Numerical simulation of a draining vessel

M I Nasyrova and P A Kulakov
Ufa State Petroleum Technological University, 1, Kosmonavtov St., Ufa, 450044, Russia
E-mail: nasyrova.marina@mail.ru

Abstract. In this paper the models of a theoretical tank and a real size draining vessel are described and numerically simulated. The main idea is to demonstrate the Toricelli’s law theorem for fluid dynamics for ideal liquid both for a theoretical vertical cylindrical tank and for a different geometry horizontal vessel of the same volume as the tank and compare the total time of liquid leaving the space of both of the reservoirs. The simulation is a multiphase problem containing two phases, which are water and air, is based on the VOF (volume of fluid) method discrete elements method (DEM) for simulating the sand bed formation for outlet nozzle capacity evaluation. As a result there are two-phase (gas-liquid) models of an “ideal” tank and an industrial vessel numerically calculated for determining and comparing the total time of water draining, and an extra model showing the dependence of sediment layer thickness on the outlet nozzle capacity.

1. Introduction
In oil industry there are many different types of equipment used for storage of liquids and gases. A cylindrical storage vessel is one of the most popular type of fluid storage equipment and it is widely used for storage due to being less expensive than other types of storage tanks, like a sphere, for example [1].

As industrial vessels are usually filled with a definite amount of liquid, they experience hydrostatic pressure. Hydrostatic pressure is the pressure exerted by a fluid at a given point within the fluid, due to the force of gravity, and increases in proportion to depth measured from the surface because of the increasing weight of fluid exerting downward force from above [2]. An industrial vessel or a tank experiences hydrostatic pressure either during a hydro testing or during keeping the liquid product inside.

Since a typical vessel represents a thin-walled construction, it is limited to stand the exact pressure value for an exact time period. Thus, there is a necessity to drain the liquid phase out of the apparatus either after the definite time of the exploitation or after a hydro testing.

A drain system that is connected directly to pressure vessels is called a “pressure” or “closed” drain system. A drain system that collects liquids that spill on the ground is an “atmospheric”, “gravity”, or “open” drain. The liquid in a closed drain system must be assumed to contain dissolved gases that flash in the drain system and can become a hazard if not handled properly. In addition, it must be assumed that a closed drain valve could be left open by accident. Once the liquid has drained out of the vessel, a large amount of gas will flow out of the vessel into the closed drain system (gas blowy) and will have to be handled safely [3]. Thus, closed drain systems should always be routed to a pressure vessel and should never be connected to an open drain system.
Due to the fact every kind of action during exploitation of industrial apparatus must be strictly planned and controlled, calculating the exact time of draining a tank or a vessel is a particular interest in this paper. This is basically reached by knowing or calculating the product outlet nozzle’s passage capacity, that’s why this value has the most important meaning in the entire draining calculation process. Knowing the exact time of draining a vessel gives an opportunity to schedule the work plan, correct the calendar schedule and even predict a range of risks caused by unexpected time loss.

Another problem is connected with separation of sediments from liquid caused by long settling of liquid in the vessel. The bottom sediment can include sand, wax or paraffin-like substance that make the draining process difficult due to clogging the outlet nozzle.

The newest calculation methods are based on finite element method (FEM). In oil industry there are many different types of equipment used for storage of liquid and gases. A cylindrical storage vessel is one of the most popular type of fluid storage equipment and it’s widely used for storage due to being less expensive than other types of storage tanks, like a sphere, for example [1].

As industrial vessels are usually filled with a definite amount of liquid, they experience hydrostatic pressure. Hydrostatic pressure is the pressure exerted by a fluid at a given point within the fluid, due to the force of gravity, and increases in proportion to depth measured from the surface because of the increasing weight of fluid exerting downward force from above [2]. An industrial vessel or a tank experiences hydrostatic pressure either during a hydro testing or during keeping the liquid product inside.

Since a typical vessel represents a thin-walled construction, it’s limited to stand the exact pressure value for an exact time period. Thus, there is a necessity to drain the liquid phase out of the apparatus either after the definite time of exploitation or after a hydro testing.

