CFD Simulation of Heat Transfer of Liquid Nitrogen in Heat Exchanger

Rahul Patel[1], Rahul Paliwal[2]

[1]PG Student, [2]Associate Professor, L. D. College Of Engg ., Ahmedabad Gujarat, India.

Abstract: In the present work computational fluid dynamics (CFD) was used to investigate the two phase cross flow of liquid nitrogen and gaseous nitrogen in tub-in-tube heat exchanger. The behaviour of the flow was investigated in heat exchanger along the entire tube length. The study has further been expanded to see pressure drop and heat transfer phenomena in two phase flow of liquid nitrogen. ANSYS CFX and FLUENT modules are used to get the required simulation results. Simulation results are in good agreement with the data.
Keywords: CFD simulation, Heat transfer, Two-phase Flow, Flow boiling, Liquid nitrogen, Heat exchanger

I. INTRODUCTION

Applications of cryogenic can be found in variety of fields such as physics, biology, medicine, engineering and industry [1]. Liquid nitrogen, liquid hydrogen, liquid oxygen is cryogenic fluid. Liquid nitrogen is available easily so it has wide use in industries and also for conducting different experiments. When liquid nitrogen flows through tube gets heated and under goes a phase change from liquid to vapour. It is called flow boiling which has two phase flow. In case of cryogenic fluid, flow boiling has a high heat transfer rates which makes it widely applicable for such as to cool devices like rocket motors and nuclear reactors. There are many studies on the refrigerants and also some on liquid nitrogen. But most of them are done mainly for micro channels. Kandlikar [2] and thome [3] has reviewed that the flow boiling heat transfer is divided into nucleate boiling dominated region and convective boiling dominated region. Some studies [4], [5] of heat transfer, pressure drop and critical heat flux in a small tube with refrigerant as a working fluid show the heat transfer is influenced by the nucleate boiling. S. L. Qi et al. [6], [7] investigated the flow boiling of liquid nitrogen about heat transfer and pressure drop and found that ONB can be marked by the phenomena that mass flux drops suddenly while pressure drop increases. They also reported that for low mass quality region, heat transfer is governed by nucleate boiling and heat transfer coefficient is dependent on heat flux and pressure, but independent to mass flux. Whereas for high mass quality region, heat transfer coefficient is dominated by mass flux, insensitive to heat flux. Most of the researches about CFD simulation of two phase flow are related to water [8], [9], refrigerants [10], [11],[12] and other liquids. There are not much reported researches about CFD simulation for two phase flow of cryogenic fluids.

II. CFD SIMULATION

A. Operating Conditions and Physical Properties

CFD simulations have been carried out to observe the heat transfer phenomena of flow boiling liquid nitrogen through tube-in-tube heat exchanger. Experimental data has been taken from the thesis [13]. Simulation has been conducted in the following conditions. The tube-in-tube heat exchanger has a length of 32.23 m. Both the tubes are made of copper. Liquid nitrogen flows through inner tube while gaseous nitrogen flows in annular space. Diameter and thickness of inner tube is 54.8mm and 2.77mm respectively. Diameter of outer tube is 108mm and thickness is 3.05mm respectively.

Table 1 shows other physical properties and operating conditions, which have been applied for the simulations.

TABLE I Physical Property and Operating Conditions Used for Simulations

Parameter	Value
LN_2 Density	777.72 Kg/m^3
LN_2 Viscosity	1.57×10^{-4} Pa-Sec
LN_2 Specific heat capacity	2.10×10^3 J/kg-K
LN_2 Thermal Conductivity	0.02535 W/m K
GN_2 Density	8.748 Kg/m^3
GN_2 Viscosity	5.80×10^{-6} Pa-Sec
GN_2 Specific Heat	1.120×10^3 J/kg K
GN_2 Thermal Conductivity	7.821×10^{-3} W/m K
Wall Density	7920 kg/m^3
Wall Specific heat capacity	0.375×10^3 J/kg-K
Wall Thermal conductivity	8.26 /m K

