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

International Journal of Mechanical Engineering

The Numerical Investigation of Natural Convection of Finned Heated Pipes Inside Cylindrical Enclosure using COMSOL Multiphysics

Rassol H. Rasheed [1*], Majid H. Majeed [2], Ahmed Q. Mohammed [3]

The Technical Engineering College, The Midland Technical University/ Baghdad- Iraq.

Abstract - The natural convection inside the cylindrical enclosure within two heated finned pipes has been studied numerically. The enclosure of four symmetrical pipes where the finned heated pipes are used in various geometries are designed and modeled. The momentum and heat equations are combined by using COMSOL Multiphysics finite element method package commercial software. The various parameters are introduced as boundary conditions and geometrical configurations. The impact of fins number, fins radius ratios (R) , aspect ratio (A.S) and fins geometry on heat transfer performance (Nusselt Number) is included in present investigation. The fins number of 12-16, radius ration (R)=2 to 3 and A.S=1.83 to 2.7 are used. The results show that The heat transfer enhancement rate for 14 fins is 22 percent, and for 16 fins it is 33 percent. It is projected that the heat transfer enhancement rate for 18 fins will be half of 16. The heat transfer increase is around 2.16 percent for R=2.5 and R=3 in the hot area and 7.38 percent for R=2.5 and 3 in the cold region. Where R is more than 2.5, the heat transfer enhancement is not possible to increase. Increased heat transfer surface area and the creation of deep channels are the results of raising the radius ratio but the too deep channels form dead zones makes a heat transfer enhancement barrier. By 0.96 percent, the highest Nu is attained when log (Ra)= 7.46 and As=2.7. By reducing the distance between the pipes, the heat transfer dissipation across the fluid is increased. Between the close-by pipes, a greater volume of hot fluid will be created. The circular fins have heat transfer performance higher than the rectangular fins by 4.55 % where the turbulence intensity in circular fins higher than the longitudal fins. The temperature and velocity contours are obtained for various parameters.

Index Terms - Numerical, Natural convection, finned pipes, COMSOL Multiphysics, Enclosure.

INTRODUCTION

Depending on the flow arrangement and geometric parameters, natural convection heat transfer can be classified as exterior or interior. The fluid is bounded or contained by solid substrates all across the internal flow configuration, whereas the fluid is bounded or enclosed by a solid object during the exterior flow configuration. Internal fluxes include the flow through tubes and ducts, which are the most common instances. External flows include fluid flowing over a flat plate, cylinders, and spheres, to name a few. Free convective flows, on the other hand, are a complicated process due to the necessity of linking the hydrodynamic and thermal transport fields. The internal flow configuration is more complicated than the outward flow configuration in terms of precision [1]. External flow issues are simulated using the classic boundary layer assumption, which assumes that the homogeneous boundary layer's outer area remains unaltered [2] [3]. The linkages between the boundary layer and the center, on the other hand, are a primary source of complication in the natural convection issue for the internal convection design [4] [5].

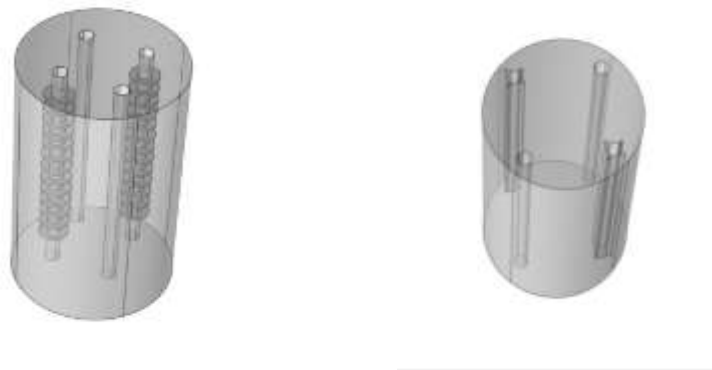
The numerical investigation in free convection systems has great attention in last few years. The finite element method is used to reduced the investigations cost with high development, analysis and modeling impact. One of the basic mechanisms of heat transport is natural convection. To correspond to the equations of mass, momentum, and energy conservation of the flow field in the enclosed space, the pressure-velocity approximations of the Navier–Stokes momentum equations and energy equation are utilized. For oversell the field parameters, a nonlinear joined solution technique with various-noded triangular component finite difference system is used to solve the finite element models of the dimensionless interfacing equations with connected boundary conditions[6]. The significances of heat generations, aspect ratios, various Prandtl numbers, and locations of the thermally active section of the side walls on the flow pattern, streamlines, heat distribution, isotherms, mid-height velocity, and rate of heat transfer from the walls of the enclosure are linked in various investigations[7]. It's a function of the Rayleigh number, the heated element size, and it increases as the Rayleigh number, the number of heaters, and the heater size increases. In general, the Nusselt number and volume exchange rate increases as the Rayleigh number, heated element size increases. The impact of these variables on the Nusselt number and volume exchange rate are both significant [8]**.**

Various instigators dealt with numerical investigations where the impact of Ra, fins presence and heated element geometry is investigated such as: **Ouzzane and Galanis, 2002**, The combined non-linear partial fifferential equation improving laminar – mixed convective heat transfer in a parallel pipe and conduction in fins have been investigated computationally. The consequences showed that the effective heat and hydrodynamic contour were quite various and the thermal loss rate from the top of fins was more significant then that from the fins bottom for various operational heat and temperature conditions. **Aydin** and Using an appropriate boundary conditions**, Pop, 2005 [9]** analyzed numerically the steady two - dimensional natural convective flow and heat exchange of micropolar fluids in compartments with a central location separate heater in one of its side panels. Each other sidewall was kept at a constant temperature, and the horizontal walls were presumed to be insulated. Using a finite element model**, Saha et al., 2007 [6]** numerically investigated free convection inside a two-dimensional rectangular enclosure. The leading to rise was protected, two vertical walls were kept at a continual low temp, the bottom wall was maintained at constant high temp, and the non-heated segments of the bottom wall were regarded adiabatic in this study. The goal of this project was to demonstrate the computational attitude's ability to handle such configurations. The Grashof number range was 103 to 106 and Prandtl number was taken as 0.71, in which the air was used. This investigations had indicated the significance of various aspect ratios, range of 0.5 to 1, and inclination degree of the enclosure from 0º to 30º on the thermo-fluid individuality. Results were existing in the form of streamline and isotherm. Unsteady boundary - layer free convective heat in a casing with partly heat active side panels and heat generation / absorption were studied numerically by **Kandaswamy et al., 2008 [7].** Segments of the side panels and longitudinal walls that were not heat active were insulated. The finite volume technique with a power law scheme was used to solve the combining equations. Heat generation, aspect ratios, numerous Prandtl numbers, and the placement of the thermodynamically active segment of the side panels on the fluid flow, streamlines, produce heat, isotherm models, semi speed, and heat transfer rate from of the enclosed walls were also discussed. Resulting in an increase in flowing fluid, the thermal efficiency increased with the increase the Grashof amount and decreased with the increasing heat production. Whenever the cold and hot thermally energetic places are located in the center of the side panels, the heat exchange is greatest. Numerically boundary - layer natural ventilation in inclined rectangle abscesses with a regionalized heat source was explored by **Elsherbiny and Ragab in 2013** [10]. The preservation equations governing mass, motion, and heat energy, as well as their boundary conditions, were solved using a mathematical model. Graphs of streamline as well as isotherm contour plots were used to summarize the results.. With a mean difference of less than 11.5 percent, a correlation has been developed to represent the current numerical heat transfer results. **Gavara and Soni [11]** used numerical modeling to investigate stable laminar convective heat transfer inside a rectangular cavity exterior with identical heating systems on the bottom wall. Three cooling configurations were investigated using convective heat transfer: (1) cooling just at left vertical surface, (2) cooling mostly at the left and right vertical surfaces, and (3) cooling at the upper side wall. The cold cavity walls were kept adiabatic while another cavity walls have been cooled constant temperature. Despite the dimensional and thermal symmetrical of the combinations, both the wall cooling and top wall cooling performance results showed inverted flow and thermal trends for elevated Rayleigh numbers. The cooling performance of left wall cooling was found to be highly sensitive to cavity inclination. Correlations for dimensionless maximum temperature were presented. **Mokhtari et al., 2017** [12], Both in flow regimes, the combined convection of a 3 -D cubic duct containing different layouts of fins was statistically defined and analyzed. The objective of this research was to see if fins configurations can improve heat exchange when the stream was highly viscous as well as the fluid was air. Its bottom conduit barrier has an uniform thermal state in their systems, whereas the two walls and upper wall had insulation. When the basic fin arrangement is replaced with inclined fins, heat transfer significantly improves (about 40–50% in the laminar flowing fluid and 15–20% in turbulent flow compared to base plates heat exchange). Furthermore, the impact of limiting heat input on temperature profile is thoroughly examined. **El-Shorbagy et al., 202**1 [13], On a suitable heat sinks, the researchers numerically modelled the convective heat transfer of Al2O3/CuO–H2O hybrid nanofluids (HNFs) (HS). An isothermal HS carries the (HNFs) flow. On the upper wall, the NF stream was handled as a slip flow. The NF flow is influenced by a homogeneous magnetic field. To solve the algebraic problems, the control volume approach was combined with the SIMPLE algorithm. Fin depth was varied between 0.1 and 0.3 for this purpose, with four distinct layouts for fins of various thicknesses studied. The results showed that dispersing a 3% volume of nanocomposites (NPs) in water at a 1:1 ratio increased heat transfer by 2.78 percent and entropy generation (Sgen) by 6.59 percent, correspondingly. Dispersing CuO NPs were approved as having a greater effect on the heat transfer improvement than Al2O3 NPs. At varying Richardson numbers (Ri), a change in together for or pattern resulted in differing behaviour. **Alshara et al., 2016** [14], In 3- D laminar mixed convective heat transfer coefficient from extended surfaces in a rectangular duct with slope, an experiential evaluation was carried out. This channel's lowest material was exposed to a homogenous heat flow, while another sides were isolated. In a various orientation rectangular duct, a series of empirical experiments for air flow with mixed convective heat transfer coefficient with longitudinal fins was performed. Sloped orientations with perpendicular in the ranges of 0- 75°. At various Reynolds numbers (1000- 2300) and the ranges of Grashof number (3x108-1x109), the study was developed to assess the influence of slope on local reference temperature, heat transfer rate, and mean heat transfer rate. The outcomes of the empirical analysis suggest that optimal angle for maximal heat transmission is 45 degrees. The direction of the fins arrangement is also critical for improving heat exchange. **Rout et al. , 2012** [15], For a stable and laminar flowing fluid within a pipe under made by mixing flow regime, the temperature profile of a from within finned pipe has already been computationally calculated for different fin count, tallness, and structure by attempting to solve continuity equation of mass, dynamism, and energy utilizing Fluent 12.1 for a stable and laminar flowing fluid within a pipe. It was discovered that amount of fins necessary to keep the tube wall temp to a minimal had such an optimal value. Fin height tallness had an upper limit above which the temperature profile is unaffected by fin height. During mixed fluid flow, the top surface of a straight pipe had a greater mean temperature than the bottom part. The effect of fin form on heat transfer rate demonstrates that pyramidal fins have the lowest thermal resistance relative to rectangular- and T-shaped fins. Aside from the thermal properties, the pressure drop produced by the existence of fins had also been studied. **Kopec´ and Z˙ elasko , 2021** [16], The article's optimization estimates were relevant to longitudinally finned pipes of a hvac system evaporation working in natural steam outside air circulation circumstances. An unique, undulating fin form distinguishes the finned surfaces. The paper describes the methods used to find the best geometric parameters for a finned tube, in which heat simulations were done using the numerical method that represent a convective heat transfer flow on the finned surfaces. The optimal solution was driven by the lowest mass of the fin, and hence design variables corresponded to the number of fins (n = 6), fin heights (h = 0.065), and fin depth (s = 0.0015 m) in the situation of maximizing heat input with least mass of a fin. The tubes with 10 fins with a length of h = 0.11 m and a depth of s = 0.0018 m permitted optimal heat input at the estimated mass of the fins inside the exchangers tubes design, according to optimization estimates for maximum efficiency of the exchangers at constant load. For any mass and optimum thermal performance, the article presents a simpler technique of estimating the ideal geometrical features of the contour. **Dogan and Sivrioglu, 2010** [17], For such a wide variety of altered Rayleigh numbers and varying fin height and distances, mixed convection transport via longitudinal fins on the inside of a horizontal tube has already been explored. The effect of spacing, fin height, and heat flow intensity on mixed convective heat transfer coefficient using rectangular fin arrays warmed from underneath in a horizontal duct were investigated in an empirical parameterized research. The best fin spacing for optimal heat transmission has been looked at. The constant heat parameter was achieved during the trials, and gas was employed as the coolants. When using gas velocity solenoid valve, the speed of fluid exiting the duct was maintained virtually stable (0.15 - 0.16 m/s) such that the Reynolds value was constantly around 1500. Tests with altered Rayleigh numbers 3x 107 -8x 108 with Richardson number 0.4 -5 were carried out. Fin height was changed from Hf/H = 0.25 to Hf/H = 0.80, and non - dimensional fin separation was adjusted from S/H = 0.04 to S/H = 0.018. The findings of an experimental investigation on mixed convective heat transfer coefficient demonstrate that ideal fin separation for maximal heat transfer is S = 8–9 mm, and the optimal fin separation is dependent on the amount of Ra*. **Khan and Mishra, 2016** [18], This research focused on the computational model of a tube having inner fins that is exposed to mixed convection and undergoes natural and forced rate of heat transfer. The study was carried out using the FEV tool ANSYS FLUENT, with the SIMPLE method being utilized to solve the tube's equilibrium equations. The effect of several factors such as fin height, fin shape (rectangular, T-shaped triangular), heat flux, Reynolds number, and tube inclination (horizontal and vertical) on tube temperature rise has been investigated. It has been discovered that heat transmission is considerable for triangular fins, and that heat transfer is greater in perpendicular direction due to buoyancy and boundary layer heat exchange than in parallel configuration. The collected data were compared to existing publications and found to be acceptable accord.

