Simulation of Heat Removal for Air Distribution Unit with Parallel Fin Using Air at Velocity of 1 m/s

Muhamad Nurfikri Bin Azmi, Hanafiah Bin Zainal Abidin,

Faculty of Chemical Engineering, Universiti Teknologi Mara

***Abstract* — Solid waste generation is a major concern in today's vast population as a result of economic development, rapid urbanisation and industrialisation. There are too much solid waste generated to the extent that the landfill sites is not practical anymore so an alternative disposal method should be taken which is waste to energy technology (WTE) and focusing on incineration method. The study is conducted mainly because of one part of air nozzle which is air duct cannot handle heat stress due to high operating temperature of incinerator. In order to overcome the problem, cooling media which is air is added between insulator and nozzle. The objectives of the study are to study the air distribution pattern in the incinerator and to improve the heat removal through simulation of air distribution unit with parallel fin using air using Computational Fluid Dynamics (CFD) simulation. The study have some limitation to achieve the desired outcome such as using ANSYS 15.0, use common air nozzle design, use rotary kiln incinerator and many more. Studies on the past researcher's research is done based on the major aspect which are solid waste, waste to energy technologies (WTE) and computational fluid dynamics (CFD). The simulation is conducted through four main steps started with specify and modelling the design, undergoing meshing process, conducting solving process and last but not least analysing the obtained result to determine the desired outcome. Results from the simulation shows that presence of cooling media, fin and changes in surface area greatly affect the temperature and turbulence intensity distribution in the air nozzle.**

***Keywords* —** *ANSYS 15.0, Air nozzle, Waste, Turbulence Intensity, Temperature.*

I. INTRODUCTION

Malaysia is a developed country that currently grows in populations which also resulted to the growing of solid waste generation [1]. It is not practical to rely only on the landfill sites to handle the solid waste [2]. Many methods can be used to dispose the increasing number of waste and one of them is waste to energy technology (WTE) that can convert the waste into electrical energy and heat energy [3]. Thermal treatment technologies is among the famous technology that yield energy in the form of heat and electricity which include pyrolysis, gasification and incineration [4].

Conventional incinerator commonly involve organic matter of waste as feedstock to be reacted with oxygen under aerobic combustion process in a furnace under high pressure and adequate temperature [3]. In Malaysia, rotary kiln type incinerator is used and there have been a few rotary kiln incinerator although in a small scale located at Labuan, Langkawi, Pangkor and Tioman since 1996 [5]. One drawback of using the incinerator is that its efficiency rapidly decreases over time. This is because of one part of air nozzle which is air duct cannot handle heat stress due to high operating temperature of incinerator. So, a software is introduced to analyse the process involving fluid flow, heat transfer and associated phenomena such as chemical reactions by using computer based simulation known as Computational Fluid Dynamics (CFD) [6].

CFD conduct numerical simulation of fluid flow which has proved to be a very good help for specific industry such as helping the chemical engineers to maximize yield from the equipment [7]. In the CFD, there is one major equation being used for the calculation which is Navier Stoke Equation that involve mathematical model of the motion of a fluid [8]. Through CFD, effect of air as cooling media and used of parallel fin as mixer and heat distributor can be determined in details.

II. METHODOLOGY

A. Specify and Modelling Design

ANSYS 15.0 software is installed and Workbench program is used to run the simulation. The fluid domain can be drawn based on three plane (X, Y and Z plane). The drawing of simulated air nozzle is constructed based on the existing standard design of the air nozzle in the rotary kiln incinerator in the industry. For the alternative way, the drawing also can be drawn in AUTOCAD software before being converted into fluid domain and then can be exported into the Workbench 15.0 for next simulation.

