

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.

Eigenfrequency=0.0014427i Hz

Shell Geometry (shell)

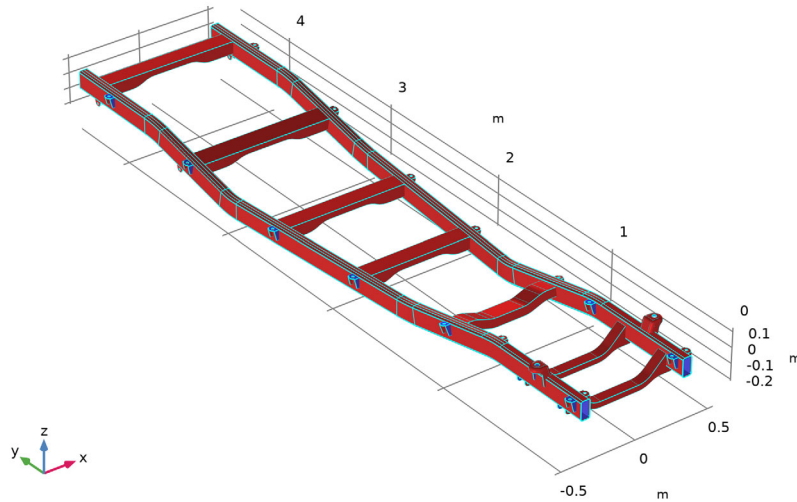


Figure 1: Shell geometry.

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.

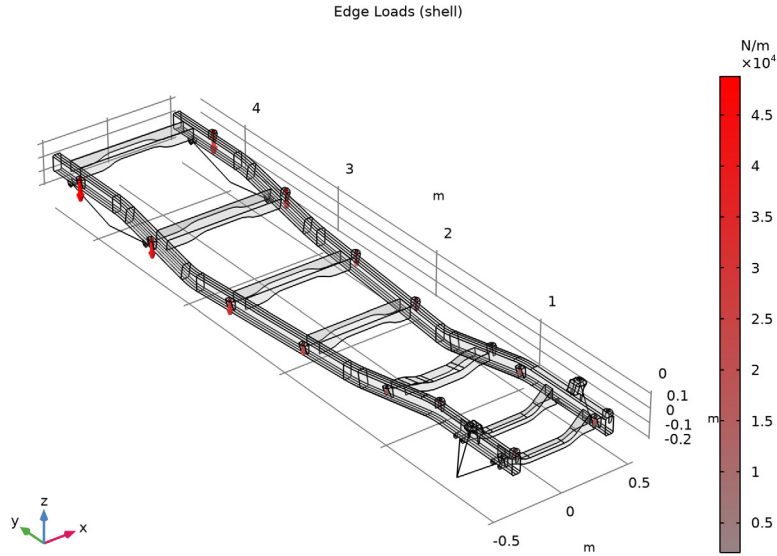


Figure 2: Load distribution.

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 (27.2 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 Hz) also to a bending mode, in the lateral direction.

Eigenfrequency=27.207 Hz

Surface: Displacement magnitude (m)

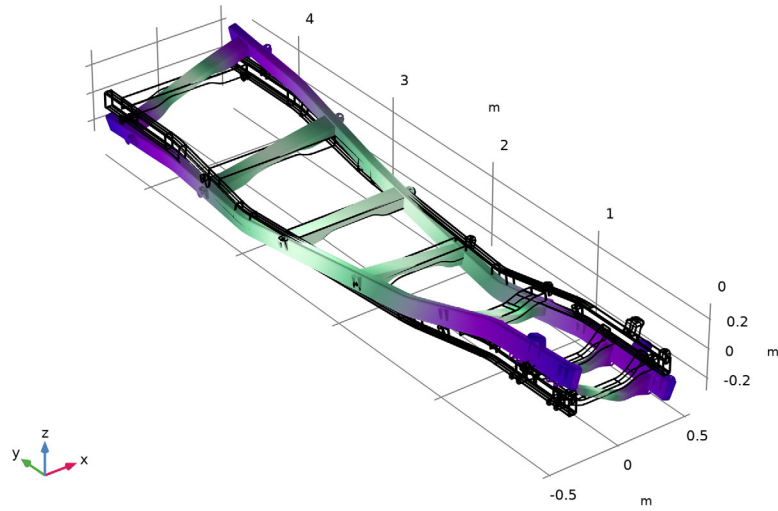


Figure 3: First natural frequency (torsion).

Eigenfrequency=30.414 Hz

Surface: Displacement magnitude (m)

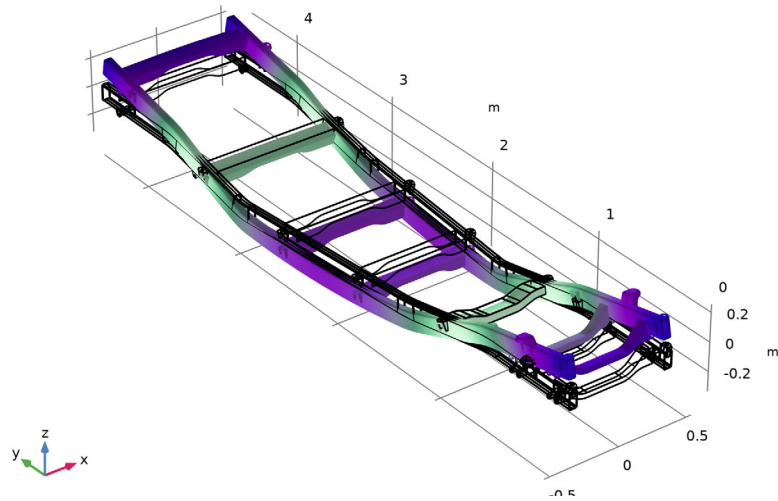


Figure 4: Second natural frequency (bending).

