Program system for analysis of rotor dynamics of turbomachines

DYNAMICS R4

User Guide

2024

Alfa-Tranzit Co., Ltd

Thank you for purchasing Dynamics R4 program, developed by our Alfa-Tranzit Co., Ltd for solution of many practical problems you can meet in rotordynamics research

Alfa-Tranzit Co., Ltd and its engineers do their best that Dynamics R4 would be the best among other programs for rotordynamics turbomachinery and meet your requirements.

We are ready to respond any proposals and consider them in further versions of the system.

We are always ready to help you!

Web-site: www.alfatran.com
support@alfatran.com

CONTENTS

1 INTRODUCTION

DYNAMICS R4 is a new generation of software package for turbomachinery rotordynamics problems.
This program was developed by people employed at the Moscow Aviation Institute (MAI) rotordynamics laboratory for a long time – since 1963 till present time.

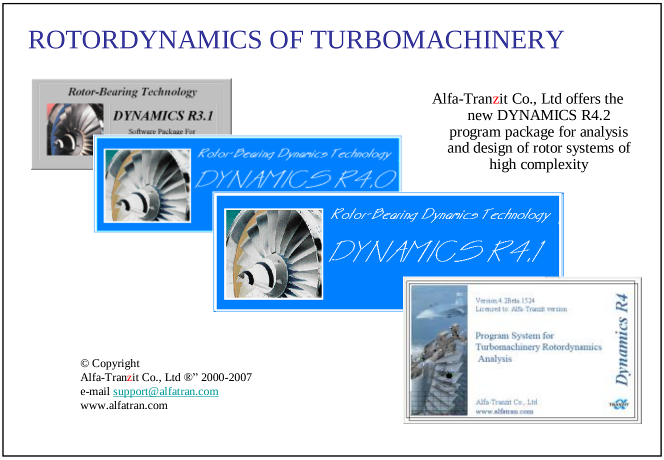
The first version of program software was developed in 1969 for axial symmetric models of gas-turbine engines and rotor-bearing structures (Chronin D.V, professor of MAI, Ivanov A.V., PhD, lecturer of MAI). The programming language is a machine-language code.
The second version of program to solve complete rotordynamics problems was developed in 1974 for anisotropic systems of gas-turbine engines (Chronin D.V, Ivanov A.V., Leontiev M.K., PhD, professor of MAI, member of SAE, academician). Programming language is ALGOL. Operating system is OS (IBM 360). Later this program was developed for transient response of nonlinear rotor-bearing structures (Chronin D.V, Leontiev M.K.) The program was provided for the design, development, fabrication and application of rotating systems and during subsequent 20 years it was the basic code in practice of the most design offices in USSR. The programming language is Fortran-4.

In the early 90's **the second version of program** (linear statement) was reprogrammed for new generation of computers (PC) and called **DYNAMICS** (Leontiev M.K.). The programming language is DOS-Fortran. The operating system is DOS.

The coding of **DYNAMICS R3.0**® software package was sponsored by Samsung® Aerospace (Republic of Korea) and developed by Alfa-Tranzit® Co., Ltd (Russia) - Leontiev M.K., Ivanov A.V and Degtiarev A.A, PhD, engineer. The programming language is C++ for operating systems Windows 95/98/2000/NT.

The widely known in Russia **DYNAMICS R3.1**. ® was programmed in 2001 and it has been used by many Russian companies.

DYNAMICS R4 ® is a new software product from Alfa-Tranzit Co., Ltd. The software(its various versions) may be applied to solution of a wide range of linear and non-linear multi-shaft rotordynamics problems. Its objective is to predict accurately the vibration characteristics and dynamic response of rotor structures in high speed rotating machinery. This analysis can be conducted during both the design phase and the machine operating when the causes of a particular failure or the reasons for poor performance are met.



The objects of research in **DYNAMICS R4**® are gas-turbine engines, power plants, air compressors, starters, turbo-expanders, turbo-driven pump assemblies, any kinds of gear systems, etc. The software may be used at different stages of design, development and operating.

DYNAMICS R4 is the software package specifically developed for design, analysis and troubleshooting of many kinds of rotating machinery. It can be also used for development of model based diagnostic algorithms.

1.1 General Properties

- Variety of practical rotor dynamic problems of rotating machinery that can be solved in Dynamics R4
- Analysis of linear and non-linear rotating structures
- High accuracy and speed of computations
- Adaptive methods of numerical integration in transient analysis
- Modular architecture of program system that allows leading its development and improvement
- Possibility of user’s algorithms and elements development and their integration into software
- Advanced system of information including help functions, warnings and error messages
- 49 examples of models and solutions that show program functionalities including results of calculations, exercises and instructions on program work
- Possibility to use the software for deep study of linear and non-linear rotor systems dynamics
- A user-friendly Russian- and English - s language interface

1.2 Functional Capacity

- Modeling of multi-shaft and multi-level rotor structures including cases, mountings and foundation
- 3-dimensional location of subsystems (free orientation of spin axes) - coaxial, or nonaligned, or crossing
- One-dimensional and two-dimensional parametric analysis – time variation of speed, geometry, location, stiffness and damping coefficients, loads, etc.
- Numerous stationary and non-stationary external loads
- Large quantity of various types of rotor nodes
- Any bar-and-pic charts of operating modes of rotating machines
- Modeling and analysis of any kinds of gears used in rotating machinery

1.3 General Problems of Linear Dynamics

- Calculation of natural frequencies and mode shapes of non-rotating and undamped systems
- Damped natural frequencies and mode shapes of rotating systems
- Natural frequency maps

- Stability maps
- Kinetic and potential energy distribution on a rotor model element for every mode shape
- Critical speeds calculation
- Critical speed maps
- Unbalance response
- Computation of rotor systems with time depended stiffness and damping matrixes
- Static deformations and reactions in supports

1.4 General Problems of Non-linear Dynamics

- Transient response due to acceleration and deceleration of rotor system
- Transient response of rotating structures due to various non-stationary loads
- Modeling of rotor structures with nonlinear supports and seals of general kind
- Calculation of systems with clearances and rubbings
- Computation of rotor systems with journal bearings
- Modeling and analysis of rotor systems supported by rolling bearings
- Computation of rotor systems with squeeze-film dampers
- Stability thresholds computation

1.5 User interface

- Text and graphic preprocessor for multi-level rotor structures creation
- Standard elements library for rotor system modeling
- Material database support
- 2D and 3D visualization of rotor models
- Exchangeable or removed groups of modeling elements or data for alternative calculations within a single dynamic model
- 2D and 3D results output
- Numerous types of output data – displacements, rotations, velocities, accelerations, reactions, forces and moments
- Continuous animation
- Graphical post-processor for output data after the transient analysis
- User-defined forms of protocols and reports based on the model data and calculation results

NOTE: The program is being constantly developed. New functions, elements and algorithms are being designed. In this connection this version of **User's Guide** may slightly differ from the program details

NOTE: Some algorithms described in this Guide may be absent in the supplied code

2 DEVELOPMENT OF SOFTWARE VERSIONS

With releasing new versions, users are sent notifications about possibility to download a new version. Notifications give the complete list of new functionality and corrected problems. Also, a user can look through changes list on our web-site: http://www.alfatran.com/dynamics4_new.shtml.

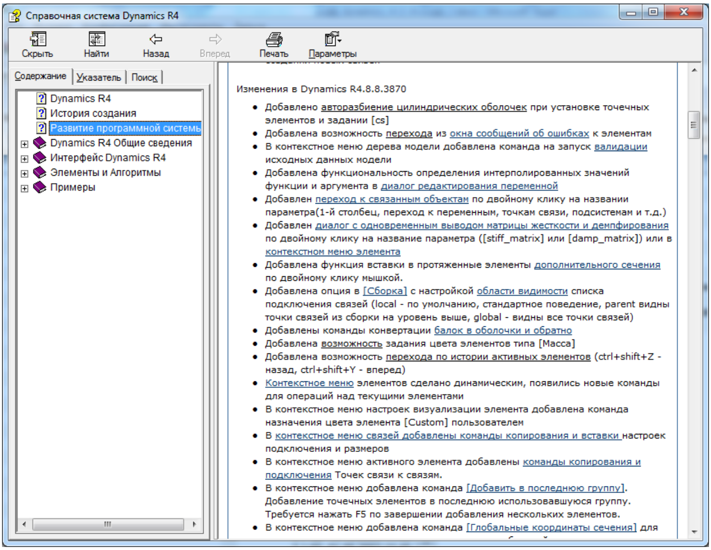


Figure 2.1

A user can study the history of changes in Help System (**Figure 2.1**). It worth noticing that a user can find additional information on the changes following hyperlink. "Help system change log" is link to the list of new materials in Help (**Figure 2.2**).

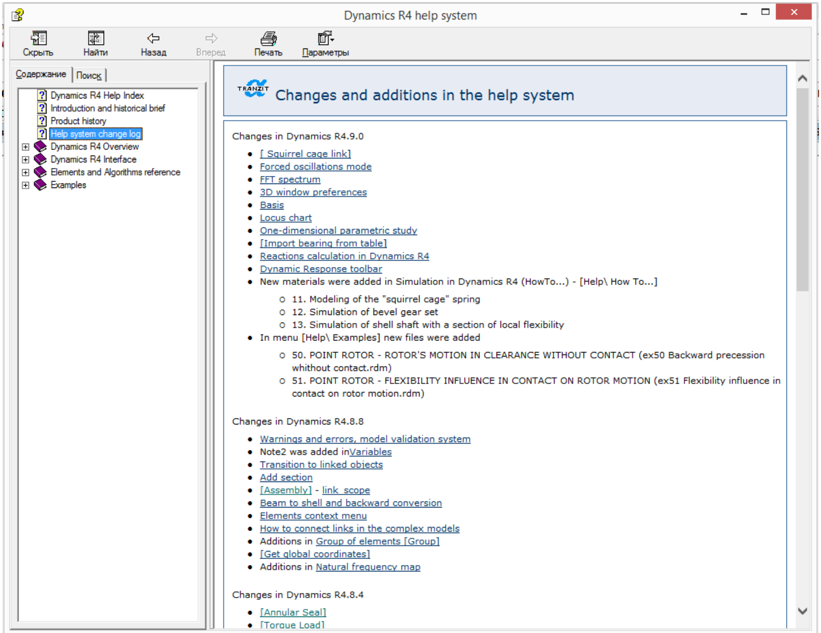


Figure 2.2

3 DYNAMICS R4 INSTALLATION NOTES

The program can be installed in the following way:

Insert the installation CD into a drive.

Run the file **setup_x86.exe** for 32-bit Windows and **setup_x64**.exe for 64-bit ones. You should not start the file Dynamics4.msi on the computer where Dynamics R4 was not installed before.

After that follow the instructions appearing on the screen while installing. When installation is finished in the directory chosen by a user, a new group Dynamics R4 will be created with an icon on the Desktop

The program can work on your computer only if there is electronic protection dongle on your computer.
To continue connect the security device (a dongle) to your computer through USB port. Windows will automatically install new hardware after you connect it.

After installing the program, the user can observe some installation information by [**About Dynamics R4]** command, that is, **Figure 3.1**:

- program version number (please, send this number when sending messages to the Licenser)
- software packages which were used in Dynamics R4
- license data
- license text

- technical support information
- information about disabled options of Dynamics R4 version for Licensee

To uninstall **Dynamics R4**, use the Windows [**Add or Remove…]** applet.

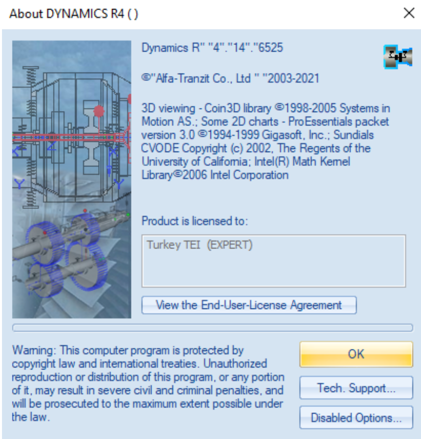


Figure 3.1

3.1 Dynamics R4 (floating license)

Server - computer with installed Guardant Net and Guardant Net Server software.

Client – workstation with installed Dynamics R4. (server can be started on the client also)

1. Settings on the server side

1.1 Setup drivers for Guardant dongles. (Can be downloaded also from - http://www.guardant.ru/support/download/drivers/)

1.2 Copy in any folder (specified by system administrator) Guardant Net Server software. (grdsrv.exe and grdsrv.ini)

1.3 Guardant Net Server includes next files:

- **GrdSrv.exe** — server Guardant Net (server side);

- **GrdSrv.ini** — file with Guardant Net Server settings. Should be placed in the same directory with GrdSrv.exe (server side);
- **GnClient.ini** — file with Guardant Net client settings. Should be placed in the same directory with Dynamics.exe (client side);
- **GrdMon.exe** — utility for monitoring of the current Guardant Net server state. Can be used for checking of availability of the Guardant Net Server and number of available licenses of Dynamics R4 (can be used only for monitoring of the server launched in the same LAN, client side);

1.4 **Server should be accessible from client by TCP/IP protocol (on default TCP 3182, UDP 3183, UDP 3184; for http access 3185)**

1.5 Run Guardant server software (**GrdSrv.exe**)

For working of GrdSrv.exe as service it should be launched in console (cmd.exe) with options **GrdSrv.exe /s [q]**

2. Client side settings

2.1 Setup Dynamics R4 (run setup_x86.exe or setup_x64.exe)

2) Edit in gnclient.ini (in any text editor) in the installation folder

For example: " c:\Program Files \Alfatran\DynamicsR4\Bin\ gnclient.ini"

```
BC_ADDR=255.255.255.0
IP_NAME=192.168.0.144
```

In IP_NAME server IP address should be specified.

3) Give write permissions to Bin folder in order to run Dynamics R4.

4 STARTING DYNAMICS R4

4.1 Run in graphical mode

The startup **DYNAMICS R4** window is shown in **Figure 4.1.**

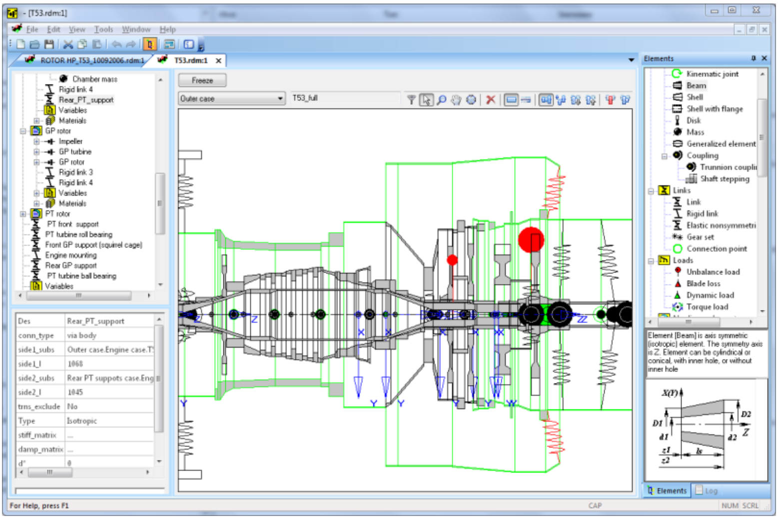


Figure 4.1

The startup window consists of several input and output data areas:

- The constant central window provides the modeling process, model displacement and outputs the calculation results.
- The right window includes tabs **[Elements**] and **[Log]** and may be opened or closed by the user. These tabs may be placed in any part of the screen.

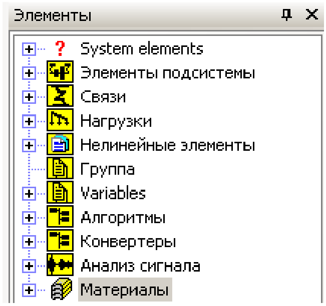


Figure 4.2

The **[Elements**] tab includes libraries of elements, links, algorithms for model analysis and materials. The tab bottom field displays current information about the element, or algorithm choice, Figure 4.2**.**

The **[Log]** tab may be opened instead of the **[Elements]. [Log]** displays current reports of the code in text format. The reports consist of calculation proceeding, error warnings, fulfilling of the orthogonality conditions, etc.

- The left top window includes the model data, set of subsystems and elements, materials and the calculation algorithm menu.
- The left bottom window provides input, editing and imaging of subsystem parameters and algorithm tuning.

NOTE: A user may change the areas sizes or their location with the cursor using the **"Drag & Drop"** technology. Text or graphic information may be changed in a standard way.

The code displays the standard Windows command line and the code status line shown in Figure 4.3 .

Figure 4.3

To open or close the command line, click the command **[View]/ [Toolbar].**

The command **[File]** includes standard commands and the **[Reload subsystems].** This command refreshes the total system model after the submodels editing.

The **[Tools]** opens **[Options]** or **[Edit Materials DB]** commands**.**

The **[Options]** command **(Figure 4.4)** does preliminary setup the procedure options.

The options are:

- The startup opens the last project opened before the previous shutdown
- Matching the main window size at startup
- Matching the project window size at startup
- Number of files saved in the file list
- Switching between Russian and English interface language
- Sound notifications upon calculation completion. Different notifications for successful and unsuccessful calculation
- Autosave parameters settings (on/off, autosave timer - program restart is needed after parameter changing)
- Settings for running of x64 console solver for models with more than 700-800 stations (basis calculation)

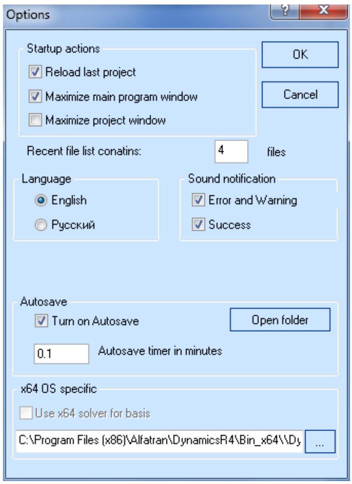


Figure 4.4

The following buttons are used in the log panel:

 - **[Tab Bar]** opens right window containing **[Elements**] or **[Log]** tabs

 - creates and outputs the calculation result protocol

 - undoes the last action

 - redoes the previously undone action

 - opens a program window with information about default configuration

4.2 Run in command line mode

Dynamics R4 can be used in command line mode. To this end one has to execute the DynConsole.exe file in Windows command line, or x64/DynConsole.exe for the 64-bit version. DynConsole.exe has the following arguments: [model file name] [-option1] [-option2] ..., where [model file name] – a *.rdm project file name. If the program is run with no options specified, the entire set of the algorithms added previously by the user to the *.rdm file will be executed (see paragraph 6.1 for detailed information about the project structure). As the computation has completed, the dialog box of the output protocol data selection will start. This default behavior can be altered by the user providing additional options to DynConsole.exe. There is the list of the available options:

- [-script] – suppresses the protocol dialog box appearance after the calculation has completed.
- [-script [-result_file output_file_name]] – generates the protocol and saves it into “output_file_name” file (.txt extensions will be automatically appended) after the calculation has completed.
- [-script [-result_file output_file_name] [-result_file_extension output_file_extension]] - generates the protocol and saves it into “output_file_name.output_file_extension” file after the calculation has completed.
- [-run_script [script file name]] – run a Python script that uses the simulation command interface (see section 21) to automatically modify the model, and then start the model calculation.

To view DynConsole.exe help run DynConsole.exe [-? | -help].
To check the program version run DynConsole.exe [-V].

Examples:

```
DynConsole.exe –V
DynConsole.exe test_cfg.rdm
DynConsole.exe test_cfg.rdm –script
DynConsole.exe test_cfg.rdm -script -result_file test.txt
DynConsole.exe test_cfg.rdm -script -result_file testOutput -result_file_extension log
DynConsole.exe test_cfg.rdm -run_script test.py
DynConsole.exe test_cfg.rdm -force_save  -run_script test.py
```

5 LOADING AND THE MODEL DATA SAVING

The first step of a user must be selection of an existing project or creating of a new one:

1. An old project loading. Any old project is loaded with **[File] [Open]** command.

2. A new project creation. A new project is created with **[File] [New]** command (the active status of program after starting is the **Dynamics R4** project).

3. New or edited project data are saved by **[File] [Save]** or **[File] [Save as…]** commands at any moment including computation process.

NOTE: When the **[Reload last project]** option of the **[Options]** menu is active, the code loading automatically opens the latest project opened before the previous code shutdown

In order to load the model developed in the previous version of **Dynamics 3.1** program, it should be converted into **Dynamics R4** format. For this purpose the model should be saved in the corresponding converter, which is supplied to the **Dynamics R3.1** users.

6 PROJECT

6.1 Project data

A project is defined as a structure that includes information about the object under investigation. The project includes two main parts – initial data about the model and set of algorithms chosen by a user for investigations conducting. There is also an additional information – materials database and information for leading of variant calculations.

A project is presented as a hierarchy, or a tree consisting of structural elements, links between them, loads, material tables, variables, etc. An example tree is shown in **Figure 6.1**

The project tree opens when the project file ***.rdm** is loaded.

General data of the project may be opened by highlighting of the first line in the project menu. A user may change the default project name **[System]** to any other. When the command is done, the window appears in the left bottom part and shows general parameters and location of the project (**Figure 6.2)**. The default values of the global frame location are 0 (zero).

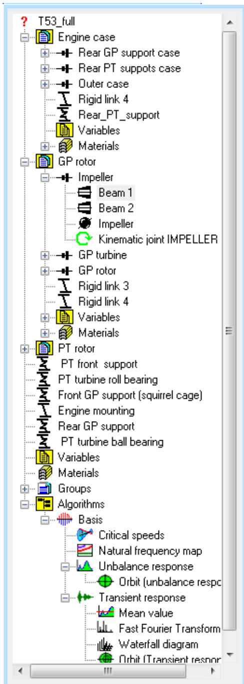


Figure 6.1

There are the project data:

- **[Des] -** designation of the project file displayed in the first line of the project tree
- **[def_units] -** default units for new model objects and output
- **[title]** - project title, can be used for short description of the dynamical model
- **[description]** – project description, can be used for complete description of the project
- **[name] –** name of the project developer
- **[modified]** - date of last modification
- **[x,y,z] -** system of translational coordinates
- **[eps_x, eps_y, eps_z]** - system of rotational coordinates

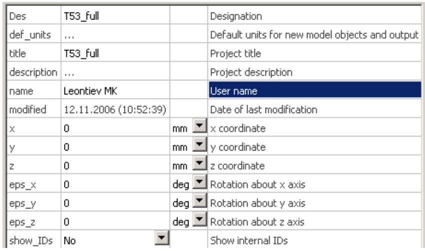


Figure 6.2

The default values of linear and angular location of the global co-ordinates are zero (0) turn and co-ordinate beginning.

> **NOTE:** All the project data are used for protocol preparation

6.2 Units

The system of units is determined by a user. It is possible to choose the International System of Units (SI system) or English System of Units. Also for SI system a user can change unit **m** into **mm**.
On default the new project uses the International System of Units (SI system). In accordance with this system some units are, **Table 6.1**:

Table 6.1

length (diameter)	mm	**pressure**	Pa
clearance	mm	**density**	kg/m^3
mass	kg	**angle**	rad (deg)

force	N	**frequency**	rpm (Hz)
moment	Nm	**magnitude**	mm
linear stiffness, etc	N/m, etc.	**velocity**	mm/sec
linear flexibility, etc	m/N, etc.	**acceleration**	mm/sec^2
damping	Nsec/m	**inertia moment**	kgm^2
static moment	kgm		

NOTE: Change of the unit system leads to automatic recalculation of values corresponding to change in parameters

6.3 Warnings and errors

Dynamics R4 checks correctness of input and calculated data. If necessary, it provides a warning info that does not block calculations. Warning examples are given in **Figure 6.3** and **Figure 6.4** .

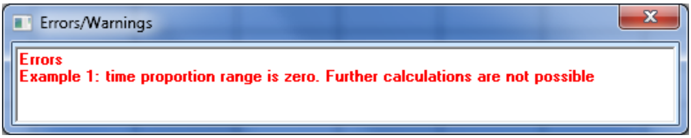


Figure 6.3

Errors prevent the following calculations.

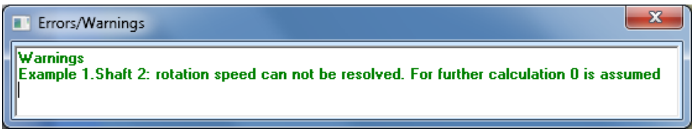


Figure 6.4

Errors are highlighted in red, warnings - in green.

In the regime of modeling (2D window is active) there is a possibility to move from errors given by the system of validation of the model parameters to the problem elements. In order to move, you should put the cursor on the line with the error and click on the [Go] button, **Figure 6.5.**

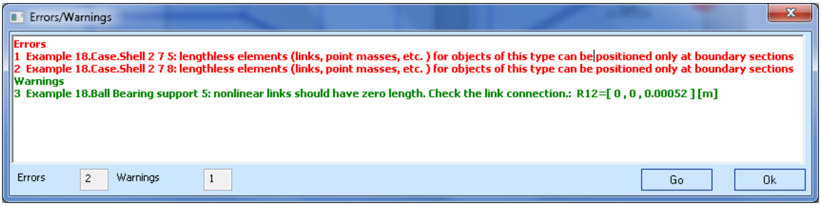


Figure 6.5

A user can also check a model using the ”Validation”command available in the context menu of the model tree or program menu [Tools]\[Validate],, **Figure 6.6.**

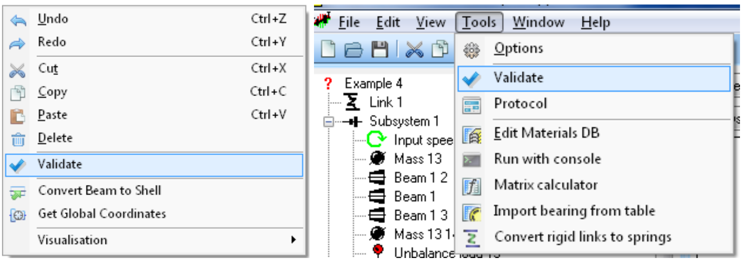


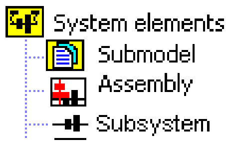
Figure 6.6

7 ARCHITECTURE OF DYNAMICAL SYSTEM IN DYNAMICS R4

7.1 Dynamical systems structural units

A dynamic system model may include the following structural units, **Figure 7.1**.

Figure 7.1



- Submodels **[Submodel],** Assemblies **[Assembly],** Subsystems **[Subsystem]** appear when building the dynamic system model in **Dynamics R4.**

Assemblies **[Assembly]** and subsystems **[Subsystem]** belong to the dynamic system model **[System]** formed in the Dynamics R4 environment.

Submodels **[Submodel]** are stored in external files and are loaded together with the model opening. A user may operate these structure units without loading the system model **[System].**

All structural units of a dynamic system are connected by links**.** Connection is done using **[Connection point]** or by direct definition of the links boundary sections.

Pictures below show possible assemblies of structure units within a dynamic system model.
A system **[System]** may include a **[Submodel],** an assembly, a subsystem and links, **Figure 7.2**.

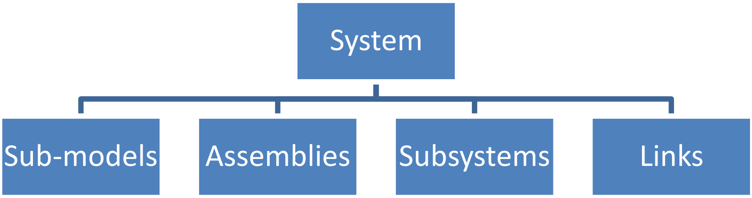


Figure 7.2

An assembly may include a **[Submodel],** a **[Subsystem],** other assemblies **[Assembly]** and links between them, **Figure 7.3.**

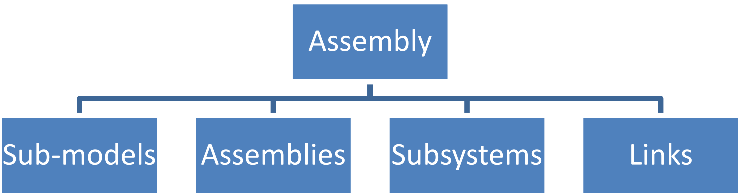


Figure 7.3

A **[Subsystem]** is built of linear elements; loads may be applied only to subsystems, **Figure 7.4.**

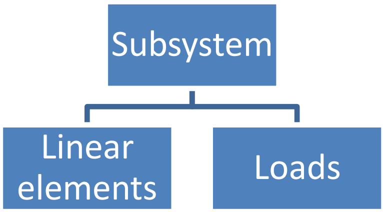


Figure 7.4

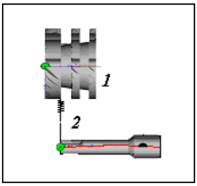
A system **[System]** with unique name is stored in special file.

NOTE: Any structural element may have unique name given by a user

7.2 [Assembly] element

The element serves for unification of several submodels, subsystems, etc. in the assembly which is a part of the whole model of the investigated object as an independent structural unit. The assembly may be edited only as a part of the whole model. The assembly is defined by its name and position of its local coordinate system in the global one belonging to the whole model, **Figure 7.5.**

Menu symbols	Input box



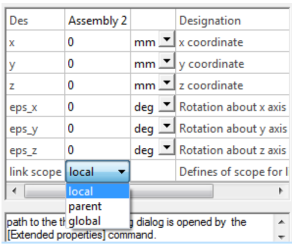


Figure 7.5

Several options define the visibility scope for connection points in the combo boxes in the links properties (linear and nonlinear). The choice is possible: **[local]** – by default, standard behavior like in the old versions. Only connection points of the current and child assemblies are visible. **[parent]** – in additionally points from the parent assembly are visible. **[global]** – all connection points are visible.

The order of the elements such as [Assembly] and [Submodel] may be changed in the model tree. For this the commands of the context menu “Move Up”, “Move Down” may be used, **Figure 7.6**.

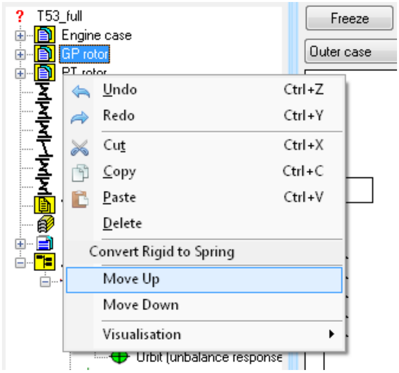


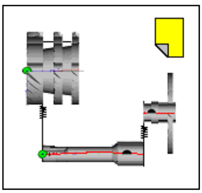
Figure 7.6

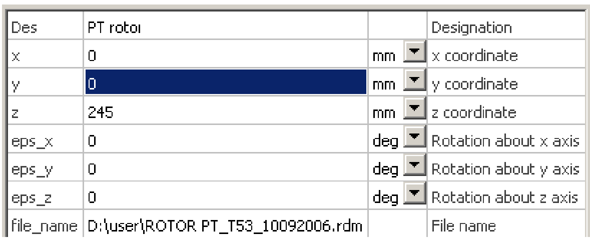
Change in sequence of structural elements in the model tree helps to change visualization at assemblies spacing.

7.3 [Submodel] element

The element **[Submodel]** is for composing of several assemblies and/or subsystems into a submodel. The submodel is assigned by its name and the local coordinate system defining its position in the global coordinate system of the whole model, **Figure 7.7.**

Menu symbols	Input box
Submodel	





Des	PT rotor		Designation
x	0	mm	x coordinate
y	0	mm	y coordinate
z	245	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
file_name	D:\user\ROTOR PT_T53_10092006.rdm		File name

Figure 7.7

When a submodel is added, its location in the file structure should be given. For this purpose:

- Highlight the submodel in the main project tree, **Figure 7.8**.

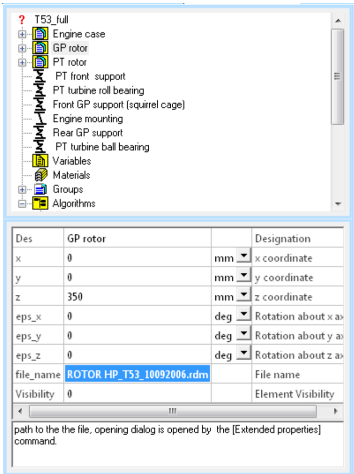


Figure 7.8

- Show the submodel path in the **[file name]**

The file opening dialog is opened by the **[Extended properties]** command.

To do it, select the input field with the cursor and press the right mouse button. Then find the required file with standard actions, **Figure 7.9.**

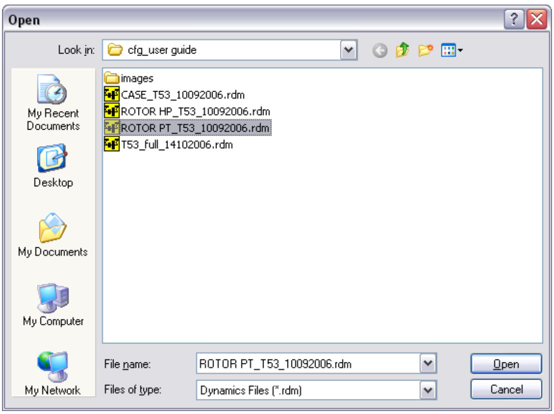


Figure 7.9

This procedure is to be repeated for all submodels. If a submodel path or the related configuration are changed the commands **[File]**, **[Reload submodels]** should be done. As the result the project model will be renewed by the re-edited models. The current submodel can also be renewed if choosing the command [Restart submodel] in the context menu of the element.

NOTE: If the submodel path consists only of a file name, the system attempts to open submodel in the catalog where the main model is placed

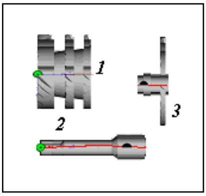
NOTE: Editing of sub-model in systems and assemblies is blocked

7.4 [Shaft] element

A subsystem or shaft is a structural element built of simpler elements - beams, shells, discs, etc. The location of a subsystem is determined by a set of linear and angular coordinates, **Figure 7.10**.

Menu symbols | **Input box**

Shaft



Des	Подсистема 1		Designation
x	0	mm	x координата
y	0	mm	y координата
z	40.0844	mm	z координата
eps_x	0	deg	Поворот подсистемы вокруг оси X
eps_y	0	deg	Поворот подсистемы вокруг оси Y
eps_z	0	deg	Поворот подсистемы вокруг оси Z
material	задается пользователем		Выбор материала
E	2.1e+011	N/m2	Модуль упругости
Nue	0.3		Коэффициент Пуассона
rho	7850	kg/m3	Плотность материала
Ln_dec	0		Логарифмический декремент
cs	1e+007	mm	Шаг разбиения

Figure 7.10

A subsystem material may be either defined by a user or chosen from the database. On default a subsystem material is given to all elements belonging to a subsystem.

The **[cs]** parameter is used by all the program algorithms. Its purpose is splitting of long cylindrical beams and cylindrical shells into short segments (beams/shells). This process improves accuracy of simulating of real beam structures due to fuller consideration of inertial parameters at discrete modeling.

Splitting of beams after specifying the **[cs]** value is carried out automatically. The **[cs]** value is assigned in mm and determines the length of parts which the long beams are divided into.

NOTE: Beam elements and cylindrical shells are automatically divided in points of links attachment or point elements setting

To add a subsystem, a user applying the **“drag and drop”** technology transfers element from elements library to a central window. The coordinate axes and **[Beam]** element is appeared in a central window.

8 SUBSYSTEMS ELEMENTS AND THEIR PARAMETERS

Subsystem is a part of a dynamical system. For example, for a twin-shaft gas turbine engine three subsystems can be determined – low pressure rotor, high pressure rotor and case with mounts.

Subsystems are connected with each other by links that describe characteristics of supports, structural members, a hanger, etc.

In the program library the subsystems elements are represented by the following strings, Figure 8.1.

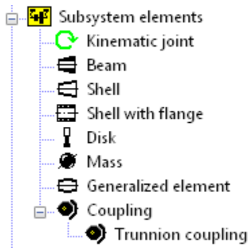


Figure 8.1

8.1 General properties of the elements and their parameters

Each element is assigned by the set of parameters that defines its location in the model or in the subsystem. Some of the parameters have the same names and meanings for all elements, **Table 8.1.**

Table 8.1

Parameter	Description
Des	On default the variable has the text value **[Name_Number]**. The Number indicates the sequential number of a standard element in a model. For example, **[Beam 12]**. A user can change the name of an element by any other
Material	The variable indicates the way to define the material properties: from data base; by direct input of data; one material is defined for all subsystem elements
segRef	The variable indicates the way to describe element: its position and geometry. In the first case it is possible to set length of the element by means of start and end coordinates **[z1]** and **[z2]**. The variables can be edited. In the second case a user can set the length of the element by means of the variable **[ls].** Variables **[z1]** and **[z2]** are not edited.
ls	Length of element. Variable **[ls]**can be edited when **[segRef]** is **[length]**
z1	Coordinate **[z1]** indicates the position of left side of element in direction Z.
1. **Z2**	Coordinate **[z2]** indicates the position of right side of element in direction Z.
Type	The variable defines different characteristics of elements, for example: • for beams and shells – cylindrical or orthotropic • for an abstract element – isotropic or orthotropic • for a link – isotropic or orthotropic, etc.
Conn_type	The variable specifies a type of connection of subsystems with each other - **[via body]** – the connection is defined without additional conditions. The point of connection is determined by the subsystem name and **z** coordinate; - **[via connection point]** – the connection point must be determined beforehand. The point is specified by the subsystem name
side1_subs	The variable indicates a subsystem to connect the boundary section 1 of the link. On default the subsystem is a foundation (zero subsystem)
side2_subs	The variable indicates a subsystem to connect the boundary section 2 of link. On default the subsystem is a foundation (zero subsystem)
UT_x	Displacement along **X** axis
UT_y	Displacement along **Y** axis
UT_z	Displacement along **Z** axis
UR_x	Rotation about **X**
UR_y	Rotation about **Y**
UR_z	Rotation about **Z**
2. **D***	The variable defines the inner size of a link in a model. The size value is used only for link visualization
D*	The variable defines the outer size of a link in a model. The size value is used only for link visualization
B*	The variable specifies the width of a link in a model. The size value is used only for a model visualization
t_st_type	The variable gives a way to define initiating time (start time) of the force: - **[real]** uses **[time parameter]** from element **[Kinematic joint]** - **[relative]** uses integration time given in **[Transient response]** algorithm
t_st_rel	Initiating time of a force (proportion start time)
t_st_real	Initiating time of a force (real start time)
E	Modulus of elasticity
Nue	Poisson's ratio
Rho	Material density
Ln_dec	Logarithmic decrement

> **NOTE:** If a link is inclined its stiffness coefficients are determined under assumptions of the first border section **sides 1** fixing and the second **sides 2** loading
>
> **NOTE:** Flexibility matrix in lengthy elements is obtained under conditions of restraint of the left boundary section 1 and loading of the right section 2

> **NOTE:** All dimensions of an element are given in a local system of coordinates

At double click by left mouse button on the area with the name of the parameter where the variable is attached, the window of characteristics of the linked variable will be activated

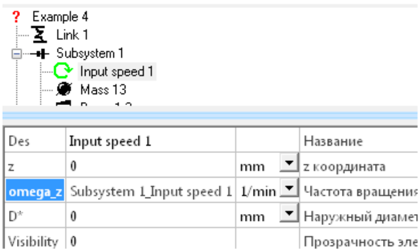


Figure 8.2

At double click on the name of the link attachment area, move to the link point or the closest element in the subsystem takes place.

Output of calculated parameters (mass-inertia, flexibility) for line elements ([Beam], [Shell], [Shell with flange]) is available in [El. results] dialog (Figure 8.3) since Dynamics R4.11. These parameters are also accessible via [Protocol] functionality.

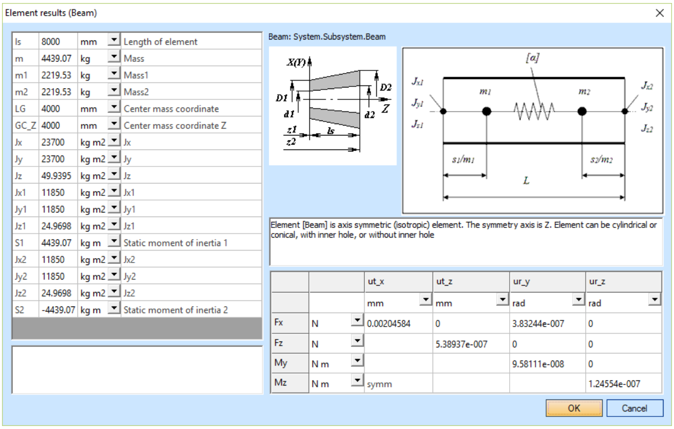


Figure 8.3

NOTE: You can switch to the previous active element by clicking ctrl+shift+z. To the next one - ctrl+shift+y.

Sometimes it is required to add an additional section in the model, not applying automatic subdividing of beam and shell elements. Or it may be required to divide elements which can't be divided automatically. For cylindrical elements it may be done before using the elements copy-paste function.

Now at double mouse click the dialog window (**Figure 8.4**) appears. There it is possible to edit the coordinate of the element insert either in the local coordinate system or in the global one. The section insert into the global coordinate system may be useful at setting the section for the link attachment at the regime [via body], when the coordinates are given in the global coordinate system of the model.

This functionality may be also used for approximate coordinates obtainment in the model.

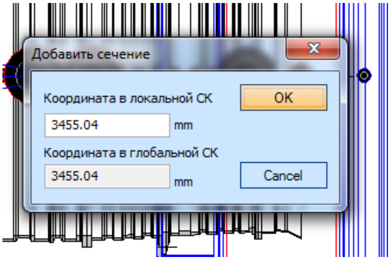


Figure 8.4

NOTE: Parameters not relating to the geometry of the lengthy element will be duplicated.

NOTE: Pressing of Ctrl+2 or Ctrl+4 for active parameter in element property table leads to appropriate multiplication. Simultaneous pressing of ctrl+shift+2 or ctrl+shift+4 leads to appropriate division. Functionality is intended for data input from drawings (for example, radius input, or drawing scale consideration)

8.2 [Kinematic joint] element

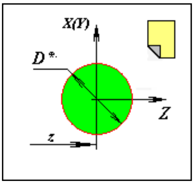
The element defines rotational speeds and speed ranges of the subsystem. It is used for rotating and non-rotating subsystems, Figure 8.5. In last case rotation speed is always zero.

The law of changing rotational speed versus time may differ:

- a user can input constant rotational speed of rotating subsystem. In this case it is impossible to calculate the critical speeds;
- a user can define rotation speed of a rotor versus time varying from 0 to maximum. Data input is carried out by means of **[Variables]** commands.

Menu symbols	Input box

Kinematic joint



Des	Kinematic joint 2		Designation
z	0	mm	z coordinate
omega_z	0	1/min	Velocity about z direction
D*	0	mm	Outer diameter

Figure 8.5

Input of rotational speeds is carried out using the following steps:

1. A user adds [Kinematic joint] element in the corresponding subsystem. On default the name of element is the name of subsystem.

> **NOTE:** For given element the place (coordinate z) in the subsystem does no matter

2. A user creates new variable by means of commands **[Variable], [Value variable].** In the left lower window the **[Value variable]** input box is appeared **Figure 8.6.**

Des		Designation
value	0	Value

Figure 8.6

3. Using the right mouse button, a user opens list of the following commands, Figure 8.7.

4. Using the command **[Extended properties]**, a user can open the **[Value parameter]** input box, **Figure 8.8 .**

The input box is opened by the mouse left button click.

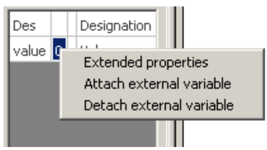


Figure 8.7

NOTE: It is recommended to name all **[Value variables]** elements in correspondence with the name of variable parameter

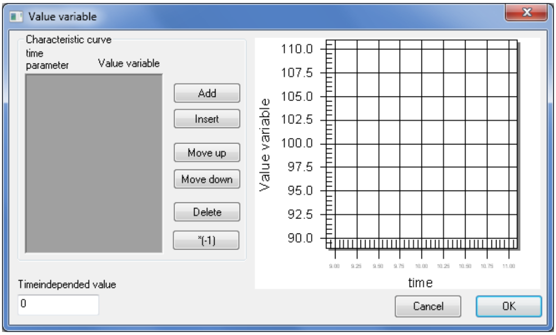


Figure 8.8

5. Then a user adds the corresponding number of time parameters. For them the variable value **[Value variable]** will be obtained, **Figure 8.9**

Number of points, addition, movement, etc. may be controlled by buttons.

NOTE: [time parameter] (noted as t_pr in the algorithms settings) is an abstract parameter that can bring to the same scale change in the parameters that are different in their physics.

Table 8.2 gives different variants of assigning law of change in rotating speed. All of them define acceleration from 0 to 10000 rpm. In (1) variant parameter t_pr represents rotating speed. In (2) variant percentage at the regime is shown, in (3) - modified speed. If at run-up the rotor accelerates up to 10000 rpm for 12 sec, (4) variant allow parametrizing the parameters at the test time (it can be useful in the [Transient response] algorithm])

Table 8.2

1		**2**		**3**		**4**	
t_pr	**Value**	**t_pr**	**Value**	**t_pr**	**Value**	**t_pr**	**Value**
0	**0**	**0**	**0**	**0**	**0**	**0**	**0**
10000	**10000**	**100**	**10000**	**1**	**10000**	**12**	**10000**

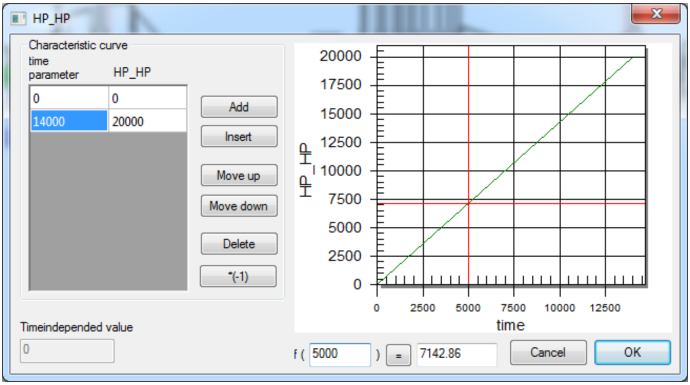


Figure 8.9

The window buttons give opportunity to control the input and editing processes.
Fields under the plot allow to know interpolated value of the function or the argument.

6. The last step is connection **[Value variable]** to variable parameter **[omega_z]**. A user opens the **[Kinematic joint]** element and executes the command **[Attach external variable],**Figure 8.10**.**

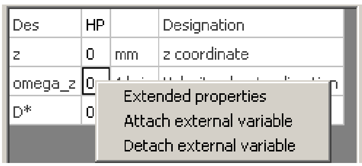


Figure 8.10

NOTE: If in a subsystem the **[Kinematic joint]** is not defined, **the subsystem speed is assumed to be equal to 0.** This assumption generates a corresponding warning

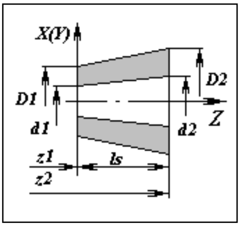
NOTE: If a subsystem speed is constant, it may be directly input in the element parameters line besides the **[Extended properties]** command

8.3 [Beam] element

The element **[Beam]** is an axis symmetric (isotropic) element, **Figure 8.11.**

Menu symbols

Beam



Input box

Des	Beam_1		Designation
segRef	length		Measurement
ls	29	mm	Length of element
z1	56	mm	Start coordinate
z2	85	mm	End coordinate
Type	Cone		Type
d1	26	mm	Inner start diameter
D1	40	mm	Outer start diameter
d2	26	mm	Inner end diameter
D2	40	mm	Outer end diameter
material	From database		Material
material	titan		Material from database

Figure 8.11

The element can be cylindrical or conical, with or without inner hole. Cross-sectional area changes linearly. Using **[cs]** of the **[Options]** command, the automatically subdivision into shorter parts may be done.

For each individual beam the elastic and mass parameters are computed and can be observed in the protocol output.

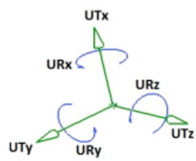


Figure 8.12

The rule of signs for determining of elastic coefficients is shown in **Figure 8.12.**

NOTE: For thin-walled conical sections elements [Shell] should be used instead of [Beams]

NOTE: On default all beam elements are calculated with consideration of shear deformation

The commands to convert elements from one type to another were added into the context menu of beams and shells, **Figure 8.13**
. Thin-shelled elements of cases and journals should be simulated by shell elements. If on the modeling stage they were simulated by beam elements, they may be substituted using the commands in the context menu.

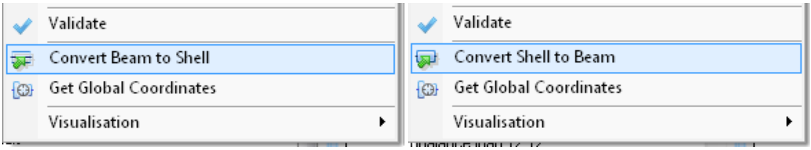
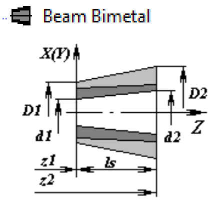


Figure 8.13

8.4 [Beam Bimetal]

[Beam Bimetal] - is an axis symmetric (isotropic) element, Figure 8.14. Allows to set various properties of materials in the radial direction.

