Numerical Simulation of Liquids Draining From a Tank Using OpenFOAM

Numerical simulation of liquids draining from a tank using OpenFOAM

Fadhilah Mohd Sakri[1*], Mohamed Sukri Mat Ali[1], Sheikh Ahmad Zaki Shaikh Salim[1] and Sallehuddin Muhamad[2]

[1] Wind Engineering Laboratory, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100, Kuala Lumpur, Malaysia.
[2] Department of Engineering, UTM Razak School of Engineering and Advanced Technology, Universiti Teknologi Malaysia, 54100, Kuala Lumpur, Malaysia.

*eypasakri@gmail.com

Abstract. Accurate simulation of liquids draining is a challenging task. It involves two phases flow, i.e. liquid and air. In this study draining a liquid from a cylindrical tank is numerically simulated using OpenFOAM. OpenFOAM is an open source CFD package and it becomes increasingly popular among the academician and also industries. Comparisons with theoretical and results from previous published data confirmed that OpenFOAM is able to simulate the liquids draining very well. This is done using the gas-liquid interface solver available in the standard library of OpenFOAM. Additionally, this study also able to explain the physics flow of the draining tank.

1. Introduction

Liquid storage tank is important to maintain a system run continuously or when required without any shortage in the supply of the working liquids. The use of liquid storage tanks can be found on the space vehicle for its propellant storage, oil industries for its short and long term storages at the refineries and also water tank for the domestic use. The draining of liquids from the storage tank is usually made by pressured system and/or drain by gravity at which the liquid is forced out through a small pipe outlet.

The process of draining liquids in a tank generates several flow phenomena. One of that is the formation of air-core vortex. Air-core vortex is rotational motion of the liquid at which air entraining the vortex through its core. When the core of the vortex extend till the bottom of the tank, the rate of draining process is reduced and the flow at the outlet is highly rotational and unsteady. If this happen, vibration may occurs that then reducing the life of the storage tank.

Various passive flow control mechanisms have been suggested and applied to reduce the effect of the air-core vortex generated in the draining tank. The most common device is the vortex breaker [1]. Vortex breaker is a device that stop the angular velocity of the liquid by using its radial vanes. There are studies [2] to improve the efficiency of the vortex breaker that includes the design of the baffles and vanes. In addition to the vortex breaker, the formation of the air-core vortex can also be reduced by shaping the geometry of the outlet [3]. However, the efficiency all this passive vortex controls are still lower than that the theoretical value that assumes the air-core vortex is not developed at all (irrotational flow).

Therefore, the study of liquid draining process needs to be revisited to understand in detail the fundamental physic flow of the generation of the air-core vortex. With the rapid advance of

computational performance and many accurate numerical models have been developed to simulate a broad range of fluids problems, numerical simulation is the best tool to get all the information required in the investigation of liquid draining of a tank.

　　This paper presents a numerical simulation of a liquids draining from a tank using OpenFOAM free CFD software package[4]. OpenFOAM has the capabilities to simulate a wide range of flow problems. Its extensive range of features are contributed voluntarily by the CFD user community around the world. Thus, OpenFOAM increasingly popular than other commercial CFD packages. In this paper, results obtained from OpenFOAM are compared with the theoretical and previous published results of similar case. The proper environment variable setting in OpenFOAM for simulating the liquids draining problem is also explained in this paper.

2.　Description of the problem geometry

A cylindrical tank of diameter (D) 90mm and length (L) of 450mm is filled partially with water. The initial height of the water measured from the bottom of the tank ( ) is 350mm. A drain nozzle is located at the centre of the bottom surface of the tank. The drain nozzle diameter (d) is 6mm and length (l) of 15mm. The top and bottom of the tank is open, i.e., atmospheric condition. The fluid is drained by gravity, g, downward. This problem geometry is intentionally made the same as the experimental and numerical investigations of Park & Sohn [5] so that a comparison study can be made. Figure1 show the schematic diagram of the problem geometry.
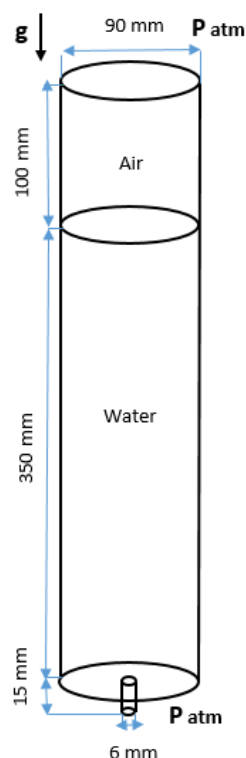

