Model created in COMSOL Multiphysics 6.4

Ladder Frame

Introduction

The ladder frame, or ladder chassis, is a common structure used in some light truck and SUV designs. An advantage of this structure is its simplicity: two longitudinal members connected by multiple transverse members.

The purpose of the chassis is to connect the upper body, the engine, and the wheel system. The stiffness of the chassis is important for the stability of the vehicle under operating conditions.

In this model, an eigenfrequency analysis is first performed on the unconstrained chassis structure. This is followed by a static analysis including a simplified suspension system with loads corresponding to the engine, the upper body, and an estimated payload. For the static load case, a weld verification at a critical location is also performed.

Model Definition

The ladder frame consists of a steel structure that is 4.5 m long and 1 m wide at maximum, as shown in Figure 1. The longitudinal members have a rectangular section with a thickness of 8 mm. The transverse members have a C-shape section of 5 mm thickness. The geometry is imported from a STEP-file as a solid object.

.

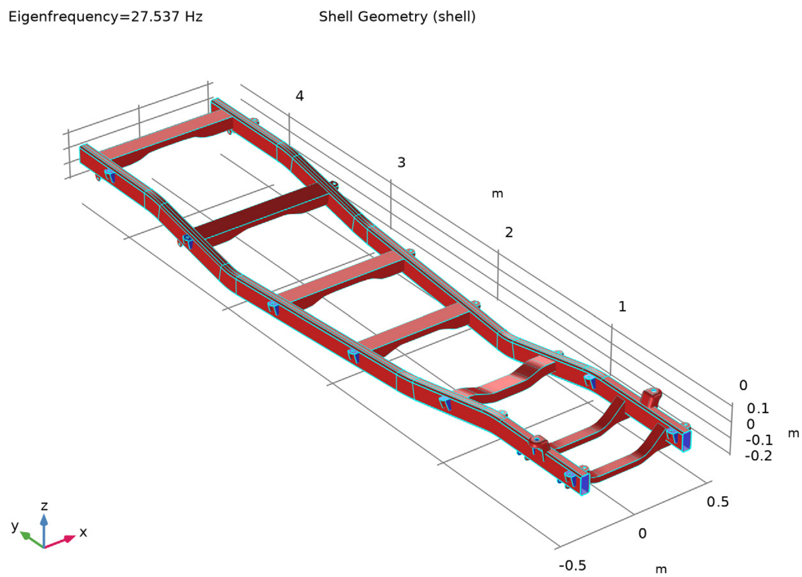


Figure 1: Shell geometry.

The nine lowest eigenfrequencies (natural frequencies) are computed. The eigenfrequency analysis is performed on the structure absent any connections to other vehicle components, and absent external constraints. This means that the six lowest eigenfrequencies correspond to rigid body modes, and are numerically zero. Higher modes correspond to deformations of the ladder frame.

For the stationary analysis, a simplified suspension component system is used to prevent over-constraint conditions when directly applying external loads to the structure.

The front suspension system is modeled using a 10^6 N/m spring constant, and the rear suspension system is modeled using a spring constant of $3 \cdot 10^6$ N/m.

The applied loads consist of

- the weight of the chassis
- the weight of the engine, estimated at 400 kg
- the weight of the upper body, combined with the payload, estimated at 2000 kg

Figure 2 shows the applied load on the chassis for the static analysis.

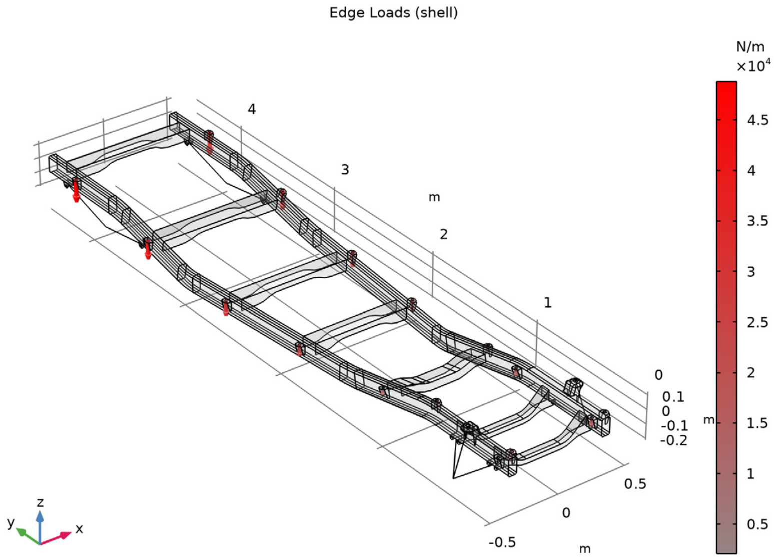


Figure 2: Load distribution.

Results and Discussion

Figure 3, Figure 4, and Figure 5 show the first, second, and third natural frequencies of the chassis, respectively (disregarding the six rigid body modes). The first frequency (27.5 Hz) corresponds to a torsion mode, the second one (30.4 Hz) to a bending mode in the vertical direction, and the third one (47.6 Hz) also to a bending mode, in the lateral direction.

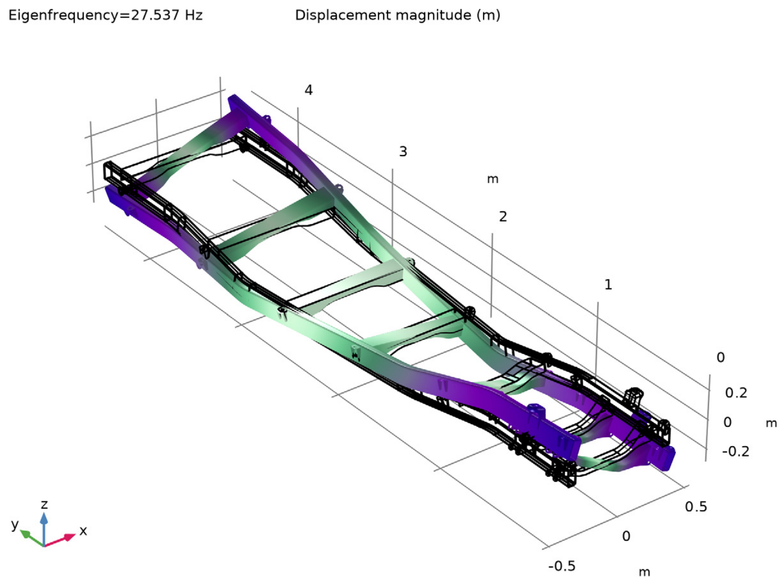


Figure 3: First natural frequency (torsion).

