Created in COMSOL Multiphysics 6.2

Stresses and Heat Generation in Landing Gear

Introduction

This model simulates the dynamics of the shock absorber used in a landing gear mechanism using the Multibody Dynamics interface present in COMSOL Multiphysics. It analyzes the stresses in the landing gear components and the energy dissipated in the absorber during the landing of an aircraft. The heat transfer analysis is also performed, in which the rise of temperature in the components due to heat generated in the shock absorber is calculated by coupling the Multibody Dynamics interface with the Heat Transfer in Solids interface.

Model Definition

The landing gear assembly, shown in Figure 1, consists of two tires, a shock-absorber piston, and a shock-absorber cylinder. Other components in the landing gear, which are not relevant in this analysis, are not modeled.

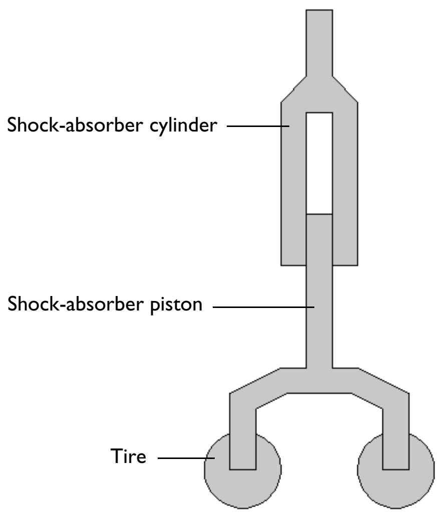

Figure 1: Geometry of the landing gear.

The shock-absorber piston and cylinder are connected through a prismatic joint that has only one translational degree of freedom. The joint is loaded with a spring and a damper to absorb the shock during the landing. This is modeled using the spring and damper subnode of the prismatic joint.

To model the effect of the tires, the bottom surfaces of the shock-absorber piston are loaded with equivalent spring and damper. The energy dissipated in the damper is used as a heat source on the common boundaries of the shock-absorber piston and cylinder. Remaining boundaries are considered to be insulated. The heat generated on the boundaries is conducted into the landing gear components during the landing, which increases their temperature.

Results and Discussion

Figure 2 displays the stress generated in the landing gear components at the time when the displacement reaches its maximum.

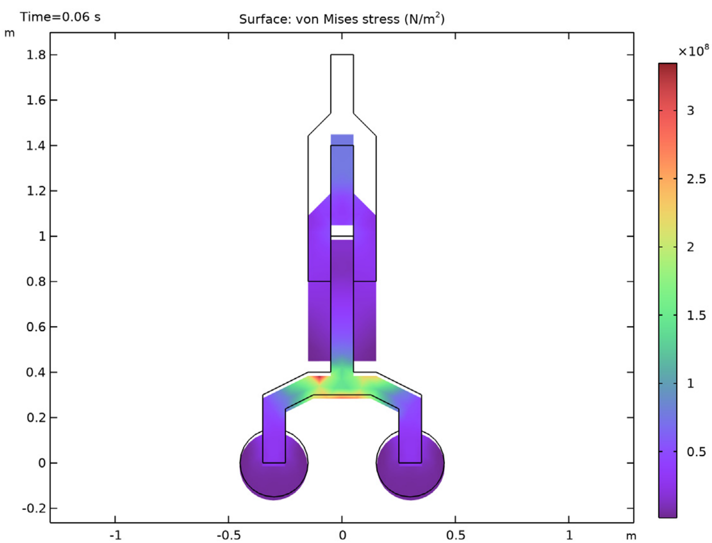

Figure 2: von Mises stress distribution in the deformed landing gear.

Figure 3 shows the relative displacement between the shock-absorber piston and cylinder during landing. The amplitude of the relative displacement is very high initially, but it decays rapidly due to the energy loss in the shock absorber and tires.