[

Eigenfrequency=47.05 Hz

Surface: Displacement magnitude (m)

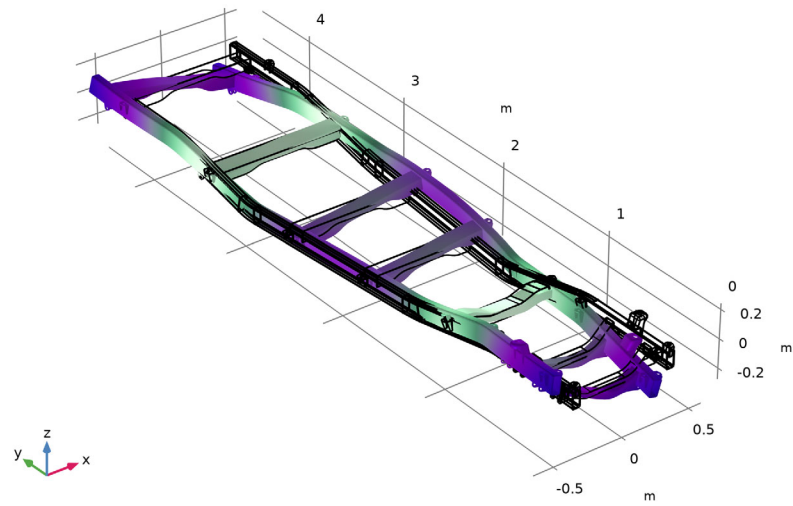


Figure 5: Third natural frequency (bending).

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

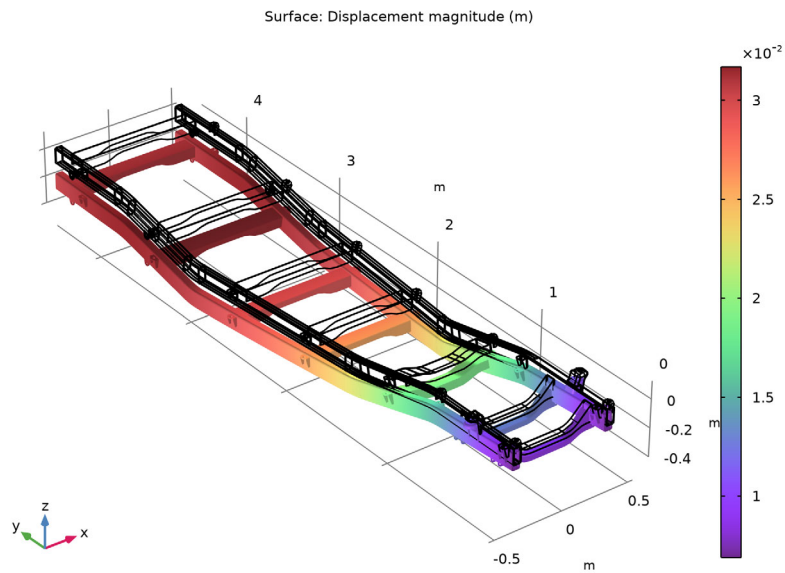


Figure 6: Chassis displacement.

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.

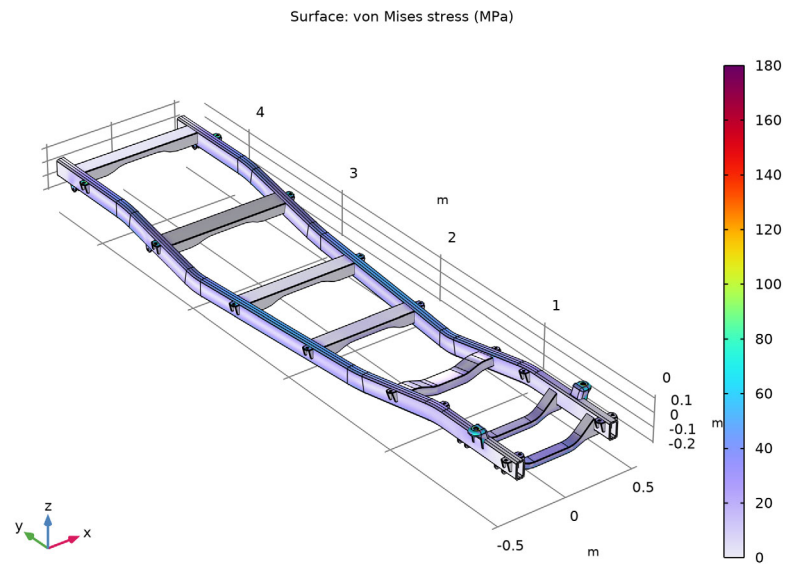


Figure 7: von Mises Stress distribution in the chassis.

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.

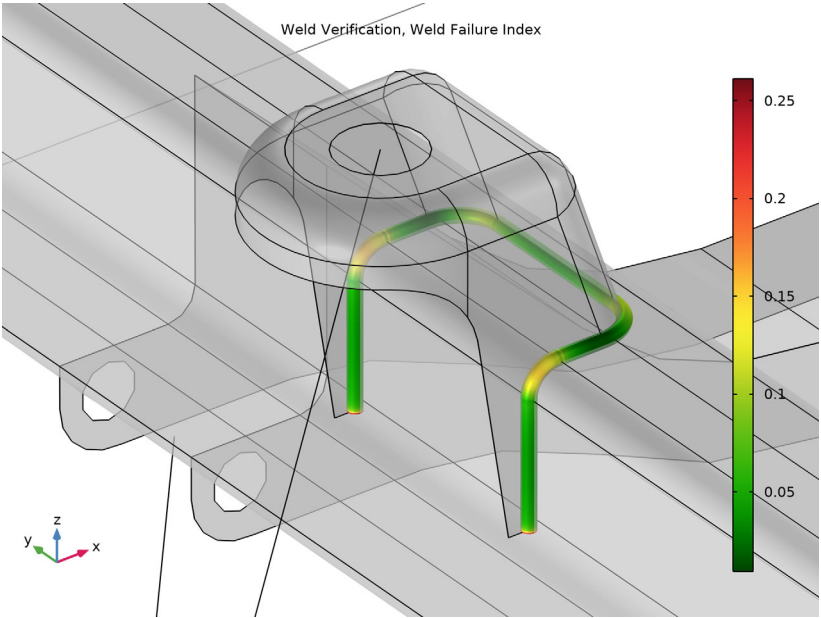


Figure 8: Weld verification.

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..

