ICIC Express Letters
Volume **12**, Number **7**, July **2018**
ICIC International ©2018 ISSN 1881-803X
pp. 655–662

PERFORMANCE EVALUATION OF OPEN-SOURCE STRUCTURAL ANALYSIS SOLVER, CALCULIX AND CODE-ASTER, FOR LINEAR STATIC AND CONTACT PROBLEMS

Seung-Keun Park, Dong-Woo Seo, Heeseok Jeong and Myungil Kim*

Department of Supercomputing M&S Technology Development
Korea Institute of Science and Technology Information
245 Daehak-ro, Yuseong-gu, Daejeon 34141, Korea
{ skpark; seodongwoo; hsjeong }@kisti.re.kr
*Corresponding author: mikim@kisti.re.kr

Received December 2017; accepted March 2018

Abstract. *This paper evaluates simulation results obtained using CalculiX and Code-Aster, which are open-source structural analysis solvers for modeling and simulation (M&S). Linear static analyses are conducted with a pipe model under a pressure load and a cantilever model under a remote force. Contact analyses are tested using a two-beam model with tie, sliding and general contact conditions. Outputs obtained using CalculiX and Code-Aster agree well with those of the commercial software, ANSYS. These results suggest that CalculiX and Code-Aster are reliable, and can be used to improve efficiency of product development in manufacturing and engineering.*
Keywords: Modeling & simulation, Small and medium sized enterprises, Open-source solver, CalculiX, Code-Aster, ANSYS

1. **Introduction.** Modeling and simulation (M&S) is an important technology in many areas such as production manufacturing, automotive industry and civil engineering [1,2]. Various commercial M&S software (SW) have been widely used to reduce production development cost and time while improving products. However, the high cost of commercial SW discourages its use in production development, especially by small and medium sized enterprises (SMEs). Open-source SW can be an alternative to resolve this cost problem [3,4]. However, many people are skeptical about the reliability of results obtained using open-source SW. Therefore, the outputs of open-source SW should be compared to those of commercial SW using various examples.

This paper evaluates two open-source structural analysis solvers, CalculiX and Code-Aster. CalculiX and Code-Aster are thermo-mechanical finite element (FE) solvers that support numerous analysis types including linear static, contact, modal, nonlinear material, heat transfer, and implicit/explicit dynamic analyses [3,4]. CalculiX is widely used as a finite element (FE) solver in several applications such as FreeCAD [5], CastNet [6], Mecway [7] and SimScale [8], because of its ease of use. Code-Aster has its own pre-platform and post-platform, which is Salome-Meca [4], and is also used as a solver in SimScale [8]. The results from the two open-source SW are compared with the results from the commercial SW, ANSYS [9], which is regarded as correct. The versions used are CalculiX 2.11 [3] and Code-Aster 12.6 [4]. To visualize the results from CalculiX, an in-house developed SW (HEMOS) [10] is used; to visualize the results of Code-Aster, Salome-Meca [4] is used. The remainder of this paper is organized as follows. Section 2 presents the results of linear static analysis with pressure and remote force. Section 3 reports tie, sliding and general contact analyses and compares the results. Section 4 concludes the paper.

DOI: 10.24507/icicel.12.07.655

2. **Results of Linear Static Analysis.** This section compares the results of linear static analysis calculated using open-source SW (CalculiX and Code-Aster) and the results calculated using commercial SW (ANSYS). Various types of loads such as facial loads (i.e., pressure) and concentrated load (i.e., remote force) are tested. RBE2 element, which is called a kinematic coupling element in ABAQUS, and the RBE3 element, which is a distributing coupling element in ABAQUS, are used to apply remote force at a remote point [11]. The accuracy of the results is evaluated by the error of maximum displacements

$$e_u = |\hat{u}_{\max} - \bar{u}_{\max}|/|\hat{u}_{\max}|, \tag{1}$$

and von-Mises stresses

$$e_\sigma = |\hat{\sigma}_{\max} - \bar{\sigma}_{\max}|/|\hat{\sigma}_{\max}|, \tag{2}$$

in each case, normalized to the maximum values calculated using ANSYS, where $\hat{u}_{\max}$ and $\bar{u}_{\max}$ represent maximum displacement calculated by commercial and open-source SW, respectively, and $\hat{\sigma}_{\max}$ and $\bar{\sigma}_{\max}$ represent maximum von-Mises stress calculated by commercial and open-source SW, respectively.

