Simulation studies on cooling of cryogenic propellant by gas bubbling

Pavitra Sandilya\textsuperscript{a,\textasteriskcentered}, Pritam Saha\textsuperscript{a}, Sonali Sengupta\textsuperscript{b}

\textsuperscript{a}Cryogenic Engineering Centre, Indian Institute of Technology Kharagpur, Kharagpur 721302, India
\textsuperscript{b}Department of Chemical Engineering, Indian Institute of Technology Kharagpur, Kharagpur 721302, India

Abstract

Injection cooling was proposed to store cryogenic liquids (Larsen et al. [1], Schmidt [2]). When a non-condensable gas is injected through a liquid, the liquid component would evaporate into the bubble if its partial pressure in the bubble is lower than its vapour pressure. This would tend to cool the liquid. Earlier works on injection cooling was analysed by Larsen et al. [1], Schmidt [2], Cho et al. [3] and Jung et al. [4], considering instantaneous mass transfer and finite heat transfer between gas bubble and liquid. It is felt that bubble dynamics (break up, coalescence, deformation, trajectory etc.) should also play a significant role in liquid cooling. The reported work are based on simple assumptions like single bubble, zero bubble deformation, and no inter-bubble interactions. Hence in this work, we propose a lumped parameter model considering both heat and mass interactions between bubble and the liquid to gain a preliminary insight into the cooling phenomenon during gas injection through a liquid.

Keywords: Injection cooling; Cryogenic storage; Heat and Mass transfer; Bubble dynamics.

1. Introduction

Injection cooling offers a cost effective means of storing cryogenic liquids for long time. The method is operationally simple. The cooling is obtained due to the evaporation. When an insoluble gas passes through a liquid medium in the form of bubble the evaporation of liquid takes place due to the difference between the saturation pressure of the liquid and partial pressure of the evaporating component in the gas bubble. The cooling is obtained due to the simultaneous heat and mass transfer from the liquid medium to the gas medium. Cooling continues until the partial pressure of the evaporating component inside the bubble equals the saturation pressure of the liquid. Enough contact time as well as contact surface area are to be provided to effect greater and faster cooling. These are dictated by the bubble size and shape and two-phase flow dynamics, which in turn depends on the physical configuration of the gas-liquid system and the operating conditions. In the literature, there are only a few studies on liquid cooling by gas injection, probably because of their limited cooling capacity and hence limited application.

\textsuperscript{\textasteriskcentered} Dr. Pavitra Sandilya. Tel.: +91 - 3222 - 283596.
E-mail address: profsandilya@gmail.com
Larsen et al. [1] observed cooling in LOX-He, LN₂–He, and LOX–N₂ systems. Schmidt [2] proposed "Randolph and Vaniman" method to determine the cooling rate in case of injection cooling. Subcooling of about 5 °C over a period of about 5 hours was demonstrated. Rudimentary theoretical analysis based on lumped parameter modelling has been presented by Cho et al. [3]. The lumping was done for the overall storage tank without considering the temperature distribution inside the storage vessel and the bubble dynamics. In this present study finite heat and mass transfer between the liquid and the gas are considered for modelling and simulation.

2. Development of the mathematical model

A two dimensional model has been developed to study the cooling of cryogenic propellants by gas bubbling. The model considers finite heat and mass transfer between the liquid and gas phase along with the bubble motion and distribution. The liquid column is taken to be rectangular in shape. A sparger or submerged orifice is considered to inject the gas into the liquid column. The Schematic representation of injection cooling system is shown in Fig. 1. The following assumptions has been made for modelling the injection cooling system.

- Two dimensional mathematical model is considered.
- Injected gas is insoluble in liquid.
- Zero heat inleak from the surroundings to the column.
- Constant evaporation rate from the liquid to gas.
- Constant specific bubble interfacial area.

