Created in COMSOL Multiphysics 6.0

Hanging Cable

Introduction

A cable is a structural member that has stiffness only in its tangential direction, but virtually no bending stiffness. When supported only at its two ends, it deflects under gravitational load. An idealized hanging cable can be analyzed analytically and forms a deflection curve known as a catenary (derived from the Latin word *catena* for chain). In COMSOL Multiphysics, hanging cables, chains, or strings can be modeled with the Truss interface.

Model Definition

The example consists of a cable carrying two street lamps. The model setup, the global coordinate system and relevant properties are indicated in Figure 1. The cable has an initial length L, and it is supported at two points spaced by a length $d < L$ apart from each other. Three load cases are analyzed:

1 Cable only with gravitational load
2 Cable and lamps with gravitational load
3 Cable and lamps with gravitational *and* wind loads

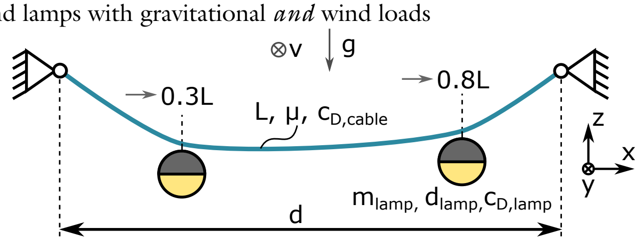

Figure 1: Cable carrying two lamps. The gravitational acceleration is parallel to the z direction, and the wind is parallel to the y direction.

The lamps are assumed to be spherical in shape and mounted at distances $0.3L$ and $0.8L$ along the cable's initial length. The wind vector is assumed to point in the positive y direction, and the forces acting on the lamps and the cable are estimated as

$$F_D = c_D \frac{\rho}{2} v^2 A$$

where c_D is the drag coefficient, ρ the air density, and A the reference area for the drag.

The cable is made of steel (see Table 1), and behaves linear elastically.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	CABLE
Young's modulus	200 GPa
Poisson's ratio	0.3
Density	7850 kg/m^3

An analytical solution can be derived for the first load case (a cable with gravitational load only). The cable setup and relevant properties are indicated in Figure 2. To the right, a free-body diagram is shown for the cable section between points A and B.

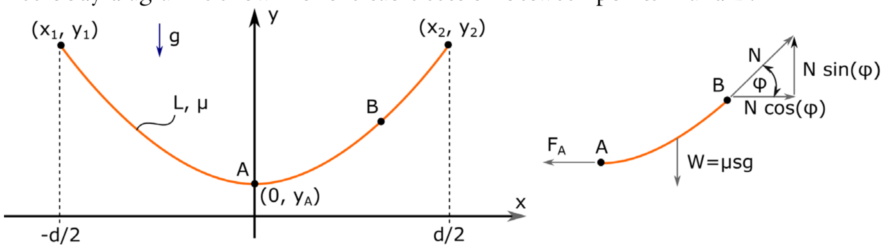

Figure 2: Deformed cable under its self-weight. A free-body diagram for the cable segment between points A and B is shown to the right.

As before, the cable has an initial length L, and its support points are spaced by a distance d apart from each other. The analytical treatment assumes that the stiffness in tangential direction is high, and any cable elongation is negligible; that is, the length of the deformed cable is still equal to the original length L.

The analytical treatment of a hanging cable is a classical problem of calculus of variations. The derivation of the catenary can be found in many text books. Typically, the curve is derived as a constrained variation problem, by constructing a functional that incorporates the potential energy of the system plus a constraint equation (the arc length of the deflected cable is L) with a Lagrange multiplier. The application of the Euler–Lagrange equations then lead to a differential equation which when solved yields the catenary curve.

