Lab 5 - Importing Solid Model Parts & Assemblies

Overview

In this lab we will be exploring how to import solid model parts and assemblies, created in a CAD program such as SolidWorks or Fusion 360, into a Simulink model for simulation. We will begin by observing a simple pendulum dynamic system in Simulink. We will import into the Simulink model two different solid model parts designed in the shape of asymmetric dumbbells and observe how the two different models behave.

After observing the pendulum system, we will look at importing a multibody robotic assembly into Simulink for simulation. After the import, we will correct the performance of the model and use a scope to observe the middle joint's movement which will evolve as changes are made.

Resources Required:
MATLAB
Simulink
Simscape Multibody

Pendulum.zip
multibodyRobot.zip

Description of the System

The pendulum system, is a simple model in which a pre-established block will act as a world frame and fulcrum point to the imported dumbbell solid models. There will be two variations of the dumbbell that we will use. The two dumbbell variations can be seen in figure 1. We will use these two variations to observe how slight changes in geometry can affect the physical behavior of them acting as pendulums.

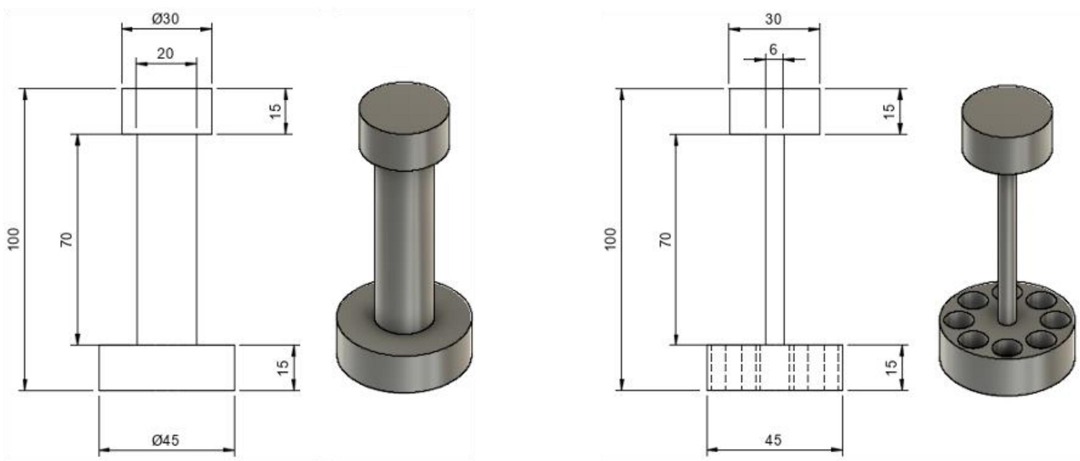


Figure 1: Dumbbell Models for Pendulum System

The multibody robotic assembly system, is a 4 linked arm with a tube end-effector. It is comprised of 3 HEBI module actuators that operate in series to drive motion at the joints and ultimately produce movement in the tube end-effector. The model of this system can be seen in figure 2.

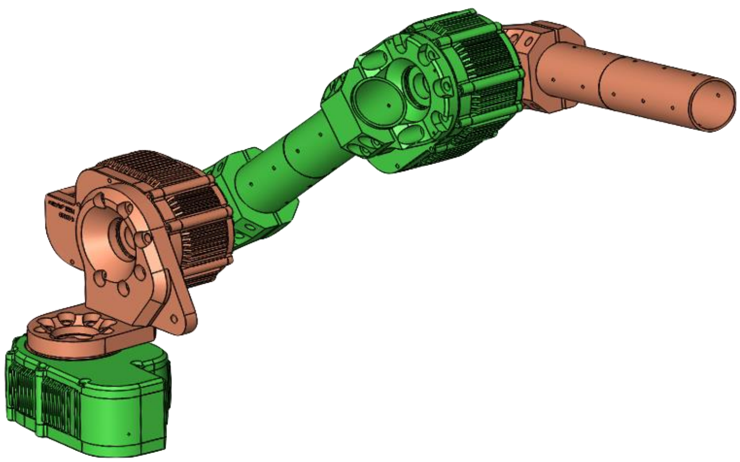


Figure 2: HEBI Robotic Assembly

Task 1: Importing Solid Models as a Simple Pendulum

In this task, we will observe the behavior of solid models acting as dumbbell pendulums in a Simulink model. To begin, retrieve the Simulink file titled “SimplePendulum.slx” and the solid model parts tilted “dumbbell.STEP” and “dumbbell2.STEP” within the “Pendulum.zip” folder.

