

# Structural Analysis of Fiberglass Handle of Fast Grip Wrench

Author: Lilit Mazmanyan

Akian College of Science and Engineering & United Testing Laboratory

Software: Pro/ENGINEER, Pro/MECHANICA Year:2005



## Introduction

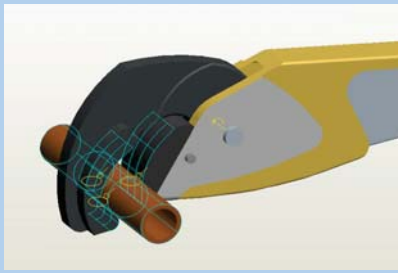


The companies that have the capacity to incorporate simulation into their design process can streamline their workflow. This is especially true when the company is engaged in product optimization tasks. CAE simulation tools can significantly reduce the number of iterations while testing the prototypes. They can help the companies to reduce the product cost and production time. Presented is a "Design for Industry" project. Its goal is to improve the design of the handle of the fast grip wrench so that it can bear the test load of 7000 N. The real-life tests are done in the United Testing Laboratory (picture on the left). In order to implement the computer simulation, we created the 3D model of the wrench. Its exploded view is shown at right.

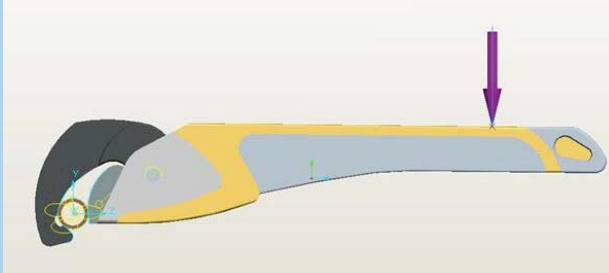


## Mechanism Design and Static Analysis

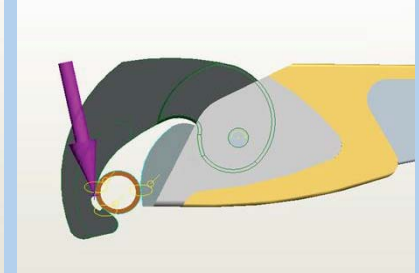
The Mechanism Design module enabled us to calculate forces occurring in the model under applied loads. In order to retrieve this information we replaced the real connections in the model with Pro/ENGINEER idealization features: cam-follower connections created for the contact regions between the wrench jaws and the pipe and the pin connection - for the joint between the upper jaw and the handle (see the picture on the left). In addition, we applied the test load on the handle. After the test conditions were specified via the mentioned idealizations, we ran a force balance analysis to find the reaction force on the jaw of the wrench.



Cam Follower and Pin Connections



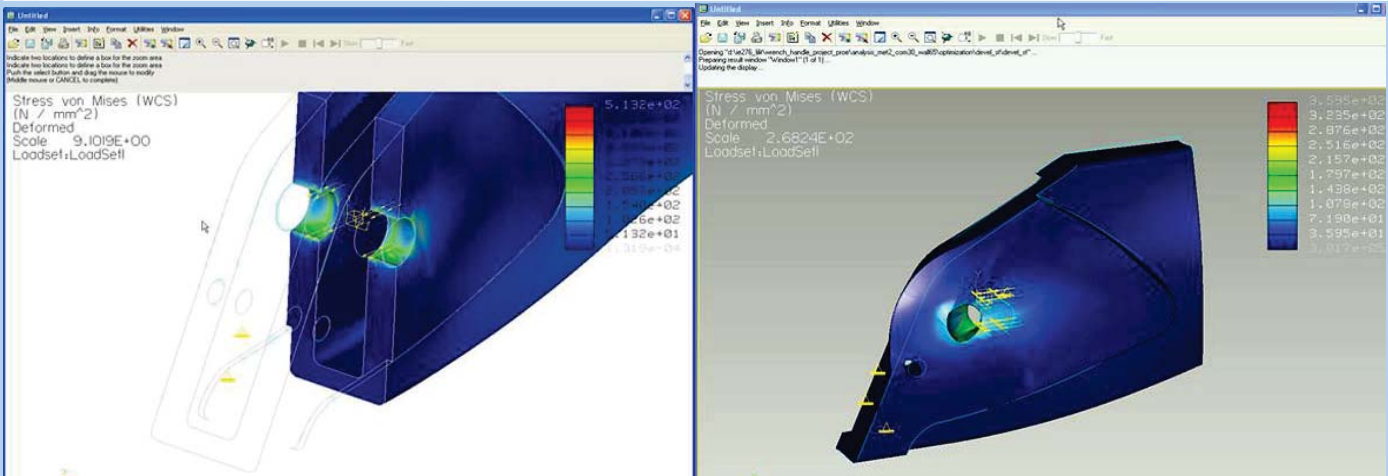
Applied Test Load of 7000 N.



Reaction Force Found via Static Analysis

## Structural Analysis

The reaction forces obtained in the Mechanism Design module were transferred into Pro/Mechanica. The first static analyses showed that the resulting stresses exceeded the allowable stress for the given material. We ran sensitivity analyses and found the dependence of the stress on the three variable parameters (the thickness of the handle in two different locations and the thickness of the metallic reinforcement). As a result, we found the range of those parameters for the optimization analysis. The goal of the latter was to minimize the mass of the model, given the yield stress of the material. The modified model that satisfied the test requirements was found and recommended to UTL.



The stress in the original model exceeds the allowable stress. The result window at right shows that the improved model will withstand the test load.