From a mechanical perspective, the catenary can also be derived by simply formulating the equilibrium equations using a free-body diagram in a deformed configuration, as shown on the right in Figure 2. Choosing point A to be the lowest deflection point simplifies the analysis. Force balance in the x and y directions yield

$$N\cos\varphi - F_A = 0 \text{ and } N\sin\varphi - W = 0$$

where N is the tangential force at point B, F_A the tangential force at point A (only acting in the horizontal direction). The variable W is the weight given as $\mu g s$, where μ is the mass per length, g the gravitational acceleration, and s is the arc length of the cable segment. Rearranging and dividing the equations yields

$$\frac{\sin\varphi}{\cos\varphi} = \tan\varphi = \frac{dy}{dx} = \frac{\mu g s}{F_A}.$$

Here, we have also used that $\tan(\phi)$ is the slope of the curve dy/dx. Differentiation with respect to x yields a 2nd-order nonlinear differential equation for the deflection $y(x)$:

$$y'' = \frac{\mu}{F_A} \cdot \frac{ds}{dx} = \frac{\mu g}{F_A} \cdot \sqrt{1+y'^2} = \frac{1}{a} \cdot \sqrt{1+y'^2}$$

In the last step, the constant $F_A/(\mu g)$ was replaced by a different constant, a. The general solution for this differential equation is

$$y(x) = a\sinh(c_1)\sinh\left(\frac{x}{d}\right) + a\sinh(c_1)\sinh\left(\frac{x}{d}\right)$$

where the constants c_1 and c_2 can be found from the boundary and symmetry conditions. This leads to the final expression for the catenary:

$$y(x) = a\left[\cosh\left(\frac{x}{a}\right) - \cosh\left(\frac{d}{2a}\right)\right].$$

The expression contains the unknown parameter a, which can be found by using the constraint that the length of the catenary must equal L:

$$L = \int_{\frac{d}{2}}^{-\frac{d}{2}} \sqrt{1+y'^2}\,dx = 2a\sinh\left(\frac{d}{2a}\right)$$

This equation can be only be solved numerically for a. Note that the solution does not depend on any material properties. Cables of equal lengths and uniform density take on the same deformed shape, independently of their respective weights.

The computed and analytical solutions agree very well when considering the first load case (cable deforming under its own weight); see Figure 3.

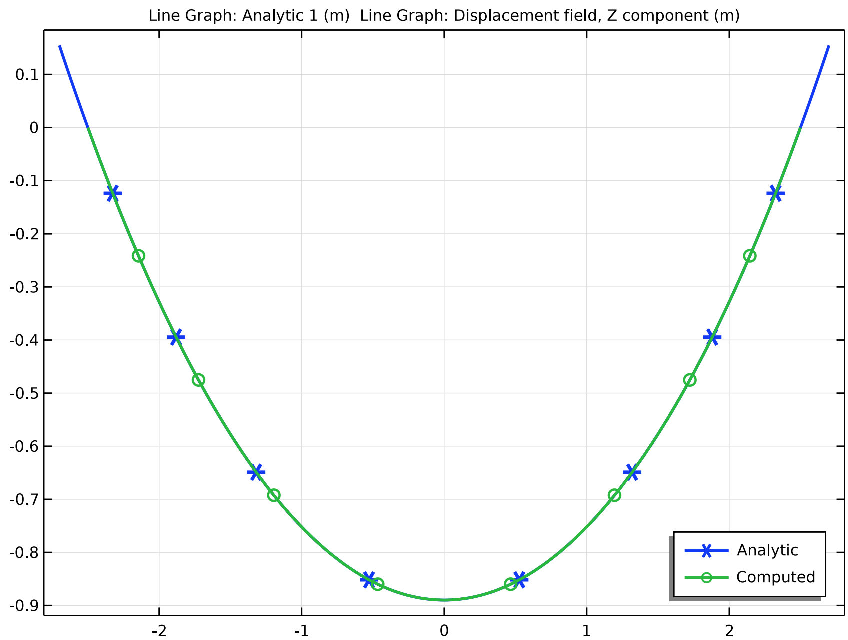

Figure 3: Cable sag due to self-weight (load case 1). Comparison of the analytical and the solution computed using the Truss interface.

Figure 4 shows the vertical sag (w) for all three load cases, as well as the deflection magnitude (in the y and z directions) for the last load case which includes the wind load in the y direction. The lamps are hung asymmetrically, leading to the nonsymmetric deflection curves. The cable is stiff in its axial direction and does not exhibit any significant elongation. With gravity and wind loads acting on the lamps, the maximum cable deflection, or sag, decreases.

