Created in COMSOL Multiphysics 5.6

Pratt Truss Bridge

Introduction

This example is inspired by a classic bridge type called a Pratt truss bridge. You can identify a Pratt truss by its diagonal members, which (except for the very end ones) all slant down and in toward the center of the span. All the diagonal members are subject to tension forces only, while the shorter vertical members handle the compressive forces. Since the tension removes the buckling risk, this allows for thinner diagonal members resulting in a more economic design.

A *truss structure* supports only tension and compression forces in its members and you would normally model it using bars, but as this model uses 3D beams it also includes bending moments to some extent in a *frame structure*. In the model, shell elements represent the roadway.

Model Definition

BASIC DIMENSIONS

The length of the bridge is 40 m, and the width of the roadway is 7 m. The main distance between the truss members is 5 m.

ANALYSIS TYPES

The model includes two different analyses of the bridge:

- The goal of the first analysis is to evaluate the stress and deflection fields of the bridge when exposed to a pure gravity load and also when a load corresponding to one or two trucks cross the bridge.
- Finally, an eigenfrequency analysis shows the eigenfrequencies and eigenmodes of the bridge.

LOADS AND CONSTRAINTS

To prevent rigid body motion of the bridge, it is important to constrain it properly. All translational degrees of freedom are constrained at the leftmost horizontal edge. Constraints at the right-most horizontal edge prevent it from moving in the vertical and transversal directions but allow the bridge to expand or contract in the axial direction. This difference would however only be important if thermal expansion was studied.

Figure 1 shows the bridge geometry.

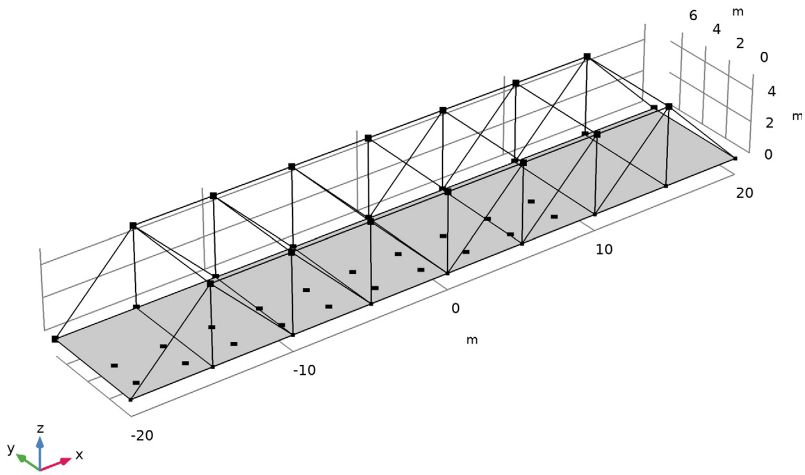


Figure 1: The geometry of the bridge

The first study uses several load cases. In the first load case the effects of self-weight are analyzed. The following load cases compute the solution when two trucks are moving over the bridge. The weight of each truck is 12,000 kg, the wheelbase is 6 m, the axle track is 2 m, and the weight is distributed with one third on the front axle and two thirds on the rear axle. The right side wheels of the truck are 1 m from the edge of the bridge.

In the second study the natural frequencies of the bridge are computed.

MATERIAL PROPERTIES AND CROSS SECTION DATA

The material in the frame structure is structural steel. The roadway material is concrete; the effect of reinforcement is ignored. The frame members have different cross sections:

- The main beams along the bridge have square box profiles with height 200 mm and thickness 16 mm. This is also true for the outermost diagonal members.
- The diagonal and vertical members have a rectangular box section 200 mm-by-100 mm and a thickness of 12.5 mm. The large dimension is in the transverse direction of the bridge.

- The transverse horizontal members supporting the roadway (floor beams) are standard HEA100 profiles.
- The transverse horizontal members at the top of the truss (struts) are made from solid rectangular sections with dimension 100 mm-by-25 mm. The large dimension is in the horizontal direction.