B. 3D Modelling and Meshing

Here the profile has been generated for the length of the tube. The part file for each separate domain has been created in CREO. After that step file of assembly have been created and called for investigation in ANSYS CFX and FLUENT module.
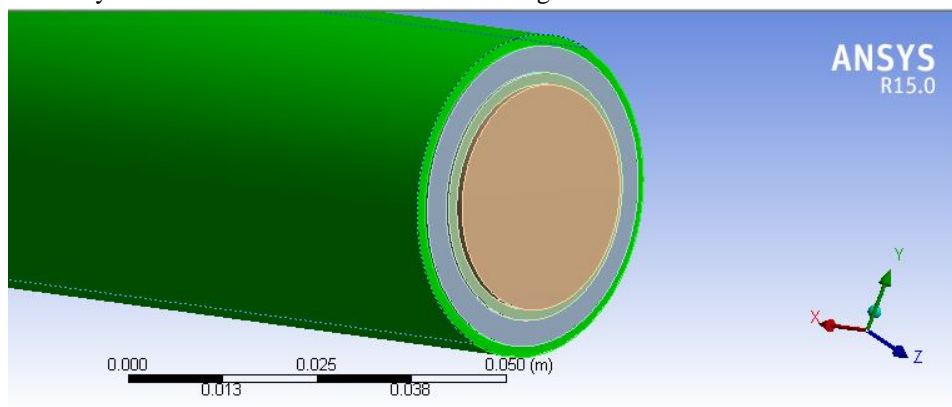

Fig. 1 Copper Tube in Model Assembly

Fig. 1 shows the outer tube which of coper material with its specified dimention. One another pipe is going to be assembled with this pipe. The fluids which in this case are liquid nitrogen and gasseous nitrogen are going to be poured inside the assembly of these parallel tubes.
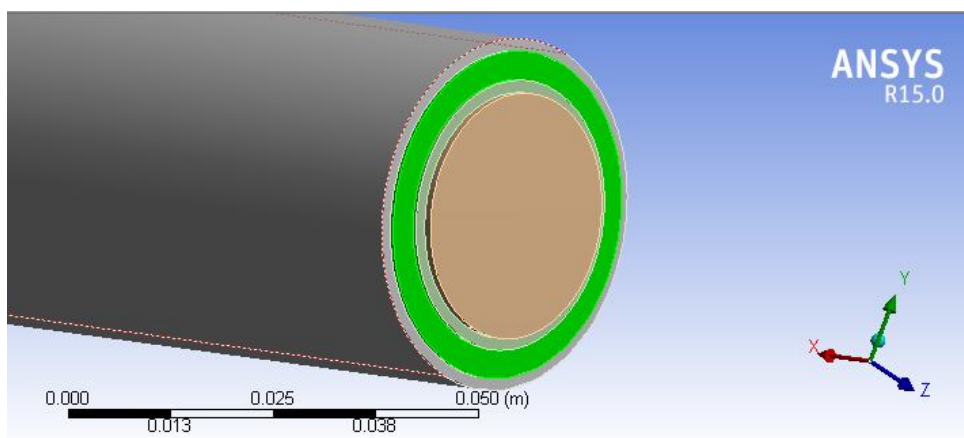

Fig. 2 Nitrogen Gas Passage in Parallel Flow Assembly

Fig. 2 covers gaseous domain containing nitrogen gas. The nitrogen gas flows in between annular space between tubes. Here also the gas is flown with specific flow criteria of nitrogen gas shown in TABLE I.
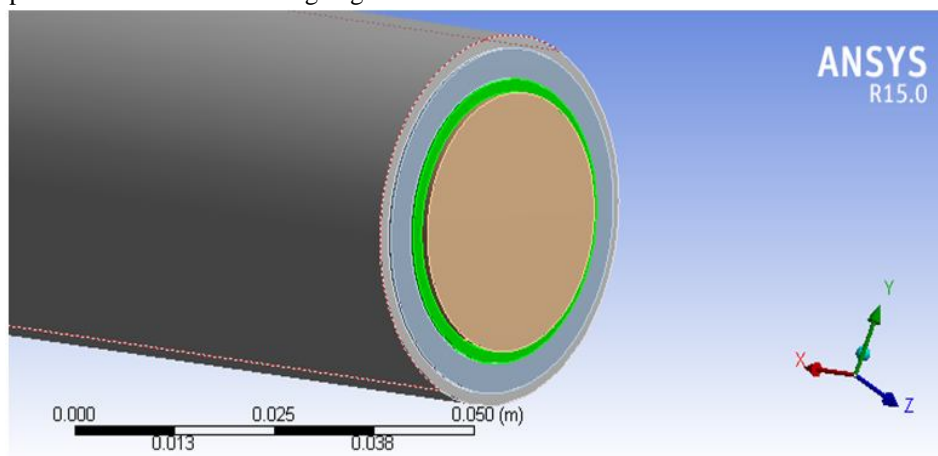

