Journal of Mechanical Engineering Research and Developments
ISSN: 1024-1752
CODEN: JERDFO
Vol. 43, No. 4, pp. 134-149
Published Year 2020

Advanced Thermal Analysis of Three-Dimensional Conjugate Heat Transfer with Radial Radiation in Horizontal Pipe for Sustainability

Ahmed M. Ajeena* and Hayder S. Al-Madhhachi

Department of Mechanical Engineering, College of Engineering, University of Kufa, Iraq

* *Corresponding Author Email: Ahmedm.dhayea@uokufa.edu.iq*

ABSTRACT: An advanced analysis of thermal behavior of three-dimensional conjugate heat transfer in a horizontal pipe with a fluid flow in a steady and transient states is performed. A numerical procedure of three-dimensional conjugate heat conduction in a solid wall and heat convection in a laminar flow with radial radiation is developed by analyzing a computational modeling of the solid-fluid domains. The procedure is based on the assumption that the gradient of temperature and heat flux of the solid-fluid domains are dependent on the thermal equivalent between conduction, convection and radiation heat transfers, which is true since the thermal loads on the pipe are reasonably identified. Annealed stainless steel as a common solid pipe is used with two types of fluids (air and water) for sustainability assessment. The results of the thermal analysis are represented by various parameters in the steady and transient states to emphasize the effect of the three-dimensional conjugate heat transfer combined with radial radiation on the fluid flow. These parameters are the temperature distribution and the heat flux from the ambient to the fluid flow and the fluid velocity. The environmental impacts on the pipe are estimated in terms of Air Acidification and Water Eutrophication. It can be concluded from the thermal analysis that the computational modeling of the thermal loads recommends for different types of engineering devices such as a convergent-divergent nozzle.

KEYWORDS: Thermal analysis; Conjugate heat transfer; Laminar fluid flow; Horizontal pipe; Sustainability.

INTRODUCTION

Heat transfer in solids and liquids regions are usually governed by conduction, convection and radiation heat transfer processes. These processes play an essential role in the thermal energy systems [1]. Conjugate heat transfer refers to an interaction of conduction in a solid wall and convection in a fluid flow. Conjugate heat transfer occurs in many important engineering devices, such as heat exchangers, nozzles, diffusers, microelectronic devices, ducts and pipes [2]. In the industrial applications, water and air are commonly used through various pipes for different working situations such as: heating and cooling systems, water treatment technologies, air conditioning and filling and emptying transportable tanks in homes, institutions and centers. There are several problems associated with water and air flows, such as water and air leaks and water condenses outside a pipe at the cold climate [3]. Fluid leaks, in general, occur at a location where an equipment connected directly to the compressed air ducts or saturated water pipes. Expansion and contraction processes as a result of cycling, opening, closing and vibrating which they are common causes of dropping and loosening at the connections, and thus fluid leaks problem [4]. The key parameters that could be changed in order to find the optimum design would be the pipe geometry, materials or even the control of pressure and mass flow rate of the fluid flow on a computational modeling program [5]. Direct numerical simulations of fluid flow mixed with convection in a differential heating process of a vertical channel were executed [6]. Their analysis concentrated on the classification of solid-wall with fluid-flow structures. The researchers found that the structures with a strong influence on the walls are generally responsible for locally large thermal energy rates. Another problems commonly referred as the volumetric heat conduction problems have been considerably studied in the literature for both two-dimensions and three-dimensions cases and recently has also been experimentally investigated [7-10]. In addition, there are several techniques in literature have been successfully applied to different kinds of problems related to fluid and thermal optimizations. The principles of these techniques are discussed in detail [11].

A studied topology optimization for coupled fluid-solid thermal problem to design and model heat exchangers under a constant input power using sequential quadratic programming [12]. The authors emphasized the role of appropriate material interpolation schemes in producing a perfect fluid/solid designs. A group researchers used a multistage optimization approach to model a non-conventional of a two-dimensions design of a heat sink under forced convection in COMSOL [13]. The modeling and simulation fluid flow methods of two phases in a pipe are relatively well considered. It could be conduct satisfactorily a true simulations of pressure drop and reproduce fluid-flow systems from a number of experiments [14-16]. Simultaneous with the regular advances associated with computing power, computational procedures and other approaches, it's a realistic developing to employ numerical simulations of several types of two and three dimensions multiphase pipe flow, including geometries and shapes such as T-junction risers, wavy pipe, ventilated vertical pipe, horizontal loop and a single-phase flow simulation in the annulus with a rotating inner or outer cylinder [17-23]. However, the temperature and heat transfer studies of the horizontal pipes continues to increase with the recent trend of the thermal engineering [24-26]. Arıcı and Kaya investigated the effects of the step change exposed to the outside boundary condition [27]. They have involved externally insulated upstream and consistently heated downstream parts of pipe wall for Peclet number range from 5 to 50 in the thermal analysis. The influence of axial conduction for both the fluid and the solid sections was taken into consider and it was concluded that the influence of wall axial conduction can directly change the influence of fluid axial conduction. The conjugate heat transfer problem of the pipes exposed to the step change was considered by Rzehak and Kriebitzsch under fully developing flow condition for both transient and steady-state conditions [28]. They concluded that for large values of the Peclet numbers, the fluid axial conduction is neglected and results generally depend on the wall characteristics relatively than on flow conditions.

The general trends noted from the studies in literature used a finite element method formulation and over the computational domain based on finite volume method to compute the fluid flow and temperature fields. A few studies investigated the conjugate heat transfer subjected to a radial radiation issues under the transient state in a horizontal specific pipe. As for the choice of objective function, most of the studies in literature consider either temperature distribution in a pipe or fluid flow properties. Almost all the studies in literature use a gradient-based optimization algorithm coupled with insulated systems for gradient computation. But most importantly, all these numerical techniques in literature, they are too dispersed without enough computations the sustainable models and the environmental impacts of the models. The key objectives of the research are to modeling and analyzing the conjugate heat transfer problem subjected to a radial radiation with a laminar water or air flow in a horizontal pipe to understand and recognize the distributions of the temperature and heat flux of the solid wall and fluid flow. Furthermore, this study could be a typical study and it exists in this field in a real world applications because it presents in details the effects of the conjugate heat transfer combined with radiation in a horizontal pipe on a fluid flow in the steady and transient conditions. The study is well estimated the environmental impacts of the horizontal pipe in terms of the air acidification and the water eutrophication.

PROBLEM STATATEMENT

Conduction heat transfer in solid wall combining with convection fluid flow subjected to radial radiation in a horizontal pipe is referred to as conjugate heat transfer problem. Conjugate heat transfer in pipe is a significant aspect in the design and analysis of thermal engineering devices. Numerical simulation approach can solve both the three-dimensional conduction equation and the fully developed energy equation of the fluid flow. The solution can be employed to estimate the environmental impacts on the horizontal pipe.

