

CFD MODELING OF TWO-PHASE FLUID SEPARATION IN A FLASH TANK USED IN A VAPOR INJECTION HEAT PUMP CYCLE

*Xing Xu, Ph.D. Student, Center for Environmental Energy Engineering
University of Maryland, College Park
3163 Glenn L. Martin Hall Bldg., MD 20742, USA*

*Yunho Hwang, Associate Director and Research Professor
Center for Environmental Energy Engineering, University of Maryland, College Park
3163 Glenn L. Martin Hall Bldg., MD 20742, USA*

*Reinhard Radermacher, Director and Professor
Center for Environmental Energy Engineering, University of Maryland, College Park
3163 Glenn L. Martin Hall Bldg., MD 20742, USA*

*Hung M. Pham, Manager, System Technology, Emerson Climate Technologies
1675 West Campbell Road, P.O. Box, Sidney, OH 45365, USA*

Abstract: A vapor refrigerant injection heat pump cycle with a flash tank has proven to be effective in significantly improving system performance at low ambient temperatures. However, ensuring effective liquid and vapor separation in the flash tank becomes one of the major challenges with regards to the reliable operation of this type of system. This paper investigates the liquid-vapor two-phase separation behavior in a flash tank by utilizing the Computation Fluid Dynamics (CFD) and flow visualization. A 3-dimensional flash tank model was built and utilized to simulate the two-phase separation under different liquid levels and a wide range of operating conditions. The modeling results were compared with the actual two-phase flow visualization results obtained through experiments. Good agreement between the modeling and experimental results was reached. The flash tank liquid level, inlet velocity, and flash tank size have been studied and found to be critical parameters for reliable two-phase separation. Results of this work could establish design guidelines for the design of a flash tank for a vapor injection heat pump cycle.

Key Words: vapor injection, CFD, two-phase flow, refrigerant charge management

1 INTRODUCTION

Recently there has been an increased interest in the study of the vapor injection cycle with a flash tank (Ma and Zhao, 2008; Cho et al., 2009; Wang et al., 2009; Xu et al., 2010). The vapor injection cycle employs the concept of introducing an additional refrigerant path to inject vapor refrigerant into the compressor. A flash tank is typically used to provide the liquid and vapor separation. The vapor refrigerant is injected to the compressor, and the liquid refrigerant flows through the evaporator. The flash tank cycle has been well known for its prominent feature of great capacity and coefficient of performance (COP) improvements, especially at low ambient temperatures. However, the liquid-vapor separation in the flash tank is very critical to reliable system operation. If the flash tank is designed or selected inappropriately, liquid refrigerant might be injected to the compressor, which would cause great damage to the compressor. As a consequence, it's necessary to investigate the two-phase flow and separation behavior inside the flash tank, and to find out the best design of the flash tank for a vapor injection system.

Computation fluid dynamics (CFD) has been known to be a useful tool for multiphase flow modeling. Advances in computational fluid mechanics have provided the basis for further

insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach (FLUENT 6.3 manual). The Euler-Lagrange approach treats each fluid phase as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. A fundamental assumption made in this model is that the dispersed second phase occupies a low volume fraction. This model is applicable for the modeling of spray dryers and coal or liquid fuel combustion. However, it's inappropriate for the modeling where the volume fraction of the second phase is not negligible. In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phase, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations that have similar structure for all phases. In the Euler-Euler model, the volume of fluid (VOF) model is widely used. It is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, the motion of large bubbles in a liquid, and the steady or transient tracking of any liquid-gas interface (FLUENT 6.3 manual). In the application of the flash tank, the main motivation of research is to track the liquid-vapor interface and to model the bubbling flow. Therefore the VOF model is appropriate for the flash tank two-phase flow modeling.

A number of research groups have employed the VOF model to simulate the liquid-vapor two-phase flow. Raynal and Harter (2001) studied the gas-liquid flow through reactors internals using the VOF model. They studied two configurations. The first configuration consists of a mixing box orifice inlet through which liquid flows as a film sheared by a gas flow. The second configuration consists of the two-phase flow through a downcomer of a distributing tray. Two and three dimensional CFD simulations using the VOF approach have been used to compute both flows for similar flow conditions as used in their experiments. Good agreement was reached between the experiment and modeling results. This provided useful means for them to achieve better design rules for gas-liquid reactors for industrial processes. Ghorai and Nigam (2006) conducted a local modeling of wavy stratified air-water flow in a pipe using the VOF model from FLUENT 6.0. The variables they studied include: gas velocity, volume fraction of liquid and interfacial roughness. The numerical profiles of longitudinal velocity above and below the waves have been compared to experimental profiles of Strand and Lopez. The error was found to be within $\pm 10\%$. They also developed a correlation to determine the ratio of the interfacial friction factor to the wall friction factor in the case of wavy flow to the smooth flow. Schepper et al. (2008) modeled the water-air gas-liquid two-phase co-current horizontal flow regimes using the VOF model, and compared with the experimental data taken from the Baker chart. They also modeled flow regimes of gas-oil liquid-vapor flow as predicted by the Baker chart. They concluded that all horizontal flow regimes appearing in the Baker chart can be simulated using the existing codes of the VOF model. Vashisth and Nigam (2009) studied the flow profiles and interfacial phenomena for two-phase flow in coiled tubes. They used a 3-dimensional CFD model using the VOF approach to predict the development of velocity fields, local and average friction factor, interfacial friction factor, phase distribution and entry length using a commercial CFD package FLUENT 6.2. A correlation was developed to determine the entry length of flow in coiled tubes. The model predictions are in good agreement with the independent experimental datasets in the literature. Parvareh et al. (2010) studied the two-phase flow regimes in horizontal and vertical tubes experimentally and theoretically. A 3-dimensional CFD modeling was carried out in order to model gas-liquid two-phase flow using the VOF model. The theoretical model has been successfully used as a reliable tool to distinguish the

flow regimes and gas holdup in two-phase fluid flow. They used an electrical resistance tomography system to visualize the flow regimes. The images from the electrical resistance tomography measurement and corresponding captured photographs for different flow regimes have been compared with the CFD predictions and a good qualitative agreement was observed. From the literature it was found that the VOF indeed provides good insights into two-phase flow. To the best knowledge of the authors, there was no research on implementing the CFD technique to model the two-phase flow in the flash tank in a heat pump cycle. Therefore this could be an effective method to bring insights into the two-phase flow behavior and provide design guidelines of the flash tank.

