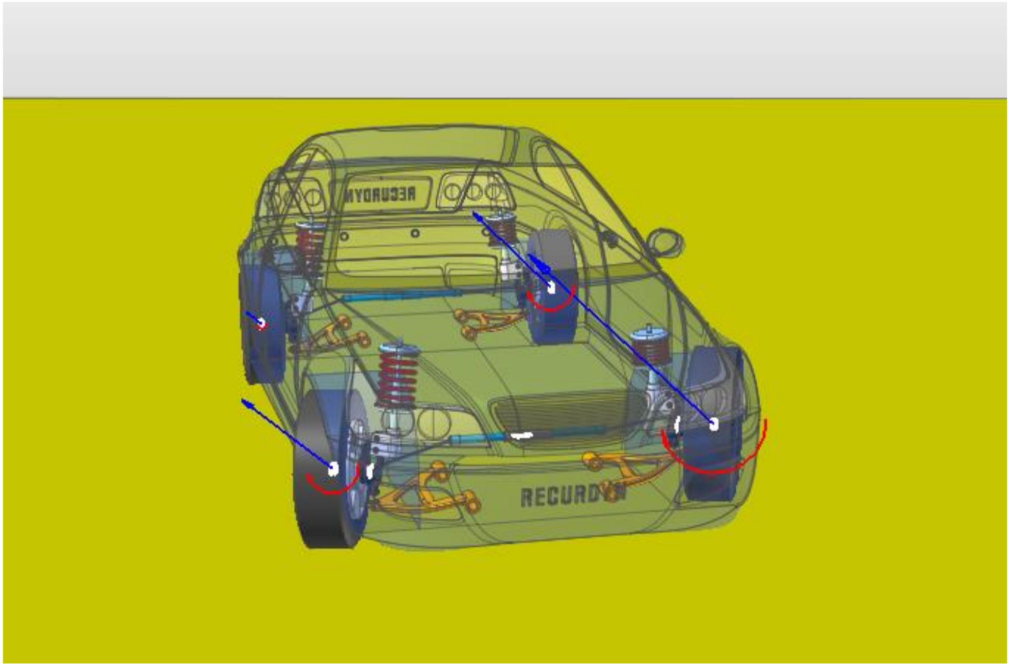
RECURDYN

Driving J-Turn Tutorial (Tire)



FunctionBay

Copyright © 2020 FunctionBay, Inc. All rights reserved.

User and training documentation from FunctionBay, Inc. is subjected to the copyright laws of the Republic of Korea and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. FunctionBay, Inc. hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the FunctionBay, Inc. copyright notice and any other proprietary notice provided by FunctionBay, Inc. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of FunctionBay, Inc. and no authorization is granted to make copies for such purpose.

Information described herein is furnished for general information only, is subjected to change without notice, and should not be construed as a warranty or commitment by FunctionBay, Inc. FunctionBay, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the Republic of Korea and other countries. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks of FunctionBay, Inc. or Subsidiary

RecurDyn is a registered trademark of FunctionBay, Inc.

RecurDyn/Professional, RecurDyn/ProcessNet, RecurDyn/Acoustics, RecurDyn/AutoDesign, RecurDyn/Bearing, RecurDyn/Belt, RecurDyn/Chain, RecurDyn/CoLink, RecurDyn/Control, RecurDyn/Crank, RecurDyn/Durability, RecurDyn/EHD, RecurDyn/Engine, RecurDyn/eTemplate, RecurDyn/FFlex, RecurDyn/Gear, RecurDyn/DriveTrain, RecurDyn/HAT, RecurDyn/Linear, RecurDyn/Mesher, RecurDyn/MTT2D, RecurDyn/MTT3D, RecurDyn/Particleworks I/F, RecurDyn/Piston, RecurDyn/R2R2D, RecurDyn/RFlex, RecurDyn/RFlexGen, RecurDyn/SPI, RecurDyn/Spring, RecurDyn/TimingChain, RecurDyn/Tire, RecurDyn/Track_HM, RecurDyn/Track_LM, RecurDyn/TSG, RecurDyn/Valve

are trademarks of FunctionBay, Inc.

Edition Note

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

Table of Contents

Chapter

1

Overview

Task Objectives

The contact force between the tire and the ground is very important when analyzing the behavior of a running vehicle. It is difficult to express the tire contact force as general contact because friction, slip, and others arising from various dynamic loads must be considered. RecurDyn offers UIs and several types of Tire Solvers to easily define the contact force between the tires and the ground.

In this tutorial, you will learn how to create a UA-Tire-type tire using the GTire Toolkit. The GTire Toolkit allows you to easily create and define the geometries of tires and wheels, and you can easily change several types of Tire Solvers.

The model used in this tutorial is a front wheel drive model with the Macpherson Strut Suspension structure, which is configured to check the dynamic characteristics of the vehicle when the J-Turn is performed by changing the steering angle during driving.

- Vehicle modeling by importing CAD and subsystem files
- Create GTire of GRoad- and UA-Tire-type
- Analysis of driving by P control and Steering control
- Tire Property modification and result analysis
-

Prerequisites

This tutorial is for users familiar with Basic Tutorial provided by RecurDyn. Therefore, in order to use this tutorial, it is necessary to understand the above mentioned textbook first to improve the understanding of this tutorial.

Task

This tutorial consists of the following steps, and the time required is shown in the following table.

Tasks	**Time (min.)**
Setting up the simulation environment	5
Vehicle modeling	15
Analysis of driving	20
Modification and analysis of Tire Property	10
Change and analysis of GRoad	10
Total	**60**

Estimated Time to Complete this Task

75 minutes

Chapter

2

Setting up the simulation environment

Task Objectives

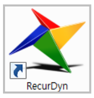
Set Unit, Gravity, and Icon Size to run the simulation.

Estimated Time to complete this task

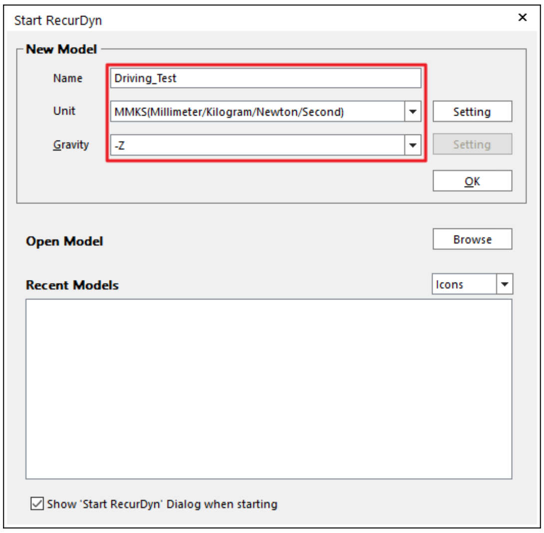
5 minutes

Start RecurDyn

Creating a new model



1. Double-click the **RecurDyn** icon on the desktop.
2. **RecurDyn** runs and the **Start RecurDyn** dialog box appears.
3. In the **Name** field, enter the name of the model as **Driving_Test**.
4. Set the **Unit** to **MMKS**.
5. Set **Gravity** to **–Z**.
6. Click **OK**.

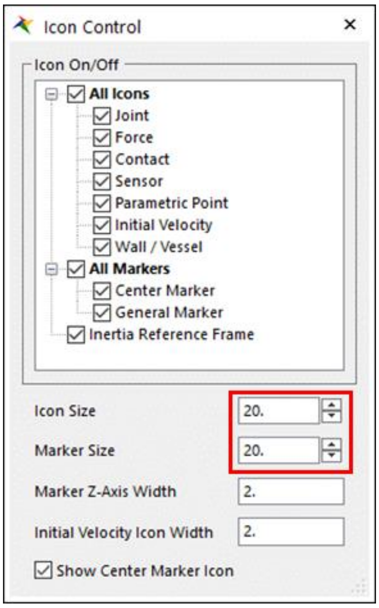


Resizing icons and markers

Adjust the size of icons and markers to make the model more visible.

Changing the size of icons and markers

7. On the **View Control** toolbar, click **Icon Control**.
8. **Icon Control** dialog box appears.
9. Set Icon Size and Marker Size to 20.
10. Close **Icon Control** dialog box.



Chapter

3

Vehicle modeling

Task Objectives

Vehicle's CAD files and subsystem files are imported to define the chassis and suspension, and GRoad and GTire are created to complete the vehicle modeling.

Try the following modeling.

- Import CAD files to create Chassis Body
- Import subsystem files to create Suspension
- Create GRoad by using Box Geometry
- Creating GTire of UA-Tire type

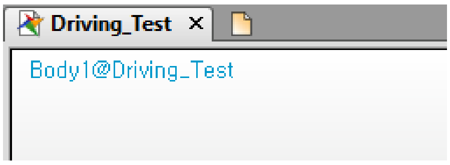
Estimated Time to complete this task

15 minutes

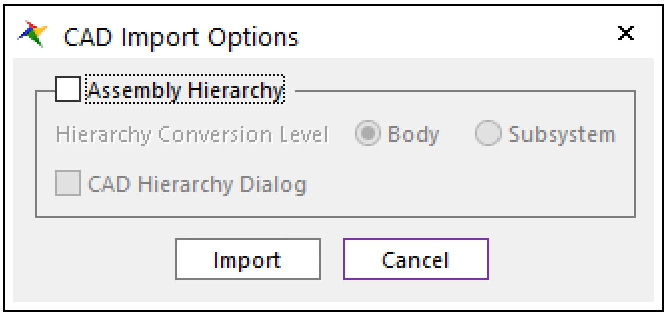
Creation of Chassis Body

Creating Chassis Geometry

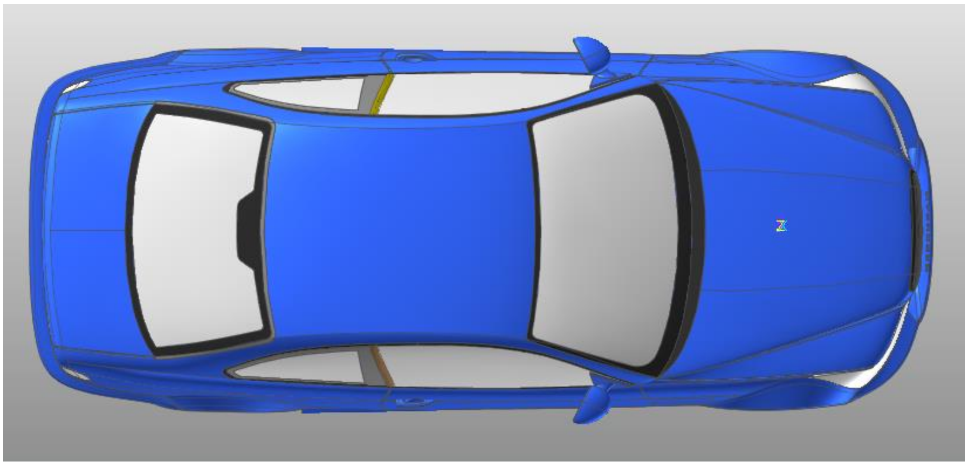
1. To enter the Edit mode of the Body, click **General** in the **Marker and Body** group on the **Professional** tab.
   - **Body1@Driving_Test** is displayed in the upper left corner of the Working Window as shown on the right.

