Lab 1: Introduction to ADAMS-VIEW

Released: August 22, 2016

1 The ADAMS environment

Open ADAMS-VIEW and click on **New Model**. In the dialog box (see Fig. 1) change:

- The name of the model: SCARA
- The unit system: **m**,kg,N,s,deg
- The working directory: Z:\Documents\Your_working_Directory

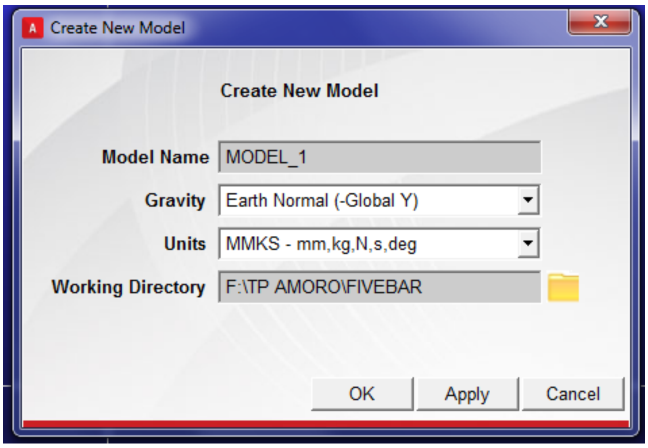


Figure 1: New model dialog box.

During the first part of this lab, you will design and analyze a 2R planar robot. For that, you should first build two type of devices:

1. "Parts" which correspond to the bodies to be created
2. "Joints" which by default are passive

First, you will create reference points using the table editor (**Tool/Table Editor**-Fig.2). Select "Points" in the selector at the bottom of the window and create three points at (0,0,0), (0.5,0,0) and (0.8,0,0). The points must appear in ADAMS window as in Fig. 3.

In ADAMS window, you can zoom by pressing 'Z', rotate by pressing 'R' and translate by pressing 'T'. You will now create your first link. Select the link

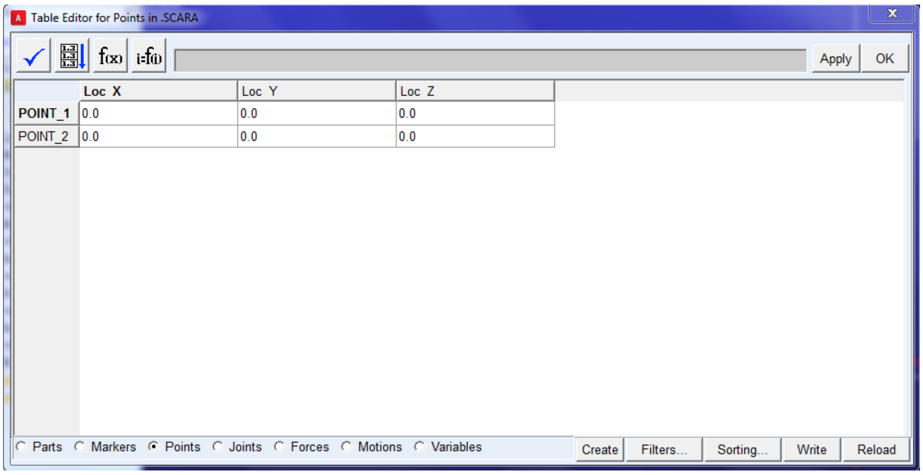


Figure 2: The table editor.

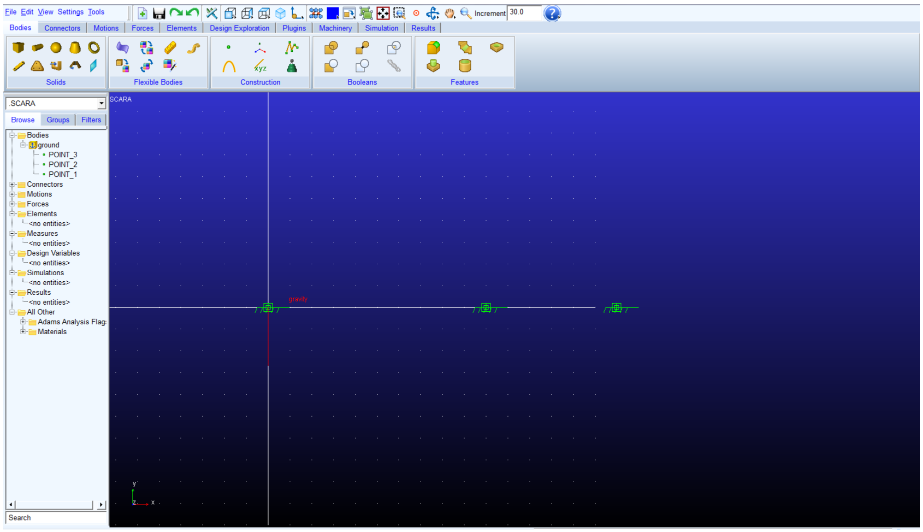


Figure 3: The points created in ADAMS environment.

