Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin

e-ISSN	: 2549-1999	No.	: 6	Month	: December
p-ISSN	: 1693-5462	Volume	: 22	Year	: 2024

Article Processing Dates: Received on 2023-09-27, Reviewed on 2024-10-27, Revised on 2024-11-13, Accepted on 2024-11-17 and Available online on 2024-12-30

Pre-design of bottom ash cooling using CFD simulation: a case study of the coal generator power plant in the PT. BEST Tanjung Enim of South Sumatera

Ozkar F. Homzah[1,*], Ella Sundari[1], Firdaus[1], Muhammad Irfan Dzaky[1], Fatahul Arifin[1], Damar Adji Prayoga[2], Kabul[3]

[1]Mechanical Engineering Department, Politeknik Negeri Sriwijaya, Palembang, 30128, Indonesia

[2]Applied Undergraduate Student, Mechanical Engineering Department, Politeknik Negeri Sriwijaya, Palembang, 30128, Indonesia

[3]PT. Bukit Energi Service Terpadu (BEST), Tanjung Enim, 31712, Indonesia

*Corresponding author: ozkarhomzah@polsri.ac.id

Abstract

The manual removal of bottom ash from the boiler is a hazardous activity due to the extremely high temperatures involved, ranging from 700 to 800°C. The objective of this research is to develop a cooling device to facilitate the bottom ash removal process for workers at PT. BEST Tanjung Enim and to measure the reduction in ash temperature achieved by cooling device. The design was created using computer aided design software and Computational Fluid Dynamics (CFD) simulations. Subsequently, a hypothesis is proposed, defining the bottom ash material as Silicon Dioxide (SiO_2) and equalizing the density of the bottom ash. To ascertain the temperature drop, variations in the screw conveyor's rotation speed were tested at 40, 33, and 12 rpm, paired with cooling water velocities of 0.5, 0.75, and 1 m/s respectively. The findings of study indicated that a screw conveyor speed of 12 rpm combined with a cooling water velocity of 0.5 m/s yielded the lowest bottom ash temperature, reaching 402°C, thus significantly reducing the need for manual handling when the ash temperature remains at 800°C. Further study should explore the application of Discrete Element Method (DEM) simulations for modelling bulk material behaviour and the integration of additional cooling media to enhance system performance.

Keywords:

Design, bottom ash, cooling device, CFD, performance.

1 Introduction

A Circulating Fluidized Bed (CFB) boiler is a specific type of boiler that employs a bed material as a medium for heat transfer to coal, thereby exhibiting distinctive characteristics [1]. This technology is well known in the power generation industry both in terms of economics and the resulting exhaust emissions [2]. The CFB boiler uses coal as the main feed in the combustion system. This coal combustion process produces waste as bottom ash and fly ash. Bottom ash is ash formed from the combustion process in the furnace in the form of solids that are not carried away by flue gas and settle at the bottom of the boiler [3]. The CFB boiler's operation and production of bottom ash result in increased pressure within the wind box, which in turn affects the efficiency and safety of operators in regulating steam pressure and draining bottom ash. The latter process at PT Bukit Energi Servis Terpadu (BEST), the operator, is still conducted manually.

Based on the data obtained from field observations, the bottom ash there has a temperature of about 700 - 800°C and based on the bottom ash temperature reference of 850°C [4], with the high temperature, the worker carries the bottom ash manually, directly carrying the bottom ash to the storage. Therefore, a tool is needed to help workers carry out bottom ash drain activities. The function of the tool is to reduce the temperature and produce dry bottom ash. In this research, testing is being carried out using Computational Fluid Dynamics (CFD) simulation.

CFD is the study of how to predict fluid flow, heat transfer, and chemical reactions by solving the numerical equations of fluid dynamics [5],[6]. A collection of numerical modeling methods called CFD is used to simulate and test the behavior of single-phase and multi-phase fluid flows coupled with mass transfer, heat transfer, and chemical processes [7]. Many studies have used CFD to determine heat transfer [8],[9],[10]. In the heat transfer procedures, the outcomes of numerical calculations are employed to determine the best results [11],[12],[13]. Therefore, researchers used computational fluid dynamics to determine the temperature drop of bottom ash. However, due to the inability of computer devices to simulate bulk material, the objective of this study is to replace bottom ash material with SiO_2 as a solid material, with a density value similar to that of bottom ash [14]. The study aims to use CFD ANSYS simulation as a first step of pre-design for a bottom ash cooling device. Furthermore, the objective of this research is to ascertain the bottom ash temperature as a dependent variable toward the screw conveyor rotation speed and cooling water speed such as the independent variables.