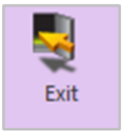


2. In the **File** menu, click **Import**.
3. Set Files of type to ParaSolid File (*.x_t,*.x_b …).
4. Select the **Chassis.x_t** file in the GTire tutorial folder. (File path: <InstallDir> /Help /Tutorial/Toolkit/Tire/DrivingJTurn/Chassis.x_t)
5. Click the **Open** button. The **CAD Import Options** window appears. Clear the **Assembly Hierarchy** checkbox and click the **Import** button.



6. The message window displays a message that the import has been successfully completed, and the Chassis is shown as follows.

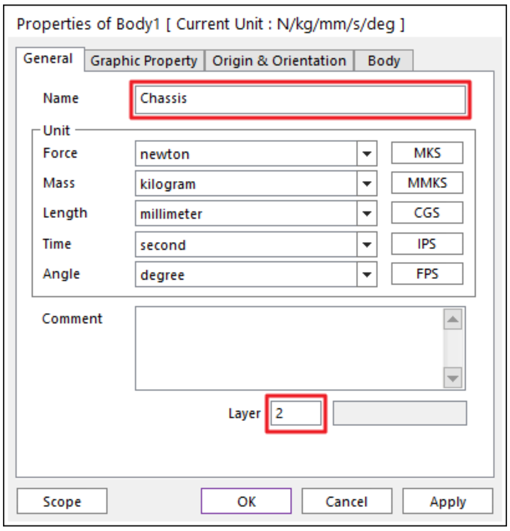




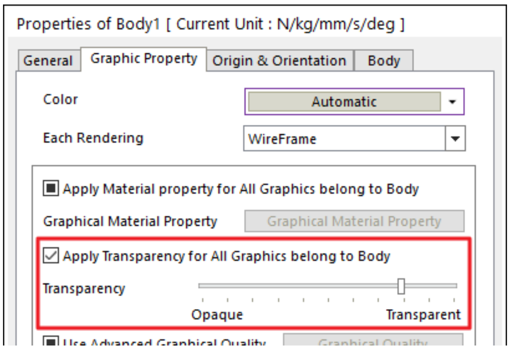
7. Click **Exit** on the **Geometry** tab to exit the Body Edit mode.

Changing the property of Body1

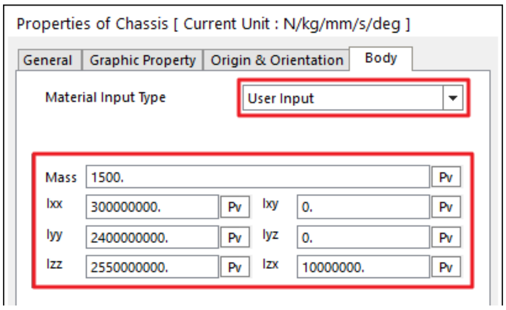
1. Open the property dialog box for **Body1**.
2. Click the **General** tab and make the following changes.
    - **Name**: Chassis
    - **Layer**: 2



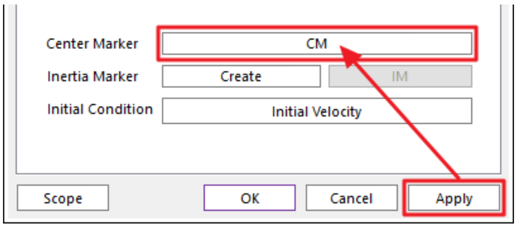
3. Click the **Graphic Property** tab.
4. Check **Apply Transparency for All Graphics belong to Body** and change the Transparency Level as shown on the right.



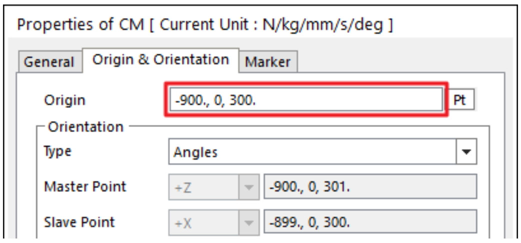
5. Click the **Body** tab.
6. The material input data of Chassis is very important in vehicle analysis. Therefore, it is necessary to apply it to the body property based on the experimental measurement value or the reliable data.
7. Change the **Material Input Type** to **User Input** and change the information as shown below.
    - **Mass**: 1500
    - **Ixx**: 300000000
    - **Iyy**: 2400000000
    - **Izz**: 2550000000
    - **Ixy**: 0
    - **Iyz**: 0
    - **Izx**: 10000000



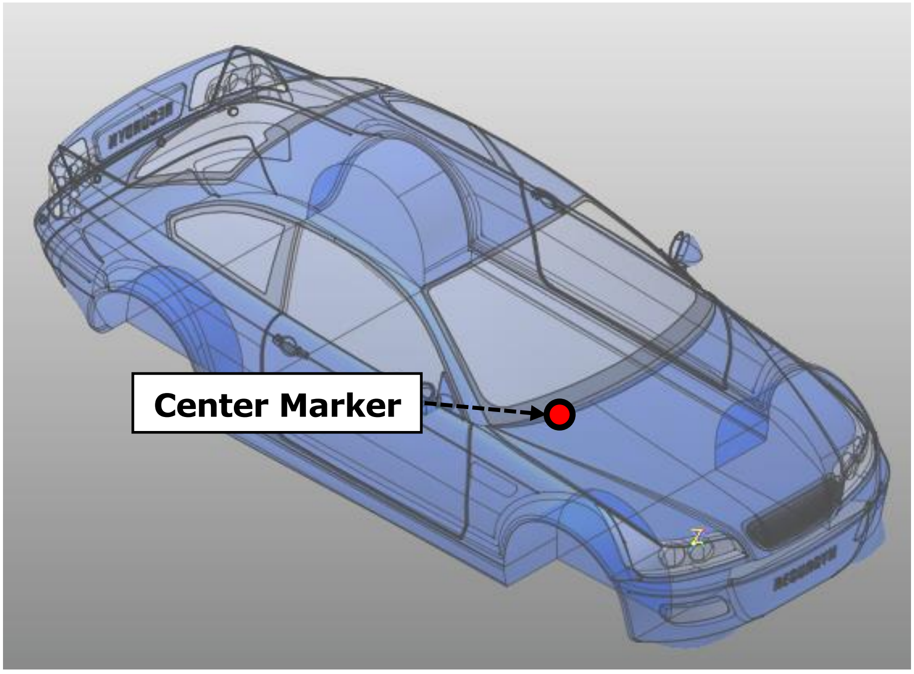
8. Click **Apply** to apply the **User Input** status.
9. If you do not apply the User Input status, Center Marker cannot be modified.
10. Click **CM** to open the property dialog box of **Center Marker**.



11. In the Property dialog box of the CM, change **Origin** to **-900, 0, 300** and click **OK**.
12. Click **OK** of the property dialog box of body to apply changes.



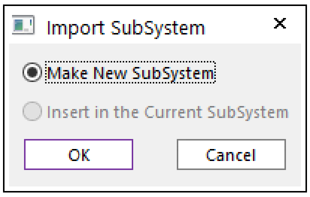
13. **Chassis** is shown as below.



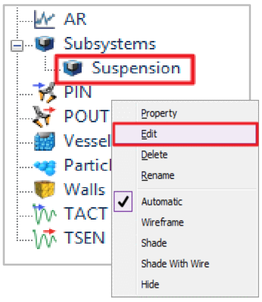
Creation of Suspension Subsystem

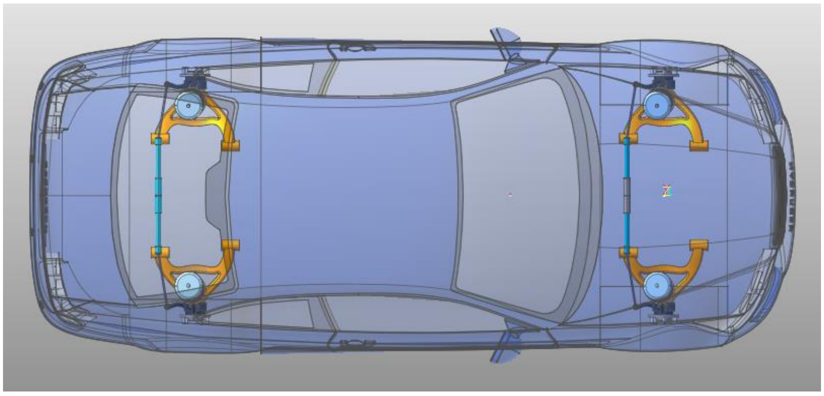
Loading Suspension Subsystem

1. In the **File** menu, click **Import**.
2. Set Files of type to RecurDyn Subsystem File (*.rdsb).
3. Select the **Suspension.rdsb** file in the GTire tutorial folder. (File path: <InstallDir> /Help /Tutorial/Toolkit/Tire/DrivingJTurn/Suspension.rdsb)
4. Click **Open** to open the Import Subsystem dialog box.
5. Click **OK**.

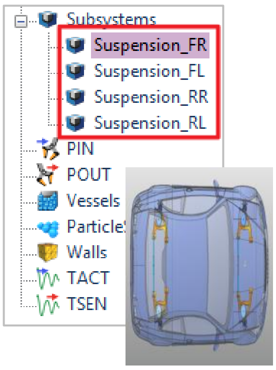


6. A subsystem called **Suspension** is created in the database and Suspension appears inside the chassis as shown below.

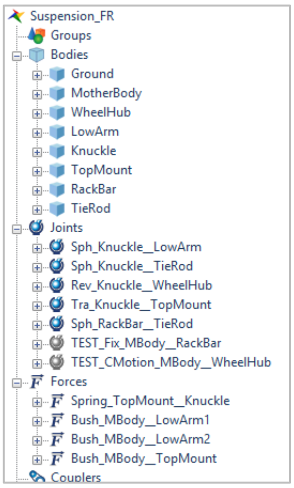


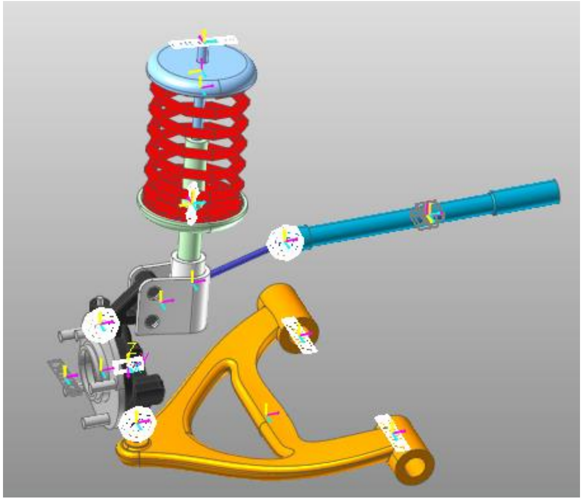


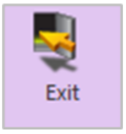
7. When entering the **Suspension** Subsystem from the database, it is divided into four subsystems in front, back, left and right.



8. Again, if you enter one of the four **Suspension** subsystems in the Database, you will see that the MBD modeling of each element is finally done.



