Created in COMSOL Multiphysics 6.2

Large Deformation Analysis of a Beam

This model is licensed under the COMSOL Software License Agreement 6.2.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Model Definition

In this example you study the deflection of a cantilever beam undergoing very large deflections. The model is called “Straight Cantilever GNL Benchmark” and is described in detail in section 5.2 of NAFEMS *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks* (Ref. 1). A schematic description of the beam and its characteristics is shown in Figure 1.

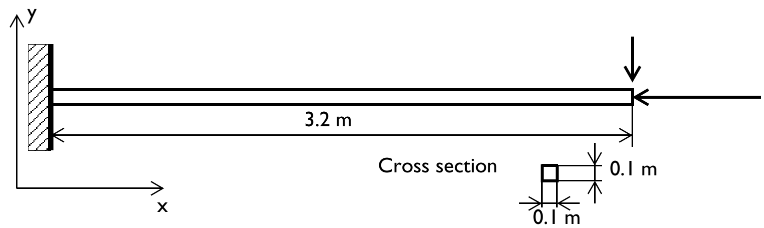

Figure 1: Cantilever beam geometry.

GEOMETRY

- The length of the beam is 3.2 m.
- The cross section is a square with side lengths 0.1 m.

MATERIAL

The beam is linear elastic with $E = 2.1\cdot10^{11}$ N/m^2 and $\nu = 0$.

CONSTRAINTS AND LOADS

- The left end is fixed.
- The right end is subjected to a total load of $F_x = -3.844\cdot10^6$ N and $F_y = -3.844\cdot10^3$ N.

MODELING IN COMSOL

This problem is modeled separately using both Solid Mechanics and Beam interfaces and the results are compared with the benchmark value. Using the Solid Mechanics interface, the problem is modeled as a “plane stress” problem considering that out-of-plane dimension is small. Poisson’s ratio ν is set to zero to make the boundary conditions consistent with the beam theory assumptions. The load on the right end of the beam is modeled as a uniformly distributed boundary load, corresponding to the specified total load.

In the second part of this problem, a linear buckling analysis study is carried out to compute the critical buckling load of the structure.

Results and Discussion

Due to the large compressive axial load and the slender geometry, this is a buckling problem. If you are to study the buckling and postbuckling behavior of a symmetric problem, it is necessary to perturb the symmetry somewhat. Here the small transversal load serves this purpose. An alternative approach would be to introduce an initial imperfection in the geometry.

Figure 2 below shows the final state with the **1:1** displacement scaling.

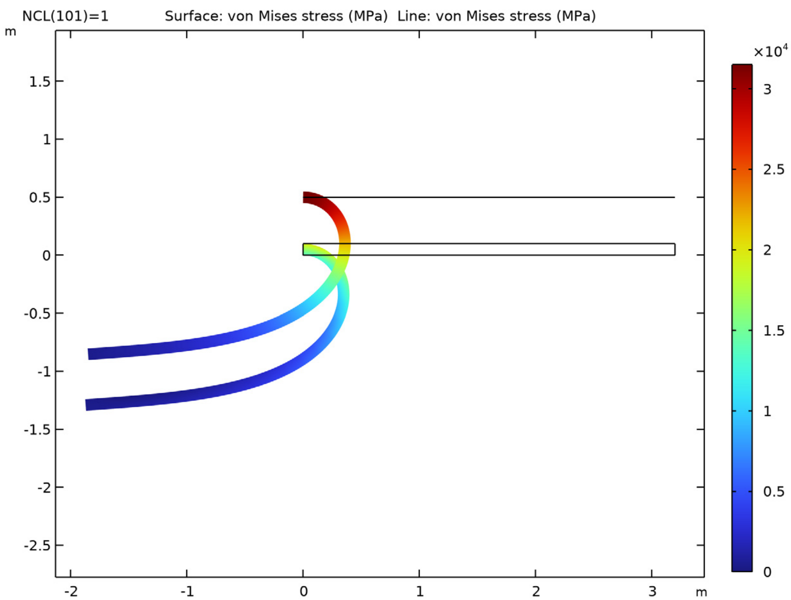

Figure 2: The effective von Mises stress of the deformed beam.

The horizontal and vertical displacements of the tip versus the compressive load normalized by its maximum value are shown in Figure 3.

