Created in COMSOL Multiphysics 5.6

Connecting Beams and Solids

Introduction

The selection of element types for modeling different parts of a structure depends on a variety of factors. Even though solid elements have the advantage of representing the behavior of any structure in detail, their use increases the number of degrees of freedom compared to other structural elements, which often results in significantly higher computational efforts. Under such circumstances, instead of modeling the entire structure using solid elements, you can use a combination of different elements by employing certain engineering assumptions. In order to model slender structures with significant bending and torsional stiffness, use of beam elements is a good alternative. In a structure having solid and beam elements, it is essential to couple these two element types accurately.

This example demonstrates the use of the **Solid-Beam Connection** multiphysics coupling in order to create a transition between a domain modeled with solid elements and an edge modeled with beam elements. Two different connection types are discussed, and a comparison of the stress distribution at the interface is performed.

Model Definition

The model consists of an I-section beam with a circular hole through its flange. The beam is fixed at one end and subjected to different point loads at the free end. In this example, you model the beams using two different structural elements. While the presence of a circular hole in the first beam necessitates the use of solid elements, the outer part of the beam can be modeled using beam elements, as s shown in Figure 1.

The transition between the two parts of the beam is created using the **Solid-Beam Connection** multiphysics coupling. In order to compare the two different solid-beam connection types, two instances of the solid-beam assembly with different types of couplings are used. The **Solid boundaries to beam points, transition** connection type is used to model the transition in the first instance, and the **Solid boundaries to beam points, general** connection type is used in the second one. The warping of the cross section, which is only present in the solid elements is computed by solving for the special warping variables in a separate study. Additional details about these connection types can be found in the documentation for *Couplings Between Structural Mechanics Interfaces* in the *Structural Mechanics Module User's Guide*.

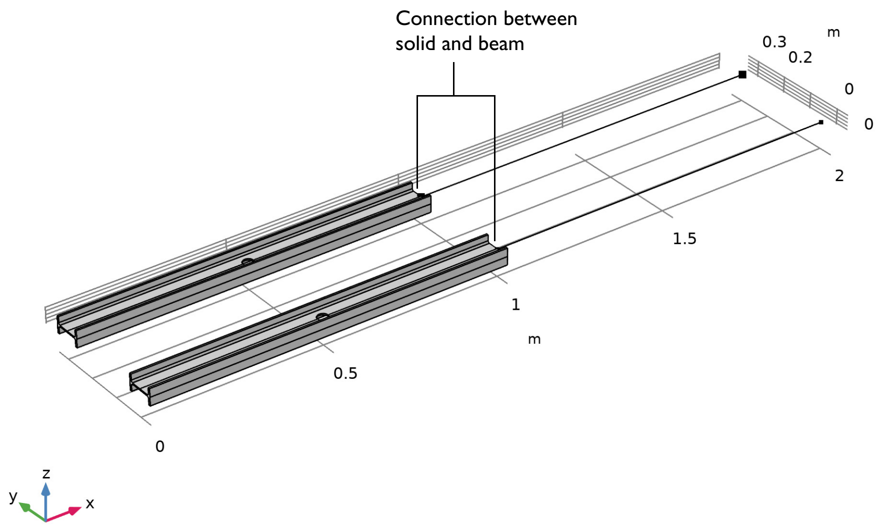


Figure 1: The geometry used for demonstrating two different solid-beam connections.

GEOMETRY

- Length of each beam: 2 m. The transition is at the midpoint.
- Cross section of the beam: European standard I-section beam (IPE 80) of 80-by-46 mm with 3.8 mm web thickness and 5.2 mm flange thickness.
- Radius of the circular hole: 20 mm.

MATERIAL

The material is Structural Steel with the following data:

- Young's modulus, $E = 200$ GPa.
- Poisson's ratio, $\nu = 0.33$.

CONSTRAINTS

The inner end of the solid modeled beam is fixed.

