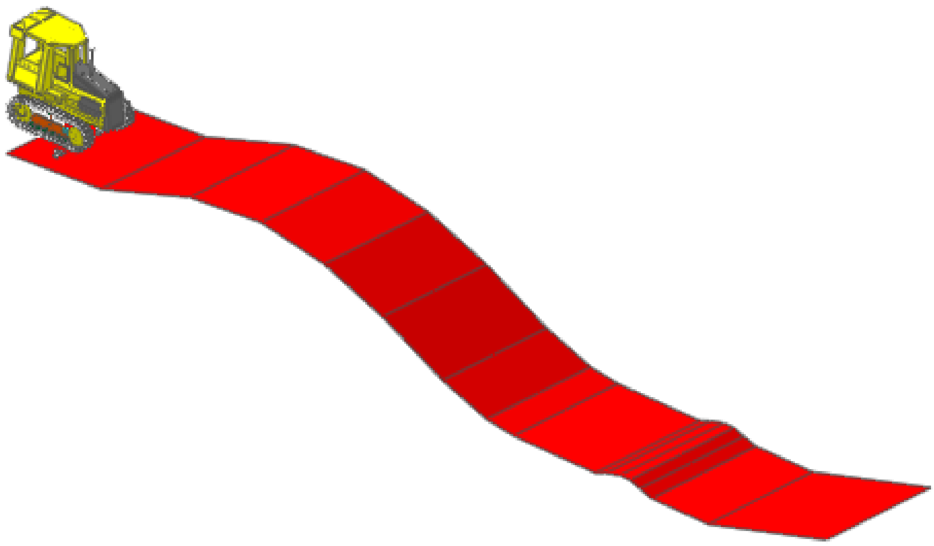
Low-mobility Tracked Vehicle Tutorial (Track_LM)



FunctionBay

Edition Note

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

Table of Contents

Chapter

1

Getting Started

Objective

This tutorial introduces you with the Low-Mobility (LM) Track Transport Toolkit and how to simulate the behavior of tracked vehicles on rigid, soil, and snow terrains. You will learn how to define undercarriage components and track shoes. You will also learn how to efficiently define the assembled track that will wrap around the undercarriage components. The tutorial is made up several main chapters, followed by two optional exercises. You can choose to work through any of the optional exercises after creating the model as instructed in main chapters.

The process that you will follow in the tutorial and the process automation tools that you will use to define the LM Track are similar to the processes and tools used to define belt/pulley and chain/sprocket subsystems. Therefore, the concepts that you learn while performing this tutorial will help you learn how to use the belt and chain toolkits more quickly. In addition, if you have already worked through a belt or chain tutorial, you will find this tutorial easier to work through than someone who has no experience with the belt or chain tutorials.

The terms low-mobility and high-mobility refer to two different styles of tracked vehicles. The low mobility toolkit is focused on:

- Tracked construction equipment with discrete steel shoes.
- Track shoes that are defined as an extruded profile.
- External guides that keep the track shoes from falling off of the rollers.
- Single sprocket that contacts the shoes at their centers.

The high-mobility toolkit is focused on:

- Tracked military vehicles.
- Track shoes that are defined as blocks with attached cleats.
- Center guides on the track shoes that pass-through grooves in the roadwheels and the idler to keep the track shoes from falling off of the rollers.
- Dual sprocket that contacts the shoes at the pins that protrudes from the sides of the track shoe.

The names low-mobility and high-mobility refer to the vehicle's speed of operation. Typically, construction equipment operates at a slower (or low) speed while the military equipment operates at a fast (or high) speed, especially during transport.

Audience

This tutorial is intended for experienced users of RecurDyn

Prerequisites

Users should first work through the 3D Crank-Slider Tutorial and the Engine with Propeller Tutorial

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 setup	15
Track component definition	35
Track subsystem completion	15
Full-vehicle model development	15
Subsystem tuning(optional)	20
Addition of blade linkage(optional)	30
Total	**130**

Estimated Time to Complete

2 hours

Chapter

2

Setting Up the Simulation Environment

In this chapter, you will start RecurDyn and set up its environment, including importing the completed bulldozer chassis with cab. You will also merge the imported geometry into one body and set its center of mass.

Task Objective

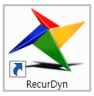
Learn to set up the RecurDyn environment and import data.

Estimated Time to Complete

15 minutes

Starting RecurDyn

To start RecurDyn and create a new model:

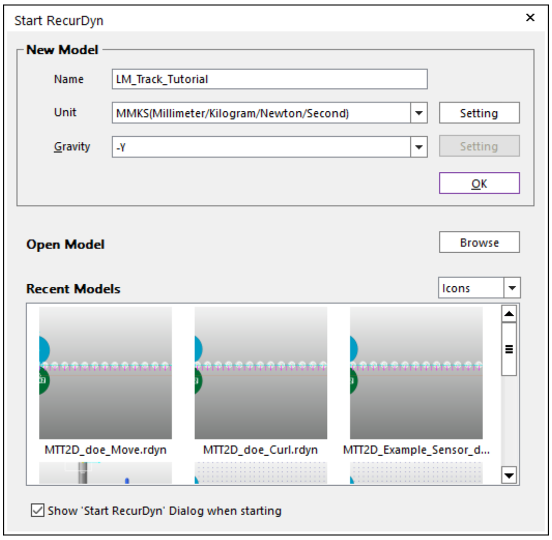


1. On your **Desktop**, double-click the **RecurDyn** icon.

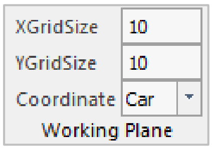
   RecurDyn starts and the **Start RecurDyn** dialog box appears.

2. Ensure the units are the same as those shown in the Start RecurDyn dialog box to the right. If not, click **MMKS**.

3. Change the model name to **LM_Track_Tutorial** as shown in the window, and then click **OK**.



4. Set the grid spacing to **10** mm.



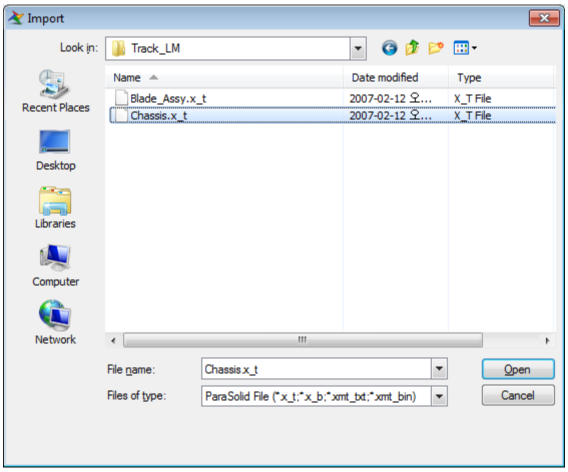
Importing the Chassis Geometry

You will begin to model the tracked vehicle by importing an already completed bulldozer chassis.

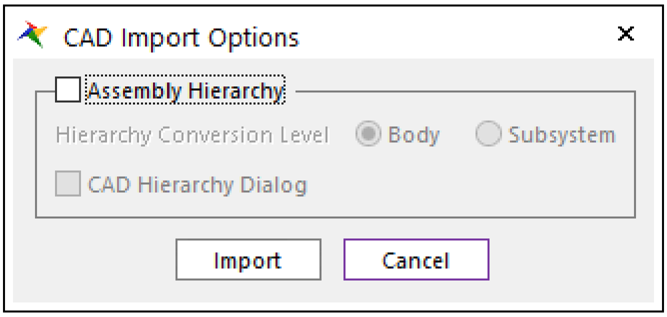
To import the chassis geometry:

1. From the **File** menu, choose **Import**.

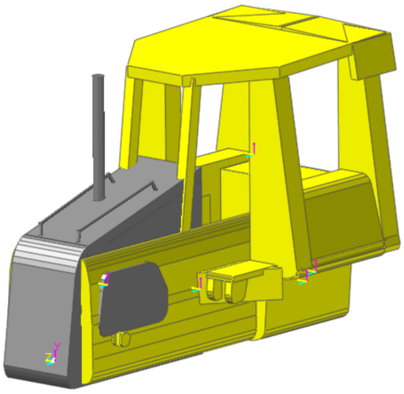
2. In the Open dialog box, set file type to **ParaSolid File**, and from the **Track LM** Files folder, select the file **Chassis.x_t** as shown in the dialog box on the right. (**The file location**: <Install Dir> /Help /Tutorial /Toolkit /Track_LM /LowMobilityTrackedVehicle, ask your instructor for the location of the directory if you cannot find it).



3. Click **Open**. The **CAD Import Options** window appears. Clear the **Assembly Hierarchy** checkbox and click the **Import** button.



The bulldozer chassis appears as shown below after rotating it and setting the render mode to Shade with wire.

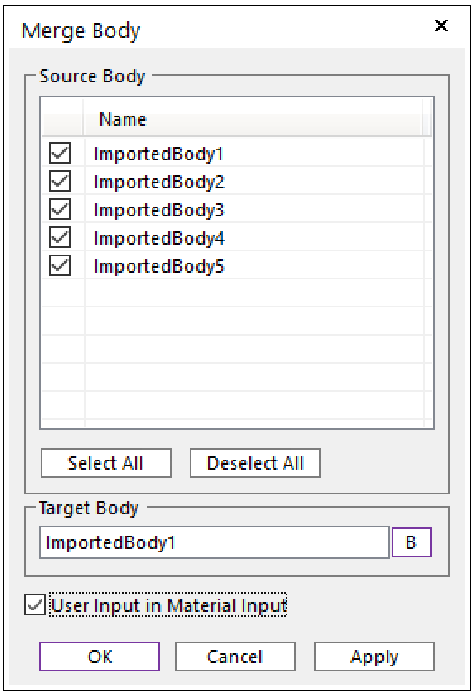


Merging the Bodies

The chassis is made up of five bodies. You are going to merge them into one body to make it easier to work with the chassis. In addition, you are going to change the center of mass to reflect the new merged body.

