedom which were fixed in a rigid link.

Current elastic link may be replaced using context menu (Fig. 5.2) for the element which is active in the model tree.

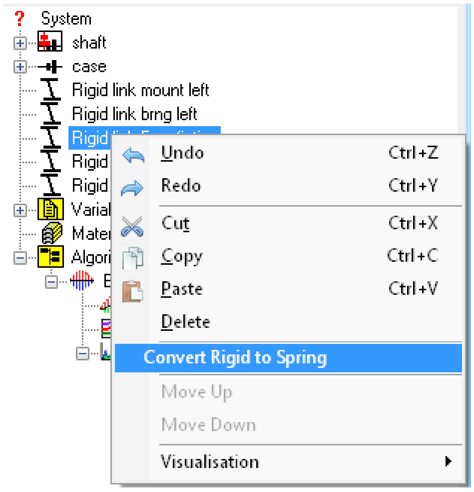
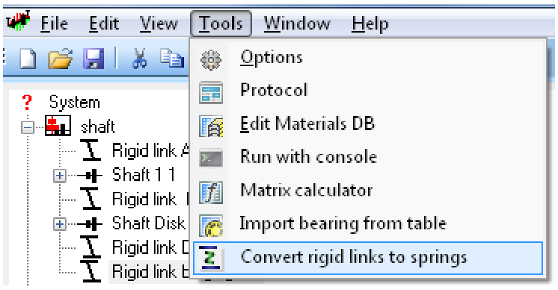


Fig. 5.2

The menu [Service\Convert rigid links by elastic links] may also be used.



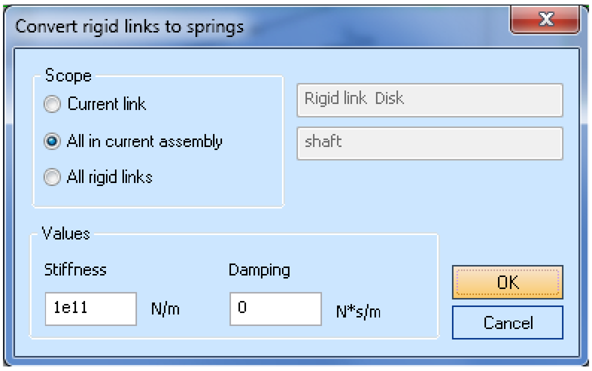


Fig. 5.3

Meanwhile the dialog window with conversion settings appears (Fig. 5.3).

Using these settings, equivalent stiffness (recommended values 1e10-1e12 N/m) and the type of the substitute may be set. Besides current link, all rigid links may be converted in the current assembly or in the whole model.

Note: such conversion does not eliminate problem in modeling, it helps to localize it. Search of mistake in modeling should not be stopped by converting rigid links into elastic ones and obtaining acceptable orthogonality of mode shapes.

6. Modeling of step change in shaft diameter [Shaft Stepping]

When modeling elements in Dynamics R4, assumption about absolutely rigid sections of elements is accepted. In some cases it may result in some inaccuracy of calculation. For example, at torsion of the shaft with step change in diameter there is an effect of penetration of the section with smaller diameter into the section with bigger diameter. It leads in some cases to difference of torsion flexibility obtained for beam elements in Dynamics R4 with experiment and FEM calculations at modeling without such assumption.

According to the requirements of API 684 standard [1], it is necessary to take into account penetration factor λ (also described as PF – “Penetration Factor”) when calculating flexibility for shaft torsion. Effect can be taken into consideration through lengthening of the smaller diameter shaft section with corresponding shortening of the bigger diameter shaft section by calculated λ. λ is calculated taking into account smaller diameter and ratio of bigger diameter to smaller one, according to the Table 6.1. It is reasonable to consider this effect at significant ratio D_2/D_1 (see Fig. 6.2), for example, $D_2/D_1 \geq 1.7$.

Starting from Dynamics R4.8.1 to consider the penetration effect, the corresponding element **[Shaft Stepping]** may be used. Fig. 6.1 gives dependence of error in calculation of torsion flexibility at constant ratio D_2/D_1.

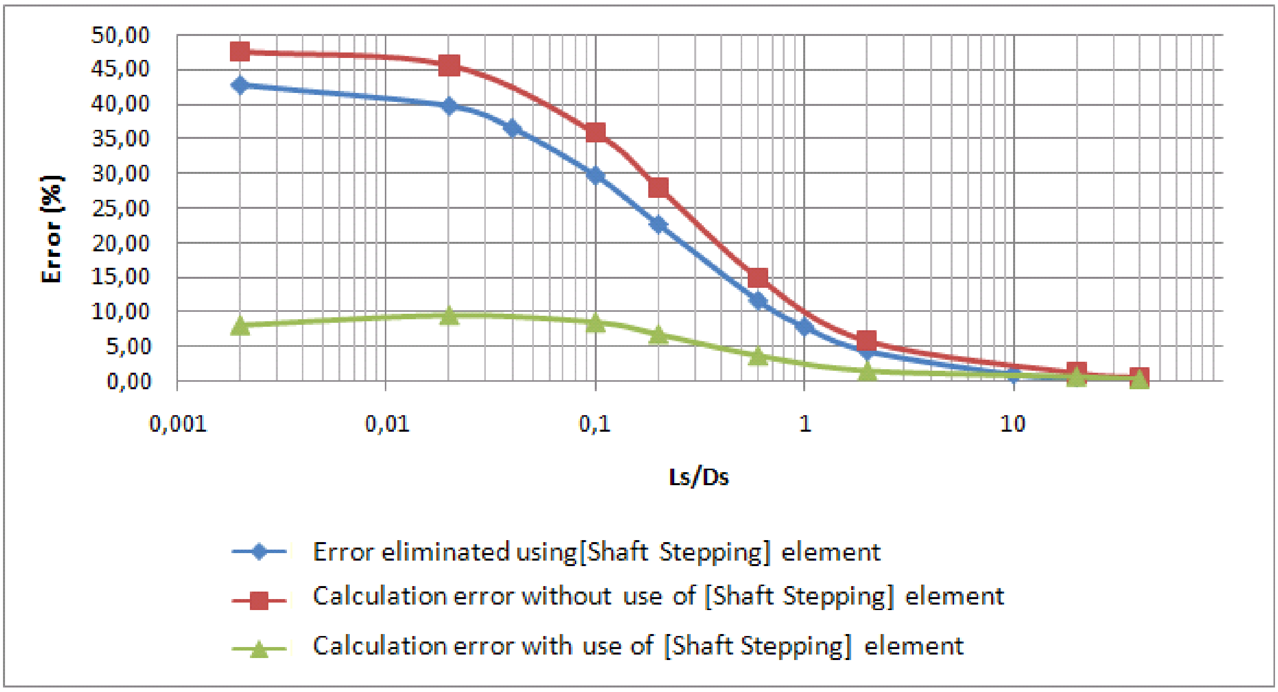


Fig. 6.1

In calculations in Dynamics R4 the penetration effect may be taken into account using either the element [Coupling] or the [Shaft Stepping] element.

6.1 Consideration of penetration effect using element [Coupling]

In Dynamics R4 additional flexibility, appearing at step change in diameter, may be considered using the following recommendations [1], [2]:

1) Calculate geometrical parameters of the beam (diameter D_e and length L_e), that is equivalent (in torsion flexibility) to the shaft section with the step;
2) Obtain torsion flexibility of equivalent beam ur_z$_e$ [rad];
3) Calculate difference Δur_z [rad] between torsion flexibility of the equivalent beam ur_z$_e$ [rad] and the shaft section with step change in diameter ur_z [rad], consider it in coupling.

6.1.1 Calculation of parameters of equivalent beam

In standard API 684 the equation (1) is given that is used to calculate geometry of the equivalent beam for case of step change in the shaft diameter.

$$L_e = \left(\frac{L_1+\lambda}{D_1^4} + \frac{L_2-\lambda}{D_2^4}\right) D_e^4\,, \tag{6.1}$$

where L_1, L_2, D_1, D_2 – geometrical parameters of the step shaft;
L_e, D_e – geometrical parameters of the equivalent beam;

Λ (Fig. 6.2) –penetration depth of the shaft section of length L_1 of smaller diameter D_1 into the section with bigger diameter D_2 and length L_2;

Fig. 6.2 and Table 6.1 present penetration factor PF= λ /D1 to smaller diameter vs ratio of bigger diameter D_2 to smaller one D_1 and also pictorial diagram of the penetration effect principle.

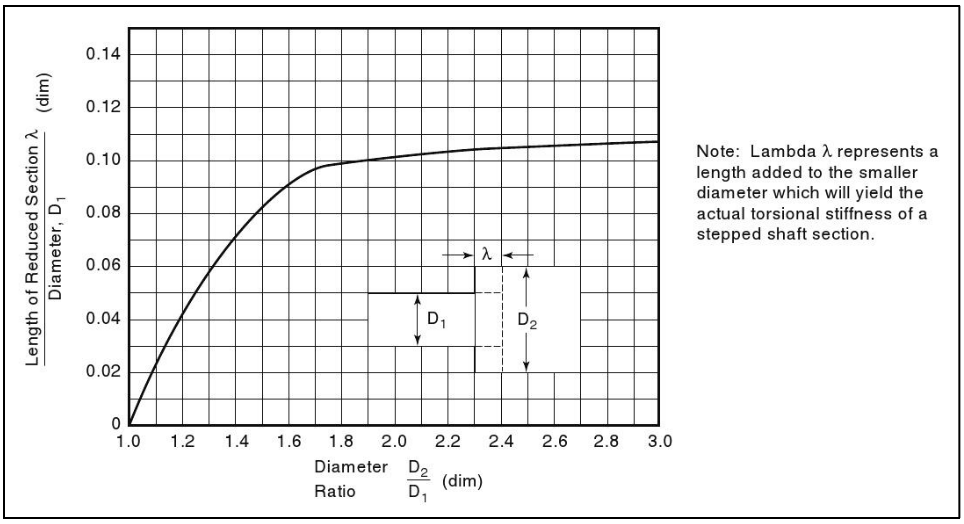


Fig. 6.2

Table 6.1

D_2/D_1	PF
1.00	0
1.25	0.055
1.50	0.085
2.00	0.100
3.00	0.107
∞	0.125

As the condition $D_1 < D_2$ should be met, the following new notations are introduced for ease in perception (Fig. 6.3):

- $L_1 = L_S$, $D_1 = D_S$ (“S = small” – the parameter concerns to the smaller diameter section);

- $L_2 = L_L$, $D_2 = D_L$ (“L = large” – the parameter concerns to the bigger diameter section).
- L_E, D_E – length and beam of the equivalent beam correspondingly.

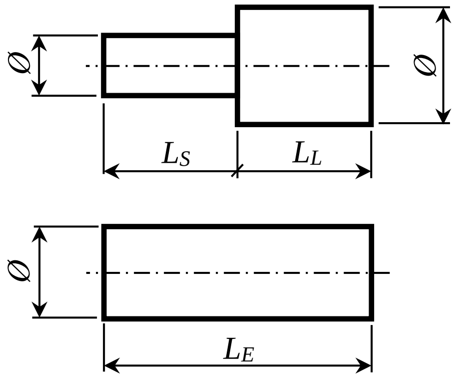


Fig. 6.3

It is better to set LE=LS+LL and calculate parameter DE using the equation. Thus, we obtain the following from (6.1):

$$D_E = \sqrt[4]{\frac{L_E}{\frac{L_S+PF}{D_S^4}+\frac{L_L-PF}{D_L^4}}} \tag{6.2}$$

6.1.2 Calculation of torsion flexibility of equivalent beam

To calculate torsion flexibility of the equivalent beam, it is necessary to build the model in Dynamics R4, presented in Fig. 6.4. Among the subsystem characteristics set the same material as for step shaft.

The step shaft is considered in this example. Table 6.2 gives its parameters.

Table 6.2

	Material			Geometry						
Parameter	ρ	E	μ	L_S	D_S	L_L	D_L	L_E	D_E	PF
Value	7850	210000	0.3	100	50	100	50	200	58.2145	0.1
Unit	kg/m^3	N/mm^2	---	mm						---

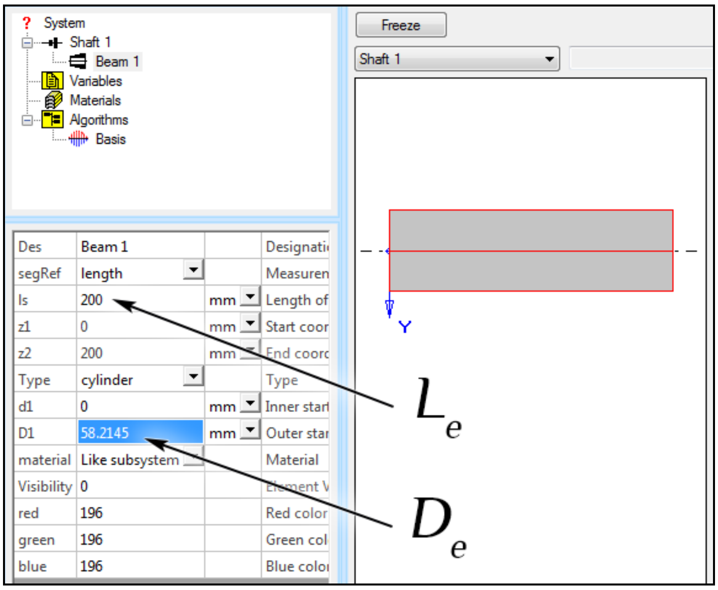
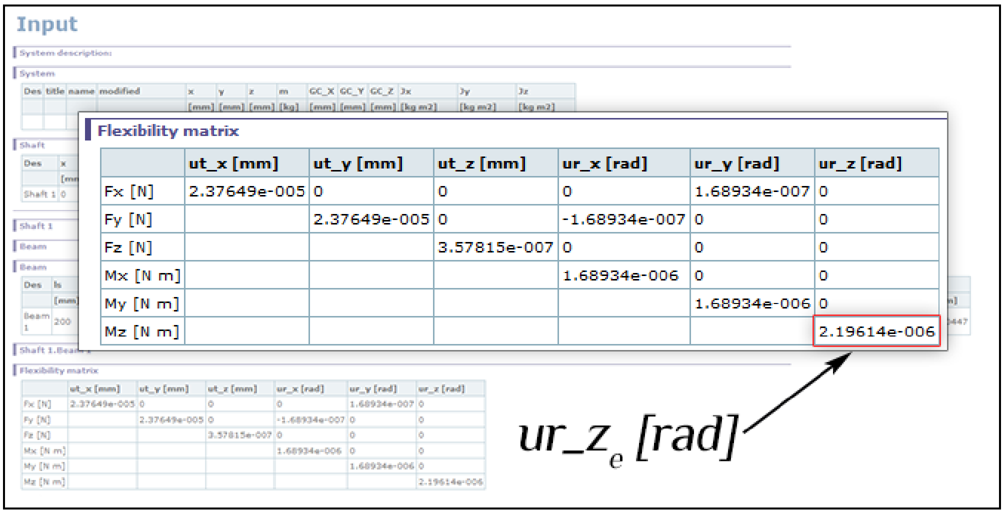


Fig. 6.4

After setting all the parameters it is necessary to create the model protocol in order to obtain flexibility ur_z_e [rad] (Fig. 6.5) from it.



Flexibility matrix

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	2.37649e-005	0	0	0	1.68934e-007	0
Fy [N]		2.37649e-005	0	-1.68934e-007	0	0
Fz [N]			3.57815e-007	0	0	0
Mx [N m]				1.68934e-006	0	0
My [N m]					1.68934e-006	0
Mz [N m]						2.19614e-006

Fig. 6.5

6.1.3 Calculation of Δur_z [rad] and its consideration in coupling

Δur_z [rad] is calculated in the following way:

$$\Delta\, ur\_z = ur\_z_e - ur\_z \qquad (6.3)$$

When modeling the step shaft in Dynamics R4, it is necessary to insert the [Coupling] element in the section where change in diameter takes place, insert the value Δur_z in its characteristics into the dURz/dMz area. Unit to be chosen is 1/N*m (see Fig. 6.6).

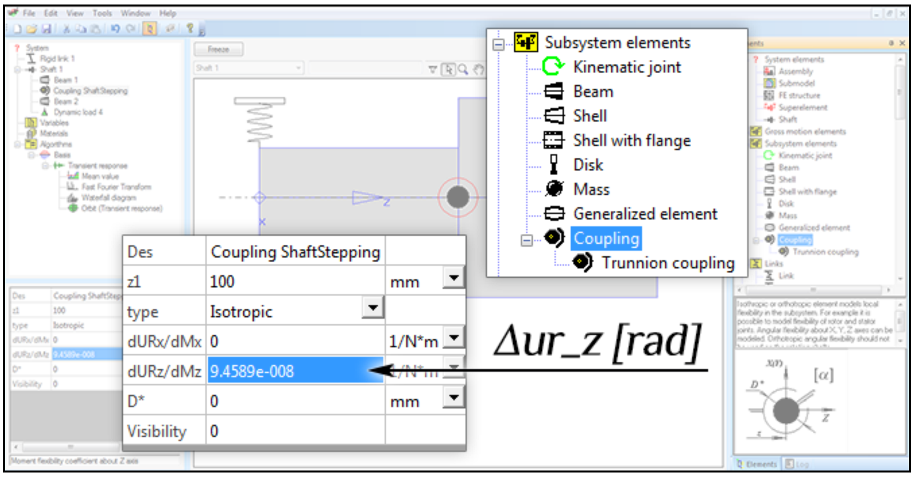


Fig. 6.6

6.2 Use of [Shaft Stepping] element

Starting from Dynamics R4.8.1 version, the **[Shaft Stepping]** element may be used to take into account the penetration effect (Fig. 6.7). Fig. 6.8 shows the element settings.

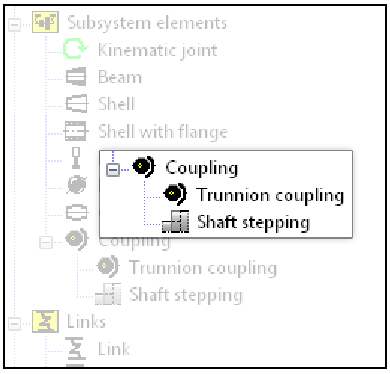


Fig. 6.7

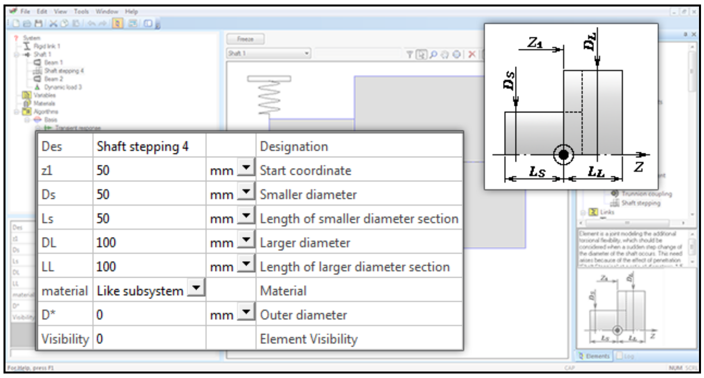


Fig. 6.8

6.3 Literature

1. API recommended practice 684, second edition, august 2005.

2. “Теория колебаний”, Бабаков И.М., изд. “Наука”, 1968.

3. ГОСТ 25.504-82 Расчеты и испытания на прочность. Методы расчета характеристик сопротивления усталости, 01.07.1983.

7. Modeling of disk skew

7.1 Modeling using [Unbalance Load]

When the disk axis does not coincide with the rotation axis of the rotor, additional unbalanced loads appear in the dynamic system. Noncoincidence of principal axes of inertia with the rotor coordinate system also influence change in natural frequencies. Because the angles are small, this influence may be neglected. If it is necessary to simulate the rotor disk skew in Dynamics R4, the **[Unbalance Load]** element may be used, where moment load created by centrifugal moment of skew disk is set (Fig. 7.1).

Des	Unbalance load +t/-t		Designation
z1	3403.3	mm	Start coordinate
F unb	0	gcm	unbalance force
F phase	0	deg	unbalance force phase
M unb	0.024	kg m m	unbalance moment
M phase	0	deg	unbalance moment phase
D*	0	mm	Outer diameter
Visibility	0		Element Visibility

Fig. 7.1

The following data are necessary for modeling:

- m (kg) – disk mass;
- I_d, I_p (kg*m^2) – diametral and polar inertia moments correspondingly;
- τ– inclination (skew) disk angle (Fig. 7.2).

Static unbalance (force unbalance) may be also taken into consideration. For this it is necessary to have disk eccentricity data - ε (mm).

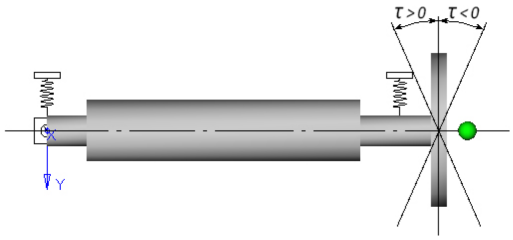


Fig. 7.2

Moment unbalance given in the element [Unbalance Load] settings is calculated using the following equation [7.1]:

$$M_{unb} = -(I_d - I_p) \cdot \sin(\tau) \cdot \cos(\tau) \tag{7.1}$$

Because the skew angle is small, product of trigonometric functions may be neglected and replaced by the angle in radian. Considering the assumptions, the following equation may be used:

$$M_{unb} = (I_p - I_d) \cdot \tau \tag{7.2}$$

Setting the moment unbalance, it is necessary to use the scheme presented in Fig. 7.3 In the scheme moment unbalance is presented as a pair of static unbalances modeling moment load. Angle φ presents phase (M_phase), given in the element settings [Unbalance Load].

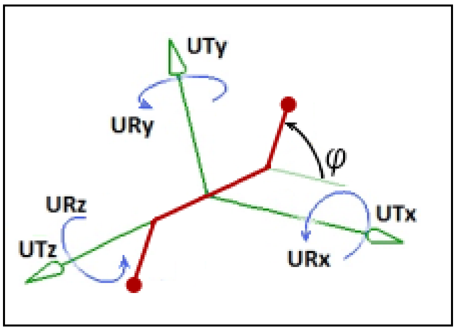


Fig. 7.3

7.2 Example

In [2], [3] the outboard rotor with disk skew is considered, and joint influence on unbalance behavior of the disk skew and its mass unbalance is taken into account. Fig. 7.4 shows the rotor model considered in [2], [3] articles, its geometric and mass-inertia parameters and the results of unbalance behavior calculation (rotating speed is in Hz in X-direction). Fig. 7.5 presents the model built in Dynamics R4 and the result of calculation of dynamic behavior using the [Unbalance Response] algorithm. Mass-inertia characteristics correspond to those presented in article [2]. The obtained unbalance Munb=0.024 kg*m2.

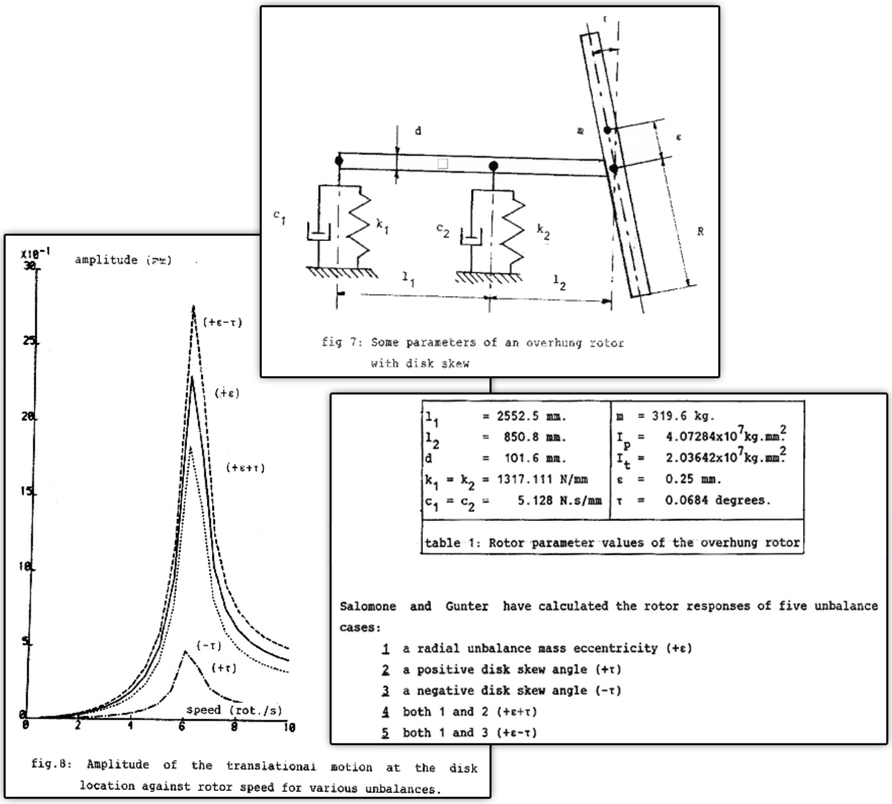


fig 7: Some parameters of an overhung rotor with disk skew

l_1 = 2552.5 mm.	m = 319.6 kg.
l_2 = 850.8 mm.	I_p = 4.07284x10^{7}kg.mm.^{2}
d = 101.6 mm.	I_t = 2.03642x10^{7}kg.mm.^{2}
$k_1 = k_2$ = 1317.111 N/mm	ε = 0.25 mm.
$c_1 = c_2$ = 5.128 N.s/mm	τ = 0.0684 degrees.

table 1: Rotor parameter values of the overhung rotor

Salomone and Gunter have calculated the rotor responses of five unbalance cases:

1 a radial unbalance mass eccentricity (+ε)
2 a positive disk skew angle (+τ)
3 a negative disk skew angle (−τ)
4 both 1 and 2 (+ε+τ)
5 both 1 and 3 (+ε−τ)

fig.8: Amplitude of the translational motion at the disk location against rotor speed for various unbalances.

Fig. 7.4

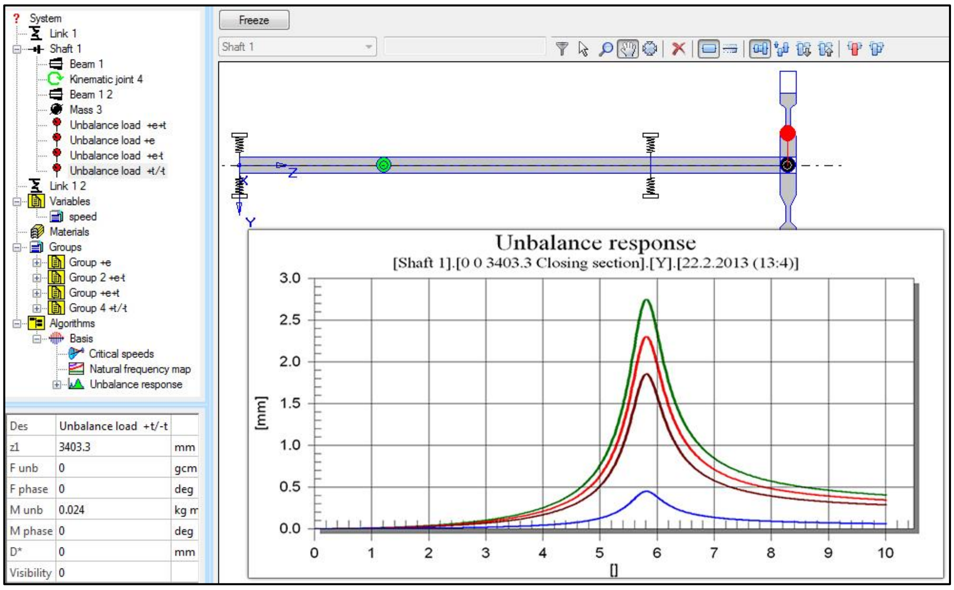


Fig. 7.5

7.1 Literature

1. "Handbook of Turbomachinery", Marcel Dekker, Inc., 1995.

2. "Synchronous Unbalance Response of an Overhung Rotor with Disk Skew" D. J. Salamone and E. J. Gunter, Transactions of the ASPIE, Journal of Engineering for Power, Vol.102, No.4, October 1980, pp.749-755.

3. "Unbalance behavior of a multi-disk rotor with mass eccentricity, residual shaftbow and disk skew" by Marc Crooijmans, Eindhoven, 21.11.1983

8. Modeling of bowed shaft [Shaft Bow]

When the bowed shaft rotates, the unbalanced forces, which are necessary to take into account solving the rotordynamics tasks, arise. Bow may take place as a result of breakdown in manufacturing process, temperature deformations of the shaft. To model, the **[Unbalance Load]** element, simulating unbalance load (radial and moment ones), may be used. Unbalanced forces have characteristic distributed along the shaft length.

8.1 Calculation of unbalance load

Unbalance, set into the element, is calculated using the equation 1:

$$F_{unb} = M \cdot e, \tag{8.1}$$

where: M – shaft mass, (kg);

e – eccentricity created by the shaft bow (cm).

Unbalances distribution is set by a user depending on requirements of the specific task. If it is necessary, the shaft is divided into several parts, and the **[Unbalance Load]** element is placed in calculated centre of mass of every part. Load, set in every element, is calculated separately using the equation (1), where mass of the chosen section is applied instead of the shaft mass and the eccentricity of the centre of mass of the chosen section is set.

8.2 Calculation of parameters of shaft section

In order to obtain mass and centre of mass of the shaft section, it is necessary to do the following:

1) to model the shaft section in Dynamics R4 or copy it from the given rotor model into the separate model
2) display the model log.

Fig. 8.1 gives the sought model parameters.

