2012

Pressure Drop of Two-Phase Flow Through Horizontal Channel with Smooth Expansion

Nurdil Eskin
eskinn@itu.edu.tr

Emrah Deniz

Follow this and additional works at: http://docs.lib.purdue.edu/iracc

http://docs.lib.purdue.edu/iracc/1255

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html
Pressure Drop of Two-Phase Flow through Horizontal Channel with Smooth Expansion

Nurdil ESKIN\textsuperscript{1}\textsuperscript{*}, Emrah DENIZ\textsuperscript{2}

\textsuperscript{1}Istanbul Technical University, Faculty of Mechanical Engineering, Istanbul, Turkiye
eskinn@itu.edu.tr

\textsuperscript{2}Istanbul Technical University, Faculty of Mechanical Engineering, Istanbul, Turkiye
emrah.deniz@itu.edu.tr

\textsuperscript{*} Corresponding Author

ABSTRACT

Industrial systems at where gas-liquid flow is extensively used generally have complex geometries including singularities like expansion, contraction, bends and orifices. Thus two-phase flow characteristics, such as pressure drop, through these singularities should be identified in order to be used to design these systems. In this study, pressure drop of air-water flow through horizontal channel having smooth expansion is investigated experimentally. Internal diameter of the channel expands from 40 mm to 50 mm. Hence area ratio ($\sigma$) of the singularity is calculated to be 0.64. Flow rate for water is constant at 3 l/s while that for air is taken as 30, 50 and 60 l/min. In the experiments, effects of volumetric void fraction and injector type on two-phase pressure drop are examined. Flow under consideration is also modeled via commercial software, GAMBIT (v. 2.3.16) and ANSYS (v. 12). Experimental results are then compared with those obtained by the numerical simulation.

1. INTRODUCTION

Gas-liquid two-phase flow takes the leading role in applications including evaporation and/or condensation, flow through cross-sectional area change (i.e. nozzle, diffuser) and sudden pressure drop (i.e. flashing). These systems normally have complex geometries composed by singularities like expansion, contraction, bends, orifices, etc. Thus two-phase flow characteristics, such as pressure drop, phase distribution, two-phase pattern, through these singularities should be identified in order to be used in designing of the system. Besides that, pressure drop relations with respect to mass flow rate for two-phase flows through these singularities are important for the control and operation of such industrial plants as chemical reactors, power generation units, refrigeration apparatuses, oil wells, and pipelines. The proper design and control of such systems need reliable predicting procedures to evaluate local pressure losses as well as upstream and downstream effects.

Two-phase circuits are mainly composed of various singularities that provoke important flow modifications. Among these singularities, the sudden expansion and contraction have been analyzed experimentally and theoretically in several studies published in the literature. Ahmed et al. (2008) investigated two-phase flow through a horizontal sudden expansion, experimentally. They used air and oil as fluids, two different area ratios and concluded that upstream flow pattern and area ratio are dominant on phase distribution and developing length after expansion. Ahmed et al. (2007) developed an analytical formulation for the pressure recovery of two-phase flow across a sudden expansion based on the experiments performed using oil-air two-phase flows. Similar studies have been performed by Aloui et al. (1999) for bubbly flow in an axisymmetric sudden expansion. In this study the singular pressure drop has been obtained as a global formulation. Fossa and Guglielmini (1998) experimentally investigated the void fraction distributions in horizontal pipes with sudden area contraction. They measured the
instantaneous electrical impedance of the air-water mixture in order to obtain the cross-sectional average of void fraction along the test pipes. They also investigated experimentally pressure drop through singularities such as thin and thick orifices in their study in 2002 (Fossa and Guglielmini, 2002). Bertola (2002, 2004) used single-fiber optical probes to investigate the time-averaged structure of gas–liquid horizontal flow through a sharp-edged sudden area contraction. It is stated that the distribution of the gas and the liquid close to the pipe contraction are quite different from that of a homogeneous flow, so that the commonly used void fraction and pressure drop correlations are unable to predict the flow behavior near a singularity. It is also concluded that the sudden contraction considerably affects the gas distribution in both upstream and downstream pipes, and its effect grows more and more as the flow approaches the singularity. Delgado-Tardaguila (2008) studied on channel having divergence and convergence type singularities and proposed a modified correlation for static pressure change. Water and air at atmospheric conditions are used as fluids and effects of liquid and gas flow rate, volume fraction of the gas and distance from the singularity, on pressure change are investigated. A comparison of experimental and numerical profiles with handbook solution is found to be in good agreement.