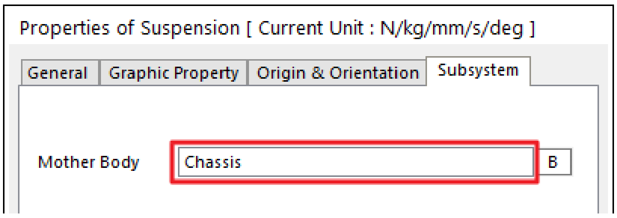




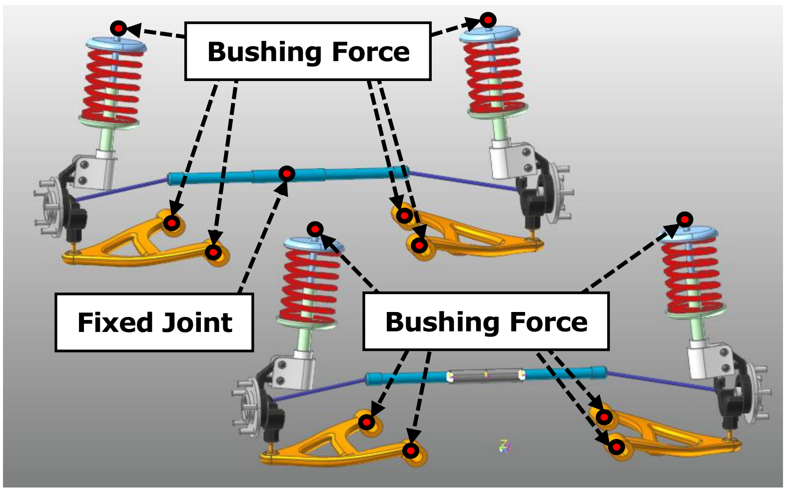
9. Once the subsystem analysis is complete, click **Exit** on the **Professional** tab to go to the top level.

Changing Subsystem Mother Body

1. Open the property dialog box for **Suspension** Subsystem.
2. On the Subsystem tab, change **Mother Body** to **Chassis**.
3. Click **OK**.



Entities shown in the figure below are modeled with Mother Body in Subsystem. In the above, the mother body is changed to the chassis, and the connection between the entities and Chassis is defined. In other words, you can see that **Suspension** is modeled with **Chassis**.



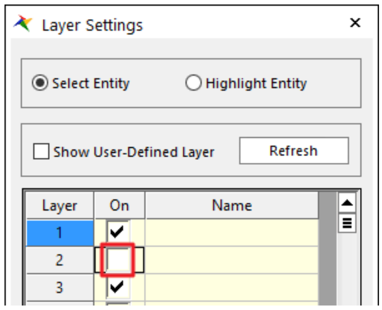
Create Translate Joint for Steering

There is no modeling of the steering in the Suspension Subsystem. Create a Translate Joint between Chassis and RackBar to configure the Steering System.

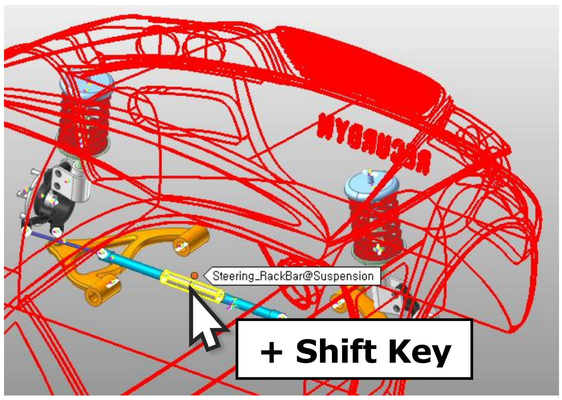
Creating Translate Joint

1. Since you need to click the **RackBar** hidden in the chassis, open the **Layer Settings** dialog box in advance.

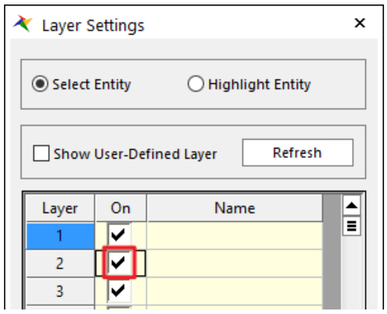
2. On the **Professional** tab, click **Translate** in the **Joint** group.
3. Set the Creation Method to **Body, Body, Point, Direction**.
4. In the Working Window, click **Chassis** as the first Body.
5. In order to click the **RackBar** hidden in the **Chassis**, uncheck **On** of **Layer 2** in the Layer Settings dialog box.



6. To click the RackBar in the Subsystem as the second Body, hold down the **Shift** key and click **Steering_RackBar@Suspension** as shown on the right.
7. When two bodies are selected, enter the remaining two values as shown below.
   - **Point**: -216, 0, 176
   - **Direction**: 0, 1, 0



8. Once **TraJoint1** is created, place a check mark in **On** of **Layer2** again in the Layer Settings dialog box and click **Close** to close the dialog box.

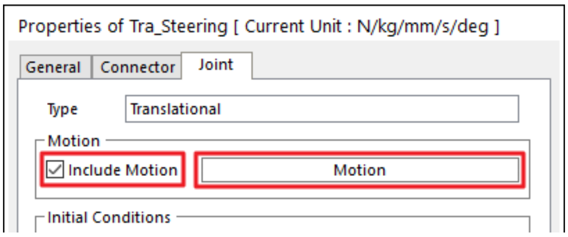


9. Open the Property dialog box for **TraJoint1** and change the **Name** to **Tra_Steering**.

Entering Motion in the Translate Joint

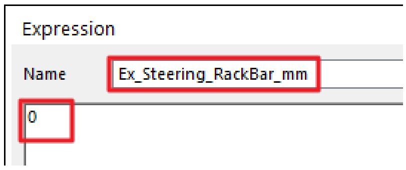
There must be no movement in the steering system to drive the vehicle straight ahead. Define the Expression so that there is no motion and enter it into the Motion of the Translate Joint.

1. Open the property dialog box for **Tra_Steering**.
2. On the Joint tab, click **Include Motion**.
3. Click **Motion** to open the Motion dialog box.
4. Set the **Type** to **Displacement(Time)** and click **EL**.

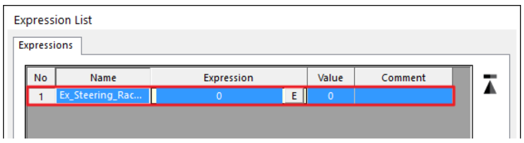


5. When the **Expression List** dialog box appears, click **Create** to open the dialog box.

6. When the **Expression** dialog box appears, enter as follows:
   - **Name**: Ex_Steering_RackBar_mm
   - Content: 0
7. In the **Expression** dialog box, click **OK**.

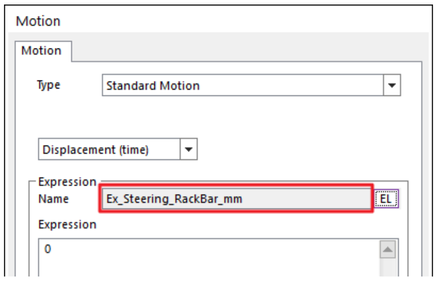


8. When the **Expression List** dialog box appears again, activate **Ex_Steering_RackBar_mm** in the list and click **OK**.

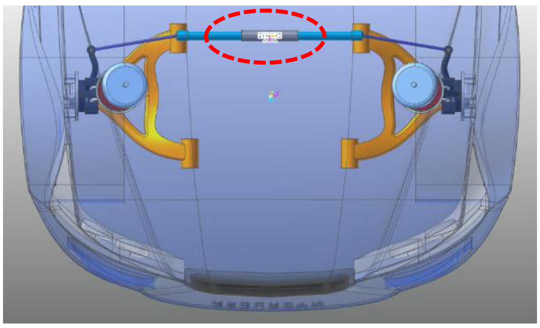


9. Make sure that Expression is entered in the **Motion** dialog box as shown in the figure on the right and click **OK**.

10. Click **OK** in the **Tra_Steering's Property** dialog box.



11. As shown in the figure on the right, Motion is displayed on the Translate Joint.

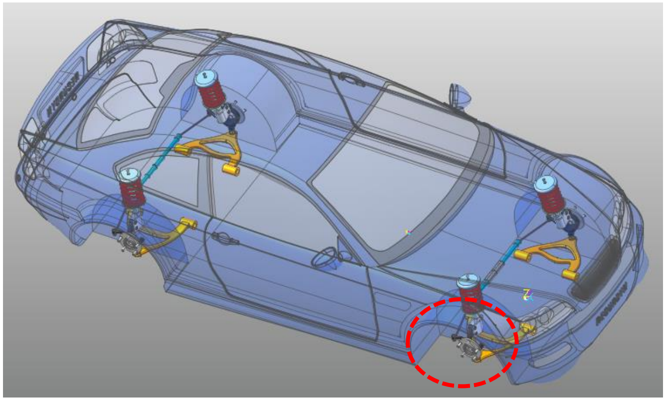


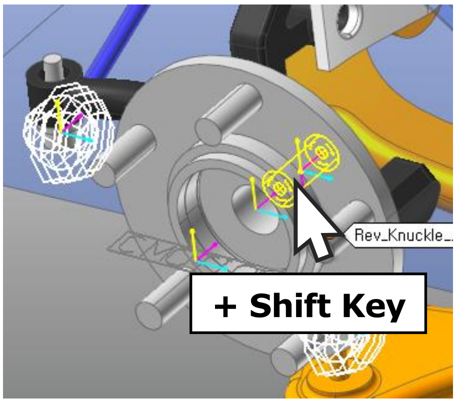
Creation of Rotational Axial Force for Power

Create a **Rotational Axial Force** on both **WheelHubs** of the front Suspension to represent the power of the front wheel drive vehicle. Create a rotational axial force by using Revolute Joint located between Knuckle and **WheelHub** already defined in Suspension Subsystem.

Creation of Rotational Axial Force

1. On the **Professional** tab, click **Rot.Axial** in the **Force** group.
2. Set the Creation Method to **Joint**.
3. To create the right front wheel power, hold down the **Shift** key and click **Rev_Knuckle__WheelHub@Suspension_FR@Suspension** at the center of WheelHub in the right subsystem.



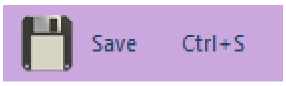


4. On the **Professional** tab, click **Rot.Axial** again in the **Force** group.
5. Set the Creation Method to **Joint**.
6. To create the left front wheel power, hold down the **Shift** key and click **Rev_Knuckle__WheelHub@Suspension_FL@Suspension** at the center of WheelHub in the left subsystem.

7. Open the Property dialog box for **RotationalAxial1** and change the **Name** to **Rot_Torque_FR**.
8. Open the Property dialog box for **RotationalAxial2** and change the **Name** to **Rot_Torque_FL**.

To save the model

- Basic vehicle modeling has been completed. On the **File** menu, click **Save** to save the model.



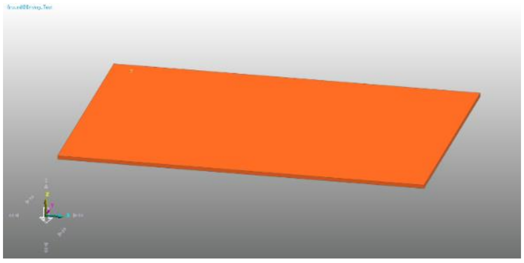
Creation of GRoad

GTire's Contact requires the definition of Groad unlike the general Contact entities.

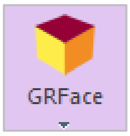
Creating Box Geometry for creation of GRoad

1. To enter the Edit mode of the Ground, click **Ground** in the **Marker and Body** group on the **Professional** tab.
2. When you enter Ground Edit mode, it will appear as **Ground@Driving_Test** in the upper left corner of the Working Window.

