Tutorials

Tutorial 1 - Getting Started

Objective

In this tutorial we guide you step by step into the work with SimulationX. Using simple example models you acquire the necessary modeling skills for SimulationX.
Based on a two-mass oscillator we explain the structure of a model. You can easily repeat this on your own computer.
Many actions can be performed in several ways. In this introduction only one is usually demonstrated and used.

- *Overview of the Graphical User Interface*
- *Working with existing example models*
- *Creating your own model*
- *Running a simula-tion*
- *Observation of results*

Graphical User Interface (GUI)

The working area of SimulationX can be subdivided into different windows and areas (Figure 1).

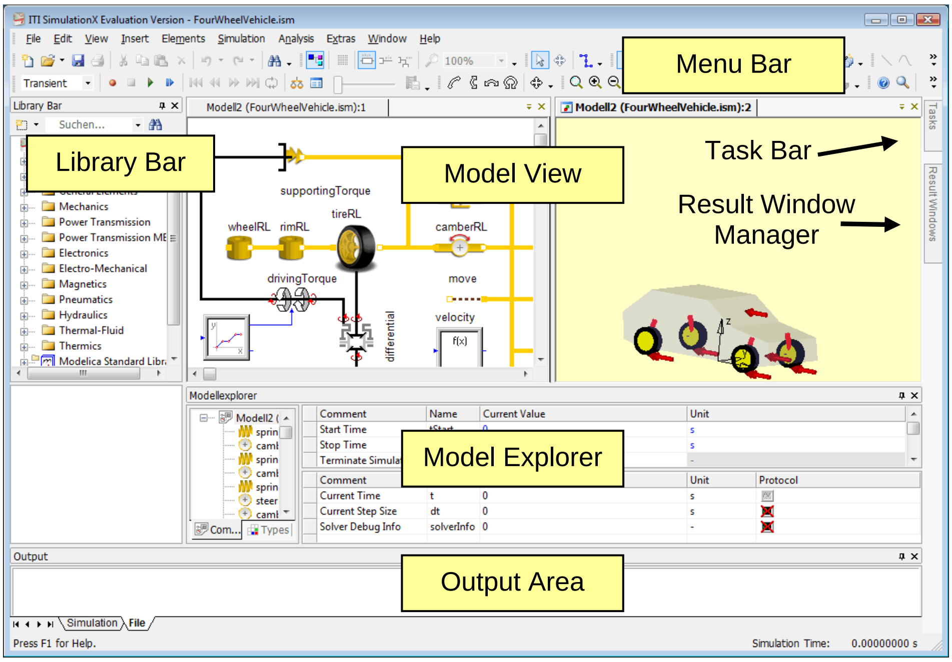
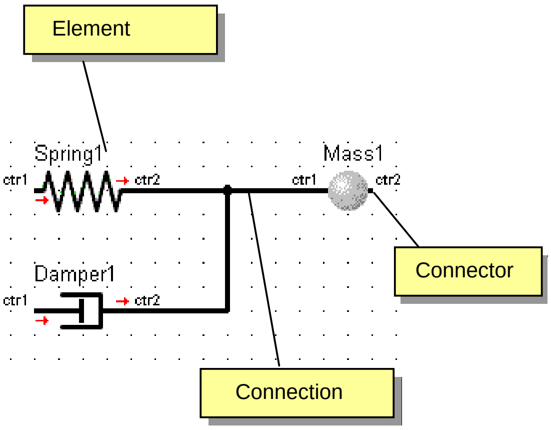


Figure 1: Graphical User Interface

The library bar offers access to the installed element types. For clarity, the element types are subdivided into libraries (groups). In the tree view, element types and libraries are shown according to their hierarchy. Element types in the library view are represented by symbols, which are administered in folders.

Within the "Favorites" library you can create your own groups. As in Windows, links are created by Drag&Drop of element types and libraries. Inside the "Favorites" library you can copy, move, and delete sublibraries and links.

The model view serves for the graphical representation of the structure and the modification of the simulation model. Elements and connections are the components of a simulation model. Elements have connectors that can be linked together via a connection. Connections can be branched arbitrarily, i.e., you can link more than two connectors to the same connection.



