Numerical Analysis of Refrigerant Flow in Capillary Tubes

Asst. Lect. Ahmed M. Ibarhim
Refrigeration and Air Conditioning Eng. Dept.
Technical College, Baghdad, Iraq

Lect. Ma'athe Abdul-Wahed
Mechanical Eng. Dept., College of Eng.
Al-Mustansiriya University, Baghdad, Iraq

Abstract

The capillary tube is a common expansion device used in small sized refrigeration and air-conditioning systems and whose function is to reduce the high pressure in the condenser to low pressure in the evaporator and therefore drops the temperature of the refrigerant.

The main focus lies on the analysis of the refrigerant flow behavior in the capillary tube, including pressure drop, temperature and quality distribution along the tube by using a commercial code FLUENT V.6 for the simulation.

The numerical results are compared with experimental results from internet web site and show a good agreement. It shows the capability of the code to simulate the flow and phase change in the capillary tube.

الخلاصــــــة

الانبوبه الشعريه هي اكثر اجهزه التمدد شيوعا تستخدم في انظمه التبريد و التكيف الصغيره الحجم و وظيفتها تخفيض الضغط العالي في المكثف الى ضغط واطئ في المبخر.

الهدف الرئيسي هو تحليل سلوك الجريان لمائع التبريد في الانبوبه الشعريه تشمل توزيع انخفاض الضغط درجه الحراره و النوعيه (كسر الجفاف) على طول الانبوبه باستخدام برنامج FLUENT للمحاكاة.

النتائج النظريه تم مقارنتها مع نتائج عمليه و أظهرت تطابق جيد. اظهرت الدراسه مقدره البرنامج لمحاكاة الجريان و تغير الطور في الانبوبه الشعريه.

1. Introduction

The capillary tube is a constant area expansion device used in a vapor-compression refrigeration system located between the condenser and the evaporator and whose function is to reduce the high pressure in the condenser to low pressure in the evaporator. The capillary tube expansion devices are widely used in refrigeration equipment, especially in small units such as household refrigerators, freezers and small air conditioners. Its simplicity is the most important reason to continue using it instead of other expansion devices. It is a long copper tube (1-6 m) with a very small inner diameter. The flow inside the capillary tube of a refrigeration system can be divided into a sub-cooled liquid region from the entrance to the point in which the fluid reaches saturated conditions, and a two-phase flow region after that point until the end of the capillary tube.

Many numerical and experimental investigations have been developed in order to study the refrigerant flow in capillary tubes. Normally, the numerical studies concerning capillary tubes found in the literature are targeted to the calculation of the mass flow rate through the tube for given geometry and operative conditions Bittle et. al. [1,2], Wongwises [3,4]. Experimental studies were investigated on the performance of refrigerants in-several capillary tubes length and geometry for air-conditioners Ro S. T. et. al. [5].

Another experimental study was taken under account the influence of geometrical parameters on capillary behavior i.e. the influence of length, inner diameter and type of refrigerants. The experimental device has been fully instrumented with pressure, temperature, and power sensors. The vapor-compression system was composed mainly of a compressor equipped with an accumulator, condenser, evaporator, and adjustable expansion device. Both condenser and evaporator were equipped with pre and post heat exchangers to control the flow conditions at the capillary tube test section. A straight capillary tube was installed together with various measuring stations equipped with sensors for temperature and pressure. Those sensors were necessary for measurement of the temperature and the pressure profiles along the capillary tube length, S. M. Sami et. al. [6].

The author has evaluated the pressure, temperature and quality distribution along the tube, which plays the most important influence on the capillary tube performance and compare the results with the experimental results extracted from S. M. Sami et. al. [6].

2. Numerical Procedure

One of the main objectives of FLUENT is to reduce the maximum calculation time. This makes FLUENT a very comfortable analysis and optimization tool to solve fluid flow problems involving phase change. A fixed-grid methodology is used, in which an equation for the latent heat content is solved and the enthalpy equation is solved with extra source terms due to the phase change.

