Pinball 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 all types of contacts, from simple to complex and with body geometry created in RecurDyn, as well as geometry that is imported from CAD software. Consideration of contacts is needed to model designs that have interesting responses to model changes.

In this tutorial, you'll act as a company developing a novel pinball machine that includes a higher level of vertical motion. One aspect of the model is that the ball goes up and down a curved ramp as it is propelled from its starting point. The purpose of the tutorial is to select the spring that can store sufficient energy to propel the ball over the vertical obstacle.

This tutorial provides the first exposure to the modeling. You will learn about below.

- Create geometry.
- Define contacts between bodies.
- Define a parametric value.
- Run a design study.

You will also learn below.

- Simulate a small portion of a pinball game, where balls contact with each other and guides that act as boundaries.
- Study the relationship between the driving force of the ball launcher and the response of the system.

Audience

This tutorial is intended for new users of RecurDyn. All new tasks are explained carefully.

Prerequisites

Users should firstly work through the 3D Crank-Slider Tutorial and the Engine with Propeller Tutorial, 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)
Setting Up Your Simulation Environment	5
Creating Geometry	5
Creating Force and Contact	15
Creating Expression Scope and Performing Analysis	10
Performing a Design Study	30
Total	65

Estimated Time to Complete

65 minutes

Chapter
2

Setting Up Your Simulation

Task Objective

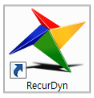
Learn how to set up the simulation environment, including units, materials, gravity, and the working plane.

Estimated Time to Complete

5 minutes

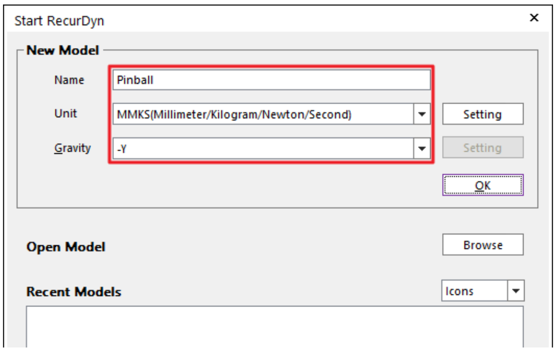
Starting RecurDyn

To start RecurDyn and create a new model



1. On your Desktop, double-click the **RecurDyn** icon.

   RecurDyn starts and the **Start RecurDyn** window appears.
2. Enter the name of the new model as **Pinball**.
3. Change **Unit** to **MMKS**.
4. Change **Gravity** to **-Y**.
5. Click **OK**.



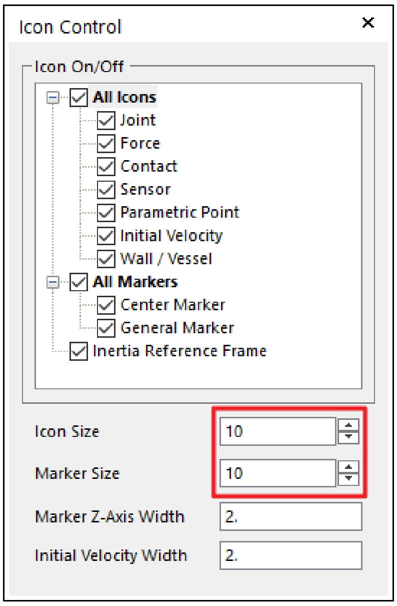
Adjusting the Icon and Marker Size

You are now going to change the icon and marker size to 10 pixels so you can view the model better.

To change the icon and marker size

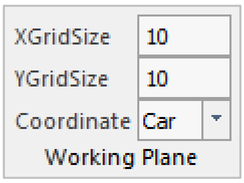
1. In the **View Control Toolbar**, click **Icon Control**.

   The Icon Control window appears.
2. Set **Icon Size** and **Marker Size** to **10**.
3. Close **Icon Control** window.



To set the grid size to 10

From the **Working Plane** group in the **Home** tab, set the grid size to **10** in both text boxes by placing the cursor in each text box, and then pressing the **Enter** key.



Chapter

3

Creating Geometry

You will use wireframe geometry to define the ball guides in this pinball model. You will define all of the guides on the ground body and define several balls at the model level as individual bodies.

Task Objective

Learn to create:

- Line and arc geometry that will guide the motion of the balls.
- Spherical geometry that represents the three balls in the model.
- Circle geometry inside ball for 2D Contact between circle and guide.

Estimated Time to Complete

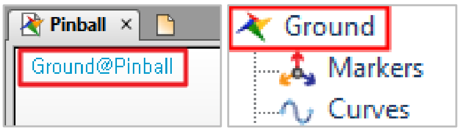
10 minutes

Creating the Guide Geometry

To create the straight guide geometry

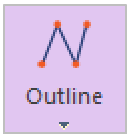
1. To enter Body Editing mode for the ground body, from the **Body** group in the **Professional** tab, click **Ground**.

   You know that RecurDyn is in Body Editing Mode for the Ground body because

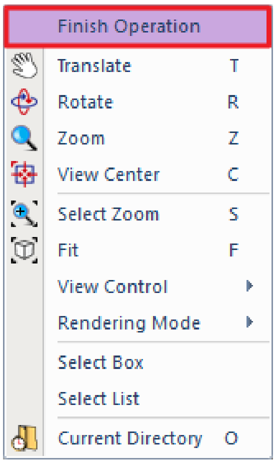
   

   - The title in the upper left corner of the working model window changes to **Ground@Pinball**.
   - The top item in the database window is **Ground**.

2. Change the working plane to the **XY Plane**.



3. From the **Curve** group in the **Ground** tab, click **Outline**.
4. Set the Creation Method toolbar to **MultiPoint**.
   - **Point 1**: 0, 0, 0
   - **Point 2**: 110, 0, 0

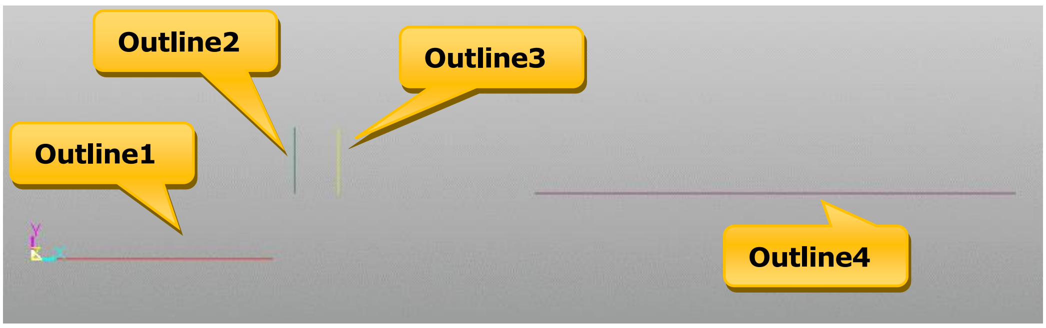


5. Right-click and then click **Finish Operation** (the top item in the menu that appears) to finish the definition of the outline.
6. Repeat **Steps 3-5** three times using following information
   - **Point 1**: 120, 30, 0
   - **Point 2**: 120, 60, 0

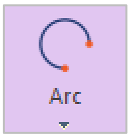
   - **Point 1**: 140, 30, 0
   - **Point 2**: 140, 60, 0

   - **Point 1**: 230, 30, 0
   - **Point 2**: 450, 30, 0

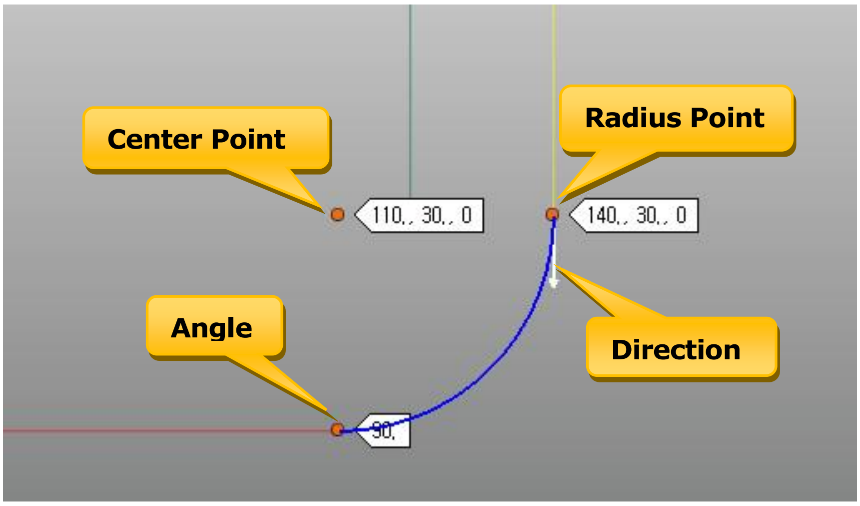
The following appears in the working model window.



