Instability of a Space Arc Frame

Model Definition

In this example you study the lateral deflection of a space frame subjected to concentrated vertical loading at four different points. A small lateral load is applied to break the symmetry of the structure. The model is described in detail in section 6.3 of Ref. 1, where it is called "Space frame subjected to concentrated loading". A schematic description of the frame and loads are shown in Figure 1. There are two types of members used in the frame, marked as 1 and 2 respectively.
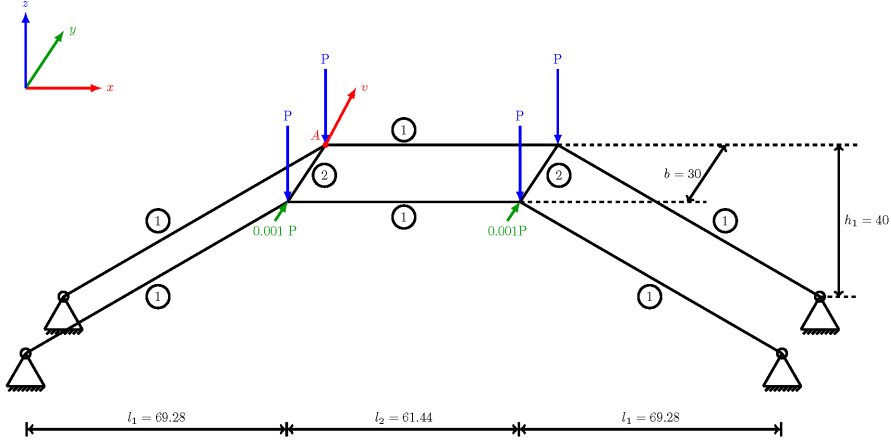

Figure 1: Space frame geometry.

GEOMETRY

- Cross section properties of type 1 members are $A_1 = 0.5$, $I_{y1} = 0.4$, $I_{z1}=0.133$.
- Cross section properties of type 2 members are $A_2 = 0.1$, $I_{y2} = 0.05$, $I_{z2}=0.05$.

The local y direction coincides with the global y direction.

The torsional constant is not supplied in the reference, so the common approximation J = $I_y + I_z$ is used.

MATERIAL

Linear elastic with $E = 4.32 \cdot 10^5$ and G = $1.66 \cdot 10^5$.

CONSTRAINTS AND LOADS

- All the base points of the frame are pinned.
- The four corners at the top are subjected to vertical loads P, ranging from 0 to 8.65, acting downward.

- The front two corners are subjected to lateral loads of $0.001 \cdot P$.

Results and Discussion

With only vertical loads active on the frame this is a symmetric problem. Hence, it is necessary to perturb the symmetry somewhat to induce a controlled instability. The small lateral loads serve this purpose. As an alternative, you could introduce an initial imperfection in the geometry.

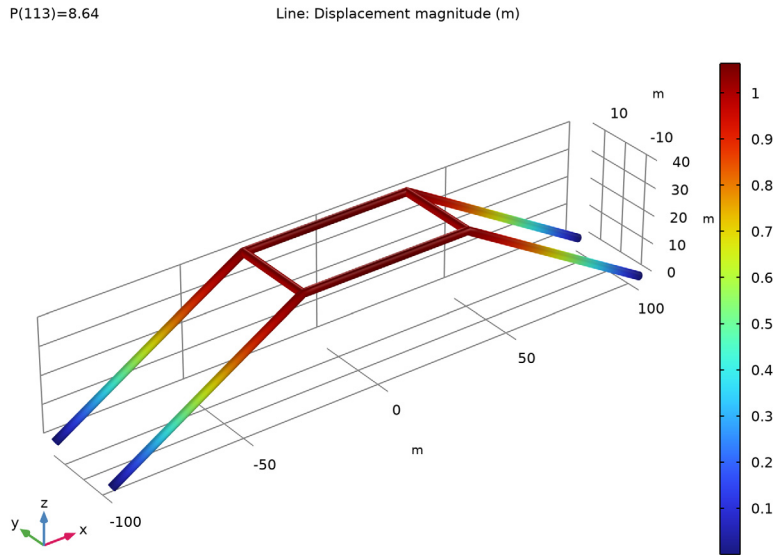Figure 2 below shows the final state of the frame.

P(113)=8.64   Line: Displacement magnitude (m)

Figure 2: Final state of the deformed frame.

