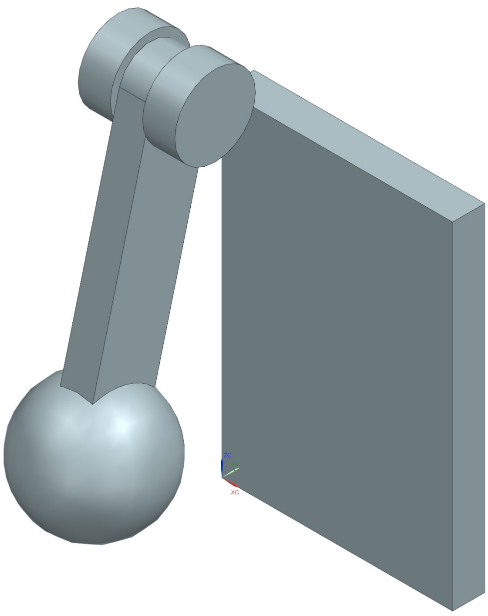
LIÈGE université
Sciences Appliquées

SIEMENS NX TUTORIALS

Université de Liège - Faculté des sciences appliquées

NX Motion (dynamic) :

The pendulum



NX 2206 - Bolyn (inspired by previous works)- November 2024

1. Introduction

NX Motion gives the opportunity to simulate mechanisms kinematically or dynamically. This tutorial is a dynamical simulation of a pendulum hitting a wall. Dynamic simulations allow us to use or compute forces, whereas it is not possible with kinematic simulations, for instance.

We strongly advise you to complete the assembly and kinematic simulation tutorials before taking this one. These tutorials are available in French from the "Communication Graphique" course at the University of Liège.

2. Reminder of NX Motion

The two first steps involved in parametrising a dynamic simulation are identical to those for a kinematic simulation:

1. Defining motion bodies
2. Defining joints

However, please note that unlike in kinematic simulations, the masses and inertias of the bodies will be taken into account. Make sure that the geometry and material of the parts are correctly defined before starting a dynamic simulation.

The next steps are specific to dynamic simulation:

3. Setting loads
4. Defining contacts if required

Contacts are difficult to simulate, as we will see later in this tutorial. It is recommended to replace contacts with joints as much as possible. The purpose of this tutorial is to look at different features of NX Motion for dynamic simulations, which is why the contacts are detailed (furthermore, the collision between the pendulum and the wall cannot be defined differently).

2.1 Creating file

Create a new *sim* file with NX Motion as the solver, but deactivate the joint wizard. As you can see in the title of the window, NX Motion is now activated, and you can read on the status bar that the Gruebler count is zero.

2.2 Defining motion bodies

We need to define each body (there are three: the pendulum, the wall, and the hinge). To do so, use the function *Motion Body* available in the panel *Home*.

For each body, select the object representing the body, then give them their name and validate with *OK* or *Apply* (*OK* will close the window but not *Apply*). Note that by default the mass properties are computed automatically (computation based on the geometry and the material of the model), but you can manually change them with the *User defined* mode (for instance, if the 3D model is a simplified version of the real part). The case for the pendulum is presented in Figure 1.

2.2 Defining kinematic joints

Now the kinematic relations between the bodies must be defined, but before doing so, we must not forget to fix the bodies that will not move.

To fix a body (the hinge and the wall in this tutorial), there are two options:

- Fixing the body without creating a joint
- Creating a fix joint (visible in the motion navigator)

You can fix a body without creating a joint by clicking on *Fix the Motion Body without Joint* after a right click on the Motion Body in the Motion Navigator. Or by simply ticking on this option when creating the Motion Body. If you want to create a joint, the steps are the same as for other joints (see below).