LOAD

In order to compare the behavior of structure under various loading conditions, six different load cases are analyzed separately. A unit point load (as force or moment) is applied at the free end of the beam in three different directions.

Results and Discussion

The von Mises stress distribution for the load case with moment about the X-axis (the "twisting moment" load case) is plotted and compared for both connection types. As shown in Figure 2, the stress distribution seems to be comparable for both cases except near the solid-beam connections.

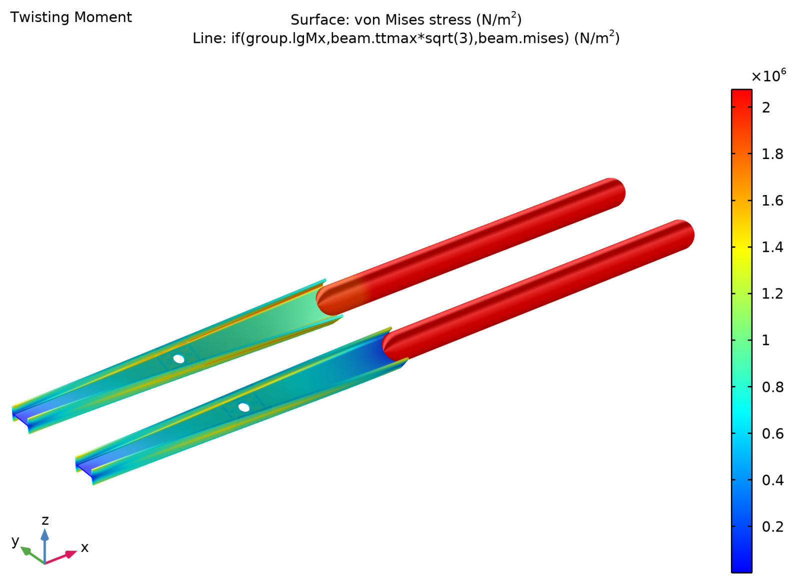


Figure 2: The von Mises stress distribution of the Twisting Moment load case.

To compare the behavior of both connection types, the von Mises stress distribution for the load case with moment about the Z axis (the "bending moment Z" load case) is plotted, along with the difference in von Mises distribution between the two. As can be seen in Figure 3, the stress distribution is comparable throughout the structure except near the solid-beam interface.

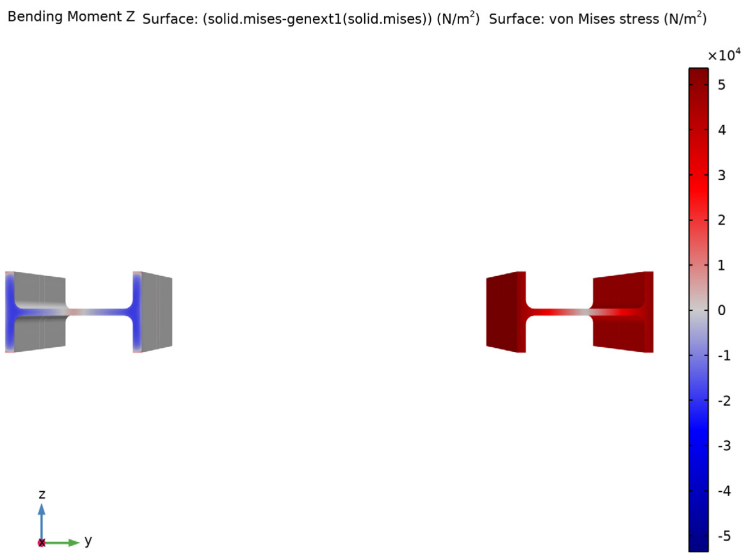


Figure 3: Comparison of von Mises stress distribution under Bending Moment Z load case.

To compare the performance of the two connection types, the traction distribution at the solid-beam interface for different load cases are plotted as shown in Figure 4 and Figure 5. For the load case with twisting moment, the transition connection type shows an in-plane distribution of traction, while the traction in the second case is predominantly out of plane. Similarly, for the load case with moment about the Z-axis, a uniform distribution of traction across the cross section is observed for the first type compared to the general connection type in the second case.

