Advanced Techniques for Predicting Mechanical Product Design via COMSOL Multiphysics

Shoubing Zhuang[1]
[1]CAEaid, Inc.
*Corresponding author: 15804 Jeffs Ln, Austin, TX 78717, shoubing.zhuang@caeaid.com

Abstract: Mechanical product design is crucial in many engineering field and lots of industries. To reduce the cost and shorten the design life cycle, CAE simulation has been widely applied for predicting virtual product design. Because of complicated geometry, material/geometrical nonlinearity and complex conditions, it is painful to efficiently and accurately simulate and predict challenging mechanical product designs, especially for those having joint connections or contact interactions. Both joint connections and contact interactions result in severe nonlinearity and stress concentration in local areas, so it is prohibitive to predict product designs with these two features via CAE simulation. COMSOL is known for its powerful Model Builder and complete simulation capabilities. In this paper, COMSOL Multiphysics is applied to predict typical mechanical product designs, where rigid connectors represent joint connections and penalty method executes contact interactions.

Keywords: Product Design, Joint, Rigid connector, Contact, Penalty Method

1. Introduction

Mechanical product design appears in many engineering field and lots of industries, such as automotive components, mechanism design, computer hardware, semiconductor devices, and offshore structures. CAE simulation has been widely applied for predicting and simulating virtual product designs to help reduce the cost and shorten the design life cycle. However, it is still painful to efficiently and accurately simulate and predict challenging mechanical product designs because of complicated geometry, material/geometrical nonlinearity and complex conditions, especially for those having joint connections [1] or contact interactions [2, 3]. Both joint connections and contact interactions result in severe nonlinearity and stress concentration in local areas, so it is prohibitive to predict product designs with these two features via CAE simulation.

In this paper, the feature of rigid connector in COMSOL Multiphysics is presented to represent joint connections and simplify corresponding simulation. Contact executed by penalty method is also discussed and investigated. Two typical mechanical design models are simulated and discussed for the purpose of verification.

2. Joint Represented by Rigid Connector

Joints have utmost importance commonly in virtual mechanical products. However, product designers usually pay more attention to prediction and evaluation of whole mechanical design rather than the small geometrical features such as joints. Moreover, in the prediction and simulation process, joints generate tons of elements which significantly increase the computation time and memory allocation.

COMSOL provides a powerful feature – rigid connector, which is a special kinematic constraint that works like attaching all connected boundaries by a common rigid body. One rigid connector can be attached to one or several boundaries.

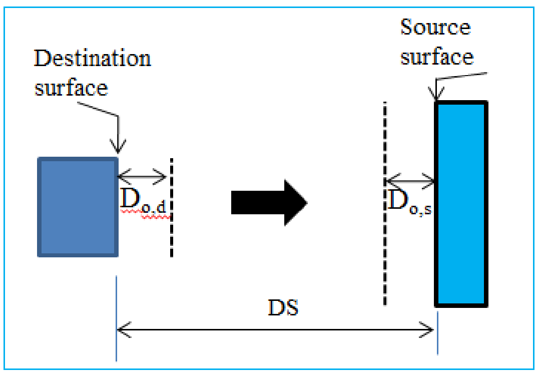
To represent the real mechanism of a joint connection in virtual design, it is necessary to well set up the rigid connector, including coordinate system, center of rotation, prescribed displacement at center of rotation (u_{0x}, u_{0y}, u_{0z}), and prescribed rotation (Ω_x, Ω_y, Ω_z).