NUMERICAL SIMULATION PROCEDURE

Flow simulation procedure is discussed in details which includes the geometry and the mesh density of the model, the thermal loads on the model (including both solid and fluid domains), the physical situation, governing equations and the boundary conditions.

Geometry and Mesh Density

The geometry of the pipe as shown in Figure.1 consists of 0.5 m length with outer cylinder diameter of 0.1 m and inner cylinder diameter of 0.08 m. As pointed in Table 1, the engineering material (annealed stainless steel) is used to study the effects of the conjugate heat transfer combined with radiation on the velocity, temperature and heat flux distributions. For a precise simulation, three mesh densities are shown in Figure. 2 (a, b and c) and summarized in Table 2. According to the total nodes and elements, the fine mesh density is chosen in the thermal analysis.

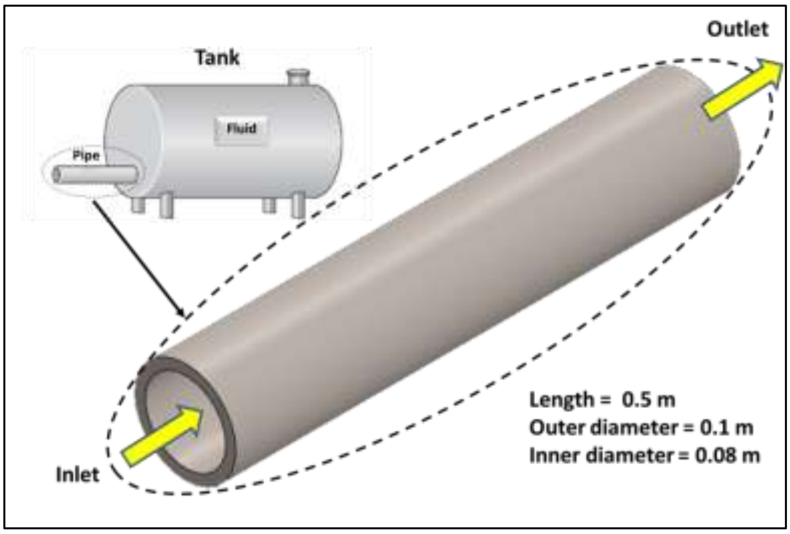


Figure 1. Geometry of the pipe.

Table 1. Properties of the engineering material used as a model.

Property	Annealed Stainless Steel
Elastic Modules	207 x 10^9 N/m^2
Poisson's Ratio	0.27
Density	7860 kg/m^3
Tensile Strength	6.85 x 10^8 N/m^2
Thermal Conductivity	16.3 W/m.K
Specific Heat	502 J/kg.K

Table 2. Solid-Mesh information of the pipe.

Mesh Density	Coarse	Medium	Fine
Mesh Type	Solid Mesh	Solid Mesh	Solid Mesh
Mesh Used	Standard Mesh	Standard Mesh	Standard Mesh
Jacobian Points	4 Points	4 Points	4 Points
Element Length	20 mm	10 mm	5 mm
Tolerance	1 mm	0.5 mm	0.25 mm
Mesh Quality	High	High	High
Total Nodes	4141	21394	127366
Total Elements	2037	11898	78140
Maximum Aspect Ratio	6.0228	4.97	3.855
Percentage of elements with Aspect Ratio < 3	97.3	99.2	99.9
Percentage of elements with Aspect Ratio > 10	0	0	0

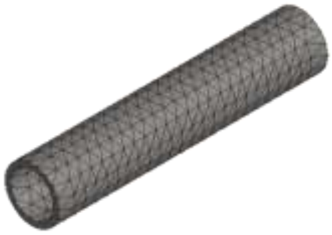
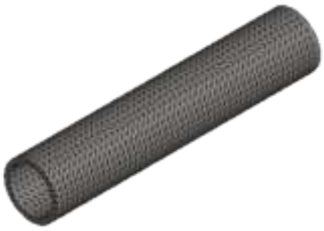
		
a. Coarse	b. Medium	c. Fine

Figure 2. Mesh density refinery.

Thermal Loads on the Solid Pipe

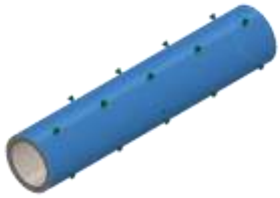
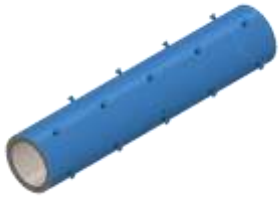
To perform thermal analysis, thermal loads such as; temperature, heat flux (including: conduction, convection and radiation modes) and heat power (as an external heat) are clearly defined. The thermal loads can describe on faces, edges, and vertices of the solid pipe (See Table 3).

Physical Situation

The steady and transient conditions of the conjugate heat transfer problem subjected to radial radiation involving convection (from the wall surface to the flowing fluid) and conduction (through the solid wall) in a horizontal pipe, as shown schematically in Figure. 3. The solid walls exposed uniformly to the radial radiation q_{rad} and transfer to the fluid which moves with a velocity u.

Table 3. Thermal loads on the solid pipe.

Load Name	Load Image	Details
Initial Temperature		Entities: 1 face State: Fluid Type of fluid: Water / Air Temperature: 293.2 K
Initial Temperature		Entities: 1 face State: Solid Type of Solid: Annealed Stainless Steel Temperature: 293.2 K
Heat Power		Entities: 0 face Heat Power Value: 0 W/m^2 Heat Power Direction: None

Conduction		Entities: 1 face Conduction Type: Radial Thermal Conductivity: 16.3 W/m.K Outer Surface Area: 0.157 m^2
Convection		Entities: 1 face Flow Type: Laminar Heat Transfer Coefficient: For water: 100 W/(m^2·K) For Air: 10 W/(m^2.K) Inner Surface Area: 0.126 m^2
Radiation		Entities: 1 face Radiation Type: Radial Emissivity: 0.87 Inner Surface Area: 0.157 m^2

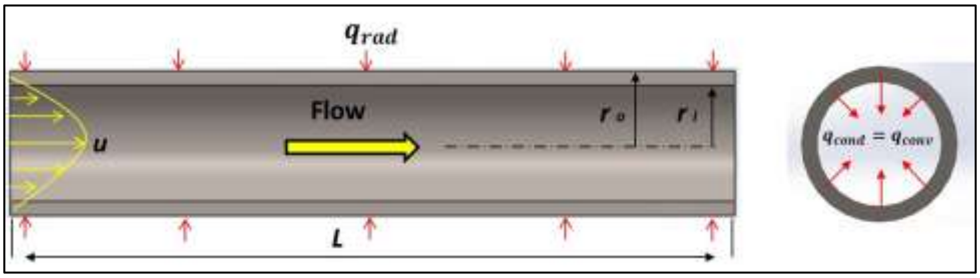


Figure 3. Schematic diagram of the physical situation.