The analysis considers a pipe model (Figure 1 left) which is symmetric to the xy-plane. A pressure of 10MPa is applied to the inner surface (Figure 1 left, black) and both ends are fixed ($d_x = d_y = d_z = 0.0$ mm). The symmetric condition (i.e., $d_z = 0.0$ mm) is applied to the faces that are normal to the xy-plane. The mesh model (Figure 1 right) has 5395 nodes and 900 elements. To test the solvers with the same conditions, the mesh model was generated with structured mesh and all elements are quadratic. The pipe has Young's modulus $E = 210$ GPa and Poisson's ratio $\nu = 0.3$, which are representative material properties of alloy steel. A full integration scheme was used for Gaussian integration [12,13].

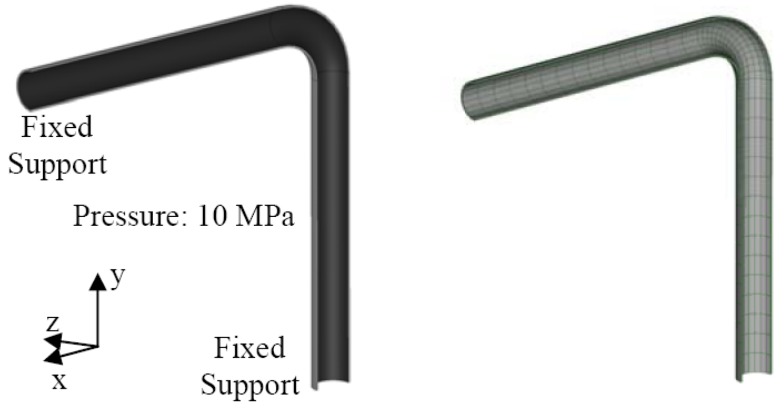
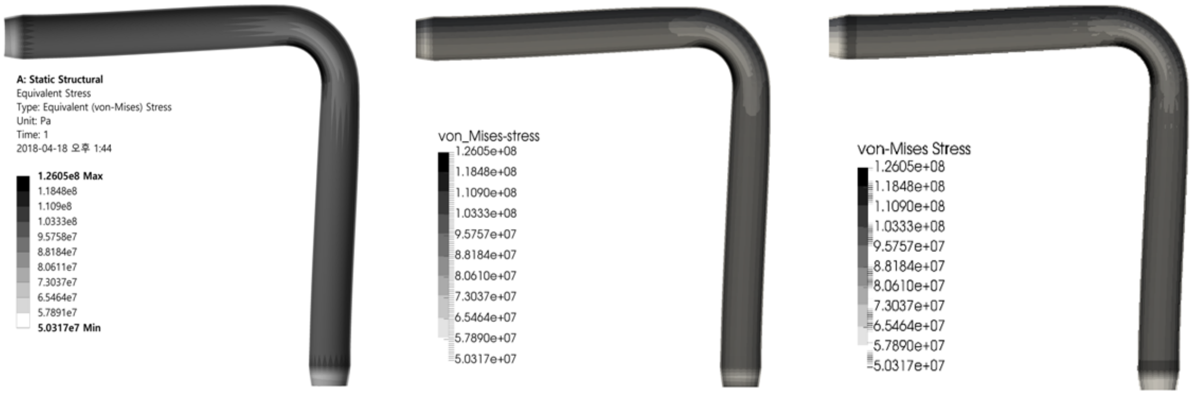


FIGURE 1. Symmetric pipe model and its mesh model

The overall distributions predicted by the linear static analysis with the three solvers were very similar (Figure 2). All solvers yield the same maximum displacements, and the von-Mises stress differed from ANSYS results by 0.68% and 0.28% for CalculiX and Code-Aster (Table 1). These similarities imply that open-source solvers are reliable when they use the same mesh model and integration scheme as ANSYS for linear static analysis under pressure load.