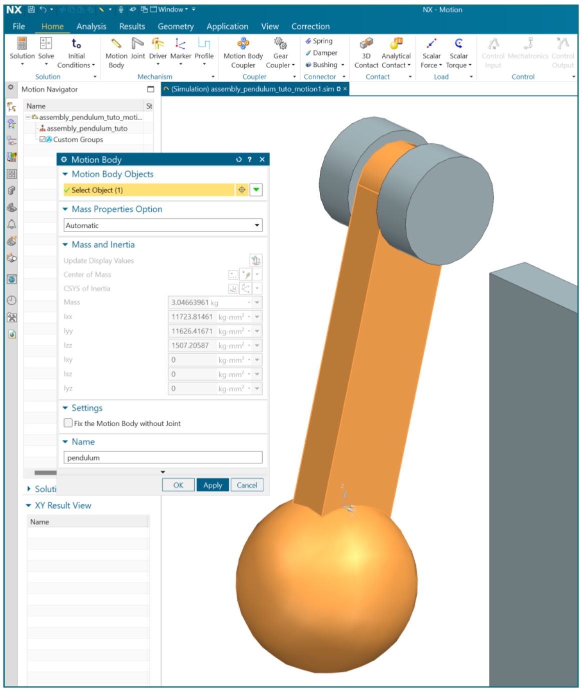


Figure 1—Defining motion body

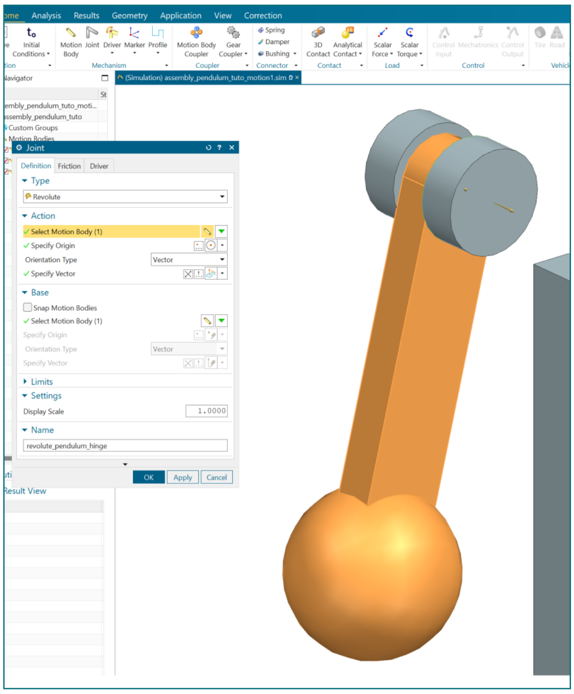


Figure 2—Defining kinematic joint

To create the revolute joint (this joint allows only one rotation, which is the relation between the pendulum and the hinge), click on *Joint* and select *Revolute* in section *Type*. In the *Action* section, select the *Motion Body* representing the pendulum, then define the axis of the joint (around which the rotation is allowed):

- Select a point on the axis. You can easily select one with the *Arc/Ellipse/Sphere Center* option.
- Define the vector of the axis. Since it is the axis of the hinge, you can use options such as *Curve/Axis Vector* or *Two Points Vector*. You can also use the X-axis because the assembly was realized such that the axis of the hinge is parallel to it, but it is not always the case (thus using absolute axes is usually not recommended).

Before validating this joint, define in *Base* section the body to which the joint is referring. If you do not select a *Motion Body*, the reference will be the absolute referential (actually, in our case, it does not really change the motion of the pendulum since the hinge is fixed, but it can induce errors in the computations because the Gruebler count may not be correct).

3. Adding loads

For dynamic simulations, it is possible to add forces or torques. For the pendulum case, we will apply a torque on the hinge axis and a force on the spherical part of the pendulum to swing it towards the wall.

3.1. Torque

Open *Scalar Torque* of the Load section to set the constant torque. By selecting the type as *Revolute Joint*, you can easily select the one from the hinge and set its value to 500 Nmm (you can easily select objects through the *Motion Navigator*). Of course, do not forget to give an understandable name to that load.

The load we just set is constant in time, which means that during the whole simulation the torque applied will constantly be 500 Nmm. It happens that loads are applied for a specific time; that is what we are going to see with the force in the next section.

3.2. Force

To create a force, open *Scalar Force* (next to *Scalar Torque*).

For our case, we want to do a little push horizontally at the beginning of the simulation. To do so (see Figure 3):

- Set the direction as Y and the application type as "Action only".
- Select the pendulum as the motion body, and with the *Arc/Ellipse/Sphere Center* option, select the center of the sphere as the origin of the action.
- In Magnitude section, set the type as "Function" and set the name of the force.

