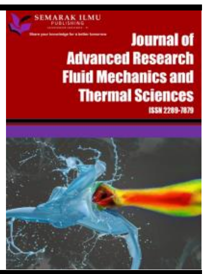
Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879




# Numerical Modelling and Analysis of Externally Blown Heated Pipes Applicable for Furnace

Muhkamad Wakid[1], Aan Yudianto[1,*], Agus Widyianto[1]

[1] Department of Automotive Engineering Education, Faculty of Engineering, Universitas Negeri Yogyakarta, 55281, Indonesia

**ARTICLE INFO**

***Article history:***
Received 18 April 2022
Received in revised form 22 August 2022
Accepted 5 September 2022
Available online 28 September 2022

***Keywords:***
Heat transfer; furnace; CFD; temperature; velocity; pressure

**ABSTRACT**

The heat transfer system of the pipe for furnace application was newly introduced and need to be analyzed. The study aims to numerically model and investigate the externally blown heated pipe for furnace. The furnace itself is the heat source for briquette production inside the oven. The investigation includes the analysis of temperature, air flow velocity and also pressure inside the pipes. The investigation employed the Computational Fluid Dynamics methodology for numerically analysis the intended results. Three cases were introduced in this study, having differences in the air intake velocity which was set to 5 m/s (case 1), 3 m/s (case 2) and 1 m/s (case 3). The main objective is to have the highest outlet temperature to be blown in the heating chamber. Case 1 results in the lowest resulting temperature which only 77 ℃, while case 3 yields in the highest output temperature more than 100 ℃. In terms of velocity, it is clear that the highest velocity intake affects also in the increasing value of the velocity inside the pipe. Case 1 results in maximum of 35 m/s air velocity. The pressure distribution also results in the similar trend. The case 1 yields in the maximum pressure of 650 Pa and case 3 produce the pressure nearly zero. However, case 3 results in the highest temperature for furnace to be blown in the heating chamber of the oven system.

## 1. Introduction

The emerging technology in heating systems for an oven is nowadays greatly improved by the emerging some applicable systems, such as protection system, application of PID algorithm, predictive thinking, and heating management systems [1]. The application of new technology in the heating system is majorly due to the essence of the heat transfer and efficiency of the system. However, not all the already ongoing production using furnace technology for heating system utilize the mentioned technology. The small to medium businesses keep conventional technology for a production system which mainly due to cost reasons, including furnace technology for oven briquette production [2–4], which is relevant to this work. Some others also combine the conventional one and add some optimization in terms of structure and thermal efficiency [5], control system [6], design

* *Corresponding author.*
*E-mail address: aan.yudianto@uny.ac.id*

https://doi.org/10.37934/arfmts.100.1.3043

and analysis [7–9], and energy-saving production planning [10]. The main results of previous research activity are to enhance the furnace performance in terms of energy efficiency. Therefore, the previous research indicates the importance of heat transfer analysis in a heating system, including a furnace.

The development of computer programs and the application of software to design and evaluate system performance have also been greatly enhanced. It is proven to be a cost-effective analysis for supporting nearly perfect system production. This technology includes the use of numerical modeling to design [11], analyze [12], investigate [13], and improve the furnace system performance [14]. The analysis is also not limited to the entire system itself but also to the subsystem for in detail explanation and investigation, such as the investigation of heat transfer in pipe [15], carrier [16], melting process [17], and other related works [18,19]. Moreover, the previous works also involve numerical analysis by means of Computational Fluid Dynamics for system analysis, not only related to heat transfer [20,21]. The mentioned past research successfully conducted computational research which was validated by experiment to ensure the capability of the numerical modelling that have been performed. Therefore, the use of computational modeling and simulation is one of the methods that have been widely employed for system analysis and evaluation for heat transfer related research and it is proved to have a realistic value.

This study focuses on the heat transfer analysis of heated pipes for furnace application. The equipment was introduced as the heat source for a unit of the oven for briquette manufacturing. The furnace comprises an independent chamber containing heat source which is heated by electric of manual stove using light petroleum gas. The fire from the stove is heating the unit of pipes above the stove. As an effect, the pipes undergo external heating on their surface. An external blower is attached on one side of the pipe for blowing fresh and clean air from the outside of the furnace, flowing through the heated pipe. As a result, the output air flowing through the pipe is heated and directed to the heating chamber, which is located on the other side of the furnace. This method is employed to ensure that only clean air is blown to the heating chamber since the external air source is isolated by the pipes from the heat source. An exhaust is attached at the top of the furnace to allow the exhaust gas to flow to the outside of the furnace. The schematic of the furnace can be observed in Figure 1 (a).

The mentioned previous studies have conducted numerous analyses about the system and subsystem related to the heating technology and application. However, the newly built furnace for briquette production was introduced, and the analysis of the newly built heating system for the furnace requires further analysis, and the literature related to the current work is still limited. The innovation of the heating chamber is to separate the main heating chamber where the heat source is located and the external air which is blown. The system allows clean air only, which is flown through the heating chamber. The study aims to numerically model and analyzes the temperature, pressure, and velocity of the heated pipes that is used for furnace application for oven heating system. The study impacts the more effective briquette production by supporting the numerical modeling and analysis of the resulting heat for the furnace system.

## 2. Methodology

### *2.1 Model Explanation*

A simplified model of the system was introduced to focus on the unit pipes with inlet and outlet parts, as it is depicted in Figure 1 (b). The location for the external blower is modeled as the air inlet direction with a certain air inlet velocity. The heated pipe is modeled as the same as the real condition in terms of the dimension and location, including the separation distance among the pipes. The outlet part of the furnace, allowing the clean heated air to flow to the heating chamber, was modeled by air outlet direction, similar to the inlet one. By using this simplification model, it is possible to observe the heat transfer from the heated pipe to the air flowing through the pipes.

A full-scale model was introduced to capture the heat transfer phenomenon in the system. There are ten pipes in the system having vertical and horizontal spacing distances, as it is depicted in Figure 2. The horizontal distance is 160 mm, and the vertical distance is 80 mm. The pipe itself has 70 mm in diameter, and the length of the pipe between the inlet and outlet location is 1,200 mm. The inlet and outlet parts were similar in shape, having a thickness of 100 mm and a height is 320 mm. The shape was chosen to simulate the direction of the fresh inlet air and the heated air at the outlet part of the system. The complete explanation of the model dimension is in Figure 2.

