XVI International Conference on Computational Plasticity. Fundamentals and Applications
COMPLAS 2021
E. Oñate, D. Peric, M. Chiumenti and Eduardo de Souza Neto (Eds)

A COMPARISON BETWEEN COMMERCIAL AND OPEN-SOURCE SOFTWARE FOR FINITE ELEMENT ANALYSIS OF ELASTO-PLASTIC BENDING

AHMED ABDELATY[+*], CHRISTIAN REISINGER[+], WOLFGANG KUNZE[§] AND CHRISTIAN ZEHETNER[+]

[+]University of Applied Sciences Upper Austria
Stelzhamerstraße 23, 4600 Wels, Austria
e-mail: christian.zehetner@fh-wels.at - web page: https://www.fh-ooe.at/

[§]Salvagnini Maschinenbau GmbH
Dr.-Guido-Salvagnini-Straße 1, 4482 Ennsdorf, Austria
e-mail:wolfgang.kunze@salvagninigroup.com - web page: http://www.salvagnini.at

Key words: Computational Plasticity, Elasto-plastic Bending, Finite Element Method, Abaqus, Salome-Meca

Abstract. Nowadays, simulation is becoming more and more important in industries. Here we consider a typical industrial application in the field of sheet metal bending. A high number of simulations is necessary during the development process to perform parameter studies and optimizations. On the other hand, simulation tools should be also available for the customers of these machines, e.g., to plan the production of very specific profiles. In such cases, the optimal process parameters only can be found by simulation.

Very important in this context are the license costs for commercial simulation software. Frequently, the simulations are not limited by computational power but by the number of available licenses, such that the duration for parameter studies is elongated. Also, with license costs it very expensive to provide a simulation platform to the customers. The presented case study has been carried out with the goal of comparing possible open source alternatives to expensive commercial Finite Element software.

Exemplarily, we consider the elasto-plastic bending of a cantilever, using the Johnson-Cook constitutive law. For this test case, a three dimensional Finite Element analysis is performed, comparing the results of open-source software (Salome-Meca) and a commercial counterpart (Abaqus). Different element types and mesh sizes are compared, the usability of both tools, and the computational time.

Considering the obvious price difference, both platforms show comparable results. Comparing the functionality of both programs, both are capable for modelling highly detailed and complex models for elasto-plastic material processing. However, for understanding the structure of the user interface of Salome-Meca is far more time consuming. Additionally, the performance of Salome-Meca on different operating systems is compared: Salome-Meca on Linux, Salome-Meca on Linux, installed in a virtual machine on Windows, and finally Salome-Meca on Windows. All in all, it turned out that depending on the specific application Salome-Meca can be a powerful alternative to Abaqus for the considered industrial application.

1 INTRODUCTION

In industries, simulation becomes more and more important for optimizing products, processes and systems. The present work refers to the industrial application of an automatic panel bender of Salvagnini Group [1]. In an already two decades lasting research project, a system of simulation models for the complete machine and process has been created, and the final outcome is a digital twin [2]. The latter is used for designing new machine types, for optimization, parameter studies, and also for an adaptive bending strategy.

During the long period of the research project, the number and size of simulations has grown continuously. For Finite Element computations of the nonlinear process involving plastic deformation of the sheet metal, contacts and large deformations, the commercial Finite Element software Abaqus [3] has mainly been used. Among others, advantages are the user-friendly interface, high reliability, and highly optimized numerical algorithms. On the other hand, high license costs are the main disadvantage. For research and development activities, these costs are acceptable. However, it becomes more and more important to do online simulations on the machine by the customer, e.g. for special bending applications and profile shapes. For this case, it would be a great advantage to have open source simulation tools without license costs.

More than ten years ago, the software Salome-Meca, [4], already has been used in this research project as open source tool for simple linear elastic Finite Element computations. Salome-Meca consists of Salome for pre-and postprocessing, and Code_Aster as Finite Element solver. It is also part of the CAELinux project, [5], which is a Ubuntu-based Linux distribution for scientific computation. In our research project, this software has been used to implement an automatic simulation framework for a tool analysis, controlled by a Pyhton script.