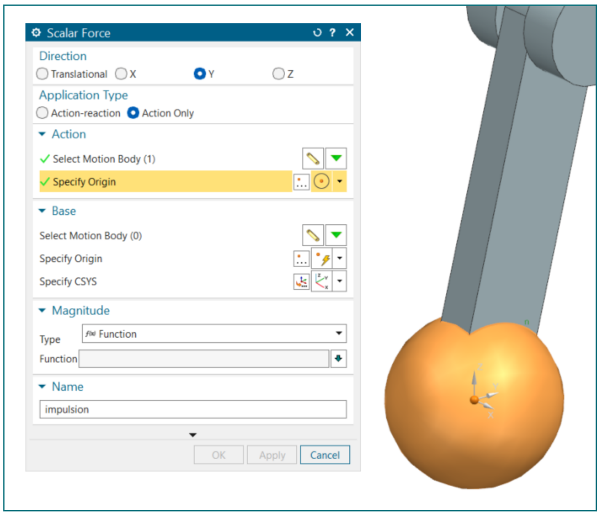


Figure 3—Defining a force

Now we need to define the function. NX offers many different possibilities through the *Function Manager* (common math functions, etc.), but we will create our own function with a spreadsheet.

4. Function with spreadsheet

Before closing the window, we need to define the function through the *Function Manager*: next to the field Function of the Magnitude section, click on the green arrow then *Function Manager*. The *Function Manager* allows us to create our own functions, which can be defined by common math functions (Math mode) or by tables of values (Table in AFU mode).

Select the *Table in AFU* Mode then create a new function with the button at the bottom left (representing a pencil on a graph, see Figure 4 on the next page). In the new window:

- Select ID in *Creation Steps* section then set a good name for your table.
- Select *XY Data* (XY button in *Creation Steps*) and click on the Excel button (called Key In from Spreadsheet) to open a spreadsheet file (as shown in Figure 5).

The spreadsheet already has two columns called X_Value and Y_Value, which are in our case the time and the magnitude of the force, respectively. To realize our little push, as we explained before, put the values detailed in Table 1.

Before closing the spreadsheet, update the values by clicking in the spreadsheet on *Update Table function* in the ribbon *Complements* (or *Add-ins*). Once done, close the spreadsheet and validate your function with the button *OK* in the *XY Function Editor* window and the other windows previously opened.