The purpose of present investigation is to develop numerical simulation of two heated finned pipes inside the pressurized cylindrical enclosure natural convection. To fined the impact of various parameters such as fins number, radius ratio, aspect ratio and fins geometry on heat transfer coefficient based on finite element method solution. To develop thermal and momentum contours for present investigation.

NUMERICAL MODELING

The cylindrical enclosure is used to place four pipes with equal distances from the center. The two of these pipes are subjected to uniform heat power while the others are subjected to ambient temperature from the interior surface. Many parameters such as Rayleigh number is used for various distances, orientations, fins numbers, fins size, fins type in free convection additional to Ri in mixed convection case. The geometries are presented in Figure 1.



a. circular finned test section   b. rectangular finned test section

Figure 1: Geometrical configuration.

The numerical investigation should be generated based on high quality assumptions, hypothesis and theories. The assumptions are placed by modifying the geometry coordinate, physics conservation equations, boundary conditions and study pattern. For the present investigations, the assumptions are summarized as following:

- Steady state models.
- The geometry coordinate is three dimensions with constant surface area and symmetric with finite slides with z direction.
- The boundary walls are non- slip conditions.
- The fluid flow is laminar.
- Constant physical properties except the density variation with temperature (Bossinesseq approximation).
- The radiation effect and the energy dissipations had been neglected.

The main applied physics in present work are momentum transport (turbulent k-ε) and heat transfer through fluid and solid. The complex partial differential equations are converted in to algebraic equations based on force balance and heat balance with control system by mean of mesh distribution. The Multiphysics coupling is enabled by interaction between velocity and temperature distribution, the velocity components is used for convection purpose in heat transfer equation while the temperature can manipulate the physical properties which are polynomial expressions, following conservation equations are used for momentum:

- Momentum Equation

$$\rho(U.\nabla)U = \nabla.(-\rho L + K) + F + \rho g \quad (1)$$

- Continuity equation

$$\nabla.(\rho U) = 0 \quad (2)$$

- Kinetics expressions

$$K = (\mu + \mu_t)(\nabla U + (\nabla U)^T) - \frac{2}{3}(\mu + \mu_t)(\nabla U)l - \frac{2}{3}\rho k l \quad (3)$$

$$\rho(U.\nabla)k = \nabla.\left[\left(\mu + \frac{\mu_t}{\delta_k}\right)\nabla k\right] + \rho_k - \rho£ \quad (4)$$

$$\rho(U.\nabla)£ = \nabla.\left[\left(\mu + \frac{\mu_t}{\delta_£}\right)\nabla £\right] + C_{£1}\frac{£}{k}\rho_k - C_{£2}\,\rho\frac{£^2}{k} \quad (5)$$

$$\mu_t = \rho\frac{C_\mu k^2}{£} \quad (6)$$

$$\rho_k = \mu_t\left[\nabla U{:}\left(\nabla U + (\nabla U)^T\right) - \frac{2}{3}(\nabla.U)^2\right] - \frac{2}{3}\rho k \nabla.U \quad (7)$$

where: The k-£ turbulent model constants which are derived from COMSOL software for general fluid , C£1=1.33, C£2=1.92, Cμ=0.09 , δk=1 and δ£= 1.3. For heat conservation, following equation is used:

- Heat equations

$$\rho C p U \nabla T + \nabla.q = Q_{sum} \quad (8)$$

and $q = -\zeta\,\nabla T$ (9)

The main boundary conditions for free convection:

- At the inner pipe: Constant heat flux surface
- At the outer cylinder: Isotherm cold surface temperature
- Wall function for interior and exterior walls

The main boundary conditions for mixed convection:

- Constant heat flux surface
- Isotherm cold surface temperature
- Wall function for interior and exterior walls
- Inlet flow: from top to bottom
- Outlet pressure: outlet pressure
- Inlet Temperature: the temperature of inlet fluid

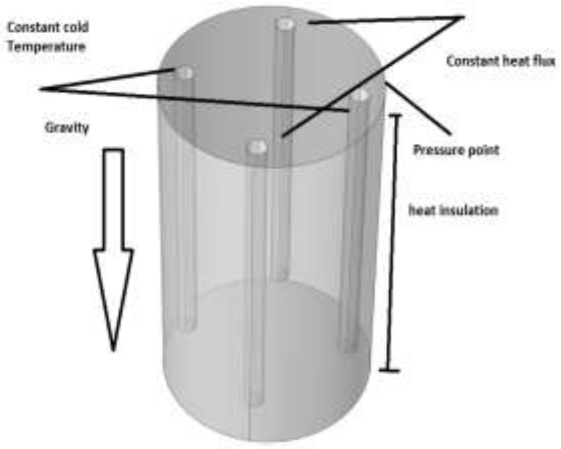
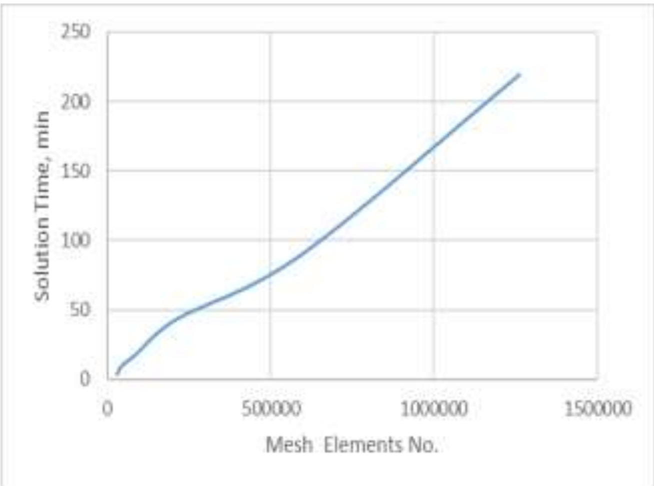


Figure 2: Boundary conditions.