To merge the bodies:

1. From the **Tools** group in the **Home** tab, click **Merge**.



2. Select all the bodies in the **Merge Body** dialog box.
3. Enter **ImportedBody1** as the target body, either by typing the name into the **Target Body** text box or by clicking the **B** (Body) button and clicking the **cab geometry**. The target body is the body into which all the bodies will be merged.
4. Check the **User Input in Material Input** option to set the **Body Material Type** to **User Input**.
5. Click **OK**.
6. Display the Property dialog box for the merged body, **ImportedBody1** and do the following:

Tip: In the Database window, right-click **ImportedBody1** and click **Property**.

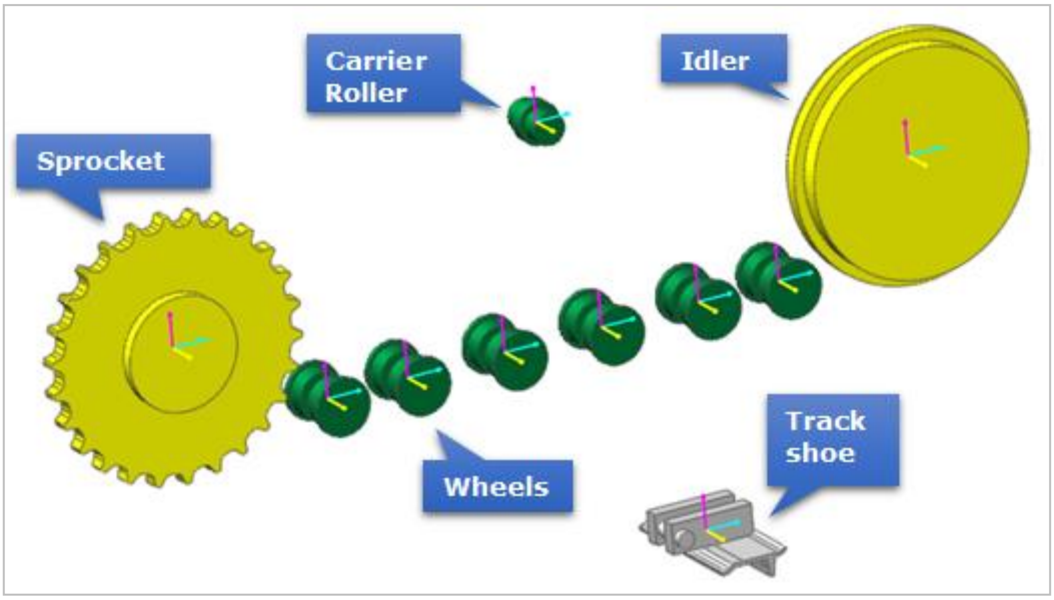
   - Click the **General** tab, if necessary, and change the name of **ImportedBody1** to **Chassis**.
   - Click **Apply**.
   - From the **Body** Tab, Click **CM** and change the **Origin** to **-1400, 500, 0**.
   - Click **OK** to close the CM dialog box.
   - Click **OK** again to close the **Chassis Properties** dialog box.
7. Save the **RecurDyn** model as **LM_Track_Tutorial.rdyn**.

Chapter

3

Defining the Track Componets

In this chapter, you will define the objects that make up the track using the **TrackLMSub1** tools. In the next chapter, you will assemble the objects together. The objects that you will create are shown below



Task Objective

Learn to create the objects that define a track and understand the process of creating a track using the **Track LM** tools.

Estimated Time to Complete

35 minutes

Defining the Right Track Assembly

You will create a low-mobility track subsystem which then provides you with access to standard entities in the LM Track Transport Toolkit, such as:

- Sprockets
- Rollers used to define road wheels, idlers, and carrier rollers
- Track links, lateral guards for the track, and a terrain builder

To define a new low-mobility track subsystem:

- From the **Subsystem Toolkit** group in the **Toolkit** tab, click the **Track (LM).**

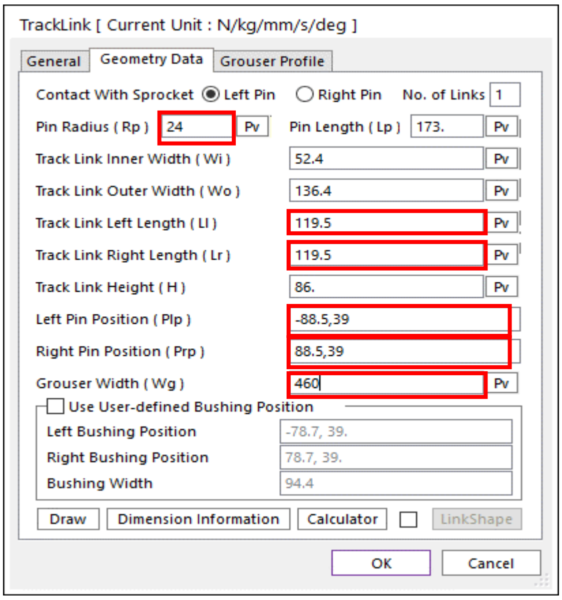
After **RecurDyn** creates the subsystem, it is in Subsystem Edit mode. The name in the upper left of the graphics window is **TrackLM1@LM_Track_Tutorial** and the top node in the Database window is TrackLM1.

Creating a Track Shoe

You will create a track shoe and define the grouser profile by importing a set of xy data points. You could create the set of points by exporting a profile from your CAD software or by calculating the points in Excel. The points are simply exported as two columns that are separated by a delimiter, such as a comma or a space. You should change the extension of the file name to .mat.

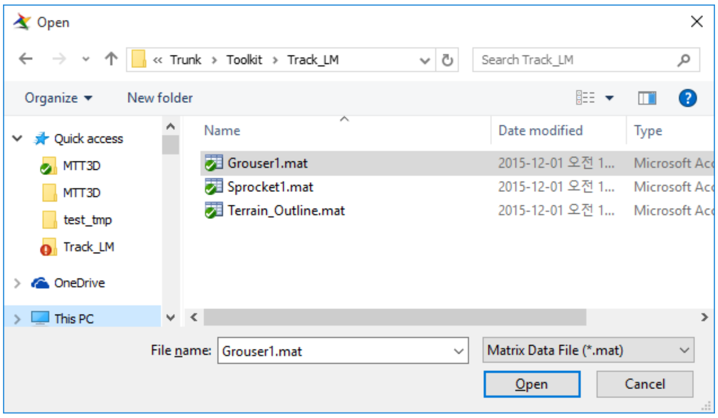
To create a track shoe:

1. From the **Link** group in the **Track (LM)** tab, click the **Link**. (This procedure proceed in the subsystem edit mode named TlackLM1)



2. Click the location **-1400, -900, 0.**
3. In the **TrackLink** dialog box, click the **Geometry Data** tab and fill in the text boxes as shown in the dialog box on the right. You will adjust the:
   - **Pin Radius: 24**
   - **Track Link Left Length: 119.5**
   - **Track Link Right Length: 119.5**
   - **Left Pin Position: -88.5, 39**
   - **Right Pin Position: 88.5, 39**
   - **Grouser Width: 460**
4. To view a graphical description of the geometry data, click **Dimension Information**.
5. Keep the **TrackLink** dialog box open.

To import a new track link grouser profile:



1. In the **TrackLink** dialog box, click the **Grouser Profile** tab, and then click **Import**.
2. Select the **Grouser1** file (.mat file extension may or may not be visible) from the **Track LM Files** directory (**The file location:** <Install Dir> /Help /Tutorial /Toolkit /Track_LM /LowMobilityTrackedVehicle) and click **Open**.

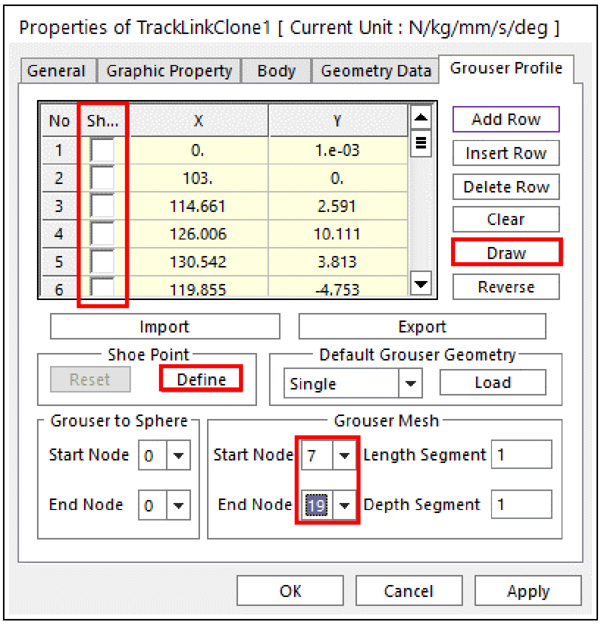
The grouser coordinates appear as shown at the right.

3. Click **Draw** to display the grouser profile as shown.

   As you review the grouser profile, you will see that nodes (points) 7 through 19 cover the section of the grouser that will contact the terrain. You want those nodes to be included in the contact calculations for hard terrains.

4. In the spreadsheet of the **TrackLink** dialog box, check the boxes under the Shoe column for points **8, 10, 12, 14, 16**, and **18.**

5. To set up the grouser points, in the Shoe Point section, click **Define**.



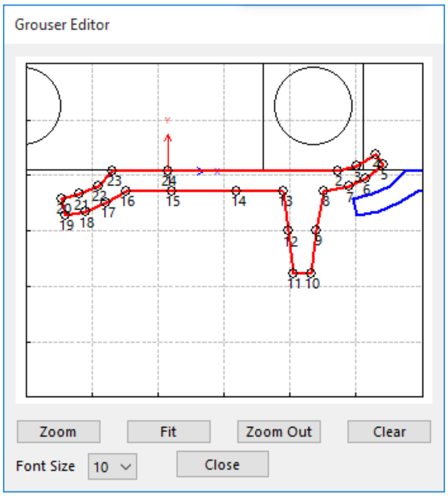
You also want to set up the contact calculations for soft terrains (soils, sands, snow, and so on).

