Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 192 (2017) 609 – 614

Procedia Engineering

www.elsevier.com/locate/procedia

TRANSCOM 2017: International scientific conference on sustainable, modern and safe transport

CFD simulation of hydraulic tank

Martin Močilan[a]*, Milan Žmindák[a], Peter Pecháč[a], Peter Weis[a]

[a]Univerzity of Zilina, Faculty of Mechanical Engineering, Univerzitná 1, 010 26 Žilina , Slovak Republic

Abstract

The aim of this paper is the examination of fluid dynamics in a tank. The use of modern CAD and CFD techniques in the conception and simulation of industrial products has huge applications in the mechanical, automotive and aerospace industries. This paper includes all the steps from treatment of CAD geometry up to the analysis of simulation results. The presented approach involved CAD simplification, meshing of the geometry, CFD simulation and analysis of the simulation results.
A case study of a hydraulic tank partially filled with hydraulic oil was simulated in this paper using Volume of Fluid (VOF) multiphase model. Simulations compared the amplitude of sloshing in tank. In this paper, a part of the project which aims to develop a computer aided methodology for developing/designing of the fuel tanks based on static and dynamic analysis is presented.

Peer-review under responsibility of the scientific committee of TRANSCOM 2017: International scientific conference on sustainable, modern and safe transport

Keywords: CFD ; Fuel Tank ; Sloshing ; Fluid-Structure Interaction

1. Introduction

Introduction sloshing refers to the motion of free liquid surface inside a tank. To estimate the sloshing effects acting on the container, an accurate modeling of the free-surface waves is necessary [1].

Over the past decades, it has been recognized that it is important to take the nonlinearity of sloshing waves into account. In terms of the free-surface non-linearity, one major difficulty is that the fully-nonlinear boundary conditions have to be satisfied on the free surface not known a priori [1].

It seems that the best options for solving the fully-nonlinear problem are numerical methods [2,5].

* Corresponding author. Tel.: +421 41 513 2951; fax: +421 41 56 52 940.
E-mail address: martin.mocilan@fstroj.uniza.sk

Peer-review under responsibility of the scientific committee of TRANSCOM 2017: International scientific conference on sustainable, modern and safe transport
doi:10.1016/j.proeng.2017.06.105

At present, numerical methods for surface wave problems are based on either the Navier–Stokes (NS) equations or potential flow theory. Due to the advantages in the efficiency and accuracy, the potential-flow-based methods which assume the fluid to be incompressible, inviscid and flow irrotational are very popularly used in sloshing studies. For the time-domain simulation, a potential-flow-based method usually has two key modules: One is for solving the Boundary Value Problem (BVP) of the velocity potential, which could be achieved by Boundary Element Method (BEM), Finite Element Method (FEM), Finite Difference Method (FDM), Pseudo-Spectral Method and so on; The other is for updating the boundary position and boundary conditions. For the free surface boundary, two procedures, the Mixed-Euler–Lagrange (MEL) and semi-Lagrangian (SL) procedure, are mostly adopted to update the free surface position and the corresponding boundary conditions [6].

In recent times, the Computational Fluid Dynamics (CFD) analysis is playing a vital role in analyzing the different design models and helps in saving time and cost by eradicating the need for construction of several prototypes in the design and testing phase [6].

In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler- Euler approach.

The VOF formulation in ANSYS FLUENT is generally used to compute a time-dependent solution, but only for problems with a steady-state solution; it is possible to perform a steady-state calculation provided solution is independent of initial guess. In case of vortex formed system with liquid-gas interface, the solution depends on the initial liquid height and hence transient solution method should be chosen.

The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the *q* (fluid's volume fraction) phase, this equation has the following form:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right) + \nabla.\left(\alpha_q\rho_q\,\vec{v}_q\right) = S_{\alpha q} + \sum_{p=1}^{n}\left(\dot{m_{pq}} - \dot{m_{qp}}\right)\right] \tag{1}$$

Where m_{qp} is the mass transfer from phase *q* to phase *p* and m_{pq} is the rate of mass transfer from phase *p* to phase *q*. By default, the source term on the right-hand side of equation (1) S_{aq}, is zero, but you can specify a constant or user-defined mass source for each phase. Volume fraction in the cell is denoted α_q. [7-8]

2. Computational study and boundary conditions

The analyzed problem consists of a closed tank, which is filled with hydraulic oil by 65% and the rest 35% of its volume is filled with air. The total volume of the hydraulic tank is 0.203 m^3, the volume of hydraulic oil is 0.132 m^3 which corresponds with 65% of tank volume. The tank is made from steel whose material properties are shown in table 1. Thickness of sheet metal walls is 6mm.

