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 crosses 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 left-most 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.

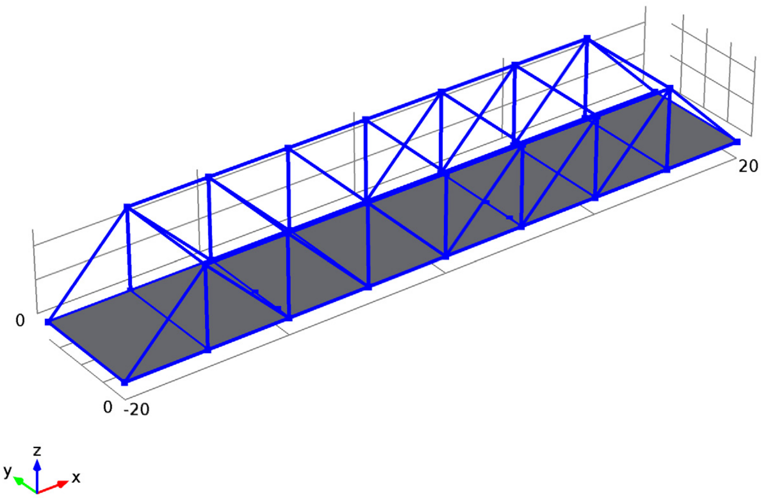

Figure 1: The geometry of the bridge

The first study uses several load cases. In the first load case the effects of self weight is 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 also includes the outermost diagonal members.

- The diagonal and vertical members have a rectangular box section 200x100 mm, with 12.5 mm thickness. 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 100x25 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.

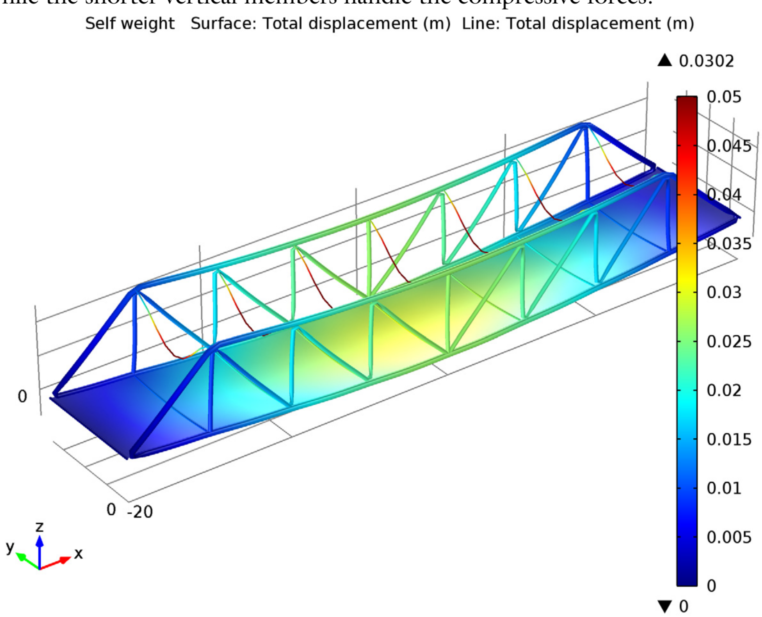

Figure 2: Deformation under self weight.