There are different types of connectors, such as mechanical (linear and rotary), hydraulic, and electrical connectors, as well as signal inputs and outputs. Only connectors of the same type can be connected to each other. Each connector possesses an unambiguous name with respect to the corresponding element. These names can be made visible via the menu "View/Pin Labels".

Figure 2: Components

The model explorer offers access to the properties of the components of a simulation model. The element and class hierarchy of the model is represented in a tree view. The parameters and results of a selected component are shown in a table and can be modified there.

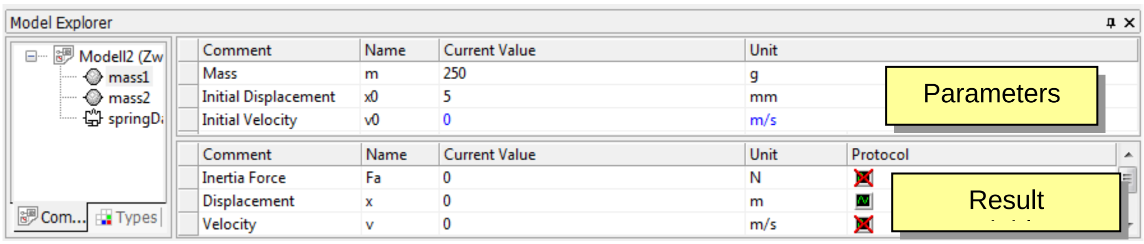


Figure 3: Model explorer

Messages, warnings, and errors are recorded in the output area. These messages are assigned to different categories (e.g. simulation, file). The content of the output block can be saved, exported as text, and printed.

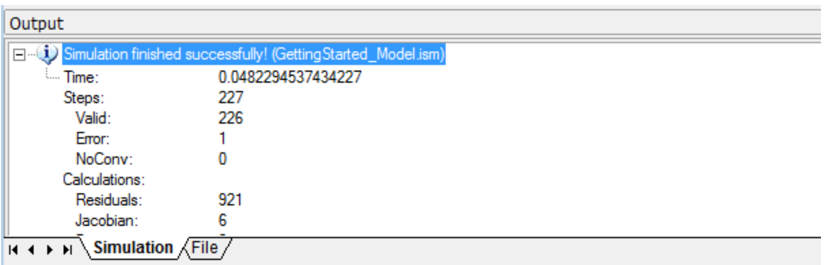


Figure 4: Output area

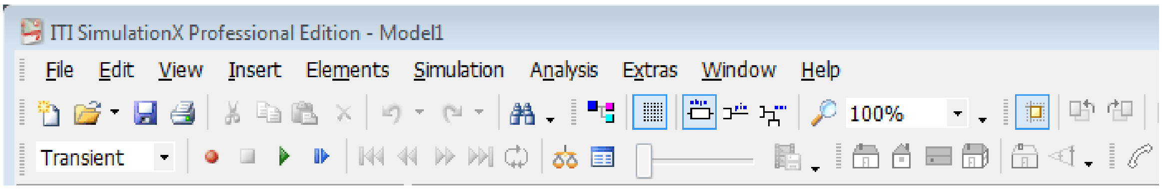
Working with Sample Models

With the installation of SimulationX you received also a collection of example models. You can open these models, change parameters and perform simulations. You will find the example models in the directory ...\SimulationX 3.0\Samples\... There are different sub-directories for the different libraries, such as, e. g.,

- *...\SimulationX 3.0\Samples\Mechanics\... for mechanics (general)*
- *...\SimulationX 3.0\Samples\Pneumatics\... for pneumatics*
- *...\SimulationX 3.0\Samples\Hydraulics\... for hydraulics*

etc.

a.) Opening a Model File



To open a model file, click on the button or select "File/Open...". After opening, the model structure will be displayed in the model view and the model explorer, and some result windows with already calculated result curves appear.

b.) Changing Parameters

If the model has already been simulated, you should first reset the simulation by clicking on the button or selecting "Simulation/Reset". Now you can double-click on any symbol of an element in order to open the parameters dialog. For each parameter in SimulationX, you can enter either constant numerical values, mathematical expressions or logical conditions. For detailed information about the elements (parameters, result variables, assumptions and calculation) you can press the "Help" button ( ). The online help system will appear, which provides the required information. For entering numbers into the parameter fields, first select the desired unit and then enter the numerical value. The parameter value will be automatically converted if you change the unit afterwards. You can prevent the conversion by pressing the Shift key during the unit selection.