Results and Discussion

Figure 2 and Figure 3 illustrate the result. Figure 2 shows the displacements, and it can be seen that the maximum deflection amounts to 3 cm on the roadway. The distribution of axial forces (Figure 3) demonstrates the function of the frame: The interplay of members in tension and compression contribute to the load carrying function. The upper horizontal members are in compression and the lower in tension. The force in the lower members is much smaller, since the load is also shared by the roadway in this example. The diagonal members are subject to tension forces only, while the shorter vertical members handle the compressive forces.

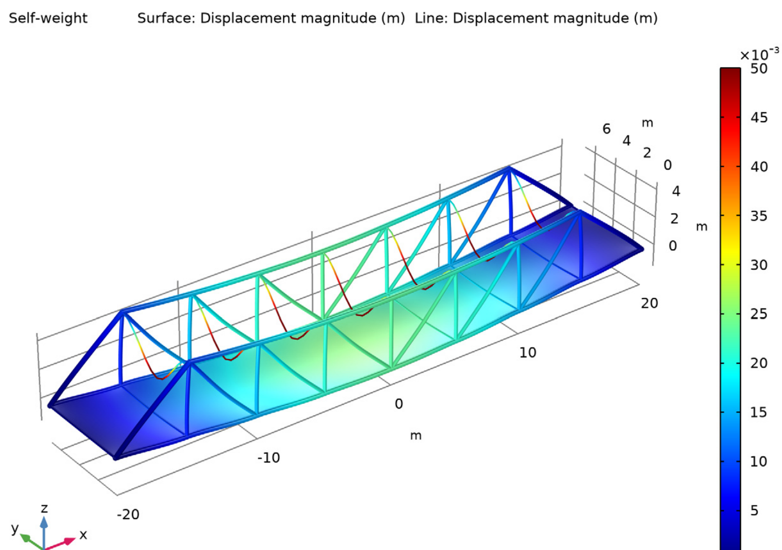


Figure 2: Deformation under self-weight.

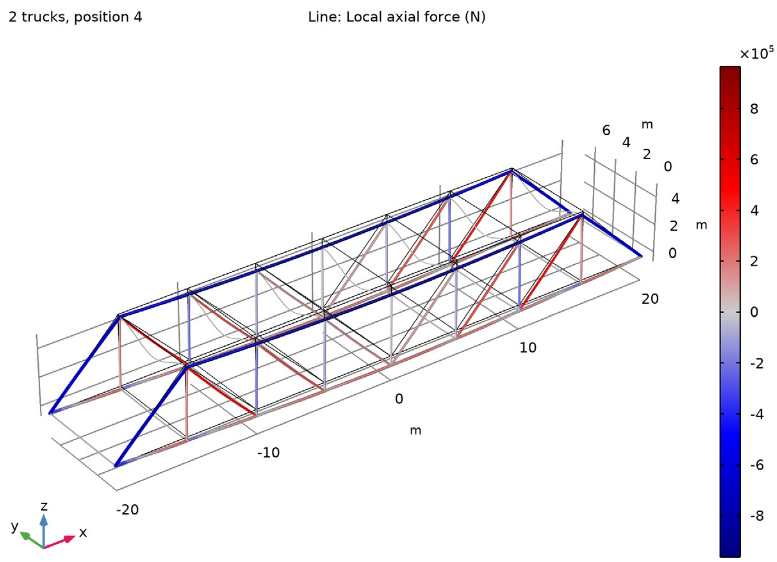


Figure 3: The axial forces in the beams. Red is tension and blue is compression.

To study the effects of trucks moving over the bridge, several load cases represent the position of the trucks. The trucks are moved 3 m along the bridge for each load case.

Figure 4 shows the stress distribution in the roadway when the first truck has passed the bridge center and the second truck has entered the bridge deck.