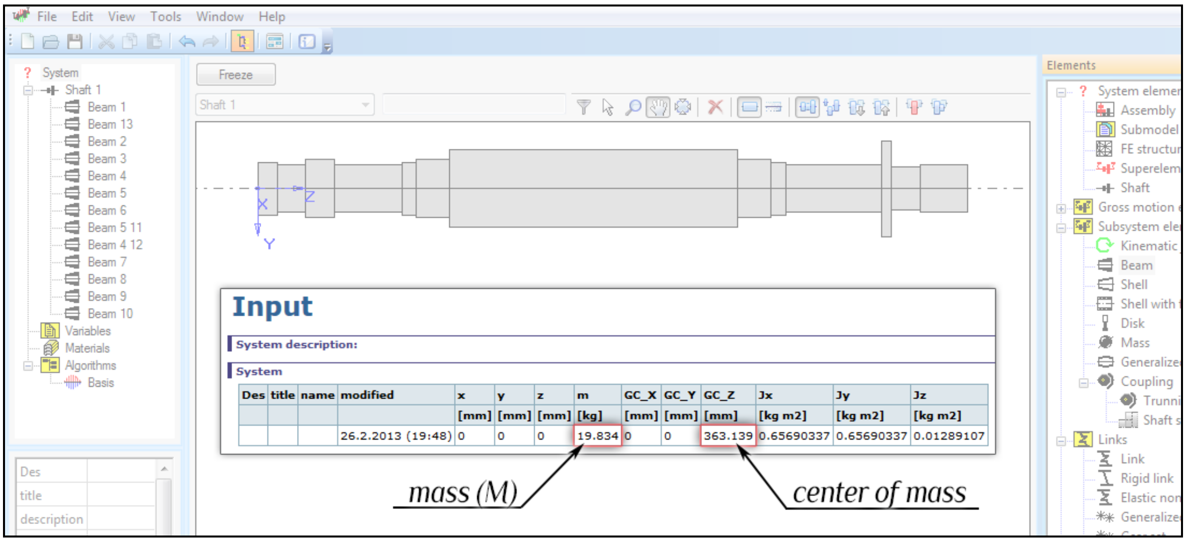


Fig. 8.1

8.3 Eccentricity calculation

In order to obtain eccentricity in any point, it is necessary to have function describing the shaft bending shape. Supposing that bending is described by quadratic function, and the function passes 3 points in the $A_i(z_i;y_i)$ format, where z_i – coordinate of section with experimentally obtained eccentricity, y_1 – eccentricity of the given section.

In this case quadratic function is the following:

$$y = az^2 + bz + c, \text{ where} \tag{8.2}$$

$$a = \frac{y_i - \frac{z_3\cdot(y_2-y_1)+z_2y_1-z_1y_2}{z_2-z_1}}{z_3(z_3-z_2-z_1)+z_1z_2} \tag{8.3}$$

$$b = \frac{y_2-y_1}{z_2-z_1} - a(z_1+z_2) \tag{8.4}$$

$$c = \frac{z_2y_1-z_1y_2}{z_2-z_1} + az_1z_2 \tag{8.5}$$

As a result of substitution of coordinates of the three points and obtainment of function describing the shaft bending, the eccentricity in the centre of mass of the investigated shaft section may be obtained. For this it is necessary to make z equal to the coordinate of the centre of mass (Fig. 8.1) and obtain y that is the sought eccentricity.

Generally it is necessary to obtain the equation for the curve approximating the experimental results, for example, using the least square method.

8.4 Disk inclination at rotor bow

When the disk is fit on the rotor shaft directly and “Shaft Bow” effect appears, disk will be inclined in correspondence with the shaft axis incline in the disk section, so the “Disk Skew” effect arises. It is described in the part 5 “Modeling of disk skew of rotor”. In this case the disk inclination angle may be calculated considering that derivative of the shaft bow function in the specific point is equal to tangent of inclination angle of the tangent line to the plot of the given function in the specific point. This gives us equation for the disk inclination angle:

$$\tau = arctg\left(y'(z_D)\right) \tag{8.6}$$

where:

- y’ – derivative of function describing the shaft bow;
- z_D – coordinate of centre of mass/ hub middle/ section of the disk setting along its axis.

For the quadratic function accepted in this method, there is the following derivative:

$$y' = 2az + b \tag{8.7}$$

8.5 Example

As an addition to the guide, the example of the abstract rotor with the shaft bow and the disk inclination appearing as a result is created. Fig. 8.2 gives the result of calculation of unbalance behavior (X-axis represents frequency in Hz).

Fig. 8.3 shows the model with added unbalance loads and the plot showing the accepted shaft bow. The model parameters are obtained on the basis of the model log and presented in appendix.

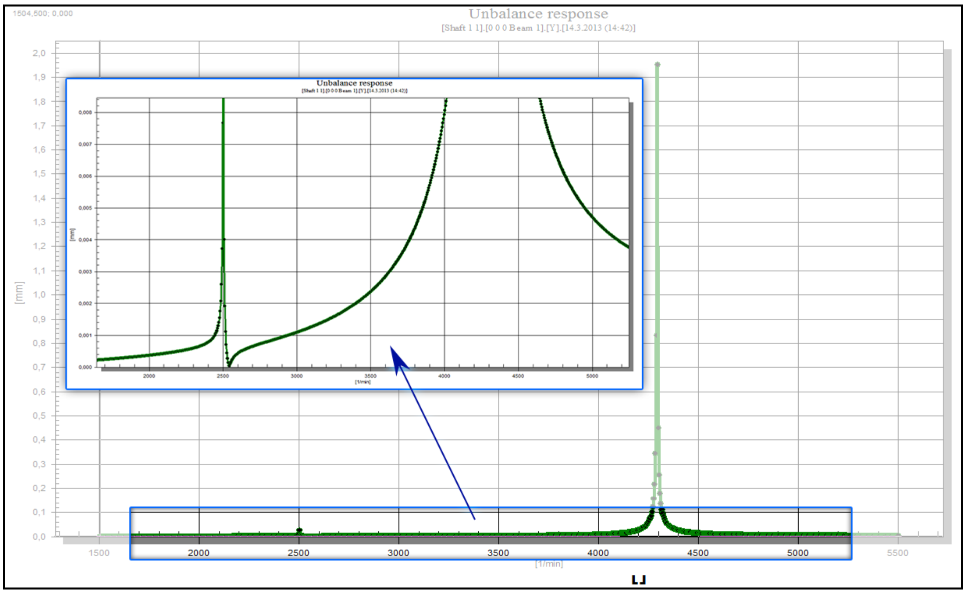


Fig. 8.2

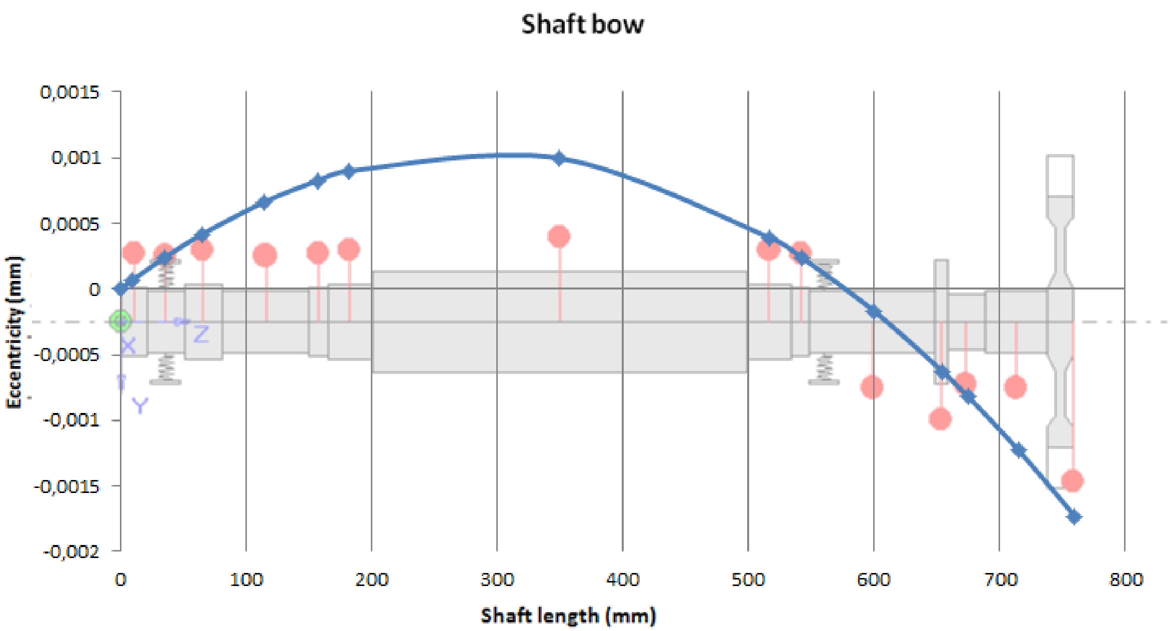


Fig. 8.3

8.6 Appendix

System description: Modeling of bowed shaft rotor
System

Des	title	name	m	GC_Z	Jx	Jy	Jz
			[kg]	[mm]	[kg m2]	[kg m2]	[kg m2]
Example	Shaft Bow	AlfaTranzit	22.334	406.443	1.08917505	1.08917505	0.21289107

Shaft 1
Beam

Des	ls	z1	z2	d1	D1	d2	D2	LG
	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]
Beam 1	20	0	20	0	55	0	55	10
Beam 13	30	20	50	0	50	0	50	15

Beam 2	30	50	80	0	60	0	60	15
Beam 3	70	80	150	0	50	0	50	35
Beam 4	15	150	165	0	55	0	55	7.5
Beam 5	35	165	200	0	60	0	60	17.5
Beam 6	300	200	500	0	80	0	80	150
Beam 5 11	35	500	535	0	60	0	60	17.5
Beam 4 12	15	535	550	0	55	0	55	7.5
Beam 7	100	550	650	0	50	0	50	50
Beam 8	10	650	660	0	100	0	100	5
Beam 9	30	660	690	0	45	0	45	15
Beam 10	50	690	740	0	50	0	50	25

Disk

Des	ls	z1	z2	d_hub*	D_rim*	m	type	Jx	Jy	Jz
	[mm]	[mm]	[mm]	[mm]	[mm]	[kg]		[kg m2]	[kg m2]	[kg m2]
Disk 14	20	740	760	0.1	200	2.5	Isotropic	0.1	0.1	0.2

Unbalance load

Des	z1	F unb	F phase	M unb	M phase
	[mm]	[gcm]	[deg]	[g cm cm]	[deg]
Unbalance load 15	10	0.003	0	0	0
Unbalance load 16	35	0.011	0	0	0
Unbalance load 17	65	0.028	0	0	0
Unbalance load 18	115	0.072	0	0	0
Unbalance load 19	157	0.023	0	0	0
Unbalance load 20	182	0.07	0	0	0
Unbalance load 21	350	1.2	0	0	0
Unbalance load 22	517	0.03	0	0	0
Unbalance load 23	542	0.007	0	0	0
Unbalance load 24	600	0.026	180	0	0
Unbalance load 25	655	0.039	180	0	0
Unbalance load 26	675	0.031	180	0	0
Unbalance load 27	715	0.095	180	0	0
Unbalance load 28	760	0	0	11.527	180

Link

Des	conn_type	side1_l	side1_subs	Type
		[mm]		
Link 1	via body	35	Shaft 1	Isotropic
Link 2	via body	561.564	Shaft 1	Isotropic

Link 1
Stiffness matrix

Stiffness matrix						
	Fx [N]	Fy [N]	Fz [N]	Mx [N m]	My [N m]	Mz [N m]
ut_x [m]	1e+008	0	0	0	0	0
ut_y [m]		1e+008	0	0	0	0
ut_z [m]			1e+008	0	0	0
ur_x [rad]				0	0	0
ur_y [rad]					0	0
ur_z [rad]						0

Link 2
Stiffness matrix

Stiffness matrix						
	Fx [N]	Fy [N]	Fz [N]	Mx [N m]	My [N m]	Mz [N m]
ut_x [m]	1e+008	0	0	0	0	0
ut_y [m]		1e+008	0	0	0	0
ut_z [m]			1e+008	0	0	0
ur_x [rad]				0	0	0
ur_y [rad]					0	0
ur_z [rad]						0

9. Critical speed map of two support rotor

9.1 Calculation of critical speed map

In Dynamics R4 the map of critical speeds depending on different nonlinear parameters, for example, the supports stiffness, may be obtained. For this the **[Parameter Analysis]** algorithm must be used.

After adding the algorithm into the model, it must be adjusted (Fig. 9.1). Here (example №21) the rotor on two supports – “Front support” и “Rear support” – is considered. The supports stiffness is assigned in [Variables]. Link with these variables is defined in the “Parameter analysis” window; it is output by double click on the […] field of the parameter [inp_parameters]. To obtain the 2D-map, all variables must be added into the field “Primary variables”. “Used guide” gives the more detailed description of the algorithm in the section 15.7 “ [Parametric analysis]” or Help.

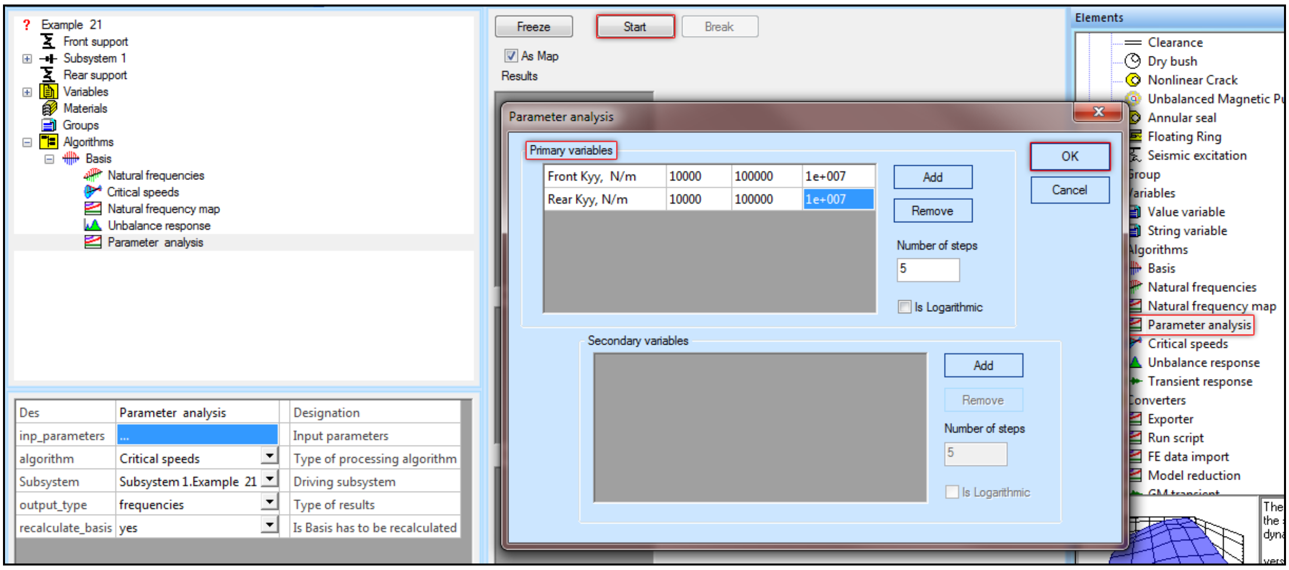


Fig. 9.1

To start calculation, the window of the variable choice must be closed clicking “OK” and “Start” in the upper screen part. Fig. 9.2 gives the result of critical speed map calculation.

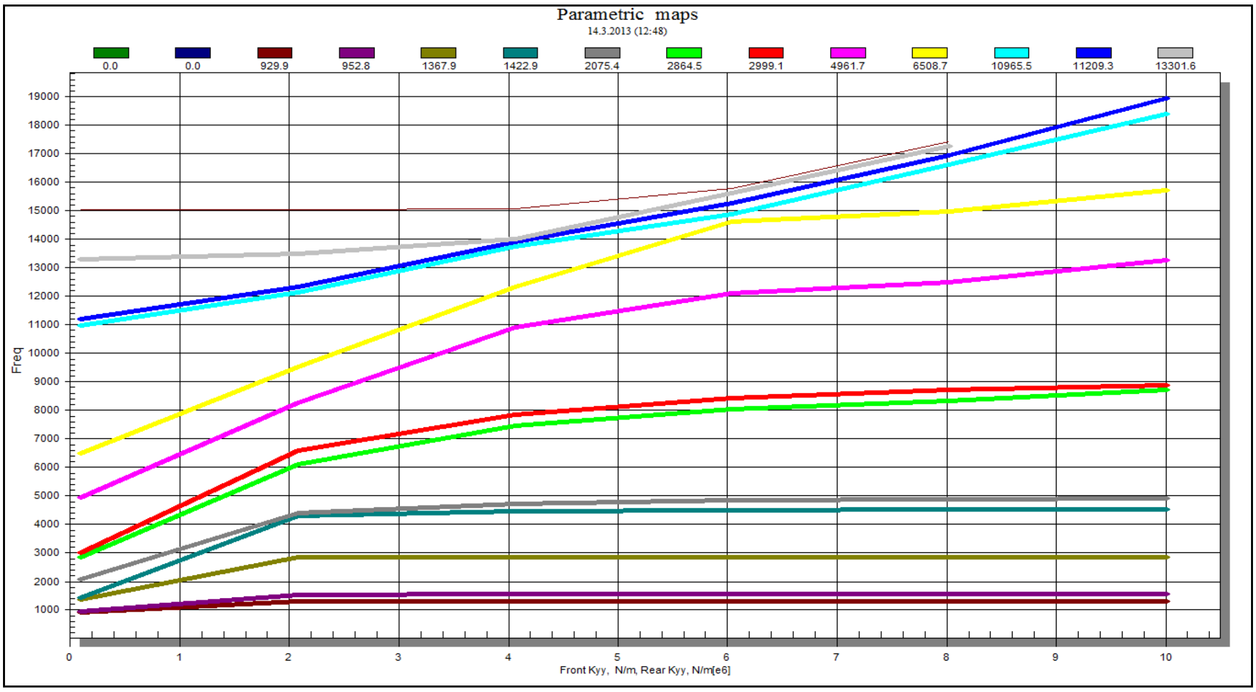


Fig. 9.2

9.2 Export of critical speed map

To export the map, the "Export…" dialog may be used. It is available clicking the right mouse button on the map (Fig. 9.3). Export is possible in three formats: MetaFile, BMP (image), Text/Data only. There are additional settings for every variant. For the following work with critical speed map in the word processor like Microsoft Excel it is advisable to choose text export format in the clipboard.

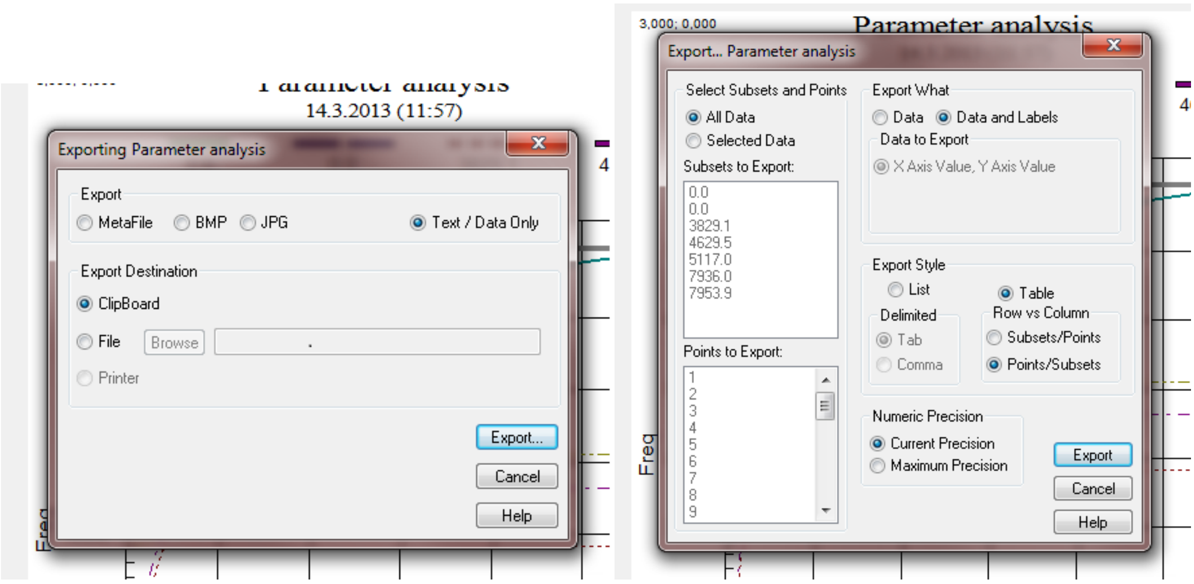


Fig. 9.3

After export in the clipboard certain results must be formatted. For this:

- Paste the clipboard content into the "Excel" table;
- Copy data from the obtained table without upper line (frequencies designations) and left column (regimes designations) and paste into the "Note";

- Replace (the dialog is called by pressing Ctrl+H, use the button “Replace all”) the elements “, “ (comma with space character) by Tab (beforehand copy the output result into clipboard when pressing the Tab button);
- Copy the obtained result from the note into the Excel table. It will be pasted in the way that is appropriate to continue work: with X- and Y-axes values separated in columns.

Using the data obtained in Excel, the critical speed map may be built (the most appropriate variant is “point diagram with smooth curves and markers”). Fig. 9.4 gives the result of the map creation.

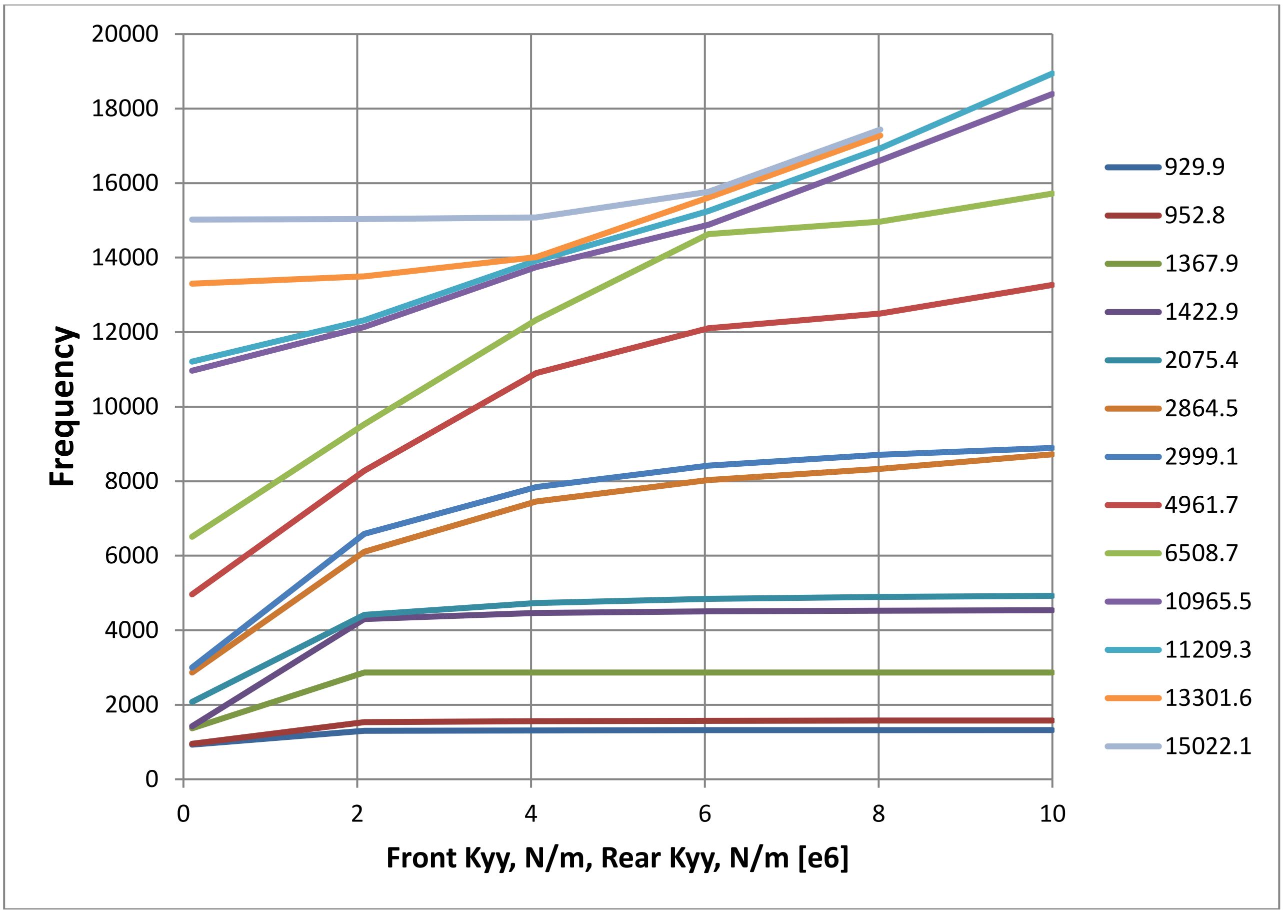


Fig. 9.4

9.3 Optimization of calculation of critical speed map

The obtainment of critical speed map by varying stiffness in basis link was presented above. It leads to necessity to recalculate basis for every point on the map. For complicated models time of basis calculation may be over a minute. In such cases in order to exclude basis recalculation it is advisable to use the link that is taken into consideration only in the modal algorithms– [Elastic nonsymmetric link].

9.4 Superposition of graphs of supports stiffness

When it is necessary to meet the requirements of the API 684 about superposition of graphs of supports stiffness values on the critical speed map (Fig. 9.5), the map and change in supports stiffness depending on the regimes (it is especially actual for journal bearings) must be exported in text format into external word processor (for example, Microsoft Excel), and graphs must be built in it. In order to obtain data on the bearing stiffness change (if they are set into the model), the dialog

[Input bearing from table] may be used. It is available in the "Tools" menu.(Fig. 9.6) User Guide gives more details about the dialog in 14.5 "Import dialog of parameterized data of stiffness and damping in journal bearings". Data may be obtained by direct copying of the data table and used to build the critical speed map Fig. 9.7 presents the result of superposition of the supports stiffness on the critical speed map. The example №45 is used here.

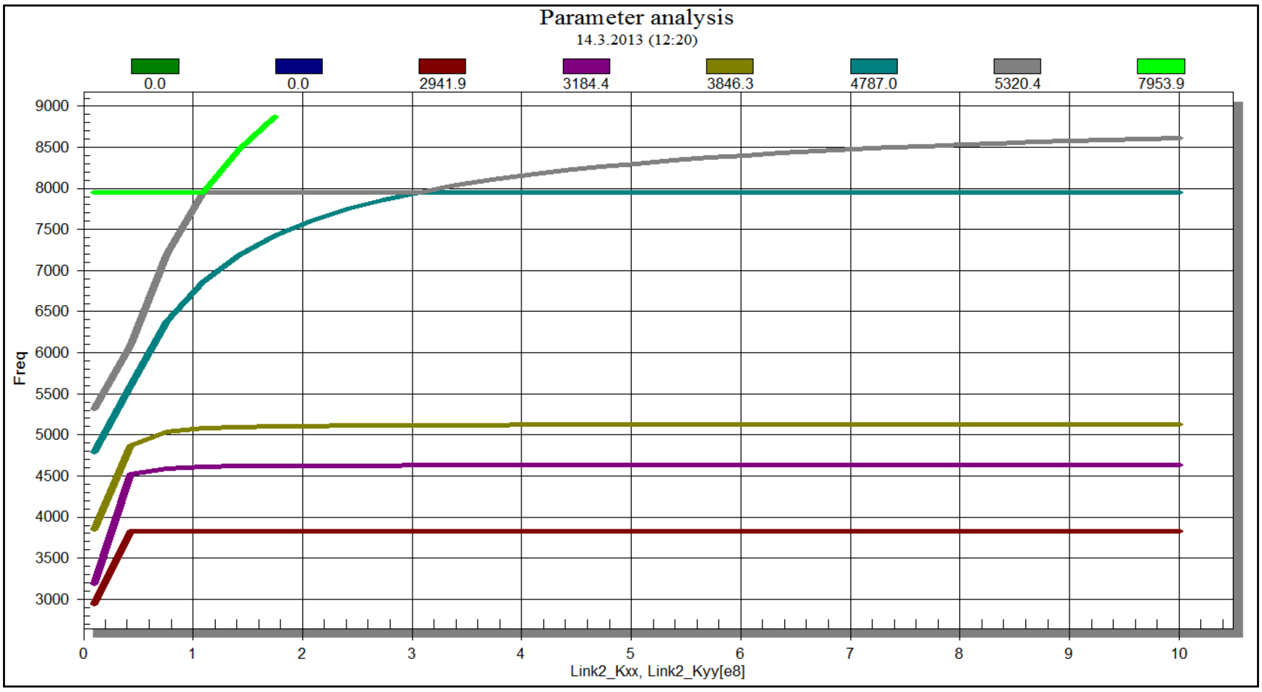
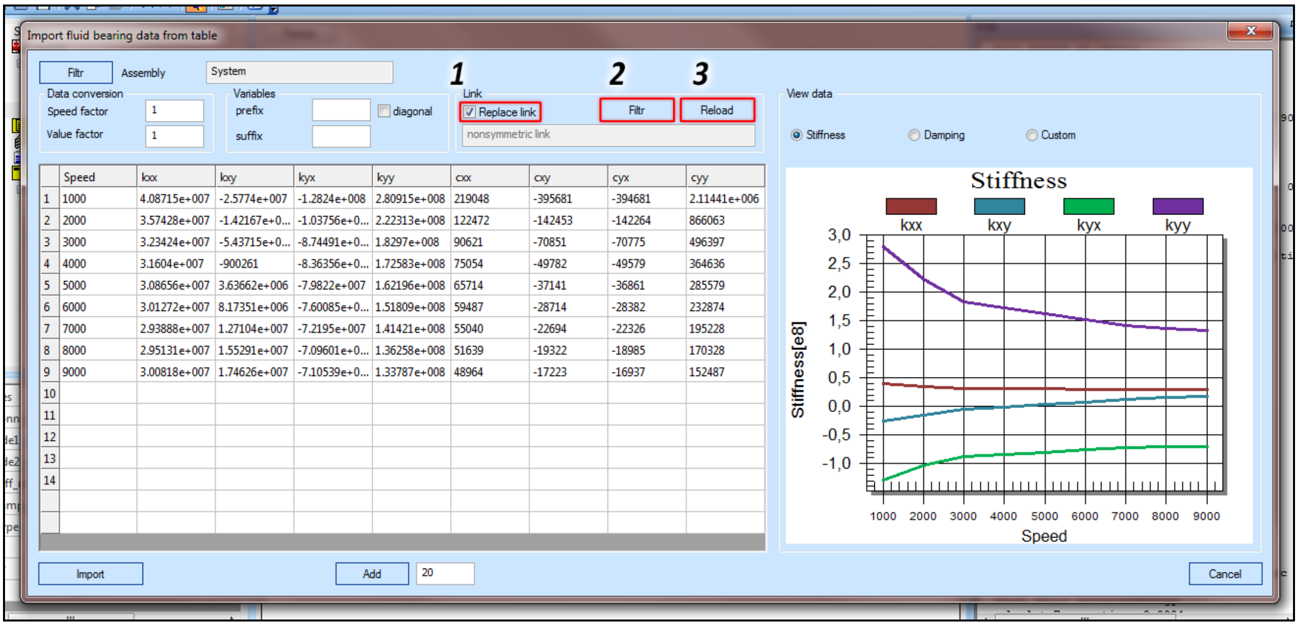


Fig. 9.5



	Speed	kxx	kxy	kyx	kyy	cxx	cxy	cyx	cyy
1	1000	4.08715e+007	-2.5774e+007	-1.2824e+008	2.80915e+008	219048	-395681	-394681	2.11441e+006
2	2000	3.57428e+007	-1.42167e+0...	-1.03756e+0...	2.22313e+008	122472	-142453	-142264	866063
3	3000	3.23424e+007	-5.43715e+0...	-8.74491e+0...	1.8297e+008	90621	-70851	-70775	496397
4	4000	3.1604e+007	-900261	-8.36356e+0...	1.72583e+008	75054	-49782	-49579	364636
5	5000	3.08656e+007	3.63662e+006	-7.9822e+007	1.62196e+008	65714	-37141	-36861	285579
6	6000	3.01272e+007	8.17351e+006	-7.60085e+0...	1.51809e+008	59487	-28714	-28382	232874
7	7000	2.93888e+007	1.27104e+007	-7.2195e+007	1.41421e+008	55040	-22694	-22326	195228
8	8000	2.95131e+007	1.55291e+007	-7.09601e+0...	1.36258e+008	51639	-19322	-18985	170328
9	9000	3.00818e+007	1.74626e+007	-7.10539e+0...	1.33787e+008	48964	-17223	-16937	152487
10									
11									
12									
13									
14									

Fig. 9.6

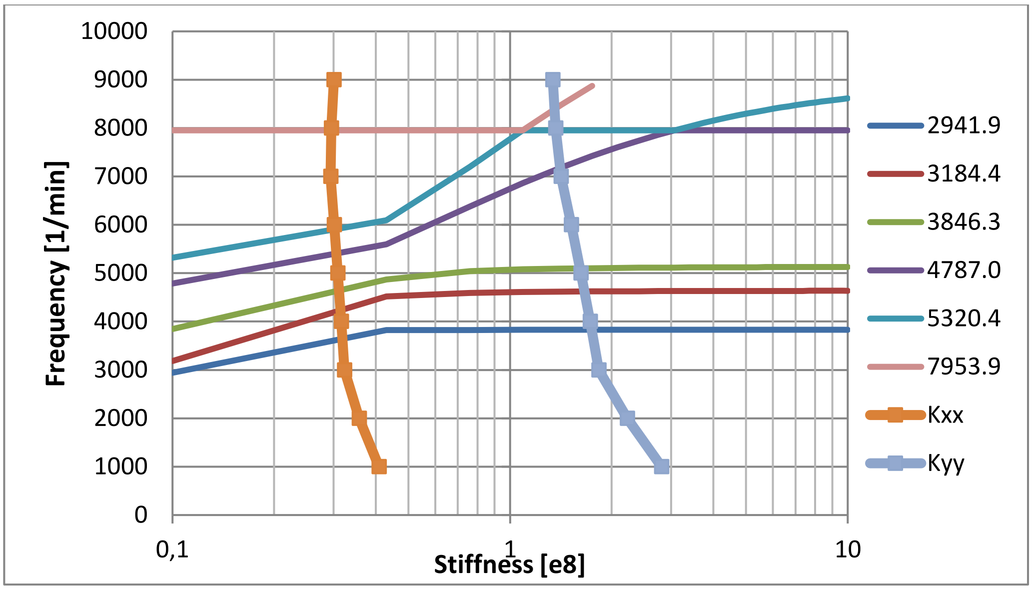

10000
9000
8000
7000
6000
5000
4000
3000
2000
1000
0
Frequency [1/min]
0,1
1
10
Stiffness [e8]
2941.9
3184.4
3846.3
4787.0
5320.4
7953.9
Kxx
Kyy

Fig. 9.7

10.Modeling of particular cases of rotors disks

10.1 Modeling of flexible disk

When the disks of the rotor wheels are modeled, using the built-in element “disk”, the disk is modeled as an absolutely rigid element with two calculated sections not depending on its length along rotation axis. When it is necessary to simulate the disk as an elastic element (when modeling massive disks of big diameter, especially being structural elements), the method of the disk modeling by two point masses may be used.