Tests of RBE2 and RBE3 elements under remote force consider an L-shaped model (Figure 3 left). The fixed support condition is applied at the bottom face. The top face and a remote point (Figure 3 left, black) are linked with the RBE2 and RBE3 elements. RBE2 restricts the deformation of the linked nodes on the black face, whereas RBE3 element allows deformation of these nodes. Concentrated loads of 1000N are applied at the remote point toward the $-y$-direction for load-case I, and toward the z-direction for load case II. This model also has $E = 210$ GPa and Poisson's ratio $\nu = 0.3$. The mesh



(a) ANSYS (b) CalculiX (c) Code-Aster

Figure 2. Deformed shapes and stress distributions of the pipe model

Table 1. Maximum displacements and von-Mises stresses for three solvers

Solver	**Displacement (mm)**		**von-Mises Stress (MPa)**	
ANSYS	0.442		126.05	
CalculiX	0.442	**0.00%**	125.19	**0.68%**
Code-Aster	0.442	**0.00%**	126.40	**0.28%**

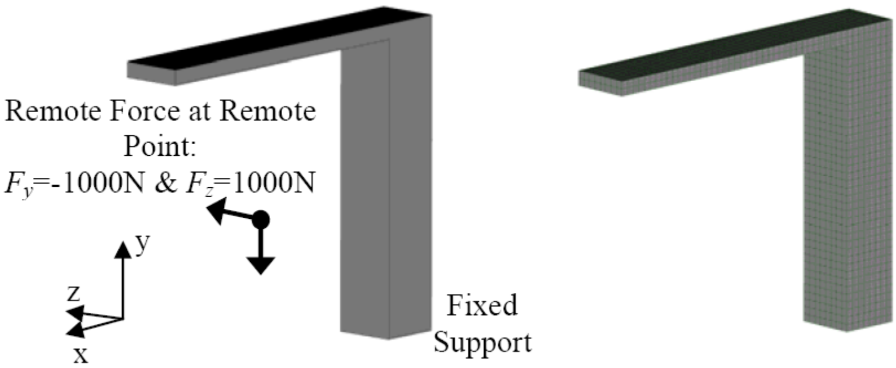
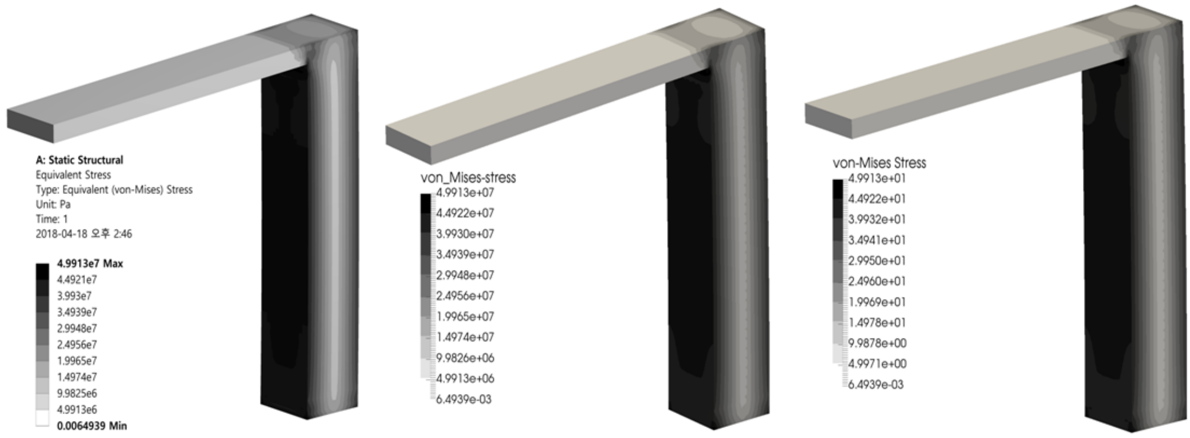


