THE USE OF AN OPEN SOURCE FEA SOLVER ON A STANDARD ENGINEERING PROBLEM

Tomáš Gajdoš[1], Petr Tomek[1]

Abstract	*The article deals with a FEA results comparison of an open source FEM software CODE_ASTER with a commercial software SolidWorks Simulation and Abaqus. The problem of a statically loaded spherical cap is solved. Result evaluation is aimed at loss of stability, i.e. limit load (equilibrium curve) when loss of stability occurs and its corresponding shape of deformation. The response of the structure is described by a load-deflection curve of a top of a spherical cap in axial direction.*
Keywords	*Abaqus, Code_Aster, FEA, FEM, Loss of stability, Salome_Meca, SolidWorks Simulation, Spherical cap*

1 INTRODUCTION

There is many well-known commercial FEM software available on the market that can solve many complex problems of engineering mechanics. Some of them are Abaqus and SolidWorks Simulation. SolidWorks Simulation is aimed at a user-friendly environment and it is primarily designated for solving common engineering problems. Because of that, it is easy to understand by beginners. A big advantage is the integration of the FEM module directly in a 3D parametric CAD SolidWorks. This makes the changes of a geometry of a numerical model very easy to do. Abaqus is able to solve more complex problems than SolidWorks Simulation, but a deeper knowledge of FEM is required. The free and mapped mesh is supported by Abaqus. SolidWorks Simulation supports only triangular shell elements (free mesh) for shells. Both of the mentioned FEM software have their pros and cons, and each of them can be more or less suitable for a different application.

In contrast, there is an area of open-source FEA software. These applications may seem a bit untrustworthy first, but there are open-source FEA solvers that can compete with a commercial one well. One of them is the FEA solver called Code_Aster.

Code_Aster is a finite element analysis solver developed by a Research and Development department of a French company Électricité De France (EDF). Its development began in 1989 and since 2001 is Code_Aster distributed under GNU GPL license. It offers tools to solve a number of static, dynamic, thermal and eigenvalue problems. There can be solved problems which contain nonlinear behaviour of material, geometric nonlinearity (large displacements) or contact. The beam, shell, or volume elements are supported. Code_Aster is integrated as a FEM module in the Salome environment, which aggregates it with preprocessing and postprocessing tools. The whole is then called Salome_Meca and it is distributed under GNU LGPL license.

---

[1] University of Pardubice, Department of mechanics, Materials and Machine Parts; Studentská 95, 532 10 Pardubice 2, Czech Republic

*Corresponding Author: Tomáš Gajdoš, E-mail: tomas.gajdos@student.upce.cz

In comparison with SolidWorks Simulation and Abaqus is the Code_Aster more similar to Abaqus. The biggest obstacle is maybe the fact that there are a French terms used in Code_Aster and also that the official English documentation is computer translated from French. Needless to say that the translation is at a very good level and, for a FEA educated person, it is mostly sufficient. Another problem may be that Salome_Meca is mainly developed for Linux operating systems.

Some of the papers where the Code_Aster was used are mentioned next. The paper (Strozzi and Lugli, 2020) examined the problem of low cycle fatigue. This paper deals with a problem of a thick cylindrical shell closed with a head. A penalty function was used for a contact between shell and a head and a static finite element analysis was solved. Another paper (Cazin et al., 2020) deals with a dynamically loaded demining tiller tool with forces initiated by blades hitting the ground. Solidworks was used for model creation in this paper, which was then exported to Salome-Meca. A frictional contact was used, while a nonlinear transient (time-history) analysis was solved by a Code_Aster. Another interesting use is mentioned in the paper (Antonutti et al., 2018) in which the dynamic behaviour of a floating wind turbine is solved. The use of an interesting CABLE element was demonstrated, which was also used in paper (Cheng et al., 2022) that deals with a dynamic response of the fish farming cage (net) to hydrodynamic forces. The paper (Peč et al., 2019) investigates the cyclic plasticity material models included in Code_Aster. The stresses induced by thermal conduction are examined in (Cherednichenko et al., 2020). A comparison of Code_Aster with CalculiX and Ansys on a problem of linear elasticity and contact is mentioned in paper (Park et al., 2018). In all mentioned papers were the results of simulations made in Code_Aster sufficient and in agreement with an experimental approach if there was one made. This fact makes Code_Aster even more interesting and catalyses the interest in his further examination of possible uses.

