

Gearbox Tutorial (DriveTrain)





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

Getting Started	5
Objective	. 5
Prerequisites	. 6
Procedures	. 6
Estimated Time to Complete	.6
Setting Up the Simulation Environment	7
Task Objective	. 7
Estimated Time to Complete	. 7
Starting the RecurDyn	. 8
Importing the Gearbox Geometry	. 8
Adjusting the Icon, Marker Size and Layer	10
Saving the Model	11
Creating the Shaft	12
Task Objective	12
Estimated Time to Complete	12
Creating the Shaft	13
Saving the Model	16
Creating the Bearing	17
Task Objective	17
Estimated Time to Complete	17
Creating the Bearing	18
Saving the Model	20
Creating the Gear	21
Task Objective	21
Estimated Time to Complete	21
Creating the Gear	22
Saving the Model	27
Creating the Joint and Force	28
Task Objective	28
Estimated Time to Complete	28
Creating the Expression	29
Creating the Joint	29
Creating the Force	31
Performing Dynamic/Kinematic Analysis	32
Analyzing the Simulation Result	33
Task Objective	33
Estimated Time to Complete	33
Analyzing the Shaft	34
Analyzing the Bearing	37

Applyzing the Coor	27
Analyzing the Gear	
Involute Analytic Contact	38
Task Objective	38
Estimated Time to Complete	38
Creating the Involute Analytic Contact	39
Performing Dynamic/Kinematic Analysis	40
Analyzing the Shaft	40
Analyzing the Bearing	41
Analyzing the Gear	41
Campbell Diagram	42
Task Objective	42
Estimated Time to Complete	42
Simulate the Campbell Diagram Model	43
Adjusting the Analysis Tab	43
Adjusting the Plot Tab	44
Adjusting the Campbell Diagram	45



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



1 hours and 45 minutes



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.



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.
- 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.

Start RecurDyn			×
New Model -			
Name	GearBox		
Unit	MMKS(Millimeter/Kilogram/Newton/Second)	Setting	
<u>G</u> ravity	-ү т	Setting	
		<u>O</u> K	
Open Model		Browse	
Recent Mode	Is	Icons	•
Show 'Start	RecurDyn' Dialog when starting		

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.

★ CAD Import Options		x
Assembly Hierarchy —		
Hierarchy Conversion Level	Body	○ Subsystem
CAD Hierarchy Dialog		
Import	Cancel	

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.

≷ Icon Control	×	
Icon On/Off All Icons Force Contact Sensor Initial Velocity Wall / Vessel All Markers General Marker Inertia Reference Fr	t	
Icon Size	25 *	
Marker Size	25	
Marker Z-Axis Width	2.	
Initial Velocity Icon Width	2.	
Show Center Marker Ico	n	

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.

Layer S	ettings			×
Select	Entity	OHigh	light Entity	
Show	User-De	fined Layer	Refresh	
Layer	On	Nan	ne	
1	V			
2		Hous	ing	
3				
4	~			
5	~			
6	~			
7	~			
8	~			
9				
10				
11	~			
				<u> </u>
Show A		Close		



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**.)





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.



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.



- Shaft
- 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	Туре
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

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

2. Shaft Section

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

3. FDR Section

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	Туре
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**.)





Creating the Bearing

Task Objective

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



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.

E	earing [Current Uni	t : N/kg/mm/s/deg]			
[General Bearing				
	Center Point	232.5, 250., 175.	Normal Direction	1., 0, 0	
	Bearing Library	Bearing Library		-	Library
	Bearing Type	Deep groove ball bearing (single row)			-

- 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

Bearing Librar	y			×
Bearing Type	Deep groove	ball bearing (single ro	ow)	•
Diameter	Inside	○ Outside	85.000	▼ mm
Bearing	Timken 6317 (d=85.000 mm, D=180	.000 mm, B=41.000 mm)	•
Internal Clearance	C2	▼ 0.		
		OK Can	ncel	

- 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**.)