The horizontal displacement of point A on the frame versus the compressive load is shown in Figure 3. Data obtained from Ref. 1 is marked on the same curve. The agreement with the data from the reference is very good.
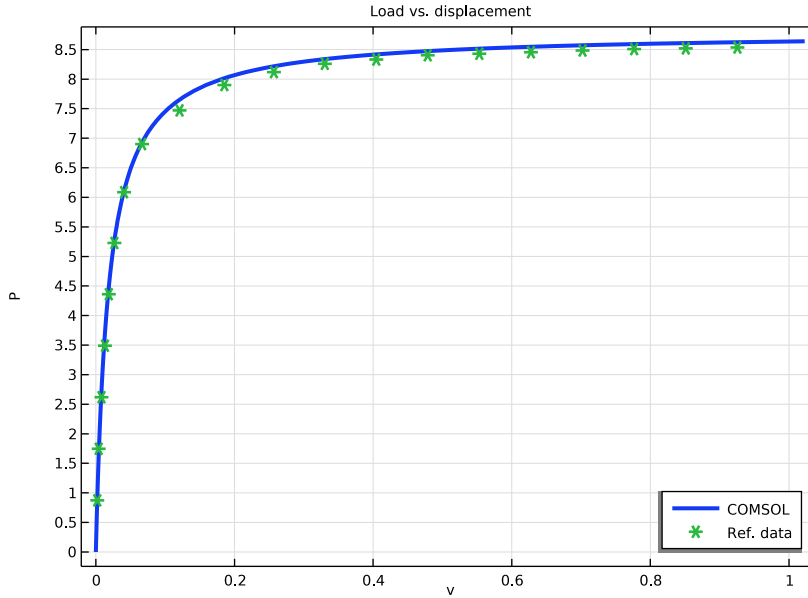
Figure 3: Load vs. displacement.

The plot of the lateral deflection shows that an instability occurs at a parameter value close to 8.0. In practice, the critical load of an imperfect structure is often significantly lower than that of the ideal structure.

Linear buckling analysis also gives the first critical buckling load as **8.67** which matches well with the critical load obtained from the above analysis. The corresponding buckling mode shape is shown in the Figure 4 below.

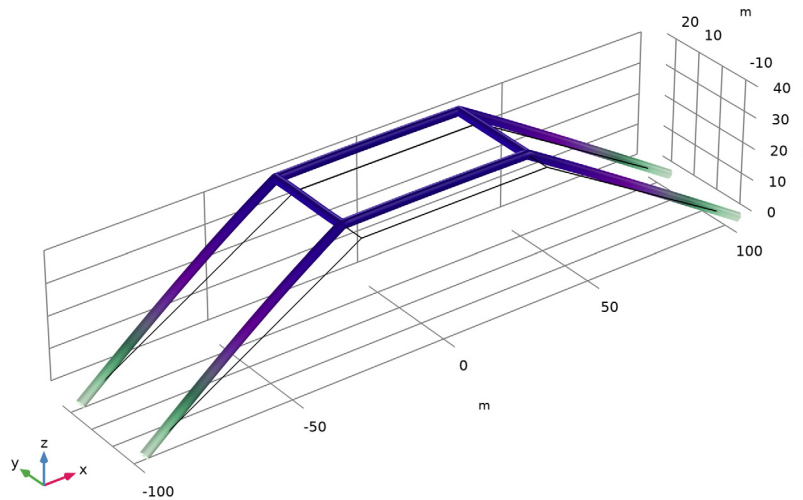Critical load factor=8.6708   Line: Displacement magnitude (m)

Figure 4: First buckling mode.

Reference

1. Z.X. Li and L. Vu-Quoc, *A Mixed Co-rotational 3D Beam Element for Arbitrarily Large Rotations*, Advanced Steel Construction Vol. 6, No. 2, 767-787, 2010.

Application Library path: `Structural_Mechanics_Module/ Verification_Examples/space_frame_instability`

Modeling Instructions

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

N E W

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

Define the load parameter as well as the geometric data.

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
    space_frame_instability_parameters.txt.

GEOMETRY 1

Since the frame is symmetric, create only one quarter of the geometry and use two mirror
operations to obtain the full geometry.

Polygon 1 (pol1)

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

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

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type -l1-l2/2 -l2/2 -l2/2 0.

5 In the **y** text field, type -b/2 -b/2 -b/2 -b/2.

6 In the **z** text field, type 0 h1 h1 h1.

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 -l2/2, **y** to -b/2, and **z** to h1.