The goal of the present work is to find out if Salome-Meca would also be appropriate for simulating elasto-plastic bending processes in our research project. For this sake, a simple cantilever beam subject to a transverse pressure load is analyzed. The results of Salome-Meca and Abaqus are compared, considering elasto-plastic deformations based on the Johnson-Cook material model [6]. In this paper, first results of this analysis are shown, in which two simplifications have been done: First, a quasi-static case is taken into account, neglecting the influence of the strain rate. Secondly, a geometric linear analysis (linearized strains) is performed. The next planned step are to extend the models considering large strains and rate-sensitivity.

The paper is organized as follows: First, the simulation models are described, as well as the used material law. Then, the results obtained by Abaqus and Salome-Meca are compared for different configurations, considering linear elastic (Hooke) and elasto-plastic behavior (Johnson-Cook). It turns out that the selection of elements and numerical settings must be done carefully to get the same results. Finally, a very good agreement of Abaqus and Salome-Meca results is demonstrated for the chosen settings.

2 FINITE ELEMENT MODELING

The Finite Element Method (FEM), e.g. [7], is well established in industries for solving and optimizing different kinds of engineering design problems. There are many commercial and open source software tools for all fields of physics. The goal of this study is to test the usability of the open-source FEM software Salome-Meca for the analysis of industrial problems involving elasto-plasticity, and compare it with the commerical software Abaqus.

No doubt, Salome-Meca is a powerful software that is used by many professionals, given the diverse and complex applications a FEM software could be of help to. However, many companies and users might be skeptical when having to deal with a new software with a different layout and module topology. Also, the usability of open source tools frequently is less comfortable compared to commercial software, and often there is the doubt if the results are correct.

In this study several cases have been analysed to compare the results and performance of 3D Finite Element analyses on both programs. In a parameter study different kinds of elements, mesh sizes, load cases, and solver settings have been compared. It has turned out that the numerical settings have to be done very carefully to obtain comparable results.

2.1 Test case

In the following we consider a cantilever beam loaded by a transverse pressure as shown in Figure 1. At the left end, the beam is clamped by setting all displacements on the left face in the yz-plane to zero, $U1 = U2 = U3 = 0$. As load a homogeneously distributed pressure on the top face normal to the xz-plane is defined.

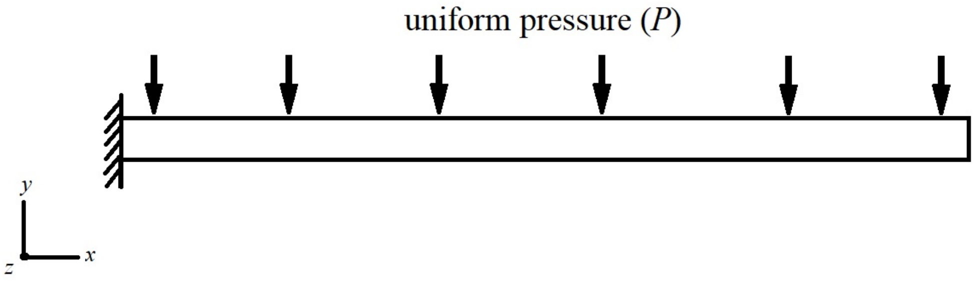

Figure 1: Boundary conditions for the bending test case

The magnitude of the pressure was varied in the range $0 \leq P \leq 1\ MPa$ with increments of 0.1 MPa for the nonlinear elasto-plastic simulations and $0 \leq P \leq 15\ MPa$ with 1 MPa increments for the linear elastic simulations. The simulation time is set to 1 s with a maximum time increment of 0.1 s and a minimum time increment of 10^{-6} s. For a first basic comparison of the two simulation programs, a small strain analysis is performed by setting *DEFORMATION='PETIT'* in Code_Aster and *nlgeom=NO* in Abaqus.

2.2 Material model

In the considered industrial application, elastic and plastic deformations play an important role. At the beginning of the bending process the strains are small and thus elastic. After reaching the yield limit, plastic strains occur. Finally, after removing the load, there is an elastic springback.

The elasto-plastic behavior of a material is usually determined by a uni-axial tensile test. For the sheet metals typically used on the panel bender, the elastic and plastic domain can be clearly determined. For small strains, the stress-strain diagram can be approximated by a linear function, yielding Hooke's law as

$$\sigma = E\,\varepsilon, \tag{1}$$

where σ is the true stress (Cauchy stress) and ε is the true strain (Hencky strain), and E is Young's modulus, [8, 9]. Hooke's law is valid for stresses lower than the yield stress σ_Y, i.e. $\sigma \leq \sigma_Y$.