Menu symbols	Input box

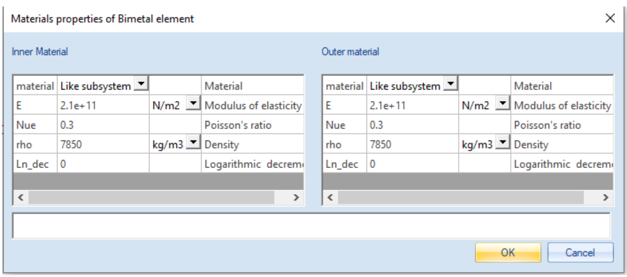


Des	Beam Bimetal 1		Designation
segRef	z1 and z2		Measurement
ls	100	mm	Length of element
z1	0	mm	Start coordinate
z2	100	mm	End coordinate
Type	Cone		Type
d1	0	mm	Inner start diameter
D1	50	mm	Outer start diameter
d2	0	mm	Inner end diameter
D2	50	mm	Outer end diameter
D1_in	40	mm	Inner part - Outer start diameter
D2_in	40	mm	Inner part - Outer end diameter
material	...		material
el. vis.	color...		el. vis.
Visibility	0		Element Visibility
el. vis.	color...		el. vis.
El. results	...		El. results

Figure 8.14

The element can be cylindrical or conical, with or without inner hole. Cross-sectional area changes linearly. Using **[cs]** of the **[Options]** command, the automatically subdivision into shorter parts may be done.

Material properties are set in the dialog (Figure 8.15) called by double clicking on the [material] parameter or from the context menu of the parameter.


Materials properties of Bimetal element

Inner Material

material	Like subsystem		Material
E	2.1e+11	N/m2	Modulus of elasticity
Nue	0.3		Poisson's ratio
rho	7850	kg/m3	Density
Ln_dec	0		Logarithmic decrem

Outer material

material	Like subsystem		Material
E	2.1e+11	N/m2	Modulus of elasticity
Nue	0.3		Poisson's ratio
rho	7850	kg/m3	Density
Ln_dec	0		Logarithmic decrem

OK Cancel

Figure 8.15

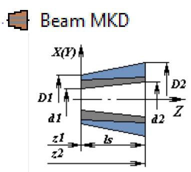
In addition to bimetallic shafts (for example, gear wheels), the element is suitable for modeling a distributed mass that is not involved in the main stiffness of a shaft or housing. For example, in generators and electric motors. Suitable for modeling attachment parts made of different materials mounted with interference.

8.5 [Beam MKD]

[Beam MKD] - is an axis symmetric (isotropic) element, Figure 8.16. Allows you to set different diameters for stiffness and mass characteristics.

The element can be cylindrical or conical, with or without inner hole. Cross-sectional area changes linearly. Using **[cs]** of the **[Options]** command, the automatically subdivision into shorter parts may be done.
Mass diameters are set larger or equal to the stiffness ones. But the calculation will be carried out correctly for arbitrary ratios of diameters.

Menu symbols



Input box

Des	Beam MKD 2		Designation
segRef	z1 and z2		Measurement
ls	100	mm	Length of element
z1	100	mm	Start coordinate
z2	200	mm	End coordinate
Type	Cone		Type
d1_M	0	mm	Inner start mass diameter
D1_M	50	mm	Outer start mass diameter
d1_S	0	mm	Inner start stiffness diameter
D1_S	50	mm	Outer start stiffness diameter
d2_M	0	mm	Inner end mass diameter
D2_M	50	mm	Outer end mass diameter
d2_S	0	mm	Inner end stiffness diameter
D2_S	50	mm	Outer end stiffness diameter
material	Like subsystem		Material
el. vis.	color...		el. vis.
Visibility	0		Element Visibility
el. vis.	color...		el. vis.
El. results	...		El. results

Figure 8.16

The element is suitable for implementing the recommendations of the API standard (Figure 8.17).

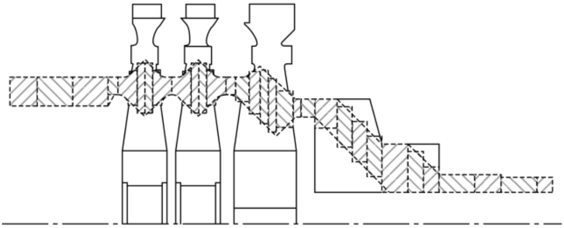


Figure 2-7—Modeling of Curvic Coupling Joints

Figure 8.17

Element visualization can be separately configured for mass-inertial and stiff parts (Figure 8.18).

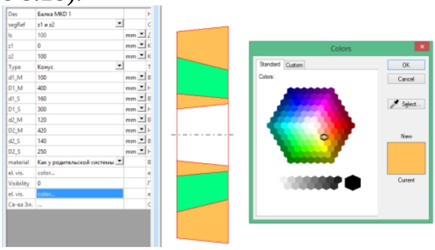
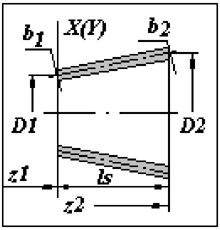


Figure 8.18

8.6 [Shell] element

Menu symbols	Input box

Shell



Des	Shell 2		Designation
segRef	z1 and z2		Measurement
ls	0	mm	Length of element
z1	100	mm	Start coordinate
z2	100	mm	End coordinate
b1	2	mm	Start thickness
D1	50	mm	Middle start diameter
b2	2	mm	End thickness
D2	50	mm	Middle end diameter
material	Like subsystem		Material
Visibility	0		Element Visibility
red	196		Red color component
green	196		Green color componen
blue	196		Blue color component

Figure 8.19

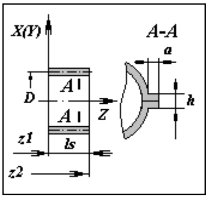
The element **[Shell]** is an axially symmetric (isotropic) cylindrical or conical element**, Figure 8.19.** built around the symmetry axis **Z**. Cross-sectional area changes linearly. Geometry is defined by the length, middle diameter and shell thickness in the start and end section.

Starting from 4.8.8 version, cylindrical shells may be automatically divided into subsections depending on the settings of the [shell autosplit] parameter in the root element [System] of the model tree. The parameter [without cs] - subdividing takes place only where links and point elements are set. When choosing [always] - the parameter [cs] of automatical subdividing of the subsystem elements will be taken into account additionally.
The shell element simulates behavior of the conical shell elements of the design from flange to flange (between stiff sections). It is not recommended to divide conical parts between flanges without validation in FEM.

NOTE: To simulate thin-walled conical elements of the design, the [Shell] element should be used

8.7 [Shell with flange] element

Menu symbols	Input box



Des	Shell with flange 1		Designation
segRef	z1 and z2		Measurement
ls	100	mm	Length of element
z1	0	mm	Start coordinate
z2	100	mm	End coordinate
b	2	mm	Thickness
D	50	mm	Middle diameter
a	5	mm	Width of flange
h	10	mm	Thickness of flange
material	Like subsystem		Material
Visibility	0		Element Visibility
red	196		Red color component
green	196		Green color componen
blue	196		Blue color component

Figure 8.20

The element **[Shell with flange]** is an orthotropic element, **Figure 8.20.** It can be only cylindrical. The element is used for modeling of split compressors of gas turbine engines and turbochargers with longitudinal flanges.

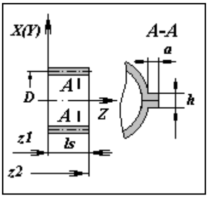
8.8 [Squirrell Cage (Full matrix)] element

[Squirrell Cage (Full matrix)] - a special element in DYNAMICS R4 for modeling the stiffness of an elastic support of the “Squirrell Cage” type. It is an extended element of the subsystem. The specified parameters (Figure 8.21) make it possible to obtain a complete compliance matrix. Additional parameters for design can also be obtained.

To calculate the coefficients of the flexibility matrix, the numerical-analytical method proposed in [10] is used. The assumption that the cross section of the elastic elements has a rectangular shape. It should be borne in mind that the manufacture of a squirrel wheel using a cutter will give a rectangular hole shape and a trapezoidal cross-section of elastic elements. In this case, the width of the elastic elements is set according to the average diameter of the squirrel wheel. Or it is automatically calculated when specifying the hole width [b_hole].

The [Yes/No] switch of the [add output] parameter allows you to show additionally calculated parameters for the “squirrel wheel”, Figure 8.22.

Menu symbols	Input box
Squirrel Cage (Full matrix)	see Figure 8.21



Des	Squirrel Cage (Full matrix) 1			Designation
segRef	z1 and z2			Measurement
ls	100	mm		Length of element
z1	0	mm		Start coordinate
z2	100	mm		End coordinate
width for	hole			Choose bar or hole width input
b_hole	8	mm		Squirrel cage hole width
b	3.38279	mm		Bar width 1
h	3.8	mm		Bar height 1
r	4	mm		Squirrel cage bar rounding radius
n	43			Number of bars
D cage	152	mm		Squirrel cage diameter 1
material	Like subsystem			Material
add output	No			show additional output
Kr	1.64981e+06	N/m		Stiffness coefficient
Visibility	0			Element Visibility
el. vis.	color...			Element visualisation
El. results	...			Element results

Figure 8.21

add output	yes		show additional output
delta	0.1	mm	Radial clearance.
F	2000	N	load in radial direction
ut	1.21226	mm	Radial deflection
max stress	29.8101	MPa	Maximal alterating stress. e=1
stress	361.375	MPa	Stress due to load.
Fmax	164.981	N	Max load in radial direction.
Kr	1.64981e+06	N/m	Stiffness coefficient
Visibility	0		Element Visibility
el. vis.	color...		Element visualisation
El. results	...		Element results

Figure 8.22

By double clicking on [El. results] (Figure 8.22) or using the command in the context menu on the field […] you can view the resulting squirrel cage flexibility matrix.

The parameter [Kr] is the radial stiffness of the squirrel wheel for the case of application only by radial degrees of freedom in connections with zero distance between connection nodes. Corresponds to the inversion of the radial flexibility coefficient. The complete stiffness matrix for coupling with non-zero node spacing (Figure 8.23) can be obtained by inverting the flexibility matrix.

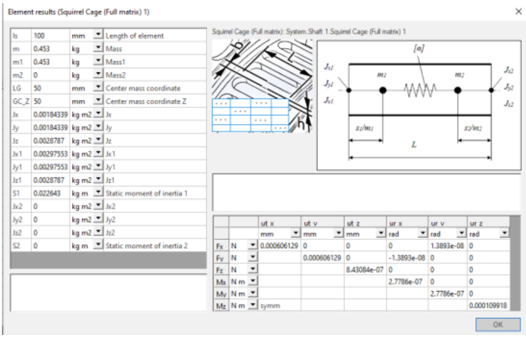


Figure 8.23

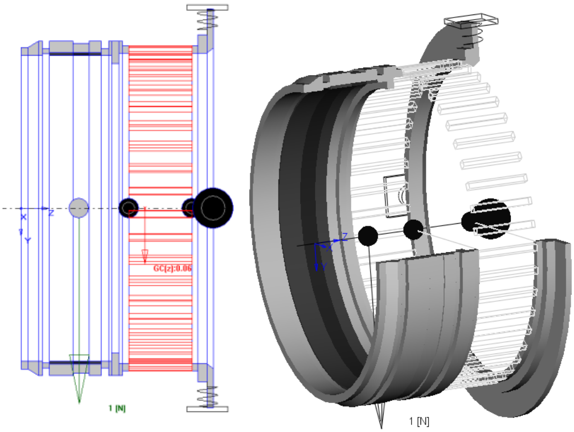
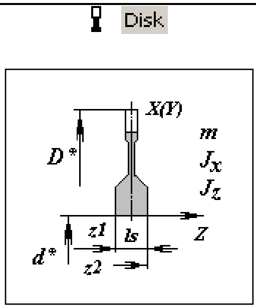


Figure 8.24

8.9 [Disk] element

The element **(Figure 8.25)** simulates typical compressor or turbine stages. A disk is defined by inertia characteristics - mass, transverse and axial moments of inertia and a width between two boundary sections which the other elements are attached to.

Menu symbols	Input box
	

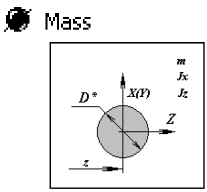
Des	Disk		Designation
segRef	length		Measurement
ls	57.5	mm	Length of element
z1	259	mm	Start coordinate
z2	316.5	mm	End coordinate
m	8.92	kg	Mass
type	Isotropic		Type
Jx	0.0183	kg m2	Jx
Jz	0.0336	kg m2	Jz
d*	28	mm	Inner diameter
D*	200	mm	Outer diameter

Figure 8.25

A user can change the form of the disk by means of **d*** and **D*** sizes. These values are used only for visualization purpose.

8.10 [Mass] element

The **[Mass]** element models the mass element concentrated in a point, Figure 8.26. It is defined by inertial parameters. It does not have length.

Menu symbols	Input box
	

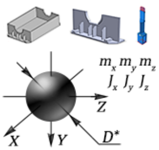
Des	Mass_1		Designation
z1	149	mm	Start coordinate
m	6.832	kg	Mass
type	Isotropic		Type
Jx	0.02	kg m2	Jx
Jz	0.027	kg m2	Jz
D*	20	mm	Outer diameter

Figure 8.26

8.11 [Mass pedestal] element

The **[Mass pedestal]** element models the mass element concentrated in a point, Figure 8.27. It is defined by inertial parameters. It does not have length. Allows to define different mass values in radial and axial directions. Intended for simulation of foundation (pedestal) natural frequencies for reduced model.

Menu symbols	Input box
Mass pedestal	(see input box table below)



Des	Mass pedestal 6		Designation
z1	0	mm	Start coordinate
mx	100	kg	Mass in X direction
my	110	kg	Mass in Y direction
mz	80	kg	Mass in Z direction
Jx	100	kg m2	Jx
Jy	100	kg m2	Jy
Jz	100	kg m2	Jz
R*	0	mm	Fictive radius
Vis as	default		Element Visualisation as...
Visibility	0		Element Visibility

Figure 8.27

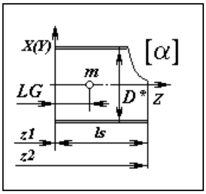
8.12 [Generalized] element

Generalized element (**Figure 8.28)** is used to model a complex element of dynamic structures which properties are known from tests or FEM programs. It can be specified either as an orthotropic or isotropic element. The element can be reduced up to absolutely rigid or up to flexible massless mode.

The input data are length, mass, inertia moments, matrix of flexibility coefficients. Mass center is specified with the distance from the left side of element.

The variable **[inertia_cos]** gives the way of inertia parameters modeling. The mass can be set in the mass center of the element, or partitioned for both of the element ends. In the latter case the static moments are also defined for the both of masses.

Menu symbols	Input box
Abstract element	



Des	MasslessElement_1		Designation
segRef	length		Measurement
ls	55.5	mm	Length of element
z1	52	mm	Start coordinate
z2	107.5	mm	End coordinate
type	general		Type of element
type	Isotropic		Type
inertia_cos	center of mass		Coordinate system for inertia properties
m	0	kg	Mass
LG	0	mm	Center mass coordinate
Jx	0	kg m2	Jx
Jz	0	kg m2	Jz
flex_matrix	...		Flexibility matrix
D*	140	mm	Outer diameter
Ln_dec	0		Logarithmic decrement

Figure 8.28

The elastic properties of the element are defined by a symmetric flexibility matrix (isotropic or orthotropic), **Figure 8.29**. Selected on the figure elements are taken into account. The others are neglected.

Designation of rows and columns were changed since Dynamics R4.11 version. Displacements and rotations are in the rows, loads in the columns. Order of coefficients in flexibility was not changed.

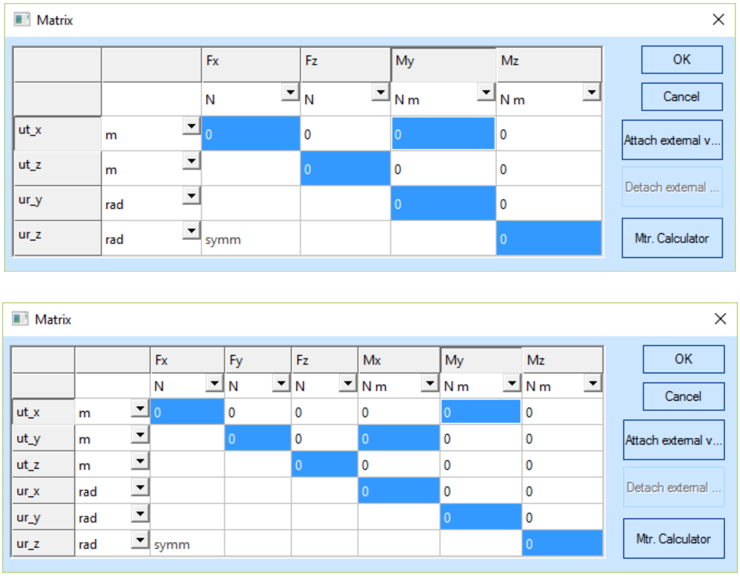


Figure 8.29

The coefficients of the matrix are defined accordingly to sign rule represented in **Figure 8.12.**

NOTE: At input of the flexibility matrix its coefficients must be in a correspondence with each other. The coefficients should reflect real properties of the design described by this element

NOTE: If mass characteristics are determined at the mass center of the generalized element, then mass characteristics are set at the left section of generalized element. If this section is rigid mass characteristics are not taken into account in calculation

Description of the element parameters is given in the **Table 8.3**.

Table 8.3

Name	Parameter description
Type **General,** **Massless** **Rigid**	The variable describes the element type. The following versions are available: - General form element, - Zero inertia element; - Zero flexibility element. The editing table opens only parameters required for this type of element
Type **Isotropic,** **Orthotropic**	The variable describes symmetry of the element properties: - Isotropic one has circular symmetry versus the **Z** axis; the parameters with **"y"** index are automatically initialized by the parameters with **"x"** one and not accessible for editing, - Orthotropic one has two orthogonal symmetry planes. The parameters are defined separately in **YOZ** and **XOZ** planes in the local co-ordinate system and may differ.
inertia_cos **Boundary,** **Center of mass**	The variable describes the inertia parameters of the model with the following options: - The inertia properties are concentrated in the element center of mass, - The inertia properties are distributed between the ends, - Distribution is defined by a user
m	Element mass
LG	The **Z**-coordinate, distance between the element left end and its center of mass.
Jx	Mass moment of inertia about the **X** axis
Jy	Mass moment of inertia about the **Y** axis
Jz	Mass moment of inertia about the **Z** axis
m1	Mass in the element left end
Jx1	Inertia moment of the element left part around the **X** axis in the general co-ordinate system **XYZ,** located in the left section
Jy1	Inertia moment of the element left part around the **Y** axis in the general co-ordinate system **XYZ,** located in the left section
Jz1	Inertia moment of the element left part around the **Z** axis in the general co-ordinate system **XYZ,** located in the left section
S1	Static moment of the element left part around the X, Y and Z axes located in the element left end
m2	Mass in the element right end
Jx2	Inertia moment of the element right part around the **X** axis in the general co-ordinate system **XYZ,** located in the left section

Jy2	Inertia moment of the element right part around the **Y** axis in the general co-ordinate system **XYZ,** located in the left section
Jz2	Inertia moment of the element right part around the **Z** axis in the general co-ordinate system **XYZ** located in the left section
S2	Static moment of the element right part around the X, Y, Z axes located in the element left end.
dUTx/dFx	The direct coefficient of force flexibility, links the right end displacement in X direction to the load applied along X axis; the left end is fixed
dUTx/dMx	The cross-influence coefficient of flexibility links the right end displacement in **X** direction to the moment applied around X-axis; the left end is fixed
dURx/dMx	The direct coefficient of moment flexibility links the right end angular displacement around **X** axis and the moment applied around X-axis; the left end is fixed
dUTy/dFy	The direct coefficient of force flexibility links the right end displacement in **Y** direction and the load applied along Y-axis; the left end is fixed
dUTy/dMy	The cross-influence coefficient of flexibility links the right end displacement in **Y** direction and the moment applied around Y-axis; the left end is fixed
dURy/dMy	The direct coefficient of moment flexibility links the right end angular displacement around **Y** axis and the moment applied around Y-axis; the left end is fixed
dUTz/dFz	The direct coefficient of force flexibility links the right end displacement in **Z** direction and the load applied along Z-axis; the left end is fixed.
dURz/dMz	The direct coefficient of moment flexibility links the right end angular displacement around **Z** axis and the moment applied around Z-axis; the left end is fixed.
D*	The variable sets the abstract element size, or diameter for the model visualization

8.13 Visualization of the [Disk], [Mass], [Generalized element] elements

Starting from Dynamics 4.6.5, there is a possibility of extended setting of visualization of the [Disk], [Mass], [Generalized element] elements. In addition to displaying elements “on default” [standard], there is a possibility of advanced setting in the [disk] and [file] modes.

For the [Disk] element the [standard] and [disk] modes may be used more often. The [disk] mode allows setting dimensions of hub and rim diameters, blades height. The predominant choice of settings for this element is the [whole] mode. In this regime both 2D visualization parameters and displaying of 3D model from file in the VRML format may be adjusted. If the directory is empty, 3D visualization is done automatically on the basis of the given dimensions of the disk.

For the [Mass] and [Generalized element] elements the [disk] and [file] visualization modes may be useful. In the [file] mode the element's 3D visualization may be accomplished by displaying the 3D model loaded from the file in the VRML format. In this case 2D displaying is shown by the rectangle of the given dimensions. When loading from the file the setting of a scaling factor of 3D model (**[to_m_factor]** – a factor of converting the model units in meters) may be required. Complex design elements (for instance, the fan disk, complex elements of supports) changing the typical product appearance may be visualized from the file. To specify mass-inertia and elastic design characteristics with the input of the obtained data in **[Generalized element**] it is preferable to load visualization of such elements from the file.

The **[Mass]** element may be successfully used to set the mass-inertia characteristics of disks and their extended visualization. The **[without rim]**, **[without hub]**, **[blades only]** modes may be used together with beam elements. For example, the hub (if it belongs to the load-bearing contour) may be simulated by the beam element with no density and the corresponding modulus of elasticity. The **[blades only]** mode is used if the web and hub of the disk are absent (for example, blades are placed on the compressor).

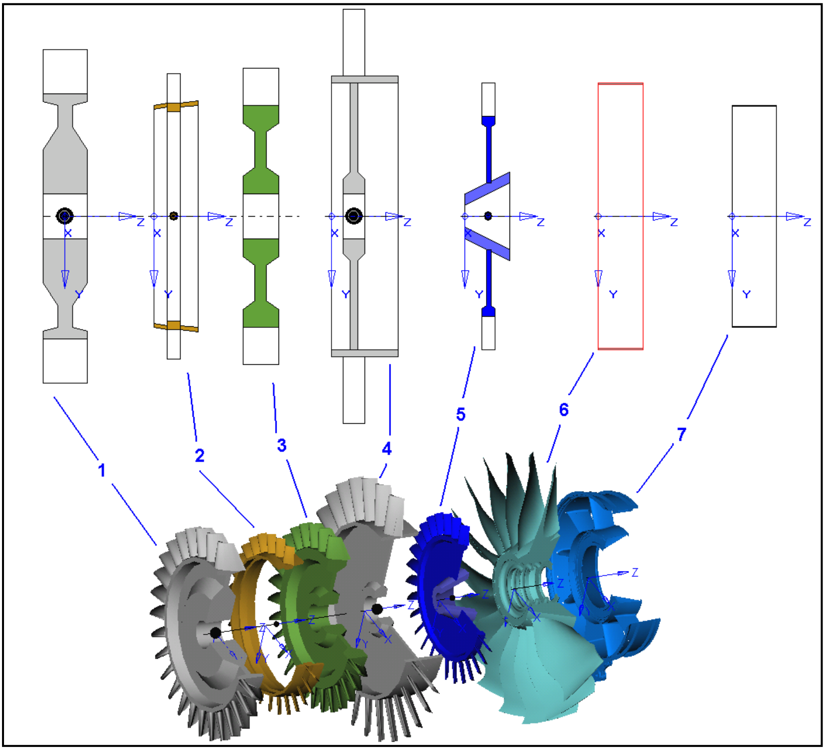


Figure 8.30

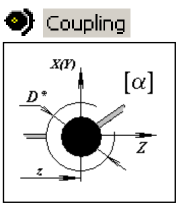
The following alternatives are shown by figures in **Figure 8.30**:

1. Visualization of mass as disk in the **[whole]** mode
2. Visualization of mass as disk in the **[blades only]** mode. The part of the compressor is simulated by the beam elements
3. Disk visualization in the **[standard]** mode
4. Visualization of mass as disk in the **[without rim]** mode. The rim is simulated by the beam element.
5. Visualization of mass as disk in the **[without hub]** mode. The hub is simulated by the beam elements
6. Disk visualization in the **[file]** mode with 3D display of the fan model.
7. Visualization of the generalized element in the **[file]** regime with display of load-bearing case elements.

8.14 [Coupling] element

General case

Axis symmetrical (isotropic) or orthotropic element allows introducing moment and torsion flexibility concentrated in the point, **Figure 8.31.** It is possible to model local flexibilities such as bending flexibility of flange joint.

Menu symbols	Input box
	see table below

Des	Coupling 30		Designation
z1	355.267	mm	Start coordinate
dURx/dMx	0	1/N*m	Flexibility about x-axis
D*	0	mm	Outer diameter

Figure 8.31

> **NOTE:** Do not use orthotropic moment flexibility for rotating rotor.

The visualization of a coupling element used in a 2D model is shown in **Figure 8.32.**

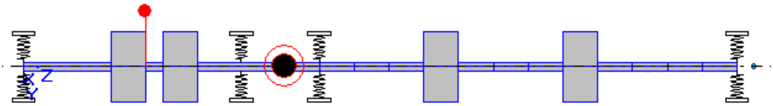
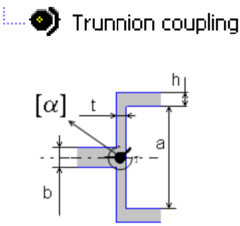


Figure 8.32

Trunnion coupling

When a beam length is much smaller than its diameter the element may be considered as a plate. The plate has a small bending flexibility, that may be neglected, and a high moment one.
The plate geometry is described by the beam element. The moment flexibility is simulated using the [**Trunnion coupling**], **Figure 8.33.**

Menu symbols	Input box



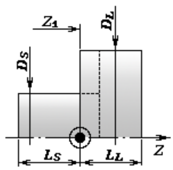
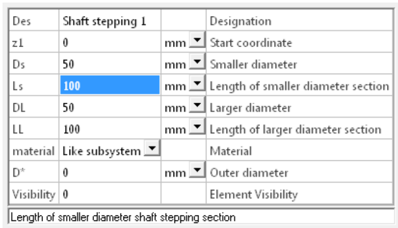
Des	Trunnion coupling 1		Designation
z1	0	mm	Start coordinate
b	50	mm	Rod diameter
t	25	mm	Plate width
a	1080	mm	Inner cylinder diameter
h	25	mm	Cylinder thickness
material	Like subsystem		Material
D*	0	mm	Outer diameter
Visibility	0		Element Visibility

Figure 8.33

The flexibility values are available in the **[Protocol]**. The element should be used under the following conditions: ***a-b>>t* *and* *a>>b***.

Shaft stepping

Element is a joint modeling the additional torsional flexibility, which should be considered when a sudden step change of the diameter of the shaft occurs. This need arises because of the effect of penetration (Shaft Stepping) when a ratio of diameters> 1.5.,**Figure 8.34.**

Menu symbols	Input box
 	

Des	Shaft stepping 1		Designation
z1	0	mm	Start coordinate
Ds	50	mm	Smaller diameter
Ls	100	mm	Length of smaller diameter section
DL	50	mm	Larger diameter
LL	100	mm	Length of larger diameter section
material	Like subsystem		Material
D*	0	mm	Outer diameter
Visibility	0		Element Visibility

Length of smaller diameter shaft stepping section

Figure 8.34

9 LINEAR LINKS BETWEEN SUBSYSTEMS

The following linking elements between subsystems can be used when modeling:

- Elastic-damping link of a general type **[Link]**
- Rigid link of a general type **[Rigid link]**
- Elastic-damping link with nonsymmetrical stiffness and damping matrixes **[Elastic nonsymmetric link]**
- Gearing of a general type **[Abstract gear set]**
- Element **[Connection point]**

In the element library these elements are presented (**Figure 9.1**):

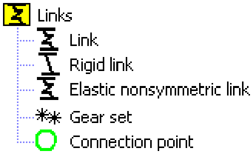


Figure 9.1

In the context menu of any link there is possibility to copy the characteristics of connection and visualization settings of the active link into clipboard, **Figure 9.2**.

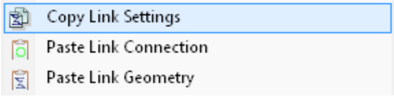


Figure 9.2

After the settings are copied using the command **[Copy Link Settings]**, it is possible to make the other link active (for example, by left mouse button) and paste the link settings from the clipboard **[Paste Link Connection]** or parameters of the link visualization **[Paste link geometry]**. The command data hasten a user's work with complex models and allow decreasing risk of appearing mistakes like a human factor when replacing one-type links for another or arrangement of parallel links.

To simplify and hasten work on links connection in complex models, you should use the following commands of the context menu. In the connection point menu, the command **[Use CP for connection]** allows copying information about the point of interest into clipboard for attachment. After this, it becomes possible to attach the copied connection point to the first (Side1) or the second (Side2) sides of the link, **Figure 9.3**.

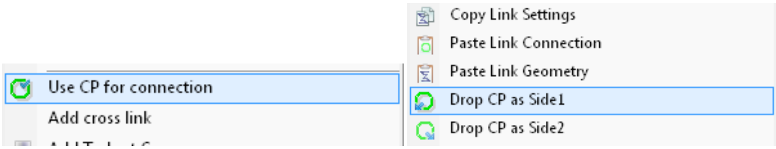


Figure 9.3

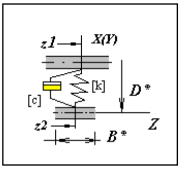
9.1 Elastic-damping link of a general type [Link]

The element connects two subsystems (including zero subsystem) by means of a linear elastic-damping link, **Figure 9.4**.

The link may be represented by two types – **[Isotropic]** link and link of general type **[Full].** Elastic properties of the link are assigned by stiffness and damping matrixes. Dimension of matrix for general type link is 6x6. Values of stiffness coefficients are obtained under boundary **section 1** loading when boundary **section 2** is fixed.

Menu symbols	Input box

Link



Des	Front support		Designation
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
side1_l	0	mm	side1 offset
side2_subs			Side2 subsystem
trns_exclude	yes		Exclude from transient analysis
Type	Isotropic		Link type
stiff_matrix	...		Stiffness matrix
damp_matrix	...		Damping matrix
d*	0	mm	Inner diameter
D*	200	mm	Outer diameter
B*	100	mm	width

Figure 9.4

The elastic properties of a link are defined by the stiffness matrix, **Figure 9.5**. In the presented example stiffness coefficient **Fx/ut_x** is defined as a variable depending on time.

Matrix

		ut_x	ut_y	ut_z	ur_x	ur_y	ur_z
		mm	mm	mm	rad	rad	rad
Fx	N	226813	-7.59053e-009	-6.03043e-008	4.6194e-006	-7.36009e+007	6.22057e-006
Fy	N		154068	2.64582e-006	4.99951e+007	-0.000424997	3.26069e-005
Fz	N			1.47009e+006	3.37564e-005	-2.59549e-005	-3.73168e-005
Mx	N mm				3.89468e+010	-0.337385	0.0251888
My	N mm					5.90387e+010	-0.00307817
Mz	N mm	symm					1.61439e+010

OK / Cancel / Attach external v... / Detach external ... / Mtr. Calculator

Figure 9.5

The damping properties of a link are defined by the damping matrix, **Figure 9.6**.

Matrix

		vt_x	vt_y	vt_z	vr_x	vr_y	vr_z
		m/s	m/s	m/s	rad/s	rad/s	rad/s
Fx	N	1000					
Fy	N		1000				
Fz	N			1000			
Mx	N m				100		
My	N m					100	
Mz	N m						100

OK / Cancel / Attach external v... / Detach external ... / Mtr. Calculator

Figure 9.6

Designation of rows and columns were changed since Dynamics R4.11 version. Displacements and velocities are in the columns, loads in the rows.

Order of coefficients in stiffness and damping matrixes was not changed (Table 9.1).

The dialog with simultaneous output of stiffness and damping matrixes is available by double click on the name ([stiff_matrix] or [damp_matrix]) or in the context menu of the element, **Figure 9.7.**

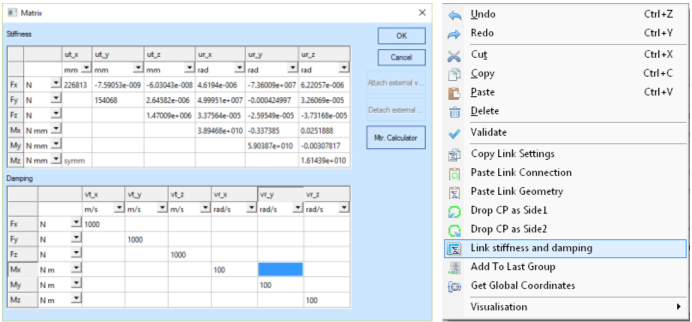


Figure 9.7

A link may be either anisotropic or isotropic. The first one has equal stiffness and damping coefficients in **X** and **Y** directions; the second one has a user-defined set of coefficients.

Table 9.1

	Ut_x[m]	Ut_y[m]	Ut_z[m]	Ur_x[rad]	Ur_y[rad]	Ur_z[rad]
Fx[N]	$K_{xx} = \frac{\partial F_x}{\partial x}$	$K_{xy} = \frac{\partial F_x}{\partial y}$	$\frac{\partial F_x}{\partial z}$	$\frac{\partial F_x}{\partial r\_x}$	$\frac{\partial F_x}{\partial r\_y}$	$\frac{\partial F_x}{\partial r\_z}$
Fy[N]	$K_{yx} = \frac{\partial F_y}{\partial x}$	$K_{yy} = \frac{\partial F_y}{\partial y}$	$\frac{\partial F_y}{\partial z}$	$\frac{\partial F_y}{\partial r\_x}$	$\frac{\partial F_y}{\partial r\_y}$	$\frac{\partial F_y}{\partial r\_z}$
Fz[N]	$\frac{\partial F_z}{\partial x}$	$\frac{\partial F_z}{\partial y}$	$\frac{\partial F_z}{\partial z}$	$\frac{\partial F_z}{\partial r\_x}$	$\frac{\partial F_z}{\partial r\_y}$	$\frac{\partial F_z}{\partial r\_z}$
Mx [N-m]	$\frac{\partial M_x}{\partial x}$	$\frac{\partial M_x}{\partial y}$	$\frac{\partial M_x}{\partial z}$	$\frac{\partial M_x}{\partial r\_x}$	$\frac{\partial M_x}{\partial r\_y}$	$\frac{\partial M_x}{\partial r\_z}$
My [N-m]	$\frac{\partial M_y}{\partial x}$	$\frac{\partial M_y}{\partial y}$	$\frac{\partial M_y}{\partial z}$	$\frac{\partial M_y}{\partial r\_x}$	$\frac{\partial M_y}{\partial r\_y}$	$\frac{\partial M_y}{\partial r\_z}$
Mz [N-m]	$\frac{\partial M_z}{\partial x}$	$\frac{\partial M_z}{\partial y}$	$\frac{\partial M_z}{\partial z}$	$\frac{\partial M_z}{\partial r\_x}$	$\frac{\partial M_z}{\partial r\_y}$	$\frac{\partial M_z}{\partial r\_z}$

Coefficients of the stiffness and flexibility matrixes may be determined as varying values within the rotation frequency range. If so, the **Natural Frequencies** and **Natural Frequency Map** algorithms operate.

> **NOTE:** Variables attached to stiffness and matrix cells are considered in SI units. Changing of units in combo boxes does not affect on values in variables.

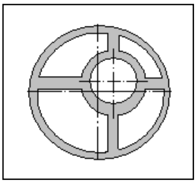
For some unsteady tasks (subsystems are connected only by non-linear links) the basis calculation should take into account the subsystems coupling. In this case linear link is included into the points of nonlinear links attachment. During unsteady analysis this link is excluded. It is done by the **[yes]** value setting in the **[trns_exclude]** parameter.

> **NOTE:** The link **[trns_exclude]** excludes only the stiffness part of a link in the unsteady analysis; the damping part is not excluded

9.2 [Rigid link] element

Link allows connecting of subsystems and defining properties of a link (boundary conditions) in all degrees of freedom – either it is fixed or free, **Figure 9.8.**

Menu symbols



Input box

Des	Rigid link 1		Designation
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
in_cos	global		Coordinate system for side1 section
side1_l	0	mm	side1 offset
side2_subs			Side2 subsystem
UTx	fixed		Translation in x-direction
UTy	fixed		Translation in y-direction
UTz	fixed		Translation in z-direction
URx	fixed		Rotation about x-axis
URy	fixed		Rotation about y-axis
URz	fixed		Rotation about z-axis
d*	0	mm	Inner diameter
D*	150	mm	Outer diameter
B*	20	mm	width

Figure 9.8

Table 9.2

UTx	Variable defines properties of a link in **X** axis direction - fixed or free
URx	Variable defines properties of a link about **X** axis - fixed or free
UTy	Variable defines properties of a link in **Y** axis direction - fixed or free
URy	Variable defines properties of a link about **X** axis - fixed or free
UTz	Variable defines properties of a link in **Z** axis direction- fixed or free
URz	Variable defines properties of a link about **Z** axis- fixed or free

9.3 [Elastic non-symmetric link] element

The element connects two subsystems (including zero one) by means of a linear elastic-damping link described by nonsymmetrical stiffness and damping matrixes **(Figure 9.9)**.

The element is used in linear analysis of systems with variable stiffness supports; the support stiffness coefficients change versus time or rotation speed. This link is not considered in the basis calculation and is used together with the main link element **[Link]**.

This kind of link may be used for modeling of nonlinear supports of any types (journal bearings, tilting-pad bearings, squeeze-film dampers, rolling bearings, etc) when the data concerning their stiffness and damping coefficients versus rotating speed are presented (given by a manufacturer or found by any other way). In this case those coefficients are given by variable parameters in the whole-time range of rotating speeds.

Dimension of stiffness and damping matrixes for general type link is 6x6. Stiffness of the main link element **[Link]** is assigned by a symmetrical or diagonal stiffness matrix that is used for the basis calculation. Coefficients of the non-symmetrical stiffness matrix are additions to the symmetrical matrix.

Values of coefficients are defined in boundary **section 1** when boundary **section 2** is fixed.

Stiffness of the main link element **[Link]** is specified by a symmetrical or diagonal stiffness matrix that is used for the basis calculation. Coefficients of the non-symmetrical stiffness matrix are additives to the symmetrical matrix.

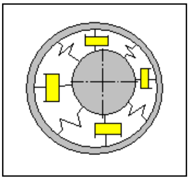
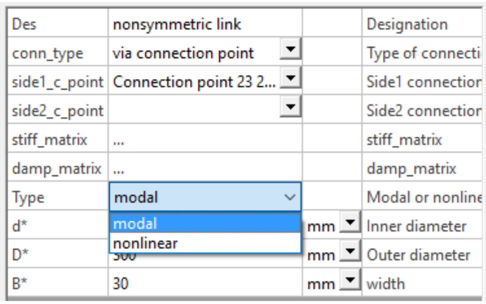
Menu symbols	Input box
Elastic nonsymmet 	

Figure 9.9

The elastic properties of a link are defined by the nonsymmetrical stiffness matrix, **Figure 9.10.** As opposed to Elastic-damping link of a general type, the matrix is full.

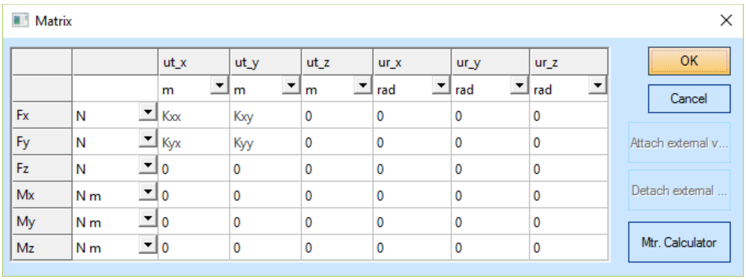


Figure 9.10

> **NOTE:** The nonsymmetrical stiffness matrix is not used in the basis calculation. Rigid body modeshapes in Basis algorithm results are needed for correct consideration of this link. User can use [Link] elements with relatively low stiffness values (for example 1000 N/M) in radial direction.

The damping properties of a link are defined by the nonsymmetrical damping matrix, Figure 9.11.

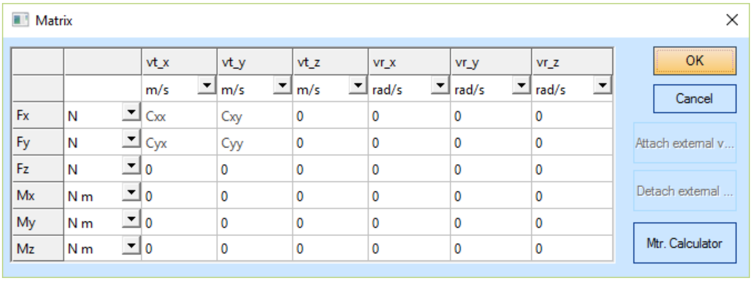


Figure 9.11

If variables are connected to the link, then when you double-click on the link or the name of the stiff_matrix or damp_matrix parameter, the “Import fluid bearing data” from table dialog is displayed.

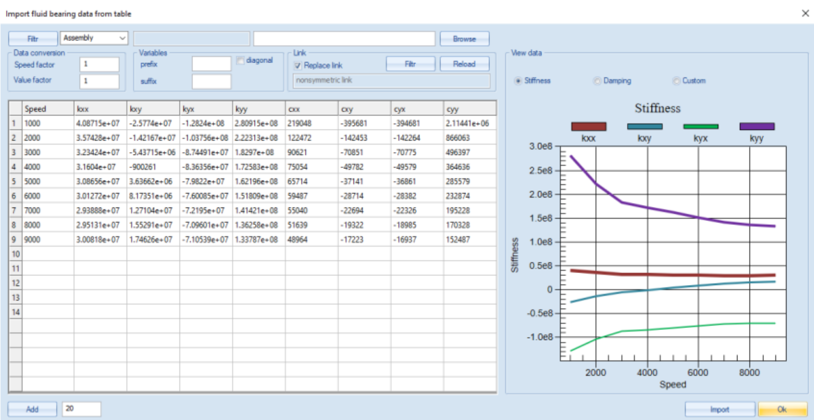


Figure 9.12

Sometimes it becomes necessary to calculate / check the system without modal link. Take into account the stiffness of the support in the basis, in this case the command of the context menu "Convert to symmetric Link" can be used indicating the mode for which the interpolated values of the stiffness and damping coefficients will be taken. In this case, the old link will be deleted and replaced with an element of the [Link] type. If you need to preserve the old link, then after conversion, copy the new link to the clipboard (Ctrl + C) and cancel the conversion of the modal link (Ctrl + Z). Paste link from clipboard. (Ctrl + V).

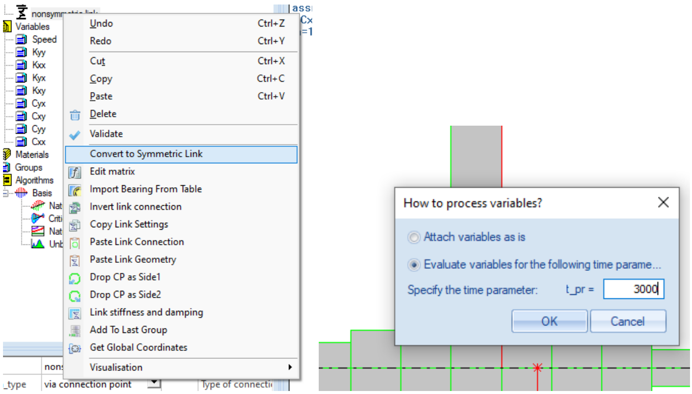


Figure 9.13

Parameter [Type] (Figure 9.9) defines the mode of Elastic non-symmetric link calculation. With [modal] type is conventional functionality. The link is taken into account as modal addition to reduced modal system of equations in this mode (in the left side of equation of motion). There is no functionality in program for automatic definition of reactions for such links in this mode. In order to calculate reactions user should multiply stiffness and damping matrixes by displacements and velocity columns. Link is taken into account in all algorithms exept [Basis] in this mode.

$$[K]*\{u\}+[C]*\{v\}=\{R\}$$

With [nonlinear] mode link is considered in the right part of equation as reaction. It is taken into account only in [Nonlinear analysis] in this mode. Reactions can be obtained in a similar way as in any other nonlinear links.
Figure 9.14 shows options for calculation optimization. With option “6x6” full stiffness and damping matrix are taken into account. With option 2x2 only 2 degrees of freedom (DOF) are taken into account. It is used for fluid bearings for example. In this case only Kxx, Kxy, Kyx, Kyy, Cxx, Cxy, Cyx, Cyy are used. With option “UtxUtyUtz” only Kxx, Kyy, Kzz, Cxx, Cyy, Czz are used.

Des	nonsymmetric link		Designation
conn_type	via connection point		Type of connection
side1_c_point	Connection point 23 2...		Side1 connection point
side2_c_point			Side2 connection point
stiff_matrix	...		stiff_matrix
damp_matrix	...		damp_matrix
Type	nonlinear		Modal or nonlinear link type
Type	2x2		Matrix content
misalignment switch	6x6		Use misalignment settings
d*	2x2 UtxUtyUtz	mm	Inner diameter
D*	300	mm	Outer diameter
B*	30	mm	width

Allows optimizing calculations by definition of the input content

Figure 9.14

Option [misalignment switch] (Figure 9.15) allows to define static preload of the support due to constant displacements of side2 vs side1 of the link (misalignment of bearing bore vs bearings centers line). Functionality can be used for supports alignment of heavy multi supports rotors due static deflection line. Intended for use in rotor systems with more than two supports. Also, it can be used as a spring preload along with angular contact ball bearing.

Des	nonsymmetric link		Designation
conn_type	via connection point		Type of connection
side1_c_point	Connection point 23 2...		Side1 connection point
side2_c_point			Side2 connection point
stiff_matrix	...		stiff_matrix
damp_matrix	...		damp_matrix
Type	nonlinear		Modal or nonlinear link type
Type	2x2		Matrix content
misalignment switch	yes		Use misalignment settings
ut_x	0	mm	Displacement in x direction
ut_y	0	mm	Displacement in y direction
ut_z	0	mm	Displacement in z direction
d*	0	mm	Inner diameter
D*	300	mm	Outer diameter
B*	30	mm	width

Use additional settings for definition of misalignment properties in the link. It can be used for definition static displacement between shaft and case in the bearing for example

Figure 9.15

9.4 [Gear set] element

[Gear set] element models any kinds of gears used in industry (spur gears, helical gears, conical gears, hypoid gears). Connection stiffness is defined by the contact stiffness in gear (the local flexibility of the teeth coupling and gear rim are not considered).

The type of a gear – cylindrical, conical or hypoid – is specified by the placement of subsystems in space. There should be correlation between subsystems location and common point at pitchlines of gears.

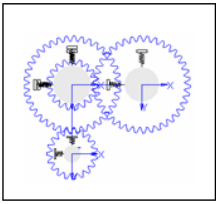
In order to determine the location and geometrical dimensions of a gear transmission a user should primarily choose the type of a gear.

The type of a gear is determined by **[helical]** or **[bevel]** parameters.

For straight-toothed and helical cylindrical pair the parameter **[helical]** having the following data is chosen, **Figure 9.16:**

- point of a gear connection to a shaft. Shafts are connected through nodal points **[Side1 connection point]** and **[Side1 connection point].** These points are taken from the common list that is formed beforehand with the **[Connection point]** element;
- gear type **[helical]** or **[bevel];**
- teeth coupling: external or internal;
- number of teeth in the leading **[z1]** and driven **[z2]** gears;
- the pitch of gear **[module]** being common to both wheels**;**
- the pressure angle **[alpha_wn] ;**
- the helix angle **[beta_w1] ;**
- the contact stiffness **[Kc]**;
- pressure line length **[b].**

Menu symbols	Input box
Gear set	



Des	Gear set 1 38/45		Designation
side1_c_point	CP3 gear z45		Side1 connection point
side2_c_point	CP8 gear z38		Side2 connection point
type_gearset	helical		Gearset type
gearing	outer		Gearing type
z1	45		Number of teeth
z2	38		Number of teeth
module	6	mm	Module
alpha_wn	20	deg	Operating pressure angle
beta_w1	0	deg	Working helix angle
Kc	1e+010	N/m^2	Contact stiffness
b	94	mm	Contact length

Figure 9.16

The parameter **[bevel]** having the following additional data is chosen for conical pair, **Figure 9.17**:

Des	Gear set 5		Designation
side1_c_point	Connection point 1		Side1 connection point
side2_c_point	Connection point 1		Side2 connection point
type_gearset	bevel		Gearset type
z1	50		Number of teeth
z2	50		Number of teeth
module	5	mm	Module
alpha_wn	20	deg	Operating pressure angle
beta_w1	0	deg	Working helix angle
beta_w2	0	deg	Working helix angle
delta1	45	deg	Pitch angle
delta2	45	deg	Pitch angle
Kc	1e+010	N/m^2	Контактная жесткость
b	60	mm	Контактная длина
C	0	s/m	Contact damping

Figure 9.17

- **working helix angle for leading gear [beta_w1]**
- **working helix angle for driven gear [beta_w2]**
- **pitch angle for leading gear [delta1]**
- **pitch angle for driven gear [delta2]**

The stiffness of pair gear is defined as contact specific stiffness **Kc ~1…2e10** N/m2.