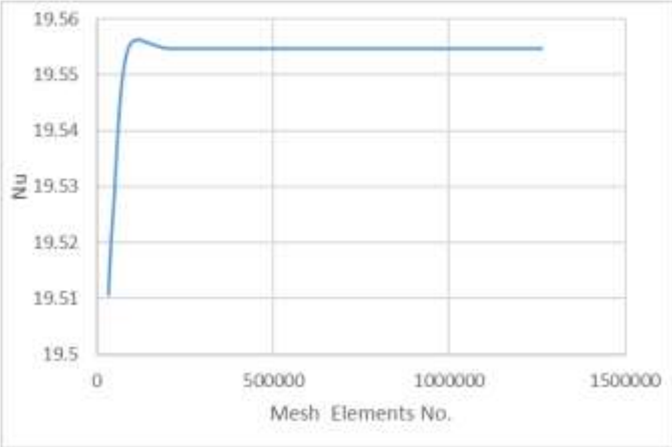
The mesh dependent method is the solution major optimization. The mesh optimization is important feature for Finite Element Method investigation. The physical memory of the personal computer needs longer time to handle the model solution with high mesh elements number, and it's needed to fine the most accurate mesh with shorter solving time. For present system, coarser option gives the optimum solution. The solution time in coarser is lower than coarse, normal …. etc. and close value of them as shown in table 1

Table 1: Mesh dependent analysis.

type of element	number of element	time (min)	Nu
Extremely coarse	31506	4	19.51079841
Extra course	45299	10	19.5254757
Coarser	87707	18	19.55462738
Coarse	209386	43	19.55462738
Normal	595766	90	19.55462738
Fine	1262006	219	19.55462738



A. Solution time.



B. Nusselt Number

Figure 3: Mesh optimization.

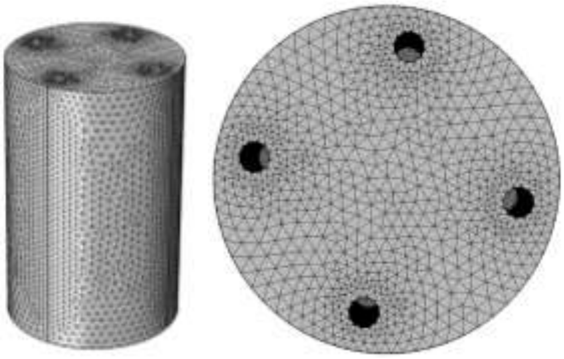


Figure 4: Normal mesh

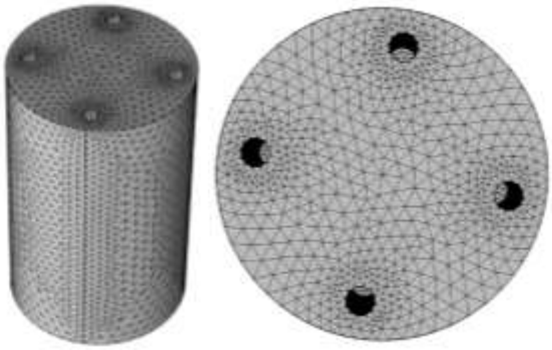


Figure 5: Coarse mesh

Results and Discussion

The natural convection of circular fins

The effect of fins number

Fig. 6 and 7 show the Nu vs. log Ra for various circular fins number for hot and cold pipes where the two heated pipes configuration operation is used. The Nu values rise as the number of fins, while the heat transfer enhancement rate decreases as the number of fins increases. The heat transfer enhancement rate for 14 fins is 22 percent, and for 16 fins it is 33 percent. It is projected that the heat transfer enhancement rate for 18 fins will be half of 16. The increase in heat transport will be 38.5 percent. For the total fins number, the cold pipe has the approximation near values of the hot pipe.

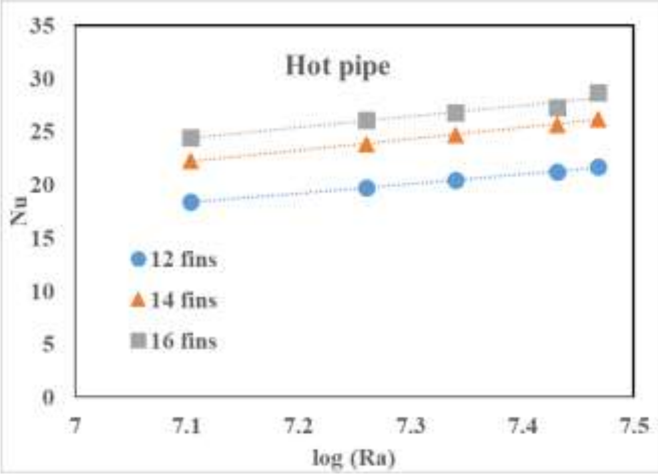


Fig. 6: Nu vs. log (Ra) for various fins number in hot pipe region.

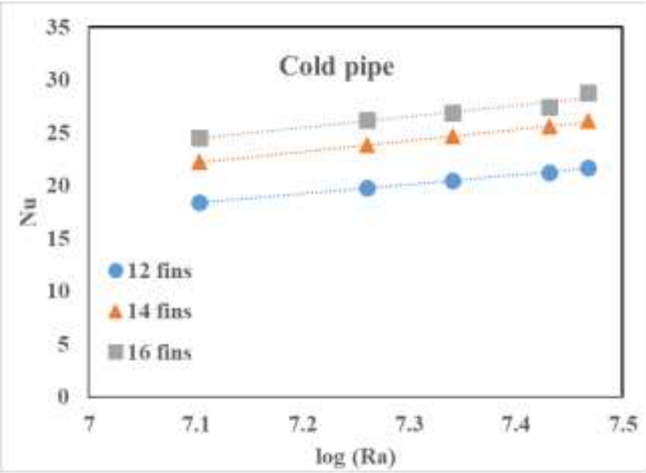


Fig. 7: Nu vs. log (Ra) for various fins number in cold pipe region.

By increasing the number of fins in addition to the mini channels existing in the hot pipe wall, the heat transfer area is enhanced, and the heat transfer resistance is reduced to a minimum. Because of the turbulence intensity generated within the current system, the hot and cold fluid particles would be efficiently combined.

The effect of fins radius ratio (R)

The radius ratio (R) is the ratio of the radius of the fins to the radius of the pipe. Figures 8 and 9 demonstrate Nu vs. log Ra for different circular radius ratios for hot and cold pipes, using two heated pipes and 12 fins. The heat transfer increase is around 2.16 percent for R=2.5 and R=3 in the hot area and 7.38 percent for R=2.5 and 3 in the cold region. Where R is more than 2.5, the heat transfer enhancement is not possible to increase. Increased heat transfer surface area and the creation of deep channels are the results of raising the radius ratio but the too deep channels forms dead zones makes a heat transfer enhancement barrier.

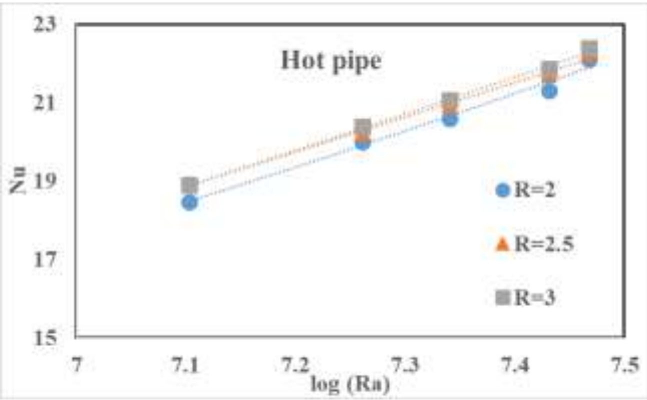


Fig. 8: Nu vs. log (Ra) for various fins R in hot pipe region.

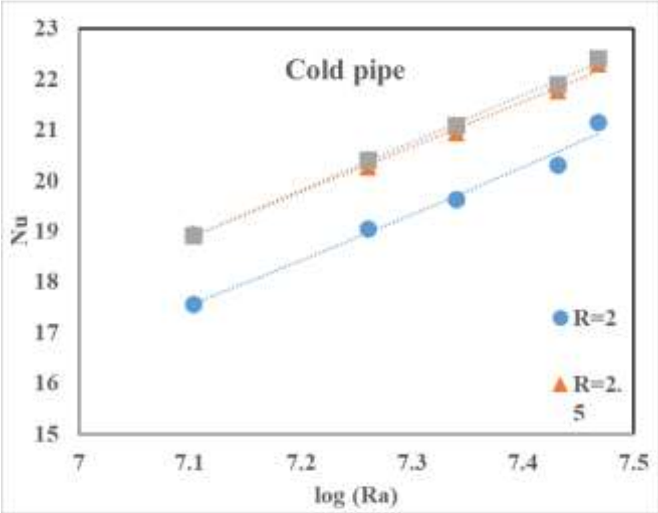


Fig. 5.9: Nu vs. log (Ra) for various fins R in cold pipe region.