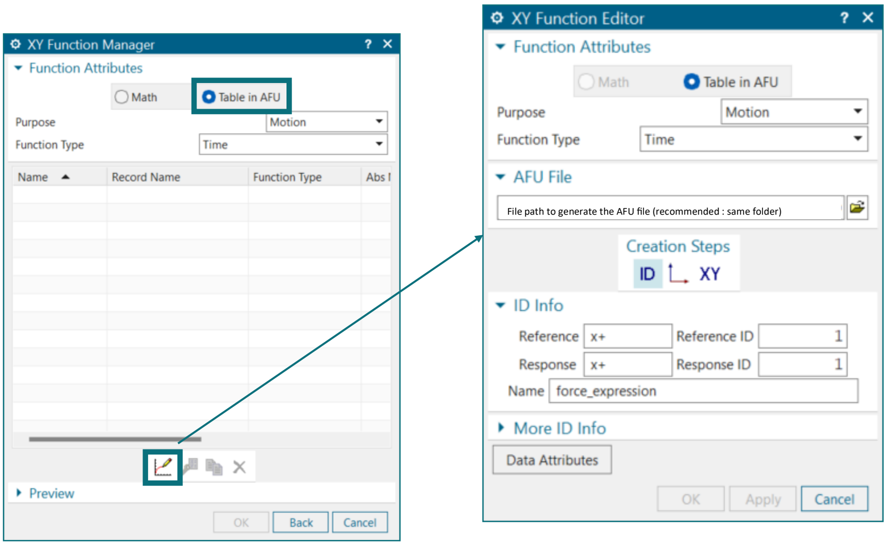


Figure 4—Create a new table

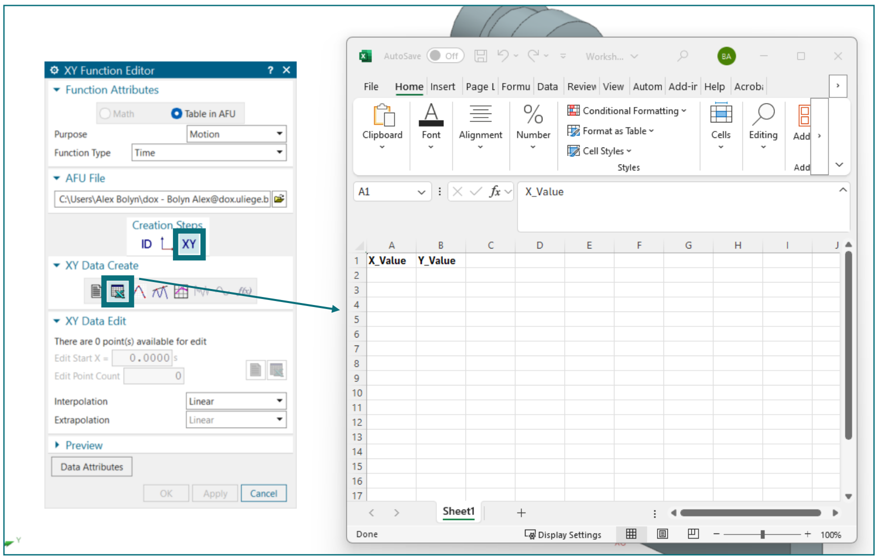


Figure 5—Defining table from spreadsheet

Time (s)	Load (N)
0	0
0.09	0
0.10	300
0.11	0
1	0

Table 1—Values of the force function

5. Defining a 3D contact

In the case of a simple contact between two parts (such as our case), NX provides a useful tool to define the contact. With this defined contact, it is possible to compute the impact force, for instance.

Click on *3D Contact*, which is available in the *Contact* section. With the type *CAD Contact* selected, set the pendulum as the *Action* body and the wall as the *Base* body. We need now to define the contact parameters.

The contact parameters constitute the most complicated part in defining a contact since they are determined experimentally and depend on factors such as the type of material and the condition of their surface. You can find starting values in Annexe A for different materials.

In our case, both the pendulum and the wall are in steel. In the *3D Contact* window, activate the Coulomb friction and fill the requested parameters with the values provided in Annexe A. Give a correct name to this contact, then validate to create it.

All the parameters are now set, it is time to start the computations.

6. Simulation

As for cinematic solutions, we must set the parameters of the solution before starting the actual simulation. To do so, click on *Solution* and set it as follow:

- Be sure *Dynamic Analysis* is set.
- The simulation will be on the first second of movement. Thus, set *Print Interval Definition* as *Fixed Print Interval* and set the time of simulation from 0 to 1 with a timestep of 0.02s. It is actually equivalent to setting *Print Interval Definition* as *Fixed Number of Steps* with 50 steps.
- Verify that the gravity is applied in the –Z direction.
- Give a correct name to the solution, then validate.

Now that the solution exists and is activated, press on *Solve* to start the computation. A window pops up to show the steps computed. If everything went fine, the status should be 100% for the last step computed.

The results of the simulation are now available. You can go in the *Results* panel and press *Play* to see the animation of the movement, but the most important part are the data you can access to.

7. Extracting data

Once the simulation is done, you can access data such as displacement, velocity, acceleration, or force for body motion or joints (also loads and contacts). Deploy the *XY Result View* section under the *Motion Navigator* and select the object you require the information in the *Motion Navigator*.

For instance, we can look at the angle of the pendulum according to its revolute joint. To do so, select the revolute joint in the *Motion Navigator*, then in the *XY Result View* go to Relative/Displacement and right click on RZ then plot in the popping menu (or double click on it). Relatively, a revolute joint frees the rotation around its Z axis.