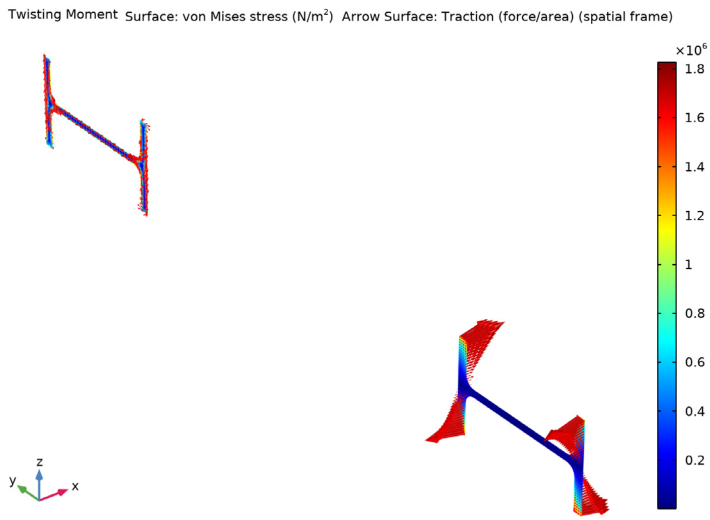


Figure 4: Traction distribution at the solid-beam interface for the twisting moment load case.

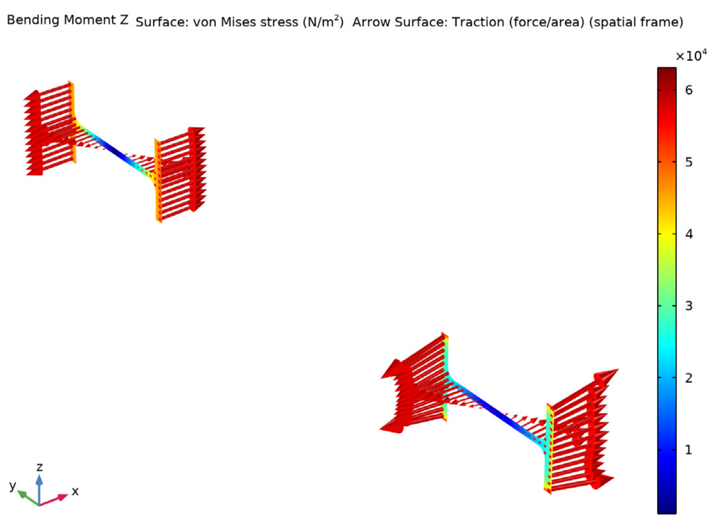


Figure 5: Traction distribution at solid-beam interface for bending moment Z load case.

Notes About the COMSOL Implementation

The built-in multiphysics coupling **Solid-Beam Connection** is used to connect the Solid and Beam interfaces in this model. The connection type **Solid boundaries to beam points, transition** includes warping of the cross section. The variables of warping are separately solved for in a stationary step, by clearing the selection of all other physics interfaces except the **Solid-Beam Connection** multiphysics coupling. In the subsequent study steps, these variables are manually suppressed to solve for the other variables of the study. If warping is insignificant for a certain connection, then you can select to ignore it, thus simplifying the solver setup.

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/beam_solid_connection`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 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 `beam_solid_connection_parameters.txt`.

Load Group Mx

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load and Constraint Groups>Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Mx` in the **Label** text field.

3 In the **Parameter name** text field, type `lgMx`.

Load Group My

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group My` in the **Label** text field.

