



Model created in COMSOL Multiphysics 6.4

Ladder Frame Geometry

Introduction

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 defeaturing tools that can be applied to delete geometric details that are not relevant for the analysis. This example generates the shell geometry of a ladder frame, or ladder chassis, from an imported solid structure for the eigenfrequency and static analyses described in the *Ladder Frame* model located in the Structural Mechanics Module Application Library.

Model Definition

The ladder frame geometry is imported from a STEP file as solid objects. The step-by-step instructions demonstrate how to generate a shell model from the imported solids.

Read on to find out more about the following topics:


- How to convert an imported CAD geometry into a shell representation using the Midsurface and Extract features.
- How to generate parameters from the dimensions of the geometry using Distance Measurement and Centroid Measurement features.

Application Library path: Design_Module/Tutorial_Examples/
ladder_frame_geometry

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click  **Blank Model**.

ADD COMPONENT




In the **Home** toolbar, click  **Add Component** and choose **3D**.

GEOMETRY I

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



- 2 From the **Geometry representation** list, choose **CAD kernel**.
- 3 Select the **Design Module Boolean operations** checkbox.

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 `ladder_frame_geom.step`.
- 5 Locate the **Simplify and Repair** section. Select the **Remove redundant edges and vertices** checkbox.
- 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 **impl.id5**, 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 **impl.id5**, select Point 43 only.
- 6 Locate the **Parameter Names** section. In the **Distance** text field, type `th1`.

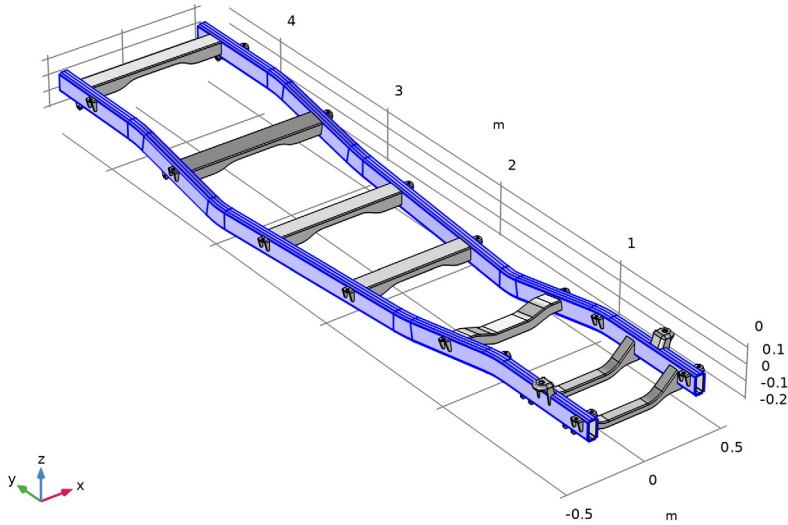
Distance Measurement 2 (dm2)


- 1 In the **Geometry** toolbar, click  **Measurements** and choose **Distance Measurement**.
- 2 On the object **impl.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

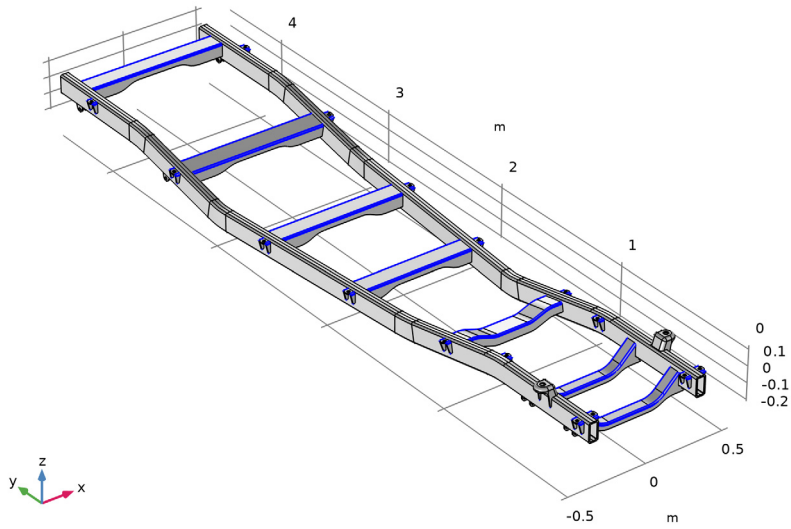
- 1 In the **Geometry** toolbar, click  **Selections** and choose **Explicit Selection**.
- 2 In the **Settings** window for **Explicit Selection**, type `Longitudinal` members in the **Label** text field.