> **NOTE:** Dimensions and inertia of the gear coupling is simulated with beam elements **[Beam]** or **[Generalized element]**

NOTE: When the gear dimensions are not correctly assigned, its image may also be wrong. So, the gear image may be used for the input check

NOTE: The gear ratio does not determine speed of the driven or driving shaft. A user should input the rotation speeds with the **[Kinematic joint]** element as preliminary

An example of a dynamic system with two gear transmission stages is shown in **Figure 9.18.**

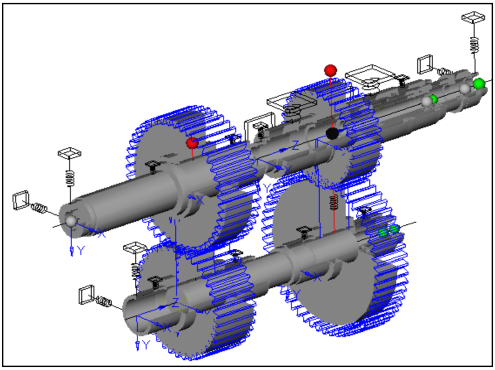


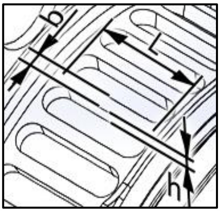
Figure 9.18

9.5 Elastic support [Squirrel Cage]

[Squirrel cage] – a special element in Dynamics R4 for simulating of radial stiffness squirrel cage elastic supports. This element allows to parameterize its parameters in tasks like **Critical speed maps** and **Natural Frequency maps**, **Figure 9.19**. Additional parameters for design purposes can be obtained also.

Menu symbols	Input box

Squirrel Cage



Des	Squirrel Cage 1			Designation
conn_type	via body			Type of connection
side1_subs	Shaft 1.Squirrel cage example			Side1 subsystem
side1_l	0	mm		side1 offset
side2_subs				Side2 subsystem
trns_exclude	No			Exclude from transient analysis
type	single			Squirrel cage type
Lb	75	mm		Squirrel cage bar length
b	4.01	mm		Squirrel cage bar width
h	4.5	mm		Squirrel cage bar height
n	96			Number of bars
E	2.1e+011	N/m2		Modulus of elasticity
Cs	0	N*s/m		Damping coefficient
	...			Squirrel Cage extended
add output	No			show additional output
Kr	1.72894e+008	N/m		Stiffness coefficient
d*	0	mm		Inner diameter
D*	100	mm		Outer diameter
B*	0	mm		width

Figure 9.19

The link may be represented by two types – **[single]** link and link of general type **[double].** Elastic properties of the link are assigned by radial stiffness and damping coefficients in X and Y direction.

For stiffness coefficient calculation the analytical formula is used, which is used in Damper R3.1 also. **Figure 9.20** 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

$$K = \frac{nEbh(b^2 + kh^2)}{2l^3},$$

where

n - number of bars;

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

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

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

Maximal stress in the bar

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

where

$$\varphi = \arctan\frac{b}{hk^{\frac{2}{3}}} + n\pi;\ n = 0 \text{ or } 1$$

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

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

Figure 9.20

In case of **[double]** type of **[Squirrel cage]** link overall stiffness is calculated as Kr=Ksc1*Ksc2/(Ksc1+Ksc2), where Ksc1 and Ksc2 are calculated for each part of squirrel cage.

In order to change parameters of each double Squirrel cage part [part#] combobox should be used, **Figure 9.21**.

type	double		Squirrel cage type
part #	first		Squirrel cage part properties
Lb	75	mm	Squirrel cage bar length
b	4.01	mm	Squirrel cage bar width
h	4.5	mm	Squirrel cage bar height
n	96		Number of bars
E	2.1e+011	N/m2	Modulus of elasticity

type	double		Squirrel cage type
part #	second		Squirrel cage part properties
Lb2	75	mm	Squirrel cage bar length
b2	4.2	mm	Squirrel cage bar width
h2	4.5	mm	Squirrel cage bar height
n2	96		Number of bars
E2	2.1e+011	N/m2	Modulus of elasticity

Figure 9.21

Yes/No combo box [add output] allows to show additional calculated parameters for the Squirrel cage, **Figure 9.22**. These parameters can be used in the squirrel cage design stage.

type	single		Squirrel cage type
Lb	75	mm	Squirrel cage bar length
b	4.01	mm	Squirrel cage bar width
h	4.5	mm	Squirrel cage bar height
n	96		Number of bars
E	2.1e+011	N/m2	Modulus of elasticity
Cs	0	N*s/m	Damping coefficient
	...		Squirrel Cage extended
add output	yes		show additional output
delta	0.1	mm	Radial clearance. For statical stress calculation
F	100	N	load in radial direction
ut	0.0144156	mm	Deflection in radial direction under load
max stress	62.0544	MPa	Maximal alterating stress. e=1
stress	4.56888	MPa	Stress due to load.
Fmax	693.691	N	Max load in radial direction. delta=ut
Kr	6.93691e+006	N/m	Stiffness coefficient

Figure 9.22

Additional features for design of squirrel cage can be obtained via **[Squirrel cage extended]** dialog. This dialog can be launched by double clicking on appropriate property (see **Figure 9.23**) or via **[Extended properties]** command in parameter context menu.

This dialog (**Figure 9.24**) allows checking the influence of input parameters on output characteristics like stiffness, stress, max loads, and deflection. Changing of one of input parameters is defined as % from the initial value. Vertical and horizontal red lines show the value of the original settings.

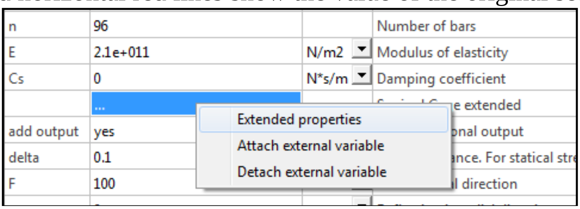


Figure 9.23

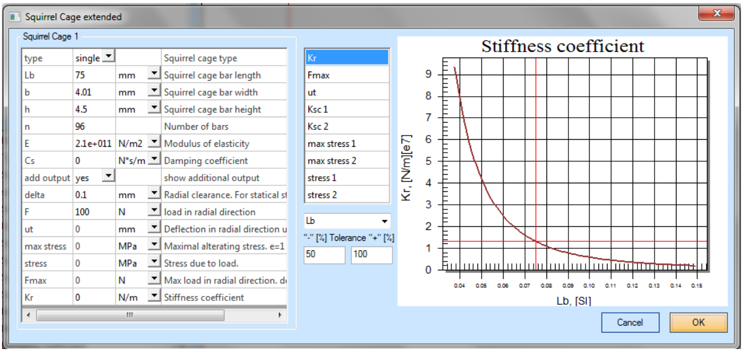


Figure 9.24

> **NOTE:** The link **[trns_exclude]** excludes only the stiffness part of a link in the unsteady analysis; the damping part is not excluded

9.6 [Connection point] element

The element assigns a subsystems' point which can be used for connection of subsystems with each other by means of any kinds of links**, Figure 9.25.**

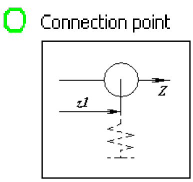
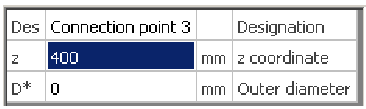
Menu symbols	**Input box**
	

Figure 9.25

Its location is determined by **Z** coordinate in a local coordinate system of the exact subsystem.

The boundary sections of links connect to assigned points by means of **[via connection points]** command. This command is chosen from links combo box **[conn_type]**.

10 STATIC AND DYNAMIC LOADS

To analyze dynamic response of a rotor system a user can choose the following elements:

- **[Unbalance load]**
- **[Axial force]**
- **[Blade loss]**
- **[Dynamic load]**

In the main menu of the Dynamics R4 program the elements are represented by the following symbols, **Figure 10.1.**

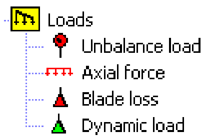
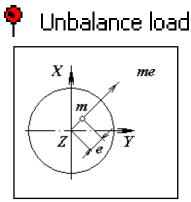
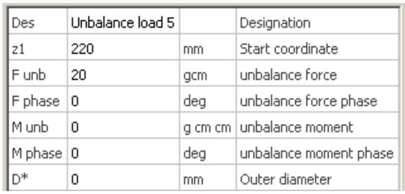


Figure 10.1

NOTE: All loads may be located in any point of a model excluding points inside disks and abstract elements. The loads can be located only in boundary sections of these elements.

10.1 [Unbalance load] element

The **[Unbalance load]** element models imbalances in rotating parts of dynamic systems, Figure 10.2.

Menu symbols	Input box
	

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	0	g cm cm	unbalance moment
M phase	0	deg	unbalance moment phase
D*	0	mm	Outer diameter

Figure 10.2

Table 10.1 Parameters description

Parameters	**Designation**	**Parameter description**
Start coordinate	**z1**	[z1] coordinate in the direction of Z axis, specifies the unbalance location in the XYZ local coordinate system
Linear unbalance	**unb F**	Value of linear imbalance (mass* eccentricity)
Unbalance phase	**phase F**	Phase of linear imbalance, defines angular displacement of unbalances between each other
Angle unbalance	**unb M**	Angle unbalance (mass moment of inertia * misalignment angle)
Angle unbalance phase	**phase M**	Phase of angle imbalance, defines displacement of annular unbalances between each other.

Figure 10.3 shows the sign rule for phase angles.

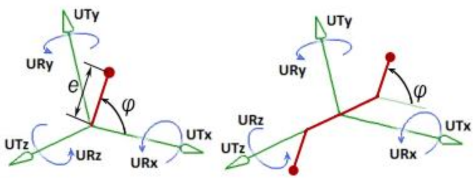


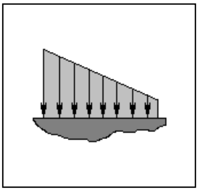
Figure 10.3

10.2 [Axial force] element

The element [Axial force] models axial force acting along axis Z of a subsystem. This load changes bending stiffness of a subsystem and leads to change of natural frequencies, **Figure 10.4** (this element is not included in shipment versions)

Menu symbols	Input box

Axial force



Point force

Des	Axial force 26		Designation
type	point		Load type
z1	802.05	mm	Start coordinate
Fz1	1800	N	force in Z direction at start coordinate

Constant value along Z axis

Des	Axial force 26		Designation
type	line(const)		Load type
z1	802.05	mm	Start coordinate
z2	1000	mm	End coordinate
Fz1	1800	N	force in Z direction at start coordinate

Variable value along Z axis

Des	Axial force 26		Designation
type	line(line)		Load type
z1	500	mm	Start coordinate
z2	800	mm	End coordinate
Fz1	2000	N	force in Z direction at start coordinate
Fz2	2000	N	force in Z direction at end coordinate

Figure 10.4

Table 10.2

Designation	**Parameter description**
z1	Coordinate [z1] in direction Z indicates the position of left side of element
z2	Coordinate [z2] in direction Z indicates the position of right side of element

type: **point** **line (const)** **line (line)**	Parameter defines type of axial load distribution along subsystem: • axial force in a point (acts only in this point); • axial force loading several segments of the subsystem (constant force along subsystem). Used for tie rods, for example; • axial force loading several segments of the subsystem (linear law of force change).
Fz	Axial force in Z direction. Compression or stretching is determined by a sign. "-" – compression load.

Significant changes in bending stiffness can be observed in case of axial loads values similar to Euler critical force of such rod compression in axial direction. There is a dialog (Figure 10.5) for calculation of Euler critical force.

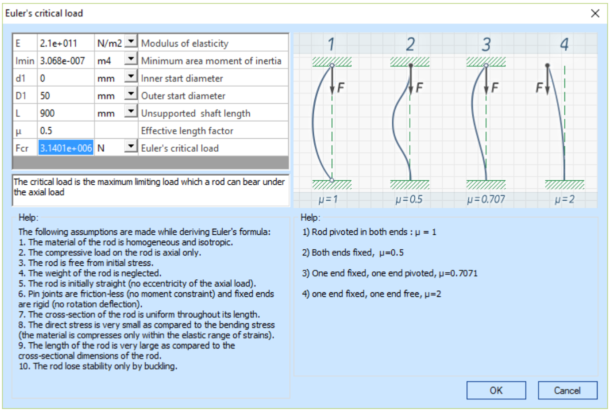
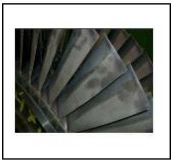


Figure 10.5

10.3 [Blade loss] element

The element models a sudden increase in imbalance of a rotor system (for example, due to blade loss), **Figure 10.6.** This element is used only in transient analysis.

Menu symbols	Input box
Blade loss	



Des	Blade loss 6		Designation
z1	309.615	mm	Start coordinate
D*	0	mm	Outer diameter
t_st_type	real		Start time definition
t_st_real	0	s	Real start time
F unb	0	gcm	unbalance force
F phase	0	deg	unbalance force phase

Figure 10.6

This element is used only in transient analysis.

Table 10.3

Designation	Parameter description
z1	Coordinate [z1] in direction Z indicates the position of sudden change of unbalance (for example, blade loss)
F unb	Linear unbalance appearing after blade loss
F phase	Phase of linear unbalance

The other parameters are presented in general description.

10.4 [Dynamic Load] element

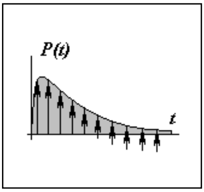
The element models different types of external excitation acting on a rotor system. There are the following loads, **Figure 10.7**:

- **[general] load**
- **[step] load**
- **[impulse] load**
- **[harmonic] load**
- **[unbalance] load**

The loads may be assigned at any degree of freedom. Simulation of polyharmonic loading is described by several elements.

Menu symbols	Input box

Dynamic load



- general

Des	External load 6		Designation
l1	0	mm	Start coordinate
D*	5	mm	diameter
Type	general		External load type
Fx	10	N	force in X direction
Fy	0	N	force in Y direction
Fz	0	N	force in Z direction
Mx	0	N m	moment in X direction
My	0	N m	moment in Y direction
Mz	0	N m	moment in Z direction

- step

Des	External load 6		Designation
l1	0	mm	Start coordinate
D*	5	mm	diameter
Type	step		External load type
t_st_type	real		Start time definition
t_st_real	3	s	Real start time
Fx	10	N	force in X direction
Fy	0	N	force in Y direction
Fz	0	N	force in Z direction
Mx	0	N m	moment in X direction
My	0	N m	moment in Y direction
Mz	0	N m	moment in Z direction

- impulse

Des	External load 6		Designation
l1	0	mm	Start coordinate
D*	5	mm	diameter
Type	impulse		External load type
t_st_type	real		Start time definition
t_st_real	3	s	Real start time
t_imp	0		impulse duration
Fx	10	N	force in X direction
Fy	0	N	force in Y direction
Fz	0	N	force in Z direction
Mx	0	N m	moment in X direction
My	0	N m	moment in Y direction
Mz	0	N m	moment in Z direction

- harmonic

Des	External load 6		Designation
l1	0	mm	Start coordinate
D*	5	mm	diameter
Type	harmonic		External load type
t_st_type	real		Start time definition
t_st_real	3	s	Real start time
Freq	3000	1/min	Frequency
init_phase	0	grad	initial phase
Fx	10	N	force in X direction
Fy	0	N	force in Y direction
Fz	0	N	force in Z direction
Mx	0	N m	moment in X direction
My	0	N m	moment in Y direction
Mz	0	N m	moment in Z direction

- unbalance

Des	Dynamic load 1		Designation
z1	0	mm	Start coordinate
D*	0	mm	Outer diameter
Type	unbalance		External load type
t_st_type	real		Start time definition
t_st_real	0	s	Real start time
rot_fctr	0.9		Rotation factor
F unb	100	gcm	unbalance force
F phase	0	deg	unbalance force phase
M unb	0	g cm cm	unbalance moment
M phase	0	deg	unbalance moment phas
Visibility	0		Element Visibility

Figure 10.7

There are some other parameters:
t_imp - duration of load impulse
Freq – frequency of harmonic load
Initialphase- initial phase of harmonic load.
This element is used only in transient analysis.

For types of loading General, Impulse, Step, loading is output in 2D plot window as an arrow with direction corresponding to the force components and inscription with a value.
This element is taken into consideration in the [Transient response] algorithm

10.5 [Torque Load] element

The [Torque load] element represents an external dynamic load applied to the shaft station. It is used for simulation of the dynamical variations of the torque load. For example, harmonic excitations can appear as a result of 2-phase, 3-phase short circuit in the electrical machines. Torque load is

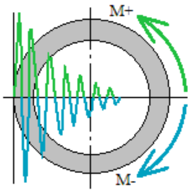
defined as a harmonic series. In order to simulate various behavior of dynamic torsion moment, a user can assign amplitude values for torque components, time constants and phases.
The following harmonic series is used to define the mathematical model of torque moment variations.

$$T = \begin{cases} T_{rated}[T_o e^{-a_0 t} + T_1 e^{-a_1 t}\sin(\omega t + \varphi_1) + T_2 e^{-a_2 t}\sin(2\omega t + \varphi_2)], t > t1 \\ T_{rated}, \quad t \le t1 \end{cases}$$

where T_{rated}, T_o, T_1, T_2, - the engine torque moment and coefficients for amplitude components of torque load variations., a_0, a_1, a_2 – time constants for decay process definition, φ_1, φ_2 – phase angles, ω – grid frequency.

Menu symbols

Torque load



Input box

Des	Torque load		Designation
z1	450	mm	Start coordinate
D*	0	mm	Outer diameter
t_st_type	real		Start time definition
t_st_real	1	s	Real start time
Trated	31831	N m	Steady state driving torqu
T0	0.837		Aperiodic torque compon
T1	9.234		First order harmonic torqu
T2	0.435		Second order harmonic tc
a0	14.7		Time constant of aperiodi
a1	13.8		Time constant of first ord
a2	13.8		Time constant of first ord
f	3600	1/min	Grid frequency
phase1	0	deg	Phase angle 1st harmonic
phase2	0	deg	Phase angle 2nd harmoni
Visibility	0		Element Visibility

Aperiodic component in definition of torque load excitation

Torque load T_{rated} acts during all integration time. This element is used only in transient analysis.

11 NONLINEAR LINKS

In the main menu of the Dynamics R4 program the [Nonlinear link] elements are represented by the following strings, **Figure 11.1**
.

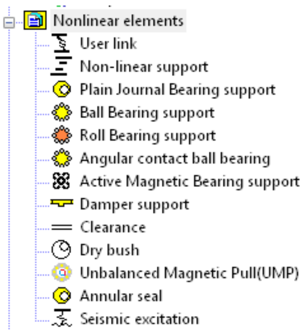


Figure 11.1

NOTE: Nonlinear links are available only for transient analysis. Linear analysis is not possible

NOTE: Distance and reciprocal turns between sections where a link is attached should be equal to zero. It means that both link sections should be in the same coordinate systems

11.1 User-programmable element [User link]

It allows working out the user's own algorithms of nonlinear effects in rotating machinery to program a new link between subsystems (journal bearing, magnetic bearing, etc.) and also to integrate them into the **Dynamics R4** program system, **Figure 11.2.**
The examples of such elements are a journal bearing of the other type, a magnetic support, a gas dynamic bearing, etc.

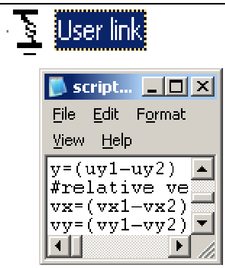
Menu symbols	Parameters window
	

Figure 11.2

The new element may be of any complexity and can be used in a rotor model in combination with other non-linear elements of **Dynamics R4.** The language of programming is the script language Python (www.python.org) built-in in the **Dynamics R4** program system and can be easily studied by a user. Example of a script window is shown at **Figure 11.3.**

NOTE: A new algorithm written on script language can be programmed by a developer of Dynamics R4 and included in a program system. It allows increasing the computation speed of a dynamic system including the new element in many times

NOTE: If you have your own algorithm, programmed on FORTRAN, C++, etc., please, contact us and you will receive the full information about including the new element in a program system

This element is used only in transient analysis.

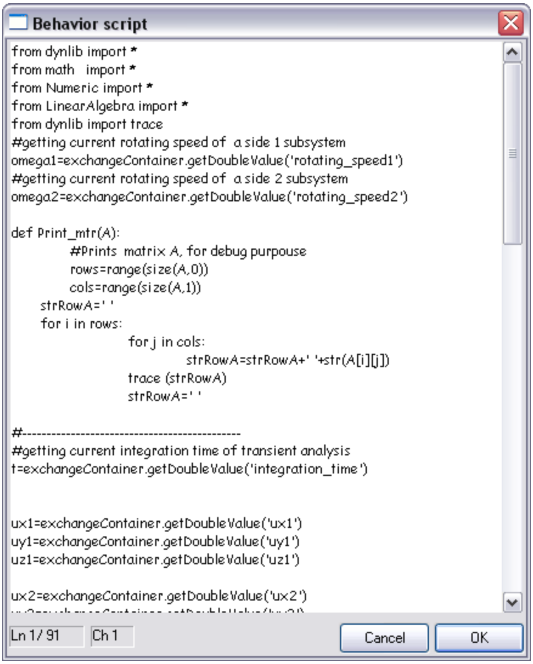


Figure 11.3

Table 11.1 Input data in User element

Link boundary section 1	Link boundary section 2
Linear displacements ux1;uy1; uz1 **Linear velocities** vx1; vy1; vz1 **Angle displacements** tx1; ty1; tz1 **Angle velocities** vtx1, vty1, vtz1	**Linear displacements** ux2; uy2; uz2 **Linear velocities** vx2; vy2; vz2 **Angle displacements** tx2; ty2; tz2 **Angle velocities** vtx2, vty2, vtz2

Among the input data there are also rotating speeds **omega1** and **omega2** of the subsystems connected by a link.
For further information on use of these parameters see section “Using Python in UserLink element”

Table 11.2 Output data in User element

Link boundary section 1	Link boundary section 2
Linear forces fx1; fy1; fz1 **Moments about axes** mx1; my1; mz1	**Linear forces** fx2; fy2; fz2 **Moments about axes** mx1; my1; mz1

11.2 Template script

A script template may be used for creation of the script element. A user can obtain the template of script program from the program main menu by double click of **[Template script]** command, and then paste it in the corresponding place of the model.

The **[Template script]** text is represented below, **Table 11.3**.

Table 11.3

```
from dynlib import *
from math   import *
from dynlib import trace

#getting current rotating speed of  a side 1 subsystem
omega1=exchangeContainer.getDoubleValue('rotating_speed1')
#getting current rotating speed of  a side 2 subsystem
omega2=exchangeContainer.getDoubleValue('rotating_speed2')

#--------------------------------------------
#getting current integration time of transient analysis
t=exchangeContainer.getDoubleValue('integration_time')

#getting translational  displacement
ux1=exchangeContainer.getDoubleValue('ux1')
uy1=exchangeContainer.getDoubleValue('uy1')
uz1=exchangeContainer.getDoubleValue('uz1')

ux2=exchangeContainer.getDoubleValue('ux2')
uy2=exchangeContainer.getDoubleValue('uy2')
uz2=exchangeContainer.getDoubleValue('uz2')

#getting translational  velocity
vx1=exchangeContainer.getDoubleValue('vx1')
vy1=exchangeContainer.getDoubleValue('vy1')
vz1=exchangeContainer.getDoubleValue('vz1')

vx2=exchangeContainer.getDoubleValue('vx2')
vy2=exchangeContainer.getDoubleValue('vy2')
vz2=exchangeContainer.getDoubleValue('vz2')
```

```
#getting rotational  displacement
Tx1=exchangeContainer.getDoubleValue('tx1')
Ty1=exchangeContainer.getDoubleValue('ty1')
Tz1=exchangeContainer.getDoubleValue('tz1')

Tx2=exchangeContainer.getDoubleValue('tx2')
Ty2=exchangeContainer.getDoubleValue('ty2')
Tz2=exchangeContainer.getDoubleValue('tz2')

#getting rotational velocity
Tx_dot1=exchangeContainer.getDoubleValue('vtx1')
Ty_dot1=exchangeContainer.getDoubleValue('vty1')
Tz_dot1=exchangeContainer.getDoubleValue('vtz1')
Tx_dot2=exchangeContainer.getDoubleValue('vtx2')
Ty_dot2=exchangeContainer.getDoubleValue('vty2')
Tz_dot2=exchangeContainer.getDoubleValue('vtz2')

#initializing variables for reactions

fx1=fx2=fy1=fy2=fz1=fz2=mx1=mx2=my1=my2=mz1=mz2=0
#--------------------------------------------------
#----------Place  your code here-----
#--------------------------------------------------
#----------------------------------------------------#
#returning calculated reactions to container
#force reactions
exchangeContainer.setDoubleValue('fx1',fx1)
exchangeContainer.setDoubleValue('fy1',fy1)
exchangeContainer.setDoubleValue('fz1',fz1)
exchangeContainer.setDoubleValue('fx2',fx2)
exchangeContainer.setDoubleValue('fy2',fy2)
exchangeContainer.setDoubleValue('fz2',fz2)

#moment reactions
exchangeContainer.setDoubleValue('mx1',mx1)
exchangeContainer.setDoubleValue('my1',my1)
exchangeContainer.setDoubleValue('mz1',mz1)
exchangeContainer.setDoubleValue('mx2',mx2)
exchangeContainer.setDoubleValue('my2',my2)
exchangeContainer.setDoubleValue('mz2',mz2)
```

Developers may also use additional math program modules developed for scientific computing and based on Python. **NumPy, LinearAlgebra**, etc. are among them. Additional modules, including graphics packages, are available for downloading via the Web.

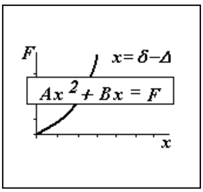
NOTE: A user may transmit a script element into the library of elements and algorithms. For this create a new folder in the **\Dynamics R4\Script\Python\UserElement** and copy the files into the folder. Files from the neighboring folders related to the script files should be changed. Editing may be hold with the help of a text editor such as **Notepad**

11.3 [Non-linear support] element

The element simulates a link between two subsystems that has nonlinear radial stiffness and damping, **Figure 11.4.**
The link may have clearance. The radial nonlinear stiffness after clearance's taking up is given by dependence between force and deformation which is described by $Ax^2+Bx=F(x)$ polynomial. Such link may simulate bearing support, for example, for which such dependence is obtained. A damping can also be considered. The element may also be used for modeling of the other types of reference nodes or links between a rotor and a case, a rotor and a rotor for which data about radial stiffness are given.

1

Menu symbols	Input box
Non-linear support	



Des	Non-linear support 5		Designation
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
side1_l	30	mm	side1 offset
side2_subs			Side2 subsystem
A	2.33e+012		Polynomial coefficient A
B	6.09e+007		Polynomial coefficient B
delta	0	mm	Radial clearance
Cb	1800	N*s*m	Damping coefficient
d*	0	mm	Inner diameter
D*	100	mm	Outer diameter
B*	50	mm	width

Figure 11.4

This element is used only in transient analysis.

11.4 [Plain journal bearing] element

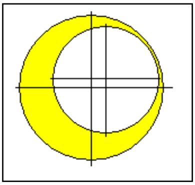
This element allows computing the rotating system with two notable cases of plain bearing: the "short" bearing (Open Ended) and the "long" bearing (Closed Ended), **Figure 11.5.**

Different kinds of cavitations of fluid films are considered: uncavitated (2π-film) bearing (Full Film) and cavitated (π-film) bearing (Half Film).

It allows computing dynamical systems in steady-state and non-steady state conditions. Inlet oil pressure may be considered.

Menu symbols	Input box

Plain Journal Bearing support



Des	Plain Journal Bearing support 1		Designation
conn_type	via body		Type of connection
side1_subs	output shaft z33		Side1 subsystem
side1_l	87.7	mm	side1 offset
side2_subs			Side2 subsystem
Type	short		Design type
R	50	mm	Bearing radius
film	2pi-film		Cavitation film model
Lr	109.5	mm	Bearing length
delta	0.12	mm	Radial clearance
mue	0.0072	Ns/m2	Lubricant absolute viscosity
P0	0	N/m2	Supply pressure to the bearing
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	50	mm	width

Figure 11.5

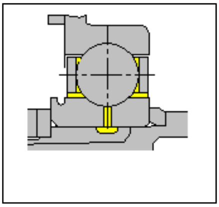
This element is used only in transient analysis.

11.5 [Ball bearing] element

The element is represented by a link between two subsystems. It models ball bearings mounted between a rotor and a stator/rotor, **Figure 11.6**.

The ball bearing model takes into account a radial clearance, number of rolling elements under radial load and contact stiffness in loading area. Damping can also be considered for a bearing. Contact stiffness in dependence on **Kp_input** parameter may be assigned or calculated automatically.

Menu symbols	Input box



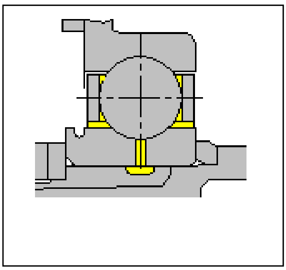
Des	Ball Bearing support 3		Designation
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
side1_l	1	mm	side1 offset
side2_subs	Subsystem 2		Side2 subsystem
side2_l	1	mm	side2 offset
Dr	11.9062	mm	Rolling elements diameter
z	11		Rolling elements count
Di	40	mm	Inner race diameter
deltai	0	mm	Inner race clearance
deltao	0	mm	Outer race clearance
Kp_input	input		Type of Hertz stiffness input
KH	1.334e+010	N/m	Hertz stiffness
Cb	2940	N*s/m	Damping coefficient
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Figure 11.6

This element is used only in transient analysis.

11.6 [Angular contact ball bearing] element

Menu symbols	Input box



Des	Ball Bearing 5DOF support		Designation
conn_type	via body		Type of connection
side1_subs	Вал		Side1 subsystem
side1_l	0	mm	side1 offset
side2_subs			Side2 subsystem
Dr	22.225	mm	Rolling elements diameter
z	18		Rolling elements count
Dm	164.93	mm	Pitch diameter (reference diameter)
Kp_input	input		Type of Hertz stiffness input
Kpi	0	N/m	Hertz contact stiffness of inner race
Kpe	0	N/m	Hertz contact stiffness of outer race
zp	0	mm	Axial distance from bearing rolling element centre
zq	0	mm	Axial distance from bearing rolling element centre
Rp	82.803	mm	Radial distance from bearing axial line to nominal
Rq	0	mm	Radial distance from the bearing axial line to nom
Ri	11.45	mm	Inner race radius of curvature
Ro	11.45	mm	Outer race radius of curvature
alfa_i	0	deg	Nominal contact angle in Ball bearing at inner rac
Phi0	90	deg	Angle position of the 1st rolling element
input_type	coordinates		Input parameters type
inertia	No		Calculate with inertia
conditions	both		Take into account temperature, mounting fits
	...		Temperature and mounting fits considerat
ext output	expert		Calculate intermediate parameters
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	15	mm	width

Figure 11.7

The element is represented by a link between two subsystems. It models a ball bearing mounted between a rotor and a stator/rotor**.**
The ball bearing model takes into account a radial clearance, curvature radiuses of inner and outer race, number of rolling elements under radial load, their inertia and contact stiffness in loading area.
Contact stiffness in dependence on the **Kp_input** parameter may be assigned or calculated automatically.
The system may be considered without balls inertia and with it. The element takes into account 5 degrees of freedom of inner race: 3 transitional and 2 rotational.

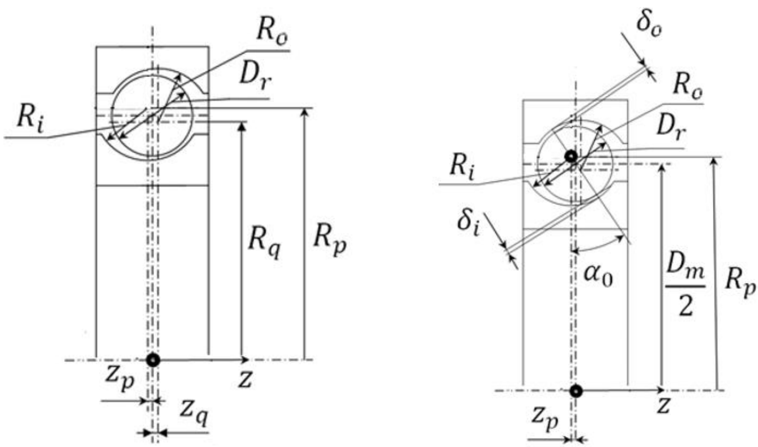


Figure 11.8

Table 11.4

Parameter	Description
Des	On default the variable has the text value **[Element_Number]**. The Number indicates the sequential number of the element in the rotor model. A user can change the name of element (for example, into subsystem name)
conn_type	The variable defines a type of connection of subsystems between each other - [via body] – the connection is defined without preliminary determination . Connection point is specified by the subsystem name and z coordinate; - [via connection point] – the connection is attached to the preliminary defined point with the given name. The point is set by the name of the subsystem to which it belongs and z coordinate (in the coordinate system that is local for the given subsystem). This type of the connection attachment is more convenient for editing and it is the only variant for creation multilevel out-of-alignment spaced subsystems
side1_subs	The variable indicates a subsystem for boundary section 1 of an elastic

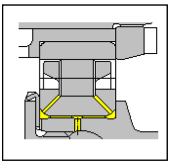
	link. On default the subsystem is a foundation (zero subsystem)
side1_I	The coordinate indicates position of an elastic link boundary section 1 in the subsystem
side2_subs	The variable indicates a subsystem for boundary section 2 of elastic link. On default the subsystem is foundation (zero subsystem)
side2_I	The coordinate indicates position of elastic link boundary section 1 in subsystem
Dr	Rolling elements diameter. It is used at contact stiffness calculation
z	Number of balls
Dm	Pitch diameter. Diameter of rolling elements center
deltai	Inner race clearance in contact direction. This parameter is available in [input]=clearance mode
delta0	Outer race clearance in contact direction. This parameter available in [input]=clearance mode
Kp_input	Type of Hertz stiffness input. It may be calculated according to the bearing parameters
Kpi	Hertz contact stiffness of inner race. This parameter is available at the [Kp_input] mode
Kpe	Hertz contact stiffness of outer race. This parameter is available at the [Kp_input] mode
zp	Axial distance from bearing rolling element centre to nominal inner race centre of curvature. Sign “-” means a contact point on the left ring part. This parameter is used in [input]=coordinates mode
zq	Axial distance from bearing rolling element centre to nominal outer race centre of curvature Sign “-” means a contact point on the left ring part. This parameter is used in [input]=coordinates mode
Rp	Radial distance from the bearing axial line to nominal inner race centre of curvature. This parameter is used in [input]=coordinates mode
Rq	Radial distance from the bearing axial line to nominal outer race centre of curvature. This parameter is used in the [input]=coordinates mode
Ri	Inner race radius of curvature. It is used at contact stiffness calculation
Ro	Outer race radius of curvature. It is used at contact stiffness calculation
alfa_i	Nominal contact angle in Ball bearing at inner race. This parameter available at the [input]=clearance mode
alfa_o	Nominal contact angle in Ball bearing at outer race. This parameter available at the [input]=clearance mode
Phi0	Angle position of the 1st rolling element. This parameter takes virtually no influence on the results in calculation with rotation. It may have small influence on the results in calculation of static radial loads
input_type	Input parameters type. It allows choosing different set of input data. The [coordinates] mode uses rings curvature centers coordinates for geometry definition. This is default mode, allows more precise calculations. The [clearance] mode uses clearances, contact angles, balls diameter for geometry definition
inertia	It allows choosing calculation with inertia of balls or without it
Mr	Ball element mass. This parameter is available at the [inertia]=yes mode
conditions	This list allows choosing different options. [none] - no temperature and mounting fits conditions consideration; [temperature] - only linear expansion is taken into account; [mounting fits] - only mounting conditions are considered; [both] - all parameters are in use

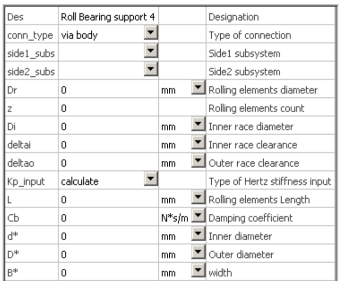
ext.output	When "Default" is selected, intermediate parameters for this nonlinear element will be calculated. When “Expert” is chosen, additional parameters will be shown as results of calculation of the current nonlinear element. Some parameters take additional calculations that may slow the process down. When "No" is selected, no additional parameters are presented
d*	The variable specifies the inner diameter of the element in the model. The size value is used only for model visualization
D*	The variable assigns the outer diameter of the element in the model. The size value is used only for model visualization
B*	The variable defines the width of link in the model. The size value is used only for the model visualization

11.7 [Roll bearing] element

The element is represented by a link between two subsystems. It models roll bearings mounted between a rotor and a stator (a rotor and a rotor). The roll bearing model takes into account a radial clearance, the number of rolling elements under radial load and contact stiffness in loading area. A damping also can be considered for a bearing, **Figure** 11.9.Figure 11.9This element is used only in transient analysis.

Menu symbols | **Input box**





Des	Roll Bearing support 4		Designation
conn_type	via body		Type of connection
side1_subs			Side1 subsystem
side2_subs			Side2 subsystem
Dr	0	mm	Rolling elements diameter
z	0		Rolling elements count
Di	0	mm	Inner race diameter
deltai	0	mm	Inner race clearance
deltao	0	mm	Outer race clearance
Kp_input	calculate		Type of Hertz stiffness input
L	0	mm	Rolling elements Length
Cb	0	N*s/m	Damping coefficient
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

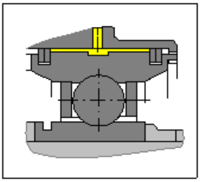
Figure 11.9

11.8 [Damper support] element

This element models two well-known types of hydrodynamic dampers: the "short" damper (without seal rings) and the "long" damper (with seal rings along the edges) and their combinations, **Figure 11.10**. Different kinds of fluid film cavitations are considered: uncavitated (2π-film) damper and cavitated (π-film) damper. There is laminar flow. It allows computing dynamical systems in steady-state and non-steady state conditions.
This element is used only in transient analysis.

Menu symbols

Damper support



Input box

Des	Damper support 2		Designation
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
side1_l	0	mm	side1 offset
side2_subs			Side2 subsystem
Type	short		Design type
R	75	mm	Damper radius
film	pi-film		Cavitation film model
Lr	37.5	mm	Squeeze film bearing land length
delta	0.225	mm	Radial clearance
mue	0.0172	Ns/m2	Lubricant absolute viscosity
P0	0	N/m2	Supply pressure to the damper
d*	0	mm	Inner diameter
D*	30	mm	Outer diameter
B*	10	mm	width

Figure 11.10

11.9 Elastic-damping constraint [Clearance]

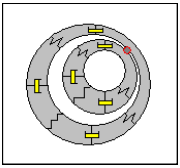
The element represents elastic-damper bending limiter. It allows simulating of systems with clearances, different types of contacts and dynamic characteristics of complex constructions of different reference nodes elements, **Figure 11.11**. In the model the element is represented by a link between two subsystems of general kind.

This allows modeling of most cases of instability in rotating systems with clearances. It takes into account any kinds of contact with external and internal damping, with friction in a contact point, weight and circumferential irregularity of clearance.

Menu symbols	Input box

Menu symbols

Clearance



Input box

Des	Clearance 2 2			Designation
conn_type	via body			Type of connection
side1_subs				Side1 subsystem
side2_subs	Subsystem 1			Side2 subsystem
side2_l	1	mm		side2 offset
L1	0	mm		Displacement L1
L2	0	mm		Displacement L2
L3	0	mm		Displacement L3
k1	1600	N/m		Damper package 1 stiffness
c1	0	N*s/m		Package 1 damping
k2	1e+010	N/m		Damper package 2 stiffness
c2	0	N*s/m		Package 2 damping
R	50	mm		Packet radius
delta	1	mm		Radial clearance
e1	0	mm		Assembly eccentricity 1
e2	0	mm		Assembly eccentricity 2
Teta1	0	deg		Initial angle position of e1
Teta2	0	deg		Initial angle position of e2
mue	0	Ns/m2		Friction coefficient
Vsmin	0.001			Minimal slipping speed
d*	0	mm		Inner diameter
D*	5	mm		Outer diameter
B*	0	mm		width

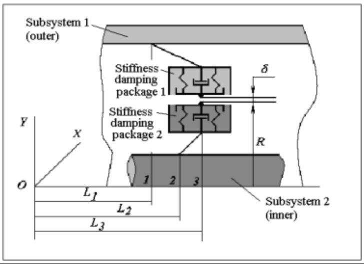


Figure 11.11

> **NOTE:** The **side_1** parameter should correspond to the outer subsystem, the **side_2** – to the inner one.

> **NOTE:** The element should be attached to both subsystems. The lack of fastening of one side (**side_1** or **side_2**) is not allowed.

Table 11.5 Element parameters

Designation	Description
L1	The coordinate indicates position of fixing point of package 1 in subsystem
L2	The coordinate indicates position of fixing point of package 2 in subsystem
L3	Coordinate of contact point. On default L1=L2=L3=0
k1	Damper package 1 stiffness
c1	Damping coefficient of outer damper package 1
k2	Damper package 2 stiffness
c2	Damping coefficient of inner damper package 2
R	Outer radius of inner damper package 2
delta	Radial clearance
e1	Initial assembly eccentricity of damper package 1
e2	Initial assembly eccentricity of damper package 2
Teta1	Initial phase angle of damper package 1
Teta2	Initial phase angle of damper package 2
mue	Coefficient of friction in a contact point
Vsmin	Minimal slipping speed in a contact point. Vsmin=0.001 m/sec

This element is used only in transient analysis.

11.10 [Dry bush] element

The element simulates dry bush support. Point or line contact type can be selected. In case of point contact this element can be used as fast substitution element for **Clearance** element, **Figure 11.12.**

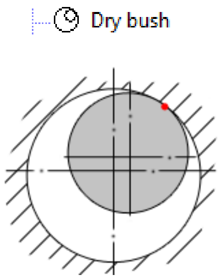
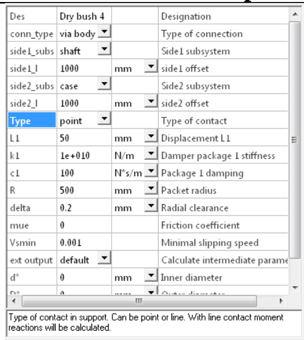
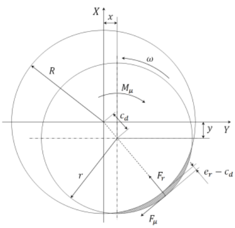
Menu symbols	Input box
	 

Figure 11.12

Table 11.6 Element parameters

Designation	Description
Type	Type of contact in support. Can be point or line. With line contact moment reactions will be calculated.
L1	The coordinate indicates position of fixing point of package 1 in subsystem
k1	Stiffness coefficient of outer damper package 1
c1	Damping coefficient of outer damper package 1
R	Outer radius of inner damper package 2
delta	Radial clearance
mue	Coefficient of friction in a contact point
Vsmin	Minimal slipping speed in a contact point.

11.11 [Active magnetic bearing support] element

The element is represented by a link between two subsystems. It models active magnetic bearing (AMB), **Figure 11.15**. There are three main types of AMB which are modeled: radial, cone and axial. Two different input types are possible - **[standard]** (common parameters set) and **[expert]**. Expert parameter input is used for simulation of special cases and requires knowledge of the algorithm of magnetic bearings. The model takes into account: radial clearance, maximal current and current density, electromagnetic poles number, control poles number, coil remove option. The control pole is the axis that coincides with direction of the resultant force from the closest to it electromagnet. Current to these electromagnets is calculated from the displacements along the axis of the pole control. Electromagnets are evenly distributed at the control poles, so the number of electromagnetic poles in the support must be multiple of the control poles number, **Figure 11.14**. Two control options exist - **[PD – controller]** and **[PID – controller]**.

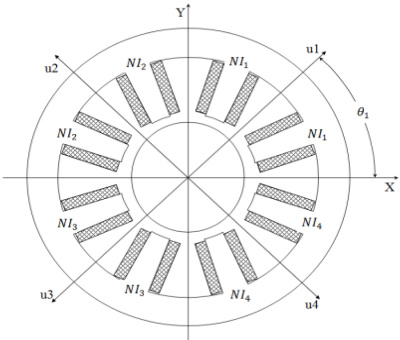


Figure 11.13

Figure 11.13 shows elements with main geometrical parameters.
The maximum allowable current in the coil is saturation current. Minimum number of electromagnetic poles and the poles of control is 3.

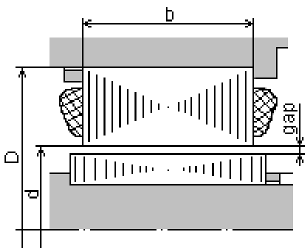
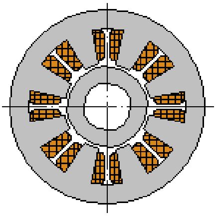


Figure 11.14

Menu symbols	Parameters window
Active Magnetic Bearing support	



Des	Active Magnetic Bearing support 2 3 2		Designation
conn_type	via body		Type of connection
side1_subs	Shaft 1		Side1 subsystem
side1_l	1000	mm	side1 offset
side2_subs			Side2 subsystem
Type	Isotropic		Link type
Opt.Type	standard		Options level type
Geometry Type	Cone		Geometry bearing configuration type
d1	260	mm	Inner start diameter
d2	188	mm	Inner end diameter
D	280	mm	Outer diameter
b	55	mm	magnetically active part width
Imax	5	A	Saturation current
Np	16		Number of poles
Phi_P1	0	deg	Angle position of the pole N1
Ncp	4		Number of control poles
Teta_CP1	0	deg	Angle position of the control pole N1
Flux direction	removable_coils		magnetic flux direction
Flux direction	heteropolar		magnetic flux direction
Jmax	6	A/mm2	Maximum Current Density
Controller	PID controller		Controller type
Kp	100000		Proportional part of the PID-controller
Ki	1.7e+007		Integral part of the PID-controller
Kd	1000		Differential part of the PID-controller

Figure 11.15

Control parameters K_p, K_i, K_d correspond to the following:

$$I = K_p u(t) + K_i \int_0^t u(t)dt + K_d \frac{du(t)}{dt}$$

where: I – current, fed control pole, u(t) - displacement in control pole plane, t – current time, K_p, K_i, K_d - proportional, integral and differential coefficients.

PD – controller realize proportional – differential control law.

PID – controller realize proportional – integral – differential control law.

This element is used only in transient analysis.

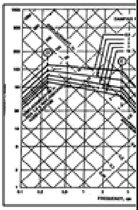
NOTE: Input parameters type (**[Isotropic]** or **[Full]**) and direction of magnetic flux options are not supported in Dynamics R4.6.

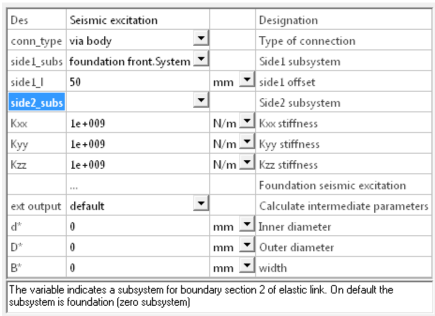
11.12 Seismic excitation [Seismic excitation]

At the **[Transient response]** algorithm calculation, the link allows modeling of kinematic excitation of the installation foundation and transient processes appearing at it. One of the frequent cases is the installation work under earthquake conditions. The element allows adjusting stiffness of the basement link with the installation in three directions and assigning spectrum of seismic excitation response from the part of the basement. Excitation is defined as a polyharmonic signal acting simultaneously along three directions. Time range of excitation is divided into several ones. They are the ranges of divergent, constant and decaying amplitudes.

Menu symbols	**Parameters window**

Seismic excitation





Des	Seismic excitation		Designation
conn_type	via body		Type of connection
side1_subs	foundation front.System		Side1 subsystem
side1_l	50	mm	side1 offset
side2_subs			Side2 subsystem
Kxx	1e+009	N/m	Kxx stiffness
Kyy	1e+009	N/m	Kyy stiffness
Kzz	1e+009	N/m	Kzz stiffness
	...		Foundation seismic excitation
ext output	default		Calculate intermediate parameters
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

The variable indicates a subsystem for boundary section 2 of elastic link. On default the subsystem is foundation (zero subsystem)

Figure 11.16

Stiffness of the model link with the basement is assigned by the stiffness factors Kxx, Kyy, Kzz.

NOTE: To calculate the **[Basis]** algorithm, presence of linear links between installation and basement is necessary. Stiffness for displacement is defined as relatively small (in comparison with stiffness in non-linear element). Freedom degrees of rotation angles may be fixed by significant moment stiffness (1e10-1e11 N/m)

To adjust the parameters of seismic excitation it is necessary to call the dialog of extended setting by double click at «…» () or by **“Extended properties”** command in the context menu.