Figure 3. L-shape model and its mesh model

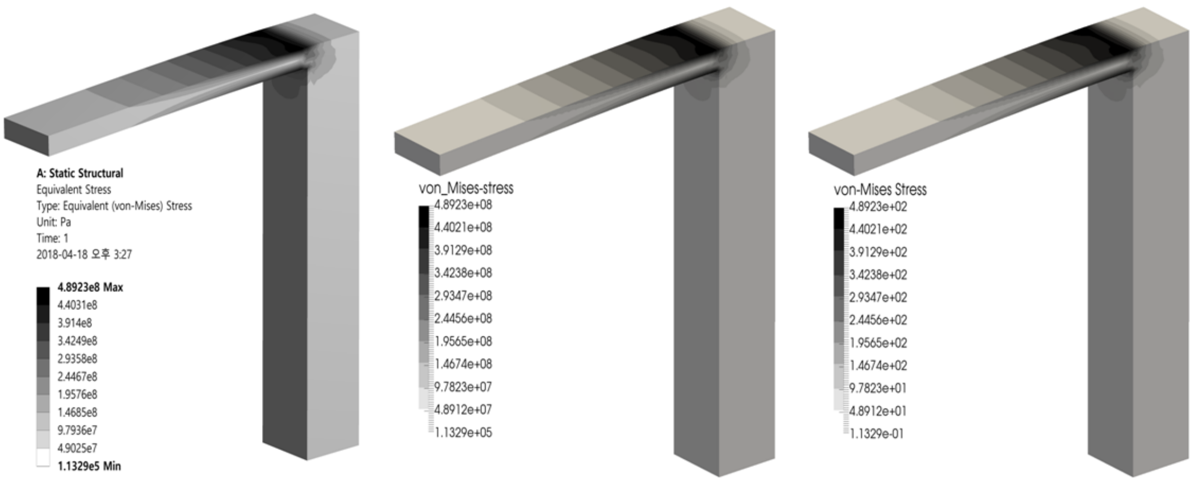
model (Figure 3 right) consists of 3200 quadratic hexahedron elements and 16306 nodes including the node at the remote point.

For load-case I, the von-Mises stresses calculated by CalculiX and Code-Aster are very similar to the results calculated by ANSYS (Figures 4 and 5). Because the RBE2 element does not allow the top face to deform, the stress at the top face is almost zero (Figure 4). However, the RBE3 element does allow the top face to deform, and the vertical column is 64 times stiffer against bending than the horizontal beam (i.e., the column (20 mm) is four times thicker than the beam (5 mm) and the bending stiffness is proportional to the third power of the thickness), the horizontal beam behaves like a cantilever (Figure 5). For load-case II, the von-Mises stresses calculated by CalculiX and Code-Aster are similar to the results obtained using ANSYS (Figures 6 and 7). In Figure 6, the top face is rigid, so the exerted torque twists the vertical column along the y-axis. The stresses calculated by the three solvers have similar distributions (Figure 6). However, in Figure 7, the horizontal beam is flexible, so the remote force twists it along x-axis. The deformed shape and the stress distribution calculated by CalculiX (Figure 7(b)) are different from the results obtained by ANSYS and Code-Aster. Additional studies of RBE3 element with CalculiX, confirm that the location of the remote point does not affect the analysis results;



(a) ANSYS (b) CalculiX (c) Code-Aster

FIGURE 4. Deformed shapes and stress distributions with RBE2 element for load-case I

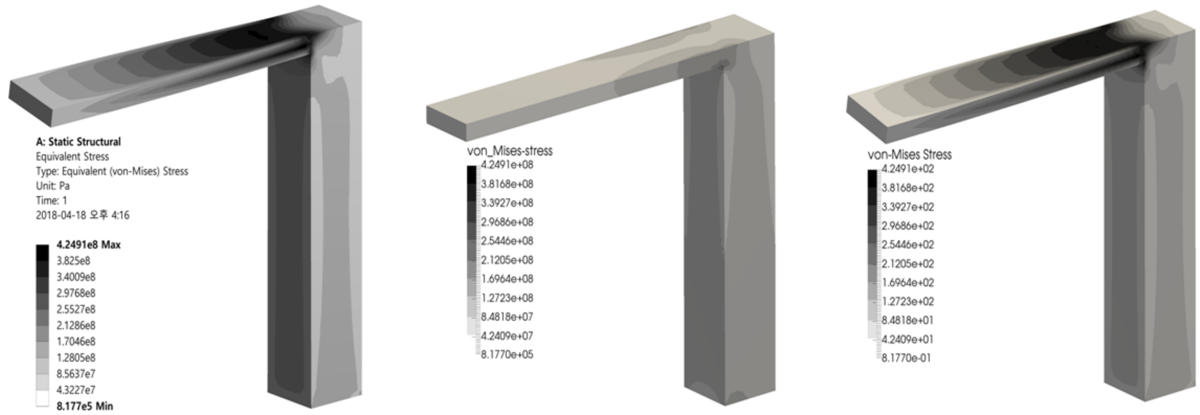


(a) ANSYS (b) CalculiX (c) Code-Aster

FIGURE 5. Deformed shapes and stress distributions with RBE3 element for load-case I