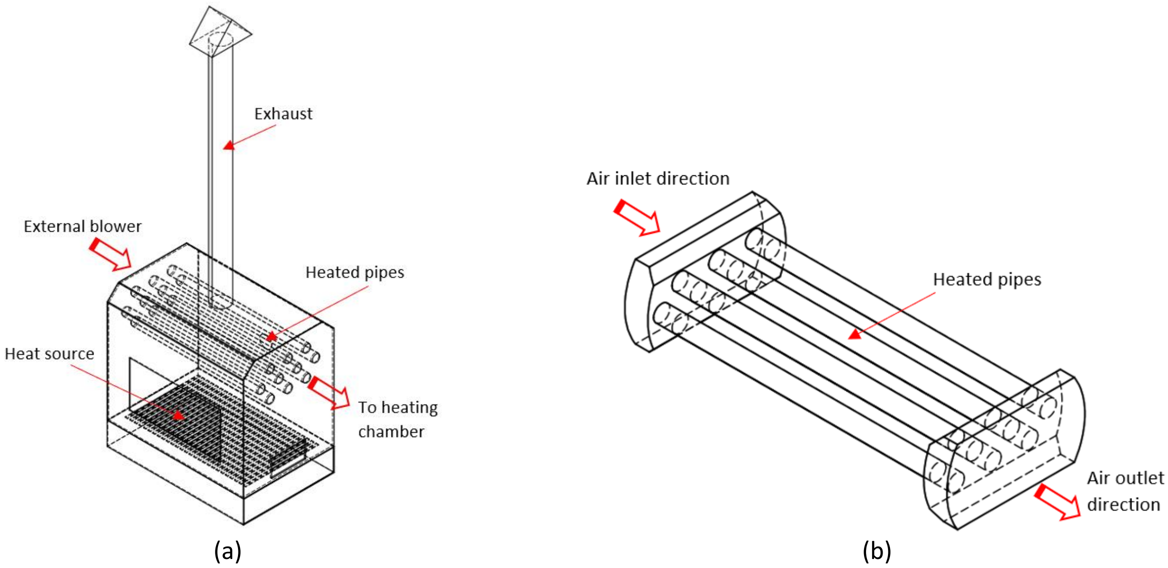



**Fig. 1.** Schematic picture of the furnace (a) and heated pipes model (b)

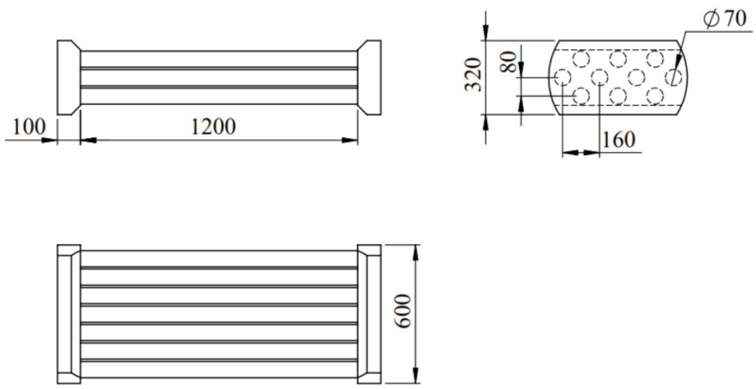



**Fig. 2.** Model dimension of the heated pipes (dimension in mm)

## *2.2 Mesh Generation*

The Computational Fluid Dynamics (CFD) simulation was preferred to study the heat transfer from the introduced system, and it has been widely used by other previous researchers [22–25]. The CFD simulation generated the mesh to numerically calculate the heat transfer in the system. The tetrahedron element was chosen. The maximum element size was 50 mm, with a maximum size of 100 mm. The growth rate setting was 1.2 to ensure the mesh transition between the smaller to the higher ones. Mesh quality checking was introduced with a skewness target of 0.9. Smoothing mesh techniques were performed with a medium level of smoothness. Since there is a movement of air inside the pipe, the inflation methodology was introduced at the surface of the pipes. There are three inflation layers applied with a growth rate of 1.2 and a maximum thickness of 5 mm. The smooth transition of the inflation layer was also applied. As a result, there are 135,261 nodes and 407,449 elements were generated as it is depicted in Figure 3. The mesh results is representative enough to simulate the behavior of the real condition of the model.

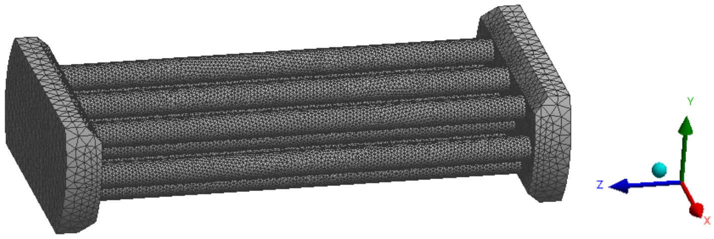


**Fig. 3.** Mesh results of the model

*2.3 Boundary Conditions*

Table 1 shows the detail of the boundary condition applied to the model. The face where external blowing is attached was set to the inlet face having air intake velocity, which varies between 5 m/s, 3 m/s, and 1 m/s depending on the case being investigated. The inlet face is the location in which the external blower is attached. The external blower has a velocity setting, which will be applied in the furnace system. The blower and exit wall were defined as the wall having a temperature for environmental temperature of 29.85 °C. It is assumed that the blower and exit wall were directly connected to the external room temperature. Instead, the pipe wall was the one that was heated by a heat source inside the furnace. It is assumed in this study that the wall temperature for the pipe is 400 °C. Finally, the outlet face was set to pressure outlet boundary setting to capture the resulting heat transfer through the system for the case of 5 m/s, 3 m/s and 1 m/s velocity inlet. Figure 4 explains the air-flowing condition and its direction from the inlet face to the outlet face through the heated pipe.

**Table 1**
Boundary condition setting of the simulation

| Face | Properties | Temperature |
|---|---|---|
| Inlet Face | Velocity inlet (5, 3, 1 m/s) | 29.85 |
| Pipe Wall | Wall | 400 |
| Blower and exit wall | Wall | 29.85 |
| Outlet Face | Pressure outlet | dependance |

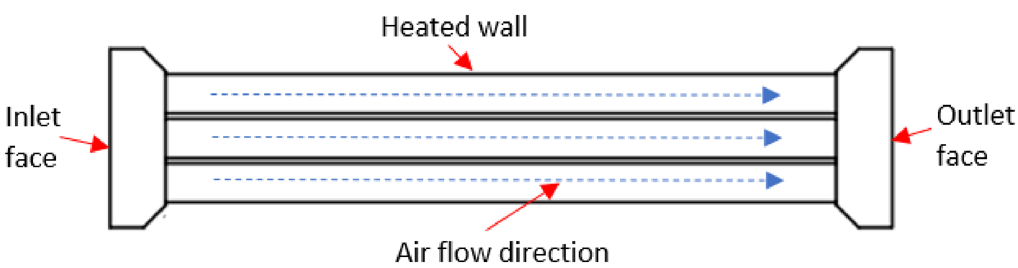



**Fig. 4.** Boundary condition

### *2.4 Material Properties*

Air and steel materials were involved in the simulation. The entire solid model was steel, having a density of 8,030 kg/m$^3$. The steel thermal conductivity is 16.27 w/m-k, and the specific heat is 502.48 j/kg-k. Instead, the standard material properties for the fluid are the air having a density of 1.225 kg/m$^3$, and thermal conductivity of 0.0242 w/m-k. The air-specific heat is 1006.43 j/kg-k. The viscosity of the air was set to constant. The other material properties for both steel and the air were also set to a constant value. The material properties involved were summarized in Tables 2 and 3.

