Created in COMSOL Multiphysics 5.6

Maximizing the Buckling Load of a Diagonal Brace

Introduction

Buckling is a common failure mode for slender structures. One can distinguish between tensile and compressive modes, but it is rare to see buckling failure due to a tensile load. The buckling load can normally be increased by adding material, but oftentimes it is possible to achieve the same effect by relocating material.

Model Definition

The model studies a structural member, which provides diagonal stiffness in a larger structure. The buckling load of the member is maximized by changing two geometrical parameters, the thickness and the width. The case of a member with a length of 4 m as well as 6 m are studied. The member has a slender geometry as illustrated in Figure 1.

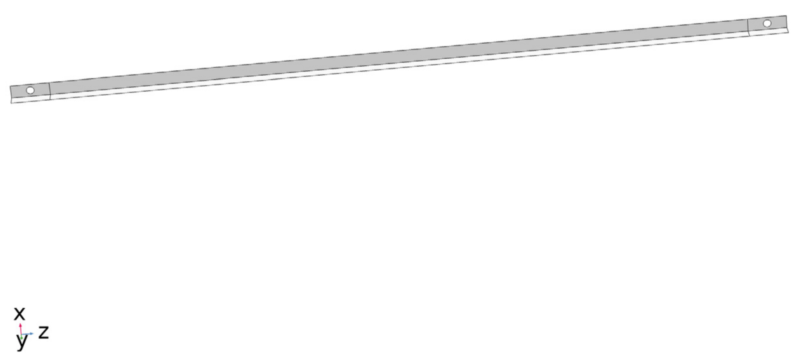

Figure 1: The computational domain is symmetric, but this is not exploited in the model.

The member is made of steel, and the thickness is small, so its structural response can be well characterized using a shell model. The brace is put in compression using a **Rigid Connector** boundary condition with an **Applied Force** feature. The initial critical load factor and volume is first calculated and then a second study dealing with the optimization is set up.

Results

The result of the optimization for a length of 6 m is shown in Figure 2.The Buckling load has been increased by 140%. The bound of the volume constraint is scaled with the brace length, so the 6 m brace is 50% heavier than the 4 m brace. This causes the optimization result for a 4 m brace to have a smaller width and larger thickness than the 6 m brace.

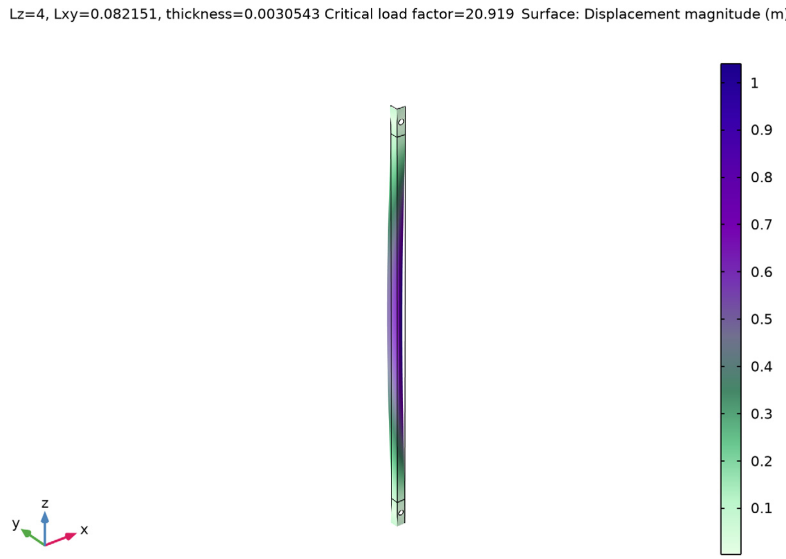

Figure 2: The optimized 6 m member is shown for the 1st buckling mode.

Application Library path: `Optimization_Module/Design_Optimization/diagonal_brace_buckling_optimization`

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>Shell (shell)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Linear Buckling**.
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
Lz	6[m]	6 m	Length of brace
Lxy	5[cm]	0.05 m	Brace width
thickness	5[mm]	0.005 m	Brace thickness
Lz1	3*Lxy	0.15 m	Brace end length
Rhole	Lz1/10	0.015 m	Hole radius
Fload	1[kN]	1000 N	Load

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type Lxy.
4 In the **Depth** text field, type Lxy.
5 In the **Height** text field, type Lz.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Lz1`
`Layer 1`	`Lz-2*Lz1`

Boundaries to Delete

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Boundaries to Delete` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **x minimum** text field, type `Lxy*0.1`.

5 In the **y minimum** text field, type `Lxy*0.1`.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Selection** list, choose **Boundaries to Delete**.

Cylinder 1 (cyl1)

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Rhole`.

4 In the **Height** text field, type `Rhole`.

5 Locate the **Position** section. In the **x** text field, type `Lxy/2`.

6 In the **y** text field, type `-Rhole/2`.

7 In the **z** text field, type `Lz1/2`.

8 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Cylinder 2 (cyl2)

1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Position** section.

3 In the **z** text field, type `Lz-Lz1/2`.

Difference 1 (dif1)

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

2 Select the object **del1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the objects **cyl1** and **cyl2** only.

6 Click the **Show Grid** button in the **Graphics** toolbar.

The geometry should now look like that in Figure 1.

Internal Edges

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Internal Edges` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **z minimum** text field, type `Lz1*0.999`.

