THE OPPORTUNITY OF USING CLOUD-BASED COMPUTING IN NUMERICAL SIMULATIONS ON STRUCTURAL ANALYSIS - CASE STUDY

VLAD ANDREI CIUBOTARIU[*1], COSMIN GRIGORAS[1], CRINA MARIA RADU[1], CATALIN NICOLAE TAMPU [1], VALENTIN ZICHIL[1]

[1] *"Vasile Alecsandri" University of Bacau, Calea Marasesti 157, Bacau, 600115, Romania*

Abstract: To make the most accurate behavioral assessments of mechanical parts, engineers rely heavily on numerous software solutions. These software solutions are able to solve more or less complex problems with a fairly high degree of accuracy and similarity to physical experiments. The common feature of these software solutions is that they need generous resources from the computer on which they are installed. Can't it be done differently? One possible answer may be the use of cloud-based solutions. The main purpose of this paper is to make a comparison and find the limitations of such a solution relative to the established ones in the field of finite element analysis. Therefore, this paper is a case study in which an industrial component – beam bracket – is subjected to a structural analysis on three different software solutions. The reference systems are: ANSYS, a solution dedicated to finite element analysis and SolidWorks Simulation, a solution often used in industry for the assessment of mechanical structures. The cloud-based solution is SimScale, a software product based on open-source codes: Code_Aster and CalculiX.

Keywords: finite element method, cloud-based solver, Ansys, SolidWorsk, SimScale

1. INTRODUCTION

Finite element analysis – FEA – is a method of numerically solving differential equations that occur mainly in engineering and other related fields. Typical areas of interest include traditional areas of structural analysis, heat transfer, fluid flow and others. FEA is a special numerical method for solving partial differential equations in two or three spatial variables. To solve a problem, FEA divides a complex system into smaller, simpler parts, which are named finite elements [1, 2].

A variety of specializations in mechanical engineering - aeronautics, biomechanics and automotive - typically use FEA to design and develop products. Several modern software solutions include specific components, such as structural and fluid, thermal and even electromagnetic work environments. In a structural analysis, FEA is particularly helpful in visualizing changes in stiffness, strength and also in minimizing weight, material consumption and cost reduction [3].

One of the most important computer-aided engineering software products in the field of FEA is ANSYS. This software solution is almost a synonym of finite element simulations. Compared to other similar products, ANSYS has extraordinary capabilities in multiphysics analysis and is used extensively in both industry – research, product development – and academia [4]. Another capable computer-aided engineering software product used for performing various tedious jobs in finite element analysis is SolidWorks Simulation – hereinafter referred to as

[*] Corresponding author, e-mail: vlad.ciubotariu@ub.ro
doi.org/10.29081/jesr.v28i1.004

SW Sim. This software solution can solve problems from a wide range of real-world loading conditions starting from linear and nonlinear static or dynamic analysis all the way to buckling, thermal and fatigue analysis [5].

This paper aims to make a comparison of these two very widespread software solutions with a slightly newer cloud-based solution, named SimScale. This computer-aided engineering software product is based on cloud-computing where the backend of the platform uses two open-source codes like Code_Aster and CalculiX [6]. This software can also perform simulations in the field of computational fluid dynamics through an open-source code such as OpenFOAM [7]. At the same time, this case study aims to find the limitations involved in cloud computing. The complexity of the analyzed components plays an essential role in the computing speed, but especially in the accuracy of the obtained data.

2. ANALYSIS SETUP AND METHODOLOGY

2.1. About the industrial component in discussion

When a steel beam-column structure, such as a manufacturing plant is constructed, its concrete filled steel tube (CFST) columns are erected before the beams can be elevated, positioned and welded or bolt connected [8]. The function of a beam bracket is to precisely position a beam and safely transfer the loads from the beam to the column. The loads are determined by a thorough analysis of the entire structure subject to design loads, such as dead load, live load, earthquake, wind load, etc. [4]. An example of such a framework is presented in Figure 1.