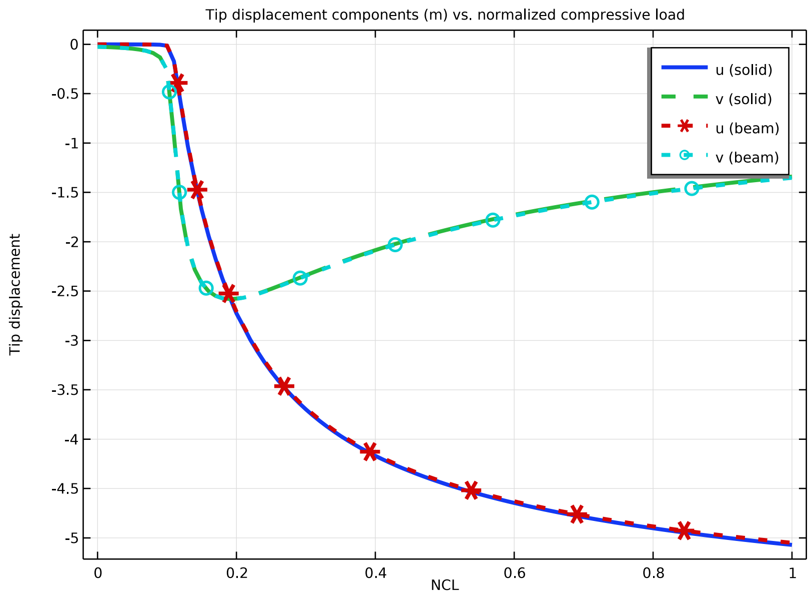

Figure 3: Horizontal and vertical tip displacements versus normalized compressive load.

Table 1 contains a summary of some significant results. Because the reference values are given as graphs, an estimate of the error caused by reading this graph is added:

TABLE 1: COMPARISON BETWEEN MODEL RESULTS AND REFERENCE VALUES.

QUANTITY	COMSOL (SOLID)	COMSOL (BEAM)	REFERENCE
Maximum vertical displacement at the tip	-2.58	-2.58	-2.58 ± 0.02
Final vertical displacement at the tip	-1.34	-1.35	-1.36 ± 0.02
Final horizontal displacement at the tip	-5.07	-5.05	-5.04 ± 0.04

The results are in excellent agreement, especially considering the coarse mesh used.

The plot of the axial deflection reveals that an instability occurs at a parameter value close to 0.1, corresponding to the compressive load $3.84 \cdot 10^5$ N. It is often seen in practice that the critical load of an imperfect structure is significantly lower than that of the ideal structure.

This problem (without the small transverse load) is usually referred to as the Euler-1 case. The theoretical critical load is

$$P_c = \frac{\pi^2 EI}{4L^2} = \frac{\pi^2 \cdot 2.1 \cdot 10^{11} \cdot (0.1^4/12)}{4 \cdot 3.2^2} = 4.22 \cdot 10^5 \text{ N}$$

Figure 4 shows the first buckling mode of the beam computed from a linear buckling analysis.

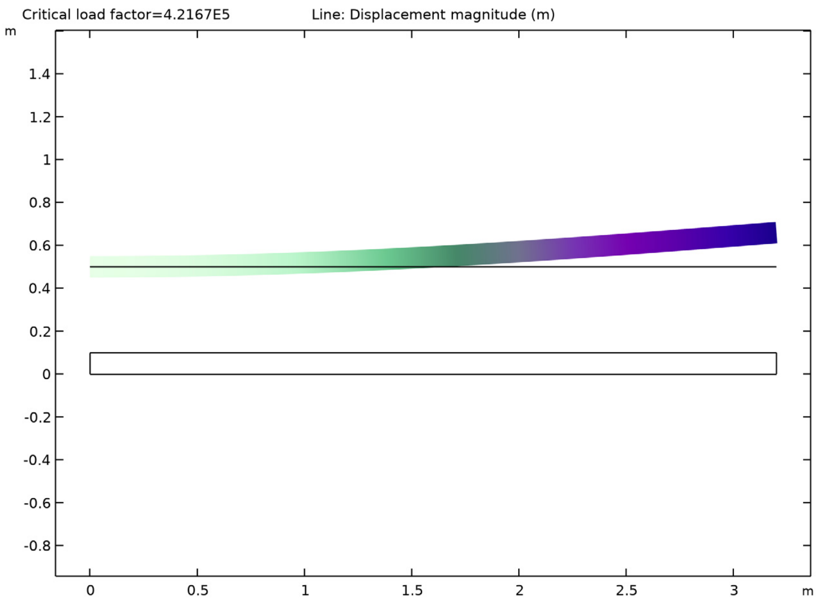

Figure 4: First buckling mode of the beam.

Reference

1. A.A. Becker, *Background to Finite Element Analysis of Geometric Non-linearity Benchmarks*, NAFEMS, Ref: -R0065, Glasgow, 1999.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/large_deformation_beam`

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 **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Structural Mechanics>Beam (beam)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
8 Click **Done**.

GLOBAL DEFINITIONS

Define parameters for the geometric data, compressive and transverse load components as well as a parameter that you will use to gradually turn up the compressive load.