icon  in **Bodies/Solids**. A new dialog box appears, allowing you to set some dimensions of the link (Fig. 4). Check and set the parameters:

- Width: 10 cm
- Depth: 5 cm

Then select POINT_1 then POINT_2 to create the link. You can then create a second link between the POINT_2 and POINT_3 with the same parameters. The links created must appear as in Fig. 5. Two parts have been created in the directory "Bodies" in the tree on the left. When creating the links, ADAMS automatically generates **MARKERS**. The markers in ADAMS are defined with

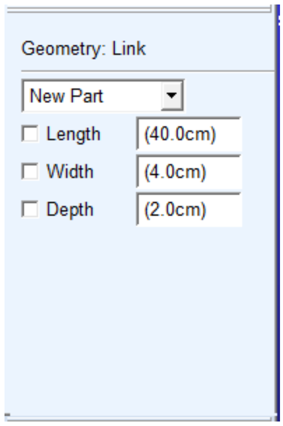


Figure 4: Link parameters.

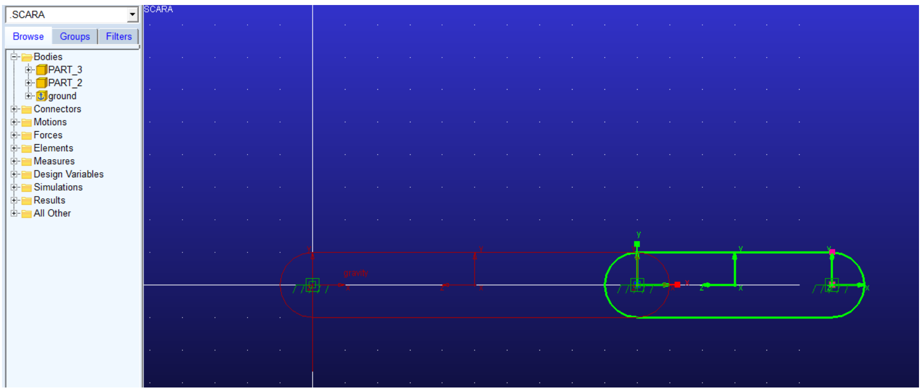


Figure 5: The two links created.

a position, orientation and the body they are attached to. The coordinates of the marker will evolve during the simulation depending of the reference body (no motion of the markers attached to the ground). The markers created with the links are the two extremities and the center of mass of the links. You can create also your own markers.

Now you will create the joints between the parts. Select the revolute joint icon  in **Connectors/Joints**. By default, you create a joint by clicking successively on the two parts then on joint location. Note that the order on which you click on the parts will influence the sense of rotation when you will impose motions on to the joints. Create two joints:

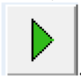
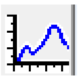
- between PART_1 and the ground at POINT_1
- between PART_2 and PART_1 at POINT_2

Two joints have been created in the directory "Connectors" in the tree on the left.

Finally, you can apply a motion into the joints. For that, select  in

Motions/Joint Motions. You can select the speed in the parameter dialog box. Set 10 deg/sec for both motor then click on the joints to apply the desired motions. Two motions have been created in the directory "Motions" in the tree on the left. You can modify the motion parameters motion with **Right Click/Modify** on the motion. Try to change the motion velocity.

Congratulations! You have built your first ADAMS mockup!

You can now proceed to the simulation. Select the simulation icon  in **Simulation/Simulate**. In the simulation dialog box, set the simulation time (5 sec.) and the number of computation steps (200). Then click on start button . Once the simulation is done, you can plot the result by clicking . The PostProcessor window is displayed. Several results may be plotted by selecting elements in the lowest part of the window (see Fig. 6).

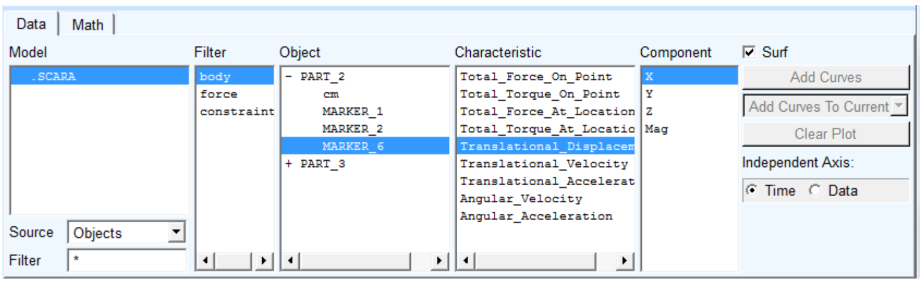


Figure 6: Selection table to plot results

Plot:

- The translational displacement of the end-effector (the point belonging to PART_2, at 0.3m from the second joint axis)
- The translational velocity and acceleration of the same point
- The input torques for both motors
- The reaction forces and torques into the 2 joints
- The power consumption of both actuators

This ends the first part of the present tutorial. Adams offers much more possibilities for the analysis of the behavior of a mechanism that you can find out yourself.

