Created in COMSOL Multiphysics 6.1

Dynamics of Helical Gears

Introduction

This model illustrates the dynamics of helical gears. It is built using the gears functionality in the Multibody Dynamics interface in COMSOL Multiphysics.

A transient study is performed to analyze the effect of constant gear mesh stiffness, varying gear mesh stiffness, and the transmission error on the angular velocity of the driven gear and the contact force. An eigenfrequency analysis is performed to compute the natural frequencies and mode shapes of the gear pair for rigid and for elastic gear mesh.

Model Definition

The geometry of helical gears shown in Figure 1.

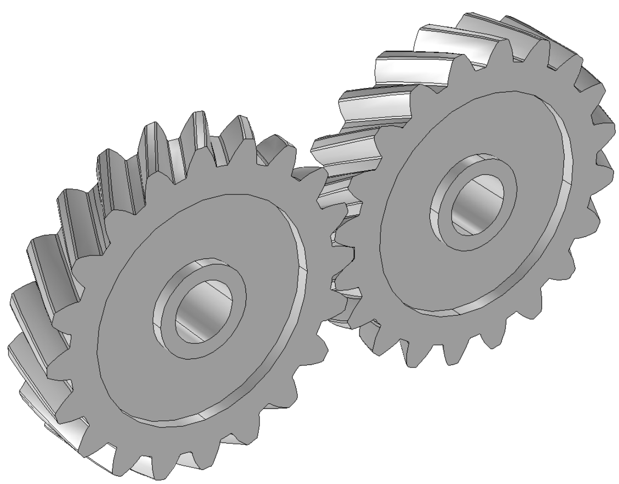

Figure 1: Model geometry.

GEAR PROPERTIES

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

TABLE 1: GEAR PROPERTIES.

PROPERTIES		WHEEL	PINION
Number of teeth	n	20	20
Pitch diameter	d_p	100 mm	100 mm
Pressure angle	α	25°	25°

TABLE 1: GEAR PROPERTIES.

PROPERTIES		WHEEL	PINION
Helix angle	β	30°	-30°
Center of rotation	$\mathbf{x}_c$	(0, 0, 0) mm	(100, 0, 0) mm
Axis of rotation	$\mathbf{e}_g$	(0, 1, 0)	(0, 1, 0)

TIME-DEPENDENT ANALYSIS

The time-dependent analysis is performed to analyze the dynamics of helical gears. The following gear meshes are considered while computing the contact force and the speed of the driven gear:

- Rigid gear mesh
- Elastic gear mesh with constant stiffness
- Elastic gear mesh with varying stiffness
- Elastic gear mesh with constant stiffness and transmission error

In all the cases, the driver gear rotates with an angular velocity of 100 rad/s, and a resisting torque of 10 Nm is applied to the driven gear. The analysis is performed for 3 mesh cycles, and the number of time steps per mesh cycle is 50.

Case-1: Rigid gear mesh

In this case, the gear mesh is assumed rigid, and there is no flexibility in the system.

Case-2: Elastic gear mesh with constant stiffness

In this case, the gear mesh is assumed elastic. The stiffness of a gear tooth is 10^7 N/m. The contact ratio in a mesh cycle is assumed constant. The gear mesh damping is also added, and it is 0.05 % of the mesh stiffness.

Case-3: Elastic gear mesh with varying stiffness

This case is similar to the case-1. However, in this case the contact ratio is varying in a mesh cycle. The maximum contact ratio in a mesh cycle is 2, and the next tooth engagement position in mesh cycle is 0.8. The variation of mesh stiffness of the gear pair in a mesh cycle is shown in Figure 2.