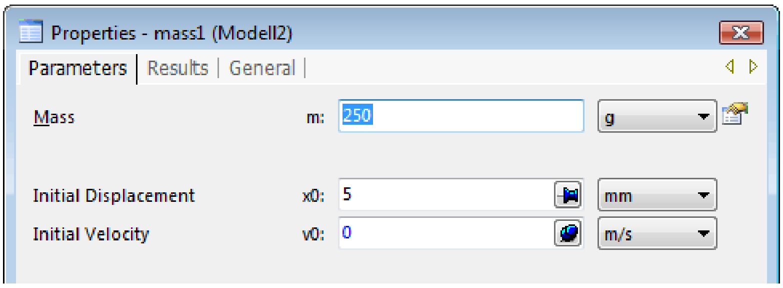


Figure 5: Parameters dialog

In order to save simulation results for later displaying, you should activate the protocol attribute (→) for the desired result quantities.

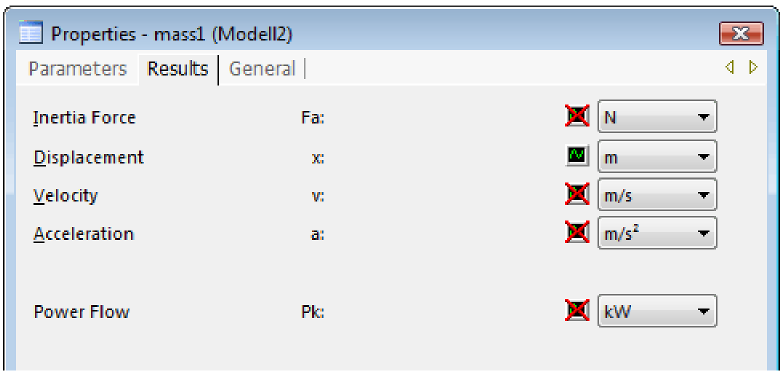


Figure 6: Result variables dialog

On the page "General" in the property window you can change the name of the element, assign a comment, and adjust the position of the element label in the model view (Figure 7).

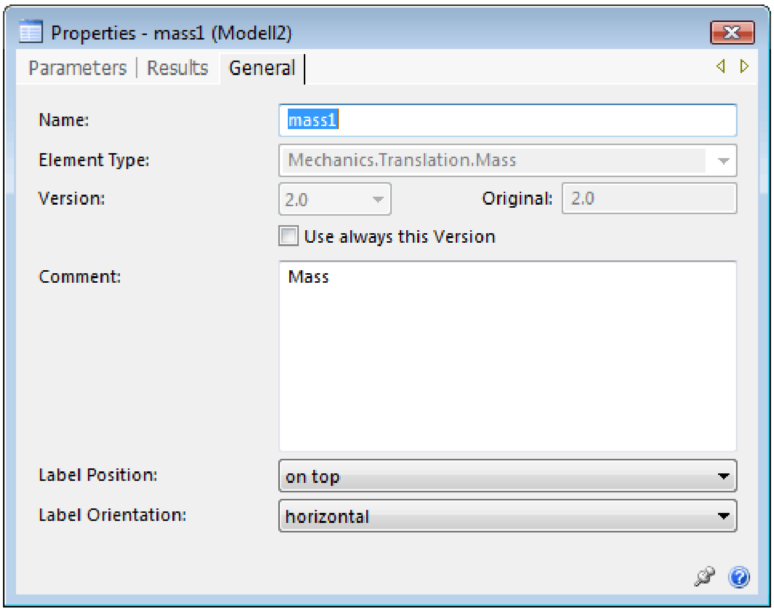


Figure 7: Property window

Close the parameter dialog with the close button ( ) or by clicking outside the dialog window.

c.) Running a Simulation

Start the simulation by clicking on the button or by selecting "Simulation/Start". The simulation will now run up to the specified stop time. You can observe the current simulation time in the lower right corner (Simulation Time: 1.00000000 s).