Fig.3 Internal Copper Pipe Containing Liquid Nitrogen

Fig. 3 covers internal copper tube containing liquid nitrogen. The tubing walls are considered to be adiabatic and conductive by nature. Here temperature of gaseous nitrogen is greater than that of liquid. So heat will tend to flow from gaseous nitrogen towards liquid nitrogen through the inner tube. Therefore conductive nature of tube material is taken into account for the simulation.

Fig. 4 Covers liquid nitrogen within it which helps in transferring the heat between surfaces. Fig. 5 represents meshing of tube assembly. The elements used are finite shell elements for copper tubes. Hexahedral and tetrahedral elements are used to present fluid domains. TABLE II shows total number of nodes and elements generated by meshing.
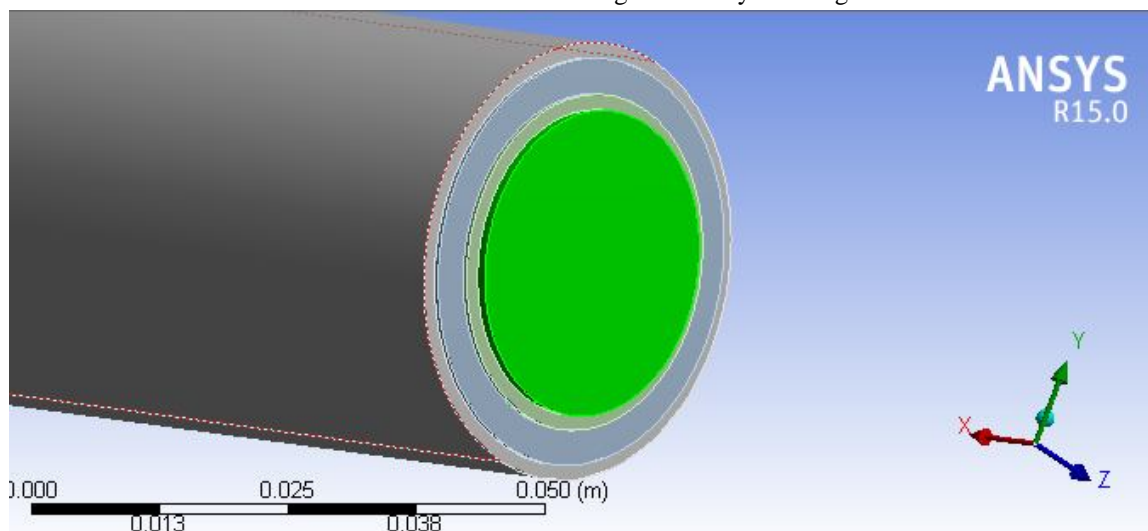

