Model created in COMSOL Multiphysics 6.4

Spot Weld Connection in a Double Hat Beam

Introduction

A double hat beam is a commonly used structure in vehicle body construction, because of its light weight, its resistance to bending, and its ability to absorb energy. Such a beam is made of two thin-walled profiles with a hat shape, connected using spot welds. The number of spot welds, the nugget diameter, as well their locations can highly affect the global stiffness of the structure. In this example, the natural frequencies of a double hat beam made of steel are studied for different sets of spot welds locations.

This model serves as an example of how to model spot welds in COMSOL Multiphysics.

Model Definition

The beam geometry is made of two single hat-shaped plates with the following dimensions:

TABLE 1: SINGLE HAT PLATE GEOMETRY DEFINITIONS.

DIMENSION	TOP HAT PLATE	BOTTOM HAT PLATE
Length (mm)	500	500
Width (mm)	150	150
Flange width (mm)	28	28
Height (mm)	40	20
Hat width (mm)	75	80
Thickness (mm)	1.5	1.5

This model uses shell elements, so the geometry is only represented as embedded 3D faces representing the midsurface of the full geometry as shown in Figure 1.

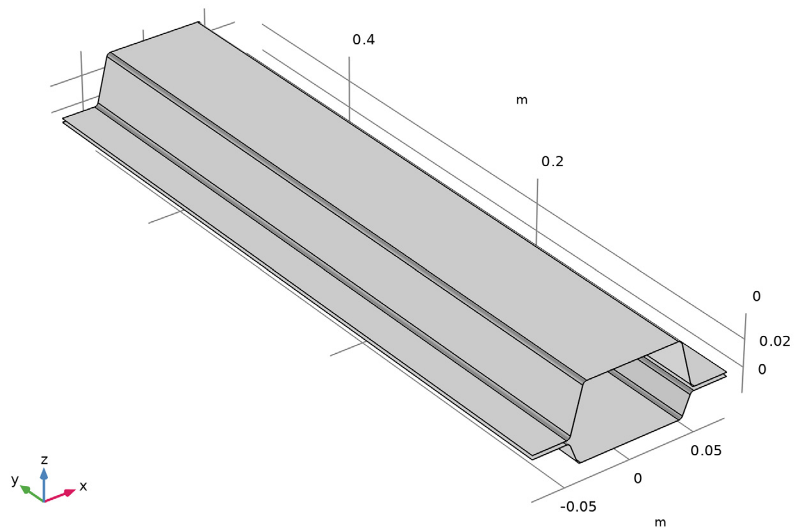

Figure 1: Double hat beam geometry.

The plates are connected on both sides of the flanges by ten spot welds with a diameter of 6 mm. The spot welds are equally spaced along the beam (the y direction) with a distance of 50 mm. In a first step, all the nuggets are aligned in the x direction. In a second step, an offset parameter is introduced to generate a misalignment in the x direction.

The built-in material Structural Steel is used to provide the material properties.

No constraints are applied to the structure. That is, free-free modes are considered. The rigid body modes are, however, not computed.

The mesh, shown in Figure 2, consists of a structured quad mesh, defined with a maximum element size of 6 mm.

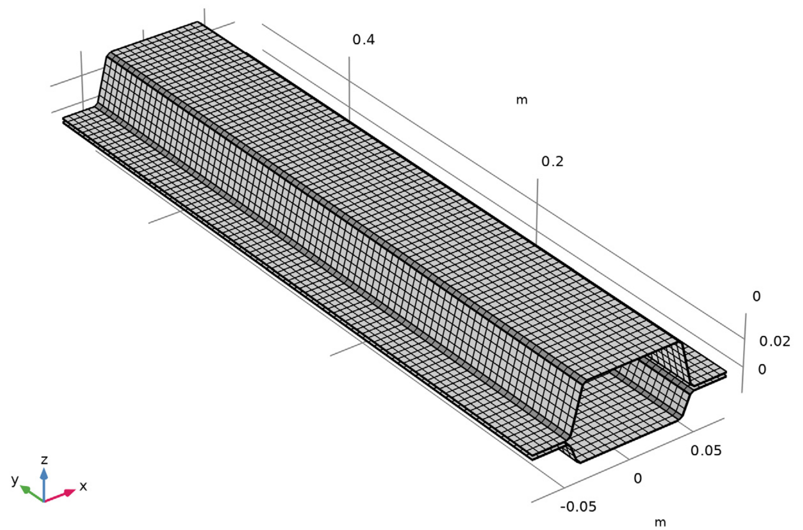

Figure 2: Mesh.