**Table 2**
Steel material properties

| Properties | Value |
|---|---|
| Density | 8030 kg/m3 |
| Thermal Conductivity | 16.27 w/m-k |
| Specific Heat (Cp) | 502.48 j/kg-k |

**Table 3**
Air material properties

| Properties | Value |
|---|---|
| Density | 1.225 kg/m3 |
| Thermal Conductivity | 0.0242 w/m-k |
| Specific Heat (Cp) | 1006.43 j/kg-k |
| Viscosity | Constant |

## 3. Results
### *3.1 Resulting Temperature*

This section explains the resulting temperature at the middle plane of the pipes where four pipes are horizontally assembled. The study of heat transfer in pipes has been widely conducted in many other cases [26–28], and it results in similar results. The explanation focuses on the exit part of the pipes prior to the outlet face, as it is explained in Figure 5. The resulting heat transfer at the outlet part of the pipe was compared at three different inlet velocity settings: 5 m/s, 3 m/s, and 1 m/s. The increasing outlet temperature was observed at the outlet part of the pipe at three different inlet velocities. The minimum observed temperature is 30℃ since this temperature was set at inlet air temperature, indicating the blower flows the fresh air. When the air intake velocity is 5 m/s, the outlet temperature is approximately 77 ℃. When the air intake velocity is decreased, the average outlet velocity at the pipe is about 83℃. Instead, When the air intake velocity was applied at the lowest setting, which is 1 m/s, it occurs that the outlet temperature at the observed pipe of the pipe is the highest among the others, reaching about 103 ℃. It is also observed that the highest temperature is observed at the wall of the pipe since it is assumed the heat is transferred from the heat source of the stove in the furnace. Heat propagation is also affected by material [29]. The heat propagation is also observed more uniformly when the lowest inlet air velocity is applied in the system. This phenomenon occurs relatively equally at all outer parts of the pipe in the heating system and the results agree with previous research investigating the flow regime in a pipe [30].

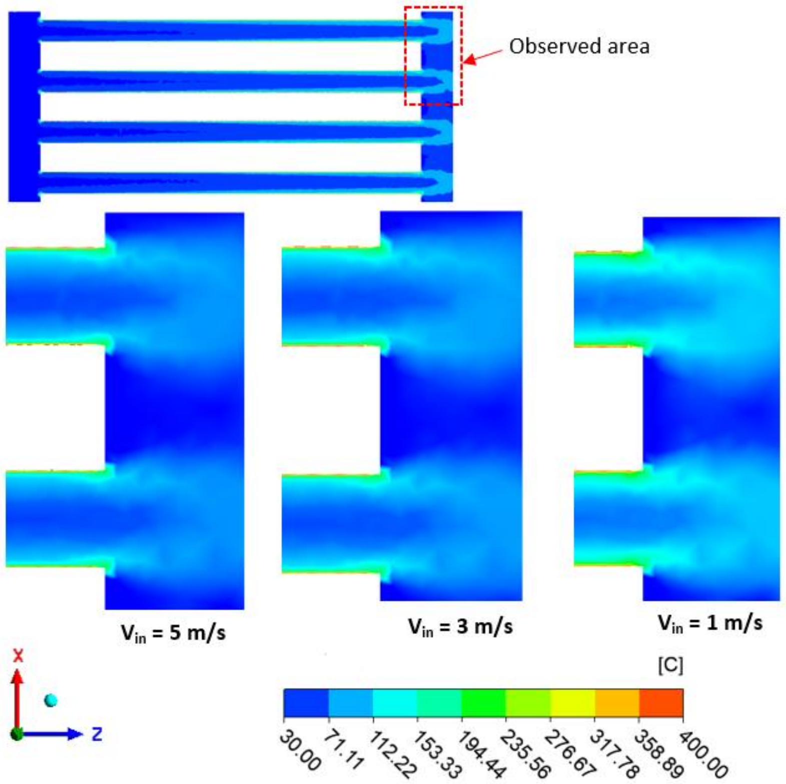



**Fig. 5.** Temperature portrayed at the exit face of the pipes

### *3.2 Pressure Distribution*

In this section, the observation of pressure distribution on the selected plane in the middle of the pipe is discussed. The pressure investigation becomes important as it has been observed by previous researchers for some other types of fluids as well [28, 31–33]. A relatively high-pressure distribution is observed when the system is blown with the air having the highest velocity inlet 5 m/s. However, when the air intake velocity was applied at 1 m/s, the pressure difference was observed to be relatively small. The entire phenomenon can be shown in Figure 6. The left part of the pipe is the inlet face, while the right part of the system is the outlet of the system. The highest pressure happens at the area of the inlet face, which is about 650 Pa. It happens when the air intake velocity is applied at 5 m/s. In this inlet velocity case, the pressure is getting decreased at the opening part of the pipe. The value drops to approximately -88 Pa before it increases again to the value of 20 Pa in the middle part of the pipe. Then, the value becomes neutral at 0 Pa at the outlet part of the pipe.

When the air intake velocity is 3 m/s, the highest pressure in the system was observed at only 125 Pa at the opening part of the pipes. In this case, the pressure drop is not really significant since the lowest pressure occurs is about 16 Pa in the middle part of the pipe. The pressure is going to zero again at the exit part of the pipe. However, at the inlet velocity 1 m/s, the pressure is relatively uniform inside the pipe. At the part of the inlet face, the highest pressure is only about 122 Pa. When the air is flown inside the pipe, the pressure is nearly zero along the pipe until it exits to the outlet face.