- 3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select the **Group by continuous tangent** checkbox.
- 5 In the **Graphics** window, select the outer boundaries of the longitudinal members. By using **Group by continuous tangent**, it is sufficient to select one face for each object. The contributing boundaries are shown in the figure below:



- 6 In the **Geometry** toolbar, click  **Defeaturing and Repair** and choose **Delete Fillets**. All objects are automatically added to the **Input objects** list.
- 7 In the **Model Builder** window, click **Geometry I**.
- 8 Remove the longitudinal members *impl.id5*, *impl.id8*, *impl.id92*, and *impl.id95* from the list.
- 9 In the **Tools** window for **Delete Fillets**, locate the **Delete Fillets** section.
- 10 In the **Maximum fillet radius** text field, type 7 [mm].

11 Click **Find Fillets**.




12 Click **Delete All**.

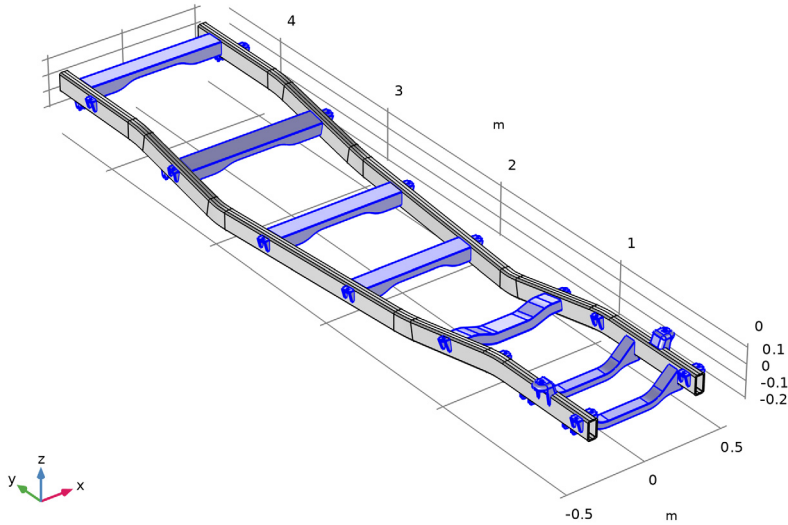
Convert the transverse beams and the brackets to a midsurface representation.

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(1) dfi1(2) dfi1(3) dfi1(4) dfi1(5) dfi1(6) dfi1(7) dfi1(8)
dfi1(9) 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(24) dfi1(25) dfi1(26) dfi1(27) dfi1(28) dfi1(29)
dfi1(30) dfi1(31) dfi1(32) dfi1(33)

- 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**.

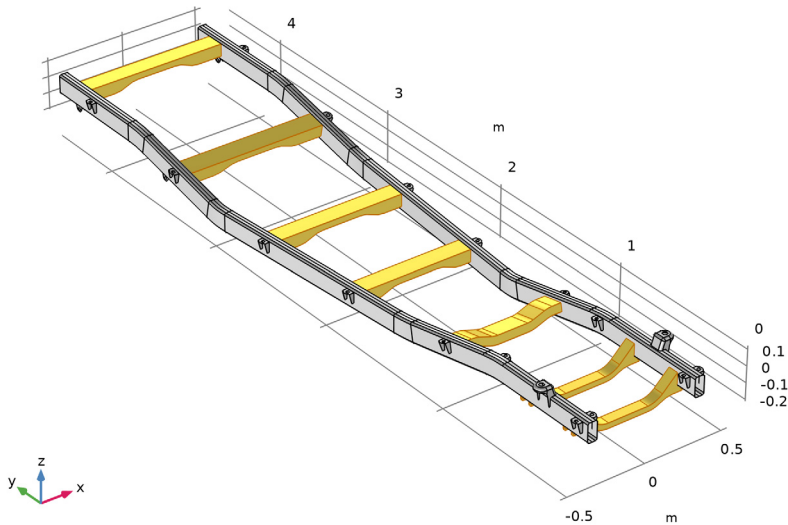
- 5 Click  **Build Selected**.

Union 1 (uni1)

- 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** checkbox.
- 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  **Virtual Operations** and choose **Remove Details**.
- 2 In the **Settings** window for **Remove Details**, click  **Build Selected**.