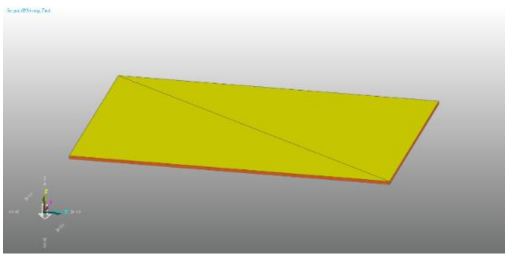
3. Click **Box** in the **Marker and Geometry** group on the **Ground** tab.
4. Set the Creation Method to **Point, Point** and enter as follows:
   - **Point**: -5000, 5000, -170
   - **Point**: 80000, -50000, -1000
5. **Box1** will be created as shown below.



Creating GRoad



1. Click **GRFace** in the **Road Data** group on the **Ground** tab, and type as below.
   - **Face**: Box1.Face1
   - **Direction**: 0, 0, 1 (Normal Vector)
   - **Direction**: 1, 0, 0 (Heading Vector)
2. When the Save As dialog box appears, save the **GRoad.rdf** file in the path where the model is saved.
3. GRoad1 will be created as shown below.



4. Click **Exit** on the **Ground** tab to exit the **Ground Edit** mode.

Creation of GTire

Solver types supported by GTire include UA-Tire, MF-Tire, F-Tire, etc. This tutorial uses the UA-Tire type, which is intuitive and easy to use.

Copying UA-Tire type Tire file

- Copy the **UATire_MMKS.tir** file from the GTire tutorial folder and paste it into where the model is saved. (File path: <InstallDir> /Help/Tutorial/Toolkit/Tire/DrivingJTurn/UATire_MMKS.tir)
- The example files used in this tutorial are defined for the UA-Tire Solver specification. The UA-Tire file configuration is as follows.

```
[HEADER]
FILE_TYPE                                   = 'tir'
$-----------------------------------------------------
[UNITS]
LENGTH                                      = 'mm'
FORCE                                       = 'newton'
ANGLE                                       = 'radians
MASS                                        = 'kg'
TIME                                        = 'second'
$-----------------------------------------------------
[MODEL]
VENDER_TIRE_TYPE                            = 'RD-UA'
$-----------------------------------------------------
[DIMENSION]
UNLOADED_RADIUS                             = 320
WIDTH                                       = 210
RIM_RADIUS                                  = 200
$-----------------------------------------------------
[PARAMETER]
$ vertical stiffness (Force/Length)
RADIAL_STIFFNESS                            = 190.0
$ vertical damping ratio
RADIAL_DAMPING_RATIO                        = 0.01
% longitudinal slip stiffness (Force/ratio)
LONGITUDINAL_STIFFNESS                      = 200000
% lateral slip stiffness (Force/angle)
LATERAL_STIFFNESS                           = 50000
% camber stiffness (Force/angle)
CAMBER_STIFFNESS                            = 3000
% rolling registance length (Length)
ROLLING_RESISTANCE                          = 3
% maximum friction coefficient
FRICTION_MAX                                = 1.1
% minimum friction coefficient
FRICTION_MIN                                = 0.8
```

Unit

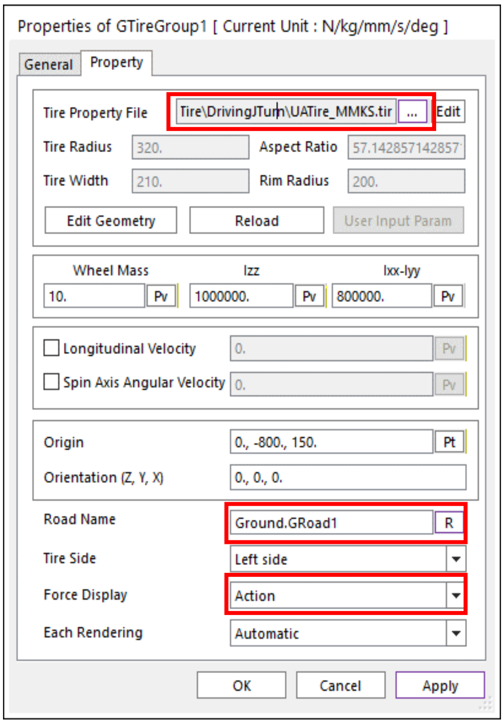
Solver Type

Tire Feature

Contact Parameter

Creating GTire

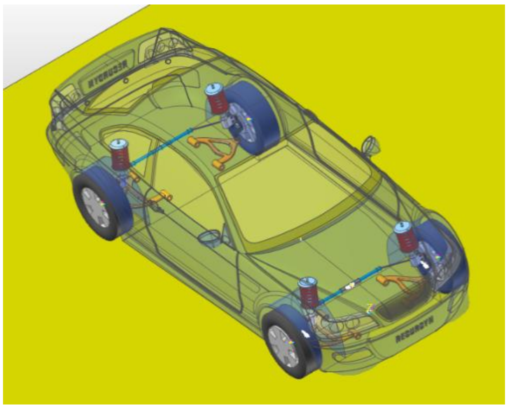
1. Click **GTire** in the **Toolkit** group on the **Toolkit** tab.
2. Set **Creation Method** to **Point, WithDialog**, and enter **0, -800, 150** for **Point**.
3. When the **GTireGroup** dialog box appears, click '**...**'.
4. When the Open dialog box appears, link the **UATire_MMKS.tir** file copied above.
5. When connected, geometry information such as Tire Radius and Tire Width is automatically updated.
6. Click **R** to select Road.
7. Select **Ground.GRoad1** created above.
8. Change the **Force display** to **Action**.
9. Click **OK**.
10. **GTireGroup1** is created in the database.
11. Open the Property dialog box for **GTireGroup1** and change the **Name** to **GTire_FR**.



12. Repeat steps **2-10** by using the information below.
    - **Point**: 0, 800, 150
    - **Name**: GTire_FL

    - **Point**: -2746.8298265481, -800, 150
    - **Name**: GTire_RR

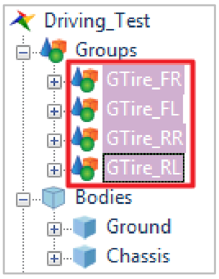
    - **Point**: -2746.8298265481, 800, 150
    - **Name**: GTire_RL



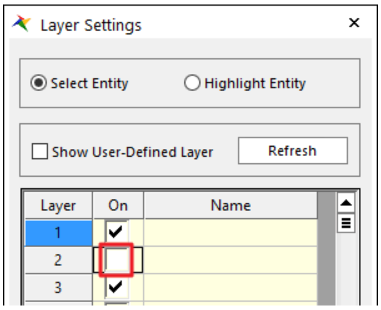
You must constrain the four GTires created above to each WheelHub with a Fixed Joint.

Creating a Fixed Joint between GTire and WheelHub

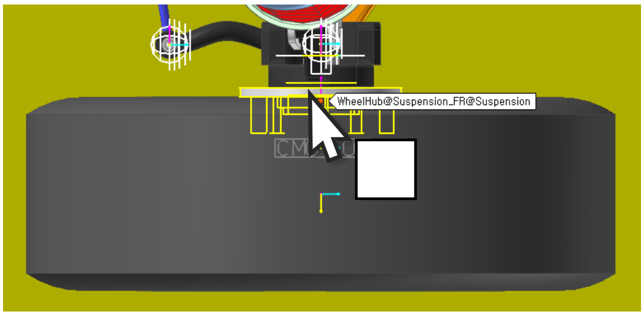
1. In the **Working Plane Toolbar**, select **Change to XY**.
2. In order to zoom in only vehicle, multi-select the every **GTire** entities in Database and click **Fit** in the View Control Toolbar.



3. In order to click **WheelHub** hidden in the **Chassis**, uncheck **On** of **Layer 2** in the **Layer Settings** dialog box.
4. Then it looks like the picture below.



5. On the **Professional** tab, click **Fixed** in the **Joint** group.
6. Set the Creation Method to **Body, Body, Point**.
7. To click the **WheelHub** in the Subsystem as the first Body, hold down the **Shift** key and click **WheelHub@Suspension_FR@Suspension** as shown below.



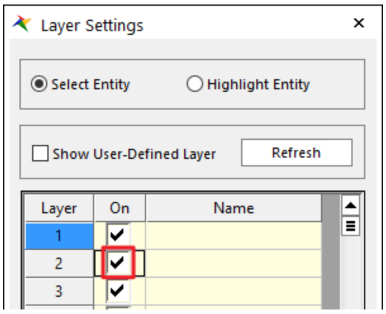
8. Select **GTire_FRBody** as the second Body.
9. Enter **0, -800, 150** as the Point.

10. Open the Property dialog box for the created **Fixed1** and change the Name to **Fix_GTire_FR**.
11. Repeat steps **3-8** by using the information below.
    - **Body**: WheelHub@Suspension_FL@Suspension
    - **Body**: GTire_FLBody
    - **Point**: 0, 800, 150
    - **Name**: Fix_GTire_FL

    - **Body**: WheelHub@Suspension_RR@Suspension
    - **Body**: GTire_RRBody
    - **Point**: -2746.8298265481, -800, 150
    - **Name**: Fix_GTire_RR

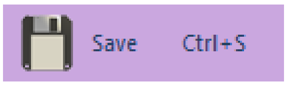
    - **Body**: WheelHub@Suspension_RL@Suspension
    - **Body**: GTire_RLBody
    - **Point**: -2746.8298265481, 800, 150
    - **Name**: Fix_GTire_RL

- Once four Fixed Joints are created, place a check mark in **On** of **Layer2** again in the Layer Settings dialog box and click **Close** to close the dialog box.



To save the model

- From Road to Tire, all the modeling required for the analysis is complete. On the **File** menu, click **Save** to save the model.



Chapter
4

Analysis of driving

Task Objectives

Use the completed vehicle model to perform the following analysis of driving.

- Implement the analysis of vehicle landing
- Perform analysis on straight driving
- Analyze J-Turn driving

Estimated Time to complete this task

20 minutes

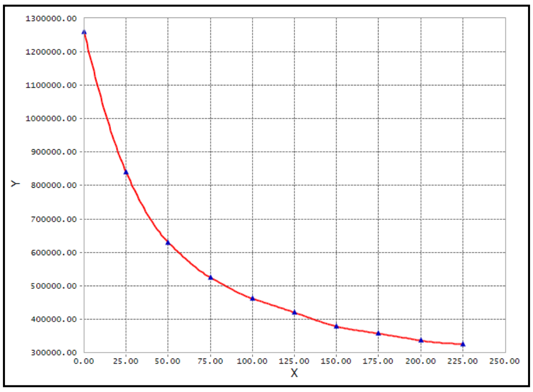
Preparation for the analysis of driving

Define the Torque for generating a driving speed of the vehicle as a spline curve and implement it as Expression. Then use the Expression created to apply it to the two Rotational Axial Forces created in Chapter 3.

Define the Torque of running speed as Spline

1. On the **SubEntity** tab, click **Spline** in the **Expression** group.
2. When the **Spline List** dialog box appears, click **Create**.
3. When the **Spline** dialog box appears, click **Add Row** and enter the values below.
   - **Name**: Sp_Torque_kmph_Nmm
   - **Data**:

No	X	Y1
1	0	1260000
2	25	840000
3	50	630000
4	75	525000
5	100	462000
6	125	420000
7	150	378000
8	175	357000
9	200	336000
10	225	325500

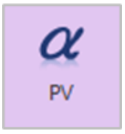


4. The above information is the torque (N*mm) that each wheel receives relative to the vehicle speed (km/h).

5. Click **OK**.
6. Click **OK** in the Spline List dialog box.

The target speed (km/h) is defined as PV for easy speed control.