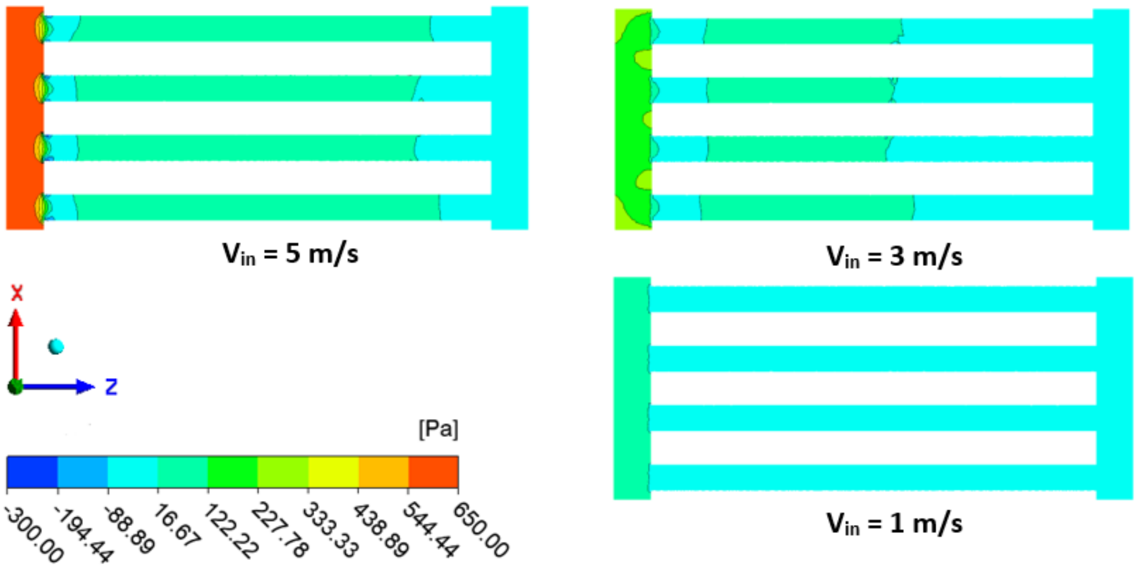



**Fig. 6.** Pressure distribution at the selected middle plane view at a different air inlet velocity

### *3.3 Velocity Propagation*

The velocity propagation of the selected plane at different inlet velocity conditions is described in Figure 7. The highest velocity inside the pipes is observed when the 5 m/s velocity inlet is applied. When the blower flows the fresh air at the inlet side of the system, the air velocity distribution varies inside the system. There are some locations that have nearly zero velocity both before the inlet pipe and between the outlet pipe. The highest velocity occurs in the area at the beginning of the pipe, which reaches about 35 m/s. This air velocity decreases as the airflow through the pipe until it exits the pipe. The outlet air velocity at the exit part of the pipe is about 27 m/s. The corresponding selected air intake velocity at 5 m/s produces the highest air velocity in the furnace system.

When the air intake velocity is set to 3 m/s, the air intake velocity is not as high as in the previous case. The average air velocity inside the pipe is relatively more uniform than it is compared to the previous one. The average value of the air velocity in the pipes is between 19 m/s to 27 m/s. A similar value is also observed at the exit part of the pipes, which corresponds to the exit air velocity when the air intake velocity is set to 3 m/s. Moreover, when the case of 1 m/s air intake velocity is applied at the inlet face, even lower air velocity also occurs in the pipes. The average air velocity in the pipe is merely between 3.89 m/s to 7.78 m/s. The velocity differences is evaluated as not too high as it is compared to the other two previous cases.

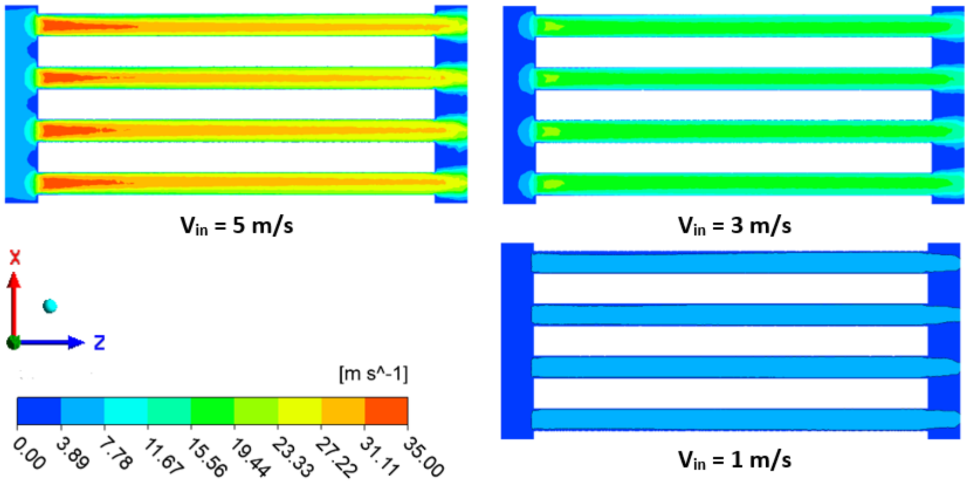



**Fig. 7.** Velocity at the selected middle plane view at a different air inlet velocity

### *3.4 Alteration of Air Temperature, Velocity, and Pressure*

This section discusses the results obtained from the observation of the measured line in the pipe. The selected line was chosen at one of the pipes at the center line, starting from the beginning of the pipe section close to the inlet face until the extended line after the pipe intersecting the outlet face of the heater system model. The description of measured line is described in Figure 8.

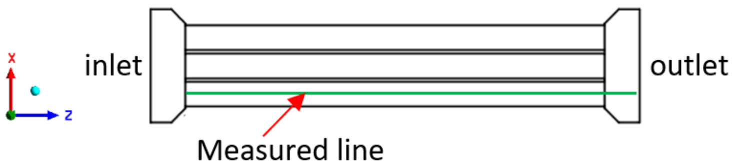



**Fig. 8.** The location of the measured line in the pipe

The temperature alters exponentially along the pipes in three different applied cases. Figure 9 compares the temperature propagation measured in the selected line when the inlet velocities 5 m/s, 3 m/s, and 1 m/s were applied. A relatively same temperature alteration has occurred in all three applied cases until the air flowed at 60 cm from the opening part of the pipe. After this location, the heat propagation changes differently in three cases applied. When case 1 is applied, the 5 m/s inlet velocity makes the temperature at the exit part of the pipe become about 77 ℃. Instead, when the air intake velocity of 3 m/s is set to the model, after the location at 60 cm from the opening part of the pipe, the temperature highly increases and exponentially rises until about 80 ℃ at the exit part of the pipe. Lastly, when 1 m/s of inlet velocity is applied to the model, the resulting temperature at the exit part of the pipe reaches more than 100℃. With the result obtained, it can be derived that the lower the air intake velocity yields the highest resulting air temperature at the outlet part of the pipe, giving the highest resulting temperature among others. Here, the types of fluids and the pressure flowing through the pipes also affect the heat transfer, as is also agreed with other research results [34–36].