1. Once, you have the files retrieved, open the “SimplePendulum.slx” file. The Simulink block diagram should look as it does in figure 3.

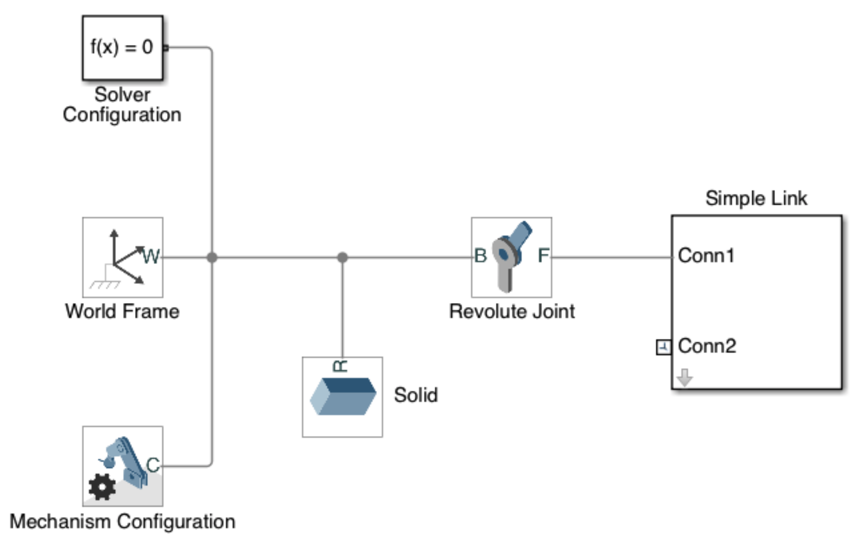


Figure 3: Simple Pendulum Model

2. Run the simulation and view the simple pendulum model in the Mechanics Explorer window when it opens up.

3. Enable the view of the coordinate frames within the model by right-clicking on the model window and selecting “Show Frames”. Enable the view of the model’s COMs (Center Of Masses) by also right-clicking on the model window and selecting “Show COMs”. Alternatively, you can also access these by clicking “View” within the top menu bar under the “MECHANICCS EXPLORERS” tab. The view should look as it does in figure 4.

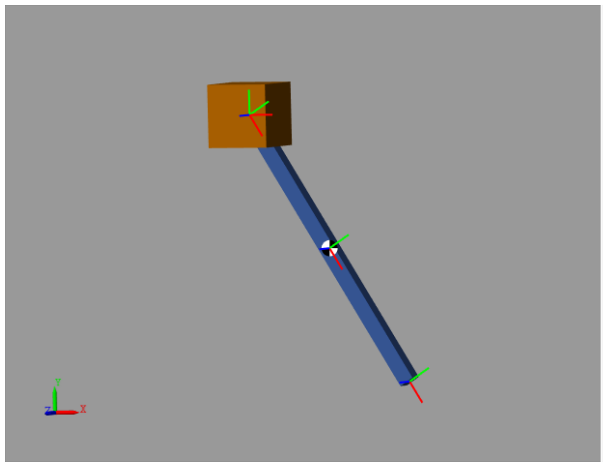


Figure 4: Simple Pendulum within Mechanics Explorer Simulation Window

4. Now, we will adjust the block diagram to produce a response in which the oscillations will die out. In the block diagram, open the “Revolute Joint” block and then expand the “Internal Mechanics” tab.
5. Add a damping coefficient of 0.001 N*m/(rad/s), as shown in figure 5. Re-run the simulation and observe the effect of the damping on the system behavior.

