Numerical study on air-core vortex in a dual port cylindrical tank for lower speed of rotations

Sоорaj PV, Joshy PJ
School of Engineering, Cochin University of Science & Technology, Cochin-22, Kerala

Abstract
Rotating cylindrical tank filled with liquid engendered an air core during draining. The extension of the air-core into the drain port reduces the flow area and consequently the flow rate. The ongoing study focuses on the numerical simulation results of such vortex formation in a rotating cylindrical tank with dual ports in which speed of rotation as the variable parameter here by keeping centre to centre distance fixed.

Nomenclature
C = Centre to Centre distance between the dual ports located at an equal distance from the base plate centre (mm)
d = Port diameter (mm)
D = Cylinder diameter (mm)
N = Rotational speed (rpm)

1. Introduction

Vortex is a rotating mass of fluid that forms a vacuum at the centre, draws the liquid into it. Vortex formation at intakes is a significant hydraulic engineering problem. The main reasons considered in the formation of vortex are the inertial distribution, fluid instability. Vortex extending to the free surface can lead to air entrainment that can cause mechanical damage and loss of pump performance. The phenomenon is of useful pertinence in the fuel feed systems in spacecrafts and rockets. The air core has a spiral shape, which perforates through the upper surface of the liquid to the base of the tank. Consequently, the effective cross-sectional area for the drain port becomes narrow because the area parting with the gas phase expands, reducing the draining flow rate remarkably. This paper focuses to assess the results of numerical simulations of the vortex formed during draining in a rotating cylindrical tank.

Numerous experimental investigations have been conducted on air core vortex formed during draining. More than one and a half centuries ago, Rankine et al. (1858) proposed a two-zone model to find out the tangential velocity within a liquid tank with a stable gas-core. Studies on vortexing in propellant tank were first carried out by Abramson et al. (1962) using a cylindrical tank [1]. Experimental investigations reported by Lubin and Springer (1967) have presented empirical relations for the dip formation during liquid draining [2]. The influence of various parameters like initial height of fluid, surface tension, viscosities on dip formation were analysed by them using a cylindrical tank with circular port. Granger et al. (1968) observed large surge velocities and rapid
intensification of the air-core vortex only for particular ranges of drain port diameters when rotational flow currents were present in the liquid column in the tank. Ramamurthi and Tharakan (1995) found out the parameters affecting the gas-core formation and its growth an instantaneous suppression, but the suppressed vortex recovered after stopping the vibration [7]. Ramamurthi and Tharakan (1993) assessed effect of bell-mouthed, stepped and cylindrically shaped drain ports on suppression of air core vortex. Gowda et al. (1996) analysed the gas-core dynamics for liquids that are initially subjected to rigid body rotation. The effect of storage tank cross-section on vortex formed during draining was conducted by Gowda et al. (1996) and compared a storage tank with rectangular and square cross-section with cylindrical tank [6]. Influence of a dish or cup shaped suppressor positioned over a circular drain hole for vortex suppression was also studied. Gowda and Udhayakumar (2005) was able to prevent vortex formation during draining subjected to initial fluid rotation with the introduction of a vane type suppressor above drain port [10]. A variety of multiphase flow simulations, particularly related to gas-entrainment systems have been reported in the literature (Cogan et al. 2008, Li et al. 2008, Mahyari et al. 2010, Satpathy et al. 2013) [8][3]. Numerical studies of tank draining systems have mostly focused on cylindrical tanks with circular cross section and axi-symmetrically placed drain port. Mahyari et al. (2010) performed a CFD study to validate against their own experiments and analytical expressions for the development of effective vortex breakers during tank draining [4]. Robinson et al. (2010) have attempted the validation of their CFD predictions with the experimental work of Lubin and Springer (1967) using commercial CFD tools [9]. Park and Sohn (2011) have carried out systematic experimental and CFD study of gas-core vortex formation [5]. However, liquid draining through the cylindrical tanks is a function of parameters, such as pressurization, drain port size, shape, etc. 

The current study focuses on the numerical simulations and analysis of air-core vortex using dual port in a rotating cylindrical tank keeping centre to centre distance same by taking drain ports diametrically opposite.

2. CFD Methodology

The commercial CFD code ANSYS FLUENT 15, 16 were used to simulate the transient draining of liquid from a cylindrical tank. FLUENT uses the finite volume method to break the problem domain into control volumes.

