Paper ID: 38

Numerical Analysis of Airflow and Temperature Distribution of an Air-Conditioned Room

Badhon Chakraborty[1], M A Wadud[1], Md. Mahbub Morshed Haque[1], Suchinto Roy Dhrubo[1]
[1]Department of Mechanical Engineering, Khulna University of Engineering & Technology (KUET), Khulna-9203, Bangladesh
E-mail: badhonchakraborty.kuet.me@gmail.com

Abstract

A comprehensive numerical analysis of airflow and temperature distribution within air-conditioned chambers is presented in this study. To address the governing equations of heat transfer and fluid flow, a three-dimensional model of a conventional room was constructed by employing computational fluid dynamics (CFD) via ANSYS Fluent. To gain insight into the efficacy and efficiency of different air conditioning system configurations, a thorough examination was conducted of the simulation results. The significance of numerical analysis in the development of air conditioning systems is highlighted by these results, which indicate that it can improve indoor comfort and energy efficiency. Utilizing CFD to optimize air vent placement and flow rates, evaluating the performance of various air conditioning configurations, and designing bespoke solutions for structures with unconventional geometries are all aspects highlighted in the study. To assess thermal comfort, three different air conditioning configurations operating at identical mass flow rates were investigated: a solitary AC duct positioned 2.73 meters above the floor, two AC ducts aligned along the same wall at an identical height, and two AC ducts facing each other, with the level of each duct being 2.73 meters. The room in question had the following dimensions: 6.85 × 5.73 × 3.52 meters. The third case was determined to be the optimal configuration, in which ducts positioned in opposition to one another produced superior distributions of temperature and air velocity, thereby improving the overall thermal comfort. Using intricate numerical simulations, this study establishes a methodological framework for attaining effective and efficient indoor climate management and illustrates the critical significance of precision air conditioning unit placement.

Keywords: Airflow Distribution, Temperature Distribution, Numerical Analysis, ANSYS Fluent, Computational Fluid Dynamics (CFD)

1. Introduction

People are spending more and more time inside, which means that air conditioning systems are becoming more and more necessary. Despite being more comfortable, these systems often lead to health problems such fatigue, coughing, and headaches [1]. Thus, in order to combine comfort and energy economy, air conditioning must be optimized. The benefits of thermal comfort outweigh the energy expenses by a large margin, particularly in the workplace. To address these issues, the use of computational fluid dynamics (CFD) is growing in the optimization of cooling unit placement and performance [2]. The realistic modelling of air temperature and velocity distribution provided by CFD allows for the assurance of thermal comfort with reduced energy use. Human well-being is directly impacted by building energy usage, as noted by Chung et al. [3], with the cooling system often using the most energy [4]. This research uses computational fluid dynamics (CFD) to measure the air velocity and temperature in a room with air conditioners at 2.5, 2.0, and 1.5 meters above the floor. The goal is to determine where comfort maintenance is most effective. The impact of changes in occupancy and human posture on airflow and temperature distribution is also examined in this research.

2. Mathematical Model

Design & Plane Selection

The research was carried out in a chamber that was devoid of heat and measured 3.52 meters in height, 5.73 meters in width, and 6.85 meters in length. The installation of air conditioning ducts was tested using three different layouts with different configurations. In Case 1, there was a single air conditioning duct that was 2.73 meters above the ground. When it came to Case 2, the duct was cut in half and placed on the same wall, with a height of 2.73 meters and a distance of 2.2 meters between each section. In

case 3, there were two duct sections that were positioned such that they faced each other and were the same height. A three-dimensional model of the space was constructed with the help of the Design Modular tool, and the space was simulated using ANSYS Fluent. The study was carried out with the assistance of ANSYS FLUENT 2020 R1.

