Created in COMSOL Multiphysics 6.2

Linear Buckling Analysis of a Truss Tower with Dead Loads

Introduction

Trusses are commonly used to create light structures that can support heavy loads. When designing such a structure, it is important to ensure its safety. For a tower made of bars, buckling can cause the structure to collapse due to compressive loads that come from external vertical loading and from the self weight of the structure. Linear buckling analysis is a common way to calculate the critical buckling load. The calculated critical load factor is usually interpreted as a common multiplicative factor on all loads. Sometimes, you want to evaluate the critical value for a specific load, keeping some other loads constant. Distinguishing between constant (dead) loads and varying (live) loads makes it possible to get the critical load factor for only certain loads.

This example shows how to perform a linear bucking analysis with dead loads on a guyed truss tower where the pretension force in the guy wires and the gravity force are the dead loads. The vertical force at the top of the tower is the live load.

Model Definition

The model geometry consists of a 19 m tall truss tower with a rectangular section. The critical buckling load is computed using the linear buckling analysis available in the Truss interface.

The geometry is the periodic structure represented in Figure 1. It consists of 19 blocks of trusses. Each block has a width of 0.45 m, a depth of 0.40 m and a height of 1.0 m. The trusses that are perpendicular to the ground are thicker and have an outer radius of 15 mm and an inner radius of 10 mm. The remaining trusses have an outer radius of 10 mm and an inner radius of 7 mm. The tower is made out of structural steel, which is one of the predefined materials in the material library. Two sets of four guys are fixed at each corner of the tower, at two different heights. The tower and guys are fixed at the ground level, the guys are prestressed at 4 kN. Gravity is applied to the structure. A vertical load is applied at the four top points, one fourth of the unit load each. Hence, the critical load factor returned by the linear buckling analysis corresponds to the external load that would cause the collapse of the structure.

The guys are modeled using the Wire interface. As long as the guys are in tension, they act as bars, so it would also have been possible to model them in the Truss interface.

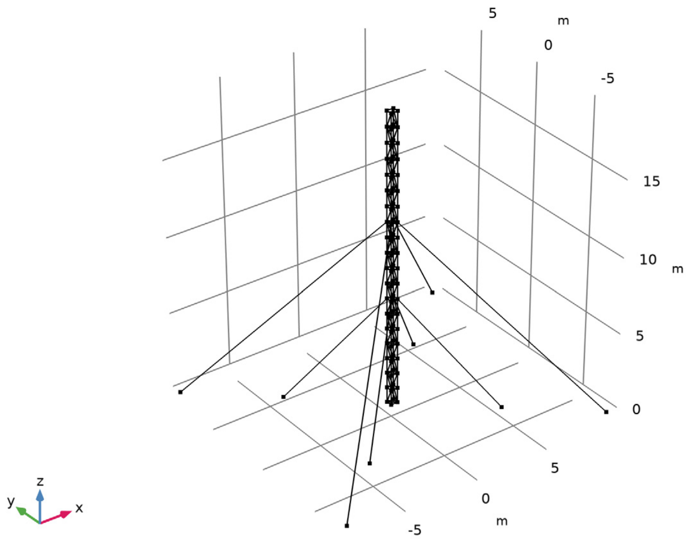


Figure 1: Geometry of the guyed truss tower.

In linear buckling theory the stiffness matrix K is separated into a linear part and a linearized nonlinear part which is proportional to the applied load

$$K = K_L + \lambda K_{NL}(P_0)$$

In the presence of dead loads, the loads are separated into constant dead loads Q_0 and live loads P_0

$$.K = K_L + K_{NL}(Q_0) + \lambda K_{NL}(P_0)$$

For a better linearization of nonlinear problems, the live loads are not calculated separately, but associated with the dead loads. Hence, an initial stationary solution is calculated with dead loads only. Then, a second solution is calculated with dead and live loads. The stiffness matrix is then

$$K = K_L + K_{NL}(Q_0) + \lambda(K_{NL}(Q_0 \cup P_0) - K_{NL}(Q_0))$$

$$K = K_L + (1 - \lambda)K_{NL}(Q_0) + \lambda K_{NL}(Q_0 \cup P_0)$$

These two solutions are then selected as the linearization point and live load solution in the linear buckling study step.

