ISSN: 0974-5823 Vol. 7 No. 2 February, 2022

International Journal of Mechanical Engineering

The Validation of Heat Transfer Characteristics of Pipes System Array inside the Pressurized Cylindrical Container

Hiyam Majeed Esmael*

The Technical Engineering College, The Midland Technical University, Baghdad, Iraq

Ammar A. Beddai

The Technical Engineering College, The Midland Technical University, Baghdad, Iraq

Majid H. Majeed

The Technical Engineering College, The Midland Technical University, Baghdad, Iraq

Abstract - Combining different four similar circular rectangular tubes effectively solved the practical investigation of convective heat transfer research in pressured container. The current experiment was conducted using a pressurized tank with a 0.3 m diameter and 0.45 m height within 0.43 m height and 0.025 m side length internal rectangular pipes. Pressure difference (1.5-2 bar), pipe Ra configurations, and container orientations are among the characteristics employed. For both the cold and hot sides, the results are Nusselt numbers. Both sides have a reverse reaction, and the parametric analysis reveals the appropriate behavior due to the interplay of gravity and pressure forces. Whenever the upper pipe is enclosed horizontally in 2 bar, the optimal circumstances are found, with a thermal transfer enhancement of roughly 24 percent. The maximum heat transfer error validation with numerical simulation is about 18.5 % and with previous work investigation is approximated to 19.01 %.

Index Terms - Enclosure, Free convection, Nu, Rectangular pipe.

INTRODUCTION

Free convection heat transfer is necessary in a wide range of industrial and domestic applications. The most important topic for manufacturing proposal, process, inquiry, and development is the analysis of heat transfer for cooling or heating purposes. The growth of heat transfer by employing exceptional techniques improves higher mean excellences, resulting in more efficient power conservation and administration [1]. Raising the heat transfer coefficient by means of physical characteristics configuration and by area by geometrical and design configurations limits the rate of heat transfer. Because to boundary layer interactions between the cold and hot surfaces, the complexity of free convection grows in an irregular pattern, forming an unusual free stream structure and an unusual boundary layer, similar to forced convection [2], [3]. The behavior of free convection is explained by the density difference in gravity direction caused by buoyancy [4], [5] A heated surface in a circular container is an active example of free convective heat transfer. Within a hot surface, the momentum rate of natural convection is minimal (no-slip condition). By the action of displacement and heat transmission among fluid particles, the momentum rate rose significantly away from the heated domain inside the free stream. Understanding the active free convection criterion helps with the design, operation, and thermal analysis of a variety of industrial and urban heating and cooling systems [6].

Previous research looked at natural convection heat transfer inside an enclosure under various operating parameters and geometries such as: Kitamura et al. (1999), [7], investigated the heat transfer free convection by displaying the free stream configuration in the region of the external surface of a cylindrical pipe and calculating the external surface temperature allocation, the natural convection behavior. According to the authors, 3D free stream separation occurs first at the pipe's uneven limits and then transforms to the turbulent non-fully developed scenario. The researchers also discovered that the confined Nu increases significantly in section of the non-developed region and turbulent stream, despite the fact that these sections are used on an undersized division of the pipe's exterior surface. Empirical expressions for the man Nu were obtained as prevails in the following:

$$Nu = 0.6\, Ra_D{}^{*0.2} \quad \text{, at } 3\times10^{8}<Ra<2.5\times10^{10} \qquad (1)$$

Habeeb, 2010 [8] investigated The impact of a hot square pipe placed on a cooled elliptical pressurized enclosure of a laminar free convection system numerically. For two positions of the main axis of the elliptical pressurized tank, horizontal and vertical), this statement was solved using the PDE Algorithm. Speed vectors, streamlines, heat lines, and Nu were the most common outcomes. The results revealed that increasing the amplitude of the enclosure's main axis increased the average Nu and decreased the momentum strength for the whole Nu. However, when Ra increased, the results changed dramatically. Salman and Shamikh, in (2012) [9] investigated numerically of free convection behavior in a closed pressurized annulus region is formed by heated square pipe in a larger constant temperature surface of cooled cylinder pipe. The numerical simulation was carried out by using profitable CFD package (FLUENT 6.3). The average Nu from the investigational results was compared with that resulted from the CFD package. The stream lines and heat lines characteristics for various operating and geometric conditions were involved from side to side stream lines and isotherms contours that obtained from the CFD package. The Aspect ratio range from 0.2 to 0.5 was also used. The results prevailed that the Nu is affected directly by aspect ratio, the increasing of aspect ratio leads to Nu increasing. Also the consequence prevailed that as Ra increasing leads to Nu increasing with significant effect. Altaee et al., (2017) [10], studied the numerical analysis of a heated triangular body inside a cold rectangular chamber filled with air. The influence of triangular body tilt and Ra on the numerical study was examined using the ANSYS 16 CFD software. They were able to create streamline counters with varied inclination degrees and Ra. Pandey et al., in (2019) [11] investigated numerically and empirically, a full description of natural convection in enclosures with and without interior bodies. Square, circular, and elliptical cylinders are frequently used as internal bodies. Ali (2017) [12] analyzed the influence on located and average natural convection flow from sloped rectangular tubes in air at 30 incidence angles to the horizontally, 45 inclination angles to the horizontal, and 60 inclination angles to the horizontal. His research indicated that the correlation obtained for each inclination angle indicated that natural convection seems to have an insufficient reliance on the angle of inclination; thus, overall correlations were acquired for any and all cylinders at all incidence angles for the laminar regime: one for local laminar profiles and the other for multilateral laminar characteristics.

The impacts of many factors on the flow regime and thermal fields, such as internal body location, Rayleigh number, aspect ratio, inclination angle, and number of internal bodies, have been thoroughly explored. The scope of present investigation is to develop parametric trend of Ra, pressure and enclosure orientation on the heat transfer performance.

THE METHODOLOGY

The investigation of natural convection inside a pressurized annular tank with four rectangular pipes is part of the experimental investigation. The goal of this experiment is to see how heat fluxes, the number of heated rectangular pipes, the orientations of the pressured vessel, and the pressure affect the heat transfer coefficient and Nusselt number. For this study, precise measurement instruments were utilized to collect data, and a high-quality statistical technique was applied to discover the empirical connection. Figure 1 shows the experimental set up. The pressurized tank, pipes with inner heaters, ambient air tubes, air compressor, variac, voltmeter, ammeter, pressure gauge, and data-logger are the major components of the experimental rig. The current work's rig configuration is used to test the various parameters. The pressurized vessel is a cylindrical container with a thick iron wall and dimensions of 0.45 m in height and 0.3 m in diameter. Four rectangular holes at the top of the vessel, the hot four square pipes are placed in fixed distance. The 0.11 m difference in distance between the center of each pipe. The two pipes include heaters with a height of 0.4 m and a maximum power of 1000 W that are fixed parallel to the tube to provide a consistent heat flow. The tube's power may be varied using a manual voltage regulator, whereas the other pipes are merely filled with ambient air. The pipes have a height of 0.43 m and a side length of 0.25 m. During the experiment, the pressure vessel is kept sealed and insulated.

