JMES The International Journal of
Mechanical Engineering and Sciences
http://iptek.its.ac.id/index.php/jmes/index

Numerical Study of Temperature and Air Velocity Distribution in Oil Filling Factory

Hernawan Novianto and Prabowo*

Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia

Received: 10 June 2017, Revised:8 July 2017, Accepted: 15 August 2017

Abstract

This paper describes the use of CFD modeling to analyze the thermal comfort in the oil filling factory which has an area of 3000 m^2. The need for this analysis comes from an uncomfortable condition that is felt by the workers of the factory. A 3D simulation using FLUENT software 6.3.26 conducted to analyze the temperature and velocity distribution in the plant room. The water is assumed as an incompressible ideal gas, steady flow, turbulence models used $k-\varepsilon$ standard, the SIMPLE algorithm and second order upwind discretisation. Analysis was conducted on existing models and propose models, whereby on a model propose, the diffuser is installed above the workers with a height of 4,2 m above the floor, the velocity of supply air diffuser is varied from 1,5 m/s, 2 m/s, and 2,5 m/s. The simulation results show that the temperature distribution in the existing conditions in the range of about 34-36°C, this value exceeds the thermal comfort standards specified by ASHRAE. The simulation results show that the proposed model better temperature distribution, where the temperature is generated in the range of ASHRAE thermal comfort criteria, ranging from 24-26°C, and the supply air velocity at the diffuser inlet of 1,5 m/s recommended for use in AHU system. For the 20 units of the diffuser with inlet velocity of 1,5 m/s, the mass flow rate that should be handled by a cooling device is 9 kg/s and require a cooling capacity of 0,128 MW. This is 58% more efficient than cooling the entire room factory.

Keywords: Distribution of Temperature, Velocity Distribution, CFD modeling.

1. Introduction

Planning air conditioning systems not only overcome the problem of thermal comfort but also reduces energy use is something that is needed today. The use of CFD for analyzing the temperature in the room has increased, such as apartment [1], the office [2], clinic [3], the museum [4], stadium [5], theatre [6] and the airport terminal [7]. In this study, CFD is used to analyze the temperature and air velocity distribution in the oil filling factory room in Gresik, Indonesia, Figure 1 shown layout of the plant which has an area of 3000 m^2. The need for this analysis came from uncomfortable conditions felt by the workers of the factory. Some actions have been carried out such as adding 24 units of ventilators and 7 units of blower fan, but the room temperature is still high.

Several studies conducted to investigate the thermal comfort in large buildings equipped with a cooling system. Yong wang et al. [8] studied full and stratified air conditioning design in buildings with large space. They compare and analyze the indoor comfort parameters such as temperature and air velocity, result in recommendations to use of stratified air conditioning design for large space buildings. Q. Li et al. [9] evaluate the thermal environment in the room of the railway station building, they are also investigating on the temperature and air velocity, and found that the mid-height of the building is a good position for the air cooling supply and horizontal angle of 0° supply is recommended.

The above literature is about room temperature, and nothing has been said about the temperature on people's body surface. Therefore, the present work investigates numerically the temperature distribution on worker body surface. The purpose of this research is to investigate the thermal comfort of workers if the spot cooling conducted. The influence of the air supply to the temperature and air velocity distribution on the body surface discussed. From these discussions, advice on optimizing the design of air conditioning made. The main objective is to introduce an effective way to assess and optimize the design of air conditioning.

2. Model Description

2.1. Configuration

Investigated oil filling factory is located in Gresik, Indonesia, a building size of about 75m (length) x 40m (width) x 13,4m (high). The size is too complex to be analyzed, therefore line 2 shown in Figure 2 is selected for sample to be analyzed, and the model is simplified.

*Email:prabowo@me.its.ac.id
Phone:+628

This model has a size of about 10,3 m x 40 m x 13,4 m. Geometry of diffuser which is located at a height of 4,2 m from the floor on each workers is added to analyze the effect of cooling diffuser to thermal comfort.