Creating the Gear

Task Objective

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



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

- Gear Ge	eometry										
Normal	Module		5.	Gear	Gear Type				Helix Right Hand 🔻		
Pressure	e Angle at Nori	mal Section	20.		Helix	Ang	le at Referenc	e Circle	20.		
Gear	No. of Teeth	Face Width	Profile Shift Coefficient	Details	Profile		Tolerance	Modific	ation	Material	
1	73	120.	0.4405		Factors					18CrNiMo7-6, 💌	
2	31	120.	0.2477		Factors					18CrNiMo7-6, 🔻	

- 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.

[Gear 1] Tolerance			×
Tooth Tolerance Type		User Input	•
Tooth Thickness (Upper / Lower)	-	0.	0.
☑ Tip Diameter (Upper / Lower)		0.	0
Root Diameter (Upper / Lower)		0.	0
	Close		

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.

Pair	Base Gear	Action Gear	Center D	istance	Backla	ash	Axial Offset	Rot. Angle	Cont	Meta Model	Import	Export	KISSsoft UI	
1	1	2	280.	Calc.	1.368	Calc.	0.	0.						
						_								
Pair	Base Gear	Action Gear	Center D	istance	Backla	ash	Axial Offset	Rot. Angle	Cont	Meta Model	Import	Export	KISSsoft UI	
1	1	2	280.	Calc.	0.013	Calc.	0.	0.						

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.

Cylindrica	alGear [Curre	nt Unit : N/ko	J/mm/s/deg]										
General	Cylindrical Gea	ir											
Assem	bly Reference P	oint (Gear1)	-55., 250., 175.			G	ear Nor	mal Direction		1., 0, 0			
Assem	bly Reference D	irection	0, 0, -1.			Pt .	Use Kl	SSsoft Z12 M	odule Files		Z12 Modu	le Files	
Gear O	Geometry ——												
Norma	al Module		5			G	ear Type	e		Helix Right	Hand		•
Pressu	ire Angle at Nor	mal Section	20.			Н	elix Ang	gle at Referen	ce Circle	20			
Gear	No. of Teeth	Face Width	Profile Shift Co	efficient	Details	Profil	e	Tolerance	Modific	ation	Materia	1	
1	73	120	0.4405			Factors				1	8CrNiMo7-6	💌	
2	31	120	0.2477			Factors				1	8CrNiMo7-6	💌	
Gear P	kdd D	elete	Insert										
Pair	Base Gear	Action Gear	Center Distance	Backla	ash	Axial Offset	Rot. A	Angle Cont	Meta Mo	del Impor	t Export	KISSsoft UI	
1	1	2	280. Calc.	0.013	Calc.	0.	0						
Gear		elete	Insert	Each R	endering	1 Automa	tic		▼ Forre	Display	Inactivate		
Cont		active				Automa					(ок	Cancel

- 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.

Gear Force Type	KISSsoft Force (Meta Model)	-	
	Inactivate KISSsoft Force		Í
	KISSsoft Force (Meta Model)		

- 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



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



- 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



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

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

To create the revolute joint

Revolute

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.

- 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.

Motion		
Motion		
Туре	Standard Motion	
Vola site (t	Initial disp.	
Velocity (t	ime) • 0. PV	
- Expression Name	Ex_Vel EL	
Expression		
10*pi*tim	e 🔺	
	OK Cancel Apply	

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.

Joint	*	
Joint	N	
Point, Direction	1	
Body, Body, Point, Direction		

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 Popup 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.



Dyn/Kin

- 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)



Analyzing the Simulation Result

Task Objective

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



20 minutes

Analyzing the Shaft

Adjust the icon control

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

🗄 🖽 🙏 🖽 🗐 💼	D d	d 🗗	P	P 1	- 🕹	Ŵ	Ŷ	Ŷ	Ŧ
lcon Control	1								

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.

🗄 🎞 🙏 🗛 🖾 💋 💋 🗗 🖉	▛ॖ ਿ P P 1 - ▼ 🕹 💊 '	V	Ŷ	Ŧ
	Wireframe With Silhouettes	Ĺ		

≷ Icon Control		×
Icon On/Off All Icons All Icons Force Contact Sensor Parametric Point Initial Velocity All Markers Center Marker General Marker Inertia Reference Fr	t	
lcon Size	25	* *
Marker Size	25	* *
Marker Z-Axis Width	2.	
Initial Velocity Icon Width	2.	
Show Center Marker Ico	n	

Adjust the contour

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.