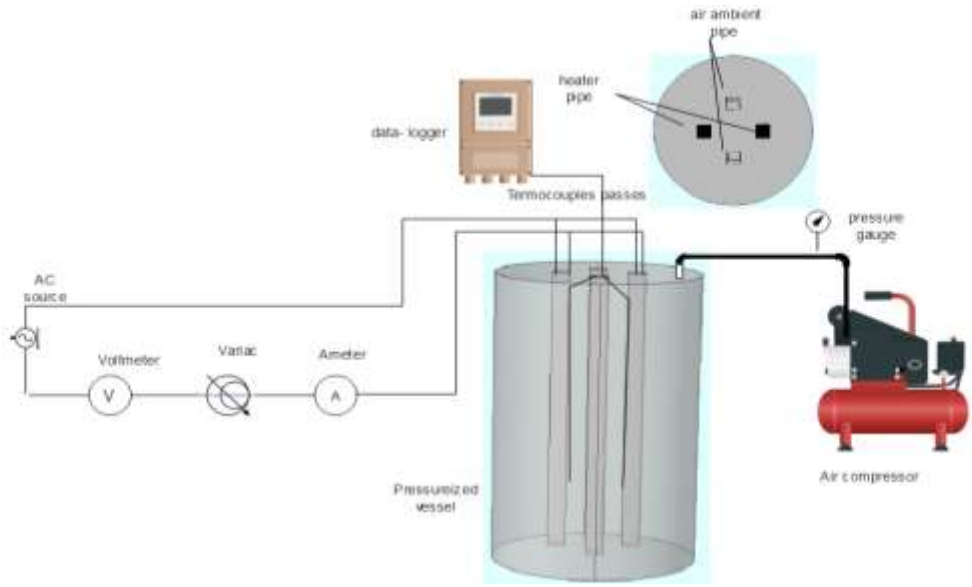

FIGURE 1A

RIG SETUP OF EXPERIMENTAL WORK

The following components can be used to summarize the experimental procedure: The compressor is turned on until the pressure within the pressurized vessel reaches the specified level. The heaters are turned on, the voltage regulator adjusts the voltage to the desired level, and the temperature is measured at steady state, when the temperature values are steady constant. The procedure can be repeated for different voltages (10-110 V) at gauge pressures (0.5-1 bar). To accomplish free convective heat transfer, the aforementioned pressures and voltages can be utilized for one heater instead of two heaters, and three pipes will be filled with ambient air. For the given characteristics, the pressurized vessel can be used both vertically and horizontally.

THE DATA COLLECTION

The data was collected as temperature distribution inside circular enclosure. The constant surface heat flux can be calculated from the following expression:

$$q = \frac{V \times I - Q_{rad}}{A} \qquad (2)$$

Where: V is electrical potential, I is electric current, A is outer surface area of inner cylinder, and Q_{rad} is the heat transfer by radiation between circular cylinder and triangular cylinder which can be calculated as follows:

$$Q_{rad} = \frac{\sigma\left[(T_i+273)^4-(T_o+273)^4\right]}{\left[\frac{1-E_1}{AE_1}+\frac{1}{AF_{12}}+\frac{1-E_2}{AE_2}\right]} \qquad (3)$$

Where

T_{in} : is the average hot temperature of inner square cylinder wall

T_{out}: is the average cold temperature of outer enclosure

σ = Stefan Boltzmann constant = 5.66×10-8 W/m2 °K4

E1=E2= emissivity of the iron inner and outer surfaces=0.21.

F1-2=Radiation View factors≈0.1

The heat transfer coefficient can be calculated from the following expression:

$$h = \frac{q}{T_{in}-T_{\infty}} \qquad (4)$$

Where T_∞ is average hot air temperature inside the enclosure.

The Nusselt number can be calculated from follows:

$$Nu = \frac{hL}{\kappa} \qquad (5)$$

The Rayleigh number can be calculated from the following:

$$Ra = \frac{g\beta(T_{in}-T_{out})L^3\, \mathrm{p}^2}{\mu^2}\, Pr \qquad (6)$$

And $\mathrm{p} = Mwt\frac{P}{RT}$

The logarithmic mean Nusselt number data can be fitted as linear relation with logarithmic Rayleigh number by using Ms-Excel trend-line to obtain the empirical correlation for each as expressed as follows:

$$Nu = cRa^n \qquad (7)$$

Where c and n are empirical constants which can be change by changing the enclosure

All the air physical properties $\rho, \mu, \nu, and \kappa$ were evaluated at the average mean film temperature (T_f).

$$T_f = \frac{T_h+T_\infty}{2} \qquad (8)$$

THE RESULTS AND DISCUSSION

1. The Effect of Pressure on Nusselt Number

Figures 2 to 4 show the Nu vs. Ra plots for various P where the vertical orientation of two pipes configuration are applied and Δx distance for cold pipe. The effect of operational pressure on Nu has reverse behavior to hot side. The increasing of operational pressure leads to decreasing the Nu values of Δx =16 cm, the heat transfer performance decreases by increasing the pressure and return to increase by increasing the operation pressure above 1.75 bar because of increasing density difference leads to Ra increasing.

The present figures indicate the increase of pressure increase the cold Nu and decrease the hot Nu. For vertical equal areas (hot and cold pipes in two configurations), the pressure increasing means higher amount of cold fluid dropping toward the gravity center while lower amount of hot fluid will be raised due to density increasing. For Δx=16 cm, The longer distance will reduce the effect of pressure between the cold and hot regions. The shorter distance where the higher pressure promotes higher cold fluid dropping makes the less temperature difference in cold region and the hot region fluids will be cooled significantly making the higher temperature difference The empirical correlating of Ali et al 2017 is applied to validate with present work.

$$Nu = 1.109\, Ra*^{0.193} \ \text{for} \ 10^7 < Ra* < 2\times10^{12} \qquad (9)$$

Where Ra*=Ra × Nu

The solution of momentum and energy equation have been included in present work by coupling and solving laminar fluid flow and heat transfer by using COMSOL Multiphysics. The physical properties of air (density, viscosity, thermal conductivity, heat capacity) are also definite in the software. In this work, the coupled equations of heat transfer and momentum transport were used to describe the natural convection behavior by obtaining stream lines, velocity profile and isothermal lines distribution.

The main transport equations that used in the present work are:

- **Continuity equation**

$$\frac{\partial \rho}{\partial t} = -\nabla . (\rho U) \tag{10}$$

- **Momentum equation**

$$\rho (U . \nabla) U = \nabla \left(\mu \left(\nabla U + (\nabla U)^T \right) \right) + \rho\, g \tag{11}$$

- **Energy equation**

$$\rho\, Cp\, U . \nabla T = \nabla (-k\, \nabla T) \tag{12}$$

The boundary conditions are as shown in Figure 3. 1, the free convection boundary conditions:

- The outer surface of heated pipe of constant heat flux.
- The constant surface temperature of ambient air pipes.
- The pressure of vessel.

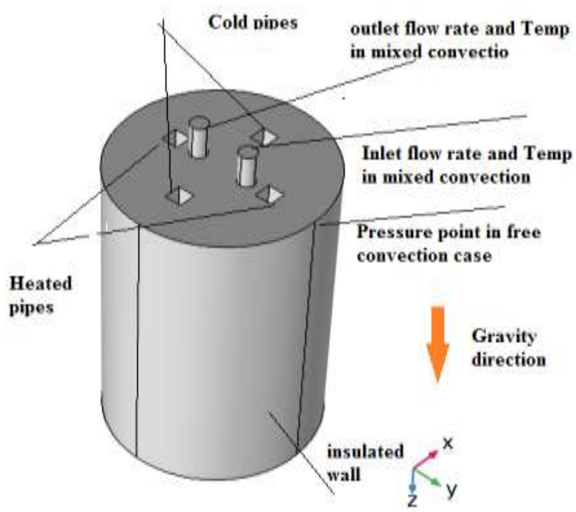

FIGURE 1B

BOUNDARY CONDITIONS OF PRESENT INVESTIGATIONS

2. The Effect Pressure and Diagonal Distance

Figures 2-4 show the Nu vs. Ra plots for various Δx where the vertical orientation of two pipes configuration are applied and various p or hot pipe. The increasing of diagonal distance decreases Nu at P=1.5 bar. At higher operation pressures (1.75 and 2 bar), the Nu increases by increasing the diagonal distance.