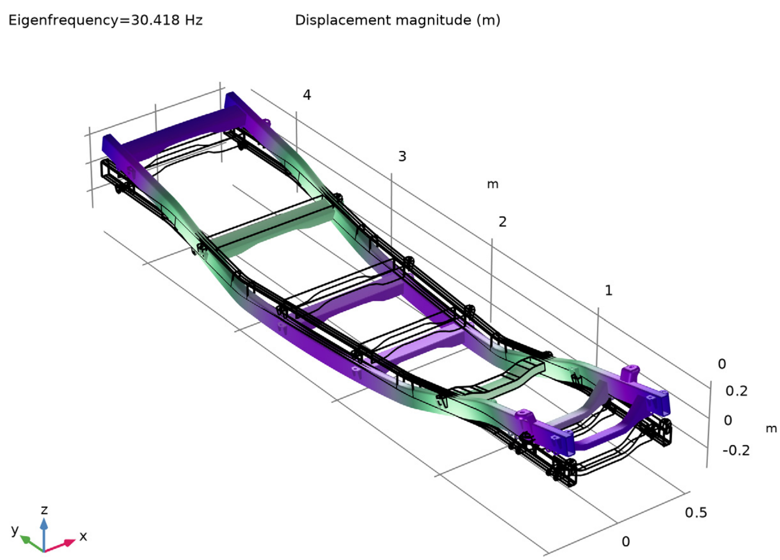


Figure 4: Second natural frequency (bending).

[

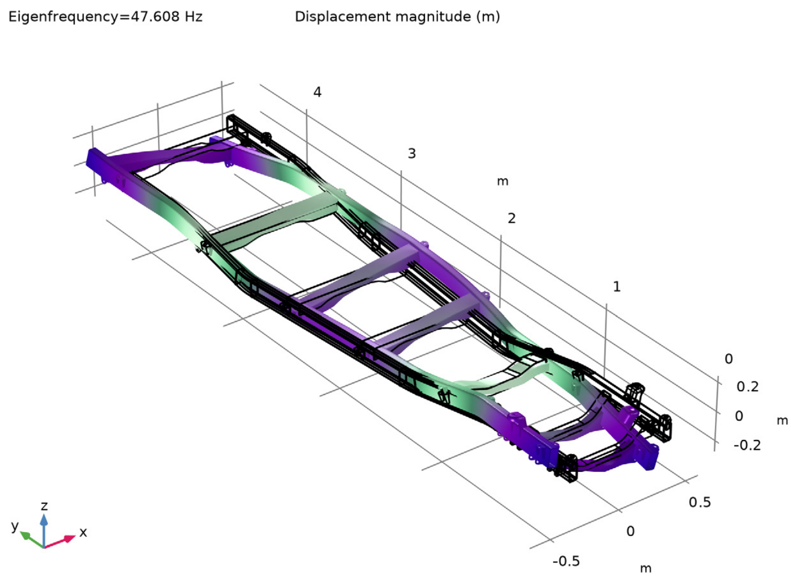


Figure 5: Third natural frequency (bending).

Figure 6 shows the displacement of the chassis subjected to engine and upper body weight, as well as the payload.

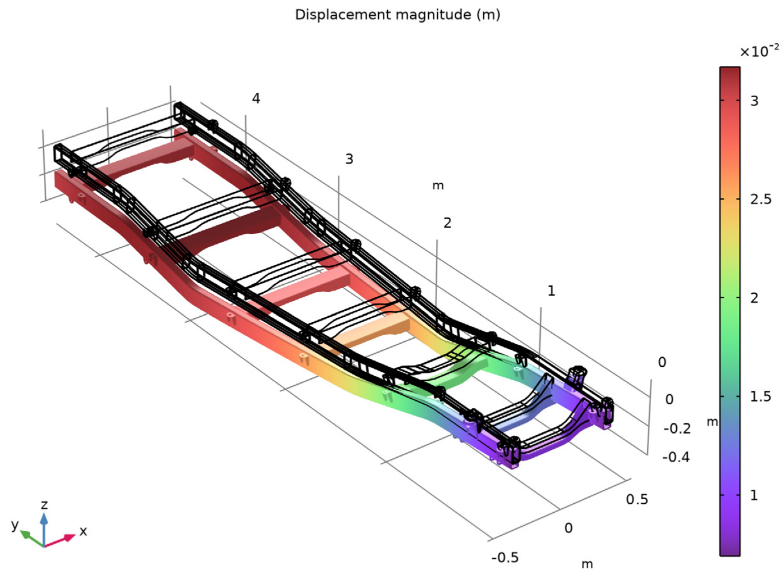


Figure 6: Chassis displacement.

Figure 7 shows the von Mises stress distribution in the chassis. Note that there is a stress concentration where the spring suspension is connected to the bracket. To improve the visualization of the stress distribution in the chassis, the maximum range value is set to 180 MPa.

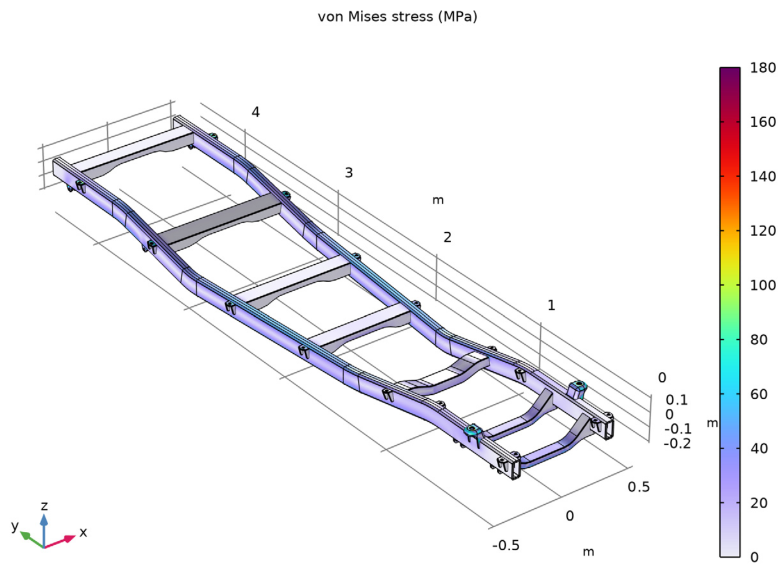


Figure 7: von Mises Stress distribution in the chassis.