To create the arc guide geometry



1. From the **Curve** group in the **Ground** tab, click the **Arc**.
2. Set the Creation Method toolbar to **Point, Point, Direction, Angle**.
    - **Center Point**: 110, 30, 0
    - **Radius Point**: 140, 30, 0
    - **Direction**: 0, -1, 0
    - **Angle**: 90

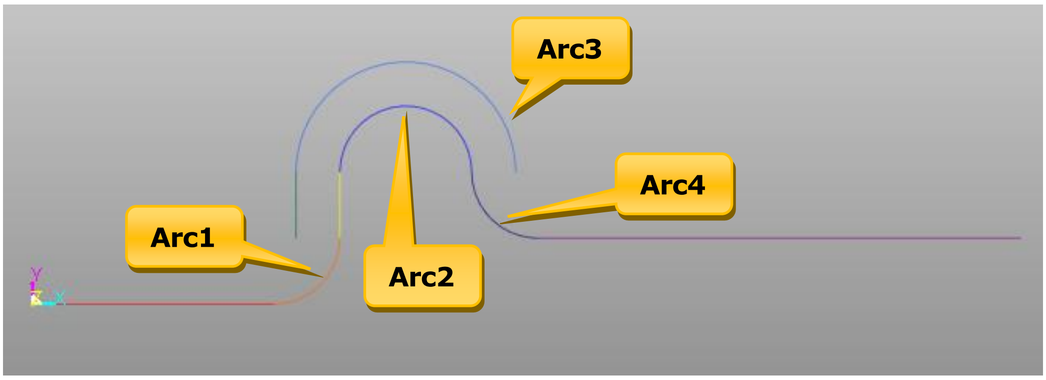


3. Repeat **Steps 1-2** three times using following information
    - **Center Point**: 170, 60, 0
    - **Radius Point**: 140, 60, 0
    - **Direction**: 0, 1, 0
    - **Angle**: 180

    - **Center Point**: 170, 60, 0
    - **Radius Point**: 120, 60, 0
    - **Direction**: 0, 1, 0
    - **Angle**: 180

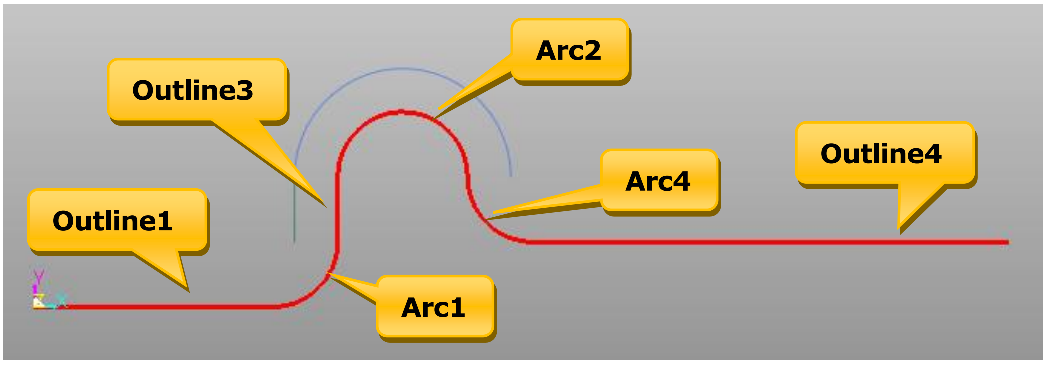
    - **Center Point**: 230, 60, 0
    - **Radius Point**: 200, 60, 0
    - **Direction**: 0, -1, 0
    - **Angle**: 90

The geometry appears as shown in the figure below.

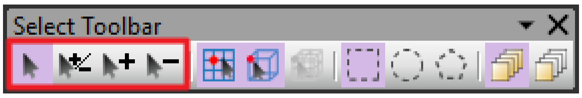


To create EdgeCurve geometry with existing curves

1. From the **Curve** group in the **Ground** Tab, click **Edge**.
2. Set the Creation Method to **MultiEdge**.
3. Use the **Select Toolbar** to curves except **Outline2** and **Arc3** as shown in the figure below.



Tip: Use Select Toolbar to easily select Multi Edges

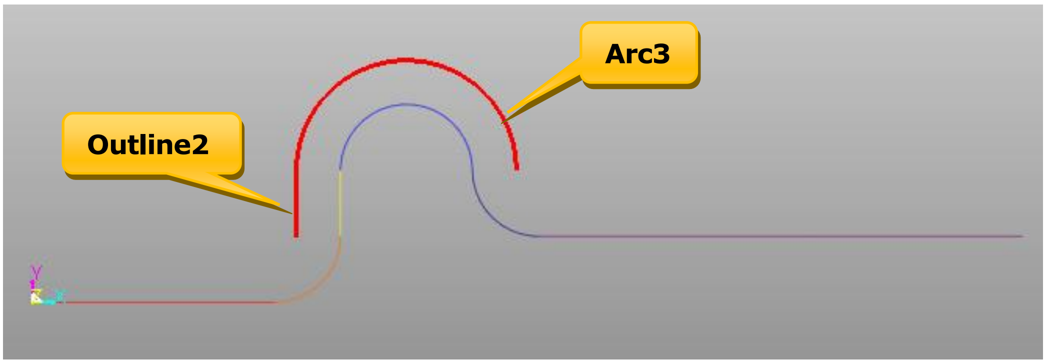


You can change **Select State** of mouse cursor with **Select Toolbar**.

- **Select(Default)**: It clears already selected entities when selecting new entities.
- **Add or Remove**: Reverse Select State of already selected entities. When selecting entities, it deselects already selected entities and selects deselected entities.
- **Add**: It keeps already selected entities and can only add other entity newly.
- **Remove**: It only deselect selected entities.

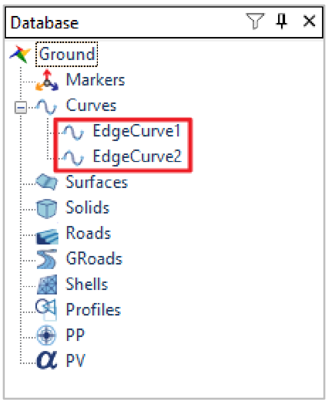
* Other functions in **Select Toolbar** will vary with different Edit Modes.

4. When you select all the curves in lower part, right-click on working window and click **Finish Operation**.
5. From the **Curve** group in the **Ground** Tab, click **Edge** again.
6. Set the Creation Method to **MultiEdge**.
7. Use the **Select Toolbar** to **Outline2** and **Arc3** as shown in the figure below

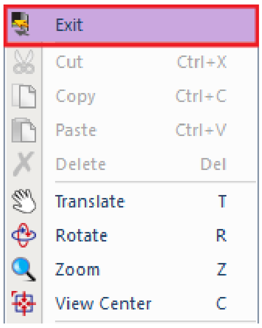


8. When you select all the curves in lower part, right-click on working window and click **Finish Operation**.

9. Delete all other curves except **EdgeCurve1** and **EdgeCurve2**.



10. Click **Exit** to exit the body edit mode. (**Exit** button is in Ribbon Menu, it is also in right-click menu when you right-click on Working Window.)