Once literature is surveyed, it can be seen that most of the studies investigating two-phase flow characteristics through singularities commonly considered sudden changes like sudden expansion or sudden contraction. Furthermore most of the experimental data refer to restrictions having an area ratio less than 0.5. In the presented work, pressure drop along adiabatic air-water flow through horizontal channel having smooth expansion with an area ratio of 0.64 is researched experimentally and numerically. Pressure transducers measuring gauge and differential pressure are employed to determine the values in the experiments. Experimental data is also compared with the results of numerical modeling performed via commercial software, GAMBIT (v. 2.3.16) and ANSYS FLUENT (v. 12). The local pressure drop has been evaluated for upstream, smooth expansion singularity and downstream of the horizontal adiabatic channel and, compared with the experimental data.

2. EXPERIMENTAL SETUP AND CONDITIONS

Schematic of the experimental facility is given in Figure 1. Water at atmospheric conditions is taken from a tank of 300 liters by a pump having maximum capacity of 18 m³/h. Outlet of the pump, water is filtered up to 5 µm, exposed to gauge pressure measurement and then sent to the electro-magnetic type flow meter for flow rate (i.e. 3 l/s) measurement. Meanwhile, air is compressed to 4 bar absolute pressure, filtered from the moisture and the particles and measured by a rotameter to determine the flow rate (i.e. 30, 50 and 60 l/min). Compressed air is then injected to the water flow by an injector at where the two-phase flow is obtained. Air-water flow flowing through the test section is subject to the measurements by means of pressure transducers. All pipes in the facility are made of transparent acrylic in order to observe the flow.

![Figure 1: Schematic of the facility](image-url)
Geometrical details and location of the pressure transducers on the test section are shown in Figure 2. There are three pipes named upstream pipe, singularity pipe and downstream pipe with prescribed lengths and diameters. Enlarging section is in the singularity pipe and will be called as singularity section. Black bars stand for positions at where the pressure measurements are performed.

![Figure 2: Detailed scheme of the test section](image)

Four differential pressure transducers, having certificated calibration, are utilized to measure the pressure drop at injector and along upstream, singularity and downstream pipes. Measurement range and total accuracy of the transducers are 0-10 kPa and ± 0.25 %, respectively. Data given by the transducers are stored in the computer via a data logger (National Instruments USB-6229) and a virtual instrument (.vi) composed in LabVIEW (v. 8.5). Kolmogorov time-scale for single-phase water flow with 3 l/s, through the test section is estimated by the correlations written below, and found to be between 0.0035 and 0.0075 seconds which give the frequency values between 285 and 134 Hz. Thus, acquisition frequency for the pressure measurements is determined to be 1000 Hz. by taking Nyquist frequency into account.

\[
\tau_\eta = \left( \frac{\nu}{\varepsilon} \right)^{1/2} \\
\varepsilon = c_\mu \varepsilon^{3/4} k^{3/2} l^{-1} \text{ where } c_\mu = 0.09, l = 0.07D \\
k = \frac{3}{2} \left( U l \right)^2 \text{ where } I = 0.16 \text{ Re}^{-1/8} \\
\nu = \mu / \rho
\]

Table 1: Operational conditions and geometrical details of the test section

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Gas phase / Liquid phase</th>
<th>Air / Water</th>
</tr>
</thead>
<tbody>
<tr>
<td>L_1 = 1500 mm</td>
<td>Mass flow rate for gas (m_{air})</td>
<td>0.0006125, 0.001021 and 0.001245 kg/s</td>
</tr>
<tr>
<td>L_2 = 330 mm</td>
<td>Mass flow rate for liquid (m_{water})</td>
<td>2.99 kg/s</td>
</tr>
<tr>
<td>L_3 = 382 mm</td>
<td>Volumetric void fraction (\beta)</td>
<td>13.87 %, 21.74 % and 25.31 %</td>
</tr>
<tr>
<td>D_1 = 40 mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>D_2 = 50 mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>\delta = 9°</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

International Refrigeration and Air Conditioning Conference at Purdue, July 16-19, 2012
During the experiments, flow rate for water is kept constant at 3 l/s while that for air is taken as 30, 50 and 60 l/min. Thus volumetric void fraction ($\beta = \dot{Q}_{\text{air}} / \dot{Q}_{\text{total}}$) of the cases are estimated to be 13.87 %, 21.74 % and 25.31 %, respectively. Expansion angle ($\delta$) is also kept constant at 9º for all cases. Operational conditions and the geometrical details of the test section can be seen in Table 1.

Two injectors with different designs are employed in the experiments, in order to examine the effect of bubble formation on pressure drop and flow pattern. To be able to distinguish, the injectors will be called according to distribution of their holes such as circular injector and in-line injector. Circular injector has 16 holes with 2 mm diameter while there are 32 holes of 1 mm diameter located on three horizontal rods of in-line injector. Illustration of the injectors is given in Figure 3.