Figures 5-7 show the Nu vs. Ra plots for various Δx where the vertical orientation of two pipes configuration are applied and various p or cold pipe. The reverse hot pipe response is observed but with close values for Δx= 8 and 11 cm with significant decreases for Δx=16 cm.

The low pressure (1.5 bar) provides the high thermal resistance for hot side making the temperature difference between the hot surface and fluid domain is maximum as possible unlike the cold stream where the temperature difference is minimum. For the rest of pressures, i.e. higher operational pressures, the pressure force acts as modifier to heat transfer rate minimizing the thermal resistance in hot region and vice versa for cold side.

The another reason is the thermal contact time, the increasing of thermal contact time in hot stream leads to decrease the thermal contact time in cold stream and vice versa some time, the increasing of contact time in any region leads to decrease or increase of temperature difference depending upon displaced volume due to free convection. The presence of high volume transport from hot region to cold region has the low transportation of cold fluid volume to hot region making the minimum mixing tendency in hot region and maximum fluid particles mixing in cold region in low pressure for high Δx. The Δx increasing would make a restriction to fluid transportation in one compartment and effective transportation in the other.

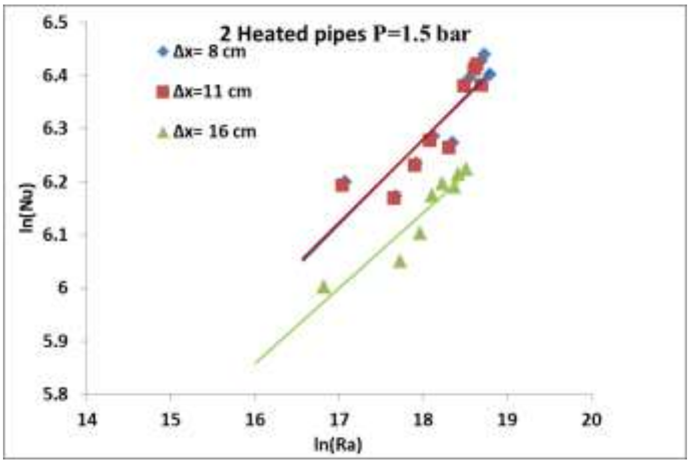

FIGURE 2

HOT SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=1.5 BAR)

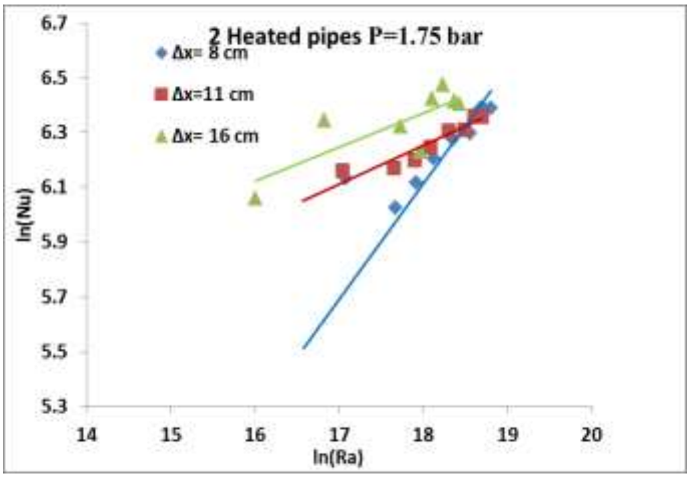

FIGURE 3

HOT SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=1.75 BAR)

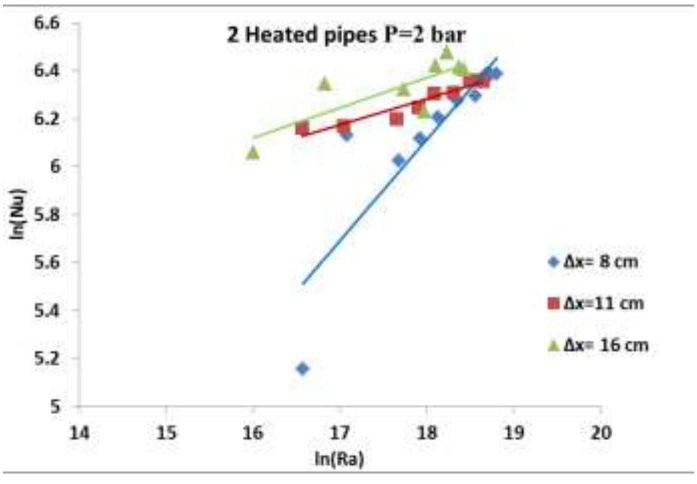

FIGURE 4

HOT SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=1.5 BAR)

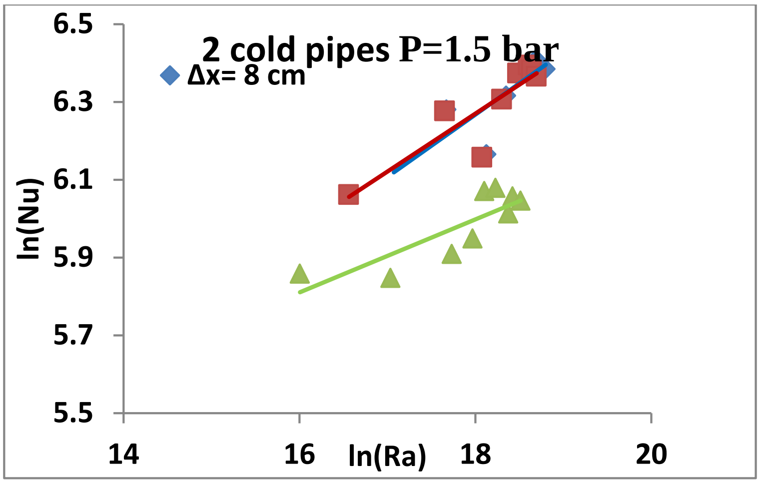

FIGURE 5

COLD SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=1.5 BAR)

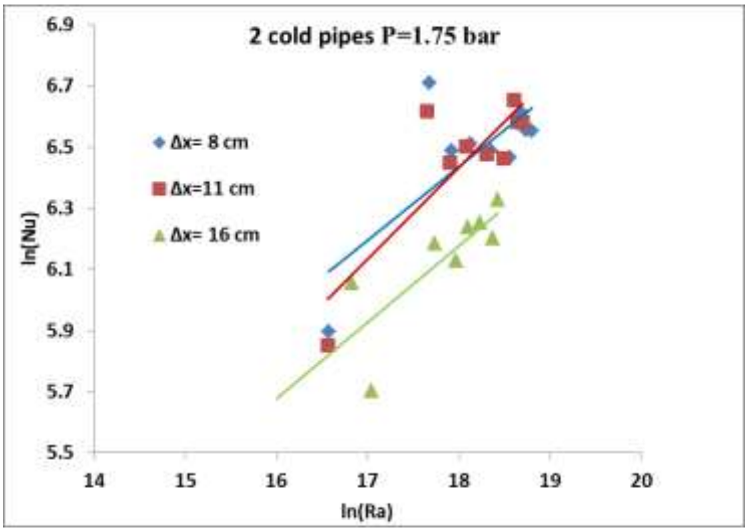

FIGURE 6

COLD SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=1.75 BAR)

