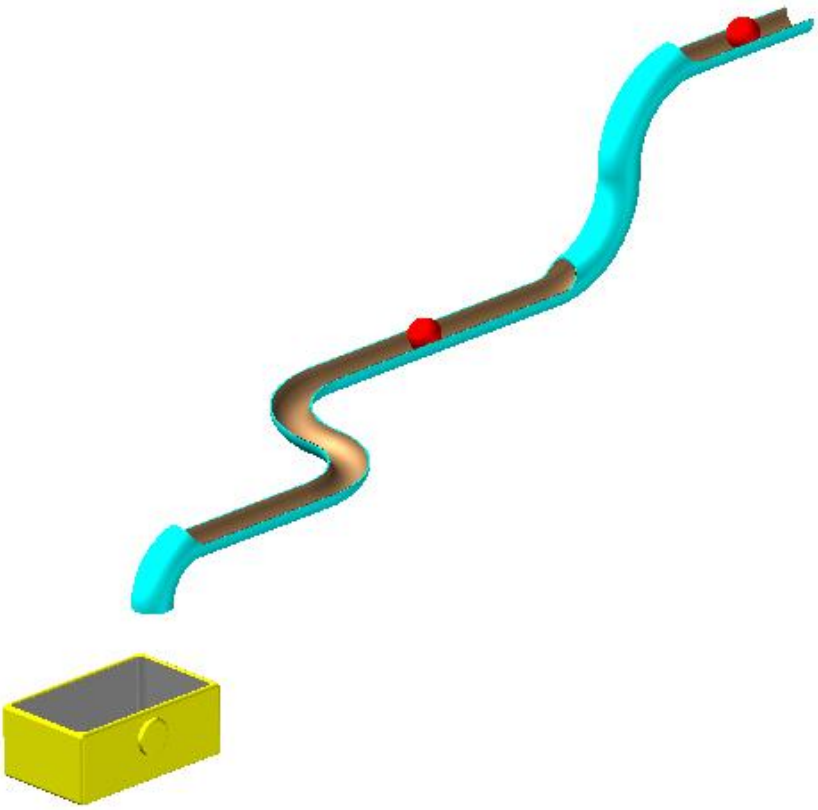
Ball Return Tutorial (Professional)



Edition Note

This document describes the release information of **RecurDyn V9R4**.

Table of Contents

Chapter 1

Getting Started

Objective

The modeling and simulation of contact between bodies are important topics in multibody dynamics. RecurDyn has powerful capabilities to define and simulate 3D contacts. The example in this tutorial is to model a bowling ball return system and the container that collects the balls.

In this tutorial, you learn how to generate 3D contacts with enough detail to be able to model sophisticated contacts on your own. For simple demonstrations, it may seem appealing to simply define complete body-to-body contact. Real models, however, are often too complex to simulate efficiently when every possible contact is considered. RecurDyn offers the powerful capability to select faces to create logical contact surfaces. When you define the contacts, you can then use the logical contact surfaces. Then you can do contact modeling effectively.

This tutorial provides an introduction to 3D contact modeling by using imported geometry. You will learn how to:

- Create multi-face contact surfaces from imported geometry.
- Define 3D contacts between bodies.
- Adjust contact parameters to correctly represent the physical system.
- Determine a load case for use in structural analysis.

You will:

- Simulate the behavior of two balls that travel through a pipe system and drop into a container.
- Study the applied loads to the Container Body from the simulation results

Audience

This tutorial is intended for new users of RecurDyn who previously learned how to create geometry, joints, force entities, and 2D contacts. All new tasks are explained carefully.

Prerequisites

Users should firstly work through the 3D Crank-Slider, Engine with Propeller, and Pinball (2D contact) tutorials, or the equivalent. We assume that you have a basic knowledge of physics.

Procedures

The tutorial is comprised of the following procedures. The estimated time to complete each procedure is shown in the table.

Procedures	Time(Minutes)
Simulation environment set up	1
Geometry import	5
Joint and force definition	10
3D contact definition	5
Analysis and review	5
Contact parameter adjustment	10
Analysis and review	15
Total	51

Estimated Time to Complete

This tutorial takes approximately 51 minutes to complete

Chapter

2

Setting Up Your Simulation Environment

Task Objective

Learn how to set up the simulation environment

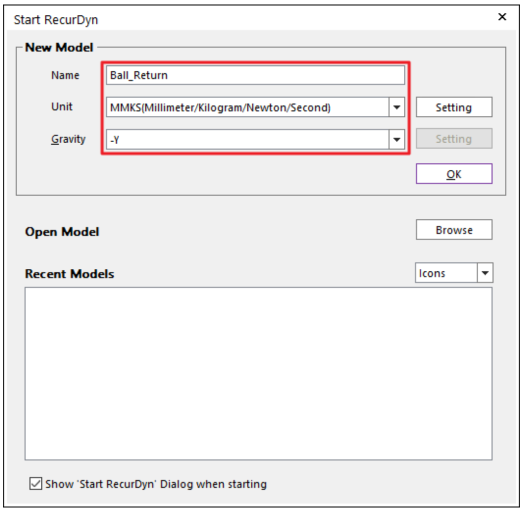
Estimated Time to complete

1 minute

Starting RecurDyn

To start RecurDyn and create a new model:

1. On your Desktop, double-click the **RecurDyn** icon
2. **RecurDyn** starts and the **Start RecurDyn** dialog box appears.
3. Enter the name of the new model as **Ball_Return**.
4. Set **Unit** to **MMKS**.
5. Set **Gravity** to **-Y**.
6. Click **OK**.



Chapter

3

Importing Geometry

You will import solid geometry to define the ball channel and container in this model. You will define contact surfaces using the imported geometry.

Task Objective

Learn to:

- Import solid geometry in Parasolid format.
- Create a contact surface from the imported geometry using the Face Surface tool.

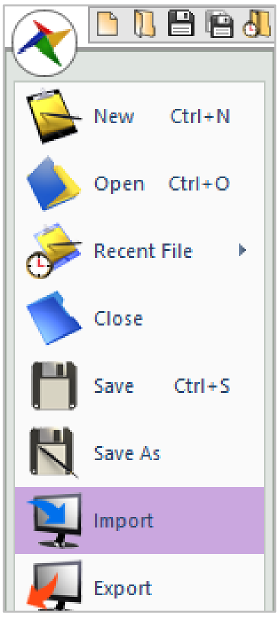
Estimated Time to Complete

5 minutes

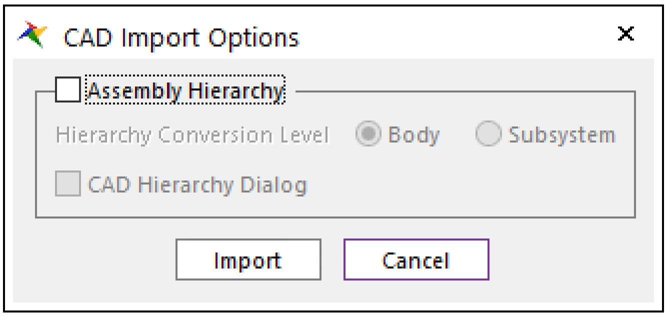
Importing the CAD Geometry

Now you will import the CAD geometry for the piping system and the container. In this case, the geometry was modeled in the CAD system in the correct locations. You do not need to adjust the geometry position.

To import the ball return system geometry:



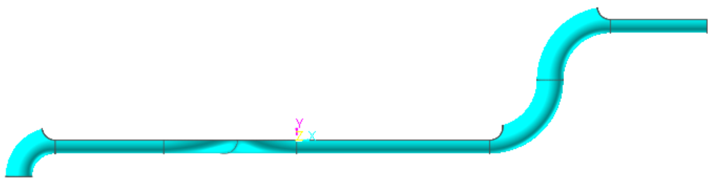
1. From the **File** menu, click **Import**.
2. Set **Files of type** to **ParaSolid File (*.x_t,*.x_b ...)**.
3. In the **Basic Tutorials** folder of the RecurDyn installation, select the file: **Return.x_t (The file location:** <Install Dir> /Help /Tutorial /Professional /BallReturn)
4. Click **Open**. The **CAD Import Options** window appears. Clear the **Assembly Hierarchy** checkbox and click the **Import** button.



5. Click **Import**. The Output window appears and displays a message indicating the success of the import.

Tip: You can toggle **Message** window by clicking **Message** button from **Window** group in the **Home** tab.