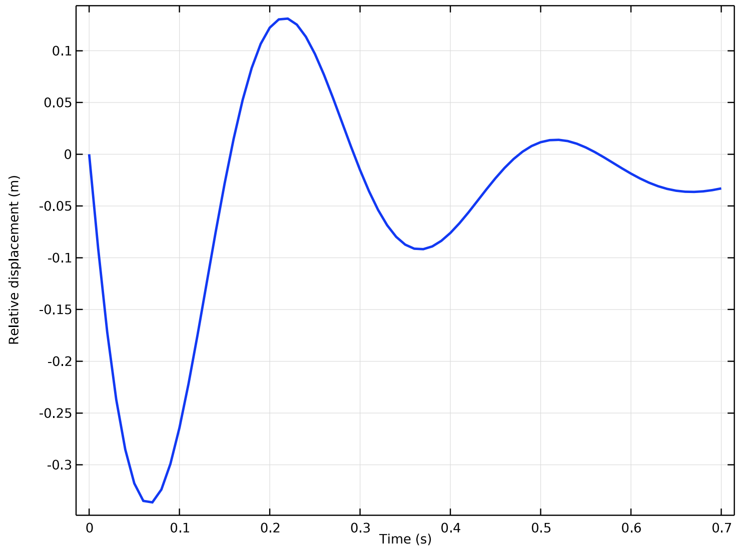

Figure 3: Relative displacement between the shock-absorber piston and cylinder.

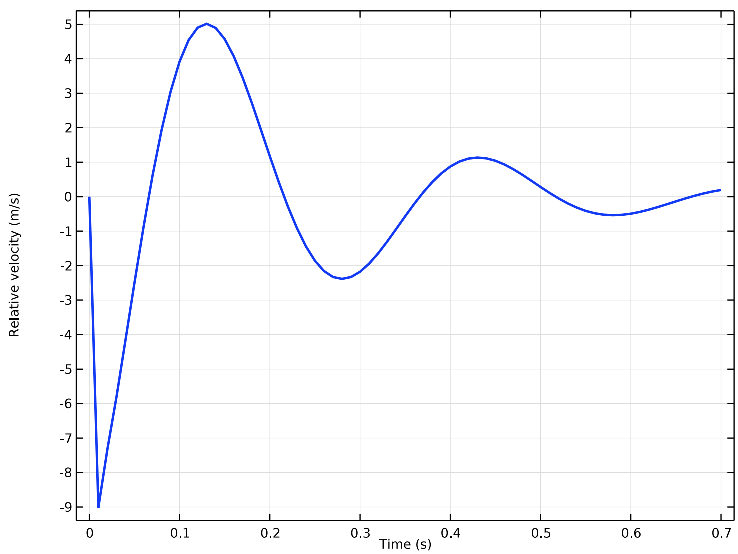

Figure 4: Relative velocity between the shock-absorber piston and cylinder.

[Figure 4](#) shows the relative velocity variation between the shock-absorber piston and cylinder during landing. The relative velocity is also very high initially, and it decays to zero due to the energy loss in the shock absorber and tires during the course of the landing.

The force component in y direction in the prismatic joint is shown in Figure 5. The force also decays with time and subsequently acquires a steady state value equivalent to the weight of the whole supported system.

Figure 6 shows the variation in the energy components during the landing. Initially, the kinetic energy is very high. But due to the dissipation in the subsequent cycles, it decays to zero. The other forms of energies, such as strain energy and potential energy, are very small as compared to the kinetic energy. This happens because most of the kinetic energy is stored in the shock absorber initially and then dissipated before getting converted to any other forms of energy.

The temperature rise in the components due to heat dissipated in shock-absorber is shown in Figure 7. The maximum temperature rise occurs near the piston-cylinder boundaries.