The effect of fins aspect ratio (As)

Hot pipe Nu vs. Ra for different Aspect ratios (As) with 12 fins number and R=1.83 is shown in Figure 10. The Nu range values increase with each aspect (AS), with considerable alterations for the whole Ra. By 0.96 percent, the highest Nu is attained when log (Ra)= 7.46 and As=2.7. By reducing the distance between the pipes, the heat transfer dissipation across the fluid is increased. Between the close-by pipes, a greater volume of hot fluid will be created.

Figure 11 displays the cold pipe Nu vs. Ra for different Aspect ratios (As) with 12 fins and R=1.83. This scenario functions similarly to the hot pipe example. Where log (Ra)= 7.43155 and R=3, the maximal heat transfer improvement is roughly 1.315 percent. The greatest heat transfer rate from hot pipes will be transported to bulk fluid, and the same rate will be transferred to cold pipes, resulting in efficient heat transfer mixing with the highest volume mixing rate possible.

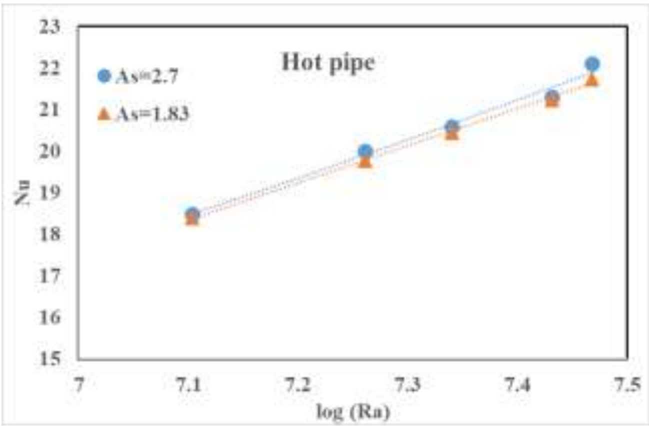


Fig. 10: Nu vs. log (Ra) for various fins As in hot pipe region.

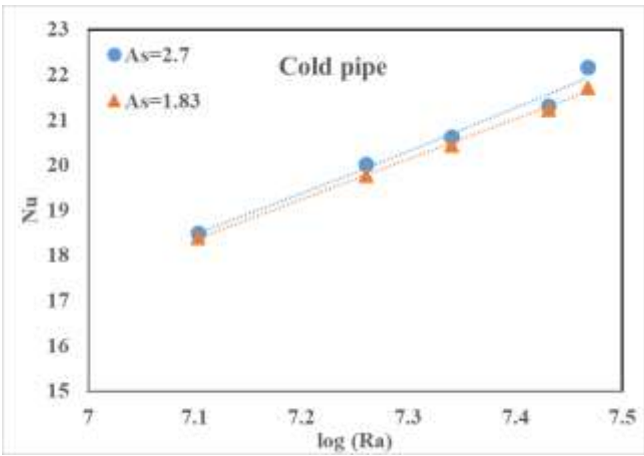


Fig. 11: Nu vs. log (Ra) for various fins As in cold pipe region.

The natural convection of longitudal fins

The rectangular (logitudal) fins is placed in current investigation as equivalent surface area to circular fins for various fins numbers and radius ration (R).

The effect of equivalent circular fins number

Fig. 12 and 13 show the Nu vs. log Ra for various equivalent circular fins number for hot and cold pipes where the two heated pipes configuration operation is used. The Nu values increase as the equivalent number of fins, The heat transfer rate improvement increases by increasing equivalent number higher than 14 fins. The heat transfer enhancement of 4 % is achieved when the 14 fins is used and 16 % is obtained at 16 equivalent fins. The enhancement of heat transfer is lower than circular because the fins is act as heat transfer area improver only with lower turbulence intensity generator.

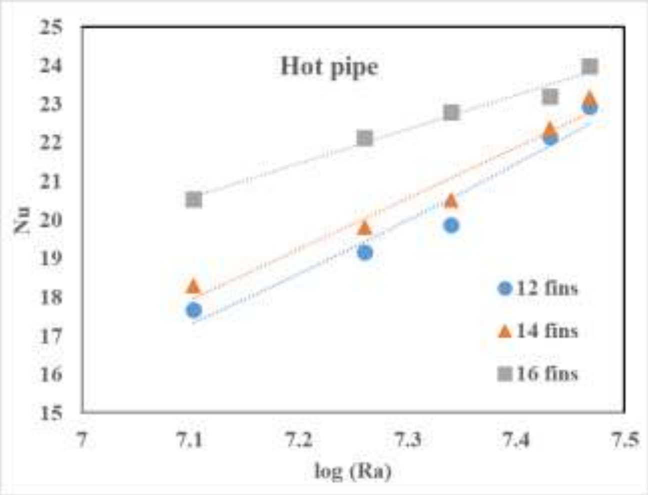


Fig. 12: Nu vs. log (Ra) for various equivalent fins number in hot pipe region.

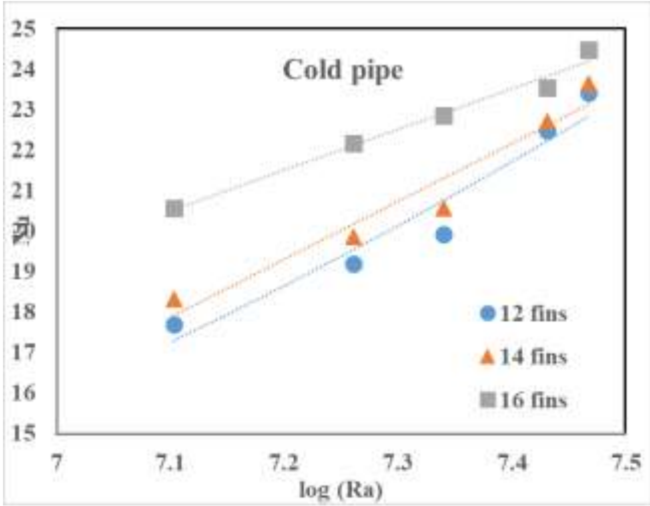


Fig. 13: Nu vs. log (Ra) for various equivalent fins number in cold pipe region.

The effect of equivalent circular fins R

Using two heated pipes, Figures 14 and 15 show Nu vs. log Ra for various comparable circular radius ratios for hot and cold pipes. In both hot and cold climates, the heat transmission increase is roughly 8.5 percent for R=2.5 and 13.22 percent for R=3. The rate of heat transmission accelerates as R increases. R increases the length of longitudal fins with non-channeling development, resulting in the smallest dead zone possible.

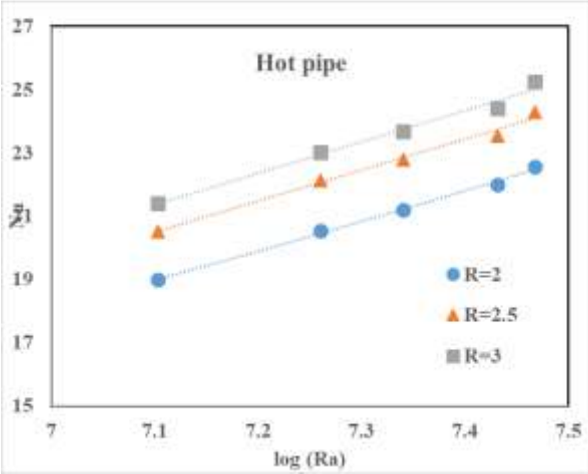


Fig. 14: Nu vs. log (Ra) for various equivalent fins R in hot pipe region.

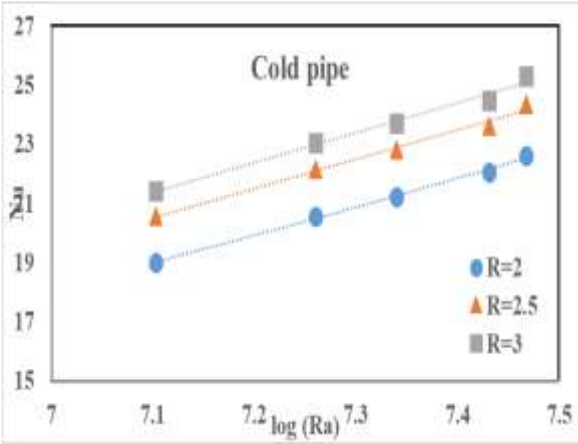


Fig. 15: Nu vs. log (Ra) for various equivalent fins R in cold pipe region.

The effect of equivalent circular fins As

Hot pipe Nu vs. Ra for different equivalent Aspect ratios (As) with 12 fins number and R=1.83 is shown in Figure 16. The Nu range values increase with each aspect (AS), with considerable alterations for the whole Ra. By 1.6 percent, the highest Nu is attained when log (Ra)= 7.46 and As=2.7. By reducing the distance between the pipes, the heat transfer dissipation across the fluid is increased. Between the close-by pipes, a greater volume of hot fluid will be created. The same behavior is observed in figure 17 in cold pipe region. The aspect ratio has no significant effect on free convection inside the enclosure.

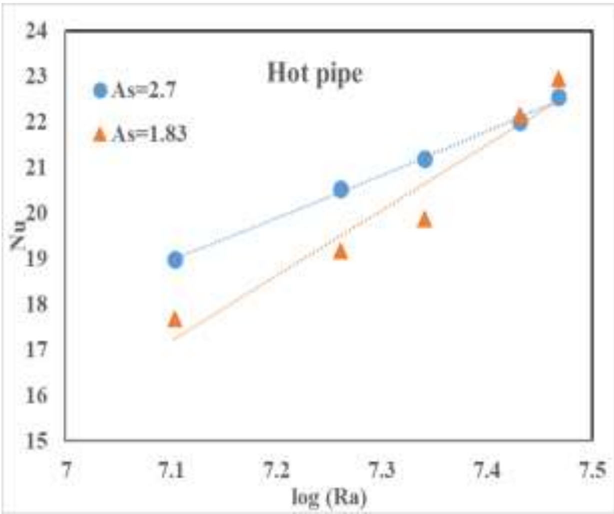


Fig. 16: Nu vs. log (Ra) for various equivalent fins As in hot pipe region.