The beam bracket consists of a L-seat plate which present a triangular stiffener welded to it. General specification of the beam bracket with a 45° middle stiffener is presented in Figure 2. The specification of the square steel tube for the columns usually is 150 × 150 × 6 mm and 194 × 150 × 6 mm for the steel H-beams. To join these components, the beam bracket is welded to the column and connected to the beam with M12 high-strength bolts.

The beam bracket was modeled using SolidWorks CAD, after which the model was saved in the universal *.step format so that it could be imported into the other two software solutions, even if ANSYS uses a CAD modeler such as SpaceCleim.

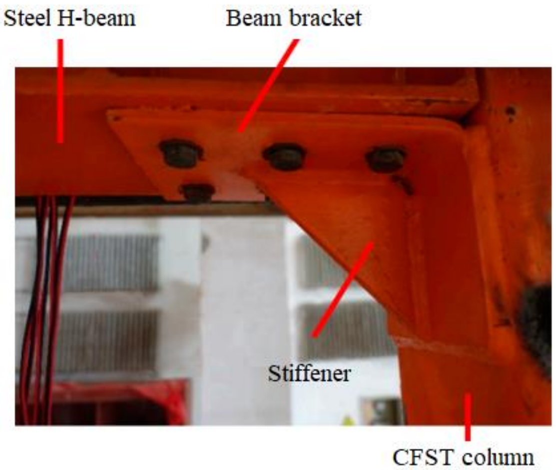

Fig. 1. Example of framework which includes a beam bracket with stiffener [8].

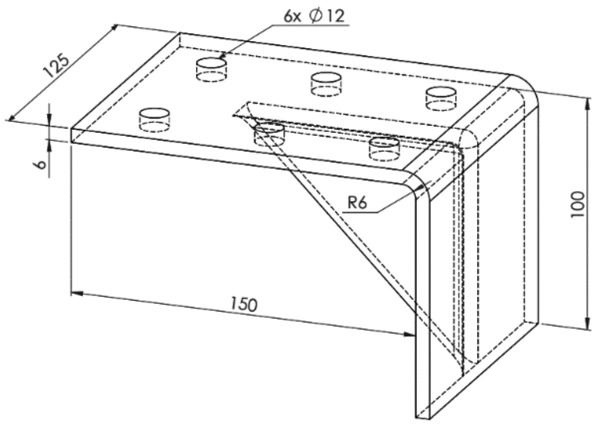

Fig. 2. General specification of the beam bracket with a 45° middle stiffener.

2.2. FEA setup

In the present case study, linear static structural simulation is used for the model presented in Figure 2. All three above-mentioned software products were used similarly to make this comparison meaningful. Apart from the geometric model, additional information is required to perform the simulations. This information is generally divided into two categories: information on the material properties and information on the environment conditions (restrictions and loads).

The beam bracket is made of structural steel which typically has a Young’s modulus of 200 GPa, the Poisson's ratio is around 0.3 and has a yield strength of approximately 250 MPa. The yield strength is very useful to assess the safety factor.

The load is 5 kN uniformly distributed on the L-seat plate. This value emerged from various analysis of the entire structure, which is not the subject of this paper. As to the support conditions, it was assumed the beam bracket's back face is rigidly welded on the steel column, therefore is considered a fixed support.

Regarding the discretization, a curvature-based mesh with the elements size between 4 and 1.2 mm was used, this being considered a standard discretization, both in ANSYS and in SW Sim. Solution accuracy depends not only on mesh density but also on mesh quality. For example, in complex nonlinear problems, poorer mesh quality often leads to more computing time or even failure of finding a solution. Achieving a high mesh quality is, however, not trivial [4].

In this case study, the factor of safety – hereinafter referred to as FoS – is the ratio between the yield strength and the maximum equivalent stress calculated on the beam bracket in discussion under the above-mentioned load.

3. RESULTS AND DISCUSSION

Once performed, the simulations showed extraordinary similarities between all above-mentioned software. As somehow expected, ANSYS showed the highest value of equivalent stresses obtained after the beam bracket was loaded with the 5 kN. Table 1, below, shows the von Mises equivalent stress, maximum displacement and FoS values obtained from the simulations. The table also shows the percentage differences obtained in comparison to the data considered as most accurate, namely the ANSYS values.