6. In the **TrackLink** dialog box, set the **Start Node** to **7** and the **End Node** to **19** in the **Grouser Mesh** section.

7. Click **OK** to exit the **TrackLink** dialog box.

8. Display the **TrackLink1** Property dialog box and set its color to **light gray** (**Graphic Property** tab).

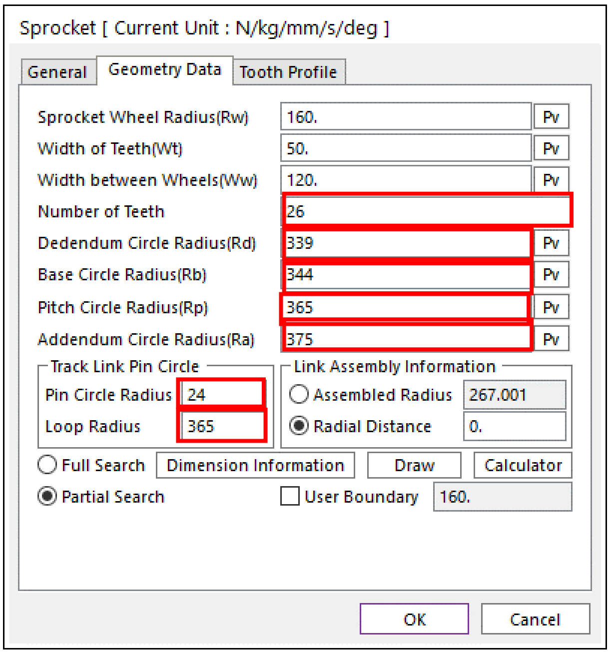
9. Click **OK**.



Creating a Sprocket

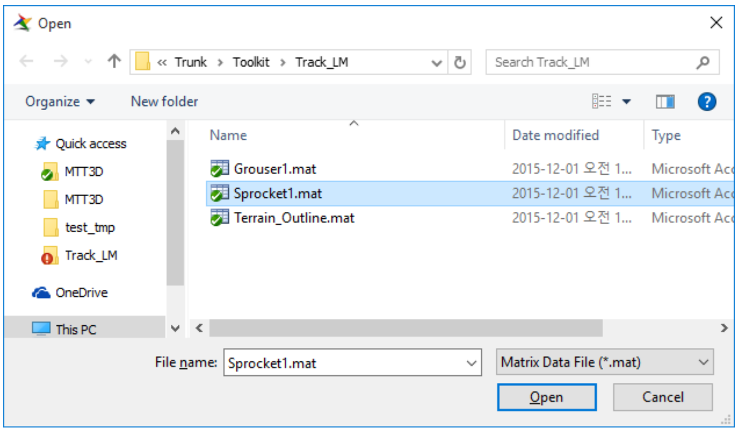
To create a sprocket:

1. From the **Sprocket** group in the **Track (LM)** tab, click the **Sprocket**.
2. Click the location -**2900, 0, 0**.
3. In the **Sprocket** dialog box, click the **Geometry Data** tab.
4. Click **Dimension Information** to see the meaning of the various entries.
5. Fill in the text boxes as shown in the dialog box on the right. You will adjust the:
   - **Number of Teeth: 26**
   - **Dedendum Circle Radius (Rd): 339**
   - **Base Circle Radius (Rb): 344**
   - **Pitch Circle Radius (Rp): 365**
   - **Addendum Circle Radius (Ra): 375**
   - **Pin Circle Radius (Rp): 24**
   - **Loop Radius: 365 (typically the same as the pitch circle radius)**
6. Keep the dialog box open.



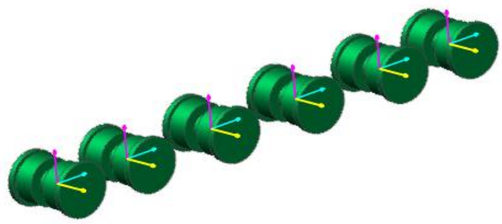
To import a new sprocket tooth profile:

1. In the **Sprocket** dialog box, click the **Tooth Profile** tab, and then click **Import**.
2. Select the **sprocket1** file (.mat file extension may or may not be visible) from the **Track LM Files** directory. (**The file location:** <Install Dir> /Help /Tutorial /Toolkit /Track_LM /LowMobilityTrackedVehicle)



3. Click **Open**.
4. Click **OK**.
5. Display the Sprocket Property dialog box and set the color of the sprocket tooth profile to **yellow (Graphic Property** tab).
6. Click **OK**.

Creating a Set of Road Wheels



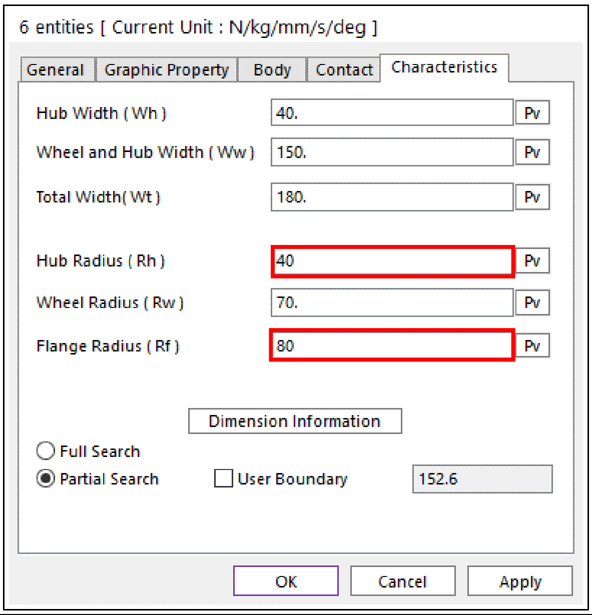
In this section, you will create six road wheels, as shown in the figure on the right. You will set their properties and change their names.

To create a set of road wheels:

1. From the **Flange** group in the **Track (LM)** tab, click **Single**.

2. Create six road wheels with centers at the locations specified in the table below. You can create each road wheel individually and modify its properties individually, or you can create one road wheel, modify its properties, make a copy, move it to the correct position and repeat for the six road wheels. If copying, remember to turn off the **Shift when Pasting** option (see **Home Tab→Setting Group→Display→Advanced).** In the next step, you will rename the road wheel names as listed in the table.

Set the single flange	To point (X,Y)	With a distance (radius)	X offset (if copying)	Change name to Road Wheel #
SingleFlange1	-2460, -250	70		6
SingleFlange2	-2220, -250	70	240	5
SingleFlange3	-1930, -250	70	290	4
SingleFlange4	-1640, -250	70	290	3
SingleFlange5	-1350, -250	70	290	2
SingleFlange6	-1110, -250	70	240	1

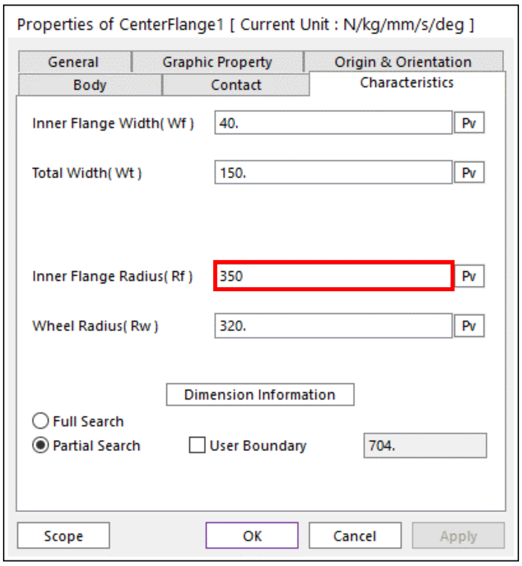


3. Display the **Properties** dialog box for each road wheel.
   - Click the **Characteristics** tab and set the parameters to be the same as in the figure and listed below:
     - **Hub Radius (Rh): 40**
     - **Flange Radius (Rf): 80**
   - Change their color to **green**.
   - Rename them from **SingleFlange1** through **SingleFlange6** to **RoadWheel_6** through **RoadWheel_1**, renaming them in descending order, as shown in the table above.

Creating an Idler

To create an idler:

1. From the **Flange** group in the **Track (LM)** tab, click the **Center**.

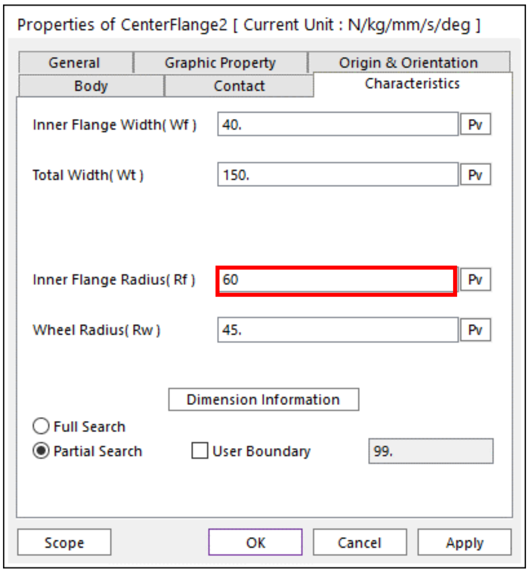


2. Create a flange with a center at **-700, 0, 0**. and a radius (distance) of **320** (enter it in the **Command Toolbar** text box or drag the mouse horizontal or vertical until the readout is **320**).
3. Display the **CenterFlange1** Properties dialog box to set the color of the idler to **yellow** (**Graphic Property** tab).
4. Click the **Characteristics** tab and set **Inner Flange Radius (Rf)** to **350**.
5. Click the **General** tab and set the name to **Idler**.
6. Click **OK**.