The main objective of this paper is to compare the results of the finite element analysis of a standard engineering problem from Code_Aster with results from SolidWorks Simulation and Abaqus. The comparison is focused on the ease of use of the selected FEM software and the correlation of the results.

2   DESCRIPTION OF A SOLVED PROBLEM

The problem of a statically loaded spherical cap with a low half-angle was solved. This problem is a very good example of the importance of a nonlinear simulation with geometrical nonlinearity (large displacements). The descripted spherical cap is nonstandard construction with a large initial bending state (nonlinear behaviour during loading). This problem cannot be solved by a linear buckling analysis LBA (linear loss of stability). A limit state of this type of construction is expected as a nonlinear collapse or as a nonlinear buckling. The limit load is expected to be significantly lower than the critical load from the linear buckling analysis.
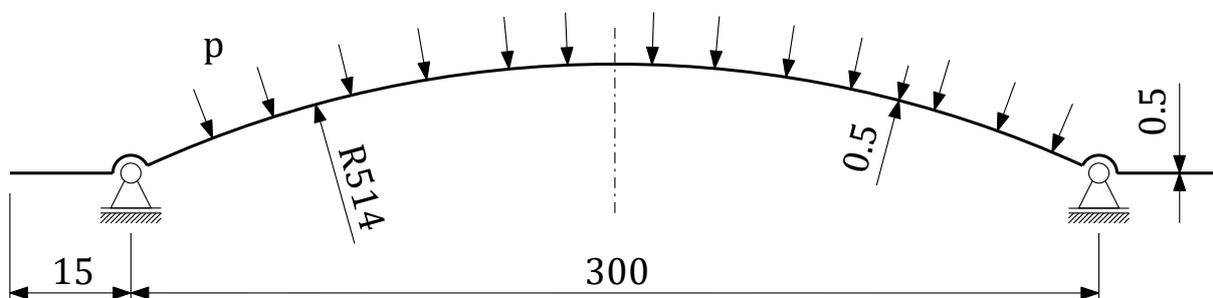
Fig. 1 Dimension of a considered spherical cap, units in millimetres

The dimensions and geometry of a numerical model are shown in Fig. 1. The boundary conditions were defined by zero normal displacements of the nodes on the outer ring of the spherical cap and zero tangential displacements of nodes of elements on the inner edge of this ring. A load was defined as

a pressure on the convex face of a spherical part of a cap (see Fig. 1). A nonlinear von Mises bilinear material model (linear hardening) was used. The material properties are shown in Tab. 1.

Tab. 1 Material model

Young's modulus	E	$2 \cdot 10^5$	MPa
Poisson's ratio	μ	0.3	–
Yield strength	$R_{p0.2}$	180	MPa
Tangent modulus	E_T	20	MPa

The numerical analyses were performed with a triangular (free) and quadrilateral (mapped) mesh. The results of the SolidWorks simulation were calculated only with a triangular mesh. In all cases, the material and geometric nonlinearities were used. The second order (parabolic) shell elements were used in all cases. An Arc-Length method was used to control the process of nonlinear analysis. In the case of SolidWorks, a direct sparse solver was used. In case of Abaqus and Code_Aster, the default solver was used.