Contour Option —		Band Option		Vi	iew / I	Reference M	lode / Referen	nce Mark	er –
Animation Status	SMISES	Legend Type	Display 🔻		Sel	Body	Node ID	Sel	Or
Туре	Stress 💌	Location	Bottom	. [~	Shaft1	10001		
Component	SMISES 💌		Show Text Legend		-	Shaft2 Shaft3	10001		
Display Vector	1	Band Level(10~50)	50	ק ^י					
Uniform	Simple				•	ш			
Contact Surface	Only	Style Option		7					
User defin	ed contact surface	Color Option	Edit		ontou	ir Data Trac	e		_
© Oser denni	cu contact surface	Colors	Spectrum 🔻	-	Sel	Body	Node	ID	
Contact pa	itches only	Style	Smooth						
Min/Max Option —		Text Color	Text Color 🗸						
SIMISES		Exceed Max Color	Max Color 🔹						
Type Display		Less than Min Colo	Min Color 👻	illo		Add		Delete	
(Calculation				ontou	r Element 9	et Selection -		
					Sel	Body	Contour F	Part	
Max 13.37	13.3769								
Show Min/Max	Enable Log Scale								
User Defined Ma	ix Color								
Lines Define and Mis	n Color	Mesh Lines	Line Color 🔹			Add		Delete	
Oser Defined Mil									

To play an animation

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.

ection	All Sections				
Component	SMISES				
Reference Fr	ame				
Node	10001	N	OMarker		
		Update F	Plot Data		
Animation Fr	ame				
Synchron	ize with Animation Contro	ol			
				Time	Frame
· []				0.014	36
		— Ger	neral Sections		
13 60					
15.60		~			
10.20				, f	
SI 6.80					
8	\wedge				
5.40					
0.00					
4	146.00	292.00	438.0	0 584.0	00
0.0		Assist F	View la coment		

Analyzing the Bearing

Result

- 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.





Involute Analytic Contact

Task Objective

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



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.

Icon Control	×
I con On/Off	Point ity el ker rker ce Frame
lcon Size	25.
Marker Size	25.
Marker Z-Axis Width	2.
Initial Velocity Width	2.

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.

Involute

4. **Repeat** the **Step 1~3** for the **CylindricalGearGroup2**.

To create the 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 Popup 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.



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.





Scope

Control

Analyzing the Bearing

Result

- 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**.
- 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.







Campbell Diagram

Task Objective

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



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.

- ovn/Kin
- 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.

Input Data —								
Input File								
Time TIM	E	Plot Index	0					
Tacho Rec	uest/Expressions/ExRq1/F1(Ex_Tacho)	Plot Index	0					
Signal Rec	Plot Index	0						
Tachometer Type Tacho 💌 Pulse/rev 1.								
Interpolatio	Linear 🔻 dt	0.						
	Update Signal Information							

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.

Use Recommend Frame Settings						
Frame Number	190	R	Frame Length(sec)	5.08e-002		
Frame Size	128	-	Overlap(%)	81.1023622047244		
RPM Compute	Average	-	Residual Number	6		
Delta Time	1					
Advanced Setting		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.

1 □ - 1 0 + 1 0 F 1. 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
1 - 1 5 + 1 F
[™]

- 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.

Legend Location Left Style Style				Smooth 🔻
- Color Se Minimur	n Value 0.	7863.50064902726		
No	Start Value	End Value	Color	Add
1	0.	800.	С	la la catal
2	800.	1600.	С	Insert
3	1600.	2400.	С	Delete
4	2400.	3200.	С	
5	3200.	4000.	С	
6	4000.	4800.	С	
7	4800.	5600.	С	
8	5600.	6400.	С	
9	6400.	7200.	С	
10	7200.	8000.	С	Eq. Division
				Color Tool

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



Thanks for participating in this tutorial!

GEARBOX TUTORIAL (DRIVETRAIN)