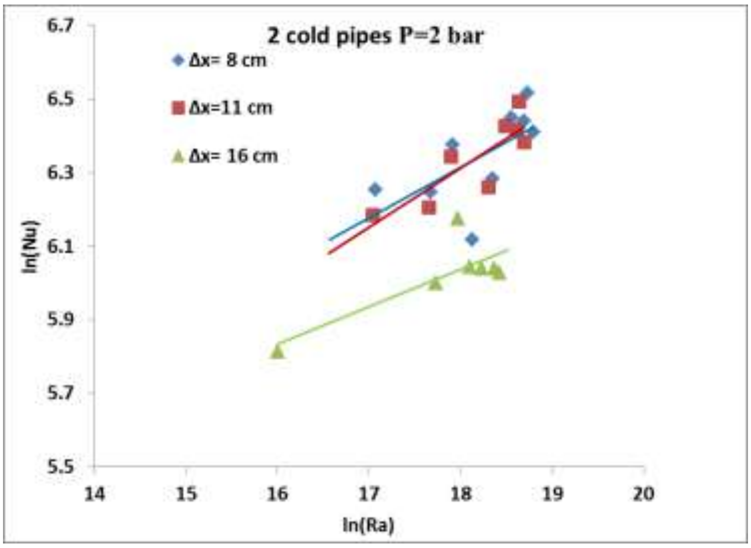

FIGURE 7

COLD SIDE (NU) VS. (RA) FOR VARIOUS ΔX, 2 HOT PIPES CONFIGURATION IN VERTICAL (P=2 BAR)

3. The Effect of Pipes Configuration on Nusselt Number

Figure 8 shows the Nu vs. Ra plots for various heated pipe operation configuration where Δx=8 cm and p=1.5 bar in hot pipe. The one heated pipe is not varied significantly with two heated pipe within the present system.

Figure 9 shows the Nu vs. Ra plots for various heated pipe operation configuration where Δx=8 cm and p=1.5 bar in cold pipe. The one heated pipe , cold Nu is less than the cold Nu in two heated pipe operation because the cold heat transfer rate is less than hot heat transfer by three because the heat that comes from the heated pipe is dispersed to the three cold pipes effectively. The Nu number of cold pipes is slightly less than hot pipe Nu.

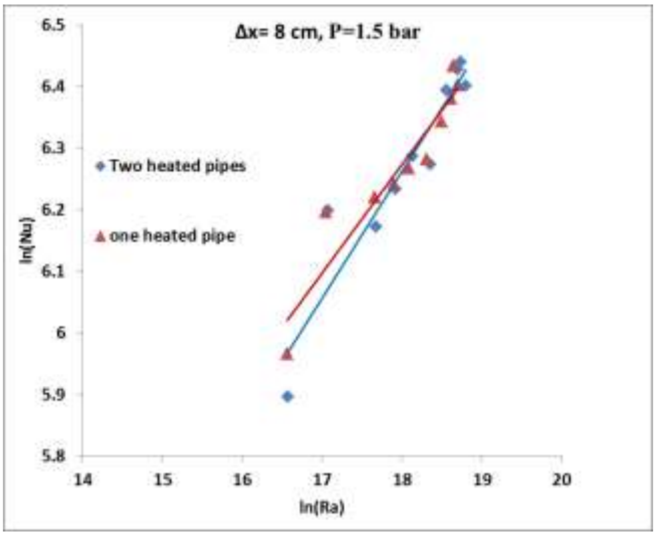

FIGURE 8

HOT SIDE (NU) VS. (RA) FOR VARIOUS HEATED PIPES OPERATIONS

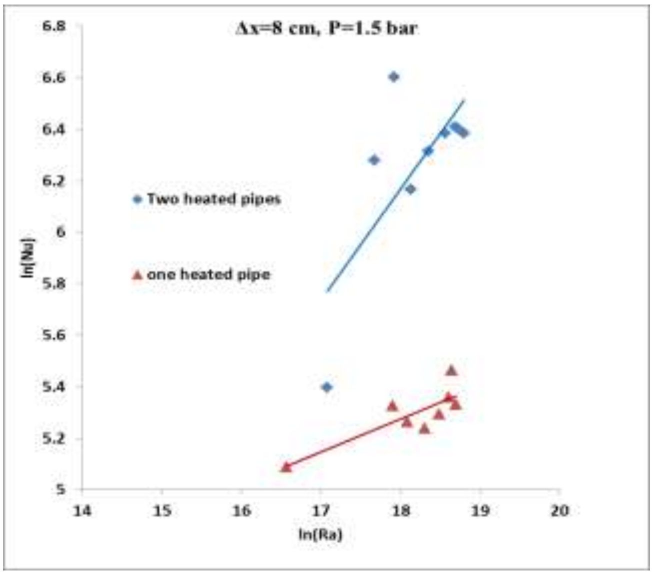

FIGURE 9

COLD SIDE (NU) VS. (RA) FOR VARIOUS HEATED PIPES OPERATIONS

4. The Effect of Pipes Orientations on Nusselt Number

Figures 10 and 11 show the Nu vs. Ra plots where the horizontal orientation of two pipes configuration are applied and various Δx for hot and cold pipes respectively. The orientation changing prevails the changing in characteristics length, the vertical pipes uses the pipe length as characteristics length while the horizontal case, the pipe side length is used as characteristics length. The comparison shows that the Ra of the both systems can be integrated easily because varying in values and ranges.

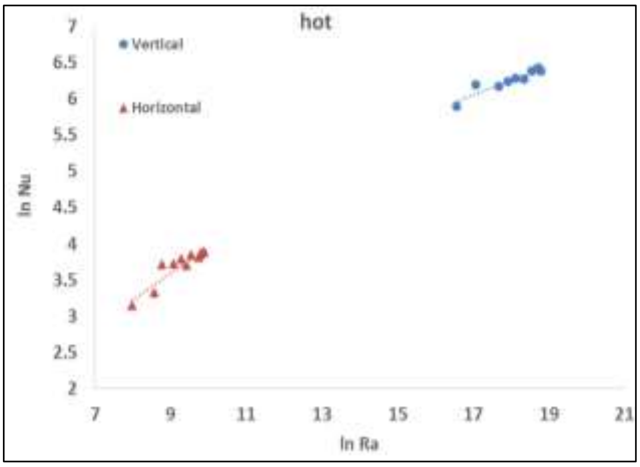

FIGURE 10

HOT SIDE (NU) VS. (RA) OF 2 HOT PIPE CONFIGURATION IN VARIOUS ORIENTATIONS (ΔX=8 CM)

