Progress in Engineering Application and Technology Vol. 1 No. 1 (2020) 30–38
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: https://publisher.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 0000-0000

Air Flow Distribution Analysis by Using CFD Simulation

Sakira Ramli[1], **Fatimah Mohamed Yusop**[2*]

[1]UTHM Kampus Pagoh, Hab Pendidikan Tinggi Pagoh,
KM1, Jalan Panchor, 84600 Panchor, Johor.

*Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2020.01.01.004
Received 20 September 2020; Accepted 12 November 2020; Available online 02 December 2020

Abstract:A modern building has requirements need to consider increasing the quality of the building. In design the building, fresh air, cooling and heating is the important role to consider. The Heating, Ventilation and Air conditioning (HVAC) is most important part in building. Centralized system is the common use especially in large building. Usually, the system are using diffuser for air inlet and grille for air return. Therefore, the aim of this study is to determine the effectiveness of diffusers and grilles position in order to distribute the cool air into building. The objectives of this study are to perform a simulation in building and validate the simulation data with actual data. The CAD model was developed by using SolidWorks software. Then, the simulation was performed by using ANSYS-FLUENT software. The simulation result was validated by compare actual data. The result of temperature of air velocity of the room is not acceptable according to MS:1525:2014. The minimum velocity of the simulation data is 0.001m/s and the temperature is 17.0 °C and the maximum velocity is 3.5m/s and the temperature is 18.5 °C. The difference velocity of actual data and simulation for maximum and minimum velocity is 29.90 % and 99.50 % and for the maximum and minimum temperature difference is 0.53 % and 4.20 %. The minimum air velocity that acceptable for room air velocity based on Ms:1525:2014 is 0.15 m/s and the temperature is 23.0 °C. Therefore, the improvement of HVAC design is required.

Keywords: Centralized system, Computer Fluid Dynamic, velocity, temperature, ANSYS-FLUENT

1. Introduction

A building has requirements need to consider to increase the quality of the building. The process of build need many consideration to make sure the building safe for user. In building, the important part after construct the building is the services system that apply in building such as electrical system,

*Corresponding author: fatimahy@uthm.edu.my

telecommunication system, water supply system, drainage system and most important part is heating, ventilation and air conditioning system (HVAC).

HVAC system is an important part of system that can increase the performances and quality of the building. A HVAC system provides is to meet the requirement of comfort, cost, efficiency and aesthetic appeal.[1] The cooling in building is important and need to improve day by day in order to provide comfortable indoor environment. The absence of comfort condition will disturb people and effect to health. According to America Society Heating, Refrigeration and air conditioning Engineering (ASHRAE)-55 Standard, thermal comforts are state of mind of the fulfilment of the thermal condition.

The two important factors for indoor environment comfortable are indoor air quality (IAQ) and thermal comfort. In thermal comfort, the basically consist air humidity, air temperatures, mean radiant temperature, air velocity and activity of occupant. Based on ASHRAE standard 55-1981 the temperature and humidity ranges are comfortable to most people in large sedentary activity. In addition, air indoor quality and thermal comfort have significant impact to the occupant output productivity. The thermal comfort achieve by varying the condition of air supply. Based on the ASHRAE-55 standard section 5, the design of thermal environment condition which required is to satisfy a specific percentage of the tenant of the space. Air indoor quality (AIQ) is state as cleanness of air in room with regard human health.[2]

1.2 Problem Statement

The large buildings nowadays are mostly using ductwork in HVAC system. The ductwork distribution can increase the performance and minimize the cost. The ductwork have outlet and inlet vent that mostly use diffuser and grille. The design of the outlet vent must be consider the location installation because can affect the air distribution and air flow pattern in room.[9] Maintaining a balance air distribution through slot and diffusers is important in ensure comfort condition. In an indoor space, the unbalanced distribution air may sense lower or sense higher ambient parameter values.[3]

The poor air flow distribution can give an impact to the occupant activity. The poor air distribution will increase the odor and contaminant in spaces.[7] For example, the location of grille is close to the supply diffuser, air by passes the room and flow direct back into the return duct.[8] Other that, the closer can make a heat transfer and effect the circulation in system. Based on ASHRAE chapter 57, the intake should be located in the stagnant zone to return the warmest room air during heating. The air surround in area but the air characteristic is different based on the location. Recently, CFD are convenient tools are used to simulate the air flow. CFD analysis aids in understanding and optimizing the flow behavior through the complete intake system.[10] Therefore through this study will simulate the air flow distribution inside the building and able to determine the practical location of diffuser and grille that give good impact on thermal comfort.