Figure 8 shows that the weld failure index between the suspension bracket and the longitudinal beam is below 1, when the ladder frame is subjected to the external loading.

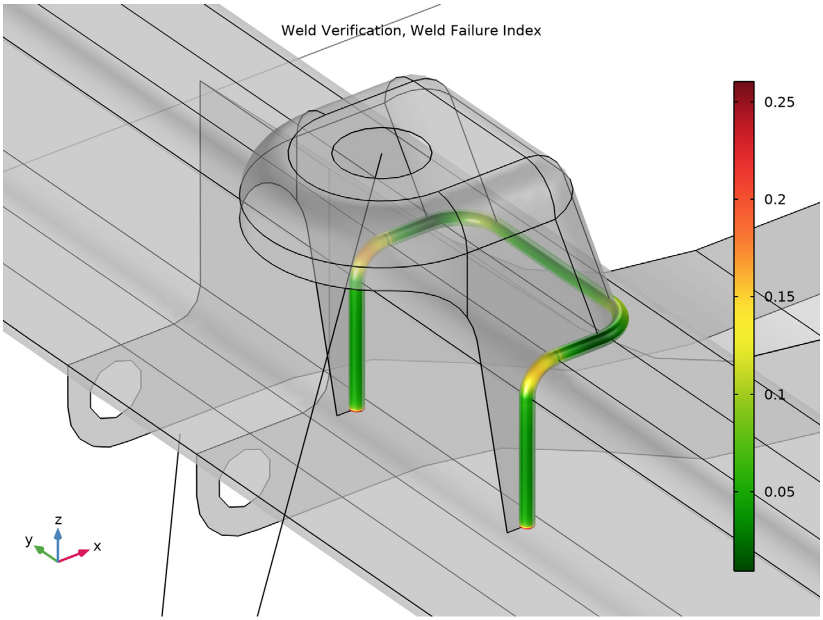


Figure 8: Weld verification.

Modeling in COMSOL Multiphysics

For geometries with a high aspect ratio in one direction, shell elements are preferred compared to solid elements. Using the Design Module, you can convert a solid geometry representation to a surface object by removing its thickness. In addition, the Design Module includes geometry defeaturing and measurement tools. For instance, you can delete fillets that are not relevant to the analysis to reduce the number of elements.

Using the Resultant load type, you can specify a given force at a specific application point; the corresponding load is then distributed at the geometry location.

To verify that the different parts in the Shell interface are well connected to each other, you can use the connected region indicator.

Figure 9 shows how the suspension bracket and the longitudinal members are connected to each other.

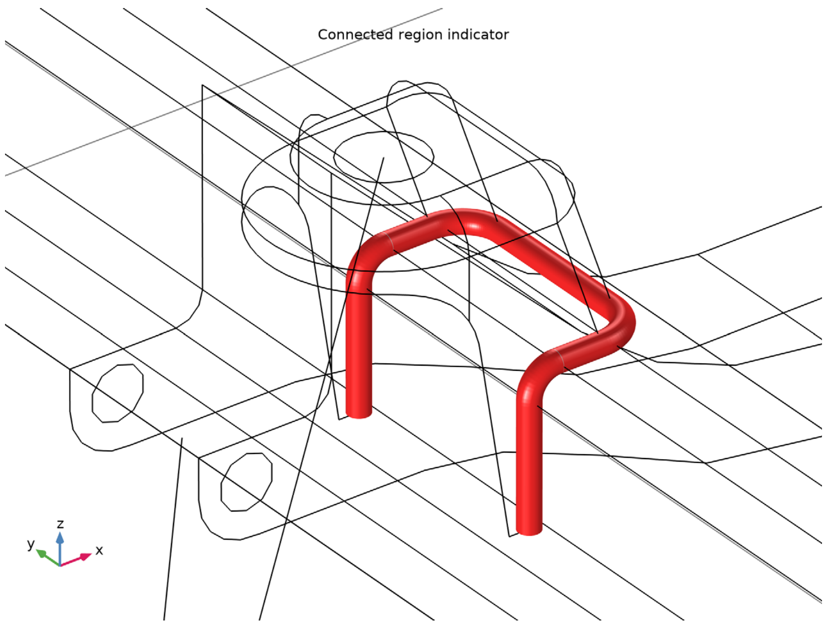


Figure 9: Connected region indicator.

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/ladder_frame`

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 > Shell (shell)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

4 Select the **Design Module Boolean operations** checkbox.

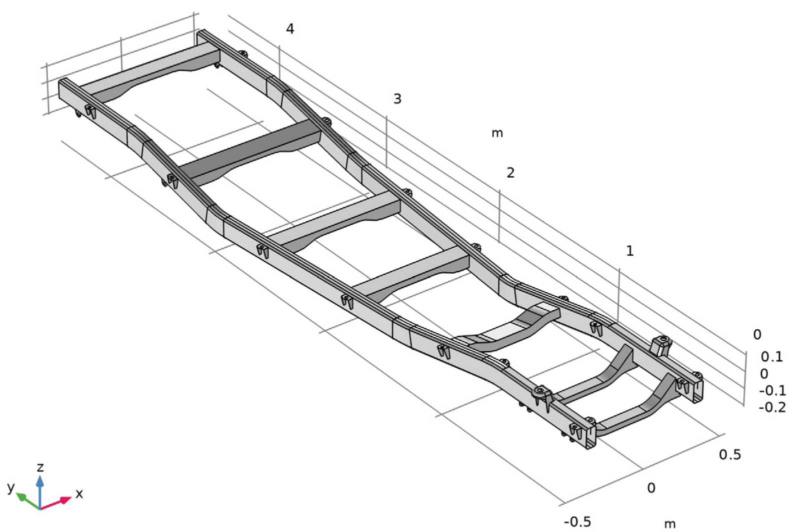
The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the tutorial under `applications/Design_Module/Tutorial_Examples`. Otherwise, insert the geometry sequence as follows:

5 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

6 Browse to the model's Application Libraries folder and double-click the file `ladder_frame_geom_sequence.mph`.

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

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



ADD MATERIAL

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

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

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

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

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

SHELL (SHELL)

Thickness and Offset 1

1 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

2 In the d_0 text field, type `th2`.

Thickness and Offset 2

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Longitudinal members**.

