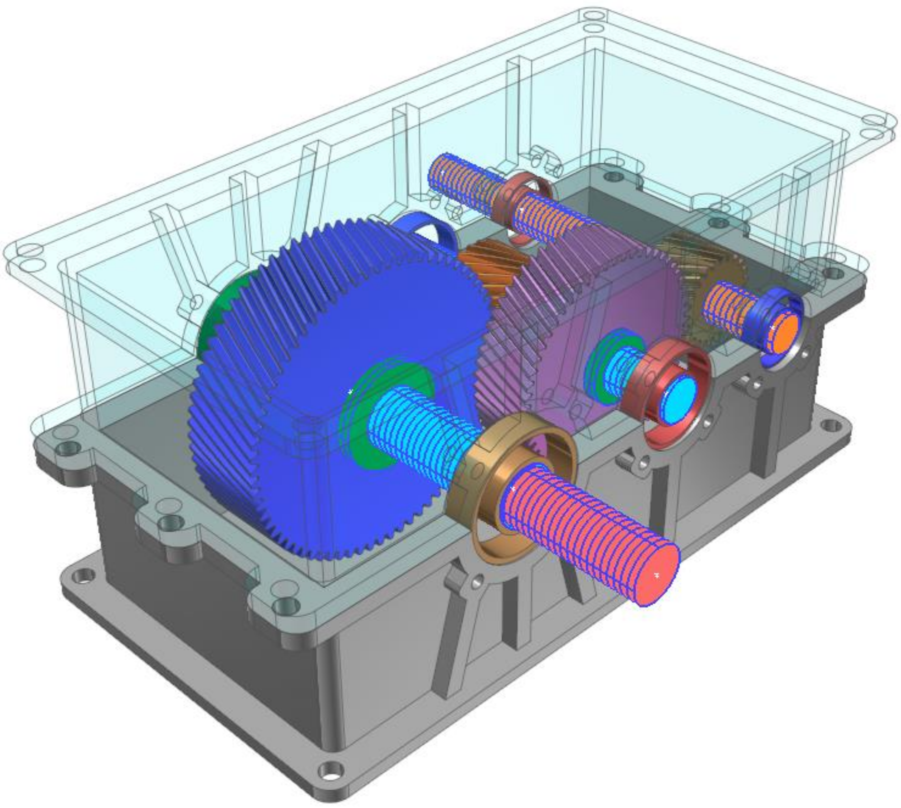
Gearbox Tutorial (DriveTrain)



FunctionBay

Edition Note

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

Table of Contents

Chapter

1

Getting Started

DriveTrain Toolkit can design the mechanical system composed of shaft, gear, bearing, etc. It can consider the material and dynamic properties of its parts. For shaft, it can design and analyze the shaft with the various radii using the FE Beam Element. For gear, it co-simulates with KISSsoft for more accurate and detailed result and it can be used with Involute Analytic Gear Contact for reduced simulation time with accurate result. For bearing, it also co-simulates with KISSsoft for more accurate and detailed result.

In this tutorial, you will learn about the process of simulation of gearbox system using DriveTrain Toolkit. You can learn how to use the new functions and analyze the simulation results.

Objective

You will also learn below.

- Create a shaft, bearing, gear
- Create an analytic gear contact
- Analyze the simulation result of shaft, bearing, gear

Prerequisites

This tutorial is for the users who already learned the basic tutorial and the FFlex tutorial. Therefore, you should first work through the tutorials that mentioned earlier to enhance the understanding of this tutorial. Also, we assume that you have a basic knowledge of dynamics and the Finite Element Method.

Procedures

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

Procedures	Time (minutes)
Simulation environment setup	10
Create shaft	10
Create bearing	10
Create gear	15
Create joint, force	20
Analyze the simulation result	20
Create involute analytic contact	20
Total	105

Estimated Time to Complete

1 hours and 45 minutes

Chapter

2

Setting Up the Simulation Environment

Task Objective

In this chapter, you will start the RecurDyn and setup its environment, including importing the completed gearbox CAD and changing its names and layer.

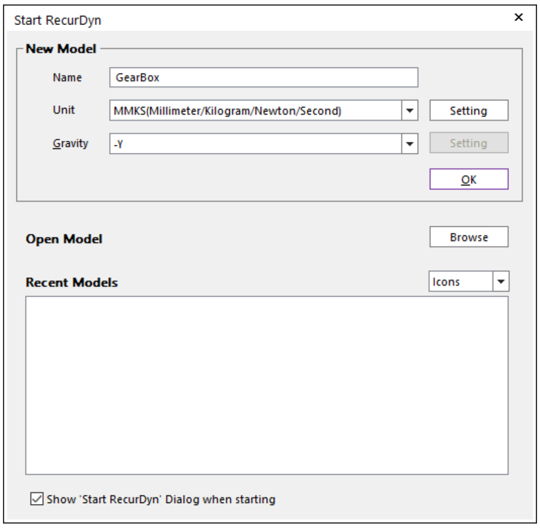
Estimated Time to Complete

15 minutes

Starting the RecurDyn

To start the RecurDyn and create a new model:

1. On your Desktop, double-click the **RecurDyn** icon and the **New Model** dialog box will appear.
2. Change the **Model Name** to **GearBox** as shown at right.
3. Ensure the units are the same as those in the **Start RecurDyn** dialog box shown at right. If not, click **MMKS**. (Millimeter/Kilogram/Newton/Second)
4. Click **OK**.

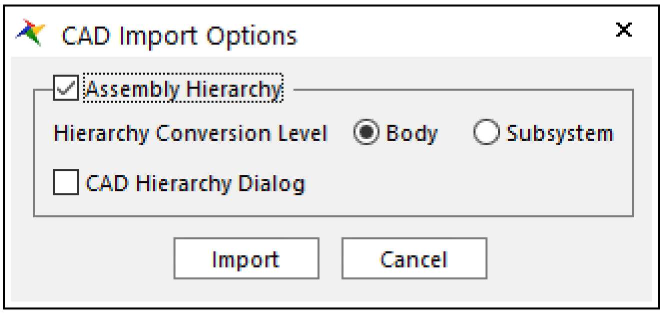


Importing the Gearbox Geometry

You will begin to model the gearbox by importing an already completed gearbox CAD.

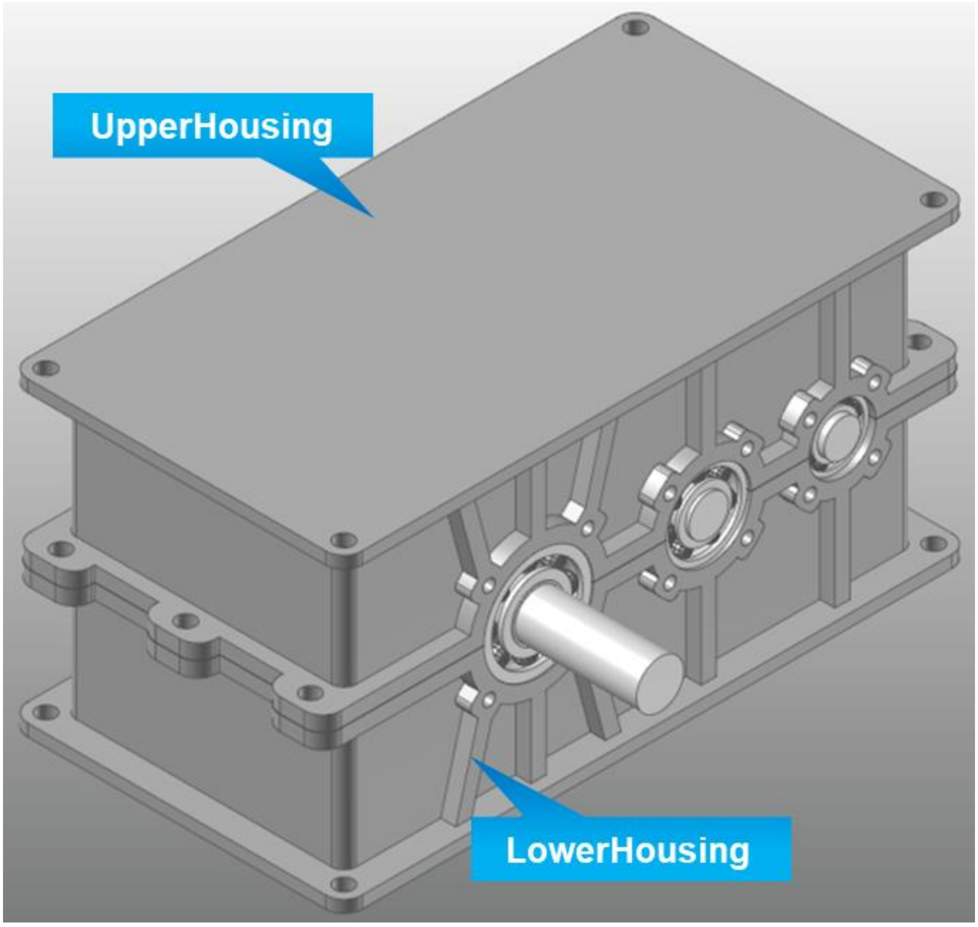
To import the gearbox CAD

1. From the **File** menu, Choose **Import**.
2. In the Open dialog box, select the file **GearBoxCAD.x_t**. (**The file location:** <InstallDir>/Help/Tutorial/Toolkit/DriveTrain/GearBox).
3. Click **Open**. The **CAD Import Options** dialog appears. Make sure the option **Assembly Hierarchy** is checked and the option **Body** is checked in the **Hierarchy Conversion Level** and click **Import**.



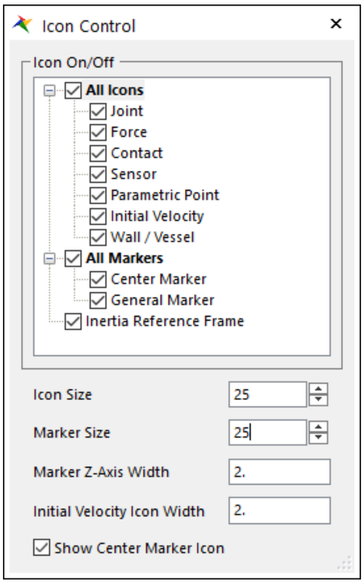
To change the name of CAD and set layer

1. Right-click the **Upper Housing** body as shown in the figure below and click **Properties** in the Pop-up menu.
2. In the **General** tab, you will adjust the value using the information below and click **OK**.
   - **Name**: UpperHousing
   - **Layer**: 2
3. For the **Lower Housing** body, change the **Name** to **LowerHousing** and **Layer** to **2**.

