ISSN: 2536-4383

Computational Simulation of Two Phase Flow in a Refrigerant Distributor

Mehmet Direk[a,*] Mehmet Harun Sökücü[b], M. Ekrem Çakmak[a] and Ergin BAYRAK[c]

[a]Yalova University, Department of Energy Systems Engineering, Yalova, 77200, TURKEY

[b]Friterm A.Ş., R&D Department, Istanbul, 34957, TURKEY

[c]Alanya Alaaddin Keykubat University, Department of Air Conditioning and Refrigeration Technology, 07425 Alanya, Antalya, TURKEY

Abstract

In order to keep pressure drops at lower levels and to maintain uniform refrigerant distribution, flow behavior of refrigerants in the distributors at the entrance of evaporators should be elucidated. The aim of this study is to obtain a validated computational simulation of two-phase flow distribution in conventional and new designs refrigerant distributor. The results of the computational simulations of the distributor are validated against the experimental results. The influence of the flow distribution on the evaporator performance has been considered. Distribution of mass flow rate in the distributor channels and pressure drop are numerically calculated and then compared to the data obtained from the experimental facility. As a result, the difference between numerical and experimental study of the pressure drop is less than 2%.

Key words: Refrigerant distributor, computational simulation, two phase flow

Soğutucu Akışkan Distribütöründe İki Fazlı Akışın Bilgisayarlı Simülasyonu

Öz

Soğutucu akışkan basınç düşümünün düşük seviyede tutulması ve evaporatör girişinde homojen bir dağılımın sağlanması amacıyla distribütördeki akışın anlaşılması gerekmektedir. Bu çalışmanın amacı geleneksel olarak kullanılan ve yeni dizayn edilmiş bir soğutucu akışkan distribütöründe iki fazlı akış karakteristiklerinin elde edilmesi ve doğrulanmasıdır. Bu amaçla deney veriler kullanılarak bir simülasyon modeli hazırlanmıştır. Distribütördeki iki fazlı akış dağılımının evaporatör performansı üzerindeki etkisi göz önünde bulundurulmuştur. Distribütör kanallarındaki kütlesel debilerin dağılımı ve basınç düşümleri numerik olarak hesaplanmış ve deney düzeneğinden alınan sonuçlar ile karşılaştırılmıştır. Sonuç olarak, deney düzeneğinden alınan basınç düşümü sonuçlarının numerik olarak elde edilen sonuçlarla % 2'den daha az bir farkla örtüştüğü görülmüştür.

Anahtar Kelimeler: Soğutucu akışkan distribütörü, bilgisayarlı simülasyon, iki fazlı akış

[*] Corresponding author
E-mail: mehmetdirek@hotmail.com

Received: 16.11.2016
Accepted: 27.10.2017

Introduction

In refrigeration cycles evaporators are used to evaporate refrigerant fluid while absorbing heat from the air passing through the evaporator coils. The most important factors affecting the efficiency of evaporators may be counted as pressure drops and heat transfer rates. In order to perform sufficient heat transfer and balanced pressure distribution inside micro channels of evaporator coil, a distributor should uniformly distribute refrigerant into micro channels. Designs of distributors are essential in terms of proper distribution of the gas/liquid mixture. Homogeneous distribution is dependent on geometry and flow profile in evaporator channels. Non-uniform airflow profiles diminish the heat transfer rate about 6% [1]. Cooling capacity and Coefficient of Performance (COP) of the system decrease by as much as 15-10% in case of a non-uniform distribution in the fin-and-tube evaporator [2].