To change the preset value of the stop time, open the simulation control panel by selecting "Simulation/Transient Settings…". Now you can edit the simulation parameters (e.g. "tStop").

d.) Opening a Result Window

If you have activated the protocol attribute (→) for result quantities before running the simulation, you can now plot these result as y = f(t) diagrams. You can open a result window by clicking with
right mouse button on an element and selecting the desired result quantity from the pop-up menu.

If no result quantities are available for an element, no protocol attributes have been activated ( ).

The work with result windows is described in the sequel.

Creating your own model

Now we develop the initially mentioned "Two-Mass Oscillator" model. The individual steps are explained, further details are found in the subsequent chapters.

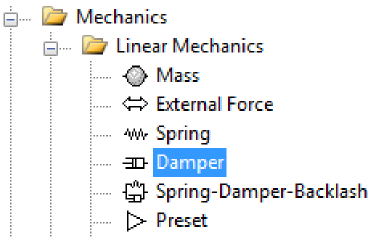
When creating your own model, you should always start with a new file (button or menu "File/New"). Then proceed as follows:

a.) Selecting elements

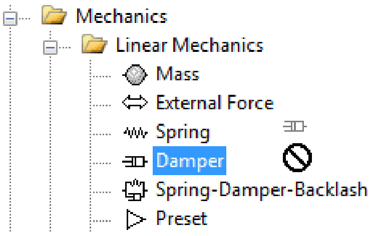
We now assemble our first simple model - the "two-mass oscillator".

To place a new element using Drag&Drop into the model view, proceed as follows:

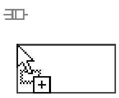
1. Find the corresponding element type in the tree view of the library bar.



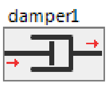
2. Click with the left mouse button on the entry in the tree and keep the mouse button pressed.



3. With the left button pressed move the mouse pointer to the position in the model view where the new element is to be inserted.



4. When releasing the mouse button the element is created at the desired position in the model.



You can simplify the positioning of elements by activating the option "Snap to grid" (Menu "Elements").

To place several elements of one type into the model view, you can repeat the procedure before

Working example:
Select two masses (mass1, mass2) and a spring damper element (springDamper1) from the library "Linear mechanics" and place them in the model view.

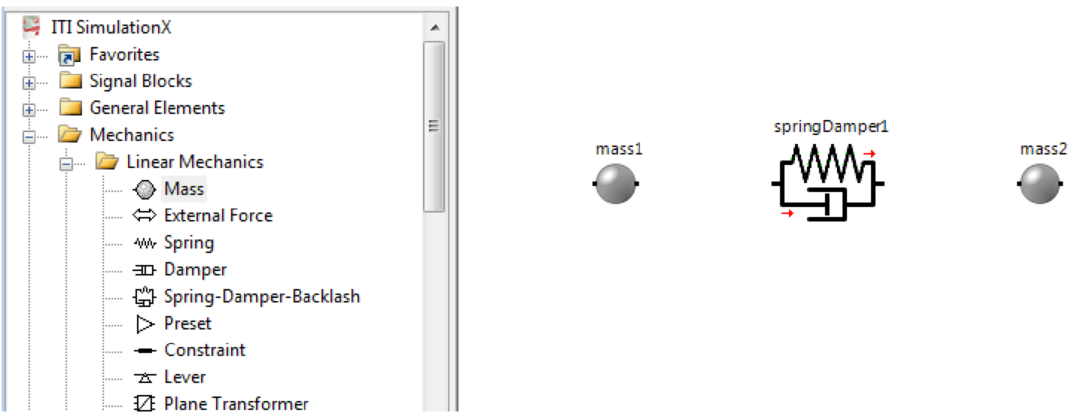


Figure 8: Library bar and the model

The elements of the model can still be manipulated, i.e., they can be shifted, rotated and mirrored.

This element is selected by a mouse click. Now it can be shifted with pressed left mouse button to a new location in the model view. By means of the tools  you can rotate the element turn and with  you can flip it horizontally or vertically. Thus you can arrange the elements in the desired position and orientation for connecting them.

b.) Connecting Elements

The next step is to connect the elements in order to obtain the desired model structure.
To create a connection between two connectors you can do the following:

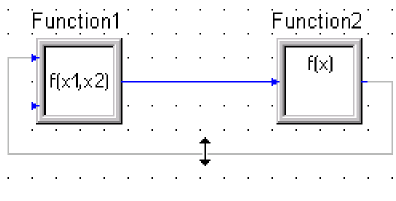
1. Place the mouse pointer over the connector, from which you want to draw the new connection. The changed mouse pointer as well as the changed color of the connector indicates that you have met the connector.

