17th International Research/Expert Conference
”Trends in the Development of Machinery and Associated Technology”
TMT 2013, Istanbul, Turkey, 10-11 September 2013

NUMERICAL STRESS ANALYSE OF WELDED JOINTS ON PRESSURE VESSELS

Adil Muminović, Isad Šarić, Vahid Avdić
Faculty of Mechanical Engineering
Sarajevo, Bosnia and Herzegovina

ABSTRACT
The main goal of this paper is to give an example of how numerical FEM simulation in combination with parametric design features can be used very effectively in estimating stress in welded joints on pressure vessels. In order to achieve this goal, authors choose a specific pressure vessel design for compressed air in accordance with specialised literature. A complex analytical calculation was conducted followed by a numerical simulation in the CATIA V5 software. All parts of the pressure vessel were designed with the parametric design feature that the CATIA V5 software offers. In this way, quick design changes where possible. Authors choose 3 different versions of the same pressure vessel where the geometry was modified to find the best possible solution. Also, welded joints were simulated as a single part using the more common basic workbenches of the CATIA V5 software environment instead of specialised welding workbenches which cost extra. Finally, a summarize in form of a diagram is given where analytical and numerical results are compared. The paper concludes with a analyze of analytical and numerical results by estimating the error from the numerical simulation and giving guidlines for further investigation.
Keywords: Pressure vessels, welded joints, numerical stress analyse, CATIA V5 system

1. INTRODUCTION
Pressure vessels represent a inevitable element in modern thermal, hydro, chemical or water supply systems. They primarily serve as a tank for different fluids such as liquids, gases or mixtures of powder materials. These pressure vessels are closed type containers whereby the inner pressure is much higher then the surrounding atmospheric pressure. This pressure difference causes a very dangerous threat to the surrounding area where the pressure vessel is placed. Because of this fact, pressure vessels must be designed very strictly according to engineering regulations. Considering the specific field of use there are few pressure vessel types but they mainly share all one basic constructive layout which consists of several cylindrical segments, two heads, an inlet valve and an outlet valve. After on, all mentioned parts are welded together (Figure 1).

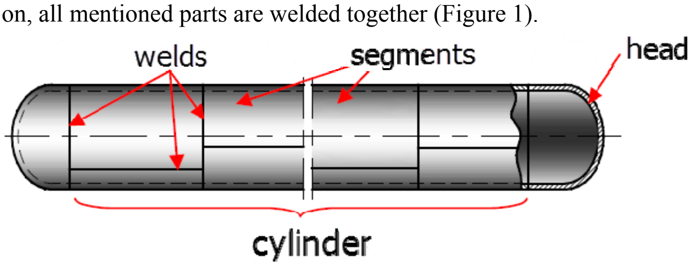
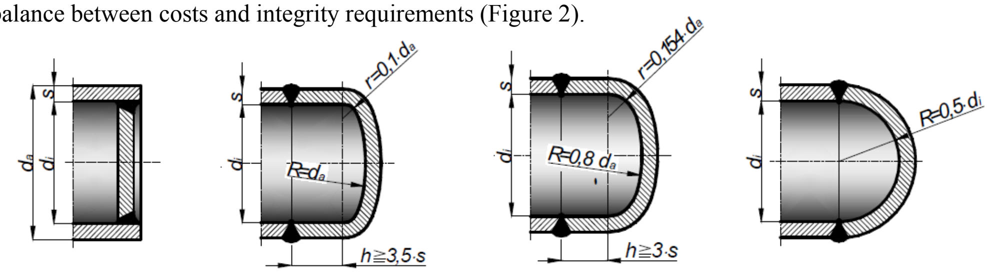


Figure 1. Basic pressure vessel layout (inlet and outlet valve are not shown)

The segments, which can take on different shapes (not just cylindrical), represent the main body of the vessel and withstand the greatest part of the inner pressure. After they are welded together they share a common axis and can adopt several shapes like cylindrical, spherical or conic. The second most important part of a pressure vessel is the head. From all possible shapes, the spherical is the optimal one. The choice of spherical shape for the head becomes more obvious if the vessel withstands a specifically high pressure. However, the manufacturing process of such shapes is very expensive in comparison to less complex forms which in turn cannot withstand very high pressures. It is always a balance between costs and integrity requirements (Figure 2).