On the other hand, with the current experimental techniques it is very difficult to directly analyses pressure gradients and flow profiles due to micro scale flow encountered in the distributors. Moreover, the presence of two-phase flow (gas and liquid) makes the situation more complicated [3, 4] Therefore, computational simulations may supply these data easily. However, in the beginning there are many computational limitations to be dealt with such as turbulence, presence of two-phase [5, 6 and 7]. To best of our knowledge, not many studies are present about two phase flow in distributors. Li et al. [8] employed Fluent 5.4 (2003) for computational fluid dynamics analysis of distributors with different geometric properties as well as effect on pressure drop. They have found that, among three two-phase models included in Fluent 5.4, the IPSA (inter-phase slip algorithm) produced results in close agreement with their experimental results. They also demonstrated that spherical base distributor yielded the best distribution regarded flow distribution in distributors. Nakayama et al. [3] they conducted a comparison between new design distributors that had a capillary mixing space with the orifice of the conventional distributor. Due to vertical orientation with better mixing of the new type capacity can be increased by 1.2% compared to the conventional distributor. On the other hand, the most general model for solving multiphase flows is Eulerian model which are different levels of application and accuracy [8]. Fei and Hrnjak [9] modeled the two-phase flow in a header with the Eulerian model due to flow in the header has droplet mist flow characteristics. Abishek et al. [10] they conducted computational simulation study the effect of flow configuration (parallel/counter) on the heat transfer and two-phase flow characteristics of a double-pipe evaporative heat exchanger. The governing equations solved with Eulerian multiphase modeling framework in FLUENT for the conjugate non-isothermal two-phase flow problem with phase change. Yuan et al. [11] modeled the two-phase flow with computational fluid dynamics (CFD) software employed ANSYS-CFX. Their simulation based on the Eulerian–Eulerian approach in which the gaseous phase is accepted as continuum and liquid phase as dispersed phase.

In this study it is expected to develop a cost-effective and reliable test method to assess the distributors used in the evaporators manufactured by FRITERM, A.Ş. as an alternative to experimental analysis conducted in the R&D facilities. For this purpose ANSYS Fluent 15.0 is used to simulate two-phase flow in a distributor generally used by FRITERM, A.Ş. The Eulerian model included in Fluent 15.0 is chosen because the Eulerian model has similar properties with the IPSA model, which is not readily present in the Fluent version used in the current study. The results of the CFD simulations of the distributor are validated against the experimental results supplied by R&D Department of FRITERM, A.Ş.. The validated method is expected to be used to develop a better design of the distributor investigated in the current study (on-going project) without a need for experimental analysis. The numerically predicted two- phase flow characteristics such as mass flow rate distribution, volume fraction and pressure droop in are compared with experimental results for conventional (case 1) and new designs (case 2) distributor.

The Simulation Model

Computational simulations are performed to simulate two-phase flow inside of the distributor by using Fluent 15.0. It is assumed that the flow is incompressible viscous turbulent flow and the process is adiabatic. Velocity inlet and pressure outlet is selected to be the boundary conditions.

The Eulerian Multiphase Model

According to Fluent 15.0 user's manual Eulerian Multiphase Model (EMM) evaluate the system of mass, momentum and energy equations separately for each phase (gas and liquid phases). Addition to the interfacial lift and virtual mass forces included in the inter-phase slip algorithm model [8] wall lubrication force, virtual mass force, and turbulence dispersion force are considered in the Eulerian Multiphase Model.

As described in the FLUENT 15.0 manual [12]: for each phase, the momentum conservation equation is

$$\frac{\partial}{\partial t}\left(a_q\, \rho_q \bar{V}_q\right) + \bar{V}\left(\, a_q\, \rho_q \bar{V}_q\, \bar{V}_q\right) = -a_q \bar{V}_p + \bar{V}\bar{\bar{\tau}}_q a_q\, \rho_q \vec{g}$$

$$+\sum_{p=1}^{n}(\vec{R}_{pq}+\dot{m}_{pq}\vec{V}_{pq}-\dot{m}_{qp}\vec{V}_{qp})+(\vec{F}_{q}+\vec{F}_{lift,q}+\vec{F}_{wl,q}+\vec{F}_{vm,q}+\vec{F}_{td,q}$$