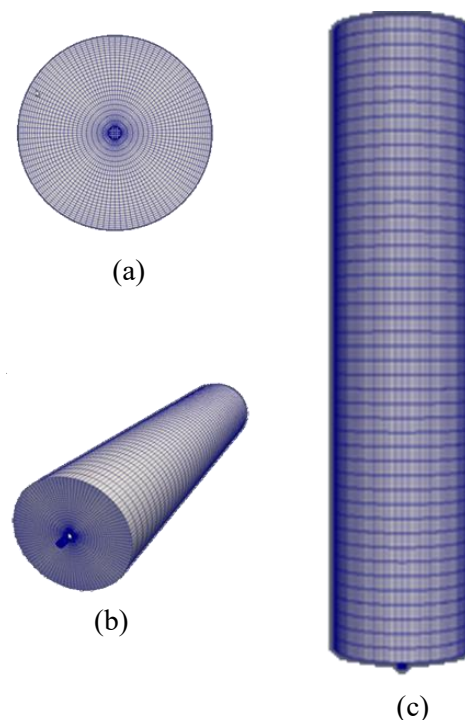Figure 1. Schematic diagram

Figure 2. Draining tank mesh and geometry structure in (a) top, (b) projection and (c) side views.

3. Analytical estimation

The time for the liquid to complete draining can be estimated using the equation of conservation of mass. The mass relation for a control volume undergoing any transient process is given by – Prateep Basu , Dheraaj Agarwal , T. John Tharakan and A. S [6] – Y.A. Cengel, J.M. Cimbala [7] and reformulate here:

$$\dot{m}_{in} - \dot{m}_{out} = \frac{dm_{CV}}{dt} \tag{1}$$

$$\dot{m}_{in} = 0$$

$$\dot{m}_{out} = (\rho v A)_{out} = \rho \sqrt{2gh}\, A_n \tag{2}$$

Mass of water in the tank at anytime :

$$m_{cv} = \rho v = \rho A_t h \tag{3}$$

Where $A_n = \pi D_n^2/4$ is the cross sectional area of the nozzle, $A_t = \pi D_t^2/4$ is the cross sectional area of the tank, D_n is diameter of the nozzle and D_t is diameter of the tank. Substitute equation (2) and (3) into equation (1):

$$-\rho\sqrt{2gh}\,\frac{\pi D_t^2}{4} = \frac{d(\rho A_t h)}{dt} \tag{4}$$

After rearrange equation (4);

$$dt = -\frac{D_t^2}{D_n^2}\frac{dh}{\sqrt{2gh}} \tag{5}$$

Then taking time integral of equation (5) from t=0 to t=t at which $h = h_0$ to $h = h_2$. Finally, the draining time can be expressed theoretically with the following equation

$$t = \frac{\sqrt{h_0} - \sqrt{h}}{\sqrt{\frac{g}{2}}}\left(\frac{d_t}{d_n}\right)^2 \tag{6}$$

Here, t is the draining time, h_o the initial water level, h the water level at a t time, d_t the tank diameter, d_n the nozzle diameter, and g gravitational acceleration. Equation (6) is derived with the assumption that the flow is irrotational and inviscid.

4. Numerical model

The two phases (liquid and air) flow system is modelled by assuming each phase is a continuous phase. Averaging procedure that is based on the Volume of Fluid Method (VOF), a Eulerian-Eulerian approach, is used in the description of the flow behaviour. In the VOF method, only a single set of governing equations is required. Thus, it's computationally less expensive than the Eulerian model (another type Eulerian-Eulerian approach) that has a number of governing equation, depending on the number of phase.