Return Pipe is shown in working window and **ImportedBody1** is listed in the Database.



Tip: Changing Model View

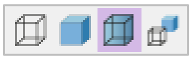
- You can automatically fit the model view to **Working Window** using **View Control Toolbar**.

  - Clicking **Fit** in **View Control Toolbar** will fit the model view to Working Window. (Keyboard shortcut for this function is **'F' Key**.)

  - Clicking **View Snap** in **View Control Toolbar** will change the model view to Plane view or Isometric view according to current perspective.

(Keyboard shortcut for this function is **'G' Key**.)

- You can change the graphic rendering mode by changing **Rendering Mode** in **Render Toolbar**.

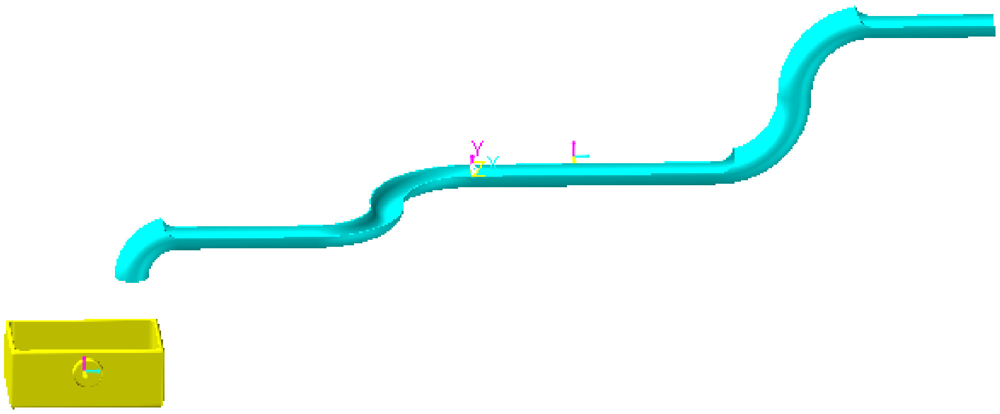


  - **Wireframe**: You can see entities as wire frame.
  - **Shade**: You can see entities as shade.
  - **Shade with Wire**: You can see entities as shade with wire.
  - **Each Render**: You can see entities as specified rendering mode of each entity.

To import the Container geometry:

Follow the instructions above, but this time, select the file, **Container.x_t,** from the **Basics Tutorials** folder of the RecurDyn installation.
(**The file location**: <InstallDir>/Help/Tutorial/Professional/BallReturn)

The geometry representing the container appears in the working window (see the figure below) and the body name **ImportBody2** appears in the Database window.



To set the names of both bodies:

1. Display the **Properties** dialog box for **ImportBody1**.
2. Select the **General** tab and Change **Name** to **ReturnPipe**.
3. Click **OK** to exit the Properties dialog box.

4. Display the **Properties** dialog box for **ImportBody2**.
5. Select the **General** tab and Change **Name** to **Container**
6. Click **OK** to exit the Properties dialog box.

Creating the Ellipsoid Geometry

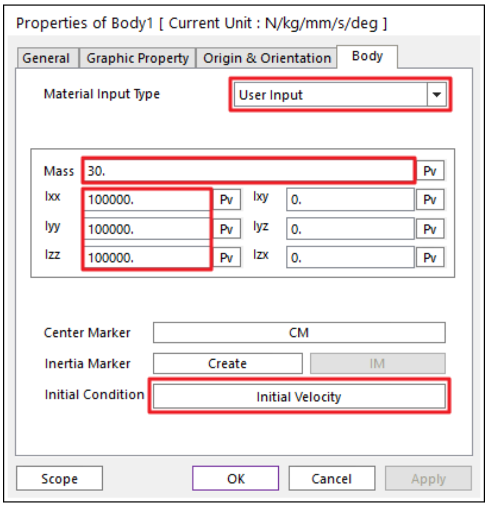
You will create the two bowling balls that are included in this simulation. This process is not only the creation of the ball geometry, but also the setup of the correct mass properties, initial conditions, and names. Ball_1 will be given an initial velocity of 2 m/sec to the left, which provides the energy the initiates the action in the simulation.

To create the ball geometry:

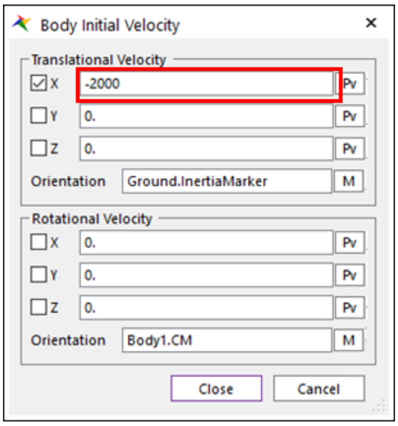
1. From the **Body** group in the Professional tab, click the **Ellipsoid**. Enter the following:
   - **Point**: 3000, 1000, 0
   - **Distance**: 90
2. From the **Body** group in the Professional tab, click the **Ellipsoid**. Enter the following:
   - **Point**: 700, 0, 0
   - **Distance**: 90

To update the ball bodies:

1. Display the Properties dialog box for **Body1**, the first ball you created:
   - In the **General** tab, change **Name** to **Ball_1**.
   - In the **Graphic Property** tab, change **Color** to **Red**.
   - In the **Body** tab, change the values using information below.
     - Change **Material Input Type** to **User Input**.
     - Change **Mass** to **30**.
     - Change **Mass Moment of Inertia (Ixx, Iyy, Izz)** to 100000.

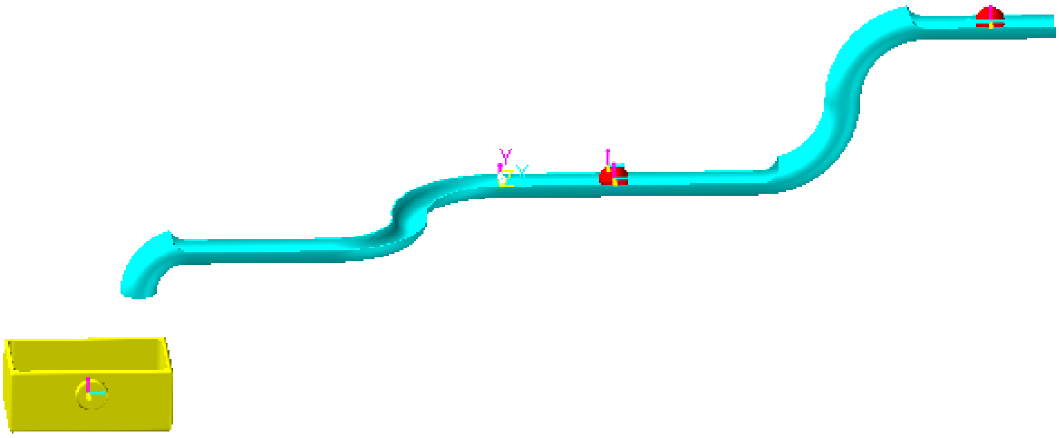


   - Click **Initial Velocity**. And change the values using information below.
     - In the window that appears, select **X Translational Velocity** and set the value of the velocity to **-2000**.
     - Click **Close** to exit the Body Initial Velocity dialog box.
   - Click **OK** to exit the Properties dialog box.



2. Display the Properties dialog box for **Body2**:
   - In the **General** tab, change **Name** to **Ball_2**
   - In the **Graphic Property** tab, change **Color** to **Red**.
   - In the **Body** tab, change the values using information below.
     - Change Material Input Type to User Input.
     - Change **Mass** to **30**.
     - Change Mass Moment of Inertia (Ixx, Iyy, Izz) to 100000.
   - Click **OK** to exit the **Ball_2** Properties dialog box.

Your model appears as shown in the figure below.



Saving the model

From the **File** menu, click **Save.**

Chapter

4

Defining Joints and Forces

In this chapter, you will create all the necessary joints and forces.

Task Objective

You will create:

- A fixed joint to hold the ball return geometry in place.
- A revolute joint to define the pivot point of the container.
- A rotational spring to define the coil spring that controls the rotation of the container as the balls drop into it.