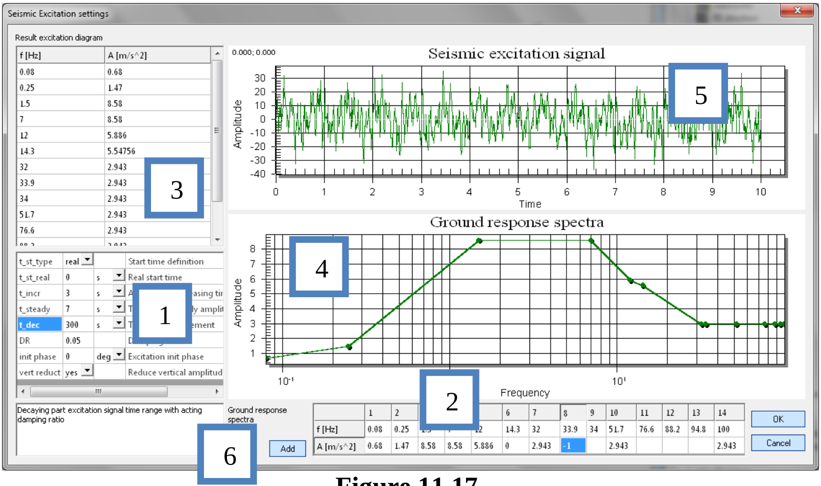


Figure 11.17

Figure 11.17 shows dialog of setting seismic excitation influencing the basement. The following control elements are shown using figures:

1. The table with the characteristics of time ranges of seismic excitation. Figure 11.18 shows different stages of seismic excitation– areas of divergent, constant and decaying amplitudes of the input excitation. When the time range t_dec is over, seismic excitation is switched off. It is recommended to set this value beyond integration time.
2. The table of frequencies and amplitudes of harmonic components which determine the response spectrum at seismic excitation. The significant points of the diagram of response spectrum should be assigned. If the additional harmonics (such as those corresponding to the natural frequencies of design) are required to be defined, then negative number may be put in the sell for amplitude or the sell may remain empty. In this case amplitude values for these frequencies will be interpolated according to the given significant points. If one of harmonic components should be switched off temporarily then zero may be put in the amplitude sell.
3. The resultant table of the response spectrum with the interpolated amplitude values.

4. Response spectrum presented as amplitude-frequency characteristic.
5. Amplitude-time characteristic of polyharmonic signal for the area of constant amplitudes.
6. The button "Add" is for adding the additional harmonic components into response spectrum from seismic excitation.

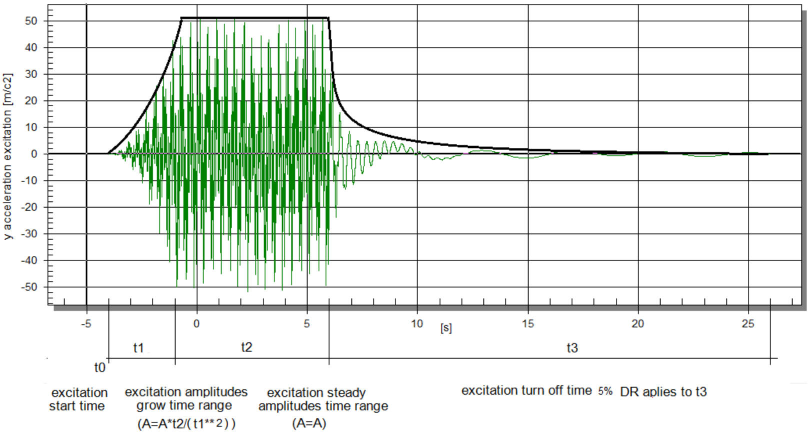
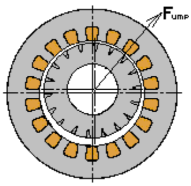
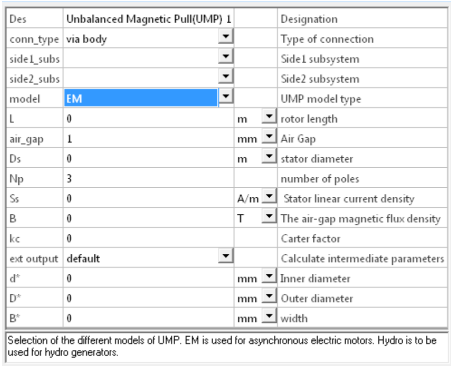


Figure 11.18

11.13 Unbalanced magnetic pull

It is the link modeling the effect of unbalanced magnetic pull (UMP) in electric machines. In case of appearance of air clearance irregularity between a rotor and a stator, as a result of static and dynamic rotor eccentricity, electromagnetic field provokes a one-sided radial force applied to the rotor centre and directed at the part of the minimum air clearance. UMB works to increase the rotor stiffness. Unbalanced magnetic pull is inherent in hydro generators and electric motors.

Menu symbols	Parameters window
	

Des	Unbalanced Magnetic Pull(UMP) 1		Designation
conn_type	via body		Type of connection
side1_subs			Side1 subsystem
side2_subs			Side2 subsystem
model	EM		UMP model type
L	0	m	rotor length
air_gap	1	mm	Air Gap
Ds	0	m	stator diameter
Np	3		number of poles
Ss	0	A/m	Stator linear current density
B	0	T	The air-gap magnetic flux density
kc	0		Carter factor
ext output	default		Calculate intermediate parameters
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Selection of the different models of UMP. EM is used for asynchronous electric motors. Hydro is to be used for hydro generators.

Figure 11.19

Table 11.7

model	Choice of the model type for UMP calculation. EM is used for asynchronous electric machines. Hydro is used for multipolar machines such as generators for hydro turbines
L	Length of the rotor part with winding
air_gap	Nominal air clearance between rotor and stator
Ds	Inner stator diameter
Np	Number of poles
Ss	Linear current density in winding
B	Nominal value of magnetic field induction in air clearance
kc	Carter factor (Factor of air clearance) may be obtained as ratio of maximum magnetic induction in clearance to the middle one

Figure 11.20 shows interaction between the models of the rotor system and UMP model while integrating

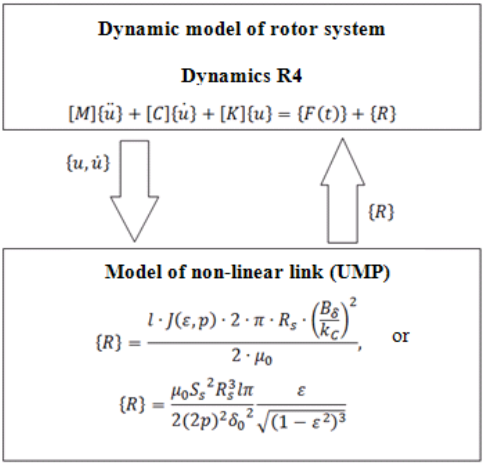
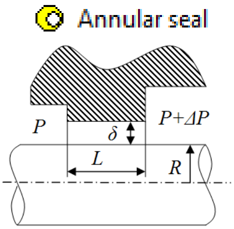


Figure 11.20

In case of UMP modeling for asynchronous machine, the first equation is used. In case of a hydro generator the second equation is used. It is attributed to some peculiarities of every type of machines.

11.14 [Annular seal] element

This element models interaction between a rotor and a stator in case of incompressible fluid with constant viscous flowing through small clearance under pressure - **Annular seal** (**Figure 11.21**). The mathematical model of this element takes into account: the algorithm which describes the bulk flow with centering force (Lomakin effect), inlet whirl factor, entrance loss factor. The mean axial velocity is defined by an iterative process. The rotordynamic coefficients are calculated in the centered rotor position.

Menu symbols	Input box
	see table below

Des	Annular seal 1		Designation
conn_type	via body		Type of connection
side1_subs			Side1 subsystem
side2_subs			Side2 subsystem
R	120	mm	Annular seal radius
L	8	mm	Annular seal length
delta	0.25	mm	Radial clearance
dP	5e+006	N/m2	Pressure drop
mue	0.000183	Ns/m2	Dynamic viscosity
rho	1000	kg/m3	Fluid density
alpha	0		Inlet whirl factor
zeta	0.1		Entrance loss factor
ext output	default		Calculate intermediate parameters
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Figure 11.21

11.15 Calculation of stiffness and damping in non-linear elements

If the extended choice **[expert]** is defined in parameters of a non-linear element, the coefficients of stiffness and damping matrixes are calculated additionally in correspondence with the freedom degrees of this element. These coefficients are calculated by numerical differentiation that leads to slowdown of total time of the model calculation.

> **NOTE:** Calculation of stiffness and damping for the **[Dry bush]**, **[Clearance]** elements is not provided.

Figure 11.22 presents an example of stiffness and damping coefficients output for a journal bearing.

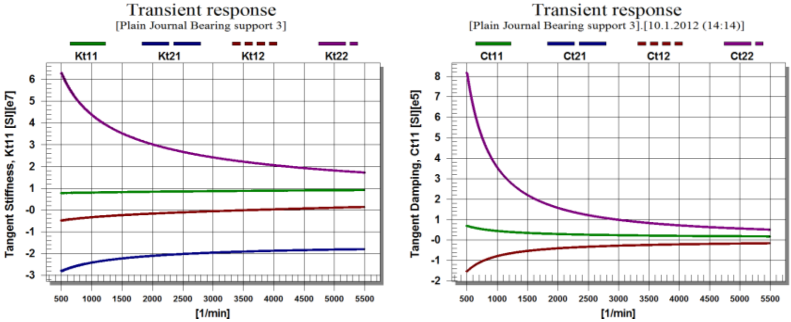


Figure 11.22

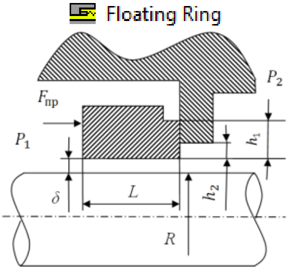
Factors numeration corresponds to the cells numeration in the 6x6 matrix, where the first value corresponds to the row, the second one-to the column, starting from 1. Thus, cross factor Kyx will be noted as Kt21, and moment stiffness along y axis at loading along x axis - Kt15.

11.16 [Floating ring seal] element

This element models an interaction between rotor and stator in case of incompressible fluid with constant viscous flowing between small clearance under pressure - **Annular seal**, **Figure 11.23**.

The mathematical model of this element takes into account: the algorithm which describes the bulk flow with centering force (Lomakin effect), inlet whirl factor, entrance loss factor. The mean axial velocity is defined by an iterative process. The rotordynamic coefficients are calculated in the centered rotor position. At the results output you should consider: Kxx=Kyy, Kyx=-Kxy, Cxx=Cyy, Cyx=-Cxy.

Menu symbols	Input box



Des	Floating Ring 1		Designation
conn_type	via body		Type of connection
side1_subs			Side1 subsystem
side2_subs			Side2 subsystem
h	13	mm	Floating ring thickness
R	120	mm	Seal shaft radius
L	8	mm	Length of the Floating ri
delta	0.25	mm	Radial clearance. Half dia
dP	5e+006	N/m2	Pressure drop
mue	0.000183	Ns/m2	Dynamic viscosity
mue_fr	0.1		Coefficient of friction bet
Fpr	0	N	Pressing force of the sprii
rho	1000	kg/m3	Fluid density
alpha	0		Inlet whirl factor
zeta	0.1		Entrance loss factor
r_r	0		Mean surface roughness
r_s	0		Mean surface roughness
ext output	default		Calculate intermediate pa

Figure 11.23

NOTE:

The ring is considered to be ideal without any diffuser or confuser in terms of its geometry.

Clearance nonequality between the ring and the shaft, shape deviation of the seal surfaces in longitudinal and cross-section direction are not taken into account.

Change in friction force at change in contact area between the ring and the shaft are not considered in case of the ring movement.

Rotordynamic coefficients are applied for the central rotor position.

Fluid is incompressible and homogeneously viscous.

NOTE: Side 1 – rotor, side 2 – case.

Table 11.1. gives list of expert parameters

Table 11.1

Parameter	Description
Delta_Pressure	pressure difference at regimes of the seal operation, Pa
Delta_ux, Delta_uy	clearance change between the ring and the rotor, mm
Ring_u_x, Ring_u_y, Rotor_u_x, Rotor_u_y	displacements of the ring and the rotor relatively the case, used to build motion orbits, mm.
e_ring	absolute value of displacement vector (eccentricity)of the ring, mm
vec_FFrictionRest, vec_FFrictionSlip	absolute value of friction and sliding force, N
vec_FHudrodyn	absolute value of hydrodynamic force, N
vec_FInertia	absolute value of force making the ring move, N
vec_vel_ring	absolute value of speed vector of the ring, mm/s

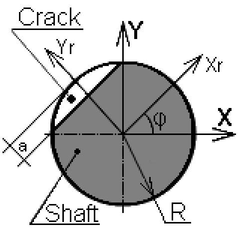
11.17 [Crack] element

The element represents link between two rod subsystems, the shaft parts separated by crack (Figure 11.24
). The element simulates change in local moment stiffness at the crack section while the shaft is rotating.

The "Crack" element is used together with the basis linear link whose moment stiffness along X and Y axes is equal to minimum moment stiffness of open crack $\mathrm{k}^{\mathrm{init}}\mathrm{min} = \mathrm{k}_{\mathrm{rx}}^{init}$. The "Crack" element reproduces cyclic increase and decrease in stiffness while the rotor is rotating relatively to $\mathrm{k}_{\mathrm{rx}}^{init}$.

Menu symbols	Parameters window

Crack



Des	Nonlinear Crack		Designation
conn_type	via connection point		Type of connectio
side1_c_point	Connection point 1.Shaft 1.System		Side1 connection p
side2_c_point	Connection point 2.Shaft 2.System		Side2 connection p
model	breathing		Crack model
Kr_input	calculate		Calculate or input
a	3	mm	Crack depth
D	10	mm	Diameter of shaft s
teta	0	deg	Crack position spir
Nue	0.3		Poisson's ratio
E	2.1e+011	N/m2	Modulus of elastic
Kr basis	15244.3	N*m	From basis Link: ro
Kr calc	15244.3	N*m	For Basis:Place this
ext output	expert		Calculate intermed
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Figure 11.24

The element's reactions are calculated in the following way:

$$\begin{Bmatrix} M_x \\ M_y \end{Bmatrix} = \begin{bmatrix} k_{rx}(\theta)C_1{}^2 + k_{ry}(\theta)S_1{}^2 - k_{rx}^{init} & k_{rx}(\theta)C_1S_1 - k_{ry}(\theta)C_1S_1 \\ k_{rx}(\theta)C_1S_1 - k_{rx}(\theta)C_1S_1 & k_{rx}(\theta)C_1{}^2 + k_{rx}(\theta)S_1{}^2 - k_{rx}^{init} \end{bmatrix} \cdot \begin{Bmatrix} u_{rx} \\ u_{ry} \end{Bmatrix},$$

$$k_{rx}(\theta) = \frac{2 \cdot k_{rx}^{init}}{(1 - \cos(\theta))},$$

$$k_{ry}(\theta) = \frac{2 \cdot k_{ry}^{init}}{(1 - \cos(\theta))}, \text{where}$$

$k_r(\theta)$ – coefficient of moment stiffness in rotating coordinate system,
θ - angle between the shaft rotation phase and precession phase,
k_r^{init} – initial value of the moment stiffness coefficient, corresponds to open crack,
u_r – the section rotation,
M – bending moment.

The element gives an opportunity to simulate constantly open crack, in this case stiffness in rotating coordinate system remains constant. Initial stiffness values are calculated automatically for the set of given parameters. Initial coefficients of moment stiffness can be defined by a user manually.

At output in the "expert" regime, the output of additional parameters takes place:

- Krx [N*m/rad] – local moment stiffness in crack section around Xr in the coordinate system rotating together with the shaft.
- Kry [N*m/rad] – local moment stiffness in the crack section around Yr axis in the coordinate system rotating together with the shaft.
- Kfx [N*m/rad] – local moment stiffness in the crack section along X axis in global coordinate system.
- Kfy [N*m/rad] – local moment stiffness in the crack section along Y axis in global coordinate system.
- Kfxy, Kfyx [N*m/rad] – local cross moment stiffness in crack section in global coordinate system.

NOTE

1 If stiffness of basis link is unknown beforehand, only geometrical crack parameters are given, it is necessary to calculate one point at transient response of the system when basis link has zero stiffness matrix. In the window of time signal output you should choose output of inner parameters of the "Crack" element. Then you should copy stiffness coefficient _Krx from signal and paste it as moment stiffness coefficients along X and Y axes in stiffness matrix of basis link. If moment stiffness of basis link is known beforehand, it is necessary to input the values into the corresponding interface field of the "Crack" element.

2 The element should be always fixed to two subsystems. It is not allowed not to fix it from one of sides (**side_1** or **side_2**).

3 In case of manual assigning of initial coefficients of the crack moment stiffness it is necessary to define coefficients in correspondence to the coordinate system shown in the picture.

4 With decrease in crack size, error of automatic calculation of initial stiffness increases.

11.18 [Shaft misalignment] link

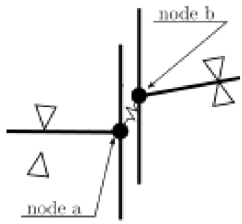
The element models a link between two rotor subsystems (the shaft parts) coupled together with misalignment (see Figure 11.25). The element "Shaft misalignment" reproduce cyclic variation of the coupling stiffness matrix during rotation accounting for relative parallel and angular misalignment.

Table 11.2

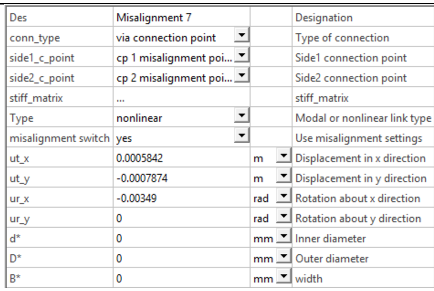
Parameter	Description
stiff_matrix	Stiffness matrix 6x6 of the coupling without misalignment
ut_x, ut_y	Radial displacement of the node "b" relative to the node "a"
ur_x, ur_y	Rotation of the node "b" relative to the node "a"

Menu symbol

Misalignment (shaft)



Parameters window



Des	Misalignment 7		Designation
conn_type	via connection point		Type of connection
side1_c_point	cp 1 misalignment poi...		Side1 connection point
side2_c_point	cp 2 misalignment poi...		Side2 connection point
stiff_matrix	...		stiff_matrix
Type	nonlinear		Modal or nonlinear link type
misalignment switch	yes		Use misalignment settings
ut_x	0.0005842	m	Displacement in x direction
ut_y	-0.0007874	m	Displacement in y direction
ur_x	-0.00349	rad	Rotation about x direction
ur_y	0	rad	Rotation about y direction
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Figure 11.25

Note: Foundation of the Dynamics R4 algorithms is the small deformations assumption, e.g. the model nodes never move out of their initial planes attached to the rotor system cross sections. Therefore, the element "Shaft misalignment" is capable to reproduce small misalignment and is not applicable for modeling of spatially rotating frames.

Note: Node "a" of the element "Shaft misalignment" have to be always connected to the left subsystem, while node "b" - to the right one. Both nodes "a" and "b" must be located at the same cross section of the rotor system, e. g. the axial distance between them has to be 0.

Note: Equal rotation speeds have to be specified for each of the subsystems connected by the element "Shaft misalignment".

12 GROUPS AND VARIABLES

12.1 Group of elements [Group]

The element is used for variant calculations, Figure 12.1.

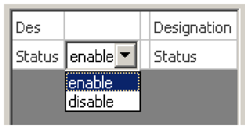
Menu symbols	Input box
	

Figure 12.1

A user can select some elements of the rotor model and exclude one or several groups from them. Each group has an appointed status **[enable]** or **[disable].** Subject to that status the group elements will be used or not used in the rotor model when calculation is carried out.

> **NOTE:** In group user can include only elements with zero length (linear and nonlinear links, masses, couplings, etc.)

Procedure to create a group of elements is presented below:

- Start the project
- Add new subsystems elements that will be included in the group
- Double click on the element [**Group]** to create the element [**Group]** in the project. The first group will be named as [**Group 1].** The second one – as [**Group 2],** etc.
- Select the subsystems elements and drag them into the field of the element **[Group 1]** or the other group (**"drag and drop"** Windows technology)

- Assign the status of [**Group …]** element, **[Enable]** or **[Disable]**

NOTE: The elements of all groups are output in the project model screen regardless of the group status

NOTE: Any element may be placed into the group, except elements with length

When dragging the point element or link into the group, the model tree is renewed and collapses. This may be inconvenient in case of adding several lengths less elements or links into the same group in the nested assemblies for example. The command **[Add To Last Group]** may be used after any element addition into desired group using dragging. When addition of all elements is finished, the key **F5** should be pressed to refresh the model tree, **Figure 12.2.**

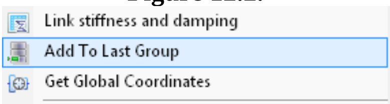


Figure 12.2

12.2 [Variables]

The program includes two variables types **[Value variable], [String variable]**, **Figure 12.3.**

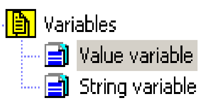
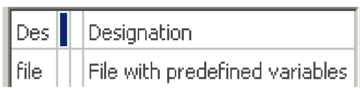
Menu symbols	Input box
	

Figure 12.3

The element on default always exists in project file.

At double click by left mouse button on the field with the name of the parameter, which the variable is attached to, the window of the linked variable characteristics will be activated, **Figure 12.4**. At double click on the name of the link attachment field, switch to the nearest element in the subsystem takes place.

After this you can switch to the previous active element by pressing key combination **[ctrl+shift+z]**.

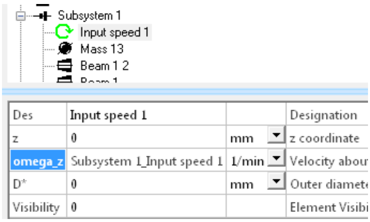


Figure 12.4

12.2.1 Value variable

The **[Value variable]** is used in the following actions:

- to define a variable value irrespective to time that may be assigned to a few parameters of the dynamic system;
- to specify a value of a time dependent variable that may be assigned to a few parameters of the dynamic system.

Rotation speed, support stiffness, support location, dimension or mass parameters may be among the parameters.
The **[String variable]** is used in creation of text variables and in user-programmable elements. For example, a script algorithm may be connected to a few equal non-linear links. This allows editing a script algorithm in one place.
The variables [**Value variable], [String variable]** are moved to a project file by double click of mouse left button.

The **[Value variable]** input is carried out by the following steps:

- Using cursor, select the **[Value variable]** line in the project and designate a name of variable (for example, **[Subsystem 1_Input speed 1], Figure 12.5**;

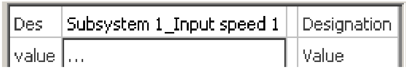


Figure 12.5

- Using cursor and right button of a mouse, a user opens the list of the following commands, **Figure 12.6;**

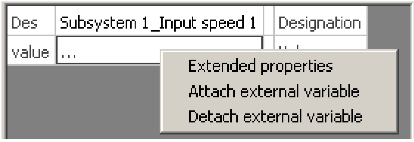


Figure 12.6

- Using command **[Extended properties],** a user inputs parameter value versus time parameter, **Figure 12.7.** Also it is possible to obtain the interpolated values of the function and the argument.

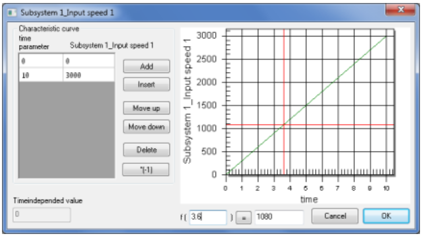


Figure 12.7

NOTE: Pay attention to the following - [**Time independed value]** has a value that is used for computation of system basis. On default the **[Time independed value]** is zero. User can change this value for any other

NOTE: In any algorithm variables are extrapolated beyond the defined utmost values. Only in linear algorithms rotating speed is not extrapolated, and calculation is carried out up to t_pr corresponding to the maximum value of the speed variable

- A user closes window with **[OK]** and a variable automatically becomes an external variable with the name **[Subsystem 1_Input speed 1**;

- A user selects the variable parameter (for example, omega_z) (**Figure 12.8**) and assigns an external variable to the variable parameter by means of the [Attach external variable] command, **Figure 12.9**;

Des	Input speed 1		Designation
z	450	mm	z coordinate
omega_z	0	1/min	Velocity about z direction
D*	0	mm	Outer diameter

Figure 12.8

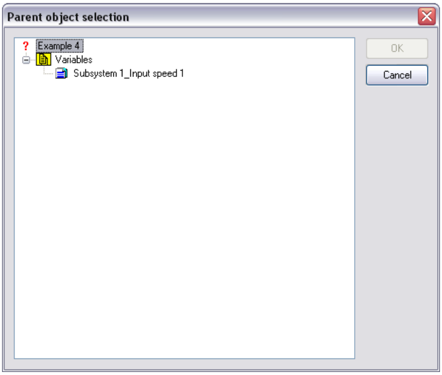


Figure 12.9

12.2.2 String variable

The [**String variable]** is used to form text variables. The element is often used for user-programmable elements. For example, one script algorithm may be attached to several identical non-linear links. This allows editing of the script algorithm only in one place and once.

Building of the [**String variable]** is carried out by the following steps:

- Using cursor, select the **[String variable]** line in the project and designate a name of text variable (for example, **[Rolling bearing script]**), **Figure 12.10**;

Des	Rolling bearing script	Designation
value		Value

Figure 12.10

- Use the **[Extended properties]** command for input of a text variable, **Figure 12.11;**
-

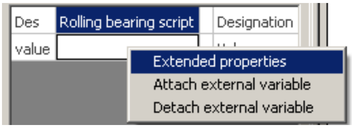


Figure 12.11

- When this command is completed, the text variable input window appears there, **Figure 12.12;**

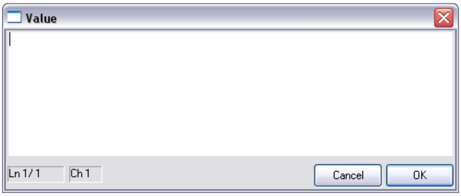


Figure 12.12

- The algorithm text is input in the script language, **Figure 12.13;**

Figure 12.13

- When the text is input with **[OK]** button, the value string show three dots that indicate assigning of a value to the **[Rolling bearing script]** variable, **Figure 12.14;**

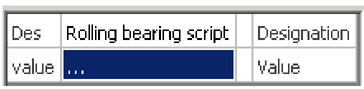


Figure 12.14

- Select the target parameter for the external variable, for example, [Ball bearing], in the tree. The bottom screen shows the element input window, **Figure 12.15;**

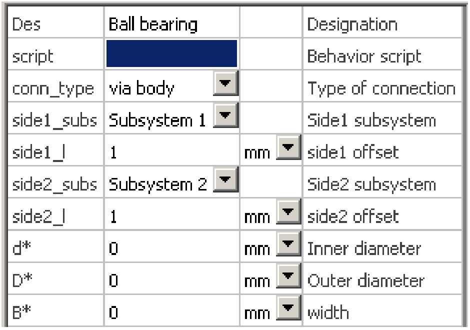


Figure 12.15

- A click with the mouse right button opens the command list for the external variables, **Figure 12.16;**

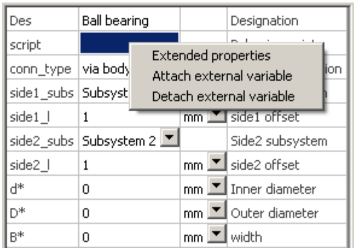


Figure 12.16

- Select the variable and carry out the **[Attach external variable]** command**.** There appears the external variable list that consists of variables of all types including text variables, **Figure 12.17;**

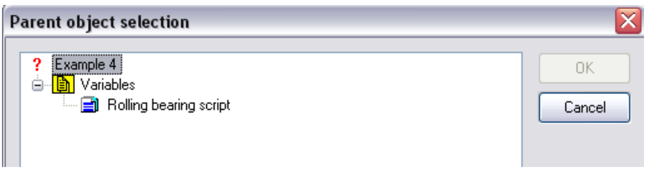


Figure 12.17

- Select the variable and assign its value with **[OK]**, **Figure 12.18.**

Des	Ball bearing		Designation
script	Rolling bearing		Behavior script
conn_type	via body		Type of connection
side1_subs	Subsystem 1		Side1 subsystem
side1_l	1	mm	side1 offset
side2_subs	Subsystem 2		Side2 subsystem
side2_l	1	mm	side2 offset
d*	0	mm	Inner diameter
D*	0	mm	Outer diameter
B*	0	mm	width

Figure 12.18

13 DYNAMIC SYSTEM SIMULATION

13.1 Screen image control

Figure 13.1 demonstrates a screen for simulation of a new dynamic system**.**

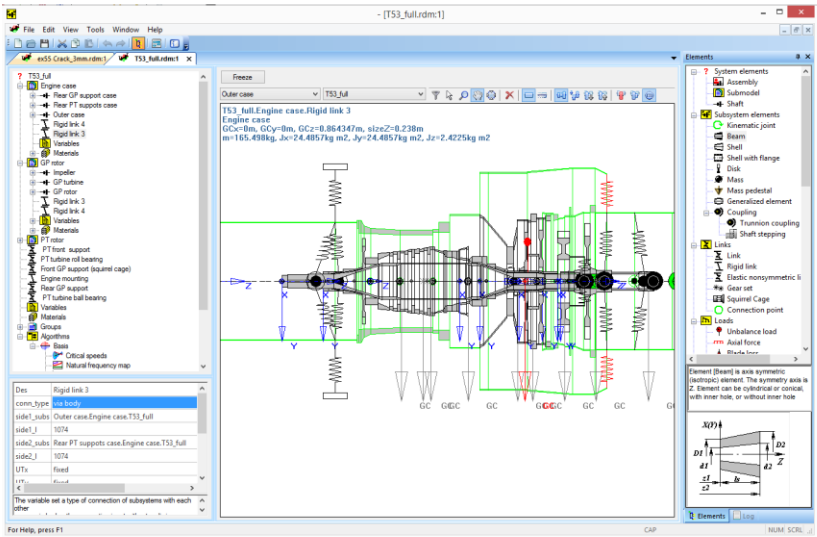


Figure 13.1

The upper part of the model input-output window has a line with the screen image buttons, **Figure 13.2**:

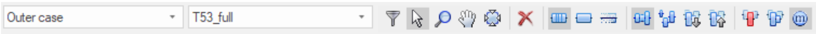


Figure 13.2

The buttons are the following:

- a list with the name of the active subsystem.

- shows the name of the assembly or the submodel which was selected in the filter dialog

- call of filter dialog. The dialog with the model structure will be shown. If a user chooses an assembly or a submodel, then only nested subsystems and links will be shown

- mouse cursor activity;

- image scale control;

- move the picture in the window;

- match the picture and the window size;

- delete an element. The element may be shown in the model or with the cursor in the project menu;

- display a 3-D project model with intermediate boundary elements lines

- display a 3-D project model;

- display the model skeleton;

- assemble subsystems on a common axis;

- move subsystems separately;

- expand assemblies on one level;

- undo expansion on one level;

- display a single active subsystem;

- visualize only current assembly/submodel.

- turn on/off output of mass and inertia properties in 2D window

13.2 Creation of single subsystem model

Creation of a model begins from definition of submodels, assemblies within each of them, subsystems inside assemblies, etc.

The result is the hierarchy of the main structure units, for example, Figure 13.3**.**

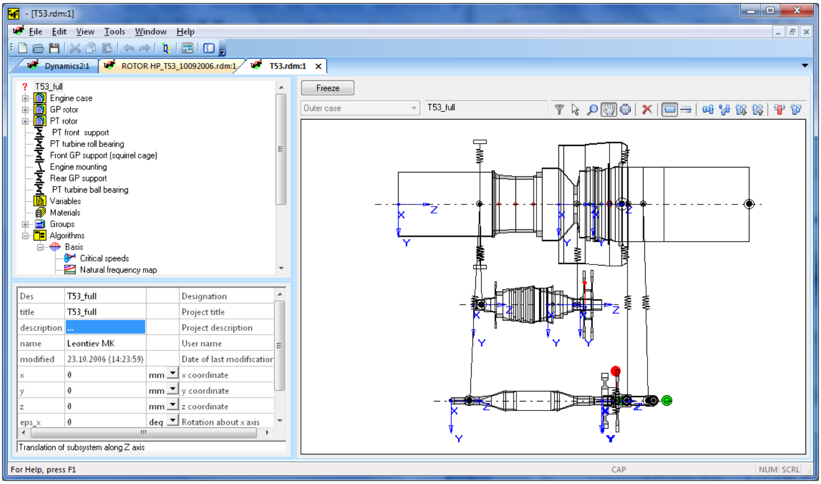


Figure 13.3

Each of the structure units may match with one or a few subsystems.

The subsystems are built subsequently.
1. Select the structure unit in the project menu in order to include it into the subsystem.

Select the **[Subsystem**] element and move it from the library by “Drag and Drop” into the input-output window.

The screen (**Figure 13.4**) displays the first beam element taken by default. Later the element and its parameters may be corrected or deleted.

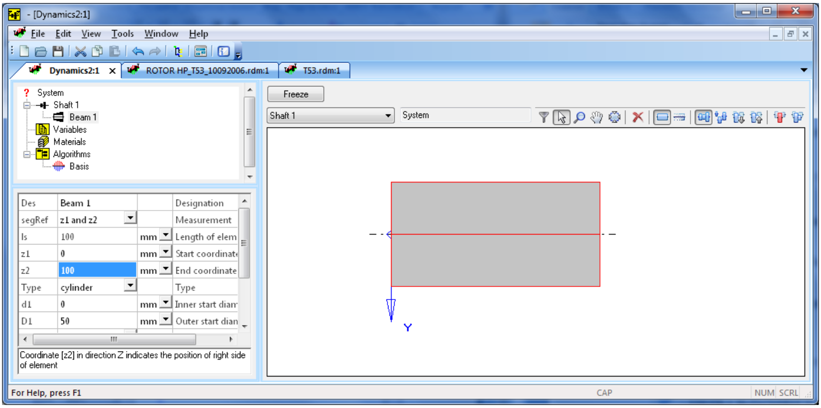


Figure 13.4

NOTE: Keep in mind that any subsystem must be of a positive length

2. Here the user defines general properties:

- Unique name

NOTE: The subsystem should have a unique name related to the structure. It helps not to feel lost in complicated dynamic systems consisting of a few subsystems

- 3-D location is defined by coordinates of the system left end. By default all the systems are created co-axially;
- Subsystem material. If it is necessary, the material of some elements may be re-determined.

3. Assemble the model of linear elements by using the “Drag and Drop” function.

The element parameters may be edited together with the element input or with the model editing. For this purpose select the special element (**Figure 13.5**) with the cursor or in the project menu. After the element selection the left bottom window **[Element data window]** opens the element parameters that may be edited. The data may be input with a cursor click in any point of the screen.

NOTE: To select an element, place the cursor on it and click the left mouse button. The selected element becomes active and its contour is highlighted with red. Links with the zero subsystem, or with the basement are selected through the project menu

NOTE: All the elements with determined dimensions are displayed in the proper scale

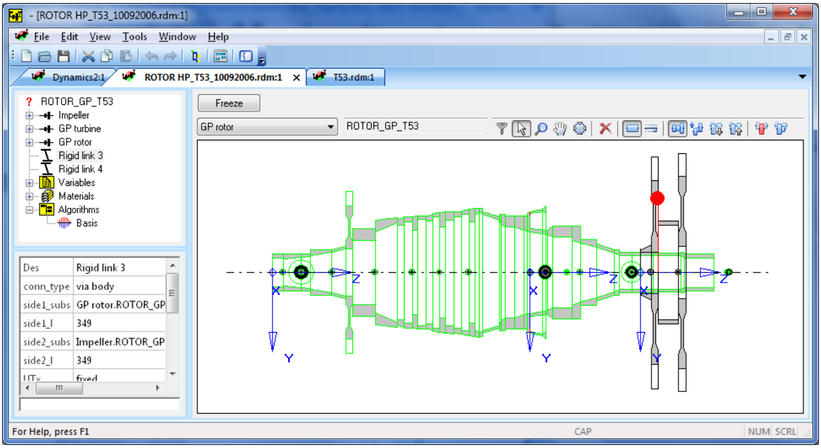


Figure 13.5

The selected element may be deleted, or copied, or inserted into any subsystem. The right mouse button opens the context command menu, **Figure 13.6**.

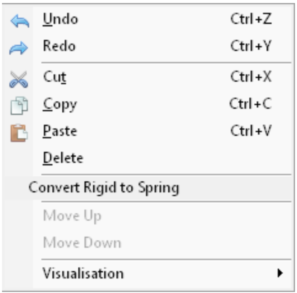


Figure 13.6

You may take an element from the library and insert it into any place of the model. This action changes length of a subsystem.

NOTE: If a subsystem consists of many elements or the elements' length is small, they may be displayed as a line. This image is not convenient for the selection, so select one subsystem or use the **[Zoom]** function

Beam or shell elements may be amended by addition of joint, or mass, or load elements. If so the beam or shell will be split into two parts in the addition section.

Sometimes it is required to add the section in the model without recourse to autodivision of beam and shell elements. Or it is needed to divide the elements into those, which autodivision can't be applied to. For cylindrical elements it could be also done before using the function of the elements' copy-paste.

Now at double mouse click in the 2D panel, the dialog window (**Figure 13.7**) appears where it is possible to edit the coordinate of the element paste either in the local coordinate system or in the global one. Paste of the section in the global coordinate system can be useful at setting the section for the link attachment in the [via body] regime where coordinates are given in the global coordinate system of the model.

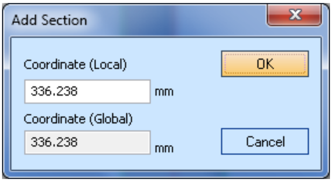


Figure 13.7

This functionality may be also used for approximate determination of the coordinates in the model.

> **NOTE:** All element parameters except geometry will be duplicated

When checking the model, distances between different model stations (for example, between supports) are sometimes required to be obtained. As the elements' coordinates in subsystems are given in the local coordinate system, this may cause difficulties.

The command of the context menu **[Get Global Coordinates]** is available for elements, links, subsystems and assemblies, **Figure 13.8.** The output is done in the **[Log]** window in the following format «**[name of element]** x1 y1 z1 x2 y2 z2». For lengthy elements, subsystems, links the start and end coordinates are output. For point elements, the coordinates 1 and 2 are duplicated. For assemblies the coordinates of origin of the assembly's coordinate system are output.

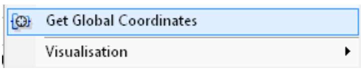


Figure 13.8

User can change transparency and colors of the model elements with the help of "Visualisation" context menu command (Figure 13.9). This command can be applied to single elements, or all elements inside shaft or assembly structures.

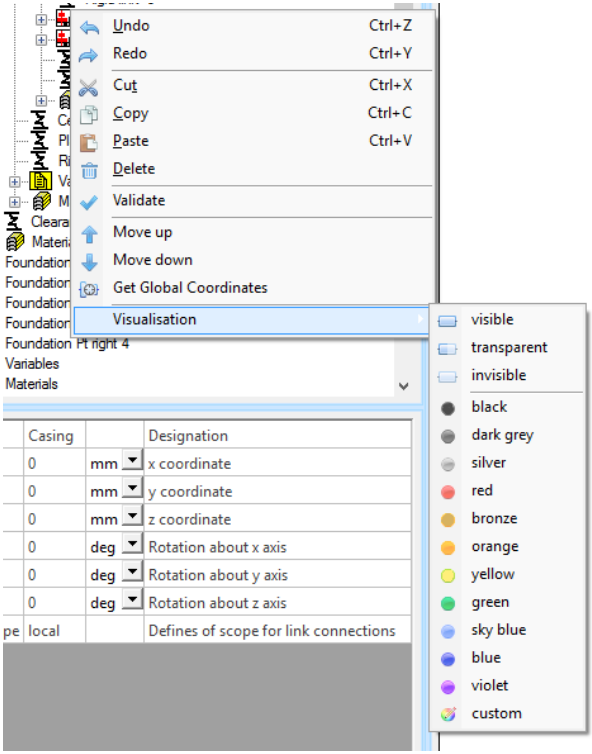


Figure 13.9

13.3 Creation of a multi-subsystem model

Creation of a multi-subsystem model is done in the following steps, **Figure 13.10.**

- Separate subsystems are created. The subsystems number is not limited.

- The subsystems are linked. Each link simulates a linear or non-linear element of the real dynamic structure. The number of links is not limited.

NOTE: Link may be directly attached to the subsystem. In this case position of its boundary point is determined by Z coordinate. Link may also be attached to link point that had been assigned beforehand. For spaced systems the variant of link attachment is preferable

- Links parameters are determined.

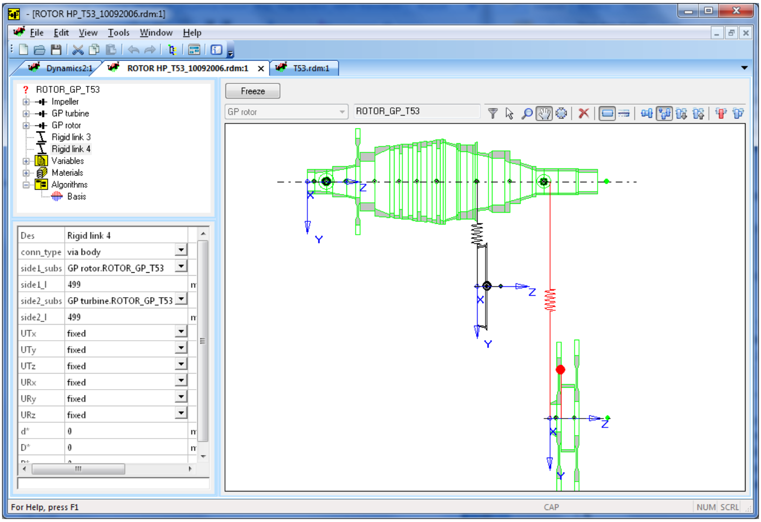


Figure 13.10

NOTE: Links parameters may be input or edited through links selection in the project

NOTE: Unique names of the subsystem links are very useful. This measure may help in analysis of calculation results of complicated systems

Figure 13.11 illustrates a system consisting of six subsystems. All the subsystem links are elastic or rigid. The axes are spaced conventionally.

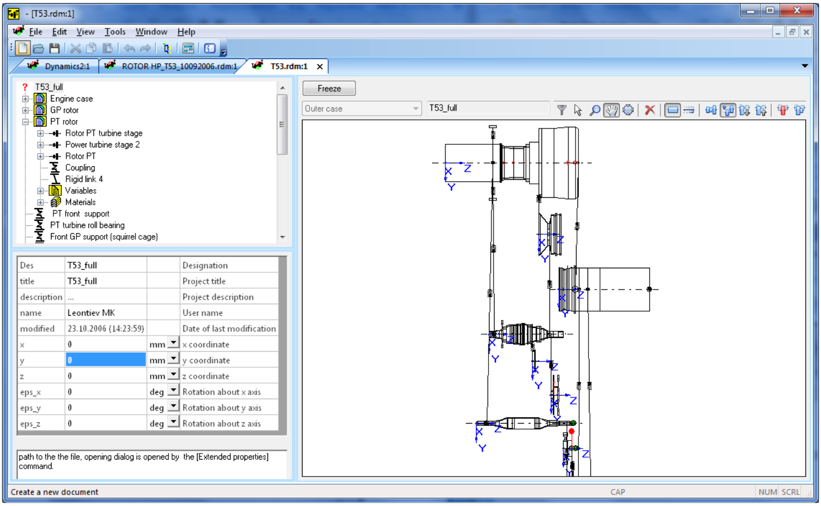


Figure 13.11

Figure 13.12 shows the initial position of subsystems.

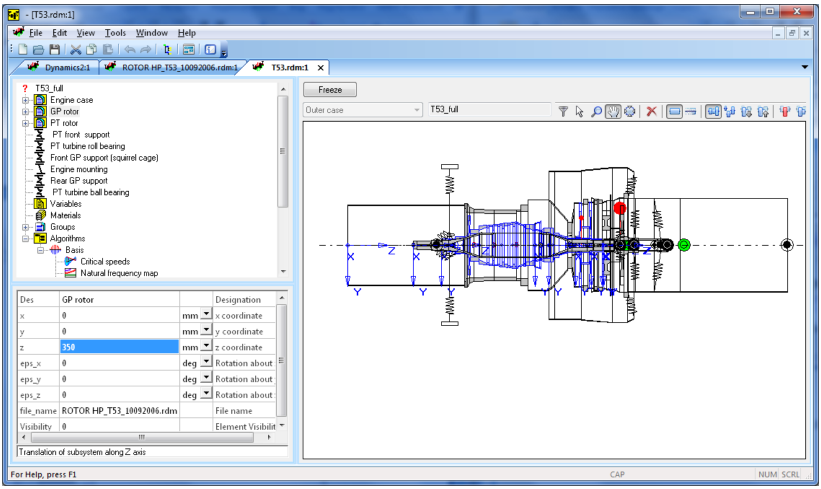


Figure 13.12

13.4 Spacing of assemblies and submodels

Building the complex model, it is desirable to structure it properly. Spacing of assemblies and submodels may help significantly in the further work at the model. In addition to convenience of work in the model tree, visualization of the subject of inquiry at spacing of the model's components is improved.

Figure 13.13 shows an example of the consecutive model's spacing into submodels and assemblies.

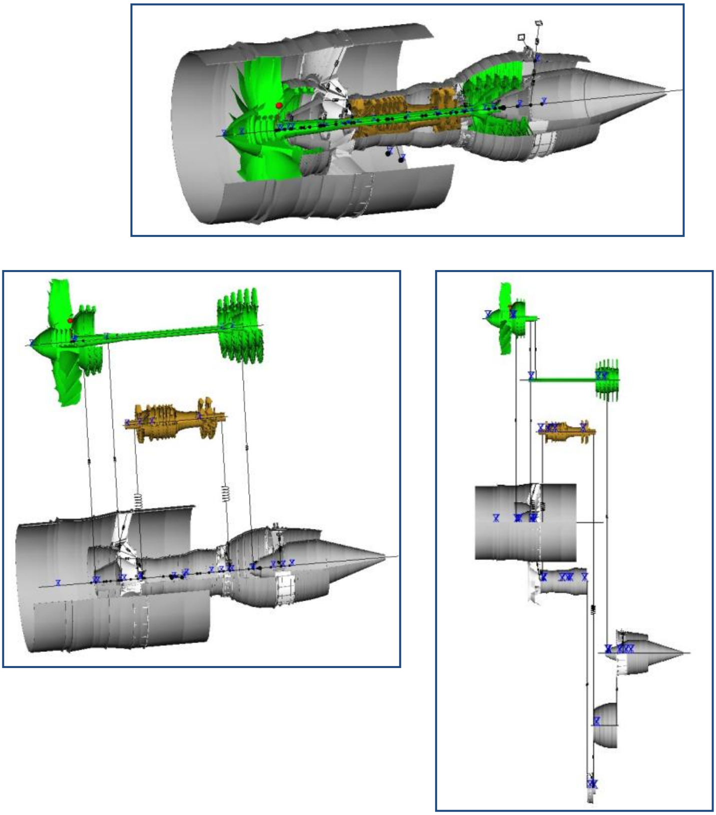


Figure 13.13

To visualize mode shapes, it is preferable to single out into separate structural elements not according to the engine modular design but according to its belonging to cases and rotors.

NOTE: Every pressing the “Expand assemblies” button spaces model to one nesting level.

The order of arrangement of such elements as **[Assembly**] and **[Submodel]** may be changed in the model tree. For this the commands of the context menu “Move Up”, “Move Down” may be used, **Figure 7.6.** Change in the order of sequence of structural elements in the model tree is for change in visualization at spacing of assemblies.

13.5 Import dialog of parameterized data of stiffness and damping in journal bearings

Stiffness and damping parameters in journal bearings depend significantly on the rotor speed. **Figure 13.14** shows the table with the calculated coefficients in the XLPocket program (www.rmt-inc.com)).

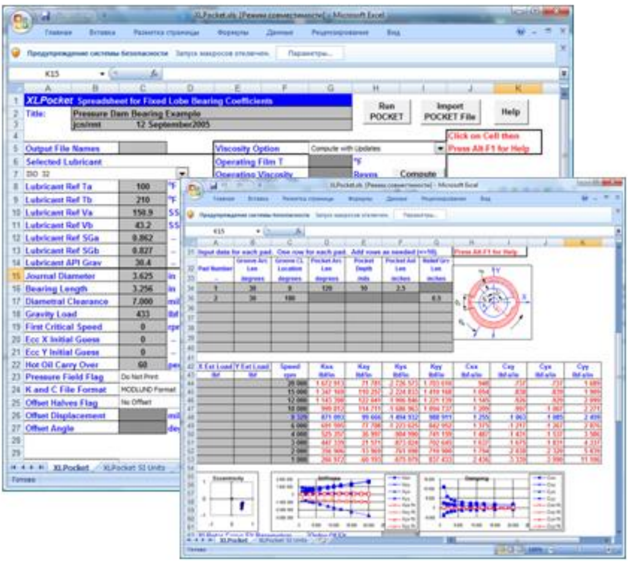


Figure 13.14

Such bearings are modeled with the use of parameterization of the corresponding sells in the stiffness and damping matrix of the **Elastic nonsymmetric link.**

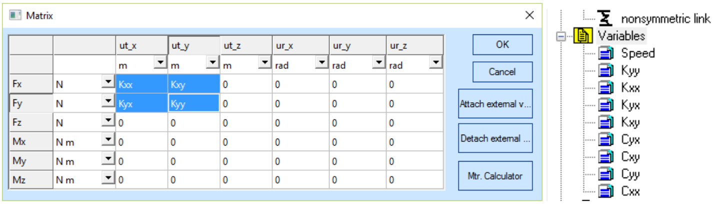


Figure 13.15

This type of a link is taken into account in the modal analysis algorithms (in all algorithms except Basis algorithm). It is also possible for this type of a link to define nonsymmetrical stiffness and damping matrixes typical of the journal bearings, **Figure 13.15.**

For the correct calculation with **Elastic nonsymmetric link**, a spurious link should be present to calculate basis and to obtain the whole set of shapes. There is a special dialog of the bearing data input in the table form to automatize the initial data input. The dialog may be called from the "Service\Import of bearing table" menu, **Figure 13.16**.