To simulate the disk using this method, you should know its mass-inertia and stiffness characteristics. It is also necessary to know mass-inertia characteristics for two disk halves: inner and outer. Division is done in terms of the assignment conditions at cylindrical section. For example, one part may include the hub and half of the disk cloth, the other one – the rim with the blades fixed on it and the other half of the cloth.

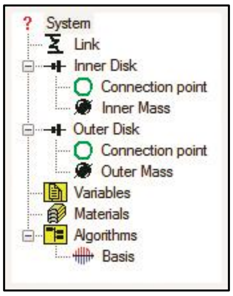


Fig. 10.1

Fig. 10.1 gives the structure of the flexible disk model. It represents two subsystems (one point mass in each one – “Inner Mass” and “Outer Mass”) and the elastic link between them. One subsystem must be considered as an inner half of the disk, the other one – as an outer. In accordance with the accepted division, mass-inertia characteristics are defined. A link between subsystems is simulated by the **[Link]** element where stiffness (and damping if it is necessary) characteristics of the disk cloth are assigned, as it is shown in Fig. 10.2.

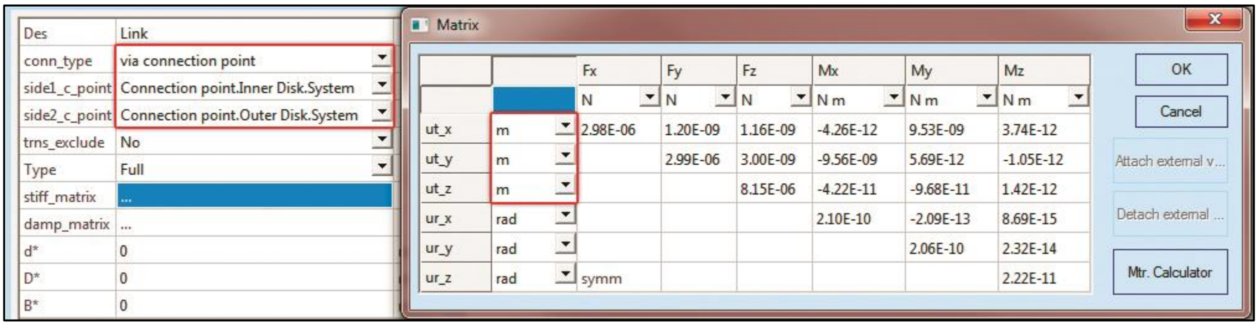


Fig. 10.2

In Dynamics R4 it is possible to image the point masses as disks. Here it is reasonable to set up visualization of one mass as a disk, and make the other one invisible (transparent). In order to change transparency of the element, you may vary either the “Visibility” parameters in the element setup or choose one of three variants (“Visible”, “Transparent”, “Invisible”) in the element characteristics in the model tree, clicking on it by right mouse button (Fig. 10.3). Fig. 10.4 gives an example of the settings for “Inner Mass”.

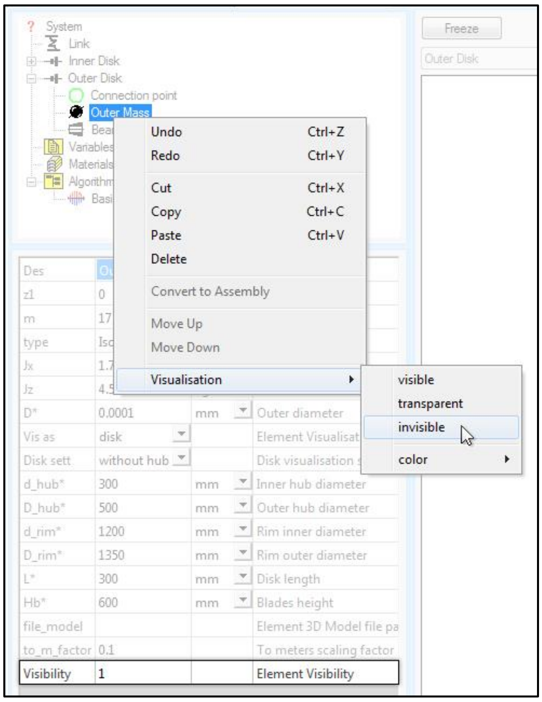


Fig. 10.3

Des	Inner Mass		Designation
z1	0	mm	Start coordinate
m	20	kg	Mass
type	Isotropic		Type
Jx	0.8	kg m2	Jx
Jz	1.1	kg m2	Jz
D*	0.0001	mm	Outer diameter
Vis as	disk		Element Visualisation as...
Disk sett	whole		Disk visualisation settings
d_hub*	300	mm	Inner hub diameter
D_hub*	500	mm	Outer hub diameter
d_rim*	1200	mm	Rim inner diameter
D_rim*	1350	mm	Rim outer diameter
L*	300	mm	Disk length
Hb*	600	mm	Blades height
file_model			Element 3D Model file path
to_m_factor	0.1		To meters scaling factor
Visibility	0		Element Visibility

Fig. 10.4

10.2 Modeling of offset disk

When it is necessary to take into consideration stiffness characteristics of the installation and/or fixing disk on the shaft (for example, when mounting a disk on the shaft using conical pin), modeling may be done using the method of the offset disk.

In this method mass-inertia characteristics of the disk and stiffness (when it is necessary – and damping) characteristics of fixing/ disk support on the shaft are necessary. Disk is simulated in the subsystem separated from the shaft by the **[Disk]** or **[Mass]** element and is attached to it using the **[Link]** element between subsystems. Fig. 10.5 gives the model example on the method of the offset disk.

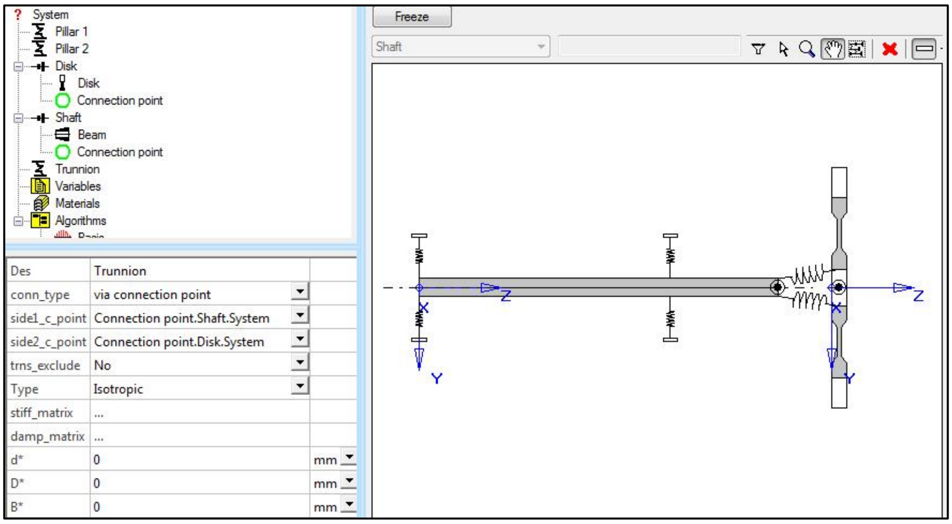


Fig. 10.5

Link stiffness is obtained using the method of the corresponding section for case elements (section 4

11.Modeling of the “squirrel cage” spring

When modeling elastic-damper support of the rotor, it is important to simulate properly an elastic element. This example presents the methodology of simulation of the “squirrel cage” spring (Fig. 11.1). Similar approach may be used when modeling any design elements of the rotor’s supports if the geometry is complicated and there are specific design decisions which can’t be simulated by solid of revolution.

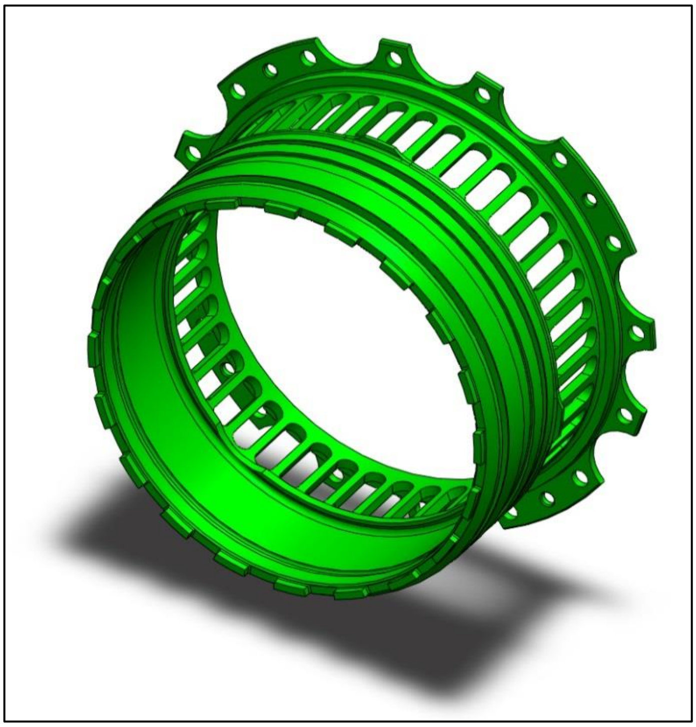


Fig. 11.1

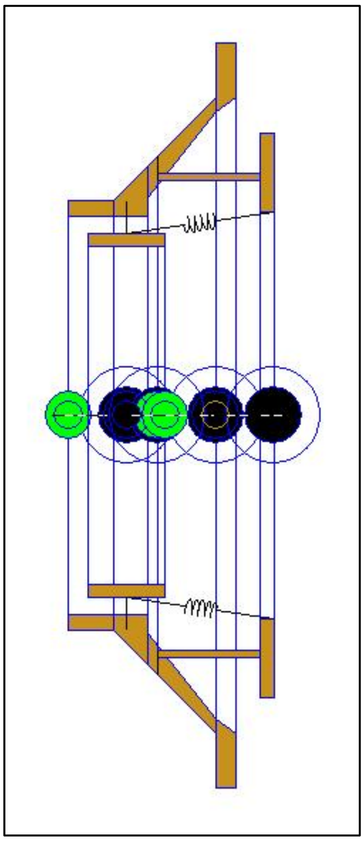


Fig. 11.2

One of the simulation options is to create the elastic link between the model’s sections connected by the elastic part of the support with the stiffness matrix corresponding to the stiffness characteristics of the Squirrel cage. These characteristics may be obtained using FEM-calculation. Fig. 11.2 gives the example of simulation of the elastic-damper support of the rotor; stiffness characteristics of the spring are taken into account using the element [Link].

When modeling in the FEM-system, the "squirrel cage" model must be fixed rigidly to the flange which connects it with the support and should be loaded by unit forces and moments alternately along 6 degrees of freedom corresponding to the actual spring’s operation. In the considered example it is necessary to load the point on the “squirrel cage” axis that is rigidly connected with the surface which the outer bearing ring is mounted on. Fig. 11.3 presents the scheme of fixing and loading of this “squirrel cage” of this example. The restraint (a rigidly fixed surface with no possibility to move and rotate at all freedom degrees) is denoted by hatching; the loading surface that is rigidly connected with the loading point placed on the axis -- by the thick line; alternate application of unit loads at 6 freedom degrees – by the vector F.

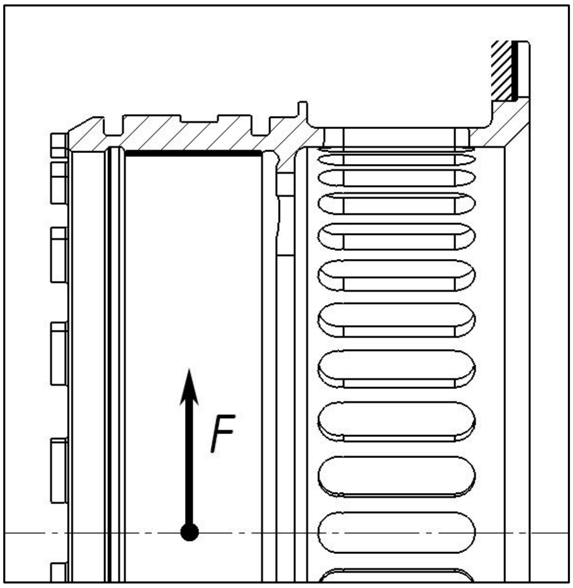


Fig. 11.3

Flexibility matrix (Table 11.1) should be the result of FEM-calculation. It represents the matrix 6x6, where the rows correspond to deformations along 6 freedom degrees, columns – applied loads.

Table 11.1

Flexibility						
	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	2.3239E-05	-2.4061E-08	1.3254E-10	-2.0582E-11	-6.1731E-09	3.1495E-10
Fy [N]	-2.4061E-08	2.3268E-05	4.8544E-10	6.1983E-09	2.0744E-11	-3.5339E-10
Fz [N]	1.3254E-10	4.8545E-10	8.2392E-07	6.0184E-12	-9.4609E-12	-5.0732E-13
Mx [N mm]	-2.0582E-11	6.1983E-09	6.0184E-12	2.7413E-10	4.6970E-14	-3.0982E-13
My [N mm]	-6.1731E-09	2.0744E-11	-9.4609E-12	4.6970E-14	2.7410E-10	-2.7865E-13
Mz [N mm]	3.1495E-10	-3.5339E-10	-5.0733E-13	-3.0982E-13	-2.7865E-13	2.9879E-09

To obtain stiffness matrix, you must invert flexibility matrix. For example, the matrix can be inverted easier in the DYNAMICSR4 matrix calculator, using the function «C=Invert (LA)» (Fig. 11.5). Flexibility matrix obtained before in the FEM-system should be pasted as the matrix “A”. After that you should click the button corresponding to the inversion operation («C=Invert (LA)»). The inverted matrix (stiffness matrix in this case) appears as the matrix “C” in the calculator.

Note: Matrix calculator can be launched from the menu “Tools” – “Matrix Calculator” (Fig. 11.4)

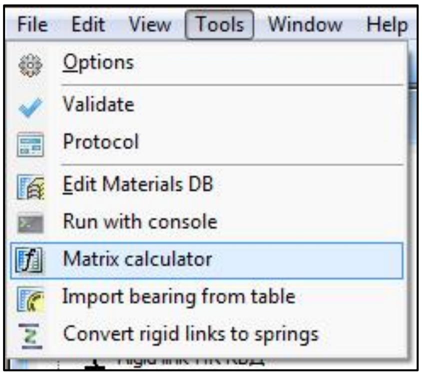


Fig. 11.4 Launch of matrix calculator

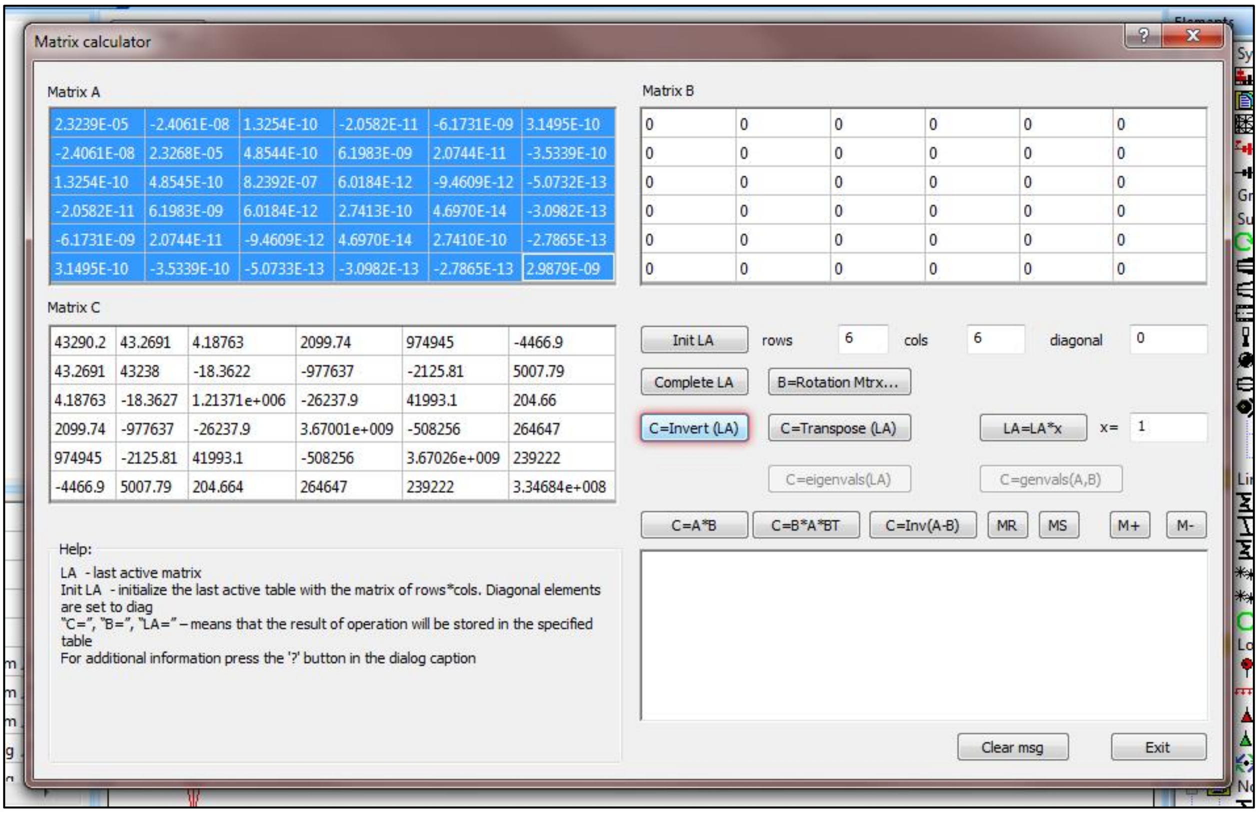


Fig. 11.5 Operation in matrix calculator

You must insert the obtained stiffness matrix as the stiffness matrix of the link simulating "the squirrel cage" (Fig. 11.6). At this stage it is imperative to take into account measurement units in the FEM-system. So, if millimeters are used then it is necessary to set preliminary millimeters as the units to measure both deformations and unit moments. That is, when obtaining flexibility matrix, deformations of 4, 5 6 freedom degrees were obtained at loading by the moment of 1 [H*мм], so it is necessary to get displacements in [mm], and to use stiffness in [mm] after that. If loading accounts for the moment of 1 [H*м], you must use meters.

Important: DYNAMICSR4 takes into consideration the measurement units. So, if you change measurement units after defining the numerical values (in this case after pasting stiffness matrix), the displayed numerical values will change in correspondence with dimension change.

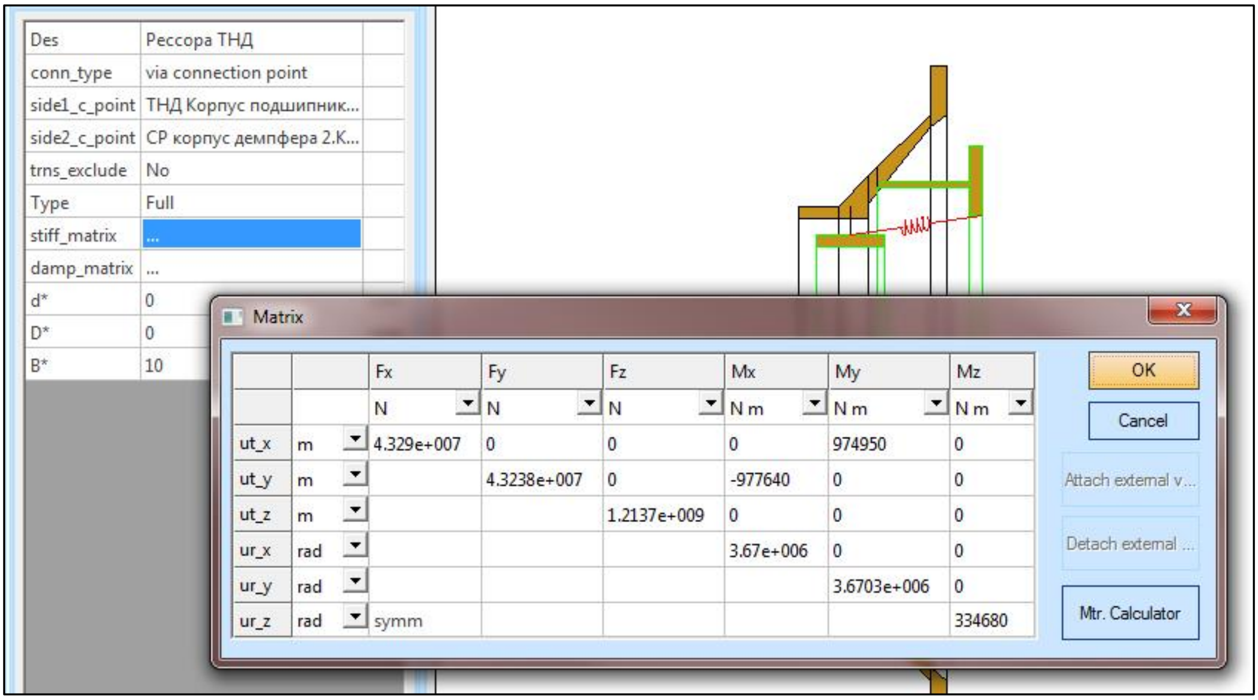


Fig. 11.6

There is an alternative variant how to take into account stiffness characteristics of the elastic element of the support - to use radial link considering only radial stiffness (axial and moment stiffness are not taken into account). In this case it is possible not to use FEM-simulation, but use analytical calculation of radial stiffness of the elastic element of the support in the DamperR3.1 program (Fig. 11.7). The calculation methodology is given in the part "3.3.2 Flexible element analysis" of the "DAMPER SUPPORTS" document.

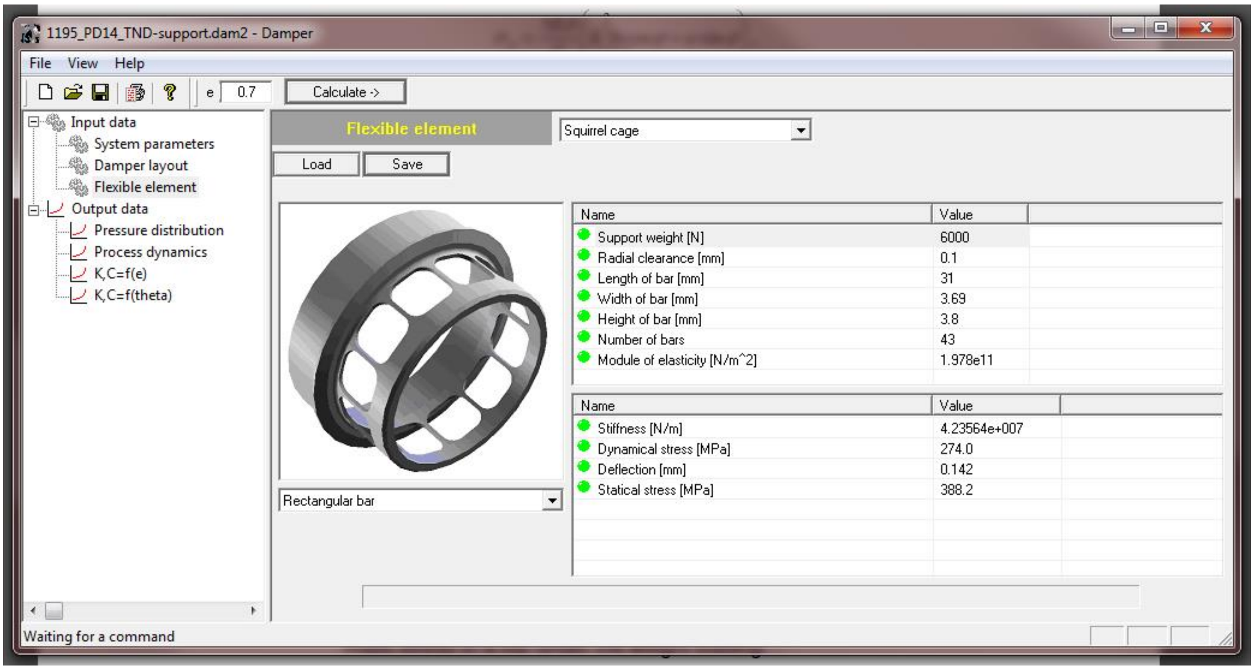


Fig. 11.7 Calculation of elastic element of support in DamperR3.1

It is also possible to calculate stiffness coefficient directly using the analytical formula, which is used in DamperR3.1. Fig. 11.8 gives the extract from the part "3.3.2 Flexible element analysis" of the document "DAMPER SUPPORTS" with description of the formula.

An FE stiffness can be calculated by

$$K = \frac{nEab(a^2 + kb^2)}{2l^3},$$

where

n - number of bars;

a, b, l - width, thickness and length of a bar accordingly;

E - Young module of the bar material at operating temperature.

$k = \dfrac{1}{\left(1 + \dfrac{2\sqrt{ab}}{l}\right)^3}$ - correction coefficient, depending on the flexible web dimensions

Maximal alterating stress in the bar

$$\sigma_d = \frac{3E\delta}{l^2}\left(k^{\frac{2}{3}} b\cos\varphi + a\sin\varphi\right),$$

where

$$\varphi = \arctan\frac{a}{bk^{\frac{2}{3}}} + n\pi; \quad n = 0 \text{ or } 1$$

δ - radial clearance

Static displacement under the support weight G loading

$$\delta_0 = G/K \quad .$$

Static stress in a bar under the weight loading

$$\sigma_{\delta_0} = \sigma_d \frac{\delta_0}{\delta} \quad .$$

Fig. 11.8 Analytical calculation of characteristics of the elastic element of the “squirrel cage” support

Adding of calculated radial stiffness into DYNAMICS R4model

Fig. 11.9 gives examples of the stiffness matrixes of the link simulating the elastic element of the support with the defined radial stiffness that is calculated in DamperR3.1 or analytically. The upper table shows stiffness matrix of the “Isotropic” type, lower – the “Full” type.