Volume of fluid. The problem under investigation involves two phases of fluids, i.e., liquid and air. In a finite volume of space, the flow is modelled using the Unsteady Reynolds Averaged Navier Stokes Equations. In Volume of Fluid (VOF) method, α is a function that indicates the relative fraction between liquid and gas in each cell of physical domain. The changes of its properties in each cell can be following [4] :

$$\rho = \alpha\, \rho_l + (\, 1 - \alpha\,)\rho_g$$

$$\mu = \alpha\, \mu_l + (\, 1 - \alpha\,)\mu_g$$

$$\bar{U} = \alpha\, \overline{U_l} + (\, 1 - \alpha\,)\bar{U}_g \qquad (7)$$

Here, ρ is the density and μ is the dynamic viscosity, while the subscripts l and g indicate the liquid and gas. $\bar{U}$ is the mean velocity of fluid and it is transported with the function α . The governing equations (8) is valid for incompressible and newtonian phases (liquid and air). This two phases can be considered as an isothermal fluid system.[4]

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\, \bar{U}) = 0 \; ; \; \textbf{Incomp.} \Rightarrow \frac{\partial \bar{U}_i}{\partial x_i} = 0 \; ; \; i = x,y,z$$

$$\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_j\frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(v\frac{\partial \bar{U}_i}{\partial x_j} - \overline{U'_\iota U'_J}\right) + f_i + \sigma\xi\nabla\alpha$$

$$\frac{\partial k}{\partial t} + \bar{U}_j\frac{\partial k}{\partial x_j} = -\overline{U'_\iota U'_J}\frac{\partial \bar{U}_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j}\left[(v + v_T)\,\overline{U'_\iota U'_J}\,\right)\frac{\partial k}{\partial x_j}\right]$$

$$\frac{\partial \varepsilon}{\partial t} + \bar{U}_j\frac{\partial \varepsilon}{\partial x_j} = -C_{1\varepsilon}\frac{\varepsilon}{k}\,\overline{U'_\iota U'_J}\frac{\partial \bar{U}_i}{\partial x_j} - C_{2\varepsilon}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_T}{\gamma_0}\right)\frac{\partial \varepsilon}{\partial x_j}\right]$$

$$\frac{\partial \alpha}{\partial t} + \nabla.(\alpha\, \bar{U}) + \nabla.\big(\alpha\,(\, 1 - \alpha\,)(\bar{U}_l - \bar{U}_g\,)\big) = 0$$

$$\xi = -\nabla.\left(\frac{\nabla\alpha}{|\nabla\alpha|}\right) \qquad (8)$$

In this equations, σ is surface tension, ξ is the curvature of free surface, ε is the dissipation rate of the turbulent kinematic energy, k is the turbulent kinematic energy, v_T is the turbulent kinematic viscosity , v is the kinematic viscosity, and $C_{1\varepsilon}, C_{2\varepsilon}$ and γ_0 are characteristic constants for the $k - \varepsilon$ model : $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92 and γ_0=1.30.

5. Mesh generation and boundary conditions
Figure-2 shows the computational domain, which is a cylindrical in shape with a height of 450 mm and 90 mm of diameter. The tank is designed with a discharge port of 6mm diameter at the centre of the bottom surface.Top and outlet surfaces are exposed to the atmosphere condition.

The mesh is made ftom 150,000 hexahedral cells. In order to represent this geometry, a three dimensional structured computational grid has been applied. The mesh is constructed using snappyHexMesh, a preprocessing utility available in OpenFOAM. This utility allows a CAD drawing in stl format to be meshed.

Three different patches have been defined in the computational domain. The top surface is defined as Inlet, where the boundary condition is set as shown in Table-1. While the side surface is defined as Walls and the bottom surface is defined as Outlet. The boundary conditions (BC) have been adapted for the current case in Table-1. The definition of each boundary condition is explained in Table-2.

Table 1. Boundary conditions

	Type of Patches		
	Inlet	**Outlet**	**Walls**
U	*pressureInletOutletVelocity*	*zeroGradient*	*fixedValue*
ρ rgh	*totalPressure*	*fixedValue*	*bouyantPressure*
V_T	*calculated*	*calculated*	*nutkWallfunction*
k	*inletOutlet*	*zeroGradient*	*kqRWallFunction*
ε	*inletOutlet*	*zeroGradient*	*epsilonWallFunction*
α	*inletOutlet*	*zeroGradient*	*zeroGradient*

Table 2. Description of boundary conditions

Type	Description of boundary conditions
zeroGradient	Normal gradient of ϕ is zero
fixedValue	Value of ϕ is specified
pressureInletOutletVelocity	Combination of *pressureInletVelocity* and *inletOutlet* . (pressureInletVelocity : When $\mathcal{P}$ is known at inlet, U is evaluated from the flux, normal to the patch)
totalPressure	Total pressure $\mathcal{P}_0 = \mathcal{P} + \frac{1}{2}\rho|U|^2$ is fixed; when U changes, $\mathcal{P}$ is adjusted accordingly
inletOutlet	Switches U and between *fixedValue* and *zeroGradient* depending on direction of U.
bouyantPressure	Sets fixedGradient pressure based on the atmospheric pressure gradient
calculated	Boundary field ϕ derived from other fields
nutkWallfunction	on corresponding patches in the turbulent fields k and nut.
kqRWallFunction	on corresponding patches in the turbulent fields k, q and R
epsilonWallFunction	on corresponding patches in the epsilon field