After exceeding the yield stress, $\sigma > \sigma_Y$, materials show hardening behavior based on their lattice structure [9]. There are many ways to describe a flow curve [10]. Here, we use the Johnson-Cook plasticity model as it is very suitable for the metal sheets used in the industrial panel bender, as shown in [11]. According to [6] and [12] the Johnson-Cook hardening model can be expressed as

$$\sigma\left(\varepsilon_p, \dot{\varepsilon}_p\right) = \underbrace{\left(A + B\varepsilon_p^n\right)}_{\text{hardening}} \underbrace{\left[1 + C\ln\left(\frac{\dot{\varepsilon}_p}{\dot{\varepsilon}_{p,0}}\right)\right]}_{\text{strain rate}} \underbrace{\left[1 - \left(\frac{T - T_r}{T_m - T_r}\right)^m\right]}_{\text{thermal softening}}, \tag{2}$$

considering three effects: The first one is hardening under quasi-static conditions, i.e. for deformations rising so slowly that the strain rate does not play a role. The second term considers the rate sensitivity, i.e. an increasing strain rate causes an increase of hardening. In case of high strain rates thermal softening becomes relevant, represented by the third term: the material warms up so that there is a reduction of strength. In [11] these effects have been identified for a typical industrial sheet metal.

In Eq. (2) $\varepsilon_p = \varepsilon - \frac{\sigma}{E}$ is the plastic part of strain, $\dot{\varepsilon}_p = \frac{\varepsilon_p}{dt}$ is the plastic part of the strain rate $\dot{\varepsilon} = \frac{\varepsilon}{dt}$, $\dot{\varepsilon}_{p,0}$ is the quasi-static reference strain rate, and T is the temperature. Moreover, the following parameters in Eq. (2) must be identified by experiments: A is the quasi-static yield stress, B the strength coefficient, n the hardening exponent, C the rate-sensitivity, T_m the melting temperature, T_r the reference temperature, and m is a temperature coefficient. For the material DC01 which is typically used on the industrial panel bender, these parameters have been identified in [11].

In this paper the first step of our study is presented, only considering the quasi-static hardening: Setting $C = 0$ and $m = 0$ in Eq. (2) we obtain

$$\sigma = \ A + B\,\varepsilon_p^n, \tag{3}$$

which in the literature is also referred to as Ludwik-Hollomon law, [13, 14]. As shown in [11], it is a very good approximation for the considered cold-rolled mild steel.

2.3 Model parameters

The metal sheets processed on the industrial panel bender can be considered as plates. For our software comparison purposes, we can reduce the number of Finite Elements by modelling a beam. Thus, the lateral dimension is reduced compared to the sheets on the machine. The structure is defined as a homogeneous solid beam with a length of 20 mm, and a square cross-section with side length of 1 mm.

For the material parameters $E = 2.1 \cdot 10^6\ N/mm^2$, $A = 200\ N/mm^2$, $B = 600\ N/mm^2$ and $n = 0.19$ are used. The stress-strain diagram for these parameters is shown in Figure 2. Moreover, the Poisson ratio is set to $\nu = 0.3$. In the simulation model the material behavior is defined in two ways: first by specifying the Johnson-Cook parameters A, B and n, and secondly, the flow curve in Figure 2 is defined as a table.

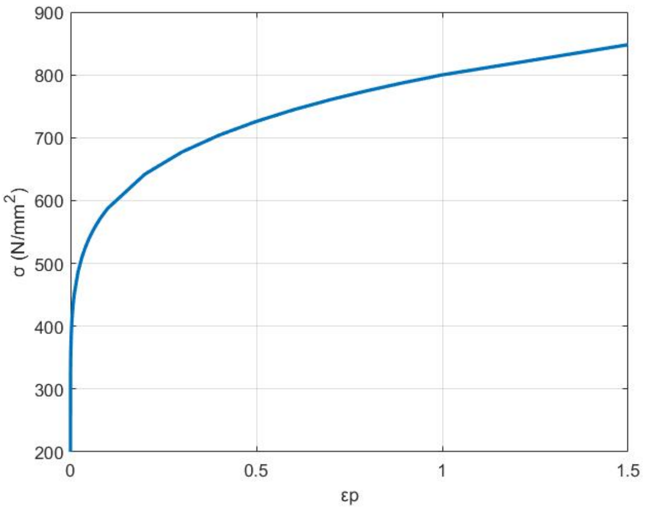

Figure 2: Stress strain curve of the chosen material based on Johnson-Cook parameters

2.4 Meshing

As well known, meshing has a high impact on the results of a Finite Element simulation. To find out the convergence behavior of the two considered software tools, different linear and quadratic element types and mesh sizes have been analysed, starting with 160 elements, and ending up with a fine mesh of 12800 elements.