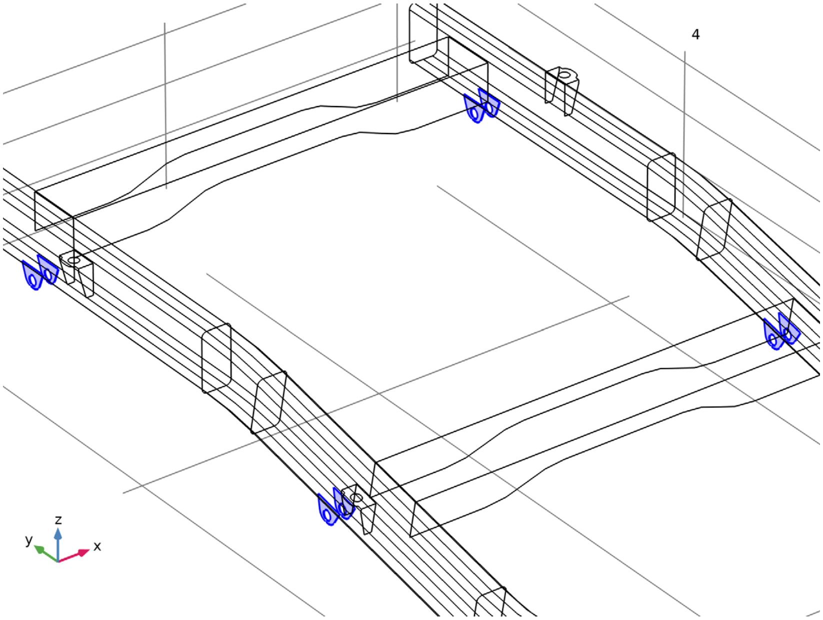
4 Locate the **Thickness and Offset** section. In the d_0 text field, type `th1`.

5 From the **Position** list, choose **Top surface on boundary**.

Thickness and Offset 3

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

2 Select Boundaries 33, 34, 47, 48, 291, 292, 306, and 307 only.



3 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

4 In the d_0 text field, type `th1`.

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 **Extra coarse**.

4 Click **Build All**.

STUDY 1

Step 1: Eigenfrequency

As there are no constraints in the model, the eigenvalue solver will also compute the rigid body motion modes. The corresponding eigenfrequencies are very small. To exclude these modes, search for eigenfrequencies with a larger real part than the shift value (1 Hz).

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

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

3 Select the **Desired number of eigenfrequencies** checkbox.

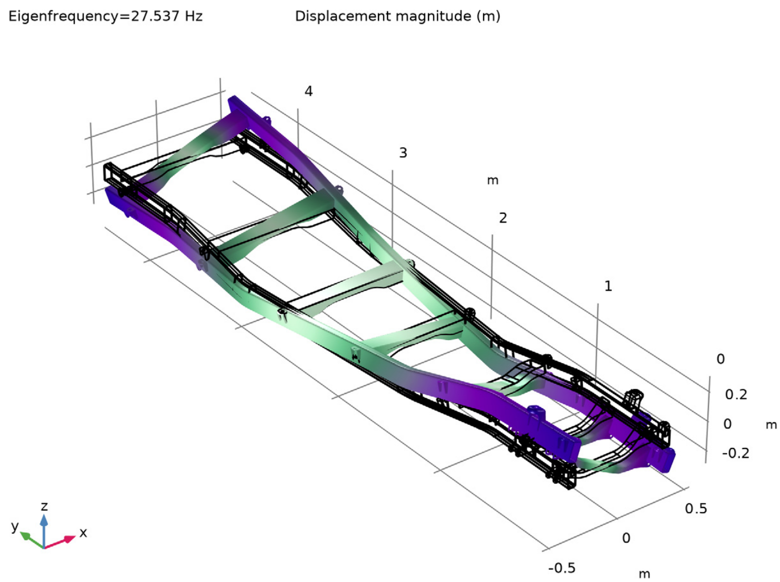
4 From the **Search method around shift** list, choose **Larger real part**.

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

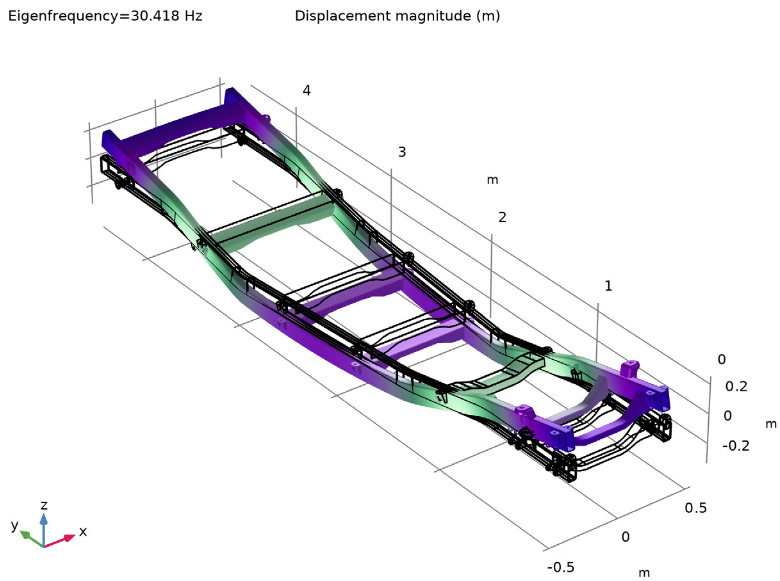
RESULTS

Mode Shape (shell)

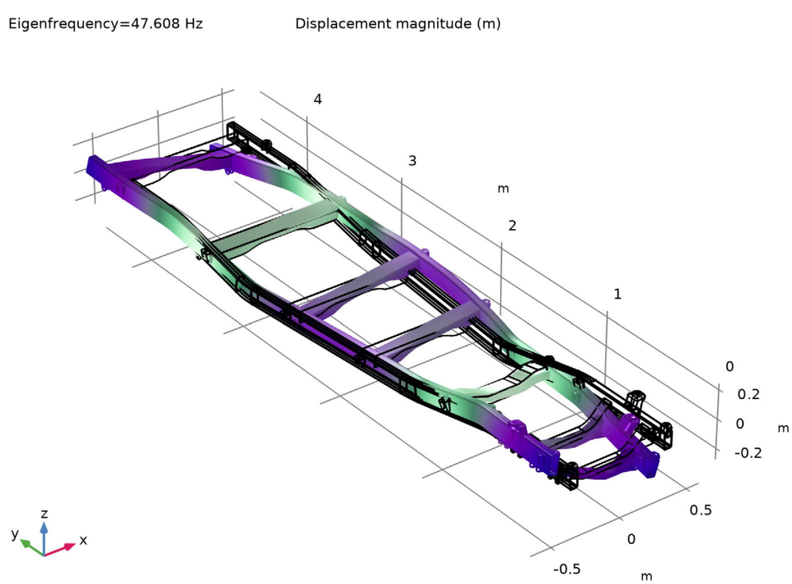
1 In the **Mode Shape (shell)** toolbar, click **Plot**.