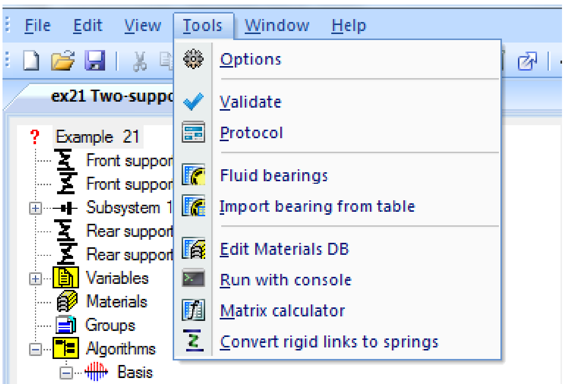


Figure 13.16

Figure 13.17 shows the dialog of the table data processing. It allows generating of new links in the model with the automatic attachment of variables and data editing in links which already exist in the model.

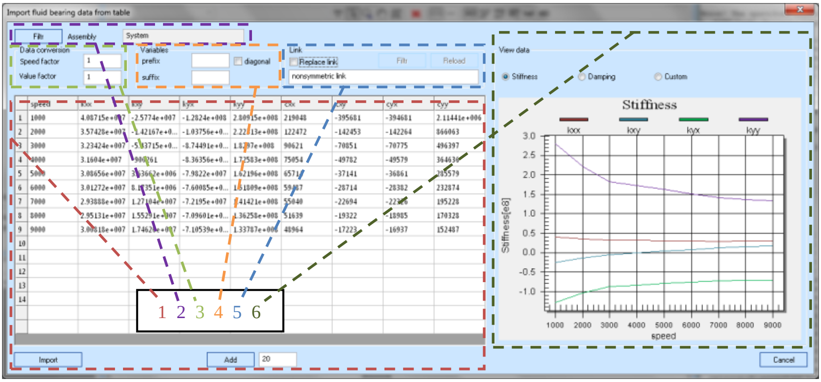


Figure 13.17

The following groups of elements are denoted by figures in **Figure 13.17**:

1. There is a table containing rotating speeds, stiffness and damping data. When pasting in the program, rotating speeds are interpreted as time_proportion (the first column in variables), stiffness and damping are in the SI system (N/m, N*sec/m)
2. It defines an assembly in the model for pasting new Elastic nonsymmetric link and set of variables.
3. They assign coefficients to convert table data. They are used to correct speed in time proportion, and to convert stiffness and damping from paste values into SI system.
4. Control elements to attach prefix and suffix to the variable name. It is necessary when there are several bearings in the model in order to separate the names of the variables attached to different links. In case if the «diagonal» element is ticked, only diagonal elements are processed.
5. The group of elements for assigning the link characteristics. Determination of new link name of loading data from the links that exist in the model. To activate elements of loading control, it is necessary to mark the "Replace Link" point.
6. Visualization of data that are input in the table of stiffness and damping. For example, if «Stiffness» is ticked, only columns corresponding to stiffness are output. At «Custom» choice, those columns are output where the first sells are highlighted.

To visualize mode shapes it is preferable to highlight some structural elements not according to the modular design of an engine but according to belonging to cases and rotors

NOTE: every pressing of the **[Expand assemblies]** button spaces the model to one nesting level

Data import on stiffness and damping from a text file is also possible. The file should consist of 9 columns with data separated by white-space. White-space should terminate the row. The first column – rotating speeds, the other 8 columns – stiffness (kxx, kxy,kyx,kyy) and damping (cxx, cxy, cyx, cyy). The first row is ignored.

14 FLUID BEARINGS ANALYSIS

14.1 Introduction

DynFB – a computer program specially designed for static and dynamic performance assessment of different types of fluid bearings under stationary working conditions.

14.2 Hydrodynamic bearing model

The mathematical model of the fluid film is based on the well know Reynolds equation, which was modified according to the Constantinescu turbulence model, together with the energy conservation equation for thin films. The Reynolds equation is based on the following fundamental assumptions:

- The fluid is incompressible.
- The flow takes place in a thin film.
- Fluid inertia is not taken into account.

The Reynolds equation makes it possible to compute a static pressure in the fluid flow and has the following form:

$$\frac{\partial}{R^2\partial\varphi}\left(\frac{h^3}{12k_x\mu}\frac{\partial p}{\partial\varphi}\right)+\frac{\partial}{\partial z}\left(\frac{h^3}{12k_z\mu}\frac{\partial p}{\partial z}\right)=\frac{\partial h}{\partial t}+\frac{1}{2}\omega\frac{\partial h}{\partial\varphi},$$

where R – journal radius, μ – dynamic viscosity of the fluid, h – radial clearance between the bearing journal and the bearing housing, ω – journal rotation speed, φ, z - angular and axial coordinates, t – time, k_x, k_z – turbulence coefficients, which are calculated by the following formulas:

$$k_x = 1 + 0.04417 \cdot (k^2 Re)^{0.725},$$
$$k_z = 1 + 0.0247 \cdot (k^2 Re)^{0.65},$$
$$k = 0.125 \cdot Re^{0.07},$$

where Re – local Reynolds number inside the flow $Re = \frac{\rho R \omega h}{\mu}$.

The energy equation for a fluid bearing is also obtained based on the assumptions relevant for a thin film fluid flow:

- convection in $\boldsymbol{\varphi}$ and $\boldsymbol{z}$ directions significantly predominate over diffusion
- heat capacity and thermal conductivity of the fluid flow are considered to be constant in all the points of the computational domain
- a stationary process is considered, therefore time derivatives are zero
- in a turbulent flow, the effect of the turbulent mixing far outweighs the fluid molecular diffusivity. Consequently, increase of the temperature during viscous dissipation has a tendency to uniform distribution across the film thickness.

Therefore, the energy equation has the following form:

$$\rho c h \left(U \frac{\partial T}{R \partial \varphi} + W \frac{\partial T}{\partial z} \right) = \frac{12\mu}{h^2} \left(k_z W^2 + k_x \frac{R^2 \omega^2}{12} + k_x \left(U - \frac{\omega R}{2} \right)^2 \right) + Q_s,$$

$$U = \frac{-h^2}{12 \mu k_x} \frac{\partial p}{R \partial \varphi} + \frac{R\omega}{2},$$

$$W = \frac{-h^2}{12 \mu k_z} \frac{\partial p}{\partial z},$$

$$Q_s = h_j (T - T_j) + h_b (T - T_b),$$

where ρ – the fluid mass density, c – the fluid heat capacity, T – temperature, U, W – velocities of the flow in circular and axial directions, respectively, Q_s – heat flow into the journal and the housing walls, h_j, h_b – heat transfer coefficients from the fluid to the journal and the housing, respectively, T_j, T_b – temperature of the journal and the housing, respectively. Connection between the Reynolds equation and the energy equation is provided through the dynamic viscosity (a function of the temperature) and the fluid flow velocity (a function of the fluid pressure).

The cavity region is defined by the Swift-Stieber boundary conditions:

$$p = 0,$$

$$\frac{\partial p}{\partial \varphi} = 0.$$

The force acting on the journal is defined as the integral of the static pressure field obtained from the solution of the Reynolds equation. The equality between the specified external load and the computed reaction force of the fluid film determines the rotor steady-state position. Static and dynamic performance parameters are calculated for the obtained steady-state condition.

14.3 Main assumptions of the hydrodynamic bearing mathematical model

- The fluid is incompressible;
- The fluid is Newtonian;
- The fluid film is thin $\frac{\boldsymbol{c}}{\boldsymbol{R}} = \boldsymbol{O}(\mathbf{10^{-3}})$;
- Fluid inertia is not taken into account;
- Surfaces of the journal and the housing are rigid and isothermal;
- Heat transfer by the fluid from one pad to another is taken into account through a given hot oil carry over coefficient;
- Possible misalignment between the journal and the housing is not considered into account;

14.4 Bumped type air-foil bearing model

For calculation gas pressure on the walls of the shaft and bearing housing, the Reynolds equation for a compressible fluid is used, which has the following form:

$$\frac{\partial}{R^2 \partial \varphi}\left(p \frac{h^3}{12\mu} \frac{\partial p}{\partial \varphi}\right) + \frac{\partial}{\partial z}\left(p \frac{h^3}{12\mu} \frac{\partial p}{\partial z}\right) = \frac{1}{2}\omega \frac{\partial}{\partial \varphi}(ph) + \frac{\partial}{\partial t}(ph),$$

where R – shaft radius, h - clearance distribution, μ – dynamic viscosity of the oil, p – static pressure, ω – angular velocity of the shaft, φ, z – spatial coordinates, t – time.

The pressure at the ends of the bearing is equal to the ambient pressure:

$$p(\varphi, 0) = p(\varphi, L) = p_a,$$

where p_a – ambient pressure (usually equal to static atmospheric).

If the bearing consists of separate segments or leaves, then the boundary conditions at the leading and trailing edges of the segments are ambient pressure.

$$p(\varphi_1, z) = p(\varphi_2, z) = p_a.$$

Unlike bearings with rigid walls, the clearance in air-foil bearings depends not only on spatial coordinates, but also on pressure. This is because the leaves are compliant and deformed under gas pressure $h = f(\varphi, z, p)$. The clearance value in the bearing is determined by two terms. Primarily the clearance is determined by the displacement of the shaft relative to the housing $h_0(X, Y)$, but in addition to the displacement, the clearance is determined by the leaf deformation $\delta(p)$. DynFB uses the well-known Heshmat model to consider the deformation of the foil leaf lying on the bump-strip layer.

$$h(\varphi, z) = h_0 + \delta = c + X\cos(\varphi) + Y\sin(\varphi) + K_1(p - p_a),$$

where c – bearing clearance, K_1 – flexibility coefficient of the bump-foil structure. The coefficient K_1 is determined by the properties of the bumped strip material and its geometry (see Figure 14.1):

$$K_1 = \left(\frac{\alpha c}{p_a}\right),$$

$$\alpha = \frac{2 s_0 p_a l_0^3}{E t_0^3 c}(1 - \nu^2),$$

where s_0 - pitch between two bumps, l_0 - original half-length of a bump, t_0 - thickness of a bump, h_0 - original height of a bump, E - Young's modulus of a bump, ν - Poisson ratio of a bump, α – dimensionless compliance.

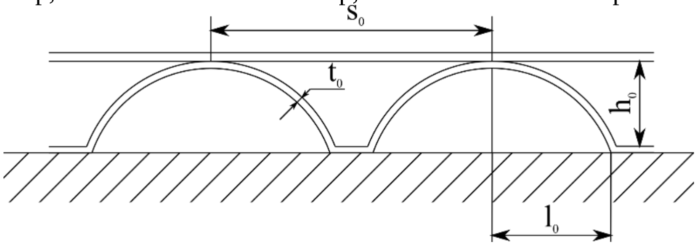


Figure 14.1

The Heshmat model is based on a number of assumptions:

- The stiffness of the foil is taken to be uniformly distributed and constant throughout the bearing surface.
- The coefficient $\boldsymbol{\alpha}$ is a constant and thus independent of the amount of the bump deflection.
- The foil is assumed not to "sag" between bumps, but rather to follow the deflection of the bumps themselves.
- The deflection of the foil in its response to the acting forces is dependent on the local effect only. i.e., on the force acting directly over the particular point.

- Friction between the foil, bumps and housing is considered negligible.

The construction of foil journal bearings essentially does not permit the generation of subambient pressures. Since when the condition $p < p_a$ is reached, the foil will lift up from the bumped strip under the action of the ambient pressure and deflect towards the shaft, reducing the clearance value until the pressures on both sides of the foil are equalized (see Figure 14.2). This local area has no effect on the value of the radial force. However, this effect will affect the pressure field in the film. According to Heshmat, the above phenomenon is essentially similar to the appearance of a cavitation zone in conventional hydrodynamic journal bearings. To take this into account, in DynFB an additional pressure boundary condition is applied:

$$P = (p/p_a) = 1, \text{где } p < p_a,$$

which is similar to the Gümbel boundary condition in hydrodynamic journal bearings.

In gas bearings with rigid walls the subambient pressure zone persists, and its account does not require any additional efforts.

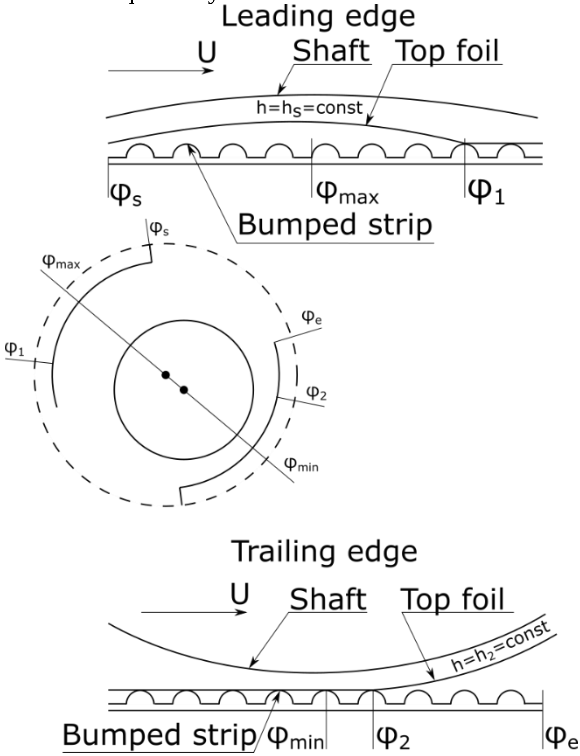


Figure 14.2

14.5 Main assumptions of the gas bearing mathematical model

- The gas film is thin, therefore the fluid inertia force and pressure gradient across the film thickness are considered negligible;
- The flow is laminar;
- The flow is isothermal;
- The fluid in the film is a prefect gas, i.e. for an adiabatic isothermal process the mass density is proportional to the pressure $\boldsymbol{p/\rho = Const}$.

14.6 Calculation results

For each steady-state condition of the journal bearing, which is defined by the journal rotation speed and the external load, following set of parameters is calculated:

Table 14.1

Name	Dimension	Comment
Stiffness coefficient [K]	N/m	Stiffness coefficients for two degrees of freedom in X and Y directions
Damping coefficient [C]	N*m/s	Damping coefficients for two degrees of freedom in X and Y directions
Steady-state coordinates of the rotor [X/c, Y/c]	-	The rotor steady-state condition coordinates X, Y related to the value of a given gap
Eccentricity of the rotor [Epp]	-	The rotor eccentricity related to the value of a given gap
The rotor circumferential angular coordinate [Psi]	deg	The rotor circumferential angular coordinate measured from the negative load vector (minus Y) in direction of the shaft rotation.
Maximum pressure [Pmax]	Pa	The value of the fluid film maximum pressure
Angle of maximum pressure [Pmax_angl]	deg	Circumferential angular coordinate of the maximum pressure point.

Minimal clearance [Hmin]	-	The value of minimal fluid film clearance related to the value of a given clearance
Angle of minimum clearance [Hmin_angl]	deg	Circumferential angular coordinate of the minimum clearance point.
Threshold rotating speed [ThresSpeed]	rpm	Threshold rotating speed, above which the rigid rotor will lose stability on the journal bearing for a given weight force
Is the bearing stable [IsStable]	YES / NO	Is the rigid rotor stable on the journal bearing for given weight force and rotating speed.
Hydrodynamic friction power loss [PwrLoss]	W	Power loss due to hydrodynamic friction in the fluid film.
Side leakage [SideFlow]	lpm	Oil leakage from the bearing open ends. (For hydrodynamic bearings only.)
Maximal temperature [Tmax]	deg. C	Maximum temperature in the fluid film.
Critical mass [CritMass]	kg	Threshold mass of the rigid rotor, above which the rotor will remain stable on the journal bearing at a given rotating speed
Pads displacements	[rad, m, m]	Pads displacements in their local coordinate systems: one rotation and two translations. These parameters are calculated only for tilting pad journal bearings.

14.7 Common initial data

Rotating speed of the rotor ω [rpm] is a list of rotating speeds for which the bearing parameters will be calculated.

External loads Fx, Fy [N] are the external load vector components. They can be constant or functions of the rotating speed:

$$F = F_0 + F_1 \cdot \omega + F_2 \cdot \omega^2.$$

Lubricant properties:

- Name of the lubricant (for fluids only). If the name does not exist in the lubricants database, the user must add new position into the database and provide the lubricant properties.
- The lubricant mass density under working temperature [kg/m^3]. It will be taken into consideration if the direct input of the oil properties is turned on. In this case the energy equation is excluded from the calculation and mass density is considered equal to the input value and constant across the whole fluid film.
- The fluid dynamic viscosity [Pa*sec]. Similarly to the previous item it is taken into consideration only in the case of direct input of the oil properties is turned on.

Oil input properties (for hydrodynamic bearings only):

- Hot oil carry over coefficient. It determines which part of heat generated in the previous pad will be carried by oil into the next pad. Limit values [0; 1].
- Inlet oil temperature [℃].

Additional properties (for hydrodynamic bearings only):

- Cavitation zone boundary conditions. Gümbel boundary condition:

$$\textbf{If } \boldsymbol{P < 0}, \textbf{then } \boldsymbol{P = 0}$$

Reynolds (Swift – Stieber) boundary condition:

$$p = 0, \frac{\partial p}{\partial \varphi} = 0.$$

- Computation grid nodes number m, n in circumferential and axial directions, respectively.
- Heat analysis regime: isothermal – the fluid film temperature is considered constant and equal to a given value and uniform across the whole film, not isothermal – the temperature of the fluid film is nonuniform and determined from the coupled solution of the Reynolds equation and the energy equation.
- Heat analysis boundary conditions: adiabatic – no heat exchange between the fluid film and the shaft and the housing surfaces, input temperature of the shaft and the housing surfaces – the surfaces are considered to be isothermal and have constant temperature equal to a given value.

14.8 Plain journal bearing.

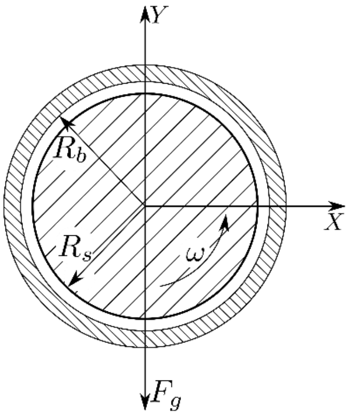


Figure 14.3

Table 14.2

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	c

14.9 Axial groove journal bearing.

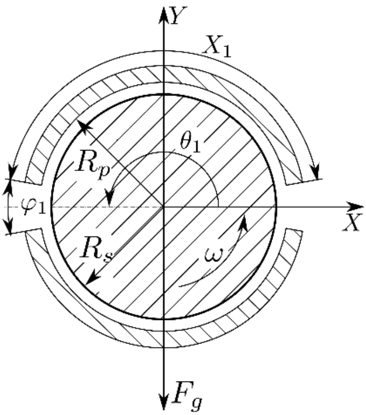


Figure 14.4

Table 14.3

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	c
4	Angular length of a pad	deg	X_i	Pd_Arc_Ln[]
5	Angular position of the groove centerlines	deg	θ_i	groove_Arc_pos[]
6	Angular length of a groove	deg	φ_i	Grv_Arc_Ln[]

14.10 Offset half journal bearing.

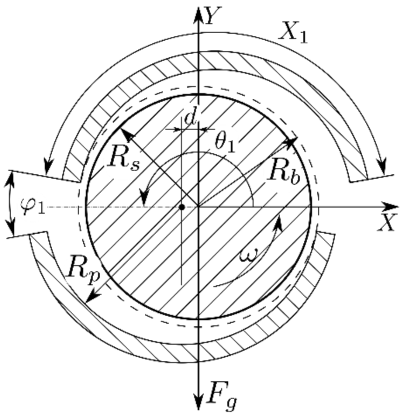


Figure 14.5

Table 14.4

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance (Manufacturing, not assembly)	m	Cp=Rp-Rs	c
4	Angular length of a pad	deg	X_i	Pd_Arc_Ln[]
5	Angular position of the groove centerline	deg	θ_i	groove_Arc_pos[]
6	Angular length of a groove	deg	φ_i	Grv_Arc_Ln[]
7	Pad offset	m	d	Lin_offset

14.11 Elliptical and multi lobe journal bearing.

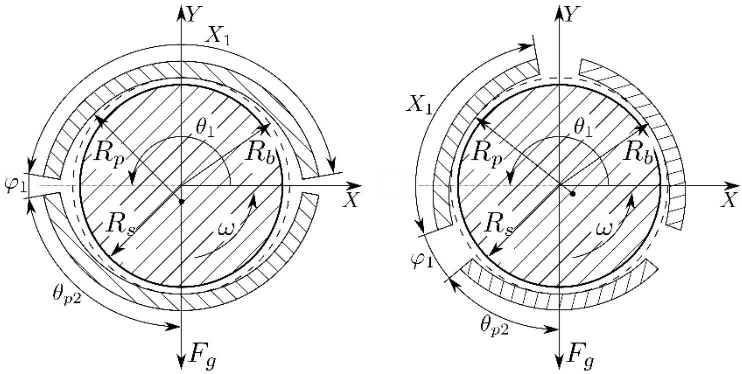


Figure 14.6

Table 14.5

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance (assembly)	m	Cb=Rb-Rs	c
4	Angular length of a pad	deg	X_i	Pd_Arc_Ln[]
5	Angular position of the groove centerlines	deg	θ_i	groove_Arc_Pos[]
6	Angular length of a groove	deg	φ_i	Relief_Grv_Len[]
7	Lobe preload	--	mf	preload[]
8	Lobe offset	--	af	offset[]

Preload is the ratio of the distance between the pad curvature center and the bearing geometric center to the radial clearance:

$$mf = 1 - \frac{c_b}{c_p}.$$

Lobe offset is the ratio of the converging pad length to the full arc length:

$$af = \frac{\theta_{pi}}{X_i}.$$

14.12 Tapered land journal bearing.

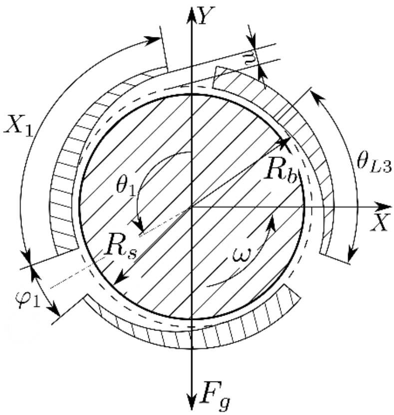


Figure 14.7

Table 14.6

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	c
4	Angular length of a pad	grad	X_i	Pd_Arc_Ln[]
5	Angular position of the groove centerlines	grad	θ_i	groove_Arc_pos[]
6	Angular length of a groove	grad	φ_i	Grv_Arc_Ln[]
7	Taper angular length	grad	θ_{Li}	Taper_Arc_Len[]
8	Lobe undercut	m	u	lob_undercut[]
9	Lobe undercut axial length	m	L_l	Taper_Ax_Len[]

14.13 Pressure dam journal bearing.

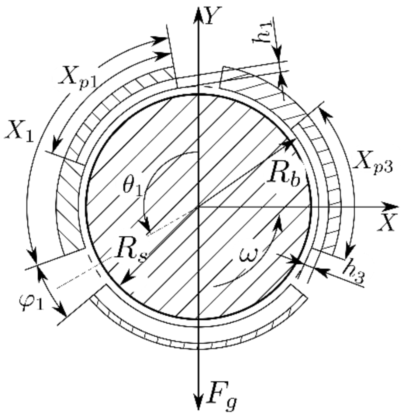


Figure 14.8

Table 14.7

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	c
4	Angular length of a pad	deg	X_i	Pd_Arc_Ln[]
5	Angular position of the groove centerlines	deg	θ_i	groove_Arc_pos[]
6	Angular length of a groove	deg	φ_i	Grv_Arc_Ln[]
7	Pocket angular length	deg	X_{pi}	Pocket_Arc_Len[]
8	Pocket depth	m	h_i	Pocket_Depth[]
9	Pocket axial length	m	h_{gi}	Pocket_Axl_Len[]
10	Axial length of a circumferential groove	m	L_{gp}	Relief_Grv_Len[]

14.14 Tilting pad journal bearing.

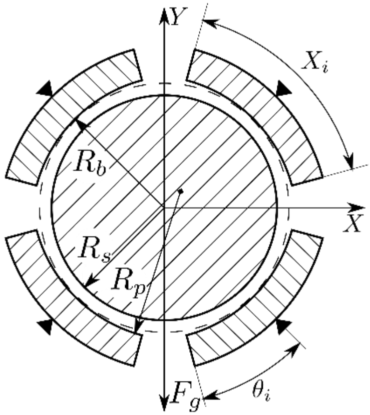


Figure 14.9

Table 14.8

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance (assembly)	m	Cb=Rb-Rs	c
4	Angular length of a pad	deg	X_i	PadArcLen
5	Pad preload	--	mf	PadPreload
6	Pad pivot offset	--	af	PadPivotOffset
7	Gravity force direction	--	df	GravityForceDirection
8	Enable pad inertia	--	pi	PadInertiaInclude
9	Enable pivot stiffness	--	pstfi	PivotStiffInclude

Preload is the ratio of the distance between the pad curvature center and the bearing geometric center to the radial clearance:

$$mf = 1 - \frac{c_b}{c_p}.$$

Pad pivot offset is the ratio between the angular offset from the pad leading edge to the pad pivot and the angular length of the pad:

$$af = \frac{\theta_i}{X_i}.$$

The direction of the gravity force can aligned either with the line passing between the pads or the line connecting centers of the two opposite pad. In fact, this parameter defines the orientation of the pads relative to the coordinate system and, in practice, the pads are usually oriented in one of the two aforementioned positions with respect to the gravity force direction.

Mass properties of the pads can be considered in the analysis. This feature is based on the assumption that the pads oscillate with the frequency of direct synchronous whirling motion of the rotor.
If pads mass properties are accounted for, the user must input some additional data (see Figure 14.10):

Table 14.9

#	Name	Units	Designation
1	Pad mass	kg	mp
2	Pad mass moment of inertia about the center of gravity	kg*m^2	Ip
3	Distance from pivot point to CG horizontal	m	B
4	Distance from pivot point to CG vertical	m	C

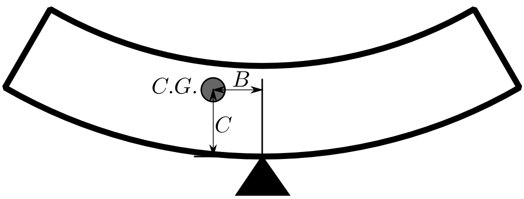
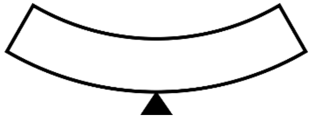
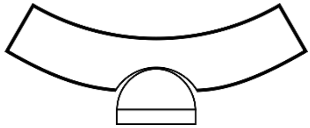
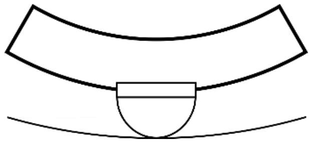
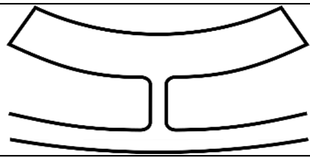


Figure 14.10

If the pad pivot is designed such that it can not be modeled as an ideal hinge, it is necessary to take into account its stiffness properties in the calculation of the dynamic characteristics of the bearing. At the moment, the following types of suspension segments are implemented in DynFB:

Table 14.10

Ideal hinge	
Spherical pivot – ball in sphere	
Cylindrical line contact model	
Spherical pivot – ball in cylinder	
Web pivot	

Initial data for the spherical and cylindrical pivot looks the same, but have different meanings depending on the selected type (see Figure 14.11**,** Figure 14.12):

Table 14.11

#	Name	Units	Designation
1	Pivot material elastic modulus	Pa	Ep
2	Housing material elastic modulus	Pa	Eh
3	Pivot material Poisson ratio	--	vp
4	Housing material Poisson ratio	--	vh

5	Pivot diameter	m	Dp
6	Housing diameter	m	Dh
7	Pad effective length	m	Leff

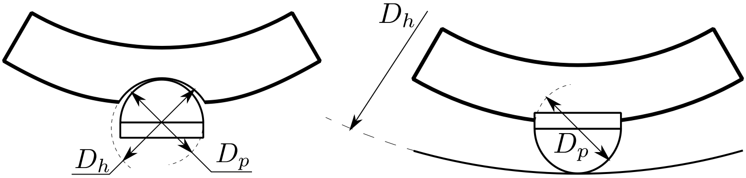


Figure 14.11

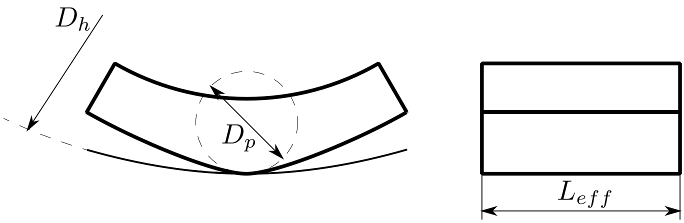


Figure 14.12

The web pivot is modeled as a cantilever beam with a rectangular cross section (see Figure 14.13).

Table 14.12

#	Name	Units	Designation
1	Pivot material elastic modulus	Pa	Ep
2	Web effective length	m	Leff
3	Web cross section area	m^2	A
4	Web cross section area moment of inertia in bending direction	m^4	I

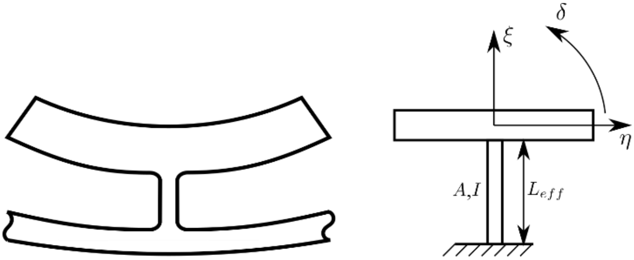


Figure 14.13

14.15 Gas bearing.

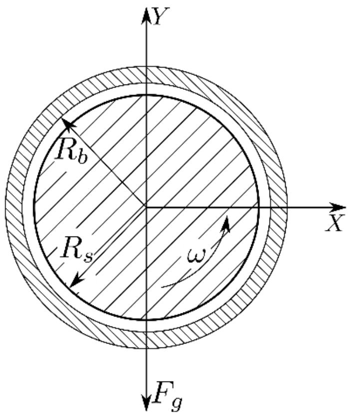


Figure 14.14

Table 14.13

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	C
4	Orientation angle	deg	α	OrientationAngle

14.16 Multi lobe gas bearing.

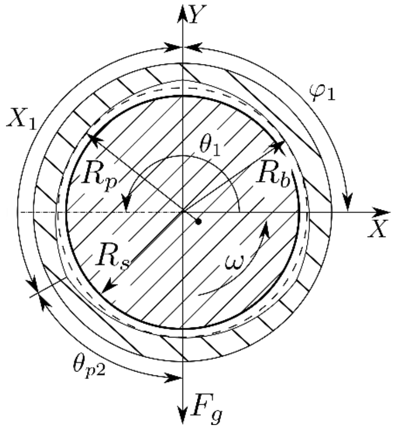


Figure 14.15

Table 14.14

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance (assembly)	m	Cb=Rb-Rs	c
4	Angular length of a pad	deg	X_i	Pd_Arc_Ln[]
5	Angular position of the first pad leading edge	deg	φ_1	firstEdgeLocation
6	Lobe preload	--	mf	preload[]
7	Lobe offset	--	af	offset[]

14.17 Air foil bearing.

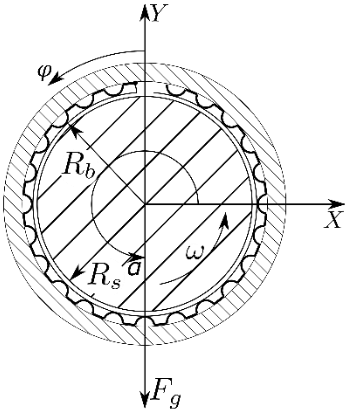


Figure 14.16

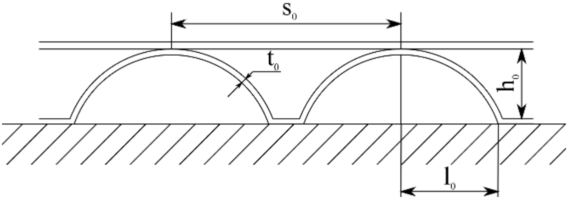


Figure 14.17

Table 14.15

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance (assembly)	m	Cb=Rb-Rs	c
4	Height of a bump	m	h_0	BMh
5	Half-length of a bump	m	l_0	BMl
6	Bumped strip width	m	w	BMw
7	Thickness of a bump	m	t_0	BMt

8	Pitch between two bumps	m	s_0	BMs
9	Poisson ratio of a bump	--	ν	BMnu
10	Young's modulus of a bump	Pa	E	BME
11	Friction coefficient	--	Fmu	Fmu
12	Structural loss factor	--	η	eta
13	Orientation angle	deg	α	OrientationAngle

When simulating such bearings, essentially two basic friction models are used: the viscous friction model and the hysteresis friction model.

$$P = K_b(1 + i\eta)U,$$

Where P – dimensionless pressure, K_b – dimensionless coefficient of stiffness of an elastic part, i – imaginary unit, η – structural loss factor. It is easy to see that when using the hysteresis friction model, the pressure is a complex number, which means that the forces in the bearing will also be complex. In this case, the Reynolds equation for the derivatives of pressure with respect to displacement (the method of infinitesimal deviations) is solved in a complex form, which makes it possible to obtain a matrix of mechanical impedance. And from its matrices of stiffness and damping of the bearing.

All basic DynFB equations in terms of gas dynamic bearings are based on a viscous friction model. However, provided that the rotor performs a circular precession, it is possible to easily make the transition from the hysteresis friction model to the viscous friction model and obtain the equivalent viscous damping coefficient.

$$C_{eq} = \frac{\eta K_b}{\gamma},$$

Thus, the structural loss factor η can be taken into account in the already implemented mathematical model.

14.18 Annular seal

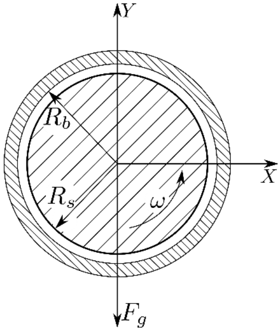


Figure 14.18

For the calculation of the Annular seal, the template "t31 AnnularSealBlack 1200" (using the Black & Jenssen model [42]) or "t32 AnnularSealChilds05 37360" (using the Childs model [43]) can be selected. The templates are set to calculate seal for 1200 rpm and 37360 rpm respectively.

Assumptions:
1) short seal analysis
2) Vibrations are small relative to a centered position.

Table 14.16

#	Name	Units	Designation	Designation in GUI
1	Journal radius	m	Rs	R
2	Bearing length	m	L	L
3	Radial clearance	m	Cb=Rb-Rs	C
4	Orientation angle	deg	α	OrientationAngle
5	Pressure Drop constant part	Pa		deltaP0
6	Pressure Drop linear part P=deltaP1*rpm	Pa*rpm		deltaP1
7	Pressure Drop linear part P=deltaP2*rpm^2	Pa*rpm^2		deltaP2
8	Inlet loss factor	-		inlet_loss
9	Inlet Swirl Ratio	-		Inlet_Swirl_Ratio

Pressure drop is calculated as:

$$deltaP = deltaP0 + deltaP1 * \omega + deltaP2 * \omega^2$$

14.19 Graphical user interface

There is a graphical user interface built in Dynamics R4 providing all necessary facilities to work conveniently with DynFB. The main dialog box can be opened from the “Tools\Fluid bearings” menu, See Figure 14.19.

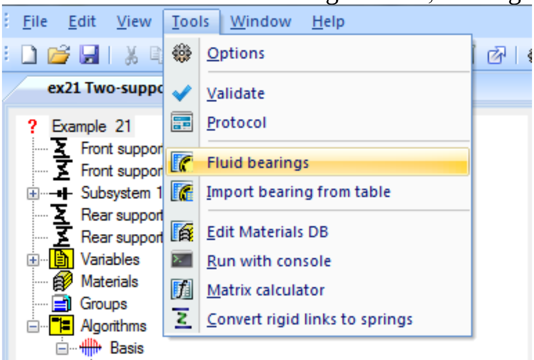


Figure 14.19

The DynFB dialog box is presented on the Figure 14.20. The dialog provides functionality to prepare bearing input data, run calculation, view the calculation results, create in the active model new links modeling the bearing parameterized by the rotation frequency.

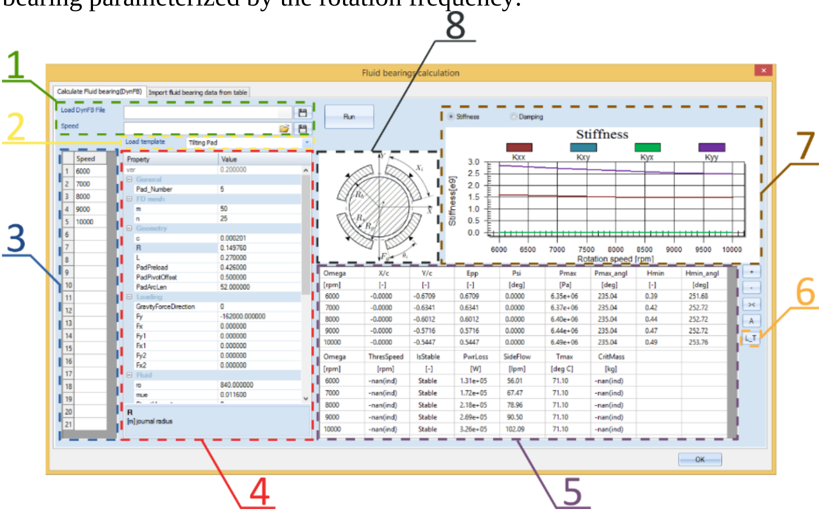


Figure 14.20

Numbers in the figure define following groups of options:

1. Paths to the computation model configuration file and the list of rotation speeds. Buttons to save the corresponding files.
2. Choice of the bearing type, the following options are available:
   - Two axial groove journal bearing “ t0 Two Axial Groove ”
   - Offset half journal bearing “ t1 OffsetHalf Bearing ”
   - Elliptical bore journal bearing “ t2 Elliptical ”
   - Multi lobe journal bearing “ t2 Multi-Lobe ”
   - Tapered land journal bearing “ t3 Taper Land ”
   - Pressure dam journal bearing “ t4 Pressure Dam ”
   - Plain journal bearing “ t5 Plain Journal ”
   - Tilting pad journal bearing with ideal hinge pivot “ Tilting Pad ”
   - Tilting pad journal bearing with spherical pivot – ball in sphere “ Tilting Pad Ball Contact ”
   - Tilting pad journal bearing with cylindrical pivot (line contact model) “ Tilting Pad Line Contact ”
   - Tilting pad journal bearing with web pivot “ Tilting Pad Web Pivot ”
   - Plain gas bearing
   - Multi lobe gas bearing
   - Air foil bearing
3. List of rotation speeds for which the bearing parameters are calculated
4. Main input area
5. Results output area (see Table 14.1). The results are presented in the convenient tabular form. Intermediate information is printed into this area directly from DynFB during the calculation process. This information can be used to monitor convergence of the solution verifying values of the support reactions forces (Fx, Fy): they must be equal to the external loads with the opposite sign. In addition, the control can performed monitoring the computed values of the dimensionless (normalized to the gap) rotor displacements. If those are significantly greater than one, the calculation has probably converged to incorrect results and the input data needs to be checked.
6. The user can view the DynFB output even after the calculation has completed by pressing the L_T button (see Figure 14.21).
7. Chart depicting the variation of the bearing stiffness or damping with respect to the rotation speed.

8. The chosen bearing type.

After the calculation for the selected configuration has completed, the obtained stiffness and damping coefficients are automatically transferred to the "Import fluid bearing from table" tab (see Figure 14.22) of is the "Import bearing from table" dialog (see paragraph 13.5). With the aid of this dialog, the user can easily import a parameterized link that reproduces behavior of the analyzed journal bearing into the model of the rotor dynamic system.

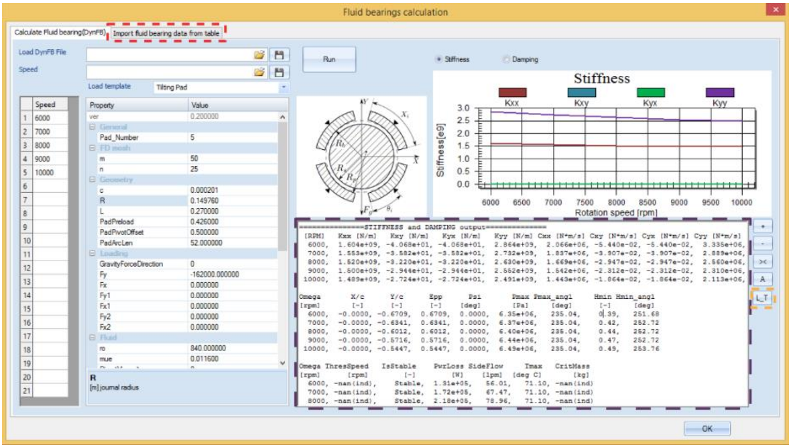


Figure 14.21

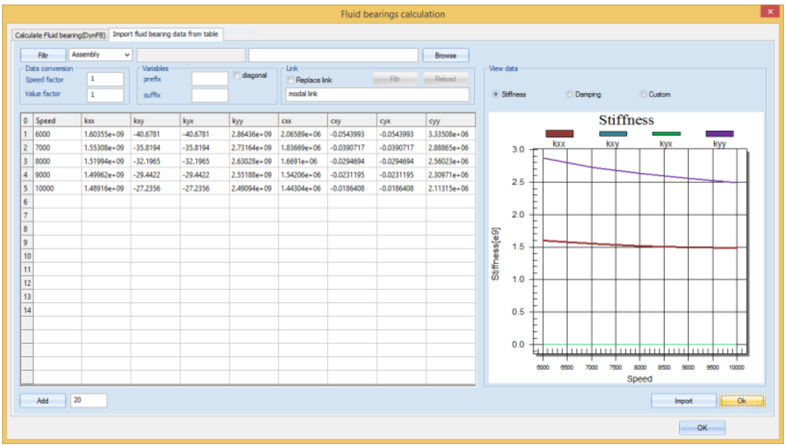


Figure 14.22

15 ALGORITHMS OF LINEAR SYSTEMS ANALYSIS

The steady state analysis uses the following algorithms:

- Basis calculation **[Basis]**
- Natural frequencies calculation **[Natural frequencies]**
- Natural frequency map **[Natural frequency map]**
- Critical speeds **[Critical speeds]**
- Steady state unbalance analysis **[Unbalance response]**
- Parameter analysis **[Parameter analysis]**

The algorithms above are aimed to analyze linear systems. Analysis of non-linear systems is given in the transient analysis section.

If the model includes non-linear elements, the linear algorithms ignore them. In this case the basis may be insufficient for further analysis and the calculation accuracy may be low. For example, if a rotor is supported only by journal bearings, the analysis results will be obtained for a free rotor.

The algorithms set of a project is built of the algorithm database. To place an algorithm into a project, select it in the database and click twice with the left button, **Figure 15.1.** The algorithm will be placed automatically in the database of the projects algorithm.

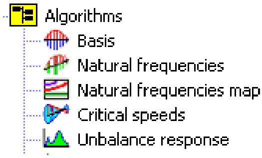


Figure 15.1

Any algorithm has a function window for the parameters input-output. The window is opened by selection of the algorithm. When the parameters are specified, the calculation is started with the **[Start]** button in the top part of the screen.

NOTE: While running a project calculation algorithm, it is preferable not to run other program codes

15.1 Basis

The dynamic response calculation is based on the modal analysis method. The modal analysis is based on the preliminary calculated set of natural frequencies and modes.

IMPORTANT NOTE: The set is calculated within a given frequency range under assumptions of zero damping and non-rotating rotors

The basis includes frequencies and modes of all types of oscillations, vertical and horizontal transversal, axial and torsional.

The basis is calculated after the model creation by pressing the button **[Start].** After calculation the frequencies and modes are sorted versus their type in the frequency list. When a frequency is selected in the list, its corresponding mode is displayed.

Figure 15.2 shows the basis frequency window. When a frequency is selected, the main window shows the natural mode.

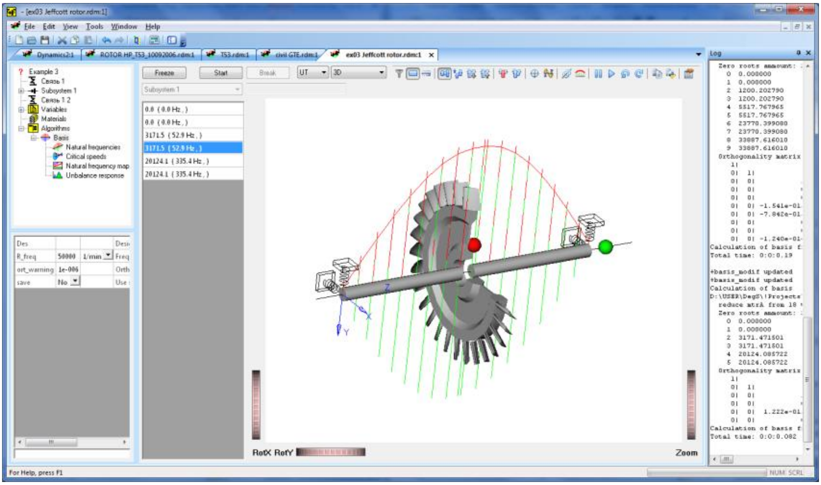


Figure 15.2

The function buttons provide different formats of the result display.

The basis analysis includes calculation of the modes orthogonality matrix. The matrix reflects the model correctness and the calculation accuracy. The Natural frequency list together with the orthogonality matrix is opened by the **[Log]** button, **Figure 15.3.**

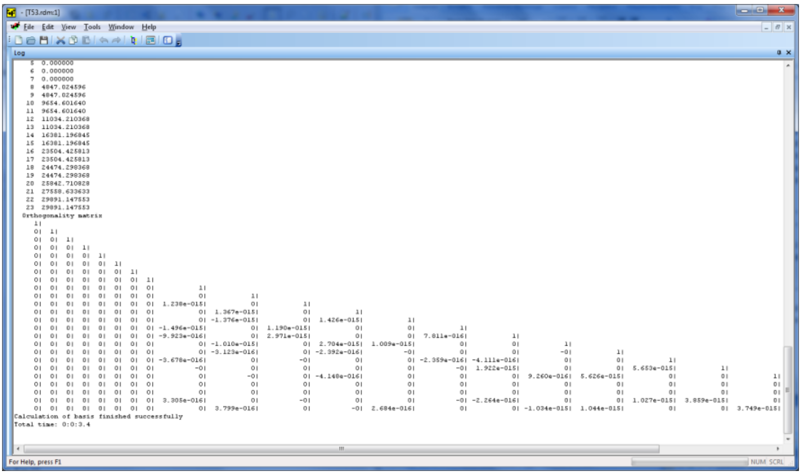


Figure 15.3

Together with the graphic image, the mode shapes may be shown in a table form. Also, the table shows distribution of potential and kinetic energies in subsystems, links and elements. The output may be done in two ways:

- by function **[Protocol]** (ref. chapter 17);
- in the main output window by command **[Table], Figure 15.4.**

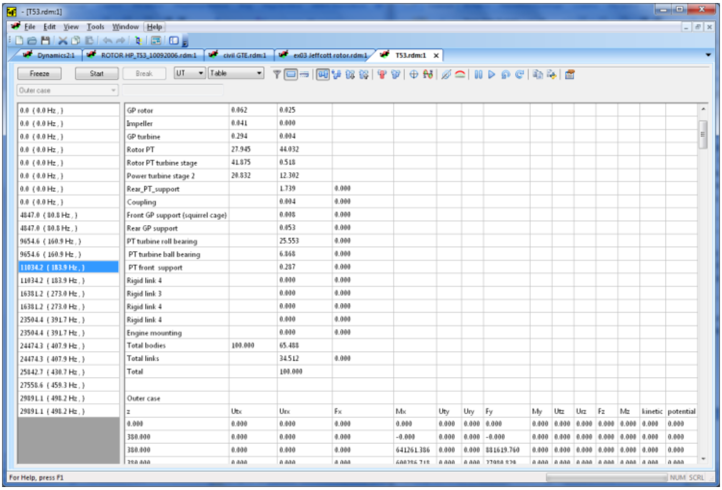


Figure 15.4

The data are output for each mode shape separately.

NOTE: As to avoid errors related to parametric studies the basis is re-calculated automatically before calculation of any algorithm if needed. Automatic basis recalculation takes place in case of parameters changing with influence on basis (geometry, mass and inertia, stiffness, new elements addition and etc.).
To speed up the calculations, it is recommended to calculate the Basis manually. Then, when starting each other algorithm, the recalculation of the basis mode shapes will not be carried out, which can take considerable time on complex systems.

NOTE: All calculated mode shapes are presented in the normalized form. The norm means reduction in relation to the mode kinetic energy

When analyzing the sensitivity of a dynamic system, it is useful to use information about the distribution of energies over links, subsystems, and their elements.

The distribution of kinetic energies in the [Basis] algorithms and the potential of inertia and gyroscopic forces in the [Natural frequencies] and [Critical speeds] algorithms show the degree of excitability of individual elements of the rotor-case model of a dynamic system. The elements with the highest kinetic energy will correspond to the antinode points on the mode shapes. Unbalances in these elements will have the greatest impact on the vibration level.