Estimated Time to Complete

10 minutes

Attaching the Return Pipe to Ground

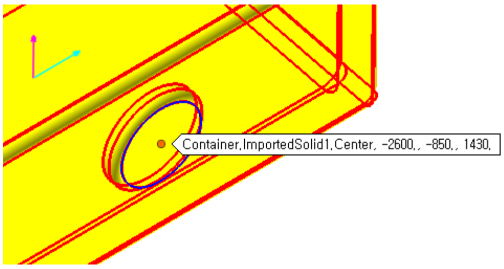
To attach the Return pipe to ground using a fixed joint:

1. Click the **Fixed** Joint, from the **Joint** group in the **Professional** tab.
2. Set the creation method to **Body, Body, Point**.
3. To select **Ground** as **Base Body**, click anywhere in the Working Window where there is no geometry. (Or you can enter Ground in the Command Toolbar)
4. Click the **ReturnPipe** geometry as **Action Body**.
5. Enter **1600, 0, 0** as **Point** in the Command Toolbar.

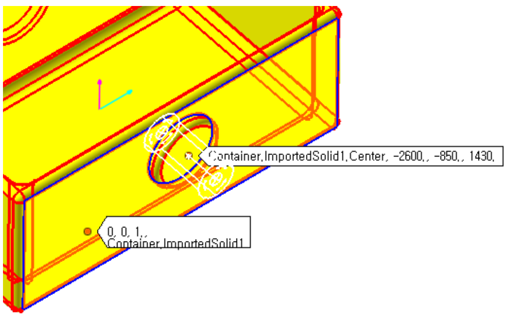
To attach Container to ground using a Revolute Joint:

1. From the **Joint** group in the **Professional** tab, click the **Revolute Joint**
2. Set the Creation Method toolbar to **Body, Body, Point, Direction**.
3. To select **Ground** as **Base Body**, click anywhere in the Working Window where there is no geometry.
4. Click the **Container** geometry as **Action Body**.

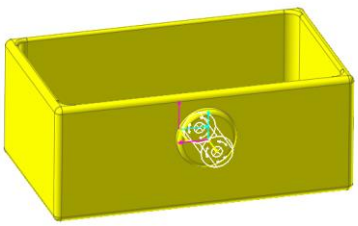
   You will zoom to the **Container** and define **Point** and **Direction** using geometrical information.
5. Move the cursor over to the inner circle of the fillet on the end of the post as shown at right. When the Tooltip indicating **Point** value is (**-2600, -850, 1430)**, click the left mouse button to input **Point**.



6. Move the cursor over to the square Face of the box as shown in right. When the Tooltip indicating, **Normal Vector** is **(0,0,1)**, click the left mouse button to input **Direction**.



RevJoint1 is created as shown below.

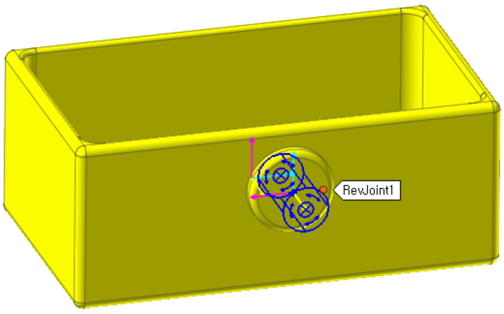


Defining the Rotational Spring

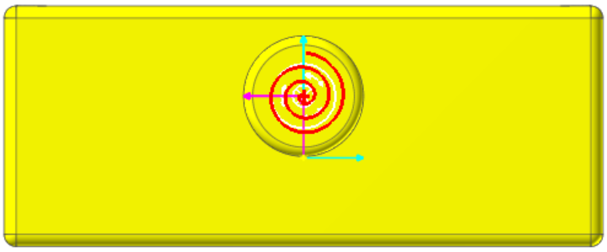
You will create a rotational spring that will control the rotation of the container.

To create the rotational spring:

1. From the **Force** group in the **Professional** tab, click the **Rotational Spring**.
2. Set the Creation Method toolbar to **Joint**.
3. Click the icon of **RevJoint1** that you just created.



RotationalSpring1 is created as shown below.



Chapter
5

Defining 3D Contact

In this chapter, you will create all the contacts between the balls, the balls and ReturnPipe, and the balls and the Container.

Task Objective

You will create contact definitions between:

- Ball_1 and the ReturnPipe
- Ball_2 and the ReturnPipe
- Ball_1 and the Container
- Ball_2 and the Container
- Ball_1 and Ball_2

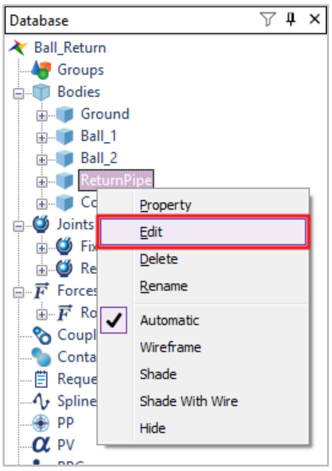
Estimated Time to Complete

10 minutes

Defining Contact between Ball and the ReturnPipe

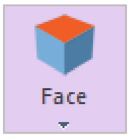
You will define FaceSurface in solid geometry of **ReturnPipe** which is needed to create Contact. And you will define contact between two Balls by using that FaceSurface.

To create the Contact Surface on the Return geometry:



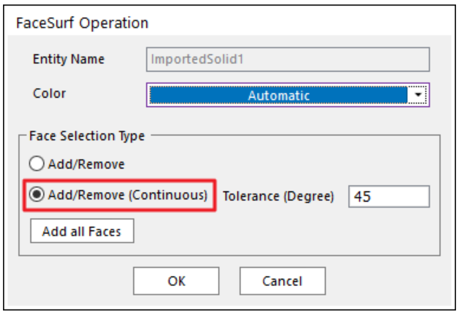
1. Change to **Body-Editing Mode** of the **ReturnPipe** body:
   - In the Database window, right-click **ReturnPipe**.
   - From the menu that appears, click **Edit**.

   The Return piping system is now the only geometry in the working window.

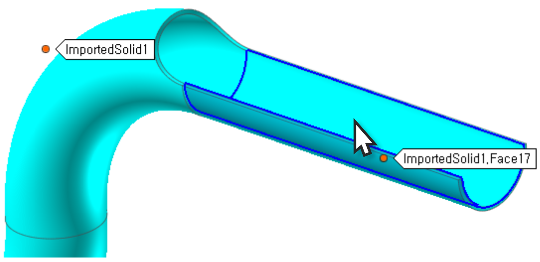


2. From the **Surface** group in the **Geometry** tab, click the **Face.**
3. Set the creation method to **Solid(Sheet), MultiFace.**
4. Click **ImportedSolid1** as **Solid**.

   The **FaceSurf Operation** dialog box appears.

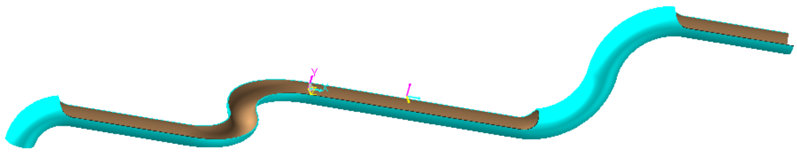


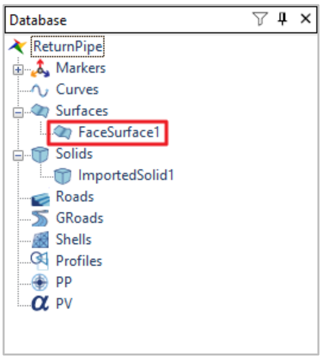
5. Click **Add/Remove (Continuous)** option.
6. Click inner faces of Pipe as shown below.



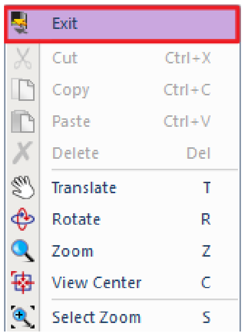
7. When all the inner faces are selected, click **OK** in **FaceSurf Operation** dialog box.

The newly defined contact surface appears as an item in the Database window under the Surfaces category, **FaceSurface1** as shown below.