Creating a Carrier Roller

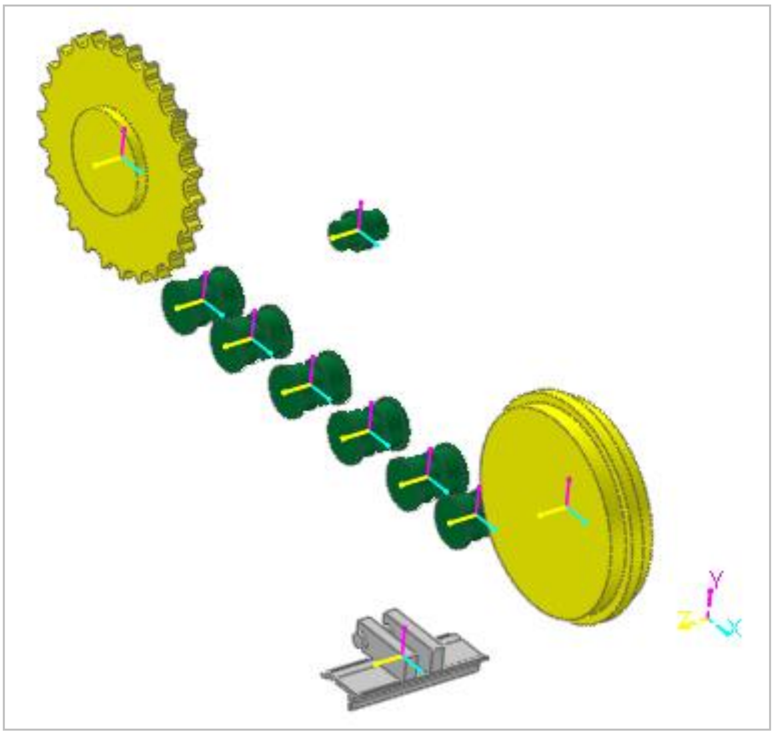
To create a carrier roller:

1. From the **Flange** group in the **Track (LM)** tab, click the **Center**.



2. Create a flange with a center at **-1780, 360, 0** and a radius (distance) of **45** (enter it in the **Input Window** text box).
3. Display the **CenterFlange1** Properties dialog box to set the roller color to **green** (**Graphic Property tab**).
4. Click the **Characteristics** tab and set **Inner Flange Radius (Rf)** to **60**.
5. Click the **General** tab and change the name to **Carrier**.
6. Click **OK**.

Your track subsystem should look like the figure below (rotated, and with the render mode set to **Shade With Wire**).



You are now ready to assemble the track.

Chapter

4

Finishing the Track Subsystem

In this chapter, you will complete the modeling of the subsystem by creating the track, creating the track frame geometry and attaching the track components to the track frame. You will add a motion to the sprocket and be prepared to simulate the behavior of the track subsystem.

Task Objective

Learn to assemble a track, create a track frame, and attach the track components to the track frame.

Estimated Time to Complete

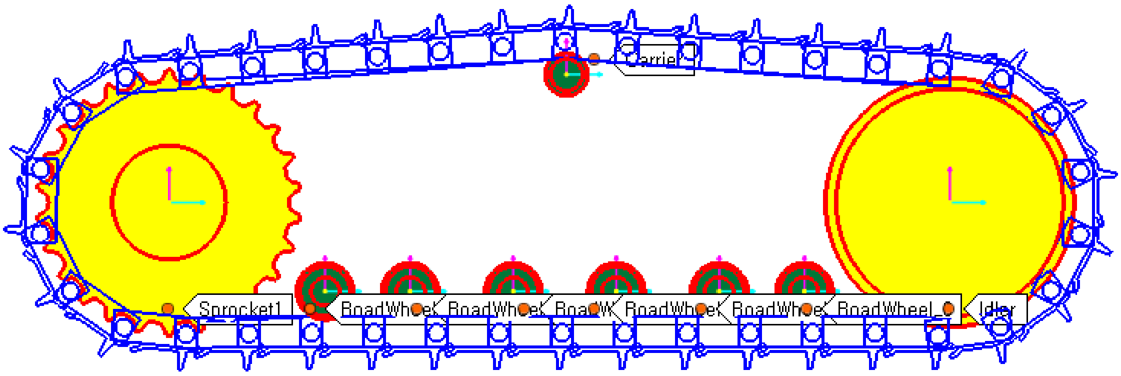
30 minutes

Assembling the Track

To assemble the track:

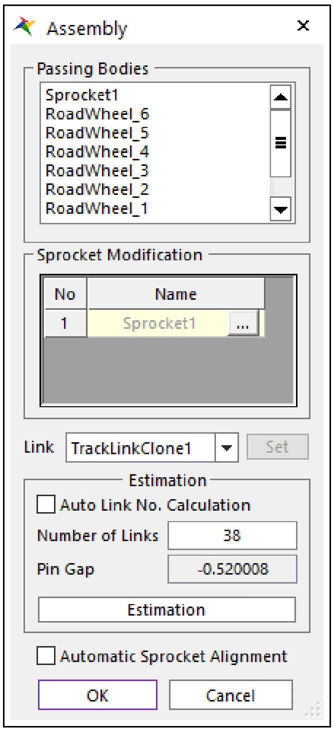
1. From the **Assembly** group in the **Track (LM)** tab, click the **Assembly**.

2. Starting with the sprocket, select each item in the track in a counterclockwise order until you have completed the loop as shown in the figure following.



When you are done, a dialog box appears as shown on the right.

3. Click **OK** and **RecurDyn** adds the components to the database.



Creating a Track Frame

To create a track frame body:

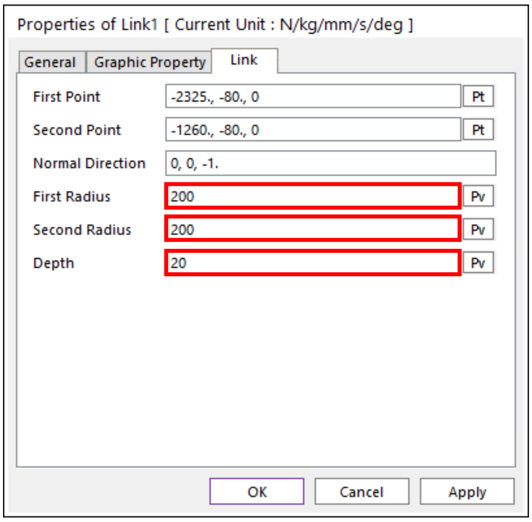
1. From the **Marker and Body** group in the **Professional** tab, click the **Link**.

2. Click the following two points to create the link.
   - **-2325, -80, 0**
   - **-1260, -80, 0**

Editing the Track Frame Body

To edit the track frame body:

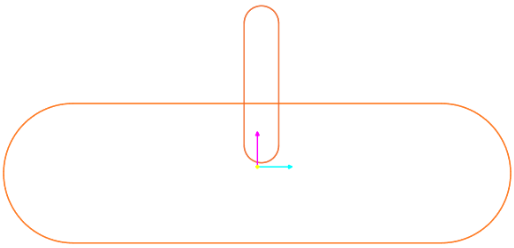
1. Enter **Body Edit** mode for the link body you just created by either.
   - Double-clicking on the geometry of the body that you just created.
   - In the Database window, right-clicking **Body1**, and selecting **Edit**.
2. Display the Properties dialog box for the link geometry and do the following.
   - Change the **First Radius** and **Second Radius** values to **200** and the **Depth** to **20**.
   - Click **OK**.



Creating the Carrier Holder

To create the carrier holder:

1. From the **Marker and Geometry** group in the **Geometry** tab, click the **Link**.



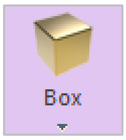
2. Click the following two points to create the link.
   - **-1780, 0, 0**
   - **-1780, 350, 0**
3. Display the Properties dialog box for the second link geometry and do the following.
   - Change the **First Radius** and **Second Radius** values to **50** and the **Depth** to **20**.

- Click **OK**.

4. Click the **Exit** arrow to exit **Body Edit** mode and return to the subsystem.
5. Display the **Properties** dialog box for this body and:
   - Set the name to **Track_Frame**.
   - In the **Graphic Property** tab, change the **color** to orange.
   - Click **OK**.

Creating a Tensioner Body

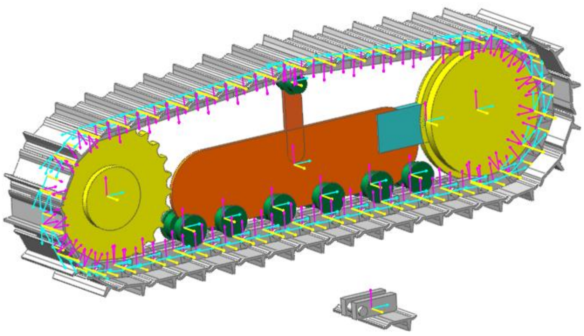
To create a tensioner body:



1. From the **Marker and Body** group in the **Professional** tab, click the **Box**.
2. Click on the following two points to create the box.
   - **-1300, 100, 0**
   - **-700, -100, 0**
3. Double-click the tensioner to enter **Body Edit** mode.
4. Display the Properties dialog box for the tensioner and do the following.
   - In the **Box** tab, change **Depth** to **40**.
   - In the Origin & Orientation tab, change Origin to (-1000, 0, -20).
5. Click **OK**.

6. Click the **Exit** arrow to return to the subsystem edit mode.
7. Display the **Properties** dialog box for the tensioner and do the following.
   - Click the **General** tab, if necessary, and set the name to **Tensioner**.
   - Click the **Graphic Property** tab and change the color to **Aqua**.
8. Click **OK**.

The track assembly now appears as shown in the figure on the right.



Creating Joints

In this section, you will create a series of revolute and fixed joints to properly attach the sprocket, road wheels, idler, and carrier to the track frame to the **Mother Body** (local ground in subsystem).

To create joints:

- Using the tools of the Joint group in the Professional tab, create the following joints as specified in the table below. Use the Body, Body, Point option for joints located away from the body geometry.