2. Press the left mouse button and move the mouse pointer to the target connector of the new connection, while keeping the mouse button pressed.

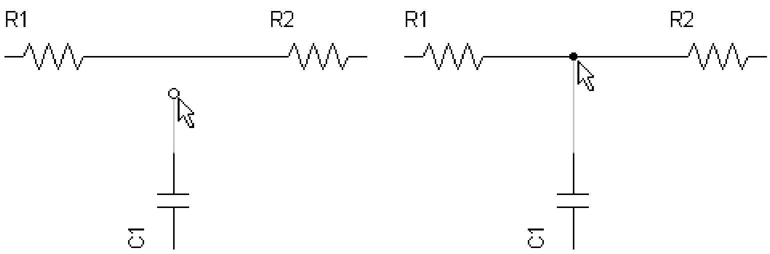
3. When you release the mouse button the connection is created.

You can abort the creation of a new connection at any time by releasing the mouse button over an empty region in the model view or by using the escape key.

The routing of a connection will be determined automatically, but a change of the path is possible at any time. To do these move the mouse over a connection, while pressing the Alt-key. The mouse pointer shows you in which direction you can move the selected segment of the connection line.



To improve the clarity of the model, you can also branch connections. So, you can create connec-tions between free connectors and existing connections in both directions.



Working example:
Connect the two masses with the spring damper element according to the following structure.

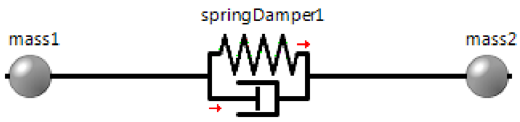
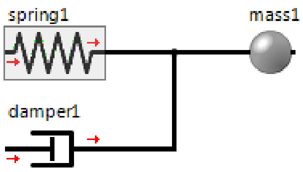


Figure 9: Model structure

Please note that you can connect only elements from the same physical domain (e.g., a mechanical spring will not connect to a hydraulic throttle). SimulationX prevents automatically the creation of such a connection.

c.) How to Select an Element

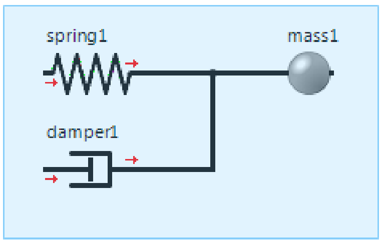
An element is selected by a mouse click. Once selected it will be optically accentuated by a frame. Single elements can be selected also by selection of the corresponding entry in the model explorer.



How to Select Several Elements

To select several elements there are two possibilities:

1. Draw a frame around the respective elements.



2. An element can be added to the current selection by pressing the Shift key and clicking with the mouse on the element. The removal of an element from the selection is performed in the same way.

d.) Entering Parameters

In order to be able to work with our model, we first have to enter the desired element parameters.

Select the component (element or connection) which you want to edit, either in the model view or in the tree view of the model explorer. A component, selected in the model view, is selected also in the tree view of the model explorer and vice versa.
For the selected component the related parameters and result variables are shown in two tables. For editing an entry click on the desired table field. Now you can edit the contents of the field or choose an entry from the corresponding selection list.
As long as the field is edited, the corresponding parameter is not updated. The transfer of the new value to the parameter takes place only after completion and validation of the input. For the completion of the input there are the following possibilities:

- *Change to another row using the cursor keys ↑ and ↓ or a click on the new field*
- *Press the Return-key*
- *Change the focus to another window, e.g. by a click to the model view.*

Modifications can be undone using the Undo button or the menu "Edit/Undo".

To toggle the protocol attribute of a result variable click on the corresponding symbol in the protocol column (→).

Comment	Name	Current Value	Unit	Protocol
Pressure	p	0	bar	
Temperature	T	-273.15	°C	

The current value of a result variable cannot be changed. However, the unit of measurement, in which the variable should be shown, can be changed.