LOCAL BUCKLING

In addition to the buckling of the whole structure, the safety against buckling of individual truss members must be studied. The critical compressive load for an individual bar can be calculated from material properties and the geometry of the bar:

$$F_{c,\,\mathrm{bar}} = \frac{\pi^2 E I_{\mathrm{min}}}{(K_{\mathrm{bar}} L_{\mathrm{bar}})^2}$$

Here, I_{min} is the moment of inertia in the weakest direction, based on the cross section data, and L_{bar} is the length of the bar. In this case the effective length factor K_{bar} is kept to its default value of 1, which corresponds to a pinned-pinned configuration. From this critical force and the compressive axial force N, the failure index f_{i} and the local buckling safety factor s_{f} can be calculated:

$$f_{\mathrm{i}} = \frac{-N}{F_{c,\,\mathrm{bar}}}$$

$$s_{\mathrm{f}} = \frac{1}{f_{\mathrm{i}}}$$

In presence of dead loads, the aim is to find the multiplicative factor to live loads which is needed to reach the critical force:

$$-N(Q_0) + \lambda(-N(Q_0 \cup P_0) + N(Q_0)) = F_{c,\,\mathrm{bar}}$$

$$f_{\mathrm{i}}(Q_0) + \lambda(f_{\mathrm{i}}(Q_0 \cup P_0) - f_{\mathrm{i}}(Q_0)) = 1$$

Thus, the safety factor of the live load can be calculated from the live and dead load solutions:

$$s_{\mathrm{f}}(P_0) = \frac{1 - f_{\mathrm{i}}(Q_0)}{f_{\mathrm{i}}(Q_0 \cup P_0) - f_{\mathrm{i}}(Q_0)}$$

Results and Discussion

The two first buckling modes, corresponding respectively to buckling in the depth direction (y direction) and width direction, are shown in Figure 2 and Figure 3. The prestressed guys make it possible to increase the critical load factor up to 92 kN and 117 kN, respectively.

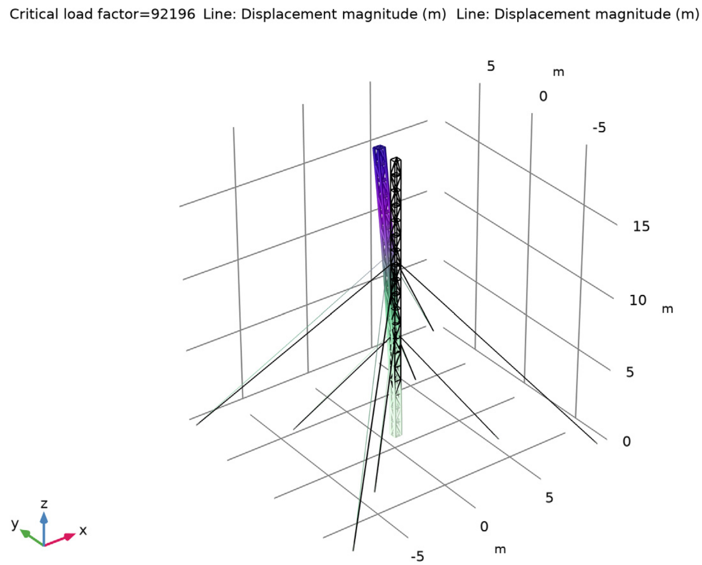


Figure 2: First buckling mode.