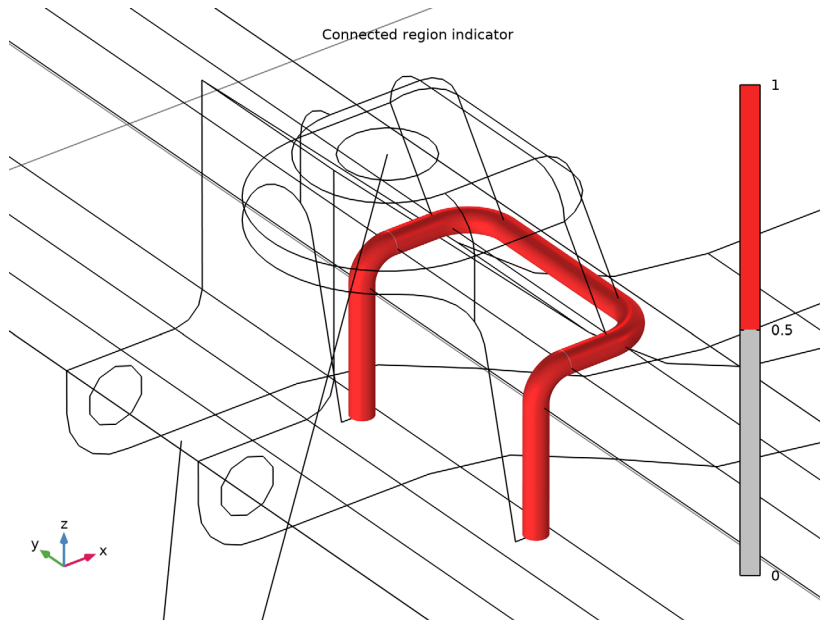



Figure 9: Connected region indicator.

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 I



- 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**.

Import 1 (imp1)


- 1 In the **Home** toolbar, click  **Import**.
- 2 In the **Settings** window for **Import**, locate the **Import** section.
- 3 Click  **Browse**.
- 4 Browse to the model's Application Libraries folder and double-click the file `ladder_frame_geom.step`.
- 5 Find the **Import options** subsection. Select the **Remove redundant edges and vertices** check box.
- 6 Click  **Build Selected**.


One way to retrieve the thickness of the beams is to use the **Distance Measurement** tools.

Distance Measurement 1 (dm1)



- 1 In the **Geometry** toolbar, click  **Measurements** and choose **Distance Measurement**.
- 2 On the object **imp1.id107**, select Point 44 only.
- 3 In the **Settings** window for **Distance Measurement**, locate the **Geometric Entity Selection** section.
- 4 Click to select the  **Activate Selection** toggle button for **Second entity**.
- 5 On the object **imp1.id107**, select Point 43 only.
- 6 Locate the **Parameter Names** section. In the **Distance** text field, type `th1`.

Distance Measurement 2 (dm2)

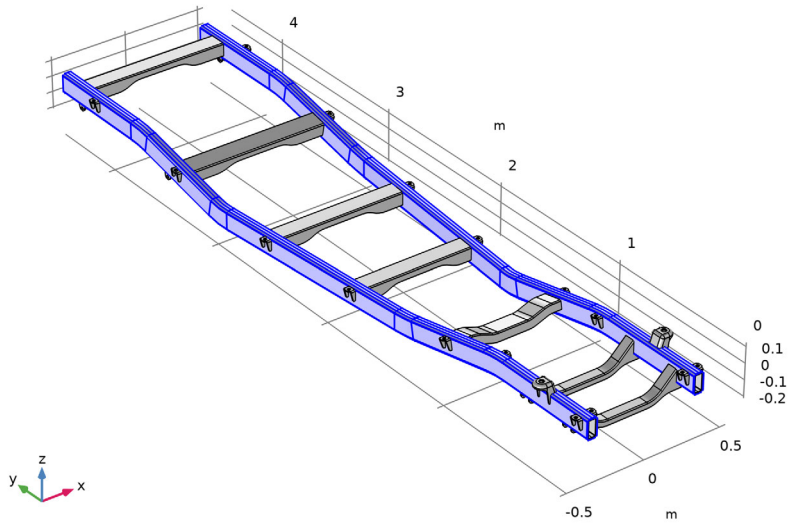
- 1 In the **Geometry** toolbar, click  **Measurements** and choose **Distance Measurement**.
- 2 On the object **imp1.id101**, select Point 65 only.
- 3 In the **Settings** window for **Distance Measurement**, locate the **Geometric Entity Selection** section.

- 4 Click to select the  **Activate Selection** toggle button for **Second entity**.
- 5 On the object **impl.id101**, select Point 67 only.
- 6 Locate the **Parameter Names** section. In the **Distance** text field, type th2.

Longitudinal members


- 1 In the **Geometry** toolbar, click  **Selections** and choose **Explicit Selection**.
- 2 In the **Settings** window for **Explicit Selection**, type Longitudinal members in the **Label** text field.
- 3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 In the **Graphics** window toolbar, click ▼ next to  **Select Boundaries**, then choose **Group by Continuous Tangent**.

- 5 In the Graphics window, select the outer boundaries of the longitudinal members as shown in the figure below:

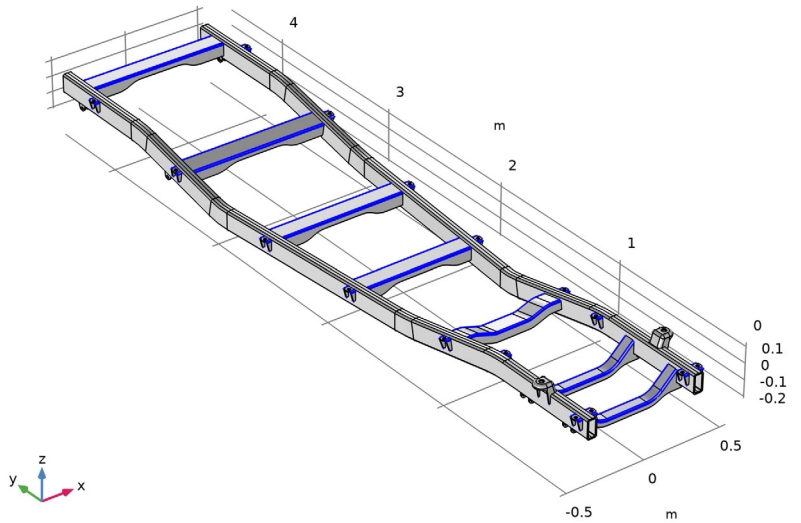


In the table below you can find the corresponding selected entities:

Entities to select
imp1.id107: 1, 3-14, 25, 28, 29, 32, 33, 36, 37, 40, 45, 46, 54-62, 64-66, 71, 72, 83-85, 92-94
imp1.id110: 3, 7, 8, 11, 12, 15, 17, 21, 24, 26, 28, 31, 32, 35, 36, 39, 40, 43, 47, 48, 51-53, 56, 58, 61, 63, 65, 76-78, 80, 81, 84-86, 93-95, 99-102
imp1.id38: 2, 4, 6, 8, 22-25, 27, 31-35, 44-47, 52, 55, 57-61, 63, 67, 70, 74, 75, 78, 79, 82, 83, 86, 88, 91, 92, 95, 96, 99, 101, 102
imp1.id41: 1-8, 10-16, 29-32, 41-44, 48, 51, 62-65, 67-72, 76, 79, 87, 90, 91, 94, 95, 98

- 6 In the **Geometry** toolbar, click  **Defeaturing and Repair** and choose **Delete Fillets**.
- 7 In the **Model Builder** window, click **Geometry I**.
- 8 In the **Tools** window for **Delete Fillets**, locate the **Delete Fillets** section.
- 9 In the **Maximum fillet radius** text field, type 7[mm].
- 10 Click **Find Fillets**.

- 11 In the Graphics window, click the fillets inside the longitudinal members to remove them from the selection list.



The following entities are now selected in the fillet selection list:



Fillet selection

5 - 84, 89 - 136

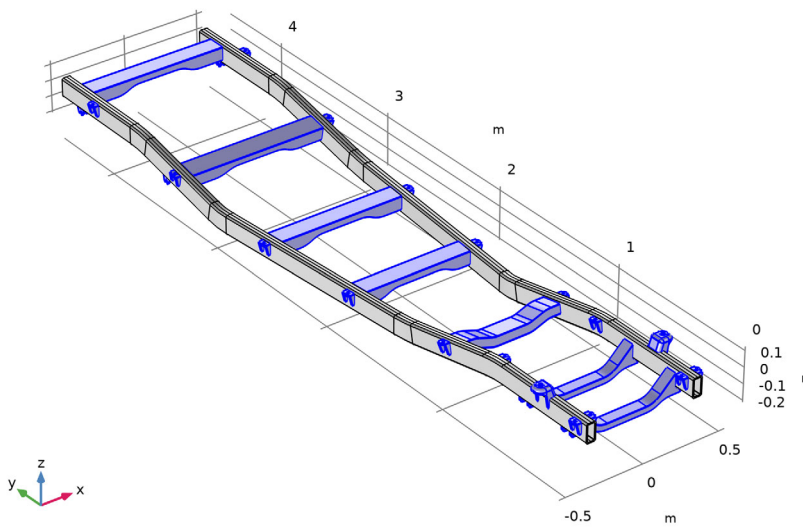
- 12 Click **Delete Selected**.

Convert the transverse beams and the brackets to a midsurface representation.

Midsurface 1 (mid1)

- 1 In the **Geometry** toolbar, click  **Conversions** and choose **Midsurface**.
- 2 Click the  **Select All** button in the **Graphics** toolbar.

- 3 In the Graphics window, click on the longitudinal members to remove them from the selection list.





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




Input objects
dfi1(10), dfi1(11), dfi1(12), dfi1(13), dfi1(14), dfi1(15), dfi1(16), dfi1(17), dfi1(18), dfi1(19), dfi1(20), dfi1(21), dfi1(22), dfi1(23), dfi1(26), dfi1(27), dfi1(28), dfi1(29), dfi1(3), dfi1(30), dfi1(31), dfi1(32), dfi1(33), dfi1(34), dfi1(35), dfi1(36), dfi1(37), dfi1(4), dfi1(5), dfi1(6), dfi1(7), dfi1(8), dfi1(9)

- 4 In the **Settings** window for **Midsurface**, click  **Build Selected**.
The longitudinal members are not represented by the midsurface but by the top surface.


Extract 1 (extract1)

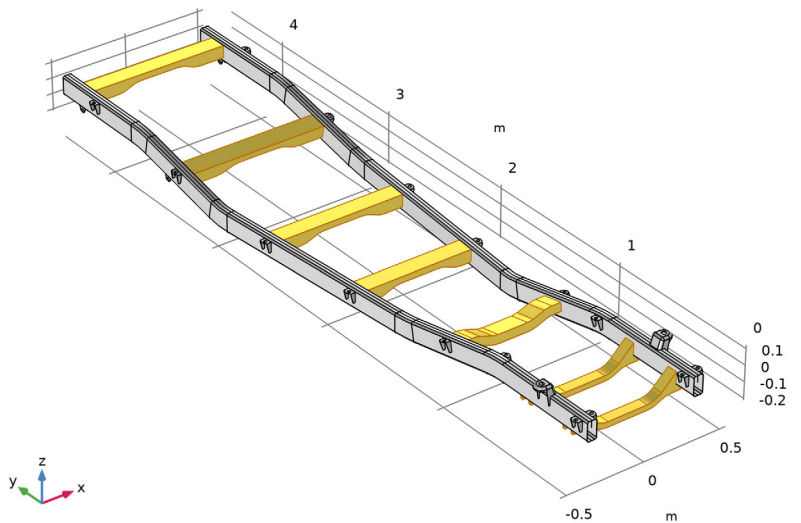
- 1 In the **Geometry** toolbar, click  **Extract**.
- 2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.
- 3 From the **Selection** list, choose **Longitudinal members**.
- 4 From the **Input object handling** list, choose **Remove**.
- 5 Click  **Build Selected**.

Union 1 (un1)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 Click the  **Select All** button in the **Graphics** toolbar.
- 3 In the **Settings** window for **Union**, click  **Build Selected**.

Cross members

- 1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.
- 2 In the **Settings** window for **Box Selection**, type **Cross members** in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Select the **Group by continuous tangent** check box.
- 5 Locate the **Box Limits** section. In the **x minimum** text field, type 0.
- 6 In the **x maximum** text field, type 0.