Where, q_{cond} is the conduction heat transfer through the annealed stainless steel, q_{conv} is the convection heat transfer between the inside wall surface and the fluid (air or water), L is the length of the pipe and r_o and r_i are the outer and inner radius of the pipe. Table 4 presented the properties of the fluids (air and water) using for the thermal analysis.

Table 4. Fluids properties.

Property	Air	Water	Unit
Density	1.1	1000	Kg/m^3
Thermal Conductivity	0.027	0.61	W/m.K
Specific Heat	1000	4200	J/kg.K

Governing Equations of the Fluid flow

The behavior of the air and water in the numerical simulation is specified by the conservation laws: 1) Continuity equation, 2) Newton second law and 3) First law of thermodynamics equations which described by the following expressions:

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \cdot \vec{v}\right) = S_m \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho \cdot \vec{v}\right) + \nabla\left(\rho \cdot \vec{v} \cdot \vec{v}\right) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \cdot \vec{g} + \vec{F} \tag{2}$$

$$\frac{\partial}{\partial t}(\rho \cdot E) + \nabla \cdot \left(\vec{v}(\rho E + p)\right) = \nabla \cdot \left(k_{eff} \nabla T - \sum_j h_j \cdot \vec{J}_j + \left(\bar{\bar{\tau}}_{eff} \cdot \vec{v}\right)\right) + S_h \tag{3}$$

Assumptions

There are specific assumptions which are not influence on the energy balance pointed as following: 1) The flow is fully developed laminar. 2) The heat generation due to friction is negligible. 3) The thermal conductivity of the material pipe doesn't change with temperature. 4) There is no piping losses, which means all heat transfer across the pipe are equivalent (q_{rad} = q_{cond} = q_{conv}).

Boundary Conditions (B.Cs.)

The fluids used in this study are water and vapour designed as the primary and secondary phase, respectively. Boundary conditions used for this model are explained in the following subsections:

Inlet B.C.

The fluid specific initialization used in this study at t = 0 is water or air. The fluid temperature entering the horizontal pipe is 20 °C and the volume fraction water and air is (0.001, 0.999) respectively. The reference pressure is selected (1 atm), and the buoyancy reference density is 1.225 kg.m^3

Outlet B.C.

The outlet boundary condition is set depending on the ambient temperature and reference pressure.

RESULTS AND DISCUTIONS

The variations of the fluid velocity in the pipe are shown in the Figures (4 – 10). The pipe is filled with two working fluids (air or water) at a specific values of pressure, temperature and volume flow rate (P_{inlet}, T_{inlet} and $\dot{V}_{inlet}$ at the inlet) and (P_{outlet}, T_{outlet} and $\dot{V}_{outlet}$ at the outlet), respectively.

Fluid Velocity Variation in the Pipe at Constant Pressure and Temperature

When the temperature and the pressure at the inlet and at the outlet are constants (293.2 K and 101.325 kPa), the maximum velocity of air and water in the pipe are 107.9 m/s and 103.1 m/s respectively, as shown in Figures. 4 and 5. The volume flow rate is considered as a constant value. The heat transfer coefficient of water and air are 100 W/m^2.K and 10 W/m^2.K ,respectively. The movement of the air molecules is at random impingement with each other and with the surface of the solid pipe. The conjugate heat transfer combined subjected to radial radiation is affected sensibly on the molecules movements of water and air. Air molecules are at a considerably higher velocity than the water molecules.

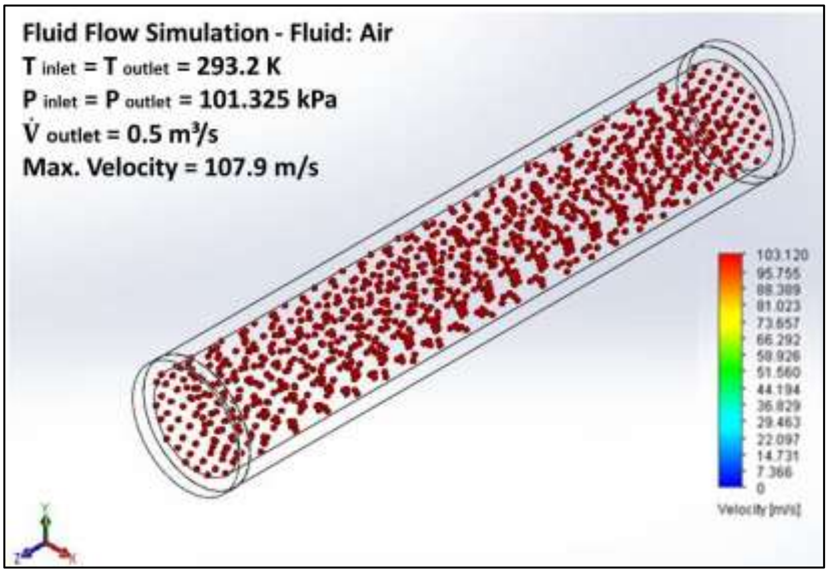


Figure 4. The variation of the air velocity at the constant P and T values.

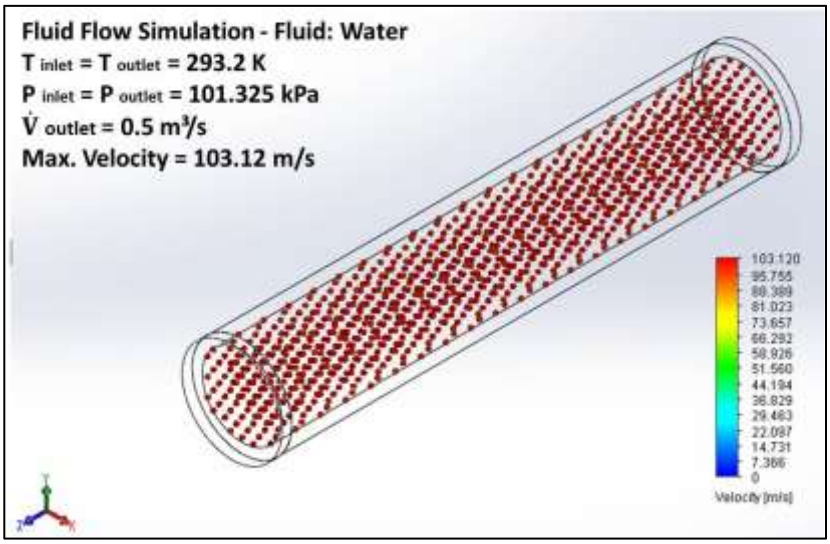


Figure 5. The variation of the water velocity at the constant P and T values.