3   RESULTS AND DISCUSSION

The resultant loading curves (load vs displacement of a chosen node) are shown in Fig. 2. The curves are plotted for a node in the centre of the spherical cap. The term *Tria* stands for a triangular mesh, and *Quad* stands for a quadrilateral mesh. A value on the horizontal axis represents a displacement in the z axis (axis of symmetry) of a cap. A negative value represents a displacement in the negative direction of the z axis (see Fig. 1). Fig. 2 shows that the nonlinear collapse with a snap-through occurs at first and is followed by a nonlinear buckling. This happens in all solved variants.
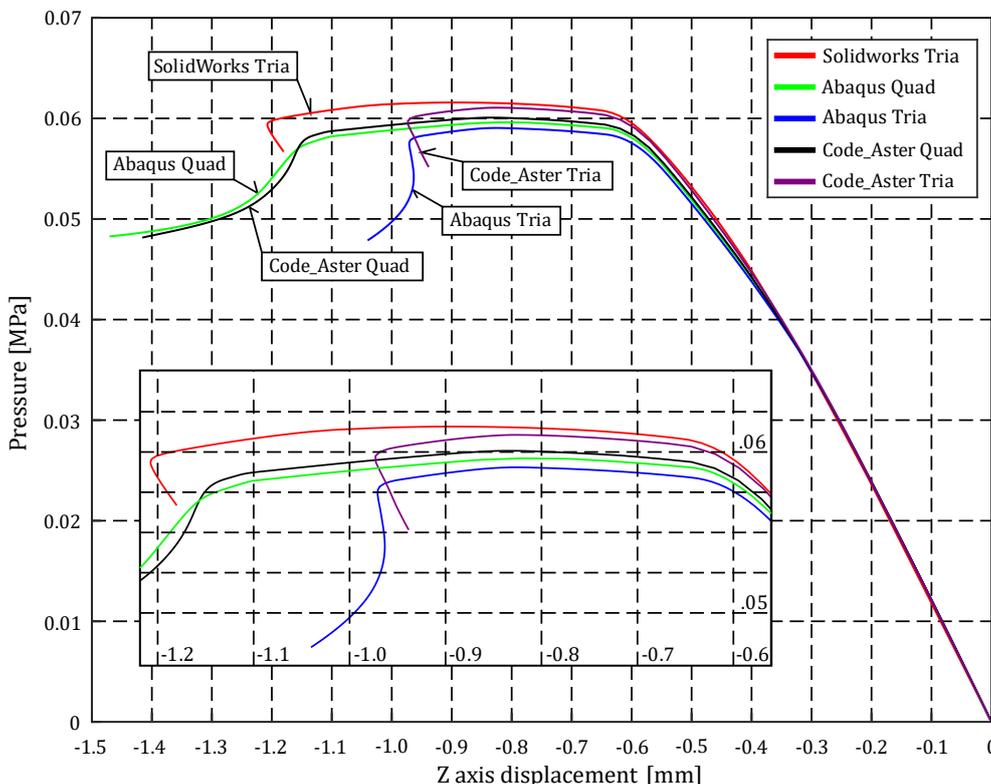
Fig. 2 Load-displacement curves

The limit loads and corresponding displacements in z axis are listed in Tab. 2 and Tab. 3. Table 2 shows the values for a nonlinear collapse and Tab. 3 for a nonlinear buckling. For the purpose of this article, the decimals were rounded to 4 digits.

Tab. 2 Limit loads and corresponding z axis displacements for a nonlinear collapse

	Triangular mesh	Quadrilateral mesh
SolidWorks Simulation	0.0616 MPa -0.8872 mm	—
Abaqus	0.0590 MPa -0.8271 mm	0.0596 MPa -0.8201 mm
Code_Aster	0.0611 MPa -0.8277 mm	0.0601 MPa -0.8347 mm

The results show that the limit load of a nonlinear collapse occurs approximately at 0.06 MPa. It may seem that the SolidWorks Simulation is the least conservative, and Abaqus with triangular mesh is the one that gives the most conservative limit load. The important fact is that the difference between values is not more than 5 %. The nonlinear collapse happens at lower displacements for Abaqus and Code_Aster than for SolidWorks Simulation, but this is also negligible for practical use. The same happens for the second limit state. An interesting fact is that even the use of a quadrilateral and triangular mesh does not have a significant influence on the numerical results.

