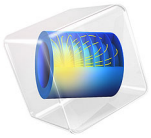

Model created in COMSOL Multiphysics 6.4

Door Slam Analysis

Introduction

This example illustrates the modeling of a car door slam event. This type analysis is important for several reasons:

- It should be easy to close the door at different speeds, both by a soft closure and by an aggressive slam.
- Usually there are loudspeakers mounted in the doors. Such components can be sensitive to excessive accelerations.
- The hinges must be able to withstand the forces caused by repetitive door slams.
- The pressure on the seals when the door is closed is an important factor for keeping noise a dust out of the compartment.
- The sound emitted when a car door is closed is important to the perception of quality.

The model in this example is simplified in several ways, most importantly:

- The geometry of a real car door is far more complex.
- There are no components, such as loudspeakers, in the door.
- The glass pane is omitted.
- The seals are made from homogeneous rubber. A real door seal will usually have an internal air cavity with intermittent evacuation holes, which makes the stiffness strongly velocity dependent.
- The locking mechanism is simplified.

Model Definition

Figure 1 shows the geometry of the car door and the frame in the car body. .

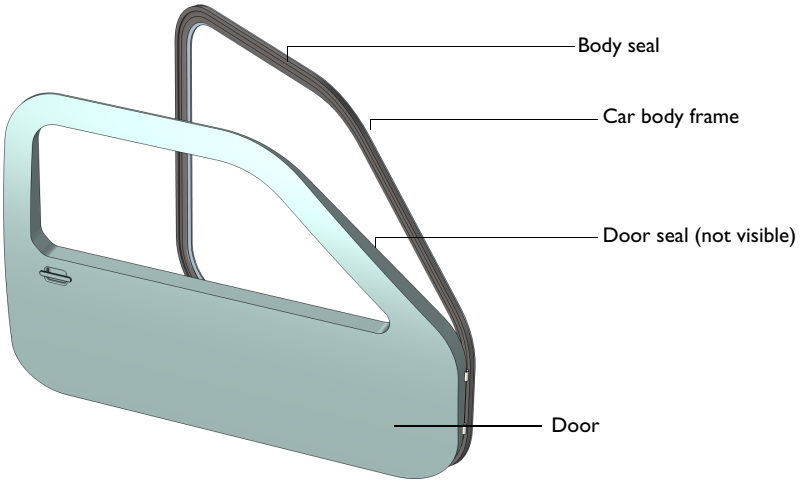

Figure 1: Geometry of the car door and the frame.

The material in the door is aluminum, with data taken from the Material Library. The rubber material in the seals have the following properties:

- Material model: Mooney-Rivlin
- C_{10} = 0.37 MPa
- C_{01} = 0.11 MPa
- Mass density 1100 kg/m^3

The door starts at position where it is opened 30°. It is given an initial angular velocity of 2 rad/s, corresponding to a velocity of about 1.4 m/s at the outermost point.

A penalty method is chosen for the contact modeling. The penalty factor is set fairly high in order to avoid excessive overclosure of the contact boundaries. The penalty function is set to **Smooth ramp**. This can improve performance significantly for impact problems.

To model the locking mechanism, simple unidirectional damper with a high viscosity is used. As soon as the velocity if the door is reversed, this damper is activated and effectively stops the rebound. The damper is implemented using a weak contribution.

Results and Discussion

The stress distribution in the door after the system has come to rest is shown in Figure 2. This stress state is an effect of that the locking mechanism keeps the door pressed against seals.