Effect of Increased Pressure of the fluid at Constant Temperature on the Velocity

An isothermal process is a process that operating at unchanged temperature. When the pressure at the inlet is doubly compressed than at the outlet, while the temperatures at the inlet and outlet are constants, the maximum velocity of air and water in the pipe are 334.5 m/s and 327.2 m/s respectively, as shown in Figures. 6 and 7. The volume flow rate is also considered as a constant value. The heat transfer coefficient of water and air are 100 W/m^2.K and 10 W/m^2.K ,respectively. The conjugate heat transfer combined subjected to radial radiation is affected sensibly on the molecules movements of water and air. Under this process, Air molecules are at a slightly higher velocity than the water molecules. The process of increasing the pressure at constant temperature could be happen at the vacuum situation. Vacuum situation is generally more complex than a conventional process, and its use is controlled to the thermal applications that result in steady condition.

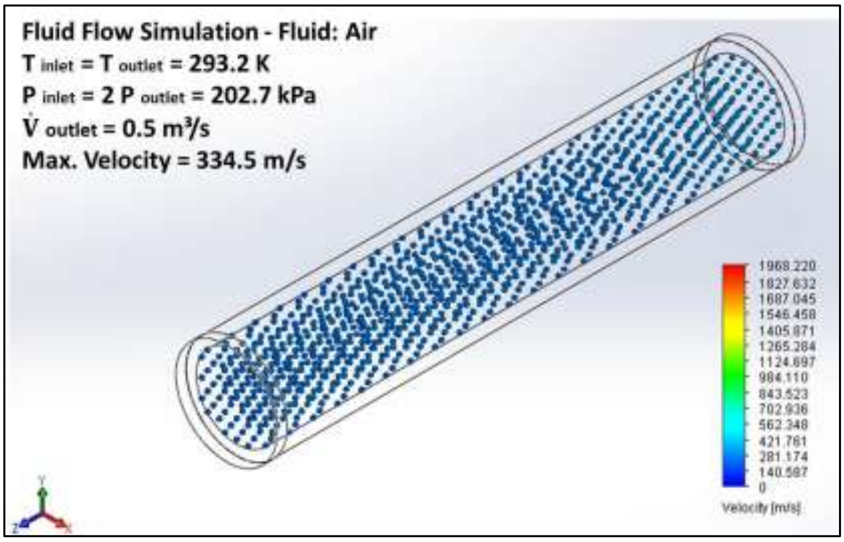


Figure 6. The variation of the air velocity at the increased pressure and constant temperature.

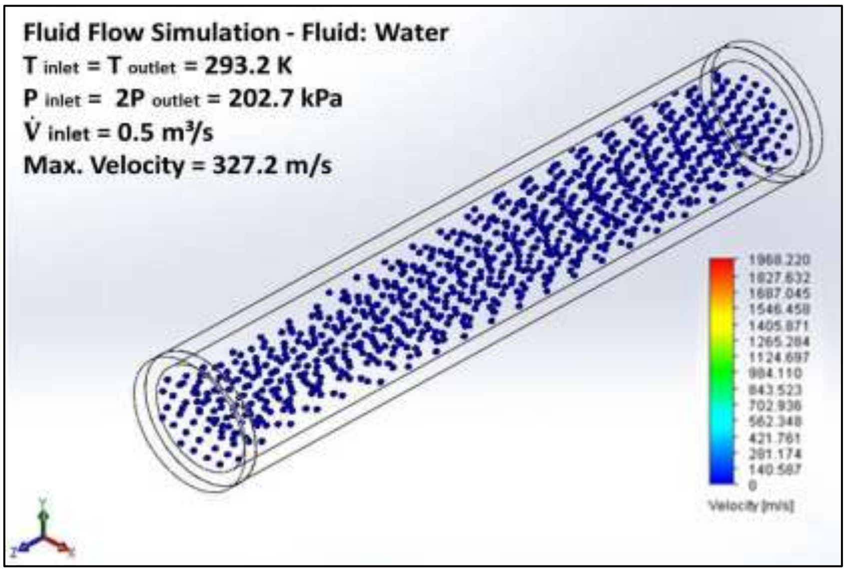


Figure 7. The variation of the water velocity at the increased pressure and constant temperature.

Effect of Increased Temperature of the fluid at Constant Pressure on the Velocity

An isobaric process is a process that operating at unchanged pressure. When the temperature at the inlet is doubly increased than at the outlet, while the pressure at the inlet and the outlet are constants, the maximum velocity of air and water in the pipe are 111.2 m/s and 102.9 m/s respectively, as shown in Figures. 8 and 9. The volume flow rate is also considered as a constant value. The heat transfer coefficient of water and air are 100 $W/m^2.K$ and 10 $W/m^2.K$,respectively. The conjugate heat transfer combined subjected to radial radiation is affected sensibly on the molecules movements of water and air. Under this process, Air molecules are also at a slightly higher velocity than the water molecules. At these velocities, the fluids move smoothly and orderly without lateral mixing in layers.

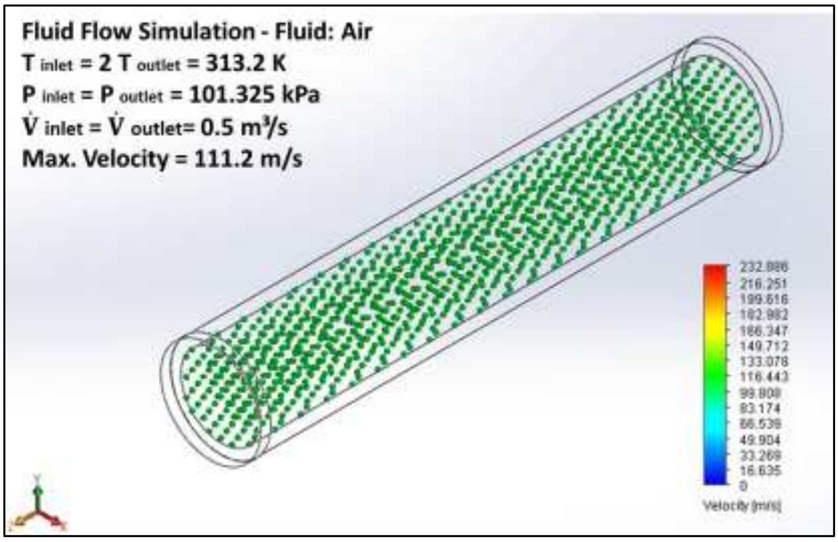


Figure 8. The variation of the air velocity at the increased temperature and constant pressure.