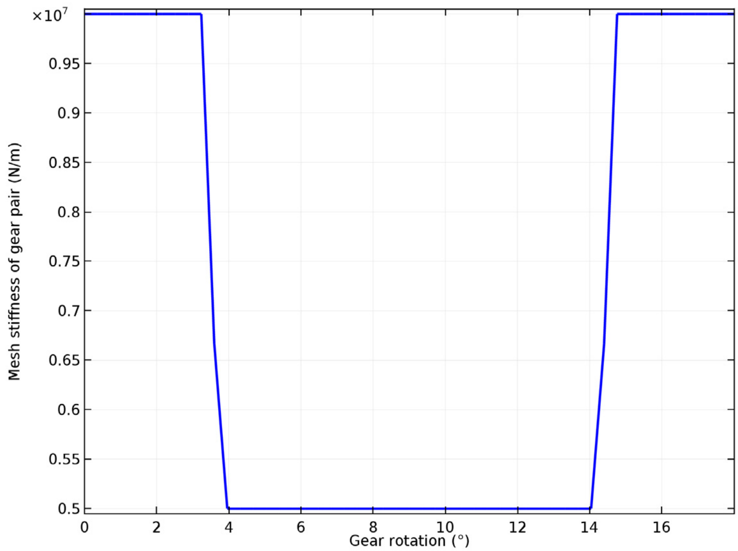

Figure 2: The variation of mesh stiffness in a mesh cycle (case-3).

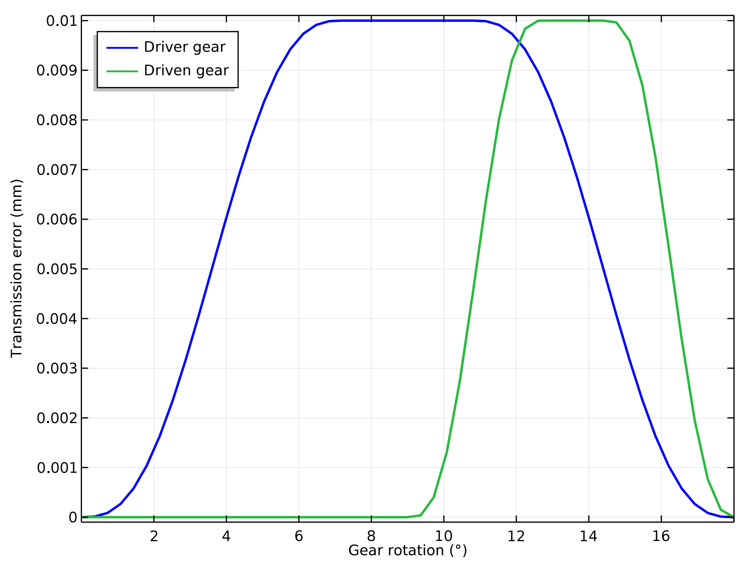

Figure 3: The variation of transmission error in a mesh cycle (case-4).

Case-4: Elastic gear mesh with constant stiffness and transmission error

This case is similar to the case-1. However, in this case the static transmission error is also added on both gears. The maximum static transmission error on each gear is 0.01 mm. The variation of transmission error on each gear in a mesh cycle is shown in Figure 3.

EIGENFREQUENCY ANALYSIS

An eigenfrequency analysis is performed to compute the natural frequencies and mode shapes of the helical gear pair. The following gear meshes are considered:

- Rigid gear mesh
- Elastic gear mesh with constant stiffness

In all cases, both gears are free to rotate about their axis. The driven gear is mounted on an elastic joint. The translational and rotational stiffness of the elastic joint is $2 \cdot 10^7$ N/m and $2 \cdot 10^7$ Nm/rad, respectively.

Results and Discussion

TIME DEPENDENT ANALYSIS

The angular velocity of the pinion or driven gear for various gear meshes is shown in Figure 4. It can be seen that for the rigid gear mesh, the angular velocity of the driven gear is constant. For constant stiffness case, the angular velocity of the driven gear fluctuates in the beginning, and it settles down to the mean value after a while.

For varying stiffness or transmission error cases, the angular velocity of the driven gear keeps changing with the gear rotation. The variation of angular velocity is periodic, and it is repeated in the next mesh cycle.

Figure 5 shows the variation of contact force for various gear meshes. It can be seen that for the rigid gear mesh, the contact force has a constant value of 254.8 N. For constant stiffness case, the contact force fluctuates in the beginning, and it settles down to the mean value after a while.

For the varying stiffness case, the contact force keeps changing with the gear rotation. The variation of contact force is periodic, and it is repeated in the next mesh cycle. The maximum and minimum values of the contact force in a mesh cycle are approximately 440 N and 150 N, respectively. For the transmission error case, the behavior of the contact force is similar to that in the varying stiffness case.

Figure 6 and Figure 7 show the variation of the reaction forces and reaction moments at the center of the driver gear for the gear mesh having varying stiffness.