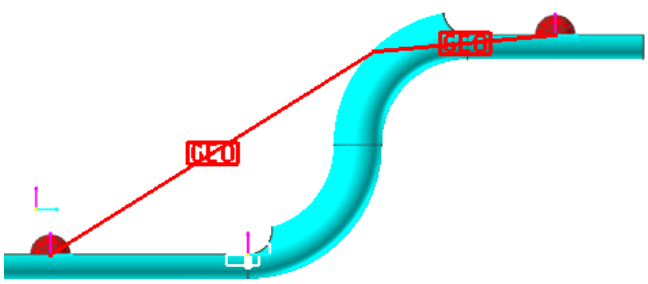
8. Exit the body-editing mode for the Return body by right-clicking the working window, then clicking **Exit**.

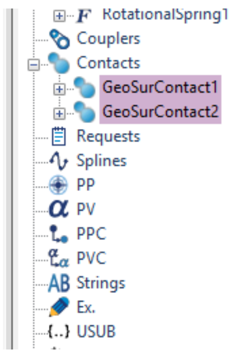


To create the contact between Ball1 and the ReturnPipe

1. Click **GeoSph Contact** from **Contact** group in **Professional** tab.
2. Set creation method to **Surface(PatchSet), MultiSphere**.
3. Select **ReturnPipe.FaceSurface1** as Surface(PatchSet).
4. Select **Ball_1.Ellipsoid** and **Ball_2.Ellipsoid1** as MultiSphere in this order.
5. Right-Click in Working Window and click Finish Operation.

GeoSurContact1 and **GeoSurContact2** is created in Database as shown below.



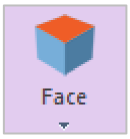


Defining Contact between Ball and the Container.

Same as **ReturnPipe**, you will define FaceSurface in **Container** which is needed to create Contact. And you will define contact between two Balls by using that FaceSurface.

To create contact surface in Container

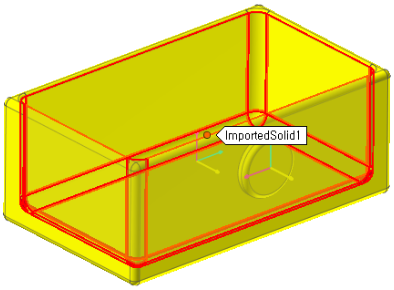
1. Change to **Body-Editing Mode** of the **Container** body:
   - In the Database window, right-click **Container**.
   - From the menu that appears, click **Edit**.



2. Click **Face** from **Surface** group in the **Geometry** tab.
3. Set creation method to **Soild(Sheet), MultiFace**.
4. Click **ImportedSolid1** as **Solid**.
5. Select **Add/Remove** option.
6. In **Select Toolbar**, change **Select State** to **Add**.

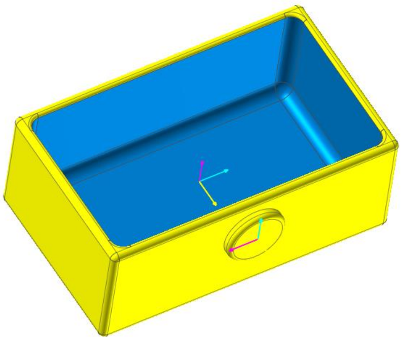
7. Add by clicking inner faces of **Container** as shown at right.

   (If you clicked the wrong faces, change **Select State** to **Delete** and click the wrong faces.)

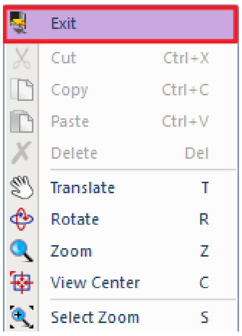


8. When all the inner faces are selected, click **OK** in the dialog box.

   FaceSurface1 is created as shown below.



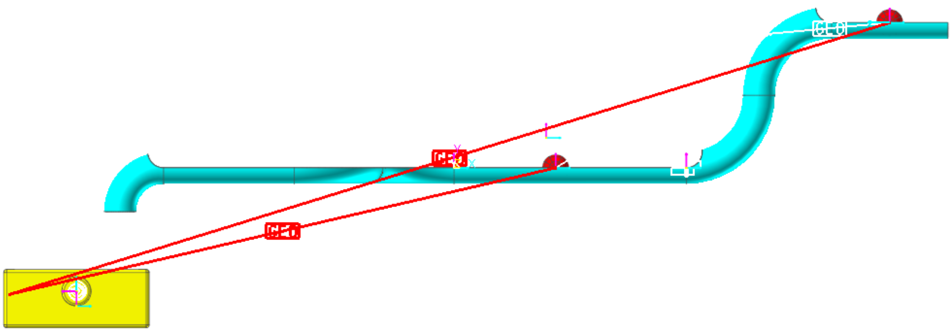
9. Exit the body-editing mode for the Return body by right-clicking the working window, then clicking **Exit**.



To create Contact between Ball and Container.

1. Click **GeoSph Contact** from the **Contact** group in the **Professional** tab.
2. Set creation method to **Surface(PatchSet), MultiSphere**.
3. Select **Container.FaceSurface1** as Surface(PatchSet).
4. Select **Ball_1.Ellipsoid1** and **Ball_2.Ellipsoid1** as MultiSphere in this order.
5. Right-Click in Working Window and click **Finish Operation**.

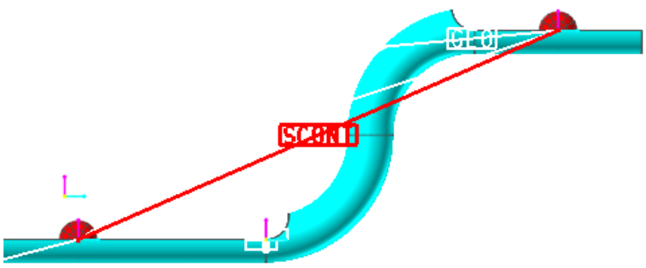
   GeoSurContact3 and **GeoSurContact4** is created in Database as shown below.



Defining Contact between Ball_1 and Ball_2

To create the contact between the balls:

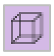
- Click the **Sph-Sph** from the **Contact** group in the **Professional** tab and click the following:
  - **Sphere**: Ball_1.Ellipsoid1
  - **Sphere**: Ball_2.Ellipsoid1



Adjust Contact Surface Resolution of Return

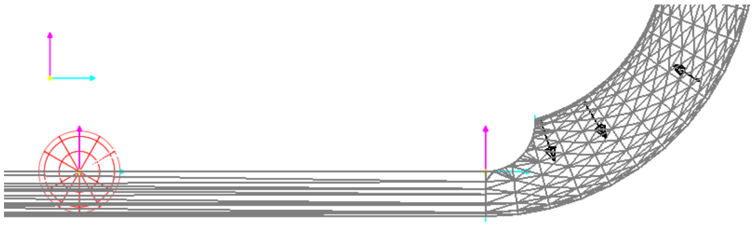
You will adjust the Faceting Resolution of contact surface of GeoSphere Contact for more accurate simulation results.

To adjust Faceting Resolution of GeoSurContact1:



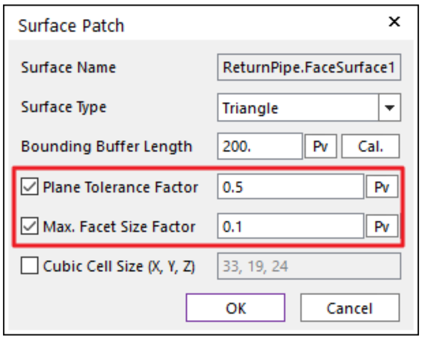
1. Change **Rendering Mode** to **Wireframe** in **Render Toolbar** to see the faceting status of contact surface more clearly.
2. Open Property dialog of **GeoSurContact1**.

   Faceting status of **ReturnPipe.FaceSurface1** is shown as picture below.



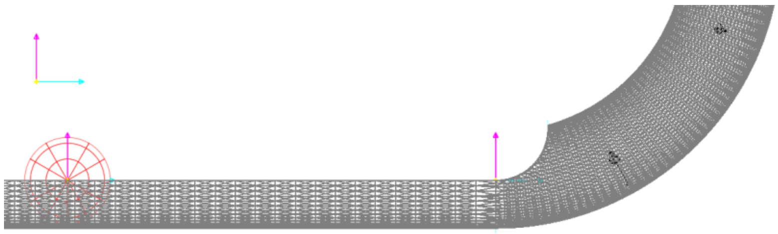
3. Click **Contact Geometry** at **Base Geometry** in **Geo Contact** tab.

   Surface Patch dialog is shown.