3 In the **Parameter name** text field, type `lgMy`.

Load Group Mz

1 Right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Mz` in the **Label** text field.

3 In the **Parameter name** text field, type `lgMz`.

Load Group Fx

1 Right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Fx` in the **Label** text field.

3 In the **Parameter name** text field, type `lgFx`.

Load Group Fy

1 Right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Fy` in the **Label** text field.

3 In the **Parameter name** text field, type `lgFy`.

Load Group Fz

1 Right-click **Load and Constraint Groups** and choose **Load Group**.

2 In the **Settings** window for **Load Group**, type `Load Group Fz` in the **Label** text field.

3 In the **Parameter name** text field, type `lgFz`.

The following section provides step by step instructions to create the geometry from scratch. If you do not want to build the geometry yourself, you can load the geometry sequence from the stored model. In the **Model Builder** window, under

Component 1 (comp1) right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `beam_solid_connection.mph`.

You can then skip the section about creating **Geometry 1** and continue to the **Definitions** section.

GEOMETRY 1

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. From the **Data source** list, choose **Vectors**.
4. In the **x** text field, type `L 2*L`.
5. In the **y** text field, type `0 0`.
6. In the **z** text field, type `0 0`.
7. Click **Build Selected**.
8. Click the **Zoom Extents** button in the **Graphics** toolbar.

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 **zy-plane**.
4. Click **Build Selected**.

Work Plane 1 (wp1)>Plane Geometry

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

PART LIBRARIES

1. In the **Home** toolbar, click **Windows** and choose **Part Libraries**.
2. In the **Part Libraries** window, select **Structural Mechanics Module>Beams>European Standard>IPE_beam** in the tree.
3. Click **Add to Geometry**.

GEOMETRY 1

Work Plane 1 (wp1)>European IPE-beam 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry** click **European IPE-beam 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, click **Build Selected**.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
L

4 Select the **Reverse direction** check box.

5 Click **Build Selected**.

Cylinder 1 (cyl1)

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type R.

4 In the **Height** text field, type L.

5 Locate the **Position** section. In the **x** text field, type L/2.

6 In the **z** text field, type -L/2.

7 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **ext1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **cyl1** only.

6 Click **Build Selected**.

Copy 1 (copy1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

4 In the **y** text field, type `0.3*L`.

5 Click **Build Selected**.

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

Work Plane 2 (wp2)

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 **zy-plane**.

4 In the **x-coordinate** text field, type `0.45*L`.

5 Click **Build Selected**.

Work Plane 3 (wp3)

1 Right-click **Work Plane 2 (wp2)** and choose **Duplicate**.

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

3 In the **x-coordinate** text field, type `0.55*L`.

4 Click **Build Selected**.

Partition Objects 1 (par1)

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

2 Select the objects **copy1(1)** and **dif1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 From the **Work plane** list, choose **Work Plane 2 (wp2)**.

6 Click **Build Selected**.

Partition Objects 2 (par2)

1 Right-click **Partition Objects 1 (par1)** and choose **Duplicate**.

2 Select the objects **par1(1)** and **par1(2)** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Work plane** list, choose **Work Plane 3 (wp3)**.

5 Click **Build Selected**.

Union 1 (uni1)

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

2 Select the object **par2(2)** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Union 2 (uni2)

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

2 Select the object **par2(1)** only.

3 In the **Settings** window for **Union**, 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**.

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

Fixed End

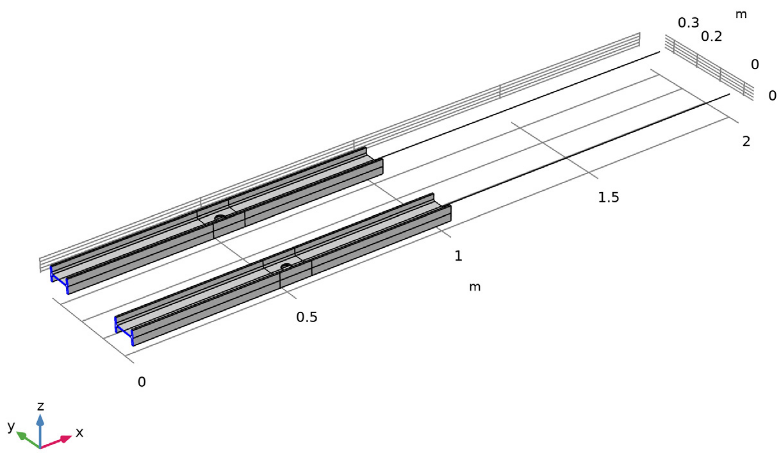
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