Note that all stress values as such are of little value because of the crude representation of the door and the low-order elements used,

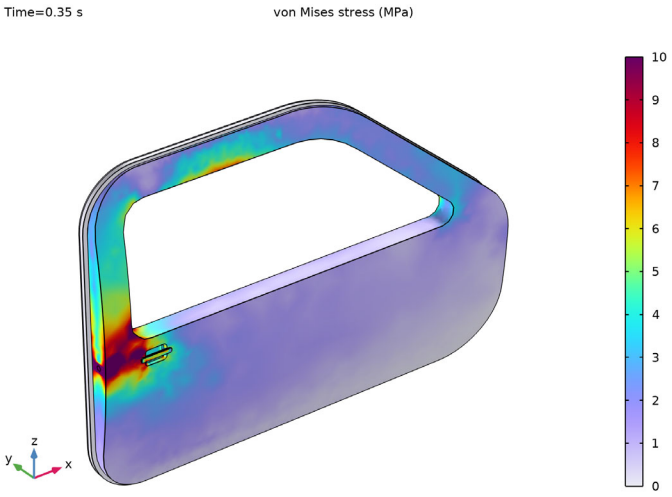

Figure 2: Equivalent von Mises stress in the door at the end of the event.

In Figure 3 and Figure 4, the stress distributions in the two seals at the end of the event are shown.

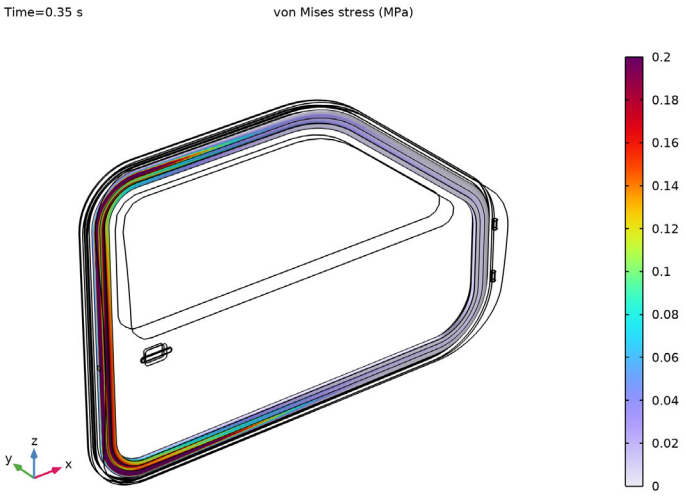

Figure 3: Stress distribution in the seal on the door at the end of the event.

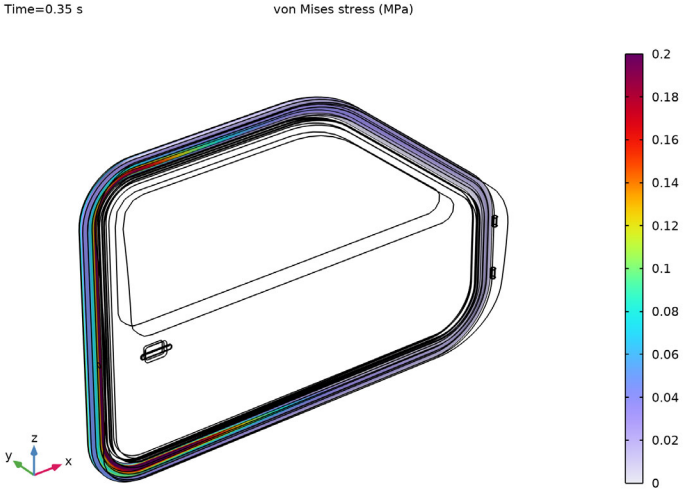

Figure 4: Stress distribution in the seal at the end of the event.

In Figure 5, the acceleration normal to the door is shown in two points on the door. This type of results are useful for evaluating the forces in components mounted on the door.