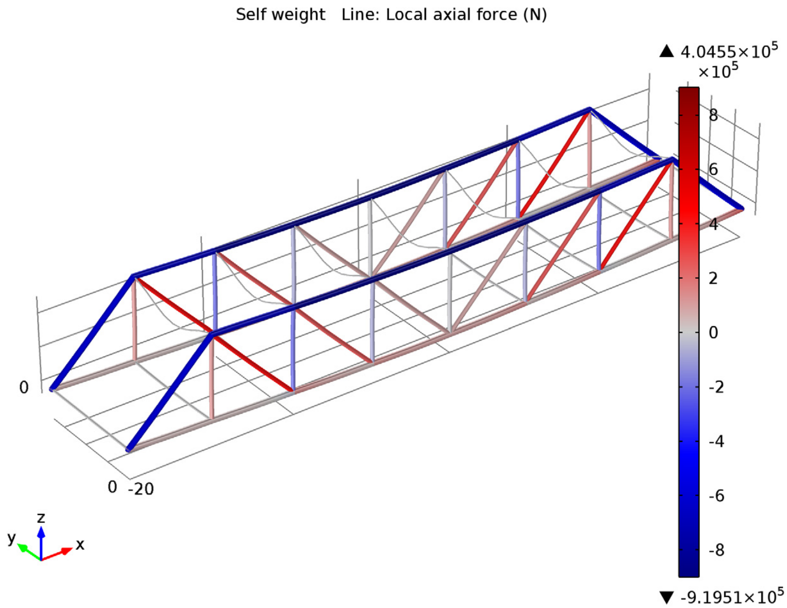

Figure 3: The axial forces in the beams. Red is tension and blue is compression.

To study the effects of trucks moving over the bridge several load cases represent the position of the trucks, which 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.

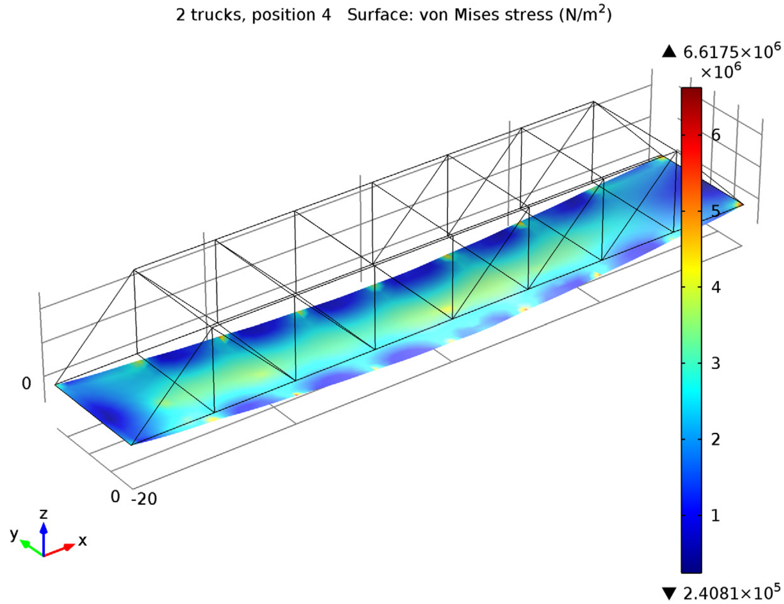

Figure 4: Truck load analysis: Stresses in the bridge deck with two trucks are on the bridge.

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 9th eigenmode of the bridge, which is the fundamental mode for the roadway. The first eight eigenmodes only involve displacements of the weak members at the top of the truss.

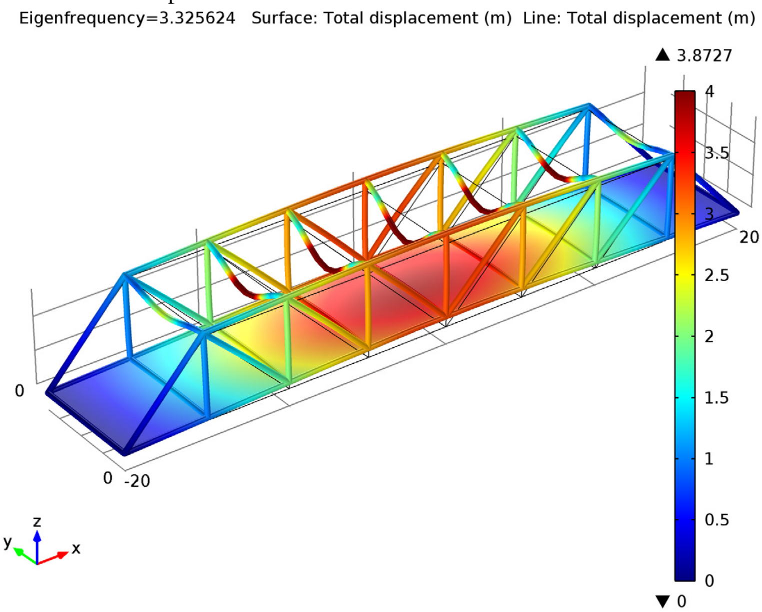

Figure 5: The 9th eigenmode.

