Numerical simulation based on the volume-of-fluid approach for compressible two-phase flow in the cylindrical reservoir

I V Morenko
Institute of Mechanics and Engineering - Subdivision of the Federal State Budgetary Institution of Science «Kazan Scientific Center of the Russian Academy of Sciences», Russian Federation

E-mail: morenko@imm.knc.ru

Abstract. The process of underwater implosion of the air sphere in the cylindrical reservoir is considered. Numerical simulation of the two-phase flow is based on the volume of fluid approach. Both air and water are considered compressible. Menter’s Shear Stress Transport turbulence model is chosen for the closure of Unsteady Reynolds-averaged Navier-Stokes equations. The solution of the problem is carried out by the finite volume method. The implosion event is on the order of milliseconds. The simulation results demonstrate that the pressure wave with high peak pressure moves from centre of the gas volume to reservoir wall. A comparison of pressure time histories were made between experimental data and numerical simulation results.

1. Introduction

The underwater implosion caused by external high pressure generates a pressure pulse in the surrounding water. The rapid collapse of the air structure is potentially damaging to adjacent structures. Understanding the mechanics of implosion is critical to the safe and efficient design of underwater devices and structures.

The underwater implosion is used in engineering applications. For example, oceanographic signaling device is proposed for measuring the depth of the seabed. Acoustic waves emanated from broken glass spheres are used to indicate the contact of the equipment with the seabed.

Very few experimental studies of underwater implosion exist in the literature, largely because of a limited number of test facilities capable of conducting implosion. The underwater implosion of cylindrical structures containing low-pressure gas and surrounded by high-pressure water were investigated in [1-4]. Underwater implosion experiments were conducted with thin-wall glass spheres to determine the influence that structural failure has on the pressure pulse in [5].

The underwater implosion is a fluid-structure interaction problem. Fluid-structure interaction problems are a broad class of problems involving both solid and fluid mechanics. The solution of the problem in the complete formulation is very difficult. In this study, only the hydrodynamic part of the problem is considered. It is believed that the shell of the low pressure structure is destroyed instantly.

2. Governing equations

A cylindrical reservoir with a radius of \( R = 0.76 \) m and a height of \( H = 3.65 \) m is filled with water (figure 1). The hydrostatic pressure is \( P^0 \). Thin-wall glass sphere of radius \( r = 0.0381 \) m is placed on
the axis of the reservoir. The sphere is filled with air. The initial air pressure is \( P_0 \). The axis \( Ox_3 \) of the Cartesian coordinate system \( x_1x_2x_3 \) is directed along the axis of the reservoir.

It is believed that submerged enclosed shells collapses instantaneously. When constructing a mathematical model, the time of collapse of a glass sphere is neglected. We also neglect possible chemical processes, condensation and evaporation of liquid from the interphase surface.

**Figure 1.** Scheme of the cylindrical reservoir.

For computer simulation of the two-phase flow with free surface the Volume of Fluid (VOF) \([6]\) method is selected. The motion of the medium is modeled by a single set of equations of motion, energy for all phases. VOF methods are the most popular numerical approaches used to predict interface motion \([7-8]\).

The two-phase medium consists of liquid and gas phases. Let \( \alpha \) is the volume fraction of the liquid phase. If the liquid is in the control volume, then \( \alpha = 1 \), if the gas, then \( \alpha = 0 \). The value ranging \( 0 < \alpha < 1 \) represents the interface.

The density \( \rho \) and the viscosity \( \mu \) are specified in terms of the indicator function \( \alpha \) by the following equations

\[
\rho = \rho_l \alpha + \rho_g (1-\alpha), \quad \mu = \mu_l \alpha + \mu_g (1-\alpha).
\]

where the subscripts \( l \) and \( g \) denote the liquid and gaseous phase, respectively.

The effects of compressibility are defined for each phase. The equations of state are given by

\[
\rho_g = \rho_g / R_g T, \quad \rho_l = \rho_{l0} + \rho_l / R_l T,
\]

here \( R_g \) is the universal gas constant, \( R_l = \text{const} \), \( \rho_{l0} = \text{const} \).

