Model created in COMSOL Multiphysics 6.4

Vibrations in a Compound Gear Train

Introduction

This model simulates the vibrations in a compound gear train. It is built using the gears functionality in the Multibody Dynamics interface in COMSOL Multiphysics.

The spur gears are used to model the gear train. The gears are mounted on rigid shafts, and the shafts are supported by the elastic housing on both the ends. The gear mesh is assumed elastic with varying stiffness, which is the source of sustained vibration. A transient analysis is performed to compute the dynamics of the gears as well as the vibrations in the housing.

The contact modeling is used for the computation of the gear mesh stiffness. A parametric analysis is performed to compute the gear mesh stiffness as a function of gear rotation in one mesh cycle.

Model Definition

The geometry of the compound gear train where shafts are mounted on an elastic housing is shown in Figure 1.

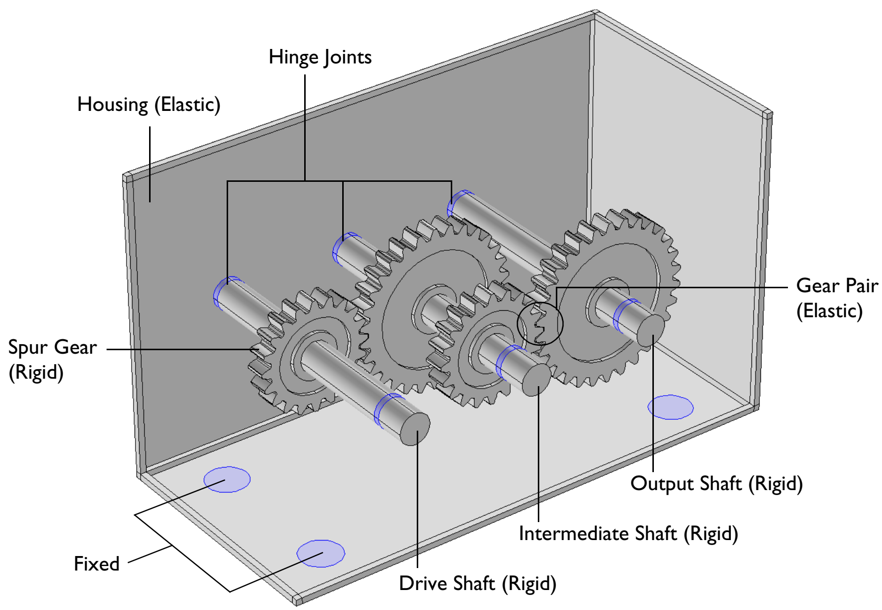

Figure 1: Model geometry of the compound gear train. The fixed and hinged locations are highlighted.

This model has two parts:

- In the first part of the model, the variation of gear mesh stiffness in a mesh cycle is computed using the contact analysis.
- In the second part of the model, the computed gear mesh stiffness is used in gear pair nodes to compute the dynamics of gears and the vibrations in the housing.

PART-I: COMPUTATION OF THE GEAR MESH STIFFNESS

In the compound gear train shown in Figure 1, there are two pairs of spur gears. Both of them are of the same size and made up of same material, hence the gear mesh stiffness can be computed for any single pair of gears. Moreover, to reduce the computation time, the cross-section of gears (shown in Figure 2), is used to compute the gear mesh stiffness.

In this model, both gears are hinged at their respective centers. A contact is modeled between the teeth of the gears.

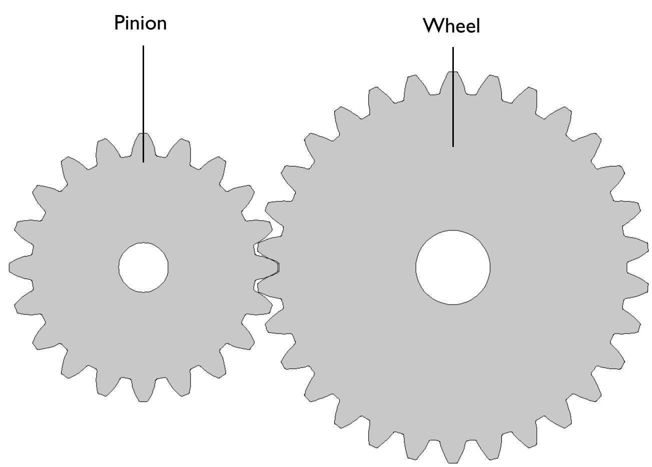

Figure 2: Model geometry for the gear mesh stiffness computation.

Gear Properties

The properties of the wheel and pinion are given in the table below:

TABLE 1: GEAR PROPERTIES

PROPERTIES		PINION	WHEEL
Number of teeth	n	20	30
Pitch diameter	d_p	50 [mm]	75 [mm]
Pressure angle	α	25 [deg]	25 [deg]
Gear width	w_g	10 [mm]	10 [mm]

Structural steel is used as a material for both gears.

Contact Condition

A penalty contact is modeled between the two gears. The boundaries of the gears, which are in contact with each other, are shown in Figure 3.

