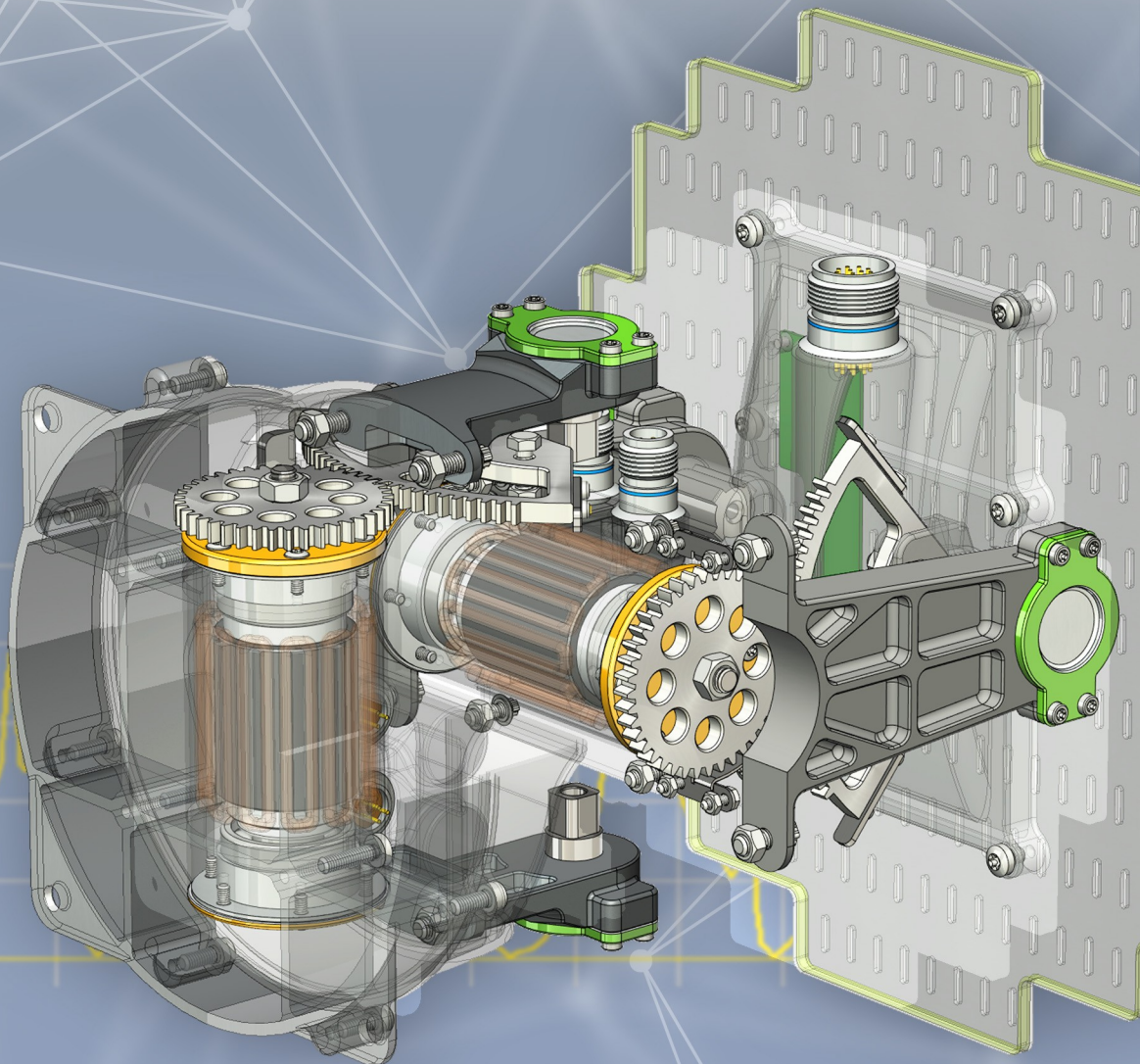


# T·FLEX Dynamics 17

USER MANUAL

MOTION SIMULATION



© Top Systems, 1992-2022

This Software and Related Documentation are proprietary to Top Systems. Any copying of this documentation, except as permitted in the applicable license agreement, is expressly prohibited.

The information contained in this document is subject to change without notice and should not be construed as a commitment by Top Systems who assume no responsibility for any errors or omissions that may appear in this documentation.

T-FLEX Parametric CAD, T-FLEX CAD, T-FLEX Analysis, T-FLEX Dynamics are trademarks of Top Systems.

All other trademarks are the property of their respective owners.

## Table of Contents

Introduction.....	4
The Structure of a Dynamic Calculation Study.....	6
Rules of Performing Dynamic Calculations.....	8
Study Management Utilities.....	8
Body Fixation.....	10
Creating Study.....	10
Study Parameters.....	13
Joints.....	17
Loads.....	20
Force.....	20
Linear Velocity.....	23
Rotation.....	24
Torque.....	25
Initial Velocity.....	26
Spring.....	28
Sensors.....	31
Results.....	34
Running Calculations.....	37
Utilities of Displaying Calculation Results.....	42
Limitations of the Express Dynamics Module.....	45

## Introduction

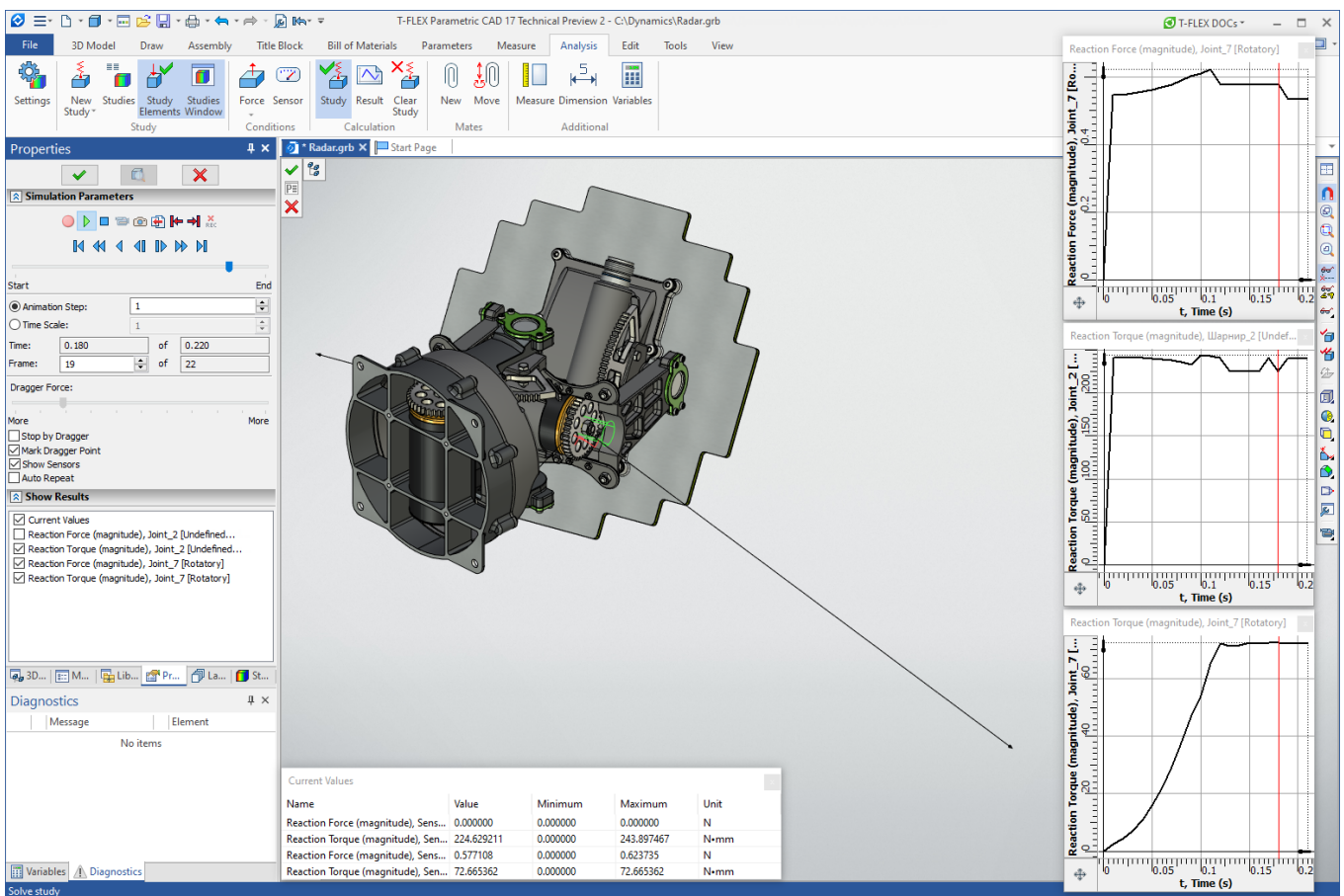
T-FLEX Dynamics module is intended for quantity and quality analysis of mechanisms kinematics and dynamics. The module is focused on analysis of mechanical systems subjected to external and internal forces, mass-inertial characteristics of individual parts of the system and interaction of mechanical system components. T-FLEX Dynamics is used, for example, to simulate the movement of mechanisms with complex relationships when designing new machinery or equipment, to check jamming, to simulate falls and interactions of bodies at collisions.

T-FLEX Dynamics module is integrated with T-FLEX CAD. They use common principles, common user interface elements and unified geometric model. Full associativity of initial model and calculated data and possibility to use calculation results for the further researches using T-FLEX CAD tools are supported.

Express Dynamics Module has some [limitations](#) on the types of loads and on the conclusion of the results of dynamic analysis.

The dynamical analysis system is capable of solving the following studies:

- Analysis of the motion trajectory, velocity and acceleration of any point of a component in a mechanical system under applied forces;
- Analysis of performance times of a mechanical system (the time to the target point, the time for vibrations to settle, etc.);
- Analysis of forces building up in components of a mechanical system during motion (reaction forces at the supports, joints, etc.).



A mechanism's model is defined as a system of solid bodies, joints, and loads. The system is created on the basis of T-FLEX CAD three dimensional geometric model and mates. Solver takes into account mass-inertial characteristics of three dimensional model bodies.

The data for the analysis is automatically accessed directly from the geometrical model created in the T-FLEX CAD system. The familiar tools of T-FLEX CAD are used in modeling, with mates and degrees of freedom employed to define relations between three-dimensional bodies. The system also provides the means of modeling the contact between arbitrary solid bodies, being capable of processing a simultaneous contact of hundreds and thousands of solid bodies of arbitrary shape.

Loads on solid bodies are defined as the initial linear and angular velocities, forces, moments (torque), springs, gravity, etc. To read the results, special sampling elements are used. Handling of the results of calculations is limited for the free module (the tools to obtain numerical calculation results are not available). In the commercial module, the calculation results are displayed as graphs, dynamic vector arrows and as an array of numbers (graph points). Numerous values are available for analysis: coordinates, velocities, accelerations, reaction forces in joints, forces in springs, etc. The user can observe the model behavior from any viewpoint immediately during the actual calculation. One can create animation clips from the obtained results of a dynamic calculation.

## The Structure of a Dynamic Calculation Study

The commands of the dynamic motion study are available on the **Analysis** tab of the Ribbon panel. Additionally, this menu contains the commands of the finite element analysis. Some of them are common across both study types.

To run dynamic calculations, the user creates a **Dynamic Motion study**. It consists of the following components:

- [bodies of the study](#);
- [study parameters](#);
- [joints](#);
- [sensors](#);
- [loads](#);
- [results](#).

### Contacts

One of the main features of the system is a realistic modeling of contacts between assembled bodies. The user is relieved from the task of defining contact points manually. It is possible to specify contacts calculation of the individual pairs of bodies and parameters of impact and friction individually for each pair of the bodies. All calculations are run by the system automatically. There are two methods of contacts definition:

- The first one is based on the use of a set of analysis functions for contacts and intersections based on the Parasolid kernel, which allows to perform precise analysis of the geometry in the area (or several areas) of the contact that gives the system possibilities for realistic modeling of bodies contacts and impacts.
- The second one is based on analysis of faceted representation of the geometry. The method allows to improve performance and stability in comparison to the bodies interaction check using Parasolid.

### Friction and Impact Parameters

The system can model the characteristics of interaction between system elements.

For joints:

- **Friction Coefficients** of the static friction, dynamic friction and viscous friction.
- **Tightness**. For some joint types, one can account for the tightness of the connection between parts. That means, parts are connected with some additional force and are subjected to the elastic strain due to a mismatch in the mating dimensions. The parameter units are those of a force. It determines the additional reaction in the joint, that will be accounted for when calculating the friction forces.
- **Geometrical Dimensions** of a joint are accounted for when calculating friction in the joint.

For the contacts:

- **Friction Coefficients** of the static friction, dynamic friction, rolling friction and pivoting friction.
- **Restitution Coefficient** is used to describe an impact and defines the fraction of the Solids' mechanical energy that remains after the impact.
- **Minimum Velocity**, above which an impact is considered elastic (a weak collision is considered nonelastic).

### Results

Results are recorded based on readings collected by sensors, or displayed in the way of graphical marks (arrows) in the real time while running calculations. Results are usually stored in the way of plots (graphs).

