NUMERICAL INVESTIGATIONS OF AIR-CORE VORTEX FORMATION IN A ROTATING DUAL PORT CYLINDRICAL TANK

Sooraj PV, Joshy PJ

School of Engineering, Cochin University of Science & Technology, Cochin-22, Kerala

Abstract
An air-core is the vortex flow generated when the rotating liquid in a tank drains into a narrow drain port. The discharge rate of the fluid will be affected due to the gradual reduction in cross sectional area of the drain port. The study is initiated by the necessity to control air-core vortex in space vehicles where similar flow patterns are observed. The prevailing study focuses to dissect the numerical simulation results of such vortex formation in a rotating cylindrical tank with dual ports. The drain port diameter is the variable parameter here by keeping centre to centre distance immutable.

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)

1. Introduction
Vortex is a swirling mass of fluid associated with fluid flows. Vortex formation at intakes is a significant hydraulic engineering problem. The reason for such vortex formation has been ascribed to many factors in the past like changes in inertial distribution, shape of the vessel or fluid instability. The trouble is that the 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 practical relevance in the fuel feed systems in spacecrafts and rockets. Undesirable effects like friction drag, high decibel noise formation in jet engine exhaust etc., occurs due to the vortex formation. 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 ($u_\phi = 0$) and the remaining quantities are independent of $\phi$:

$$
\begin{align*}
  \mathbf{r} : & \quad \rho \left( \frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z} \right) = -\frac{\partial p}{\partial r} + \mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_r}{\partial r} \right) + \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2} \right) + \rho g_r \\
  \mathbf{z} : & \quad \rho \left( \frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_z}{\partial r} \right) + \frac{\partial^2 u_z}{\partial z^2} \right) + \rho g_z, \\
  \frac{1}{r} \frac{\partial}{\partial r} \left( r u_r \right) + \frac{\partial u_z}{\partial z} & = 0.
\end{align*}
$$
3. Computational Mesh

A three dimensional tetrahedral structured grid was generated to represent test geometry. The number of cells were 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.

![Geometric representation of the model.](image)

**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 modeling the flow are given in Table.
Wall | Viscous / No Slip ($u_z; u_r = 0$)
---|---
Inlet | Pressure Inlet (Gauge pressure = 0)
Outlet | Pressure Outlet (Gauge pressure = 0)
Interface | Stress free, Wall adhesion effects

Table 1: Boundary conditions

**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 behavior 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 rotation speed constant. Here the influence of drain port diameter on vortex formation was analysed.

**Effect of drain port diameter**

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, thereby 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 drain port size increases the mass flow rate.

Case 1: drain port size (d) = 6 mm; (d /D) = 0.046

i) Vorticity Magnitude

The results shown above are of the vorticity magnitude. Small vorticity cells were detected near the side wall because of the no-slip effect. The darker blue shades represent lowest vorticity downstream of the flow. Hence, it is inferred from the result that vorticity component was feeble in the fluid domain. Hence the presence of vortex was assumed to be absent.
ii) Turbulence Intensity

![Contours of turbulence intensity at the dual port drain outlet, (d /D) = 0.046](image)

There was no considerable turbulence intensity variation observed in the fluid domain. Lower turbulence at the drain ports compared to adjacent walls. Hence there was no chance for vortex initiation or propagation. The strength of vortex was highest at the region adjacent to the walls (since turbulence intensity lines are maximum there). Small vorticity cells were detected near the side wall because of the no-slip effect. These vorticity cells are well known as the secondary vortex having a circumferential axis. Here the simulations were done by keeping the centre to centre distance constant as, \( C = 25 \text{ mm} \). In the absence of a dip, gas-liquid interface and the liquid beneath it were of zero pressure gradients. The free surface is the zero pressure line, along with another one inside the flow regime at early times when dip was not formed. The absence of gas-core formation may be attributed to the lack of sufficient angular momentum in the liquid especially towards the bottom of the tank. This enabled the liquid layers to move radially inwards, and prevented any accumulation of the axial component of vorticity.
Case 2: drain port size \( (d) = 12 \text{ mm}; \ (d /D) = 0.092 \)

i) **Vorticity Magnitude**

![Vorticity Contours](image)

Fig. 4: Contours of vorticity magnitude at the dual port drain outlet, \((d /D) = 0.092\)

Here vorticity effects are clearly visible from the result shown above. Maximum vorticity was measured at the drain port entry. The swirling liquid near the drain-port lead to the formation of a small low pressure zone, which thus resulted into the deformation of the free surface above it.

ii) **Pressure variation**

The pressure gradient far away from the centerline is almost constant, and there is a large radial pressure gradient in the vicinity of the discharge port. The pressure at the axis of the tank is less than that of its neighbourhood. This leads to the draws-down of the centre of the free surface, thereby forming a dip. The total pressure along the interface varies thereafter, due to the presence of radial pressure gradient. The net pressure force does not pass through the center of mass and the resulting torque changes the vorticity.
Fig. 5: Contours of pressure at the dual port drain outlet, \((d / D) = 0.092\)

Case 3: drain port size \((d) = 20\text{ mm}\); \((d / D) = 0.192\)

i) Vorticity Magnitude

Fig. 6: Contours of vorticity magnitude at the dual port drain outlet, \((d / D) = 0.192\)
Magnitude of vorticity was little bit higher in the vicinity of the drain port compared to the last one. The drain flow rate initially decreased due to the abrupt generation of the air core, and as a result, the liquid in the tank is pushed to the side wall. However, the pushed liquid immediately undergoes a strong vortexing flow with a circumferential axis. In the next instant, this strong secondary vortex near the bottom wall entrains liquid into the drain port. This makes the air core extremely thin at the bottom center. For higher drain port sizes, there will be an effective reduction in the flow area from the outlet.

With further progress in draining, the swirl velocity increases, which induces high centrifugal force on the liquid as it passes through the drain-port. This increase in centrifugal force, creates a further pressure drop in the vicinity of the axis thereby pulling the cover gas through the outlet which was exposed to atmospheric conditions.

i) Relative velocity magnitude

![Image of contours]

Fig. 7: Contours of relative velocity magnitude at the dual port drain outlet, \( \frac{d}{D} = 0.192 \)

As expected when pressure drops near the regions of drains port, swirl velocity increases. There observed a high pressure zone between the two drain ports which in turn stopped the dangling of the vortex swirls.
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 drain port size on the vortex formation was brought out with some numerical results. It was found that, for very small drain-port sizes \(d/D < 0.04\), the dip formation does not result in gas-core vortex. However, for higher drain-port sizes, the vortex formation was ineludible.

5. Reference


[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