The fundamental equations governing capillary tube flow are continuity and Navier-Stokes equations

$$[\rho \, u \, A] = 0 \quad \text{.......... (1)}$$

where:

ρ *: is the density.*

u: is the velocity of refrigerant, and

A: is the cross section area of capillary tube.

$$u_t + \nabla . (u u) = -\frac{1}{\rho}[\nabla p - \nabla .(2\mu S)] + \frac{1}{\rho} F_{SF} , \nabla . u = 0 \quad \text{.......... (2)}$$

where:

p: is the pressure, and

S: is the deformation tensor which is given by:

$$S = \frac{1}{2}\left(\nabla u + [\nabla u]^T\right) \quad \text{.......... (3)}$$

where:

F_{SF}: is continuum surface force vector.

The above equation is dependent on the volume fractions of all phases through the properties ρ and μ. These properties were calculated by the following equations:

$$\rho = \sum \alpha_k \, \rho_k \quad \text{.......... (4)}$$

and,

$$\mu = \frac{\sum \alpha_k \, \rho_k \, \mu_k}{\sum \alpha_k \, \rho_k} \quad \text{.......... (5)}$$

where:

α_k , ρ_k *and* α_k *: are the volume fraction, density and viscosity of the* kth *fluid, respectively.*

In a two-phase system, the following possibilities arise in a particular cell.

$$\alpha_2 = \begin{cases} 0, & (\text{Fluid 1}) \\ 1, & (\text{Fluid 2}) \\ 0 \langle \alpha_2 \langle 1 & (\text{Interface between two fluids}) \end{cases} \quad \text{.......... (6)}$$

The interface between two fluids was tracked by volume fraction function α_k. It convicts with the flow and conservation of this function can be represented with the help of interface mass balance conditions by pure convection equation:

$$\frac{\partial \alpha_k}{\partial t} + \mathbf{u} . \nabla \alpha_k = 0 \quad \text{.. (7)}$$

The volume fraction for the primary phase was not solved and was obtained from the following equation:

$$\sum \alpha_k = 1 \quad \text{.. (8)}$$

These equations can be integrated and solved simultaneously by an iterative process for each control volume with the help of finite volume method (FVM). The process can be repeated along the capillary tube length including single-phase and two-phase cells. For the two phases flow region a separated flow model is assumed.

The model was developed with the following considerations:

1. The capillary tube is horizontal (gravity effects are neglected), and of constant cross section.
2. The flow in the capillary tube is steady, one-dimensional.
3. When the fluid reaches the saturation pressure, the fluid starts to evaporate.
4. The fluid is always in local thermodynamic equilibrium corresponding to its local pressure.

Pressure losses can be defined by:

1. Pressure drop by entrance effects (from the upstream tube to the capillary tube)
2. Pressure drop by friction: Single-phase friction from entrance up to the saturation point, and two phase friction from saturation point up to the end.

The conservation equations and equations of state constitute a system of non linear equations. For a given capillary tube geometry (inner diameter and length) and given operative conditions, the whole problem becomes fully implicit. The calculation of the capillary tube performance for given operating conditions (upstream and downstream) starts from the inlet section and proceeds till the end of the capillary tube. Integrating the conservation equations over length, for non critical conditions, the calculated capillary tube length must be the real capillary tube length. The process start calculating the pressure, temperature and then the quality at each cell can be calculated.

Structured rectangular grid generated with the help of Gambit 2.2 (Fluent v.6 Inc., USA) was used for simulation. A complete and detailed explanation of the numerical method with the equations can be found in FLUENT software user manual.

3. Description of Capillary Tube

A tube with a length of 222.3 cm and an inside diameter of 2.159 mm. The refrigerant is R-22 with an inlet temperature of 26 C, inlet pressure of 1100 kPa and mass flow rate of 15 g/sec.

4. Results and Discussion

The described capillary tube model has been validated with numerical investigation by using the commercial CFD code FLUENT v.6 program for the simulation. As can be observed in resulting Figures the model has a good agreement for experimental results [6].

Figure (1) shows the variation of pressure drop over the length of the capillary tube. The result shows that there is no choke flow phenomenon was observed during the study because there was no large drop in pressure at the exit of the tube. The results show a good agreement between numerical and experimental [6]. (The pressure unit in psig for comparison purpose).