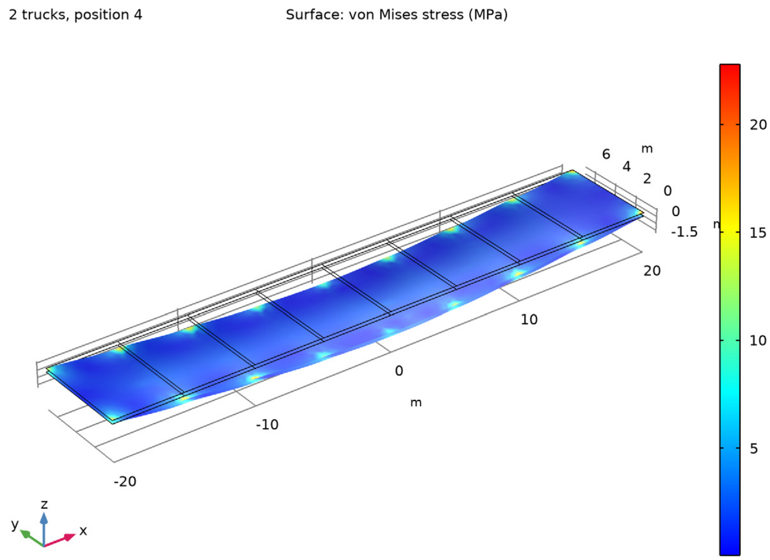


Figure 4: Truck load analysis: Stresses in the bridge deck with two trucks on the bridge.

The study of eigenfrequencies is important with respect to the excitation and frequency content from various loads such as wind loads and earthquakes.

[Figure 5](#) shows the 10^{th} eigenmode of the bridge, which is the fundamental mode for the roadway. The first eight eigenmodes only involve displacements of the weak struts at the top of the truss.

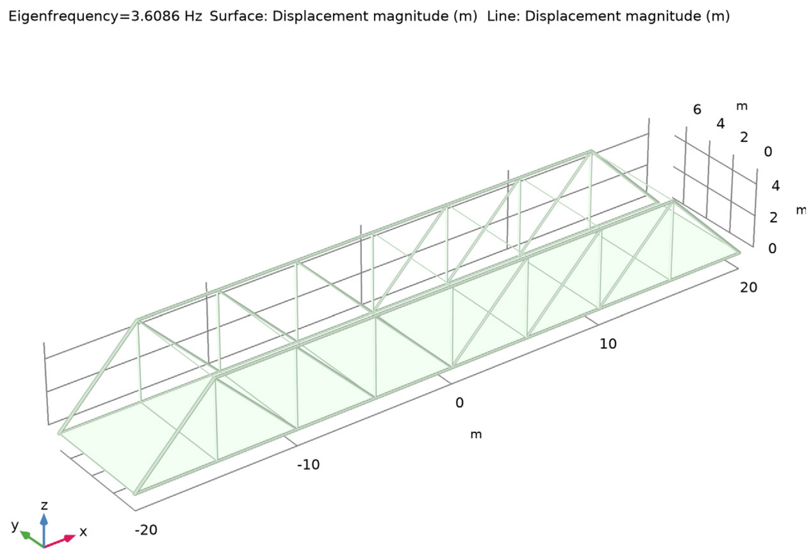


Figure 5: The 10^{th} eigenmode.

Notes About the COMSOL Implementation

You can define load cases to activate and deactivate loads within a study. All the loads need to be defined in the Model Builder. Under the Stationary Study node, you can then select which load or constraint to activate for a specific load case. Moreover for each load case you can modify the value of the applied load by changing its weight factor.

☑ Define load cases

Load case	lg1	Weight	lg2	Weight	lg3	Weight	lg4	Weight	lg5	Weight
Self-weight	☐	1.0	☐	1.0	☐	1.0	☐	1.0	☐	1.0
1 truck, position 1	☑	2.0	☐	1.0	☑	1.0	☐	1.0	☐	1.0
1 truck, position 2	☐	1.0	☑	2.0	☐	1.0	☑	1.0	☐	1.0
1 truck, position 3	☐	1.0	☐	1.0	☑	2.0	☐	1.0	☑	1.0

When combining two different physics interfaces, each have individual sets of degrees of freedom as a default. In structural mechanics, you usually want these to be equal. You can set such connections across various structural mechanics interfaces using built in multiphysics connection features. In this particular model, the **Shell-Beam Connection** features under the **Multiphysics** node are used to set up the connection between the two physics.