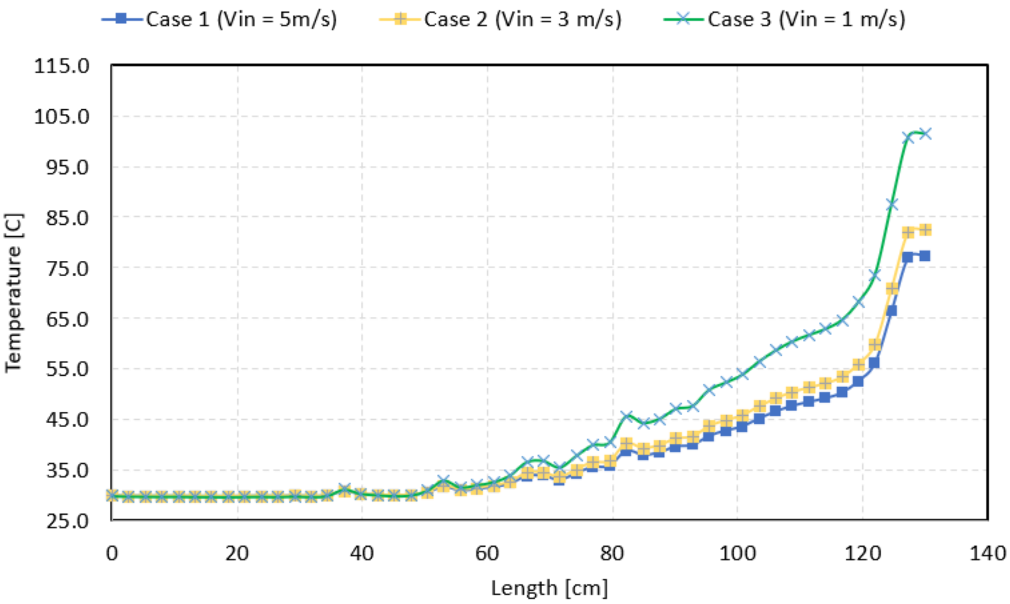



**Fig. 9.** Temperature propagation of three different cases at measured line

Figure 10 compares the velocity propagation at the selected measured line of three different cases applied. Case 1, when the inlet velocity is 5 m/s, results in the highest velocity propagation inside the pipe. The highest velocity when case 1 is applied is 33 m/s at the beginning of the pipe. The value is generally decreasing as the airflow through the pipe reaches the end of the pipe. The lowest velocity is at the exit part of the pipe, having a value of about 24 m/s. Moreover, when case 2 is applied in the simulation, the 3 m/s inlet velocity yields the highest velocity at the beginning of the pipe, reaching about 20 m/s. The value gradually decreases to approximately 14 m/s when the air exits the pipe. In contrast, the lowest air intake velocity results in also lowest velocity propagation in the pipe. The value of the air velocity is between 4.8 m/s to 6.5 m/s. The value is smaller at the exit part of the pipe. There is also an interesting phenomenon that occurs at the beginning of the pipe, when the air starts entering the pipe. The value jumps to a certain number when the air is blown to the pipe. This is due to sudden changes in the cross-section area between the inlet face and the pipe.

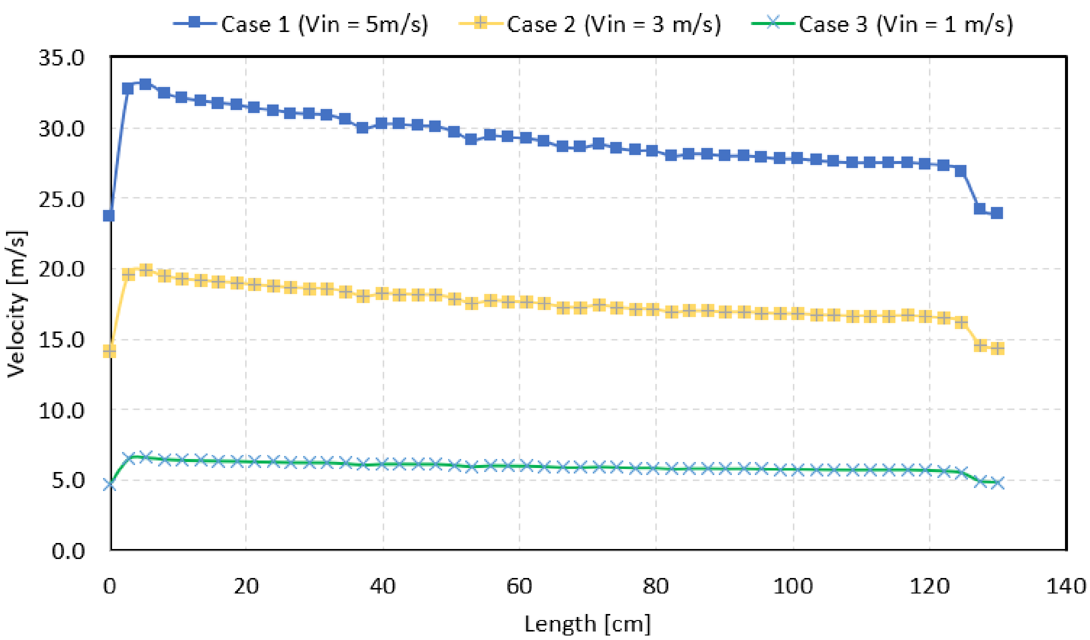



**Fig. 10.** Velocity propagation of three different cases at measured line

Figure 11 depicts the comparison among cases applied in this study in terms of pressure distribution and propagation inside the selected measured line in the pipe. The pressure drop was observed in the measured line in all three cases. High pressure was identified at the opening section of the pipe, and it dropped to below zero value for the case when 5 m/s inlet velocity was applied. The value is then increased to slightly more than 50 Pa at 20 cm to 60 cm of the pipe length. Then, the pressure gradually decreases to 0 Pa. A similar phenomenon also occurs when case 2 is applied. Pressure drops happen at the beginning of the pipe length. It increases and gradually decreases to the value of zero Pascal at the end of the pipe. However, the value is not as high as the one in Case 1. In contrast, the pressure propagation when only 1 m/s velocity inlet is applied yields a relatively similar value close to 0 Pa.