Calculations are based on the method of computing the system state frame by frame. The information about the system state is recorded for each frame. To calculate each frame of a recording, the system performs a certain number of modeling steps. This can be one step per frame, or a thousand. The user can specify the normal number of steps for the system to use. If the specified normal number of steps is not enough to achieve an accurate frame, then the system will use more steps, up to the maximum value also specified by the user.


The automatic assessment of the required number of steps (from normal to the maximum) may not deliver optimum in some cases. Therefore, in some examples, for instance, when small quickly moving bodies collide, the normal number of steps in a frame must be rather large.

## Rules of Performing Dynamic Calculations

Dynamic calculations are based on a special system object – a **study**. The study unites data and elements required for running specific calculations of a model. It contains the data that define the direction of gravity, the default properties of the study elements (joint properties, friction forces, contact properties), the time factor in the modeled process, as well as the information about the used bodies, loads, and properties of relations between individual components, etc. After completing the calculations, the study will also include the calculation results. A dynamic motion study is associatively related with the 3D model. As some parameters or contents of the model change, the study would automatically adjust.

### Study Management Utilities

To display the list of a model's studies, there is a special **Studies** window. It contains the portion of the current model tree, which represents the contents of the engineering analysis studies (similar functions are provided in the **3D model** tool window. The window provides a quick access to elements of each study. Dynamic motion study elements (joints, loads, sensors, results) are arranged into groups. The studies tool window is common for the studies in the finite element and in the dynamic calculation. The command to open this window is called as follows:

Icon	Ribbon
	Analysis > Study > Studies Window
Keyboard	Textual Menu
<3MW>	Analysis > Show Studies Window

Similar to other types of engineering studies that can be contained in a T-FLEX CAD model (finite element studies), it is possible to have several dynamic motion studies in one document. Each of the studies may contain its own set of elements and boundary conditions in order to find solutions in different formulations or under different loads.

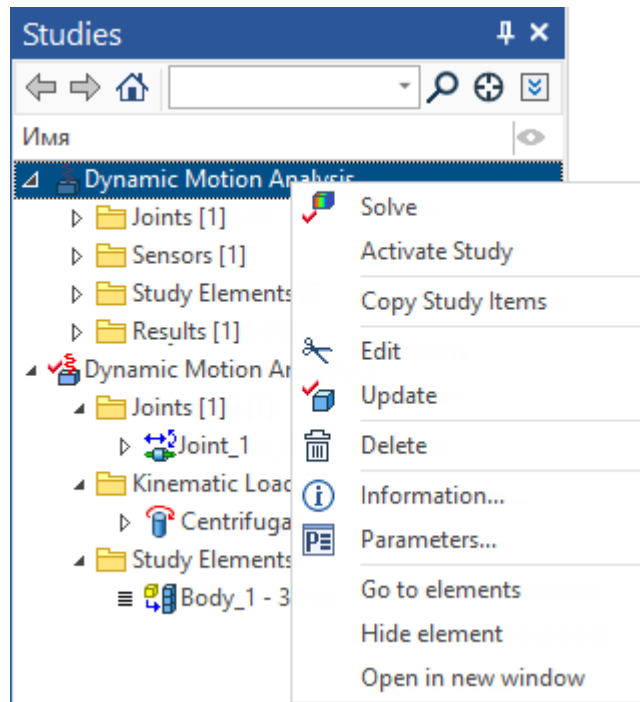
Most of study management commands are available in the context menu that can be accessed in this window.

The study currently being worked on is called active. The active study's icon has a red check in the studies window. To make another study active, use the context menu command **Activate**.


The **Update** command is available in the context menu of the selected Study. It is used to quickly refresh the study data after making changes to the model, if the full recalculation was skipped.

To a get a study or to modify its properties, use the commands **Edit** and **Properties**.

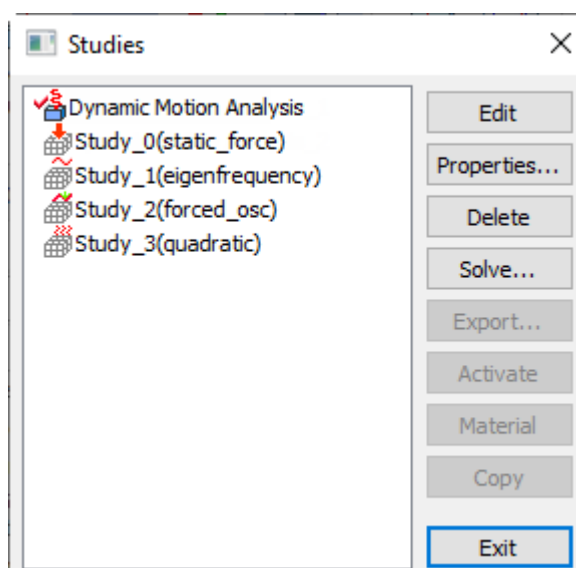




A special command is provided for managing the list of studies:


Icon	Ribbon
	Analysis > Study > Studies
Keyboard	Textual Menu
<3ML>	Analysis > Studies

The dialog window of this command displays the list of all studies existing in the current document. This window is common for both the dynamic motion studies and finite element analysis. The buttons for calling the main commands are to the right of the list. The export, copying and material property definition commands are not available in the dynamic calculation case.



If several studies are created in the system, only one of them will be active. New elements are created, and calculations are run, only in the active study.


## Body Fixation

All bodies added or created in the scene can be fixed. To do this, select the body in the scene and activate **Fix Component**  option in its context menu .

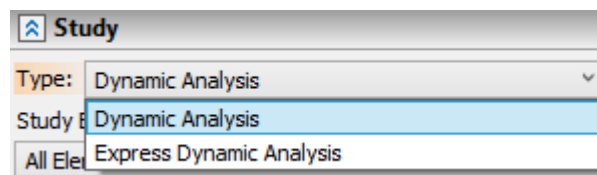
Fixation control is possible only if there is at least one mate in the document, or a dynamic motion study.


## Creating Study

To create a new dynamic motion study, use the command:

Icon	Ribbon
	Analysis > Study > New Study > Dynamic Motion Study
Keyboard	Textual Menu
-	Analysis > New Study > Dynamic Motion Study

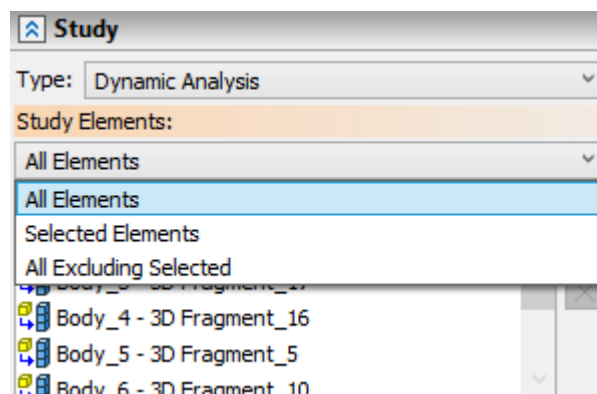
This command allows creating two types of studies – **Express Dynamic Analysis** and **Dynamic Motion**. The **Express Analysis** is shipped free together with the T-FLEX CAD 3D program and has certain [limitations](#).



A study can be created "as is", based on the data of the 3D model. To create such a study, just use the option  in the properties window or in the automenu. Contact between bodies is ignored by default (whereas the friction is accounted for by default). The necessary number of joints is automatically created, based on the mates and degrees of freedom in the 3D model.

The study's properties window has numerous parameters to allow for a precise study setup. For the user's convenience, the window is arranged into blocks.

In the **Solids** block of parameters, you can manually specify the model Solids that shall be included in the dynamic motion study. This setting is needed in the case when the model has more bodies than required for calculations. This also saves the computational resources, by excluding insignificant objects from calculations.



By default, all objects in the model are displayed in the list. There are two options for generating a list of design objects. Depending on which side it is more convenient to select from, you can select objects that actually participate in the study (the switch in **Selected Elements** position), or switch to **All Excluding selected** mode, and then all objects of the model, except for the selected ones, will participate in the study.

To select solids to be included in the study, use the automenu option:

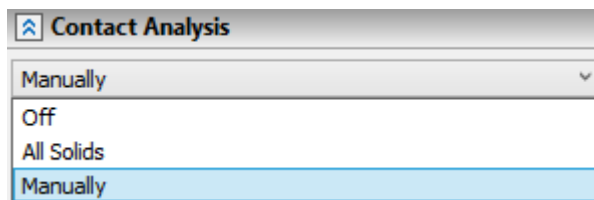


<S> Select Solids of motion analysis

The list of selected solids is edited using the **[Delete Element from the List]**  and **[Clear]**  buttons.

The **Contact Analysis** block allows engaging the check of mutual body penetration. At the top of the parameters block, there is a switch made into a combo box. It has three positions:

- **Off** - no check of body contacts is performed, and the rest of parameters are inaccessible.
- **All Solids** - automatic mode, in which the analysis of contacts between all bodies participating in the calculation is performed, with the ability to set [individual friction and impact settings](#) for each contacting pair.

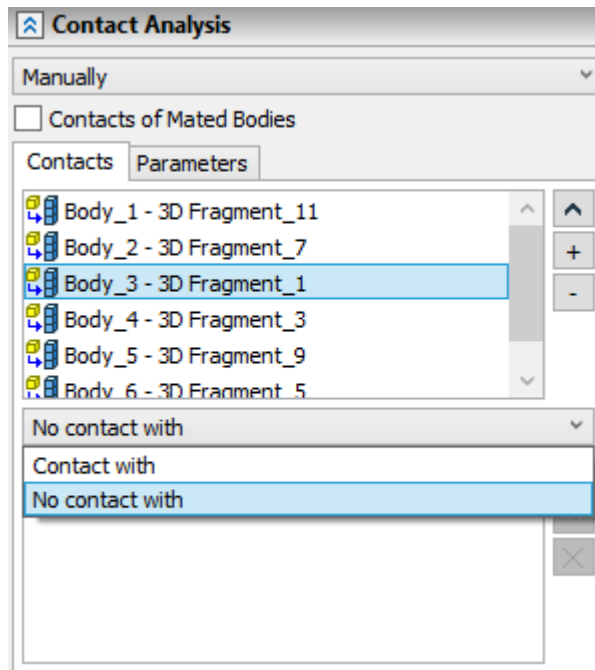




- **Manually** - the mode of specifying specific pairs of contacting bodies with the ability to set [individual friction and impact settings](#) between them. This mode can also be helpful, if it is known exactly that certain solids in the study will never interact, and their mutual contact can be ignored. This refined study setting sometimes helps significantly to save computational resources.

The **Contacts of Mated Bodies** parameter is intended for accounting of contacts of mated bodies.

It should be borne in mind that mating objects are usually close to each other, and therefore there is a significant risk of unnecessary contacts that can lead to incorrect results of dynamic calculations.

Below there are **Contacts** and **Parameters** tabs. On the **Contacts** tab there is a field with all the objects of the study. For each object of this field, two lists are automatically generated: objects with which it contacts and, conversely, does not contact. These lists are displayed alternately below the main list depending on the switch, which has two positions: **Contact with** and **No contact with**. Initially, the system creates contacts between all objects of the study. It is allowed to adjust the contact lists for each solid.





The [**Consider All Solids Contact**]  and [**Consider No Contacts**]  buttons, located in the objects field, are designed to quickly form and delete all possible contacting pairs.

To define contacts, first select the intended solid in the first field. After that, set the switch as desired and select objects using the automenu option:



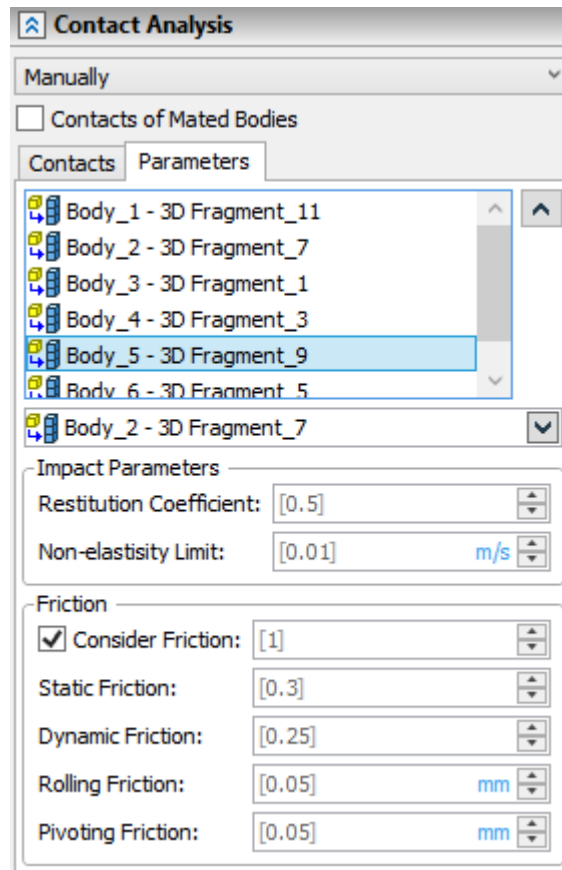
<R> Select Solids used in contact analysis

The selected objects are added to the contact list. The selected pair of solids is highlighted with different colors in the 3D window. The list of selected solids is edited using the [**Delete Element from the List**]  and [**Clear**]  buttons.