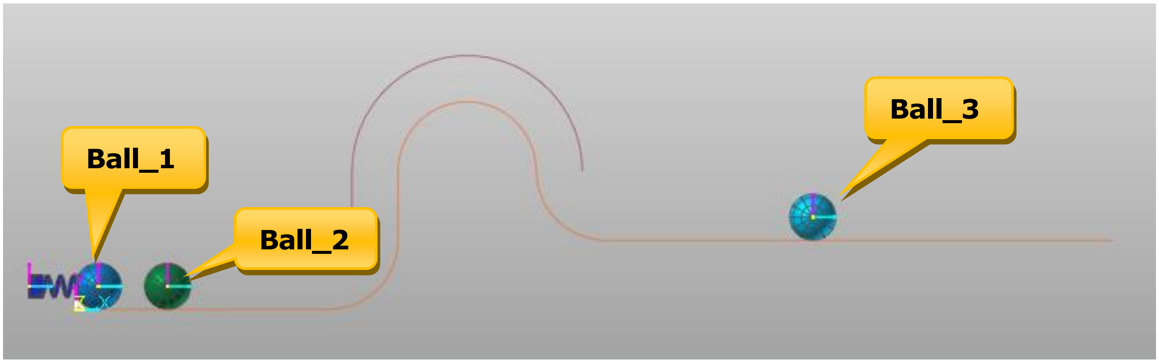


Creating the Ball Geometry

To create the Ball geometry and Circle Geometry

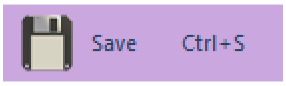
1. From the **Marker and Body** group in the **Professional** tab, click **Ellipsoid**.
2. Set the Creation Method toolbar to **Point, Distance.**
   - **Point**: 10, 10, 0
   - **Distance**: 10
3. Open the properties dialog and rename the **Body1** to **Ball_1**.
4. Repeat **Steps 1-3** twice using following information.
   - **Point**: 40, 10, 0
   - **Distance**: 10
   - **Body Name**: Ball_2

   - **Point**: 320, 40, 0
   - **Distance**: 10
   - **Body Name**: Ball_3



Saving the Model

Take a moment to save your model before you continue with the next chapter. (**Tip**: From the **File** menu, click **Save**.)



Chapter 4

Creating Force and Contact

The behavior of this model is driven by forces. A compressed spring drives Ball_1 into Ball_2. The continuing motion of the balls results from gravity forces, contacts between balls, and contacts between the balls and the geometry.

Task Objective

Learn to create three types of force elements:

- Compressed spring that will act on Ball_1
- Sphere To Sphere contacts between the balls
- Geo Curve contacts between the balls and the Guides.

Estimated Time to Complete

20 minutes

Defining the Compressed Spring

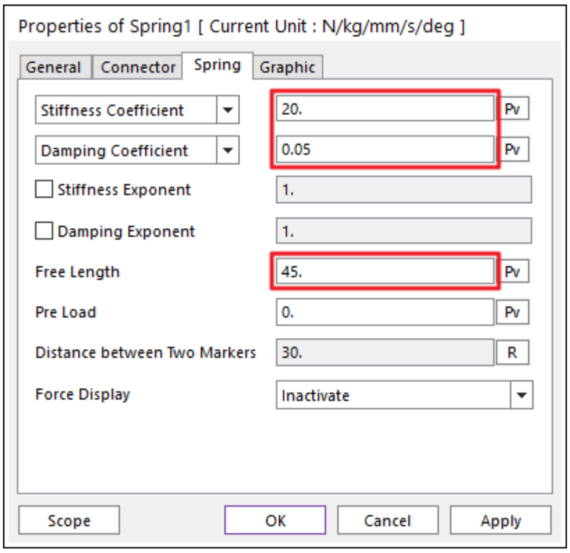
You will create a spring force and adjust its properties to reflect that it is a compressed spring. As a result, Ball_1 will be pushed to the right when the simulation begins.

To create the spring

1. From the **Force** group in the **Professional** tab, click **Spring**.
2. Set the Creation Method toolbar to **Body, Body, Point, Point**.
3. Click in the background of the Working window to select the **Ground** body to be the base body of the Spring.
4. Click on the **Ball_1** geometry to select Ball_1 to be the action body of the **Spring**.
5. Click on the following locations in the Working window.
   - **Point1**: -20, 10, 0
   - **Point2**: 10, 10, 0

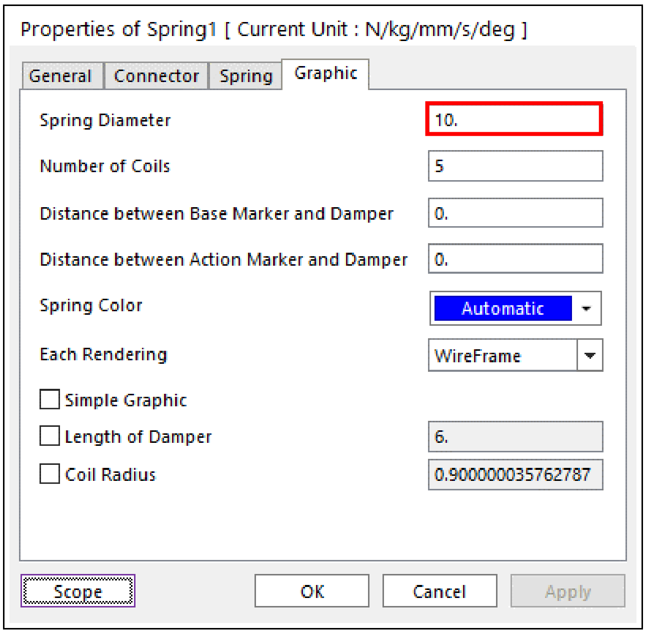
To adjust the spring properties

1. Display the Properties window for the spring, which will have the name of Spring1.
2. In the **Spring** tab, change.
   - **Stiffness Coefficient**: 20
   - **Damping Coefficient**: 0.05
   - **Free Length**: 45



The length as defined is 30 mm. By changing the Free Length to 45, you are indicating that the spring is compressed by 15 mm. Given the spring Coefficient of 10, a load of 150 N will be applied to the ball at the beginning of the simulation. Once the spring length increases to 45 mm, the force becomes zero.

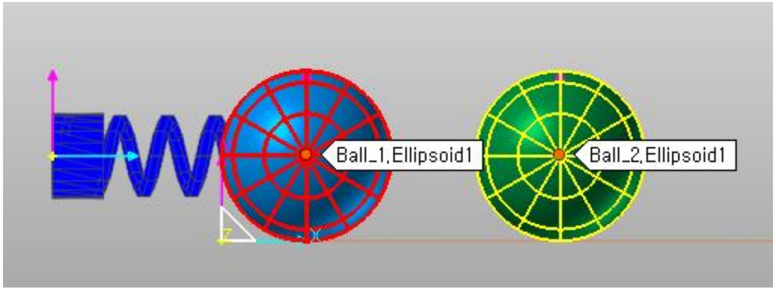
3. In the **Graphic** tab, change the **spring diameter** value to **10**.
4. Click **OK.**



Defining the Contact between the Balls

To create the contact between the balls

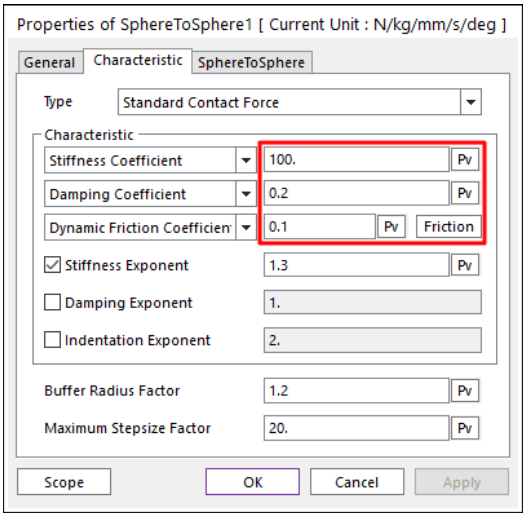
1. From the **Contact** group in the **Professional** tab, click **Sph-Sph**.
2. Set the Creation Method toolbar to **Sphere, Sphere**.
3. Select the following geometries.
   - **Sphere**: Ball_1.Ellipsoid1
   - **Sphere**: Ball_2.Ellipsoid1



4. From the **Contact** group in the **Professional** tab, click **Sph-Sph** and select the following geometries.
   - **Sphere**: Ball_2.Ellipsoid1
   - **Sphere**: Ball_3.Ellipsoid1

To adjust the contact between the balls

1. Open the **Properties** dialog of **SphereToSphere1**.
2. In the **Characteristics** tab, change:
   - **Stiffness Coefficient:** 100
   - **Damping Coefficient**: 0.2
   - **Dynamic Friction Coefficient**: 0.1
3. Click **OK**.
4. Change the **SphereToSphere2** Properties with same settings above.