Unsteady Reynolds-averaged Navier-Stokes (URANS) equations are used to compute the turbulent compressible flows. The continuity equation takes the form

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0.
\]

The momentum equation takes the form
where \( u_i \) is the component of the averaged velocity vector \( \mathbf{u} \), \( t \) is the time, \( p \) is the pressure, \( \mu_{\text{eff}} = \mu + \mu_t \) is the effective viscosity, \( \mu_t \) is the turbulent viscosity, \( \delta_j \) is the Kronecker delta, \( \kappa = -\nabla \cdot n \) is the curvature of the interface, \( n = \nabla \alpha / |\nabla \alpha| \) is a unit vector normal to the interface, \( \sigma \) is the surface tension coefficient.

The transport of \( \alpha \) in time is expressed by an advection function:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) + \nabla \cdot ((1 - \alpha) \alpha \mathbf{u}) = -\frac{\alpha}{\rho_i} \frac{D\rho_i}{Dt}.
\]  

The necessary compression of the surface is achieved by introducing an extra artificial compression term \( \mathbf{u}_s \),

\[
\mathbf{u}_s = \mathbf{u}_i - \mathbf{u}_s.
\]

Conservation of energy equation can be written

\[
\frac{\partial (\rho T)}{\partial t} + \frac{\partial (\rho Tu_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \varphi_{\text{off}} \frac{\partial T}{\partial x_j} \right) - \left( \frac{\alpha}{c_{s,i}} + \frac{1 - \alpha}{c_{s,g}} \right) \left( \frac{\partial \rho K}{\partial t} + \frac{\partial}{\partial x_j} (\rho K u_i) + \frac{\partial}{\partial x_j} (\rho u_i) \right),
\]  

where \( T \) is the temperature, \( K = \frac{\left| \mathbf{u} \right|^2}{2} \) is the specific kinetic energy, \( \varphi_{\text{off}} = \frac{\alpha \lambda_i}{c_{s,i}} + \frac{(1 - \alpha) \lambda_s}{c_{s,g}} + \frac{\mu_t}{\Pr_t} \),

\( c_s \) – specific heat capacities at constant volume, \( \Pr_t = 0.85 \) is the turbulent Prandtl number.

The summation over \( i \) from 1 to 3 is assumed. The equations (1)-(5) are supplemented by the turbulence closure model, the initial and boundary conditions.

For the closure of URANS, the Menter’s Shear Stress Transport (SST) [9] turbulence model is chosen. SST is a widely used [10] and robust two-equation turbulence model:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = P' - \beta' \rho o k + \frac{\partial}{\partial x_j} \left( \mu + \sigma_t \mu_t \right) \frac{\partial k}{\partial x_j},
\]  

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \frac{\tau_{n_i}}{\nu_t} P' - \beta_o \rho \omega^2 + \frac{\partial}{\partial x_j} \left( \mu + \sigma_o \mu_t \right) \frac{\partial \omega}{\partial x_j} + 2(1 - F_i) \rho_o \sigma_o \frac{k}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j},
\]  

where \( k \) is the turbulence kinetic energy \( \omega \) is the rate of dissipation of the eddies.

In equations (6) to (7), the following notations for auxiliary relations and closure coefficients are used: \( P' = \min \left( P, 20 \beta' o k \right) \), \( P = \tau_{n_i} \frac{\partial u_i}{\partial x_j} \), the stress tensor \( \tau_{n_i} = 2 \mu_t \delta_{ij} \), the strain-rate tensor is defined as \( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \). The turbulent viscosity is defined by \( \mu_t = \frac{\rho a_k}{\max \left( \alpha_i \omega, \Omega F_2 \right)} \),

\[ \Omega = \sqrt{2W_i W_i}, \quad \omega = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right), \]  

second blending function \( F_2 \) is written \( F_2 = \tanh \left( \arg_2 \right) \),

\[
\arg_2 = \max \left( 2 \sqrt{ \frac{k}{\beta \omega d \omega}, \frac{500 \nu}{d'} \omega \right).
\]
The blending function $F_i$ is given by $F_i = \tanh(\text{arg}_i)$.