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 step 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	Wei
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, they will as a default have individual sets of degrees of freedom. In structural mechanics, you will usually want them to be equal, so you rename the displacements in the second interface to have the same name as in the first interface. In this case, where both beams and shells have rotational degrees of freedom, also these will in general need to treated. The representation of rotation is however different in the Shell and Beam interface, so the simple renaming cannot be used. In the example you can see how the rotations in the beam elements are prescribed to follow those in the shell to which they are connected. In models like this, the results will be virtually the same if the connection of rotations is omitted, as long as the mesh is reasonably fine.

Model Library path: `Structural_Mechanics_Module/Civil_Engineering/pratt_truss_bridge`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Shell (shell)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **Structural Mechanics>Beam (beam)**.

6 Click **Add Selected**.

Make the beam and shell elements have common displacements.

7 Find the **Dependent variables** subsection. In the **Displacement field** edit field, type `u`.

8 In the **Displacement field components** table, enter the following settings:

u
v
w

9 Click **Next**.

10 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

11 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
2 In the **Parameters** settings window, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Description
width	7[m]	Width of bridge
height	5[m]	Height of bridge
spacing	5[m]	Spacing between members along the bridge
length	40[m]	Total bridge length
truck_weight	12000[kg]	Total truck weight

4 Right-click **Global Definitions** and choose **Load Group**. Repeat this nine times so that you get ten load groups.

GEOMETRY 1

In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Work Plane**.

Plane Geometry

Create the bridge deck.

Rectangle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type `spacing`.
4 In the **Height** edit field, type `width`.
5 Locate the **Position** section. In the **xw** edit field, type `-length/2`.
6 Click the **Build All** button.

Array 1

1 Right-click **Plane Geometry** and choose **Transforms>Array**.
2 In the **Array** settings window, locate the **Size** section.
3 From the **Array type** list, choose **Linear**.

4 Select the object **r1** only.

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

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

7 Click the **Build All** button.

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

Plane Geometry

Make it possible to create beams along the edges.

Convert to Curve 1

1 Right-click **Plane Geometry** and choose **Conversions>Convert to Curve**.

2 In the **Convert to Curve** settings window, locate the **Input** section.

3 Select the **Keep input objects** check box.

4 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

5 Click the **Build All** button.

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

7 Click the **Zoom Extents** button on the Graphics toolbar.

Start creating the truss.

Work Plane 2

1 Right-click **Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Work Plane** section.

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

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 2** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click the **Add Linear** button.

5 Find the **Control points** subsection. In row **2**, set **xw** to `spacing`.

6 In row **2**, set **yw** to `height`.

7 Find the **Added segments** subsection. Click the **Add Linear** button.

8 Find the **Control points** subsection. In row **2**, set **yw** to `0`.

9 Click the **Build All** button.

Bézier Polygon 2

1 Right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click the **Add Linear** button.

5 Find the **Control points** subsection. In row **1**, set **yw** to `height`.

6 In row **2**, set **xw** to `spacing`.

7 In row **2**, set **yw** to `height`.

8 Click the **Build All** button.

Array 1

1 Right-click **Plane Geometry** and choose **Transforms>Array**.

2 In the **Array** settings window, locate the **Size** section.

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

4 Select the objects **b1** and **b2** only.

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

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

7 Click the **Build All** button.

Bézier Polygon 3

1 Right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click the **Add Linear** button.

5 Find the **Control points** subsection. In row **1**, set **xw** to `length/2-spacing`.

6 In row **1**, set **yw** to `height`.

7 In row **2**, set **xw** to `length/2`.

8 Click the **Build All** button.

Mirror 1

1 Right-click **Plane Geometry** and choose **Transforms>Mirror**.
2 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.
3 In the **Mirror** settings window, locate the **Input** section.
4 Select the **Keep input objects** check box.
5 Click the **Build All** button.
6 Click the **Zoom Extents** button on the Graphics toolbar.