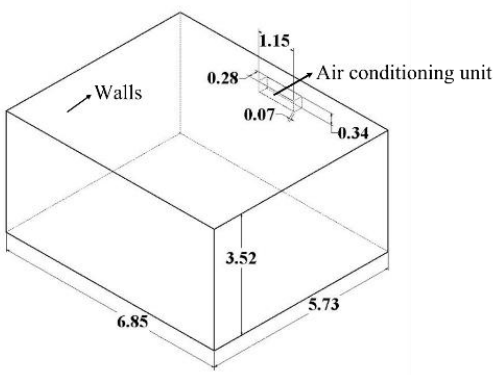


Fig. 1. Heat Free Chamber Dimension

Table 1. Duct Location Specification

Duct Location Specification				Plane Selection		
Cases	No. of Duct	Duct Height	Duct Location	Planes	Plane Height	Reference (Average Height of People)
Case 1	1	2.73m	Middle of the Wall	Plane A	1.72m	In Standing Position
Case 2	2		Same Wall (2.2m Apart)	Plane B	1.22m	In Sitting Position (Office)
Case 3	2		Opposite Wall	Plane C	0.85m	In Sitting Position (Prayer Room)

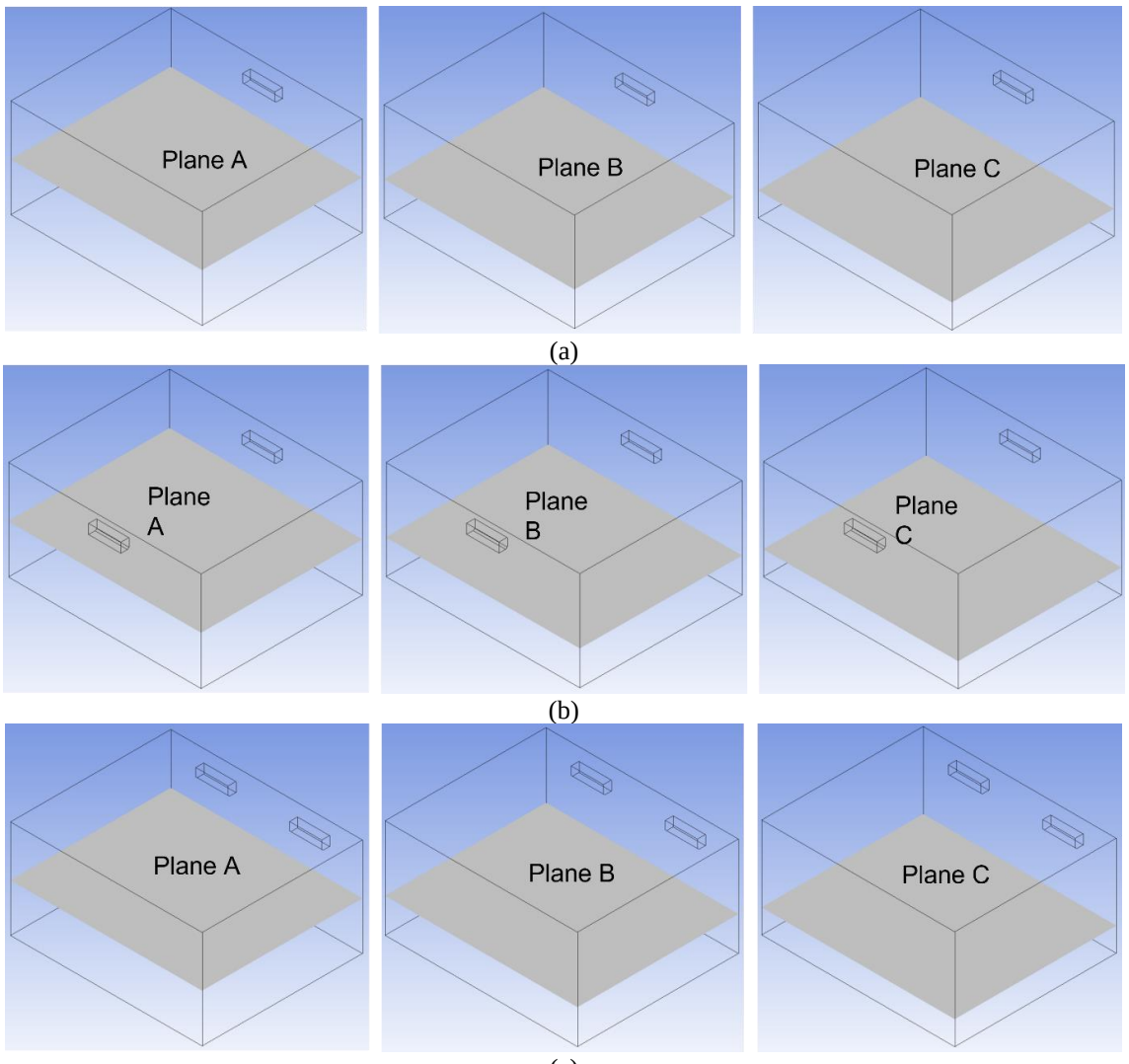
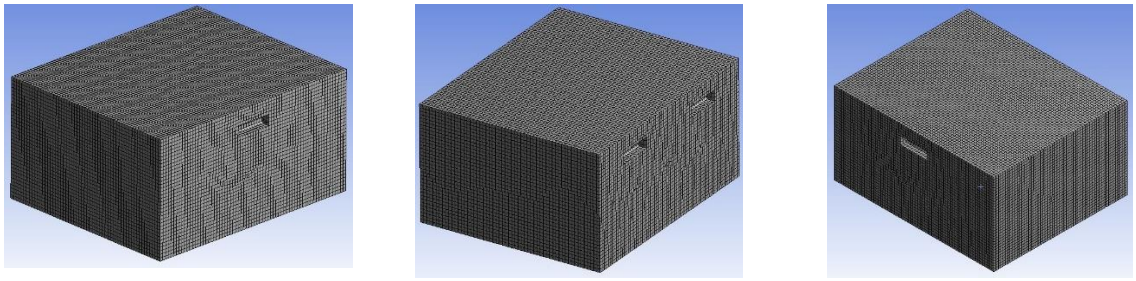