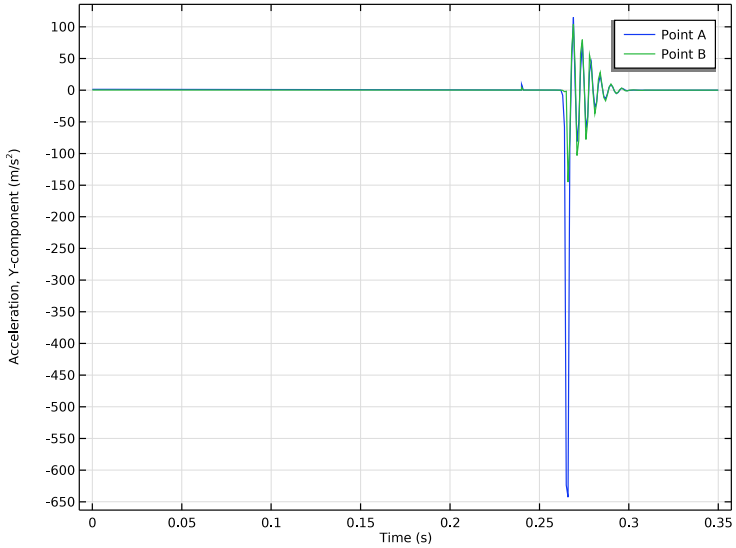

Figure 5: Acceleration in two points in the door during the door slam event.

The forces and moments acting on the hinges are shown in Figure 6 to Figure 9.

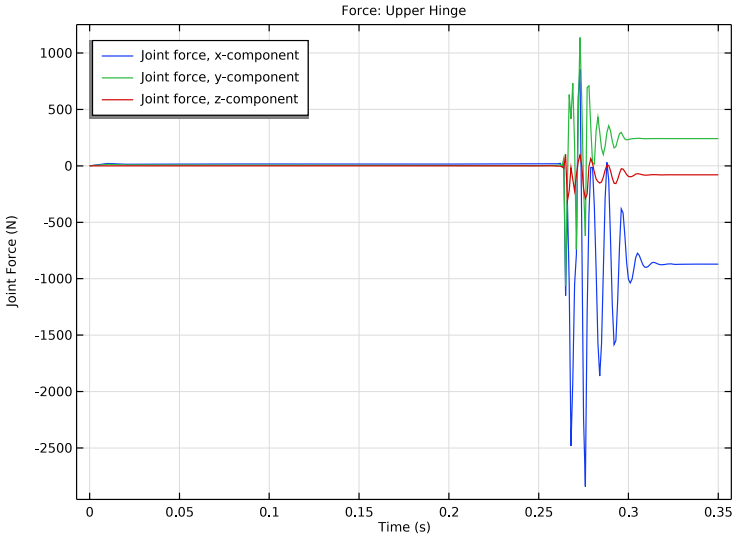

Figure 6: Forces in the upper hing during the door slam event.

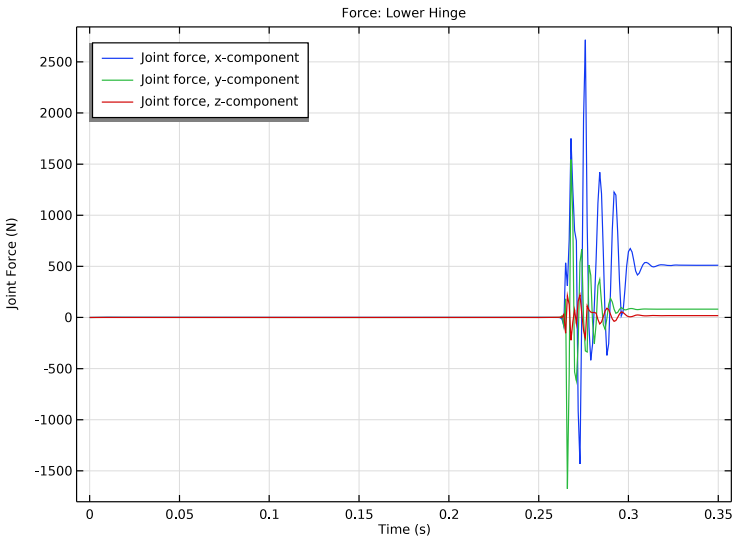

Figure 7: Forces in the lower hinge during the door slam event.