The **Parameters** tab allows manually defining the individual impact and friction settings for each pair of interacting solids.

At the beginning, the system sets all parameters of all pairs of bodies as [**From study**], so that their values are copied from the **Study Parameters** dialog.

To define individual contact parameters between two solids, such a pair shall first be selected in the fields of the properties window. The first solid is selected in the upper field. The second solid is selected from the latter list. The selected objects are highlighted with different colors in the 3D window. After selecting a pair, you can define its individual contact properties at the lower part of the properties window.



There is also an alternative way to select a pair with the ability to specify solids directly in the 3D window. For this it is convenient to use the automenu options. After clicking the option,



<C> Select Solids to define contact parameters

the system goes into the mode of defining individual contact parameters, and two additional options appear in the automenu to select the first and the second solid:




<F> Select first Solid



<G> Select second Solid

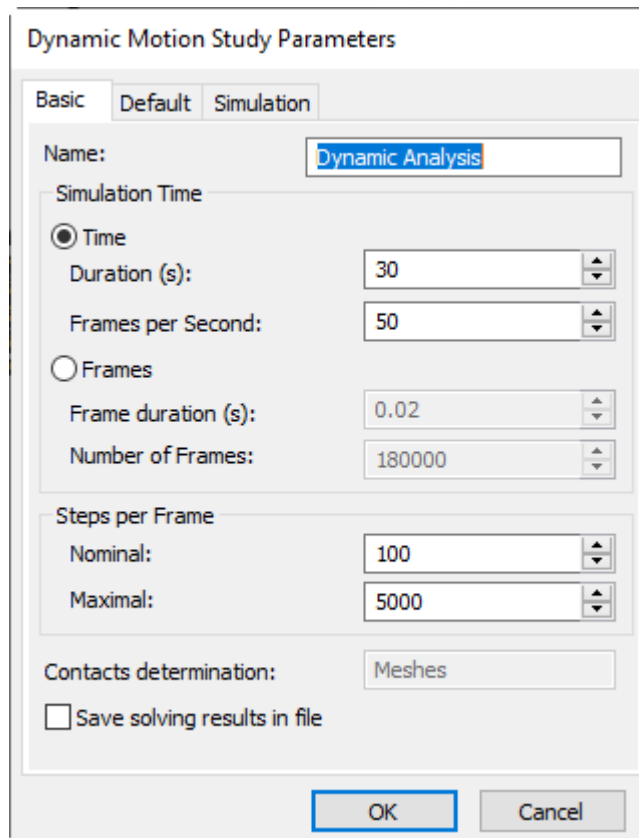
These options activate in turns. The selected objects are marked with different colors in the 3D window.

## Study Parameters

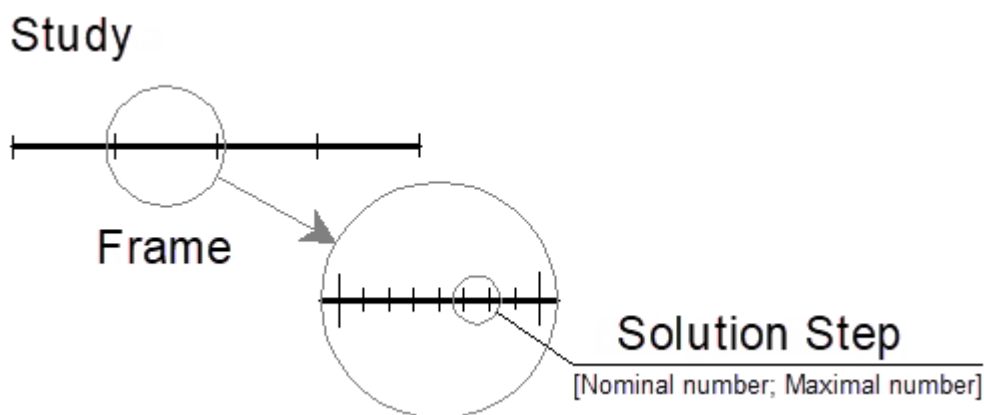
The option  calls the dialog to define the time characteristics and accuracy of parameters of a calculation, as well as a geometrical properties of certain objects in the current dynamic motion study. The dialog has three tabs.

On the **Basic** tab you can define (or modify when you edit) the name of the dynamic motion study. The **Simulation Time** group of interface controls defines the time-related parameters and the modeling step for calculating the current study. These parameters can be defined in two ways. You can define the overall duration and the number of frames per second, or define the duration of one frame and the total number of frames.

Time-related parameters can be modified in the calculation parameters just before starting calculations.



The group of parameters **Steps per Frame** defines the number of iterations found by the system to find the solution for each frame. There are two parameters: the first one indicates the normal number of steps to find the precise solution, whereas the second parameter limits the maximum number of steps for the system to find the solution, if the normal number of steps is not enough.



The **Contacts determination** parameter displays a predefined [method for calculating contacts](#) between assembly bodies. The method is selected in the command **ST: Document Settings > 3D > Method of Contacts Definition**. In this case, no restrictions are imposed on the shape of the contacting bodies - the bodies can be absolutely arbitrary.

The **Save solving results in file** parameter includes a mode in which all calculation results are saved in a separate file with the model. This allows you to analyze the results of a previously calculated and saved task without having to re-perform the calculation. It should be borne in mind that saving the calculation results in a model document leads to an increase in the file size of the document.

On the **Default** tab you define the default parameters for the study's interacting components.

The screenshot shows the 'Dynamic Motion Study Parameters' dialog box with the 'Default' tab selected. The parameters are as follows:

Section	Parameter	Value
Joint Dimensions	Length:	50
	Width/Radius:	10
Friction Parameters	<input checked="" type="checkbox"/> Consider Friction:	1
	Static Friction:	0.3
	Dynamic Friction:	0.25
	Viscous Friction:	0
	Rolling Friction:	0.05
	Pivoting Friction:	0.05
	Tightness:	0
Impact Parameters	Restitution Coefficient:	0.5
	Non-elasticity Limit:	10

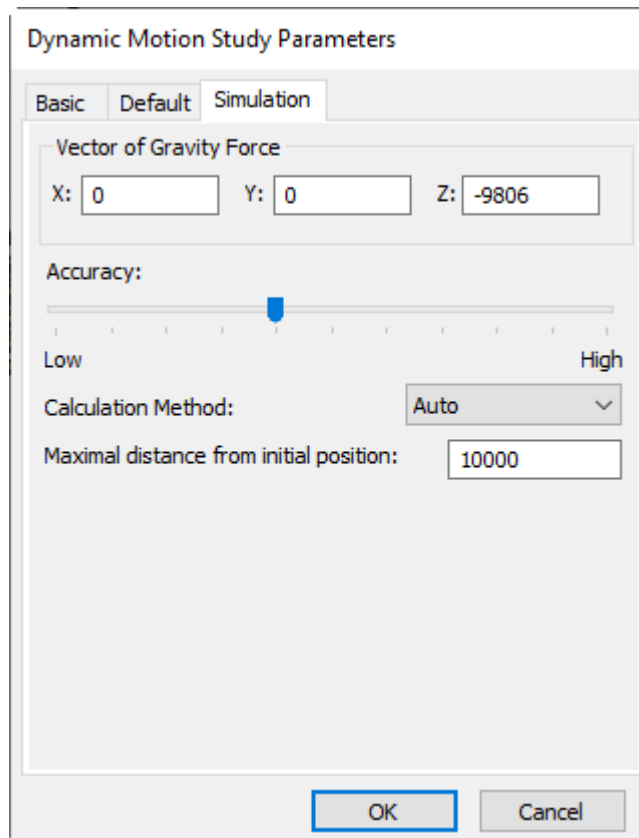
In the **Joint Dimensions** group you define the geometrical dimensions for the joints, whose dimensions could not be determined by the system based on the assigned mates and degrees of freedom. You can set a default for the two dimensions - **Length** and **Width/Radius**.

In the **Friction Parameters** group you define the default friction parameters. Those parameters will be supplied to various joints at creation. You enter in the respective fields the coefficients of the static friction, dynamic friction (at sliding), viscous friction (additional velocity-dependent friction), rolling friction, pivoting friction. The two latter parameters are considered only if the body contact is accounted for.

The **Tightness** parameter units are those of a force. It determines the additional reaction in the joint, that will be accounted for when calculating the friction forces.

In the **Impact Parameters** group you make the settings to evaluate impacts. **Restitution Coefficient** defines the fraction of the kinetic energy that remains after the impact. The value "1" means a purely elastic impact, and "0" – purely nonelastic. The **Non-elasticity Limit** parameter sets the velocity at the impact, below which the impact would be considered nonelastic.

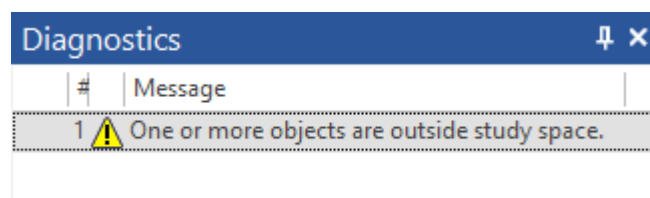
On the **Simulation** tab you define the gravity parameters (my default, the Earth's G-force acts along the Z-axis).



The **Accuracy** parameter defines a certain relative corridor, within which the system finds the solution. The parameter tells the system, how accurately the joints will be accounted for in the simulation. The more accuracy is prescribed, the higher will be the quality of the results, but the system will work slower. Please also note that too strict accuracy requirements may cause the system to simply stop functioning because of a failure to find the solution.

The **Calculation Method** parameter is one of the two available calculation methods - **Precise** or **Fast**. The **Precise** method shall be used with systems measuring up to several dozen of bodies and joints. Calculations could become intolerably slow in a system comprising hundreds of joints. In such cases, you should use the **Fast** method, which is as good with large systems as it is with small ones. The **Auto** mode allows the system to select a calculation method depending on the model complexity.








The **Maximal distance from initial position** sets the limits of the calculation space. Thus, if the elements of the study are located outside the calculation space (before the start or during the calculation process), the system will issue a corresponding notification in the **Diagnostics** window with the simultaneous stopping of the calculation.





## Joints



Joints define relations and interaction between individual bodies in a dynamic motion study. Those are automatically created in the study based on the degrees of freedom and mates defined in the model. The system allows modeling friction in typical types of joints, plus defining impact parameters in one-sided contacts. Presently, the following joint types are realized in the system:






- **Spherical Joint** . Rotation is permitted in all directions, but with no .
- **Rotational Joint** . Rotation is only permitted about a certain axis. Other translation and rotation in any other direction are not allowed. This imitates a door hinge.
- **Translational Joint** . Only a translation in a certain direction is permitted.
- **Cylindrical Joint** . Rotation is only permitted about a certain axis, together with the translation along it.
- **Helical Joint** . This imitates fastening with a screw. It is created based on the transmission mates. Rotation about one axis is permitted, with a simultaneous translation along it.
- **Contact-Type Joint** . Contact-type joints define a relation between a pair of body elements that are connected with other joints. Contact pair members can vary, as: point-point, point-curve, line-plane, plane-body, plane-plane, etc. The relation can be tangency, distance, or distance with an inequality (in the latter case, you have a one-sided contact joint).
- **Generic Joint** . All other combinations of relations, which the system couldn't classify as any known joint, will still be accounted for in the study by being converted into a generic joint. Such relations are considered ideal. For them, you cannot specify friction parameters or geometrical properties.

Each joint type has certain properties that the user can control, for example, geometrical dimensions, friction parameters, etc.

**Remark:** the Cardan joint is considered as the system of three bodies rather than a single joint.


Joints are created automatically at the time of the study creation, based on the mates and the specified degrees of freedom of the 3D model elements. As you make changes to the model's mates, the set of joints will change when the study is updated. The user can additionally adjust the mechanical properties by defining friction parameters. The set of properties may differ, depending on the joint type. The following table comprises the settings that can be adjusted, for all joint types.