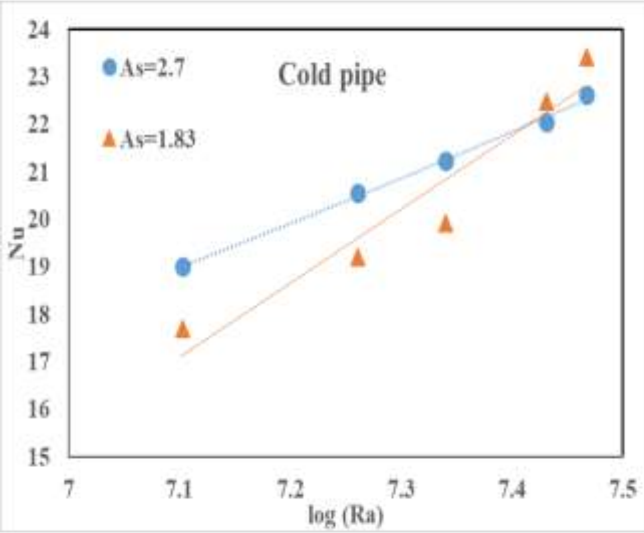
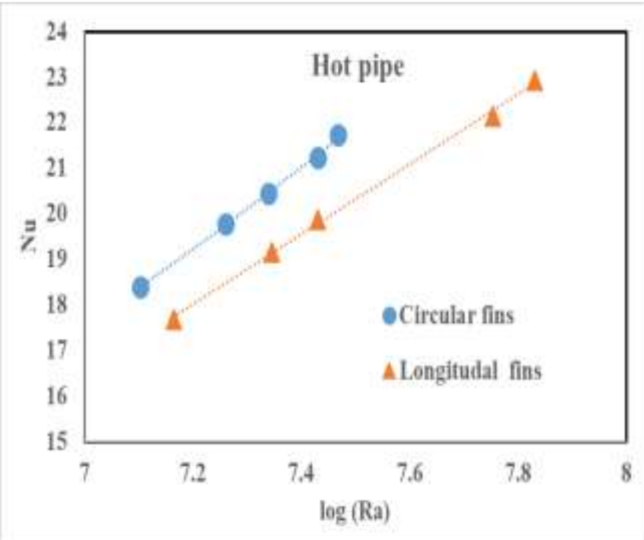


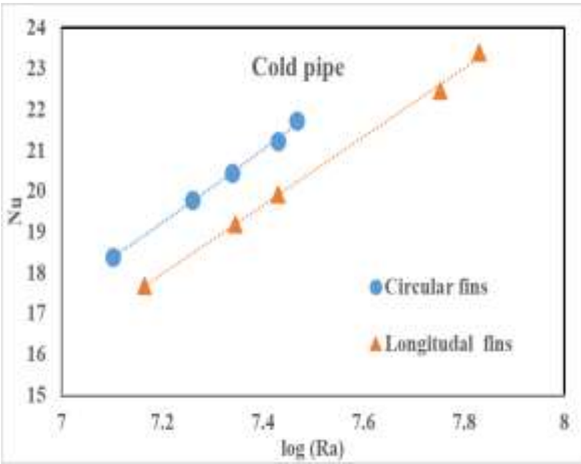
Fig. 17: Nu vs. log (Ra) for various equivalent fins As in cold pipe region.

The comparison between the fins geometries

Figures 18 show the Nu vs. log (Ra) for various fins geometries of the same heat transfer area (12 circular fins), R and As. The circular fins has heat transfer performance higher than the rectangular fins by 4.55 % where the turbulence intensity in circular fins higher than the longitudal fins.



a. hot pipe region

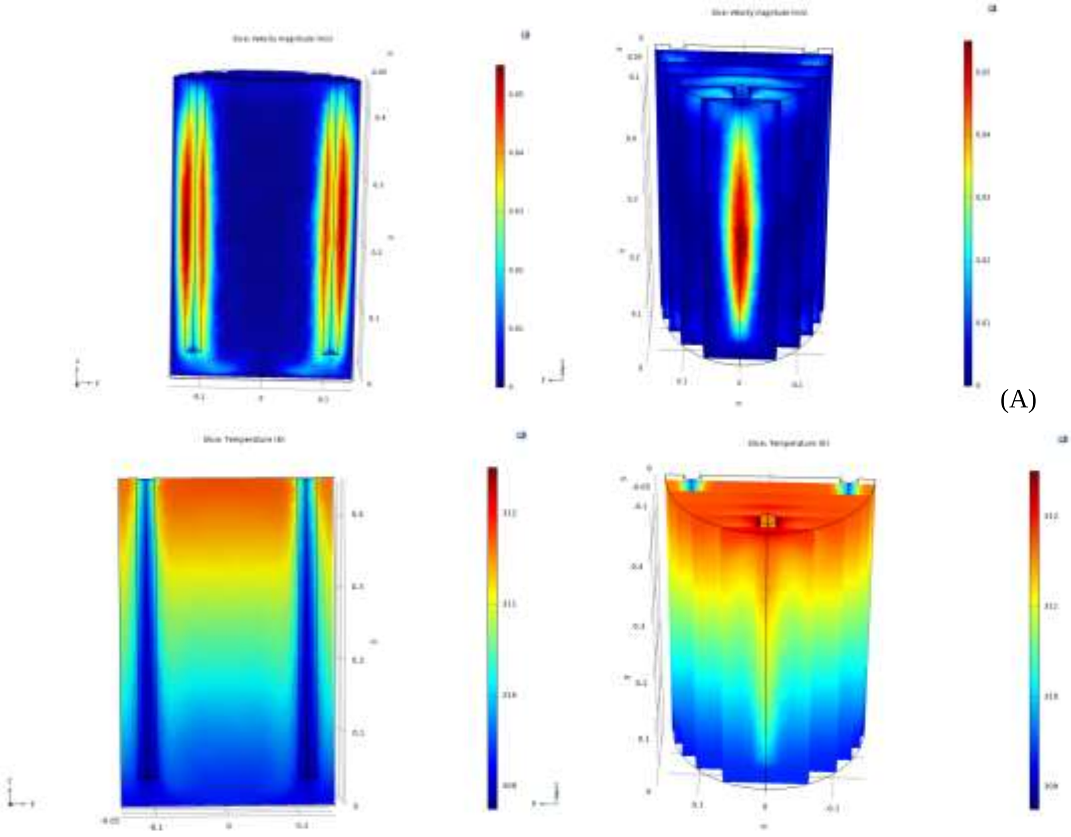


b. cold pipe region

Fig. 18: Nu vs. log (Ra) for various fins geometries.

The contour analysis of free convection

Figure 19 shows the contour analysis (velocity and temperature) of free convection; flat pipe is used as heated pipe. The velocity has high values nearby the pipes (cold and hot pipes), while the free stream has distributed grades from the highest value to lowest value. The hot fluid is submerged in the top of enclosure by action of density difference, so the maximum temperature values is placed upward and vice versa.



(A)

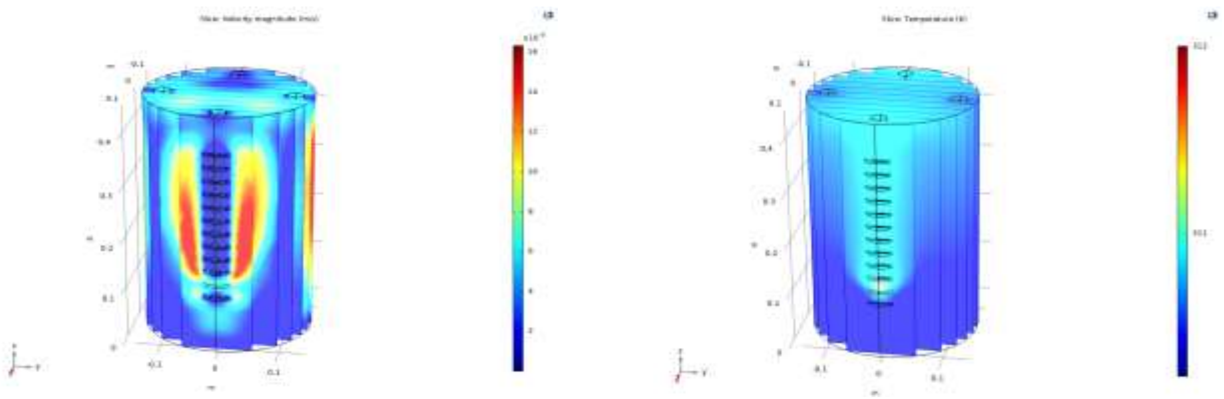
(B)

Fig. 19: Contour analysis of free convection of flat pipe: A. Velocity, B. Temperature.