Fig. 2. Chamber 3D Model with Plane Selection for (a) Case 1, (b) Case 2 and (c) Case 3

Mesh Generation

In computational fluid dynamics (CFD) simulations, reliable results depend on a well-executed meshing procedure. In order to do this, face meshing to generate an adequate mesh was used in conjunction with a high relevance center and smoothing technique. Gravitational acceleration was taken into account in the simulation, and the Fluent setup launcher was configured with double precision. The realizable k-ε model, which has a scalable wall function and transport equations for turbulent kinetic energy and its dissipation rate, was used to represent turbulent flow. A constant pressure condition was employed at the outlet, while velocity type was used as the inlet boundary condition. For momentum, continuity, energy, K, and epsilon, the convergence requirements were set at 10^{-3} and for the energy equation, at 10^{-6}. The simulation was started and run through to the end.



(a) (b) (c)

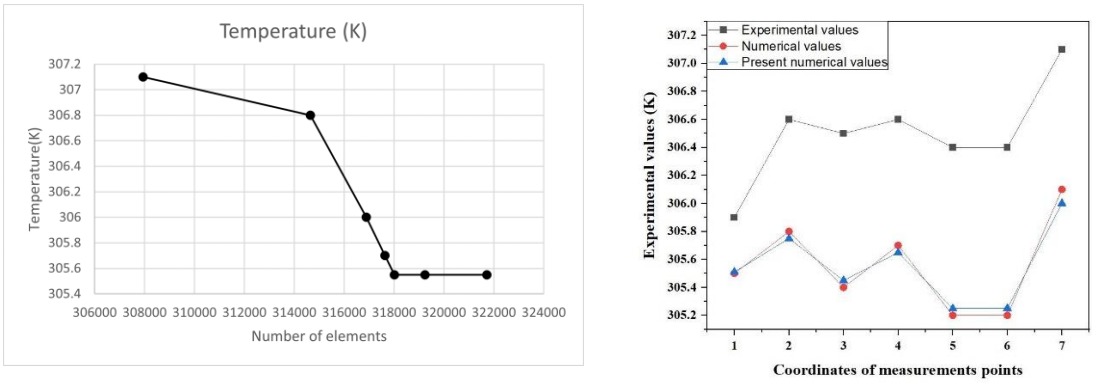
Fig. 3. Mesh Generation for (a) Case 1, (b) Case 2 and (c) Case 3

Boundary Condition

Table 2. Boundary Condition for different Cases

	Case 1	Case 2	Case 3
Inlet Temperature	290 Kelvins	290 Kelvins	290 Kelvins
Inlet Velocity	2 m/s	1 m/s	1 m/s
Flow Direction (from the AC bottom surface)	45^0 down	45^0 down	45^0 down
Outlet	Pressure Outlet	Pressure Outlet	Pressure Outlet
Room Walls	305 Kelvins	305 Kelvins	305 Kelvins
Mass Flow Rate	0.1945 kg⁄s	0.1945 kg⁄s	0.1945 kg⁄s