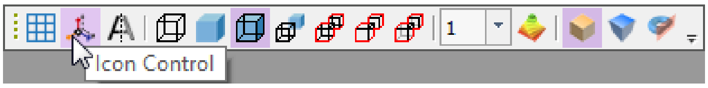


Adjusting the Icon, Marker Size and Layer

To adjust the icon and marker size

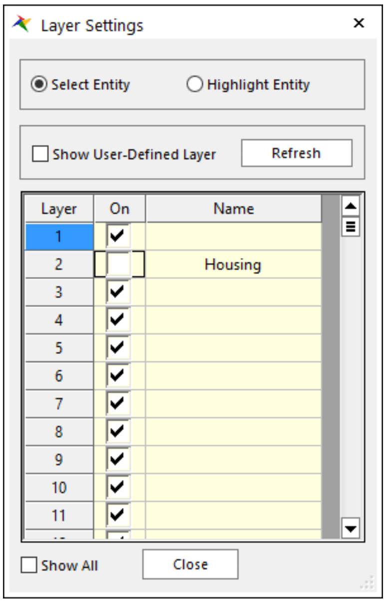


1. In the **Render Toolbar**, click the **Icon Control** as shown in the figure below.

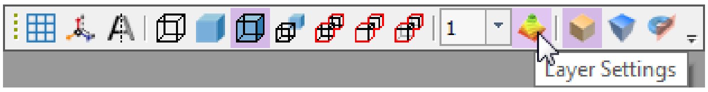


2. Set the **Icon Size** and **Marker Size** to **25** as shown at right.

To adjust the layer setting

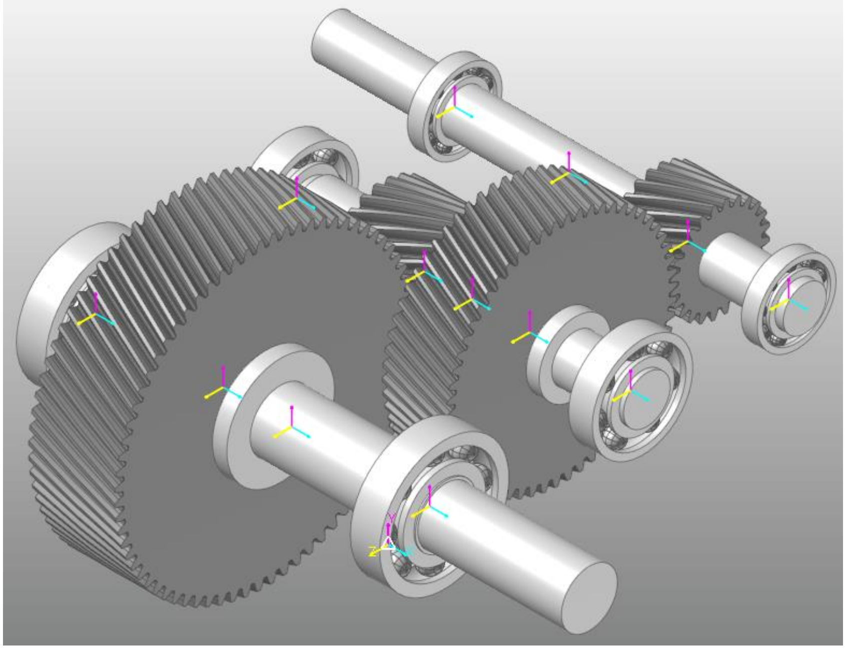


1. In the **Render Toolbar**, click **Layer Settings** as shown in the figure below.



2. Change the **Name** of Layer 2 to **Housing** and **Check Off** the **Layer On** option as shown at right.

The model appears as shown in the figure below.



Saving the Model

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

Chapter 3

Creating the Shaft

Task Objective

In this chapter, you will learn how to use the shaft modeler that can create various types of shaft sections which are composed of finite beam element with different circular cross sections.

Estimated Time to Complete

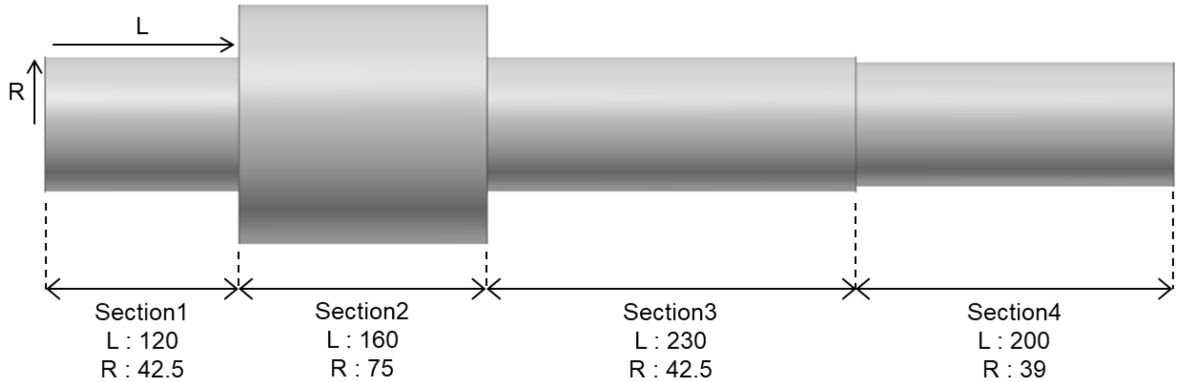
10 minutes

Creating the Shaft

You will create the shaft with beam element to analyze the stress and deformation of shaft in gearbox system.

To create the Shaft1

Shaft Modeler in RecurDyn defines the section as parts with same radius, length and material. Number of shaft section increases from starting point towards axial direction. Design of the Shaft1 is shown in the figure below.



1. From the **Shaft** group in the **DriveTrain** tab, click **Shaft**.
2. Set the creation method to **Point, Direction, WithDialog** and input the value using following information.
   - **Point:** -255, 250, 175
   - **Direction**: 1, 0, 0
3. After **Shaft** dialog appears, click **Add** button 3 times to make 4 **Sections** and adjust the values using following information.

Section	L	Ro	Ri	Element Size
1	120	42.5	0	10
2	160	75	0	10
3	230	42.5	0	10
4	200	39	0	10

4. Click **FDR** button which is located next to the **Sections** box. After **FDR** dialog appears, click **Add** button 3 times to make 3 **FDRs** and adjust the values using the following information.

No	Center Position	Width	Type
1	22.5	41	RBE2
2	200	120	RBE2
3	487.5	41	RBE2

5. Click **Close** to exit **FDR** dialog.
6. Click **OK** to create **Shaft**.
7. **Delete** the existing **Shaft body** where **Shaft1** has been created.

Note: FDR
Most of the time Shafts are attached with the machine elements like gear, bearing, etc. Connectors like pin, key, spline, snap ring are used to attach those elements. In RecurDyn, connections of these machine element are expressed as FDR element which is rigid element. If you look at the center position of the FDR which is master node, you can see that the size of master node is bigger than the size of other nodes.

Note: FDR Tolerance
When you create a FDR element, 3 nodes, center position and two end sides of FDR, are added to the existing nodes. The distance between existing node and the added FDR node can be very small. The FDR Tolerance is used to ignore the small element caused by creation of FDR. For example, if FDR Tolerance is 0.01 and the distance between two nodes are 0.009, the added FDR Element is ignored and altered as existing node.

To create the Shaft2

Create the **Shaft2** using same method as above using the following information.

1. **Point, Direction, WithDialog**
   - **Point:** -255, 250, -105
   - Direction: 1, 0, 0
2. **Shaft Section**

Section	L	Ro	Ri	Element Size
1	50	32.5	0	10
2	235	37.5	0	10
3	125	50	0	10
4	100	32.5	0	10

3. **FDR Section**

No	Center Position	Width	Type
1	22.5	33	RBE2
2	200	120	RBE2
3	347	90	RBE2
4	487.5	33	RBE2

To create the Shaft3

Create the **Shaft3** using same method as above using the following information

1. **Point, Direction, WithDialog**
   - **Point:** -402, 250, -325
   - Direction: 1, 0, 0
   -

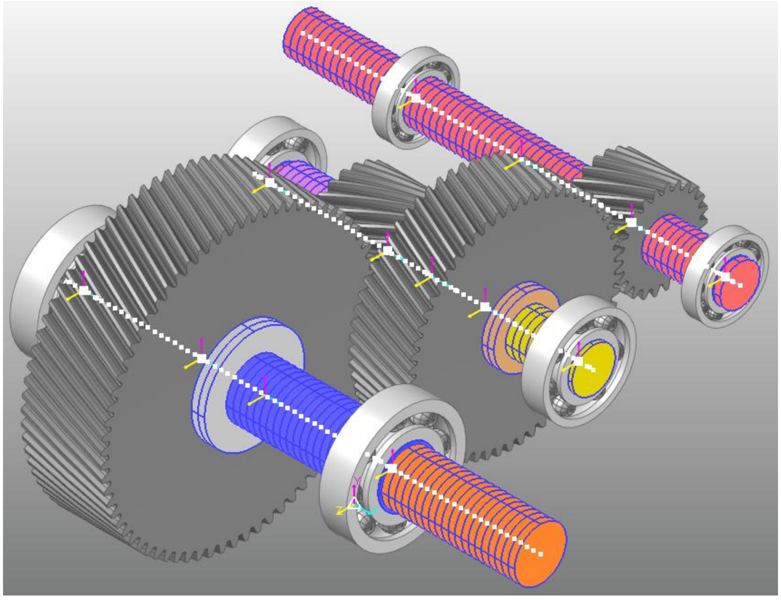
2. **Shaft Section**

Section	L	Ro	Ri	Element Size
1	657	30	0	10

3. **FDR Section**

No	Center Position	Width	Type
1	169.5	22	RBE2
2	494	90	RBE2
3	634.5	22	RBE2

The model appears as shown in the figure below.



Saving the Model

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

Chapter

4

Creating the Bearing

Task Objective

In this chapter, you will learn how to create a Ball Bearing using a **KISSsoft Ball Bearing Library**.

Estimated Time to Complete

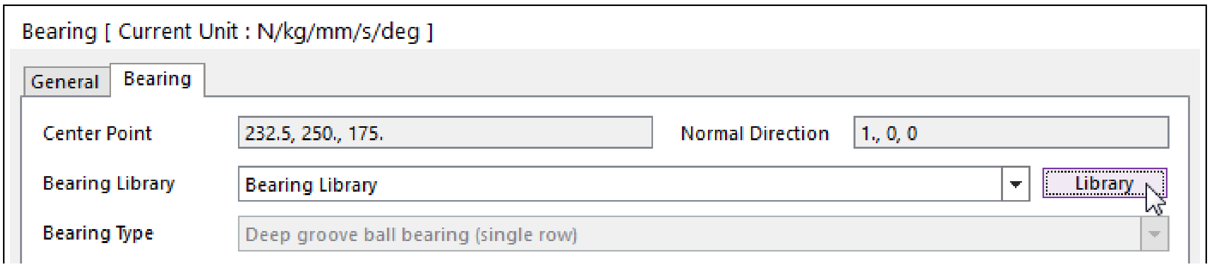
10 minutes

Creating the Bearing

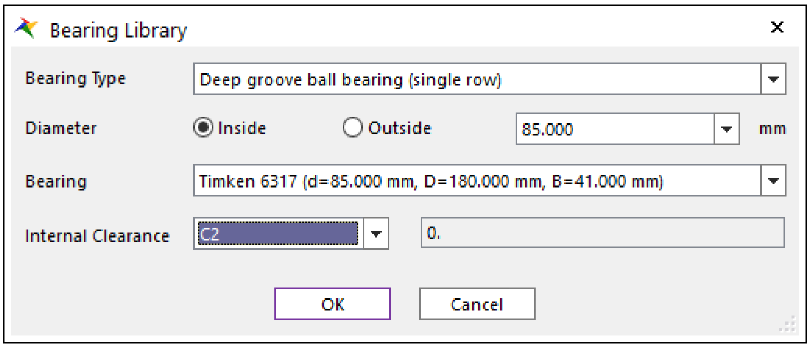
You will create a **KISSsoft Ball Bearing** to analyze the dynamic behavior of the **Ball Bearing** that attached to the shaft.

To create the BearingGroup1, 2

1. From the **KISSsoft** group in **DriveTrain** tab, click **Bearing**.
2. Set the creation method to **Point, Direction, WithDialog** and input the value using following information.
   - **Point:** 232.5, 250, 175
   - Direction: 1, 0, 0
3. After the **Bearing** dialog appears, click the **Library** button next to the Bearing Library.



4. After the **Bearing Library** dialog appears, adjust the values using the following information.
   - **Bearing Type**: Deep groove ball bearing (single row)
   - **Diameter**: (Inside) 85.000 mm
   - **Bearing**: Timken 6317 (d=85.000 mm, D=180.000 mm, B=41.000 mm)
   - **Internal Clearance**: C2



5. Click **OK** to close the **Bearing Library** dialog.
6. Click **OK** in the **Bearing** dialog to create **BearingGroup1**.
7. **Delete** the existing **Bearing body** where the **BearingGroup1** has been created.
8. **Repeat** the **Step 1~7** but change the **Point** value in the **Step 2** as (-232.5, 250, 175).