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/pratt_truss_bridge`

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 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

Load Groups

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Node Group**.
2 In the **Settings** window for **Group**, type `Load Groups` in the **Label** text field.

Load Group 1

1 Right-click **Load Groups** and choose **Load and Constraint Groups>Load Group**.
2 Repeat this nine times so that you get ten load groups.

GEOMETRY 1

The geometry sequence for the model (see Figure 1) is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Appendix — Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `pratt_truss_bridge_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Parameters 1

Add the non geometrical parameters.

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
truck_weight	12000[kg]	12000 kg	Total truck weight

DEFINITIONS

Create groups for the different beam sections.

BeamsTransvBelow

1 In the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `BeamsTransvBelow` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2+1)`.
5 In the **x maximum** text field, type `length/2+1`.
6 In the **y minimum** text field, type `1`.
7 In the **y maximum** text field, type `width-1`.
8 In the **z minimum** text field, type `-1`.
9 In the **z maximum** text field, type `1`.

BeamsAllBelow

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

2 In the **Settings** window for **Box**, type `BeamsAllBelow` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `-1`.

4 In the **y maximum** text field, type `width+1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

BeamsTransvAbove

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `BeamsTransvAbove` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2+1)`.

5 In the **x maximum** text field, type `length/2+1`.

6 In the **y minimum** text field, type `1`.

7 In the **y maximum** text field, type `width-1`.

8 In the **z minimum** text field, type `height-1`.

9 In the **z maximum** text field, type `height+1`.

BeamsDiag

1 In the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `BeamsDiag` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-(length/2-spacing+1)`.

5 In the **x maximum** text field, type `length/2-spacing+1`.

6 In the **y minimum** text field, type `-1`.

7 In the **y maximum** text field, type `width+1`.

8 In the **z minimum** text field, type `1`.

9 In the **z maximum** text field, type `2`.

AllBeams

1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select the **All edges** check box.

BeamsMain

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `BeamsMain` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **AllBeams** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **BeamsTransvBelow**, **BeamsTransvAbove**, and **BeamsDiag**.

10 Click **OK**.

Add the materials.

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>Concrete**.

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

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

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

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

MATERIALS

Structural steel (mat2)

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

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

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

SHELL (SHELL)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

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

3 In the d text field, type `0.25`.

Add self-weight for the bridge deck.

Gravity 1

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

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

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

Pinned 1

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

2 Select Edge 1 only.

Prescribed Displacement/Rotation 1

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

2 Select Edge 78 only.

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

4 Select the **Prescribed in y direction** check box.

5 Select the **Prescribed in z direction** check box.

Add the possible loads from the truck wheels.

Point Loads

1 In the **Model Builder** window, right-click **Shell (shell)** and choose **Node Group**.

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

Point Load 1

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

2 Click **Load Group** and choose **Gravity Load**.

3 Select Points 3 and 4 only.

4 In the **Settings** window for **Point Load**, locate the **Force** section.

5 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
0	y
-truck_weight*g_const/6	z

Point Load 2

1 Right-click **Point Load 1** and choose **Duplicate**.

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

3 In the **Model Builder** window, click **Point Load 2**.

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

5 Click **Clear Selection**.

6 Select Points 5 and 6 only.

7 Repeat this duplication procedure so that you get ten **Point Load** features. The loaded points and corresponding load groups are summarized in the table below:

Point load	Points	Load goup
Point Load 3	11, 12	Load Group 3
Point Load 4	15, 16	Load Group 4
Point Load 5	19, 20	Load Group 5
Point Load 6	25, 26	Load Group 6
Point Load 7	27, 28	Load Group 7
Point Load 8	33, 34	Load Group 8
Point Load 9	37, 38	Load Group 9
Point Load 10	41, 42	Load Group 10

BEAM (BEAM)

Set the cross-section data of the different beam types.

Cross Section Main

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**, type `Cross Section Main` in the **Label** text field.