2 Research Methods

This study used Autodesk Inventor to design the cooling device and Ansys Fluent to simulate the screw speed and water cooling. The screw conveyor rotation speed is 40, 33, and 12 Rpm, and then at each screw conveyor rotation speed, there is a variation in cooling water speed of 0.5, 0.75, and 1 m/s. The mixing of Silicon Dioxide (SiO_2) in natural latex, specifically focusing on a high silica content regime [15] found that the scaling criterion was consistently applied to the mixing of high silica natural latex, with no deviation exceeding 10%. Therefore, the study replaced the bottom ash material with SiO_2 as a non-Newtonian fluid and equalized the density of the bottom ash. CFD simulation using SiO_2 as the majority chemical composition which is around 50% of bottom ash is illustrated in Table 1.

Fluent software is used in the simulation to solve partial differential equations governing several types of flow. Control volumes are used, where each node corresponds to a computational domain so that the partial differential equations are integrated into each limited volume [16]. This simulation uses the energy and k-omega models, a form of Reynolds-Averaged Navier-Stokes (RANS) model that is frequently utilized in CFD is the k-omega model [17], uses materials in Table 2, and uses boundary conditions based on the variations previously described.

The CFD simulation process is generally divided into three steps: pre-processing, processing, and post-processing [18]. The study had four steps of the research work. In the first stage, we first model the form of the tool and the fluid flow that will exist in solid form. The next step is meshing the geometry that has been made. The second stage is to determine the simulation model that will be carried out or determine what phenomena will exist in the simulation. The third stage is to determine the material to be used, both the material of the tool and the fluid used, and determine the conditions or parameters of each part of the geometry, such as the inlet and outlet. The last stage is to perform the simulation stage or generated simulation. Then, the fifth stage is to determine whether the results have achieved what is desired. If the results are correct, a visual representation can be captured; otherwise, the process must return to stages 1 to 4 to identify the parts requiring modification or improvement.

Table 1. Compositions of bottom ash [14]

Power Plant Station/ Chemical Composition (%)	Spanish Power Plant	Guru Hargobind Thermal Power Plant Bathinda, India	TNB Electric Power Plant Perak, Malaysia	Seocheon Coal-Fired Thermal Power Plant, South Korea
SiO_2	52.30	56.44	54.80	44.2
Al_2O_3	25.14	29.24	28.50	31.5
Fe_2O_3	9.23	8.44	8.49	8.9
CaO	2.37	0.75	4.20	2.0
MgO	1.84	0.40	0.35	2.6
Na_2O	0.66	0.09	0.08	-
K_2O	3.72	1.29	0.45	-
TiO_2	1.45	3.36	2.71	2.4
P_2O_5	0.25	-	0.28	-
SO_3	0.03	0.24	-	-
LOl	1.07	0.89	2.46	-

Fig. 1 is the mesh used in the geometry, and polyhedral mesh is used in geometry. The polyhedral mesh type creates good meshing (time and model quality of computation) for simulations running inside the chamber [19]. In the CFD modeling of industrial processes, polyhedral meshes have previously been used (e.g. air convective heat transfer around cylindrical objects), and have shown better performance (cost of computation and simulation precision) [20]. With a total number of cells of 428734, an average skewness value of 0.035, and a maximum of 0.59, it is already in the good category.