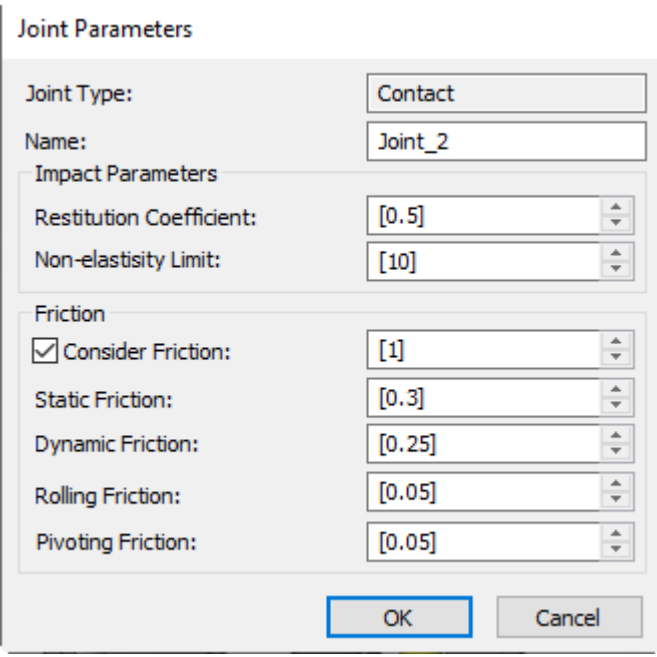
Joint Type	Geometrical Parameters	Friction and Impact Parameters
 Spherical joint	<ul style="list-style-type: none"> <li>• Radius</li> </ul>	<ul style="list-style-type: none"> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Viscous friction</li> <li>• Tightness</li> </ul>
 Rotational joint	<ul style="list-style-type: none"> <li>• Radius</li> <li>• Length</li> <li>• Displacement</li> </ul>	<ul style="list-style-type: none"> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Viscous friction</li> <li>• Tightness</li> </ul>

Joint Type	Geometrical Parameters	Friction and Impact Parameters
 Translational joint	<ul style="list-style-type: none"> <li>• Length</li> <li>• Width</li> <li>• Thickness</li> <li>• Rotation</li> <li>• Displacement</li> </ul>	<ul style="list-style-type: none"> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Viscous friction</li> <li>• Tightness</li> </ul>
 Cylindrical joint	<ul style="list-style-type: none"> <li>• Radius</li> <li>• Length</li> <li>• Displacement</li> </ul>	<ul style="list-style-type: none"> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Viscous friction</li> <li>• Tightness</li> </ul>
 Helical joint	<ul style="list-style-type: none"> <li>• Radius</li> </ul>	<ul style="list-style-type: none"> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Viscous friction</li> <li>• Tightness</li> </ul>
 Contact-type joint	-	<ul style="list-style-type: none"> <li>• Restitution Coefficient</li> <li>• Non-elasticity limit</li> <li>• Static friction</li> <li>• Dynamic friction</li> <li>• Pivoting Friction</li> <li>• Rolling Friction</li> </ul>
 Generic joint	-	-

The way of modeling the friction phenomenon depends on the geometrical dimensions and sometimes on the joint orientation (in the case of a translational joint). Most of joints employ the geometrical parameters dependent on the overall dimensions (radius and length – for a rotational, cylindrical and helical joints, as well as width and thickness - for the translational one). Those define the physical dimensions of a joint. The geometrical parameters also include translation and rotation. Translation defines the displacement of the joint from the original position of the geometrical elements (surfaces of the faces, the point, etc.), which define the given joint. Translation occurs along the joint's axis. Rotation defines the angle, by which the block of the translational joint is rotated. Rotation occurs about the joint's axis.

Whenever a joint is created, its parameters are initialized with the data from [the study's general properties](#). Certain geometrical parameters (radius, length, etc.) may sometimes be obtained from the 3D model elements that define mates. If the system fails to obtain a joint's geometry, then this data is also accessed from the study's general properties.

If necessary, you can call the joint properties dialog and define the individual properties. The properties dialog can be called by double-clicking the joint in the studies window , or via the context menu.



The image shows a 'Joint Parameters' dialog box with the following fields and values:

Property	Value
Joint Type:	Contact
Name:	Joint_2
Impact Parameters	
Restitution Coefficient:	[0.5]
Non-elasticity Limit:	[10]
Friction	
<input checked="" type="checkbox"/> Consider Friction:	[1]
Static Friction:	[0.3]
Dynamic Friction:	[0.25]
Rolling Friction:	[0.05]
Pivoting Friction:	[0.05]

Buttons: OK, Cancel

## Loads

Loads - these are special objects created by the user before running calculations. You can specify forces, moments and rotatory effects in the way of loads. For some bodies it is possible to define the linear and angular velocity at the initial instant. A special load type is a spring. Thanks to the capability of defining the spring stiffness via a plot, this load type generally supports not just simplest springs with a linear dependency, but also bipolar forces governed by arbitrary laws.

The main types of loads:


- [Force](#). Creates a concentrated force acting in the specified direction. This force can be defined with a value or via a plot. In the latter case, you can specify a variable value that depends on time or a sensor parameter.
- [Linear Velocity](#). Implies linear movement of an object at a specified constant speed in a specified direction.
- [Rotation](#). Serves to create an actuator that spins the body about the specified axis with the specified angular velocity. The maximum torque developed by the actuator can be limited by the user as desired.
- [Torque](#). Serves to define the vector of an external moment acting on the body.
- [Initial Velocity](#). The user can specify the initial velocity for any Body in the study (the translational velocity of the Body's center of mass and the angular velocity). By default, when you start modeling, body velocities are zero.
- [Spring \(bipolar force element\)](#). Serves to create springs, dampers or arbitrary bipolar concentrated forces defined via a plot. This load can also be used in modeling a linear actuator that pushes two bodies with the specified velocity or acceleration. This can be used for modeling a hydraulic cylinder, to name an example.
- [Gravity](#). The system can account for any G value, and in any arbitrary direction, when running calculations.

Loads can be created only after creating the study. If several studies exist in the model, then the specified loads pertain only to the active study.

## Force

A **Force** can be applied at any point in the space, and at the same time it becomes bound to a specific body involved in the study. The direction of a force is specified separately. The direction can be fixed in the space, or it may vary depending on the position of any body in the dynamic study. To define the direction of the force action, select any additional body among those involved in the study.

To create a force, use the command:

Icon	Ribbon
	Analysis > Conditions > Force
Keyboard	Textual Menu
<3MF>	Analysis > Load > Force

Selection of the body that are affected by the force carried out by automenu options:



<B> Select Body

The system allows you to select simultaneously several bodies that are affected by force. In this case, the load will act with the specified value on each body.

The following automenu option is used to set the point of force application:



<E> Select Point

The selection of an object that determines the direction of the force is carried out using the following automenu option:



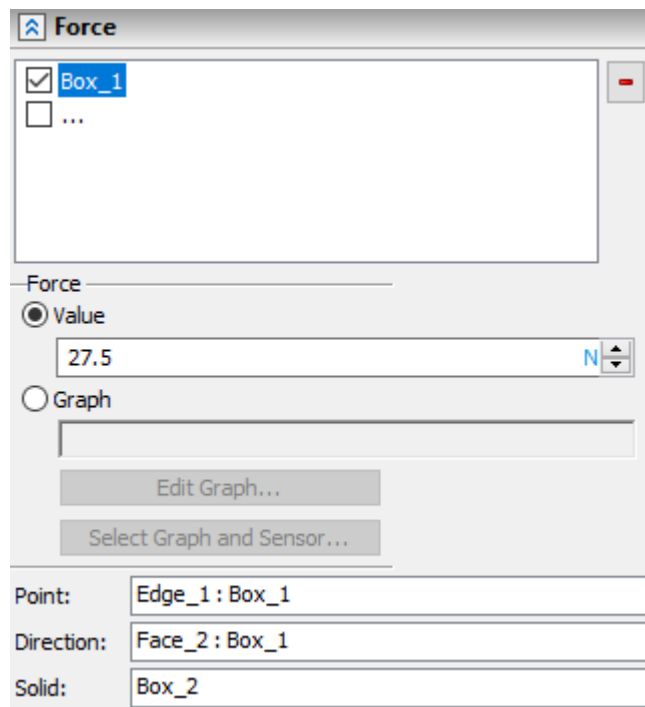
<D> Select Direction


To select a body with respect to which the direction is fixed, the following automenu option is intended:



<R> Select Body to Attach Direction

All selected objects are entered in the corresponding fields of the properties window. The flag located next to the selected loaded object in the list allows you to invert the direction of the force on this object.



Deleting a selected object from any field of the properties window during editing is carried out using the <Del> or the [Delete]  button. To delete the object that sets the direction, a separate option is provided for the automenu:



<U> Cancel Direction Selection

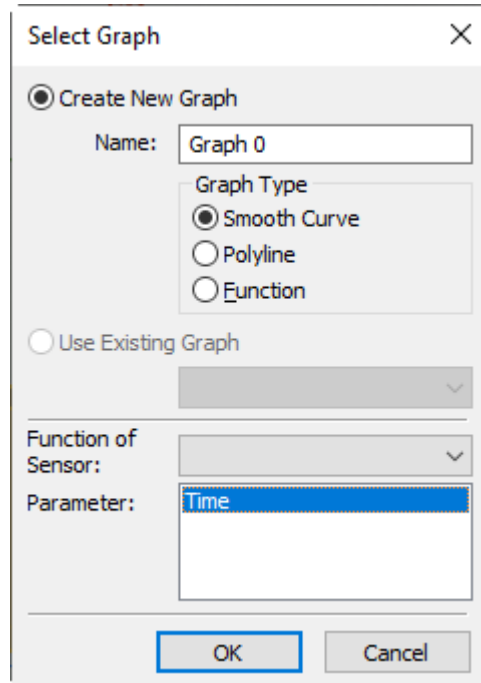
If the point of application of force is not specified, then the load will be applied to the center of mass of the selected body (set of bodies).

In the properties window the units and the numerical value of the force are set. To define the numerical value, set the radio switch into the **Value** position.

The force can also be defined with a plot (the radio switch in the **Graph** position). Its magnitude may be dependent on time or on any reading from any sensor of the given dynamic motion study. To define a dependency with a plot (graph), click the button [Select Graph and Sensor...].

After clicking the button, a dialog appears, in which you need to specify a plot. In this case, you can either create a new plot or use one existing in the system. After that, you need to specify, which parameter to use by the system as the argument to read the functional dependency. For a force, you can use either the time counter "Time", or some reading from any existing sensor. The sensor selection is done in the drop-down list **Function of Sensor**. After selecting a sensor, you can select one of its

parameters in the adjacent field. If no sensor is selected, then you can use only the time counter "Time" as the parameter.

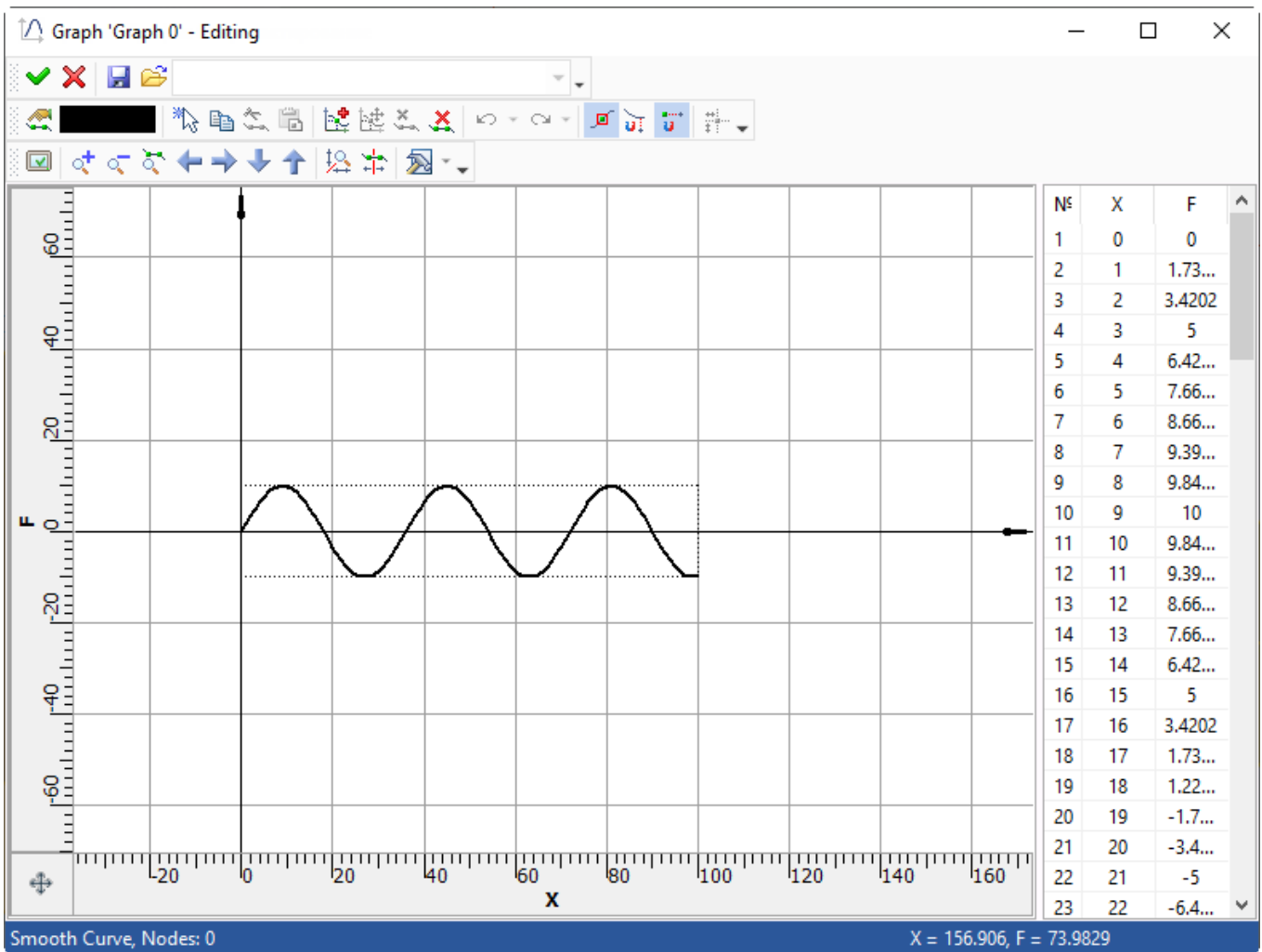


To create a new plot, set the switch to the **Create New Graph** position, after which you can enter the name of the new plot and its type in the respective fields. In the lower part of the dialog, select the desired parameter to read in the functional dependency. After clicking [OK], the graph editor will open, in which you can define a new functional dependency.