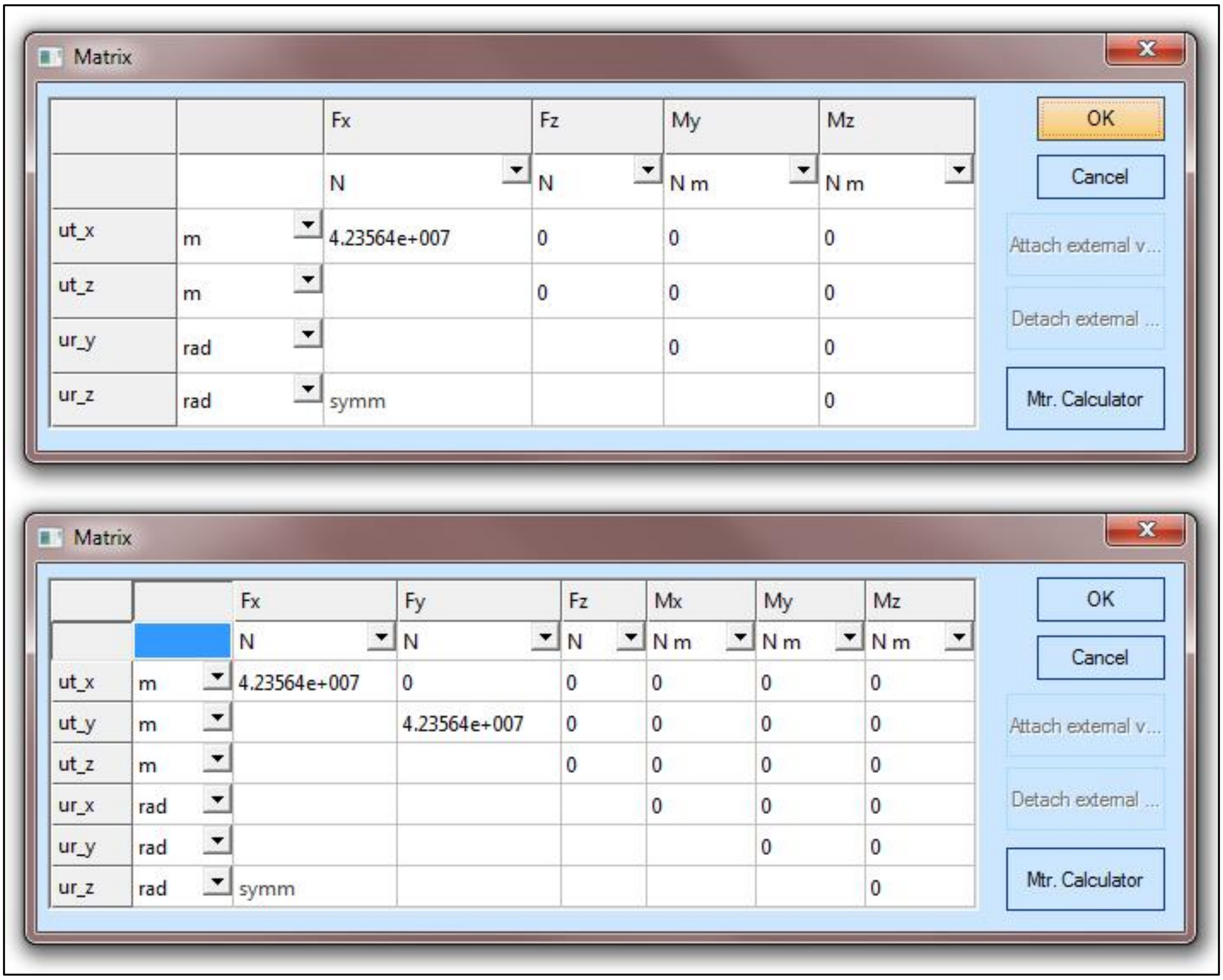


Fig. 11.9 Use of stiffness coefficient calculated in DamperR3.1 or analytically in stiffness matrix of link simulating elastic element of support

Simulation of mass-inertia characteristics of elastic element of support in DYNAMICS R4 models

In order to consider mass-inertia characteristics of the “squirrel cage”, it is necessary to obtain mass and inertia moments in the CAD/CAE – system relatively to the coordinate system placed in the loading point of the “squirrel cage” with the axes location consistent with DYNAMICSR4.

The obtained mass and inertia moments must be assigned in the element [mass] in the DYNAMICSR4 model. The element is inserted in the subsystem of the outer bearing ring with assignment of the centre mass coordinate of the elastic element as the coordinate of Z-element (Fig. 11.10). It is important to define mass in this subsystem in order to have natural frequencies of the outer bearing ring being in the calculation range of the basis. It is necessary for obtainment of correct values of the reactions forces in the supports. If there are not these oscillations in the basis, value of reactions in the support may be incorrect.

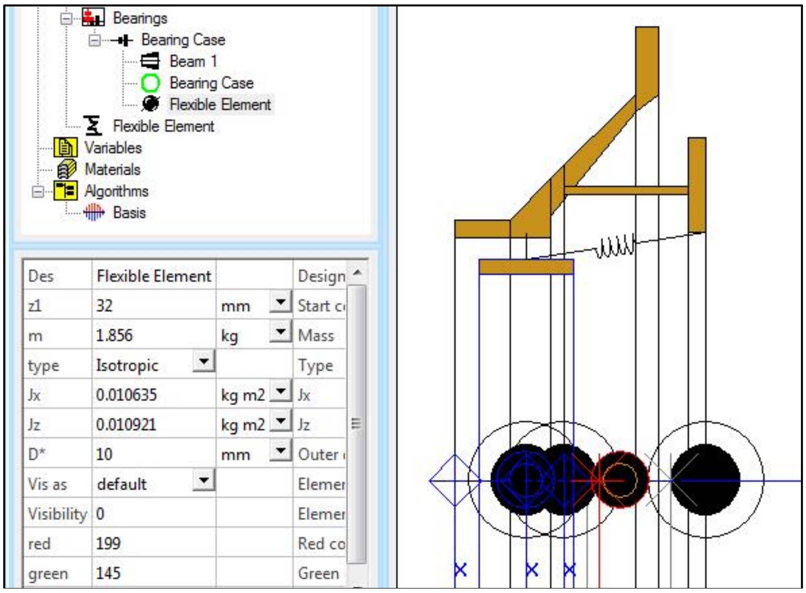


Fig. 11.10 Taking into consideration of mass-inertia characteristics of elastic element

12.Simulation of bevel gear set

Gear sets are simulated using a linear link [Gear Set]. The following data needed to simulate a bevel gear set in Dynamics R4:

- Parameters of gear sets (the teeth number, module, contact angle, helix angle (for bevel gear sets), helix angles of pitch cones)
- Geometrical parameters of the gear set that are necessary to reconcile subsystems.

The feature of simulation of bevel gear sets in Dynamics R4 is necessity to reconcile subsystems where shafts with gear sets are simulated. The position of the gear set in the subsystem is defined by the coordinate of the connection point [Connection Point], corresponding to the given gear set (it is defined in the settings of the [Gear Set] element). The connection point should be at the distance equal to the half of the reference diameter of another gear set (if rotation axes cross at right angle). If rotation axes of the gear set are not perpendicular, it is necessary to use trigonometric functions to find the coordinates of the connection point. It must be calculated considering the subsystem offset if it takes place (defined in the subsystem settings). Fig. 12.1 presents the scheme.

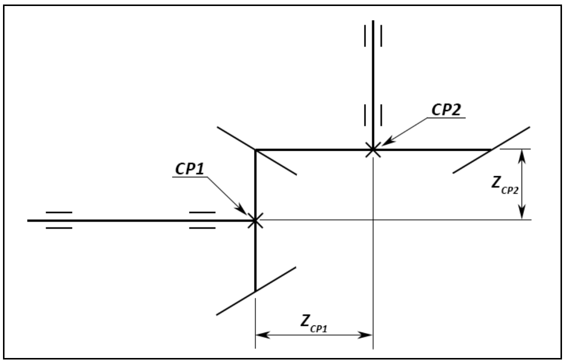


Fig. 12.1

<u>Note:</u> it is necessary to take into account that spur and helical gears are calculated using different equations, particularly their reference diameters. If the other parameters are equal, a helical gear has bigger reference diameter than a spur one.

<table>
<tr><td>Calculation</td><td>Gear type</td><td>spur</td><td>helical</td></tr>
<tr><td>Reference diameter</td><td rowspan="2">spur</td><td>$d = z * m$</td><td>$d_w = z * \frac{m_n}{\cos\beta}$</td></tr>
<tr><td>Module</td><td>m, is chosen from standard ones</td><td>m_n – normal module (see State Standard 9563-60), is chosen from standard ones</td></tr>
<tr><td rowspan="2">Average module</td><td rowspan="3">bevel</td><td>$m_m = m_e \frac{R_m}{R_e}$,
m_e – heel end module, equal to m</td><td>$m_m = m_n \frac{R_m}{R_e}$</td></tr>
<tr><td colspan="2">R_m – mean cone distance,
R_e – outer cone distance</td></tr>
<tr><td>Average reference diameter</td><td colspan="2">$d_m = z \cdot m_m$</td></tr>
</table>

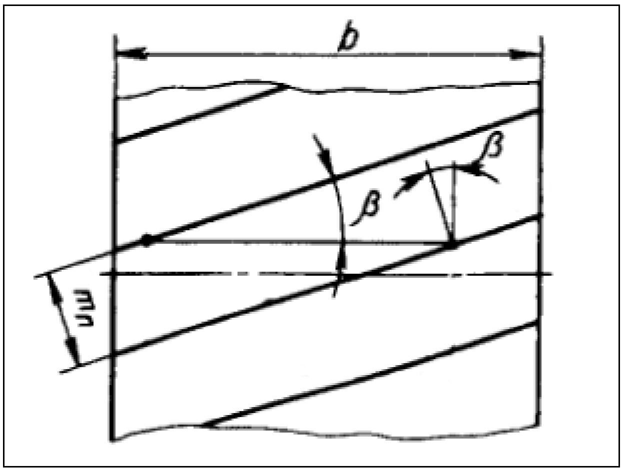


Fig. 12.2

12.1 Stiffness characteristics of gears

As opposed to spur gears, stiffness characteristics of gear journals are very important. It is particularly urgent if a journal is of an evident conical type (Fig. 12.3).

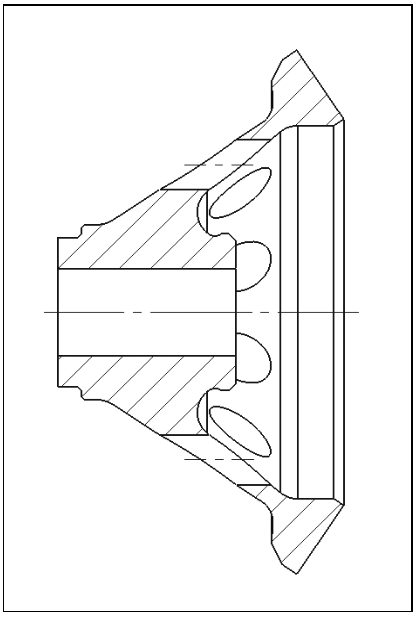


Fig. 12.3

In order to take into consideration stiffness and/or flexibility of the journal, you must use one of two variants:

- simulation using [Generalized Element];
- simulation using elastic-damping link [Link].

12.1.1 Simulation using [Generalized Element].

Let us consider simulation of bevel gear when one of gears is modeled by the [Generalized Element] element. Fig. 12.4 shows the shaft subsystem built using this method.

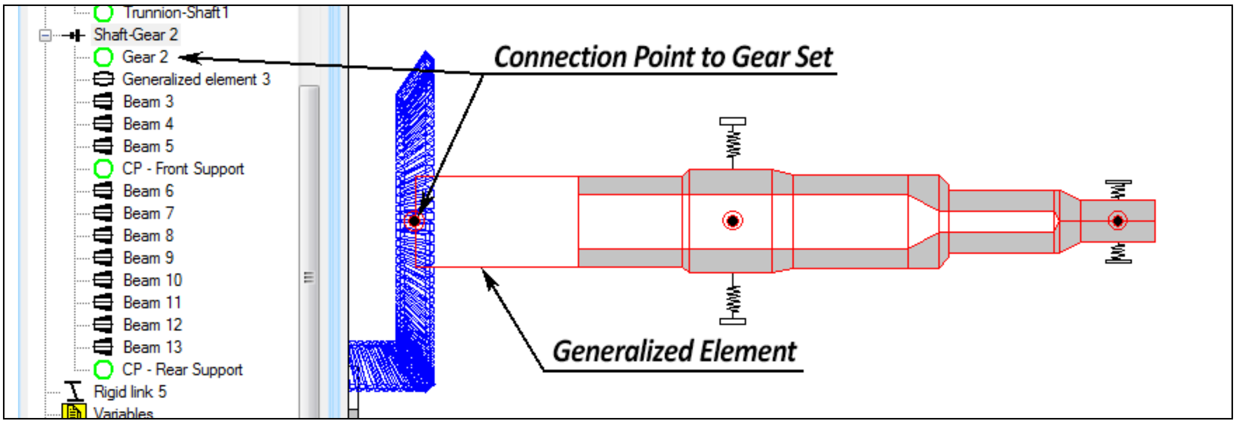


Fig. 12.4

In this case stiffness characteristics are defined in the [Generalized Element] settings as the flexibility matrix obtained in FEM by the method of consecutive measurement of displacements and rotation angles of the loading point relatively to the fixing point as a result of loading by unit forces and moments. Loads should be applied to the surface of the gear inner cone (the gear is simulated without teeth - they are considered in [Gear Set]). Fig. 12.5 gives the scheme of fixing and loading. Mass-inertia characteristics, which may be obtained in the CAD-system, are also assigned in [Generalized Element]. Inertia moments should be obtained in the centre of mass of the simulated element. The centre mass coordinate of the [Generalized Element] element is defined in its settings and counted from its last section on the left (z_1 coordinate). Fig. 12.6 gives an example of settings of the [Generalized Element] parameters.

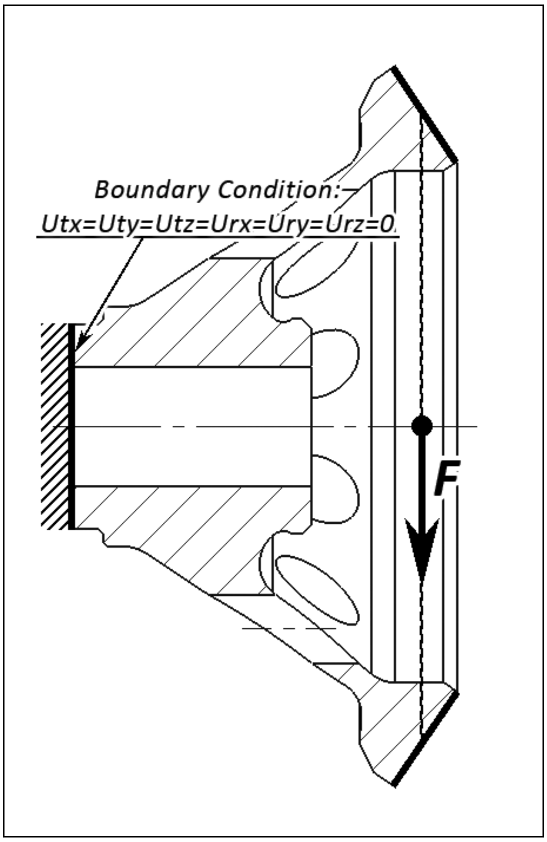


Fig. 12.5

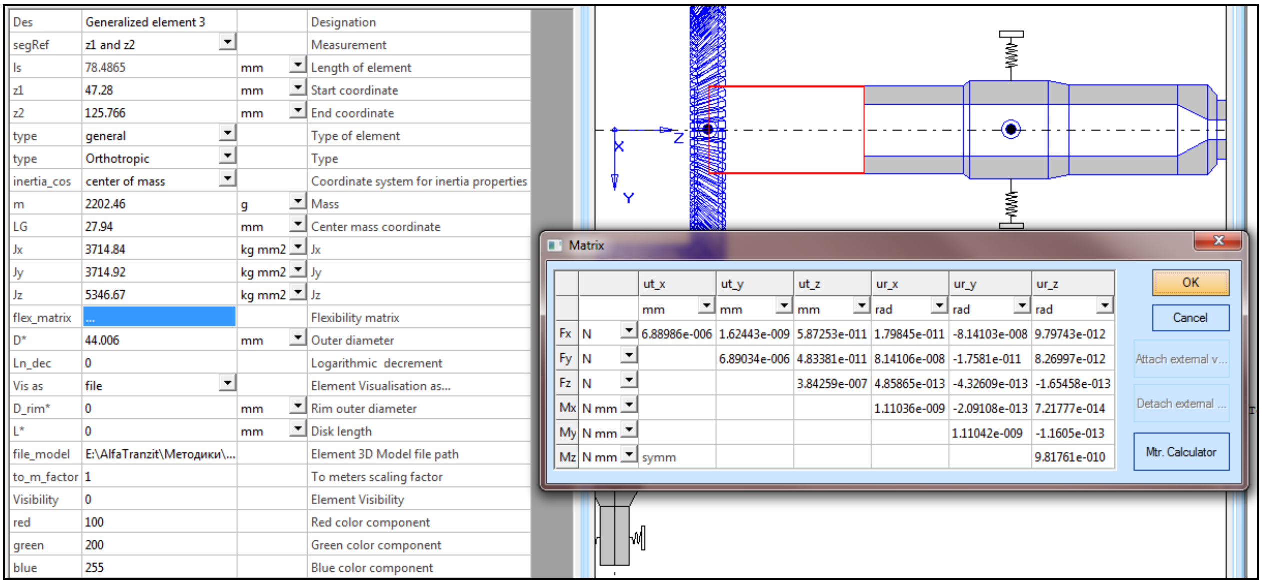


Fig. 12.6

Note: When using visualization of generalized element with help of external 3D-file (in the VRML-format, for example) you must take into account that the origin of the coordinate system will be placed in the element section with z1 coordinate.

Note: The peculiarity of the [Generalized Element] is possibility to place point elements (mass, connection point, unbalance, etc.) only in boundary sections.

12.1.2 Simulation using elastic-damping link [Link]

This method is urgent if it is necessary to consider the journal damping abilities and if the centre of mass of the journal is unknown. In this case the gear is simulated in the other subsystem separated from the shaft and consisted of the point mass [Mass] and the [Connection Point].Mass and inertia characteristics of the journal is assigned in the [Mass] element, its stiffness characteristics – in the [Link] element connecting the gear subsystem (connection point placed together with point mass that simulates the gear mass-inertia characteristics) and the shaft subsystem. The connection point in the shaft subsystem must be placed either in the boundary section (as in the given example) or at the centre of the surface where the gear if fastened (for example, in the centre of spline coupling). This is conditioned by passage of force loads appearing in gearing and transmitting onto the shaft supports. Fig. 12.7 gives the example of the model.

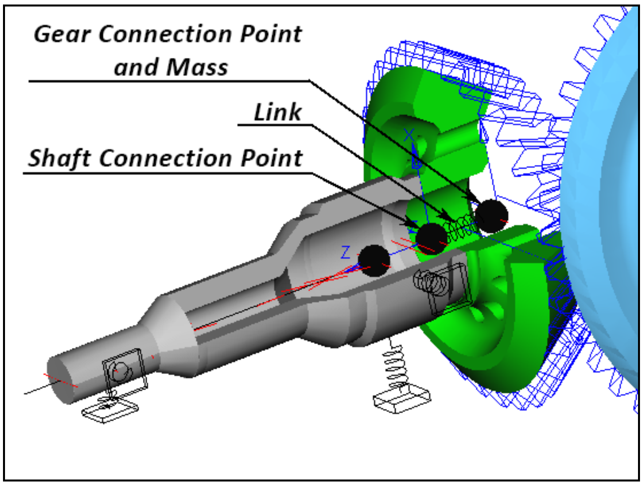


Fig. 12.7

12.2 Results of simulation

Fig. 12.8 presents the results of calculation of the [Basis] algorithm for the model of the gear simulated using the both methods (the blue gear is simulated using the element [Generalized Element], the green gear – by the point mass [Mass]).

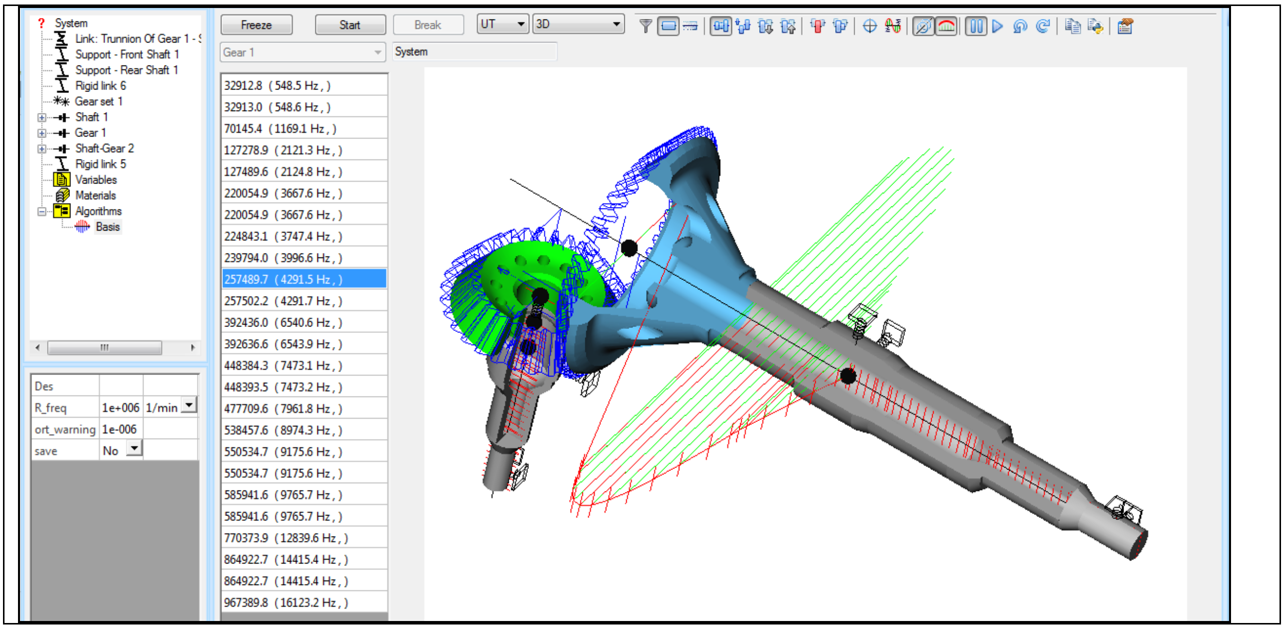


Fig. 12.8

13.Simulation of shell shaft with a section of local flexibility

This methodology is suitable to simulate shafts with flexibility section of any geometry such as the cavity, the cross bore, the groove, etc. This example considers simulation of a shaft with a section of complex shape given in Fig 13.1. The shaft material is steel with density of 7850 kg/m^3, modulus of elasticity of 2.1e+005 MPa, Poisson's ratio of 0.3. This design solution may be used to lower critical rotating speed at bending mode shape.

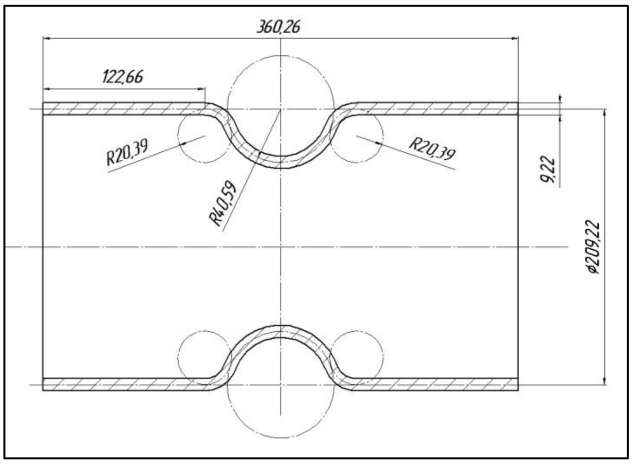


Fig 13.1 Geometry of flexible section of shaft

In order to simulate the shaft with complex geometry, you should use the element [Generalised element] to simulate this section. In the elements parameters you must define the section length, its mass, the coordinate of the mass centre, inertia moments calculated relatively to the mass centre and flexibility matrix of the section.

To obtain the flexibility matrix of the section, you may use the FEM system. For this you must build the FEM-model of the section (it is recommended to take into account certain length of the shaft of simple geometry in order to identify influence of boundary effects and stress raisers on the shaft flexibility with higher accuracy). Fig. 13.2 shows the FEM-model of the considered section built using the shell elements.

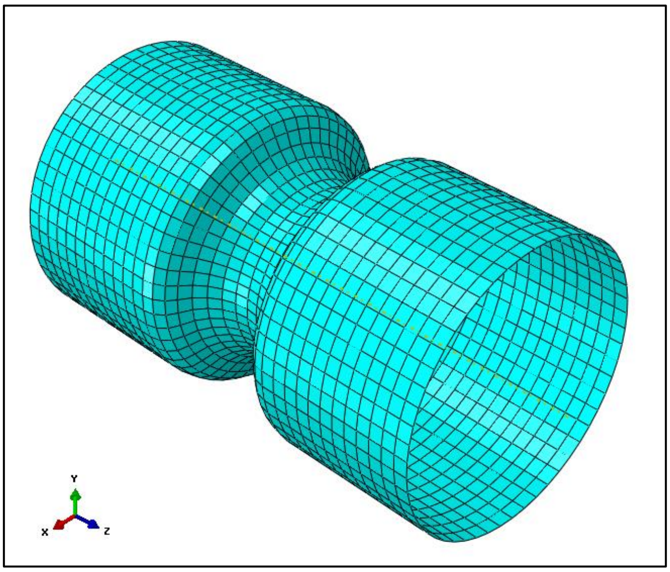


Fig. 13.2 FEM-model of flexible section of shaft

Flexibility matrix of the section is calculated as measurement of displacements and rotation angles at all freedom degrees of one boundary section at its alternate loading by unit forces and moments at all freedom degrees when fixing the other boundary section. Fig. 13.3 gives the scheme of flexibility matrix obtainment.

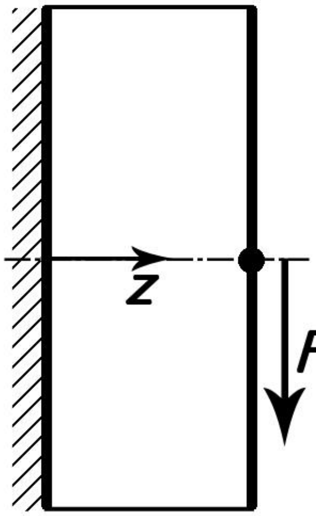


Fig. 13.3 Scheme of boundary conditions of FEM-model

As a result of FEM-calculation, the full flexibility matrix of the section must be obtained (Table 13.1) and pasted in the corresponding dialog window at operation with parameters of generalized element simulating the shaft section in the DYNAMICSR4 model (Fig. 13.4).