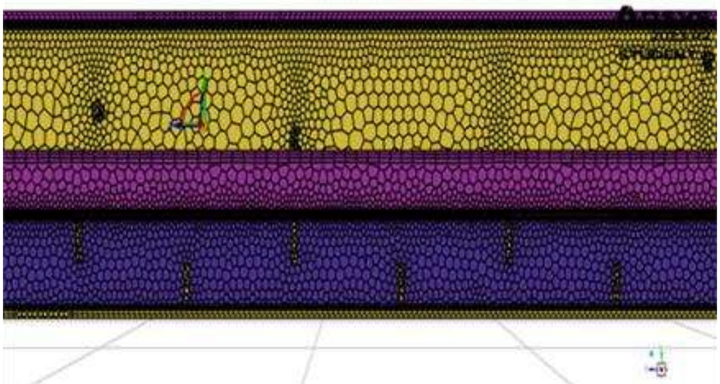


Fig. 1. The result of polyhedral meshing used.

The screw conveyor length, diameter, and screw pitch are 15240 mm, 508 mm, and 508 mm, respectively in Fig. 2 within a two and three-dimensional design of the bottom ash cooler. This study also used the tool's geometry was divided into real condition simulations such as the symmetry method.

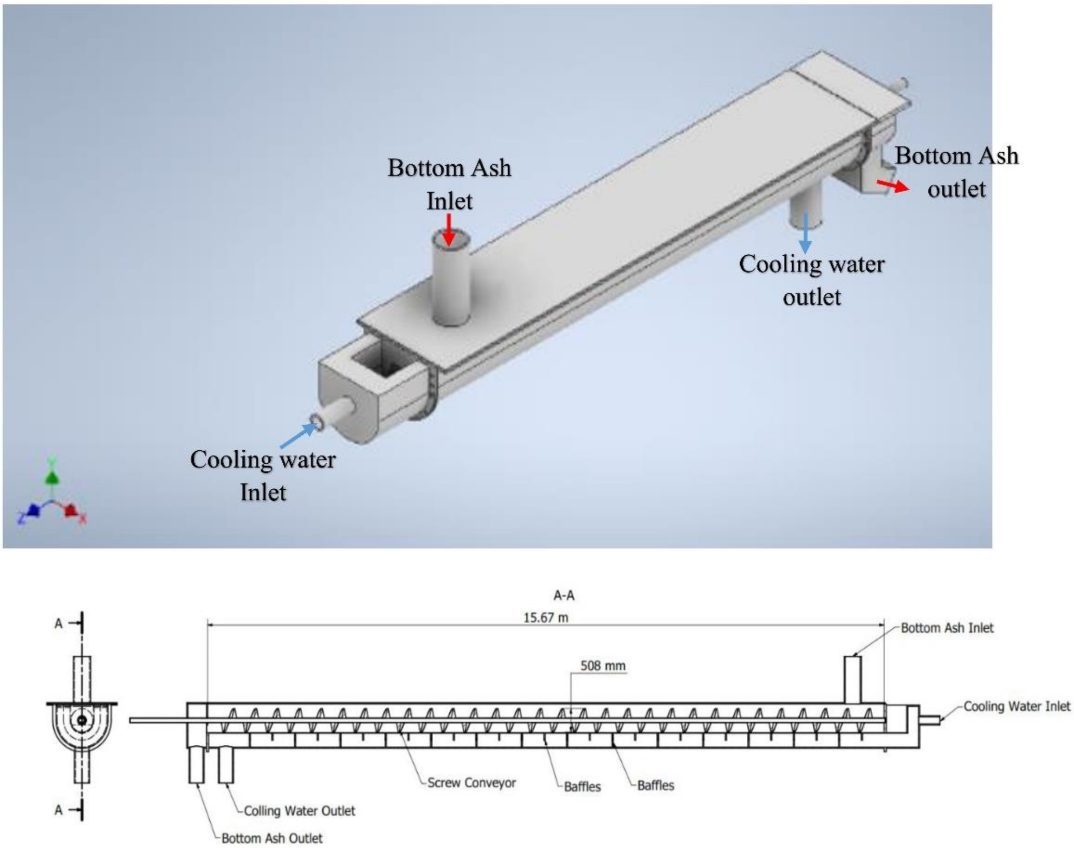


Fig. 2. Bottom ash cooling device design.

2.1 Tool and Materials

The tools employed for designing and simulating the cooling device consist of a laptop, which serves as a platform for conducting tests, and two software applications, namely Autodesk Inventor and Ansys. Additionally, the materials utilized in this cooling device's design are outlined in Table 2.