Tab. 1 Material properties

Material	Elastic Modul	Poisson´s Ratio	Density
Steel	206 GPa	0,29	7 827 kg/m^3
Hydraulic Oil	-	-	900kg/m^3

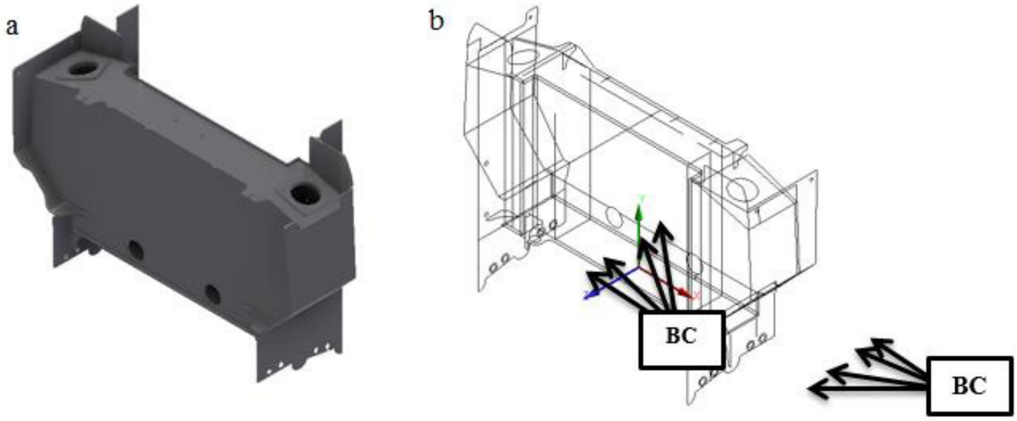

Fig.1 a.) CAD model of the Hydraulic tank b.) Wire model of the hydraulic tank with boundary conditions.

Coordinate system and boundary conditions for static solution is shown in Fig. 1 b. the tank was loaded in all directions by acceleration of 3G.

Tab. 2 Load Case

X	3G
Y	- 3G
Z	3G

The tank is fixed on 8 places as can be seen on Fig. 1 All degrees of freedom were fixed at these 8 places. These boundary conditions were used for static analysis and also FSI analysis of the hydraulic tank.

3. Static analysis of hydraulic tank

Hydrostatic pressure was used as load during static analysis. The pressure distribution can be seen on Fig. 3. The computational model of fluid was not used during static analysis. The approach involved the free surface at an angle of 45 degrees. The distribution of the free surface can be seen in Fig. 2.

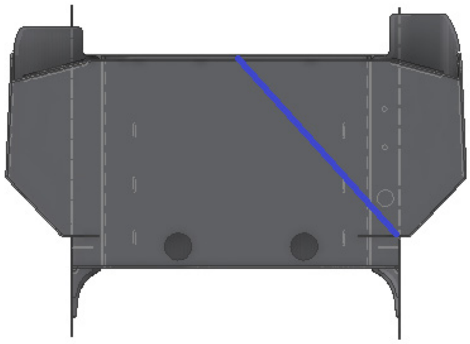

Fig.2 The distribution of the free surface in the tank.

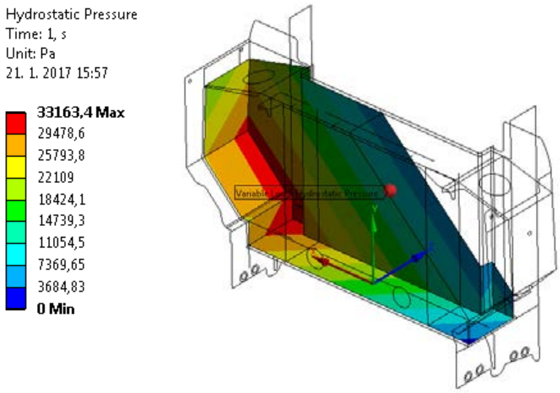

Fig. 3. Hydrostatic pressure.

The stress results of static structural analysis for surface is presented in the figure (Fig. 4).