Table 1. Values obtained from simulations regarding the beam bracket in discussion using ANSYS, SW Sim and SimScale softwares.

Software solution	Comparison data values					
	Von Mises equivalent stress [MPa]	%	Maximum displacement [mm]	%	Factor of safety	%
ANSYS	186.07	0	0.340	0	1.344	0
SW Sim	185.51	-0.301	0.330	-2.941	1.348	0.2977
SimScale	177.30	-4.713	0.301	-11.470	1.410	4.9419

At a first analysis of the obtained data, it was found that the stress distribution in the analyzed beam bracket is quite similar in all three software solutions used in this study. The difference in color intensity of the graphic area is not necessarily important, because the values obtained are revealing.

Figures 3, 6, 9 and Table 1 show that the values of the von Mises equivalent stress obtained after loading the beam bracket are very similar. The data obtained by SW Sim (185.51 MPa) is almost identical to those of ANSYS (186.07 MPa), where the SimScale software (177.3 MPa) shows a negative difference of only 4.7% compared to the data obtained by the latter software.

Regarding the displacement magnitude (Figures 4, 7, 10), the same trend is observed, even if the difference is slightly higher between the value obtained by SimScale and the other two software. In this case there is a difference of 11.47% of the value obtained by SimScale (0.301 mm) compared to the one obtained by ANSYS (0.34 mm). Again, the values obtained by SW Sim (0.33 mm) are very close to those obtained by the latter software.

As mentioned above, the calculation of FoS is performed as a ratio between yield strength and the maximum value of von Mises equivalent stress obtained from the simulations. With regard to SimScale, there is no possibility here to graph a representation of the most affected areas of the analyzed part from the point of view of FoS, although it can be assumed, without fear of error, that this graph may be somehow similar to that of the von Mises equivalent stress distribution.

In this particular case there is also a difference of 4.94 % between the FoS obtained by SimScale simulation (1.41) and the calculation performed by ANSYS (1.34). Again, the difference of only 0.298 % is insignificant for the FoS found through SW Sim (1.35) compared to ANSYS.

The distribution of equivalent stress according to FoS (Figures 5 and 8) is concentrated in the area before the gusset, noting that SW Sim displays a more radiant distribution of these stress comparing to ANSYS.

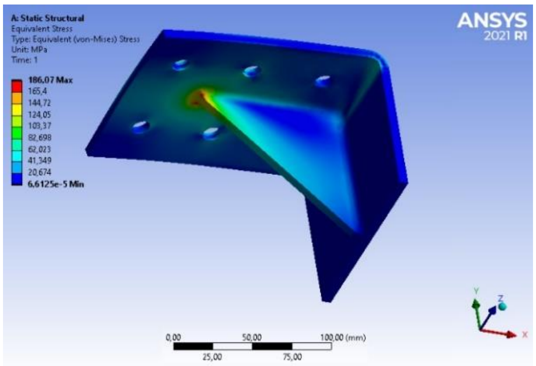

Fig. 3. Maximum von Mises equivalent stress values obtained through ANSYS.

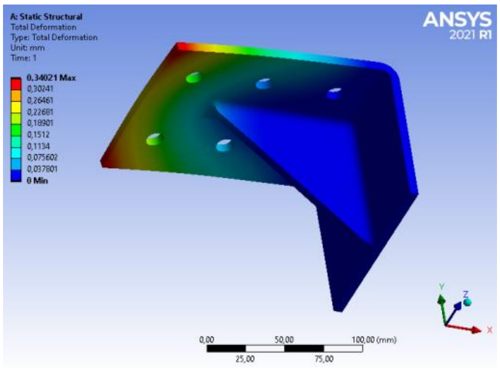

Fig. 4. Maximum displacement magnitude of the beam bracket obtained through ANSYS.