The dissipation energy distribution in elements of the [Link] type shows their contribution of the to the damping of oscillations in the mode shape under study.

The most useful in the study of a dynamic system is the distribution of potential energies over structural elements (in the [Natural Frequencies] and [Critical speeds] algorithms, the potential of elastic forces). The more potential energy in a section, the more deformations occur in this element. For rigid body mode shapes of oscillations, the dominance of potential energy in Links is typical. Bending, torsional mode shapes are characterized by maximum energy in subsystems (shafts, housings). If the potential energy in the Links is small, then the absorption of energy by this mode of vibration will be inefficient.

In the algorithms [Basis], [Natural frequencies] and [Critical speeds] it is possible to display information on energy distributions. In Table mode, information is displayed on the frequency selected in the list.

When [kinetic energy], [potential energy], [dissipation energy] is selected from the drop-down list, a summary table is displayed for all or selected frequencies.

According to the summary tables of energies, the following functionality is implemented:

1. Table output for all frequencies or for selected ones (selected with the Ctrl button pressed). It is preferable to select the desired shapes while viewing in 3D mode. Form selection is preserved when switching to other algorithms and back. If only one form is selected, then the table is displayed for all forms.

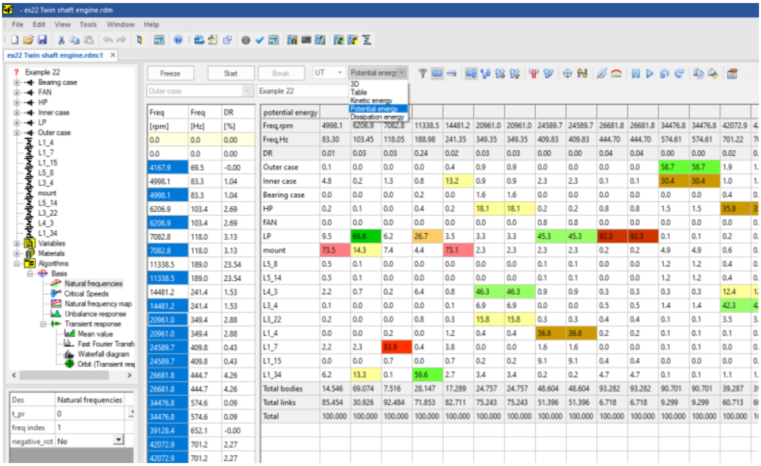


Figure 15.5

2. Color highlighting is done in 10% increments. Cells with energies less than 10% are not highlighted.

3. The table can filter out subsystems with energy levels in all forms below a given level, for example 1% (e.g. sensors, “rigid” links). The sum of energies by subsystems and Links is shown for the positions displayed in the table. If some subsystems and Links are filtered out, then the sum of energies will be less than 100% (used to control the representativeness of the table).

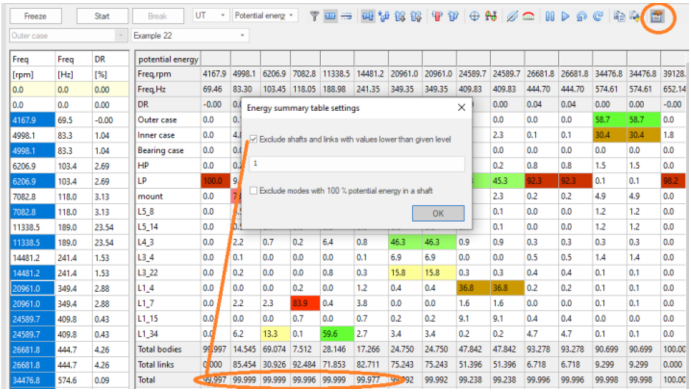
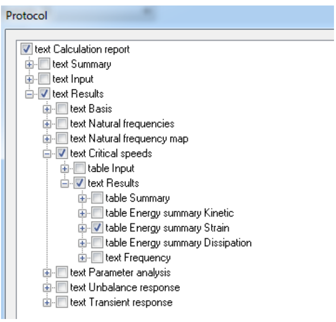


Figure 15.6

4. Frequencies with 100% potential energy may be excluded from the table (thus there may be fewer columns than the frequencies selected in the list). In most models, these will be torsional and less often axial oscillations of the rotors.

5. When you click on a cell in the energy table in the list of frequencies, the corresponding form is activated. You can switch to 3D mode and view this mode shape directly without resetting the frequency selection.

In the protocol mode it is possible to output information on energy in one table for all frequencies, subsystems and links (**Figure 15.7**). Tables are output separately for kinetic and potential energies.



Energy summary Strain

Designation	0	0	2308.77	2476.77	3107.9	5041.67	6339.81
	0	0	0.0229	0.0224	0	0.0025	0
	[%]	[%]	[%]	[%]	[%]	[%]	[%]
HP	0.0	0.0	0.5	0.6	0.0	0.26	2.98
LP	0.0	0.0	5.63	7.1	100.0	85.22	0.0
L2_2	0.0	0.0	0.05	0.26	0.0	0.0	97.02
L1_2	0.0	0.0	0.16	0.19	0.0	2.03	0.0
L1_19	0.0	0.0	0.17	0.2	0.0	0.39	0.0
L1_30	0.0	0.0	0.91	1.33	0.0	8.49	0.0
L1_52	0.0	0.0	1.18	0.99	0.0	0.78	0.0
L1_58	0.0	0.0	91.38	89.34	0.0	2.82	0.0

Figure 15.7

The protocol data may be copied to clipboard and then exported to some programs, for example **Microsoft Office Excel ®,** for further graphic presentations.

The accuracy of the calculation results of all algorithms in the program depends on fullness of basis calculation. The more shapes are calculated in basis, the more accurate the results of modal algorithms. But the fuller basis is calculated, the longer calculation of the other algorithms. Algorithms of equations systems reduction on the basis of the Modal synthesis methods allow receiving fast calculation and acceptable results accuracy. In the majority of cases for Unsteady calculations (only linear elements, “small” gyroscopes) it is enough to use doubled in correspondence to operating speed basis calculation frequency range. In case of the presence of non-linear elements, elements with high gyroscopic moments (depend on the value of diametric inertia moments and rotating speed), quasi-linear links with significantly changing stiffness and damping coefficients at regimes (especially typical of journal bearings), it is necessary to define the system at optimum ratio of efficiency and calculation accuracy. For this several calculations are done for different basis ranges and that range is determined where change in results is insignificant. In linear algorithms they may be frequencies in the **[Natural frequency map]**, position of resonances on the amplitude-frequency characteristic of the **[Unbalanced response]** algorithm. For example, 5-10% accuracy of oscillation frequency or resonance position may be enough for a user.

It is possible to check reactions correctness in Nonlinear analysis by summing of reactions of all links and comparing with the applied load (loads sum). This is the most exact method but it may be difficult to apply in complicated models. Several calculations may also be carried out with different basis range. Change in reactions value is controlled.

15.2 Control of output and imaging of the modes calculation results

Figure 15.8 shows output control and imaging buttons related to the natural frequencies and mode shapes.

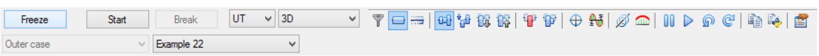


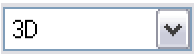
Figure 15.8

- Save pictures constantly in the central window. The pictures may include a model or analysis results. For example, a user may fix the transient analysis results and continue working with the model.

- Startup of a dynamic model calculation with an earlier selected algorithm.

- Break of the calculation procedure.

- Identifier of the mode shape imaging. A mode may be presented in displacements, or turn angles, or forces, or moments.



- Mode shape output format, graphic 3D, or table.

- Change to output of one subsystem.

- Subsystem selection. Command works after pressing of the button .

- Displays the precession orbits in the screen plane (to show the critical speeds modes). Pressing the button several times changes the model view. Left-side view, right-side view

- Shows the mode shape in the screen plane. Sequential pressing changes view in XZ, YZ planes

- Highlights or deletes the precession motion orbits of the rotor system points (to display critical mode shapes).

- switching on of the shape line display

- Controls of the mode shape image animation.

- Pause.

 - Start the animation.

- Discrete animation in counter clockwise direction.

- Discrete animation in clockwise direction.

- Copy the mode shape into the exchange buffer.

 - Copy the mode shape into the exchange buffer with automated insertion into the last active opened Word document (installed Active Python 2.7 is needed)

 - Opens window to set the display options of the 3D view of the present model (Figure 15.9)

The options include:

- Defining of the window background color
- Control of the mode shapes visibility through the model
- Coefficient of scale of the subsystems and assemblies frame is 0 – display is absent
- Coefficient of scale of bending mode shapes
- Coefficient of scale of torsion mode shape
- Scale of displacement of the shape in 3D animation regime and unbalanced response
- Scale of the orbits in 3D window of the orbit postprocessing

"Wheels" in the bottom part of the screen rotate the image around the X, or Y axis, or scales the image.

A small cursor movement with the left button fixed on the mode image may provoke the model image and shapes to move in the indicated direction. The movement may be stopped with the following left button pressing.

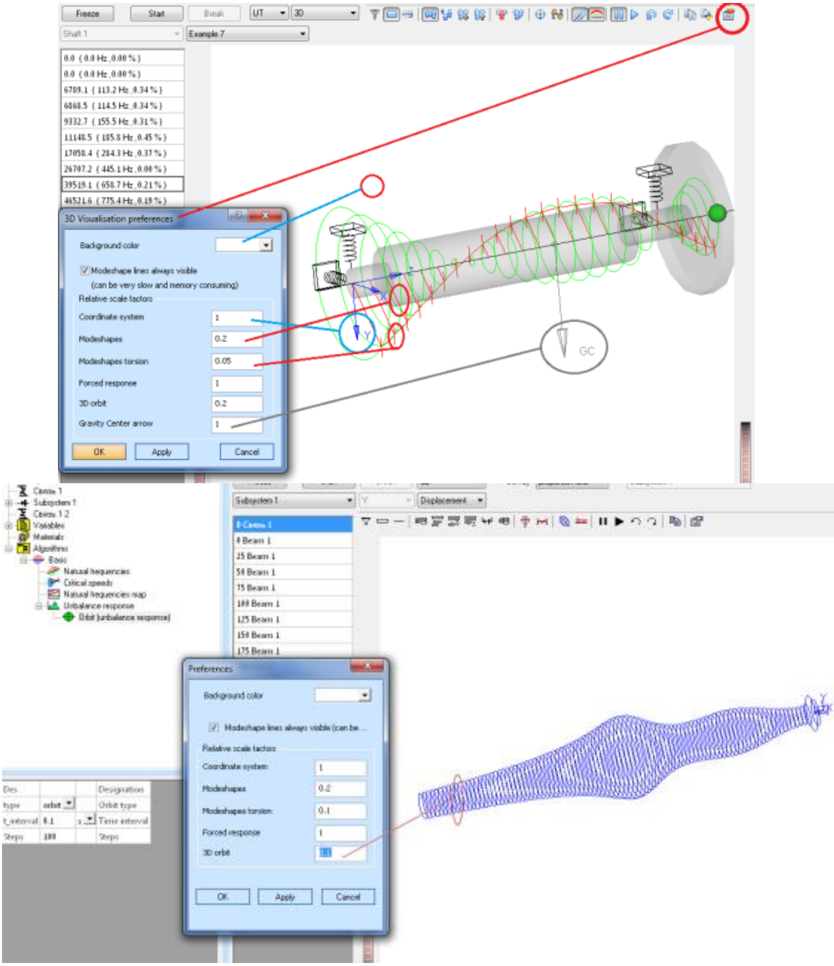


Figure 15.9

15.3 Natural frequencies

This algorithm calculates natural frequencies and modes with consideration of the system damping and rotation.

On the contrary to the basis algorithm the modes are calculated with consideration of the rotor rotation. The rotor moves along circular orbits, **Figure 15.10.**

The direct and the reverse precession motions are highlighted with green and blue respectively.

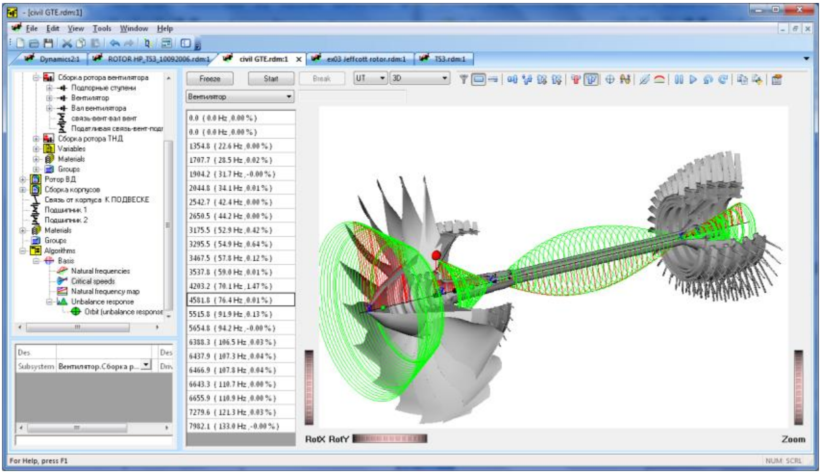


Figure 15.10

In the **[Log]** natural frequencies values are output as complex numbers with both real and image parts.

The algorithm control window is shown in Figure 15.11.

Des			Designation
t_pr	5000		Time parameter

Figure 15.11

The rotors speeds will be selected from a regime in accordance with the **t_pr** parameter.

Together with the graphic the mode may be presented in the table form.

Distributions of the potential and kinetic are also available

15.4 Natural frequency map

The **[Natural frequency map]** shows the influence of the rotor speed upon the frequencies of direct and reverse precessions, **Figure 15.12**.

The map shows the natural frequencies lines and the rotor speed lines. The descending and ascending lines show reverse and direct precessions respectively. Intersections of the rotor speed lines with the natural frequencies ones correspond to the system critical speeds.

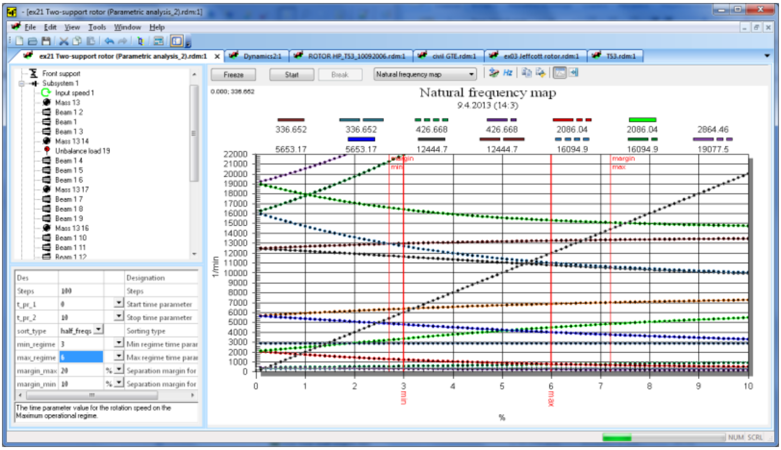


Figure 15.12

The Natural frequency map is given in the user determined frequency range from **[t_pr_1]** to **[t_pr_2]**.

The number of calculation steps within the given range is defined by the parameter **[Steps].** Parameters **[max_regime]** and **[min_regime] –** are the time parameter values for the rotation speed on the minimum and maximum operational regimes. Parameters **[margin_max]** and **[margin_min]** are additional boundary lines defined by a user. Parameter [**update_results**] - if "Yes" is selected then the data on the chart will be updated during the calculation with a given step [**upd_step**] as a percentage of the total calculation, the graph will be redrawn. If set to 0, it will be updated at every calculation step.

The calculation starts with the **[Start]** button**. Figure 15.13** shows the parameter window.

Des			Designation
Steps	100		Steps
t_pr_1	0		Start time parameter
t_pr_2	1e+06		Stop time parameter
sort_type	half_freqs		Sorting type
zoom	No		Zoom NFM max(25%,SMmax) above maximum regime
min_regime	0		Min regime time parameter
max_regime	1e+06		Max regime time parameter
margin_max	20	%	Separation margin for max regime
margin_min	5	%	Separation margin for min regime
update_results	yes		Update results while calculation
upd_step	10		Results update step as a percentage of the total calculation

Figure 15.13

 - The button gives opportunity to add the Natural frequencies algorithm into the model applying settings corresponding to the current point on the frequency plot. Thus, if a user wants to look at the mode shape for the specific frequency at the certain regime, he may click at the point on the line at the Natural frequencies plot; a cursor will be placed at the point on the plot. After that a user may press a button and go to the new created algorithm **[Natural frequencies]**. Meanwhile, the parameter t_pr corresponds to the current point, and the needed frequency will be highlighted in the frequency range after recalculation.

 - Button for the models with contrarotating rotors to build the Natural frequency map, the output of the rotors speeds at modulus is reasonable. Pressed button (on default) serves for compulsory display in the positive area of the rotor speeds in negative direction.

The Stability map may be output together with the Natural frequency map, **Figure 15.14.**

Depending upon the "time proportion" parameter, each mode is described with its **[Logarithmic Decrement],** or the relative **[Damping Ratio].**

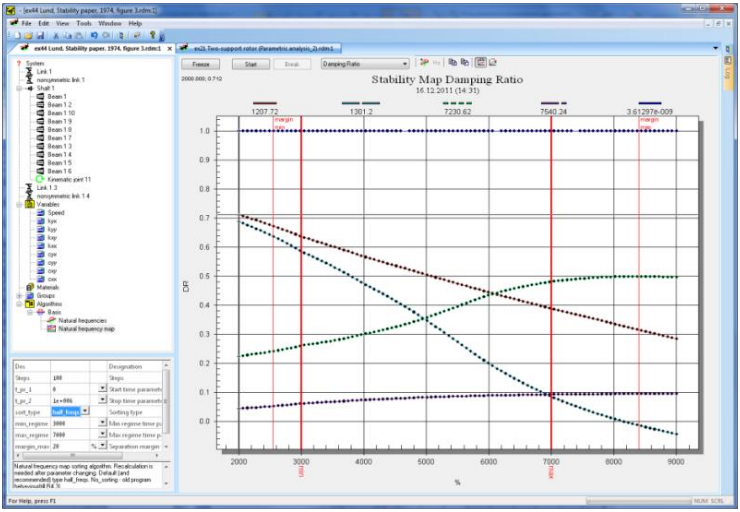


Figure 15.14

A relation between the damping ratio coefficient and the Natural Frequency may be obtained by Locus chart Logarithmic Decrement (Locus Lndec) and Locus chart Damping Ratio (Locus DR), **Figure 15.15**.

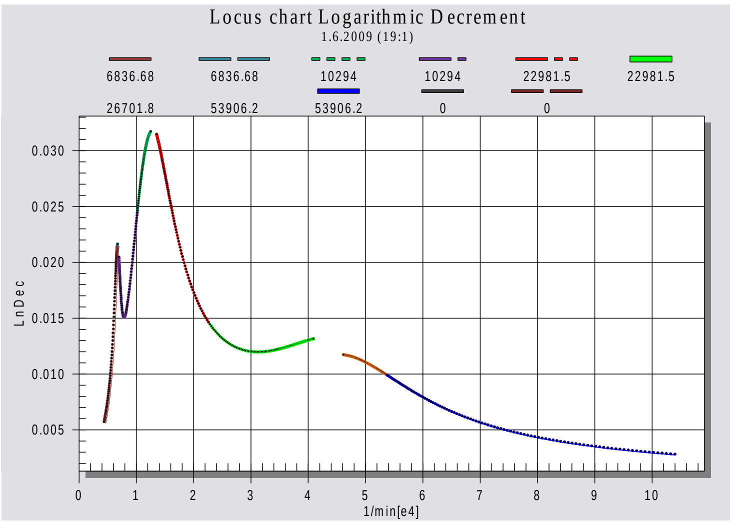


Figure 15.15

For the map Locus DR, the results output may be obtained additionally in correspondence with the requirements of the API 610 standard - Impellers (**Figure 15.16**).

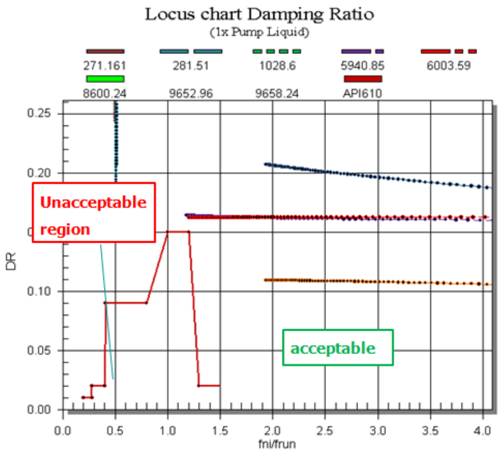


Figure 15.16

Ratio of natural frequencies to rotating speed at the regime (fri/frun) is given in X-direction. There is an additional output of the boundary of the region if found in which it is necessary to carry out additional calculations of the system.

15.5 Critical speeds

The **[Critical speeds]** algorithm calculates free and damped mode shapes of the system. In a multi-rotor system the modes may be calculated for each of the rotors separately, **Figure 15.17**

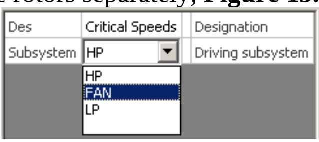


Figure 15.17

Critical speeds modes are displayed in the output window similarly to the natural modes, **Figure 15.18** and **Figure 15.19.**

NOTE: The critical speeds are calculated only when the frequency range is defined with the **[Kinematic joint]**

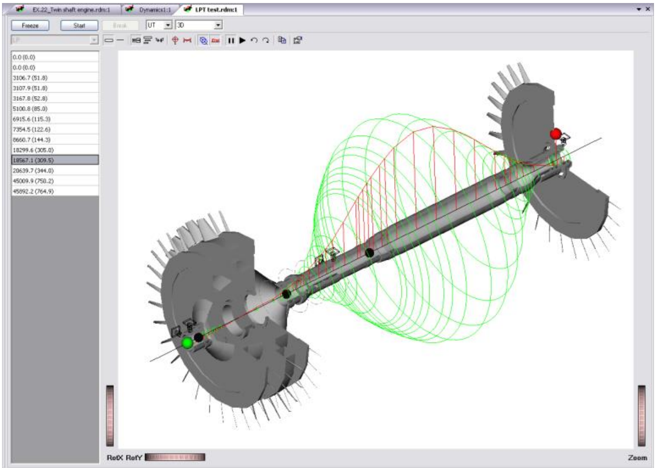


Figure 15.18

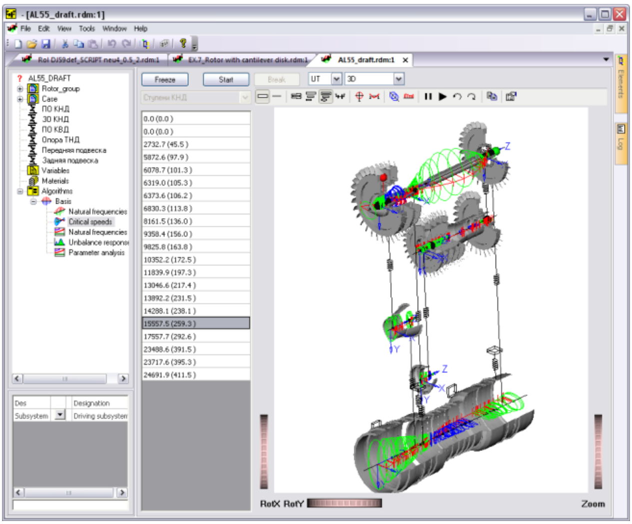


Figure 15.19

Like for the natural modes here are available distributions of potential and kinetic energies.

15.6 Unbalance response

Select the **[Unbalance response]** option to compute response to rotating mass imbalance.

The forcing function will be defined by imbalances you enter by means of **[Unbalance load]** element.

The **[Unbalance response]** algorithm shows behavior of the system under unbalance loads.

The unbalance response may be calculated for cases of a single unbalance, or systems of unbalances in one or a few rotors.

The steady state unbalance response gives displacement values in all points of the model. The displacement occurs with the rotor frequency.

Dynamic behavior of a multi-rotor system may be calculated as a transient analysis.

NOTE: The unbalances are determined together with the model building. A group of unbalances may be created with the **[Group]** element. Using different groups gives influences of different unbalance groups

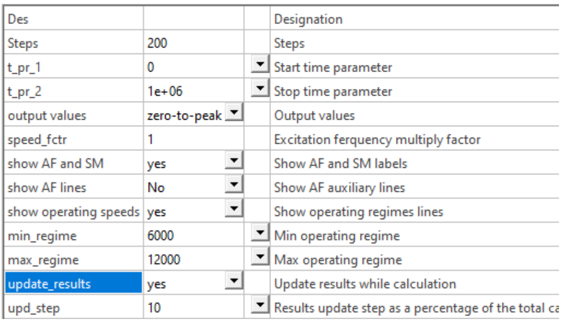
The **[Unbalance response]** algorithm is followed by its input and output fields.

Figure 15.20 shows the input window for the calculation and results output steps. The steps determine input and output of Magnitude-Frequency (MF) and Phase-Frequency (PF) functions.

Table 14.1 Description of elements parameters

Designation	Parameter description
Steps	Number of points of Amplitude-frequency characteristic in whole calculation range. On graph points are connected by linear sections.
t_pr_1	Parameter determines left boundary of Amplitude

	Frequency characteristic output
t_pr_2	Parameter determines right boundary of Amplitude Frequency characteristic output
show AF and SM	Yes or No. Control of values AF SM output on plot (see below)
show AF lines	Yes or No. Output of additional lines on the plot to calculate AF
show operating speeds	Yes or No. Output of additional lines showing boundary of operating modes on the output
min_regime	t_pr corresponding to the minimum regime
max_regime	t_pr corresponding to the maximum regime
update_results	The data on the chart is updated during the calculation with a given step [**upd_step**]
upd_step	during the calculation with a specified step as a percentage of the total calculation, the graph will be redrawn. If set to 0, it will be updated at every step.



Des		Designation
Steps	200	Steps
t_pr_1	0	Start time parameter
t_pr_2	1e+06	Stop time parameter
output values	zero-to-peak	Output values
speed_fctr	1	Excitation ferquency multiply factor
show AF and SM	yes	Show AF and SM labels
show AF lines	No	Show AF auxiliary lines
show operating speeds	yes	Show operating regimes lines
min_regime	6000	Min operating regime
max_regime	12000	Max operating regime
update_results	yes	Update results while calculation
upd_step	10	Results update step as a percentage of the total ca

Figure 15.20

The output control window is shown in **Figure 15.21.**

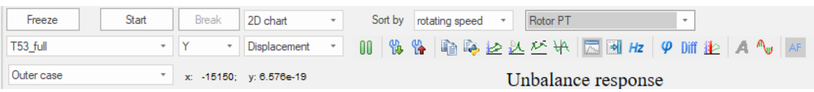
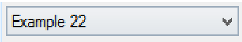
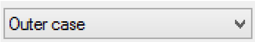


Figure 15.21

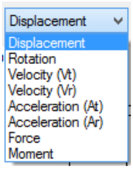
The unbalance response output is controlled by the following windows:



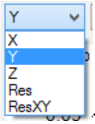
- filter at assemblies for the subsystems and links list



- Subsystem for the data output;

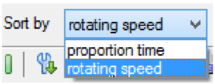


- choice of the parameter to output data;

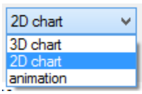


- choice of the projection of the vector parameter. Res – output of resultant values ($\sqrt{x^2+y^2+z^2}$); ResXY – output of resultant values ($\sqrt{x^2+y^2}$);

- choice of the basic subsystem for the data output;



- choice of the parameter of the X axis;



- the plot format choice;

 - calculation stopping;

 - save the plot settings (the program remembers the plot settings between startups);

 - apply the previously saved settings to the plot;

 - copy the plot into the clipboard. Image of the plots is copied with the size 1500* 750;

 - the plot is copied into clipboard and pasted into the latest opened Word document (the presence of the installed ActivePython 2.7 is necessary in the system);

- values for the highlighted point on the plot are copied into the clipboard and output in the log panel for one or several plots;

- start the dialog of the plot export;

- output of legends for extremums on the plot. Only legend of the biggest at magnitude value is output at the pressed button Ctrl. The legend for the current point on the plot is output at the pressed Shift button;

- set the cursor by the point index on the chart or the value on the X axis in the [Unbalance response] and [Transient response] algorithms

- buttons of control by output tof the legend to the plots;

translation of the axis from rpm to Hz when sorting by revolutions in the results of the [Imbalance behavior] and [Nonlinear analysis] algorithms

- button of adding phase-frequency characteristics to the plot of amplitude-time-characteristics

- is used to calculate and output deformations and reactions;

- Simultaneous output of several plots with independent Y axes. For example, superposition of rotating speed chart on the vibration displacement plot

- output of amplification factors and reserve in detuning from operating modes in correspondence with API RP684.

If 2D plot is output, a user should determine the output subsystem and section, **Figure 15.22.**

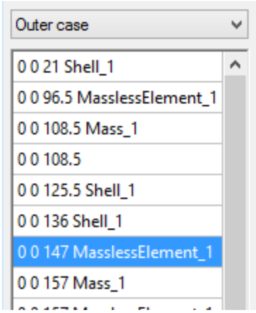


Figure 15.22

Figure 15.23 shows the window of the obtained amplitude-time characteristics. Simultaneously a user can output amplitude-time characteristics at all or several sections of the subsystem .

Note: Unbalance response results shows **half** of peak-to-peak values.

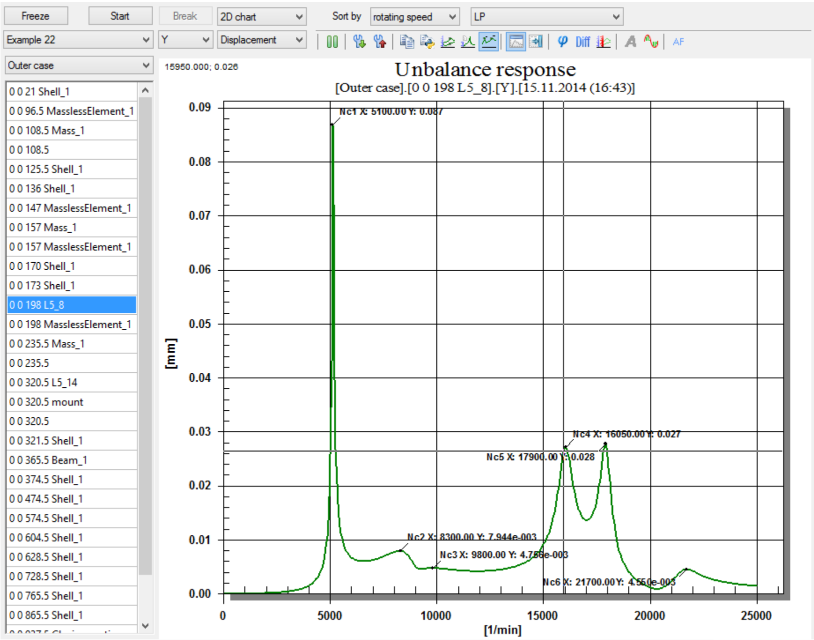


Figure 15.23

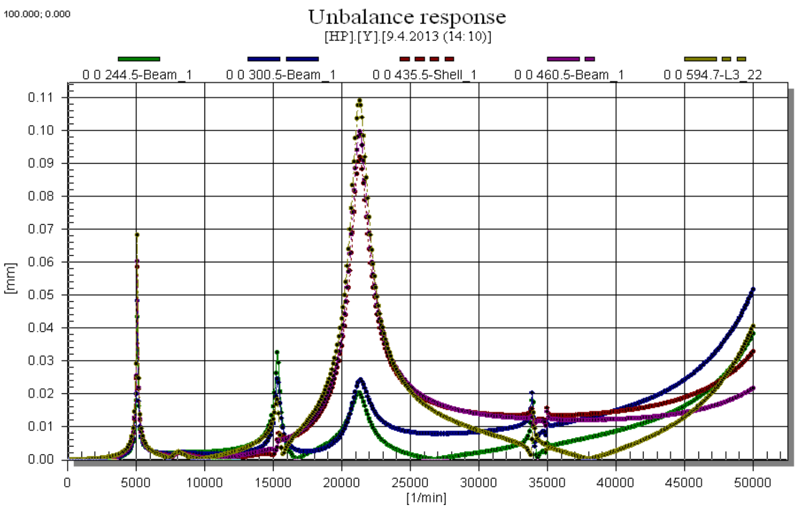


Figure 15.24

Also, the user can create a list of sections from different subsystems and connections. To do this, add it to the new list through the context menu of the section. Then add other sections of the model to this list. A position with the created list will appear in the list of subsystems above the sections.

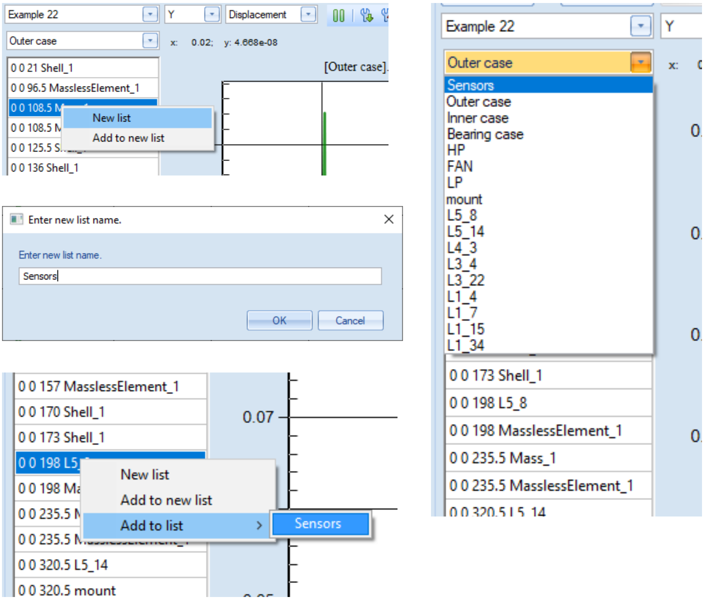


Figure 15.25

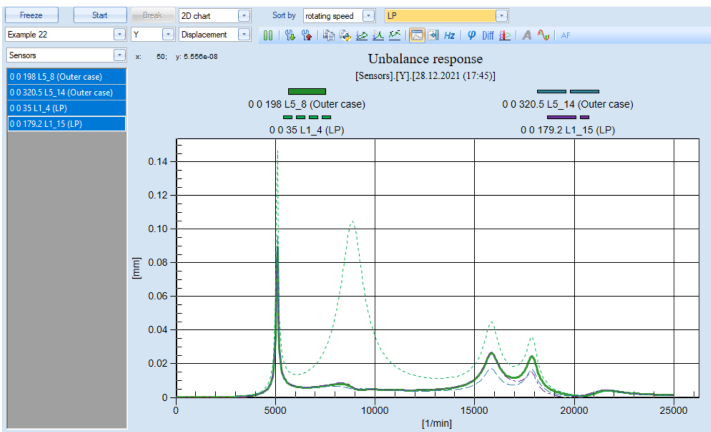


Figure 15.26

The context menu of stations from the created list contains commands for deleting sections and deleting a list.

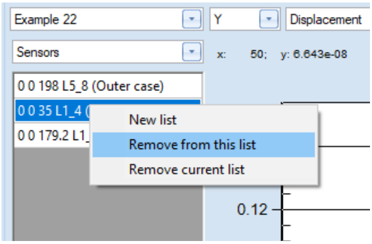


Figure 15.27

Sensor’s functionality is used when it is necessary to display the results on one graph from sections in different subsystems/shafts or Links. For example, display the response for all bearings or sensor mounting sections.

Button [AF] () - Amplification Factors – gives opportunity to output plots of amplitude-frequency characteristics in correspondence with recommendations of the API RP684 standard

API Standard Paragraphs Rotordynamic Tutorial: Lateral Critical Speeds, Unbalance Response, Stability, Train torsionals, and Rotor Balancing (API RP 684), Second Edition, august 2005.

Figure 15.28 gives the parameters controlled according to API.
Nc1 – the first rotor critical speed
Ac1 - vibration amplitude of the first critical speed
Ncn - critical speed with number n
Nmc - maximum rotating speed
N1 – smaller speed corresponding to the amplitude 0.707 Ac1
N2 – bigger speed corresponding to the amplitude 0.707 Ac1
N2-N1 – peak width with half of oscillations energy
AF=Nc1/(N2-N1) – amplification factor (Amplification Factor)
SM - reserve in resonance detuning (Separation Margin)
SMr – required separation margin
SMa – actual separation margin

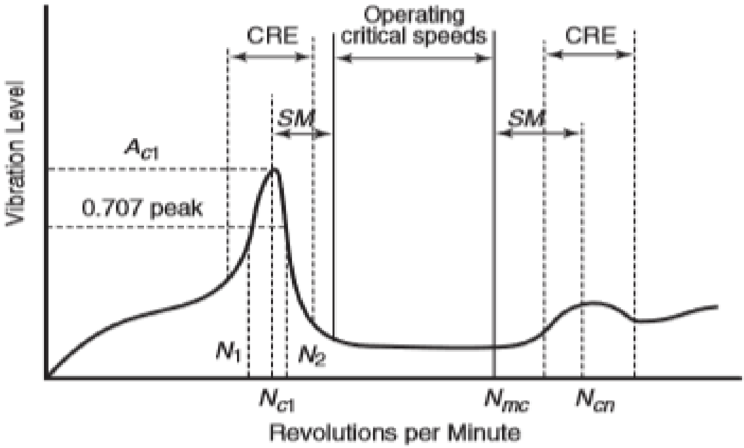


Figure 15.28

If due to big damping values at critical speed, N1 or N2 cannot be obtained, the legend "AF=?" is attached to the plot.
According to API, critical speeds with AF<2.5 are considered critically damped, and SM obtainment is not required.
If AF>= 2.5 and critical speed is lower than operating modes range, SM (in percentage) should be not less than the one calculated using the equation or 16% (the smallest value is taken)

$$SM = 17 * \left(1 - \frac{1}{AF - 1.5}\right)$$

If AF>= 2.5 and critical speed is higher than operating modes range, SM (in percentage) should be not less than the one calculated using the equation or 26% (the smallest value is taken)

$$SM = 10 + 17 * \left(1 - \frac{1}{AF - 1.5}\right)$$

Except output of information on the plot (**Figure 15.29**), data are also output in the log.
If the required SMr is bigger than actual SMa, the legend to this peak is given in red. SM for the peaks being within the operating modes range is not calculated.

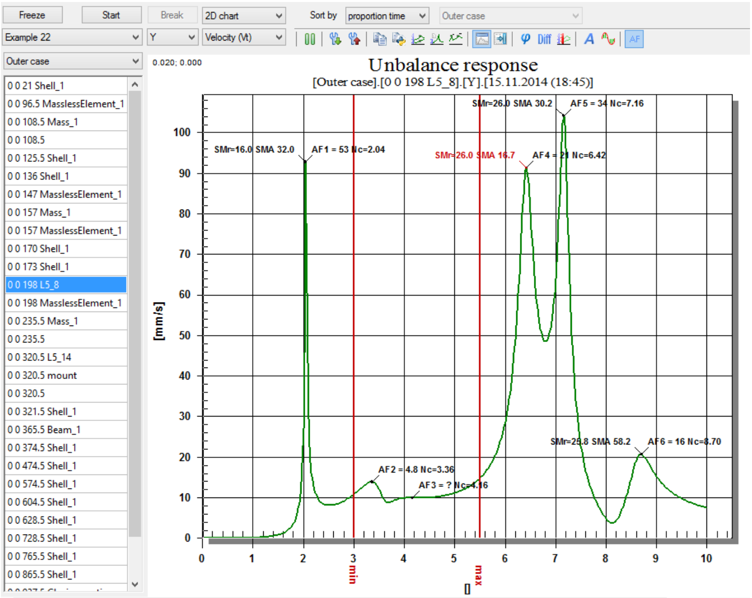


Figure 15.29

The  button is used to calculate deformations or reaction forces. For this, pressing **[Ctrl]** button, a user points to the two curves whose difference he would like to get. For example, deformation of the shaft middle relatively to its support may be obtained. The support reaction may be also obtained using this function (if highlight forces before the link and after it)

Output of a 3D plot of the unbalance response is similar. **Figure 15.30** and **Figure 15.31** shows response curves for displacement and force.

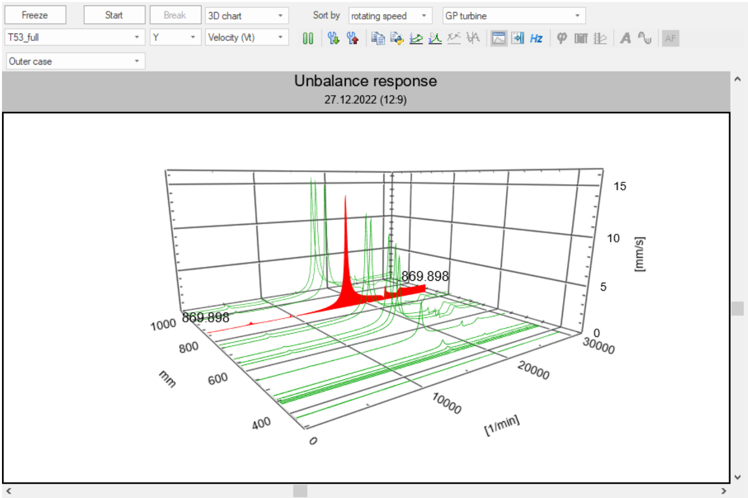


Figure 15.30

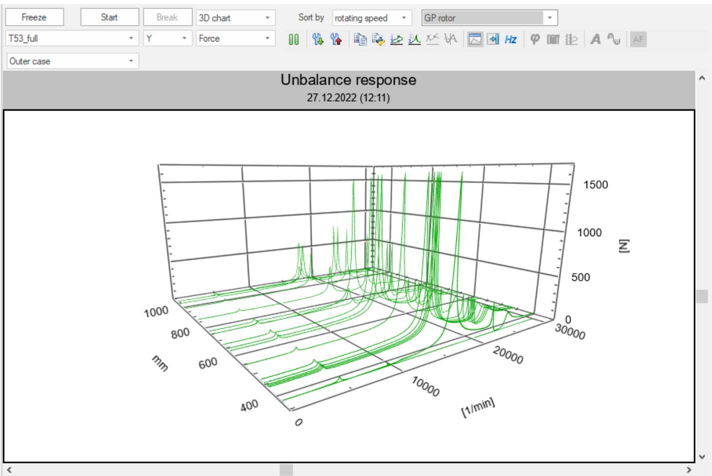


Figure 15.31

One more option for the unbalance response output is the **[Orbit (Unbalance response)]** algorithm that shows the motion orbits. The algorithm may be taken from the algorithm library into the project database.

The **[Orbit (Unbalance response)]** algorithm has its output control windows for the motion orbits (**Figure 15.32**).

Des			Designation
type	orbit		Orbit type
t_interval	10	s	Time interval
Steps	10		Steps

Figure 15.32

and the phase shifts (**Figure 15.33**).

Des		Designation
type	phase diagramm	Orbit type

Figure 15.33

The plots formats are controlled by the graphic menu. To call the menu (**Figure 15.34**) put the cursor into the plot field and press the right button.

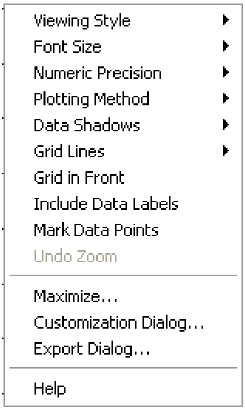


Figure 15.34

The plots may be edited (**Figure 15.35)**, or copied in different formats into clipboards, or into a file, or to a printer (**Figure 15.36)**.

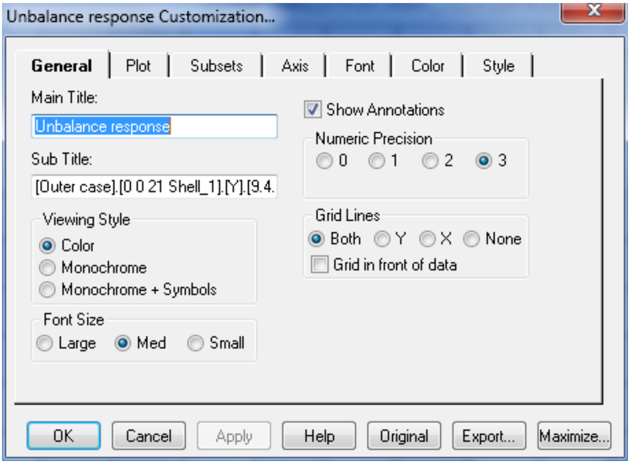


Figure 15.35

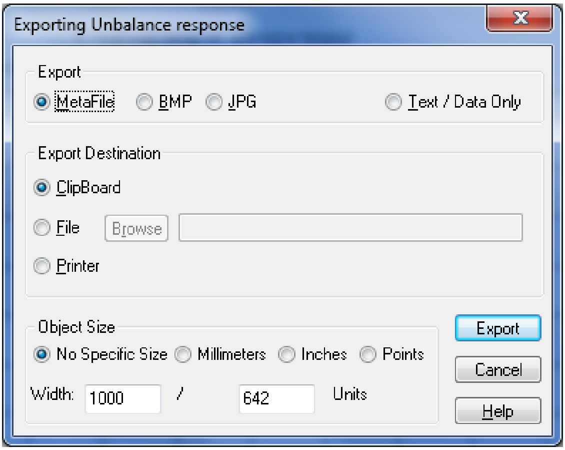


Figure 15.36

It is also possible to output data in the text format and use them in other systems, for example, **Microsoft® Excel**. The text format output control is shown in **Figure 15.37**.

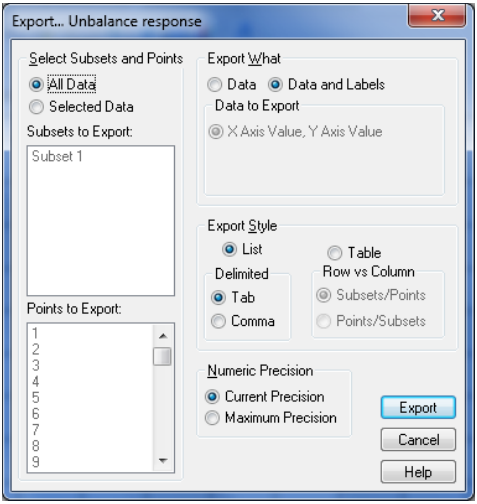


Figure 15.37

15.7 Parametric analysis

15.7.1 One-dimensional parametric study

The parametric study relates the system parameters and its dynamic properties. The study may be carried out versus different parameters of the model.

The analysis result is a relation map of the Natural frequency, or the critical speed or the Logarithmic decrement versus the verified parameter.

NOTE: The parametric analysis map may be obtained for any parameter of the system that describes a link, or a station, or a coupling, of a mass, or a disc. The location coordinates may be verified for the zero length elements, couplings, links, and masses.

A parametric study map may be obtained by the following actions:

- Select the verified parameter. For example, it may be the stiffness coefficient **kyy = [ut_x/Fx]** in the stiffness matrix **[k]** that determines the support stiffness

- Create the variable, for example **[Front kyy, N/m]**

- Assign a constant value to the variable. For example, the value 10000 N/m that will be used for the basis calculation is assigned to the **[Front kyy, N/m]**, **Figure 15.38**

Figure 15.38

- Attach the variable to the stiffness matrix coefficient **[ut_x/Fx], Figure 15.39**

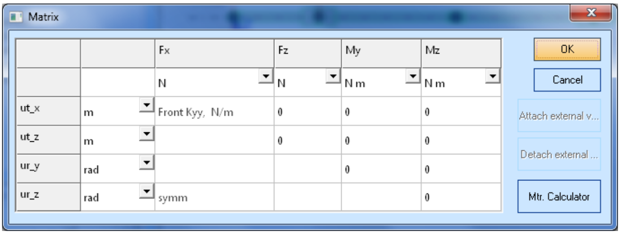


Figure 15.39

- Add the **[Parameter analysis]** to the project algorithm list

- Select the **[Parameter analysis]** in the algorithm list. In the left bottom screen there appears the algorithm parameter window, **Figure 15.40**.The options here are:

 - determine the algorithm type – [**Critical speeds]** or [**Natural frequencies]**;

 - determine the rotor or subsystem for the natural frequency analysis;

 - determine the output - [**Natural frequencies]** or **[Decrements]**;

Des	Critical Speed Map	Designation
inp_parameters	...	inp_parameters
algorithm	Critical speeds	Type of processing algorithm
Subsystem		Driving subsystem
output_type	frequencies	Type of results
recalculate_basis	yes	Is Basis has to be recalculated
update_results	yes	Update results while calculation
upd_step	10	Results update step as a percentage of the total calculation

Figure 15.40

Parameter [**update_results**] - if “Yes” is selected then the data on the chart will be updated during the calculation with a given step [**upd_step**] as a percentage of the total calculation, the graph will be redrawn. If set to 0, it will be updated at every calculation step.

In the **[Natural frequencies**] algorithm the time parameter **[t_pr]** determines the investigated regime, **Figure 15.41**.

Des	Parametric maps	Designation
inp_parameters	...	Input parameters
algorithm	Natural frequencies	Type of processing algorithm
t_pr	8	Time parameter
output_type	frequencies	Type of results
recalculate_basis	yes	Is Basis has to be recalculated

Figure 15.41

- select the **[Recalculate Basis]** value. If the value is **YES,** the basis will be re-calculated, if – **NO,** the basis is not re-calculated. The latter is used when the verified parameter does not influence the basis.