1.3 Objective

The objectives of this study are:

- To perform air simulation in the building by using ANSYS-FLUENT
- To validate the air properties ANSYS-FLUENT data with actual data.

1.4 Scope of study

The scopes of this study are:

- Use ANSYS-FLUENT software for perform the air simulation.
- Validate the simulation data by comparing temperature and velocity with actual data

2. Methodology

Figure 1 show the flow chat of methodology to complete the study of air flow distribution by using CFD. The flow created to help achieve the objective of the study. The first objective is to identify the building physical, develop CAD model and run the simulation using ANSY-FLUENT. For second objective validate data simulation and analysis the data with actual data.

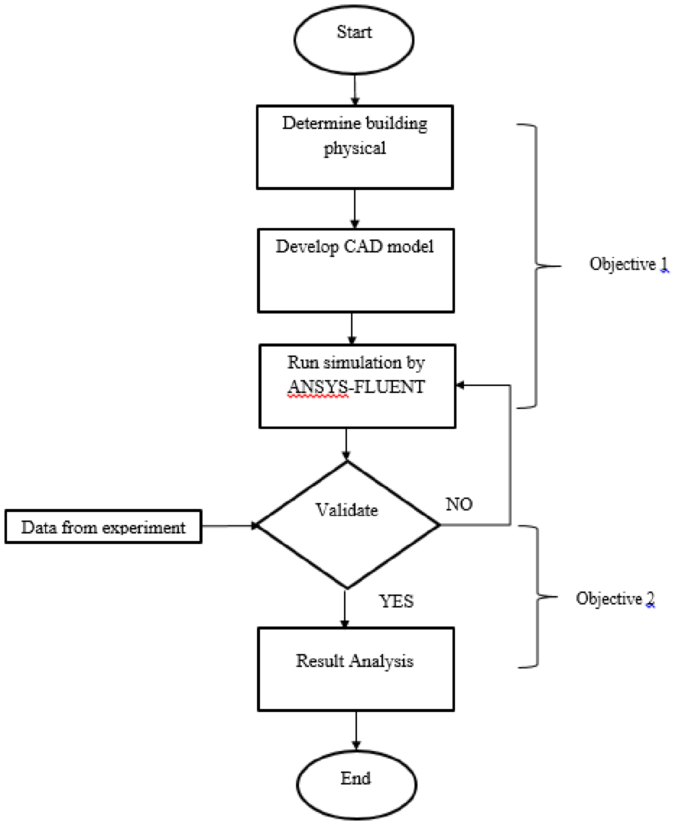


Figure 1: Flowchart of methodology

2.1 Building Specification

The location of the building is at Block C University Tun Hussien Onn Campus Pagoh, Johor. The process to determine the building physical have include the size of the building and number of outlet and inlet of space. Figure 2 shows the layout of selected building. The size of building is 25563 mm x 13625 mm and have 26 units diffusers for of air supply and 4 unit of grilles for return air.

Figure 2: Layout of selected building

2.2 Develop CAD model

The second stages is to develop the CAD model for the space by using SolidWork software. The CAD model is including the size of the diffuser inlet and grilles outlet. The size of the diffuser is 600mm x 600mm that refer to ASTI ceiling diffuser catalogue and TROX grilles catalogue for grilles size. The grilles size is 1225mmx 525mm. Figure 3 shows the CAD model for building that developed by using SolidWork software.