2 EXPERIMENTAL SETUP

The flash tank studied is part of a vapor injection system, as shown in Figure 1. Liquid refrigerant exits the condenser, and is expanded through the upper-stage expansion valve. Then the two-phase refrigerant enters the flash tank for separation. Vapor refrigerant is injected to the vapor-injected compressor. Liquid refrigerant exits the flash tank, and flows through the lower-stage expansion valve, and then flows into the evaporator for evaporation. The flash tank used in the experimental setup is shown in Figure 2. It is a cylindrical vessel that has one inlet and two outlets. The two-phase refrigerant entering the flash tank from the port located in the middle part of the tank. The inflow is tangential to the wall of the flash tank, and this design is to facilitate the liquid and vapor separation in the flash tank. The liquid refrigerant exits the flash tank from the port located at the bottom of the tank, and vapor refrigerant leaves the tank from the port at the top of the tank. A flow visualization window has been installed to the wall of the flash tank in order to visualize the two-phase separation and flow regime. Table 1 shows the specifications of the flash tank. In the experimental study of the heat pump, the system was operated at different ambient temperatures. The highest ambient temperature for cooling mode reached 46°C, and the lowest ambient temperature for heating mode reached -18°C. The flow behavior of two-phase refrigerant at different operating conditions was observed through the visualization window in the flash tank.

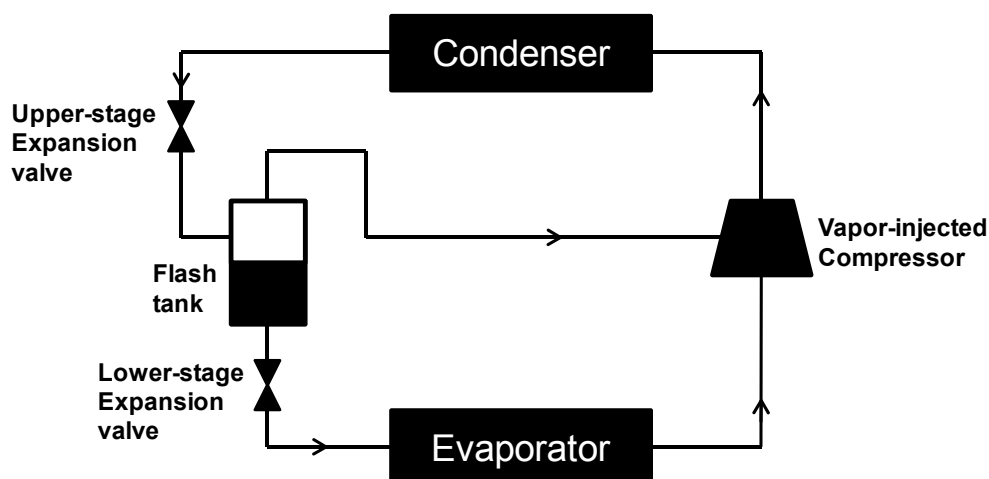


Figure 1: Schematic of the vapor injection system with a flash tank

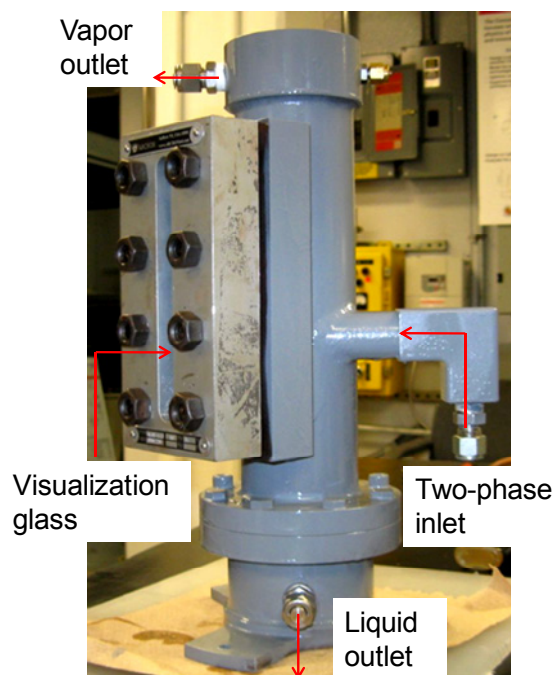


Figure 2: Flash tank used in the experimental setup

Table 1: Specifications of the flash tank in the experiment

Parameter	Unit	Dimension
Flash tank height	cm	32.0
Flash tank diameter	cm	7.0
Two-phase inlet inner diameter	cm	2.2
Liquid outlet inner diameter	cm	0.8
Vapor outlet inner diameter	cm	0.8

2 TWO-PHASE FLOW MODELING

The commercial software package FLUENT 6.3 has been selected for the two-phase flow simulation. FLUENT provides the flexibility of selecting the discretization schemes for the governing equations. The discretized equations, along with the initial and boundary conditions, have been used to obtain the final numerical solutions. Parallel solving option is also available from FLUENT, which enhances the computation efficiency for multi-core computers. The presented modeling results were performed on a 64-bit computer with 8-core and 18-G ram. The modeling focused on investigating the two-phase flow regimes and phase distributions, and therefore the heat transfer is not considered in the modeling work.