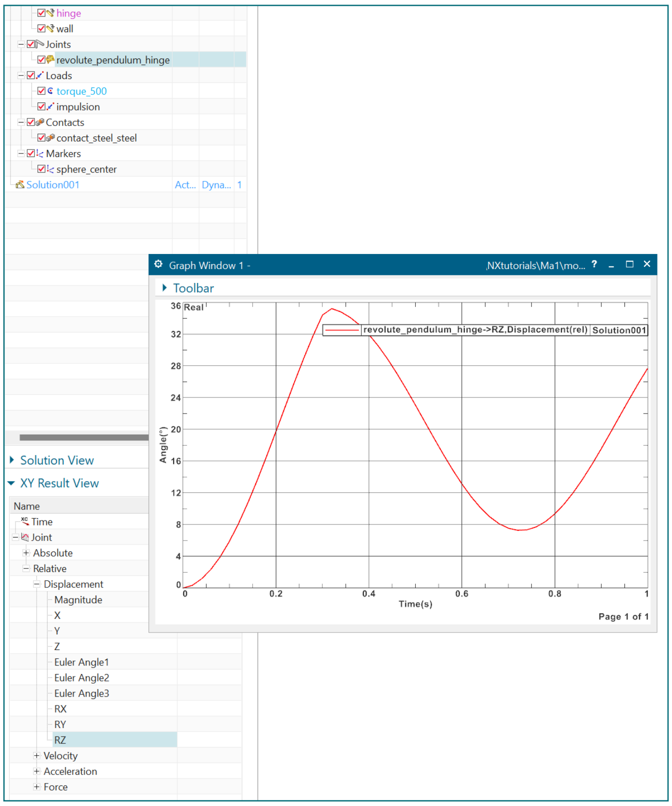


Figure 6—Plotting the relative angle of the revolute joint

When you select a *Motion Body*, the information provided is about the center of gravity of the object. If you need information at a specific point of a motion body, it is necessary to create a *Marker*. It is shown in Figure 6 the creation of a marker at the center of the sphere of the pendulum.

Be careful that you have to rerun the simulation if you want the results.

If you want to plot the contact force, select the contact and absolute force in the *XY Results View*. However, the time of contact is very small compared to the print interval, thus the force might not have been measured. To modify the solution parameters, right-click on the solution in the *Motion Navigator* then click on *Solution attributes...* (do not forget to rerun the simulation after). A timestep of 1 ms or less should be enough (plot of the force on Figure 8).

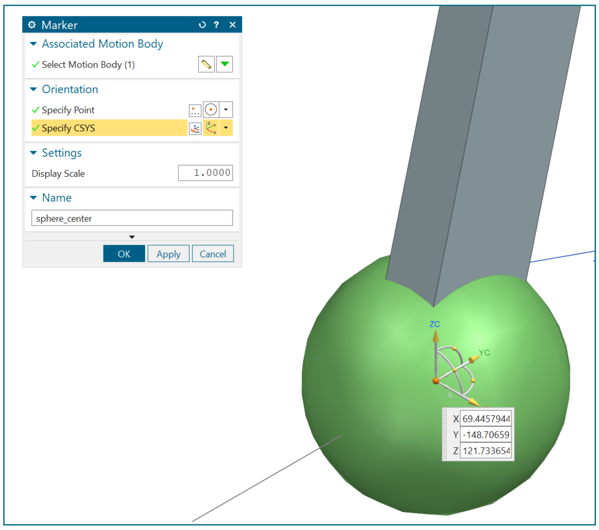


Figure 7—Setting a marker at the sphere center

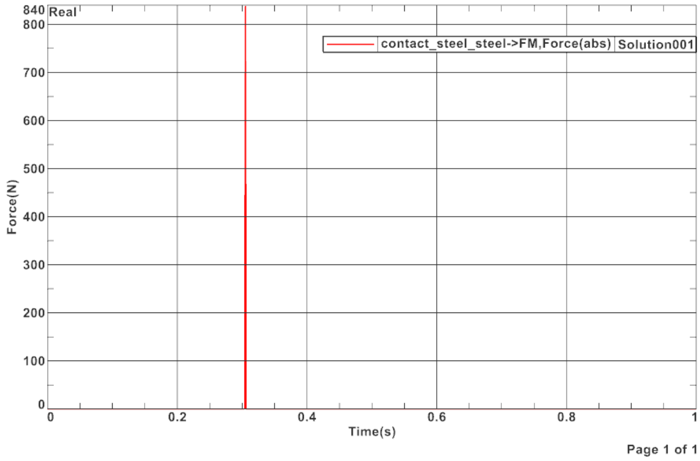


Figure 8—Contact force between the pendulum and the wall

In the plot window, the toolbar provides useful functions such a *Capture Image* that capture the plot. However, the quality is often not good, so it is better to export it as a CSV file to plot it with another software. You can export a plot by right-clicking on the graph, then *Export...* . You can find more details about the plots in NX in the tutorials of the "Communication Graphique" course.