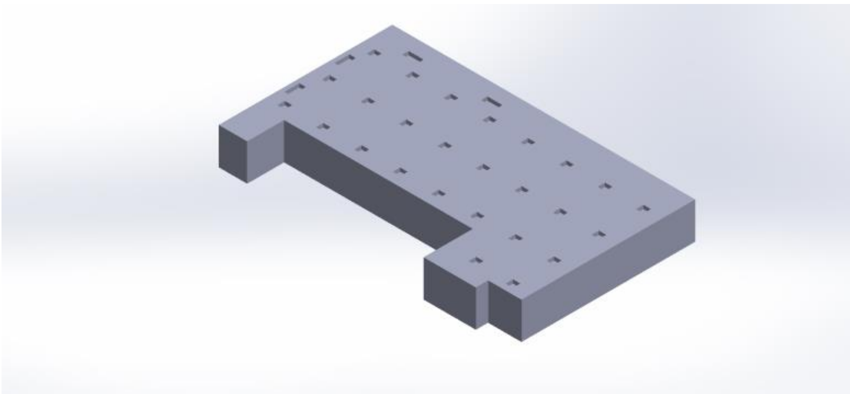


Figure 3: Show the CAD model using solidwork

2.3 Simulation process

The next stages is to perform the simulation by using ANSYS-FLUENT. Basically, simulation involved three process, called pre-processing, solver and post-processing in sequence as depicted in Figure 4.

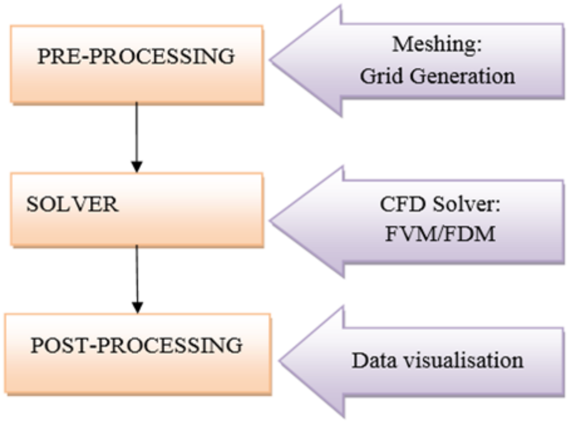


Figure 4: Basic Process in simulation

2.5.1 Grid generation

Grid generation is a process of meshing the CAD model into element. This study was used geometry for display mesh. The number of the element of the model is 3837 as shown in Figure 5 and Figure 6.

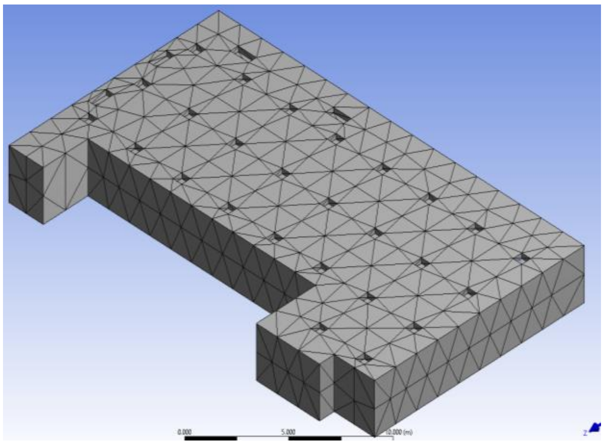
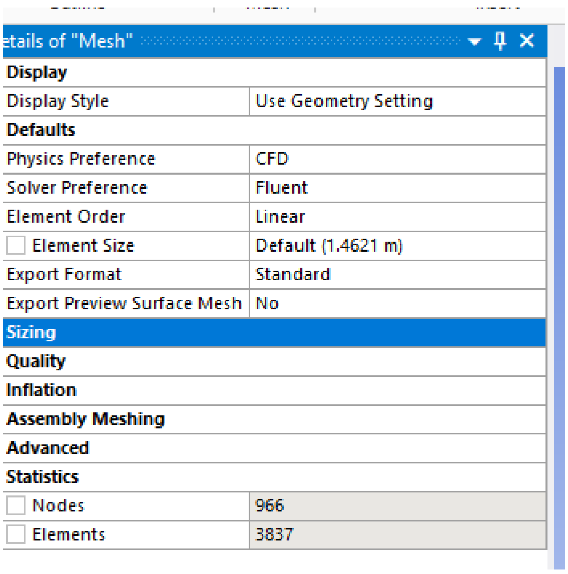


Figure 5: Mesh of the CAD model


etails of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	Default (1.4621 m)
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	
Nodes	966
Elements	3837

Figure 6: Properties of the mesh

2.5.2 Set up solver for simulation

The next process is setting up boundary condition and scheme for solver. The selected turbulence model is k-epsilon. The boundary condition set up for diffuser is velocity inlet and air return as pressure outlet.