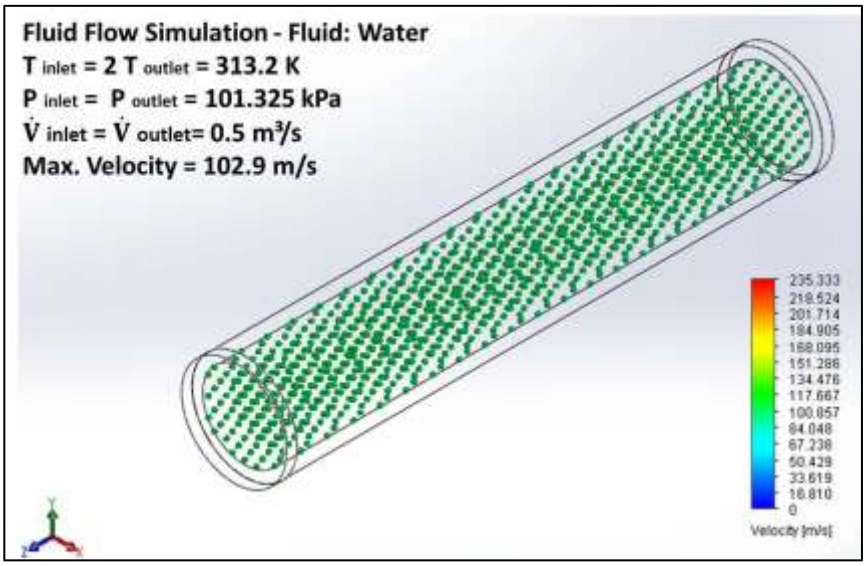


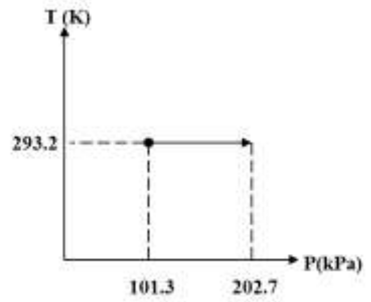
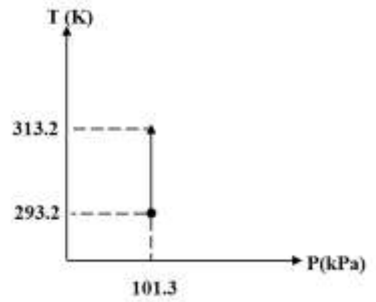
Figure 9. The variation of the water velocity at the increased temperature and constant pressure.

Velocity Analysis at the Isothermal and Isobaric Processes

The analysis of the air and water velocities at the isothermal and isobaric processes are presented in Table 5. In general, The air velocities at both processes are a considerably higher than the water velocities at constant mass flow rate, this is because of level of the energy of two phases of the fluids. The energy level increases at these processes and causes to rise the velocity of the fluid. The maximum velocity (for air 334.5 m/s) is recorded at the isothermal process and doubled pressure, while the computational modeling is recorded the minimum value of the water velocity (102.9 m/s) at the isobaric process and doubled temperature.

Table 5. The velocity analysis of air and water at the isothermal and isobaric processes.

	Isothermal process (at T=293.2 K)	Isobaric process (at P=101.325 kPa)

Fluid Type				
	Max. Velocity at P $_{inlet}$=101.325 kPa	Max. Velocity at P $_{inlet}$=202.7 kPa	Max. Velocity at T $_{inlet}$=293.2 K	Max. Velocity at T $_{inlet}$=313.2 K
Air	107.9 m/s	334.5 m/s	107.9 m/s	111.2 m/s
Water	103.1 m/s	327.2 m/s	103.1 m/s	102.9 m/s

Thermal Behavior at the Steady State Condition

The majority of thermal devices operate for long intervals of time under the consistent conditions, and they are recognized as steady state devices. The processes involving such pipes can be illustrated reasonably by a certain idealized conditions. However, in the thermal analysis of the specific heat transfer problem through the pipe in which the same conditions apply, the research is primarily interesting in the spatial distributions of properties, particularly temperature and heat flux.

Temperature Distribution of the Solid Pipe

The temperature distribution of the solid pipe is noticeably uniform at the steady state condition as shown in Figure. 10. The emissivity of ambient is considered as a constant value. Hence, the steady temperature of the solid pipe is 293.2 K.

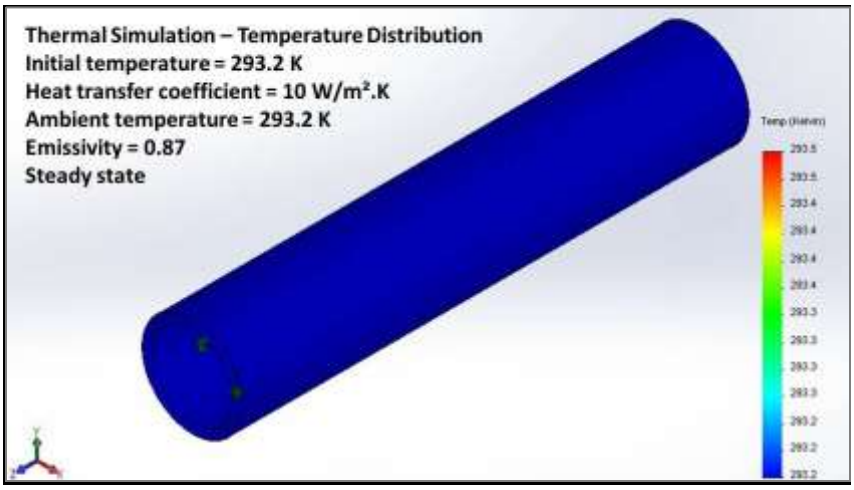


Figure 10. The temperature distribution of the solid pipe under the steady state condition.

Heat Flux Distribution of the Solid Pipe

The heat flux distribution of the solid pipe is noticeably uniform at the steady state condition as shown in Figure. 11. The emissivity of ambient is considered as a constant value. Hence, the steady heat flux of the solid pipe is 166.7 W/m^2.

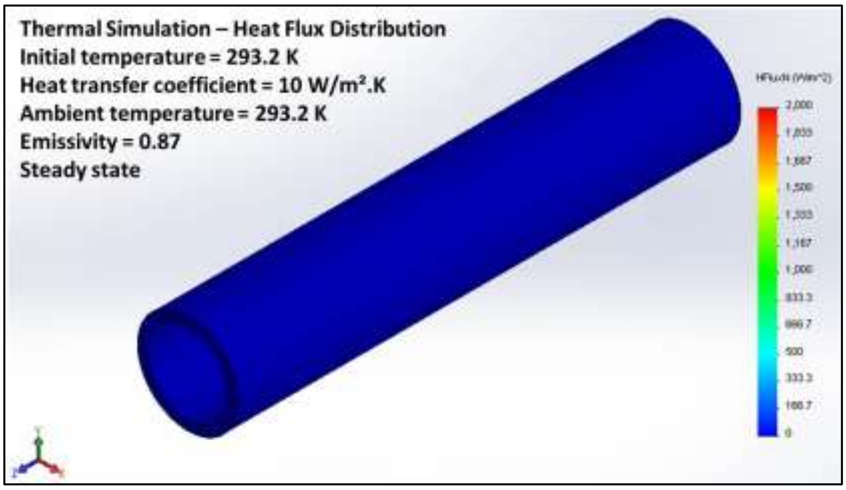


Figure 11. The heat flux distribution of the solid pipe under the steady state condition.

Thermal Behavior at the Transient Condition

The analysis for the thermal behavior of the solid pipe is extended with the presentation under one hour of the transient condition.