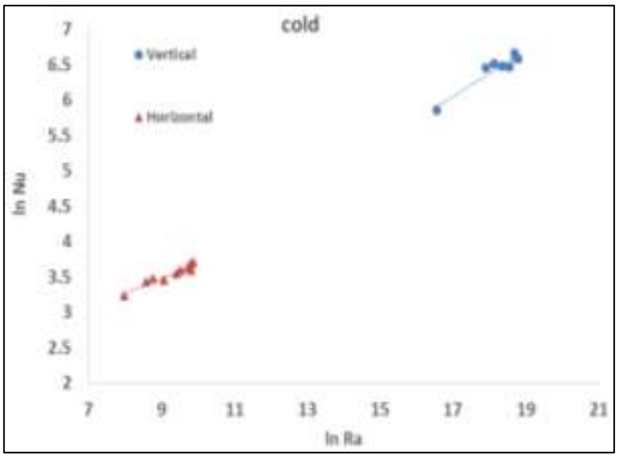

FIGURE 11

COLD SIDE (NU) VS. (RA) OF 2 HOT PIPE CONFIGURATION IN VARIOUS ORIENTATIONS (ΔX=8 CM)

5. Literature Validation in Free Convection

Figures 12 -19 show the Nu vs. Ra plots for various P, orientations, operation configuration and 110 mm distance as comparing with Ali [2017] correlation. Ali [2017] is applicable for hot side only because the modified Ra should be applied with in the present system. The good agreement with the proposed correlation is observed for whole operation parameters. The maximum error % is observed in one top pipe configuration in horizontal by 19.1 %.

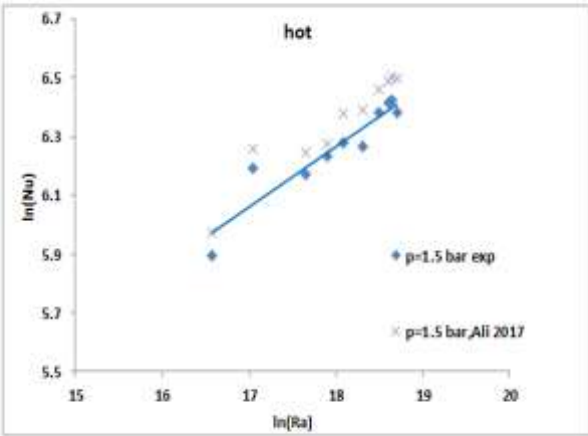

FIGURE 12

THE LITERATURE VALIDATION IN TWO PIPES CONFIGURATION IN VERTICAL ORIENTATION

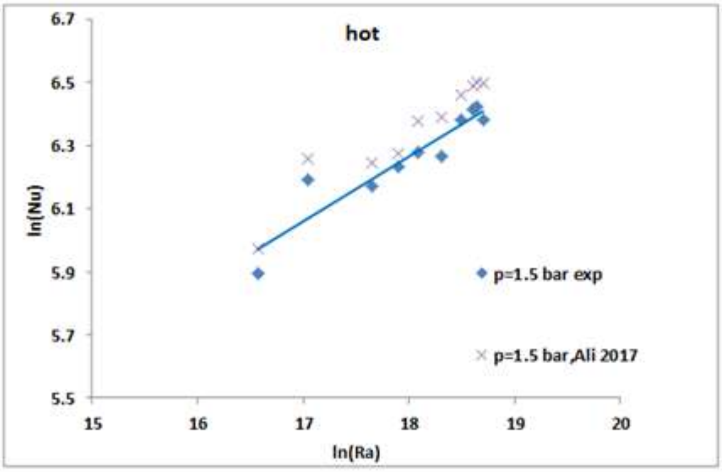

FIGURE 13

THE LITERATURE VALIDATION IN TWO PIPES CONFIGURATION IN HORIZONTAL ORIENTATION

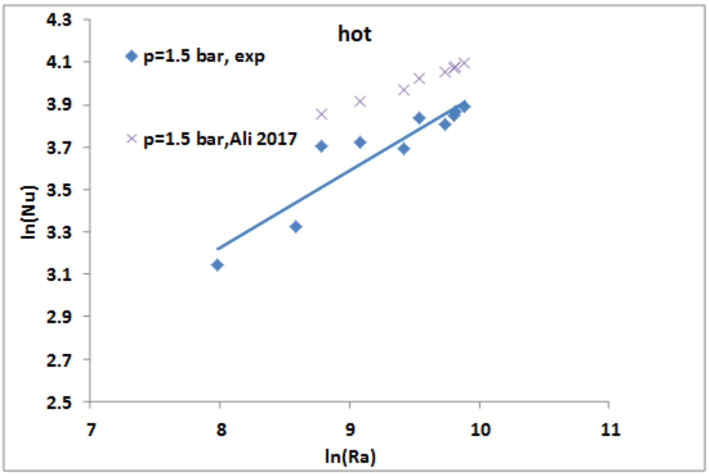

FIGURE 14

THE LITERATURE VALIDATION IN ONE PIPE CONFIGURATION IN VERTICAL ORIENTATION

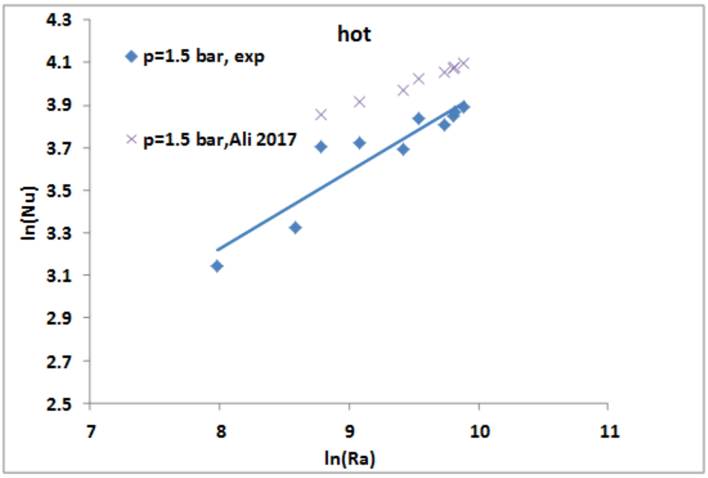

FIGURE 15

THE LITERATURE VALIDATION IN ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, TOP

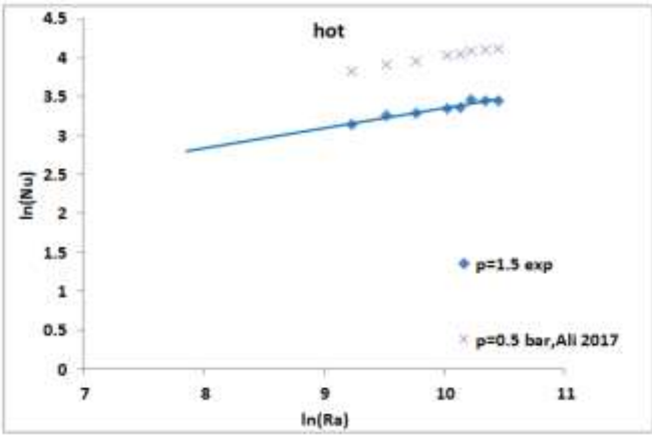

FIGURE 16

THE LITERATURE VALIDATION IN ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, BOTTOM

6. *The Numerical Work Discussion in Free Convection*

Figures 17 and 18 show the Nu vs. Ra plots for various P, orientations, operation configuration and 110 mm distance as comparing with CFD results. The CFD numerical solution is applicable for hot and cold pipes. The maximum error % is found in one pipe configuration in vertical and horizontal top by 18.5 % approximately. The two pipes configuration (horizontal and vertical) and one pipe configuration from bottom has high validation intensity.