Clean up the geometry, removing unnecessary details.

Remove Details 1 (rmd1)

- 1 In the **Geometry** toolbar, click  **Remove Details**.
- 2 In the **Settings** window for **Remove Details**, click  **Build Selected**.

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 Right-click and choose **Add to Component 1 (comp1)**.
- 5 In the **Home** 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.

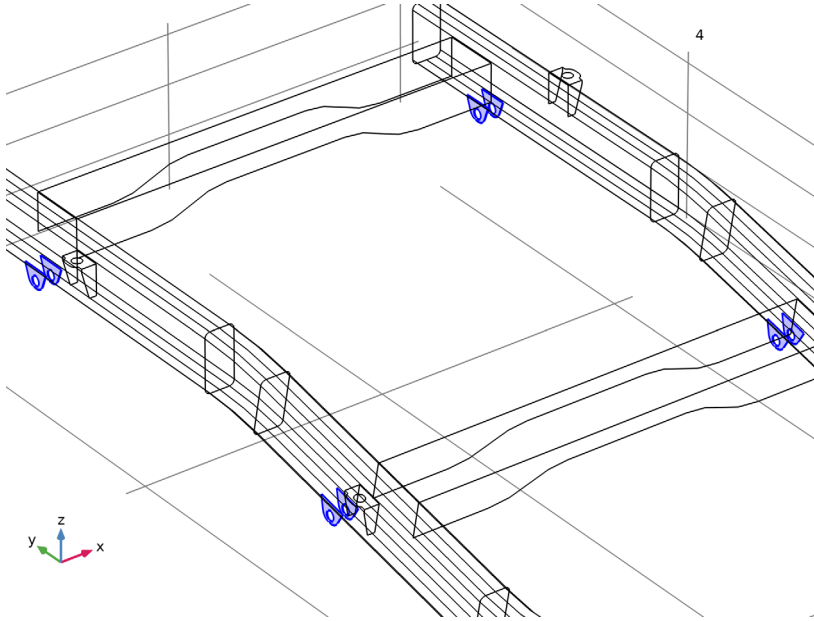
Thickness and Offset 2

- 1 In the **Model Builder** window, right-click **Shell (shell)** 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 **Finer**.
- 4 Click  **Build All**.

STUDY 1

Step 1: Eigenfrequency


As there are no constraints in the model, the first six eigenfrequencies correspond to rigid body motion. Increase the number of desired eigenfrequencies to compute the interesting ones.

- 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** check box. In the associated text field, type 9.
- 4 In the **Search for eigenfrequencies around shift** text field, type 15[Hz].
- 5 In the **Home** toolbar, click  **Compute**.

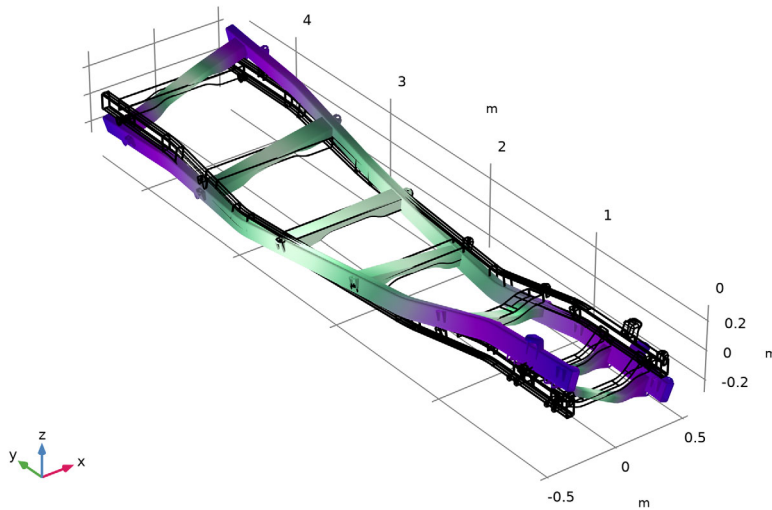
RESULTS

Mode Shape (shell)

- 1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 2 From the **Eigenfrequency (Hz)** list, choose **27.207**.
- 3 In the **Mode Shape (shell)** toolbar, click  **Plot**.

Eigenfrequency=27.207 Hz

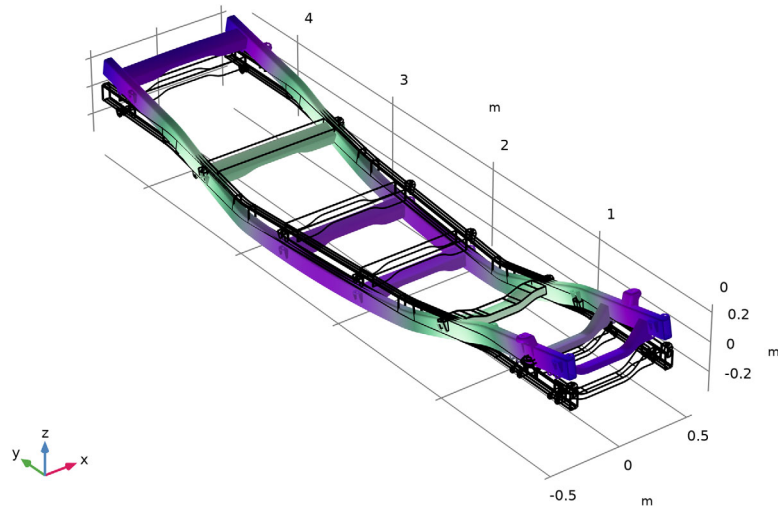
Surface: Displacement magnitude (m)