Temperature Distribution of the Solid Pipe after One Hour

The temperature distribution of the solid pipe is almost uniform at the transient condition as shown in Figure12. The emissivity of ambient is considered as a constant value. Hence, the maximum temperature is 295.3 K after one hour exposing to the radial radiation. There is a minor variation of the solid temperature through the cross sectional area of the horizontal pipe.

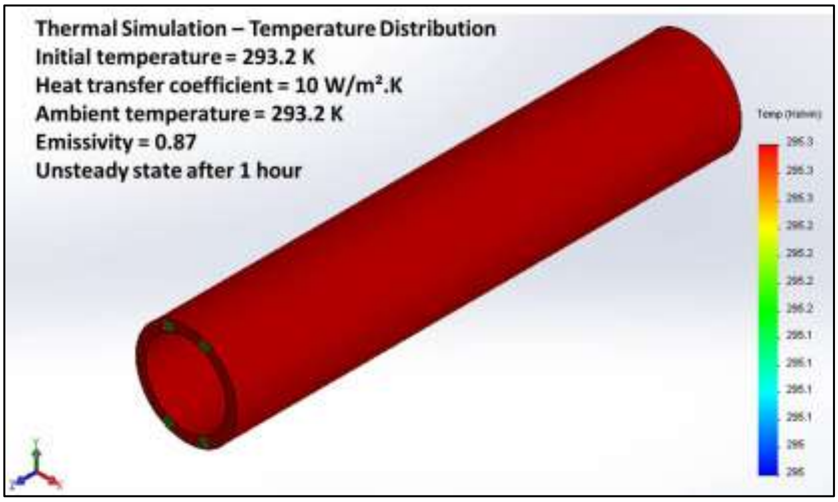


Figure 12. The temperature distribution of the solid pipe under the transient condition.

Heat Flux Distribution of the Solid Pipe after One Hour

The heat flux distribution of the solid pipe is noticeably non-uniform at the transient condition as shown in Figure 13. After one hour exposing to the radial radiation, the maximum and minimum heat fluxes of the solid pipe are about 1600 W/m^2 and 1200 W/m^2 ,respectively. Its interesting issue to consider the heat flux at the cross sectional area of the input face. The values of the heat flux at the input face with the parametric distance are shown clearly in Figure 14.

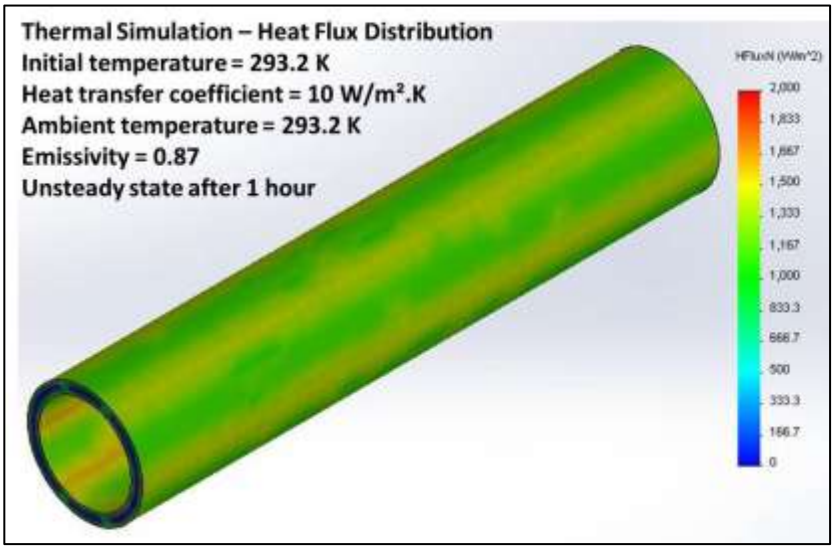


Figure 13. The heat flux distribution of the solid pipe under the transient condition.

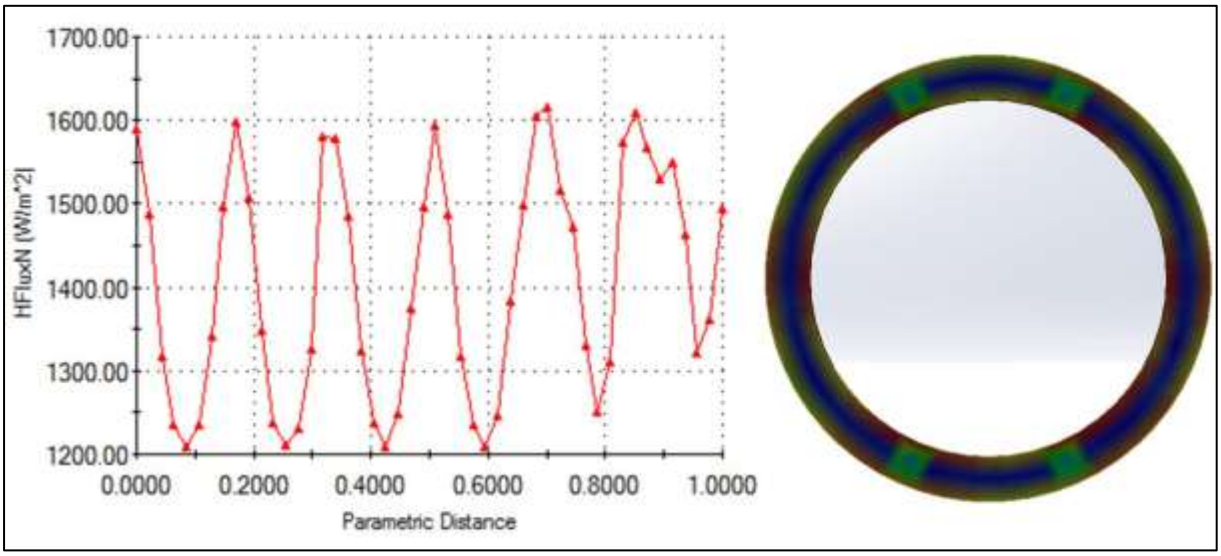


Figure 14. The values of the heat flux at the input face of the solid pipe under the transient condition.

Effect of Heat Transfer Modes on the Water Temperature at the Transient Condition

The effect of the heat transfer modes (conduction, convection and radiation) on water temperature at the transient condition after 1, 2 and 3 hours is shown in Figure 15. Its significant consideration to analyze the transient state, three-dimensional heat flow through the pipe which is exposed to the radial radiation. In that case, the heat is transferred by conduction through the solid material of the pipe to the flowing water by the convection heat transfer. The maximum water temperature is 296.8 K after 3 hours of the heat transfer.