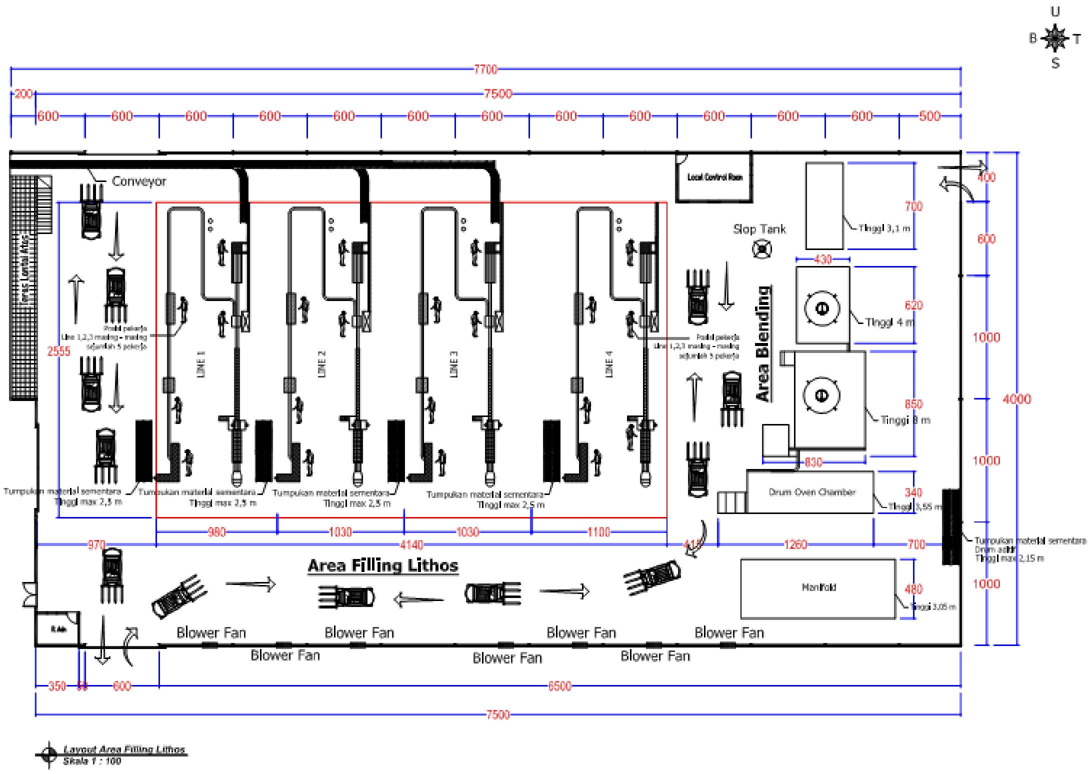

Figure 1. Oil-Filling Plant Layout.

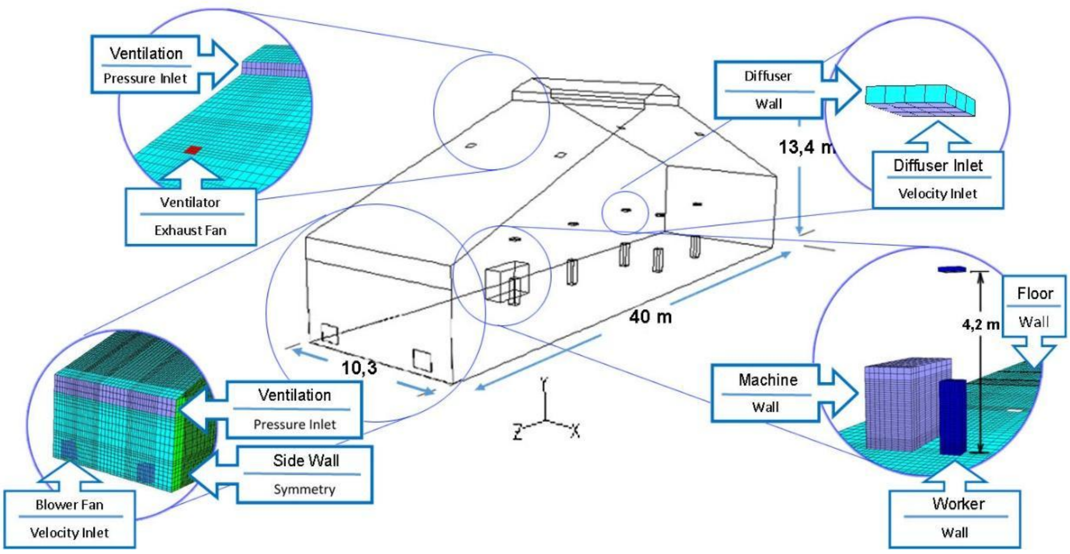

Figure 2. Model Geometry.

2.2. Meshing

Type of meshing used in this simulation is tetrahedral and hexahedral mesh with a mesh number of 685.374 Cell Mesh and 727.788 Node Mesh. The results meshing shown in Figure 3.

2.3. Boundary Conditions

Domain of the study is shown in Figure 4, where in the model of existing, blower fan is activated at velocity inlet magnitude 6 m/s and the geometry of the diffuser is hidden (OFF), contrary to the propose models, blower fan is turned off and the diffuser is turned on (ON), the velocity of the air supply from the diffuser was varied from 1,5 m/s, 2 m/s, and 2,5 m/s, with a supply temperature of about 20°C. the setting of boundary conditions are shown in Table 1. A 3D-simulation using FLUENT 6.3.26 software conducted to analyze the temperature and velocity distribution in the plant room. The air is assumed as an incompressible ideal gas, steady flow, turbulence models used is the standard k- ε, SIMPLE algorithm and second order upwind discretization. residuals for parameters energy set at 10^{-6}, while the other set of 10^{-4}.