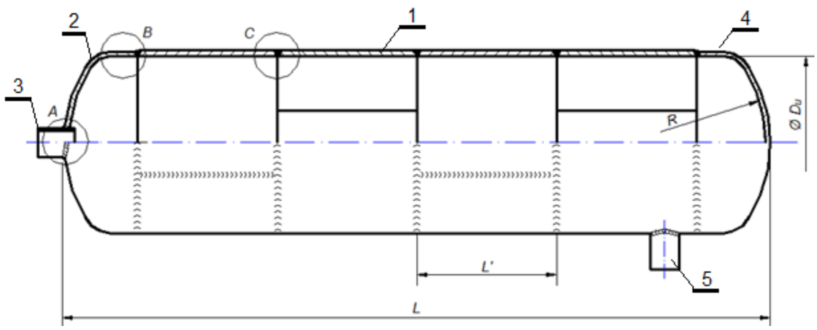


a) b) c) d)

Figure 2. Different types of pressure vessel heads a) Flanged head; b) Shallow head; c) Deep head; d) Hemispherical head

2. ANALYTICAL STRESS CALCULATION OF PRESSURE VESSEL WITH DEEP HEADS

In order to have a valuable reference which can be used later on to compare results with the numerical simulation, an analytical calculation was conducted first. According to the chosen pressure vessel for compressed air (Figure 3) a comprehensive project calculation was conducted. It included the determination of vessel dimensions and weld stress for three chosen versions of pressure vessel. The pressure was the same in all three cases and was set to 1,4 MPa.

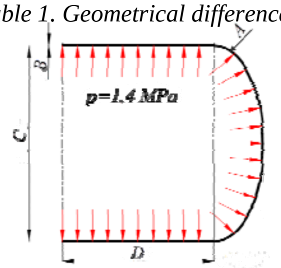


1 – Segment
2 – Left Head
3 – Input Valve
4 – Rigth Head
5 – Output Valve

Figure 3. Example of the analyzed pressure vessel for compressed air

The geometrical difference between the chosen three version of pressure vessels is given in Table 1. The analytical calculation was conducted according to documentation given in [1].

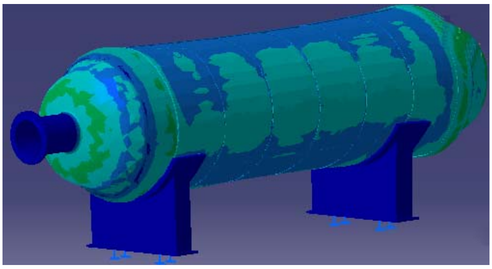
Table 1. Geometrical differences between three observed pressure vessel versions with attached sketch



	Version 1	Version 2	Version 3
Head Thikness A (mm)	13	10	7
Segment Thikness B (mm)	17	13	10
Inner diameter C (mm)	1300	1000	700
Segment width D (mm)	574	624	549

3. NUMERICAL ANALYSE

The numerical simulation was conducted in the CATIA V5 software. All parts where modeled with parametric design features so that they could be very easily modified according to Table 1. Since the mesh generation is a crucial element of every FEM analyze, building a very fine mesh was especially important for getting relevant results. The CATIA V5 software enables one to choose between two types of different Finite Elements: Linear and Parabolic, whereby the Absolute sag parameter controls the final fineness of the mesh. For this kind of calculation a Parabolic type of Finite Element was chosen whilst the Absolute sag parameter was set to 7,218 mm. It is important to point out that the welded joints where simulated by assigning them a less more qualitative material then that one from the segments and heads. Visual results are shown on Figure 4.



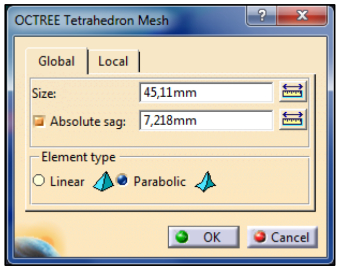


Figure 4. Example of the analyzed pressure vessel for compressed air

A detailed preview with different kinds of visualization methods can be exported later on. For the sake of this paper a general preview for the different types of stress is given in Table 2. Obtained results indicate that there are no evitable errors in the post processor stage of the CATIA V5 solver which can occur. All values are relatively close to each other and do slightly vary according to the chosen version of the pressure vessel. According to Table 2 the best possible solution regarding low weld stresses would be version 2. Nevertheless, for a final verification of the gathered data via numerical simulation a comparison between analytical and numerical results must be conducted to ensure correct conclusion. Hereof a set of three analytical calculation was performed for each version 1, 2 and 3. Results are compared and final conclusions are made in Table 2.

4. DISCUSSION AND CONCLUSION

The main goal of this paper was to give an example of how numerical FEM simulation in combination with parametric design feature can be used very effectively in designing pressure vessels. Furthermore using the parametric design feature, basic stress calculations for different geometrical definitions can be achieved within a shorter period of time. The second goal of this paper was to visualize the possibility that stress of welded joints don't has to be evaluated in special workbenches of the CATIA V5 environment such as the Welding workbench. Therefore the whole problem was designed in common workbenches whereby the welds were simulated by a distinct part with a less qualitative material. The justification of such an approach is given in Table 2, where the numerical and analytical results are compared.