$$\text{arg}_i = \min \left[ \max \left( 2 \frac{\sqrt{k}}{\beta \omega d}, \frac{500 \omega d}{d^2 \omega} \right), \frac{4 \sigma_{a2} k}{C D_{a2} d^2} \right], \quad \text{CD}_{a2} = \max \left( 2 \sigma_{a2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} 10^{-20} \right),$$

$d$ is the distance to the nearest wall.

Constants $\gamma_{a2}$, $\beta_{a2}$, $\sigma_{a2}$ are defined as $\gamma_{a2} = F_i \gamma_{a1} + (1 - F_i) \gamma_{a2}$, $\beta_{a2} = F_i \beta_{a1} + (1 - F_i) \beta_{a2}$, $\sigma_{a2} = F_i \sigma_{a1} + (1 - F_i) \sigma_{a2}$; $\gamma_1 = \frac{\beta_1}{\beta'} - \frac{\sigma_{a1} \kappa^2}{\sqrt{\beta'}}$, $\gamma_2 = \frac{\beta_2}{\beta'} - \frac{\sigma_{a2} \kappa^2}{\sqrt{\beta'}}$.

The closure constants of the model $\sigma_{a1} = 0.85$, $\sigma_{a2} = 1.0$, $\sigma_{a3} = 0.5$, $\sigma_{a4} = 0.856$, $\beta_1 = 0.075$, $\beta_2 = 0.0828$, $\beta' = 0.09$, $\kappa' = 0.41$, $a_1 = 0.31$.

Basic equations (1)-(7) are supplemented by the initial and boundary conditions.

At the initial time $t = 0$ the velocity is zero on the whole domain. The pressure in the area $\Omega_l$ occupied by the liquid is $P = P_l^0$, and in the area $\Omega_g$ filled with gas is $P = P_g^0$; the temperature $T = T^0$.

At the boundary $\Gamma$ of the computational domain we use the following boundary conditions:

$$u_{\Gamma} = 0, \quad \frac{\partial p}{\partial n} = 0, \quad \frac{\partial \alpha}{\partial n} = 0, \quad T_{\Gamma} = T_{\Gamma}, \quad k_{\Gamma} = 0, \quad \phi_{\Gamma} = 10 \frac{6y}{\beta_1 d_1},$$

where $\beta_1 = 0.075$, $d_1$ is the wall step, $n$ is the unit normal vector.

Equations (1)-(7) are solved using the software OpenFOAM [11] licensed under the GNU General Public Licence. The hexahedral mesh is generated. Mesh thickening is performed in the region of large gradients of flow parameters. The solution of the problem is carried out by the finite volume method. The preconditioned conjugate gradient method is used for solving the pressure correction equation. The iterative procedure is used for solving equations for velocity. Discrete velocity and pressure values are calculated at the centers of the mesh cells. The time step $\Delta t$ is chosen so that the maximum Courant number $Co = u \Delta t / \Delta x$ in the whole domain does not exceed 0.03. Where $u$ is velocity magnitude and $\Delta x$ is the mesh size.

3. Results of numerical simulations

The following thermal and physical properties of the gas are given: the coefficient of kinematic viscosity $\nu_g = 1.46 \times 10^{-5}$ m$^2$/s, the specific heat at constant pressure $c_{pg} = 1006$ J/(kg·K), the thermal conductivity $\lambda_g = 0.024$ W/(m·K), as well as liquid properties: the density $\rho_l = 10^3$ kg/m$^3$, the coefficient of kinematic viscosity $\nu_l = 10^{-6}$ m$^2$/s, the specific heat at constant pressure $c_{pl} = 4182$ J/(kg·K), the thermal conductivity $\lambda_l = 0.6$ W/(m·K). In addition, $R_g = 8.314$ J/(mol·K) is the universal gas constant, $R_l = 3000$ J/(mol·K), $\sigma = 0.07$ N/m is the surface tension coefficient. The liquid pressure in the area $\Omega_l$ is $P_l^0 = 6.996$ MPa, and the gas pressure in the area $\Omega_g$ is $P_g^0 = 0.1$ MPa. At the initial time $t = 0$ the temperature is $T^0 = 300$ K.