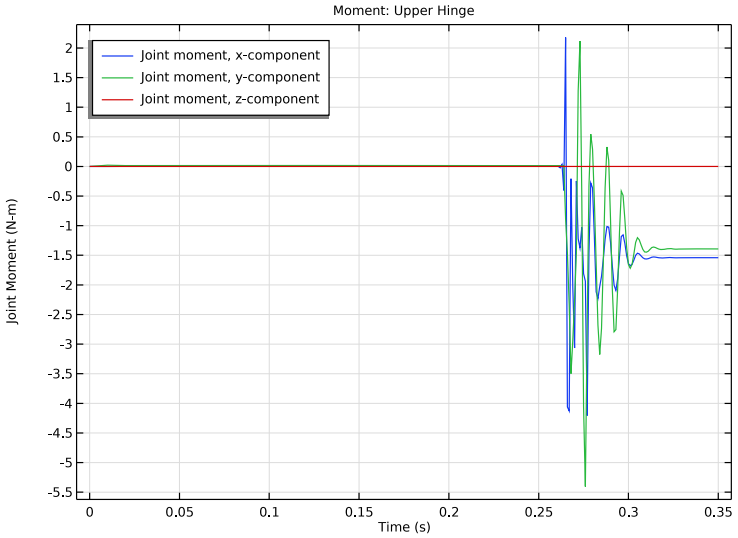

Figure 8: Moments in the upper hinge during the door slam event.

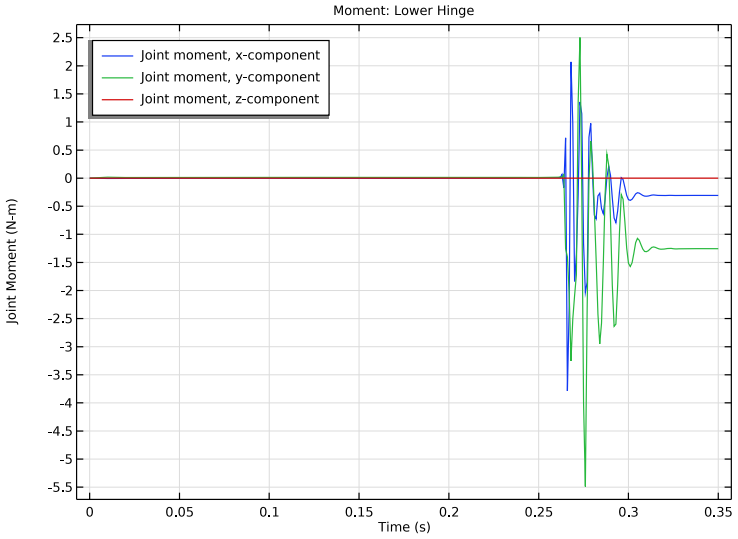

Figure 9: Moments in the lower hinge as during the door slam event.

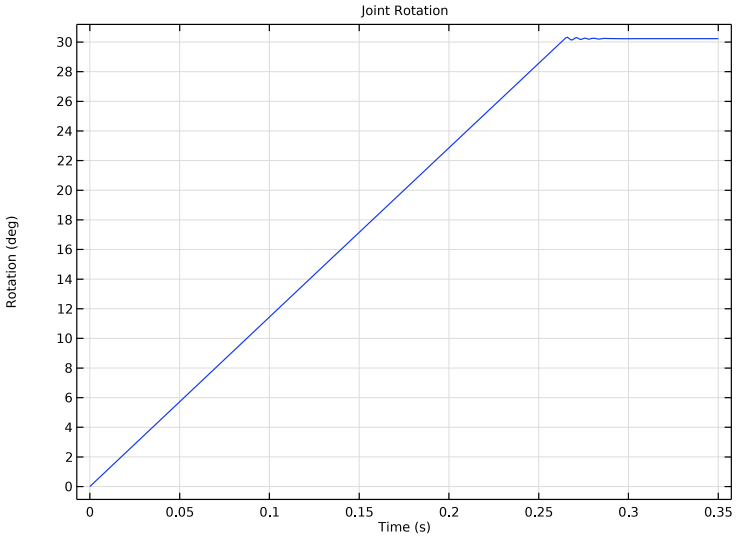

Figure 10: Rotation in the upper hinge during the door slam event.

