Created in COMSOL Multiphysics 5.3

Stresses and Strains in a Wrench

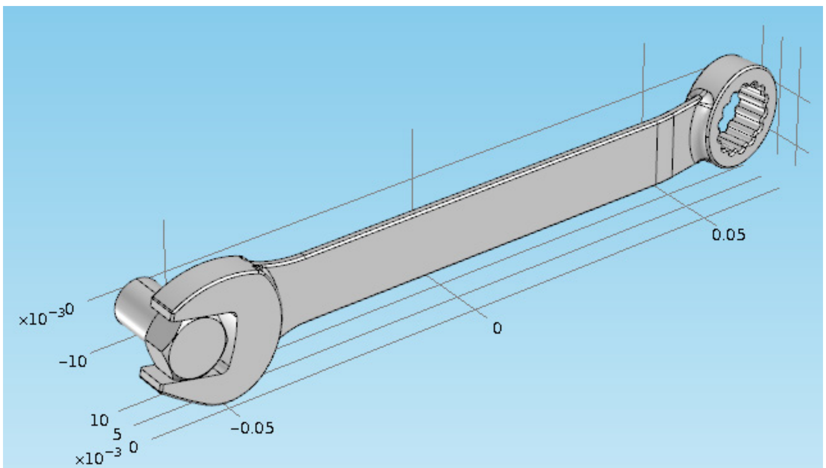
Introduction

This tutorial demonstrates how to set up a simple static structural analysis. The analysis is exemplified on a combination wrench during the application of torque on a bolt.

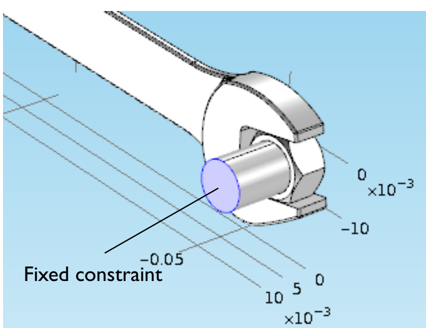
Despite its simplicity, and the fact that very few engineers would run a structural analysis before trying to turn a bolt, the example provides an excellent overview of structural analysis in COMSOL Multiphysics.

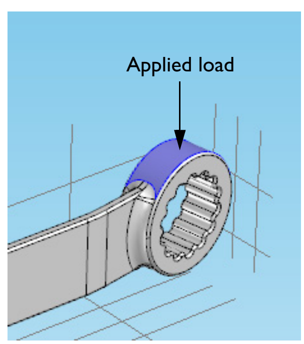
Model Definition

The model geometry is shown below.



The bolt's fixed constraint is at the cross section shown below. A load is applied at the box end of the combination wrench.





Here, assume that there is perfect contact between the wrench and the bolt. A possible extension is to apply a contact condition between the wrench and the bolt where the friction and the contact pressure determines the position of the contact surface.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/wrench`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

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

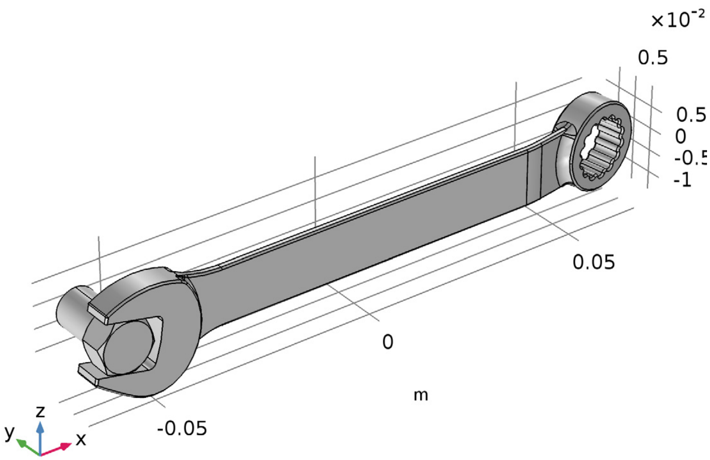
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

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

5 Click **Build All Objects**.

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



ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-In>Structural steel**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
F	150[N]	150 N	Applied force

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select Boundary 35 only.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 111 only.

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

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

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

0	x
0	y
-F	z

The minus sign means that the force is applied downwards.

MESH 1

Use finer mesh because the geometry contains small edges and faces.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (solid)** and choose **Enable**.

5 In the **Settings** window for **Iterative**, locate the **General** section.

6 From the **Preconditioning** list, choose **Right**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot group shows the von Mises stress in a **Surface** plot with the displacement visualized using a **Deformation** subnode. Change to a more suitable unit as follows.

Surface 1

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

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

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

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

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Surface 1

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

2 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Surface**.

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Solid Mechanics>Strain>Principal strains>solid.ep1 - First principal strain**.

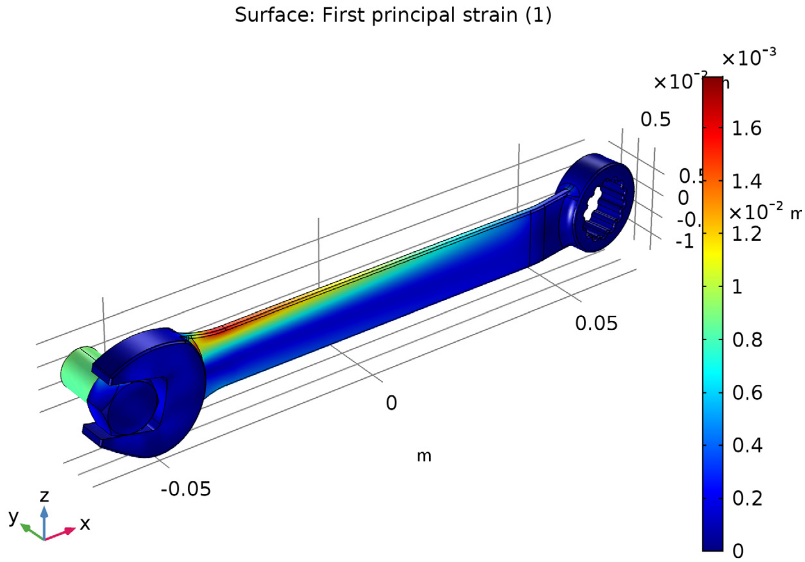
4 On the **3D Plot Group 2** toolbar, click **Plot**.

3D Plot Group 2

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 2** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `First Principal Strain` in the **New label** text field.

3 Click **OK**.



Notice that the maximum principal strain is lower than 2%, a result that satisfies the small strain assumption.