Annexe A : contact material properties

Source : https://docs.sw.siemens.com/en-US/doc/209349590/PL20190529153447339.motion/id562936

Those are typical values that can be used as a starting point.

Materials :

A- Steel (dry); B- Steel (greasy); C- Aluminum (dry); D- Aluminum (greasy);
E– Acrylic; F– Nylon; G– Rubber (dry); H– Rubber (greasy)

Mat. 1	Mat. 2	Stiffness (N/mm)	Stiffness exponent	Material damping (N*s/mm)	Penetra-tion depth (mm)	Stiction velocity (mm/s)	Friction velocity (mm/s)	Static coefficient of friction	Dynamic coefficient of friction	Restitu-tion co-efficient
A	A	100000	1.5	50	0.1	0.1	10	0.3	0.25	0.15
B	B	100000	1.5	50	0.1	0.1	10	0.08	0.05	0.15
B	A	100000	1.5	50	0.1	0.1	10	0.08	0.05	0.15
C	C	34000	1.5	30	0.1	0.1	10	0.25	0.2	0.2
C	A	34000	1.5	30	0.1	0.1	10	0.25	0.2	0.2
C	B	34000	1.5	30	0.1	0.1	10	0.08	0.05	0.2
D	D	34000	1.5	30	0.1	0.1	10	0.05	0.03	0.2
D	A	34000	1.5	30	0.1	0.1	10	0.05	0.03	0.2
D	B	34000	1.5	30	0.1	0.1	10	0.05	0.03	0.2
D	C	34000	1.5	30	0.1	0.1	10	0.05	0.03	0.2
E	E	1200	2	0.7	0.1	0.1	10	0.15	0.1	0.4
E	A	1200	2	0.7	0.1	0.1	10	0.15	0.1	0.4
E	B	1200	2	0.7	0.1	0.1	10	0.08	0.05	0.4
E	C	1200	2	0.7	0.1	0.1	10	0.15	0.1	0.4
E	D	1200	2	0.7	0.1	0.1	10	0.15	0.1	0.4
F	F	4000	2	0.15	0.1	0.1	10	0.13	0.09	0.5
F	A	4000	2	1.5	0.1	0.1	10	0.13	0.09	0.5
F	B	4000	2	1.5	0.1	0.1	10	0.08	0.05	0.5
F	C	4000	2	1.5	0.1	0.1	10	0.13	0.09	0.5
F	D	4000	2	1.5	0.1	0.1	10	0.05	0.03	0.5
F	E	4000	2	1.5	0.1	0.1	10	0.13	0.09	0.5
G	G	3000	1.1	0.6	0.1	0.1	10	0.7	0.55	0.8
G	A	3000	1.1	0.6	0.1	0.1	10	0.3	0.25	0.8
G	B	3000	1.1	0.6	0.1	0.1	10	0.08	0.05	0.8
G	C	3000	1.1	0.6	0.1	0.1	10	0.25	0.2	0.8
G	D	3000	1.1	0.6	0.1	0.1	10	0.05	0.03	0.8
G	E	3000	1.1	0.6	0.1	0.1	10	0.15	0.1	0.8
G	F	3000	1.1	0.6	0.1	0.1	10	0.13	0.06	0.8

Mat. 1	Mat. 2	Stiffness (N/mm)	Stiffness exponent	Material damping (N*s/mm)	Penetra-tion depth (mm)	Stiction velocity (mm/s)	Friction velocity (mm/s)	Static coefficient of friction	Dynamic coefficient of friction	Restitu-tion co-efficient
H	H	3000	1.1	0.6	0.1	0.1	10	0.5	0.43	0.8
H	A	3000	1.1	0.6	0.1	0.1	10	0.3	0.25	0.8
H	B	3000	1.1	0.6	0.1	0.1	10	0.08	0.05	0.8
H	C	3000	1.1	0.6	0.1	0.1	10	0.25	0.2	0.8
H	D	3000	1.1	0.6	0.1	0.1	10	0.05	0.03	0.8
H	E	3000	1.1	0.6	0.1	0.1	10	0.15	0.1	0.8
H	F	3000	1.1	0.6	0.1	0.1	10	0.15	0.1	0.8
H	G	3000	1.1	0.6	0.1	0.1	10	0.5	0.43	0.8