3 Locate the **Cross-Section Definition** section. From the list, choose **Common sections**.

4 From the **Section type** list, choose **Box**.

5 In the h_y text field, type `200[mm]`.

6 In the h_z text field, type `200[mm]`.

7 In the t_y text field, type `16[mm]`.

8 In the t_z text field, type `16[mm]`.

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

Cross Section Diagonals

1 In the **Physics** toolbar, click **Edges** and choose **Cross-Section Data**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section Diagonals` in the **Label** text field.

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

4 Locate the **Cross-Section Definition** section. From the list, choose **Common sections**.

5 From the **Section type** list, choose **Box**.

6 In the h_y text field, type `200[mm]`.

7 In the h_z text field, type `100[mm]`.

8 In the t_y text field, type `12.5[mm]`.

9 In the t_z text field, type `12.5[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Cross Section Diagonals** node, then 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

Cross Section Transv Below

1 In the **Physics** toolbar, click **Edges** and choose **Cross-Section Data**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section Transv Below` in the **Label** text field.

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

4 Locate the **Cross-Section Definition** section. From the list, choose **Common sections**.

5 From the **Section type** list, choose **H-profile**.

6 In the h_y text field, type `96[mm]`.

7 In the h_z text field, type `100[mm]`.

8 In the t_y text field, type `8[mm]`.

9 In the t_z text field, type `5[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Cross Section Transv Below** node, then 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
0	Y
1	Z

Cross Section Transv Above

1 In the **Physics** toolbar, click **Edges** and choose **Cross-Section Data**.

2 In the **Settings** window for **Cross-Section Data**, type `Cross Section Transv Above` in the **Label** text field.

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

4 Locate the **Cross-Section Definition** section. From the list, choose **Common sections**.

5 In the h_y text field, type `100[mm]`.

6 In the h_z text field, type `25[mm]`.

Section Orientation 1

1 In the **Model Builder** window, expand the **Cross Section Transv Above** node, then 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

1	X
0	Y
0	Z

Add the self-weight of the beams.

Gravity 1

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

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

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

Create connections between beams and shells.

MULTIPHYSICS

Shell-Beam Connection 1 (shbc1)

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

2 In the **Settings** window for **Shell-Beam Connection**, locate the **Connection Settings** section.

3 From the **Connection type** list, choose **Shared edges**.

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

5 Locate the **Edge Selection** section. Click **Clear Selection**.

6 Select Edges 2, 8, 18, 30, 40, 50, 62, and 72 only.

7 Locate the **Connection Settings** section. From the **Offset definition** list, choose **Offset vector**.

8 Specify the $\mathbf{d}_0$ vector as

0	X

beam.hy_box/2	Y
-beam.hz_box/2	Z

Shell-Beam Connection 2 (shbc2)

1 Right-click **Shell-Beam Connection 1 (shbc1)** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Edges 4, 13, 25, 35, 45, 57, 67, and 76 only.

5 Locate the **Connection Settings** section. Specify the $\mathbf{d}_0$ vector as

0	X
-beam.hy_box/2	Y
-beam.hz_box/2	Z

Shell-Beam Connection 3 (shbc3)

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

2 In the **Settings** window for **Shell-Beam Connection**, locate the **Connection Settings** section.

3 From the **Connection type** list, choose **Shared edges**.

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

5 Locate the **Edge Selection** section. From the **Selection** list, choose **BeamsTransvBelow**.

6 Locate the **Connection Settings** section. From the **Offset definition** list, choose **Offset vector**.

7 Specify the $\mathbf{d}_0$ vector as

0	X
0	Y
-beam.hy_H/2	Z

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 **Extremely fine**.