More detailed information on creating and editing plots can be found in the T-FLEX CAD help, chapter "Service 3D Tools and Entities", section "Graphs".

To use an existing plot, set the switch in the **Use Existing Graph** position and select a plot from the list. Also, specify the desired parameter to read in the plot. To confirm this selection, click the [OK] button.


The name of the plot will appear in the special field of the main properties window, with the driving parameter entered next in parentheses. There will be also the button [Edit Graph], with which you can call the graph editor to modify the functional dependency.



## Linear Velocity

The **Linear Velocity** load allows to create an external action that provides a linear movement of the selected solid at a constant speed in a given direction. As a limiter, it is allowed to set a limiting force, upon reaching which the impact will stop for the duration of the counter force.

To create a load, use the command:

Icon	Ribbon
	Analysis > Conditions > Linear Velocity
Keyboard	Textual Menu
<3MR>	Analysis > Load > Linear Velocity

To select the body, to which the load is applied, use the automenu option:



<B> Select Body

The system allows you to select simultaneously several bodies. In this case, the load will act with the specified value on each body. This load acts at the center of mass of the selected body.

To select a body with respect to which the direction is fixed, the following automenu option is intended:




<R> Select Body to Bind Translation Direction

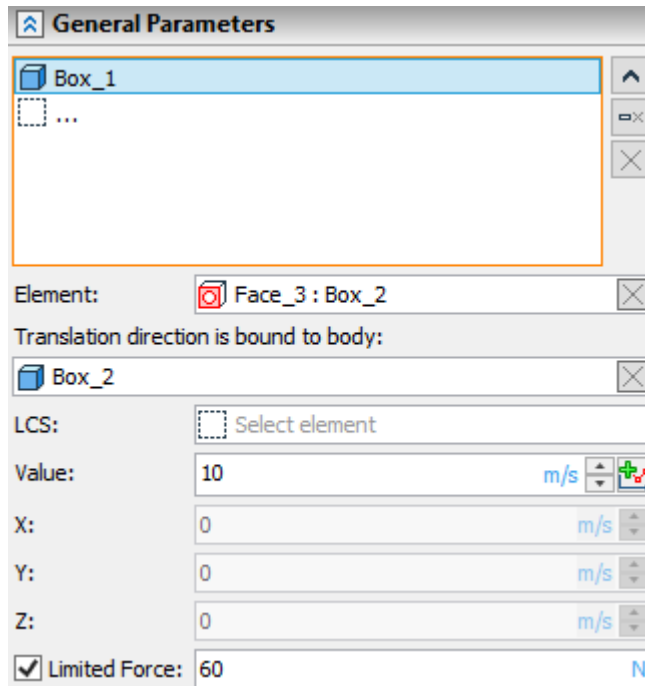
The selection of an object that determines the direction of movement is carried out using the following automenu option:



 <D> Select Direction of Movement

Also, to set the direction of movement, it is possible to use a local coordinate system with the expansion of the velocity vector along the axes of this coordinate system. The following automenu option is used to select a local coordinate system:

 <C> Select Coordinate Sistem of movement

After selecting the LCS, fields become available for specifying the numerical components of the velocity along each axis.



All selected objects are displayed in the corresponding fields of the parameters window. The selection can be canceled using the <Del>, the [Delete Element from the List]  and [Clear]  buttons. Separate automenu options are used to deselect the direction element and coordinate system:

 <U> Cancel Direction Selection

 <K> Cancel LCS Selection

Specifying the value of the linear speed and limiting force is carried out in the corresponding fields. These fields provide the ability to change the unit of measurement.


In addition to a constant, the linear velocity can be set in the form of a graph. Its value can depend on time or on any readings of any sensor of the active study. To do this, use the button. The further algorithm for creating a graph is similar to the actions described for the [Force](#) load.

## Rotation

The **Rotation** load allows to create an external force that rotates the selected body with a constant angular velocity about the specified axis. You can specify the maximum moment as the limit that cannot be exceeded by the **Rotation** load, with the force ceasing for the duration of the countering moment action.

To create a rotation, use the command:



Icon	Ribbon
	Analysis > Conditions > Rotation
Keyboard	Textual Menu
<3RB>	Analysis > Load > Rotation

To select the body, to which the load is applied, use the automenu option:



<B> Select Body

The system allows selecting several bodies simultaneously. In this case, the load of the specified magnitude will be applied to each of them.

The rotation direction (as a vector) is defined using the following automenu option:



<A> Select Axis of Rotation

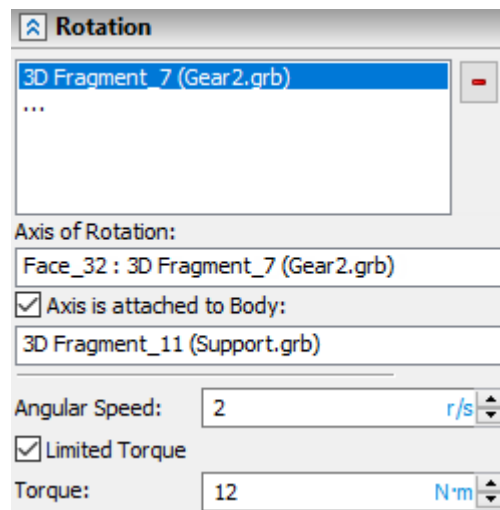
To define the axis, you can select any 3D object suitable for defining a direction.


The rotation direction can be related (bound) to any movable or still body in the study, using the automenu option:



<R> Select Body to Attach Axis of Rotation

All selected objects are entered in the corresponding fields of the properties window.



Deleting a selected object from any field of the properties window during editing is carried out using the <Del> or the [Delete]  button. To remove the object that defines the axis of rotation, set special option automenu:




<C> Cancel Axis Selection

The numerical value of the angular velocity and the units are defined in the properties window. If necessary, it is allowed to separately set the maximum torque value. To do this, enable the **Limited Torque** parameter and set the corresponding value.

## Torque

This type of load simulates a moment applied to the selected body or set of bodies. To create a moment, use the command:

Icon	Ribbon
	Analysis > Conditions > Torque
Keyboard	Textual Menu
<3MQ>	Analysis > Load > Torque

To select the body, to which the load is applied, use the automenu option:



<E> Select Element for Loading

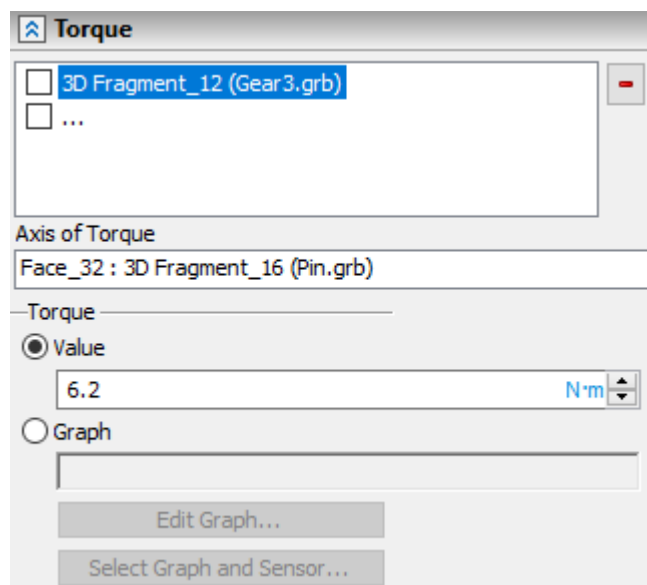
The system allows selecting several bodies simultaneously. In this case, the load of the specified magnitude will be applied to each of them.


The moment (torque) application direction (as a vector) is defined using the following automenu option:



<D> Select Axis of Rotation

All selected objects are entered in the corresponding fields of the properties window. The flag next to the selected load object in the list allows you to invert the load. Thus, it is possible to create a moment between the bodies.



Deleting a selected object from any field of the properties window during editing is carried out using the <Del> or the [Delete]  button. To remove the object that defines the axis of rotation, set special option automenu:




<U> Cancel Axis Selection

The torque value and the units are defined in the properties window. The torque value can be defined with a plot. For example, you can define a dependency of the torque on the angular velocity, using the sensor that is measuring the angular velocity. The steps to define the plot for the torque are fully identical to the steps described for the [Force](#) load type.

## Initial Velocity

The **Initial Velocity** load type creates an initial impulse applied to the selected body. To create the initial velocity, use the command:

Icon	Ribbon
	Analysis > Conditions > Initial Velocity
Keyboard	Textual Menu
<3MV>	Analysis > Load > Initial Velocity

To select the body, to which the load is applied, use the automenu option:



<B> Select Body

The system allows selecting several bodies simultaneously. In this case, the load of the specified magnitude will be applied to each of them. The selected objects are entered in the list in the properties window. This load acts at the center of mass of the selected body.

The user can create a linear or angular initial velocity. To define the initial velocity of each type, there are two tabs in the properties window with identical sets of parameters.

To select the direction, use the automenu options:



<D> Select Direction of Movement



<D> Select Axis of Rotation

When defining the direction, you can specify the numerical value of the velocity in the **Value** field.

It is also possible to use the local coordinate system in order to define the velocity direction, by having the velocity vector decomposed by the axes of this coordinate system. To select a local coordinate system, use the automenu options:



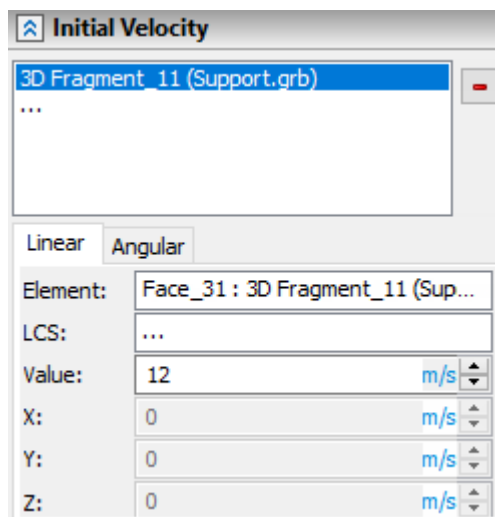
<C> Select Coordinate System of Movement




<C> Select Coordinate System of Rotation Axis

After selecting the LCS, the fields to define the velocity components along each axis become accessible.

All selected objects are entered in the corresponding fields of the properties window.



Deleting a selected object from any field of the properties window during editing is carried out using the <Del> or the [Delete]  button. To remove objects that define the direction and the axis of rotation, respectively, are separate automenu options:



<U> Cancel Direction Selection



&lt;U&gt; Cancel Axis Selection

Cancellation of selected LCS is carried out using the following options of the automenu:



&lt;K&gt; Cancel LCS Selection



&lt;K&gt; Cancel LCS Selection

## Spring

This load type allows creating a linear spring or a bipolar force element (a spring with nonlinear properties) between two points of two bodies included in the dynamic motion study. This load can also help modeling a linear actuator that induces a relative motion of two points of different bodies with the specified velocity, acceleration, or by the specified distance. The linear actuator also allows modeling such mechanism operation, as that of a hydraulic cylinder.

To create a spring, use the command:

Icon	Ribbon
	Analysis > Conditions > Spring
Keyboard	Textual Menu
<3MG>	Analysis > Load > Spring

To select the first and the second point of the load application, use the automenu options:



&lt;1&gt; Select Vertex



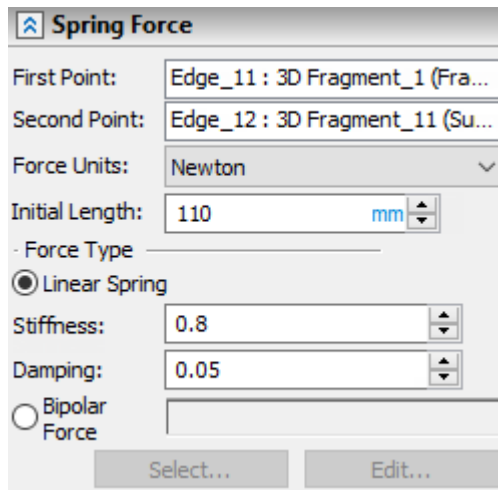
&lt;2&gt; Select Vertex

You can use any two characteristic points on the two bodies included in the dynamic motion study, as the locations of the load application. The selected elements are entered in the fields **First Point** and **Second Point** of the properties window.

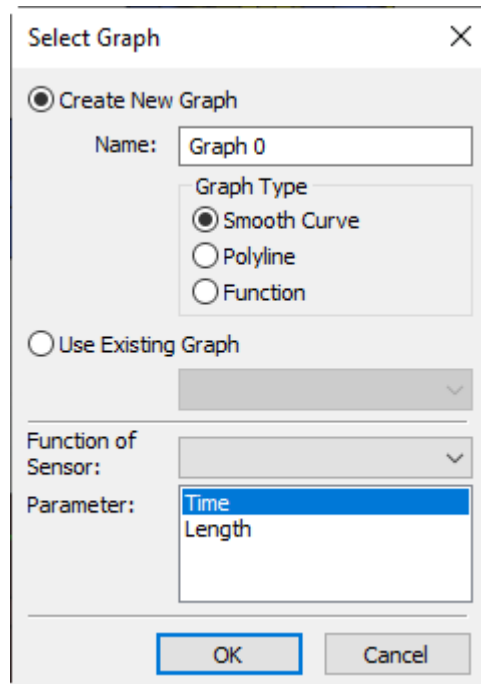
The **Initial length** parameter defines the distance between the spring ends, which corresponds to the absence of load (the free state). The distance is entered in model units. Upon any change in the distance, the spring will develop a respective resistance (compression or tension), according to the specified stiffness, proportional to the elongation.