(a) ANSYS (b) CalculiX (c) Code-Aster

FIGURE 6. Deformed shapes and stress distributions with RBE2 element for load-case II



(a) ANSYS (b) CalculiX (c) Code-Aster

Figure 7. Deformed shapes and stress distributions with RBE3 element for load-case II

Table 2. Max. displacements and von-Mises stresses for two load-cases and RBE elements

Element	Load Case	**Displacement (mm)**			**von-Mises Stress (MPa)**		
		ANSYS	CalculiX	Code-Aster	ANSYS	CalculiX	Code-Aster
RBE2	Case I	0.224	0.213	0.213	49.91	51.72	49.96
			4.91%	**4.91%**		**3.63%**	**0.10%**
	Case II	0.349	0.332	0.332	66.68	67.91	66.43
			4.87%	**4.87%**		**1.84%**	**0.37%**
RBE3	Case I	2.95	2.83	2.83	489.23	513.25	490.52
			4.07%	**4.07%**		**4.91%**	**0.26%**
	Case II	0.761	0.611	0.727	424.91	189.17	420.87
			19.71%	**4.47%**		**55.47%**	**0.95%**

this observation means that the RBE3 element in CalculiX is not reliable especially when the torsional force is applied to the structure. The maximum displacements and von-Mises stresses calculated using CalculiX and Code-Aster are quite close to those calculated by ANSYS except the CalculiX result of load-case II with element RBE3 (Table 2).

3. **Results of Contact Analysis.** This section compares the results for contact analysis calculated using CalculiX and Code-Aster to those calculated using ANSYS. Analyses are conducted for tie, sliding and general contacts (i.e., bonded, no separation and frictional contacts in ANSYS). However, CalculiX does not support the analysis of sliding contact, so only Code-Aster is used for this comparison. For the general contact analysis, all solvers use penalty method [11,14] and the number of sub-steps is set to 10.

The analysis considers a two-beam model (Figure 8). The three contact conditions are defined on the surface at which the beams contacted each other, and fixed support conditions are assigned to both side surfaces. A pressure load of 1 MPa is applied to one face above the contact area (Figure 8 top, black). This model also has $E = 210$ GPa and Poisson's ratio $\nu = 0.3$. The mesh model (Figure 8 bottom) is composed of 5120 quadratic hexahedron elements and 26802 nodes.

The deformed shapes and overall stress distributions produced by the models looked similar for all contact cases (Figures 9-11). When the two bodies are tied (Figure 9), both tangential and normal displacements are restricted on the contact surfaces; therefore, the

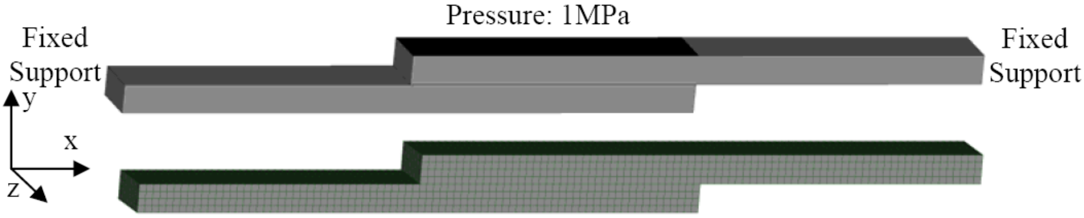
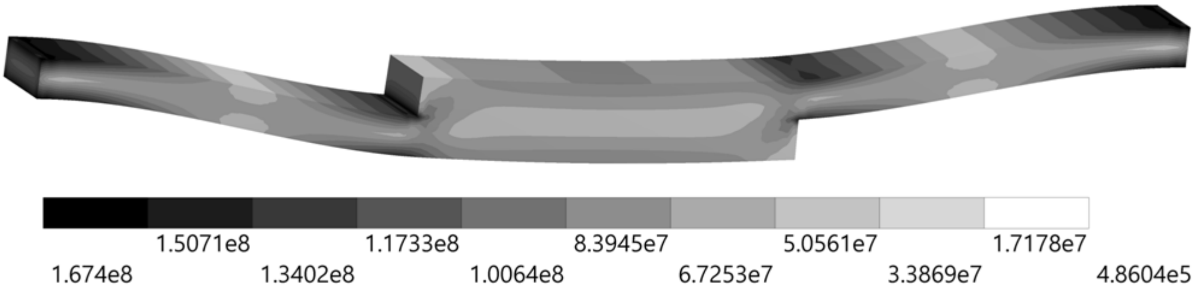
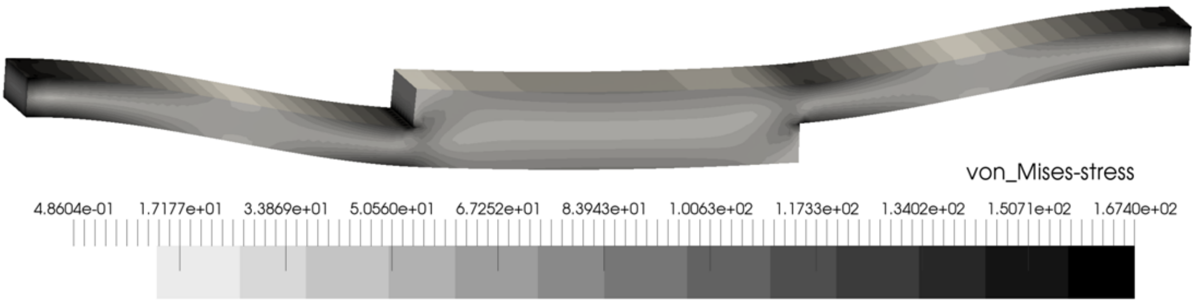