The mesh is not adapted to the locations of the spot welds.

Results and Discussion

The eigenfrequencies of the beam are summarized in Table 2 below. Figure 3 and Figure 4 show the corresponding mode shapes.

TABLE 2: NATURAL FREQUENCIES.

MODE NUMBER	FREQUENCY (HZ), NO OFFSET	FREQUENCY (HZ), 4 MM OFFSET
1	384	410
2	391	423
3	431	433
4	456	460
5	731	738
6	876	878

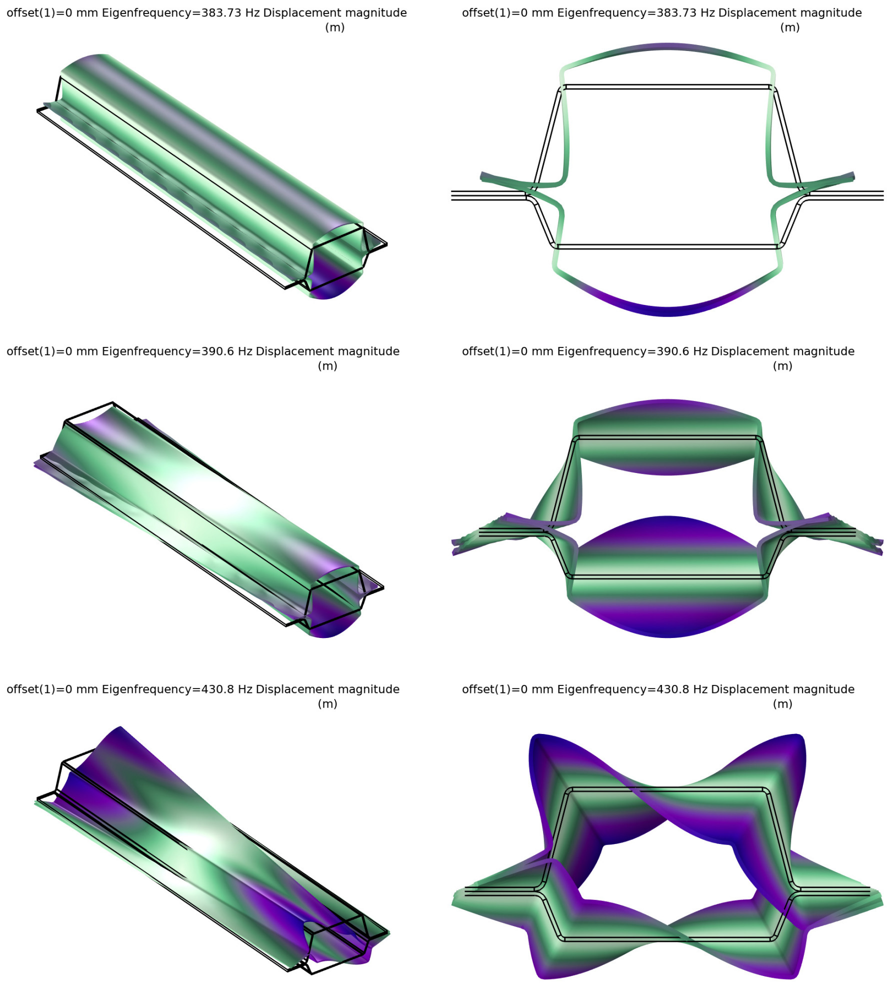

Figure 3: Mode shapes (mode 1 to 3).