To define the linear spring properties, set the switch to the **Linear Spring** position and define the stiffness coefficient and damping.

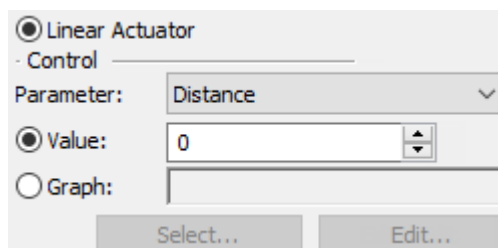
**Stiffness** defines the strain depended on the spring elongation and has the units of [Force/Length]. **Damping** depends on the velocity of tension/compression and has the units of [Force\*Seconds/Distance]. The force the units are specified using the parameter **Force Units**. Other units depend on the document settings (the command **ST:Set Document Parameters**).



To define a functional dependency between the force developed by the spring and the elongation, set the switch to the position **Bipolar Force**. The rest of the steps for defining the plot for a spring is identical to those described for the loads [Force](#). In the spring case, you can use an additional driven parameter **Length**, that allows using the current spring elongation as the base for reading the plot.



To define a linear actuator, set the switch to the **Linear Actuator** position. You can select **Distance**, **Speed** or **Acceleration** as the driving parameter. The driving parameter is defined by a constant **Value** or by **Graph**.



You can specify the compression or tension force limit for the linear actuator, depending on the direction of the actuator's operation. By default, the **Not Limited** setting is used. If necessary, you can specify a constant value or a plot to define the limiting force versus the time.

· Compression Force

Not Limited

Limited by Value:

Limited by Graph

Select... Edit...

· Stretching Force

Not Limited

Limited by Value:

Limited by Graph

Select... Edit...

There is a special section of the properties window to define visualization parameters to display special decorations that appear as springs, when working with the dynamic motion study. You can specify the spring **Diameter** and the number of **Coils**.

**Geometry**

Diameter:


Coils:

## Sensors

Sensors are special objects in a dynamic motion study that serve to read and transmit the calculation results to the result-rendering utilities. Sensors can be of several types, depending on the measured object:






- **Body.** This sensor serves to measure coordinates, linear and angular velocities and accelerations, as well as the equivalent force and torque (moment) acting on the body. The equivalent force or moment is the sum of all external loads applied to the body.
- **Point.** This sensor serves to measure coordinates, linear velocities and accelerations of any point of the mechanical system.
- **Joint.** This sensor serves to measure the reaction forces and moments in a joint, friction forces, velocities, acceleration and coordinates of a joint. You select one of the study's joints as the object to host the sensor.
- **Force Element.** This sensor is used to measure the actual values of force and/or torque applied to the body.
- **Distance.** This sensor can read the distance between two points connected to the arbitrary bodies of the study, the velocity, with which a distance changes, and acceleration. You select two model points as the base objects to create this sensor. Additionally, for each of the points you select a body in the study, with which it moves together, as the mechanical system moves.
- **Two Solids.** This sensor serves to measure the reaction and friction forces at the contact of two bodies. You select two bodies of the study as the base of the sensor creation.

Sensors can be created only for the active study of the dynamic motion study, using the command:

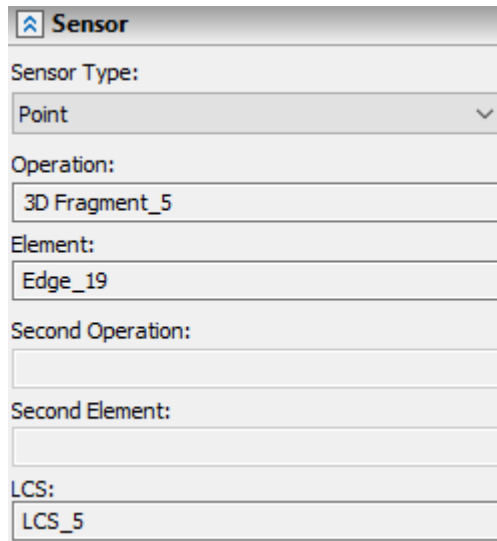
Icon	Ribbon
	Analysis > Conditions > Sensor
Keyboard	Textual Menu
<3MD>	Analysis > Sensor

In the properties window you can specify the type of the sensor being created.

After calling the sensor creation command, various options appear in the automenu to help selecting objects. The set of options depends on the type of the sensor being created:

-  <B> Select Solid
-  <G> Select Second Solid
-  <V> Select Point
-  <F> Select Second Point
-  <C> Cancel Selection

In the 3D window, the sensors are displayed as spherical graphical objects. In the properties window, there is a collection of fields (**Operation**, **Element**, **Second Operation**, **Second Element**), in which the names of the selected object appear. Deleting a selected object from any field of the properties window during editing is carried out using the <Del>.



By default, all measurements are made in the global coordinate system of the model. If necessary, for sensors (with the exception of the types of sensors **Force Element** and **Two Solids**), it is allowed to specify the LCS in which it is necessary to make measurements using the automenu option:



<L> Select LCS

The selected LCS is displayed in the corresponding field of the properties window. The selection is canceled using the following automenu option:



<D> Cancel LCS Selection

In the **Value** block, you can customize the display of auxiliary graphics objects, that will the outputting certain sensor readings during the calculation process (see the table below).

Parameter	Drawing Style	Sensor
Trajectory	3D Curve	Point, Body, Joint <sup>a</sup>
Reaction Force	Vector	Joint <sup>a</sup> , Two solids <sup>b</sup>
Reaction Torque	Vector	Joint <sup>a</sup> , Two solids <sup>b</sup>
Friction Force	Vector	Joint <sup>a</sup> , Two solids <sup>b</sup>
Friction Torque	Vector	Joint <sup>a</sup> , Two solids <sup>b</sup>
Linear Velocity	Vector	Body
Angular Velocity	Vector	Body
Linear Acceleration	Vector	Body
Angular Acceleration	Vector	Body
Active Force	Vector	Body
Active Torque	Vector	Body
Velocity	Vector	Point



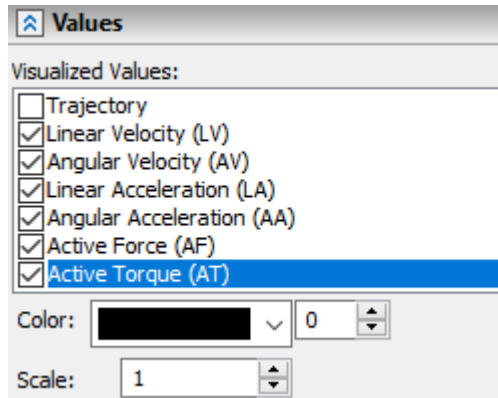
Parameter	Drawing Style	Sensor
Acceleration	Vector	Point

Explanations

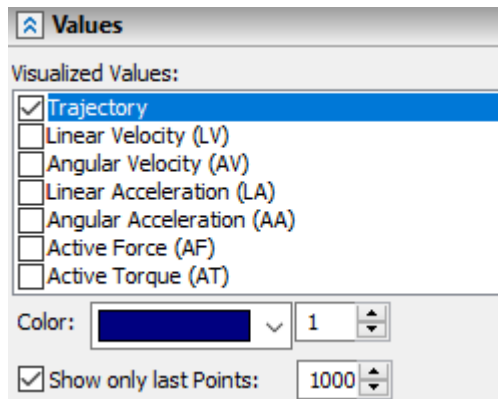
<sup>a</sup> the coordinates are computed, and then the trajectory is displayed of the point that is the center of the contact area between the two bodies;

<sup>b</sup> the coordinates of the joint center are computed.

You can define the scale and color settings for the graphics objects that appear as a vector arrow.




For objects in the form of a spatial curve, it is allowed to limit the length of the curve in addition to color, by setting a fixed number of the last calculated points using the **Show only last Points** parameter.

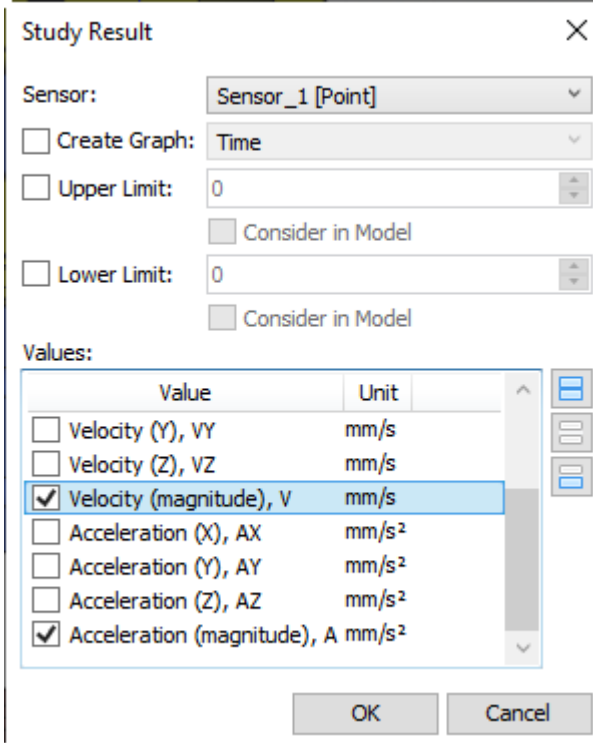


## Results

Results - special study objects intended for reading sensor readings during dynamic calculation. To create a result, use the command:

Icon	Ribbon
	Analysis > Calculation > Result
Keyboard	Textual Menu
<3MZ>	Analysis > Result

Also, this command can be called from the context menu when selecting a sensor in the 3D model tree or in the Studies window. This command is available if there is at least one sensor in the study. After calling the command, the result creation dialog appears.






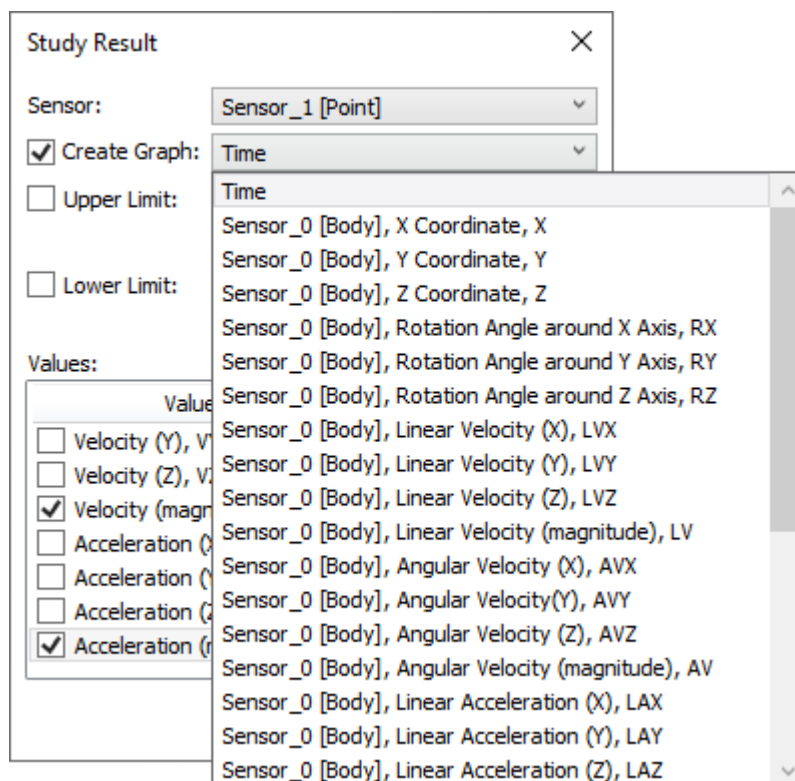
The result is created based on the values of a specific sensor, the set of which depends on its type. The set of values is shown in the table below.