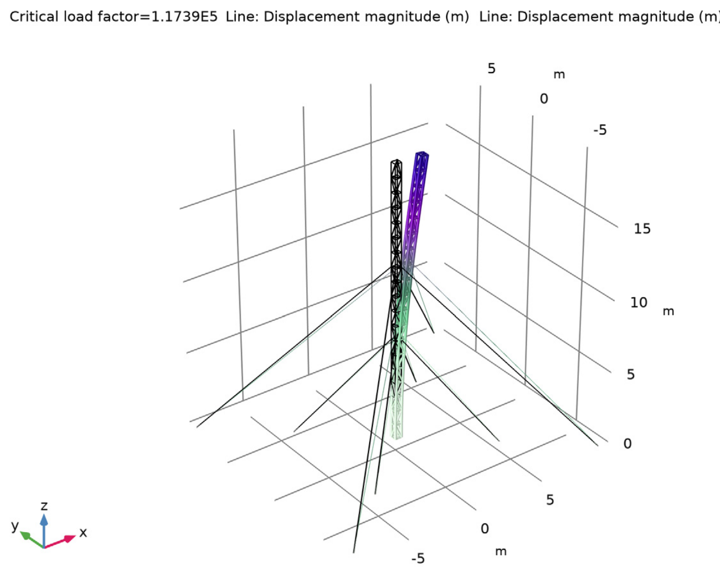


Figure 3: Second buckling mode.

Figure 4 plots the local buckling index for live loads in the tower. The calculated minimal value for safety factor is 131 kN for the diagonal bars at the base of the tower.

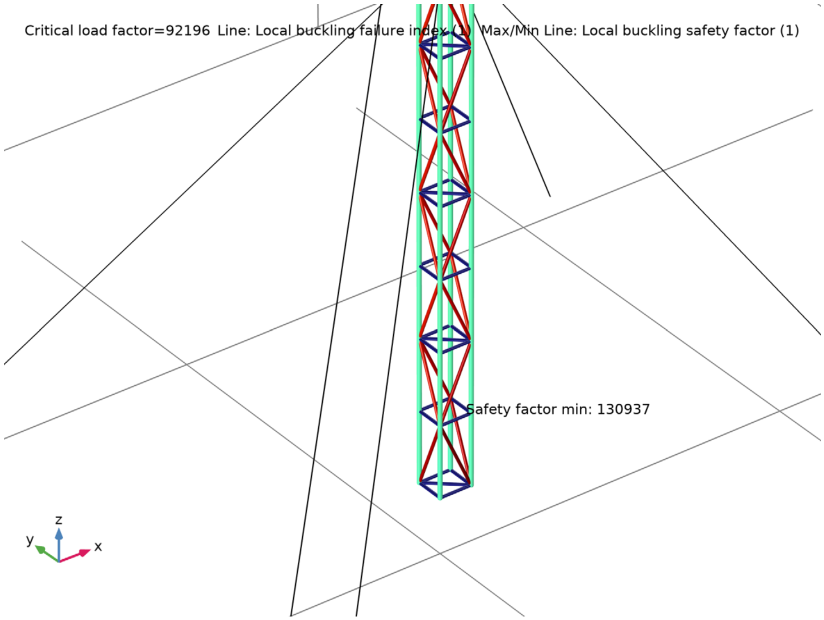


Figure 4: Maximum local buckling index, with value of minimal safety factor.

Notes About the COMSOL Implementation

The **Linear Buckling** study step has a section for setting the linearization point and the live load solutions. In the absence of dead loads, only one solution is needed, so the live load option must be kept at the default setting **Same as linearization point**. When dead loads are present, two different solutions must be selected. They can come, for example, from two different study steps, from two different parameter values of an auxiliary sweep, or from two load cases. In this model, two stationary study steps are used, the first one with live load disabled.

Application Library path: `Structural_Mechanics_Module/Buckling_and_Wrinkling/truss_tower_buckling_guys`

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>Truss (truss)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Wire (wire)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Linear Buckling**.

8 Click **Done**.

GLOBAL DEFINITIONS

Tower Geometric Parameters

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

2 In the **Settings** window for **Parameters**, type `Tower Geometric Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Guy Geometric Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Guy Geometric Parameters` in the **Label** text field.

