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 $E = 4.32 \times 10^5$ and $G = 1.66 \times 10^5$.

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 downwards.
The front two corners are subjected to lateral loads of 0.001\( P \).

**Results and Discussion**

With only a 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.

*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.
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. Corresponding buckling mode shape is shown in the Figure 4 below.

Figure 3: Load vs. displacement.
Figure 4: First buckling mode.

Reference


Application Library path: Structural_Mechanics_Module/Verification_Examples/space_frame_instability

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 **Preset Studies>Stationary**.
6. Click **Done**.

**GLOBAL DEFINITIONS**

Define the load parameter as well as the geometric data.

**Parameters**

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

*Bézier Polygon 1 (b1)*

1. On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.
2. In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
3. Find the **Added segments** subsection. Click **Add Linear**.
4. Find the **Control points** subsection. In row 1, set \( x \) to \(-11 - 12/2\) and \( y \) to \(-b/2\).
5. In row 2, set \( x \) to \(-12/2\), \( y \) to \(-b/2\), and \( z \) to \(h1\).
6. Find the **Added segments** subsection. Click **Add Linear**.
7. Find the **Control points** subsection. In row 2, set \( x \) to 0.

*Bézier Polygon 2 (b2)*

1. On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.
2. In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
3. Find the **Added segments** subsection. Click **Add Linear**.
4. Find the **Control points** subsection. In row 1, set \( x \) to \(-12/2\), \( y \) to \(-b/2\), and \( z \) to \(h1\).
5 In row 2, set \( x \) to \(-12/2\) and \( z \) to \( h_1 \).

*Mirror 1 (mir1)*

1 On 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 \( y \) text field, type 0.
7 In the \( z \) text field, type \( h_1 \).
8 Locate the *Normal Vector to Plane of Reflection* section. In the \( y \) text field, type 1.
9 In the \( z \) text field, type 0.

*Mirror 2 (mir2)*

1 On 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 On the *Geometry* toolbar, click *Build All*.
8 Click the *Go to Default View* button on 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:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>E</td>
<td>4.32e5</td>
<td>Pa</td>
<td>Basic</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>G</td>
<td>1.66e5</td>
<td>N/m²</td>
<td>Bulk modulus and shear modulus</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>0</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

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 Basic Section Properties section.

3 In the A text field, type A1.

4 In the Izz text field, type Iz1.

5 In the Iyy text field, type Iy1.

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

Section Orientation 1

1 In the Model Builder window, expand the Cross Section Data 1 node, then click Section Orientation 1.

2 In the Settings window for Section Orientation, locate the Section Orientation section.

3 From the Orientation method list, choose Orientation vector.

4 Specify the V vector as

<table>
<thead>
<tr>
<th>0</th>
<th>x</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>y</td>
</tr>
<tr>
<td>0</td>
<td>z</td>
</tr>
</tbody>
</table>
Cross Section Data 2
1 On 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 Basic Section Properties section.
4 In the \(A\) text field, type \(A_2\).
5 In the \(I_{zz}\) text field, type \(I_{z2}\).
6 In the \(I_{yy}\) text field, type \(I_{y2}\).
7 In the \(J\) text field, type \(I_{y2}+I_{z2}\).

Section Orientation 1
1 In the Model Builder window, expand the Cross Section Data 2 node, then click Section Orientation 1.
2 In the Settings window for Section Orientation, locate the Section Orientation section.
3 From the Orientation method list, choose Orientation vector.
4 Specify the \(V\) vector as

\[
\begin{array}{c}
1 \\
0 \\
0
\end{array} \begin{array}{c}
x \\
y \\
z
\end{array}
\]

Pinned 1
1 On the Physics toolbar, click Points and choose Pinned.
2 Select Points 1, 2, 11, and 12 only.

Point Load 1
1 On 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 \(F_P\) vector as

\[
\begin{array}{c}
0 \\
0 \\
-P
\end{array} \begin{array}{c}
x \\
y \\
z
\end{array}
\]
2 Select Points 3 and 8 only.

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

4 Specify the $F_p$ vector as

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>x</td>
</tr>
<tr>
<td>0.001*P</td>
<td>y</td>
</tr>
<tr>
<td>0</td>
<td>z</td>
</tr>
</tbody>
</table>

**MESH 1**

1 In the Model Builder window, under Component 1 (comp1) click Mesh 1.

2 In the Settings window for Mesh, locate the Mesh Settings 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. Locate 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:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P$</td>
<td>range(0, 0.1, 8)</td>
</tr>
<tr>
<td></td>
<td>range(8.02, 0.02, 8.65)</td>
</tr>
</tbody>
</table>

**Solution 1 (sol1)**

1 On 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 Locate the **Method and Termination** section. In the **Maximum number of iterations** text field, type 40.

13 On the **Study** toolbar, click **Compute**.

**RESULTS**

**Stress (beam)**

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

2 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 **Results>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>Beam>Displacement> beam.disp - Total displacement**.

3 On the **Displacement (beam)** toolbar, click **Plot**.

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

   Compare load-displacement curve with values from the reference.
**ID Plot Group 9**

1. On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
2. In the **Settings** window for **ID Plot Group**, type **Load vs displacement** in the **Label** text field.
3. Locate the **Plot Settings** section. Select the **x-axis label** check box.
4. In the associated text field, type **v**.
5. Select the **y-axis label** check box.
6. In the associated text field, type **P**.
7. Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
8. 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:

<table>
<thead>
<tr>
<th></th>
<th>COMSOL</th>
</tr>
</thead>
</table>

10. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type **3**.

**Table 1**

1. On 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 **Settings** window for **Table**, type **Ref data** in the **Label** text field.
Table Graph 1
1 In the **Model Builder** window, under **Results** 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 In the **Number** text field, type 20.

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

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

8 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Legends</th>
<th>Ref. data</th>
</tr>
</thead>
</table>

**Load vs displacement**

1 In the **Model Builder** window, under **Results** click **Load vs displacement**.

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

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

4 On the **Load vs displacement** toolbar, click **Plot**.

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

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

**ROOT**

On the **Home** toolbar, click **Windows** and choose **Add Study**.

**ADD STUDY**

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**>
   **Linear Buckling**.

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

4 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

**STUDY 2**

**Step 1: Stationary**

On the **Home** toolbar, click **Compute**.
First default plot from the buckling analysis shows the first buckling mode shape as shown in Figure 4.

RESULTS

Mode Shape (beam)

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

2 On the Mode Shape (beam) toolbar, click Plot.

3 Click the Zoom Extents button on the Graphics toolbar.