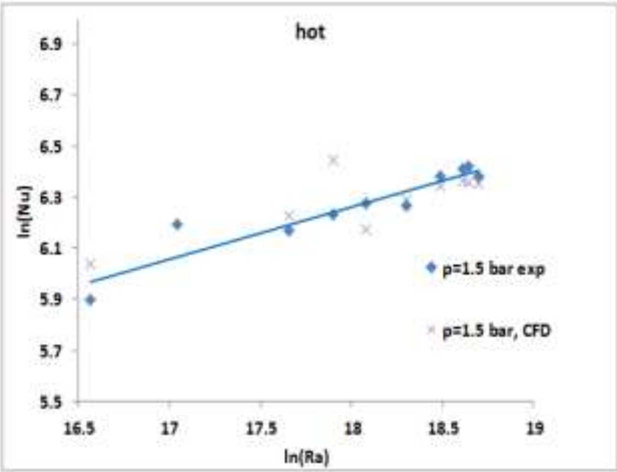

FIGURE 17

THE CFD VALIDATION IN TWO PIPES CONFIGURATION IN VERTICAL ORIENTATION, HOT SIDE

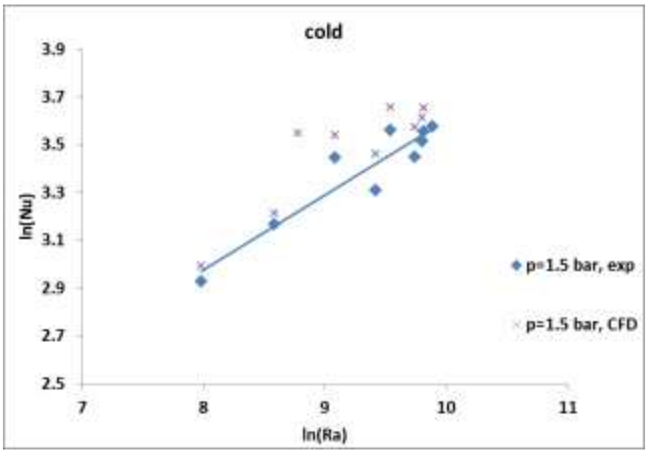

FIGURE 18

THE CFD VALIDATION IN TWO PIPES CONFIGURATION IN VERTICAL ORIENTATION, COLD SIDE

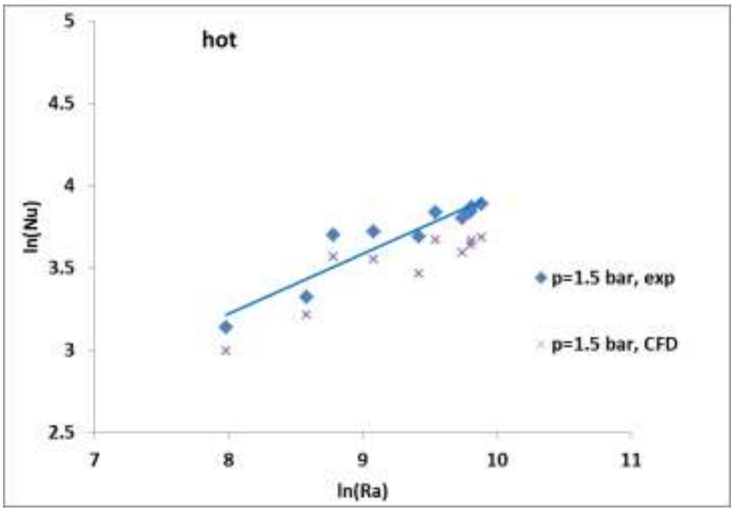

FIGURE 19

THE CFD VALIDATION IN TWO PIPES CONFIGURATION IN HORIZONTAL ORIENTATION, HOT SIDE

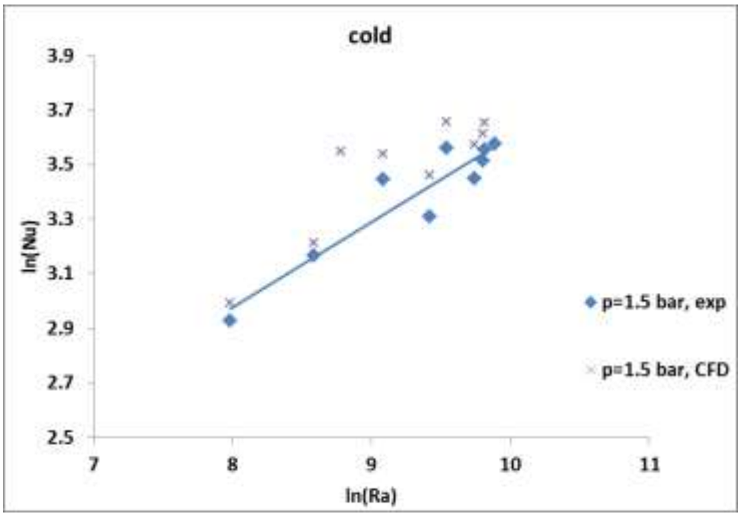

FIGURE 20

THE CFD VALIDATION IN TWO PIPES CONFIGURATION IN HORIZONTAL ORIENTATION, COLD SIDE

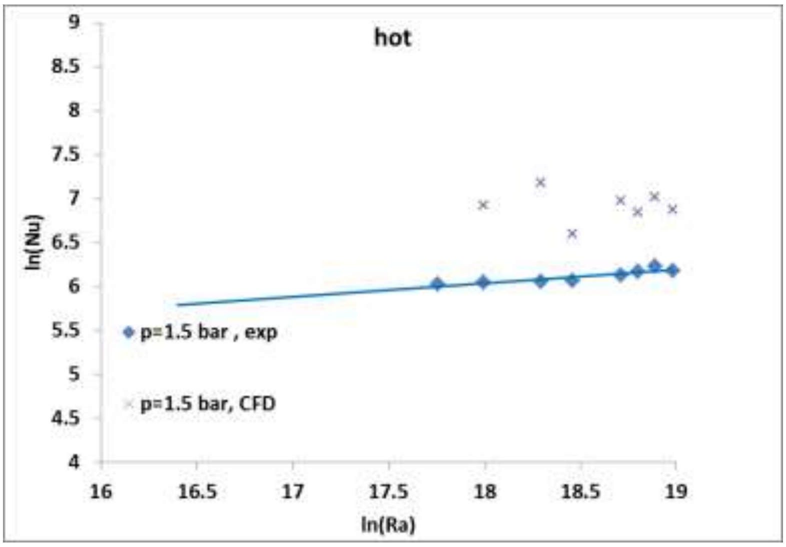

FIGURE 21

THE CFD VALIDATION IN ONE PIPE CONFIGURATION IN VERTICAL ORIENTATION, HOT SIDE

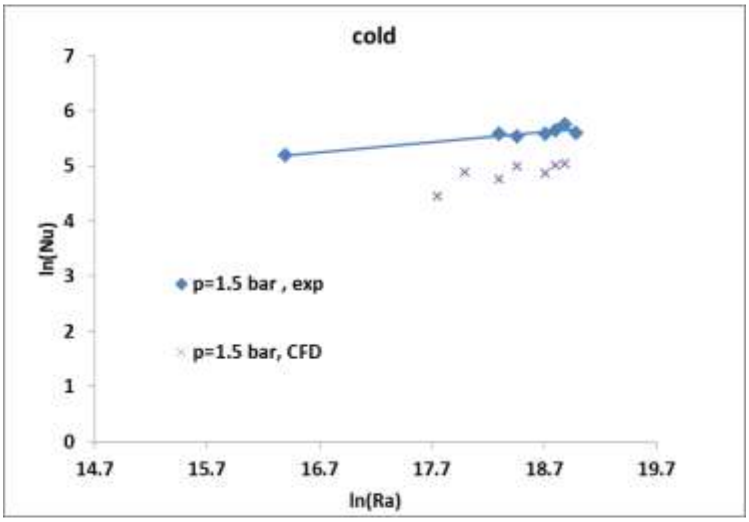

FIGURE 22

THE CFD VALIDATION IN ONE PIPE CONFIGURATION IN VERTICAL ORIENTATION, COLD SIDE

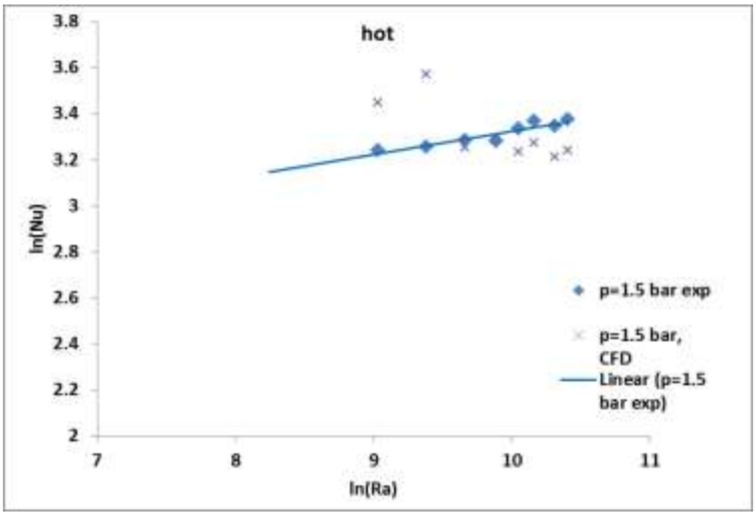

FIGURE 23

THE CFD VALIDATION IN TOP ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, HOT SIDE