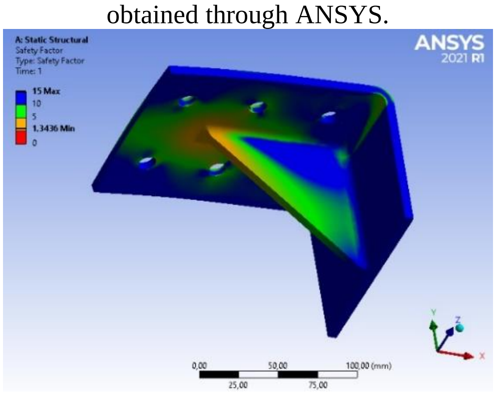

Fig. 5. Safety factor value calculated by ANSYS.

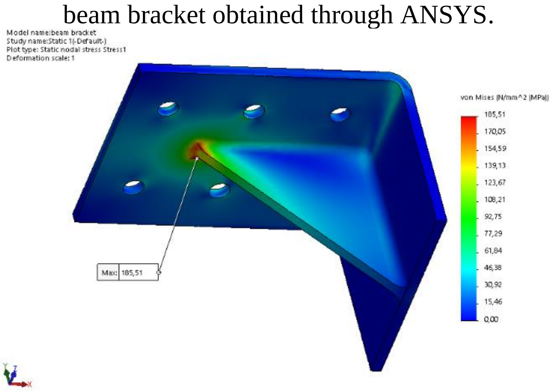

Fig. 6. Maximum von Mises equivalent stress values obtained through SW Sim.

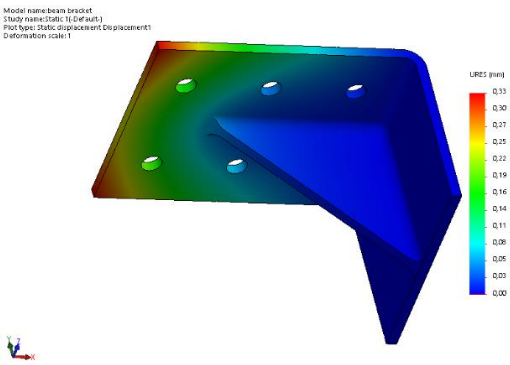

Fig. 7. Maximum displacement magnitude of the beam bracket obtained through SW Sim.

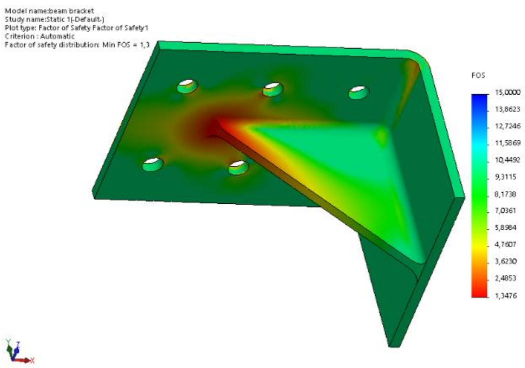

Fig. 8. Safety factor value calculated by SW Sim.

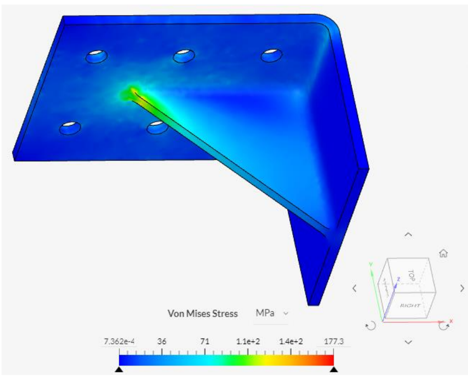

Fig. 9. Maximum von Mises equivalent stress values obtained through SimScale.

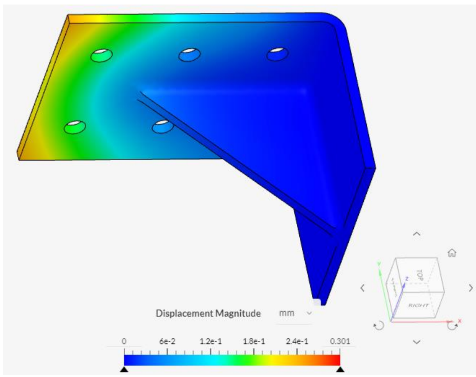

Fig. 10. Maximum displacement magnitude of the beam bracket obtained through SimScale.