Note that you can freely download the Student version of Adams (limited to the creation of 12 bodies only) directly on the Adams website.

2 The ADAMS/Control environment

Adams offers the possibility to interface a mockup with Simulink in order to control it directly with Matlab. This can be done thanks to the Adams/Controls module.

Note that some Matlab versions are not compatible with the current version of Adams. For knowing which version is compatible with yours, have a look at the Adams Release documents.

Close your current Adams model and open the database "SCARA_IRCCyN.bin". This file includes the mockup of a planar 2R robot. Contrary to what you have done in Section 1, the parts of the present model have not been designed in the Adams environment, but have been designed with CATIA software and then imported into Adams. At the end of this lab, the lecturer will explain to you how to build a mockup made up of parts designed with CATIA.

It may be necessary to activate ADAMS/Control plugin. To do that, select Tools/Pugin Manage and check "load" and "load at startup" for ADAMS/Control.

In order to interface your Adams mockup with Simulink, the following procedure must be used.

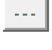
2.1 Definition of the state variables

It is first necessary to define the state variables defining the inputs of your model (the motor positions, or velocities, or even input efforts, etc.) AND the outputs (usually, the motor positions / angles corresponding to the measures of the encoders for a real system, but also some additional measures such as the end-effector pose, etc). To create a state variable, the following procedure should be followed.

Click on create a state variable  in **Elements/System Elements**. The window "Create State Variable" opens. Let us first create the variable *q1* corresponding to the position of the actuator 1 (input):

- In "Name", write q1
- In "F(time,...)" input 0. We will define this variable as an input and then it will be modified by Matlab during the co-simulation. It is not necessary to parametrize it now.
- Then click "Apply". Your input variable is created

Create a second input variable *q2*.

For output variables, an additional step is necessary. In this case the variable is defined by ADAMS. Let us now define the variable x, position of the end effector as an output. Create a variable as before, but in "F(time,...)" click to the right button  in order to define the function that represents the output . In the scrolling menu, select "displacement". Select the function "Displacement along x" and click on "Assist" (see Fig. 7). Displacements, velocities, angles,... are measured between markers. Here, the desired measure is between the end-effector and the reference. Right-click into the fields to select the appropriate markers. You can use "Pick" to select then in the 3D environment or "Browse" to choose them in the Database Navigator. The field "Along marker" is used to define the marker in which the result is expressed. The default marker (used by ADAMS when no input in the grey fields) is represented in the bottom-left

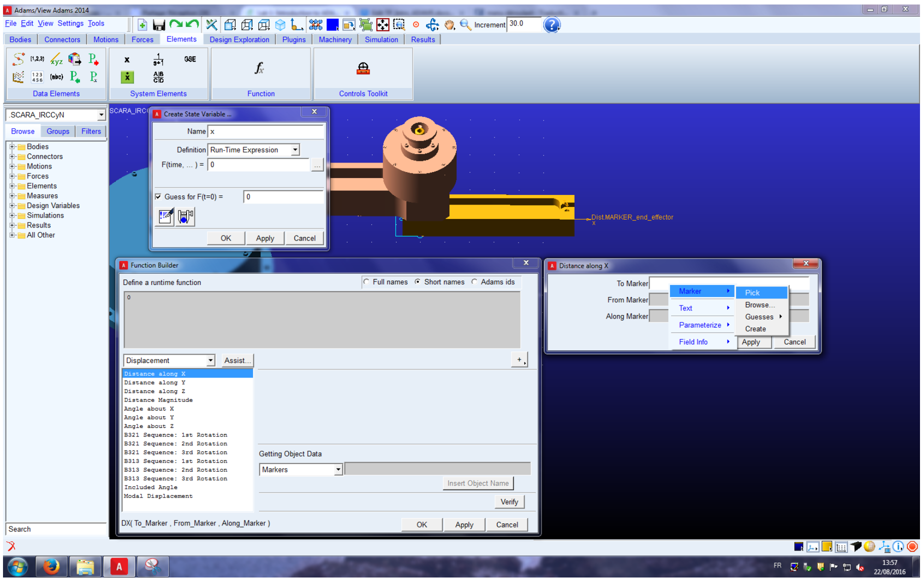
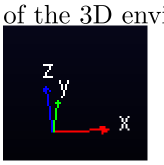


Figure 7: Pamaretrization of an output

of the 3D environment.