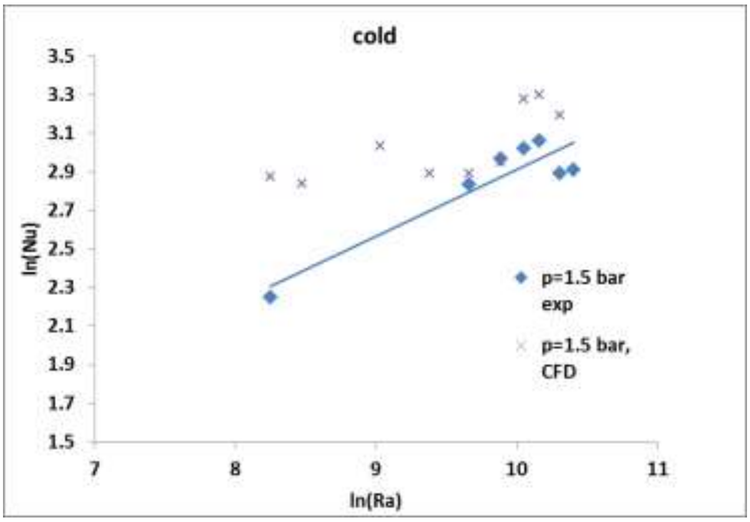

FIGURE 24

THE CFD VALIDATION IN TOP ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, COLD SIDE

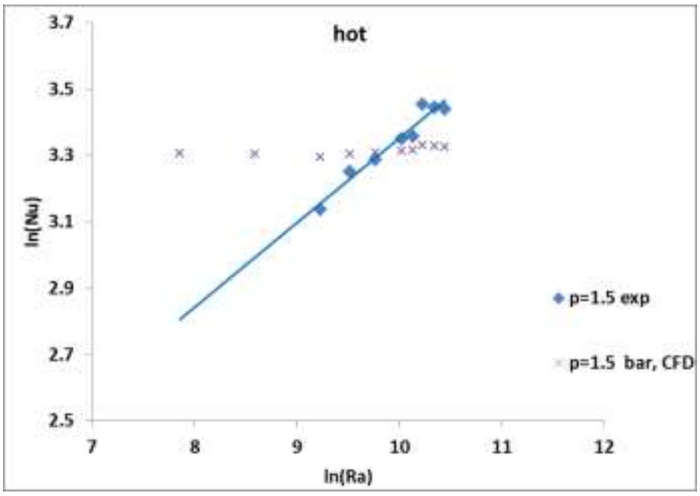

FIGURE 25

THE CFD VALIDATION IN BOTTOM ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, HOT SIDE

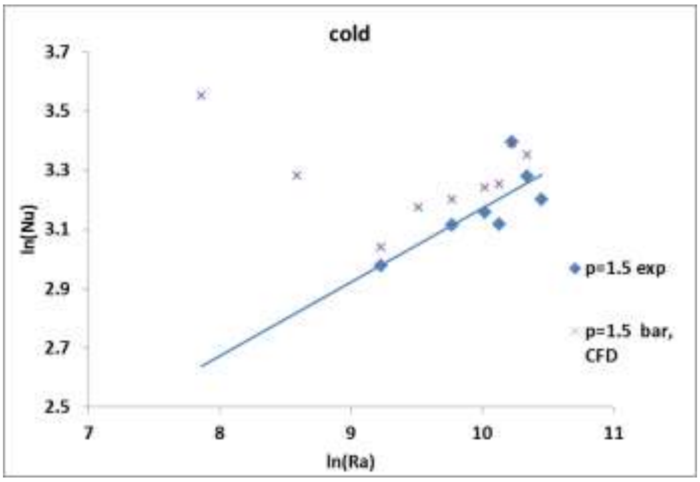

FIGURE 26

THE CFD VALIDATION IN BOTTOM ONE PIPE CONFIGURATION IN HORIZONTAL ORIENTATION, COLD SIDE

Figures 27-30 show the CFD temperature continuous for various orientations and operation configuration. The hot fluid distribution tends to get upward in opposite to cod fluid. The vertical temperature distribution forms layers for various heights can be observed clearly from the top view. The horizontal orientation promotes distortion in temperature contour due to the cold and hot fluids interaction making destruction in cold and boundary layers which is resulted due to boundary layers interaction.