(1)

Where $\bar{\bar{\tau}}_q$ is the qth phase stress-strain tensor

$$\bar{\bar{\tau}}_q = a_q\mu_q(\bar{V}\vec{V}_q+\bar{V}\vec{V}_q^T)+a_q\left(\lambda_q-\frac{2}{3}\mu_q\right)\bar{V}\vec{V}_q\bar{\bar{I}} \quad (2)$$

Here $\mu_q$and λ_q are the shear and bulk viscosity of phase q, $\vec{F}_q$ is an external body force, $\vec{F}_{lift,q}$ is a lift force $\vec{F}_{wl,q}$ is a wall lubrication force $\vec{F}_{vm,q}$ is a virtual mass force, and $\vec{F}_{td,q}$ is a turbulence dispersion force (in the case of turbulence flows only) $\vec{R}_{pq}$ is an interaction force between phases, and p is the pressure shared by all phases."

Mesh and Boundary Conditions

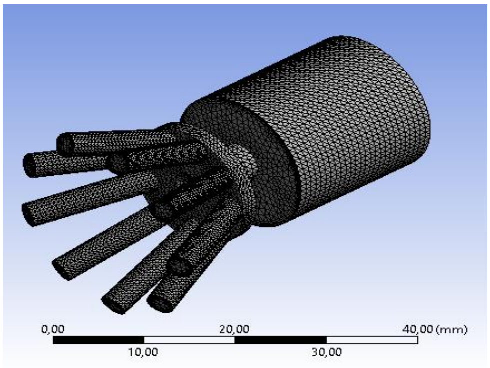
Velocity inlet and pressure outlet are selected to be the boundary conditions. Mesh statistics that apply to distributor are given Table 1. View of the distributor geometry and mesh for case 1 (a) and case 2 (b) can be seen in Fig. 1. Fig. 2 demonstrates position of distributor channels. Since the results from the last two mesh systems are similar, the results presented in the paper are generated using the second mesh system with 203703 nodes. Inlet boundary conditions of distributors are depicted in Table 2.

Table 1. *Mesh statistics of distributors*

	Case 1	**Case 2**
Nodes number	67572	65300
Element number	205464	203703

Table 2. *Inlet boundary conditions for distributors*

	Case 1	**Case 2**
Mass flow rate (kg/s)	0.104	0.094
Quality	0.29	0.3
Gas phase volume rate (m^3/s)	0.0000691	0.0000625
Liquid phase volume rate (m^3/s)	0.0007500	0.0006782
Void fraction	0.7657	0.7657
Gas phase velocity (m/s)	0.77	0.94
Liquid phase velocity (m/s)	2.57	3.12



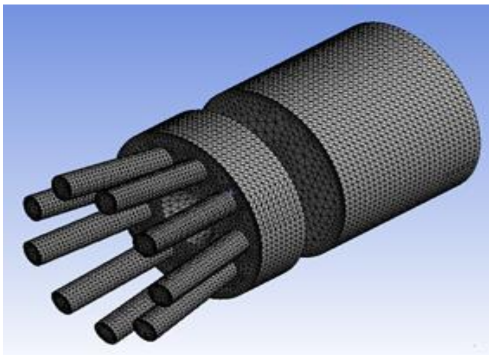


Figure 1*. View of the distributor geometry of case-1 (a) case-2 (b) and mesh*

ISSN: 2536-4383

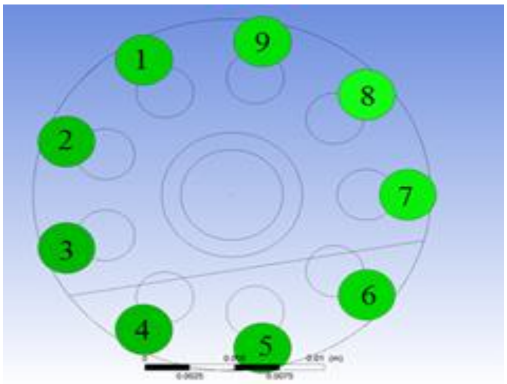


Figure 2. *Position of distributor channels*

Fluid Properties

R404A refrigerant is used as a refrigerant in the experiments. Refrigerant is introduced to ANSYS Fluent material list. Properties of the refrigerant are shown Table 3.