A series of numerical calculations is carried out. It is established, that the duration of a typical implosion event is on the order of milliseconds. In the initial phase, the liquid rushes into the low-pressure region. The air volume decreases, the air density increases. In the surrounding fluid the rarefaction wave is formed (figure 2 on left). When the gas volume reaches a minimum value, a high-pressure wave is generated at the centre of the gas volume. The spherical wave moves to reservoir wall (figure 2 on right).
Figure 2. Pressure fields at time $t = 0.25 \, mc$ (on left) and $t = 0.75 \, mc$ (on right).

The pressure was measured at a control point in the surrounding fluid, which is located at a distance of 0.1 m from the sphere center. The pressure history at the control point is shown in figure 3. When the rarefaction wave passes through the control point, then the fluid pressure is less than the hydrostatic pressure $P_H \ll P_f$ is recorded. The experimental data [5] and numerical simulation results are compared. Where $t_{exp}$ and $t_{num}$ is the time moment of the beginning of the implosion process in the experiment and in the numerical calculations, respectively. The time period of rarefaction in the experiment is longer than in the calculations. This fact is explained by the fact that the simulation does not take into account the destruction of the thin-wall glass sphere.

In experiment [5] three pressure sensors measured the pressure. There is a difference in the peak pressure between all sensors, indicating some asymmetry exists in the implosion. Sensors recorded the peak values: 27.0, 25.3, 31.1 MPa. The peak pressure 24.4 MPa in the numerical simulation is less than that in the experiment. The most interesting effects occur during the first cycle of compression-expansion of the air sphere. Secondary pulsations are less intense. The amplitude of the oscillations of the thermodynamic parameters decreases. In the second cycle of the implosion process the pressure is again decreases and then the local maximum pressure 9.2 MPa is observed. Finally, the fluid return to equilibrium.
Figure 3. The dependence of the pressure at the control point from the time: – calculation, ● - experiment [5].

4. Conclusions
In the present work, attention was focused on studying the generation of pressure waves from the implosion. The results of numerical simulation showed that during the implosion of the air sphere in the initial phase, the rarefaction wave moves from air-water interface. In the second phase, the high-pressure spherical wave is generated in surrounding water. A volume of fluid method implemented in OpenFOAM is a predictive tool for the simulation of the hydrostatically-induced implosion.

References
[1] Gupta S, LeBlanc J M, Shukla A 2014 International Journal of Solids and Structures 51 3996–4014
[2] Farhat C, Wang K G, Main A, Kyriakides S, Lee L-H, Ravi-Chandar K and Belytschko T 2013 International Journal of Solids and Structures 50 2943–2961
[3] Ikeda C M, Wilkerling J and Duncan J H 2013 Proc R Soc A. 469 20130443
[4] Shukla A, Rajapakse Y D S and Hynes M E 2013 Springer Science & Business Media 364
[5] Turner S E 2007 Journal of the Acoustical Society of America 121(2) 844–52
[6] Hirt C W, Nichols B D 1981 Journal of Computational Physics 39(1) 201–225
[7] Morenko I V and Volkov Y A 2018 Proceedings of the higher educational institutions. Energy sector problems 20(1-2) 128-137
[8] Ferrer P J M, Causon D M, Qian L, Mingham C G and Ma Z H 2016 Computers and Fluids 125 116–129
[9] Menter F R 1994 AIAA J 32(8) 1598–1605
[10] Gubaidullin D A and Snigerev B A 2018 High Temperature 56(1) 61–69
[11] OpenFOAM – The open source CFD toolbox. URL: http://www.openfoam.com