Created in COMSOL Multiphysics 6.1

Bracket — Initial-Strain Analysis

Introduction

The various examples based on a bracket geometry form a suite of tutorials which summarizes the fundamentals when modeling structural mechanics problems in COMSOL Multiphysics and the Structural Mechanics Module.

In this example you learn how to introduce a prestrain to a structure and investigate how it affects the assembly.

It is recommended that you review the *Introduction to the Structural Mechanics Module*, which includes background information.

Model Definition

This tutorial is an extension of the example described in the section “The Fundamentals: A Static Linear Analysis” in the *Introduction to the Structural Mechanics Module*. The same model is also available as a standalone model in the Application Libraries as *Bracket - Static Analysis*.

In the previous example, the pin was only considered as providing a load, whereas in this example, the pin is actually modeled as shown in Figure 1.

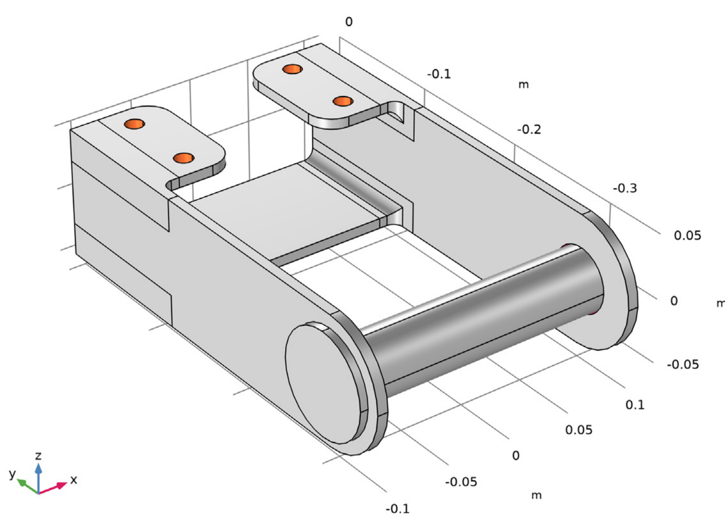

Figure 1: Bracket geometry.

An initial strain simulates that the pin is 1 mm too short in the axial direction. This could, for example, happen if there was a mismatch in dimensions due to manufacturing tolerances.

Results

Figure 2 shows how the pin compresses the bracket arms, and that the largest stresses are found in the region where the bracket arms are joined to the bolt supports.

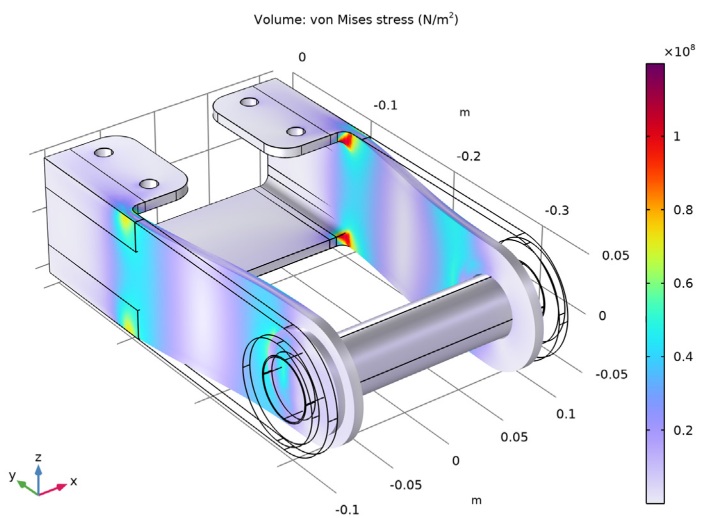

Figure 2: Von Mises stress distribution in the bracket. The deformation is exaggerated.

Figure 3 shows the third principal strain in order to visualize the total strain in the structure. As the pin is stiff when compared to the bracket, the total strain in the pin is almost the same as the initial strain given in the example.