The time step is managed by Courant number, which was typically chosen between 0.1 and 0.4 depending on the mesh density. The time step sizes were varied accordingly. The CFD simulation involves tracking the gas-liquid interface using the VOF approach implicit scheme was used for time discretisation. The flow field for modeling this flow was assumed to be an axisymmetric swirling type, although the actual draining process accompanies a spiral motion on the free surface. So as to capture the gas vortex, whirling stream and turbulence viably, standard k–ω display for axisymmetric streams is applied, which uses second order differencing schemes for turbulent kinetic energy and dissipation rate. Navier Stokes equations are the governing equations, which can be represented in 3D cylindrical coordinates. A very common case is axisymmetric flow with the assumption of no tangential velocity ($u_\theta = 0$) and the remaining quantities are independent of $\phi$: 
3. Computational Mesh

A three dimensional tetrahedral structured grid was generated to represent test geometry. The number of cells was varied from 300000 to 700000 typically, for the meshes with configuration $d/D$ ratio as 12/130, so as to achieve mesh independent solutions, which is assumed to be achieved when consecutive solutions show a change less than 1% in the outflow mass flow rate. The differencing schemes used were second order upwind for momentum and turbulent kinetic energy. Pressure was discretized using the PRESTO! Scheme, which is similar to the staggered-grid schemes used with structured meshes.

\[
\begin{align*}
\mathbf{r}: & \quad \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \left( \nabla \times \left( \nabla \times \mathbf{u} \right) \right) + \rho \mathbf{g} \\
\mathbf{z}: & \quad \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \left( \nabla \times \left( \nabla \times \mathbf{u} \right) \right) + \rho \mathbf{g}.
\end{align*}
\]

\[
\frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial \mathbf{u}}{\partial r} \right) + \frac{\partial \mathbf{u}}{\partial z} = 0.
\]

Fig. 1: Geometric representation of the model.
Boundary conditions: A large gas to liquid volume ratio was assumed so that the gas pressure above the liquid could be considered as constant. The boundary conditions for modelling the flow are given in Table.

| Model Geometry |
|----------------|
| To numerically investigate the tank draining problem, the models were particularly chosen so that boundary conditions should be properly satisfied. The cylindrical domain was designed with 130 mm diameter and 500 mm height. The drain port diameter was varied from 8mm to 20 mm where centre to centre distance was kept constant as 25 mm. |

4. Numerical results and discussion

VOF model has been shown to give a good account of draining characteristics in a qualitative and quantitative sense in the simulation of liquid draining from cylindrical tank. Generation of air core implies process of air entrainment in the cylindrical domain by park and sohn (2011). The dimple of the free surface at the early stage was due to the rigid body behaviour of the liquid in a tank before the draining was initiated. But sinking of the free surface was found near to the center area. Later air core was completely generated. The air core penetrated from the top face of the free surface to the drain port. The generated air core was sustained throughout draining. The effective exit section was reduced because the air occupied the center portion of the drain nozzle. Thus, the flow rate for draining decreased.

Vortex formation in dual port design

After setting the boundary conditions, respective mesh motion was achieved by setting the drain port diameter constant. Here the influence of speed of rotation on vortex formation was analysed.

Effect of speed of rotation

In the eccentric dual port system initially a dip forms and air entrains in the course of draining. When water was allowed to drain through the dual port rotating cylinder, the pressure just above the drain port was less than that of the atmospheric pressure because the velocity was sufficiently high. This draws the centre of the free surface down, there by forming a dip. The air pushes the vertical column of fluid over the drain port down, which results in air entrainment followed by air-core vortex.
It is obvious that an increase in speed of rotation increases the turbulent velocity thus the vorticity magnitude.

**Case 1: Rotational speed, \(N = 100\) rpm**

**i) Vorticity Magnitude**

![Fig. 2: Contours of vorticity at the dual port drain Outlet when given cylinder rotation, \(N = 100\) rpm](image)

The darker blue shades represent lowest vorticity downstream of the flow. So, it is inferred from the result that vorticity component is weak in the fluid domain. Hence there is no vortex formation. The vorticity magnitude and radial velocity profiles in eccentric dual port system are negligibly small compared to single port design. This is because; in the eccentric dualport system pressure gradient is not sufficient to enter the air core into the drain outlet.

**ii) Radial Velocity**

![Fig. 3: Contours of radial velocity at the dual port drain Outlet when given cylinder rotation, \(N = 100\) rpm](image)
Radial velocity profiles are almost uniformly maintained throughout the whole liquid zone even if the liquid level decreased over time. As there is no significant speed of rotation, there are no flow-driving forces except gravity; therefore, the considerable radial and axial velocities near the drain port are entirely due to the sudden contraction of the flow passage.

However, significant velocity magnitude is observed on the free surface. According to the kinematic condition on a free surface, a particle on a free surface should dip in its position on the free surface. Thus, the low swirl case had small dip on the free surface, and the free surface dimples during nearly all of the draining time. The swirl velocity at the early stage was caused by the sidewall rotation before the draining. Thus, the magnitude was comparatively high near the side wall. As time goes on, the maximum swirl velocity zone moved to the bottom area because of the growth of the boundary layer on the side wall. Such a boundary layer also formed on the bottom wall region, is called the Ekman spiral layer.