4 Locate the **Mesh Settings** section. From the **Sequence type** list, choose **User-controlled mesh**.

Distribution 1

1. In the **Model Builder** window, 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 2.
4. Click **Build All**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

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**, click to expand the **Study Extensions** section.
3. Select the **Define load cases** check box.
4. Click **Add** nine times.
5. In the table, select the load cases and set the related weight according to the table:

Load case	Active load groups and weights
Self-weight	
1 truck, position 1	lg1: 2.0; lg3: 1.0
1 truck, position 2	lg2: 2.0; lg4: 1.0
1 truck, position 3	lg3: 2.0; lg5: 1.0
1 truck, position 4	lg4: 2.0; lg6: 1.0
2 trucks, position 1	lg1: 1.0; lg5: 2.0; lg7: 1.0
2 trucks, position 2	lg2: 1.0; lg6: 2.0; lg8: 1.0
2 trucks, position 3	lg1: 2.0; lg3: 1.0; lg7: 2.0; lg9: 1.0
2 trucks, position 4	lg2: 2.0; lg4: 1.0; lg8: 2.0; lg10: 1.0

6. In the **Home** toolbar, click **Compute**.

RESULTS

Stress (shell)

The default plot is the stress plot for the shells using the last load case, see Figure 4.

Surface 1

1. In the **Model Builder** window, expand the **Stress (shell)** 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 the **Zoom Extents** button in the **Graphics** toolbar.

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

Add a new plot containing both shell and beam results, and examine the self-weight load case to reproduce Figure 2.

Displacement

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

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

3 Locate the **Data** section. From the **Load case** list, choose **Self-weight**.

Surface 1

Right-click **Displacement** and choose **Surface**.

Deformation 1

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

Line 1

1 In the **Model Builder** window, right-click **Displacement** and choose **Line**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Beam>Displacement> beam.disp - Displacement magnitude - m**.

You can indicate the dimensions of the beams by drawing them with a size depending on the radius of gyration.

1 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

2 In the **Tube radius expression** text field, type `comp1.beam.re`.

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

Surface 1

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

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `0.05`.

Deformation 1

In the **Model Builder** window, right-click **Line 1** and choose **Deformation**.

Now plot the axial force in beams like in Figure 3.

1 In the **Settings** window for **Deformation**, locate the **Expression** section.

2 In the **X component** text field, type u2.

3 In the **Y component** text field, type v2.

4 In the **Z component** text field, type w2.

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

Displacement

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

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

3 Clear the **Plot dataset edges** check box.

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

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

Investigate the forces in the beams.

Axial Force (beam)

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

2 In the **Model Builder** window, expand the **Results>Section Forces (beam)** node, then click **Axial Force (beam)**.

3 In the **Axial Force (beam)** toolbar, click **Plot**.

Stress (shell)

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

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

Create an animation of the trucks passing the bridge.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 In the **Number of frames** text field, type 9.

4 In the **Frame number** text field, type 9.

You can easily remove unused plots to clean up the structure in the **Results** tree. An alternative could have been to clear **Generate default plots** in the Study feature, but then you would have needed to create the current plot manually.

In the **Model Builder** window, select **Results>Stress Bottom (shell) 1**, hold down the Shift key, and then select **Results>Torsion Moment (beam) 1**. Right-click and choose **Delete**.

Before adding a new study you can group all current plots. First ungroup the load plots and section forces plots created by default.

Applied Loads (beam), Applied Loads (shell), Section Forces (beam)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Applied Loads (shell)**, **Section Forces (beam)**, and **Applied Loads (beam)**.

2 Right-click and choose **Ungroup**.

3 In the **Model Builder** window, hold down the Shift key to select all plot groups under **Results**, right-click the selected nodes and choose **Group**.

Stationary Results

1 In the **Model Builder** window, under **Results** click **Group 3**.

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

Now add an eigenfrequency study.

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> Eigenfrequency**.

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

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

STUDY 2

Step 1: Eigenfrequency

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

2 Select the **Desired number of eigenfrequencies** check box.

3 In the associated text field, type `12`.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (shell)

Select the first mode involving the roadway, see Figure 5.

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

2 From the **Eigenfrequency (Hz)** list, choose **3.6086**.

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

4 In the **Model Builder** window, expand the **Mode Shape (shell)** node.

Line 1

1 In the **Model Builder** window, expand the **Results>Mode Shape (beam)** node.

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

Line 1

1 In the **Model Builder** window, right-click **Mode Shape (shell)** and choose **Paste Line**.

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

3 From the **Plot** list, choose **Surface 1**.

Deformation

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `0.2`.

