Created in COMSOL Multiphysics 6.2

Bracket — Inertia Relief 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 perform an inertia relief analysis. This type of analysis is intended for situations, where an unconstrained structure is accelerated by external forces. The applied loads and the inertial forces caused by the acceleration will then be in balance. This type of analysis is common in, for example, aerospace engineering.

From an engineering point of view, analyzing the behavior of an unconstrained bracket can be of little interest. Thus, the purpose of this example is just to demonstrate how to set up an inertia relief analysis.

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

Model Definition

The model used in this guide is a bracket made of steel. The geometry is shown in Figure 1.

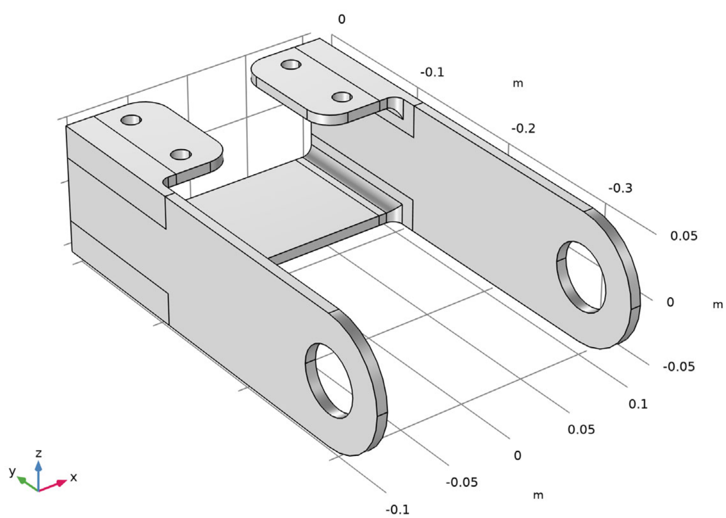

Figure 1: Bracket geometry.

In this analysis, the bracket is unconstrained. One of the arms is loaded upward and the other downward. The loads are applied as total forces of 65 N, acting in the opposite directions on the inner surfaces of the holes.

Results

Figure 2 shows the von Mises stress distribution together with an exaggerated picture of the deformation.

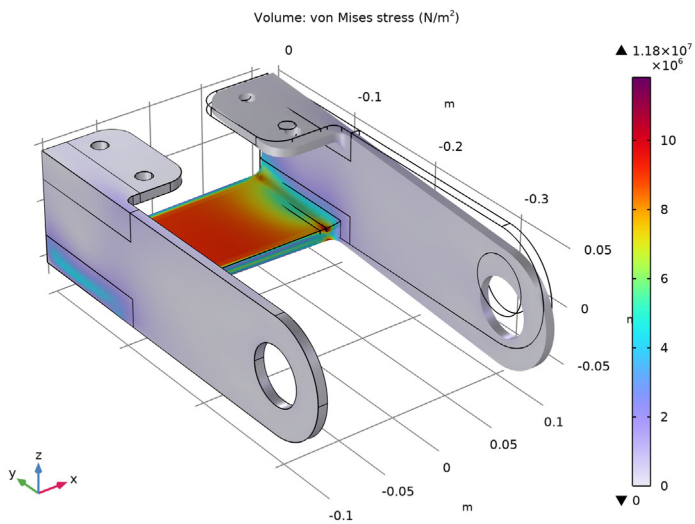

Figure 2: Von Mises stress distribution in the bracket.

It should, however, be noted that the displacement field computed in an inertia relief analysis is not unique. It is relative to an arbitrary rigid body motion.

Figure 3 shows the acceleration. Since there is no resultant force, the motion will mainly consist of a rotational acceleration around the y-axis.

