Aerospace Science and Technology 32 (2014) 60–65

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Prediction of gas-core vortices during draining of liquid propellants from tanks

Dheeraj Agarwal [a,*], Prateep Basu [b], T. John Tharakan [c], A. Salih [d]

[a] Mechanical Integration Division, ISRO Satellite Centre (ISAC), Bangalore, India
[b] Liquid Propellant Servicing Facilities, Satish Dhawan Space Centre, Sriharikota, India
[c] Flow & Acoustics Division, Liquid Propulsion Systems Centre, Trivandrum, India
[d] Dept. of Aerospace Engineering, Indian Institute of Space Science & Technology, India

ARTICLE INFO

Article history:
Received 28 December 2012
Received in revised form 6 November 2013
Accepted 4 December 2013
Available online 12 December 2013

Keywords:
Draining flow
Gas-core vortex
Propellant tank
Discharge port
CFD
VOF

ABSTRACT

The gas–liquid interface dips during the draining of a liquid through a discharge port of a vessel or a tank. The dip develops into a gas-core vortex which subsequently enters the discharge port. This entry can be either gradual or sudden, depending on the intensification of the rotational flow currents during the draining process. The extension of the gas-core into the drain port reduces the flow area and consequently the flow rate. In liquid propellant stage of rockets, this phenomenon can have an adverse effect on the performance of engine and may lead to under utilisation of the propellant. In this paper, the authors have tried to give theoretical explanations for the formation and intensification of such gas-core vortices, using the results obtained by simulating a gas–liquid flow, drained axisymmetrically from an orifice beneath the liquid. These investigations have been carried out using the commercial ANSYS Fluent code. The flow is modelled using the volume of fluid (VOF) method, which obtains the volume fraction of fluid elements throughout the domain and tracks the gas–liquid interface motion as it descends during drainage. The effect of initial swirl velocity in the liquid and the shape of the container on the formation of these gas-core vortices are studied using the results obtained from numerical simulations to get the physical insight of the involved flow mechanism.

1. Introduction

Of the various dynamic liquid phenomena that may occur in a tank, the rotational motions of liquids are of special interest. These rotational motions are due to the formation of vortices during draining. The reason for such vortex formation has been attributed to many factors in the past like changes in inertial distribution, shape of the vessel or fluid instability. The air core has a spiral shape like a tornado, which pierces through the upper surface of the liquid to the bottom of the tank. Consequently, the effective cross-sectional area for the drain outlet becomes narrow because the area parting with the gas phase increases, reducing the draining flow rate significantly. Typically, in a sump section with an exit pipe leading to a pump, this entrained gas can cause cavitation, vibrations, structural damage and loss of efficiency in turbines or pumps, when it enters in the pipelines. Hence this behaviour at intakes is an important problem in hydraulic engineering. The motivation for this work stems from the propellant management during draining of propellant from tanks, where such air-core vortex is formed during drainage of liquid propellant from tanks to the engine. It is an undesirable phenomenon because it results in premature ingression of pressurant gas into the feed line with propellant still remaining in the tank. Pump fed liquid rocket engines must therefore be cut-off before the pressurant gas enters the pump to prevent cavitation, and possible launch failure.

Lubin and Springer [4] investigated the formation of such air-vortices for the gravity draining of an initially quiescent column of liquid. The height of the liquid column at which the dip enters the drain port is termed as *critical height*. Lubin and Springer [4] gave a simple analytical expression, predicting the height of the liquid surface in the tank at which this dip forms. Zhou and Graebel [10] performed a numerical study by assuming the potential flow model. They found for the one-layer-fluid case that when a dip forms, the computed critical heights agree with Lubin and Springer's [4] analytical solution, but only for moderate values of Froude number. An analytical study of this problem was done by Stokes et al. [9]. Ramamurthi and Tharakan [7] carried out experiments with cylindrical tanks, and studied the intensification of this vortex over time. Different configurations and sizes of drain ports have been shown to strongly influence the height of air-core vortex formed during the draining. The rapidly intensifying vortex, quasi-steady vortex and decaying air vortex were identified by Ramamurthi and Tharakan [6] through their experiments.

* Corresponding author.
E-mail addresses: dheeraj.iist@gmail.com (D. Agarwal), prateepbasu@gmail.com (P. Basu), tharakan12@yahoo.com (T. John Tharakan), salih@iist.ac.in (A. Salih).

1270-9638/$ – see front matter © 2013 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.ast.2013.12.001

Nomenclature

Q	Volume flow rate .... m^3/s	H_c	Critical height .... m
d	Drain port diameter .... m	μ	Dynamic viscosity .... Pa s
D	Vessel diameter .... m	ζ	Vorticity
ρ	Density .... kg/m^3	G	Circulation

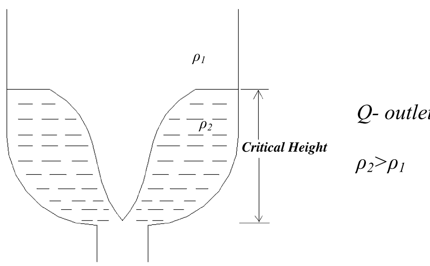

Fig. 1. Schematic of gas-vortex formation during draining.