- Go to the **[inp_parameters]** line and open the algorithm parameters input window with the **[Extended properties]** command, **Figure 15.42.**

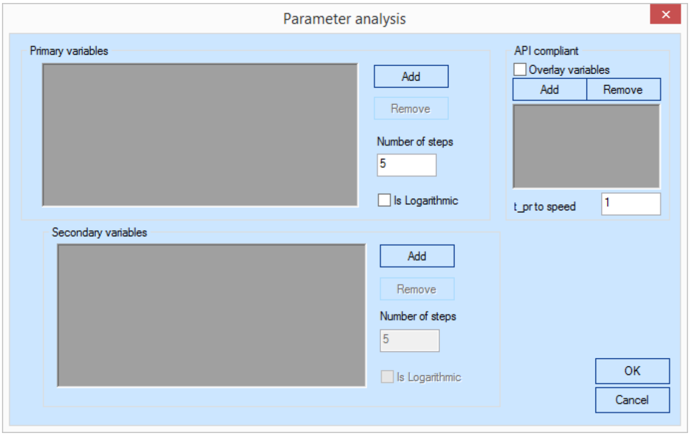


Figure 15.42

- With the button **[Add]** add the initial variable **[Primary values]**, **Figure 15.43**

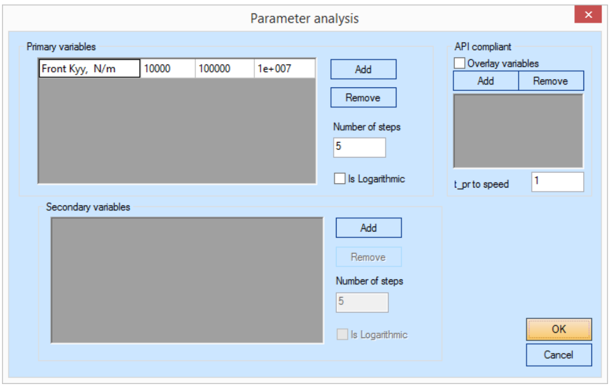


Figure 15.43

- Input the variable boundaries, **Figure 15.44**

In the part "API compliant" a user can define the additional values on the plot. For example, they may be the values of stiffness of journal bearings at the regimes when building the critical speed map.

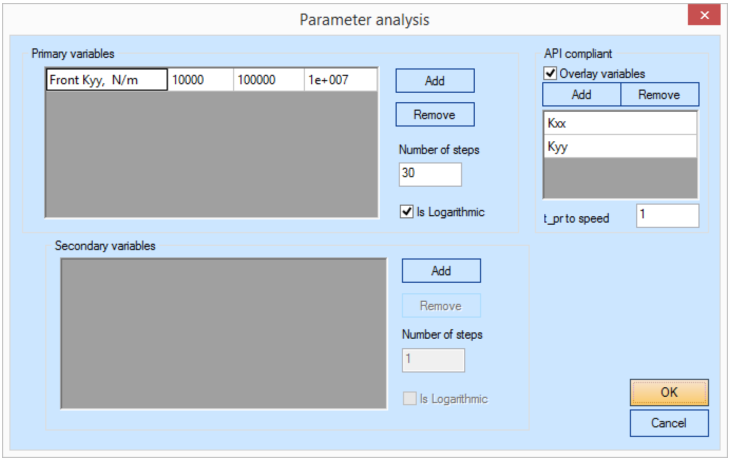


Figure 15.44

- Input the **[Number of steps]** for the parametric analysis map.

NOTE: The Parametric analysis step may change under the logarithmic dependence by selection of the **[Is Logarithmic]** mark placing a flag there. This flag should be set in case if the range of the variable change accounts for several degrees.

- Begin the calculation by **[Start].** The result will be, for example the critical speed spectrum at the lower value of the verified parameter, **Figure 15.45**.

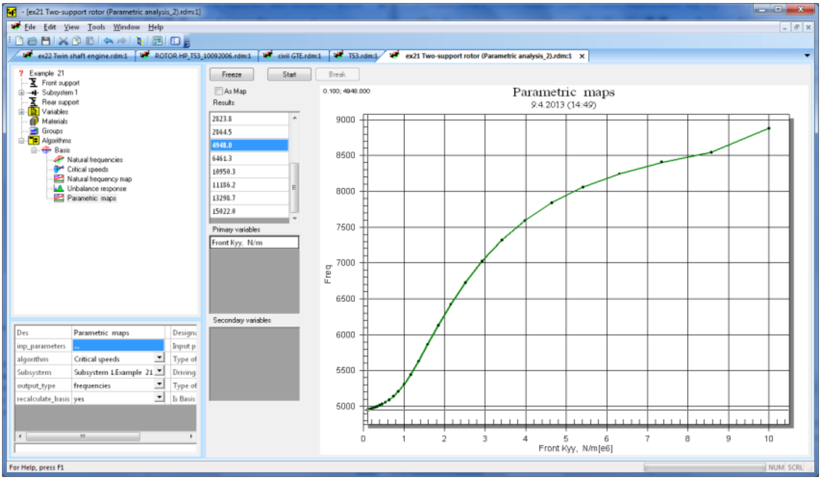


Figure 15.45

- If the flag "As Map" is set, a user can obtain the parametric analysis map, for example, the critical speed map, **Figure 15.46.**

NOTE: The number of primary values [Primary values] may be unlimited

Values of the calculated frequencies or decrements may be obtained by the **[Export dialog]** command of the graphic output menu. The commands sequence **[Export dialog] – [Text/data only] – [Export…] – [Export]** forms a text file containing the frequency or decrement values versus the verified parameter.

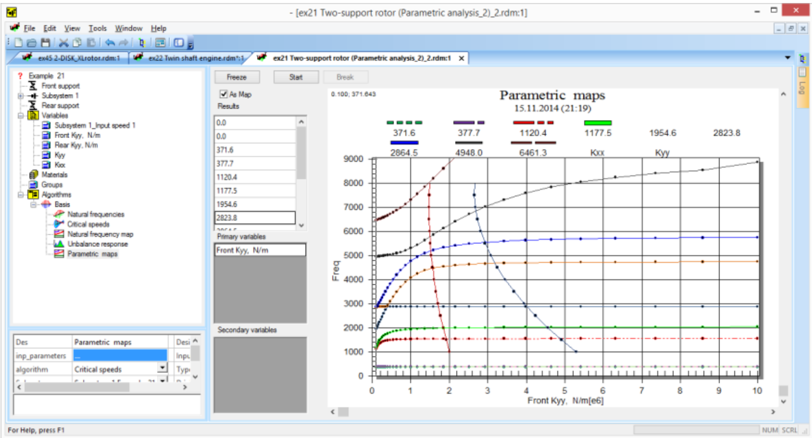


Figure 15.46

15.7.2 Two-dimensional parametric study

When a two-dimensional analysis is needed, a **[Secondary variables]** should be added**.** Parameters of this variable are input in a similar way to the one-dimensional analysis, **Figure 15.47.**

> **NOTE:** The number of secondary variables **[Secondary values]** is not limited

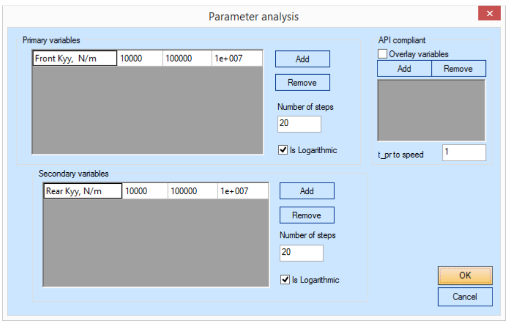


Figure 15.47

The first result of the calculation is the frequency spectrum calculated at the lowest values of variables. For any of the frequencies may be selected a surface where every point reflects a critical speed with the number equal to the initial spectrum, **Figure 15.48**

NOTE: If variables are attached to the links of the [Link] type with take part in the algorithm [Basis] calculation, when calculating every parameter combination, basis is recalculated. For complex models it may take a lot of time. In such cases it is recommended to use the link of the [Elastic link] which is not used in basis calculation.

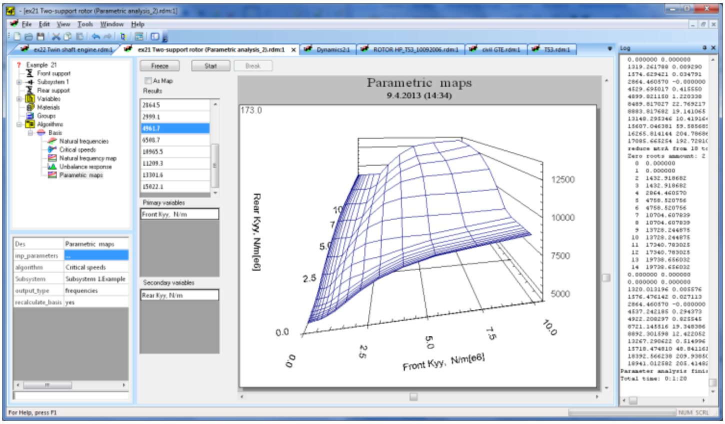


Figure 15.48

Values of the frequencies or decrements may be obtained by the **[Export dialog]** command of the graphic menu which may form a text file with the map data.

15.8 Aero loads simulation

Aero loads in labyrinth seals or flowpath elements may cause a rotor instability. Natures of these loads are similar and are related with gas flow non-symmetry in blades or seals.

Aero excitement criteria described in this chapter give evaluations of the non-stability effects. The criteria are included into Dynamics R4.

The algorithm result is a stiffness matrix that simulates aero interaction between a rotor and a stator. The matrix contains two stiffness coefficients for axial or centrifugal compressor or turbine stages.

The cross-coupling stiffness matrix 6x6 has the following view:

$$\begin{bmatrix} 0 & K_{xy} & 0 & 0 & 0 & 0 \\ K_{yx} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

15.8.1 Alford's formula

The Alford's formula gives cross stiffness aero links between rotor and stator in axial or radial stages:

$$K_{xy} = -K_{yx} = \frac{\beta \cdot P}{\omega \cdot D \cdot H},$$

where

K_{xy} , K_{yx}	Н/м	Cross - coupling stiffness coefficients
P	Н·м /s	Stage power
ω	rad/s	Rotor speed
D	м	Blade pitch diameter
H	м	Blade height
β		Stage efficiency factor

β=0.5	For shrouded axially bladed disks
β =1.5	For un-shrouded axially bladed disks
β =2 … 3	For un-shrouded radial flow impellers
β =5 … 10	For extreme cases, overhung impellers

15.8.2 Modified Alford's equation

The American Petroleum Institute (API) has implemented new rotordynamic stability specification for centrifugal compressors. The specifications consist of a Level 1 analysis that approximates the destabilizing effects of the labyrinths seals and aerodynamic excitations.
A modified Alford's equation is used to approximate the destabilizing effects. If the compressor fails the Level 1 specification, a more sophisticated Level II is used. It gives a detailed labyrinth seal analysis.

$$K_{xy} = -K_{yx} = \frac{\beta \cdot 9554 \cdot P}{N \cdot Dc \cdot Hc} \cdot \rho_{ratio}$$

Kxy , Kyx	Н/м	Cross-coupling stiffness coefficients
9554	m·N·rpm/kW	Empirical dimensional constant
P	kW	Stage power
N	rpm	Rotor speed
Dc	м	Impeller Diameter
Hc	м	Minimum width of the impeller or discharge volute
ρ_{ratio}	-	$\rho_{ratio} = \rho_{out} / \rho_{in}$
ρ_{in}	kg/m3	Inlet Gas Density
ρ_{out}	kg/m3	Outlet Gas Density
β	-	Stage efficiency factor

15.8.3 Wachel's equation

It is well-known Wachel's equation, which is used to estimate the overall aerodynamic cross-coupling applied on the compressor rotor in the stability analysis. Wachel's cross-coupling equation resulted from correlation of rotor instability experienced by centrifugal compressors and has taken into account all destabilizing effects from impeller, labyrinth seals, oil seals, etc.
Gas properties, stages power absorbed, and impeller/diffuser geometries should be known to use this formula.

$$K_{xy} = -K_{yx} = \frac{955 \cdot P \cdot M}{N \cdot D \cdot H} \cdot \rho_{ratio} ,$$

K_{xy} , K_{yx}	Н/м	Cross -coupling stiffness coefficients
955	N·m·rpm/kW	An empirical dimensional constant
P	kW	**Stage power**
N	rpm	**Rotor speed**
M	а.е.м	**Gas Molecular Weight**
D	м	**Impeller Discharged Diameter**
H	м	**Gas Passage Width**
ρ_{ratio}	-	$\rho_{ratio} = \rho_{discharge} / \rho_{suction}$
$\rho_{suction}$	kg/m3	Inlet Gas Density
$\rho_{discharge}$	kg/m3	Outlet Gas Density

15.8.4 Cross–coupling stiffness coefficients and link script file

Executive script file **run_af_.py** calculates cross-coupling stiffness coefficients and creates a link described with a stiffness matrix with variable coefficients. Dynamics R4 starts the file by the **Run script** command. The user prepares the file input in advance. File **af_code.py** contains calculation functions and must be located in the same folder.

Running the input data file (run_af_.py)

The data can be input in different measuring systems. For English system flag is assigned as (**is_English_units=True),** else in SI unit as **(is_English_units=False).**

	Speed	**Stage Power**	**Discharge Diameter**	**Gas Passage Width**	**Density Ratio**	**Gas Molecular Weight**	**Beta**	**Kxy**
English based	rpm	hp	in	in	-	**a.e.m.**	-	**lbf/in**
SI based	rpm	kW	m	m	**0**	**a.e.m.**	+	**N/m**

For each rotor wheel the calculation type is defined by **calcType =[1,2,2,...]** as:
1 – Wachel equation,
2 – Alford equation,
3 – Modified Alford equation.

Different stages may be calculated by different equations.

For each wheel the parameters are input in a 1-D array in an order shown in the table below:

Speed	Stage Power	Discharge Diameter	Gas Passage Width	Density Ratio	Gas Molecular Weight	Beta
1000	10	24	3	0	0	2
2000	50	24	3	0	0	2
3000	250	24	3	0	0	2

The absent parameters are replaced with zeros.

	Speed	Stage Power	Discharge Diameter	Gas Passage Width	Density Ratio	Gas Molecular	Beta
Wachel equation	+	+	+	+	+	+	**0**
Alford equation	+	+	+	+	**0**	**0**	+
Modified Alford equation	+	+	+	+	+	**0**	+

For example, a 1-D array for the script file input in English units for two stages and three regimes is the following:

Stage1=[
1000,10, 24,3, 0,0,2,
2000,50, 24,3,0 ,0,2,
3000,250,24,3,0,0, 2
]
Stage2=[
1000,10, 24,3,0, 0,3,
2000,50, 24,3,0, 0,3,
3000,250,24,3,0, 0,3
]

For example, calculated stages may be listed as **stage_list=[Stage1, Stage2]**

There are a few calculation options:
Option 1. Calculation for a compressor as a total resulting with a total K_{xy}**.**. Then the script variables are:

isVariant1=True
isVariant2=False

Option 2. Calculation for a single stage resulting with its K_{xy}:

isVariant1=False
isVariant2=True
stage_number=0

Value of the **stage_number =k** variable defines its number in the **stage_list [0…n]** list.

The script operation results in a general configuration link [**Aerodynamic_Force_genLink]** with variables **Kxy_N/m Kyx_N/m** connected to the related stiffness coefficients in the link stiffness matrix.

So the user must connect the link in the load application station.

16 TRANSIENT RESPONSE

16.1 General information

The transient response algorithms provide analysis of linear and non-linear systems in steady state or transient approaches.

The algorithm is based on direct adoptive integration of the motion equations. The step of integration is selected automatically in accordance with the given analysis accuracy and current gradients of modelled process.

The complete set of transient analysis algorithms is formed from the algorithm database by a left button double click on the **[Transient analysis]** line. Together with the main algorithm, the project receives the results post-processing algorithms, **Figure 16.1**.

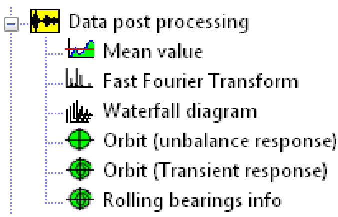


Figure 16.1

The post-processing algorithms are:

- Calculation of mean values **[Mean value]**
- **[Fast Fourier Transform]** algorithms
- Waterfall diagrams of vibration spectrums **[Waterfall diagram]**
- Motion orbits **[Orbit (Transient response)]**

> **NOTE:** Any algorithm may be removed from the project algorithm menu by **[Delete]** command

The algorithm set window (**Figure 16.2)** has a window for the setup controls. The window is used for the both linear and non-linear systems.

Des			Designation
expert	yes		Show expert options
start_action	none		Start action
process_action	calculate		Process action
process_file_save	none		Process save results
step method	bdf		Linear multistep method
iter method	newton		Iteration method
rel tolerance	1e-06		Relative tolerance
displ_abs_tolerance	1e-06	mm	Displacements absolute tolerance (noise level)
speed_abs_tolerance	1e-06	mm/s	Speeds absolute tolerance (noise level)
t_pr_1	0		Start time parameter
t_pr_2	0.1		Stop time parameter
intgr_time	0.1	s	Integration time
init_time	1.5	s	Initial integration time
init_toler	same		Tolerances for init_time
init_output_step	0.0001	s	Initial output step
init_damping	1000	N*s/m	Initial damping
output_step	0.0001	s	Output step
modal_det	500		Modal stiffness and damping refinement
With own weigth	yes		With own weigth
g_x	0	mm/s^2	Gravity acceleration in x direction
g_y	-9810	mm/s^2	Gravity acceleration in y direction
g_z	0	mm/s^2	Gravity acceleration in z direction
With maneuver	yes		With maneuver loads due to gyroscopic moments
v_x	-0.105	rad/s	Maneuver angular velocity around X axis
v_y	0	rad/s	Maneuver angular velocity around Y axis

Value of damping in additional time of integration. The damping is changed from setting value of damping to system state

Figure 16.2

Table 16.1 Algorithm parameters

Parameters	Designation	Description
Des		Text format variable, by default an empty field. The user may input here a new algorithm name that will be followed by matching amendment of the algorithm menu.
start_action	none	The variable determines the way of data processing: **[none]** – Data are not saved and not loaded **[load]** – Load file **[save]**– Save available results
process_action	calculate	Code processing, calculation or no calculation
process_file_save	none	Data saving during the integration procedure: - Do not save - Save with a determined time step
step_method	bdf	Method for the step of integration determining
iter method	newton	Determines the iteration method for the integration
real_tolerance	1e-006	Relative parameter error for the integration (1e-3…. 1e-15)
display abs tolerance	1e-006	Absolute parameter error for the integration
Speed abs tolerance	1e-006	
t_pr_1	0	The beginning value of the investigated time interval parameter
t_pr_2	10	The end value of the investigated time interval parameter
intgr_time	10	Integration time appointed for the investigated time range
init_time	1	This is the initial integration time. This variable is input to avoid initial distortions of speed, mass, etc. The method to avoid the distortions is damping
init_damping	0	Damping value. This is input only for the initial integration time. This variable changes linearly from the input value to zero
output_step	0.0001	Step size for the integration results output, 0.0001 by default. All intermediate values are omitted. Step reduction may slow the calculations. (For a satisfying rate the computer memory should be 2048…4096 MB). The step size is used as the initial step size, and the method varies it down as needed.

With own weight	Yes/No	Determines influence of the weight consideration.
a_x	0	X direction acceleration.
a_y	-9.81	Y direction acceleration.
a_z	0	Z direction acceleration.
With evolution	Yes/No	Determines whether the gyroscopic effects occurring due to the aircraft's maneuver will be taken into account.
v_x	-0.105	Maneuver angular velocity around X axis
v_y	0	Maneuver angular velocity around Y axis

Time signal calculation in **[Transient response]** for complicated models may take a lot of time. Calculation results of adjusted model may be saved in file on a disk for the following analysis of the obtained data without their recalculation, **Figure 16.3.**

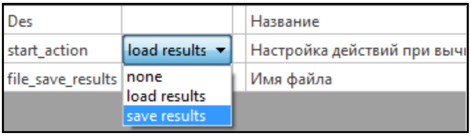


Figure 16.3

Saving. To save the results, choose the point **[save results]** in the algorithms characteristics in the list **[start_action].** Highlight any other area. The parameter setting directory to the file saving is left in characteristics. Saving dialog is called through context menu on the **[file_save_results]** area. Saving takes place when the **[Start]** is pressed and after closing of saving directory dialog.

Data loading. The item **[load results]** should be chosen in the **[start_action]** list. Directory to the file with results is defined similarly to saving. Results loading takes place after pressing of **[Start]** button. Data may also be loaded in an empty model. The dialog of file opening is called in File/Open menu. Extension filter is specified on "All files". Results file is opened with **trs** extension. File **trs** may also be opened by double click. At such loading the empty model including the file specified to load file by the **[Transient response]** algorithm is created.

NOTE: Calculation stop is needed in order to close the window during **[Transient response]** calculation

An example of transient analysis results is shown in **Figure 16.4**

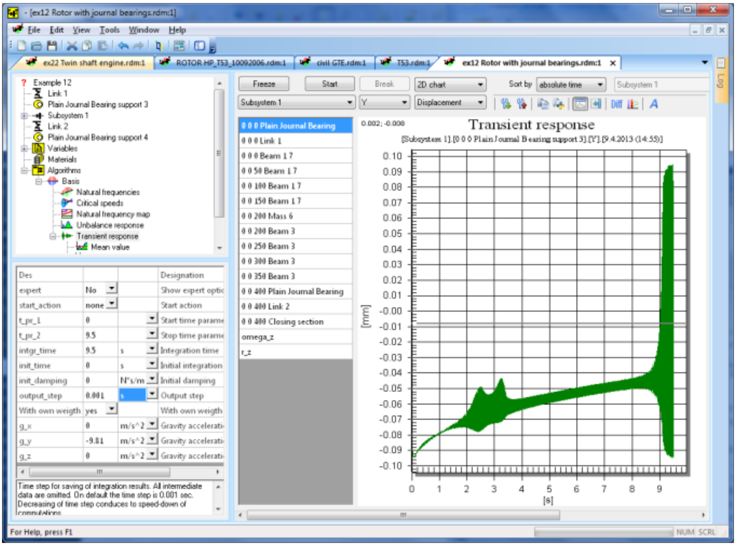


Figure 16.4

After time-signal obtainment the additional information may be obtained using postprocessing algorithms.

A user may observe results of unsteady calculation while calculating. For this he should start calculation, stop it and start again. Choosing the necessary section to output, he sees the result obtained by the moment of this section highlighting. Besides sorting of results in the current section at time and rotating speed, a user may output the dependence of the parameter change in one section relative to another, **Figure 16.5**

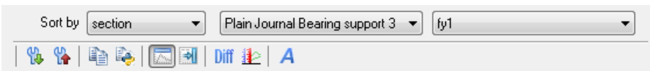


Figure 16.5

Superposition function is used for simultaneous data output at several sections in their own axes of coordinates. For example, superposition of the displacement and rotating speed diagram.

A - the button outputs all displacements and rotations for the present cursor position on graph in the log. Output format: u_x u_y u_z ur_x ur_y ur_z.

16.2 Reactions determination

Reactions in linear links may be obtained in the **[Unbalance response]** and **[Nonlinear analysis]** algorithms, using the following algorithm, **Figure 16.6:**

1. In the opening list choose a subsystem where the link of interest is attached
2. In the opening list of axes projections choose the axis where the reaction will be obtained, for example, Y
3. Choose type of the output data **[Force]** in the opening list
4. In the subsystems sections list choose two sections (with pressed **[Ctrl]** button) corresponding to the point of the link attachment and the section following it with the same coordinate. It may be the first section of the beam element, point mass or the other link
5. Tick the item "Calculate difference"

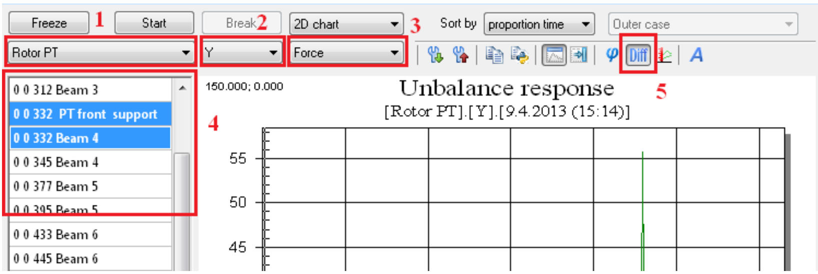


Figure 16.6

NOTE: This methodology is inapplicable to obtainment of reactions in linear links

NOTE: This methodology is applicable only for links attached to the sections where distance and reciprocal rotations are equal to zero. Otherwise, results may be false

For the elements of the [Link] type, reaction may be obtained at forces output in the Internal link section additionally.

All the links from the **[Nonlinear elements]** group are taken into account only while calculating the **[Nonlinear analysis]** algorithm. Reaction in nonlinear link may be obtained from additional output parameters of the link of interest in the list. For this you should, **Figure 16.7** :

1. In the opening list choose nonlinear link of interest
2. Choose reaction in the fixing point along the axis of interest. For example, non-linear link is attached to the shaft by the first point and to the case by the second one. To obtain reaction in the Y direction on case it is necessary to choose the “fy2” parameter from the list

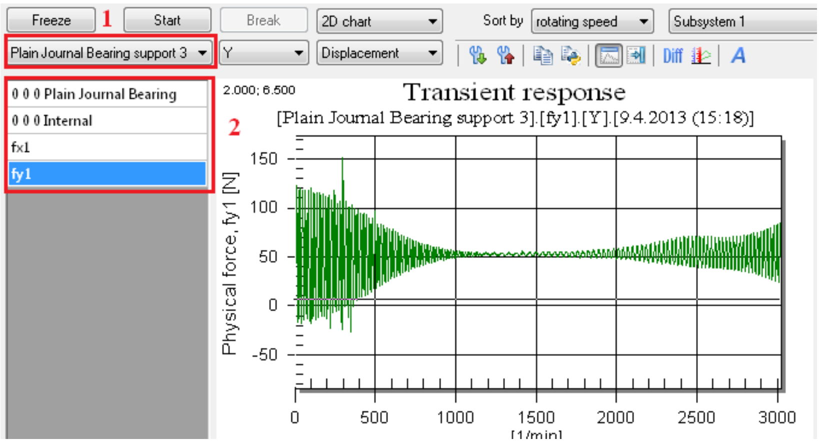


Figure 16.7

NOTE: Obtaining reactions you should remember about influence of basis calculation on values obtained in other algorithms

16.3 Progress messages during integration

During the integration process the messages / warning window displays current information on the integration progress and the elapsed time, **Figure 16.8**

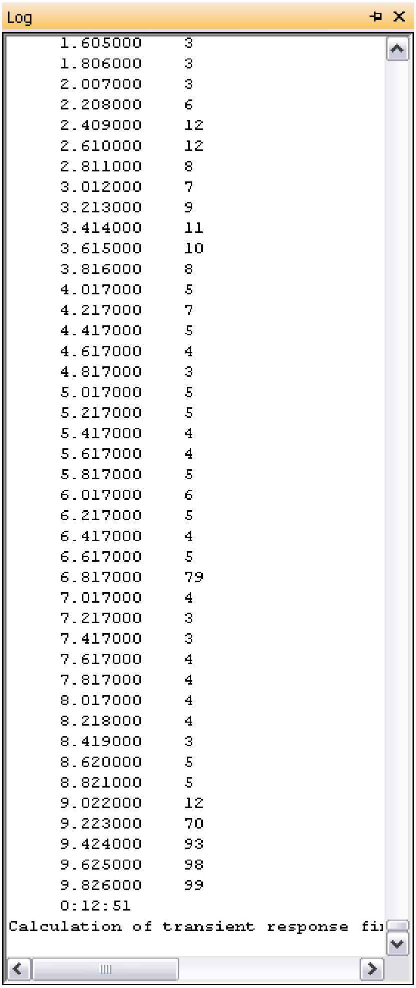


Figure 16.8

In this screen the first column is the time elapsed, the second column shows the integration time duration for the corresponding period**.**

The integration process data may also be obtained through the output parameters combo box(**Figure 16.9).**

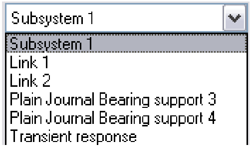


Figure 16.9

Select in the box **[Transient response]**.

Select a command in the opened window, **Figure 16.10**

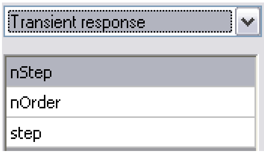


Figure 16.10

The following parameters may be displayed during and/or after the calculation:

- **[nStep]** – the number of the internal integration steps, **Figure 16.11**

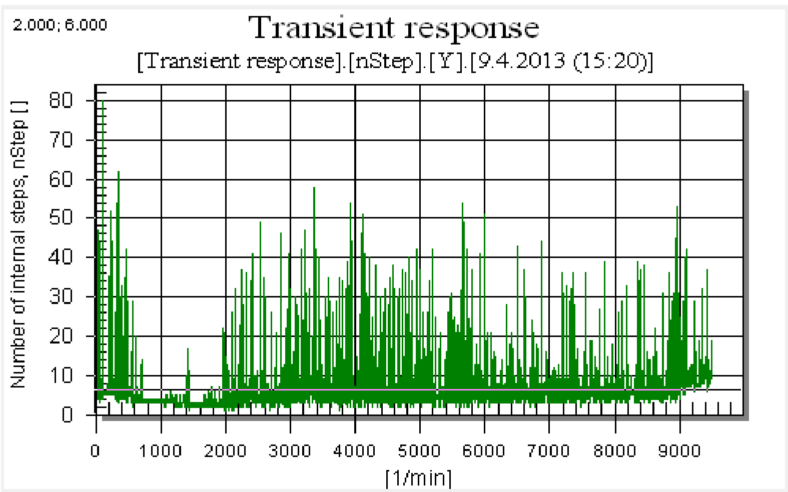


Figure 16.11

- **[nOrder]** - order of the approximating polynomial function, **Figure 16.12**

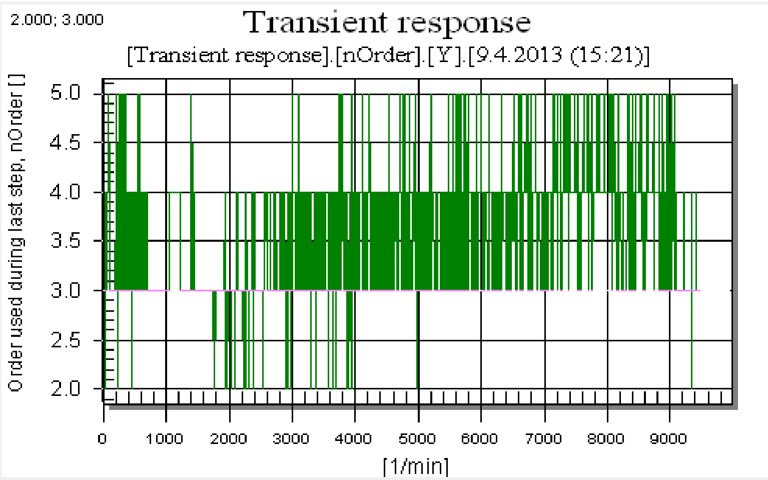


Figure 16.12

- **[step]** - Length of the step used at the end of the time interval, **Figure 16.13**

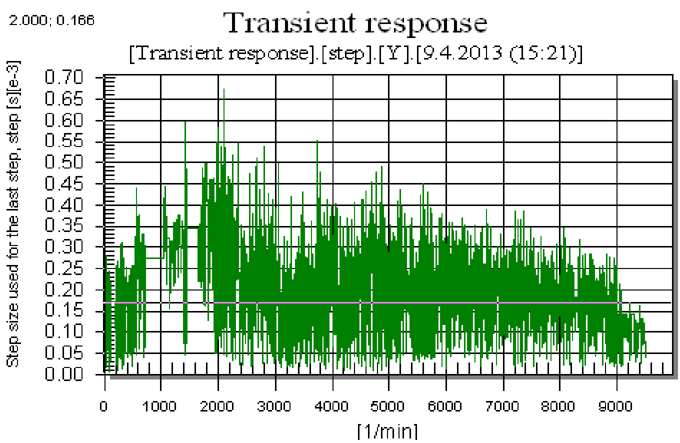


Figure 16.13

16.4 Recommendations to obtain integration tolerances

The user may control the integration process defining not only time integrating parameters (integration time and step), but also specifying tolerances on the parameters that influence convergence process to solution. If the values of these parameters are chosen correctly, a user may increase significantly the accuracy of the obtained results and reduce time to analize results.

Control of integration process is hold using the following parameters: **[rel_tolerance], [displ_abs_tolerance]** and **[speed_abs_tolerance], Figure 16.14.**

rel tolerance	1e-006		Относительная погрешность
displ_abs_tolerance	1e-006	mm	Displacements absolute tolerance (noise level)
speed_abs_tolerance	1e-006	mm/s	Speeds absolute tolerance (noise level)

Figure 16.14

[rel_tolerance] – the parameter estimates relative error of the results obtained while integrating. The value of the parameter may be in the range from **1.0e-3** to **1.0e-15**. For example, **rel_tolerance=1.0e-4** means that relative error accounts for **0.01%.** It is not recommended to use the value of the parameter less than **1.0e-3**. However, this parameter should not be too small, in order to avoid zone of the rounding mistakes of the machine arithmetic.

[displ_abs_tolerance]/[speed_abs_tolerance] - the parameters estimate absolute error of the parameters (displacements, speeds) obtained while integrating. They may be so small that it is pointless to use relative error. For example, if the **y(i)** parameter starts not from zero, but it goes to zero in the course of time, so relative error control stops being effective, because the value of **y(i)** parameter is in the noise zone. The value of these parameters is also in the range from **1.0e-3** to **1.0e-15**.

Errors are controlled at every time step so the final (global) mistakes are the result of the mistakes at every time step.
It may be made a rule to define the value of errors decreasing their values from actually desired range of the final result error by **0.01** factor. For example, if you want the accuracy of integration parameters to be **0.01%,** so the **[rel_tolerance]** parameter value equal to **1.0e-6** should be chosen.

In any case it is desirable for a user to carry out a series of calculation experiments for the exact model in order to find the best values of tolerances on the parameters obtained during integration, to see difference in the results at their variation.

16.5 Forced oscillations mode

For the [Transient response] algorithm it is possible to output vibration displacements, vibration speeds, vibration accelerations and loads along all directions at the length of the rotor system components

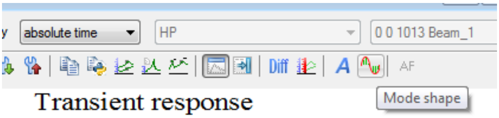


Figure 16.15

Mode shapes under current forces actions are output for the current time moment. A user chooses a data range (zooms the plot area of interest) and the moment time of interest and (clicks at the plot point) in Amplitude Time Characteritics. After pressing the button of the mode shape output (**Figure 16.15**), mode shape plots are built for all the subsystems in the chosen assembly (**Figure 16.16**). When changing the direction (X, Y, Z), of the type of output parameter or current assembly, mode shapes are rebuilt.

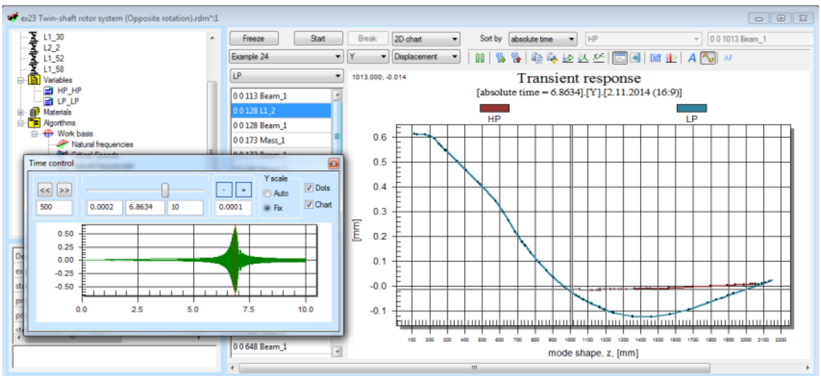


Figure 16.16

The dialog of mode shapes control gives an opportunity to manage change of current position of the mode shapes output.

Figure 16.17 shows control elements.

1. Buttons with arrows launch automatical change in time moment. The area below gives an opportunity to edit an update rate in msec. Time step is edited in the field (3)
2. Manual editing of the time signal boundaries and current time graduation
3. Change in the time graduation by the value given in the field in seconds.
4. Control of ordinate axis scale. In the Auto regime the axis scale changes automatically at minimum-maximum value of mode shape. At the Fix regime the scale is fixed automatically at current values.
5. The area of display of the signal Amplitude Time Characteristics for the highlighted section before entering the regime of mode shapes display. Current time moment of the mode shapes output is shown by vertical red line.
6. Control of output of sections' points on the plots.

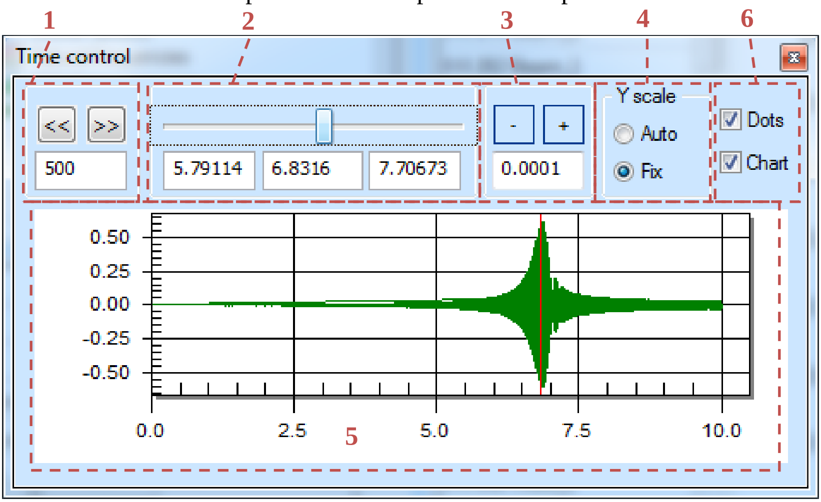


Figure 16.17

16.6 Gyroscopic moments during aircraft steady maneuvers

Maneuvers in which the angular velocity vector from evolution does not coincide in direction with the angular velocity vector of the rotating rotors are taken into consideration. During a steady maneuver of the aircraft gyroscopic moments act on the all elements of the GTE in a plane orthogonal to the motion path along with inertial loads, which in turn act in the plane of the maneuver. Gyroscopic moments can be evaluated using the equation:

$$\overline{M_g} = \bar{L} \times \bar{\Omega},$$

where $\bar{\Omega}$ - aircraft angular velocity vector, $\bar{L}$ - angular momentum of an engine rotor: $\bar{L} = J_p\bar{\omega}$, since the rotor rotates around its axis of symmetry, where J_p - moment of inertia of the rotor around the axis of rotation, $\bar{\omega}$ – angular velocity vector of the rotor. Thus, the value of the gyroscopic moment can be calculated using the equation:

$$|\overline{M_g}| = J_p|\bar{\Omega}||\bar{\omega}|\sin(\theta),$$

where θ – angle between two vectors $\bar{\Omega}$ and $\bar{\omega}$.

Obviously, in case of co-rotation of the rotors, the gyroscopic moments will have the same direction, and in a counter-rotation engine the moments will have opposite directions. This fact should be taken into account in assessment of the loads acting on the rotor supports during aircraft maneuvers.

For linearized models, one should remember that the matrices of the stiffness and damping coefficients of elements that are nonlinear in nature should be linearized with the new loads.

NOTE: gyroscopic moments are calculated for a constant speed corresponding to the t_pr1 mode. If the settings of the [Transient response] algorithm specify the t_pr1 and t_pr2 modes with different rotor speeds, then the change in rotation speed by the modes will not be taken into account when calculating gyroscopic moments. DYNAMICS R4 will display a warning message at the end of the calculation.

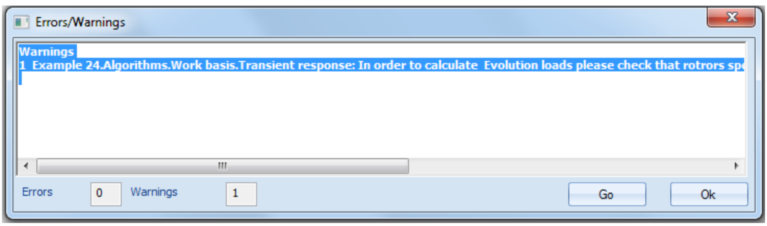


Figure 16.18

17 WAVEFORMS POSTPROCESSING

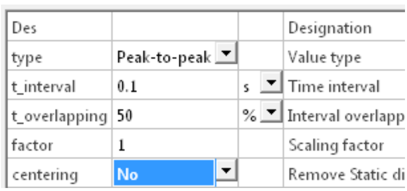
The post-processing algorithms automatically start with default parameters. The parameters may be changed. An algorithm re-starts by repeated selection of an output section.

17.1 Mean value

Two options are available here:

- [Peak-to-Peak]
- [Root mean square]

The **[Peak-to-Peak]** or the **[Root mean square]** algorithm may be selected from the fall-down menu, **Figure 17.1**.



Des			Designation
type	Peak-to-peak		Value type
t_interval	0.1	s	Time interval
t_overlapping	50	%	Interval overlapp
factor	1		Scaling factor
centering	No		Remove Static di

Figure 17.1

Table 17.1 Algorithm parameters

Parameters	Designation	Description
Value type	type	Algorithm choice [Peak-to-Peak] [Root mean square]
Time interval	t_interval	Time interval for the mean value calculation
Interval overlapping	t_overlapping	Length of the interval overlapping with the neighboring intervals
Data output scaling factor	factor	Factor of scaling of calculation data
For RMS	centering	For the mean value algorithm

algorithm static displacement will be neglected		statical component of the signal is excluded. It may appear as a result of statical force action (for example, weight force). To exclude it correctly, time range should be long enough and include several periods of carrier frequency. In case of variable statical component (for example effect of floating-up the rotor supported by journal bearings) – guite short one. Default value is “No”.

For Peak-to-peak algorithm parameter [factor] is available. It allows results scaling.

For RMS algorithm static displacement will be neglected. Results depend on the selected time interval. Time interval should include several periods of carrier frequency (should be long enough). And for models with variable static displacements (for example, with Plain Journal Bearings) should be short enough. The default choice is "No".

The **[t_interval]** and **[t_overlapping]** intervals are output equally for the both algorithms.

Plots may be displayed versus time, or rotor speed. Examples of mean value **[Peak-to-Peak]** plots are given in **Figure 17.2.**

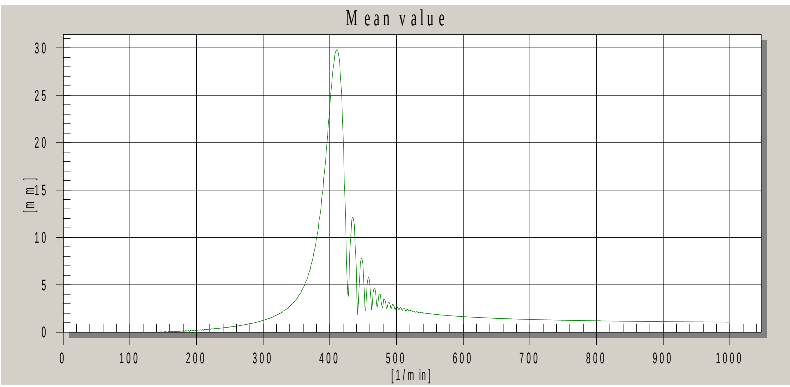


Figure 17.2

17.2 Fast Fourier Transform

The FFT parameters window is shown in **Figure 17.3.**

Des	FFT		Designation
window	Hanning		Window function
resampling step	1		Resampling load step
t_st_real	7	s	Real start time
t_fft_interval	0.33	s	Time interval for FFT
f1	0	Hz	Start frequency
f2	700	Hz	Stop frequency

Figure 17.3

Table 17.2 Algorithm parameters

Parameters	**Designation**	**Description**
Real start time	**t_st_real**	Beginning time for the FFT realization interval
Window function	**window**	Type of window function that is foreground on the chosen range
Resampling load step	**resampling step**	Step of time signal loading to decrease sampling frequency
Time interval	**t_interval**	Length of the FFT realization interval
Start frequency	**f1**	Left border of the frequency range
Stop frequency	**f2**	Right border of the frequency range

A user himself may define the time start of realization **[t_st_real],** for which FFT will be obtained and length of time realization **[t_interval].** The realization length should not exceed the length of the remained part of the time signal.
Frequency range from **[f1]** to **[f2]** to output spectra may be also defined.
An example of the frequency spectrum is given in **Figure 17.4**.
A user can mark peaks on the plot. Place vertical lines of multiple harmonics in relation to the current cursor position on the plot (click on the point on the plot and click on the corresponding function).

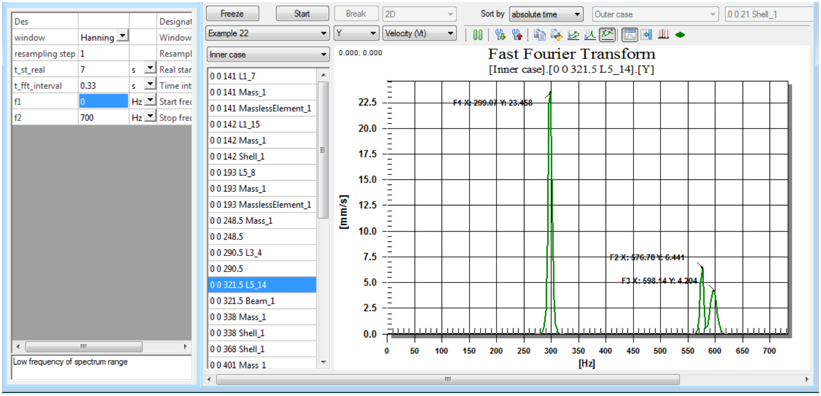


Figure 17.4

Before FFT signal may be processed additionally by applying of the window function ().

You should remember that FFT are calculated for discrete signals with the number of points that are multiple to the 2 degree (for example, ..., 4096, 8192, ...). Maximum value in peaks rate in the Fourier spectrum is equal to half of sampling rate.

Information about the number of points in the asked buffer length, sampling rate of initial signal and data on the number of points and Amplitude-Time Characteristics length of the FFT used in calculation is printed in the log panel.

This information is useful to optimize the parameters of the algorithm [Waterfall diagram]

```
Requested t_fft_interval= 0.50[sec]; Number of points = 50001 Initial Sample rate=1.00000e+005 [Hz]
Actual t_fft_interval = 0.328[sec]; Number of points= 32768; Sample rate=1.00000e+005 [Hz]
```

When increasing sampling rate, accuracy of obtaining high frequencies in the spectrum increases, accuracy of obtaining low frequencies decreases. If a user is interested in low frequencies, sampling rate may be decreased using the parameter [resampling step]. For resampling step=2 sampling rate becomes twice lower.

User can output multiplications for selected harmonic. For using this feature user should select the point of interest by clicking on the chart and press appropriate button . Vertical lines for 0.5x,1x...5x will be presented.

17.3 Waterfall diagram

The waterfall diagram parameters window is shown in **Figure 17.5**.

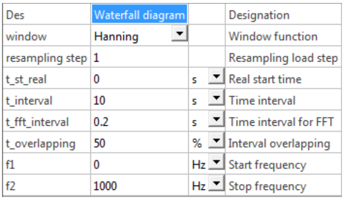


Figure 17.5

Algorithm is intended for calculations of spectrum in accordance with the given parameters of integration. The spectrums can be sorted against proportion time, absolute time and rotating speed.

Table 17.3 Algorithm parameters

Parameters	Designation	Description
Window function	window	Type of window function that is overlaid on the chosen range of time signal of window function
Resampling load step	**resampling step**	Step of loading time signal to decrease sampling rate
	t_st_real	Time of Amplitude-Time characteristics start to build waterfall diagram
Time interval	**t_interval**	Variable defining length of realization to calculate set of spectra for diagram
Time interval for FFT	**t_fft_interval**	Variable defining length of realization for spectrum calculation
Interval overlapping	t_overlapping	Variable defining realization intervals overlapping for mean spectrum calculation
Start frequency	**f1**	Beginning of the frequency range
Stop frequency	**f2**	End of the frequency range

All the parameters above are user's input.

The diagram 3D view may be rotated with standard Windows vertical and horizontal scroll bars movements.
The other example of a cascade diagram is shown in **Figure 17.6**.
Description of window functions and recommendations to set time interval for spectra see in the section Fast Fourier Transform Algorithm
Display Control:

- Mouse wheel - zoom in / out
- Click + move mouse - rotate the chart
- Press + left shift + move the mouse - move the chart
- Highlighting the spectrum and mode values with a single left-click (LMB).
- Arrows from the keyboard can be used to change the selected spectrum
- When double-clicking with LMB, flags are added to the picks.

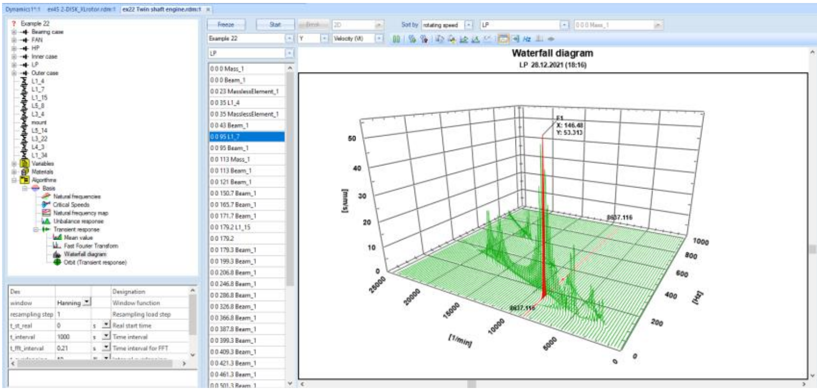


Figure 17.6

17.4 Orbits (Transient response)

The motion orbits parameters window is shown in **Figure 17.7**.