2.1 VOF Model: Governing Equations

The differential equations governing the turbulent flow in the flash tank can be written in tensor forms as follows:

The mass conservation equation is:

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \vec{u}) = 0 \quad (1)$$

The momentum conservation equation is:

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla P + \nabla [\mu(\nabla \vec{u} + \nabla \vec{u}^T)] + \rho \vec{g} + \vec{F} \quad (2)$$

Physical interpretations can be given to the different terms in the above equations. In the mass conservation equation, the first term represents the variation of density, and second term represents the convection contribution. In the momentum conservation equation, the first term on the left-hand side is the change of momentum, and the second term represents the change of momentum contributed by convection. The first term on the right-hand side is the effect from pressure difference. The second term originated from the viscous effect of the fluids. The third and fourth terms on the right-hand side represent the gravity effect and external forces, respectively. In the presented study, refrigerant properties of the liquid and vapor phases have been applied. The refrigerant used in the system is R-410A. The properties appearing in equations (1) and (2) are related to the volume fraction of all phases as follows:

$$\rho = \sum \alpha_k \rho_k \quad (3)$$

$$\mu = \frac{\sum \alpha_k \rho_k \mu_k}{\sum \alpha_k \rho_k} \quad (4)$$

The volume fraction of each fluid α_k is calculated by tracking the interface between different phases throughout the solution domain.

$\alpha_k = 0$: the cell does not contain the k th fluid;

$\alpha_k = 1$: the cell is full of the k th fluid;

$0 < \alpha_k < 1$: the cell contains the interface between the k th fluid and one or more other fluids.

2.2 Mesh Geometry

A 3-dimensional geometry the same as the flash tank in the experimental setup was created in Gambit. Figure 3 shows the schematic of the flash tank constructed in Gambit. This model includes 32,136 mesh cells. As can be seen from Figure 3-b, "A" represents the two-phase refrigerant inlet; "B" represents the vapor refrigerant outlet, and "C" represents the liquid refrigerant outlet. From Figure 3-c it can be seen that the inlet is in the tangential direction of the flash tank wall.

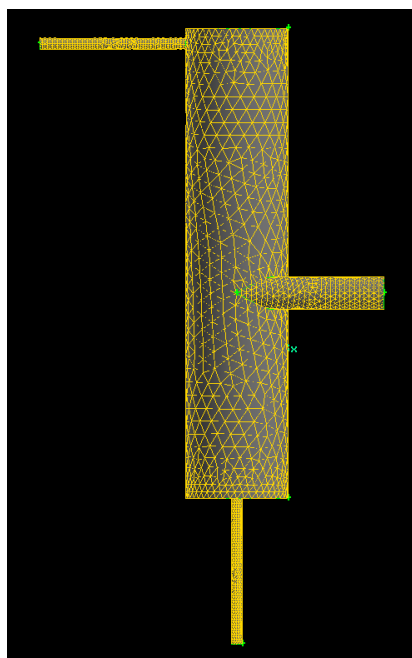


Figure 3-a: Front view of the flash tank meshed in Gambit

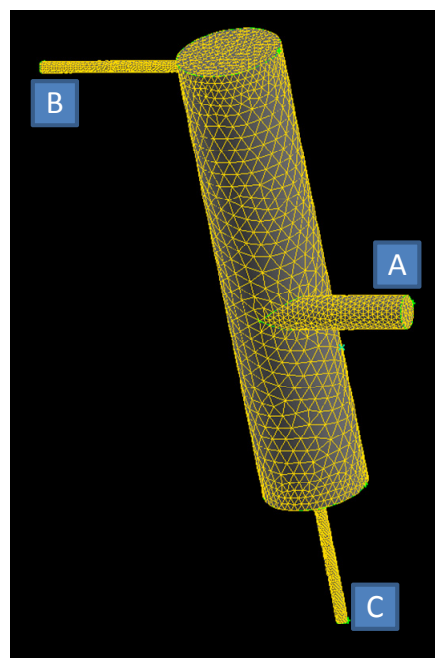


Figure 3-b: Side view of the flash tank meshed in Gambit

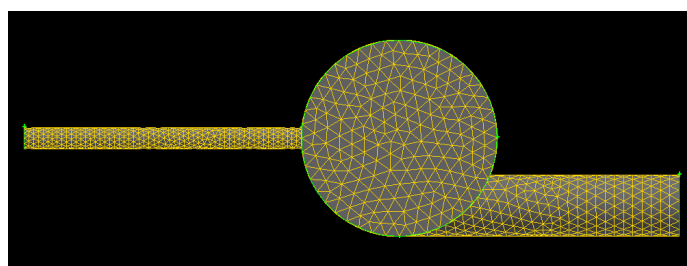


Figure 3-c: Top view of the flash tank meshed in Gambit

2.3 Boundary Conditions

Velocity boundary condition has been assigned to the inlet of the flash tank. This enables the advantage of easily varying the velocity to investigate its effect on the two-phase flow behavior. The velocity boundary condition in FLUENT also provides the convenience to define the vapor fraction of the two-phase flow. The outlet boundary conditions for the vapor and liquid have been assigned to be pressure outlets. This is because the outflows in the vapor and liquid lines are essentially driven by the pressure difference in the tank and the boundaries.