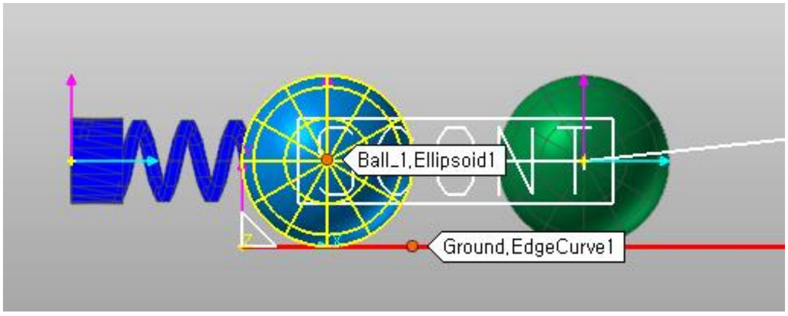


Defining Contact Between the Balls and Guides

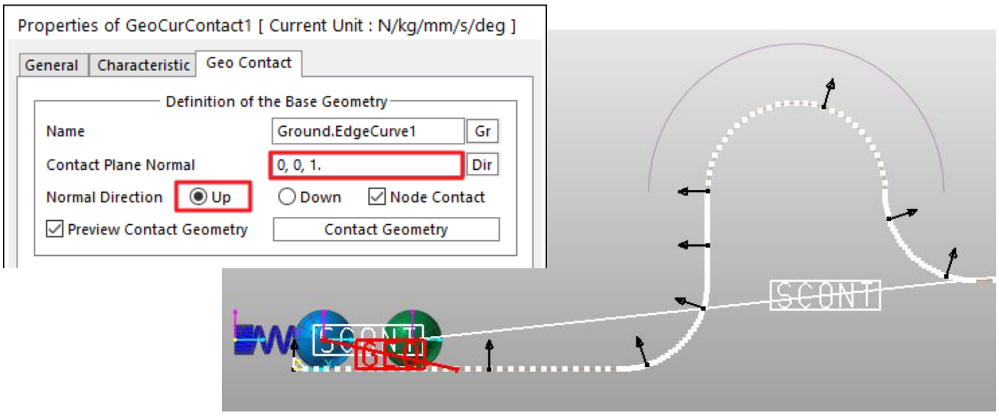
The first ball (**Ball_1**) is constrained by the spring and will only contact the straight line as the left of the guide geometry.

To create the contact between the first ball (Ball_1) and the guide geometry

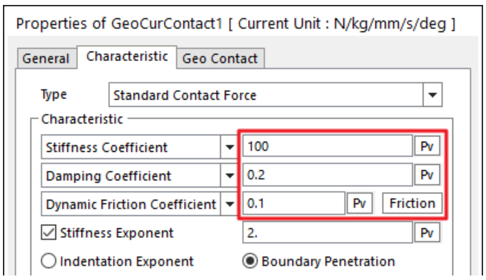
1. From the **2D Contact** group in the **Professional** tab, click **GeoCir** and select geometries below.
   - **Curve**: Ground.EdgeCurve1
   - **Circle(Sphere)**: Ball_1.Ellipsoid1



2. Open the Properties dialog of **GeoCurContact1**.
3. In **Geo Contact** tab, change the normal directions in Preview as shown below.
   - **Contact Plane Normal(Base/Action)**: 0, 0, 1
   - **Normal Direction(Base)**: Up

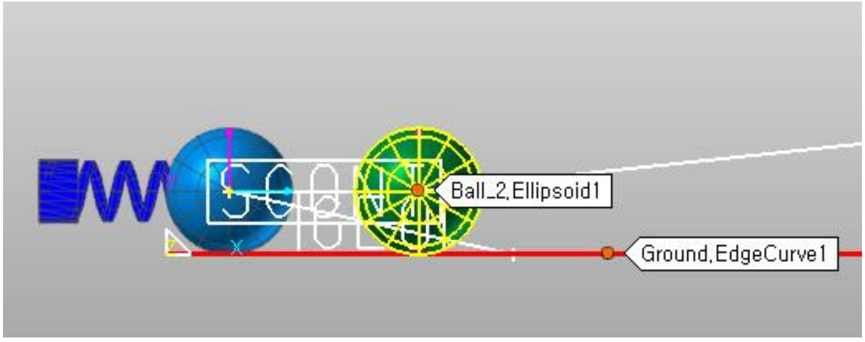


4. In the **Characteristics** tab, change.
   - **Stiffness Coefficient**: 100
   - **Damping Coefficient**: 0.2
   - **Dynamic Friction Coefficient**: 0.1
5. Click **OK**.

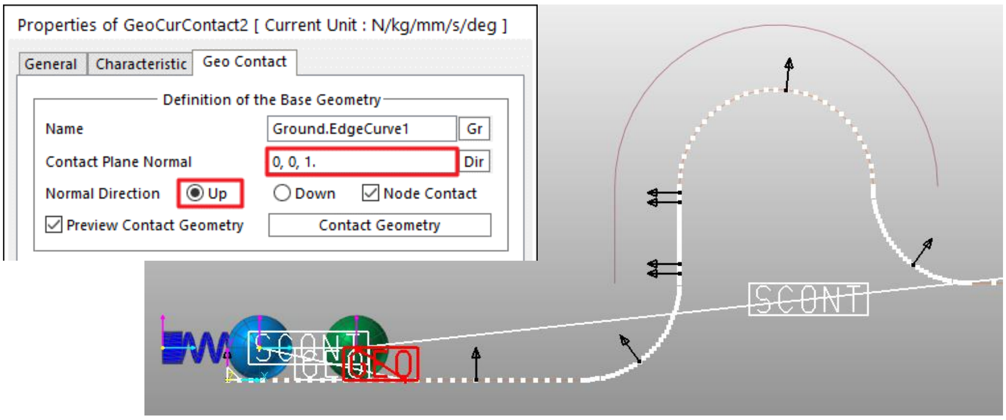


To create the contact between the second ball (Ball_2) and the guide geometry

1. From the **2D Contact** group in the **Professional** tab, click the **GeoCir** and select following geometries.
   - **Curve**: Ground.EdgeCurve1
   - **Circle(Sphere)**: Ball_2.Ellipsoid1



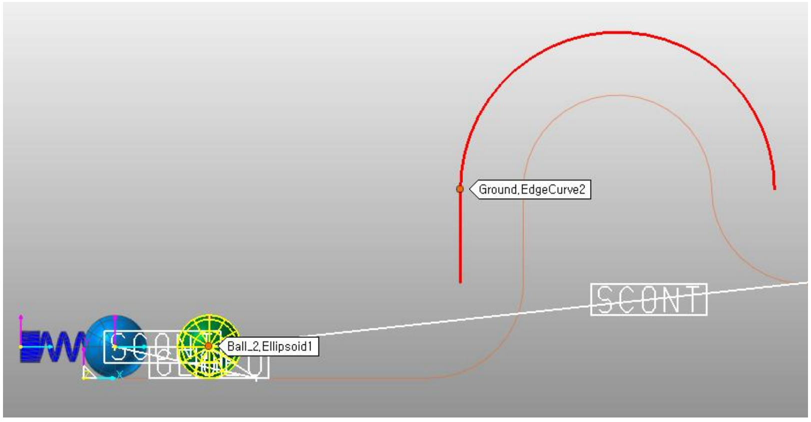
2. Open the Properties dialog of **GeoCurContact2**.
3. In **Geo Contact** tab, change the normal directions in Preview as shown below.
   - **Contact Plane Normal(Base)**: 0, 0, 1
   - **Normal Direction(Base)**: Up