To create the BearingGroup3, 4

Create the **BearingGroup3, 4** using same method as above using the following information.

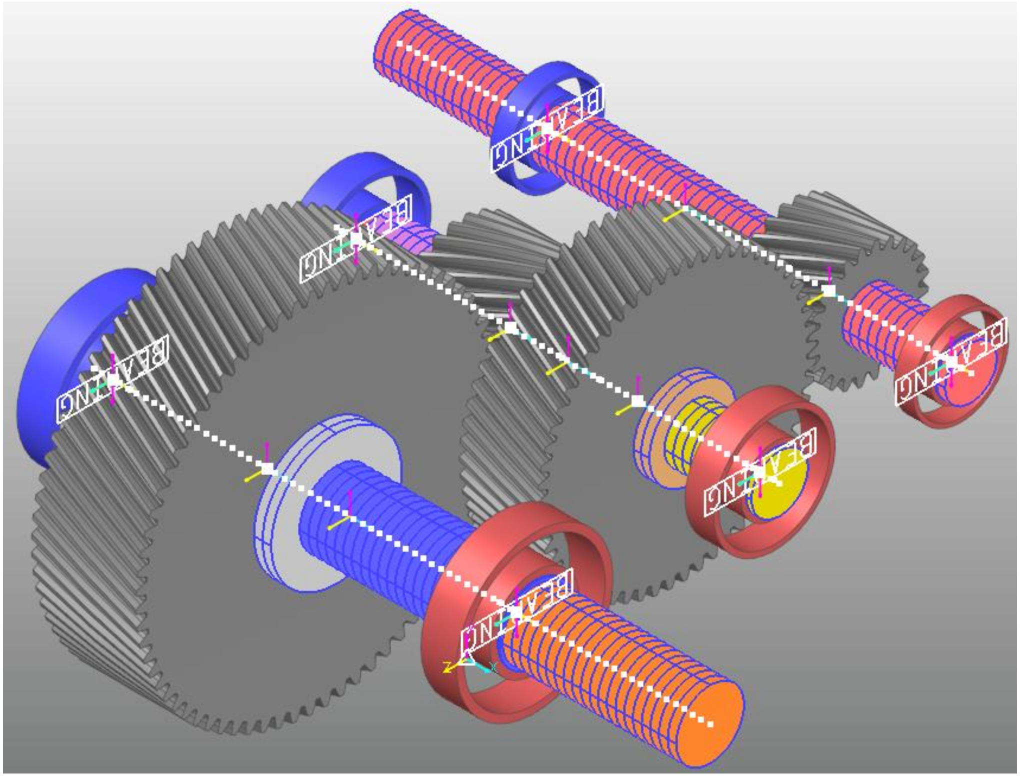
1. **Point, Direction, WithDialog**
   - **Point:** (232.5, 250, -105), (-232.5, 250, -105)
   - **Direction**: 1, 0, 0
2. **Bearing Library**
   - **Bearing Type**: Deep groove ball bearing (single row)
   - **Diameter**: (Inside) 65.000 mm
   - **Bearing**: Timken 6313 (d=65.000 mm, D=140.000 mm, B=33.000 mm)
   - **Internal Clearance**: C2

To create the BearingGroup5, 6

Create the **BearingGroup5, 6** using same method as above using the following information.

1. **Point, Direction, WithDialog**
   - **Point:** (232.5, 250, -325), (-232.5, 250, -325)
   - **Direction**: 1, 0, 0
2. **Bearing Library**
   - **Bearing Type**: Deep groove ball bearing (single row)
   - **Diameter**: (Inside) 60.000 mm
   - **Bearing**: Timken 6212 (d=60.000 mm, D=110.000 mm, B=22.000 mm)
   - **Internal Clearance**: C2

The model appears as shown in the figure below.



Saving the Model

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

Chapter

5

Creating the Gear

Task Objective

In this chapter, you will create a Gear Pair using a **KISSsoft Gear Train**.

Estimated Time to Complete

15 minutes

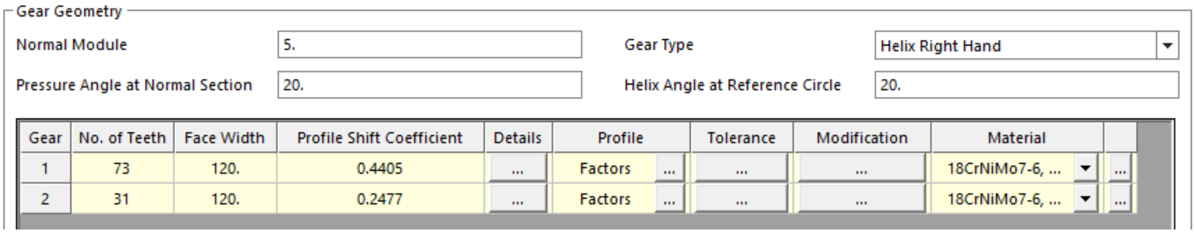
Creating the Gear

You will create a KISSsoft Gear Train to analyze the dynamic behavior of a gear in a gearbox system.

To create a CylindricalGearGroup1

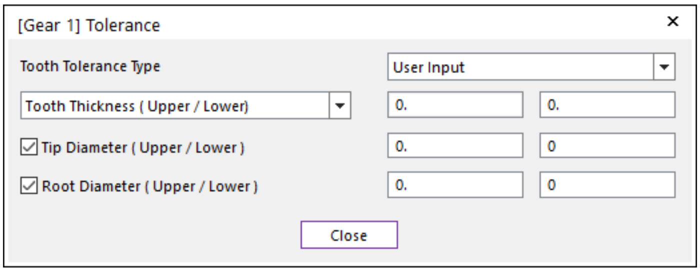
1. From the **KISSsoft** group in the **DriveTrain** tab, click **GearTrain**.
2. Set the creation method to **Point, Point, Direction, WithDialog** and input the value using the following information.
   - **Point:** -55, 250, 175
   - **Point:** -55, 250, -105
   - **Direction**: 1, 0, 0
3. After the **CylindricalGear** dialog appears, input the values in the **Gear Geometry** section using the following information.
   - **Normal Module**: 5
   - **Gear Type**: Helix Right Hand
   - **Helix Angle at Reference Circle**: 20
4. For rest of the parameters of **Gear1, 2,** input the values using following information. The dialog will appear as shown in the figure below.

Gear	No. of Teeth	Face Width	Profile Shift Coefficient
1	73	120	0.4405
2	31	120	0.2477



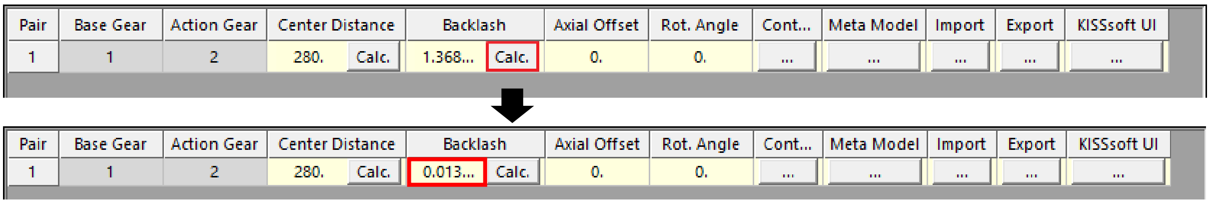
5. Click the **'…'** button in the **Details** of **Gear1**. Change the **Inner Diameter** value to **150** and click **Close** to exit the dialog. Change the **Inner Diameter** of **Gear2** to **75** using above method.
6. Click the **'…'** button in the **Tolerance** of **Gear1**. After **[Gear 1] Tolerance** dialog appears, input the value using following information and click **Close** to exit the dialog.
   - Tooth Tolerance Type: User Input
   - Tooth Thickness (Upper/Lower): (0, 0)
   - Tip Diameter (Upper/Lower): (Check) (0, 0)
   - Root Diameter (Upper/Lower): (Check) (0, 0)

- Then the dialog appears as shown in the figure below.



Note: Tooth Tolerance
In the Tolerance dialog, you set the tolerances of tooth thickness, tip diameter and root diameter. If you set the Upper / Lower value as 0, it means no tolerance in gear shape. If you set the Upper / Lower values, KISSsoft automatically calculate the appropriate value between the upper and lower limit.

7. **Repeat** the **Step 6** to the **Tolerance** of **Gear2**.
8. In **Gear Pair** section, change the **Center Distance** value to **280** and click the **Calc.** button in the **Backlash** to calculate backlash.

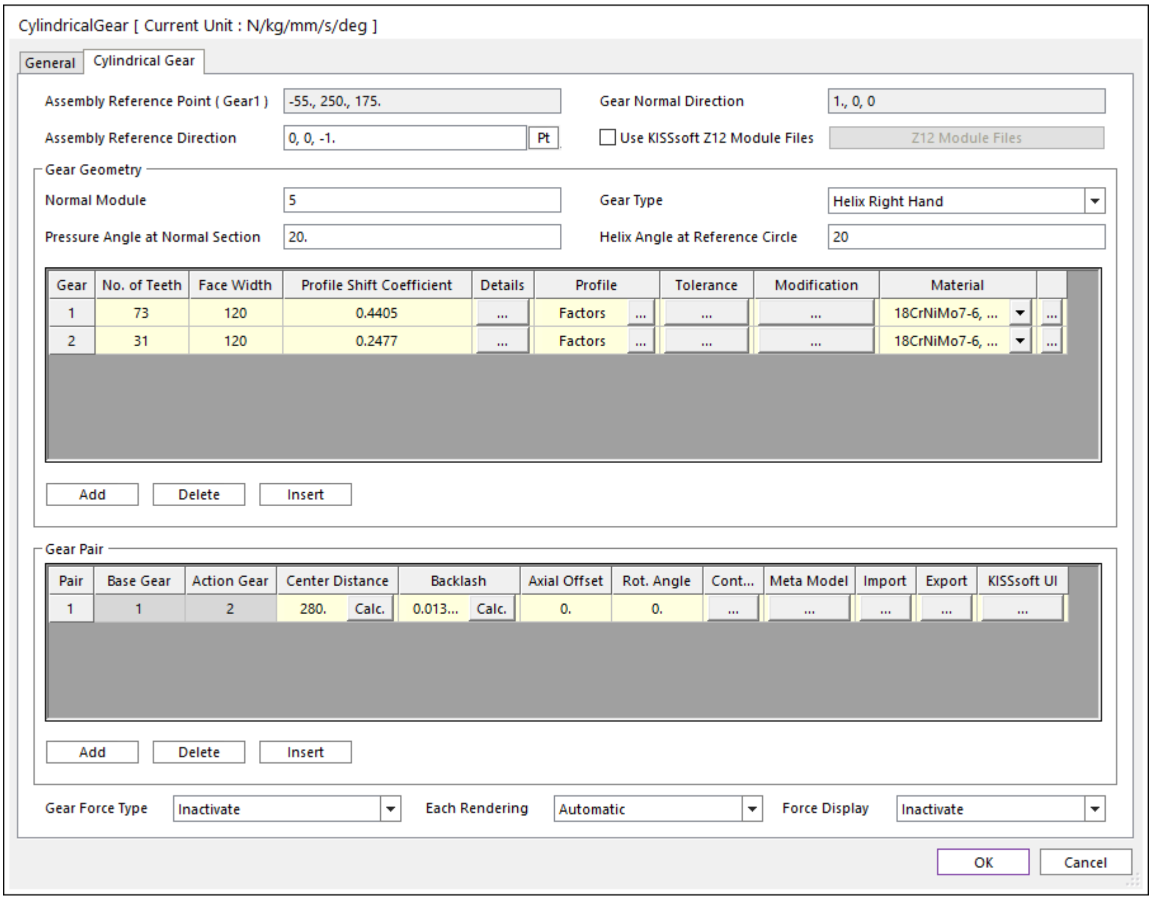


Note: Contact Analysis
In Contact Analysis dialog, the Number of Meshing Positions determines the number of calculation following the Path of Contact between a meshing teeth pair. The Number of Slices determines the number of axial cross sections of gear pair where the Meshing Position is calculated. For example, if you set the Number of Meshing Position as 17 and 11 as the Number of Slices, the 17 meshing positions are distributed evenly throughout the path of contact in each one of 11 slices while the surface of gear tooth contacts the tooth surface of other gear.