In a similar bending analysis of Hemanth [15], the results of different element types were compared to an analytical analysis of James [16]. Their results showed that the fully integrated linear elements (C3D8) and the reduced integration linear elements (C3D8R) produced less accurate results on Abaqus, except with the enhanced hourglass control option.

In our first simulations using linear elements in Salome-Meca, we observed a similar trend. Finally, we compared 3D linear and quadratic fully integrated elements in Salome-Meca and Abaqus.

In Abaqus C3D8 are linear 3D hexahedral continuum elements with eight nodes and eight integration points. As a variant C3D8R uses reduced integration with only one integration point. Hexahedral continuum elements with quadratic trial functions are C3D20 with 20 nodes and 27 integration points, and accordingly C3D20R are elements with reduced integration.

Using Salome-Meca the beam is meshed with hexahedral elements in Salome. Then, in Code_Aster the element type is set to fully integrated elements by the keyword *Modelisation='3D'*, and to reduced integration by *Modelisation='3D_SI'*. Quadratic elements can be used by additionally using the keywords *CREA_MAILLAGE* and *LINE_QUAD*.

3 SIMULATION RESULTS

Initially the models used the preset linear elements in both simulation tools: C3D8R with reduced integration in Abaqus, and elements with full integration in Code_Aster, *Modelisation='3D'*. As expected, these results showed a quite high discrepancy. After using reduced integration elements also in Code_Aster, *Modelisation='3D_SI'*, the discrepancies were reduced, but still too high. To clarify, we started with unit test cases.

3.1 Element Type Comparison

We considered three different load cases for a single Finite Element to investigate the discrepancies between the two programs: a uni-axial displacement-driven tensile test, a tensile test case with pressure load, and a bending test case applying a transverse pressure. The element types considered for the unit tests were linear with reduced integration (C3D8R), linear fully integrated (C3D8) and quadratic fully integrated elements (C3D20). The unit test results for the element types are shown in Table 1.

Table 1: Element type unit test comparison for linear elastic material

Element Type	Test Case	VMIS stress results on Abaqus	VMIS stress results on Salome-Meca	% Difference
Linear (C3D8R)	Displacement-driven tensile test	20990.9	20203.6	3.75%
	Pressure load tensile test	149.837	136.5143	8.89%
	Bending test	0.998975	0.9229019	7.62%
Linear (C3D8)	Displacement-driven tensile test	23239.1	21752.06	6.4%
	Pressure load tensile test	160.44	141.3884	11.87%
	Bending test	1.92664	1.465674	23.93%
Quadratic (C3D20)	Displacement-driven tensile test	20940.9	20940.5	0.002%
	Pressure load tensile test	157.902	157.901	0.001%
	Bending test	2.30959	2.30958	0%

Because of the high deviations obtained by the linear elements, and the good coincidence of the quadratic ones, we continued our analysis with the latter.

3.2 Linear Elastic Bending

The linear elastic case was the simplest with regard to simulation effort and time. The maximum displacement occurs at nodes of the free beam end, and the maximum v.Mises stress at the clamped end. Accordingly, in Figures 3 and 5 the displacement is plot for a node at the free beam end, and the v.Mises (VMIS) stress for the integration point closest to the clamped end and the upper side of the beam, as marked in the contour plots in Figures 7 and 8.

The maximum displacement and the v.Mises (VMIS) stress plots shown in Figures 3 and 5 illustrate the linear behavior with the increase in load pressure. The simulations were run up to a load pressure of $15MPa$, and the results show very good coincidence.