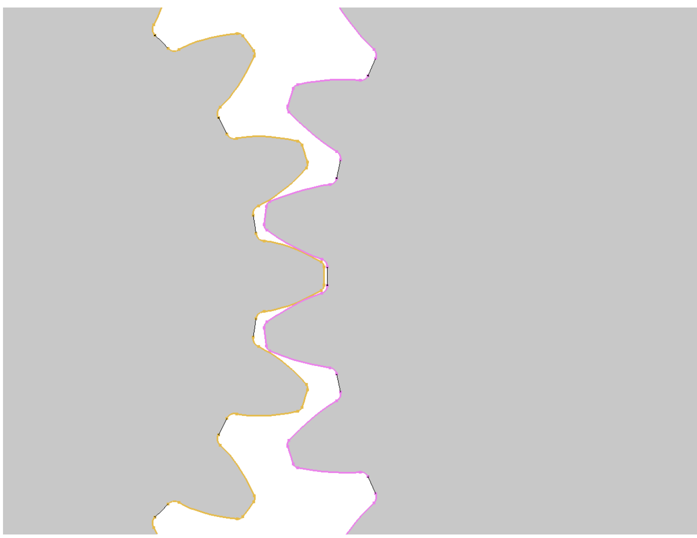

Figure 3: Contact pair boundaries.

Constraints

The rotation of the pinion (θ_p) about the out-of-plane axis is prescribed. It is incrementally increased in such a way that the pinion is rotated for two mesh cycles.

The rotation of the wheel (θ_w) about the out-of-plane axis is also prescribed. The wheel rotation is given by:

$$\theta_w = -\frac{\theta_p}{g_r} + \theta_t$$

where g_r is the gear ratio and has a value of 1.5, and θ_t is the twist and has a value of $0.5°$.

Gear Mesh Stiffness

The wheel is given a twist (θ_t) and the required twisting moment (T) is computed on the hinge joint.

The torsional stiffness of the gear pair can be defined as:

$$k_t = \frac{T}{\theta_t}$$

The stiffness along the line of action can be defined as:

$$k_g = \frac{4k_t}{(d_{pw} \cdot \cos\alpha)^2}$$

where d_{pw} is the pitch diameter of the wheel, and α is the pressure angle.

The gear mesh stiffness is different for different positions of gears in a mesh cycle. Hence, both the gears are rotated parametrically to compute the variation of gear mesh stiffness in a mesh cycle.

Mesh

The mesh used in both the gears while computing the gear mesh stiffness is shown in Figure 4.

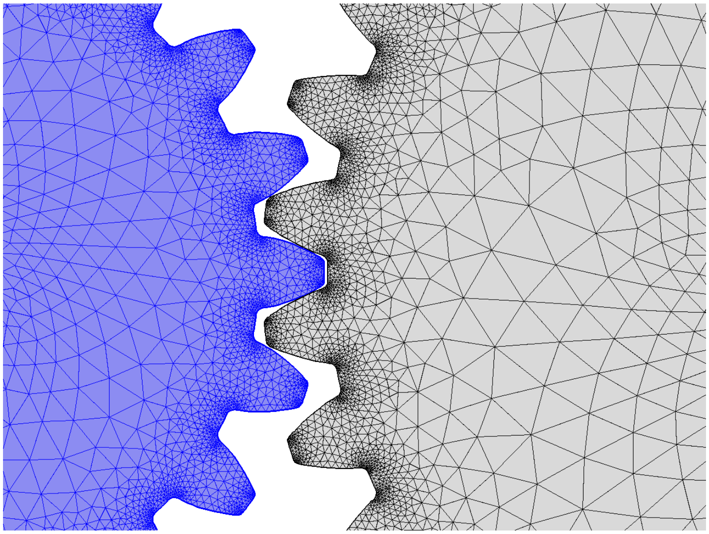

Figure 4: Mesh used in the gears.

PART-2: VIBRATIONS IN A COMPOUND GEAR TRAIN

The geometry of the compound gear train is shown in Figure 1. It has a drive shaft connected to an intermediate shaft. The intermediate shaft is connected to the output shaft. The spur gears are used to connect different shafts.

Both ends of the shafts are connected to the elastic housing through hinge joints. The housing is mounted to the ground at the locations shown in Figure 1.

Shafts and Gears

All the shafts and gears are assumed rigid, whereas the gear mesh is assumed elastic. The stiffness of the gear mesh is computed in the first part of the model. The properties of different gears are given in Table 1. Structural steel is used as a material for shafts and gears.

There are two spur gears mounted on the intermediate shaft. The first gear is considered as a part of the rigid shaft, and the second gear is connected to the shaft through a fixed joint.

Housing

The shafts are supported on a housing. The housing is assumed as a linear elastic material. Structural steel is used as a material for the housing.

Note: To reduce the computation time, linear elements are used in the discretization while modeling the housing. To get more accurate results, quadratic elements as well as more than one mesh elements in the thickness direction can be used.

Constraints and Loads

- The drive shaft rotates with an angular velocity of 600 rad/s.
- An external torque of 100 N-m is applied on the output shaft.

Results and Discussion

Figure 5 shows the von Mises stress distribution in the gear pair. At this position in mesh cycle, two pair of teeth are in contact with each other. It can be seen that the stresses are high at the contact points as well as at the roots of the teeth.