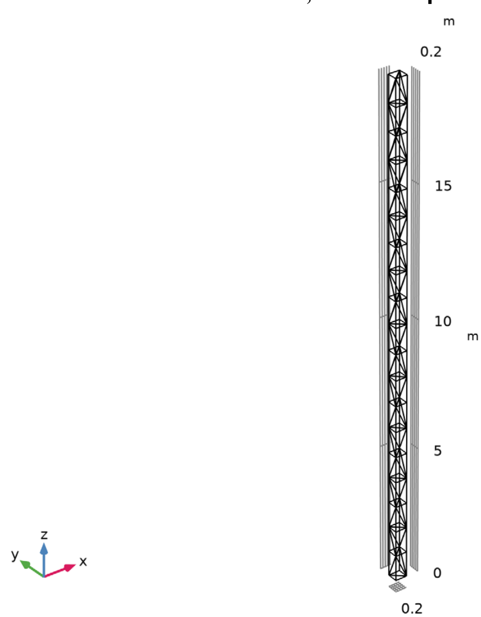
3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
gnd_attach_guy1	6[m]	6 m	Ground attachment distance of guy 1
gnd_attach_guy2	12[m]	12 m	Ground attachment distance of guy 2
guy_d	5[mm]	0.005 m	Diameter of guy 1 and guy 2

Name	Expression	Value	Description
guy_height1	7[m]	7 m	Height of guy 1
guy_height2	12[m]	12 m	Height of guy 2
guy_area	pi/4*guy_d^2	1.9635E-5 m²	Area of guy 1 and guy 2
kA	3.927e6[N]	3.927E6 N	Axial stiffness
rhoL	0.15413[kg/m]	0.15413 kg/m	Mass per unit length

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `truss_tower_buckling.mph`.
3 In the **Geometry** toolbar, click **Build All**.
4 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.



Array 1 (arr1), Array 2 (arr2), Block 1 (blk1), Convert to Curve 1 (ccur1), Line Segment 1 (ls1), Line Segment 2 (ls2), Mirror 1 (mir1), Polygon 1 (pol1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Block 1 (blk1)**, **Polygon 1 (pol1)**, **Line Segment 1 (ls1)**, **Line Segment 2 (ls2)**, **Convert to Curve 1 (ccur1)**, **Mirror 1 (mir1)**, **Array 1 (arr1)**, and **Array 2 (arr2)**.

2 Right-click and choose **Group**.

Tower

In the **Settings** window for **Group**, type `Tower` in the **Label** text field.

Line Segment 3 (ls3)

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 In the **z** text field, type `guy_height1-mod(guy_height1,height)`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **x** text field, type `-gnd_attach_guy1*width/sqrt(width^2+depth^2)`.

7 In the **y** text field, type `-gnd_attach_guy1*depth/sqrt(width^2+depth^2)`.

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

9 In the **New Cumulative Selection** dialog box, type `Guy 1` in the **Name** text field.

10 Click **OK**.

11 In the **Settings** window for **Line Segment**, click **Build All Objects**.

Line Segment 4 (ls4)

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 In the **z** text field, type `guy_height2-mod(guy_height2,height)`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **x** text field, type `-gnd_attach_guy2*width/sqrt(width^2+depth^2)`.

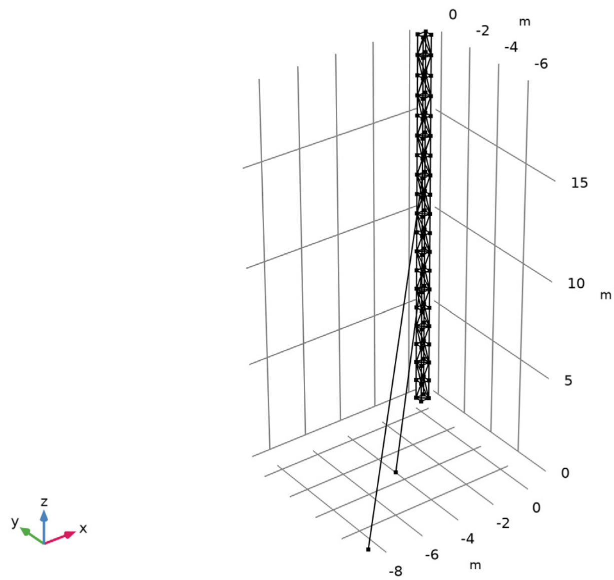
7 In the **y** text field, type `-gnd_attach_guy2*depth/sqrt(width^2+depth^2)`.