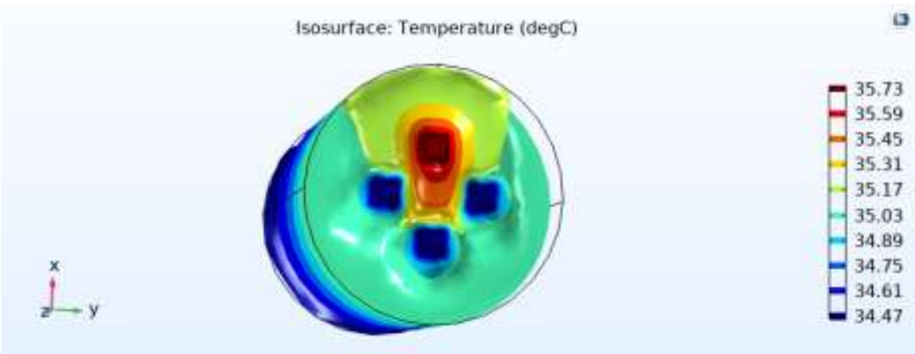

FIGURE 27

THE TEMPERATURE CONTOUR OF VERTICAL ONE PIPE CONFIGURATION

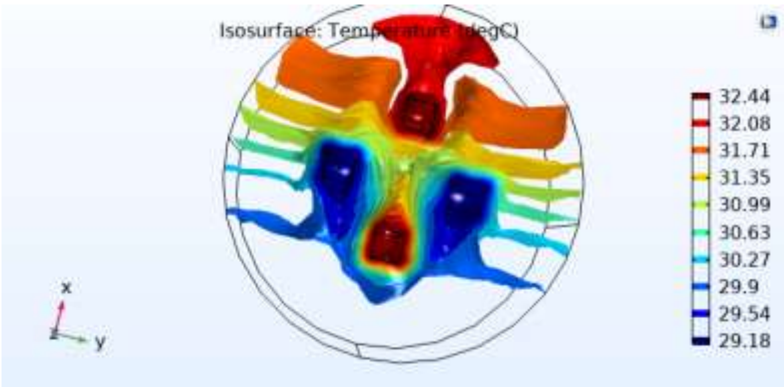

FIGURE 28

THE ISOTHERMAL CONTOUR OF HORIZONTAL TWO PIPES CONFIGURATION

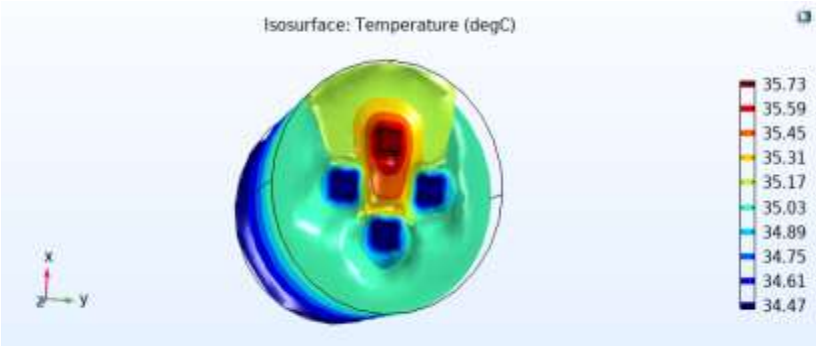

FIGURE 29

THE TEMPERATURE CONTOUR OF HORIZONTAL ONE TOP PIPE CONFIGURATION

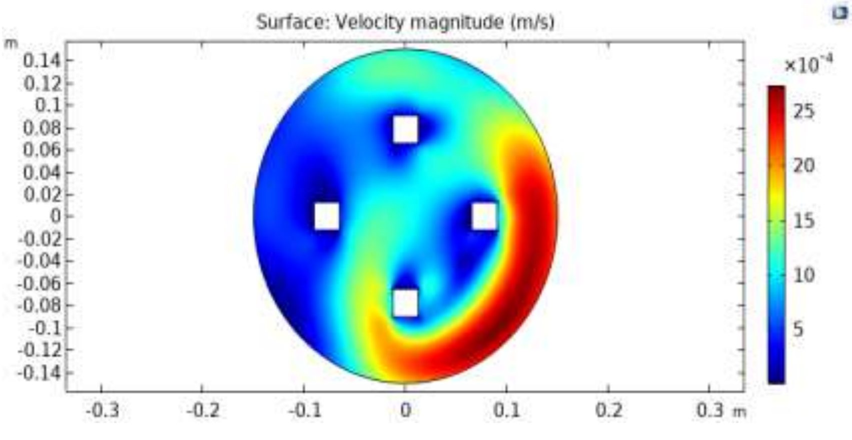

FIGURE 30

THE ISOTHERMAL CONTOUR OF HORIZONTAL ONE BOTTOM PIPE CONFIGURATION

CONCLUSION

The experimental investigation of pressurized tank free convection for various operation conditions has been established successfully. The results indicate the effect of pressure, enclosure orientation and pipes configuration on cold and hot Nusselt numbers. The increasing of cold Nu is meaning the decreasing in hot Nu generally. The pressure has direct effect on Nusselt number. The horizontal orientation has 5 % heat transfer enhancement from the vertical case for two hot pipes configuration. The heat transfer improvement of 9.5 % is also observed when the one hot pipe configuration is used. In vertical orientation. The optimum operating condition is observed when the top one hot horizontal pipe configuration is used, the enhancement is 24 % approximately. The present work has good validation with empirical correlation of previous investigation and CFD simulation. The maximum heat transfer error validation with numerical simulation is about 18.5 % and with previous work investigation is approximated to 19.01 %.

REFERENCES

[1] M. Sheikholeslami, M. Hatami, M. Jafaryar, F. Farkhadnia, D.D. Ganji, and M. Gorji-Bandpy, "Thermal management of double-pipe air to water heat exchanger," *Energy Build.*, vol. 88, pp. 361–366, 2015, doi: 10.1016/j.enbuild.2014.11.076.

[2] G. Barakos, E. Mitsoulis, and D. Assimacopoulos, "Natural convection flow in a square cavity revisited: Laminar and turbulent models with wall functions," *Int. J. Numer. Methods Fluids*, vol. 18, no. 7, pp. 695–719, 1994, doi: 10.1002/fld.1650180705.

[3] H. Yao, *"Studies of Natural Convection in Enclosures Using the Finite Volume Method,"* vol. M.

[4] R.S. Brodkey and H.C. Hershey, Transport Phenomena: A Unified Approach. Singapore: McGraw-Hill Series, 1998.

[5] A.J. Chamkha and M.A. Ismael, "International Journal of Thermal Sciences Conjugate heat transfer in a porous cavity fi lled with nano fl uids and heated by a triangular thick wall," *Int. J. Therm. Sci.*, vol. 67, pp. 135–151, 2013, doi: 10.1016/j.ijthermalsci.2012.12.002.

[6] L. Theodore, *Heat Transfer Applications for the Practicing Engineer.* New Jersey: John Wiley & Sons, Inc, 2011.

[7] K. Kitamura, Kami-iwa, F., and T. Misumi, "Heat transfer and fluid flow of natural convection around large horizontal cylinders," *Int. J. Heat Mass Transf.*, vol. 42, pp. 4093–4106, 1999.

[8] S.J. Habeeb, "Investigation of heat transfer phenomena and natural flow behavior around a heated square cylinder placed in a cooled elliptical enclosure," *Mathematics*, pp. 1–18, 2010.

[9] L.A. Salman and A.J.J. Shamikh, "Experimental and theoritical study of natural convection heat transfer of heated square cylinder placed inside a cooled circular enclosure filled with air," *Iraqi J. Mech. Mater. Eng.*, vol. 12, no. 3, pp. 564–581, 2012.

[10] A.H. Altaee, Farooq Hassan Ali, and Q.A. Mahdi, "Natural Convection Inside Square Enclosure Containing Equilateral Triangle with Different Orientations," J. Univ. Babylon, vol. 25, no. 4, pp. 1194–1205, 2017.

[11] S. Pandey, Y.G. Park, and M.Y. Ha, "An exhaustive review of studies on natural convection in enclosures with and without internal bodies of various shapes," Int. J. Heat Mass Transf., vol. 138, pp. 762–795, 2019, doi: 10.1016/j.ijheatmasstransfer.2019.04.097.

[12] Ali, M., & Sadek, S. (2018). Free convection heat transfer from different objects. *Heat Transfer-Models, Methods and Applications, 1,* 1-20. http://doi.org/10.5772/intechopen.75427