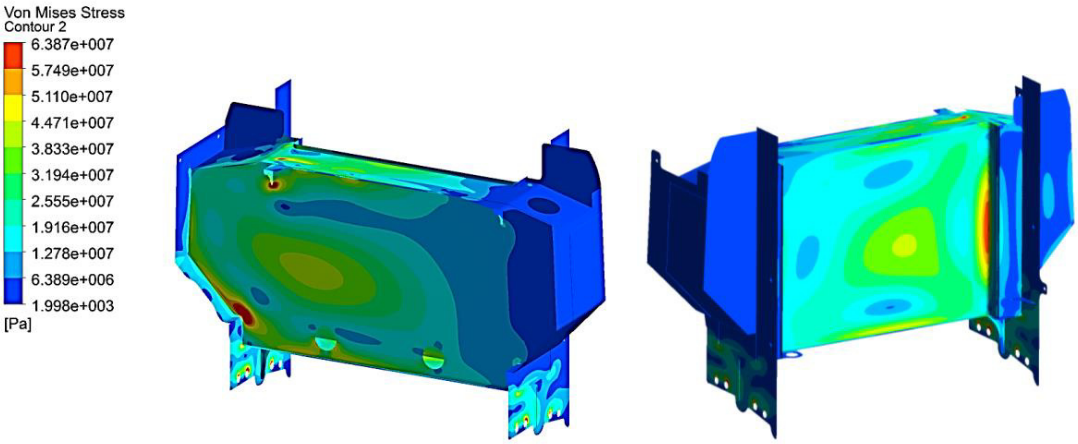

Fig. 4. The stress results for surface 1.

4. Fluid structure interaction of the hydraulic tank

The analyzed problem consisted of a closed hydraulic tank which was filled with hydraulic oil by 65% and 35% of air. The tank was at rest at the beginning of the simulation and it was subjected to time-dependent acceleration at time t = 2s. The goal was to find the maximum stress in the tank during acceleration. Multiphase model with two phases was used. One phase was air and the second phase was hydraulic oil. Volume of Fluid (VOF) multiphase model in ANSYS FLUENTTM 12.0 was used to predict the motion of the fuel inside the tank when the tank was under accelerated motion. The VOF model was designed to determine the position of interface between two or more immiscible fluids. Volume fraction of each of the fluids in each computational cell was tracked throughout the domain by sharing a single set of momentum equations between the fluids. The model relied on the fact that the fluids were not interpenetrating. The free surface shape prediction results of dynamic simulation at the time of t = 2s are represented in figure (Fig. 5.).

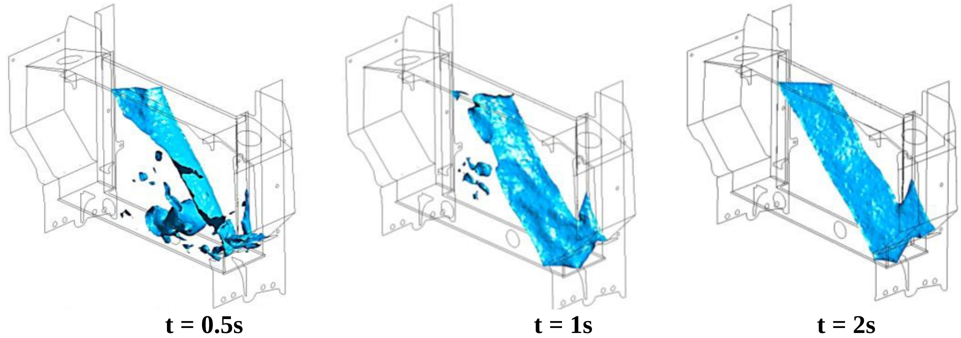

Fig. 5. Liquid interface at various time steps for tank without baffles.

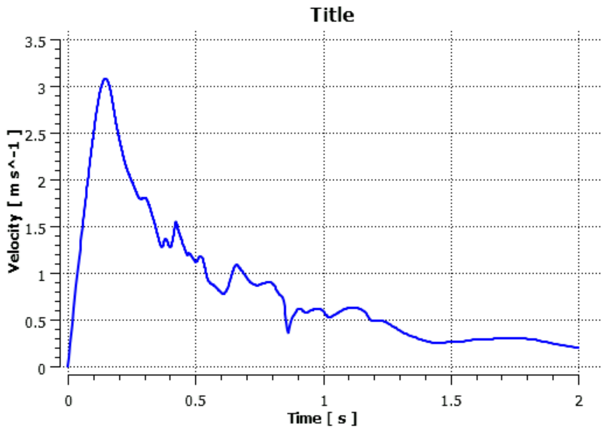

Fig. 6. The graph of process liquid flow velocity in the tank without baffles during 2 seconds.