Tab. 3 Limit loads and corresponding z axis displacements for a nonlinear buckling

	Triangular mesh	Quadrilateral mesh
SolidWorks Simulation	0.0596 MPa -1.2064 mm	—
Abaqus	0.0579 MPa -0.9686 mm	0.0580 MPa -1.1121 mm
Code_Aster	0.0598 MPa -0.9731 mm	0.0587 MPa -1.1063 mm

The analytical approach mentioned next. The method used is based on the procedure from ECCS (Rotter and Schmidt, 2008), but uses the coefficients mentioned in the monography (Paščenko et al., 2012). For the problem solved in this paper, the reference load is

$$p_{\mathrm{Rcr}} = 1.21 \, C_{\mathrm{cr}} \, E \left(\frac{t}{R}\right)^2 = 1.21 \cdot 0.414 \cdot 2 \cdot 10^5 \cdot \left(\frac{0.5}{514}\right)^2 \doteq 0.095 \text{ MPa} , \tag{1}$$

$$p_{\mathrm{Rpl}} = R_{\mathrm{p0.2}} \, C_{\mathrm{pl}} \left(\frac{2t}{R}\right) = 180 \cdot 0.179 \cdot \left(\frac{2 \cdot 0.5}{514}\right) \doteq 0.063 \text{ MPa} . \tag{2}$$

The characteristic pressure for buckling is then

$$p_{\mathrm{Rk}} = \chi \, p_{\mathrm{Rpl}} = 0.676 \cdot 0.063 = 0.043 \text{ MPa} . \tag{3}$$

Different deformed shapes were obtained for limit loads. Needless to say, that the deformed shape according to a limit state of loss of stability depends on many factors. These are, for example, mesh geometry, material properties, or boundary conditions. For a real part, there is also the occurrence of an initial imperfections which comes from manufacturing and can also highly affect the deformed shape.
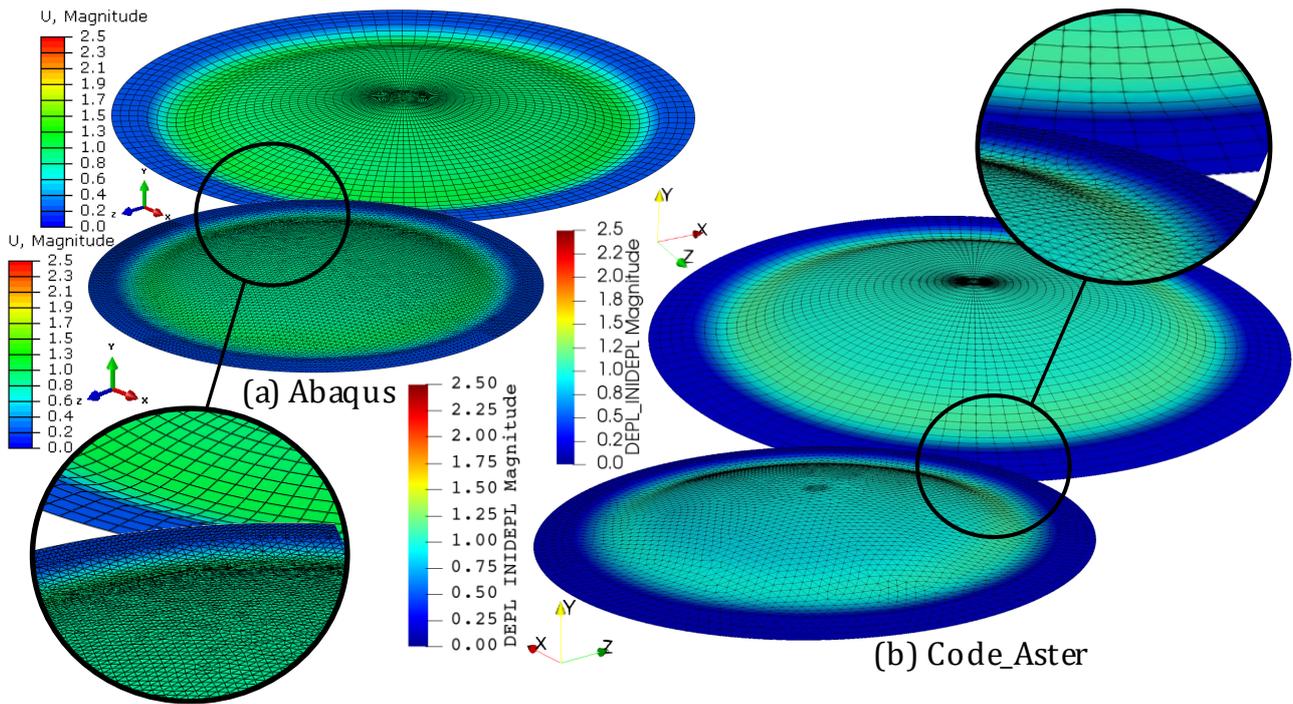
Fig. 3 Nonlinear collapse, deformed shape, scale factor 10