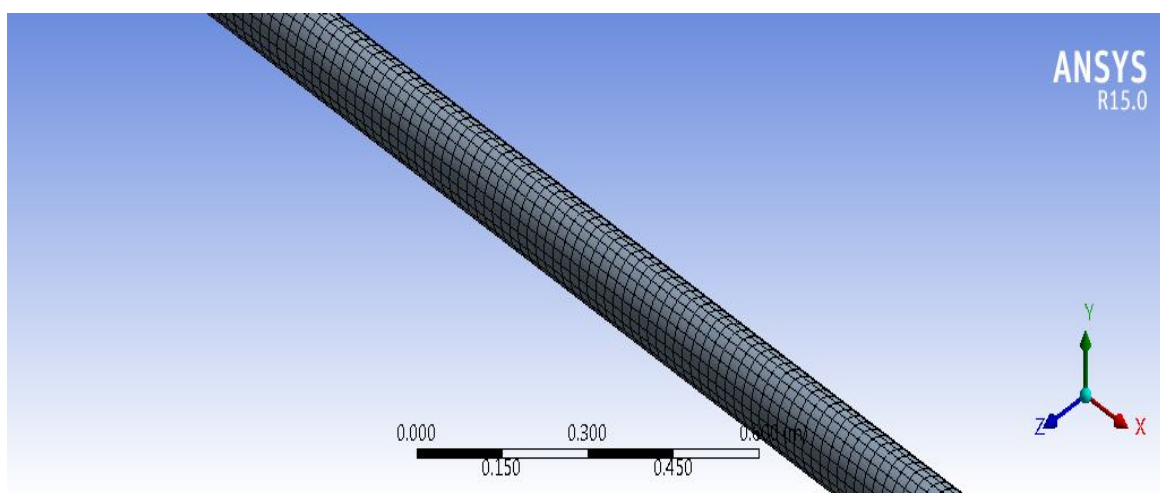Fig. 4 Liquid Nitrogen Concideration

Fig. 5 Meshed Profile of Assembly of Fluid and Solid Interface Profile in Tube

TABLE II
Data Regarding Node and Elements

Nodes	777383
Elements	137202

III.RESULTS AND DISCUSSION

The pipe is too much large to observe so after defining all the four domains the constrained assembly has been cut in different small sections so that it can be more effective to observe central zone.

Fig. 7 represents total heat flux developed inside the tube at cut sectioned zone. It shows vector of presentation for heat flux. The heat flux exchange at central zone is less and at the zone of intersection the heat flux is more. This shows the heat exchange capabilities of system. Maximum heat exchanges have been found at the zone of interface of fluid and solid interface inside.
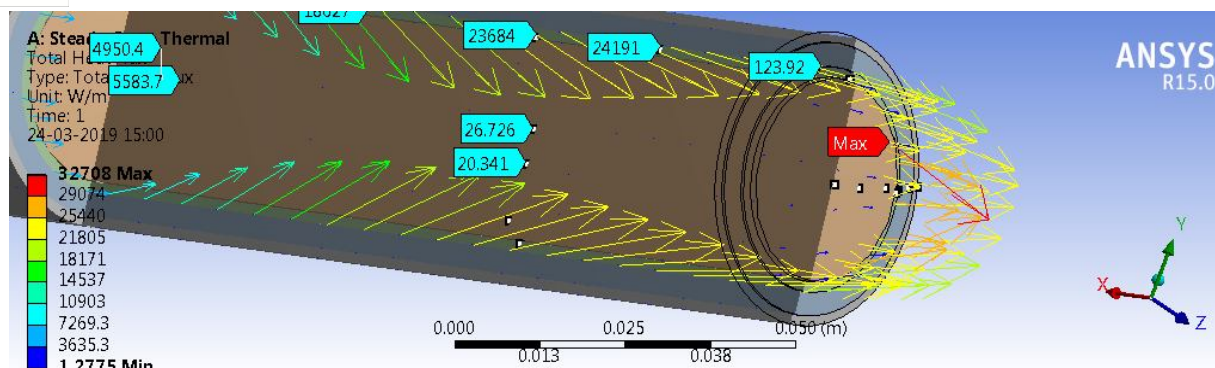
Fig. 7 Total Heat Flux in Tube

The fig. 8 represents tube heating with directional heat exchange. The minimum heat flux been found at the opening of tube, inside the flow region the heat flux varies between maximum to minimum value depending on contact interface.
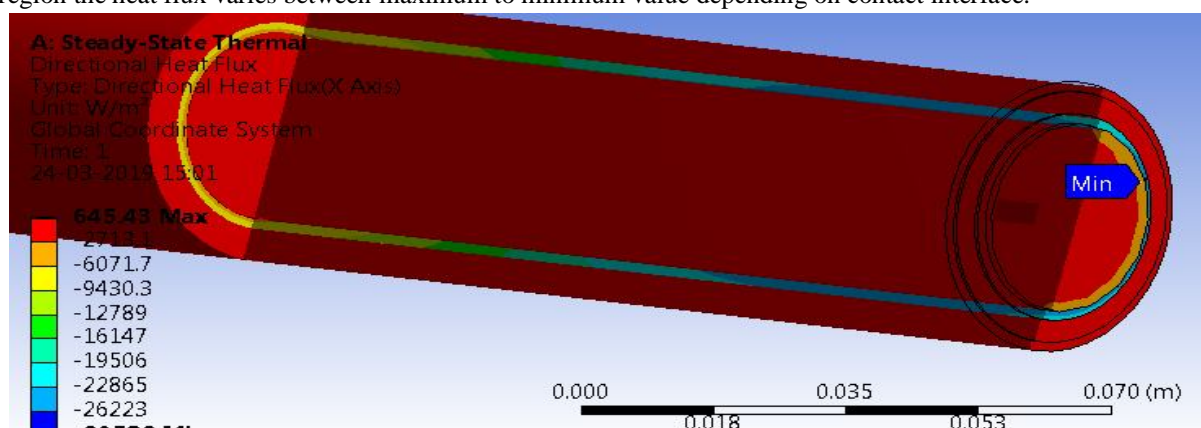