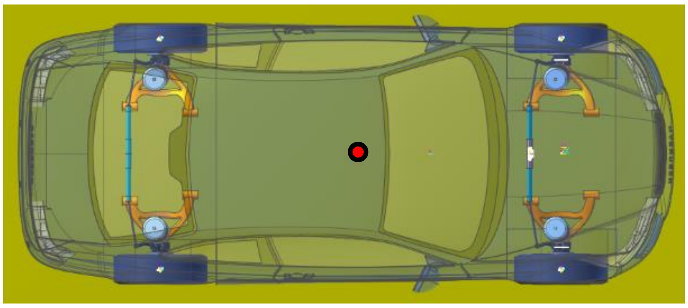
Defining PV



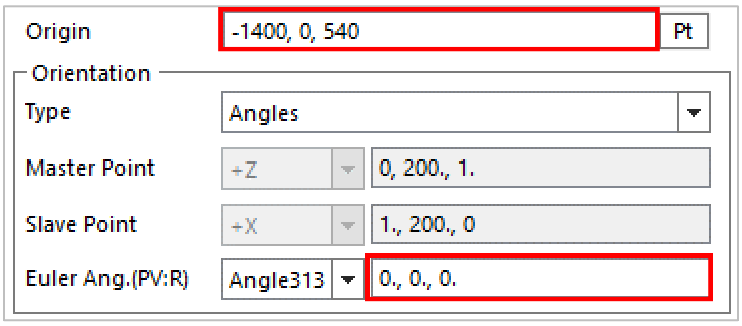
1. On the **SubEntity** tab, click **PV** in the **Parameter** group.
2. When the PV List dialog box appears, click **Add** to define the values below.
   - **InputV_kmph: 0**
3. Click **OK**.

Creating Reference Marker to measure vehicle speed

1. In the Working Plane Toolbar, select Change to XY.
2. In order to zoom in only vehicle, multi-select the every **GTire** entities in Database and click **Fit** in the View Control Toolbar.
3. On the **Professional** tab, click **Marker** in the **Marker and Body** group.
4. Set the Creation Method to **Body, Point**, and enter as follows:
   - **Body**: Ground
   - **Point**: -1400, 0, 540
5. Repeat steps **3** by using the information below.
   - **Body**: Chassis
   - **Point**: -1400, 0, 540
6. Two markers are created in the center of the chassis as shown below.

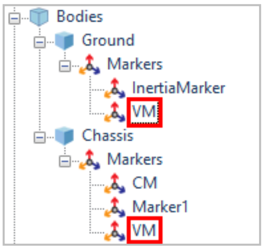


7. In Database, select the Marker last created from the **Markers** under **Ground** and open the Property dialog box.
8. Go to the Origin & Orientation tab and make sure the same information as below is entered.
   - **Origin**: -1400, 0, 540
   - **Orientation**: 0, 0 ,0

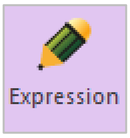


9. Since it is a marker which will be used as a reference marker, please note that the analysis result may vary greatly depending on the origin and orientation.
10. After confirming that the Property values are OK, change the **Name** to **VM** in the General tab.
11. Click **OK**.

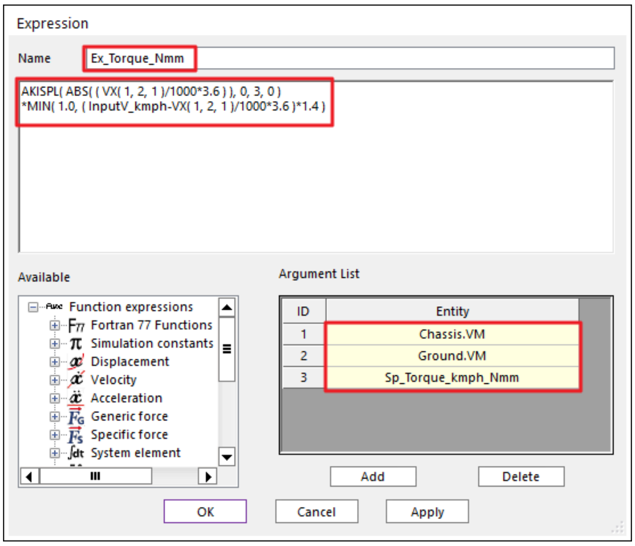
12. Repeat steps **6-8** for the markers of **Chassis**.



Defining Torque Expression



1. On the **SubEntity** tab, click **Expression** in the **Expression** group.
2. When the Expression List dialog box appears, click **Create** to open the dialog box.
3. When the Expression dialog box appears, enter as follows:
   - **Name**: Ex_Torque_Nmm
   - **Content**:
     - **AKISPL(ABS((VX(1, 2, 1)/1000*3.6)), 0, 3, 0)*MIN(1.0, (InputV_kmph-VX(1, 2, 1)/1000*3.6)*1.4)**
   - **Argument** :
     - **ID1: Chassis.VM**
     - **ID2: Ground.VM**
     - **ID3: Sp_Torque_kmph_Nmm**

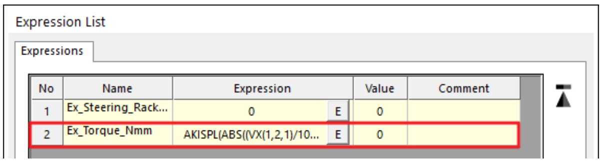


4. The detailed look at the configuration of **Expression** defined above is as follows.
   - The **AKISPL** function returns the torque value using the spline information defined above.
   - Use spline's X value for the **VX** function to input the vehicle's travelling direction speed.
   - The vehicle speed is converted to km/h and an absolute value is taken by the **ABS** function.
   - Multiply the gap between the input velocity and the current speed by the Gain value (1.4) to implement a simple P control. (The gain value can be changed depending on situations.)
   - Use the **MIN** function so that the P control syntax does not exceed 1.
5. Click **OK** in the Expression dialog box.
6. Click **OK** in the Expression List dialog box.

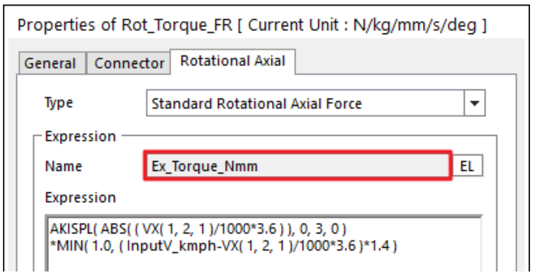
Enter the Expression you created above in the two **Rotational Axial Forces**.

Entering Torque Expression

1. Open the Property dialog box of **Rot_Torque_FR** in the database.
2. Click **EL**.
3. When the Expression List dialog box appears, activate **Ex_Torque_Nmm** in the list and click **OK**.



4. Make sure that Expression is entered on the Property dialog box of **Rot_Torque_FR** and click **OK**.



5. Open the Property dialog box of **Rot_Torque_FL** in the database, and repeat steps **1-4**.

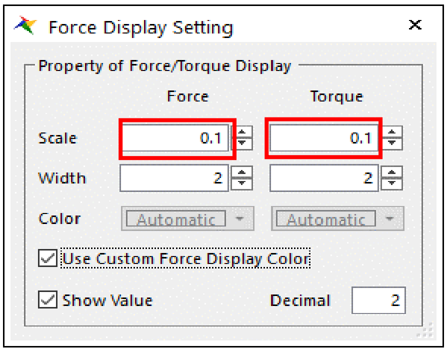
Implement the analysis of vehicle's settling on the road

Run Dynamic/Kinematic to see if the created vehicle is duly settled the road.

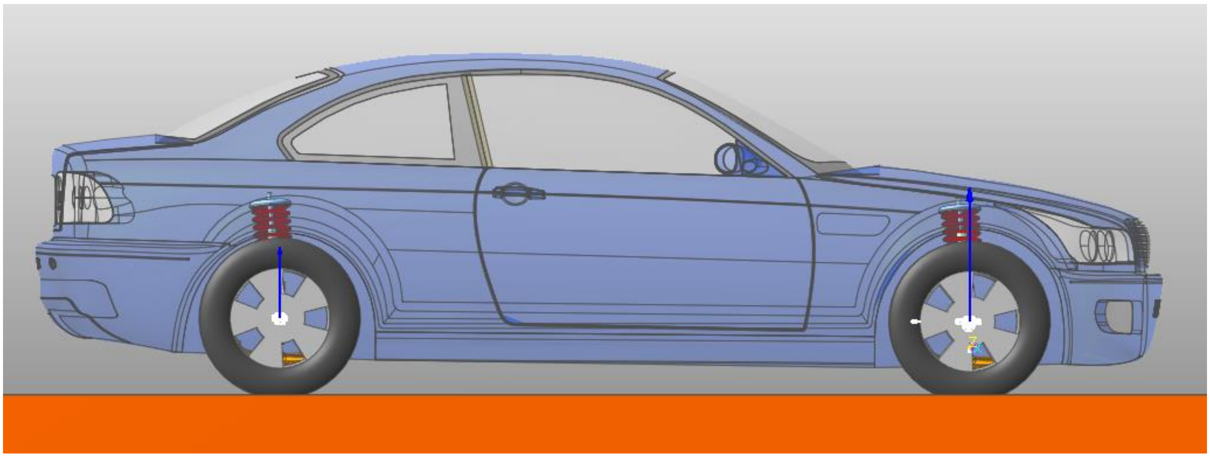
Implement the analysis of vehicle landing

1. In the **Simulation Type** group on the **Analysis** tab, click **Dyn/Kin** to open the Dynamic/Kinematic dialog box.
2. On the **General** tab, values are defined as follows.
   - **End Time**: 5
   - **Step**: 100
3. On the Parameter tab, change the **Maximum Time Step** to **1.e-3**.
4. Click **Simulate** to proceed with the analysis.

5. In the Animation Control group on the Anaylsis tab, click Force Display Setting.
6. Change the **Scale** of **Force** and **Torque** to **0.1** and close the dialog box.



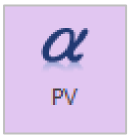
7. Play back the animation to see if the vehicle duly reaches the road as shown below.



Perform analysis on straight driving

To create torque for the vehicle, change the target velocity defined by PV.

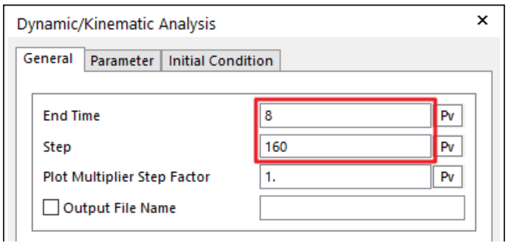
Changing velocity PV



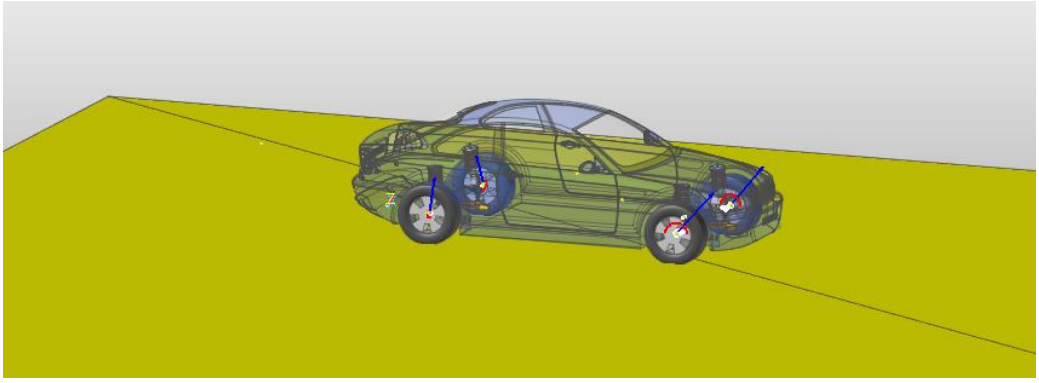
1. On the **SubEntity** tab, click **PV** in the **Parameter** group.
2. In the PV List dialog box, change **InputV_kmph** to **40**.
3. Click **OK**.