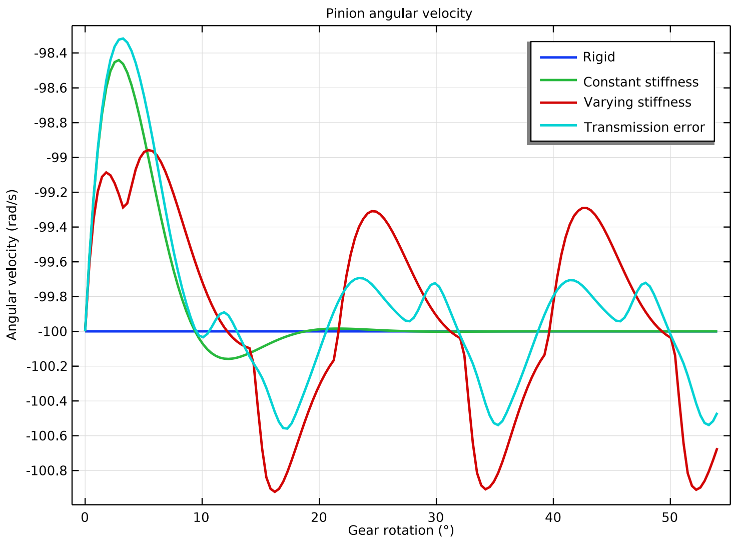

Figure 4: The variation of pinion angular velocity with gear rotation.

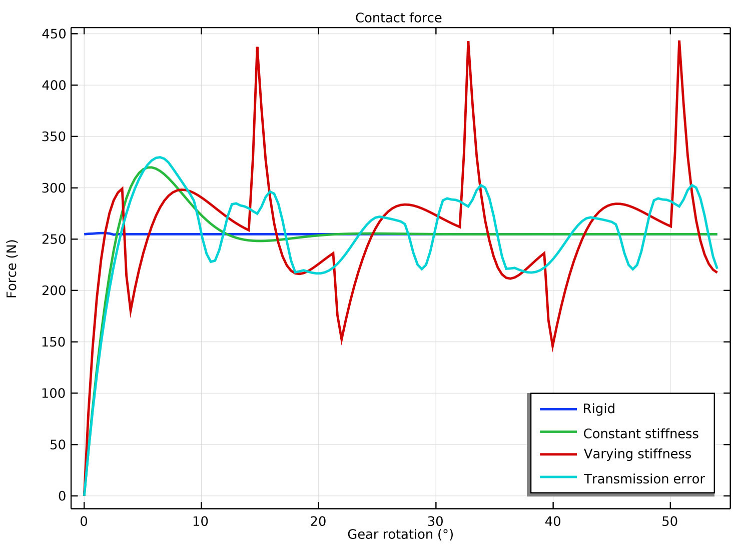

Figure 5: The variation of contact force with gear rotation.

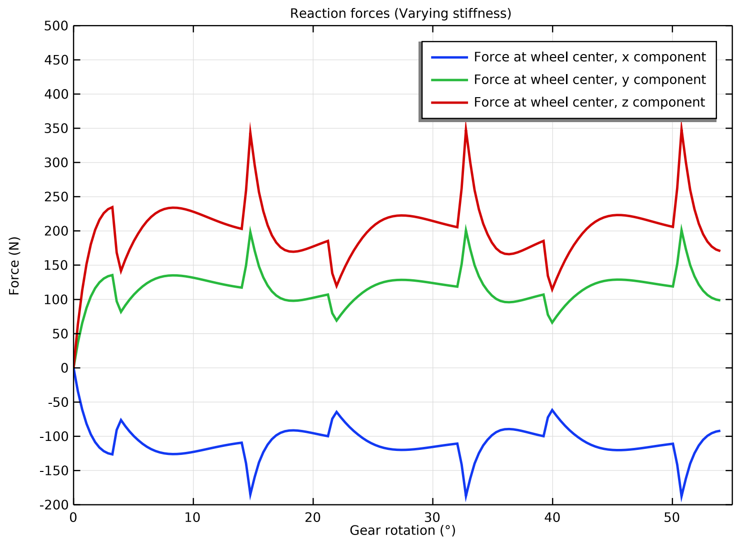

Figure 6: Reaction forces at the center of driver gear (case-3).