Figure 6 shows the variation of computed gear mesh stiffness. The gear mesh stiffness is plotted with the pinion rotation, and it is computed for two mesh cycles. It can be seen that the gear mesh stiffness is periodic, and its variation is the same in the second mesh cycle.

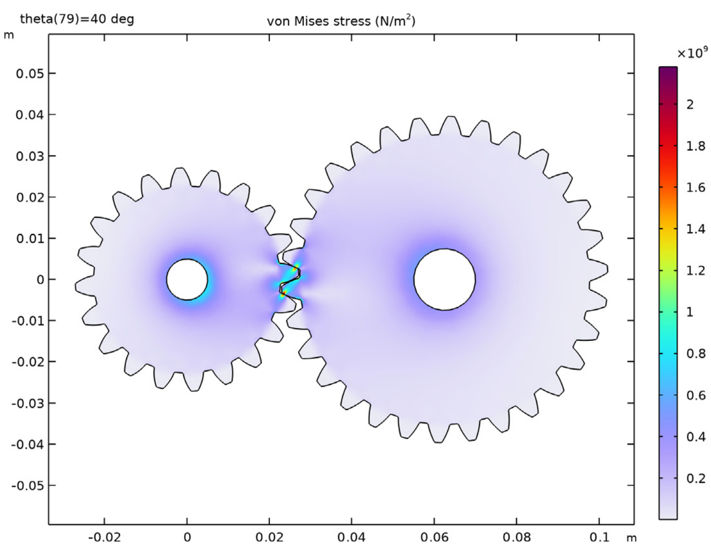

Figure 5: von Mises stress distribution in the gear pair at a particular position in mesh cycle. At this position the contact ratio is 2.

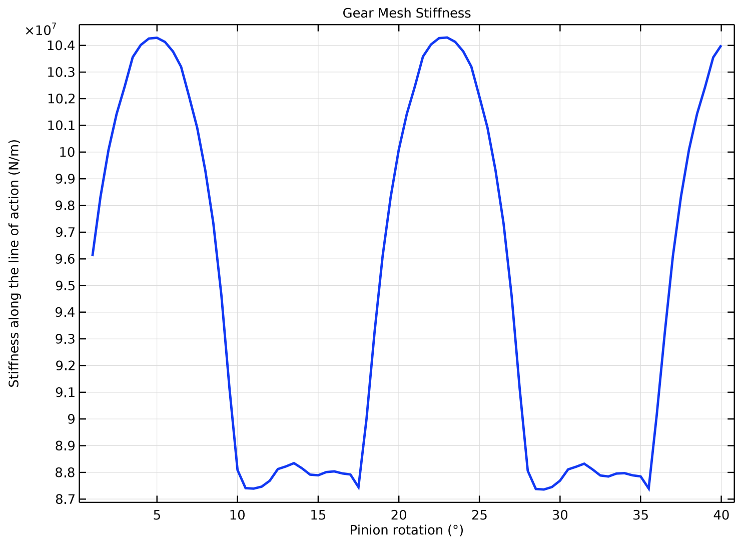

Figure 6: The variation of gear mesh stiffness with pinion rotation.

Within a mesh cycle, the gear mesh stiffness increases in the beginning, and later it decreases. This happens because of the change in contact ratio. In the beginning, the contact ratio changes from 1 to 2, and later it switches back from 2 to 1.

The displacement in the gears and the normal acceleration in the housing are shown in Figure 7. The variation of contact force in both gear pairs is shown in Figure 8. It can be seen that the contact force continuously changes as the gear rotates. The variation is periodic in a mesh cycle. It can also be seen that the mean value of contact force on the second gear pair is higher than that for the first gear pair. This is because the speed of the intermediate shaft is higher than that for the output shaft. The ratio of the mean values of contact forces is almost the same as the gear ratio.

Figure 9 and Figure 10 show the variation of angular velocity and acceleration of the drive shaft, intermediate shaft, and output shaft. The ratio of the mean values of angular velocity of different shafts is the same as the gear ratio. The drive shaft angular velocity is constant, as it is prescribed, however the angular velocity of the intermediate and output shaft keep oscillating about their respective mean values. The source of sustained oscillations is the varying stiffness of the gear mesh. It can also be noted that the oscillations in the speed of the output shaft are higher than that of the intermediate shaft.

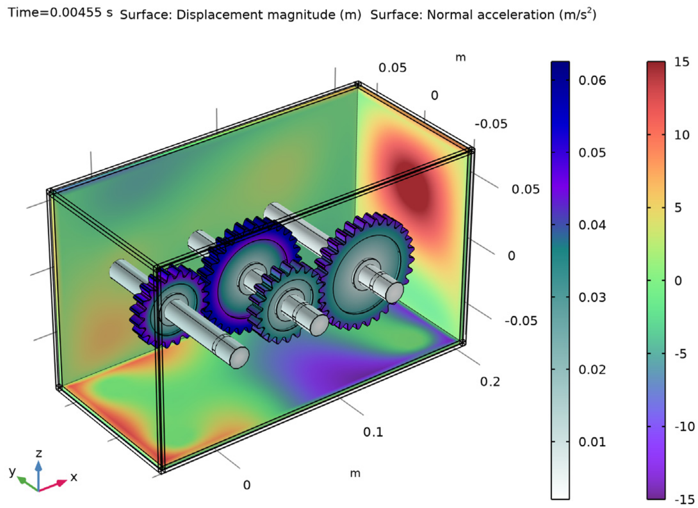

Figure 7: The displacement is gears and the normal acceleration in the housing at a particular instance.