Table 13.1

	ut_x [mm]	ut_y [mm]	ut_z [mm]	ur_x [rad]	ur_y [rad]	ur_z [rad]
Fx [N]	1.45E-05	1.43E-18	1.28E-13	-6.32E-21	6.78E-08	-1.15E-18
Fy [N]	-9.33E-18	1.45E-05	9.74E-17	-6.78E-08	-2.87E-20	-2.21E-14
Fz [N]	1.28E-13	3.95E-17	1.26E-06	1.34E-20	8.06E-16	1.17E-20
Mx [N mm]	-5.92E-19	-6.78E-08	4.53E-19	3.76E-10	-3.33E-21	-8.77E-19
My [N mm]	6.78E-08	2.66E-19	8.06E-16	-1.31E-21	3.76E-10	-6.26E-21
Mz [N mm]	-1.16E-18	-2.21E-14	7.95E-21	-8.77E-19	-6.17E-21	1.13E-10

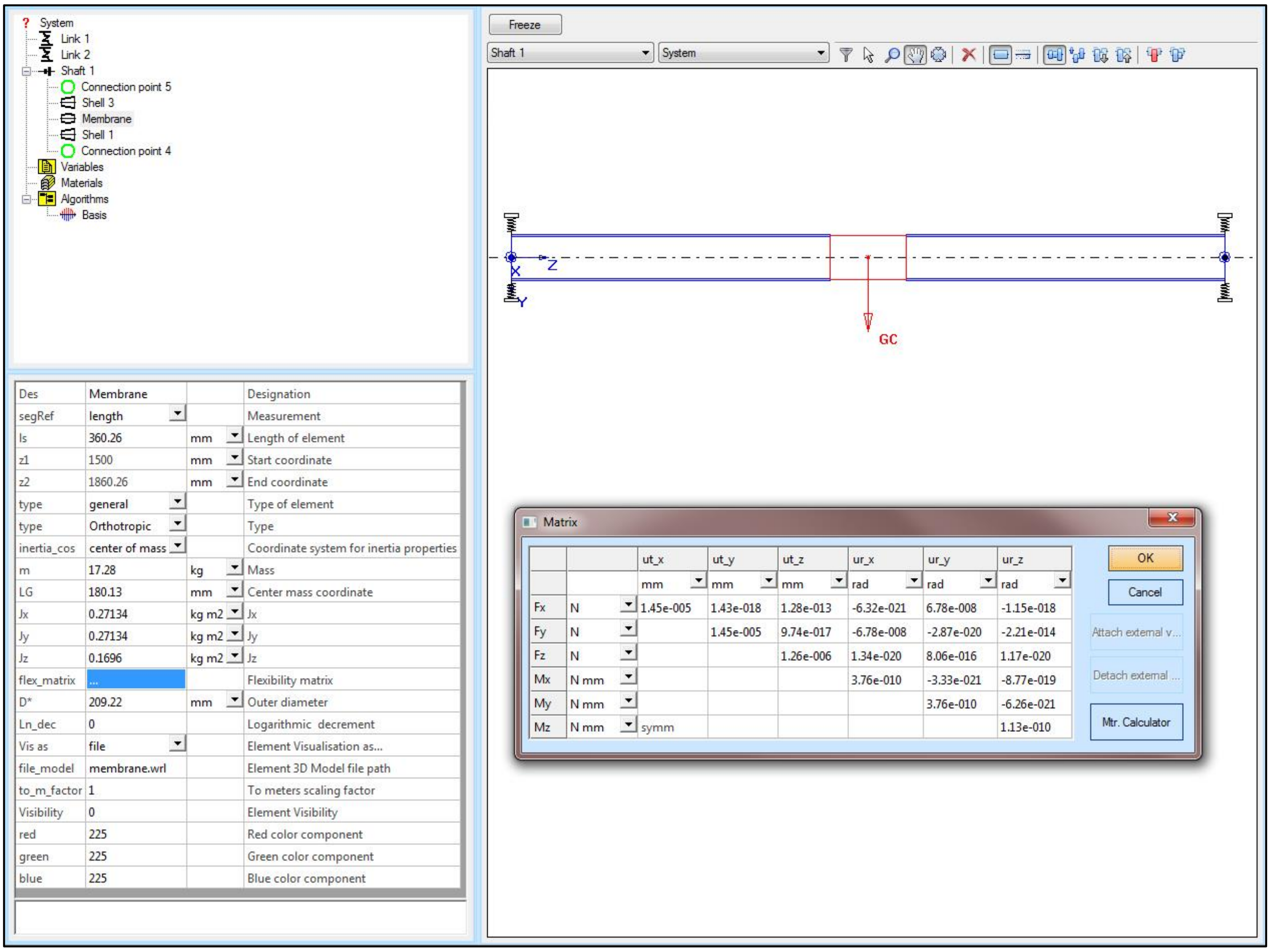


Fig. 13.4 Setting of parameters of generalized element

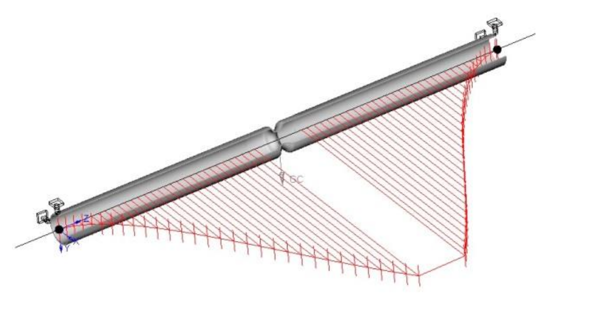
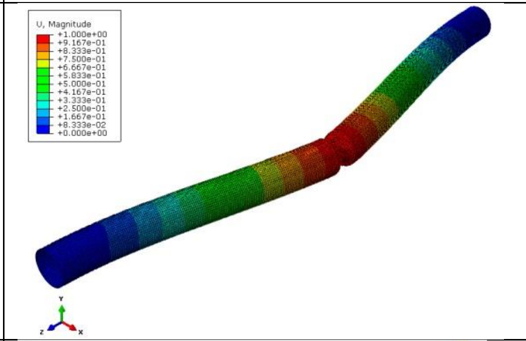
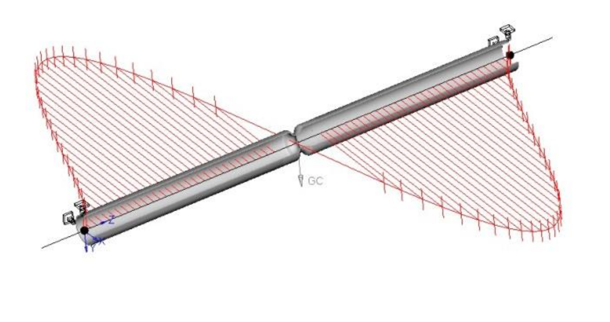
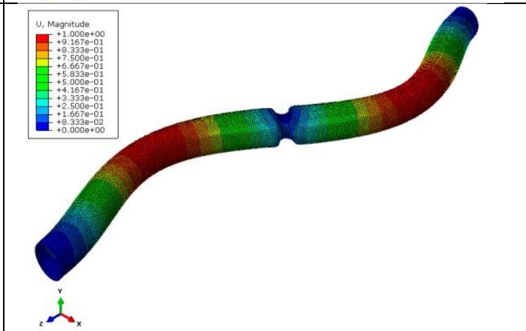
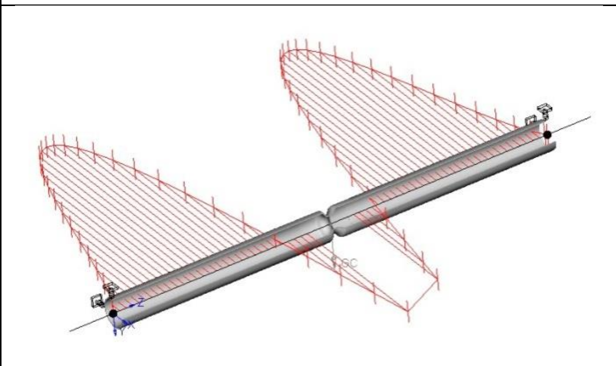
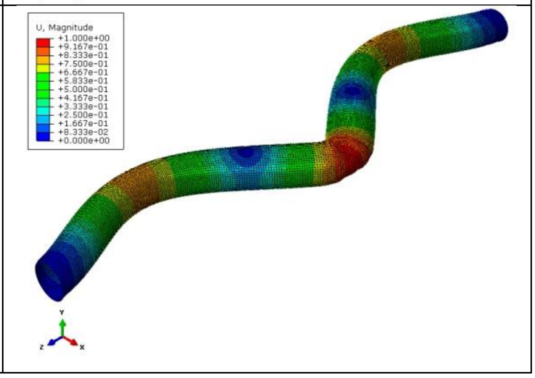
In this example cylindrical sections of the shafts represent thin shells of the constant diameter and thickness in correspondence with the boundary of complex section. The length of every cylindrical section is 1500 mm.

In order to validate simulation correctness, calculation results of natural frequencies of the given shaft in DYNAMICS R4 are compared with those in FEM-system at rigid restraint of the shaft boundary sections. Table 13.2 and Table 13.3 give the results and show high accuracy of similar geometry modeling in DYNAMICSR4.

Table 13.2

Shape	Frequencies (Hz)		Comparison (%)
	DYNAMICS R4	МКЭ	
1 flexural	89.5	91.522	2.2
2 flexural	292.2	293.05	0.3
3 flexural	409.9	445.83	8.1
1 torsional	480.2	482.39	0.5
2 torsional	884.2	890.82	0.7
3 torsional	1416.2	1447.20	2.1
1 axial	759.3	768.94	1.3
2 axial	1187.2	1191.10	0.3
3 axial	2212.8	2291.70	3.4

Table 13.3

Shape	DYNAMICS R4	FEM
1 flexural		
2 flexural		
3 flexural		

14.Simulation of influence of axial load on natural frequencies

Axial loads in rotating machines may influence flexural stiffness of shafts. Axial force acting on compressing decreases natural frequencies of the shaft (lowers stiffness). On stretching – increases. Let us consider simulation by example of cantilever rod of length 8 m, diameter 0.3m. Automatic division of the model into sections, parameter cs = 50 mm.

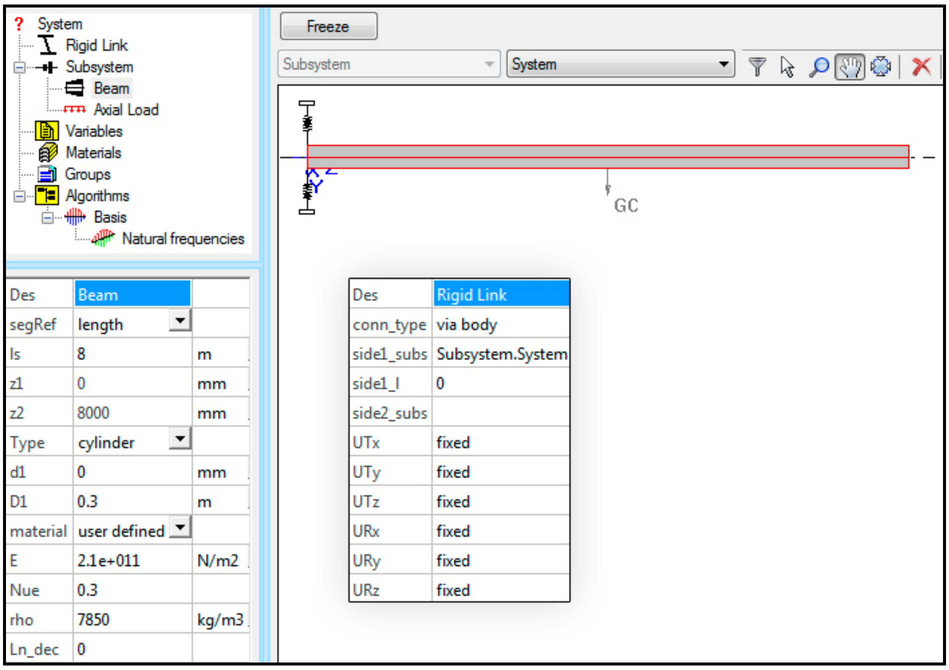


Fig. 14.1 – Model of cantilever rod

Influence of axial load on natural frequencies is simulated by the element [Axial load] (Fig 14.2). Coordinates given in the element's settings assign loaded section of the rod. Constant type of loading means that diagram of strains is constant, even, as for a coupling bolt. Positive sign of force means stretching, negative – compressing. For more obvious display of the effect, force should be comparable to critical load (Euler force) for the present rod. As it is necessary to take into consideration that the given element does not influence calculation of [Basis] algorithm – natural frequencies with axial load consideration should be calculated using the [Natural frequencies] algorithm.

Natural frequencies of the simulated rod without load, with load on stretching and compressing are given in the table 1.1.

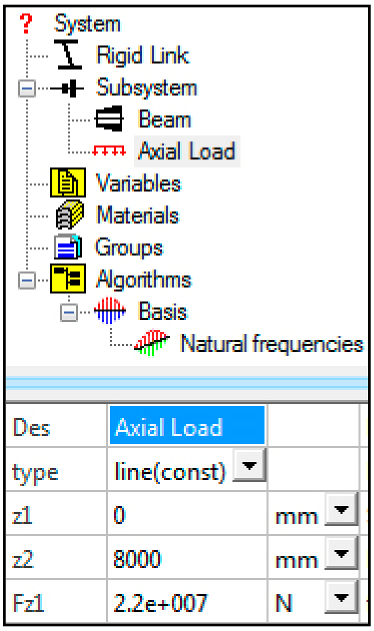


Fig 14.2 – Element [Axial load]

Table 14.1 – Natural frequencies of cantilever beam in Dynamics R4

shape	frequency before applying load, Hz	frequency after applying load 2.2E+007 N, Hz		frequency change, %	
		on stretching	on compressing	on stretching	at compression
1 flexural	3.40	8.30	0.00	144.1	100.0
2 flexural	21.10	30.40	6.20	44.1	70.6
3 flexural	58.60	67.80	47.80	15.7	18.4
1 torsional	100.20	100.20	100.20	0.0	0.0
4 flexural	113.50	122.20	104.00	7.7	8.4
1 axial	161.60	161.60	161.60	0.0	0.0
5 flexural	184.70	193.00	176.10	4.5	4.7
6 flexural	271.00	278.80	263.00	2.9	3.0
2 torsional	300.70	300.70	300.70	0.0	0.0
7 flexural	370.80	378.20	363.30	2.0	2.0
8 flexural	482.60	484.90	475.60	0.5	1.5
2 axial	484.90	489.60	484.90	1.0	0.0

To validate operation of the element, FEM –simulation of the identical model is carried out. The model is built using beam quadratic elements which consider shift. Division into elements is similar to the Dynamics R4 model – length of every separate element is 50 mm. Material parameters of both models are also similar. Boundary conditions are fixing at six freedom degrees from one end and loading by axial force from the other one – correspond to the Dynamics R4 model. Table 14.2 gives the obtained results and their comparison with Dynamics R4 results

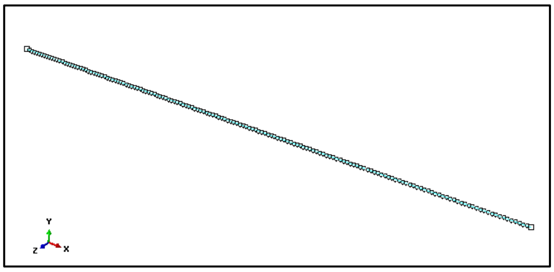
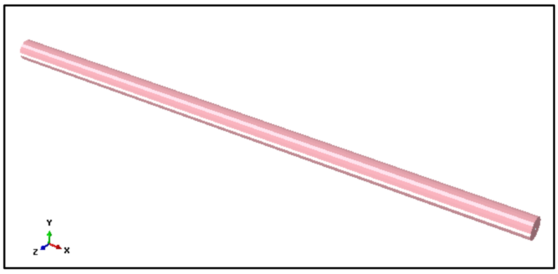



Figure 14.3 – FEM-model of rod

To estimate influence of axial force on the beam natural frequencies, you may compare acting force with Euler force. Euler force (1) is critical force of axial compression at which first natural frequency is equal to zero. Frequencies of other mode shapes also decrease, but to a lesser degree. In the considered case critical Euler force is ≈3.33E+006 N (for E = 2.1E+05 MPa, J = 3.976E-004 m^4, l = 8 m, π=3.142)

$$Fcr = \frac{\pi^2 EJ}{4l^2} \quad (1)$$

Table 14.2 – Natural frequencies of cantilever rod, FEM – calculation, comparison with Dynamics R4

shape	frequency before applying load, Hz	frequency after applying load 2.2E+007 N, Hz		comparison with Dynamics R4, %		
		on stretching	on compressing	without loading	at stretching	at compressing
1 flexural	3.39	8.28	0.00	-0.3	-0.2	0.0
2 flexural	21.14	30.41	6.10	0.18	0.0	-1.6
3 flexural	58.74	67.99	47.71	0.24	0.3	-0.2
1 torsional	100.24	100.36	100.12	0.04	0.2	-0.1
4 flexural	113.88	122.88	104.05	0.33	0.6	0.0
1 axial	161.63	161.51	161.75	0.02	-0.1	0.1
5 flexural	185.71	194.44	176.51	0.55	0.7	0.2
6 flexural	272.99	281.47	264.20	0.73	1.0	0.5
2 torsional	300.72	301.07	300.36	0.01	0.1	-0.1
7 flexural	374.39	382.65	365.91	0.97	1.2	0.7
8 flexural	488.52	484.53	480.29	1.23	-0.1	1.0
2 axial	484.89	496.59	485.25	0.00	1.4	0.1

In general case equation to determine critical load is of the form (2) (Feodosiev V.I. *Strength of materials: Institute textbook. – 10-nth edition, revised and complemented edition– M.: Bauman Moscow State Technical University Press,1999. – 592 p.*). Here μ=1/n is the value which is reciprocal to the n number of half-waves of sinusoid at which the rod bends (Fig14.4). Constant μ is called length coefficient and product μl – effective rod length. Effective length is length of half-wave of sinusoid at which this rod bends.

$$Fcr = \frac{\pi^2 EJ}{(\mu l)^2} \tag{2}$$

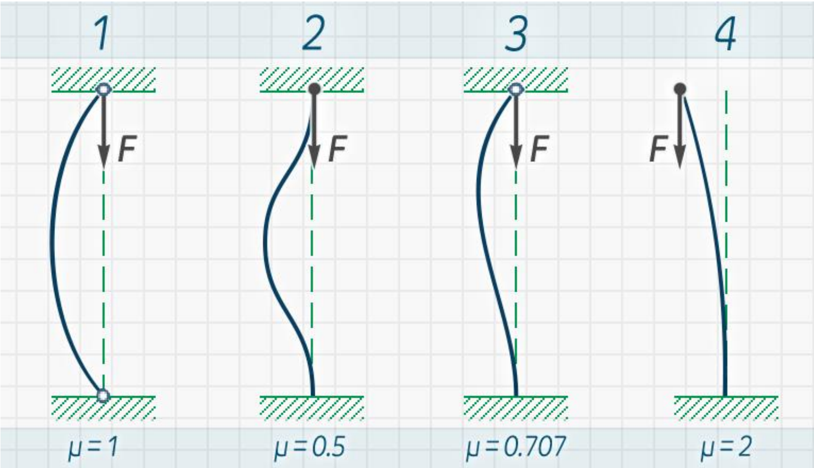


Fig14.4 – Bending of rod

15.Defining geometry of the rotor system by line elements

Dynamics R4 interface has various functionality created to simplify building of the model and the following work with it. In this part, possible variants to work with line elements (two nodal elements having the 'length' parameter) of the [Beam], [Shell], [Generalized element] type are considered. Ways to add the elements into the model, their editing, etc. are described. Methods are shown on the example of the [Beam] element, but the other similar elements may be also applied.

15.1 Variants to add elements into the model

1) Double click on the elements panel

Create the new model (menu File/New) By double click on the elements tree on the right create the new shaft. Go to the parameters of the Beam element (in the left panel) and change value of parameter segRef from "z1 and z2" into "length".

Double click on the element of interest on the elements panel – and it will be added into the model with certain set of parameters "on default". The element will be added on the right from active element (Figure 15.1) and will become active itself. *(The active element in the visualization area is highlighted by red contour)*

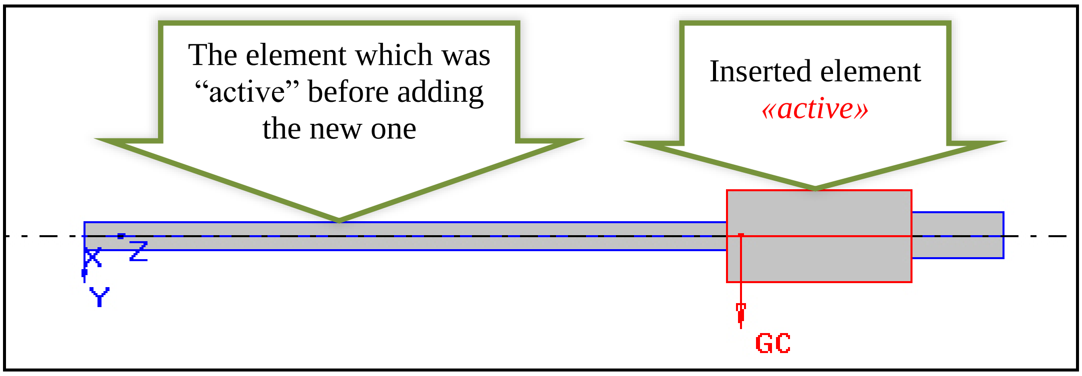


Figure 15.1 - Adding new element by double click

2) Context menu of the elements panel ("Append to model")

The method is similar to the previous one. At right click on the element, context menu appears with the only button "Append to model" (Figure 15.2), at left click the element on the right from active element will be added. The added element will become active automatically.

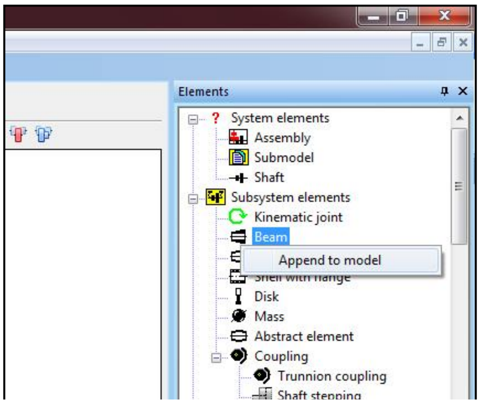


Figure 15.2 Menu "Append to model"

3) Dragging from the elements panel , "drag-and-drop" (cursor activation for this element)

The method is convenient if several one-type elements should be added. Parameters of each one may differ from each another.

Click by left mouse button on the element in the elements panel on the right and pressing the mouse button, drag it in 2-D area of the model visualization and after that release it. Then the mouse cursor "arrow" will be replaced by cursor typical of the chosen model element (Figure 15.3), which testifies activation of the regime of the adding of this element into the model. In this regime at left click on the visualization area the element will be placed where cursor points.

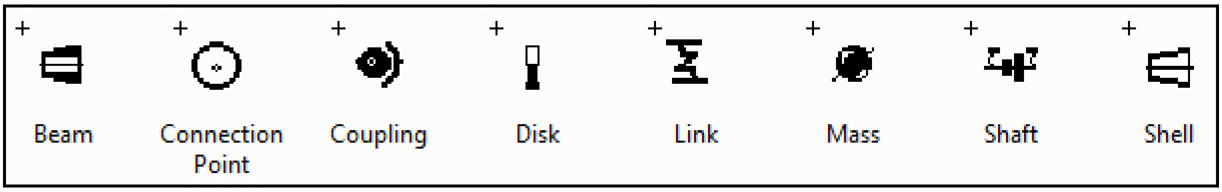


Figure 15.3 Examples of cursors of Dynamics R4 elements

4) Adding intermediate sections

The element may be divided into two ones adding intermediate section. For this it is necessary to click twice on the element in the area of the model visualization by cursor "Pointer) ("arrow"). After that the dialog of correcting Z coordinate of the added section appears(Figure 15.4). On default the coordinate of the section which a user pointed at by cursor is defined. Press the button «OK». Active element will be divided into two elements.

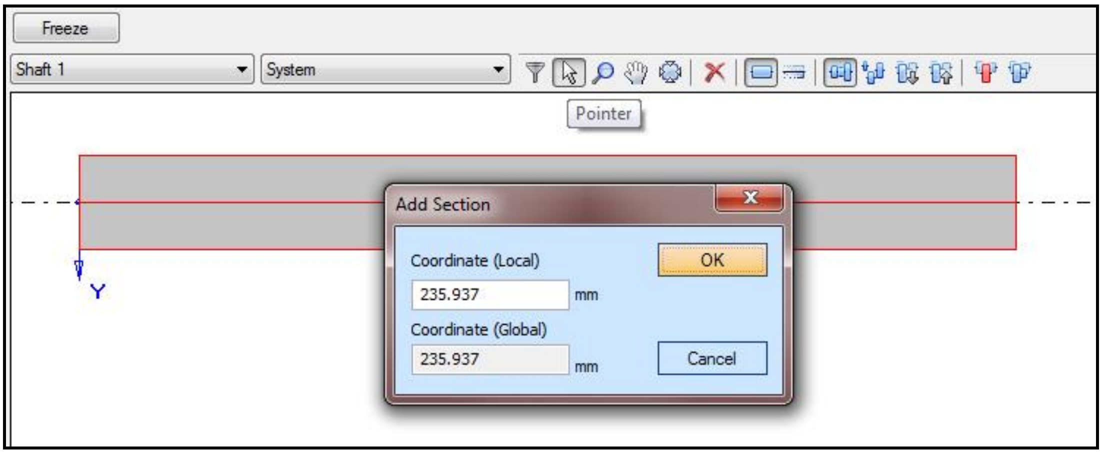
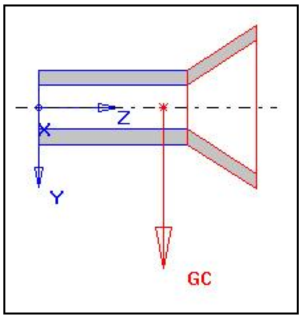


Figure 15.4 The section adding

5) Copying of the previously created element



It is convenient to use such technique for shaft or case consecutive modeling if the right section of i-element has the same geometry as the left section of i+1-element (Figure 15.5). You can copy the active element by three methods :

- To press the button "Copy" on the toolbar ;
- To click by right button in the model tree on copied element and choose "Copy" in the appearing menu (Figure 15.6):

- Press standard combination Ctrl+C on the keyboard.

Figure 15.5

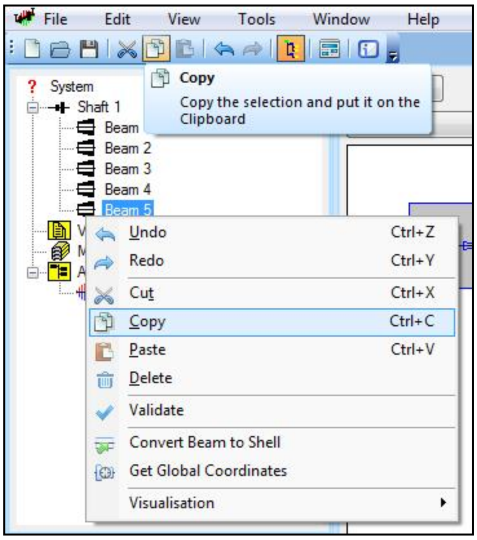


Figure 15.6 – Copying of the element

To paste the copied element, you may also apply one of the three methods

- Press the button «Paste» on the instruments panel;
- Click by right mouse button in the model tree on the copied element and choose [Paste] in the appearing context menu (Figure 15.6).
- Press standard combination Ctrl+V on the keyboard.

As opposed to copying, the element is not inserted immediately; regime of paste of this element is activated. Methods to work with it are similar to the previously described method of dragging elements “drag-and-drop” with the only difference that when pasting the copied element, its parameters are saved and the element becomes the copy of the previous one except the coordinates of its positioning depending on the active method of the element positioning.

15.2 Methods of the element positioning

When adding into the model elements having the length and at their removing from it, it is necessary to consider active method of the element positioning. As you can see in screenshot, there are two methods: «length» and «z1 and z2» (Figure 15.7).

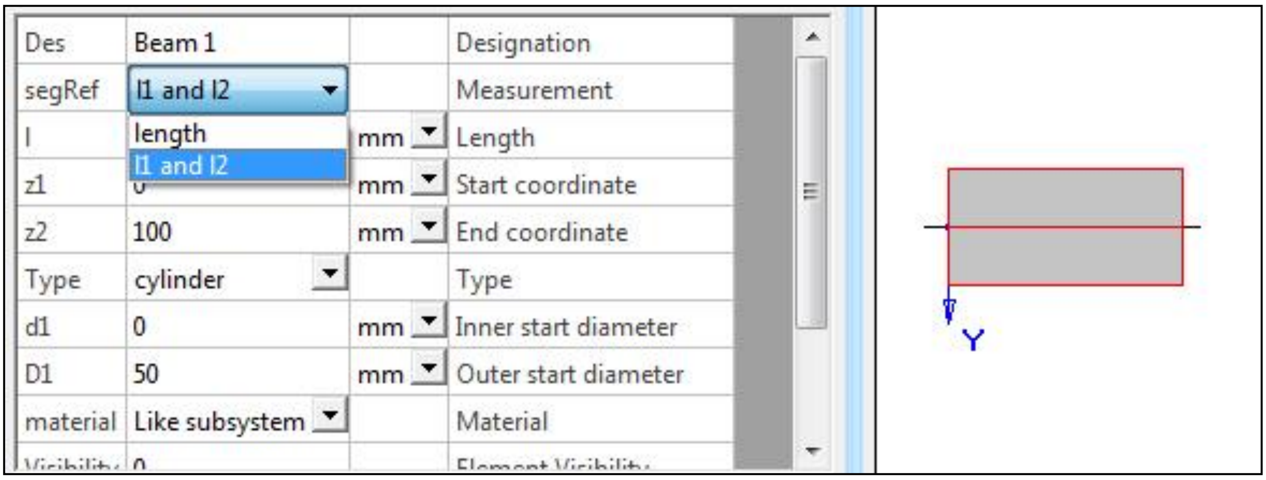


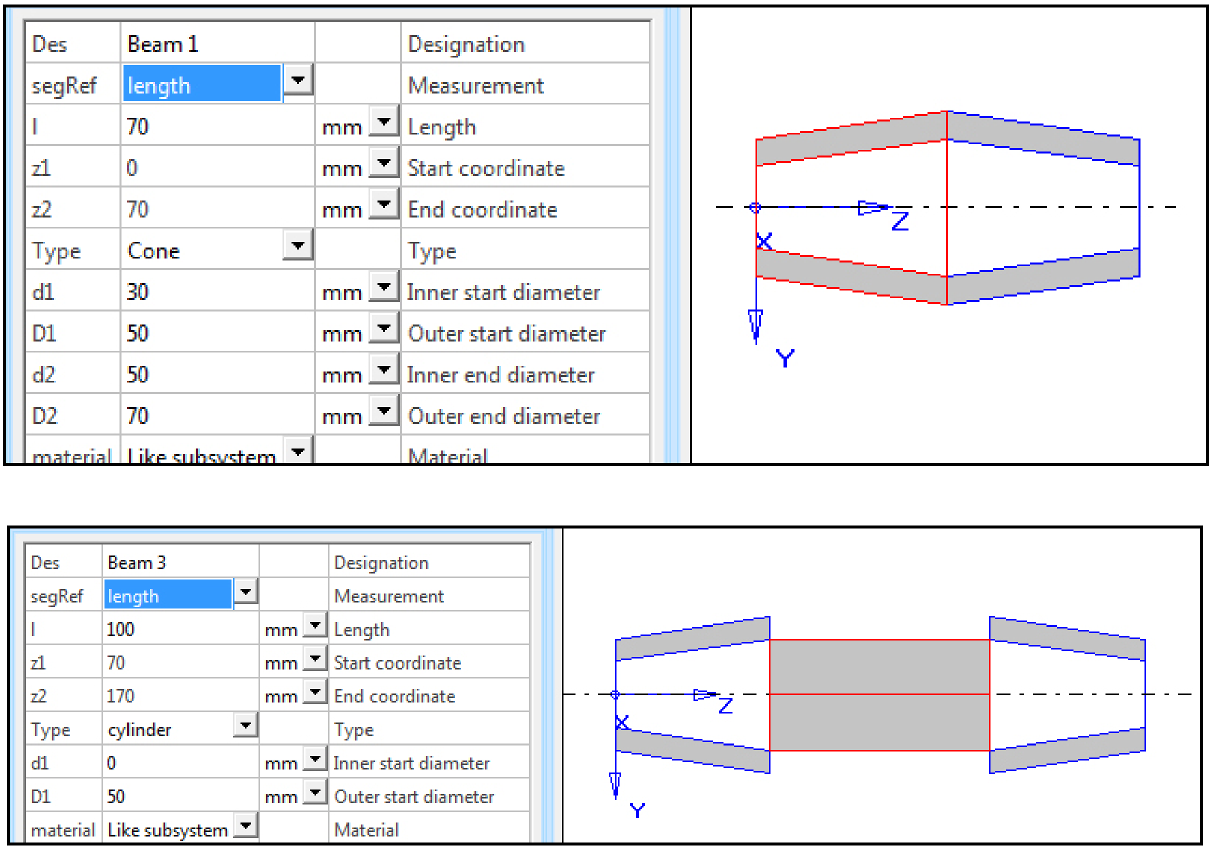
Figure 15.7 – Methods of element positioning

When working with active variant “length” a user can’t change coordinates of boundary sections of the element z1 and z2 – and can change only the parameter ls – “The element length”. When adding and removing the elements, the principle to save the length of every element works– right boundary section of the whole model changes correspondingly.

Let us examine this in details. For this purpose we create the model with two beam elements (Figure 15.8- A). To visualize, their geometry is changed in comparison with the variant “on default”.

Let us make the left element as active one and add one more beam element by double click. It will be pasted between the previously created elements, i.e. on the right from the one which was active (Figure 15.8-B). Inserted element will have the length of 100 mm (on default), and the right one will move to the right, and its geometry will remain the same – length, diameters and the other parameters will not change.