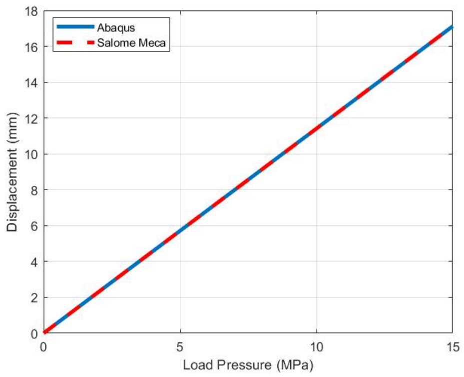

Figure 3: displacement, linear elastic case, Maximum displacement = $17.12mm$

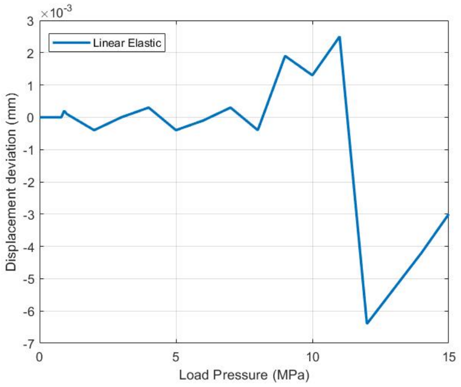

Figure 4: Absolute displacement deviation, linear elastic case, $-0.05\% \leq deviation \leq 0.03\%$

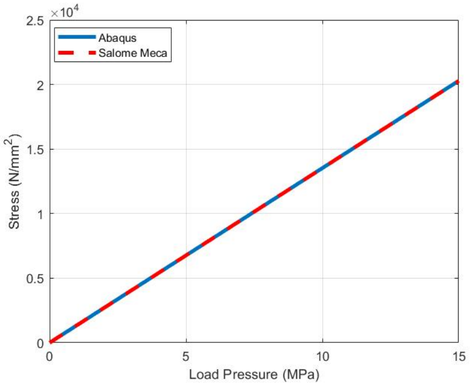

Figure 5: VMIS stress, linear elastic case, Maximum stress = $20280N/mm^2$

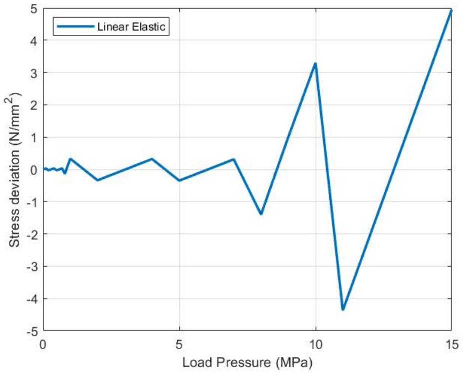

Figure 6: Absolute VMIS stress deviation, linear elastic case, $-0.03\% \leq deviation \leq 0.02\%$

The absolute deviation between Abaqus and Salome-Meca results shown in Figures 4 and 6 illustrate that the discrepancies are almost negligible, since the maximum deviation for the displacements is $-0.05\% \leq deviation \leq 0.03\%$, and for the VMIS stresses $-0.03\% \leq deviation \leq 0.02\%$.

The contour plots for both programs in Figures 7 and 8 are also comparable, if we consider that they are created in a different way: The results rendering on Code_Aster illustrates the stresses for every integration point in the contour plot. On the contrary, Abaqus only displays an average of the 27 integration points for each element. Thus, the contour plots are slightly different.

Figure 7: VMIS stress contour plot for linear elastic case on Code_Aster

Figure 8: VMIS stress contour plot for linear elastic case on Abaqus

3.3 Nonlinear Elasto-plastic Bending

For the nonlinear elasto-plastic bending case the material defined by the Johnson-Cook parameters was also simulated on both, Abaqus and Salome-Meca. The material law was defined in two ways: by the Johnson-Cook parameters A, B. n, and as a tabular input. The results show slight differences, also with respect to the computational performance, as discussed in the following.

The elasto-plastic simulations were run up to a load pressure of $1MPa$. Figure 9, exemplarily shows the deformed shape for a load pressure of $P = 0.6MPa$, yielding a maximum tip displacement of $1.66mm$.