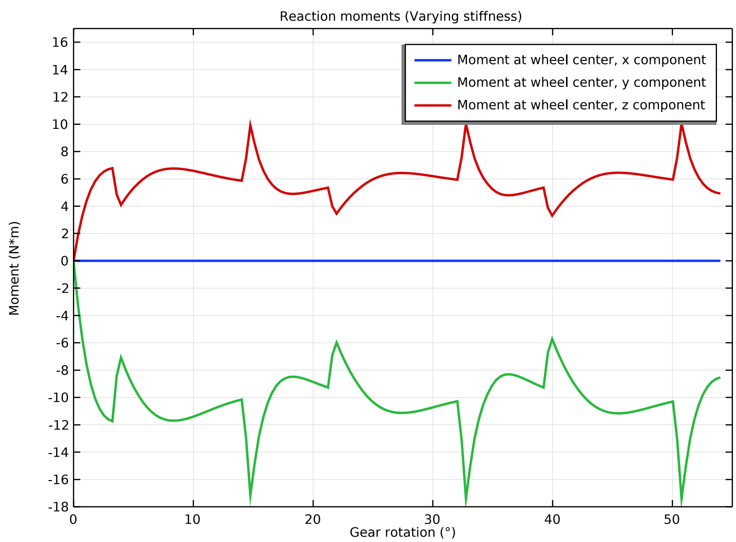

Figure 7: Reaction moments at the center of driver gear (case-3).

EIGENFREQUENCY ANALYSIS

The results of the eigenfrequency analysis, for both the gear meshes, are given below:

TABLE 2: NATURAL FREQUENCIES AND MODE SHAPES

MODE NUMBER	NATURAL FREQUENCY (RIGID MESH) (HZ)	NATURAL FREQUENCY (ELASTIC MESH) (HZ)	MODE TYPE
First	0*	0*	Rigid body rotation
Second	-	293.7+583.2i	Gear mesh twist
Third	601.3	590.3+71.7i	Elastic joint: rotation
Fourth	728.1	728.1	Elastic joint: translation
Fifth	728.1	728.1	Elastic joint: translation
Sixth	2417	2669+143.5i	Elastic joint: rotation

* Ideally the frequency of rigid body rotation should be zero. However, due to small numerical errors, it is not exactly zero but very close to zero.

It can be seen in Table 2 that the second mode is present only when the gear mesh is elastic. This mode corresponds to the twisting of the gear mesh. The rest of the modes that correspond to the displacement and rotation of the elastic joint are present for both gear meshes. It can be seen that the elasticity of the gear mesh affects only the rotational modes. The translational modes are the same in both the cases.

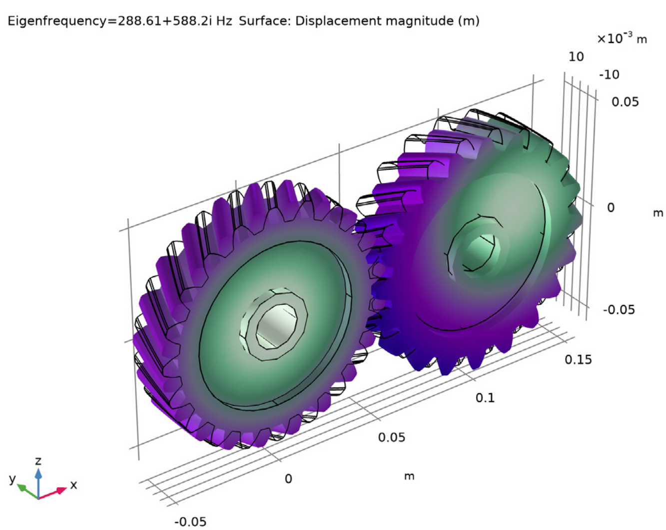

Figure 8: Second mode of the helical gear pair with an elastic mesh.