2.4 Solution Procedure

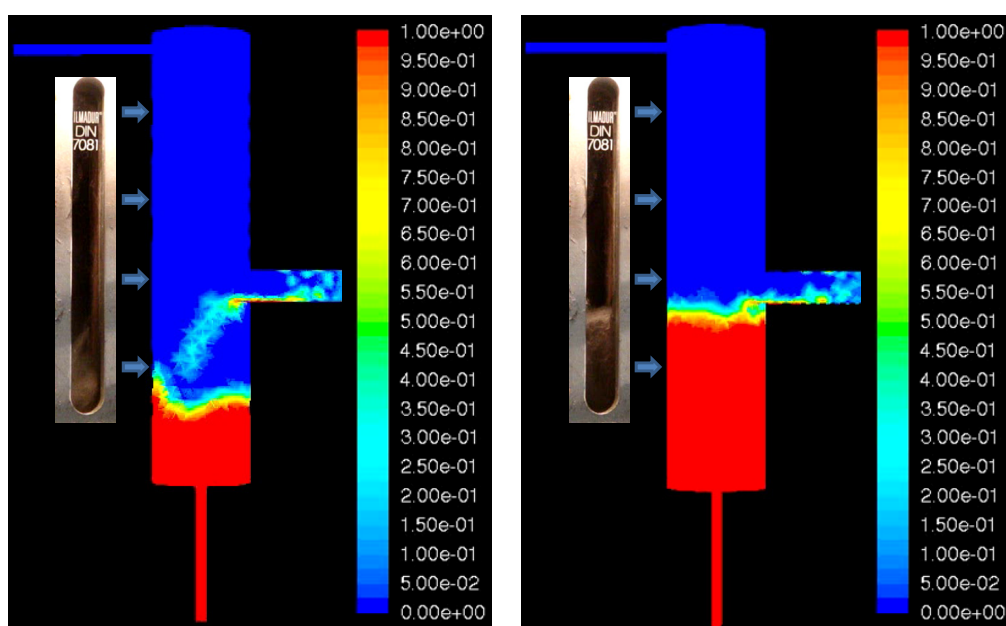
FLUENT is based on the finite volume method, which takes the average value of each cell in the control volume. To investigate the dynamic flow behavior of the two-phase flow, the unsteady solver has been selected. The time step has been selected to be 0.0002 s. RNG k-epsilon model has been selected for the viscous model. The RNG k-epsilon model was derived using a rigorous statistical technique called renormalization group theory. Compared to the standard k-epsilon model, the RNG model considers the effect of swirl on turbulence, enhancing accuracy for swirling flows (FLUENT 6.3 manual). Combination of the PISO algorithm for pressure-velocity coupling and a first-order upwind calculation scheme for the determination of momentum and volume fraction were performed in the calculations. The

PISO algorithm is particularly useful for all transient flow calculations. The applied convergence criterion uses the FLUENT default value of 0.001.

3 RESULTS AND DISCUSSIONS

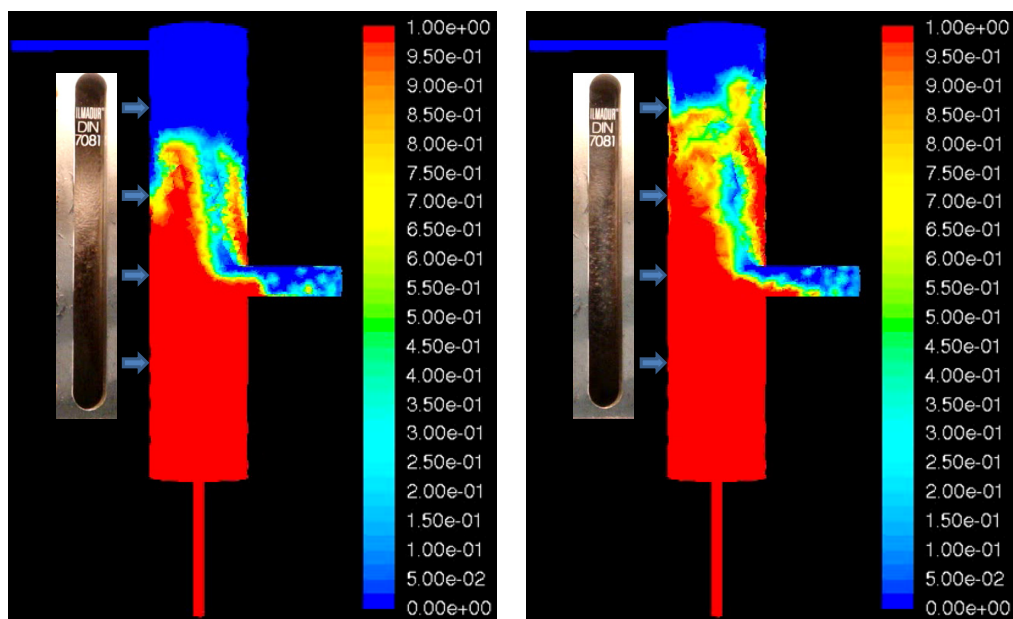
3.1 Validation of the Experimental Results

In order to perform the CFD modeling analysis to provide design guidelines of the flash tank, the modeling results need to first be validated by the experimental results. The experimental data at an ambient temperature of -18°C was used, since it's the most severe operating condition of the heat pump system. The flash tank pressure was found to be 714 kPa, and the inlet two-phase flow velocity and vapor quality were found to be 0.5 m/s and 0.2, respectively. In FLUENT, the inlet boundary condition requires the definition of volume fraction, therefore the vapor volume fraction 0.92 was used. Figure 4 shows the experimental and modeling results comparison. Figure 4a to Figure 4d shows the two-phase flow regimes of liquid refrigerant filling up 20%, 40%, 60%, and 80% of the flash tank. The pictures on the left side of each figure were taken from the experiments. Through the visualization glass the two-phase flow regime in the flash tank can be easily observed. The flash tank modeling results are shown on the right side of each figure. The blue color represents that the volume is fully occupied by vapor, and red color represents that the volume is fully occupied by liquid. In Figure 4a, the liquid fills up 20% of the flash tank by height. It can be seen that the small turbulence can be observed on liquid-vapor interface. Some bubbles were generated due to the two-phase flow impinging onto the liquid-vapor interface, and this can also be seen from the experiment result. Figure 4b shows the liquid fills up 40% of the flash tank. The liquid-vapor interface is clearly visible in this case, and there is only a thin layer of bubbles in the liquid-vapor interface. Figure 4c shows that the liquid level reaches 60% of the tank height. Strong turbulence can be seen from both the modeling and experimental results. The liquid-vapor interface is also not quite clear in this case. As the liquid level increases to 80% of the tank, stronger turbulence can be observed. A number of bubbles can also be seen from the experimental result, occupying almost 50% of the flash tank. It's also likely that some liquid droplets might enter the vapor outlet. The modeling and experimental results show good agreement, and this indicates that the model is accurate in predicting the two-phase flow regimes in different scenarios.