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 `large_deformation_beam_parameters.txt`.

   By restricting the range of parameter **NCL** to [0, 1], it serves as a compressive load normalized by the maximum compressive load.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `1`.
4 In the **Height** text field, type `d`.

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (m)	y (m)
0	5*d
1	5*d

4 Click **Build All Objects**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

GLOBAL DEFINITIONS

In this example, the same material data will be referenced for the **Solid Mechanics** and **Beam** interfaces. Hence it can be added as a **Global Material**. Using a **Material Link** node, assign the **Global Material** to the domains, boundaries, and edges of the structure.

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Poisson's Ratio**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties>Young's Modulus**.

8 Click **Add to Material**.

9 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	7850	kg/m³	Basic
Poisson's ratio	nu	0	1	Basic
Young's modulus	E	2.1e5[MPa]	Pa	Basic

MATERIALS

Material Link 1 (matlnk1)

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

Material Link 2 (matlnk2)

1. Right-click **Materials** and choose **More Materials>Material Link**.
2. In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 4 only.

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

SOLID MECHANICS (SOLID)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2. In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.
3. From the list, choose **Plane stress**.
4. Locate the **Thickness** section. In the d text field, type `d`.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. Select Boundary 1 only.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundary 5 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. From the **Load type** list, choose **Total force**.
5. Specify the $\mathbf{F}_{tot}$ vector as

NCL*F_Lx	x
F_Ly	y

BEAM (BEAM)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Beam (beam)**.
2. In the **Settings** window for **Beam**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.

4 Select Boundary 4 only.

Cross-Section Data 1

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

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

3 From the **Section type** list, choose **Rectangle**.

4 In the h_y text field, type `d`.

5 In the h_z text field, type `d`.

Fixed Constraint 1

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

2 Select Point 3 only.

Point Load 1

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

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

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

NCL*F_Lx	x
F_Ly	y

4 Select Point 6 only.

Add another unit point load for the linear buckling analysis.

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

Point Load 2

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

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

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

-1	x
0	y

MESH 1

Edge 1

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

2 Select Boundaries 2–4 only.

Distribution 1

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

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

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

4 Select Boundary 4 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `40`.

Distribution 2

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

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

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

4 Select Boundaries 2 and 3 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `20`.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

STUDY 1

Step 1: Stationary

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

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

3 Select the **Include geometric nonlinearity** check box.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Beam (beam)>Point Load 2**.

6 Click **Disable**.

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list
NCL (Normalized compressive load)	`range(0,0.01,1)`

10 Right-click **Study 1>Step 1: Stationary** and choose **Get Initial Value for Step**.

STUDY 1

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Stationary Solver 1**.

2 In the **Settings** window for **Stationary Solver**, locate the **General** section.

3 In the **Relative tolerance** text field, type `1e-4`.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

5 Right-click **Stationary Solver 1** and choose **Segregated**.

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

7 From the **Termination technique** list, choose **Iterations**.

8 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step**.

9 In the **Settings** window for **Segregated Step**, locate the **General** section.

10 In the **Variables** list, select **Displacement field (material and geometry frames) (comp1.beam.uLin)**.

11 Under **Variables**, click **Delete**.

12 Under **Variables**, click **Delete**.

13 Click to expand the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

14 In the **Model Builder** window, right-click **Segregated 1** and choose **Segregated Step**.

15 In the **Settings** window for **Segregated Step**, locate the **General** section.

16 Under **Variables**, click **Add**.

17 In the **Add** dialog box, in the **Variables** list, choose **Rotation field (material and geometry frames) (comp1.beam.thLin)** and **Displacement field (material and geometry frames) (comp1.beam.uLin)**.

18 Click **OK**.

19 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

20 From the **Nonlinear method** list, choose **Automatic (Newton)**.

21 In the **Maximum number of iterations** text field, type 200.

22 In the **Tolerance factor** text field, type 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 **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Stress (beam)**.

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

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Results>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 Right-click **Line 1** and choose **Copy**.

Stress (solid)

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

Surface 1

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

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

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

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

6 Click **OK**.

Line 1

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

2 In the **Settings** window for **Line**, click to expand the **Inherit Style** section.

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

4 Clear the **Tube radius scale factor** check box.

Stress (solid and beam)

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

2 In the **Settings** window for **2D Plot Group**, type `Stress (solid and beam)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Material (X, Y, Z)**.