4. Change **Plane Tolerance Factor** to **0.5**
5. Check the **Max. Facet Size Factor** option and change it to **0.1**.
6. Click **OK** in Surface Patch dialog.

   You can see that facet size has become more detailed as shown below.



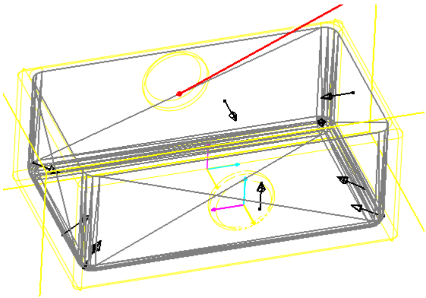
7. Click **OK** in **GeoSurContact1** dialog.

You don't need to change contact surface of **GeoSurContact2** because it is defined as same geometry of **GeoSurContact1**.

To adjust Faceting Resolution of GeoSurContact3:

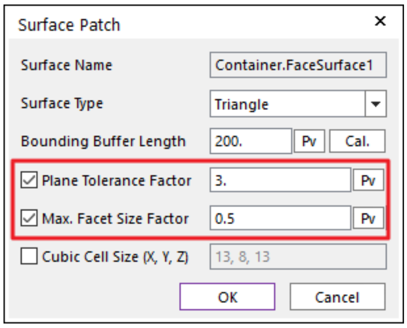
1. Open the Property dialog of GeoSurContact3.

   Faceting status of **Container.FaceSurface1** is shown as picture below.



2. Click **Contact Geometry** at **Base Geometry** in **Geo Contact** tab.

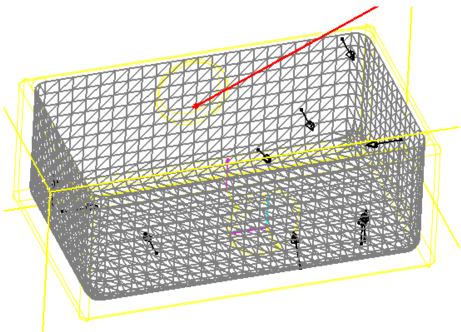
   Surface Patch dialog is shown.



3. Check the **Max. Facet Size Factor** option and change it to **0.5**

4. Click **OK** in Surface Patch dialog.

   You can see that facet size has become more detailed as shown below.



5. Click **OK** in **GeoSurContact3** dialog.

   You don't need to change contact surface of **GeoSurContact4** because it is defined as same geometry of **GeoSurContact3**.

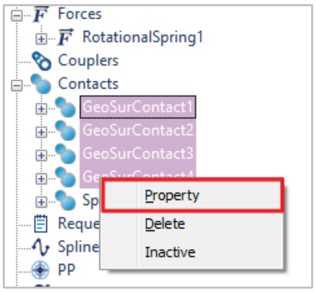
6. Change **Rendering Mode** to **Shade With Wire** in **Render Toolbar**.

To modify the property of GeoSurContact

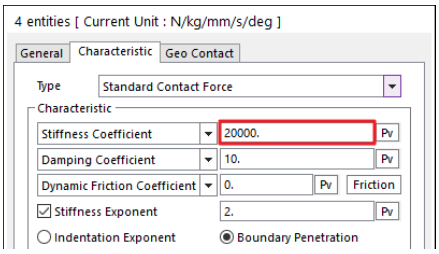
You will modify the value about the contact property between ReturnPipe and Ball.

1. From **GeoSurContact1** to **GeoSurContact4**, select 4 Contacts from Database. Then right-click holding **Ctrl Key** to click **Property**.

   Multi-Property dialog is shown which can change the properties of selected entities.

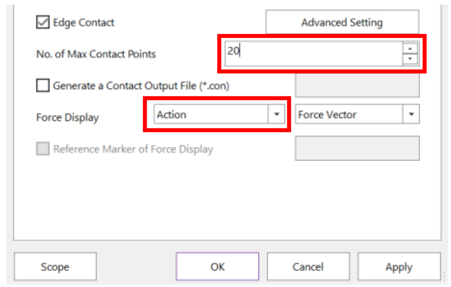


2. In **Characteristic** tab, change the values using information below.
   - **Stiffness Coefficient**: 20000



You will set the options about **Force Display** to visually check the contact force of **ReturnPipe** and **Two Ball** in animation.

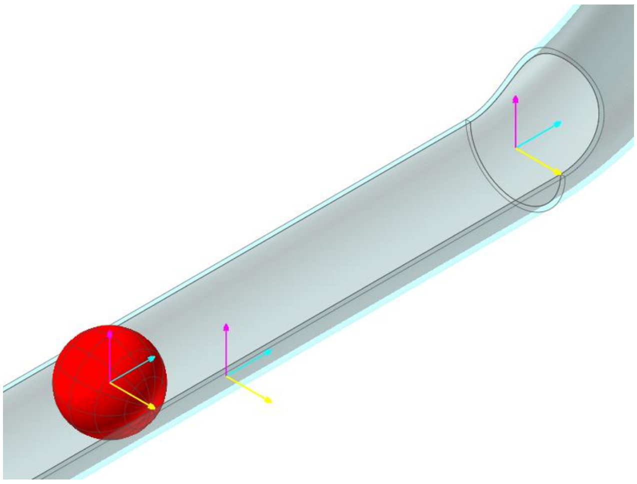
3. Open Property dialog of GeoSurContact2.
4. In **Geo Contact** tab, change the values using information below.
   - **No. of Max Contact Points**: 20
   - **Force Display**: Action
5. Click **OK**.

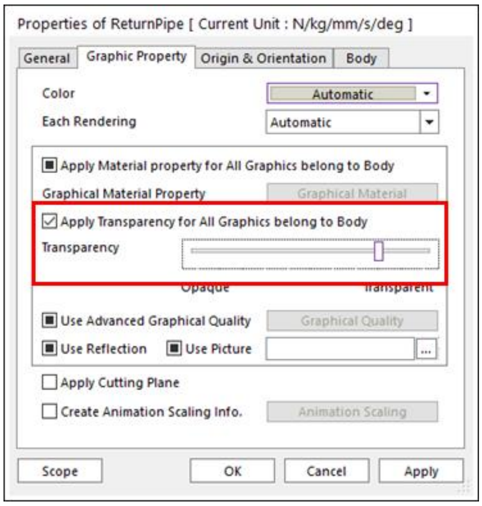


To modify the graphic property of ReturnPipe

You will adjust graphic transparency of **ReturnPipe** to visually check the contact force of **ReturnPipe** and **Ball_2** in animation.

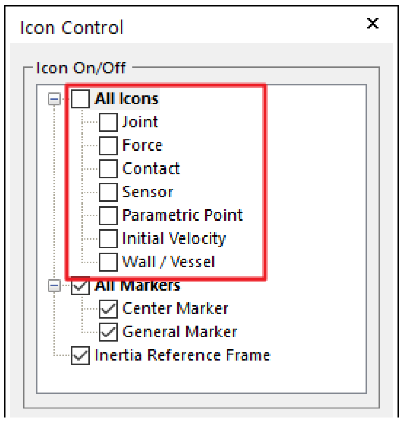
1. Open Property dialog of **ReturnPipe**.
2. In **Graphic Property** tab, check **Apply Transparency for All Graphics belong to Body** and move **Transparent Level** slider to right as shown below.
3. Click **OK**.





To hide the icon

1. In the **View Control Toolbar**, click **Icon Control**.
2. Uncheck **All Icons** to disappear the icons on Working Window.
3. Close **Icon Control** window



Saving the Model

Take a moment to save your model before you continue with the next chapter.

Chapter

6

Analyzing and Reviewing the Model

In this chapter, you will run a simulation of the model you just created. You will examine the results and think about what to change to make sure the results are correct.

Task Objective

Learn to use the more advanced aspects of plotting, including playing animations with the plots and using the **Plot Template** and **Trace Data** tool.

Estimated Time to Complete

5 minutes

Performing Dynamic/Kinematic Analysis

In this section, you will run a dynamic/kinematic analysis to view the effect of forces and motion on the model you have just created.

To change options about Simulation Output.