A drain system that is connected directly to pressure vessels is called a “pressure” or “closed” drain system. A drain system that collects liquids that spill on the ground is an “atmospheric,” “gravity,” or “open” drain. The liquid in a closed drain system must be assumed to contain dissolved gases that flash in the drain system and can become a hazard if not handled properly. In addition, it must be assumed that a closed drain valve could be left open by accident. Once the liquid has drained out of the vessel, a large amount of gas will flow out of the vessel into the closed drain system (gas blowy) and will have to be handled safely [3]. Thus, closed drain systems should always be routed to a pressure vessel and should never be connected to an open drain system.

Due to the fact every kind of action during exploitation of industrial apparatus must be strictly planned and controlled, calculating the exact time of draining a tank or a vessel is a particular interest in this paper. This is basically reached by knowing or calculating the product outlet nozzle’s passage capacity, that is why this value has the most important meaning in the entire draining calculation process. Knowing the exact time of draining a vessel gives an opportunity to schedule the work plan, correct the calendar schedule and even predict a range of risks caused by unexpected time loss.

2. Calculation details
2.1 Brief concept of axisymmetric and non-axisymmetric loading of thin-walled shells
A shell of revolution is called axisymmetric if it is loaded with loads uniformly distributed about its axis. Examples of axisymmetric shells are spherical, cylindrical, conical and elliptical shells of revolution, loaded with internal gas or liquid pressure [4].

A simple example of an axisymmetric shell is a vertical cylindrical storage tank.

In a horizontal apparatus filled with water, each point of the cross section experiences different pressure and, therefore, this shell is no longer considered to be axisymmetric.

A hydrostatic pressure distribution diagram for cylindrical axisymmetric and non-axisymmetric shells respectively are shown in figure 1.

It is important to consider whether the shell is axisymmetric or not, since the type of the shell depends on both the distribution of hydrostatic pressure and, consequently, the stresses in the metal of the shell, and the rate of flow of fluid from the shell, depending on the location of the drainage hole, which is one the main objectives affecting the calculation result in this paper.
2.2 Volume of fluid (VOF) method

The volume-of-fluid (VOF) is one of the most widely spread and used method in computational fluid dynamics (CFD) nowadays.

The VOF model allows modelling two or even more immiscible fluids by tracking the volume fraction of each of the fluids throughout the domain and solving a set of volume fraction, momentum, energy and additional scalar equations, including the main equation which is so called fraction function $C$ [5–8].

The fraction function represents the integral of fluid characteristic function in the control volume, which is the volume of a computational grid cell. The volume fraction of the entire fluid domain is tracked through every cell of the grid, however all the fluid domains share the same momentum equations. If the computational cell is empty, not filled with the fluid, the fraction function value is zero ($C = 0$); if the cell is 100%-full with the fluid, the fraction function value is 1 ($C = 1$). So if two different fluids are separated by a fluid interface, the fraction function in the interface cell ranges like $0 < C < 1$ [9–11].

In VOF method the evolution of the $m$-th fluid in the whole system is governed by solving the transport equation

$$\frac{\partial C_m}{\partial \tau} + \mathbf{u} \nabla C_m = 0,$$

where $\mathbf{u}$ id the velocity. Function $C_m$ is the volume fraction of the $m$-th fluid in the system [12].

The volume of the fluids is considered to be constant, thus the properties of the fluids can be calculated using the volume fraction function in the computational cell [13]. For instance, density on the fluid can be defined as

$$\rho = \sum_{m=1}^{n} \rho_m C_m.$$  \hspace{1cm} (2)

The main advantage of using the VOF method is the fact the calculations require minimum time and memory storage. Moreover there is an opportunity to deal with difficult non-linear tasks and track the fluid cells in more complicated models connected with multiphase problems [14–15].

However, due to the problem of superfluous diffusion in the transport equation there is a question about calculating and tracking the fluid-fluid interface because of so called smudging the interface area between two immiscible fluids [16].
2.3 Toricelli’s law

Studying a theoretical model of a vertical cylindrical tank, full of uncompressible (ideal) fluid, the instantaneous flow out of the nozzle on the bottom of the tank is calculated using an equation

\[ Q = \frac{dV}{dt}, \]

which basically means the instantaneous flow of the fluid equals the change of the volume of fluid in time [17–18].