The deformed shapes that were obtained are shown in Figs. 3-5. These figures show the deformed model (displacements in z axis) after the limit state occurs. The scale factor 10 was used in all figures. The expected symmetric deformed shape can be seen in Figs. 3 and 4a, as usually happens at nonlinear collapse. The deformed shape after nonlinear buckling occurs is shown in Figs. 4b and 5.
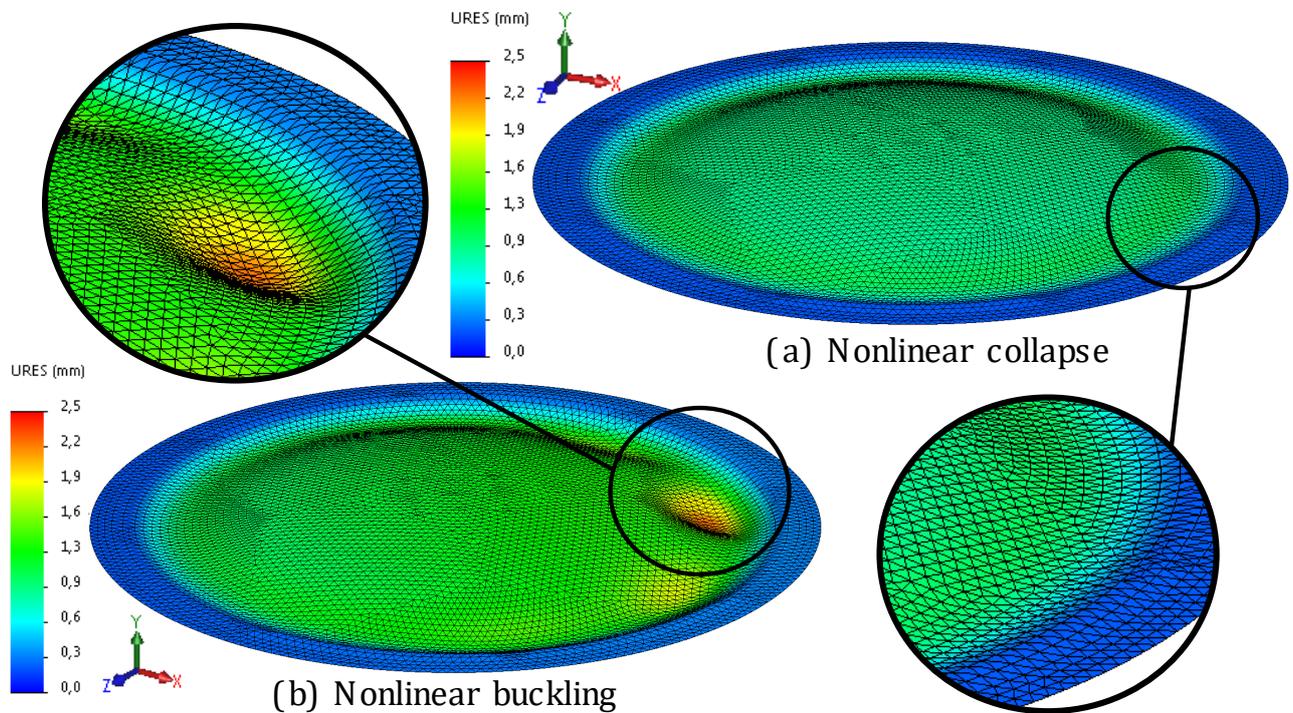
Fig. 4 SolidWorks results, deformed shape, scale factor 10