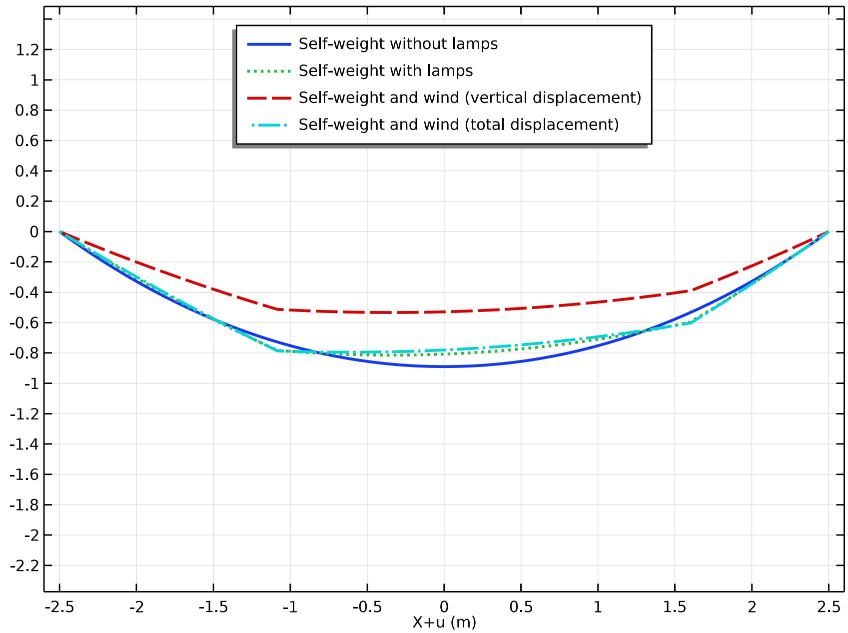

Figure 4: Vertical and total cable sag for the three load cases.

The distribution of the axial forces are compared in Figure 5.

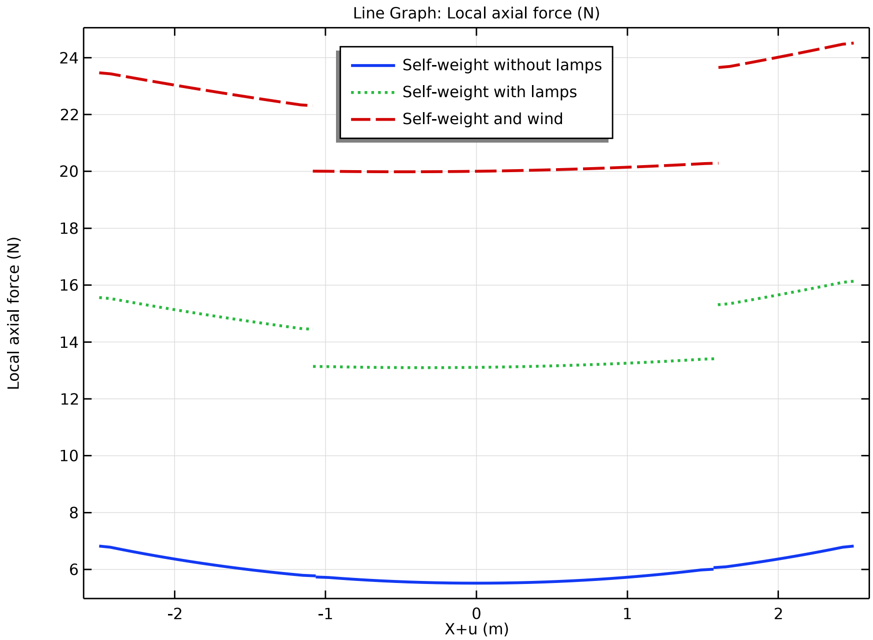

Figure 5: Axial force along the cable for the three load cases.

Notes About the COMSOL Implementation

The analytic expression for the catenary includes a scaling factor a. As shown above, its value is expressed in terms of a transcendental function; that is, no closed-form solution exists. However, the value for a can be easily obtained numerically. To do so in COMSOL Multiphysics, a **Global ODEs and DAEs** interface is added to the model.

