COMSOL tutorial 1: Maximal stress in a wrench

Follow the tutorial "Example 1: Structural Analysis of a Wrench", except "Convergence Analysis", described in the following pdf document:

IntroductionToCOMSOLMultiphysics.pdf

This pdf file should be in the following folder:

COMSOL (Installation directory)/Multiphysics/doc/pdf/COMSOL_Multiphysics/

This document can also be found in the help menu within Comsol application.

When you have finished the tutorial, decrease the material stiffness by half and evaluate the effect on the (von Mises) stress and the (first principal) strain.

Note. If you get a memory error, reduce the mesh size.