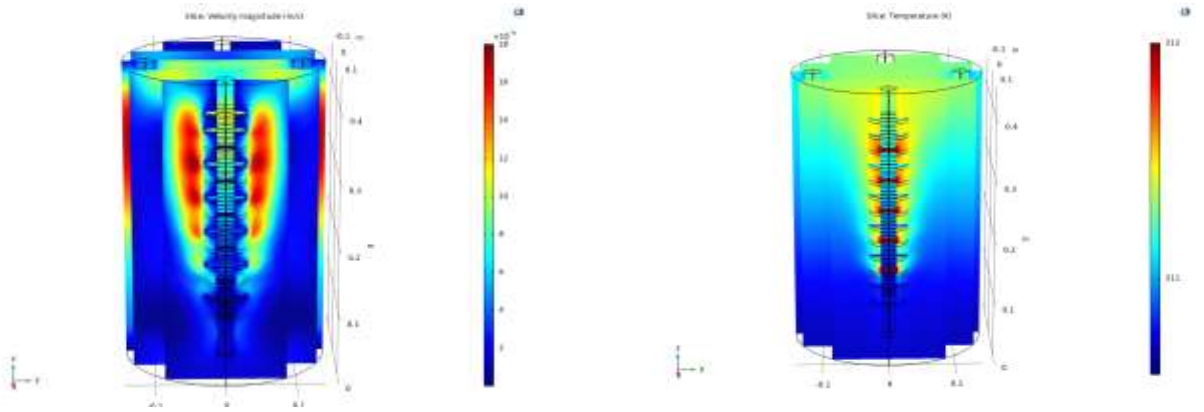
Figure 20 shows the contour analysis (velocity and temperature) of free convection; circular finned pipe is used as heated pipe. From photos, it seems that the hot and cold fluids are mixed sufficiently by applying the fins, the mixing tendency increases by increasing fins but the sufficient increasing generates drawbacks making the constant heat transfer enhancement.

A

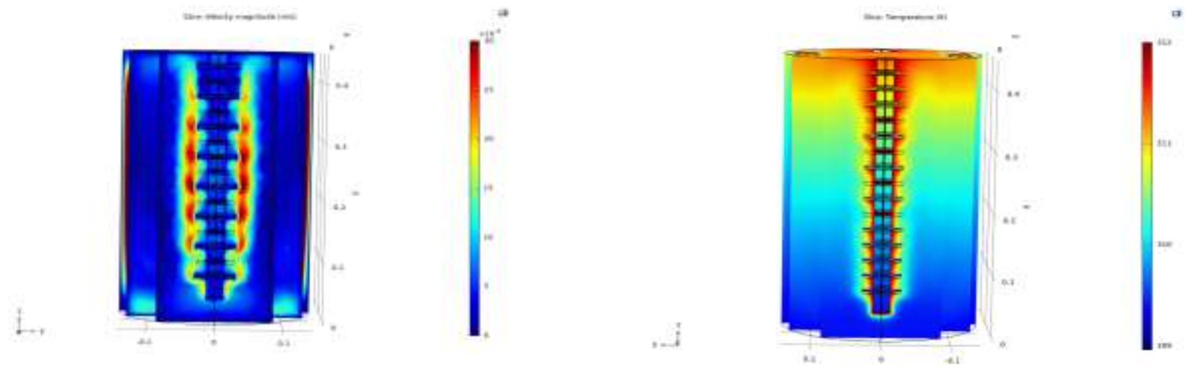
B



12 Fins



14 fins

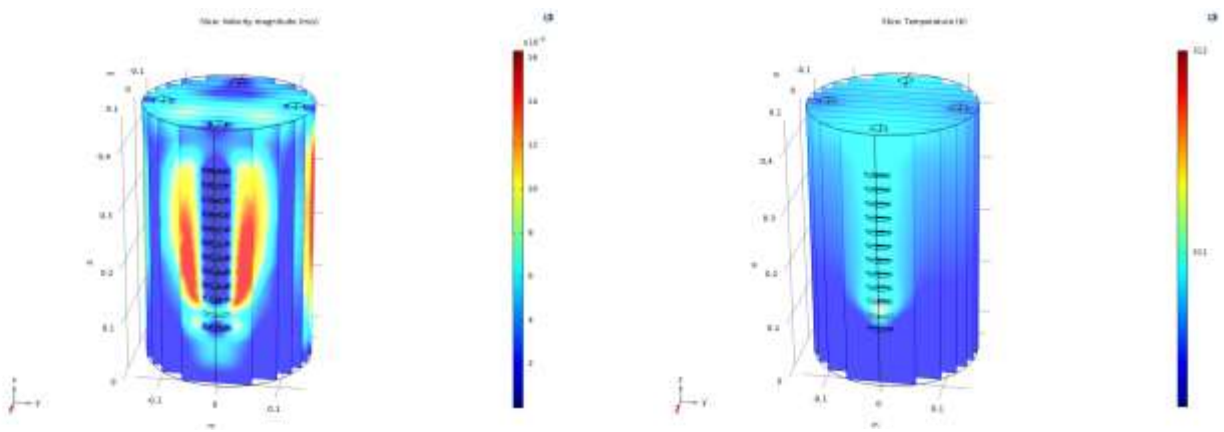


16 fins

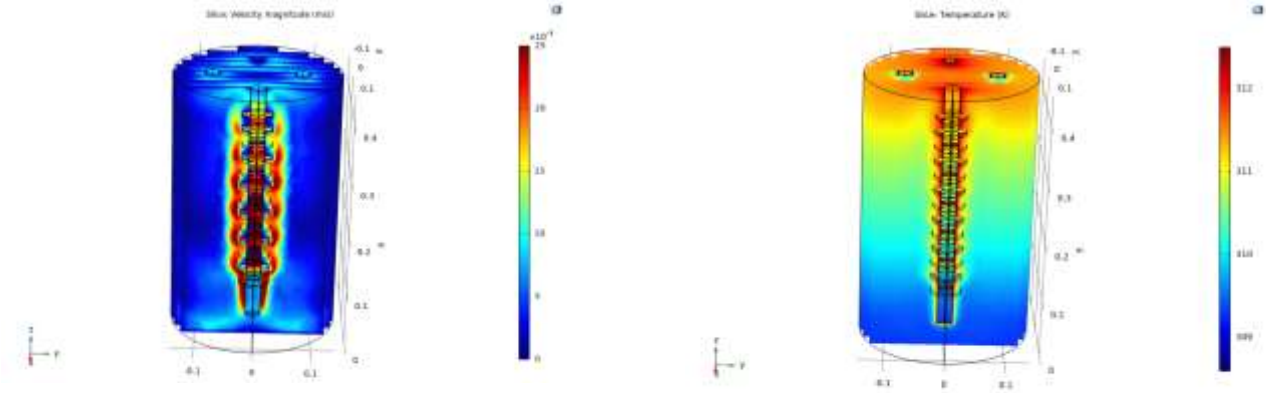
Fig. 20: Contour analysis of free convection of circularfinned pipe: A. Velocity, B. Temperature.

By increasing the number of fins in addition to the mini channels existing in the hot pipe wall, the heat transfer area is enhanced, and the heat transfer resistance is reduced to a minimum. Because of the turbulence intensity generated within the current system, the hot and cold fluid particles would be efficiently combined.

The increasing of aspect ratio promotes high fluid mixing tendency. The As=2.7 provides higher velocity distribution in the present investigation. The temperature profile has wider fluid heating which refers to higher amount of heat transfer rate as shown in fig. 21.