A) B)



Des	Beam 1		Designation
segRef	length		Measurement
l	70	mm	Length
z1	0	mm	Start coordinate
z2	70	mm	End coordinate
Type	Cone		Type
d1	30	mm	Inner start diameter
D1	50	mm	Outer start diameter
d2	50	mm	Inner end diameter
D2	70	mm	Outer end diameter
material	Like subsystem		Material

Des	Beam 3		Designation
segRef	length		Measurement
l	100	mm	Length
z1	70	mm	Start coordinate
z2	170	mm	End coordinate
Type	cylinder		Type
d1	0	mm	Inner start diameter
D1	50	mm	Outer start diameter
material	Like subsystem		Material

Figure 15.8 – Paste of element of the positioning way “length’ (A-before paste, B-after paste)

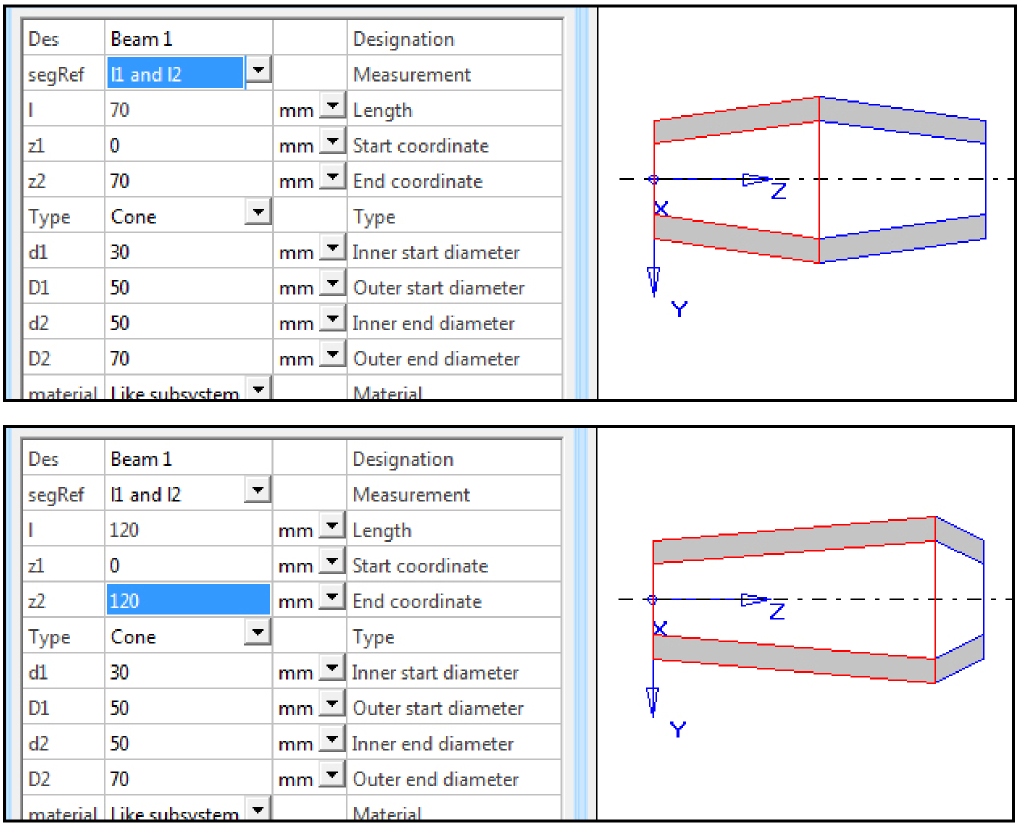
When working with active way of positioning «z1 and z2» a user can’t change directly the parameter ls («Length of the element») – he may edit the parameters z1 and z2, representing the coordinates of the boundary sections of the elements. It is also necessary to take into account that the parameter z2 of i-element corresponds to the parameter z of 1 i+1-element, i.e. the section between two neighboring elements belongs to both of them. It means that at this way of positioning «z1 and z2» a user edits not the elements but coordinates of the sections.

Let us consider the example. We create the model with two beam elements (Figure 15.9Figure 15.8 - A).

We make the left beam element as active one and increase its parameter z2. As a result you can see that the section between the elements of the model has moved to the right, and boundary sections stayed as they were (Figure 15.9- B).

If at this way of positioning the new element is added by double click – it will be added into the model with zero length (Figure 15.10).

If by dragging "drag-and-drop" – the coordinate z2 and the beam element diameter will depend on cursor position (Figure 15.11).



A)

B)

Figure 15.9 – Change in section coordinate at the way of positioning "z1 and z2" (A - before change, B-after change)

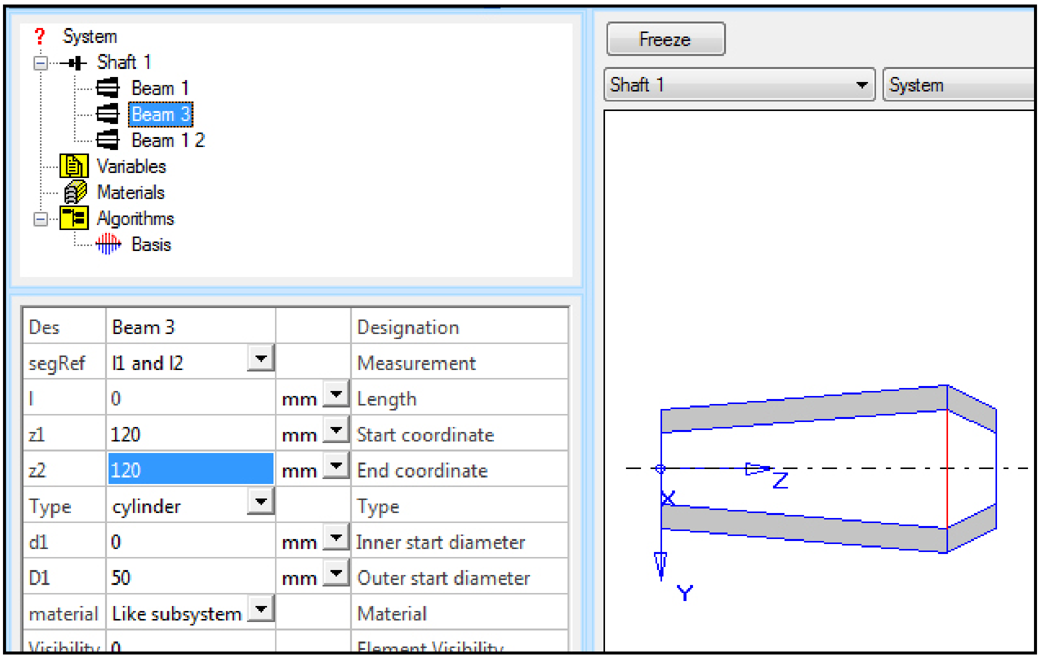


Figure 15.10 – Result of adding element [Beam] by double click at active positioning method "z1 and z2"

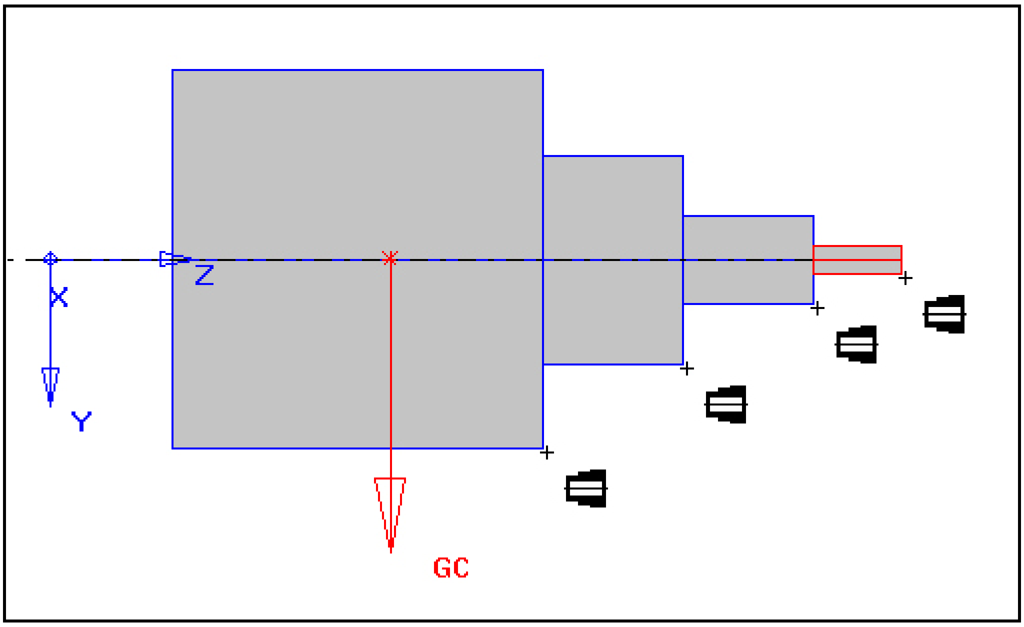


Figure 15.11 – Adding of the element [Beam] by the way “drag-and-drop” at active way of positioning element “z1 and z2”

16. Gas turbine engines model structure

16.1 The model structure

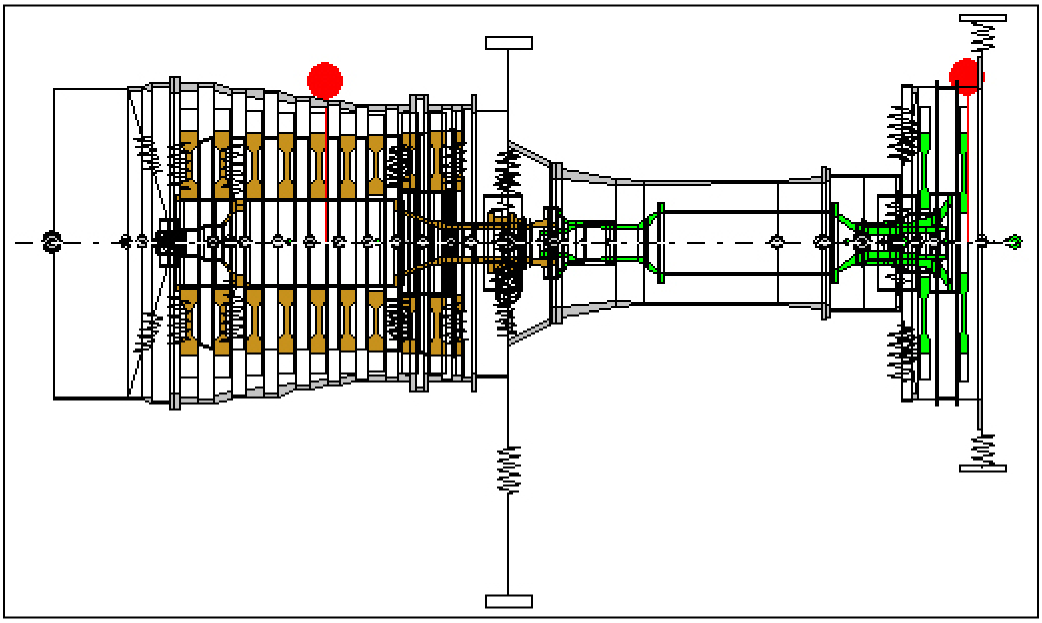
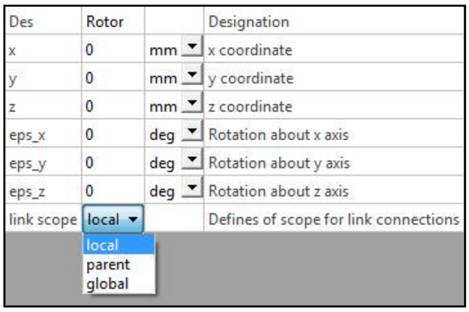


Figure 16.1 Gas turbine engine model

Figure 16.1 gives the model of single shaft gas-turbine engine including the compressor rotor, the turbine rotor and the stator part (the case, the supports) . When building the model it is recommended to work out the model structure thoroughly using not only subsystems but the elements [Assembly]. This element gives an opportunity to structure the model better gathering inside itself subsystems, links between them, variables related to the given subsystems and links . Also several similar links such as those simulating bearings or engine mount/ pylon may be included into the assembly.

The element [Assembly] Assembly is used to unite several submodels, subsystems, etc. into an assembly which will be a part of the whole model of the investigated object as a separate structural unit. The assembly may be edited only being the part of the whole model. It is assigned by the name and position of its local coordinate system in the global one belonging to the whole model, Figure 16.2



Des	Rotor		Designation
x	0	mm	x coordinate
y	0	mm	y coordinate
z	0	mm	z coordinate
eps_x	0	deg	Rotation about x axis
eps_y	0	deg	Rotation about y axis
eps_z	0	deg	Rotation about z axis
link scope	local		Defines of scope for link connections

Figure 16.2

The parameter [link scope] defines the visibility scope of the links connection points in the combo box in the links characteristics (linear or non-linear). It is possible to choose: local – on default, typical behavior as in previous program versions. Only links' points in the current and nested assemblies are output. parent – you can also observe links' points from the assembly that is a level higher, global – you can see all links points in the model.

Figure 16.3 presents the structure of the gas turbine engine using assemblies.

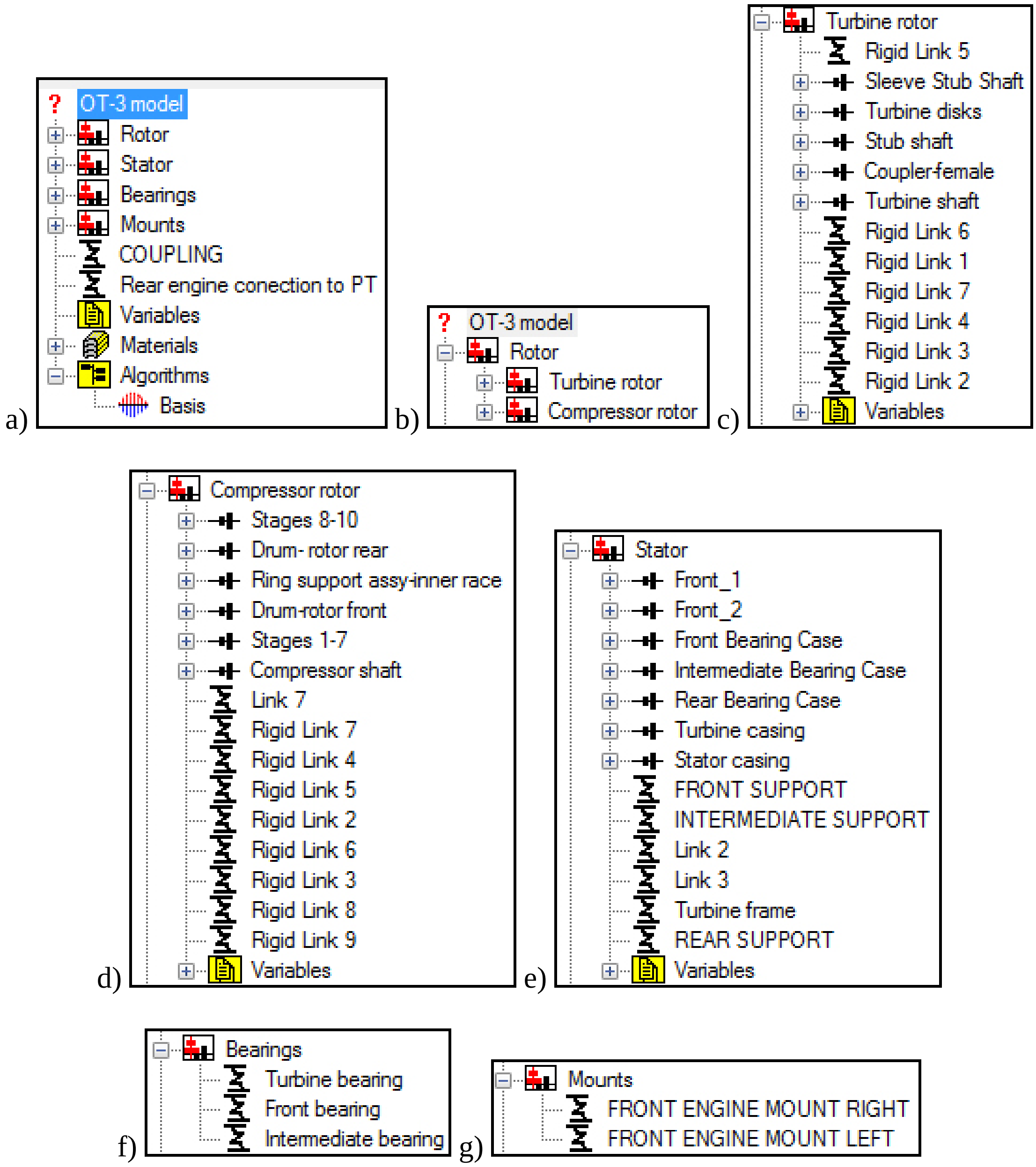


Figure 16.3 – Structure of gas turbine engine model; a) general model structure; b) “Rotor” assembly; c) “Turbine rotor” assembly; d) “Compressor rotor” assembly, e)” Stator” assembly; f)”Bearings” assembly; g) “Mounts” assembly

As the presented above model structure shows, subsystems related to the same structural unit – for example, the turbine rotor subsystems, are united into the corresponding assembly «Turbine rotor». Links connecting subsystems inside one assembly are also distributed among assemblies. For example, there are the subsystems “”Stages 1-7” and “Stages 8-10” in the subsystem “Compressor rotor”. Division of one rotor into subsystems is usually related to division into modules and mass-inertial validation of the model. The link connecting these two subsystems are also added into the assembly “Compressor rotor”. Links used to connect subsystems into different assemblies are on more general level of the model structure – for example, link «COUPLING», simulating coupling of joint between the compressor rotor with turbine rotor.

Links simulating bearings (Figure 16.3 – f) and a mount (Figure 16.3 – e) are placed in separate assemblies. It is also done for more convenient and clear model structuring. However, trying to unite links into a similar assembly a user can face the problem that in the list of connection points of a link attachment (edit link connection) there is nothing. This is related to the {Assembly} element settings on default, specifically with the parameter link_scope – it is responsible for visuability of links points of different levels in links settings in this assembly(see above).

Due to such structuring, several engineers can work easier with the model, it is simpler to find needed element at the model editing and variation of some parameters.

This functionality of Dynamics R4 gives also opportunity to space models not only at subsystems but at assemblies in the area of visual displaying (Figure 16.3). Figure 16.4 compares both variants of the model spacing.

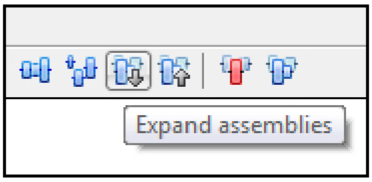


Figure 16.4 – Functionality "expand assemblies"

a) b)

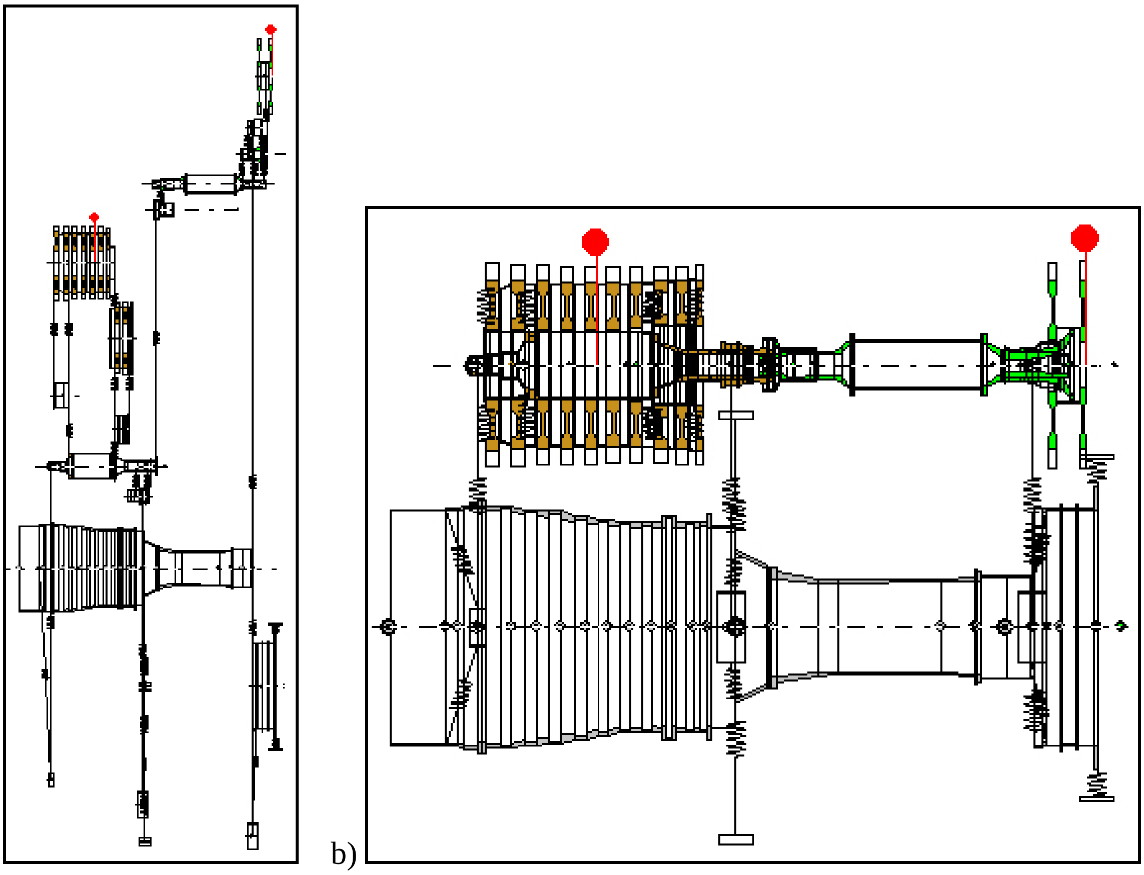


Figure 16.5 – Variants of spacing models at structural units: a) spacing at subsystems, b) spacing at assemblies

For example, the variant to space at assemblies is convenient to describe the model in the report documentation: the models of the compressor and turbine, the case model and links between them are highlighted separately.

The option of spacing at subsystems is convenient at work with separate assembly. In order to make spacing at subsystems for specific assembly it is necessary to use functionality with the sign "Filter" and choose the assembly of interest in it (Figure 16.5).

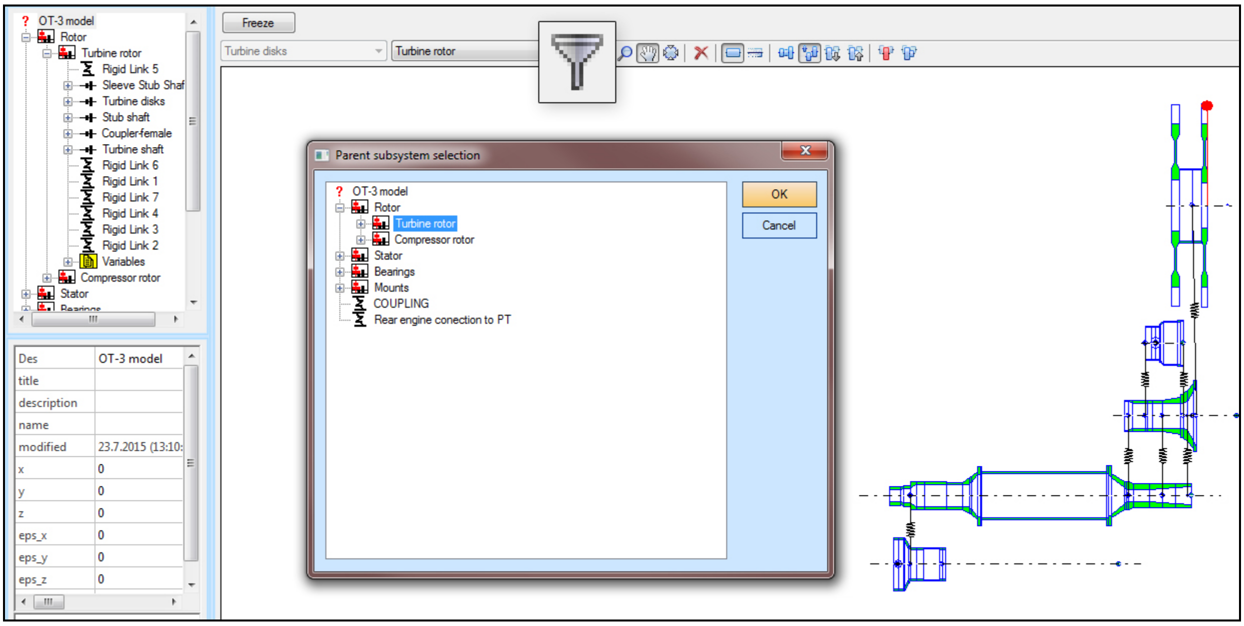


Figure 16.6 – Spacing of assembly “Turbine rotor” at subsystems

Functions “Active subsystem” and “Active assembly” (Figure 16.7) allow to work with the complex multilevel model. Shows only the structural unit which is being worked with at the moment.

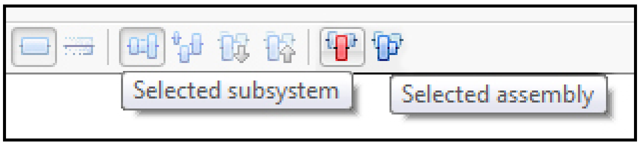


Figure 16.7 – Functionality “Selected subsystem” and “Selected assembly’

17.Simulation of non-synchronous excitation in the linear analysis

17.1 Simulation principles

The non-synchronous excitation is the excitation with a frequency not equal to the rotor rotation frequency. In the Dynamics R4 linear analysis all unbalances must act in sub-systems with equal rotor's rotations. This assumption means that in a multi-rotor system with different rotation speeds the [Unbalance Load] elements may be applied only to rotors with equal rotation speeds. The elements may be added or removed with the [Group] element (ref. User Manual Item 12.1 "Groups of elements [Group]". The non-linear algorithm [Transient Response] this requirement is absent, the multi-rotor systems may have different rotation speeds, including the rotors counter-rotation, any rotor may have unbalance loads. On the other side the [Non-linear Analysis] of complicated systems with numerous sections and links may need larger resources and time. Thus it is reasonable to simulate the so-called "rotors sliding" in the linear analysis.

The linear analysis of a rotor with the non-synchronous excitement requires two linear calculations, one for the rotor unbalance loads and the second for the non-synchronous loads. The analysis results are combined, the Dynamics R4 allows results export in the convenient Microsoft Office Excel format.

The example below is the non-synchronous excitement of a rotor with a mid-span disc. The model and its parameters are shown in figures 17.1 to 17.6. The system model consists of two sub-systems, the rotor with a mid-span disc on flexible supports [Shaft] and the non-synchronous excitement model [Sliding]. The link between the sub-systems is absolutely rigid. The both sub-systems accelerate, the rotor from 0 to 10000rpm, the [Sliding] sub-system from 0 to 20000rpm.

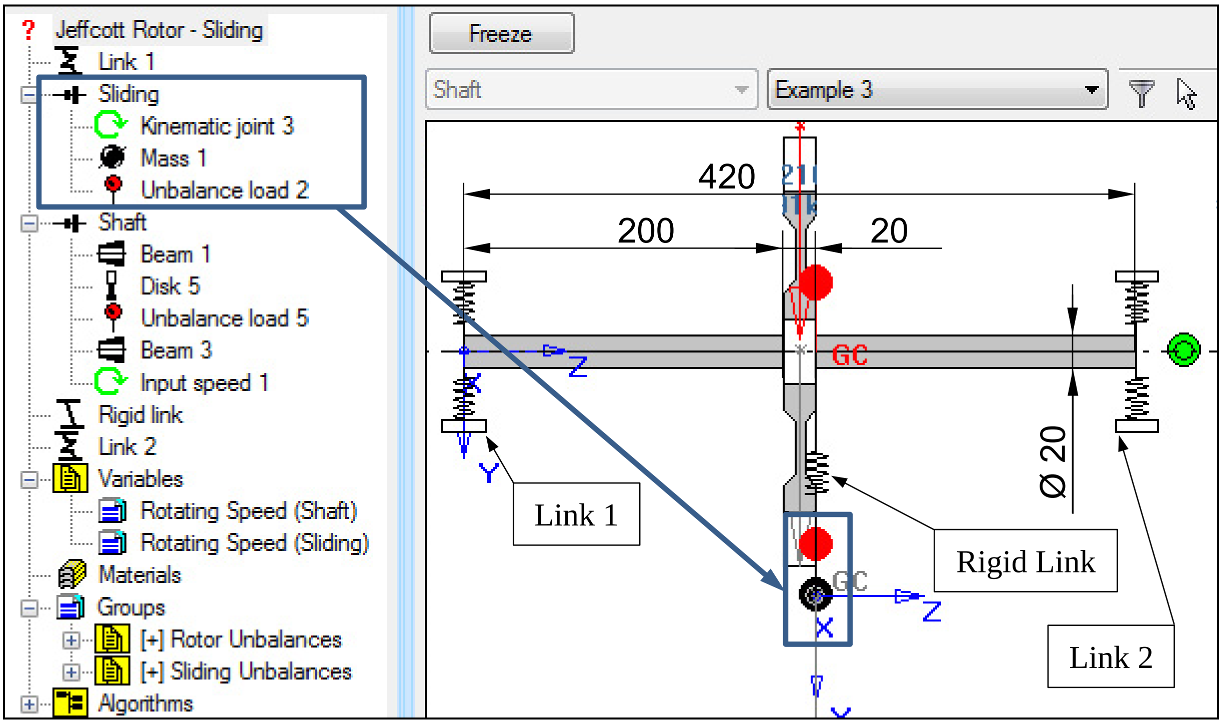
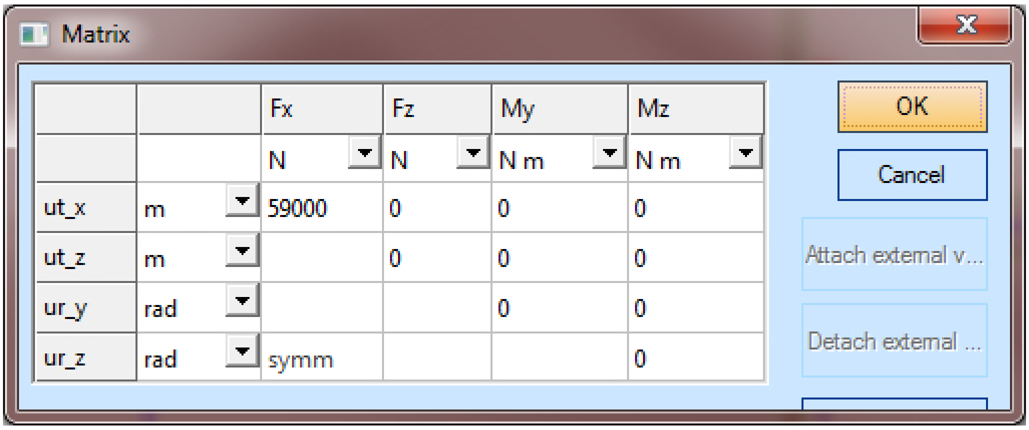
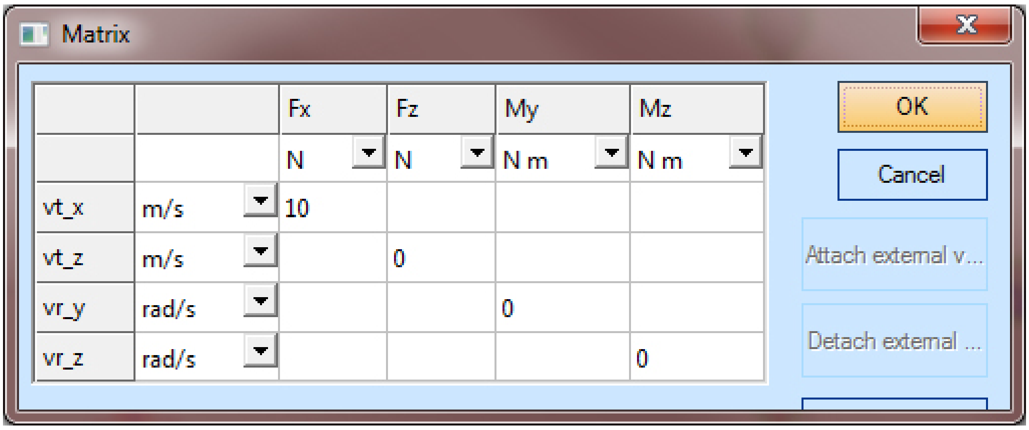


Figure 17.1 - Dynamics R4 rotor model



		Fx	Fz	My	Mz
		N	N	N m	N m
ut_x	m	59000	0	0	0
ut_z	m		0	0	0
ur_y	rad			0	0
ur_z	rad	symm			0

Figure 17.2 – Stiffness matrix of the rotor supports [Link 1] and [Link 2]



		Fx	Fz	My	Mz
		N	N	N m	N m
vt_x	m/s	10			
vt_z	m/s		0		
vr_y	rad/s			0	
vr_z	rad/s				0

Figure 17.3 – Damping matrix of the rotor supports [Link 1] and [Link 2]

Des	Disk 5		Designation
segRef	length		Measurement
ls	20	mm	Length of element
z1	200	mm	Start coordinate
z2	220	mm	End coordinate
m	1.5	kg	Mass
Jx	0.001	kg m2	Jx
Jz	0.002	kg m2	Jz

Des	Mass 1		Designation
z1	0	mm	Start coordinate
m	0.001	kg	Mass
type	Isotropic		Type
Jx	0.001	kg m2	Jx
Jz	0.001	kg m2	Jz

Figure 17.4 - Disc and lumped mass parameters in the [Sliding] sub-system.

