Created in COMSOL Multiphysics 6.2

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 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 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.

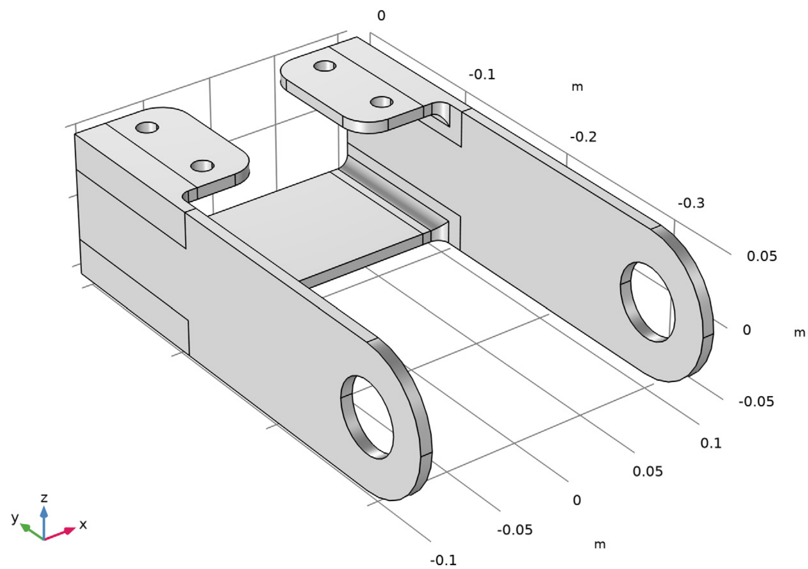

Figure 1: Bracket geometry.

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 surface which would be pressed against the mounting plate by the bolted joints.

The total spring stiffness that the bracket experiences is assumed to be 100 MN/m in the two in-plane directions, and 50 MN/m in the vertical direction.

Results and Discussion

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

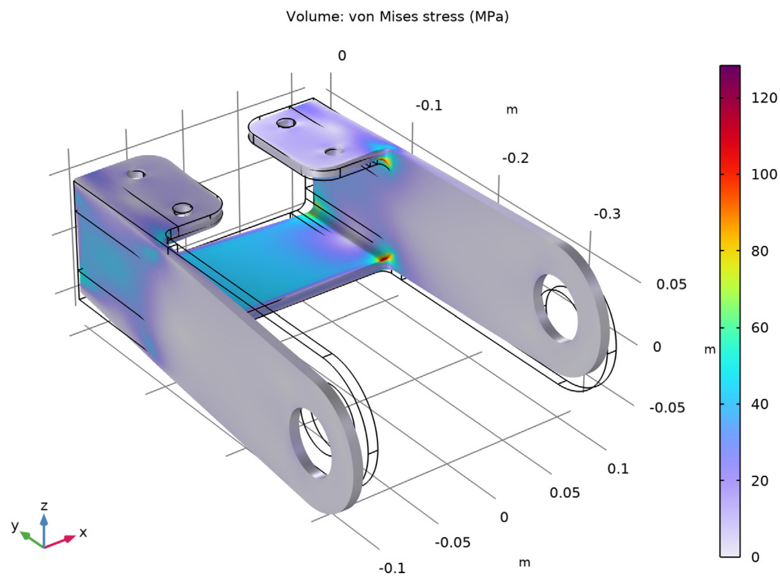

Figure 2: The Von Mises stress distribution.

The maximum stress in the bracket when connected using spring foundation is about 125 MPa. The stress distribution differs significantly from when the bolt holes are fixed. (see the Results section in the tutorial *Bracket - Static analysis*). The singular stress fields around the bolt holes are no longer present. A more important difference is that the stress at the fillets is much higher than before, and distributed in a different way. 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.

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
`P0`	`2.5[MPa]`	2.5E6 Pa	`Peak load intensity`
`kxy`	`1e8[N/m]`	1E8 N/m	`Spring coefficient in x and y direction`
`kz`	`5e7[N/m]`	5E7 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 table, enter the following settings:

Argument	Unit
F	Pa
py	m
px	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-PinHoleY,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 Select Boundaries 15, 37, 50, and 65 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 In the $\mathbf{k}_{\mathrm{tot}}$ 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.

Volume 1

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

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

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

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