The stress results of dynamic simulations in the time 2 second are presented in the figure (Fig. 7.).

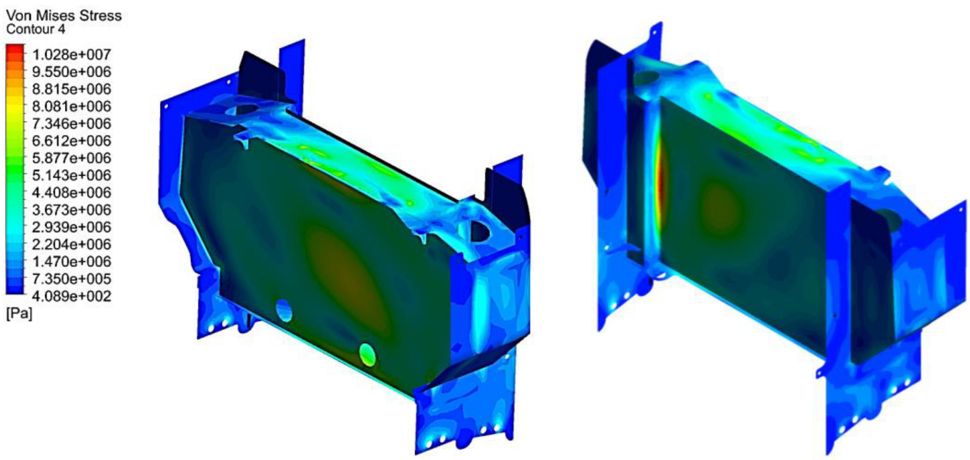

Fig. 7. The max. stress results in time 2 s.

5. Conclusion

During solution of the problem, simulation of the flow of hydraulic oil in the tank proved to be an important factor. Comparison of results from static simulation and fluid structure interaction shows that these methods produce different results. For static analysis, the maximum stress on surface was 63,87 MPa.

Static analysis can be performed using significantly smaller computational time than FSI, however the results produce higher stresses which do not correspond with the results of FSI. Using only static analysis during the design of tank can lead to oversized design, which could be considered as wasteful and undesired.

In the further research, different other configurations will be analyzed to optimize the design of the tank by further reducing the stress and sloshing phenomenon.

Acknowledgement

This work was supported by the Slovak Grant Agency No. VEGA 1/0983/15 and No. VEGA 1/0787/15.

References

[1] Singala, V, Bajaj, J, Awalgaonkara, N, Tibdewalc, S. 2014. CFD Analysis of a kerosene fuel tank to reduce liquid sloshing, Procedia Engineering 69, 1365 – 1371, 2014.
[2] Stankovičová, Z; Dekýš, V; Novák, P. 2016. Time average synchronization in thermoelastic stress analysis, Procedia Engineering 136, 204-210, 2016.
[3] Kopas, P., Vaško, M., Handrik, M. 2014. Computational Modeling of the Microplasticization State in the Nodular Cast Iron. Applied Mechanics and Materials, Vol. 474, 2014, p. 285-290, ISSN 1660-9336.
[4] Sága, M., Vaško, M., Pecháč, P. 2014. Chosen Numerical Algorithms for Interval Finite Element Analysis. In Procedia Engineering, Vol. 96, 2014, p. 400–409, ISSN 1877–7058.
[5] Sapietová, A., Sága, M., Novák, P., Bednár, R., Dižo, J. 2011. Design and Application of Multi-software Platform for Solving of Mechanical Multi-body System Problems. In Mechatronics: Recent Technological and Scientific Advances, 2011, p. 345-354, ISBN 978-3- 642-23243- 5.
[6] Močilan, M. – Žmindák, M. – Pastorek, P. (2015): Dynamic analysis of Fuel tank: Procedia Engineering. Vol. 136, pages 45-49.
[7] Janna, W. S. 2009. Introduction to Fluid Mechanics,Vol. 4. New York: CRS Press, 2009. ISBN 978-1-4200-8524-2.
[8] Zhang, Ch. 2015. Application of an improved semi-Lagrangian procedure to fully-nonlinear simulation of sloshing in non-wall-sided tanks, Applied Ocean Research 51, 74–92, 2015.