Model created in COMSOL Multiphysics 6.4

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. In many cases, this is a useful approximation, but sometimes you may need to consider the flexibility 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 connected 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 shown in Figure 1.

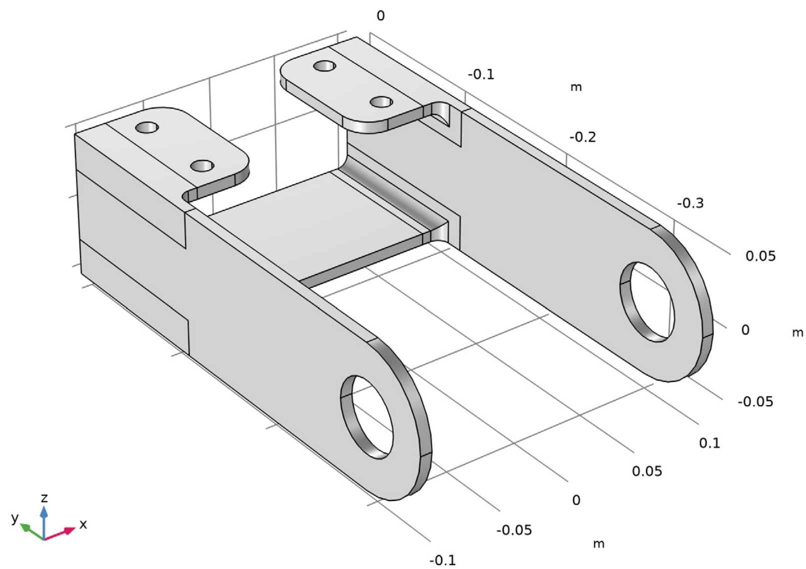

Figure 1: Bracket geometry.

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, as such, are ignored. The spring stiffness is distributed over the upper flat surfaces with the bolt holes. Large part of these surfaces will be pressed against the mounting plate by the bolted joints. The spring constants should then reasonably well approximate the stiffness of the mounting plate.

The total spring stiffness that the bracket experiences is assumed to be 1 MN/mm in the two in-plane directions, and 2 MN/mm in the vertical direction.

Results and Discussion

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

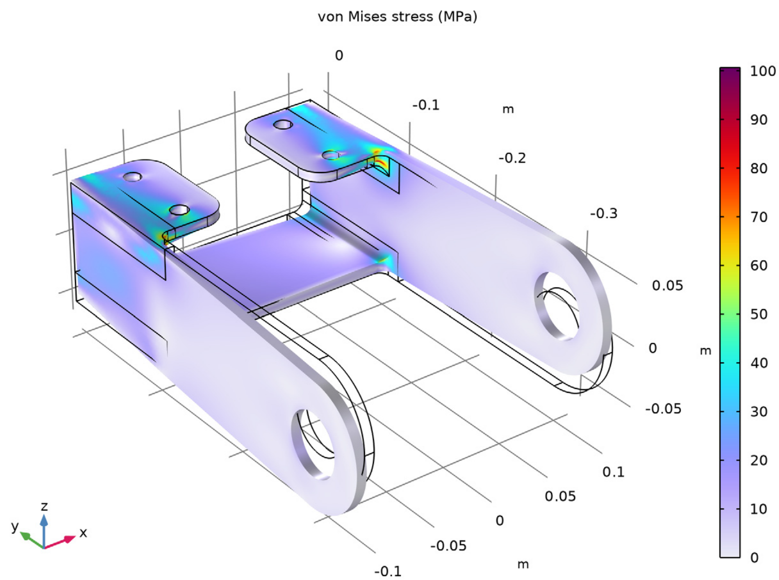

Figure 2: The Von Mises stress distribution.

The maximum stress in the bracket when connected using spring foundation is slightly above 100MPa, a significant increase from the situation where the bolt holes were fixed. There are also general differences in the stress distribution (see the Results section in the tutorial *Bracket - Static analysis*). The singular stress fields around the bolt holes are no longer present. This is an effect of the flexibility that a spring connection provides.

In order to supply proper values for the springs in a case like this, it is necessary that the stiffness of the supporting structure can be estimated. This can possibly be done in a separate finite element analysis.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_spring`

Modeling Instructions

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module** > **Tutorials** > **bracket_basic** in the tree.

3 Click **Open**.

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
kxy	1[MN/mm]	1E9 N/m	Spring coefficient in x and y directions
kz	2[MN/mm]	2E9 N/m	Spring coefficient in z direction

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

Apply a boundary load to the bracket holes. Since the entire circumference of each hole is selected, the expression for the pressure must be truncated so that it acts only on the intended 180 degrees

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

`if(abs(sys2.phi)<pi/2,-p0*cos(sys2.phi),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 Select Boundaries 37 and 50 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

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

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

6 Specify the $\mathbf{k}_{tot}$ matrix as

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 the **Add Study** button 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 **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

Volume 1

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

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