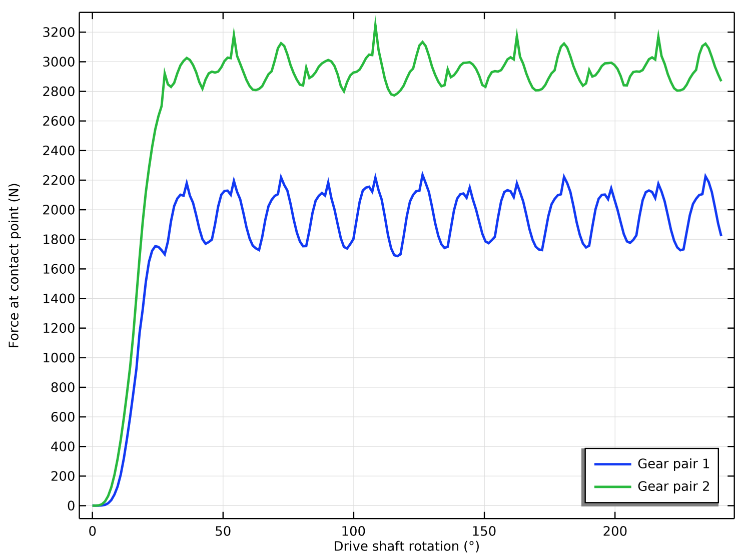

Figure 8: The variation of contact forces.

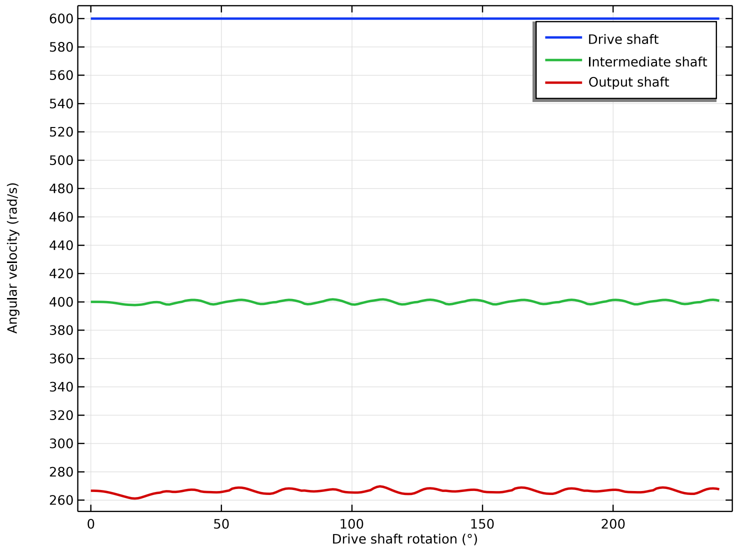

Figure 9: The variation of angular velocity of different shafts.

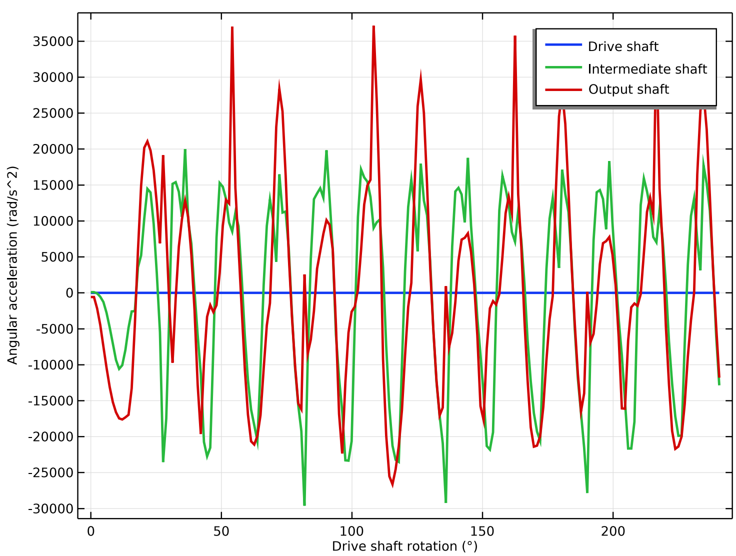

Figure 10: The variation of angular acceleration of different shafts.

Figure 11 shows the variation of normal acceleration at a location of attachment-5 in the housing. The corresponding frequency content is shown in Figure 12. In the frequency content, it can be seen that there are two dominant frequencies near 550 Hz and 2000 Hz at which the housing is vibrating. The normal acceleration result can be further used to compute the noise emitted by the housing.