Table 2. Comparison of analytical and numerical results

		Version 1	Version 2	Version 3
Analytical calculation	Weld between head and segment (MPa)	34,65	34,65	34,65
	Radial segment weld (MPa)	26,42	26,58	24,15
	Longitudinal segment weld (MPa)	56,86	56,16	50,85
Numerical calculation	Weld between head and segment (MPa)	34,86	32,17	31,96
	Radial segment weld (MPa)	27,16	25,81	20,52
	Longitudinal segment weld (MPa)	60,09	60,07	55,01

For better visualisation the content from Table 2 is visually presented in diagram 1. According to diagram 1 differences between analytical nad numerical results for all three versions of pressure vessels are minimal and can therefore be neglected. This fact confirms the previously mentioned claim that more common basic workbenches in the CATIA V5 software can be used for quick stress analyze of welded joints instead of specialised welding workbenches which cost extra.

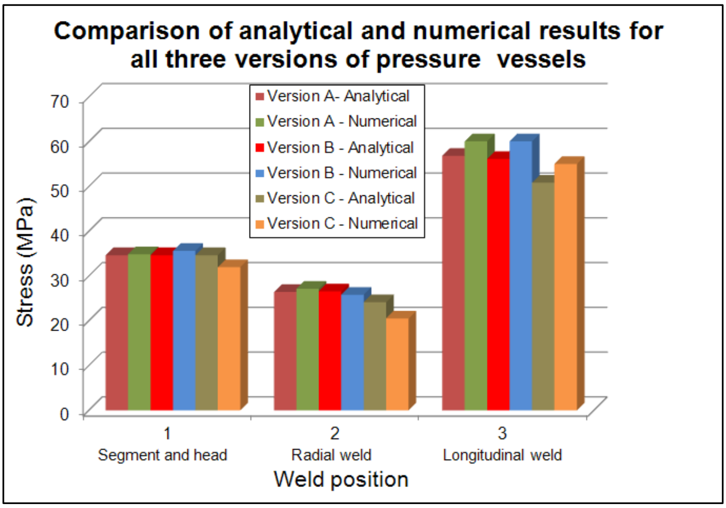


Diagram 1. Comparison of analytical and numerical results for all three versions of pressure vessels

The difference between analytical and numerical calculations is about 5 to 8 percent with one outlier of approximately 15 %. These results are absolutely satisfying if the chosen method of approximation is taken into account. Further investigation guidelines could be to choose different types of Finite elements or to test the same three versions of pressure vessels with different software's by choosing different Finite element types that these software offer, fragment the mesh more and then measure the time the solver needs to calculate a certain type of Finite element in combination with the mesh density.

6. REFERENCES

[1] Repcic, N. and A. Muminovic. Mechanical Elements I. Faculty of Mechanical Engineering Sarajevo, Sarajevo, 2013.

[2] Muminovic, A., I. Saric and E. Mesic. Computer-Aided Design (CAD). Faculty of Mechanical Engineering Sarajevo, Sarajevo, 2012. ISBN 978-9958-601-41-5, COBISS.BH-ID 19995654

[3] Isad Saric, Nedzad Repcic and Adil Muminovic (2012). Mechanical Transmissions Parameter Modelling, Mechanical Engineering, Dr. Murat Gokcek (Ed.), ISBN: 978-953-51-0505-3, COBISS.BH-ID 20189958, InTech

[4] Zamani, N.G. CATIA V5 FEA Tutorials Release 19. University of Windsor, Schroff Development Corporation (SDC Publications), 2010. ISBN 978-1-58503-586-1

[5] Šarić, I., N. Repčić and A. Muminović. „Parameter Modelling of Gears". Proceedings of the 14th International Research/Expert Conference „Trends in the Development of Machinery and Associated Technology" TMT 2010, Mediterranean Cruise, 11-18 September 2010, pp. 557-560. ISSN 1840-4944

[6] Saric I., N. Repcic and A. Muminovic. „3D Geometric parameter modelling of belt transmissions and transmissions gear". Technics Technologies Education Management – TTEM, Vol. 4, No. 2, 2009. pp. 181-188. ISSN 1840-1503

[7] Šarić, I., A. Muminović and N. Repčić. „Geometric Modelling and Stress Analysis of Model Device for Torsion". Proceedings of the 6th International Scientific Conference on Production Engineering „Development and Modernization of Production – RIM 2007", Plitvice Lakes, Croatia, October 24th-26th 2007, pp. 143-144. ISBN 978-9958-9262-1-1 (Full article on additional CD)

[8] Muminović, A., I. Šarić and N. Repčić. „Stress Analysis of Model Device for Torsion". Proceedings of the 11th International Research/Expert Conference „Trends in the Development of Machinery and Associated Technology" TMT 2007, Hammamet, Tunisia, 5-9 September 2007, pp. 731-734. ISBN 978-9958-617-34-8