Bézier Polygon 4

1 Right-click **Plane Geometry** and choose **Bézier Polygon**.
2 In the **Bézier Polygon** settings window, locate the **General** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click the **Add Linear** button.
5 Find the **Control points** subsection. In row **2**, set **yw** to `height`.
6 Click the **Build All** button.
7 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.

Copy 1

1 Right-click **Geometry 1** and choose **Transforms>Copy**.
2 In the **Copy** settings window, locate the **Displacement** section.
3 In the **y** edit field, type `width`.
4 Select all edges that are not part of the bridge deck. (**wp2.arr1(1,2)**, **wp2.arr1(3,1)**, **wp2.arr1(2,1)**, **wp2.arr1(1,1)**, **wp2.mir1(1)**, **wp2.b3**, **wp2.arr1(3,2)**, **wp2.arr1(2,2)**, **wp2.mir1(6)**, **wp2.mir1(7)**, **wp2.b4**, **wp2.mir1(2)**, **wp2.mir1(3)**, **wp2.mir1(4)**, and **wp2.mir1(5)**).
5 Click the **Build All** button.

Bézier Polygon 1

1 Right-click **Geometry 1** and choose **More Primitives>Bézier Polygon**.
2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.
3 Find the **Added segments** subsection. Click the **Add Linear** button.
4 Find the **Control points** subsection. In row **1**, set **x** to `-length/2+spacing`.
5 In row **1**, set **z** to `height`.

6 In row **2**, set **x** to `-length/2+spacing`.

7 In row **2**, set **y** to `width`.

8 In row **2**, set **z** to `height`.

9 Click the **Build All** button.

Array 1

1 Right-click **Geometry 1** and choose **Transforms>Array**.

2 Select the object **b1** only.

3 In the **Array** settings window, locate the **Size** section.

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

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

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

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

Point 1

1 Right-click **Geometry 1** and choose **More Primitives>Point**.

2 In the **Point** settings window, locate the **Point** section.

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

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

Array 2

1 Right-click **Geometry 1** and choose **Transforms>Array**.

2 Select the object **pt1** only.

3 In the **Array** settings window, locate the **Size** section.

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

5 In the **y size** edit field, type `2`.

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

7 In the **y** edit field, type `2`.

8 Click the **Build All** button.

DEFINITIONS

Create groups for the different beam sections.

Box 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Box**.

2 In the **Box** settings window, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **x minimum** edit field, type `-(length/2+1)`.
5 In the **x maximum** edit field, type `length/2+1`.
6 In the **y minimum** edit field, type `1`.
7 In the **y maximum** edit field, type `width-1`.
8 In the **z minimum** edit field, type `-1`.
9 In the **z maximum** edit field, type `1`.
10 Right-click **Model 1>Definitions>Box 1** and choose **Rename**.
11 Go to the **Rename Box** dialog box and type `BeamsTransvBelow` in the **New name** edit field.
12 Click **OK**.

Box 2

1 Right-click **Definitions** and choose **Selections>Box**.
2 In the **Box** settings window, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **x minimum** edit field, type `-(length/2+1)`.
5 In the **x maximum** edit field, type `length/2+1`.
6 In the **y minimum** edit field, type `1`.
7 In the **y maximum** edit field, type `width-1`.
8 In the **z minimum** edit field, type `height-1`.
9 In the **z maximum** edit field, type `height+1`.
10 Right-click **Model 1>Definitions>Box 2** and choose **Rename**.
11 Go to the **Rename Box** dialog box and type `BeamsTransvAbove` in the **New name** edit field.
12 Click **OK**.

Box 3

1 Right-click **Definitions** and choose **Selections>Box**.
2 In the **Box** settings window, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **x minimum** edit field, type `-(length/2-spacing+1)`.