Notes About the COMSOL Implementation

Two physics interfaces are used in the model: Solid Mechanics and Joints. All types of joints can be modeled either using the Joints interface or using the Multibody Dynamics interface. If this analysis had involved only linear elastic material, it had been sufficient yo use only a Multibody Dynamics interface. Since the rubber seals are modeled as hyperelastic, the Solid Mechanics interface has to be used. The hinge joints is then most easily introduced by adding a Joints interface. In either case, the Multibody Dynamics module is required.

Linear shape functions are used in this example to speed up the solution. First order elements are known to have bad predictive capabilities for stresses, so this type of modeling is only suitable for determining displacements, velocities, and accelerations.

In this example, a **General Contact Pair** node together with a **General Contact** node in the Solid Mechanics interface is used for modeling the contact between seals, car frame, and car door. An alternative is to use three **Contact Pair** nodes together with a **Contact** node. The latter method is computationally more efficient, and will in this case lead to about 30% shorter solution time. The setup is however mote complicated, since you need to define

three contact pairs together with their boundary selections, and also take meshing constraints into account.

The locking of the door is modeled by activating a strong viscous damper at the lock position at the moment when the door starts to bounce back. This is implemented in a **Weak Contribution** node.

Application Library path: `Multibody_Dynamics_Module/Tutorials/door_slam_analysis`

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** > **Solid Mechanics (solid)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Structural Mechanics** > **Joints (joints)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies** > **Time Dependent**.
8. Click **Done**.

GEOMETRY 1

You can import the geometry of the door assembly by browsing to the model's Application Libraries folder.

1. In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node, then click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Advanced** section.
3. From the **Geometry representation** list, choose **CAD kernel**.

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 `door_slam_analysis.mphbin`.

5 Click **Import**.

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

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
`omegaInit`	`2[rad/s]`	2 rad/s	`Initial angular velocity`

DEFINITIONS

Now, create selections in the geometry. You will use them later when setting up the physics and mesh.

Door Seal

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

2 Select Domain 4 only.

3 In the **Settings** window for **Explicit**, type `Door Seal` in the **Label** text field.

Body Seal

1 Right-click **Door Seal** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Body Seal` in the **Label** text field.

3 Locate the **Input Entities** section. Click  **Clear Selection**.

4 Select Domain 1 only.

Seals

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

2 In the **Settings** window for **Union**, type `Seals` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click  **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Door Seal** and **Body Seal**.

5 Click **OK**.

Body

1 In the **Model Builder** window, right-click **Door Seal** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Body` in the **Label** text field.

3 Locate the **Input Entities** section. Click  **Clear Selection**.

4 Select Domain 2 only.

Door

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

2 In the **Settings** window for **Explicit**, type `Door` in the **Label** text field.

3 Locate the **Input Entities** section. Click  **Clear Selection**.

4 Select Domains 3 and 5 only.

Hinges

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

2 In the **Settings** window for **Explicit**, type `Hinges` in the **Label** text field.

3 Locate the **Input Entities** section. Click  **Clear Selection**.

4 Select Domains 6 and 7 only.

Door with Seal and Hinges

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

2 In the **Settings** window for **Union**, type `Door with Seal and Hinges` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Selections to add** list box, select **Body Seal**.

4 Under **Selections to add**, click  **Delete**.

5 Under **Selections to add**, click  **Add**.

6 In the **Add** dialog, select **Door** in the **Selections to add** list.

7 Click **OK**.

8 In the **Settings** window for **Union**, locate the **Input Entities** section.

9 Under **Selections to add**, click + **Add**.

10 In the **Add** dialog, select **Hinges** in the **Selections to add** list.

11 Click **OK**.

Hinge 1: Inner Boundaries

1 In the **Model Builder** window, right-click **Door Seal** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Hinge 1: Inner Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Clear Selection**.