Surface 1

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

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

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type `4`.

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

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

7 In the **Model Builder** window, hold down the shift key to select all the newly added plot groups under **Results**, right-click the selected nodes and choose **Group**.

Eigenfrequency Results

1 In the **Model Builder** window, under **Results** click **Group 4**.

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

Prepare for an export to file of the beam section forces and stresses at element level for the main beams.

Study 1/Solution 1 (3) (sol1)

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

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** and choose **Duplicate**.

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 **Edge**.

4 From the **Selection** list, choose **BeamsMain**.

Data 1

1 In the **Model Builder** window, under **Results>Datasets** right-click **Study 1/ Solution 1 (3) (sol1)** and choose **Add Data to Export**.

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

3 From the **Parameter selection (Load case)** list, choose **From list**.

4 In the **Load cases** list, select **Self-weight**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
dom		Entity index
beam.Nxl	kN	Local axial force
beam.Myl	kN*m	Bending moment, local y direction
beam.Tzl	kN	Shear force, local z direction
beam.Mzl	kN*m	Bending moment, local z direction
beam.Tyl	kN	Shear force, local y direction
beam.Mxl	kN*m	Torsional moment, local x direction
beam.mises	MPa	von Mises stress

6 Locate the **Output** section. From the **Geometry level** list, choose **Line**.

7 Click to expand the **Advanced** section. Clear the **Full precision** check box.

Appendix — Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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
width	7[m]	7 m	Width of bridge
height	5[m]	5 m	Height of bridge
spacing	5[m]	5 m	Spacing between members along the bridge
length	40[m]	40 m	Total bridge length

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

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

Create the bridge deck.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `spacing`.

4 In the **Height** text field, type `width`.

5 Locate the **Position** section. In the **xw** text field, type `-length/2`.

6 In the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

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

3 From the **Array type** list, choose **Linear**.

4 Select the object **r1** only.

5 In the **Size** text field, type `length/spacing`.

6 Locate the **Displacement** section. In the **xw** text field, type `spacing`.

7 In the **Work Plane** toolbar, click **Build All**.

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

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

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

Start creating the truss.

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 **xz-plane**.

4 Click **Show Work Plane**.

Work Plane 2 (wp2)>Polygon 1 (pol1)

1 In the **Work Plane** 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. From the **Data source** list, choose **Vectors**.

5 In the **xw** text field, type `0 spacing spacing spacing`.

6 In the **yw** text field, type `0 height height 0`.

Work Plane 2 (wp2)>Line Segment 1 (ls1)

1 In the **Work Plane** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **yw** text field, type `height`.

6 Locate the **Endpoint** section. In the **xw** text field, type `spacing` and **yw** to `height`.

7 In the **Work Plane** toolbar, click **Build All**.

Work Plane 2 (wp2)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

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

3 From the **Array type** list, choose **Linear**.

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

5 In the **Size** text field, type `length/(2*spacing)-1`.

6 Locate the **Displacement** section. In the **xw** text field, type `spacing`.

7 In the **Work Plane** toolbar, click **Build All**.

Work Plane 2 (wp2)>Line Segment 2 (ls2)

1 In the **Work Plane** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5 Locate the **Starting Point** section. In the **xw** text field, type `length/2-spacing` and **yw** to `height`.
6 Locate the **Endpoint** section. In the **xw** text field, type `length/2`.
7 In the **Work Plane** toolbar, click **Build All**.

Work Plane 2 (wp2)>Mirror 1 (mir1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
3 In the **Settings** window for **Mirror**, locate the **Input** section.
4 Select the **Keep input objects** check box.
5 In the **Work Plane** toolbar, click **Build All**.
6 Click the **Zoom Extents** button in the **Graphics** toolbar.