According to the Toricelli’s law the velocity of the fluid flow through a hole of a thin wall, located on a definite h height over the surface, is the velocity of a body freefalling from a h height [19]

\[ v = \sqrt{2gh} \]

where \( g \) is gravitational acceleration.

Fluid flow rate can be calculated as

\[ Q = av \]

where \( a \) is the cross-sectional area of the nozzle, m².

Since the main interest in this paper is calculation the time of totally emptying the tank, there is the final equation for calculation the total time of fluid leaving the inner space on the tank based on the Toricelli’s law [20–21]

\[ T = \frac{R^2}{r^2} \frac{2H}{\sqrt{g}} \]

where \( R \) is the cylindrical tank inner radius, \( r \) is the nozzle inner radius and \( H \) is the total fluid height inside the tank.

2.4 Concept of nozzle capacity

The calculation of the nozzle capacity is based on a number of so-called calculated models representing certain assumptions that simplify the actual physical picture of the flow.

The calculation is based on the calculated model, and then amendments are made to the result, taking into account the imperfection of the model and the additional influence of various factors [22].

All calculations of the nozzle bandwidth are based on the ideal nozzle model.

According to this model, the flow through the nozzle is firstly calculated for adiabatic (without heat exchange with the environment) and isentropic (without hydraulic friction loss) flow through the nozzle. Then correction factors (for example, additional friction, local or hydrostatic pressure loss) are applied to the calculated nozzle bandwidth.

Since the models in this paper are considered to be close to ideal, the nozzle bandwidth can be substituted with volume flow rate

\[ Q = \frac{V}{t} \]

where \( V \) is the volume of fluid passing through the flow cross-section during time \( t \), m³.

Or the nozzle bandwidth for the current problem also can be calculated according to (5).

3. Boundary conditions

Both of the reservoirs are of the same volume which is 3.85 m³. The tank is a vertical cylindrical reservoir with both a flat top and a flat bottom. The vessel is a horizontal apparatus consisting of a cylindrical shell and two elliptical dishes on both of the ends. It should be noticed that the nozzles filled with the fluid are not included in the total fluid volume. Geometry parameters of the tank and the vessel are shown in Figure 2.

The tank is supposed to be a typical cylindrical axisymmetric shell, so it is considered that the pressure value at any cross-sectional area of the shell is constant. It also should be mentioned that the nozzle (the orifice) for water draining is located symmetrically in the middle of the bottom of the tank.

The vessel represents a more complicated task since it’s not only a non-axisymmetric shell model, but also includes a couple of nozzles for water draining and for air entering, that are located on the
sides of the vessel. Comparing to the tank the complexity of the vessel model is explained by the absence of a free surface on top—this element is actually substituted by a nozzle for air entering. Moreover the vessel construction includes a couple of elliptical dishes that makes it more complex to calculate analytically.

Figure 2. Geometry parameters of the tank and the vessel

The nozzle inner diameter is 200 mm both for the tank and for the vessel for the numerical calculation.

The numerical calculation for both of the reservoirs is a pressure-based task, based on the VOF method with two eulerian phases, where air is considered to be the primary phase and water is the secondary one. Implicit body force formulation is enabled in the task.

Both of the fluids (water and air) are set as ideal fluids. Surface tension between water and air is 0.072 H/m. The entire volume both inside the tank and the vessel are initialized as water, so the volume fraction function by the beginning of the calculation equals 1 for the water and 0 for the air phase for the whole computational grid area.

Operating conditions are normal, so operating pressure is atmospheric which is 1 atm.

Boundary conditions for numerical calculation for both of the reservoirs include gravitational acceleration which equals 9.81 m/s.

4. Calculation results discussion

According to the research work (6) for the tank there are values of fluid velocity and total time of fully emptying the tank depending on the nozzle inner diameter in Table 1.

Table 1. Values of fluid velocity and total time of fully emptying the tank depending on the nozzle inner diameter

| Nozzle inner diameter, mm | Total draining time, s |
|---------------------------|------------------------|
| 200                       | 30.27                  |
| 150                       | 53.80                  |
| 100                       | 121.07                 |

The maximum velocity magnitude is detected at the highest liquid level which is 3.467 m.