6. Results and discussions

6.1. *No Generation of Air Core*. Figure 3 shows the process of liquid draining inside a tank for the current case. The results show that there are no generation of air core vortex is formed. It was happened because there was no initial rotation was imparted to the liquids before the liquid was draining. As stated in [6] the air core vortex happens when there is initial rotation. This initial rotation create a dip.
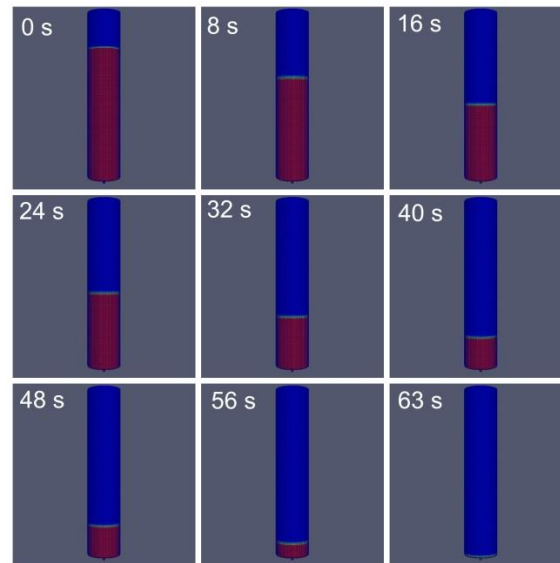

Figure3. Liquid draining process inside tank from the numerical simulation (0-63s)

Table 3. Comparison of drain time for different liquid level.

Water Height (mm)	Park & C.H Sohn Exp.[s]	Park & C.H Sohn Theo.[s]	Park & C.H Sohn Num. [s]	Study Case Num. [s]
350	0.0	0.0	0.0	0.0
330	2.21	2.00	1.98	1.93
310	4.28	4.07	5.14	4.97
290	6.57	6.20	7.81	7.53
270	8.89	8.41	9.96	9.76
250	11.44	10.70	12.52	12.30
230	14.12	13.08	16.54	16.12
210	16.59	15.57	19.04	17.82
190	19.29	18.18	22.26	22.04
170	22.11	20.94	24.17	23.89
150	25.13	23.86	26.29	25.98
130	28.22	26.98	30.65	30.26
110	31.63	30.35	35.03	34.79
90	35.26	34.05	39.98	39.63
70	37.56	38.19	45.87	45.62
50	43.68	42.97	50.21	50.04
30	48.82	48.86	55.38	55.13
10	55.12	57.41	59.86	59.43
0	58.16	69.08	63.18	63.0

6.2. *Comparison of the Draining Time*. Table 3 compares the draining time for liquid level obtained experimentally, numerically, and theoretically by Park and Sohn's. A good agreement between the numerical study of Park and Sohn is obtained.

Figure-4 shows the pattern of the draining time for the current study case compared with the result obtained by Park and Sohn's. The results show that the draining time for the current study, which had done by using numerical method is in a good agreement with Park and Sohn cases.
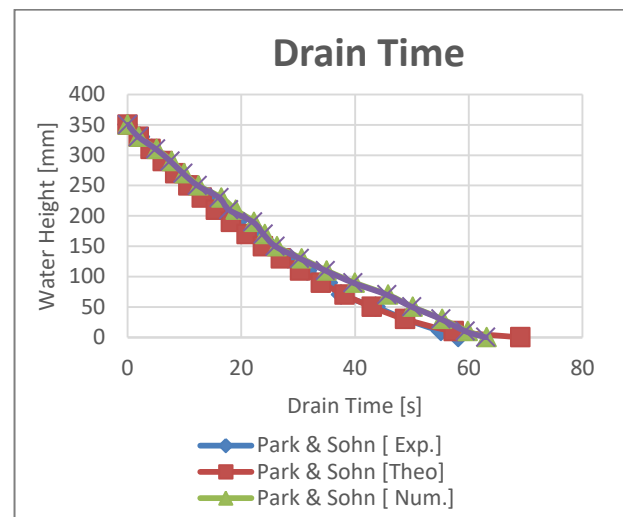
Figure-4. Drain time by experimental, theoretical and numerical methods.