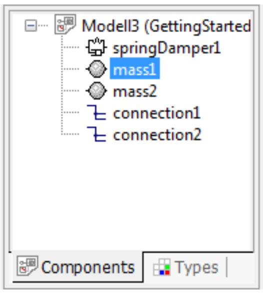
Working example:
Enter the following parameters:

mass1	*Mass*	*m*	*250*	*g*
	Initial Displacement	*xo*	*5*	*mm*
mass2	*Mass*	*m*	*2*	*kg*
springDamper1	*Stiffness of Contact*	*k*	*1*	*N/mm*
	Damping of Contact	*b*	*2*	*Ns/m*

Table 1: Parameters of the elements

You can do it by using the Property dialog of an element (double click on it) or by using the model explorer. With click on an element appear its parameters in the model explorer.

Now in the top section of the model explorer (s. Figure 10) the parameters are accessible for changing and in the lower part the result variables can be accessed.



Comment	Name	Current Value	Unit	Quantity	Current Value in SI-Unit
Mass	m	250	g	Mass	0.25
Initial Displacement	x0	5	mm	Displacement	0.005
Initial Velocity	v0	0	m/s	Velocity	0

Comment	Name	Current Value	Unit	Protocol
Inertia Force	Fa	0	N	
Displacement	x	0	mm	
Velocity	v	0	m/s	
Acceleration	a	0	m/s²	
Power Flow	Pk	0	kW	

Figure 10: Parameters of mass1 in the model explorer

Note that as decimal separators always uses one point (no comma!).

Further you activate the protocol attributes for the result variables which can be logged, so that they can plot these during or after the simulation.

Activate protocol attributes for the following result variables:

- *mass1* *Displacement (x)*
- *mass2* *Displacement (x)*
- *springDamper1* *Internal Force (Fi)*
- *springDamper1* *Displacement Difference (dx)*

e.) Running the Simulation

With the prepared sample model you can perform all computations implemented in SimulationX:

- *Simulation in the transient mode*
- *Computation of the equilibrium*
- *Linear model analysis (natural frequencies and mode shapes)*

We will restrict ourselves to the simulation in the transient mode. Figure 11 shows the dialog of the simulation control. You open the dialog of the simulation control panel using the menu "Simulation/ Transient Settings".

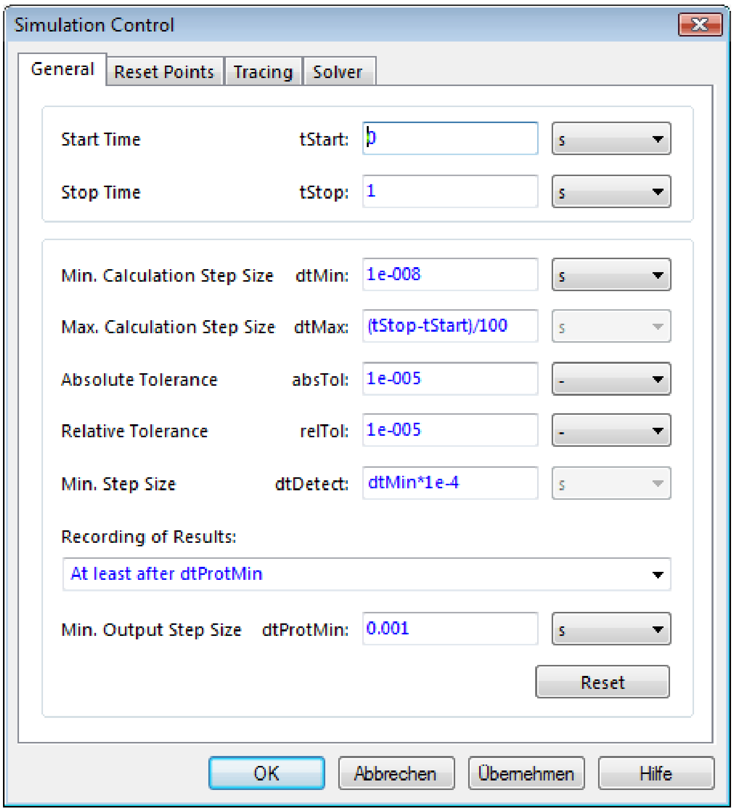


Figure 11: Property window "Simulation"

Now you can start the simulation via the menu "Simulation/Start" or with the button in the tool bar. The computation takes place up to the given stop time. The default value for the stop time is 1s. You can change this value in "Simulation/Transient Settings".

f.) Opening a Result Window

Let us now display the results of the simulation. Activated protocol symbols can be dragged into the model view or an already opened result window. For this click on the protocol symbol and move it to the desired place, while keeping the button pressed. When releasing the button, the result protocol is shown either in a new or in an existing result window.