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

9 In the **New Cumulative Selection** dialog box, type `Guy 2` in the **Name** text field.

10 Click **OK**.

11 In the **Settings** window for **Line Segment**, click **Build All Objects**.



Mirror 2 (mir2)

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

2 Select the objects **ls3** and **ls4** only.

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 `width/2`.

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

7 In the **z** text field, type `0`.

8 Click **Build All Objects**.

Mirror 3 (mir3)

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

2 Select the objects **ls3**, **ls4**, **mir2(1)**, and **mir2(2)** only.

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 **y** text field, type `depth/2`.

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

7 In the **z** text field, type `0`.

8 Click **Build All Objects**.

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

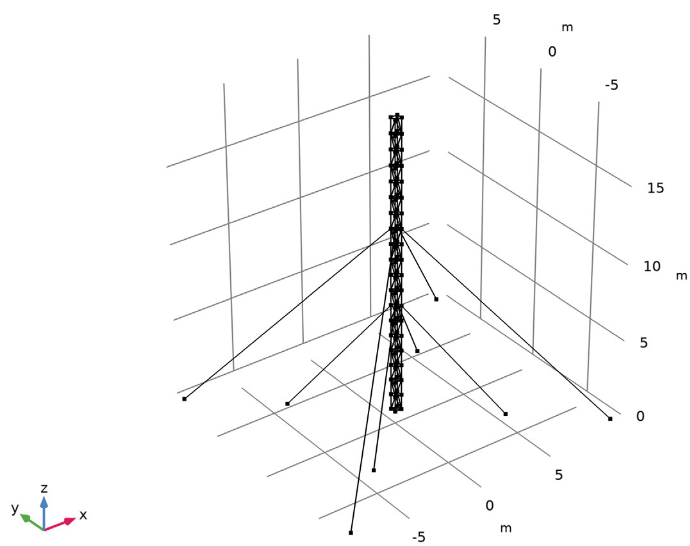
Line Segment 3 (ls3), Line Segment 4 (ls4), Mirror 2 (mir2), Mirror 3 (mir3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Line Segment 3 (ls3)**, **Line Segment 4 (ls4)**, **Mirror 2 (mir2)**, and **Mirror 3 (mir3)**.

2 Right-click and choose **Group**.

Guys

1 In the **Settings** window for **Group**, type `Guys` in the **Label** text field.



DEFINITIONS

Vertical Edges

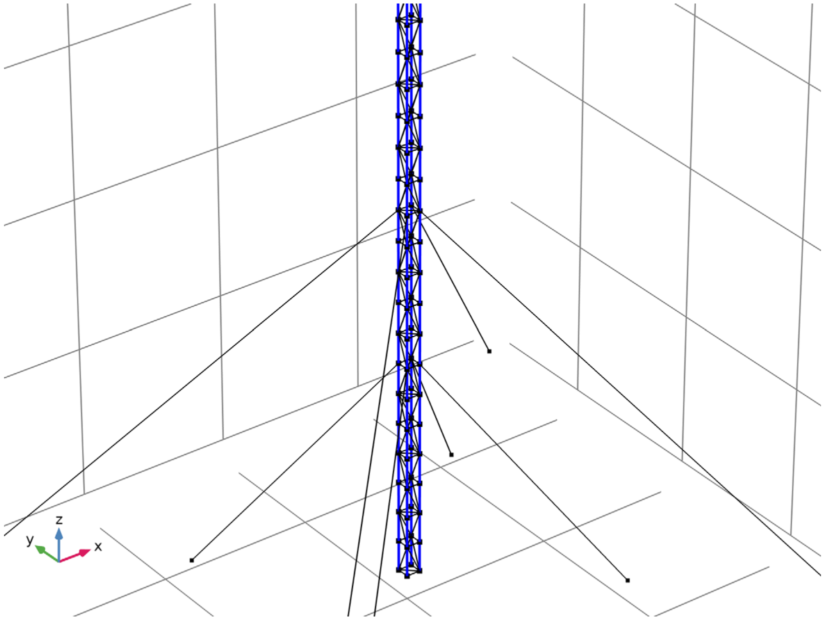
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Vertical Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 5, 112, 180, and 240 only.