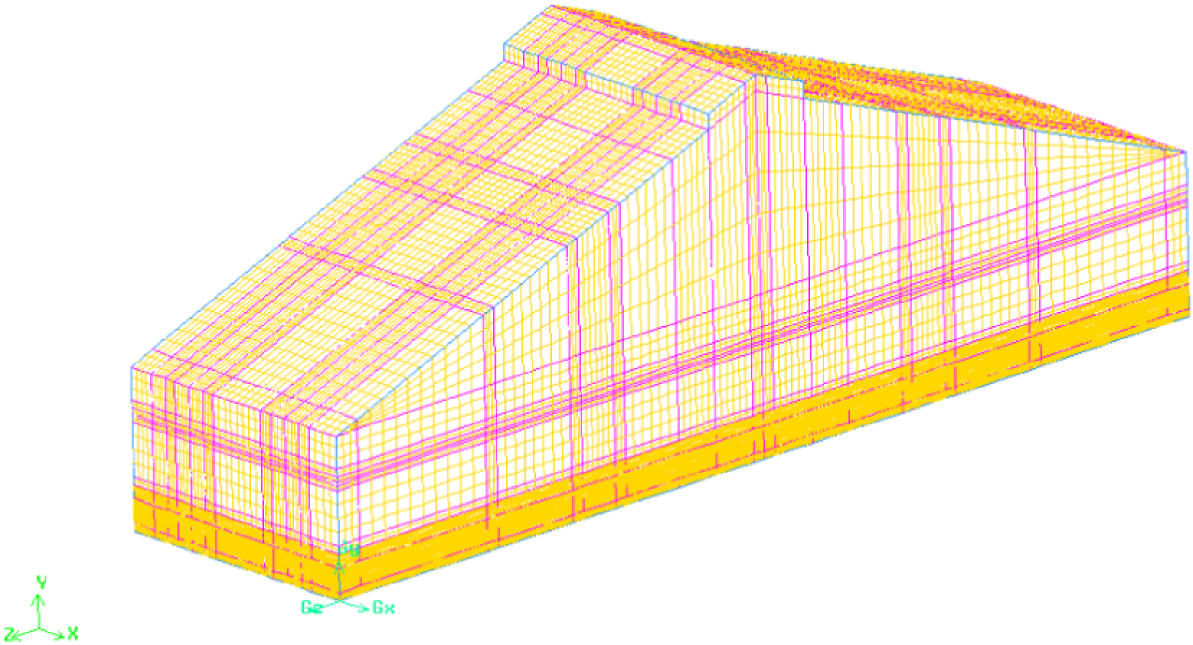

Figure 3. Meshing of the model.

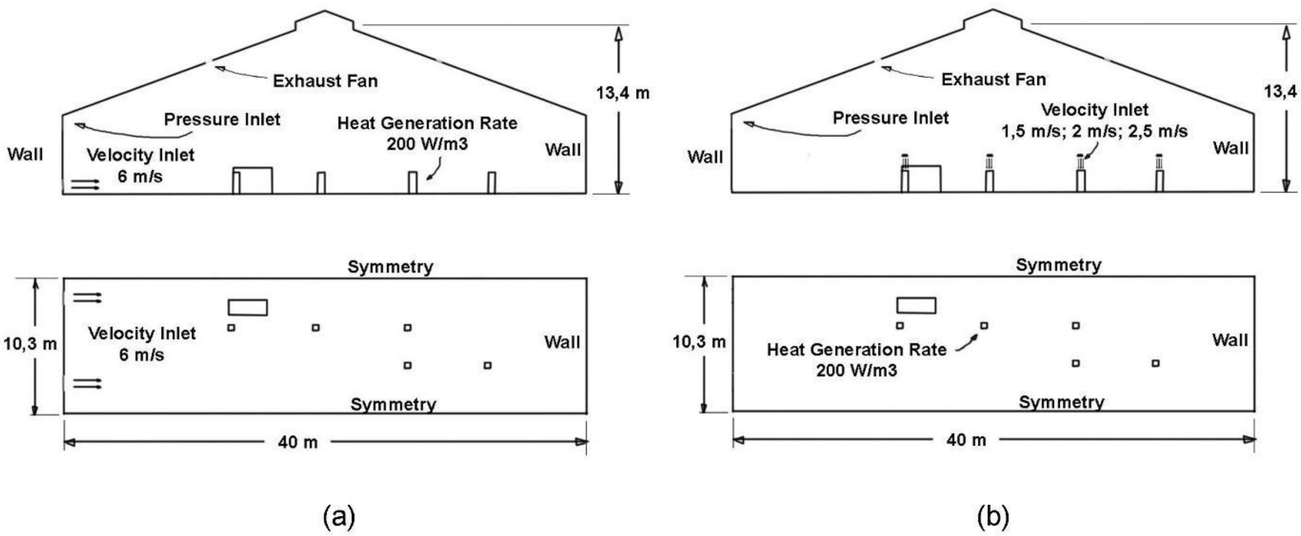

Figure 4. Domain of Investigation (a) Existing Model (b) Propose Model.