4 Click → **Plot Next.**

Eigenfrequency=30.414 Hz

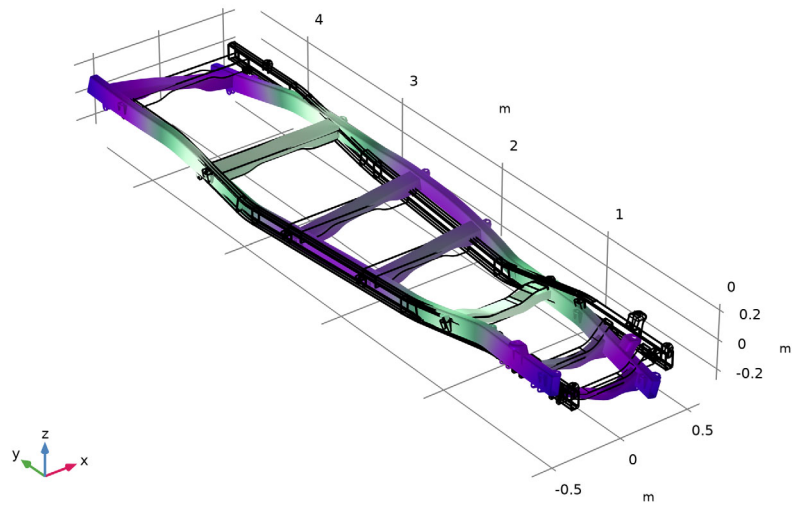
Surface: Displacement magnitude (m)



5 Click → **Plot Next.**



Eigenfrequency=47.05 Hz

Surface: Displacement magnitude (m)



Add the shell geometry plot as in [Figure 1](#).

ADD PREDEFINED PLOT

- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1)>Shell>Shell Geometry (shell)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** 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 I

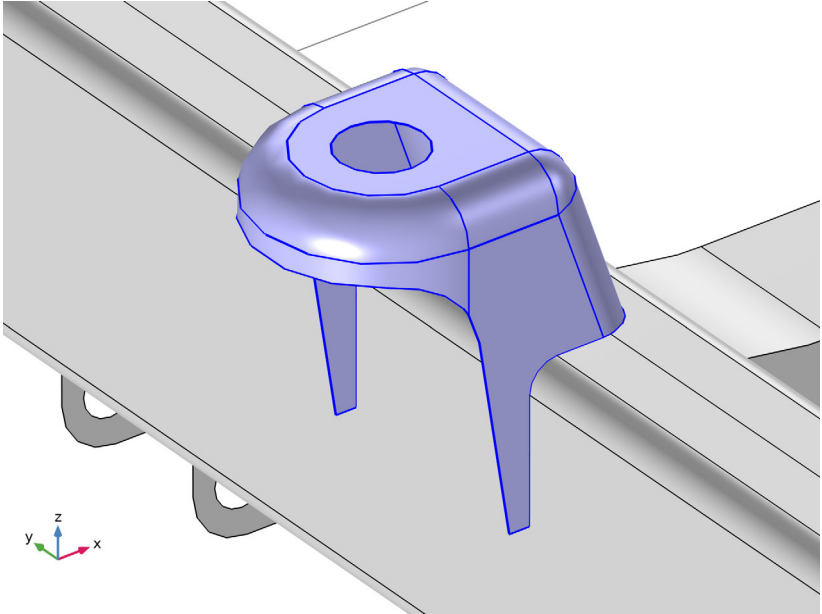
Cross members (boxsell)

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

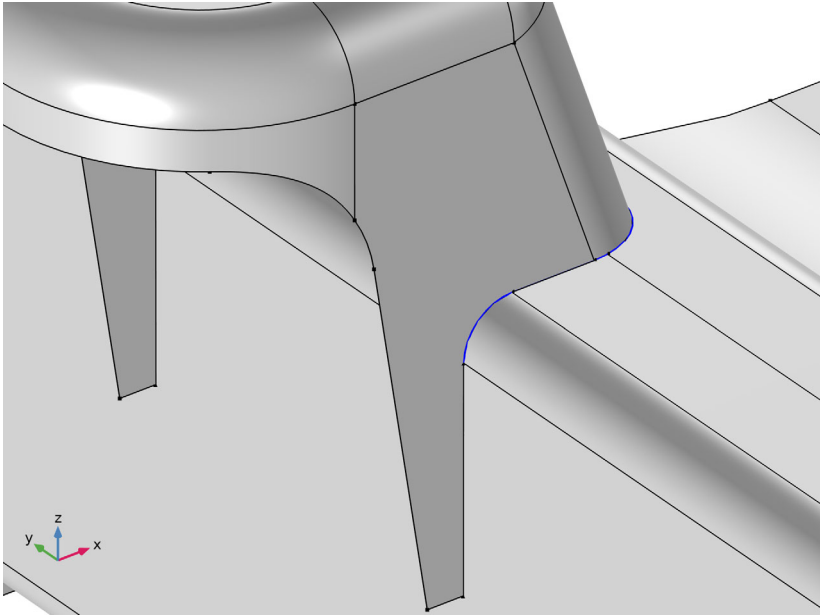
Extract 2 (extract2)

- 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 In the Graphics window, select the edges which are adjacent to the suspension bracket and connect to the longitudinal member.




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

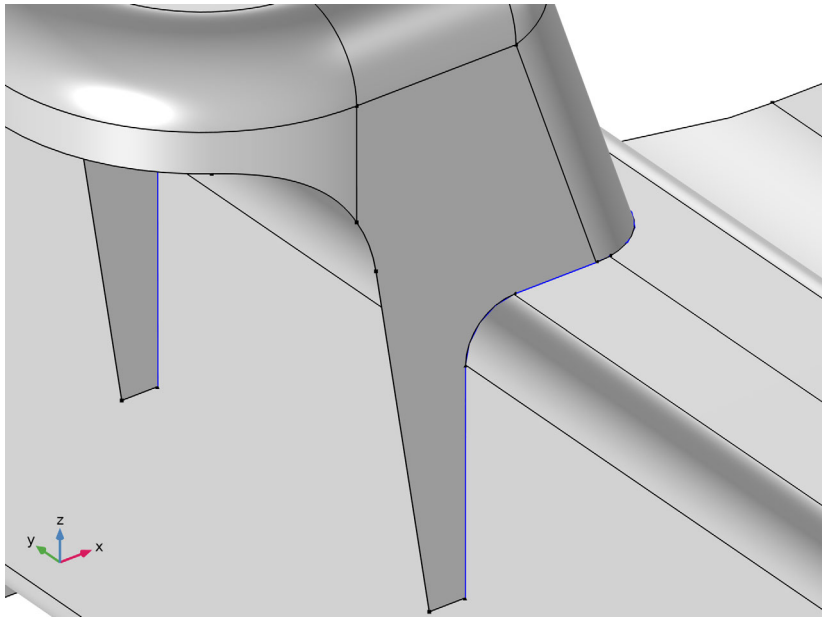
Entities to select

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 In the Graphics window, select the edges which 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 (cm)

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

2 On the object **extract2(2)**, select Points 551–554 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): 555, 556, 557, 558
Centroid Measurement 3 (cm3)	extract2(2): 614, 652
Centroid Measurement 4 (cm4)	extract2(2): 707, 713, 742, 748
Centroid Measurement 5 (cm5)	extract2(2): 708, 714, 743, 749

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_fw1`.

