Created in COMSOL Multiphysics 6.2

Stress Analysis of a Portal Crane

Introduction

Portal cranes, also known as gantry cranes, are common for handling heavy loads, for example in harbors and on industrial sites. A portal crane consists of a more or less rectangular frame, where the payload is hoisted from a trolley which runs along the upper horizontal beam called the bridge. Often, the crane runs on tracks in the direction perpendicular to the frame.

Figure 1: The Goliath crane in the shipyard of Harland & Wolff in Belfast.

In this example, a portal crane is analyzed using the Beam interface. The crane is subjected to loads from self-weight, payload, and thermal expansion.

Model Definition

GEOMETRY

An overview of the crane geometry is shown in Figure 2. Three different beam cross sections are used:

- The main horizontal beam, the bridge, has an HEA500 profile.

- The supporting columns have box cross sections, varying from 100 mm-by-100 mm at the ground level to 200 mm-by-100 mm at the connection to the bridge. The wall thickness is 10 mm.
- The horizontal crossbars between the columns have square box sections, 80 mm-by-80 mm with a wall thickness of 8 mm.

The geometry is parameterized, and the following values are used:

- Bridge width: 12 m.
- Crane height: 5 m.
- Distance between columns at ground plane: 2 m.

The material of all members is steel.

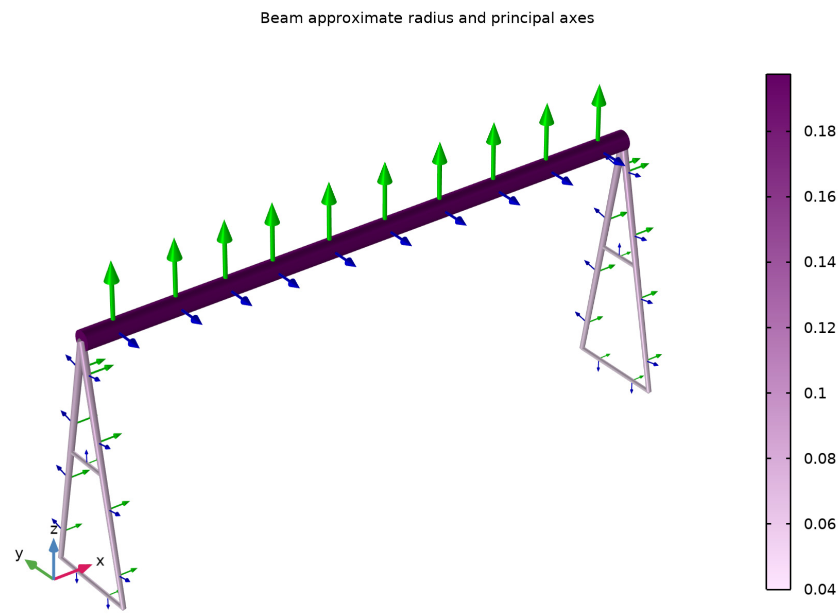

Figure 2: A sketch of the crane, with the stiffness of the members indicated (using size, color surface, and arrows in the principal directions).

BOUNDARY CONDITIONS

The columns are assumed to run on rails. The vertical and transverse displacements are constrained at all four lower ends of the columns. In addition, the displacement in the rail direction is constrained for two of the columns to make the model stable. Because all loads act in the vertical direction, this does not affect the results.

One end of the bridge is hinged with respect to its supporting columns. Because the structure is otherwise symmetric, the location is selected arbitrarily.

LOADING

Three different load cases are considered:

- Thermal load: The maximum temperature of the crane can rise to 50°C on a hot day. The stress-free assembly temperature is set to 20°C.
- Self weight: In addition to the weight of the frame, the weight of the trolley carrying the payload (200 kg) is taken into account.
- Payload: 15 ton are applied as a uniform load over a distance of 0.8 m of the bridge. This is the trolley width. The center of the trolley is placed at 3 m from the hinged end of the bridge.

Results and Discussion

The thermal expansion does not cause any stresses, since the frame is statically determinate. Obtaining a statically determinate structure is a reason for introducing hinges in this type of frame. A statically determinate structure has several advantages

- No stresses will be introduced during mounting, not even in case of geometrical mismatches due to manufacturing tolerances.
- A homogeneous increase in temperature will not introduce any stresses.
- The stress analysis is simplified, since the force distribution is not affected by the stiffness of individual members and joints.

The stress distribution caused by self-weight and payload are shown in Figure 3 and Figure 4, respectively.