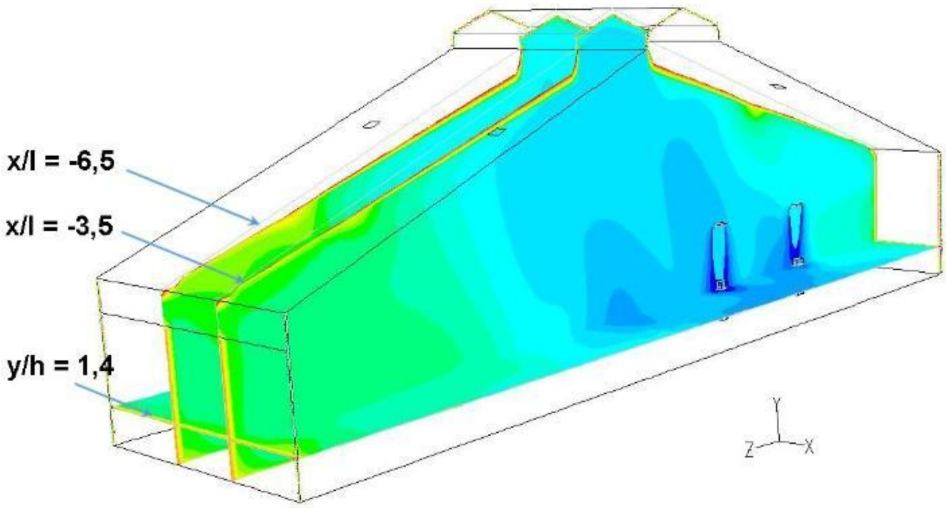

Figure 5. The Observation Field.

Table 1. Boundary Conditions

Model	Boundary Condition	Remarks
Blower Fan	Velocity inlet	V = 6 m/s;
		T = 34 °C
Diffuser Inlet	Velocity Inlet	T = 20 °C;
		V = 1,5 m/s,
Ventilator	Exhaust Fan	2 m/s, 2,5 m/s
		P = 0 Pa
Ventilation	Inlet Pressure	Initial P = 0 Pa (gauge);
Roof	Wall	T = 34 °C
		T = 50 °C
Floor	Wall	T = 34 ° C
Engine	Wall	T = 60 °C
Worker	Wall	Heat Generation Rate = 200 W/m^3
sidewalls (not real)	Symmetry	No set up required

2.4. Post Processing

The visualization of simulation results in the form of qualitative data and quantitative data, qualitative data in the form of temperature and air velocity contours and quantitative data in the form of a graph or numerical data. Field of observations are shown in Figure 5, where x/l= -6,5 is a field where the first, second, and third workers is located, the x/l=3,5 is the fourth and fifth workers located, and y/h = 1,4 is a field that has a height of 1,4m from the floor

3. Results and Discussion

3.1. Existing and Propose Simulation Results Analysis

Figure 6a shows airflow pathline of existing models, in which the air flow from the fan straight into the room, split into several directions after crashing machines and workers. Airflow spread after crashing the machine causes the flow does not fill the entire space, especially the occupied zone, beside the air flow brought in the room by the blower fan is high in temperature, consequently not able to cool down the occupied zone temperature. The temperature contour of the occupied zone shown in Figure

7, is ranging of 34-36°C. The value is exceeds the thermal comfort standards specified by ASHRAE

Figure 6b shown airflow pathline of propose models, cold air will spread when hit the workers and will cross every workers side. When the cold air touching the surface of the floor, the air stream will flow upwards. So the cold air will spread into the room. In the occupied zone , the air flow decrease in velocity. This shows the circulation formed by the effect of the cooling air and the hot air in the room. Figure 8 shows the differences of existing model and propose models simulation in contour and graph of temperature, in the contours of temperature of propose models, air supply velocity of the diffuser is able to cool down the air temperature in the occupied zone, the temperature distribution on the surface of the workers is in the range of 23,9°C – 24,7°C.