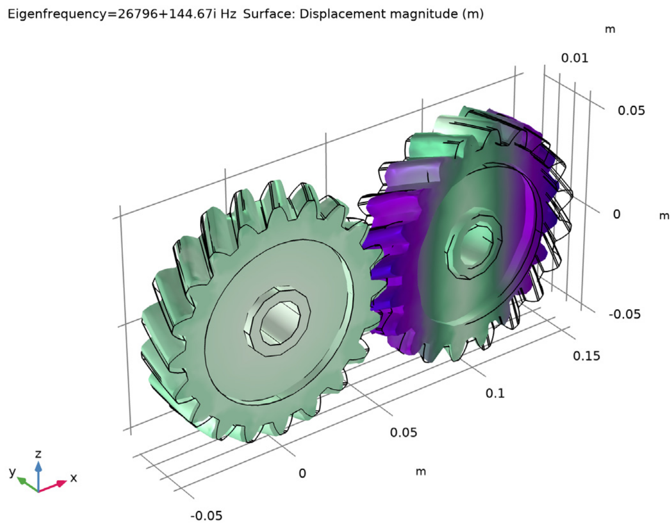

Figure 9: Sixth mode of the helical gear pair with an elastic mesh.

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 in the **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 constraint 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/helical_gear_pair`

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 **3D**.

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>Time Dependent**.

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 `helical_gear_pair_parameters.txt`.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Multibody Dynamics Module>3D>External Gears>helical_gear** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

Helical Gear 1 (pi1)

1 In the **Home** toolbar, click **Build All**.

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

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

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

4 In the table, enter the following settings:

Name	Expression	Value	Description
n	`n`	20	Number of teeth
dp	`dp`	0.1 m	Pitch diameter
alpha	`alpha`	25 °	Pressure angle
beta	`beta`	30 °	Helix angle
lsr	`0`	0	Shaft length to pitch diameter ratio (Set 0 for no shaft)
egy	`1`	1	Gear axis, y component
egz	`0`	0	Gear axis, z component

Helical Gear 2 (pi2)

1 In the **Geometry** toolbar, click **Parts** and choose **Helical Gear**.

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`	20	Number of teeth
dp	`dp`	0.1 m	Pitch diameter
alpha	`alpha`	25 °	Pressure angle
beta	`-beta`	-30 °	Helix angle
lsr	`0`	0	Shaft length to pitch diameter ratio (Set 0 for no shaft)
xc	`dp`	0.1 m	Gear center, x coordinate
egy	`1`	1	Gear axis, y component

Name	Expression	Value	Description
egz	0	0	Gear axis, z component
th	360/(2*n)[deg]	9 °	Mesh alignment angle

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** check box.

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

Define **Rectangle** functions to use them in the expressions of the transmission error.

DEFINITIONS

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Rectangle**.

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

3 In the **Lower limit** text field, type `0.2`.

4 In the **Upper limit** text field, type `0.8`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.4`.

Rectangle 2 (rect2)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Rectangle**.

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

3 In the **Lower limit** text field, type `0.6`.

4 In the **Upper limit** text field, type `0.9`.

5 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `0.2`.

ADD MATERIAL

1 In the **Home** 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 **Add to Component** in the window toolbar.

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

MULTIBODY DYNAMICS (MBD)

Add **Helical Gear** nodes and specify the gear properties. For the automated creation of **Helical Gear** nodes from geometry parts, use **Automated Model Setup** section of Multibody Dynamics node.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.

2 In the **Settings** window for **Multibody Dynamics**, click **Physics Node Generation** in the upper-right corner of the **Automated Model Setup** section. From the menu, choose **Create Gears**.

Helical Gear 1

1 In the **Model Builder** window, expand the **Gears** node, then click **Helical Gear 1**.

2 In the **Settings** window for **Helical Gear**, locate the **Initial Values** section.

3 From the list, choose **Locally defined**.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 Specify the ω vector as

0	x
omega	y
0	z

Helical Gear 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Multibody Dynamics (mbd)>Gears>Helical Gear 2** node, then click **Helical Gear 2**.

2 In the **Settings** window for **Helical Gear**, locate the **Center of Rotation** section.

3 Specify the $\mathbf{X}_c$ vector as

dp	x
0	y
0	z

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

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 Specify the ω vector as

0	x
-omega	y
0	z

Add **Hinge Joints** to connect the two gears with the ground.

Hinge Joint: Fixed-Gear 1

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

2 In the **Settings** window for **Hinge Joint**, type `Hinge Joint: Fixed-Gear 1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Fixed**.

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

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

0	x
1	y
0	z

Hinge Joint: Fixed-Gear 2

1 Right-click **Hinge Joint: Fixed-Gear 1** and choose **Duplicate**.

2 In the **Settings** window for **Hinge Joint**, type `Hinge Joint: Fixed-Gear 2` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Destination** list, choose **Helical Gear 2**.