5 In the **x maximum** edit field, type `length/2-spacing+1`.
6 In the **y minimum** edit field, type `-1`.
7 In the **y maximum** edit field, type `width+1`.
8 In the **z minimum** edit field, type `1`.
9 In the **z maximum** edit field, type `2`.
10 Right-click **Model 1>Definitions>Box 3** and choose **Rename**.
11 Go to the **Rename Box** dialog box and type `BeamsDiag` in the **New name** edit field.
12 Click **OK**.
13 Right-click **Definitions** and choose **Selections>Explicit**.
14 In the **Explicit** settings window, locate the **Input Entities** section.
15 From the **Geometric entity level** list, choose **Edge**.
16 Select the **All edges** check box.

Difference 1

1 Right-click **Definitions** and choose **Selections>Difference**.
2 In the **Difference** settings window, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Edge**.
4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5 Go to the **Add** dialog box.
6 In the **Selections to add** list, select **Explicit 1**.
7 Click the **OK** button.
8 In the **Difference** settings window, locate the **Input Entities** section.
9 Under **Selections to subtract**, click **Add**.
10 Go to the **Add** dialog box.
11 In the **Selections to subtract** list, choose **BeamsTransvBelow**, **BeamsTransvAbove**, and **BeamsDiag**.
12 Click the **OK** button.
13 In the **Model Builder** window, right-click **Difference 1** and choose **Rename**.
14 Go to the **Rename Difference** dialog box and type `BeamsMain` in the **New name** edit field.
15 Click **OK**.

MODEL 1

Add the materials.

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
2 In the **Material Browser** window, locate the **Materials** section.
3 In the tree, select **Built-In>Concrete**.
4 Right-click and choose **Add Material to Model** from the menu.
5 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.
6 In the **Material Browser** window, locate the **Materials** section.
7 In the tree, select **Built-In>Structural steel**.
8 Right-click and choose **Add Material to Model** from the menu.

Structural steel

1 In the **Model Builder** window, under **Model 1>Materials** click **Structural steel**.
2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Edge**.
4 From the **Selection** list, choose **All edges**.

SHELL

1 In the **Model Builder** window, under **Model 1** click **Shell**.
2 In the **Shell** settings window, locate the **Thickness** section.
3 In the d edit field, type `0.25`.

Add self weight for the bridge deck.

Body Load 1

1 In the **Model Builder** window, right-click **Shell** and choose **Body Load**.
2 In the **Body Load** settings window, locate the **Force** section.
3 In the $\mathbf{F}_\mathrm{V}$ table, enter the following settings:

0	x
0	y
-g_const*shell.rho	z

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.

Set the boundary conditions of the bridge.

Pinned 1

1 Right-click **Shell** and choose **Pinned**.

2 Select Edge 1 only.

Prescribed Displacement/Rotation 11

1 Right-click **Shell** and choose **Prescribed Displacement/Rotation**.

2 Select Edge 78 only.

3 In the **Prescribed Displacement/Rotation** settings window, 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 Load 1

1 Right-click **Shell** and choose **Points>Point Load**.

2 In the **Model Builder** window, under **Model 1>Shell** right-click **Point Load 1** and choose **Load Group>Load Group 1**.

3 In the **Model Builder** window, click **Point Load 1**.

4 Select Points 3 and 4 only.

5 In the **Point Load** settings window, locate the **Force** section.

6 In the $\mathbf{F}_p$ table, enter the following settings:

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 **Model Builder** window, under **Model 1>Shell** right-click **Point Load 2** and choose **Load Group>Load Group 2**.

3 In the **Point Load** settings window, locate the **Point Selection** section.

4 Click **Clear Selection**.

5 Select Points 5 and 6 only.

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

Point load	Points	Load group
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, 41	Load Group 10

Make the beam and shell elements have equal rotations.

BEAM

Prescribed Displacement/Rotation 11

1 In the **Model Builder** window, under **Model 1** right-click **Beam** and choose **More>Prescribed Displacement/Rotation**.