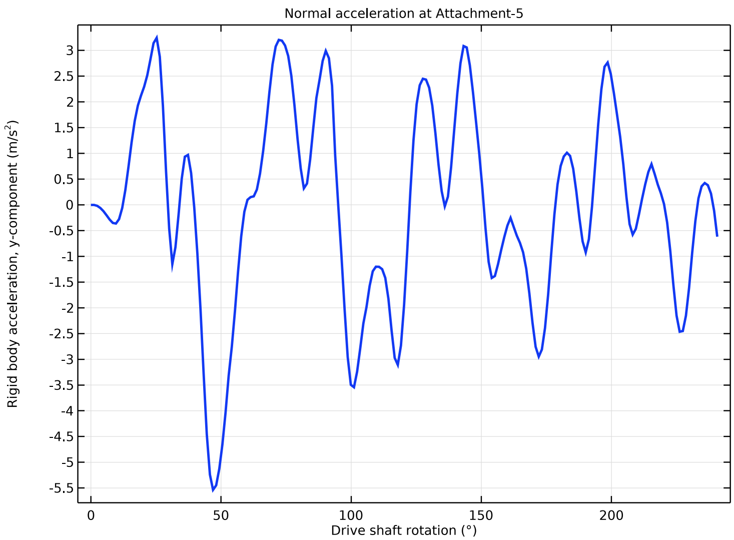

Figure 11: The variation of normal acceleration on a point in the housing.

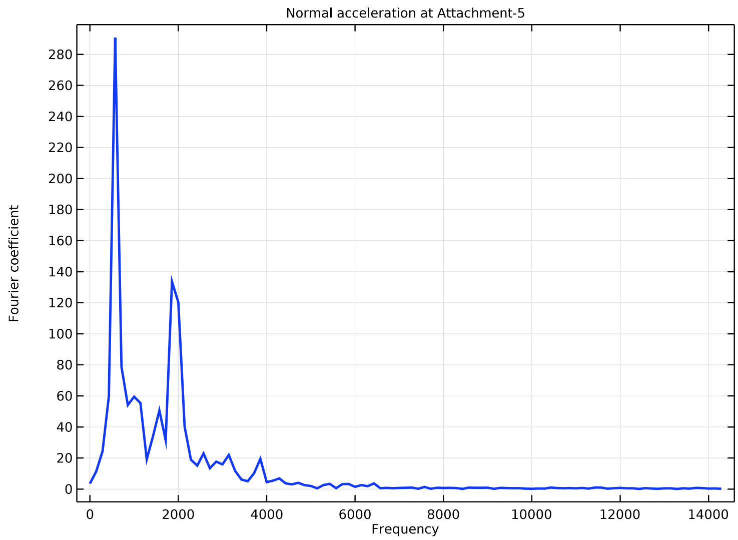

Figure 12: The frequency content in the normal acceleration of the housing.

Notes About the COMSOL Implementation

- To build a gear geometry, you can import a gear part from the **Parts Library** and customize it by changing its input parameters. Alternatively, you can also create an equivalent disc or cone to represent the gear.
- All the gears are assumed rigid. The elasticity of a gear mesh can be included on **Gear Pair** nodes using the **Gear Elasticity** subnode.
- All the **Gear Pair** nodes are assumed ideal and frictionless. You can add **Transmission Error**, **Backlash**, or **Friction** subnodes when required.
- To constrain the motion of a gear, you can use **Prescribed Displacement/Rotation** or **Fixed Constraint** subnodes. Alternatively, you can mount the gears on a shaft or on the ground through various **Joint** nodes.
- The contact force on a **Gear Pair** is computed using **Weak constraints** or **Penalty method**. By default, the contact force computation is turned off. Use the weak constraints method for more accurate contact forces. However, you preferably opt for the penalty method for large rigid body systems.

Application Library path: `Multibody_Dynamics_Module/Tutorials,_Transmission/gear_train`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics** > **Multibody Dynamics (mbd)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `gear_train_parameters.txt`.

PART LIBRARIES

1 In the **Geometry** toolbar, click **Part Libraries**.
2 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
3 In the **Part Libraries** window, select **Multibody Dynamics Module** > **2D** > **External Gears** > **spur_gear_2d** in the tree.
4 Click **Add to Geometry**.

GEOMETRY 1

To customize the gear geometry, enter the gear parameters in the input parameters of the part.

Spur Gear (2D) 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Spur Gear (2D) 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
n	n_pn	20	Number of teeth
dp	dp_pn	0.05 m	Pitch diameter
alpha	alpha	25 °	Pressure angle

4 Locate the **Selection Settings** section. Select the **Keep noncontributing selections** checkbox.

Spur Gear (2D) 2 (pi2)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Spur Gear (2D)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
n	n_wh	30	Number of teeth
dp	dp_wh	0.075 m	Pitch diameter
alpha	alpha	25 °	Pressure angle
xc	(dp_pn+dp_wh)/2	0.0625 m	Gear center, x-coordinate
th	360/(2*n_wh)[deg]	6 °	Mesh alignment angle

4 Locate the **Selection Settings** section. Select the **Keep noncontributing selections** checkbox.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** checkbox.

Define a contact pair between both the gears using the selection provided by the geometry parts.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Pairs** > **Contact Pair**.