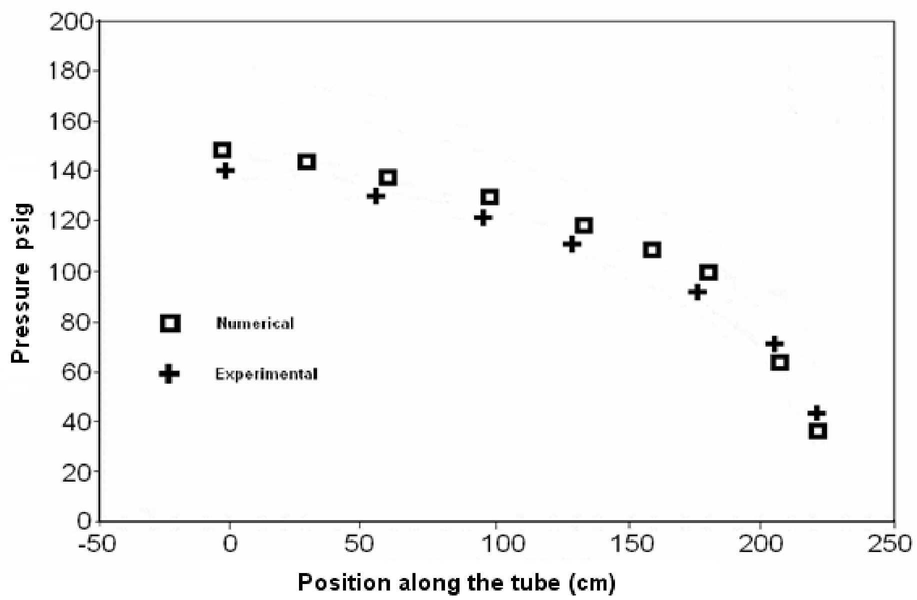

Figure (1) Variation of pressure over the length of the tube

Figure (2) shows the variation of temperature distribution over the length of the capillary tube. The results show a good agreement between numerical and experimental results.

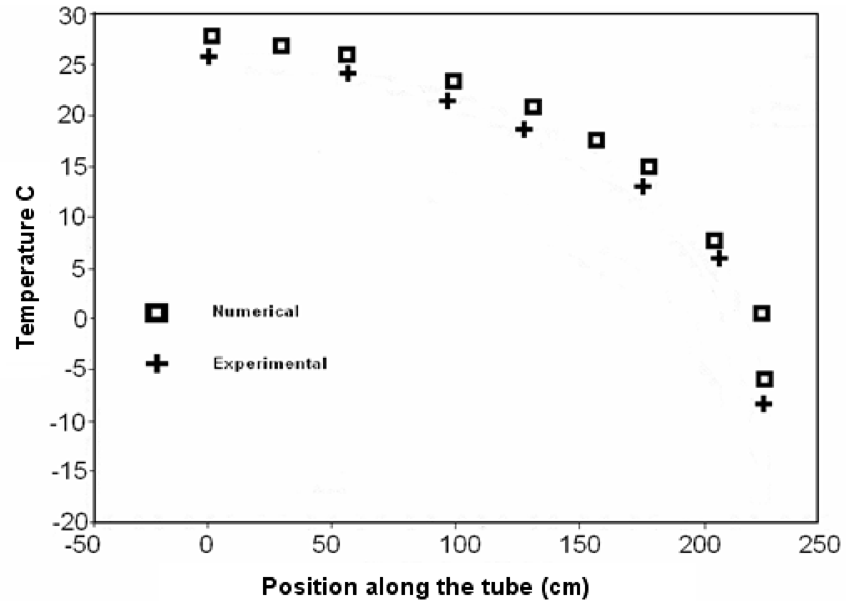

Figure (2) Variation of temperature over the length of the tube

Figure (3) shows the variation of quality distribution over the length of the capillary tube. This figure shows the phase change behavior of refrigerant R-22 in the capillary tube, starting from the sub-cooled liquid region from the entrance in which the quality X less than zero (Sub-cool), to the point in which the fluid reaches saturated conditions in which the quality X equal to zero, then a two-phase flow region in which the quality X greater than zero after that point until the end of the capillary tube.