Volume fraction of water phase at the beginning and at the end of the tank and the vessel draining respectively are shown in figure 3.

As the result it took 31.5 s to totally empty the tank and 41.93 s for the vessel.

Comparing the numerical calculation time (31.5 s) to the theoretically calculated value (30.27 s) it should be mentioned that the result is almost the same. The slight difference might be caused by the rough geometry for numerical simulation.
Figure 3. Volume fraction of water phase at the beginning and at the end of the tank and the vessel draining respectively: a – model of a tank draining; b – model of a vessel draining

Talking about the result for the vessel, it took about 30% more time to empty the space inside the reservoir, which is caused by:

- a much lower height of the liquid as the initial parameter, and as a consequence, lower hydrostatic pressure for emptying the tank comparing to the tank;
- non-standard geometry of the reservoir;
- asymmetrical nozzle location, so the weight of the water wasn’t concentrated evenly in the nozzle area due to the hydrostatic pressure.

There is also another problem including nozzle capacity evaluation due to impurities sedimentation on the bottom of the vessel, representing the third discrete phase. The sand is granular material with mean particles size 150 mkm. The calculation of sedimentation process was stopped until the height of the sand layer reached value of 10 mm.

Bottom sediment distribution by time residence (hours) is shown in figure 4.

Figure 4. Sediment forming a sand bed on the bottom of the vessel causing a draining problem

As it is seen from fig.4 draining process discrete phase forms a thick layer on the bottom of the vessel, mainly caused by gravity force. Draining process of the vessel was calculated taking into account the sand bed presence.

According to the research work (7) the bottom nozzle capacity was calculated. Dependence of the nozzle capacity (for a nozzle with 100 mm inner diameter) on the sand bed layer thickness is shown in table 2. It is evident that clogging up the nozzle cross sectional area with sediment increases the total time of the draining process for the vessel.

According to the calculation results in Table 2 it is evident that the sediments layer growth causes less nozzle capacity and at a critical value of the sand bed thickness (approximately 200 mm for this case) there is no way for fluid to leave the space of the vessel due to clogging up the liquid outlet, so in this case the nozzle capacity is supposed to equal zero.
Table 2. Dependence of nozzle capacity and total draining time on the sediment layer thickness for the vessel

| Sediment layer thickness, mm | Nozzle Capacity, $x10^{-4}$ m$^3$/s | Total draining time, s |
|-----------------------------|-----------------------------------|------------------------|
| 1                           | 318                               | 121.07                 |
| 5                           | 277                               | 138.50                 |
| 10                          | 213                               | 180.78                 |
| 50                          | 154                               | 250.30                 |
| 100                         | 80                                | 480.65                 |
| 200                         | 0                                 | --                     |

5. Conclusion

To sum up, the time calculation methods overview and experiment was made in order to demonstrate different ways of solving the problem of industrial vessels total time emptying, which are one of the most important aspects in projects planning, risks and time loss prediction. Using numerical calculation methods allows not only saving time while planning the process, but also creating a range of ways for solving the same problem.

Comparing the numerical calculation time (31.5 s) to the theoretically calculated value (30.27 s) it should be mentioned that the result is almost the same. The slight difference is caused by near wall effects calculation moments in the numerical model.

Talking about the result for the vessel, it took about 30 % more time to empty the space inside the reservoir, which is caused by:

- non-uniform vessel cross section hydrostatic pressure distribution;
- a much lower height of the liquid as the initial parameter;
- a different shape of the reservoir;
- asymmetrical nozzle location, so the weight of the water was not concentrated evenly in the nozzle area due to the hydrostatic pressure.

According to the results it should be noticed that the theoretical method of calculating the total time for draining a reservoir could be applied including appropriate correction factors since the real result depends on the geometry imperfection, real liquid compressibility, additional forces (frictional, inertial etc.) and others.

Taking into account nozzle capacity calculation it is possible to conclude sediment formation is a burning industrial problem causing not only clogging up the outlet nozzles, but also a range of unexpected troubles. One of the ways to cope with sedimentation is to maintain the homogeneity of the fluid inside the vessel by constant phases mixing process.