4 In the **Stress (solid and beam)** toolbar, click **Plot**.

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

Add a dataset to use for plotting of the results at the tip of the solid beam.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

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

3 In the **X** text field, type `1`.

4 In the **Y** text field, type `d/2`.

5 Click **Plot**.

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

Tip Displacement

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

2 In the **Settings** window for **1D Plot Group**, type `Tip Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.

Point Graph 1

1 Right-click **Tip Displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X-component**.

3 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`u (solid)`

Point Graph 2

1 In the **Model Builder** window, right-click **Tip Displacement** and choose **Point Graph**.

2. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>v - Displacement field, Y-component**.
3. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
4. From the **Width** list, choose **3**.
5. Locate the **Legends** section. Select the **Show legends** check box.
6. From the **Legends** list, choose **Manual**.
7. In the table, enter the following settings:

Legends
v (solid)

Point Graph 3

1. Right-click **Tip Displacement** and choose **Point Graph**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. Locate the **Selection** section. Click to select the **Activate Selection** toggle button.
5. Select Point 6 only.
6. Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Beam>Displacement>Displacement field - m>u2 - Displacement field, X-component**.
7. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
8. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
9. From the **Positioning** list, choose **Interpolated**.
10. From the **Width** list, choose **3**.
11. Locate the **Legends** section. Select the **Show legends** check box.
12. From the **Legends** list, choose **Manual**.
13. In the table, enter the following settings:

Legends
u (beam)

14. Right-click **Point Graph 3** and choose **Duplicate**.

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Beam> Displacement>Displacement field - m>v2 - Displacement field, Y-component**.

3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

4 From the **Positioning** list, choose **Interpolated**.

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

Legends
`v (beam)`

6 In the **Tip Displacement** toolbar, click **Plot**.

Tip Displacement

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Tip displacement components (m) vs. normalized compressive load`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Tip displacement`.

7 In the **Tip Displacement** toolbar, click **Plot**.

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

Evaluate the deformation of the structure.

Point Evaluation 1

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

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

3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 From the **Parameter selection (NCL)** list, choose **Last**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X-component**.

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

Expression	Unit	Description
u	m	Solid: x-disp

7 Click **Evaluate**.

8 Right-click **Point Evaluation 1** and choose **Duplicate**.

Point Evaluation 2

1 In the **Model Builder** window, click **Point Evaluation 2**.

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

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

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

5 Select Point 6 only.

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

Expression	Unit	Description
u2	m	Beam: x-disp
uFinal_Ref	m	Reference value for final horizontal displacement at the tip

7 Click next to **Evaluate**, then choose **Table 1 - Point Evaluation 1**.

Point Evaluation 1

In the **Model Builder** window, right-click **Point Evaluation 1** and choose **Duplicate**.

Point Evaluation 3

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
v	m	Solid: y-disp

4 Click next to **Evaluate**, then choose **New Table**.

Point Evaluation 2

In the **Model Builder** window, right-click **Point Evaluation 2** and choose **Duplicate**.

Point Evaluation 4

1 In the **Model Builder** window, click **Point Evaluation 4**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`v2`	`m`	`Beam: y-disp`
`vFinal_Ref`	`m`	`Reference value for final vertical displacement at the tip`

4 Click ▾ next to **Evaluate**, then choose **Table 2 - Point Evaluation 3**.

Point Evaluation 3

In the **Model Builder** window, right-click **Point Evaluation 3** and choose **Duplicate**.

Point Evaluation 5

1 In the **Model Builder** window, click **Point Evaluation 5**.

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

3 From the **Parameter selection (NCL)** list, choose **All**.

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

Expression	Unit	Description
`abs(v)`	`m`	`Solid: y-disp`

5 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

6 Click ▾ next to **Evaluate**, then choose **New Table**.

Point Evaluation 4

In the **Model Builder** window, right-click **Point Evaluation 4** and choose **Duplicate**.

Point Evaluation 6

1 In the **Model Builder** window, click **Point Evaluation 6**.

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

3 From the **Parameter selection (NCL)** list, choose **All**.

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

Expression	Unit	Description
`abs(v2)`	`m`	`Beam: y-disp`
`abs(vMax_Ref)`	`m`	

5 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

6 Click ▾ next to **Evaluate**, then choose **Table 3 - Point Evaluation 5**.

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 **Preset Studies for Selected Physics Interfaces>Linear Buckling**.

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: Stationary

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

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)**.

4 Click **Disable in Model**.

5 In the tree, select **Component 1 (comp1)>Beam (beam)>Point Load 1**.

6 Click **Disable**.

Step 2: Linear Buckling

1 In the **Model Builder** window, click **Step 2: Linear Buckling**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)**.

5 Click **Disable in Model**.

6 In the tree, select **Component 1 (comp1)>Beam (beam)>Point Load 1**.

7 Click **Disable**.

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

RESULTS

Mode Shape (beam)

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

Point Evaluation 7

1 In the **Results** toolbar, click **Point Evaluation**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Select Point 6 only.

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

Expression	Unit	Description
`Fcr`	`N`	`First critical buckling load`

6 Click **Evaluate**.