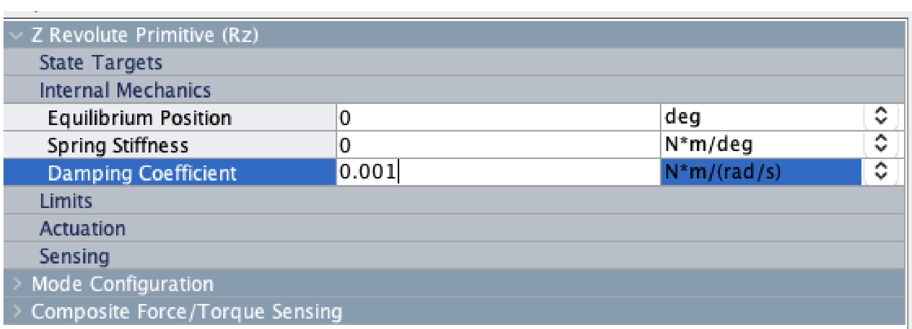


Figure 5: Simple Pendulum Damping Coefficient

6. Navigate back to the simple pendulum block diagram and open the “Simple Link” subsystem by clicking on the “Simple Link” tab between the Simulink ribbon and block diagram workspace. You can also access it by right-clicking on the “Simple Link” block and selecting either “Open in New Tab” or “Open in New Window” from the menu.
7. Within the subsystem, delete the “Solid”, “Rigid Transform1”, and “Conn2” blocks, as well as the lines connecting them together. At this point, if you update the model with “ctrl+d” (“cmd+d” if using a mac), you will notice that the simple rectangular prism pendulum disappears from the Mechanics Explorer window. All that is left is the brown block acting as the world frame.

8. Now we will replace the simple rectangular prism pendulum with our first custom dumbbell. While in the "Simple Link" subsystem, add a "File Solid" block to the workspace and connect the "File Solid" block to the "Rigid Transform" block. The subsystem block diagram will now look like figure 6.

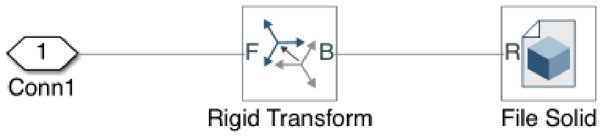


Figure 6: Simple Link Subsystem

9. Open the "File Solid" block and under the "Geometry" tab, click on the far-right side of the "File Name" field. We are clicking on a file browser button next to the text entry box. Note that this button is somewhat invisible initially, but is indicated with three little dots once selected. Figure 7 below highlights this button.

Figure 7: File Name Browse Button

10. Once the file browser window opens, navigate to and select the "dumbbell.STEP" file you saved and open the file. The file path will then populate in the "File Name" field indicating a successful import.
11. We now want to adjust the model's inertia parameter. While still in the File Solid window, expand the "Inertia" tab and for the "Based on" field, select "Custom Density" from the drop-down menu. Keep the default value of 1000 kg/m^3.
12. Update the File Solid visualization, by pressing F5 ("fn+F5" for mac) hot key. The results will appear as they do in figure 8. Exit the "File Solid" window by clicking "OK"