3. Contact Executed by Penalty Method

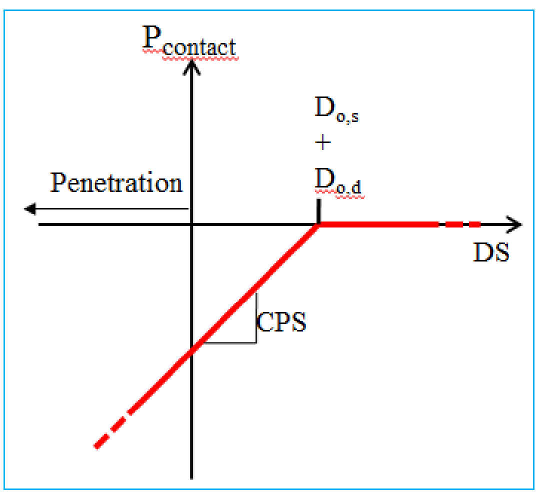
In COMSOL Multiphysics, there are two contact algorithms available: the augmented Lagrangian method and the penalty method. The penalty method has an advantage of no increase in degrees of freedom of global system [4, 5]. Moreover, it can be easily implemented into existing software. It is also known that the performance of the penalty method depends on the penalty parameter, as shown in Table of Appendix. A large penalty parameter results in slow convergence, while a small penalty parameter may reduce the accuracy. In

COMSOL Multiphysics, the penalty method usually shows better convergence, and the default contact pressure penalty factor usually works quite well.

Figure 1.a shows that the distance (DS) of a slave node on the destination boundary is calculated between the slave node and its mapping on the source boundary. DS is used to check contact status. In COMSOL Multiphysics, there are two contact surface offsets: one is the contact surface offset from geometric destination surface ($D_{o,d}$), and the other is the contact surface offset from geometric source surface ($D_{o,s}$), Note that these two offsets are also used in contact checking and contact pressure calculation, as show in Figure 1.b, based on contact pressure penalty factor (CPS). Note that if DS is larger than the sum of $D_{o,d}$ and $D_{o,s}$, the contact pressure is zero.



(a)



(b)

Figure 1. Contact executed by penalty method: (a) definition of contact distance and contact surface offsets, (b) Calculation of contact pressure.

If there is bad convergence, the contact pressure penalty factor can be reduced, while if there is a big penetration, the contact pressure penalty factor can be enlarged. There is another trick that a small value of contact surface offset can be used to reduce the penetration without the increase of the contact pressure penalty factor.

4. Numerical Examples

This section presents two numerical examples to demonstrate the key discussed methods and technologies.

4.1 Piston

The CAD model of the piston is imported into COMSOL. “Steel AISI 4340” is applied to the model from “Built in” material library of COMSOL. The basic material properties are shown in Table 2 of Appendix. The boundary conditions include a rotation of 6.28 rot and a pressure of 100 Pa, as shown in Figure 3.

Figure 4 shows 5 rigid connectors (RC1~RC5) are defined to represent joint connections in the piston model, where global coordinate system is applied. The prescribed displacements (u0x, u0y, u0z) at center of rotation and prescribed/constrained rotation (Ωx, Ωy, Ωz) for each rigid connector are set up:

RC1:

$$u_{0x}^{1}=u_{0y}^{1}=u_{0z}^{1}=0$$

$$\Omega_{x}^{1}=6.28\,\text{rad},\ \Omega_{y}^{1}=\Omega_{z}^{1}=0$$

RC2:

$$u_{0x}^{2}=u_{0x}^{3},\ u_{0y}^{2}=u_{0y}^{3},\ u_{0z}^{2}=u_{0z}^{3}$$

$$\Omega_{y}^{2}=\Omega_{z}^{2}=0$$

RC3:

$$u_{0x}^{3}=0$$

$$\Omega_{y}^{3}=\Omega_{z}^{3}=0$$

RC4:

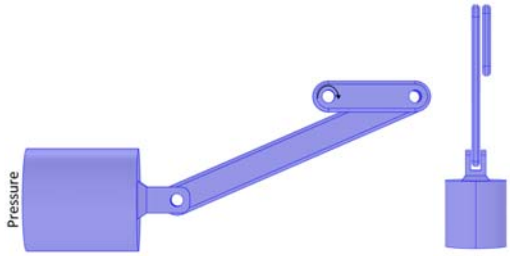
$$u_{0x}^{4}=u_{0x}^{5},\ u_{0y}^{4}=u_{0y}^{5},\ u_{0z}^{4}=u_{0z}^{5}$$

$$\Omega_{y}^{4}=\Omega_{z}^{4}=0$$

RC5:

$$u_{0x}^{5}=0,\ u_{0z}^{5}=0$$

$$\Omega_{y}^{5}=\Omega_{z}^{5}=0$$



(a) (b)

Figure 3. The schematic of piston: (a) YZ view; (b) XY view.

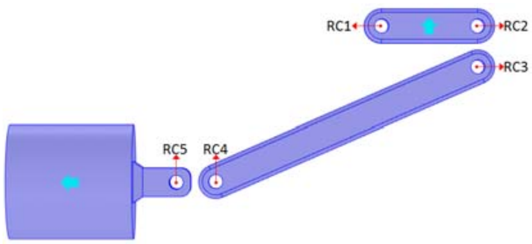
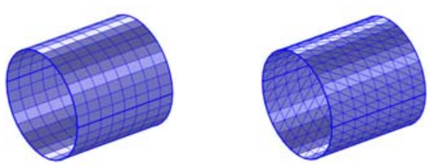


Figure 4. Defined rigid connectors (YZ view).

Considering the cylindrical surface shape of joint holes, "Mapped" node and "Distribution" node in COMSOL are first used to generate rectangular surface mesh, and then rectangular surface mesh is converted into triangular surface mesh by use of "Convert" node.

Once the special options are applied, 'Free tetrahedral' is selected with the meshing properties in Table 4 of the Appendix. The meshed piston is meshed with tetrahedral elements as shown by Figure 6.



(a) (b)

Figure 5. Meshed joint surface: (a) "Mapped" mesh; (b) "Convert" mesh.

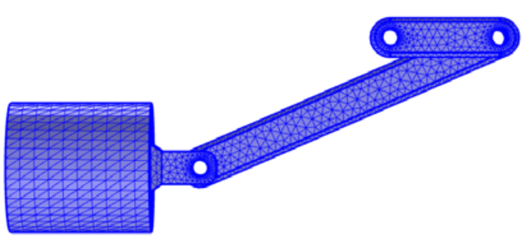


Figure 6. Meshed piston.

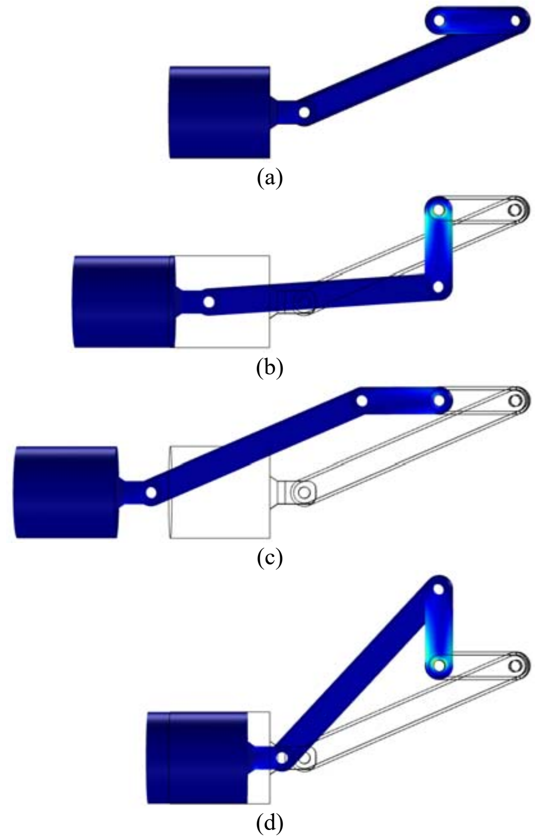


Figure 7. Simulation results of piston at the rotation of: (a) 0 rad; (b) 1.57 rad; (c) 3.14 rad; (d) 4.71 rad.