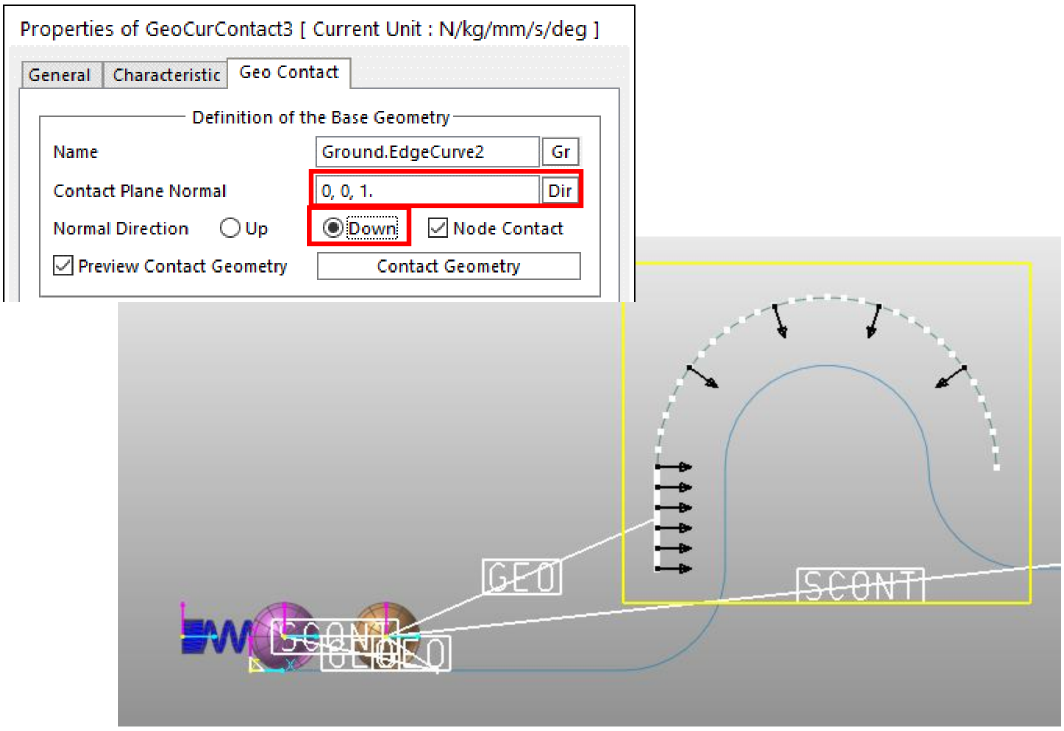
4. In the **Characteristics** tab, change
   - **Stiffness Coefficient**: 100
   - **Damping Coefficient**: 0.2
   - **Dynamic Friction Coefficient**: 0.03
5. Click **OK**.

To create the contact between the Ball_2 and the arc Guide geometry

1. From the **2D Contact** group in the **Professional** tab, click the **GeoCir** and select following geometries.
   - **Curve**: Ground.EdgeCurve2
   - **Circle**: Ball_2.Ellipsoid1



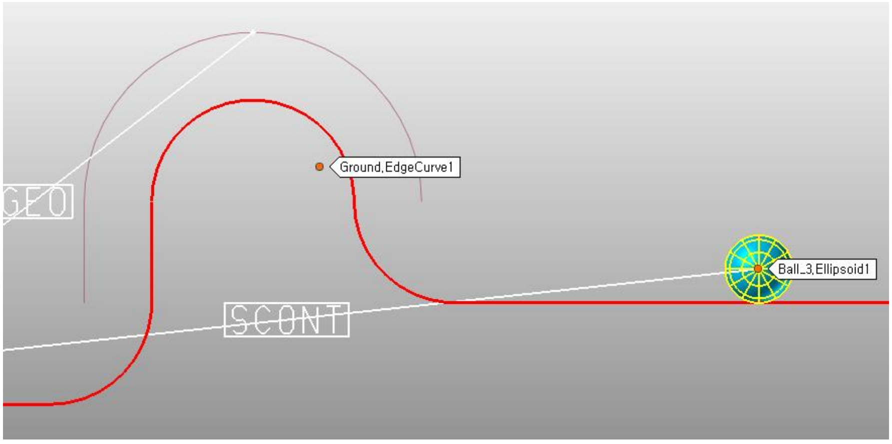
2. Open the Properties dialog of **GeoCurContact3**.
3. In **Geo Contact** tab, change the normal directions in Preview into the following configurations as shown below.
   - **Contact Plane Normal(Base)**: 0, 0, 1
   - **Normal Direction(Base)**: Down



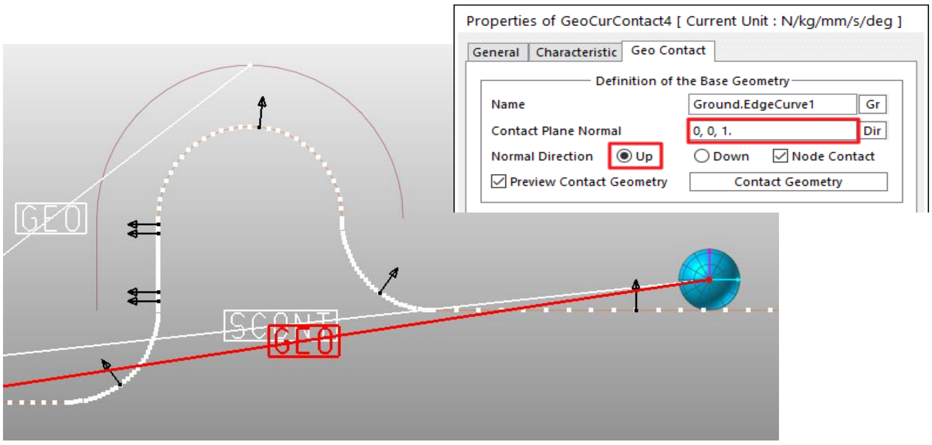
4. In the **Characteristics** tab, change.
   - **Stiffness Coefficient**: 100
   - **Damping Coefficient**: 0.2
   - **Dynamic Friction Coefficient**: 0.03
5. Click **OK**.

To create the contact between the Ball_3 and the lower Guide geometry

1. From the **2D Contact** group in the **Professional** tab, click the **GeoCir** and select following geometries.
   - **Curve**: Ground.EdgeCurve1
   - **Circle(Sphere)**: Ball_3.Ellipsoid1



2. Open the Properties dialog of **GeoCurContact4**.
3. In **Geo Contact** tab, change the normal directions in Preview into the following configurations as shown below.
   - **Contact Plane Normal(Base)**: 0, 0, 1
   - **Normal Direction(Base)**: Up



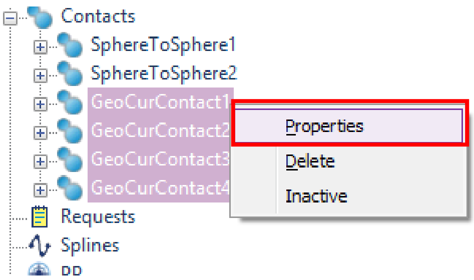
4. In the **Characteristics** tab, change
    - **Stiffness Coefficient**: 100
    - **Damping Coefficient**: 0.2
    - **Dynamic Friction Coefficient**: 0.1
5. Click **OK**.

To define the Smooth Node Contact and define force displays

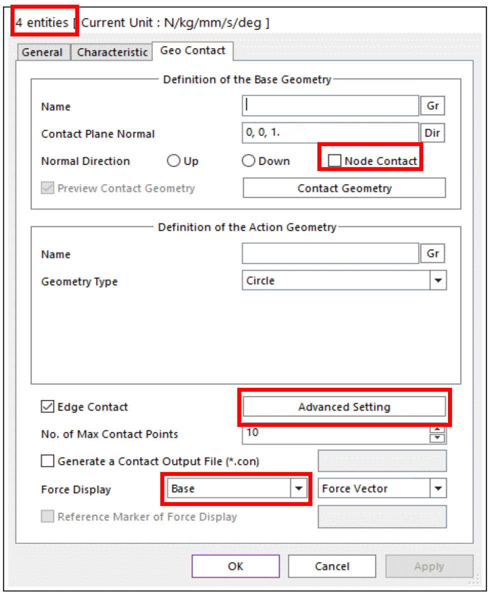
Set the additional options to make smooth contact between guides and balls and define **Force Display** options for the visualization of Contact Forces in animation.

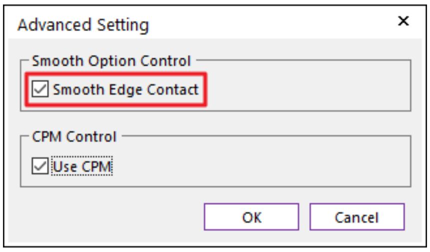
1. From **GeoCurContact1** to **GeoCurContact4**, select 4 Contacts from Database. Then right-click holding **Ctrl Key** to click **Properties**.

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



2. In **Geo Contact** tab, Uncheck **Node Contact** of **Base Geometry**.
3. In **Geo Contact** tab, click **Advanced Setting** button.
4. In **Advanced Setting** dialog, check on the **Smooth Edge Contact** option and click **OK**.