6.3. *Velocity Profile in the Tank*. Figure 5, 6 and 7 show the velocity profile when H=2, H=0 and H=outlet. Here, H represent the water level in the cylindrical tank, where H=outlet is the water level at discharged nozzle. In these 3 figures, the changing pattern of velocity profile seem sequentially at t=0.5s, t=30s and t=60s. We can understand this flow through the tank by using Newton's Law. When water starts to drain out, the flow in the tank is the fastest in the centre compared to the water in contact with the side wall. Here all the water at the same distance from the centre moves down in the tank at the same speed, except the flat layer of water. Besides that, the pressure difference between the outlet and inlet is pushed along the tank. There are also pressure and mass differences along each water level in every second. Large mass or water level correspond to large pressure and consequently smaller values of water velocity in the tank.
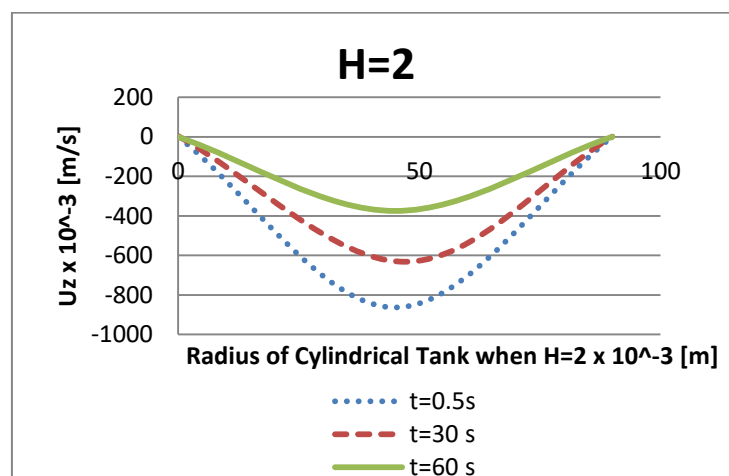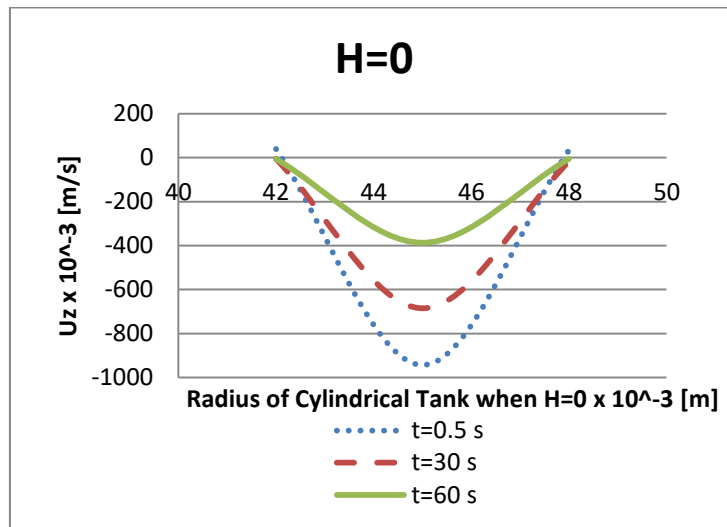