6 Locate the **Endpoint** section. In the **x** text field, type -12/2 and **z** to h1.

Mirror 1 (mir1)

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

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

3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

5 Locate the **Point on Plane of Reflection** section. In the **x** text field, type -12/2.

6 In the **z** text field, type h1.

7 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type 1.

8 In the **z** text field, type 0.

Mirror 2 (mir2)

1 In the **Geometry** 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 Locate the **Normal Vector to Plane of Reflection** section. In the **x** text field, type 1.

6 In the **z** text field, type 0.

7 In the **Geometry** toolbar, click ▦ **Build All**.

8 Click the ⤢ **Go to Default View** button in the **Graphics** toolbar.

BEAM (BEAM)

Linear Elastic Material 1

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

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Young's modulus and shear modulus**.

MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	4.32e5	Pa	Young's modulus and shear modulus
Shear modulus	G	1.66e5	N/m²	Young's modulus and shear modulus
Density	rho	0	kg/m³	Basic

The density is set to zero since it is not used in the present analysis.

BEAM (BEAM)

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 In the A text field, type A1.

4 In the I_{zz} text field, type Iz1.

5 In the I_{yy} text field, type Iy1.

6 In the J text field, type Iy1+Iz1.

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 Data 2

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

2 Select Edges 3, 5, 9, and 11 only.

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

4 In the *A* text field, type A2.

5 In the I_{zz} text field, type Iz2.

6 In the I_{yy} text field, type Iy2.

7 In the *J* text field, type Iy2+Iz2.

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, 2, 11, and 12 only.

Point Load 1

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

2 Select Points 3, 5, 8, and 10 only.

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

4 Specify the $\mathbf{F}_P$ vector as

0	x
0	y
-P	z

Point Load 2

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

2 Select Points 3 and 8 only.

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

4 Specify the $\mathbf{F}_P$ vector as

0	x

0.001*P	y
0	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 **Fine**.

STUDY 1

Step 1: Stationary
Use geometric nonlinearity since the problem is expected to have an instability.

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.

Set up parametric sweep for the load.

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

5 Click ＋ **Add**.

Due to instability, the load increment for **P**>8 is reduced.

6 In the table, enter the following settings:

Parameter name	Parameter value list
P (Load)	range(0,0.1,8) range(8.02, 0.02, 8.65)

Solution 1 (sol1)
1 In the **Study** toolbar, click ┌┬┐ **Show Default Solver**.

Scale the dependent variables appropriately.

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

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

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

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

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Rotation field (comp1.beam.thLin)**.

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

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

9 In the **Scale** text field, type pi/10.

Increase the maximum allowed number of iterations due to the expected instability.

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

11 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

12 In the **Maximum number of iterations** text field, type 40.

13 In the **Study** toolbar, click ![Compute] **Compute**.

RESULTS

Displacement (beam)
In the **Settings** window for **3D Plot Group**, type Displacement (beam) in the **Label** text field.

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

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**.

3 In the **Displacement (beam)** toolbar, click ![Plot] **Plot**.

4 Click the ![Zoom Extents] **Zoom Extents** button in the **Graphics** toolbar.

Compare the load-displacement curve with the values from the reference.

Load vs. Displacement
1 In the **Home** toolbar, click ![Add Plot Group] **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type v.

5 Select the **y-axis label** check box. In the associated text field, type P.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type Load vs. displacement.

Point Graph 1

1 Right-click **Load vs. Displacement** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type P.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type beam.uLinY.

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

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

9 In the table, enter the following settings:

Legends
COMSOL

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

Reference Data

1 In the **Results** toolbar, click ⊞ **Table**.

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

3 Click ⊞← **Import**.

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

5 In the **Label** text field, type Reference Data.

Table Graph 1

1 In the **Model Builder** window, right-click **Load vs. Displacement** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

6 In the **Number** text field, type 20.

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

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

9 In the table, enter the following settings:

Legends
Ref. data

Load vs. Displacement

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

2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 In the **Load vs. Displacement** toolbar, click ◉ **Plot**.

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

Next, you verify the critical buckling load by performing the **linear buckling** analysis.

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

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

The default plot from the buckling analysis shows the first buckling mode shape as shown in Figure 4.

RESULTS

Mode Shape (beam)

1 In the **Mode Shape (beam)** toolbar, click ◉ **Plot**.

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