Perform analysis on straight driving

1. In the **Simulation Type** group on the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic** dialog box.
2. On the General tab, values are defined as follows.
   - **End Time**: 8
   - **Step**: 160



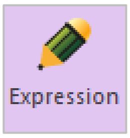
3. Click **Simulate** to proceed with the analysis.
4. Play back the animation to see if the vehicle drives straight as shown below.
5. Since the steering motion is fixed, the vehicle will travel straight at 40km/h.



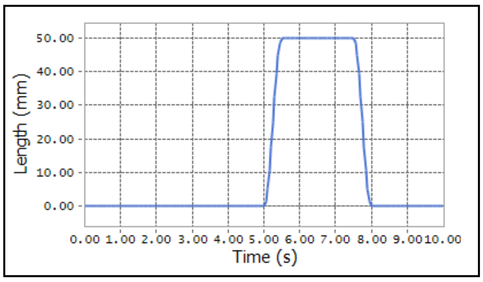
Analyze J-Turn driving

During drving, control the steering to analyze the J-Turn driving.

Modifying the Expression of Steering



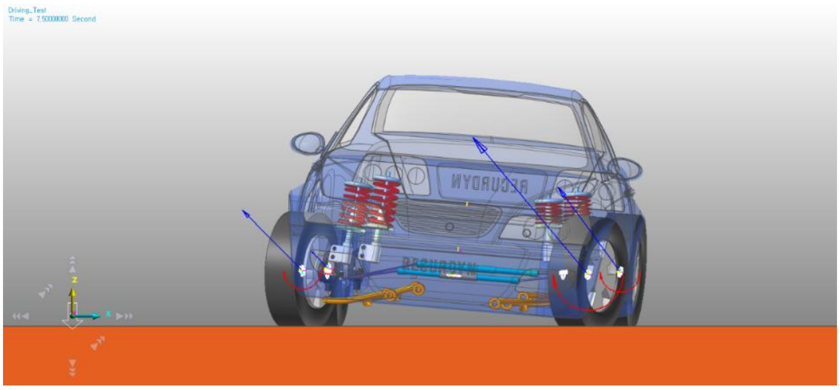
1. On the **SubEntity** tab, click **Expression** in the **Expression** group.
2. When the Expression List dialog box appears, click **E** in the list to modify **Ex_Steering_RackBar_mm**.
3. When the **Expression** dialog box appears, modify the **Content** as shown below.
   - **Content**:
     - **Step(Time-5,0,0,0.5,50) - Step(Time-7.5,0,0,0.5,50)**



4. The Expression defined above is a formula to move the **RackBar** by 50mm between 5 and 8 seconds and undo the moving. When the **RackBar** moves 50mm, the Tire will turn about 12 degrees.
5. Click **OK** in the **Expression** dialog box.
6. Click **OK** in the **Expression List** dialog box.

Analyze J-Turn driving

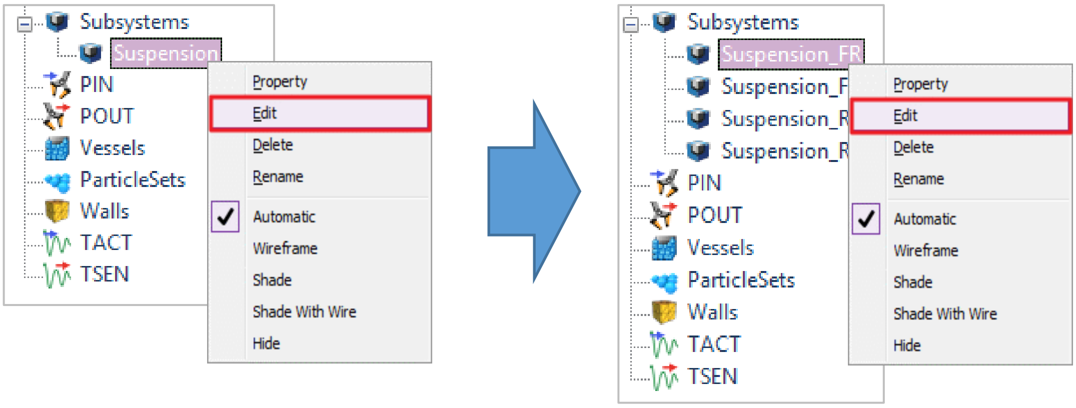
1. In the **Simulation Type** group on the **Analysis** tab, click **Dyn/Kin** to open the Dynamic/Kinematic dialog box.
2. On the **General** tab, values are defined as follows.
   - **End Time**: 10
   - **Step**: 200
3. Click **Simulate** to proceed with the analysis.
4. Play back the animation to see if the vehicle performs J-Turn driving as shown below.



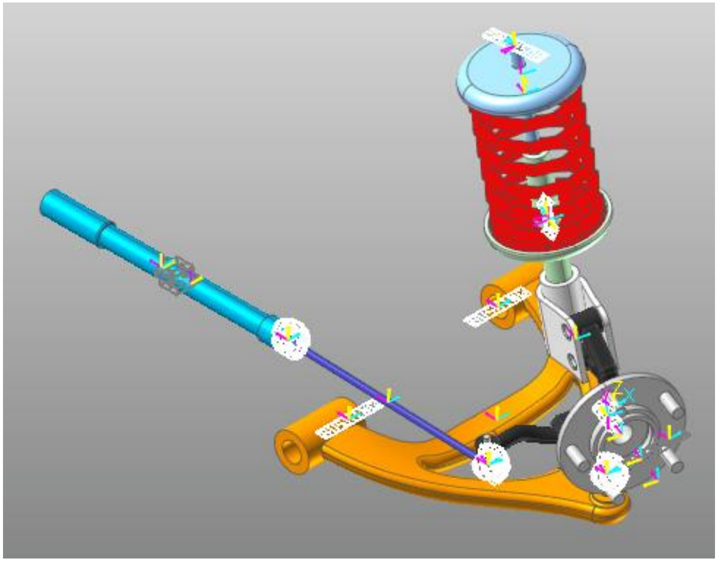
The result of animation analysis can be played back inside the subsystem as well. When played back inside the subsystem, it is played based on the Mother Body.

Watching an animation by using the subsystem

1. In the database, right-click **Suspension** Subsystem and click **Edit** in the pop-up menu.
2. In the database, again, right-click **Suspension_FR** Subsystem and click **Edit** in the pop-up menu.



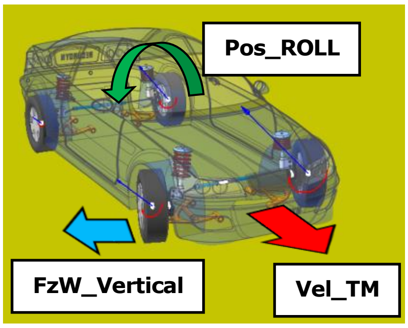
3. Play back the animation to observe the behavior of the suspension.
4. The animation is played back based on the movement of Chassis, which is the mother body of the subsystem. Therefore, the Suspension connected to the Chassis is seen as a fixed state.

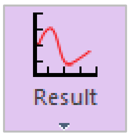


Drawing a plot

Draw a plot to see the dynamic behavior of a vehicle driving J-Turn.

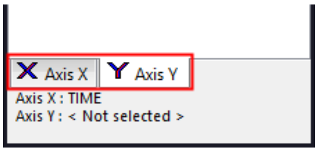
Drawing a plot for J-Turn analysis

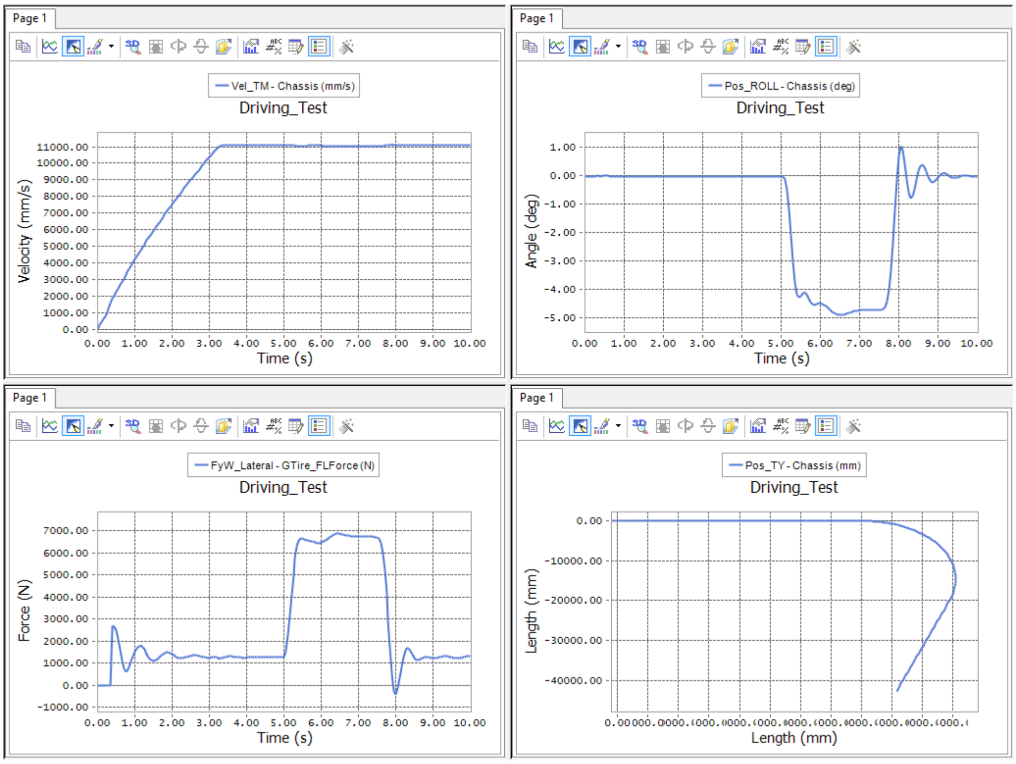




1. In the **Plot** group on the **Analysis** tab, click **Result**.
2. Enter the Plot Window.

3. In the **Windows** group on the **Home** tab, click **All**.
4. Use the information below to draw a plot.
   - **Top left Window**: Bodies/Chassis/Vel_TM
   - **Top right Window**: Bodies/Chassis/Pos_ROLL
   - **Bottom left Window**: Tire/GTire_FLForce/FyW_Lateral
   - Bottom right Window:
     - **AxisX: Bodies/Chassis/Pos_TX**
     - **AxisY: Bodies/Chassis/Pos_TY**

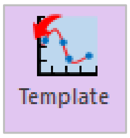




The result of plot analysis shows that the vehicle travels at a constant speed after 3.4 seconds. You can also see that when you start J-Turn after 5 seconds, the roll angle of the vehicle and the lateral force of tires occur. Finally, you can see the turning behavior of the vehicle numerically by X-Y position information.