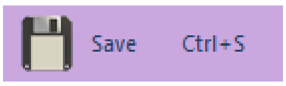




5. In **Geo Contact** tab, change **Force Display** to **Base**.
6. Click **OK** in Multi-Property dialog.

Saving the Model

Take a moment to save your model before you continue with the next chapter. (**Tip**: Click **Save** in **File** Menu)



Chapter

5

Creating Expression Scope

Task Objective

In this chapter, you'll run a simulation of the model you just created. Furthermore, you will define the expression measuring the distance between two bodies and run the simulation. Also, you will monitor the expression result through the scope.

Estimated Time to Complete

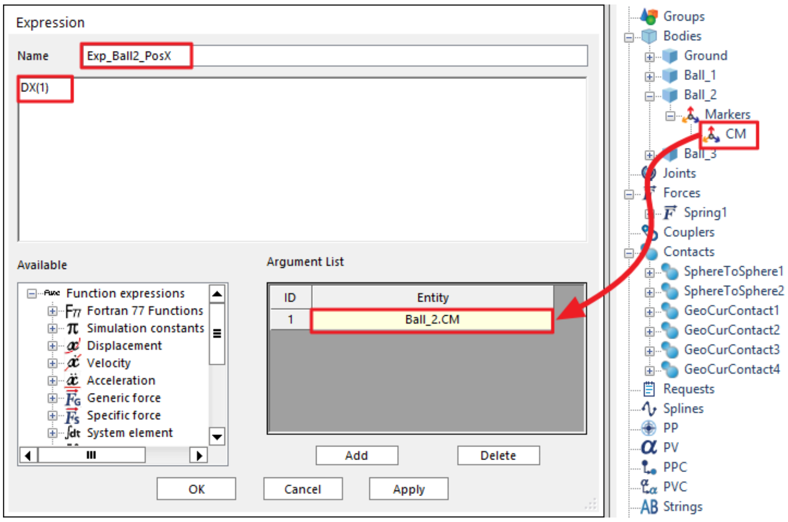
30 minutes

Defining Expression

You will define the expression measuring distance between two bodies.

1. From the **Expression** group in the **SubEntity** tab, click **Expression.** The Expression List dialog will appear.
2. Click **Create**. The **Expression** Dialog will appear.
3. Change **Name** to **Exp_Ball2_PosX**.

   Enter the following expression in the large text box: **DX(1)**
4. Click **Add** located below **Argument List**.
5. In the **Database** window, click **+** in front of **Ball_2** and **Markers**.
6. Drag **CM** in **Ball_2** markers and drop in empty text box in the **Argument List**.

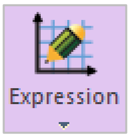


7. Click **OK** in the **Expression** Dialog.
8. Click **OK** in the **Expression List** Dialog.

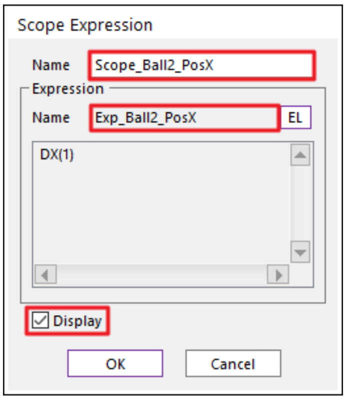
Tip: Changing Markers in Expression
In the above **Expression**, the Center Marker(**CM**) used in the **Argument List** means the marker of the mass center of the body. You can use the **General Marker** which you can define instead of **CM** in **Expression.**
You can create the **General Marker** by clicking **Marker** from the **Marker and Body** group in the **Professional** Tab.

Creating Expression Scope

You can monitor the result of **Exp_Ball2_PosX** with **Scope** without going through **Plot**.



1. From the **Scope** group in the **Analysis** tab, Click **Expression**.
2. Change **Name** to **Scope_Ball2_PosX**.
3. Click **EL** button to select **Exp_Ball2_PosX** and Click **OK**.
4. Check **Display** and Click **OK**.



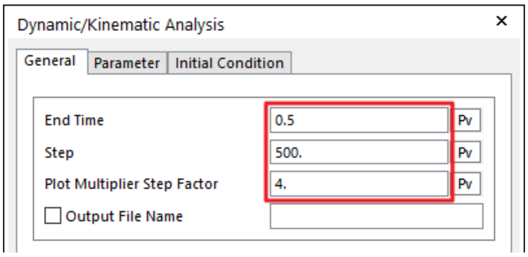
Now you are ready to monitor the **Scope** of **Expression**.

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 just created.

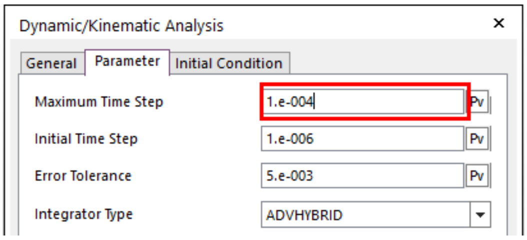
To perform a dynamic/kinematic analysis

1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
2. In the **General** tab, define the end time of the simulation and the number of steps:
   - **End Time**: 0.5
   - **Step**: 500
   - **Plot Multiplier Step Factor**: 4

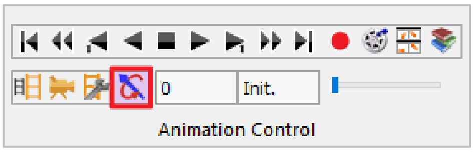


3. In the **Parameter** tab, define the **Maximum Time Step** to **1.e-004**.
4. Click **Simulate**.

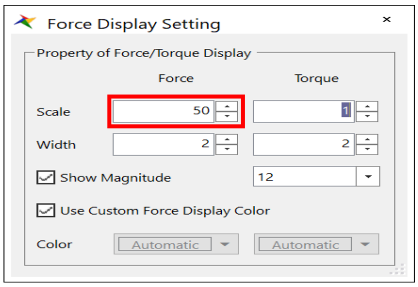
   RecurDyn calculates the motions and forces in the balls, spring and contacts. There will be 2000 plot outputs stored because the Number of Steps is 500 and the Plot Multiplier Step Factor is 4.



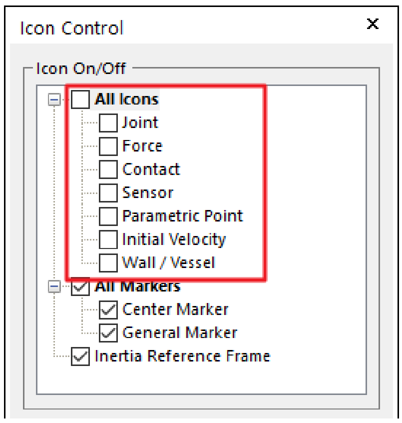
5. From the **Animation Control Group** in **Analysis** tab, click **Force Display Setting**.



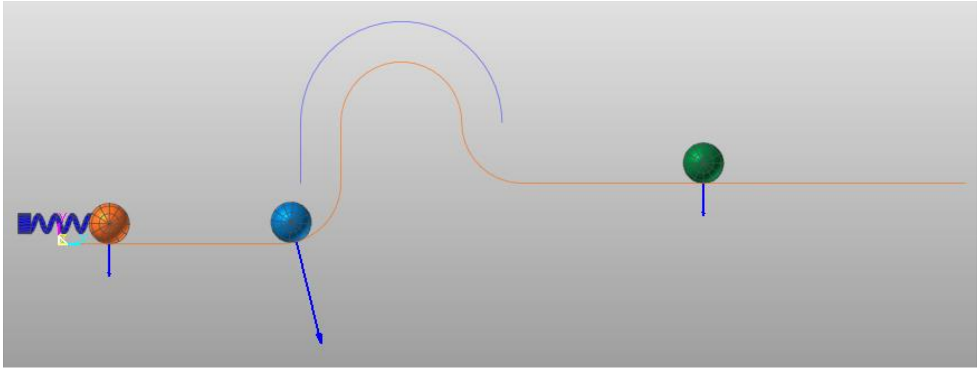
6. Change **Scale** to **50** to clearly monitor the **Force Display** result of **Contact**.