B. Meshing Process

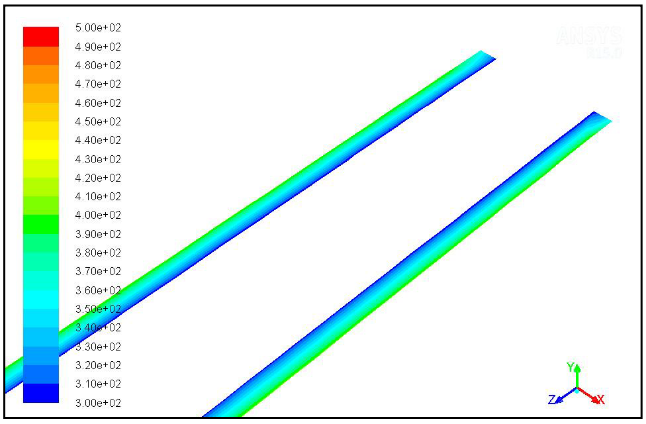
Click on "mesh" command at the outline and click on the surface of the drawing to identify the inlet, outlet and the wall of fluid domain as to differentiate from one boundary to another boundary. The sizing of the meshing is set from course size into fine sizing so that detail simulation can be conducted. Apply the "selected geometry" command so that the mesh can be generated. If no boundary has been assigned, the geometry will automatically set as wall.

C. Solver / Setup

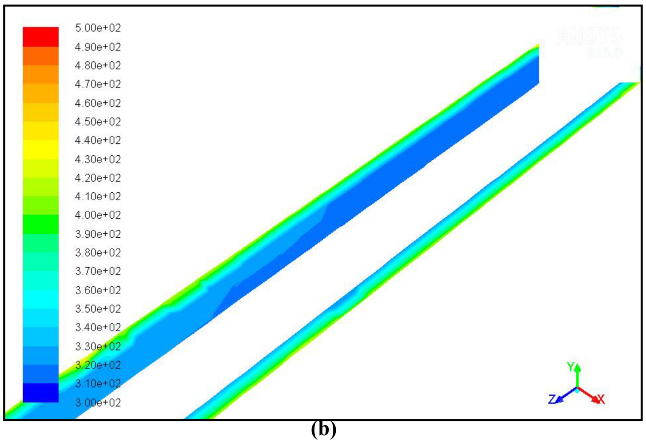
Solver or setup is a step to run the simulation of the simulated air nozzle by inputting several parameters to achieve the desired result. Once the "Setup" command is clicked, there is "Solution Setup" section where the parameters can be inserted. Energy and viscous which consist of Energy Equation and k-Epsilon Models respectively are important that need to be used in the simulation. Inlet temperature of 300 K and velocity of 1 m/s were set at the inlet while temperature of the heat transfer wall is set at 500 K. the iteration was then run until the calculation become converged. Results in term of temperature and turbulence intensity can be viewed for further discussions.

III. RESULTS AND DISCUSSION

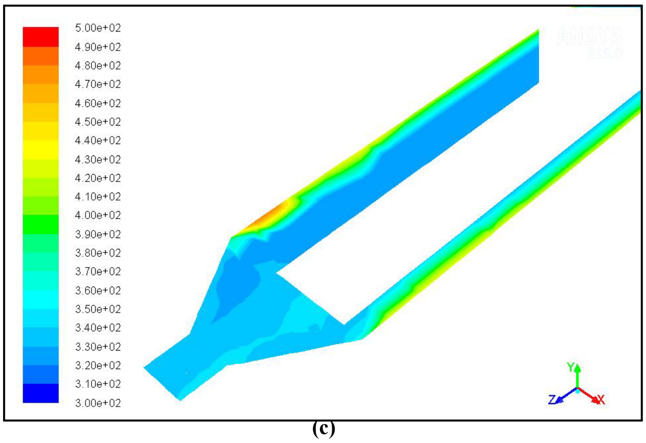
A. Temperature



(a)



(b)



(c)