Hinge Joint: Fixed-Gear 2 (Elastic)

1 Right-click **Hinge Joint: Fixed-Gear 2** and choose **Duplicate**.

2 In the **Settings** window for **Hinge Joint**, type `Hinge Joint: Fixed-Gear 2 (Elastic)` in the **Label** text field.

3 Locate the **Joint Elasticity** section. From the list, choose **Elastic joint**.

Joint Elasticity 1

1 In the **Model Builder** window, click **Joint Elasticity 1**.

2 In the **Settings** window for **Joint Elasticity**, locate the **Spring** section.

3 In the $\mathbf{k}_u$ text field, type `ku`.

4 In the $\mathbf{k}_\theta$ text field, type `kth`.

Hinge Joint: Fixed-Gear 1

Prescribe the motion of the wheel.

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint: Fixed-Gear 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: Fixed-Gear 2

In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Hinge Joint: Fixed-Gear 2**.

Applied Force and Moment 1

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

Apply the resisting torque on the pinion.

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`.

Use multiple **Gear Pair** nodes, with different properties, to connect the two gears in different cases.

Gear Pair: Rigid

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

2 In the **Settings** window for **Gear Pair**, type `Gear Pair: Rigid` in the **Label** text field.

3 Locate the **Gear Selection** section. From the **Wheel** list, choose **Helical Gear 1**.

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

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

Gear Pair: Constant Stiffness

1 Right-click **Gear Pair: Rigid** and choose **Duplicate**.

2 In the **Settings** window for **Gear Pair**, type `Gear Pair: Constant Stiffness` in the **Label** text field.

3 Locate the **Gear Pair Properties** section. Select the **Include gear elasticity** check box.

Gear Pair: Varying Stiffness

1 Right-click **Gear Pair: Constant Stiffness** and choose **Duplicate**.

2 In the **Settings** window for **Gear Pair**, type `Gear Pair: Varying Stiffness` in the **Label** text field.

Gear Pair: Transmission Error

1 Right-click **Gear Pair: Varying Stiffness** and choose **Duplicate**.

2 In the **Settings** window for **Gear Pair**, type `Gear Pair: Transmission Error` in the **Label** text field.

3 Locate the **Gear Pair Properties** section. Select the **Include transmission error** check box.

Gear Elasticity 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Constant Stiffness** click **Gear Elasticity 1**.

2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.

3 In the $k_{\mathrm{t,wh}}$ text field, type `kt`.

4 In the $k_{\mathrm{t,pn}}$ text field, type `kt`.

5 Locate the **Mesh Damping** section. In the c_{g} text field, type `(ct[s]/100)*mbd.grp2.kg`.

Gear Elasticity 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Varying Stiffness** node, then click **Gear Elasticity 1**.

2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.

3 In the $k_{\mathrm{t,wh}}$ text field, type `kt`.

4 In the $k_{\mathrm{t,pn}}$ text field, type `kt`.

5 From the **Contact ratio in mesh cycle** list, choose **Varying**.

6 In the ζ text field, type `0.8`.

7 Locate the **Mesh Damping** section. In the c_{g} text field, type `(ct[s]/100)*mbd.grp3.kg`.

Gear Elasticity 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Transmission Error** click **Gear Elasticity 1**.

2 In the **Settings** window for **Gear Elasticity**, locate the **Mesh Stiffness** section.

3 In the $k_{\mathrm{t,wh}}$ text field, type `kt`.

4 In the $k_{\mathrm{t,pn}}$ text field, type `kt`.

5 Locate the **Mesh Damping** section. In the c_{g} text field, type `(ct[s]/100)*mbd.grp4.kg`.

Transmission Error 1

1 In the **Model Builder** window, click **Transmission Error 1**.

2 In the **Settings** window for **Transmission Error**, locate the **Transmission Error** section.

3 In the e_{wh} text field, type `et*rect1(mbd.grp4.thm_wh/(2*pi/n))`.

4 In the e_{pn} text field, type `et*rect2(mbd.grp4.thm_pn/(2*pi/n))`.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

Compute the solution for a rigid pair.

STUDY 1: TRANSIENT (RIGID)

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1:` in the **Label** text field.

3 In the **Label** text field, type `Study 1: Transient (Rigid)`.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1: Transient (Rigid)** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,dT,T)`.