a – Liquid filling up 20% of the flash tank height

b – Liquid filling up 40% of the flash tank height



c – Liquid filling up 60% of the flash tank height

d – Liquid filling up 80% of the flash tank height

Figure 4: Experimental and modeling data comparison with liquid filling up different levels in the flash tank. The pictures on the left-hand side of each figure were taken from the experiment. The view angle of the visualization window is from the left to right. In the modeling result, blue color represents the volume is fully occupied by vapor, and red color represents the volume is fully occupied by liquid

3.2 Effect of the Inlet Velocity

As the model has already been validated by the experimental results, it's interesting to investigate the flow behaviors in broader conditions. The two-phase flow inlet velocity has been varied to study its effect on the two-phase flow regimes. Figure 5 shows the liquid filling up 20% of the flash tank by height with inlet velocities of 0.5 m/s, 0.8 m/s, 1.1 m/s and 1.4 m/s, respectively. It can be seen that as the inlet velocity increases, the two-phase flow tends to impinge against the wall of the vessel. The liquid-vapor interface also becomes more wavy as the inlet velocity increases. In these four cases, the liquid-vapor interfaces are clearly visible. Figure 6 shows the liquid filling up 40% of the flash tank with inlet velocities of 0.5 m/s, 0.8 m/s, 1.1 m/s and 1.4 m/s, respectively. There is no significant difference between these cases. With 1.4 m/s inlet velocity, there are more bubbles in the liquid-vapor interface.

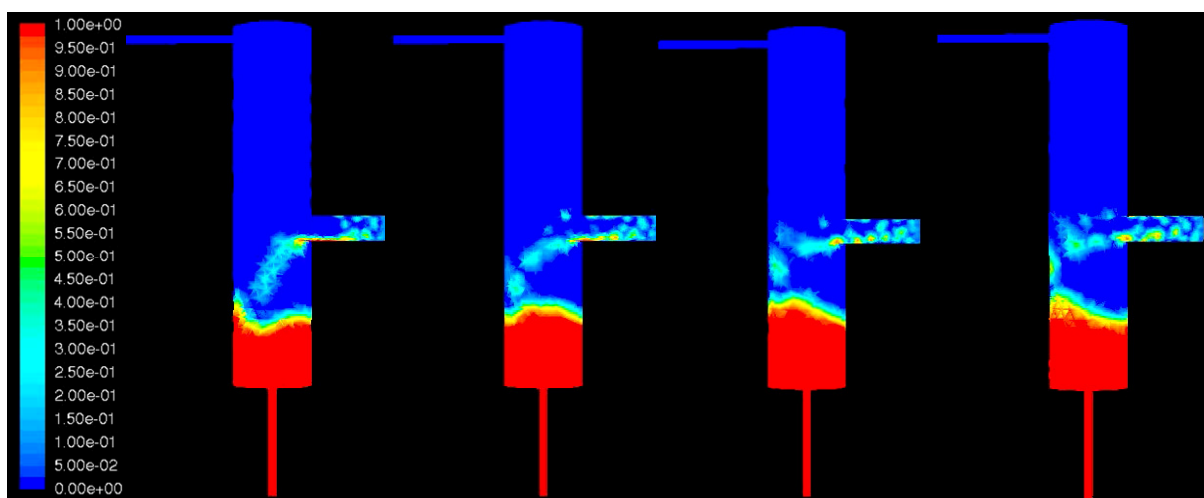


Figure 5: Liquid filling up 20% of the flash tank by height; from left to right the inlet velocity is 0.5 m/s, 0.8 m/s, 1.1 m/s, and 1.4 m/s, respectively

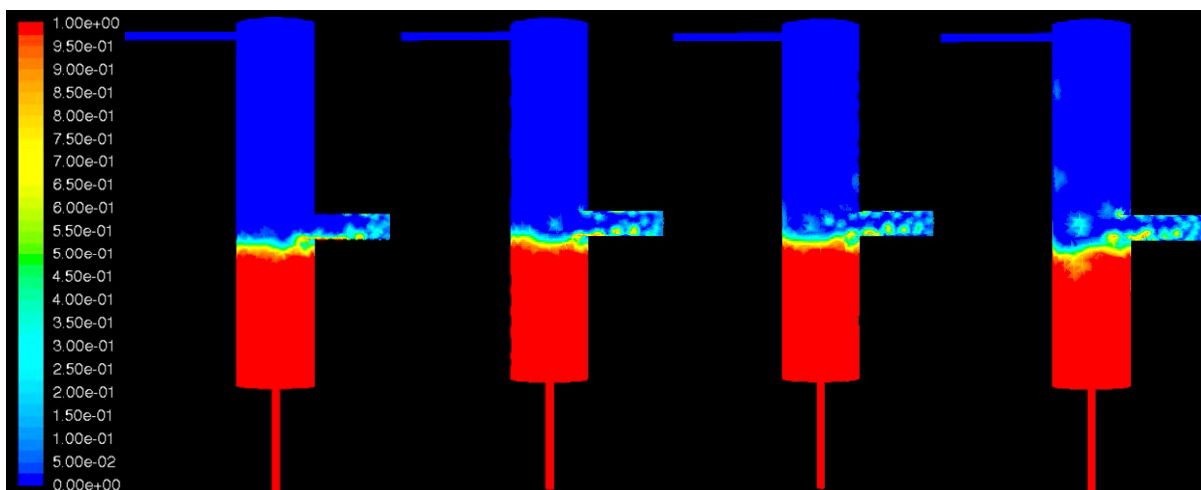


Figure 6: Liquid filling up 40% of the flash tank by height; from left to right the inlet velocity is 0.5 m/s, 0.8 m/s, 1.1 m/s, and 1.4 m/s, respectively