Most cable problems are geometrically nonlinear. A wire which is not in tension is numerically unstable. Physically, it wrinkles in an unpredictable manner. In order to start the analysis in this case, an initial stress is added by giving an approximate expression for the expected deflection using the **Initial Values** feature.

Analyzing this type of problem poses some numerical challenges. In a cable which is not pretensioned, the strains will typically be very small. Still the force balance is computed from the stress in each element, a stress that is proportional to the strain. The strains are computed as derivatives of the displacements, which typically are large. The numerics will thus contain computing a small difference between two large numbers. This is the reason why it is recommended to use linear element shape functions for this type of analysis.

More information and tips about the modeling of cables can be found in the section *Modeling with Truss Elements* in the *Structural Mechanics Module User's Guide*.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/hanging_cable`

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 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 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`L_cable`	`5.4[m]`	5.4 m	`Cable length`
`u_support`	`0.2[m]`	0.2 m	`End point displacement`

Name	Expression	Value	Description
d_support	L_cable - 2* u_support	5 m	Support distance
m_lamp	0.5[kg]	0.5 kg	Lamp mass
D_cable	5[mm]	0.005 m	Cable diameter
D_lamp	400[mm]	0.4 m	Lamp diameter
q_dyn	(1.225[kg/m^3]/2)* (15[m/s])^2	137.81 Pa	Dynamic pressure

To easily activate or deactivate gravity or wind loads, add two load groups.

Lamp Weight

1 Right-click **Global Definitions>Parameters 1** and choose **Load Group**.

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

Wind Load

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Load Group**.

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

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **More Primitives>Polygon**.

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

4 In the table, enter the following settings:

x (m)	y (m)	z (m)
-L_cable/2	0	0
L_cable/2	0	0

Partition Edges 1 (pare1)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **pol1**, select Edge 1 only.

3 In the **Settings** window for **Partition Edges**, locate the **Positions** section.

4 In the table, enter the following settings:

Relative arc length parameters
0.3
0.8

5 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.

TRUSS (TRUSS)

Cross-Section Data 1

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

2 In the A text field, type `pi*(D_cable/2)^2`.

Prescribed Displacement 1

1 In the **Model Builder** window, right-click **Truss (truss)** and choose **Prescribed Displacement**.

2 Select Points 1 and 4 only.

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

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `-sign(X)*u_support`.

6 Select the **Prescribed in y direction** check box.

7 Select the **Prescribed in z direction** check box.

Gravity 1

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

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

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

Edge Load: Wind on Cable

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

2 In the **Settings** window for **Edge Load**, type `Edge Load: Wind on Cable` in the **Label** text field.

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

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

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

0	x
1.1*q_dyn*(L_cable*D_cable)	y
0	z

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

Point Load: Wind on Lamps

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

2 In the **Settings** window for **Point Load**, type `Point Load: Wind on Lamps` in the **Label** text field.

3 Select Points 2 and 3 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_{\mathrm{P}}$ vector as

0	x
0.45*q_dyn*pi*(D_lamp/2)^2	y
0	z

5 In the **Physics** toolbar, click **Load Group** and choose **Wind Load**.

Point Mass: Lamps

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

2 In the **Settings** window for **Point Mass**, type `Point Mass: Lamps` in the **Label** text field.

3 Select Points 2 and 3 only.

The lamp mass can be included in load group 1 by conditionally activating the mass using the load group's weight factor.

4 Locate the **Point Mass** section. In the *m* text field, type `if(group.lg1, group.lg1*m_lamp, 0)`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

0	X
0	Y
(-L_cable/2 + X)*(L_cable/2 + X)*(2/L_cable^2)*sqrt(L_cable^2 - d_support^2)	Z

A cable which is not in tension is not numerically stable. To improve solver convergence an initial guess is provided in terms of a parabolic function.

MESH 1

Edge 1

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

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

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `L_cable/60`.

5 In the **Minimum element size** text field, type `L_cable/60`.

6 Click **Build All**.

GLOBAL DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `y_catenary` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `a*(cosh(X/a) - cosh(0.5*d_support/a))`.

