

Advanced Techniques for Predicting Mechanical Product Design Via COMSOL Multiphysics

S. Zhuang¹

¹CAEaid, Inc., Austin, TX, USA

Abstract

Finite element analysis (FEA) has a crucial and practical application to help predict and optimize product design in many engineering fields, especially in Mechanical product design area. However, because of complicated geometries, high nonlinearity, and complex constraints, how to well use FEA in product design prediction still remains as an extremely challenging obstacle for many industrial applications.

When performing mechanical product design prediction, it is significant but too challenging to represent joints, define contact pairs and contact settings, create boundary conditions, and select material models. Lots of time it might be thought as a very painful process to obtain a successful solution and diagnose the convergence issue.

COMSOL Multiphysics® software has capabilities of providing excellent mesh quality, representing joints via rigid connectors, performing contact analysis, and representing nonlinear materials. This topic presents advanced techniques for performing reliable and efficient mechanical product design simulation and prediction via COMSOL® software. First, significant challenges of mechanical simulation are presented. Second, the techniques and tricks to use several key features (rigid connector, contact pair, contact definition, time step range, etc.) are discussed. Third, the painless procedures to diagnose the model issues are addressed. Lastly, typical models including piston, structural beam, and gear contact, are applied to demonstrate the key presented technologies.

After this topic, the audiences will be able to apply COMSOL Multiphysics® software to (1) well use rigid connectors to represent different kinds of joints in mechanism design prediction, (2) perform convergent and efficient contact analysis, and (3) define multiple time zones for complicated nonlinear FEA.

Reference

S. Zhuang, M. G. Lee, Y. T. Keum, J. H. Kim, and R. H. Wagoner, "Improved contact procedure for implicit finite element sheet forming simulation", *International Journal for Numerical Methods in Engineering*, 1759–1779(2010).

D. Peric and D. R. J. Owen, "Computational Model for 3-D Contact Problems with Friction Based on the Penalty Method", *International Journal for Numerical Methods in Engineering*, 1289-1309(1992).

P. Wriggers, "Computational Contact Mechanics", West Sussex: John Wiley & Sons, 2002.

Figures used in the abstract

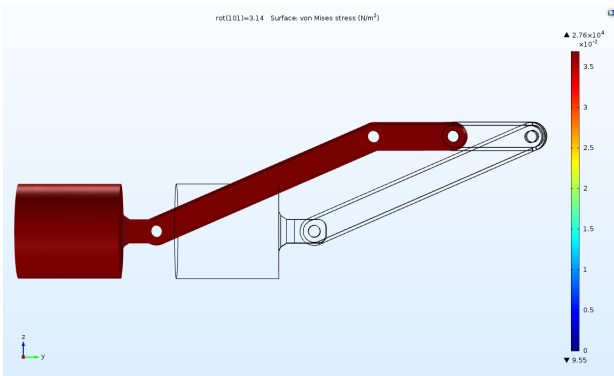


Figure 1: Piston simulation

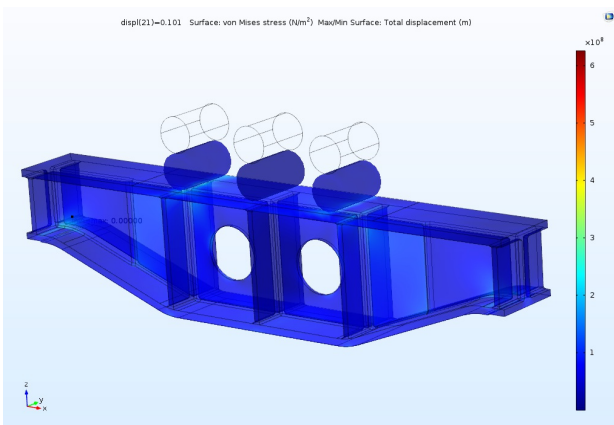


Figure 2: Contact with steel beam

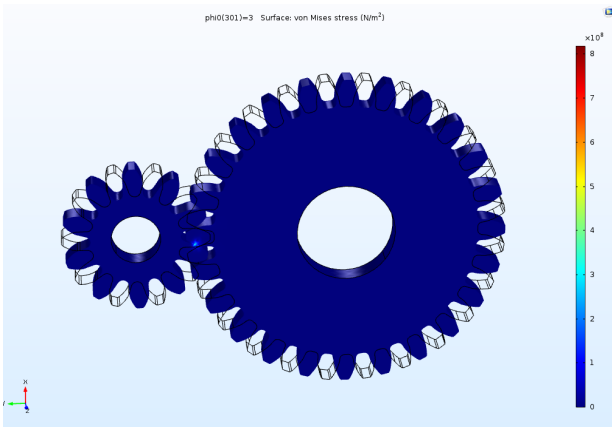


Figure 3: Gear contact analysis