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

6 Select Boundaries 57, 58, 60, 68, and 74 only.

Hinge 2: Inner Boundaries

1 Right-click **Hinge 1: Inner Boundaries** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Hinge 2: Inner Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

4 Select Boundaries 55, 56, 59, 67, and 70 only.

Average 1 (aveop1)

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

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

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

4 Select Boundary 18 only.

5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 Right-click and choose **Paste**.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
vel	aveop1(d(v,TIME))	m/s	Velocity
disp	aveop1(v)	m	Displacement

General Contact Pair 1 (p1)

Create contact pairs between different components, which can come into contact when the door closes. To make the contact search more efficient, select only domains that can come into contact.

1 In the **Definitions** toolbar, click **Pairs** and choose **General Contact Pair**.

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

3 From the **Selection** list, choose **Manual**.

4 Locate the **Domain Selection** section. Click to select the **Activate Selection** toggle button.

5 Select Domains 1–4 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

3 From the **Displacement field** list, choose **Linear**.

Hyperelastic Material: Seals

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, type `Hyperelastic Material: Seals` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Seals**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney–Rivlin, two parameters**.

Damping 1

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

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

3 In the α_{dM} text field, type `0.5`.

4 In the β_{dK} text field, type `0.002`.

Contact Model 1

1 In the **Model Builder** window, expand the **General Contact 1** node, then click **Contact Model 1**.

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

3 Find the **Penalty function** subsection. From the list, choose **Smooth ramp**.

Fixed Constraint: Body

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

2 In the **Settings** window for **Fixed Constraint**, type `Fixed Constraint: Body` in the **Label** text field.

3 Select Boundary 5 only.

Attachment: Upper Hinge

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

2 In the **Settings** window for **Attachment**, type `Attachment: Upper Hinge` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hinge 1: Inner Boundaries**.

Attachment: Lower Hinge

1 Right-click **Attachment: Upper Hinge** and choose **Duplicate**.

2 In the **Settings** window for **Attachment**, type `Attachment: Lower Hinge` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Hinge 2: Inner Boundaries**.

Attachment: Lower Hinge, Attachment: Upper Hinge

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)**, Ctrl-click to select **Attachment: Upper Hinge** and **Attachment: Lower Hinge**.

2 Right-click and choose **Group**.

Attachments

In the **Settings** window for **Group**, type `Attachments` in the **Label** text field.

Initial Values: Door

1 In the **Physics** toolbar, click **Domains** and choose **Initial Values**.

2 In the **Settings** window for **Initial Values**, type `Initial Values: Door` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Door with Seal and Hinges**.

4 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

5 In the **Structural velocity field** vector, enter

`omegaInit*(Y-solid.att1.xcy)`	X
`-omegaInit*(X-solid.att1.xcx)`	Y

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

8 Click **OK**.

9 Click the **Show More Options** button in the **Model Builder** toolbar.

10 In the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

11 Click **OK**.

Weak Contribution: Locking Condition

1 In the **Physics** toolbar, click **Global** and choose **Weak Contribution**.

The locking mechanism is modeled by a high viscous damping which is activated once the door tries to bounce back.

2 In the **Settings** window for **Weak Contribution**, type `Weak Contribution: Locking Condition` in the **Label** text field.

3 Locate the **Weak Contribution** section. In the **Weak expression** text field, type `if(disp>0.34&&vel<=0,-1e8*vel*test(disp),0)`.

JOINTS (JOINTS)

Upper Hinge

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

2 In the **Settings** window for **Hinge Joint**, type `Upper Hinge` in the **Label** text field.

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

4 From the **Destination** list, choose **Attachment: Upper Hinge (solid)**.

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

0	x
1	z

Lower Hinge

1 Right-click **Upper Hinge** and choose **Duplicate**.

2 In the **Settings** window for **Hinge Joint**, type `Lower Hinge` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Destination** list, choose **Attachment: Lower Hinge (solid)**.