Value	Unit of measurement	Sensor
Coordinate	mm	Body, Point, Joint, Two solids
Rotation Angle around axis	rad	Body
Vector	mm	Distance
Distance	mm	Distance
Velocity	mm/s	Point, Distance
Linear Velocity	mm/s	Body

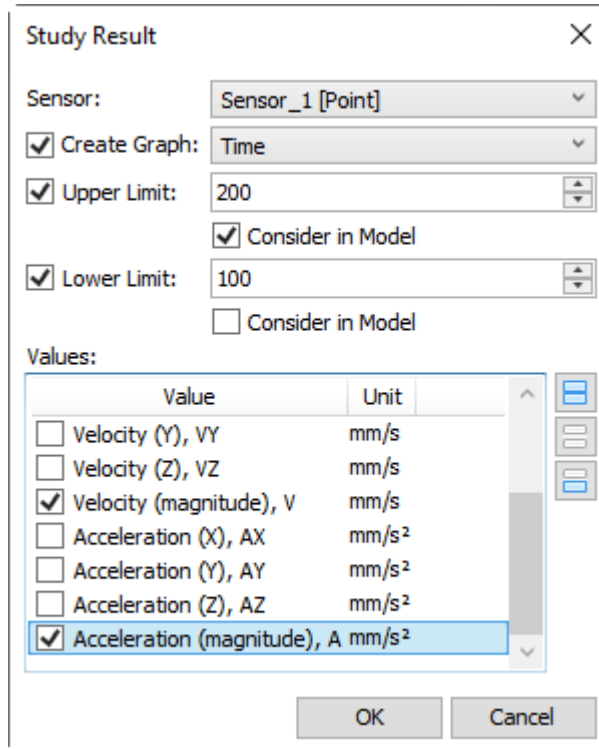
Value	Unit of measurement	Sensor
Angular Velocity	rad/s	Body
Acceleration	mm/s <sup>2</sup>	Point, Distance
Linear Acceleration	mm/s <sup>2</sup>	Body
Angular Acceleration	rad/s <sup>2</sup>	Body
Action Force	N	Body, Force
Action Torque	N*mm	Body, Force
Reaction Force	N	Joint, Two solids
Reaction Torque	N*mm	Joint, Two solids
Friction Force	N	Joint, Two solids
Friction Torque	N*mm	Joint, Two solids
Kinetic Energy	J	Body

Additionally, it is possible to select X, Y, Z - components in the form of a separate graph for each value.

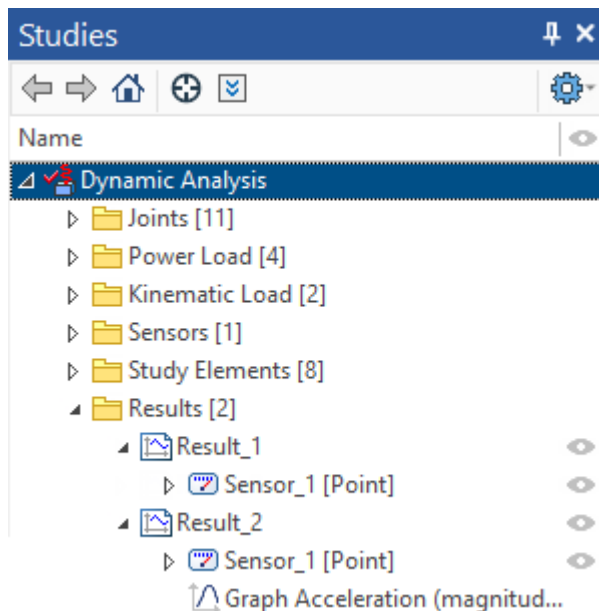
In the **Sensor** drop-down list, a sensor is selected from among those existing in the study. In the **Values** field, you need to set labels opposite those indicators on the basis of which you want to create results. Comprehensive values management is achieved through the use of the **[Select All]** , **[Unselect All]** , **[Invert Selection]**  buttons. The **Create Graph** parameter allows you to generate a result in the form of a graphical dependence of the current value on time or the value of one of the existing sensors.



Also, for each value, it is possible to determine the range of acceptable values by setting limitation. Thus, during the dynamic calculation, the system informs the user about the achievement of the set limits by displaying the "!" Sign in the **Current Values** window. The parameters **Consider in Model** are intended to stop the movement of the mechanism when the set limit value is reached.




After selecting the necessary values and closing the dialog, new objects of the **Result** type are created in the system, which record data in the process of calculating the study. The results formed in the form of graphs can be viewed outside the study of dynamic calculation by conventional means for working with graphs.



## Running Calculations

The calculation is performed using the following command:

Icon	Ribbon
	Analysis > Calculation > Study
Keyboard	Textual Menu
<3MY>	Analysis > Study

Also, this command can be called from the context menu when selecting the study in the 3D model tree or in the **Study** window.

After calling the command, the system goes into the mode of waiting for the result of the calculations or rendering the calculated results. The parameters to control the process and view the results appear in the properties window.

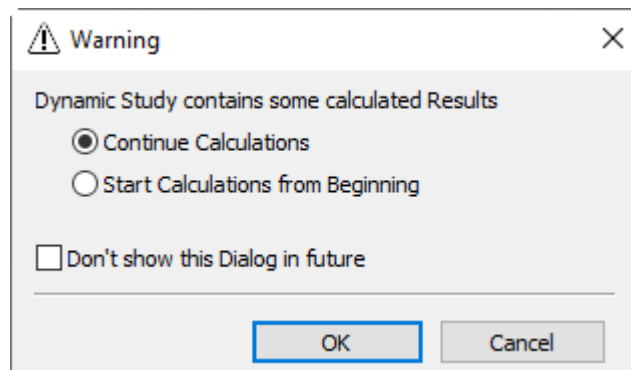
This command can work in two modes – in the mode of calculating, or in the mode of viewing the ready results. Upon the first launch of the **Study** command, the system doesn't have any results yet. You can judge that by the inaccessible buttons to view the results. At this point, you can only start new calculations.

You control the calculations process via the properties window, in which there are necessary control elements.

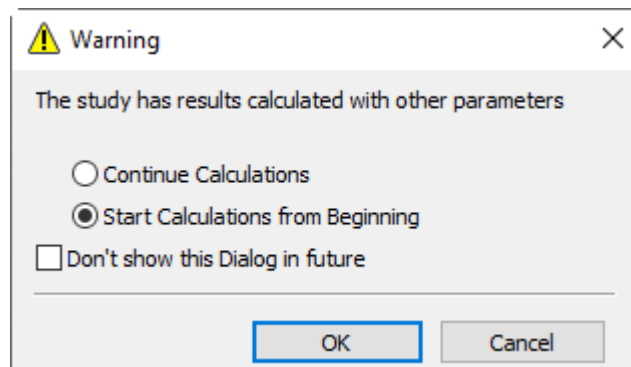


**Record**. Starts the calculations. Depending on the current parameters of the study, if there are earlier calculation results, the system offers the following options:

- **Continue Calculations**. Relevant if there are previously calculated results for the current parameters of the study;



- **Start Calculations from Beginning**. It is relevant for cases when the current parameters of the study differ from the parameters of the previously calculated results.





- **Play.** Starts the playback of the calculation results. The button is accessible if there are saved results of calculations.



- **Pause.** Pause playback of the calculation result. The button is available after starting playback of the calculation result.




- **Stop.** Stops the calculation process or the results playback.




- **Create Video File.** Call the video recording parameters dialog:

The **Parameters** group:

- **Starting Frame.** Specifies the frame from which the record begins.
- **Ending Frame.** Specifies the frame after which the record will stop.
- **Step.** Step of the video recording.

The button  allows to choose the optimal value of the frame recording step without changing the video duration. I.e. the step is increased and several frames will not be used, but the length of the video will not change.

- **Time Scale.** Specifies the scale for the video recording time. Used in the case when a short video (for example, a duration of 1 second) includes many frames (for example, 300 frames).

The button  allows you to select the optimal video duration without changing the number of frames. I.e. the number of frames will remain the same, but the duration of the video will change. The video duration time is specified in the motion analysis study parameters.

The parameters of the **Photorealistic View Generation** group allow you to create photorealism using one of the available mechanisms:

- **OptiX.** The mechanism uses NVidia OptiX. It uses video card for calculations.
- **Embree.** The mechanism uses Embree technology – a ray-tracing kernel, developed by Intel. It uses CPU for calculations.
- **POV-Ray.** The mechanism uses POV-ray technology, which uses a ray-tracing method.

The **Image quality** parameter is available for the photorealism by means of Embree and OptiX, which allows setting the number of iterations of rendering for each frame. You can select a standard option from the list or select **User-defined** and set the value manually in the **Number of Ray Tracing Iterations** field.

Creating a photorealistic view increases the recording time of the AVI file, but it can significantly improve its quality.

The **View Point** group determines the position and direction of the recording camera:

- **Use Current View Point.** The view point is defined by the system camera present in any 3D window by default.
- **Use Camera.** Allows to specify a user-defined camera.

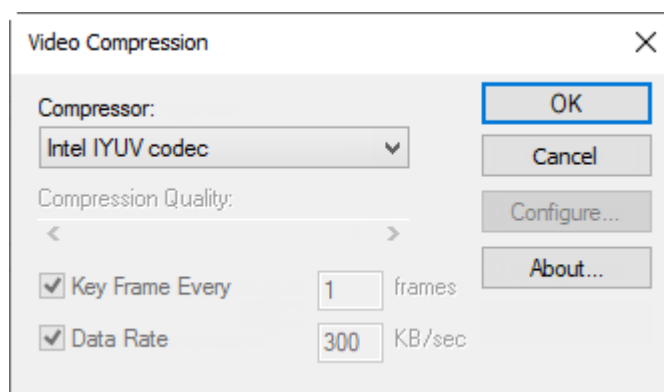
The **Output File** group:

- **File Name.** Allows you to set the file name, its storage directory and select the video file format - WMV or AVI.
- **Video quality.** Specifies image quality for WMV files.

The graphic button [**Compression...**] allows for AVI files. It invokes the dialog box for customizing multimedia file compression parameters:

- **Compressor.** Selects a compressing application.
- **Compression Quality.** Defines the quality grade for multimedia file compression.
- **Key Frame Every.** Defines the number of frames between the key frames.
- **Data Rate.** Defines the data transfer rate (kilobytes per second).

The graphic button [**Configure...**] brings up the dialog box for entering the required settings of the selected compressing application.



The **Size** group:

- **Use Current View Size.** With the flag turned on, the whole contents of the current 3D view window are recorded. With the flag off, the user can define one's own width and height parameters of the image being saved.

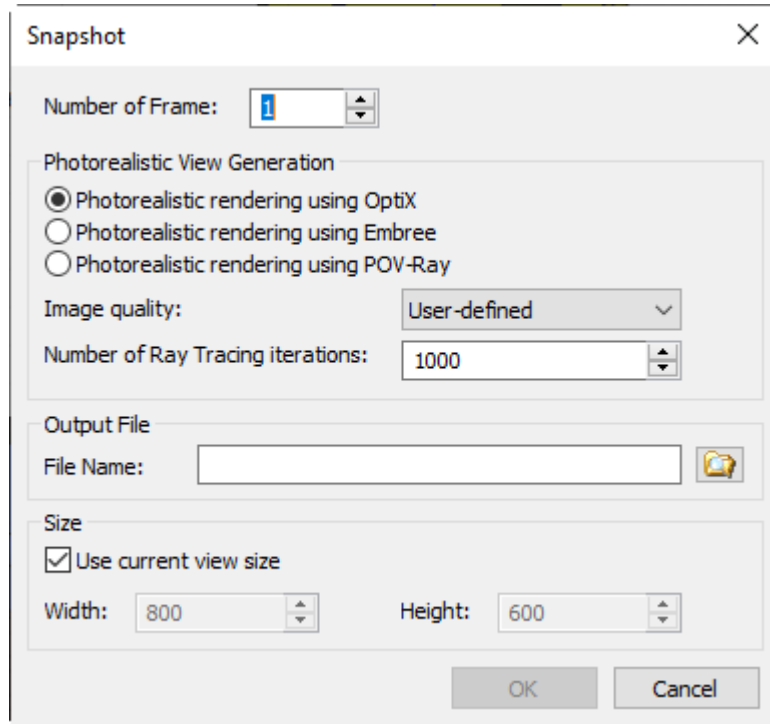
The **Reverse** parameter allows you to record video in the opposite direction.

The **Logotype** group is intended for overlaying a logo on a video with an indication of its position.



**Take Snapshot.** The option allows to save photorealistic snapshot of the current or specified frame in PNG, JPG or BMP.

A special dialog is used to customize the snapshot.



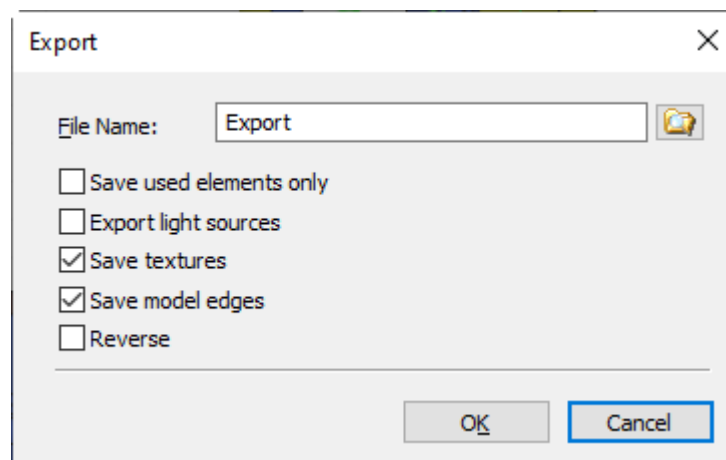
The **Number of Frame** parameter is intended for setting the sequence number of the frame on the basis of which the picture will be taken.

Other parameters are similar to the [video recording settings](#) dialog.



**Export Results.** The button contains a drop-down list with the following options:

- **Export Animation.** The option allows you to export the animation obtained during the analysis of motion to grid formats: WRL, X3D, 3D PDF, U3D, POV. After the call, the following dialog appears:



**File Name.** Allows you to specify the file name, its storage directory, as well as select a grid format.





**Save used elements only.** Allows you to upload to the file only the elements involved in the study.

**Export light sources.** Allows you to upload light sources to a file

**Save textures.** The textures of the objects will be uploaded to the file. Otherwise, textures will not be added to the file, and body material will be displayed instead.