Table 3. *Properties of refrigerant (R404A)*

Molecular weight (kg/kmol)	96.7
Liquid phase density (kg/m^3)	1072
Gas phase density (kg/m^3)	40.38
Liquid phase viscosity (kg/(ms)	0.00016129
Gas phase viscosity (kg/ms)	0.00001197
Surface tension (N/m)	0.003

Experimental Method

A R404A tube-plate-fin evaporator with 9 circuits is placed in the experimental facility. Experimental facility is illustrated in Fig. 3. Experimental facility mainly involves calorimetric room and conditioning room. Fin and tube heat exchangers are located in the calorimetric room where the test operations are performed. The calorimetric room setup is consisted of a test section, where evaporator is installed; an air handling unit to maintain a constant air temperature and humidity; a refrigeration section to regulate the temperature and flow rate of the refrigerant feed to the test unit during the experiments.

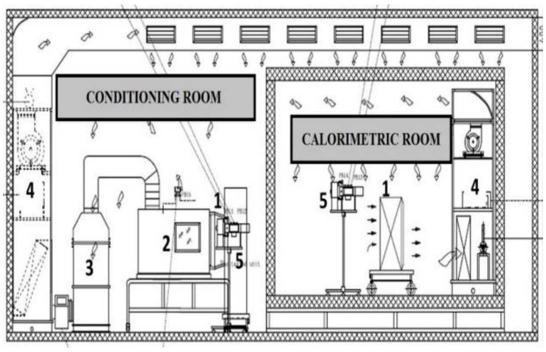


Figure 3. *Schematic of experimental R-404A refrigeration cycle (1. Product that is tested, 2. Air Receiver Room, 3. Flowrate Measurement Room 4. Air Handling Unit, 5. Air Sampler)*

The refrigerant used in these experiments is R404A and the refrigerant system had an auxiliary line operated with water for the purpose of controlling the refrigerant temperature at such as the condenser subcooling temperature. Besides, this refrigerant line have a shell and tube condenser used for the system's condenser and auxiliary evaporator in order to adjust evaporation pressure. An electronic expansion device (EXV) is also used. Cooling cycle with refrigerant distributor is depicted in Fig. 4. Table 4 shows the characteristics of unit cooler used in this study. Experimental conditions are shown in Table 5.

ISSN: 2536-4383

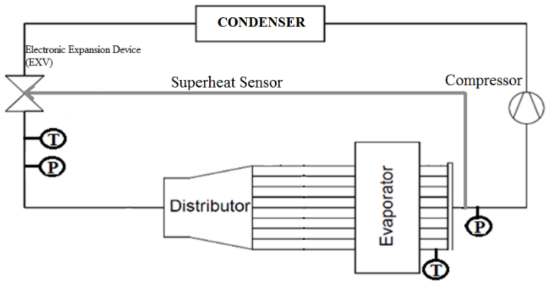


Figure 4. *Cooling cycle with refrigerant distributor*

In order to validate the simulation results, experiments are performed with both geometries. Fig. 4 demonstrates a vapor compression cycle with a refrigerant flow distributor. After electronic expansion device, refrigerant come to the distributor and is divided into multiple flow circuits.

