MECHANIKA
CZASOPISMO TECHNICZNE WYDAWNICTWO
TECHNICAL TRANSACTIONS POLITECHNIKI KRAKOWSKIEJ
MECHANICS
2-M/2010
ZESZYT 8
ROK 107
ISSUE 8
YEAR 107

MARIUSZ DOMAGAŁA*

ANALYSIS OF LIQUID MOTION IN THE MOBILE TANK

SYMULACJA RUCHU CIECZY W ZBIORNIKU

A b s t r a c t

This paper present the analysis of the free surface flow conducted in Ansys CFX package. The aim of CFD analysis was to simulate fluid behavior in a rectangular shape tank with two surge plates during short time longitudinal acceleration 1 g. Performed simulation allowed to obtain information about the liquid motion in the tank as well as to estimate forces acting on surge plates during liquid motion.

Keywords: CFD analysis, free surface flow

S t r e s z c z e n i e

W artykule przedstawiono symulację ruchu cieczy znajdującej się w prostopadłościennym zbiorniku z przegrodami podczas krótkotrwałego przyspieszenia wzdłużnego o wartości 1 g. Symulację powierzchni swobodnej cieczy przeprowadzono z zastosowaniem narzędzi CFD i systemu Ansys CFX. Przeprowadzone obliczenia numeryczne pozwoliły uzyskać informacje o kształcie powierzchni swobodnej cieczy przy częściowym wypełnieniu zbiornika oraz określić siły działające na przegrody zbiornika podczas ruchu cieczy. Analizę CFD wykonano w akademickim centrum komputerowym „Cyfronet" w Krakowie.

Słowa kluczowe: analiza CFD, powierzchnia swobodna

* PhD. Mariusz Domagała, Institute of Applied Infomatics, Faculty of Mechanical Engineering, Cracow University of Technology.

1. Introduction

Mobile tanks are widely used from decades on railway and road transport to carry various liquids from water to petrol products and even liquefied gases. During this time there was a huge development in the field of tank design, materials and manufacturing processes. Nowadays tanks are able to transport not only one type of fluid but the whole family of fluids with similar properties. Due to the differences in liquids density and limitations in permissible loads or requirements for some liquids it may happened that tank will be filled up only in the part of its nominal capacity. It means that during operation liquid sloshing within the tank. To avoid unexpected forces during that process tanks are equipped with surge plates. Such cases are regulated by international standards. According to EN 13530-2 regulations each tank that is provided to carry liquids which capacity is less than 80% of nominal tank capacity should be equipped with surge plates. Such plates should also resist dynamic forces that appear during tank operation. For example tanks that might be transported on railway have to fulfill requirements which are acceleration +/– 2g longitudinal direction and +/–1 g transverse direction. The EN 13530-2 standard claims that surge plates should resist forces that appears during such acceleration and defines load on surge plates as a pressure arising from liquid captured between plates.

Real phenomenon that appears in the tank during operation is a complex problem which is difficult to simulate. One of the way might be experiments, but it is very expensive if has to be performed for each type of tank and carrying fluid. Very helpful might be lastly developed tools which is called Computational Fluid Dynamics (CFD). Available capabilitiess of CFD tools allows to simulate free surfaces flow which have fund application mostly in environmental engineering. More and more application of free surface simulation in mechanical engineering application is also met. This paper presents an application of Ansys CFX code to simulate liquid behavior in the nonpresurrised rectangular shape mobile tank. It is presented results of simulation for liquid sloshing for longitudinal acceleration 1g.

2. Theoretical descriptions

In general the interface surface between liquid and air might be obtained as it presented in the work [1].

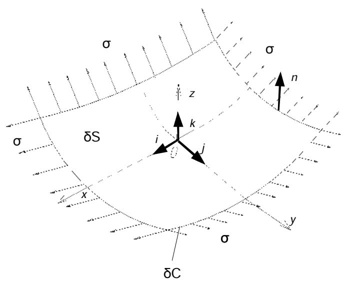

Fig. 1. A slice of free surface

Rys. 1. Wycinek powierzchni swobodnej

Assuming that the surface tension is constant $\sigma = \text{const}$ on slice of surface *S*, for center of coordinate system (which is assumed to be on surface *S*), plane that is tangential to the surface *S* might be described by formula

$$z = F(x, y) = 0 \tag{1}$$

Force acting on boundary *C* is determined in the following way

$$\begin{aligned} -\sigma\oint_{\sigma C} \mathbf{n}\times \mathrm{d}\,\mathbf{x} &= -\mathbf{k}\sigma\oint_{\sigma C}\left(-\frac{\partial F}{\partial x}dy + \frac{\partial F}{\partial x}dx\right) - \mathbf{i}\sigma\oint_{C} dy + \mathbf{j}\sigma\oint_{C} dx = \\ &= \mathbf{k}\sigma\int_{S}\left(\frac{\partial^2 F}{\partial x^2} + \frac{\partial^2 F}{\partial y^2}\right) \end{aligned} \tag{2}$$

where:

F	–	body forces,
σ	–	surface tension,
i, j, k	–	unit vector,
S	–	surface,
C	–	boundary over surface *S*.