2 In the **Settings** window for **3D Plot Group**, click → **Plot Next**.



3 Click → **Plot Next**.



Add the shell geometry plot as in Figure 1.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Shell Geometry (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Shell Geometry (shell)

In the second part of the analysis, the suspensions are added to the model. Also, a weld verification analysis is performed for the connection between one of the suspension brackets and the longitudinal beam.

GEOMETRY 1

Cross members (boxsel1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Cross members (boxsel1)**.

2 In the **Settings** window for **Box Selection**, click **Build Selected**.

Suspension Bracket

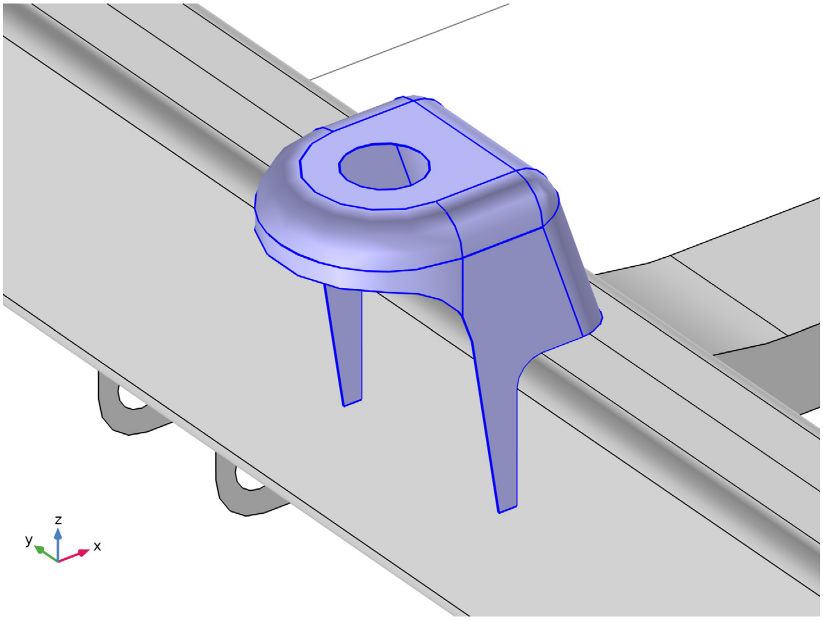
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 In the **Label** text field, type `Suspension Bracket`.

5 In the **Graphics** window, select the boundaries as in the figure below:



In the table below you can find the corresponding selected entities:

Entities to select
`uni1: 75, 78, 86-90, 121-125, 130`

Extract 2 (extract2)

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

2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.

3 From the **Selection** list, choose **Suspension Bracket**.

4 From the **Input object handling** list, choose **Create remainder object**.

Suspension Bracket (Edge)

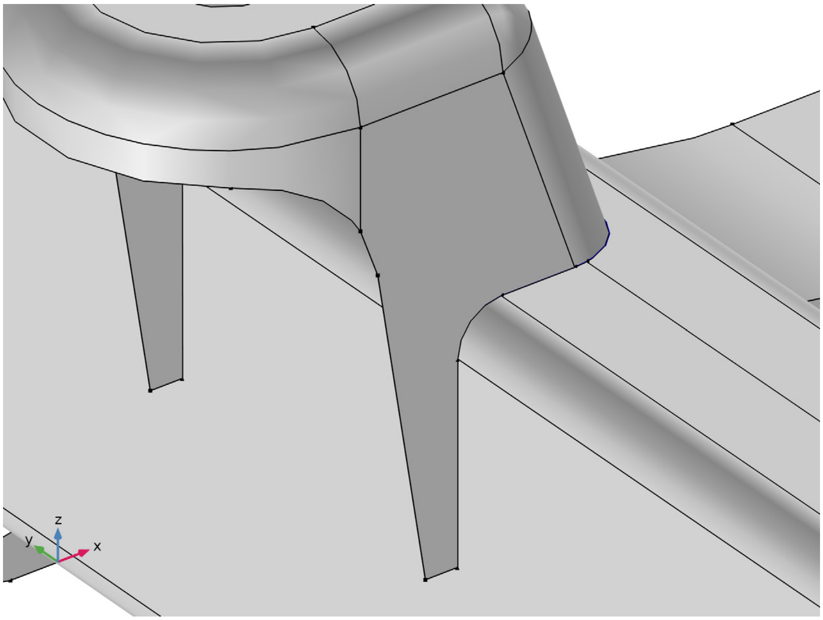
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Suspension Bracket (Edge)` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Edge**.

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

5 In the **Graphics** window, select the edges that are adjacent to the suspension bracket and connect to the longitudinal member.



In the table below you can find the corresponding selected entities:

Entities to select
`extract2(1): 22-27, 40-44`

Long Beam (Edge)

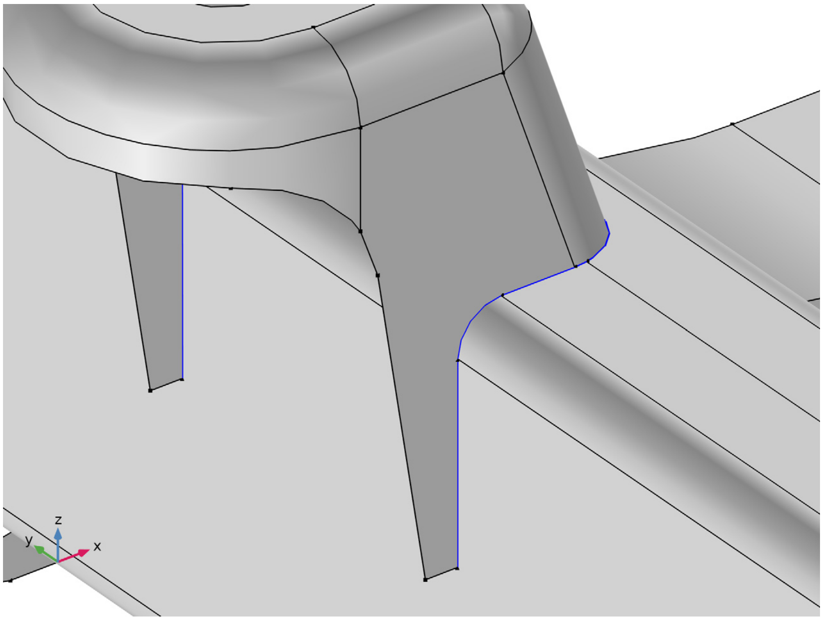
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Long Beam (Edge)` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Edge**.