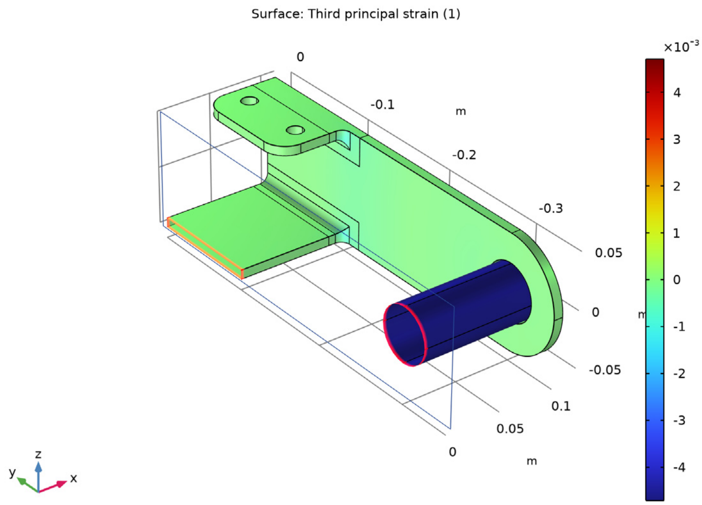

Figure 3: Strain distribution in the bracket.

Notes About the COMSOL Implementation

Initial stresses and strains can be specified in the **Initial Stress and Strain** subnode to a material model. Think of the strain or stress that you introduce as an inelastic contribution, which is not necessarily constant over the simulation. You can define a stress/strain distribution with constant values or as an expression which can, for example, be space or time dependent. The initial stresses and strains can also be results from another study, or even from another physics interface in the same study. The **External Stress** and **External Strain** subnodes can be used for similar purposes, but provide more options.

The structure is modeled as an assembly, so that the pin and bracket are considered as different objects. They are then connected using a **Continuity** condition. This means that the bracket and the pin can be meshed independently, so that the original mesh on the bracket can be kept unaltered.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_initial_strain`

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**.

GLOBAL DEFINITIONS

Parameters 1

In the Parameters table, define a strain value that corresponds to a reduction of the pin length from 215 mm to 214 mm.

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
`L0`	`215[mm]`	0.215 m	`Initial pin length`
`L`	`214[mm]`	0.214 m	`Current pin length`
`InitStrain`	`(L-L0)/L0`	-0.0046512	`Pin strain`

GEOMETRY 1

Add the pin geometry to the bracket assembly by importing it into the existing geometry.

Import 2 (imp2)

1 In the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 From the **Source** list, choose **COMSOL Multiphysics file**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `bracket_pin.mphbin`.

6 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Click **Build Selected**.

SOLID MECHANICS (SOLID)

Adding Initial Stress and Strain

Specify the initial strain under the **Linear Elastic Material** node.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Select Domain 2 only.

The prestrain direction is the axial direction of the bolt, which coincides with the global *X* direction.

5 Locate the **Initial Stress and Strain** section. In the ε_0 table, enter the following settings:

InitStrain	0	0
0	0	0
0	0	0

MESH 1

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

Size 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1> Free Tetrahedral 1** node, then click **Size 1**.

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

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

4 Select Domains 5, 6, 10, and 11 only.

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

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

RESULTS

Stress (solid)

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

Third Principal Strain

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

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

Surface 1

1 Right-click **Third Principal Strain** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>Principal strains>solid.ep3 - Third principal strain**.

3 In the **Third Principal Strain** toolbar, click **Plot**.

4 Locate the **Coloring and Style** section. From the **Scale** list, choose **Linear symmetric**.

5 In the **Third Principal Strain** toolbar, click **Plot**.

6 Click the **Clipping** button in the **Graphics** toolbar.

7 In the **Graphics** window toolbar, click next to **Clipping**, then choose **Add Clip Plane**.

8 In the **Graphics** window toolbar, click next to **Clipping Active**, then choose **Show Gizmos**.

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