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

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

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint: Fixed-Gear 2 (Elastic)**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Constant Stiffness**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Varying Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Transmission Error**.

8 Click **Disable**.

Solution 1 (sol1)

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

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

Displacement (mbd)

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

Add another **Time dependent** study and compute the solution for a gear pair with constant stiffness.

ADD STUDY

1 In the **Study** 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 Click **Add Study** in the window toolbar.

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

STUDY 2: TRANSIENT (CONSTANT STIFFNESS)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2: Transient (Constant Stiffness)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2: Transient (Constant Stiffness)** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,dT,T)`.

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

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

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint: Fixed-Gear 2 (Elastic)**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Rigid**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Gear Pair: Varying Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Transmission Error**.

8 Click **Disable**.

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**, locate the **Absolute Tolerance** section.

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

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

Add another **Time dependent** study and compute the solution for a gear pair with varying stiffness.

ADD STUDY

1 In the **Study** 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 Click **Add Study** in the window toolbar.

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

STUDY 3: TRANSIENT (VARYING STIFFNESS)

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3: Transient (Varying Stiffness)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3: Transient (Varying Stiffness)** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,dT,T)`.

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

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

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint: Fixed-Gear 2 (Elastic)**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Rigid**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Gear Pair: Constant Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Transmission Error**.

8 Click **Disable**.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

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

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

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

Add another **Time dependent** study and compute the solution for a gear pair with a transmission error.

ADD STUDY

1 In the **Study** 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 Click **Add Study** in the window toolbar.

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

STUDY 4: TRANSIENT (TRANSMISSION ERROR)

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study 4: Transient (Transmission Error)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 4: Transient (Transmission Error)** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,dT,T)`.

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

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

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Hinge Joint: Fixed-Gear 2 (Elastic)**, **Component 1 (comp1)> Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Rigid**, **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame> Gear Pair: Constant Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd), Controls spatial frame>Gear Pair: Varying Stiffness**.

8 Click **Disable**.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver 1**.

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

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

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

RESULTS

Use the following instructions to plot the pinion angular velocity and the contact force for the gear pairs as shown in Figure 4 and Figure 5 respectively.

Pinion angular velocity

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pinion angular velocity` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Angular velocity (rad/s)`.

Global 1

1 Right-click **Pinion angular velocity** 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)>Multibody Dynamics>Gear pairs>Gear Pair: Rigid>Pinion>mbd.grp1.tht_pn - Pinion angular velocity - rad/s**.

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

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Hinge joints>Hinge Joint: Fixed-Gear 1>mbd.hgj1.th - Relative rotation - rad**.

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

6 Select the **Description** check box. In the associated text field, type `Gear rotation`.

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

8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Rigid

Global 2

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

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

3 From the **Dataset** list, choose **None**.

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

Expression	Unit	Description
mbd.grp2.tht_pn	rad/s	Pinion angular velocity

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Transient (Constant Stiffness)/Solution 2 (sol2)**.

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

Legends
Constant stiffness

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **None**.

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

Expression	Unit	Description
mbd.grp3.tht_pn	rad/s	Pinion angular velocity

5 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Transient (Varying Stiffness)/Solution 3 (sol3)**.

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

Legends
Varying stiffness

Global 4

1 Right-click **Global 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **None**.

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

Expression	Unit	Description
mbd.grp4.tht_pn	rad/s	Pinion angular velocity

5 Locate the **Data** section. From the **Dataset** list, choose **Study 4: Transient (Transmission Error)/Solution 4 (sol4)**.

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

Legends
Transmission error

Pinion angular velocity

1 In the **Model Builder** window, click **Pinion angular velocity**.

2 In the **Pinion angular velocity** toolbar, click **Plot**.

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

Contact force

1 Right-click **Pinion angular velocity** and choose **Duplicate**.

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

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Force (N)`.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **Contact force** node, then click **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**
mbd.grp1.Fc	N	Force at contact point

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
mbd.grp2.Fc	N	Force at contact point

Global 3

1 In the **Model Builder** window, click **Global 3**.

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

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
mbd.grp3.Fc	N	Force at contact point

Global 4

1 In the **Model Builder** window, click **Global 4**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
mbd.grp4.Fc	N	Force at contact point

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

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

Use the following instructions to plot the reaction forces and moments at the wheel center when the gear pair has varying stiffness as shown in Figure 6 and Figure 7 respectively.