Fig. 8 Directional Heat Flux

Fig. 9 represents the heat flux inside the copper tube which clearly shows a conductive behaviour of heat transfer, from outside to inside. It also helps in supplying fluidic heat from out to inside direction of tube.
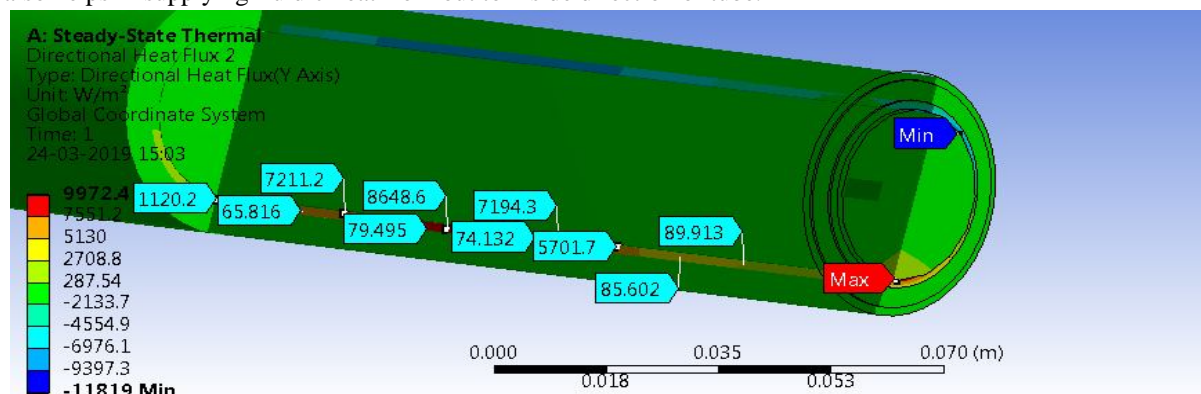

Fig. 9 Heat Flux Distribution in Tube

The longitudinal section is not able to represent all information regarding profiles so transverse cut section have been taken under consideration and here the profile shows heat distribution in all domains inside out. The fig. 10 represents fluid domain. The solid domains are removed to see the heat transfer at central zone. The heat at central zone is more and at the contact interface is less. This also covers velocity profile which shows the central zone gets velocity changes due to interface contacts. The pressure profile observation can also help in studying the mass flow rates.
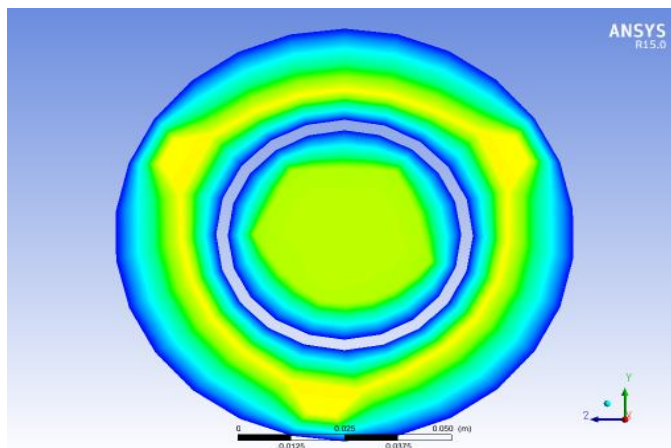
Fig. 10 Heat Distribution in Plane Central Cross Section