Figure 7 shows the simulated results of the states at the rotation with a value of 0, 1.57, 3.24, and 4.71 rad. By use of rigid connector, the joint connections can be appropriately represented, and the mechanism of the piston model can be predicted and simulated.

4.2 Steel Beam under Contact

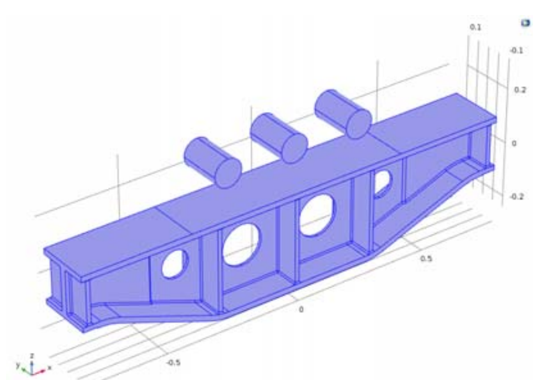


Figure 8. The schematic of steel beam under contact.

The CAD model of the steel beam is imported into COMSOL. Three cylinders are built using COMSOL Model Builder. “Structural steel” is applied to steel beam and three cylinders from “Built in” library of COMSOL. The basic material properties are shown in Table 4 of the Appendix.

Two surfaces on both sides of the bottom are wholly fixed on X-, Y- and Z- translations. The three cylinders move downward to be in contact with the steel beam. The contact pair defined appears in Figure 10, where the source boundary is selected from the steel beam and the destination boundaries are selected from the three cylinders. Here, the contact pressure penalty factor and contact surface offsets are using the default values.

Considering the regularity and symmetry of contact surfaces, “Mapped” node is used to generate rectangular mesh, which is transformed to triangular mesh by “Convert” node. After that, “Free tetrahedral” is selected to generate mesh for the whole model, shown in Figure 12.

Figure 13 shows the simulated results of Von Mises stress. Except for the contact areas, the high stress distribution in the steel beam is the interesting point to the mechanical designer. Using penalty method, COMSOL Multiphysics can well predict virtual structures with contact.

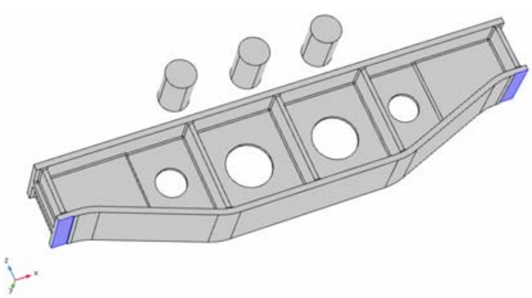


Figure 9. Fixed surfaces for steel beam.

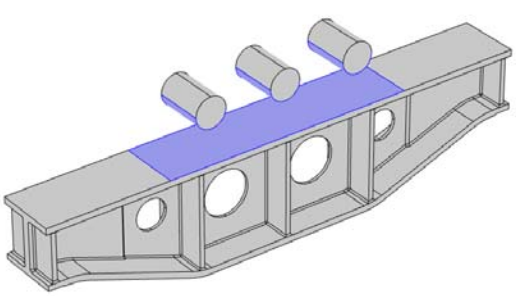


Figure 10. Definition of contact pair between cylinders and steel beam.

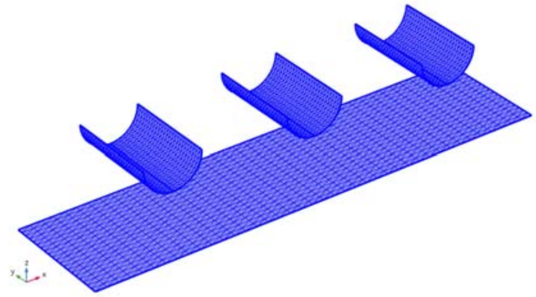


Figure 11. Meshed contact surfaces.