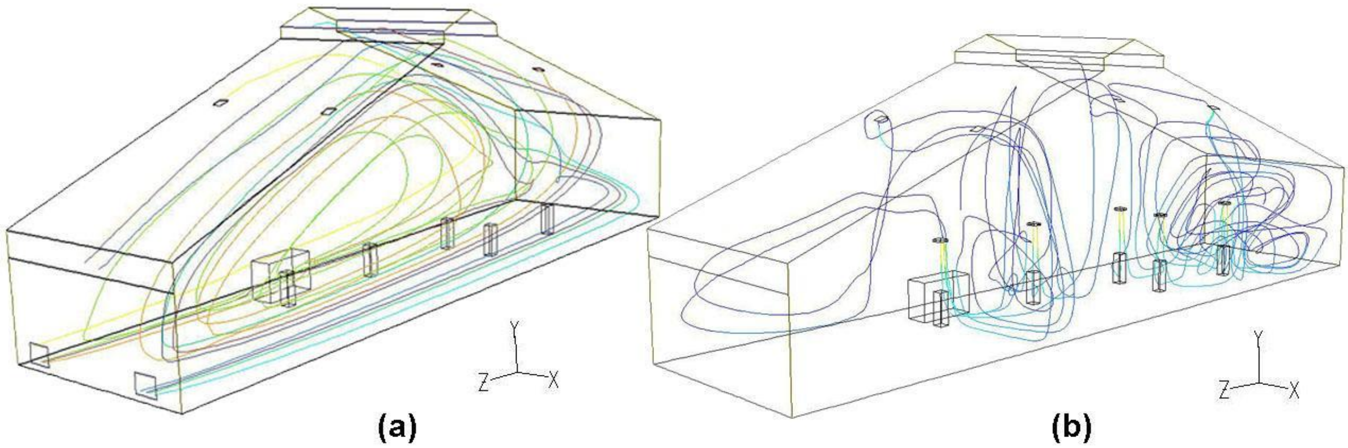

Figure 6. Air Flow Pathline (a) Existing Model (b) Propose Model.

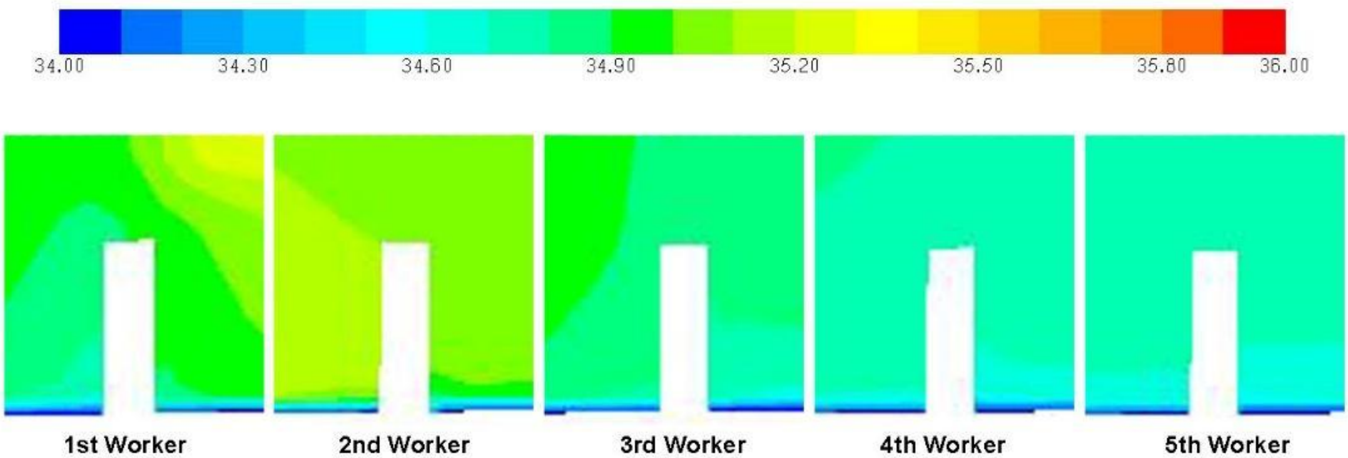

Figure 7. Temperature Contours on Existing Model.