4 In the **Arguments** text field, type `X, a, d_support`.

5 Locate the **Units** section. In the **Function** text field, type `m`.

6 In the table, enter the following settings:

Argument	Unit
X	m
a	m
d_support	m

The analytic solution, `y_catenary`, takes the parameter a as an input. Its value can be obtained from a transcendental function which can be easily solved for with a **Global ODEs and DAEs** interface.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>0D**.

ADD PHYSICS

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

2 Go to the **Add Physics** window.

3 In the tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

4 Click **Add to Component 2** in the window toolbar.

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

GLOBAL ODES AND DAES (GE)

Global Equations 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
a	L_cable - 2*a* sinh(d_support/(2*a))	3	0	Catenary parameter

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `length` in the text field.

6 Click Filter.

7 In the tree, select **General>Length (m)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, select **General>Length (m)** in the tree.

12 Click **OK**.

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 Click to expand the **Study Extensions** section. Select the **Define load cases** check box.

5 Click **Add** three times.

6 In the table, enter the following settings:

Load case	lg1	Weight	lg2	Weight
`Self-weight without lamps`		1.0		1.0
`Self-weight with lamps`	√	1.0		1.0
`Self-weight and wind`	√	1.0	√	1.0

Solution 1 (sol1)

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

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

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

4 In the **Relative tolerance** text field, type `1e-6`.

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

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

7 From the **Nonlinear method** list, choose **Constant (Newton)**.

8 In the **Maximum number of iterations** text field, type `250`.

9 In the **Study** toolbar, click **Compute**.

RESULTS

Line 1

1 In the **Model Builder** window, expand the **Results>Force (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 `10`.

Line 1

1 In the **Model Builder** window, expand the **Results>Stress (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 `10`.

Displacement

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Axis** section. Select the **Preserve aspect ratio** check box.

5 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Line Graph 1

1 Right-click **Displacement** and choose **Line Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `w`.

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

6 In the **Expression** text field, type `X+u`.

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
`Self-weight without lamps`
`Self-weight with lamps`
`Self-weight and wind (vertical displacement)`

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 In the **Width** text field, type `2`.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

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

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

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `-sqrt(v^2+w^2)`.

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

Legends
`Self-weight and wind (total displacement)`

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

Force

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

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

3 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

Line Graph 1

1 Right-click **Force** and choose **Line Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `truss.Nxl`.

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

6 In the **Expression** text field, type `X+u`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

8 In the **Width** text field, type `2`.

9 Locate the **Legends** section. Select the **Show legends** check box.

10 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

11 In the **Force** toolbar, click **Plot**.

Analytic vs. Computed Solution

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

2 In the **Settings** window for **1D Plot Group**, type `Analytic vs. Computed Solution` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (Load case)** list, choose **First**.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1 Right-click **Analytic vs. Computed Solution** and choose **Line Graph**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `y_catenary(X, comp2.a, d_support)`.

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

6 In the **Expression** text field, type `X`.

7 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

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

9 Locate the **Legends** section. Select the **Show legends** check box.

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

11 In the table, enter the following settings:

Legends
Analytic

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

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

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `X+u`.

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

Legends
Computed

6 In the **Analytic vs. Computed Solution** toolbar, click **Plot**.

Maximum Sag

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

2 In the **Settings** window for **Evaluation Group**, type `Maximum Sag` in the **Label** text field.

Line Maximum 1

1 Right-click **Maximum Sag** and choose **Maximum>Line Maximum**.

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

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

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

Expression	Unit	Description
abs(w)	cm	Maximum vertical sag (computed)

Line Maximum 2

1 In the **Model Builder** window, right-click **Maximum Sag** and choose **Maximum>Line Maximum**.

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

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

4 From the **Parameter selection (Load case)** list, choose **First**.

5 Locate the **Selection** section. From the **Selection** list, choose **All edges**.

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

Expression	Unit	Description
-y_catenary(X, comp2.a, d_support)	cm	Maximum vertical sag (analytic)

7 In the **Maximum Sag** toolbar, click **Evaluate**.