Nonlinear buckling demonstrates nicely how different parameters affect the final deformed shape. There can be seen that the triangular mesh gives the shape most close to reality and the quadrilateral mesh gives a completely different result. However, there can be seen a good correlation of results when the mesh of the same type is compared.
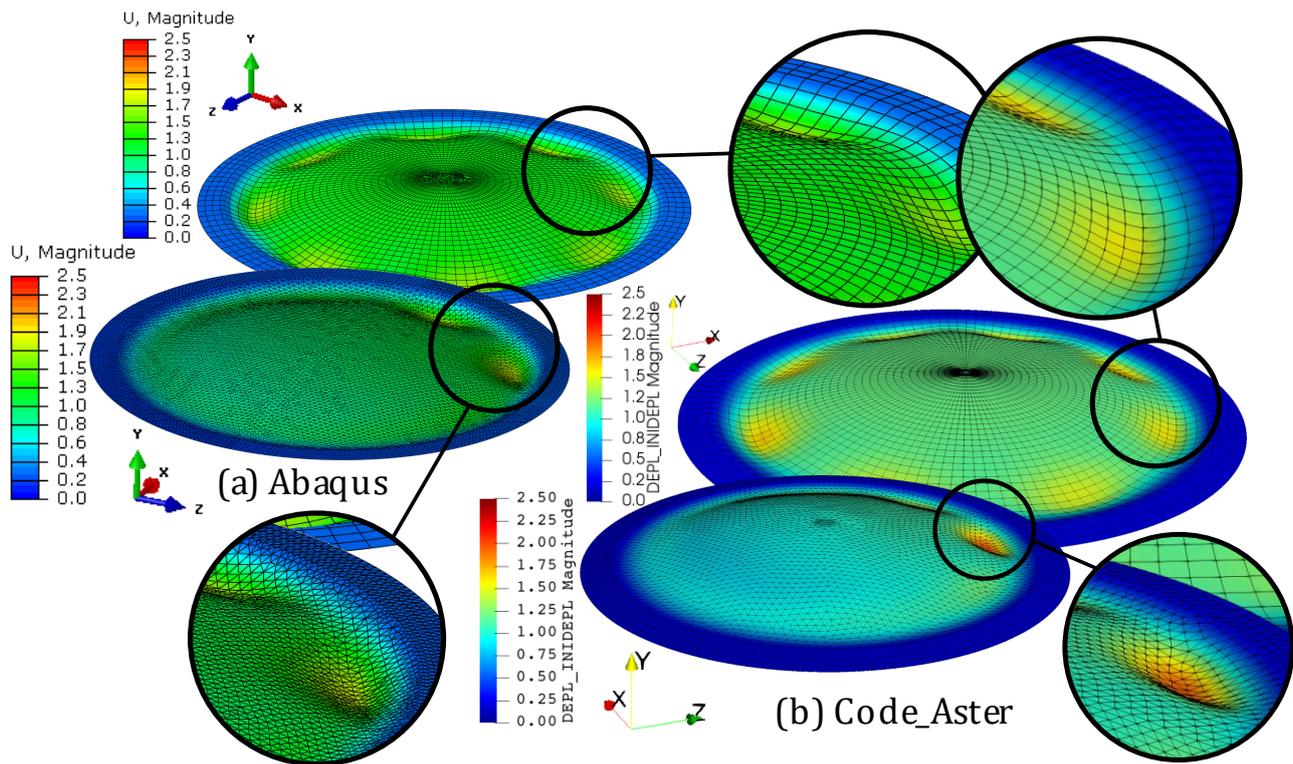
Fig. 5 Nonlinear buckling, deformed shape, scale factor 10

All examples took a similar time to solve. The overall time needed for preprocessing was the longest for Code_Aster and the least for SolidWorks Simulation. The preprocessing for Code_Aster was done in the Aster Study module, which offers most of the analysis parameters to choose of the tested applications. SolidWorks Simulation has the most user-friendly environment of tested applications, but the analysis parameters are least customizable. Abaqus falls somewhere in the middle, as it allows one to edit many analysis parameters and has a more easy-to-orient user environment than Salome_Meca.