Table 4. *Characteristics of Evaporator*

Parameters	Value
Design cooling capacity (kW)	13.5
Number of circuit	9
Number of rows	4
Fin Material	Aluminum
Fin thickness (mm)	0.15
Fin pitch (mm)	4
Tube (circuit) material	Cupper
Tube thickness (mm)	0.32
Tube inner / outer diameter (mm)	11.36/12

Table 5. *Experimental conditions*

Refrigerant Name	R404A
Evaporation Temperature (°C)	-4
Condensation Temperature (°C)	40
Subcooling Temperature (°C)	10
Superheating Temperature (°C)	3
Calorimetric Room Temperature (°C)	1

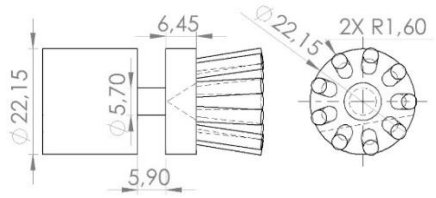
However, as the heat transfer rate of air side is calculated from electrical loads, that of refrigerant side is calculated from

$$\dot{Q}_{r,tot} = \dot{m}_r(h_{r,before\,exp\,valve} - h_{r,o})$$
(3)

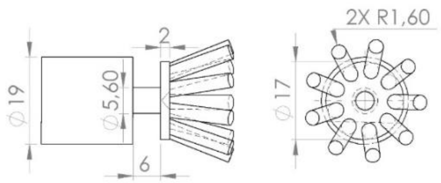
Where, $\dot{m}_r$ is the refrigerant mass flow, $h_{r,before\,exp\,valve}$ and $h_{r,o}$ are enthalpy values of before expansion valve and outlet of evaporator, respectively. The heat transfer rate of refrigerant side is admitted as cooling capacity because the uncertainty value of refrigerant side is smaller when compared with heat transfer rate of air side. Table 6 demonstrated the specifications of the instrumentation. Both distributors has nine channels and an orifice that is located at the centerline of the the distributor as it seen in Fig. 5.

Table 6. *Specifications of the instrumentation*

Measured variable	Instrument	Range	Uncertainty
Temperature (°C)	K-type thermocouple (PT100)	-50-500	± 0.3%
Pressure (bar)	Pressure transmitter	0-50	±0.3%
Humidity (%)	Hygrometer	0-100	±0.3%
Refrigerant flow rate (kg/h)	Flow meter	0-500	±0.6%



(a)



(b)

Figure 5*. Geometry of the distributor case 1 (a) and case 2 (b)*

In addition, the uncertainty of related parameters, tabulated at Table 7, is calculated according to Stephanie Bell's study 'A Beginner Guide to Uncertainty of Measurement' [13] The calculations are performed with the both distributor

Table 7*. Estimated uncertainties of parameters*

Parameters	Maximum Uncertainty
Heat transfer rate [kW]	2.35%
Refrigerant pressure difference [Pa]	0.259%

Results and Discussion

The distribution performance is evaluated by the difference in the mass flow rate and pressure drop in the distributor for both case. The maximum mass flow rates difference tells how uniformly the mass flow rate is distributed [6]. Two-phase mass flow rate distribution for case 1 remains almost the same because two-phase flow is distributed properly as it is seen in Fig. 6. The orientation and length of the orifice have the main impact on flow distribution.

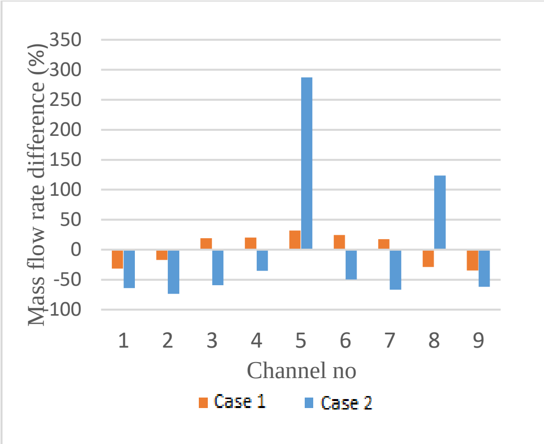


Figure 6. *Two-phase simulation results for mass flow distribution*

Fig. 6 also implies Case 1 distributor has much more uniform distribution of mass flow rate among the different channels. Channel no. 5 and no. 8 are more different than the average value for case 2.