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

4 On the object **fin**, select Boundaries 1 and 69 only.

It might be easier to select the boundaries by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



Solid Beam Interface

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

2 In the **Settings** window for **Explicit Selection**, type `Solid Beam Interface` in the **Label** text field.

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

4 On the object **fin**, select Boundaries 68 and 136 only.

Free End

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

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

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

4 On the object **fin**, select Points 178 and 180 only.

DEFINITIONS

General Extrusion 1 (genext1)

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

2 Select Domains 4–6 only.

3 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

4 In the **x-expression** text field, type `X`.

5 In the **y-expression** text field, type `Y+0.3*L`.

6 In the **z-expression** text field, type `Z`.

7 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

ADD MATERIAL

Structural steel is used for all the components of the structure, and thus it can be added as a **Global Material** in the model. Using the **Material Link** node, you can assign the **Global Material** to different domains, boundaries, and edges of the structure.

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 Click the right end of the **Add to Component** split button in the window toolbar.

5 From the menu, choose **Add to Global Materials**.

6 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material Link 1 (matlnk1)

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

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More Materials>Material Link**.

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

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

4 Select Edges 309 and 310 only.

Add physics settings for the **Solid Mechanics** interface.

SOLID MECHANICS (SOLID)

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

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

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

Fixed Constraint 1

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

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

3 From the **Selection** list, choose **Fixed End**.

Add physics settings for the **Beam** interface.

BEAM (BEAM)

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

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

3 Click **Clear Selection**.

4 Select Edges 309 and 310 only.

Cross-Section Data 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Basic Section Properties** section.

3 In the A text field, type `A`.

4 In the I_{zz} text field, type `Izz`.

5 In the e_z text field, type `ez`.

6 In the I_{yy} text field, type `Iyy`.

7 In the e_y text field, type `ey`.

8 In the J text field, type `J`.

9 Click to expand the **Stress Evaluation Properties** section. In the h_y text field, type `hy`.

10 In the h_z text field, type `hz`.

11 In the w_{t} text field, type `Wt`.

12 In the μ_y text field, type `muy`.

13 In the μ_z text field, type `muz`.

Section Orientation 1

1 In the **Model Builder** window, click **Section Orientation 1**.

2 In the **Settings** window for **Section Orientation**, locate the **Section Orientation** section.

3 From the **Orientation method** list, choose **Orientation vector**.

4 Specify the *V* vector as

0	X
1	Y
0	Z

Point Load 1

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Moment** section. Specify the $\mathbf{M}_\mathrm{P}$ vector as

1	x
0	y
0	z

5 In the **Physics** toolbar, click **Load Group** and choose **Gravity Load**.

Point Load 2

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Moment** section. Specify the $\mathbf{M}_\mathrm{P}$ vector as

0	x
1	y
0	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group 2**.

Point Load 3

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Moment** section. Specify the $\mathbf{M}_\mathrm{P}$ vector as

0	x
0	y
1	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group 3**.

Point Load 4

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

1	x
0	y
0	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group 4**.

Point Load 5

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
1	y
0	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group 5**.

Point Load 6

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

2 In the **Settings** window for **Point Load**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Free End**.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
0	y
1	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group 6**.

Add couplings between the solid and the beam elements.

MULTIPHYSICS

Solid-Beam Connection Transition

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.

2 In the **Settings** window for **Solid-Beam Connection**, type `Solid-Beam Connection Transition` in the **Label** text field.

3 Locate the **Connection Settings** section. From the **Connection type** list, choose **Solid boundaries to beam points, transition**.