Note: Difference between RecurDyn Gear Toolkit
The Gear Train dialog in **DriveTrain** Toolkit is created based on the **KISSsoft User Interface**. Some of the nomenclature is different from the RecurDyn Gear Toolkit. Below is the comparison of those parameters that has the same meaning but different names.

KISSsoft	RecurDyn
Normal Module	Module
Pressure Angle at Normal Section	Pressure Angle
Helix Angle at Reference Circle	Helix Angle
Face Width	Gear Width
Profile Shift Coefficient	Addendum Modification Coefficient
Inner Diameter	Hole Radius
Dedendum Coefficient	Dedendum Factor
Root Radius Coefficient	Hob Rack Radius Coefficient
Addendum Coefficient	Addendum Factor

Then, the **CylindricalGear** dialog appears as shown in the figure below.



9. Click **OK** in **CylindricalGear** dialog to create **CylindricalGearGroup1**.
10. **Delete** the existing **Gear body** where the **CylindricalGearGroup1** has been created.

Note: Gear Modification
If you click the Modification button, you can create the various types of tooth modification. Profile/Tooth Modification has different input value regarding its type. For detailed information, refer to the manual. (DriveTrain > Functions for DriveTrain > KISSsoft > Gear Train > Properties > Modification)

To create a CylindricalGearGroup2

Create the second Gear Pair using same method as above using following information.

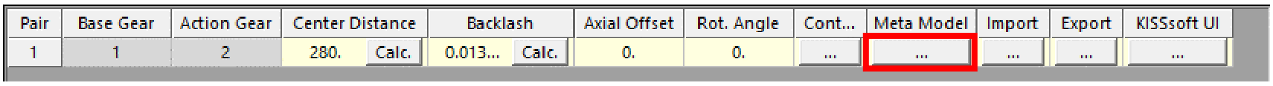
1. **Point, Point, Direction, WithDialog**
   - **Point:** 92, 250, -105
   - **Point:** 92, 250, -325
   - **Direction**: 1, 0, 0
2. **Gear Geometry**
   - Normal Module: 5
   - **Gear Type**: Helix Right Hand
   - **Helix Angle at Reference Circle**: 20

Gear	No. of Teeth	Face Width	Profile Shift Coefficient
1	59	90	0.2193
2	23	90	0.1595

3. **Inner Diameter**
   - **Gear1**: 100
   - **Gear2**: 60
4. **Gear1**, **Gear2 Tolerance**
   - **Tooth Tolerance Type**: User Input
   - **Tooth Thickness (Upper/Lower)**: (0, 0)
   - **Tip Diameter (Upper/Lower)**: (Check) (0, 0)
   - **Root Diameter (Upper/Lower)**: (Check) (0, 0)
5. Change the **Center Distance** to **220** and click **Calc.** button of **Backlash**.

To adjust the Contact

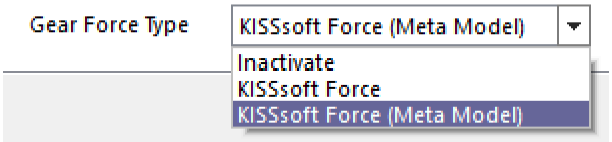
1. In the **Database**, right-click the **CylindricalGearGroup1** and click **Properties** in the Pop-up Menu.
2. In the **Gear Pair** section, Click the **...** button in the **Meta Model**.



3. Check the **Import Meta Model File(*.gmm)** and click the **...** button.
4. **Open** the GearBox_GearForce1.gmm file from directory below.

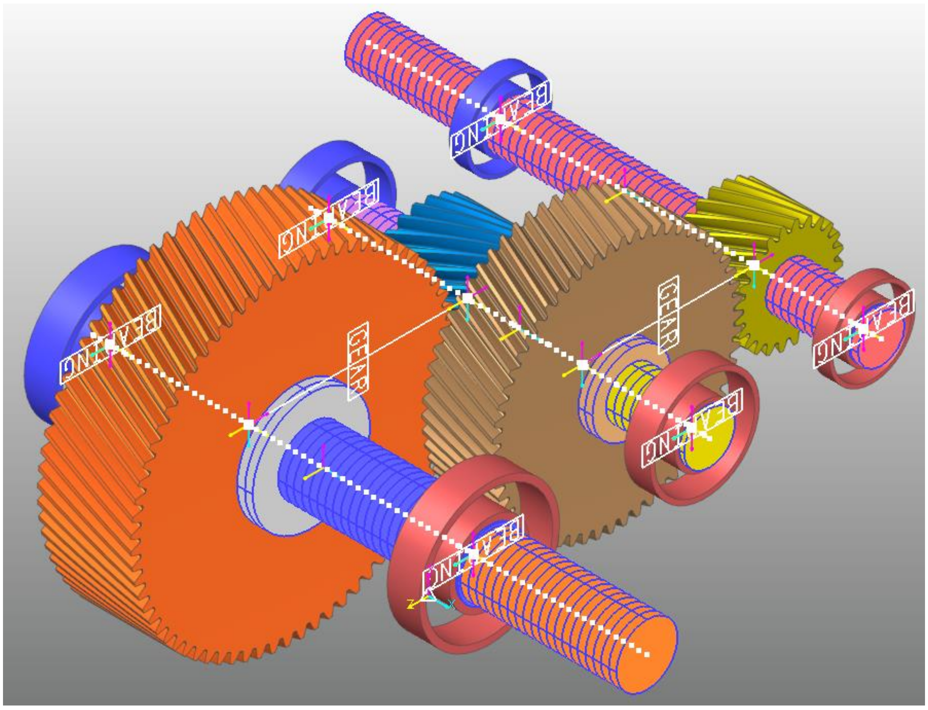
- <InstallDir>\Help\Tutorial\Toolkit\DriveTrain\GearBox

5. Close the **Meta Model** dialog.
6. Set the **Gear Force Type** to **KISSsoft Force (Meta Model)** as figure below.



7. Click **OK** to close the dialog.
8. Repeat the **Step 1~7** for the **CylindricalGearGroup2** but changing the file to GearBox_GearForce2.gmm.

   Then, model appears as shown in the figure below.



Saving the Model

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

Chapter

6

Creating the Joint and Force

Task Objective

In this chapter, you will create the Joints and Forces.

- **Fixed Joint** between **Ground** and **Housing**
- **Fixed Joint** between **Housing** and **Outer Bearing**
- **Fixed Joint** between **Shaft** and **Inner Bearing**
- **Revolute Joint** between **Ground** and **Shaft**
- **Rotational Axial Force** between **Ground** and **Shaft**

Estimated Time to Complete

20 minutes

Creating the Expression

You will create the expression which will be applied to Joints and Forces,

1. From the **SubEntity** tab, click **Expression**.
2. In the **Expression List** dialog, click **Create**.
3. In the **Expression** dialog, create the expression using the following information.

Name	Expression
Ex_Vel	10*pi*time
Ex_Torque	-640000

Creating the Joint

To create the fixed joint

1. From the **Joint** group in the **Professional** tab, click **Fixed Joint**.
2. Set the creation method to **Body, Body, Point** and input the value using the following information.
   - **Body**: Ground
   - **Body**: LowerHousing
   - **Point**: 245, -5, 485
3. Create the **Fixed Joint** between the **Housing** and **Bodies** using the following information.

Name	Body	Body	Point
Fixed1	Ground	LowerHousing	245, -5, 485
Fixed2	LowerHousing	UpperHousing	205, 250, 500
Fixed3	LowerHousing	BearingOuterBody1	232.5, 250, 175
Fixed4	LowerHousing	BearingOuterBody2	-232.5, 250, 175
Fixed5	LowerHousing	BearingOuterBody3	232.5, 250, -105
Fixed6	LowerHousing	BearingOuterBody4	-232.5, 250, -105
Fixed7	LowerHousing	BearingOuterBody5	232.5, 250, -325
Fixed8	LowerHousing	BearingOuterBody6	-232.5, 250, -325

4. Create the **Fixed Joint** between the **Shaft** and **body** using the following information.

Name	Body	Body	Point
Fixed9	Shaft1	BearingInnerBody1	232.5, 250, 175
Fixed10	Shaft1	CylindricalGear1	-55, 250, 175
Fixed11	Shaft1	BearingInnerBody2	-232.5, 250, 175
Fixed12	Shaft2	BearingInnerBody3	232.5, 250, -105
Fixed13	Shaft2	CylindricalGear2	-55, 250, -105
Fixed14	Shaft2	CylindricalGear3	92, 250, -105
Fixed15	Shaft2	BearingInnerBody4	-232.5, 250, -105
Fixed16	Shaft3	BearingInnerBody5	232.5, 250, -325
Fixed17	Shaft3	CylindricalGear4	92, 250, -325
Fixed18	Shaft3	BearingInnerBody6	-232.5, 250, -325

To create the revolute joint

You will create the Revolute Joint to set the motion in the gearbox.

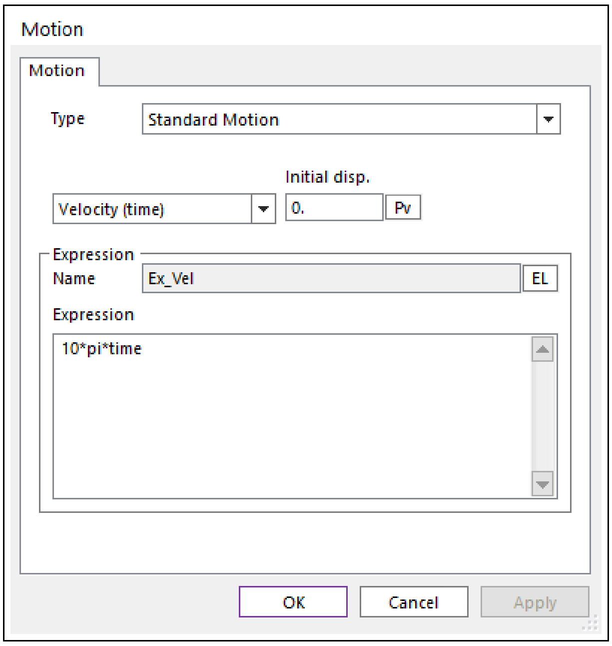
1. From the **Joint** group in the **Professional** tab, click **Revolute Joint**.
2. Set the creation method to **Body, Body, Point, Direction** and input the value using following information.
   - **Body**: Ground
   - **Body**: Shaft3
   - **Point**: -402, 250, -325
   - **Direction**: 1, 0, 0
3. Create the **Revolute Joint** between the **Shaft** and **Ground** using the following information.

Name	Body	Body	Point	Direction
RevJoint1	Ground	Shaft3	-402, 250, -325	1, 0, 0
RevJoint2	Ground	Shaft1	455, 250, 175	1, 0, 0

To adjust the motion

You will set the **Motion** in the **RevJoint1**.

1. In the **Database**, right-click the **RevJoint1** and click the **Properties** in the Pop-up menu.
2. Check the **Include Motion** and click **Motion**.
3. Change **Displacement (time)** to **Velocity (time)** and click **EL**.
4. Select **Ex_Vel** and click **OK**. The dialog appears as shown right.