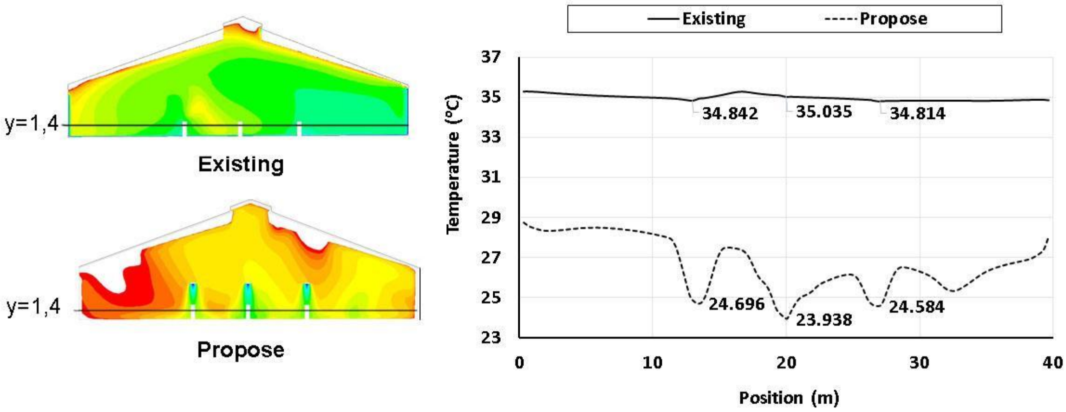

Figure 8. Graph of Temperature at y = 1,4 on x/l = -6,5.

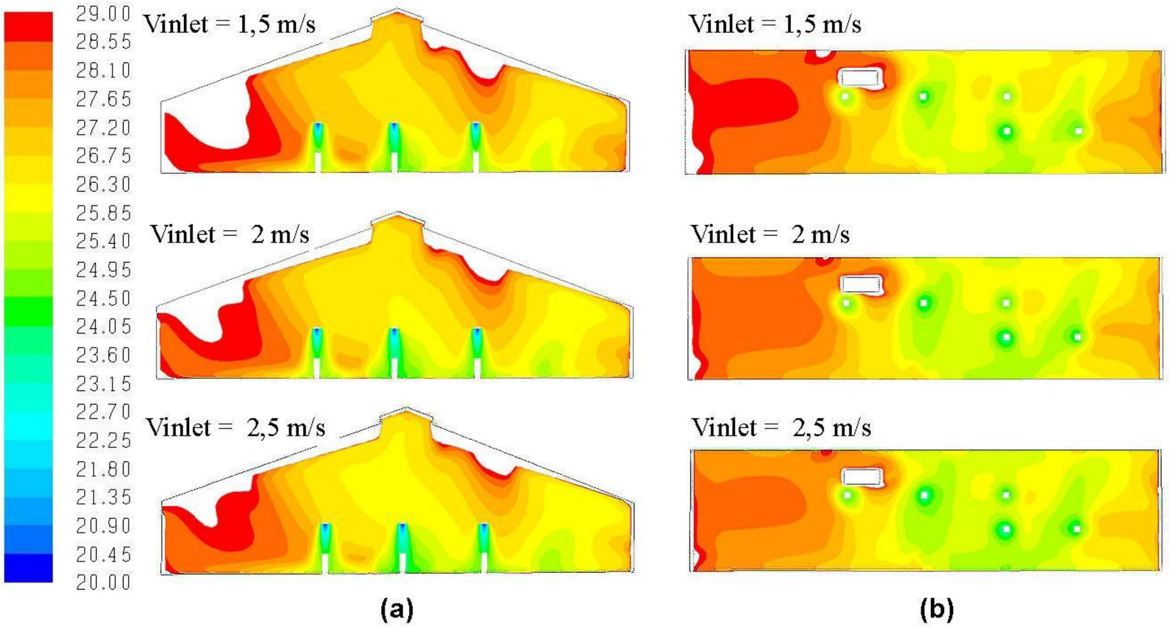

Figure 9. Effect of Diffuser's Velocity Inlet Variations to The Room Temperature.

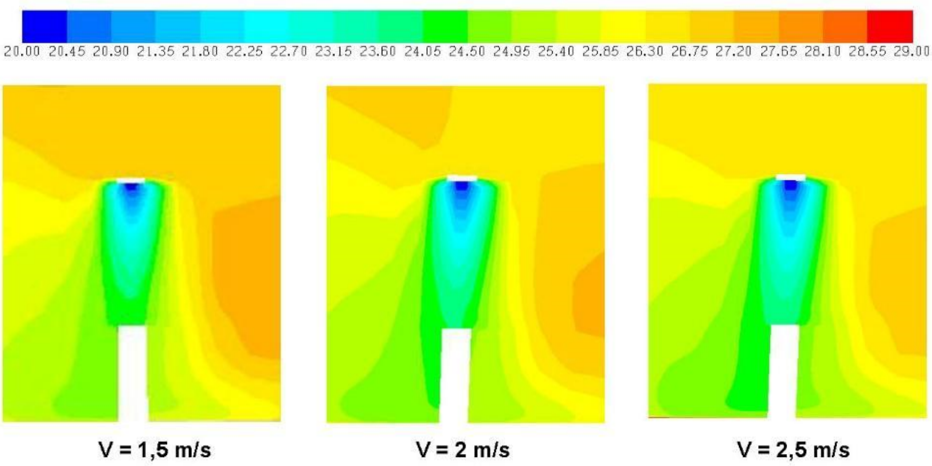

Figure 10. Effect of Diffuser's Velocity Inlet Variations On 5^{th} Worker's Temperature.