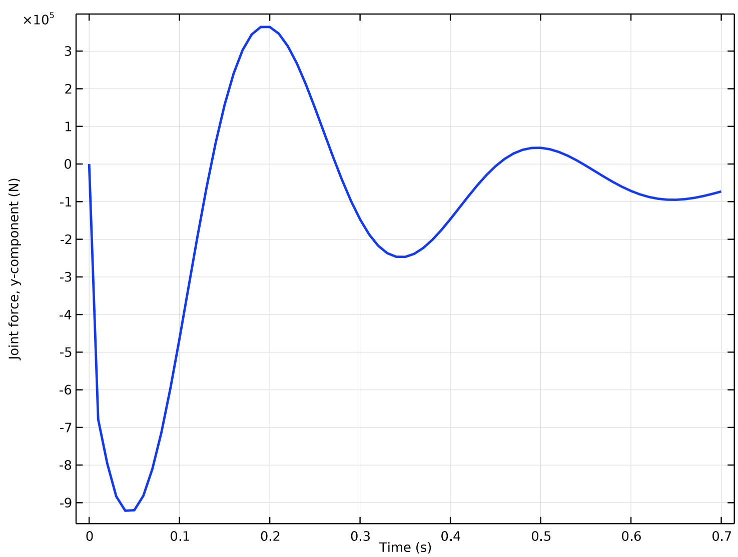

Figure 5: The y component of force in the shock-absorber.

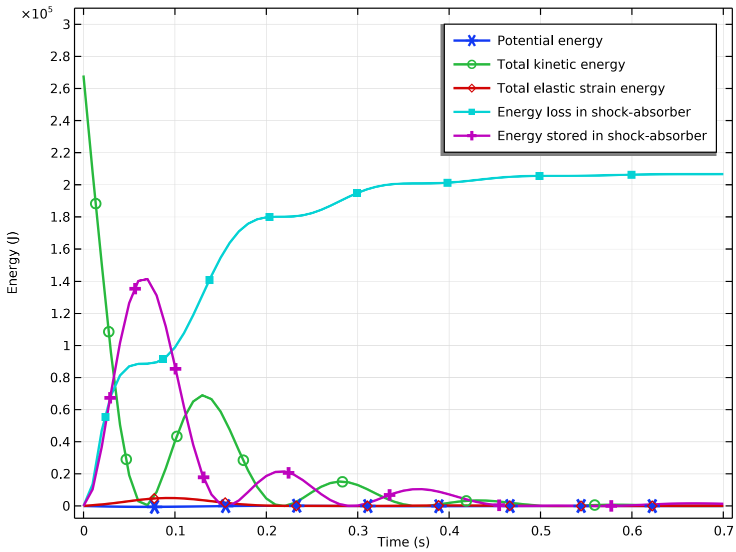

Figure 6: Variation of different types of energy in the gear during landing.

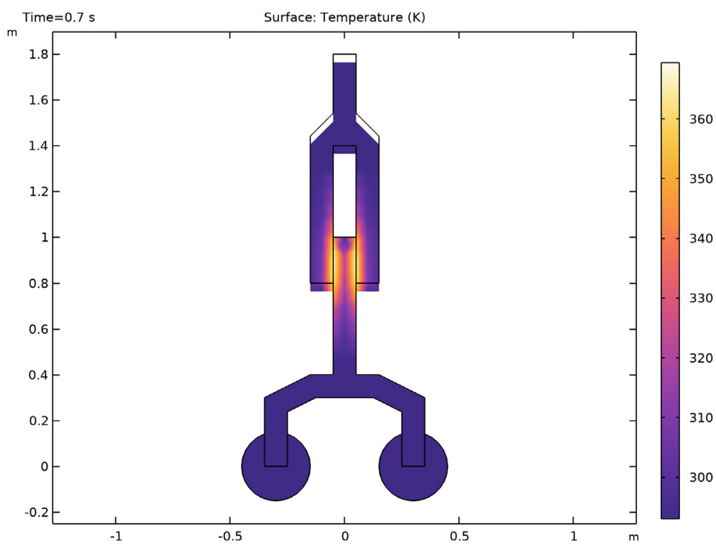

Figure 7: Temperature profile in the landing gear.

Application Library path: `Multibody_Dynamics_Module/Automotive_and_Aerospace/landing_gear`

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 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 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 In the table, enter the following settings:

Name	Expression	Value	Description
m	5000[kg]	5000 kg	Mass of the aircraft
vi	10[m/s]	10 m/s	Downward velocity of the aircraft
k_sa	2.5e6[N/m]	2.5E6 N/m	Spring constant of shock absorber
c_sa	5e4[N*s/m]	50000 N·s/m	Damping coefficient of shock absorber

Name	Expression	Value	Description
k_t	2.5e7[N/m]	2.5E7 N/m	Stiffness of tire
c_t	2.5e6[N*s/m]	2.5E6 N·s/m	Damping coefficient of tire
d	0.1[m]	0.1 m	Out of plane dimension

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `landing_gear.mph`. You can then continue to the **Add Material** section below.