MATERIALS

Assign material properties. Use Aluminum for door and body.

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** > **Aluminum**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

MATERIALS

Add model specific data for the rubber in the seals.

Seal Rubber

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Seal Rubber` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Seals**.

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

Property	Variable	Value	Unit	Property group
Model parameters	C10	`0.37[MPa]`	Pa	Mooney-Rivlin
Model parameters	C01	`0.11[MPa]`	Pa	Mooney-Rivlin
Density	rho	1100	kg/m³	Basic

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 199, 209, and 300 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

Free Quad 1

1 In the **Mesh** toolbar, click  **More Generators** and choose **Free Quad**.

2 Select Boundaries 41, 42, 47, 48, 51, 54, 63, 66, 72, and 76 only.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

Swept 1

1 In the **Mesh** toolbar, click  **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

4 Select Domains 1, 2, 4, and 6–34 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domains 1, 2, 4, 12–15, 18, 19, 24, 25, 28, 29, 31, and 34 only.

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

4 In the **Number of elements** text field, type 40.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 Select Domains 8–11, 16, 17, 20–27, 30, and 32–34 only.

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

4 In the **Number of elements** text field, type 12.

Distribution 3

1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domains 6 and 7 only.

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

4 In the **Number of elements** text field, type 6.

Free Triangular 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 14, 15, 18, and 29–35 only.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

2 Select Boundaries 30–35 only.

3 In the **Settings** window for **Size**, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Minimum element size** checkbox. In the associated text field, type `0.0015`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 2** and choose **Size**.

2 Select Boundaries 14, 15, 18, and 29 only.

3 In the **Settings** window for **Size**, locate the **Element Size** section.

4 From the **Predefined** list, choose **Extra fine**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

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(0,0.01,0.24) range(0.24,0.0005, 0.35)`.

Solution 1 (sol1)

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

The default solver suggests a segregated solver, which is not appropriate for this type of tight coupling between two physics interfaces.

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

3 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** and choose **Fully Coupled**.

By allowing some more iterations, you can avoid many time step changes if the nonlinear solver is on the verge of reaching convergence.

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 `6`.

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

RESULTS

1 Click the **Show Grid** button in the **Graphics** toolbar.

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

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, type `stre` in the text field.

5 In the tree, select **Solid Mechanics** > **Stress tensor (N/m^2)**.

6 Click **OK**.

7 In the **Settings** window for **Preferred Units**, locate the **Units** section.

8 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

9 Select the **Apply conversions to expressions with the same dimensions** checkbox.

Volume 1

1 In the **Model Builder** window, expand the **Results** > **Stress (solid)** node, then click **Volume 1**.

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

3 From the **Unit** list, choose **MPa**.

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

5 In the **Maximum** text field, type `10`.

6 Click to expand the **Quality** section. From the **Resolution** list, choose **Normal**.

7 From the **Smoothing** list, choose **Everywhere**.

8 From the **Smoothing threshold** list, choose **None**.

9 In the **Stress (solid)** toolbar, click **Plot**.

Stress: Door Seal

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Selection** section.

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

4 From the **Selection** list, choose **Door Seal**.

5 In the **Label** text field, type `Stress: Door Seal`.

Volume 1

1 In the **Model Builder** window, expand the **Stress: Door Seal** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Range** section.

3 In the **Maximum** text field, type `0.2`.

4 In the **Stress: Door Seal** toolbar, click **Plot**.

Stress: Body Seal

1 In the **Model Builder** window, right-click **Stress: Door Seal** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Stress: Body Seal` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Body Seal**.

4 In the **Stress: Body Seal** toolbar, click **Plot**.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.

3 In the **X** text field, type `-0.6 0.1`.

4 In the **Y** text field, type `-0.4`.

5 In the **Z** text field, type `0.2`.

6 From the **Snapping** list, choose **Snap to closest boundary**.

Acceleration

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

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

Point Graph 1

1 Right-click **Acceleration** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Acceleration and velocity** > **Acceleration - m/s²** > **solid.u_ttY - Acceleration, Y-component**.