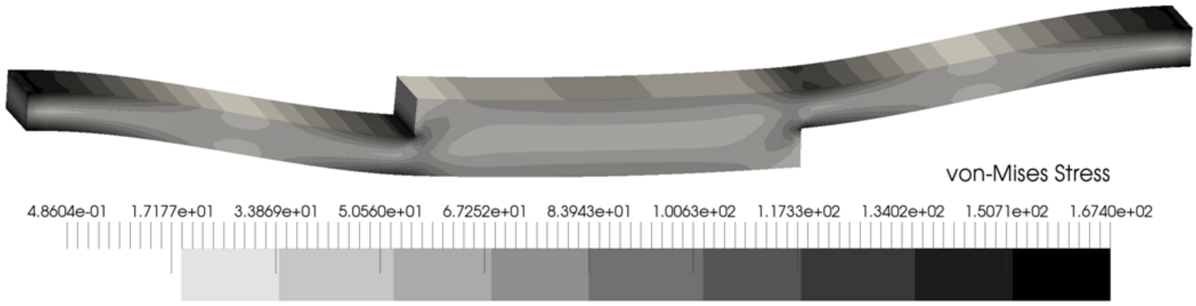
Figure 8. Two-beam model and its mesh model



(a) ANSYS



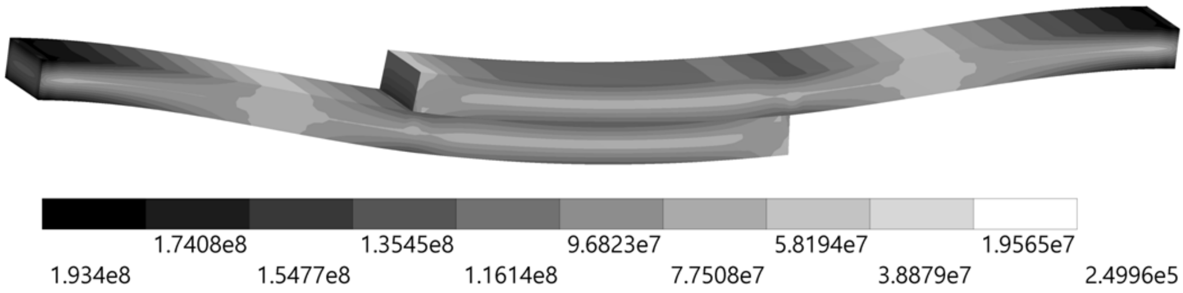
(b) CalculiX



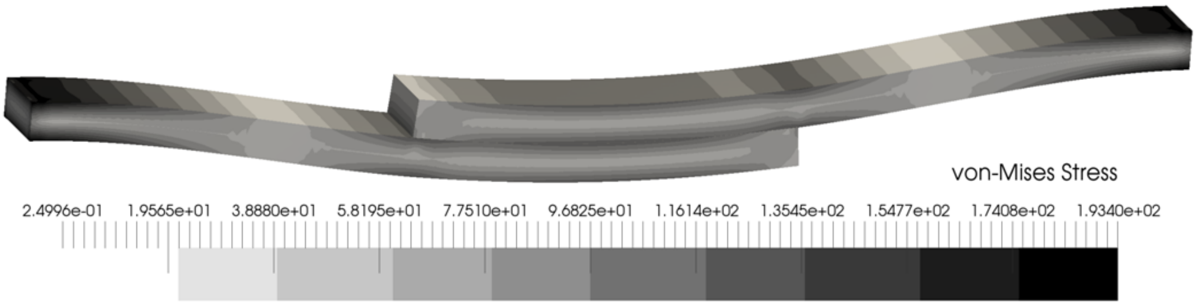
(c) Code-Aster

Figure 9. Deformed shapes and stress distributions with tie contact condition

nodes on the contact surfaces are aligned exactly and the two beams have continuous stress distribution around the contact surfaces. When the contact condition is set to sliding, only normal displacements between the both contact surfaces are restricted; therefore, the tangential displacements are different on the two sides and the stress distribution is not continuous around the contact surfaces (Figure 10). When the contact is general, both tangential and normal displacements are allowed between the contact surfaces; therefore, the two bodies are detached (Figure 11). Even though the general contact analysis is a nonlinear problem, the open-source SW yields similar results to those of ANSYS. Table 3 compares the maximum displacements and von-Mises stresses for three types of contact analysis with three solvers. In most cases, the errors of the calculated displacements are less than 1%, and the errors of the stresses are 0.5% to 5.14% except the tie contact