3 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

4 From the **Selection** list, choose **Gear teeth, contact surface (Spur Gear (2D) 1)**.

5 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Gear teeth, contact surface (Spur Gear (2D) 2)**.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Structural steel**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MULTIBODY DYNAMICS (MBD)

1 In the **Settings** window for **Multibody Dynamics**, locate the **Thickness** section.

2 In the d text field, type `wg`.

Contact 1

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Click **Add**.

4 In the **Add** dialog, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

The resolution of the contact condition will be important for the gear stiffness computation. A penalty solution artificial flexibility. Use the Nitsche method to improve the accuracy of the contact condition.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the list, choose **Nitsche**.

Attachment 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Attachment**.

2 Select Boundaries 52, 53, 64, and 65 only.

Hinge Joint 1

1 In the **Physics** toolbar, click **Global** and choose **Hinge Joint**.

2 In the **Settings** window for **Hinge Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Fixed**.

4 From the **Destination** list, choose **Attachment 1**.

Rotate the first gear to compute the mesh stiffness for various positions in a mesh cycle.

Prescribed Motion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.

2 In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.

3 In the θ_p text field, type `theta`.

Attachment 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Attachment**.

2 Select Boundaries 246, 247, 257, and 262 only.

Hinge Joint 2

1 In the **Physics** toolbar, click **Global** and choose **Hinge Joint**.

2 In the **Settings** window for **Hinge Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Fixed**.

4 From the **Destination** list, choose **Attachment 2**.

Rotate the second gear using the first gear rotation and the gear ratio. Moreover, specify additional rotation to twist the gear pair and compute the stiffness.

Prescribed Motion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.

2 In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.

3 In the θ_p text field, type `-theta/gr+twist`.

4 Click to expand the **Reaction Force Settings** section. Select the **Evaluate reaction forces** checkbox.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
T	mbd.hgj2.pm1.RM	N·m	Required twisting moment
kt	T/twist	N·m/rad	Torsional stiffness
kg	kt/(dp_wh/2* cos(alpha))^2	N/m	Stiffness along the line of action

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta (Pinion rotation)	range(1,0.5,40)	deg

Reduce the number of nonlinear iterations before a cutback is made. Without this change, the solver may perform too many iterations when in fact a cutback is better for convergence.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 In the **Maximum number of iterations** text field, type `8`.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Use the following instructions to plot the von Mises stress and the gear mesh stiffness as shown in Figure 5 and Figure 6 respectively.

Stress

1 Right-click **Displacement (mbd)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Stress` in the **Label** text field.

Surface

1 In the **Model Builder** window, expand the **Displacement (mbd) 1** node, then click **Results** > **Stress** > **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `mbd.mises`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

5 In the **Stress** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **kg - Stiffness along the line of action - N/m**.

3 Click **Evaluate**.

Gear Mesh Stiffness

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Gear Mesh Stiffness` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 Right-click **Gear Mesh Stiffness** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **kg - Stiffness along the line of action - N/m**.

3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `theta`.

5 From the **Unit** list, choose **°**.

6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

8 In the **Gear Mesh Stiffness** toolbar, click **Plot**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Stress**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `40`.

After the computation of the gear mesh stiffness, the next step is to use it on **Gear Pair** nodes to model the vibration in a gear train assembly.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **3D**.

GEOMETRY 2

Import 1 (imp1)

1 In the **Geometry** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Source** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `gear_train.mphbin`.

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Geometry 2** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** checkbox.

5 In the **Geometry** toolbar, click **Build All**.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics** > **Multibody Dynamics (mbd)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.

5 Click the **Add to Component 2** button in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

MULTIBODY DYNAMICS 2 (MBD2)

Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Directly import the gear mesh stiffness in an **Interpolation Function** from a **Result table** evaluated in the previous analysis.

DEFINITIONS (COMP2)

Interpolation 1 (int1)

1 In the **Model Builder** window, expand the **Component 2 (comp2)** > **Definitions** node.

2 Right-click **Component 2 (comp2)** > **Definitions** and choose **Functions** > **Interpolation**.

3 In the **Settings** window for **Interpolation**, locate the **Definition** section.

4 From the **Data source** list, choose **Result table**.

5 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 2 and column number 1.

6 In the **Name** text field, type `gearStiffness`.

Step 1 (step1)

1 In the **Definitions** toolbar, click f(x) **More Functions** and choose **Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `0.5[ms]`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-3`.

Explicit 1

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 71, 72, 75, and 76 only.

5 Select the **Group by continuous tangent** checkbox.

Explicit Selections

Similarly add more **Explicit** selections using the information given in the following table:

Name	Selection
Explicit 2	`69, 70, 73, 74`
Explicit 3	`79, 80, 82, 84`
Explicit 4	`77, 78, 81, 83`
Explicit 5	`87, 88, 90, 92`
Explicit 6	`85, 86, 89, 91`

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Structural steel**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MULTIBODY DYNAMICS 2 (MBD2)

Add a **Spur Gear** node and specify it's properties.