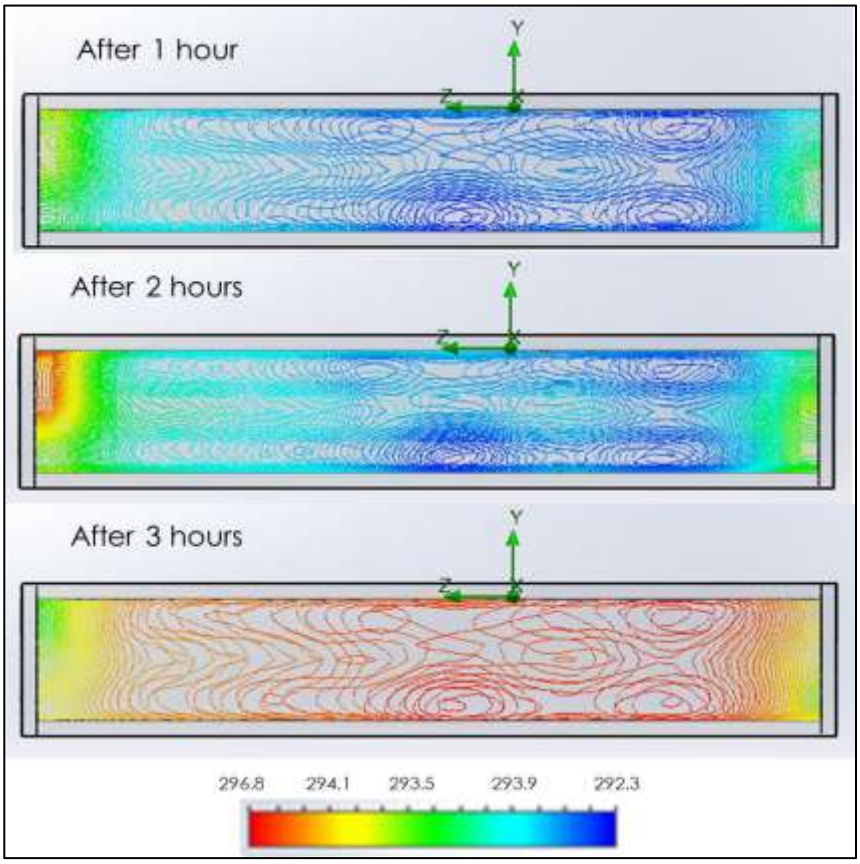


Figure 15. Water temperature in the pipe at the transient condition.

Environmental Impacts on the Pipe

The selection of manufacturing region evaluates the energy sources and the technologies used in the modeled engineering material designing and manufacturing stages of the life cycle of the product. The use region is utilized to evaluate the energy sources consumed during the using of the engineering material phase and the destination for the product at its end of the material life. At the same time with the manufacturing region, the use region is also utilized to evaluate the environmental impacts associated with transfering the product from the location of the manufacturing to the consuming. Environmental impacts are estimated using sustainability assessment methodology. When determine the environmental impacts during 20 years, assessment of the life cycle observes at what occurs in the production, use and final disposal steps. This involves the impact of the transportation that happens between the stapes. Assessments on the engineering material used, manufactured and other features can result in many different effects on the environment. Table 6 illustrates the environmental impacts on the pipe in terms of Air Acidification and Water Eutrophication.

Table 6. The environmental impacts on the pipe during 20 years.


Air Acidification		Water Eutrophication	
Material:	16 kg SO_2e	Material:	6.6 kg PO_4e
Manufacturing:	0.00 kg SO_2e	Manufacturing:	0.00 kg PO_4e
Transportation:	0.224 kg SO_2e	Transportation:	0.046 kg PO_4e
End of Life:	3.4 kg SO_2e	End of Life:	0.443 kg PO_4e
19 kg SO_2e		7.1 kg PO_4e	

Where Air Acidification consists primary of sulfur dioxide (SO_2) and other acidic releases to air cause an raises in the acidity of the rain, which is normal amount (19 kg) according to the world health organization. Large amounts of (SO_2) can cause the ground and the water toxic for the plants and the aquatic life. Water Eutrophication is the additional amounts of nutrients are accumulated to a water ecosystem, this can be a problem causes eutrophication. Nitrogen and phosphorous from a waste water and the agricultural fertilizers affects an overabundance of algae to bloom, which then reduces the amount oxygen in water and accordingly results in the death of both plants and animals life. This impact (7.1 kg) is normal value according to the world health organization which is measured in kg phosphate equivalent (PO_4).

CONCLUSIONS

The thermal behavior of three-dimensional conjugate heat transfer subjected to the radial radiation in the horizontal pipe with water and air flow in the steady and transient states is analyzed. The following points are concluded:

- Developing the numerical procedure of the conjugate heat transfer subjected to radial radiation by employing the thermal loads in the computational modeling.
- Optimizing the sustainable pipe using annealed stainless steel as a common solid pipe with two types of fluids (air and water).
- Analyzing the effects of the isothermal and isobaric processes on the fluid velocity.
- Analyzing the distributions of the temperature and heat flux of the solid pipe under the steady and transient conditions.
- Reasonable environmental impacts on the pipe in terms of Air Acidification and Water Eutrophication.
- It can be concluded from the thermal analysis that the computational modeling of the thermal loads is recommended for different types of engineering devices such as a convergent-divergent nozzle as shown in Figure.16.

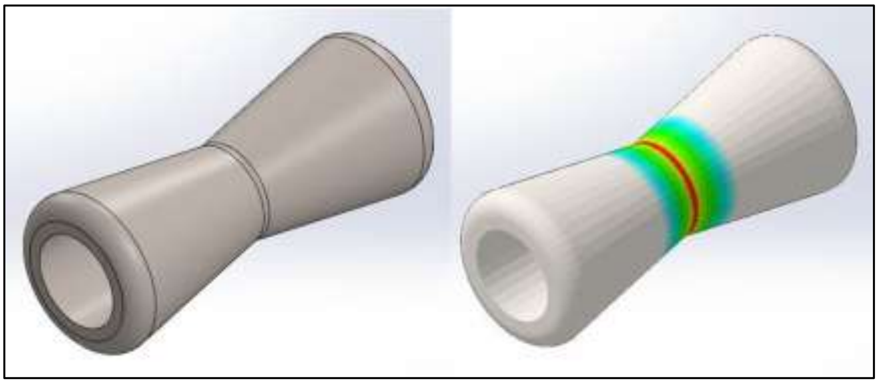


Figure 16. A convergent-divergent nozzle recommended for thermal analysis.

Nomenclature			
E	Energy (J)	Creek Symbols	
F	Force (N)	*v*	Specific Volume (m^3/kg)
g	Ground Acceleration (m/s^2)	*ρ*	Density (kg/m^3)
h	Heat Transfer Coefficient ($W/m^2.K$)	Subscripts	
k	Thermal Conductivity (W/m.K)	*cond.*	Conduction
L	Length (m)	*conv.*	Convection
P	Pressure (Pa)	*eff*	Effective
q	Heat Transfer (W)	*in*	Inlet
T	Temperature (^oC)	*out*	Outlet
t	Time (s)	*rad.*	Radiation
V	Volume Flow Rate (m^3/s)		

ACKNOWLEDGEMENT

We wish to acknowledge the financial and scientific supports from the Iraqi Ministry of Higher Education and Scientific Research, University of Kufa.

