Created in COMSOL Multiphysics 6.2

Bracket — Parametric 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.

This example includes computing the solution to a case where the direction of the load is changed using a parametric sweep over a set of directions.

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

Model Definition

This model 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*.

The geometry is shown in Figure 1.

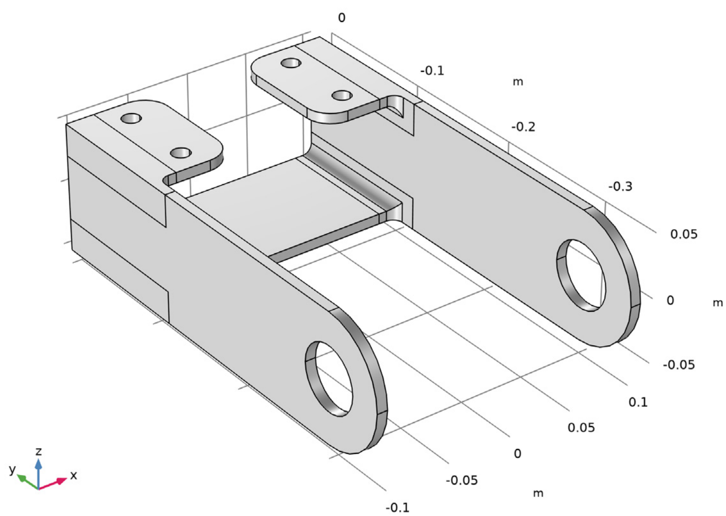

Figure 1: Bracket geometry.

In this analysis, the mounting bolts are assumed to be fixed and securely bonded to the bracket. To model the external load from the pin, you specify a surface load p with a trigonometric distribution on the inner surfaces of the two holes:

$$p = P_0 \cos(\alpha - \theta_0) \qquad -\frac{\pi}{2} < \alpha - \theta_0 < \frac{\pi}{2}$$

where P_0 is the peak load intensity. The main direction of the load is defined by θ_0, the angle from the y-axis. The load on the two holes acts in opposite directions. The orientation of the load is controlled by a local coordinate system with axis directions generated using the sweep parameter `theta0`.

Results

Figure 2 shows the von Mises stress distribution corresponding to a twisting load case, where the load acts in the positive z direction in the left arm and in the negative z direction in the right arm.

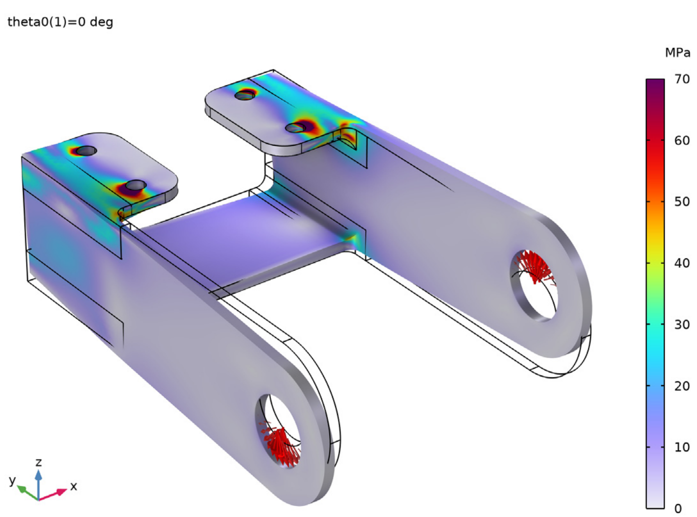

Figure 2: Von Mises stress in a twisting load case (parameter theta0 = 0°).

Figure 3 shows the von Mises stress distribution corresponding of a tensile load in the right arm and a compressive load in the left arm. The maximum von Mises stress value is significantly lower in this case..

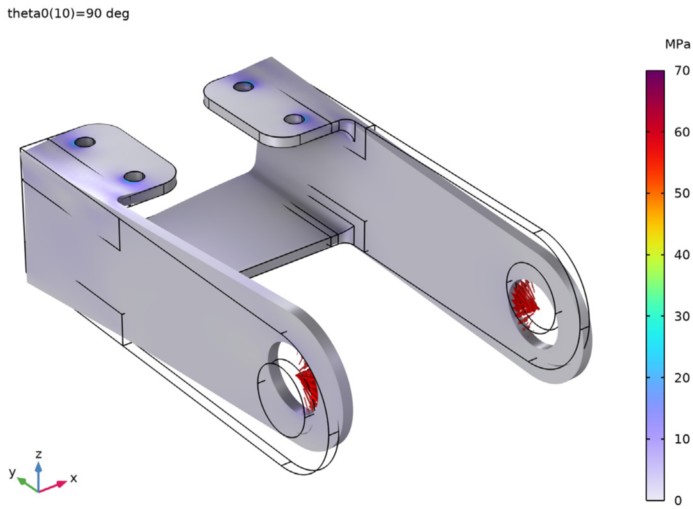

Figure 3: Von Mises stress in a tensile and compressive load case (parameter theta0 = 90°).