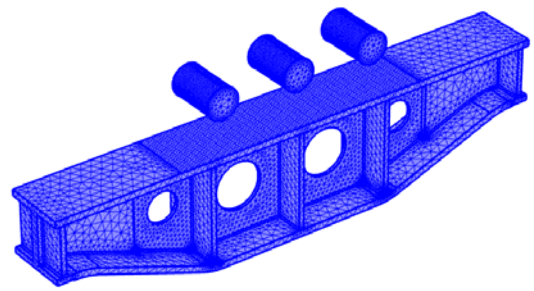


Figure 12. Meshed cylinders and steel beam.

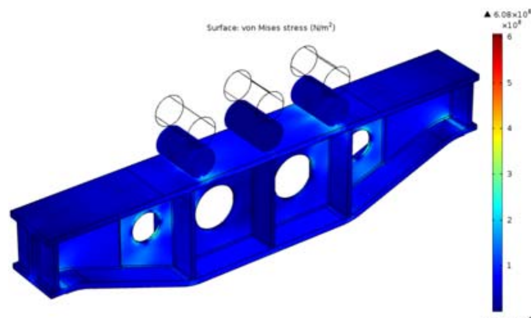


Figure 13. Von Mises stress distribution.

5. Conclusions

This paper presents the use of COMSOL Multiphysics that successfully predicts and simulates two typical mechanical designs.

By use of rigid connector nodes to represent joint connections, the severe nonlinearity, contact interaction and stress concentration in local joint connection areas of the piston model can be neglected, which greatly reduces the total DOF and then enhances the performance of modeling, meshing and simulation.

Contact definition can be easily set up via COMSOL Multiphysics. By use of penalty method, steel beam under contact is successfully simulated with good convergence.

6. References

1. L. Molnár, K. Váradi, etc., “Stress Analysis of Bolted Joints Part II. Contact and Slip Analysis of a Four Bolt Joint”, *Modern Mechanical Engineering*, **4**, 46-55 (2014)
2. O. C. Zienkiewicz and A. Francavilla, “A Note on Numerical Computation of Elastic Contact Problems”, *International Journal for Numerical Methods in Engineering*, **9**, 913-924 (1975)
3. P. Wriggers, “Computational Contact Mechanics”, West Sussex: John Wiley & Sons (2002)
4. S. Zhuang**,** “Advanced Techniques and Painless Procedures for Nonlinear Contact Analysis and Forming Simulation via Implicit FEM”, *AIP Conference Proceedings*, **1532**, 499-504 (2013)
5. K. W. Man, "Contact mechanics using boundary elements", ed. C. A. Brebbia and J. J. Conner, UK: Computational Mechanics Publications (1994)

7. Acknowledgements

The author would like to thank David Kan at COMSOL, Inc. for valuable technical support and discussions.

8. Appendix

Table 1: Influence of contact pressure penalty factor on convergence and Accuracy

Contact Pressure Penalty Factor	Convergence	Accuracy
↑	↓	↑
↓	↑	↓

Table 2: Material properties of Steel AISI 4340

Material Property	Value	Unit
Density	7850	kg/m^3
Young’s Modulus	205	GPa
Poisson’s Ration	0.33	_

Table 3: Meshing properties for piston model

Meshing Property	Value	Unit
Predefined Element Size	Extra fine	_
Maximum Element Size	0.0209	m
Minimum Element Size	8.94E-4	m
Maximum Element Growth Rate	1.35	_
Curvature Factor	0.3	_
Resolutions of Narrow Regions	0.85	_

Table 4: Material properties of Structural Steel

Material Property	Value	Unit
Density	7850	kg/m^3
Young’s Modulus	200	GPa
Poisson’s Ration	0.33	_