Des			Designation
t_st_real	0	s	Real start time
t_interval	10	s	Time interval

Figure 17.7

It is possible to output orbits of any section and in any time interval. The interval is defined by its beginning **[t_st_real]** and length **[t_interval].**

Table 17.4 Algorithm parameters

Parameters	Designation	Description
Real start time	**t_st_real**	Beginning of the orbit output interval
Time inteval	**t_interval**	Length of the orbit output interval

The orbits are displayed in 2D and 3D forms. Examples of 2D and 3D orbit images are given in **Figure 17.8**.

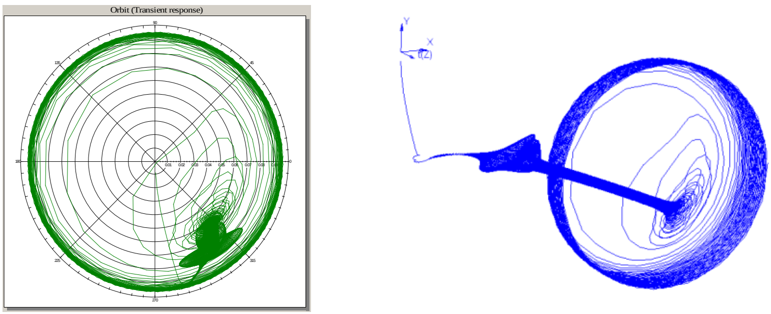


Figure 17.8

17.5 Rolling bearings info

There are two options of work with Dynamics R4 interface in order to carry out analysis. Data may be obtained within the bounds of time signal analysis from the results of the **[Nonlinear analysis]** algorithm and by use of the **[Rolling bearings info]** algorithms of signals postprocessing, **Figure 17.9**.

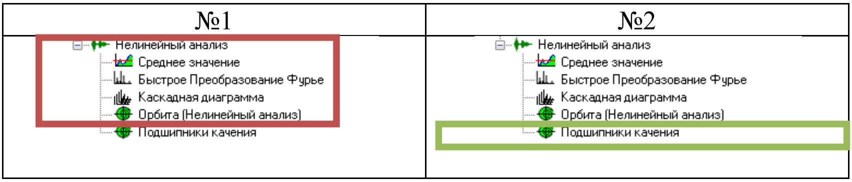


Figure 17.9

If the more detailed analysis of the bearing for the exact regime is needed, the **[Rolling bearing info]** algorithm should be used.

For algorithm's work the calculated results of the **[Nonlinear analysis]** algorithm is required. Data for the **[Rolling bearing info]** algorithm may be loaded from the previously saved calculation for the current model. **Figure 17.10** presents the output window.

The algorithm interface is presented by several main blocks. In the window upper part, the list of parameters/sections of the investigated rolling bearing and amplitude-time characteristics graph of the chosen parameter are placed. This part of the window is used for control of choice of operating regimes.

In the lower window part, the elements of control of results at current rolling bearing are presented. There is a list of regimes for analysis of the bearing conditions with the elements of control of this list, output window of a text log at the current regime and a button of control of information export into clipboard.

Chosen by user regimes for bearing conditions analysis are highlighted in red vertical lines and figures.

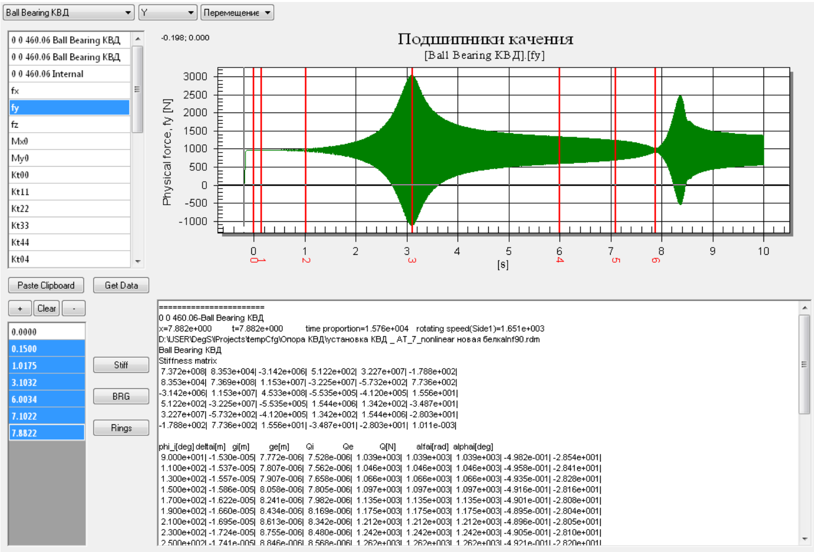


Figure 17.10

Let us consider sequence of a user's actions at work with this algorithm. First, with the help of a cursor the point of the investigated parameter is chosen in the amplitude-time characteristics of the displayed parameter. The [Get Data] Get Data button is pushed for the immediate bearing data output. Log window shows data concerning parameters of bearing conditions for the chosen regime. There are: the whole stiffness matrix, a table of rolling elements parameters, a table of linear and angular rings displacements, reactions in bearings. To make a list of regimes, the button + may be pressed instead of [Get Data] button. The temporary value of the chosen point will appear among the regimes list. In the sequel it is enough to highlight the regime of interest in the list for a bearing data output. The [Stiffness], [BRG], [Rings] buttons control data export.

Stiff - copying of stiffness matrix into clipboard;

BRG - copying of a table containing rolling elements parameters into clipboard;

Rings - copying of a table including displacements and reactions in bearings rings into clipboard.

Data are copied into clipboard in the format suitable for insert into MS Excel.

If several sections are highlighted in the regimes list, then the information about the last regime is displayed. Using the buttons of copy into clipboard the summary tables embracing all regimes are written.

Table 17.5

	Fx[N]	Fy[N]	Fz[N]	Mx[N/m]	My[N/m]	Mz[N/m]
ux[m]	7.36E+08	-6.94E+02	2.58E+04	1.53E+01	3.22E+07	-3.74E+02
uy[m]	-6.94E+02	7.35E+08	1.14E+07	-3.22E+07	-1.50E+01	-2.00E+00
uz[m]	2.58E+04	1.14E+07	4.52E+08	-4.82E+05	2.33E+03	0.00E+00
urx[rad]	1.53E+01	-3.22E+07	-4.82E+05	1.54E+06	0.00E+00	0.00E+00
ury[rad]	3.22E+07	-1.50E+01	2.33E+03	0.00E+00	1.54E+06	-3.78E+01
urz[rad]	-3.74E+02	-2.00E+00	0.00E+00	0.00E+00	-3.78E+01	0.00E+00

18 OUTPUT OF RESULTS - PROTOCOL

The user may output the model data and the calculation results with by means of the function **[Protocol]** available in the Instruments panel or started with the button . The button opens a tree window, **Figure 18.1**.

The output protocol format, **HTML, MSWord** or **ASCII** is user determined. The HTML is recommended and is active at default. User selects in the protocol tree the output data, shows the data saving catalog and inputs the protocol name with the **[Generate output]** button.
The protocol format is controlled in a tree form.
Selection or release of a nod element leads to changes in all levels. For example, selection of **[text input]** causes output of all model elements parameters. The protocol tree contains all element types. If a type is not used in the model, it is not displayed in spite of the selection.

> NOTE: The output data amount may be remarkably reduced by exclusion of calculated parameters of beams and shells like flexibility matrixes or mass and inertia parameters

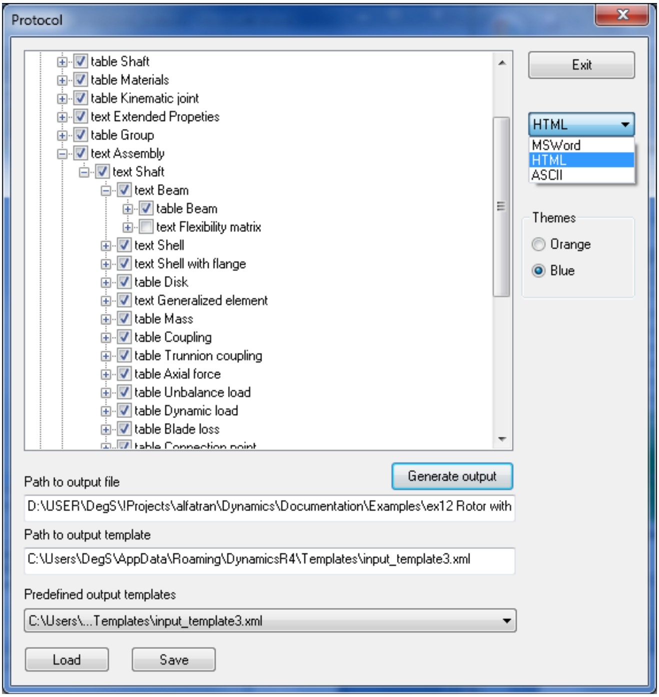


Figure 18.1

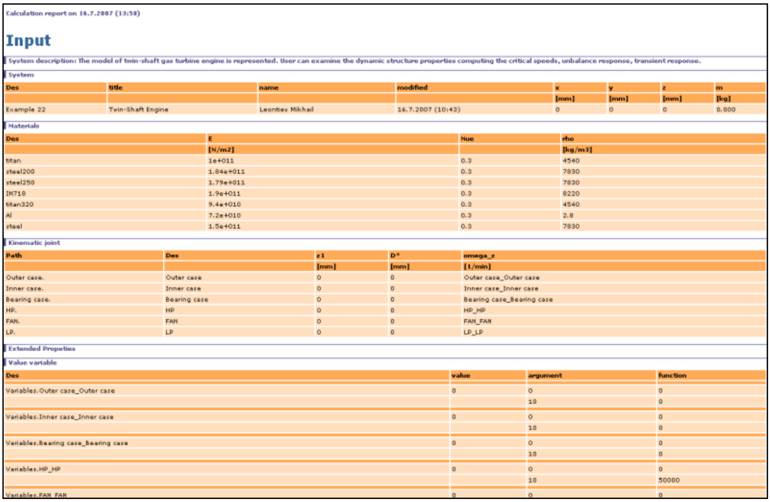
NOTE: The input data assembling per the subsystem elements is located in the: **[text Input]->[text Assembly]->[text Subsystem]** tree

NOTE: It is very useful to output only the required data. Selection of all elements increases the protocol generation time; the text information may increase up to a few hundred pages. The output protocol sample may be determined and saved. The save path is input in the bottom dialogue window.

A user can copy saved protocol template into "[User Application Folder]\DynamicsR4\Templates" folder. Then it will appear in the dropdown list of the predefined templates.

At default the protocol file is created in the folder of the model file and has the same name. The name extension suits the output format.

Different protocol formats are shown in **Figure 18.2, Figure 18.3** and **Figure 18.4.**



Calculation report on 16.7.2007 (13:58)

Input

System description: The model of twin-shaft gas turbine engine is represented. User can examine the dynamic structure properties computing the critical speeds, unbalance response, transient response.

System

Des	title	name	modified	x [mm]	y [mm]	z [mm]	m [kg]
Example 22	Twin-Shaft Engine	Leontiev Mikhail	16.7.2007 (10:43)	0	0	0	0.000

Materials

Des	E [N/m2]	Nue	rho [kg/m3]
titan	1e+011	0.3	4540
steel200	1.84e+011	0.3	7830
steel250	1.79e+011	0.3	7830
IN718	1.9e+011	0.3	8220
titan320	9.4e+010	0.3	4540
Al	7.2e+010	0.3	2.8
steel	1.5e+011	0.3	7830

Kinematic joint

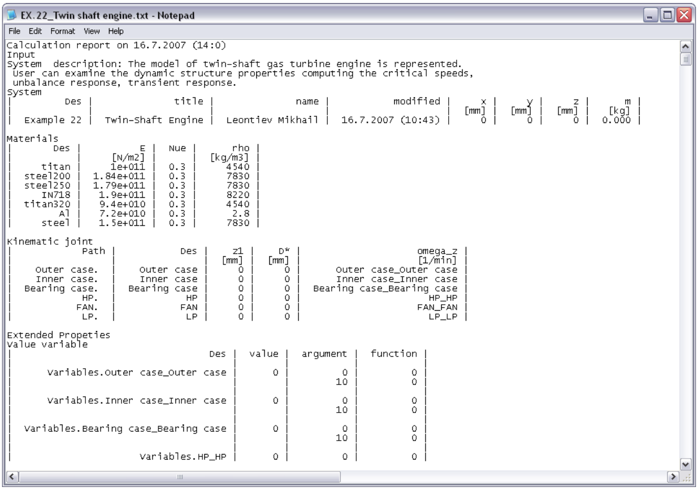
Path	Des	z1 [mm]	D* [mm]	omega_z [1/min]
Outer case.	Outer case	0	0	Outer case_Outer case
Inner case.	Inner case	0	0	Inner case_Inner case
Bearing case.	Bearing case	0	0	Bearing case_Bearing case
HP.	HP	0	0	HP_HP
FAN.	FAN	0	0	FAN_FAN
LP.	LP	0	0	LP_LP

Extended Propeties

Value variable

Des	value	argument	function
Variables.Outer case_Outer case	0	0	0
		10	0
Variables.Inner case_Inner case	0	0	0
		10	0
Variables.Bearing case_Bearing case	0	0	0
		10	0
Variables.HP_HP	0	0	0
		10	50000
Variables.FAN_FAN	0	0	0

Figure 18.2



EX.22_Twin shaft engine.txt - Notepad

File Edit Format View Help

```
Calculation report on 16.7.2007 (14:0)
Input
System  description: The model of twin-shaft gas turbine engine is represented.
 User can examine the dynamic structure properties computing the critical speeds,
 unbalance response, transient response.
System
|        Des |             title |             name |            modified |    x |    y |    z |     m |
|            |                   |                  |                     | [mm] | [mm] | [mm] |  [kg] |
| Example 22 | Twin-Shaft Engine | Leontiev Mikhail |   16.7.2007 (10:43) |    0 |    0 |    0 | 0.000 |

Materials
|       Des |         E |  Nue |     rho |
|           |    [N/m2] |      | [kg/m3] |
|     titan |    1e+011 |  0.3 |    4540 |
|  steel200 | 1.84e+011 |  0.3 |    7830 |
|  steel250 | 1.79e+011 |  0.3 |    7830 |
|     IN718 |  1.9e+011 |  0.3 |    8220 |
|  titan320 |  9.4e+010 |  0.3 |    4540 |
|        Al |  7.2e+010 |  0.3 |     2.8 |
|     steel |  1.5e+011 |  0.3 |    7830 |

Kinematic joint
|           Path |            Des |   z1 |   D* |                   omega_z |
|                |                | [mm] | [mm] |                   [1/min] |
|    Outer case. |     Outer case |    0 |    0 |     Outer case_Outer case |
|    Inner case. |     Inner case |    0 |    0 |     Inner case_Inner case |
|  Bearing case. |   Bearing case |    0 |    0 | Bearing case_Bearing case |
|            HP. |             HP |    0 |    0 |                     HP_HP |
|           FAN. |            FAN |    0 |    0 |                   FAN_FAN |
|            LP. |             LP |    0 |    0 |                     LP_LP |

Extended Propeties
Value variable
|                                  Des |  value |  argument |  function |
|                                      |        |           |           |
|      Variables.Outer case_Outer case |      0 |         0 |         0 |
|                                      |        |        10 |         0 |
|                                      |        |           |           |
|      Variables.Inner case_Inner case |      0 |         0 |         0 |
|                                      |        |        10 |         0 |
|                                      |        |           |           |
|  Variables.Bearing case_Bearing case |      0 |         0 |         0 |
|                                      |        |        10 |         0 |
|                                      |        |           |           |
|                      Variables.HP_HP |      0 |         0 |         0 |
```

Figure 18.3

Calculation report on 16.7.2007 (14:1)

Input

System description: The model of twin-shaft gas turbine engine is represented. User can examine the dynamic structure properties computing the critical speeds, unbalance response, transient response.

System

Des	title	name	modified	x	y	z	m
				[mm]	[mm]	[mm]	[kg]
Example 22	Twin-Shaft Engine	Leontiey Mikhail	16.7.2007 (10:43)	0	0	0	0.000

Materials

Des	E	Nue	rho
	[N/m2]		[kg/m3]
titan	1e+011	0.3	4540
steel200	1.84e+011	0.3	7830
steel250	1.79e+011	0.3	7830
IN718	1.9e+011	0.3	8220
titan320	9.4e+010	0.3	4540
Al	7.2e+010	0.3	2.8
steel	1.5e+011	0.3	7830

Kinematic joint

Path	Des	z1	D*	omega_z
		[mm]	[mm]	[1/min]
Outer case.	Outer case	0	0	Outer case_Outer case
Inner case.	Inner case	0	0	Inner case_Inner case
Bearing case.	Bearing case	0	0	Bearing case_Bearing case
HP.	HP	0	0	HP_HP
FAN.	FAN	0	0	FAN_FAN
LP.	LP	0	0	LP_LP

Extended Propeties
Value variable

Des	value	argument	function
Variables.Outer case_Outer case	0	0	0
		10	0
Variables.Inner case_Inner case	0	0	0

Figure 18.4

19 MATERIAL DATABASE

The Material database formed by the users is presented in the main Elements library, **Figure 19.1.**

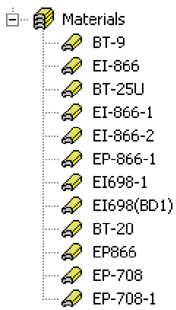
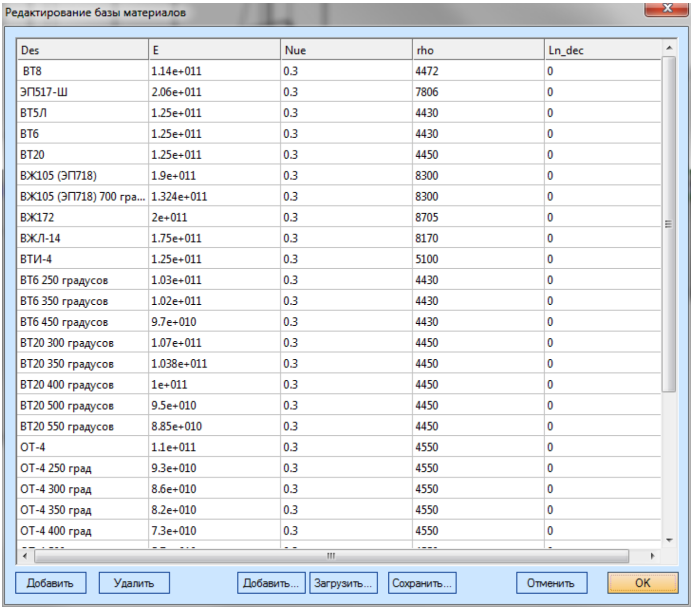


Figure 19.1

The material properties are input by editing in the **Figure 19.2** window under commands **[Tools]**, **[Edit Materials DB]**.



Des	E	Nue	rho	Ln_dec
ВТ8	1.14e+011	0.3	4472	0
ЭП517-Ш	2.06e+011	0.3	7806	0
ВТ5Л	1.25e+011	0.3	4430	0
ВТ6	1.25e+011	0.3	4430	0
ВТ20	1.25e+011	0.3	4450	0
ВЖ105 (ЭП718)	1.9e+011	0.3	8300	0
ВЖ105 (ЭП718) 700 гра...	1.324e+011	0.3	8300	0
ВЖ172	2e+011	0.3	8705	0
ВЖЛ-14	1.75e+011	0.3	8170	0
ВТИ-4	1.25e+011	0.3	5100	0
ВТ6 250 градусов	1.03e+011	0.3	4430	0
ВТ6 350 градусов	1.02e+011	0.3	4430	0
ВТ6 450 градусов	9.7e+010	0.3	4430	0
ВТ20 300 градусов	1.07e+011	0.3	4450	0
ВТ20 350 градусов	1.038e+011	0.3	4450	0
ВТ20 400 градусов	1e+011	0.3	4450	0
ВТ20 500 градусов	9.5e+010	0.3	4450	0
ВТ20 550 градусов	8.85e+010	0.3	4450	0
ОТ-4	1.1e+011	0.3	4550	0
ОТ-4 250 град	9.3e+010	0.3	4550	0
ОТ-4 300 град	8.6e+010	0.3	4550	0
ОТ-4 350 град	8.2e+010	0.3	4550	0
ОТ-4 400 град	7.3e+010	0.3	4550	0

Figure 19.2

The following data are input:

- Material name **[Designation]**
- Young modulus **[E]**
- Poisson coefficient **[rho]**
- Reduced mass **[Nue]**
- Logarithmic decrement **[Ln_dec]**

A material may be applied to project elements by double click of the left button. The material data may be used for determining system elements.

NOTE: The Material database may be transmitted to another computer by copying the file **[Program setup directory]\Data\materials.xml**

The Material database is used when the same materials are used in different projects. Changes of a material property by editing, deletion or addition of a material do not influence the existing materials data. The amendments will become valid after the code re-start.

NOTE: Relation of a material property with a temperature is reflected by addition of a few materials with the same names each marked with a corresponding temperature

Control elements:

- Add – adds template of the material position with default characteristics in database
- Delete – deletes current or highlighted line in the materials table
- [Add...] – gives opportunity to choose the file with the saved material base and add it to the materials table
- [Load...] – gives opportunity to choose the file with the saved materials base and replace the table content by files data
- [Save...] – gives opportunity to choose the file to save copy of the Material base in file on the disk.
-

Note: Materials database file is placed in the folder with a user's settings. for example, in Windows XP «C:\Documents and Settings\[User Name]\Application Data\DynamicsR4\materials.xml». The button **[save...]** does not influence file location

20 USING PYTHON IN [USER LINK] ELEMENT

It allows developing the user's own algorithms of nonlinear effects in rotating machinery to program a new link between subsystems (a journal bearing, a magnetic bearing, etc.) and their integrating into the **Dynamics R4** program system.

20.1 Script relations with Dynamics R4

The main element to connect the script with the program is [exchangeContainer] object.

Data exchange between the program integrator and the script algorithm goes through this object. When a script introduces a non-linear algorithm its input data are displacements, velocities, accelerations and rotation angles in sections connected with the UserScriptLink. These parameters are listed in Table 11.1.

A variable inside a container is referred by its name.

Example:

```
ux1=exchangeContainer.getDoubleValue('ux1')
Tx1=exchangeContainer.getDoubleValue('tx1')
```

Script output parameters are force and moment reactions listed in

Table 11.2

Example:

```
exchangeContainer.setDoubleValue('fx1',fx1)
```

20.2 Additional parameters

A non-linear element algorithm often needs additional information on sub-systems rotation speeds, current integration time, phase shift, etc. listed in **Table 20.1**

Table 20.1

Parameter	Type	Description
time_proportion	double	Current relative time, or abscissa in velocity change relation
integration_time	double	Current absolute time, or integration time
rotating_phase1, rotating_phase2	double	Current absolute time, or integration time
rotating_speed1, rotating_speed2	double	Current speeds of subsystems rotation
call_status	int	Returns information flag on integration step -1 – first algorithm call (firstCall) 0- correct step (processCall) 1- last iteration (lastCall) 2- step to be saved (processCallSave)

Example:

```
omega1=exchangeContainer.getDoubleValue('rotating_speed1')
phase1=exchangeContainer.getDoubleValue('rotating_phase1')
flag= exchangeContainer.getIntValue('call_status')
```

20.3 Exchange Container object functions

getDoubleValue('Var_name'**), getIntValue(**'Var_name'**)**

These functions get variables' values from the Container by the variables' names, the names are marked with quotes. Examples concerned are given in parameters description.

setDoubleValue('Var_name', var),
setIntValue('Var_name', var)

This function defines the parameter value 'Var_name' from the script variable, or creates a new parameter in exchangeContainer.

Example:

```
force_x1=100.5
exchangeContainer . setDoubleValue (‘fx1’, force_x1)
```

NOTE: If the sent quoted parameter name is absent in the exchangeContainer, the new parameter will be added to the container. This option may be used for exchange of data common for all iterations. See the example in IsExist function.

IsExist ('Var_name','Type')

This function checks availability of a variable in the container. The input parameters are two strings, the variable name and the variable type. The returned value is a Boolean True or False.

Example:

```
test_var=0
if False==exchangeContainer.IsExist(' test_var ','double'):
       #will execute only at first iteration
       #calculation of some data
        test_var=CalcSomething(…)
        trace(' test_var ='+str(test_var))
        exchangeContainer.setDoubleValue(' test_var ', test_var)
    else:
        test_var =exchangeContainer.getDoubleValue(' test_var ')
```

20.4 Print development info in the program log file.

trace('massage')

Prints needed info in the Program Log window.

Example:

```
trace(' test_var ='+str(test_var))
```

Here function str(test_var) transforms a number into a string (Python function)

20.5 Acceleration of the Script element function

You can speed up calculation of a non-linear algorithm programmed in the User Script Link. To do this, the script is divided into two logical parts: the first is initialization, the second is the main part, which must be called on each step of the transient response calculation. This second part of the algorithm has to be moved to the special function which will be called many times, the rest of the script will be executed at once at the very beginning of the calculation. To this end the user have to define a function with a special name "dyn_loop()", if the function is not defined the entire script will be called on each time step during the transient response calculation.

```
#code that will be called at the beginning of the calculation (initialization)

def dyn_loop():
          #  the main algorithm is here
```

In the script editing window it may look like this, **Figure 20.1.**

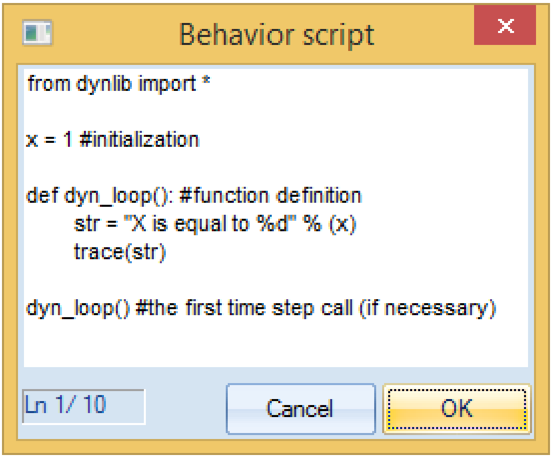


Figure 20.1

21 SIMULATION COMMAND INTERFACE

The Command interface extends the user's capabilities in models build and parametric studies. It is possible to create converters from simulation formats or from external systems into the **Dynamics R4** format. It is possible to input table form data automatically. Also it is possible to build small "calculators" to calculate elements data.

Converters. Sometimes the input data are given in a table form. This may be data from an external system, or drawing review results, or results of stiffness matrix calculations. The command simulation interface helps to input these data.

Calculators. Sometimes the input data need preliminary calculations, for example support stiffness, or flange flexibility. This calculation may be done in the script and resulting values may be used for an element insertion. For details see **Dynamics R4** reference.

Parameterization. The user may create calculations of a complicated function values or describe a complicated parameter variation.

Figure 21.1 shows a script example to describe a cantilever beam of 10 elements.

```
from dynlib import *

#//////////////////////////////////////////////////////////
document=exchangeContainer.getObject("document")
root=document.getRootObject()
bldr=ModelBuilderRDM(root)
#//////////////////////////////////////////////////////////

bldr.setCurrentMaterial(7850,2.1e+011,0.3,0)
obj_Subsystem_1=bldr.shaft("Subsystem 1",0,0,0,0,0,0,0.01)
bldr.clearCurrentMaterial()

for i in range(10):
        bldr.beam("Beam "+str(i+1),0.1,0,0.02)

obj_Rigid_link_1=bldr.rigidLink("Rigid link 1",None,None,1,1,1,1,1,1)
bldr.connectLink2(obj_Rigid_link_1,obj_Subsystem_1,0,None,0)

trace("Script finished")
```

Figure 21.1

A script produced model may be loaded by **[File/Start script].**
A standard created model may be exported into a script by the menu command **[File/Export to script].**

Note: To work with scripts we recommend to install **Python SDK (www.python.org)**

21.1 Converters

The command simulation interface facilitates input of table format data.

Together with the reference system, we supply the data converter for the data calculated in **Xlpocket** and **Xtltpad** (www.rmt-inc.com).

These codes (**Figure 20.2**) calculate stiffness and damping of different sleeve bearings, cylindrical, elliptical, or segment.

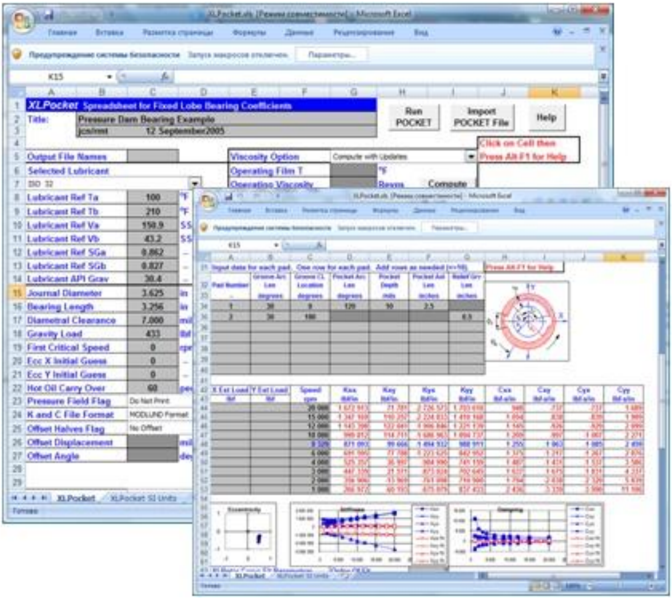


Figure 20.2

It is necessary to input the data into **Dynamics R4, Figure 20.3**

Speed rpm	Kxx lbf/in	Kxy lbf/in	Kyx lbf/in	Kyy lbf/in	Cxx lbf-s/in	Cxy lbf-s/in	Cyx lbf-s/in	Cyy lbf-s/in
20 000	1 672 913	71 781	-2 726 573	1 703 610	948	-737	-737	1 689
15 000	1 347 169	110 257	-2 224 833	1 410 168	1 054	-838	-839	1 909
12 000	1 143 398	122 041	-1 906 846	1 221 139	1 145	-926	-929	2 099
10 000	999 812	114 711	-1 686 963	1 094 737	1 209	-997	-1 007	2 271
8 329	871 093	99 666	-1 494 932	988 911	1 255	-1 063	-1 085	2 459
6 000	691 595	77 788	-1 223 625	842 952	1 375	-1 217	-1 267	2 876
4 000	525 357	36 997	-984 990	741 159	1 487	-1 431	-1 537	3 586
3 000	447 339	21 571	-873 024	702 645	1 637	-1 675	-1 831	4 337
2 000	356 906	-13 969	-761 098	710 900	1 794	-2 038	-2 320	5 839
1 000	266 972	-60 193	-675 079	837 433	2 436	-3 339	-3 990	11 106

Figure 20.3

The table shows stiffness and damping coefficients calculated for 12 speeds. A manual input requires creation of 8 variable models, creation of a link with an asymmetric stiffness matrix and connection of the variable to each of the matrix coefficients. Each variable shall contain 12 regimes. This is a time and labor consuming job.

Scripts carry out this work automatically and thus exclude possible errors. To start the scripts it is necessary to transfer the data from the table into the **Brgs_Input.py** file and create an array **Bearing1** (the name is arbitrary).

```
Bearing1=[
20000,1672913,71781,-2726573,1703610,948,-737,-737,1689,
15000,1347169,110257,-2224833,1410168,1054,-838,-839,1909,
12000,1143398,122041,-1906846,1221139,1145,-926,-929,2099,
10000,999812,114711,-1686963,1094737,1209,-997,-1007,2271,
8329,871093,99666,-1494932,988911,1255,-1063,-1085,2459,
6000,691595,77788,-1223625,842952,1375,-1217,-1267,2876,
4000,525357,36997,-984990,741159,1487,-1431,-1537,3586,
3000,447339,21571,-873024,702645,1637,-1675,-1831,4337,
2000,356906,-13969,-761098,710900,1794,-2038,-2320,5839,
1000,266972,-60193,-675079,837433,2436,-3339,-3990,11106
]
```

1. Copy the table with stiffness and damping coefficients into an empty **Excel** file, add a letter as the last column.
2. Save the file as **csv** with comma separators
3. Open the file with a text editor, **notepad**, for example. If the separators are semicolons, replace them with commas and delete all letters from the last column and the comma after the last element. Now the data are ready for the script input.
4. Insert the array into square brackets.

The reference examples archive has an import example of three bearings data. The speed variable connected to the created sub-system is added to the model. Addition of variables and two links is the result of script running. One link is described by nonsymmetrical stiffness and damping matrixes. This link is not taken into account in Basis algorithm. The second one is used in Basis.

NOTE: User should check the Basis calculation range for results correctness.

Archive contents:

BldrBearings.py – running as < File>, <Run script>
Brgs_Input.py – calculates bearings input data
Bearing.py - auxiliary module (no editing)
Interpolation.py - auxiliary module (no editing)

NOTE: Stiffness and damping values are created in SI units (N/m, N*s/m). The script needs point digits separator. Digits groups cannot be separated with blanks.

22 MATRIX CALCULATOR

When an engineer simulates complex design units, the necessity in matrix calculations may arise. For example, as a result of calculation in Finite Element Method programs for common purposes the flexibility matrixes of engine structural elements were obtained. Compliance matrix inversion may be required in order to input data about the received elastic characteristics in a model using links.

For a user's convenience the matrix calculator was added to Dynamics R4. It may be found in the menu "Tools \ Matrix calculator", **Figure 22.1**.

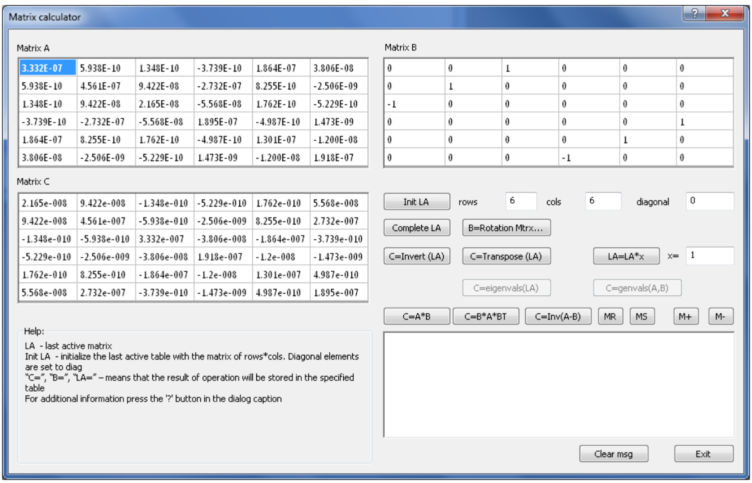


Figure 22.1

The calculator window includes three tables presenting data concerning A, B, C matrixes, buttons of matrix operations and report windows.

Matrixes may be input using clipboard. They may be copied from other tables in the program, protocol files, Excel, Word or from the text file (delimiters – commas and tabulation).

Before matrix insert the required number of rows and columns should be assigned The sign “LA” on the button implies that operation will be made at the matrix from the last active table (Last Active). The signs “C=”, “B=”, “LA=” show where the operation result is stored.

Description of operations:
“**Init LA**” – initialization of the last active matrix (LA) according to the defined number of rows and columns with the completion of the diagonal.
“**Complete LA**” – filling of the left lower matrix triangle as the right upper triangle (symmetrically).It may be used when inserting a matrix from the links into Dynamics R4.
“**B=Rotation Matrix…**” – call of the dialog of creation of the 6*6 rotation matrix from a right coordinate system into another one.
“**C=Invert(LA)**” – LA inverting with the result record in C (translation flexibility matrixes into stiffness ones and vice versa)
“**C=Transpose (LA)**” –LA transposition with the result record in C
“**LA=LA*x**” – multiplication of LA by a number
“**C=A*B**” – multiplication of A and B matrixes by the result record in C
“**C=B*A*BT**” – matrix permutation
“**C=Inv(A-B)**” – inverting of the result of substraction matrix B from A. (compensating links)
“MR, MS, M+,M-” – operation of work with memory and LA – reading from memory in LA; saving of LA in memory; summation of LA and a matrix in memory; substraction from matrix in LA memory

Obtainment of rotation matrix
When the “**B=Rotation Matrix…**” is pressed, the dialog of obtainment of rotation matrix for move from one coordinate system to another one is called. It is used for conversion of stiffness and flexibility matrixes (6*6).

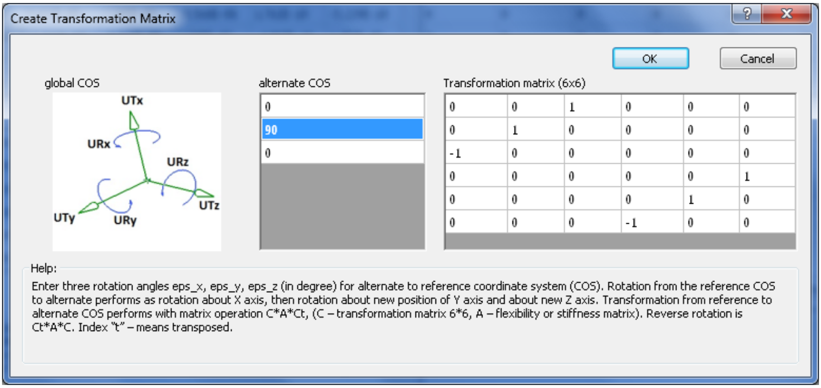


Figure 22.2

Enter three rotation angles eps_x, eps_y и eps_z for an alternate coordinate system (COS), **Figure 22.2**. The units are degrees. Global COS is rotated into alternate one by consecutive rotation about X axis (eps_x), then – about new position of Y axis (eps_y), after that – about new position of Z axis (eps_z). Transformation from global to alternate COS is implemented by the matrix operation C*A*Ct, where C is 6*6 rotation matrix, A – flexibility or stiffness matrix in global COS. Reverse rotation is Ct*A*C. Index "t" means transposed.

23 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.

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

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

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 for the element which is active in the model tree.

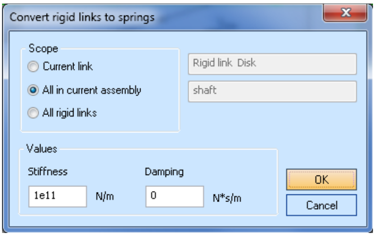


Figure 23.1

The menu [Service\Convert rigid links by elastic links] may also be used. Meanwhile, the dialog window with conversion settings appears, **Figure 23.1** .

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

NOTE: Additional information on this matter can be found in the "Simulation in Dynamics R4 (HowTo…)"

24 HOT KEYS

To speed up some operations, the program has the following hotkeys. LMB - left mouse button.

Hot keys	Context	Description
Ctrl +G	Group	Enables / disables the current group.
Ctrl +G	Groups	Adds new group
Ctrl +G	Point element	Adds current element to last active/used group
Ctrl	2D panel	The old mode of selection of active elements, elements of only the active assembly are selected
Ctrl	Variables, Element's library	Structuring groups of variables. When holding down the Ctrl key and in case of inactive element, [Variables] in the model tree, when added by a double click, prompts you to select a parent element
Ctrl+Shift+V	Variables, Model tree	Create a new variable in the [Variables] node selected in the model tree
Ctrl+Shift+V	Element parameter	Create a new variable and attach to the current parameter of the active element

Ctrl+L	2D panel, Model tree	Creates a [Link] element in the active assembly
Ctrl+Shift+L	2D panel, Model tree	Creates a [Link] element with a parent assembly request
Shift+L	2D panel, Model tree	Adds [Elastic nonsymmetric link]
Ctrl +ЛКМ click	2D panel, Model tree	The element is added with dimensions according to the position of the cursor, and not the dimensions of the previous element.
Ctrl+2	Element parameter	Multiplies the current parameter of the element by 2
Ctrl+4	Element parameter	Multiplies the current parameter of the element by 4
Ctrl+Shift+2	Element parameter	Divides the current parameter of the element by 2
Ctrl+Shift+4	Element parameter	Divides the current parameter of the element by 4
Shift	Element drag and drop	While holding down the Shift key, the drag and drop of an object leads to copying the element to a new position, and deleting from the old one
ctrl+shift+Z	Model tree	Moving through the history of active elements. Activates the previous active item
ctrl+shift+Y	Model tree	Moving through the history of active elements. Activates the next active item

25 LITERATURE

1. **Иванов А.В.** Расчет резонансных режимов газотурбинных двигателей, Труды Московского авиационного института, т. 245, 1972, стр. 66-77.

2. **M.K. Leontiev, E.V.Borzdyko and S.L.Zvonarev.** Gas-turbine dynamic response analysis through simulation. Proceedings of the international conference on rotating machine dynamics ROTORDYNAMICS'92. Poster Session Papers. Hotel Des Bains, Venice, Italy, 28-30 April 1992, pp. 1-10.

3. **M.K. Leontiev, S.L.Zvonarev.** Improvement of Gas-Turbine Engine Dynamic Structure With Rotor Rubbing Through Mathematical Simulation. Proceedings of the international conference VIBRATION & NOISE '95. Venice, Italy, 1995, pp. 641-649

4. **Леонтьев М.К., Иванов А.В., Дегтярев А.А.** Программная система для расчета и анализа роторной динамики турбомашин Dynamics R3.1. Свидетельство об отраслевой разработке №2716. Отраслевой фонд алгоритмов и программ. Государственный информационный центр информационных технологий. Министерство образования Российской Федерации. 2003 г.

5. **Леонтьев М.К.** Современные методы расчета динамических характеристик роторных систем. NASTRAN или DYNAMICS? “Двигатель”, 2004, №3 (33), С.14-16.

6. **Иванов А.В., Леонтьев М.К.** Модальный анализ динамических систем роторов. “Известия высших учебных заведений. Авиационная техника”. 2005, №3. С.31-35.

7. **Д.В. Хронин.** Теория и расчет колебаний в двигателях летательных аппаратов, М.:Машиностроение, 1980 г.

8. **Ivanov A., Leontiev M., (2004)** Vibrations of multi-shaft systems with gears. Consulting agreement on rotordynamic analysis and other services. Stage 1. Technical Report, INA - Moscow aviation institute, 34 pp.

9. **Ivanov A., Leontiev M.** Vibrations of multi-shaft systems with gears. Consulting agreement on rotordynamic analysis and other services. Stage 5. Technical Report. INA- Alfa-Tranzit Co. Ltd, December, 2003. 40pp.

10. **Дегтярев С.А., Леонтьев М.К., Попов В.В**. Совершенствование методики определения матрицы податливости «Беличьего колеса» в опорах роторов авиационных газотурбинных двигателей // Вестник Самарского университета. Аэрокосмическая техника, технологии и машиностроение. 2016. №1.

11. **Биргер И.А., Шорр Б.Ф., Иосилевич Г.Б.** Расчет на прочность деталей машин. М.: Машиностроение. 1993. Иосилевич Г.Б., Строганов Г.Б., Маслов Г.С. Прикладная механика. М.: Высшая школа, 1989.

12. **А.И.Белоусов, В.Б.Балякин, Д.К.Новиков.** Теория и проектирование гидродинамических демпферов опор роторов. - Самара: Издательство СНЦ РАН. 2002. – 335 с.

13. **William T. Tomson.** Theory of Vibration with Applications. Prentice - Hall, 1981.

14. **Bohm,T. Tame** The Menace of Turbine Vibration, SAE Journal, February 1966, pp.44-48

15. **Hibner, H.** Dynamic Response of Viscous-Damped Multi-Shaft Jet Engines, AIAA J.Aircraft,Vol.12, No.4, 1975, pp.305-312.

16. **Childs,D.W. A** Modal Transient Rotor-Dynamic Model for Dual-Rotor Jet Engine Systems, Journal of Engineering for Industry. Trans.ASME, August 1976, pp.876-882.

17. **Gunter, E.J.,Li,D.F**. Component Mode Synthesis of Large Rotor Systems, Trans.ASME, Vol.104, Jule 1982, pp.552-560.

18. **Marmol, A.A., Akin,J.T.** Rapid Verification of Engine Rotor Case Flexibilities by the Modal Comparison Algorithm, Journal Aircraft, Vol.12. No.4 1975.

19. **Mohan, S.Hahn,E.J.** Design of Squeeze-Film Damper Supports for Rigid Rotors, Journal of Engineering for Industry, Trans.ASME, August 1974, pp.976-982.

20. **Bhat, S.T., Buono,D.F., Hibner,D.H** Analysis of High Load Dampers, NASA-CR-165503, Final Report, 1981, pp.79.

21. **M.A. Prohl**, "A General Method for Calculating Critical Speeds of Flexible Rotors", Journal of applied Mechanics, Vol. 12, 1945, pp. 142-148.

22. **Timoshenko,** "Vibration Problems in Engineering", D. Van Nostrand Company, 3rd Edition, 1955.

23. **Gunter, E. J.,** "Dynamic Stability of Rotor Bearing Systems", NASA SP-113, Washington, D.C., NASA 1966.

24. **Johne M. Vance,** Rotordynamics of Turbomachinery, A Willey-Interscience Publication, 1988

25. **Hurty, W.S.,** 1965**,** "Dynamic Analysis of Structural Systems Using Component Modes", AIAA Journal, Vol. 3, No.4, pp 678-685.

26. **Craig, R.R. and Bampton, M.C.C**., 1968, "Coupling of Structures for Dynamic Analysis", AIAA Journal, Vol.6, No.7, pp.1313-1319.

Sellgren, U., 2003, "Component Mode Synthesis/A method for efficient dynamic simulation of complex technical systems", Technical report. ISSN 1400-1179, pp.

27. **Geradin, M., Cardona, A**., 2000,"Flexible Multibody Dynamics/A Finite Element Approach", JOHN WILLEY & SONS, LTD, pp. 327

28. **Ivanov A., Leontiev M.,** (2004) Vibrations of multi-shaft systems with gears. Consulting agreement on rotordynamic analysis and other services. Stage 1. Technical Report, INA- Moscow aviation institute, pp. 34

2

29. **Felippa C.A., Park K.C.,** "The Construction of Free-Free Flexibility Matrices for Multilevel Structural Analysis", Report No. CU-CAS-01-06, College of Engineering University of Colorado, June 2001, pp.33

30. **David P. Fleming, Jerzy T. Sawicki, J.V. Poplawski,** Unbalance Response Prediction for Accelerating Rotors With Load Dependent Nonlinear, June 2005, NASA/TM-2005-213801

31. **Childs, Dara.W.** "Turbomachinery rotordynamics: phenomena, modeling and analysis". JOHN WILLEY & SONS, INC.1993. pp. 476

32. **S. D. Cohen and A. C. Hindmarsh,** "CVODE, A Stiff/Nonstiff ODE Solver in C," Computers in Physics, Vol. 10, No. 2 (March-April 1996), pp. 138-143

33. **NASA Conference Publication 2250** “Rotordynamic Instability Field Problems, Rotordynamic Instability Problems in High Performance Turbomachinery”, Texas A&M University, May 10-12, 1982, pages 1-19.

34. **Jimming Li, Pranabesh De Choudhury, Mike Sharples, John Wright** “Experiences and Applications of API 617 Full Stability Analysis (Level - II) on Centrifugal Compressors”.

35. **Alford, J.S.,** 1965, “Protecting Turbomachinery from Self-Excited Rotor Whirl”, ASME Journal of Engineering for Power, 87 (4), pp.333-344

36. **J.C.Wachel** “Rotordynamic Instability Field Problems, Rotordynamic Instability Problems in High-Performance Turbomachinery”, NASA Conference Publication 2250, Texas A&M University, May 10-12, 1982, pages 1-19.

37. **API Standard 617**, 2002, Axial and Centrifugal Compressors and Expander-Compressors for Petroleum, Chemical and Gas Industry Services, Second Edition, American Petroleum Institute, Washington, D.C

38. **API Standard 684,** 2005, Tutorial on Rotordynamics: Lateral Critical Speeds, Unbalance response, Stability, Train Torsional and Rotor Balancing,, Second Edition, American Petroleum Institute, Washington, D.C.

39. **John Nicholas, John A. Kocur** “Rotordynamic design of Centrifugal Compressors in Accordance with the New API Stability Specification”, Proceedings of the 34-th Turbomachinery Symposium, Turbomachinery Labarotory, Texas A&M University, College Station, TX, pp. 25-34, 2005

40. **Heshmat H., Walowit J. A., Pinkus O.** “Analysis of gas-lubricated foil journal bearings”, Journal of Lubrication Technology, Vol. 105, No. 4, 1983, pp. 647-655.

41. **Kim Y. C., Lee D. H., Kim K. W.** “Performance analysis of double-bump air foil bearings”, Tribology Transactions. Vol. 51, No. 3, 2008, pp. 285-295.

42. **Black, H. F., Jenssen, D. N.,** Dynamic Hybrid Bearing Characteristics of Annular Controlled Leakage Seals, Proc Instn Mech Engrs, 1970, Vol. 184, pp. 92- 100.

43. **Childs, D. W.** (July 1, 1983). "Dynamic Analysis of Turbulent Annular Seals Based On Hirs’ Lubrication Equation." ASME. J. of Lubrication Tech. July 1983; 105(3): 429–436.

© Copyright
Alfa-Tranzit Co., Ltd ®” 2000-2024