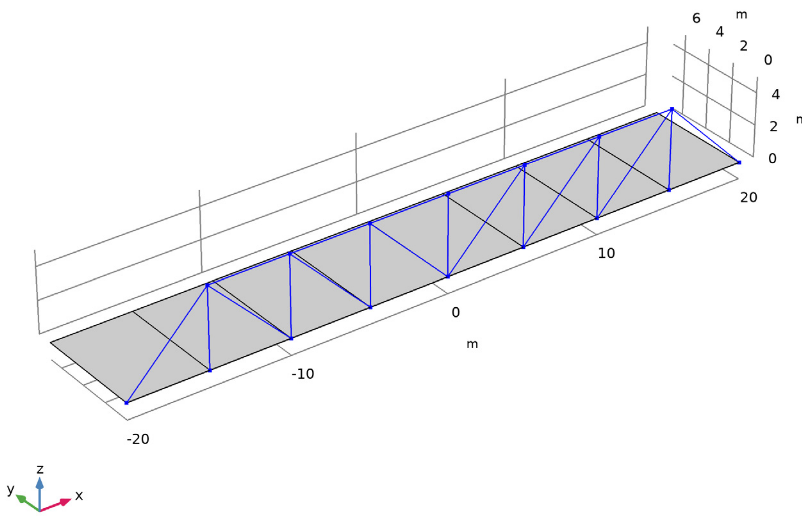
Work Plane 2 (wp2)>Line Segment 3 (ls3)

1 In the **Work Plane** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5 In the **yw** text field, type `height`.
6 In the **Work Plane** toolbar, click **Build All**.
7 Right-click **Geometry 1** and choose **Build All**.

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2 In the **Settings** window for **Copy**, locate the **Displacement** section.
3 In the **y** text field, type `width`.

4 Select the object **wp2** only.



5 Click **Build All Objects**.

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

5 Locate the **Starting Point** section. In the **x** text field, type `-length/2+spacing`.

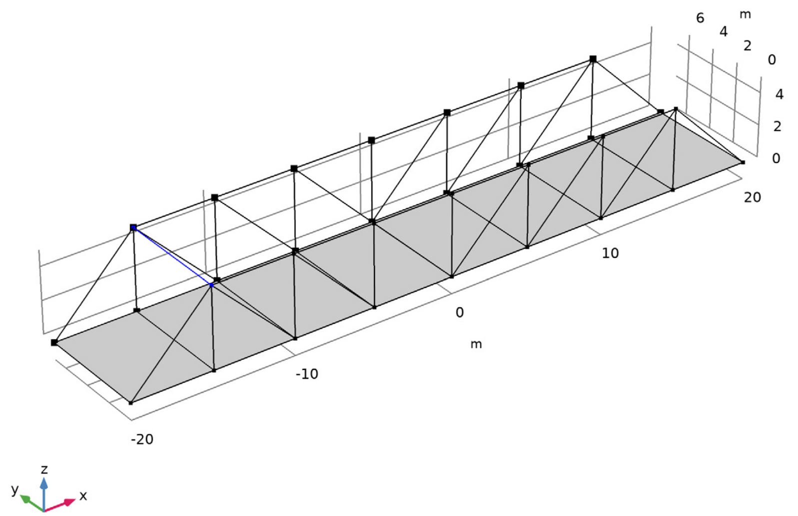
6 Locate the **Endpoint** section. In the **x** text field, type `-length/2+spacing`, **y** to `width`, and **z** to `height`.

7 Click **Build All Objects**.

Array 1 (arr1)

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

2 Select the object **ls1** only.



3 In the **Settings** window for **Array**, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 Locate the **Displacement** section. In the **x** text field, type `spacing`.

6 Locate the **Size** section. In the **Size** text field, type `length/spacing-1`.

Create points in the positions where the loads from the truck wheels are to be applied.

Point 1 (pt1)

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

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

3 In the **x** text field, type `-19`.

4 In the **y** text field, type `1`.

Array 2 (arr2)

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

2 Select the object **pt1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `10`.

5 In the **y size** text field, type 2.

6 Locate the **Displacement** section. In the **x** text field, type 3.

7 In the **y** text field, type 2.

8 Click **Build All Objects**.