Create and apply a plot template to repeat the same plot easily.

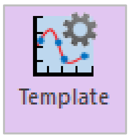
Exporting plot template

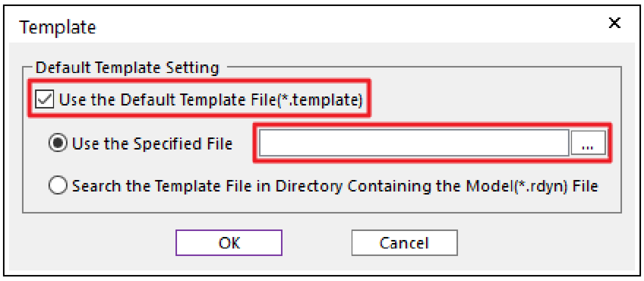


1. With four curves drawn, click **Template** in the **Export** group on the **Home** tab.
2. When the Export Plot Template Setting dialog box appears, click Export to save the template file.

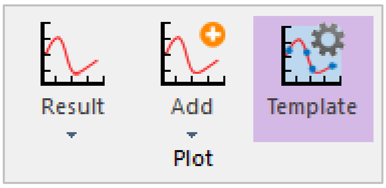
Connecting the plot template file

1. Return to the modeling Window.
2. In the **Plot** group on the **Analysis** tab, click **Template**.
3. When the **Template** dialog box appears, turn on the **Use the Default Template File(*.template)** option.
4. Select **Use the Specified File** and connect the Plot Template file created above.





5. Click **OK** to close the dialog box.
6. When the **Plot Template** is connected, the **Template** icon becomes active as shown on the right.



Chapter

5

Modification and analysis of Tire Property

Task Objectives

After modifying the contact property of UA-Tire, perform the analysis of driving. Try to understand the impact of parameter change through the plot analysis of analysis results.

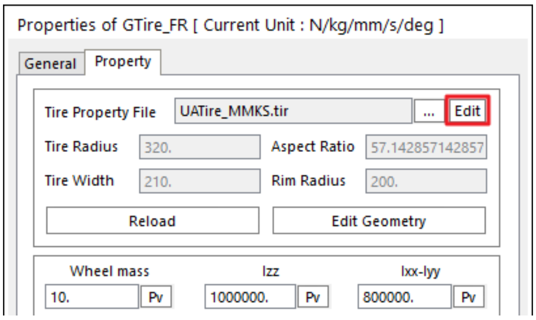
Estimated Time to complete this task

10 minutes

Modification and analysis of UA-Tire Property

There are several parameters in the Tire file that affect behaviors. Especially, during the J-Turn driving, the behaviors are greatly affected by the lateral force applied to the side of the Tire. Modify the parameters that affect the lateral force and try the same analysis.

Implement the analysis after modifying the UA-Tire Property

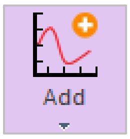


1. Open the Property dialog box of **GTire_FR** in the database.
2. Click **Edit** to open the **UATire_MMKS.tir** file.
3. The file of **UA Tire** opens as a text format in Notepad as shown below.
4. Edit **LATERAL_STIFFNESS** as **30000**.

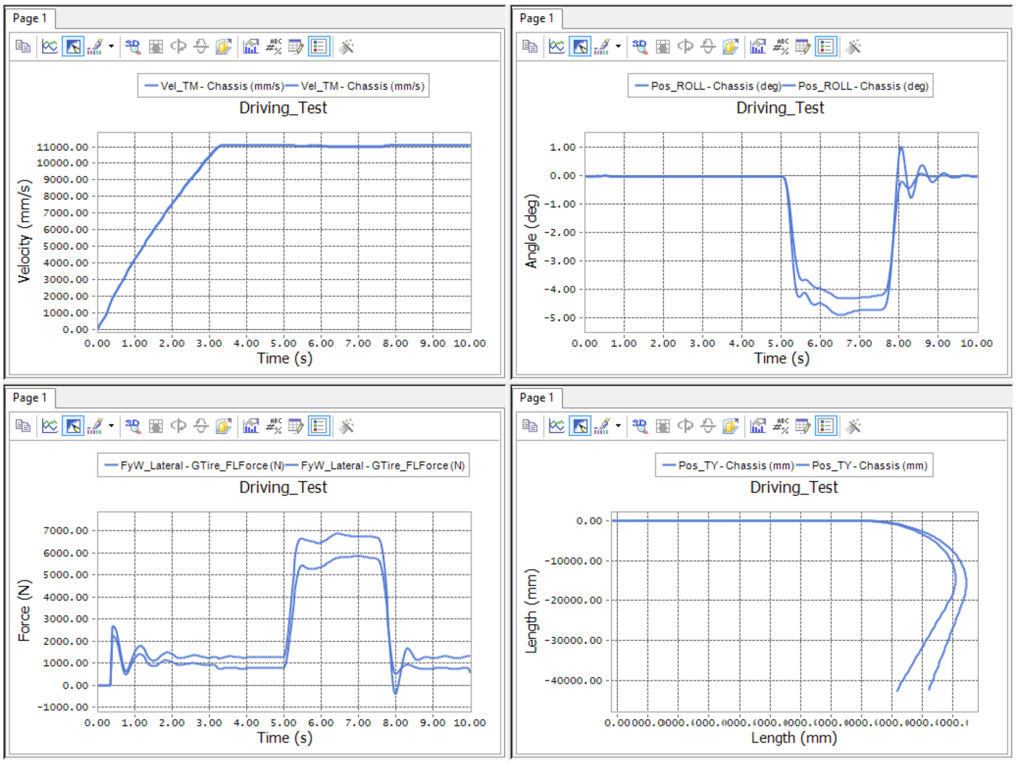
```
$---------------------------------------------------------------parameter
[PARAMETER]
$ vertical stiffness (Force/Length)
RADIAL_STIFFNESS                    = 190.0
$ vertical damping ratio
RADIAL_DAMPING_RATIO                = 0.01
% longitudinal slip stiffness (Force/ratio)
LONGITUDINAL_STIFFNESS              = 200000
% lateral slip stiffness (Force/angle)
LATERAL_STIFFNESS                   = 30000
% camber stiffness (Force/angle)
CAMBER_STIFFNESS                    = 3000
% rolling registance length (Length)
ROLLING_RESISTANCE                  = 3
% maximum friction coefficient
FRICTION_MAX                        = 1.1
% minimum friction coefficient
FRICTION_MIN                        = 0.8
```

5. In the **File** menu of Notepad, click **Save** to close Notepad.
6. Again, click **OK** in the Property dialog box of **GTire_FR**.
7. Since the four **GTire** properties share the same Tire file, your action affects all the others.

Tip: Tip: Editing Tire Property directly with a text editor
You can also edit the Tire file directly by opening it in a text editor on the Windows Explorer without having to open it in the GTire Property dialog box. The Property value thus edited is applied and then analysis is performed.

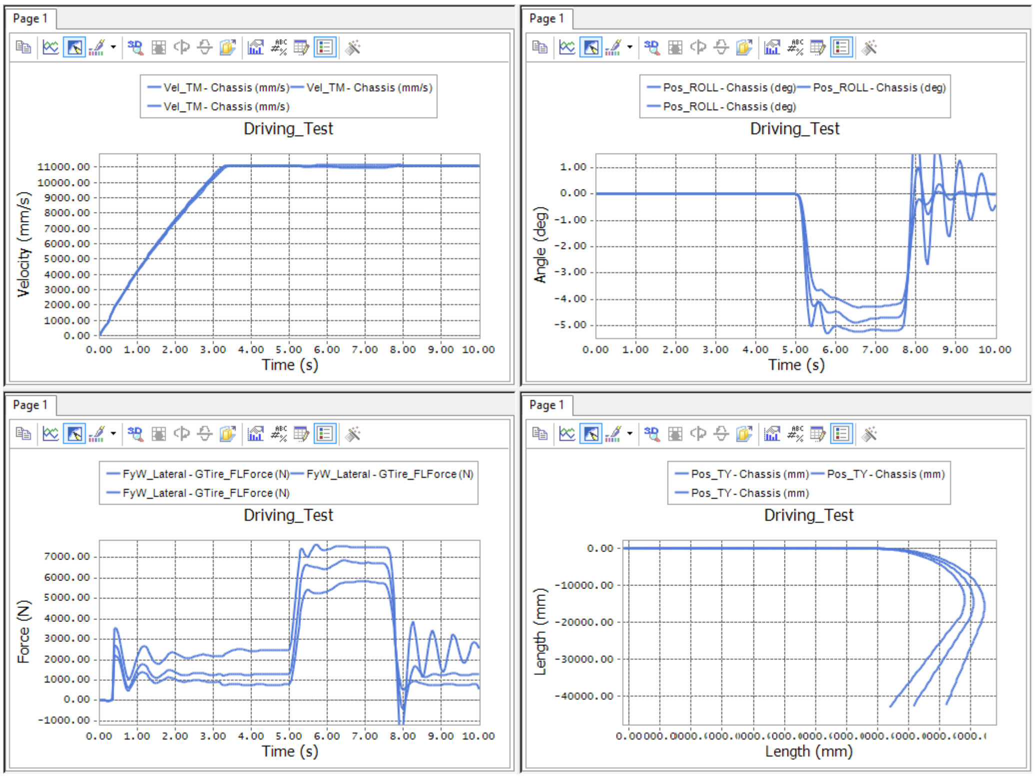


8. In the **Simulation Type** group on the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic** dialog box.
9. Click **Simulate** to proceed with the analysis.
10. Once the analysis is completed, click **Add** in the **Plot** group on the **Analysis** tab.
11. Since the Plot Template has been defined in Chapter 4, curves of newly analyzed results are drawn in the same form over the previously drawn Plot.



12. If you analyze the plot results, the Lateral Force becomes smaller after lowering LATERAL_STIFFNESS to 30000. So, the vehicle is pushed outward in the direction of turn. You can also see that the roll angle of the chassis becomes smaller during the turn between 5 seconds and 8 seconds.

13. Repeat steps **1-8** by using the information below.
    - **LATERAL_STIFFNESS: 100000**
14. Curves of newly analyzed results are drawn additionally over the previous plot.



15. If you analyze the plot result again, raising **LATERAL_STIFFNESS** to **100000** increases the lateral force as opposed to the previous result. So, the vehicle is pulled inward in the direction of turn. You can also see that the roll angle of the chassis becomes larger during the turn between 5 seconds and 8 seconds. Because of this change, the amplitude of the roll becomes very large immediately after the turn, that is, after 8 seconds.

Chapter

6

Change and analysis of GRoad

Task Objectives

After changing GRoad, perform the analysis of driving.

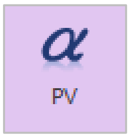
Estimated Time to complete this task

10 minutes

Change and analysis of GRoad

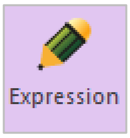
Analyze the behavior of vehicle during a straight driving while changing the ground.

Changing velocity PV



1. On the **SubEntity** tab, click **PV** in the **Parameter** group.
2. In the PV List dialog box, change **InputV_kmph** to **30**.
3. Click **OK**.