5 Select the **Group by continuous tangent** check box.



Transversal Truss Edges

1 In the **Definitions** toolbar, click **Complement**.

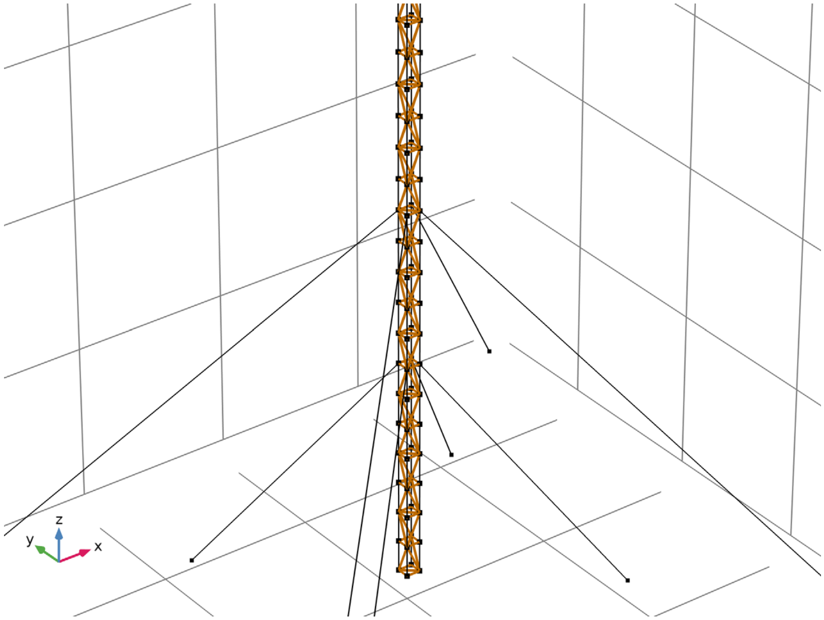
2 In the **Settings** window for **Complement**, type `Transversal Truss Edges` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog box, in the **Selections to invert** list, choose **Vertical Edges**, **Guy 1**, and **Guy 2**.

6 Click **OK**.



Tower

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Tower` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Vertical Edges** and **Transversal Truss Edges**.

6 Click **OK**.

Guys

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Guys` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Guy 1** and **Guy 2**.

6 Click **OK**.

TRUSS (TRUSS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Truss (truss)**.

2 In the **Settings** window for **Truss**, locate the **Edge Selection** section.

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

Cross-Section Data (Vertical Edges)

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

2 In the **Settings** window for **Cross-Section Data**, type `Cross-Section Data (Vertical Edges)` in the **Label** text field.

3 Locate the **Cross-Section Definition** section. From the **Section type** list, choose **Pipe**.

4 In the d_o text field, type `do1`.

5 In the d_i text field, type `di1`.

Cross-Section Data (Other Truss Edges)

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

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

3 From the **Selection** list, choose **Transversal Truss Edges**.

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

5 In the d_o text field, type `do2`.

6 In the d_i text field, type `di2`.

7 In the **Label** text field, type `Cross-Section Data (Other Truss Edges)`.

Pinned 1

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

2 Select Points 1–5, 25, 45, 65, and 85–88 only.

Gravity 1

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

Point Load 1

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

2 Select Points 24, 44, 64, and 84 only.

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

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

0	x
0	y
-1[N]/4	z

WIRE (WIRE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Wire (wire)**.

2 In the **Settings** window for **Wire**, locate the **Edge Selection** section.

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

Use the same dependent variable in both physics interfaces in order to share the displacement where the guys are attached to the tower.

4 Click to expand the **Dependent Variables** section. In the **Displacement field (m)** text field, type `u`.

Elastic Wire 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Wire (wire)** click **Elastic Wire 1**.

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

3 In the A text field, type `guy_area`.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Edge Selection** section.

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

4 Locate the **Initial Stress and Strain** section. In the N_i text field, type `4[kN]`.

Pinned 1

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

2 Select Points 1–4 and 85–88 only.

Gravity 1

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

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 Click **Add to Component** in the window toolbar.

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

MATERIALS

Structural steel (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Tower**.

Wire material

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

2 In the **Settings** window for **Material**, type `Wire material` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Guys**.