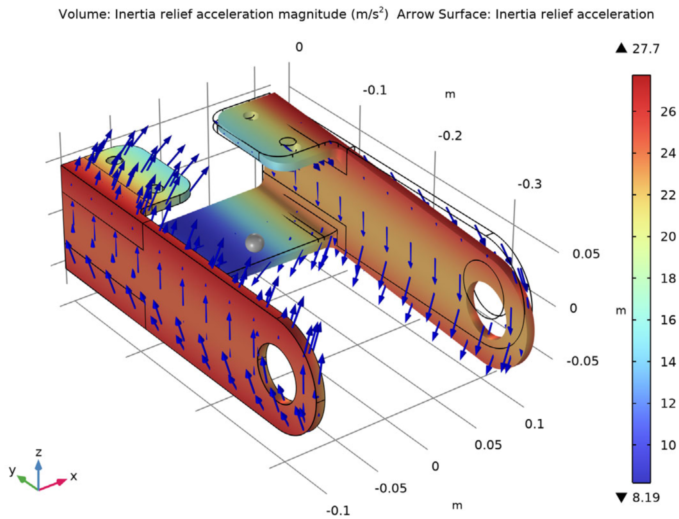

Figure 3: Inertia relief acceleration. The gray sphere in the middle is not part of the structure, it just indicates the location of the center of mass of the bracket.

Notes About the COMSOL Implementation

To perform an inertial relief analysis, you add an **Inertia Relief** node to the physics interface. In most cases, you do not need to change any settings. You should not use any other types of constraints or elastic supports. The structure must be able to move freely as rigid body.

Then, you click the **Create Load Groups and Study** button. A special set of load groups will be added, together with a new study. This study consists of two stationery study steps. In the first step, a number of load cases (six in 3D) are solved. Using these load cases, a rigid body acceleration is determined, using the conditions that the external loads are balanced by inertia forces. In the second study step, the external load and the computed inertia forces are all combined together to form the total solution.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_inertia_relief`

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

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
Ftot	65[N]	65 N	Boundary load

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Right-click **Component 1 (comp1)>Solid Mechanics (solid)>Fixed Constraint 1** and choose **Delete**.

Boundary Load 1

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

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

3 Specify the $\mathbf{F}_A$ vector as

F	z

4 Select Boundary 5 only.

5 From the **Load type** list, choose **Total force**.

6 Specify the $\mathbf{F}_{tot}$ vector as

Ftot	z

7 Right-click **Boundary Load 1** and choose **Duplicate**.

Boundary Load 2

1 In the **Model Builder** window, click **Boundary Load 2**.

2 Select Boundary 75 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_{tot}$ vector as

-Ftot	z

Inertia Relief 1

1 In the **Physics** toolbar, click **Domains** and choose **Inertia Relief**.

2 In the **Settings** window for **Inertia Relief**, click **Automated Model Setup** in the upper-right corner of the **Inertia Relief** section. From the menu, choose **Create Load Groups and Study**.

DEFINITIONS

Mass Properties 1 (mass1)

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

2 Right-click **Definitions** and choose **Physics Utilities>Mass Properties**.

INERTIA RELIEF STUDY

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

RESULTS

Stress (solid)

1 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

2 Select the **Show maximum and minimum values** check box.

3 Right-click **Results>Stress (solid)** and choose **Duplicate**.

Inertia relief acceleration (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Inertia relief acceleration (solid)` in the **Label** text field.

Volume 1

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

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Acceleration and velocity>solid.air - Inertia relief acceleration magnitude - m/s²**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

5 Click **OK**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Inertia relief acceleration (solid)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Acceleration and velocity>solid.airX,...,solid.airZ - Inertia relief acceleration**.

3 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

4 From the **Color** list, choose **Blue**.

5 From the **Arrow length** list, choose **Normalized**.

6 Select the **Scale factor** check box. In the associated text field, type `0.001`.

Inertia relief acceleration (solid)

In the **Model Builder** window, click **Inertia relief acceleration (solid)**.

Point Trajectories 1

1 In the **Inertia relief acceleration (solid)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Type** list, choose **None**.

4 Find the **Point style** subsection. From the **Type** list, choose **Point**.

5 Click **Replace Expression** in the upper-right corner of the **Trajectory Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Mass Properties 1>mass1.CMX,..., mass1.CMZ - Center of mass**.

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

7 Locate the **Coloring and Style** section. In the **Point radius expression** text field, type `0.008`.

8 Select the **Radius scale factor** check box. In the associated text field, type `1`.

9 From the **Color** list, choose **Gray**.

10 In the **Inertia relief acceleration (solid)** toolbar, click **Plot**.

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