Des	Unbalance load 5		Designation
z1	220	mm	Start coordinate
F unb	20	gcm	unbalance force
F phase	0	deg	unbalance force phase
M unb	20	g cm cm	unbalance moment
M phase	0	deg	unbalance moment phase

Figure 17.5 - Rotor unbalance load parameters in the [Sliding] sub-system.

Des	Shaft		Designation
material	user defined		Material
E	2.1e+011	N/m2	Modulus of elasticity
Nue	0.3		Poisson's ratio
rho	7850	m	Density
Ln_dec	0		Logarithmic decrement
cs	25	mm	calculation step

Figure 17.6 - Rotor sub-system parameters.

17.2 Linear calculation of non-synchronous excitement

It is mentioned above that it is necessary to carry out two algorithms [Unbalance Response] and combine the calculation results. The first calculation considers the rotor unbalances. The [Rotor Unbalances] group must be in the "Enable" state and the [Sliding Unbalances] group "Disable" (Figure 17.7). The result will be the Magnitude-Frequency chart produced by the rotor internal unbalances (Figure 17.8).

The second calculation considers unbalances of the sub-system that produces the non-synchronous excitation. So the groups' status must be changed to the opposite [Rotor Unbalances] to "disable" and [Sliding Unbalances] to "enable". The result will be the Magnitude-Frequency chart produced by the external excitation, in our example by the non-synchro rotation of the sub-system [Sliding] (Figure 17.9).

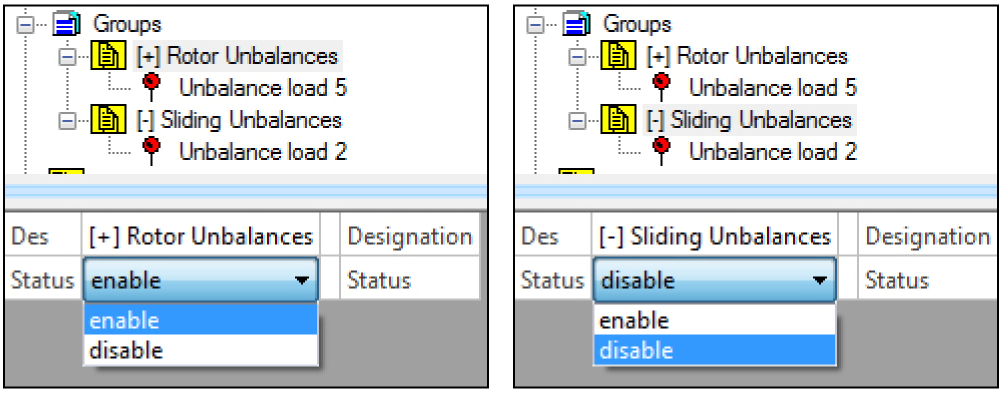


Figure 17.7 – Changes of [Group] elements status

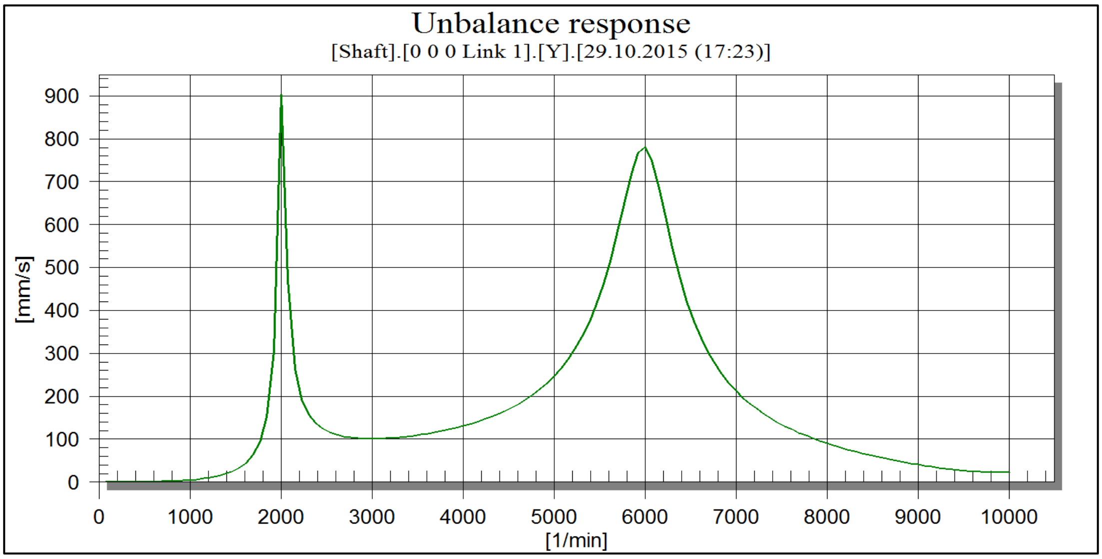


Figure 17.8 - Calculation results of [Unbalance Response] with rotor unbalance enabled.

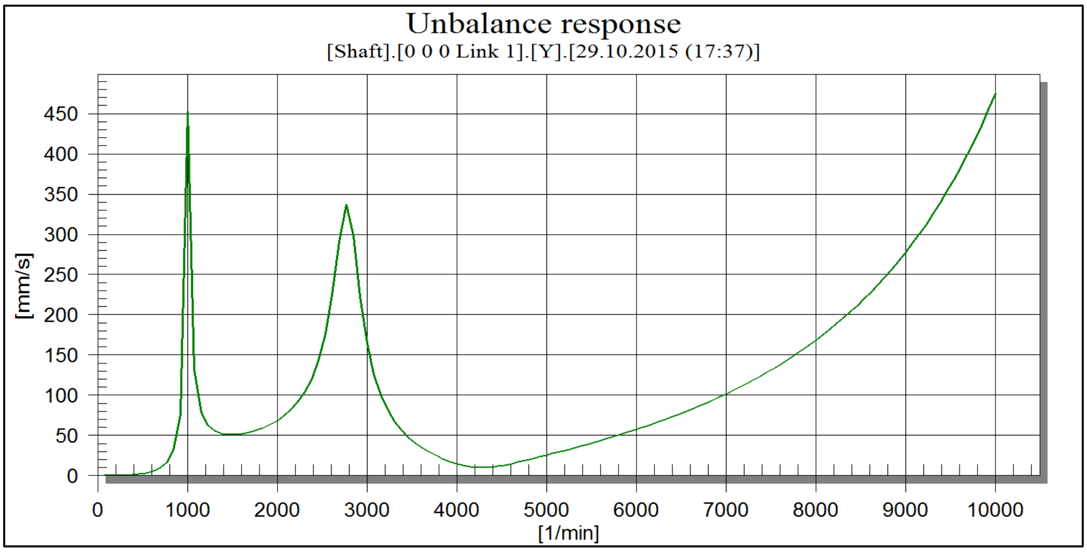


Figure 17.9 - Calculation results of [Unbalance Response] with sub-system [Sliding] unbalance enabled. The results are shown for the rotor rotation speed (sub-system [Shaft])

The results may be combined with the export function *(ref. "User Manual" chapter 14.6 [Unbalance response]).* The combined results will be show later together with the non-linear calculation results.

17.3 Non-synchronous excitation Non-linear calculation

The Dynamics R4 unsteady analysis does not limit application of [Unbalance load] elements with different rotation speeds. Thus it is possible to switch all groups with the [Unbalance Load] elements to the "Enable" and carry out the [Non-linear analysis].

NOTE: This calculation example does not consider weight loads.

Figure 17.10 shows the non-linear analysis resylts as a displacement vs time plot. The complcated dynamic behaviour is explained in figures 17.11 - 17.15 that show this relation with extended time scale.

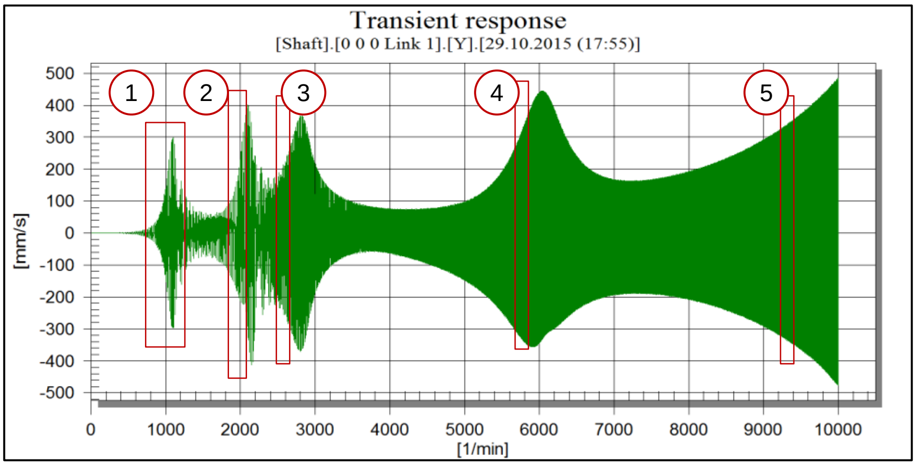


Figure 17.10 – Non-linear analysis results, 0-10000rpm speed range.

Figure 17.11 shows the transient response in the 1050rpm-1400rpm speed range. The displacement signal shows a deep low frequency magnitude modulation and a small 2X component.

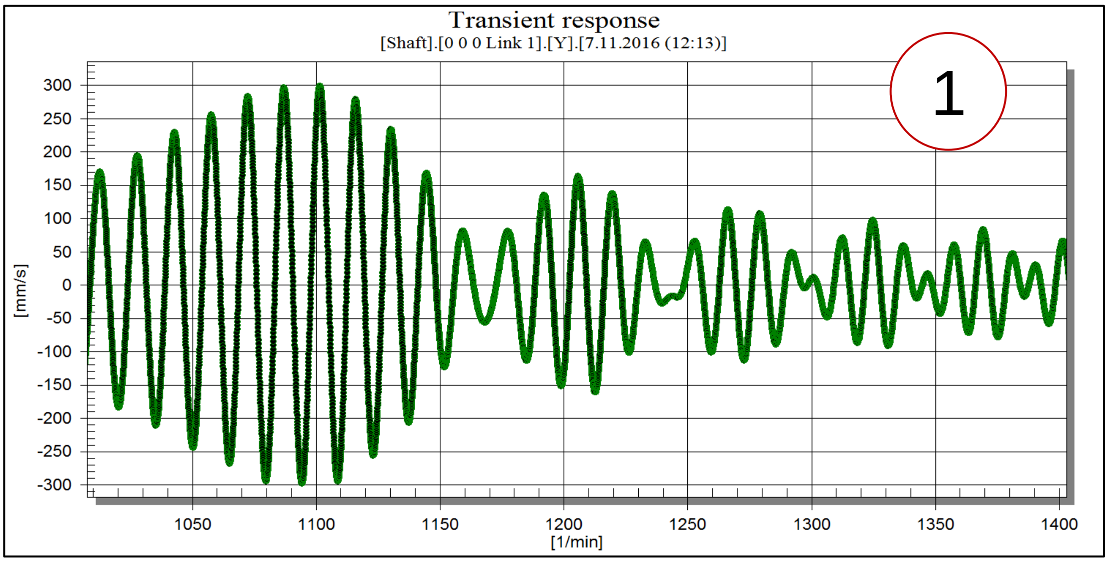


Figure 17.11 - Non-linear analysis results, 1000-1400rpm speed range.

Figure 17.12 also shows in the 2100-2350rpm speed range a modulation but the 2X component is more visible.

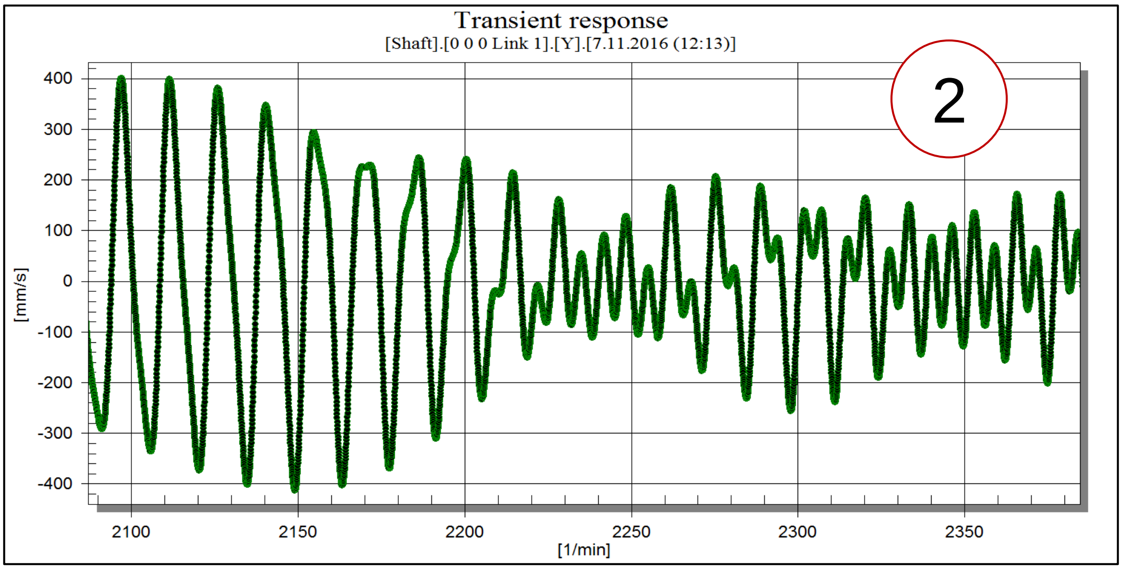


Figure 17.12 - Non-linear analysis results, 2000-2400rpm speed range.

In Figure 17.13 the 2X influence is distinctly seen as the ~15% deep 2X modulation. (Obviously, the RMS presentation of this case gives only the mean values, the modulation view is lost.)

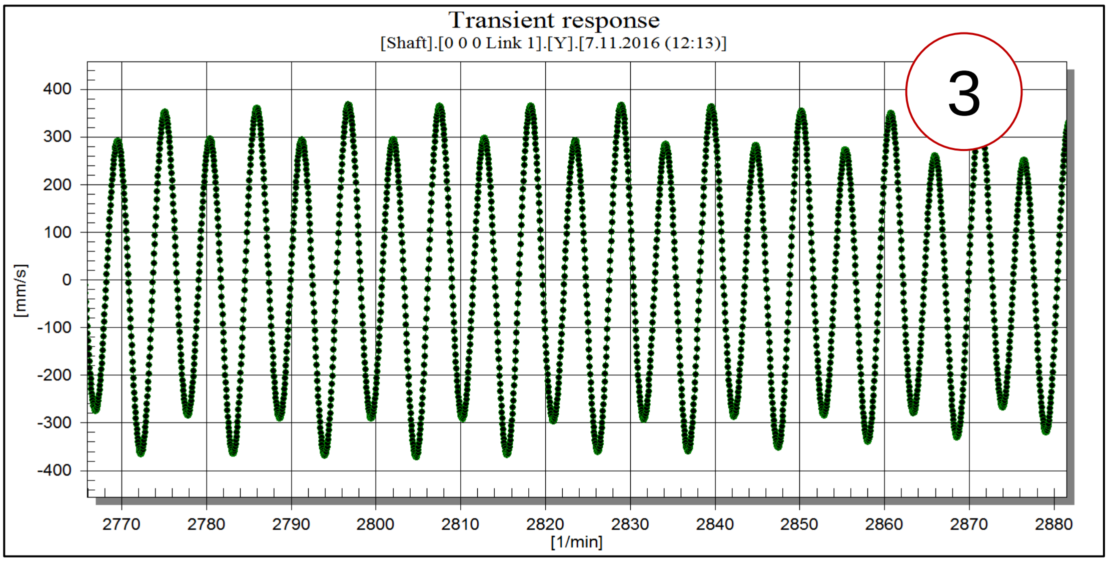


Figure 17.13 - Non-linear analysis results, 2700-2900rpm speed range.

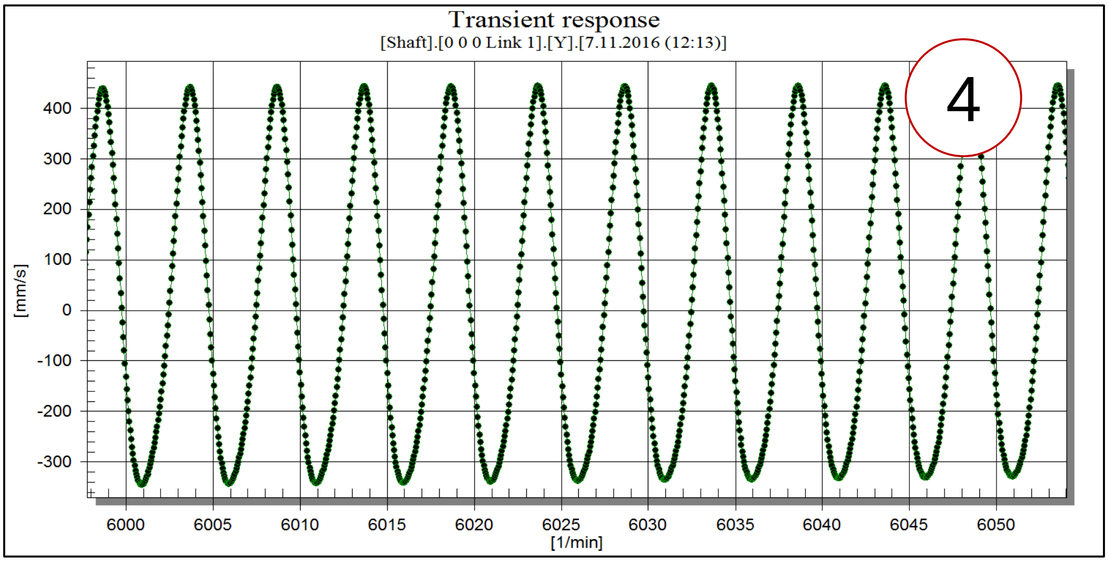


Figure 17.14 - Non-linear analysis results, 6000-6100rpm speed range.

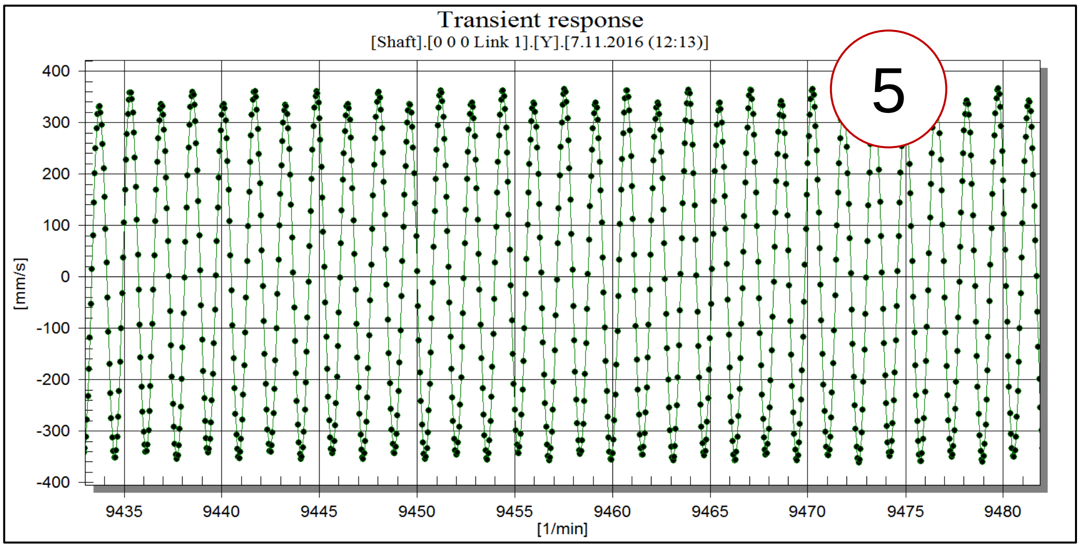


Figure 17.15 - Non-linear analysis results, 9400-9500rpm speed range.

The specific signal structure may be explained by transient modulation of the dominating 1X signal with the 2X non-synchronous component. It is worth mentioning that in many cases this may be a valuable diagnostic sign, for example in multi-rotor structures. The RMS presentation (Figure 17.16) and the waterfall diagram (Figure 17.17) reflect the general system status but don't show the internal interaction details.

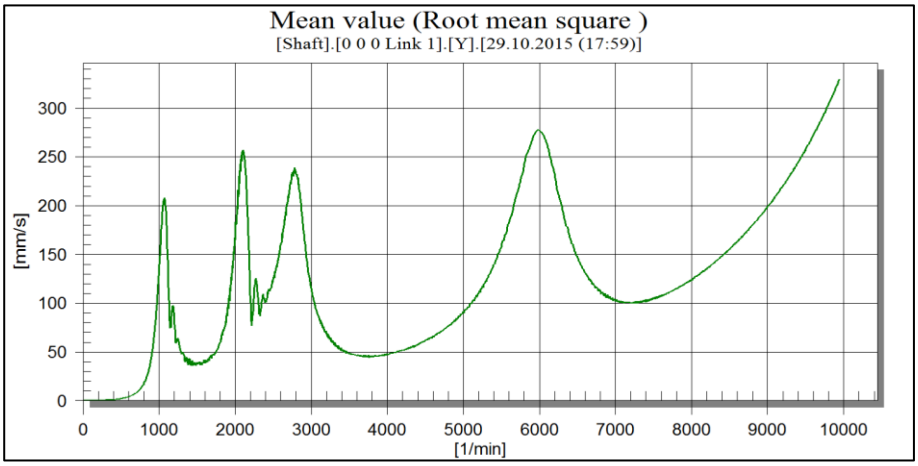


Figure 17.16 - RMS values of the [Non-linear analysis] calculation

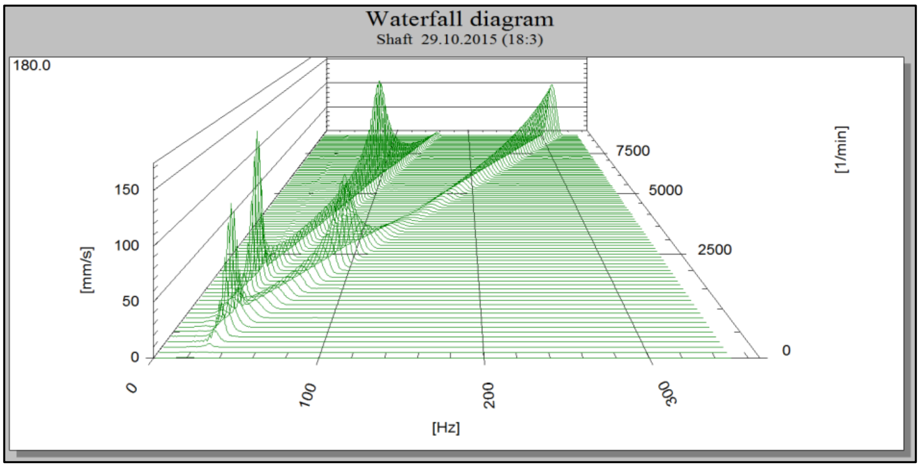


Figure 17.17 - Waterfall diagram

17.4 Results comparison

Figure 17.18 shows the both calculation results, linear and non-linear. One may conclude that the linear analysis with combined calculation results shows all resonance frequencies which is confirmed by the non-linear analysis. The magnitude differences are caused by specific features of the non-linear analysis.

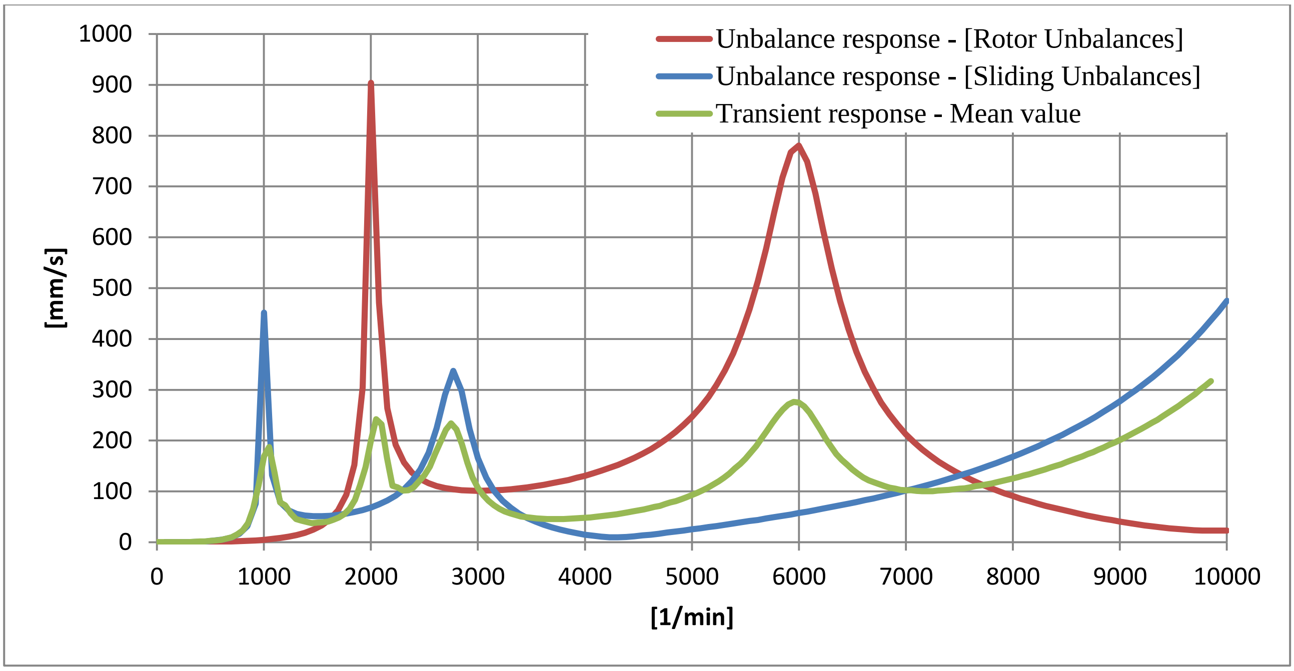


Figure 17.18 – Comparison of the linear [Unbalance response] and non-linear [Transient response] analysis

18. Application of command simulation interface, to automatic data table input for a model creation

User manual, clause 20 "Simulation command interface": *Command interface extends user's capabilities in models creation and parametric studies. It allows writing converters from simulation formats, or external codes formats, to the Dynamics R4 format. Table form data may be input automatically. Also a user may create so-called "calculators", or small codes for calculation of elements' performance.*

Converters. *Sometimes the input data for model creation are presented in table forms. The data may be data of external codes, or drawing dimensions, or results of bearing stiffness related to its operating mode. Manual input of large data tables is labor consuming and may cause input errors. The Simulation Command Interface helps this work.*

The example shows creation of a rotor model and a script tuning. The model parameters are input in the form of Microsoft Office Excel table. Table 18.1 shows an example of rotor elements parameters.

All parameters are described in the first column "A". Here one box, not one line, corresponds to one element. The element parameters are split with blanks.

Similar parameters in different elements may have different meanings. In the example parameters #4 and #5 in the element [Beam] describe inner and outer diameters, in the [Shell] element they reflect the shell mean diameter and wall thickness.

The table is saved in the **.csv** format.