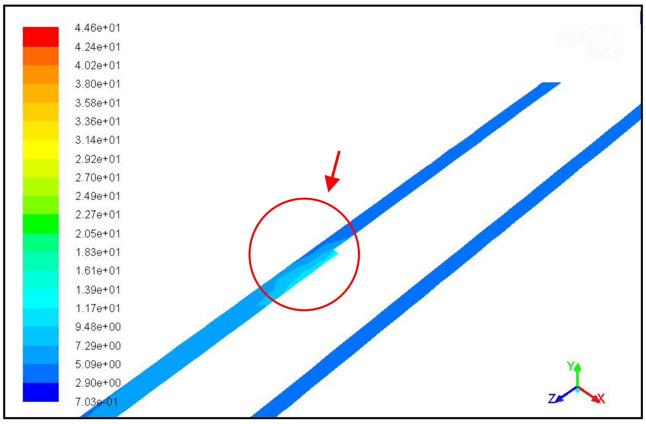
Fig. 1: Temperature distribution at (a) inlet, (b) middle and (c) outlet section.

Air and parallel fin plays an important role in determining the temperature distribution at the wall of air nozzle. The inner wall temperature of the air nozzle only experienced a slight increase in temperature and still can be maintained at low temperature because of the presence of air as cooling media which prevent direct heat stress into the inner wall by absorbing the remaining heat.

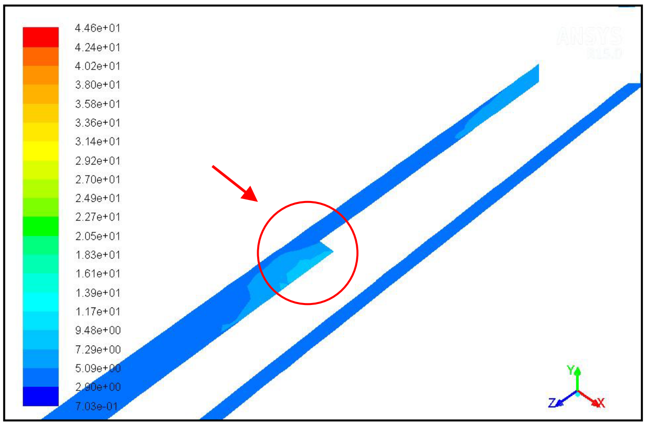
Parallel fin are installed at every side of the nozzle. As the air travel throughout the nozzle, instead of going in one direction, the air and the heat are distributed evenly throughout the wall for a better surface area for the heat exchanger process to take place. This is proven by comparing the temperature gradient at all the three cutting plane where a uniform temperature distribution can be seen at the side that have more fin compared to the side with less fin.

Air travel from the inlet and exit at the outlet of air nozzle and during that time, the temperature of the wall keep building up and increases along the nozzle. This is because as the air continuously being supplied, the combustion process became more and more vigorous and the heat stress formed from the process keep increasing and affect the overall temperature of the air nozzle's wall. Even though the temperature increases, the efficiency of the air nozzle is still at its best because the air nozzle still can withstand the temperature with the help of cooling media.

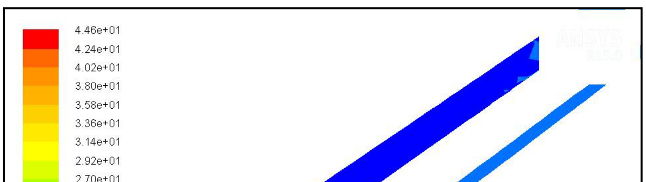
B. Turbulence Intensity



(a)



(b)



(c)

Fig. 2: Turbulence intensity at (a) inlet, (b) middle and (c) outlet section.

Based on the figures above, it can be seen that turbulence intensity is mainly affected by surface area and number of parallel fin in the air nozzle. Generally, air nozzle has three stages of design which are drying (first 3 metre), combustion (2 metre) and cooling section (last 5 metre).

From the drying section to the combustion and even to the cooling section, there is an increase of area which also resulted to the changes in the turbulence intensity to become more intense. In the small area, the air flow is limited as there is not enough space for the turbulence to be occur but at the transition point where small area go to the large area, the air suddenly gain larger space for the turbulence of air to be occur more vigorously and intensely before started to flow uniformly.

It is proven from the figure that the turbulence intensity increases especially at the transition point. Other than the surface area, the number of parallel fin also affect the turbulence intensity.