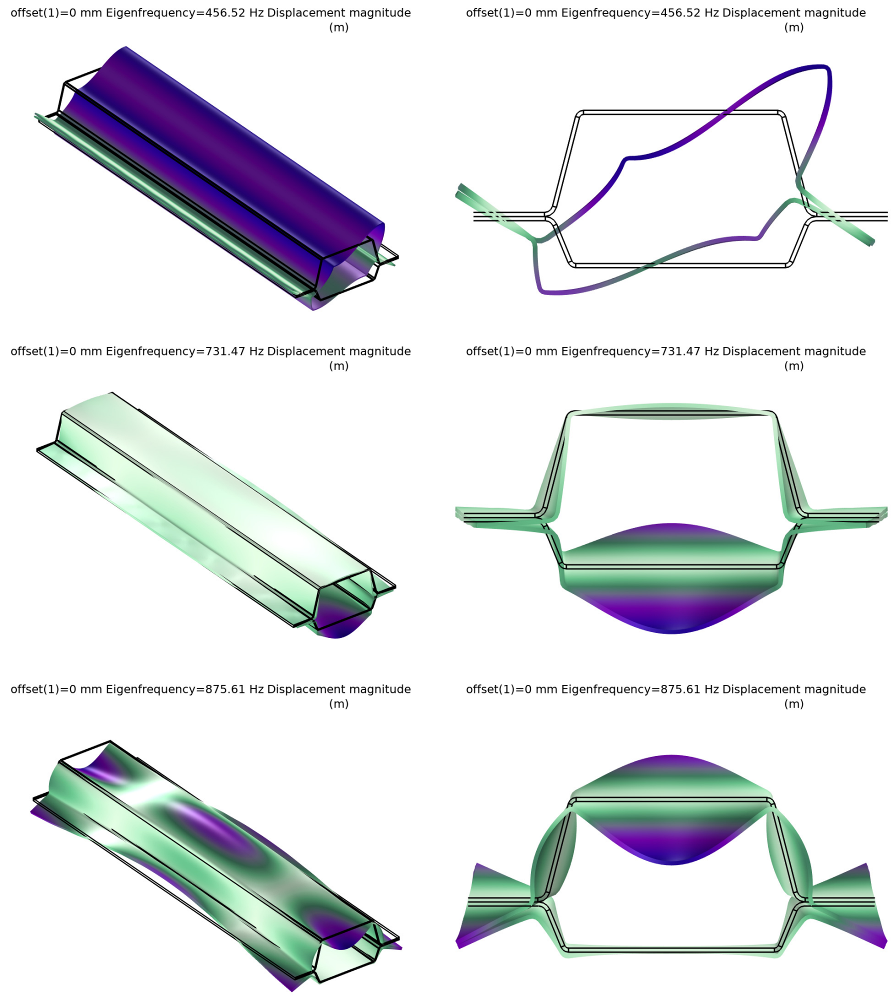

Figure 4: .Mode shapes (mode 4 to 6).

It can be noticed that the nugget position affects mainly the two first modes. The two first frequencies increase significantly with the offset, while the other ones show little change. Increasing the offset even more would even cause a swap in the ordering of modes 2 and 3.

Notes About the COMSOL Implementation

The **Spot Welds** feature for shell connections creates cohesive forces at the specified nugget location. The nugget is defined by its center coordinates and diameter. The coordinates can be stored in a text file and imported into the model. The location is independent of the mesh, so you do not need to regenerate a mesh after adding, or changing, a weld location. The given location does not need to be exactly on the destination surface, as it is automatically projected to the nearest destination boundary.

It is recommended that the element size around the weld locations is not larger than the nugget diameter. You can however use even larger elements, and still maintain a reasonable accuracy by manually adjusting the number of integration points. This will ensure proper force and moment transmission. The integration order is controlled in the **Quadrature Settings** section in the **Spot Welds** node.

Figure 5 shows the nugget location and the active integration points together with the mesh on the destination boundaries for both cases, with and without offset. This plot can be used for model checking purposes and is accessible from the results templates.

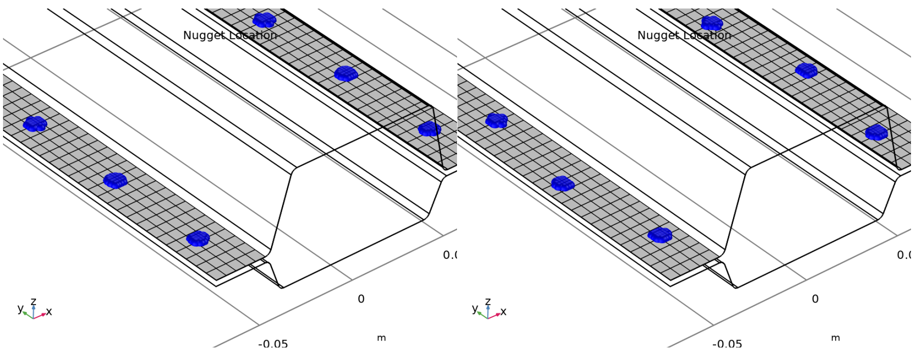

Figure 5: Active integration point at the nugget locations without offset (left) and with 4 mm offset (right).