Mesh Independence Test & Model Validation



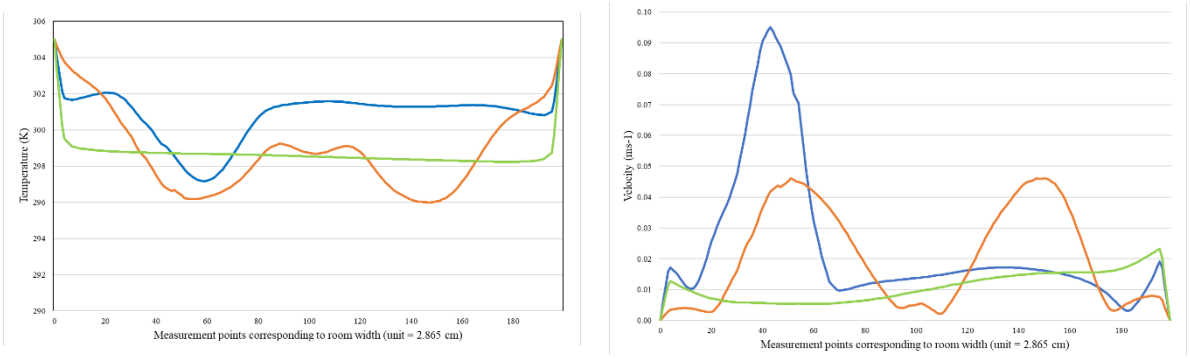
(a) (b)

Fig. 4. (a) Mesh Independence Test and (b) Model Validation

3. Result & Discussion

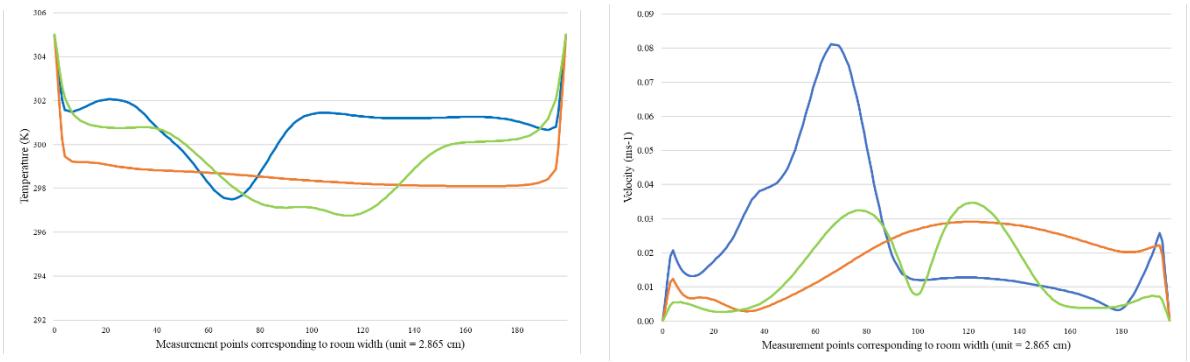
The purpose of this study is to explore a broad variety of various air conditioning (AC) duct layouts in order to discover the method in which these layouts effect the temperature as well as the velocity of airflow in a certain region. There were three distinct designs that were taken into consideration: one air conditioning duct that was situated 2.73 meters above the base floor, two air conditioning ducts that

were equal in height and were positioned on the same wall, and two air conditioning ducts that were situated on different walls. Each and every one of these designs was taken into consideration.



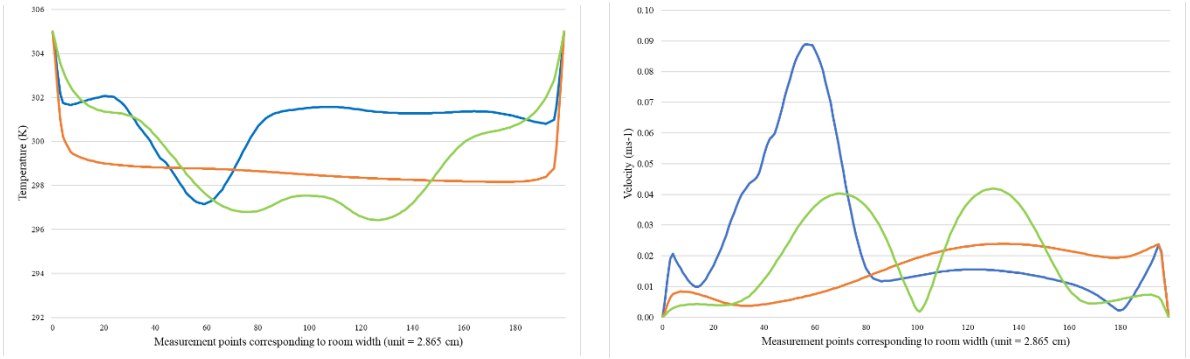
(a) (b)