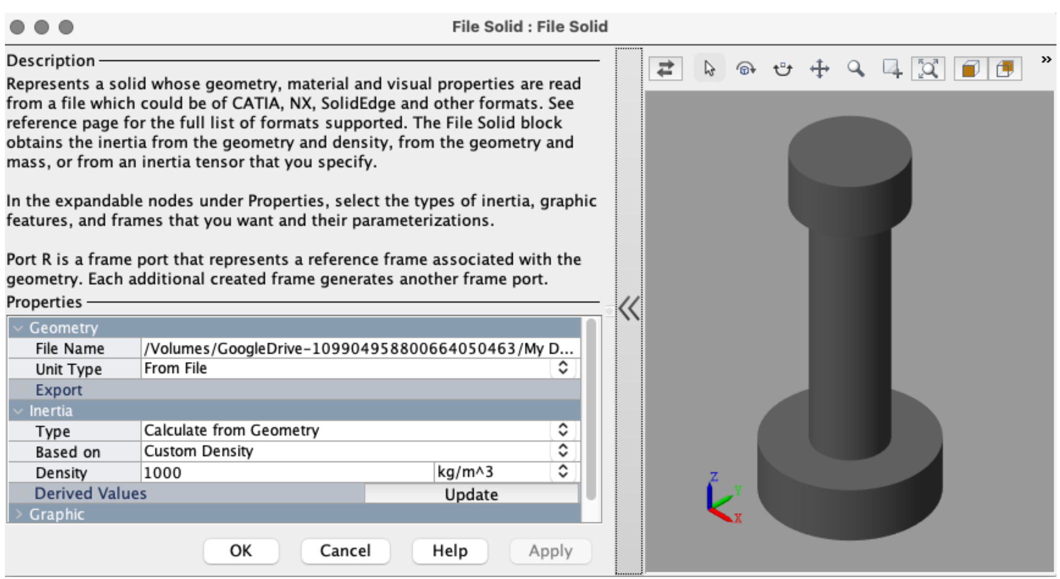


Figure 8: File Solid Window

13. Update the model and graphics. Within the Mechanics Explorer window, you will notice the dumbbell model is now in the Mechanics Explorer window with the world frame block. However, the orientation and link between the two bodies is wrong. We will now correct this.
14. Return back to the "File Solid" window for the dumbbell. Expand the "Frames" tab to reveal two fields. Add a new frame by clicking the yellow plus icon to the right of the "New Frame" field.
15. The window will change for you to select the parameters for the new frame. Under the "Frame Origin" section, choose the origin to be "Based on Geometric Feature". Click the smaller circular surface at the top of the model. Then click the "Use Selected Feature" button in the "Frame Origin" section.
16. In the "Frame Axes" section, change the following:
    a. Primary Axis: +Z
    b. Along Reference Frame Axis: +X
    c. Secondary Axis: +X
    d. Along Reference Frame Axis: +Z

    Figure 9 indicates how the selections should correctly appear. After verifying these are correct, save and close the block.

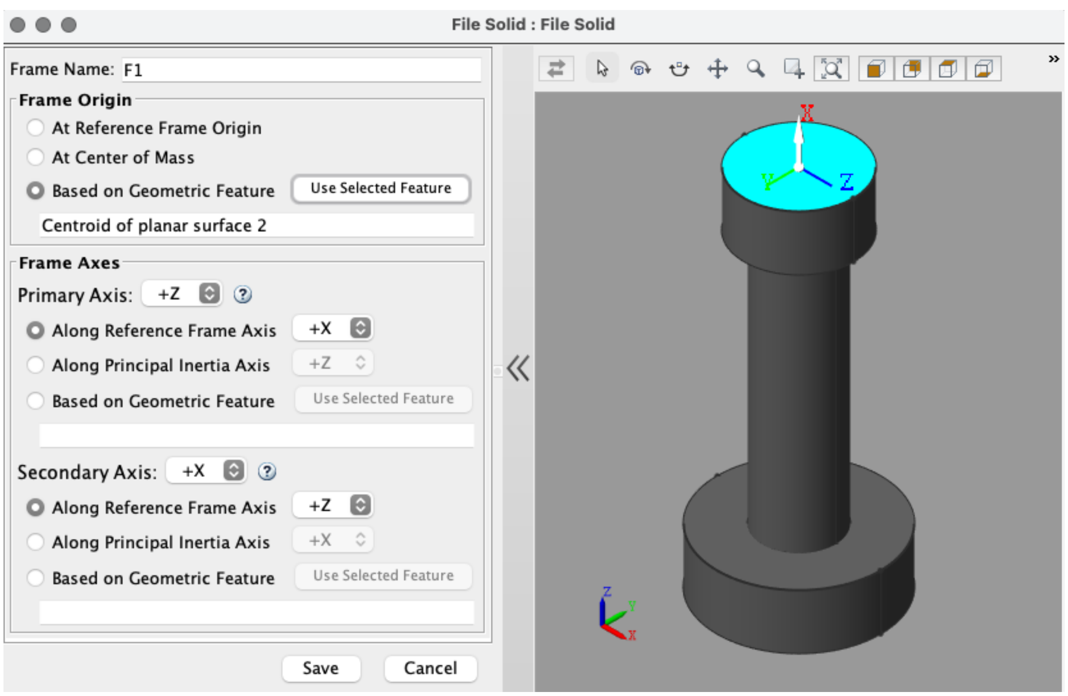


Figure 9: Dumbbell Frame Creation

17. We now need to establish the connection between the newly created frame on the dumbbell and the world frame block. In the "Simple Link" block diagram, delete the signal line between the "Rigid Transform" block and the "R" connection point on the "File Solid" block. Then establish a connection between the "Rigid Transform" block and the "F1" connection point on the "File Solid" block. It should look as it does in figure 10.

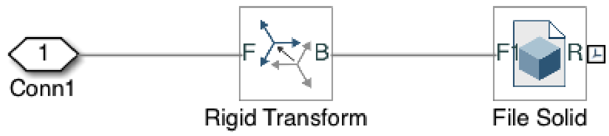
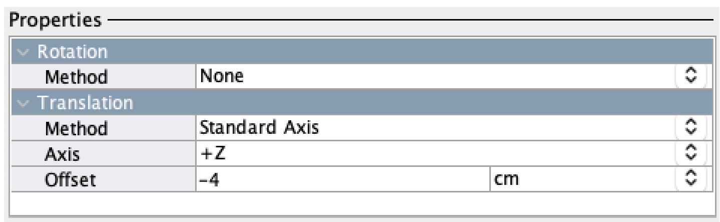


Figure 10: New Frame Connection in Simple Link Subsystem

18. Open the “Rigid Transform” block then expand the “Translation” tab and change the following parameters:
    a. Axis: +Z
    b. Offset: -4 cm

    The changes should appear as they do in figure 11.