![Circular and In-line Injectors](image)

**Figure 3:** Circular (left) and in-line (right) injector used in the experiments

### 3. NUMERICAL MODELING

In order to calculate the pressure drop at each cross section along the pipe theoretically, velocity field and void fraction distribution of the phases have to be defined in detail. In this study, it is done by using commercial software based on finite volume method. Construction of the numerical domain and calculation are performed by GAMBIT (v. 2.3.16) and ANSYS FLUENT (v. 12), respectively. Since air-water flow is obtained by injecting air to the water flow via an injector in the experiments, as shown in Figure 1, it can be stated that the inlet conditions of two-phase flow through the test section (i.e. upstream, singularity and downstream pipes) are determined by the outlet conditions of the injector. Therefore numerical analysis of the flow will be performed in two steps including modeling of the flow through the injector and, then through the test section. Shape and diameter of the bubbles in the flow are affected by design of the injector and forces (i.e. drag and lift) between the phases. Break-up and/or coalescence between the bubbles also modify the diameter, instantly. In the present study, a constant diameter value is estimated for the bubbles at the injector outlet thanks to the correlation written in literature (Kunii, 1991). Also, it is assumed that there is no break up/coalescence between the bubbles in the flow.

$$V_{\text{bubble}} = 1.138 \left( \frac{v}{N} \right)^{6/5} g^{3/5}$$

Here, $V_{\text{bubble}}$ is the volume of an individual bubble, $v$ is the total volumetric flow rate of air through the injector, $N$ is number of the holes in the injector and $g$ is the gravitational acceleration. Diameter is then extracted from the bubble volume, assuming that the bubble is spherical.
3.1 Modeling of the Flow through the Injector

Numerical modeling of the circular injector will be presented solely in this study. Numerical geometry constructed for the injector is given in Figure 4. Here, holes of the injector are represented by means of rods with 100 mm length which inserted to a pipe created in 3D. In order to ease to have structural mesh, rods are constructed with square cross-section having 2 mm of hydraulic diameter. According to the simulation, air flows through these rods while water enters the pipe from the remaining cross-section of the inlet and, two-phase flow to be considered is obtained just after the outlet of the rods where the phases infiltrate each other. At the end of each simulation for different air flow rates, phase velocity and void fraction profiles obtained at that location are extracted from the simulation in order to be introduced as the inlet condition for the analysis of the flow through the test section.

Figure 4: Numerical injector

94,969 nodes are considered to be adequate for the injector simulation according to the grid independency examination performed by testing several domains with different node densities. Mesh structure of the domain is given with the water-inlet region shown in Figure 5. In ANSYS FLUENT (v.12), boundary conditions like “velocity inlet” is taken as the inlet condition for water and air, while “interior” and “outflow” are employed as the outlet condition for the rods and the pipe, respectively. Eulerian (Dispersed) multiphase model is used for the analysis. Reynolds Stress Model is utilized to model the turbulent flow. Standard Wall Functions are operated for near wall treatment. Phase Coupled SIMPLE scheme for pressure-velocity coupling, Green-Gauss Cell Based option for gradients and Second Order Upwind for spatial discretization are chosen.

Figure 5: Mesh structure at water-inlet of the injector
3.2 Modeling of the Flow through the Test Section

Second step of the analysis is modeling of two-phase flow through test section having three regions called upstream, singularity and downstream pipe. It is performed separately by taking the outlet conditions of the former pipe as the inlet conditions of the following. Numerical domain of the system is constructed in 3D and symmetrical with respect to y-axis. Node density at cross-section of the pipes are adjusted to be equal to that of the injector in order to avoid numerical errors due to interpolation thus 348,558 nodes are used for upstream and downstream pipes while 119,274 for those of the singularity pipe. y-directional node density for the upstream pipe and mesh structure for the enlarging section are illustrated in Figure 6.

![Figure 6: Mesh structure for upstream-inlet and enlarging section](image)

Referring to the study in the literature (Deniz and Eskin, 2011), Reynolds Stress Model is chosen to model turbulence. As in the injector simulation, Eulerian (Dispersed) multiphase model is used for the analysis. Standard Wall Functions are utilized for near wall treatment. Phase Coupled SIMPLE scheme for pressure-velocity coupling, Green-Gauss Cell Based option for gradients and Second Order Upwind for spatial discretization are chosen.

4. RESULTS AND DISCUSSIONS

Experimental results and comparison of them with those of numerical are given in the following figures. In these figures, "singularity section" stands for the enlarging portion of the "singularity pipe" which has length of L₃ as shown in Figure 2.

4.1 Experimental Results

Pressure drop values with respect to volumetric void fraction of the flow are given in Figure 7 and Figure 8 for circular and in-line injector, respectively.