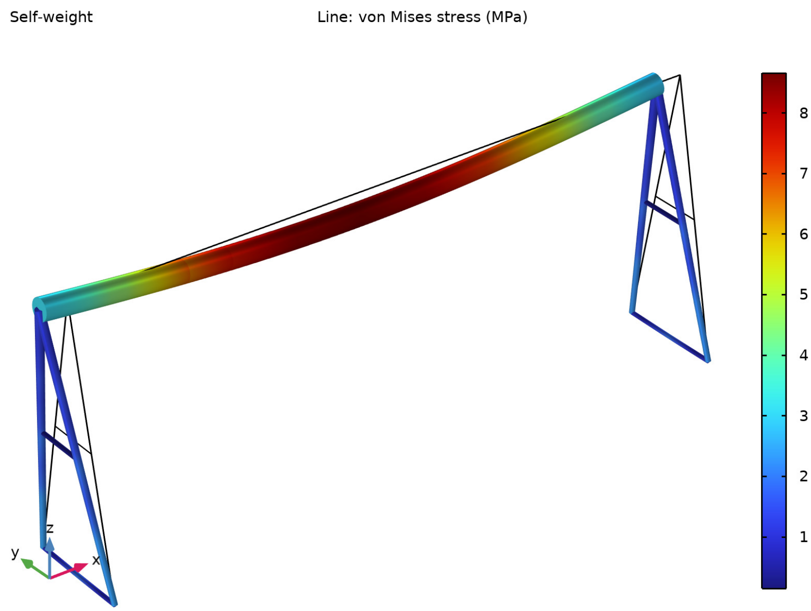

Figure 3: Equivalent stress caused by self-weight.

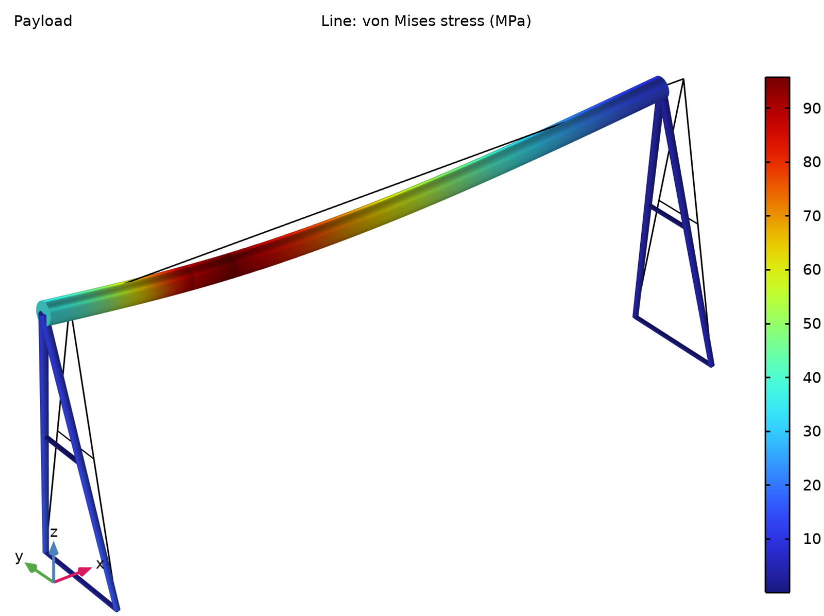

Figure 4: Equivalent stress caused by payload.

In a real-life analysis of this type of structure, there are several other effects that would have to be taken into account. For example:

- Dynamic effect of the payload. This can often be treated as a static load with a safety factor which allows for dynamic effects.
- Different trolley positions.
- Fatigue, since the trolley is moving.
- Local stresses at the joints between columns and bridge.
- Local stresses in the bridge under the trolley.

Notes About the COMSOL Implementation

The **Beam End Release** node is used to insert a hinge. When more than two beams meet at a point where some degrees of freedom are decoupled, it is necessary to specify how the beams are connected to each other. This is done by adding **Edge Group** subnodes. All edges that are selected in a single edge group are considered to be rigidly connected to each other at the joints. In this case the two column beams are placed in an edge group.

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/portal_crane`

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>Beam (beam)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

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 In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
`0`	`columnDistance/2`	`0`
`0`	`columnDistance/4`	`height/2`
`0`	`0`	`height`
`0`	`-columnDistance/4`	`height/2`
`0`	`-columnDistance/2`	`0`

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Columns` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Polygon**, click **Build Selected**.

Line Segment 1 (ls1)

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

2 On the object **pol1**, select Point 5 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **pol1**, select Point 1 only.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Crossbars` in the **Name** text field.

8 Click **OK**.

Line Segment 2 (ls2)

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

2 On the object **pol1**, select Point 4 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **pol1**, select Point 2 only.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Crossbars**.

7 Click **Build All Objects**.

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 **x** text field, type `width`.