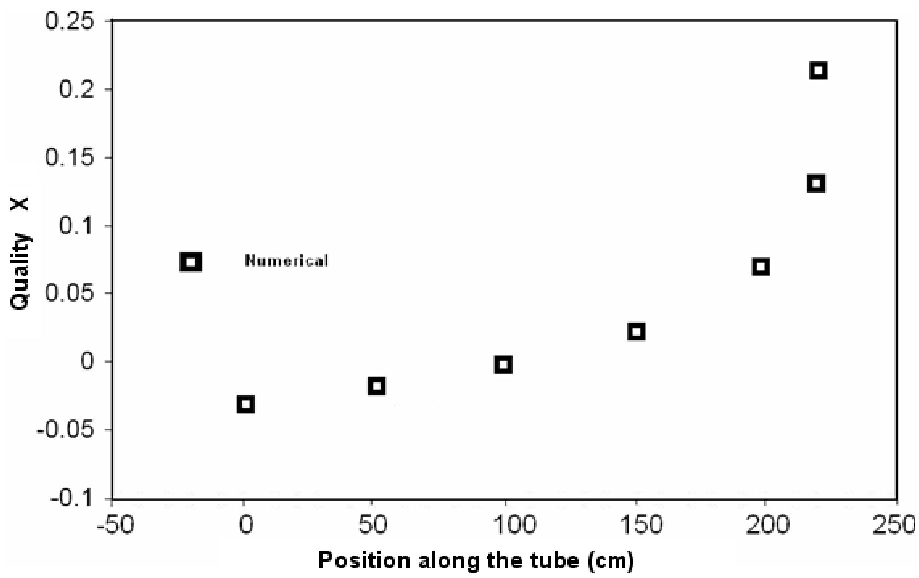

Figure (3) Variation of quality over the length of the tube

5. Conclusions

A numerical model for calculating pressure, temperature and quality distribution through the capillary tube working with pure refrigerant R-22 has been carried out by means of the integration of the conservation equations (mass, momentum and energy) over individual cell. The model assumes thermodynamic equilibrium and one-dimensional two-phase flow.

The FLUENT program is able to predict the behavior of the refrigerant flow with the phase change during the throttling process in the capillary tube. In general, a good agreement between numerical and experimental results has been observed. No choke flow phenomenon was observed at the exit of the tube because there was no large drop in pressure occurs in the tube.

6. References

1. Bittle, R., Pate, M., ***"A Theatrical Model for Predicting Adiabatic Capillary Tube Performance with Alternative Refrigerants"***, ASHRAE Trans. and Research, Vol. 102, No. 2, 1996, pp. 52-64.

2. Bittle, R., et. al., ***"A Generalized Performance Prediction Method for Adiabatic Capillary Tubes"***, HVAC and Research, Vol. 4, No. 1, 1998, pp. 27-44.

3. Wongwises, S., Chan, P., ***"Two-Phase Separated Flow Model of Refrigerant Flowing through Capillary Tubes"***, Int. Comm. Heat Mass Transfer, Vol. 27, No 3, 2000a, pp. 343-356.

4. Wongwises S., et. al., ***"A Comparison of the Flow Characteristics of Refrigerants Flowing through Adiabatic Capillary Tubes"***, Int. Comm. Heat Mass Transfer, Vol. 27, No. 5, 2000b, pp. 611-621.

5. Ro, S. T., Kim, S. G., Kim, M. S., 2002, ***"Experimental Investigation of the Performance of R22, R407C and R410A In-Several Capillary Tubes for Air-Conditioners"***, Int. Journal of Refrigeration, Vol. 25, 2002, pp. 521-531.

6. Sami, S. M., Maltais, H., and Desjardins, D. E., ***"Influence of Geometrical Parameters on Capillary Behavior with New Alternative Refrigerants"***, (internet communication), 2002.