The time required to solve the FEM analysis depends, among other things, on the number of steps used. This time can be significantly reduced by using the automatic stepping function. This option is offered in all of the tested solvers. There is more than one option in Code_Aster on how it can be done. In the problem solved in this paper, the interval cutting in half was used. This approach was combined with a manual step addition at the points where the analysis did not converge well. In SolidWorks Simulation and Abaqus, only the maximal and minimal step length adjustment with the increasement of the number of all steps was needed.

In the case of Code_Aster, there is a definition of many additional parameters necessary while shell elements are used. The mesh is needed to be parabolic (second order elements) and the function to add middle nodes is required to be called in a solver manually. The raw result is then shown with those middle nodes with zero displacements, so the known condensation of the middle node needs to also be done manually. This can be done by projecting the result on the mesh with elements without the middle node. This gives the required result without zero displacements in the middle of an element. Needless to say, that this needs to be done only for a field of displacements. The resultant stress field also requires an additional function to be called during simulation to obtain a separate result for the TOP and BOTTOM faces and the MIDDLE surface. An important piece of knowledge is that all of the steps mentioned can be included in a separate command file and then imported when needed. This reduces the time needed for preprocessing, so it can be almost compared with the preprocessing time in Abaqus.

Since all solvers come with different postprocessing environments, a bit of knowledge is needed to obtain the required results. There were no bigger obstacles in any of the used applications, as all of them are very well documented. The interesting part is the postprocessing in the Salome platform for Code_Aster which

happens in ParaView. This application uses an easy-to-understand concept of filters which are applied to obtain desired results.

The result of analytical approach (see Eqs. 1-3) shows a value of limit load about 28 % lower than the numerical approach gives. This can be sensed as an ordinary result as the analytical approach should be always more conservative than a numerical one. The reason can be found in the simplifications which are assumed for an analytical approach. Thus, it can be assumed that the values obtained for the limit load are in a good correlation with the experimental and analytical approach. The same result gives the comparison of a deformed shape with those obtained experimentally in the publication (Tomek, 2013). As was mentioned before, the real deformed shape should be similar to the shape obtained for a triangular mesh in this paper. Even if the deformed shapes for a different type of mesh differ, the limit load remains approximately the same. This is an important fact, thanks to which differences of a deformed shape can be assumed as a minor parameter.

4   CONCLUSIONS

This paper deals with a comparison of an open source FEM solver Code_Aster and its preprocessing and postprocessing environment with selected proprietary FEM applications. Comparison was performed on the problem of loss of stability of a spherical cap. This problem was solved with a shell mesh consisting of parabolic triangular (free mesh) and quadrilateral (mapped mesh) elements. The Arc-Length method was used to control the process of nonlinear analysis in all cases.

The use of preprocessing and postprocessing tools and the user environment of the used applications were compared. There is a big advantage of SolidWorks which, as the only one of the tested software, offers the parametric CAD for preprocessing. This feature makes the numerical model preparation and mainly its edition very easy to do and notably reduces the preprocessing time. However, all of the used applications support the use of imported formats like STEP or IGES which can be used for mesh creation and analysis parameters setup. This lowers the meaning of this advantage to the level where the question on use of separate application is mandatory or not. There is a nice user environment in SolidWorks Simulation that makes preprocessing consisting of the definition of material parameters, boundary conditions, load, and mesh creation very fast. However, there are not as many options as is offered in the Abaqus or Code_Aster. Needless to say, that for a standard engineering problems it is mostly sufficient.

Another interesting feature which is offered in Abaqus and Code_Aster is the possibility to change the analysis parameters manually in input file. This can be helpful if there are many analyses needed to be done, but they require, for example, different material properties. These can be then easily changed straight in the input (command) file by some batch processing application.

The way Code_Aster works gives, for example, the option to define a custom analysis step in different parts of an analysis. This allows the users to completely control the number of analysis steps in the parts of the analysis they are interested in and does not leave it fully on the automatic step function.

Post-processing was performed with the tools offered with a particular solver. All of the tools of the used solvers are documented very well and there were no bigger obstacles which would make some of them significantly worse or better.