Reaction forces (Varying stiffness)

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Reaction forces (Varying stiffness)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Transient (Varying Stiffness)/Solution 3 (sol3)**.

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

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Force (N)`.

Global 1

1 Right-click **Reaction forces (Varying 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)>Multibody Dynamics>Gear pairs>Gear Pair: Varying Stiffness>Wheel>Force at wheel center - N>mbd.grp3.F_whx - Force at wheel center, x component**.

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

Expression	Unit	Description
mbd.grp3.F_whx	N	Force at wheel center, x component
mbd.grp3.F_why	N	Force at wheel center, y component
mbd.grp3.F_whz	N	Force at wheel center, z component

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

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

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

7 Select the **Description** check box. In the associated text field, type `Gear rotation`.

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

9 In the **Reaction forces (Varying stiffness)** toolbar, click **Plot**.

Reaction forces (Varying stiffness)

1 In the **Model Builder** window, click **Reaction forces (Varying stiffness)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `0`.

5 In the **x maximum** text field, type `55`.

6 In the **y minimum** text field, type `-200`.

7 In the **y maximum** text field, type `500`.

8 In the **Reaction forces (Varying stiffness)** toolbar, click **Plot**.

Reaction moments (Varying stiffness)

1 Right-click **Reaction forces (Varying stiffness)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Reaction moments (Varying stiffness)` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Moment (N*m)`.

Global 1

1 In the **Model Builder** window, expand the **Reaction moments (Varying stiffness)** node, then click **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
mbd.grp3.M_whx	N*m	Moment at wheel center, x component
mbd.grp3.M_why	N*m	Moment at wheel center, y component
mbd.grp3.M_whz	N*m	Moment at wheel center, z component

Reaction moments (Varying stiffness)

1 In the **Model Builder** window, click **Reaction moments (Varying stiffness)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 In the **y minimum** text field, type `-18`.

4 In the **y maximum** text field, type `17`.

5 In the **Reaction moments (Varying stiffness)** toolbar, click **Plot**.

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>Eigenfrequency**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Add an **Eigenfrequency** study and compute the solution for a rigid gear pair.

STUDY 5: EIGENFREQUENCY (RIGID)

1 In the **Model Builder** window, click **Study 5**.
2 In the **Settings** window for **Study**, type `Study 5: Eigenfrequency (Rigid)` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 5: Eigenfrequency (Rigid)** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `5`.
4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
5 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd)>Hinge Joint: Fixed-Gear 1>Prescribed Motion 1**, **Component 1 (comp1)>Multibody Dynamics (mbd)>Hinge Joint: Fixed-Gear 2**, **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Constant Stiffness**, **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Varying Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd)>Gear Pair: Transmission Error**.
6 Click **Disable**.
7 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (mbd)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2 From the **Eigenfrequency (Hz)** list, choose **24269**.

Add another **Eigenfrequency** study and compute the solution for a gear pair with constant stiffness.

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> Eigenfrequency**.

4 Click **Add Study** in the window toolbar.

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

STUDY 6: EIGENFREQUENCY (CONSTANT STIFFNESS)

1 In the **Model Builder** window, click **Study 6**.

2 In the **Settings** window for **Study**, type `Study 6: Eigenfrequency (Constant Stiffness)` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 6: Eigenfrequency (Constant Stiffness)** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Multibody Dynamics (mbd)>Hinge Joint: Fixed-Gear 1>Prescribed Motion 1**, **Component 1 (comp1)>Multibody Dynamics (mbd)> Hinge Joint: Fixed-Gear 2**, **Component 1 (comp1)>Multibody Dynamics (mbd)> Gear Pair: Rigid**, **Component 1 (comp1)>Multibody Dynamics (mbd)> Gear Pair: Varying Stiffness**, and **Component 1 (comp1)>Multibody Dynamics (mbd)> Gear Pair: Transmission Error**.

5 Click **Disable**.

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

RESULTS

Mode Shape (mbd) 1

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

2 From the **Eigenfrequency (Hz)** list, choose **288.61+588.2i**.

3 In the **Mode Shape (mbd) 1** toolbar, click **Plot**.

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

5 From the **Eigenfrequency (Hz)** list, choose **26796+144.67i**.

6 In the **Mode Shape (mbd) 1** toolbar, click **Plot**.

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

Animation 1

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