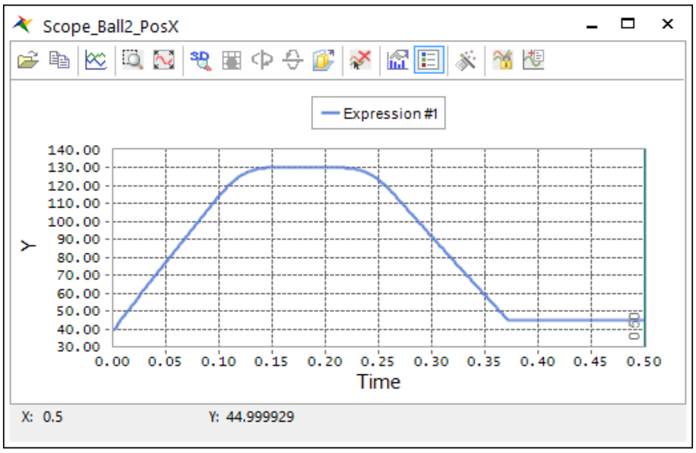
7. In the **View Control Toolbar**, click **Icon Control**.
8. Uncheck **All Icons** to disappear the icons on Working Window.
9. Close **Icon Control** window.



10. From **Animation Control** Group in **Analysis** tab, click **Play** button to play the animation.



You will see the result of **Expression Scope** as shown below



You look in the spring catalog and you find that this spring is available in 1 mm increments from 45 mm through 54m. You want to try out different lengths of the spring, but you don't want to have run the various cases and keep track of the results manually.

You want to do an automated design study, which is explained in next chapter.

Chapter

6

Performing a Design Study

Task Objective

In this chapter, you will rerun the simulation under a design study environment to decide how to select a spring that will provide the right energy to move the ball over the hump.

Estimated Time to Complete

30 minutes

Performing a Design Study

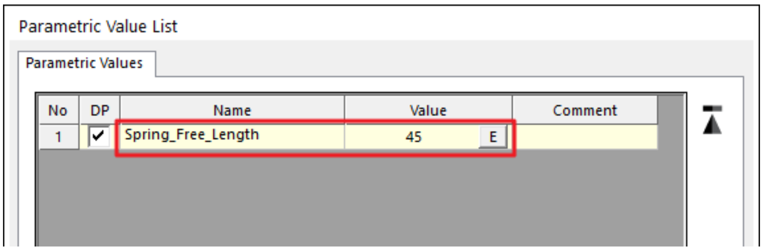
The steps to performing a design study for this model are below.

1. Define the free length of the spring as a **Parametric Value** that can be adjusted during the design study.
2. Define a **Design Variable** from the Parametric Value, where you add information about the data boundaries. The range of the free length of the spring is from **45** to **54 mm**.
3. Define a **Performance Index**. This is an outcome of interest for the design study.

   In this case, you will measure the maximum position of **Ball_2** in the X direction. If the maximum position of **Ball_2** is more than **170**, then you know that it has passed the top of the vertical obstacle.
4. Set the **Number of Trials** for this design study, which is **10**
5. **Run** the Design Study. Outputs will automatically be directed to different output files.
6. **Review** and **Plot** the results from the design study.
7. **Animate** a particular trial.

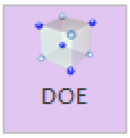
To define a parametric value

1. Display the **Properties** window for the **Spring1**.
2. Next to the **Free Length** text box, click the **Pv** button.
3. When the Parametric Value List window appears, click the **Add** button and change like as below.
   - **Name**: Spring_Free_Length.
   - **Value**: 45



4. Click **OK** to exit the Parametric Value List window.
5. Click **OK** to exit the **Spring1** Properties window. You will see that the name of the **Parametric Value** appears in the Free Length text box.

To define a design variable:



1. Click the **DOE** from the **Simulation Type** group in the **Analysis** tab.

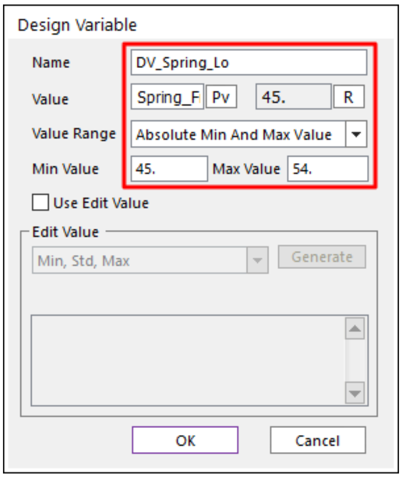
   The Design Study window appears

2. In the **Design Variables** section of the Design Study window, click the **Add** button.

   The **Design Variable List** window appears.

3. In the **Design Variable List** window, click the **Create** button.

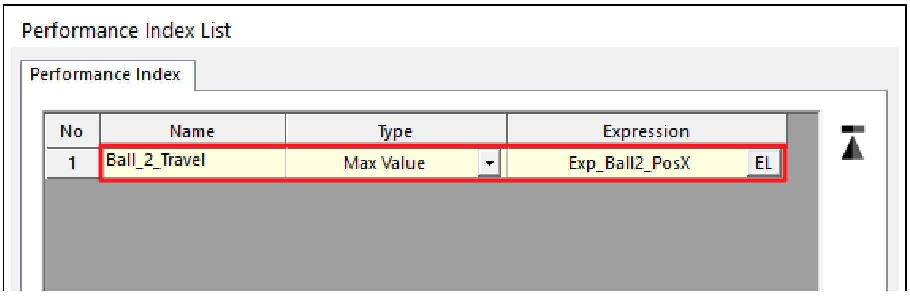
   The **Design Variable** window appears.



4. Do the following in the window:
   - **Name**: DV_Spring_Lo.
   - **Value**: Spring_Free_Length
   - **Value Range**: Absolute Min And Max Value
   - **Min Value**: 45
   - **Max Value**: 54
5. Click **OK** to exit the Design Variable window.
6. Click **OK** to exit the Design Variable List window.

To define a performance Index

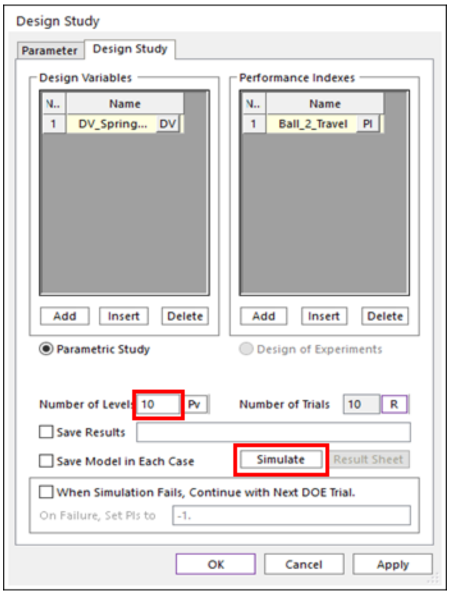
1. Click the **Add** button in the **Performance Indexes** section of the Design Study window. The **Performance Index List** window appears.
2. Click the **Add** button in the Performance Index List window.
   - **Name**: Ball_2_Travel
   - **Type**: Max Value
   - **Expression**: Exp_Ball2_PosX



3. Click **OK** to exit the Performance Index List window.

To set the number of trials

In the **Number of Levels** text box in the Design Study window, type **10**.



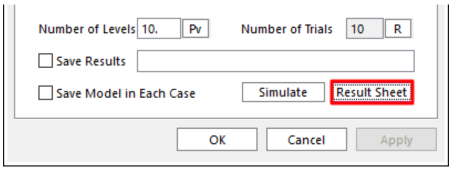
To run the design study

In the **Design Study** window, click the **Simulate** button.

10 runs occur in the output window. When the runs are complete, the Design Study window reappears.