Figure 7 shows the liquid filling up 60% of flash tank by height. From left to right the inlet velocities are 0.5 m/s, 0.8 m/s, 1.1 m/s and 1.4 m/s, respectively. Since the liquid level is above the inlet of the flash tank, the flow regimes are significantly different compared to liquid filling up 20% and 40% of the flash tank. It can be seen that with the increase of inlet velocity, the two-phase flow is pushing the liquid more toward the flash tank wall on the left side. Therefore the turbulence also becomes stronger as the inlet velocity increases. Large bubbles can be seen with inlet velocities of 1.1 m/s and 1.4 m/s. At 1.4 m/s inlet velocity, some liquid droplets can be seen in the vapor outlet due to the turbulence generated in the flash tank. Figure 8 shows the liquid filling up 80% of the flash tank with inlet velocities of 0.5 m/s, 0.8 m/s, 1.1 m/s and 1.4 m/s, respectively. It can be seen that almost half of the tank is occupied by two-phase region. With 0.8 m/s inlet velocity, the liquid is about to be pushed into the vapor outlet. As inlet velocity increases to be 1.1 m/s and 1.4 m/s, liquid refrigerant is injected to the liquid outlet of the vessel. From the experimental point of view, this should be avoided since the injection of liquid refrigerant may cause great damage to the compressor. From the flow regimes with different inlet velocities, it can be seen that higher inlet velocity yields stronger turbulence in the flash tank. This is particularly obvious when the liquid level exceeds the flash tank two-phase flow inlet. Therefore the design of the flash tank should enlarge the flash tank inlet diameter in order to avoid high velocity of two-phase inflow.

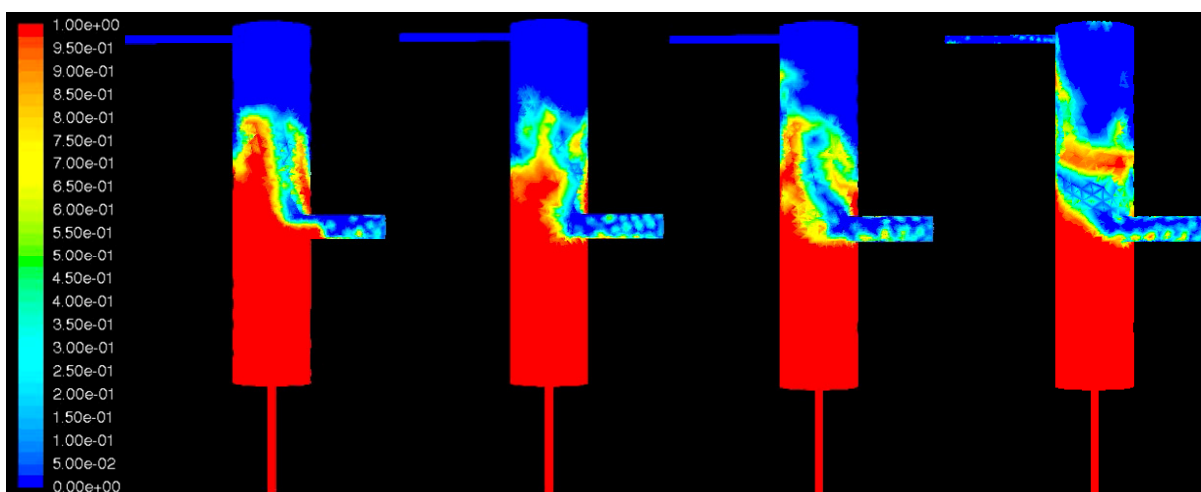


Figure 7: Liquid filling up 60% of the flash tank by height; from left to right the inlet velocity is 0.5 m/s, 0.8 m/s, 1.1 m/s, and 1.4 m/s, respectively

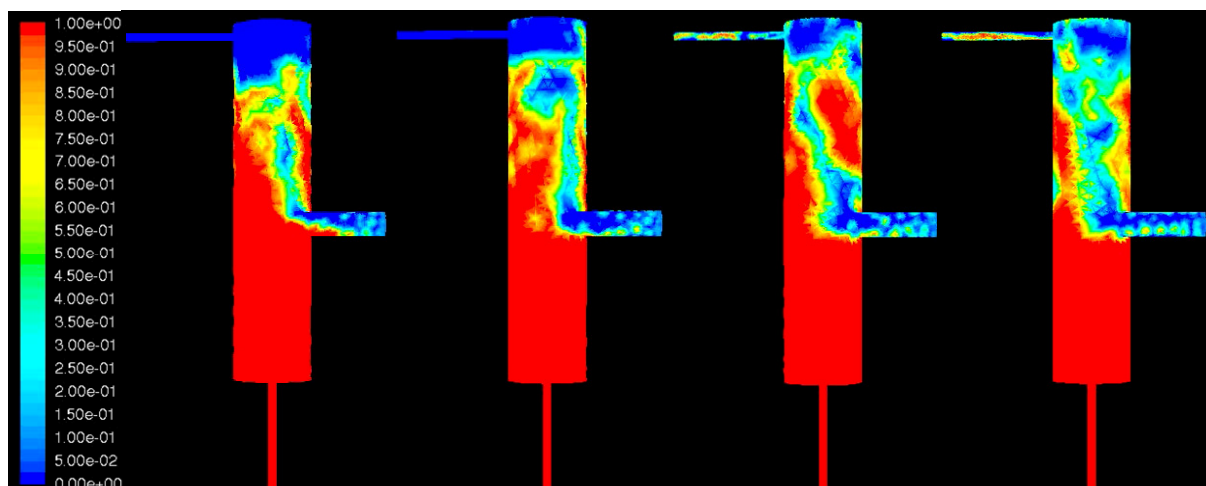


Figure 8: Liquid filling up 80% of the flash tank by height; from left to right the inlet velocity is 0.5 m/s, 0.8 m/s, 1.1 m/s, and 1.4 m/s, respectively

