Model created in COMSOL Multiphysics 6.4

Bracket — Inertia Relief Analysis

Introduction

The various examples based on a bracket geometry form a suite of tutorials summarizing the fundamentals of 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 relevant background information.

	In the *Structural Mechanics Modeling* chapter of the *Structural Mechanics Module User's Guide*: *Inertia Relief Study*.

Model Definition

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

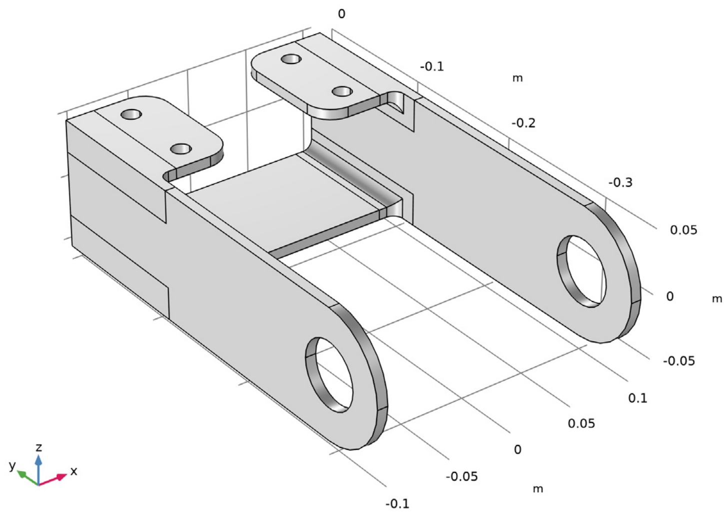

Figure 1: Bracket geometry.

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 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.

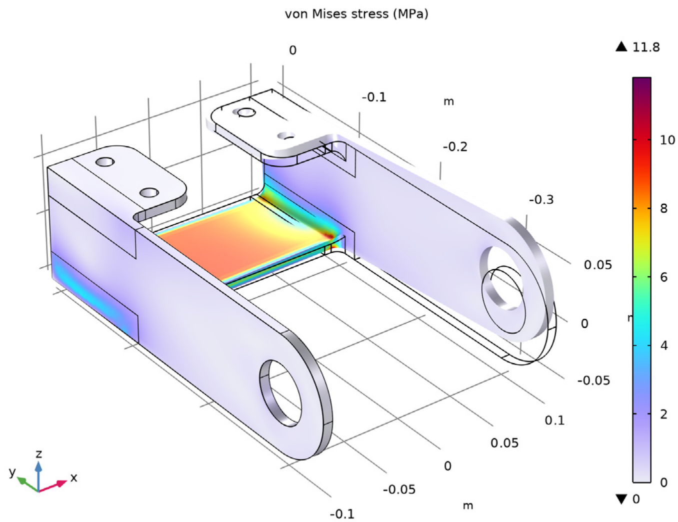

Figure 2: von Mises stress distribution in the bracket.

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 mainly consists of a rotational acceleration around the y-axis.

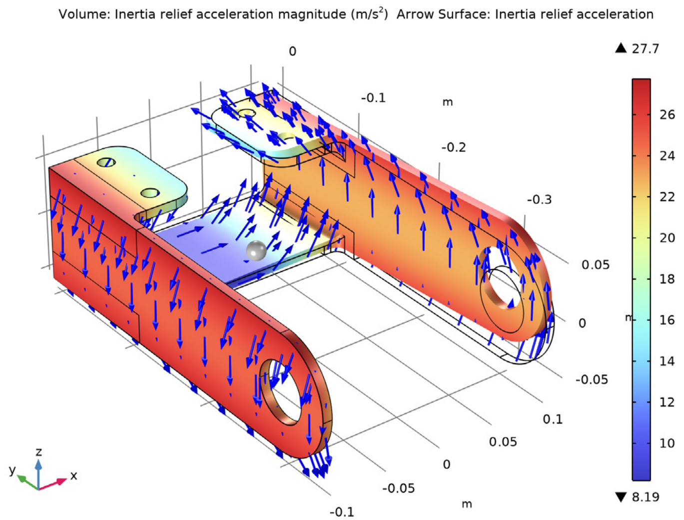

Figure 3: Inertia relief acceleration. The gray sphere in the middle is not part of the structure; it just indicates the location of the center of mass of the bracket.

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 stationary 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`

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
Fh	65[N]	65 N	Force per hole

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 **Physics** toolbar, click **Boundaries** 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

`if(abs(sys2.phi)<pi/2,-p0*cos(sys2.phi),0)`	n

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.

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**.

3 In the **Settings** window for **Mass Properties**, locate the **Density** section.

4 From the **Density source** list, choose **From physics interface**.

INERTIA RELIEF STUDY

In the **Study** 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** checkbox.

Inertia relief acceleration (solid)

1 Right-click **Stress (solid)** and choose **Duplicate**.

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/s^2**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

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** checkbox. 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** checkbox.

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.