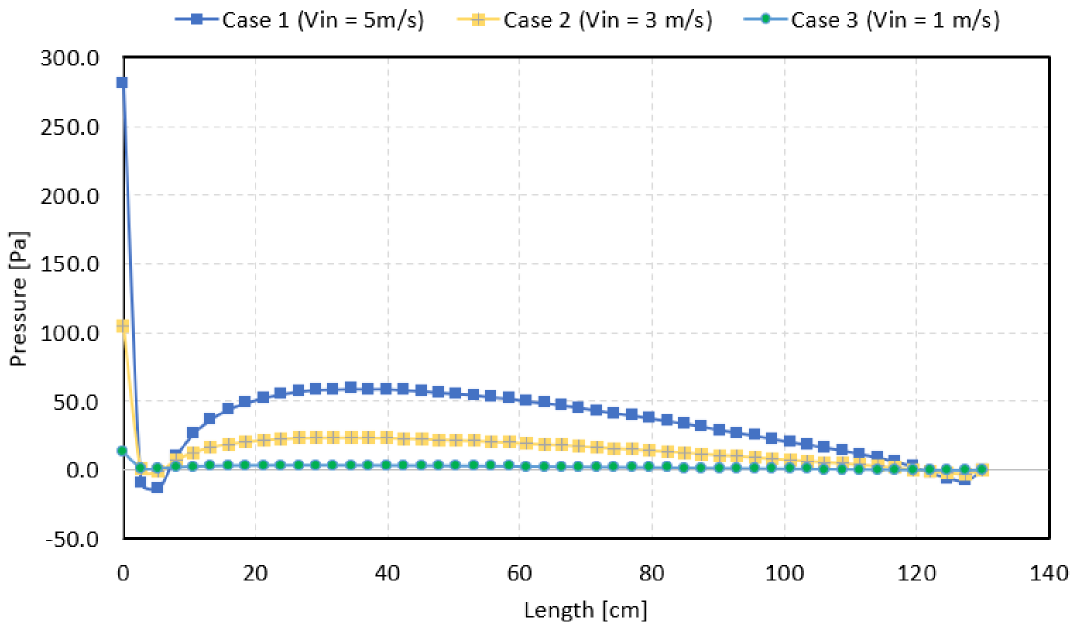



**Fig. 11.** Pressure propagation of three different cases at measured line

The point of view of temperature observation at the selected cross-sectional area is discussed in this section. Different cross-sections, namely cross-sections A, B, C, and D, were chosen to even discuss the thermal distribution and thermal transfer in the system. This becomes essential since the boundary layer in each pipe surface also affects the heat transfer [37]. Cross section A is chosen at the beginning point of the pipe close to the inlet face. Lastly, cross-section D is chosen at the exit part of the pipe nearby the outlet face. Two sections, which are B and C, were chosen equally in between. The graphical cross-sectional explanation is depicted in Figure 12. Moreover, Figure 13 compares the temperature distribution at the selected cross-sectional area at the velocity inlet 5 m/s as case 1, 3 m/s as case 2, and 1 m/s as case 3. The observation at cross-section A does not show any striking difference in the temperature resulting. However, the difference in the temperature distribution is clearly shown starting from the cross-section B to D. It is clear that the lower the velocity inlet applied, the wider the heat transfer from the pipe wall to the air flowing in the pipe. Moreover, the slower the velocity applied to the system, it results in the wider the thermal propagation at the end of the pipe (cross-section D).