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/double_hat_spot_welds`

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**.

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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `double_hat_spot_welds_parameters.txt`.

PART 1

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts** > **2D Part**.
2 In the **Settings** window for **Part**, locate the **Input Parameters** section.
3 In the table, enter the following settings:

Name	Default expression	Value	Description
L	l1	0.075 m	Top beam width
H	h1	0.04 m	Beam height

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.

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

x (m)	**y (m)**
`-width/2`	`th`
`-width/2+flange`	`th`
`-L/2`	`H-th`
`L/2`	`H-th`
`width/2-flange`	`th`
`width/2`	`th`

5 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **pol1**, select Points 2–5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `fillet`.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Part Instance 1 (pi1)

In the **Work Plane** toolbar, click **Part Instance** and choose **Part 1**.

Work Plane 1 (wp1) > Part Instance 2 (pi2)

1 In the **Work Plane** toolbar, click **Part Instance** and choose **Part 1**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
L	`l2`	0.08 m	Top beam width
H	`h2`	0.02 m	Beam height

4 Locate the **Position and Orientation of Output** section. In the **Rotation angle** text field, type 180.

5 Click **Build Selected**.

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
length

4 Select the **Reverse direction** checkbox.

5 Click **Build All Objects**.

6 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 Click the **Add to Component** button in the window toolbar.

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 th.

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

Spot Welds 1

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

2 Select Boundaries 1 and 17 only.

3 In the **Settings** window for **Spot Welds**, locate the **Boundary Selection, Destination** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 2 and 18 only.

6 Locate the **Nugget Location** section. Click **Load from File**.

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

8 Locate the **Nugget Properties** section. In the d_{nug} text field, type `d_nugget`.

MESH 1

Mapped 1

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

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `d_nugget`.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

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 **Study Settings** section.

3 In the **Search for eigenfrequencies around shift** text field, type `200[Hz]`.

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

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

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
offset	0[mm]	0 m	Nugget position offset

SHELL (SHELL)

Spot Welds 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Spot Welds 1**.
2. In the **Settings** window for **Spot Welds**, locate the **Nugget Location** section.
3. In the table, enter the following settings:

Nugget	X (m)	Y (m)	Z (m)
1	-62.5e-3+offset	25e-3	0
2	-62.5e-3-offset	75e-3	0
3	-62.5e-3+offset	125e-3	0
4	-62.5e-3-offset	175e-3	0
5	-62.5e-3+offset	225e-3	0
6	-62.5e-3-offset	275e-3	0
7	-62.5e-3+offset	325e-3	0
8	-62.5e-3-offset	375e-3	0
9	-62.5e-3+offset	425e-3	0
10	-62.5e-3-offset	475e-3	0
11	62.5e-3-offset	25e-3	0
12	62.5e-3+offset	75e-3	0
13	62.5e-3-offset	125e-3	0
14	62.5e-3+offset	175e-3	0
15	62.5e-3-offset	225e-3	0
16	62.5e-3+offset	275e-3	0
17	62.5e-3-offset	325e-3	0
18	62.5e-3+offset	375e-3	0
19	62.5e-3-offset	425e-3	0
20	62.5e-3+offset	475e-3	0

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
offset (Nugget position offset)	0 4	mm

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

RESULTS

Shell

1 In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Shell**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Eigenfrequencies (Study 1)

1 In the **Model Builder** window, under **Results** click **Eigenfrequencies (Study 1)**.

2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 In the **Eigenfrequencies (Study 1)** toolbar, click **Evaluate**.

RESULT TEMPLATES

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

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

3 In the tree, select **Study 1/Parametric Solutions 1 (sol2)** > **Shell** > **Nugget Location (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.