Fig. 11 shows that Flow pressure profile is presented here in internal fluid domain and outer fluid domain. The flow velocity variation is not major in internal domain as the region pr cross section of flow is consistent. The outer flow profile shows major variation as the pressure profile changes more as the hollow profiles for observation is available.
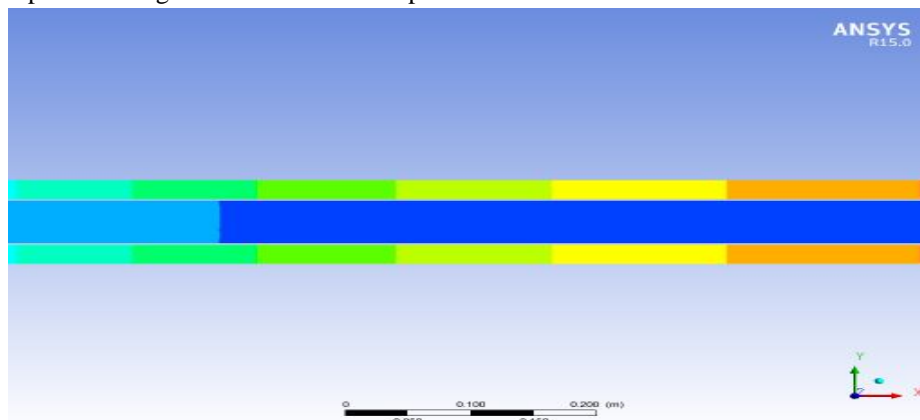

Fig. 11 Flow Pressure Profile in Liquid and Gaseous Nitrogen

Fig. 12 shows temperature gradient vector plot for fluid domains inside inner tube and in-between inner and outer tube. The more dense vectors shows more interactive changes and the less dense regions represents even distribution of temperature at various zones of flow.
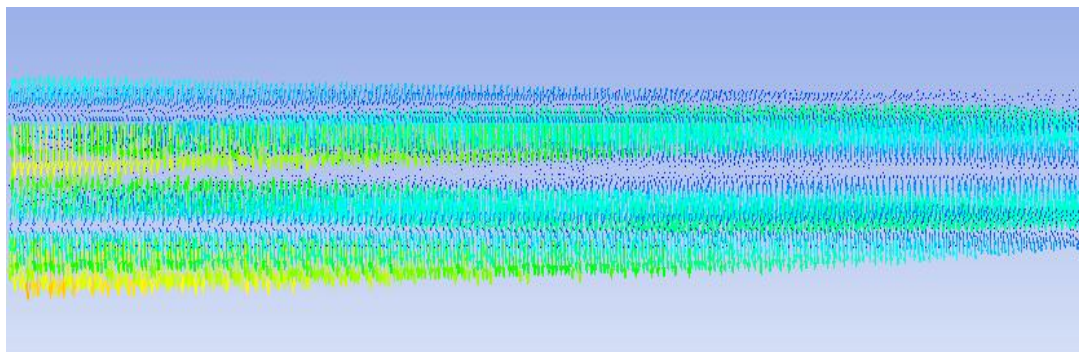

Fig. 12 Temperature Gradient Vector Observations

In Fig. 13 the results of CFD is compared against thesis results. CFD results are differing from the thesis results but the behaviour followed by the results is similar to thesis results. Here we can clearly see that a trend line generated by the CFD analysis follows same trend lines generated by the thesis results. The more physics consideration can give exact confirmation regarding results.
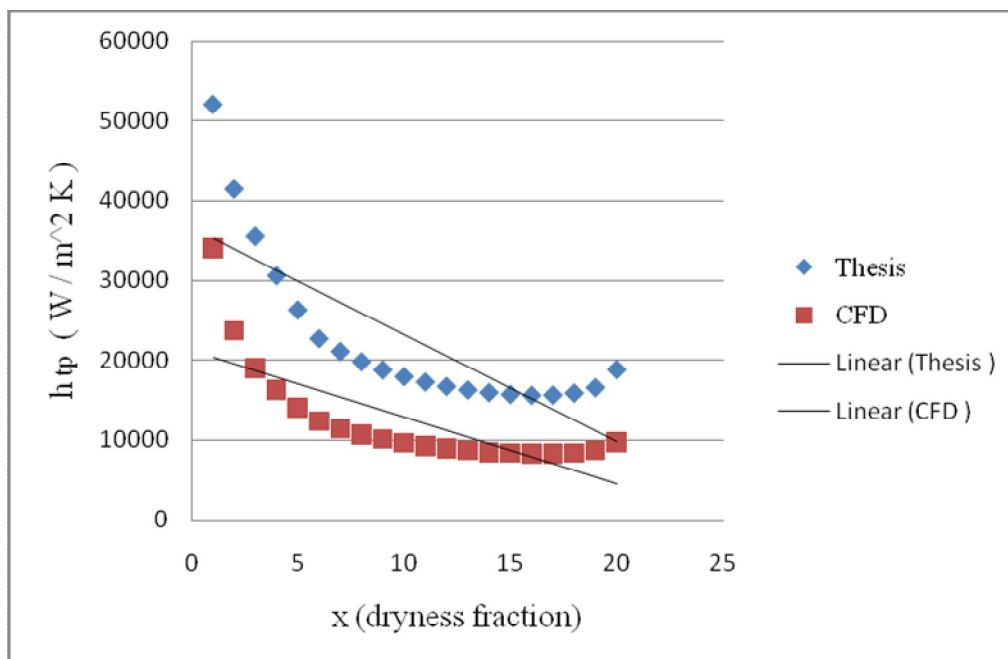
Fig. 13 Thesis vs CFD Result for Heat Tansfer Coefficient