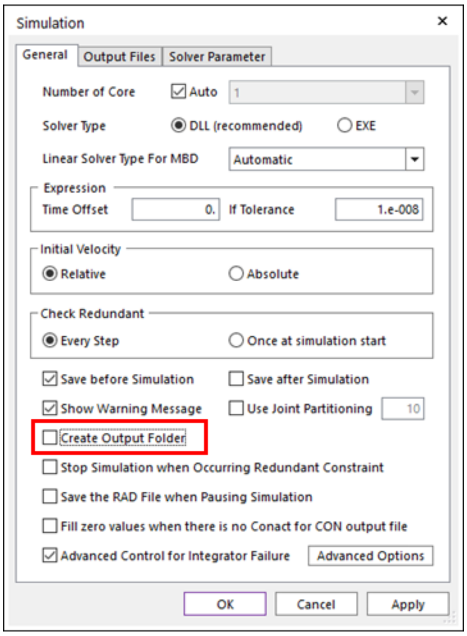
To use **Output File Name** function efficiently, the **Create Output Folder** option is turned off.

1. From the **Model Settings** group in the **Home** tab, click **Simulation**.

   The Simulation dialog box will appear.

2. Select the **General** tab. Make sure the **Create Output Folder** check box is not checked.

3. Click **OK**



To perform a Dynamic/Kinematic analysis:

1. Click the **Dyn/Kin**. from the **Simulation Type** group in the **Analysis** tab.

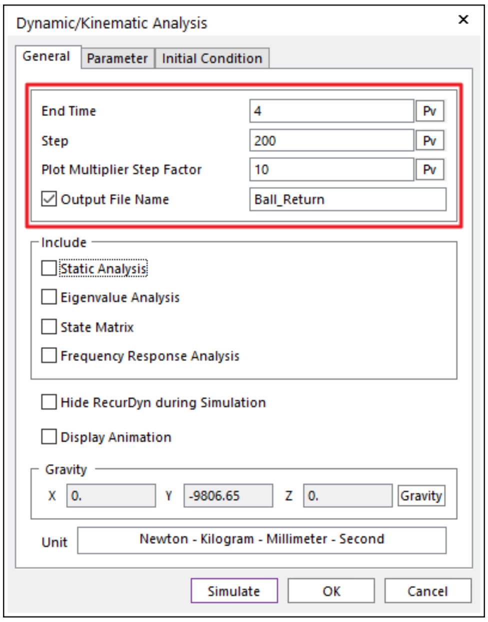
   The Dynamic/Kinematic Analysis dialog box appears.

2. Define the **End Time** of the simulation and the number of **Steps**:

   - **End Time**: 4
   - **Step**: 200
   - **Plot Multiplier Step Factor**: 10

3. Select **Output File Name**, and then enter the output file name: **Ball_Return**

4. Click **Simulate**.

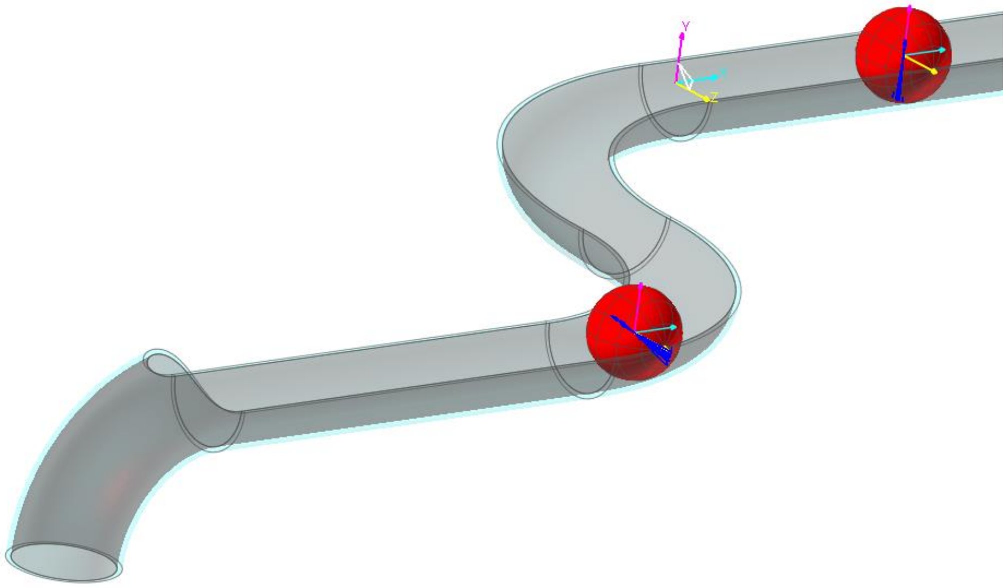


Examine the result.

To examine the animation result.

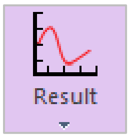
- Click **Play** from the **Animation Control** group in the **Analysis** tab to play animation.

  Ball_1 (in the right) moves to the left and affects **Ball_2**. **Ball_2** moves along the path in **ReturnPipe** and fall into the **Container**.



Overall behavior seems appropriate. However, if you move the **Animation** step by step and observe **Ball_2**, it doesn't roll because **ReturnPipe** and **Ball_2** don't have friction force between them.

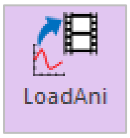
To examine the Plot result.



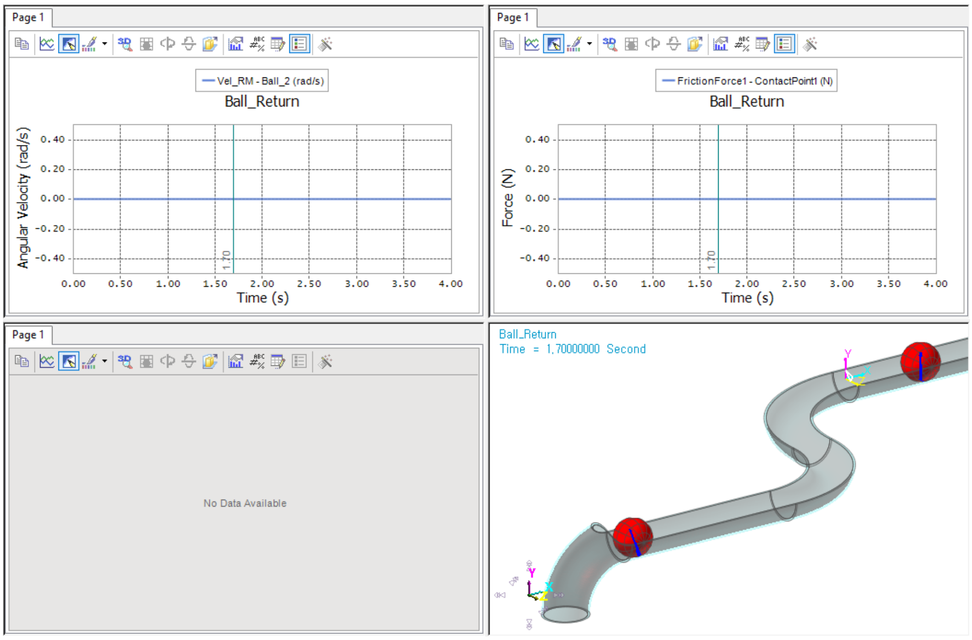
1. Click **Result** from the **Plot** group in the **Analysis** tab.

   Working window changes to Plot Window.

2. Click the **All** from the **Windows** group in the **Home** tab to display four plotting windows.
3. Click the **Upper left** window to make that pane active and double click **Vel_RM** in the Database following path below.
   - **Ball_Return → Bodies → Ball_2 → Vel_RM**
4. Click the **upper right** window to make that pane active and double click **FrictionForce1** in the Database following path below.
   - **Ball_Return → Contact → Geo Contact → GeoSurContact2 → ContactPoints → ContactPoint1 → FrictionForce1**



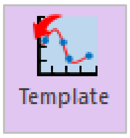
5. Click the **lower right** window to make that pane active and click **LoadAni** from **Animation** group in the **Tool** tab.
6. Click **Yes** when the warning message appears.
7. Click **Play** button and check the animation.



As we already checked, **Ball_2** doesn't roll because **ReturnPipe** and **Ball_2** don't have friction force between them. For more realistic results, you will modify the friction information in the Contact between **ReturnPipe** and **Ball_2**.

To create Plot Template

You will create Plot Template to reduce the inconvenience of redrawing plot results after each simulation.