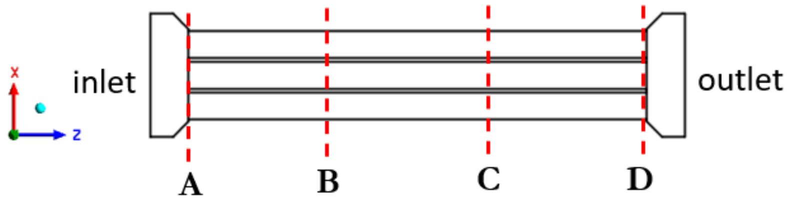


**Fig. 12.** Selected Cross section A to D

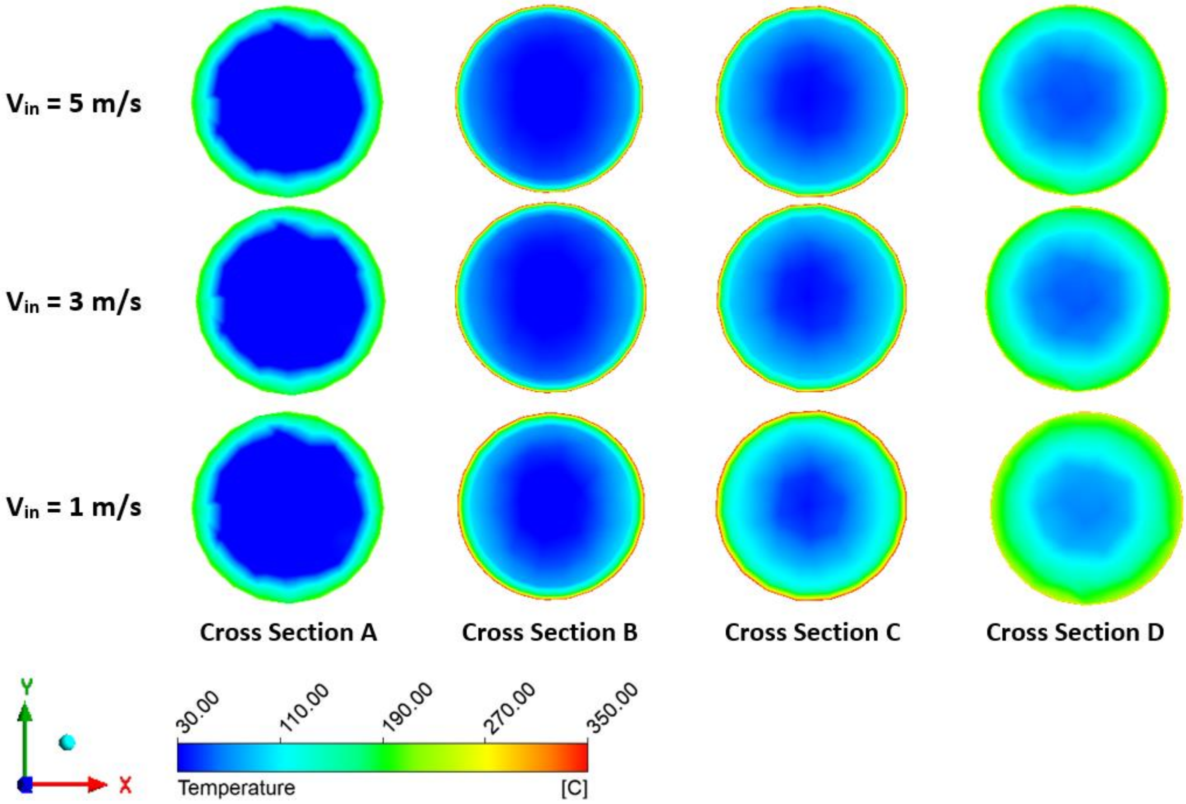


**Fig. 13.** Temperature distribution at selected cross-section at three different cases

## 4. Conclusions

The study evaluates the thermal transfer, pressure distribution, and velocity propagation of the heated air flowing through the pipes in a furnace in three different cases applied. It can be derived from the results and discussions that the velocity at the inlet part of the pipe affects the heat propagation, thermal distribution, velocity, and also pressure distribution along the pipes. The smaller velocity applied in the inlet part of the pipe, the resulting thermal transfer at the outlet of the pipe is higher, reaching a value of more than 100 °C when 1 m/s of inlet velocity is set. In terms of the resulting pressure, the higher velocity inlet value resulting higher pressure also in the pipe. Similar results also occur for velocity distribution inside the pipe. All in all, the objective of the resulting highest temperature in the heating chamber of the oven can be achieved by applying 1 m/s inlet velocity to get the highest temperature at the outlet when the pipes is heated at 400 °C. However, the study to evaluate the effect of different pipe temperatures need to be further observed, which will become the next research activity of this project. The evaluation and validation of the real condition will also be planned to be conducted. The results of the study give the contribution to the manufacturer to evaluate the resulting temperature of a similar furnace system based on the user’s need.

**Acknowledgement**
Universitas Negeri Yogyakarta fully funds this research in this work with grant number SP DIPA-023.17.2.677509/2022. The work is also supported by the Design and Simulation Laboratory in the Department of Automotive Engineering Education and Licensed Software by the CADD Laboratory in the Department of Mechanical Engineering Education Universitas Negeri Yogyakarta.

## References

[1] Wang, Yu, Yunxiu Sai, and Dongliang Yuan. "Key technologies for energy saving and emission reduction of heating furnaces in petrochemical plants." *Thermal Science* 25, no. 4 Part B (2021): 3159-3168. https://doi.org/10.2298/TSCI2104159W
[2] Lubis, M. F., N. Matondang, and Y. Nabilah. "Production of biomass briquettes from septic tank at pilot scale." In *IOP Conference Series: Materials Science and Engineering*, vol. 1122, no. 1, p. 012084. IOP Publishing, 2021. https://doi.org/10.1088/1757-899X/1122/1/012084
[3] Rinanda, Aditiya Darma, Wahidin Nuriana, and Sutrisno Sutrisno. "Pengaruh Variasi Tekanan Terhadap Kerapatan, Kadar Air Dan Laju Pembakaran Pada Biobriket Limbah Kayu Mahoni." *JURNAL PILAR TEKNOLOGI Jurnal Ilmiah Ilmu Ilmu Teknik* 6, no. 1 (2021): 21-24. https://doi.org/10.33319/piltek.v6i1.67
[4] Ifa, La. "Production of bio-briquette from biochar derived from pyrolysis of cashew nut waste." *Ecology, EnvironmentandConservation (EEC)* 25 (2019): 125-131.
[5] LIU, Jing, Jia-dong LI, Hao-jie WANG, and Zhao-dong WANG. "Numerical Simulation to Optimize Heating System in Vacuum Carburizing Furnace." *Journal of Northeastern University (Natural Science)* 40, no. 5 (2019): 641.
[6] Xu, Weide, Junfeng Zhang, and Ridong Zhang. "Application of multi-model switching predictive functional control on the temperature system of an electric heating furnace." *ISA transactions* 68 (2017): 287-292. https://doi.org/10.1016/j.isatra.2017.02.001
[7] Lin, G, Wang. S, X. Zhang, J. Gao, and D. Gao, "Optimal Design and Analysis of Heating System of Czochralski Single Crystal Furnace," *Rengong Jingti Xuebao/Journal of Synthetic Crystals* 50, no. 8 (2021): 1541-1551.
[8] Li, Z, Li. C, F. Qin, and Y. Hu, "Heating process analysis and system design of continuous annealing furnace for cold-rolled stainless steel," *Jinshu Rechuli/Heat Treatment of Metals* 46, no. 3 (2021): 191-196.
[9] Nizhegorodov, A. I. "Nonlinear Heating System of Modular-Pouring and Platform Furnaces for Firing Bulk Materials." *Refractories and Industrial Ceramics* 62, no. 4 (2021): 384-391. https://doi.org/10.1007/s11148-021-00630-4
[10] Cheng, Qiang, Jinlong Gong, Gang Xiao, Congbin Yang, Zhifeng Liu, and Baobao Qi. "Research on energy-saving production planning of periodic forging resistance furnace." *Journal of Cleaner Production* 275 (2020): 122897. https://doi.org/10.1016/j.jclepro.2020.122897
[11] Chakraborty, Saurav, and Prabal Talukdar. "Efficient Modeling and Optimal Design of Coal Fired Pusher Type Reheating Furnace." *Heat Transfer Engineering* 42, no. 22 (2021): 1949-1968. https://doi.org/10.1080/01457632.2020.1834218
[12] Badshah, Saeed, Muhammad Atif, Ihsan Ul Haq, Suheel Abdullah Malik, Mujahid Badshah, and Sakhi Jan. "Thermal analysis of vacuum resistance furnace." *Processes* 7, no. 12 (2019): 907. https://doi.org/10.3390/pr7120907
[13] Mayrhofer, Markus, Michael Koller, Peter Seemann, Rene Prieler, and Christoph Hochenauer. "CFD investigation of a vertical annealing furnace for stainless steel and non-ferrous alloys strips–A comparative study on air-staged & MILD combustion." *Thermal Science and Engineering Progress* 28 (2022): 101056. https://doi.org/10.1016/j.tsep.2021.101056
[14] Matykhin, V. I., A. V. Matykhina, and S. E. Punenkov. "Improvement of Thermal Performance of a Shaft Furnace for Asbestos Raw Material Drying." *Refractories and Industrial Ceramics* 62, no. 6 (2022): 628-633. https://doi.org/10.17073/1683-4518-2021-12-7-13
[15] Jouhara, Hussam, Sulaiman Almahmoud, Daniel Brough, Valentin Guichet, Bertrand Delpech, Amisha Chauhan, Lujean Ahmad, and Nicolas Serey. "Experimental and theoretical investigation of the performance of an air to water multi-pass heat pipe-based heat exchanger." *Energy* 219 (2021): 119624. https://doi.org/10.1016/j.energy.2020.119624
[16] Li, Cheng, Wei-Jun Zhang, Zhi Yi, Na Li, and Kai Li. "Impact Analysis of Intermediate Heat Carrier on Heat Transfer in Furnace." *Mathematical Problems in Engineering* (2020). https://doi.org/10.1155/2020/7646329
[17] Ohol, Sandeep, V. K. Mathew, Savita Shinde, and G. Balachandran. "Heat balance analysis in electric arc furnace for process improvement." In *E3S Web of Conferences*, vol. 170, p. 02012. EDP Sciences, 2020. https://doi.org/10.1051/e3sconf/202017002012