Case 2: Rotational speed, $N = 150$ rpm

i) Turbulent intensity

![Contours of turbulent intensity at the dual port drain outlet when given cylinder rotation, $N = 150$ rpm](image)

When cylindrical rotation starts, the fluid particles in the middle of the volume of the tank don't move because none of the agencies which creates turbulence have yet come to play there. Next to the wall however the fluid does circulate because of the viscous stresses. These viscous stresses gradually decrease outwards from the wall and more and more fluid particles circulate. In the end, viscosity brings all the fluid in to a perfect solid body rotation. At this point paradoxically, the viscous forces vanish all together and there is nothing to force further changes in the turbulence.

ii) Velocity Magnitude

The fluid particles near the side wall attain the rotational energy due to no slip effect, so the magnitude of velocity will be at its maximum near the wall. But the velocities of the particles at the centre will be almost zero or negligible and after the passage of time the particles near the centre attains velocity, yet lower compared to the particles near the cylindrical wall. The dip shape of the
free surface at the early stage was due to the rigid body behavior. The depression occurs on the free surface due to the pressure gradient existing between the free surface and the bottom of tank. The pressure force due to the rotation is minimum at the axis of rotation with its magnitude increasing radially outwards. Velocity profiles in eccentric dual port system are negligibly small compared to single port design. This is because; in the eccentric dual port system pressure gradient is not sufficient to enter the air core into the drain outlet.

![Fig. 5: Contours of velocity magnitude at the dual port drain Outlet when given cylinder rotation, N= 150 rpm](image)

**Case 3: Rotational speed, N = 250 rpm**

i) **Vorticity magnitude**

As said, more and more vorticity is induced near the side wall of the cylindrical domain which decays before reaching the central axis of rotation. Hence, the dip formed due to the rigid body rotation fails to form the fully developed vortex and the pressure gradient will not be sufficient to generate the air-core.

![Fig. 6: Counters of vorticity magnitude at the dual port drain outlet when given cylinder rotation, N= 250 rpm](image)
ii) Turbulent intensity

Fig. 7: Counters of turbulence intensity in the domain when given cylinder rotation, N= 250 rpm

Turbulence intensity is also negligible at the drain port. Lower turbulence at the drain ports compared to adjacent walls. So there is no chance for vortex initiation or propagation. But as speed of rotation increases, there observed an increase in turbulent intensity compared to the other remaining cases.

Fig 8 : Counters of turbulence intensity at the dual port drain outlet when given cylinder rotation, N= 250 rpm
5. Summary and Conclusions

Draining of liquid from a cylindrical tank was numerically investigated to study the gas-core vortex formation. VOF flow model was used to track the gas–liquid interface. The role of speed of rotation on the vortex formation was brought out with some numerical results. Speed of rotation is also a parameter which effects the formation of vortex. Preferably given low speed rotations, where vortex formation and its various characteristics are almost constant, which was elaborated with the help of the numerical results. However, for higher speed of rotations, the vortex formation was ineludible.

6. Reference

[1]. Abramson, H. N., and Chu, W. H. (1962)., Some studies of liquid rotation and vortexing in rocket propellant tanks (No. TN D 1212), National Aeronautics And Space Administration Washington DC.
[2]. Lubin, B. T., and Springer, G. S. (1967)., The formation of a dip on the surface of a liquid draining from a tank, Journal of Fluid Mechanics, 29(02), 385-390.
[3]. Satpathy K, Velusamy K, Patnaik B S V and Chellapandi P 2013 Numerical simulation of liquid fall induced gas entrainment and its mitigation Int. J. Heat Mass Transfer 60 335–54
[4]. Mahyari M N, Karimi H, Naseh H and Mirshams M 2010 Numerical and experimental investigation of vortex breaker effectiveness on the improvement in launch vehicle ballistic parameters J. Mech.Sci. Technol. 24 1997–2006
[5]. Park I S and Sohn C H 2011 Experimental and numerical study on air cores for cylindrical tank draining.Int. Commun. Heat Mass Transfer 38 1044–9
[6]. Lakshmana Gowda, B. H., Joshy, P. J., and Swarnamani, S. (1996). Device to suppress vortexing during draining from cylindrical tanks. Journal of spacecraft and rockets, 33(4), 598-600.
[7]. K Ramamurthi and T John Tharakan. Intensification of a vortex during free draining. The Canadian Journal of Chemical Engineering, 73(3):292{299, 1995.
[8]. Cogan S J, Sheard G J and Ryan K 2008 Axisymmetric Hopf bifurcation in a free surface rotating cylinder flow CTAC-2008: Proc. 14th Conf. on Biennial Computational Techniques and Applications , G N Mercer and A J Roberts ANZIAM J. 50 251–65
[9]. Robinson A, Morvan H and Eastwick C 2010 Computational investigations into draining into an axisymmetric vessel J. Fluids Eng. 132 1104–10
[10]. Gowda B H L and Udhayakumar H 2005 Vane-type suppressor to prevent vortexing during draining from cylindrical tanks J. Spacecr. Rockets 42 381–3