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

Property	Variable	Value	Unit	Property group
Axial stiffness	k_A	kA	N	Elastic wire
Mass per unit length	rho_L	rhoL	kg/m	Elastic wire

MESH 1

Edge 1

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

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

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

Distribution 1

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 1.

Distribution 2

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

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

3 From the **Selection** list, choose **Guy 1**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `10`.

Distribution 3

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

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

3 From the **Selection** list, choose **Guy 2**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `10`.

5 Click **Build All**.

The buckling study requires two stationary study steps to compute live and dead load solutions.

STUDY 1

Step 3: Stationary 2

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 Right-click **Step 3: Stationary 2** and choose **Move Up**.

Use the first stationary study step to solve for the dead loads only.

Stationary, Dead Loads Only

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

2 In the **Settings** window for **Stationary**, type `Stationary, Dead Loads Only` in the **Label** text field.

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

4 In the tree, select **Component 1 (comp1)>Truss (truss)>Point Load 1**.

5 Click **Disable**.

In the second stationary study step, both live and dead loads are applied.

Stationary, Dead + Live Loads

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

2 In the **Settings** window for **Stationary**, type `Stationary, Dead + Live Loads` in the **Label** text field.

You need to generate the solver sequence to enable the selection of solutions in the linear buckling steps.

Solution 1 (sol1)

In the **Study** toolbar, click **Show Default Solver**.

Step 3: Linear Buckling

1. In the **Model Builder** window, under **Study 1** click **Step 3: Linear Buckling**.
2. In the **Settings** window for **Linear Buckling**, locate the **Study Settings** section.
3. In the **Desired number of buckling modes** text field, type 2.
4. Locate the **Values of Linearization Point** section. From the **Settings** list, choose **User controlled**.
5. From the **Use** list, choose **Solution Store 1 (sol2)**.
6. Locate the **Live Loads Solution** section. From the **Settings** list, choose **User controlled**.
7. In the **Study** toolbar, click **Compute**.

RESULTS

Line 1

1. In the **Model Builder** window, expand the **Mode Shape (truss)** node, then click **Line 1**.
2. In the **Settings** window for **Line**, locate the **Coloring and Style** section.
3. In the **Radius scale factor** text field, type 6.

Line 1

1. In the **Model Builder** window, expand the **Results>Mode Shape (wire)** node.
2. Right-click **Line 1** and choose **Copy**.

Mode Shape (truss)

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

Line 2

1. In the **Model Builder** window, click **Line 2**.
2. In the **Settings** window for **Line**, click to expand the **Inherit Style** section.
3. From the **Plot** list, choose **Line 1**.

Mode Shape (truss)

1. In the **Model Builder** window, click **Mode Shape (truss)**.
2. In the **Mode Shape (truss)** toolbar, click **Plot**.

Add variables to assess the risk of local buckling.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
lbs_f	(1-withsol('sol2', truss.lbf_i))/(withsol('sol3', truss.lbf_i)-withsol('sol2', truss.lbf_i))		Local buckling safety factor
lbf_i	1/lbs_f		Local buckling failure index

STUDY 1

In the **Study** toolbar, click **Update Solution**.

RESULTS

Local Buckling

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Local Buckling` in the **Label** text field.

Line 1

1 Right-click **Local Buckling** and choose **Line**.

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

3 In the **Expression** text field, type `lbf_i`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

5 From the **Line type** list, choose **Tube**.

6 In the **Tube radius expression** text field, type `truss.re`.

7 Select the **Radius scale factor** check box. In the associated text field, type `2`.

8 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

9 In the **Local Buckling** toolbar, click **Plot**.

Local Buckling

In the **Model Builder** window, click **Local Buckling**.

Max/Min Line 1

1. In the **Local Buckling** toolbar, click **More Plots** and choose **Max/Min Line**.
2. In the **Settings** window for **Max/Min Line**, locate the **Expression** section.
3. In the **Expression** text field, type `lbs_f`.
4. Locate the **Display** section. From the **Display** list, choose **Min**.
5. Locate the **Text Format** section. In the **Prefix** text field, type `Safety factor `.
6. In the **Local Buckling** toolbar, click **Plot**.