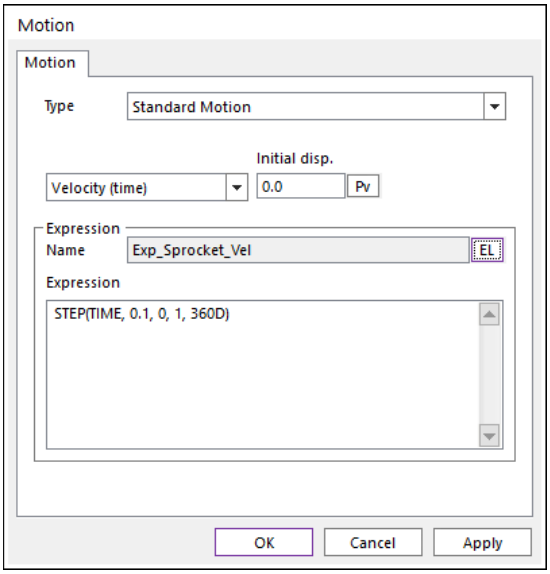
Create the joint of type	First Body	Second Body	At the location
Revolute	Sprocket1	MotherBody	-2900, 0, 0
Revolute	Idler	Tensioner	-700, 0, 0
Revolute	RoadWheel_6	Track_Frame	-2460, -250, 0
Revolute	RoadWheel_5	Track_Frame	-2220, -250, 0
Revolute	RoadWheel_4	Track_Frame	-1930, -250, 0
Revolute	RoadWheel_3	Track_Frame	-1640, -250, 0
Revolute	RoadWheel_2	Track_Frame	-1350, -250,0
Revolute	RoadWheel_1	Track_Frame	-1110, -250,0
Revolute	Carrier	Track_Frame	-1780, 360,0
Fixed	Tensioner	Track_Frame	-1200, 0, 0
Fixed	Track_Frame	Mother Body	-1780, 0, 0

Adding a Motion Input

In this section, you are going to add a motion input to the revolute joint of the sprocket that smoothly brings the sprocket velocity up to one revolution per second (360 degrees per second). The velocity will begin to ramp up at 0.1 seconds and will be fully developed at 1.0 second.

To add a motion input:

1. Right-click the revolute joint at the center of the sprocket (RevJoint1) and click **Properties**.
2. Check the **Include Motion** check box.
3. Click **Motion** button.
4. Set the unlabeled pull-down menu to **Velocity**.
5. Click **EL** to view the Expression List.
6. Click **Create** to create an expression.
7. Change the expression name to **Exp_Sprocket_Vel**, and define the expression as:
   - **STEP(TIME, 0.1, 0, 1, 360D)**
8. Click **OK** four times to exit all the way out of the Joint Properties dialog box.



Validating the Track Subsystem Definition

In this section, you will run a simulation to validate the subsystem. The track system is fixed in space and there is no terrain for the track to contact. You will be able to see the sprocket motion and the motion of the track over the road wheels, idler, and carrier.

To validate the subsystem:

1. Run a simulation for **5** seconds and **250** steps.

2. View the animation by using the animation control buttons.

3. Select the **Exit** arrow to go to the assembly mode.

Chapter

5

Developing and Running the Full-vehicle Model

In this chapter, you will develop a full-vehicle model by making a copy of the completed track system and attaching both track subsystems to the chassis. You will build a terrain and then run the tracked vehicle over the terrain.

Task Objective

Learn to combine two subsystems with other geometry at the main model level and how to define a terrain.

Estimated Time to Complete

30 minutes

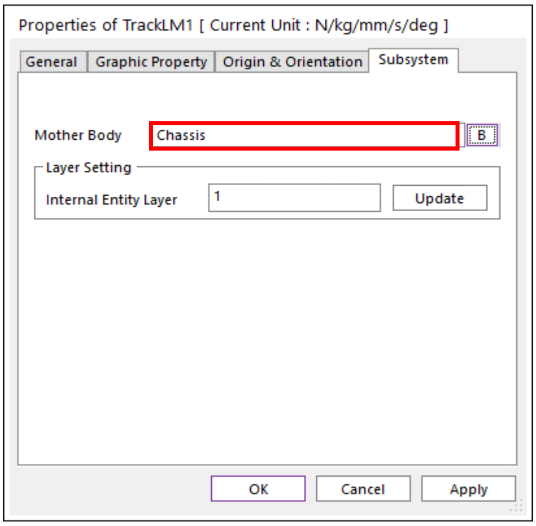
Adjusting the Subsystem Properties

There are two subsystem properties that you must adjust at the model level:

- Change the name of the subsystem to make it easier to identify.
- Define the mapping so the Mother Body in the subsystem corresponds to the Chassis body at the Model level. This mapping is very useful because it helps you simulate the subsystem in a fixed position (attached to the Mother Body) while also properly attaching the subsystem to the Chassis when you simulate at the Model level.

To adjust the subsystem properties:

1. Display the Properties dialog box for **TrackLM1** subsystem.
2. Click the **General** tab and change the name of the subsystem to **Right_Track_Assy**.
3. Display the Properties dialog box for the track subsystem.
4. In the **Subsystem** tab, change the **Mother Body** to the **Chassis** body.
5. Click **OK**.



Positioning and Testing the Right Track Assembly

You will move the right track subsystem into position with respect to the chassis.

To position the subsystem:

- Select the **Right_Track_Assy** subsystem, and display the **Basic Object Control** dialog box to move the subsystem:
  - Enter **250** mm in the **Offset Value** text box and click **+X**.
  - Enter **300** mm in the **Offset Value** text box and click **–Y**.
  - Enter **1000** mm in the **Offset Value** text box and click **+Z**.

To test the subsystem:

1. Add a fixed joint between the chassis and ground to hold up the chassis for this test. The location of three fixed joint does not matter.
2. Run a simulation for **5 seconds** and **250 steps** and compare the results to the previous animation.

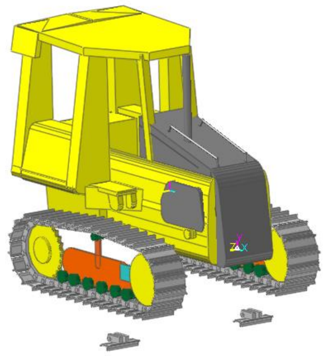
Creating the Left Track Subsystem

You will create the left track subsystem by making a copy of the existing track subsystem and by moving the new track subsystem to the other side of the chassis. You will then test this step.

To create the left track subsystem:

1. From the **Setting** group in the **Home** tab, click **General**.
2. If needed, clear the selection of the box next to **Shift When Pasting** (you may have already cleared this box when you were creating the road wheels).
3. Click **OK** to exit the **General Settings** dialog box.
4. Select the right track subsystem, copy it, and paste it (this will take a few seconds).
5. Display the Properties dialog box for the new track assembly and do the following:
    - Click the **General** tab, if necessary, and change the name to **Left_Track_Assy**.
    - Click the Origin & Orientation tab, and change Origin to 250, -300, -1000.
    - Click the **Subsystem** tab, and change **Mother Body** to **Chassis**.
    - Click **OK**.

    Your model should appear as shown on the right.



To test the subsystem:

- Run a simulation for 5 seconds and 250 steps and compare the results to the previous animation.

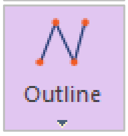
Defining a Terrain

Edit the ground body and define a terrain.

To define the terrain:

No.	X	Y
1	-3,500	-1,600
2	250	-1,350
3	3,700	-250
4	6,500	0
5	9,050	-450
6	11,450	-1,350
7	13,950	-2,600
8	15,750	-3,250
9	16,850	-3,400
10	20,100	-3,400
11	20,500	-3,250
12	21,050	-3,100
13	21,450	-3,100
14	21,850	-3,250
15	22,350	-3,400
16	24,600	-3,400
17	29,200	-2,100

1. To enter the ground edit mode, click the **Ground** of the **Marker and Body** group in the **Professional** tab.



2. Click the **Outline** of the **Curve** group in the **Ground** tab.
3. Create an outline by using the data in the table to the right.
4. When you finish selecting the points, right-click and click **Finish Operation**.

Tip: Another option is for you to create a dummy outline and import the coordinates from the file **Terrain_Outline:**f

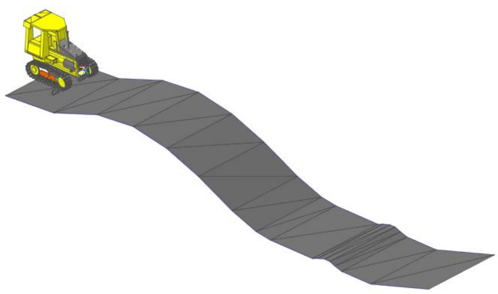
1. Create an outline with two points that are anywhere on the screen. Right-click and select **Finish Operation**.
2. Right-click the outline and select **Properties**.
3. Click **Import** and select the file **Terrain_Outline**(.mat file extension, **The file location:** <Install Dir>/Help/Tutorials/Toolkit/Track_LM/LowMobilityTrackedVehicle).

5. Make a copy of the outline (copy and paste) and move the outline **2500** mm in the +z direction and the first outline **2500** mm in the –z directions.

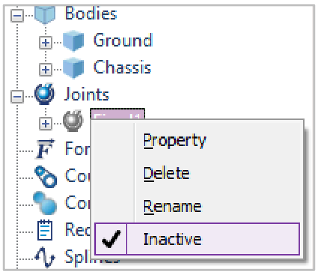
6. To create a terrain (road surface), click the **Spline Road** of the **Road Data** group in the **Ground** tab.
7. Click the near outline, then the far outline, then right-click and click **Finish Operation**.
   - **RecurDyn** creates the road data entity.

8. Select the **Exit** arrow to return to the assembly mode.

   The result is shown in the right figure.