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

5 In the **Graphics** window, select the edges that are adjacent to the longitudinal member and connect to the suspension bracket .



In the table below you can find the corresponding selected entities:

Entities to select
`extract2(2): 275, 277, 278, 280, 312, 314, 341, 342, 362, 364, 384`

GLOBAL DEFINITIONS

Suspension Coordinates

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Suspension Coordinates` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

GEOMETRY 1

Centroid Measurement 1 (cm1)

1 In the **Geometry** toolbar, click **Measurements** and choose **Centroid Measurement**.

2 On the object **extract2(2)**, select Points 543–546 only.

Centroid Measurement 2-5

Proceed to create four additional centroid measurements with the following settings:

Name	**Vertex Selection**
Centroid Measurement 2 (cm2)	`extract2(2): 547, 548, 549, 550`
Centroid Measurement 3 (cm3)	`extract2(2): 606, 644`
Centroid Measurement 4 (cm4)	`extract2(2): 699, 705, 734, 740`
Centroid Measurement 5 (cm5)	`extract2(2): 700, 706, 735, 741`

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `geom1.cm1.x`.

5 In the **y** text field, type `geom1.cm1.y`.

6 In the **z** text field, type `geom1.cm1.z`.

7 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

8 In the **x** text field, type `x_fwl`.

9 In the **y** text field, type `y_fwl`.

10 In the **z** text field, type `z_fwl`.

Line Segment 2-5

Proceed to create four additional line segments with the following settings:

Name	**Starting point coordinates**	**End point coordinates**
Line Segment 2 (ls2)	`geom1.cm2.x, geom1.cm2.y, geom1.cm2.z`	`x_fwl, y_fwl,z_fwl`
Line Segment 3 (ls3)	`geom1.cm3.x, geom1.cm3.y, geom1.cm3.z`	`x_fwl, y_fwl, z_fwl`
Line Segment 4 (ls4)	`geom1.cm4.x, geom1.cm4.y, geom1.cm4.z`	`x_rwl, y_rwl, z_rwl`
Line Segment 5 (ls5)	`geom1.cm5.x, geom1.cm5.y, geom1.cm5.z`	`x_rwl, y_rwl, z_rwl`

Union 2 (uni2)

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

2 Select the objects **ls1**, **ls2**, **ls3**, **ls4**, and **ls5** only.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog, type `Truss` in the **Name** text field.

6 Click **OK**.

Mirror 1 (mir1)

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

2 Select the object **uni2** only.

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

4 Select the **Keep input objects** checkbox.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type `1`.

6 In the **z** text field, type `0`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Truss**.

8 Click **Build Selected**.

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**.

Upper Body Bracket

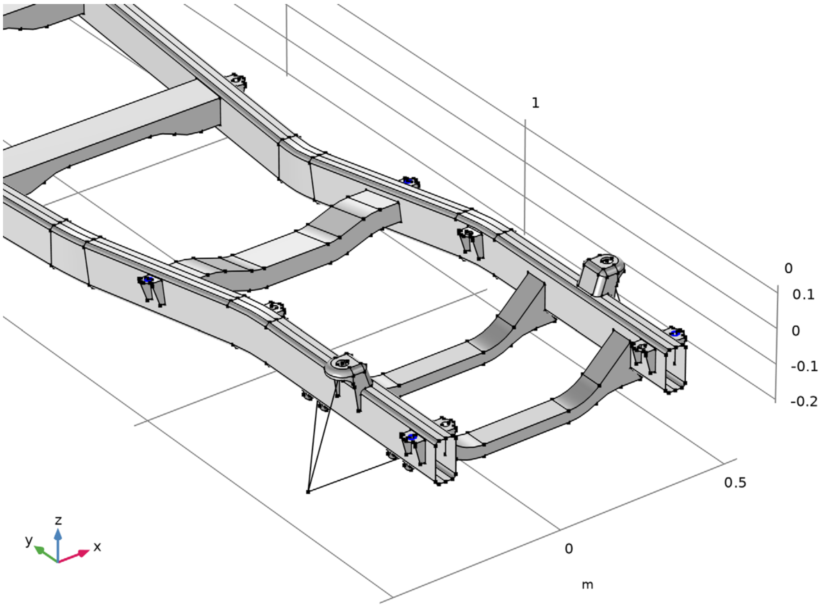
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Edge**.

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

5 In the **Graphics** window, select the edges around the hole for all brackets which are located at the exterior of the frame. This is shown in the figure below. Note that only the four brackets at the front of the frame are shown. The full selection includes twelve brackets.



In the table below you can find the corresponding selected entities:

Entities to select
`extract2(2): 21, 22, 25, 26, 64, 65, 211, 212, 239, 240, 256, 257, 856, 857, 864, 865, 898, 899, 1004, 1005, 1076, 1077, 1082, 1083`

Engine Bracket

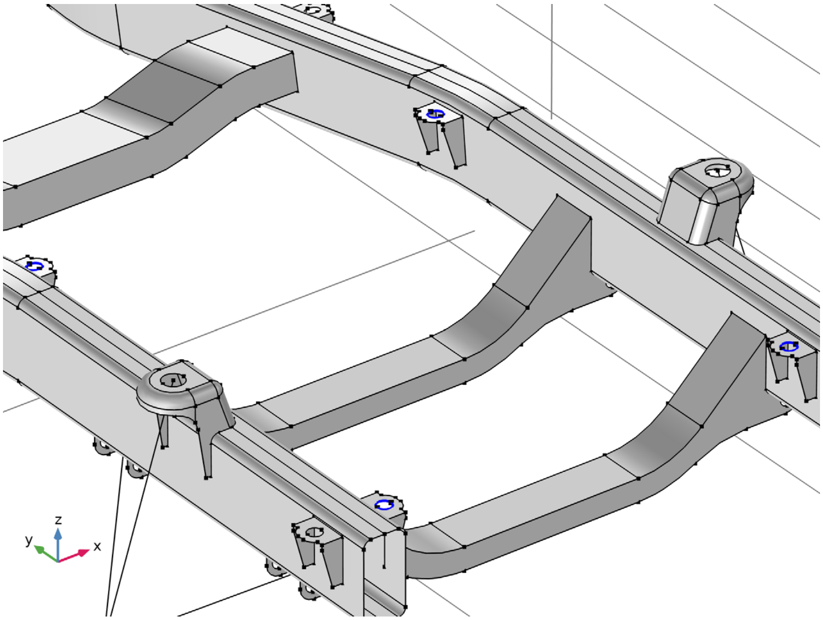
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Edge**.