5 Click to expand the **Legends** section. Select the **Show legends** checkbox.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Point A`
`Point B`

8 In the **Acceleration** toolbar, click **Plot**.

Force: Upper Hinge

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Joint Force (N)`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Force: Upper Hinge** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Joints** > **Hinge joints** > **Upper Hinge** > **Joint force - N** > **All expressions in this group**.

Force: Upper Hinge

1 In the **Model Builder** window, click **Force: Upper Hinge**.

2 In the **Force: Upper Hinge** toolbar, click **Plot**.

Force: Lower Hinge

1 Right-click **Force: Upper Hinge** and choose **Duplicate**.

2 In the **Model Builder** window, click **Force: Upper Hinge 1**.

3 In the **Settings** window for **1D Plot Group**, type `Force: Lower Hinge` in the **Label** text field.

Global 1

1 In the **Model Builder** window, 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)** > **Joints** > **Hinge joints** > **Lower Hinge** > **Joint force - N** > **All expressions in this group**.

Force: Lower Hinge

1 In the **Model Builder** window, click **Force: Lower Hinge**.

2 In the **Force: Lower Hinge** toolbar, click **Plot**.

Moment: Upper Hinge

1 Right-click **Force: Lower Hinge** and choose **Duplicate**.

2 In the **Model Builder** window, click **Force: Lower Hinge 1**.

3 In the **Settings** window for **1D Plot Group**, type `Moment: Upper Hinge` in the **Label** text field.

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

Global 1

1 In the **Model Builder** window, 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)** > **Joints** > **Hinge joints** > **Upper Hinge** > **Joint moment - N·m** > **All expressions in this group**.

Moment: Upper Hinge

1 In the **Model Builder** window, click **Moment: Upper Hinge**.

2 In the **Moment: Upper Hinge** toolbar, click **Plot**.

Moment: Lower Hinge

1 Right-click **Moment: Upper Hinge** and choose **Duplicate**.

2 In the **Model Builder** window, click **Moment: Upper Hinge 1**.

3 In the **Settings** window for **1D Plot Group**, type `Moment: Lower Hinge` in the **Label** text field.

Global 1

1 In the **Model Builder** window, 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)** > **Joints** > **Hinge joints** > **Lower Hinge** > **Joint moment - N·m** > **All expressions in this group**.

Moment: Lower Hinge

1 In the **Model Builder** window, click **Moment: Lower Hinge**.

2 In the **Moment: Lower Hinge** toolbar, click **Plot**.

Joint Rotation

1 Right-click **Moment: Lower Hinge** and choose **Duplicate**.

2 In the **Model Builder** window, click **Moment: Lower Hinge 1**.

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

4 Locate the **Legend** section. Clear the **Show legends** checkbox.

5 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Rotation (deg)`.

Global 1

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

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

3 Click **Clear Table**.

4 In the table, enter the following settings:

Expression	Unit	Description
`-joints.hgj1.th`	`deg`	`Rotation`

Joint Rotation

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

2 In the **Joint Rotation** toolbar, click **Plot**.

Force: Lower Hinge, Force: Upper Hinge

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Force: Upper Hinge** and **Force: Lower Hinge**.

2 Right-click and choose **Group**.

Joint Forces

In the **Settings** window for **Group**, type `Joint Forces` in the **Label** text field.

Moment: Lower Hinge, Moment: Upper Hinge

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Moment: Upper Hinge** and **Moment: Lower Hinge**.

2 Right-click and choose **Group**.

Joint Moments

In the **Settings** window for **Group**, type `Joint Moments` in the **Label** text field.

Stress

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

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

3 Locate the **Frames** section. In the **Number of frames** text field, type `200`.

Displacement

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

2 In the **Settings** window for **Animation**, type `Displacement` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Acceleration**.