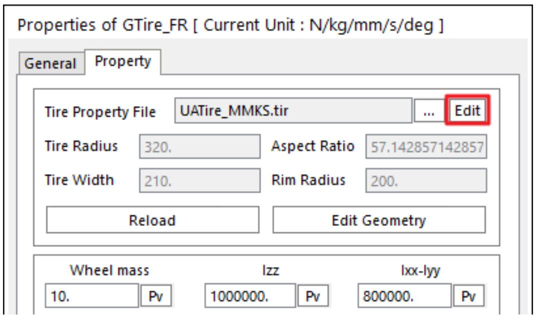
Modifying the Steering Expression for straight driving



1. On the **SubEntity** tab, click **Expression** in the **Expression** group.
2. When the Expression List dialog box appears, click **E** in the list to modify **Ex_Steering_RackBar_mm**.
3. When the **Expression** dialog box appears, modify the **Content** to **0**.
4. Click **OK** in the **Expression** dialog box.
5. Click **OK** in the **Expression List** dialog box.

Modifying the UA-Tire Property

1. Open the Property dialog box of **GTire_FR** in the database.
2. Click **Edit** to open the **UATire_MMKS.tir** file.
3. Edit **LATERAL_STIFFNESS** as **50000**.



Create GRoad by loading the ground information file provided in this tutorial.

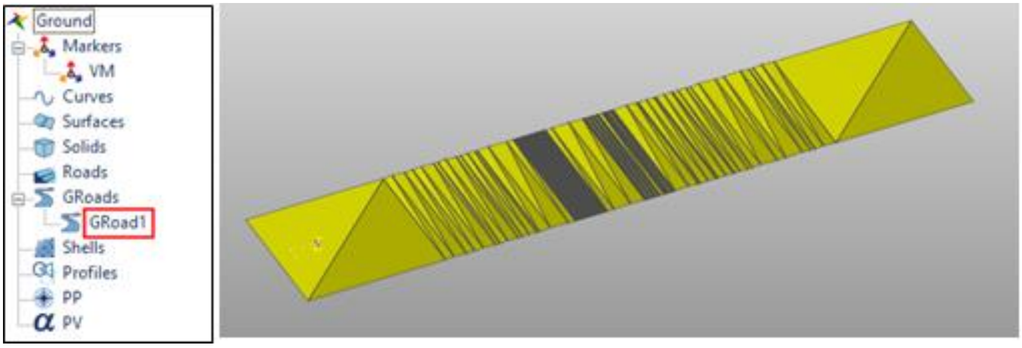
Copying GRoad files

- Copy the **GRoad_Rough.rdf** file and the **GRoad_Hill.rdf** file from the GTire tutorial folder and paste it into where the model is saved.
- (File path: <InstallDir> /Help/Tutorial/Toolkit/Tire/DrivingJTurn/GRoad_Hill.rdf)

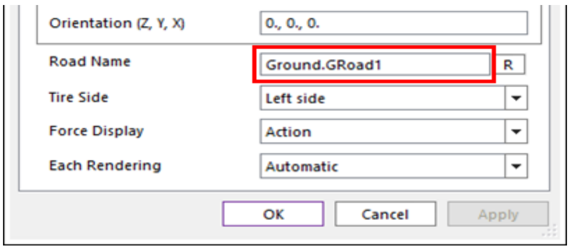
Analyzing after changing to rough GRoad

1. To enter the Edit mode of the Ground, click **Ground** in the **Marker and Body** group on the **Professional** tab.
2. When you enter Ground Edit mode, it will appear as **Ground@Driving_Test** in the upper left corner of the Working Window.
3. Delete **Box1** and **GRoad1** from the database.

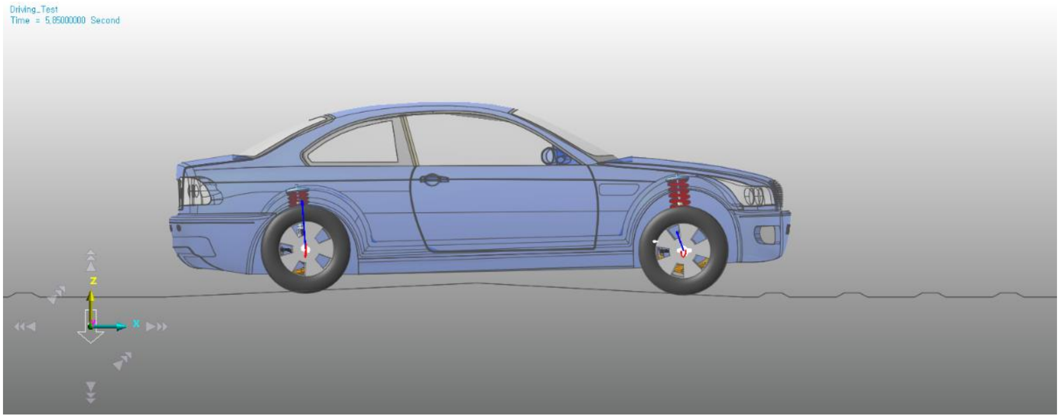
4. In the **Road Data** group on the **Ground** tab, click **GRImport** to import **GRoad_Rough.rdf** copied above.
5. **GRoad1** is recreated in the database, and the rough surface is shown as below.



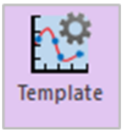
6. Click **Exit** on the **Ground** tab to exit the Ground Edit mode.
7. In the Property dialog box for four GTires, enter **Road name** as **Ground.GRoad1**.



8. In the **Simulation Type** group on the **Analysis** tab, click **Dyn/Kin** to open the **Dynamic/Kinematic** dialog box.
9. On the **General** tab, values are defined as follows.
   - **End Time**: 10
   - **Step**: 1000
10. Click **Simulate** to proceed with the analysis.
11. Play back the animation to check the behavior of the vehicle. You can see the vehicle sway back and forth due to the rough ground.



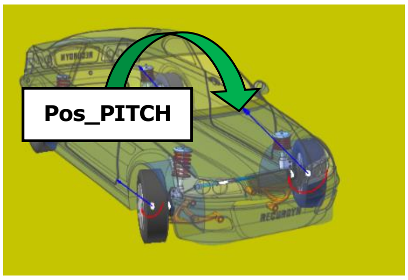
Unlink the Plot Template File

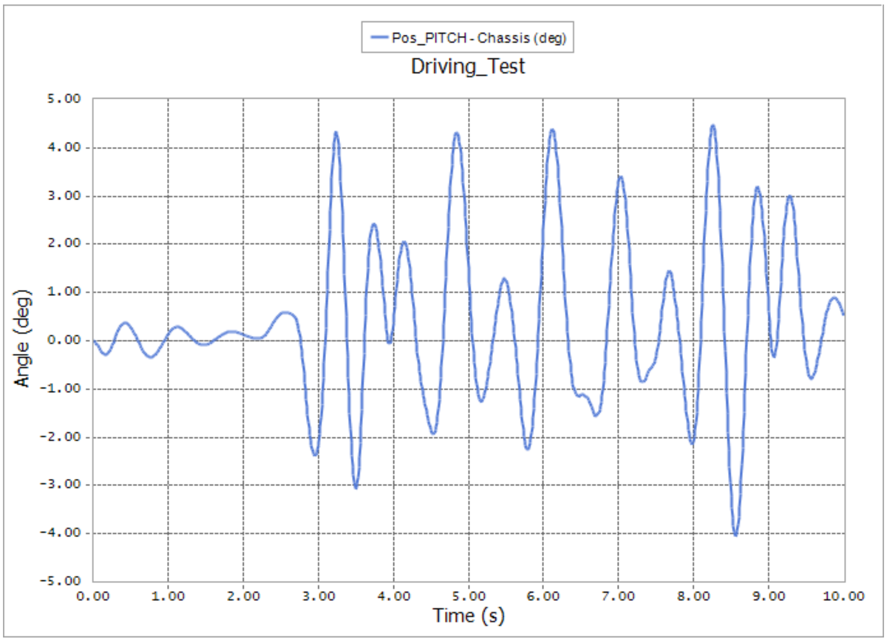


1. In the **Plot** group on the **Analysis** tab, click **Template**.
2. When the Template dialog box appears, turn off the **Use the Default Template File(*.template)** option.
3. Click **OK** to close the dialog box.

Drawing a Plot for rough surface analysis

1. In the **Plot** group on the **Analysis** tab, click **Result**.
2. Enter the **Plot Window**.
3. Use the information below to draw the vehicle's Pitch information.
   - **Bodies/Chassis/Pos_PITCH**

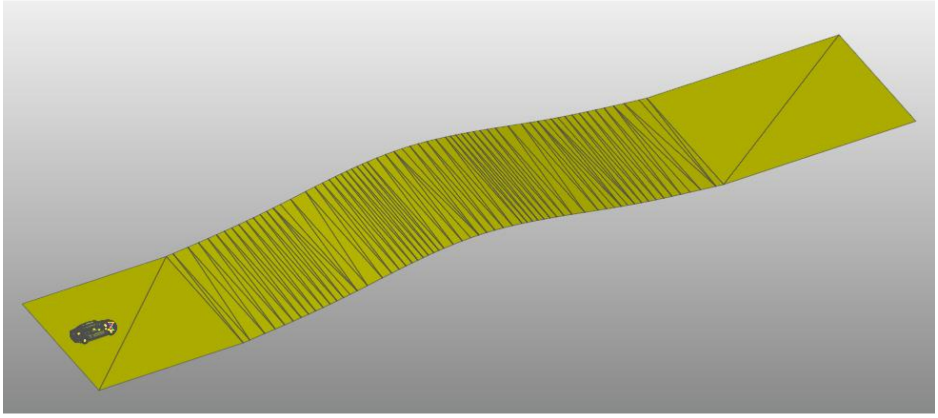


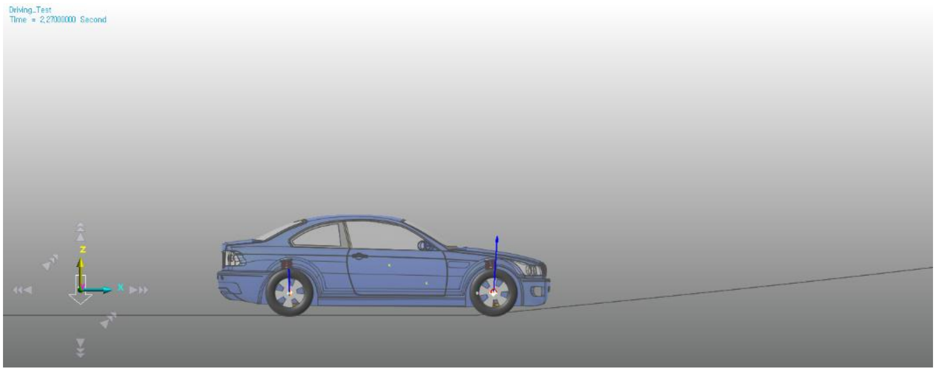


As you saw in the animation, you can also see on the plot numerically that the pitch angle increases due to the irregular ground surface.

Analyzing after changing to GRoad with hills

- In the same way as before, delete the existing **GRoad** in the Edit mode of Ground and then open **GRoad_Hill.rdf** and try the same analysis.





Various analyses of driving (reference)

You can think about additional driving analyses by considering several factors as follows:

- Modify other properties than **LATERAL_STIFFNESS** and then drive
- Using various geometries to change the **GRoad** and then drive
- Change Input Velocity and Steering and then drive
- Change spring property of Suspension and then drive

Thanks for participating in this tutorial