[18] Yudianto, A., Saeid Ghafari, Pierre Huet, and Muhkamad Wakid. "Evaluation of the Temperature Distribution and Structural Deformation of the Car Dashboard Subjected to Direct Sunlight." In *Journal of Physics: Conference Series*, vol. 1273, no. 1, p. 012076. IOP Publishing, 2019. https://doi.org/10.1088/1742-6596/1273/1/012076

[19] Adiyasa, I. W., A. P. Prasetyono, A. Yudianto, P. P. W. Begawan, and D. Sultantyo. "Bilinear interpolation method on 8x8 pixel thermal camera for temperature instrument of combustion engine." In *Journal of Physics: Conference Series*, vol. 1700, no. 1, p. 012076. IOP Publishing, 2020. https://doi.org/10.1088/1742-6596/1700/1/012076

[20] Yudianto, Aan, I. Wayan Adiyasa, and Afri Yudantoko. "Aerodynamics of Bus Platooning under Crosswind." *Automotive Experiences* 4, no. 3 (2021): 119-130.

[21] Yudianto, Aan, Herminarto Sofyan, and Naufal Annas Fauzi. "Aerodynamic Characteristics of Overtaking Bus under Crosswind: CFD Investigation." *CFD Letters* 14, no. 8 (2022): 20-32. https://doi.org/10.37934/cfdl.14.8.2032

[22] Shetty, Divya D., Mahamad Sulthan, Mohammad Zuber, Irfan Anjum Badruddin, and Chandrakant R. Kini. "Computational Design and Analysis of a Novel Battery Thermal Management System of a Single 26650 Li-Ion Battery Cell for Electric Vehicle Application." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 93, no. 2 (2022): 61-75. https://doi.org/10.37934/arfmts.93.2.6175

[23] Maragkos, Georgios, and Tarek Beji. "Review of convective heat transfer modelling in CFD simulations of fire-Driven Flows." *Applied Sciences* 11, no. 11 (2021): 5240. https://doi.org/10.3390/app11115240

[24] de Albuquerque, Clarissa Detomi, Sébastien Curet, and Lionel Boillereaux. "A 3D-CFD-heat-transfer-based model for the microbial inactivation of pasteurized food products." *Innovative food science & emerging technologies* 54 (2019): 172-181. https://doi.org/10.1016/j.ifset.2019.04.007

[25] Setoodeh, Hamed, Amirhosein Moonesi Shabestary, Wei Ding, Dirk Lucas, and Uwe Hampel. "CFD-modelling of boiling in a heated pipe including flow pattern transition." *Applied Thermal Engineering* 204 (2022): 117962. https://doi.org/10.1016/j.applthermaleng.2021.117962

[26] Li, Chao, Yanling Guan, Ruitao Yang, Xiong Lu, Wenxue Xiong, and Anjie Long. "Effect of inner pipe type on the heat transfer performance of deep-buried coaxial double-pipe heat exchangers." *Renewable energy* 145 (2020): 1049-1060. https://doi.org/10.1016/j.renene.2019.06.101

[27] Gupta, Naveen Kumar, Arun Kumar Tiwari, and Subrata Kumar Ghosh. "Heat transfer mechanisms in heat pipes using nanofluids–A review." *Experimental Thermal and Fluid Science* 90 (2018): 84-100. https://doi.org/10.1016/j.expthermflusci.2017.08.013

[28] Elfaghi, Abdulhafid MA, Alhadi A. Abosbaia, Munir FA Alkbir, and Abdoulhdi AB Omran. "CFD Simulation of Forced Convection Heat Transfer Enhancement in Pipe Using Al2O3/Water Nanofluid." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 44-49.

[29] Ling, Yang Dong, and Shuichi Torii. "Heat transfer enhancements in heat pipe constructed with a copper porous microstructure." *International Journal of Green Energy* 18, no. 2 (2021): 166-171. https://doi.org/10.1080/15435075.2020.1847114

[30] Dong, Feng, Shuo Zhang, Xuewei Shi, Hao Wu, and Chao Tan. "Flow regimes identification-based multidomain features for gas–liquid two-phase flow in horizontal pipe." *IEEE Transactions on Instrumentation and Measurement* 70 (2021): 1-11. https://doi.org/10.1109/TIM.2021.3089767

[31] Zhang, Junhong, Weidong Li, Jiewei Lin, Yongbo Qiu, Yi Yuan, and Tianyi Zhou. "Failure analysis of a high-pressure fuel pipe of engine." *Engineering Failure Analysis* 103 (2019): 70-81. https://doi.org/10.1016/j.engfailanal.2019.04.040

[32] Wang, Zhen-yu, Yang Zhao, Guo-wei Ma, and Zhi-guo He. "A numerical study on the high-velocity impact behavior of pressure pipes." *Journal of Zhejiang University-SCIENCE A* 17, no. 6 (2016): 443-453. https://doi.org/10.1631/jzus.A1500112

[33] Utomo, Allessandro Setyo Anggito, F. Fariz, and S. A. Lambang. "Reduction of Drag Resistance by Pressure Drop in Pipeline with 3-Dimensional Design Optimization." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 83, no. 2 (2021): 44-53. https://doi.org/10.37934/arfmts.83.2.4453

[34] Burggraf, Jonas, Peter Farber, Kavien Raaj Karpaiya, and Peer Ueberholz. "Numerical investigation of laminar flow heat transfer of TiO2-Water nanofluid in a heated pipe." *Heat Transfer Engineering* 42, no. 19-20 (2021): 1635-1647. https://doi.org/10.1080/01457632.2020.1818379

[35] Liu, Jialun, Huixiong Li, Xianliang Lei, Qian Zhang, and Liangxing Li. "An improved model on flow distributions of supercritical pressure water in parallel heated pipes." *Applied Thermal Engineering* 130 (2018): 793-803. https://doi.org/10.1016/j.applthermaleng.2017.10.170

[36] Isa, N. Mat, A. F. Ab Rahman, and A. Sadikin. "Numerical Simulation of Splitting Devices in Horizontal Pipeline." *Journal of Advanced Research in Applied Mechanics* 5, no. 1 (2015): 8-14.

[37] Zabrodina, Ekaterina Alekseevna, Ol'ga Vasil'evna Nikolaeva, Nikolai Nikolaevich Fimin, and Valerii Mikhailovich Chechetkin. "Hydrodynamic flows in heated pipes with boundary layer computed using the BGK

model." *Computational Mathematics and Mathematical Physics* 61, no. 11 (2021): 1895-1909. https://doi.org/10.1134/S0965542521110166