5 In the **z maximum** text field, type `Lz1*1.001`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Cylinder Selection 1 (cylsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

2 In the **Settings** window for **Cylinder Selection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Edge**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `Rhole*1.01`.

5 Locate the **Position** section. In the **x** text field, type `Lxy/2`.

6 In the **z** text field, type `Lz1/2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Cylinder Selection 2 (cylsel2)

1 Right-click **Cylinder Selection 1 (cylsel1)** and choose **Duplicate**.

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

3 In the **z** text field, type `Lz-Lz1/2`.

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.

SHELL (SHELL)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d text field, type `thickness`.

Rigid Connector 1

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

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

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

4 Locate the **Prescribed Displacement at Center of Rotation** section. Select the **Prescribed in x direction** check box.

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

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

7 Locate the **Prescribed Rotation** section. From the **By** list, choose **Constrained rotation**.

8 Select the **Constrain rotation around z-axis** check box.

9 Select the **Constrain rotation around x-axis** check box.

Rigid Connector 2

1 Right-click **Rigid Connector 1** and choose **Duplicate**.

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

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

4 Locate the **Prescribed Displacement at Center of Rotation** section. Clear the **Prescribed in z direction** check box.

Applied Force 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.

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

3 Specify the **F** vector as

0	x
0	y
-Fload	z

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 **Internal Edges**.

Distribution 1

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

Mapped 1

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

2 Select Boundaries 3 and 4 only.

Distribution 1

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

2 Select Edge 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 20.

Free Triangular 1

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

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

3 From the **Geometric entity level** list, choose **Remaining**.

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

DEFINITIONS

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **General>Variable Utilities** in the tree.

3 In the tree, select the check box for the node **General>Variable Utilities**.

4 Click **OK**.

Mass Properties 1 (mass1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variable Utilities>Mass Properties**.
2 In the **Settings** window for **Mass Properties**, locate the **Source Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **All boundaries**.
5 Locate the **Density** section. From the **Density source** list, choose **From physics interface**.

INITIAL DESIGN

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Initial Design` in the **Label** text field.
3 In the **Home** toolbar, click **Compute**.

RESULTS

Mass and Critical Load Factor

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Mass and Critical Load Factor` in the **Label** text field.

Global Evaluation 1

1 Right-click **Mass and Critical Load Factor** and choose **Global Evaluation**.
2 In the **Settings** window for **Global Evaluation**, click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Definitions>Mass Properties 1>mass1.mass - Mass - kg**.
3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Shell>shell.LFcrit - Critical load factor**.
4 In the **Mass and Critical Load Factor** toolbar, click **Evaluate**.

Mode Shape (shell), Shell Geometry (shell), Thickness and Orientation (shell)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mode Shape (shell)**, **Shell Geometry (shell)**, and **Thickness and Orientation (shell)**.
2 Right-click and choose **Group**.

Initial Design

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

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
`M0`	`23.5[kg]`	23.5 kg	`Initial Mass`

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Lz (Length of brace)	`4 6`	`m`

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, type `Optimization` in the **Label** text field.

Optimization

1 In the **Study** toolbar, click **Optimization**.

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

3 From the **Method** list, choose **COBYLA**.

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

Expression	Description	Evaluate for
abs(comp1.shell.LFcrit)		Linear Buckling

This means that both compressive and tensile failure modes are considered.

5 From the **Type** list, choose **Maximization**.

6 From the **Solution** list, choose **Minimum of objectives**.

7 Locate the **Control Variables and Parameters** section. Click **Add** twice.

8 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
Lxy (Brace width)	5[cm]	5[cm]	1[cm]	20[cm]
thickness (Brace thickness)	5[mm]	5[mm]	1[mm]	20[mm]

9 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Definitions>Mass Properties 1>comp1.mass1.mass - Mass - kg**.

10 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound	Evaluate for
comp1.mass1.mass/M0		Lz/(6[m])	Linear Buckling

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

RESULTS

Mode Shape (shell) 1, Shell Geometry (shell) 1, Thickness and Orientation (shell) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mode Shape (shell) 1**, **Shell Geometry (shell) 1**, and **Thickness and Orientation (shell) 1**.

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

Optimization

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

2 Right-click **Optimization** and choose **Plot**.

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

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Mass and Critical Load Factor** right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Optimization/Parametric Solutions 1 (sol5)**.

4 In the **Mass and Critical Load Factor** toolbar, click **Evaluate**.

The critical load factor has been improved with a factor of 2 for the 6 m brace.

Create a new plot for the model thumbnail.

Thumbnail

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Shell 1**.

Surface 1

Right-click **Thumbnail** and choose **Surface**.

Deformation 1

In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

Surface 2

1 In the **Model Builder** window, under **Results>Thumbnail** right-click **Surface 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Shell 2**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 Click the **Go to XY View** button in the **Graphics** toolbar.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Deformation 1**.

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

3 In the **x component** text field, type `u-4`.

4 Right-click **Deformation 1** and choose **Copy**.

Line 1

1 In the **Model Builder** window, right-click **Thumbnail** and choose **Line**.

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

3 From the **Dataset** list, choose **Shell 2**.

4 Locate the **Expression** section. In the **Expression** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

8 Clear the **Color** check box.

Deformation 1

1 In the **Model Builder** window, right-click **Line 1** and choose **Paste Deformation**.

2 In the **Thumbnail** toolbar, click **Plot**.

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