![Figure 7: Pressure drop with respect to axial direction when circular injector is used](image)
From the figures it can be seen that the effect of volumetric void fraction on pressure drop is significant at the injector rather than the other regions because of the jet effect of the air while being injected to the water flow. Pressure recovery due to the enlarging cross-section can be analyzed thanks to the values belonging to the singularity section for both injectors. It can be stated that the recovery is higher while the in-line injector is used. However, pressure drop downstream the singularity can be lowered by using the circular injector. Since it is observed that flow pattern turns out to be slug flow as volumetric void fraction increases and, there is flow separation just after the expansion, it has been hard to make a certain conclusion about effect of volumetric void fraction on two-phase pressure drop for the regions investigated.

![Figure 8: Pressure drop with respect to axial direction when in-line injector is used](image)

Effect of injector design on two-phase pressure drop is given in Figure 9. It can be concluded that using circular injector decreases the pressure drop along the test section although the difference is not much between that of in-line.

![Figure 9: Effect of injector on pressure drop](image)

Numerical and experimental values of two-phase pressure drop along upstream, singularity and downstream pipes are compared according to the volumetric void fraction of the flow with circular injector, in Figure 10 and Figure 11.
As a result of the comparison shown in the figures, it can be declared that the numerical modeling well predicts the pressure drop at lower values of volumetric void fraction. Divergence increases with increasing $\beta$ because assumptions considered in the modeling (i.e. constant bubble diameter and no breakup/coalescence between the bubbles) are lack of defining the effects caused by the bubble interactions that increase with $\beta$, either. A novel modeling taking the bubble interactions into account would give more reasonable results.

Figure 10: Comparison of numerical and experimental values of two-phase pressure drop for $\beta = 21.74\%$.

Figure 11: Comparison of numerical and experimental values of two-phase pressure drop for $\beta = 25.31\%$.

Numerical pressure drop values with respect to the volumetric void fraction of the flow with circular injector are plotted in Figure 12. It can be concluded that the pressure drop increase with increasing $\beta$.
5. CONCLUSIONS

Pressure drop of adiabatic air-water flow through horizontal pipe with smooth expansion is experimentally investigated in this study. Measurements are carried out by means of differential pressure transducers along four regions named injector and upstream, singularity and downstream pipes. Two differently-designed injectors (i.e. circular and in-line) are utilized in the experiments in order to examine the effect of bubble formation on two-phase pressure drop. Numerical modeling of the flow is also performed via commercial software based on control volume method and the results are compared with those obtained experimentally. Conclusions can be listed as follows.

- Effect of volumetric void fraction on pressure drop is significant at the injector rather than the other regions because of the jet effect of the air while being injected to the water.
- Pressure recovery due to the expansion is higher while the in-line injector is used. However, pressure drop downstream the singularity can be lowered by using the circular injector.
- Due to the variation of the flow pattern and to the flow separation observed downstream the expansion, effect of volumetric void fraction on two-phase pressure drop is hardly concluded.
- Employing circular injector results lower pressure drop along the test section.
- Numerical modeling well predicts the pressure drop at lower values of volumetric void fraction. Assumptions considered in the modeling (i.e. constant bubble diameter and no breakup/coalescence between the bubbles) are disable to define the effects caused by the bubble interactions, properly, thus discrepancy increases with increasing $\beta$.
- A novel model taking bubble interactions into account can be taken as the future work of the study.

NOMENCLATURE

- $D$: diameter of the pipe (mm)
- $L$: length of the pipe (mm)
- $V$: volume ($cm^3$)
- $m$*: mass flow rate (kg/s)
- $\dot{Q}$*: volumetric flow rate ($m^3/s$)
- $N$: number of the injector holes
- $g$: gravitational acceleration ($cm/s^2$)
- $y$: distance along radial direction (mm)
- $k$: turbulence kinetic energy ($m^2/s^2$)
- $l$: turbulence length scale (m)
- $U$: free-stream velocity (m/s)

* $\dot{m}$ and $\dot{Q}$ are used interchangeably for mass and volumetric flow rates.
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{\mu}$</td>
<td>$k - \varepsilon$ model parameter</td>
<td>(–)</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>(–)</td>
</tr>
<tr>
<td>$\Delta P$</td>
<td>pressure drop</td>
<td>(kPa)</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>dissipation rate</td>
<td>(m$^2$/s)</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinematic viscosity</td>
<td>(m$^2$/s)</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>area ratio</td>
<td>(–)</td>
</tr>
<tr>
<td>$\mu$</td>
<td>dynamic viscosity</td>
<td>Pa.s</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Kolmogorov time scale</td>
<td>s</td>
</tr>
<tr>
<td>$\tau_0$</td>
<td>volumetric flow rate</td>
<td>(cm$^3$/s)</td>
</tr>
<tr>
<td>$\theta$</td>
<td>expansion angle</td>
<td>(°)</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>non-dimensional location</td>
<td>(–)</td>
</tr>
</tbody>
</table>

**REFERENCES**