9 In the **y** text field, type `y_fw1`.


10 In the **z** text field, type `z_fw1`.

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)	<code>geom1.cm2.x</code> , <code>geom1.cm2.y</code> , <code>geom1.cm2.z</code>	<code>x_fw1</code> , <code>y_fw1</code> , <code>z_fw1</code>
Line Segment 3 (ls3)	<code>geom1.cm3.x</code> , <code>geom1.cm3.y</code> , <code>geom1.cm3.z</code>	<code>x_fw1</code> , <code>y_fw1</code> , <code>z_fw1</code>
Line Segment 4 (ls4)	<code>geom1.cm4.x</code> , <code>geom1.cm4.y</code> , <code>geom1.cm4.z</code>	<code>x_rw1</code> , <code>y_rw1</code> , <code>z_rw1</code>
Line Segment 5 (ls5)	<code>geom1.cm5.x</code> , <code>geom1.cm5.y</code> , <code>geom1.cm5.z</code>	<code>x_rw1</code> , <code>y_rw1</code> , <code>z_rw1</code>

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 box, 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** check box.
- 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)

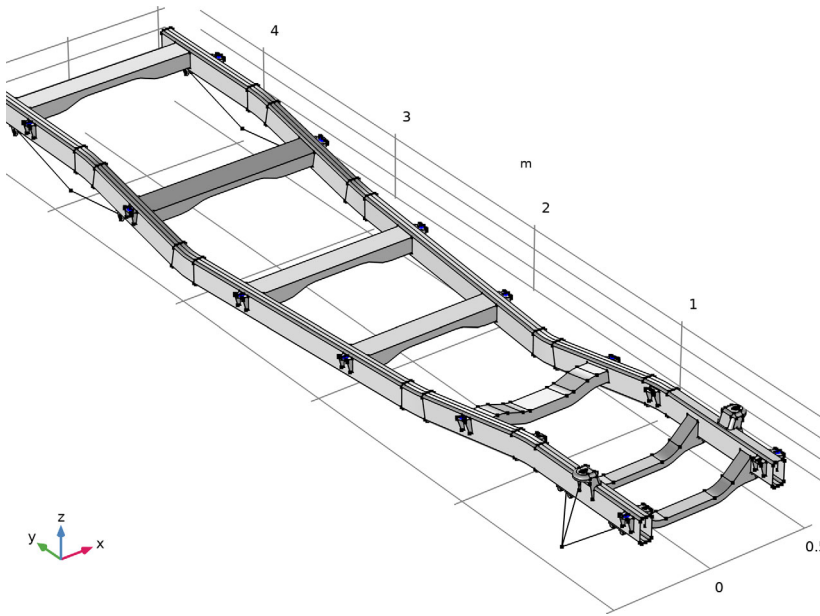
- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
- 2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
- 3 From the **Action** list, choose **Form an assembly**.

DEFINITIONS

Upper body bracket

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Upper body bracket** in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.
- 4 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.



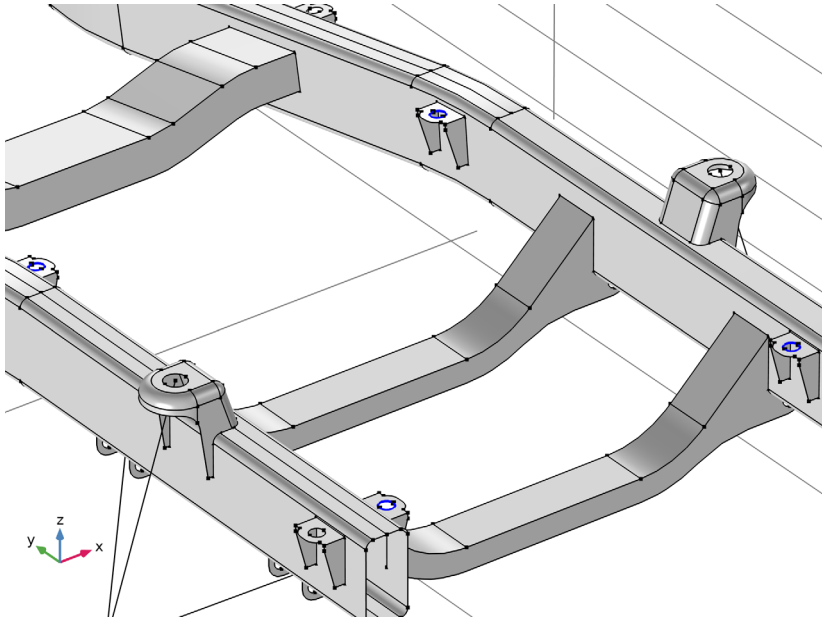
In the table below you can find the corresponding selected entities:

Entities to select
17, 18, 21, 22, 58, 59, 181, 182, 205, 206, 222, 223, 700, 701, 706, 707, 735, 736, 824, 825, 886, 887, 892, 893

Engine bracket

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type *Engine bracket* in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

- 4 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
411, 412, 417, 418, 483, 484, 497, 498

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 I

- 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** check box.

- 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 σ_{\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.

3 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, 611, 612, 613
Attachment 3 (att3)	697, 698
Attachment 4 (att4)	303, 304, 305, 306
Attachment 5 (att5)	307, 308, 309, 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 **Physics** 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 **Add to Component 1** in the window toolbar.

5 In the **Physics** 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 $1\text{e}15$.
- 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`.
- 10 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
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 $1\text{e}6[\text{N/m}]$.