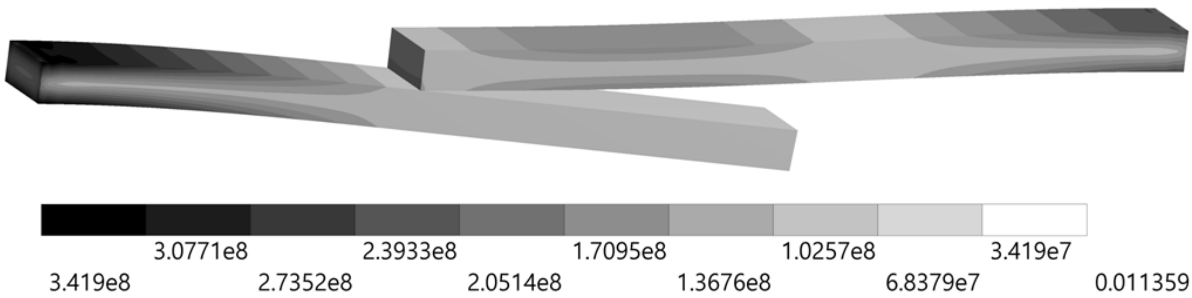


(a) ANSYS

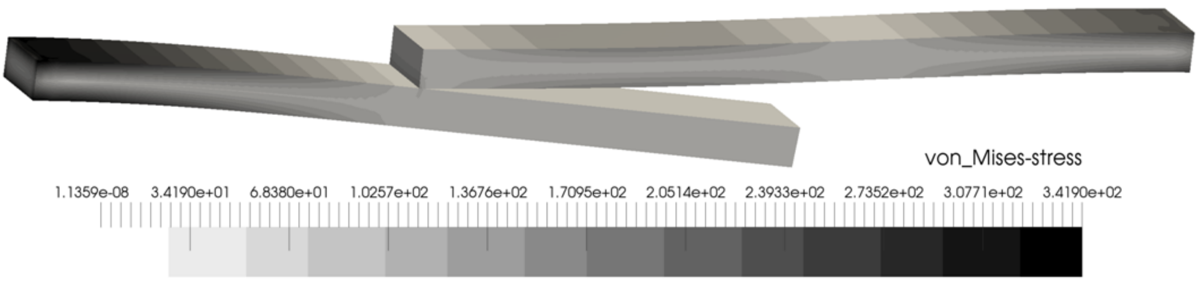


(b) Code-Aster

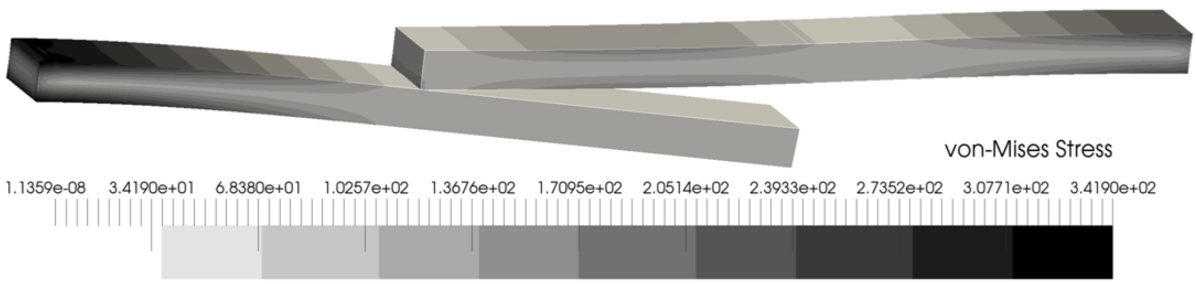
Figure 10. Deformed shapes and stress distributions with sliding contact condition



(a) ANSYS



(b) CalculiX



(c) Code-Aster

Figure 11. Deformed shapes and stress distributions with general contact condition

Table 3. Max. displacements and von-Mises stresses for two load-cases and RBE elements

Contact Type	**Displacement (mm)**			**von-Mises Stress (MPa)**		
	ANSYS	CalculiX	Code-Aster	ANSYS	CalculiX	Code-Aster
Tie	0.176	0.176	0.176	167.40	186.29	176.00
		0.00%	**0.00%**		**11.28%**	**5.14%**
Sliding	0.274	–	0.274	193.40	–	194.50
		–	**0.00%**		–	**0.57%**
General	1.358	1.354	1.355	341.90	340.00	355.80
		0.29%	**0.22%**		**0.56%**	**4.07%**

calculated using CalculiX (Table 3). These overall results demonstrate that the open-source SW is reliable for contact analyses as well.

4. **Summary and Conclusion.** Two open-source structural analysis solvers (CalculiX, Code-Aster) in linear static and contact analyses are evaluated by comparing their results with those obtained using a commercial solver (ANSYS). The quadratic hexahedron, RBE2 and RBE3 elements are used for the verification. Pressure and remote force are tested; the tie, sliding and general contact conditions are used in the contact analysis. Results obtained using the three examples indicate that CalculiX, Code-Aster yield accurate results of linear static and contact analysis with various elements and load types, except the RBE3 element in CalculiX. In that case, CalculiX yields large error under torsional force, so care must be applied when that element is used. Future research will consider other types of the analysis, such as nonlinear materials, modal and heat transfer analyses. These solvers should also be evaluated in practical, complex and large-sized problems.