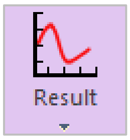
To test the full-vehicle model with the terrain:



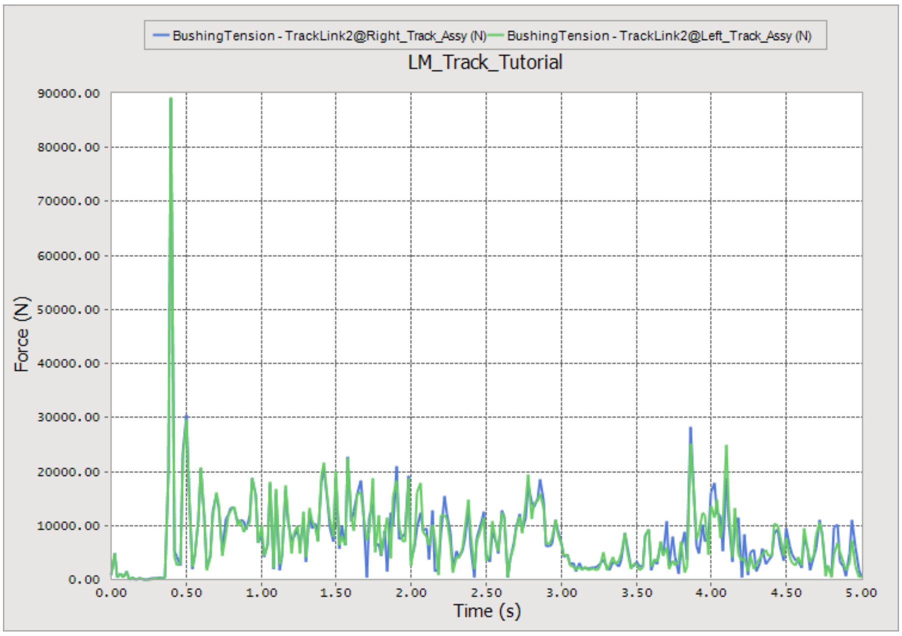
1. In the Database window, expand **Joints**, and right-click the name of the fixed joint (between the chassis and ground) that appears and select **Inactive** to make the joint inactive.

   The joint icon in the Database window and in the graphics window turns gray. You are making the fixed joint inactive to allow the bulldozer to drop down and make contact with the terrain.

2. Run a simulation for **5 seconds** and **250 steps** and compare the results to the previous animation.



3. Display the **Plot** window and plot the tension in the tracks by expanding the **Track-Force**, expanding each track link entity and double-clicking **Bushing Tension**.



4. If you have the time, run the simulation again, but this time for **12 seconds** and **600 steps**. You will see that the bulldozer will traverse much of the terrain.

5. Save the **RecurDyn** model.

The remaining sections of the tutorial contain optional exercises.

Chapter
6

Track Subsystem Tuning (Optional)

In this chapter, you will consider a variety of topics related to the detailed modeling of tracked vehicles. In the earlier chapters of this tutorial, you accepted many default values as you created the track components and the track assembly. Here you will find out how to modify some interesting track system values.

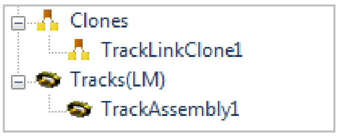
Task Objective

Learn to work with the various value and parameters associated with the track assembly and the track components. As you use the LM Track toolkit to model your own products, you will adjust these parameters according to your engineering and test data to develop validated models.

Estimated Time to Complete

20 minutes

Adjusting the Track Assembly Parameters

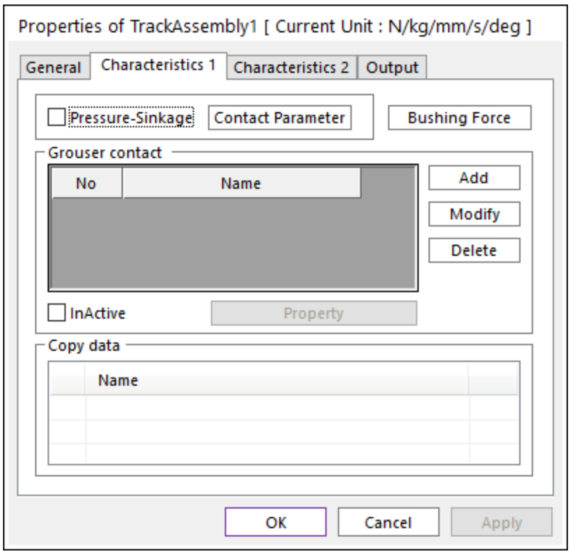


When you execute the **Track Assembly** operation, **RecurDyn** creates a **Track Assembly** entity. It appears near the bottom of the Database window as shown in the figure on the right. Right-click the Track Assembly entity and click **Properties** to display its Properties dialog box. It contains several tabs, which are explained in the next sections.

General Tab:

Use the **General** tab to modify the **Name** of the **Track Assembly**.

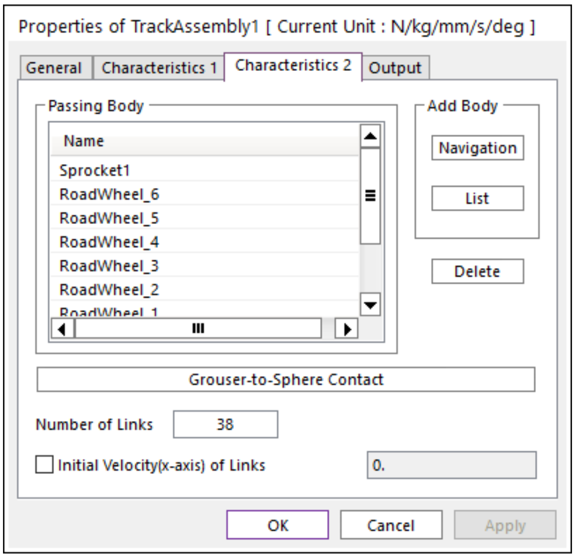
Characteristics 1 Tab:



Use the following options:

- Set track stiffness by clicking the Bushing Force.
- Set the translational and rotational stiffness and damping values for all degrees of freedom for the two bushings used to connect each track link to the next.
- Set the track tension by setting a preload in the Radial direction of the bushing force.
- Define the interaction between the track and a soft soil (clays, sands, loams, even snow) by selecting Pressure-Sinkage and then clicking Contact Parameter. You can select the soil type from the select list and click Load to load the Bekker parameters for that soil. You can modify the parameters and save them to a file to create your own library of soils.

Characteristics 2 Tab:

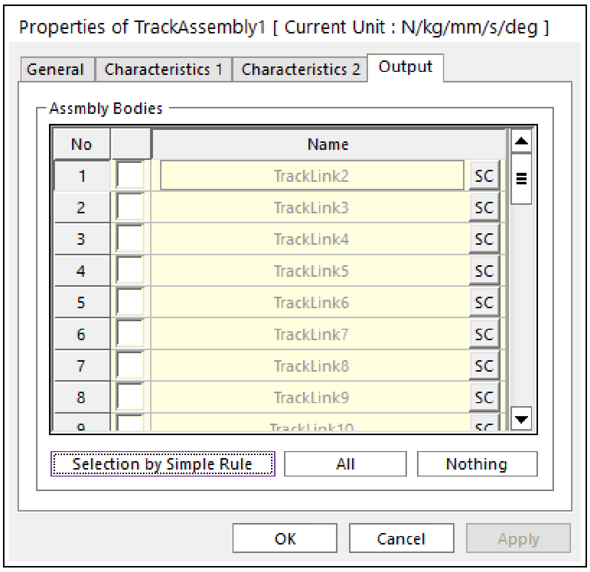


The Characteristics 2 tab is used to define or modify the list of entities that the track contacts.

- If you want to add another carrier roller, you will create the new roller and then use Navigation (for clicking from the graphics screen) or List to select the new roller from a list.
- Use Grouser-to-Sphere Contact to select spheres that have been defined to represent rocks in the terrain or any other supplemental contact with the track.
- Set the Initial Velocity of the track by checking Initial Velocity (x-axis) of Links box at the bottom of the dialog box and entering the desired initial velocity in the text box.

Output Tab:

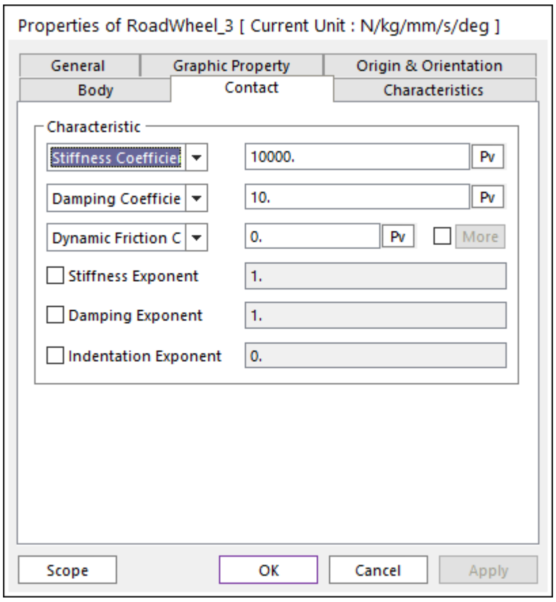
You can use the Output tab to customize which track links have their outputs saved from the simulation. The default is to save only the data from the first link, TrackLink2. You can manually check the boxes or use **Selection by Simple Rule** to automate the selection of links according to a numbering increment.



Adjusting the Track Component Parameters

Contact between components and the track:

The contact characteristics between any track wheel or sprocket to the track assembly can be different. For that reason, the contact parameters are set in the **Contact Characteristic** tab of each component. You can set the stiffness, damping, and friction according to the materials used in each component and the state of the component (that is, according to wear).



Setting Up a Mechanical Track Tensioner

To create a mechanical track tensioner, do the following for each track subsystem.

