

# 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.



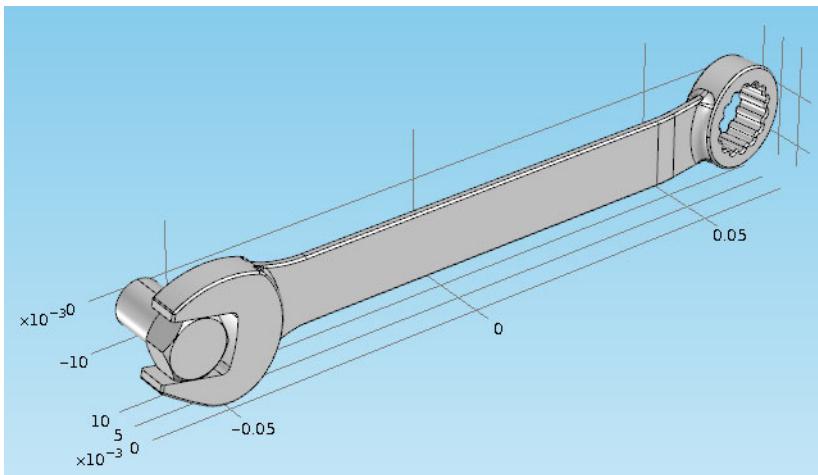
Another, more extensive variant of this tutorial is available in the *Introduction to COMSOL Multiphysics* document.

---

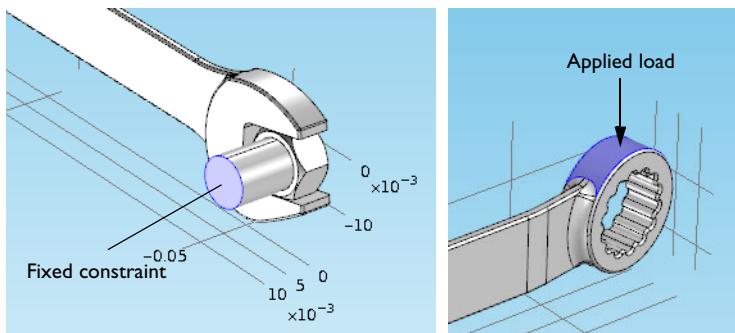
## *Model Definition*

---

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**.

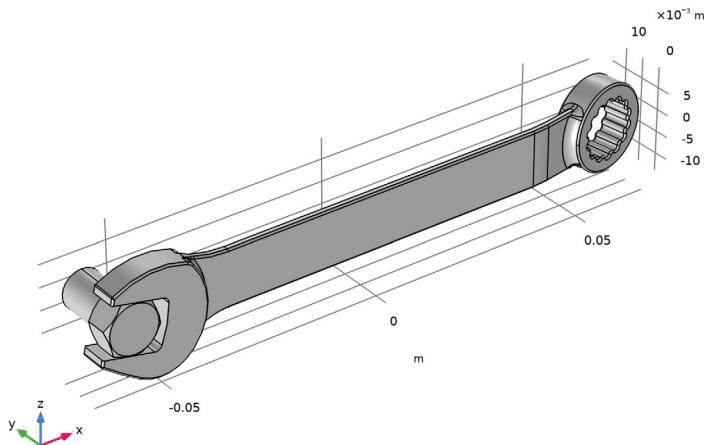
#### **MODEL WIZARD**

- 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 **General Studies>Stationary**.
- 6 Click **Done**.

## GEOMETRY I

*Import 1 (imp1)*

- 1 In 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 in the **Graphics** toolbar.



## ADD MATERIAL

- 1 In 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 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

## 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
F	150[N]	150 N	Applied force

## SOLID MECHANICS (SOLID)

### Fixed Constraint 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.
- 2 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- 3 Select Boundary 35 only.

### Boundary Load 1

- 1 In 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  $\mathbf{F}_{\text{tot}}$  vector as

0	x
0	y
-F	z

The minus sign means that the force is applied downward.