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

5 Click **Build Selected**.

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

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
`0`	`0`	`height`
`trolleyPos-trolleyWidth/2`	`0`	`height`
`trolleyPos+trolleyWidth/2`	`0`	`height`
`width`	`0`	`height`

4 Click **Build All Objects**.

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>Structural steel**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

BEAM (BEAM)

Cross Section: Bridge

1 In the **Settings** window for **Cross-Section Data**, type `Cross Section: Bridge` in the **Label** text field.

2 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **H-profile**.

3 In the h_y text field, type `490[mm]`.

4 In the h_z text field, type `300[mm]`.

5 In the t_y text field, type `23[mm]`.

6 In the t_z text field, type `12[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
0	Y
1	Z

Cross Section: Columns

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

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

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

4 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **Box**.

5 In the h_y text field, type `100[mm]+100[mm]*(Z/height)`.

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

7 In the t_y text field, type `10[mm]`.

8 In the t_z text field, type `10[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

1	X
0	Y
0	Z

Cross Section: Crossbars

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

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

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

4 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **Box**.

5 In the h_y text field, type `80[mm]`.

6 In the h_z text field, type `80[mm]`.

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

8 In the t_z text field, type `8[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

1	X
0	Y
0	Z

Pinned 1

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

2 Select Points 1 and 8 only.

Prescribed Displacement/Rotation 1

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

2 Select Points 5 and 12 only.

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

4 From the **Displacement in x direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

Beam End Release 1

1 In the **Physics** toolbar, click **Points** and choose **Beam End Release**.

2 Select Point 3 only.

3 In the **Settings** window for **Beam End Release**, locate the **Release Settings** section.

4 Find the **Rotation** subsection. Select the **Release in Y direction** check box.

Edge Group 1

1 In the **Physics** toolbar, click **Attributes** and choose **Edge Group**.

Because three beams meet at the hinge, you must indicate how they are connected.

2 Select Edges 3 and 5 only.

Gravity 1

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

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

GLOBAL DEFINITIONS

Load Group: Gravity

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

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

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

BEAM (BEAM)

Trolley Self-Weight

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

2 In the **Settings** window for **Edge Load**, type `Trolley Self-Weight` in the **Label** text field.

3 Select Edge 8 only.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

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

0	x
0	y
-trolleyWeight*g_const	z

6 Locate the **Edge Selection** section. Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Trolley` in the **Selection name** text field.

8 Click **OK**.

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

Payload

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

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

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

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

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

0	x
0	y
-payload*g_const	z

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

GLOBAL DEFINITIONS

Load Group: Payload

1 In the **Model Builder** window, under **Global Definitions>Load and Constraint Groups** click **Load Group 2**.

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

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

BEAM (BEAM)

Linear Elastic Material 1

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

Thermal Expansion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 Click **Go to Source** for **Temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.

2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.

3 Find the **Expression for remaining selection** subsection. In the **Temperature** text field, type `maxTemp`.

BEAM (BEAM)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Beam (beam)> Linear Elastic Material 1** click **Thermal Expansion 1**.

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

GLOBAL DEFINITIONS

Load Group: Temperature

1 In the **Model Builder** window, under **Global Definitions>Load and Constraint Groups** click **Load Group 3**.

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

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

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**.

5 In the table, enter the following settings:

Load case	lgG	Weight	lgP	Weight	lgT	Weight
Self-weight	√	1.0		1.0		1.0

6 Click **Add**.

7 In the table, enter the following settings:

Load case	lgG	Weight	lgP	Weight	lgT	Weight
Payload		1.0	√	1.0		1.0

8 Click **Add**.

9 In the table, enter the following settings:

Load case	**lgG**	**Weight**	**lgP**	**Weight**	**lgT**	**Weight**
Temperature		1.0		1.0	√	1.0

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

RESULTS

Line 1

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

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

3 From the **Unit** list, choose **MPa**.

4 Click the **Show Grid** button in the **Graphics** toolbar.

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

The default plot shows the last load case; the thermal loading. The stresses are essentially zero since the frame is statically determinate. Next, consider the self-weight case.

6 Click **Plot First**.

The stresses from the self-weight are also small. This is what you would expect in a crane, since it should be possible to add a large payload. Next, move to the results for the payload.

7 Click **Plot Next**.

Check that the beams are correctly oriented.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Beam>Beam Orientation (beam)**.

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

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

RESULTS

Beam Orientation (beam)

1 In the **Beam Orientation (beam)** toolbar, click **Plot**.

The green arrows show the local Y directions, and the blue arrows show the local Z directions. The arrow sizes indicate the stiffness in each direction (actually the square root of the stiffness to give the arrows better visibility). The radius and the grayscale of the beam structure indicate the dimensions of the beam. Note the gradient in stiffness in the vertical direction of the columns.