4 Select the **Manual control of selections** check box.

5 Locate the **Boundary Selection, Solid** section. Select the **Activate Selection** toggle button.

6 Select Boundary 136 only.

7 Locate the **Point Selection, Beam** section. Select the **Activate Selection** toggle button.

8 Select Point 179 only.

Solid-Beam Connection General

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Solid-Beam Connection**.

2 In the **Settings** window for **Solid-Beam Connection**, type `Solid-Beam Connection General` in the **Label** text field.

3 Locate the **Connection Settings** section. Select the **Manual control of selections** check box.

4 Select Boundary 68 only.

5 Locate the **Point Selection, Beam** section. Select the **Activate Selection** toggle button.

6 Select Point 177 only.

MESH 1

Free Triangular 1

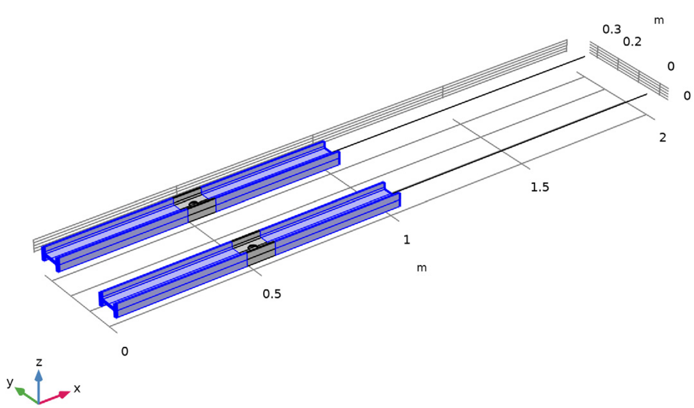
1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2 Select Boundaries 1, 68, 69, and 136 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5 In the associated text field, type `0.002`.
6 Click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 1, 3, 4, and 6 only.



Distribution 1

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

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

3 In the list, choose **3** and **6**.

4 Click **Remove from Selection**.

5 Select Domains 1 and 4 only.

6 Locate the **Distribution** section. From the **Distribution type** list, choose **Explicit**.

7 In the **Relative placement of vertices along edge** text field, type `range(0,0.015,1)`.

8 Click **Build Selected**.

Distribution 2

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

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

3 In the list, choose **1** and **4**.

4 Click **Remove from Selection**.

5 Select Domains 3 and 6 only.

6 Locate the **Distribution** section. From the **Distribution type** list, choose **Explicit**.

7 In the **Relative placement of vertices along edge** text field, type `range(0,0.015,0.92) range(0.92,0.005,1)`.

8 Select the **Reverse direction** check box.

9 Click **Build Selected**.

Free Tetrahedral 1

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

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

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

4 Select Domains 2 and 5 only.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.002`.

6 Click **Build Selected**.

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

2 Select Edges 309 and 310 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type 10.

4 Click **Build All**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics (solid)	√	Physics settings
Beam (beam)		Physics settings

4 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

Stationary 2

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Define load cases** check box.

4 Click **Add** six times.

5 In the table, leave all weights unchanged and modify the other columns as follows:

Load case	lgMx	lgMy	lgMz	lgFx	lgFy	lgFz
Twisting Moment	√					
Bending Moment Y		√				

Load case	lgMx	lgMy	lgMz	lgFx	lgFy	lgFz
Bending Moment Z			√			
Axial Force				√		
Transverse Force Y					√	
Transverse Force Z						√

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Dependent Variables**, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

6 In the **Settings** window for **Field**, locate the **General** section.

7 Clear the **Solve for this field** check box.

8 In the **Model Builder** window, click **Dependent Variables 2**.

9 In the **Settings** window for **Dependent Variables**, locate the **General** section.

10 From the **Defined by study step** list, choose **User defined**.

11 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 2** node, then click **Displacement field (comp1.u)**.