Properties

Rotation		
Method	None	
Translation		
Method	Standard Axis	
Axis	+Z	
Offset	-4	cm

Figure 11: Rigid Transform Properties

19. Update the model and graphics. Within the Mechanics Explorer window, you will see the model correctly connected as it does in figure 12.

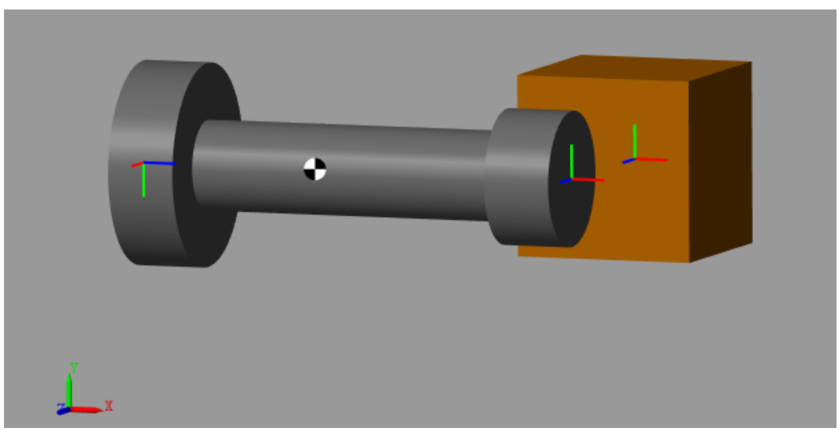


Figure 12: Dumbbell Pendulum

20. Now we need to add a scope to observe the behavior of the pendulum. Back in the “SimplePendulum” block diagram, open the “Revolute Joint” block. Expand the “Sensing” tab and click the check box for the “Position” field and apply. You will notice a newly created connection point on the “Revolute Joint” block. Click “OK” to save and exit.
21. In the block diagram workspace, add a “PS-Simulink Convertor” and “Scope” block. Then connect the blocks together to complete the sensing output. The diagram should appear as it does in figure 13.

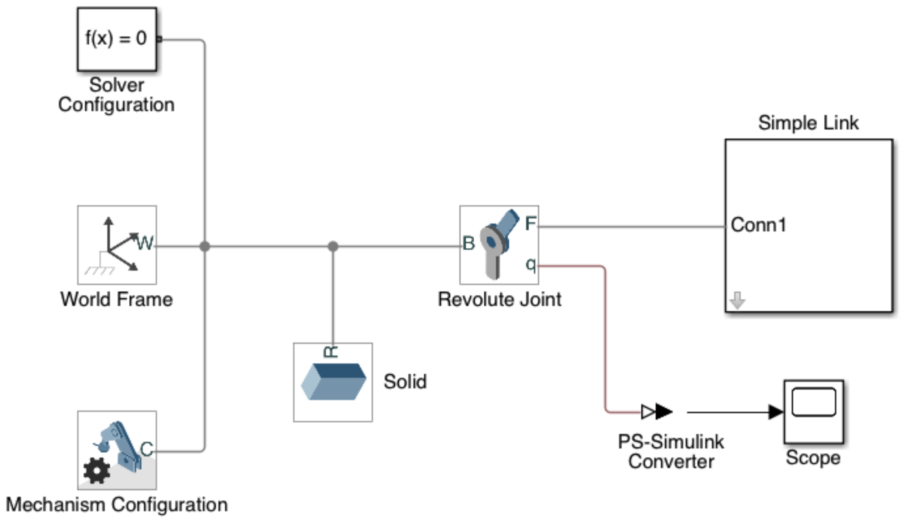


Figure 13: Simple Pendulum Block Diagram with Position Scope

22. Change simulation time to 5s. Run the simulation and observe the response.
23. Format and save the final block diagram and the scope response for the dumbbell pendulum's position appropriately and include them in your report.
24. Save, a screenshot of the pendulum model as it appears in the "Mechanics Explorer" window and include it in your report.
25. We now want to observe how the same pendulum model behaves with the second dumbbell variant as it has a slightly modified geometry. Navigate back to the "File Solid" block in the "Simple Link" subsystem and replace the "dumbbell.STEP" file by importing in the "dumbbell2.STEP" file.
26. Update the model, open the scope, run the simulation, and observe the response. You should see a somewhat different behavior in the pendulum response.
27. Ensure your plot is formatted appropriately, save it, and include it in your report.
28. Save, a screenshot of the pendulum model as it appears in the "Mechanics Explorer" window and include it in your report.