Fig. 4. (a) Temperature Distribution and (b) Velocity Distribution of Plane A for Case 1 (—), Case 2 (—) and Case 3 (—)



(a) (b)

Fig. 5. (a) Temperature Distribution and (b) Velocity Distribution of Plane B for Case 1 (—), Case 2 (—) and Case 3 (—)



(e) (f)

Fig. 6. (a) Temperature Distribution and (b) Velocity Distribution of Plane C for Case 1 (—), Case 2 (—) and Case 3 (—)

To begin, the temperature range stayed quite consistent over the length of the experiment, remaining somewhat close to 305 Kelvin (31.85 degrees Celsius) during the entirety of the trial. In contrast, the temperature steadily declined in the second and third circumstances, with the lowest values being attained when two ducts were used. This brought about the best possible results. This indicated that the temperature was progressively falling into a lower range. This was the situation in each of the three different scenarios that may have occurred. The third scenario, in which the ducts were arranged in such a manner that they faced each other, demonstrated a far more successful performance in terms of

decreasing the temperatures across the board. When compared to the circumstance in which the ducts were situated on the same wall, this was a significant step forward.

Every single instance demonstrated that the airflow velocity reduced as the distance from the air conditioning ducts increased. This was the case in every single instance. This was shown in each and every incident that took place. According to the results of the research, the installation of two ducts on the same wall resulted in the highest airflow velocities, while the installation of a single duct resulted in the lowest airflow speeds (the lowest airflow speeds). It was discovered, however, that the temperature and velocity were more vulnerable to change depending on the conditions when there were a greater number of ducts.

Table 3. Result Summery

			Case 1	Case 2	Case 3
Plane A	Temperature (℃)	For Comfort	21 to 23		
		Minimum	24.15	22.97	21.50
	Velocity (m/s)	For Comfort	0.2 to 0.4		
		Maximum	0.10	0.05	0.20
Plane B	Temperature (℃)	For Comfort	21 to 23		
		Minimum	24.49	25.10	23.76
	Velocity (m/s)	For Comfort	0.2 to 0.4		
		Maximum	0.08	0.03	0.19
Plane C	Temperature (℃)	For Comfort	21 to 23		
		Minimum	24.15	25.16	23.42
	Velocity (m/s)	For Comfort	0.2 to 0.4		
		Maximum	0.09	0.11	0.30

The results indicate that the number of air conditioning ducts in a building as well as the location of those ducts have a significant influence on the factors that determine the thermal comfort and airflow dynamics of the building. This is the case regardless of whether the ducts are located in the building or outside the building. A design that has two ducts that are situated on opposing walls provides superior performance when it comes to the capacity of temperature regulation within the design.

9. Conclusion

This study emphasizes the need of numerical analysis when using airflow and temperature dispersion models to optimize air conditioning systems. With the use of CFD simulations and the construction of a 3D model, the study successfully investigated thermal comfort in various air conditioning setups. Among the three configurations that were examined, case 3 turned out to have the best thermal comfort settings. With two AC ducts facing each other and 2.73 meters above the ground, the space of 6.85 x 5.73 x 3.52 meters produced the optimum temperature dispersion. A further piece of evidence supporting the results came from the mass flow rate, which remained constant in all situations. This research emphasizes the need of properly designing your system to achieve both comfort and energy economy.

11. References

[1] Sookchaiya Thammanoon, Monyakul Veerapol, Thepa Sirichai. Assessment of the thermal environment effects on human comfort and health for the development of novel air conditioning system in tropical regions. Energy Build 2010;42(10):1692–702.

[2] Huizenga, Charlie, Abbaszadeh, Sahar, Zagreus and Leah; “Air quality and thermal comfort in office buildings: Results of a large indoor environmental quality survey”, Proceeding of Healthy buildings, Lisbon, Vol. III, 2006, pp. 393-397.