Replace the Fixed joint with a Translational joint:

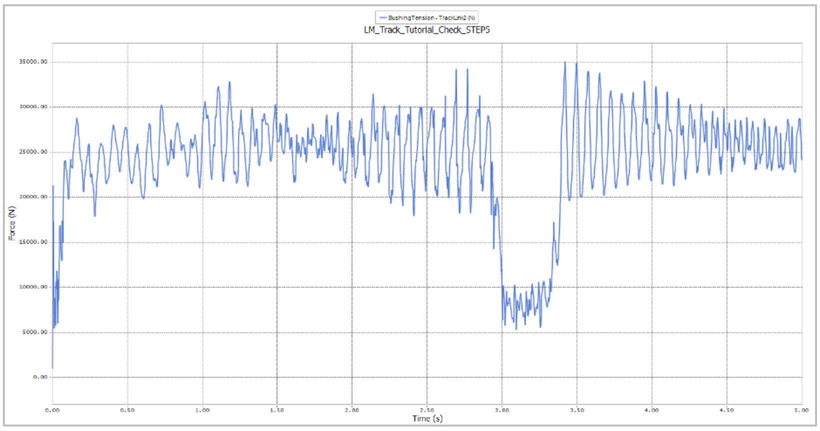
1. Enter the **Right_Track_Assy** subsystem edit mode. (if necessary)
2. Delete the **Fixed** joint between the **Track_Frame** and the **Tensioner** by selecting the name **Fixed1** in the Joints section of the database window, clicking the right mouse button with the cursor still over the name, and selecting the **Delete** option in the menu that pops up.
3. Create a **Translational** joint between the **Track_Frame** and the **Tensioner** by selecting the **Translate** icon of the **Joint** group in **Home** the tab and selecting Body, Body, Point, Direction input method.
    - **Body1**: Track_Frame
    - **Body2**: Tensioner
    - **Joint Origin Point: -1200, 0, 0** (Use markers remaining from Fixed joint)
    - **Position Vector: Tensioner.Box1 (1,0,0)** (Select the horizontal edge on the top of the Tensioner body or select the edge using the Flip List to orient the joint to be horizontal)

Add a Tensioning Spring

1. Add a **Spring** (of the **Force** group in the **Professional** tab) between the **Track_Frame** and the **Tensioner** (you can select the locations of the ends of the spring).
    - **Body1**: Track_Frame
    - **Body2**: Tensioner
    - **Point 1**: -1500, 0, 0 (on the Track_Frame)
    - **Point 2**: -1200, 0, 0 (on the Tensioner)
2. Bring up the **Properties** of the **Tensioner Spring** by placing the cursor over the **Spring1** in the Force group of the Database window, clicking the right mouse button, and selecting the **Properties** option.
3. Under the **Spring** tab, set the parameters listed below. The spring is set to a low stiffness value so that the spring force remains close to a constant value as the Tensioner stretches the track.
    - **Stiffness Coefficient**: 0.1
    - **Damping Coefficient**: 100
    - **Pre Load**: 50000
4. Under the **Graphic** tab, set the **Spring Diameter** to **50**.
5. Click **OK** to exit the Properties dialog box.

Validate the Track Tensioner

1. Run a simulation for **5** seconds, **250** Steps, and with the **Plot Multiplier** set to 8.
2. Play back the animation and you will see that the idler stretches forward by a small amount in response to the track tensioner.
3. Open the plot window and open the **Track-Force** group, open the **TrackLink2**, plot the **BushingTension**. You will see that the average track tension is approximately 50% of the force in the Tensioner Spring (25,000 N). The low tension occurs when TrackLink2 is in contact with the sprocket.



4. Adjust the spring properties as needed to attain the desired track tension.
5. Repeat what you just did (replace fixed joint by translation joint and spring) at **Left_Track_Assy** for next optional tutorial.

Chapter

7

Adding the Blade Linkage (Optional)

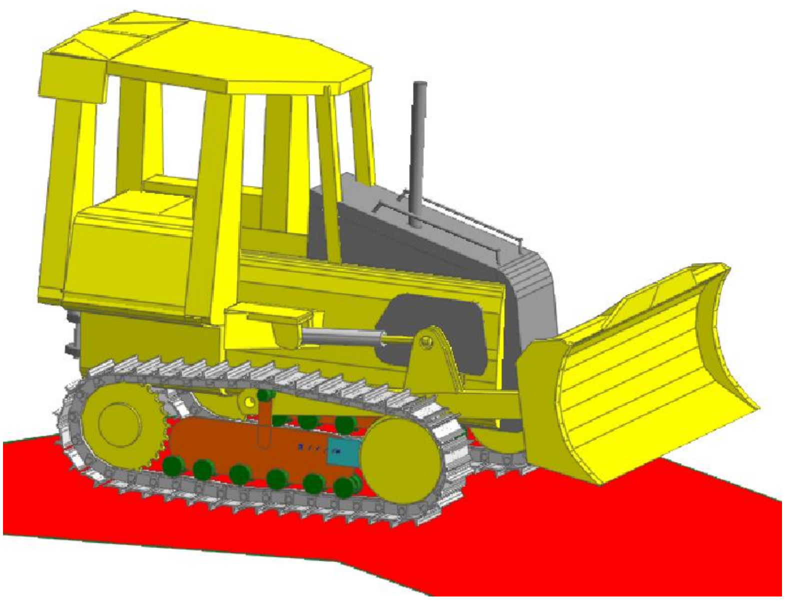
In this chapter, you will add a blade assembly (one blade and two side rails to the bulldozer model that you developed earlier. You will add hydraulic cylinders and the constraints needed to raise and lower the blade. You will run the updated bulldozer over the terrain.

Task Objective

Learn how to use predefined subsystems and how to add constraints between bodies at the model level and bodies in a subsystem.

Estimated Time to Complete

30 minutes



Saving a New RecurDyn Model

Because this is an optional exercise, it is a good idea to save the **RecurDyn** model to a new file so that at a later time you can separately refer to the basic bulldozer model, as well as to the bulldozer model with a blade assembly.

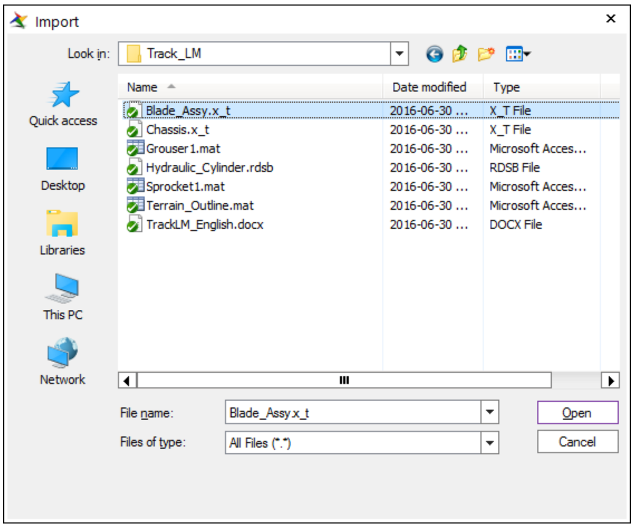
To save a new RecurDyn model:

1. From the **File** menu, choose **Save As**.
2. Enter the file name **LM_Track_Blade.rdyn**.

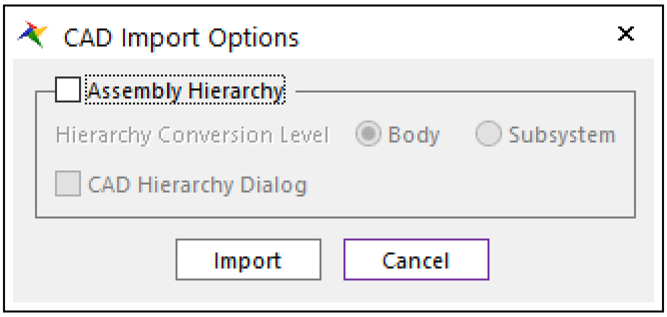
Importing and Aligning the Blade Assembly Geometry

The blade assembly consists of the blade and the two side rails. Once imported, the assembly needs to be moved into place with respect to the chassis.

To import the blade assembly geometry:



1. From the **File** menu, choose **Import**. (In the assembly mode).
2. In the Open dialog box, set file type to **ParaSolid File**, and from the **Track LM Files** folder, select the file **Blade_Assy.x_t** as shown in the dialog box on the right. (**The file location:** <Install Dir> /Help /Tutorials /Toolkit /Trak_LM /LowMobilityTrackedVehicle , ask your instructor for the location of the directory if you cannot find it).
3. Click **Open**. The **CAD Import Options** window appears. Clear the **Assembly Hierarchy** checkbox and click the **Import** button.



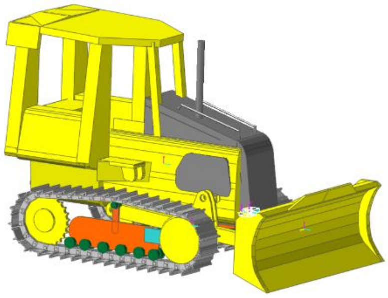
To place the blade assembly geometry:

Display the Basic Object Control window.

- Make sure that ImportedBody1, ImportedBody2, and ImportedBody3 are selected.
- Select the **Rotate** tab, set **Degree** to **180**, and rotate about the global **Y** axis (in either direction) to flip the blade assembly around.

- Click the **Translate** tab, set the **Offset Value** to **3500**, and move the blade assembly in the –**X** direction.
- Set the **Offset Value** to **200** and move the blade assembly in the **–Y** direction.

The model should appear as shown in the figure on the right.



To name the blade assembly bodies:

- Display the **Properties** dialog box for each blade assembly body and rename each as follows.
  - **ImportedBody1** to **Right_Rail**
  - **ImportedBody2** to **Left_Rail**
  - **ImportedBody3** to **Blade**

Adding Hydraulic Cylinders

A hydraulic cylinder consists of the upper cylinder body and a cylinder rod. It can be tedious to create and align the cylinder bodies with the two connecting points. Instead, you can import a hydraulic cylinder system and then position them using parametric points.

To import the hydraulic cylinder subsystem:

1. From the **File** menu, choose **Import**.
2. In the Open dialog box, the file type should default to **RecurDyn Subsystem File**. If not, select that file type.
3. From the **Track LM Files** folder, select the file **Hydraulic_Cylinder.rdsb** as shown in the dialog box. (**The file location:** <Install Dir>/Help / Tutorials /Toolkit /Track_LM /LowMobilityTrackedVehicle, ask your instructor for the location of the directory if you cannot find it).
4. Click **Open**.