To show completion of this task, save your formatted plots for the two dumbbell responses and show the TA.

Task 2: Importing Multi-Bodied Assembly for Simulation

For task 2, we will be importing the HEBI multi-body assembly into Simulink for simulation. To begin, be sure you have saved the necessary folder titled "multibodyRobot" and all of its contents.

1. To import the model assembly, open MATLAB's main user interface. Then navigate to your saved "multibodyRobot" folder within the "Current Folder" pane to the left of the "Command Window"
2. In the command window, type the command "smimport('multibodyRobot.xml')" and hit enter to issue the command. Figure 14 shows how the view will look for the command prior to hitting enter.

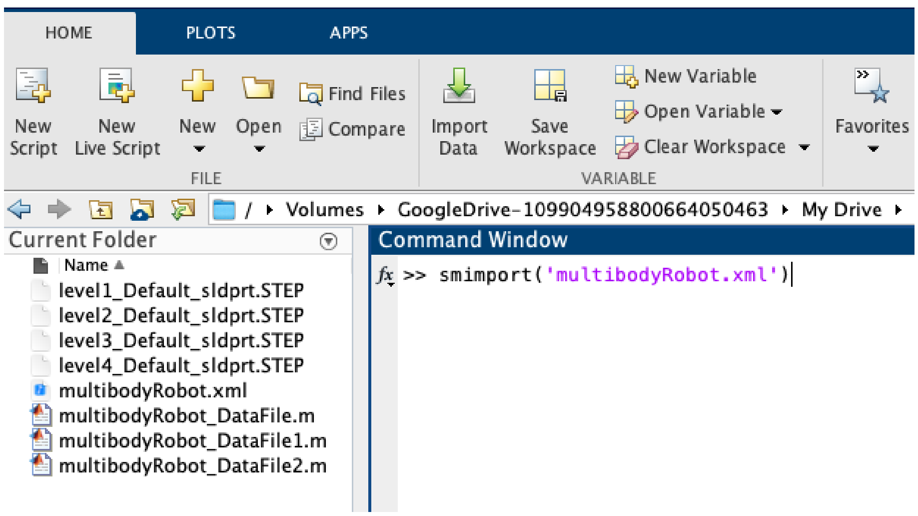


Figure 14: Importing multibodyRobot Assembly

3. After the command is finished, the "multibodyRobot" Simulink block diagram will open, as seen in figure 15. Now, run the simulation. Since this model has many details, it will take some time to process. Once, complete the Mechanics Explorer window will open showing the simulation. The motion of the robot will be chaotic. The following steps will correct it.

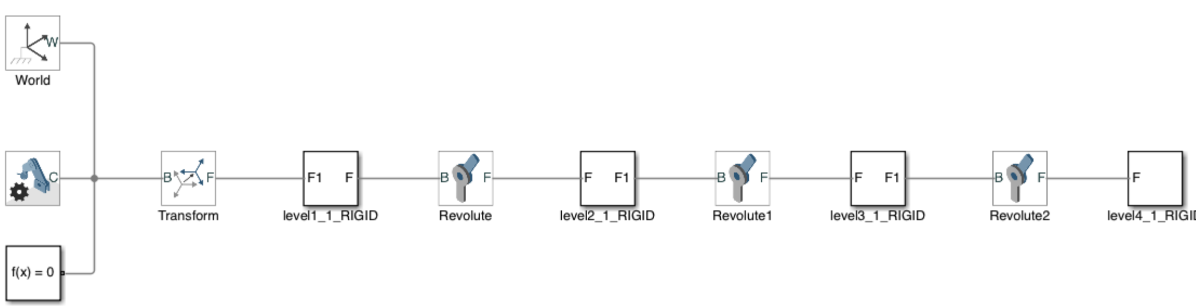


Figure 15: multibodyRobot Block Diagram

4. First, we will make clear the different bodies in the assembly. Go to the block diagram and open the "level1_1_RIGID" sub assembly.

5. Within the “level1_1_RIGID”, open the “Solid” block. Under the properties, expand the “Graphic” then “Visual Properties” tabs. Enter in the color value of [0 0.8 0] into the “Color” field and apply the change. The resulting change should look as it does in figure 16.