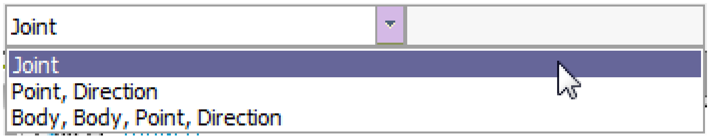
Creating the Force

To create the rotational axial force

You will create the Rotational Axial Force to set the input torque of gearbox.

From the **Force** group in the **Professional** tab, click **Rotational Axial Force**.

1. Set the creation method to **Joint** as shown in the figure below.



2. Click **RevJoint1** to create **RotationalAxial1**.

To create the rotational axial force

1. In the **Database**, right-click the **RotationalAxial1** and click the **Properties** in the Pop-up menu.
2. In the **Rotational Axial Force** tab, click the **EL** button.
3. After **Expression List** dialog appears, select **Ex_Torque** by clicking it and click **OK**.
4. Click **OK** to close the dialog.

Performing Dynamic/Kinematic Analysis

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

1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
2. In the **General** tab, define the end time of the simulation and number of steps:
   - **End Time**: 2
   - **Step**: 5000
   - **Plot Multiplier Step Factor**: 2
3. Click **Simulate.** It will take approximately 3 minutes to complete the analysis. (CPU: Intel® Core™ i7-6700K CPU @ 4.00GHz)

Chapter

7

Analyzing the Simulation Result

Task Objective

In this chapter, you will analyze the result of gearbox simulation.

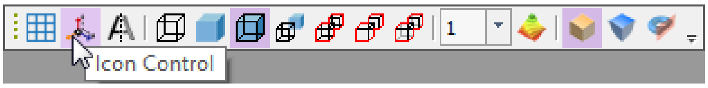
Estimated Time to Complete

20 minutes

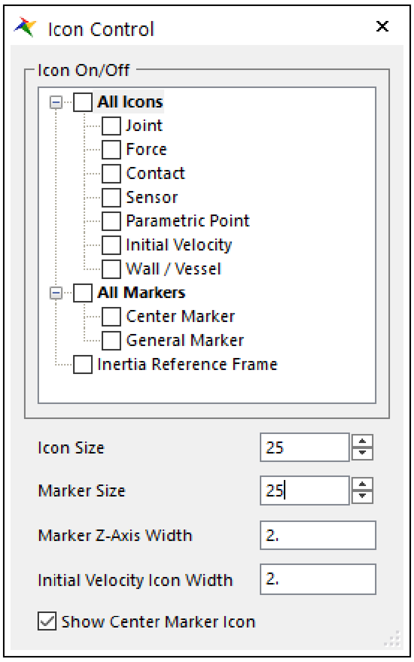
Analyzing the Shaft

Adjust the icon control

1. In the **Render Toolbar**, click **Icon Control** as shown in the figure below.

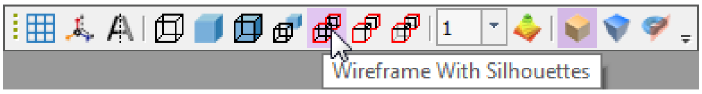


2. **Check Off** the **All Icons**, **All Markers** and **Inertia Reference Frame** as shown at right.

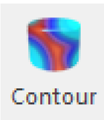


Adjust the rendering mode

1. In the **Render Toolbar**, click **Wireframe with Silhouettes.**

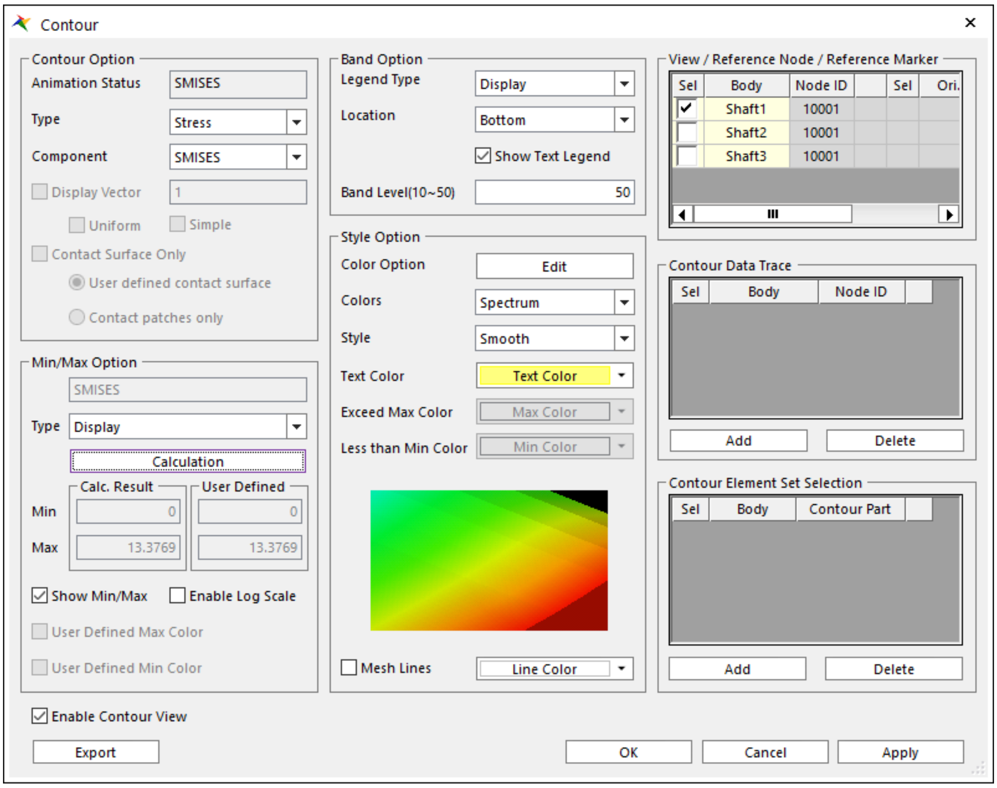


Adjust the contour



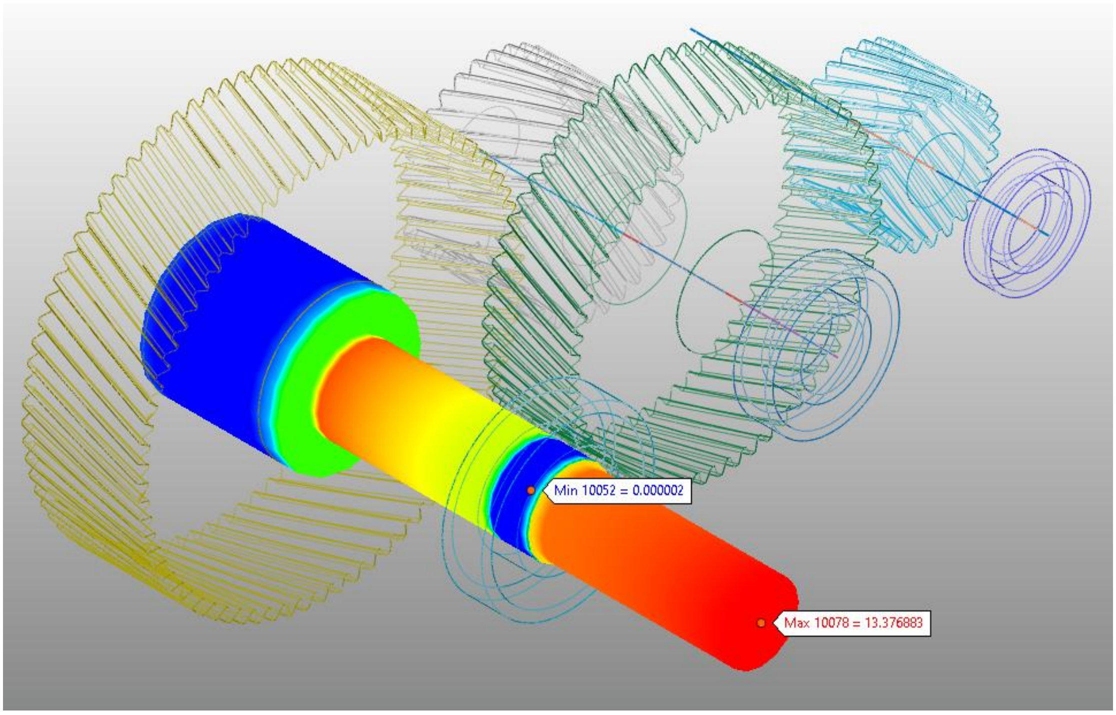
1. From the **Shaft** group in the **DriveTrain** tab, click **Contour**.
2. In the **Contour Option**, change the **Type** as **Stress** and **Component** as **SMISES**.
3. In the **Style** option, change the **Style** from **Stepped** to **Smooth**.
4. In the right-side of the dialog, check off the **Shaft2, Shaft3** in the **View/Reference Node/Reference Marker**.
5. Click the **Calculation** button, check the **Show Min/Max** option and click **OK**.

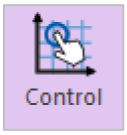
Then, the **Contour** dialog appears as shown in the figure below.



To play an animation

1. From the **Animation Control** group in the **Analysis** tab, click **Play**.
   Then, maximum stress occurs in the **Shaft1** after 36 frames.

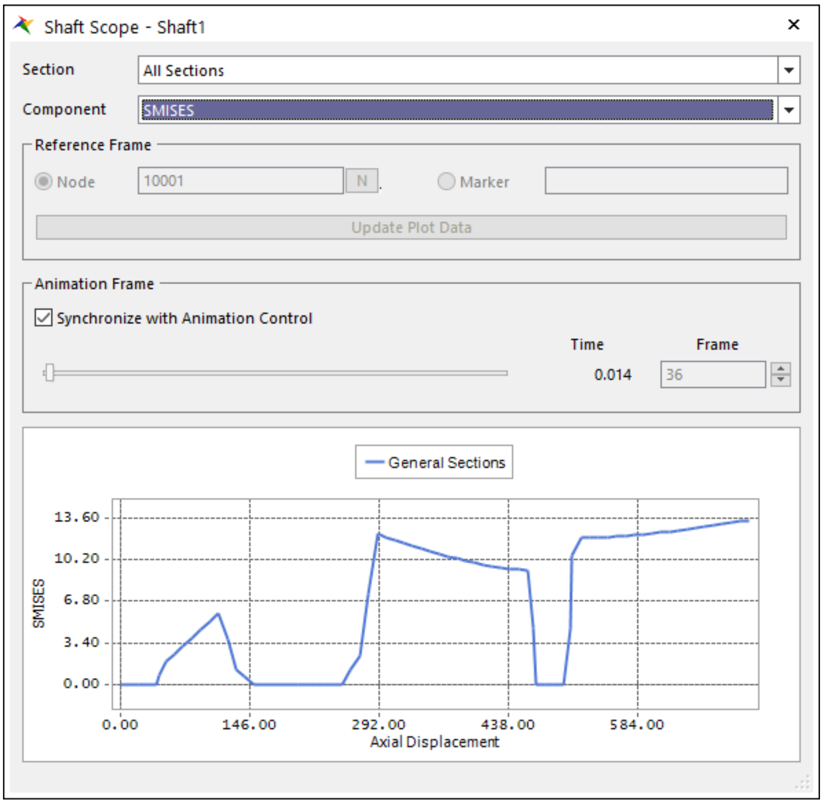




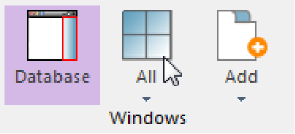
To view the shaft1 scope

1. From the **Shaft** group in the **KISSsoft** tab, click **Scope Control**.
2. After the **Shaft Scope Control** dialog appears, check **Use** option next to the **Shaft1** and click **Display**.
3. After the **Shaft Scope – Shaft1** dialog appears, adjust the **Component** as **SMISES**.
4. Make sure the **Synchronize with Animation Control** option is checked in the **Animation Frame** section.
5. From the **Animation Control** group in the **Analysis** tab, click **Animation Play**.