A1 fx station type L D/Dmid d/b E Rho Nue

	A	B	C
1	station type L D/Dmid d/b E Rho Nue Comment		
2	1 beam 100 80 0 2.1e11 7850 0.3		
3	2 beam 100 80 0 2.1e11 7850 0.3 brng		
4	3 beam 100 100 50 2.1e11 7850 0.3		
5	4 beam 100 100 50 2.1e11 7850 0.3		
6	5 beam 100 100 50 2.1e11 7850 0.3		
7	6 beam 100 100 50 2.1e11 7850 0.3		
8	7 beam 100 100 50 2.1e11 7850 0.3		
9	8 beam 100 80 0 2.1e11 7850 0.3		
10	9 beam 100 80 0 2.1e11 7850 0.3 brng		
11	10 shell 100 96 2 2.1e11 7850 0.3		
12			
13			

Fig. 18.1 – Rotor parameters in Microsoft Office Excel

Dynamics R4 cannot operate directly with the **.csv** tables, but it operates Python language scripts that may use the table data. The simulation script interface is described in details in the special note (18.3). Below is shown code of the script tuned for operating the mentioned table.

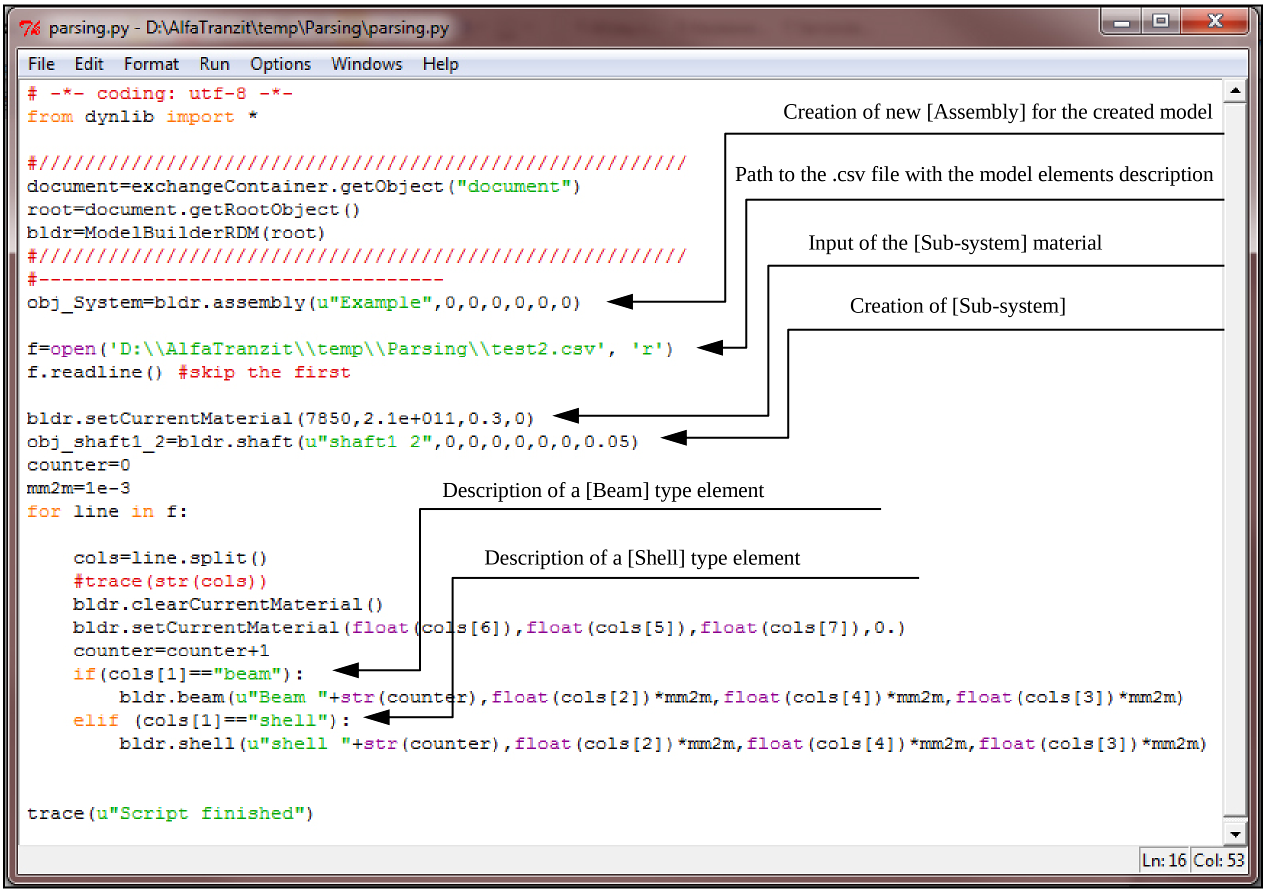


Fig. 18.2

This code includes creation of [Beam] and [Shell] elements but Dynamics R4 functionality allows creation with scripts of all system elements. Additional creation of an element needs two lines and is similar to the [Shell] creation but with the parameters format described in the Note.

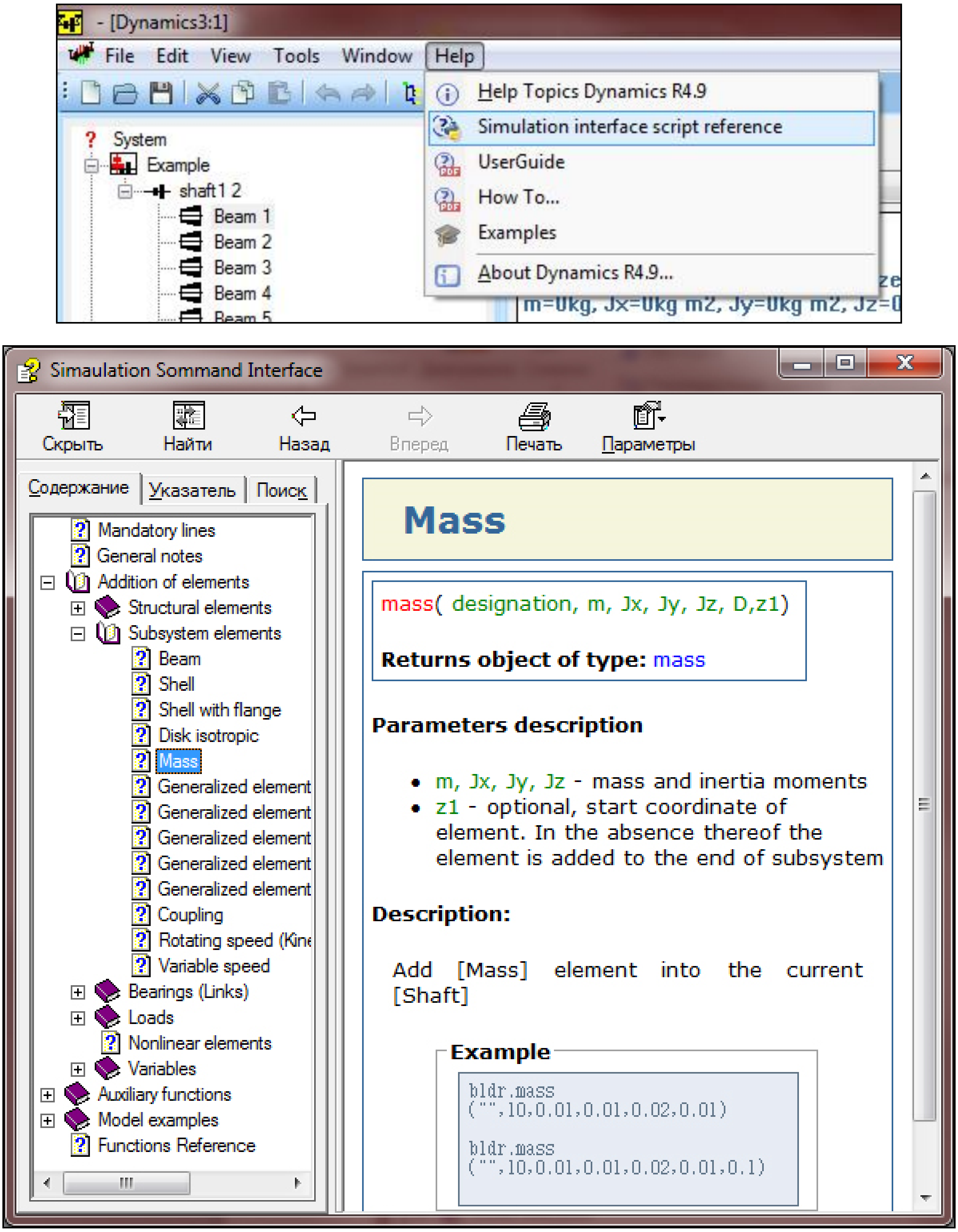


Fig. 18.3 – Call of a reference on the simulation command interface

After the .csv table with the model parameters is tuned and the Python script with descriptions of all elements in use and a reference to the .csv table in use it is necessary to call in Dynamics R4 running this script (18.4 - 18.5).

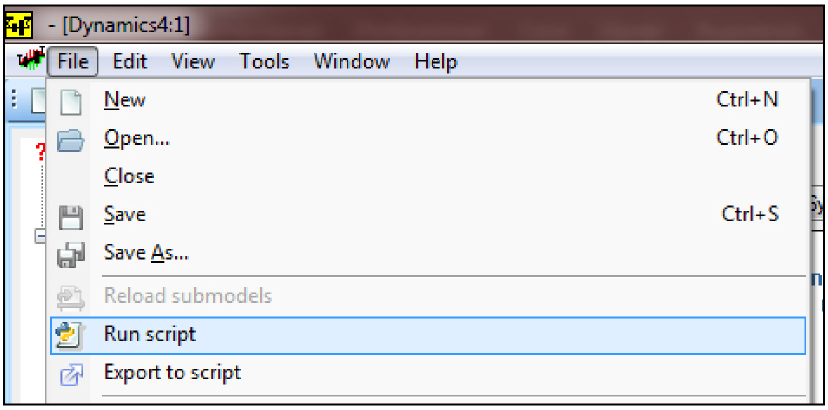


Fig. 18.4 – Command “Run script” in the code menu

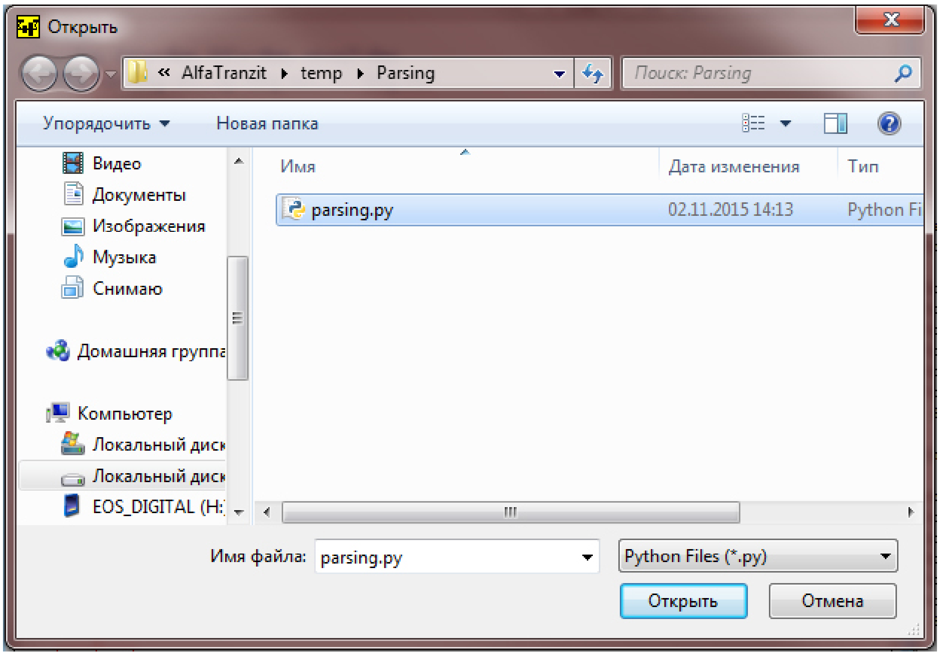


Fig. 18.5 – Script selection

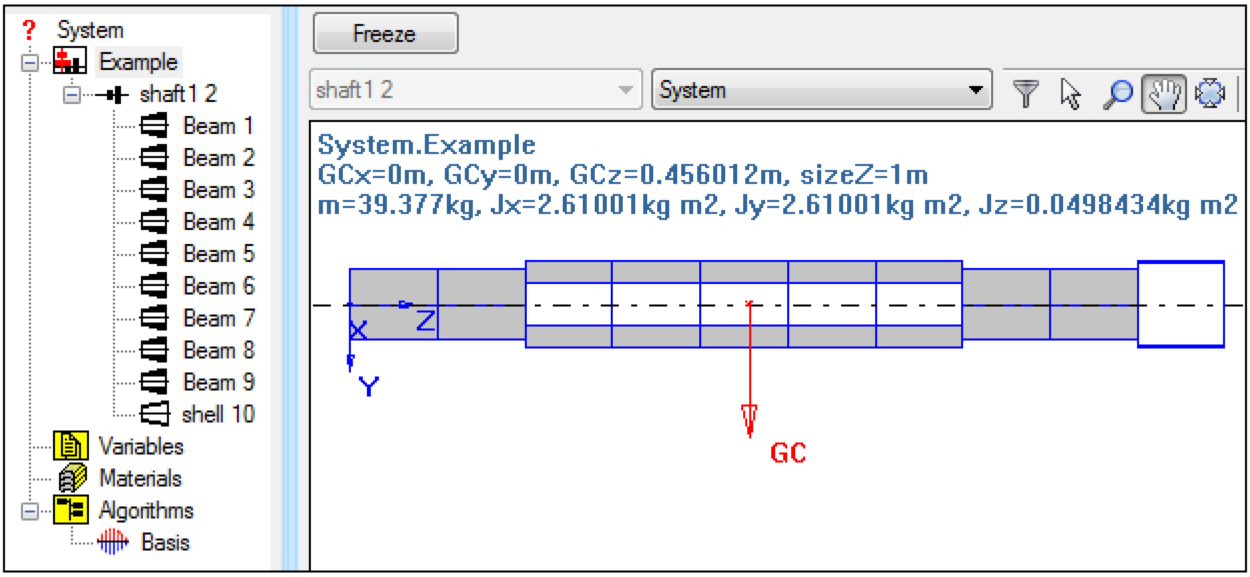


Fig. 18.6 – Script operation result

19. Modeling of a rotor with a bearing misalignment

The displacement of the bearing housing in the three-support rotor leads to a change in the line of static deflection and the redistribution of reactions in the support of the rotor. Unlike rotor misalignment, such a displacement does not lead to a change in the vibrations of the linear rotor system, but can affect the dynamics of the nonlinear system. This can occur if the stiffness of the supports depends on the load. For example, in hydrodynamic dampers in the gap, the rotor eccentricity can change, which will lead to a change in the damping in the support.

The modeling procedure is presented on the example of non-stationary analysis of a linear system.

Figure 19.1 shows a rotor with three bearings. Support nodes are modeled by isotropic linear elements of “Link” type. The stiffness of the support nodes is presented in the table 19.1

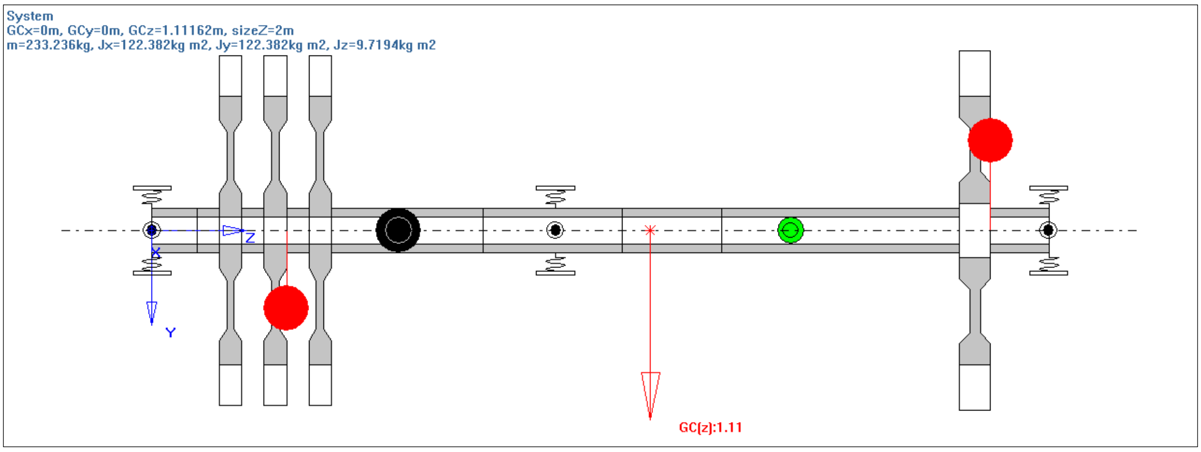


Figure 19.1 Rotor system model

Table 19.1

Stiffness matrix coefficient	Front support	Middle support	Rear support
	N/m	N/m	N/m
K_{XX}	1e+007	1e+008	1e+007
K_{YY}	1e+007	1e+008	1e+007

The static deflection line of the rotor under the action of gravity is shown in Figure 19.2. The basis used in the calculation is limited by the frequency R_freq = 30000 1/min.

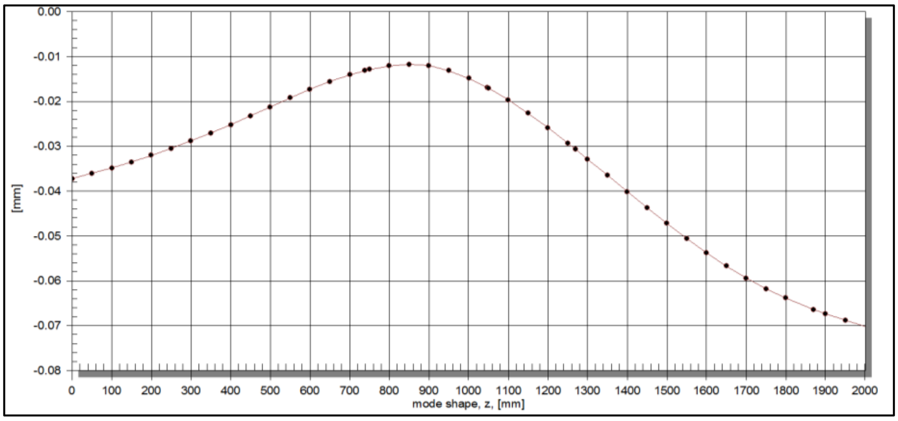


Figure 19.2 The static deflection line of the rotor

The calculated reactions in the supports are shown in the table 19.2

Table 19.2

Support number	1	2	3	Sum
Support reaction, N	372.951	1202.63	701.635	2277.216

It should be noted that the total mass of the rotor is 233.236 kg, which corresponds to the total weight: $F_W = 233.236 \cdot 9.81 = 2288.0452$ N. Thus, it is possible to determine the error in calculating the total reaction in the supports as $\varepsilon = \left(1 - \frac{R_W}{F_B}\right) \cdot 100\%$, where R_W - The total reaction in the supports calculated in DYNAMICS R4. In this case, the calculation error R_W is $\varepsilon = \left(1 - \frac{2277.216}{2288.0452}\right) \cdot 100\% = 0.473\%$.

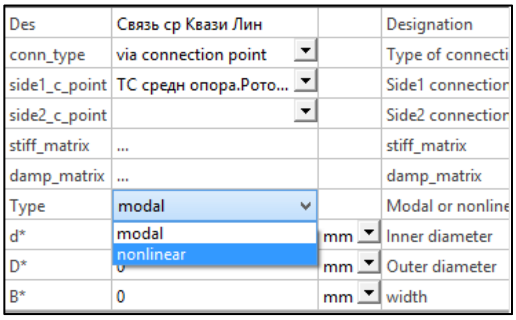


Figure 19.3

<u>The order of modeling the displacement of the support:</u>

a) It is necessary to create a quasilinear element «Elastic nonsymmetric link» with the same stiffness and damping parameters as the existing link and replace the existing linear link with it, the latter can be deleted or disabled in the group.

b) The created quasilinear link must be switched to the mode «nonlinear» see Figure 19.3, which makes it possible to set the misalignment of the housing. At this stage, it makes sense to calculate the reactions in the supports in the task of calculating the static deformation under the influence of the force of weight. The results obtained should coincide with a similar calculation with a linear relationship.

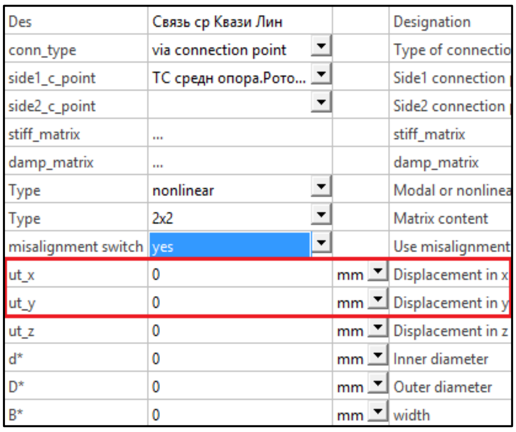


Figure 19.4

c) To enable the ability to set the misalignment of the support, it is necessary to switch the "misalignment switch" to "yes". This action activates the ut_x and ut_y fields. (see.Figure 19.4), which allow you to set the offset of the support, respectively, in the XY plane.

After replacing a linear link with a quasilinear one, the reactions are calculated under the influence of the force of weight. The result of the calculation of the reaction for the average support in the case of the use of quasilinear coupling is 1220.67 N, which is 18 N more than in the case of linear communication.

The offset of the middle support is set to 50 μm in the –Y direction. The results of calculating the elastic line of rotor deformation under the action of gravity are presented in Figure 19.5, other model settings are not changed.

IMPORTANT!!! Provided that the first connection node is connected to the rotor, and the second to the housing, the offsets ut_x and ut_y are set with the opposite sign.

It can be seen from the results that the elastic line of the rotor has changed: an additional displacement of 50 μm appeared in the middle support.

The calculated support reactions are presented in table 19.3. The support reaction in the displaced support was expectedly reduced, and the reactions in the other supports were redistributed accordingly. In this case, the total reaction was 2286.703 N, that is, the error in calculating the reaction decreased to$\varepsilon = 0.06\%$. Error reduction is associated with the use of quasilinear link.

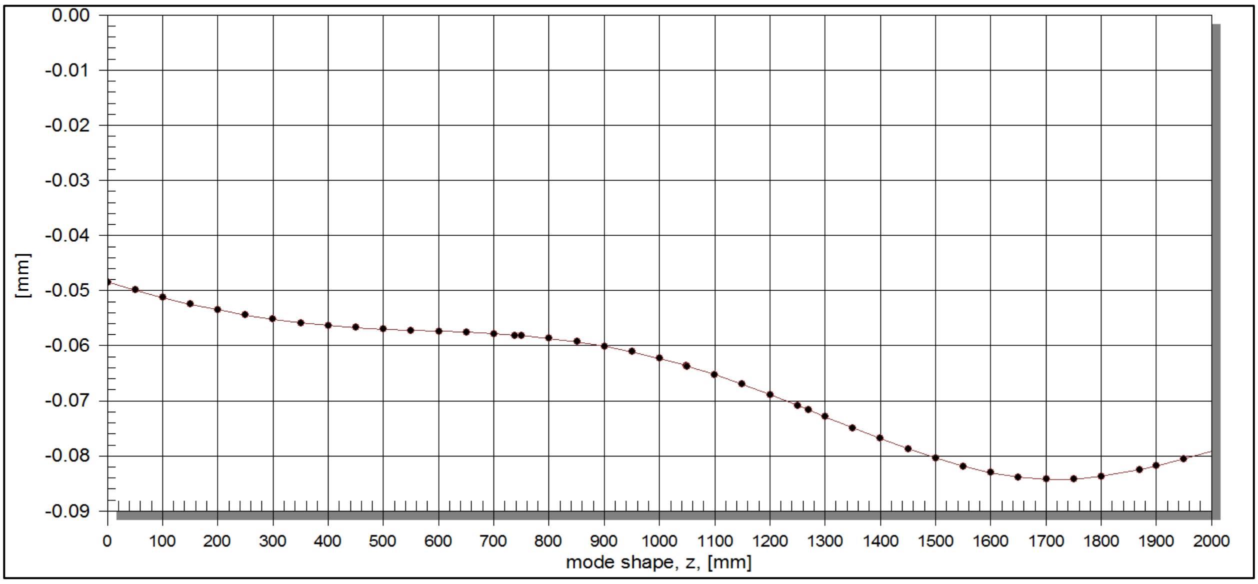


Figure 19.5 The static deflection line of the rotor under the action of a gravity load with a misalignment of the support by 50 μm

Table 19.3

Support number	1	2	3	Sum
Support reaction, N	484.566	1011.01	791.127	2286.703

To assess the effect of the displacement of the rotor support on the dynamics of the rotor system, the RMS of the rotor vibration velocity in the cross section of the second support is calculated while run up from 0 to 10,000 rpm in 5 seconds. The RMS calculation time intervals were 0.02 sec. External loads are weight force and unbalanced force from rotor imbalance. The results for centered supports are shown in Figure 19.6, the results for the case of the middle support misalignment by 50 μm in the –Y direction are presented in Figure 19.7.

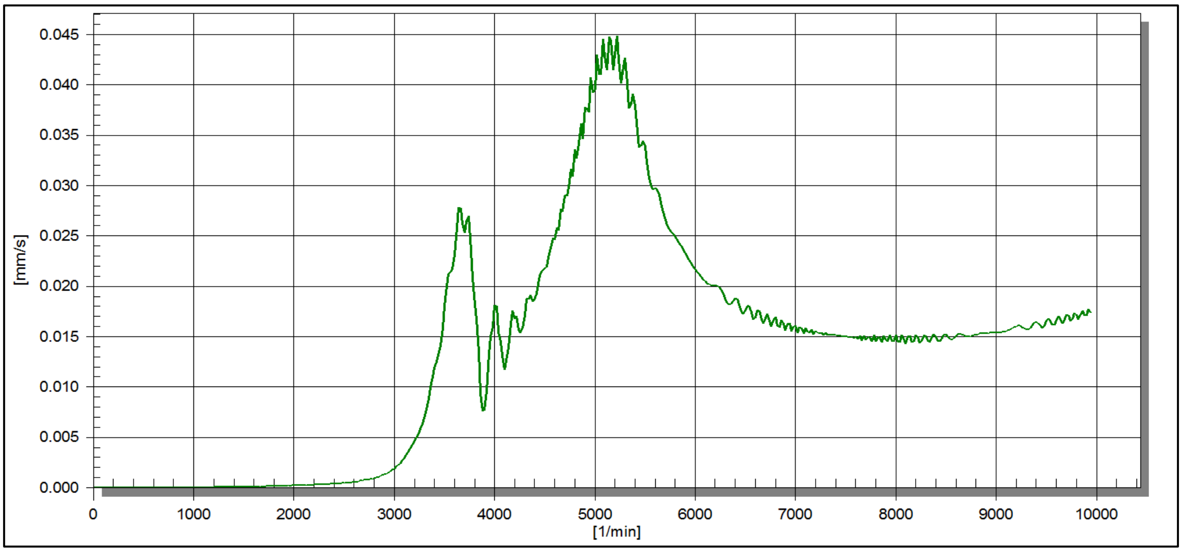


Figure 19.6 – RMS of the rotor vibration velocity in the cross section of the support 2 in the Y direction, support offset 0 µm

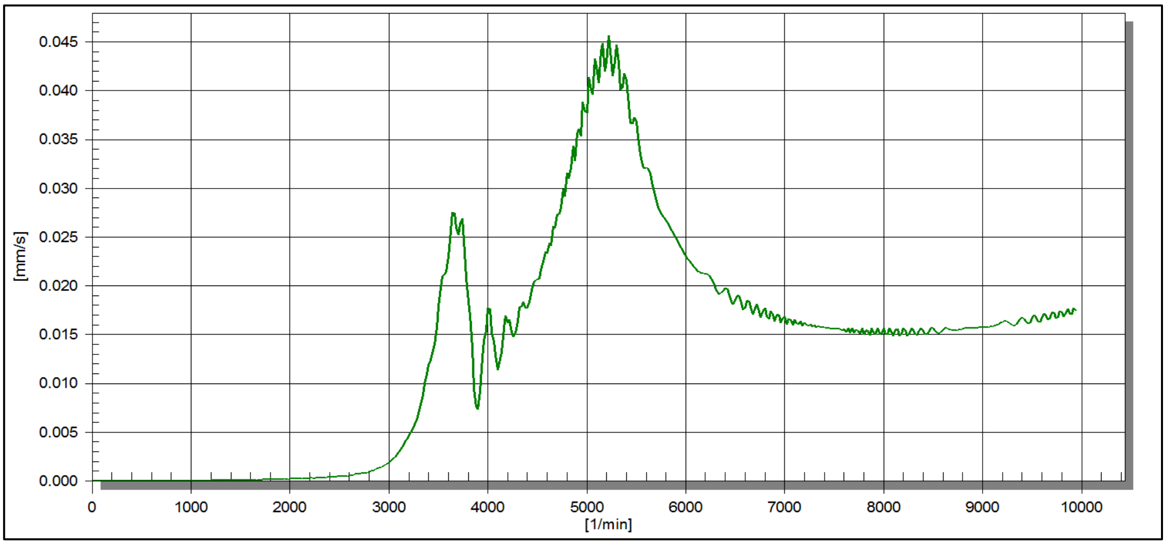


Figure 19.7 – RMS of rotor vibration velocity in the cross section of the support 2 in the Y direction, support offset 50 µm

From the results of Figure 19.6, Figure 19.7 one can conclude that the shift of the average support by 50 microns did not affect the dynamics of the rotor system. Due to the fact that due to the displacement of the support, the stiffness characteristics of the rotor system have not changed, the inertial characteristics also remained the same.

In a real rotating system, a change in bearing load affects the stiffness and damping of bearings. To a lesser extent, the effect is observed in rolling bearings, to a greater extent in sliding bearings. In plain bearings, the limit of stability loss can also change. In general, an increase in load leads to an increase in the stiffness of the support. In hydrodynamic dampers, the eccentricity of the orbit of the rotor precession in the gap is of importance. In this case, a decrease in the eccentricity in the gap can lead to the reduction of damping and an increase in vibrations. An increase in eccentricity can lead to a sharp increase in damping and freezing of the support (the damper will turn into an absolutely rigid support), which will lead to a change in the dynamic system of the rotor and a shift in the resonances of the system.

Alfa-Tranzit Co. Ltd.
support@alfatran.com
www.alfatran.com