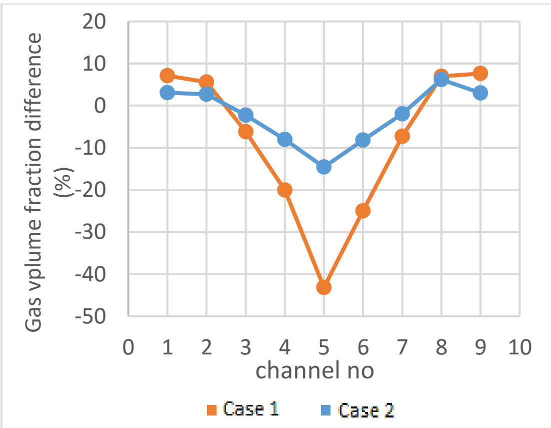


Figure 7. *Two-phase simulation results for gas volume fraction distribution*

ISSN: 2536-4383

Two-phase simulation results for gas volume fraction distribution can be seen in Fig. 7. Because of the diameter of the connecting pipe two-phase flow is not uniform at the orifice of the distributor. The maximum difference is occurred in channel 5 due to both orientation and the gravity effect. Simulation results of pressure drop for both case are shown in Fig. 8. Due to the expansion of refrigerant after nozzle part in the distributor, refrigerant pressure drop is increased rapidly as it seen in the Fig. 8.

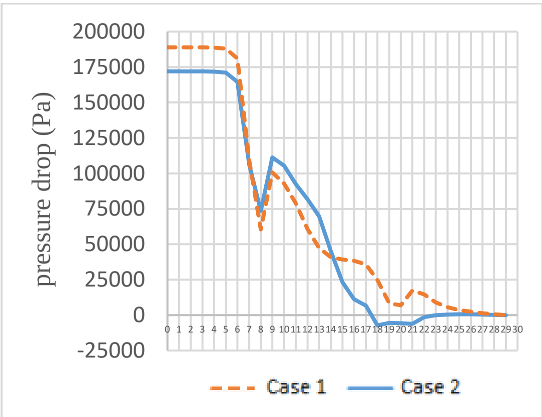


Figure 8. *Two phase simulation results for pressure drop in the distributor for case 1 and case 2*

Comparisons between the experimental and numerical results of pressure drops are shown in Fig 9. The ratio of pressure drop of the experimental result is slightly higher than the numerical results. The difference between numerical and experimental results of the pressure drop is less than 2% for both distributors. The computational pressure drop results of case 1 and 2 are in good agreement with the experimental results as shown in Fig. 9.

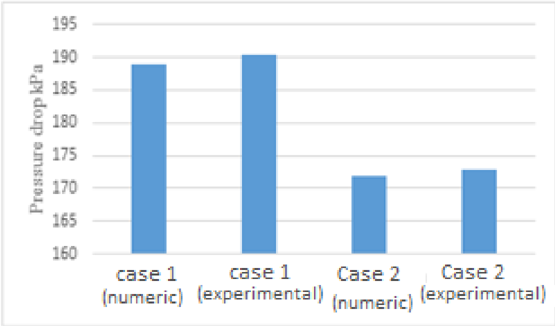


Figure 9. *Total pressure drop in the distributor experimental and numerical results*

Table 8 summarizes the experimental results. The numerical results of the pressure drop agree with the experimental results quite well.