Table 2. Material properties

Materials	Parameter	Range
Bottom ash	Temperature (°C)	700-800
	Quantity (Ton/ hour)	±20
Silicon dioxide	Density (kg/m^3)	2648
	Spesific heat (J/Kg.K)	680
	Thermal conductivities (W/m.K)	1.3
Water	Density (kg/m^3)	998.2
	Spesific heat (J/Kg.K)	4182
	Thermal conductivities (W/m.K)	0.6
304 stainless steel	Density (kg/m^3)	8000
	Spesific heat (J/Kg.K)	500
	Thermal conductivities (W/m.K)	16.2

2.2 Mathematical Model

The governing equation is Eq. 1 [21] for a momentum equation.

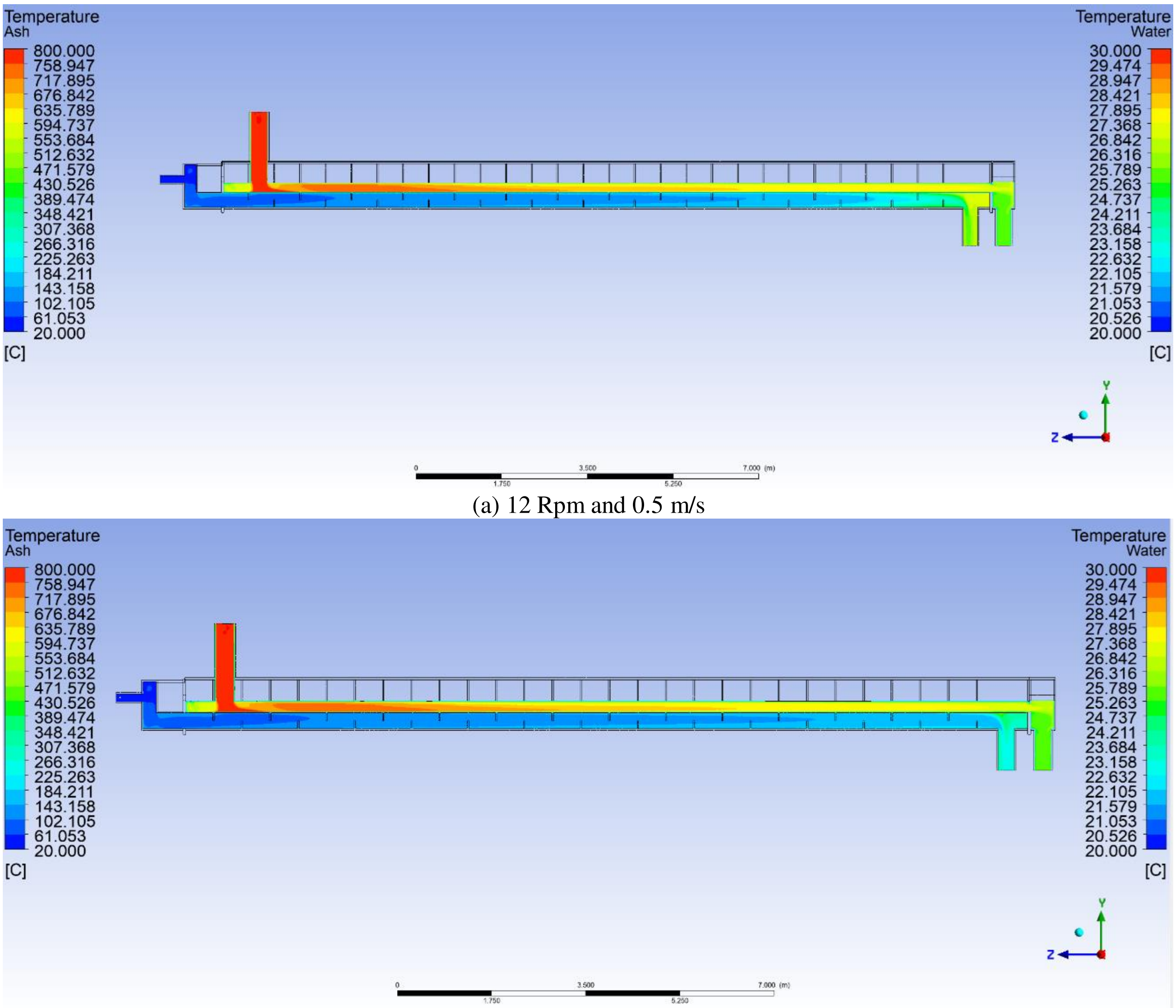
$$v_j \frac{\partial v_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_i}\left[(v + vi)\frac{\partial vi}{\partial xj}\right] \qquad (1)$$