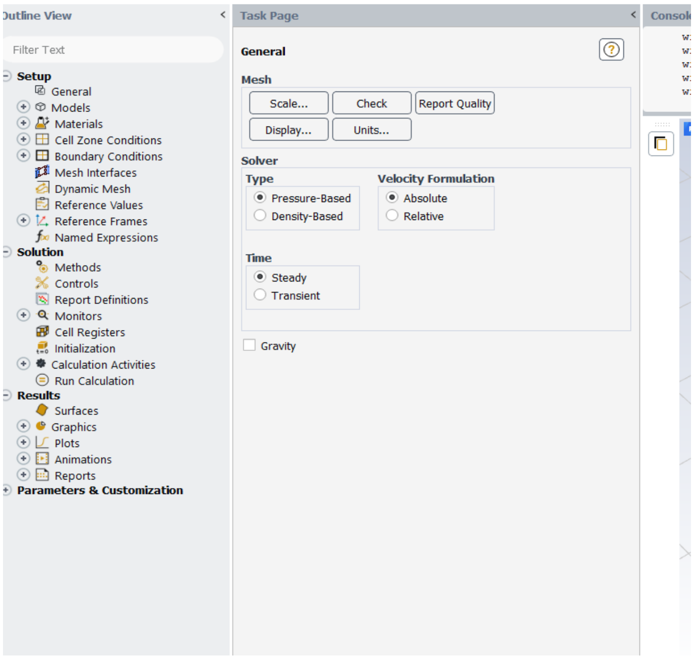


Figure 7: Selection of turbulence model

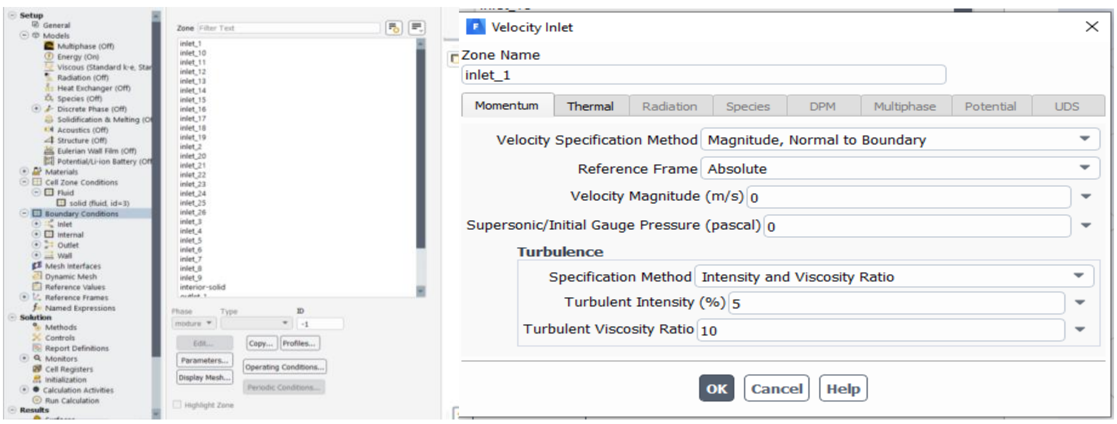


Figure 8: Boundary condition

4. Result and Discussion

Through the simulation, the result of the temperature and air velocity was obtained. Figure 9 and 10 show the temperature distribution in the building from the YX view and XZ view.