   Then, you can see the changes of the **SMISES** of **Shaft1** with **Animation** as shown in the figure below.

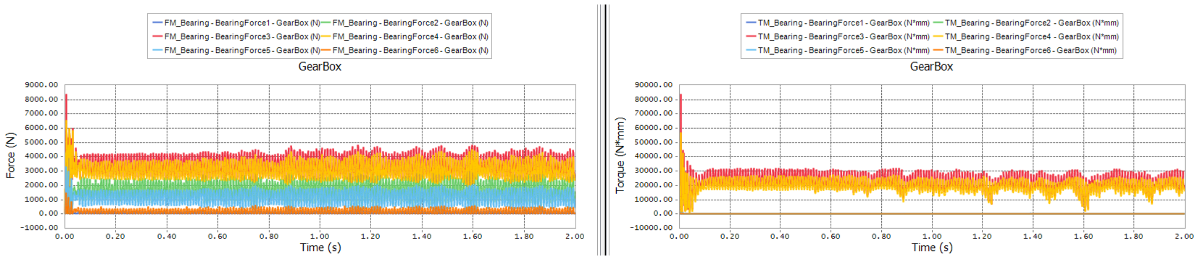


Analyzing the Bearing



1. From the **Plot** group in the **Analysis** tab, click **Plot Result**.
2. From the **Windows** group in the **Home** tab, click **Show All Windows**.
3. In the **Plot Database**, click the '**+**' button next to the **Force**.
4. Click the '**+**' button next to the **DriveTrain_BearingForce**.
5. Click the '**+**' button next to the **BearingForce1**.
6. Click the upper-left pane of the **Plot Window** and double-click the **FM_Bearing** from the **Plot Database**.
7. Click the upper-right pane of the **Plot Window** and double-click the **TM_Bearing** from the **Plot Database**.
8. **Repeat** the **Step 5~7** for the **BearingForce2~6**.

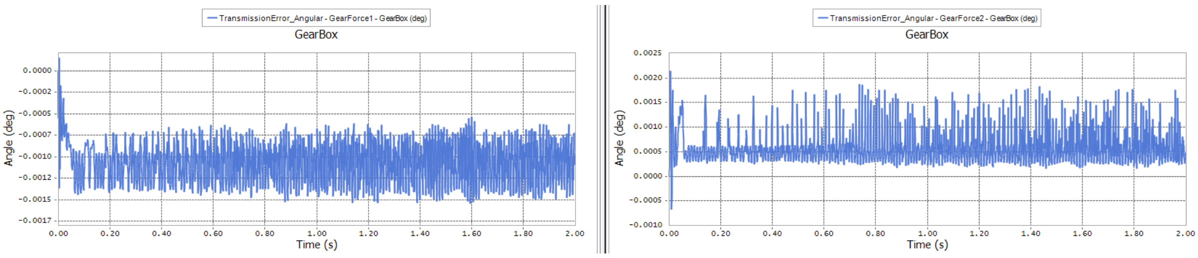
   Then, **Plot Window** appears as shown in the figure below.



Analyzing the Gear

1. Click the '**+**' button next to the **DriveTrain_GearForce**.
2. Click the each '**+**' button next to the **GearForce1** and **GearForce2**.
3. Click the lower-left pane of the **Plot Window** and double-click the **TransmissionError_Angular** from the **GearForce1**.
4. Click the lower-right pane of the **Plot Window** and double-click the **TransmissionError_Angular** from the **GearForce2**.

   Then, **Plot Window** appears as shown in the figure below.



Chapter
8

Involute Analytic Contact

Task Objective

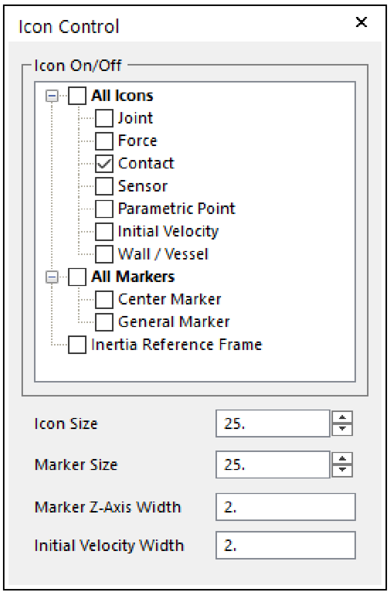
In this chapter, you will learn how to use the Involute Analytic Contact.

Estimated Time to Complete

20 minutes

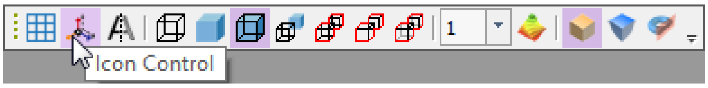
Creating the Involute Analytic Contact

Return to the GearBox model, you will learn how to use RecurDyn Involute Analytic Contact instead of KISSsoft Gear Contact.



Adjust the icon control

1. In the **Render Toolbar**, click **Icon Control** as shown in the figure below.



2. Check the **Contact** as shown at right.

Inactive the kisssoft gear contact

1. In the **Database**, right-click the **CylindricalGearGroup1** and click **Property** in the Pop-up menu.
2. Change the **Gear force Type** option to **Inactivate** at the bottom of Property dialog.
3. Click **OK** to close the dialog.
4. **Repeat** the **Step 1~3** for the **CylindricalGearGroup2**.

To create the involute analytic contact

1. From the **Contact** group in **DriveTrain** tab, click **Involute Analytic Contact**.
2. Set the creation method to **KISSsoft body, KISSsoft body** and select the **CylindricalGear1** and **CylindricalGear2**.
3. **Repeat** the **Step 1~2** for the **CylindricalGear3**, **CylindricalGear4**.

To adjust the involute analytic contact

1. In the **Database**, right-click the **GearContactInvolute1** and click **Property** in the Pop-up menu.
2. In the **Gear Involute Contact** tab, adjust the values using the following information.
   - **No. of Slices in Width**: 31
   - No**. of Maximum Involute Profile**: 10
3. In **Advanced Option**, check **Tooth Flexibility**.
4. **Repeat** the **Step 1~3** for the **GearContactInvolute2**.

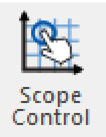
Performing Dynamic/Kinematic Analysis

In this section, you will run a dynamic/kinematic analysis to view the effect of Involute Analytic Contact on the model you just created.

1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
2. Click **Simulate**.

Analyzing the Shaft

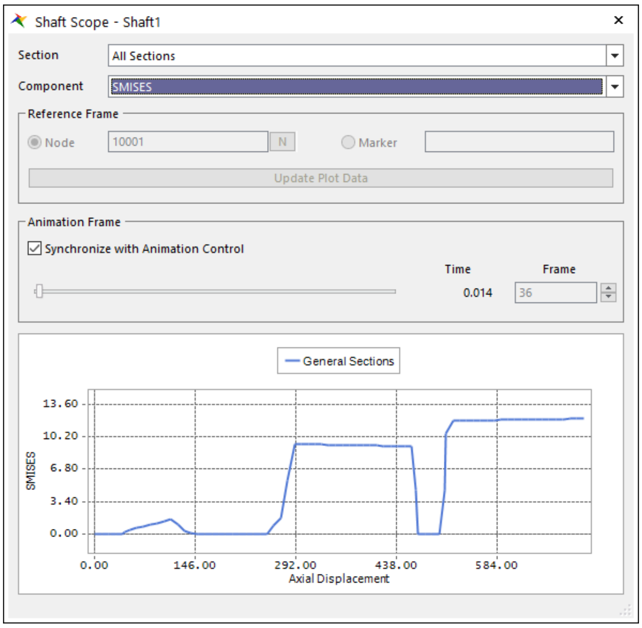
To view the shaft1 scope



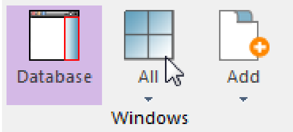
1. From the **Shaft** group in the **KISSsoft** tab, click **Scope Control**.
2. After the **Shaft Scope Control** dialog appears, check **Use** option next to the **Shaft1** and click **Display.**
3. After the **Shaft Scope – Shaft1** dialog appears, adjust the **Component** as **SMISES**.
4. Make sure the **Synchronize with Animation Control** option is checked in the **Animation Frame** section.
5. From the **Animation Control** group in the **Analysis** tab, click **Animation Play**

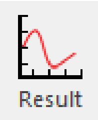
   Then, you can see the changes of the **SMISES** of **Shaft1** with **Animation** as shown in the figure below.

   The graph is almost same as the SMISES graph from **Chapter 7**.



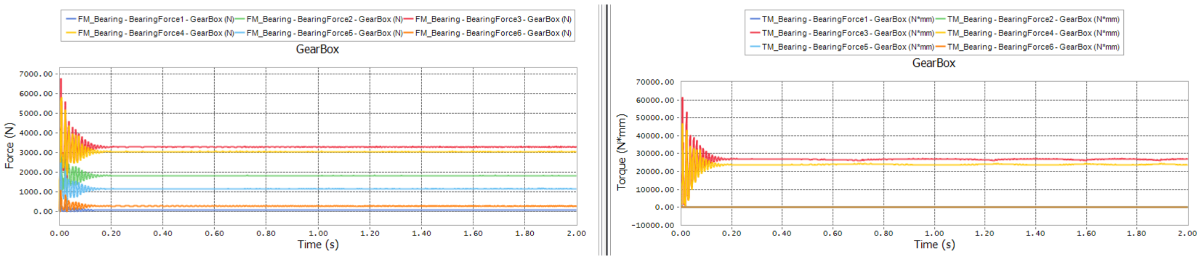
Analyzing the Bearing





1. From the **Plot** group in the **Analysis** tab, click **Plot Result**.
2. From the **Windows** group in the **Home** tab, click **Show All Windows**.
3. In the **Plot Database**, click the '**+**' button next to the **Force**.
4. Click the '**+**' button next to the **DriveTrain_BearingForce**.
5. Click the '**+**' button next to the **BearingForce1**.
6. Click the upper-left pane of the **Plot Window** and double-click the **FM_Bearing** from the **Plot Database**.
7. Click the upper-right pane of the **Plot Window** and double-click the **TM_Bearing** from the **Plot Database**.
8. **Repeat** the **Step 5~7** for the **BearingForce2~6**.

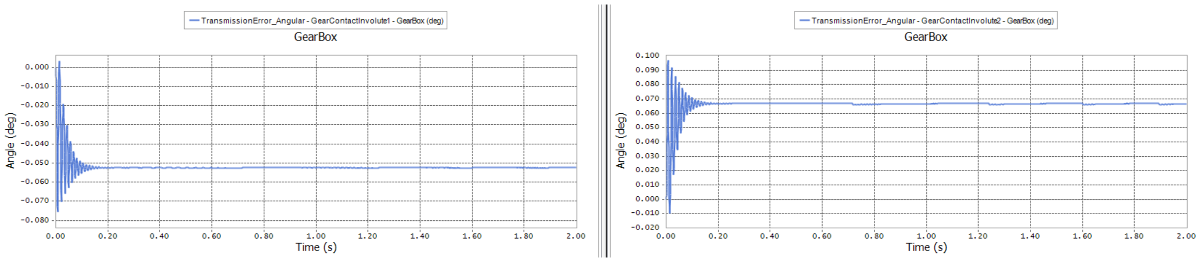
   Then, **Plot Window** appears as shown in the figure below



Analyzing the Gear

1. Click the '**+**' button next to the **Contact**.
2. Click the '**+**' button next to the **Gear Involute Contact**.
3. Click the each '**+**' button next to the **GearContactInvolute1** and **GearContactInvolute2**.
4. Click the lower-left pane of the **Plot Window** and double-click the **TransmissionError_Angular** from the **GearContactInvolute1**.
5. Click the lower-right pane of the **Plot Window** and double-click the **TransmissionError_Angular** from the **GearContactInvolute2**.