A graphical comparison on the von Mises equivalent stress data and the displacement magnitude between these three software solutions can be found in Figure 11.

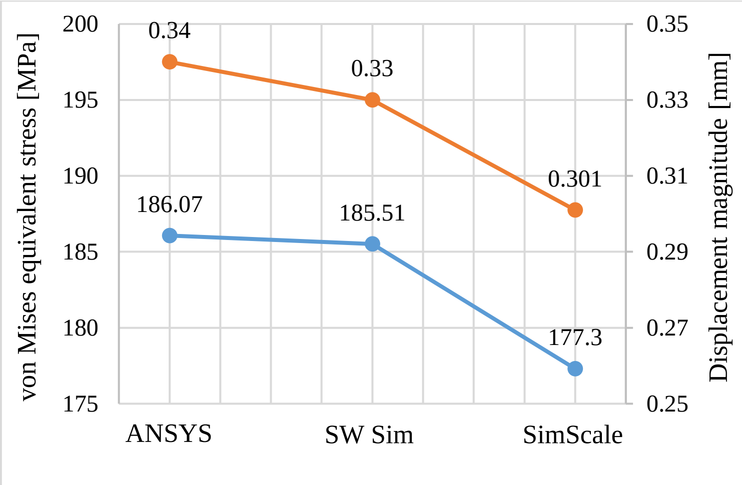

Fig. 11. Comparison on the von Mises equivalent stress and displacement magnitude between the software solutions.

4. CONCLUSIONS

Following the obtained results, it can be said that the software solution based on cloud computing performs well in terms of calculating the equivalent stress distribution allowing to obtain good data. These data, however, underestimate the mechanical behavior of the studied parts, but do not differ significantly from the established software solutions. In this case, the differences between the values obtained are less than 5 %.

The same cannot be said about estimating calculated displacements. In this case, the differences are slightly larger, approaching 11.5 %. This percentage can be dangerous for components important for the integrity of certain types of mechanical structures.

Regarding the calculated safety factor, of course, the estimation of the maximum equivalent stresses has a very important role. Therefore, in the case study of this paper, data obtained by the software solution based on cloud computing are acceptable. And in this case, the differences are below 5 %.

Analyzing all of the above, it can certainly be concluded that the use of software solutions based on cloud computing – in this case, SimScale – is recommended for parts with a rather low complexity, the emphasis being on estimating the equivalent stress distribution to the detriment of displacement assessment.

REFERENCES

[1] Daryl, L. L., A first course in the finite element method, Ed. Cengage Learning, USA, 2011.
[2] Reddy, J.N., An introduction to the finite element method (Third ed.), Ed. McGraw-Hill, USA, 2006.
[3] Kiritsis, D., Eemmanouilidis, Ch., Koronios, A., Mathew, J., Engineering asset management, Proceedings of the 4th World Congress on Engineering Asset Management (WCEAM), 2009, p. 591–592.
[4] Lee, H.H., Finite element simulation with ANSYS workbench 2019 – theory, applications, case studies, SDC publications, USA, 2019.
[5] Weber, M., Verma, G., SolidWorks simulation black book, Ed. CADCAMCAEWORKS, USA, 2016.
[6] König, P., Mit SimScale und make gratis simulieren lernen wie die profis, 2016, https://www.heise.de/-3175633 (21.03.2021).
[7] Digital Engineering 24/7, Enhancing cycling performance via simulation, 2016, https://www.digitalengineering247.com/article (21.03.2021).
[8] Suizi, J., Wanlin, C., Zibin, L., Wei, D., Yingnan, S., Experimental study on a prefabricated lightweight concrete-filled steel tubular framework composite slab structure subjected to reversed cyclic loading, Applied Sciences, 2019, vol. 9, no. 6, p. 1264.