Spur Gear 1

1 In the **Physics** toolbar, click **Domains** and choose **Spur Gear**.

2 Select Domains 27 and 28 only.

3 In the **Settings** window for **Spur Gear**, locate the **Gear Properties** section.

4 In the n text field, type `n_pn`.

5 In the d_{p} text field, type `dp_pn`.

6 In the α text field, type `alpha`.

7 Locate the **Gear Axis** section. Specify the $\mathbf{e}_g$ vector as

0	x
1	y
0	z

8 Locate the **Initial Values** section. From the list, choose **Locally defined**.

Initial Values 1

1 In the **Model Builder** window, expand the **Spur Gear 1** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values: Rotational** section.

3 Specify the ω vector as

omega	y
0	x
0	z

Spur Gears

Duplicate the **Spur Gear 1** to create more gears using the information given in the following table:

Name	Selection (Domains)	Number of Teeth	Pitch Diameter	Initial Angular Velocity
Spur Gear 2	29, 30	n_wh	dp_wh	-omega/gr
Spur Gear 3	31	n_pn	dp_pn	-omega/gr
Spur Gear 4	32, 33	n_wh	dp_wh	omega/gr^2

Now add **Gear Pair** nodes to connect two spur gears.

Gear Pair 1

1 In the **Physics** toolbar, click **Global** and choose **Gear Pair**.

2 In the **Settings** window for **Gear Pair**, locate the **Gear Selection** section.

3 From the **Wheel** list, choose **Spur Gear 1**.

4 From the **Pinion** list, choose **Spur Gear 2**.

5 Locate the **Gear Pair Properties** section. Select the **Include gear elasticity** checkbox.

6 Locate the **Contact Force Computation** section. From the list, choose **Computed using weak constraints**.

Gear Elasticity 1

1 In the **Model Builder** window, expand the **Gear Pair 1** node, then click **Gear Elasticity 1**.
2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.
3 From the **Specify** list, choose **Total stiffness of gear pair**.
4 In the k_g text field, type `gearStiffness(mbd2.grp1.thm_wh*180/pi)`.

Gear Pair 2

1 In the **Physics** toolbar, click **Global** and choose **Gear Pair**.
2 In the **Settings** window for **Gear Pair**, locate the **Gear Selection** section.
3 From the **Wheel** list, choose **Spur Gear 3**.
4 From the **Pinion** list, choose **Spur Gear 4**.
5 Locate the **Line of Action** section. From the **Obtained by rotation of tangent in** list, choose **Counterclockwise direction**.
6 Locate the **Gear Pair Properties** section. Select the **Include gear elasticity** checkbox.
7 Locate the **Contact Force Computation** section. From the list, choose **Computed using weak constraints**.

Gear Elasticity 1

1 In the **Model Builder** window, expand the **Gear Pair 2** node, then click **Gear Elasticity 1**.
2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.
3 From the **Specify** list, choose **Total stiffness of gear pair**.
4 In the k_g text field, type `gearStiffness(mbd2.grp2.thm_wh*180/pi)`.

Fixed Joint 1

1 In the **Physics** toolbar, click **Global** and choose **Fixed Joint**.
2 In the **Settings** window for **Fixed Joint**, locate the **Attachment Selection** section.
3 From the **Source** list, choose **Spur Gear 2**.
4 From the **Destination** list, choose **Spur Gear 3**.

Attachment 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Attachment**.
2 In the **Settings** window for **Attachment**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Explicit 1**.

Attachments

Similarly add 5 more **Attachments** using the information given in the following table:

Name	Selection
Attachment 2	Explicit 2
Attachment 3	Explicit 3
Attachment 4	Explicit 4
Attachment 5	Explicit 5
Attachment 6	Explicit 6

Hinge Joint 1

1 In the **Physics** toolbar, click **Global** and choose **Hinge Joint**.

2 In the **Settings** window for **Hinge Joint**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Attachment 1**.

4 From the **Destination** list, choose **Spur Gear 1**.

5 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

Hinge Joints

1 Duplicate the **Hinge Joint 1** to create more joints using the information given below:

Name	Source	Destination
Hinge joint 2	Attachment 2	Spur Gear 1
Hinge joint 3	Attachment 3	Spur Gear 2
Hinge joint 4	Attachment 4	Spur Gear 2
Hinge joint 5	Attachment 5	Spur Gear 4
Hinge joint 6	Attachment 6	Spur Gear 4

Prescribe the rotation of the drive shaft and add external load on the output shaft.

2 In the **Model Builder** window, click **Hinge Joint 1**.

Prescribed Motion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.

2 In the **Settings** window for **Prescribed Motion**, locate the **Prescribed Rotational Motion** section.

3 From the **Prescribed motion through** list, choose **Angular velocity**.

4 In the ω_p text field, type `omega`.

Hinge Joint 5

In the **Model Builder** window, under **Component 2 (comp2)** > **Multibody Dynamics 2 (mbd2)** click **Hinge Joint 5**.

Applied Force and Moment 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force and Moment**.

2 In the **Settings** window for **Applied Force and Moment**, locate the **Applied On** section.

3 From the list, choose **Joint**.

4 Locate the **Applied Force and Moment** section. In the M text field, type `-T_ext*step1(t)`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `67-68, 93-94` in the **Selection** text field.