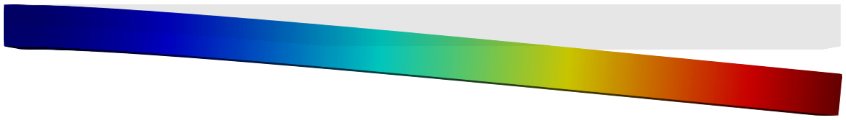

Figure 9: Displacement contour plot for nonlinear elasto-plastic case on Code_Aster

The maximum displacement at the free end is shown in Figure 10 as a fuction of the load pressure. The curve illustrates the linear relationship up to the yield point at $P = 0.2MPa$. Afterwards, the relationship becomes nonlinear because of plasticity. The displacement results show good coincidence: The absolute deviations between Abaqus and Code_Aster are shown in Figure 11. The relative deviations are in the range $-0.07\% \leq deviation \leq 0.03\%$.

In Figure 10 the difference between the tabular input versus the Johnson-Cook parameter definition is illustrated by the error bars. Comparing the differences of the two simulation programs in Figure 11, the simulation with tabular input seems to be more stable having no discrepancy up to $P = 0.5MPa$, while for the Johnson-Cook parameter definition the results seem to have more deviations. However, the deviations are very low.

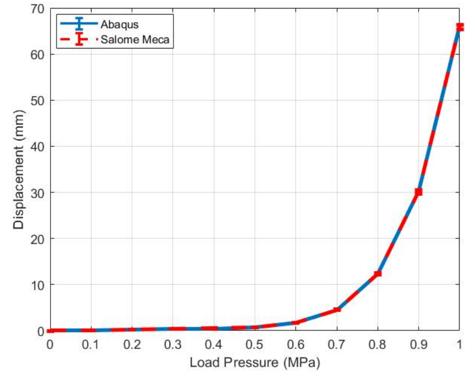

Figure 10: Displacement, nonlinear elasto-plastic case, Maximum displacement of 65.88 mm

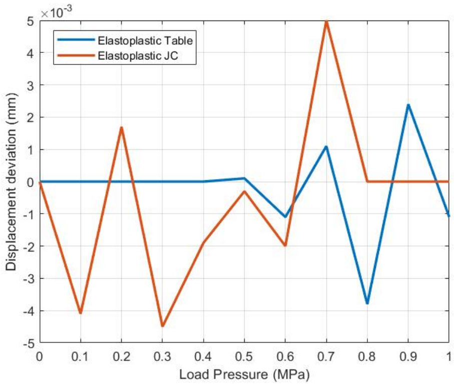

Figure 11: Absolute deviation, nonlinear elasto-plastic case, $-0.07\% \leq deviation \leq 0.03\%$.

Similarly the v.Mises (VMIS) stress plot in Figure 12 illustrates the linear behavior up to the yield point at $P = 0.2\ MPa$, and the nonlinear plastic behavior for $P > 0.2\ MPa$. For the VMIS stress the absolute deviations between Abaqus and Salome-Meca results, shown in Figure 13, for both material models show good coincidence. The VMIS stress deviation is in the range $-0.09\% \leq deviation \leq 0.29\%$.

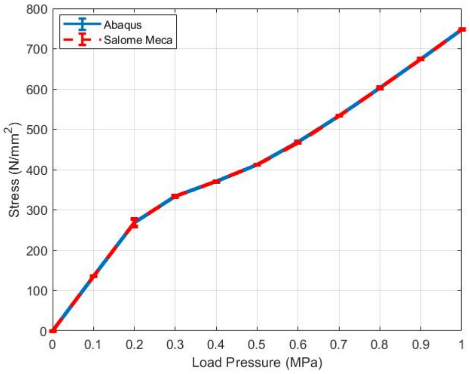

Figure 12: VMIS stress, nonlinear elasto-plastic case, Maximum stress = $747.9N/mm^2$

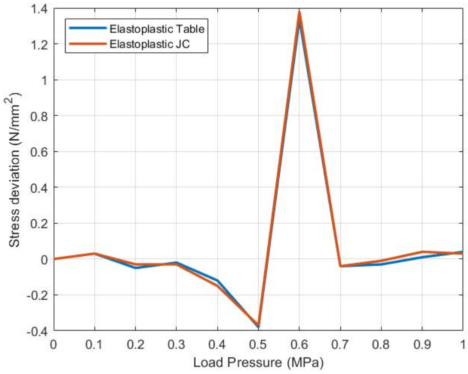

Figure 13: Absolute VMIS stress deviation, nonlinear elasto-plastic case, $-0.09\% \leq deviation \leq 0.29\%$.