[3] Chung LP, Ahmad MH, Ossen DR, Hamid M. Application of CFD in prediction of indoor building thermal performance as an effective predesign tool towords sustainability. World Applied Sciences Journal 2014; 30:269-79.

[4] Kwong QJ, Adam NM, Sahari BB. Thermal comfort assessment and potential for energy efficiency enhancement in modern tropical buildings: A review. Energy and Buildings 2014; 68:547-57.

[5] Papakonstantinou KA, Kiranoudis CT, Markatos NC. Computational analysis of thermal comfort: The case of the archaeological museum of Athens. Applied Mathematical Modelling 2000; 24:477–94.

[6] Kendrick. An overview of combined modeling of heat transport and air movement. International Energy Agency. Technical Note AIVC 40.

[7] Bonefacic I, Frankovic B, Vilicic I, Glazar V. Numerical modelling of temperature and air flow distribution in enclosed room. Proceedings of the Heat-SET 2007: Heat Transfer in components and systems for sustainable energy technologies 2007; Vol II: 1055-62.
[8] Florian Antretter, Andreas Holm, Achilles Karagiozis, Samuel Glass, Interior Temperature and Relative Humidity Distributions in Mixed-Humid and Cold Climates as Building Simulation Boundary Conditons. ASHRAE, 2010. Referece: Rutman, E., Fournier, F., Jurkowski, R., Bailly, A., Franković, B., Viličić, I., 2002, Industrial Approach for CFD Modelling Applications for Air-conditioning and Heat Exchanger Systems, Energy and the Environment 2002, Vol. 2, Opatija, Croatia, pp.11-18.
[9] Lee, S.C., Cheng, C.Y., Chen, C.K., 1997, Finite elements solutions of laminar and turbulent flows with forced and mixed convection in an air-cooled room, Numerical Heat Transfer, Part A, 31, pp. 529–50.
[10] Sinha, S. L., 2001, Behaviour of Inclined Jet on Room Cooling, Building and Environment 36, pp. 569-578.
[11] Richtr, J., Katolicky, J., Jicha, M., 2001, Numerical modeling of the influence of angle adjustment of A/C diffuser vanes on thermal comfort in a computer room, Seventh International IBPSA Conference, Rio de Janeiro, Brazil, pp. 1135-1141.
[12] Zukowski, M., 2003, A Numerical Analisys of Heat and Mass Transfer in a Room with the Air-Underfloor Heating, 8th International IBPSA Conference, Bialystok Technical University, Poland.
[13] Wierciński & Aldona Skotnicka-Siepsiak. Application of CFD for Temperature and Air Velocity distribution calculation in a Ventilated Room.
[14] H. Begdouri, “Analysis of the impact of the location of a window type air-conditioner on thermal comfort in an office room”, M S thesis, 2005.
[15] Y. Huo, Y. Gao and W. K. Chow, “Locations of Diffusers on Air Flow field in an office”, Seventh Asia-Pacific conference on Wind Engineering, 2009.
[16] I. Bonefacic, B. Frankovic, I. Vilicic and V. Glazar, “Numerical Modeling of Temperature and Air Flow Distribution in Enclosed Room”, Strojniški vestnik - Journal of Mechanical Engineering, 2015, pp. 641-650.
[17] N. Jusoha, R. A. Bakara, A. R. Ismailb and T. Z. S. Ali, “Computational analysis of thermal building in a no-uniform thermal environment”, Energy Procedia 68, 2015, pp. 438 – 445.
[18] ISO 7730. Moderate thermal environment: determination of the PMV and PPD indices and specification of the conditions for Thermalcomfort. International Standard Organization: Geneva; 1994.
[19] Zukowski M. A numerical analysis of heat and mass transfer in a room with the air-underfloor heating. 8th International BPSA Conference. Eindoven: Netherlands 2003:1497-504.
[20] ASHRAE ANSI/ASHRAE Standard 55. Thermal environmental conditions for human occupancy.1992.
[21] Refrigeration and Air Conditioning - C. P. Arora.
[22] FLUENT 19.2(academic version) User's Guide, FLUENT Inc.,2017.
[23] Patankar, S.V., 1980, Numerical heat transfer and Fuid Flow, Washington DC, Hemisphere.