## 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 **Physics-Controlled Mesh** section.
- 3 From the **Element size** list, choose **Finer**.
- 4 Click  **Build All**.

## STUDY 1

If your computer has more than 4 GB of RAM, you can skip the following four instructions and continue directly to the section [Compute the Solution](#). Otherwise, follow these steps to use an iterative solver:

### *Solution 1 (sol1)*

- 1 In 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 **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Suggested Iterative Solver (solid)** and choose **Enable**.

Iterative solvers require less memory but can be less robust than direct solvers.

## COMPUTE THE SOLUTION

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

## RESULTS

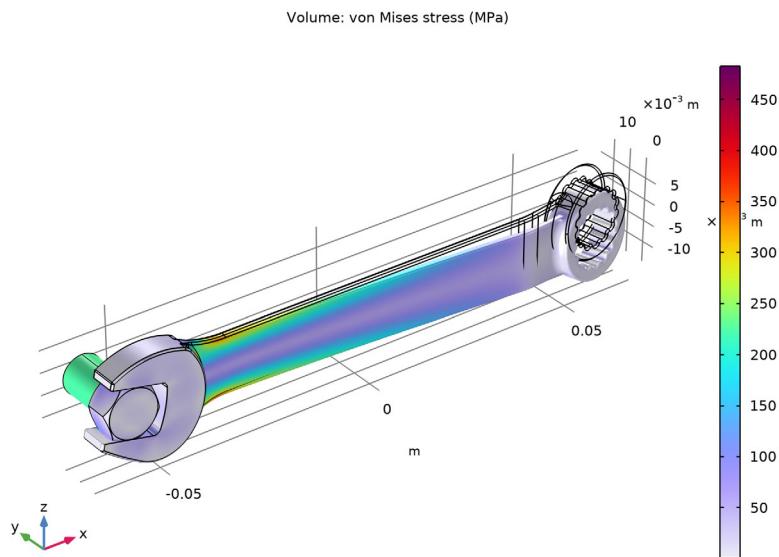
### *Stress (solid)*

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.

### *Volume 1*

- 1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 In the **Stress (solid)** toolbar, click  **Plot**.

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



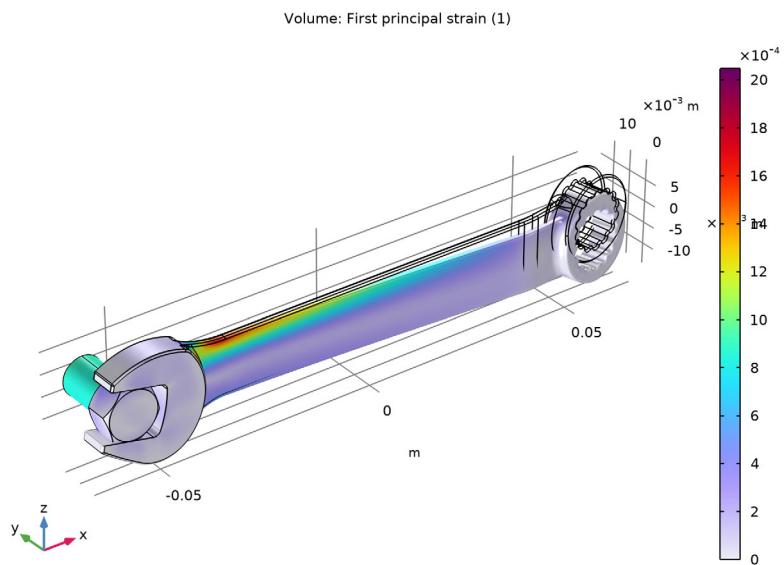
#### *First Principal Strain*

- 1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type **First Principal Strain** in the **Label** text field.

#### *Volume I*

- 1 In the **Model Builder** window, expand the **First Principal Strain** node, then click **Volume I**.
- 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>Strain>Principal strains>solid.ep1 - First principal strain**.

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



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