Table 8. *Experimental results of evaporator with case 1 and case 2 distributor*

Experimental values	Case 1	Case 2
Evaporator pressure (kPa)	527	527
Fluid temperature before expansion device (°C)	29.89	30.09
Fluid pressure before expansion device (MPa)	1.53	1.58
Evaporator superheating (K)	3	3
Total pressure drop in evaporator (kPa)	202.399	184.909
Cooling capacity	12.8	11.5

Conclusions

In this study, it was expected to develop a cost-effective and reliable test method to assess the evaporator distributors. For this aim, a CFD simulation was performed to obtain the distributor flow configuration. Simulation results were validated with the experimental results for case 1 and case 2 distributors. The Eulerian

ISSN: 2536-4383

Multiphase Model was used to simulate two-phase flow in case 1 and case 2 distributors.

The results showed two-phase flow was divided equally into the each channel regardless to the mass flow rates for case 1. The pressure drops obtained from CFD analysis for case 1 and 2 distributors are in a good agreement with the experimental results. Finally, the difference between numerical and experimental results of the pressure drops is less than 2%.

Acknowledgement

This project was supported by Türkiye Bilimsel ve Teknolojik Arastirma Kurumu (Scientific and Technological Research Council of Turkey) (TUBITAK) under 2209/A program (project number:1919B011503311). The authors appreciate the support provided by FRITERM, A.Ş.

References

[1] Lee, J., Kwon, Y.-C., Kim, M.H., (2003) An improved method for analyzing a fin and tube evaporator containing a zeotropic mixture refrigerant with air maldistribution. Int. J. Refrigeration 26, 707-720.

[2] Kim, J. H., Braun, J., Groll, E. (2009), A hybrid method for refrigerant flow balancing in multi-circuit evaporators: upstream versus downstream flow control. Int. J. Refrigeration 32, 1271-1282, USA

[3] Nakayama, M., Y. Sumida, S. (2000), Development of a refrigerant two-phase flow distributor for a room air conditioner, International Refrigeration Conference at Purdue, pp. 313-319, USA

[4] Wen, M.-Y., Lee, C.-H., Tasi, J.-C., (2008) Improving two-phase refrigerant distribution in the manifold of the refrigeration system, App. Thermal Eng., 28:2126-2135.

[5] Gang Li , Steven Frankel , James E. Braun (2002), Application of CFD Models To Two Phase Flow In Refrigerant Distributors, International Refrigeration and Air Conditioning Conference, Paper 592, USA

[6] Gang Li , Steven Frankel , James E. Braun and Groll, E. A. (2002), Evaluating The Performance Of Refrigerant Flow Distributors, International Refrigeration And Air Conditioning Conference, Paper 593, USA

[7] Dong, Z. G. and Bean, J., 2006, Experimental research and CFD simulation on microchannel evaporator header to improve heat exchanger efficiency, International Refrigeration and Air Conditioning Conference. Paper 753.

[8] Gang Li , Steven Frankel , James E. Braun and Groll, E. A. (2005), Application of CFD Models To Two Phase Flow In Refrigerant Distributors, HVAC&R Research, VOLUME 11, NUMBER 1

ISSN: 2536-4383

[9] Fei, P., Hrnjak, P.S. (2004) Adiabatic Developing Two-phase Refrigerant Flow in Manifolds of Heat Exchangers in Manifolds of Heat Exchangers, ACRC, University of Illinois, 2004. Tech. Rep. TR-225.

[10] Abishek, S., King, A.J.C., Ramesh N., (2017), Computational analysis of two-phase flow and heat transfer in parallel and counter flow double-pipe evaporators, International Journal of Heat and Mass Transfer, Volume 104, January Pages 615-626.

[11] Yuan P., Jiang G.B., He Y.L., Tao W.Q., Performance simulation of a two-phase flow distributor for plate-fin heat exchanger (2016) *Applied Thermal Engineering 99 1236–1245

[12] ANSYS Meshing User's Guide, (2013), ANSYS, Inc. Release 15.0, USA.

[13] Bell S., (1999), A Beginner's Guide to Uncertainty of Measurement, Measurement Good Practice Guide", No:11(2), Thermal and Length Metrology National Physical Laboratory, Middlesex, United Kingdom.