Description:
ρ = Density
υ_i = Velocity component in the x_i coordinate direction
v = Kinematic viscosi

3 Results and Discussion

This tool is simulated with the help of Ansys Fluent software by varying the screw conveyor rotation speed and water speed, which we will know from the color difference in the simulation results display. In this simulation, the tool's geometry is divided into the symmetry assumption to economize the mesh generated due to mesh limitations such as viscous k-omega, turbulent intensity of only 5%, and diameter hydraulic of 350 mm. Some studies use the symmetry assumption [22], [23], [24].

Fig. 3 is the graphical result of the screw rotation speed of 12 Rpm and 33 Rpm respectively for water velocity is 0.5 m/s, which shows that at the 126th iteration, the line is straight and then converges at the 400th iteration. The results were confirmed completely, the fact that the graph line is even or straight indicates that the temperature distribution is in an equilibrium state. Based on the results of the three screw speeds, it is known that the 12 Rpm speed produces the lowest ash temperature of 402ºC and water temperature cooling with natural is to 6.5ºC.



(a) 12 Rpm and 0.5 m/s

(b) 33 Rpm and 0.5 m/s

Fig. 3. CFD simulation for bottom ash temperature results.

According to Fig. 4 of the simulation findings, the screw conveyor's 40 Rpm rotation speed and cooling water velocity for 0.5m/s and 0.75 m/s produces the lowest bottom ash outlet of 450ºC. The screw conveyor's 33 Rpm rotation speed and cooling water velocity 0.5 m/s produce the lowest bottom ash outlet of 450ºC. The screw conveyor's 12 Rpm rotation speed and velocity of cooling water 0.5 m/s produce the lowest bottom ash outlet of 402ºC.

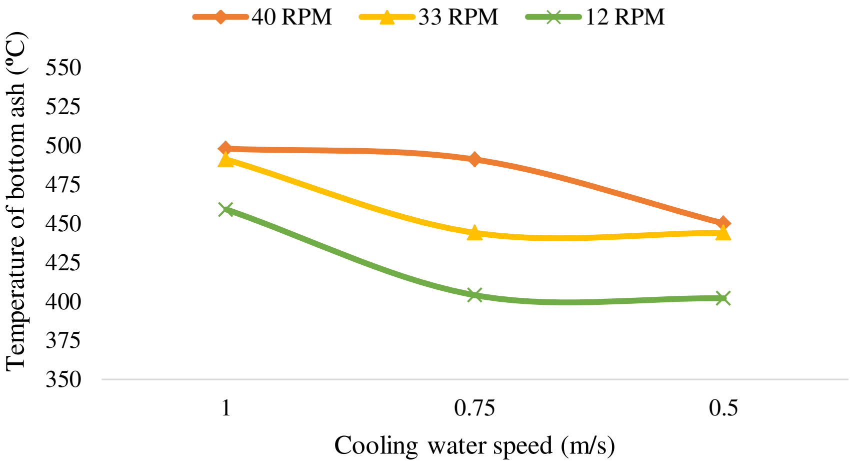


Fig. 4. Graph of simulation results that have been carried out.

4 Conclusion

This study simulated the temperature reduction of bottom ash using water cooling in Ansys Fluent. The results indicate that the maximum temperature drop achieved was 402°C, obtained with a screw conveyor rotation speed of 12 rpm and a cooling water velocity of 0.5 m/s. Among the three cooling water velocities tested, 0.5 m/s proved to be the most effective. Based on this study, the reduction in bottom ash temperature to 402°C allows PT BEST operators to handle bottom ash without the risk of exposure to its initial high temperature of 800°C during transportation to temporary storage. However, the temperature reduction cannot yet be considered optimal, as the simulation was conducted with an assumption that approximates the bottom ash material as silicon dioxide. Further studies should incorporate Discrete Element Method (DEM) simulations to model the behaviour of bulk materials more accurately and explore its integration with Ansys Fluent. Additionally, other cooling media could be introduced to achieve faster cooling and lower bottom ash temperatures, improving the efficiency of the cooling process.