To build the geometry from scratch, continue here.

GEOMETRY 1

Circle 1 (c1)

1. In the **Geometry** toolbar, click **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `0.15`.
4. Locate the **Position** section. In the **x** text field, type `-0.3`.

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. From the **Data source** list, choose **Vectors**.
4. In the **x** text field, type `-0.35 -0.35 -0.35 -0.15 -0.15 -0.05 -0.05 -0.05 -0.05 0 0 0 0 -0.15+0.1*(sqrt(5)-2) -0.15+0.1*(sqrt(5)-2) -0.25 -0.25 -0.25`.
5. In the **y** text field, type `0 0.3 0.3 0.4 0.4 0.4 0.4 1 1 1 1 0.3 0.3 0.3 0.3 0.3-0.05*(sqrt(5)-1) 0.3-0.05*(sqrt(5)-1) 0`.

Polygon 2 (pol2)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. From the **Data source** list, choose **Vectors**.
4. In the **x** text field, type `-0.15 -0.15 -0.15 -0.05 -0.05 -0.05 -0.05 0 0 0 0 -0.05 -0.05 -0.05`.

5 In the **y** text field, type `0.8 1.3+0.1*sqrt(2) 1.3+0.1*sqrt(2) 1.4+0.1*sqrt(2) 1.4+0.1*sqrt(2) 1.8 1.8 1.8 1.8 1.4 1.4 1.4 1.4 0.8`.

6 Click **Build Selected**.

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

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **pol1** only.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **copy1** only.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mir1(2)** and **pol1** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mir1(3)** and **pol2** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Union 3 (uni3)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **dif1**, **mir1(1)**, and **uni1** only.

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 Click **Build Selected**.
5 Click the **Go to Default View** button in the **Graphics** toolbar.
6 Click the **Zoom Extents** button in the **Graphics** toolbar.

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)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.
2 In the **Settings** window for **Multibody Dynamics**, locate the **Thickness** section.
3 In the *d* text field, type `d`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 From the list, choose **Locally defined**.
4 Specify the d**u**/dt vector as
5 In the table, enter the following settings:

`0`	X
`-vi`	Y

Attachment 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Attachment**.
2 Select Boundaries 34 and 40 only.

3 In the **Settings** window for **Attachment**, locate the **Connection Type** section.

4 From the list, choose **Flexible**.

Attachment 2

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

2 Select Boundaries 10 and 14 only.

3 In the **Settings** window for **Attachment**, locate the **Connection Type** section.

4 From the list, choose **Flexible**.

Prismatic Joint 1

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

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

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

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

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

0	x
-1	y

Spring and Damper 1

1 In the **Physics** toolbar, click **Attributes** and choose **Spring and Damper**.

2 In the **Settings** window for **Spring and Damper**, locate the **Spring and Damper: Translational** section.

3 In the k_{u} text field, type `k_sa`.

4 In the c_{u} text field, type `c_sa`.

Use the **Added Mass** node to account for the mass of the aircraft.

Added Mass 1

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

2 Select Boundaries 37 and 39 only.

3 In the **Settings** window for **Added Mass**, locate the **Added Mass** section.

4 From the **Mass type** list, choose **Total mass**.

5 From the list, choose **Diagonal**.

6 In the **m** table, enter the following settings:

0	0
0	m

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

The **Spring Foundation** node is used to account for the elastic and damping effects of the tires.

Spring Foundation 1

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

2 Select Boundaries 2 and 19 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{tot}$ table, enter the following settings:

0	0
0	k_t

7 Click to expand the **Viscous Damping** section. From the **Damping type** list, choose **Total damping constant**.

8 From the list, choose **Diagonal**.

9 In the $\mathbf{d}_{tot}$ table, enter the following settings:

0	0
0	c_t

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

3 From the **Selection** list, choose **All domains**.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

The heat generated due to the energy dissipation in the absorber is modeled as a heat source. This heat source is distributed on the common boundaries of the shock-absorber piston and cylinder.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Q	mbd.prj1.Qdamper	W	Heat generated per second
Wp	intop1(mbd.rho*g_const*v*mbd.d)	J	Potential energy
h_sa	timeint(0,t,Q)	J	Energy loss in shock-absorber
Ws_sa	mbd.prj1.Wspring	J	Energy stored in shock-absorber

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Physical Model** section.