12 In the **Settings** window for **Field**, locate the **Scaling** section.

13 From the **Method** list, choose **Manual**.

14 In the **Scale** text field, type `1.0e-10`.

15 In the **Model Builder** window, click **Displacement field (comp1.u2)**.

16 In the **Settings** window for **Field**, locate the **Scaling** section.

17 From the **Method** list, choose **Manual**.

18 In the **Scale** text field, type `1.0e-10`.

19 In the **Model Builder** window, click **Warping function (comp1.sbc1.ww)**.

20 In the **Settings** window for **Field**, locate the **General** section.

21 Clear the **Solve for this field** check box.

22 In the **Model Builder** window, click **Warping constant (comp1.sbc1.C0)**.

23 In the **Settings** window for **State**, locate the **General** section.

24 Clear the **Solve for this state** check box.

25 In the **Model Builder** window, click **Warping constant (comp1.sbc1.C1)**.

26 In the **Settings** window for **State**, locate the **General** section.

27 Clear the **Solve for this state** check box.

28 In the **Model Builder** window, click **Warping constant (comp1.sbc1.C2)**.

29 In the **Settings** window for **State**, locate the **General** section.

30 Clear the **Solve for this state** check box.

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

DEFINITIONS

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.

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

3 Clear the **Show grid** check box.

Use the following instructions to plot the von Mises stress distribution shown in Figure 2.

RESULTS

Stress (solid)

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Load case** list, choose **Twisting Moment**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Line 1

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

2 Right-click **Line 1** and choose **Copy**.

Line 1

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

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

3 In the **Expression** text field, type `if(group.lgMx,beam.ttmax*sqrt(3),beam.mises)`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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

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

Use the following instructions to plot the comparison of von Mises stress distribution shown in Figure 3.

Comparison

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Comparison` in the **Label** text field.

3 Locate the **Data** section. From the **Load case** list, choose **Bending Moment Z**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

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

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

3 In the **Expression** text field, type `(solid.mises-genext1(solid.mises))`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 Select the **Symmetrize color range** check box.

Surface 2

1 In the **Model Builder** window, right-click **Comparison** and choose **Surface**.

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

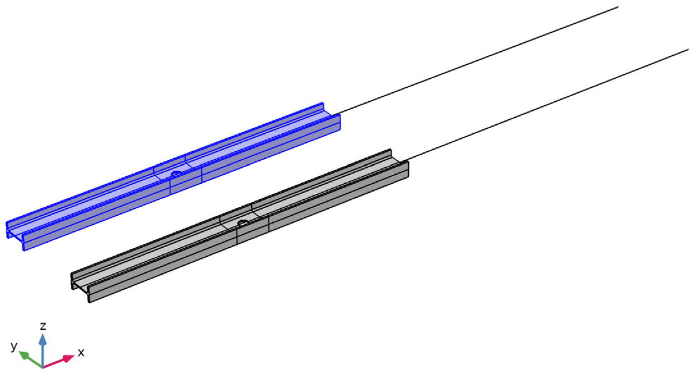
3 In the **Expression** text field, type `solid.mises`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 Select Boundaries 69–136 only.



3 In the **Comparison** toolbar, click **Plot**.

To see the stress distribution at the solid-beam interface, select the **YZ View**.

4 Click the **Go to YZ View** button in the **Graphics** toolbar.

For different load cases, the traction at the solid-beam interface is compared for both Solid-Beam Connection types as shown in Figure 4 and Figure 5.

Study 1/Solution 1 (3) (sol1)

In the **Results** toolbar, click **More Datasets** and choose **Solution**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Solid Beam Interface**.

Traction

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Traction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 From the **Load case** list, choose **Twisting Moment**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

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

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

3 In the **Expression** text field, type `solid.mises`.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Traction** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>solid.Tax,...,solid.Taz - Traction (force/area) (spatial frame)**.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `1000`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 In the **Traction** toolbar, click **Plot**.

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

Traction

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Load case** list, choose **Bending Moment Z**.

Arrow Surface 1

1 In the **Model Builder** window, click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 In the **Number of arrows** text field, type `120`.

4 In the **Traction** toolbar, click **Plot**.