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

5 In the **Graphics** window, select the edges around the holes for all brackets located at the interior of the frame, as in the figure below:



In the table below you can find the corresponding selected entities:

Entities to select
`extract2(2): 457, 458, 463, 464, 621, 622, 639, 640`

SHELL (SHELL)

The geometry is formed as an assembly which means that the suspension bracket and the beam are no longer connected.

Edge to Edge 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge to Edge**.

2 In the **Settings** window for **Edge to Edge**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Suspension Bracket (Edge)**.

4 Locate the **Edge Selection, Destination** section. From the **Selection** list, choose **Long Beam (Edge)**.

5 Locate the **Connection Settings** section. From the **Connected location, destination** list, choose **Top surface**.

6 Select the **Weld verification** checkbox.

7 Click to expand the **Equation** section. Locate the **Weld Properties** section. In the a text field, type 3[mm].

8 In the σ_{eq}^{max} text field, type 500[MPa].

9 In the $\sigma_{\perp}^{max}$ text field, type 500[MPa].

Add the upper body weight and the payload, distributed over the supporting brackets. The load is defined using the resultant and the center of gravity.

Upper Body and Payload

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 In the **Settings** window for **Edge Load**, type Upper Body and Payload in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **Upper Body Bracket**.

4 Locate the **Force** section. From the **Load type** list, choose **Resultant**.

5 Specify the **F** vector as

-2000[kg]*g_const	z

6 From the **Application point defined using** list, choose **Coordinates**.

7 Specify the $\mathbf{x}_a$ vector as

3[m]	Y
1[m]	Z

Add the load corresponding to the engine weight.

Engine

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 In the **Settings** window for **Edge Load**, type Engine in the **Label** text field.

3 Locate the **Edge Selection** section. From the **Selection** list, choose **Engine Bracket**.

4 Locate the **Force** section. From the **Load type** list, choose **Resultant**.

5 Specify the **F** vector as

-400[kg]*g_const	z

6 From the **Application point defined using** list, choose **Coordinates**.

7 Specify the $\mathbf{x}_a$ vector as

0.4[m]	Y
0.1[m]	Z

Gravity 1

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

Attachment 1

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

2 Select Edges 606–609 only.

Attachment 2

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

Attachment 2-10

Proceed to create nine additional attachments with the following settings:

Name	Edges
Attachment 2 (att2)	610-613
Attachment 3 (att3)	697, 698
Attachment 4 (att4)	303-306
Attachment 5 (att5)	307-310
Attachment 6 (att6)	910, 911
Attachment 7 (att7)	795, 796, 836, 837
Attachment 8 (att8)	797, 798, 838, 839
Attachment 9 (att9)	87, 88, 124, 125
Attachment 10 (att10)	89, 90, 126, 127

ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics > Truss (truss)**.

4 Click the **Add to Component 1** button in the window toolbar.

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

TRUSS (TRUSS)

1 In the **Settings** window for **Truss**, locate the **Edge Selection** section.

2 From the **Selection** list, choose **Truss**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Truss (truss)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `1e15`.

4 From the ν list, choose **User defined**. From the ρ list, choose **User defined**.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 647 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in x direction** list, choose **Prescribed**.

5 In the u_{0x} text field, type `shell.att1.u`.

6 From the **Displacement in y direction** list, choose **Prescribed**.

7 In the u_{0y} text field, type `shell.att1.v`.

8 From the **Displacement in z direction** list, choose **Prescribed**.

9 In the u_{0z} text field, type `shell.att1.w`.

Prescribed Displacement 2

Right-click **Prescribed Displacement 1** and choose **Duplicate**.

Prescribed Displacement 2-10

Proceed to create nine additional prescribed displacements with the following settings:

Name	**Edges**
Prescribed Displacement 2 (disp2)	648
Prescribed Displacement 3 (disp3)	649
Prescribed Displacement 4 (att4)	623
Prescribed Displacement 5 (disp5)	624
Prescribed Displacement 6 (disp6)	622
Prescribed Displacement 7 (disp7)	651
Prescribed Displacement 8 (disp8)	652

Name	Edges
Prescribed Displacement 9 (disp9)	619
Prescribed Displacement 10 (disp10)	620

Front Suspension

1 In the **Physics** toolbar, click **Edges** and choose **Spring–Damper Material**.

2 In the **Settings** window for **Spring–Damper Material**, type `Front Suspension` in the **Label** text field.

3 Select Edges 904 and 945 only.

4 Locate the **Spring–Damper** section. In the *k* text field, type `1e6[N/m]`.

Rear Suspension

1 In the **Physics** toolbar, click **Edges** and choose **Spring–Damper Material**.

2 In the **Settings** window for **Spring–Damper Material**, type `Rear Suspension` in the **Label** text field.

3 Select Edges 907, 908, 946, and 947 only.

4 Locate the **Spring–Damper** section. In the *k* text field, type `3e6[N/m]`.

Prescribed Displacement 11

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.

2 Select Edges 907, 908, 946, and 947 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in x direction** list, choose **Prescribed**.

Prescribed Displacement 12

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Points 618 and 653 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

Pinned 1

1 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 621 and 650 only.

MESH 2

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

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

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

4 Click **Build All**.

5 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

6 From the list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Mesh 2** click **Size**.

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

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

Size 1

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

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

3 From the **Selection** list, choose **Suspension Bracket**.

4 Locate the **Element Size** section. Click the **Custom** button.

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

6 Select the **Maximum element size** checkbox. In the associated text field, type `5e-3`.

Size 2

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

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

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

4 From the **Selection** list, choose **Suspension Bracket (Edge)**.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `1e-3`.

Size 3

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

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

3 From the **Selection** list, choose **Long Beam (Edge)**.

4 Click **Build All**.

ADD STUDY

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

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

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

4 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

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

RESULTS

Stress (shell)

In the **Model Builder** window, expand the **Stress (shell)** node.