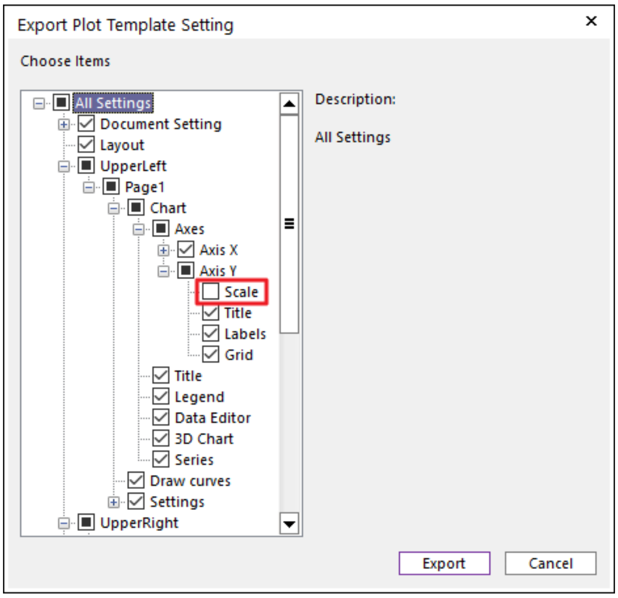


1. Click **Template** from the **Export** group in the **Home** tab.

   The **Export Plot Template Settings** dialog is shown. The dialog shows the list of properties of drawn plots.

2. Exclude below items from the list.
   - **UpperLeft → Page1 → Chart → Axes → Axis Y → Scale**
   - **UpperRight → Page1 → Chart → Axes → Axis Y → Scale**

   (If you don't exclude Scale of Axis Y, it won't automatically scale when drawing the next plot.)



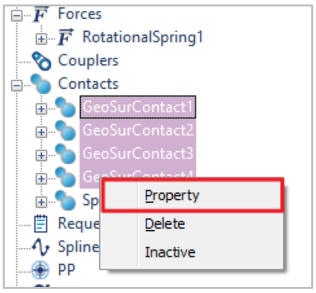
3. Click **Export** to save Template file.

Running a New Simulation

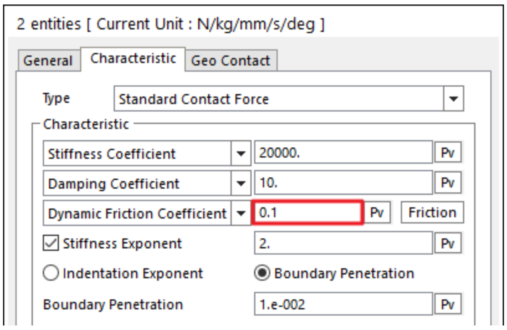
To define the friction value for the contact between Ball and ReturnPipe.

1. Select **GeoSurContact1** and **GeoSurContact2** from Database. Then right-click holding **Ctrl Key** to click **Property**.

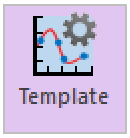
   Multi-Property dialog is shown which can change the properties of selected entities.



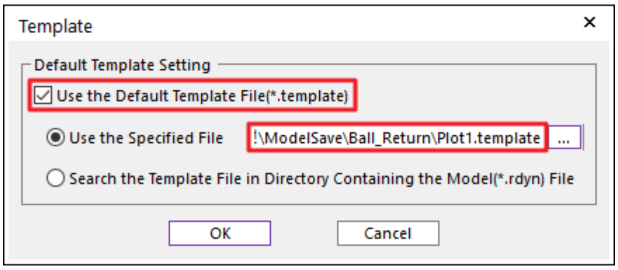
2. In **Characteristic** tab, change the values using information below.
   - **Dynamic Friction Coefficient**: 0.1
3. Click **OK.**



Connect Plot Template file before running a new simulation.



1. Click **Template** from the **Plot** group in the **Analysis** tab.
2. Check **Use the Default Template File(*.template)** when the Template dialog is shown.
3. Select **Use the Specified File** and enter the file path where you saved the *.template file.
4. Click **OK.**

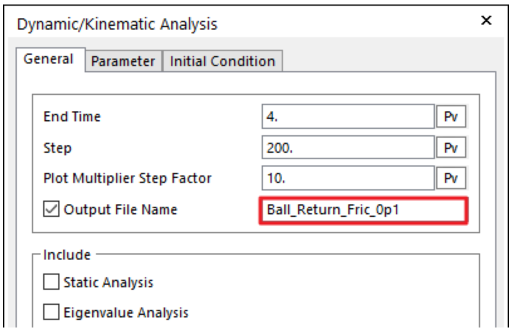


To perform a dynamic/kinematic analysis:

1. Click the **Dyn/Kin** from the **Simulation Type** in the **Analysis** tab.

   The Dynamic/Kinematic Analysis dialog box appears.

2. Change the file name: **Ball_Return_Fric_0p1**

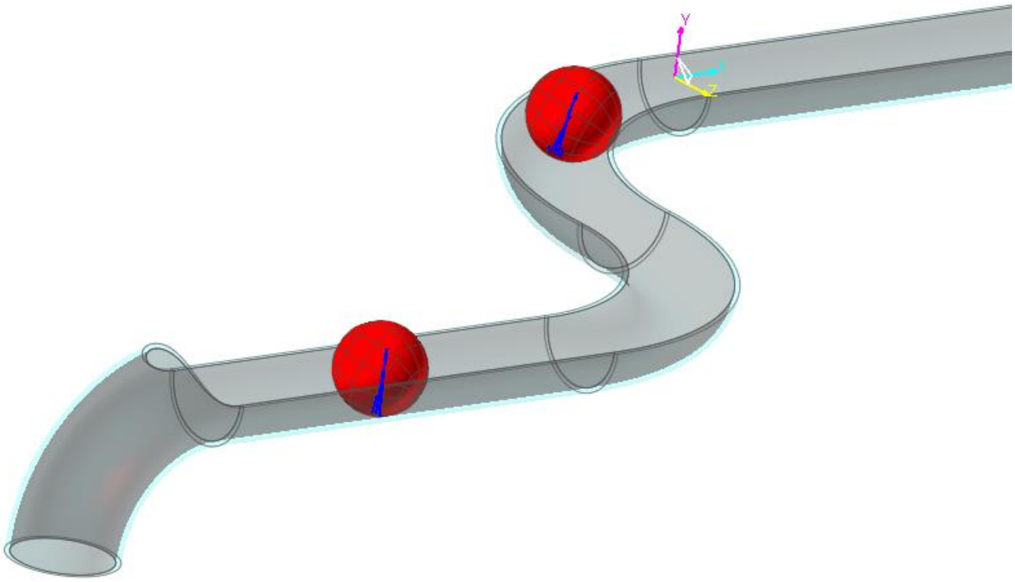


3. Click **Simulate**.

To examine the animation result

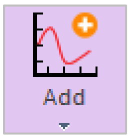
- Click **Play** from the **Animation Control** group in the **Analysis** tab.

  If you move the **Animation** step by step and observe **Ball_2**, it rolls because **ReturnPipe** and **Ball_2** now have friction force between them.



Comparing the Results of the Two Simulations

To compare the result from previous analysis using Plot Add.