Robinson et al. [8] simulated the draining of a cylindrical tank, using computational fluid dynamics and validated the predictions using the experimental work of Lubin and Springer [4]. The versatility of volume of fluid (VOF) approach in modelling such transient draining flows was proven by Robinson et al. [8]. Recently, Park and Sohn [5] modelled such draining vortices, concluding that the generation of air-cores is related to both the Ekman suction and the draining effect. Fig. 1 here shows an artist's impression of the axisymmetric draining phenomenon.

In the published work on numerical modelling of vortices, Basu et al. [1] studied the parameters that affect the formation and growth of such vortices for rectangular bottomed tanks. In this paper, the influence of those parameters on the formation and intensification of gas-vortices in a cylindrical flat bottom tank are studied to predict the characteristics of vortex formation. The present investigations have been carried out through simulations done using the commercial ANSYS Fluent 13 code.

2. Problem formulation

The unsteady flow of a viscous liquid through a discharge port located beneath the gas–liquid interface was considered. A two-phase model was employed with three different propellants filling the bottom half of the tank, one at a time, and gaseous nitrogen being the other fluid. The tank top is kept open so that only atmospheric pressure acts on the liquid at the bottom. The effects of earth's rotation and surface tension forces are neglected in this analysis. The axisymmetric equation set for the mass and momentum conservation was selected, similar to Park and Sohn [5]. To find the swirl velocity component in a two-dimensional grid system, the conservation equation for the tangential momentum was applied in addition to the basic axisymmetric governing equation set as follows:

$$\frac{1}{r}\frac{\partial}{\partial r}(ru_r)+\frac{\partial u_z}{\partial z}=0 \tag{1}$$

$$\left.\begin{aligned}
&\left(\frac{\partial u_r}{\partial t}+u_r\frac{\partial u_r}{\partial r}+u_z\frac{\partial u_r}{\partial z}\right)\\
&\quad=-\frac{\partial p}{\partial r}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right)+\frac{\partial^2 u_r}{\partial z^2}-\frac{u_r}{r^2}\right]+\rho g_r\\
&\rho\left(\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+u_z\frac{\partial u_z}{\partial z}\right)\\
&\quad=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\frac{\partial^2 u_z}{\partial z^2}\right]+\rho g_z
\end{aligned}\right\} \tag{2}$$

$$\begin{aligned}
&\frac{\partial}{\partial z}(r\rho u_z u_\theta)+\frac{\partial}{\partial r}(r\rho u_r u_\theta)\\
&\quad=\frac{\partial}{\partial z}\left(r\mu\frac{\partial u_\theta}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r^3\mu\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)\right)-\rho u_\theta u_r
\end{aligned} \tag{3}$$

Here, z is the axial component of the coordinate system, r the radial coordinate component, u_r the radial velocity component, u_θ the tangential velocity (swirl) component, u_z the axial velocity component, ρ is the density of the liquid, and μ is its dynamic viscosity. The CFD simulation, which involves tracking the gas–liquid interface using VOF method, solves the continuity equation for a volume fraction of gas. The common momentum equation shared between the two phases is as follows:

$$\left.\begin{aligned}
&\frac{\partial}{\partial t}(\rho u_i)+u_j\frac{\partial u_i}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}\right)\right)+\rho g_i\\
&\text{where}\quad \mu=F\mu_1+(1-F)\mu_2,\ \rho=F\rho_1+(1-F)\rho_2
\end{aligned}\right\} \tag{4}$$

VOF method is more efficient than the Euler–Euler approach because of the common momentum equation. This simplification is valid only when there is no slip and no interpenetration between the phases.

3. Computational methodology

The computational methodology followed for modelling the flow is the same as that of Basu et al. [1]. Since the physical situation to be modelled here is transient, the Non-Iterative Time Advancement scheme was used, which performs one-iteration per time step and thereby significantly speeds up transient simulations. The time step is controlled by Courant number, which is typically chosen between 0.1 and 0.3 depending on the mesh density. The resulting time steps were of the order of 10^{-5} s. Implicit scheme is used for time discretisation, and the geometrical reconstruction scheme is used to obtain the face fluxes whenever a cell is completely filled with one phase or another. In order to capture the gas vortex, swirling flow and turbulence effectively, standard k–ω model for axisymmetric flows is applied, which uses second order differencing schemes for turbulent kinetic energy and dissipation rate, the central differencing scheme being used for the diffusion of turbulence energy.

3.1. Computational mesh

A two dimensional unstructured grid is generated to represent test geometry. The number of cells is varied from 2000 to 60 000 typically, for the meshes with configuration d/D ratio as 1/6, 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. Unstructured grids are used to capture the high gradients generated at the edges due to curved bottom. Fig. 2 shows a typical mesh used for the simulations.

3.2. Boundary conditions

The boundary conditions used for modelling the flow are given in Table 1.

A large gas to liquid volume ratio was assumed so that the gas pressure above the liquid could be considered as constant.