3.3 Effect of the Flash Tank Dimensions

Theoretically, a flash tank with larger volume should be able to provide more efficient separation of liquid and vapor. However, the large volume also means higher refrigerant charge amount for a heat pump system. Therefore it's necessary to provide flash tank designs that ensure efficient liquid-vapor separation, while also requires minimum refrigerant charge amount. The flash tank used in the experiment functions well to separate liquid and vapor when the liquid level does not exceed 60% of the tank. It's interesting to see whether a flash tank with reduced dimensions would also function well in the given conditions. Two flash tanks with modified geometries have been studied. Table 2 shows the specifications of the flash tanks with modifications. Flash tank 1 has 80% of the original flash tank height and diameter, and flash tank 2 has 60% of the original flash tank height and diameter, respectively. The inlet and outlets of the flash tank have been set to be the same as the original flash tank design. Figure 9 shows the physical comparison between the three flash tanks investigated.

Table 2: Specifications of the flash tanks with modifications

Parameter	Unit	Flash tank 1 dimension	flash tank 2 dimension
Flash tank height	cm	25.6	19.2
Flash tank diameter	cm	5.6	4.2
Two-phase inlet inner diameter	cm	2.2	2.2
Liquid outlet inner diameter	cm	0.8	0.8
Vapor outlet inner diameter	cm	0.8	0.8

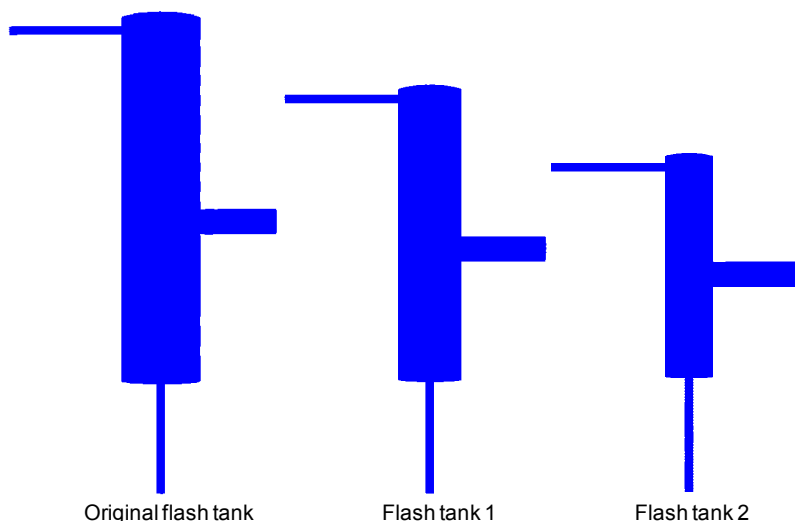


Figure 9: Physical comparison between the original flash tank and flash tank 1 and 2

Figures 10 and 11 show the two-phase flow regimes with liquid filling up the flash tank with flash tank 1 and flash tank 2, respectively. As the size of the flash tank decreases, the effect of inlet flow impinging onto the wall becomes stronger. It can also be seen that for flash tank 2, when the liquid level exceeds 60% of the tank height, some liquid droplets can be seen in the vapor outlet. For flash tank 1, however, there is no liquid droplet seen in the vapor outlet when the liquid reaches 60% of the flash height. For both cases, it can also be seen that liquid is present in the vapor outlet when the liquid level reaches 80% of the flash tank height. Moreover, as the flash tank size decreases, it's more likely for liquid to enter the vapor outlet since the response time for the expansion valve that controls the two-phase inflow becomes shorter. Therefore flash tank 2 seems to be too small for the current flow condition, and flash tank 1 seems to be suitable if the liquid level is not significantly higher than 60% of the flash tank height.

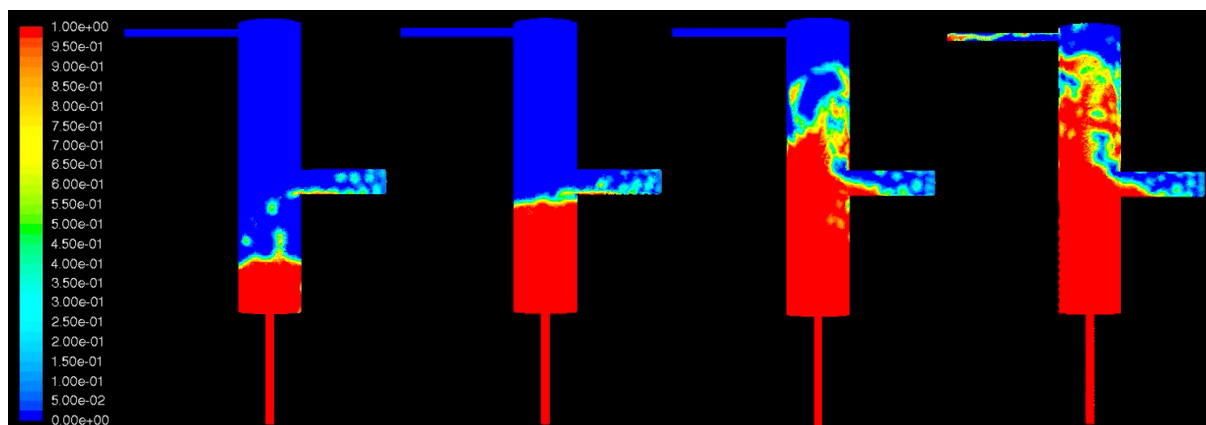


Figure 10: Two-phase flow regimes with liquid filling 20%, 40%, 60% and 80% of the flash tank 1; the flash tank diameter and height are 80% of the original flash tank dimensions, and the inlet and outlets are the same as the original flash tank