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 $3\text{e}6[\text{N/m}]$.

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**, click  **Build All**.
- 3 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

- 4 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** check box. 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** check box. In the associated text field, type 1e-3.
- 8 Right-click **Size 2** and choose **Duplicate**.

Size 3

- 1 In the **Model Builder** window, click **Size 3**.
- 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 table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

3 In the **Home** 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** check box.


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**.

ADD PREDEFINED PLOT

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

2 Go to the **Add Predefined Plot** window.

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

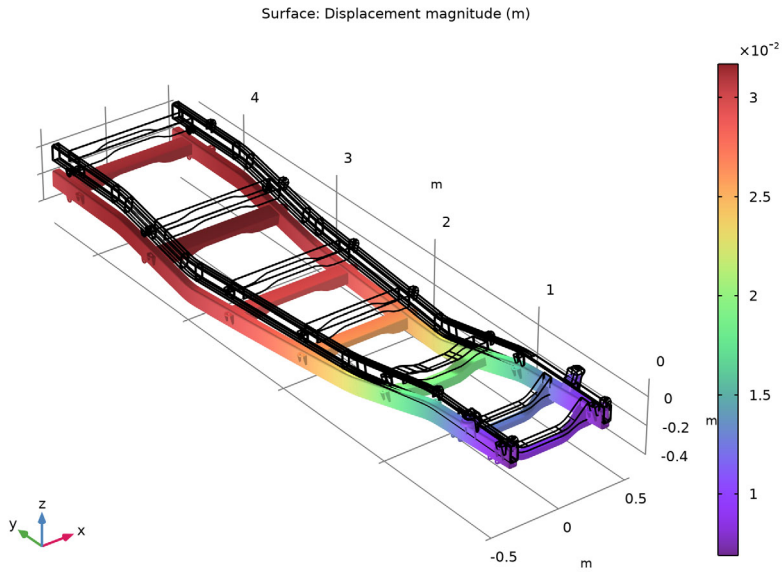
4 Click **Add Plot** in the window toolbar.

- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.


RESULTS

Displacement (shell)

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



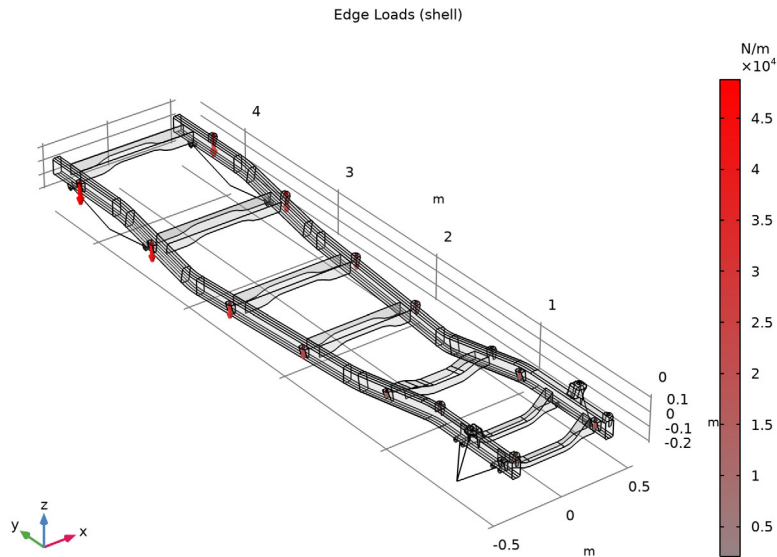
ADD PREDEFINED PLOT

- 1 Go to the **Add Predefined Plot** window.
- 2 In the tree, select **Study 2/Solution 2 (sol2)>Shell>Applied Loads (shell)>Edge Loads (shell)**.
- 3 Click **Add Plot** in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS


Edge Loads (shell)

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



The instructions below show how to generate [Figure 8](#).

ADD PREDEFINED PLOT

- 1 Go to the **Add Predefined Plot** window.
- 2 In the tree, select **Study 2/Solution 2 (sol2)>Shell>Weld Failure Index (shell)**.
- 3 Click **Add Plot** in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Surface 1


- 1 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 I

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

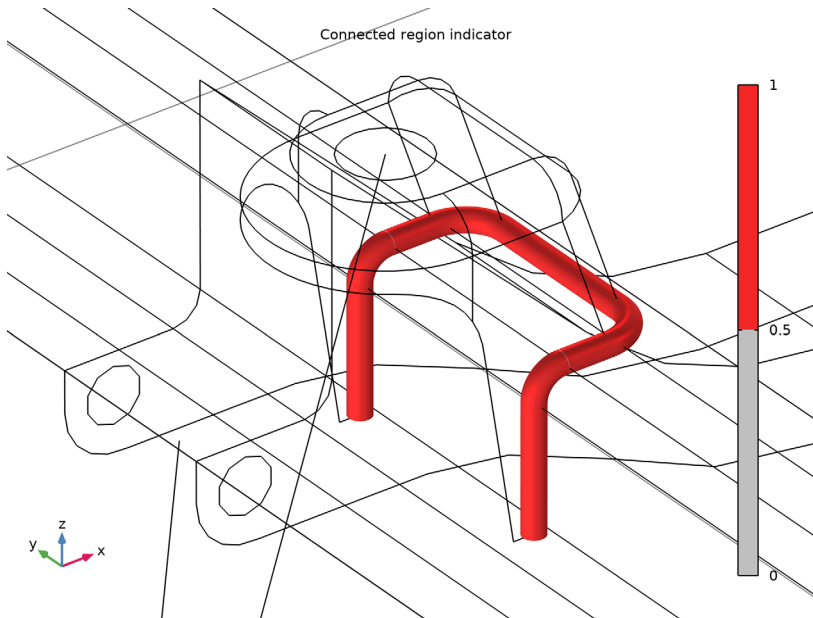
Finally, display the weld verification result as in [Figure 9](#).

ADD PREDEFINED PLOT

- 1 Go to the **Add Predefined Plot** window.
- 2 In the tree, select **Study 2/Solution 2 (sol2)>Shell>Connected Region Indicator (shell)**.
- 3 Click **Add Plot** in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Connected Region Indicator (shell)





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** check box.
- 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**.