Create also the output variable y.

2.2 Parameterizing the motion of the actuators

Right click on the JOINT_1, and select "modify/Impose Motion". The window on Fig. 8 appears. To control the joint position with the variable *q1* input

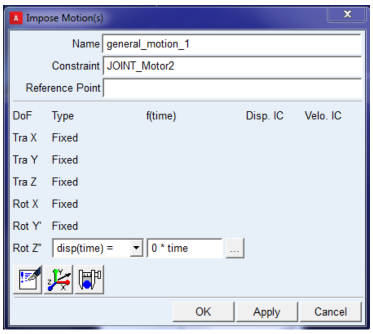
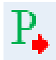


Figure 8: Pamaretrization of a joint motion

"disp(time)=VARVAL(q1)" and click "OK". Repeat the procedure for the second joint.

2.3 Creation of the plant

To create the inputs, select  in **Elements/Data Elements**. In Variable Name, input "q1,q2".

To create the outputs, select  in **Elements/Data Elements**. In Variable Name, input "x,y".

Click on  in **Plugins/Controls** and select **Plant export** and specify (see Fig. 9):

- the File prefix
- the input and output signals by clicking to "From Pinput" / "From Poutput" (remove the blank)
- Target Software: Matlab
- Solver C++
- Click OK

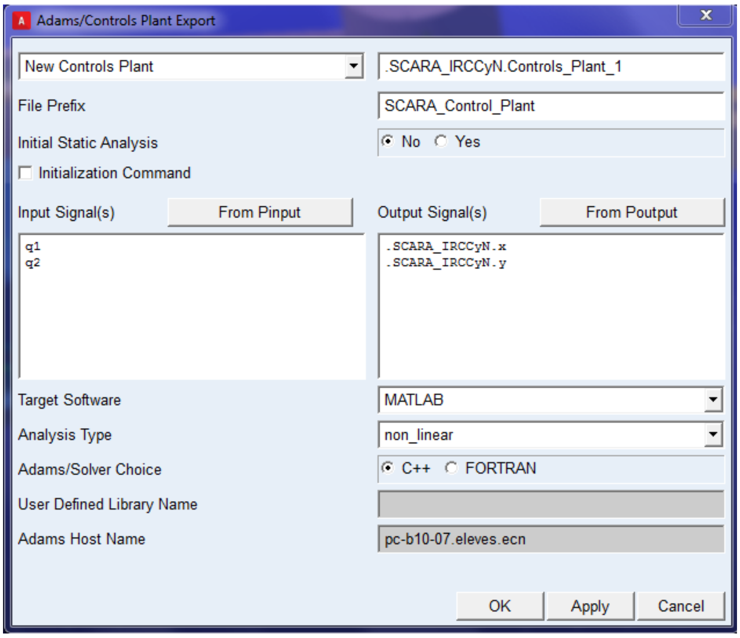


Figure 9: Plant exportation

3 Interfacing ADAMS with MATLAB

If you are not familiar with the MATLAB/Simulink environment please ask your professor. He will explain you the basic knowledge necessary for the labs.

In your working directory, you can now find a Matlab file corresponding to your control plant. Run this file. Then, enter the command "adams_sys"

A Simulink window will open after few seconds in which you have a block "adams_sub". This block amounts to the subsystem that has been created in the previous stages.

Copy the block "adams_sub" in a new Simulink window.

This block has two entries corresponding to the variables *q1* and *q2* defined in the Adams plant and two outputs x and y.

By double-clicking on the block "adams_sub", a detail of the block appears.

Open the block "MSC Software/ADAMS plant" and set the following parameters:

- For "animation mode", choose either batch (no mechanism visualization) or interactive (the simulation can run while seeing the mechanism moving in real time).
- Set the sampling time at 0.01 sec (Communication interval)
- Set the simulation mode to Discrete

Then, go back to the Simulink window, in which you have copied the block "adams_sub".

Add two ramp signals as inputs for $q1$ and $q2$ variables. Use a slope of 30 deg/s (be careful, inputs for ADAMS are considered as radians). Then run your Simulink model. Observe the results for x and y output variables.

For the 2R robot, we have

$$\begin{aligned} x &= l_1 \cos q_1 + l_2 \cos(q_1 + q_2) \\ y &= l_1 \sin q_1 + l_2 \sin(q_1 + q_2) \end{aligned} \tag{1}$$

Write this mathematical model in a "Matlab Function" block, and with $l_1 = 0.5$ m and $l_2 = 0.3$ m and connect it to the same inputs. Compare the results.