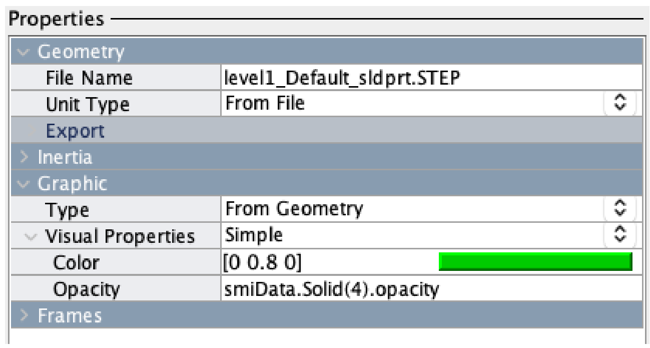


Figure 16: Color Change Property

6. Repeat the same steps to change the color for the “Solid” block within the “level3_1_RIGID” subsystem.
7. Update the model and graphics. Within the Mechanics Explorer window, you should see the model as it is shown in figure 17.

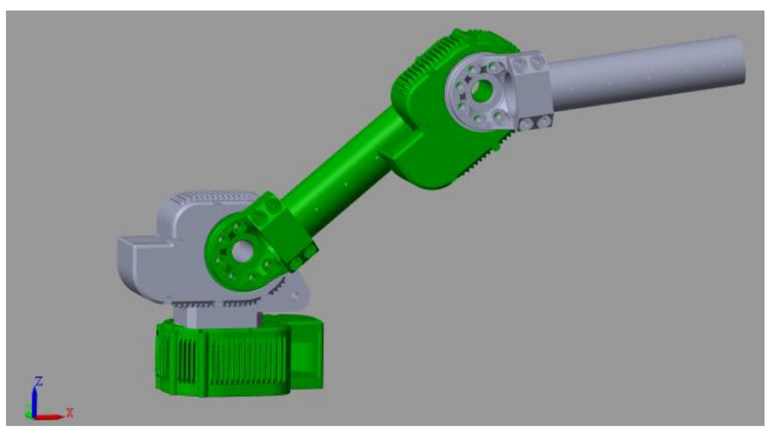


Figure 17: Colored Multibody Robot

8. Now we will fix the revolute joints and the model environment. In the “multiBodyRobot” block diagram, open the “Revolute” block. Expand the “State Targets” then “Specify Position Target” tabs and change the “Value” field to 0 deg. This change should look as it does in figure 18.

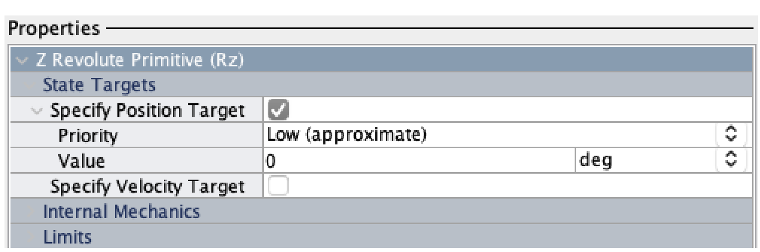


Figure 18: Revolute Joint Position Target

9. Repeat this change for the other to revolute joints in the model. That is for "Revolute1" and "Revolute 2"
10. Open the "Configuration Parameters" window by clicking the "Model Settings" icon. Change the following settings:
    a. Stop time: 3 (seconds)
    b. Solver: ode23t (mod.stiff/Trapesoidal)
    c. Max step size: 0.01 (seconds)

    The changes should appear as they do in figure 19. Apply the changes and exit by clicking "OK".

Figure 19: Configuration Parameters

11. Now we will add a scope to the middle joint to observe it's behavior. Open the "Revolute1" block and add a velocity sensor.
12. Add a "PS-Simulink Convertor" and "Scope" block to the diagram and connect them as shown in figure 20.

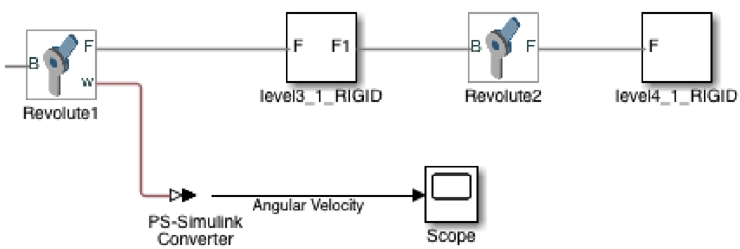


Figure 20: Middle Joint Angular Velocity Output

13. Open the "Mechanism Configuration" block and change gravity to apply in the -Z direction. It must be entered in as [0 0 -9.81] in the "Gravity" field. Figure 21 shows how this should look.