IV. CONCLUSIONS

In this paper two-phase flow heat transfer of liquid nitrogen is investigated using ANSYS CXF module. The plotted results shows temperature variation representing the calculative result profiles. The mass density gradient plot also represents density variation in terms of vector density change. Simulation results differing from the data but shows same trend with it which can be improved with more physics considerations.

REFERENCES

[1] J. C. Erdmann, "Application of cryogenic techniques to materials research," Washington, USA: Elsevier, vol. 9, pp. 411-421, December 1969.
[2] S.G. Kandlikar, "Fundamental issues related to flow boiling in mini-channels and micro-channels," Experimental Thermal and Fluid science, vol. 26, pp. 389-407, 2002.
[3] J. R. Thome, "Boiling in micro-channel: a review of experiment and theory," International Journal of Heat Fluid Flow, vol. 25, pp. 128-139, 2004.
[4] G. M. Lazarek, S. H. Black, "Evaporative heat transfer, pressure drop and critical heat flux in a small vertical tube with R-113," International Journal of Heat Mass Transfer, vol. 25, pp. 945-960, 1982.
[5] T. N. Tran, M. W. Wambsganss, D. M. France, "Small circular and rectangular channel boiling with two refrigerants," International. Journal Multiphase Flow, vol. 22, pp. 485-498, 1996.
[6] S. L. Qi, P. Zhang, R. Z. Wang, L. X. Xu, "Flow boiling of liquid nitrogen in micro-tubes: Part I – The onset of nucleate boiling, two-phase flow instability and two-phase flow pressure drop," International journal of Heat and Mass Transfer, vol. 50, pp. 4999-5016, October 2007.
[7] S. L. Qi, P. Zhang, R. Z. Wang, L. X. Xu, "Flow boiling of liquid nitrogen in micro-tubes: Part II –Heat transfer characteristics and critical heat flux," International journal of Heat and Mass Transfer, vol. 50, pp. 5017-5030, October 2007.
[8] C. K. Sandra, De Schepper, J. H. Geraldine, B. M. Guy, "CFD modeling of all gas-liquid and vapor-liquid flow regimes predicted by the Baker chart," Chemical Engineering Journal, vol. 138, pp.349-357, 2008.
[9] Q. H. Mazumder, S. A. Siddique, "CFD Analysis of Two-Phase Flow Characteristics in a 90 Degree Elbow," Mechanical Engineering University of Michigan-Flint 303 East Kearsley Street, Flint, MI 48502, USA, vol.3. pp. 165-175, 2011.
[10] P. Bhramara, V. D. Rao, K. V. Sharma and T. K. K. Reddy, "CFD Analysis of Two Phase Flow in a Horizontal Pipe – Prediction Pressure Drop" International Journal of Mechanical and Mechatronics Engineering, vol.2, pp. 476-482, 2008.
[11] Bostjan Koncar, Eckhard Krepper, "CFD simulation of convective flow boiling of refrigerant in a vertical annulus," Nuclear Engineering and Design, vol.238, pp. 693-706, 2008.
[12] Pramod Kumar, M. R. Nagraj, "CFD Analysis of Fluid Flow and Heat Transfer in Two Phase Flow Micro-channels," International Journal of Engineering Research & Technology, vol. 2, pp. 2132-2143 August 2013.
[13] Manoj Gupta, "design, optimation and development of controlled cool down system for thermal shield of SST-I," Mechanical Engineering thesis, L. D. College of Engineering, Ahmedabad, Gujarat, India, pp. 111-125, 2007.

INTERNATIONAL JOURNAL
FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)