Acknowledgment

This work is financially supported by Politeknik Negeri Sriwijaya through Penelitian Kerjasama Dosen dan Mahasiswa with the year 2023.

References

[1] C. Erwald Mataufani, A. Sidiq, and J. Arifin, “Analisis Pengaruh Kalori Batubara Dan Excess Air Pada Boiler Cfb Dengan Metode CFD (ANSYS)”, JUTIN, vol. 6, no. 1, pp. 27–40, Apr, 2023.

[2] I. Yuliyani and M. A. M, “Analisis Sistem Ruang Bakar Boiler Jenis Fluidized Bed Combustion Untuk Pltu Kapasitas 8 Mw,” *Jurnal Teknik Energi*, vol. 9, no.1, pp. 1–8 Nov, 2019. doi:10.35313/energi.v9i1.1638

[3] H. Winarno, D. Muhammad, R. Ashyar, and Y. Gusti Wibowo, “Pemanfaatan Limbah Fly Ash Dan Bottom Ash Dari Pltu Sumsel-5 Sebagai Bahan Utama Pembuatan Paving Block,” *Jurnal Teknika*, vol. 11, no. 1, 2019. doi:10.30736/jt.v11i1.288

[4] S. V Thite and S. B. Ingole, “Modeling Of Heat Transfer In Fluidized Ash Cooler,” *International Journal of Research in Advent Technology*, vol. 2, no. 3, pp. 69–74 March, 2014.

[5] M. B. N. Royyana, U. Budiarto, and G. Rindo, "Analisa Bentuk Oscillating Water Column Untuk Pemanfaatan Gelombang Laut Sebagai Sumber Energi Terbarukan Dengan Metode Computational Fluid DYNAMIC (CFD)," *Jurnal Teknik Perkapalan*, vol. 3, no. 1, Feb. 2015.

[6] F. Tuakia, “Dasar-Dasar CFD Menggunakan Fluent,” 1st ed., Bandung: Informatika Bandung, 2008.

[7] B. Chezeau and C. Vial, “Modeling And Simulation Of The Biohydrogen Production Processes,” Elsevier B.V., 2019. doi:10.1016/B978-0-444-64203-5.00019-8.

[8] K. Nakaso, H. Mitani, and J. Fukai, “Convection Heat Transfer In A Shell And Tube Heat Exchanger Using Sheet fins For Effective Utilization Of Energy,” *International Journal of Heat and Mass Transfer*. Vol 82, pp 581–587. 2015, doi: 10.1016/j.ijheatmasstransfer.2014.11.033.

[9] Y. Wang, X. Gu, Z. Jin, K. Wang, “CHARACTERISTICS OF HEAT TRANSFER FOR TUBE BANKS IN Crossflow AND ITS RELATION WITH THAT IN SHELL-AND-TUBE HEAT EXCHANGERS,” *International Journal of Heat and Mass Transfer*. Vol 93, pp 584–594. 2016, doi: 10.1016/j.ijheatmasstransfer.2015.10.018

[10] S. Araavind, A.S. Athreya, “CFD Analysis Of Shell And Tube Heat Exchanger For Pre-Heating Of Biodiesel,” Int. Adv. Res. Sci. Eng. Vol 6, 687–693. Jul 2017.

[11] W.G. Osley, P. Droegemueller, P. Ellerby, “CFD Investigation Of Heat Transfer And flow Patterns In Tube Side Laminar flow And The Potential For Enhancement,” Chem. Eng. Trans. Vol 35, 997–1002. 2013, doi: 10.3303/CET1335166

[12] O. Karar, S. Emani, S.M. Gounder, M.M. Myo Thant, H. Mukhtar, M. Sharifpur, M. Sadeghzadeh, “Experimental And Numerical Investigation On Convective Heat Transfer In Actively Heated Bundle-Pipe,” Eng. Appl. Comput. Fluid Mech. Vol 15 No 1, 848 –864, Apr 2021, doi:10.1080/19942060.2021.1920466.