   Then, **Plot Window** appears as shown in the figure below.



Chapter

9

Campbell Diagram

Task Objective

In this chapter, you will learn how to use the Campbell Diagram.

Estimated Time to Complete

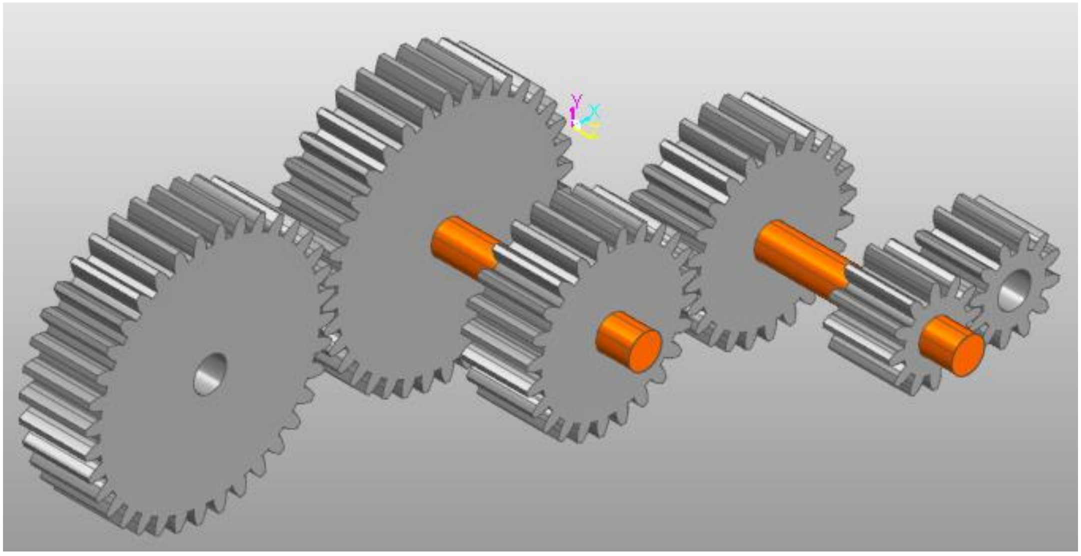
20 minutes

Simulate the Campbell Diagram Model

You will open and simulate the already saved Campbell Diagram model.

To open the Campbell Diagram model

1. From the **File** menu, click **Open**.
2. From the **Drivetrain** tutorial directory, select the file Capmbell_Diagram.rdyn.
- Path: <InstallDir>\Help\Tutorial\Toolkit\Drivetrain\GearBox
3. Click **Open**.
   The model appears as shown in the figure below.



To save the Campbell Diagram model

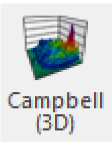
1. From the **File** menu, click **Save As**.
2. Save the model in different directory, because you cannot simulate in **tutorial** directory.

To simulate the Campbell Diagram model

You will run the Dynamic/Kinematic analysis because the model is already set.

1. From the **Simulation Type** group in the **Analysis** tab, click **Dyn/Kin**.
2. Click **Simulate**. It will take approximately 30 seconds to complete the analysis. (CPU: Intel® Core™ i7-6700K CPU @ 4.00GHz)

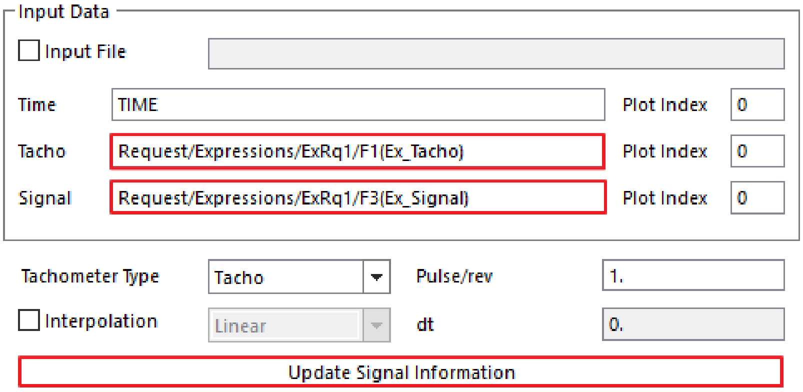
Adjusting the Analysis Tab



To adjust the Input Data

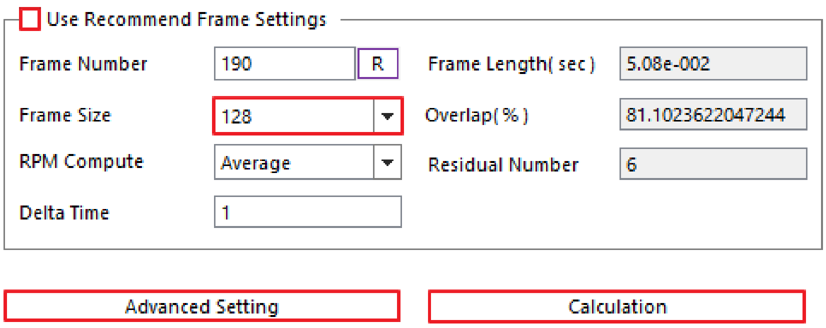
1. From the **Plot** group in the **Analysis** tab, click **Plot Result**.
2. From **Tool** tab, click **Campbell (3D)**.

3. In **Plot Database**, expand the **Request\Expression\ExRq1**.
4. **Drag & Drop** the **F1(Ex_Tacho)** in the **Tacho** from **Input Data**.
5. **Drag & Drop** the **F3(Ex_Signal)** in the **Signal** from **Input Data**.
6. Click the **Update Signal Information** to update the below parameters.



To adjust the Frame Settings

1. Check off the **Use Recommended Frame Settings**.
2. Change **Frame Size** to **128**.
3. Click **Advanced Setting**.
4. Change **Window Type** to **Bartlett**.
5. Click **Calculation**.

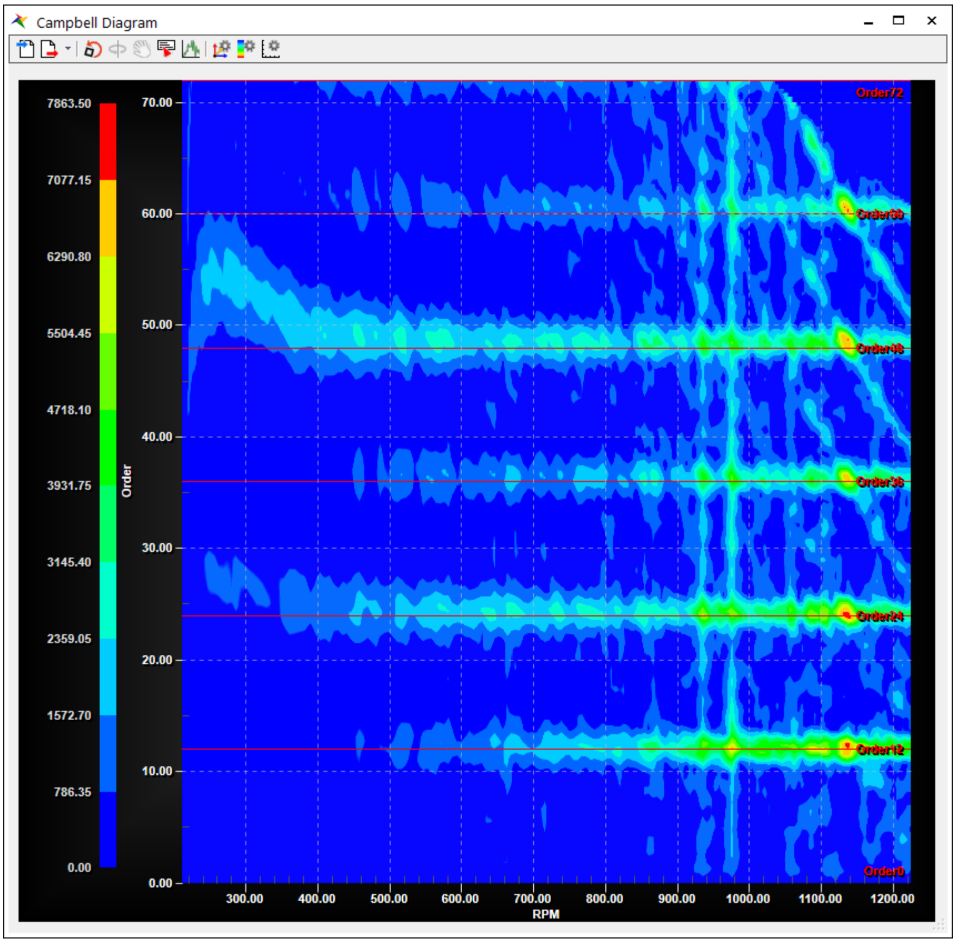


Adjusting the Plot Tab

1. From **Campbell Diagram** dialog, click **Plot** tab.
2. Change **Graph Type** to **Color Map (2D)**.
3. Change **Graph Option** to **RPM-Order**.
4. Check **Draw Order Line**.
5. Adjust the **Order Line** parameters using the following information.
   - **Minimum Order**: 0
   - **Maximum Order**: 72

- **Gap**: 12
- **Resolution**: 100

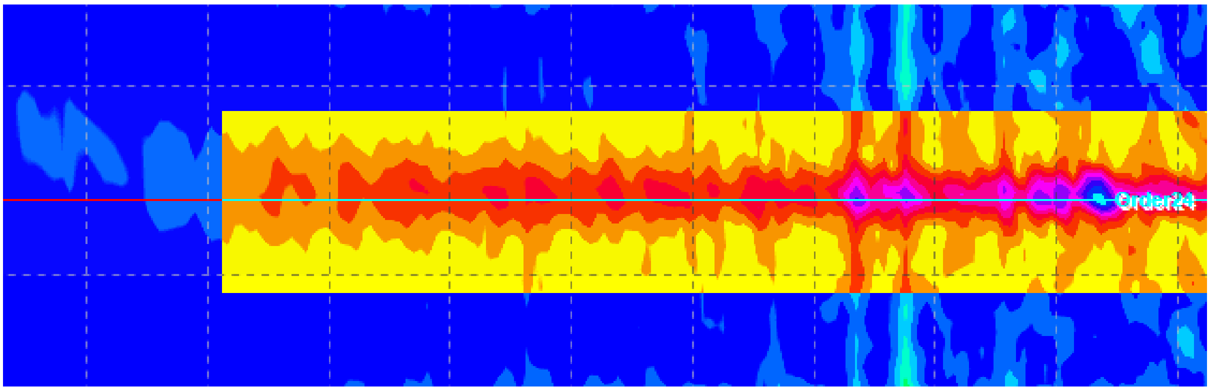
6. In the below-part of the dialog, click **Plot** to open the graph. The graph will be shown as below.



Adjusting the Campbell Diagram

To use Zoom

1. **Drag** as below figure to include **Order 24**.