2 In the **Prescribed Displacement/Rotation** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **BeamsTransvBelow**.

4 Locate the **Prescribed Rotation** section. Select the **Prescribed in x direction** check box.

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

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

7 In the θ_{0x} edit field, type `shell.thx`.

8 In the θ_{0y} edit field, type `shell.thy`.

Prescribed Displacement/Rotation 12

1 In the **Model Builder** window, right-click **Beam** and choose **More>Prescribed Displacement/Rotation**.

2 In the **Prescribed Displacement/Rotation** settings window, locate the **Prescribed Displacement** section.

3 Select the **Prescribed in x direction** check box.

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

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

6 Locate the **Prescribed Rotation** section. Select the **Prescribed in x direction** check box.

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

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

9 In the θ_{0x} edit field, type `shell.thx`.

10 In the θ_{0y} edit field, type `shell.thy`.

11 Select Edge 1 only.

Prescribed Displacement/Rotation 13

1 Right-click **Beam** and choose **More>Prescribed Displacement/Rotation**.

2 In the **Prescribed Displacement/Rotation** settings window, locate the **Prescribed Displacement** section.

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

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

5 Locate the **Prescribed Rotation** section. Select the **Prescribed in x direction** check box.

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

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

8 In the θ_{0x} edit field, type `shell.thx`.

9 In the θ_{0y} edit field, type `shell.thy`.

10 Select Edge 78 only.

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

Cross Section Data 1

1 In the **Model Builder** window, under **Model 1>Beam** click **Cross Section Data 1**.

2 In the **Cross Section Data** settings window, locate the **Cross Section Definition** section.

3 From the list, choose **Common sections**.

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

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

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

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

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

9 Right-click **Model 1>Beam>Cross Section Data 1** and choose **Rename**.

10 Go to the **Rename Cross Section Data** dialog box and type `Cross Section Main` in the **New name** edit field.

11 Click **OK**.

Section Orientation 1

1 In the **Section Orientation** settings window, locate the **Section Orientation** section.

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

3 In the V table, enter the following settings:

0	x
1	y
0	z

Cross Section Data 2

1 In the **Model Builder** window, right-click **Beam** and choose **Cross Section Data**.

2 In the **Cross Section Data** settings window, locate the **Edge Selection** section.

3 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 edit field, type `200[mm]`.

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

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

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

10 Right-click **Model 1>Beam>Cross Section Data 2** and choose **Rename**.

11 Go to the **Rename Cross Section Data** dialog box and type `Cross Section Diagonals` in the **New name** edit field.

12 Click **OK**.

Section Orientation 1

1 In the **Section Orientation** settings window, locate the **Section Orientation** section.

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

3 In the V table, enter the following settings:

0	x
1	y
0	z

Cross Section Data 3

1 In the **Model Builder** window, right-click **Beam** and choose **Cross Section Data**.

2 In the **Cross Section Data** settings window, locate the **Edge Selection** section.

3 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 edit field, type `96[mm]`.

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

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

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

10 Right-click **Model 1>Beam>Cross Section Data 3** and choose **Rename**.

11 Go to the **Rename Cross Section Data** dialog box and type `Cross Section Transv Below` in the **New name** edit field.

12 Click **OK**.

Section Orientation 1

1 In the **Section Orientation** settings window, locate the **Section Orientation** section.

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

3 In the *V* table, enter the following settings:

0	x
0	y
1	z

Cross Section Data 4

1 In the **Model Builder** window, right-click **Beam** and choose **Cross Section Data**.

2 In the **Cross Section Data** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **BeamsTransvAbove**.

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

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

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

7 Right-click **Model 1>Beam>Cross Section Data 4** and choose **Rename**.

8 Go to the **Rename Cross Section Data** dialog box and type `Cross Section Transv Above` in the **New name** edit field.

9 Click **OK**.

Section Orientation 1

1 In the **Section Orientation** settings window, locate the **Section Orientation** section.

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

3 In the *V* table, enter the following settings:

1	x
0	y
0	z

Add the self weight of the beams.