Working example:
We create a result display for the variable mass1.x in a y=f(t) diagram.

In order to open the appropriate result window, you use the Drag&Drop procedure described above. Select the element "Mass1" in the model view by a mouse click. In the model explorer the available result variables are displayed. You click the protocol attribute for the displacement of "Mass1" with the left mouse button and drag it into the model view. When releasing the mouse button the result window becomes visible.

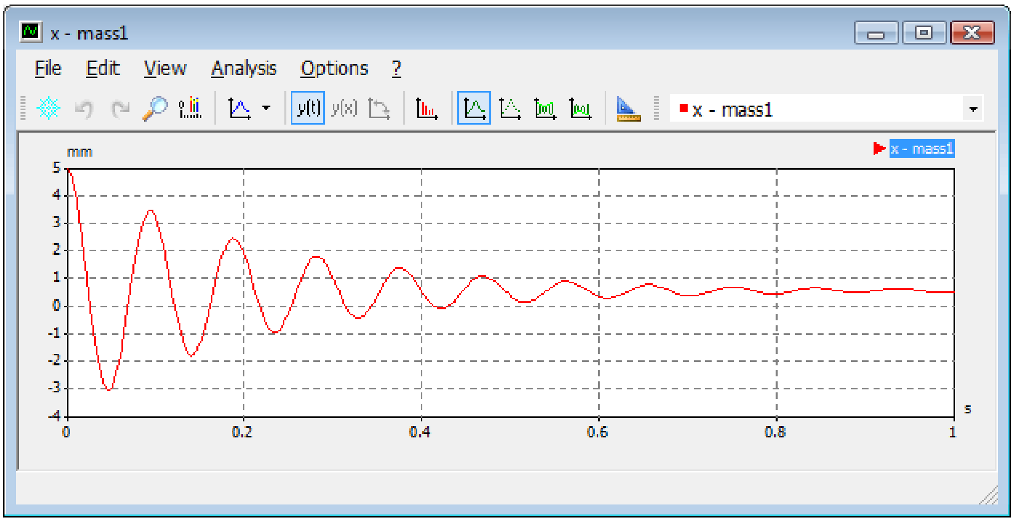


Figure 12: Result window for "mass1.x"

Working example:

Now we add the result variable "mass2.x" to the already existing result window.

For this select the element "mass2" in the model view and drag the protocol attribute symbol for the displacement into the existing result window. The window now shows two curves.