Figure 5. Velocity profile when H=2m

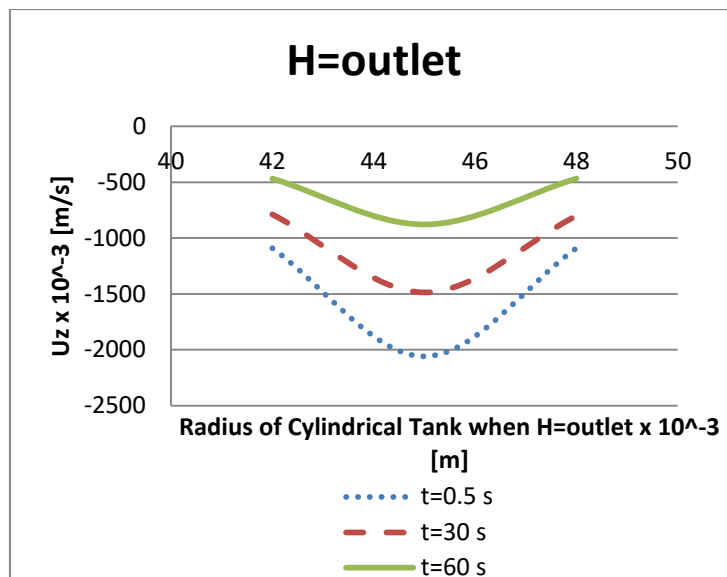Figure 6. Velocity profile when H=0m

Figure 7. Velocity profile when H=outlet

6.4. *Velocity Vector in the Tank*. Figure 8, 9 and 10 show the instantaneous velocity vector in the tank at t=0.5s, t=30s and t=60s respectively. The velocity vector in the outlet varies across the diameter. The size of the arrow corresponds to the velocity magnitude. This arrow represent large velocity magnitude. Figure 9 and Figure 10 show the interesting flow feature. The arrows seem to have a reverse direction, where the flow moving upward. It is called a reverse jet case. The reverse jet occurred because of the strong compression at the bottom centre of the tank. This strong compression initiates high velocity, at which the velocity is maximum. The collection of fluid in this high velocity region (trough of the free surface) makes some of the fluid to move upward [9].
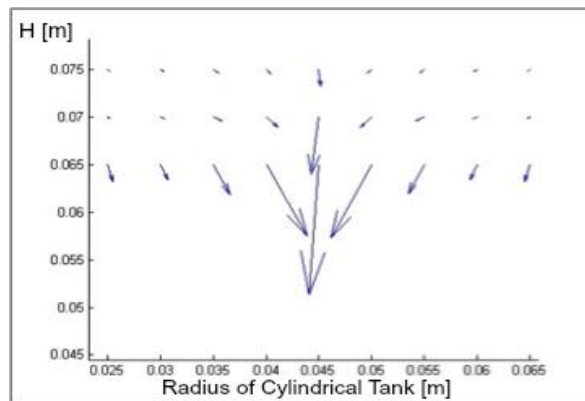
Figure 8. Velocity Vector in the outlet at t=0.5s
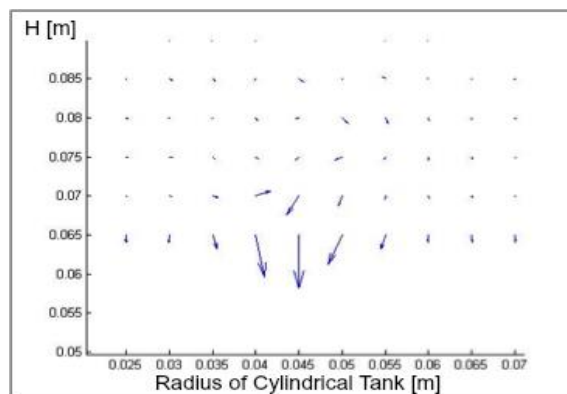
Figure 9. Velocity Vector in the outlet at t=30s
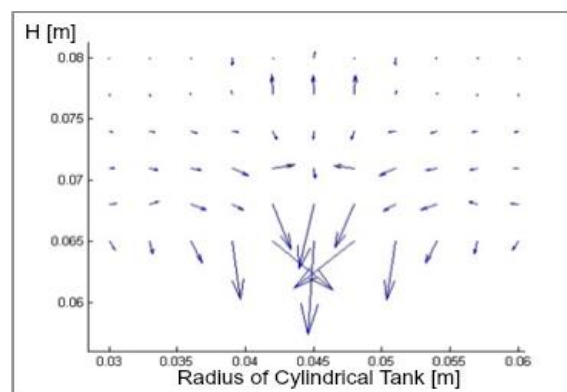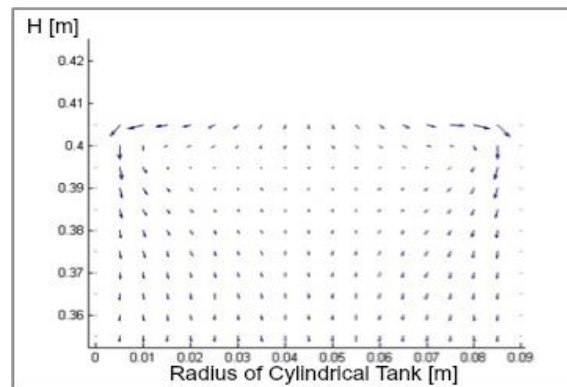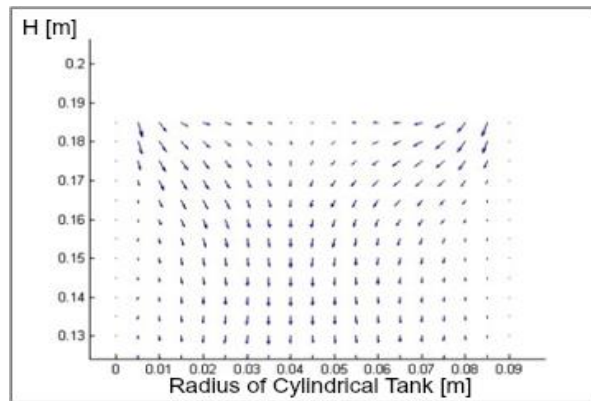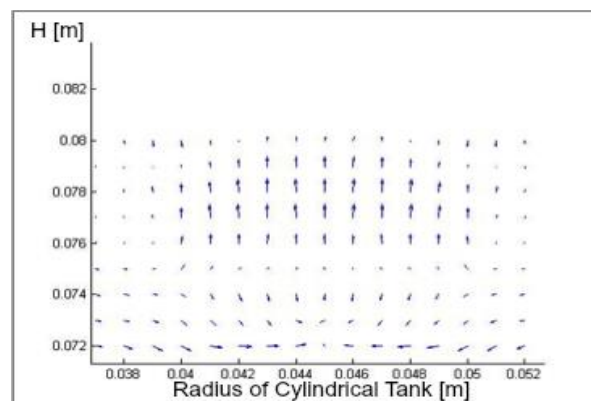