Despite the calculations being considered to be close to ideal models, numerical calculation provides quite accurate information due to a more complex system of initial and boundary conditions for the experiment and a possibility to solve the problem using discrete phase method to reach a closer to real model result.

References

[1] Couper J R, Fair J R, Penney W Roy and Walas S M 2004 Chemical process equipment. Solution and design 2nd ed pp 655–75
[2] Nakayama Y and Boucher R F 1999 Introduction to fluid dynamics pp 30–8
[3] Tanaka R, Uddin M A and Kato Y 2018 Flow characteristics related to liquid/liquid mixing pattern in an impeller-stirred vessel ISIJ Int pp 620–66
[4] Wagner W and Gruttmann F 1994 A simple finite rotation formulation for composite shell elements Engineering Computations 145–76
[5] Xu W and YangY-R 2011 Numerical simulation of multiphase flow based on VOF (Sichuan Daxue Xuebao, Gongcheng Kexue Ban) pp 294–6
[6] Smirnov B M and Stephen Berry R 2015 Growth of bubbles in liquid Chem. Cent. J. 1–8
[7] Wang J, Doussin J F, Perrier S, Perraudin E, Katrib Y, Panguie E and Picquet-Varrault B 2011 Design of a new multi-phase experimental simulation chamber for atmospheric photosmog, aerosol and cloud chemistry research Atmos. Meas. Tech. 2465–94
[8] Boisson N and Malin M R 1996 Numerical prediction of two-phase flow in bubble columns Int. J. Numer. Methods Fluids 1289–310
[9] Waltrich P J, Falcone G and Barbosa J R Jr 2011 Performance of vertical transient two-phase flow models applied to liquid loading in gas wells Proc. SPE Annu Tech. Conf. Exhib pp 3568–82
[10] Lee Y K, Ahn K H and Lee S J 2014 Local shear stress and its correlation with local volume fraction in concentrated non-Brownian suspensions: Lattice Boltzmann simulation Phys. Rev. E Stat. Nonlinear Soft Matter Phys. 062317
[11] Sun X, Smith T R, Kim S, Ishii M and Uhle J 2003 Interfacial structure of air-water two-phase flow in a relatively large pipe Exp. Fluids 206–219
[12] Pfleger D, Gomes S, Gilbert N and Wagner H-G 1999 Hydrodynamic simulations of laboratory scale bubble columns fundamental studies of the Eulerian-Eulerian modelling approach Chem. Eng. Sci. 5091–5099
[13] Asad Sangabi F and Talebbeydokhti N 2011 Numerical modeling of two phase flow in Morning-Glory spillways Dam Maint. Rehabil. Proc. Int. Congr. Dam Maint. Rehabil pp 529–35
[14] Marder M, Chen C-H and Patzek T 2015 Simple models of the hydrofracture process Phys. Rev. E Stat. Nonlinear Soft Matter Phys. 062408
[15] Al Jibory M W, Al Turaihi R S and Al Jibory H N 2018 An experimental and numerical study for two phase flow (water-air) in rectangular ducts with compound turbulators IOP Conf. Ser. Mater. Sci. Eng. 012049
[16] Tanaka R, Uddin M A and Kato Y 2018 Flow characteristics related to liquid/liquid mixing pattern in an impeller-stirred vessel ISIJ Int. 620–6
[17] Swanepoel K J 1999 The fermat-toricelli problem and a balancing game J. Geom. 23–24
[18] Yetilmezsoy K 2017 IMECE—Implementation of mathematical, experimental, and computer-based education: A special application of fluid mechanics for civil and environmental engineering students Comput Appl Eng Educ 833–860
[19] Thevenin D and Janiga G 2014 Fluid dynamics for engineers pp 68–9
[20] Otto J 2018 Toricelli’s law for large holes pp 5–7
[21] Bistafa Sylvio R 2015 First Theoretical constructions to the fluid mechanics problem of the discharge Advances if historical studies 172–99
[22] Korelshtein L B 2016 Calculation of safety values bandwidth in the dfaft of the new State Standard Calculations and design 40–3