Edge Load 1

1 In the **Model Builder** window, right-click **Beam** and choose **Edge Load**.

2 In the **Edge Load** settings window, locate the **Edge Selection** section.

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

4 Locate the **Force** section. From the **Load type** list, choose **Edge load defined as force per unit volume**.

5 In the **F** table, enter the following settings:

0	x
0	y
-g_const*beam.rho	z

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extremely fine**.

4 Click the **Build All** button.

STUDY 1

Step 1: Stationary

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

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

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

4 Click **Add**.

5 In the table, enter the following settings:

Load case
Self Weight

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lg1	Weight	lg3
1 truck, position 1	√	2.0	√

8 In a similar way, add seven more load cases with the properties given in the table below:

Load case	Active load groups and weights
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: 21.0
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

9 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The default plot is the stress plot for the shells using the last load case. Add a new plot containing both shell and beam results, and examine the self weight load case.

3D Plot Group 10

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

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

3 From the **Load case** list, choose **Self Weight**.

4 Right-click **Results>3D Plot Group 10** and choose **Surface**.

5 Right-click **Results>3D Plot Group 10>Surface 1** and choose **Deformation**.

6 Right-click **Results>3D Plot Group 10** and choose **Line**.

7 In the **Model Builder** window, under **Results>3D Plot Group 10** click **Line 1**.

8 In the **Line** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Beam>Displacement>Total displacement (beam.disp)**.

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

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

10 In the **Tube radius expression** edit field, type `mod1.beam.rgy+mod1.beam.rgz`.

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

12 In the **Model Builder** window, under **Results>3D Plot Group 10** click **Surface 1**.

13 In the **Surface** settings window, click to expand the **Range** section.

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

15 In the **Maximum** edit field, type `0.05`.

16 In the **Model Builder** window, under **Results>3D Plot Group 10** right-click **Line 1** and choose **Deformation**.

17 Click the **Plot** button.

18 In the **3D Plot Group** settings window, locate the **Plot Settings** section.

19 Clear the **Plot data set edges** check box.

20 Click the **Plot** button.

21 In the **Model Builder** window, under **Results>3D Plot Group 10** click **Line 1**.

22 In the **Line** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Beam>Section forces>Local axial force (beam.Nxl)**.

23 In the **Model Builder** window, under **Results>3D Plot Group 10** click **Line 1**.

24 In the **Line** settings window, locate the **Inherit Style** section.

25 From the **Plot** list, choose **None**.

26 Click to expand the **Range** section. Select the **Manual color range** check box.

27 In the **Minimum** edit field, type `-9e5`.

28 In the **Maximum** edit field, type `9e5`.

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

30 In the **Model Builder** window, under **Results>3D Plot Group 10** right-click **Surface 1** and choose **Disable**.

You can also look at the stress results for the different load cases including the truck weight.

Stress Top (shell)

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

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

3 From the **Load case** list, choose **2 trucks, position 4**.

4 Click the **Plot** button.

Create an animation of the trucks passing the bridge.

5 Click the **Player** button on the main toolbar.

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

6 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**.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Eigenfrequency**.

3 Click **Finish**.

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.

2 In the **Eigenfrequency** settings window, locate the **Study Settings** section.

3 In the **Desired number of eigenfrequencies** edit field, type 12.

4 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

Select the first mode involving the roadway.

RESULTS

Mode Shape (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.

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

3 From the **Eigenfrequency** list, choose **3.325624**.

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

Mode Shape (beam)

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

Mode Shape (shell)

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

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

3 In the **Line** settings window, locate the **Inherit Style** section.

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

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

6 In the **Deformation** settings window, locate the **Scale** section.

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

8 In the associated edit field, type `0.2`.

9 In the **Model Builder** window, under **Results>Mode Shape (shell)** click **Surface 1**.

10 In the **Surface** settings window, locate the **Range** section.

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

12 In the **Maximum** edit field, type `4`.

13 Click the **Plot** button.