   The **Import SubSystem** window appears.
5. Click **OK** to create the hydraulic cylinder subsystem.

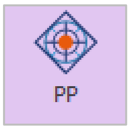
To create the second hydraulic cylinder subsystem:

1. In the **Database** window, click the **SubSystem1** and use the **Copy** and **Paste** commands to create a second hydraulic cylinder subsystem, **C1_Subsystem1**.
2. Display the **Properties** dialog box for each subsystem and rename each as follows.
   - **SubSystem1** to **Left_Cyl**
   - **C1_Subsystem1** to **Right_Cyl**

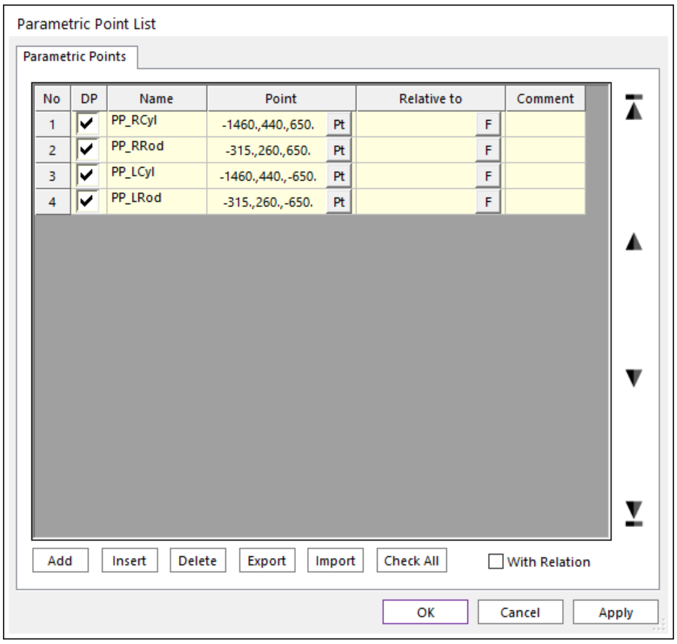
Positioning the Hydraulic Cylinders

There are parametric points in the hydraulic cylinder subsystems. The hydraulic cylinders will position themselves if you define parametric points at the attachment points of the cylinders in the model level and link (connect) those parametric points to certain parametric points in the subsystems.

To define the parametric points at the model level:

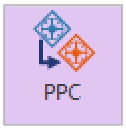


1. From the **Parameter** group in the **SubEntity** tab, click the **Parametric Point (PP)**.
2. Create four parametric points and fill in the text boxes as shown in the figure and as explained below.



   - Click **Add** four times.
   - Name the first point as **PP_RCyl** and the second point as **PP_RRod**.
   - Assign the first point the coordinates of **-1460, 440, 650** and the second point the coordinates of **-315, 260, 650**.
   - Copy the data from the first point to the third point and the data from the second point to the fourth point. Change the **R** to **L** in the names as needed to indicate Left rather than Right. Change the sign of the **Z-coordinates** to be **negative** to put the third and fourth points on the other side of the chassis.
3. Click **OK** to create the points and exit the Parametric Points window.

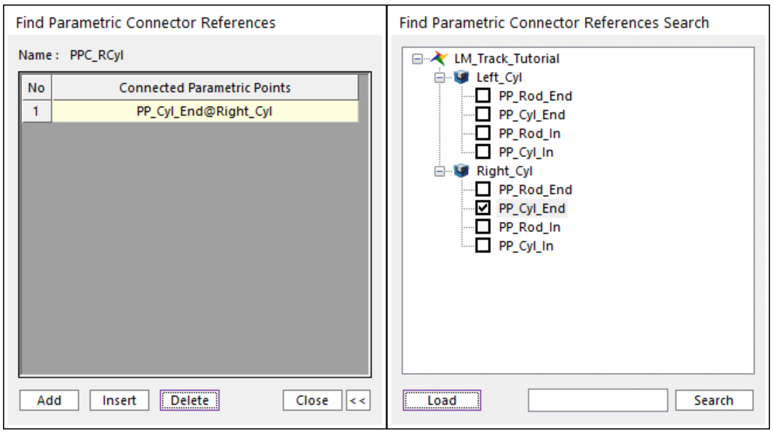
To link the parametric points and position the hydraulic cylinders:



No	Name	Point		Refs	
1	PPC_RCyl	PP_RCyl	Pt	0	...
2	PPC_RRod	PP_RRod	Pt	0	...
3	PPC_LCyl	PP_LCyl	Pt	0	...
4	PPC_LRod	PP_LRod	Pt	0	...

1. From the **Parameter** group in the **SubEntity** tab, click the **Parametric Point Connector (PPC)**.
2. In the **Parametric Point Connector List** window, click **Add** four times.
3. Name the first connector **PPC_RCyl** and the second point **PPC_RRod**.
4. Click the **Pt** button for the first connector and drag the name **PP_RCyl** parametric point by using the mouse or input directly the position information in the **Input window Toolbar**.

5. Click the **...** button.
   - The Find Parametric Connector References dialog box appears.
6. Click **>>** in the lower right corner.
   - The **Search** dialog box appears.
7. Check the box for **PP_Cyl_End** for the **Right_Cyl** subsystem. Then, click the **Load** button.



8. Click **Close** in the Find Parametric Connector References dialog box.
9. Create four parametric points and fill in the text boxes as shown in the figure and as explained below.

Parametric Point Connector	Parametric Point Name (model level)	Parametric Point Name (subsystem level)
PPC_RCyl	PP_RCyl	PP_Cyl_End@Right_Cyl
PPC_RRod	PP_RRod	PP_Rod_End@Right_Cyl
PPC_LCyl	PP_LCyl	PP_Cyl_End@Left_Cyl
PPC_LRod	PP_LRod	PP_Rod_End@Left_Cyl

10. Click **OK** to close the **Parametric Point Connector List** window and create the connectors.

    The two cylinders are now located in the proper locations.

Adjusting Size, Colors, and Motion of Hydraulic Cylinders

Now that the hydraulic cylinders are in position, it is easy to see that the size of the cylinder components is too small. You should also adjust the color of the components to match the bulldozer. Finally, you need to add a motion input to the translational joint to contract the hydraulic cylinders by 100 mm so that the blade is in a transport position.

To adjust the size of the components of each hydraulic cylinder:

1. Enter the **Edit** mode for one of the hydraulic cylinders.
2. Edit the **Parametric Values** for the hydraulic cylinder subsystem:
   - Set **PV_Cyl_Radius** to **60**.
   - Set **PV_Rod_Radius** to **30**.

To adjust the color of the components of each hydraulic cylinder:

1. Display the Properties window for **Body1** (cylinder rod) and change the color to **Gray-25%**.
2. Display the Properties window for **Body2** (upper cylinder) and change the color to **yellow**.

To adjust the motion of each hydraulic cylinder:

1. Display the Properties window for joint **TraJoint1**.
2. In the **Joint** tab, click **Include Motion** and click **Motion**.
3. Click **EL** to display the **Expression List** window.
4. Click **Create** to display an **Expression** window.
   - Change Name to Exp_Cyl_Length
   - Enter **STEP(TIME, 0.1, 0.0, 0.5, -100)** in the text box.
   - Click **OK** to exit the **Expression** window.
5. With Expression 7 selected, click **OK** to exit the **Expression List** window.
6. Click **OK** to exit the Motion window.
7. Click **OK** to exit the TraJoint1 window.

8. Click the **Exit** arrow to return to the assembly mode.
9. Repeat steps 1 through 12 above for the other cylinder.

Defining Constraints for the Blade Assembly

In this section, you will create two revolute joint, four spherical joints, a **CMotion** constraint, and a pair of bushings to properly attach the blade assembly components to each other and to the chassis.

To define the constraints:

- Using the tools of the Joint group in the Professional tab, create the following joints as specified in the table below. Use the Body, Body, Point option to make sure that you are referencing the correct set of bodies, especially when one of them is within one of the two hydraulic cylinder subsystems (see tip below).

Tip: You will notice that when creating a joint, force, or contact; you cannot select a body that is within a subsystem in the normal fashion. However, if you first press down on the **Shift** key, you will find that are able to select the body with the subsystem.

Create the connection of type	First Body	Second Body	At the location
Joints:			
Revolute	Chassis	Body2@Right_Cyl	-1460, 440, 650
Revolute	Chassis	Body2@Left_Cyl	-1460, 440, -650
Spherical	Chassis	Right Rail	-1775, -90, 650
Spherical	Chassis	Left Rail	-1775, -90, -650
Spherical	Body1@Right_Cyl	Right Rail	-315, 260, 650
Spherical	Body1@Left_Cyl	Left Rail	-315, 260, -650
CMotion	Right Rail	Blade	885, -130, 0 (for both points)
Forces:			
Bushing	Right Rail	Blade	659, -280, 650
Bushing	Left Rail	Blade	659, -280, -650

Adjust the CMotion to fix the blade angle:

1. Display the Properties dialog box for **CMotion1**.
2. Set the **Cartesian Motion Type** to **Displacement (time)** and **RZ**.
3. Create an expression that has a **constant_angle_of_zero** (0.0).

To test the full vehicle model with the blade assembly:

- Run a simulation for 5 seconds and 250 steps and compare the animation and plot results to those of the previous animation.

You should observe that the blade rises up to a transport position and then remained fixed as the bulldozer traverses the terrain. You may notice a small vertical motion of the blade relative to the linkage when the bulldozer hits the terrain near the beginning of the simulation. This is due to the compliance of the bushings.

Do you observe any difference in the overall behavior of the bulldozer? You should not see any large changes in behavior, but small changes due to the forward transfer of the center of gravity of the bulldozer due to the weight of the blade assembly. You could plot the vertical loads at the sprocket and the idler for the two cases and see the effect of the weight transfer.

Thanks for participating in this optional exercise!