**Save model edges.** Allows to unload model edges into a file. Otherwise, an edgeless image is exported. This parameter is available only for WRL and X3D formats.

**Reverse.** Animations will be exported in the opposite direction.

- **Unload Geometry to File** . The option allows you to upload the model in its current position to a separate file. After calling this option, a standard export dialog appears with a choice of the save directory.
- **Unload Geometry to File and Create Analysis Study** . The option allows you to upload the model in its current position to a separate file with the subsequent creation of the Analysis study. After calling this option, a standard export dialog appears with a choice of the save directory.



- **Trim Left.**





- **Trim Right.**

These buttons clip the record of the saved result at the current playback instant. The options are used to remove unnecessary fragments of the recording.



- **Delete Record.** Deletes saved results - before performing new calculations.

The saved calculations render the model behavior in the state in which it was back at the moment of calculating the results. If changes are introduced in the model that need to be accounted for in calculation (for example, it could be the changes in the properties/number of joints, load parameters, study conditions, etc.), then you shall delete the saved results using the button  and then start a new calculation with the button .



- **To Start.** Transfer of calculation animation to the beginning of playback.



- **Rewind.** A button that, when held, rewinds the animation to the start of playback.



- **Playback.** Start playback of the calculation results in the opposite direction from the current frame.



- **Previous Frame.** Go to the previous frame.



- **Next Frame.** Go to the next frame.

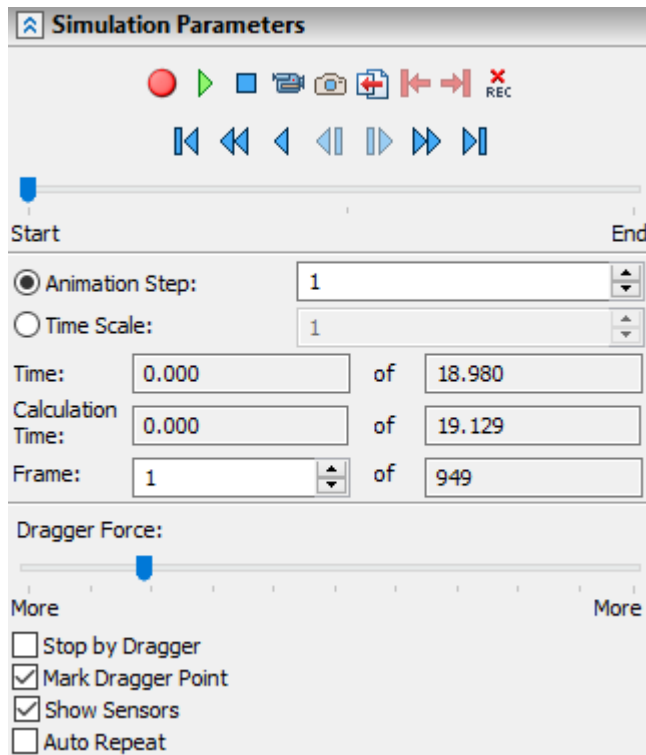





- **Fast Forward.** A button that, when held, rewinds the animation to the end of playback.



- **To End.** Transfer calculation animation to the end of playback.

Below the panes that control the calculation, there is a special slider, that allows setting the time counter to any instant. A scale with time intervals appears on the slider bar as the total time increases. By using the **Animation Step** parameter, you can set the speed of the result playback. You can use only the integer numbers here. In fact, this parameter sets the step to pick the frames from the array of results. It can be conveniently used when creating animated clips in the real time, if the calculations were run with a large number of frames per second in order to obtain quality results. An alternative way to control the playback speed is provided by the parameter **Time Scale**. Displayed below are the time parameter for each frame, total time counter and the frame counter.



When the calculations run, the user can grab any movable part with the cursor and act on it with a relative force, which depends on the distance to the grabbing point and on the **Dragger Force** parameter setting. To grab, you make a pause  with . Next, while holding down , you need to "pull" the part in the desired direction.


**Dragger Force** – this is a special control element that sets a relative dragger force to act on the model while running calculations.

**Stop by Dragger.** This option automatically pauses the calculations process each time when the dragger is engaged. It is convenient to use whenever the dynamic calculation is conducted with additional loads imposed by the dragger, and the instants must be excluded when the model is off the dragger.

**Mark Dragger Point.** This option enables the display of a crosshair mark which denotes the application point of the dragger force.

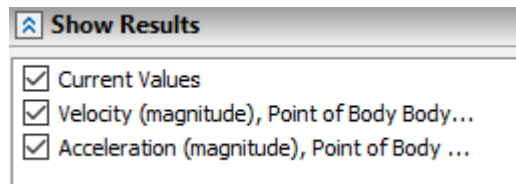
**Show Sensors.** Designed to display sensors in a 3D scene.

**Auto Repeat.** This option turns on an automatic replay loop.

When running calculations, it may often be necessary to set different intervals for the time counter, and different modeling parameters. In the automenu there is the option  to call the dialog for managing some general study parameters. This dialog duplicates the respective portions of the [general study settings](#) dialog and has two tabs, **[Basic]** and **[Simulation]**. It helps quickly access the necessary study settings directly in the calculation running command. For example, the group of parameters **Steps per Frame** control is the number of calculation steps. The calculation speed and accuracy depend on these settings. Sometimes, when running preliminary trial calculations, one may need to set a higher speed at the expense of the accuracy.

## Utilities of Displaying Calculation Results

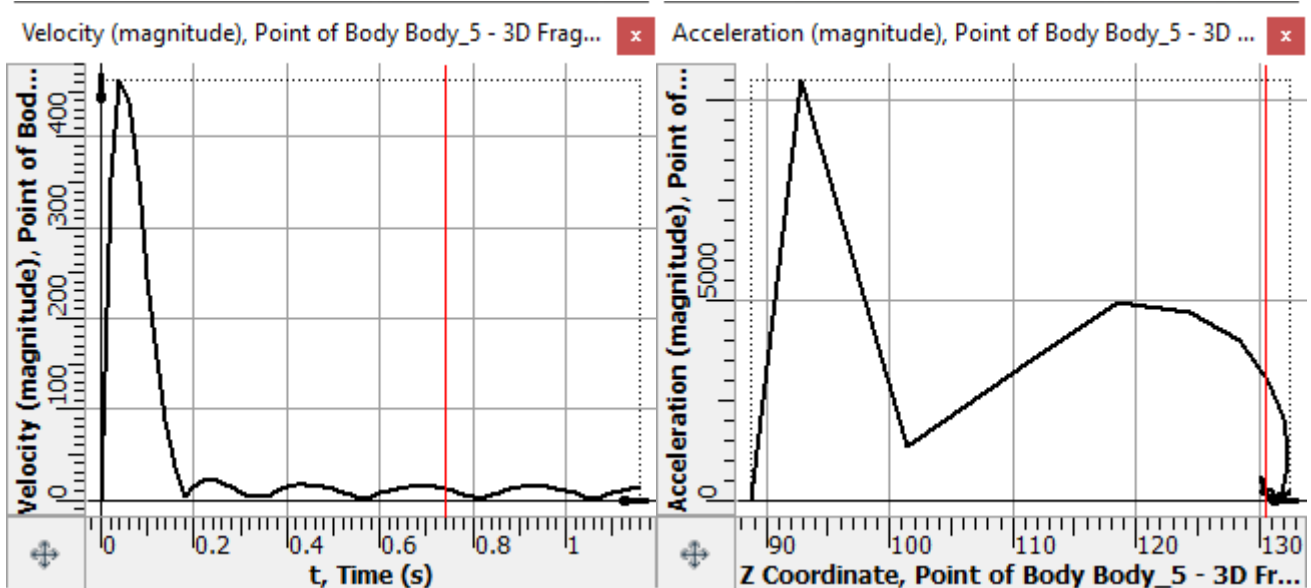
In the process of dynamic calculation, it is allowed to display all pre-generated study results. For this, in the calculation parameters window, the **Show Results** block is provided, containing the corresponding list.



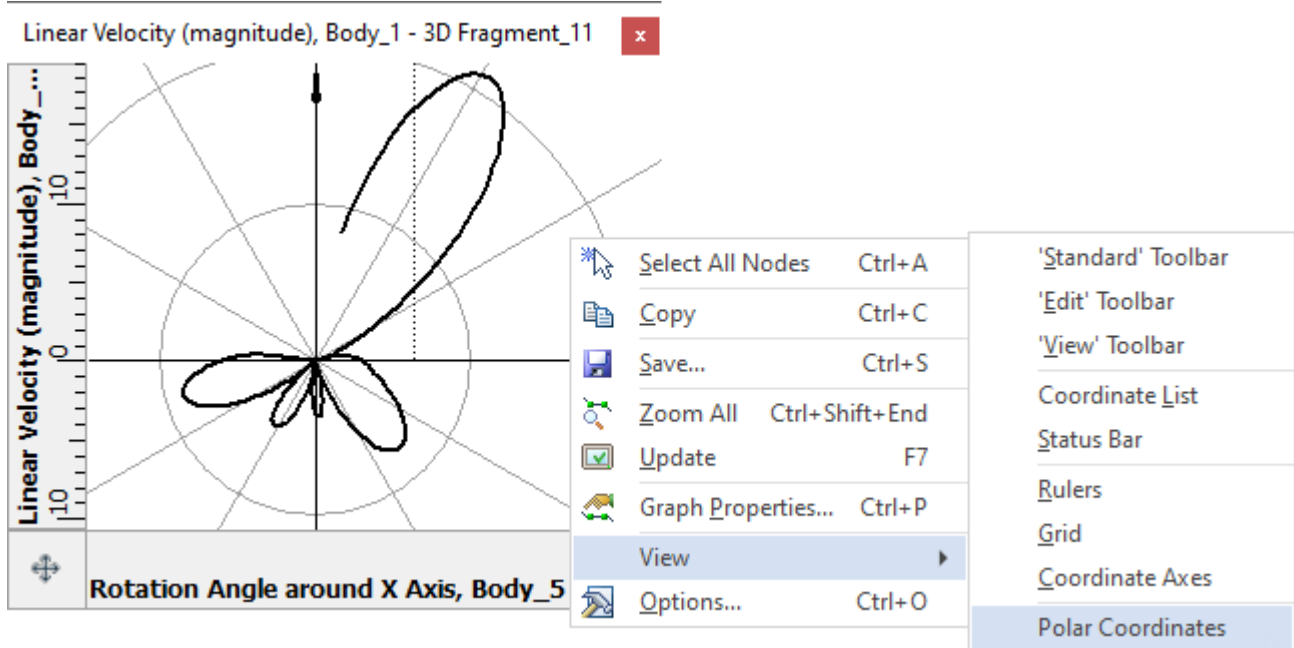
The **Current Values** window is intended to display the current values of the initial indicators of the sensors, as well as the extremum points achieved during the calculation. If a [limitation](#) is set for any indicator, then when the set value is reached, the "!" Sign is displayed in the corresponding columns.

Current Values <span style="float: right;">x</span>				
Name	Value	Minimum	Maximum	Unit
Z Coordinate, Sensor_1, Point of B...	247.026781	149.064624	284.159370	mm
Velocity (magnitude), Sensor_1, P...	193.354962	0.000000	618.256287	mm/s
Acceleration (magnitude), Sensor...	4397.102808	0.000000	10274.652726	mm/s <sup>2</sup>

The remaining windows are designed to display the indicators of sensors for which the result is formed in a graph. Each dynamic window with a chart has the same functions for managing schedules as the regular chart editor. The image of the graph in the window is automatically scaled to constantly display the active area.



For sensors with axis rotation angles, the calculation results are displayed in polar coordinates by default. Switching to the display of the graph in cartesian coordinates is carried out through the context menu.



When playing back the results, graphics are also displayed. To display the current moment of time in each chart, a special label **Argument Position Marker** is dynamically moved.

The user can adjust the size of the floating windows and the scale of the graphs at his discretion. If necessary, each chart can be examined in detail in a special chart editor.

More detailed information on creating and editing plots can be found in the T-FLEX CAD help, chapter "Service 3D Tools and Entities", section "Graphs".

For sensors of the **Point**, **Body** and **Joint** type, it is possible to create a 3D path based on the trajectory of their movement. Thus, the problem of obtaining the trajectory of movement of individual links of the mechanism is solved. Trajectory creation is carried out using the corresponding context menu option of the selected sensor.

After the dynamic calculation, it is allowed to export the sensor readings to a text document for use in other systems. Export is carried out using the corresponding context menu option of the selected result in the task window or in the model tree.

## Limitations of the Express Dynamics Module

There is a build-in module of Express Dynamics Analysis in T-FLEX CAD. It is a free non-commercial version that shows some functionality of full Dynamics module. Express Dynamics Analysis module has some restrictions in forces and dynamic motion results output:

1. Utilities of Displaying Calculation Results is restricted - the results are not displayed in the form of graphs.
2. The capability of animation clip generation is restricted.

Contact us to request information about T-FLEX software, our Academic Program, or if you have ideas on cooperating with Top Systems

**[www.tflex.com/mail](http://www.tflex.com/mail)**

Contact Us



[www.tflex.com](http://www.tflex.com)

(7-499) 973-20-34  
(7-499) 973-20-35