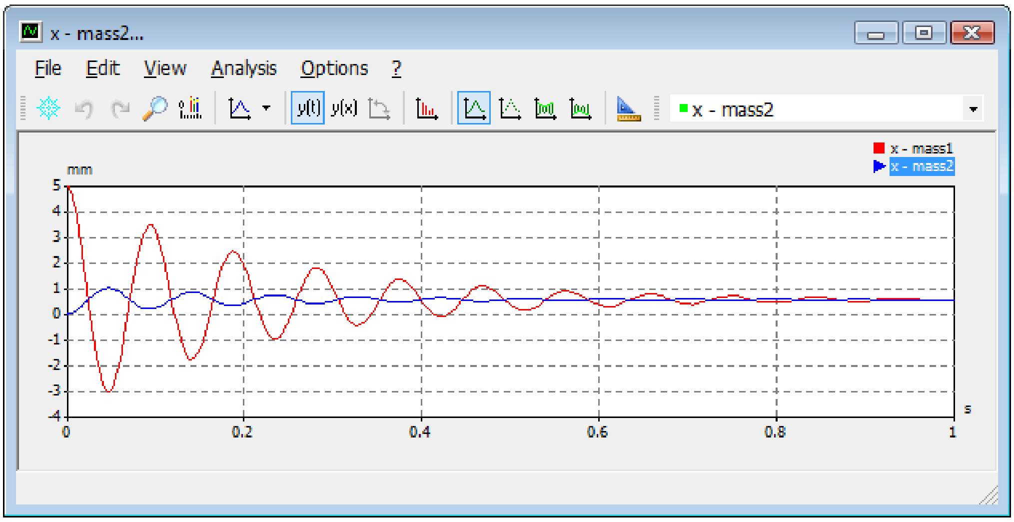


Figure 13: Result window with 2 curves

Now we would like to display the displacement difference (dx) of the spring damper element (SpringDamper1) over the internal force of the element (y(x) diagram). We have already activated the corresponding protocol attributes, so the results were stored during the simulation. Now you create a common result window containing dx and Fi (as described above for the mass displacement). The result display is switched to the y(x) mode by pressing the y(x) button in the result window toolbar. You now should see the curve as shown in Figure 14. You can swap the two axes by pressing the button.

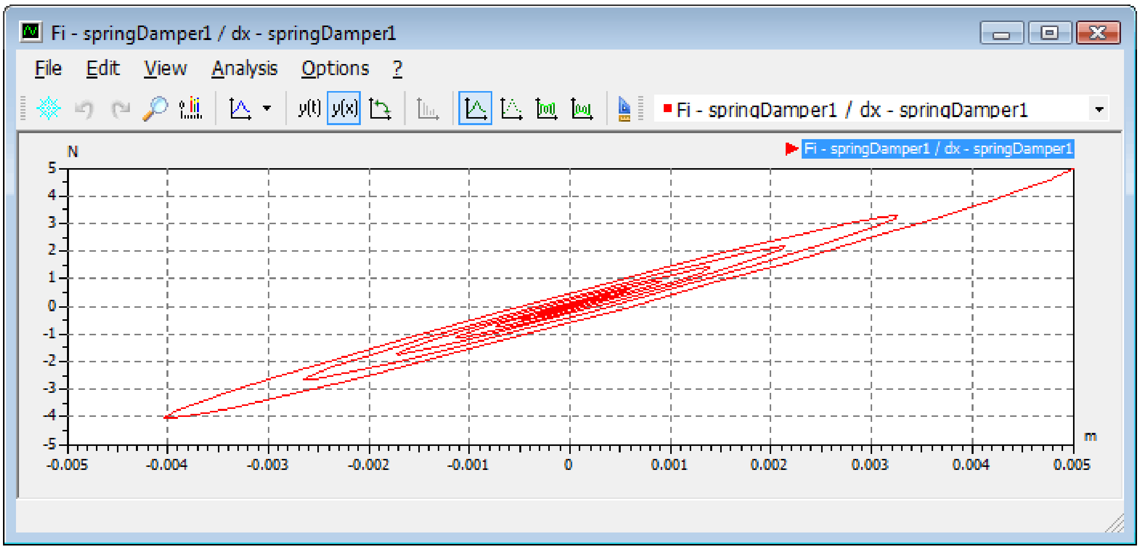


Figure 14: Result window with a y(x)-representation

Now you can play with the model. You can reset the simulation with the button, apply parameter or structural changes to the model and start the simulation again. When resetting all result curves are deleted.

Note: If you would like to preserve a result curve, you can freeze it with the button before resetting. Thus the curve remains visible and you can directly observe the effects of you parameter changes by comparing the new result with the frozen curve.

ITI SimulationX for heterogeneous system modeling and analysis

Use they the possibilities of ITI SimulationX for a fast and efficient problem solution as well as during the evaluation and optimization of technical systems:

- *Parameter Studies automatically (button or menu Analysis/Variants Wizard …)*
- *Linear system analysis: Analyze Natural Frequencies and Mode Shapes of your system (button or menu Analysis/Natural Frequencies …)*
- *Linear system analysis: Input-Output Analysis (button or menu Analysis/Input-Output Analysis)*
- *Extend existing elements (button or menu Elements/Derive)*
- *Creating Compounds (button or menu Elements/Summarize)*
- *Use the TypeDesigner/FluidDesigner for creating your own element types and fluids*
- *Implement your own user-specific algorithms*
- *Co-Simulation*
- *Code Export*
- *Steady state simulation*