Derivatives $\frac{\partial^2 F}{\partial x^2}$ and $\frac{\partial^2 F}{\partial y^2}$ represents curvature $\frac{1}{R_1}, \frac{1}{R_2}$ of surface in two perpendicular planes. Sum of these curvatures is constant and is called Gauss curvature.

$$\sigma\left(\frac{\partial^2 F}{\partial x^2} + \frac{\partial^2 F}{\partial y^2}\right) \equiv \sigma\left(\frac{1}{R_1} + \frac{1}{R_2}\right) \tag{3}$$

Surface S is in equilibrium when liquid tension force is balanced by pressure forces. In relation to surface unit equilibrium condition is as follows

$$\left(\frac{1}{R_1} + \frac{1}{R_2}\right) = \Delta p \tag{4}$$

For assumption that steady state conditions are considered there is no problem to determine the equation of free surface flow. But for transient conditions which are typical for mobile tanks during operation it is very difficult or even not possible. The only way is using numerical methods which are called CFD (Computational Fluid Dynamics). During the years of development CFD methods offers more and more possibilities which also includes multiphase flows and free surface flow. There are few numerical approaches that are meet in commercial CFD codes, the main two groups are [2]:
- interface tracking methods,
- interface capturing methods.

Capabilities of CFD tools are still expanding what gives possibility to simulate very complex phenomena. Despite this fact there are also conducted independent work on free surface simulations [3].

3. Object of simulation

The object of simulation is a rectangular shape tank with two surge plates shown in Fig. 2 with dimensions 6000 x 2000 x 3000 (in mm). The main goal of the simulation was to simulate liquid (water) behavior in the tank at longitudinal acceleration 1 g.

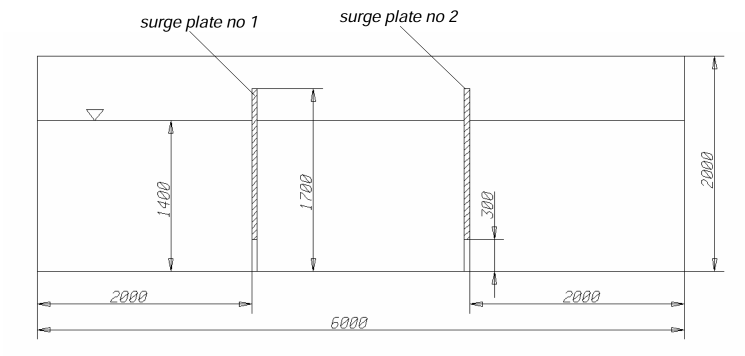

Fig. 2. Dimensions of tank that was used in CFD simulation

Rys. 2. Wymiary zbiornika użytego w symulacji CFD

To shorten time of calculation it was assumed 2D model shown on Fig. 3.

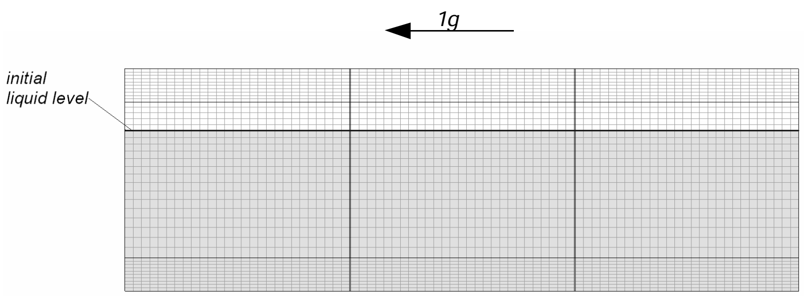

Fig. 3. Grid of the tank

Rys. 3. Model dyskretny zbiornika

4. CFD analysis

CFD analysis was conducted in Ansys CFX code. It was assumed that the initial level of liquid is 1400 mm, lower than height of surge plates. In CFD analysis was assumed that the acceleration 1g acts on the tank in direction perpendicular to surge plates for time 0.2 s (Fig. 4). Furthermore, it was assumed that there is no heat transfer in the model and both fluids (air at room temperature 25°C) are homogenous and phenomena that appears on vicinity to the walls might be neglected. The goal of CFD analysis was to investigate behavior of water in short time of longitudinal acceleration which may appear during tank operation. It was assumed that time of simulation will be 5.5 seconds.