[Figure 4](#) shows the von Mises stress distribution corresponding to a load orientation of 130°.

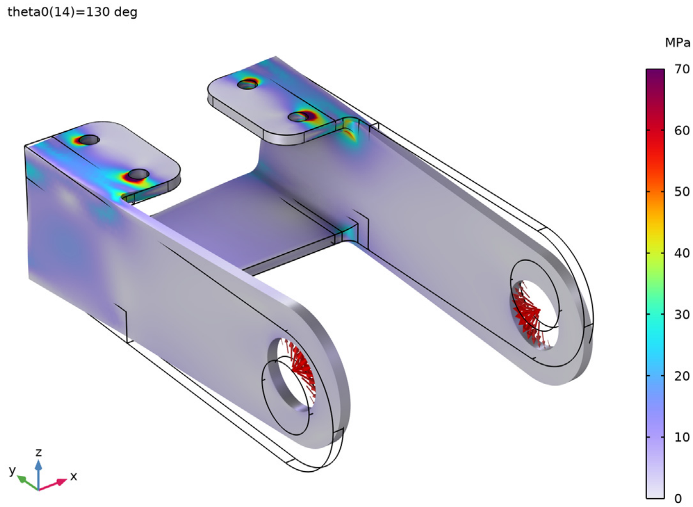

Figure 4: Von Mises stress for parameter theta0 = 130°.

[Figure 5](#) shows that the x-component of the reaction force in all bolts for all load cases.

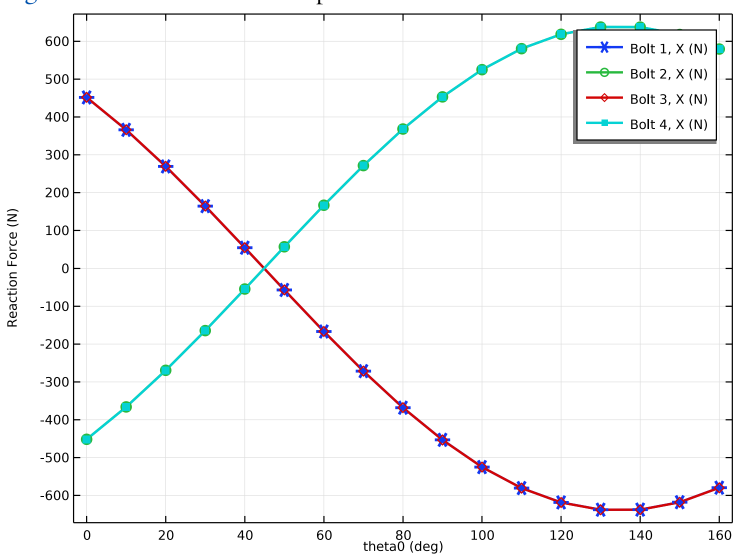

Figure 5: Reaction forces (x-component) as a function of angle.

Figure 6 shows the y-component and Figure 7 shows the *z*-component of the reaction force in all bolts for all load cases.

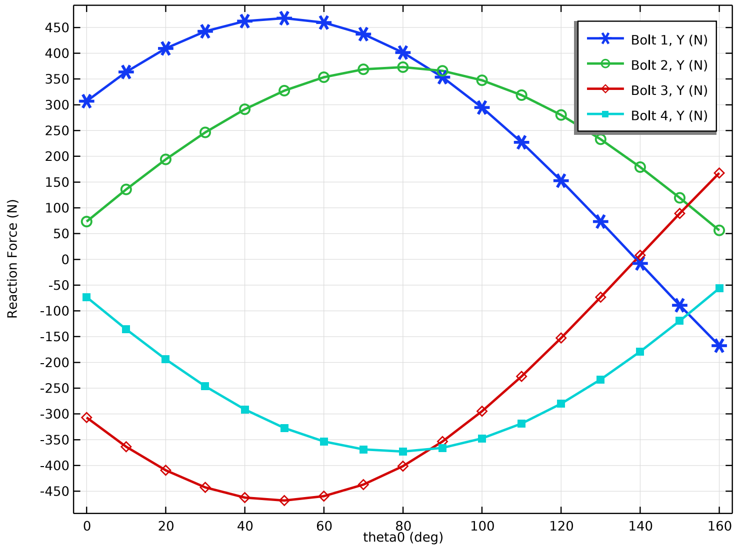

Figure 6: Reaction forces (y-component) as a function of angle.