Surface 1

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

2 In the **Settings** window for **Surface**, 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 `180`.

Deformation

In the **Model Builder** window, right-click **Deformation** and choose **Disable**.

Stress (shell)

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 In the **Stress (shell)** toolbar, click **Plot**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 2/Solution 2 (sol2)** > **Shell** > **Displacement (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

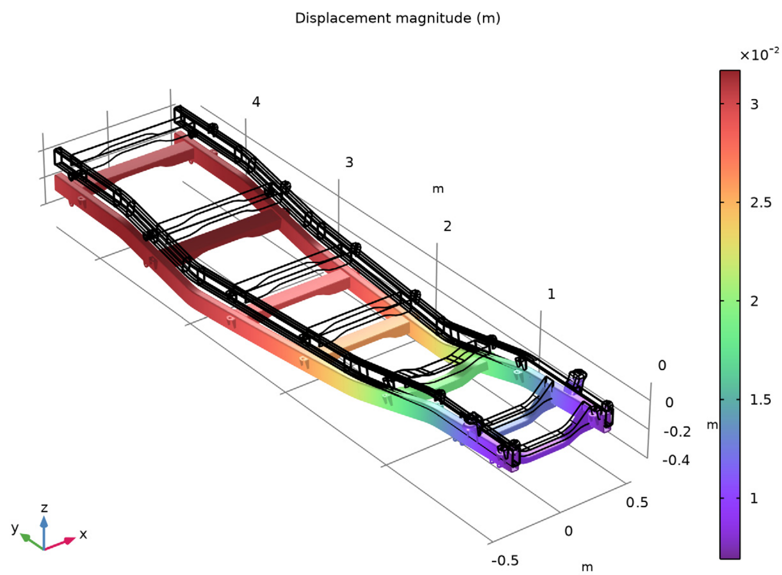
5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Displacement (shell)

1 In the **Displacement (shell)** toolbar, click **Plot**, to generate the displacement plot as in Figure 6.

2 In the **Model Builder** window, under **Results** click **Displacement (shell)**.



RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 2/Solution 2 (sol2)** > **Shell** > **Applied Loads (shell)** > **Edge Loads (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

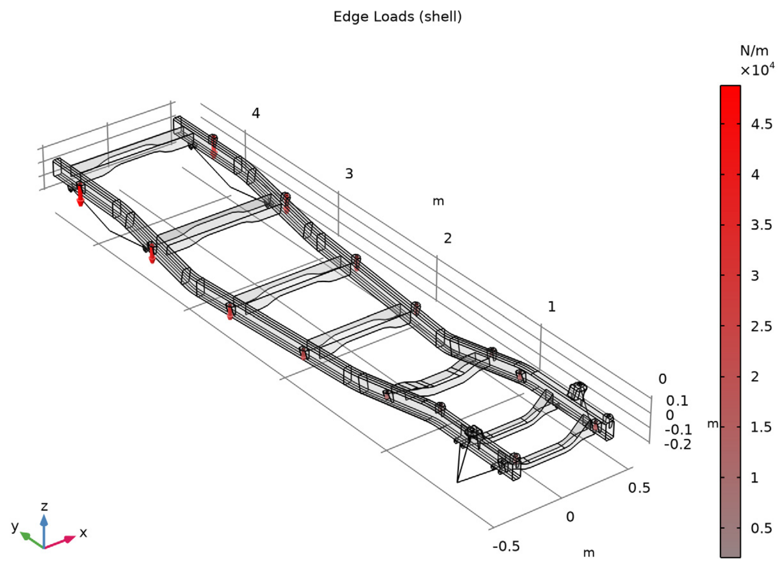
5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Edge Loads (shell)

1 In the **Edge Loads (shell)** toolbar, click **Plot**, to generate the plot as in Figure 2.

2 In the **Model Builder** window, under **Results** click **Edge Loads (shell)**.



The instructions below show how to generate Figure 8.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 2/Solution 2 (sol2)** > **Shell** > **Weld Failure Index (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Surface 1

1 In the **Model Builder** window, right-click **Weld Failure Index (shell)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

Transparency 1

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

Zoom in on the weld to reproduce the scene in Figure 8.

Finally, display the weld verification result as in Figure 9.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

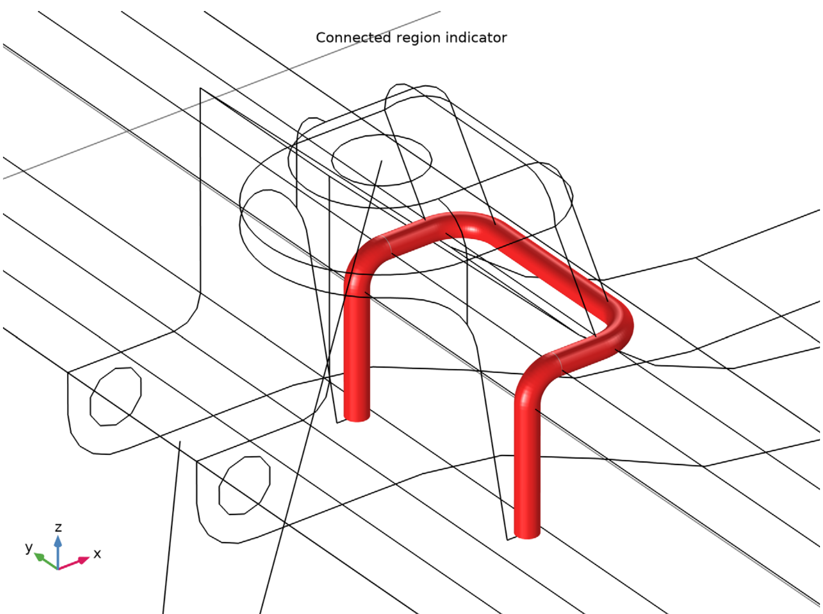
3 In the tree, select **Study 2/Solution 2 (sol2)** > **Shell** > **Connected Region Indicator (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Connected Region Indicator (shell)



1 Click the **Zoom Extents** button in the **Graphics** toolbar to see the full geometry again.

STUDY 1

To run the first study again, you need to make the following settings.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** 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** checkbox.

4 Select all nodes between **Upper body and payload** and **Attachment 10**, both included. Also, select **Truss (truss)**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)** > **Truss (truss)**.

7 Click **Disable in Model**.