As for the linear elastic contour plots, the nonlinear elasto-plastic contour plots are also comparable with slight differences, if we consider the different rendering of the two programs. The contour plots for the Johnson cook material model on Code_Aster and Abaqus are shown in Figures 14 and 15.

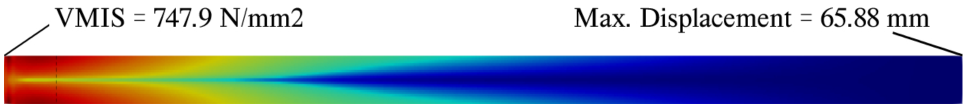

Figure 14: VMIS stress contour plot for nonlinear elasto-plastic case on Code_Aster

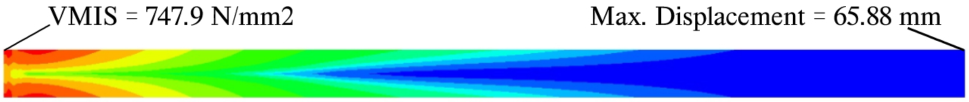

Figure 15: VMIS stress contour plot for nonlinear elasto-plastic case on Abaqus

4 USABILTY REVIEW

The first impressions using both, Abaqus and Salome-Meca, can be summarized as follows: On a first look Salome-Meca is composed of multiple modules, for geometry, meshing and Code_Aster for analysis, which can be used as a part of Salome-Meca or as standalone module. The layout and user interfaces seem to be less interactive compared to Abaqus. Moreover, the support material for Salome-Meca is strictly through their user's manual and Code_Aster documentation. On the other hand, Abaqus has a similar extensive user manual and documentation, although the interface is more integrated and self-explanatory for beginners and intermediate users. Abaqus is the more common software with more community support in terms of professional studies, software tutorials and examples. On the other hand, the community forum for Code_Aster was quite beneficial and the test case assistant templates help ease in beginners getting accustomed to the software. Since Salome-Meca is Linux based, it benefits from running on lighter operating system compared to Windows and also runs fine on a virtual machine. Error troubleshooting was not a problem in both programs. However, on Code_Aster it requires a certain level of know-how to be able to tackle some simulation errors.

Salome-Meca was originally designed for Linux. It is also possible to run it under Linux in a virtual machine on Windows. Nowadays, also versions for Windows are available. In our study, we compared the performance of the two simulation programs for the following platform configurations:

- Abaqus 2019 on Windows 10
- Salome-Meca 2019.0.1 running on CAELinux 2020 (Ubuntu 18.04) installed on a virtual machine (VMWare 16) under Windows 10.
- Salome-Meca 2019.0.1, running on CAELinux 2020 (Ubuntu 18.04) installed as native operating system.
- Salome-Meca 2019.0.3, running on Windows 10.

The comparison of the computational performance with these configurations is shown in Figure 16. In all test cases Abaqus is considerably faster than Salome-Meca. Considering the computation time difference, Salome-Meca still performs adequately on all platforms tested. An interesting result is the better performance on the windows machine and the Linux virtual machine on Windows compared to the native Linux machine. As expected, the best running stability was obtained on the native Linux machine.