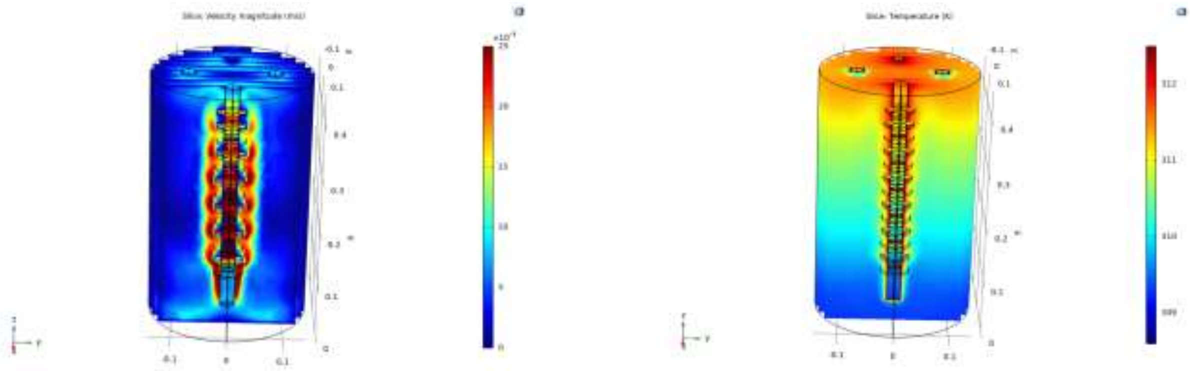


As=1.83

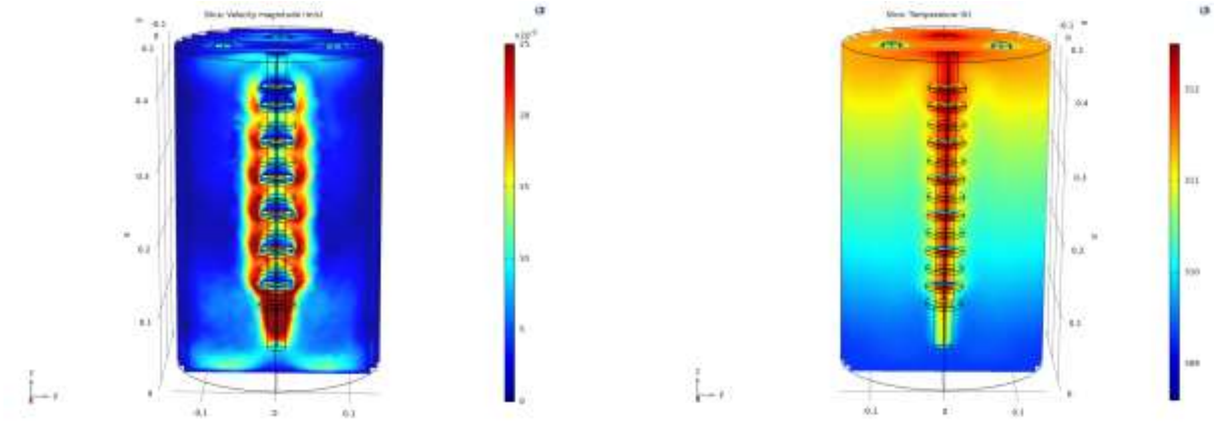


As=2.7

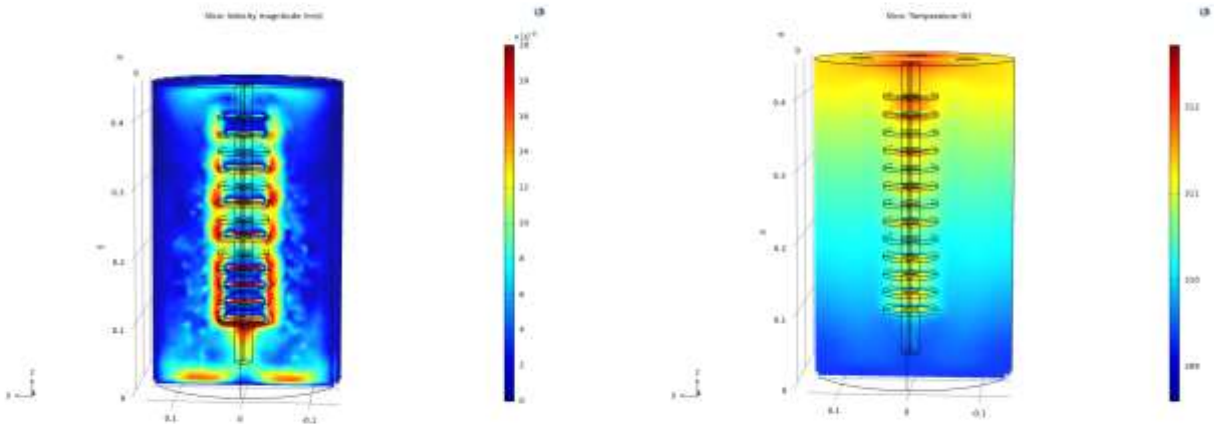
Fig. 21: Contour analysis of free convection of circular finned pipe of various As.



R=2



R=2.5

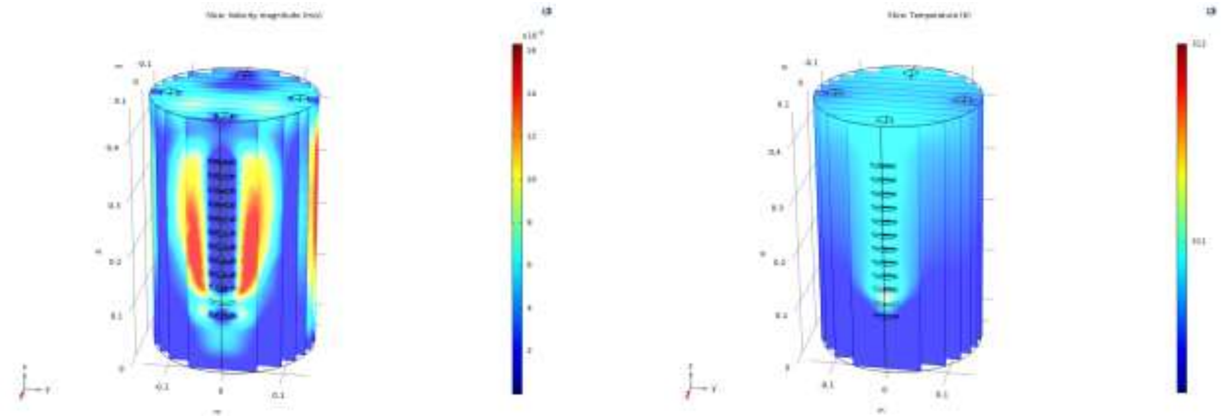


R=3

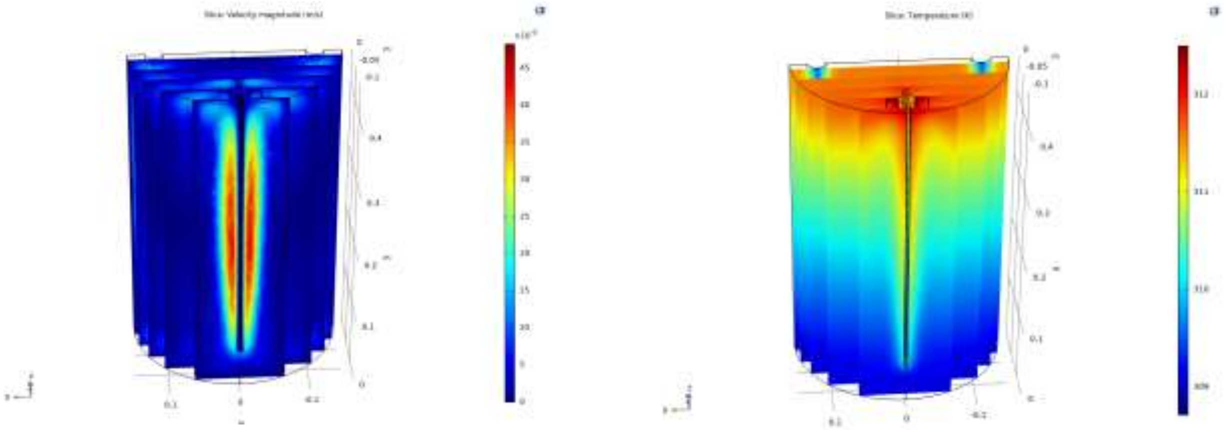
Fig. 22: Contour analysis of free convection of circular finned pipe of various R.

Figure 22 shows the contour analysis (velocity and temperature) of free convection; circular finned pipe is used as heated pipe for various R. The increasing of R leads to increase of the velocity volume and the heat transfer dispersion through maximum volume as possible. The increasing of R means heat transfer area increasing which enhances the heat transfer performance with corresponding of the channeling effect for high R.

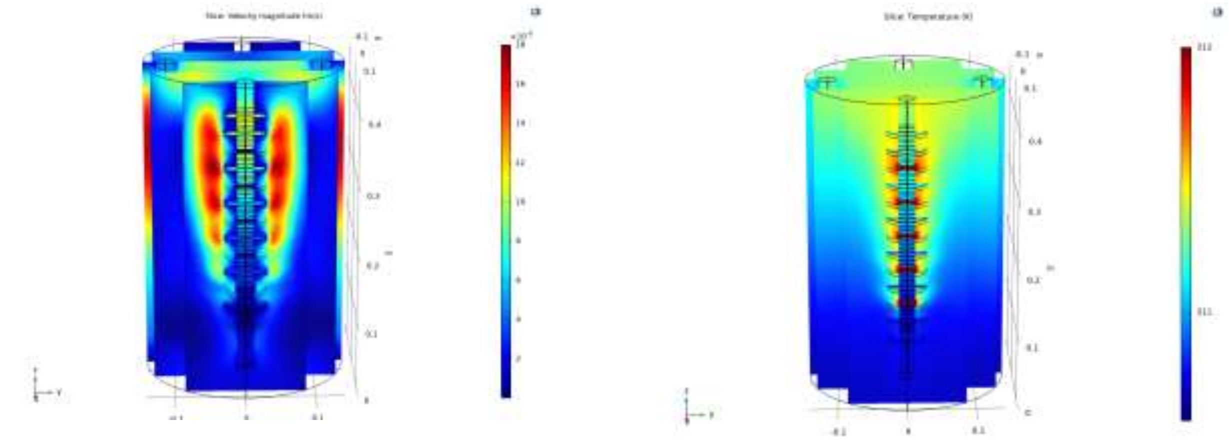
Figure 23 shows the contour analysis (velocity and temperature) of free convection; finned pipe is used as heated pipe for various fins geometries. The rectangular fins has the same surface area of circular fins. The channeling effect is not presented in rectangular fins (longitudal). The mixing of hot and cold fluids has more sufficient in longitudal finned pipe more than the circular one.