3 In the d_z text field, type d.

Boundary Heat Source 1

1 In the **Physics** toolbar, click **Pairs** and choose **Boundary Heat Source**.

2 In the **Settings** window for **Boundary Heat Source**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Identity Boundary Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Heat Source** section.

7 From the **Heat source** list, choose **Heat rate**.

8 In the P_{b} text field, type Q.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

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

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** 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(0,0.01,0.7)`.

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

RESULTS

Follow these instructions to generate a plot similar to the one shown in Figure 2:

Stress

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

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

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Material (X, Y, Z)**.

4 Locate the **Data** section. From the **Time (s)** list, choose **0.06**.

Surface

1 In the **Model Builder** window, expand the **Stress** node, then click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Multibody Dynamics>Stress>mbd.misesGp - von Mises stress - N/m²**.

3 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

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

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

Follow these instructions to generate a plot similar to the one shown in Figure 7:

Surface 1

1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node, then click **Surface 1**.

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

3 From the **Resolution** list, choose **No refinement**.

Deformation 1

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `1`.

4 In the **Temperature (ht)** toolbar, click **Plot**.

Follow these instructions to generate a relative displacement plot similar to the one shown in Figure 3:

Relative Displacement

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 Right-click **Relative Displacement** 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>Prismatic joints>Prismatic Joint 1>mbd.prj1.u - Relative displacement - m**.

3 Click to expand the **Legends** section. Clear the **Show legends** check box.

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

5 In the **Relative Displacement** toolbar, click **Plot**.

Relative Displacement

Follow these instructions to generate a relative velocity plot similar to the one shown in Figure 4:

1 In the **Model Builder** window, right-click **Relative Displacement** and choose **Duplicate**.

Relative Velocity

1 In the **Model Builder** window, under **Results** click **Relative Displacement 1**.

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

Global 1

1 In the **Model Builder** window, expand the **Relative Velocity** node, then click **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 1 (comp1)>Multibody Dynamics>Prismatic joints>Prismatic Joint 1>mbd.prj1.u_t - Relative velocity - m/s**.

3 In the **Relative Velocity** toolbar, click **Plot**.

Relative Velocity

Follow these instructions to generate a joint force plot similar to the one shown in Figure 5:

1 In the **Model Builder** window, right-click **Relative Velocity** and choose **Duplicate**.

Joint Force

1 In the **Model Builder** window, under **Results** click **Relative Velocity 1**.

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

Global 1

1 In the **Model Builder** window, expand the **Joint Force** node, then click **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 1 (comp1)>Multibody Dynamics>Prismatic joints>Prismatic Joint 1>Joint force - N>mbd.prj1.Fy - Joint force, y-component**.

3 In the **Joint Force** toolbar, click **Plot**.

To generate the energy plot shown in Figure 6, follow the instructions below.

Energy

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

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

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

Global 1

1 Right-click **Energy** 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>Wp - Potential energy - J**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Wk_tot - Total kinetic energy - J**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Global>mbd.Ws_tot - Total elastic strain energy - J**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>h_sa - Energy loss in shock-absorber - J**.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Ws_sa - Energy stored in shock-absorber - J**.

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

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Energy** toolbar, click **Plot**.

Energy

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

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

3 Select the **y-axis label** check box. In the associated text field, type `Energy (J)`.

4 Locate the **Axis** section. Select the **Manual axis limits** check box.

5 In the **y maximum** text field, type `3.1e5`.

6 In the **Energy** toolbar, click **Plot**.

Use the instructions below to generate an animation of the stress distribution in the landing gear components.

Animation 1

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

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

3 From the **Target** list, choose **Player**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `100`.

Finally, follow these instructions to generate an animation of the temperature distribution in the components.

Animation 2

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

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

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Temperature (ht)**.

5 Locate the **Frames** section. In the **Number of frames** text field, type `100`.