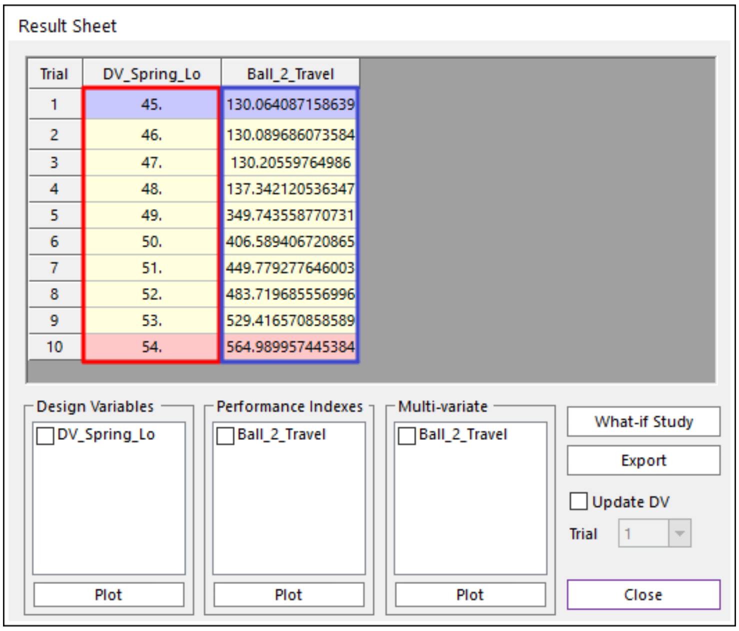
To review and plot results

1. Click the **Result Sheet** button.



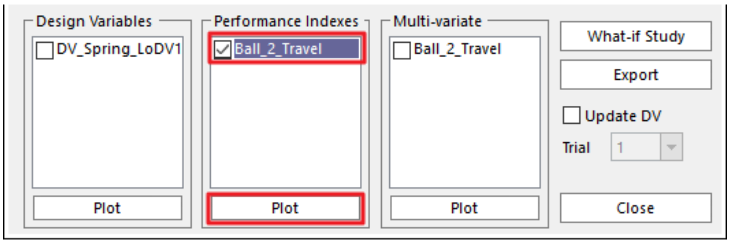
A list of the 10 runs appears:

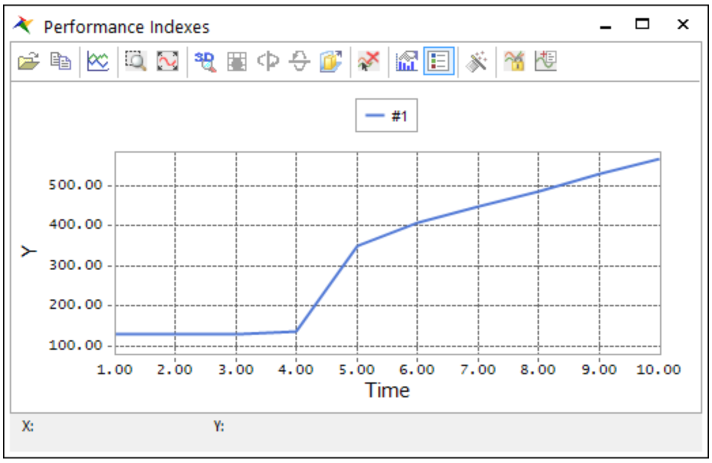
- The first column displays the value of the Free Length of the spring.
- The second column displays the maximum X value of the center of Ball_2.



2. To plot the Ball_2_Travel Performance Index, under the **Performance Indexes** heading, click the box in front of **Ball_2_Travel**, and then click **Plot**.

3. The following plot appears when you click the **Plot** button





4. Click the **x** in the upper right corner and to close the plot and click **Close** in the Result Sheet window.

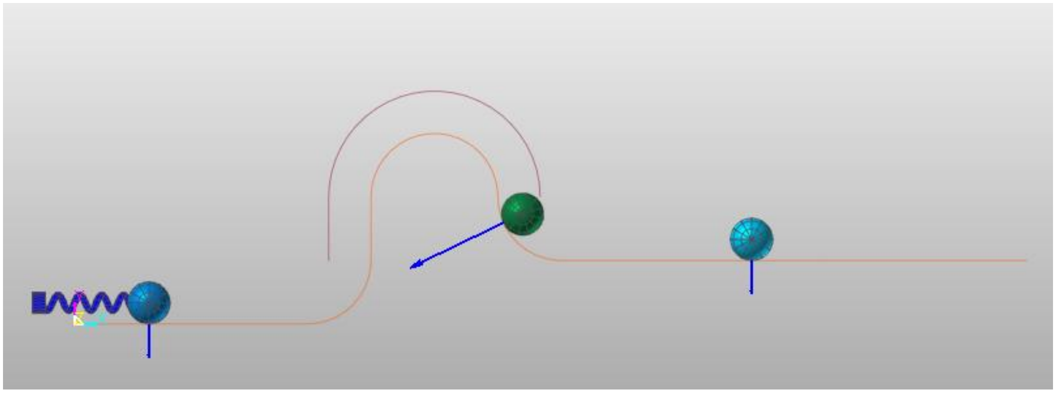
5. Click **OK** to close the Design Study window.

Animating the Results of a Trial

You'll first take a look at Trial 5, which is the firstly trial where Ball_2 makes it over the vertical obstacle.

To animate the results of a particular trial:

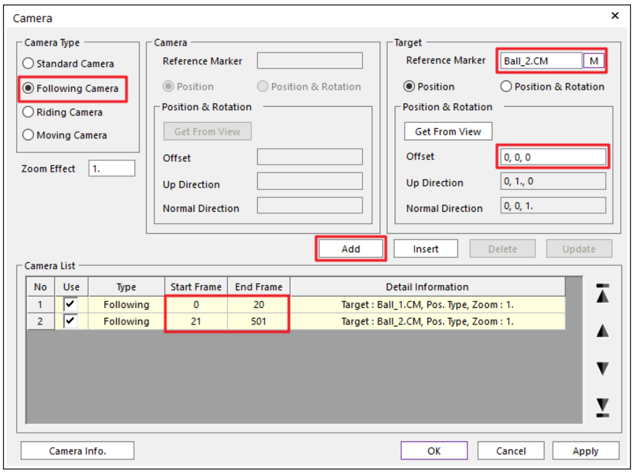
1. From the **File** menu, click **Import**.
2. Set **Files of type** to **RecurDyn Animation Data File (*.rad)**.
3. Double-click **Pinball_5.rad**, which contains the animation results for Trial 5
4. Animate the model using the **Play** button on the Animation toolbar



To animate the result using select camera

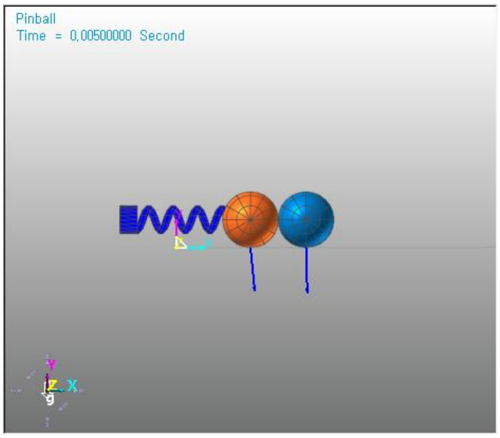
1. From the **Animation Control** group in the **Analysis** tab, click **Select Camera**.

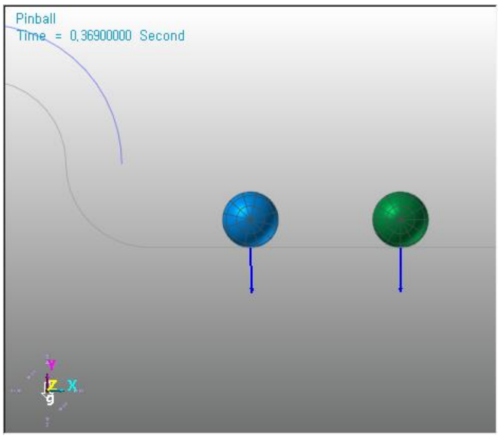
   The Camera window appears.
2. In **Camera Type** section, Select **Following Camera**.
3. In **Target** section, enter the following information.
   - **Reference Marker**: Ball_1.CM
   - **Offset**: 0, 0, 0
4. Click the **Add** button to add camera. Enter the following information in the Camera List.
   - **Start Frame**: 0
   - **End Frame**: 20



5. Repeat **Steps 2-4** twice using following information:
   - **Reference Marker**: Ball_2.CM
   - **Offset**: 0, 0, 0
   - **Start Frame**: 21
   - **End Frame**: 501

From 0 to 20 frame, camera follows **Ball_1_CM** and from 21 to 501 frame, camera follows the **Ball_2_CM** as shown below. You can do better posting if you set up with other **Camera Types** on your desired frames.





Ideas for Further Exploration

You can gain a good understanding of the behavior of the pinball model by looking at the animations through several trials. Here are more things to consider:

- The model used a low coefficient of friction (0.1). It represents the friction of a polished steel ball contacting with a smooth, dry surface. Low friction is a good assumption for the new pinball machine, but what if when the machine becomes old (by a dirt and some corrosion)?
    - How will results be changed when the friction coefficient is changed?
    - What could you do to study the effects of increased friction in this model?
    - Does increased friction require a stronger or weaker spring?

Thanks for participating in this tutorial!