REFERNCES

[1] S. He, K. He, and M. Seddighi, "Laminarisation of flow at low reynolds number due to streamwise body force". Journal of Fluid Mechanics, Vol. 809, Pp. 31–71, 2016.

[2] J.H. Bae, J.Y. Yoo, H. Choi, and D.M. McEligot, "Effects of large density variation on strongly heated internal air flows". Physics of Fluids, Vol. 18, No. 7, Pp. 1–25, 2006.

[3] C. Friedemann, M. Mortensen, and J. Nossen, "Gas–liquid slug flow in a horizontal concentric annulus, a comparison of numerical simulations and experimental data". International journal of heat and fluid flow, Vol. 78, Pp. 1–14, 2019.

[4] D. Feldmann, C. Bauer, and C. Wagner, "Computational domain length and Reynolds number effects on large-scale coherent motions in turbulent pipe flow". Journal of Turbulence, Vol. 19, No. 3, Pp. 274–295, 2018.

[5] I. Bicalho, D. dos Santos, C. Ataide, and C. Duarte, "Fluid-dynamic behavior of flow in partially obstructed concentric and eccentric annuli with orbital motion". International Journal of Heat and Fluid Flow, Vol. 137, Pp. 202–213, 2016.

[6] A. Fabregat, J. Pallares, A. Vernet, I. Cuesta, J.A. Ferre, and F.X. Grau, "Identification of near-wall flow structures producing large wall transfer rates in turbulent mixed convection channel flow", Computers and Fluids, Vol. 39**,** No. 1, Pp. 15–24, 2010.

[7] J. Dirker, and J.P. Meyer, "Topology optimization for an internal heat conduction cooling scheme in a square domain for high heat flux applications", Journal of Heat Transfer, Vol. 135, No. 11, Pp. 111010, 2013.

[8] G. Marck, M. Nemer, and J.L. Harion, "Topology optimization of heat and mass transfer problems: laminar flow". Numerical Heat Transf. Part B, Vol. 63, No. 6, Pp. 508–539, 2013.

[9] F.H. Burger, J. Dirker, and J.P. Meyer, "Three-dimensional conductive heat transfer topology optimization in a cubic domain for the volume-to-surface problem". International Journal of Heat and Mass Transfer, Vol. 67, Pp. 214–224, 2013.

[10] V. Subramaniam, T. Dbouk, and J.L. Harion, "Topology optimization of conductive heat transfer devices: an experimental investigation", Applied Thermal Engineering, Vol. 131, Pp. 390–411, 2018.

[11] O. Sigmund, and K. Maute, "Topology optimization approaches", Structural and Multidisciplinary Optimization, Vol. 48, No. 6, Pp. 1031–1055, 2013.

[12] T. Matsumori, T. Kondoh, A. Kawamoto, and T. Nomura, "Topology optimization for fluid–thermal interaction problems under constant input power", Structural and Multidisciplinary Optimization, Vol. 47, No. 4, Pp. 571–581, 2013.

[13] S. Zeng, B. Kanargi, and P.S. Lee, "Experimental and numerical investigation of a mini channel forced air heat sink designed by topology optimization", International Journal of Heat and Mass Transfer, Vol. 121, Pp. 663–679, 2018.

[14] R. Issa, and M. Kempf, "Simulation of slug flow in horizontal and nearly horizontal pipes with the two fluid-model", International Journal of Multiphase Flow, Vol. 29, Pp. 69–95, 2003.

[15] H. Al-Madhhachi, and G. Min, "Key factors affecting the water production in a thermoelectric distillation system", Energy Conversion and Management, Vol. 165, Pp. 459–464, 2018.

[16] E.I. Housz, G. Ooms, R. Henkes, M. Porquie, A. Kidess, and R. Radhakrishnan, "A comparison between numerical predictions and experimental results for horizontal core-annular flow with a turbulent annulus", International Journal of Multiphase Flow, Vol. 95, Pp. 271–282, 2017.

[17] R. Tunstall, and A. Skillen, "Large eddy simulation of a t-junction with upstream elbow: the role of dean vortices in thermal fatigue", Applied Thermal Engineering, Vol. 107, Pp. 672–680, 2016.

[18] L. Xing, H. Yeung, J. Shen, and Y. Cao, "Numerical study on mitigating severe slugging in pipeline/riser system with wavy pipe ", International Journal of Multiphase Flow, Vol. 53, Pp. 1–10, 2013.

[19] M. Xiang, S. Cheung, G. Yeoh, W. Zhang, and J. Tu, "On the numerical study of bubbly flow created by ventilated cavity in vertical pipe", International Journal of Multiphase Flow, Vol. 37, Pp. 756–768, 2011.

[20] A. Hossain, J. Naser, and M. Imteaz, "Cfd investigation of particle deposition in a horizontal looped turbulent pipe flow", Environmental Modeling Assessment, Vol. 16, Pp. 359–367, 2011.

[21] S. Chung, and H. Sung, "Large-eddy simulation of turbulent flow in a concentric annulus with rotation of an inner cylinder", International Journal of Heat and Fluid Flow, Vol. 26, Pp. 191–203, 2005.

[22] M. Escudier, I. Gouldson, P. Oliveira, and F. Pinho, "Effects of inner cylinder rotation on laminar flow of a Newtonian liquid through an eccentric annulus", International Journal of Heat and Fluid Flow, Vol. 21, Pp. 92–103, 2000.

[23] M. Hadžiabdić, K. Hanjalić, and R. Mullyadzhanov, "Les of turbulent flow in a concentric annulus with rotating outer wall", International Journal of Heat and Fluid Flow, Vol. 43, Pp. 74–84, 2013.

[24] H. Noh, and S. Kim, "Numerical simulation of pulsating heat pipes: Parametric investigation and thermal optimization", Energy Conversion and Management, Vol. 203, Pp. 112237, 2020.

[25] H. Al-Madhhachi and G. Min, "Effective use of thermal energy at both hot and cold side of thermoelectric module for developing efficient thermoelectric water distillation system", Energy Conversion and Management, Vol. 133, Pp. 9–14, 2017.

[26] R. Rabiee, B. Rajabloo, M. Desilets, and P. Proulx, "Heat transfer analysis of boiling and condensation inside a horizontal heat pipe", International Journal of Heat and Mass Transfer, Vol. 139, Pp. 526-536, 2019.

[27] M.E. Arıcı and M.E. Kaya, "Conjugate heat transfer analysis for laminar flow in pipes having a step change in boundary conditions and wall conductivity", Journal of Mechanical Engineering Science, Vol. 122, Pp. 58–63, 2017.

[28] R. Rzehak, and S. Kriebitzsch, "Multiphase cfd-simulation of bubbly pipe flow: a code comparison", International Journal of Multiphase Flow, Vol. 68, Pp. 135–152, 2015.