Created in COMSOL Multiphysics 5.6

Bracket — Spring Foundation Analysis

Introduction

A fixed, fully constrained, boundary condition contains the assumption that the analyzed structure is attached to an infinitely stiff support. While in many cases this is a useful approximation, sometimes you may need to consider the stiffness of the supporting structure in your model. In COMSOL Multiphysics you can do this by using the Spring Foundation boundary condition.

In this example, you study the stress in a bracket subjected to external loads. The stiffness of the mounted bolts connecting support is modeled with spring foundations.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This model is an extension of the model example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

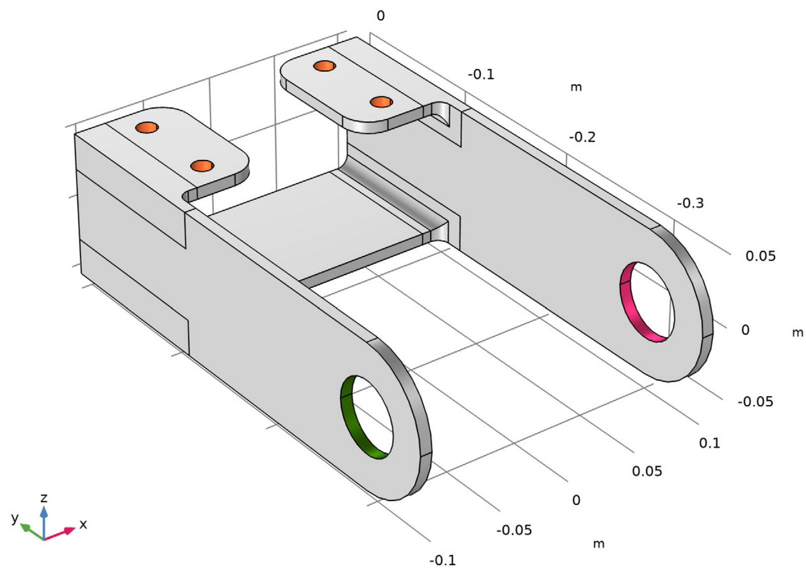

Figure 1: Bracket geometry.

The load is applied in the positive z-direction in the bracket left arm and in the negative z-direction in the bracket right arm, the same as in the original model.

The bolts are assumed to be elastic, defined with spring foundations.

Results and Discussion

Figure 2 shows the von Mises stress on a deformed geometry.

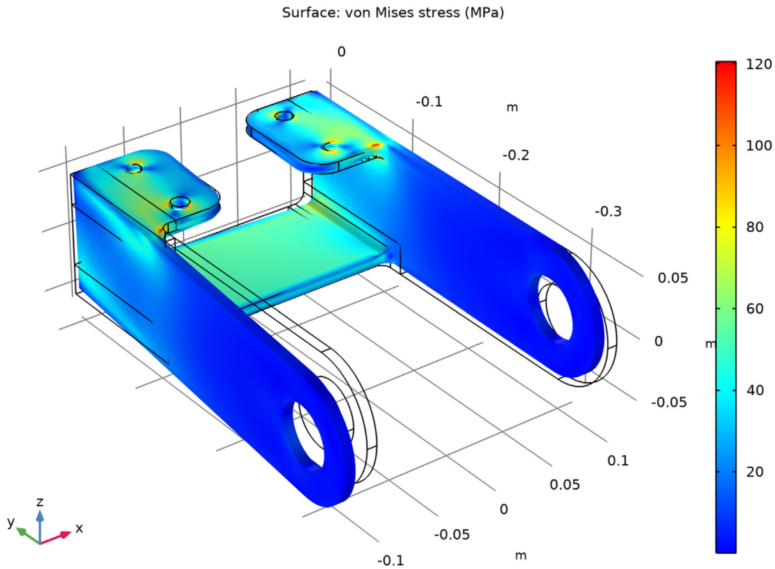

Figure 2: The Von Mises stress distribution.

The maximum stress in the bracket when connected using spring foundation is about 120 MPa. This is significantly lower than when using rigidly mounted bolts (see the Results section in the tutorial *Bracket - Static analysis*). There the maximum stress is about 190 MPa. The difference in the stress is caused by the flexibility that a spring connection provides.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_spring`

Modeling Instructions

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

Browse to the model's Application Libraries folder and double-click the file `bracket_basic.mph`.

COMPONENT 1 (COMP1)

Add the two new parameters for the spring coefficients of the external structure to the table.

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
P0	2.5[MPa]	2.5E6 Pa	Peak load intensity
YC	-300[mm]	-0.3 m	Y coordinate of hole center
kxy	1e6[N/m]	1E6 N/m	Spring coefficient in x and y direction
kz	1e8[N/m]	1E8 N/m	Spring coefficient in z direction

DEFINITIONS

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

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

6 In the **Function** text field, type `Pa`.

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

Apply a boundary load to the bracket holes.

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

3 From the **Selection** list, choose **Pin Holes**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
0	t2
load(-P0,Y-YC,Z)*(sign(X)*Z>0)	n

Fixed Constraint 1

In the **Model Builder** window, right-click **Fixed Constraint 1** and choose **Disable**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 In the **Settings** window for **Spring Foundation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Bolt Holes**.

4 Locate the **Spring** section. From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{tot}$ table, enter the following settings:

kxy	0	0
0	kxy	0
0	0	kz

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 **General Studies>Stationary**.

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

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

STUDY 1

Step 1: Stationary

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

RESULTS

Stress (solid)

The default plot shows the von Mises stress distribution, shown in Figure 2.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (solid)** toolbar, click **Plot**.