The present study has been carried out in the bubbly region only. Also the volume of fluid (VOF) method is used to determine the gas-liquid interfacial behaviour and characteristics. Both gas and liquid are considered as incompressible and Newtonian fluid.

2.1. The volume of fluid (VOF) method

The volume of fluid method was proposed by Hirt and Nicholas [5] to track or capture the interface in a two phase flow system. The formulation of the volume of fluid (VOF) rely on the volume fraction of the fluid. The summation
of all the phases present in a multiphase flow system is taken as unity. If the value of volume fraction is known at all the location of a computational grid then the fluid properties as well as the variables can be represent in terms of volume average values. Also all the phases share the fields of variables. Equation 1 is the volume of fluid equation representing the continuity equation for volume fraction of the $k^{th}$ phase in a multiphase flow system.

$$\frac{\partial \alpha_k}{\partial t} + \bar{u} \nabla \alpha_k = \frac{S_m}{\rho_k}$$  \hspace{1cm} (1)

Equation 1 can be written for gas-only cell, liquid-only cell and the cell containing gas-liquid interface. The source term $S_m$ in Eq. 1 can be defined as the mass source and it is zero for the gas-only and liquid-only cells.

$$\sum_{k=1}^{n} \alpha_k = 1$$ \hspace{1cm} (2)

Equation 2 is the summation equation for volume fractions of all the phases present in a multiphase flow system. In a typical gas-liquid two phase flow system if the volume fraction of the gas phase is taken as $\alpha_g$, then the density and the viscosity in each computational cell can be represented by Eq. 3 and Eq. 4 respectively.

$$\rho_{lg} = \alpha_g \rho_g + (1 - \alpha_g) \rho_l$$ \hspace{1cm} (3)

$$\mu_{lg} = \alpha_g \mu_g + (1 - \alpha_g) \mu_l$$ \hspace{1cm} (4)
2.1.1. Momentum equation

The general form of momentum equation used in volume of fluid method (VOF) is given by Eq. 5. The velocity field throughout the domain is calculated from this momentum equation and is shared by all the phases in a multiphase flow system. The term \( \mathbf{F} \) in Eq. 5 accounts the effect of surface tension force.

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \left[ \mu \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) \right] + \rho \mathbf{g} + \mathbf{F} \tag{5}
\]

Equation 5 can be written for different computational cells containing only gas, only liquid and the gas-liquid interface.

2.1.2. Energy equation

Equation 6 depicts the energy equation used in volume of fluid (VOF) method. The temperature field obtained by solving the energy equation is shared among different phases like momentum equation. The term \( E \) in Eq. 6 is the mass averaged energy defined by Eq. 7.

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\mathbf{u} (\rho E + p)) = \nabla \cdot \left( k_c \nabla T \right) + S_h \tag{6}
\]

\[
E = \frac{\sum_{k=1}^{n} \alpha_k \rho_k E_k}{\sum_{k=1}^{n} \alpha_k \rho_k} \tag{7}
\]

Where, \( E_k \) depends on the temperature and the specific heat of the phase. The term \( S_h \) in Eq. 6 is the volumetric heat source used to calculate heat transfer between phases during evaporation. It may be obtained by multiplying latent heat with the mass transfer rate and is zero for the gas-only and the liquid-only cells. Equation 6 can also be written for each computational cell containing only gas, only liquid and the gas-liquid interface.

The rate of mass transfer during evaporation can be calculated by Eq. 8. The coefficient \( c \) in Eq. 8 can be described as relaxation time.

\[
\dot{m}_{evp} = c \alpha_g \rho_g \left( T_g - T_{sat} \right) \tag{8}
\]

2.2. Initial and boundary conditions

The initial and boundary conditions used in mathematical modelling are given as follows

At \( x = 0, T_l = T_{l0} \) and \( T_g = T_{g0} \); At \( x = 0 \) and \( x = R_{col}, \frac{\partial T_i}{\partial x} = 0 \)