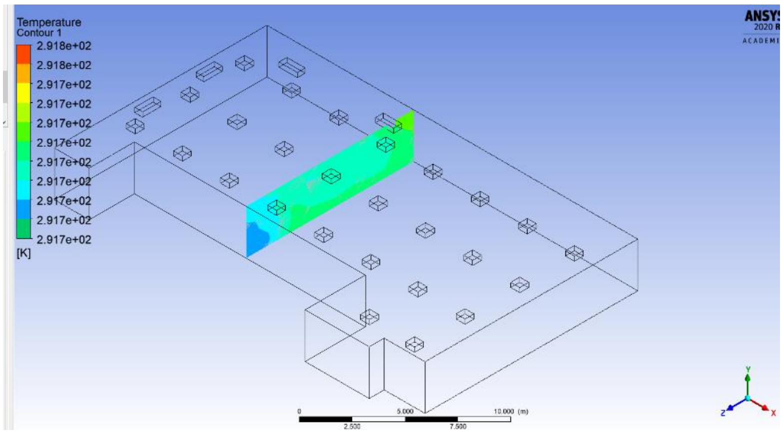


Figure 9: Temperature distribution from YZ view

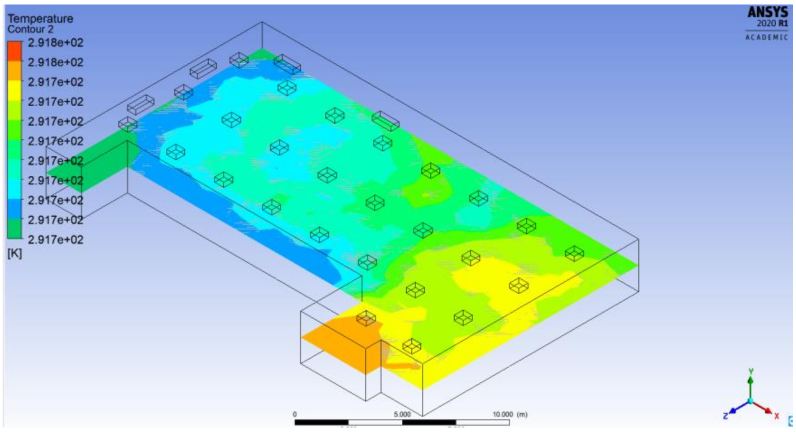


Figure 10: Temperature distribution from XZ view

The temperature result show the minimum temperature in "Makmal Struktur Ringan" is 291.8K(17.00 °C) and the maximum temperature is 291.7K (18.55 °C). The distribution of temperature at grilles area are lower than others. Besides that, based on the ASHRAE standard provide for HVAC design state the selected outlet is provide to adequate and direct away from return or exhaust it did not affect the short circuit. [4]

Meanwhile, Figure 11 and Figure 12 show the result of air velocity from YZ view and XZ view. The minimum velocity in room is 0.001 m/s and the maximum velocity is 3 m/s. The result indicated that front side which is far from grille has lowest air velocity. Dan Int - Hout in ASHRAE journal have state "In the real world, air movement toward return openings is imperceptible except within a few inches of the opening. In ASHRAE chapter 20 have state a guidelines for location of the return grilles it is "Openings should be located to minimize short-circuiting of supply air''. Addition, The return air should located at sufficient distance from the supply outlet to avoid short circuit.[5]