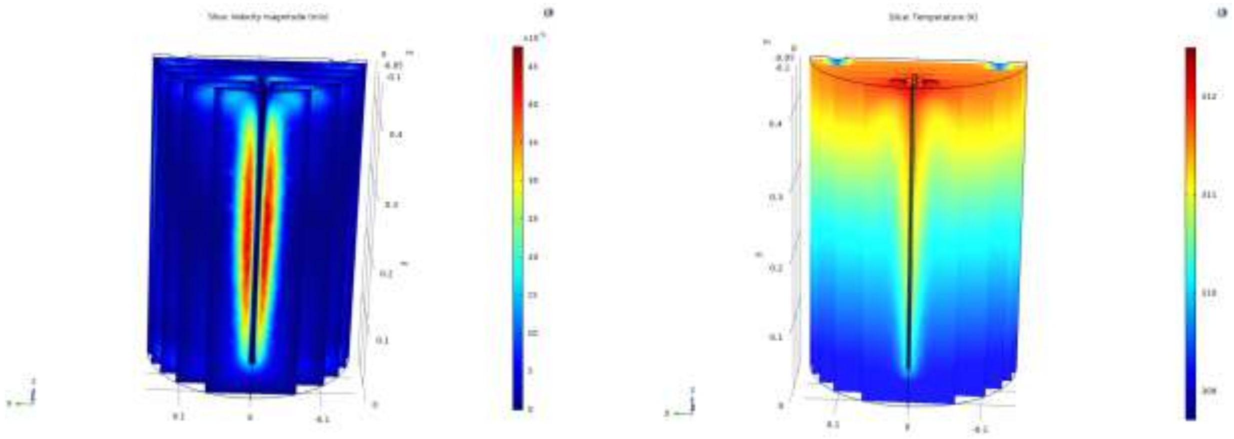
A.Circular finned pipes 12 fins



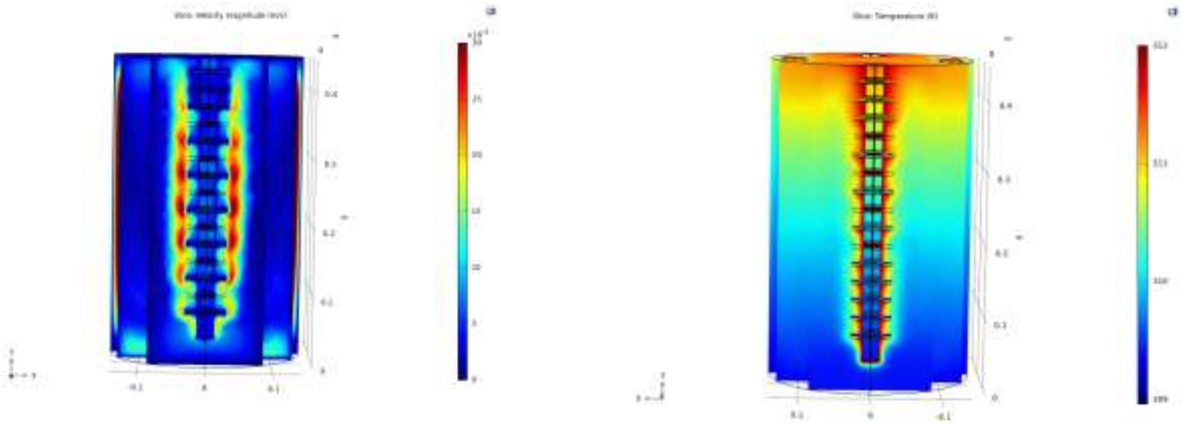
B.Rectangular finned pipes 12 equivalent fins.



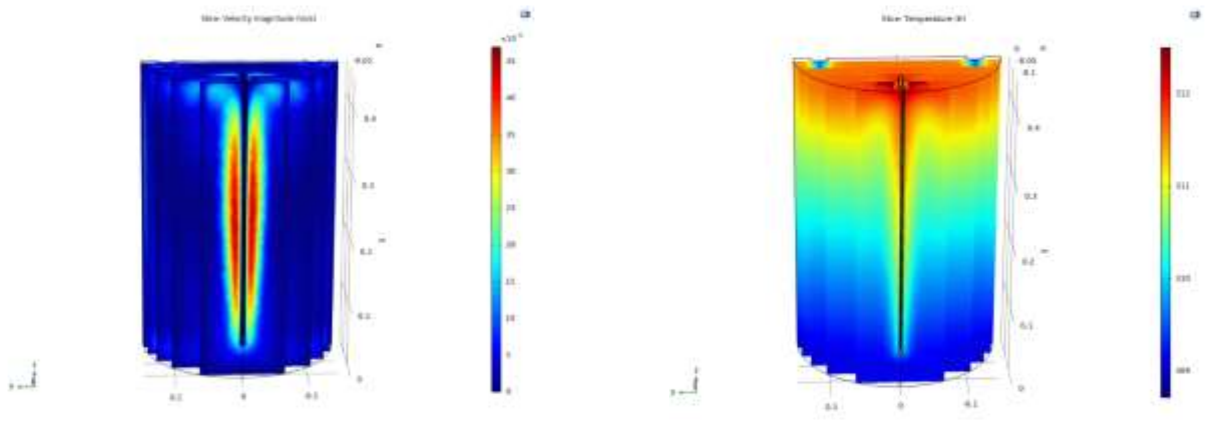
C.Circular finned pipes 14 fins



D.Rectangular finned pipes 14 equivalent fins.



E. Circular finned pipes 16 fins



F. Rectangular finned pipes 16 equivalent fins.

Fig. 23: Contour analysis of free convection of finned pipe of various fins geometries.

Conclusion

The numerical simulation of natural convection inside the cylindrical enclosure within two heated finned pipes has been studied successfully. The various parameters and parametric impact has been developed where the interaction is observed. The increasing of Ra , fins number, radius ratio (R) and aspect ratio (As) increases the Nusselt number for various geometries. The results show that The heat transfer enhancement rate for 14 fins is 22 percent, and for 16 fins it is 33 percent. It is projected that the heat transfer enhancement rate for 18 fins will be half of 16. The heat transfer increase is around 2.16 percent for R=2.5 and R=3 in the hot area and 7.38 percent for R=2.5 and 3 in the cold region. Where R is more than 2.5, the heat transfer enhancement is not possible to increase. Increased heat transfer surface area and the creation of deep channels are the results of raising the radius ratio but the too deep channels form dead zones makes a heat transfer enhancement barrier. By 0.96 percent, the highest Nu is attained when log (Ra)= 7.46 and As=2.7. By reducing the distance between the pipes, the heat transfer dissipation across the fluid is increased. Between the close-by pipes, a greater volume of hot fluid will be created.

The circular fins have heat transfer performance higher than the rectangular fins by 4.55 % where the turbulence intensity in circular fins higher than the longitudal fins. The thermal contours show the behavior of parametric analysis and interaction graphically and give indication of heat transfer improvement due to geometrical aspects.

References

[1] M. K. S. Kumar, “CFD Analysis of natural cnvection in different heated enclosures,” National Institute of Technology Rourkela, 2009.

[2] D. Das, M. Roy, and T. Basak, “International Journal of Heat and Mass Transfer Studies on natural convection within enclosures of various (non-square) shapes – A review,” Int. J. Heat Mass Transf., 2016.

[3] H. Ghodsinezhad, “Experimental Investigation on Natural Convection of Al2O3-water Nanofluids in Cavity Flow byNo Title,” University of Pretoria, 2016.

[4] A. A. Samarskii, “Numerical simulation of convection/diffusion phase change problems-a review,” /,,I J. H<w .WN.SS Trm\/e, vol. 36, no. 17, pp. 4095–41, 1993.

[5] B. V. K. S. Sai, K. N. Seetharamu, and P. A. Aswathanarayana, “Finite element analysis of heat transfer by natural convection in porous media in vertical enclosures Investigations in Darcy and non-Darcy,” Int. J. Numer. Methods Heat Fluid Flow, vol. 7, no. 4, pp. 387–400, 1997.

[6] G. Saha, S. Saha, M. Q. Islam, and M. A. R. Akhanda, “Natural convection in enclosure with discrete isothermal heating from below,” J. Nav. Archit. Mar. Eng., vol. 4, pp. 1–13, 2007.

[7] P. Kandaswamy, N. Nithyadevi, and C. O. Ng, “Natural convection in enclosures with partially thermally active side walls,” pp. 1–9, 2008.

[8] E. Bilgen and A. Balkaya, “Natural convection on discrete heaters in a square enclosure with ventilation ports,” Int. J. Heat Fluid Flow, vol. 29, pp. 1182–1189, 2008.

[9] O. Aydin and I. Pop, “Natural convection from a discrete heater in enclosures filled with a micropolar fluid,” Int. J. Eng. Sci. 43, vol. 43, pp. 1409–1418, 2005.

[10] S. M. Elsherbiny and E. H. Ragab, “Laminar natural convection in inclined rectangular cavities with a localized heat source,” Alexandria Eng. J., vol. 52, no. 3, pp. 249–257, 2013.

[11] R. P. Soni and M. R. Gavara, “Numerical Heat Transfer , Part A : Applications Natural convection in a cavity surface mounted with discrete heaters and subjected to different cooling configurations,” vol. 7782, no. July, 2016.

[12] M. Mokhtari, M. Barzegar Gerdroodbary, R. Yeganeh, and K. Fallah, “Numerical study of mixed convection heat transfer of various fin arrangements in a horizontal channel,” Eng. Sci. Technol. an Int. J., vol. 20, no. 3, pp. 1106–1114, 2017.

[13] M. A. El-Shorbagy, E. A. Algehyne, M. Ibrahim, V. Ali, and R. Kalbasi, “Effect of fin thickness on mixed convection of hybrid nanofluid exposed to magnetic field-Enhancement of heat sink efficiency,” Case Stud. Therm. Eng., vol. 26, no. April, p. 101037, 2021.

[14] A. K. ALShara, F. Abood, and S. M. Shrama, “An experimental investigation of mixed convection heat transfer in a rectangular enclosure,” south Tech. Univ. Int. Conf., vol. 42, no. March, 2016.

[15] S. K. Rout, D. P. Mishra, D. Nath Thatoi, and A. K. Acharya, “Numerical analysis of mixed convection through an internally finned tube,” Adv. Mech. Eng., vol. 2012, no. January, 2012.

[16] P. Kopeć and B. Niezgoda-żelasko, “Optimisation of the geometric parameters of longitudinally finned air cooler tubes operating in mixed convection conditions,” Processes, vol. 9, no. 1, pp. 1–15, 2021.

[17] M. Dogan and M. Sivrioglu, “Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel,” Int. J. Heat Mass Transf., vol. 53, no. 9–10, pp. 2149–2158, 2010.

[18] Junaid Khan and Shashank Mishra, “Computational Analysis of Mixed Convection of Internal Finned Tube,” Int. J. Eng. Res., vol. V5, no. 08, pp. 561–567, 2016.