Figure 10. Velocity Vector in the outlet at t=60s

Figure 11. Velocity Vector on the surface at t=0.5s

Figure 12. Velocity Vector on the surface at t=30s

Figure 13. Velocity Vector on the surface at t=60s

Figure-11, 12 and 13 show the instantaneous velocity vector on the surface at t=0.5s, t=30s and t=60s respectively. No reverse jet is observed at t=0.5s and t=30s. The reverse jet is only occurred when the liquid is almost drained completely, i.e. t=60s. This because the top surface of the liquid is near to the outlet.

7. Conclusions

Liquid draining from a tank was numerically investigated by using OpenFOAM framework. Volume of Fluid (VOF) method was applied to trail the air-liquid interface. A good agreement is obtained between the current study and previous available published data.

8. Acknowledgement

REFERENCE

[1] Tamagna U., Subhash T. S. and Thomas M. 2012. Design of Gas-Liquid Separator for Complete Degasing.

[2] Kiyoshi K., Toru K., Hideyo N., Keisuke Y. and Masanobu F. 2010. Experimental and Numerical Simulation Study of Liquid-Propellant Draining from Rocket Tanks.

[3] Sam M., Patnaik B. S. V. and Tharakan T. J. 2014. Numerical Study of Air-Core Vortex Dynamics During Liquid Draining from Cylindrical Tanks.

[4] Ludmila M. R., Mariano I. C. and Enzo A. D. 2014. Hydrodynamic Transient Assessment of a Draining Tank.

[5] II S. P. and Chang H. S. 2011. Experimental and Numerical Study on Air Cores for Cylindrical Tank Draining.

[6] Prateep B., Dheraaj A., Tharakan T. J. and Salih A. 2013. Numerical Studies on Air-Core Vortex Formation During Draining of Liquids from Tanks.

[7] Cengel Y. A. And Cimbala J. M. 2008. Fluid mechanics Fundamentals and Applications, McGraw-Hill Higher Education. pp. 166-189

[8] Adam R., Herve M. and Carol E. 2010. Computational Investigations Into Draininig in an Axisymmetric Vessel.

[9] Qiao N. Z. and Graebel W. P. 1990. Axisymmetric draining of a cylindrical tank with a free surface.

[10] Souders M. J., Huntington R. L., Corner H. G., and Emert F. L. 1938. Performance of Bubble-Plate Columns. Ind. Eng. Chem., 30(1), pp. 86–91.

[11] Barry T. L. and George S. S. 1966. The Formation of a Dip on the Surface of a Liquid Draining from a Tank.

[12] OpenFOAM. 2014. http:www.openfoam.com.