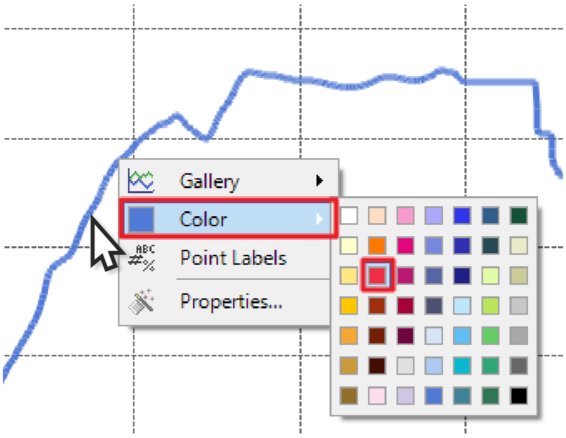


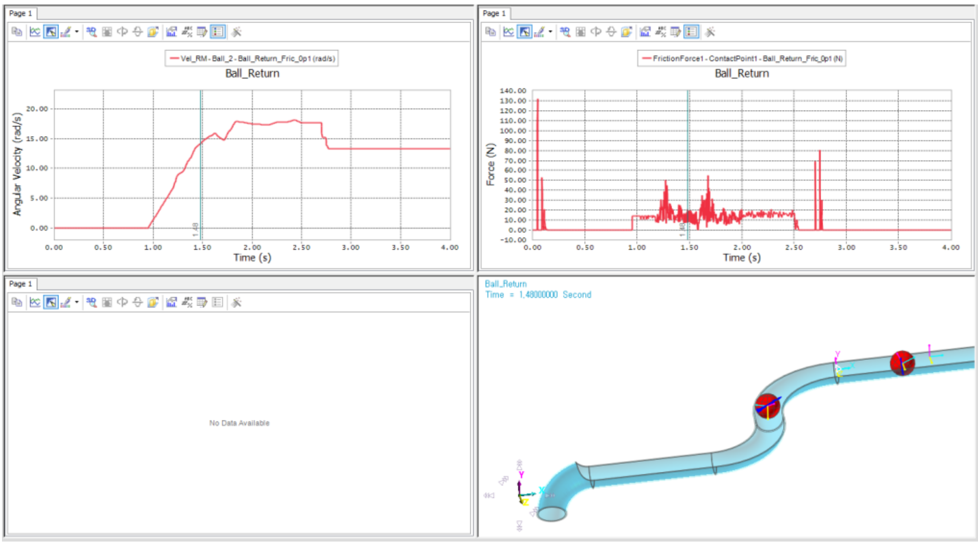
1. Click **Add** from the **Plot** group in the **Analysis** tab.

   Previous Plot Window opens again and plot from new simulation result is automatically drawn. You will change the color of graph to separate the overlapped two graphs.

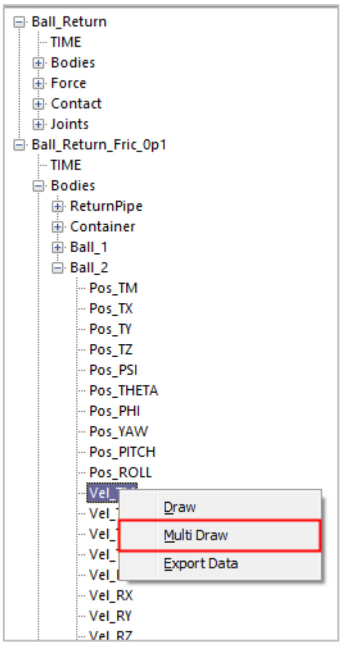
2. In the **Plot Window**, select the newly drawn graph and right-click it.

3. Change the **Color** to **Red** when the Pop-up menu appears.





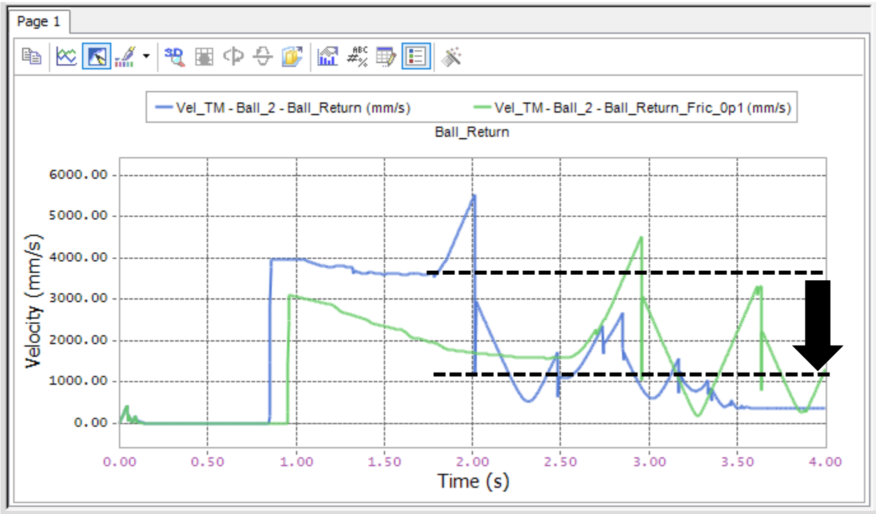
If you compare the two plot results, you can check that Contact Friction Force is generated between **Ball_2** and **ReturnPipe**. Also, you can check that Angular Speed is generated from Contact Friction Force.



To draw the graph simultaneously using Multi Draw.

1. Click the **lower left** window.
2. Unfold to see **Vel_TM** in the Database following path below.
   - **Ball_Return → Bodies → Ball_2 → Vel_TM**
3. Right-click it and click **Multi Draw** in the pop-up menu.

   Graph about **Ball_Return** and **Ball_Return_Fric_0p1** are simultaneously drawn.

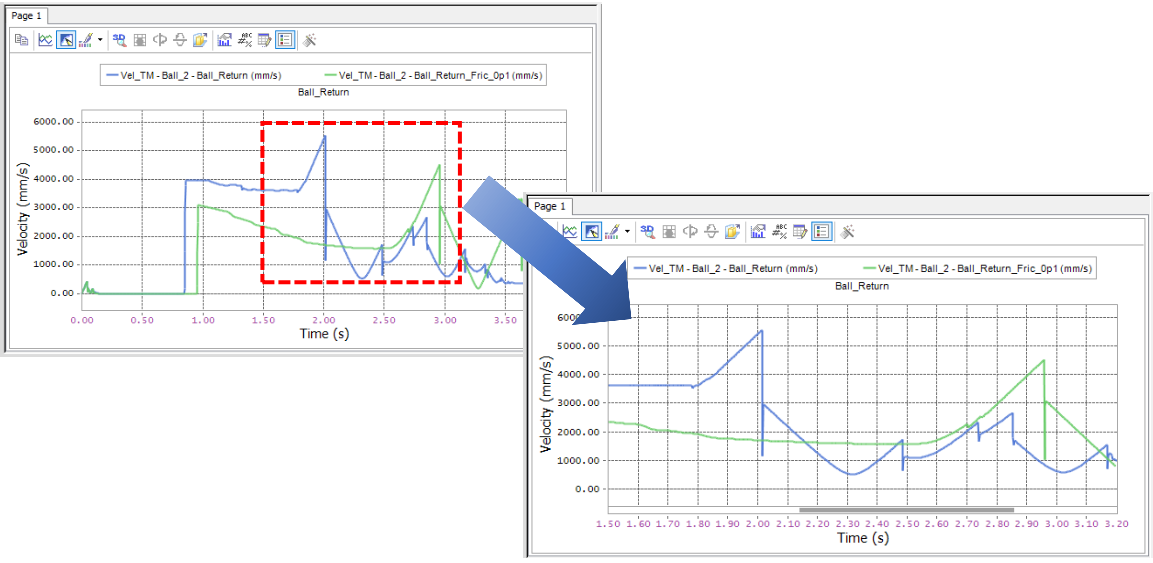


If you compare the two results, you can see that the velocity of **Ball_2** has decreased when it moves in the **ReturnPipe** because of friction force.

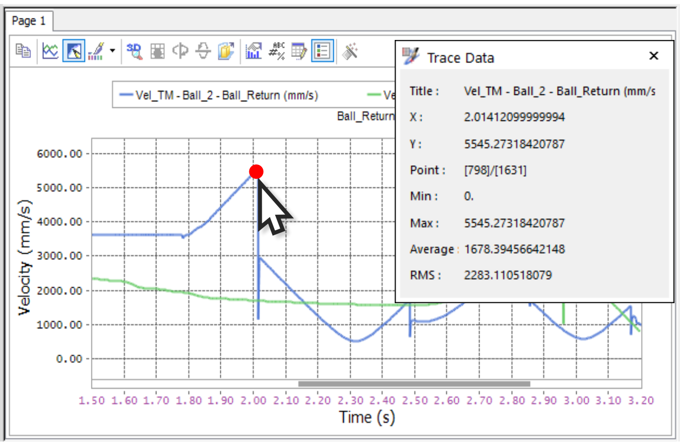
To find the peak value using Trace Data:

1. Click **Zoom** from the **View** group in the **Home** tab.
2. In the Plot Window where the **Vel_TM** graph is, drag the area where the peak value is located as shown below

   You can see the area you dragged magnified as shown below. The highest value of **Vel_TM** graph is where the **Ball_2** free-falls.



3. From the **Tools** group in the **Home** tab, click **Trace**.
4. Place the cursor at the peak value of the graph. Data Display appears as shown below.
   - The first row contains the animation frame number.
   - The second row contains the **X** value (time).
   - The third row contains the **Y** value (velocity).



Thanks for participating in this tutorial!