3.2. Effect of Air Velocity Variations on Inlet Diffuser

Figure 9 shows the contour of the air temperature distribution, higher velocities at the inlet diffuser, wider in cooled area coverage, this is indicated by the distribution of red color that shrinking with increasing of air supply velocity at the inlet diffuser, to be replaced by cooler color.

Figure 10 shows the effect of air supply variations at inlet diffuser to 5^{th} worker, in which the increasing air supply velocity at the inlet diffuser, the air temperature at the body surface of the worker will decrease, which is in line with the theory that an increase in air velocity, increasing the value of Reynolds Number, Nusselt Number and the heat transfer coefficient. When heat flux is constant, increasing heat transfer coefficient causes the difference of surface temperature and the infinity temperature smaller, which means the surface temperature is almost equal to the infinity temperature, or in other words the surface temperature gets colder.

Effect of air supply velocity at the inlet diffuser against the other workers, shown by Figure 11, In the same of the air supply velocity, the temperature at the body surface of the 1^{st} worker is most high than the others, caused by the worker's position is closest to the heat source, followed by 3^{rd} workers then workers 5. While temperatures at 2^{nd} worker and 4^{th} worker most cold, caused by their positition are among the other workers, beside influence by the diffuser is located above their, also get the effect of diffusers above 1^{st}, 3^{rd}, and 5^{th} workers. More detailed information of temperature distribution shown in the graph in Figure 12, which demonstrated the influence of the air supply velocity on temperature of each body surface of workers, the higher air velocity, causing the body surface of workers colder, this is indicated by movement of the plotting line to the left in the chart, temperature line of 1^{st} worker located at the right of another graph, is showing that the temperature at 1st worker highest. Temperature line at the bottom of the worker appears to rise sharply, this is because under the influence of heat from the floor.

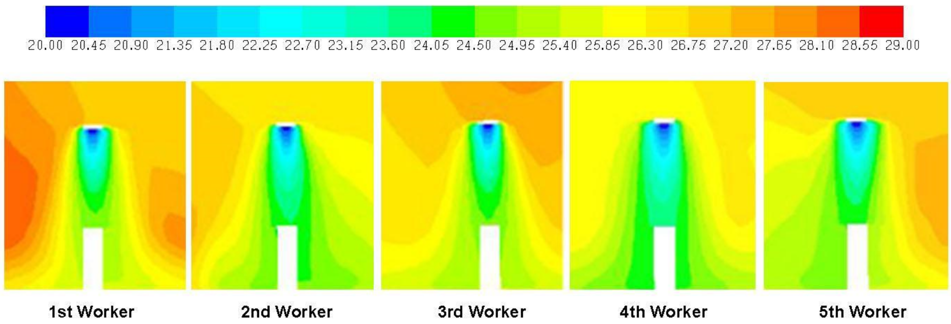

Figure 11. Temperature Contour at Each Worker.

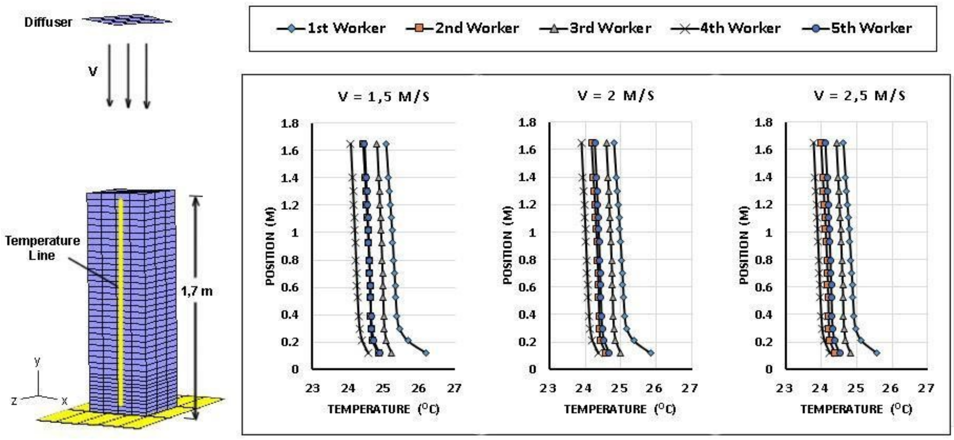

Figure 12. Graph of The Velocity Inlet Variations to The Worker's Body Surface Temperature.