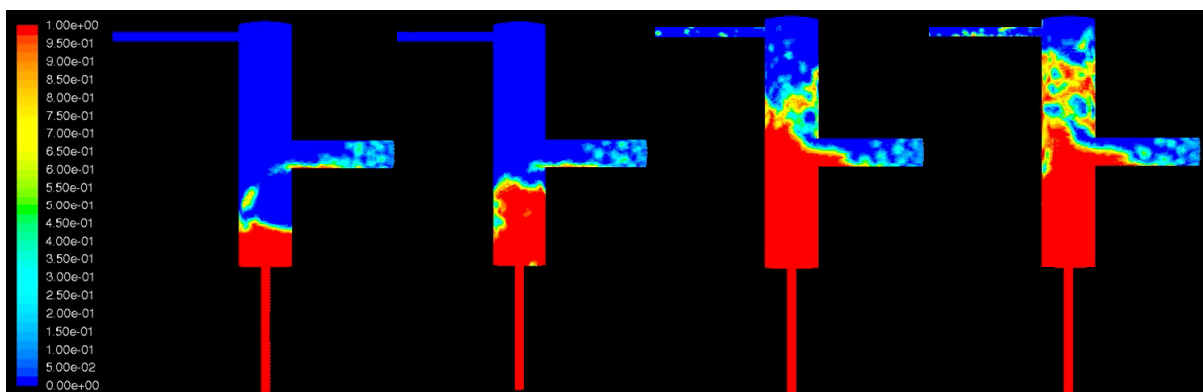


Figure 11: Two-phase flow regimes with liquid filling 20%, 40%, 60% and 80% of the flash tank 2; the flash tank diameter and height are 60% of the original flash tank dimensions, and the inlet and outlets are the same as the original flash tank

4 CONCLUSIONS

This paper investigates the two-phase flow regimes and separation in a flash tank by utilizing CFD and flow visualization. A 3-dimensional flash tank model was built and utilized to simulate the two-phase separation under different operating conditions. Gambit was selected as the meshing tool, and FLUENT 6.3 was utilized as the solver for the 3-dimensional model. A flow visualization window was installed in the flash tank to study the two-phase flow experimentally. The two-phase flow regimes with liquid level filling up 20%, 40%, 60% and 80% of the flash tank were modeled and compared with experimental results. Good agreement between modeling and experimental results has been reached. The inlet two-phase flow velocity was also varied from 0.5 m/s to 1.4 m/s, with an increment of 0.3 m/s. It was found that higher inlet velocity results in stronger turbulence in the flash tank, and this is particularly obvious when the liquid level exceeds the two-phase flow inlet. Therefore it's recommended that the flash tank should be designed in such a way that high inlet velocity should be avoided. The flash tank size has also been reduced to be 80% and 60% of the original dimensions in the model to study whether a smaller tank would also provide efficient two-phase separation. It has been found that the flash tank with 80% of the original flash tank dimensions seems to be suitable for the current system, and the flash tank with 60% of the original dimensions seems to be too small. Further work is needed to find a critical parameter that combines the parameters of the inlet velocity and flash tank diameter. The presented work provides insights into the two-phase flow in the flash tank in a heat pump system, and identifies a few key parameters related to the flash tank design. However, some more efforts are needed to study the geometry of the flash tank, such as the aspect ratio in order to provide the optimum design of the flash tank.

5 REFERENCES

Cho, H., Baek, C., Park, C., Kim, Y., 2009. "Performance evaluation of a two-stage CO₂ cycle with gas injection in the cooling mode operation," *International Journal of Refrigeration*, Vol. 32, pp. 40-46.

FLUENT 6.3 manual:

http://my.fit.edu/itresources/manuals/fluent6.3/help/html/ug/main_pre.htm

Ghorai, S., Nigam, K., 2006. "CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes," *Chemical Engineering and Processing*, Vol. 45, pp. 55-65.

Ma, G.Y., Zhao, H.X., 2008. "Experimental study of a heat pump system with flash tank coupled with scroll compressor," *Energy and Building*, Vol. 40, pp. 697-701.

Parvareh, A., Rahimi, M., 2010. Alizadehdakhel, A., Alsairafi, A., "CFD and ERT investigations on two-phase flow regimes in vertical and horizontal tubes," *International Communications in Heat and Mass Transfer*, Vol. 37, pp. 304-311.

Raynal, L., Harter, I., 2001. "Studies of gas-liquid flow through reactors internals using VOF simulations," *Chemical Engineering Science*, Vol. 56, pp. 6385-6391.

Schepper, S., Heynderickx, G., Marin, G., 2008. "CFD modeling of all gas-liquid and vapor-liquid flow regimes predicted by the Baker chart," *Chemical Engineering Journal*, Vol. 128, pp. 349-357.

Vashisth, S., Nigam, K., 2009. "Prediction of flow profiles and interfacial phenomena for two-phase flow in coiled tubes," *Chemical Engineering and Processing: Process Intensification*, Vol. 48, pp. 452-463.

Wang, X., Hwang, Y., Radermacher, R., 2009. "Two-stage heat pump system with vapor-injected scroll compressor using R410A as a refrigerant," *International Journal of Refrigeration*, Vol. 32, pp. 1442-1451.

Xu, X., Hwang, Y., Radermacher, R., Pham, H.M., 2010. "Control strategy and refrigerant charging management of vapor injection system with a flash tank," *Journal of Refrigeration and Air-Conditioning*, Vol. 10, pp.119-126 (In Chinese).