Acknowledgment. This research was supported by Information & Communications Technology Promotion (IITP) grant funded by Korea Institute of Science and Technology Information (KISTI) and Korea government (MSIT) (No. 2017-0-00350, Shape Pattern Modeling Method based on 3D CAD and Integration Analysis Platform Development for Manufacturing Engineering).

REFERENCES

[1] Deloitte and Council on Competitiveness, *2016 Global Manufacturing Competitiveness Index*, Deloitte, London, 2016.
[2] Korea Institute of Science and Technology Information (KISTI), *Supercomputing Based Modeling and Simulation 4.0 for Smart Manufacturing Innovation*, Daejeon, 2014.
[3] G. Dhondt, *CalculiX CrunchiX User's Manual Version 2.11*, http://www.dhondt.de/, 2016.
[4] A. Jean-Pierre, *Beginning with Code_Aster – A Practical Introduction to Finite Element Method Using Code_Aster Gmsh and Salome*, https://www.code-aster.org/, 2013.
[5] *FreeCAD*, https://www.freecadweb.org/.
[6] *CaseNet*, http://www.dhcae-tools.com/CastNet.html.
[7] *Mecway*, http://www.mecway.com/.
[8] *SimScale*, https://www.simscale.com/.
[9] *ANSYS*, http://www.ansys.com/.
[10] *HEMOS*, https://hemos-cloud.kisti.re.kr.
[11] G. Dhondt, *The Finite Element Method for Three-Dimensional Thermo-mechanical Applications*, John Wiley & Sons, 2004.
[12] L. E. Malvern, *Introduction to the Mechanics of a Continuous Medium*, Prentice-Hall, NJ, 1969.
[13] R. D. Cook, *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, 2007.
[14] P. Wriggers, *Computational Contact Mechanic*, John Wiley & Sons, 2002.