Temperature distribution range of each worker in variations of air supply velocity shown in Table 2, For all variations of the air supply velocity, Each of which produce a comfortable temperature on the workers. Therefore the slowest air velocity of 1,5 m/s is recommended as the air supply velocity at the inlet diffuser in Air Handling Unit

Table 2. Effect of Air Supply Velocity on Worker's Body Surface Temperature

Velocity of Air Supply	Worker Body
1,5 m / s	24,0 to 27,0°C
2 m / s	23,8 to 26,6°C
2,5 m / s	23,7 to 26,3°C

(AHU) system design, For 20 units of diffuser, the mass flow rate must be supplied by the cooling device is 9 kg/s, therefore require a cooling capacity of 0,128 MW. This means that spot cooling method 58% more efficient than cooling the entire room.

4. Conclusions

1. On existing simulation, blower fan is not able to overcome the problems of existing thermal comfort. The simulation results show that the temperature in the existing conditions is in the range of about 34-36°C, this value exceeds the thermal comfort standards specified by ASHRAE.

2. On propose models simulation, adding diffusers show better temperature results, in which the variations of supply air velocity at the inlet diffuser, each generates a comfortable temperature, ie in the range of 24-26°C

3. The increasing velocity of air supply at the inlet diffuser, the air temperature on the surface of the worker's body will decrease, which is in line with the theory that an increase in air velocity, increase in Reynolds number, Nusselt Number and heat transfer coefficient. For constant heat flux, increasing heat transfer coefficient, the difference of surface temperature and the infinity temperature will be smaller, which means the surface temperature is almost equal to the infinity temperature, or in other words the surface temperature gets colder.

4. Beside influenced by the velocity of the supply air at the inlet diffuser, the temperature distribution on the body surface of workers affected by the position of each worker, For the same air supply velocity, the body surface temperature on the 1st worker is most high than the others, caused by the worker's position is most close to the heat source, followed by 3^{rd} workers then workers 5. While temperatures at 2^{nd} worker and 4^{th} worker most cold, caused by their positition are among the other workers, beside influence by the diffuser is located above their, also get the effect of diffusers above 1^{st}, 3^{rd}, and 5^{th} workers.

5. From the results, air supply velocity of 1,5 m/ is recommended as the air supply velocity at the inlet diffuser in Air Handling Unit (AHU) system design, For 20 units of diffuser, with 50cm x 50 cm of size, the mass flow rate must be supplied by the cooling device is 9 kg/s, therefore the system require a cooling capacity of 0,128 MW. This means that spot cooling method 58% more efficient than cooling the entire room.

References

[1] M. Bojic, F. Yik, and T. Lo, "Locating air-conditioners and furniture inside residential flats to obtain good thermal comfort," *Energy and Buildings*, vol. 34, no. 7, pp. 745–751, 2002.

[2] W. N. Hien, W. Liping, A. N. Chandra, A. R. Pandey, and W. Xiaolin, "Effects of double glazed facade on energy consumption, thermal comfort and condensation for a typical office building in singapore," *Energy and Buildings*, vol. 37, no. 6, pp. 563–572, 2005.

[3] A. Stamou and I. Katsiris, "Verification of a cfd model for indoor airflow and heat transfer," *Building and Environment*, vol. 41, no. 9, pp. 1171–1181, 2006.

[4] C. Helmis, J. Tzoutzas, H. Flocas, C. Halios, O. Stathopoulou, V. Assimakopoulos, V. Panis, M. Apostolatou, G. Sgouros, and E. Adam, "Indoor air quality in a dentistry clinic," *Science of the Total Environment*, vol. 377, no. 2, pp. 349–365, 2007.

[5] T. Karimipanah, H. Awbi, M. Sandberg, and C. Blomqvist, "Investigation of air quality, comfort parameters and effectiveness for two floor-level air supply systems in classrooms," *Building and Environment*, vol. 42, no. 2, pp. 647–655, 2007.

[6] K. Cheong, E. Djunaedy, Y. Chua, K. Tham, S. Sekhar, N. Wong, and M. Ullah, "Thermal comfort study of an air-conditioned lecture theatre in the tropics," *Building and Environment*, vol. 38, no. 1, pp. 63–73, 2003.

[7] K. Papakonstantinou, C. T. Kiranoudis, and N. Markatos, "Computational analysis of thermal comfort: the case of the archaeological museum of athens," *Applied Mathematical Modelling*, vol. 24, no. 7, pp. 477–494, 2000.

[8] Y. Wang, K. K. Wong, H. Du, J. Qing, and J. Tu, "Design configuration for a higher efficiency air conditioning system in large space building," *Energy and Buildings*, vol. 72, pp. 167–176, 2014.

[9] Q. Li, H. Yoshino, A. Mochida, B. Lei, Q. Meng, L. Zhao, and Y. Lun, "Cfd study of the thermal environment in an air-conditioned train station building," *Building and Environment*, vol. 44, no. 7, pp. 1452–1465, 2009.