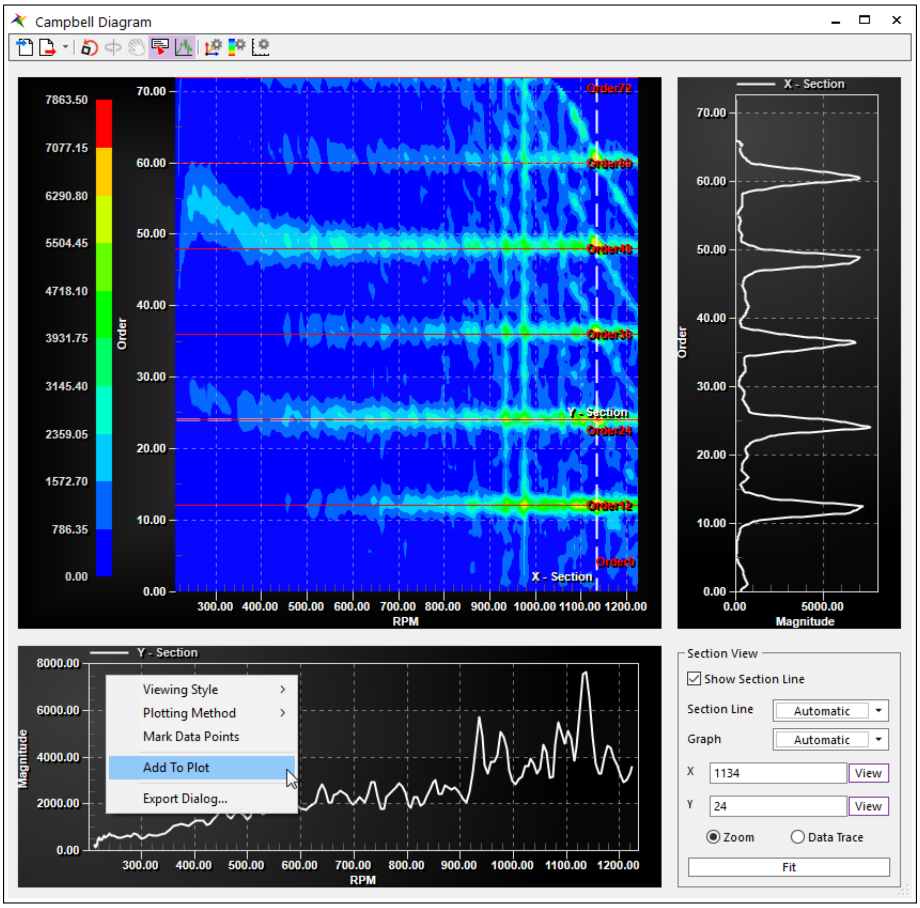
2. The selected part is **Zoomed**.
3. From **Toolbar**, click **Initialize View** to initialize the view.

To use Section View

1. From **Toolbar**, click **Section View**.

2. At the left-side of the **Section View**, click **Trace Data**.

3. To set the **Section**, left click in the graph where **X, Y** is **(1134, 24)**.
4. Right-click in the **Y-Section** as shown in the figure below and click **Add to Plot** in the Pop-up menu.



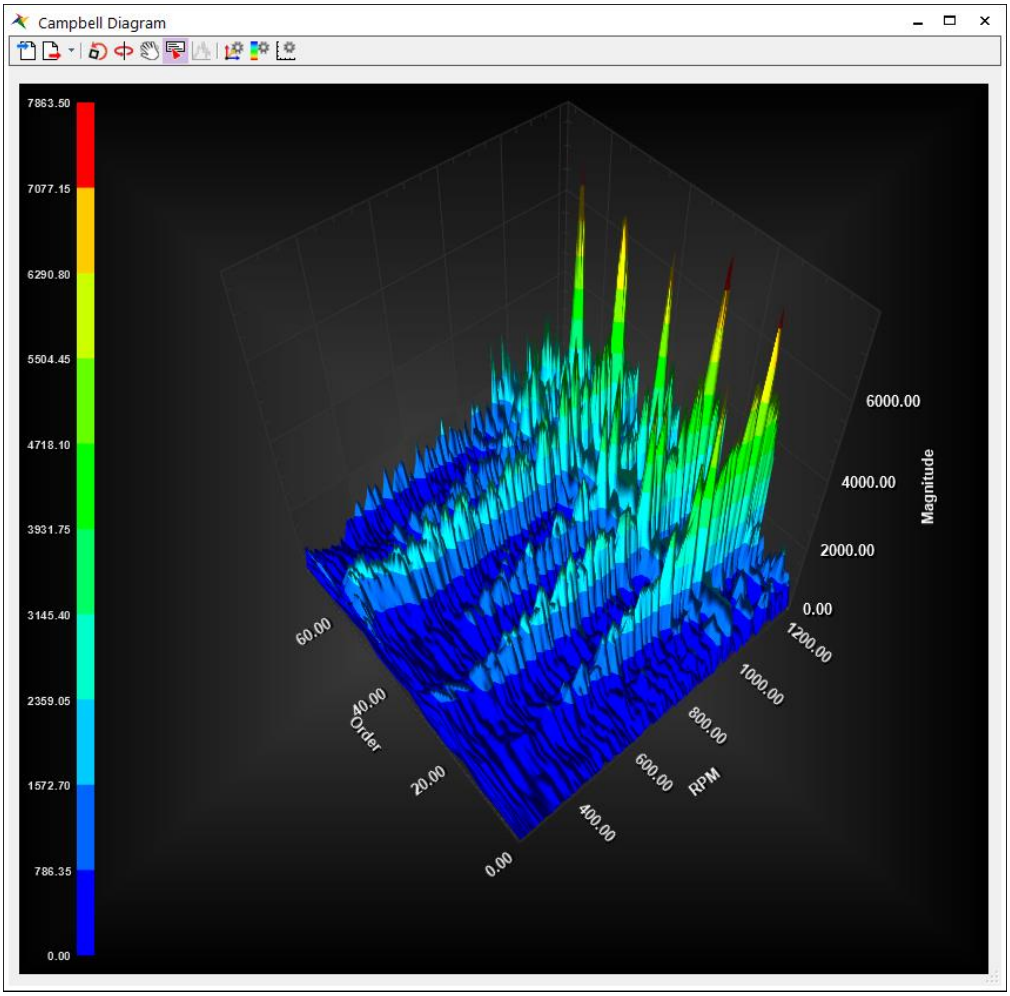
5. **Y-Section Data** is drawn in the **Plot**.

To use 3D plot

1. Close the **Graph** and go back to the **Campbell Diagram** dialog.
2. In the **Plot** tab, change the **Graph Type** to **Surface Contour (3D)**.

3. Click **Plot** in the below-part of the dialog.

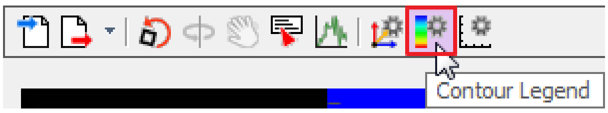
   3D Plot will be shown as figure below.



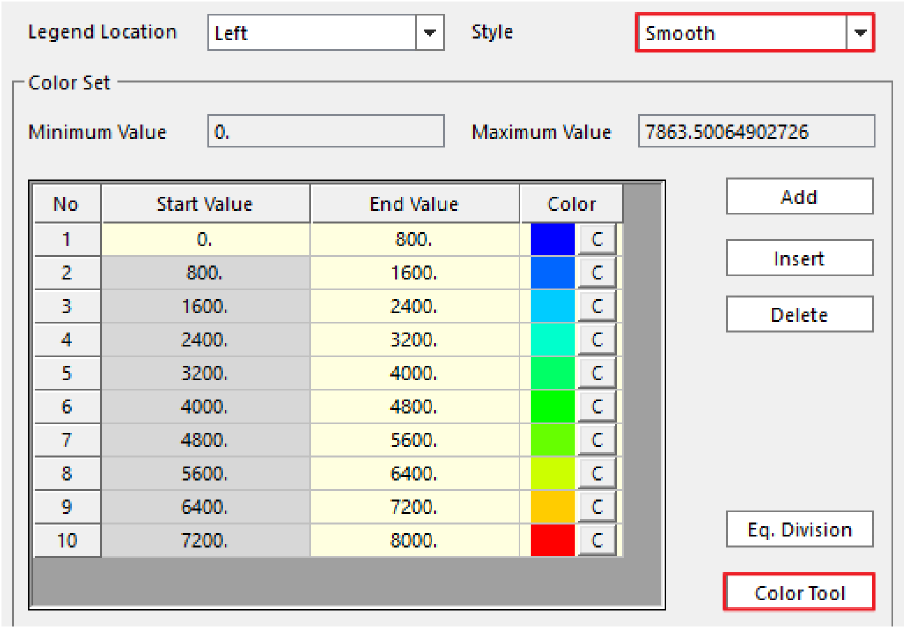
Note: Mouse Operation
Rotate View: Dragging with left click will rotate the view.
Translate View: Dragging with left click while pressing Shift key will rotate the view.
Light Change: Dragging with middle click (mouse scroll button) will change the location of the light source.

To adjust the Contour

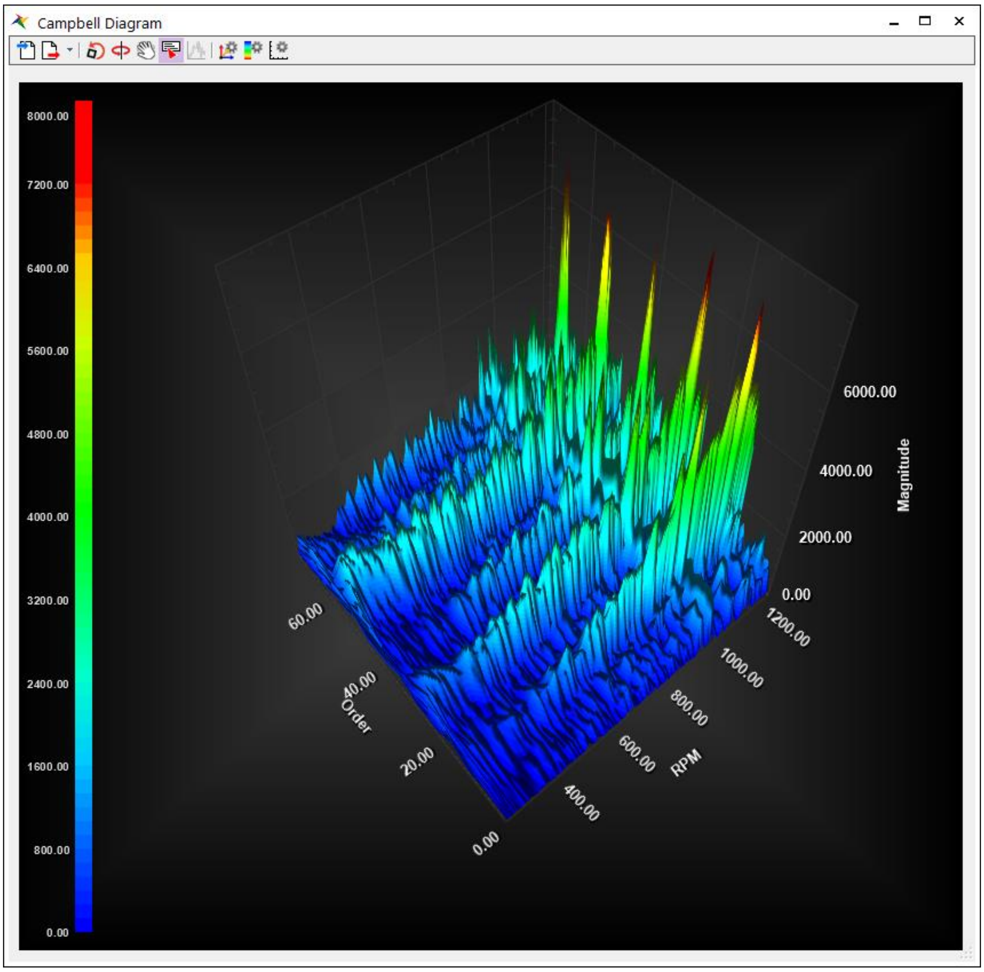
1. In the **Toolbar**, click **Contour Legend**.



2. Change **Style** to **Smooth**.
3. In the **Color Set** section, click **Color Tool**.
4. In the **Color Tool** dialog, change **Max Value** to **8000**, click **Update** and **OK**.
5. **Contour Legend** dialog will be shown as below.



6. Click **OK** to apply the changes. The graph will be shown as below.



Thanks for participating in this tutorial!