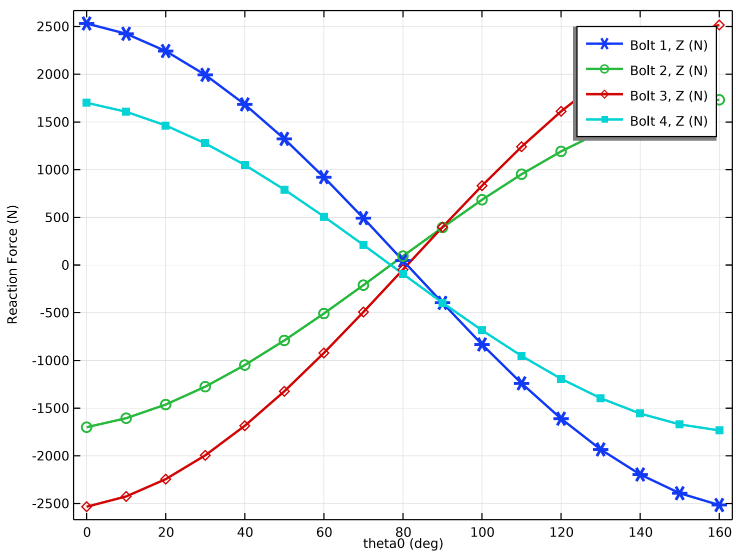

Figure 7: Reaction forces (z-component) as a function of load angle.

Finally, the variation of the stress with the load angle in some points is studied in Figure 8. It is clear that the load angles that induce a bending/twisting state cause higher stresses than those which give predominantly tension and compression.

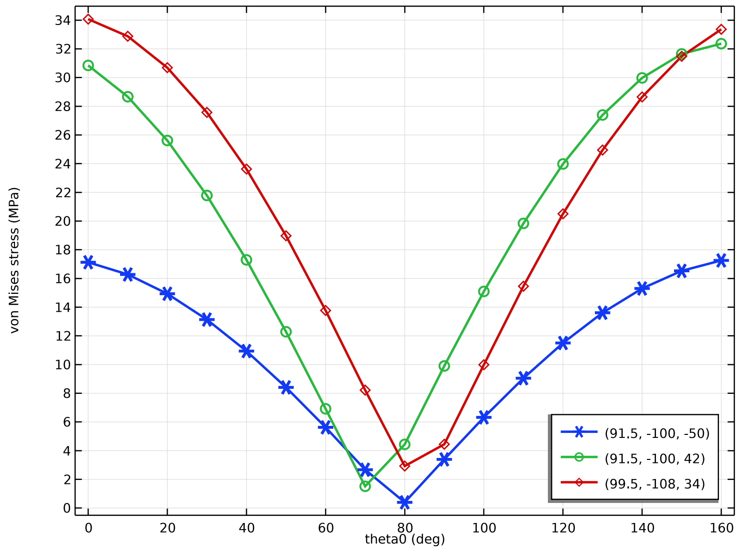

Figure 8: von Mises stress as function of the load angle for three different high stress locations.

Notes About the COMSOL Implementation

In COMSOL Multiphysics, you have two ways to perform parametric studies — using either a **Parametric Sweep** node or the **Auxiliary sweep** from the **Stationary Solver** node. In this example, either method can be used. An **Auxiliary sweep** is used here, but the continuation solver is not used. The continuation solver uses the solution from the previous parameter as an initial guess to calculate the current parameter value, which is the preferred option for nonlinear problems. Using the **Parametric Sweep** node is necessary for applications requiring, for example, geometric parameterization.

It can be noted that the stiffness matrix in this case is only inverted once. The solver automatically recognizes the fact that it is only the loads, and thus the right hand side of the system of equations, that are changing between the parameter steps.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_parametric`

Parametric studies can be set up from scratch or, as in this example, added to an existing study.

APPLICATION LIBRARIES

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

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

3 Click **Open**.

RESULTS

Stress (solid)

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

GLOBAL DEFINITIONS

In this model, the stress in the bracket is computed for different load orientations. First add a parameter to set the load direction angle.

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
theta0	0[deg]	0 rad	Load direction angle

DEFINITIONS

You will now create a local coordinate system that will rotate with the load orientation.

Cylindrical System 2 (sys2)

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

2 Right-click **Component 1 (comp1)>Definitions** and choose **Coordinate Systems>Cylindrical System**.

3 In the **Settings** window for **Cylindrical System**, locate the **Settings** section.

4 Find the **Origin** subsection. In the table, enter the following settings:

x (m)	**y (m)**	**z (m)**
0	PinHoleY	0

5 Find the **Longitudinal axis** subsection. In the table, enter the following settings:

x	**y**	**z**
1	0	0

6 Find the **Direction of axis ϕ=0** subsection. In the table, enter the following settings:

x	**y**	**z**
0	sin(theta0)	cos(theta0)

Analytic 1 (load)

Change the expression for the load distribution.