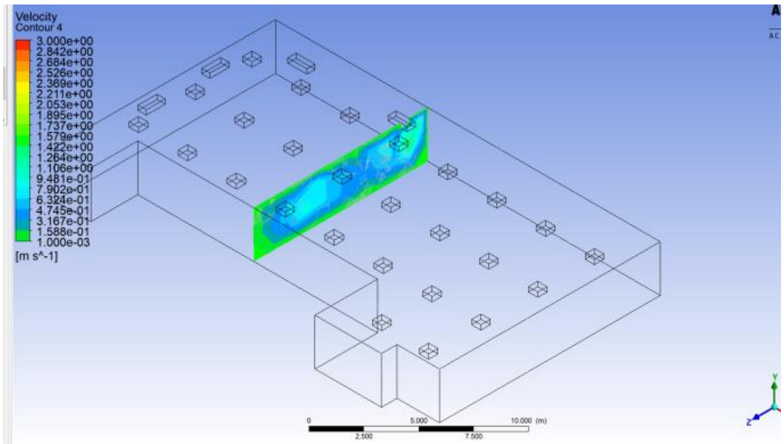


Figure 11: Velocity distribution from YZ view

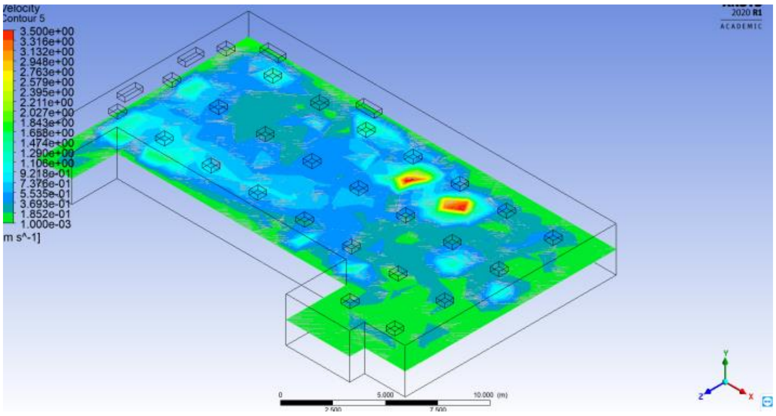


Figure 12: Velocity distribution from XZ view

4.1 Comparison result with actual data

Table 1 show the summary of temperature and velocity of actual and simulation data. The maximum temperature of the simulation is 18.50 % is 0.53 % lower and actual data. The difference between the minimum temperature for actual data and data simulation is 4.20 %. The 20.00 % is a different for maximum velocity of actual and simulation data. The difference of minimum velocity of actual and simulation data is 99.50 %.

Table 1: Comparison data between actual and simulation

	Temperature		Velocity	
	Max	Min	Max	Min
Actual Data	18.70 °C	18.5°C	5.562 m/s	0.466 m/s
Simulation Data	18.5 °C	17.0°C	3.500 m/s	0.001m/s
% different	0.53 %	4.20 %	29.90 %	99.50 %

5. Conclusion

An air flow distribution study in "Makmal Struktur Ringan" was present by using ANSYS-FLUENT software. The result indicated that diffuser and grille postion give impact to air distribution. At the back area of building which is around grille have lower temperature and higher velocity compare to the front area. Meanwhile, the simulation results was validated through temperature result with low percentage difference. The velocity result given higher different due to simulation constrain. The simulation performed by using student version of ANSYS-FLUENT software that have limited number of elements can be applied.

Through this study show that, the simulation can be used as a tool in order to predict the air distribution that provide by HVAC system. So that, other that follow ASHRAE standard for HVAC design, simulation can be use in design of system. This able to increase efficiency of HVAC system and reduce energy, cost and failure.

5.1 Challenging doing air flow distribution using CFD

- To learn Solidwork and ANSYS-FLUENT.
- To understand the simulation went getting floating point and to get a suitable setting
- Setting the time to measure the velocity and temperature in room.

Acknowledgement

The authors would also like to thank the Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Arsha Viswambharan,Sheetal Kumar Patidar,Khyati Saxena. (2014). Sustainable HVAC Systems in Commercial And. *International Journal of Scientific and Research Publications*, 1

[2] Kanbashi, A. (2017). STUDIES IN THE RESEARCH PROFILE BUILT ENVIRONMENT. *Experimental study of*,

[3] Dhakar, P. S. (2018). CFD Analysis of Air Conditioning in Room Using Ansys Fluent.*Research Gate*, 2-3

[4] Saadeddin, K. A. (2016). Open Praire. *The Effects of Diffuser Exit Velocity and Distance Between Supply and Return Apertures on the Efficiency of an Air Distribution System in an Office Space*,

[5] Dan Int-Hout and Leon Kloostra, Air Distribution for Large Spaces, Practical Guide, ASHRAE Journal, April 1999, 55-57

[6] Nielsen, P. V. (2015). Buildng and Environment. *Fifty years of CFD for room air distribution*, 79.

[7] S. Gilani, H. Montazeri. (2015). CFD simulation of stratified indoor environ ment in displacement ventilation. 1

[8] Wasserman, S. (2016, November 22). *engineerig.com*. Retrieved Dicember 1, 2019, from Choosing the Right Turbulence Model for Your CFD Simulation: https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/13743/Choosing-the-Right-Turbulence-Model-for-Your-CFD-Simulation.aspx

[9] Chen, Q. (2008). *Ventilation performance prediction for building: Amethod overview and recent applicationn.*

[10] Srujal Shah & Kari Dufva. (2017). *CFD MODELING OF AIRFLOW IN A KITCHEN ENVIRONMENT*. South-Eastern : Theseus.