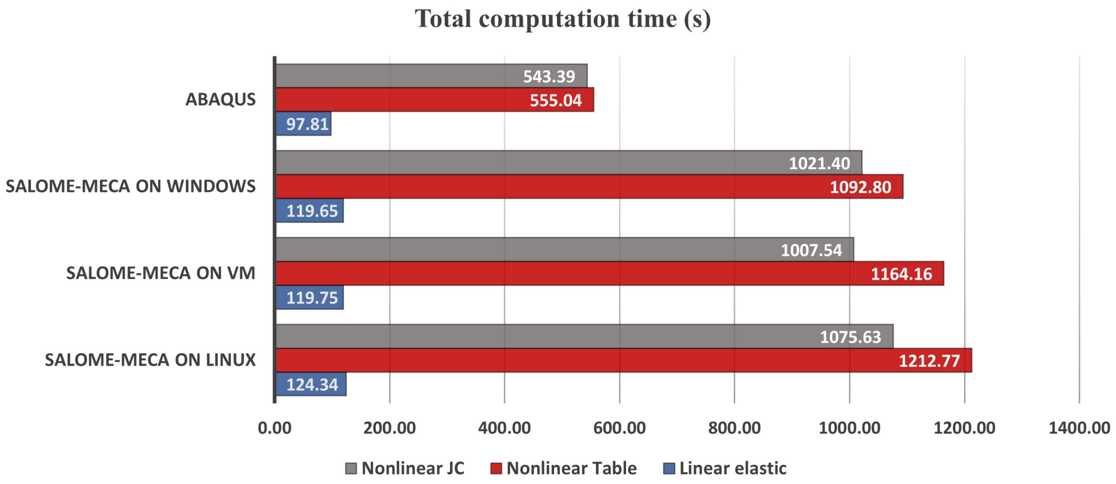

Figure 16: Computation time comparison between Abaqus and Salome-Meca on Windows, Linux and Virtual Machine

5 CONCLUSION

Both, Abaqus and Salome-Meca are well established professional programs capable of complex Finite Element analyses. For our test case of plastic bending, the results are comparable, but the simulation time is higher in Salome-Meca. Comparing different platforms, it turned out that Salome-Meca works well on all of them, with only very slight differences of the computation time.

Great advantage of Salome-Meca are the license costs: Since the prices for commercial software like Abaqus are quite high, Salome-Meca is an interesting alternative, especially for research or small companies. For our considered industrial example of the panel bender, Salome-Meca definitely is an interesting alternative for Abaqus.

However, several questions have to be answered in subsequent studies, as already has been started: First of all the high discrepancies of the linear elements should be investigated in more detail. Moreover, further effects are planned to be considered: large strains, higher deformations, and the influence of strain rate and temperature.

ACKNOWLEDGEMENT

This work has been supported by the COMET-K2 “Center for Symbiotic Mechatronics” of the Linz Center of Mechatronics (LCM) funded by the Austrian federal government and the federal state of Upper Austria.

REFERENCES

[1] Salvagnini Maschinenbau GmbH, www.salvagninigroup.com

[2] Zehetner, C., Reisinger, C., Kunze, W., Hammelmüller, F., Eder, R., Holl H. and Irschik, H. High-quality sheet metall production using a model based adaptive approach. *Procedia Computer Science* (2021), **180**:249-258.

[3] Abaqus User Manual, Simulia User Assistance 2019, Dassault Systemes (2019).

[4] Code_Aster and Salome-Meca, www.code-aster.org

[5] CAELinux, www.caelinux.com

[6] Johnson, G.R. and Cook, W.H. Fracture characteristics of three metals subjected to various strains, strain rates, temperatures and pressures. *Engineering Fracture Mechanics* (1985), **21**:31-48.

[7] Bathe, K.J. *Finite Element Procedures*, Prentice Hall (2014).

[8] Hearn, E.J. *Mechanics of Materials 1.* Butterworth Heinemann, (1997), Chapter 14:361-400.

[9] Chen, W.F. and Han, D.J. *Plasticity for Structural Engineers*, Springer, (1988).

[10] Prakash, M. and Dixit, U.S. *Plasticity. Fundamentals and application.* Butterworth Heinemann, CRC Press, (2015).

[11] Katzmayr, F., Reisinger, C. , Gross, T. , Sieberer, S., Kunze, W., Wagner, L. and C. Zehetner. Material models for highly dynamic metal forming processes. *Proceedings of the XVI International Conference on Computational Plasticity. Fundamentals and Applications*, COMPLAS, (2021).

[12] Sobolev, A.V. and Radchenko, M.V. Use of Johnson–Cook plasticity model for numerical simulations of the SNF shipping cask drop tests. *Nuclear Energy and Technology.* (2016) **2**:272-276.

[13] Ludwik, P. *Elemente der Technologischen Mechanik*, Springer Verlag, (1909).

[14] Hollomon, J.H. Tensile deformations. *Trans. Metall. Soc.* (1945), AIME 162: 268-290.

[15] Hemanth R.H., Ruchin P., Gourav G., Venkatesha K.S., Ravi Kumar G.V.V., Sklyut H., Kulak M., Heinimann M. Performance Evaluation of Finite Elements for Analysis of Advanced Hybrid Laminates; *Proceedings of the ABAQUS User's Conference*, (2010)

[16] James, M.G. and Timoshenko, S.P. *Mechanics of Materials.* PWS Publisher, (1984).