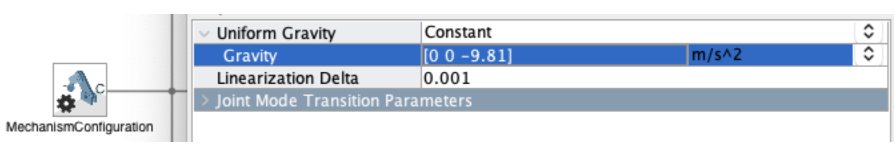


Figure 21: Gravitational Setting

14. Run the simulation and view the scope. Format the scope appropriately and save the plot to include in your report.

15. Now we will apply spring stiffness to the joints. The "Spring Stiffness" property is found within the "Revolute Joint" block under the "Internal Mechanics" tab. As shown in figure 22. Open each of the revolute joint blocks and apply the following spring stiffness values to the joints.
    a. Joint 1 ("Revolute" block): 1 Nm/rad
    b. Joint 2 ("Revolute1" block): 1 Nm/rad
    c. Joint 3 ("Revolute2" block): 0.1 Nm/rad

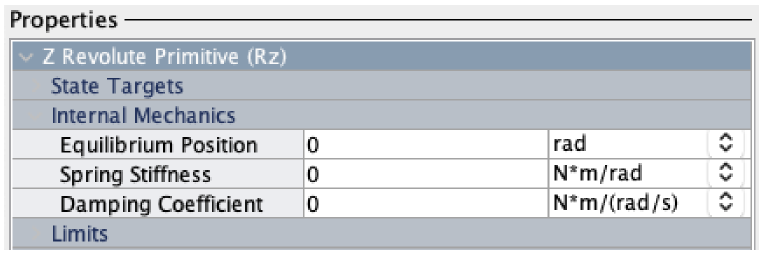


Figure 22: Internal Mechanics Properties

NOTE: Be mindful of the units. By default, the units are set use degrees, you will need to change the units to use radians by selecting the correct units with the drop-down selector.

16. After entering the spring stiffness values, run the simulation and view the scope. Make sure the scope is formatted appropriately and save the plot to include in your report.
17. Now return back to each of the revolute joint blocks and apply the following damping coefficient values to the joints. The "Damping Coefficient" property is just below the "Spring Stiffness" property.
    a. Joint 1 ("Revolute" block): 0.05 Nm/(rad/s)
    b. Joint 2 ("Revolute1" block): 0.05 Nm/(rad/s)
    c. Joint 3 ("Revolute2" block): 0.005 Nm/(rad/s)

18. After entering the damping coefficient values, run the simulation and view the scope. Make sure the scope is formatted appropriately and save the plot to include in your report.

NOTE: By this point, you should have three saved plots for this task. One plot showing the scope response after each of the three property changes in steps 13, 15, and 17.

19. Lastly, capture two screenshots of the Mechanics Explorer view of the model in an orientation of your choice. DO NOT move the orientation or zoom of the viewer between the screenshots. The screenshots should be taken at 0.0 seconds and 3.0 seconds to show the starting and ending positions.
20. Be sure to also format and save the final block diagram and include it in your report.

To show completion of this task, save your three formatted plots and the two screenshots of the robot at its starting and ending positions, and show the TA.

Post-Lab Questions

From Task 1:

1. View the scope from the first pendulum (dumbbell.STEP). About how many seconds pass before the oscillations die out (<10% of initial magnitude)?
2. View the scope from the second pendulum (dumbbell2.STEP). About how many seconds pass before the oscillations die out?
3. What causes the change in duration for the oscillations between the two pendulums?

From Task 2:

4. Taking in consideration the motion of the robot between its starting and ending positions, what units are being used for angular velocity in this simulation
5. Manipulate any of the parameters within the table below to achieve the following motion path for the tube end-effector:

   <u>Starting Position is aligned with the Z-axis and final position is parallel to the XY-plane.</u>

   Report your results in a table form as shown here: (be aware of units)

	State Position Target (deg)	**Equilibrium Position (deg)**	**Spring Stiffness (units)**	**Damping Coefficient (units)**
Joint 1				
Joint 2				
Joint 3				

   Capture and include a screenshot of the robots starting and final position for this exercise to show successful results with your values.