At \( y = 0, T_g = T_{g, in} \) and \( \frac{\partial T_i}{\partial y} = 0 \); At \( y = L_{col}, \frac{\partial T_i}{\partial y} = 0 \)

3. Results and Discussion

Numerical simulation of injection cooling was carried out using ANSYS® Fluent 15 Academic version. The height and edge length of the liquid column were taken as 80 cm and 10 cm respectively. Step size of 1 mm was used in the VOF model of Fluent to simulate the two phase flow system. Pressure based solver was chosen for the simulation. Liquid nitrogen (LN2) was taken as the primary or continuous phase and gaseous Helium (GHe) was taken as the secondary or dispersed phase. The operating conditions of the numerical simulation are given in Table 1.
Table 1. Operating conditions for numerical simulation.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Symbols</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liquid temperature</td>
<td>$T_l$</td>
<td>77.3 K</td>
</tr>
<tr>
<td>Inlet Gas temperature</td>
<td>$T_g$</td>
<td>77.3 K</td>
</tr>
<tr>
<td>Column pressure</td>
<td>$P$</td>
<td>1 atm</td>
</tr>
<tr>
<td>Liquid density</td>
<td>$\rho_l$</td>
<td>806.08 kg/m$^3$</td>
</tr>
<tr>
<td>Gas density</td>
<td>$\rho_g$</td>
<td>0.1625 kg/m$^3$</td>
</tr>
<tr>
<td>Volume fraction of gas at inlet</td>
<td>$\alpha_g$</td>
<td>1</td>
</tr>
<tr>
<td>Liquid velocity</td>
<td>$u_l$</td>
<td>0 m/s</td>
</tr>
<tr>
<td>Gas velocity</td>
<td>$u_g$</td>
<td>0.1, 0.25 and 0.30 m/s</td>
</tr>
<tr>
<td>Surface tension</td>
<td>$\sigma$</td>
<td>0.00885 N/m</td>
</tr>
</tbody>
</table>

The surface tension force was included to study the bubble deformation. Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm was used to solve the pressure-velocity coupled equations. PRESTO! scheme was used to calculate the pressure field. Second order upwind scheme was incorporated to discretize the momentum and energy equation. Geo-Reconstruct scheme was used to discretize and solve the volume fraction equation.

Numerical simulation was carried out at gas velocities of 0.1, 0.25 and 0.3 m/s. Fig. 2 (a) and (b) depict the contour plots of volume fraction for liquid nitrogen at 5.0 and 10.0 seconds for a gas velocity of 0.1 m/s. The contour plots of volume fraction show the size and distribution of the helium gas bubbles in the liquid nitrogen column. Fig. 2 (c) and (d) show the contour plots of temperature distribution of liquid nitrogen and gaseous helium bubbles throughout the column at 5.0 and 10.0 seconds for a gas velocity of 0.1 m/s. The variation of temperature for liquid nitrogen with time at different gas velocities was plotted in Fig. 3. From the figure it is seen that cooling of up to 0.003271...
K, 0.002419 K and 0.000626 K was obtained in 3.5 seconds at gas velocities of 0.10, 0.25 and 0.30 m/s respectively. Rate of cooling varies depending upon the gas velocity. It is also seen that at low gas velocity higher cooling rate was obtained. This may be due to the large contact time between the gas bubble and liquid as the bubble residence time increases with the decreasing gas velocity. However, more rigorous studies are being carried out to account for the bubble dynamics.

![Graph showing variation of liquid temperature with respect to time at different gas velocities.](image)

**Fig. 3.** Variation of liquid temperature with respect to time at different gas velocities.

### 4. Conclusion

A preliminary study on the injection cooling of cryogenic liquid was carried out numerically using liquid nitrogen and gaseous helium. The study indicates that the cooling rate of the liquid depends on the gas velocity. Future studies of injection cooling are aimed at studying the effects of the shape and size of the bubbles, and also the variable evaporation rate along with two phase hydrodynamics.

### References