5 Click **OK**.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.

5 Click **Build All**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Multibody Dynamics (mbd)**.

5 Click the **Add Study** button in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,3.5e-5,7e-3)`.

3 From the **Tolerance** list, choose **User controlled**.

4 In the **Relative tolerance** text field, type `1e-6`.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Study 2/Solution 2 (3) (sol2)

Create more **Datasets** for better visualization of the gear train assembly.

Study 2/Solution 2 (4) (sol2)

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** > **Study 2/Solution 2 (3) (sol2)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 27–33 only.

Study 2/Solution 2 (5) (sol2)

Right-click **Study 2/Solution 2 (3) (sol2)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 13, 16, and 22 only.

Use the following instructions to plot the displacement and the normal acceleration as shown in Figure 7.

Displacement (mbd2)

1 In the **Model Builder** window, under **Results** click **Displacement (mbd2)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **0.00455**.

Surface

1 In the **Model Builder** window, expand the **Displacement (mbd2)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (4) (sol2)**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralis**.

Surface 2

1 Right-click **Surface** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (5) (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `mbd2.an`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **SpectrumLight**.

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

7 In the **Minimum** text field, type `-15`.

8 In the **Maximum** text field, type `15`.

Displacement-Normal Acceleration

1 In the **Model Builder** window, under **Results** click **Displacement (mbd2)**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement-Normal Acceleration` in the **Label** text field.

3 In the **Displacement-Normal Acceleration** toolbar, click **Plot**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Use the following instructions to plot the time variation of contact forces on both the gear pairs as shown in Figure 8.

Contact force

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Contact force` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 Right-click **Contact force** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Multibody Dynamics 2** > **Gear pairs** > **Gear Pair 1** > **mbd2.grp1.Fc - Force at contact point - N**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Multibody Dynamics 2** > **Gear pairs** > **Gear Pair 2** > **mbd2.grp2.Fc - Force at contact point - N**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mbd2.grp1.Fc`	N	`Force at contact point`
`-mbd2.grp2.Fc`	N	`Force at contact point`

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Multibody Dynamics 2** > **Hinge joints** > **Hinge Joint 1** > **mbd2.hgj1.th - Relative rotation - rad**.

7 Locate the **x-Axis Data** section. From the **Unit** list, choose **°**.

8 Select the **Description** checkbox. In the associated text field, type `Drive shaft rotation`.

9 In the **Contact force** toolbar, click **Plot**.

10 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

11 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`Gear pair 1`
`Gear pair 2`

Follow the instructions to plot the angular velocity and acceleration of different shafts as shown in Figure 9 and Figure 10 respectively.

Contact force

1. In the **Model Builder** window, click **Contact force**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Lower right**.
4. In the **Contact force** toolbar, click **Plot**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

Angular Velocity

1. Right-click **Contact force** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Angular Velocity` in the **Label** text field.
3. Locate the **Plot Settings** section.
4. Select the **y-axis label** checkbox. In the associated text field, type `Angular velocity (rad/s)`.
5. Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Global 1

1. In the **Model Builder** window, expand the **Contact force 1** node, then click **Results** > **Angular Velocity** > **Global 1**.
2. In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Multibody Dynamics 2** > **Hinge joints** > **Hinge Joint 1** > **mbd2.hgj1.th_t - Relative angular velocity - rad/s**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
-mbd2.hgj3.th_t	rad/s	
mbd2.hgj5.th_t	rad/s	Relative angular velocity

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Drive shaft
Intermediate shaft
Output shaft

Angular Velocity

1 In the **Model Builder** window, click **Angular Velocity**.

2 In the **Angular Velocity** toolbar, click **Plot**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

Angular Acceleration

1 Right-click **Angular Velocity** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type Angular Acceleration in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type Angular acceleration (rad/s^2).

Global 1

1 In the **Model Builder** window, expand the **Angular Velocity 1** node, then click **Results** > **Angular Acceleration** > **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
mbd2.hgj1.th_tt	rad/s^2	Relative angular acceleration
-mbd2.hgj3.th_tt	rad/s^2	
mbd2.hgj5.th_tt	rad/s^2	Relative angular acceleration

4 In the **Angular Acceleration** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Use the following instructions to plot the time variation of normal acceleration and its frequency spectrum as shown in Figure 11 and Figure 12 respectively.

Normal Acceleration

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Normal Acceleration` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Normal acceleration at Attachment-5`.

Global 1

1 Right-click **Normal Acceleration** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
mbd2.att5.u_tty	m/s^2	Rigid body acceleration, y-component

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `mbd2.hgj1.th`.

6 From the **Unit** list, choose **°**.

7 Select the **Description** checkbox. In the associated text field, type `Drive shaft rotation`.

8 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

9 Locate the **Legends** section. Clear the **Show legends** checkbox.

10 In the **Normal Acceleration** toolbar, click **Plot**.

11 Click the **Zoom Extents** button in the **Graphics** toolbar.

Normal Acceleration: Frequency

1 In the **Model Builder** window, right-click **Normal Acceleration** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Normal Acceleration: Frequency` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Normal Acceleration: Frequency** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Discrete Fourier transform**.

4 From the **Show** list, choose **Frequency spectrum**.

5 In the **Normal Acceleration: Frequency** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Animation 2

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Displacement-Normal Acceleration**.

4 Locate the **Frames** section. In the **Number of frames** text field, type 50.