1 In the **Model Builder** window, click **Analytic 1 (load)**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `F*cos(p)`.

4 In the **Arguments** text field, type `F, p`.

5 Locate the **Units** section. In the table, enter the following settings:

Argument	**Unit**
F	Pa
p	rad

SOLID MECHANICS (SOLID)

Change the boundary loads to consider the parameterized direction.

Boundary Load 1

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

2 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`load(P0*sign(X),sys2.phi)*(sign(X)*(abs(sys2.phi)-pi/2)<0)`	r
0	phi
0	a

STUDY 1

Add an auxiliary sweep parameter, and compute the results.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Load direction angle)	`range(0,10,160)`	`deg`

6 From the **Run continuation for** list, choose **No parameter**.

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

RESULTS

Stress (solid)

The default plot shows the solution for the last parameter value (`160[deg]`). You can easily change the parameter value to display the plot and then compare solutions for different load cases.

The following instructions reproduce Figure 2 to Figure 4.

1 In the **Settings** window for **3D Plot Group**, click **Plot First**.

2 Locate the **Data** section. From the **Parameter value (theta0 (deg))** list, choose **90**.

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

4 From the **Parameter value (theta0 (deg))** list, choose **130**.

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

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

Evaluation Group: Reactions

You will now create a plot showing how the reaction forces vary with the load angle.

1 In the **Model Builder** window, click **Evaluation Group: Reactions**.

2 In the **Evaluation Group: Reactions** toolbar, click **Evaluate**.

EVALUATION GROUP: REACTIONS

1 Go to the **Evaluation Group: Reactions** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Reaction Force, X-component

1 In the **Model Builder** window, under **Results** click **1D Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Reaction Force, X-component` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Reaction Force (N)`.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Plot columns** list, choose **Manual**.

4 In the **Columns** list, choose **Bolt 1, X (N)**, **Bolt 2, X (N)**, **Bolt 3, X (N)**, and **Bolt 4, X (N)**.

5 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 In the **Reaction Force, X-component** toolbar, click **Plot** to show Figure 5.

Reaction Force, X-component

In the **Model Builder** window, right-click **Reaction Force, X-component** and choose **Duplicate**.

Reaction Force, Y-component

1 In the **Model Builder** window, expand the **Results>Reaction Force, X-component 1** node, then click **Reaction Force, X-component 1**.

2 In the **Settings** window for **1D Plot Group**, type `Reaction Force, Y-component` in the **Label** text field.

Table Graph 1

1. In the **Model Builder** window, click **Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. In the **Columns** list, choose **Bolt 1, Y (N)**, **Bolt 2, Y (N)**, **Bolt 3, Y (N)**, and **Bolt 4, Y (N)**.
4. In the **Reaction Force, Y-component** toolbar, click **Plot** to show Figure 6.

Reaction Force, Y-component

In the **Model Builder** window, right-click **Reaction Force, Y-component** and choose **Duplicate**.

Reaction Force, Z-component

1. In the **Model Builder** window, expand the **Results>Reaction Force, Y-component 1** node, then click **Reaction Force, Y-component 1**.
2. In the **Settings** window for **1D Plot Group**, type `Reaction Force, Z-component` in the **Label** text field.

Table Graph 1

1. In the **Model Builder** window, click **Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. In the **Columns** list, choose **Bolt 1, Z (N)**, **Bolt 2, Z (N)**, **Bolt 3, Z (N)**, and **Bolt 4, Z (N)**.
4. In the **Reaction Force, Z-component** toolbar, click **Plot** to show Figure 7.

   Create a graph of how the stress varies with the load angle in some interesting points.

Stress as Function of Load Angle

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Stress as Function of Load Angle` in the **Label** text field.

Point Graph 1

1. Right-click **Stress as Function of Load Angle** and choose **Point Graph**.
2. Select Points 86, 88, and 103 only.
3. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4. In the **Expression** text field, type `solid.mises`.
5. From the **Unit** list, choose **MPa**.
6. Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

   Annotate the graphs by the location of each point.

7. Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Evaluated**.

9 In the **Legend** text field, type `(eval(X,mm), eval(Y,mm), eval(Z,mm))`.

10 In the **Stress as Function of Load Angle** toolbar, click **Plot**.

Stress as Function of Load Angle

1 In the **Model Builder** window, click **Stress as Function of Load Angle**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

5 In the **Stress as Function of Load Angle** toolbar, click **Plot**.