As a results of simulation was presented force that appears on both surge plates (see Fig. 5) and some shape of water phase in selected time steps (see Fig. 6–8).

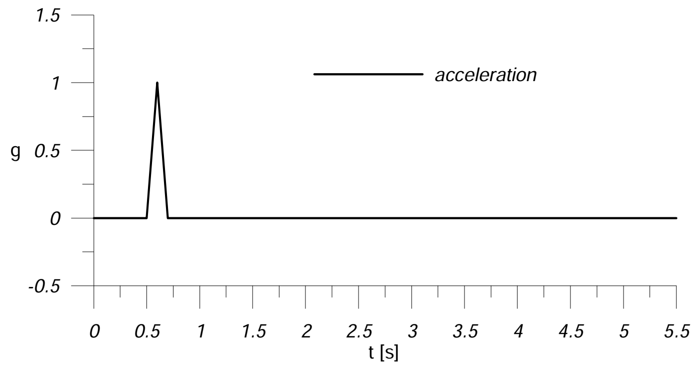

Fig. 4. Acceleration acting on the tank

Rys. 4. Przyspieszenie działające na zbiornik

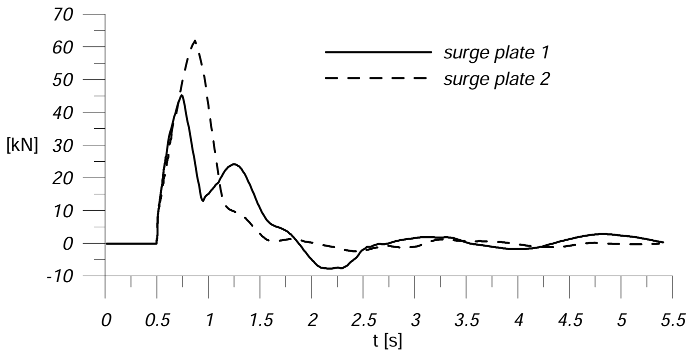

Fig. 5. Forces acting on surge plates

Rys. 5. Siły działające na przegrody zbiornika

Fig. 6. Water surface in time 0.6 s

Rys. 6. Objętość cieczy dla czasu 0,6 s

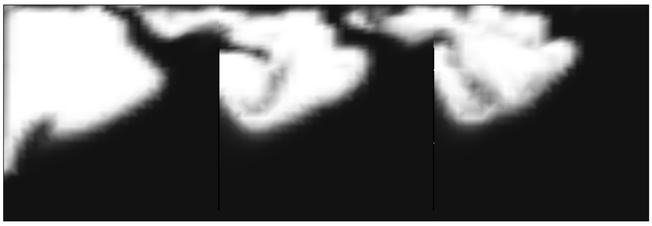

Fig. 7. Water surface in time 1.0 s

Rys. 7. Objętość cieczy dla czasu 1,0 s

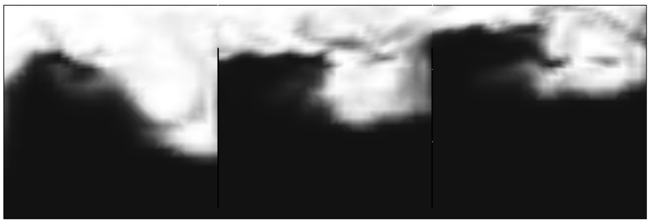

Fig. 8. Water surface in time 1.4 s

Rys. 8. Objętość cieczy dla czasu 1,4 s

5. Conclusions

This paper presents an application of Ansys CFX code to simulate behavior of liquid in the tank with longitudinal acceleration 1g. CFD analysis allowed to simulate liquid sloshing when tank is supplied with surge plates and estimate forces that acting on surge plates.

References

[1] Gryboś R., *Podstawy mechaniki płynów*, cz. 1, *Kinematyka, dynamika cieczy i gazów, hydrostatyka*, PWN, Warszawa 1998.

[2] Ferziger J. H., Peric M., *Computational Methods for Fluid Dynamics, 3rd Edition*, Springer Verlag, 2002.

[3] Warmowska M., *Numerical simulation of liquid motion in partly filled tank,* Opuscula Mathematica, Vol. 26, No. 3, 2006.

[4] Godderidgel A.B. Phillips, Lewis S., Turnock S.R., Hudson D.A., Tan M., *The simulation of free surface flows with Computational Fluid Dynamics,* ANSYS UK User Conference, 2008.