Parallel fin is functioned to help for better air and heat distribution. Due to the fin, the air is distributed evenly and mixed well with each other which is one of the main reason that affect the turbulence intensity at the nozzle.

IV. Conclusion

The simulation is conducted to to study the air distribution pattern in the incinerator and to improve the heat removal through simulation of air distribution unit with parallel fin using air using Computational Fluid Dynamics (CFD) simulation. Result from the simulation shows that presence of cooling media and fin greatly affect the temperature and turbulence intensity distribution in the air nozzle. It can be seen that with an addition of air as cooling media, the temperature of the nozzle especially wall can be lower so that the efficiency of the nozzle can be maintained as the cooling media absorb the excess heat stress from the surrounding that can cause the nozzle to be bend. Parallel fin is the fin used and based on the simulation, the fin really help to mix and distribute the air and heat evenly throughout the nozzle wich resulted to a better turbulence intensity. In a nut shell, the simulation conducted is a success based on the parameters that being discussed and some suggestion can be done to improve the results of the simulation.

V. RECOMMENDATIONS

Some suggestions or recommendations can be done to improve the results of the simulation. Instead of using air as the cooling media, water also can be used as alternative because water is a better heat absorber compared to air. Next, fin play a major part that contribute to a better air and heat distibution. Spiral fin provide a better air distribution compared to parallel fin because the spiral fin cover the whole air nozzle for a large surface area for the air to be mixed and flow.

Acknowledgment

Thank you to my supervisor, Sir Hanafiah Bin Zainal Abidin for his supervisions, teaching and supports in the process of completing this research project. His helps through suggestions and constructive comment really helps a lot in assisting me with this project. Next, I would like to express my gratitude to my colleagues for helping me with the research project with all the discussion and help that I could possibly get. Furthermore, it is hard for me to complete the research project without the endless support and encouragement from my beloved family. Last but not least, I'm forever grateful to Allah S.W.T for giving me strength and patience throughout the process of completing the research. Without all of these support, this research project is really challenging to be completed. Thank you all.

References

[1] Tan, S. T., Ho, W. S., Hashim, H., Lee, C. T., Taib, M. R., & Ho, C. S. (2015). Energy, economic and environmental (3E) analysis of waste-to-energy (WTE) strategies for municipal solid waste (MSW) management in Malaysia. Energy Conversion and Management, 102, 111–120. https://doi.org/10.1016/j.enconman.2015.02.010

[2] Moh, Y. C., & Abd Manaf, L. (2017). Solid waste management transformation and future challenges of source separation and recycling practice in Malaysia. Resources, Conservation and Recycling, 116,1–14. https://doi.org/10.1016/j.resconrec.2016.09.012

[3] Moya, D., Aldás, C., López, G., & Kaparaju, P. (2017). Municipal solid waste as a valuable renewable energy resource: A worldwide opportunity of energy recovery by using Waste-To-Energy Technologies. Energy Procedia, 134, 286–295. https://doi.org/10.1016/j.egypro.2017.09.618.

[4] Jacobi, J., & Vlahoplus, C. (2011). Waste to Energy Technology, (June).

[5] Sharifah, A. S. A. K., Abidin, H. Z., Sulaiman, M. R., Khoo, K. H., & Ali, H. (2008). Combustion characteristics of Malaysian municipal solid waste and predictions of air flow in a rotary kiln incinerator. Journal of Material Cycles and Waste Management, 10(2), 116–123. https://doi.org/10.1007/s10163-008-0207-3.

[6] Versteeg, H. K., & W, M. (2007). An Introduction to Computational Fluid Dynamics. World Wide Web Internet And Web Information Systems (Vol. M). https://doi.org/10.2514/1.22547

[7] Rumsey, C. (2006). Introduction: Computational Fluid Dynamics Validation for Synthetic Jets. AIAA Journal, 44(2), 193–193. https://doi.org/10.2514/1.22547

[8] Dobek, S. (2012). Fluid Dynamics and the Navier-Stokes Equation. Cs.Umd.Edu.