[13] S. Rana, M. Zunaid, R. Kumar, “CFD Approach For The Enhancement Of Thermal Energy Storage In Phase Change Material Charged Heat Exchanger,” Case Stud. Therm. Eng. Vol 33, 101921, 2022, doi: 10.1016/j.csite.2022.101921

[14] N.I.R.R. Hannan, S. Shahidan, N. Ali, and M.Z. Maarof, "A Comprehensive Review On The Properties Of Coal Bottom Ash In Concrete As Sound Absorption Material," *MATEC Web of Conferences*, Vol 103, p. 01005, Apr 2017, doi: 10.1051/matecconf/201710301005.

[15] Ekaroek Phumnok, Phonsan Saetiao, Panut Bumphenkiattikul, Sukrittira Rattanawilai, Parinya Khongprom, "CFD simulation of silica dispersion/natural rubber latex mixing for high silica content rubber composite production", RSC Advances, vol.14, issue.18, 2024, pp. 12612-12623, https://doi.org/10.1039/d4ra01348d

[16] M. R. Safaei, B. Rahmanian, and M. Goodarzi, “Numerical Study Of Laminar Mixed Convection Heat Transfer Of Power-Law Non-Newtonian Fluids In Square Enclosures By Finite Volume Method,” *International Journal of the Physical Sciences* Vol. 6, pp. 7456 - 7470, Des 2011, doi: 10.5897/ijps11.1092.

[17] A. Putra Maulana, Indrayani, F. Arifin, “CFD Analysis In Investigating The Impact Of Turbine Blade Number On The Performance Of Hydro Turbine,” *Jurnal Polimesin,* vol. 21 no. 3, 2023, doi: https://dx.doi.org/10.30811/jpl.v21i3.3671.

[18] A. Rosidi, Giarno, D. Haryanto, N. Adji Wahanani, Y. Dwi Setyo Pambudi, M. Hadi Kusuma, “The Simulation Of Heat Transfers And Flow Characterization On Wickless Loop Heat Pipe,” *Jurnal Polimesin,* vol. 20 no.1, 2022, doi: https://dx.doi.org/10.30811/jpl.v20i1.2497.

[19] H. Chen, X. Zhou, Z. Feng, and S-J. Cao, “Application Of Polyhedral Meshing Strategy In Indoor Environment Simulation: Model Accuracy And Computing Time,” *Indoor and Built Environment*, Vol 31 No 3, pp 719-731, 2021 doi: 10.1177/1420326X211027620.

[20] G. Balafas, “Polyhedral Mesh Generation For Cfd-Analysis Of Complex Structures,” Master Thesis, Technical University of Munich, Munich, 2014.

[21] D. Kurmanova, N. Jaichibekov, A. Karpenko, and K. Volkov, “Modelling And Simulation Of Heat Exchanger With Strong Dependence Of Oil Viscosity On Temperature,” Fluids, vol. 8, no. 3, p. 95, Mar. 2023, doi: 10.3390/fluids8030095.

[22] I. G. Garcia, “Enhancement Of Heat Transfer Using Nanofluids: Application To An Automotive Radiator”, Theses. Chemical Engineering, Universitat de Barcelona, Barcelona, 2019.

[23] L. Hill, “A Computational Fluid Dynamic, Experimental And Empirical Analysis Of University Of Plymouth’s Brunel Laboratories Shell And Tube Heat Exchanger”, *The Plymouth Student Scientist*, Vol 15 No 1, pp. 69-103. 2022.

[24] M. Khoshvaght Aliabadi, M. Gholam Samani, F. Hormozi, and A. Haghighi Asl, “3D-CFD Simulation And Neural Network Model For The J And F Factors Of The Wavy Fin-And-Flat Tube Heat Exchangers,” Brazilian Journal Of Chemical Engineering, Vol. 28, No. 03, pp. 505 - 520, Jul - Sep 2011, doi: 10.1590/S0104-66322011000300016.