Results obtained from all solvers are shown in figs. 2-5. Limit loads are listed in table 2 and 4. There can be seen that all results converge between solvers very well. All of the FEM software tested can be used to design a safe construction of the type which was solved in this paper.

References

Antonutti, R., Peyrard, C., Incecik, A., Ingram, D. and Johanning, L. **2018**. Dynamic mooring simulation with Code_Aster with application to a floating wind turbine. *Ocean Engineering*, 151, pp.366-377. doi:10.1016/j.oceaneng.2017.11.018

Aubry, Jean-Pierre. **2013**. *Beginning with Code_Aster*. Framasoft. ISBN 979-10-92674-03-3.

Bushnell D. **1985**. *Computerized buckling analysis of shells*. Martinus Nijhoff Publishers. Dordrecht. ISBN 90-247-3099-6.

Cazin, D., Braut, S., Božić, Ž. and Žigulić, R. **2020**. Low cycle fatigue life prediction of the demining tiller tool. *Engineering Failure Analysis*, 111. doi:10.1016/j.engfailanal.2020.104457

Cheng, H., Li, L. and Ong, M.C. **2022**. Comparative study of five commonly used gravity type fish cages under pure current conditions. *Ocean Engineering*, 250. doi:10.1016/j.oceaneng.2022.110977

Cherednichenko, A.V., Maksimova, E.A. and Savelyeva, I.Yu. **2020**. Calculation Of The Heat-Stressed State Of The Disk Using Free Software Code_aster. *IOP Conference Series: Materials Science and Engineering*, 747(1). doi:10.1088/1757-899X/747/1/012104

Cook, R., D. **1995**. *Finite element modelling for stress analysis*. New York: Wiley. ISBN 0471107743.

Electricíe de France. **2018**. *Great principles of operation of Code_Aster, U1.03.00*. Available at: https://www.code-aster.org/V2/doc/default/en/man_u/u1/u1.03.00.pdf [Accessed: 2022, April 1].

Electricíe de France. **2018** *Introduction to Code_Aster*, *U1.02.00*. Available at: https://www.code-aster.org/V2/doc/default/en/man_u/u1/u1.02.00.pdf [Accessed: 2022, April 1].

Gnu.org. *Licenses - GNU Project - Free Software Foundation*. [online] Available at: https://www.gnu.org/licenses/licenses.html [Accessed 26 May 2022].

Park, S.-K & Seo, D.-W & Jeong, H. & Kim, M. **2018**. Performance evaluation of open-source structural analysis solver, CalculiX and Code-Aster, for linear static and contact problems. *ICIC Express Letters*, 12(7), pp.655-662. doi:10.24507/icicel.12.07.655.

Paščenko, P., Tomek, P., Středová, D. **2012**. *Stabilita kulového vrchlíku*. Pardubice: Univerzita Pardubice. ISBN 978-80-7395-697-4.

Peč, M., Šebek, F., Zapletal, J., Petruška, J. and Hassan, T. **2019**. Automated calibration of advanced cyclic plasticity model parameters with sensitivity analysis for aluminium alloy 2024-T351. *Advances in Mechanical Engineering*, 11(3). doi:10.1177/1687814019829982

Rotter, J. M. (editor), Schmidt, H. (editor). **2008**. *Buckling of steel shells: European design recommendations*. 5th edition. Published by ECCS. ISBN 92-9147-000-92.

Salome Platform. *Salome Platform - The open-source platform for numerical simulation*. [online] Available at: https://www.salome-platform.org/ [Accessed 26 May 2022].

Strozzi, A., Lugli. R. **2020**. Design of a pressure vessel shell-bottom closure achieved via a segmented locking ring. *International Journal of Pressure Vessels and Piping*, 188. doi:10.1016/j.ijpvp.2020.104240

Tomek, P. **2013**. *Vliv počátečních imperfekcí na pevnost a stabilitu tenkostěnných skořepinových konstrukcí v dopravní technice*. Habilitation work. Univerzita Pardubice.

www.code-aster.org. *Code_Aster*. [online] Available at: https://www.code-aster.org/ [Accessed 26 May 2022].